Editors:
Dr. Mostafa RANJBAR
M. Cihat YILMAZ

ICAMMEN@ybu.edu.tr
ICAMMEN 2018 Ankara, TURKEY
Dear valuable scholars, as the scientific chairman of ICAMMEN2018, it is my ultimate pleasure to welcome you sincerely to our conference. Me and my conference team are so excited to see you all here in Ankara. Ankara is the capital city of Turkey. In this city, there are several top-level state and private universities, research and industrial companies. Also, there are several others around the Ankara. They need to be connected locally and internationally. Therefore, ICAMMEN2018 has been established to provide a platform for such important and strategic scientific interaction in local and international levels. ICAMMEN 2018 has prepared to host scholars from the Turkey, neighbors and other countries to show their latest advances in mechanical and mechatronics fields in the highest level. We announced the first conference call in June 2018. Five months is a very short time to arrange such international conference. However, we faced with an unbelievable and prominent level of interests from Turkish and international colleagues. In a couple of months from the announcing of conference, we are proud to indicate that we got almost 200 good submissions from more than 400 authors from around 30 different countries in five continents like, Germany, USA, Canada, Columbia, Peru, Ecuador, Brazil, France, Italy, Portugal, United Kingdom, Slovenia, Slovakia, Hungary, Algeria, Ghana, Cameron, Nigeria, India, Iran, Iraq, Kuwait, Qatar, Saudi Arabia, Pakistan, Thailand, Malaysia, and Australia. Also, we received very good quality submissions from top rooted Turkish universities and industries like Middle East Technical University, Istanbul Technical University, Yıldız Technical University, Hachette University, Gazi University and all other universities that I cannot mention them here. We have also some valuable researchers who will present their latest studies from big companies like TOFAŞ, TUSAŞ, MAN, MKE, AVL and Valeo. At least 100 papers will be presented in the parallel technical sessions on solid mechanics, thermofluid/energy, manufacturing and material engineering during today afternoon and tomorrow morning.

With all this background, I wish you all a very beneficial conference and to make long term collaborations with each other’s.
Dear Ladies and Gentlemen. I want to say welcome to our conference with my sincere feelings. It is our first international conference in Mechanical Engineering field. It is a big step for us since we are a newly established university.

Let me give you some brief information about our faculty. We are providing education in English language to 2000 students with a good international student ratio in 8 different departments which are Electrical and electronics engineering, Computer engineering, Mechanical engineering, Material engineering, Civil engineering, Industrial engineering, Energy systems engineering and Mathematics. We have 142 academic staff. Furthermore, we have several foreigner academic members in our faculty. Furthermore, we are offering MSc and PhD programs in 5 majors including Electrical and electronics engineering, Computer engineering, Mechanical engineering, Material engineering and Energy systems engineering to more than 700 graduate students.

AYBU as a state university which is in the capital city of Turkey, we are aiming to take a key role in research & development, manufacturing and defense technologies by using of being in Ankara which is the capital city and the hearth of the Turkish defense industries.

Currently, we have focused on the multidisciplinary engineering fields. Several newly defined courses in the department of mechanical engineering are available for the first time in the region. Also, we are doing our best to be the pioneer of design and development of the new generation of Auxetics Structures and Metamaterials in the Turkey and the middle east. Furthermore, we are aiming to expand our regional scientific and industrial cooperation with our neighbor countries. This can be very beneficial for the region. Developing of joint national brands in various engineering fields is our wish.

ICAMMEN2018 can be a good platform for beginning of such local and international cooperation. I hope that you enjoy from ICAMMEN2018 and wish you a pleasant stay in Ankara.
Dear Ladies and Gentlemen.

I would like to welcome you all to our university. Although Ankara Yıldırım Beyazıt University is a newly established institution, it has managed to be a center of attraction for the academicians who are true professionals in their fields and has attracted the attention of successful students. The experienced, active and dynamic academic staff of Ankara Yıldırım Beyazıt University is mostly consisted of academicians who have obtained their undergraduate/graduate degrees from the pioneer universities of USA, Europe and Turkey. With almost 500 faculty members and more than 1000 academic staff in total, the university has managed to advance its academic human resources perfectly. Also, the university attracts attention of the students of high quality by offering English education. Furthermore, the university has been taking significant steps to position itself among the pioneer universities. At this regard, the undergraduate placement examinations result show that AYBU is placed within the first 5 in the university preferences among the Turkish universities. The university vision is to become a pioneer not only in Ankara or Turkey, but also in the world.

Our aim is to expand the scientific cooperation with our neighbor countries to establish an ecosystem for research and development and economic progress of this region which has a population number even more than European union. This can be done by developing joint industrial projects to develop joint national brands in various sections of engineering so that the people of our country and region can use them with a lower price and higher quality. We believe in developing of regional economy by developing scientific and industrial ties. In fact, together we are stronger.

Thank you very much for your attention and wish you a successful conference.
CONFERENCE MANAGING COMMITTEE

Prof. Dr. Metin DOĞAN
Honorary Chairman & Rector - Ankara Yıldırım Beyazıt University, TURKEY

Assoc. Prof. Dr. Mostafa RANJBAR
Scientific Chairman - Ankara Yıldırım Beyazıt University, TURKEY

Prof. Dr. Fatih V. ÇELEBİ
Chairman & Dean - Ankara Yıldırım Beyazıt University, TURKEY

Prof. Dr. Sadettin ORHAN
Ankara Yıldırım Beyazıt University, TURKEY

Prof. Dr. Veli ÇELİK
Ankara Yıldırım Beyazıt University, TURKEY

M. Cihat YILMAZ
Conference Secretary - Ankara Yıldırım Beyazıt University, TURKEY
INTERNATIONAL SCIENTIFIC COMMITTEE

Prof. Dr. Abdolreza OHADI - Amirkabir University of Technology, IRAN
Prof. Dr. Bekir Sami YILBAS - King Fahd University of Petroleum and Minerals, SAUDI ARABIA
Prof. Dr. Davood YOUNESIAN - Iran University of Science & Technology, IRAN
Prof. Dr. Fabrizio SCARPA - University of Bristol, UK
Prof. Dr. Ghodrat KARAMI - North Dakota State University, USA
Prof. Dr. Hamid AHMADIAN - Iran University of Science & Technology, IRAN
Prof. Dr. Hans-Jürgen HARDTKE - Technical University of Dresden, GERMANY
Prof. Dr. Hasan AMCA - Eastern Mediterranean University, TRNC - KKTC
Prof. Dr. Hasan DEMİREL - Eastern Mediterranean University, TRNC - KKTC
Prof. Dr. Ion STIHARU - Concordia University, CANADA
Prof. Dr. Karen ABRINIA - University of Tehran, IRAN
Prof. Dr. Kiumars MAZAHERI - Tarbiat Modares University, IRAN
Prof. Dr. Krzysztof WOJCEICHOWSKI - Polish Academy of Sciences, POLAND
Prof. Dr. Mojtaba AHMADI - Carleton University, CANADA
Prof. Dr. Steffen MARBURG - Technical University of Munich, GERMANY
Assoc. Prof. Dr. Andrew PELOW - Zayed University, UAE
Assoc. Prof. Dr. Ghulam HUSSAIN - GIK Institute, PAKISTAN
Assoc. Prof. Dr. Hasan HACİŞEVKİ - Eastern Mediterranean University, TRNC - KKTC
Assoc. Prof. Dr. Jamal AHMAD - Khalifa University, UAE
Assoc. Prof. Dr. Ali Vatankhah BARENJI - MIT, USA
Assoc. Prof. Dr. Dariush FIRUZI - McMaster University, CANADA
Assoc. Prof. Dr. Qasım ZEESHAN - Eastern Mediterranean University, TRNC - KKTC
Assist. Prof. Dr. Mohammad Sadegh MAZLOOMI - University of British Columbia, CANADA
Assist. Prof. Dr. Neriman ÖZADA - Eastern Mediterranean University, TRNC - KKTC
Assist. Prof. Dr. Roham RAFIEE - Iran University of Science & Technology, IRAN
Assist. Prof. Dr. Nima KAZEMI - MIT, USA
Assist. Prof. Dr. Mohsen PIRMOHAMMEDI - Azad University, IRAN
Assist. Prof. Dr. Davut SÖLYALI - Eastern Mediterranean University, TRNC - KKTC
Dr. Ali Vatankhah BARENJI - MIT, USA
Dr. Dariush FIRUZI - McMaster University, CANADA
Dr. Hossein TOWSFYAN - University of Southampton, UK
Dr. Mohammad Sadegh MAZLOOMI - University of British Columbia, CANADA
Dr. Stefan SCHNEIDER - Knorr Bremse Company, GERMANY
Mr. Mohammed ZALI - Engine Research & Design Company, IRAN
NATIONAL SCIENTIFIC COMMITTEE

Prof. Dr. Adem ÇİÇEK - Ankara Yıldırım Beyazıt University, TURKEY
Prof. Dr. Bülent YEŞİLATA - Ankara Yıldırım Beyazıt University, TURKEY
Prof. Dr. Erkan DOKUMACI - Dokuz Eylül University, TURKEY
Prof. Dr. Erol ARCAKLIOĞLU - Ankara Yıldırım Beyazıt University, TURKEY
Prof. Dr. İpek BAŞDOĞAN - Koç University, TURKEY
Prof. Dr. Mehmet Ali GÜLER - TOBB University, TURKEY
Prof. Dr. Mehmet ÇALIŞKAN - Middle East Technical University, TURKEY
Prof. Dr. Mehmet SUNAR - Ankara Yıldırım Beyazıt University, TURKEY
Prof. Dr. Oğuzhan YILMAZ - Gazi University, TURKEY
Prof. Dr. Sadettin ORHAN - Ankara Yıldırım Beyazıt University, TURKEY
Prof. Dr. Ünal ÇAMDALI - Ankara Yıldırım Beyazıt University, TURKEY
Prof. Dr. Veli ÇELİK - Ankara Yıldırım Beyazıt University, TURKEY
Assoc. Prof. Dr. Arif ANKARALI - Ankara Yıldırım Beyazıt University, TURKEY
Assoc. Prof. Dr. Mehmet BARAN - Ankara Yıldırım Beyazıt University, TURKEY
Assist. Prof. Dr. Fatih GÖNCÜ - Ankara Yıldırım Beyazıt University, TURKEY
Assist. Prof. Dr. Fatih ÖKTEM - Ankara Yıldırım Beyazıt University, TURKEY
Assist. Prof. Dr. Fatma D. GÜZEL - Ankara Yıldırım Beyazıt University, TURKEY
Assist. Prof. Dr. Hakan ARSLAN - Kırıkkale University, TURKEY
Assist. Prof. Dr. İhsan TOKTAŞ - Ankara Yıldırım Beyazıt University, TURKEY
Assist. Prof. Dr. Kemal BİLEN - Ankara Yıldırım Beyazıt University, TURKEY
Assist. Prof. Dr. Ramazan KARSLIOĞLU - Ankara Yıldırım Beyazıt University, TURKEY
Assist. Prof. Dr. Yasin SARIKAVAK - Ankara Yıldırım Beyazıt University, TURKEY
Dr. Egemen TİNAR - Arçelik Group, TURKEY
Dr. Mustafa İLARSLAN - AIRBUS Defence & Space, TURKEY
Dr. Reza VATANKHAH - Hacettepe University, TURKEY
Dr. Selçuk ALİMDAR - Tofaş, TURKEY
Dr. Selçuk ÇELİKEL - Ford Otosan, TURKEY
KEYNOTE SPEAKERS

Prof. Dr. Steffen MARBURG
Technische Universität München, GERMANY

The research interests of Steffen Marburg (b. 1965) encompass the development and application of numerical methods for vibroacoustics and aeroacoustics, the experimentally based virtual prototyping of complex models in combination with parameter identification, the consideration and identification of parameter variations and structural acoustic optimization. The applications are manifold and include automotive parts, ships, electric tools and musical instruments. Marburg graduated from TU Dresden where he was awarded a doctoral degree in 1998. He became a junior professor for Structural Acoustic Optimization/Boundary Element Methods in 2004. In 2010 he moved to the University of the Federal Armed Forces in Munich and became a full professor for Technical Dynamics. In 2015 he was awarded the new professorship for Vibroacoustics at TUM. Marburg is associate editor of the peer-reviewed journal Acta Acustica united with Acustica, editor of the Journal of Computational Acoustics and chair of the TC Computational Acoustics of the European Acoustics Association.

Keynote Speech: From Axioms of Continuum Mechanics to Quiet Structures - A Brief Survey of Vibroacoustic Modelling and Optimization

This talk will discuss the ingredients of simulation of low-frequency structural acoustic problems. Low frequency problems are defined in terms of the wave number and thus, assumes that only a limited number of waves is counted in the computational domain. Usually, a low frequency problem assumes normalized wave numbers $kL < 30\ldots100$ with $k$ as the wave number and $L$ as a characteristic length of the model. It is a weak assumption though. Often, low frequency acoustic problems are understood as purely deterministic, whereas high frequency problems are seen as non-deterministic and hence, statistic problems. This presentation will survey vibroacoustic modelling, model adjustment and model optimization. For this, modelling starts with the axioms of continuum mechanics in which linear elasticity is assumed for structures and fluids are compressible and lossless. Interaction between structure and fluid is formulated by two interface conditions. Since an analytical solution is only possible for a limited number of simple cases, industrially relevant problems are usually solved numerically, usually by using finite and boundary element methods, i.e. FEM and BEM. In many cases, it is easy to yield an accurate numerical solution. However, practical problems require knowledge of model parameters which account for the major problem in structural acoustics. Material parameters, parameters of joints and boundary conditions are often difficult to determine accurately and, additionally, exhibit different kinds of uncertainties. Parameter adjustment requires experimental investigations such as modal analysis. When finally arrived at a suitable simulation model, the results need to be assessed. For this interior and exterior problems are distinguished. While interior problems are often assessed based on local quantities, e.g. sound pressure at one or a few points, the radiated sound power accounts for a suitable quantity for exterior problems. Simulation models are optimized by minimizing an objective function to find a (much) better design than originally available. Throughout this talk, most of these steps will be presented together with different academic and industrial applications.
Prof. Dr. Bekir Sami YILBAS  
King Fahd University of Petroleum and Minerals, SAUDI ARABIA

Bekir Sami Yilbas obtained his PhD degree in Mechanical Engineering from Birmingham University in UK in 1982. He worked and affiliated with the University of Birmingham, Glasgow University, Erciyes University, University of Ontario Institute of Technology, Korean Institute of Science and Technology, Massachusetts Institute of Technology, and others. He is currently a Distinguished University Professor at King Fahd University of Petroleum & Minerals in Saudi Arabia. His research area covers laser machining and applications, surface sciences and engineering, thermal processing, and energy materials. He published over 800 papers in international journals and presented over 100 papers in conferences. He received many awards over the years due to his scientific achievements. Some of these include President of India’s Prize for 1988, the best researcher awards from KFUPM (1997, 2002, 2007), Silver Jubilee Medal for the outstanding achievements in Materials and Manufacturing 2005 by Silesian University of Technology, Poland, Doctor of Engineering Degree from Birmingham University (2005), Donald Julius Groen Prize for 2007 from by Institution of Mechanical Engineers (IMechE), Manufacturing Industries Division, UK, Professor W. Johnson International Gold Medal for 2008 by awarded by the Advances in Materials and Processing Technologies Steering Committee. Professor Fryderyk Staub Golden Owl Award by World Academy of Metals, and Almarai’s Distinguished Scholar Prize, awarded by King Abdulaziz City of Science and Technology in Saudi Arabia. He contributed to teaching and training of many graduate students in Mechanical Engineering and related fields.

**Keynote Speech: Texturing of Alloy Surfaces Towards Self Cleaning Applications**

Texturing of surfaces remains critical for self-cleaning applications. In the present study, laser gas assisted and repetitive pulse treatment of Ti6Al4V alloy surface is presented. The resulting surface texture characteristics and wetting state are analyzed using the analytical tools. The surface energy of the laser treated surface is determined adopting the contact angle method. It is demonstrated that laser repetitive pulse treatment results in hierarchically distributed micro/nano pillars. The wetting state of the laser treated surface remains hydrophilic because of the large gap size between the micro/nano pillars. The surface free energy of the laser treated surface is similar to that corresponding to the TiN coated surfaces, which is attributed to the nitride compounds formed during the laser treatment.
Prof. Dr. Davood YOUNESIAN  
Iran University of Science and Technology, IRAN

Dr. Younesian is Professor of Vibration and Acoustics at Iran University of Science and Technology. He received his PhD degree in Mechanical Engineering from Sharif University of Technology in 2005. He then attended a fellowship program at Institute of Sound and Vibration (ISVR) in the UK-2005, and a post-doctoral fellowship at University of Ontario Institute of Technology in Canada-2006. He has also served as Visiting Scholar at UC Berkeley. He is now acting as president of Iranian Society of Acoustics and Vibration (ISAV). His main research focus is on the modeling and identification of nonlinearities in dynamical systems. Professor Younesian has recently focused in application of nonlinear systems in vibration energy harvesting. He has authored or co-authored over two hundred publications including four books, journal articles, monographs, reports and conference papers.

**Keynote Speech: Application of Multi-Stable Systems in Broad-band Vibration Energy Harvesting**

Performance and characteristic behavior of different types of multi-stable systems in vibration energy harvesting is presented in this talk. A universal mechanism is introduced to provide diverse types of the broad-band energy harvesters. A short review of the new achievements and applications of the multi-stable systems is addressed. For a typical design, numerical results are presented for monochromatic and irregular wave regimes. Over a parametric study, effects of different design parameters including the power take off coefficient and characteristics of the restoring system on the efficiency of energy harvesting device are evaluated. It is shown how and in what extent, bistable and tristable systems can enhance the capture width ratio of the conventional point absorbers. It is shown that robustness of these two multi-stable systems with respect to the frequency and damping off-tuning is quite significant.
Prof. Dr. Zafer Evis
Middle East Technical University, TURKEY

Dr. Zafer Evis’s degrees are from Middle East Technical University (B.S., 1996) and from Rensselaer Polytechnic Institute (M.S., 1999; Ph.D., 2003, Post Doc., 2004). He is currently a Professor of Engineering Sciences at Middle East Technical University. His research includes the use of nanotechnology in biomedical applications. Specifically, his research focuses on synthesis and evaluation of bioceramics, scaffolds, biomimetic coatings, and biodegradable alloys. To date, his lab group has generated over 8 book chapters, 27 invited presentations, 62 peer-reviewed literature articles, 59 conference presentations, 13 projects, 2 awards, 11 M.S. thesis, 7 Ph.D. thesis, and 2 Post Doc. studies. He is an Associate member of Turkish Academy of Sciences.

Keynote Speech: Mechanical Properties of Polymer / Bioceramics Based 3D Porous Scaffolds in Hard Tissue Engineering

In recent years, significant progress has been performed in designing and producing polymer/ceramic based 3D porous scaffolds for hard tissue engineering applications. Conventional metals used in hard tissue applications currently are unsatisfactory due to density and mechanical mismatch between the implant and the hard tissues. Therefore, metal implants with higher mechanical strength than bone result in stress shielding. Moreover, some components of these materials might be toxic under in-vivo conditions. Polymer/bioceramic composites are currently investigated to solve the problems arising from the conventionally used biometals. This study investigates the contribution of pore parameters (Total porosity, pore size, pore distribution, pore morphology) on mechanical characteristics of scaffolds for hard tissue applications.
Prof. Dr. Fahrettin ÖZTÜRK
Turkish Aerospace Industry TAI, TURKEY

Dr. Fahrettin Ozturk is currently Vice President at Turkish Aerospace Industries, Inc. (TAI), Ankara, Turkey. He has responsibility for Strategy and Technology Management. Dr. Ozturk received all his degrees in Mechanical Engineering. His undergraduate degree from Selcuk University, Konya, Turkey in 1992, M.Sc. from University of Pittsburgh, PA, USA in 1996, and Ph.D. from Rensselaer Polytechnic Institute, Troy, NY, USA in 2002. Before joining the TAI, he has worked in universities and industry for many years. He is also jointly working in the Mechanical Engineering Department at Ankara Yildirim Beyazit University, Ankara, Turkey.

Keynote Speech: Composite Applications in Aerospace Industry

Combinations of lightness and performance (high strength, toughness, product life etc.) of aerostructures are extremely important in aerospace industry. Composites materials are key candidates to satisfy lightweight and exceptional mechanical properties requirements. Complex parts can easily be produced by using different manufacturing methods. The advantage of the parts help reduce number of parts, fasteners, and the assembly time in aircrafts. In recent years, the use of composite materials in aerospace industry have been increased gradually. Previously they are mostly used as secondary parts now they are highly used for primary structures such as fuselages and wings. Nowadays, more than 50% of some commercial aircrafts parts are composites. In near future, it is expected that the ratio will be increased more. In this present study, the applications of composite materials in aerospace industry will be summarized and discussed.
Dr. Mostafa Ranjbar got his PhD from Technical University of Dresden in Germany. He has worked in various top international universities and companies. He is an expert in the field of multidisciplinary engineering design optimization of complex structures. He has established a research group on Auxetics structures in Ankara Yıldırım Beyazıt University. He is currently working on the design of new Auxetics gradient sandwich panels with a focus on their vibroacoustic performance. He has published several papers in this fields in composite structures journal and journal of smart structures and materials. He is pioneer of Auxetics structures design and optimization field in the region and the middle East.

**Keynote Speech: From Cellular Solids to Auxetic Structures**

Cellular Solids can be found in nature, medicine and engineering applications. They are lightweight, can undergo large deformations, good insulation properties and can cover large surface areas. Auxetic structures are a branch of Cellular Solids which represent negative Poisson ratio futures. Their shear stiffness, plain strain fracture toughness, vibroacoustic performance and indentation resistance are superb.
# ICAMMEN 2018 CONFERENCE PROGRAM

**Thursday, 8th of November 2018**  
Venue: AYBU Faculty of Engineering and Natural Sciences (15 Temmuz Şehitleri Binası Etlik / ANKARA)

<table>
<thead>
<tr>
<th>Time</th>
<th>Session</th>
<th>Speaker(s)</th>
<th>Location</th>
</tr>
</thead>
<tbody>
<tr>
<td>08:30-10:00</td>
<td>Registration @ Block A, Ground Floor</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
| 10:00-10:30 | Opening Speech                                                          | Dr. Mostafa Ranjbar  
Conference Scientific Chairman  
*Ankara Yıldırım Beyazıt University* | Conference Hall (A212) |
| 10:00-10:30 | Welcome Speech                                                           | Prof. Dr. Fatih V. Çelebi  
Conference Chairman & Dean of Faculty of Engineering and Natural Sciences  
*Ankara Yıldırım Beyazıt University* |                  |
| 10:00-10:30 | Welcome Speech                                                           | Prof. Dr. Metin Doğan  
Honorary Chairman & Rector of *Ankara Yıldırım Beyazıt University* |                  |
| 10:30-11:00 | Application of Multi-stable Systems in Broad-band Vibration Energy Harvesting | Prof. Dr. Davood Younesian  
*Iran University of Science and Technology, Tehran, IRAN* |                  |
| 11:00-11:30 | Texturing Of Alloy Surfaces Towards Self Cleaning Applications           | Prof. Dr. Bekir Sami Yilbaş  
*King Fahd University of Petroleum and Minerals, SAUDI ARABIA* |                  |
| 11:30-11:45 | Coffee Break                                                            |                                                                           | 2nd and 3rd floor |
| 11:45-12:15 | Composite Applications in Aerospace Industry                             | Prof. Dr. Fahrettin Öztürk  
*Turkish Aerospace Industry (TUSAŞ), TURKEY* | Conference Hall (A212) |
| 12:15-12:45 | Mechanical Properties of Polymer / Bioceramics Based 3D Porous Scaffolds in Hard Tissue Engineering | Prof. Dr. Zafer Evis  
*Middle East Technical University (METU), TURKEY* |                  |
| 12:45-14:00 | Lunch (sponsored by Conference Committee)                               |                                                                           | 9th floor |
| 14:00-14:30 | From Cellular Solids to Auxetic Structures                              | Dr. Mostafa Ranjbar  
*Ankara Yıldırım Beyazıt University* | Conference Hall (A212) |
| 14:30-15:00 | From Axioms of Continuum Mechanics to Quiet Structures - A Brief Survey of Vibroacoustic Modelling and Optimization | Prof. Dr. Steffen Marburg  
*Technische Universität München, GERMANY* |                  |
| 15:00-15:15 | Coffee Break                                                            |                                                                           | 3rd floor |
| 15:15-16:30 | Paper Presentations (1st Round)                                         |                                                                           |                  |
| 16:30-16:45 | Coffee Break                                                            |                                                                           | 3rd floor |
| 16:45-18:00 | Paper Presentations (2nd Round)                                         |                                                                           |                  |
| 19:00-20:30 | GALA DINNER (Thursday 8th of November 2018)                             |                                                                           |                  |
### ICAMMEN 2018

**Friday, 9th of November 2018**

**Venue:** AYBU Faculty of Engineering and Natural Sciences (15 Temmuz Şehitleri Binası Etlik / ANKARA)

<table>
<thead>
<tr>
<th>Time</th>
<th>Event</th>
<th>Location</th>
</tr>
</thead>
<tbody>
<tr>
<td>09:00-10:15</td>
<td>Paper Presentations (3rd Round)</td>
<td></td>
</tr>
<tr>
<td>10:15-10:30</td>
<td>Coffee Break</td>
<td>3rd floor</td>
</tr>
<tr>
<td>10:30-12:00</td>
<td>Paper Presentations (4th Round)</td>
<td></td>
</tr>
<tr>
<td>12:00-14:00</td>
<td>Lunch (sponsored by Conference Committee)</td>
<td>9th floor</td>
</tr>
<tr>
<td>14:00-15:00</td>
<td>WORKSHOP #1 <code>Workshop on Auxetics Structures, Theory, Simulation and Applications</code></td>
<td>A 303</td>
</tr>
<tr>
<td>15:00-16:00</td>
<td>WORKSHOP #2 <code>Railway Noise and Vibration: Prediction, Measurement and Control</code></td>
<td>A 307</td>
</tr>
<tr>
<td>16:00-19:00</td>
<td>Visit to Ankara Hacı Bayram Mosque and Hamamönü Historical Places (sponsored by Conference Committee)</td>
<td></td>
</tr>
</tbody>
</table>
### CONFERENCE PROGRAM

**Thursday, 8th of November 2018**  
**Paper Presentations (1st Round), 15:15-16:30**

<table>
<thead>
<tr>
<th>Section 1</th>
<th>Chairman: Prof. Dr. Sadettin ORHAN (A 307)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ferina Saati and Steffen Marburg</td>
<td><strong>COMPARISON OF INVERSION MODELS FOR CHARACTERIZATION AND UNCERTAINTY MODELLING OF RIGID FRAME POROUS MATERIALS</strong></td>
</tr>
<tr>
<td>Ali Hosseinkhani, Davood Younesian and Mostafa Ranjbar</td>
<td><strong>A SHORT REVIEW ON VIBRO-AcouSTIC BEhAVIOR OF AUXETIC TWO-DIMENSIONAL STRUCTURES</strong></td>
</tr>
<tr>
<td>Hojjat Ghahramanzadeh Asl, Salim Cam, Osman Orhan and Özgü Bayrak</td>
<td><strong>ANALYSIS OF DENTAL IMPLANT ACCORDING TO NUMBER OF COMPONENTS IN TERMS OF APPLICABILITY AND MECHANICAL BEHAVIOR</strong></td>
</tr>
<tr>
<td>Mustafa Sarioğlu, Mehmet Seyhan and Yahya Erkan Akansu</td>
<td><strong>INVESTIGATION OF HALF NACA 0018 AIRFOIL ON TRUCK-TRAILER</strong></td>
</tr>
<tr>
<td>Mustafa Latif Çobankaya and Şerafettin Erel</td>
<td><strong>COMPARISON OF ELECTRICITY GENERATION IN A NEW PHOTOVOLTAIC APPLICATION BASED ON THE FIXED AND SINGLE AXIS SUN TRACKING SYSTEMS</strong></td>
</tr>
<tr>
<td>Mohammad Farid Khansanami, Davood Younesian and Mostafa Ranjbar</td>
<td><strong>A SHORT REVIEW ON THE MANUFACTURING, MECHANICAL PROPERTIES AND POTENTIAL APPLICATIONS OF THE AUXETIC BEAMS</strong></td>
</tr>
<tr>
<td>Kenan Nur, Ebubekir Beyazoglu and Erhan Pulat</td>
<td><strong>COMPUTATIONAL INVESTIGATION OF CONFINED WALL INCLINATION EFFECTS ON IMPINGING JET HEAT TRANSFER</strong></td>
</tr>
<tr>
<td>Abdullah Düzcan and Yusuf Ali Kara</td>
<td><strong>WIND ENERGY POTENTIAL IN ECEABAT, TURKEY</strong></td>
</tr>
<tr>
<td>Majid Karim Nejhad</td>
<td><strong>EXPERIMENTAL INVESTIGATION OF THERMOCHEMICAL HEAT STORAGE SYSTEM USING HYDRATED SALT BASED COMPOSITE SORBENTS FOR BUILDING SPACE HEATING APPLICATIONS</strong></td>
</tr>
<tr>
<td>Nima Khosravi and Mohammad Charidi</td>
<td><strong>INVESTIGATION OF DESICCANT ENHANCED EVAPORATIVE COOLING SYSTEM FOR BUILDING APPLICATION</strong></td>
</tr>
<tr>
<td>Arian Bahrami and Hasan Hacisevki</td>
<td><strong>VORTEX SHEDDING FROM TWO TANDEM NORMAL FLAT PLATES</strong></td>
</tr>
<tr>
<td>Fatih Selimefendigil and Hakan Öztop</td>
<td><strong>MIXED CONVECTION OF NANOFUID IN A VENTED CAVITY WITH MAGNETIC FIELD</strong></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Section 2</th>
<th>Chairman: Dr. Hasan ÖZCAN (A 303)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mohammad Farid Khansanami, Davood Younesian and Mostafa Ranjbar</td>
<td><strong>A SHORT REVIEW ON THE MANUFACTURING, MECHANICAL PROPERTIES AND POTENTIAL APPLICATIONS OF THE AUXETIC BEAMS</strong></td>
</tr>
<tr>
<td>Kenan Nur, Ebubekir Beyazoglu and Erhan Pulat</td>
<td><strong>COMPUTATIONAL INVESTIGATION OF CONFINED WALL INCLINATION EFFECTS ON IMPINGING JET HEAT TRANSFER</strong></td>
</tr>
<tr>
<td>Abdullah Düzcan and Yusuf Ali Kara</td>
<td><strong>WIND ENERGY POTENTIAL IN ECEABAT, TURKEY</strong></td>
</tr>
<tr>
<td>Majid Karim Nejhad</td>
<td><strong>EXPERIMENTAL INVESTIGATION OF THERMOCHEMICAL HEAT STORAGE SYSTEM USING HYDRATED SALT BASED COMPOSITE SORBENTS FOR BUILDING SPACE HEATING APPLICATIONS</strong></td>
</tr>
<tr>
<td>Nima Khosravi and Mohammad Charidi</td>
<td><strong>INVESTIGATION OF DESICCANT ENHANCED EVAPORATIVE COOLING SYSTEM FOR BUILDING APPLICATION</strong></td>
</tr>
<tr>
<td>Arian Bahrami and Hasan Hacisevki</td>
<td><strong>VORTEX SHEDDING FROM TWO TANDEM NORMAL FLAT PLATES</strong></td>
</tr>
<tr>
<td>Fatih Selimefendigil and Hakan Öztop</td>
<td><strong>MIXED CONVECTION OF NANOFUID IN A VENTED CAVITY WITH MAGNETIC FIELD</strong></td>
</tr>
<tr>
<td>Section 3 - Chairman: Prof. Dr. Adem Çiçek (A 315)</td>
<td></td>
</tr>
<tr>
<td>---</td>
<td>---</td>
</tr>
<tr>
<td>Selim Gürgen</td>
<td>ROLLER HEMMING OPERATION ON A VEHICLE DOOR</td>
</tr>
<tr>
<td>Selim Gürgen</td>
<td>RHEOLOGY OF CORNSTARCH SUSPENSION WITH CARBIDE PARTICLES</td>
</tr>
<tr>
<td>Selim Gürgen</td>
<td>INFLUENCE OF MANUFACTURING CONDITIONS ON WEAR PROPERTIES OF UHMWPE</td>
</tr>
<tr>
<td>Burçin Kaygusuz and Sezer Özörenç</td>
<td>MECHANICAL PROPERTIES OF PLA/PHA BLENDS FOR FUSED DEPOSITION MODELING APPLICATIONS</td>
</tr>
<tr>
<td>Fırat Memu, Nuri Durdu, Kaan Çiloğlu, Burcu Arslan Hamat and Yavuz Güleç</td>
<td>EFFECT OF MACHINING AND HOT ISOSTATIC PRESSING ON MECHANICAL PROPERTIES OF Ti-6Al-4V MANUFACTURED BY ELECTRON BEAM MELTING</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Section 4 - Chairman: Prof. Hasan Okuyucu (A 319)</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Wünesche Michael, Chuanzeng Zhang and Felipe Garcia-Sanchez</td>
<td>NUMERICAL INVESTIGATIONS OF LAYERED AND FUNCTIONALLY GRADED PIEZOELECTRIC MATERIALS WITH CRACKS UNDER IMPACT LOADINGS</td>
</tr>
<tr>
<td>Mehmet Fatih Oktem, Bahadir Aydas and Hasan Topacli</td>
<td>TECHNOLOGY AND INVESTIGATION OF MECHANICAL AND THERMAL PROPERTIES OF BIOCOMPATIBLE POLYURETHANE BASED SHAPE MEMORY POLYMER</td>
</tr>
<tr>
<td>Sencer Aydın and Sezer Özörenç</td>
<td>EFFECTS OF ANNEALING ON THE STRUCTURE AND MECHANICAL PROPERTIES OF 3D PRINTED PLA COMPONENTS</td>
</tr>
<tr>
<td>Iremnur Akçaçoca, Fatma D. Güzel, Ayşegül Bereketlioğlu, Araz N. Dizaji, Hamed Ghorbandpoor and Damion Corrigan</td>
<td>FABRICATION OF A MICROHEATER FOR BACTERIAL LYSIS</td>
</tr>
<tr>
<td>Mithat Gokhan Atahan</td>
<td>NUMERICAL INVESTIGATION OF THE EFFECTS OF ADHESIVE DEFECTS ON THE LOW-SPEED IMPACT BEHAVIOUR OF ADHESIVELY BONDED SINGLE-LAP JOINTS</td>
</tr>
<tr>
<td>Vahidullah Taç and Ercan Gürses</td>
<td>MICROMECHANICAL CHARACTERIZATION OF NANOTUBE REINFORCED POLYMERS</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Poster Presentation</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Aydin Güneş, Abdullah Aslan, Emin Salur and Ömer Sinan Şahin</td>
<td>EFFECT OF PRODUCTION TEMPERATURE ON VIBRATION BEHAVIOR OF COMPOSITE MATERIALS</td>
</tr>
<tr>
<td>Aydin Güneş, Emin Salur, Abdullah Aslan and Ömer Sinan Şahin</td>
<td>VIBRATION BEHAVIOR OF METAL MATRIX COMPOSITE MATERIALS PRODUCED AT DIFFERENT PRESSURES</td>
</tr>
<tr>
<td>Section 5 - Chairman: Dr. Gilsim TOPATEŞ (A 307)</td>
<td>Emre Alpman</td>
</tr>
<tr>
<td>---</td>
<td>---</td>
</tr>
<tr>
<td></td>
<td>Baris Zeytinci, Mehmet Ali Guler and Mehmet Sahin</td>
</tr>
<tr>
<td></td>
<td>Burcu Küçukoğlu Doğan and Tuncay Karaçay</td>
</tr>
<tr>
<td></td>
<td>Merve Akkaya and Mostafa Ranjbar</td>
</tr>
<tr>
<td></td>
<td>Emre Demirci and Ali Rıza Yıldız</td>
</tr>
<tr>
<td></td>
<td>Reza Mohammadzadeh Gheshlaghi and Hamit Akbulut</td>
</tr>
<tr>
<td>Section 6 - Chairman: Dr. Hasan ÖZCAN (A 303)</td>
<td>Ahmed Kouidri and Brahim Madani</td>
</tr>
<tr>
<td></td>
<td>Ilker Goktepeli, Ulas Atmaca and Sercan Yagmur</td>
</tr>
<tr>
<td></td>
<td>Okan Kocabıyık, Hürem Akbıyık, Mehmet Seyhan, Yahya Erkan Akansu and Mustafa Sarioğlu</td>
</tr>
<tr>
<td></td>
<td>Burak Tüzüner, Emre Alpman and Levent Ali Kavurmacıoğlu</td>
</tr>
<tr>
<td></td>
<td>Yahya Erkan Akansu, Mehmet Seyhan, Hürem Akbıyık and Okan Kocabıyık</td>
</tr>
<tr>
<td></td>
<td>Fikriye Bilaloğlu and Nezaket Parlak</td>
</tr>
<tr>
<td></td>
<td>Osman Öztürk, Mete Kalyoncu and Ali Ünüvar</td>
</tr>
<tr>
<td>Section 7 - Chairman: Dr. Yasin SARIKAVAK (A 315)</td>
<td>Section 8 - Chairman: Dr. İhsan TOKTAŞ (A 319)</td>
</tr>
<tr>
<td>--------------------------------------------------</td>
<td>--------------------------------------------------</td>
</tr>
<tr>
<td>Sadettin Orhan, Fahrettin Öztürk and Jabbar Gattmah</td>
<td>Mustafa Yildiz and Fatih Göncü</td>
</tr>
<tr>
<td>ANALYSIS OF COLD ROLLING PROCESS WITH DIFFERENT PARAMETERS USING FINITE ELEMENT METHOD</td>
<td>ESTIMATION OF EFFECTIVE ELASTIC PROPERTIES OF HONEYCOMBS WITH 1st ORDER CIRCULAR HIERARCHY</td>
</tr>
<tr>
<td>Jabbar Gattmah, Muzher Taha Mohamed and Suha K. Shihab</td>
<td>Seyyid Ali Özlçelik, Recep Emre Başar and Mostafa Ranjbar</td>
</tr>
<tr>
<td>SHEET METAL FORMING PROCESSES FOR VARIOUS MATERIALS USING FINITE ELEMENT ANALYSIS</td>
<td>A REVIEW ON SANDWICH PANEL STRUCTURES IN AUTOMOTIVE INDUSTRY</td>
</tr>
<tr>
<td>Bilal Kurşuncu and Cemre Can Alkan</td>
<td>Seyyid Ali Özlçelik and Mostafa Ranjbar</td>
</tr>
<tr>
<td>THE INVESTIGATION OF NANOCOMPOSITE HARD COATED CARBIDE CUTTING TOOL CUTTING PERFORMANCE IN THE MILLING OF INCONEL 718 SUPERALLOY USING ECO-FRIENDLY CUTTING FLUID IN THE MQL SYSTEM</td>
<td>A REVIEW ON DESIGN OF AUXETIC SANDWICH PANEL STRUCTURE</td>
</tr>
<tr>
<td>Bilal Kurşuncu and Cemre Can Alkan</td>
<td>Bahaa Eddin Dahkoul, Madiyar Alkhozhayev and Suat Kuyumcu</td>
</tr>
<tr>
<td>EXAMINATION OF THE EFFECT OF CRYOGENIC HEAT TREATMENT ON NANO-LAYERED CARBIDE CUTTING TOOLS IN MILLING INCONEL 718 SUPERALLOY</td>
<td>AUXETIC STRUCTURES: A REVIEW</td>
</tr>
<tr>
<td>Sait Özmen Eruslu</td>
<td>Murat Ünal, Dilan Kutmaral, Metehan Altay and Fatih Göncü</td>
</tr>
<tr>
<td>THE EFFECT OF PARTICLE TYPE AND DISTRIBUTION ON BENDING ANALYSIS OF GLASS PARTICLE REINFORCED COMPOSITE BEAMS</td>
<td>A REVIEW OF AUXETIC STRUCTURES WITH CHIRAL CORES FOR MORPHING WING APPLICATIONS</td>
</tr>
<tr>
<td>Bilal Kurşuncu and Cemre Can Alkan</td>
<td>Halim Kovacı</td>
</tr>
<tr>
<td>EXAMINATION OF THE EFFECT OF CRYOGENIC HEAT TREATMENT ON NANO-LAYERED CARBIDE CUTTING TOOLS IN MILLING INCONEL 718 SUPERALLOY</td>
<td>IMPROVING STATIC PROPERTIES AND FATIGUE LIFE OF METALLIC BONE PLATES MADE OF Ti6Al4V ALLOY USING PLASMA NITRIDING</td>
</tr>
</tbody>
</table>

Thursday, 8th of November 2018
Paper Presentations (2nd Round), 16:45-18:00
Friday, 9th of November 2018
Paper Presentations (3rd Round), 09:00-10:15

<table>
<thead>
<tr>
<th>Section 9 - Chairman: Dr. Mostafa RANJBAR (A 307)</th>
<th>1-D MODELLING AND SYSTEM OPTIMIZATION OF TRUCK POWERTRAIN SYSTEM USED WITH PTO FRONT DAMPER</th>
</tr>
</thead>
</table>
| Mehmet Onur Genç, Tolga Çakmak, Ahmet Koray Pehlivan and Necmettin Kaya | Kaan Ozgokhan and Hakan Yazıcı
| AN ARTIFICIAL NEURAL NETWORK (ANN) APPROACH FOR SOLUTION OF THE TRANSCENDENTAL EQUATION OF LONGITUDINAL VIBRATION |
| Gözde Rabia Aktaş, Abdullah Emül and Sadettin Orhan | DÜZ PERDELİ BİR SUSTURUCUDA PERDE KONUMUNUN SES DÜŞÜMÜNE ETKİSİNİN ARAŞTIRILMASI |
| Zeynep Mutlum, Hakan Arslan and Doğan Can Mutlum | THE INVESTIGATION OF STRESS DISTRIBUTION ON THE TRACTOR PTO WET CLUTCH COVER BY USING FINITE ELEMENT METHOD |
| Mehmet Onur Genç, Süleyman Konakçı and Necmettin Kaya | O zgur Erdogan
| DESIGN OF RUBBER DAMPER OF CLUTCH DISC USING RESPONSE SURFACE METHODOLOGY |

<table>
<thead>
<tr>
<th>Section 10 - Chairman: Dr. Metin AKTAŞ (A 303)</th>
<th>DESIGN AND APPLICATION OF A NEW SPLITTER PLATE-PRESSURE PROBE FOR WIND TUNNEL FLOW MEASUREMENTS</th>
</tr>
</thead>
<tbody>
<tr>
<td>Kasim Biber</td>
<td>A NEW TRANSONIC AIRFOIL DESIGNED AND OPTIMIZED FOR A FUTURE MALE UAV CONCEPT</td>
</tr>
<tr>
<td>Kasim Biber</td>
<td>CALCULATION OF PROPELLER-SLIPSTREAM DRAG AND ITS APPLICATION ON AN AIRPLANE PERFORMANCE</td>
</tr>
<tr>
<td>Kasim Biber</td>
<td>HYSTERESIS EFFECTS ON WIND TUNNEL TESTS OF AIRFOILS AND WINGS</td>
</tr>
<tr>
<td>Kasim Biber</td>
<td>DESIGN AND THERMAL DISTRIBUTION ANALYSIS OF 2 PHASE FLUID SYSTEM</td>
</tr>
<tr>
<td>Atakan Zeybek</td>
<td>NUMERICAL ANALYSIS OF THE THERMAL DISTRIBUTION OF A MACHINERY DRIVER’S CAB</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Section 10 - Chairman: Dr. Metin AKTAŞ (A 303)</th>
<th>DESIGN AND APPLICATION OF A NEW SPLITTER PLATE-PRESSURE PROBE FOR WIND TUNNEL FLOW MEASUREMENTS</th>
</tr>
</thead>
<tbody>
<tr>
<td>Kasim Biber</td>
<td>A NEW TRANSONIC AIRFOIL DESIGNED AND OPTIMIZED FOR A FUTURE MALE UAV CONCEPT</td>
</tr>
<tr>
<td>Kasim Biber</td>
<td>CALCULATION OF PROPELLER-SLIPSTREAM DRAG AND ITS APPLICATION ON AN AIRPLANE PERFORMANCE</td>
</tr>
<tr>
<td>Kasim Biber</td>
<td>HYSTERESIS EFFECTS ON WIND TUNNEL TESTS OF AIRFOILS AND WINGS</td>
</tr>
<tr>
<td>Atakan Zeybek</td>
<td>DESIGN AND THERMAL DISTRIBUTION ANALYSIS OF 2 PHASE FLUID SYSTEM</td>
</tr>
<tr>
<td>Hazal Koyuncu and Veli Çelik</td>
<td>NUMERICAL ANALYSIS OF THE THERMAL DISTRIBUTION OF A MACHINERY DRIVER’S CAB</td>
</tr>
<tr>
<td>Section 11 - Chairman: Prof. Dr. Adem Çiçek (A 315)</td>
<td>Reza Vatankhah Barenji</td>
</tr>
<tr>
<td>Section 11 - Chairman: Prof. Dr. Adem Çiçek (A 315)</td>
<td>Mehmet Alper Sofuoğlu</td>
</tr>
<tr>
<td>Section 11 - Chairman: Prof. Dr. Adem Çiçek (A 315)</td>
<td>Mehmet Alper Sofuoğlu</td>
</tr>
<tr>
<td>Section 11 - Chairman: Prof. Dr. Adem Çiçek (A 315)</td>
<td>Mehmet Alper Sofuoğlu</td>
</tr>
<tr>
<td>Section 11 - Chairman: Prof. Dr. Adem Çiçek (A 315)</td>
<td>Mehmet Alper Sofuoğlu</td>
</tr>
<tr>
<td>Section 11 - Chairman: Prof. Dr. Adem Çiçek (A 315)</td>
<td>Reza Vatankhah Barenji</td>
</tr>
<tr>
<td>Section 12 - Chairman: Dr. Fatih Öktem (A 319)</td>
<td>Gülsüm Meryem Dursun and Cihangir Duran</td>
</tr>
<tr>
<td>Section 12 - Chairman: Dr. Fatih Öktem (A 319)</td>
<td>Gülsüm Meryem Dursun and Cihangir Duran</td>
</tr>
<tr>
<td>Section 12 - Chairman: Dr. Fatih Öktem (A 319)</td>
<td>Emine Feyza Şükür, Süheyla Kocaman and Gürol Önal</td>
</tr>
<tr>
<td>Section 12 - Chairman: Dr. Fatih Öktem (A 319)</td>
<td>Yasin Öztürk, Güven Çankaya, Fatma D. Güzel and Halil İbrahim Sağ</td>
</tr>
<tr>
<td>Section 12 - Chairman: Dr. Fatih Öktem (A 319)</td>
<td>Sinem Demirsoy, Tuğçe Bayraktar, Gülsüm Meryem Dursun and Cihangir Duran</td>
</tr>
<tr>
<td>Section 12 - Chairman: Dr. Fatih Öktem (A 319)</td>
<td>Ramazan Karslıoğlu and Hatem Akbulut</td>
</tr>
<tr>
<td>Section 12 - Chairman: Dr. Fatih Öktem (A 319)</td>
<td>Zeycan Kalkan, Ece Gurcan, Uğur Barut and Ramazan Karslıoğlu</td>
</tr>
<tr>
<td>Section 13 - Chairman: Dr. Fatih GöNCÜ (A 307)</td>
<td>Section 14 - Chairman: Prof. Dr. Ünal ÇAMDAİL (A 303)</td>
</tr>
<tr>
<td>-----------------------------------------------</td>
<td>--------------------------------------------------</td>
</tr>
<tr>
<td><strong>Ismail Özçil, E. İlhan Konukseven and A. Buğra Koku</strong></td>
<td><strong>Ozan Kaya and Ahmet Sinan Oktem</strong></td>
</tr>
<tr>
<td><strong>Mehmet Emin Aktan and Erhan Akdoğan</strong></td>
<td><strong>Onur Kaya and Ahmet Sinan Oktem</strong></td>
</tr>
<tr>
<td><strong>Ozge Albayrak, Kutluk Arikan and Salar Rahimi</strong></td>
<td><strong>Onur Gunel and Mostafa Ranjbar.</strong></td>
</tr>
<tr>
<td><strong>İsmail Özçil, E. İlhan Konukseven and A. Buğra Koku</strong></td>
<td><strong>Mahmut Cihat Yılmaz and Sadettin Orhan</strong></td>
</tr>
<tr>
<td><strong>SLIPPAGE ESTIMATION OF A TWO WHEELED MOBILE ROBOT USING DEEP NEURAL NETWORK</strong></td>
<td><strong>CHARACTERIZATION OF OSCILLATIONS DURING FLOW BOILING OF WATER IN PARALLEL MICROCHANNELS</strong></td>
</tr>
<tr>
<td><strong>DESIGN AND DEVELOPMENT OF FOREARM PERIPHERAL MEASUREMENT DEVICE FOR USE IN INTELLIGENT ROBOTIC REHABILITATION</strong></td>
<td><strong>CONTACT FORCE CALCULATION OF AN ELECTRICALLY ACTUATED MICRO CANTILEVER BEAM SWITCH</strong></td>
</tr>
<tr>
<td><strong>A SINGLE DEGREE OF FREEDOM ROBOT ARM FOR HUMAN ROBOT INTERACTION</strong></td>
<td><strong>BUCKLING OF ISOTROPIC COLUMNS WITH UNCERTAIN DIMENSIONS AND MATERIAL PROPERTIES</strong></td>
</tr>
<tr>
<td><strong>A TEST BENCH TO STUDY THE TRANSLATIONAL PLATFORM STABILIZATION</strong></td>
<td><strong>REVIEW ON AUXETIC MATERIALS</strong></td>
</tr>
<tr>
<td><strong>Ozge Albayrak, Kutluk Arikan and Salar Rahimi</strong></td>
<td><strong>Mert Gülüm and Atilla Bilgin</strong></td>
</tr>
<tr>
<td><strong>DESIGN AND DEVELOPMENT OF FOREARM PERIPHERAL MEASUREMENT DEVICE FOR USE IN INTELLIGENT ROBOTIC REHABILITATION</strong></td>
<td><strong>DENSITY and VISCOSITY CHANGES of CORN OIL BIODIESEL-DIESEL FUEL BINARY BLENDS</strong></td>
</tr>
<tr>
<td><strong>DESIGN AND DEVELOPMENT OF FOREARM PERIPHERAL MEASUREMENT DEVICE FOR USE IN INTELLIGENT ROBOTIC REHABILITATION</strong></td>
<td><strong>TEMPERATURE AND BIODIESEL FRACTION DEPENDENCE OF DENSITY OF BIODIESEL-DIESEL FUEL BLENDS</strong></td>
</tr>
<tr>
<td><strong>DESIGN AND DEVELOPMENT OF FOREARM PERIPHERAL MEASUREMENT DEVICE FOR USE IN INTELLIGENT ROBOTIC REHABILITATION</strong></td>
<td><strong>DENSITY VARIATION OF ETHYL ESTER-DIESEL-BUTANOL TERNARY BLENDS</strong></td>
</tr>
<tr>
<td><strong>DESIGN AND DEVELOPMENT OF FOREARM PERIPHERAL MEASUREMENT DEVICE FOR USE IN INTELLIGENT ROBOTIC REHABILITATION</strong></td>
<td><strong>TURBOJET ENGINE AIR INTAKE MODAL ANALYSIS USING IMPACT TESTING METHOD</strong></td>
</tr>
<tr>
<td><strong>DESIGN AND DEVELOPMENT OF FOREARM PERIPHERAL MEASUREMENT DEVICE FOR USE IN INTELLIGENT ROBOTIC REHABILITATION</strong></td>
<td><strong>INVESTIGATION OF EXERGY COSTS OF AIR AND SOIL SOURCE HEAT PUMPS WORKING WITH DIFFERENT REFRIGERANTS</strong></td>
</tr>
<tr>
<td><strong>DESIGN AND DEVELOPMENT OF FOREARM PERIPHERAL MEASUREMENT DEVICE FOR USE IN INTELLIGENT ROBOTIC REHABILITATION</strong></td>
<td><strong>FARKLI YAYICİLİĞE SAHİP İKİ, ÜÇ VE DÖRT CAMLI PENCERELERE BAĞLI YAKIT TÜKETİMİ</strong></td>
</tr>
<tr>
<td><strong>DESIGN AND EXPERIMENTAL ANALYSIS OF HEAT STORAGE AIR COLLECTORS</strong></td>
<td><strong>DESIGN AND EXPERIMENTAL ANALYSIS OF HEAT STORAGE AIR COLLECTORS</strong></td>
</tr>
</tbody>
</table>
### Friday, 9th of November 2018
Paper Presentations (4th Round), 10:30-12:00

<table>
<thead>
<tr>
<th>Section 15 - Chairman: Dr. Kemal BİLEN / Dr. Ramazan KARSLIOĞLU (A 315)</th>
<th>Mehmet Seyhan, Mustafa Sarioğlu and Yahya Erkan Akansu</th>
<th>INVESTIGATION OF LEADING EDGE TUBERCLES ON NACA 0015 AIRFOIL</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hudaelaslam Abdali Salem Mohamed, Kayhan Dağdıır, Kemal Bilen and Erol Arcaklıoğlu.</td>
<td>INVESTIGATION OF THE EFFECT OF NANOREFRIGERANTS ON PERFORMANCE OF THE VAPOR COMPRESSION REFRIGERATION CYCLE: A REVIEW STUDY</td>
<td></td>
</tr>
<tr>
<td>Abid Ustaoglu and Murat Bertan Parıltı</td>
<td>CONSIDERING OF DEVELOPMENT POTENTIAL OF ABSORPTION REFRIGERATION SYSTEM UTILIZING ADVANCED EXERGY ANALYSIS METHOD</td>
<td></td>
</tr>
<tr>
<td>Abid Ustaoglu and Murat Bertan Parıltı</td>
<td>THERMODYNAMIC ANALYSIS OF ABSORPTION REFRIGERATION CYCLE UTILIZING ENHANCED EXERGY ANALYSIS METHOD</td>
<td></td>
</tr>
<tr>
<td>Edwin Ocaña, Leonardo Santana and Jorge Lino</td>
<td>STUDY OF PERMEABILITY AND HYDRAULIC CONDUCTIVITY OF 3D PRINTED PLASTER PARTS BY BINDER JETTING</td>
<td></td>
</tr>
<tr>
<td>Batukan Cem Tarhan and Kemal Bilen</td>
<td>A REVIEW ON OPTIMIZATION OF FUEL CELLS</td>
<td></td>
</tr>
<tr>
<td>Ozra Shirzadeh Shagaghí and Ersun Işçiöğlu</td>
<td>STUDENT’S PERSPECTIVES OF ICT USAGE FOR EDUCATIONAL PURPOSES: A CASE STUDY OF EASTERN MEDITERRANIAN UNIVERSITY MECHANICAL ENGINEERING</td>
<td></td>
</tr>
<tr>
<td>Onur Güngör and Veli Çelik.</td>
<td>SAFETY EVALUATION OF A FIBER REINFORCED COMPOSITE WRAPPED STEEL CYLINDER UNDER DYNAMICALLY APPLIED AXIALLY NON-UNIFORM INTERNAL SERVICE PRESSURE DISTRIBUTION</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Section 16 - Chairman: Dr. Begüm ÜNİVEROĞLU (A 319)</th>
<th>Barhm Mohamad</th>
<th>A REVIEW OF FLOW ACOUSTIC EFFECTS ON A COMMERCIAL AUTOMOTIVE EXHAUST SYSTEM</th>
</tr>
</thead>
<tbody>
<tr>
<td>Barhm Mohamad</td>
<td>A REVIEW OF FLOW ACOUSTIC EFFECTS ON A COMMERCIAL AUTOMOTIVE EXHAUST SYSTEM: METHODS &amp; MATERIALS</td>
<td></td>
</tr>
<tr>
<td>Mert Gülüm and Atilla Bilgin</td>
<td>PRODUCTION OF CORN AND HAZELNUT OIL METHYL ESTERS USING SODIUM HYDROXIDE AND METHYL ALCOHOL</td>
<td></td>
</tr>
<tr>
<td>Alperen Acar and Özen Ü. Çolak Çakır</td>
<td>MATERIAL MODEL DEVELOPMENT FOR NANOCOMPOSITES AND FINITE ELEMENTS METHOD IMPLEMENTATION</td>
<td></td>
</tr>
<tr>
<td>Özgen Colak and Deniz Uzunsoy</td>
<td>MACROSCOPIC TESTING AND NANOINDENTATION OF GRAPHENE-EPOXY NANOCOMPOSITES</td>
<td></td>
</tr>
<tr>
<td>Umut Koltan and Özgen Çolak</td>
<td>MECHANICAL PROPERTIES OF FLOWFORMED AISI 1.4330 GUN BARREL</td>
<td></td>
</tr>
<tr>
<td>Ismail Özen and Yasin Varol</td>
<td>INVESTIGATION OF THE EFFECT ON AERODYNAMIC PERFORMANCE OF STEPPED DIFFUSER IN A LAND VEHICLE</td>
<td></td>
</tr>
</tbody>
</table>
Effect of Production Temperature On Vibration Behavior of Composite Materials .............................................. 526
Vibration Behavior of Metal Matrix Composite Materials Produced At Different Pressures .......................... 530
Multi Objective Optimization of Cutting Parameters In A Single Pass Turning Operation Using The Bees Algorithm ........................................................................................................................................... 534
A Review of Auxetic Structures With Chiral Cores For Morphing Wing Applications ................................. 540
Comparison of Electricity Generation In A New Photovoltaic Application Based On The Fixed And Single Axis Sun Tracking Systems ............................................................................................................. 542
HYBRIDIZING BEST-WORST METHOD WITH DIFFERENT MCDM METHODS TO SOLVE NON-CONVENTIONAL MACHINING PROBLEMS

Mehmet Alper Sofuoğlu
The University of Eskişehir Osmangazi, Department of Mechanical Engineering, Eskişehir, Turkey
asofuoglu@ogu.edu.tr

ABSTRACT
Significant developments occurred in the machining operations with the rapid growth of information/space technology. Several non-conventional techniques become popular to machine complex shaped parts and hard to machine materials. To select appropriate non-conventional manufacturing techniques, multi-criteria decision-making models (MCDM) are often used. In this study, novel hybrid decision-making models are developed to choose proper non-traditional machining method. Best-Worst method (BWM) is combined with three different MCDM models. Engineers and operators can use the proposed models in the selection of non-conventional machining methods.

Keywords: MCDM, Best Worst method, Reference Ideal Method, Non-traditional machining, TOPSIS

NOMENCLATURE
AJM Abrasive jet machining
CHM Chemical machining
ECM Electrochemical machining
EDM Electrical discharge machining
EBM Electron beam machining
LBM Laser beam machining
USM Ultrasonic machining

1. INTRODUCTION

Important developments occurred in the manufacturing operations with the rapid change of technology. Therefore, traditional machining methods are combined with non-traditional machining methods. In recent years, several different non-conventional techniques have been developed. Generally, complicated shaped parts and hard to machine materials are processed using these techniques (Stout, 1992; Griffiths, 2001; Puertas and Luis Perez, 2003; Rajurkar and Ross, 1992; Yao et al., 2005).

In recent years, there have been several studies on multi-criteria decision-making models (MCDM). For MCDM techniques, several studies have been performed in the material science (Mardani et al., 2015; Jahan et al., 2011; Cavallini et al., 2013; Chatterjee et al., 2009; Shanian et al., 2008; Mayyas et al., 2011), production technologies (Streimikiene et al., 2012), mass production (Chang et al., 2013), manufacturing sector (Bagočius et al., 2013), manufacturing systems (Jana et al., 2013), global production (Tzeng and Huang, 2012) and production strategies (Yurdakul, 2004). Several studies have been published on MCDM for machining/material area (Buyurgan and Saygin, 2008; Îc et al., 2012; Yurdakul and Îc, 2009). Many studies have been conducted to analyze material selection problem via TOPSIS, ELECTRE, etc. (Rahman et al., 2012; Jahan and Edwards, 2013; Çalışkan, 2013; Chatterjee and Chakraborty, 2012; Khorshidi and Hassani, 2013).
One of the novel methods developed recently is BWM (Razaei, 2015). This method scores only the best and worst criteria. Therefore, pairwise calculations are only between best and worst criteria and calculations are simple. Besides, it is more consistent than AHP method. Reference Ideal method (RIM) is another new MCDM method to rank the alternatives (Cables et al., 2016).

When examining the past studies, it was observed that it is ambiguous in the weighting of machining characteristics. Generally, the weighting process is not based on a specific method. In this study, BWM is hybridized with three MCDM methods (Reference Ideal Model, TOPSIS, VIKOR). Furthermore, RIM and BWM are used for the first time in machining operations when examining studies carried out in the literature. Furthermore, these proposed hybrid models have not been developed before.

In this study, BWM, RIM, TOPSIS, and VIKOR methods are hybridized for the selection problems in machining. Two case studies are taken from the literature to test the models. Developed models are tested by using these non-traditional machining selection problems. In the second part, MCDM methods are examined. In the third part, case studies are given. In the fourth part, the results of hybrid MCDM techniques are explained. In the last part, the results are discussed, and the conclusion section is given.

2. METHODS

2.1. BWM

BWM is a newly developed method to determine criteria weights. The calculation procedure is given below (Razaei, 2015):

1: Specify decision-making criteria (c1, c2.....cn).
2: Specify the best and the worst criterion
3: Scoring of the best criterion versus the other criteria
   \[ a_{bj} = (a_{b1}, a_{b2}.... a_{bn}) \]
   \( a_{bj} \): The comparison scores of the best criterion B with the jth criterion.
4: Scoring of the other criteria versus the worst criterion
   \[ a_{jw} = (a_{1w}, a_{2w}.... a_{nw})^T \]
   \( a_{jw} \): The comparison scores of the worst criterion w with the jth criterion.
5: Find the optimum weights (w1*, w2*, w3* …… wn*) and index for consistency ratio (\( \epsilon^* \))

The method is defined below (Eq.1-4).

Min \( \epsilon \)
subject to

\[ \frac{w_j}{w_j} - a_{bj} \leq \epsilon \]  
\[ \frac{w_j}{w_j} - a_{jw} \leq \epsilon \]  
\[ \sum_j w_j = 1 \]  
\[ w_j \geq 0 \]

Consistency ratio formula is shown in Eq.5

\[ Consistency \ ratio = \frac{\epsilon^*}{consistency \ index} \]
2.2. RIM

In this method, different intervals are determined, so it includes subjectivity. The computation procedure is given below (Cables et al., 2016):

1: Normalization: In this step, reference ideal interval is ascertained. Eq. 6 gives the distance to reference ideal interval.

\[ d_{\text{min}}(x, [C, D]) = \min (|x - C|, |x - D|) \]  

Eq. 6

\[ X \] is the valuation for a given approach and the interval. [C, D] is the reference ideal. Normalization calculations are given in Eq.7.

\[ f(x, [A, B], [C, D]) = \begin{cases} 
1 & \text{si } x \in [C, D] \\
1 - \frac{d_{\text{min}}(x, [C, D])}{|A - C|} & \text{si } x \in [A, C] \land A \neq C \\
1 - \frac{d_{\text{min}}(x, [C, D])}{|D - B|} & \text{si } x \in [D, B] \land D \neq B
\end{cases} \]  

Eq. 7

where,


[C, D] Reference Ideal interval.

\[ x \in [A, B] \text{ and } [C, D] \subset [A, B] \text{ should be satisfied.} \]

2: Computation of the weighted normalized matrix Y

3: Computation of the variation for each option \( A_i \) (Eq.8-9).

\[ I_i^+ = \sqrt{\sum_{j=1}^{n} (y_{ij}' - w_j)^2} \]  

Eq. 8

\[ I_i^- = \sqrt{\sum_{j=1}^{n} (y_{ij}')^2} \]  

Eq. 9

\( m \) is the number of options, \( n \) is the number of criteria. \( y_{ij} \) is weighted normalized matrix values of ith option and jth criterion, \( w_j \) is the weight value for the jth criterion.

4: Computation of the relative index value \( R_i \) (Eq.10)

\[ R_i = \frac{I_i^-}{I_i^+ + I_i^-} \]  

Eq. 10

5: Ranking of the options.

3. CASE STUDIES

Two case studies taken from the literature are used to test the developed models. These case studies are briefly explained in this section. The detailed explanation of these studies is given in the reference (Yurdakul and Coğun, 2003).
3.1. Case Study-1

Electron beam machining is used for the drilling of turbine engine combustor domes. The wall thickness of the part is 1.1 mm, and the part is perforated with 3748 holes. Workpiece material is superalloy (35HRC). The diameter of the hole is 0.9 mm. The size tolerance is 0.05 mm, and the depth of the hole is 1.1 mm.

The criteria-alternatives matrix is given in Table 1. Seven non-traditional machining processes are presented according to six criteria. The objective of the study is to minimize surface finish, surface damage, taper, cost and maximize material removal rate (MRR), workpiece material (WM) (Yurdakul and Çoğun, 2003).

Table 1. Criteria/alternative matrix of the study

<table>
<thead>
<tr>
<th>Alternatives/criteria</th>
<th>Surface finish</th>
<th>Surface damage</th>
<th>Taper</th>
<th>MRR</th>
<th>WM</th>
<th>Cost</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.AJM</td>
<td>0.6</td>
<td>2.5</td>
<td>0.005</td>
<td>50</td>
<td>3</td>
<td>4</td>
</tr>
<tr>
<td>2.USM</td>
<td>0.5</td>
<td>25</td>
<td>0.005</td>
<td>500</td>
<td>1</td>
<td>5</td>
</tr>
<tr>
<td>3.ECM</td>
<td>1</td>
<td>0</td>
<td>0.001</td>
<td>2000</td>
<td>3</td>
<td>4</td>
</tr>
<tr>
<td>4.CHM</td>
<td>2</td>
<td>5</td>
<td>0.3</td>
<td>40</td>
<td>2</td>
<td>2</td>
</tr>
<tr>
<td>5.EDM</td>
<td>2</td>
<td>20</td>
<td>0.001</td>
<td>800</td>
<td>3</td>
<td>7</td>
</tr>
<tr>
<td>6.EBM</td>
<td>3</td>
<td>100</td>
<td>0.02</td>
<td>2</td>
<td>2</td>
<td>1</td>
</tr>
<tr>
<td>7.LBM</td>
<td>1</td>
<td>100</td>
<td>0.05</td>
<td>2</td>
<td>2</td>
<td>1</td>
</tr>
</tbody>
</table>

3.2. Case Study-2

Ultrasonic machining is used for precision drilling of a lot of holes at the same time. It is used to drill 930 holes with 0.64 mm diameter and 90 holes with 1.53 mm diameter. The operation is performed with stainless steel hypodermic needles as tools and 320 grit boron carbide as abrasives. Workpiece material is ceramic (non-conductive).

The criteria-alternatives matrix is given in Table 2. Five non-traditional machining processes are presented according to seven criteria. The study aims to minimize tolerance, surface finish, surface damage, taper, WM, cost and to maximize MRR (Yurdakul and Çoğun, 2003).

Table 2. Criteria/alternative matrix of case study-2

<table>
<thead>
<tr>
<th>Alternatives/criteria</th>
<th>Tolerance</th>
<th>Surface finish</th>
<th>Surface damage</th>
<th>Taper</th>
<th>MRR</th>
<th>WM</th>
<th>Cost</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.AJM</td>
<td>0.05</td>
<td>0.6</td>
<td>2.50</td>
<td>0.005</td>
<td>50</td>
<td>3</td>
<td>4</td>
</tr>
<tr>
<td>2.USM</td>
<td>0.013</td>
<td>0.5</td>
<td>25.00</td>
<td>0.005</td>
<td>500</td>
<td>3</td>
<td>5</td>
</tr>
<tr>
<td>3.CHM</td>
<td>0.03</td>
<td>2</td>
<td>5.00</td>
<td>0.3</td>
<td>40</td>
<td>1</td>
<td>2</td>
</tr>
<tr>
<td>4.EBM</td>
<td>0.02</td>
<td>3</td>
<td>100.00</td>
<td>0.02</td>
<td>2</td>
<td>3</td>
<td>1</td>
</tr>
<tr>
<td>5.LBM</td>
<td>0.02</td>
<td>1</td>
<td>100.00</td>
<td>0.05</td>
<td>2</td>
<td>3</td>
<td>1</td>
</tr>
</tbody>
</table>
4. RESULTS AND DISCUSSION

4.1. The Rankings of Case Study-1

In the case study -1, BWM is used to calculate criteria weights. Pairwise comparison of the study is presented in Table 3.

<table>
<thead>
<tr>
<th>Worst criterion: Surface damage</th>
<th>Surface finish</th>
<th>Surface damage</th>
<th>Taper</th>
<th>MRR</th>
<th>WM</th>
<th>Cost</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.5</td>
<td>1</td>
<td>1</td>
<td>0.33</td>
<td>0.25</td>
<td>0.14</td>
<td></td>
</tr>
<tr>
<td>Best criterion: Cost</td>
<td>5</td>
<td>7</td>
<td>3</td>
<td>2</td>
<td>3</td>
<td>1</td>
</tr>
</tbody>
</table>

Criteria weights of the study are shown in Table 4. The cost has the highest criterion weight. MRR, WM, taper, surface finish and surface damage are sorted in descending order respectively. In Table 5, the objective function value and consistency ratio are given. Consistency ratio is lower than 0.1, so the analysis is consistent.

<table>
<thead>
<tr>
<th>Criteria</th>
<th>Weights</th>
<th>Objective function</th>
<th>Consistency ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>Surface damage</td>
<td>0.0550</td>
<td></td>
<td>0.058</td>
</tr>
<tr>
<td>Surface finish</td>
<td>0.0805</td>
<td></td>
<td>0.013</td>
</tr>
<tr>
<td>Taper</td>
<td>0.0997</td>
<td></td>
<td></td>
</tr>
<tr>
<td>WM</td>
<td>0.1745</td>
<td></td>
<td></td>
</tr>
<tr>
<td>MRR</td>
<td>0.2094</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Cost</td>
<td>0.3808</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Criteria weights used in the study are 0.0805, 0.055, 0.0997, 0.2094, 0.1745, and 0.3808 for surface finish, surface damage, taper MRR, WM, and cost, respectively.

Range and reference ideal matrices are determined as follows.

Range matrix:
[A,B] = [0.5,3,0,100,0.001,0.3,2,2000,1,3,1,7]

Reference ideal matrix:
[C,D] = [0.5,0.6,0.2,5,0.001,0.005,800,2000,2,3,1,2]

RIM, TOPSIS and VIKOR models are developed. v is taken as 0.5 for VIKOR model. Vector normalization is used for TOPSIS model. The rankings of the models are shown in Table 5. The rankings are tested by using Spearman Correlation Test.
Table 5. Developed models for case study-1

<table>
<thead>
<tr>
<th>Developed models</th>
<th>Rankings</th>
</tr>
</thead>
<tbody>
<tr>
<td>RIM</td>
<td>5-6-1-3-7-4-2</td>
</tr>
<tr>
<td>TOPSIS</td>
<td>5-6-1-4-7-3-2</td>
</tr>
<tr>
<td>VIKOR</td>
<td>3-6-1-5-7-4-2</td>
</tr>
</tbody>
</table>

Correlation matrix of different methods is presented in Table 6. It is obtained that there is no difference between rankings at 5% significance level. When case study-1 is taken into consideration, electrochemical machining is the best alternative, whereas electrical discharge machining is the worst.

Table 6. Correlation matrix

<table>
<thead>
<tr>
<th>r/p</th>
<th>RIM</th>
<th>TOPSIS</th>
<th>VIKOR</th>
</tr>
</thead>
<tbody>
<tr>
<td>Yurdakul and Çağun (2003)</td>
<td>0.964/0.000</td>
<td>1.000/0.000</td>
<td>0.893/0.007</td>
</tr>
</tbody>
</table>

4.2. The Rankings of Case Study-2

Comparison of the best and worst criteria according to the other criteria is given in Table 7. Based on case study-2, the worst criterion is surface damage, whereas tolerance is selected as the best criterion. In Table 8, tolerance has the highest criteria weight which is 31%, whereas surface damage and taper have the lowest criteria weight which is 4%. Table 8 shows the objective function value and consistency ratio. Consistency ratio is nearly 0.1, so the analysis is consistent.

Table 7. Pairwise comparison of case study-2

<table>
<thead>
<tr>
<th></th>
<th>Tolerance</th>
<th>Surface finish</th>
<th>Surface damage</th>
<th>Taper</th>
<th>MRR</th>
<th>WM</th>
<th>Cost</th>
</tr>
</thead>
<tbody>
<tr>
<td>Best criterion: Tolerance</td>
<td>1</td>
<td>2</td>
<td>7</td>
<td>7</td>
<td>2</td>
<td>3</td>
<td>3</td>
</tr>
<tr>
<td>Worst criterion: Surface damage</td>
<td>0.14</td>
<td>0.2</td>
<td>1</td>
<td>1</td>
<td>0.2</td>
<td>0.33</td>
<td>0.33</td>
</tr>
</tbody>
</table>

Table 8. Weights of criteria and the other indices of case study-2

<table>
<thead>
<tr>
<th></th>
<th>Weights</th>
<th>Objective function</th>
<th>Consistency ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>Surface damage</td>
<td>0.042</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Surface finish</td>
<td>0.1934</td>
<td></td>
<td>0.1045</td>
</tr>
<tr>
<td>Taper</td>
<td>0.042</td>
<td></td>
<td></td>
</tr>
<tr>
<td>MRR</td>
<td>0.1934</td>
<td></td>
<td></td>
</tr>
<tr>
<td>WM</td>
<td>0.1094</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Cost</td>
<td>0.1094</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tolerance</td>
<td>0.3105</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
For the case study-2, BWM is used to calculate criteria weights. Criteria weights used in the study are 0.3104, 0.1934, 0.042, 0.042, 0.1934, 0.1094, 0.1094 for tolerance, surface finish, surface damage, taper, MRR, WM and cost, respectively.

Range and reference ideal matrices are determined as follows.

Range matrix:
\[ [A,B] = [0.013,0.05,0.5,3,2.5,100,0.005,0.3,2,500,1,3,1,5] \]

Reference ideal matrix:
\[ [C,D] = [0.5,0.6,0.2,5,0.001,0.005,800,2000,2,3,1,2] \]

RIM, TOPSIS and VIKOR models are developed. \( v \) is taken as 0.5 for VIKOR model. Vector normalization is used for TOPSIS model. The results of developed models are shown in Table 9. Spearman Correlation Test is performed to obtain correlation coefficients.

Table 9. The rankings of developed models for case study-2

<table>
<thead>
<tr>
<th>Developed models</th>
<th>Rankings</th>
</tr>
</thead>
<tbody>
<tr>
<td>RIM</td>
<td>5-1-2-4-3</td>
</tr>
<tr>
<td>TOPSIS</td>
<td>5-1-4-3-2</td>
</tr>
<tr>
<td>VIKOR</td>
<td>5-1-3-4-2</td>
</tr>
</tbody>
</table>

Correlation matrix of different methods is presented in Table 10. According to Spearman Correlation Test, there is no difference between rankings at 5% significance level except RIM.

Table 10. Correlation matrix

<table>
<thead>
<tr>
<th>r/p</th>
<th>RIM</th>
<th>TOPSIS</th>
<th>VIKOR</th>
</tr>
</thead>
<tbody>
<tr>
<td>Yurdakul and Çağun (2003)</td>
<td>0.7/0.188</td>
<td>1.0/0.000</td>
<td>0.9/0.037</td>
</tr>
</tbody>
</table>

When case study-2 is taken into account, ultrasonic machining is an appropriate alternative, whereas abrasive jet machining is the worst alternative.

5. CONCLUSION

In this research, novel hybrid MCDM models are suggested. BWM is used to calculate criteria weights and three MCDM methods are used to find the final rankings. The proposed models are tested using these studies. The obtained results show that calculated rankings are nearly the same. The developed model can be used for various problems. Also, different MCDM models may be hybridized with BWM and RIM.

REFERENCES


PREDICTION OF STABLE DEPTH OF CUTS USING PROBABILISTIC APPROACH IN TURNING

Mehmet Alper Sofuoğlu
Eskişehir Osmangazi University, Mechanical Engineering Department, Eskişehir, Turkey
asofuoglu@ogu.edu.tr

ABSTRACT

The calculation of stability boundaries of machining operations requires the modeling and parameterization of cutting forces and dynamic machine characteristics. Stability lobe diagrams can represent these boundaries. Uncertainties of machining parameters will lead to changes in the stability lobe diagram. Chatter vibrations in turning are one of the most critical problems that cause low workpiece quality and manufacturing efficiency. Therefore, determination of stable cutting depths is essential. Different vibrational characteristics affect stable cutting depths in machining operations. The vibration characteristics of these operations show randomness. In this study, a probabilistic approach and regression model are combined for turning operation to establish confidence intervals for stability diagrams.

Keywords: Chatter vibrations, stable cutting depths, stochastic approach, turning operation

1. INTRODUCTION

The determination of stability boundaries of machining operations is an important task and requires the modeling of the dynamic machine characteristics. To prevent chatter vibrations during machining, stability lobe diagrams are generally used, and the stability boundaries are considered. Uncertainties of machining parameters will induce changes in the stability lobe diagram. Several investigators presented various methods to compute confidence levels of calculated stability lobe diagrams. Different past methods for estimating chatter stability lobes assumed that vibrational parameters (mass, stiffness, and cutting coefficients) are constant. Nonetheless, in practical applications, these parameters can change (Shin and Lee, 2015; Li et al., 2015; Li et al., 2016; Chai et al., 2015, Löser and Großmann, 2016; Huang et al., 2016). Thomas et al. (2003) studied the effects of machining parameters on the cutting force, tool vibration, and change in modal parameters. Lee and Donmez (2007) examined the change in tool point dynamics describing the dynamics of the tool-holder-spindle system and machining stability. Liu et al. (2016) examined the probability distributions of model parameters using turning experiments. For machining parameters, Sahali et al. (2015) suggested a genetic algorithm to perform a multi-response robust optimization. In the machining operation, interval or fuzzy theory was used to show the uncertainty.

In this study, a probabilistic approach was proposed to predict stable cutting depths. Turning operation was investigated. A probabilistic approach was modeled using vibrational characteristics (natural frequency (w_n), stiffness coefficient (k) and damping coefficient (s)) affecting stable cutting depths in turning operations. An experimental study was conducted and a previous experimental study was also used (Türkeş, 2007). Probabilistic approach and regression model were combined for turning operation to establish confidence intervals for stability diagrams. Also, Mersenne Twister algorithm was used in random number generation for vibration characteristics. In the second section experimental and statistical model are explained. In the third section, the results of the model are discussed. In the final section, conclusions and recommendations are given.

2. EXPERIMENTAL/STATISTICAL STUDY

To investigate the vibration parameters affecting the stable cutting depth, different data in the lathe were obtained by the experimental work. Hammer test for turning operation is given in Figure 1. The parameters of the turning operation are w_n, k, and s. The experiments were carried out for Al-2024 materials using hammer test and modal analysis. The cutting coefficients (k,s) and natural frequency are obtained after modal analysis. CUT-PRO software was used during modal analysis. Also a study conducted by Türkeş (2007) was used (Table 1). The results of the performed experimental tests are shown in Table 2.

EASYFIT software was used during analysis to analyse the experimental data. This software tests 61 different continuous distributions. Using EASYFIT software, the goodness of fit test was performed by examining 61 continuous distributions. Kolmogorov Smirnov, Anderson Darling, and Chi-Square tests were used in this context. When the parameters of the distributions were determined, the maximum likelihood method was used.
Mersenne Twister algorithm is used to derive random numbers for distributions. This algorithm can generate pseudorandom numbers at high quality. Pseudo-random number generators have been extensively used in numbers of applications, especially simulation and cryptography. This algorithm is preferred in many statistical simulations because it gives positive results at high speeds. In this context, 10000 data are derived from the study.

Table 1. Experimental results of stable cutting depth in turning (AISI-1050) (Türkeş, 2007)

<table>
<thead>
<tr>
<th>The number of revolutions (rpm)</th>
<th>Tool overhang length (mm)</th>
<th>Natural frequency (Hz)</th>
<th>k (N/m)</th>
<th>s (%)</th>
<th>Stable cutting depth (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>90</td>
<td>80</td>
<td>1101.0</td>
<td>1.02E+07</td>
<td>3.97E-02</td>
<td>4.5</td>
</tr>
<tr>
<td>125</td>
<td>80</td>
<td>1101.0</td>
<td>1.02E+07</td>
<td>3.97E-02</td>
<td>4.0</td>
</tr>
<tr>
<td>180</td>
<td>80</td>
<td>1101.0</td>
<td>1.02E+07</td>
<td>3.97E-02</td>
<td>3.5</td>
</tr>
<tr>
<td>250</td>
<td>80</td>
<td>1101.0</td>
<td>1.02E+07</td>
<td>3.97E-02</td>
<td>2.8</td>
</tr>
<tr>
<td>355</td>
<td>80</td>
<td>1101.0</td>
<td>1.02E+07</td>
<td>3.97E-02</td>
<td>2.3</td>
</tr>
<tr>
<td>500</td>
<td>80</td>
<td>1101.0</td>
<td>1.02E+07</td>
<td>3.97E-02</td>
<td>1.8</td>
</tr>
<tr>
<td>710</td>
<td>80</td>
<td>1101.0</td>
<td>1.02E+07</td>
<td>3.97E-02</td>
<td>1.3</td>
</tr>
<tr>
<td>90</td>
<td>90</td>
<td>944.6</td>
<td>5.71E+06</td>
<td>4.15E-02</td>
<td>4.0</td>
</tr>
<tr>
<td>125</td>
<td>90</td>
<td>944.6</td>
<td>5.71E+06</td>
<td>4.15E-02</td>
<td>3.5</td>
</tr>
<tr>
<td>180</td>
<td>90</td>
<td>944.6</td>
<td>5.71E+06</td>
<td>4.15E-02</td>
<td>3.0</td>
</tr>
<tr>
<td>250</td>
<td>90</td>
<td>944.6</td>
<td>5.71E+06</td>
<td>4.15E-02</td>
<td>2.4</td>
</tr>
<tr>
<td>355</td>
<td>90</td>
<td>944.6</td>
<td>5.71E+06</td>
<td>4.15E-02</td>
<td>2.0</td>
</tr>
<tr>
<td>500</td>
<td>90</td>
<td>944.6</td>
<td>5.71E+06</td>
<td>4.15E-02</td>
<td>1.5</td>
</tr>
<tr>
<td>710</td>
<td>90</td>
<td>944.6</td>
<td>5.71E+06</td>
<td>4.15E-02</td>
<td>1.1</td>
</tr>
<tr>
<td>90</td>
<td>100</td>
<td>839.3</td>
<td>5.24E+06</td>
<td>1.75E-02</td>
<td>3.8</td>
</tr>
<tr>
<td>125</td>
<td>100</td>
<td>839.3</td>
<td>5.24E+06</td>
<td>1.75E-02</td>
<td>3.3</td>
</tr>
<tr>
<td>180</td>
<td>100</td>
<td>839.3</td>
<td>5.24E+06</td>
<td>1.75E-02</td>
<td>2.8</td>
</tr>
<tr>
<td>250</td>
<td>100</td>
<td>839.3</td>
<td>5.24E+06</td>
<td>1.75E-02</td>
<td>2.3</td>
</tr>
<tr>
<td>355</td>
<td>100</td>
<td>839.3</td>
<td>5.24E+06</td>
<td>1.75E-02</td>
<td>1.8</td>
</tr>
<tr>
<td>500</td>
<td>100</td>
<td>839.3</td>
<td>5.24E+06</td>
<td>1.75E-02</td>
<td>1.3</td>
</tr>
<tr>
<td>710</td>
<td>100</td>
<td>839.3</td>
<td>5.24E+06</td>
<td>1.75E-02</td>
<td>1.0</td>
</tr>
</tbody>
</table>
Table 2. Experimental results of stable cutting depth in turning (Al-2024)

<table>
<thead>
<tr>
<th>The number of revolutions (rpm)</th>
<th>Tool overhang length (mm)</th>
<th>Natural frequency (Hz)</th>
<th>k (N/m)</th>
<th>s (%)</th>
<th>Stable cutting depths (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>355</td>
<td>80</td>
<td>1343.2</td>
<td>3.70E+06</td>
<td>6.50E-02</td>
<td>4.7</td>
</tr>
<tr>
<td>500</td>
<td>80</td>
<td>1343.2</td>
<td>3.70E+06</td>
<td>6.50E-02</td>
<td>4.1</td>
</tr>
<tr>
<td>710</td>
<td>80</td>
<td>1343.2</td>
<td>3.70E+06</td>
<td>6.50E-02</td>
<td>3.5</td>
</tr>
<tr>
<td>355</td>
<td>90</td>
<td>1190.1</td>
<td>2.0E+06</td>
<td>7.79E-02</td>
<td>4.3</td>
</tr>
<tr>
<td>500</td>
<td>90</td>
<td>1190.1</td>
<td>2.0E+06</td>
<td>7.79E-02</td>
<td>3.6</td>
</tr>
<tr>
<td>710</td>
<td>90</td>
<td>1190.1</td>
<td>2.0E+06</td>
<td>7.79E-02</td>
<td>3.1</td>
</tr>
<tr>
<td>355</td>
<td>100</td>
<td>971.9</td>
<td>1.40E+06</td>
<td>2.20E-02</td>
<td>3.6</td>
</tr>
<tr>
<td>500</td>
<td>100</td>
<td>971.9</td>
<td>1.40E+06</td>
<td>2.20E-02</td>
<td>3.1</td>
</tr>
<tr>
<td>710</td>
<td>100</td>
<td>971.9</td>
<td>1.40E+06</td>
<td>2.20E-02</td>
<td>2.6</td>
</tr>
<tr>
<td>355</td>
<td>110</td>
<td>849.8</td>
<td>1.11E+06</td>
<td>4.40E-02</td>
<td>3.4</td>
</tr>
<tr>
<td>500</td>
<td>110</td>
<td>849.8</td>
<td>1.11E+06</td>
<td>4.40E-02</td>
<td>2.9</td>
</tr>
<tr>
<td>710</td>
<td>110</td>
<td>849.8</td>
<td>1.11E+06</td>
<td>4.40E-02</td>
<td>2.4</td>
</tr>
</tbody>
</table>

3. RESULTS AND DISCUSSIONS

3.1. The Results of Turning Operation

Different distributions were tested during the analysis (61 distributions). The goodness of fit tests for the coefficient of rigidity (k) were conducted. When the sequences are examined, it is seen that the best distribution is the Log logistic (3P) distribution according to Kolmogrov Smirnov, Anderson Darling, and Chi-Square. The goodness of fit test results for the Log-logistic distribution (3P) and the parameter values of the distribution according to the maximum likelihood estimation method are given in Table 3.

Table 3. The results of the goodness of fit test for Log-logistic distribution and parameter values of the distribution

<table>
<thead>
<tr>
<th>Statistic</th>
<th>Kolmogrov Smirnov</th>
<th>Anderson Darling</th>
<th>Chi Square</th>
</tr>
</thead>
<tbody>
<tr>
<td>p-value</td>
<td>0.09251</td>
<td>0.47089</td>
<td>0.51188</td>
</tr>
</tbody>
</table>

Parameter values: α=1.2115; β=2.7376e6; γ=4.2576e5

The goodness of fit tests for the coefficient of damping were performed. When the rankings are examined, it is obtained that the best distribution is the Log Pearson (3P) distribution. The goodness of fit test results for the Log Pearson distribution (3P) and the parameter values of the distribution according to the maximum likelihood estimation method are shown in Table 4.

Table 4. The results of the goodness of fit test for Log Pearson distribution and parameter values of the distribution

<table>
<thead>
<tr>
<th>Statistic</th>
<th>Kolmogrov Smirnov</th>
<th>Anderson Darling</th>
<th>Chi Square</th>
</tr>
</thead>
<tbody>
<tr>
<td>p-value</td>
<td>0.8225</td>
<td>*</td>
<td>0.99168</td>
</tr>
</tbody>
</table>

Parameter values: α=81.437; β=0.05505; γ=-7.8611

The goodness of fit tests for natural frequency were performed. The goodness of fit test results for the gamma distribution (3P) and the parameter values of the distribution according to the maximum likelihood estimation method are given in Table 5.
Table 5. The goodness of fit test for Gamma distribution and parameter values of the distribution

<table>
<thead>
<tr>
<th></th>
<th>Kolmogrov</th>
<th>Anderson</th>
<th>Chi</th>
</tr>
</thead>
<tbody>
<tr>
<td>Statistic</td>
<td>0.0894</td>
<td>0.38226</td>
<td>0.35365</td>
</tr>
<tr>
<td>p-value</td>
<td>0.85171</td>
<td>*</td>
<td>0.99651</td>
</tr>
</tbody>
</table>

Parameter values:
α=4.1233; β=155.46; γ=450.33

3.2. Regression Analysis for Turning Operation

Using the experimental data from Tables 1-2, the regression equations were derived which are given below (Eqs. 1-2). Eq. 1 is related to Türkeş’s study (Türkeş, 2007), and the other equation is associated with experimental work (Table 2). These equations were derived to obtain confidence interval curves and to show these curves clearly at the same time (Figs. 2-3).

\[
\text{Stable cutting depth}_1 = 0.138 - (4.43e - 3)\text{number of revolutions} + (5.5e - 3)\text{natural frequency} \\
- (1.9e - 7) k - 5.06 s \\
(R^2=0.89, \text{Adj. } R^2= 0.876, S=0.38)
\]

\[
\text{Stable cutting depth}_2 = 2.85 - (3.06e - 3)\text{number of revolutions} + (1.69e - 3)\text{natural frequency} \\
- (1.2e - 7) k + 2.14 s \\
(R^2=0.987, \text{Adj. } R^2= 0.98, S=0.097)
\]

In Figure 2, experiments performed in three different tool overhang lengths and graphs obtained at 5% - 95% confidence level are shown. It is seen that the experiments took place within the 5% - 95% confidence level. Stable cutting depths vary between 0.5 and 3.8 mm.

![Figure 2. Stable cutting depths of AISI-1050 material with different confidence levels and experiments.](image)

In Figure 3, the experiments performed in four different tool overhang lengths and the graphs obtained at the 5% - 95% confidence level are shown. It is seen that the tests took place within the 5% - 95% confidence interval. Stable cutting depths vary between 2 and 8.5 mm. As can be seen from the Figs. 2-3, the confidence interval can be narrow (Fig. 2) and wide (Fig.3) because different developed distributions affect lower and upper intervals differently.
4. CONCLUSIONS

In this study, a probabilistic approach was proposed to predict stable cutting depths in turning process. The log-logistic distribution for k, the log Pearson distribution for s, and the gamma distribution for \( w_0 \) were found suitable for turning operation. Regression analysis was performed for the turning process using experimental studies from the literature and confidence levels were determined. This study helps engineers and operators to prevent chatter vibrations using confidence level stability diagrams in the manufacturing environment. In forthcoming studies, the approach of confidence level might be used for the other milling (end milling, up milling, down milling), and drilling operations. Different integrated models (probabilistic-analytical) can be developed. Also, different probabilistic-analytical can be developed for turning operation.

REFERENCES


LOW-FREQUENCY VIBRATION ASSISTED MACHINING METHODS:
A CASE STUDY OF A TITANIUM ALLOY

Mehmet Alper Sofuoğlu
The University of Eskişehir Osmangazi, Mechanical Engineering Department, Eskişehir, Turkey
asofuoglu@ogu.edu.tr

ABSTRACT

In the aviation industry, several machining methods have been proposed to improve machining efficiency. One of the innovative machining methods is vibration assisted machining, which shows better results compared to the traditional machining in the literature. This method increases manufacturing performance and workpiece quality. The essential characteristic of the vibration assisted machining method is that it makes intermittent contact which increases the surface quality of the workpiece and decreases the cutting energy. Permanent tool-workpiece contact is periodic. The process temperature is lower than traditional machining because of the separation between the workpiece-cutting tool. In a vibration assisted machining method, the removal rate of the chips in each vibration cycle is less than conventional machining. In general, reduction of cutting force, tool wear stresses and temperature are obtained. Low-frequency vibration assisted machining is a method which uses controlled external vibrations to enhance surface quality of the workpiece and tool life during operation. In this technique, vibrations with small amplitude and frequency are applied to enhance cutting performance. In this study, a literature search was conducted for different low-frequency machining techniques. The advantages and disadvantages of the methods have been examined. A simulation case study of low-frequency vibration turning for Ti6Al4V alloy was presented the results of this analysis was discussed. The analysis was conducted using the Thirdway AdvantEdge program.

Keywords: Low-frequency vibration assisted machining, Thirdway AdvantEdge program, titanium alloy

1. INTRODUCTION

In the aerospace industry, there are several novel machining methods developed in recent years. One of the novel methods is vibration-assisted machining. This process is an intermittent cutting process which decreases effective stress and cutting energy with the help of vibration application (Astashev and Babitsky, 1998; Kumar et al., 2014; Thoe et al., 1998; Skelton, 1969). Continuous tool-workpiece contact is intermittent with repetitive penetration and withdrawal of the cutting tool. Therefore, the time spent at the actual cut-off time is less than all machining times. Cutting temperature is reduced due to the intermittent contact between the tool and the workpiece (Babitsky et al., 2013). However, in the vibration assisted machining method, the rate of chip removal at each vibration cycle is less than the conventional machining. Unlike static cutting forces in traditional machining, cutting forces vary widely because of the intermittent tool-workpiece contact (Ding et al., 2010). In general, these processes have resulted in reduced strength, low stress and temperature, better surface roughness and longer tool life (Patil et al., 2014).

Vibration has been applied to conventional machining processes (Kumar et al., 2014). High-frequency vibration (5 kHz to 40 kHz) is often used for different machining processes. Low-frequency vibration (values less than 1 kHz) is used in non-conventional machining methods, including hole drilling, electro-erosion, and electrochemical machining. Compared with high-frequency vibration assisted machining, low-frequency vibration assisted machining has improved process performance with lower system modifications and costs (Skelton, 1968; Skelton, 1969). In a low-frequency vibration assisted machining process, higher cutting depth operations can be performed than high-frequency vibration assisted machining process because the vibration amplitude, which causes a high dynamic load, is generally higher. Therefore, higher chip removal rates can be obtained with low-frequency vibration assisted machining applications.

Limited research has been conducted on the low-frequency vibration assisted machining process. Low-frequency vibration is first studied by R.C. Skelton which was applied with a frequency between 0 and 125 Hz on the lathe.
Low cutting forces-temperatures, tool wear, and good surface quality have been observed (Skelton, 1968; Skelton, 1969). Kim et al. (2009) used the vibrations of 300-500 Hz in the turning of aluminum and STAVAX. A smooth textured surface is formed with the help of the vibration. The low-frequency vibration was first used in the hole drilling process by Adachi et al. (2004). Chern and Chang (2006) used the frequency range (100 Hz-10 kHz). High dimensional accuracy-tool life and low surface quality were obtained. In low-frequency vibration assisted milling process, intermittent chips were obtained. Much of low-frequency vibration-assisted machining literature has examined the hole drilling of high-strength materials, composites, and polymers (Kumar et al., 2014; Adachi et al., 2004). The results show that the process can improve the manufacturing efficiency significantly. In this work, low vibration assisted machining process was examined, and the advantages-disadvantages of this process were discussed. A simulation case study of low-frequency vibration turning for Ti6Al4V alloy was presented. The results of this analysis was discussed. The analysis was conducted using Thirdway AdvantEdge program. In the second section, the details of the process are given. In the third section, a simulation study with the results is explained. Then, the conclusion section is given.

2. DETAILS OF THE LOW FREQUENCY VIBRATION ASSISTED PROCESS

In vibration assisted machining, small amplitude vibrations are applied to the cutting insert or workpiece to increase machining efficiency, and a lot of work has been performed on this process. Usually, traditional vibration-assisted machining can be applied in two ways: vibrations in the direction of cutting speed (direction of cutting speed) and vibrations in the part to be machined (direction of workpiece) (Brehl and Dow, 2008; Lauwers et al., 2014). To apply the intermittent cutting method with the application of vibration in the cutting speed direction, the cutting speeds have to satisfy the condition ω.A > V, where ω is the angular speed frequency, 2A is the amplitude between the peak points, and V is the cutting speed. For this reason, ultrasonic vibrations are used in such vibration assisted machining methods and are only applied to ultra-precision machining at low cutting speeds. However, an improved vibration-assisted machining process in the direction of the cutting speed known as elliptical vibratory machining has been developed and practically used. When vibration is used in the direction of the part to be machined (Mann et al., 2011; Moscoso et al., 2005), the surface roughness is decreased (Schubert et al., 2011), and intermittent chip formation is obtained (Guo et al., 2012; Okamura et al. 2006; Smith et al., 2009).

In conventional vibration assisted machining, vibrations are produced via piezoelectric or magneto-resistive actuators (Moriwaki and Shamoto, 1991; Shamoto and Moriwaki, 1999; Gou et al., 2012) or a linear motor (Okamura et al., 2006). In contrast to these methods, in the low-frequency vibration assisted machining method, the vibration is produced directly without using any device other than NC tool benches and a NC program.

In the experimental setup, a dynamometer was installed to measure the shear forces of the three components, and a radiation thermometer was installed on the cutting tool tip to measure the tool surface temperature. The NC controller is connected via a PMCIA-LAN card, and the position feedback data is measured with a servo tuning software to determine the correct position of the tool throughout the cutting operation. The improved process includes the real machining time and the non-machining time in each vibration; which brings intermittent machining to these lathe operations. It can be controlled easily by modifying the amount of feed, feed rate, back feed amount and back feed speed.

3. SIMULATION STUDY (A CASE STUDY)

To show low-vibration assisted machining process, a simulation study was conducted. The simulation study was conducted using the Thirdway AdvantEdge program. The image from the program is given in Fig.1. The study was carried out for Ti6Al4V. Dry cutting conditions were used in the study. 2-dimensional simulation studies in machining are preferred because of their simplicity compared to 3D simulation studies. The workpiece is modeled as plastic. The tool is rigidly modeled.
The Johnson-Cook material model was used during the simulation. The equations of this model are given below (Eqs. 1-2). The coefficients are shown in Table 1.

\[
\sigma = \left( A + B \varepsilon^m \right) \left[ 1 + C \ln \left( \frac{\varepsilon}{\varepsilon_0} \right) \right] \left( 1 - T^* \right)^n
\]

\[
T^* = \frac{T - T_{\text{room}}}{T_{\text{melt}} - T_{\text{room}}}
\]

\[ \varepsilon \] : Plastic strain rate / reference plastic strain rate

\[ \varepsilon_0 \] : Sensitivity of strain rate for the material

\[ T_{\text{room}} \] : Room temperature

\[ T_{\text{melt}} \] : Melting temperature of the material

\[ A, B, C, m \] : Material coefficients

**Table 1.** Johnson Cook coefficients of Ti6Al4V used in the simulations (Özel et al., 2010)

<table>
<thead>
<tr>
<th>Material</th>
<th>A (MPa)</th>
<th>B (MPa)</th>
<th>C</th>
<th>n</th>
<th>m</th>
<th>\varepsilon_0</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ti6Al4V</td>
<td>724.7</td>
<td>683.1</td>
<td>0.035</td>
<td>0.47</td>
<td>1</td>
<td>2000</td>
</tr>
</tbody>
</table>

The material of the cutting tool is WC. The rake angle is 0° and the relief angle is 0°. The hone radius is 0.02 mm. The friction model was used as Coulomb, and the value is 0.5. The cutting tool is vibrated in the feed direction. The frequency of the vibration is 13.875 Hz. The number of revolutions is 555 rpm. Workpiece diameter is 2.75 inches. The feed rate is 0.012 inch / rev. Table 2 gives the mechanical and thermal properties. Standard and chip breaking are selected in the simulation mode. The maximum number of nodes is set to 72000.
Table 2. Mechanical-thermal properties of Ti6Al4V material (Özel et al., 2010)

<table>
<thead>
<tr>
<th>Mechanical / Thermal properties</th>
<th>Ti6Al4V</th>
</tr>
</thead>
<tbody>
<tr>
<td>Elasticity modulus (MPa)</td>
<td>0.7412 +113375</td>
</tr>
<tr>
<td>Thermal expansion (mm.mm⁻¹°C⁻¹)</td>
<td>3×10⁻⁹T +7×10⁻⁶</td>
</tr>
<tr>
<td>Thermal conductivity (watt.m⁻¹°C⁻¹)</td>
<td>7.039×10⁰¹¹T</td>
</tr>
<tr>
<td>The coefficient of emissivity</td>
<td>0.7</td>
</tr>
<tr>
<td>Poisson ratio</td>
<td>0.31</td>
</tr>
</tbody>
</table>

Figures 2-3 show the cutting forces (Fx and Fy). The mean cutting forces of the low-vibration assisted turning were obtained as 60.3 and 21.1 N, respectively. In Figure 4, the maximum temperatures are given for the insert. Examination of the results shows that the maximum cutting tool temperature for the low-vibration assisted machining is close to 1000°C. Fig. 5 shows chip formation. It is observed that the formation of chips is continuous.

![Figure 2. Fx forces](image)

![Figure 3. Fy forces](image)

![Figure 4. The maximum cutting tool temperature](image)
4. CONCLUSIONS

In this study, a literature search was conducted for different low-frequency machining techniques. The advantages and disadvantages of the methods have been examined. In this technique, vibrations with small amplitude and frequency are applied to improve cutting performance. This process also helps to break the chip easily to prevent continuous chip during machining. In the future studies, different hybrid machining methods (low-frequency vibration assisted machining+hot machining or cryogenic machining) might be proposed.

REFERENCES


ROLLER HEMMING OPERATION ON A VEHICLE DOOR

Selim Gürgen

Eskişehir Vocational School, ESOGU, Eskişehir, 26110, Turkey
sgurgen@ogu.edu.tr

ABSTRACT

In the present study, roller hemming operation was investigated using a sample automobile door. Two hemming stages such as one pre-hemming and one final hemming were used to close the flange angle of 90º. Bending angles for each stage were varied and their influences were discussed in terms of deformation and hemming force. Furthermore, the effect of hemming line geometry was discussed considering a complex region on the product.

Keywords: Roller hemming, sheet metal, finite element method.

1. INTRODUCTION

In automotive industry, roller hemming operation predominates the sheet metal joining operations due to the advantages of flexible manufacturing. The process includes bending of outer panel onto inner panel and therefore, joining is completed mechanically. To perform this, a roller guided by a robotic arm passes over outer panel edges and bends the edges onto inner panel. In order to prevent excessive deformation on the panels, the process uses successive passes of roller and generally two or three stages are sufficient to complete the joining. Roller hemming is a good alternative to other joining operations such as welding, riveting and adhesive bonding. The process provides better visibility than welding and riveting while providing stronger joining in comparison to adhesive bonding. Doors, hoods, sunroofs, wheel arches and tailgates are the major areas for roller hemming applications [1].

Hemming on straight edge-flat surface components is performed easily however, in automotive applications, joining panels may have flat, concave or convex surfaces and hemming line may be straight, concave or convex. In most cases, complex geometries cause undesired deformations on the components. Wrinkling is common in convex edge components whereas splitting is seen in concave edge components due to the effects of tangential forces. In addition to these, misalignment in supporting results in recoiling and warping on hemming lines [2-5].

2. NUMERICAL MODELING

In the present work, the door geometry was supplied from LSTC. The door has a critical region for hemming operation as shown in Figure 1. This region has an acute radius curve form on the hemming line that may accommodate undesired deformations and large plastic strain. The process includes three main components on the door such as flange, outer panel and inner panel. Flange is the folded edge of outer panel and has a height of 10 mm in this study. Flange angle of 90º was closed two-stage hemming operations using three different bending angle pairs such as (30º/60º), (45º/45º) and (60º/30º). Thickness of the flange and inner panel was 0.7 and 1.0 mm respectively. The roller was 30 mm diameter cylinder that was modeled as rigid in the simulations. Anisotropic elastoplastic materials properties were supplied from LSTC library. To simplify the simulations, inner panel was modeled as rigid. In the numerical modeling, shell elements were used due to their time effective properties in comparison to solid elements. Mesh size reduced at the hemming line and adaptive remeshing option was enabled to enhance the accuracy. Explicit time integration with time step of \(-5\times10^{-6}\) s was used in the simulations.
3. RESULTS AND DISCUSSION

Figure 2 shows the maximum plastic strain on the hemming line after each operation. It is seen that maximum values are approximately 0.35 mm/mm after pre-hemming stages for each bending angle pair. However, it reaches the order of 0.85 mm/mm after final hemming stages due to excessive bending of flange. Despite very close results, bending angle pair of (30º/60º) yields relatively lower plastic strain among the studied bending angles. Comparing (45º/45º) and (60º/30º), it is possible to mention that strain formation is significantly identical according to the analyses. Based on the simulations, maximum deformation area is located at the acute radius form on the hemming line as shown in Figure 3. In comparison to straight edge regions, curve edges generate additional tensile forces in the direction of hemming line during the bending of flange and therefore, the material is subjected to excessive deformation which may lead to micro-cracks on the surface.
In terms of hemming forces, bending angle exhibits considerable influence in roller hemming operation. Figure 4a shows pre-hemming forces on the roller for each bending angle pair. Based on the graph, force curves exhibit a rapid peak at the beginning of the operation due to the first contact between the roller and flange. Force curves have an unstable profile for the sample geometry because the hemming line includes several curves. For example, the pre-hemming roller passes over the aforementioned acute radius form at the process completion rate of 40%. It is seen that hemming force drastically reduces at the top of the radius form, however, the roller requires higher forces during climbing. Comparing the bending angle pairs, hemming force increases as the bending angle increases. From the graph, maximum hemming force is approximately 900 N for the bending angle of 30° however, it is 1300 N the bending angle of 60°. Figure 4b shows final hemming forces on the roller for each bending angle pair. Clearly seen that hemming forces significantly increase in final hemming with respect to pre-hemming stage. To flatten out the wrinkles on the flange, higher forces are generated in final hemming. Similar to pre-hemming stage, hemming forces exhibit a drastic increase at the roller entrance. On the other hand, bending angle pair of (30°/60°) exhibits the highest force generation due to the largest angle to close the flange. Considering lower range of hemming forces both for pre-hemming and final hemming stages, (45°/45°) is the most preferable bending angle pair among the others.

![Figure 4. Hemming forces for (a) pre-hemming stages and (b) final hemming stages](image)

4. CONCLUSIONS

In this study, roller hemming operation was numerically conducted on a sample automobile door to investigate the influence of bending angle on the plastic deformation in hemming line. In the material selection, BH180 steel was preferred due to its extensive utilization in automotive industry. The results exhibited that high level plastic strains are located on the hemming line however, the material is capable to complete the process without any failure due to its good formability characteristics. On the other hand, strain evolution varies in a close range even though the bending angles in each stage change from 30° to 60°. However, force on the roller is directly related to bending angle because the force profile drastically increases as bending angle increases.

REFERENCES


DESIGN AND THERMAL DISTRIBUTION ANALYSIS OF 2 PHASE FLUID SYSTEM

Atakan Zeybek

Istanbul Technical University, Mechanical Engineering, Istanbul, Turkey
zeybeka@itu.edu.tr

ABSTRACT

Heating systems are getting more momentous due to being requirement, crowd, and their energy consumptions. With developments of design and manufacturing, producing heater tools are becoming easier. Although there are so many heating mechanisms, the radiator is the incomparable one of them because of price, ease of use, easy-to-produce. For the reason that the design of radiator can be changed in terms of dimensions, calculating thermal dispersion around this rigid body is one of the most significant problems for the radiator manufacturer. Even there are numerous studies for calculating the analytical value of heat conduction and convection problem, finite element analysis gives us favorable technic for that calculation. In this paper, it is presented that thermal, pressure, velocity distribution around the radiator - 2 phase system- using ANSYS Fluent 16.0.

Keywords: CFD, Heat Transfer, Radiator, Thermal Distribution

1. INTRODUCTION

There are numerous amounts of radiator designs in terms of view, efficiency and energy requirement; furthermore, various manufacturers are trying to produce more and develop new models which will be efficient and have a delightful view. Computational Fluid Dynamics (CFD) simulation software is getting the more significant effect on analyzing thermal engineering problems. As an example of it, heat exchanger simulations can be solved with using CFD simulations precisely [1]. Another thermal simulation instances can be count as heat transfer of solar collector [2], electronic equipment [3] and gas turbines [4]. ANSYS Fluent, which was used in this study, gives the user a chance that enables them to analyze more precise. In this study, regular radiator design was created and will be explained. Subsequently, thermal distribution of this radiator will be demonstrated in the regular box-shaped room with using ANSYS Fluent. Using many simplifications, the aim in this study focused on the distributions of physical values mentioned; therefore, every point or region in the simulation volume have been tried to be presented.

2. MODELING

To make model easy to export and to use with other package programs, SpaceClaim was used. Firstly, the average rough dimensions of radiators which are used in current systems have been taken as 1200x600x60 mm to design the radiator in the limits of suitable for 37.5m³ (dimensions of 3x5x2.5m) room. While designing radiator, it was assumed that the radiator is made up of sheet metal, so the hollow body designing method has been used to create a (Computer Aided Design) CAD model of the radiator (figure 1). The design of our model cannot be used as an exact model because some features like input/output ports and inner water channels do not exist on it. On the other hand, our model provides the specifications for being an appropriate model which are mass, surface area and dimensions. These physical quantities define whole thermal analyze parameters which are inevitable for thermal simulations. The orientation of the radiator in the room can be seen in figure 2. The place of radiator has been chosen in the middle of the biggest wall and 30 cm above the ground in the room because the results would be symmetric in terms of temperature, velocity and pressure distribution.
3. ANALYSIS

3.1 Boundary Conditions
The radiator was though as a heat source which has a constant temperature so, there should be a value of it. For regular conditions, it has been assumed a radiator has 50 °C outer surface temperature. Because of this assumption, only the convection effect between the outer surface and the room air will be taken into account; therefore, the convection and conduction heat transfer mechanisms inside the radiator will be ignored. Because outer temperature have been utilized, ignoring the heat transfers into the coil would not be wrong. Even there are radiation effect on the outer surface of the radiator, it was assumed that the surface brightness and roughness were less enough to left this effect out of account. The figure 3 explains the 3 heat transfer mechanisms (which are convection between room air and wall, conduction through the wall and convection between the wall and outer atmosphere) around the wall.

For heat transfers with convection, below equation is used [4]:

\[ q_k = -kA \frac{dT}{dx} \]  

(1)

Also heat transfers with conduction, below equation is used [4]:

\[ q_c = \bar{h}A(T_s - T_{f,\infty}) \]  

(2)

3.2 Mesh
Creating mesh is one of the significant factors that improve the precision of CFD simulations especially in fluid mechanics. In this system, the mesh has been created with the contact region sizing so, the smallest mesh is on the radiator surface and it is getting large when it reaches the outer surface of air volume.
4. SIMPLIFICATIONS

4.1 Boundary Condition Simplifications

Because the radiator releases the heat through the water flows, the temperature of the surface of radiator also decreases in that way [5]. Even there is that kind of temperature effect, in this study, the temperature of the radiator has been taken a constant value which has 50°C of temperature; therefore, this simplification has been done in order to simplify the analysis and running time.

4.2 Design Simplifications

Even traditional radiator has so many design specifications, the simplified model had to use for computational calculations. If the complex geometry is used, not only would the amount of mesh increase, but also the calculation time would last extraordinarily long. To make design simple, two vertical flat plate and some fin-like pockets have been created.

4.3 Calculation Simplifications

Firstly, Although convective heat transfer coefficient (h) of around radiator outer surface depends on plenty amount of design features such as the geometry of radiator and the volume of the room, vertical flat plate approach has been utilized in order to calculate this value. To find this, the equations were sequentially calculated [6, 7, 8]:

\[
Gr = \frac{g\beta(\Delta T L^3)}{\nu^2} \tag{3}
\]

\[
Ra = GrPr = \frac{g\beta(T_s - T_\infty)x^3}{\nu u} \tag{4}
\]

\[
\overline{Nu} = 0.825 + \left\{ \frac{0.387RaL^{1/6}}{1+0.492(Pr)^{1/6}} \right\}^2 \tag{5}
\]

Average convective heat transfer coefficient was calculated from Nusselt number [9]:

\[
\overline{Nu} = \frac{hL}{k} \tag{6}
\]
5. RESULTS

After running the simulation in the CFD, the results could be observed for 2500 seconds. For thermal distribution of 2 plane view can be seen in figure 5.

![Figure 5. 2 view temperature distribution](image)

It was obvious that the temperature of the room air is high around the radiator, and it is decreasing when becoming distant. Moreover, the temperature boundary layer can be seen in the magnified view around the radiator (figure 6).

![Figure 6. Isothermal curves around the radiator](image)

In the same way, pressure distribution can be seen in the figure 7.

![Figure 7. Pressure distribution](image)

The velocity of air inside has been demonstrated in figure 8 below,
Because of the variable air density, this kind of a velocity distribution was predictable. The maximum speed value of one dimensionless particle in the air is 0.383 m/s. Apart from the velocity vector graphics, the velocity through the middle axis of the side view radiator has been plotted (figure 9). This graphic indicates the velocity distribution of simulation volume from floor to ceiling.

![Figure 8. Velocity vectors](image1.png)

6. CONCLUSION AND RECOMMENDATIONS

The CFD simulation have been run for determining the temperature, velocity and pressure distribution of radiator in the room. All results have been indicated in the results section. According to results, the maximum temperature was equal to radiator surface temperature and it decreases slightly through the temperature boundary layer. After boundary layer temperature drops significantly. The maximum velocity was 0.383 m/s and the graphic represents velocity distribution. In the pressure distribution results, it can be seen that the upper part of the radiator inside has been focused in terms of pressure. This effect can be explained by transitive fluid flow by heat transfer. For next researches, boundary conditions can be enhanced and advanced model can be used for better results.

REFERENCES

A SINGLE DEGREE OF FREEDOM ROBOT ARM FOR HUMAN-ROBOT INTERACTION

Özge Beyza Albayrak¹, Salar Rahimi², Kutluk Bilge Arıkan³

¹TED University, Department of Mechanical Engineering, Kültür Mah. Ziya Gökalp Cd. No:48 Ankara, TURKEY
²METU, Department of Mechanical Engineering, Üniversiteler Mahallesi, Ankara/TURKEY
³obeyza.albayrak@tedu.edu.tr, ²salar.rahimi1995@gmail.com, ³kutluk.arikan@tedu.edu.tr

ABSTRACT

This article presents the 1 degree of freedom (dof) robot arm in the form of a slider-crank mechanism to be used for the purpose of target tracking. By using a linear spring, the slider of the robot arm can be coupled to another slider which is guided by the human subject who is tracking the same target. Such an interaction may help the human subject to predict the motion of the target [15]. The mathematical modeling of the dynamical system, controller design, simulations, and the physical system are briefly presented. The designed position controller for the slider is implemented on the physical system, as well. The real performance is compared with that of the simulations. The purpose of this setup is to analyze the human’s motor performance when it interacts with a robot arm. The motivation is to improve the setup to be used for the patients with movement disorders [2].

Keywords: Robot arm, Slider-crank, Movement disorder, Target tracking, Prediction.

1. INTRODUCTION

Human-robot interaction has been a challenging interest for researchers from different disciplines for various purposes. Some models suggest that haptic interaction enables subjects to improve their motor performance by making estimations about the partner’s movement, since it gives proprioceptive clues to the subjects [15]. Considering the target tracking of the human using the arm and eyes, haptic interaction between the human subject and the robot is assumed to be providing additional information to the human subject to make predictions about the future motion of the target [5]. One of admissible candidates in the brain is the cerebellum to perform the prediction. Cerebellum takes role to coordinate the movement by supporting motor functions [5]. It is the center that performs the adaptive motor control and cerebellum has a significant role in coordinating and learning the motion of the target trajectory. Making a prediction about the next target movement is the keyword for the human subject, because improved motor coordination depends on predictive information about movement [4].

Damage to cerebellum impairs the ability to control the movement. As a consequence, some functional diseases appear in the brain, resulting movement disorders. Cerebellum dysfunction (CD) and Parkinson’s disease (PD) are two of them. These diseases are very critical and serious disorders. In a study, authors performed a target tracking task for both patients with movement disorders and for healthy people [2]. They developed a compensation method to improve the performance of hand movements of patients using an additional equipment, a robot arm, attached to human arm. They observed that the additional force assists the patient’s arm and compensates the error by improving the control of hand movement. Therapy assisting devices are used to train patient’s wrists and forearms during training programs by using a robot arm equipment in order to improve PD patient’s motor performance [10]. The robot arm provides an external proprioceptive clue, a haptic interaction provided by proprioception [7], to the patient’s arm. These studies reflect our desire to test our setup on patients with PD, when there is a haptic connection between the patient and the position controlled robot arm.

Our aim is to investigate subjects’ motor performances during a target tracking task, when they are interacted with a robot arm. Thus, the purpose of the setup is to perform a target tracking task when there is a haptic interaction, a physical connection between the subject and the robot arm. This connection is created by a spring which acts as an assisting force to the subject during the task and helps he/she to predict the movement of the target. The important point is, the physical connection does not provide a direct information about the target’s position, it only helps the subject to make estimations. Parents helping their children to walk for the first time provides a physical assistance to the child, since
there is an information exchange between the parent and the child [15]. The physical interaction between the baby and the parent is called haptic interaction and it helps, assists the baby to improve his or her motor performance. “What” kind of information is exchanged between partners and “how” this information affects the coordination and the movement of one’s remains unknown [12]. Based on these theories, our biggest motivation is to carry out some tasks when there is a connection, an information exchange, between the subject and the robot arm and observe the cerebellum activation, also to analyze the improvement of the motor performance of patients.

In this paper, we briefly present the experimental setup, mathematical modeling and controller design implementations. This is the starting phase of our research which aims to study the contribution of the cerebellum to the target tracking of subjects with haptic interaction. In this setup both the human and robot arm are tracking targets that make exactly the same motions. Slider mass (point C) is tracking the target on the front by the closed loop control system using the robot arm and human operator is tracking another target on the front that is exactly making the same motion as the one tracked by the slider mass.

2. EXPERIMENTAL SETUP

Slider-crank mechanism converts the rotational motion into translational motion [11]. It is used as a single dof robot arm that mimics the motion of the human arm.

The slider crank mechanism shown in the Figure 1 is the representative of 1 degree of freedom robot arm and by using the joint angles $\theta_2$ and $\theta_3$, the desired displacement $x_c$ can be determined. Figure 2 shows the physical system which is developed in ME 336 Modeling and Control of Dynamical Systems course as a course project. Figure 3 presents the setup that uses 2 manipulandum type robots to implement the human-human and human-robot type haptic interactions during target tracking [15]. The interaction is implemented by the virtual spring and damper components. In this paper, we propose a simplified setup to implement interaction while tracking a 1 dof target. We use a physical spring to the implement interaction.

![Figure 1. Slider-crank mechanism as a single dof robot arm](image)

![Figure 2. Our slider-crank mechanism](image)
Length of the links and the geometric parameters of the physical system are listed in Table 1.

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Values (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>$a_2$</td>
<td>0.20</td>
</tr>
<tr>
<td>$a_3$</td>
<td></td>
</tr>
<tr>
<td>$d_1$</td>
<td>0.25</td>
</tr>
<tr>
<td>$d_2$</td>
<td>0.25</td>
</tr>
</tbody>
</table>

The slider mass displacement can be determined from the kinematic calculations of the system. Both the inverse and forward kinematics of the slider crank mechanism are obtained using the Loop Closure Equations.

### 2.2. Loop Closure Equation Method

The loop closure equation below is used to derive the forward and backward kinematics of the slider crank mechanism.

$$d_1 + a_2 e^{i\theta_2} + a_3 e^{i\theta_3} = d_2 l + x_c$$ \hspace{1cm} (1)

Inverse kinematics gives the joint angles $\theta_2$ and $\theta_3$ with respect to the desired displacement of the slider mass, $x_c$, using the known parameters of the mechanism $d_1, d_2, a_2, a_3$. Therefore, to determine the inverse kinematic equations, $\theta_2$ and $\theta_3$ values should be eliminated separately, by writing down Equation 1 in two different forms. Multiplying the two different form of loop closure equation by their conjugate provides two expressions in terms of $\theta_2$ and $\theta_3$ individually. Rewriting the loop closure equation in different forms:

$$a_3 e^{i\theta_3} = d_2 l + x_c - a_2 e^{i\theta_2} - d_1$$ \hspace{1cm} (2)

$$a_2 e^{i\theta_2} = d_2 l + x_c - a_3 e^{i\theta_3} - d_1$$ \hspace{1cm} (3)

When Equation 2 is multiplied by its conjugate, $\theta_3$ will be eliminated and the obtained equation only contains $\theta_2$ as a joint angle. Multiplying Equation 3 by its conjugate gives the opposite. Rearranged versions of Equation 2 and 3 can be determined as:

$$K_1 + K_2 \cos \theta_2 + K_3 \sin \theta_2 = 0$$ \hspace{1cm} (4)

$$K_4 + K_5 \cos \theta_3 + K_6 \sin \theta_3 = 0$$ \hspace{1cm} (5)

$K_1, \ldots, K_6$ are constants. In order to solve these equations following transformations are performed.

$$\cos \theta = \frac{1 - \tan^2 \left( \frac{\theta}{2} \right)}{1 + \tan^2 \left( \frac{\theta}{2} \right)} \hspace{1cm} \sin \theta = \frac{2 \tan \left( \frac{\theta}{2} \right)}{1 + \tan^2 \left( \frac{\theta}{2} \right)}$$ \hspace{1cm} (6)

If we assume that $x = \tan \left( \frac{\theta}{2} \right)$, Equation 6 can be written in terms of $x$ to form a second order polynomial. Therefore, the expression becomes:
The obtained values for $\theta_2$ form the reference input for the closed loop controller of the 1 dof robot arm.

The forward kinematics of the mechanism is also considered in the same way that we solve inverse kinematics for our system. The logic behind direct kinematics is to obtain the displacement of the slider mass, $x_c$ and $\theta_3$ as the outputs due to the given input angle, $\theta_2$.

### 2.3. Equation of Motion

The kinetic equations of the system are obtained by dividing the system by drawing the free body diagrams (FBD) for the links of the slider crank and the slider mass separately. The forces acting on the link AB is shown in Figure 4. The link inertias and the accelerations are small and quasi-static analysis is performed for the links. Whereas, a dynamic equation is derived for the slider mass that has higher mass.

![Free Body Diagram of Link AB](image)

**Figure 4.** Free Body Diagram of Link AB

Considering the forces acting on the horizontal and vertical directions and the moment, the conditions for equilibrium are simply stated as:

\[
\begin{align*}
\sum F_x &= 0 \quad F_2 + F_4 = 0 \quad (9) \\
\sum F_y &= 0 \quad F_1 + F_3 = 0 \quad (10) \\
\sum M_G &= 0 \quad -F_1 \frac{a_2}{2} \cos \theta_2 + F_2 \frac{a_2}{2} \sin \theta_2 + F_3 \frac{a_2}{2} \cos \theta_2 - F_4 \frac{a_2}{2} \sin \theta_2 = -T_2 \quad (11)
\end{align*}
\]

The free body diagram for the second link BC is shown in Figure 5. The static equilibrium of the part for both the forces and the moment is also stated is the following equations.

\[
\begin{align*}
\sum F_x &= 0 \quad -F_3 + F_6 = 0 \quad (12) \\
\sum F_y &= 0 \quad -F_4 + F_5 = 0 \quad (13) \\
\sum M_G &= 0 \quad -F_3 \frac{a_2}{2} \cos \theta_3 - F_4 \frac{a_2}{2} \sin \theta_3 - F_5 \frac{a_2}{2} \sin \theta_2 - F_6 \frac{a_2}{2} \cos \theta_3 = 0 \quad (14)
\end{align*}
\]
The third divided part of the mechanism is the slider mass and the free body diagram of the third part shown in Figure 6 is pretty different comparing to other two parts introduced above.

Forces acting on the slider mass can be determined as:

\[
\sum F_x = ma_x \\
\sum F_y = 0
\]

\[
-F_6 - F_5 = m_4 \ddot{x}_c - F_e \\
-F_6 + F_7 = 0, \quad T_4 = 0
\]

The force \( F_e \) represents the damping and defined as "\( c \dot{x}_c \)." Additionally, \( F_e \) represents the interaction force coming due to the spring. \( m_4 \) is considered as the mass of the slider. Therefore, its weight is considerably small, since we do not want our system to have a hard time on moving the slider mass while tracking the moving object.

Finally, we get 9 equations and 9 unknowns, \( \{ F_1, F_2, F_3, F_4, F_5, F_6, F_7, T_4, \ddot{x}_c \} \) from the kinetics of the system. Solving the equations simultaneously, acceleration, \( \ddot{x}_c \) is obtained as follows.

\[
\ddot{x}_c = \frac{F_e - c \dot{x}_c}{m_4} - \frac{T_4 \cos \theta_3}{a_2 m_4 \sin(\theta_2 + \theta_3)}
\]

Damping constant is expressed as \( c \), the torque applied by the motor is \( T_2 \).

3. MOTOR MODELLING

In this study, the slider-crank mechanism is driven by a Maxon DC EC-max motor in order to rotate the link AB. The parameters of the DC motor are shown in Table 2.
The back emf voltage is $v_b(t)$ is written as follows. $K_b$ is the back emf constant.

$$v_b(t) = K_b \frac{d \theta_m(t)}{dt} \quad (18)$$

From the Kirchhoff's voltage law, the following equation can be written.

$$V_b(s) + LsI(s) + RI(s) = E(s) \quad (19)$$

Motor torque, $T_m$ is proportional to the armature current $i(t)$ and $K_t$ is the motor torque constant. Thus, Laplace transform of motor torque is,

$$T_m(s) = K_t I(s) \quad (20)$$

The inductance value $L$ is a very small w.r.t. the internal resistance $R$. Thus, it can be considered as zero. According to Equation 20, current $I(s) = T_m(s)/K_t$. Furthermore, substituting Equation 18 and 20 into Equation 19 can provide a different form of the loop equation as follows:

$$K_b s \theta_m(s) + R \frac{T_m(s)}{K_t} = E(s) \quad (21)$$

The inertia of the motor $J_m$ and the damping is represented by $D_m$. The rotational dynamics for the motor is given as below.

$$T_m(s) = (J_m s^2 + D_m s) \theta_m(s) \quad (22)$$

Substituting Equation 22 into Equation 21 to obtain a hybrid expression, we get:

$$K_b s \theta_m(s) + R \frac{(J_m s^2 + D_m s) \theta_m(s)}{K_t} = E(s) \quad (23)$$

Transfer function representation for the voltage input, $V$ and $\theta_2$ can be determined as:

$$\frac{\theta_2}{V} = \frac{K_t n/R}{s^2 + K_b K_t n^2 / J s} \quad (24)$$

There is a disturbance torque added to the system as well and it is expressed as $T_d$ in the mathematical modeling. In simulations, the form of the disturbance torque, $T_d$ is step input.

The transfer function between disturbance torque, $T_d$ and $\theta_2$ can be expressed as:

$$\frac{\theta_2}{T_d} = \frac{1}{J s^2} \quad (25)$$
4. CONTROLLER DESIGN

PID controller is designed to control the position of the link AB and the structure of the overall model is shown in Figure 7.

![Figure 7. The Proposed Model](image)

The controller gains are obtained using a linearized model. Designed control system is implemented on both the linearized and nonlinear models. The design specifications are the percent overshoot, $\%OS = 2\%$ and the settling time $T_s = 1$ sec. Controller gains are presented in Table 3.

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Gains</th>
</tr>
</thead>
<tbody>
<tr>
<td>$K_p$</td>
<td>21.73</td>
</tr>
<tr>
<td>$K_i$</td>
<td>62.73</td>
</tr>
<tr>
<td>$K_d$</td>
<td>2.98</td>
</tr>
</tbody>
</table>

5. RESULTS & DISCUSSION

5.1. Simulation Results

Figure 8 shows the responses of the linear model due to different reference inputs and a disturbance torque added to the system as a step function. Figure 8(a) and Figure 8(c) stands for the displacement of the slider mass, Figure 8(b) and (d) are the plots for the rotation of the crank shaft. Figure 8(a) and (b) are the responses of a sine wave with an amplitude of 0.125 and frequency at 1.25 Hz.

![Figure 8. Linear Model Simulation Results](image)
Figure 9 shows the simulations responses of the nonlinear system. The input signal is a sine wave with a 0.125 initial value. The amplitude of the sine wave is 0.05 and the frequency is set to 0.62 Hz.

![Figure 9. Nonlinear Model Simulation Responses](image)

Figure 10 shows the nonlinear system performance in case of a complex target motion.

![Figure 10. Nonlinear Model Simulation Results obtained by Multiple Sine Inputs](image)

5.2. Physical Implementation

In the physical implementation, the slider crank mechanism is driven by the dc motor and the moving target is excited by the stepper motor. The National Instruments (NI) data acquisition board is used to implement the real time control with the Simulink Realtime Desktop software. The proximity sensor is used to measure the position of the slider mass. The position of the target for the robot arm is assumed to be measured by the inputs to the stepper motor.

The experimental result in the case of the sine wave reference position for the slider mass, i.e. harmonically moving target, is shown in Figure 11. The blue line is the desired position and the red line is the actual position of the slider mass.

![Figure 11. Reference position and the actual position of the slider mass, point C](image)
CONCLUSIONS & FUTURE SCOPE

In this paper, we briefly present our human-robot interaction setup. Both the human and the robot arm are tracking the 1 dof target with or without a physical coupling. The performances can be evaluated using the mathematical measures based on the sensor data. The initial phase of the system is developed in the ME 336 Modeling and Control of Dynamical Systems course as a course project. The linear and nonlinear models are derived and PID controller is designed and implemented on these models. In the physical implementations, Simulink Realtime Desktop is utilized. The performance of the designed controller is compared with that of the simulations. The deviations of the actual response from the simulations is mainly due to the parametric uncertainties such as the friction. After improving the performance of the control system, physical tests with the interactions will be done. Based on the acquired human and robot data, we will focus on the prediction models to express the haptic interaction and to assess the role of cerebellum.

REFERENCES


MACROSCOPIC TESTING AND NANOINDENTATION OF GRAPHENE-EPOXY NANOCOMPOSITES

Özgen Çolak¹ and Deniz Uzunsoy²

¹Yıldız Technical University, Department of Mechanical Engineering, Beşiktaş, İstanbul, Turkey
²Bursa Technical University, Department of Metallurgy & Materials Engineering, Bursa, Turkey
¹ozgen@yildiz.edu.tr and ²deniz.uzunsoy@btu.edu.tr

ABSTRACT
The purpose of this work is to investigate the mechanical properties of graphene-epoxy nanocomposites at nano and macro scale. The determination of localized mechanical properties of graphene-polymer nanocomposites plays a great role in studying internal properties and failure mechanisms in these materials. Nanoindentation technique is one of the increasingly used methods for characterization of local properties and mechanical behavior of a wide variety of materials. Elasticity modulus and hardness in addition to plastic or viscoplastic properties can be estimated by this technique.

In this work, using the electric arc discharge method, graphene platelets (GPL) are obtained from graphite. Dispersion of GPL in epoxy matrix is done with sonication. Following the manufacturing of graphene-epoxy nanocomposites, nanoindentation experiments (loading, unloading and cyclic loading) are performed at load and displacement controlled modes. Mechanical properties such as hardness and elasticity modulus are directly obtained from load-displacement curves. In addition to nanoscale experiments, the macroscopic behaviors of graphene-epoxy nanocomposites are investigated using uniaxial tensile testing and hardness measurements. The mechanical properties at different scales are compared.

Keywords: graphene-epoxy nanocomposite, nanoindentation, manufacturing, mechanical properties

1. INTRODUCTION
Nanocomposites are composite materials in which the matrix material is reinforced by one or more separate nanomaterials in order to improve performance properties. The most common materials used as matrix in nanocomposites are polymers (e.g. epoxy, nylon, polyepoxide, polyetherimide), ceramics (e.g. alumina, glass, porcelain), and metals (e.g. iron, titanium, magnesium). Nanomaterials are generally considered as the materials that have a characteristic dimension (e.g. grain size, diameter of cylindrical cross-section, layer thickness) smaller than 100 nm.

Graphene is considered a two-dimensional carbon nanofiller with a one-atom-thick planar sheet of $sp^2$ bonded carbon atoms that are densely packed in a honeycomb crystal lattice. Graphene is predicted to have remarkable properties, such as high thermal conductivity, superior mechanical properties and excellent electronic transport properties.

Epoxy materials infused with graphene exhibited better performance. In fact, adding graphene equal to 0.1% of the weight of the composite boosted the strength and the stiffness of the material to the same degree as adding CNTs equal to 1% of the weight of the composite. This gain, on the measure of one order of magnitude, highlights the promise of graphene.

In this work, the mechanical properties of graphene-epoxy nanocomposite are investigated using nanoindentation and macroscopic testing. The properties which are obtained at different scale are compared.

2. MANUFACTURING OF GRAPHENE-EPOXY NANOCOMPOSITE

2.1 Dispersion of GPL in Epoxy Matrix:
There are different methods for dispersion of graphene platelets (GPL) in epoxy matrix. However, the one used in this work is as follows: GPL was dispersed in acetone using sonicator and epoxy resin was added and sonicated for 1.5 h. Acetone is used as solvent and needs to be removed then. Therefore, the mixture is heated in a magnetic stir plate using a Teflon coated magnetic bar and acetone is evaporated. To make sure all acetone is removed, the mixture is placed in a vacuum chamber for 12 h. Then curing agent (2120 Epoxy hardener) was added and mixed using shear mixer. Again in vacuum chamber, the mixture is degassed. Since the specimen are used for nanoindentation, a good surface profile is needed. Therefore, in clean room, silicon wafer is used. Using spin coating, graphene-epoxy nanocomposite film is created on silicon wafer. Then, the nanocomposite is cured at room temperature. Schematic showing the dispersion of graphene sheets in the epoxy matrix \textit{via} solution mixing with high amplitude ultrasonic agitation is given in Figure 1.

3. NANOINDENTATION

There are different ways to experimentally characterize nanocomposites. For example, tensile and flexural tests (mostly conducted on Instron or MTS machines), impact tests (conducted on pendulum impact testing machine), and micro-compression tests. Nanoindentation test is one of the most effective and widely used methods to measure the mechanical properties of materials. This technique uses the same principle as microindentation, but with much smaller probe and loads, so as to produce indentations from less than a hundred nanometers to a few micrometers in size.

Hardness (H) and elastic modulus (E) are calculated from the load-displacement curve obtained from a nanoindentation test. As the indenter penetrates into the specimen, the loading curve climbs up. At some point, the maximum load $P_m$ is reached, and then followed by the unloading. If the material is perfectly elastic and has no hysteresis, the loading curve and the unloading curve will be identical. $h_m$ gives a measure of the total maximum deformation, while $h_f$ represents the maximum permanent (plastic) deformation (final penetration depth).

3.1. The Procedure to Calculate the Hardness and Reduced Modulus

Unloading curve is used to calculate elasticity modulus of the material. It is well approximated by power law relation given below.

$$P = \alpha (h - h_f)^m$$  \hspace{1cm} (1)

The derivative of the power law relation with respect to $h$ is evaluated at the maximum load to calculate the contact stiffness, $S$.

$$S = \frac{dP}{dh}$$  \hspace{1cm} (2)

The contact depth, $h_c$, is calculated as
To account for edge effects, the deflection of the surface at the contact perimeter is estimated by taking the geometric constant $\varepsilon$ as 0.75. The Hardness is defined as

$$H = \frac{P_{\text{max}}}{A}$$

where $P_{\text{max}}$ is the maximum indentation force and $A$ is the resultant projected contact area at that load. The Reduced Modulus ($E_r$) is calculated with

$$E_r = \frac{1}{E} = \left(1 - \nu^2\right)_{\text{sample}} + \left(1 - \nu^2\right)_{\text{indenter}}$$

For a standard diamond indenter probe, $E_{\text{indenter}}$ is 1140 GPa and $\nu_{\text{indenter}}$ is 0.07. Poisson’s ratio varies between 0 and 0.5 for most materials.

Reduced modulus is defined as also,

$$E_r = \frac{S \sqrt{\pi}}{2 \sqrt{A}}$$

Rearranging this equation yields

$$A = \frac{\pi}{4} \left( \frac{S}{E_r} \right)^2$$

To determine the area function, a series of indents at various contact depths are performed in a sample of known elastic modulus and contact area $A$ is calculated using previous equation. A plot of calculated area as a function of contact depth is created and six order polynomial is found,

$$A = C_0 h_c^6 + C_1 h_c^5 + C_2 h_c^4 + C_3 h_c^3 + C_4 h_c^2 + C_5 h_c$$

Indentation is a highly nonlinear problem. It involves large plastic deformation, material nonlinearity, and contact. In order to better understand and characterize the mechanical properties and to provide guidelines for proper design of experiments, finite element method is often used to simulate the nanoindentation tests. The primary mechanical properties extracted from a nanoindentation test are the hardness and the elastic modulus.

4. EXPERIMENTAL RESULTS

Displacement controlled nanoindentation experiments are performed at different location to investigate heterogeneity of the material. Four results are given in Figure 2. Due to heterogeneity of the material discrepancy is observed.

![Figure 2](image_url)

**Figure 2.** Displacement controlled nanoindentation at different locations at the same conditions. (Graphene content is 0.1 wt%)

In nanoindentation, depth has effect on material response. To investigate change in material properties with changing depth, load controlled experiments are performed at different load levels and result are plotted in Figure 3.
Cyclic behavior of graphene-epoxy nanocomposite under load controlled loading is investigated and ratcheting which is strain accumulation is observed due to load controlled loading with nonzero mean force, Figure 4.

Figure 4. Load controlled cyclic loading. Ratcheting (strain accumulation) is observed like many other materials.

Using the load-displacement curves, elasticity modulus and hardness of the material is calculated and given in Table 1.

Table 1. Elasticity modulus and hardness obtained from load controlled nanoindentation

<table>
<thead>
<tr>
<th>Load (µN)</th>
<th>Loading rate (µN/s)</th>
<th>Er (GPa)</th>
<th>E (GPa)</th>
<th>Hardness (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>2000</td>
<td>20</td>
<td>1.47</td>
<td>1.34</td>
<td>73</td>
</tr>
<tr>
<td>2000</td>
<td>200</td>
<td>2.85</td>
<td>2.60</td>
<td>170</td>
</tr>
<tr>
<td>4000</td>
<td>200</td>
<td>2.36</td>
<td>2.15</td>
<td>136</td>
</tr>
<tr>
<td>6000</td>
<td>200</td>
<td>1.72</td>
<td>1.56</td>
<td>114</td>
</tr>
<tr>
<td>8000</td>
<td>20</td>
<td>1.38</td>
<td>1.25</td>
<td>90</td>
</tr>
<tr>
<td>8000</td>
<td>200</td>
<td>1.43</td>
<td>1.30</td>
<td>110</td>
</tr>
</tbody>
</table>

Table 2. Graphene epoxy nanocomposite nanoindentation experiments (Displacement controlled)

<table>
<thead>
<tr>
<th>Max. Displacement (nm)</th>
<th>Displacement rate (nm/s)</th>
<th>Er (GPa)</th>
<th>E (GPa)</th>
<th>Hardness (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1000</td>
<td>200</td>
<td>3.70</td>
<td>3.37</td>
<td>230</td>
</tr>
<tr>
<td>1000</td>
<td>20</td>
<td>3.01</td>
<td>2.74</td>
<td>173</td>
</tr>
</tbody>
</table>
5. MACROSCOPIC TESTING

Uniaxial tensile tests were performed using dumbbell-shaped samples according to ASTM D638-14, through the universal mechanical testing instrument (Shimadzu AGS-X) with 10 KN maximum load capacity and 2 mm/min crosshead speed. At least three samples from each group were tested and their averages were taken. Stress-strain behavior of graphene-epoxy nanocomposite and elasticity modulus of GNP/epoxy for two different weight fraction are given in Figure 5. Elasticity modulus of graphene epoxy nanocomposite is calculated from initial slope of stress-strain curve and it is around 2.41 GPA for 0.1 wt% graphene.

![Graphene-Epoxy Tensile Test](image)

**Figure 5.** Stress-strain behavior and Young’s modulus of GNP/epoxy for two different weight fraction obtained from tensile test.

On the other hand, the Vickers hardness of graphene-epoxy nanocomposite is measured as 16 HV (157 MPa).

6. CONCLUSIONS

Nano indentation technique is used for characterization the local properties of graphene-epoxy nanocomposites. In addition, the macroscopic behaviors of graphene-epoxy nanocomposites are investigated using uniaxial tensile testing and hardness measurements. The mechanical properties such as elasticity modulus and hardness are determined at different scales and compared.

It is observed that increase in load level yields a decrease in modulus and hardness obtained from nanoindentation due to a combined surface and tip effect. Hardness levels off when the indentation depth is increased. Increasing loading rate results an increase in E and hardness. Resistant to deformation increases at high loading rate since there is no time for relaxation. Comparing the elasticity modulus obtained from tensile test and the nanoindentation reveals that elasticity modula obtained from different scaled experiments are close to each other even though the strain rates are not the same. Since the material is rate dependent, loading rate has an effect on the behavior. Vickers hardness is obtained as 157 MPa, on the other hand, the harness measured from nanoindentation depends on the loading rate, control type and depth. It is 73-230 MPa range.

REFERENCES


MECHANICAL PROPERTIES OF FLOWFORMED AISI 1.4330 GUN BARREL

Umut K. Koltan and Özgen Ü. Çolak
Yıldız Technical University,
Department of Mechanical Engineering, Beşiktaş, İstanbul
1umutkoltan@gmail.com, 2ozgen@yildiz.edu.tr

ABSTRACT
Flowforming which is an advanced cold forming technology is used for producing high precision thin and thick walled symmetric components with single and multiple internal and external contours on work pieces. Advantages of this technique are: work hardening effect, increased tensile strength, grain orientation in the direction of deformation, dimensional accuracy and high surface quality. Flowforming technology has been used in defense and aerospace industries for years.

In this work, the grooved gun barrel is manufactured by flowforming method. The mechanical properties of as-received AISI 1.4330 steel and flowformed gun barrel made from AISI 1.4330 steel are compared. To observe the changes in the properties, uniaxial tensile are performed for undeformed and flowformed AISI 1.4330. In addition, hardness and internal structure of both specimens are investigated using Vickers hardness test and optical microscopy respectively. The improvements in the mechanical properties are observed.

Keywords: flowforming, gun barrel, mechanical properties.

1. INTRODUCTION
Flow form is a cold forming method, also known as sputtering. In this method, the axial and radial forces are applied to the work piece on the circular section mold which rotates around its axis. As a results of these forces, work piece undergoes a plastic deformation, thinning in the wall thickness and elongation are observed. During this shape change, the hardening occurs and the tensile strength and hardness values of the material increase significantly.

There are number of advantages of the flow form process. It enables the production of circular parts with high precision and high surface quality. It increases the life of the parts under torsional stress due to the preload applied to the parts in the torsional direction. It is possible to produce circular parts with different diameters in one piece. Also, it allows the production of parts that cannot be produced by machining due to its inner diameter forms. Due to all these advantages over conventional manufacturing techniques, flow forming technology has been used in defense industry, aviation and space, automotive industry, petrochemical industry and mining industry, (Mohebbi and Akbarzadeh, 2009).

In this work, conventional manufacturing techniques used in grooved gun barrel manufacturing is reviewed. Then an alternative method which is flow form has been considered for the grooved barrel production. Following the manufacturing of grooved gun barrel, the mechanical properties of as-received AISI 1.4330 and flowformed one are investigated by performing tensile tests and hardness measurements. Improvements in all mechanical properties are observed.

2. CONVENTIONAL METHODS FOR GROOVED GUN BARREL MANUFACTURING
Barrel manufacturing has been commonly carried out by three main methods: broaching, radial forging and button rifling. In the broaching method, the groove set geometries are formed by removing the chip from the inner diameter surface. In this method, the microstructure of the barrel material does not change. The method allows for the manufacturing of large-diameter barrels, but has limited possibilities to provide dimensional accuracy in the manufacture of long workpieces. Due to the long production time, it is not generally preferred for mass production.
In the button rifling method, there is a mandrel that forces the material to be pressed by moving it back and forth towards the inner diameter of the cylindrical workpiece. During the movement of the mandrel, hardening occurs on the workpiece inner diameter. In addition, mirror surfaces are obtained.

In the radial forging method, the cylindrical workpiece is forged on a mandrel which has rifling form. During the radial forging process, the workpiece is forced to plastic deformation under high radial loads so residual stresses occur in the material internal structure. It affects negatively the performance of the barrel. For example, the barrel temperature increases during fire, with this warming, the bond forces are weaken, internal stresses in the places where the inter-bond forces weaken causes the barrel to deform. This will adversely affect the accuracy of barrel targeting. For this reason, the heat treatment is very important. In addition, the production of barrel with radial forging is much faster than other conventional methods, therefore, it is more preferred in serial manufacturing, (Podder et al. 2012).

3. FLOWFORM METHOD FOR GROOVED GUN BARREL MANUFACTURING

Apart from conventional methods, the flowform method is used in grooved gun barrel manufacturing. Flowform is an advanced cold forming technology and used for producing high precision thin and thick walled symmetric components. In this method, the axial and radial forces are applied to the work piece which is on the circular section mold. During the process, the mold rotates around its axis. As a result of the applied axial and radial forces, work piece undergoes a plastic deformation, thinning in the wall thickness and elongation are observed. By choosing this method, high costs in the classical production methods, long production processes, cracking in the groove set bottom, abrasions on the groove set are eliminated, (Maj et al. 2018).

The productions were carried out by using the flow form method on the 40 mm barrel gun barrel. Until obtaining the gun barrel with the desired properties, the process parameters and the mold dimensions were changed. The products obtained were controlled in three dimensionally with high precision measurement devices. Then, the final product was obtained in the desired properties. Figure 1 shows 40 mm diameter barrels produced by REPKON.

After the flow form, only the outer diameters of the barrels seen in Figure 1 were turned and the coating process was carried out to prevent corrosion. The groove sets in the inner diameter have been left in post process and not subjected to any treatment. While the extra-polishing process was required in the classical methods after the process, any polishing process is not needed in flow formed gun barrels. It was seen that the groove set surfaces after the flowform process were in the mirror brightness.

4. EXPERIMENTAL WORK

4.1 Uniaxial Tensile Tests

As stated above, the changes in the mechanical properties of the material occur after the flowform process. In order to understand the nature and quantity of these changes, the material coded 1.4330, which is used in the manufacturing of gun barrel, was subjected to tensile test before flowform process. In addition, tensile testing of the flowformed material is carried out and the test results are given in Figure 2.
When the results are compared, it is observed that elasticity modulus did not change. However, yield stress increases around 2% in the flowformed material. Main difference is observed in the ultimate stress. Almost 70% increase is seen. On the other hand, nonlinear hardening behavior is much more prominent in flowformed one. Break stresses are almost the same. Strain at break is somewhat bigger in the undeformed material. Mechanical properties of 1.4330 steel and flowformed one are given in Table 1.

Table 1. Mechanical properties of 1.4330 steel and flowformed one obtained from tensile test.

<table>
<thead>
<tr>
<th></th>
<th>1.4330 steel (as-received)</th>
<th>Flowformed 1.4330 steel</th>
</tr>
</thead>
<tbody>
<tr>
<td>Elasticity modulus (GPa)</td>
<td>168.14</td>
<td>170.25</td>
</tr>
<tr>
<td>Yield stress (MPa) (%0.2 offset)</td>
<td>582.43</td>
<td>594.61</td>
</tr>
<tr>
<td>Ultimate stress (MPa)</td>
<td>740.79</td>
<td>1261.68</td>
</tr>
</tbody>
</table>

4.2. Hardness Measurements

Hardness measurements were made on the surface of the material prior to forming (preform) and after forming. Vickers was preferred as a method of measurement and the results were charted and compared. Yivset and Yivset-XYZ measurements are taken in accordance with the distances shown in the following illustration. Vickers hardness measurements were made by applying a load of 300 g (HV 0,3).

4.2.1 Material Hardness Measurement in As-received 1.4330 Steel

The hardness value and the submergence images of the material are given below before being subjected to the flowform process (Figure 3). The obtained hardness value is 257.33 HV.

Figure 2. Stress-strain behavior of 1.4330 steel and flowformed steel under uniaxial tension

Figure 3. Hardness measurements before flowform process.
4.2.2 After Flow Forming, Material Hardness Measurement Results

The results of the measurements made in order to observe the changes that may occur in the hardness of the material subjected to the flowform are given in Table 2 and the points taken in the grooved barrel are given in Figure 4.

![Figure 4](image1.png)  
**Figure 4. Points on the barrel where the measurements are done.**

![Figure 5](image2.png)  
**Figure 5. Hardness measurements on flowformed gun barrel**

Comparing the hardness measurements taken on the pre-forming material and from the section close to the groove set reveals that 80 Vickers increase in material hardness is observed. This improvement on the hardness will reduce wear on the groove set.
5. COMPARISON OF FLOWFORMED GUN BARREL WITH TRADITIONAL GUN BARREL

5.1. Microstructure Analysis

Samples were taken from two different gun barrels and the microstructures of the samples were examined by 200x magnification. Inner, middle and outer section of samples were examined respectively. The results are depicted in Figure 6. Morphology of flowformed barrel shows an internal structure that the grain size reduces from the inside out. There is no change in the internal structure of the traditional barrel in terms of grain size. Reduction of the grain size of the materials allows the yield and tensile strength to increase according to the Hall-Petch principle. The direction of the grain structure reduces the possibility of notching by reducing the stress concentration points, thus increasing material life and strength.

<table>
<thead>
<tr>
<th>Inner Section</th>
<th>Floformed Barrel Microstructure</th>
<th>Traditional Barrel Microstructure</th>
</tr>
</thead>
<tbody>
<tr>
<td>Middle Section</td>
<td><img src="image1" alt="Floformed Barrel Microstructure" /></td>
<td><img src="image2" alt="Traditional Barrel Microstructure" /></td>
</tr>
<tr>
<td>Outer Section</td>
<td><img src="image3" alt="Floformed Barrel Microstructure" /></td>
<td><img src="image4" alt="Traditional Barrel Microstructure" /></td>
</tr>
</tbody>
</table>

Figure 6. Microstructure of Gun Barrel
When Figure 7 is examined, it is seen that grain structure is directed in the flowformed gun barrel. These orientations in the grain structure led to the formation of layers such as onion skins. This orientation and layers increase friction and notch strength in the groove-set region.

<table>
<thead>
<tr>
<th>Flowformed Barrel Yivset Microstructure</th>
<th>Traditional Barrel Microstructure</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image1.jpg" alt="Flowformed Barrel Yivset Microstructure" /></td>
<td><img src="image2.jpg" alt="Traditional Barrel Microstructure" /></td>
</tr>
</tbody>
</table>

**Figure 7. Gun barrel yivset microstructure**

### 6. CONCLUSIONS

In this work, the grooved gun barrel is manufactured by flowforming method instead of conventional methods such as broaching, radial forging and button rifling. The mechanical properties of as-received AISI 1.4330 steel and flowformed gun barrel made from AISI 1.4330 steel are investigated. Uniaxial tensile tests are performed for undeformed and flowformed AISI 1.4330. In addition, hardness and internal structure of both specimens are investigated using Vickers hardness test and optical microscopy respectively.

It is observed that ultimate stress is increased almost 70% while small changes are observed in elasticity modulus and yield stress. Also, hardness values increase in preferred areas in flowformed barrel. When microstructures of both of the specimens are examined, a significant difference attracts the attention: in flowformed barrel grain size reduces from the inside out, however there is not such a situation in traditional barrel and its microstructure is homogeneous everywhere. Due to the reduction of the grain size in the flowformed material, the yield and tensile strength are increased according to the Hall-Petch principle.

### ACKNOWLEDGEMENTS

This work is supported by YTU, BAP. (Project number: FYL-2018-3335)

### REFERENCES


A TEST BENCH TO STUDY THE TRANSLATIONAL PLATFORM STABILIZATION

Ayşen Süheyla Bağbasi\textsuperscript{1,a}, Yağmur Ersan\textsuperscript{1,b}, Betül Oktay\textsuperscript{1,e}, Ahmet Turgut Başaranoğlu\textsuperscript{1,d}, Salar Rahimi\textsuperscript{2,e}, Hassan Gol Mohammedzadeh\textsuperscript{3,f}, Kutluk Bilge Arıkan\textsuperscript{1,g}

\textsuperscript{1}Department of Mechanical Engineering, Faculty of Engineering, TED University, Ankara, Turkey
\textsuperscript{2}Department of Mechanical Engineering, Faculty of Engineering, Middle East Technical University, Ankara, Turkey
\textsuperscript{3}Department of Mechatronics Engineering, Faculty of Engineering, Atılım University, Ankara, Turkey

\textsuperscript{a}asuheyla.bagbasi@tedu.edu.tr, \textsuperscript{b}yagmur.ersan@tedu.edu.tr, \textsuperscript{e}betul.oktay@tedu.edu.tr, \textsuperscript{d}ahmet.basaranoglu@tedu.edu.tr, \textsuperscript{e}salar.rahimi1995@gmail.com, \textsuperscript{f}hassan.golmohamadzadeh@atilim.edu.tr, \textsuperscript{g}kutluk.arikan@tedu.edu.tr

ABSTRACT

This paper briefly presents the experimental setup that is designed to study the stabilization of the translational platform on which a camera is placed. It is developed by the third-year students within the scope of the `Modeling and Control of Dynamic Systems' course in order to implement the control systems and disturbance rejection algorithms. A spring-mass-damper system that can vibrate in the horizontal plane is produced. A four-bar mechanism is mounted on the vibrating mass and it is used to generate the rectilinear motion for the camera which is placed on the coupler link of the mechanism. The mathematical model of the dynamical system is developed, and the control systems are designed based on the simulations. Designed controllers are implemented on the physical system using Simulink Desktop Real-Time software and the performances are evaluated. The main motivation is to design control architectures for the purpose of translational and rotational disturbance rejection which are inspired by the birds.

Keywords: Disturbance rejection, platform stabilization, four-bar mechanism

1. INTRODUCTION

Platform stabilization is required in various fields such as film, automotive and the defense industry. For example, Steadicam and Fig Rig are good examples of the camera stabilization in film industry [1]. Active stabilization techniques are used in advanced systems [2]. In robotics, platform stabilization is implemented, as well. As a typical example a head stabilization platform in [3] is designed to reject the disturbances caused by the six-legged Rhex robot. Camera is mounted on the platform and its orientation is controlled by using the closed loop control algorithm [4-6].

In nature, platform stabilization is performed by the biological creatures, as well. The authors in [7], did experiments with pigeons and examined the head stabilization. According to them, bird head stabilization is important to stabilize the image on the retina during walking. Multimodal sensory feedback from the eyes, the vestibular system and the proprioception in the body and neck are required to perform head stabilization. The reactions of the birds against the rotational and translational disturbances are different. In rotational motion, they can stabilize their heads perfectly. Conversely, it is poor during translational perturbation. The vision has a small but measurable effect on head stabilization during translation [8].

In the above-mentioned studies, the aim is to reject the disturbances due to the base motions. This paper presents the experimental setup that is designed to study the stabilization of the translational platform on which a camera is placed, Figures 1 and 2. It is developed by the third-year Mechanical Engineering students within the scope of the ‘ME 336 - Modeling and Control of Dynamic Systems’ course. A spring-mass-damper system that can vibrate in the horizontal plane is produced. A four-bar mechanism is mounted on the vibrating mass and it is used to generate the rectilinear motion for the camera which is placed on the coupler link. The mass can be disturbed by
the slider-crank mechanism as shown in Figure 1. The four-bar linkage system was placed on the mass of the system to reject the base motion.

![Figure 1. 3D model of the system](image1.png)

![Figure 2. Camera stabilization system with the four-bar mechanism](image2.png)

2. MATHEMATICAL MODEL OF THE SYSTEM

Four-bar mechanism is used in this study in the form of a parallelogram. With the parallelogram four-bar linkage in Figure 3, the orientation of the coupler does not change during the motion and image stabilization could be provided.

![Figure 3. Representation of four bar mechanism](image3.png)

Angle between x axes and third rod assumed as zero because of parallelism. Free body diagrams of the system are given in Figures 4-7. Due to the small inertia and accelerations, quasi-static motion assumption is made for the motions of the links 2 and 4.
Following equations are derived using the Newton’s Law based on the free body diagram in Fig. 4.

\[ \sum F_x = 0; \quad F_1 + F_3 = 0 \]  \hspace{1cm} (1)

\[ \sum F_y = 0; \quad F_2 + F_4 = 0 \]  \hspace{1cm} (2)

\[ \sum M_{G2} = 0; \quad F_1 \left( \frac{a_2}{2} \right) \sin \theta_2 - F_2 \left( \frac{a_2}{2} \right) \cos \theta_2 - F_3 \left( \frac{a_2}{2} \right) \sin \theta_2 + F_4 \left( \frac{a_2}{2} \right) \cos \theta_2 + T = 0 \]  \hspace{1cm} (3)

Following equations are derived using the Newton’s Law based on the free body diagram in Fig. 5.

\[ \sum F_x = m_3 \ddot{x}_{G3}; \quad -F_3 + F_5 = m_3 \ddot{x}_{G3} \]  \hspace{1cm} (4)

\[ \sum F_y = m_3 \ddot{y}_{G3}; \quad -F_4 + F_6 = m_3 \ddot{y}_{G3} \]  \hspace{1cm} (5)

\[ \sum M_{G3} = 0; \quad F_4 \left( \frac{a_3}{2} \right) + F_6 \left( \frac{a_3}{2} \right) = 0 \]  \hspace{1cm} (6)
Following equations are derived using the Newton’s Law based on the free body diagram in Fig. 6.

\[ \sum F_x = 0 ; -F_5 + F_y = 0 \]  \hspace{1cm} (7)

\[ \sum F_y = 0 ; -F_6 + F_8 = 0 \]  \hspace{1cm} (8)

\[ \sum M_{G4} = 0 ; F_5 (\frac{a_4}{2}) \sin \theta_2 + F_6 (\frac{a_4}{2}) \cos \theta_2 + F_7 (\frac{a_4}{2}) \sin \theta_2 + F_8 (\frac{a_4}{2}) \cos \theta_2 = 0 \]  \hspace{1cm} (9)

\[ \sum F_x = m_1 \ddot{x}_{G1} ; -F_1 + F_7 + F_{in} = m_1 \ddot{x}_{G1} \]  \hspace{1cm} (10)

\[ \sum F_y = m_1 \ddot{y}_{G1} = 0 ; -F_2 - F_8 = 0 \]  \hspace{1cm} (11)

\[ \sum M_{G1} = 0 ; F_2 (\frac{a_1}{2}) - F_8 (\frac{a_1}{2}) + T_{reaction} = 0 \]  \hspace{1cm} (12)

Twelve equations are used to find the following unknowns.

\[ F_1 = \frac{T}{\sin(\theta_2) a_2} \]  \hspace{2cm} \[ F_7 = 0 \]

\[ F_2 = 0 \]  \hspace{2cm} \[ F_8 = 0 \]

\[ F_3 = -\frac{T}{\sin(\theta_2) a_2} \]  \hspace{2cm} \[ \ddot{\theta}_2 = \frac{-c \dot{x}_{G1}}{a_2 m_1} + \frac{F_{in}}{a_2 m_1} - \frac{k \dot{x}_{G1}}{a_2 m_1} - \frac{(m_1 + m_3) T}{a_2^2 m_1 m_3} \]

\[ F_4 = 0 \]  \hspace{2cm} \[ \ddot{y}_{G3} = 0 \]

\[ F_5 = 0 \]  \hspace{2cm} \[ T_{react} = 0 \]

\[ F_6 = 0 \]  \hspace{2cm} \[ \ddot{x}_{G1} = \frac{-c \dot{x}_{G1}}{m_1} + \frac{F_{in}}{m_1} - \frac{k \dot{x}_{G1}}{m_1} - \frac{T}{a_2 m_1} \]

Loop closure equation (LCE) is used to simulate the kinematics of the mechanism. The link lengths are selected so as to reach ±5 cm camera translation with the admissible transmission angles. For a parallelogram type of mechanism LCE may not be necessary. However, alternative mechanisms were considered at the beginning of the project and the standard LCE is derived and utilized. Using the simulations of the forward and backward kinematics, the link lengths are selected as follows: lengths of the rod 1 and 3 are 0.08 m, lengths of the rod 2 and 4 are 0.1 m. Simulation of the mechanism to validate the ±5 cm camera translation is presented in Fig. 8.
3. CONTROL SYSTEM

The linearized dynamical equations are given as below.

\[
\ddot{\theta}_2 = -c \dot{x}_{G1} \frac{F_{in}}{a_2 m_1} + \frac{k x_{G1}}{a_2 m_1} \frac{(m_1 + m_3)T}{a_2^2 m_1 m_3}
\]  

(15)

\[
\ddot{x}_{G1} = -c \dot{x}_{G1} \frac{F_{in}}{m_1} + \frac{k x_{G1}}{m_1} - \frac{T}{a_2 m_1}
\]  

(16)

\[
T = \frac{K_t V}{R_a} - \frac{\theta_2 K_i K_p r^2}{R_a}
\]  

(17)

\(T\) is the motor torque generated by the dc motor which is coupled to link 2. Simulations in Simulink are performed using the model in Fig. 10.
PID controller is designed to reach the desired transient response specifications and tuned using the simulations. Following plot presents the performance of the closed loop system as the disturbance force is applied.

![Figure 11. Performance of model in Simulink](image)

Disturbances usually appear in systems and degrade the system performance. Disturbance rejection is of great importance for control systems. Also, acceleration feedback can be used to improve the performance of the system for the purpose of disturbance attenuation. It has the effect of virtually increasing inertia, damping and stiffness. It can be used to increase the disturbance rejection without sacrificing tracking performance. Because of that, acceleration feedback algorithm is utilized in the simulations and in physical implementations. If the value of disturbance is known, feedforward method could be used to reduce the effect of the disturbance. Alternatively, disturbance observers (DOB) are used to reject the disturbance [9-11]. In this study, DOB is implemented in the simulations, as well.

### 4. EXPERIMENTAL RESULTS

The aim of this project is to reject the disturbances due to the base motions. A spring mass damper system which can vibrate in the horizontal plane is produced. A four-bar mechanism is used to produce translational motion. The designed controllers are implemented on the physical system using the Simulink Real-Time Desktop software and the performances are interpreted. The physical system is shown in the schematics in Fig. 12.

![Figure 12. Connections to operate setup in real time](image)

The performance of the PID controller is evaluated in various disturbances. Typical results are presented in the following figures.
The base is moved harmonically in Figure 13. The disturbance is implemented as a step type of base displacement in Figure 14. On the other hand, the acceleration feedback is utilized in Figure 15.

5. DISCUSSION and CONCLUSION

An experimental setup is designed and produced to study the platform stabilization with the implementation of different kinds of control systems. Position feedback is processed in a PID type control system. Nonlinear and linearized models are derived and utilized. The use of acceleration feedback may improve the disturbance rejection performance. DOB type architecture is studied, as well. However, it requires more effort to tune and get the satisfactory results. The bioinspired control architectures will be designed and implemented as the future work. It is experienced that the physical characteristics of the mechatronic components such as the actuators and sensors have great importance in the control system implementation.
REFERENCES


RHEOLOGY OF CORNSTARCH SUSPENSION WITH CARBIDE PARTICLES

Selim Gürgen
Eskişehir Vocational School, ESOGU, Eskişehir, 26110, Turkey
sgurgen@ogu.edu.tr

ABSTRACT
Cornstarch particles in a water medium exhibit non-Newtonian properties although the liquid medium of water is a Newtonian fluid. The change in the rheological behavior is described by the effect of cornstarch particles that enhance the hydrodynamic forces and inter-particle contacts in the suspension. In addition to this, carbide particles were included in a cornstarch/water suspension and the effect of this second phase was investigated. According to the rheological results, the suspension showed significant changes in terms of critical shear rate, critical viscosity, maximum viscosity and thickening ratio, as the carbide particles are included in the mixture.

Keywords: Non-Newtonian fluid, cornstarch, suspension, carbide particles, rheology.

1. INTRODUCTION
Shear thickening fluid (STF) is a kind of non-Newtonian fluids exhibiting viscosity increase under loading. STFs are fabricated dispersing nano or micro-sized colloidal particles in a carrier liquid. Shear thickening mechanism is explained by hydro-clustering theory. In this theory, hydrodynamic forces are the major factor on the viscosity increase. As the shear rate increases in the suspension, hydrodynamic forces grow on the particles and therefore, particles are brought together and forming large particle clusters. These particle groups are called hydro-clusters and hinder the fluid flow and consequently lead to a viscosity increase in the mixture. As shear rate increases in the suspension, hydrodynamic forces grow and hydro-clusters are extended to a wider scale in the suspension [1,2]. Hydro-clustering theory was verified by computational studies and dynamic simulations in the literature [3,4]. Even though the hydro-clustering theory explains the shear thickening behavior, recent investigations suggest that particle contacts are also important mechanism in the viscosity increase. At low shear rates, hydrodynamic forces predominate the suspension with a contactless microstructure. However, at high shear rates, particles in the suspension physically contact each other by the effect of elevated hydrodynamic forces. Therefore, liquid film between two particles may be completely removed such a high loading. For this reason, force chains are formed via particles within the hydro-clusters and these chains grow in the suspension as shear rate increases during the loading.

In the light of these theories, two different STFs were investigated in the present work. The first one was fabricated dispersing cornstarch particles in a water medium. The second one was derived from the first suspension dispersing carbide particles as a second phase. Therefore, multi-phase STF concept was introduced in this study. The suspensions were subjected to rheological measurements in a stress-controlled rheometer. In the measurements, shear thickening parameters such as critical shear rate, critical viscosity, maximum viscosity and thickening ratio were found and these parameters were compared for each suspension.

2. MATERIALS AND METHOD
In the present work, STF was fabricated dispersing cornstarch particles in water. The concentrations of the components in the neat STF were selected as 55 wt% for cornstarch and 45 wt% for water. For the multi-phase STF, 10 g of silicon carbide particles were added into 100 g of a neat STF. Figure 1 shows the scanning electron microscope image of silicon carbide particles.
3. RESULTS AND DISCUSSION

Figure 2 shows the rheological curves of the neat and carbide added STFs. From the given chart, viscosity increase is obvious as shear rate increases for both suspensions. This behavior is called shear thickening and fluids exhibiting this kind of viscosity increase are called shear thickening fluids (STFs). There are three stages in the rheological curve of STFs. First stage is known as shear thinning and during this stage, viscosity of the suspension decreases until a certain point. Upon reaching this critical point, shear thickening onsets and suspension viscosity exhibits a drastic rise. The onset point is called critical shear rate. The second stage is known as shear thickening and this stage extends to a maximum point of viscosity. Last stage begins after the maximum viscosity point and this one is seen at the elevated shear rates. In this stage stability problems may be seen due to spilled sample between the rheometer plates. However, generally shear thinning predominates this stage.

![Rheological curves of the suspensions](image)

**Figure 2.** Rheological curves of the suspensions

<table>
<thead>
<tr>
<th>Suspension</th>
<th>Critical shear rate (1/s)</th>
<th>Critical viscosity (Pa.s)</th>
<th>Max viscosity (Pa.s)</th>
<th>Thickening ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cornstarch/Water</td>
<td>10.85</td>
<td>0.056</td>
<td>0.38</td>
<td>6.64</td>
</tr>
<tr>
<td>Cornstarch/Water/SiC</td>
<td>4.91</td>
<td>0.160</td>
<td>1.36</td>
<td>8.50</td>
</tr>
</tbody>
</table>

**Table 1.** Shear thickening parameters for the suspensions
Table 1 gives the shear thickening parameters for the suspensions. According to these results, critical shear rate, at which the suspension starts to thicken, is 10.85 and 4.91 s\(^{-1}\) for the suspensions cornstarch/water and cornstarch/water/SiC respectively. It shows that shear thickening point reduces as carbide particles are included in the suspension. On the other hand, critical viscosity, which is the viscosity of suspension at the critical shear rate, enhances with the addition of carbide particles. In fact, addition of carbides enhances the density of particles in the suspension and thereby increasing the initial viscosity of the mixture. For this reason, viscosity curve of the mixture with carbide additives is above the neat STF for all shear rate points. From this fact, it is expected finding that critical viscosity enhances adding carbide particles into the suspension. Another important parameter is maximum viscosity for the STFs. It is seen that neat STF reaches the peak viscosity of 0.38 Pa.s whereas addition of carbides enhances the maximum viscosity level to 1.36 Pa.s. Indeed, it is again related to the mixture density as discussed in the previous stage. Inclusion of carbide particles in the suspension enhances the initial viscosity and therefore, viscosity increase reaches upper levels for denser STFs. However, thickening ratio, which is found dividing maximum viscosity by critical viscosity, reduces adding carbides in the mixture. This parameter shows the intensity of shear thickening behavior in the mixture and considering the results, the inclusion of carbide additives enhance the shear thickening behavior. The neat STF has a thickening ratio of 6.64 whereas this jumps to 8.50 with the addition of carbides in the suspension. Considering all these results, it is possible to mention that additive particles are beneficial for the shear thickening mechanism of cornstarch/water suspension.

4. CONCLUSIONS

In this study, a non-Newtonian fluid was fabricated dispersing cornstarch in a water medium. The suspension exhibited shear thickening behavior under a changing shear rate. In order to investigate multi-phase concept, carbide particles were added into the cornstarch/water suspension and the influence of additive particles were discussed. Based on the rheological measurements, carbide particles enhance the shear thickening mechanism in the cornstarch/water suspension. Carbide additives reduce the critical shear rate of the mixture which shows that thickening onsets at lower loadings on the suspension. Furthermore, initial viscosity and peak viscosity are both increased in the suspension by the effect of carbide additives. These changes stem from the increase in the particle density. In addition, thickening ratio introduces an increase adding carbide particles in the mixture.

REFERENCES

INFLUENCE OF MANUFACTURING CONDITIONS ON WEAR PROPERTIES OF UHMWPE

Selim Gürgen

Eskişehir Vocational School, ESOGU, Eskişehir, 26110, Turkey
sgurgen@ogu.edu.tr

ABSTRACT

Ultra-high molecular weight polyethylene (UHMWPE) based products are widely utilized in industry due to their superior wear resistance. In this study, UHMWPE powder was sintered in a compression molding machine to fabricate disc shaped specimens. The influence of manufacturing conditions such as molding pressure was investigated using three different levels. The specimens were subjected to wear tests in a ball-on-disk configuration and a wolfram carbide (WC) ball was utilized as a counterpart in the tests. Based on the results, both molding pressure exhibited significant influences on the wear properties of UHMWPE specimens.

Keywords: UHMWPE, wear, compression molding, molding pressure.

1. INTRODUCTION

Ultra-high molecular weight polyethylene (UHMWPE) based products are widely utilized in industry due to their superior wear resistance. The primary application areas for this material are fishing lines, sailing equipment, climbing ropes, body armor and implant for different body parts such as spine, knee and hip. In the industrial applications, various kinds of manufacturing methods are utilized to produce components made of UHMWPE. Compression molding is one of these methods extensively preferred for manufacturing. In this method, generally powder form material is utilized. UHMWPE powder is heated in a mold cavity and a pressure is applied in the mold until the powder has cured. This method is very suitable for mass-production because the initial investment is only required for mold and die. After this non-recurring cost, the production requires only material cost on the line. Even though it is seen as a straight-forward method, the process should be designed precisely. Molding pressure is one of the most important parameter in compression molding technique since the product properties are heavily dependent on it. Insufficient pressures lead to fusion defects which are seen as crack evolution and porous structure in the products. On the other side, over-pressures result in degradation of material and extra energy consumption in the process. For this reason, optimum design for the molding pressure is crucial in compression molding method in common with all engineering operations.

In the present study, the wear behavior of UHMWPE specimens was investigated. The specimens were produced from the powder form of UHMWPE using compression molding method. In order to investigate the role of molding pressure on the wear properties of UHMWPE, molding pressure was varied from 50 to 250 bar. Wear tests were conducted in a ball-on-disk configuration against a wolfram carbide (WC) ball. In addition to wear behavior, microhardness of the specimens was measured according to the hardness Vickers scale. Therefore, the relationship between the wear behavior and hardness was also discussed in the results.

2. MATERIALS AND METHOD

In this work, UHMWPE powder was used in compression molding method to fabricate 30 mm diameter disc shaped specimens. In order to investigate the molding pressure on the wear behavior of the specimens, three different pressures were applied in the molding stage. The pressure levels decided as 50, 150 and 250 bar. Molding temperature was remained constant at 150°C which is just above the melting point of UHMWPE. In the molding process, UHMWPE powder was subjected to heating for 3 min and a successive cooling for 2 min. In the wear test, a ball-on-disk configuration was utilized with a 3 mm diameter WC ball as a counterpart. The test was carried out with a linear speed of 20 cm/s for a distance of 500 m under a load of 10 N.
3. RESULTS AND DISCUSSION

Figure 1 shows the specific wear rate and microhardness values with respect to molding pressure. Specific wear rate was calculated from Eq.1 where “\(\Delta V\)” is the volume loss, “\(L\)” is the applied load and “\(d\)” is the sliding distance. Microhardness of the specimens was measured using a Future-Tech FM-700 system with a load of 25 gf according to the hardness Vickers scale. As given in the graph, specific wear rate of the specimens decreases with an increase in the molding pressure. On the other side, hardness results grow as the molding pressure increases. In fact, wear rate and hardness is directly related to each other as a general rule in tribology science that wear rate of a material is reduced by enhancing the hardness of the material. Based on the results, it can be mentioned that higher molding pressures provide improved consolidation in the specimens through accelerating the diffusion between the UHMWPE particles. This process stems from the better compressibility at high pressures which removes the gaps between the particles and increases the particle contacts during the molding [1-3]. Specimens subjected to high molding pressures possess more compact microstructures and thereby having higher hardness values. For this reason, these specimens exhibit advanced wear resistance and thus, a lower wear rate during the wear tests. Figure 2 shows the correlation between the specific wear rate and microhardness of the specimens. From this chart, it is obvious that hardness is the major mechanism to improve the wear resistance of the specimens.

Specific wear rate = \(\Delta V/(L.d)\)  \hspace{1cm} (1)

![Figure 1. Specific wear rate and microhardness values with respect to molding pressure](image1)

![Figure 2. Correlation between specific wear rate and microhardness](image2)
4. CONCLUSIONS

In this study, UHMWPE powder was sintered to produce disc shaped specimens in a compression molding machine. The effect of molding pressure was investigated changing the pressure from 50 to 250 bar during the sintering process. The specimens were subject to wear tests against a ball made of WC. In the results, specific wear rates and microhardness for each specimen were found. Based on the results, molding pressure of 250 bar leads to a more compacted and consolidated microstructure in the sintered specimens with respect to those produced at lower molding pressures. Therefore, UHMWPE sintering at higher pressures provides improved wear resistance and hardness.

REFERENCES

BUCKLING OF ISOTROPIC COLUMNS WITH UNCERTAIN DIMENSIONS AND MATERIAL PROPERTIES

Onur Kaya, Ahmet Sinan Oktem

1,2 Gebze Technical University, Mechanical Engineering, Kocaeli, Turkey
1 Corresponding Author: Onur Kaya, onur.kaya@gtu.edu.tr

ABSTRACT

Elastic buckling load for columns made of isotropic material for different shapes of cross-sections is studied by taking the variations in the material property and dimensions into consideration. Axially loaded columns are considered as simply supported from both ends. Critical buckling load may be adversely affected as there may be issues such as voids or defects during production or inability to obtain the desired quality in the dimensions of the material. Therefore, the Young’s Modulus of material and its dimensions are taken as uncertain. The classical Euler formula is used to determine the critical buckling load for different uncertainty levels of the material and dimensions. The uncertain buckling load expressions are derived for the effects of the uncertain material parameters and uncertain dimensions. The least favorable value of the buckling load is obtained via convex modeling corresponding to the worst case material properties and dimensions. Finite element method (FEA) is also employed in order to compare the results with the analytical model. The results obtained with analytical model is in close agreement with the finite element counterparts. It’s seen that even minor variations in the material properties and dimensions lead to considerable loss of critical buckling load.

Keywords: Buckling, Uncertainty, Convex Modeling

INTRODUCTION

The buckling occurs if an axially compressive load is subjected to a member. When applied load reaches a point that makes the column unstable, large laterally deflections occur on the member and this situation called buckling. Critical buckling loads take a crucial role in structural stability and it is a fundamental concern for engineers whether the structure is stable or not.

So far, extensive studies that includes deterministic approaches, with deterministic parameters, are investigated for buckling of columns. However, in reality, material properties and the dimensions should be taken as uncertain because of voids or defects during production or inability to obtain the desired quality in the dimensions of the material. These defects or voids and variations on the dimensions may adversely influence the critical buckling load. Therefore, uncertainty phenomena should be considered to obtain realistic critical buckling load. Several methods such as convex modeling, probabilistic and statistical approaches may be employed in order to investigate the uncertainty for the structures. As probabilistic and statistical approaches require probabilistic density function and experimental data, convex modeling which requires less information may be used for uncertainty applications [1 – 4].

Previously, Morgan EJ et al. [5] investigated an experimental study. Differences between the critical buckling loads for experimental results and theoretical results have been reported. This may be caused by defects in the material. Besides, several researches are published that includes uncertainty of material properties, dimensions, imperfect support conditions and deviations on the applied loads [6 – 10].

Shu Zhang et al. [11] studied uncertain initial geometrical imperfection on the buckling of a thin plate. In order to define imperfections, they have employed double trigonometric series and then Monte Carlo simulation was used for determining the range of maximum deflection. Oktem AS and Adali S [3] investigated buckling of nanocomposite columns with uncertain material properties and used convex modeling in order to determine the least favorable situation of critical buckling load at different uncertain levels. Radebe IS and Adali S [4] reported minimum cost design of cylinders with uncertain material properties. Cylinders were hybrid cross – plied and were exposed to external pressure. According to uncertainty levels, convex modeling was used to calculate the minimum buckling pressure for various parameters.
In this paper, the elastic buckling load for columns made of isotropic material for different shapes of cross-sections is studied by taking the variations in the material property and dimensions into consideration for different cross sections. Axially loaded columns are considered as simply supported from both ends. By using convex modeling, least favorable value of critical buckling loads for different shape of cross sections are determined and the effect of the worst-case of material properties and dimensions on critical buckling load are investigated. Finite element method (FEA) is also employed in order to compare the results with the analytical model.

**BUCKLING OF COLUMNS**

Euler – Bernoulli beam theory is used to derive the critical buckling load for isotropic materials. Boundary conditions at the ends of the columns are considered as simply supported.

\[
M = -EI \frac{d^2w}{dx^2} \tag{1}
\]

and

\[
EI \frac{d^2w}{dx^2} + Pw = 0 \tag{2}
\]

where \(w, M, E, I, P\) are the lateral deflection, bending moment, Young’s Modulus, area moment of inertia and applied load, respectively. By using Eq. (1) and (2), the Euler’s critical buckling load can be expressed as below:

\[
P_{cr} = \frac{n^2 \pi^2 EI}{L^2} \tag{3}
\]

where \(L\) and \(n\) are the length of column and buckling mode number, respectively and it should be noted that the critical buckling load for isotropic materials which includes uncertainty can be described as

\[
\overline{P}_{cr} = \frac{n^2 \pi^2 E\overline{I}}{L^2} \tag{4}
\]

**UNCERTAINTY ANALYSIS**

Uncertain material constants of isotropic materials can be denoted in terms of deterministic values \(E, L, B, H, b, h, D, d\) which are shown in Table 1 and also the parameters that are taken as uncertain for four different shapes of cross-sections are indicated in detail. Fig.1 shows the shapes of cross-sections and dimensions considered as uncertain.
Table 1. Uncertain parameters used in convex model (Over-bar indices on symbols indicate uncertain parameter)

<table>
<thead>
<tr>
<th>Number</th>
<th>Cross-Sections</th>
<th>Material Properties</th>
<th>Dimensions</th>
</tr>
</thead>
<tbody>
<tr>
<td>I</td>
<td></td>
<td>( \bar{E}^I = E^I (1 + \delta^I) )</td>
<td>( \bar{b}^I = b^I (1 + \delta^I) ) ( \bar{h}^I = h^I (1 + \delta^I) ) ( \bar{L}^I = L^I (1 + \delta^I) )</td>
</tr>
<tr>
<td>II</td>
<td></td>
<td>( \bar{E}^{II} = E^{II} (1 + \delta^{II}) )</td>
<td>( \bar{B}^{II} = B^{II} (1 + \delta^{II}) ) ( \bar{H}^{II} = H^{II} (1 + \delta^{II}) )</td>
</tr>
<tr>
<td>III</td>
<td></td>
<td>( \bar{E}^{III} = E^{III} (1 + \delta^{III}) )</td>
<td>( \bar{D}^{III} = D^{III} (1 + \delta^{III}) ) ( \bar{L}^{III} = L^{III} (1 + \delta^{III}) )</td>
</tr>
<tr>
<td>IV</td>
<td></td>
<td>( \bar{E}^{IV} = E^{IV} (1 + \delta^{IV}) )</td>
<td>( \bar{D}^{IV} = D^{IV} (1 + \delta^{IV}) ) ( \bar{d}^{IV} = d^{IV} (1 + \delta^{IV}) ) ( \bar{L}^{IV} = L^{IV} (1 + \delta^{IV}) )</td>
</tr>
</tbody>
</table>

\( \delta_i \) (i=1…6) values can be positive or negative that satisfy the worst-case situation. These parameters are calculated by using convex modeling which leads to least favorable critical buckling load. To determine the uncertainty level, the constraint equation should satisfy the following condition and it is given as below:

\[
\sum_{i=1}^{m} \delta_i^2 \leq \beta^2
\]  

(5)

where \( \beta \) indicates the uncertainty level and \( m \) shows the number of parameters that are taken as uncertain. \( \beta = 0 \) represents the deterministic situation.

**CONVEX ANALYSIS**

The convex analysis is used in order to obtain the minimum critical buckling load for a given level of uncertainty. \( \delta_i \) values are determined by using method of Lagrange multipliers. The formulation is described as below:

\[
L(\delta, \lambda) = P_c + \lambda \left( \sum_{i=1}^{m} \delta_i^2 - \beta^2 \right)
\]  

(6)

\( \delta_i \) parameters and Lagrange multiplier ( \( \lambda \) ) are calculated by using eq. (6) as below:

\[
\frac{\partial L(\delta, \lambda)}{\partial \delta_i} = 0, \quad \frac{\partial L(\delta, \lambda)}{\partial \lambda} = 0
\]  

(7)
Based on eqns. (5 – 7), the calculated uncertain parameters given in Table 1 and non-deterministic critical buckling load ($P_{cr}$) that gives the worst-case situation for the different kind of cross-sections as a function of $\beta$ are given in Table 2.

**Table 2.** Obtained uncertain parameters and critical buckling load equations for worst-case situation

<table>
<thead>
<tr>
<th>Number</th>
<th>Cross-Sections</th>
<th>$\delta_i$</th>
<th>Obtained Critical Buckling Load For Worst – Case</th>
</tr>
</thead>
<tbody>
<tr>
<td>I</td>
<td><img src="image1" alt="Cross-Section" /></td>
<td>$\delta_1 = \frac{-\beta}{\sqrt{15}}$, $\delta_2 = \frac{-\beta}{\sqrt{15}}$, $\delta_3 = \frac{-3\beta}{\sqrt{15}}$, $\delta_4 = \frac{2\beta}{\sqrt{15}}$</td>
<td>$P_{cr}(\beta) = \frac{\pi^2 Ebh^3}{12L^2} \left(1 - \frac{\beta}{\sqrt{15}}\right)^2 \left(1 - \frac{3\beta}{\sqrt{15}}\right)^3 \left(1 + \frac{2\beta}{\sqrt{15}}\right)^2$</td>
</tr>
<tr>
<td>II</td>
<td><img src="image2" alt="Cross-Section" /></td>
<td>$\delta_1 = \frac{-(n_1 - n_2)/\beta}{\sqrt{15(n_1^2 + n_2^2) + 6n_1n_2}}$, $\delta_2 = \frac{n_1\beta}{\sqrt{15(n_1^2 + n_2^2) + 6n_1n_2}}$, $\delta_3 = \frac{-3n_1\beta}{\sqrt{15(n_1^2 + n_2^2) + 6n_1n_2}}$, $\delta_4 = \frac{2(n_1 + n_2)/\beta}{\sqrt{15(n_1^2 + n_2^2) + 6n_1n_2}}$</td>
<td>$P_{cr}(\beta) = \left(1 + \frac{-(n_1 - n_2)/\beta}{\sqrt{15(n_1^2 + n_2^2) + 6n_1n_2}}\right) \left(1 + \frac{2(n_1 + n_2)/\beta}{\sqrt{15(n_1^2 + n_2^2) + 6n_1n_2}}\right)^2$</td>
</tr>
<tr>
<td>III</td>
<td><img src="image3" alt="Cross-Section" /></td>
<td>$\delta_1 = \frac{-\sqrt{21}/\beta}{\sqrt{5(n_1 - n_2)^2 + 16(n_1^2 - n_2^2)}}$, $\delta_2 = \frac{-4\beta}{\sqrt{21}}$, $\delta_3 = \frac{2\beta}{\sqrt{21}}$</td>
<td>$P_{cr}(\beta) = \frac{\pi^3 Ebh^4}{64L^2} \left(1 - \frac{\beta}{\sqrt{21}}\right)^4 \left(1 + \frac{2\beta}{\sqrt{21}}\right)^2$</td>
</tr>
<tr>
<td>IV</td>
<td><img src="image4" alt="Cross-Section" /></td>
<td>$\delta_1 = \frac{-(n_1 - n_2)/\beta}{\sqrt{5(n_1 - n_2)^2 + 16(n_1^2 - n_2^2)}}$, $\delta_2 = \frac{-4n_1\beta}{\sqrt{5(n_1 - n_2)^2 + 16(n_1^2 - n_2^2)}}$, $\delta_3 = \frac{4n_1\beta}{\sqrt{5(n_1 - n_2)^2 + 16(n_1^2 - n_2^2)}}$, $\delta_4 = \frac{2(n_1 - n_2)/\beta}{\sqrt{5(n_1 - n_2)^2 + 16(n_1^2 - n_2^2)}}$</td>
<td>$P_{cr}(\beta) = \left(1 + \frac{-(n_1 - n_2)/\beta}{\sqrt{5(n_1 - n_2)^2 + 16(n_1^2 - n_2^2)}}\right) \left(1 + \frac{2(n_1 - n_2)/\beta}{\sqrt{5(n_1 - n_2)^2 + 16(n_1^2 - n_2^2)}}\right)^2$</td>
</tr>
</tbody>
</table>

**Note:** The equations for $P_{cr}(\beta)$ are given for each cross-section type in the table, taking into account the parameters $\delta_i$ calculated based on eqns. (5 – 7).
RESULTS AND DISCUSSIONS

Fig. 2 indicates that the result obtained by the current analytical model is in close agreement with the finite element model carried out by ANSYS commercial software.

As nominal values, the dimensions b and h are taken as 30 mm and 20 mm, respectively in the rectangle section column and for the rectangular tube the dimensions are B = 30 mm and H = 20 mm with its inner dimensions b = 20 mm and h = 10 mm. In addition to that, the outer diameter of circular section and circular tube columns taken as 30 mm while the inner diameter is 20 mm for the circular tube column (Fig. 1). The column length is taken as L = 1000 mm for all the structures. Young’s modulus used in convex model is taken as 200 GPa for a steel material. The normalized critical buckling load for all the calculations is given in eq. (8)

\[ P_0 = \frac{P_{cr}(\beta)}{P_{cr}(0)} \]  

where \( P_0, P_{cr}(\beta), P_{cr}(0) \) are the normalized value, the uncertain critical buckling load and the deterministic buckling load, respectively. Variation of normalized \( P_0 \) with increasing level of uncertainty is presented in Fig. 3. As the level of uncertainty increased, the normalized buckling load decreased. It is interesting to note that the most effected structure is the column with the hollowed circular cross section with the increase of level of uncertainty.

This is because of the fourth powers of outer and inner diameter in the critical buckling load equation. While the uncertainty level increases, the outer diameter becomes smaller and the inner diameter gets bigger. This leads to a more reduction in the critical buckling load than the other columns considered in this work.
CONCLUSIONS AND RECOMMENDATIONS

The results obtained from the analytical model are in close agreement with the finite element counterparts. It’s seen that even minor variations in the material properties and dimensions lead to considerable loss of critical buckling load. It can be concluded that the uncertainty phenomenon takes a vital role for reliability of columns. These results represent more realistic cases and they are particularly important for the early design stages of such structures for engineers.

APPENDIX

Expressions which are used in equations (see Table 2) are given below.

\[ n_1 = \frac{\pi^2 EBH^3}{12L^2} \]
\[ n_2 = \frac{\pi^2 Ebh^3}{12L^2} \]
\[ r_1 = \frac{\pi^3 ED^4}{64L^2} \]
\[ r_2 = \frac{\pi^3 Ed^4}{64L^2} \]
REFERENCES


NUMERICAL INVESTIGATION OF THE EFFECTS OF ADHESIVE DEFECTS ON THE LOW-SPEED IMPACT BEHAVIOUR OF ADHESIVELY BONDED SINGLE-LAP JOINTS

Mithat Gokhan Atahan

Abdullah Gül University, Department of Mechanical Engineering, Kayseri, Turkey
Corresponding Author: Mithat Gokhan Atahan, gatahan@erciyes.edu.tr

ABSTRACT

Increasing use of adhesive joints in industry such as aerospace and automotive has made it necessary to determine the behavior of joints under impact loading. The aim of this study is the numerical investigation of the low-speed impact behavior and damage analysis of single-lap joints consisting of Al 5754-0 adherends and Araldite® 2015 adhesive with a central adhesive defect. The damage initiation and propagation in the adhesive layer were determined using the finite element method. The adhesive layers were divided into three regions as upper and lower adhesive interfaces regions and adhesive middle region in order to determine the initiation and propagation of the damage in the adhesive. While the upper and lower adhesive interfaces were modeled with cohesive zone model, elastic-plastic material model was implemented for the middle adhesive region and adherends. The effects of the overlap length (25 and 40 mm) and adhesive surface condition were investigated on the damage initiation and propagation in the adhesive layer. It has been determined that the increase in overlap length and the defect rate in the adhesive significantly affect the initiation and propagation of the damage in the single-lap joints. In the case of defect rate above 46% in the adhesive layer, a large increase in adhesive damage was observed and the maximum contact force value and total contact duration were considerably affected.

Keywords: Cohesive, joint, adhesive, impact, defect.

1. INTRODUCTION

Adhesive joints are widely used in industrial applications since they have many advantages compared to other mechanical joining methods. One of the advantages of adhesive joints is that they can be combined with different materials that cannot be combined with other mechanical joining methods. Materials that cannot be combined with many mechanical joining methods, especially composite materials, can be combined with adhesive joints without reducing the strength of the materials. The widespread use of composite materials has increased the importance of determining the mechanical behavior of adhesive joints. Adhesive bonding strength depends on the mechanical properties of adhesive and bonding surface condition. The clean and rough adhesion surfaces provide better adhesion strength [1-3]. Adhesion defects can affect the joint strength as they reduce the adhesion effect between the interfaces. Several studies examining the effect of adhesive defects on joint strength are available in the literature. Ribeiro et al. [4] examined the mechanical behavior of single-lap joints with a central defect in the adhesive layer for different adhesives and overlap lengths. They used the cohesive zone model (CZM) in their numerical studies to evaluate the adhesive damage and joint strength. They determined the effect of adhesive defect size on the joint behavior. De Moura et al. [5] investigated the effect of adhesive defect on the mechanical behavior of carbon-epoxy composite single-lap joints. The interface finite elements and damage model in numerical analyses allowed the determination of the initiation and propagation of damage. They examined the effect of adhesive defect on the joint strength and adhesive failure type. They determined that the joint strength was slightly affected by the presence of the defect. Heidarpour et al. [6] experimentally examined the effect of size and shape of 2D and 3D adhesive defects on ultimate shear strength of single-lap joints. In their work, they used square, circular and triangular adhesive defects to examine the effect of adhesive strength. They determined that as the area of defect increased, the joint strength decreased approximately linearly for single-lap joint with 3D defects. However, in the case of 2D defects in the single-lap joints, the joint strength reduced non-linearly for the single-lap joint with 2D defect. It is necessary for many industrial applications that the adhesively bonded joints serve under impact loading. For this reason, many researchers tend to examine the behavior of adhesively bonded joints under impact loading. Aga and Woldesenbet [7] examined the effect of adhesive thickness on impact behavior. Park and Kim [8] investigated the effect of transverse ice impact on damage resistance of...

According to the literature review done, there is no study to investigate the behavior of the adhesively bonded single-lap joint including defect under bending impact load. Considering the industrial areas where adhesive joints are used, it is important to determine the damage initiation and propagation under the impact load. The main purpose of this study is to numerically investigate the effect of adhesive defect on the impact behavior of single-lap joint. The damage initiation and propagation in the adhesively joints were determined using cohesive elements.

2. FINITE ELEMENT MODELING APPROACH

The effect of adhesive defects on low velocity impact behavior of single-lap joints was investigated with three dimensional dynamic explicit finite element analysis using ABAQUS/Explicit software [15]. The numerical model used in this study is similar to the numerical model verified with experimental study in the literature [14]. In this study, the overlap length ($b$) was selected as 25 and 40 mm, the impact energy level ($E$) was determined as 6 J and the adhesive defect rates were implemented as 0%, 46% and 70%. The holding pressure at the ends of the joint contacting the apparatus was applied as 20 bar. The upper holding part was allowed to move only along y-axis direction while the bottom holding part was fixed in all directions (Figure 1).

![Figure 1. Boundary and loading situation of single-lap joint](image)

The semi-cylindrical impactor with a radius of 5 mm was modeled as rigid. The regions of 0.02 mm thickness that contact with the adherend interfaces of the adhesive were modeled using the cohesive zone model. In addition, cohesive behavior interaction was implemented between the adhesive layers. Aluminum adherend materials and the middle region of the adhesive were modeled using the elastic-plastic material model (Figure 2).

![Figure 2. Numerical modeling approach of adhesive and adherends](image)
The stress-strain curve values of Al 5754-0 and Araldite® 2015 adhesive were obtained from the literature [9,14,16]. The cohesive zone model parameters were used to determine the initiation and propagation of damage in the adhesive interface region [17,18]. The damage initiation in the cohesive region was determined by the maximum nominal stress criterion. The damage propagation of adhesive was determined using the failure displacement value of 0.065 mm obtained from the MMF (mixed mode flexure) test of the Araldite® 2015 adhesive [17]. In the middle region of the adhesive, the failure strain of 0.17 and absolute failure displacement of 0.065 mm were used for determining the damage initiation and propagation, respectively [9,17]. The finite element mesh distribution of single-lap joint was constructed with bias command to decrease the mesh density from the impact region to the edges. The impactor was modeled with R3D4 element. The aluminum adherends and the adhesive middle region were modeled using the C3D8R element, while the adhesive interface region was modeled using COH3D8 element type.

3. NUMERICAL RESULTS

The contact force-time and contact force-displacement graphs of single-lap joints under low-speed impact loading are presented in Figure 3. The contact force-time and contact force-displacement curves have a similar tendency for joints having overlap length of 25 mm with 0% and 46% defect. A similar trend was observed in the joints with a length of 40 mm. When the defect rate increased to 70%, the overlap length of 25 mm was insufficient for the applied impact energy. The single-lap joint with 25 mm overlap length was completely separated for the 70% defect rate, while the overlap length of 40 mm was able to carry the applied impact energy level.

![Figure 3. Contact force-time and contact force-displacement diagrams of adhesively single-lap joints](image)

When the overlap length is increased from 25 mm to 40 mm, the total bonding area is increased. Although the defect rate level (70%) is the same for both joints, the total remaining bonding area in the joints is different. Therefore, the total amount of bonding area is important for the joints to carry the applied impact energy. The maximum contact force value occurred at 70% bonding defect for the joint with an overlap length of 40 mm. This is due to the fact that the internal resistance in the local transition zones that have defect affects the maximum contact force value. Total contact time and total displacement values were close to each other, although the defect rate increased. However, the total contact time and displacement decreased due to the complete separation of the joint with overlap length of 25 mm and the defect rate of 70%. Figure 4 illustrates the method of determining the permanent central deflection and axial separation length of single-lap joints.

![Figure 4. Illustration of permanent central deflection (δ) and axial separation length (Δ)](image)

![Figure 4. Illustration of permanent central deflection (δ) and axial separation length (Δ)](image)
The permanent central deflection and axial separation values for the different defect rates and overlap lengths of the single-lap joints are presented in Figure 5. If the defect rate increased from 0% to 46% for a single-lap joint with an overlap length of 25 mm, an increase in the axial separation value of 2.6 mm was achieved. When the defect rate increased to 70%, the joint was completely separated. When the defect rate increased from 0% to 70% for overlap length of 40 mm, only 7 mm increase in axial separation value occurred. Since the increasing overlap length increases the total adhesion area, it can carry the impact energy applied even if the defect rate reaches very high levels (70%). The central permanent displacement value for the joint with a length of 25 mm decreased from 15.7 mm to 7.2 mm in case the defect rate increased to 70%. For the joint length of 40 mm, the central permanent displacement value increased by only 1.8 mm. Figure 6 shows the initiation and propagation of damage in single-lap joints having overlap length of 25 mm. Adhesive damage is rated with a damage index ranging from 0 to 1. When the adhesive is completely damaged, the damage index reaches 1 (gray color). 0 (blue color) indicates the area that is not damaged. Adhesion damage considerably increased between 4.5-8 ms for a non-defective joint. The damage has progressed slightly slower at later intervals. There was an increase in the damage progress in the area where the defect was found for the joint with 46% defect. If the defect ratio reaches 70%, the damage progressed from the upper interface to the other adhesive layers at 5.1 ms. Adhesive damages have evolved from the outer edge of the adhesive layer to the middle region. It was found that the increase in the rate of defect decreased the time of adhesive damage initiation and increased the damage progression. The damage showed more balanced progression when the length of overlap was 40 mm due to increased total bonding area (Figure 7). The increase of adhesive damage progression was observed when the damage reached the defect zone similar to the joints having overlap length of 25 mm. When the defect ratio value was 70%, the damage initiated at the top interface and progressed along all adhesive layers. Figure 8 shows the time-dependent variation of axial separation length (Δ) of single-lap joints for each defect rate. The increase of the defect rate of joints with an overlap length of 25 mm increased the rate of damage progression. The damage progressed very rapidly between 5-10 ms and then progressed more balanced for the defect rates of 0% and 46%. The damage progression increased very rapidly until complete separation occurred for the defect rate of 70%. In addition, the damage initiation time was close to each other for each defect rate. The damage progression rate was close to each other in joints with a defect ratio of 0% and 46% for joints with an overlap length of 40 mm. In case the defect ratio reached 70%, it was determined that the damage progression rate increased at a time interval of 0.5 ms and then increased as balanced. In this study, it was determined that the total bonding area was effective in the behavior of the joints under the impact load. Total interfacial adhesive areas of joints having an overlap length of 25 mm and defect rates of 0%, 46% and 70% are 625 mm², 336 mm² and 184 mm², respectively. In addition, the total adhesive areas at the same defect rates for overlap length of 40 mm are 1000 mm², 541 mm² and 296.5 mm², respectively. The joints with a 336 mm² bonding area were able to carry the impact load of 6 J without being completely separated, while the 184 mm² bonding area was completely separated. In addition, the axial separation value at the joint with the 625 mm² bonding area was close to the separation damage at the joint with the 336 mm² bonding area. The bonding area of 296.5 mm² was sufficient to carry the impact load without completely separated for a joint with an overlap length of 40 mm. The determination of the optimum interfacial bonding area was found to be important both in terms of safety and cost. In addition to numerical analysis, it is necessary to determine the optimum bonding area by experimental study. In this way, both the effect of the adhesion defect on the behavior of joint and the determination of the optimum bonding area can be made more reliably. The results of the numerical analysis revealed that the defects in the adhesive shortened the damage initiation and increased the damage propagation. It has also been found that the increase in defect rate changes the progress of the damage between the adhesive layers. Because of these effects, taking measures to reduce the defect rate in adhesively joint design is very important for joint strength.
Figure 6. Damage initiation and propagation throughout adhesive layers in single-lap joints (h=25 mm)
4. CONCLUSIONS

In this study, the low speed impact behavior of single-lap joints with different defect rates (0%, 46% and 70%) were investigated. In addition, the effect of defect rates on the initiation and propagation of adhesive damage was examined. Four main conclusions were obtained from numerical analysis results.

(1) The increase in adhesive defects reduced the impact strength of the joints. If the total adhesive area value is below the critical value, the joint is completely separated under impact load. It is important to design the adhesively joints by determining the optimum bonding area.

(2) Total interfacial adhesive areas of joints can be increased by the increase in the overlap length. In this way, the negative effect of adhesive defect on mechanical properties of joint can be reduced.

(3) The cohesive zone model has been used reasonably to determine the damage initiation and propagation throughout adhesive layers in single-lap joint.

(4) Adhesion damage progressed from the outer side edges to the middle zone. The damage progression generally occurred along the top adhesive interface. However, the damage occurred between all adhesive layers for the
joint having overlap length of 40 mm and defect rate of 70%. This showed that the defect rate affects the initiation and propagation of adhesive damage.

ACKNOWLEDGEMENTS

This study was partially supported by Scientific Research Project Division of Erciyes University under the contract (FYL-2014-5262).

REFERENCES


INVESTIGATION OF THE EFFECT OF NANOREFRIGERANTS ON
PERFORMANCE OF THE VAPOR COMPRESSION REFRIGERATION CYCLE:
A REVIEW STUDY

HudaElaslam Abdali Salem MOHAMED, Kemal BİLEN*, Kayhan DAĞDIR, Erol ARCAKLIOĞLU
Ankara Yıldırım Beyazıt University, Mechanical Engineering Department, Ankara
*Corresponding author: kemal.bilen@ybu.edu.tr

ABSTRACT
In this study, effect of usage of nanorefrigerants being a kind of nanofluids have been investigated in Vapor
Compression Refrigeration System (VCRS). Recent developments, particularly, about experimental studies
associated with the nanorefrigerants have been handled. Nanorefrigerant technology is a new application used in
the general refrigeration systems especially in the VCRS to improve the thermal efficiency as a result of improved
thermophysical properties of the working fluid. Therefore, studies in this issue have become important.
Nanorefrigerants are the mixture of a pure refrigerant and nanoparticles. Most studies show that these mixtures
improve performance of the refrigeration systems. Experimental studies show that addition of various nanoparticle
results in different outcomes and different concentrations of the same particle additive have also different effect
on the performance of the VCRS. For example, effects of nanoparticle addition to thermophysical properties, the
increment in the thermal conductivity is 3\% and 15\% with 1\% and 5\% volumetric concentration, the increment in
the density is 2.48\% and 12.41\% with 1\% and 5\% volumetric concentration, for TiO\textsubscript{2} with R141b at 323 K. Another effect also is in
some important system characteristics, for instance, increasing the volumetric concentration of Al\textsubscript{2}O\textsubscript{3} with R134a
0.01\% and 0.02\%, the heat transfer coefficient increases by 0.54\% to 1.1\%, the Coefficient Of Performance (COP)
increases by 3.33\% to 12\%, the power consumption by compressor decreased by 1.6\% and 3.3\%, respectively.

Keywords: Nanoparticle, Thermophysical properties, Nanorefrigerants, VCRS, COP.

1. INTRODUCTION
The first refrigerant, Ethyl Ether, was used in 1834. After that several traditional refrigerants were applied such as
ammonia, CO\textsubscript{2}, hydro-carbons etc. In 1930, Chloroflorocarbons (CFCs) were used as called Freons and later
Hydrocloroflorocarbons (HCFCs) were introduced mostly in air conditioning systems. CFCs are excellent, stable
and friendly to the environmental refrigerants. However, CFCs have an important effect on the ozone (O\textsubscript{3}) layer
(namely Ozone Depletion Potential: ODP). This was the reason for phasing them out by Montreal Protocol in
1987. Moreover, CFCs have a very high Global Warming Potential (GWP). In 1987, several companies started to
replace CFCs to HCFC-22 or mixtures in the air conditioning systems because they have low ODP about 0.034.
In addition, the most widely used HCFC is R-22 having a value of 1780 for GWP and its lifetime is 12 years.
There is no information about percentage of CFCs in the total effect on global warming, but, in 1990, it was
believed that the share was approximately 15\%. However, for their chemical stability still plays a major role
compared with Hydroflorocarbons (HFCs). According to the Montreal Protocol, usage of CFCs would have been
finished by 2010 in the developing countries. Then, usage of HCFCs started and it will have last until 2040 in the
developing countries [1].

Refrigeration which can be defined as artificial cold covers a wide range of applications such as food processing,
preservation and transport, comfort cooling, commercial and industrial air conditioning, manufacturing, energy
production, health, recreation, etc. The first known machine to produce continuous cold was invented by the French
man Ferdinand Carré in 1859. This was the earliest version of ‘aqua ammonia’ absorption system. However,
commercially successful compression refrigeration systems working with ammonia were introduced in 1875.
Since then, the refrigeration technology has been grown tremendously, influencing almost all aspects of human
life [2]. Refrigeration systems can be broadly classified into two categories: 1) Steady-state refrigeration systems
in which the cooling effect is continuous, the refrigerant flow is steady and in one direction. 2) Periodic
refrigeration systems in which the cooling effect is cyclic or intermittent, the refrigerant flow varies periodically
with time and is bidirectional. On the other hand, the performance of steady-state refrigeration systems depends on the properties of the working fluid refrigerant used. Refrigeration systems can also be classified depending on the type of energy input into the following [2]:

- Mechanical energy (compression refrigerators),
- Thermal energy (absorption/adsorption refrigerators),
- Electrical energy (thermo-electric refrigerators),
- Magnetic energy (magnetic refrigerators),
- Acoustic energy (acoustic refrigerators), and,
- Light energy (optical refrigerators).

Different types of refrigeration systems with a large fraction (typically about 80%) of practical refrigerators are of vapor compression type and operate with mechanical energy input. In most cases the mechanical energy is derived from electric motors. Vapor absorption refrigerators that operate with heat input are the second most widely used refrigeration systems [2].

Vapor compression refrigeration systems (VCRSs) are responsible for about 30% of the total world energy consumption and this amount can be increased when the refrigerant leakage [3]. The vapor compression refrigeration system consists of a compressor, a condenser, a capillary tube as expansion device, and an evaporator. Refrigerant vapor is compressed in the compressor with low temperature and low pressure to high temperature and high pressure vapor that flows into the condenser and is changed into a subcooled liquid by rejecting heat to the outside of refrigeration system then the refrigerant flows into the expansion device and is expanded to low temperature and low pressure after that flows into the evaporator. At the end of these processes the refrigerant in the evaporator absorbs heat from outside of the system and flows into the compressor again [4]. Utilization vapor compression technology consumes a large amount of electricity to solve this problem must improve thermal performance of the vapor compression refrigeration system by the following: Thermal performance of vapor compression refrigeration system is called Coefficient Of Performance (COP) that is the ratio of cooling load to the network input. To increase the COP of the, cooling load capacity (refrigeration effect) must increase or work that given to the VCRS must reduce [5].

Refrigeration system always is a major sector in energy consumption in most of the countries around the world [6]. At the beginning of this decade the development of synthetic refrigerants has been started to prevent the increasing global warming effect as the result the refrigerants with high GWP started to be limited in many of application [7]. Unsaturated HFC refrigerants that are known as hydrofluoric olefins (HFO) refrigerants are must promising alternative to HFC refrigerants because of low GWP, zero ODPs, and comparative performance. However, several issues prevent usage of those refrigerants such as flammability, high cost and lack of information about their properties [7].

2. LITERATURE REVIEW

2.1 General Literature Review

The researchers reported that the use of nano-refrigerants could decrease the energy consumption of domestic refrigerators by 26% [6]. Nano-refrigerants are a new kind of refrigerants that consist of nano-meter sized materials such as nano-fiber, nano-tube, nano-wires, nano-sheet, nano-rods or droplets in a base refrigerant. Nano-refrigerants have been used to improve thermophysical properties such as thermal conductivity, density, dynamic viscosity, and convective heat transfer coefficients compared with those of base refrigerant as R134a, R407c, R404, etc. [8]. To use nano-refrigerant in the refrigeration system, the basic information about thermophysical properties of nano-refrigerant must be known [6]. There are three main advantages of using nano-particles in refrigerants as follow: 1) They can enhance the solubility between the refrigerant and lubricant. 2) They can improve the thermal conductivity and heat transfer of the refrigerant. 3) They can reduce the friction coefficient and wear rate when nano-particles are dispersed into the lubricant [9].

Synthesis of nano-particles is the first step for the preparation of nano-refrigerant. There are three different methods to synthesis nano-refrigerants. However, two-step method is mainly used for this purpose. Nano-particles can be synthesized using various approaches including chemical, physical, and biological approaches. Two-step method is accepted by the major portion of the researchers as it is simple and economic. Also, most of the metal oxides
based on nano-refrigerants are produced by two-step method because of its better stability than one-step method [10].

The challenge of this study is to make homogeneous and long-term stable nano-refrigerant with negligible agglomeration (accumulation the particles beside each other) and without affecting the thermophysical properties. The long-term stability of nano-refrigerant is a key issue for many applications. It can be mentioned that different nano-particles need their own stability method. Sometimes, these methods have to combine together while in other cases just one method would be adequate to obtain the preferred stability. To date the long-term stability of most studied nano-refrigerant is not confirmed and more basic theoretical and experimental work is required for improving the stability of nano-refrigerant. Hybrid nano-refrigerant has shown promising improvement in thermophysical properties and further can be carried out in the future [10].

The basic and most important step during the study with nano-refrigerants is its preparation process as it affects the stability and properties of nano-refrigerants. Preparation of nano-refrigerants requires some special methods for uniform and stable solution and less agglomeration and sedimentation problem. Nano-refrigerants are produced by dispersing metal, metal oxides, non-metals of nano-sized in the base refrigerant. There are mainly two methods for preparation of nano-refrigerants: One-step method and two-step method. These can be done by using the chemical method or mechanical methods [11]. There is also a third method named the vacuum SANSS method.

There are two main methods in the manufacturing of nano-materials. The ‘bottom up’ approach relies on growth and self-assembly of single atoms and molecules to form nanostructures. This approach is very powerful in creating identical structures with atomic precision. The ‘top down’ approach, on the other hand, relies on breaking down large-scale material to generate required nano-structures from them. Nano-particle manufacturing can be separated into six different categories [12]:

- Gas phase processes, including flame pyrolysis, high-temperature evaporation, and plasma synthesis,
- Chemical Vapor Deposition (CVD),
- Colloidal or liquid phase methods,
- Mechanical processes including grinding, milling, and alloying,
- Atomic and molecular beam epitaxy,
- Dip pen lithography.

Many researchers have been done on the vapor compression refrigeration system to improve the performance included 1) Using ejector, 2) Selection of appropriate refrigerants, 3) Cycle configuration and operating conditions improvement, 4) Utilizing the wasted heat of high temperature sources in the cycle [13].

Thermodynamic properties of refrigerants are required in the refrigeration process, air conditioning, and heat pump system. When pure refrigerants and refrigerants mixture are used as working fluid in the industrial application then the knowledge of thermophysical properties has importance for the design and fabrication of these devices. However, direct measurement of the thermophysical properties with a wide range of temperatures and pressures is not practical, the last decades a lot of effort has been developed to improvement of methods for estimating thermophysical properties. In order to estimate thermophysical properties of fluid, Equation Of State (EOS) plays a crucial role. Equation of state is needed to represent both volumetric and equilibrium properties of new refrigerants that be used as working fluid. In general, thermodynamic and physical properties such as enthalpy, entropy, vapor pressure and density of refrigerants can be estimated by EOS [14].

Experimental studies depends on selection of a new nano-refrigerant that will be used as a working fluid in the vapor compression refrigeration system to enhance thermal efficiency of the system. Since the mid-1980s the refrigeration and air conditioning sectors have developed to identity and employed refrigerants that have less environmental problem of stratospheric ozone depletion and global warming. For selecting the best refrigerants there are many considerations with COP that is dominant importance once other (e.g., chemical stability or lack of toxicity) have been accepted. It was classified with various methods for predicting the performance of refrigerants operating in vapor compression cycles by Domanski and et al. in 1992. They classified these methods in five categories from theoretical analysis to laboratory equipment testing: 1) Carnot cycle analysis 2) Simple methods based on fundamental observation and principles 3) Theoretical and semi theoretical cycle analysis 4) Detailed equipment simulation models and 5) Laboratory test of vapor compression equipment [15]. The thermophysical properties of nano-refrigerants are calculated by some correlation or measured by some instruments.
In some studies [16], experimental data for the thermal conductivity were compared with the models of Maxwell (1904) and Yu and Choi (2003).

In order to determine thermophysical properties of nano-refrigerants, some mathematical models can be used. The mathematical models include that the effective thermal conductivity of nano-refrigerants depends on the thermal conductivity of both the particles and the base refrigerant and the volume fraction of the mixture. The modified Maxwell model including the effect of nano-layer predicts that the very thin nano-layer has a significant impact on the thermal conductivity of nano-refrigerant, particularly when the particle diameter is less than 10 nm. However, the nano-layer impact is small, and the modified Maxwell formula reduces to the original Maxwell equation in the case of large particles [17].

2.2 A Summary of Studies Related to VCRSs

The vapor compression cycle is largely used in domestic, commercial, and industrial refrigeration systems including air conditioning systems. These systems present high energy consumption, thus reduction of energy consumption is a major concern in vapor compression refrigeration systems. For reduction of energy consumption, efficient system must be used. The most of the VCRS is running on halogenated refrigerants because of their excellent thermodynamic properties also thermophysical properties. Moreover the low price, however the international protocols (Montreal and Kyoto protocols) restrict the use of halogenated refrigerants in vapor compression refrigeration systems and the use of chlorofluorocarbon was completely stopped in most of countries. However, hydrochlorofluorocarbon refrigerants can be used until 2040 in developing countries and they should be phased out by 2030 in undeveloped countries. Many developing countries still use R134a (HFC) in refrigeration devices because of low cost and excellent thermodynamic properties as well as thermophysical properties. But, the leakage of HFC refrigerants makes contribution to the global warming [18].

Table 1 illustrates a set of previous studies which have been applied on the vapor compression refrigeration system that uses the nano-refrigerant as the working fluid and it is also indicates the findings which were accessed by the researchers.
Table 1. A summary of previous studies which used nano-refrigerants as working fluid in the vapor compression refrigeration systems (VCRSs) between 2014 and 2018.

<table>
<thead>
<tr>
<th>No</th>
<th>Author(s)</th>
<th>Journal</th>
<th>Base Fluid(s)</th>
<th>Nano-particle(s)</th>
<th>Year</th>
<th>Methods and Findings</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Singh and Lal [19]</td>
<td>International Journal of Research in Mechanical Engineering and Technology</td>
<td>R134a</td>
<td>Al₂O₃</td>
<td>2014</td>
<td>Experimental study, improvement in COP is maximum (7.2 to 8.5%) with 0.5%wt Al₂O₃.</td>
</tr>
<tr>
<td>2</td>
<td>Coumaressin and Palaniradja [20]</td>
<td>International Journal of Advanced Mechanical Engineering</td>
<td>R134a</td>
<td>CuO</td>
<td>2014</td>
<td>Experimental study, evaporator heat transfer coefficient increases with the usage of CuO.</td>
</tr>
<tr>
<td>3</td>
<td>Alawi and Sidik [21]</td>
<td>International Communications in Heat and Mass Transfer</td>
<td>R134a</td>
<td>CuO</td>
<td>2014</td>
<td>Mathematical correlations, thermal conductivity of the nano-refrigerants increases with the increase of nano-particle volume concentrations of 1 to 5% at temperatures of 300 K to 320 K.</td>
</tr>
<tr>
<td>4</td>
<td>Mishra and Jaiswal [22]</td>
<td>International Journal of Advance Research and Innovation</td>
<td>R134a, R404A, R407c</td>
<td>Cu, Al₂O₃, CuO, and TiO₂</td>
<td>2015</td>
<td>Theoretical analysis, using EES software, Al₂O₃/R134a highest COP of 35%. R404A, R407 with different nano-particle enhance in COP about 3-14% and 3-12%, respectively.</td>
</tr>
<tr>
<td>5</td>
<td>Senthilkumara and Praveenb [23]</td>
<td>International Conference on Recent Advancement in Mechanical Engineering &amp; Technology</td>
<td>R600a</td>
<td>CuO</td>
<td>2015</td>
<td>Experimental study, temperature drop in the condenser from 12.37% to 10.88% and a gain of 5.52% and 9.24% in the evaporator. COP improved 1.17%-9.14%.</td>
</tr>
<tr>
<td>6</td>
<td>Kushwaha, Shrivastava, and Shrivastava [24]</td>
<td>International Journal of Mechanical and Production Engineering</td>
<td>R134a</td>
<td>Al₂O₃</td>
<td>2016</td>
<td>TK Solver software, with 0.55 Al₂O₃, 0.55 TiO₂ and 0.6 CuO. The COP of Al₂O₃-R1234yf is 2% higher than other nano-particles and heat transfer is higher with Al₂O₃-R1234yf compared to the others.</td>
</tr>
<tr>
<td>7</td>
<td>Coumaressin, Palaniradja, Sathishkumar, and Mathivanan [25]</td>
<td>International Conference on Breakthrough in Engineering, Science &amp; Technology</td>
<td>R1234yf</td>
<td>Al₂O₃/TiO₂/CuO</td>
<td>2016</td>
<td>The experimental study, heat transfer coefficient enhancement of up to 67.30% relative to the coefficient of R600a/oil mixture without nano-particles.</td>
</tr>
<tr>
<td>8</td>
<td>Akhavan-Behabadi, Torabian, and Nasr [26]</td>
<td>International Conference, Paris, France</td>
<td>R600a/Oil</td>
<td>Multi Wall Carbon Nano-Tubes (MWCNTs)</td>
<td>2016</td>
<td>The experimental study, heat transfer coefficient enhancement of up to 67.30% relative to the coefficient of R600a/oil mixture without nano-particles.</td>
</tr>
<tr>
<td>No.</td>
<td>Authors</td>
<td>Journal/Publication</td>
<td>System Type</td>
<td>Particle Type</td>
<td>Year</td>
<td>Notes</td>
</tr>
<tr>
<td>-----</td>
<td>---------------------------------</td>
<td>--------------------------------------------------------------</td>
<td>--------------</td>
<td>---------------</td>
<td>------</td>
<td>-------</td>
</tr>
<tr>
<td>9</td>
<td>Sharma, Duggal, Dondapati, and Usurumarti [27]</td>
<td>International Conference on Electrical, Electronics, Engineering Trends, Communication, Optimization and Sciences (EEECOS)</td>
<td>R141b</td>
<td>TiO₂</td>
<td>2016</td>
<td>Mathematical correlations, the increment in the thermal conductivity is 3% and 15% with 1% and 5% volume concentration, respectively, at 323 K of TiO₂. Einstein model, the increment in the viscosity is 2.5% and 12.5% with 1% and 5% volume concentration, respectively, at 323 K of TiO₂. Xuan and Roetzel equation, the increment in the density is 2.48% and 12.41% with 1% and 5% volume concentration, respectively, at 323 K of TiO₂. Numerical study, using software FLUENT thermal conductivity increases and specific heat decreases with increase in particle concentration. Viscosity and density increase with increase in volume fraction.</td>
</tr>
<tr>
<td>10</td>
<td>Faizan and Han [28]</td>
<td>International Journal of Engineering and Science (IJES)</td>
<td>R245fa</td>
<td>Al₂O₃</td>
<td>2016</td>
<td>Theoretical analysis, with concentration ZrO₂ from 0.01% to 0.06% with particle size of 20 nm. COP improved with 33.45% when 0.06% of ZrO₂ with R152a and higher than that of R134a and R152a.</td>
</tr>
<tr>
<td>11</td>
<td>Kumar, Baskaran, and Subaramanian [29]</td>
<td>International Journal of Scientific and Research Publications</td>
<td>R152a/R134a</td>
<td>ZrO₂</td>
<td>2016</td>
<td>Experimental study, COP of VCR S is improved 12.08% with Al₂O₃ and improved 21.18% with TiO₂ compare with R134a.</td>
</tr>
<tr>
<td>12</td>
<td>Raghavalu, Reddy, Khan, Kumar, and Prashanth [30]</td>
<td>IJSRD/National Conference on Recent Trends &amp; Innovations in Mechanical Engineering</td>
<td>R134a</td>
<td>Al₂O₃-Ethylene glycol oil, TiO₂-Ethylene glycol oil</td>
<td>2016</td>
<td>Experimental study, the increments in thermal conductivity of spherical and cubic shape ZnO nanoparticles in R134a refrigerant were 25.26% and 42.5%, respectively. Numerical computations, maximum COP was 20% higher by using copper mixed in the brine water and minimum was 15% higher by using TiO₂ with size 0.00001 m mixed in the brine water as without mixing nano-particles. Theoretical analysis, R134a with Cu have the highest Effectiveness Factor (EF) approximately 3.2 at 5%v. The EF increases with increasing in %v.</td>
</tr>
<tr>
<td>13</td>
<td>Maheshwarya, Handab, and Nemadec [31]</td>
<td>International Conference on Processing of Materials, Minerals and Energy</td>
<td>R134a</td>
<td>ZnO</td>
<td>2016</td>
<td>Experimental study, addition of 0.1 %v of CuO to the lubricant enhanced the thermal conductivity of the lubricant by 12.67%. Also, CuO–R134a reduced the compressor work input by 5.2%, increased the freezing capacity of VCRS by 11.1%, and increased COP of the refrigeration system by 7.5%.</td>
</tr>
<tr>
<td>15</td>
<td>Mishra and Jaiswal [33]</td>
<td>International Journal of Research in Engineering and Innovation</td>
<td>R134a</td>
<td>Cu</td>
<td>2017</td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>Sanukrishna, Vishnu, and Jose [34]</td>
<td>Journal of Mechanical Science and Technology</td>
<td>R134a</td>
<td>CuO/Polyalkylene glycol (PAG) oil</td>
<td>2017</td>
<td></td>
</tr>
<tr>
<td>No.</td>
<td>Authors</td>
<td>Journal/Conference</td>
<td>Fluids</td>
<td>Mass Fraction</td>
<td>Year</td>
<td>Notes</td>
</tr>
<tr>
<td>-----</td>
<td>------------------------------</td>
<td>-----------------------------------------------------------------</td>
<td>-------------------------------</td>
<td>---------------</td>
<td>-------</td>
<td>----------------------------------------------------------------------</td>
</tr>
<tr>
<td>17</td>
<td>Pawale, Dhumal, and Kerkal</td>
<td>International Research Journal of Engineering and Technology (IRJET)</td>
<td>R134a</td>
<td>---</td>
<td>Al₂O₃</td>
<td>2017 Experimental study, COP was 4.17 for pure R134a. And with R134a + 0.5% Al₂O₃ and for R134a + 1% Al₂O₃ theoretical COP was 3.75 and 3.54, respectively.</td>
</tr>
<tr>
<td>18</td>
<td>Senthilkumar, Saravanan, and Kumar</td>
<td>IOP Conference Series Material Science and Engineering</td>
<td>R22</td>
<td>---</td>
<td>CuO</td>
<td>2017 Experimental study, theoretical COP of R22 was 16.05 whereas COP of R22 + CuO was 18.72.</td>
</tr>
<tr>
<td>19</td>
<td>Senthilkumar</td>
<td>International Journal of Air-Conditioning and Refrigeration</td>
<td>R134a</td>
<td>---</td>
<td>SiC</td>
<td>2017 Experimental investigations, COP of the system improved 1.17%-8.40%.</td>
</tr>
<tr>
<td>20</td>
<td>Shrivastava and Chhalotre</td>
<td>International Journal Online of Science (IJO)</td>
<td>R134a</td>
<td>---</td>
<td>Al₂O₃</td>
<td>2017 Experimental study, the power consumption reduced by 14.71% and COP increased by 28.93% with 0.06% mass fraction of Al₂O₃.</td>
</tr>
<tr>
<td>21</td>
<td>Mohod and Kale</td>
<td>International Conference, Paris, France</td>
<td>R134a</td>
<td>---</td>
<td>Al₂O₃</td>
<td>2017 Experimental study, increasing the volume concentration of Al₂O₃ 0.01% and 0.02%, the heat transfer coefficient increases by 0.54% to 1.1%. The thermal conductivity increases by 11.5% and 14.2%. The COP increases by 3.33% to 12%. The power consumption by compressor decreased by 1.6% and 3.3%, respectively.</td>
</tr>
<tr>
<td>22</td>
<td>Mahdi, Theeb, and Saed</td>
<td>Journal of Energy and Power Engineering</td>
<td>R134a</td>
<td>---</td>
<td>Al₂O₃</td>
<td>2017 Experimental study, with concentration factor 0.02 to 0.06 g by volume % of silver oxide the heating capacity was improved by 26-82%, respectively. Also the exergy destruction is decreasing in the range of 29-31.28%, 65.77-70.01%, 14.31-16.03%, 17-23% in compressor, condenser, evaporator, and expansion valve, respectively.</td>
</tr>
<tr>
<td>23</td>
<td>Mishra and Kumar</td>
<td>International Journal of Research in Engineering and Innovation (IJREI)</td>
<td>Ethyl glycol with water</td>
<td>---</td>
<td>Silver oxide</td>
<td>2017 Experimental analysis, R600a/oil/CuO increased the frictional pressure drop compared to R600a/oil mixture and by increasing the nano-particle concentration, the frictional pressure drop rises.</td>
</tr>
<tr>
<td>24</td>
<td>Sheikholeslami, Darzi, and Sadoughi</td>
<td>International Journal of Heat and Mass Transfer</td>
<td>R600a/Oil</td>
<td>CuO</td>
<td>2018</td>
<td>Experimental study, larger frictional pressure drop of R600a/POE/CuO compared to R600a/POE and by increasing the nano-particle concentration, the frictional pressure drop increase.</td>
</tr>
<tr>
<td>25</td>
<td>Darzi, Sadoughi, and Sheikholeslami</td>
<td>Physica B: Condensed Matter</td>
<td>R600a with Polyester (POE) oil</td>
<td>CuO</td>
<td>2018</td>
<td></td>
</tr>
</tbody>
</table>
3. CONCLUSIONS AND RECOMMENDATIONS

In conclusion, current studies show that the nanoparticle additive is improving the thermophysical properties of the refrigerant. Nano-refrigerants have the potential to be an alternative working fluid for vapor compression refrigeration systems. Metal oxides have mostly used in studies in this field. Generally good results have been obtained.

R134a is the most commonly used as refrigerant. However, R134a is between the refrigerants to be gradually removed from usage. Studies in this field should focus on alternative refrigerants. In future studies, alternatives should be included in the use of nanoparticles. The allotropes of carbon can be used as alternative nanoparticles. Hybrid studies can also be a good alternative.

To sum up, to develop safely, efficiently and environmentally friendly alternative refrigerants, this field needs more work. It is considered that this field will attract the interest of researchers in the future.

REFERENCES


DESIGN AND APPLICATION OF A NEW SPLITTER PLATE-PRESSURE PROBE FOR WIND TUNNEL FLOW MEASUREMENTS

Kasim Biber

Department of Mechanical Engineering, Faculty of Engineering, Bartin University, Bartin, Turkey
E-mail: kbiber@bartin.edu.tr

ABSTRACT

A new splitter plate pressure probe was designed and built for flow field measurements over a two-element airfoil in the Wichita State University 7 by 10 ft low speed wind tunnel. Wind tunnel tests were conducted at a Reynolds number of 2.2 million and Mach number of 0.13. The splitter plate was made of aluminum and had pressure taps on its one side. The probe can make many measurements in the plane of the plate simultaneously by using these pressure tabs. It was supported by a traversing strut in the wind tunnel and surveyed the flow field on the airfoil upper surface. Measured pressures were presented by isobars of coefficients. These isobars were connected with the pressure coefficients distributed over the airfoil. Results of splitter plate pressure probe compare well with other single-point measuring probes. However, results show that no matter how small the probe, its presence in the flow causes a disturbance, especially in the highly time-dependent separated flow regions.

Keywords: splitter plate-pressure probe, flow field pressures, wind tunnel instrumentation, flow field isobars, multi-element airfoils.

1. INTRODUCTION

Pressure measurements in wind tunnels are of interest not only for determining the pressure distribution on test models but also for determining flow field pressures in the wind tunnel test section. Flow field pressures are commonly measured at a single point and this measurement is repeated until all point measurements at prescribed locations are completed. This kind of point measurement may provide qualitative data when the flow is attached over the test model, but it may not be adequate when the flow is separated from the model surface.

The wind tunnel investigation reported by (Biber, 1991) involved measuring such pressures in separated flows over an airfoil with a slotted flap. One objective from these tests was to provide flow field pressure data to validate computational codes developed on multi-element airfoils. The vast majority of wind tunnel tests on multi-element airfoils had been conducted for the measurements of velocity and turbulence quantities as in (Braden continued 1983). However, for the evaluation of computational codes such as the one reported by (Stevens continued, 1971), there was a need to obtain flow field static pressures. Such pressure data were reported by (Adair and Clifton, 1988), (Nakayama and Kreplin, 1990) and (Wentz and Ostawari, 1983), but they did not include separated flow at stall conditions. Therefore, there was a need to provide experimental pressure data for validating computational codes developed for separated flows over airfoils, as in (Zumwalt and Elangovan, 1982).

Separated flow over airfoils may involve both forward and reverse flow. Its static pressures can be measured at a single point, but this type of measurement is not capable of capturing the time dependent features of separated flow. Therefore, for measuring static pressures in separated flows, a new pressure measuring instrument called splitter plate-pressure probe was developed as part of wind tunnel test program reported by (Biber and Zumwalt, 1992) and (Biber and Zumwalt, 1993). The splitter plate can make many measurements simultaneously and it can capture time averaged-static pressures for separated flow. The present paper first describes the experimental arrangement including design features of the new pressure instrument and then provides its example pressure data as obtained in the wind tunnel.

2. EXPERIMENTAL ARRANGEMENT

2.1 Wind Tunnel

The tests were conducted in the Wichita State University (WSU) 2.13 m x 3.05 m (7 ft x 10 ft) wind tunnel fitted with two inserts, providing a 2.13 m (7 ft) high and 0.914 m (3 m) wide two-dimensional test section. The test model used
was 13% thick GA(W)-2 airfoil with 25% single-slotted flap. It had a reference chord of 0.61 m (2 ft) at flap-nested position and has a cove region where the flap retracts for the single-element case. Details of tunnel, model, supporting end plates and surface pressure taps are given in (Biber and Zumwalt, 1993) and (Wentz and Ostawari, 1983).

Flap positioning was based on previous wind tunnel tests of GA(W)-2 airfoil at WSU, as reported in (Biber, 1991). The selected flap had a deflection of 30 deg from chord line and 3% chord gap without any overlap with the main wing element.

2.2 Test Conditions

Wind tunnel tests were conducted at a Reynolds number based on the reference chord of 2.2 million, Mach number of 0.13 and an indicated dynamic pressure of 1150 N/m² (24 psf). The transition on the main wing was fixed by 2.4 mm wide trip strips at 5% chord upper surface and 10% chord lower surface. The test section turbulence level was 1%, as obtained by a single hot-wire probe.

The flow field two-dimensionality in the test section was evaluated by the visualization of separation patterns on the model upper surface. Tempera and kerosene flow showed a change in the two-dimensional flow character near the side walls with the airfoil stalled. Beyond the stall, a span-wise surface flow component within the separated zone appeared and extended toward the juncture of the model and sidewall, where a vortex pair formed and shed at the model trailing edge. However, this was limited to the outer 25% of the span. No boundary layer control was employed to reduce the effects of three-dimensional flow patterns.

The sidewall boundary layer did not have significant effect on the airfoil stall characteristics. This was probably due to the relatively short upstream length of the sidewalls, 1.06 m from the model leading edge. Besides tempera and kerosene, tufts were attached to the model upper surface to supplement the preceding results and particularly to observe the probe interference with the surface flow. The tufts did not show span-wise flow at pre-stall angles of attack, but at post stall angles, it was not clear if the sideway flapping of tufts was due to three-dimensional feature of separated flows or the probe interference.

Figure 1. Geometry of splitter plate-pressure probe with 42 pressure ports. All dimensions are in millimeters.

3. DEVELOPMENT OF SPLITTER-PLATE PRESSURE PROBE

The new instrument, called splitter plate-pressure probe, was specially designed and built for mapping the flow field static pressures over the airfoil. Figure 1 shows geometric details of the new pressure probe. The idea for this instrument originated from the difficulty of measuring static pressures in the separated wake flows. The conventional pressure probes sense the pressure in the prescribed directions although the flow is multi-directional, especially, in the
The splitter plate was mainly developed for measuring flow field pressures on airfoils. During its application, the major issue was the plate size and its support system. The wind tunnel blockage and three-dimensional effects, especially, at post-stall conditions were considered as major limitations. Tests showed that the plate would flutter to some degree at some high angle of attack.

The splitter plate was supported by a tunnel traversing strut, also used for other flow field surveys in the wind tunnel. The splitter plate can make many measurements simultaneously in the plane of the plate, by the pressure taps on the plate surface. It has an elliptic leading edge and rounded bottom edge for good surface flow quality. The slope of the bottom edge is straight rather than curved to get closer to the airfoil upper surface at all selected angles of attack. The plate was made of aluminum and had 42 pressure taps on its one-side. These taps were connected to stainless steel tubes buried inside the 6.35 mm thick plate. These tubes were extended vertically as much as 508 mm along the traversing strut and then linked to the tunnel pressure transducers externally. The pressure transducers had a resolution of $\pm 2.4 \text{ N/m}^2$ (0.05 psf). The long tubing length would tend to smooth out any pressure fluctuations, so the pressure readings must be regarded as time-average values.

The flow quality on the plate surface was checked by the oil flow visualization with and without the two-dimensional inserts in the empty test section. The oil flow showed a small separation bubble at the leading edge, but the elliptic leading edge minimized the bubble size. Also, some vortices developed on the bottom edge due to its inclination with respect to the main flow. However, these flow features did not indicate a significant change in the pressure data obtained during the plate calibration.

The splitter plate surveyed the flow field at various survey stations above the airfoil model. Starting well above and downstream of the model, the survey was made by moving the plate in 127 mm horizontal (downstream) steps. This made the front two columns of pressure taps overlap the locations the last two columns of the previous position. After moving downstream of the model, the plate was moved forward and lowered, at most 76 mm. Downstream steps were made again. At least 6.35 mm clearance from the model surface was maintained. When sizable pressure differences were sensed at the same position by the plate overlap at two different steps, the data from the innermost taps on the plate were considered to be the most accurate. Otherwise, data from overlapped taps were averaged. Plate flutter limited the portion of the field that could be measured. It occurred at the most extended, that is, the lowest, strut position when in the wake of the model at high angle of attack. The flutter was monitored visually from the tunnel ceiling window and no data were taken when it was present.

A disk probe was also used to measure flow field static pressures at a single point. It is 38 mm diameter and 5 mm thickness with a pressure port on the center of each side. Its circumference was rounded to obtain a better flow quality on its surface. Before it was used, the probe was calibrated to make sure both sides of the probe reads the same static pressures when aligned with the free-stream flow.

The accuracy of splitter plate pressure probe was checked against the the disk probe. Both instruments surveyed the same stagnations, but the disk probe picked data points closer to the model upper surface because of its relatively smaller size. The data obtained from these two instruments agreed reasonably well at the maximum lift conditions. There was however some discrepancy at post stall angles of attack. This appeared at the most downward probe positions where the splitter plate read higher pressures than the disk probe.

### 4. RESULTS AND DISCUSSION

The flow field static pressures on the upper surface of two-element airfoil were measured by the splitter plate pressure probe as describe earlier. During the movement of the splitter plate, there was a gap in the pressure data from the airfoil upper surface. This gap was filled by the data obtained from the disk probe. Even though the data comparison between the splitter plate and disk probe did not show a significant difference, the disk probe was considered to yield better results close to the airfoil. This was due to the possibility of interference between the relatively large size of splitter plate and the boundary layer on the airfoil surface.

Pressure data were presented by coefficient contours of constant pressures or isobars. These contours were connected with the upper surface pressures in the airfoil bounded regions and they were left isolated in the far wake regions. They tend to approach to the airfoil a nearly orthogonal for the attached flows and parallel for the separated flows.
Figures 2 and 3 show the test results for the two-element airfoil with 30 deg flap case. The airfoil stall occurred when the angle of attack reached 12.8 deg. For the stall angle of attack, Figure 2 shows a gradual increase of contour values of pressures as moved downstream. While the most upstream contour has a value of −2.5, it becomes almost 0.0 at the wake of flap trailing edge. This means that the flow accelerated at the wing leading edge decelerates without any major steepness and reaches the free-stream pressure value in relatively short downstream from the flap.

Notice the contour value of −1.5 on the flap leading edge. This indicates the regions of accelerating flow in the gap having a relatively higher negative pressures, as expected. Also, the contours stay nearly orthogonal to the main wing because the flow is mainly attached. However, a small separated region is shown past the flap mid-chord. This is typical for the airfoil at stall angle of attack with 30 deg flap case.

![Figure 2. Flow field static pressure coefficients contours for $\alpha=12.8$ deg, stall angle of attack case.](image)

Splitter plate measurements were also presented for a representative angle of attack at post stall conditions. Figure 3 shows the pressure contour values on airfoil upper surface at 15.5 deg angle of attack. As compared to the previous figure, there are clear changes in the flow field pressure distribution. While the contour value is −2.5 at the forward portion of wing, it becomes −0.5 after the wing trailing edge. For this case, there appears to be a region of constant pressure on flap upper surface, indicating a separated wake flow. Also, notice that there are contours staying nearly parallel on the main wing and flap upper surface. This means that the flow is separated on both wing and flap for the post-stall angle of attack. For this case, there is the contour value of 0.0 further downstream from the flap. This shows that the free-stream pressure is reached further downstream as the angle of attack past the stall.

![Figure 3. Flow field static pressure coefficients contours for $\alpha=15.5$ deg, post-stall angle of attack case.](image)
5. CONCLUSIONS

A new splitter plate pressure probe was developed for a simultaneous measurement of flow field static pressures at 42 points. The new probe was employed for flow field measurements over a two-element airfoil in the Wichita State University 7 by 10 ft low speed wind tunnel. Measured pressures were presented by isobars of coefficients. These isobars were connected with the pressure coefficients distributed over the airfoil. They were nearly orthogonal to the surface in the attached flow, and parallel to the surface in the separated regions. Free-stream pressure is reached at a further downstream location when the angle of attack is increased to the post stall value. Results of splitter plate pressure probe compare well with other single-point measuring probes such as the disk probe. However, results show that no matter how small the probe, its presence in the flow causes a disturbance, especially in the highly time-dependent separated flow regions.

REFERENCES

1-D MODELLING AND SYSTEM OPTIMIZATION OF TRUCK POWERTRAIN SYSTEM USED WITH PTO FRONT DAMPER

Mehmet Onur Genç¹, Ahmet Koray Pehlivan¹ Necmettin Kaya²

¹Valeo Automotive Systems, Bursa, Turkey
²Bursa Uludag University, Mechanical Engineering Department, Bursa, Turkey

¹Corresponding Author: Mehmet Onur Genç, mehmet-onur.genc@valeo.com

ABSTRACT

In parallel to new technological developments, truck powertrain systems need to be more resistive and comply with the more powerful and fuel optimized engine types. Power Take Off (PTO) is the operational machine of the truck which has efficient usage on many commercial areas. PTO usage on truck is expected to increase engine oscillation because of the simultaneous power consume with drive side. This case leads to PTO front damper design for PTO usage on trucks in order to prevent mechanical damages and increase truck comfort.

The aim of this study is to make system optimization with 1D modeling in vehicles using PTO. Modal shapes and vibration characterization were studied on each part of the truck and most effective parameters on system were determined.

Keywords: Torsional front damper, Truck powertrain system, 1-D modelling, Vibration analysis, Modal analysis

1- INTRODUCTION

Heavy vehicle usage and efficiency recently have increased as the technology and commercial activities increase widely in all around the world. In addition to transport activities, heavy vehicles are used on many areas in which the operational equipments are used (Figure 1). Power Take Off (PTO) is the equipment which uses the engine torque at the same time with drive side. Therefore, this case needs to deep investigation for truck transmission system in order to avoid of any damage, breakages on the transmission components. PTO usage also needs optimization for dynamic balance, driving comfort, dynamic durability and safety. Figure 2 shows the sample of front damper which is used for vibration damping on the trucks used with PTO system. Front damper has no engagement/disengagement functions as in clutches, therefore the mission of the damper is just to filter vibration coming from engine. Vibration is the mechanical oscillation movement for mass or system around the equilibrium point. For this reason, it is aimed to decrease vibration to the minimum level.

Figure 1. Vehicles used with PTO system
In literature, some studies have been conducted by using 1-D modelling on the powertrain system. Verdonck et al. (2010) investigated the hybrid vehicle powertrain systems by using 1-D modelling. They used Forward Dynamic Model (FDM) and defined some contrasts for the power system parameters. Genc and Kaya (2018) have investigated the clutch damper stiffness effect on truck driving comfort by using AMESim 1-D modelling software. In the study, simulation and real test results have been compared and correlated. Genc et al. (2018) studied the modal analysis for truck powertrain system and determined related parameters which have most effective and contribute to modal shape of the system. Macor et al. (2017) have investigated hydro-mechanical hybrid powertrain system by using 1-D modelling with AMESim software. In their study, hybrid and traditional mechanical system have been compared in terms of energy savings.

In this study, a 1-D analysis is conducted and optimization with genetic algorithm by using AMESim software was performed. Firstly, serial and parallel modelling of powertrain system used with PTO are performed using AMESim software. Then optimization study is conducted with genetic algorithm and most effective system parameters have been determined.

2- MATERIAL and METHOD

a) 1-D Modelling

1-D modeling is simulating the systems that consist of real physical components using a network approach. Each block on a network correspond to physical elements, for example gears, motors, mass, springs etc. These blocks are joined by lines corresponding to the physical connections. With this approach the physical behavior of system is simulated easily. In this study, serial and parallel options of powertrain with PTO are evaluated and the compared with 1D modelling technique.

Serial Modelling

In serial modelling, PTO front damper and accessories are considered as serial and joined to powertrain system directly after engine. In this system, front damper is joined directly to fake flywheel after engine crankshaft and taken power is distributed drive side and PTO side simultaneously. Drawbacks of this system are the inefficiency on PTO activation in terms of power consumption and resonance overlaps between the components. Figure 3 is the basic example of the serial connection between the powertrain members.
Figure 3. Serial modelling of powertrain system used with PTO

Parallel Modelling

In this system front damper is located separately with drive side and can be controlled without drive side activation. Figure 4 is the basic example for the system which has parallel configuration between the PTO and drive side.

Figure 4. Parallel modelling of powertrain system used with PTO

The advantage of parallel configuration is enabling the PTO system to avoid of any resonance frequency overlap with using desired inertia and stiffness parameters.
b) System Optimization

In this study, genetic algorithm is used for system optimization. Genetic algorithm is an global optimization method for solving optimization problems and is based on natural selection represented with biological evolution. The genetic algorithm modifies a population of individual solutions at each step and produces the children for the next generation. It is selected because it is not trapped into a local optimum. This algorithm is available in AMESim software for optimization. As a design parameters, the variables which have major effect on the system outputs such as vibration are defined with constraints.

![Genetic algorithm flow](image)

**Figure 5. Genetic algorithm flow**

The flowchart of the genetic algorithm is given in Figure 5. Firstly, Initial parameters of algorithm are set to suggested values.

3- ANALYSIS and DISCUSSIONS

a) 1-D Vibration analysis

Simulation were run for both configuration serial and parallel. Key parameters such as components inertia and stiffness value are taken into account as real vehicle condition and taken by truck brand owner. Figure 6 represents the model shape diagram of serial modelling and explains the vibration contribution of each components to the powertrain system based on frequency level. Figure 6 is the example and shows one of the vibration level 19.6038 Hz which is found with eiginvalues vectors by Fast Fourier Transform (FFT). According to results, engine has high vibration (%17) at this frequency and PTO damper has the lowest vibration (%8) level compare to others. However, results can be accepted approximately close to each other.
Figure 6. Modal analyse diagram for serial modelling

Figure 7 represents the modal shape analysis of parallel configuration. Clutch system (part 3) and transmission system (part 6) have the biggest contribution to the vibration system and these results can be countermeasures for to avoid of any NVH (Noise, vibration, harshness) problems on clutch and transmission in design phase.

Figure 7. Modal analyse diagram for parallel modelling

b) System optimization

In this section the outputs of the modelling is evaluated. Parallel modelling is selected as example for the optimization with genetic algorithm. Vibration parameter is selected as output value. Table 1 shows the values of genetic algorithm parameters. Vibration value of PTO damper is selected as objective function. Inertia value of PTO accessories and PTO
damper spring stiffness value are chosen for constraints. In this study, according to request coming from truck owner, the PTO accessories inertia were taken between 1.2 kg.m$^2$ and 2.2 kg.m$^2$ as constrains (Figure 8). Spring stiffness interval is defined by Valeo between 360 Nm/° and 400 Nm/° (Figure 9). PTO damper vibration is accepted objective function and desired to be around 500 rad/s$^2$ (Figure 10). All constrains and objective key parameters were defined in AMESim genetic algorithm optimization toolbox and simulation were run.

**Table 1.** Genetic algorithm parameters used in simulation

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Population Size</td>
<td>100</td>
</tr>
<tr>
<td>Reproduction Ratio (%)</td>
<td>80</td>
</tr>
<tr>
<td>Max.Number of Generation</td>
<td>20</td>
</tr>
<tr>
<td>Mutation Probability (%)</td>
<td>10</td>
</tr>
<tr>
<td>Mutation Amplitude</td>
<td>0.2</td>
</tr>
<tr>
<td>Seed</td>
<td>1</td>
</tr>
</tbody>
</table>

Figure 8 explains the optimization iterations of PTO accessories inertia value in each cycle. According to Figure 8, the system has stabilization at near the 1.5071 kg.m$^2$ in order to have desired vibration level which is called objective value (Figure 10).

Figure 9 is the other constraint and represents the PTO front damper stiffness. It can be obtained from the graph that after the specified iterations optimum damper stiffness value has found 379 Nm/°.
Figure 9. PTO front damper spring stiffness (Constraint value-2)

Figure 10. PTO accessories inertia value optimization iterations (Objective value)

Figure 10 shows the objective parameter variation and represents the desired vibration level within the specified constraints PTO accessories and front damper spring stiffness value.

4- CONCLUSIONS

In this study the powertrain system of truck used with PTO front damper was investigated and efficient system parameters were investigated. In order to have optimum vibration level for PTO front damper, the 1-D modelling analysis has been conducted and system optimization has been performed by using specified constraints PTO accessories inertia and PTO front damper spring stiffness value. This methodology enables to get some countermeasures during design phase prior to production phase in terms of driving comfort and durability for powertrain components and provides time saving and cost. Future study will be the correlation and comparison of real vehicle test and simulation test data.
Acknowledgements
This study is supported by TÜBİTAK (Project No: 3170745) within the scope of the research project.

References
A NEW TRANSONIC AIRFOIL DESIGNED AND OPTIMIZED FOR
A FUTURE MALE UAV CONCEPT

Kasım Biber

Bartın University, Faculty of Engineering, Department of Mechanical Engineering, Bartın, Turkey
e-mail: kbiber@bartin.edu.tr

ABSTRACT
Design, optimization and analysis of a new 16% thick transonic airfoil were completed for given specifications of a conceptual Medium Altitude Long Endurance Unmanned Air Vehicle. Based on notional climb, cruise and maneuver flight conditions of the air vehicle. The airfoil specifications included a range of Reynolds number per foot from 1.7 million to 2.5 million and Mach number from 0.4 to 0.8. Shape optimization in geometry and inverse design modules of the airfoil analysis program MSES were used to design a new 16% thick reference airfoil. The performance of the reference airfoil was then optimized with an objective of minimizing drag coefficient for 7 design points with conflicting requirements in Reynolds and Mach number by using the MSES/LINDOP optimizer. The optimization results in an upward shift of drag bucket in the direction of higher lift coefficient. The endurance parameter has a linear increase within the drag bucket showing the benefit of laminar flow. Both surface pressure distribution and Mach contour plots show that supersonic compression waves on airfoil surface are terminated at Mach 0.78 with a normal shock wave and associated flow separation, which causes not only a decrease of the maximum suction pressure, but also a decrease in lift and increase in drag coefficient. The new optimized airfoil shows robust performance when operating within the specified design constraints.

Keywords: airfoil optimization, airfoil design, transonic airfoils, shock flows, compression waves

1. INTRODUCTION
A medium altitude, long endurance unmanned air vehicle (MALE UAV) has been conceptualized to provide the context of a vehicle within which an airfoil design optimization problem can be formed. The MALE UAV concept is used to generate flight conditions and desired performance characteristics for the airfoil design process. It is considered to have a maximum gross weight of 2500 kg and cruising altitude between 5000 m to 15000 m. There is no official requirement for this class of vehicle, but some notional vehicle performance objectives are formulated so that a new airfoil can be designed and optimized for its main wing.

The vehicle design problem is multi-faceted, as there are several disciplines that should be considered for design and optimization. Disciplines of propulsion, structures, payload, and aerodynamics can all be considered, along with their integration into the overall vehicle system. However, only the aerodynamic discipline is considered under the scope of this study. The objective here is to design an aerodynamically optimal airfoil shape for application in a notional MALE-UAV concept vehicle.

The concept vehicle is assumed to operate for the majority of its flight time in an efficient cruising condition at high subsonic speed, and so one objective would be to maximize the endurance of the vehicle. Maximizing flight time should improve the quality of information collected and could also save mission costs. For this objective, the airfoil design parameter would be minimum drag coefficient at a given cruise lift coefficient, or a range of cruise lift coefficients. The concept vehicle also has an objective to achieve a high transonic top speed. The ability to achieve a high dash speed would aid in vehicle responsiveness, for example, in relocation between areas of interest. A thinner airfoil may be desirable for maximizing top speed, but it could lead to increased structural weight and a heavier vehicle. For various reasons, but primarily for antenna integration, the aircraft may also require relatively thick wings. Therefore, a third objective has been to design an airfoil that has a maximum thickness ratio of 16% chord.
Because of these factors, it is likely that basing the wing designs on existing airfoil sections will provide sub-optimal performance. Thus, it becomes necessary to develop a unique class of airfoils to use as the baseline for further development efforts. The specifications for the new 16% thick airfoil include a range of Reynolds numbers per foot from 1.7 million to 2.5 million and Mach numbers from 0.4 to 0.8. With these given specifications, this paper aims at designing and optimizing a custom transonic airfoil. The long-endurance flight capability requires the aircraft to have maximum aerodynamic efficiency and minimum fuel consumption, as illustrated by their prominence in the Breguet-endurance equations. For this reason, the objective has been to extend the laminar flow over the airfoil as much as possible, leading to a substantial decrease in drag and consequent reductions in fuel consumption and pollution. However, as evidenced from some other research, reported in (Biber and Tilmann, 2004) and (Cella, continued, 2005), and (Drela, 1992), the laminar flow exhibits strong sensitivity to the leading edge sweep angle and to the environment conditions. Therefore, the current investigation focuses on two-dimensional transonic flow over a wing section having no sweep angle and taking full advantage of natural laminar flow extension.

The research required the use of the MSES computational airfoil analysis and design program package. This program package (Drela, 2004) and (Drela, 1996) is capable of design, analysis and optimization of airfoils used for both high lift systems and transonic wings. This paper first describes the computer programs and relevant methods used for the airfoil design and optimization. It then presents the polar performance of a reference airfoil in comparison to the new optimized one. The polar performance includes a graphical presentation of lift, drag and pitching moment sensitivities of the new airfoil to changes in the critical amplification factor and the location of flow transition on the airfoil surface.

2. COMPUTATIONAL TOOLS

The airfoil design effort reported here included the use of a collection of FORTRAN computer programs called the MSES, version 3.12b and its optimizer called LINDOP, version 2.50. The MSES program consists of main and supporting programs, as described in its user’s manual in (Drela, 2004) and (Drela, 1996).

Boundary layer transition in an MSES solution, as in an XFOIL solution, is triggered by either a free transition where $e^{N}$ criterion is met or a forced transition where a trip or the trailing edge is encountered, (Drela and Youngren, 2001).

The $e^{N}$ method is only appropriate for predicting transition in situations where the growth of two-dimensional Tollmien-Schlichting waves via linear instability is the dominant transition-initiating mechanism. The $e^{N}$ method is always active, and free transition can occur upstream of the trip. The $e^{N}$ method has the user-specified parameter ”$N_{cr}$”, which is the log of the amplification factor of the most-amplified frequency which triggers transition. A suitable value of this parameter depends on the ambient disturbance level in which the airfoil operates, and mimics the effect of such disturbances on transition. For the present airfoil design and analysis, the standard $N_{cr}$ value of 9 was used; however, the effect of changing the transition parameter to other values such as 4, 5, 6 and 12 was also investigated.

3. DEVELOPMENT OF NEW TRANSONIC AIRFOIL

The mid-sized UAV is envisioned to have a long endurance at medium altitudes, requiring its wing to be as aerodynamically efficient as possible. The primary requirements for the wing section considered in this study included 16% chord maximum thickness, over 50% chord laminar flow, low pitching moment, and operations at Mach numbers ranging from 0.4 to 0.8 and Reynolds numbers per foot ranging from 1.7 million to 2.5 million. Both Mach and Reynolds numbers are based on free-stream flow conditions. MSES flow parameters were defined for vortex + doublet far-field, isentropic except near shocks and free-transition for either specified angle of attack or Mach number. Reynolds number was specified for viscous analysis, but it was set to 0.0 for inviscid runs.

A compromise was made among all of the requirements to increase the operational Mach number while maintaining airfoil maximum thickness with a high L/D ratio and large drag bucket for given range of Reynolds number. Surface pressure distribution or surface geometry was changed in mixed or inverse design modules of the MSES software to meet the design objectives. The interactive and iterative work resulted in a new 16% thick airfoil considered to be a reference one, as shown in Figure 1, for the mid-size UAV wing.

Once the reference airfoil was determined, the optimization capability of LINDOP driver was implemented to obtain the new transonic airfoil with 16% chord maximum thickness ratio. For the optimization process, global variables and their corresponding fixing constraints were set to 5 for sweeps in angle of attack, 15 for sweeps in Mach number and 20 for making use of LINDOP optimizer. Incremental sweep values in angle of attack and Mach number were provided in a
separate \textit{spec.xxx} file. Geometry deformation modes were represented by some functions describing camber, and upper and lower surface of airfoil. These functions have end points; 0 at the leading edge and 1.0 at the trailing edge, as indicated in the file \textit{modes.xxx}. The particular geometric shapes are implemented in FUNCTION GFUN in the program package.

\begin{center}
\includegraphics[width=0.5\textwidth]{figure1.png}
\end{center}

\textbf{Figure 1.} Geometric comparison of 16\% thick reference airfoil with the optimized one.

The reference airfoil geometry file, \textit{blade.xxx} was used for all 7 design points selected for climb, cruise and maneuver flight conditions. MSET was run with the airfoil geometry file \textit{blade.xxx} to create one \textit{mdat.xxx}, and then the same file was copied for all 7 design points. The \textit{modes.xxx} file was also kept the same to ensure the same airfoil shape during optimization. The \textit{mses.xxx} file had a different extension for each design point with conflicting requirements.

The optimization process was started by first converging \textit{mses.xxx} files for all 7 cases. MSES sets all the geometry mode amplitudes to zero during a calculation and calculates the sensitivities of various quantities such as CL, CD, etc. to the mode displacements. These sensitivities are written out to the unformatted file \textit{sensx.xxx}, which is then read in by LINDOP optimizer. LINDOP reads all input files with some initialization and lists the available operating points and design parameters.

LINDOP in general is used to minimize an objective function $F(X_k)$ with respect to the parameters $X_k$ (see Drela 1996). In this study, the objective function $F$ was the drag coefficient $C_D$ while the parameters $X_k$ were the airfoil geometry deformation modes. One optimization step consisted of the generation of design parameter changes via line minimization in LINDOP, followed by a nonlinear MSES solution calculation. This sub-cycle was executed toward the line minimum. The gradient vector and line-minimization direction vector were generated to find the optimum. The optimization process resulted in a new 16\% thick transonic airfoil, designated as SCR-16 and shown in Figure 1 with a comparison with the reference airfoil.

The optimized airfoil clearly has better performance at transonic flow conditions. In order to investigate the cause of this improvement in design, the transition location of the optimized airfoil was compared with the reference one at $\alpha=-1$ deg, $Re=2.5$ million and $N_{cr}=9$. It was shown that the optimization results in producing a relatively larger extent of laminar flow on the airfoil. The transition occurs at about 0.82c on upper surface and 0.60c on lower surface for the optimized airfoil. However, it gradually moves upstream as the Mach number nears its critical value for given flow conditions.

The critical Mach number is the free-stream Mach at which sonic flow is first achieved on the airfoil surface. It was determined for both reference and optimum airfoils by the method described in (Anderson, 2001). For the given airfoil, a minimum value of pressure coefficient was obtained for incompressible flow by running the MSES program. The pressure coefficient was corrected by using the Prandtl-Glauert rule and plotted against the free-stream Mach number. Another curve was obtained by the variation of critical pressure coefficient with Mach number. The intersection of these two curves represents the point corresponding to sonic flow at the minimum pressure location on the airfoil. The value of free stream Mach at this intersection is, by definition, the critical Mach number. The critical Mach number has a value of 0.710 for the reference airfoil and it moves to 0.724 as a result of LINDOP optimization of the airfoil.

One of the main objectives of transonic airfoil design is to be able to increase the critical Mach number so as to obtain the highest possible drag divergence Mach. This is the Mach number for the onset of the dramatic increase in wave drag at a given angle of attack or lift coefficient, for a given maximum thickness ratio. With this objective in mind, the MSES code was run with the Mach sweep option, and the airfoil drag was monitored.
Figure 2. Drag divergence Mach comparison of reference airfoil (solid line) with the optimum one (dotted line).

Figure 3. Polar comparison of reference airfoil with the optimized one at Mach 0.4

Figure 2 shows a drag divergence plot comparing the optimized airfoil with the reference one at an angle of attack of $-1$ deg and a Reynolds number of 2.5 million. The figure has a variation of total drag and its components such as friction, pressure and wave drag coefficients with Mach number. With the airfoil optimization, there is clearly a reduction in drag coefficient, which is more significant for the pressure component during the Mach divergence. The start of wave drag is realized at Mach 0.74 for the reference airfoil, and it moves to Mach 0.75 with the optimized one. After this Mach, the drag coefficient starts diverging progressively from its profile value due to the increased compressibility effects. It is desirable to have the smallest possible initial rate of drag increase beyond the drag divergence Mach because the best cruise performance is obtained at a Mach number of 0.02–0.03 in excess of drag divergence Mach, (Torenbeek, 1982).

Figure 3 shows polar performance data comparing the reference airfoil with the optimized one at Re=2.5 million and $N_c=9$. The comparison was made at Mach 0.4. The airfoil initially has a drag bucket with a lift coefficient range of as much as 0.2. This bucket is shifted upward in the direction of higher lift coefficient with LINDOP optimization. The new optimized airfoil has an upper corner of drag bucket operating at a relatively higher lift coefficient. This shift of polar performance is also seen on the upper surface location of transition producing an increase in the extent of laminar flow.

4. POLAR PERFORMANCE OF NEW TRANSONIC AIRFOIL

The polar performance of new airfoil includes a graphical presentation of lift, drag, pitching moment characteristics in sweeps of angle of attack and Mach number. However, the flow behavior at each angle of attack can be better explained with surface pressure distributions over the airfoil.

Figure 4 shows lift, drag and pitching moment characteristics for the new optimized airfoil at Re =2.5 million and $N_c=9$. For a given angle of attack, the coefficient data is easily determined. The transition location and endurance parameter can also be found at a specific lift coefficient. At the upper corner of drag bucket, the extent of laminar flow starts decreasing on the airfoil’s upper surface while it almost stays constant on its lower surface, shown with a dashed
line. The endurance parameter \( \left( \frac{C^L}{C_D} \right) \) on the other hand increases linearly within the drag bucket and reaches to a maximum value at the upper corner of bucket. There is in fact a further increase in the endurance parameter until \( C_L = 0.8 \).

**Figure 4.** Coefficients of lift, drag and pitching moment along with transition location and endurance parameter at \( Re = 2.5 \) million, Mach 0.6 and transition-free.

Figure 5 shows surface pressure distributions for Mach numbers of 0.4, 0.6, 0.70, 0.74, 0.76, 0.77, 0.78, 0.79 and 0.80. There are negative pressures with distinct suction peaks on both upper and lower surfaces. At Mach numbers below Mach 0.75, the upper surface pressure has a gradual increase of negative pressures over the forward part until a point where the laminar separation bubble has its maximum thickness, followed by a pressure rise to the trailing edge. This is similar to the shape of roof-type pressure distributions, which delays critical Mach number by virtue of a uniform velocity at the design condition. As the Mach number increases beyond 0.75, there is a dramatic change of surface pressures in the separation bubble region. The suction level basically moves to its maximum value at Mach 0.78. This is probably due to the thickening of the boundary layer by the presence of locally supersonic flow embedded in the subsonic outer flow.

**Figure 5.** Surface pressure distributions for Mach 0.40, 0.60, 0.70, 0.74, 0.76, 0.77, 0.78, 0.79 and 0.80, showing the effects of supercritical speeds at \( \alpha = -1 \) deg, \( Re = 2.5 \) million and \( N_{cr} = 9 \).

The supersonic flow extends over a region in which there is near isentropic flow and shock-free compression waves. However, at Mach numbers above 0.78, the supersonic region is terminated by a normal shock and associated flow separation, which causes not only a decrease of the maximum suction pressure, but also a decrease in lift and increase in drag coefficient, as shown in the data tabulation of Figure 5. Notice also the decrease in magnitude of pitching moment coefficient as the Mach number is raised above the drag divergence Mach.

Figure 6 shows Mach contours to better illustrate the shock formation and associated flow events at the same Mach numbers and flow conditions as the surface pressure distributions presented in Figure 5. The shock formation is clearly visible in the dense region of Mach contours at the location of the separation bubble. Pre-compression waves are initially weak at Mach 0.74, but they get stronger with increasing Mach number and are eventually terminated with a normal shock wave. They separate the laminar boundary layer well ahead of the first shockwave, which impinges on the free-shear layer and reflects as an expansion wave. The so-called shock-induced flow separation, clearly exhibited at Mach 0.78 and beyond, limits the operational range of airfoils designed for transonic vehicle applications.
Figure 6. Mach contours for Mach 0.74, 0.76, 0.78 and 0.80 showing the shock formation on airfoil upper and lower surfaces at $\alpha = -1$ deg, $Re=2.5$ million, $N_{cr}=9$ and transition-free.

5. PERFORMANCE SENSITIVITIES TO THE FLOW TRANSITION

The new transonic airfoil was designed and optimized with conflicting requirements in a range of Reynolds and Mach numbers. Therefore, Reynolds number did not have a significant effect on the airfoil performance. However, given all of the uncertainties in design and analysis of the airfoil, it is important to evaluate the sensitivities of airfoil performance to some other values of critical amplification ratio and fixed transition. During the optimization process, the flow transition was left free on both upper and lower surfaces of the airfoil. A critical amplification ratio of 9 was used for the $e^N$ envelope method.

Figure 7. Variation of drag coefficient with Mach comparing critical amplification ratios of 4, 5, 6, 9 and 12 at $Re = 2.5$ million and $\alpha = -1$ deg.

Figure 7 shows a comparison of drag divergence Mach number obtained for $N_{cr}=9$ with those obtained for $N_{cr}=4, 5, 6$ and 12 cases at $Re = 2.5$ million and $\alpha = -1$ deg. The higher $N_{cr}=12$ infers lower disturbance levels, more typical for UAS aircraft operating at high altitudes. The lower $N_{cr}=4$, on the other hand, corresponds to ground conditions such as in a wind tunnel. However, the $N_{cr}=9$ case clearly produces the lowest drag value among all others because of its use in optimizing the airfoil for minimum drag coefficient.

The effect of transition was investigated on the airfoil performance by fixing the transition at 0.55c on upper surface and 0.35c on lower surface of the airfoil. The selection of transition points was based on the surface pressure distributions. As shown in Figure 5, the selected transition points are approximately at the start of separation bubble on airfoil surfaces. A comparison was made between free and fixed transition cases in terms of their drag divergence at $Re=2.5$ million and $\alpha = -1$ deg. As shown in Figure 8, the free transition clearly produces a lower drag coefficient because it has a larger extend of laminar flow on the airfoil. It also has a higher lift coefficient as indicated in the figure. The drag divergence Mach is about the same for both cases since the critical Mach number does not change significantly with fixing transition.
Figure 8. Variation of drag coefficient with Mach showing the effect of fixing the transition at 0.55c upper and 0.35c lower surfaces, Re = 2.5 million, N\textsubscript{cr} = 9 and \(\alpha\) = –1 deg.

The effect of fixing the transition was also investigated on the formation of shock waves in the region of drag divergence. Figure 9 shows the Mach contours for the fixed transition at 0.55c upper and 0.35c lower surfaces of the airfoil. Free-stream Mach numbers are 0.74, 0.76, 0.78 and 0.80, the same as in Figure 6 shown for the free transition case. Comparing the two transition cases, the normal shock wave is shown to move slightly upstream by fixing the transition. It is however still formed at a downstream location of transition while the flow is turbulent for both upper and lower surfaces. The location of normal shock does not seem to change significantly on the airfoil, whether the flow is laminar with free transition or turbulent with fixed transition. This shows a robustness in flow character of the new airfoil design. The new airfoil basically performs well as specified within its design constraints and shows the robustness with shock location.

Figure 9. Mach contours for Mach 0.74, 0.76, 0.78 and 0.80 showing the shock formation at \(\alpha\) = –1 deg, Re=2.5 million, N\textsubscript{cr} = 9 and transition fixed at 0.55c upper and 0.35c lower surface of the airfoil.

6. CONCLUSIONS

Design, optimization and analysis of a new 16% thick transonic airfoil were completed for given specifications of a notional MALE UAV concept. A collection of computer programs called MSES, version 3.12b, was used. Based on the climb, cruise and maneuver flight conditions of the air vehicle, the airfoil was specified for a range of Reynolds numbers per foot from 1.7 million to 2.5 million and Mach numbers from 0.4 to 0.8. The wing section considered for the new airfoil is two-dimensional with no sweep angle. For the design and optimization process, the transition was left free with N\textsubscript{cr} = 9.

The research included designing a new 16% thick reference airfoil with shape optimization in geometry and inverse design modules of the MSES program. The performance of the reference airfoil was optimized for 7 design points with conflicting requirements in Reynolds and Mach number by using LINDOP optimizer. For the free transition case at Reynolds number per foot of 2.5 million, the optimized airfoil has the following design features:

1) The flow transition occurs at approximately 80% chord upper surface and 60% chord lower surface of airfoil, taking full advantage of laminar flow as intended.
2) The optimization results in an increase of critical Mach number. While the critical Mach is 0.710 for the reference airfoil, it moves to 0.725 for the optimized one.

3) Drag divergence Mach plot shows a summation of friction, pressure and wave drag components. For the plotted data, the start of wave drag occurs at Mach 0.74 for the reference airfoil and it moves to Mach 0.75 for the optimized one.

4) The optimization also results in an upward shift of drag bucket in the direction of higher lift coefficient. The new optimized airfoil has an upper corner of drag bucket operating at a relatively higher lift coefficient.

5) The endurance parameter, $C_L^{1.5}/C_D$, shows an almost linear increase within the region of drag bucket, indicating the benefit of having a laminar flow airfoil.

6) Increasing Mach number causes the drag bucket to move upward in the direction of both higher lift and higher drag coefficients.

7) Surface pressure distribution plots show that the supersonic compression waves on the airfoil surface are terminated at Mach 0.78 with a normal shock wave and associated flow separation, which cause not only a decrease of the maximum suction pressure, but also a decrease in lift and increase in drag coefficient. This result is supplemented by Mach contour plots provided at some free stream Mach numbers.

The airfoil was optimized for a range of Reynolds numbers and therefore its performance does not change significantly with alterations in Reynolds number. However, given all of the uncertainties in design and analysis of the airfoil, it is important to evaluate the sensitivities of airfoil performance to some other flow conditions. A drag divergence Mach plot shows that when the critical amplification ratio, $N_{cr}$, is set to a different value from its standard 9, the corresponding drag coefficient increases, indicating the sensitivity of airfoil performance to the flow environment. Flow transition on airfoil was fixed approximately at the start of separation bubbles, corresponding to a transition location of 55% chord upper and 35% chord lower surfaces. Fixing the transition shows an increase of drag coefficient as expected, but does change the drag divergence Mach. Mach contour plots show that the location of normal shock on airfoil surfaces does not change significantly by fixing the transition, indicating the robustness of airfoil performance with shock location.

ACKNOWLEDGEMENTS

I wish to acknowledge the Scientific and Technological Research Council of Turkey for funding this research for a 3-month period in the Summer of 2017.

REFERENCES


CALCULATION OF PROPELLER-SLIPSTREAM DRAG AND ITS APPLICATION ON AN AIRPLANE PERFORMANCE

Kasım Biber

Bartın University, Faculty of Engineering, Department of Mechanical Engineering, Bartın, Turkey
e-mail: kbiber@bartin.edu.tr

ABSTRACT
An innovative method of calculating the slipstream drag of a propeller and accounting this method on the performance predictions of a single-propeller airplane is presented. The example airplane is equipped with a twin-pack engine driven by a tractor propeller. The analysis includes the determination of parasite and induced drag increments due to the effect of spinning a propeller, and corresponding reduction on propeller efficiency for various blade angles. It is shown that the amount of correction on propeller efficiency is relatively high at low speeds and high pitch angles, corresponding to relatively high engine power settings. The highest effect of propeller slipstream is shown to occur on the airplane performance during the second segment climb with one-engine inoperative power rating.

Keywords: slipstream drag, propeller airplane, airplane performance, propeller efficiency

1. INTRODUCTION
Propeller slipstream has seen much attention over the years because of its effect on both stability and control and performance characteristics of airplanes (Biber, 2011) and (Biber, 2006). For the stability and control estimations, the spiral air due to the rotational mass of propeller and its effect on forces and moments acting on downstream elements of airplane immersed in the propeller slipstream has to be considered. For performance calculations though, it is of particularly importance because of its effect on drag. If the propeller were running isolated, the dynamic pressure behind the propeller would be greater than the free stream for a delivery of positive thrust. As the fuselage is placed immediately behind the propeller, the slipstream becomes displaced and this in turn causes a relative increase in dynamic pressure over the fuselage. The higher dynamic pressure results in a reduction for the propeller efficiency or an increase for the drag, and this would mean more power required from the engine to drive the airplane.

During preliminary design of single engine airplanes, the slipstream drag is commonly either accounted as 7-10% increase (Hoerner, 1965) or calculated using some approximations (Finck, 1978). However, these methods are not directly sensitive to variations in propeller efficiency or engine power. The slipstream effects are, in fact, more pronounced at maximum take-off power and low speed conditions. Therefore, there is a need to introduce a new method of estimating slipstream drag. This becomes more apparent as the diameter ratio of fuselage to the propeller increases to 0.65, as is the case for the single propeller cargo airplane described in (Biber, 2006). The method presented in this paper was in fact used within the performance calculations of that same airplane.

2. DESCRIPTION OF THE EXAMPLE AIRPLANE
The example airplane considered here is a high-wing cargo airplane equipped with a twin-pack engine driven by a single propeller through a unique combining gearbox, as described in (Biber, 2011) and (Biber, 2006). Its reference wing area is 458 ft². The airplane is in FAR23 commuter category with 19,000 lbs of maximum take-off weight. Its twin-pack engine is rated at 2700 HP take-off setting, which is halved for one-engine inoperative condition. The propeller used is made of composite with six blades and has a diameter of 12.9 ft. Its activity factor is 76 and design lift coefficient is 0.42. The propeller has 1200 RPM at take-off setting, and its blade aerodynamic data is provided for performance calculations. For given power and airspeed, these calculations have typical output parameters such as propeller blade angle and efficiency along with power and thrust coefficients.
3. METHOD OF CALCULATING PROPELLER SLIPSTREAM DRAG

This section presents the method of estimating propeller slipstream drag due to the effect of spinning a propeller as in a previous paper, (Biber, 2011). The slipstream drag is accounted as an increment in linear lift range, and it consists of parasite and induced drag components.

The parasite drag is due to the increment of skin friction caused by a higher local dynamic pressure on all surface area immersed in the propeller slipstream, and it may be estimated by using the following equation, as given in (Finck, 1978)

$$\Delta C_{\text{D}0} = C_F \frac{S_I}{S} \frac{\Delta q_s}{q}$$

where $C_F$ and $q_s$ values are assumed constant over the area wetted by the slipstream. For the example airplane, $C_F$ is taken as 0.009, as suggested in (McCormick, 1995) and the ratio of surface area immersed in the propeller slipstream to the reference wing area, $S_I/S$ is taken as 0.42.

The change in dynamic pressure because of the slipstream can be written in terms of velocity ratios as,

$$\frac{\Delta q_s}{q} = \left( \frac{V_s}{V} \right)^2 - 1$$

Here the velocity of slipstream, $V_s$ is defined as the sum of free stream velocity, $V$ and incremental velocity, $\Delta V$, of ultimate slipstream wake, that is:

$$V_s = V + \Delta V$$

The incremental wake velocity can be defined by using the axial momentum equation written for the propeller thrust, $T$ as;

$$\Delta V = \frac{T}{\rho AV}$$

where $\rho AV$ is in fact the mass flow rate entering the propeller disk area.

![Figure 1. Propeller slipstream velocity $V_s$ compared with induced wake velocity $V_w$ at blade pitch angles of 18, 22 and 26 degrees.](image)

If the propeller were running without airplane parts immersed in its slipstream, the ultimate wake velocity would then be the sum of free stream velocity and twice the propeller-induced or downwash velocity immediately behind the propeller disk, as in (McCormick, 1995).

$$V_w = V + 2w$$

where the induced velocity, $w$ is defined by the equation:

$$w = 0.5 \left( -V + \sqrt{V^2 + \left( \frac{2T}{\rho A} \right)} \right)$$
Within the above equations, both \( V_s \) and \( V_w \) are functions of flight velocity, \( V \) and disk loading, \( T/A \) for a given altitude. These velocities can be non-dimensional as \( V_s/Nd \) and \( V_w/Nd \), and plotted against advance ratio \( V/Nd \) for propeller pitch angle, \( \beta \) = 18, 22 and 26 degrees, as shown in Figure 1. The comparison shows that \( V_s \) has a linear variation with flight speed. \( V_w \) values on the other hand have a curvature with a minimum, higher than \( V_s \) values at low advance ratios. They become essentially the same as they reach a value equal to about \( 1.1V \) (10% higher than \( V \)). The slipstream velocity, \( V_s \), presented here clearly shows more sensitivity at low speeds and high pitch angles where more power is extracted from the engine.

The propeller slipstream modifies the downwash over the portions of the wing and fuselage and changes the corresponding lift dependent drag. This change may be estimated by using the induced drag definition without propeller in which \( C_{D_i} \) is proportional to \( C_{L0}^2 \). In this definition, slipstream effect on drag and lift coefficients can be accounted as increments of \( \Delta C_{D_i} \) and \( \Delta C_L \) in linear range respectively. Ignoring \( \pi A_R \) division on the right side of equation temporarily,

\[
C_{Di} + \Delta C_{Di} \propto \left( C_{L0} + \Delta C_L \right)^2
\]

\[
C_{Di} + \Delta C_{Di} \propto C_{L0}^2 + 2C_{L0}\Delta C_L + \Delta C_L^2
\]

Since \( \Delta C_L^2 \) is much smaller than \( 2C_{L0}\Delta C_L \),

\[
\Delta C_{Di} \propto 2C_{L0}\Delta C_L
\]

Including \( \pi A_R \) term, the equation for the incremental induced drag coefficient becomes,

\[
\Delta C_{Di} = 2\Delta C_L \frac{C_{L0}}{\pi A_R}
\]

where \( C_{L0}/\pi A_R \) corresponds to the induced angle of wing without propeller at lift coefficient \( C_{L0} \), and \( \Delta C_L \) is contributed by the slipstream effect on wing and fuselage characteristics. The change in lift coefficient, is approximately calculated by the following equation as given in Hoerner, 1965,

\[
\Delta C_L = 0.5T_c C_{L0} (b/d)
\]

where \( T_c = T/qA \) thrust disk loading coefficient, and \( A = \pi d^2/4 \) propeller disk area.

![Figure 2. Propeller efficiency variation with advance ratio at \( \beta = 18, 22 \) and 26 degrees](image_url)

The incremental drag due to propeller slipstream may be estimated by multiplying the sum of drag coefficient values by the dynamic pressure of slipstream wake and reference wing area.

\[
\Delta D_s = (\Delta C_{D0} + \Delta C_{Di}) q_s S
\]
The slipstream drag increment results in a decrease with the propeller efficiency and this decrease can be expressed as:

$$\Delta \eta = \frac{\Delta D_s}{P} V_s$$

So, the effective propeller efficiency becomes,

$$\eta_e = \eta_p - \Delta \eta$$

The effect of slipstream drag on propeller efficiency is shown in Figure 2 in variation with advance ratio at propeller pitch angles of 18, 22 and 26 degrees. In this figure, solid lines are for the efficiency $\eta_p$ obtained from propeller performance, and dashed lines for the effective efficiency, $\eta_e$ accounting the effect of slipstream drag. The variation reaches a maximum value and then tends to decrease with increasing advanced ratio. Because of the slipstream drag, there is clearly more reduction in propeller efficiency at relatively low advance ratios and high blade pitch angles.

4. APPLICATION OF THE METHOD ON AN AIRPLANE PERFORMANCE

The slipstream drag estimation was performed in a subroutine of a computer program developed to predict the performance of example airplane. The computer program can either call the engine and propeller performance programs directly or use the maps of data obtained from the power plant. It combines power plant performance with the airframe drag polar. The drag polar was obtained from wind tunnel test of power-off airplane model and corrected for flaps up, down 15 and 33 deg flight conditions. The calculated slipstream drag increment was added to the total drag for the overall performance predictions. Spreadsheet hand calculations were used to check the program accuracy for flight segments at a particular weight, altitude, and temperature conditions.

Figure 3 shows the effects of slipstream drag on take-off field length of the example airplane. The data is presented for both engines operational case at critical flight conditions; the altitude of 5000 ft, temperature of ISA+25°C, and flaps down 15 degrees. The take off distance is calculated for ground, rotation and climb to clear 35 ft screen height. The solid line shows the field length without accounting the slipstream drag ($\Delta D_s=0$) and dashed lines with accounting the slipstream drag ($\Delta D_s$ estimated). The field length clearly increases with the inclusion of the slipstream drag, and this is more pronounced for the distance from the lift-off to the obstacle height.

![Figure 3](image-url)

**Figure 3.** Variation of take-off field length with airplane weight with and without including the propeller slipstream drag. Take-off power rating, altitude 5000 ft, temperature ISA+25°C and flaps down 15 deg.

The take-off field length requirement becomes more critical for one-engine inoperative case. This is illustrated in Figure 4 for the variation of second segment climb gradient with take-off weight. The data is given at critical flight conditions, the altitude of 5400 ft and temperature of ISA+25°C. According to FAA FAR 23 regulations, the airplane operation at this flight condition has a climb gradient of 2%. Accounting the slipstream drag, as shown with dashed line ($\Delta D_s$ estimated), the airplane is clearly projected to meet and exceed the climb requirement with its maximum take-off gross weight of 19000 lbs.

The effects of accounting slipstream drag on other flight segments for typical performance parameters were investigated. The results are tabulated in Table 1 for maximum take-off gross weight at standard day temperature of ISA and hot day temperature of ISA+25C. Flaps are considered down 15 deg for take-off and climb and up for cruise condition at 10000 ft altitude.
Figure 4. Variation of 2nd segment climb gradient with take-off gross weight with and without slipstream effects. OEI engine rating, altitude 5400 ft, temperature ISA+25C and flaps down 15 deg.

Table 1. Effect of slipstream drag on typical performance parameters of example airplane in standard and hot day temperature.

<table>
<thead>
<tr>
<th>OAT</th>
<th>ΔDₛ=0.0</th>
<th>ΔDₛ estimated</th>
<th>change, %</th>
</tr>
</thead>
<tbody>
<tr>
<td>ISA</td>
<td>1506</td>
<td>1590</td>
<td>5.28</td>
</tr>
<tr>
<td>ISA+25C</td>
<td>1912</td>
<td>2007</td>
<td>4.73</td>
</tr>
<tr>
<td>OEI 2nd segment climb gradient (rad) at 5400 ft</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>ISA</td>
<td>0.062</td>
<td>0.055</td>
<td>−11.29</td>
</tr>
<tr>
<td>ISA+25C</td>
<td>0.035</td>
<td>0.030</td>
<td>−14.28</td>
</tr>
<tr>
<td>ISA</td>
<td>2065</td>
<td>1907</td>
<td>−7.62</td>
</tr>
<tr>
<td>ISA+25C</td>
<td>1836</td>
<td>1695</td>
<td>−7.66</td>
</tr>
</tbody>
</table>

The second segment climb gradient with OEI power is the most sensitive of all parameters to the addition of slipstream drag. It results in a decrease of 14.3% at ISA+25C condition. This is primarily because the climb gradient is calculated from the difference of thrust and drag and divided by the airplane weight. When the engine thrust is halved with OEI power, the effect of changes on drag becomes more influential on the performance parameter.

Rate of climb with both engines is also sensitive to the slipstream drag with 7.6% decrease, as expected because of its direct dependence on the airplane drag. Take-off distance with both engines operational has an increase of 4.73% with the slipstream drag. The effect of slipstream on cruise performance is not significant as indicated with only 1% decrease on specific range. The performance data clearly has more sensitivity to the increase of slipstream drag in flight segments where the airplane has the climb attitude in relatively low speed conditions.

5. CONCLUSION

An innovative method of calculating the slipstream drag of a propeller and accounting this method on the performance predictions of an example airplane is presented. The method includes calculating parasite and induced drag components that primarily result from the increase of dynamic pressure in the propeller slipstream. The incremental drag due to slipstream of running propeller is shown to decrease with air speed and increase with propeller pitch angles. Propeller efficiency was corrected to account for the slipstream drag at various blade pitch angles. The amount of correction appears relatively high at low speed and high pitch angle conditions, which corresponds to relatively high engine power settings.
REFERENCES


HYSTERESIS EFFECTS ON WIND TUNNEL TESTS OF AIRFOILS AND WINGS

Kasim Biber
Bartin University, Faculty of Engineering, Department of Mechanical Engineering, Bartin, Turkey
E-mail: kbiber@bartin.edu.tr

ABSTRACT
The stall hysteresis on aerodynamic characteristics of airfoils and wings tested in low speed wind tunnels is reviewed briefly. The hysteresis was commonly known to occur on wind tunnel tests of single-element airfoils operating at low Reynolds numbers. However, it had been shown by the author that the stall hysteresis occurs on wind tunnel tests of an airfoil with a slotted flap at relatively high Reynolds number. Recent wind tunnel tests reported elsewhere in the literature show that the hysteresis effect can also present on the performance of three-dimensional wing and even possibly on airplane models when they are tested in low speed wind tunnel. In this paper, an update on aerodynamics hysteresis is given and then examples of wind tunnel data are presented to show the effects of hysteresis on two- and three-dimensional wing models.

Keywords: stall hysteresis, hysteresis effect, separation bubble, airfoil, wing, wind tunnel testing

1. INTRODUCTION
Aerodynamic hysteresis, as can be demonstrated in a wind tunnel, is a phenomenon where the stall inception and stall recovery do not occur at the same angle of attack, (Biber, 2005). If present, it can result in severe control problems in stall. The hysteresis has been commonly observed for some single airfoils operating at Reynolds numbers below 0.7 million, as shown by (Mueller, 1985) and (Marchman, 1987). At low Reynolds number, there is usually a laminar separation bubble that characterizes the flow over airfoils. The bubble contracts in size and trips the boundary layer into turbulent flow with increasing angle of attack. It bursts at stall and this causes an abrupt fall in lift coefficient. Angle of attack may have to be reduced significantly for an attached flow on upper surface. In the process, lift coefficient is not restored, causing a reduction in maximum lift coefficient. The aerodynamic hysteresis is sensitive to changes in wind tunnel flow quality and acoustics, as in (Marchman, 1987). It decreases in size and eventually disappears with Reynolds number.

The aerodynamic hysteresis was shown to occur on the wind tunnel performance of an airfoil with a slotted flap operating at relatively high Reynolds number of 2.2 million, as in (Biber and Zumwalt, 1993). The hysteresis on the flapped airfoil also occurred in the stall region and caused in average 20% drop in lift coefficient. Its size was shown to vary as function of flap angle and gap. Similar results are also reported by (Landman and Britcher, 2001) in wind tunnel tests of a three-element airfoil but at relatively low Reynolds number of 1 million. The stall hysteresis was recently observed by (Broeren continued, 2017) on wind tunnel tests of a reflection plane type wing configuration at a test Reynolds number of 1.6 million.

The hysteresis appears to be an off-design phenomenon and occurs only if the airfoil angle of attack is lowered from its post-stall range. A proper airfoil design should provide a desired performance of not only at design but also at off-design conditions. This requires a well understanding of hysteresis effects on airfoil characteristics. In this paper, the hysteresis effects have been reviewed briefly by evaluating some low speed wind tunnel data available in the literature.

2. HYSTERESIS EFFECTS ON WING MODELS
Hysteresis effects are most often confind to airfoils operating at low Reynolds numbers, as reported by (Mueller, 1985) at University of Notre Dame and (Marchman, 1987) at Virginia Tech University. Results as in (Machmann and Abtahi, 1985) show the effects of Reynolds number on the hysteresis loop for an aspect ratio 8 wing with Wortmann FX-63-137 airfoil. These tests were conducted at Virginia Tech University Stability Wind Tunnel having a very low turbulence...
level of 0.02%. The case with the lowest Reynolds number of 70000 shows a relatively lower lift curve slope and a flat-plate stall at 6 deg angle of attack. At a Reynolds number of 100000 or above, the airfoil reaches its classical linear regime. At this Reynolds number, there is an abrupt leading edge stall on the increasing angle of attack side. As the Reynolds number is increased to first 200000 and then 300000, the stall becomes more and more gradual and the hysteresis loop moves to the right in the positive angle of attack direction having the same maximum lift coefficient. At higher values of Reynolds number near 500000, the hysteresis loop is expected to be a much more post-stall phenomenon. Further increases in Reynolds number, close to the airfoil design value, the hysteresis loop may eventually disappear from the lift curve.

Results as in (Sumantran continued, 1985) show the turbulence effects on the Wortmann model wing with aspect ratio 8 at Reynolds number of 200000. Increase of turbulence level from 0.02% to 0.2% reduces the hysteresis loop significantly without changing the maximum lift and stall angle at Reynolds number of 200000. Acoustic disturbance also affects the hysteresis loop similar to that of turbulence. That is, the hysteresis loop can be reduced and even be eliminated from the airfoil performance at its lower Reynolds number range with increase in turbulence or acoustic noise.

Results as in (Marchmann continued, 1985) show the variation of lift curve with the aspect ratio for a rectangular wing of Wortmann airfoil section at Reynolds number of 200000. It is apparent that the lift curve slope and maximum lift increase with increases in aspect ratio from 4 to 8, but there is little or no further increase from 8 to 10. In general, the hysteresis loop decreases in size and moves to the left in the negative angle of attack direction with increases in aspect ratio.

The stall hysteresis phenomenon was also observed by (Boeren continued, 2017) for a wing mounted vertically on a tunnel floor, the so-called reflection plane type wing configuration. These tests were conducted in Wichita State University 7 by 10 ft low speed wind tunnel at a Reynolds number of 1.6 million and Mach number of 0.18. Performance data were acquired for angle of attack sweeps in which the angle of attack was increased from –5.8 to 17.1 deg, followed by decreasing angles to 0.6 deg.

The hysteresis loop was shown on lift and drag coefficient data as in (Boeren continued, 2017). These data exhibit a classic stall hysteresis loop where the flow on the wing is not fully recovered until the angle of attack is decreased from about 14 deg to less than 12.1 deg. For these tests, the hysteresis was not however observed at a lower Reynolds number of 0.8 million and Mach number of 0.09. The lower Reynolds number was obtained by lowering the tunnel test speed. The Wichita State University wind tunnel operates at near atmospheric pressure. Therefore, the test Reynolds number and Mach number in the tunnel test section can not be controlled independently. Reynolds number effect would be better investigated by using a pressurising wind tunnel.

3. HYSTERESIS EFFECTS ON MULTI-ELEMENT AIRFOILS

Hysteresis effects appeared on wind tunnel tests of 13% thicker GA(W)-2 airfoil with 25% slotted flap, as reported by (Biber and Zumwalt, 1993). The tests were conducted in Wichita State University 7 by 10 ft low speed wind tunnel equipped with 7 by 3 ft dimensional inserts at Reynolds number of 2.2 million and Mach number of 0.13. The airfoil model had a chord of 2 ft and span of 3 ft. The tunnel speed was raised to the test value while the model at zero angle of attack. For the hysteresis tests, the angle of attack (α) had its increasing and then decreasing sweeps within its prescribed range. Figure 1(a) shows some example wind tunnel data from these tests.

Figure 1(a) shows lift and pitching moment coefficients for flap-nested airfoil case at both increasing and decreasing angle of attack. The airfoil has a maximum lift coefficient of 1.7 and pitching moment coefficient, $C_m = -0.0759$ at 16 deg angle of attack. It has a lift coefficient of 0.4 and $C_m = -0.1$ at $α = 0$ deg. Fixed transition allowed 95% of the airfoil upper surface to have turbulent boundary-layer flow. The turbulent flow apparently did not have any bubble-induced stall hysteresis.

Figure 1(b), on the other hand shows lift and pitching moment coefficient data for 30 deg flap with optimum flap. For this case, there is clearly a hysteresis loop in the stall region. Because of this stall hysteresis, the airfoil looses 20% of its maximum lift coefficient of 3.21 with an abrupt stall after $α = 13.58$ deg. Lift coefficient appears to stay constant at 2.57, as the angle of attack changes from 14.05 to 15.55 deg, before another stall. This peculiar behavior in lift coefficient data was repeatable and better understood when the angle of attack was lowered back after a few degrees above the second stall. When $α$ passes below 14 deg, the lift coefficient is not restored to its original value, but instead exhibits a significant loss continuing until $α = 6.95$ deg. The lift curve appears to be shifted downward between $α = 6.95$ and 14.05 deg, indicating the effect of hysteresis. The pitching moment coefficient data, in contrast, shift upward (becomes less negative) in the decreasing side of hysteresis loop.
a) Absence of hysteresis on single-element airfoil case  

b) Hysteresis loop on two-element airfoil case

Figure 1. Coefficients of lift and pitching moment for the GA(W)-2 airfoil with 25% slotted flap at Reynolds number of 2.2 million. 30 deg flap with optimum gap case, (Biber, 2005).

Knowing the range of hysteresis loop from force and moment data, surface pressure coefficient (Cp) distributions were obtained to study the flowfield on airfoil elements. The measurements were made at settings of angle of attack determined by α limits and the maximum lift coefficient condition of hysteresis loop in lift curves.

For the 30-deg flap with optimum gap, the Cp distribution was obtained at α = 7, 8, 12, 12.8, 14, and 15.5 deg as shown in Figure 2. The data for an increasing α side are shown in Figure 2 (a) and for a decreasing α side in Figure 2(b). Comparing these Figures, there is a clear discrepancy seen between the increasing and decreasing α sides (7 and 14 deg) of the hysteresis loop. The highest discrepancy occurs at α = 12.8 deg, which is near the maximum lift coefficient. The suction Cp values on the main wing are lower for the decreasing side because the flap apparently does not provide the necessary downwash, and consequently the circulation for the main wing, in contrast to the case for the increasing angle of attack side.

Figure 2. Surface pressure coefficient distribution at α = 7, 8, 12, 12.8, 14, and 15.5 deg for 30-deg flap with optimum gap with increasing and decreasing α, (Biber, 2005).

For both sides of the hysteresis loop, suction Cp on the main wing shows a dramatic fall after its peak value near the leading edge. The dramatic fall is followed by a short step between the 5% and 7.5% chord locations, where the transition is fixed on the upper surface. The trip strips apparently produced a short bubble to trip the flow. This was also evident from surface flow visualizations. At a low α setting, the aft portion of flap has a nearly constant Cp distribution. The start of constant Cp is considered to be the location for flow separation, as evidenced in correlation to the flow visualization studies. The region of separated flow on flap decreases with increasing α settings and almost disappears at α = 12.8 deg before the stall. This is considered to be a typical feature of flaps designed to maximize the maximum lift coefficient of the airfoil system. At poststall α = 14 deg, suction Cp values on the flap are flattened, indicating a leading-edge stall. The flatness on the flap moves upstream onto the main wing, and, at a higher α = 15.5 deg, it leads to a
massive flow separation. When $\alpha$ is lowered back from 14 deg, the flow on the flap remains stalled until $\alpha = 7$ deg, where the airfoil lift is restored to its value obtained with increasing $\alpha$. In other words, the suppressed $C_p$ values on the flap under the influence of the thickening wing wake after the first stall are evidently not returned to their prestall values until a significant reduction in $\alpha$ occurs.

4. CONCLUSIONS

Hysteresis effects on wind tunnel performance of airfoils and wings are reviewed briefly, based on some wind tunnel test data available in the literature. The review was limited to the examining the hysteresis effects on airfoil and wing models. The common feature of hysteresis is that it occurs in the stall region when the angle of attack is lowered back from its post-stall value. The maximum lift coefficient obtained while increasing the angle of attack is not recovered during the decreasing angle of attack side. As observed from flow visualization studies, tripping the laminar flow to turbulent character is not reformed once the flow is fully separated from the upper surface of test model, at least, not until the angle of attack is lowered significantly. In another words, once the flow is separated while increasing the angle of attack, it stays separated for the decreasing side within the range of hysteresis loop. The laminar separation bubble is apparently not reformed, causing the hysteresis and therefore this may also be considered as an irreversible flow phenomenon. The size of the irreversibility depends on factors such as the Reynolds number, model aspect ratio, turbulence level in test section and contamination level in flow. Whether or not the hysteresis is formed on the other hand seems to depend on how abrupt the lift coefficient falls during the stall. The larger the fall in lift coefficient, the more chance that the hysteresis loop appears to occur.

REFERENCES


STUDY OF PERMEABILITY AND HYDRAULIC CONDUCTIVITY OF 3D PRINTED PLASTER PARTS BY BINDER JETTING

Edwin Ocaña-Garzón1,3, Leonardo Santana2,3, Jorge Lino Alves3

1 Departamento de Ciencias de la Energía y Mecánica, Universidad de las Fuerzas Armadas–ESPE, Sangolquí, Ecuador
2 International CNPq fellow, Brazil
3 Inegi, Faculty of Engineering, University of Porto, 4200-465, Porto, Portugal
emocaniaabc@gmail.com, leonardosantana29@gmail.com, falves@fe.up.pt

ABSTRACT

In the post process of infiltration in the Binder Jetting technology, the permeability ($k$) and hydraulic conductivity ($K$) are essential factors in the penetration of the infiltrant within the three-dimensional printing (3DP) model and thus get improve its mechanical strength. This study experimentally evaluated the influence of 3DP sample thickness, infiltrant type, build orientation, binder saturation and colour of model 3DP on its permeability and hydraulic conductivity. Permeability and hydraulic conductivity were determined based on Darcy's law in a custom-made falling head permeameter with low and high viscosity mineral oils that simulated the infiltrants. In addition, porosity and the pore size distribution (PSD) were estimated using the liquid saturation and mercury intrusion porosimetry (MIP) methods. In decreasing order, the build orientation and binder saturation of the sample showed a statistical significance for $k$, whereas for $K$ the infiltrant type and build orientation were significant. The cumulative volume curves of intruded/extruded mercury to the 3DP model showed liquid entrapment, this could be associated with kinetic effects during mercury extrusion, coupled with the tortuosity and surface chemistry of the pores in the porous network. More than 90% of the pores detected are macropores ($> 0.05 \mu m$), however the predominance in the absorption of liquid controls the pores between 10 to 30 $\mu m$.

Keywords: permeability, hydraulic conductivity, porous material, 3D printing, plaster

1. INTRODUCTION

Extensive studies of permeability have been conducted mainly in the fields of geology, petroleum and bioengineering, leaving only some researches in the field of three-dimensional printing (3DP) (Dias et al. 2012, Yan et al. 2017, Mitra et al. 2018). In this area, binder jetting technology requires post-processes as infiltration, being the permeability ($k$) and the hydraulic conductivity ($K$) influencing factors in the penetration of the infiltrant inside the 3DP model, thus improving its mechanical strength (Latt and Giao 2017).

Permeability is the most important physical property of a porous medium, which describes the conductivity of a porous medium with respect to fluid flow and it depends on the combination of porosity, pores size, orientation, tortuosity and interconnectivity (Dias et al. 2012). The hydraulic conductivity is a combined property of the medium and the fluid, expressing the ease with which a fluid is transported through a porous matrix (Bear 2013). The porosity is its most important geometrical property, it is a measure of the fluid storage capacity of a porous material (Pal et al. 2006).

Given the importance of the infiltration phenomena, the present study proposes to assess experimentally the influence of some process parameters on the permeability and hydraulic conductivity of plaster-based 3DP models by simulating the infiltration process with infiltrants of different viscosities.

2. MATERIALS AND METHODS

To experimentally evaluate the influence on $k$ and $K$ of the infiltrated 3DP model, an experimental design (DOE) was performed where five factors that varied in two levels (low and high) were selected. These factors are: sample thickness (A), infiltrant type (B), build orientation (C), binder saturation (D) and colour sample (E), as shown in Table 1.

<table>
<thead>
<tr>
<th>Levels</th>
<th>A: sample thickness (mm)</th>
<th>B: Infiltrant type</th>
<th>C: Build orientation</th>
<th>D: Binder saturation</th>
<th>E: Colour</th>
</tr>
</thead>
<tbody>
<tr>
<td>Low</td>
<td>3.2</td>
<td>Lv</td>
<td>Do</td>
<td>Ds</td>
<td>Cl</td>
</tr>
<tr>
<td>High</td>
<td>9</td>
<td>Hv</td>
<td>Vo</td>
<td>Hs</td>
<td>nCl</td>
</tr>
</tbody>
</table>

The sample thicknesses were 3.2 and 9.0 mm. To simulate a real infiltrant two mineral oils were selected, with low "$Lv$" and high viscosity "$Hv$". Two build print orientations were selected, the default orientation "$Do$", where the software
places the model inside the 3D printer chamber to print it in a shorter time, and a vertical orientation "Vo", with the model aligned its largest dimension along the Z-axis of the printing chamber (see Figure 1). The saturation level for the binder/volume ratio of 3D printing was established at two levels: Default saturation "Ds" (0.24 for the shell and 0.12 for the core), and high saturation "Hs" (0.24 for all volume of the printed solid). Finally, the influence of printing models with color binder "Cl" and colorless "nCl" was analysed (using a clear binder).

Two batches of sixteen cylindrical samples of 30 mm diameter were printed with plaster-based powder (VisiJet® PXL Core, 3D Systems) and water based binder solution (VisiJet® PXL, 3DSystems) in a projet 660 Pro 3D printer, (see Figure 1). A layer thickness of 0.01 mm was setting for all samples. After printing, the models were dried in the 3D printer's build chamber for 2 hours at 55°C, after which the specimens were first depowdered. Finally, samples were dried in a forced air convection oven SLW53 STD (PolEko, Poland) at 60 °C for 24 h. With the same construction procedure, two batches of cylindrical samples of 8 mm diameter and 20 mm high were printed, for the porosimetry test.

Permeability and hydraulic conductivity were determined in a custom-made falling head permeameter (Figure 2) with low and high viscosity mineral oils (Motul 7100 4t and Renolit EP2) that simulated the infiltrants.

![Figure 1. Layout of the 3DP samples in the 3D printer build chamber](image1)

![Figure 2. a) Custom-made rigid wall falling head permeameter and b) its scheme](image2)

The hydraulic conductivity and permeability constants were established according to ASTM 5856 (2007) and based on Darcy’s law equation (1) and (2), respectively (Darcy 1856), where, \( a \) (m²) is the cross-sectional area of the standpipe, \( L \) (m) is the length of sample, \( A \) (m²) is the cross-sectional area of specimen, \( t \) (s) is the time for fluid drop from \( h_0 \) to \( h_1 \)
(cm), (initial and final fluid level), \( \mu \) (Pa·s) is the fluid dynamic viscosity (or absolute), \( \rho \) (kg/m\(^3\)) is the fluid density and \( g \) (m/s\(^2\)) is the gravity acceleration.

\[
K = \frac{aL}{At} \ln \left(\frac{h_0}{h_1}\right)
\]

\[
k = K \left(\frac{\mu}{\rho g}\right)
\]

The apparent porosity (\( \phi_o \)) of the samples was estimated by liquid saturation and immersion technique, by saturating a sample (Chen et al. 2002), according to standard ASTM C20. Samples were weighted in dry state \( (m_d) \), using a scale (Mettler Toledo H31AR, d=0.1mg, Switzerland), then they were immersed in 150 ml bath of oil under 4 bar of vacuum for 3 to 5 minutes, to ensure the filling of the voids with fluid. Then, the samples weight, suspended in an immersion liquid \( (m_{sub}) \) were measured. Samples were then removed and dabbed with paper towels to clean excess liquid and weighted to determine their saturated weight \( (m_s) \). The apparent porosity \( (\phi_o) \) was calculated from Eq. (3) (Gallé 2001, Berger 2010, Xia and Sanjayan 2016).

\[
\phi_o = \frac{m_s - m_d}{m_s - m_{sub}}
\]

Additionally, porosity \( (\phi_{Hg}) \) and pore size distribution (PSD) of macropores were estimated for the most representative samples of the permeability test by the mercury intrusion porosimetry method (MIP), in a Micromeritics AutoPore IV 9500 Mercury porosimeter (Micromeritics Instrument, USA), with pressure range of 0.34 to 228 MPa (0.50 to 33000 psia). Scanning Electron Microscopy (SEM, Quanta 400, USA) was used to analyse the porous morphology of cross sections of the brittle fractured sample.

3. RESULTS AND DISCUSSIONS

Using the average time of the three tests in each condition, it was obtained for each combination the hydraulic conductivity \( (K) \), permeability \( (k) \) and apparent porosity \( (\phi_o) \). An analysis of variance (ANOVA) was carried out with a 95% confidence interval to validate the real significance of each factor on the responses \( K \) and \( k \). Figure 3 shows the means with their standard errors from ANOVA of hydraulic conductivity and permeability for all analysed factors and levels, organized from right to left according to their highest contribution. The ones with a statistically significant contribution \( (P_{value} < 0.10) \) are highlighted in the yellow box. The horizontal line plotted represents the mean of means for all factors.

The significant factors in decreasing order, for hydraulic conductivity are:

- Infiltrant type \( (B) \), build orientation \( (C) \), and their interactions \( BxC \), all of these with \( P=0.000 \)

The significant factors in decreasing order for permeability are:

- Build orientation \( (C) \), \( P=0.002 \), binder saturation \( (D) \), \( P=0.089 \), and their interactions \( AxC \), \( P=0.007 \) and \( AxD \), \( P=0.094 \).

![Figure 3. Pareto ANOVA diagram to identify the fitted mean and its standard error of a) hydraulic conductivity \( (K) \), and permeability \( (k) \) of 3DP samples, where significant factors are highlighted in the yellow square](image-url)
For hydraulic conductivity, Figure 3a), the infiltrant type (Factor B) has a substantial difference between the two levels $LV$ and $HV$ denoting the greatest contribution. This factor, despite not being dependent on the porous medium, is the most important factor for the infiltration process.

The build orientation (Factor C), showed the second and first significance for the $K$ and $k$ responses, respectively. The vertical orientation ($Vo$) showed higher values for $K$ and $k$ compared to default orientation ($Do$). This may basically be due to the parallel layers orientation to the flow, do not create barriers that hinder the passage of the fluid, as happened with the layers perpendicular to the flow, as outlined in Figure 4a).

The "binder saturation" factor ($D$) was significant only for the permeability of the 3DP sample. A default saturation level ($Ds$) showed better permeability compared to high saturation level ($Hs$). This could be because when printing a body in default saturation mode, the infiltrant will flow more easily in the zone with less concentration of binder (core), since here the powder grains keep a light linkage.

3D printed samples with the most significant parameters were tested by the MIP technique, whose permeability ($k_{Hg}$) and apparent porosity ($\phi_{Hg}$) results are shown in Table 2, which are contrasted with the techniques described above.

![Cross section of the 3DP samples mounted on the porosimeter. The fluid passes through the sample thickness "L" printed with different a) build orientation, and b) binder saturation. The length of the flow path was outlined (in red line) according to the significance found for these factors.](image)

**Figure 4.** Cross section of the 3DP samples mounted on the porosimeter. The fluid passes through the sample thickness "L" printed with different a) build orientation, and b) binder saturation. The length of the flow path was outlined (in red line) according to the significance found for these factors.

**Table 2.** Permeability ($k_{Hg}$), and apparent porosity ($\phi_{Hg}$) for samples, measured by Mercury Intrusion Porosimetry method, and previously described methods

<table>
<thead>
<tr>
<th>Run</th>
<th>Factors</th>
<th>$k$ ($\mu m^2$)</th>
<th>$\phi$ (%)</th>
<th>$k_{Hg}$ ($\mu m^2$)</th>
<th>$\phi_{Hg}$ (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>3.2 Do</td>
<td>1.04</td>
<td>47.68</td>
<td>0.40</td>
<td>46.99</td>
</tr>
<tr>
<td>3</td>
<td>3.2 Vo</td>
<td>2.93</td>
<td>40.41</td>
<td>0.31</td>
<td>45.93</td>
</tr>
<tr>
<td>13</td>
<td>9 Do</td>
<td>1.24</td>
<td>40.48</td>
<td>0.18</td>
<td>49.37</td>
</tr>
<tr>
<td>15</td>
<td>9 Vo</td>
<td>2.58</td>
<td>43.05</td>
<td>0.21</td>
<td>45.17</td>
</tr>
</tbody>
</table>

The cumulative volume of intruded/extruded mercury, and the pore size distribution curve were obtained from mercury intrusion technique by applying the Washburn equation with assumed contact angles of 130° depicted in Figure 5. The cumulative volume of intruded/extruded mercury of two consecutive intrusion/extrusion cycles, and the pore size distribution curves are showed as a function of pore diameter. The cumulative volume of intruded/extruded mercury curve clearly reveals the monomodal nature of this 3DP sample, i.e. it consists of macropores between 10 to 30 $\mu m$.

Upon depressurization specially in the first cycle, the extrusion curve does not follow the same path, compared to intrusion, indicating some mercury is permanently retained in the pores (Figure 5, horizontal cyan line, extrusion 1\textsuperscript{st} run). Usually mercury intrusion/extrusion is always accompanied by entrapment. Entrapment would indicate that the rate of mass transfer in and out of the macropores does not appear to be rapid enough to avoid fragmentation of liquid during extrusion (Thommes et al. 2008). The origin of intrusion/extrusion hysteresis and entrapment are different,
whereas hysteresis is caused by intrinsic as well as by structural effects. The entrapment phenomenon seems to be associated with kinetic effects during mercury extrusion, coupled with the tortuosity and surface chemistry of the pores in the porous network. The origin of entrapment is the slowdown factor of the dynamics associated with the fragmentation of the liquid in the void space that makes vapor transport an important part of the extrusion process (Porcheron et al. 2007). As a consequence, samples with small pores, low porosity and highly tortuous nature exhibit larger amounts of entrapment as compared to samples with large pores and high porosity (Porcheron et al. 2007, Thommes et al. 2008). Hence, in principle it should be possible to correlate the entrapment behavior with characteristic transport properties of a 3D printed sample.

The pore size distribution curve showed a distribution with a mean diameter of 11 µm and a mode of 0.018 µm. The minimum pore diameter detected was 0.005 µm, which agrees with the mesopore range detected by this technique (Thommes et al. 2008). More than 90% of the pores detected are macropores (> 0.05 µm), which makes a permeable material.

4. CONCLUSIONS

The present study evaluated experimentally the influence of the build orientation, binder saturation, printing colour and thickness parameters on the permeability and hydraulic conductivity of plaster-based 3D printing samples by simulating the infiltration process with two infiltrants of different viscosities.

The significant factors, in decreasing order for hydraulic conductivity were the infiltrant type and build orientation, while for the permeability were the build orientation and binder saturation.

A more efficient infiltration process of 3DP models (reaching greater depth or less time) will be achieved with a low viscosity infiltrant, if its predominant flow is perpendicular to a model printed in vertical orientation (highest hydraulic conductivities would be achieved). A thin thickness of model, although not fundamental by itself, when joining with these mentioned factors can also improve its hydraulic conductivity.

5. ACKNOWLEDGEMENTS

Authors gratefully acknowledge funding of Project to Universidad de las Fuerzas Armadas ESPE, Ecuador, and SAESCTN-PII&DT/1/2011 co-financed by Programa Operacional Regional do Norte (ON.2 – O Novo Norte), under Quadro de Referência Estratégico Nacional (QREN), through Fundo Europeu de Desenvolvimento Regional (FEDER).
REFERENCES
SAFETY EVALUATION OF A FIBER REINFORCED COMPOSITE WRAPPED STEEL CYLINDER UNDER DYNAMICALLY APPLIED AXIALLY NON-UNIFORM INTERNAL SERVICE PRESSURE DISTRIBUTION

Onur GÜNGÖR, Veli ÇELİK

1 MKE Kurumu Mühimmat Fabrikası Ar-Ge Müdürlüğü 71100 Merkez/Kırıkkale/Türkiye
2 Yıldırım Beyazıt Üniversitesi Doğa Bilimleri Fakültesi 06010 Keçiören/Ankara/Türkiye
a onur.gungor@mkek.gov.tr, b vlc@ybu.edu.tr

ABSTRACT

Developing lightweight guns and weapon systems have been becoming one of the complementary issues to meet logistics needs in order to outmaneuver in a war zone in recent history. The ammunition is energized and spin by the propellant and sets-grooves in the gun barrel which is the most important element of the gun system. Today's technological improvements allow the targets of longer barrel life, upgraded operating and design pressures for gun systems, ammunition and gun barrels to reach extended ranges, improved heat transfer specifications, and lighter weight gun systems to be realized.

The main aim of the study is to evaluate the safety of the steel section of a small arms gun barrel wrapped with fiber reinforced composite material. The results of the study showed that the gained safety factor was between 5% and 20% compared to unwrapped but optimized/minimized wall thickness steel gun barrel. Furthermore, test results showed that diameter shrinkage because of the negative expansion ratio of the composite material was not perpetual as the number of shot increases.

Keywords: Composite wrapped gun barrel, autofrettage, residual stress, weight optimization.

INTRODUCTION

Robust and at the same time lightweight parts are indispensable elements for increasing effectiveness of Weapon Systems which are getting prominent on the battlefield and have been getting important in recent history.

The gun barrel is the most important element of the weapon systems in which the movement of the ammunition is given by gunpowder and groove-sets. It is such an important thing that; Sultan Mehmet the Conqueror used the Sahi gun barrels he had designed himself and made the conquest by destroying the Byzantine walls in Istanbul.

Developments in today's technology make it possible to achieve longer life, higher pressures, and lower logistics burden, as well as mass reduction demands. Composite materials of advanced materials are as durable as steel and are used in the production of many parts with their lightweight characteristics.

Traditional barrel steel is a material with the ability to produce satisfactory residual stress during the autofrettage process. The work of increasing the strength/weight ratio by supporting with composites is considered to be the subject of innovative design works.

The area of the barrel where the composite material is to be applied, the mounting constraints and the autofrettage calculations are taken into consideration. In order to ensure that at least the composite reinforced barrel shows the same as steel barrel performance, composite reinforcement is applied in accordance with the dimensions and other factors of the gun barrel.

Composite material, briefly composite, is a material with at least two elements that work together with each other and have much improved mechanical and physical properties and functions than those of the elements that are considered independently. In practice, many composites consist of a matrix forming the main bulk structure and a reinforcement or filler material which changes the mechanical strength and matrix strength of the matrix and sometimes changes its thermal conductivity and electrical stiffness. This reinforcement usually consists of fibers (e.g. monofilaments, whiskers) as well as particles (e.g., particle reinforcement) or materials with a more complicated shape (e.g., web, strip, laminate). Composites are divided into three main classes according to the matrix phases [1].
Inspection of the autofrettage process, chip removal, effects of the pressure which is the result of the ignition of the propellant in order to propel the ammunition in the barrel and which creates different loading conditions at different points of the gun barrel, wrapping of it by using composite material, and the evaluation of these effects on the durability of the barrel has been the subject of various studies.

Çelik et al. [2] investigated the residual stresses of the autofrettage process on the gun barrel and optimized the dimensions of the draft gun barrel for the maximum residual stresses which carries the loads inside the gun barrel while the projectile accelerates through.

Rhee et al. [3] have investigated the delamination characteristics of multilayer carbon/epoxy composites under hydrostatic pressures ranging from 1 Bar to 3000 Bar. Impact strength increased from 2.11 kJ / m2 to 3.04 kJ / m2 (44%) under hydrostatic pressures.

Balya [4] implemented the parameters required for the design of filament winding tubes under combined loads by lamination theory and finite element methods.

Littlefield et al. [5] fabricated an 11.3 kg Carbon / PEEK (polyether ether ketone) composite, turning a 56.7 kg steel from a tank barrel with a mass of 889 kg, a length of 5460 mm and a caliber of 120 mm, the same material steel barrel has achieved mass reduction of 45.4 kg. They investigate problems that have been overcome during the design process. Finite element models were used to predict the response of the barrel to shot loads and these models were trying to be verified.

Katz et al. [6] and Emerson et al. [7] investigated the use of hybrid CMC / MMC composites in 5.56 mm calibrated barrels as two different research groups.

Tabakov et al. [8] developed theories for anisotropic thick composite cylinders under asymmetric load conditions. The number of layers has achieved full analytical results independent of the layer properties and they have indicated that the theory can be used without restriction for thick and thin-walled, isotonic, orthotropic and anisotropic layered pressure tanks and open-ended cylinders.

Kim et al. [9] found that thermal residual stresses were caused by non-isotropic thermal deformation during cooling of thick-walled phenolic composite cylinders from the curing temperature of 155 °C to room temperature. They have reduced this stress by 30% with the intelligent curing method based on the cooling and reheating they have developed.

Kulayıcıoğlu and Dirikolu [10] designed the pressure tank of the seamless metal lined and composite wound band named Type 3 by TSE with the help of network analysis and classical layer theory. Based on the Tsai-Wu damage criterion, the study considered the failure of the metal liner as the beginning of the plastic flow and the damage of the composite part as fiber breakage.

Değirmenci [11] investigated bottlenecks in the production of lightweight gun barrels capable of being produced with composite support at high temperatures and internal operating pressures.

Önder et al. [12] investigated the optimal layering of thin-walled E-glass / epoxy composite tanks with maximum bursting pressure in layered thin-walled E-glass / epoxy composite tanks. It is revealed that the most suitable winding angle is 55° in coiled composite pipelines exposed at an internal helical angle and this value is 90° in wounded composite pipes at a single angle.

Chen et al. [13] have shown that shear stress can be reduced by 20-30% and environmental stress by 7.9% using thick-walled tubing in various shapes using advanced memory material. They found that the rate of reduction for environmental stress was not sufficient but that the method did not prevent the delamination condition of the layers.

Ansari et al. [14] obtained time-dependent strain, strain, and deformation distributions by performing stress analysis based on the three-dimensional anisotropic elasticity theory for a multilayer filament winding tube subjected to repetitive internal pressure and temperature loading. It has been shown that if the fiber orientation tends to be axial and circumferential, the circumferential rotation tends towards zero and that the tendency of the fiber winding tends to increase circumferential and axial stretches as they become circumferential and longitudinal.

Martins et al. [15] identified structural and functional damage pressures for a filament winding tube in a numerical/experimental study conducted in 2012. Similar to other studies on structural damage, structural and functional damage pressures were observed to be highest at ± 55° of winding angle. In addition, with the Abaqus
Packet Program, they can create a numerical model that can provide some information to determine satisfactory, functional damage for structural damage.

Sburlati [16] obtained analytical results for a thick-walled cylinder that was exposed to internal and external pressures and whose inner wall was entirely layered or just a thin coating.

Güngör and Çelik [17] performed numerical simulations to study the effect of the pressure front and its influence on the projectile/gun tube interaction and lateral movement of the projectile in the gun tube.

In this study, by using dynamical moving pressure structural numerical calculation model developed in the former study [17], composite wrapping of a small arms gun barrel was added to the numerical model and the contribution of composite support to the safety factor of steel was evaluated throughout the barrel with respect to time (or position) in the gun barrel.

The strain levels obtained by the dynamical structural numerical model calculations and the shot test strain measurements were compared. By using strain levels with respect to the number of shot, expansion of the gun barrel was evaluated. Numerical calculations were run on the Abaqus solver [18].

MATERIAL AND METHODS

In the Composite Support Application section, calculations were made using the modeling and solver modules of ABAQUS software. Field testing and verification of the composite support were performed using factory test infrastructures.

As a method; Numerical finite element method was used and the values obtained by the calculations were compared with the measurement results obtained from the experimental method.

The application of the composite, which is supposed to increase the safety factor as a steel support, was incorporated into the numerical model developed [17].

Composite materials were chosen for their availability in our country and ease of use. The same materials as those used for lightweight weapons were used in the gun barrel winding and the comparability of the results was taken into consideration. AS4 was used as carbon fiber, APC2 for PEEK resin and 3501-6 for an epoxy resin.

The use of the thermoplastic resin with carbon fiber composite material (Carbon / PEEK) as an interlayer to the steel with the winding angle of 90 and as a outermost layer with the winding angle of 90, and thermoset resin with carbon fiber composite (Carbon / Epoxy) as an intermediate layer with the winding angle of ±45 degrees between these layers, has been evaluated as appropriate.

The composite layers with variable material properties in each direction were characterized by surface normal and appropriate axis definitions in the numerical simulation software (Figure 1).

Figure 1. Fiber winding directions
For the verification of the numerical model and evaluating the composite reinforcement part, shot tests were carried out with the small arms gun barrel (Composite Wrapped Sniper rifle). It is aimed to calibrate the model measurements to small arms gun barrel measurements and to operate them in accordance with their internal ballistics and to verify them with strain gage measurements.

For this purpose, a series of shots were carried out with the composite small arms gun barrel. The test setup is shown below (Figure 2).

![Test Setup Image](image1)

**Figure 2 Test Setup**

In the test setup, a set of strain gauges was applied as often as possible for the first 160 mm section from the chamber of the gun barrel. Because that was the highest tensile stress section of the gun barrel (Figure 3).

![Strain Gage Application Image](image2)

**Figure 3. Strain gage application for the first 160 mm of the composite wrapped gun barrel.**
The strain gages were arranged axially, at the same point like the numerical model and with an angle of 45° with the barrel axis. In this way, measurements were taken from matching points with the numerical model to compare model against measurements.

In the second part, it was aimed to confirm that the model works properly by comparing the dynamical numerical model [17] which is used in the design of the composite gun barrel and the measurement results to be able to evaluate the measurements related to the composite part.

The model for the composite wrapped light gun barrel is shown below (Figure 4). While light weapon gun barrel is being produced, the forging process is performed and was not subjected to an extra autofrettage process. For this reason, the calculations are based on the same material properties as the steel section in each direction, and the uniaxial tensile test results were utilized.

![Figure 4. Numerical Composite Barrel Model created for calculation and test measurement comparisons.](image)

The anisotropic composite material properties of fiber and resins were used. Composite material configurations were; first layer thermoplastic resin, 1 mm thick, wrapped 90° perpendicular to the barrel axis and as an interface with the steel section, second layer thermoset resin, 4 mm thick, wrapped ±45° with respect to the barrel axis and the last and top layer thermoplastic resin, 1 mm thick, wrapped 90° perpendicular to the barrel axis.

**RESULTS AND DISCUSSION**

According to the theoretical model, the barrel axis position at which the highest stress values are obtained in the composite supported barrel is 100 mm right in front of the chamber section. The highest measured strain values were realized in the direction of the 4th or 5th strain gage placed from the chamber. Similar to the theoretical numerical model, this confirms that the highest strains occurred in the first 100 mm in front of the chamber.

It was evaluated that it would be appropriate to calculate the change of the outer diameter at the point where the highest tension was detected to determine if the tension values taken over the composite support with the theoretical model values are compatible. For this purpose, the tension values of the strain gage perpendicular to the axis and measuring the circumferential strain were translated into composite outer diameter variation (contraction/expansion) values. In addition, in the theoretical dynamic model, a point is defined on the outer diameter and at the same point in the axial direction. The change of the outer diameter while the dummy projectile was passing through this point was calculated with respect to time. In the measurement results, the strains that occurred during the shot were compared to the calculated external diameter changes. The starting points of the curves moved to zero. The theoretical and actual results obtained at the moment of the shot were transformed to diameter expansion/contraction values and showed in the following graph (Figure 5).
In the case where the temperature value is not used in the numerical calculation, the simulated projectile expands the outer diameter while passing from the defined point on the outer diameter of the composite wrap, defined as the strain gauge, only in a short period of time and then decreases in time. On the other hand, the values obtained by the measurements showed that the outer diameter was realized as decreasing and increasing behavior. When evaluated in this way, the numerical calculation was solved in the form of Coupled Thermal-Displacement, where the properties of the temperature of the materials are also used. It was evaluated that the temperature may also be a factor affecting the external diameter change. The temperature was granted equal to 100 °C at every point of the barrel. When this value was selected, the previously performed shot test results were taken into account. In these tests, it was measured that the temperature did not show a tendency to increase not exceeding 100 °C [11].

When the calculation is done in this way, it is determined that the calculation results were very similar to those of the measurement results. It was also determined that the barrel had a tendency to shrink on external diameter immediately before projectile reaches the measurement point, a sudden expansion when passing through the measurement point, and a tendency to tilt back to balance after the projectile has passed (Figure 5).

An interesting result was encountered while looking at the curve of the outside diameter change over a wider period of time and the strain values of the multiple shots as the outer diameter variation.

It has been determined that when the measured values of the strain gauge in the same region are translated as outer diameter variation and when examined, the outer diameter tends to contract for a certain number of shots and have the similar form for each shot. But the diameter tends to expand again as the number of shots increases. The reason for this was considered to be the negative expansion coefficient of the carbon / epoxy resin composite. In this case, the composite reinforcement tends to squeeze the steel at first shot. As the number of shots increases, the inclination decreases and the diameter begins to return to its original state and then to expand (Figure 6).
The effects of the composite support on the capacity of the steel layer load carrying capacity were evaluated by using the safety factor of steel (Figure 7).

The bullet leaves the barrel in about 1.3 ms. Composite support has been shown to improve the steel safety factor by 5% in the front part of the steel (close to chamber), where steel wall thickness is the greatest, and to improve by 10-20% in the oncoming section close to the muzzle. It has been verified that the barrel mass can be reduced by using composite support without losing its durability. It has been shown by using numerical and experimental methods that the cargo carried by troops on a tactical level can be reduced by composite wrapping.
ACKNOWLEDGE

The author would like to thank Ministry of Science, Industry, and Technology which supported this project under the Industrial Thesis Support Program, to Ankara Yıldırım Beyazıt University and MKE Corporation and to everyone who contributed to the Project.

REFERENCES


Numerical Analysis of the Thermal Distribution of a Machinery Driver’s Cab

Hazal Koyuncu¹, Veli Çelik²

¹HIDROMEK A.Ş., Ankara, TURKEY  
ahazal.koyuncu@hidromek.com.tr

²Department of Mechanical Engineering, Yıldırım Beyazıt University, Ankara, TURKEY  
vcelik05@hotmail.com

ABSTRACT

HVAC systems are of vital importance in both comfort and safety in today's transportation vehicles. Therefore, one of the most important goals of today's vehicle manufacturers is to provide a comfortable and luxurious travel. Thermal comfort will be an advantage if it is done at a minimum cost, without changing the style of the vehicle and without increasing fuel consumption. 

In the present study, velocity and temperature distributions have been investigated numerically in a machinery driver’s cab cooled by the air conditioning system. Computational mesh has been generated using ANSYS meshing module. Commercial, Computational Fluid Dynamics (CFD) software FLUENT has been used to get numerical solutions. The necessary data have been collected during the air conditioning test conducted while the machinery was operating, and the measured data have been used as boundary conditions. Radiation effects have also been included in the numerical model in FLUENT. Then the model verification has been done by comparing the CFD predictions with the test data. Consequently, a general evaluation has been done and proposals have been presented.

Keywords: HVAC, Computational Fluid Dynamics (CFD), Flow Analysis

INTRODUCTION

Until the beginning of the 90’s, car design was mostly steered to vehicle aerodynamic or engine studies than the thermal management of the passenger compartment. Thermal comfort in vehicles was considered a luxury privilege. Over time, thermal comfort has become a more important issue for customer satisfaction. In-car air conditioning modeling is important not only for thermal comfort but also for safety. Today, thermal comfort has become a more important criterion while choosing vehicles. In-car air conditioning modeling is important not only for thermal comfort but also for safety. An uncomfortable in-car environment may cause distraction and short or long periods of sleepiness of drivers. This may jeopardize driving safety. Since weather conditions are variable, good air conditioning in the driver's cabins is a challenging process. In terms of cooling capacity, the glazed area is also very important for machinery cab.

E. Z. E. Conceição et. al (1999) [1] examined the heat transfer balance and mechanisms in vehicle cabinets. In the test study on the railway wagons, the effects of solar radiation in the cabin were examined as experimental and numerical models. Wagon glazings, ceilings and interior temperatures of the train were taken in moving and non-moving condition. They published their results comparing with the numerical model.

Aronson et. al. (2000) [2] examined the airflow distribution in the car using the ventilation method. PIV measurements were carried out in the vehicle and results were companies with the modeling predictions. They reported similar results in the overall flow area.

Currie and Mouse (2000) [3] used a Mercedes E-series car's flow model. They made changes in the size of the side and middle vents of the flow model. It was been observed that changes in vent dimensions also affect the velocity of air flow. According to these variables, the distribution of air in the vehicle was examined. They examined the changes in thermal comfort in the vehicle and published them in their articles.
In the present study, isothermal and flow field with temperature variations in a machinery driver’s cab cooled by air conditioning were analyzed. The operator cabin cooling test is performed according to ISO 10263-4 standard [6]. There is no operator in the driver’s cab during the test. The radiation model S2S, surface to surface, and the solar load calculator were activated in the FLUENT and the radiation effects were included in the numerical model. According to the seat index point defined by ISO 5353, temperature measurements were taken at certain points around the operator. Based on these points, temperature measurements have been taken by placing thermocouples at certain points around the operator's seat. The test and analysis results of these points were compared.

MATERIALS AND METHODS

Model Geometry

The computational geometry is shown in Figure 1,

![Flow volume of the machinery driver’s cab](image)

**Figure 1. Flow volume of the machinery driver’s cab**

Preparation of the Computational Fluid Dynamics (CFD) Model

Geometric details that cannot influence the flow field must be deleted before meshing. 31 Million Tetrahedral mesh was created for flow volume. Aspect Ratio was used as the boundary layer method for analysis and 8 prism layers were used for near–wall modeling. Realizable, k-epsilon turbulence model was used for the simulations. The maximum Skewness value of the flow volume was 0.91. Steady-state computations were carried out. Buoyancy effects were also included in the energy equations. The convergence criteria for the model were 1E-3 for continuity, 1E-4 for momentum equations and 1E-6 for energy. Buoyancy effect was also included in the energy equation, using a Boussinesq approximation, to catch the main points the effects of rising hot air and sinking cool air. Incompressible ideal-gas was used for air density. With the S2S radiation model and solar load calculator in fluent influences of solar radiation was included in the numerical model. S2S radiation model assumes all walls to be gray and diffuse. The data collected during the vehicle test was used as a boundary condition in the numerical model. These data were the velocity and temperature values in the vehicle vents. In addition, the ambient air temperature outside the vehicle was measured and used for the analysis. Technical specifications of materials used in machinery cabin were defined in the analysis. Convection heat transfer boundary condition was given to glasses and wall surfaces. The constant temperature measured as boundary condition was used to the wall close to the engine side.

CFD Mesh

One volume mesh is created for the CFD model. There are 8 vents on the front side and 6 vents on the rear side of the model. In total, there are 14 vents and 1 outlet of the model. The vents are oriented towards the driver and the windows. The vents are indicated as inlet conditions. So that, different air mass fluxes, inlet velocities, inlet temperature or inflow directions as well as vent areas etc. may be arranged very readily. The air outlet is on the left side wall and the boundary conditions are constant for all calculations.
The computational mesh was shown in Figure 2.

![Figure 2. Mesh and Inlets Representation on the Model](image)

**CFD Results**
As stated in ISO 10263-4, temperature measurements were taken from the positions indicated in the following model.

![Figure 3. Display of points](image)

Table 1. Differences between measurement results and analysis results

<table>
<thead>
<tr>
<th>Points</th>
<th>Average Experimental Results (K)</th>
<th>Average CFD Results (K)</th>
<th>Difference</th>
<th>% Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Head Position</td>
<td>296.35</td>
<td>298.55</td>
<td>2.2</td>
<td>0.8</td>
</tr>
<tr>
<td>Right arm position</td>
<td>295.98</td>
<td>299.74</td>
<td>3.8</td>
<td>1.3</td>
</tr>
<tr>
<td>Left arm position</td>
<td>295.16</td>
<td>295.95</td>
<td>0.8</td>
<td>0.3</td>
</tr>
<tr>
<td>Steering position</td>
<td>297.68</td>
<td>298.25</td>
<td>0.6</td>
<td>0.2</td>
</tr>
<tr>
<td>Right foot position</td>
<td>294.45</td>
<td>293.65</td>
<td>0.8</td>
<td>0.3</td>
</tr>
<tr>
<td>Left foot position</td>
<td>293.75</td>
<td>293.55</td>
<td>0.2</td>
<td>0.4</td>
</tr>
</tbody>
</table>

In the above table, the results of the CFD and the experimental results were compared. According to the results, it was seen that the results of the CFD and the results of the test were very close to each other at the shown points.

**Temperature Distribution**

In the following figures, figures were taken in different planes in order to be able to observe the temperature distribution in the machinery driver’s cab.

![Temperature Distribution Diagram](image)

Figure 4. Temperature values of the operator at the foot level

Figure 4 demonstrates varying temperature values of the operator at the foot level between 20-22 °C. Due to the radiated heat, the temperature in the cabin wall close to the engine varies between 25-27 °C.
Figure 5. Temperature distribution of the operator in chest position

Figure 5 illustrates that the temperature distribution of the operator in the chest position changes within the range of 22-26 °C around the driver. A temperature difference of 5-6 °C between the temperature distributions was determined on the right and left sides for the windshield position of the cabin. The reason for this difference is considered to be the higher air velocity of the air vents close to the air conditioning fan unit accelerating the heat transfer in those areas.

Figure 6. Temperature distribution of the operator's in head position

In the plane in Fig. 6, the temperature in the vent outlets varies between 10 and 13 °C while the temperature around the head of the operator is determined between 26-31 °C. Direct reflection of the solar load by the windshield at a certain angle and the different temperature and air velocities of the vents have caused this region to vary.
Figure 7. Temperature changes on a plane in the YZ axis

Figure 7 illustrates the temperature changes on the cabin walls and the interior surfaces of the cabin as well as the temperature changes on a plane in the YZ axis. The average temperature of the cabin interior walls was about 26 °C while the average temperature of the outer walls of the cabin was approximately 36 °C.

**Velocity Distribution**

In the following figures, figures were taken in different planes in order to be able to observe the velocity distribution in the machinery driver’s cab.

Figure 8. The velocity measurements in the operator head position
Figure 8 examines the velocity measurements in the operator head position which are considered to be important. In the analysis, the vent positions were directed directly to the operator and the maximum velocities were intended to be observed while the air conditioner was operating at full speed. The maximum value of the velocity around the operator head was approximately 0.9 m/s. The result of the analysis in the eye around the operator was approximately 0.1 m/s.

![Velocity Magnitude Contour](image)

Figure 9. Temperature distribution on a plane in the YZ axis

Figure 9 investigates velocity variations on a plane formed on the YZ axis. It was observed that the velocity values in the areas where the vents were directed were higher than in the other regions.

**RESULTS**

Flow analysis was performed for summer conditions by considering the velocity and temperature measurements taken from the vent outlets in a machinery driver’s cab. Planes were formed in the cabin with reference to the measurement points in the results of the analysis, and the temperature and air velocity values on these planes were shown by taking the contour. In addition, measurement and analysis values of the temperature in points were expressed in a comparative table. The examination of both the points taken in the cabin and the general temperature distribution reveals a maximum 1.3% difference between the results of the analysis and the measurement results. Temperature and flow velocity values met the requirements of ISO 10263-4. The results showed that the given boundary conditions were close to reality. As another indicator, the mesh and solvent method were suitable for the computational model. With these achievements, flow analyzes can be performed before waiting for the test results in similar designs. According to the results, system performance can be improved through the improvements in the design.
REFERENCES


MICROMECHANICAL CHARACTERIZATION OF NANOTUBE REINFORCED POLYMERS

Vahidullah Taç, Ercan Gürses

1,2 Dept. of Aerospace Engineering, Middle East Technical University, Ankara, Turkey
1 Turkish Aerospace Industries, Ankara, Turkey
3 Corresponding Author: Ercan Gürses, gurses@metu.edu.tr

ABSTRACT

One of the greatest challenges seen as obstructing further development and wide-scale use of carbon nanotube reinforced materials is seen as a lack of understanding and thereby of modelling of the interfacial region of carbon nanotube composites. We develop analytical models of carbon nanotube reinforced polymer composites using micromechanics and hence characterize the elastic moduli of nano composites with a special emphasis on the interfacial region. The micromechanical model allows for multiple inclusions or phases and the phases can be functionally graded. A number of parametric studies were performed to visualize the effects of various unknowns in such a nano composite on its mechanical properties. In particular the effect of number of inclusions, importance of each inclusion and length and width of the nanotube were studied in detail. The results were compared with previously obtained numerical and experimental data available in the literature.

Keywords: Carbon nanotube, nano composite, interface, interphase, micromechanics

1. INTRODUCTION

The study and modelling of carbon nanotubes (CNT) and their composites has recently gained momentum due to the potential that they show in revolutionizing the materials industry. Mechanics wise, CNTs have shown unprecedented stiffness and strength values that, coupled with their extremely large aspect ratios, make them ideal candidates for reinforcing polymers for engineering applications. While CNT-polymer composites are already being used in some rare applications, large scale implementation of this reinforcement mechanism is yet to come. Major reasons for this are usually cited as a lack of profound understanding of the mechanisms and phenomenon that take place in the interfacial region between the carbon nanotube and the polymer and lack of effective modeling techniques for their mechanical analysis.

Some criteria sought in modeling methods for mechanical analysis of CNT-polymer composites are for them to be light, easy to use and of high performance and fidelity. Micromechanics is widely regarded as a good candidate for developing models for CNT composites due to its simple and flexible nature. Numerous studies have recently focused on use of micromechanics for accurate representation of nano composites [1], but they are yet to fully culminate.

Bi-phase composites such as CNT-polymer composites have traditionally been simply represented by two micromechanical phases, or regions; the carbon nanotube and the polymer. However, as the science of nano composites progresses new observations are being made on the interfacial region of CNT-polymer composites that suggest the interfacial region has distinct mechanical properties than either CNT or polymer, and hence should be treated as a separate phase. Additionally, the polymer around the nanotube has been observed to change properties due to the presence of the CNT [2]. Building a high fidelity micromechanical model requires that this region, known as the interphase, also be taken into account.

In this study we use a micromechanical model of 4 phases consisting of the carbon nanotube, the interface, the interphase and the polymer. The mechanical properties of the interphase are modelled in a functionally graded way after observation of its density distribution in atomistic simulations. A number of parameters affecting the results of the model were studied.
in detail and recommendations made for their determination. The results are compared with other numerical and experimental studies for further validation and adjustment of the model and its parameters.

2. MODEL CONSTRUCTION AND DESCRIPTION

The micromechanical model used in this study was adopted from [3]. The model assumes $n$ ellipsoidal micromechanical inclusions of elastic constitutive nature embedded in a fictitious infinite domain. The inclusions are required to be of similar shape, such that $\frac{a_1}{b_1} = \frac{a_2}{b_2} = \frac{c_1}{c_2} = \gamma$ where $a_i, b_i$ and $c_i$ are the dimensions of the ellipsoids as shown in . The fictitious domain represents the entire nano composite from a continuum point of view. Whence the elastic properties of the infinite domain were assumed to be identical to that of the nano composite that results from this model. Initially a set of guess values were assigned to the infinite domain moduli. The micromechanical model was then used to calculate the moduli of the nano composite and the results homogenized as described in section 2.1. If the moduli of the nano composite were reasonably close to those of the infinite domain the calculation was finalized, otherwise the values obtained from homogenization were assigned to the infinite domain and the operation restarted.

![Infinite Domain](image)

**Figure 1.** Figurative representation of the inclusions in the micromechanical model, and their respective dimensions.

The elasticity tensor, $\mathbf{C}$, of a composite of $n$ inclusions is provided as

$$
\mathbf{C} = \mathbf{C}^{\text{inf}}[\mathbf{I} + (\mathbf{S} - \mathbf{I})\Lambda][\mathbf{I} + \mathbf{S}\Lambda]^{-1}
$$

$$
\Lambda = \sum_{i=1}^{n} f_i \Lambda_i
$$

where $\mathbf{I}$ is the 4th order identity tensor, $\mathbf{S}$ is Eshelby’s tensor for an ellipsoidal inclusion [4], $f_i$ are the volume fractions of the inclusions, and $\mathbf{C}^{\text{inf}}$ is the elasticity tensor of the infinite domain.

The definition of $\Lambda$ depends on how a given inclusion is graded. If the inclusion is of constant characteristics then $\Lambda$ is given as

$$
\Lambda_i = \left[\left(\mathbf{C}^{\text{inf}} - \mathbf{C}_i\right)^{-1}\mathbf{C}^{\text{inf}} - \mathbf{S}\right]^{-1}
$$

on the other hand if the inclusion is to be functionally graded the following integral form of $\Lambda$ must be used

$$
\Lambda_i = \frac{3}{1 - \gamma^2} \int_0^1 r^2 \lambda_i(r) dr
$$

$$
\lambda_i = \left[\left(\mathbf{C}^{\text{inf}} - \mathbf{C}_i(r)\right)^{-1}\mathbf{C}^{\text{inf}} - \mathbf{S}\right]^{-1}
$$
2.1. Nanotube Orientation Homogenization

In a typical nanotube reinforced nano composite the nanotubes are randomly distributed across the polymer in random orientations. From a macroscopic point of view this results in isotropic properties. Obtaining the properties of such macroscale nano composites does not require building large models consisting of large numbers of nanotubes. Instead, an orientation averaging scheme can be utilized to emulate partially or fully random distributions of CNT orientation. The orientation averaging integral of tensor $\mathbf{A}$ is denoted by $\langle \mathbf{A} \rangle$ and is given as

$$
\langle \mathbf{A} \rangle = \frac{\int_{-\pi}^{\pi} \int_{0}^{\pi/2} \int_{0}^{\pi} \bar{A}(\phi, \gamma, \psi) g(\phi, \psi) \sin(\gamma) \, d\phi \, d\gamma \, d\psi}{\int_{-\pi}^{\pi} \int_{0}^{\pi/2} \int_{0}^{\pi} g(\phi, \psi) \, d\phi \, d\gamma \, d\psi}
$$

where

$$
\bar{A}_{ijkl} = c_{ij} c_{pq} c_{kr} c_{ts} A_{pqrz}
$$

$c_{ij}$ are the direction cosines of the transformation from local fiber coordinates to global coordinates [5] and $g$ is the orientation distribution function

$$
g(\phi, \psi) = \exp(-s_1 \phi^2) \exp(-s_2 \psi^2)
$$

$g$ defines how nanotube orientation is distributed across the composite using the parameters $s_1$ and $s_2$ as follows:

- **Random Orientation**: $s_1 = 0, s_2 = 0 \quad g(\phi, \psi) = 1$
- **Aligned Orientation**: $s_1 = 0, s_2 = \infty \quad g(\phi, \psi) = \delta(\phi - 0) \delta(\psi - 0)$
- **Axisymmetric Orientation**: $s_1 = k, s_2 = \infty \quad g(\phi, \psi) = \exp(-k\phi^2) \delta(\psi - 0)$

where $\delta(x - x_0)$ is Dirac’s delta function.

2.2. Inclusion Properties

2.2.1. Carbon Nanotubes (CNT)

Carbon nanotubes are usually reported to have transversely isotropic properties with a longitudinal Young’s modulus in excess of 1 TPa. In this study we adopted the molecular dynamics results of Tsai et al. [6] for the moduli of 3 single walled zigzag CNTs with varying radii that are reported in Table 1.

<table>
<thead>
<tr>
<th>Radius [Å]</th>
<th>$E_1$ [GPa]</th>
<th>$G_{12}$ [GPa]</th>
<th>$\nu_{12}$</th>
<th>$E_2$ [GPa]</th>
<th>$\nu_{23}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.9</td>
<td>1382.5</td>
<td>1120</td>
<td>0.272</td>
<td>645</td>
<td>0.2</td>
</tr>
<tr>
<td>5.5</td>
<td>981.5</td>
<td>779.2</td>
<td>0.27</td>
<td>504</td>
<td>0.2</td>
</tr>
<tr>
<td>7.1</td>
<td>759.9</td>
<td>596.3</td>
<td>0.27</td>
<td>425</td>
<td>0.2</td>
</tr>
</tbody>
</table>

2.2.2. Interface (IF)

The interface is the empty space separating the CNT from the polymer. Its width is usually reported to be roughly equal to the average distance between graphene sheets; i.e. 3.4 Å. The interface is dominated by weak Van der Waals forces as opposed to covalent bonds in the CNT. In this study the interface is simplified as a uniform domain of isotropic elastic properties. Poisson’s ratio of the interface was assumed to be equal to 0.3, which is similar to most isotropic materials found in nature. This parameter was later verified to be of trivial significance to the end results.

The stiffness of the interface is a matter of great debate. In this study two models of the interface are utilized consisting of a soft and a hard interface to analyze the effects of interface stiffness on the nano composite properties. The two interface models are characterized relative to the bulk polymer (BP) in Table 2.
Table 2. Elastic moduli of the interface.

<table>
<thead>
<tr>
<th>Interface Model</th>
<th>Young’s Modulus</th>
</tr>
</thead>
<tbody>
<tr>
<td>Soft interface</td>
<td>$E_{IP} = 0.3E_{BP}$</td>
</tr>
<tr>
<td>Stiff interface</td>
<td>$E_{IP} = 5E_{BP}$</td>
</tr>
</tbody>
</table>

The concept of soft and stiff interphases were previously studied in [7–9] but there the interphase constituted the entirety of the interfacial region unlike in the current study where the interfacial region consists of two parts; the interface and the interphase.

2.2.1. Interphase (IP)

The interphase is a region of the polymer that changes its properties as a result of the presence of carbon nanotubes. The density distribution of the polymer in a radially outward direction from the center of the carbon nanotube in carbon nanotube-polymer composites seems to start from a higher value and decrease until matching that of the bulk polymer [10]. This motivated the use of a functionally graded model for the moduli of the interphase. We assumed the interphase Poisson’s ratio stays unmodified, but the Young’s modulus starts from a higher value and linearly decreases until matching that of the polymer.

A number of varying values have been reported for the thickness of the interphase from 3 to 26 Å [2,11], however the general consensus appears to be that the width of the interphase is independent of CNT radius.

Unless otherwise stated the interphase Young’s modulus is assumed to start from $E_{IP_1} = 2E_{BP}$ and end with $E_{IP_2} = E_{BP}$, its width is assumed to be equal to 10 Å.

2.2.1. Bulk Polymer (BP)

The absolute value of the mechanical properties of the polymer on which the nano composite is based on is not given a great importance in this study. This is because the resulting nano composite properties are expressed in reference to the bulk polymer. A polymer synthesized by Odegard et al. [12] with a Young’s modulus of 3.8 and a Poisson’s ratio of 0.4 was chosen as the matrix in this study.

3. RESULTS

It is important to emphasize the importance of use of a 4-phase model. Ignoring any of the phases in the nano composite could lead to serious error. We plotted the variation of normalized stiffness of the nano composite as a function of CNT volume fraction for a 2-phase, 3-phase and 2 4-phase models in Figure 2. In all of these CNT radius is kept at 3.9 Å. Note that the 2-phase and 3-phase models significantly underestimate the stiffness of the nano composite even compared to a 4-phase model with a soft interface. This is because these models do not take into account the effect of the interphase on the stiffness. The interphase can be the largest phase by volume depending on its width. For example, in this analysis the 4-phase models do not go beyond CNT volume fraction does not go beyond 1% because at that point the interphase constitutes almost the entirety of the nano composite.
Figure 3 shows a plot of nano composite normalized stiffness with respect to CNT radius. Two curves are plotted for the stiff and soft interfaces with a CNT volume fraction of 0.1%. The first conclusion one can draw from this plot is that as the CNT radius grows its stiffening effect diminishes. This is largely due to the fact that keeping the volume fraction the same and increasing the CNT radius effectively means decreasing the number of CNTs. This in turn decreases the portion of the polymer that has turned into interphase. A stiff interface results in higher stiffening ratios than the soft interface,
but this effect is also diminished with growing CNT radius for the same reasons cited above; that is the interface occupies smaller regions as the CNT radius grows and hence has reduced impact on the nano composite stiffness.

To study the effect of interphase width on the stiffening effect of the CNT we plotted the normalized nano composite Young’s modulus as a function of interphase width for a CNT volume fraction of 1% and CNT radius of 7.1 Å in Figure 4. It appears seen that the stiffening effect linearly increases with interphase thickness after about 5 Å. There is about 10% difference between the stiff and soft interface models, which stays constant despite the rising interphase width. Assuming a moderate interphase thickness of about 10 Å causes a 25 to 35% increase in the stiffness of the polymer using the current configuration.

In Figure 5 we compare the results of the current model with the findings of the literature using a stiff interface and a moderate interphase thickness of 10 Å for three different CNT radii. The figure shows that for a given volume fraction using a carbon nanotube of smaller radius results in higher stiffening ratios, however using small carbon nanotubes results in much higher interphases. This prevents their increased use in dilute models. Larger nanotubes, however, can be used in much higher quantities without reaching dilution limits of the nano composite. In the current configuration carbon nanotubes with a radius of 7.1 Å can be used by more than 3.8% before reaching the dilution limit of the nano composite.

The results of the current model are also compared to other experimental and numerical findings in the literature in Figure 5 [7,12–19]. Experimental results are in black, while numerical studies are noted in gray colors. Another conclusion that can be drawn from Figure 5 is that using a soft interface, interphase width of about 10 Å and interphase moduli described in 2.2.1. result in micromechanical models that have a high accuracy as compared to experimental and numerical data in the literature.
Figure 5. Stiffening ratio of the current model compared to other numerical and experimental work in the literature. Numerical studies are shown with gray markers, while experimental ones are in black.

4. CONCLUSIONS AND RECOMMENDATIONS

The research work reported in this study shows that using a 4 phase micromechanical model that accounts for all regions of the nano composite with distinct mechanical properties results in stiffening ratios that are in good agreement with the previous findings in the literature. A special emphasis must be placed on the interfacial region of the nano composite in order to obtain high performance models as this region is arguably the most important in the nano composite. The interphase is best represented using functionally graded moduli, however more atomistic simulations and analyses are required for better understanding of the exact nature of moduli distribution in the interphase.

5. REFERENCES


EFFECT OF MACHINING AND HOT ISOSTATIC PRESSING ON MECHANICAL PROPERTIES OF TI-6AL-4V MANUFACTURED BY ELECTRON BEAM MELTING

Fırat MEMU1,a, Nuri DURLU1,b, Kaan ÇİLOĞLU2,c, Burcu ARSLAN HAMAT2,d, Yavuz GÜLEÇ2,e

1 TOBB University of Economics & Technology, Mechanical Engineering Department, Ankara, Turkey
2 Turkish Aerospace, Ankara, Turkey
1 Corresponding Author: Fırat MEMU, fmemu@etu.edu.tr

ABSTRACT

Parts with desired mechanical properties especially for aerospace and medical implant industries can be produced by powder based additive manufacturing techniques. In this study, the effects of post-processing techniques such as machining and hot isostatic pressing on the mechanical properties of Ti-6Al-4V alloy manufactured by electron beam melting have been investigated. Density, yield strength, tensile strength and ductility values of the samples have been compared after each operation. Microstructures of the as electron beam melted and hot isostatically pressed samples have been examined by optical and scanning electron microscope. The results showed that machining led to higher yield and tensile strengths, whereas hot isostatic pressing significantly improves ductility of electron beam melted samples.

Keywords: Additive Manufacturing, Electron Beam Melting, Ti-6Al-4V, Hot Isostatic Pressing

1. INTRODUCTION

The Ti-6Al-4V alloy, with its heat resistance, high strength and corrosion resistance, has been frequently used a material especially in aerospace and medical industry. The difficulties and limitations in machining of titanium alloys lead to high costs [Donachie, 2000]. In recent years, especially in the manufacturing of complex shaped workpieces, additive manufacturing has become an important method of production. Electron Beam Melting (EBM) is one of these additive manufacturing techniques, which is commonly used in the manufacturing of Ti-6Al-4V alloys. In this method, the CAD file of the workpiece to be produced is transferred to the EBM machine. After this procedure, 3D part is divided into slices in the direction of production. The thickness of these slices is the layer thickness entered as the control parameter. Then, Ti-6Al-4V powders are laid on the steel plate in a vacuum chamber in order to prevent the powders from interacting with oxygen during processing. The electron beam produced by the electron gun is send onto the Ti-6Al-4V powders in order to form the layered structure of the sliced workpiece. The electron beam provides the energy required to melting and bonding of powders. After finishing a layer, the spreading and melting process is repeated with electron beam and the lower and upper layers are bonded. These processes continue until obtaining the final workpiece [Murr continued, 2012]. The temperature of the vacuum chamber is between 650-700 ºC in EBM process. The reason for this is that it is not desirable for the powders with high temperature suddenly cool down. Ti-6Al-4V powders have α-martensite microstructure due to rapid cooling conditions and residual stresses are formed in the part [Al-Bermani continued, 2010].

Post-processes can be used to increase the mechanical properties of the components produced by the EBM process. The surface roughness of the part manufactured by EBM process varies according to the beam diameter applied during the production [Ramulu continued, 2013]. After production, machining can be performed to reduce the surface roughness of the part. It has been shown that the mechanical properties of parts can be improved due to removal of surface roughness [Formanoir continued, 2016]. In the study of Formanoir et. al., the effect of machining on the tensile properties of electron beam melted Ti-6Al-4V samples has been investigated. The strength and elongation of the electron beam melted samples have been increased after machining. The study by Murr et. al. has reported that dimensions of the part should be increased as much as for the depth of cut before the machining process [Murr continued, 2009].

Hot isostatic pressing (HIP) method can be applied both to increase the density of the produced part and to improve its mechanical properties [Facchini continued, 2009]. In this method, an increase in the density of the part can be obtained by applying pressure in all directions in a certain temperature and time under inert atmosphere. A change in the
microstructure can also be obtained due to the applied temperature. Thus, both microstructure and density control can be done by HIP method. In a study by Al-Bermani et al., both the production temperature and HIP effect on the mechanical properties and microstructure of the Ti-6Al-4V samples have been investigated [Al-Bermani continued, 2010]. It has been shown that the size of the alpha-phase in the microstructure has been increased in the HIPed samples regardless of the production temperature. This change resulted in a decrease in yield and tensile strengths and an increase in elongation of electron beam melted samples. In a study by Mohammadhosseini et. al., the effect of hot isostatic pressing application on the mechanical properties of manufactured tensile and fatigue samples has been investigated [Mohammadhosseini continued, 2013]. After HIP process, a decrease in the yield and tensile strengths and an increase in elongation of electron beam melted Ti-6Al-4V samples had been observed. A significant increase in fatigue properties has been shown with HIP process. With these studies, it has been shown that the elongation of the samples produced by EBM can be improved by HIP process.

2. EXPERIMENTAL PROCEDURE

2.1. Material and Electron Beam Melting Process

Ti-6Al-4V powders with spherical morphology supplied from Arcam produced by gas atomization method have been used. Size distribution of the Ti-6Al-4V powders are between 45 and 100 µm. The chemical composition of the Ti-6Al-4V powders is shown in Table 1.

| Table 1. Chemical composition of Ti-6Al-4V powder used in EBM [*ASTM F1108, 2014] |
|-----------------|--------------|----|----|---|---|---|---|
| Element         | Ti           | Al | V  | C  | Fe | O  | N  |
| Typical (%)     | Balance      | 6  | 4  | 0.03 | 0.1 | 0.15 | 0.01 | 0.003 |
| Required* (%)   | Balance      | 5.5 - 6.75 | 3.5 - 4.5 | < 0.1 | < 0.3 | < 0.2 | < 0.05 | < 0.015 |

Cylindrical tensile specimen designed in computer aided software according to the ASTM E8/E8M standards has been transferred to the EBM machine for fabrication. The dimensions of the three-dimensional model of the specimen have been shown in Figure 1. Each layer thickness of the 3D model specimen is 90 µm. The final part is obtained with assembled the layers along the z-axis (perpendicular to the plate). Optimized process parameters from Arcam have been used to reach final shape of the tensile specimen.

![Figure 1. Technical drawing of cylindrical tensile specimen.](image)

2.2. Machining and HIP

Machining and HIP have been applied to investigate the effect of post-processing on the tensile specimens built up layer by layer. Before machining process, the gage diameter has been increased by the amount of depth of finish cutting to be removed from the surface. Finish process with 0.5mm depth of cut has been applied to decrease the roughness on the specimen surface. HIP has been carried out at 900 ºC for 2 hours with pressure of 100 MPa to the received tensile specimens. As-built (DR), as-machined (DM) and as-HIPed (DH) samples are shown in Figure 2.

2.3. Mechanical Testing

The density values of as-received, as-machined and as-HIPed samples have been measured by the Archimedes method. The gage length and the gage diameter of samples prepared according to ASTM E8/E8M standards is 30 mm and 6 mm. A mechanical test device (Instron) which has maximum 5 tons load capacity has been used to investigate the mechanical behaviour of samples. The crosshead speed has been set to 1 mm/min on the device and elongation data has been recorded by using an extensometer. The tensile stress – tensile strain graph has been defined from obtained force and elongation data. Tensile stress, percentage elongation and yield stress from 0.2% offset value have been calculated from the graph.
Samples were cut from the horizontal and vertical sections of the electron beam melted Ti-6Al-4V samples for microstructural investigation with optical and scanning electron microscope. The sections have been prepared for metallographic examination. First, rough grinding with special surface for titanium alloys followed by fine grinding with 9 µm diamond abrasive suspension have been performed. Secondly, polishing has been applied with 0.04 µm diamond abrasive suspension. The pore structures of the samples have been investigated with optical microscope.

3. RESULTS AND DISCUSSION

3.1. Density

The density values of the as-received (DR), as-machined (DM) and as-HIPed (DH) samples have been calculated according to the Archimedes method. The densities are shown in Table 2. Theoretical density of Ti-6Al-4V is 4.43 gr/cm³. Table 2 also shows percentage of density. Due to reduction in porosity, HIP process led to highest density samples.

Table 2. Density values of the samples build in EBM

<table>
<thead>
<tr>
<th>Samples</th>
<th>Density (gr/cm³)</th>
<th>Density (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>DR</td>
<td>4.31 ± 0.01</td>
<td>97.2</td>
</tr>
<tr>
<td>DM</td>
<td>4.35 ± 0.05</td>
<td>98.1</td>
</tr>
<tr>
<td>DH</td>
<td>4.40 ± 0.02</td>
<td>98.6</td>
</tr>
</tbody>
</table>

3.2. Tensile Properties

Tensile tests have been done with the as-received, as-machined and as-HIPed samples. The mechanical properties of the electron beam melted Ti-6Al-4V samples are given in Table 3.

Table 3. Tensile properties of the electron beam melted Ti-6Al-4V samples build in vertical direction.

<table>
<thead>
<tr>
<th>Samples</th>
<th>Yield Strength (MPa)</th>
<th>Tensile Strength (MPa)</th>
<th>Elongation (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>DR</td>
<td>877 ± 37</td>
<td>907 ± 46</td>
<td>3.1 ± 0.7</td>
</tr>
<tr>
<td>DM</td>
<td>970 ± 18</td>
<td>994 ± 20</td>
<td>2.9 ± 0.6</td>
</tr>
<tr>
<td>DH</td>
<td>803 ± 4</td>
<td>876 ± 6</td>
<td>5.3 ± 1.7</td>
</tr>
</tbody>
</table>

As shown in Table 3, yield and tensile strengths have been increased after machining process whereas elongation of samples is very similar. After HIP process, due to closure of porosities considerable increase in elongation was observed with some decrease in yield and tensile strengths due to coarsening of the microstructure. Engineering stress-strain diagram of the as-received (electron beam melted) and as HIPed samples is given in Fig. 3.

3.3. Pore Structure

Sections were taken from the as-received and machined samples in directions perpendicular and parallel to the build direction. Pore structures have been examined with an optical microscope and a scanning electron microscope from these sections. Before the optical microscope images have been taken, the sections were prepared metallographically. Firstly, the samples have been cut by precision diamond saw and then mounted with bakelite. The samples have been subjected to grinding and polishing processes specific to the Ti-6Al-4V alloy. Rough damages (stretches and saw marks) on the surface have been eliminated by grinding. Micro damages on the surface have been removed by polishing process and the sample surface has been brought to the same height in all directions.
Surface optical images of before and after the machining process are shown in Figure 4. Figure 4 shows that the surface roughness can reach to 200 µm after manufacturing. It has been shown in previous studies that the high roughness value creates a crack effect on the sample surface during the tensile test [Farmanoir continued, 2016]. Because the crack formation starts from the surface during the tensile test, the surfaces in Figure 4a have made crack formation easier.

After the surface roughness has been examined with an optical microscope, the effect of the HIP process has been investigated with an optical microscope. The pores on the surface of as-received samples have been re-analysed by optical microscope after HIP application. Figure 5 shows as-received and as-HIPed pore sizes. The application of high pressure and temperature have resulted in shrinkage in pore sizes. The effect of this shrinkage in density values is shown in Table 2.

The pore size of as-received sample shown in Figure 5 is nearly 100 µm. After hot isostatic pressing application, the pore size has been decreased to 15-20 µm.
Figure 6 shows SEM images of etched faces of as-received and as-HIPed samples. Figure 6a shows fine and sharp grain structure in the as-received samples, however, it is seen that these grain structures become more coarse and elliptic structure after HIP process. It has been observed that the width of the alpha structures in as-HIPed samples is higher than the as-received samples.

4. CONCLUSION

Both machining and hot isostatic pressing application have been carried out to investigate the effect of post processing on the mechanical properties of electron beam melted Ti-6Al-4V alloys. Following conclusions can be drawn:

1. With the machining of the Ti-6Al-4V alloy part produced by EBM, an increase in density, yield strength and tensile strength has been achieved, while a significant change in elongation value could not be achieved.

2. The porosity of the Ti-6Al-4V alloy part produced by EBM has been reduced after the HIP application, resulting in a significant increase in density.

3. Yield strength and tensile strength have been decreased by HIP application, while an increase in elongation has been obtained.

ACKNOWLEDGEMENTS

This study was supported by Presidency of Defence Industries, TOBB University of Economics and Technology and Turkish Aerospace under the SAYP project DDKIT1. We would also like to acknowledge Dr. N. Kaan Çalışkan from TUBITAK-SAGE for HIP experiment.

REFERENCES


IMPROVED DELAYED DETACHED EDDY SIMULATION OF DEEP CAVITIES AT SUBSONIC FLOW CONDITIONS

Burak Tüzüner¹, *Emre Alpman², Levent Ali Kavurmacoğlu³

¹Department of Mechanical Engineering, Istanbul Technical University, Istanbul, TURKEY
²Department of Mechanical Engineering, Marmara University, Istanbul, TURKEY
* Corresponding author: Emre Alpman, emre.alpman@marmara.edu.tr

ABSTRACT

This study includes high subsonic compressible flow simulations over a rectangular deep cavity. Numerical solutions are obtained using the open-source computational fluid dynamics software OpenFOAM. Improved Delayed Detached Eddy Simulations (IDDES) are used as the basic turbulence model and IDDES predictions are compared with the Reynolds Averaged Navier-Stokes (RANS) and Large Eddy Simulation (LES) predictions in terms of computational time, flow field characteristics, time averaged velocity profiles in cavity and acoustic results. Spalart-Allmaras IDDES and k-ω Shear Stress Transport(SST) IDDES methods are used as different IDDES alternatives and simulations performed for 0.8 Mach number as inlet flow condition. As RANS model, standard k-ω model is used. Standard k equation model and wall-adaptive local eddy viscosity(WALE) model used as LES models. A manual iterative procedure is used to get the desired momentum thickness and velocity profile at the upstream of cavity for inlet boundary condition. Time averaged velocity profiles on desired lines are post processed using Paraview. Pressure fluctuations are transformed from time domain to frequency domain using a GNU Octave script. All the simulations are performed in two-dimensional space.

Keywords: Deep cavity, computational fluid dynamics, OpenFOAM, hybrid LES/RANS, improved delayed detached eddy simulation

1. INTRODUCTION

Since the first foundation of Large Eddy Simulation (LES) methods in 1960s, it has been used more and more by the academic turbulence researchers. It is still not practical to use LES models in industrial applications due to the high computational costs in comparison with RANS models. The main reason for high computational cost demand of LES methods in turbulent flows is turbulent boundary layers and it is even more problematic for high Reynolds numbers. The idea of using wall-modeled LES became feasible to reduce computational cost and still have a good match with exact solution or experiments. The main advantage of wall-modelled methods is to model the turbulence in inner boundary layer and avoid high computational cost of resolving turbulent eddies in inner boundary layer (Larsson continued, 2016). In the same study (Larsson continued, 2016), it is suggested that as the Reynolds number increase, computational cost of LES models dramatically increases over wall-modelled LES models. In the earlier applications of LES methods, researchers tend to use wall stress models. Starting with late 1990s, the tendency changed to use hybrid LES/RANS models which are the main methods used in this study.

Researchers suggests in their study (Argyropoulos and Markatos, 2014) that Detached Eddy Simulation (DES) models can be used for aerospace, automotive and construction industries to investigate parts of flying vehicles, ground vehicles, flow control mechanisms, cavitation and engine air inlets. The need of eliminating log-layer mismatch (LLM) problem of DES models and constructing a concrete one set of formulas for DDES applications lead researchers to suggest IDDES model (Shur continued, 2008).

Although previous studies are conducted on the problem concerning the characterization of acoustic behavior near cavity, high subsonic flow deep cavity problem is firstly studied experimentally in the beginning of millennium where the flow field observed and captured as schlieren pictures (Forestier continued, 2003). Large eddy simulations performed with the same features of the study (Larcheveque continued, 2003). The reason to choose a deep cavity is to
reduce the third dimensional effects of cavity on flow field (Forestier continued, 2003). If the cavity length to depth ratio is smaller than 1, the cavity is defined as a deep cavity (Larcheveque continued, 2003).

Figure 1. Basic sketch of flow domain

In present study, high Mach number subsonic compressible flow over a deep rectangular cavity problem is investigated using OpenFOAM® as open source computational fluid dynamics software.

2. METHODOLOGY

OpenFOAM® v1806 provides a wide range of boundary condition, solver and numerical scheme options. Since it is described as transient solver for turbulent flow of compressible fluids, rhoPimpleFoam is chosen as solver to perform the simulations. BlockMesh utility is used to generate mesh. Geometry is divided into 4 blocks. Upstream and downstream blocks divided into 4000x60 grading elements. Cavity itself divided into 100x80 grading elements and the block between cavity, upstream and downstream is divided into 100x60 elements. Figure 2 shows the mesh density around cavity.

Figure 2. Grid around cavity.

To obtain the desired momentum thickness at the beginning of cavity, it is considered to provide velocity profile inlet boundary condition instead of uniform velocity.

$$\theta = \int_{y_{min}}^{y_{max}} \frac{y \nu}{U_e} \left(1 - \frac{y}{U_e} \right) dy$$ (1)
In equation (1), $\theta$ denotes momentum thickness, $y_{\text{min}}$ and $y_{\text{max}}$ are the lower and upper coordinates of plates, $U_e$ is the free stream velocity and $\overline{U}$ is the mean longitudinal velocity component at corresponding $y$ coordinate. To achieve the required velocity profile, 2D flow simulations in a rectangular domain is performed. The domain dimensions are 8 m x 0.1 m. Wall normal direction is divided in to 60 grading elements which is exactly same as the main simulations. Figure 3 shows the grid used for the initial simulations performed to achieve inlet velocity profile boundary condition.

Figure 3. Grading mesh in wall normal ($y$) direction for the initial simulations to get inlet boundary condition velocity profile.

Figure 4 shows the velocity field of the simulation made to get velocity profile which gives the desired momentum thickness. It is seen that a uniform velocity is applied at inlet, and a non-uniform velocity profile is developed through the x direction.

Figure 4. Velocity field of the initial simulation.

Following the simulation, fully developed velocity profile is taken at 1 mm upstream of cavity and momentum thickness is calculated for the velocity profile using a GNU Octave script.

Figure 5. Final velocity profile.
After the simulations, cell center x direction velocity values calculated using Paraview. Obtained velocity profile is modified such that freestream velocity will be equal to the value which makes the Mach number equal to 0.8. This procedure is repeated until the desired momentum thickness is achieved at 1 mm upstream of cavity. Figure 5 shows the final velocity profile obtained and used in the main simulations.

As turbulence model, k-ω Shear Stress Transport IDDES (Gritskevich continued, 2012) and Spalart-Allmaras IDDES (Shur continued, 2008) models are used. The general idea of hybrid LES/RANS models is to reduce computational cost of LES models by switching to RANS models when applicable. It is crucial to understand how IDDES models are switching from LES to RANS or vice versa to be able to comment on results. The model behaves as RANS model at inner boundary layer which is in the area flow attached to the wall and behaves like LES model out of this area. In IDDES models, not only cell dimensions, but also wall normal distance of cell is also a parameter to decide the grid filters (Verhoeven, 2011). In addition to Spalart-Allmaras IDDES and k-ω Shear Stress Transport IDDES simulations, a standard RANS k-ω Shear Stress Transport simulation, k equation LES and LES WALE simulation are performed.

3. RESULTS

Results of the CFD analysis are discussed in this section.

<table>
<thead>
<tr>
<th>Models</th>
<th>t=0 s</th>
<th>t=0.018 s</th>
</tr>
</thead>
<tbody>
<tr>
<td>k-ω Shear Stress Transport IDDES</td>
<td>3.03 s</td>
<td>90730.9 s</td>
</tr>
<tr>
<td>Spalart-Allmaras IDDES</td>
<td>4.21 s</td>
<td>89072.1 s</td>
</tr>
<tr>
<td>LES</td>
<td>3.78 s</td>
<td>45862.8 s</td>
</tr>
<tr>
<td>Standard k-ω Shear Stress Transport</td>
<td>2.89 s</td>
<td>42978.6 s</td>
</tr>
</tbody>
</table>

Table 1 show that both standard RANS k-ω SST model and k-ω SST IDDES’s execution times at first iteration are very small in comparison with other models. As the simulations proceed, LES model becomes faster. LES model and RANS k-ω SST model are nearly two times faster than IDDES models.

3.1. RANS/LES Content of IDDES Simulations

Figure 6 represents RANS (blue) and LES (red) content of IDDES near inlet location. At the initial time, IDDES simulations start as 100% RANS content and 0% LES content.

![Figure 6. LES content of k-ω SST IDDES around inlet for t=0.001, t=0.1 and t=0.2.](image-url)
As the simulation proceeds, LES becomes dominant. Although LES content becomes dominant, near wall areas do not switch to RANS because of the nature of IDDES method. At the final time step, LES content cover 82% of the total volume.

3.1. Flow Characteristics

Figure 7 represents an instantaneous visualization of flow at time is equal to 0.01325 seconds. It can be said that both IDDES and LES methods captures the reflected waves well while standard RANS k-ω SST gives more averaged results.

![Figure 7](image1)

Figure 7. Visualization of pressure field at t=0.01325 s. Above left k-ω SST IDDES, above right Spalart-Allmaras IDDES, below left standard k-ω SST and below right LES WALE model.

Figure 8 shows the vorticity fields in simulations. It is clearly seen that LES and IDDES models are capable of capturing vortices while RANS k-ω SST model gives more averaged figure as expected.

![Figure 8](image2)

Figure 8. Visualization of vorticity field at t=0.01325 s. Above left k-ω SST IDDES, above right Spalart-Allmaras IDDES, below left standard k-ω SST and below right LES WALE model.
3.2. Mean Velocities in the Vicinity of Cavity

Mean velocity profiles obtained for three different locations in the area where free stream flow and cavity flow meets. Three lines on x=0.2L, x=0.4L and x=0.8L are chosen. Then three lines with 0.4L length placed in y direction and centralized at y=0. Temporal statistics obtained for these three locations.

In Figure 9, results show that standard RANS k-ω SST model fits well with the experimental data considering horizontal velocity profiles. LES WALE model also agrees with the solution above shear layer, but it does not fit with the experiment in cavity because of over-predicted gradients.

In Figure 10, vertical velocity profile results show that until the 0.4L of cavity simulations agree with experimental data above shear layer. Simulations over-estimates vertical velocities at the 0.8L distance of cavity.
3.2. Acoustics in Cavity

Pressure fluctuations during the flow are noise sources. Pressure fluctuations obtained from simulations shown below in Figure 10.

![Figure 11. Pressure fluctuations over time on the point on the upstream wall of cavity at y= -0.7L.](image)

Figure 11 shows that until 0.008 seconds, pressure fluctuation shows different behavior than rest of the simulations. It is the time when flow reaches to the probe point which is on the upstream wall of cavity at -0.7L distance from origin in y direction. These first 0.008 second data is not considered when calculating sound pressure levels of simulations at given point. Formula given in Equation 2 is used to calculate sound pressure levels.

\[
SPL = 20 \left( \frac{p_{ref}}{p_{ref}} \right)
\]  

(2)

Here \( p_{ref} \) denotes the reference pressure which is typically equal to 2e-5 Pa. Then using the FFT function facility of GNU Octave, sound pressure level and time data transformed to frequency domain.
If we focus on the full spectrum results in Figure 12, it is seen that standard LES model is in line with the experimental data. It can be also noticed that standard RANS k-ω SST model’s fluctuations are negligible after the first four peaks.
The solution is unsteady, but the model pretends like as a steady solution after converging. It is clearly seen that all the simulations give higher noise values than experimental data. The simulations carried out modelling upper wall as a wall boundary instead of modelling as a flat plate. This causes waves to reflect from upper wall and pretend like an additional noise source which results in higher sound pressure level values at given point.

![Graph 1](Image)

![Graph 2](Image)

![Graph 3](Image)

![Graph 4](Image)

**Figure 13.** Zoomed sound pressure levels on the upstream wall of cavity at y= -0.7L.

It is seen from zoomed results in Figure 13 that all the solutions give higher sound pressure levels in comparison with experimental data. Although all simulations are successful to capture the first peak of spectrum, only standard LES model and standard k-ω SST model and Spalart-Allmaras IDDES models succeeded to capture second peak as well.
3. CONCLUSION

Transient compressible flow simulations performed for subsonic high Mach number deep cavity flows using OpenFOAM®. GNU Octave and Paraview used for post processing. IDDES model capabilities are investigated comparing with standard RANS and LES models. Flow characteristics, time averaged velocity profiles and acoustic data are analyzed. IDDES models are successful to capture the reflected waves around cavity while standard RANS k-ω SST model is not good enough to capture these waves. Time average velocity comparisons show that nearly all models are giving reasonable results for horizontal velocity. In contrast, while the models capture well above shear layer, they are over – estimates the vertical velocity profile below shear layer. Acoustic results show that a filtering process should be performed before calculating noise data to have more accurate results. It is also seen that IDDES models takes more computation time comparing with LES and RANS models for same mesh. For future work, IDDES coarse mesh and LES fine mesh simulations should be performed and compared. Three dimensional simulations can be also performed to compare.

REFERENCES

TURBOJET ENGINE AIR INTAKE MODAL ANALYSIS USING IMPACT TESTING METHOD

Özgürcan Yıldız¹a, Mümtaz Aşım Esi²b, Tuncay Karaçay²c

¹Tactical Missile Systems Directorate, Roketsan Inc. Ankara, Turkey
²Department of Mechanical Engineering, Gazi University, Ankara, Turkey

aozgurcan.yildiz@roketsan.com.tr, bmesi@roketsan.com.tr, ckaracay@gazi.edu.tr

ABSTRACT

Modal Analysis has been an improving scientific evaluation of the dynamic properties of a structure. Sometimes models produced by modal tests may have low quality because of the factors involved in the measurement. At this point, the structures near the rotating parts such as motor shafts are quite critical for a missile. It is very important that the natural frequencies of these parts do not coincide with the engine operating frequency. For this reason, the accuracy of the measurement becomes more important. Within this paper, frequency response functions and mode shapes of a turbojet air intake is determined using impact testing method from which mode shapes extracted. The air intake is both tested for free-free condition and bounded flight condition under different loadings. After that, frequency response functions and mode shapes are compared for the cases from the data obtained at same location on the structure.

Keywords: Modal analysis, Impact testing, Mode shape, Turbojet engine air intake.

1. INTRODUCTION

Impact hammer modal testing is used in a variety of engineering applications and the speed of the method is an important advantage especially for the limited time cases in the industry. If most of the design decisions could be made with impact testing, it could improve the efficiency of development process. With the rise in the missile technologies in engineering, it is becoming essential to find an effective and quick solution to dynamic characterization process. The effects of static forces can be easily predicted, but for dynamic forces unwanted responses may occur. Most of the cases finite element modelling is need verification for the nonlinear elements such as some kind of springs, bolts etc. Therefore, the dynamic character of the systems must be well established by the tests. At this point, not only the system but also the effects of each part to be added to the system on the local and global system should be examined. This report presents a case study on impact hammer testing concept for the missile technologies.

A turbojet engine typically consists of high and low-pressure compressors, combustor, and high and low-pressure turbines. In the turbojet, all the inlet air passes through each element of the engine [1]. A missile is exposed to an extraordinary dynamic behavior during lunch and maneuvering at maximum thrust. Therefore, the dynamic behavior of an inlet is very crucial. If the air intake exhibits any undesirable behavior during the flight, the engine performance could be adversely affected.

In some of the cases system cannot be described as a single degree of freedom system since systems consist infinite number of masses and connection elements. There are many methods for modeling a multi-degree of freedom system. The simplest of these forms is the method in which the mass is distributed along the system and represented as a matrix and masses are connected by elastic and damping elements which is also represented in a matrix form. Equation of motion for a multi-degree of freedom system is given by the Equation 1 where [m] mass matrix, [c] is damping matrix, [k] is stiffness matrix, \( \vec{x} \) is displacement vector and \( \vec{F} \) is force vector. [2].

\[
[m]\ddot{\vec{x}}+[c]\dot{\vec{x}}+[k]\vec{x}=\vec{F}
\]  

In its simplest form, the frequency response function (FRF) is the transfer function of a system's output against an applied input. To obtain FRF of a structure first, the analog signals are collected and filtered from high frequencies.
Then these signals are converted from analog to digital with specific converters called analog to digital converters (ADCs). In order to calculate FFT, collected data must be continuous when switching from time data to frequency data. At this point, to satisfy periodicity requirement generally windowing functions are used. Once the FFT determined, then input power spectrum, cross power spectrum and output power spectrum are calculated. These functions are used to specify FRF and coherence functions. Coherence indicates that how input signal is correlated with the output signal when determining the response of the system to the input [3].

2. EXPERIMENTAL SETUP

The aim of the test was investigating the dynamic behavior of air intake and experimental based redesign if needed. The study is conducted to validate the specification that air intake natural frequencies do not coincide with the engine operating frequency. Geometry of air intake is shown in Figure 1.

![Air intake geometry](image)

**Figure 1. Air intake geometry**

2.1 Excitation Type and Data Acquisition System

The choice of the tip in the hammer excitation is important because it determines the frequency bandwidth. In this study hard metal tip is used to excite the structure. Sufficiency of the tip can be controlled by checking dB level change before the test. It is stated that the useful frequency range is from 0 Hz to a frequency \( F \), at which point the spectrum magnitude has decayed by 10 to 20 dB [4]. Figure 2 shows that from 0 Hz to 2048 Hz energy level drops only 5 dB. To collect the data LMS SCADAS SCR205 type data acquisition system is used.

![Frequency range](image)

**Figure 2. Frequency range where the available energy is concentrated**

2.2 Excitation/Response Locations and Acquisition Settings

10 tri-axial (PCB 356A16 model) accelerometers are spread around the air inlet. For the free-free condition air intake hanged with elastic ropes. Before the test, the free bounce mode of the system is measured as 1.58 Hz. In theory that should be zero, but it is not realistic in experimental world. It is investigated and recommended as a rule of thumb that, the ratio of first rigid body frequency to the measured system frequency should be at least 10 times less [5]. Expected natural frequency values were at least 100 times higher in the case study.

Hit surfaces are chosen as flat as possible. To decrease variations between each hammer hit 20 averages are taken with implicit accept. For the bounded condition same analogy is used. The absence of any extra structure for impact hammer testing is advantageous in terms of not adding extra mass to the structure. To reduce the noise, 6 N trigger level used such that measurement start automatically when the impact hammer strikes the object and force level reached to 6 N.

Since the air intake is expected to be stiff and damped there is no windowing applied to the measurement. Therefore, there is no reduction or any artificial effect in the amplitude of the signal/window over the measurement time. Data
acquisition is applied with 3200 Hz bandwidth. Resolution of the measurement was 0.39 and so that 2.56 s acquisition time is expected to be enough for the system to be damped.

For the FRF estimation there are many methods. $\text{H}_1$ method assumes that there is no error on the references, $\text{H}_2$ method assumes that there is no error on responses. In this case, $\text{H}_\nu$ formulation is used which presuppose noise in both the input and output channels [6]. During the measurement acquisition system set up to detect and auto reject with overload and double impact. Experimental setup is shown with Figure 3.

![Figure 3. Air intake accelerometer locations and coordinates (a) Bounded condition (b) Free-free condition](image)

3. RESULTS

PolyMAX (polyreference least squares complex frequency domain method) is used for the modal parameter estimation. PolyMAX method does not have problem from numerical errors since it works with a least squares sense. The method yields a very clean stabilization diagrams without increasing the solution time [7].

3.1 Natural Frequencies

Table 1 shows the results of the modal test as free-free condition modal test and bounded condition modal test within the scope of determination of the dynamic character of the air intake. According to this, the lowest natural frequency value of air intake was obtained as 1101.85 Hz for free condition and 59.06 Hz is obtained for bounded condition. There is no evaluation is done for the longitudinal direction. The modal assurance criterion (MAC) matrix of the modes is given in Figure 4.

![Table 1. Natural frequencies of the modal test](image)

<table>
<thead>
<tr>
<th>Air intake free condition natural frequencies (Hz)</th>
<th>Air intake bounded condition lateral (Y) natural frequencies (Hz)</th>
<th>Air intake bounded condition vertical (Z) natural frequencies (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1101.85</td>
<td>369.31</td>
<td>59.06</td>
</tr>
<tr>
<td>1194.51</td>
<td>687.32</td>
<td>681.68</td>
</tr>
<tr>
<td>1404.95</td>
<td>1134.31</td>
<td>1055.43</td>
</tr>
<tr>
<td>1836.94</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
3.2 Mode Shapes

In any mode of a structure, the overall modal shape for the respective natural frequency is dominated by the resonance mode shape. Mode shapes determined for the free-free condition is given by Figure 5. Mode shapes determined for the bounded condition is given by Figure 6. In general, most displacement occurs in the regions on the mouth where the air diffuse to the engine.
4. CONCLUSIONS

All measurements are based on the principle of taking both drive and measurement points in the same way for both free-free and mounted conditions as much as possible. It should be taken into consideration that the test results presented in this report are obtained by using a fixture generated according to the connected situation scenario. It was not possible to get results with the test taken on the missile in the final case. This paper discussed the measured dynamic information to determine the dynamic behavior of an air intake under different boundary conditions.

Conclusively, MAC matrices indicate that the natural frequencies estimated by the hammer test show that vectors are well identical in the scale and modes are well excited. The mode shapes show that there is more translation and rotation movement at the boundaries of the air intake and that if the flight operation occurs at the detected natural frequencies, it may prevent the air flow to be taken into the engine efficiently.

ACKNOWLEDGEMENTS

We thank to ROKETSAN INC. TMTA Vibration and Modal Analysis Lab. in Tactical Missile System Engineering Directorate for the inspiration and all publications support and staff, who provided helpful comments on this document. Some of the references cited in this paper are included for illustrative purposes only.

REFERENCES

MATERIAL MODEL DEVELOPMENT FOR NANOCOMPOSITES AND FINITE ELEMENTS IMPLEMENTATION

Alperen Acar¹ and Özgen Ü. Çolak²

Yıldız Technical University, Department of Mechanical Engineering, Istanbul
¹alpacar@yildiz.edu.tr, and ²ozgen@yildiz.edu.tr

ABSTRACT

Cooperative – VBO theory developed initially for polymers in the work by Colak et al. (2013) is extended in such a manner that viscoelastic and viscoplastic behavior of graphene-epoxy nanocomposites can be modeled. The influences of strain rate and temperature on the mechanical behavior are included into the model. Using the Takayanagi averaging approach, activation energy and activation volume which are two scalar valued material parameters of the plastic strain rate function, are defined as functions of graphene fraction and are incorporated into cooperative-VBO model.

Newly introduced material model for nanocomposites is implemented into a commercial finite element code. A computational procedure is defined using the explicit forward gradient method. For validation purposes, numerical simulations are also carried out for the material model using finite difference Gear’s method for stiff ODE’s. The modeling capabilities of the introduced model are demonstrated by predicting uniaxial compression behavior of epoxy and graphene-epoxy nanocomposites with different strain rates.

Keywords: viscoelasticity, viscoplasticity, VBO, finite element method, implementation

1. INTRODUCTION

The studies on the use of nanocomposite materials, especially graphene-epoxy nanocomposites, in aerospace, defense and automotive industry applications are performed intensively. In this context, both material synthesis and characterization and modeling of material behaviors are important.

The development of material models (constitutive equations) and the use of these models in structural analysis (the use of finite element method in the software such as ABAQUS, ANSYS) are very important for the design of structural components. However, based on the material models in the literature, these models are not sufficient to model the viscoelastic, viscoplastic behaviors and non-linear reinforcement mechanisms of the nanocomposite materials.

It is difficult to model the mechanical properties of nanocomposites which depend on many parameters such as the strain rate, temperature, reinforcement ratio, homogeneity of the distribution of the reinforcing element. Material models need to be developed in such a way as to include the effects of the above mentioned parameters on properties.

The main objective of this work is to develop a micromechanics based material model to simulate the viscoelastic and viscoplastic behavior of graphene-reinforced nanocomposite materials and to implement it ABAQUS software. Modeling capabilities are shown by simulating the compression behavior of graphene-epoxy nanocomposite at room temperature and different strain rate. Simulation results are compared to experimental results obtained by Shadlou et al. (2014).
2. CONSTITUTIVE MODEL AND SIMULATION RESULTS

Cooperative – VBO theory developed initially for polymers in the work by Colak et al. (2013) is extended in such a manner that viscoelastic and viscoplastic behavior of graphene-epoxy nanocomposites can be modeled. To be able to model the influences of temperature and strain rate, the effective modulus definition developed by Ji et al. (2010) is coupled with temperature and rate dependent elasticity modulus equation to simulate the viscoelastic response of graphene nanocomposites. Using the Takayanagi averaging approach, activation energy and activation volume which are two scalar valued material parameters of the plastic strain rate function, are defined as functions of graphene fraction and are incorporated into cooperative-VBO model. Model equations are not depicted in this paper since a detail definitions of each parameters and state variables are needed. For details of the model, the newly published paper by Acar et al. (2018) can be reviewed.

Even though graphene-epoxy nanocomposites exhibit nonlinear viscoelastic behavior, then following the yield, some amount of hardening and fracture under tensile loading, compression behavior is quite different than the behavior in tension. Under compression upon yielding, nonlinear hardening is observed.

The compression behavior of graphene-epoxy nanocomposite at different strain rates are modeled with newly developed model. The simulation results are depicted in Figure 1. The linear viscoelastic behavior which is followed by softening and a nonlinear hardening are captured quite well. The compression tests carried out at room temperature by Shadlou et al. (2014) are used for comparison and finding out material parameters of the model. Increasing rate increases the mechanical properties.

![Graph showing simulation of compression behavior of graphene-epoxy nanocomposites at different strain rates](image1)

Figure 1. Simulation of compression behavior of graphene-epoxy nanocomposites at different strain rates and comparison with experimental data by Shadlou et al. (2014). Graphene content is 0.25wt %.

3. FINITE ELEMENT IMPLEMENTATION

Cooperative-VBO model developed for graphene-polymer nanocomposites is implemented in the commercial FE code ABAQUS/Standard using a user-defined material subroutine UMAT (ABAQUS 2013). To develop a UMAT subroutine, it is necessary to write these equations in an incremental form using a suitable integration procedure. In this work, the forward gradient method is used for numerical integration of the differential values used in the model. The finite element model is defined with cubic elements of C3D8R. Time step used is 1e-5 s for all simulations.

The new proposed integration scheme for the proposed model results are compared to the model results obtained by Gear’s method. All results are taken using ABAQUS ® 6.13-1 and Parallel Studio XE 2013 as required by the finite element code. Figure 2 shows the ODE solution and FEM solution for epoxy and graphene-epoxy nanocomposite with 0.25 wt% graphene content.
4. CONCLUSIONS

Constitutive model for graphene-polymer nanocomposites is developed by modifying the cooperative-VBO model which is for polymeric materials. Following the model development, it is implemented into FEM code to be able to use in the structural analysis of the components. Simulations are performed to model compression behavior which consists of viscoelastic behavior followed by nonlinear hardening. Good match with experimental data is observed.

5. REFERENCES


DÜZ PERDELİ BİR SUSTURUCUDA PERDE KONUMUNUN SES DÜŞÜMÜNE ETKİSİNİN ARAŞTIRILMASI

Zeynep MUTLUM1, Hakan ARSLAN2 ve Doğancan MUTLUM3

1İstanbul Tersanesi Komutanlığı, 34890 Pendik, İstanbul, Türkiye
2Kırıkkale Üniversitesi, Makine Mühendisliği Bölümü, Kırıkkale, Türkiye
3Alfa Mühendislik, 34890, İstanbul, Türkiye
zeynepmutlum1991@gmail.com

ÖZET

Bu çalışmada, düz perdeli bir reaktif bir susturucuda perdenin yerleşimleri değiştirilerek 7 farklı model elde edilmiş ve bu modeller için ses iletim kayıları, MATLAB programı kullanılarak analitik olarak elde edilmiş, sonra Akustik Empedans Tüpü Deney sistemleri kullanarak da deneySEL olarak ölçülmüştür. DeneySEL ve analitik olarak elde edilen ses iletim kayıp (TL) değerleri karşılaştırılmış ve değerlerin birbirleri ile göreceli olarak uyumlu olduğu görülmüştür. Ayrıca tek perdeli susturucu modelleri için performans karşılaştırması yapmak amacıyla 0-3000 Hz frekans bölgesi için TL-RMS değerleri elde edilmiş ve iletim kaybı açısından en iyi performansı gösteren susturucu modeli belirlenmiştir.

Anahtar Kelimeler: Düzperdeli susturucu, Akustikanaliz, Ses iletim kaybı, Gürültü azaltımı

1. GİRİŞ

Susturucular; motor, fan, silah gibi ses üreten makinaların ortaya çıkan ses basıncını, iç içe geçmiş delikli borular, genleşme odaları ya da çapı nispeten büyük olma eğiliminde bulunan delikli borular genel olarak ses şelalesi olarak tanımlanır.

Yüksek ses sonucu ortaya çıkan gürültü, insan sağlığı için zararlı olup aynı zamanda kullanıma yeri göze özellikle sahne olanın, konservasyon ve performansı arttırmak için kullanılır. Susturucular daha ziyade gürültünün indirilmesi konusunda kullanılır.


Susturucunun, kullanıldığı sistemde performansın sebebi olan gürültüyü en aza indirmesi gerekmektedir. Bunun için susturucunun kullanılandan motor özelliklerine göre (gürültü frekansı, motor devri, ve performansı) belirlenmesi gerekmektedir. Reaktif susturucularda, akış ve akustik özellikleri etkileyen iç tasarım parametreleri (porozite oranı, delik sayısı ve oda hacmi) optimizasyon ile edilmişdir. Çünkük bu parametrelerin değişimleri egzoz gazının akış hızını ve hacmini değiştirme eğiliminde olabilir [1].


Selamet (1997), düz giriş ve çıkış kanalında dairesel asimetrik genişleme odasında akustik yayılma üzerinde çok boyutlu dalga yayılma etkisini araştırmak için üç boyutlu analitik bir yaklaşım geliştirilmiş. Kanal giriş ve çıkışında düzlemlerin olmayan dalgının iletim kaybını performansı üzerinde etkileri incelenmiştir [5].Hudson (1996), kücük kalibri silahların...
susturucu tasarmında hesaplamları ağırlayan diğer modellendendi yerdenmiştir. Prototip susturucu için deneySEL
öLCMLER yapmışlar ve simulasyonlari iele kompleks modelden doğrulugunu belirlemiştir [6].


Bu çalışmada, düzel perdeli bir reaktif susturucuda perdenin konumu değişdirilerek 8 farklı model elde edilmiştir ve bu modeller için akustik hesaplamlarında Transfer Matris Metodu (TMM) kullanılarak analitik ve deneySEL olarak ses iletim kayıpları elde edilmiştir. DeneySEL ve analitik olarak elde edilen ses iletim kayıf değerleri karşılaştırılması sonucunda, STl açısından en iyi performansı gösteren susturucu modeli belirlenmiştir.

2. MATERYAL VE METOT


Şekil 1. Çalışmada kullanılan susturucunun katı modeli
Susturucular kullanım amaçlarına göre dört gruba ayrılır. Bunlar, yutucu tip susturucu, yansıtıcı tip susturucu (reaktif), melez susturucu ve aktif susturucu şeklinde sınıflandırılır. Bu çalışmada Şekil 1'de görülen yansıtıcı tip susturucu kullanılmıştır [17].

2.1. Transfer Matrisi Metodu ile Ses İletim Kaybının Bulunması

Şekil 2'de bir düz perde, iki odacıklı bir susturucunun kesit görünüşü verilmektedir. Susturucu, beş adet elemandan oluşmaktadır. Bu elemanlar arasında ani genişleşme ve ani daralma şeklinde geçiş söz konusudur. Şekilde her bir kısımda numaralandırılmış olup, susturucu kısımları için belirlenen numaralandırma, tanımlanacak olan eşitliklerde indis olarak yer alacaktır.

Bir düz perdeli, iki odacıklı susturucu için TMM ile TL hesabı aşağıdaki gibi yapılmaktadır:

\[ M_{k} = \frac{S_{1}}{S_{2}}, \quad M_{j} = \frac{S_{3}}{S_{4}}, \quad M_{4} = \frac{S_{5}}{S_{4}} \quad \text{ve} \quad M_{5} = \frac{S_{1}}{S_{5}} \]

(1)

\[ S \text{ borunun kısımlarına ait kesit alanı, } d \text{ boru çapı ve } l \text{ uzunluk değişkenleri olmak üzere,} \]

\[ S_1 = \frac{\pi d_1^2}{4}, \quad S_2 = \frac{\pi d_2^2}{4}, \quad S_3 = \frac{\pi d_3^2}{4}, \quad S_4 = \frac{\pi d_4^2}{4}, \quad S_5 = \frac{\pi d_5^2}{4} \]

(2)

Borun kısımlarına ait akustik karakteristik impedans değişkenleri olmak üzere,

\[ Y_i = \frac{\rho_0 c}{S_1}, \quad Y_2 = \frac{\rho_0 c}{S_2}, \quad Y_3 = \frac{\rho_0 c}{S_3}, \quad Y_4 = \frac{\rho_0 c}{S_4}, \quad Y_5 = \frac{\rho_0 c}{S_5} \]

(3)

\[ Y_{ij} = \frac{\rho_0 c}{S_1}, \quad Y_{ij} = \frac{\rho_0 c}{S_2}, \quad Y_{ij} = \frac{\rho_0 c}{S_3}, \quad Y_{ij} = \frac{\rho_0 c}{S_4}, \quad Y_{ij} = \frac{\rho_0 c}{S_5} \]

(4)

\[ k_{ij} = \frac{k_0}{(1 - M_{ij})}, \quad k_{ij} = \frac{k_0}{(1 - M_{ij})}, \quad k_{ij} = \frac{k_0}{(1 - M_{ij})}, \quad k_{ij} = \frac{k_0}{(1 - M_{ij})} \quad \text{ve} \quad k_{ij} = \frac{k_0}{(1 - M_{ij})} \]

(5)

olarak elde edilir.

1 Düz Boru

Susturucunun 1 nolu kısmını düz bir boru olarak ele almamızdır. Burada, transfer matrisi \( T_1 \),

\[
\begin{array}{cccc}
1 & 34 & 2 & 33 & 5 & 12 & 23 & 34 & 45
\end{array}
\]

Şekil 2. Bir düz perde ı susturucunun şematik modeli
12 Ani Genişleme

Susturucunun 12 nolu kısmını ani genişleme olarak ele alınamılır. Burada, transfer matrisi $T_{12}$,

$$T_{12} = \begin{bmatrix} 1 & (1 + K_{d12}) M_2 Y_3 - M_1 Y_2 \\ 0 & 1 \end{bmatrix}$$

olar.

$K_{d12}$ statik basınç kaybı sabiti,

$$K_{d12} = \left( \frac{S_1}{S_2} - 1 \right)^2$$

olar.

2 Düz Boru

Susturucunun 2 nolu kısmını düz bir boru olarak ele alınamılır. Burada, transfer matrisi $T_2$,

$$T_2 = \begin{bmatrix} e^{-i M_2 k_2 l_2} \cos(k_2 l_2) & i Y_2 e^{-i M_2 k_2 l_2} \sin(k_2 l_2) \\ i e^{-i M_2 k_2 l_2} \sin(k_2 l_2) & e^{-i M_2 k_2 l_2} \cos(k_2 l_2) \end{bmatrix}$$

ile elde edilir.

23 Ani Daralma

Susturucunun 23 nolu kısmını ani daralma olarak ele alınamılır. Burada, transfer matrisi $T_{23}$,

$$T_{23} = \begin{bmatrix} 1 & (1 + K_{d23}) M_3 Y_3 - M_2 Y_2 \\ 0 & 1 \end{bmatrix}$$

olar.

$K_{d23}$ statik basınç kaybı sabiti,

$$K_{d23} = \left( \frac{1 - \frac{S_1}{S_2}}{2} \right)$$

olar.

3 Düz Boru

Susturucunun 3 nolu kısmını düz bir boru olarak ele alınamılır. Burada, transfer matrisi $T_3$,

$$T_3 = \begin{bmatrix} e^{-i M_3 k_3 l_3} \cos(k_3 l_3) & i Y_3 e^{-i M_3 k_3 l_3} \sin(k_3 l_3) \\ i e^{-i M_3 k_3 l_3} \sin(k_3 l_3) & e^{-i M_3 k_3 l_3} \cos(k_3 l_3) \end{bmatrix}$$

(12)

34 Ani Genişleme

Susturucunun 34 nolu kısmını ani genişleme olarak ele alınamılır. Burada, transfer matrisi $T_{34}$,

$$T_{34} = \begin{bmatrix} 1 & (1 + K_{d34}) M_4 Y_4 - M_3 Y_3 \\ 0 & 1 \end{bmatrix}$$

(13)
\( K_{d_{34}} \) statik basınç kaybı sabiti,

\[
K_{d_{34}} = \left( \frac{S_3}{S_4} - 1 \right)^2
\]

olur.

\[ T_{d} = \begin{bmatrix}
\frac{i}{Y_{33}} e^{(-i M_{x} k_{x} l_{x}) \cos(k_{x} l_{x})} & i Y_{43} e^{(-i M_{x} k_{x} l_{x}) \sin(k_{x} l_{x})} \\
\end{bmatrix}
\]

\[ (14) \]

4 \textbf{ Düz Boru }

Susturucunun 4nolu kısmı düz bir boru olarak ele alınmalıdır. Burada, transfer matrisi \( T_{d} \),

\[
T_{d} = \begin{bmatrix}
e^{(-i M_{x} k_{x} l_{x}) \cos(k_{x} l_{x})} & i Y_{43} e^{(-i M_{x} k_{x} l_{x}) \sin(k_{x} l_{x})}
\end{bmatrix}
\]

\[ (15) \]

45 \textbf{ Ani Daralma }

Susturucunun 45 nolu kısmı ani daralma olarak ele alınmalıdır. Burada, transfer matrisi \( T_{45} \),

\[
T_{45} = \begin{bmatrix}
1 & (1 + K_{d_{45}}) M_{y_{5}} - M_{y_{4}} \\
0 & 1
\end{bmatrix}
\]

\[ (16) \]

\( K_{d_{45}} \) statik basınç kaybı sabiti,

\[
K_{d_{45}} = \left( \frac{1 - S_{5}}{2} \right)
\]

\[ (17) \]

5 \textbf{ Düz Boru }

Susturucunun 5 nolu kısmı düz bir boru olarak ele alınmalıdır. Burada, transfer matrisi \( T_{5} \),

\[
T_{5} = \begin{bmatrix}
e^{(-i M_{x} k_{x} l_{x}) \cos(k_{x} l_{x})} & i Y_{53} e^{(-i M_{x} k_{x} l_{x}) \sin(k_{x} l_{x})}
\end{bmatrix}
\]

\[ (18) \]

olarak elde edilir.

Perdesiz susturucu sistemi için toplam transfer matrisi,

\[
T = \begin{bmatrix}
T_{11} & T_{12} & T_{21} & T_{22} & T_{34} & T_{45} & T_{56}
\end{bmatrix}
\]

\[ (19) \]

ile elde edilir.

\[
T = \begin{bmatrix}
T_{11} & T_{12} \\
T_{21} & T_{22}
\end{bmatrix}
\]

\[ (20) \]

Toplam transfer matrisinin her bir elemanına bağlı olarak susturucunun TL değeri,

\[
STL = 20 \log \left[ \frac{Y_{5}^{2}}{Y_{1}^{2}} \frac{1 + M_{j_{5}}}{2(1 + M_{j_{5}})} \frac{T_{11} + T_{24} + T_{25} + T_{34} + Y_{5}}{Y_{1}} \right]
\]

\[ (21) \]

eşitliği ile elde edilir.
2.2. Deneysel Yöntemle Ses İletim Kaybının Bulunması


Şekil 3. TL Ölçüm Deney Düzeneğinin Genel Görünüşü[13]

3. BULGULAR

3.1. Deneysel Olarak Bulunan TL Grafiklerinin Karşılaştırılması

Çalışmada kullanılan susturucuya ait modeller ve bu modellere ait perde konumları Çizelge 1 de görülmektedir. Bu modeller için yapılan deneylerden elde edilen TL-Frekans grafikleri Şekil 4 te görülmektedir.

Çizelge 1. Susturucu modelleri ile bu modellere ait perde sayışı ve konumları çizelgesi

<table>
<thead>
<tr>
<th>Susturucu Model No</th>
<th>(L_1(\text{mm}))</th>
<th>(L_2(\text{mm}))</th>
<th>(L_3(\text{mm}))</th>
<th>(L_4(\text{mm}))</th>
<th>(L_5(\text{mm}))</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>45</td>
<td>70</td>
<td>5</td>
<td>65</td>
<td>15</td>
</tr>
<tr>
<td>2</td>
<td>45</td>
<td>20</td>
<td>5</td>
<td>115</td>
<td>15</td>
</tr>
<tr>
<td>3</td>
<td>45</td>
<td>40</td>
<td>5</td>
<td>95</td>
<td>15</td>
</tr>
<tr>
<td>4</td>
<td>45</td>
<td>10</td>
<td>5</td>
<td>35</td>
<td>15</td>
</tr>
<tr>
<td>5</td>
<td>45</td>
<td>60</td>
<td>5</td>
<td>75</td>
<td>15</td>
</tr>
<tr>
<td>6</td>
<td>45</td>
<td>80</td>
<td>5</td>
<td>55</td>
<td>15</td>
</tr>
<tr>
<td>7</td>
<td>45</td>
<td>120</td>
<td>5</td>
<td>15</td>
<td>15</td>
</tr>
</tbody>
</table>

Şekil 4 te verilen grafikte, Model–2, 1000 Hz ile 2200 Hz frekans bölgesinde TL açısından en kötü performansı sergilerken, Model–1’ in 1000 Hz ile 2200 Hz frekans bölgesinde TL açısından en iyi performansı verdiği görülmektedir.
Şekil 4. Deneysel Olarak Bulunan TL Grafikleri


Çizelge 2. Susturucu Modelleri için TL–RMS tablosu

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>RMS (0-1000 Hz)</td>
<td>1,621988</td>
<td>1,905652</td>
<td>1,757637</td>
<td>1,595666</td>
<td>1,768925</td>
<td>2,02992</td>
<td>2,187446</td>
</tr>
<tr>
<td>RMS (1000-2000 Hz)</td>
<td>4,908196</td>
<td>3,085257</td>
<td>4,084484</td>
<td>4,709019</td>
<td>4,614193</td>
<td>3,585391</td>
<td>2,605634</td>
</tr>
<tr>
<td>RMS (2000-3000 Hz)</td>
<td>4,172991</td>
<td>5,446479</td>
<td>5,7823135</td>
<td>4,366643</td>
<td>5,310783</td>
<td>5,960317</td>
<td>5,349429</td>
</tr>
</tbody>
</table>

3.2. STL–RMS Değerleri Bakımından Deneysel Bulguların ve Matlab Bulgularının Yorumlanması


Çizelge 3. Susturucu Modelleri için Deneysel ve Matlab'tan bulunan TL–RMS değerleri

<table>
<thead>
<tr>
<th>Model No</th>
<th>Perde Sayısı ve Pozisyonları</th>
<th>Deneysel Sonuçlar</th>
<th>Matlab Sonuçlar</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Perde tam ortada</td>
<td>5,71907</td>
<td>5,56561</td>
</tr>
<tr>
<td>2</td>
<td>Perde Giriş en yakın pozisyon (2cm)</td>
<td>5,48592</td>
<td>5,39998</td>
</tr>
<tr>
<td>3</td>
<td>Perde Giriş yakın pozisyon (4 cm)</td>
<td>5,49907</td>
<td>5,44774</td>
</tr>
<tr>
<td>4</td>
<td>Perde Giriş yakın pozisyon (6 cm)</td>
<td>5,50938</td>
<td>5,46751</td>
</tr>
<tr>
<td>5</td>
<td>Perde Girişten uzak pozisyon (8 cm)</td>
<td>5,60045</td>
<td>5,46402</td>
</tr>
<tr>
<td>6</td>
<td>Perde Girişten uzak pozisyon (10 cm)</td>
<td>5,61791</td>
<td>5,53986</td>
</tr>
<tr>
<td>7</td>
<td>Perde Girişten uzak pozisyon (12cm)</td>
<td>5,61168</td>
<td>5,53812</td>
</tr>
</tbody>
</table>
4. SONUÇLAR
Çalışmada, tek perdeli reaktif tip susturucu seçerek 7 farklı perde konumunu için ses iletim kaybı (STL) değerlerinin tespiti hem Matlab programı susturucu ile deneyisel olarak yapılmıştır. Çalışmada önce, akustik dalga teoremi (TMM) metodu ile sabit cıdarlı düz borunun, bir perdeli, susturucular için genel transfer matrisi oluşturularak STL ve RMS hesapları yapılmıştır. Bu susturucu sistemlerinin transfer matrisi, giriş ve çıkış noktalarındaki durum değişkenine bağlı olarak elde edilmiştir. Bu durum değişkenleri, akustik empedans, susturucu elemanın boyu ve dalga sayısına bağlıdır. Susturucunun ses iletim kaybı, bu değişkenler ile akustik empedans kullanılarak Matlab üzerinde hesaplanmıştır.


5. REFERANSLAR
APPLICATION OF VARIABLE PITCHING TO THE BLADES OF A VERTICAL AXIS WIND TURBINE FOR PERFORMANCE IMPROVEMENT

Emre Alpman
Department of Mechanical Engineering, Marmara University, Istanbul, TURKEY.
emre.alpman@marmara.edu.tr

ABSTRACT
This study includes an active variable pitching strategy applied to a 3 bladed H-type Darrieus VAWT in order to improve its aerodynamic performance. Here a sinusoidal pitching motion is applied to the blades of the VAWT with amplitude of 15 degrees and a circular frequency equal to the rotation rate of the rotor. A phase lag is also added to the pitching motion of the blades in order to see its effect on the aerodynamic performance. The flow simulations are performed using the open source CFD software OpenFOAM. The flow domain is divided into stationary, rotating and rotating-oscillating sub-domains and arbitrary mesh interface boundary conditions were applied at the interfaces separating these domains. Predictions showed that a variable pitching strategy with a suitable phase lag can give an optimum aerodynamic performance at a given wind speed and rotation rate.

Keywords: Variable pitching, vertical axis wind turbine, computational fluid dynamics, OpenFOAM.

1 INTRODUCTION
Wind turbines placed on top of buildings in urban environments allow the clean energy to be produced where it would be used. It is known that wind turbines operating in urban areas may be subjected to winds with varying directions (Kaldellis, et. al., 2012; Cace, et. al., 2007). Among the two types of wind turbines classified according to the rotation axis of the rotor, vertical axis wind turbines provide many advantages over horizontal axis ones like accepting wind from any direction (Cace, et. al., 2007) or allowing the gearbox and the generator be placed close to the ground (Paraschivoiu, 2002), etc. However, the blades of a VAWT operate at unsteady angle of attack even when they are subject to steady wind conditions (Cace, et. al., 2007). Therefore, there may be huge variations in the torque produced by them (Eriksson, et. al., 2008; Paraschivoiu, 1983), which is typically undesirable. Sweeping the blades of a turbine was shown to address this issue (Paraschivoiu, 1983; Usanmaz and Alpman, 2017; McIntosh and Babinsky, 2009) decrease the amplitude of the torque oscillations. However, even though sweeping the blades helps reduce the so-called torque ripple, it does not fully improve the mean torque production. This is mainly because the varying angle of attack problem encountered by the VAWT blades still remains even if the blades are swept.

Figure 1 displays the variation of angle of attack of a blade with respect to the azimuth angle during one revolution of the blade for different tip speed ratios (TSR), which is the ratio of blade speed to the freestream wind speed (Paraschivoiu, 2002). Here, the zero azimuth angle corresponds to the position when the blade leading edge is facing the oncoming wind. The velocity induced by the blade motion and the wake effects are neglected in the angle of attack calculation of Figure 1. It is clear from this figure that the angle of attack of a blade shows considerable variations and its value may become very high especially at low TSR. At these high angles of attack the blades will be subjected to dynamic stall effects and its performance will degrade (Paraschivoiu et. al., 2009).
An alternative approach to sweeping the blades is to apply a variable pitching procedure (Sagharichi et. al., 2016, Jakubowski et. al., 2018), which can provide some control over the angle of attack of the blades. In this approach the angle of attack of the blades is aimed to be kept in the attached flow region by pitching the blades about and axis typically passing through its chord. Pitching mechanism is also applied for horizontal axis wind turbines (HAWT) to prevent or delay the stall effects (Burton, et. al., 2011). However, unlike the HAWTs the pitching applied to a VAWT blade should depend on the blade position and therefore, a periodic pitching approach should be employed (Sagharichi et. al., 2016, Jakubowski et. al., 2018). In (Kirke and Lazauskas, 1991) a “self-acting” pitch control mechanism was applied to a straight bladed Darrieus type VAWT. Here the blade pitching was realized by the aerodynamic forces and inertial forces of a stabilizer mechanism rather than a forced motion. The mathematical model applied showed that this pitching mechanism improved the performance of the turbine however, it was also mentioned that the problems due to complexity of the system might outweigh the gains. A passive variable pitch mechanism was developed and tested in (Pawsey, 2012) for VAWTs. Theoretical and experimental studies performed showed that the starting performance of a VAWT can be improved by applying a passive pitch control mechanism. In (Elkhoury, et. al., 2015) an active pitch mechanism consisting of a four bar linkage mechanism was connected to a three bladed Darrieus type VAWT. The mechanism had an eccentric rotation axis which was different than the rotation axis of the rotor. This way the circular frequency of the pitch angle variation was equal to the rotation rate of the rotor. Their computational and experimental predictions showed that the pitching mechanism increased the mean power produced especially at low-to-moderate TSRs. Reference (Sagharichi et. al., 2016) contains the application of an active variable pitching mechanism to a three bladed Darrieus type VAWT, which contained a “cam with three spools rotating on it”. The pitch angle was controlled by the eccentricity between the rotation axis and the cam. The resulting pitch angle had a sinusoidal variation with a circular frequency equal to the rotation rate of the rotor. Their computational predictions showed that the variable pitching could successfully moderate stall, suppress the oscillation in torque and vortex formation. According to these information gathered from the references mentioned above it is clear that a pitching mechanism which decreases the angle of attack of the blades would successfully improve the mean and unsteady power production of a VAWT. A sinusoidal pitching strategy with a circular frequency equal to the rotation rate of the rotor would be suitable.

This study contains a periodic pitching strategy applied to the blades of a three bladed H-type Darrieus VAWT. In (Yucal, 2018) the a sinusoidal pitching strategy with circular frequency different than the rotor angular speed was tested and it was shown that the best performance was obtained when the circular frequency is equal to the rotation rate of the blades. Therefore, this approach is also embarked on in this study. The sinusoidal variation of pitch angle is similar to the one used in reference (Sagharichi et. al., 2016). However, a phase lag is also introduced to the sinusoidal term of the equation of pitching motion in order to see its effect on the unsteady performance of the VAWT.

In this study, flow field simulations are performed using computational fluid dynamics (CFD) via the open source CFD software OpenFOAM (https://openfoam.org/). The rotor has a radius of 0.5 m and contains the 16.03% thick MH-94 airfoil section (http://airfoiltools.com/airfoil/details?airfoil=mh94-il). This airfoil is selected due to its relatively high thickness and good stall characteristics. The flow field is assumed to be two-dimensional for simplicity. Numerical solutions are obtained at a rotation rate of 250 rpm and at a uniform wind speed of 5 m/s. This rotation rate is selected
because the corresponding TSR is close to the optimum TSR predicted by the QBlade software (https://sourceforge.net/projects/qblade/) which used the double multiple streamtube methodology (Paraschivoiou, 2002). The effect of the pitching procedure on the performance is investigated by comparing the torque predictions with that of the rotor with fixed pitch blades.

2 METHODOLOGY
2.1 Variable Pitching Strategy

The equation of pitching motion employed in this study is given in Equation 1.

\[
\beta(t) = \beta_0 - A \sin(\omega t - \psi)
\]

Here, \( t \) is time (in seconds), \( \beta \) is the blade pitch angle (angle between the chord line of the blade and the blade path), \( \beta_0 \) is the pitch angle for the fixed pitch case, \( A \) is the amplitude of the sinusoidal pitch, \( \omega \) is the rotation rate of the rotor (in rad/s) and \( \psi \) the phase lag. In this study \( \beta_0 \) is taken as zero and the value of \( A \) is set to 15 degrees. The phase lag values changes between 0 and 35 degrees with a step size of 5 degrees. These values are of course converted to radians before being submitted into Equation 1. Figure 2 displays the variation of angle of attack of the blade with respect to the azimuth angle for variable pitching with different phase lag values. Once again the velocity induced by the blade motion and the wake effects are neglected in the angle of attack calculation and the zero azimuth angle corresponds to the position when the blade leading edge is facing the oncoming wind. The figure also contains the angle of attack distribution for the fixed pitch case for reference.

![Figure 2 Variation of angle of attack with respect to azimuth angle for different phase lag values](image)

It is clear from Figure 2 that the angle of attack exceeds the stall value of the airfoil beyond which the lift-to-drag ratio decreases considerably (see http://airfoiltools.com/airfoil/details?airfoil=mh94-il). Applying the variable pitching technique successfully reduces the angle of attack to values corresponding to high lift-to-drag ratio values. However when the sinusoidal pitching is applied without the phase lag the angle of attack of the blade becomes too low especially when the azimuth angle is less than 60 degrees. This decreases the lift force provided by the blade. Since the blade produces the largest amount of torque when the azimuth angle is between 0 and 90 degrees (Paraschivoiou, 2002), increasing the angle of attack to better values (corresponding to larger lift-to-drag ratios) would be beneficial. It is evident from Figure 2 that applying a phase lag improves the angle of attack when azimuth angle is less than 90 degrees and as the amount of the lag is increased the variation of angle of attack also decreases in this region. However, this situation is reversed when the azimuth angle is approximately between 120 and 270 degrees. However, when the azimuth angle exceeds 180 degrees the blade is at the downstream half of the rotor area and it moves into the wake of other blades. Because of this, a blade is typically exposed to a non-uniform and lower wind speed than the freestream wind speed when its azimuth angle is between 180 and 360 degrees. Therefore, it is expected to have the gain in the upstream half of the rotor area outweigh the loss in the downstream half.

2.2 Numerical Procedure and Computational Mesh

The flow domain constructed for this study is displayed in Figure 3. This domain is very similar to the domains used in (Saghari et. al., 2016) and (Elkhoury, et. al., 2015). Here the domain extends 3 rotor diameters upstream, 10 rotor diameters downstream and 5 rotor diameters sideways from the rotation axis. The outer rectangular domain shown in
Figure 3 is stationary while larger circular domain inside the rectangular one is rotating with the blades. The three smaller circular regions located inside the larger circular region contain the blades. These regions also rotate with the blades but they also oscillate according to the sinusoidal pitching motion described in Equation 1. The freestream wind velocity is in horizontal direction from left to right.

The computational mesh generated for this domain is shown in Figure 4. This mesh consists of 234216 cells and is constructed using the open source mesh generation software cfMesh (https://sourceforge.net/projects/cfmesh/). The cell size is reduced in the vicinity of the rotating domain and the blades as shown in Figure 5 which contains a close up view of the mesh near one of the blades. In this figure the position of the blade corresponds to the zero azimuth angle and the blade pitch angle is zero. When the variable pitching procedure is applied for the rotor the blades start from a different angle as shown in Figure 6, which shows the same blade when the phase lag angle is 35 degrees.

Numerical solutions are obtained using the open source CFD software OpenFOAM. Here the pimpleDyMFoam solver of the software, which can handle incompressible flow fields with dynamic meshes, is employed for the predictions. The flow field is assumed to be fully turbulent and Menter’s k-ω SST turbulence model (Menter, 1994) is used for turbulence simulations.
Velocity is specified at the inlet boundary (leftmost boundary in Figure 3) while pressure is specified for the outflow boundary (rightmost boundary in Figure 3). Farfield boundary conditions applied for the upper and the lower boundaries of the domain and no-slip condition is applied on the blade surfaces. At the interfaces between the subdomains of the flow-domain cyclicAMI boundary condition of OpenFOAM software, which can handle sliding mesh interfaces, is applied. For the turbulence quantities, the turbulence intensity is taken to be 0.1% and the ratio of eddy viscosity to molecular viscosity was set to 0.1 at the inlet boundary. Wall functions are applied for turbulent flow on the solid boundaries with kqRWallFunction for turbulent kinetic energy and omegaWallFunction for the specific rate of dissipation.

3 RESULTS AND DISCUSSION

Numerical solutions are performed for 0.72 seconds of real time which corresponds to three rotor revolutions. As mentioned before the freestream wind speed is 5m/s and the rotor rotated at a speed of 250 rpm. The variable pitching procedure was applied according to Equation 1 along with the parameters described in section 2.1. For the fixed pitch solutions the parameter $A$ in Equation 1 is simply set to zero.

Rotor torque per unit span ($T'$) predictions obtained for fixed pitch configuration and variable pitch configurations with different phase lag angles are displayed in Figure 7. The improvement achieved by applying the variable pitching operation is evident from this figure. Even when the phase lag is not applied the amplitude of the oscillations is decreased. Variable pitching also eliminates the negative torque productions which would try to slow down the rotor. One can also observe the effect of the phase lag on the aerodynamic torque. Application of the phase lag clearly decreased the amplitude of the oscillations and increased the mean torque compared to the no phase lag predictions. However, the actual effect of phase lag angle on the mean torque and the oscillation amplitude are not very well understood from this figure. In order to see these effects better, mean rotor torque per unit span and the standard deviation of torque around the mean are displayed in Figure 8 and Figure 9, respectively. These figures clearly show that except for the phase lag angle of 5 degrees, increasing the phase lag angle gradually increases the mean torque production and decreases the standard deviation about the mean. Even though the mean torque continued to increase with phase lag angle, the standard deviation became minimum for $\psi = 30$ degrees and then slightly increased for $\psi = 35$ degrees. Based on these predictions one can conclude that an optimum performance can be achieved by applying a 30
degrees of phase lag angle. Because compared to the $\psi = 35$ degrees case the standard deviation is decreased by 5.64% with only a 0.88% decrease in the mean torque.

In order to understand the performance decrease observed for $\psi = 10$ degrees (also for $\psi = 15$ degrees), the distribution of Q-criterion is plotted for $\psi = 0, 5$ and 10 degrees at $t = 0.72$ s in Figure 10. According to the figure, the Q-criterion in the vicinity of the top and left most blades are similar however, there is a clear difference in the wake of the rightmost blade. It is obvious that the wake of this blade for the $\psi = 5$ degrees case is smaller than the others, which clearly indicates less amount of flow separation. Hence the $\psi = 5$ degrees case produced higher mean torque with less standard deviation about the mean compared to the $\psi = 0$ and 10 degrees cases.

Figure 7 Variation of instantaneous torque per unit span with time

Figure 8 Mean value of the produced torque per unit span
Figure 9 Standard deviation of produced torque per unit span about the mean value
CONCLUSIONS

An active variable pitching strategy was applied to a 3 bladed H-type Darrieus VAWT in order to improve its aerodynamic performance. Here a sinusoidal pitching motion was applied to the blades of the VAWT with amplitude of 15 degrees and a circular frequency equal to the rotation rate of the rotor. This type of pitching motion was studied before however, this study also introduced a phase lag to the pitching motion of the blades. The flowfield was assumed to be two-dimensional for simplicity and the simulations were performed using the open source CFD software OpenFOAM. The flow domain was divided into stationary, rotating and rotating-oscillating sub-domains and arbitrary mesh interface boundary conditions were applied at the interfaces separating these domains. The numerical solutions were obtained using the pimpleDyMFOam solver of OpenFOAM along with the k-ω SST turbulence model. The following conclusions could be drawn from the predictions obtained.

- As expected the variable pitching strategy improves the unsteady and mean torque production performance of a VAWT by decreasing the angle of attack of the blades.
- Adding a phase lag to the pitching motion further enhanced the performance by improving the angle of attack variation of the blade when it is at the upstream half of the rotor area.
- The phase lag addition aggravated the angle of attack distribution of the blade when it is at a downstream location, however, the improvement in the upstream location outweighed this aggravation due the non-uniform and lower wind speeds encountered by the downstream blades.
- The optimum performance was obtained for a phase angle of 30 degrees.

REFERENCES


ESTIMATION OF EFFECTIVE ELASTIC PROPERTIES OF HONEYCOMBS WITH 1st ORDER CIRCULAR HIERARCHY

Mustafa Yıldız\textsuperscript{a}, Fatih Göncü\textsuperscript{b}

Yıldırım Beyazıt University Faculty of Engineering and Natural Sciences, Mechanical Engineering Department, TURKEY
\textsuperscript{a}m.yildiz@ybu.edu.tr, \textsuperscript{b}fgoncu@ybu.edu.tr

ABSTRACT
We study analytically and numerically the in-plane mechanical response of honeycomb structures with circular hierarchy. The first-order hierarchy is introduced by replacing the junctions of cell-walls in the base hexagonal geometry with circular elements. Effective elastic modulus and Poisson’s ratio are calculated using Castigliano’s theorem applied to a representative element of the structure deduced from symmetry arguments. The effect of the hierarchy scaling factor which controls the size of the circular geometry, and the effect of wall thickness on the elastic properties is studied. We find that by changing the hierarchy scaling factor and wall thickness, structures with 2 times the stiffness and 0.3 times the effective Poisson’s ratio of a regular honeycomb having the same average density, can be obtained. Furthermore our results indicate that the circular hierarchy and changing the wall thickness offers a slight advantage over the 1\textsuperscript{st} order hexagonal hierarchy studied by Ajdari et al. [1], in terms of effective elastic properties. Finally we perform finite element simulations of infinite structures using periodic boundary conditions to validate our analytical findings and observe an excellent agreement between theory and numeric for small relative-densities.

Keywords: Effective properties, Honeycomb, Circular hierarchy, Castigliano’s Second Theorem, Finite Element Method, Periodic boundary conditions.

1. INTRODUCTION

There is an abundant use of hierarchical cellular structures made of different materials, and of various sizes in nature. This can be exemplified by a human bone rich in hierarchy [2-11]. Hierarchy makes a significant contribution to vital characteristics of organisms such as being able to survive in the environment in which they live in addition to their other features. Inspired by nature, engineers, designers, and architects incorporate hierarchical structures in their work. Examples range from architectural structures such as Eiffel’s tower, Garabit viaduct [12], laminates consisting of laminae of different orientations or sandwich composite panels with foam cores [13] at macro-scale, to polymers exhibiting structural hierarchy [7] at micro-scale.

The primary way to increase the mechanical properties by maintaining the lightweight feature of materials is to use structural hierarchy. For this purpose foams, honeycombs or truss lattices can be used in sandwich structures [14-18]. Previous studies have shown that light-weight and better-performing structures can be achieved by introducing a first-order hierarchy and adjusting the scaling factor (i.e., length scales) [6,7,18-22]. In this study, we show that the heterogeneity level (i.e., wall thickness ratio, $a_t = t_h/t_n$) also influences the mechanical properties, and can be used to tune them to in addition to the hierarchy scaling factor.

In many engineering applications such as impact/blast energy absorption, thermal or sound isolation, 2-dimensional honeycomb structures are preferred due to their low density and high effective strength [23]. Under transverse loading, in-plane elastic properties such as stiffness and strength are mainly dependent on the bending deformation of cell edges [23]. Maximum values of the bending moments in each cell wall, which influence the effective properties of the structure, occur at the corners of the honeycomb. Therefore, the in-plane elastic properties tend to increase as the amount of material located at the corners of the structure is more than the material found in the middle parts of the model walls [24-25].
In our work, we replace the corners of the regular hexagonal lattice with smaller sized circles with a radius proportional to the edge length of the hexagon by a factor of $\gamma_1$. Consequently, a first-order hierarchical hybrid structure (i.e., models with different geometries for normal and hierarchical regions) is obtained. As a result of keeping overall density constant, the wall thickness of the structure is decreased. It has been shown by Ajdari et al. [1] that first-order hierarchical structures can show superior Young's modulus compared to regular honeycomb having the same density. Figure 1 shows the structural hierarchies of honeycomb structures investigated in this study.

Figure 1. Hierarchical honeycomb structures investigated in this study.

The dimension of the hierarchy is determined according to the scaling factor which is the ratio of the specific dimension of the introduced hierarchy (edge length for hexagon geometry and radius for circular geometry), to the edge length of the regular hexagonal lattice as described in Figure 1 (i.e., $\gamma_1^{\text{hex}} = b/a$ and $\gamma_1^{\text{circ}} = R/a$). For a first-order circular hierarchical honeycomb, $0 \leq R \leq a/2$ so that, $0 \leq \gamma_1 \leq 0.5$. The case, $\gamma_1 = 0$, where there is no hierarchy corresponds to a regular honeycomb. The relative density of a structure with unit height, is the ratio of the area cell walls projected on to the plane of the structured to the full area covered by the unit cell, which is given by:

$$\rho_{\text{circ-hex}} = \frac{2t_n}{a\sqrt{3}}\left[1 + \gamma_1\left(\frac{4\pi}{3}a_t - 2\right)\right]$$

(1)

For an ordinary honeycomb structure (i.e., when $\gamma_1 = 0$) $\rho = 2t/a\sqrt{3}$ and for a first-order circular hierarchical honeycomb structure with homogenous thickness $\rho_{\text{circ-hex}}^\text{hom} = 2t\left[1 + \gamma_1\left(\frac{4\pi}{3}a_t - 2\right)\right]/a\sqrt{3}$ (i.e., the case of $a_t = 1$). It is apparent in the relative density equation that the ratio, $t/a$ has to decrease as the $\gamma_1$ increases to keep the relative density constant.

We have studied the in-plane effective elastic properties of the circular hierarchical honeycomb analytically using Castigliano’s theorem, and numerically using Finite Element Analysis (FEA). The analytical model for estimating the normalized effective elastic modulus and effective Poisson's ratio of the circular hierarchical honeycomb is presented in Section 2 and 3, respectively. In Section 2, the loading conditions used to derive analytical model and boundary conditions applied in FEA are also summarized. In Section 4, the in-plane elastic properties of the circular hierarchical honeycomb are presented and compared with the hexagonal hierarchy studied by Ajdari et al. [1]. Also, the possibilities for further improvement of the mechanical performance of the structure is discussed.

2. NORMALIZED EFFECTIVE ELASTIC MODULUS

We use Castigliano’s Second theorem to analytically estimate the effective properties of the circular hierarchical structures under in-plane uniaxial loading. In this study, it is assumed that the structure is made of an isotropic linear elastic material with elastic modulus, $E_s$. It has been shown that lattices having three-fold symmetry exhibit macroscopically isotropic behavior [26]. Therefore, it is possible to describe their mechanical behavior conveniently by using only two constants. Figure 2(a) shows the uniaxial loading and the boundary conditions of a finite size structure. Under these conditions, the contraction in the y-axis and the elongation in the x-axis of the structure are $\delta_y$ and $\delta_x$, respectively.
Circular hierarchical honeycomb consists of the unit cell specified in Figure 2(b). The far-field stress, $\sigma_{yy} = -(2/3) \frac{F}{a}$, occurs in the unit cell due to a load of $F$ imposed in y-direction. Therefore; the average strain is $\varepsilon_{yy} = -4\delta_y/a\sqrt{3}$. The mechanical behavior of the entire structure can be studied by using subassembly which is formed with $L_1$ and $L_2$ lines passing through the midpoints of the edges as seen in Figure 2(b) [1].

The theoretical analysis presented in this section follows the approach presented in Ajdari et al. [1]. To make the analysis clear, it is important to examine the hierarchical honeycomb given in Figure 2(b). The midpoints of several edges of the given structure are named $P_1$ to $P_5$. In the case of macroscopic normal stress, $\sigma_{11}$ and shear stress, $\sigma_{12}$ is the average force per unit length transmitted along a vertical line specified by $L_2$. The net horizontal and vertical forces on this line are zero because only $\sigma_{22}$ is not zero. Moreover, they do not transmit the bending moments because it would remove the horizontal symmetry of the structure. Hence, under a non-zero macroscopic stress $\sigma_{22}$, the edges cut by $L_2$, are unloaded.

Considering the edges of the structure cut by $L_1$, each bar is subjected to the force $F = -(3\sigma_{yy}a)/2$. Due to the symmetry in the structure, bending cannot occur in the struts cut by line $L_1$, otherwise symmetry would be violated. This means that no bending moment is transmitted by the bars. Net horizontal force is zero across $L_1$ because $\sigma_{12} = 0$. In the lower part of the structure cut by the $L_1$ line, a rightward force at $P_1$ is compensated by another leftward force at $P_3$. Therefore, it can be stated that the forces at points $P_1$, $P_3$, and $P_4$ are vertical and $F$ in magnitude.

Figure 2(c) presents the free body diagram of the subassembly of the circular hierarchical structure exposed under the loading case. The lower part of the subassembly causes the force and moment reactions ($N_{1p}$, $M_{1p}$, $N_{2p}$, and $M_{2p}$) at points 1 and 2. The vertical forces at points 1, 2, and 4 are used to determine the effective elastic modulus, whereas dummy horizontal forces (no actual load) are placed at 2, 3 and 4 in order to determine the lateral displacement using Castigliano’s Method, to obtain the Poisson’s ratio. By using vertical force equilibrium and moment balance equations for the subassembly, $N_{2p}$ and $M_{2p}$ can be written as a function of $N_{1p}$, $M_{1p}$, and $F$. To calculate the bending energy stored in the structure, the strain energy of the curved and linear beam elements that make up the subassembly should be calculated separately and summed as shown in the following expression: $U(F,N_{1p},M_{1p}) = \int M^2/(2EI) dX + \int \sigma_{yy} \varepsilon_{yy} \delta y dA$. 

Figure 2. (a) Uniaxial-loading of the structure with a far-field force $F^{\infty}$. (b) Symmetry axis and points of interest in a representative honeycomb. (c) Free-body diagram of the structure used for analytical calculations.
\[ \sum \left( \frac{M^2 R}{2E \delta} \right) \, d\theta, \]  
where \( M \) is the bending moment at location \( x \) for the linear beam, and \( \theta \) for the curved beam; \( R \) is the radius of the curved beam, \( E \) is the elastic modulus of the structure material, \( I \) is the cross-section area moment of inertia (The moment of inertia which is calculated using the wall thickness \( t \) for the unit depth structure with the rectangular section; i.e., \( I = t^3/12 \)). \( U \) is a quadratic function of the quantities \( F, N_{1P}, \) and \( M_{1P} \).

Because of the symmetry of the subassembly, it is assumed that there is no vertical displacement and rotation at point 1. So that, the equations \( \partial U / \partial N_{1P} = 0 \) and \( \partial U / \partial M_{1P} = 0 \) are obtained by application of Castigliano’s method. These two equivalence relations allow us to derive the functions of \( N_{1P} \) and \( M_{1P} \) expressions in terms of \( F: N_{1P} = F(0.5288 + 0.1378/\gamma_1), \) \( M_{1P} = aF(-0.0288 + 0.2532\gamma_1). \) Using the equation of \( \delta_y = dU/dF \) at point 4 allows finding the vertical displacement of the subassembly. We can express the displacement as a function of the \( F \) variable by substituting above \( N_1 \) and \( M_2 \) relations, and finally \( \delta_y = \sqrt{3}Fa^3/(72Ea_n f(\gamma_1, \alpha_t)) \) is obtained, where \( f(\gamma_1, \alpha_t) = \sqrt{3}/[\alpha_t^3(0.75 - 4.5\gamma_1 + 9\gamma_1^2 - 6\gamma_1^3) + (0.993\gamma_1^3 - 6.095\gamma_1^2 + 10.993\gamma_1)]. \) The ratio of the average stress \((-2F/3a)\) and the average strain \((-4\delta_y/a\sqrt{3})\) allows us to compute the effective elastic modulus of the model:

\[ E/E_s = \left( \frac{t_n}{a} \right)^3 f(\gamma_1, \alpha_t) \]  
(2)

For homogeneous thickness \((t_n = t_h)\), \( \alpha_t \) is equal to 1. To obtain the maximum value of the normalized effective elastic modulus, the term \( t_n/a \) in Eq.1 should be converted to its equivalent in terms of \( \alpha_t \). Then, Taylor's theorem for a function of two variables is used, as given in Eq. 3, to find the local maximum of the resulting expression of the normalized effective elastic modulus.

\[ \frac{\partial^2 (E/E_s)}{\partial \gamma_1^2} \frac{\partial^2 (E/E_s)}{\partial \alpha_t^2} \frac{\partial}{\partial \alpha_t} \left( \frac{\partial (E/E_s)}{\partial \gamma_1} \right) > 0 \]  
(3)

The equation gives the stationary point \( \gamma_1 = 0.279 \) and \( \alpha_t = 0.701 \) as shown in Figure 3, and then by substituting these values, \( E/E_s = 3.036\sigma^3 \) is obtained. The stiffness of the first-order circular hierarchical honeycomb is slightly larger than twice of the stiffness of the regular honeycomb model [23], and is also larger than the effective elastic modulus of the structure with first-order hexagonal hierarchy [1].

The variation of \( f(\gamma_1, \alpha_t) \) with respect to \( \gamma_1 \) and \( \alpha_t \).

Finite element (FE) method was used to confirm analytical calculation of the effective elastic properties of the first-order circular hierarchical honeycomb. The FE analysis, was carried out using Abaqus, where the structure is modeled with the BEAM22 quadratic beam elements which take axial and shear deformations into account in addition to the bending deformation. However, the contribution of axial and shear deformations to the response of the structure is negligible, when the beams are slender i.e. the ratio, \( a/t \) is large. The relative density of the structure is fixed by adjusting the thickness of the rectangular cross-section with the unit depth of the beams. It was also assumed that the structure is made of ABS polymer (Acrylonitrile Butadiene Styrene) with Young’s modulus \( E_s = 2300 \) MPa.

Periodic boundary conditions were applied during the analysis in order to reduce the model size. The Repetitive Unit Cells in Figure 4(a) with orthogonal lattice vectors \((a_1, a_2)\) were used in FE analysis because they allow an easier application of periodic boundary conditions.
Figure 4. (a) Repetitive Unit Cells and lattice vectors of the structure. (b) Points on the boundary where periodic boundary conditions are applied.

All nodes lying along the edges shown with the dashed lines in Figure 4(b) were connected to each other according to the constraints in Table 1. In this way, the model behaves as if it was infinitely long and wide cellular structure but free to strain laterally [27]. As a result, we obtained an infinite cellular structure using this RUC, and eliminated the size effect (i.e., a size-independent structure).

Table 1. Constraint equations used to apply periodic boundary conditions.

<table>
<thead>
<tr>
<th>Constraints:</th>
<th>Boundary Conditions:</th>
</tr>
</thead>
<tbody>
<tr>
<td>(-L_1^x + L_2^x + ref_1^x = 0)</td>
<td>(ref_1^x = 0)</td>
</tr>
<tr>
<td>(-R_1^x + R_2^x + ref_1^x = 0)</td>
<td>(ref_2^x = \text{free})</td>
</tr>
<tr>
<td>(-L_3^x + R_3^x + ref_2^x = 0)</td>
<td>(ref_1^y = \text{constant})</td>
</tr>
<tr>
<td>(-L_1^y + L_2^y + ref_1^y = 0)</td>
<td>(ref_2^y = 0)</td>
</tr>
<tr>
<td>(-R_1^y + R_2^y + ref_1^y = 0)</td>
<td>(L_1^x = 0; L_1^y = 0)</td>
</tr>
<tr>
<td>(-L_3^y + R_3^y + ref_2^y = 0)</td>
<td></td>
</tr>
</tbody>
</table>

In order to facilitate the comparison with known structures, the effective elastic modulus values of the circular hierarchical honeycomb are normalized with that of regular honeycomb, following Ajdari et al. [1]. The normalized effective elastic modulus of the first-order circular hierarchical honeycombs for all values of \(\gamma_1\) between 0 and 0.5 is given in Figure 5.

Figure 5. Normalized effective elastic modulus of the honeycomb structure with circular hierarchy and homogeneous thickness as function of the hierarchy scaling factor for different relative densities.

Neglecting the shear and axial deformations in the analytical model leads to discrepancies between theoretical and numerical results for larger relative-densities, i.e., for thicker cell walls where shear and axial deformations become
important. However, a good agreement between numerical and analytical results is observed for low relative densities, see Figure 5. (a good approximation is only seen for the lowest density [27]).

Figure 5 shows that the circular hierarchical structure with homogeneous thickness and a scaling factor of $\gamma_1 = 0.31$ has stiffness nearly 1.8 times of the regular hexagonal honeycomb. The contour map in Figure 6 shows that when both the scaling factor and the thickness ratio are left as free parameters, a maximum stiffness of approximately 2.1 times that of regular honeycomb can be obtained when $\gamma_1 = 0.28$, and $\alpha_t = 0.7$.

3. EFFECTIVE POISSON’S RATIO

For a complete identification of effective mechanical properties of the structure modeled as linear elastic, an equation in which the Poisson’s ratio can be represented in terms of $\gamma_1$ is needed. Therefore, Castigliano’s second theorem is used to compute the lateral deformation of the structure subjected to uniaxial loading. The horizontal forces in Figure 2(c) are used as the dummy force in the energy expression used in Castigliano’s theorem. Following a procedure similar to the calculation of the elastic modulus, axial and shear deformation of the beams are also ignored in this analysis.

$$N_2P$$ and $$M_2P$$ can be written as functions of $$N_{1P}, M_{1P}, P,$$ and $$F$$ by using equations of equilibrium of subassembly. The summation of bending energy in all beams express the total energy stored in the subassembly under the loading case. So, $U(P,F,N_{1P},M_{1P}) = \int M^2/(2E_2I) dX + \sum \int (M^2R)/(2E_2I) d\theta$. According to the assumption of no displacement and no rotation at nodes 1 and 2, $\partial U/\partial N_{1P} = 0$, and $\partial U/\partial M_{1P} = 0$ can be written. These two conditions enable us to write $N_{1P}$ and $M_{1P}$ in terms of $P$ and $F$. This allows us to express the bending energy of the subassembly as a function of $P$ and $F$, $U = U(P,F)$. Setting the dummy force $P$ to zero, the lateral displacement of the subassembly is computed from $\delta^L_x = (\partial U/\partial P)_{P=0}$ and the vertical displacement from $\delta^V_y = (\partial U/\partial F)_{P=0}$. The initial dimensions of the subassembly are $3a/4$ and $a\sqrt{3}/4$ in the x- and y-directions, respectively. Therefore, the effective Poisson’s ratio is obtained as $\nu = -\varepsilon_x/\varepsilon_y = -\delta^L_x/\sqrt{3}\delta^V_y$, which gives:

$$\nu = \frac{\gamma_1(-0.1655 + 1.0159\gamma_1 - 1.6407\gamma_1^2) + \alpha_t^3(-0.125 + 0.75\gamma_1 - 1.5\gamma_1^2 + \gamma_1^3)}{\alpha_t^3(-0.5 + \gamma_1)^3 + \gamma_1(-0.1655 + 1.0159\gamma_1 - 1.8319\gamma_1^2)}$$

which is shown in Figure 7.
For homogeneous thickness \((t_n = t_h)\), \(\alpha_t\) is equal to 1. To obtain the maximum value of the effective Poisson’s ratio, Taylor’s theorem for a function of two variables is used as given in Eq. 3. The value of the effective Poisson’s ratio is \(\nu = 1\) at \(\gamma_1 = 0\), and the minimum value 0.2996 is reached at \(\gamma_1 = 0.326\) and \(\alpha_t = 6.57 \times 10^{-6}\). However, this thickness ratio is too small for practical applications. Therefore, we consider physically reasonable value, \(\alpha_t = 0.1\) which provides a Poisson’s ratio, \(\nu = 0.3\) that is very close to the minimum. Theoretical results were verified with finite element analysis of the periodic structure subjected to uniaxial loading. The effective Poisson’s ratio of the first-order circular hierarchical honeycombs with homogeneous thickness for values of \(\gamma_1\) between 0 and 0.5 shown in Figure 8.

From Figures 5 and 8, it is seen that the highest normalized effective elastic modulus, 1.753 and the lowest effective Poisson’s ratio, 0.43 for the circular hierarchical honeycombs in the homogeneous thickness case \((\alpha_t = 1)\) are achieved when \(\gamma_1\) is 0.3 and 0.38, respectively. Thus, it is clear that mechanical performance is improved by using different thickness method when compared to the results of the heterogeneous case.

4. CONCLUSION

Figures 9 and 10 show the normalized effective elastic modulus and effective Poisson’s ratio for different relative densities and hierarchies, respectively. The results show that a relatively wide range of normalized elastic modulus and effective Poisson’s ratio can be obtained by tailoring the structural dimensions of the honeycombs or changing the hierarchy geometry. Furthermore, it can be seen that the first-order hexagonal hierarchical honeycombs have an effective stiffness up to 2 times of the regular honeycombs [1]. It is possible to achieve an effective elastic modulus up to 2.1 times
that of the regular honeycombs using circles instead of hexagons as hierarchy geometry and by setting different thicknesses for the hierarchy and regular sections of the honeycomb.

Figure 9. Normalized effective elastic modulus of honeycombs with circular hierarchy for different thickness ratios in comparison with the 1st order hexagonal hierarchy proposed by Ajdari et al. [1].

Moreover, following the same approach, we can obtain an effective Poisson’s ratio less than that of the hexagonal hierarchical honeycomb studied by Ajdari et al. [1]. Hierarchical structures with circular geometries can have effective Poisson’s ratio up to 0.3 times that of the regular honeycombs. Further optimization should be possible by also varying hierarchy geometries. Also, ideally hierarchy elements should be small different thickness conditions to get maximum mechanical performance. This could be achievable by appropriate adjustment of the cell thickness in different parts of the structure.

Figure 10. Effective Poisson’s ratio of honeycombs with circular hierarchy for different thickness ratios in comparison with the 1st order hexagonal hierarchy proposed by Ajdari et al. [1].
ACKNOWLEDGEMENTS

We thank professors Mehmet Ali Güler and Yasin Sarıkavak for fruitful discussions.

Mustafa Yıldız – 20.08.1991, I was born in Osmangazi, Bursa. I studied at Çelebi Mehmet High School in Bursa. I graduated from Balıkesir University Mechanical Engineering Department with the first degree. I am doing a PhD in Mechanical Engineering Department at Yıldırım Beyazıt University. After one year of experience in Balıkesir Elektromekanik Sanayi Tesis (BEST AŞ.), I have been working as a research assistant in the Department of Mechanical Engineering at Yıldırım Beyazıt University.

Fatih Göncü – is an assistant professor at the Department of Mechanical Engineering, Ankara Yıldırım Beyazıt University.

REFERENCES


SLIPPAGE ESTIMATION OF A TWO WHEELED MOBILE ROBOT USING FEEDFORWARD DEEP NEURAL NETWORK

İsmail Özçil, E. İlhan Konukseven, A. Buğra Koku

Middle East Technical University, Department of Mechanical Engineering, Ankara, Turkey
{kiozcil, konuk, kbusra}@metu.edu.tr

ABSTRACT
Mobile robot navigation is an important task for the operations of the mobile robots. Due to the wheel slippages, performance of the dead reckoning in estimating speed of the robot and the position of the robot is not sufficient. In this study to overcome the errors in navigation estimates, usage of the deep neural networks is proposed. Neural networks are used to understand the behavior of the linear and nonlinear systems. Parameter estimation of non-linear wheel-ground interaction model is difficult. Therefore, the usage of the neural networks is preferable since they do not require system models and parameters. In this work, a feed-forward deep neural network is proposed to estimate the speed and yaw angle of the 2 wheeled differentially driven mobile robot. By recording data from the training experiments of the navigation of the mobile robot, network is trained. After that, performance of the network is evaluated by plotting and tabulating outputs of the network, sensor data calculation and ground truth.

Keywords: Mobile robot, deep network, ground vehicle, dead reckoning

1. INTRODUCTION
For the indoor and outdoor wheeled mobile robots, positioning of the robot is important since all tasks depend on positioning of the robot. For the indoor applications, vision information is used to calculate position of the robot, but there may not be available camera getting image of whole working ground of the robot, or the camera on the robot may not get known objects to determine position every time. For the outdoor applications, GPS is widely used, however it may not be available according to Ward and Iagnemma due to surrounding buildings, trees or weather conditions [1]. Frequency of the GPS data may not be sufficient or ground truth data may be interrupted. During those times, dead reckoning is used. Dead reckoning is estimating position of the mobile robot by using on-board sensors of the mobile robot like IMU, encoder, etc. However, using those sensors over a time, errors start to propagate and position and velocity estimates start to have large error due to the sensor drifts and wheel slippages. Also, frequency of the GPS data may not be sufficient for slip detection and anti-slippage applications. Slippage is directly affects the trajectory followed by the robot, hence it should be compensated to follow the desired trajectory. Moreover, sensing and compensating wheel slip also improves traction of the robot and it may prevent unnecessary power usage where wheels of the robot are slipping.

1.1 Literature Survey
Chenavier and Crowley [2] used mobile robot with a camera mounted on the robot in an environment which contains some known objects with known positions to determine the position of the mobile robot. Gustafsson [3] used non-driven wheels to determine actual velocity of the vehicle. Seyr and Jakubek [4] used odometry and IMU to improve position estimates. They compare two measurements and in the according to the slippage of the robot, they give ore eight to one of the two measurements. Chonnaparamutt, Winai and Kawasaki [5] used fuzzy estimator to estimate position of a pruning robot. They also used a fuzzy controller to control speed of the robot. Ward and Iagnemma [1] used GPS data to improve velocity estimates of the robot, they update their velocity estimates at nearly 1 Hz, when they got GPS data. Hwang et al.[6] used images of a object taken by the on-board camera of the mobile robot to correct velocity and position estimates done by the odometry and IMU. Ojeda et al.[7] used current drawn by the DC motors connected to wheels to determine the slippage and amount of the slippage. Bayar et al.[8] used camera positioned above the working area of a 2 wheeled mobile robot to determine the coefficients of the traction, lateral and roll force equations derived in previous works. Later, they used those coefficients to determine position of the robot. Moreover, they showed that in different surfaces, current profile of the DC motors connected to wheels differs. Zabaleta et al.[9] used optical mouse sensors to improve position estimations of the rehabilitation robot they used in their work. However, this robot requires to work on a special textured surface. Bonarini and Matteucci [10] presented equations for a robot using 2 optical mice to improve position estimation, but they only presented modelling, there is not any experiments. Sekimori and Miyazaki [11] used multiple optic mice to
get position of the mobile robot. Optical mice can be used to improve position estimations, but optical mouse sensors should be positioned in a certain narrow height from the surface, installation and operation of those sensors in unstructured surfaces is not ideal since road surface may not be smooth and distance from sensor to ground may constantly change. Fujimoto et al. [12] used yaw moment observer to estimate cornering stiffness of an electric vehicle. But this method requires model of the vehicle, and model parameters may not be available for a small robot for indoor applications. Matuso [13] used neural networks to estimate tire-road friction force. This article shows that neural networks are useful with the non-linear systems with high number of unknowns. Hence for robotic applications with small robots where parameters of the robot are unknown, neural networks can be used. Melzi and Sabbioni[14] used a simple neural network with one hidden layer and 10 sigmoid neurons to determine slip angle of a vehicle. They used 3 neural networks in that structure, but they changed number of inputs for second and third network. In the end, they compared performances of those networks. Cirovic et al.[15] used neural network to adjust brake actuation pressure according to the adhesion coefficient between the tire and road. Tahami et al.[16] used neural network to generate yaw angle reference to the fuzzy controller that control the torque and slip of the wheels in case of an emergency like obstacle avoidance for an all-wheel drive electric vehicle.

2. NETWORK USED IN THE STUDY

In the feedforward neural networks, information flows in one direction and it makes no loop movement. They are the simplest model of neural networks. They are widely used in areas like image classification. In the python and TensorFlow environment, they are easy to implement and train. In the Figure 1, multi-layer deep feedforward neural network used in the study is represented. Information is taken by the input nodes, after that point calculations are made and information is passed to the all the layers in front of the current layer. There is no back movement and feedback loop. Flow of the information is represented by the arrows, and as it can be seen from the figure, a neuron takes information from the neurons one layer before it, and if that neuron is activated, it passes information to the neurons one layer after it. They are trained using to the training data, and through the training, optimal values are found according to loss function and optimization method. For the activation function, rectified linear unit (ReLU) is used. According to Glorot et al.[0], compared to sigmoid function, rectified linear unit function has better gradient propagation. Moreover, since it is a very simple function, it does not bring much computational load to algorithm. For the optimization, Adam optimization is used. There are 9 inputs to the network, and there are two outputs, which are speed and yaw angle of the robot. Inputs and outputs are tabulated below.

Table 1. Inputs and outputs of the network

<table>
<thead>
<tr>
<th>Label</th>
<th>Name</th>
<th>Sensor Used</th>
</tr>
</thead>
<tbody>
<tr>
<td>Input 1</td>
<td>Time</td>
<td></td>
</tr>
<tr>
<td>Input 2</td>
<td>Time Step</td>
<td></td>
</tr>
<tr>
<td>Input 3</td>
<td>Longitudinal Speed of the center of Robot</td>
<td>Encoder 1&amp;2</td>
</tr>
<tr>
<td>Input 4</td>
<td>Yaw Angle of the Robot</td>
<td>Encoder 1&amp;2</td>
</tr>
<tr>
<td>Input 5</td>
<td>Yaw Angle of the Robot</td>
<td>IMU</td>
</tr>
<tr>
<td>Input 6</td>
<td>+x Acceleration of the Center of the Robot</td>
<td>IMU</td>
</tr>
<tr>
<td>Input 7</td>
<td>+y Acceleration of the Center of the Robot</td>
<td>IMU</td>
</tr>
<tr>
<td>Input 8</td>
<td>Current Drawn by Motor 1</td>
<td>Current Sensor 1</td>
</tr>
<tr>
<td>Input 9</td>
<td>Current Drawn by Motor 2</td>
<td>Current Sensor 2</td>
</tr>
<tr>
<td>Output 1</td>
<td>Speed of the Center of the Robot</td>
<td>Camera</td>
</tr>
<tr>
<td>Output 2</td>
<td>Yaw Angle of the Robot</td>
<td>Camera</td>
</tr>
</tbody>
</table>

First and second inputs are time and time step. Third and fourth inputs are longitudinal speed of the center of the robot and yaw angle of the robot measured by the encoders connected to the wheels of the robot. Next inputs are yaw angle, +x acceleration and +y acceleration of the robot measured by the IMU placed on the robot. Last inputs are the current drawn by the DC motors connected to the wheels of the robot.
As it can be seen from the Figure 1, network used in this study composed of 1 input layer, 5 hidden layers and 1 output layer. Each hidden layer has 18 cells. 120 experiment used to train this network with 1000 epochs.

3. EXPERIMENTAL SET-UP

2 wheeled differentially driven robot is used in experiments. There are 2 wheels on the mobile robot which are connected to DC motors with 896 ppr encoders. Odometry data are obtained using those encoders. There also one caster wheel to stabilize robot. To get inertial measurement, an inertial measurement unit (Pololu MiniIMU v3) is placed on the mobile robot. This IMU contains a 3 axis accelerometer, a 3 axis gyro and a 3 axis magnetometer. Encoder, IMU and DC motor current measurement data are measured and sent simultaneously. In this thesis, those measurements alongside with the current measurements of the DC motors are combined to make better velocity estimates of the mobile robot. However, to evaluate performance of the estimates and measurements, a camera is placed on the top of the experiment area. Experiment area is 1.85 m x 1.85 m square area. The camera used to get position has 1280x720 pixels resolution and gives images with 30 frames per second. Two ACS 712 analog current sensors are connected to DC motors driving the wheels. Analog current sensor output is measured by a 16 bit ADC. DC motors are controlled by Faulhaber motion controllers and speed of the DC motors are controlled by a closed-loop control system. Sensor data are collected by Arduino Uno and then sent to the Raspberry pi2 placed on the mobile robot. Also, control of the robot is done by the same Raspberry pi2. A sigma profile aluminum beam is connected to chassis of the robot. Heaviest component of the system is battery, thus place of the battery is used to change center of gravity of the robot. By moving center of gravity of the robot to forward or backward, handling behavior of the robot is adjusted. If the center of gravity is close to front of the robot, then robot tends to understeer. If the center of the gravity is close to rear of the robot, then robot tends to oversteer. Battery is placed on the sigma profile beam, and since that beam stretch out from beyond front of the robot to beyond rear of the robot, handling characteristic can be adjusted by moving battery on the beam. In this work, behavior of the robot is expected to be understeer and robot is expected to start sliding in relatively low speeds since maximum speed of the robot is below 2 m/s. When the desired characteristic is achieved, battery is fixed at that location on the beam.
In the Figure 2, b=75 mm and a=125 mm. Experiments are done on the 1.85 m x 1.85 m experiment ground and during the experiment, reference velocity input is sent to the motion controllers of the DC motors connected to wheels of the robot. During the experiment, IMU and encoder data are collected by the raspberry pi2, and ground truth data is collected using the camera placed over the experiment set-up. In the experiments, different reference angular velocity inputs are sent to the motion controllers, hence robot follows near-circular path during the experiment. By this way, combined effects of the longitudinal and lateral slippages can be observed. For different set of experiments, different reference angular velocity inputs are sent, hence velocity of the robot is different in different set of experiments.

4. RESULTS

2 of the experiments are not used in the training, they are used to validate the results. In this section, results of those experiments are plotted and tabulated. First, speed estimates of the robot at Experiment 1 and Experiment 2 obtained from camera, encoders and network output are plotted.

![Speed estimates comparisons of Experiment 1](image1)

As it can be seen from the figures, network compensates some of the errors of encoder speed estimates. Secondly, yaw angle estimates are compared.

![Speed estimates comparisons of Experiment 2](image2)
Network gives significantly closer results to camera compared to IMU and encoder yaw angle estimates. Mean of error percentages are tabulated for the speed estimates and the yaw angle estimates for those experiments.

<table>
<thead>
<tr>
<th>Table 2. Mean of speed estimate error percentages of encoder and network output</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experiment  #</td>
</tr>
<tr>
<td>Experiment 1</td>
</tr>
<tr>
<td>Experiment 2</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Table 3. Mean of yaw angle estimate error percentages of encoder and network output</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experiment  #</td>
</tr>
<tr>
<td>Experiment 1</td>
</tr>
<tr>
<td>Experiment 2</td>
</tr>
</tbody>
</table>

There are previous work done about slippage estimation in literature. Chonnaparamutt, Winai and Kawasaki [5] used fuzzy estimators to estimate speed of a pruning robot. In the experiments, robot is moved around 0.1 m/s, and in average encoder measurements have 11.09% error and fuzzy estimator has 2.74% speed estimation error. In this work, trained network error is 8.81% and 5.74%. It is higher than the error achieved by Chonnaparamutt, Winai and Kawasaki, however robots and the experiment conditions are not the same. In this work, average speed of the robot is around 4.5 m/s, compared to 0.1 m/s average speed.

Sekimori and Miyazaki [11] used four optical mouse sensors to improve covered distance and yaw angle estimations of a differentially driven mobile robot. They moved robot in a 2.5m path and this path consists of straights and 90 degree
Average speed of the robot is 500 mm/s in the experiments. Encoders measured yaw angle by 19.26 degree average error (5.35%) and optical mouse sensors measure yaw angle by 8.58 degree average error (2.38%). Sekimori and Miyazaki reduced yaw angle estimate from 5.35% to 2.38%. In this work, yaw angle estimate error I reduced from 22.72% to 7.26%. In the second experiment, error reduced from 21.23% to 4.35%. Moreover, in those experiments, yaw angle estimate of the encoders are much higher, 40.13% and 35.34%. Despite different conditions, network gives comparable results to literature.

5. CONCLUSIONS AND RECOMMENDATIONS

In this work, errors on the speed and yaw angle data due to the slippages on the wheels of a 2 wheeled differentially driven mobile robot are reduced using a feedforward deep neural network. Proposed network's ability to capture and compensate errors due to the wheel slips in the speed estimates and the yaw angle estimates is evaluated according to the error comparisons. Reduced errors in speed and yaw angle estimates mean that is is better than dead reckoning, hence deep networks can be used to get position estimates when the ground truth data is interrupted. Network is able to reduce errors in each of the 2 experiments and it can be used to improve navigation of two wheeled mobile robots. Since most of the mobile robots have encoders and also an IMU, after a simple training of the network, navigation data can be obtained from the trained network with lower error compared to odometry and IMU. Since it is not requires system model and system parameters, it is easy to apply in different systems. Specifically for the systems where frequency of the ground data is low or interrupted, this method can be used in the intervals between receiving ground truth data.

For the future work, for different tires, ground surface conditions and for different inclinations, method can be tested since same surface, same robot and same set of wheels are used in this work. Using two cascade networks, first surface type can be determined and later according to the type of the surface, second network can be adjusted to provide required navigation data for the robot.

REFERENCES


A REVIEW ON SANDWICH PANEL STRUCTURES IN AUTOMOTIVE INDUSTRY

Seyyid Ali Özçelik*, Recep Emre Başar,
Mostafa Ranjbar
Department of Mechanical Engineering, Ankara Yıldırım Beyazıt University, Ankara, Turkey
*Corresponding author: 165103407@ybu.edu.tr

ABSTRACT

Sandwich panel structures have a core material between two layers. In general usage the core is a foam structure and the layers are metal, as aluminum. Therefore, novel applications such auxetic core or fiber reinforced are also available according to needs of the structure.

As the weight advantages of sandwich panel structures, it is a choice for lightening in many industries especially for satellites, aircraft, land transportation vehicles, ships, wind energy systems and buildings.

In this paper it is aimed to discuss recent researches about sandwich panels in automotive industry, as a lighter solution which affects the fuel consumption, vibro-acoustic point of view and manufacturing.

Keywords: Sandwich panel, structure, automotive,

INTRODUCTION

Deshmukh et al. investigated the sandwich steels for NVH (Noise Vibration Harshness) improvements. NVH is one of the performance standard point of customers view. The study includes steel applications in vehicle BIW (Body in White), the steel sandwich panels was compared to regular steel and the results were obtained. After extensive testing under a variety of conditions, Silent steel was chosen as a major contributor to reducing structure-borne noise in the vehicle. As a result, the manufacturing cost, weight, and time reduction is observed using sandwich steel panels. [1]

The loss in body weight in automotive due to sandwich panel usage in vehicle BIW was investigated by Deniz Hara and Gokhan O. Ozgen. First, vehicle floor, firewall, luggage and rear wheel were replaced with sandwich panels. The results showed that using sandwich panels with viscoelastic material core could reduce the weight roughly 60-70% and damping performance did not change. [2]

In the study of Wang et al., carbon fiber-reinforced composite sandwich structures improved at high temperatures with aluminum honeycomb structures as the core material were investigated. The base of the study was leaned on three-point bending and panel peeling tests by tests core thickness and density on laminate material properties. The effects of the core material thickness and density on the material properties of composite sandwich honeycomb structures were studied. [3]

This paper considered design variables as structural stiffness, strength, buckling resistance and the objective of the study was reduction of weight as well as cost. Aly et al. studied on alternative panels by developing design procedure for material selection and sizing. [4]

The study of S. Milton and S.M. Grove focused on the production of the sandwich panel structures for a lightweight vehicle chassis. One of the biggest downsides of the sandwich panel structures is difficulty of series production. The study suggests full integration of CAD and CAM [5].

The structural optimization and the material selection are combined in the study of Ermolaeva et al.. The material selection depends on parameters load endurance and geometric stability. The structural optimization was based on material selection. The purpose of the study was constituting a base for sustainable product design of lightweight vehicles. [6]

Imbalzano et al (2015) investigated the sandwich panels with auxetic and honeycomb cores confined between metallic facets are proposed for localized blast lading applications. The auxetic panels under dynamic loading shows progressively enhanced impact resistance. The performance under localized impact was numerically calculated. With the compared results, computation time significantly reduced. Hybrid auxetic composite panels (HACPs) could resist the impact and dissipate the imparted energy compared with the honeycomb panels [7].

The new roof structure design against roll-over crashes was analyzed in Soroosh Borazjani and Giovanni Belingardi’s study. As sandwich panel the composite face were chosen because of the ability to absorption of energy and Expanded
Polypropylene (EPP) is used as a core. As a result, the roof panel density significantly reduced, and the energy absorption increased about 15%. [8]

Ranjbar et al. used homogenized finite element models to determine the mechanical properties of the auxetic structures, the natural frequencies and radiated sound power level of sandwich panels made by the auxetic cores. First order and random optimization methods were used for the minimization of the radiated sound power level of the structures [9].

CONCLUSION

The recent developments in automotive industry showed how important the notions; emission, energy-saving and eco-friendly vehicle are. Therefore, the reducing weight of the car provides reduction of the emission as well. The main purpose of the sandwich panel structures is reducing the weight and hold the strength and stiffness of the body approximately same. However, the selection of the replacement place is important also the structure shape and material selection are crucial variables of the studies. In addition to these variables, the price should be affordable for the customers. In case of the development in mass production of the sandwich panels, these structures will be more common in automotive industry in future.

REFERENCES


[7] Gabriele Imbalzano, Steven Linforth, Tuan Duc Ngo, Peter Vee Sin Lee, Phuong Tran, (2017), Blast resistance of auxetic and honeycomb sandwich panels: Comparisons and parametric designs


A REVIEW ON DESIGN OF AUXETIC SANDWICH PANEL STRUCTURE

Seyyid Ali Özçelik*, Mostafa Ranjbar

Department of Mechanical Engineering, Ankara Yıldırım Beyazıt University, Ankara, Turkey
*Corresponding author: 165103407@ybu.edu.tr

ABSTRACT

Auxetic structures are a novel meta materials which have negative Poisson’s ratio, and result of it the structure has lateral extension for a tensile load applied expand under tension and thinner under compression. When the auxetic structure is used in a sandwich panel, both of their advantageous combined such low density, high acoustic insulation, high-energy absorption and durability at dynamic loads and fatigue.

In this study the sandwich panels, which has an auxetic core, is discussed on vibroacoustic point of view with current researches.

Keywords: Sandwich panel, auxetic, vibroacoustic, design

INTRODUCTION

Here, the last researches on the vibroacoustic design of auxetic sandwich panels are reviewed and reported. Mackerle (2002), done a bibliographical review of the FE analysis of sandwich structures, both of theoretical and practical points of view with various fields of engineering and the other topics [1]. Lew et al (2008), introduced a meta-model to solve the numerical simulations of large-scale engineering design problems of auxetic honeycombs, both for hexagonal and hexachiral configurations with lower CPU times usage [2]. Peters et al (2009), measured loss factors on different sandwich panels to determine the effects of different skin and core materials on the acoustical properties of sandwich structures. Analysis of the results has been revealed that inserting a viscoelastic material in the mid-plane of the core resulted in the highest loss factor. In addition, panels were constructed with carbon fiber skins exhibited larger loss factors than glass fiber skins. Panels were designed to achieve a subsonic wave speed did not show a significant increase in loss factor above the coincidence frequency. The data indicated that judicious choice of sandwich materials could lead to reductions in noise transmission [3].

Qing-Tian and Zhi-Chun (2010), investigated wave propagation in the sandwich panel which has an auxetic core and analyzed the characteristic of wave propagation in the panel, which was considered a three-layer. The characteristics of wave propagation of this panels, and effects of panel thickness, geometric properties of the unit cell on dispersive curves were discussed. Variations of Poisson’s ratio and core density with inclined angle were presented [4].

Tee et al (2010) investigated the flexural wave propagation properties in auxetic tetrachiral honeycombs. To calculate the dispersion characteristics and phase constant surfaces varying the geometric parameters of the unit cell, a Bloch wave approach was applied [5].

A group of researchers [6-22] has developed knowledge of multidisciplinary engineering design optimization significantly. They could apply the design optimization algorithms in engineering applications. They showed that how various optimization methods could provide better results for a specific application like machining of light alloys and mechanical structures.

Ranjbar et al (2011), studied a comparison on optimization in structural acoustic. They have used a combination of a commercial finite element software and user-programmed software. The optimization is focused on minimization of root means square level of structure-borne sound. To achieve it the process continued automatically until the given maximum number of function evaluations [11].

Ranjbar and Marburg (2013) made an analytical approximation of objective function with an artificial neural network (ANN) adjustment for a specific structural acoustic application. This approximation has been used as the replacement of the main real objective function during the optimization process. The aim of their optimization process was to detect a minimum radiated sound power level for a considered geometry with best modifications [16].

Ranjbar and Marburg (2013) has used a combined random search method and geometry modification concept to minimize the root mean square level of structure born sound for a model. The structure has been considered as a steel rectangular plate. According to the results, the method could reduce the SPL of the structure within a limited time [17].

Yang et al (2013) designed a sandwich structure with a 3D re-entrant auxetic core, which based on an analytical modelling. The produced samples were compared to other rectangular cellular sandwich structures with various experiments.
According to the results, the pre-designed auxetic cores has improved mechanical properties such as bending compliance and energy absorption [23].

Imbalzano et al (2015) studied on sandwich panels with auxetic core confined between metallic facets are proposed for localized impact resistance applications. The performance under localized impact was numerically calculated. With the compared results, computation time significantly reduced. Auxetic core panels can absorb similar amount of energy through plastic deformation, the maximum back facet displacement was reduced up to 56% [24].

Khaled and Mubarak (2015) studied on a novel design rather than fully auxetic sandwich structures; they presented to produce auxetic laminated faceplates. Their design was based on periodic cellular networks using which were imbedded in a fibre-reinforced polymer matrix. To model of the auxetic network and the faceplate, have been done by FEM [25].

Strek et al (2015) has analyzed effective properties and dynamic response of a sandwich panel with the auxetic core which between two sheets. The cellular auxetic structure had been immersed in a filler material, which has a given Poisson’s ratio. The results have shown that was possible to create an auxetic sandwich panel made of two solid material with positive Poisson’s ratio [26].

Droz et al (2016) studied on structural optimization of composite panels using wave/finite element method (WFEM). A periodic octagonal core was designed with 70% increase in transition frequency and a significant reduction of the modal density [27].

Ranjbar et al (2016) used homogenized finite element models to determine the mechanical properties of the auxetic structures, the natural frequencies and radiated sound power level of sandwich panels made by the auxetic cores. First order and random optimization methods were used for the minimization of the radiated sound power level of the structures [20].

Imbalzano et al (2017) analyzed and compared the resistance performances against the impulsive loading both of equivalent sandwich panels an auxetic and a conventional honeycomb core. Auxetic panels shown crushing behavior, effectively adapting to the dynamic loading, therefore conventional honeycomb panels deformed plastically without localized stiffness enhancement [28].

Mazloomi et al (2017) studied on the vibroacoustic behavior of sandwich structures made of the auxetic hexagonal core. The mechanical properties of the structure were determined by homogenized finite element model. The model also used to calculate natural frequencies and radiated sound power level of the structure. For optimization, genetic algorithm method was used with an interactive link between MATLAB and ANSYS software [22].

Novak et al (2018) tested and validated with LS-DYNA modelling auxetic cellular structures in two orthogonal directions. Then the computational model has been used to simulate the crush behavior of the material under blast loading. The simulation results with comparison to the monolithic plate show that 33% displacement decreasing with 6% mass optimization [29].

Wang et al (2018) has introduced a new core material for sandwich panels which called three-dimensional double-V Auxetic (DVA) and discussed its model both parametric and numeric ways. Also, optimization for dynamic response and the design variables has been defined [30].

Sarvestani et al (2018) has been studied energy absorption and structural performance of a 3D printed sandwich panel approaching with a semi-analytical and finite element method. The semi-analytical method based on conducting structural and low velocity impact analysis. For FEA side ANSYS has been used to analyze elasto-plastic behavior of the panel under a low-velocity impact. The results of both test and numerical studies reveal that the auxetic core usage in a sandwich panel has high-energy absorption capability and minimizes the response force [31].

CONCLUSION

For following areas are still open to study on auxetic structures:
Dynamics, acoustics, creating new types, optimization, application and production.

The response of auxetic materials to dynamic impact loading conditions at different strain rates is not yet sufficiently characterized and needs to be further investigated.

New advanced additive manufacturing techniques provide means to fabricate the next generation of auxetic materials with functionally graded porosity, which can be adapted to the requirements of a particular engineering application by computational simulations and optimization techniques.

Specifically designed internal cellular structure of auxetic materials provides the best desired mechanical and vibroacoustic response to particular loading conditions.

This response can for example result in constant deceleration of impacting projectile or constant reaction force on structures, which is very useful for different applications in defense engineering and crashworthiness.

The auxetic cellular materials and structures show huge potential to become important lightweight structural materials of the future with further development of additive manufacturing Technologies or with introduction of some new, more cost effective manufacturing techniques.
REFERENCES


REVIEWS ON OPTIMIZATION OF HYDROGEN FUEL CELLS
Batukan Cem Tarhan, Kemal Bilen

Ankara Yıldırım Beyazıt University, Department of Mechanical Engineering, Etlik Western Campus, Ankara, Turkey
*Corresponding author: kemal.bilen@ybu.edu.tr

ABSTRACT
A fuel cell (FC) is an electrochemical device that converts the chemical energy into electrical energy. In an FC, hydrogen, and oxygen are combined to generate electricity, heat, and water. FC is used in several areas like transportation applications, power application, etc. This study provides general information about the optimization of FCs and hydrogen FCs. There are different types of fuel cells and they have dissimilar characteristics. The study presents a review of FCs optimization which consists of fuel, cost, and size optimization. It also includes current studies on design, types, and working principle of FCs. If these features are optimized, optimal conditions provide a higher efficiency and lower cost. There are a few types of FCs. Some of them are Polymer Electrolyte Membrane Fuel Cell (PEMFC) and Direct Methanol Fuel Cell (DMFC) for low-temperature FCs, and Solid Oxide Fuel Cell (SOFC) and Molten Carbonate Fuel Cell (MCFC) for high-temperature FCs. Temperature, pressure, and density are provided operating conditions which must be optimized.

Keywords: Fuel cell, FC, Hydrogen FC, Optimization of FCs, Efficiency and cost optimization of FCs.

1. INTRODUCTION
A fuel cell (FC) is a device that generates electricity through an electrochemical reaction without combustion. In a FC, hydrogen and oxygen are combined to generate electricity, heat, and water. In the simplest case, a FC consists of two metallic electrodes dipping into an electrolyte solution. In the operating of fuel cells (FCs), a negative electrode, the anode, produces electrons by burning a fuel. The positive electrode, the cathode, absorbs electrons in reducing an oxidizing agent. The fuel and the oxidizing agent are each supplied to its electrode [1]. There are lots of types of FCs which are generally Alkaline Fuel Cell (AFC), Proton-Exchange Membrane Fuel Cell (PEMFC), Phosphoric-acid Fuel Cell (PAFC), Solid Oxide Fuel Cell (SOFC), Molten Carbonate Fuel Cell (MCFC). Working principle of PEMFC is shown in Fig. 1. Operating temperature, mechanical design, and materials of these FCs affect the efficiency, stability, and durability [3]. Variety of them causes negatively these features. In this situation, optimization gets an essential role. If they are applied correctly, these features are optimized.

Figure 1. Working principle of a common type PEMFC [2].
2. LITERATURE REVIEW

Bunin et al. [4] examined the experimental validation of a real-time optimization strategy for the optimal operation of a SOFC stack. Corrections terms were used and the difference between predicted and measured constraint values and are updated at each steady-state iteration. In the study, the effects of the parameters used in the modifier update and of the real-time optimization frequency on the general performance of the algorithm are also investigated.

In a theoretical research, Salah et al. [5] investigated that the gas channel for water exhaust of PEM fuel cell is examined to optimize using lattice Boltzmann method. Rectangular, trapezoidal, and triangular types are applied to the system. The result of this study is rectangular type is the optimum option in a moderate pressure.

A theoretical and experimental study of Liu et al. [6] deals with the optimization of dimensions of gas flow channels and walls/ribs in a PEMFC. The investigations show that how important geometry of the flow channel is. A numerical model was developed, and the model combined with the experimental tests. The results show that minimizing the width of the flow channels and ribs could improve the FC performance.

Karthikeyan et al. [7] studied theoretically the optimization of cell temperature, anode-cathode inlet velocities and, back pressure of PEMFC. The algorithm of the optimization is based on the Taguchi method. Results of the study show that the new method gives better battery performance of the cell. Power density was increased 3% in this optimization method.

The purpose of a theoretical and experimental investigation by Dokkara et al. [8] is to adapt the photovoltaic hydrogen power for the mobile station. The system is based on transceiver station. It includes a photovoltaic field, water electrolyzer, and two FCs. Each FC is used according to the daily and seasonal load of the station. The most important criteria in the optimization of weighting between two opposite objectives are used to find the compromise solutions between maximum efficiency and minimum FC size. Results show that in the summer period, more PV modules are required for reducing air conditioning loads.

An experimental study by Sebastián et al. [9] was concerned with platinum dispersion in low surface area carbon nanofibers (CNFs). Non-traditional synthesis method was applied for optimization and electroactivity of the oxygen reduction. Different parameters of surface area were examined, and the performance of the system was increased for some values such as initial Pt particles smaller than 3 nm.

Xu et al. [10] studied experimentally fuel consumption and CO₂ emission in Direct Carbon Fuel Cell (DCFC). A variety of anode and cathode configurations were examined for catalytic oxidation of carbon. Results in an area specific resistance are of only 0.41 Ω·cm² at 650 °C at the anode. Hence, the FC’s durability and efficiency were optimized.

In an experimental search by Bartrom et al. [11], Direct Formate Fuel Cell (DFFC) was improved by optimization of catalyst loading. Direct-membrane spray painting and gas diffusion layer brush painting were the most effective solution for this problem.

Caliandro et al. [12] worked theoretically within the context of sustainable energy supply and CO₂ emission reduction in a SOFC gas turbine hybrid system. This analysis reveals the importance of process integration maximizing the heat recovery and valorizing the waste heat. The analysis shows that the potential of the system converting woody biomass into electricity is greater than 70% as energy efficiency.

Yang et al. [13] investigated theoretically and experimentally, the optimizing way of channel geometries of PEMFC with a generic algorithm is being considered in this paper. A computational tool, COMSOL is used in the MatLab environment for optimization. In that console, calculations are done for a new geometry and that geometries also experimented with different ratios. As a result of these experiments and calculations, it seemed optimized.

How can the best mix of the platinum-loaded carbon catalyst and the electrolyte in a membrane electrode assembly of a proton exchange membrane FC effect on the reducing the activation resistance, which affects the electrochemical surface, activation polarization and the highest power density of the MEA be the reason of this consideration is to find out. The platinum-loaded carbon catalysts and electrolytes with the ratio as percentage by weight of 10, 20, 40 and 60% were included in these experiments. Tashima et al. [14] obtained experimentally the results of these mixing experiments are going to be used to find out which mixture can be optimal for the reducing activation resistance, which were 1.0:2.0 with 10%, 1.0:1.8 20%, 1.0:1.1 40% and 1.0:0.5 60%.

In an experimentally study by Liu et al. [15], pore-forming improvements were examined to reach the Cathode Microporous Layer (CML) that has good optimization in DMFC. Carbon material type, carbon material loading, and pore-forming agent loading in CML were analyzed. The best carbon material was observed as a carbon nanotube (CNT) that supplies a number of specifications. The optimum value of the specific power was 292 mW·cm⁻² at 80 °C.
The Micro Direct Methanol Fuel Cell (MDMFC) has three parameters the methanol flow rate, the methanol concentration, and the cell temperature. In an experimental investigation by Yuan et al. [16], these parameters were studied under different conditions. Hence, optimized results were obtained.

In a theoretical study by Mert et al. [17], multi-objective optimization was demonstrated by different parameters as maximum power production capacity, exergy efficiency, and minimum production cost in four types of FCs. Exegetical and exergoeconomical tests were applied for these systems using Multi-Objective Optimizer Package (MULOP) software. Stability and performance of FCs were improved in this study.

Fares et al. [18] analyzed theoretically and experimentally a FC hybrid vehicle and aimed to regulate dynamic efficiency. Some mechanical parts of this system were analyzed in Simulink. PID controllers solved the time and cost optimization. Lower cost and lower consumption were reached in this study.

Stack efficiency has been designed with a different way for PEMFC by Piela and Mitzel [19] experimentally. FC system design can take advantages of this new design. Three convenient implementations of a new design were suitable for PEMFC applications. As a result of many experiments, that design’s new stack yields for a 480-W stack.

Feng et al. [20] examined theoretically the optimization of the structure of a single tubular SOFC. The maximum power output was chosen as the optimization objective. There exist an optimal cathode thickness and an optimal FC length which leads to the double maximum power output. The power output of the TSOFC after constructional optimization is increased.

In a theoretical and experimental study based on the real-time and multi-objective control algorithm design by Hu et al. [21], the aim is to optimize the powertrain system. The fuel economy and the system durability are the main concern of the study. The multi-objective optimization strategy called soft-run validate itself in FC hybrid city bus in three months.

Direct borohydride fuel cell (DBFC) has good electro-activity, energy storage capacity. For the vital optimization procedures, these skills are so important in DBFCs. In an experimental work by San et al. [22], different parameters like cell temperature, borohydride concentration, an anode, and a cathode flow rates were analyzed in package programs. Some values are decided as an optimum value in terms of package program. The optimum value of power density was at 0.7 mg·cm⁻² catalyst loading.

A theoretical and experimental study by Salva et al. [23] displays the optimization of a PEMFC in various parameters such as cell temperature, pressure, relative humidity, and cathode stoichiometry. Some data were created for the performance curved and analytical model. The analytical model that shows the highest power output is applied to physical conditions with 2% error.

In the optimizing of DMFC, there are three essential performance values that are low Methanol Crossover (MCO) rate, low water-transfer coefficient, and high cell voltage. These parameters are related to the optimization of the design of the DMFC membrane electrode assembly. In a theoretical research by Jung [24], it is sighted that membrane thickness, the material of the cathode catalyst layer can be optimized and improvement of cell voltage and decreasing MCO are observed.

Forough and Roshandel [25] studied theoretically the optimization of the design and operation of a SOFC integrated system. The multi-objective approach using a genetic algorithm is applied considering two pairs of proposed objectives: (1) Maximization of output power and minimization of the electricity cost. (2) Maximization of system electrical efficiency and minimization of the electricity cost. The results showed that the ratio of electrical efficiency of the system and electricity cost are more favorable.

Abdullah and Liu [26] observed theoretically that tailoring the microstructures of SOFC electrodes can offset the adverse effects of lowering operating temperatures. A complete cell level multi-scale polarization model being micro and macro models are developed separately. As a result, the combination of nonlinear particle-size and porosity-graded electrodes enable SOFCs to operate at a reduced temperature.

Cheng et al. [27] studied theoretically multi-timescale characteristics and gas transmission delays, the thermal electrical cooperative control of SOFCs. An analysis based on the optimization method that applies to a discrete optimization problem with constraints is proposed to obtain optimal operating points. The optimization results were proven feasible by the proposed new system structure and stack inlet condition-based control strategy.

In a theoretical study by Jokar et al. [28], a FC-heat engine hybrid system was investigated, thermodynamically. There were four parameters such as energy efficiency, power density, exergy destruction rate density and ecological function.
density. These are related to compressor inlet temperature and turbine inlet temperature of the Brayton cycle, and interconnect plate area and the current density of the FC. Three tests were done and one of them was the best option and it provided the optimum conditions.

In an experimental investigation by Piela et al. [29], seven variables about the FC temperature such as the reactants’ stoichiometric ratios, the reactants’ inlet relative humidity, and the reactants’ outlet pressures are investigated. Using these parameters, a scalar function is obtained. Two different stack conditions were applied, and FC had up to over 12% and up to over 7% for the stack voltage and efficiency, respectively.

Ou et al. [30] searched experimentally that water and thermal management system were improved for the open-cathode proton electrolyte membrane. Using multiple-input-multiple-output (MIMO) fuzzy controller, non-linear dynamical FC control parameters that are the axial fan speed control for regulating temperature and the solenoid valve on/off control of the bubble humidifier for humidity variation were decided. When these variables were controlled, the system was optimized.

The purpose of a theoretical study by Zeng et al. [31] is understanding of the effects of channel geometry on the mass transfer of the PEMFC. The power consumption of flow and power output of PEMFC are constants, wideness of a bottom and top edges of the channel are variables. Results show that the trapezoidal channel is the optimal design. Furthermore, optimal design shows more uniform distribution of reactants.

In a theoretical investigation by Eveloy et al. [32], to improve the capacity and efficiency of distributed power and fresh water generation in coastal industrial facilities affected by regional water scarcity, pressurized SOFC-gas turbine hybrid is integrated with a bottoming organic Rankine cycle and seawater reverse osmosis desalination plant. This power and water co-generation system is optimized in terms of two objectives, maximum exergy efficiency and minimum cost. A compromise between efficiency and cost, is compared. The results indicate the thermodynamic and economic benefits.

The aim of a theoretical study by Habibollahzade et al. [33] is to increase the power generation/exergy efficiency and reduce total product cost/environmental contamination of SOFCs. The first model assesses the combination of a gasifier with a SOFC. In the second model, waste heat of the first model is reused in the Stirling engine. A multi-objective optimization is applied based on the genetic algorithm.

Two purge strategies, voltage-based purge, and nitrogen-based purge are examined in a theoretical research by Wang et al. [34]. The most effective factor of purge duration is scavenging velocity that mainly affects fuel less loss rate. Voltage-based is recommended because, in another type, nitrogen fraction measurement is difficult.

The purpose of a theoretical study by Chowdhury et al. [35] is to optimize the channel to land. Within this study, 73 channel geometry cases are used three-dimensional isothermal single phase is generated to examine 73 number of case studies. Optimization analyses are applied when the voltage of 0.4 V and channel depth 1.0 m fixed according to experiment; current density and pressure are the most important effects for optimization. The experiment shows that pressure drop more dependent on channel width compare to land with an anode pressure drops less significant than cathode pressure drops. On the other hand, both channel and land width have equal importance for cell current density. The best result was the channel to land width of 1.0 mm/1.0 mm for PEMFC geometry.

In a series of theoretical studies by Dincer et al. [36-52], the optimization of a variety FC is studied, thermodynamically. Comprehensive energy and exergy analyses are examined, so thermodynamic optimization parameters are determined. Investigation of performance assessment through efficiency, sustainability, and durability is the major aim of these studies using several solutions. In conclusion, design, energy, and exergy optimization were observed.

In a study by Yilanci et al. [36], a PEMFC unit in a solar-based hydrogen production system is investigated, thermodynamically. In another study by Zamfirescu and Dincer [37], SOFC is optimized with a new power and heating model. A research by Rashidi et al. [38] examined MCFC with a turbo expander. Schematics of conventional (burner only), current (burner and turbo expander), and near future (turbo expander; burner replaced by MCFC excess heat) Enbridge system are shown in Fig. 2.

The feasibility of strengthening a series of a system that has a mobility with a DMFC was investigated in a theoretical and experimental study by Rashidi et al. [39]. In a study by Mert et al. [40], a few optimization parameters like efficiency, cost, and power output were observed in PEMFC changing some conditions such as temperature, pressure, surrounding temperature and pressure, current density, humidity, and membrane thickness. Hydrogen and fuel cell technologies for better sustainability was investigated with energy and exergy analyses by Dincer and Rosen [41]. Four key parameters of sustainability are represented in Fig. 3.
In a study by Ramandi and Dincer et al. [42], both computational fluid dynamics (CFD) analysis and thermodynamic analysis are implemented in MCFC for current densities lower than 2500 A m\(^{-2}\). A research by Mert et al. [43] compared a variety of FC engine system in different operating conditions for how efficiency change. Thermoeconomic optimization based on Powell’s Method of three new trigeneration systems is done using the Rankine cycle in a study by Al-Sulaiman et al. [44]. In an investigation by Ozcan and Dincer [45], SOFC based trigeneration system was analyzed for fuel types and its utilization then the effect of exhaust gas for efficiency was observed. The system that was created with biomass gasification and SOFC is analyzed in terms of the first and second law of thermodynamics in a study by El-Emam and Dincer [46]. Schematic of the proposed integrated system is indicated in Fig. 4.

![Figure 2. Schematics of conventional, current, and near future Enbridge system [38].](image)

![Figure 3. Four key parameters of sustainability [41].](image)

![Figure 4. Schematic of the proposed integrated system [46].](image)
Urea-fed SOFC integrated with a gas turbine was investigated in terms of thermodynamic efficiency and urea fuel’s health and safety risks in a study by Abraham and Dincer [47]. Dynamic specifications of the PEMFC that has different parameters on a model is analyzed in a study by Bicer et al. [48]. A FC–photovoltaic system for vehicle application was examined in energy and exergy analyses and efficiency in the research by Ezzat and Dincer [49-50]. A PEM–wind energy system was investigated for different wind speeds and its thermodynamic analyses were performed in a study by Ishaq et al. [51]. There are several parameters for energy systems in optimizing in terms of such as the output, profit, productivity, product quality, etc. [52]. The study by Ahmadi and Dincer [52] explained the vitality of optimization in energy systems, especially with mathematical models. The mathematical optimization of a system is demonstrated in Fig. 5.

![Figure 5. The mathematical optimization of a system [52].](image)

3. CONCLUSIONS

This study is a fuel cells’ short overview that includes the short history of fuel cells, competing for power generation technologies, and fuel cell types. This overview has advantages and disadvantages of FCs, design modeling, thermodynamic characteristic, and its operation factors. Also, economic variables and cost optimization are an essential point. Successful optimization technics are getting common the fuel cells and they can be used for modern applications. FCs are a promising alternative to current energy sources. They essentially combine the energy density and the convenience of other sources with the clean and efficient operation of a variety of systems. Although certain aspects of technology such as efficient onboard storage still require some improvement, there are no reasons why FCs couldn’t become an equally convenient and attractive energy source at today. To improve the FCs’ parameters, there are a number of optimization applications. This study tried to examine these types of research and to examine how much important.

REFERENCES


NUMERICAL and FINITE ELEMENT SIMULATION of LAMINATE BEAM WITH BI-LINEAR PLASTIC BEHAVIOUR UNDER PURE BENDING CONDITIONS

Barış M. Zeytinci, Mehmet Şahin, Mehmet Ali Güler

1Söğütözü Mahallesi, Söğütözü Cd. No:43, 06510 Çankaya/Ankara, Türkiye
2Fethiye Mahallesi, Havacılık Bulvarı No:17 06980 Kahramankazan/Ankara, Türkiye
3Söğütözü Mahallesi, Söğütözü Cd. No:43, 06510 Çankaya/Ankara, Türkiye
1bzeytinci@etu.edu.tr, 2mehmet.sahin2@tai.com.tr, 3mguler@etu.edu.tr

ABSTRACT
In this paper, the behavior of a laminate I-beam with bi-linear plastic properties is investigated under pure bending conditions using numerical and finite element methods. I-beam is assumed to bend as an Euler-Bernouilli beam. It is also assumed to fail in three different ways; elastic, partially plastic and fully plastic, depending on the amount of the applied load. A pseudo code is written in PYTHON for the numerical solution of the section moment capacity of the beam. The formulation for the moment capacity and the results of numerical solution is verified using Finite Element Analysis (FEA). Four-noded SHELL 181 element meshing is applied to the geometry and bi-linear plastic behavior is defined with Bi-linear kinematic hardening rule in ANSYS 2018. Four-point bending is simulated in ANSYS. For the numerical results, five different types of laminate beam, each has different material properties, are selected and analyzed. The results of numerical solution and finite element solution are compared.

Keywords: Laminate beams, plasticity, numerical simulation, multi-layer beams, FEA of I-beam

1. INTRODUCTION
I-beams are the structural elements that are widely used in Aerospace, Ship Building and Construction industries. Day by day, engineers try to improve the geometrical and mechanical properties of I-beams. I-beams, that are widely used in the structures of the airplanes or helicopters, are still a topic of investigation in terms of strengthening, weight reduction and optimization. For example, perforated I-beams are used in the airplane structures to have more space for the cables and tubes. In Figure 1, an example of the usage of the I-beams for aerospace industry is provided.

Figure 1: Floor Beam (Celik M., Guler M.A., Orun A. E., 2016)
Generally, beams used in the structural elements, fail because of buckling, high moment load and because of high shear if the beam is perforated (Vierendel action). Laminated beams, are also used in many industries. Laminated beams started to use in the beams because of its improving mechanical properties. It is critical to investigate the mechanical behavior of these beams. This study, is primarily focused on the moment capacity of laminated I beams under pure bending conditions.

There are many studies in the literature about pure bending of I-beam. Lubarda (2017), derived microstress, moment-stress and line forces for a rigid-plastic beam o rectangular crosssection under pure bending in the model of strain-gradient plasticity and obtained closed form analytical solutions for both linear and non-linear hardening models. Ozutok and Madenci (2017), analyzed a composite laminated beam with mixed finite element(MFEM) equations based on Gateux differential(GD) using Euler-Bernoulli first order shear deformation theory. After the analysis performance of the solution is compared with some test problems. Shi et al, investigated the Multiple-Layered beams under bending with extraordinarily different elastic properties. Beam is composite formed from silicon and Polydimethylsiloxane. They provided compared with the strain distribution relations FEA.

There are some studies in the literature for the perforated composite beams. D’Mello and Tsavdaridis (2011), studied the composite perforated I-beams plastic moment capacity that has different geometrical openings on it. In their study, I-beam was composed of flange and web with different material properties. Which was used by many engineers to attain the mechanical needs. First of all, they compared their finite element with the coupon tests of (Mccutheon and Redwood, 1968). After that, they perforate the I-beams with different geometries and obtained the best result for the plastic moment. A similar study was worked by Chung, 2001, for the composite beams. Chung, 2001 used only circle perforated I-beams and gets the best results for them.

2. NUMERICAL SOLUTION

In this part of the paper, numerical simulation of plastic moment and normal stress at the center of the beam was taken into account. While making the simulations geometry of the I-beam was defined with parametrical units which can be seen in Figure 2.

![Figure 2: Geometry of I-beam](image)

I-beam was defined as parameters as also the material properties of the I-beam was defined as parameters.

$\sigma_{y(1,2)}$: Yield Strength (Depends on the laminated structure)

$E_y$: Elastic Modulus

$E_{ty}$: Tangent Modulus

$y_y$: Yielding Distance

$\kappa$: radius of curvature

While writing a script about the simulation of I-beam, phyton programming was used. As mentioned before everything of the I-beam’s geometrical and material properties was defined as parameters so that it can easily be changed and optimized. While writing the programming code I-beam was assumed to act according to Euler’s beam theory as also used by Ozutok and Madenci (2017). I-beam was investigated in three different situations. These are when the I-beam is elastic, partially plastic and plastic. Moreover, some of the other assumptions are listed below.
Consider an I-beam with N layers of different materials having different bilinear properties. The beam is assumed to have symmetrically placed layers with constant width \( b_j \), which can be shown as a function of \( y \), i.e., \( b = b(y) \). Each layer has a thickness of \( t_j = h_{j+1} - h_j \), and total height of the beam is \( h \). The beam is subjected to transverse loading causing a moment \( M \) at a point along the beam axis. The governing equation for the strain deformation relation in the beam is given as

\[
\varepsilon = \frac{y}{\rho} = \kappa y
\]  

(1.1)

Where \( \rho \) is curvature of the deflection curve and \( \kappa = 1/\rho \) is the inverse of the curvature. The corresponding stress for the strain, either elastic or inelastic, depends upon the strain level and can be expressed as

\[
\sigma = \begin{cases} 
E\varepsilon & \text{if } \varepsilon \leq \varepsilon_y \\
\sigma_y - E_y\varepsilon_y + E_y\varepsilon & \text{if } \varepsilon > \varepsilon_y 
\end{cases}
\]  

(1.2)

The differential force and moment resulting from the force on the \( j \)th layer are calculated as

\[
dF_j = \sigma(y)b(y)dy \quad j = 1, 2, 3, ..., N
\]

(1.3)

\[
M_j = \int_{b_j}^{h_{j+1}} ydF = \int_{b_j}^{h_{j+1}} yb_j\sigma dy
\]

(1.4)

These equations can be numerically calculated by using rectangular or trapezoidal rule as

\[
M = \sum_{j=1}^{N} M_j = \int_{b_1}^{h_{N+1}} yb_j\sigma dy \approx \sum_{i=1}^{K} b_i y_i \sigma_i \Delta y_i
\]

(1.5)

Where \( K \) is number of discrete integration points along the total height of the I-beam section and \( b_j \) is the width of the section at the \( i \)th station which is constant, \( b_j \), for the \( j \)th layer. The stress calculated as

\[
\sigma_i = \begin{cases} 
E\varepsilon_i & \text{if } \varepsilon_i \leq \varepsilon_{yi} \\
\sigma_{yi} - E_{yi}\varepsilon_{yi} + E_{yi}\varepsilon_i & \text{if } \varepsilon_i > \varepsilon_{yi} 
\end{cases}
\]  

(1.6)

Where all strain and material values are calculated at the \( i \)th point.

For numerical implementation, for each material

- Assign the material yield stress, elastic and tangent moduli
- Assign thickness of the layer and starting and ending coordinates of the layer along the cross-section of the beam.

The numerical solution pseudo-code can be summarized as

- Calculate the yield strain at the outermost fiber of the beam (Nth layer)
  \[
  \varepsilon_{yi} = \frac{\sigma_{yi}}{E_y}
  \]

- Calculate corresponding curvature \( \kappa_{yi} \)
\[ \kappa_{yN} = \frac{\epsilon_{yN}}{\left( \frac{h}{2} \right)} \]

- Assign a maximum curvature for the analysis at your disposal, let say \( \kappa_{\text{max}} = 20\kappa_{yN} \).
- Divide the curvature interval into steps, i.e. \( \kappa_i = \left( \kappa_{\text{max}} / \text{Number of curvature steps} \right) \)
- Start a do-loop for increasing curvature of the section.
  - Do-while (i < Number of curvature steps)
    - Do-while (j < Number of integration points)
      - Calculate strains
      - Calculate elastic or inelastic stress depending upon the strain
      - Calculate the width of the section
      - Calculate moment and moment summation

The python code to calculate the section moment capacity is shown below

```python
import numpy as np

def plasticStressDistributionAlongLaminaHeight(E,ET,Sy,b,yL,kappa):
    e_y = Sy/E
    eL = kappa*yL
    S = E*eL
    S[eL<e_y] = E*eL[eL<e_y]
    S[eL>=e_y] = Sy-ET*e_y+ ET*eL[eL>=e_y]
    dy=yL[1]-yL[0]
    ymax = yL[-1]
    Mw=0
    for Se,ye in zip(S,yL):
        Fw =Se*b*dy
        Mw+=Fw*(ye+dy/2)
    return Fw,Mw,yL,eL,np.array(S),ymax

def plasticStressDistributionAlongSectionHeight(layers,kappa):
    yL=np.array([])
    eL=np.array([])
    SL=np.array([])
    ymaxL=[]
    for layer in layers:
        E,ET,Sy,b,yLn=layer
        Fn,Mn,yLn,eLn,SLn,ymax = plasticStressDistributionAlongLaminaHeight(E,ET,Sy,b,yLn,kappa)
        yL=np.concatenate((yL, yLn), axis=0)
        eL=np.concatenate((eL, eLn), axis=0)
        SL=np.concatenate((SL, SLn), axis=0)
        ymaxL.append(ymax)
    return kappa,yL,eL,SL,ymaxL
```

3. FINITE ELEMENT SOLUTION

On this part of the paper, to prove the results of the numerical simulation, ANSYS 2018 finite element program was used. By using Solidworks 2015 I-beam was drawn. After drawing the I-beam, geometry was imported to the Finite Element Meshing program, ANSA. With ANSA 4-noded Shell 181 element, beam was meshed according to literature that is provided by D’Mello and Tsavdaridis, 2011. To optimize the solution process, mesh verification was applied for the I-beam and analysis got started.

Plastical analysis are used highly in ANSYS 2018, while making the analysis, geometry was selected to be laminate, which all has the same Tangent Modulus (E\(_t\)). While making the analysis, material is assumed to bend according to the Von-Misses Criterion. The material is defined as the kinematic hardening rule. Moreover, solver program is selected to be iterative Newton-Raphson Method. To make the bending analysis in ANSYS 2018, 4-point contact bending method
was used according to the literature which was also used by Dessouki et al., 2015. 4-point bending and meshing of the I-beam was given in Figure 3 and 4.

4. RESULTS AND COMPARISON

In able to check the numerical investigation, a geometry that is used by Chung, 2001 was selected for verification. Used parameters are presented in Table 1.

<table>
<thead>
<tr>
<th>Variable</th>
<th>Value</th>
<th>Note</th>
</tr>
</thead>
<tbody>
<tr>
<td>$h$</td>
<td>206.4 (mm)</td>
<td></td>
</tr>
<tr>
<td>$d$</td>
<td>189.9 (mm)</td>
<td></td>
</tr>
<tr>
<td>$b$</td>
<td>133.4 (mm)</td>
<td></td>
</tr>
<tr>
<td>$t_w$</td>
<td>6.32 (mm)</td>
<td></td>
</tr>
<tr>
<td>$t_f$</td>
<td>8.23 (mm)</td>
<td></td>
</tr>
<tr>
<td>$E_y$</td>
<td>200 (GPa)</td>
<td>Same for all types</td>
</tr>
<tr>
<td>$E_t$</td>
<td>1000 (MPa)</td>
<td>Same for all types</td>
</tr>
<tr>
<td>$\sigma_y$</td>
<td>~</td>
<td>Depends on layer</td>
</tr>
</tbody>
</table>

As can be seen from Table 1, Yield Stress of each layer changes depending on the user. In this study 5 different types of I-beam were chosen. These types are a, b, c, d and e. For each type, layers increase gradually. Type a consists of only one material, type b consists of two, c consists of three, d consists of four and lastly, type e consists of 5 materials. More detailed explanation of different beams is provided in Figure 6.

As explained before, plastoical analysis were done by using ANSYS 2018. To check the trueness of ANSYS 2018 plastic analysis, Plastic Moment vs. Mid-Span deflection graphic, given on Figure 5.
Plastic Moment vs. Mid-Span Deflection results of the type b (bi-material) is presented in the above figure. As can be seen from this figure, it is clearly observed that, ANSYS 2018 is able to solve plastic analysis and beam shows a bi-linear plastic behavior trend as expected. More detailed explanation of selected 5 different laminated I-beams can be seen in Figure 6.

With these 5 different types, both numerical and Finite element programs are compared. Compared results show us that, numerical investigation that was done is correct. Analysis that were done can be seen on figure 7. Please note that, these analyses were investigated for only comparing the numerical and finite element results. Because of this reason, only maximum stress is calculated for the selected beams, to see the effect of each layer’s yield stress. However, script also allows to calculate stress and moment at a desired time.
In each analysis web part divided in to equal layers and Yield Stress ($\sigma_y$) increased gradually according to literature review which can be seen from Figure 6 also. However, flange part of the beam is only consisting of one material. Results show us that by increasing the layers Yield Stress gradually, Maximum Capacity of Stress and Maximum Plastic Moment Capacity of an I-beam increases. With the help of these graphs results showed that, additive(layered) manufacturing have positive effects on I-beam and can be used while making a design. Of course, these analysis were only done to check the truthfulness of numerical investigation. Layers can be increased as many times as user wants. Also Yield Stress of each layer can change according to user.

Figure 5: Comparison of Results (a) mono-material, (b) bi-material, (c) tri-material, (d) quatro-material, (e) tetra-material
From Figure 7, some difference occurs between numerical simulation and finite element simulation which is very normal. In numerical simulations, I-beam assumed to be perfect. However, in Finite Element simulations because of mesh and other geometrical imperfections, beam act like non-symmetric. That’s why, at the center of the beam, stress is greater than 0 MPa. But, still, 95% of the results are coincide with each other.

5. CONCLUSION AND RECOMMENDATION

In Conclusion, I-beams (which is an highly used structural element in engineering), structural analysis are investigated. Bi-linear material property is selected and beam was defined with layers. Beam is assumed to fail as Euler beam and accepted to show three different trends. These are elastic, partially plastic and fully plastic. Distributed load has applied to the top flange of the beam, while bottom flange is supported from each side which is pure bending. To make the analysis parametrically, numerical derivations are used by Phyton programming language. For verification, finite element analysis was also done by using ANSYS 2018. Five different types of I-beams were selected to see the truthfulness in different types. Finite Element Results show that, numerical solutions are correct and gives the Stress Distribution of a layered I-beam under pure bending. With the help of Numerical program by Phyton, engineers can calculate I-beams; Plastic Moment, Maximum Stress and Stress Distribution at the Mid-Span, in milliseconds. Also, program gives us the chance to define the beam as layers, which is a highly used technology nowadays. Beginning with this study, future works can be done. These are;

- Symmetrical and non-symmetrical simulation of different beams.
- Simulation of an I-beam which is perforated with different geometries.

6. REFERENCES


COMPUTATIONAL INVESTIGATION OF CONFINED WALL INCLINATION EFFECTS
ON IMPINGING JET HEAT TRANSFER

Kenan Nur\textsuperscript{1}, Ebubekir Beyazoğlu\textsuperscript{2}, Erhan Pulat\textsuperscript{3}

\textsuperscript{1,2,3} Uludağ University, Department of Mechanical Engineering, Bursa, Turkey
\textsuperscript{3} Corresponding Author: E. Pulat, pulat@uludag.edu.tr

ABSTRACT

In this study, inclination angle effects of confined wall on impinging jet heat transfer are investigated computationally. Mainly five inclination angles of confinement plate are considered. Flow is two-dimensional, steady, incompressible, and turbulent. SST k-ω turbulence model was chosen since this model is recommended in some recent studies. This turbulence model is integrated-to-wall model, and near wall mesh structure is very important. Conservation equations of resulting computational model were solved by using ANSYS-Fluent code. Flow structures are completely governed by inclination angle, and center of main vortex moves to right by increasing inclination angle. In addition, predictions show that inclination angle has great influence on heat transfer in the stagnation point and near the second peak.

Keywords: Impinging jet, confined wall inclination, heat transfer, turbulence, CFD

1. INTRODUCTION

Impinging jet heat transfer, either free or confined, is the most effective heat removal method especially for high heat flux applications (EBadian and Lin, 2011). Impinging heat transfer depends on the some flow and geometric parameters such as jet Reynolds number, nozzle shape, nozzle-to-plate distance etc. (Meola continued, 1996; Morris and Garimella, 1998; Volkov, 2007). A recent comprehensive review about flow and thermal characteristics of jet impingement can be found in (Shukla and Dewan, 2017).

Although there are a lot of experimental (Garimella and Rice, 1995; Fitzgerald and Garimella, 1998; Bart continued, 2002; Huzayyin continued, 2006; Hofmann continued, 2007a; Anwarullah continued, 2012; Attalla, 2015; Ahmed continued, 2017), theoretical and computational studies (Shi continued, 2002; Hofmann continued, 2007b; Isman continued, 2008; Pulat continued, 2011; Wang continued, 2014; Modak continued, 2015; Isman continued, 2016; Zhu continued, 2017), effect of confined wall inclination was less studied.

In literature, single jet cases of confined wall inclination studies are related to only converging confined channel in laminar flow conditions (Mirando and Campos, 1999; Cavadas continued, 2006). In the context of the authors’ knowledge, diverging confined channel in turbulent flow conditions does not used in single impinging jet studies, and the aim of this computational study is to investigate the effects of inclination angle on flow structure and heat transfer of single turbulent impinging jet in diverging flow conditions.

2. MATERIALS AND METHODS

2.1. Geometry and Boundary Conditions

Two-dimensional impinging jet geometry together with boundary conditions is given in Figure 1. Since confinement wall inclination effects are considered, all geometrical parameters and boundary conditions are chosen in consistent with relevant experimental study with confined wall (van Heiningen, 1982) together with computational study that simulates this experimental study (Shi continued, 2002). Inlet length is D and uniform velocity profile is defined at the inlet. Outlet boundary condition is applied at the outlets. For all walls, no slip boundary condition is applied. Constant surface temperature Ts is assumed at impinging wall and confinement walls are adiabatic. α is inclination angle to provide divergent channel conditions in both outlets and five inclination angles are considered. Dimensions and boundary conditions are summarized in Table 1.
1st International Conference on Advances in Mechanical and Mechatronics Engineering
ICAMMEN 2018

Figure 1. Impinging geometry and boundary conditions

Table 1. Dimensions and boundary conditions

<table>
<thead>
<tr>
<th></th>
<th>Inlet</th>
<th>Outlet</th>
<th>Impinging wall</th>
<th>Confinement wall</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length</td>
<td>D=0.0141 m</td>
<td>Z=2.6D (α=0°)</td>
<td>L=70.92D</td>
<td>L - D</td>
</tr>
<tr>
<td>Velocity</td>
<td>U∞=12 m/s (Uniform)</td>
<td>-</td>
<td>No-slip</td>
<td>No-slip</td>
</tr>
<tr>
<td>Turb. intensity</td>
<td>Tu=0.02</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Length scale</td>
<td>0.07D</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Temperature</td>
<td>310 K</td>
<td>-</td>
<td>T≈348 K</td>
<td>Adiabatic</td>
</tr>
<tr>
<td>Pressure</td>
<td>-</td>
<td>P=0</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Inclination angle</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>α=0, 2, 4, 6, 8°</td>
</tr>
</tbody>
</table>

2.2. Mesh Structure and Numerical Procedure

Firstly, computations for simulated geometry of van Heiningen’s experimental study with α = 0° were performed and this case called as base case. Computational domain is divided into 33500, 51500, and 74150 cells to provide mesh independency. Nusselt number (Nu) distributions for all cell numbers and medium mesh structure (51500 cells) are given in Figure 2. As seen from this figure, Nu number distribution with the mesh structure with 51500 cells is the most consistent distribution with experimental measurements and it is reasonable to choose this medium mesh structure. In inclined confinement wall geometries, similar mesh structures were constructed for all inclination angles and it was decided to mesh structure in the light of the results of the base case.

Flow is two-dimensional, steady, incompressible, and turbulent. Constant thermophysical properties of air are considered at the inlet. SST k-ω turbulence model was used in this study since this model has been suggested in many studies as a result of comparison of this model with other models such as Std. k-ε, RNG k-ε, Realizable k-ε, and Std. k-ω (Sagot continued, 2008, Wang continued, 2014), and in addition to conventional case of impinging jet flows it was used successfully in more complex cases of impinging jet flows including swirling (Ahmed continued, 2017). As seen from Figure 2, secondary peak that observed in low Z/D cases is captured successfully by SST k-ω model. Since standard form of SST k-ω model with curvature correction, production Kato-Launder, and production limiter is used without any modification of model constants, conservation equations and turbulence model equations are not given. Since this turbulence model is integrated-to-wall model, and near wall mesh structure is very important, very fine meshes are formed.

Figure 2. a) Comparison of different number of cells with measurements and b) mesh structure (α = 0°)
near the impingement wall such that mean $y^+$ value is equal to $\sim 4$ at this wall for base case and medium cell number as seen from Figure 2. Least squares cell based is used in spatial discretization of gradient, and first order upwind is used in spatial discretization of turbulent kinetic energy. In spatial discretization of pressure, second order is used. Second order upwind is used for both density and momentum. Discretized equations are solved using ANSYS Fluent 16.0 with the SIMPLE method as pressure-velocity coupling. Number of iterations is taken as 2000.

3. RESULTS AND DISCUSSION

Jet impinging flow geometry offers one of the least complex separating and reattaching flows. Studies how the flow structure and heat transfer in jet impinging flows are affected by pressure gradient were limited to laminar converging cases, and effects of adverse pressure gradients on streamlines and heat transfer for impinging thermoflow case are presented as follows.

3.1. Streamlines

The effects of pressure gradient on streamlines are given in Figure 3 for all considered inclination angles. Predictions were obtained only for single jet Reynolds number of 10200 and Reynolds number is defined in terms of jet velocity and inlet length as follows

$$Re_j = \frac{U \cdot D \cdot \upsilon}{u} \quad (1)$$

![Figure 3. Streamlines for considered inclination angles (Re\textsubscript{j} = 10200)](image)

The reattaching shear layer is the presence of a pressure gradient and occurs on a surface, making strong interaction between the separation and reattachment process likely (Driver and Seegmiller, 1985). As seen from Figure 3, flow
structure and reattachment process is very sensitive to inclination angle of confinement plate, and reattachment length (RL) increases with increasing inclination angle. At the beginning, this increase is considerably high since reattachment length increases twice while angle is increased to 2°. Increase in reattachment length slows down after the inclination angle of 2°. After 4°, streamlines covers all computational domain and reattachment length exceeds the domain at the exit for 6° and 8°. In addition to reattachment length, center of main vortex moves to right regularly by increasing inclination angle.

3.2 Heat Transfer

The heat transfer coefficient over the impingement surface was normalized in the form of a local Nusselt number for an isothermal impingement surface, and the reference temperature \( T_r \) is the inlet jet temperature.

\[
Nu = \frac{h_x}{\nu} \quad (2)
\]

where

\[
h_x = \frac{q}{(T_s - T_r)} \quad (3)
\]

Local Nusselt (Nu) number distributions are given in Figure 4 in comparison with experimental measurements for base case of \( \alpha = 0° \). Nusselt number distribution predicted by SST k-\( \omega \) model is very well qualitatively in comparison with experimental values for the base case. Stagnation point heat transfer is overpredicted and this overprediction is attributed to excessive kinetic energy production in the stagnation region (Durbin, 1996), but this overprediction is less than overpredictions computed from k-\( \varepsilon \) based two equation models. When the jet discharge is close to the wall, the mixing layer formed at the edge of jet does not impinge directly onto the plate (as seen from Figure 5 region A), but the larger scales and enhanced turbulence levels in the mixing layer lead to a strong rise in Nusselt number (secondary peak as seen from Figure 5 region B) (Craft, continued, 1997). This secondary peak is successfully captured although its location is slightly overpredicted in Nu and underpredicted in \( x/D \) coordinates as seen from Figure 4. In wall-jet region, after \( x/D > 10 \) consistency of prediction with measurements is very well.

For the inclined cases, in general, local Nu number predictions for all inclination angle cases except \( \alpha = 8° \) are similar to base case. In stagnation region (\( x/D < 11 \)), similar predictions are obtained, but in wall-jet region (\( x/D > 10 \)), local Nu number distribution slightly increases with increasing inclination angle. Local Nu number prediction for \( \alpha = 8° \) is different in comparison with others in the stagnation region. It is needed further investigation for this difference for inclination angles that are greater than 8°.

![Figure 4. Local Nusselt number distribution for considered inclination angles (Rej = 10200)](image-url)
4. CONCLUSIONS AND RECOMMENDATIONS

SST k-ω turbulence model was used to compute the heat transfer under confined two dimensional impinging jet for various inclination angles of confinement plate. Flow structures are completely governed by inclination angle. Reattachment length increases with increasing inclination angle, but increase in reattachment length slows down after the inclination angle of 2°. In contrast, center of main vortex moves to right regularly by increasing inclination angle. Local Nusselt number distributions under specified conditions is consistent with experimental trend qualitatively for base case. In inclination angle cases of less than 8°, heat transfer from wall-jet region slightly increases with increasing inclination angle. But in the case of 8°, local Nu number distribution is different from other cases although exhibits similar trend. Further investigation is needed for inclination angles that are greater than 8°.

REFERENCES


MIXED CONVECTION OF NANOFLUID IN A VENTED CAVITY WITH MAGNETIC FIELD

Fatih Selimefendiğil¹, Hakan F. Oztöp²

¹ Department of Mechanical Engineering, Celal Bayar University, 45140 Manisa, Turkey
² Department of Mechanical Engineering, Technology Faculty, Firat University, 23119 Elazığ, Turkey
¹fthsel@yahoo.com, ²hfoztop1@gmail.com

ABSTRACT

In this study, mixed convection of CuO-water nanofluid in a vented cavity was examined under the effect of an inclined magnetic field. Galerkin weighted residual finite element method was used for the solution of the governing equations with boundary conditions. Effects of various pertinent parameters such as Reynolds number, Hartmann number and magnetic inclination angle on the fluid flow and heat transfer characteristics are analyzed. Magnetic field is found to suppress the recirculation zones in some locations of the cavity affects the variation of local and average Nusselt number. The effect of magnetic inclination angle on the heat transfer enhancement is marginal as compared to strength of the magnetic field. Various components of the hot wall contribute differently to the total average heat transfer rate with respect to changes in the magnetic field.

Keywords: MHD convection, nanofluid, vented cavity, mixed convection, finite element method

1. INTRODUCTION

Convection in cavities with inlet and outlet ports is important in many engineering applications such as in the design of ventilation systems, MEMs, solar power and many others. The complicated nature of the multiple circulation zones established in the cavity will be complicated further with the addition of natural convection effects [1-4]. Flow control and heat transfer enhancement in those systems can be obtained by many active and passive techniques. In one of these methods, nano sized particles can be added to the base fluid in order to enhance the thermal transport and increase the heat transfer rate. In this technology, the so called nanofluids technology, metallic or non-metallic of nano sized particles which have much higher thermal conductivity as compared to the heat transfer fluid is added to the base fluid [5-9]. The nanofluid technology was successfully implemented in many applications ranging from refrigeration to solar power.

Magnetic field effects can be encountered in geothermal energy extraction, metal casting, nuclear reactor cooling and many others. In convective heat transfer, an external magnetic field can be used to control the convective heat transfer and fluid flow features in cavities [10-14]. In many applications where magnetic field was used to control the convective heat transfer rate, magnetic field effects were found to reduce the convection and dampen the fluid motion. However, in separated flow applications, magnetic field has the potential to resize the separated zone and enhance the heat transfer. Adding nanoparticles to the base fluid in the applications with magnetic field has another possibility to control convection. This is due to the fact that not only the thermal conductivity of the base fluid enhances with nanoparticle addition but also the electrical conductivity as well. In the literature, a lot of relevant studies can be found where magnetic field effects are examined with nanofluids in convective heat transfer applications [15-18].

In the present study, magnetic field effects on the mixed convection of CuO-water nanofluid flow in a cavity with inlet and outlet ports were analyzed with numerical simulations. The result of this investigation will be helpful for the thermal design and optimization of systems encountered in various thermal engineering problems as mentioned above.
2. NUMERICAL MODELLING

Flow field and heat transfer characteristics in a vented cavity with magnetic field is studied. A schematic description of the model is shown in Figure 1. A square cavity of size of H with an inlet and outlet port of sizes w=0.25H is considered. A cold flow with temperature T_c and velocity u_0 enters the inlet while the walls of the cavity are assumed to be isothermal at temperature of T_h where T_h > T_c. The flow is two dimensional and steady while the Newtonian and incompressible fluid assumption is utilized. A uniform magnetic field was imposed and the cavity is filled with CuO-water nanofluid. Thermophysical properties of base fluid and nanoparticle are given in Table 1. Fluid flow and heat transfer equations are given by the following equations:

\[
\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \tag{1}
\]

\[
u \frac{\partial u}{\partial x} + \frac{\partial u}{\partial y} = -\frac{1}{\rho_{nf}} \frac{\partial p}{\partial x} + \nu_{nf} \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right) + \frac{\sigma_{nf} B_0^2}{\rho_{nf}} \left( \nu \sin(\gamma)\cos(\gamma) - u \sin^2(\gamma) \right) \tag{2}
\]

\[
u \frac{\partial v}{\partial x} + \frac{\partial v}{\partial y} = -\frac{1}{\rho_{nf}} \frac{\partial p}{\partial y} + \nu_{nf} \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right) + \frac{\sigma_{nf} B_0^2}{\rho_{nf}} \left( \nu \sin(\gamma)\cos(\gamma) - v \cos^2(\gamma) \right) + \beta_{nf} g(T - T_c) \tag{3}
\]

\[
u \frac{\partial T}{\partial x} + \frac{\partial T}{\partial y} = \alpha_{nf} \left( \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} \right) \tag{4}
\]

where the Lorentz forces due to the magnetic field are seen in the momentum equations.

The effective density and heat capacity of the nanofluid are given as:

\[ho_{nf} = \phi \rho_p + (1 - \phi) \rho_c, \quad \left( \rho C_p \right)_{nf} = \phi \left( \rho C_p \right)_p + (1 - \phi) \left( \rho C_p \right)_c \tag{5}
\]

The effective viscosity and thermal conductivity of the nanofluid are described as:

\[
\mu_{nf} = \frac{\mu}{(1 - \phi)^2}, \quad k_{nf} = k_f \left[ \frac{(k_p + 2k_f) - 2\phi(k_r - k_p)}{(k_p + 2k_f) + \phi(k_r - k_p)} \right] \tag{6}
\]

The fluid enters the inlet port with a uniform velocity of u_0 and temperature of T_c while fully developed flow conditions are assumed at the outlet port. The walls of the cavity are at constant hot temperature of T_h. The solution of the governing equations along with the boundary conditions are made by using the Galerkin weighted residual finite element method where the weak form of the governing equations is established. The weighted average of the governing equations satisfy the convergence criteria which is less than 10^{-3}. The present solver is validated by using the numerical results of [19] where natural convection effects in a cavity with magnetic field was investigated. Table 2 shows the comparison of average Nusselt number for two values of Hartman number at Grashof number of 2x10^5.

3. RESULTS AND DISCUSSIONS

Fluid flow and convective heat transfer features of a cavity with inlet and outlet ports is examined under the effects of a uniform magnetic field. The cavity is filled with CuO-water nanofluid. Effects of Reynolds number on the flow and thermal pattern distributions are shown in Figure 2 for fixed values of (Ha=20, γ=45°). As the value of Re increases, a corner vortex in the top right part of the cavity is established and its size increases with Reynolds number. The vortex center below the main below moves toward the outlet of the cavity. Thermal gradients are less clustered for the wall.
below the inlet port while they become denser with the rise of Reynolds number. The average Nusselt number versus Reynolds number shows almost a linear relation which has a positive slope.

Effects of Hartmann number on the variation of flow and thermal patterns are demonstrated in Figure 4 for fixed values of (Re=300, γ=45°). In the cavity flow application magnetic field effects were found to dampen the fluid motion and thus reduce the convective heat transfer rate. In the presence of magnetic field at Ha=20, the top right corner vortex diminishes in size as compared to the case in the absence of magnetic field. It disappears with further increment of Hartmann number to Ha=50 while the vortex below the inlet diminishes in size and its center is closing to the inlet port. Isotherms become denser on the bottom wall and right wall above the outlet at the highest value of Hartmann number due to the deterioration of vortices. Average Nusselt number versus Hartmann number plots are shown in Figure 5 for two values of Reynolds numbers. At low Reynolds number, its value first reduces until Ha=20 and increases thereafter. However, at Reynolds number of 500 where forced convective effects are dominant, its value reduces with higher values of Hartmann number. Contribution of different walls on the overall heat transfer changes in the absence and presence of magnetic field as shown in Figure 6. The contribution of walls P2 and P3 reduces with the applied magnetic field. Variation of the average Nusselt number with respect to changes in the magnetic inclination angle is shown in Figure 5 at two values of Reynolds number with Ha=20. The changes in the average heat transfer values are not significant.

Inclusion of nano sized particles improves the thermal transport features due to the higher thermal conductivity of solid particle. In the presence of magnetic field, both the thermal conductivity and electrical conductivity of the fluid changes. Effects of solid nanoparticle volume fraction on the variation of average Nusselt number in the absence (Ha=0) and presence of magnetic field (Ha=50) are shown in Table 3. For Ha=0, 27.64% enhancement in the average heat transfer is attained for the nanofluid with the highest particle solid volume fraction while this value is slightly differing in the presence of magnetic field at Ha=50 which is 25.20%.

REFERENCES


Figure 1. Schematic representation of cavity with boundary conditions

a- Re=300
b- Re=300
c- Re=500
**Figure 2.** Effects of Reynolds number on the flow and thermal pattern distributions (Ha=20, γ=45°)

**Figure 3.** Average Nusselt number versus Reynolds number (Ha=20, γ=45°)
1st International Conference on Advances in Mechanical and Mechatronics Engineering
ICAMMEN 2018

Figure 4. Effects of Hartmann number on the flow and thermal pattern distributions (Re=300, γ=45°)

Figure 5. Effects of Hartmann number on the average Nusselt number for two Reynolds number values (γ=45°)
Figure 6. Contribution of different walls on the overall heat transfer coefficient for two Hartmann number values (Re=300, γ=45°)
Figure 5. Effects of magnetic inclination angle on the average Nusselt number for two Reynolds number values (Ha=20)

Table 1: Thermophysical properties of base fluid and nanoparticle

<table>
<thead>
<tr>
<th>Property</th>
<th>Water</th>
<th>CuO</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\rho$ (kg/m$^3$)</td>
<td>997.1</td>
<td>6500</td>
</tr>
<tr>
<td>$c_p$ (J / kg K)</td>
<td>4179</td>
<td>540</td>
</tr>
<tr>
<td>$k$ (W/mK)</td>
<td>0.61</td>
<td>18</td>
</tr>
<tr>
<td>$d_p$ (mm)</td>
<td>-</td>
<td>29</td>
</tr>
</tbody>
</table>

Table 2: Code validation study: Average Nusselt number in the absence and presence of magnetic field for MHD free convection

<table>
<thead>
<tr>
<th>Ha</th>
<th>Rudraiah et al. [19]</th>
<th>Current study</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>4.919</td>
<td>4.972</td>
</tr>
<tr>
<td>100</td>
<td>1.431</td>
<td>1.389</td>
</tr>
</tbody>
</table>

Table 2: Effects of solid nanoparticle volume fraction on the average Nusselt number variation in the absence and presence of magnetic field (Re=300, $\gamma=45^\circ$)

<table>
<thead>
<tr>
<th>$\varphi$</th>
<th>Ha=0</th>
<th>Ha=50</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>58.99</td>
<td>55.79</td>
</tr>
<tr>
<td>0.01</td>
<td>63.10</td>
<td>59.33</td>
</tr>
<tr>
<td>0.02</td>
<td>67.21</td>
<td>62.86</td>
</tr>
<tr>
<td>0.04</td>
<td>75.30</td>
<td>69.85</td>
</tr>
</tbody>
</table>
TEMPERATURE PREDICTION OF HOT ULTRASONIC ASSISTED MACHINING FOR Ti6Al4V ALLOY

Mehmet Alper Sofuoğlu
The University of Eskişehir Osmangazi, Department of Mechanical Engineering, Eskişehir, Turkey
asofuoglu@ogu.edu.tr

ABSTRACT

Titanium alloys are preferable in aerospace, automotive, biomedical and petrochemical industries because of their unique properties. These materials show excellent strength and remarkable resistance of corrosion at elevated temperatures. Also, low density of titanium serves the purpose of reduction of weight in the structures. In the present study, a comparison of different turning methods was investigated considering max. cutting tool/chip temperature for Ti6Al4V alloy. No previous study has investigated temperature-time graphics in ultrasonic and hot ultrasonic assisted turning. Also, no temperature-time prediction study has been performed for these processes. In the experiments; conventional turning (CT), Ultrasonic assisted turning (UAT) and hot ultrasonic assisted turning (HUAT) processes were performed to compare the results. Temperature-time graphs (1-31 seconds) were obtained during the tests using a thermal camera system. Different cutting speeds (22, 31 and 44 m/min) were used in the experiments. Ti6Al4V was used as a workpiece material. An Artificial Neural Network (ANN) method was proposed to predict the cutting temperatures for 62 m/min. Based on the results, UAT and HUAT operations yield higher cutting tool/chip temperatures compared to the conventional turning. Also, the ANN model produced successful results in terms of R² values (R²training=0.97, R²test=0.947).

Keywords: Hot machining, cutting temperature, ultrasonic assisted machining, ANN

1. INTRODUCTION

High-accuracy precision parts are in demand for different sectors (biomedical engineering, MEMS, etc.) (Qin et al., 2010; Weule et al., 2001; Liu et al., 2004; Brinksmeier et al., 2001). Also, the use of hard/brittle materials are desired because of their excellent physical and mechanical properties (Debnath et al., 2014; Hitoshi and Nakagaea, 1995; Sakamoto et al., 2016; Cui et al., 2016; Zhang et al., 2013a, b). Such utilization varies from biomedical devices and bio-sensors made of piezoelectric materials to the automotive and battery industries (Chae et al., 2006; Zemann et al., 2014). Hence, the desire for high-accuracy precision has emphasized the importance of manufacturing techniques for hard/brittle materials to get desired tolerance and excellent surface quality.

New researches were developed for different kinds of manufacturing techniques to improve the machinability. Ultrasonic Assisted Turning (UAT) is intermittent machining based on vibrations on cutting tool produced by an ultrasonic machine. Several advantages are high surface quality, lowered cutting forces and decrease in residual stresses (Amini and Teimouri, 2016; Babitsky et al., 2003; Babitsky et al., 2004; Jiao et al., 2015; Mitrofanov et al., 2003; Nath and Rahman, 2008; Zou et al., 2015). Additionally, an extension in tool life is observed in different researches (Patil et al., 2014; Sharma et al., 2008; Brehl and Dow, 2008; Farahnakian and Razfar, 2014). In the past works, several studies have been performed to investigate UAT method. With regard to surface quality, past studies suggested models for predicting the surface quality of workpiece in terms of vibration characteristics (Shamoto et al., 2008; Shamoto et al., 1994). In the other studies, the relation between vibration characteristics and workpiece surface roughness was investigated (Cheung and Lee, 2000; Kin et al., 2002). Furthermore, several studies studied the surface texture of workpiece after vibrational machining techniques (Sajjadi et al., 2016; Zhang et al., 2015; Guo and Ehmann, 2013; Zhang et al., 2016; Amini et al., 2016; Silberschmidt et al., 2014). Hot Ultrasonic Assisted Turning (HUAT) process improves hot and ultrasonic assisted machining together. A research group performed first studies on HUAT method from Loughborough University in 2011. This method decreases cutting forces (Muhammad et al., 2011; Muhammad et al., 2013). Another advantage is observing a smoother chip flow (Muhammad et al., 2012).

Predicting a temperature-time graph is a crucial task because this situation affects tool wear behavior during machining. Therefore, machining processes can be easily controlled. Especially, in hot machining operations, the temperature should be checked regularly to prevent excessive heat. In this study, time-dependent cutting tool/chip...
temperature measurements were performed for CT, UAT and HUAT operations. The maximum cutting temperature measurements are taken for 22, 31, 44 m / min. Experiments were performed for 1-31 seconds. Artificial neural networks model was proposed for the experiments conducted to predict maximum cutting tool/chip temperatures. The results of the model were predicted for a cutting speed of 62 m / min and compared with the experimental study. No previous study has investigated temperature-time graphics in ultrasonic and hot ultrasonic assisted turning. Also, no research has been found to predict the maximum cutting temperature in the time domain. This paper has four parts. The first part deals with an experimental study. The second section presents the developed ANN model parameters. The fourth section presents the findings of the experimental-ANN research.

2. EXPERIMENTAL METHOD

A Ti6Al4V alloy with 120 mm diameter and 200 mm length was used in the machining operations. The insert code is ISCAR/DCMT 3-1 SM IC806. In the experimental work, a universal lathe was used. The frequency and amplitude of the vibration was 20 kHz and 20 µm, respectively. A thermal infrared camera (Optris PI400) recorded the cutting tool/chip temperatures on the cutting tool during the operations. In HUAT operation, the workpiece was heated up to 150°C shortly before the cutting by using a band resistance heater. The feed rate and cutting speeds were 0.1 mm/rev and 22-44 m/min, respectively. Radial cutting depth was selected as 0.1 mm. The machining time is 31 seconds which was measured after the temperature came to equilibrium in the machining process.

3. ARTIFICIAL NEURAL NETWORKS (ANN)

The SPSS Clementine program was used to develop artificial neural networks. Cutting speeds of 22,31,44 m / min, 1 - 31 second machining time and conventional (1), ultrasonic (2) and hot ultrasonic (3) assisted machining techniques were used as inputs. Maximum cutting temperature is used as an output. Four different artificial neural network training algorithms have been used (Quick, Dynamic, Multiple, Exhaustive Prune). Five different repetitions were carried out by varying the initial weights. The number of training data is 279, and the number of test data is 93. The model was confirmed for the cutting speed of 62 m / min.

4. EXPERIMENTAL AND ANN RESULTS

4.1. Experimental Results

All the experimental results are given in Figures 1-3. Figure 4 presents the max. machining tool/chip temperatures at a cutting speed of 22 m / min for CT, UAT and HUAT (approximately in the middle of the process). These figures show the maximum cutting temperature of 1-31 seconds for CT, UAT, and HUAT. It was observed that the highest cutting temperature belongs to HUAT operation. Also, in UAT, the cutting temperature is higher than the conventional turning operation.

![Figure 1. The temperature distribution of CT, UAT, and HUAT at 22 m / min](image-url)
In ultrasonic assisted turning, a temperature increase was observed with respect to conventional turning as it was externally added energy. Because of the direct heat addition in the hot ultrasonic assisted process, the cutting temperature is high compared to the other machining methods. Also, when cutting speeds increase, maximum cutting temperature increases. When the cutting speed increases, more heat is displaced. However, the tool temperature increases. Lowering the cutting speed gets higher the amount of heat flowing to the workpiece, which induces the rise of the workpiece temperature. The rise in cutting speed causes an increase in the temperatures. In UAT and HUAT, some fluctuations were observed in the time domain of the temperature because of the external applied vibration. These results corroborate the findings of a great deal of the previous work in the literature (Muhammad, 2013; Kuş et al., 2015).
4.2. ANN Results

ANN results are given in Table 1 for the accuracy measure. In terms of accuracy, the best model was obtained as the Exhaustive Prune model. The determination coefficient for training was 0.97, and the determination coefficient for testing was 0.947. Maximum cutting tool/chip temperatures for three machining methods were predicted at a cutting speed of 62 m/min. The results are given in Figure 5.

<table>
<thead>
<tr>
<th>Replication</th>
<th>Quick</th>
<th>Dynamic</th>
<th>Multiple</th>
<th>Exhaustive Prune</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>97.66</td>
<td>97.243</td>
<td>97.716</td>
<td>97.727</td>
</tr>
<tr>
<td>2</td>
<td>97.51</td>
<td>97.51</td>
<td>96.948</td>
<td>97.392</td>
</tr>
<tr>
<td>3</td>
<td>97.649</td>
<td>97.509</td>
<td>97.355</td>
<td>97.397</td>
</tr>
<tr>
<td>4</td>
<td>97.384</td>
<td>97.041</td>
<td>97.056</td>
<td>97.206</td>
</tr>
<tr>
<td>5</td>
<td>97.707</td>
<td>97.597</td>
<td>97.129</td>
<td>97.614</td>
</tr>
</tbody>
</table>
Figure 5. Experimental and prediction values of the max. machining tool/chip temperatures for CT, UAT, and HUAT at 62 m/min cutting speed.

5. CONCLUSIONS

In this study, different turning operations were performed to compare the results of the maximum cutting temperatures. In ultrasonic assisted turning, a temperature increase was observed with respect to CT. Also, in HUAT operation, the cutting temperature is high compared to the other machining methods. Also, when cutting speeds increase, maximum cutting tool/chip temperature increases. In UAT-HUAT operations, some fluctuations were observed in the time domain of the temperature. The ANN model produced successful results. The developed ANN model is simple and easy to use when Finite Element Models are taken into account. In future studies, different data mining algorithms can be used to compare ANN results.

REFERENCES


THE EFFECT OF PARTICLE TYPE AND DISTRIBUTION ON BENDING ANALYSIS OF GLASS PARTICLE REINFORCED COMPOSITE BEAMS

S. Özmen Eruslu

1Namık Kemal University Çorlu Engineering Faculty Mechanical Engineering Çorlu/Turkey
oeruslu@nku.edu.tr

ABSTRACT
In this work, the effect of particle type and distribution on bending behavior of glass particle reinforced epoxy composite beams are studied analytically. Euler Bernoulli and Timoshenko Beam deflection results are compared with numerical deflections obtained by FEM Analysis. Bending analysis of particle reinforced composites are studied for graded distribution bottom to top. The composite elastic modulus variation with thickness is found according to microct based modelling for different particle type and graded distribution. The effects of graded distribution of particles are considered by functionally graded beam equation formulations. Local stress distribution along thickness is found by using finite element analysis. The relation between the stress results and particle type is discussed at macro scale.

Keywords: Particle reinforced composites, functionally graded material, Finite element analysis, Bending deflection, Microct-based modelling.

1. INTRODUCTION
Particle reinforced composites are popular in various applications with the resistance to impact to wearing. Functionally graded forms are extensively used in particle reinforced composites with the development of spatial distribution of fibers. The advantage of functionally graded material is to smooth transition of matrix and fiber dominated regions. The control of the property transition along the thickness is very important [Udupa continued,2014]. Functionally graded materials (FGM) are used at many kind of industry with the development of production methods. Powder metallurgy and centrifugal casting techniques are used for metal matrix FGM composites. Jang and coworkers are used glass and carbon particle reinforced thermoplastic FGM composites with controlling the change of distribution particles (Jang and Lee,1998). The distribution of properties along thickness are obtained by using fabric reinforcements at many studies in the literature (Huang continued,2002), (Misra continued,2011). Particle reinforced FGM thermoplastic composite production is increased with development of newly developed production methods like selective laser sintering (Chung and Das,2006). The prediction of material properties of these types of composite is very important. Homogenization methods like Mori Tanaka are performed for material characterization of FGMs (Petterman continued,2010). Homogenization based micromechanical approaches are increasingly used with the developments of computational methods like finite element based unit cell models or representative of volume elements (Kurukuri,2005). Microct equipped three dimensional study of composite materials is alternative method with the implementation of FEA analysis. Microct imaging technique is applied for both short fiber and continuous fiber composites in the literature (Shen continued,2004),(Naouar continued,2015). Recently Homogenization based modelling of particle reinforced composites is obtained with the development of x-ray Micro-CT to create more effective characterization of spatial distributed particles (Cinar and Guven, 2018).

Mathematical modelling of FGM beams have studied since early 2000s. Euler Bernoulli beam theory based elasticity solution of FGM beams are solved by Sankar (Sankar,2001). Ding and coworkers are investigated the bending solutions of anisotropic FGM beams with arbitrary functions of thickness by elasticity solution (Ding continued,2007). The flexural bending of simply supported FGM beams are studied by using high order beam theories with varying gradation laws (Kadoli R. continued, 2008), (Oumrane,2009).

In this study MicroCt based finite element modelling is used to obtain distribution of particles along thickness. The graded distribution of particles are modelled by using functionally graded beam formulations. The flexural behavior of beams are studied analytically and numerically.
2. Beam Theory

In this section bending formulations of simply supported beam with concentrated load seen in figure 1 is given as follows.

\[
\begin{bmatrix}
N_x \\
M_x
\end{bmatrix} = \begin{bmatrix}
A_{11} & B_{11} \\
B_{11} & D_{11}
\end{bmatrix} \begin{bmatrix}
\varepsilon_0 \\
\kappa
\end{bmatrix}
\]

Here strain and curvature change is given

\[
\varepsilon_0 = \frac{du}{dx}, \quad \kappa = -\frac{d^2w}{dx^2}
\]

Beam moment may be defined by following way

\[
M_x = B_{11} u_x - D_{11} w_{xx}
\]

Where here beam moment is also defined as

\[
M_x = F_0 x / 2
\]

External load variation along x direction is zero so it is defined as

\[
\frac{dN}{dx} = 0, \quad A_{11} u_{xx} - B_{11} w_{xxx} = 0
\]

If we add this term into eq.3 we obtain the following equation

\[
w_{xx} \left( \frac{B_{11}}{A_{11}} - D_{11} \right) = \frac{F_0 x}{2}
\]

Here \( C = \left( \frac{B_{11}}{A_{11}} - D_{11} \right) \) then

\[
w_{xx} = \frac{F_0 x}{C} / 2
\]

After integrating we obtain following equations

\[
w_x = \frac{1}{C} \left( F_0 x^2 / 4 + c_1 \right)
\]

\[
w = \frac{1}{C} \left( F_0 x^3 / 12 + C_1 x + C_2 \right)
\]

\[
u = B_{11} \left( \frac{F_0 x^2}{C} / 4 + A_1 \right)
\]
The integration constants are obtained by boundary conditions seen in figure 1.

\[ x=0 \ u(0)=0, \ w(0)=0, \ x=L \ w(L)=0 \]

\[ A_1=0, \ C_2=0 \]

The deflection is symmetry at the point load and curvature is zero at that point.

\[ w_x \left( \frac{L}{2} \right) = 0 \]  
\[ C_1 = \frac{F_0 a^2}{16} \]  
\[ w(x) = \frac{1}{C} \left( \frac{F_0 a^2}{48} - 4 \left( \frac{x}{a} \right)^3 \right) \]  
\[ u = \frac{B_{11}}{A_{11}} \left( \frac{1}{C} \frac{F_0 x^2}{4} \right) \]

Timoshenko Beam Theory with shear effects are given at following form for concentrated load at the middle of the simply supported beam (Reddy, 2004)

\[ w(x) = \left( \frac{F_0 a^3}{48 E_I} \left( 3 \left( \frac{x}{a} \right)^2 - 4 \left( \frac{x}{a} \right)^3 \right) \right) - \frac{F_0 a}{2 K G_{xz} b h} \left( \frac{x}{a} \right) \]

Where rigidity \( E_I = D_{11} \) for symmetric layup \( E_I = C \) for antisymmetric layup seen in eq(7).

Here \( K \) is shear correction factor generally \( K = 5/6 \), \( G_{xz} \) is shear modulus of beam.

Numerical example is given for maximum deflection of isotropic epoxy beam in table 1 for following values.

\[ P=100N \ L=80mm \ b=5mm \ h=5mm \ E=1643 \ MPa \]

<table>
<thead>
<tr>
<th>Table 1</th>
<th>Deflection values of simply supported epoxy beam under midspan concentrated load</th>
</tr>
</thead>
<tbody>
<tr>
<td>Euler Bernoulli</td>
<td>Timoshenko Beam</td>
</tr>
<tr>
<td>Deflection (mm)</td>
<td>Deflection (mm)</td>
</tr>
</tbody>
</table>

3. Effective Material Properties of Composites

MicroCt based finite element models (FEM) are used for material properties characterization of composites. Volume fraction variation along thickness of composite beams are found from FEM. The rule of mixture homogenization procedure is used to find effective properties of beams (Karamanlı, 2016).

Elastic modulus and shear modulus along z direction is given as follows

\[ E(z)=E_1 V_1(z)+E_2 V_2(z) \]  
\[ G(z)=G_1 V_1(z)+G_2 V_2(z) \]  
\[ V_1(z)+V_2(z)=1 \]

Here \( E_1, E_2, G_1, G_2 \) are constituents of material

\( V_1(z), V_2(z) \) is denoted the volume fractions of constituents along thickness direction.

According to Euler Bernoulli Theory the extensional, coupling and bending rigidities for general form may be defined at the following form (Benatta continued, 2008)

\[ A_{11} = \int_{-\frac{h}{2}}^{\frac{h}{2}} E(z) \ dz \]  
\[ B_{11} = \int_{-\frac{h}{2}}^{\frac{h}{2}} E(z) z \ dz \]  
\[ D_{11} = \int_{-\frac{h}{2}}^{\frac{h}{2}} E(z) z^2 \ dz \]
4. Finite Element Model in Numerical Analysis

Functionally graded beam is modelled by layer based shell elements at finite element models. Shell elements are given good results for relatively thin walled composite plates and beams (Eruslu and Aydogdu, 2008). The finite element model and boundary conditions are given in figure 2.

Simply supported boundary conditions \((x=0, u=0, v=0, w=0, x=L, v=0, w=0)\) are used in the analysis.

![Finite element model and boundary conditions](image1)

Mesh convergence analysis is given at following figure, 160 elements are used at finite element model according to this study.

![Mesh Convergence Analysis](image2)
5. Numerical Results

In our numerical study variation of volume fraction along thickness is taken from our previous study seen in figure 4. In that study microct image file is exported to bitmap image files then converted to voxel data. These data is implemented to FEM to obtain distribution of particles along thickness. The glass particles were dispersed in epoxy matrix with the %5 mass fraction and free to settle down at preparation of composites. Elastic modulus results are obtained by rule of mixture formula given at eq.17 seen in figure 5.

![Variation of volume fraction along thickness](image1)

![Elastic modulus variation with thickness](image2)
It is seen that larger irregular particles are collapsed at the bottom side of composites. Particle size is effective for smooth transition of particle distribution along thickness. Flake and rod type particles are more homogenous distribution than the other particles.

In numerical and analytical analysis the elastic modulus of epoxy is $E_{ep}= 1.643$ GPa, the elastic modulus of glass particles are $E_{gl}= 70$ GPa and the Poisson ratio is 0.3. The deflection values are found for concentrated load at the midspan of the beam. The results are given for $P=100N$ load and ($L=80$mm $b=5$mm $h=5$mm) beam dimensions.

**Table 2** Deflection values of glass particle reinforced epoxy composites

<table>
<thead>
<tr>
<th>Glass Particle Types</th>
<th>Euler Bernoulli Deflection Results $w$ (mm)</th>
<th>Timoshenko Beam Deflection Results $w$ (mm)</th>
<th>Timoshenko Beam Transverse Deflection Results $u$ (mm)</th>
<th>(Ansys Shell Model) Deflection Results $w$ (mm)</th>
<th>(Ansys Shell Model) Transverse Deflection Results $u$ (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Irregular Shaped (75 µm &lt; size &lt; 150 µm)</td>
<td>3.123</td>
<td>3.145</td>
<td>0.1599</td>
<td>3.150</td>
<td>0.159</td>
</tr>
<tr>
<td>Irregular Shaped (50 µm &lt; size &lt; 75 µm)</td>
<td>1.891</td>
<td>1.917</td>
<td>0.0616</td>
<td>1.921</td>
<td>0.0615</td>
</tr>
<tr>
<td>Irregular Shaped (50 µm &lt; size)</td>
<td>3.9036</td>
<td>3.9578</td>
<td>0.0532</td>
<td>3.964</td>
<td>0.053</td>
</tr>
<tr>
<td>Spherical Particles</td>
<td>1.9224</td>
<td>1.9502</td>
<td>0.0451</td>
<td>1.953</td>
<td>0.0451</td>
</tr>
<tr>
<td>Flake Particles</td>
<td>1.4150</td>
<td>1.4343</td>
<td>0.0285</td>
<td>1.436</td>
<td>0.0285</td>
</tr>
<tr>
<td>Fiber Rod Particles</td>
<td>1.3104</td>
<td>1.3281</td>
<td>0.0246</td>
<td>1.330</td>
<td>0.0246</td>
</tr>
</tbody>
</table>

**Fig. 6** Von Mises Stress Variation along the thickness for

Irregular glass particle reinforced epoxy composites
Deflection results indicate that particle type and concentration affected overall composite stiffness. Local stresses are increasing at the bottom side of composite both for regular and irregular particles.

6. Results

Deflection results indicate that larger glass particles are concentrated at bottom and overall stiffness of composite is decreased. Particle type is seen effective tool for distribution along thickness. Flake type and rod type particles are homogeneously distributed along thickness at matrix according to the other particle types. Particle concentration at bottom sides is the reason of the local stresses. Local stresses are predicted with finite element method whereas finite element results with Shell type elements are found in good agreement with Timoshenko Beam results. Local stresses must be eliminated to avoid composite failure for these types of products. In the future works particle type concentration, local stress and load transfer mechanism relations may be studied experimentally and numerically for optimization of these types of production methods.

REFERENCES


EXPERIMENTAL STUDY OF HEAT TRANSFER DURING FLOW BOILING THROUGH A PLATE HEAT EXCHANGER SATURATED WITH METALLIC FOAM

Ahmed Kouidri, Brahim Madani

1 BP 32 Bab Ezzouar, 16111 – Algies, Algeria.
2 BP 32 Bab Ezzouar, 16111 – Algies, Algeria.

a akouidri@usthb.dz, b madani@usthb.dz,

ABSTRACT

The present communication focuses on the study of heat transfer during flow boiling through a plate heat exchanger. The heat exchanger is saturated with various metallic foam samples: Copper, NiFeAlCr and Inconel; with 20 PPI (Pore per Inch) and 93% of porosity and the working fluid is the n-pentane. The heat flux varying from 3 to 7.12 W/cm² for two velocities: 4 and 6 m/s. Two ranges of thermocouples are implanted in the channel: in the edge of the wall and in the center of channel. The blocks of metallic foam are implanted to enhance the heat transfer through the heat transfer coefficient and the boiling curve. The comparison of the present results to those given by the smooth channel, published at reference (Kouidri, Madani et al. 2015), permits to quantify the influence of metallic foam samples. The copper metallic foam sample presented the best heat transfer performance.

Keywords: Metallic foam, Heat transfer, Plate heat exchanger, Flow boiling.

1. INTRODUCTION

The bibliographic analysis shows that the copper metallic foam is more and more used due to their high thermal conductivity. (Kim 2007) reported that the flow capability is better with small pore diameter and the heat transfer coefficient can reaches 10 W/m²K in the case of water and 2.85 W/m²K for FC-72. (Lu 2008) found that the diminution in pore diameter increase both the heat transfer coefficient and the flow resistance. (Mancin, Diani et al. 2014) compared two fluids R 134a and 1234ze during flow boiling in refrigeration system. They report that the R134a present a better heat transfer coefficient compared to the other one; especially in low velocities. (Madani, Tadrist et al. 2013) analyzed the heat transfer coefficient during flow boiling in copper metallic foam. They compared their results to those predicted by Shah (Shah 1982) and (Gungor and Winterton 1986) correlations developed for smooth channel. They reported that the copper metallic foam enhance the heat transfer coefficient with factor varying from 2 to 4.

The present study shows a thermal study of a plate heat exchanger performance, which is filled with various metallic foam samples (Copper, NiFeAlCr and Inconel). We note that the NiFeAlCr and Inconel samples have never been studied by other authors in thermal engineering.

2. EXPERIMENTAL FACILITIES

We expose here, the essential of the experimental facilities, more detail are presented in references (Kouidri, Madani et al. 2015; Kouidri and Madani 2017). The n-pentane is used as working fluid; it is pumped from the tank to the test section. The fluid is evaporated in the test channel and separated in the separator; the flow rate of the separated liquid is measured using balance at the outlet of the channel. However the condensed steam returns in the tank, as showed in Fig 1 (a).

The test section built in-situ, Fig. 2 (b) is composed of two heated plates made from bronze. These plates are wrapped in PTFE (PolyTetrafluoroEthylene) sheets, thus ensuring both the mechanical retention and the thermal insulation of the channel. The test channel has following dimensions: 0.05 m (length), 0.005 m (width), 0.025 m (depth) [m³].

Internally, this channel is filled with various metallic foam samples (Copper, NiFeAlCr and Inconel), characterized in the reference (Kouidri and Madani 2016). The assembly is externally wrapped in PTFE (PolyTetrafluoroethylene) sheets which ensure its mechanical retention and thermal insulation.

Two heater cartridges located vertically in each bronze plate to heat the assembly. The applied heating power varies between 3.04 and 7 W/cm². The channel tightness is ensured using plate Viton gaskets.
K-type thermocouples have been used to measure the local temperature. Their diameter is equal to 0.5 mm for not disturb the flow. Locations of thermocouples are given in Fig. 1 (b). They are implanted in both wall and inside the channel along two vertical axes. Note that the fluid measurements are taken in the center of the channel.

The flow rate is measured using two turbine flow-meters; one for the small flow rates (0–1 l/min), while, the second is destined to measure the high flow rates (0–5 l/min). Furthermore, the set-up is equipped with a high performance acquisition chain to improve the accuracy of the measurements.

![Figure 1. Experimental facilities: (a) Hydrodynamic loop, (b) Test section. Photos reproduced from reference (Kouidri, Madani et al. 2015).](image)

2. METHODS

The experimental average heat transfer coefficient for the boiling regime is calculated using Eq. (1):

\[ h_a = \frac{q_b}{(T_w - T_{sat})} \] (1)
Where \( h_a \) is the average heat transfer coefficient, \( Q_b \) is the boiling heat flux (W/m\(^2\)), \( T_w \) and \( T_{sat} \) are the mean wall and the saturated fluid temperatures, respectively, in the boiling zone. The \( Q_b \) is defined by Eq. (2):

\[
Q_b = \frac{q - Q_{loss} - m \cdot C_p (T_{sat} - T_{in})}{S}
\]

Where \( q = UI \), \( m \) is the flow rate (kg/s), \( C_p \) is the specific heat (W/kg°C), \( T_{sat} \) is the saturation temperature, \( T_{in} \) is the inlet liquid temperature and \( S \) is the exchange surface in the boiling zone. The \( Q_{loss} \) is the heat flux loss through the PTFE sheets; it is calculated using Eq. (3):

\[
Q_{loss} = \left( \frac{k_{PTFE}}{\varepsilon_{PTFE}} \right) S_{PTFE} (T_w - T_{am})
\]

Where \( k_{PTFE} \) is the thermal conductivity of the insulation (W/mK), \( \varepsilon_{PTFE} \) is the insulation thickness (m), \( S_{PTFE} \) is the contact surface between the channel and the insulation (m\(^2\)), \( T_w \) is the wall temperature (°C) and \( T_{am} \) is the ambient temperature (°C). \( Q_{loss} \) is found between 1.5–3% of the heating power \( UI \).

The outlet vapor quality is calculated of basis of Eq. (4):

\[
x_{out} = \frac{\dot{m}_t}{\dot{m}_v}
\]

Where \( \dot{m}_t \) is the total flow rate measured at the inlet of channel and \( \dot{m}_v \) is the vapor flow rate calculated using Eq. (5):

\[
\dot{m}_v = \frac{q - Q_{loss} - m \cdot C_p (T_{sat} - T_{in})}{h_{lv}}
\]

Where \( h_{lv} \) represent the latent heat of vaporization.

The heat transfer coefficient uncertainties are calculated on the basis of the (Kline and McClintock 1953) method, using Eq. (6):

\[
\frac{\Delta h}{h} = \sqrt{\left( \frac{\Delta Q}{Q} \right)^2 + \left( \frac{\Delta V}{V} \right)^2 + \left( \frac{\Delta T_{en}}{T_{en}} \right)^2 + \left( \frac{\Delta T_p}{T_p - T_{sat}} \right)^2 + \left( \frac{\Delta T_{sat}}{T_p - T_{sat}} \right)^2 + \left( \frac{\Delta S}{S} \right)^2}
\]

### 3. RESULTS AND FINDINGS

#### 3.1. Fluid Temperature Signals

Figure 2 shows the fluid temperature signals during the convective boiling in fourth channels; the smooth channel and those saturated with metallic foam. All signals were recorded at stationary regime with a velocity equal to 0.04 m/s and heat flux equal to 30 kW/m\(^2\) at 0.035 m position.

The mean arithmetic value, standard deviation and confidence interval for each signal are presented in the Table 1. It is obvious that the Inconel samples have the highest standard deviation due to their important surface roughness.

<table>
<thead>
<tr>
<th></th>
<th>Copper</th>
<th>NiFeAlCr</th>
<th>Inconel</th>
<th>Smooth channel</th>
</tr>
</thead>
<tbody>
<tr>
<td>Velocity [m/s]</td>
<td></td>
<td>0.04</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Arithmetic mean value [°C]</td>
<td>43.86</td>
<td>37.23</td>
<td>44.07</td>
<td>39.48</td>
</tr>
<tr>
<td>Standard deviation [°C]</td>
<td>0.020</td>
<td>0.010</td>
<td>0.084</td>
<td>0.051</td>
</tr>
<tr>
<td>95% confidence interval</td>
<td>0.009</td>
<td>0.006</td>
<td>0.018</td>
<td>0.034</td>
</tr>
</tbody>
</table>
Figure 2. Fluid temperature signals at $V = 0.04 \text{ m/s}$ and $z = 0.035 \text{ m}$, (a) Copper, (b) NiFeAlCr, (c) Inconel, (d) Smooth channel.

3.2. Temperature Distribution

Figure 3 present the evolution of average wall temperature as function of the inlet velocity for the different metallic foam samples as well as for four heating powers. For both copper and Inconel samples (Fig. 3-a), the gradient $\frac{dT_{w}}{dV}$ is negative, which means that as the input speed increases the exchange is better and the wall is well cooled.

Whereas for the NiFeAlCr sample, there is an inversion point at $V = 0.04 \text{ m/s}$, beyond this value, the gradient $\frac{dT_{w}}{dV}$ becomes positive, and this for all used heat fluxes, which shows an overheating of the wall due to the poor heat exchange with the fluid.

The influence of the heat flux on the gradient $\frac{dT_{w}}{dV}$ is shown in Figure 3 (b) for the Inconel sample as an example. It is obvious that this gradient increases as the heat flux increase.

Figure 3. Average wall temperature as function of velocity, (a) For different metallic foam samples, (b) For different heat fluxes.
3.3. Heat Transfer Coefficient

The average heat transfer coefficient is calculated using Eq. 1. Figure 4 shows the evolution of the average heat transfer coefficient, in different cases, as a function of the density of heat flux \( Q_b \) and outlet vapor quality for \( V = 0.06 \text{ m/s} \).

The copper metallic foams has significantly increase the heat transfer coefficient compared to the smooth channel due to their important specific area and high thermal conductivity. The heat transfer intensification factor increase by decreasing the velocity; where the latter is equal to 30% and 70% for the velocities 0.06 and 0.04 m/s (Kouidri and Madani 2017), respectively.

Concerning the two other samples; NiFeAlCr and Inconel, the heat transfer coefficient is poor, where it is less than or equal to that of the smooth channel in the case of Inconel foam and much lower in the case of NiFeAlCr foam. This is mainly due to their very low effective thermal conductivity.

The heat transfer coefficient versus vapor quality is presented in Fig. 4 (b). It is shown that the insertion of metal foam has increased the steam production, where the metallic foam plays a very important role; in particular on the good distribution of the fluid in the channel and the creation of a nucleation sites which participate in the heat transfer and steam production.

Some researchers (Kim 2007; Lu 2008; Madani, Tadrist et al. 2013) found that there is an inverse point, beyond that, the heat transfer coefficient decreases with the increase of the vapor quality. This point depends mainly on the conditions of experimentation. In the present work, this point is not reached. The average uncertainties on the heat transfer coefficient is equal to 3%.

Figure 4. Average heat transfer coefficient for \( V=0.06 \text{ m/s} \), (a) as function of heat flux, (b) as function of outlet vapor quality

3.3. Boiling Curve

The boiling curves for the different metallic foam samples are shown in Fig. 5. These follow the classical curve established by Nukiyama (Nukiyama 1966). Qualitatively, it corresponds to the part before the critical heat flux, which is the maximum heat flux absorbed by the fluid during boiling.

The same figure shows that, all curves given by metallic foam samples tend more towards the critical heat flux; Compared to the smooth channel. We note that the copper metallic foam present the minimum wall superheat.
4. CONCLUSIONS

The thermal performance of a rectangular heat exchanger under boiling conditions was discussed with the representation of local results and heat transfer coefficient. The copper metallic foam presents a better thermal performance due to its high effective thermal conductivity. Where, the coefficient of heat transfer has been intensified with a factor equal to 30% compared to a smooth channel. The two other samples (NiFeAlCr and Inconel) showed a lower heat transfer coefficient than the smooth channel because of their contact resistance created between the wall and the metallic foam sample; from that, the use of these two materials in heat exchangers is discouraged.

ACKNOWLEDGEMENTS

This work was supported by the General Directorate for Scientific Research and Technological Development of Algeria, for which the authors are very thankful. We thank also Mr Andreas Tillmann from Alantum Europe, for the delivery of the metallic foam samples, Mr Djamel Miroud, for the characterization of metallic foam samples using the scanning electron microscopy in Laboratoire des Sciences et Génie des Matériaux, and Mr Allali Mohamed for his assistance during experimentations.

REFERENCES


FARKLI YAYICILIĞA SAHİP İKİ, ÜÇ VE DÖRT CAMLI PENCRELERE
BAĞLI YAKIT TÜKETİMİ

Okan KON

Balıkesir Üniversitesi Mühendislik Fakültesi Makine Mühendisliği Bölümü
Çağış Kampüsü
10145 Balıkesir/Türkiye
e-mail: okan@balikesir.edu.tr

ÖZET

Binaların pencerele rinden önemli ölçüde enerji kayıpları meydana gelmektedir. Bunu için pencelerin camlarına çeşitli yayıcılığı sahip kaplama yapılması, pencelerden enerji kayıplarının önlenmesinde önemli payı olacaktır. Çalışmada, 0.0-1.0 arasında yayıcılıklı iki, üç ve dört camı sahip pencerele bağlı ısıtma ve soğutma dönemi için yakıt tüketimleri hesaplanmıştır. Pencerele bağlı yakıt tüketimi bulunurken genel ısı transfer katsayısı değeri hesaplanmıştır. Genel ısı transferi değeri için iletim, taşıım ve ışınım ısı transferi değerleri bulunmaktadır. İki, üç ve dört kat camlı penceler için ara dolgusu olarak hava ve argon kullanılmıştır. Binaların yakıt tüketimi derece-gün metoduna göre yapılmıştır. Derece-gün değerleri günlük maksimum ve minimum sıcaklıklar göz önünde alınarak bulunmuştur. İstifa dönemi için yakıt olarak kömür, doğal gaz ve elektrik soğutma dönemi için elektrik düşünülmüştür. Yakıt tüketimi bulunurken Türk yalıtım standardı TS 825’i göre beş iklim bölgesi için İzmir, Balıkesir, Eskişehir, Yozgat, Kars illeri seçilmiştir.

Anahtar Kelimeler: Pencerelerden ısı transferi, Derece-gün, Yakıt tüketimi, TS 825

1. GİRİŞ

Çalışmada, amacı 0.0-1.0 arasında yayıcılıklı iki, üç, dört kat camı sahip pencerele bağlı ısıtma ve soğutma dönemi için yakıt tüketimleri hesaplanmıştır. İki, üç ve dört kat camlı penceler için ara dolgusu olarak hava ve argon kullanılmıştır. Binaların yakıt tüketimi derece-gün metoduna göre yapılmıştır. Derece-gün değerleri 21 yıllık günlük maksimum ve minimum sıcaklıklar göz önüne alınarak bulunmuştur.

2. METOD

2.1. Camın genel ısı transfer katsayısı değeri ve yakıt tüketimi hesabı

Çok camlı pencere için genel ısı transfer katsayısı [1,2],

\[
U = \frac{1}{h_p k_{	ext{cam}}} \left( \frac{U_{2-3,\text{con}}}{U_{2-3,\text{rad}}} \right) \frac{1}{h_p k_{	ext{cam}}} \frac{1}{h_p k_{	ext{cam}}}
\]

(1)

dir. Burada, dış camların iç yüzeyleri arasındaki \(U_{2-3,\text{con}}\) iletim ve \(U_{2-3,\text{rad}}\) radyasyon ısı geçiş katsayısı olarak değerlendirilir.

\[
U_{2-3,\text{con}} = \frac{1}{h \kappa_{\text{hava}}} (n-2) \left( \frac{1}{h \kappa_{\text{hava}}} \right)
\]

(2)

\[
U_{2-3,\text{rad}} = \frac{1}{\kappa_{\text{hava}}} \left( \frac{1}{h \kappa_{\text{hava}}} \right)
\]

(3)

\[
\phi(T_2 - T_3) = \frac{2(1+\theta)}{(\theta-2)} \frac{2n(1+\theta)}{\theta} \left( \frac{1}{h_p k_{\text{cam}}} \right)
\]

(3)
dir. Burada, n cam sayısı, L havaya argon tabakası kalınığı, A yüzey alanı, ε yayıcılık, σ Stefan-Boltzman sabiti, hₐ (=34 W/m².K) ve hₐ (=8.29 W/m².K) iç ve dış film islı dirençleri, kₐ camın ısı iletim katsayısı (kₐ=0.92 W/m.K), (1- ε)/ε(A) ve 1/Fᵢₙ yüzey ile alan radyasyon dirençleridir. Fij görünüm faktörü 1 alınmıştır. Cam kalınığı c, 4 mm kabul edilmiştir. Cam kalınığı değeri literatürde en çok kullanılan değer alınmıştır. Ortalama pencere içindeki ikinci ve üçüncü cam tabakası arasındaki sıcaklık 285 K (T₂+T₃)/2 iken fark ise (T₂-T₃) 10 K alınmıştır [1,2].

Çok camlı pencere için yıllık ısıtma yakıt tüketimi [3,4]

\[ M = \frac{86.400U_{HDD}}{\eta_HU_H} \]  

dir. Burada, HDD ısıtma derece-gün değeri, U (W/m².K) pencerenin ısı transfer katsayısı, Hₐ alt ısı değer ve ηₙₘ ısıtma sistemi verimidir.

Çok camlı pencere için yıllık Soğutma yakıt tüketimi [3,4], 

\[ M_{fc} = \frac{86400U_{CDD}}{COP} \]  

Burada, CDD soğutma derece-gün değeri, U (W/m².K) pencerenin ısı transfer katsayısı ve COP soğutma performansı değeridir.

2.2 Hesaplamalarda kullanılan değerler

Tablo 1’de beş farklı iklim bölgesi için seçilen İzmir, Balıkesir, Eskişehir, Yozgat, Kars ili için ısıtma ve soğutma derece-gün değerleri verilmiştir. Tablo 2’de hesaplamalarda kullanılan kömür, doğal gaz ve elektrik yakıtları ve özellikleri gösterilmiştir. Soğutma performansı katsayısı değeri (COP) 2.5 alınmıştır [4]

<table>
<thead>
<tr>
<th>Şehir</th>
<th>Bölge</th>
<th>İstıma Derece-Gün (HDD)</th>
<th>Soğutma Derece-Gün (CDD)</th>
<th>Enlem</th>
<th>Boylam</th>
<th>Yüseklik</th>
</tr>
</thead>
<tbody>
<tr>
<td>İzmir</td>
<td>1. Bölge</td>
<td>1480</td>
<td>617</td>
<td>38.43</td>
<td>27.17</td>
<td>28.55</td>
</tr>
<tr>
<td>Balıkesir</td>
<td>2. Bölge</td>
<td>2312</td>
<td>369</td>
<td>39.65</td>
<td>27.87</td>
<td>147.00</td>
</tr>
<tr>
<td>Eskişehir</td>
<td>3. Bölge</td>
<td>3239</td>
<td>201</td>
<td>39.77</td>
<td>30.55</td>
<td>801.00</td>
</tr>
<tr>
<td>Yozgat</td>
<td>4. Bölge</td>
<td>3550</td>
<td>122</td>
<td>39.80</td>
<td>34.80</td>
<td>1298.43</td>
</tr>
<tr>
<td>Kars</td>
<td>5. Bölge</td>
<td>4770</td>
<td>96</td>
<td>40.62</td>
<td>43.10</td>
<td>1775.00</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Yakıt</th>
<th>Alt Isıl Değer (Hₐ)</th>
<th>Verim (ηₙₘ)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Kömür</td>
<td>29.295 10⁶ (J/kg)</td>
<td>0.65</td>
</tr>
<tr>
<td>Doğal Gaz</td>
<td>34.526 10⁶ (J/m²)</td>
<td>0.93</td>
</tr>
<tr>
<td>Eletrik</td>
<td>3.599 10⁶ (J/kWh)</td>
<td>0.99</td>
</tr>
</tbody>
</table>

2. BULGULAR

Şekil 1’de a) İki camlı b) Üç camlı c) Dört camlı, ara boşluk dolgusu hava olan pencelerin yayıcılığı bağlı genel ısı transfer değerinin değişimi grafiği d) İki camlı e) Üç camlı f) Dört camlı, ara boşluk dolgusu argon olan pencelerin yayıcılığı bağlı genel ısı transfer değerinin değişimi grafiği verilmiştir. Şekil 2’de birinci bölgedeki İzmir ilinin a) İki camlı b) Üç camlı c) Dört camlı, ara boşluk dolgusu hava olan pencelerin yayıcılığı bağlı kömür tüketiminin değişimi grafiği d) İki camlı e) Üç camlı f) Dört camlı, ara boşluk dolgusu argon olan pencelerin yayıcılığı bağlı kömür tüketiminin değişimi grafiği gösterilmiştir. Şekil 3’de ikinci bölgedeki Balıkesir ilinin a) İki camlı b) Üç camlı c) Dört camlı, ara boşluk dolgusu hava olan pencelerin yayıcılığı bağlı doğal gaz tüketiminin değişimi grafiği d) İki camlı e) Üç camlı f) Dört camlı, ara boşluk dolgusu argon olan pencelerin yayıcılığı bağlı doğal gaz tüketiminin değişimi grafiği verilmiştir. Şekil 4’de üçüncü bölgedeki Eskişehir ilinin a) İki camlı b) Üç camlı c) Dört camlı, ara boşluk...
dolgusu hava olan pencerelerin yayıcılığa bağlı ısıtma amaçlı elektrik tüketiminin değişimi grafiği
d) İki camlı e) Üç camlı f) Dört camlı, ara boşluk dolgusu argon olan pencerelerin yayıcılığa bağlı ısıtma amaçlı elektrik tüketiminin değişimi grafiği gösterilmiştir. Şekil 5’de Beşinci bölgedeki Kars ilinin a) İki camlı b) Üç camlı c) Dört camlı, ara boşluk dolgusu hava olan pencerelerin yayıcılığa bağlı soğutma amaçlı elektrik tüketiminin değişimi grafiği d) İki camlı e) Üç camlı f) Dört camlı, ara boşluk dolgusu argon olan pencerelerin yayıcılığa bağlı soğutma amaçlı elektrik tüketiminin değişimi grafiği verilmiştir.

Şekil 1. a) İki camlı b) Üç camlı c) Dört camlı, ara boşluk dolgusu hava olan pencerelerin yayıcılığa bağlı genel ısı transfer değerinin değişimi grafiği
d) İki camlı e) Üç camlı f) Dört camlı, ara boşluk dolgusu argon olan pencerelerin yayıcılığa bağlı genel ısı transfer değerinin değişimi grafiği
Şekil 2. Birinci bölgedeki İzmir ilinin a) İki camlı b) Üç camlı c) Dört camlı, ara boşluk dolgusu hava olan pencelerin yayıcılığı bağlı kömür tüketiminin değişimi grafiği d) İki camlı e) Üç camlı f) Dört camlı, ara boşluk dolgusu argon olan pencelerin yayıcılığı bağlı kömür tüketiminin değişimi grafiği
Şekil 3. İkinci bölgedeki Balıkesir ilinin a) İki camlı b) Üç camlı c) Dört camlı, ara boşluk dolgusu hava olan pencerelerin yayıcılığa bağlı doğal gaz tüketiminin değişimi grafiği d) İki camlı e) Üç camlı f) Dört camlı, ara boşluk dolgusu argon olan pencerelerin yayıcılığa bağlı doğal gaz tüketiminin değişimi grafiği
Şekil 4. Üçüncü bölgedeki Eskişehir ilinin a) İki camlı b) Üç camlı c) Dört camlı, ara boşluk dolgusu hava olan pencerelerin yayıcılığa bağlı istıma amaçlı elektrik tüketiminin değişimi grafiği d) İki camlı e) Üç camlı f) Dört camlı, ara boşluk dolgusu argon olan pencerelerin yayıcılığa bağlı istıma amaçlı elektrik tüketiminin değişimi grafiği
İki camlı, yayıcılığı 0.0-1.0 arasında, 6,9,12,15,18 mm ara boşluklu, ara boşluk dolgusu argon olan iki, üç, dört camlı pencereye bağlı İzmir ili için 2.955-21.757 kg/m², Balıkesir ili için 4.616-33.988 kg/m², Eskişehir için 6.467-47.618 kg/m², Yozgat ili için 7.088-52.190 kg/m² ve Kars ili için 9.523-70.123 kg/m² kömür tüketimi bulunmuştur.

Yayıcılığı 0.0-1.0 arasında, 6,9,12,15,18 mm ara boşluklu, ara boşluk dolgusu hava olan iki, üç, dört camlı pencereye bağlı İzmir ili için 4.163-22.831 kg/m², Balıkesir ili için 6.504-35.666 kg/m², Eskişehir için 9.112-49.970 kg/m², Yozgat ili için 9.987-73.586 kg/m² kömür tüketimi bulunmuştur. Yayıcılığı 0.0-1.0 arasında, 6,9,12,15,18 mm ara boşluklu, ara boşluk dolgusu argon olan iki, üç, dört camlı pencereye bağlı İzmir ili için 2.955-21.757 kg/m², Balıkesir ili için 4.616-33.988 kg/m², Eskişehir için 6.467-47.618 kg/m², Yozgat ili için 7.088-52.190 kg/m² ve Kars ili için 9.523-70.123 kg/m² kömür tüketimi bulunmuştur.

**Şekil 5.** Beşinci bölgedeki Kars ilinin a) İki camlı b) Üç camlı c) Dört camlı, ara boşluk dolgusu hava olan pencerele rin yayıcılığa bağlı soğutma amaçlı elektrik tüketiminin değişimi grafiği d) İki camlı e) Üç camlı f) Dört camlı, ara boşluk dolgusu argon olan pencerele rin yayıcılığa bağlı soğutma amaçlı elektrik tüketiminin değişimi grafiği
Yayılımı 0.0-1.0 arasında, 6.9,12,15,18 mm ara boşluklu, ara boşluk dolgusu hava olan iki, üç, dört camlı pencereye bağlı İzmir ili için 2.461-12.038 kwh/m², Balıkesir ili için 2.715-12.038 kwh/m², Eskişehir için 2.513-11.476 kwh/m², Yozgat ili için 3.715-11.476 kwh/m², Yozgat ili için 0.726-3.981 kwh/m², Eskişehir için 0.572-4.604 kwh/m², Eskişehir ili için 0.406-2.987 kwh/m², soğutma amaçlı olarak elektrik tüketimi sağlanmıştır.

Yayılımı 1.0-1.0 arasında, 6.9,12,15,18 mm ara boşluklu, ara boşluk dolgusu hava olan iki, üç, dört camlı pencereye bağlı İzmir ili için 3.672-20.138 kwh/m², Balıkesir ili için 2.196-12.043 kwh/m², Eskişehir ili için 1.197-6.562 kwh/m², Yozgat ili için 0.726-3.981 kwh/m² ve Kars ili için 0.572-3.135 kwh/m² soğutma amaçlı olarak elektrik tüketimi hesaplanmıştır. Yayılımı 0.0-1.0 arasında, 6.9,12,15,18 mm ara boşluklu, ara boşluk dolgusu hava olan iki, üç, dört camlı pencereye bağlı İzmir ili için 2.606-19.191 kwh/m², Balıkesir ili için 1.558-11.476 kwh/m², Eskişehir ili için 0.849-6.253 kwh/m², Yozgat ili için 0.515-3.794 kwh/m² ve Kars ili için 0.406-2.987 kwh/m² soğutma amaçlı olarak elektrik tüketimi hesaplanmıştır.

Yayılımı 0.0-1.0 arasında, 6.9,12,15,18 mm ara boşluklu, ara boşluk dolgusu hava olan iki, üç, dört camlı pencereye bağlı İzmir ili için 3.672-20.138 kwh/m², Balıkesir ili için 2.196-12.043 kwh/m², Eskişehir ili için 1.197-6.562 kwh/m², Yozgat ili için 0.726-3.981 kwh/m² ve Kars ili için 0.572-3.135 kwh/m² soğutma amaçlı olarak elektrik tüketimi hesaplanmıştır. Yayılımı 0.0-1.0 arasında, 6.9,12,15,18 mm ara boşluklu, ara boşluk dolgusu hava olan iki, üç, dört camlı pencereye bağlı İzmir ili için 2.606-19.191 kwh/m², Balıkesir ili için 1.558-11.476 kwh/m², Eskişehir ili için 0.849-6.253 kwh/m², Yozgat ili için 0.515-3.794 kwh/m² ve Kars ili için 0.406-2.987 kwh/m² soğutma amaçlı olarak elektrik tüketimi hesaplanmıştır.

4. SONUÇLAR
Camların yayılımı (0.0, 0.25, 0.50, 0.75, 1.0) arttıkça genel ısı transfer katayış değeri artmaktadır. Fakat pencerelerin ara boşluk dolgu kalınlığı (6, 9, 12, 15, 18 mm) ve pencerelemin cam sayısı arttıkça (iki cam, üç cam, dört cam) genel ısı transfer katayış değeri azalmaktadır. Buna bağlı olarak yaktuk tüketim değerleri, camların yayılımı arttıkça azalmaktadır. Ara boşluk dolgu kalınlığı ve pencerelemin cam sayısı arttıkça yaktuk tüketim değerleri azalmaktadır. Ara boşluk dolgusu argent olan pencerelemin, ara boşluk dolgusu hava olan pencerelemin cam sayısı ile ısı transfer katayış değeri genel ısı transfer katayış değerini daha düşüktür. Bu durumargonun ışıltı katayış değerleri, havaya göre daha düşük olmasından kaynaklanmaktadır. Ara boşluk dolgusu argent olan pencerelemin, ara boşluk dolgusu hava olan pencerelemin cam sayısı ile ısı transfer katayış değerini daha düşüktür. İklim bölgeleri arttıkça ısı transfer katayış değeri düşer, doğal gaz ve elektrik tüketimi artarken, soğutma amaçlı elektrik tüketimi azalmaktadır. Sonuç olarak, ara boşluk dolgusu argent olan, düşük yayılımla, yüksek ara boşluk dolgusu kalınlığı ve yüksek cam sayısına sahip pencerelemin yaktuk tüketimini azaltabilir

KAYNAKLAR
A SHORT REVIEW ON VIBRO-ACOUSTIC BEHAVIOR OF AUXETIC TWO-DIMENSIONAL STRUCTURES

Ali Hosseinkhani, Davood Younesian and Mostafa Ranjbar

AGMAGETIC STRUCTURES

ABSTRACT

Concept of the negative Poisson’s ratio had been restricted to theory, until 1987, when Lakes fabricated a re-entrant foam structure with a negative Poisson’s ratio. Since then several efforts have been devoted to create different types of auxetic materials with negative Poisson’s ratio. Materials, with auxetic nature, generally show a number of improved characteristics compared to the conventional materials. These characteristics have persuaded researchers to conduct different theoretical, experimental, and practical studies on these materials fabrication and application. In this review paper, a brief summary on the auxetic structures including auxetic plates and sandwich panels, their applications, and their vibro-acoustic behavior is reported. In the first part of the paper, definition of auxetic behavior is presented and subsequently different characteristic improvements are addressed. The auxetic features have provided various developments to the load-carrying elements and structures. Diverse applications of such structures are classified and listed. The last part of the paper concentrates on the vibro-acoustic application and behavior of auxetic structures. A few ideas about future perspectives of vibro-acoustic investigation of auxetic structures are discussed.

Keywords: Auxetics, plate, sandwich panel, vibro-acoustic behavior, optimization

1. INTRODUCTION

The usual occurrence for a material subjected to tensile load is becoming thinner in the direction vertical to stretching direction; and that is due to the positive sign of Poisson’s ratio whose value is usually between 0.25 and 0.3, for most of materials. Auxetic materials, however, have negative Poisson’s ratio, which means they are stretched when are subjected to tensile load, and, conversely, they are shrunk when are under compression load (Figure 1). The negative sing of Poisson’s ratio (auxetic feature) makes the material to behave differently from conventional materials. Auxetic feature has brought along some benefits; as far as they have been exploited in vast applications and in various industries from biomedical to engineering.

From theoretical viewpoint, it has been proved for many years that Poisson’s ratio can have a value between -1 and 0.5 (Fung, 2017), according to thermodynamic restrictions. However, the existence of a real material with negative Poisson’s ratio was unknown. Lakes, in 1987, produced the first foam samples, which showed auxetic behavior for volumetric compression factor between 1.4 and 4; he entered the concept of negative Poisson’s ratio into real applications (Lakes, 1987). After that, many researchers have focused on fabrication and design of new auxetic structures. Auxetics have been observed naturally in the nature, in some woods, rocks, crystals, and bones (Homand-Etienne and Houpert, 1989, Yeganeh-Haeri continued, 1992). Other types of auxetic materials are exist which are not naturally-occurring, but they can be fabricated, including foams and honeycombs. Today, the range of auxetic materials mainly includes foams, honeycombs, ATG etc. (Evans and Alderson, 2000). Researchers have benefited from unique properties of auxetic structures in various applications (Crichley continued, 2013). One of these applications is in the field of vibro-acoustic. Improvements which are provided on mechanical properties of auxetic structures have paved the way of the using auxetics in treatment of vibro-acoustic behavior of plates and sandwich panels. Especially, increasing Young’s modulus and
bending stiffness are among the enhancement which are efficiently exploited in improving vibro-acoustic behavior of mechanical structures (Evans and Alderson, 1992). Auxetics by themselves have low density and low mechanical performances; therefore they are mostly exploited in composite structures and sandwich panels, and they have provided improvement on mechanical properties in comparison to conventional materials.

Several review papers concerning auxetics have already been published. Alderson studied different mechanisms of auxetic materials, their structures and their superior properties (Alderson, 1999). A review on the advances and structures of auxetic materials till 2004 is reported in (Yang continued, 2004). Geometrical structures and models of auxetics (Liu and Hu, 2010), their mechanical properties (Prawoto, 2012), mechanisms of Auxetic nanomaterials (Jiang continued, 2016), and applications of auxetics in comfort and protection (Duncan continued, 2018) have been reviewed.

This paper presents an overview on concept of auxetic materials and their unique properties. The applications which are persuaded by the unique properties are briefly introduced. Then, the concentration is given to the application of auxetics in vibro-acoustic behavior treatment of two-dimensional structures. Finally, a review on vibro-acoustic optimization of auxetics is performed; and future improvements on the treatment of auxetic structures from the view point of their vibro-acoustic behavior will be discussed.

2. PROPERTIES OF AUXETIC MATERIALS

An auxetic material is the definition of a material with negative Poisson’s ratio; it has come from Greek word of “auxetikos” which means is likely to growth. Auxetic materials become fatter when are stretched and, conversely, they become thinner when are compressed. Poisson’s ratio is defined as:

\[ \nu = \frac{-\varepsilon_j}{\varepsilon_i} \]

where \( \varepsilon_i \) is strain in the stretching direction and \( \varepsilon_j \) is strain in the direction perpendicular to stretching direction.

Mechanical properties which are related to the Poisson’s ratio can be improved by employment of the negative Poisson’s ratio. The tensile (\( E \)), shear (\( G \)), and bulk (\( K \)) moduli of an isotropic material are influenced by Poisson’s ratio as Equations (2-4) (Evans, 1991). As an example of the influences of Poisson’s ratio on these parameters, one can refer to Equation (3); in this equation, when \( \nu \) approaches -1 the shear modulus will increase; as the result, \( \nu = -1 \) is appropriate for applications where high shear moduli is needed. The Poisson’s ratio influences mechanical properties of a material; other behaviors such as stress distribution around defects and transmitted as well as reflected wave stresses are consequently affected by the Poisson’s ratio (Lakes, 1993).

\[ E = \frac{9KG}{3K+G} \]
\[ G = \frac{E}{2(1+\nu)} \]
\[ K = \frac{E}{3(1-2\nu)} \]
One of the most tangible kind of auxetics is addressed in cellular solids. Two-dimensional honeycombs with re-entered cells have presented auxetic behavior (Figure 2). An auxetic honeycomb which is built using these cellular structures can provide improved mechanical properties in comparison to conventional honeycombs.

![Figure 2](a): conventional honeycomb and (b): auxetic honeycomb

In-plane and out-of-plane properties for a unit cell of hexagonal honeycombs and foams are formulated by Gibson and Ashby (Gibson and Ashby, 1999). Bitzer considered different assemblies of unit cells and presented numerical values for mechanical properties of a hexagonal core (Bitzer, 2012). In-plane and out-of-plane Young’s moduli of a hexagonal honeycomb with a unit cell as Figure 3, are presented in (Mazloomi, 2017):

\[
E_x = E \beta^3 \left( \frac{\alpha + \sin \theta}{\cos \theta \left(1 + \left[2.4 + 1.5\nu + \tan^2 \theta + \frac{2\alpha}{\cos^2 \theta}\right]\beta^2\right)} \right)
\]

\[
E_y = E \beta^3 \left( \frac{\cos \theta}{\sin \theta (\alpha + \sin \theta) \left(1 + \left[2.4 + 1.5\nu + \cot^2 \theta\right]\beta^2\right)} \right)
\]

\[
E_z = E \beta^3 \left( \frac{\alpha + 2}{2(\alpha + \sin \theta) \cos \theta} \right)
\]

Equations (5-8) are written based on cell angle (\(\theta\)) and non-dimensionalized parameters of \(\alpha = \frac{h}{l}, \beta = \frac{l}{l}\). Other parameters are obvious by referring to Figure 3. Such relations are also presented for other mechanical properties (Mazloomi, 2017). The effects of cell angle on the Young’s moduli are depicted in Figure 4. It is observed that, in a range of cell angle, high enhancement in the Young’s modulus is achieved in x direction, while the changes in y direction are not as much. Furthermore, it is possible that providing increase on one parameter, by use of Poisson’s ratio, leads to decrease on another parameter (Lim, 2014). For example, increasing the Young’s modulus is usually along with decrease in the shear modulus. Hence, improvement on the Young’s modulus may be with the expense of decreasing Shear modulus.
Chiral lattices or chiral honeycombs are also cellular structures with the negative Poisson’s ratio. Prall and Lakes showed that the in-plane Poisson’s ratio of this honeycomb is -1 (Prall and Lakes, 1997). They conducted experimental and theoretical study in order to calculate properties of such a material. A chiral honeycomb is formed by an assembly of nodes with same radii and same length ligaments. An illustration of a chiral as well as Anti-chiral honeycomb is shown in Figure 5. Name of a chiral lattice is determined by the numbers of ligaments and whether the ligaments are attached on the same side of nodes or not (Mazloomi continued, 2018). By use of Finite Element method, Alderson et al. (Alderson continued, 2010) investigated effects of the numbers of ligaments on Young’s modulus for 3-, 4- and 6-connected chiral honeycombs. They showed that the increase in the numbers of ligaments results in increasing the Young’s modulus.

As shown by this section, E, G, and K are three important properties of a material, and they can change its mechanical and thermal properties. These parameters are influenced by Poisson’s ratio. Therefore, taking the advantageous of the negative Poisson’s ratio can provide a material with modified properties. In this paper, the main concentration is devoted to mechanical properties. More explanations about their applications and behavior is addressed in the following sections.
3. APPLICATIONS OF AUXETIC MATERIALS

One application of auxetics seems to be in building nails, rivets, and fasteners, where negative Poisson’s ratio can provide easier pull-in and higher pull-out resistance (Ren continued, 2018). Hardness or ball indentation resistance in an auxetic material is higher than same material with positive Poisson’s ratio (Alderson continued, 1994). Therefore, they are applicable in protective pad, sports applications, and impact protection equipment (Duncan continued, 2018, Duncan continued, 2018). Negative Poisson’s ratio also has made designer able to create synclast surfaces (single or doubly convex surfaces); while using a material with a positive Poisson’s ratio only can provide anticlastic surfaces. Efforts for making synclast surfaces by use of a material with a positive Poisson’s ratio lead to local buckling and collapse in the honeycomb walls (Evans, 1991); this can be done without collapse or buckling, by use of the auxetic feature. The aid of this property (negative Poisson’s ratio) have been employed in manufacturing double-curved composite structures and other complex geometries (Evans, 1991, Panico continued, 2018).

Compared to the positive Poisson’s ratio, the negative Poisson’s ratio have provided better effects on comfortable of furniture and friendliness of seats (Smardzewski, 2013, Smardzewski continued, 2013). Since auxetic structures have shown to be far better than conventional structures in energy absorption, they have been used in army-blast application (Ma continued, 2010) and crash boxes (Zhou continued, 2016, Wang continued, 2018).

One practical use of auxetics is in the field of vibro-acoustic research. Auxetic structures are “application-oriented”, that is, their properties are proper in one orientation but may not be in the others. Therefore, designers must consider in which direction they are going to provide improvement. In thin-walled structures such as plates and sandwich panels, bending vibration is the dominant one; as the conclusion, during the process of improving vibro-acoustic behavior of a structure by use of auxetic feature, increasing bending stiffness is an important subject. As aforementioned, auxetics are usually embedded as a core layer into sandwich structures. Auxetics have been used in mechanical applications where high stiffness, high vibration suppression, and high energy damping are needed. Due to their high stiffness to mass ratio, auxetics are preferable in aerospace applications. This property has also provided these materials with lower modal density compared to conventional materials (Clarkson and Ranky, 1983). Auxetics have been utilized in the structure of morphing wings (Martin continued, 2008), and they could provide better pressure distribution (Bornengo continued, 2005). Ma et al. investigated the damping performance of an anti-tetrachiral honeycomb. They used metal rubber particles to increase damping ratio of the structure (Ma continued, 2013). Shiyin et al. (Shiyin, 2015) studied trichiral structures from the view point of their vibration transmission and isolation. In the following section, more discussion on the use of auxetics in thin-walled structures is addressed.

4. VIBRO-ACOUSTIC BEHAVIOR AND OPTIMIZATION OF AUXETIC STRUCTURES

This part concentrates on application of auxetics in vibration and acoustics of two-dimensionalized structures, and the vibro-acoustic optimization of these structures. Two common types of auxetic cellular structures which are mainly considered in this paper are hexagonal honeycombs and chiral lattices (Lira continued, 2009, Chen continued, 2013, Spadoni and Ruzzene, 2012, Wang continued, 2015, Lorato continued, 2010, Abramovitch continued, 2010). Other kinds of auxetic structures also exist, such as rotating rigid units (Grima and Evans, 2000), double arrowhead honeycombs (DAHs) (Liu and Ma, 2007), angle-ply laminates (Shilko, 2008), and porous polymers (Evans and Caddock, 1989).

These two honeycombs have been utilized efficiently in vibro-acoustic structures. They have provided stiffer, stronger, and lighter materials which are desirable in aerospace and automotive industry. Auxetic honeycombs have easily provided a structure with orthotropic properties. Proper design of an auxetic structure can provide high bending stiffness, which makes them useful in vibration suppression (Scarpa and Tomlinson, 2000). These types of hexagonal honeycombs have higher out-of-plane moduli than conventional honeycombs. Formulation for linear mechanical properties of conventional and auxetic honeycombs are presented by Gibson and Ashby (Gibson and Ashby, 1999), based on the beam theory. These formulation have been developed for different arrangement of cells (Scarpa and Tomlinson, 2000). Scarpa and Tomlinson presented mechanical properties of re-entrant cell honeycombs using cellular material theory. By appropriate selection of unit cell parameters, they improved mechanical behavior of an auxetic sandwich panel (Scarpa and Tomlinson, 2000). Lim investigated the effects of negative Poisson’s ratio on the static and dynamic behavior of circular plates (Lim, 2014). Maruszewski et al. (Maruszewski continued, 2013) studied free and forced vibrations of rectangular auxetic plates.

The aim of vibro-acoustic optimization is perfect arrangement and best selection of mechanical and geometrical parameters, in order to achieve lighter and quieter structures. Methods of structural acoustic optimization have been presented by Marburg (Marburg, 2002). One main approach which is used in optimization of auxetic structures is topology optimization. This method includes optimization of geometry and optimization by creating gradient in the properties of the structure. In the geometry optimization method, cells’ angels and cells’ sizes are chosen as design parameters and Radiated Sound Power Level (RSPL) from the surface of a structure can be chosen as the objective function, as Equation (8) (Mazloomi, 2018).
RSPL = $\sqrt[2]{\int_{f_m}^{f_M} L_s(f) df} dB$

in which, $f$ is frequency in $Hz$ and $L_s(f)$ is the sound level power.

$L_s(f) = 10 \log \left( \frac{P(f)}{P_0} \right)$

$P(f)$ is the sound power radiated from a surface, and $P_0$ is the reference pressure. $P(f)$ can be achieved from the following equation

$p(f) = \rho c S v_{rms}^2(f) \sigma(f)$

where $\rho$ is air density, $c$ is speed of sound, $S$ is the surface from which the sound pressure is radiated, $v_{rms}^2(f)$ is mean squared velocity at the direction normal to surface, and $\sigma(f)$ is radiation efficiency (Mazloomi, 2018).

Gradient in mechanical properties of a cellular solid can be achieved by producing gradient in geometry of cells. Creating gradient in mechanical properties of an auxetic structure is accomplished with the target that each point will have appropriate property. By embedding appropriate mechanical properties in different points of a structure, its vibro-acoustic behavior will improve. An example of this optimization method is in blades and is applied in order to suppress their vibration level. Lira et al. (Lira continued, 2011), by using an auxetic gradient cellular core, decreased mass of an aerospace fan blade and its dynamical response. Hou et al. (Hou continued, 2013) performed Finite Element modeling of bending and failure behavior of sandwich beam with an auxetic honeycomb core. The effects of geometry gradient on mechanical performance of the beam were investigated. Shiyin et al. (Shiyin continued, 2015) analyzed wave propagation through uniform and gradient geometry auxetic beams. They observed that use of gradient in size of cells provide a wider frequency band gap.

An important issue in optimization problems is CPU time usage. In these regard, employing homogenized modeling and using metamodels are suggested. Chekkal et al. (Chekkal continued, 2010) developed the use of homogenized modeling based on the Biot’s theory. Mazloomi et al. (Mazloomi continued, 2018) utilized two solid elements per gauge thickness to present homogenized behavior of an auxetic core and defined effective mechanical properties of the homogenized core by the compliance matrix. This equalization was verified by modal analysis. The method efficiently reduced computational time. Lew et al. (Lew continued, 2008) used a metamodel to reduce computational time of simulation. They build the metamodel according to learning from data and teaching the model. Genetic Programming and Artificial Neural Networks have been used in the development of the model. Wang et al. introduced the use of auxetic structures as jounce bumpers. They used the concept of metamodel in optimization process (Wang continued, 2018). Optimization procedure was accomplished by employing an optimization algorithm.

Ranjbar applied different optimization algorithms to the problem of structural acoustics (Ranjbar continued, 2009). He made comparison on them from the view point of their convergence, accuracy, and effectiveness. Kaminakis and Stavroulakis developed an evolutionary-hybrid algorithm based on Particle Swarm Optimization (PSO) and Differential Evolution (DE) in order to optimize topography of auxetic structures (Kaminakis and Stavroulakis, 2012). Lira et al. (Lira continued, 2011) used first order optimization method to minimize weight of an auxetic blade. Ranjbar et al. (Ranjbar continued, 2016) optimized Radiated Sound Power Level (RSPL) of one-dimensional gradient hexagonal honeycomb sandwich structures by use of Random as well as first order optimization methods. Mazloomi et al. minimized mass and RSPL of two-dimensional gradient honeycomb sandwich structures using genetic algorithm (GA) and the Method of Moving Asymptotes (MMA) (Mazloomi continued, 2018). Bacigalupo et al. (Bacigalupo continued, 2017) studied in-plane wave propagation in an auxetic beam with tunable local resonators. The function of resonators was to act as a low frequency passive filter. The band gap was maximized by optimization of geometry and mechanical properties. Combination of MMA with Monte Carlo technique were used as optimization method (Bacigalupo continued, 2016).

5. CONCLUSION

In the last few years, auxetic materials have emerged with their unique properties which are practical in different technologies from biomedical to textile and mechanics. Today, the number of researches on auxetics have an ascending
trend; and still there is vast available space for studies about their manufacturing, properties, and applications. This paper presented a brief introduction on the fundamental auxetic materials. Their astonishing properties and the applications, which are persuaded by these properties, were mentioned. The main concentration was devoted to the improvements on vibro-acoustical behavior of two-dimensional structures.

Employment of auxetics in mechanical structures, from the aspect of vibro-acoustics, seems to be an open area which is expected to attract a great attention in following years. Optimization process of structural acoustic of conventional structures is already studied thoroughly. However, more studies are needed to perform on the auxetic structures. Topology optimization methods have efficiently been utilized in optimization of auxetic structures, but more study can be accomplished on them, especially experimental studies. Use of rubber fillers looks to be an optimization approach which can optimize damping ratio of an auxetic structure.

Concentration on mass minimization/optimization of auxetic structures needs to be considered more precisely. Combining the concept of negative mass/stiffness with negative Poisson’s ratio is another attracting area which may leads to more improvements on dynamical/acoustic response of two-dimensional structures.

REFERENCES


STUDENT’S PERSPECTIVES OF ICT USAGE FOR EDUCATIONAL PURPOSES: A CASE STUDY OF EASTERN MEDITERRANEAN UNIVERSITY MECHANICAL ENGINEERING*

Ozra Shirzadeh Shaghaghi, Ersun İscioglu

Information and Communication Technology Department, Faculty of Education, Eastern Mediterranean University, TRNC via Mersin 10, Gazimagusa, Turkey

*a#parvaz_83_2004@yahoo.com, b#ersun.iscioglu@emu.edu.tr

ABSTRACT

Information and Communication Technology (ICT) has an important influence in engineering education. In this study, the department of mechanical engineering of Eastern Mediterranean University (EMU) is examined for the usage of ICT by undergraduate students for learning purposes. The study was designed as a quantitative research. The study was attended by 102 students enrolled in EMU mechanical engineering undergraduate program in 2015-2016 academic years. The results were analyzed with SPSS software. As a result of the study, it is observed that the increase in computer usage and the increased availability of software tools significantly affect the learning behaviors and expectations of engineering students. In addition, the study concluded that the opinions of undergraduate students of the mechanical engineering department at EMU on the use of ICT for educational purposes were positive.

Keywords: Information and Communication Technology (ICT), Engineering Education, ICT Usage, Undergraduate Level, Mechanical Engineering Students.

1. INTRODUCTION

Information and Communication Technology (ICT) has had a profound effect to change our life. It has attracted a considerable attention among communities in the world. So that usage of ICT has become an imperative fact in modern society. Many people and countries now pay attention for understanding ICT and learning the basic skills and concepts of ICT [1].

Nowadays, many ICT are being used in educational institutes, universities and higher educational institutes to teach the engineering science to the students, e.g. power points, internet, computer simulation software etc [1]. This matter has facilitated student to access to more information via digital mobile tools and interacts with more information. Furthermore, the use of such technologies for educational applications is highly supported by students. These opportunities help students to learn inside and outside the classroom, by developing more appropriate educational materials. The concept of the computer experiment was discussed and developed. The use of computer graphics was illustrated with examples taken from current teaching material. Another new application of ICT in engineering education is the electronic mentoring of educational programs and engineering students at universities. This opportunity has been provided by ICT facilities as it is independent from geographical location and time constraints [3]. In fact, ICT provides some facilities like web-based via broadband connections that can help to access the educational resources from anywhere. For example, open courseware provides extra support to engineering students via the digital networks like Internet. Michael et al. [4] investigated the effect of ICTs on the education of engineering students. They employed ICT like video-conferencing facility in education process. They showed ICT is a strong tool for learning processes. They performed their evaluation via a questionnaire to investigate the usage of ICT in engineering education. González et al. [5] studied the ICT usage for engineering student’s educations in Spain. They showed that it is an important effect. They showed that the engineering students feel more satisfied if new ICTs are being implemented in the teaching process of courses. This study showed the positive effect of ICT on the graduate students’ learning as well. They showed the ICT can provide better learning facilitates for the engineering students and increases the responsibility and motivation of them during the learning process.

*This study carried out for the Master’s thesis of Ozra Shirzadeh Shaghaghi under the supervision of Assoc. Prof. Dr. Ersun Iscioglu.
To gain engineering education goals engineers should understand nature that goes beyond mere theory—knowledge that is traditionally gained in from the literature shows that there are rare or very few studies available on this topic. Investigation of ICTs usage in the learning process of engineering students is still an area that needs more attention. The main purpose of this article is to investigate the EMU mechanical engineering student’s opinions on ICT usage for educational purposes. The main questions can be considered as what are the opinions of mechanical engineering students on ICT usage for educational purpose? What are the opinions of mechanical engineering student’s ICT usage according to gender and educational background?

Some limitations can be seen in advanced:

1. There was only one full semester in the academic year of 2016-2017 available to perform this study.
2. Only a limited number of mechanical engineering students of EMU were available for performing the questionnaire.

The effect of ICT in education of mechanical engineering students of EMU will be investigated. The result of this study can help the university administration for the modification of course curriculums wherever which are needed. By having better knowledge about the effect of ICTs tool in the improvement of learning process of students, both side of university administration and students can have better impression. Indeed, it can help the university management to make some strategies and decision for improvement of ICTs facilities in the university according to the specific needs of university and students [1].

2. Engineering Education

Henderson and Broadbridge [2] explained lack of funding, poor material, students’ population explosion (without equivalent facilities), poor high-quality workers (in terms of trainers or teachers), industrial practice and poor position of organizations as the principal problems handled by engineering education technological development follows as the main key problems to a nation’s growth. On the other hand, usual engineering knowledge (acquired through suitable structures) plays an important character in the achievement of an elevated level of technological advancement. Ali, et al. [6], investigated contemporary trends and techniques of technology in engineering education. They indicated that predictions trends and other various viewpoints of the latest situation in the use of technology based in the prior predictions and the experience in distance, on-line and on-class engineering education. In current century use of technology in the engineering education are increased. This modern technology provides new learning process in everywhere and in different learning model. These modern technologies are very common toward latest years. In additional, these impact in both traditional face to face systems as well as in on-line or distance models. In the last 10 years new use of information and communication technologies in the learning process effective before in the last part of the 20th century. Improvement and modernization of engineering are rapidly increased; therefore, this needs for engineering educators to adapt to new realities and learn contemporary trends (Ali, et al. 2010). In the past in several countries challenges and opportunities associating to engineering education and future paths and visions have been examined and a difference of views have been displayed. As Henderson and Broadbridge [2] have stated that ICT can help engineering students to get more knowledge about their major.

Bakare [7] investigated that ICT play a critical role in education. The goal of this study was examination the importance of ICT in education to reach better teaching and learning area of students. However, this study searched statistically of importance of ICT on student’s Educational life in Eastern Mediterranean University. This study considerate 197 students in university of Eastern Mediterranean. The results showed that usage of ICT tools by students makes their work more flexible. Also, they were agreed that with use of ICT they can do their work quicker and better. The leads of the study showed that students studying in ICT saw lower issues than the students in other fields. Shirzadeh Shaghaghi [8] studied the effect of ICT on the education process of undergraduate students in mechanical engineering students of Eastern Mediterranean University. Alazam et al. [9] presented the levels of ICT skills and use of ICT in the classroom among technical and vocational teachers in Malaysia. They collected data from 329 technical and vocational partcipation of teachers. They investigated the skills of teachers in ICT, how teachers use ICT and their demographic factors. The study showed that skills of teachers in use of ICT were at average levels, also there were significant differences of skill of teachers in use of ICT as a function of demographic factors. In additional, there were meaningful relationship between ICT skills and ICT integration in classroom. Some factors related to the teachers like their ages, genders, teaching experiences, except level of educations did not impact ICT integration in classroom. Lorencowicz et al. [10] investigated the computers and internet access and usage by some students during their studies. The results were based on a survey in
2009-2012 on groups of 320 to 405 students (each year) of two universities in eastern areas of Poland. They concluded that the access of student’s ICT facilities was at an elevated level.

The traditional professors realized that the students could not get the fundamentals of engineering design by using of software, but the younger professors found that they had some challenge in the experiment of this process. The students were happy with the new teaching methodology as they found it useful for their future working environment. Also, the advantages and disadvantages of usage of some ICT tools in a first-year technical drawing course were examined. The application ICT in mechanical engineering students were investigated in two universities. The population of considered students for that study was 225 final year engineering students from two Kenyan technical universities that had mechanical engineering programme. The results of the study indicated the relationship between ICT and engineering education. To address these facts, the expansion of the curriculum in engineering education curriculum was proposed to include some courses like Computer Aided Design using modeling software, introduction to programming languages, Matlab, MathCAD and introduction to finite element modeling software [11].

It is accepted that the communication and technology are important to education [12]. However, due to the emerging of communication barriers in education, it is not an easy task to be fully accomplished. The ICT attitudes from 361 mechanical engineering students were investigated. The students used different ICT tool at work after their graduation. The ICT attitude was measured according to Likert scales [13] by SPSS software using descriptive statistics and Mann-Whitney test. The results showed the mechanical engineering students had positive ICT attitude.

3. METHODOLOGY

This study examines quantitative research. Quantitative methods indicate numerical analysis of data collected from surveys, questionnaires, and polls. Quantitative research focuses on collecting numerical data and concluding it beyond groups of people or to describe a special event. The main purpose of leading the quantitative research study is either to describe or to try the measured subjects before and after a treatment. Qualitative researchers now have the possibility to choose from an increasing array of theoretically and technically sophisticated methods.

The participants are the target group of undergraduate students of mechanical engineering department in EMU during the 2016 fall academic semester. The participants are from various countries with different ICT backgrounds.

A questionnaire is administered to the target group of undergraduate students of mechanical engineering department during the 2016 fall academic semester of Eastern Mediterranean University (EMU) in Northern Cyprus. Because of this research about gathering the data of ICT on Mechanical Engineering students. The questionnaire is based on five-point Likert scale. A psychometric response scale primarily used in questionnaires to obtain participant’s preferences or degree of agreement with a statement or set of statements. Likert scales are a non-comparative scaling method and are unidimensional (only measure a single trait) in nature. Respondents are asked to show their level of agreement with a given statement by way of an ordinal scale. Likert scales developed by Dr. Rensis Likert, who was a sociologist at the University of Michigan. His original report described “A Technique for the Measurement of Attitudes” was published in the Archives of Psychology in 1932 [13]. His aim was to improve averages of measuring psychological perspectives in a “scientific” approach. He attempted a method that would provide perspective measures that could rationally be described as measurements on a proper metric scale. Likert extended the principle of measuring characters by asking people to answer to a set of statements about a topic, in terms of the length to which they agree with them, and so drawing into the cognitive and affective parts of approaches. Likert-type or frequency scales use made decision reply formats and are designed to measure opinions or ideas (Likert, 1932). The main advances of this as A 5-point Likert scale is easy to collect the data owing to the fact of numbering of each option. In additional, as investigations can change from “one” to “five” or “low” to “high,” it further provides extra reach than a simple yes/no question. A 5-point Likert scale illustrated the measure opinions of people. The questionnaire used to gather data consisted of three parts; the first part gathered demographic characteristics (e.g., gender), and computer and Web experience (e.g., PC ownership, Web-usage frequency, and Web-usage activities). In the second part, there were 24 questions, described by five-pointed Likert Scale (from “1= always”, “2=usually”, “3=sometimes”, “4=seldom”, to “5=never”). This section means to define the expectations of students in terms of their use of ICT and it is created to measure knowledge of the possibilities suggested by ICT. The last part obtained the student’s point of view regarding the computers and ICT facilities of the faculty. Control questions were included to discover any incorrect data from the students in the questionnaire [1]. The aim of surveys is to gathering data based on impact of ICT tools on Mechanical Engineering students in Eastern Mediterranean University. The questionnaire is consisted of three sections similar with the work reported in Yaman et al. [1]. The first section of questionnaire is designed for the gather of demographic characteristics of participants like gender, age, department, type of attended high school, and sort of computer that they own and use; where they want mostly to make up their computer needs if they do not have a computer, what is the most objective to use computers of department, and the computer usage
frequency of students per day. In the second section there were 24 questions, categorized by five-point Likert Scale according to the relevant question. This section aims to determine the expectations of students in terms of their use of ICT and it is designed to measure awareness of the opportunities offered by ICT. This section included student’s use of computer for communication, courses and projects, or for entertainment purposes. Also, it considerate student’s sufficient about using computers, connect to the internet with their mobile phone, frequency of use computers of faculty, using programs which are related to their profession, necessary of expressing a lesson through computer, usage of computer for their profession. Furthermore, the questionnaire investigated student’s usage frequency of email and search engines tools for courses and entertainment, and the frequency of use from the internet for student’s projects and lessons, whether they can meet their needs (shopping) on the internet, frequency usage of computer for communication, lessons and projects and entertainment [1]. The last section elicited the student’s point of view concerning the computers and ICT facilities of the faculty. This section included; number of computers that sufficient in department, meet of the hardware of department computers with student’s needs, computer programs required in department by student’s profession, whether the faculty members use the computers and information technology tools for communication, entertainment purposes and homework/ projects purposes. In addition this study demonstrated announcements of university web pages, faculty web pages, department web pages, faculty members' web pages and communications and information services adequate, and adequate of presentations of course materials in the digital media, offered of the professional software by academician present and use in their department, efficiently use the professional software about their department, provide professional software need in their department when they need, professional software teachings will be useful to students in their professional life. Also, the students are asked whether they use the professional software and computer laboratories of department for their education, is the IT services of department enough good for the students and their educational activities [1]. Usually, mechanical engineering students in undergraduate level are using some technical software as SOLIDWORK, AUTOCAD and ANSYS. Furthermore, Microsoft office, social networks, and mobile communications are very common among them. These software and ICTs are developed by various international companies [1].

SPSS is used for the evaluation of data. SPSS Statistics is a software package used for logical batched and non-batched statistical analysis. Statistics involved in the base software are descriptive statistics, frequencies, Means and t-test and Anova. The reliability of the administrated questionnaire is measured by control questions. It is observed that 63.4% of collected data from students is valid. This indicates that the variables are reliable. The Cronbach’s alpha, which takes on values between 0 and 1 for the consistency of the questionnaire, is calculated at 0.67 of the questionnaire. Therefore, it can be seen questions are independently reliable.

The statistical analyses of collected data for each part of questionnaire have been presented in this section. Also, student’s opinions on objectives of computer usages have been shown. The items of first part can be seen in appendix of this work. It is seen that the students usually use computers their studies for research and projects and sometimes use them for entertainment. The questions and their calculated average values for descriptive statistics are given as follow. Most of the students connected with mobile phone to internet. On the other hand, the average value of using computer of faculty and department of mechanical engineering for their education purposes is 2.79. The mean of use e-mail by mechanical engineering students of EMU is 4.29. Furthermore, usage of search engines tools for working on project or lesson is common among students in EMU mechanical engineering students and the average use of search engines tools like Google for entertainment is 4.36. But the means for meeting student’s needs on internet like shopping is 2.74. Furthermore, the means of EMU mechanical engineering students for usage of faculty computers and information technology tools for communication and entertainment is 2.02 and for homework/projects purposes are 2.76. Mechanical engineering students in EMU use information and communication technology like mobile for connecting to internet, and they use email frequency. In additional, they use ICT tools for working on project and lesson; this shows a good usage of computer in the lecture by mechanical engineering students of EMU. In the same way, EMU mechanical engineering students are good usage of search engines for fun and entertainment. On the other hands, the mechanical engineering students of EMU are not interested to use computers of faculty and ICT tools for fun and communication and homework/ projects purposes. The average mean for requirement of expressing a lesson through computer is 4.17 and for essential of English language for computer usage is 4.28. Therefore, it shows that the mechanical engineering students of EMU need learn lesson through computer and they believe that knowing of English language is important. The average level of being confident about using computers and information is decreased. Also, they are seeing themselves not enough about computer usage and information technologies for their studies. Student’s opinion about the need of usage computer for their profession is 4.23. Furthermore, students believe about adequate of their ability to use computer for their profession is decreased. So, EMU mechanical engineering students indicate that they are less professional in using computer. On the other hands, the participant EMU mechanical engineering students responded that the computer usage is obligatory in their profession. The means of student’s opinion of they don’t connected to the internet with mobile is 1.73, and the average value of do not use much computer faculty is 3.36. Also, the means of student’s opinion about do not use internet much in project and lesson is 2.04. Therefore, EMU mechanical engineering students show that they are using their mobile for accessing
to internet. They are undecided to use computer of faculty. On the other hands, they are interested to use internet much in projects and lessons. Student’s view of point about satisfaction of number of computer in department. The average value of student’s opinion about satisfaction of number of computer in department is 2.86. So, mechanical engineering students in EMU believe that the number of computer in department is few and department needs more computer facilities. The mean of student’s opinions about convene of hardware of department computers with students require is 2.82 and this shows that mechanical students in EMU are agree that there is a lack of having access to the licensed professional and academic software. The average value of student’s opinions about computer programs obligatory by student’s profession is 3.52. Therefore, EMU mechanical engineering department computer hardware and specific subjects that are related to the educational programs are sufficient. Students believe about adequate of university, faculty and department web pages, announcements and communications. The mean of student’s opinion about adequate of announcements and communications services for university web pages is 3.69, for faculty web pages is 3.73. Also, the average value of student’s opinion about adequate of department web pages and other information and communication services is 3.53, and for faculty members’ web pages and other information and communication services are 3.68. Furthermore, the mean of presentations of course materials in the digital media is 3.70. Therefore, participants considered that the departmental and academic web pages are satisfactory in terms of making announcements and conveying information about the courses. Most of the students agree that there are advantages of the use of specific educational programs for their future career.

4. CONCLUSION

The aim of this study is to investigate of information and computer technology usage of the undergraduate students of mechanical engineering students of Eastern Mediterranean University for educational purposes and the relationship between the gender and educational background of students and ICTs usage. In conclusion of mechanical engineering student’s opinions of ICT usage for educational purpose, they are interested to use information and communication technology tools like mobile phone for connecting to internet, working on project and lesson and using email for communication. This shows a good usage of computer in the lecture by mechanical engineering students of EMU. Furthermore, EMU mechanical engineering students responded that the computer usage is obligatory in their profession. Mechanical engineering students in EMU believe that there is a need of having access to the licensed professional and academic software. However, ICTs facilities of EMU mechanical engineering department like the computer, video projectors etc. which are related to the educational programs, are sufficient. The participants considered that the departmental and academic web pages are satisfactory in terms of making announcements and conveying information about the courses. Most of the students agree that there are advantages of the use of specific educational programs for their future career.

In investigate of mechanical engineering student’s opinion of ICT usage according to gender and educational background, there is no significant different between male and female opinions in using information and communication technologies. In additional, students who graduated from super high school and classical high school are more interested to use ICT tools in their higher education. Because of the study, it is observed that the increase in computer usage and the increased availability of software tools significantly affect the learning behaviors and expectations of engineering students. In addition, the study concluded that the opinions of undergraduate students of the mechanical engineering department at EMU on the use of ICT for educational purposes were positive.

REFERENCES


A REVIEW ON OPTIMIZATION OF SYSTEM OF SYSTEMS

Merve Akkaya, Mostafa Ranjbar

Ankara Yıldırım Beyazıt University, Department of Mechanical Engineering, Ankara, Turkey
Corresponding Author: mranjbar@ybu.edu.tr

ABSTRACT

System of systems is a combination of goal-oriented set of systems to produce a new system which offers more functionality and performance. Also, optimization of System of Systems improves the performance of such new complex system. It brings many advantages in industrial applications like automotive engineering. This paper presents a study on optimization of system of systems (SoS). It applications in automotive systems and transportation systems are reviewed. This can provide a clear overview on the topic and gives some guidelines for future investigation and application in ongoing automotive engineering industries.

Keywords: System of systems, Optimization, Application, Design, Automotive, Review

1. INTRODUCTION

Nowadays, computers help us to get better optimization performance even for very complex systems. As a result, it can be possible to get better optimization results when subsystems have been already optimized for increasing individual quality. SoS term is arisen in which the individual systems are embedded. In fact, it consists of a set of systems which are independent from each other. When they are combined into larger systems, they become exclusive properties [1]. With these systems, very large systems may have developed by forming frameworks to integrate constituent systems. The multi objects system which consists of many individual components, can cause complicated analyses. It is explicit that a special numerical analyse is required to analysing SoS at the global level, because optimization of each individual components may not give optimality of their collaboration.

System of systems is almost new research area. While many researchers and scientists used it, there is almost no certain definition of it. But, it is obviously clear that SoS is comprised of many small systems or subsystems which come together for gain new properties [2].SoS describes integration of many small components to obtain global needs of multi-systems. For example, car and road are system while product range, integrated traffic system isSoS. Aeroplane is a system while airport or air traffic control system is SoS. Train is a system while station, signaling, rail network isSoS. Building is a system while shopping mall is a SoS. Also, there are some examples about biological, sociological, environmental, organisational and political SoS.

System of systems engineers (SoSE) should encourage multiple purposes and visions of systems. They should optimize the system of systems in a useful way. There are some differences between system engineering and System of Systems engineers (SoSE).

2. SYSTEM OF SYSTEMS (SOS) APPLICATION

The system of system is used in many science and technology fields such as military, security, aerospace, manufacturing, transport, environment systems, and disaster governing. In the following, we will discuss more about them.

In Hayden and Jeffries [4] presented a study about flexible Joint Polar Satellite System (JPSS) was objected to design forecast of weather conditions. JPSS improves weather forecast, ensures safe shipping travel and reduces
its cost, ensure safer military operations and helps energy generation. In Nanayakkara and Jamshidi [5] explained for Future Combat Missions (FCM) for defence national security. It composes of 18 manned or unmanned systems connected with information network. It has high range ability about defence, stability, manoeuvrability, survivability and reliability. SoS is caused improving performance in these cases. Samuel [6] presented the Target Evaluation and Correlation Method” (TECM) as an assessment approach to Global Earth Observation System of Systems (GEOSS). A new method which was about evaluation and correlation levels of targets was explained. Target Evaluation and Correlation Method was used to specify the Target Correlation Level (TCL). The proposed TECM process and applications were presented in the various parts of GEOSS Implementation Plan.

Klein and Vliet [7] studied the systematic review of system-of-systems in architecture research. They showed results of a systematic review from SoS architecture perspective. The purpose of this research was to analyse the scope of system of system for architecting reports. Davendraingam and DeLaurentis [8] presented a work on robust optimization framework to architecting system of systems. Optimization of architecting system of systems in resilient was presented. Interdependent systems were used with modelling fulfil overarching capability objectives. The newest robust optimization methods were applied with Mixed-Integer Program (MIP) for improving framework.

Embedded systems provide improving performance of system; perform completely new functions of system. When embedded system is used, system is changed completely. Embedded systems are using many applications. There are many explanations about automotive electronics applications and technologies. Automotive system safety analysis and cost efficiency were explained with Larsons thesis [9]. It [9] was included architecting and modelling automotive embedded systems. In Shin and Lim [10] studied about Unified Modelling Language (UML) model-based automatic test case generation for automotive embedded software testing through model-based approach. They developed a method for automatically generating software and hardware test cases. With this method, hardware and software test case generation required sources could be decreased. Also, Samuel mentioned a method for producing UML state diagrams using in automotive systems. Gulia and Chilla [11] produces some test cases with using UML diagrams. Florin and Norbert [12] also investigated optimization of emergence test cases for automotive systems.

DeLaurentis [13] performed a study of understanding transportation as a system of systems design problem. System of systems was introduced in aerospace design implications. Future transportation systems were described, as well.

3. OPTIMIZATION BASED DESIGN

Optimization is a mathematical discipline which intends to improve ability of the systems. It can be using in many fields such as architecture, nutrition, electrical circuits, economics, transportation, military, defence and [14].

Strong engineering designs based on optimization methods. Using optimization methods, inputs (analysis variables) of system should be known for making calculations. Analysis variables are composing of design variables and other quantities such as material properties. Analysing outputs are called as analysis functions such as stress, deflection, COP value [15].

Optimization is a finding an alternative way with maximization desired factors and minimizing undesired factors. General purpose is finding most cost-effective way or obtaining highest performance way with system constraints or boundaries. There are various optimization methods available. However, the most efficient and robust solution method should be selected. For each specific case selecting most appropriate optimization methods, some optimization methods are explained below.

Linear programming method can be used in wide-range of optimization problems. Many engineering, and economics problems can be solved with this method. Manufacturing, transportation and many other design problems are solved using this method. Linear objective function is minimized or maximized with respect to unknown variables and inequalities [16]. Disadvantage of this methods are lack of risk assessment and using only linear objective functions [14].
Non-Linear programming method is for solving of optimization problems with equality and inequality systems. Nonlinear objective function is minimized or maximized in unknown variables and inequalities [17]. Many algorithms may not find the global minimum. This method applicability is limited. In fact, this method is suitable for specific kinds of problems [14].

Gradient methods are using minimizing convex differentiable equation. Non-differential equations are not solved by this method. It is generally not fast. Objective function shouldn't be noisy for obtaining accurate results [14].

Gradient free methods include multiple objectives. It is using with non-differentiable equations and/or constraints. It is effective for finding local minimum nonlinearity constrained problems [18]. Evolutionary Algorithms methods are created from biological evolutionary such as mutation, recombination of genes. Finding solutions may not be cheap. Parameter adjustment is accomplished by trial-error method [14]. Probabilistic approach method is efficient in computational solutions. Also, it is less efficient than non-probabilistic ones [19].

Fuzzy Logic method writing task about the formulation may be challenging. It has more than one way for compound evidence. Problems can solve with long inference chains. It is not efficient for complex parts [20].

Swarm algorithms method inspired from natural swarm systems such as ant colony. It has low cost while it has fast solutions. It has robust solutions above complex problems. It can be practiced in a wide variety area with many problems. It hasn't central control which makes causing this system inefficient [21]. Multi objective optimization method is using many mixed optimization problems (min-max). It is recommended that changing all objectives to same type. Optimal solutions are termed as Pareto solutions. There may be multiple minimum solutions with this method [18].

There are many researches about optimization of energy systems. Hennet and Samarakou [22] had a study about optimization of combined solar and wind power plant. Also, optimal capacity of a battery storage system was investigated. Lozano, Valero and Serra [23] were presented local optimization of energy systems. This paper demonstrated about exergic and marginal costs theory from every component of system. It was found that global optimization methods were proposed method for these type calculations. Peippo and Vartiainen [24] studied about optimization procedure in order to defining optimal design buildings in early design stage. Shi et al. [25] was presented a study about PV, wind hybrid power system thermo economic analysis and created robust optimal system. Rezvan et al. [26] was studied about a robust optimization method to defining optimal energy generating technologies. Qasaimeh [27] was investigated optimization of the angle of inclination for solar energy for every month and seasons. Sauchell et al [28] presented study about integrating a device into the PV system leads to optimizing solar energy production. Good et al [29] presented optimization of solar energy potential for buildings in urban areas. They tried to find optimal solutions using solar energy in buildings of urban places. According to these examinations, different solar technologies were tried and found that solar thermal systems were substantially bigger annual yield output than PV systems. Wang et al [30] presented multi objective robust optimization of energy systems for a sustainable district in Stockholm study. They were investigated multi-objective robust optimization approach for minimizing greenhouse gases and life cycle cost viability in energy systems.

There are many methods and papers about reducing torsional vibrations for automotive systems with addition flywheels of the system [31]. These flywheels work in tune with vibration dampers [32]. In Alsuwayyan and Shaw [33] presented a study about performance and dynamic stability of general-path centrifugal pendulum vibration absorbers of automotive system. Centrifugal pendulum vibration absorbers were using many machines to decreasing torsion vibrations. With this study, a few identical centrifugal pendulum vibration absorbers performance and dynamical stabilities were researched. According to research results, performances of systems were restricted as two separate types and some advices were given for selection of parts [34] a new method was applied for optimization and development of torsion vibration dampers. Aim of this method was join the simulations into the initial stages of development. In Zink and Hausner [35] presented the centrifugal pendulum-type absorber. Decreasing fuel consumption with producing high torques at low speeds is important for obtaining driving pleasure. Centrifugal pendulum-type absorber was developed by LuK [35] (In this study, this type absorber was developed using as an isolation material in drive systems. In Mall et al [37] presented the
simulation based optimization of torsion vibration dampers in automotive power trains. Isolation materials used in automated design was researched and many drive train and dumper types were shown to make dynamic analysis of design. Numerical optimization operation was applied to find the most appropriate geometric parameters. Augmented Lagrangian Particle Swarm optimization method was applied successfully.

Suspension systems are using automotive system widely. Nakai et al [38] studied about the development suspension system with gimbals, actuators and a gain-scheduling controller for a flywheel battery on electrical or hybrid electrical vehicles. Els et al. [39] found spring and damper characteristics were needed for driving comfort. In Gysen et al [40] presented a study about design aspects of an active electromagnetic suspension system for automotive applications. An active electromagnetic suspension system was designed with tubular permanent-magnet actuator to provide extra stability and safety. Passive and new designed system was compared. Also, many researchers optimized suspension systems using genetic algorithm method [41]. MATLAB simulation, analysis tools and FEM (Finite Element Method) was used many researches to optimize suspension systems [42]. In Abbas et al [43]. presented a study about optimal seat and suspension design for a half-car with driver model using genetic algorithm to reducing vibration of human body during driving. Genetic algorithm was applied for finding optimal parameters of suspension design. With this type of algorithm, (with the determined parameters interval) closely optimal solutions were found. Genetic algorithm solutions were compared with passive suspensions. In Mitra et al [44] presented a study about design of experiments for optimization of automotive suspension system using quarter car test rig. The objective of this research was obtaining ideal suspension and steering geometry parameters. Different combination of steering geometry parameters with suspension was investigated. Then, optimum combination was found, tested and obtained results with compared other results. In Xue et al [45] presented a study about optimization of spring stiffness in automotive and rail active suspension systems. In this research, spring stiffness which is important effect on suspension systems was improved with using a new multi objective optimization method. Ride comfort, safety, reliability stiffness effect on actuator and many considers are taken into for assessment. Then, it was deduced that used new optimization method was effective and it can be used widely in automotive and rail active systems.

In recent years there are many researches relation between gear profile and kinematic flow pulsations. Bonacini [46] is one of the authors work about this relation. He also proofed theoretical flow of an EGP (External gear pump) based on involute profile of the gears. Same information was also mentioned by Ivantysyn [47]. In Vacca and Guidetti [48] presented a study about optimization of relevant design parameters of external gear pumps for automobiles. This paper was identified external gear pumps numerical analysis and procedure. Also, this paper focused on optimization pump and bearing blocks and optimized design prototype was tested. Measured results and experimental results were compared. Mucchi et al [49] studied about dynamical behaviour of gear external pumps and experimental and simulation results were compared. Fiebig and Korzyb [50] studied about vibration and dynamic loads in external gear pumps with using simulation model. In Zhao and Vacca A. [51] presented a study about formulation and optimization of involute spur gear in external gear pump for automotive. The purpose of this paper was improving design of asymmetric tooth geometry involute gear and formulation of flow rate with asymmetric and symmetric gear pump. Multi-objective numerical optimization algorithm was used as an optimization method. It was observed that researched gears how higher performance on standard gears. Also, it was shown how parameters impress tooth profile asymmetry.

Lee and Yoo [52] had a study about improving simulation program for automotive air conditioning system. With using simulation program, operation parameters effect on the system performance was investigated. In Jabardo et al [53] had a study about modelling and experimental evaluation of an automotive air conditioning system with a variable capacity compressor. Air conditioning system was using automotive sector in order to supply comfort and high efficiency. Computer simulation model was improved for automotive air conditioning system. Both simulated and experimental results were compared. According to results, simulation results were shown almost actual performance with small deviations. Skiepko [54] present a condenser model and made calculations with this model, then it was shown that flow regime and steam quality values affect heat transfer and efficiency. In Khayyam H. et al [55] had a study about reducing energy consumption of vehicle with using air conditioning system. Energy management system (EGM) and without EGM simulation was researched. With this study, EGM comfort temperature was obtained in cabin in automotive with reducing energy consumption. EGM and without EGM simulation results were compared. Tian et al 2014 [56] were modelled a parallel flow condenser
and optimized this model with Lavenberg-Marquardt algorithm method. In Shojaeefard et al [57] had a study on multi-objective optimization of an automotive louvered fin-flat tube condenser for enhancing HVAC system cooling performance. The objective of this research was creating higher coefficient of performance (COP) automotive refrigeration and cabin design. Multi objective optimization method was used in HVAC system design of this system. According to this study, optimization designed of cooling system COP value and cooling capacity value were increased when compared non-optimized model.

In Zhang [58] presented a study about design and testing of an automobile waste heat adsorption cooling system. Absorption systems were using automotive sector for reducing greenhouse gases. Bruno et al [59] study purpose was showing effect of post-combustion degree on the integrated system performance on the integration of micro gas turbines (MGT) and absorption chillers. Main advantage of this study was COP of chillers was higher and this reason were explained in the paper Critoph et al [60]. To increasing system performance, some control systems were designed. Also, adsorption cooling system was designed and tested by experimentally. According to results, this system can be used for waste heat adsorption cooling system. COP values were also so supportive according to this paper. Javani et al [61] was investigated compare of two refrigeration cycle with available waste heat as a used in hybrid and electric vehicles to cool the cabinet of automotive. In Pang et al [62] presented a review about liquid absorption and solid adsorption system for household, industrial and automobile applications. This paper involves innovative ways of adsorption system which were liquid absorption and solid absorption systems. It was observed that absorption and adsorption systems were decreasing fossil fuel usage, clean energy supplier, cheap and incrementally increasing in the future. In Verde et al [63] presented a study where dynamic performance of all system was tested and cabin temperature was estimated with wasted heat in Fiat Grande Punto vehicle. Unique design types were investigated for increasing system efficiency. It was seen that supplied engine waste heat was enough for cooling in the cabinet.

In a series of publications by Ranjbar et al. from 2007 to 2017, the concept of multidisciplinary engineering design optimization of various automotive structures was investigated and reported. They showed the effect of optimization in the quality of results. Furthermore, they presented very innovative techniques for improving the optimization results for real industrial automotive applications by using novel optimization methods [65-93].

4. CONCLUSION

The brief review of the optimization system of systems (SoS) was explained, SoS main structure and general definition of it were introduced. System of systems engineering (SoSE) duties were clarified briefly and application of SoS were mentioned with this study. Various applications of the SoS concept were explained based on mechanical engineering area. Examples of optimization of SoS in automotive engineering field like suspension system design and cooling system design were presented. A brief survey was done on application optimization on SoS. This showed that this field will be emerging in the next decade, especially by development of self-drive cars. More advance SoS will be used in electrical cars to protect the environment and provide better service to the community.

REFERENCES


[93] M. Ranjbar, M. Mirsalim, “A Feasibility Study Phase on the Hybrid Electric Vehicles For Iran “, The First International Conference on HEV’s, Iran Science and Technology University, Tehran, Iran, in Persian (April 2001).
INVESTIGATION OF EXERGY COSTS OF AIR AND GROUND SOURCE HEAT PUMPS WORKING WITH DIFFERENT REFRIGERANTS

Oğuz Ozan YOLCAN¹, Mustafa Arif ÖZGÜR¹, Burcu GÜLMEZ², Ramazan KÖSE¹

¹Kütahya Dumlupınar University, TURKEY
²Manisa Celal Bayar University, TURKEY
oguzozan.yolcan@dpu.edu.tr, arif.ozgur@dpu.edu.tr, burcugulmez@gmail.com, ramazan.kose@dpu.edu.tr

ABSTRACT

With increasing population and developing industry, energy needs are increasing day by day. Efficiency of systems should be increased and system costs should be reduced to ensure the efficient use of available resources and the amount of energy required for the economy. In this study, the exergy costs of air (ASHP) and ground (GSHP) source heat pumps to be used for heating of a house selected in the province of Kütahya, are investigated theoretically according to the different cooling fluids and 3 different condenser outlet temperatures during the heating season. In the study, R12, R22, R134a, R410A, R507A and R600a refrigerants in heat pumps were examined. In heat pumps, it was observed that the exergy costs of the components decreased as the heat sink temperature increased. In the air and ground source heat pump, the highest exergy costs are seen in the compressor and evaporator, while the lowest exergy costs are seen in the fan coil circuit. In components, the cost of exergy decreased as the condenser outlet temperature decreased. Exergy costs can be reduced by improving system components with low exergy efficiency.

Keywords: Heat pump, refrigerant, exergy cost.

1. INTRODUCTION

Turkey’s energy towards the outside is dependent fact that a large share of the home heating primary energy supply housing may run out of fossil fuels for heating, and are not harmful to the environment, which makes air-conditioning systems such as heat pumps attractive. Heat pumps can use the heat of sources such as air, water, earth, and can be used in hybrid systems such as solar powered ground source heat pumps (SAGSHP). There are many studies on heat pumps and exergy costs in the literature.

2. MATERIAL AND METHOD

In this study, heat loads were calculated by TS825 method in the heating season of a house selected in Kütahya province and the performance of heating of this house with air and ground source heat pump was investigated theoretically according to the months. In addition, 6 different refrigerants (R12, R22, R134a, R410A, R507A, R600a) and 3 different condenser temperatures (T = 30, 35 and 40 °C) were calculated separately for the heat pump. Exergy costs of the system elements have been obtained as a result of the calculations.

2.1. Thermodynamic Analysis

With the help of the second law of thermodynamics, exergy balance can be examined for the design and analysis of thermal systems. The exergy balance of a continuous continuous continuous system can be expressed as follows;

\[ E_{in} + E_Q = E_{out} + W + I \]  

(1)

Expressions in the above equation;

\[ E_{in} = \sum m_{in} e_{in} \]  

(2)
The general exergy equation is expressed by the following equation:
\[ E = (h - h_0) - T_0(s - s_0) + e_{ch} + (V_0^2 / 2) + gZ_0 \]  
(5)

### 2.2. Exergy Cost Analysis

With the help of the cost analysis, the cost of the exergy flows resulting from the interactions of the thermal system with the environment or inefficiencies experienced in the system are measured (Morrone, 2014). For this purpose, exergy costs can be determined as follows (Lazzaretto, Tsatsaronis, 2006);

\[ C_i = c_i \cdot E_i = c_i \cdot m_i \cdot e_i \]  
(6)

\[ C_o = c_o \cdot E_o = c_o \cdot m_o \cdot e_o \]  
(7)

\[ C_w = c_w \cdot W \]  
(8)

\[ C_q = c_q \cdot \dot{E}_q \]  
(9)

In these equations, \( \dot{E}_i, \dot{E}_o, W \) ve \( \dot{E}_q \) represents respectively; exergy inlet, exergy outlet, exergy transfer by work and heat transfer. Costs per unit exergy are represented by \( c_i, c_o, c_w, c_q \) expressions. \( \dot{C}_i, \dot{C}_o, \dot{C}_w, \dot{C}_q \) expressions are cost flows.

### 3. RESULTS AND DISCUSSIONS

When the COP values of the GSHP system are examined, it is seen that as the source temperature increases, the COP increases. The COP of the heat pump ranges between 2.48 and 3.84. The highest COP value is reached in October. The lower the condenser outlet temperature, the higher the COP. Heat pumps using R134a and R600a refrigerants have the highest COP value. The lowest COP values are shown in heat pumps using R507A and R410A refrigerant.
When the COP values of the GSHP system are examined, it is seen that a graph which is almost inversely proportional to the COP of the heat pump is obtained. This causes the fan coil fan to remain stable despite the reduced heat load of the selected house. The COP\textsubscript{sys} values for GSHP range from 1.75 to 2.05.

In the GSHP system, the exergy cost of the compressor decreases with the power consumed by the compressor in the months when the heat load of the house decreases. The exergy cost of the compressor varies between 0.06 - 0.34 $ / h. As the condenser outlet temperature decreases, the cost of exergy decreases. Compressor exergy cost is the highest in systems using R507A, while R134a is the lowest in the systems used.

In the GSHP system, the cost of the exergy of the condenser decreases as the source temperature increases. As the condenser outlet temperature decreases, the exergy cost of the condenser decreases. The exergy cost of the condenser
varies between 0.022 - 0.115 $ / h. The cost of the condenser exergy is the highest in the systems using R410A, while the R600a is the lowest in the systems used.

In the GSHP system, the exergy cost of the evaporator decreases as the source temperature increases. The exergy cost of the evaporator varies between 0.067 - 0.258 $ / h. The exergy cost of the evaporator for different fluids is close to each other.

In the GSHP system, the exergy cost of the expansion valve decreases as the source temperature increases. As the condenser outlet temperature decreases, the exergy cost of the expansion valve also decreases. The exergy cost of the expansion valve varies between 0.034 - 0.246 $ / h. Expansion valve exergy cost is the highest in systems using R507A, while R22 is the lowest in the systems used.
When ASHP COP values are examined, it is seen that as the source temperature rises, COP increases. The COP of the heat pump varies between 2.64 and 4.31. The highest COP value is reached in May. The lower the condenser outlet temperature, the higher the COP. Heat pumps using R134a and R600a refrigerants have the highest COP value. The lowest COP values are shown in heat pumps using R507A and R410A refrigerant.

![Figure 8. COP values of ASHP system](image)

When the COP values of the ASHP system are examined, it is seen that a graph is almost inversely proportional to the COP of the heat pump. This causes the fan coil fan to remain stable despite the reduced heat load of the selected house. COP$_{sys}$ values for ASHP range from 1.82 to 2.50.

![Figure 9. Exergy cost of ASHP compressor](image)

In the ASHP system, the exergy cost of the compressor decreases with the power consumed by the compressor in the months when the heat load of the house decreases. The exergy cost of the compressor varies between 0.05 - 0.32 $ / h. As the condenser outlet temperature decreases, the cost of exergy decreases. Compressor exergy cost is the highest in systems using R507A, while R134a is the lowest in the systems used.

![Figure 10. Exergy cost of ASHP condenser](image)
In ASHP system, the exergy cost of condenser decreases as the source temperature increases. As the condenser outlet temperature decreases, the exergy cost of the condenser decreases. The exergy cost of the condenser varies between 0.018 - 0.094 $ / h. The cost of the condenser exergy is the highest in the systems using R410A, while the R600a is the lowest in the systems used.

![Figure 11. Exergy cost of ASHP evaporator](image)

In ASHP system, the exergy cost of the evaporator decreases as the source temperature increases. The exergy cost of the evaporator varies between 0.255 - 0.623 $ / h. The exergy cost of the evaporator for different fluids is close to each other.

![Figure 12. Exergy cost of ASHP expansion valve](image)

In ASHP system, the exergy cost of the expansion valve decreases as the source temperature increases. As the condenser outlet temperature decreases, the exergy cost of the expansion valve also decreases. The exergy cost of the expansion valve varies between 0.027 - 0.265 $ / h. Expansion valve exergy cost is the highest in systems using R507A, while R22 is the lowest in the systems used.

**REFERENCES**


ABSTRACT
Inconel 718 superalloy one of the most used material in the aerospace industry due to its superior properties at elevated temperatures. However, the machinability of this material is very hard due to hard carbide particles in its composition and work hardening tendency. Therefore, it is tried to improve the machinability of Inconel 718 by using different methods. In addition to improving the machinability, environmental considerations must be taken into account. In this study, two different methods have been used with the aim of increasing the machinability. One of these methods is to increase the wear resistance of the cutting tools using hard coatings. Another method used is the minimum quantity of lubrication system (MQL) as cutting fluid for machining process. Innovative nanocomposite TiAlSiN/TiSiN/TiAlN is used as a hard coating on carbide cutting tools. In addition, environmentally friendly vegetable oil is used as a coolant in the MQL system. As a result of applied milling tests, cutting performance improved in terms of cutting force and surface roughness. According to optical microscope images taken from worn tools, it has been determined that effective wear mechanisms are abrasion and adhesion.

Keywords: MQL, hardmilling, Inconel 718, nano-composite hard coatings, carbide cutting tool, sustainable

1. INTRODUCTION
Inconel 718 is a hard material. It is eager to harden during work and it has low thermal conductivity. These properties make it a difficult-to-cut metal[1]. Cutting difficult-to-cut metals require a lot of force than cutting soft metals. Because of the high cutting forces, a lot of heat is generated and tool inserts are worn out rapidly. As a result, bad finish quality, low dimensional accuracy are obtained and tool costs increase. In order to prevent rapid wear of tools, different methods are applied like using coolants during processing and applying hard coatings on inserts.

Applying a liquid coolant between cutting tool-work material interface is the most common method to get good finish results. One of the most used cooling method is flooding. With this method, coolant is applied at high flow rates, nearly 400 L/h[2]. This method is useful in terms of heat removing and chip cleaning. But despite its advantages, this method has many disadvantages. First of all, conventional coolants that are being used for this method may cause allergic reactions on human body. On the other hand, because of the high flow rate, using and disposing cost too much. Furthermore, this method is not environmentally friendly and also has harmful effects on humans[3–5]. To overcome these issues, MQL technique is a good alternative. With MQL, very low flow rates are being used comparing with flooding. In this method, very low flow rates are used comparing with flooding. Ranging from 50 mL/h to2l/h of coolant is sprayed on workpiece during processing[3].There are plenty of works that show despite its low flow rate, MQL provides good surface finish, low cutting forces and long tool life[4, 6, 7].
Applying hard coatings on cutting tools is another way to increase tool life. With coatings, cutting tools gain various properties like being harder, more oxidation resistant and more chemically stable which make process more efficient. TiN coating was a common application because it was hard and wear resistant[8]. But it shows low performance at high temperature processes[8]. Then, researches showed that adding elements like Al and S to the TiN makes it harder, more oxidation resistant and more chemically stable[9]. Nanocomposite coatings based on TiAlSiN is preferred for cutting hard metals for its chemical stability at high temperatures[10]. TiSiN and TiAlSiN both show great resistance to plastic deformation[11]. Furthermore, TiAlSiN/TiSiN/TiAlN nanocomposite coating is showed to provide good cutting results. This coating attracted attention due to its chemical stability, low residual stress level and amazing work temperature[12]. It is showed to provide longer tool life and less rough surface[13], prevent flaking and peeling [10].

In this work, face milling operation of superalloy Inconel 718, which is a difficult-to-cut material is studied and compared under dry conditions and MQL conditions. It is thought that this work makes contribution to literature; because of the lack of face milling studies of Inconel 718 in literature. Furthermore, this is one of the pioneer works in terms of applying vegetable oil in MQL for face milling of Inconel 718.

2. EXPERIMENT

2.1. Material

Chemical composition and mechanical properties of Inconel 718 are given in Table 1 and Table 2.

| Table 1. Limiting Chemical Composition of Inconel 718, wt. %[13] |
|---------------------|-----------------|
| C                   | 0.046           |
| Si                  | 0.16            |
| Mn                  | 0.18            |
| S                   | 0.008           |
| P                   | 0.011           |
| Ni                  | 51.34           |
| Cr                  | 17.89           |
| Al                  | 0.57            |
| Ti                  | 0.99            |
| Nb                  | 5.04            |
| Mo                  | 3.14            |
| Cu                  | 0.031           |

<table>
<thead>
<tr>
<th>Table 2. Mechanical Properties of Inconel 718[13]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tensile Strength</td>
</tr>
<tr>
<td>Yield Strength</td>
</tr>
<tr>
<td>Elongation</td>
</tr>
<tr>
<td>Hardness</td>
</tr>
</tbody>
</table>
2.2. Machine and Tools

For experiments, following tools were used:

- Falco VMC 855-B CNC three-axis vertical machining center.
- SANDVIK R390-025A25-11L tool holder
- SANDVIK R390-11 T3 08M-KM H13A model cutting tools for metal cutting. The tools are coated with TiAlSiN/TiSiN/TiAlN multilayer hard coating.
- Vegetable oil as MQL coolant
- Werte Mikro STN 25 as MQL system
- Kistler multicomponent dynamometer up to 10kN type 9257B
- Mitutoyo Surftest SJ-310 for measuring surface roughness
- Stereo zoom optical microscope with imaging software

2.3. Cutting Tests

For cutting tests, face milling operation was performed on Inconel 718. The tests were performed with one tool mounted on the tool holder because it desired to see the behavior of only one tool. The cutting parameters are given on Table 3.

Table 3. Cutting parameters for face milling

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cutting Speed (Vc)</td>
<td>75 m/min</td>
</tr>
<tr>
<td>Feed Rate (fz)</td>
<td>0.05 mm/tooth</td>
</tr>
<tr>
<td>Axial Depth of Cut (ap)</td>
<td>0.1 mm</td>
</tr>
<tr>
<td>Radial Depth of Cut (ae)</td>
<td>15 mm</td>
</tr>
</tbody>
</table>

First, the tests were performed for dry cutting. Then same tests were performed using vegetable oil as coolant. For cutting with coolant, MQL system is adjusted to spray 100 ml of vegetable oil per hour. For each pass, cutting forces were measured and for every two passes, surface roughness and tool wear values were measured. Tool life was chosen to end when the notch wear reaches 0.2 mm.

3. RESULTS AND DISCUSSION

3.1. Cutting Forces

Average cutting force (F_R) values are given in Figure 1. The fluctuation in force waves are observed because of the nature of milling process[5]. In this figure, because of the lubricating effect of vegetable oil, the decrease in the fluctuation of the cutting forces is seen when we apply vegetable oil instead of performing dry cutting. This provided 30% reduction in cutting forces.
3.2. Surface Roughness

Surface roughness is a usual indicator that shows the cutting quality[5]. Comparing with dry cutting, it is not observed to happen a significant change in surface roughness when we use vegetable oil. The comparison can be observed in Figure 2. A reduction is observed around 7%. Either way, a smooth surface was obtained, but a smoother surface was obtained when vegetable oil is applied. Even if it is not significant, the effect of vegetable oil is observed in terms of surface roughness.
3.3. Tool Wear and Tool Life

It is observed that using vegetable oil decreases notch wear comparing with dry cutting. The decrease is around 23%. Because of the decrease in cutting forces, decrease in temperature at cutting zone is achieved. So, less tool wear and longer tool life is gained. It is also thought that the chip removing effect of the coolant that is used provided positive effects on tool life.
4. CONCLUSIONS

In this study, it is showed that the anti-wear performance of TiAlSiN/TiSiN/TiAlN nanocomposite coating in face milling of Inconel 718 for dry cutting and cutting with vegetable oil using MQL. The results are discussed in terms of cutting forces, surface roughness and tool wear. The results can be summarized as follows:

1) As a coolant, vegetable oil has a positive effect on cutting forces. Cutting forces were decreased up to 30%.
2) Vegetable oil is useful in increasing tool life. 23% percent reduction is achieved in tool wear for the same cutting distance with dry cutting.
3) In terms of surface roughness, vegetable oil is observed not to have a significant reduction. Anyway, 7% of reduction in average surface roughness is achieved because of the lubricating effect of vegetable oil.

5. REFERENCES


EXAMINATION OF THE EFFECT OF CRYOGENIC HEAT TREATMENT ON NANO-LAYERED CARBIDE CUTTING TOOLS IN MILLING INCONEL 718 SUPERALLOY.

Bilal Kursuncu1, Cemre Can Alkan1

1Bartin University, Engineering Faculty, Mechanical Engineering Department, Turkey

bkursuncu@bartin.edu.tr, ccalkan@bartin.edu.tr

ABSTRACT

Inconel 718 superalloy is frequently used due to its superior mechanical properties in aerospace engine parts and several parts of nuclear plants due to its superior mechanical properties such as high oxidation resistance, corrosion resistance at elevated temperatures. However, the machinability of Inconel 718 is very difficult due to its low thermal conductivity, work hardening tendency and occurrence of hard carbide particles in its chemical composition. Due to these properties of the Inconel 718, the cutting tools are worn rapidly in the milling process. Different methods are used to increase the machinability of the Inconel 718 superalloy. The most commonly used method is that the cutting tool material is coated with abrasion resistant hard coatings. In this study, the carbide cutting tools were coated with nano-layer AlTiN/TiN hard coating and then cryogenic heat treatment was applied to cutting tool due to increasing the abrasion resistance. Milling tests were carried out for the cutting parameters specified in the light of the manufacturer's catalog data and past studies. In consequence of milling operations, up to 25% improvement in tool life has been achieved with cryogenic heat-treated cutting tools. In addition, a smoother surface was obtained by milling with cryogenic heat-treated cutting tools.

Keywords: Hard milling, Inconel 718, nano-layer hard coatings, carbide cutting tool, sustainable manufacturing

1.INTRODUCTION

Inconel 718 is a difficult-to-cut material due to its superior mechanical properties at elevated temperatures [1]. These properties lead to high cutting forces and temperatures during machining at the cutting zone and causes rapid wear of cutting tools and high surface roughness values. For these reasons, the Inconel 718 superalloy is very difficult and costly to process.

So as to improve the wear resistance of cutting tools, hard film coating technique is a common process. TiN coating is being applied for a long time for its good hardness, wear resistance and chemical stability[2, 3]. But TiN coating remains insufficient for hard materials and high temperature after 500 °C applications[2, 3]. So, the urge for new coatings led researchers to TiN coatings with Al addition. Adding Al to TiN coating, researchers got better results in terms of oxidation resistance and tool durability[4]. In another study, it is stated that increase in Al proportion provides harder tools[5]. Furthermore, nanolayer coatings were developed. It is shown that nanolayer coatings provide better properties than singlelayer coatings[6]. One of the nanolayer coatings is AlTiN/TiN coating. This coating is a good choice for machining of hard-to-cut materials, and it is also useful for temperatures up to 1100 °C[6].
2. EXPERIMENT

2.1. Material

Chemical content and some mechanical properties of workpiece material are given in Table 1 and Table 2.

Table 1. Limiting chemical content of Inconel 718, wt. %[1]

<table>
<thead>
<tr>
<th>Element</th>
<th>C</th>
<th>Cr</th>
<th>Si</th>
<th>Al</th>
<th>Mn</th>
<th>Ti</th>
<th>S</th>
<th>Nb</th>
<th>Mo</th>
<th>Ni</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0.046</td>
<td>17.89</td>
<td>0.16</td>
<td>0.57</td>
<td>0.18</td>
<td>0.99</td>
<td>0.008</td>
<td>5.04</td>
<td>3.14</td>
<td>51.34</td>
</tr>
</tbody>
</table>

Table 2. Mechanical properties of Inconel 718[1]

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tensile Strength</td>
<td>1032 MPa</td>
</tr>
<tr>
<td>Yield Strength</td>
<td>1072 MPa</td>
</tr>
<tr>
<td>Elongation</td>
<td>0.14%</td>
</tr>
<tr>
<td>Hardness</td>
<td>54 HRc</td>
</tr>
</tbody>
</table>

2.2. Machine and Tools

For experiments, following tools were used:

- Three-axis vertical machining center (Falco VMC 855-B CNC).
- SANDVIK R390-025A25-11L tool holder
- SANDVIK TiN coated R390-11 T3 08M-KM H13A model cutting tools for metal cutting.
- Kistler multicomponent dynamometer up to 10kN type 9257B
- MitutoyoSurftest SJ-310 for evaluating surface roughness values
- Stereo zoom optical microscope with imaging software

2.3. Cutting Tests

For cutting tests, face milling operation was performed for two different tools and three different feed rates on Inconel 718. The tests were performed with dry conditions and one tool mounted on the tool holder because of the purpose of this is to control the cutting comportment of only one cutting tool.

The cutting parameters are given on Table 3.
Table 3. Cutting parameters for face milling

<table>
<thead>
<tr>
<th>Cutting Speed (V_c)</th>
<th>25 m/min</th>
</tr>
</thead>
<tbody>
<tr>
<td>Feed Rate (f_z)</td>
<td>0.05 mm/tooth</td>
</tr>
<tr>
<td>Axial Depth of Cut (a_p)</td>
<td>0.2 mm</td>
</tr>
<tr>
<td>Radial Depth of Cut (a_e)</td>
<td>15 mm</td>
</tr>
</tbody>
</table>

The tests were begun to be performed for AlTiN/TiN coated tools for the lowest feed rate without cryogenic heat treatment. Then same tests were performed for tools with cryogenic heat treatment for the same feed rate. Then this pattern was applied for different feed rates. The tests were performed until tools reach 0.3 mm of wear.

3. RESULTS AND DISCUSSION

Surface roughness values were recorded for each pass. The results are given in Figure 1 and Figure 2.

3.1. Tool Life

In Figure 1, it is clearly seen that cryogenic heat treatment positively effects tool life[7]. Longest cutting distance is achieved with cryogenic heat treated tool for 0.075 mm/tooth feed rate. It is followed by heat treated tools for other feed rates. Even if it is not heat treated, cutting with 0.1 mm/tooth feed rate is observed to give good results. The lowest tool life is achieved for not heat treated tool with 0.05 mm/tooth feed rate.

![Graph showing tool life in terms of cutting length for different feed rates](image)

**Figure 1.** Tool life in terms of cutting length for different feed rates
3.2. Surface Roughness

From Figure 2, the surface roughness values can be observed. The heat treated tools showed better results compared with untreated tools as expected [7]. Only untreated tool with 0.075 mm/tooth feed rate seems to have better surface roughness than any other case. But considering high amount of wear and low cutting length of this case, it is observed that heat treated tools show better results in general. Checking heat treated tools, the lowest roughness value is observed for the case when feed rate is 0.05 mm/tooth. This is also thought to occur because of short cutting length of this case. It can be said that heat treatment on tools observed not to have a noteworthy effect on surface roughness. It has effect on tool wear and provide longer tool life; but surface roughness development occurs at similar rate.

![Figure 2. Average surface roughness for different cutting conditions](image)

4. CONCLUSION

In this study, the cutting performances of AlTiN/TiN coated tools were compared for the cryogenic heat treated and not treated conditions. Additionally, feed rates were changed to also see the effects of feed rate on cutting performance and tool life. The results can be summarized as follows:

- For the same feed rates, tools that are not heat treated shows low performance when compared to heat treated tools.
- The longest tool life and the longest cutting distance is observed with cryogenic heat treated tool for 0.075 mm/tooth feed rate. But the lowest surface roughness value is observed for a non heat treated tool with 0.075 mm/tooth feed rate. This can be said to happen because of short cutting length of this case. And it can be concluded that whether the tool is heat treated or not, surface roughness development occurs at similar rates.
- The shortest tool life and the shortest cutting distance is observed with not heat treated tool for 0.05 mm/tooth feed rate. On the other hand, the highest surface roughness value is observed for an untreated tool.
with 0.05 mm/tooth. It can be said that 0.05 mm/tooth feed rate is not operational for face milling of Inconel 718.

- It can be said that optimum conditions are observed by using cryogenic heat treated tool with 0.075 mm/tooth feed rate.

5. REFERENCES


DESIGN AND EXPERIMENTAL ANALYSIS OF HEAT STORAGE AIR COLLECTORS

Ayse Bilgen Aksoy1, Burcu Gulmez2

1Manisa Celal Bayar University Energy Systems Engineering, 45400 Manisa, Turkey
2Manisa Celal Bayar University Mechanical Engineering, 45150 Manisa, Turkey
1aysebilgen.aksoy@cbu.edu.tr, 2burcugulmez@gmail.com

ABSTRACT

In addition to the use in drying ovens, collectors are system components used in air conditioning systems and hot water supply [1]. The air solar collectors are made of transparent cover, absorbent plate and casing used as a permeable surface and are widely used in such systems. In this study, energy analysis of an air solar collector which is supported by experimental data is performed in order to increase collector efficiency. The collector is designed with dimensions of 960x2250 mm and two fins are used to increase the effect of the radiation. Cast iron blocks with dimensions of 40x40 mm were placed under the absorbent surface in order to ensure a longer and more stable heat distribution by thermal storage. Temperature and velocity measurements were made at the entrance and exit of the collector. Thermodynamic properties were determined by using EES software and energy efficiency was determined at different inlet temperatures of collector. Collector efficiency was calculated to be 69.23% on average.

Keywords: collector, radiation, energy efficiency

1. INTRODUCTION

Due to the rapid development of civilization and limited fossil fuel reserves, the interest and need for renewable energy is increasing day by day [1]. Similar to basic needs such as nutrition and housing, energy is one of the basic needs of humanity. The global increase in energy demand and environmental pollution motivates relevant research and technological investments to improve energy efficiency and production. Among renewable energy sources, solar energy is the most accessible and the most accessible as well as being an environmentally friendly and unlimited energy source. Compared to other fossil-derived energy sources such as coal and oil, solar energy is considered a satisfactory source of energy because it is a reliable and clean source of energy [2].

With the increasing population and demand, solar energy used for different purposes for hundreds of years by the development of technology. The simplest system for heat energy conversion is the collectors. The solar collector is a heat exchanger that absorbs the radiation energy from the sun, converts it into internal energy and transmits this energy to the working fluid. A solar collector differs from traditional heat exchangers in various aspects [3]. There are two types of solar collectors: solar collectors that can be focused and fixed. The non-focusable solar collectors have a flat surface and do not focus on sunlight. Focusable solar collectors generally have a concave surface, only benefit from direct radiation and can provide high temperature. Focused collectors have to see the sun to work so that higher efficiency is obtained when used in areas rich with sunlight [4].

Solar collectors can be classified according to the working fluid they use. The most commonly used working fluid is air and water. Solar air collectors collect the radiation from the sun and transfer it to the air used as the working fluid. In this process, air entering the collector, conditioned by the fan, absorbs heat energy as it passes through the channel between the absorbent surface and the permeable upper surface. This heat energy which is taken into the air is sent to the environment where the process will be carried out.

Solar energy collectors have an important place in solar energy systems due to their minimum material usage. In addition, direct use of air as a working substance reduces the number of components required in the system [3]. Solar air collector (SAC) has the advantages of simple construction, convenient installation and easy maintenance. It is advantageous because of no leakage, congestion and freezing problems during application [5]. The hot air from the solar collector is widely used in building heating, hot water supply, drying of agricultural products and in different craft and industrial applications in low and medium temperatures [6].

The collectors designed to collect solar radiation and transfer them to the vehicle have different properties, but the overall system structure is the same; transparent cover, absorbent plate, tool fluid flow channel and isolation material. The incoming radiation passes through the transparent cover, while some of it is absorbed by the absorbent plate, and some of it is reflected. The heat retained by the plate is transmitted to the working fluid passing through the flow channel and transmitted to the desired location. The overall appearance of the collectors is shown in Figure 1.1.
In literature, increasing the efficiency of collector is based on increasing heat transfer area and reducing heat losses. In solar collectors, it was determined that the artificial roughness increased the heat transfer from the collector plate to the air. For this reason, general trend in the literature is based on the existence of baffles [8]. Surface roughness can be achieved by various methods such as sandblasting, machining, casting, forming, welding lines and fixing thin circular wires along the surface. Also, the efficiency of solar energy collectors has been increased with the help of fins in different geometries such as plates, blinds, convex window sashes and various wings patterns such as wavy and similar placed on the collector absorber surface [9]. In order to increase the efficiency of the collector, collector designs have been made with solvent vacuum [10] and composite absorber surface [11].

The success of solar energy applications is directly proportional to the efficiency of the collectors [6]. By decreasing the collector surface areas and increasing the output temperatures, it is thought that collector efficiency will be increased and usage in industrial area will be extended. In this study, it is desirable to design a collector which can be used with drying (dehumidification) furnaces, for heating or preheating of oven drying air, if desired, as an individual air conditioning system or as a support system.

2. MATERIAL

The test setup generally consists of the absorber plate, permeable cover, casing and auxiliary units forming a solar collector. The absorber surface in the collector was obtained by painting the sheet plate with selective matt paint. Five 40x40 mm cast iron blocks were placed under the absorber surface in order to store and stabilize the heat. In this way, it is aimed to obtain higher temperatures by condensing heat. Using plane wings, the radiation is condensed onto the absorbent plate. The collector used in the experiments is shown in Figure 2.1.
The collector is manufactured in standard dimensions, 200 mm high, 960 mm wide and 2250 mm long. The collector case is manufactured from 1 mm thick galvanized sheet and for the transparent cover, heat resistant, tempered glass is used. As the insulation material, glass wool with a heat transfer coefficient of $k = 0.035 \text{ W/m-K}$ and withstand temperatures up to $150 \, ^\circ\text{C}$ is used. Collector specifications are given in Table 2.1.

**Table 2.1. Collector specifications**

<table>
<thead>
<tr>
<th>Permeable top layer</th>
<th>Tempered glass (1 mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case size</td>
<td>200 x 960 x 2250 mm</td>
</tr>
<tr>
<td>Insulation material</td>
<td>Glass wool</td>
</tr>
<tr>
<td>Absorbent plate</td>
<td>Black matt painted sheet</td>
</tr>
<tr>
<td>Case material</td>
<td>Galvanized sheet</td>
</tr>
</tbody>
</table>

A piston is placed under the frame to change the angle of inclination of the collector case in order to reduce the negative effect of the change in the angle of the solar radiation on the variable position and seasonal variations. The collector wings are manufactured to move $90^\circ$. Tempered glass specifications are given in Table 2.2.

**Table 2.2. Tempered glass properties**

<table>
<thead>
<tr>
<th>Thickness (mm)</th>
<th>Refractive index</th>
<th>$\varepsilon$ Solar radiation 0,2-4,0 $\mu$m</th>
<th>$\varepsilon$ Emitted radiation 3,0-50 $\mu$m</th>
<th>Thermal resistance (C)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tempered Glass</td>
<td>3,2</td>
<td>1,52</td>
<td>0,79</td>
<td>0,02</td>
</tr>
</tbody>
</table>

During the experiments, the surface temperature and heat losses of the collector were visualized by the thermal imaging camera of FLIR T420. Examples of the obtained images are given in Figure 2.2 and Figure 2.3.
Air velocity was measured by anemometer. The temperatures were measured with PT type thermocouple at the input and output of the collector and the results were transferred to the computer with PLC software. A screenshot of the software is given in Figure 2.4.

2.1. PROGRAMMABLE LOGIC CONTROLLER

The PLC (Programmable Logic Controller) controller system is used to measure the physical quantities in the collector. While PLC surveys on the site, SCADA (Supervisory Control and Data Acquisition) is used for remote monitoring and monitoring of alarm conditions. In the measurements made in the collector, physical and chemical quantities to be used
in energy analysis were determined. In addition, PLC based control system is designed to be capable of remote access. Elements of the PLC system are given in Table 2.1.1.

![PLC screen display](image)

Figure 2.4. PLC screen display

The blue line in the screen display is the temperature change curve of the collector. These curves also form the graph of the time of the collector outlet temperature.

<table>
<thead>
<tr>
<th>Table 2.1.1. Elements of the PLC system</th>
</tr>
</thead>
<tbody>
<tr>
<td>CPU 1214C, Compact CPU, DC/DC/DC/DC</td>
</tr>
<tr>
<td>Analog Input, SM 1231 RTD</td>
</tr>
<tr>
<td>Power Module PM1207</td>
</tr>
<tr>
<td>Memory Card</td>
</tr>
<tr>
<td>500x350x270mm Standard Type Polyester Panel</td>
</tr>
<tr>
<td>G120C</td>
</tr>
<tr>
<td>Diameter-6 PT-100 4M Cabled Thermocouple</td>
</tr>
<tr>
<td>3 Phase 500 Watt asynchronous motor</td>
</tr>
</tbody>
</table>

3. METHOD

Depending on the inlet and outlet temperature of the fluid, useful energy available from the collector can be calculated theoretically by specific heat formula [12]:

\[
Q_u = \dot{m}c_p(T_c - T_g)
\]  

(1)

The mass air flow rate (kg/s) in the equation is \( \dot{m} \), \( c_p \) refers to the specific heat of air at constant pressure (J/kgK). \( T_g \) and \( T_c \) (K) are the incoming and outgoing air temperatures, respectively.

The most important feature of the collector is efficiency. Efficiency can be defined as the ratio of available energy collected from the collector to the solar energy coming to the collector surface within the same period [13]. And efficiency can be calculated by equation (2).

\[
\eta = \frac{Q_u/A_c}{G_T}
\]  

(2)

The expression \( \eta \) in the equation refers to efficiency. \( G_T \) is the instantaneous irradiance value (W/m²) and \( A_c \) is the collector area. If Equation 2 is rearranged, the instantaneous efficiency can be obtained from the collector;

\[
\eta = \frac{\dot{m}c_p(T_c-T_g)}{G_T A_c}
\]  

(3)
can calculable using equation (3). The $G_T$ instantaneous irradiance values in the equations were obtained by entering the collector coordinates from the site prepared by European Commision [15].

4. CONCLUSIONS AND RECOMMENDATIONS

In literature, the unloaded plate temperature can reached to 80-85 °C [14]. By the help of the wings, the condensed radiation stored within iron blocks and the temperature is obtained above 95 °C.

The experiments were repeated in September and October, especially in the autumn months. In September, measurements were made on a day when the temperature was 33 °C, $T_{\text{inlet}}$, $T_{\text{outlet}}$ temperatures and hourly velocities were obtained. The $c_p$ values were taken from the EES software as a function of the temperature as the air was considered ideal gas. Hourly radiation was taken from Pyres [15]. Using the equation (3), the useful heat and efficiency values that can be obtained from the collector are calculated on an hourly basis and given in the table. For 0.00283 mm² pipe area, the results that obtained from the experiment made on 19.09.2018 are given in Table 4.1.

<table>
<thead>
<tr>
<th>Hours</th>
<th>$\rho$ (kg/m³)</th>
<th>$A_{\text{col}}$ (m²)</th>
<th>Velocity (m/s)</th>
<th>Air Flow Rate (kg/s)</th>
<th>$c_p$ (J/kg.K)</th>
<th>$T_{\text{inlet}}$ (°C)</th>
<th>$T_{\text{outlet}}$ (°C)</th>
<th>$\Delta T$</th>
<th>Heat (W)</th>
<th>Radiation (W/m²)</th>
<th>Eff. (η)</th>
<th>% Eff.</th>
</tr>
</thead>
<tbody>
<tr>
<td>08:00</td>
<td>1.132</td>
<td>2.16</td>
<td>1</td>
<td>0.00320</td>
<td>1007</td>
<td>24</td>
<td>40</td>
<td>16</td>
<td>51.54280</td>
<td>595</td>
<td>0.040</td>
<td>4.01</td>
</tr>
<tr>
<td>09:00</td>
<td>1.064</td>
<td>2.16</td>
<td>1.2</td>
<td>0.00361</td>
<td>1008</td>
<td>26</td>
<td>60</td>
<td>34</td>
<td>123.6614</td>
<td>775</td>
<td>0.074</td>
<td>7.38</td>
</tr>
<tr>
<td>10:00</td>
<td>1.048</td>
<td>2.16</td>
<td>1.2</td>
<td>0.00355</td>
<td>1009</td>
<td>28</td>
<td>65</td>
<td>37</td>
<td>132.6806</td>
<td>905</td>
<td>0.068</td>
<td>6.78</td>
</tr>
<tr>
<td>11:00</td>
<td>1.033</td>
<td>2.16</td>
<td>1.2</td>
<td>0.00350</td>
<td>1009</td>
<td>30</td>
<td>70</td>
<td>40</td>
<td>141.3855</td>
<td>934</td>
<td>0.070</td>
<td>7.00</td>
</tr>
<tr>
<td>12:00</td>
<td>1.015</td>
<td>2.16</td>
<td>1.3</td>
<td>0.00373</td>
<td>1009</td>
<td>31</td>
<td>75</td>
<td>44</td>
<td>165.5485</td>
<td>900</td>
<td>0.085</td>
<td>8.51</td>
</tr>
<tr>
<td>13:00</td>
<td>0.998</td>
<td>2.16</td>
<td>1.4</td>
<td>0.00395</td>
<td>1010</td>
<td>32</td>
<td>82</td>
<td>50</td>
<td>199.3986</td>
<td>791</td>
<td>0.117</td>
<td>11.67</td>
</tr>
<tr>
<td>14:00</td>
<td>0.976</td>
<td>2.16</td>
<td>1.5</td>
<td>0.00414</td>
<td>1011</td>
<td>32</td>
<td>90</td>
<td>58</td>
<td>242.6008</td>
<td>626</td>
<td>0.179</td>
<td>17.94</td>
</tr>
<tr>
<td>15:00</td>
<td>0.976</td>
<td>2.16</td>
<td>1.5</td>
<td>0.00414</td>
<td>1011</td>
<td>33</td>
<td>90</td>
<td>57</td>
<td>238.4181</td>
<td>418</td>
<td>0.264</td>
<td>26.40</td>
</tr>
<tr>
<td>16:00</td>
<td>0.976</td>
<td>2.16</td>
<td>1.5</td>
<td>0.00414</td>
<td>1010</td>
<td>32</td>
<td>90</td>
<td>58</td>
<td>242.3609</td>
<td>201</td>
<td>0.558</td>
<td>55.82</td>
</tr>
<tr>
<td>17:00</td>
<td>0.992</td>
<td>2.16</td>
<td>1.4</td>
<td>0.00393</td>
<td>1010</td>
<td>31</td>
<td>84</td>
<td>53</td>
<td>210.1765</td>
<td>21</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>18:00</td>
<td>1.009</td>
<td>2.16</td>
<td>1.3</td>
<td>0.00371</td>
<td>1008</td>
<td>30</td>
<td>78</td>
<td>48</td>
<td>179.3529</td>
<td>0</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>19:00</td>
<td>1.064</td>
<td>2.16</td>
<td>1.2</td>
<td>0.00361</td>
<td>1008</td>
<td>27</td>
<td>60</td>
<td>33</td>
<td>120.0243</td>
<td>0</td>
<td>-</td>
<td>-</td>
</tr>
</tbody>
</table>

In the measurements made by operating the suction fan placed in the collector outlet, the inlet and outlet temperatures were determined as 26 °C and 72 °C respectively and the output rate was 9.4 m/s. By increasing the output rate, the efficiency increased to 72%.

The experiment made without wings on a different day, at the same temperature, $T_{\text{outlet}}$ was measured 52 °C. The instantaneous efficiency decreased to 41.97% when the fan was on and down to 20.46% when the fan was off.

When the collector efficiency was increased and the collector surface area was reduced, the system was found to be suitable for industrial and domestic use at low temperatures. Besides the disadvantage of excess weight, cast iron can be used for heat storage due to its low cost and the temperature can be obtained for longer and more stable.

In addition, turbulence in air flow with baffles to be formed on the surface of the collector will also increase the collector efficiency.

In case of using a fan in the system, the speed that provides optimum efficiency can be calculated. Thus, a more efficient collector can be designed.

This study was supported by MCBU Scientific Research Projects.
5. REFERENCES


YAPISAL MÜHENDİSLİK PROBLEMLERİİNİN GÜVENİLİRLIK TEMELLİ TASARIM OPTİMİZASYONU ÜZERİNE KARŞILAŞTIRMALI BİR ÇALIŞMA

Emre Demirci1,4, Ali Rıza Yıldız2

1 Bursa Teknik Üniversitesi, Makine Mühendisliği Bölümü, Bursa, Türkiye
2 Bursa Uludağ Üniversitesi, Otomotiv Mühendisliği Bölümü, Bursa, Türkiye
4 Sorumlu yazar: Emre Demirci, emre.demirci@btu.edu.tr

ÖZET

Anahtar Kelimeler: Güvenilirlik temelli tasarım optimizasyonu, mühendislik tasarımını, güvenilirlik analizi, hibrit gradyan analizi

1. GİRİŞ

RBDO problemlerinin çözümünde belirsizlikleri içeren olasılıksal kısıtların değerlendirilmesi için pek çok farklı yöntem geliştirilmiştir. Bu yöntemler arasındaki bağıltılığı ve verimliliğini sebebiyetle en yaygın kullanılan birinci dereceden güvenilirlik metodu (FORM)’dur (Hasofer ve Lind, 1974; Rackwitz ve Fiessler 1978). FORM temelli RBDO problemlerinin çözümü için geliştirilen yöntemler iç içe grup altında kategorize edilmektedir; çift-döngülü yöntemler, tek-döngülü yöntemler ve ayrırmışlı yöntem (Agarwal ve diğ., 2007; Bichon ve diğ., 2009; Val侦探 ve Schueller, 2010). Sik kullanılan çift-döngülü yöntemlerde iç döngü, genellikle iteratif bir prosedür içeren olasılıksal kısıtlamalar üzerinde güvenilirlik değerlendirilmesidir; optimize edici dış döngü, güvenilirlik analizi için tekrar tekrar iç döngüyü çağran optimizasyon arama sürecini kontrol eder.


Bu çalışmada, hibrit gradyan analizi (HGA) adındaki yeni bir yöntem ile mevcut AMV, CC, MCC ve CGA yöntemleri farklı RBDO problemlerine uygulanarak sonuçları karşılaştırılmıştır.

2. GÜVENİLİRLİK TEMELI TASARIM OPTİMİZASYONU (RBDO)

2.1. Genel RBDO modeli

Tipik bir RBDO probleminin formülasyonu aşağıdaki gibi verilebilir (Du ve Chen, 2004):

\[ \text{burada, } f(d) \text{ amaç fonksiyonudur. } g_i(\mathbf{d}, \mathbf{X}) = \begin{bmatrix} g_{i1}, g_{i2}, \ldots, g_{ip} \end{bmatrix} \text{ rasgele değişken vektörü ve } \mathbf{d} = [d_1, d_2, \ldots, d_n]^T \text{ tasarım değişkeni vektörüne bağlı } i. \text{ kısıtlayıcı fonksiyondur. Optimizasyon problemindeki belirsizlikler rasgele değişkenlerle ele alınır. } \mathbf{d}^l \text{ ve } \mathbf{d}^u \text{ tasarım değişkenlerinin alt ve üst limitleri. } P_{i, \beta} \text{ hedef (yada kabul edilebilir) başarısızlık olasılığı temsil eder. } P_i \{ g_i(\mathbf{d}, \mathbf{X}) \leq 0 \} \text{ kısıtlayıcı fonksiyonunun başarısızlık olasılığını belirlemek için kümulatif dağılım fonksiyonu kullanılır. Kümulatif dağılım fonksiyonunu matematiksel olarak hesaplamak mümkün olmadığı için FORM ile tahmini olarak hesaplanır. FORM ile çözümde rasgele değerler } X- \text{uzayından } U- \text{uzayına dönüştürülür:} \]

\[ U = T(X) \quad (2) \]

Tipik bir RBDO probleminin olasılıksal kısıtlayıcı PMA yönteminde aşağıdaki gibi ifade edilebilir:

\[ g_{i,p} = F_{i_p}^{-1}(\Phi(-\beta_i)) \quad (3) \]

burada, \( F_{i_p} \) ve \( \Phi \), sırasıyla kişit fonksiyonunun kümulatif dağılım fonksiyonu ve standart normal dağılım fonksiyonudur. \( \beta_i \) ise hedef güvenilirlik indeksidir.

2.2. PMA ile Birinci Dereceden Güvenilirlik Analizi

PMA temelli RBDO problemi aşağıdaki gibi yazılabilir:

\[ \text{burada, } F_{i_p} \text{ ve } \Phi \text{, srasıyla kişit fonksiyonunun kümulatif dağılım fonksiyonu ve standart normal dağılım fonksiyonudur. } \beta_i \text{ ise hedef güvenilirlik indeksidir.} \]

Olasılıksal performans fonksiyonu \( G \), aşağıdaki gibi bir optimizasyon problemi ile bulunur:
İteratif PMA algoritmaları, önceden tanımlanan hedef güvenilirlik indeksine bağlı olarak hedef güvenilirlik yüzeyindeki minimum performans hedef noktası \( \mathbf{u}_\| \beta \| \) 1 arar. Bu arama işlemi için, AMV, CC, MCC ve CGA gibi bazı yaklaşımlar geliştirilmiştir.

2.3. Hibrit Gradyan Analizi (HGA) Yöntemi

Literatürdeki MPTP arama yöntemlerinden en dik iniş yöntemi temelli yöntemler bazı yakınsama ve hesaplama yükü sorunlarına sahiptir. Kaos kontrol yöntemleri yakınsama problemlerini büyük oranda ortadan kaldırılmış ancak hesaplama yükü seçilen parametrelere göre bir hayli fazla olabilmektedir. CGA yöntemi ise içbükey kısıtlayıcı fonksiyonlarının değerlendirmesinde etkili iken, dışbükey kısıtlayıcı fonksiyonların değerlendirilmesinde ya ıraksama problemi ortaya çıkmaktadır ya da çözüm için çok fazla sayıda iterasyon gerekmektedir.

Hem içbükey hem de dışbükey yüksek seviyede doğrusal olmayan performans fonksiyonlarının PMA temelli çözümü için yeni bir Hibrit Gradyan Analizi (HGA) yöntemi geliştirilmiştir. Bu yöntemde CGA ve AMV yöntemlerinin üstünlükleri birleştirilmiştir. HGA yönteminin iteratif prosedürü aşağıdaki gibi tanımlanmıştır:

\[
\begin{align*}
\mathbf{y}(i) & = \mathbf{s}(i) > 0, \\
\mathbf{s}(i) & = \mathbf{x}(i) - \mathbf{X}(i) \\
\mathbf{y}(i) & = \nabla_{\mathbf{s}} \mathbf{g} \left( \mathbf{d}, \mathbf{X}(i) \right) - \nabla_{\mathbf{s}} \mathbf{g} \left( \mathbf{d}, \mathbf{X}(i) \right)
\end{align*}
\]

(6)

HGA yönteminin iteratif prosedürü aşağıdaki gibi tanımlanmıştır:

\[
\begin{align*}
\mathbf{U}_{\text{HCA}}(i+1) & = \beta \mathbf{n} \left( \mathbf{U}_{\text{HCA}}(i) \right) \\
\mathbf{n} \left( \mathbf{U}_{\text{HCA}}(i) \right) & = \frac{\mathbf{w}_{\text{HCA}}(i)}{\mathbf{w}_{\text{HCA}}(i)}
\end{align*}
\]

(7)

burada, \( \mathbf{n} \) normalleştirilmiş arama yönüdür. \( \mathbf{w}(i) \) ise arama yönüdür ve aşağıdaki gibi hesaplanır:

\[
\mathbf{w}(i) = \begin{cases} 
-\nabla_{\mathbf{s}} \mathbf{g} \left( \mathbf{d}, \mathbf{U}_{\text{HCA}}(i) \right) & \text{if } k = 0 \text{ or } \mathbf{y}(i)^T \mathbf{s}(i) \leq 0 \\
-\nabla_{\mathbf{s}} \mathbf{g} \left( \mathbf{d}, \mathbf{U}_{\text{HCA}}(i) \right) + \theta_{\text{HCA}}(i) \mathbf{w}(i-1) & \text{if } \mathbf{y}(i)^T \mathbf{s}(i) > 0
\end{cases}
\]

(8)

burada eşlenik gradyan güncelleme parametresi Fletcher–Reeves algoritmalarındaki gibi aşağıdaki gibi şekilde hesaplanır:

\[
\theta_{\text{HCA}}(i) = \frac{\nabla_{\mathbf{s}} \mathbf{g} \left( \mathbf{d}, \mathbf{U}(i) \right)^2}{\nabla_{\mathbf{s}} \mathbf{g} \left( \mathbf{d}, \mathbf{U}(i-1) \right)^2}
\]

(9)


3. RBDO PROBLEMLERİ

HGA yönteminin doğruluğu, verilimliği ve karalılığı üç farklı RBDO problemi aracılığıyla AMV, CC, MCC ve CGA yöntemi ile karşılaştırılmıştır. CC ve MCC yönteminin parametreleri \( \lambda = 0.1 \) ve \( \mathbf{C} = \mathbf{I} \) olarak alınmıştır. Yöntemlerin karşılaştırılmasında kısıtlayıcı fonksiyonların değerlendirilmesi (FE) sayıları, amaç fonksiyonu değerleri ve optimum tasarım noktası değerleri ele alınmıştır. Tüm güvenilirlik analizleri için yakınsama kriteri \( \left( \| \mathbf{X}(i+1) - \mathbf{X}(i) \| / \| \mathbf{X}(i) \| \right) \leq 10^{-6} \) olarak alınmıştır. Optimizasyon problemleri MATLAB fonksiyonu FMINCON ile çözülmüştür.
3.1. Matematiksel Problem

Bir matematiksel RBDO problemi aşağıda verilmiştir (Youn ve Choi, 2004):

\[
\begin{align*}
\min f(d) &= d_1 + d_2 \\
\text{kısıtlayıcı:} & P(g_i(X) \leq 0) \leq \Phi(-\beta_i), i = 1, 2, 3 \\
\text{0} & \leq d_1, d_2 \leq 10,
\end{align*}
\]

burada:

\[
\begin{align*}
g_1(X) &= \frac{x_1^2x_2}{20} - 1 \\
g_2(X) &= \frac{(x_1 + x_2 - 5)^2}{30} + \frac{(x_1 - x_2 - 12)^2}{120} - 1 \\
g_3(X) &= \frac{80}{(x_1^2 + 8x_2 + 5)} - 1
\end{align*}
\]

Hedef güvenilirlik indeksi tüm kısıtlayıcı fonksiyonlar için \(\beta_i = 3.0\), başlangıç tasarım değerleri \(d^{(0)} = [5.0, 5.0]^T\), standart sapma değeri ise \(\sigma = 0.6\) olarak alınmıştır. Rasgele değişkenler normal dağılımlıdır. RBDO sonuçları Tablo 1’de verilmiştir.

<table>
<thead>
<tr>
<th>Yöntem</th>
<th>Araştırma özellikleri</th>
<th>FE sayısı</th>
</tr>
</thead>
<tbody>
<tr>
<td>AMV</td>
<td>7.816</td>
<td>165/351/130 (646)</td>
</tr>
<tr>
<td>CC</td>
<td>7.816</td>
<td>2184/1910/1955 (6049)</td>
</tr>
<tr>
<td>MCC</td>
<td>7.816</td>
<td>1823/1110/1588 (4521)</td>
</tr>
<tr>
<td>CGA</td>
<td>7.816</td>
<td>4576/1047/3173 (8796)</td>
</tr>
<tr>
<td>HGA</td>
<td>7.816</td>
<td>171/1047/133 (1351)</td>
</tr>
</tbody>
</table>

3.2. Çatı Makası Tasarımı

Uniform yükleme altındaki çatı makası RBDO problemi olarak ele alınmıştır. Problem ile ilgili detaylı bilgiler Rashki ve diğ. (2014)’nin yaptığı çalışmada görülebilir. Problemin matematiksel ifadesi aşağıda verilmiştir:

\[
\begin{align*}
\min C(d) &= 20224A_s + 364A_c \\
\text{kısıtlayıcı:} & P \left[ g(X) = 0.03 - \left( \frac{g^2}{2} \right) \left( \frac{3.81}{A_sE_{s}} + \frac{1.13}{A_cE_{c}} \right) \right] \leq 0 \leq \Phi(-\beta_i) \\
\text{burada}: & 0.0006 \leq A_s \leq 0.0012, \ 0.018 \leq A_c \leq 0.063 \\
& [A_s, A_c]^{(0)} = [0.001, 0.042], \ \beta_i = 3.0
\end{align*}
\]

Bu problem iki normal dağılımlı rasgele tasarım değişkeni \((A_s, A_c)\) ve dört normal dağılım rasgele değişken içermektedir. Değişkenlerin istatistiksel özellikleri Tablo 2’de verilmiştir. RBDO sonuçları Tablo 3’de verilmiştir.

<table>
<thead>
<tr>
<th>Rasgele Değişken</th>
<th>L (m)</th>
<th>q (N/m)</th>
<th>A_s (m^2)</th>
<th>A_c (m^2)</th>
<th>E_s (Pa)</th>
<th>E_c (Pa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ortalama</td>
<td>12</td>
<td>20000</td>
<td>0.0383</td>
<td>0.0048</td>
<td>5.9852x10^{-5}</td>
<td>6x10^{9}</td>
</tr>
<tr>
<td>Standart sapma</td>
<td>0.12</td>
<td>1400</td>
<td>0.0048</td>
<td>0.0383</td>
<td>5.9852x10^{-5}</td>
<td>6x10^{9}</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Yöntem</th>
<th>Araştırma özellikleri</th>
<th>FE sayısı</th>
</tr>
</thead>
<tbody>
<tr>
<td>AMV</td>
<td>35.9988</td>
<td>448</td>
</tr>
<tr>
<td>CC</td>
<td>35.9988</td>
<td>4560</td>
</tr>
<tr>
<td>MCC</td>
<td>35.9988</td>
<td>3849</td>
</tr>
<tr>
<td>CGA</td>
<td>35.9987</td>
<td>9481</td>
</tr>
<tr>
<td>HGA</td>
<td>35.9988</td>
<td>457</td>
</tr>
</tbody>
</table>
3.3. Pasif Taşıt Süsponsiyon Tasarımı

Bir otomotiv mühendisliği yapısı olan araç süspansiyon sistemi RBDO problemi olarak ele alınmıştır (Hsu ve Chan, 2010; Rashki ve diğ., 2014). Problem ile ilgili detaylı bilgiler ilgili kaynaklarda görülebilir. Pasif taşıt süspansiyonu RBDO modeli aşağıdaki gibidir:

\[
\min Z^2(\mathbf{d}) = \left( \frac{\pi AV}{m^2} \right) \left[ c_s k + (M + m) c^2 k^{-1} \right]
\]

kısıtlayıcı:

\[
P_1 \left[ g_1 = \left( \frac{\pi AV m}{b_0 g^2 k} \right) \left( \left( \frac{c_s}{M + m} - \frac{c}{M} \right)^2 + \frac{c^2 + c_s k^2}{M m M^2} \right) \right. \leq 1 \right] \geq \Phi(\beta_1)
\]

\[
P_1 \left[ g_2 = 7.6394 \left( 4000 (M g)^{-1.5} c - 1 \right)^{-1} \leq 1 \right] \geq \Phi(\beta_2)
\]

\[
P_1 \left[ g_3 = 0.5 (M g)^{1/2} \left( k^2 c_s k^{-1} (M + m)^{-1} + c \right)^{1/2} \leq 1 \right] \geq \Phi(\beta_3)
\]

\[
P_1 \left[ g_4 = \left( (M + m) g \right)^{0.877} c_s \leq 1 \right] \geq \Phi(\beta_4)
\]

1300 ≤ c ≤ 1600, 300 ≤ c ≤ 500, 10 ≤ k ≤ 100

\[
d^{(0)} = [1550, 350, 40]
\]

burada, \(c_s, c\) ve \(k\) sırasıyla lastik rijitliği, süspansiyon rijitliği ve sönümleme katsayısıdır. Bu rasgele değişkenler standart sapması 10 olan normal dağılımdadır. Problemde diğer parametreler: \(M = 3.263\) kg s\(^2\)/cm, \(m = 0.816\) kg s\(^2\)/cm, \(b_0 = 0.27\), \(A = 1.0\) cm\(^2\)/cycle m and \(V = 10.0\) m/s. RBDO sonuçları Tablo 4’de verilmiştir.

<table>
<thead>
<tr>
<th>Yöntem</th>
<th>Amaç</th>
<th>Tasarım Değişkenleri ((c^*, c_s, k))</th>
<th>FE sayısı</th>
</tr>
</thead>
<tbody>
<tr>
<td>AMV</td>
<td>Yakınsamadı</td>
<td></td>
<td></td>
</tr>
<tr>
<td>CC</td>
<td>Yakınsamadı</td>
<td></td>
<td></td>
</tr>
<tr>
<td>MCC</td>
<td>Yakınsamadı</td>
<td></td>
<td></td>
</tr>
<tr>
<td>CGA</td>
<td>396095203.30</td>
<td>416.97, 1468.40, 46.83</td>
<td>1846</td>
</tr>
<tr>
<td>HGA</td>
<td>396095612.76</td>
<td>416.97, 1468.40, 46.83</td>
<td>1067</td>
</tr>
</tbody>
</table>

4. SONUÇLAR

RBDO sonuçları değerlendirildiğinde AMV yönteminin matematiksel problem ve çatı makası probleminde en etkili yöntem olduğu görülmektedir. Bu problemlerde CC, MCC ve CGA yöntemleri de doğru sonuca ulaşabilmekle ancak fonksiyon değerlendirme sayıları çok yüksek olduğu için verimli olmamaktadır. HGA yöntemi ise AMV yönteminin çok yakın fonksiyon değerlendirme sayısı ile en efektif ikinci yöntemdir.


KAYNAKLAR


A SHORT REVIEW ON THE MANUFACTURING, MECHANICAL PROPERTIES AND POTENTIAL APPLICATIONS OF THE AUXETIC BEAMS

Mohammad Farid KHANSANAMI 1, Davood YOUNESIAN 2, Mostafa RANJBAR 3

1 School of Railway Engineering, Iran University of Science and Technology, Tehran 16846-13114, IRAN
2 Department of Mechanical Engineering,
3 Ankara Yıldırım Beyazıt Üniversitesi, Ankara, TURKEY
a mf.khansanami@gmail.com, b younesian@iust.ac.ir, c mranjbar@ybu.edu.tr

ABSTRACT
Auxetic behavior was originally reported by Lakes in 1987 who addressed negative Poisson's ratio effect in polyurethane (PU) foam. Different terms such as re-entrant, anti-rubber, auxetic and dilatational structures have been so far devoted to these extraordinary materials by former researchers. The present article reviews the latest advances in auxetic beams, their structural mechanisms, performance and applications. First, characteristic behavior of a nonauxetic (Conventional honeycomb) and an auxetic (re-entrant) structure mechanism is studied. Subsequently, exclusive mechanical properties of the auxetic beams against conventional beams are investigated in this paper. The last part of the paper concentrates on the control of wave propagation in sandwich beams with an auxetic core. New concepts in modeling and applications of auxetic beams are addressed at the end.

Keywords: Auxetics, Beam, Bending, vibro-acoustic behavior, Optimization.

1. INTRODUCTION
From the last four decades a number of synthetic materials that exhibit a negative Poisson ratio have been discovered [1-3]. These materials are called ‘auxetic’ materials and they possess an internal mechanism which have some unique and superior mechanical properties. So that, they significantly increase in volume when stretched and vice versa [4-5]. This counter-intuitive behavior, allowed by the deformation mechanism of the internal structure, gives new perspectives in terms of possible applications of these materials [6-8]. Auxetic Materials demonstrate the fascinating property which rooted from many reasons such as reentrant geometry, the base material and the internal structure like hinging and stretching of the cell walls. In comparison to conventional materials auxetic materials have some mechanical and thermal properties advantageous [9], for example, low density, enhanced toughness and hardness and shear modulus, better acoustic isolation and damping, improve thermal management (for use in flame arresters, heat exchangers, heat shields), Variable permeability, better acoustic absorption, durability at dynamic loadings and fatigue, filters etc.

In terms of applications, based on the negative Poisson’s ratio, superior toughness and acoustic properties, auxetic materials are of considerable importance in industries. A number of technological applications for auxetic materials are actively being pursued, like robust shock absorbers, fasteners, surgical implants, sound absorbing materials, sports applications, impact protector devices, aircraft, land vehicles, piezoelectric sensors with applications in hydrophones and etc. The potential to create filters by auxetic materials using enhanced pore size and shape tunability, can overcome the problems of filtration with conventional materials. Likewise, auxetic materials have opened new way for exploration in many design fields including product design, fashion design and architecture.

In terms of classification [10] Several types of auxetic materials are found such as a) Re-entrant Structures which started with analytical calculations of various deformation mechanisms [11] b) Rotating Units were presented by Grima and Evans [12] c) Missing rib structure is derived from conventional honeycomb [13] which was done by Gaspar et. al [14] d) tri-Chiral lattices consist of circular nodes with same radii which are connected together by ligaments of equal length, where first time investigate by Prall and Lakes [15] e) Anti-tetrachiral lattices [16] F) Many other auxetic geometries were also developed in the past such as star [17] and double arrowhead [18] structures, Kagome patterns [19].
Most auxetic materials can be analyzed using beam theory and vibration properties. In this paper, the bending problem of auxetic beams and wave propagation in sandwich beams are reviewed, respectively.

2. BENDING OF AUXETIC BEAMS

The uniqueness and advantageous of Cantilever Bending of auxetic beams, in comparison to ordinary materials, is explored in this part. A formal correspondence between an Influence of negative values of the Poisson’s ratio on the mechanical behavior and Comparison of the circular and rectangular cross sections with conventional beams are obtained. B.Gu et al. [20] describe the bending and failure behaviour of polymorphic honeycomb. They show the gradient cores show a multiple curvature (polymorphic) behavior. M. A. Koenders tried to find auxetic properties (negative Poisson’s ratio) on a network of bending beams [21]. Ranjbar et al. [22] have used same homogenization method to determine the mechanical properties of an auxetic hexagonal sandwich structure.

Aside from the functional aspect, the auxetics also show a great potential to be used as structural materials. For example, the bending of auxetic beams present good behavior in comparison to conventional one. There are three different types of theories in term of analytical description of beam, i.e. the Euler-Bernoulli theory, the Timoshenko theory [23] and higher-order beam theories. The shear stress distribution of a beam with Circular cross-section has been given by Timoshenko [24, 25] and Cowper [26] as:

\[ \tau_{xz} = -\left( \frac{r}{r} \right) \]

\[ \tau_{yz} = \left( \frac{r}{r} \right) \]

Accordingly, the shear stress in a dimensionless form for the horizontal diameter of the cross section (x = 0) beam by setting:

\[ I = -r^4 \]

Can be written in the form:

\[ (\tau_{xz}) = \left( \frac{r}{r} \right) - \]

The minimum value of \( \tau_{xz} \) moves inward according to the relation:

\[ \pm \sqrt{ } \]

In case of cantilever bending of auxetic beams with regular rectangular Cross Sections, equations of the exact shear stress can be written as follows: [27].

\[ (\tau_{xz}) = \left\{ 2 - \begin{bmatrix} 1 & - \end{bmatrix} + \begin{bmatrix} - & 1 \end{bmatrix} \begin{bmatrix} 1 & - \end{bmatrix} \right\} \]

Where:

\[ \left\{ \begin{bmatrix} m \\ n \end{bmatrix} \right\} \left( \frac{m}{n} \right) \left( \frac{m}{n} \right) \left( \frac{m}{n} \right) \left( \frac{m}{n} \right) \]

Consequently, the shear stress can be obtained by substituting equations (6) into the (5) as follows:

\[ (\tau_{xz}) = 1 + \left( \frac{m}{n} \right) \times \left\{ 2 - \begin{bmatrix} 1 & - \end{bmatrix} + \begin{bmatrix} - & 1 \end{bmatrix} \begin{bmatrix} 1 & - \end{bmatrix} \right\} \]

Obviously, the shear stress \( \tau_{xz} \) decrease when the Poisson’s ratio of the beam material (highly auxetic and mod. auxetic regions) decrease, that’s mean for similar situations, auxetic materials are more resistant to shear forces, than “regular” materials. On the other hand at the extreme negative values of Poisson’s ratio the shear modulus tends to
infinity. The results showed that auxetic honeycombs offered increased transverse Young’s modulus compared to regular hexagonal materials with the same relative density.

3. WAVE PROPAGATION IN SANDWICH BEAMS

Acoustical and structural properties of auxetic material and optimization of the structures is one of the axial parts in the design of the passive noise control compliant structure, (including, radiated sound power and root mean square level of the structural particle velocity). Great Interest for the application of cellular solids in ultra-light structures are rootsed from their multifunctional properties. One of the core reason which make the usage of this materials most attractive is, use of this materials as a cores for panels and shells that lead to lower weight than conventional materials and their vibration control characteristics. For example, N. Wicks et al [28] works on the application of the lattice material which lead to the design of truss core sandwich constructions, he reported that his kind of materials can be more competitive with traditional sandwich honeycomb and stiffened structures. Marburg et al [29] works on Applications and methods of structural acoustic optimization with respect to passive noise control. A new configuration for a sandwich beam with honeycomb core have been explored by Ruzzene [30]. He works on sound transmission reduction of the beams with various cores and he achieved that auxetic honeycombs configuration are generally more effective for reducing the sound transmission based on Bloch’s theorem, the analysis of a unit cell show the wave propagation behavior of a periodic structure. The study of wave propagation in periodic composite materials that are called phononic crystals (PCs) has received much attention because of their applications on frequency filters, vibrationless environments for high-precision mechanical systems or design of new transducers [31-33]. The theory to predicting the vibration response of periodic structures primarily has been applied to control the width and the location of the frequency bands (by introducing geometric or material discontinuities) where waves propagate (pass band) and where they are attenuated (stop band) [34], but recently the longitudinal vibration, flexural vibration have been studied both theoretically and experimentally in periodic beams structures [35-36]. Phononic crystals (PCs) can be controlled the wave propagation by geometrical discontinuities along the structure. Massimo Ruzzene and Fabrizio Scarpa analyzed the wave propagation in periodic sandwich beams with cellular core, they developed a spectral finite element model to predict the wave propagation characteristics and the beam transverse vibration, they demonstrate the influence of the length ratio and of the internal angle as the real part of the propagation constants and show how the internal angle significantly affects the location of the internal resonance of the cell. Figure 1 shows the sketch map of periodic sandwich beam with honeycomb core which The cell is composed of two portions (A and B) within which the material of the beam have a uniform properties [37]. The characteristics of wave propagation are analyzed through a transfer matrix formulation. Jihong wen et al worked on Flexural vibration band gaps in periodic sandwich beams with auxetic core which lead to provn the PWE method as an effective method to calculate the band structure of sandwich beam[38]. Optimal design of low-frequency band gaps in anti-tetrachiral lattice meta-materials has analyzed by Andrea Bacigalupo he has been formulated a linear dynamic model to describe the wave propagation properties of a composite lattice material [39]. They also revealed that a lower number of resonators always allows the opening of one or more full band gaps in the low-frequency range. Vibration transmission and isolation performance of the trichiral structures with uniform and gradient geometry parameter validated by Xu Shiyin et al. [40]. They also investigate the effects of the cell geometry on the bandgap and prove that the variation of the unit cell related to the distribution of the bandgap.

Figure 2. Sandwich beam with periodic honeycomb core and unit cell.
4. Conclusion

Auxetic materials are establishing a new way on scientific activities. Their special features, make the behavior of this materials more appropriate for acoustic absorption, in comparison to conventional materials. In this paper mechanical properties of the auxetic beams and their good bending behavior in comparison to conventional materials are investigated. Subsequently, the special behavior of auxetic materials in control of the wave propagation by geometrical discontinuities along the structure are studied. Moreover, the influence of the length ratio, angle of each unit cells and the thickness of cell-wall on load, shear modulus and bending stiffness of composite sandwich beams as the real part of the propagation are revealed.

For the future activities the analysis of the mechanical characteristics of the auxetic truss core beams to optimize vibro-acoustic characterization performance can be considered. Likewise, design of the auxetic beams with irregular geometry to optimize configuration parameters can be an attractive issue for the scientific investigations.


Considering of Development Potential of Absorption Refrigeration System

Utilizing Advanced Exergy Analysis Method

Abid Ustaoğlu1 and Bertan Parıltı 2

1 Department of Mechanical Engineering, Bartın University, Turkey
2 Institute of Science and Technology, Bartın University, Turkey
(austaoglu@bartin.edu.tr) Email of the corresponding author

ABSTRACT

The thermodynamic analysis applied on solar based absorption cooling cycle using advanced exergetic method by means of the first and second laws of thermodynamics. NH3-H2O fluid pair is considered as working fluid in absorption cooling cycle. The limitations of conventional analysis pose a problem therefore advanced exergetic analysis was performed instead of conventional analysis. Balance equations of energy and exergy are solved by the help of Engineering Equation Solver (EES). The irreversibility split into endogenous/exogenous and avoidable/unavoidable sections utilizing advanced exergic method. The irreversibility of each constituent, coefficient of performance and second law efficiency of overall system are found. Total irreversibility rate was approximately 14.32 kW and around 50% of the total irreversibility into group of avoidable sections. The generator has the most percentage of the total avoidable irreversibility rate with 54.25% and it was followed by absorber, heat exchanger and expansion valve with about 18%, 12.91% and 5.23%, respectively.

Keywords: Advanced exergy analysis, avoidable, unavoidable, absorption refrigeration cycle, ammonia-water

I. INTRODUCTION

Researches focused on harness nonconventional renewable energy sources more than ever. Thus, the absorption refrigeration applications become more popular and the optimum alternative technology other than conventional vapour compression applications. Absorption refrigeration system causes no environmental hazardous and depletion of the ozone layer. Amonnia-water fluid pair is one of the well-selected ammonia-based working fluid as it has the unique advantages for a solar based absorption chiller applications. Moreover, ammonia (NH3) can be considered as an environment-friendly refrigerant with low global warming potential (GWP), zero CO2 footprints and ozone depletion potential (ODP) values in despite of its low toxicity. There are theoretical and experimental studies on the performance characteristics in absorption chiller applications. Sahem et al. (2014) achieved that the maximum COP is 0.8 and made a parametric study on the performance characteristics of water-lithium chloride in different temperatures. Kaushik and Arora et al. (2009) considered a double stage and single stage absorption chiller cycle for LiBr-H2O utilizing the first and second law of thermodynamic. The result of absorption chiller system showed that the COP is around 60% for double stage system while it was compared to the single stage system. Kaynakli and Kilic (2007) achieved a parametric study for different operating parameters. They emphasized solution heat exchanger and refrigerant heat exchanger differ from each other at the COP. Chua et al.(2002) performed that thermal conductance of the heat exchangers and the internal entropy production evaluated for an irreversible NH3-H2O fluid pairs for a single stage absorption chiller cycle. In the rectifier the maximum heat dissipation observed. Triple stage water–lithium bromide absorption chiller cycles have completed for the COP among different flow configurations by Kaita et al.(2002). They obtained that the parallel one in different flow configurations only has maximum value of the COP. The objective of this study is a parametric study considering the...
COP for each components in different heat sources and determine irreversibility percentages and amount of avoidable and unavoidable sections of each component.

II. MATERIALS AND METHOD

A. System Description

Figure 1 illustrates a simple absorption cooling cycle. It has eight components namely, an absorber, a pump, a generator, a condenser, a heat exchanger, two pressure reducing valves and an evaporator. NH₃-H₂O fluid pairs are selected as working fluids in the system. In this cycle, NH₃ and H₂O are used as the refrigerant and absorbent, respectively. When the refrigerant is sent in points 1, 2, 3 and 4, the solution is sent in points 5, 6, 7, 8, 9 and 10. Refrigerant existing the evaporator and weak solution (point 10) mix in the absorber. Weak solution absorbs the refrigerant and enters the pump as strong solution. Solution is compressed by the pump and heat transferred from weak solution in the heat exchanger (HEX). The strong solution in the generator is boiled by some external source. During the heating process, ammonia vapour is sent from the solution at higher pressure to condenser. The weak solution exiting the generator from (point 8) sent the HEX. Later, it emits some of its heat in HEX and flows to the expansion valve 2 where its heat and pressure decreases. For another circulation, the high pressure ammonia vapour leaving the generator from (point 1) flows into the condenser as high pressure liquid ammonia. Liquid ammonia is directed to the expansion valve 1 and then vaporized working fluid in evaporator enters to the absorber. Thus, the chiller cycle completes.

![Figure 1 - the schematic view of an absorption refrigeration cycle.](image)

B. Advanced Exergy Analysis Method

Advanced exergetic method is an extensive analysis. Thereby, the irreversibility is divided into avoidable/unavoidable and endogenous/exogenous parts. In this study, only avoidable/unavoidable sections are determined.

The total irreversibility can be found using following equation

\[ \dot{E}_{D,k} = \dot{E}_{F,k} - \dot{E}_{P,k} = T_0 \dot{S}_{P,k} = T_0 \dot{m}_k \dot{S}_{P,k} \]  

(1)

Where \( T_0 \), \( \dot{E}_{F,k} \), \( \dot{E}_{P,k} \) show reference temperature, exergetic fuel and exergetic product, respectively. Therefore, the second law efficiency can be found by following equations for \( k \)th component.
\[ \varepsilon_k = \frac{\dot{E}_{p,k}}{\dot{E}_{F,k}} = 1 - \frac{\dot{E}_{D,k}}{\dot{E}_{F,k}} \]  

(2)

Additively, the advanced exergetic method also classifies the destruction rates through each cycle constituent as avoidable \( \dot{E}_{D,k}^{AV} \) or unavoidable \( \dot{E}_{D,k}^{UN} \).

\[ \dot{E}_{D,k} = \dot{E}_{D,k}^{UN} + \dot{E}_{D,k}^{AV} \]  

(3)

In order to find the unavoidable part, each component are operated unavoidable conditions. Calculated irreversibilities belong to unavoidable parts (Petrakopoulou et al., 2012).

\[ \dot{E}_{D,k}^{UN} = \dot{E}_{p,k}^{real} \left( \frac{\dot{E}_{D,k}}{\dot{E}_{p,k}} \right)^{UN} \]  

(4)

### III. RESULTS AND DISCUSSION

Traditional and advanced exergic method have carried out by means of EES software. The pressure and the ambient temperature obtained as 101.325 kPa and 25°C, respectively. The generator, condenser, evaporator temperatures were evaluated as 100°C, 45°C and 5°C, respectively. Temperature of the exhausted gas compressing to the generator was 277°C. Besides, cooling capacity was assumed to be 20 kW. Isentropic efficiencies was assumed about 85% for the pump. The mass flow rate was 0.01834 kg/s for the primary fluid, and that of the strong and weak fluids was calculated by taking into account the cooling capacity of absorption refrigeration system using the energy balance equations. The mass flow rate was 44.5 kg/s for exhausted gas when it reaching to the generator. The mass flow rates in the condenser and evaporator were found by energy balance equations for the cooling waters.

Figure 2 illustrates the avoidable/unavoidable parts as percentage for the irreversibility rates. In the part of avoidable irreversibility is about 50% of the total irreversibility for the overall system. Most of the irreversibility is comprised of the unavoidable section, although the generator is the maximum value of the irreversibility rate. It is found to be 35.41% (0.51 kW), 44.24% (0.69 kW) and 61.18% (7.77 kW) for the avoidable sections in the condenser, evaporator and generator, respectively. The generator has the highest irreversibility as the heat transfer takes place at higher temperature difference. Besides, use of ammonia water working fluids are an advantage owing to its no flammability and zero ozone depletion potential (ODP).
Figure 2: Exergy destruction percentages and amount of avoidable/unavoidable parts of each component.

IV. CONCLUSION

An absorption chiller cycle is one of a waste heat powered cycle that has been performed utilizing traditional and advanced exergetic method. This advanced exergetic method reveals the constituents with maximum irreversibilities. Dividing the irreversibility into unavoidable-avoidable and exogenous-endogenous sections shows improvement in the exergetic method of energy conversion applications. Most part of the irreversibility observes in the generator and then absorber while the pump and expansion valves have the lowest irreversibility rates. Most share of total avoidable irreversibility observes into generator around 61.18% for the total avoidable section of the whole cycle. Enhanced analysis results demonstrate the real improvement potential and indicates the importance of irreversibility related to the constituents.

REFERENCES


ACKNOWLEDGMENTS

This present work was developed within the framework of a research project entitled with “Design, installation and performance evaluation of a novel compound parabolic concentrator-photovoltaic thermal system (PVT-CPC) (216M051)”, fully funded by The Scientific and Technological Research Council of Turkey (TUBITAK). The authors would like to thank TUBITAK for the financial support given to the projects. All responsibility for the content of this publication lies with the author. We thank for the support.
Thermodynamic Analysis of Absorption Refrigeration Cycle Utilizing Enhanced Exergy Analysis Method

Abid Ustaoğlu and Bertan Parıltı

1Department of Mechanical Engineering, Bartın University, Turkey
2Institute of Science and Technology, Bartın University, Turkey
Email of the corresponding author

Abstract

An absorption refrigeration system is performed utilizing enhanced exergy analysis method. Ammonia-water fluid pairs are selected as working fluids in the absorption cooling cycle. The first and second laws of thermodynamics are applied to each component in the system. Balance equations are solved with the help of Engineering Equation Solver (EES). In advanced exergetic analysis method, irreversibility split into endogenous/exogenous and avoidable/unavoidable sections. In this study, exergy destruction percentages, amount of endogenous and exogenous parts of each component are calculated. The irreversibility of each constituent, coefficient of performance and second law efficiency of overall system are found. Total exergy destruction rate was about 28.75 kW and about 54.44% of the total exergy destruction was falling into group of endogenous part. The others rates, condenser with 5.28%, evaporator with 5.45% and absorber with 26.34% belong to endogenous part, respectively.

Keywords: Advanced exergy analysis, endogenous, exogenous, energy analysis, absorption refrigeration, ammonia-water

I. INTRODUCTION

The increasing environmental awareness of researchers is currently leading an increasing interest in absorption cooling system against rapidly increasing fossil fuel costs. As renewable and sustainable energy sources, absorption refrigeration systems are used low temperature sources while tradional cooling systems are utilised by mechanical power. Absorption refrigeration technology use working fluid pairs such as H2O-LiBr and NH3–H2O. These fluid pairs are used refrigerant and absorbant. In the literature, there are restricted studies about advanced exergy analysis of absorption refrigeration system. There are a few studies about analyzing the performance of the system in different fluid pairs. Pilatowsky et al.(2001) applied the first law of thermodynamics using monomethylamine–water solutions in different evaporator and condenser temperature intervals. Xu et al.(2013) compared double and single stage absorption refrigeration cycle. They used LiBr-H2O as working fluid pair. Yari et al.(2011) compared two new absorption cooling system. They investigated the effect of generator temperature on the system performance. Varani et al.(2007) performed exergetic analysis of a LiBr/H2O absroption system. Liu and Wang (2004) analysed solar-based absorption cycle and determined the system performance. Rosiek et al. (2012) evaluated single effect absorption cooling cycle in terms of first and second laws of thermodynamics. The results indicated that the most significant constituent was the condenser. Furthermore, avoidable part of the irreversibilities was more effective for the whole system. The objective of this work is to determine exergy destruction percentages and amount of endogenous and exogenous parts of each component.
II. MATERIALS AND METHOD

A. System Description

Figure 1 illustrates a simple absorption cooling cycle. It has eight components namely, an absorber, a pump, a generator, a condenser, a heat exchanger, two pressure reducing valves and an evaporator. NH$_3$-H$_2$O fluid pairs are selected as working fluids in the system. In this cycle, NH$_3$ and H$_2$O are used as the refrigerant and absorbent, respectively. When the refrigerant is sent in points 1, 2, 3 and 4, the solution is sent in points 5, 6, 7, 8, 9 and 10. Refrigerant existing the evaporator and weak solution (point 10) mix in the absorber. Weak solution absorbs the refrigerant and enters the pump as strong solution. Solution is compressed by the pump and heat transferred from weak solution in the heat exchanger (HEX). The strong solution in the generator is boiled by some external source. During the heating process, ammonia vapour is sent from the solution at higher pressure to condenser. The weak solution exiting the generator from (point 8) sent the HEX. Later, it emits some of its heat in HEX and flows to the expansion valve 2 where its heat and pressure decreases. For another circulation, the high pressure ammonia vapour leaving the generator from (point 1) flows into the condenser as high pressure liquid ammonia. Liquid ammonia is directed to the expansion valve 1 and then vaporized working fluid in evaporator enters to the absorber. Thus, the chiller cycle completes.

![Figure 1 - the schematic view of a simple absorption chiller cycle.](image)

B. Advanced Exergy Analysis Method

Advanced exergetic method is an extensive analysis. Thereby, the irreversibility is divided into avoidable/unavoidable and endogenous/exogenous parts. In this study, only endogenous/exogenous sections are determined.

The total irreversibility can be found using following equation

$$\dot{E}_{D,k} = \dot{E}_{F,k} - \dot{E}_{P,k} = T_0 \dot{S}_{P,k} = T_0 \dot{m}_k \dot{s}_{P,k}$$  

(1)

Where $T_0$, $\dot{E}_{F,k}$, $\dot{E}_{P,k}$ show reference temperature, exergetic fuel and exergetic product, respectively. Therefore, the second law efficiency can be found by following equations for $k$th component
\[ \varepsilon_k = \frac{\dot{E}_{F,k}}{\dot{E}_{F,k}^T} = 1 - \frac{\dot{E}_{D,k}}{\dot{E}_{F,k}} \]  

(2)

For each constituent \( k \) of the absorption cooling cycle, the irreversibility that is owing to its operation is called to as the endogenous exergy destruction \( \dot{E}_{D,k}^{EN} \), while \( \dot{E}_{D,k}^{EX} \) (exogenous part) is the irreversibility arised from other cycle constituents. This analysis provides that should be focused on the component.

\[ \dot{E}_{D,k} = \dot{E}_{D,k}^{EN} + \dot{E}_{D,k}^{EX} \]  

(3)

III. RESULTS AND DISCUSSION

Traditional and advanced exergic method have carried out by means of EES software. The pressure and the ambient temperature obtained as 101.325 kPa and 25°C, respectively. The generator, condenser, evaporator temperatures were evaluated as 100°C, 45°C and 5°C, respectively. Temperature of the exhausted gas compressing to the generator was 277°C. Besides, cooling capacity was assumed to be 20 kW. Isentropic efficiencies was assumed about 85% for the pump. The mass flow rate was 0.01834 kg/s for the primary fluid, and that of the strong and weak fluids was calculated by taking into account the cooling capacity of absorpsion refrigeration system using the energy balance equations. The mass flow rate was 44.5 kg/s for exhausted gas when it reaching to the generator. The mass flow rates in the condenser and evaporator were found by energy balance equations for the cooling waters.

Figure 2 illustrates the endogenous/exogenous parts as percentage for the irreversibility rates. In the part of endogenous irreversibility is about 54.34% of the total irreversibility for the overall system. Most of the irreversibility is comprised of the exogenous part, although the generator is the maximum value of the irreversibility rate. It is found to be 9.69% (1.52 kW), 10.01% (1.57 kW) and 28.64% (4.49 kW) for the endogenous parts in the condenser, evaporator and generator, respectively. The generator has the highest irreversibility as the heat transfer takes place at higher temperature difference. Additionally, use of ammonia water fluid pairs are an advantage owing to its no flammability and zero ozone depletion potential (ODP).

![Figure 2: Irreversibility percentages and amount of endogenous/exogenous parts of each constituent.](image-url)
IV. CONCLUSION

Dividing the irreversibility into unavoidable-avoidable and exogenous-endogenous sections shows improvement in the exergic method of energy conversion applications. The dividing supplies the accuracy in exergic method. For the overall analysis of energy conversion applications, there are most advantages dividing the irreversibility into unavoidable-endogenous and endogenous-exogenous sections. The COP, irreversibility rate of each constituent and exergy exergetic were found for the whole system. Total exergy destruction rate was about 28.75 kW and about 54.44% of the total exergy destruction was falling into group of endogenous part while the 45.56% belongs to the part of exogenous irreversibility rate. The others rates, condenser with 5.28%, evaporator with 5.45% and absorber with 26.34% belong to endogenous part, respectively.

REFERENCES


ACKNOWLEDGMENTS

This present work was developed within the framework of a research project entitled with “Design, installation and performance evaluation of a novel compound parabolic concentrator-photovoltaic thermal system (PVT-CPC) (216M051)”, fully funded by The Scientific and Technological Research Council of Turkey (TUBITAK). The authors would like to thank TUBITAK for the financial support given to the projects. All responsibility for the content of this publication lies with the author. We thank for the support.
TRIBOLOGICAL PROPERTIES OF EPOXY BASED PHENOLIC COMPOSITES REINFORCED WITH GRAPHENE NANOPlatelet

Emine Feyza SUKUR1, Suheyla KOCAMAN2, Gurol ONAL3

1,3Konya Technical University, Department of Mechanical Engineering, Konya/TURKEY
2Konya Technical University, Department of Chemical Engineering, Konya/TURKEY

*efeyzasukur@selcuk.edu.tr , *suheyla@selcuk.edu.tr
*gonal@selcuk.edu.tr

ABSTRACT
Graphene nanoplatelet (GNP) is one of widely used nano filler to improve the overall mechanical properties and tribological performance of polymer composites due to their solid lubricant properties and thermal stability. This study investigated the wear performance of epoxy based phenolic resin nanocomposites containing graphene nanoplatelets (GNPs) at different additive rates as fillers. Effect of the graphene concentration on the mechanical and tribological properties of nanocomposites were evaluated in details. The friction and wear behaviors of the nanocomposites were tested by pin-on-disk method under constant load and sliding speed conditions without any lubrication at room temperature in ambient air. The results of wear tests indicate that GNP weight ratio and dispersion in resin have a substantial effect on both the wear and the coefficient of friction of composites.

Keywords: Graphene nanoplatelet (GNP), epoxy based phenolic resin, wear

1. Introduction
Polymeric composite materials; due to their light weight, low friction coefficient, wear resistance and self-lubricating properties, they are widely used in aerospace, automotive and chemical industries as well as various structural applications (Gill and Sidhu, 2016). Epoxy, phenolic, polyurethane and polyester resins are generally preferred as matrix in polymeric composite materials. Epoxy resins are more expensive than other thermosetting resins, but they have better mechanical properties, moisture absorption, higher resistance to corrosive liquids and environmental factors. Another positive aspect of epoxy resins is their low shrinkage during the curing process; in other words, the reduction percentages are low in volume. Because of the lack of styrene, epoxy resins have less toxic emissions during the curing process. This makes the use of epoxy resins with "open die" manufacturing technologies (eg, hand lay-up or vacuum bagging) possible (Liu, et al., 2006). Although the tribological properties of the epoxy resins are poor, this can be improved by the addition of micro and nano-sized particles into the matrix. (Kumar, et al.,, 2015). Particularly with nanoparticle additives, it is possible to increase the load resistance of polymer matrix composites, improve thermal properties, reduce friction coefficient and wear rate (Shi, et al., 2003; Chang, et al., 2005; Aytollahi, et al., 2012). When the studies in the literature are examined, it is seen that, especially CNT and graphene, a wide variety of nanoparticles such as Si3N4, SiC, ZnO, Al2O3, TiO2, SiO2, MnO2 are used as reinforcing elements to improve the different properties of composite materials (Zhang, et al., 2006; Shen et al., 2014; Dass, et al., 2014; Zhang, et al., 2008; Singh, et al., 2017). The polar groups on the surface of the graphene improve their compatibility with the polymer matrices. The graphene derivatives obtained by the oxidation method are more suitable for surface formation by bonding to functional groups in the production of graphene based materials / composites (Yazici, et al., 2016). Recently, graphene additives have been used in many studies to improve the tribological properties of polymer composites, thus the friction coefficient and wear rate are significantly reduced compared to pure polymer composites (Ren, et al., 2013). In addition, it was stated that tensile strength of the graphene added epoxy composites is higher than pure epoxy and it is also stated that Young's modulus is increased. As the hardness and elasticity of graphene layers are higher than that of epoxy matrix, the values of hardness and elasticity modulus of epoxy composites increase with increasing graphene content (Khun, et al., 2015). It should be stated that the additive ratio
of graphene and derivative additives are very important in the development of mechanical and tribological properties. Optimum conditions are generally achieved at lower additive rates, whereas as the ratio increases, it is difficult to distribute graphene homogeneously in the polymer and the resulting agglomerations cause a decrease in properties. In the literature, it has been reported in various studies that the addition of graphene and derivative nano additives to low levels of epoxy matrix is more effective in increasing tribological performance (Shen, et al., 2013).

In this study, the effect of 0.5, 1, and 1.5 wt% nano-graphene addition on epoxy based phenolic resin (EPhR) on mechanical and tribological properties was investigated in detail. Hardness measurements of the nano composites were made, tensile properties and damage mechanisms were examined with the help of SEM images. The friction and wear properties were tested at room temperature under the conditions of constant load and sliding velocity on a pin-on-disk wear tester (ASTM-G99).

2. Experimental
2.1. Material and preparation of polymer composites

The hardener cycloaliphatic amine (KH-816) and phenol epoxy resin (YDPN-631) were purchased from KUKDO Chemical Co. The graphene nanoparticles (Graphene Nanoplatelet (GNP), 99.5 +%, 6 nm, S.A.: 150 m² / g Dia: 5μm) were obtained from the Nanografi Turkey.

![Figure 1. a) Scanning electron microscopy image of GNP; b) preparation of GNP/ EPhR composites.](image)

First, the amount of epoxy resin was determined, then the graphene particles were weighed in an amount corresponding to 0.5, 1 and 1.5 wt% of the epoxy resin. A tipped sonicator was used to distribute graphene particles homogeneously in epoxy based phenolic resin. The mixtures were mixed in a sonicator for 20 minutes in 5 minute intervals and mixed with mechanical agitator after addition of 30% curing agent and then cast into tensile test molds and 90 mm circular molds designed for the production of materials for use in pin-on-disc experiments. The pure epoxy and grafted admixtures poured into the mold were initially cured at room temperature for 24 hours and then at 80 ° C for 15 hours (Sukur, et al., 2018).

2.2. Mechanical tests

Shore Durometer TH 210 tester was used to measure the hardness of composites. Tensile properties were determined using Stretch and Pressing Equipment TST-Mares/TS-mxe. The tests were carried out in three repetitions at a speed of 5 mm/min.

2.3. Tribological tests

The tribological properties of the specimens were investigated by using chrome steel ball with 6 mm diameter and 62 HRC hardness in pin-on-disc wear tester. The tests were carried out at a constant sliding velocity for 30 minutes at room temperature (25 ± 3 ° C) at a constant load. Composite samples were cleaned with acetone before and after the tests, were weighted in 10⁻⁴ grams sensitivity (Sukur, et al., 2018). The specific wear rate, Ws, was calculated using the following equation.
\[ W_s = \frac{\Delta m}{\rho F_N L} \text{mm}^3/Nm \] 

Here, \( \Delta m \) represents the weight loss (g) of the worn sample, \( F_N \) applied load (N), \( L \) sliding distance (m) and \( \rho \) density of the specimen in g/mm\(^3\).

**Figure 2.** a) Pin-on-disk test stand

### 2.4. Imaging

The structure of the interface between nano-additive and matrix and the damage mechanisms after the tensile test were investigated by scanning electron microscopy (SEM) and Philips XL30 SFEG.

### 3. Results and discussions

#### 3.1. Mechanical properties

Hardness and tensile test results of neat EPhR and GNP added EPhR composite materials are given in Table 1.

<table>
<thead>
<tr>
<th>GNP (wt%)</th>
<th>Elongation at break (%)</th>
<th>Tensile strength (MPa)</th>
<th>e-mod (GPa)</th>
<th>Hardness (ShoreD)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Neat EPhR</td>
<td>0.659</td>
<td>59.6</td>
<td>5.1</td>
<td>73</td>
</tr>
<tr>
<td>EPhR/GNP composites</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.5</td>
<td>1.01</td>
<td>63.5</td>
<td>5.5</td>
<td>77</td>
</tr>
<tr>
<td>1</td>
<td>0.84</td>
<td>74</td>
<td>6.2</td>
<td>77</td>
</tr>
<tr>
<td>1.5</td>
<td>0.6</td>
<td>60.5</td>
<td>5.6</td>
<td>78</td>
</tr>
</tbody>
</table>

With the increase of the GNP content, the hardness values of the composites showed an average increase of 6%. This increase in the hardness values of EPhR composites can be explained by the inclusion of a secondary hard phase (GNP) in pure epoxy. Table 1 shows that the basic mechanical properties of nanocomposites containing high amounts of GNP decreased, whereas the hardness values increased with the addition of increased nanoparticles. This point can be expressed by the fact that the hardness of nanocomposites is not sensitive to the presence of agglomerations as much as the basic mechanical properties (Ayatollahi, et al., 2012).
Figure 3. Hardness of neat EPhR and EPhR composites with different GNP contents.

The tensile strengths of EPhR composites are about 63.5, 74 and 60.5 MPa, respectively, at 0.5, 1, and 1.5% by weight of GNP, whereas the tensile strength of pure EPhR is about 59.6 MPa (Fig 4). It shows that addition of GNPs improves the tensile strength of EPhR composites. In addition, with a graphene content ranging from 0.5 to 1.5%, an increase in a modulus of elasticity between about 8% and 22% is observed. The tensile strength and modulus of elasticity showed the best performance with an increase of 24% and 22% in composites with 1% GNP added to pure EPhR, respectively. Neat EPhR is more brittle compared to composites, its resistance to crack initiation and propagation is weaker. Homogeneously dispersed GNP can bridge growing cracks, inhibit crack propagation and thereby improve the properties of nanomaterials (Wei, et al., 2017). However, as the additive ratio increases, it is difficult to distribute the nanoparticles homogeneously in the matrix. The nanoparticles, which are not well dispersed in the matrix, become agglomerated in the structure. Aggregate zones appear as a defect in the structure and cause a reduction in mechanical properties. The reduction in mechanical properties of 1.5% by weight of GNP can be explained by these agglomerations.

Figure 4. Tensile strength and modulus of elasticity of EPhR and EPhR composites with different GNP contents.
The surface morphologies of the EPhR matrix and the GNP composites are shown in Figure 5. When the SEM micrograph of the neat EPhR is examined, it shows that there is only one phase that is a regulative crack in the fracture surface, indicating a brittle fracture surface, which is accounted for the poor toughness of the neat EPhRs (Fig. 5a). Compared to neat EPhR in general terms, it was noted that GNP has a good distribution in composites. The micro cracks which was indication of homogeneous dispersion of GNP in the EPhR were seen from Fig. 5c. Fig. 5d shows fracture lines belonged to EPhR and smaller agglomeration originated. Homogeneity in the composites, continued to rise up to 1 wt% GNP loading, following which heterogeneity increased at 1.5 wt% causing a weakening of the mechanical properties (see Table 1). It is believed that such aggregations are due to the interaction between the matrix and the structure and amount of the nano filler material (Soydal, et al., 2018).

3.2. Tribological properties

The friction and wear properties of neat EPhR and composites with different ratios of GNP were investigated in the pin-on-disc wear test apparatus using a 6 mm chrome steel ball for 600 m at a sliding velocity of 0.5 m/s under 10 N normal load. Figure 6 shows the variation of the friction coefficients of the neat EPhR and EPhR/GNP composites tested at a sliding velocity of 0.5 m/s. When the friction coefficient curves are examined, it is seen that GNP content decreases the friction coefficient of EPhR composites at all additive ratios. This supports that graphene modified composites have lower friction than neat EPhR. The 0.5 wt% GNP additive is highly effective at stabilizing the friction coefficient curve, as well as significantly reducing the friction coefficient compared to other additive ratios. In different studies in the literature, it is stated that GNPs act as a solid lubricant for the friction surfaces. More precisely, the GNPs serve as a direct contact element between ball and composite. During the sliding, the contact between the steel ball and the composite may increase as the friction increases. The increase in hardness and elasticity module of EPhR composites with GNP content (see Table 1) reduces friction by reducing contact between steel ball and composites (Khun, et al., 2015).

Figure 5. Fracture surface images of a) neat EPhR; b) 0.5 wt% GNP/ EPhR; c) 1 wt% GNP/ EPhR; d) 1.5 wt% GNP/ EPhR.
Fig. 7a shows the mean friction coefficients of neat EPhR and EPhR composites with different ratios of GNP tested at sliding velocity of 0.5 m/s. In the test conditions specified, the friction coefficient of the neat EPhR is 0.2, whereas the 0.5, 1, and 1.5 GNP additives are 0.14, 0.17 and 0.18, respectively. The GNP addition of 0.5% resulted in a 30% reduction in friction coefficient. The friction coefficient has decreased compared to pure composites in all additive ratios, compared with 0.5%, the friction coefficient increased as the additive rates increased. When the amount of nanoparticles increases, more agglomeration is likely to occur. As the size of the aggregate zone increases, the matrix cannot completely penetrate and wet the particles. Therefore, a weak interface develops between the matrix and nanoparticles in these regions. Nanocomposites containing high rates of nanoparticles cannot fully benefit from the outstanding properties of nanomaterials. The decrease in mechanical properties and the deterioration in tribological properties in the increasing proportions of GNPs reveal the accuracy of this statement (Ayatollahi, et al., 2012). Wear-tested composites lose weight by removing material from the contact surface due to friction. The test parameters are specific and the density of the materials is known. With the help of precision scales, the weight loss in the materials is measured and then the wear rate values are calculated with the help of equation 1. It is known that the friction properties of the materials have effects on the wear rate results. The addition of GNPs into the structure results in lower shear stresses on the sliding surfaces, which leads to a reduction in the friction coefficients, which in turn reduces wear rates. Thanks to the GNPs, it is subject to less epoxy contact on the surfaces during sliding, thus less wear occurs in the GNP / EPhR composites than in the neat EPhR.
Wear test results show that the GNP additive reduces wear at all additive rates. In Neat EPhR and EPhR composites with 0.5, 1 and 1.5 GNP, the wear rate values are $19 \times 10^{-7}$, $5 \times 10^{-7}$, $16 \times 10^{-7}$ and $17 \times 10^{-7}$ mm$^3$/Nm, respectively. With the addition of GNP, the reduction in wear rates between 10% and 74% is achieved, thus improving tribological characteristics. Considering the lubricant effect of graphene, as the GNP ratio in the matrix increases, wear rate is expected to decrease further. However, when the results of Figure 7b were examined, it decreased until 0.5% contribution and then increased. One of the main reasons for this is the agglomerations that occur in the incremental additive rates. When the material is removed from the contact surface during wear, more material is lost in the agglomerated regions than in the homogeneous regions. The increase in weight loss leads to an increase in wear rate under the same conditions when other parameters are constant.

4. Conclusions

Mechanical and tribological properties of epoxy based phenolic resin nanocomposites containing different additive ratios (GNP) were investigated. The effects of GNP additive, which is between 0.5% and 1.5% by weight, on the wear performance were investigated under the constant test parameters (shear rate, load, temperature, shear distance). Main results obtained from the study:

- In EPhR composites, basic mechanical properties such as hardness, modulus of elasticity and tensile strength were achieved with 1% GNP contribution to the best values, when the rate of additive is increased, a decrease in properties has occurred.
- The hardness, elasticity modulus and tensile strength values of the nanocomposites were increased by 7%, 22% and 24%, respectively, when compared to neat EPhR.
- Friction coefficient and wear rate values were reduced by 30% and 74% by addition of GNP nanoparticles.
- The presence of agglomerations prevented further improvement in tribological properties in EPhR composites containing more than 0.5% by weight of GNP.

![Figure 7. Tribological properties of EPhR composites a) friction coefficient and b) wear rate.](image-url)
5. References


ANALYSIS OF COLD ROLLING PROCESS WITH DIFFERENT PARAMETERS
USING FINITE ELEMENT METHOD

Jabbar Gattmah¹, Fahrettin Ozturk²,³, Sadettin Orhan²

¹ Department of Material Engineering, College of Engineering, University of Diyala, Diyala, Iraq
²Department of Mechanical Engineering, Ankara Yildirim Beyazit University, Ankara, Turkey
³Turkish Aerospace Industries, Inc., Ankara, Turkey

Corresponding Author: Jabbar Gattmah, E-mail: msc_jgj_katma7@yahoo.com

ABSTRACT

In this study, a commercially available finite element software ABAQUS/Explicit is used to simulate a cold rolling process for a rectangular cross-section. AA5049, AISI1015, and 304L stainless steel materials are considered in rectangular bar shape. Effects of process parameters on the rolling process are studied in detail. The finite element model is validated with available experimental data. Results indicate that the material type has a great influence on determining the reduction of area, friction coefficient, and roller speed. The maximum von Mises stress is increased with increasing reduction of area and roller speed for AA5049 and it is almost constant for 304L stainless steel.

Keywords: Finite element method, rolling simulation, cold rolling, rolling process, rolling parameters.

1. INTRODUCTION

Cold rolling is one of the widely used manufacturing processes in metal forming industry. Different cross-sections including beam, channel, angle, bar, rod, and seamless pipe can be produced [1]. In the rolling process, there are two rotating rolls used to reduce thickness of the material while the length of the material is increased. These rollers work on the same plane and have the same geometry, one above the other with the opposite direction. Fig. 1 describes a schematic of a cold rolling process with a plastic zone. Where \( \omega \) represents the angular velocity of roller and \( \alpha \) is the angle of rolling contact between rolls and workpieces while \( h_c \) and \( h_f \) are the thickness of workpieces before and after rolling respectively. Reduction of material thickness is the main objective for the cold rolling process. During the process, the complexity of metal flow due to the deformation in the contact zone between rolls and material is taken place. The deformation of workpieces is quite complex due to the reduction of area that produces axial and radial stresses and strains. Many parameters like the roller speed, the speed of workpiece, the reduction in thickness, the roller diameters, and the friction coefficient have affected the rolling process [2-3]. In metal forming processes, the rolling, drawing, and extrusion problems can be solved using finite element method. Simulation work reduces development and manufacturing times and cost significantly. These processes consist of large deformation and complex contact issues. For this purpose, the time explicit dynamic analysis is generally utilized to solve large deformation and contact problem. The method is applied to describe the behavior of the internal stresses and plastic strains [4-5]. Several studies have been performed on the subject. In one of the study, rolling process of aluminum alloys were simulated by the finite element method and compared to experiments models [6]. Both the experimental and numerical results were in good consistency. The static cross-sectional for the strip thickness profile was predicted using a computational method and model deflection of the cluster-type rolling mill components [7]. Park and Anh [8] studied longitudinal strain and springback phenomenon that take place during the cold rolling of aluminum automotive components. An Arbitrary Lagrangian Eulerian technique was utilized to model chatter vibration in high speed rolling by finite element method [9]. The results were found consistent with experimental data. Cold rolling parameters are quite important in determining the quality of products. They are critical to achieve the rolling process at the lowest cost. Cold rolling process parameters for thin steel strips were determined to prevent the chatter which occurs by increasing the limit of
rolling speed [10]. Finite element analysis based on DEFORM-3D software was applied to simulate train axle cross wedge rolling. The tangential, axial and radial forces were calculated to compare the differences between round and square billet rolling [11]. The wedge rolling process was developed using the numerical method based on the LS-DYNA and experimental method to analyze the process parameters of the complex profile rolling [12]. Theoretical and experimental rolling of bars on a three-high skew rolling mill of aluminum was investigated with a single pass. Stefanik et al. [13] determined the state of deformation, stress, and temperature distribution during the process. The working roller was considered in a four-high rolling mill using by finite element method based on the dynamic analysis. Kapil et al. [14] selected different parameters of the rolling process to calculate the forces acting on the working rollers. The dynamic model was performed to analyze the cold rolling process by combining the electromechanical model and the deformation zone model. The results based on kinematic parameters, force, and energy parameters were obtained [15]. Artificial neural networks were also used to predict the effect of Inter-Stand tensions in controlling on other parameters such as the rolling pressure, the rolling force, the forward and backward slips, and the neutral angle [16]. Jurkovic et al. [17] performed the real production conditions of the cold rolling process of channels section using the theoretical approach and the complex experimental procedure to measure the force and deflection for the rolls.

Aforementioned studies clearly reveal that the simulation of the cold rolling process is difficult. In addition, there is also an urgent need to understand the behavior of stresses and plastic strains produced during the process. In this research, AA5049 aluminum, AISI1015 low carbon steel, and 304L stainless steel materials were considered in rectangular bar shape. The objective of this paper is to use the finite element method based on Explicit Dynamic/ABAQUS to study effects of the reduction of area, the friction coefficient between surfaces of rollers and bar, as well as rollers speed on von Mises stress and equivalent plastic strain during the process.

![Figure 1. A schematic of rolling process with deformation](image)

### 2. THE FINITE ELEMENT MODEL OF COLD ROLLING PROCESS

The cold rolling process was analyzed by finite element method. The three-dimensional dynamic explicit model was developed by using a commercially available software ABAQUS™. The rectangular cross-section bar was considered as a deformable body and two rollers were defined as rigid bodies with a constant diameter and thickness 140 mm and 30 mm, respectively. Initial dimensions of the workpiece were constant 40 mm in length, 20 mm in width, and 8 mm thick while final dimensions were changed and depend on the reduction of area. The reductions of 19%, 25%, and 31% were selected for reducing the initial thicknesses to 6.5 mm, 6 mm, and 5.5 mm respectively. AA5049, AISI1015, and 304L stainless steel materials were considered. The mechanical properties for these materials as well as the true stress and true plastic strain were applied according to the available data in literature [6,18-19]. The assembly definition of the symmetry rolling model includes two parts: the roller and the initial rectangular bar. The assembly was mated using the “dependent option” to assemble these two parts. The “translate instance option” was used to move parts to their correct positions.

In this study, the explicit dynamic analysis method was applied. It is defined as a mathematical technique for integrating the equations of motion through time. This technique is used to solve the problems that contain large deformation and nonlinear material response as in the case of the cold rolling process. Due to the contact between rollers surface and material surface, the penalty method with finite sliding for the surface to surface was utilized to resolve the tangential behavior of a mechanical contact [20]. The Coulomb friction is the basic role to define
compressive force that is proportional to the penetration of the material. Tangential behavior, and magnitude of friction coefficient were considered. Coefficients of friction 0.1, 0.2, and 0.3 were selected for all simulation models.

Definition of boundary conditions were quite difficult. Therefore, it must be applied correctly for obtaining the accurate results with a minimum error. For the cold rolling process, the initial boundary condition was the velocity of the bar while roller speed defined as an angular velocity. In this study, the liner velocity of the bar was constant 350 mm/sec that was defined according to the predefined field option. Different angular velocities were given to rotating the rollers and the reference center point was fixed at the directions of (U1, U2, U3, UR1, UR2=0) while the direction of (UR3≠0). The roller speed was given by VR3 direction as 10, 20, and 30 rad/sec. Because the rolling model had a symmetry, the half model was considered. Meshes and elements were displayed in Fig. 2. Element type was C3D8R which has 8-node linear brick, reduced integration, hourglass control. The number of elements for the initial workpiece (bar) were 25600 while for the rigid roller, 2189 elements were created.

![Finite element meshing](image)

**Figure 2.** Finite element meshing

3. MODELS VALIDATION

Finite element modeling of the cold rolling process was validated with the experimental results in the literature [6], as presented in Fig. 3. For the validation procedure, AA5049, diameter of roll was 148.8 mm, and reduction of areas were 20, 35, and 50%. The initial thickness was 6.13 mm and reduced to 4.91 mm, 3.94 mm, and 3.02 mm. The dimensions of the strip were 38.1 mm in length and 43.6 mm in width. The friction coefficient was 0.23, roller speed was 12 rad/sec, and the linear velocity of the strip was 381 mm/sec. From the Fig. 3, the results show a good coherence between the finite element and experimental results with some variation. It can be seen that both the simulation and experimental models have the same behavior, the roll force increases with increasing the reduction of area due to the increase in plastic deformation at inlet and outlet of forming zones. This validation proves that the initial boundary conditions for the simulation models were correctly applied. Experimentally, the roll forces were recorded 149, 212, and 225 kN at the reduction of areas of 20, 35, and 50%. For the finite element results, the roll forces were calculated as 165, 235, and 280 kN at the reduction of areas of 20, 35, and 50%. These results clearly proved the validation of model.

![Comparison of finite element and experimental results](image)

**Figure 3.** Comparison of finite element and experimental results
4. FINITE ELEMENT RESULTS

Simulations of the cold rolling process with explicit dynamic analysis software ABAQUS has been performed with different materials and rolling parameters. The relationships between these parameters and the von Mises stress and the equivalent plastic strain for various materials were shown, as well as the distribution of stress and plastic strain were presented.

4.1 von Mises Stress with Different Materials and Reduction of Areas

The Fig. 4 illustrates the distribution of von Mises stresses for various material at the reduction of area 31%, friction coefficient 0.1, roller speed 10 rad/sec and step of time 0.075 sec. It can be observed that the values and distribution of von Mises stress are different for each material. It is normal that the AA5049 has the least of strength and maximum von Mises stress of 351.3 MPa was measured. The maximum von Mises stress for AISI1015 and 304L stainless steel 705.6 and 712 MPa were recorded respectively.

![Figure 4. von Mises stress distribution for (a) AA5049, (b) AISI1015, and (c) 304L stainless steel](image)

The Fig. 5 shows the relationship between maximum von Mises stress and the reduction of area for different materials when friction coefficient of 0.2, roller speed of 10 rad/sec, and the step time of 0.1 sec. It can be seen that the von Mises stress increases with increasing the reduction of area for all materials due to the increase in plastic deformations in contact zone between material and rollers. The reason for this phenomenon was the increase in reduction of area which needs the increased roll force [6]. This led to the increase in plastic deformation and von Mises stresses. For the AISI1015 and 304L stainless steel, the maximum von Mises stress begins to approach to each other when the reduction of area increases.

![Figure 5. Relationship between maximum von Mises stress and reduction of area for different materials](image)

4.2 Equivalent Plastic Strain with Different Materials and Friction Coefficients

The Fig. 6 presents equivalent plastic strain distribution for different materials at friction coefficient of 0.1, reduction of area of 25%, roller speed of 10 rad/sec and step time of 0.09 sec. The figure shows that the maximum of equivalent plastic strain PEEQ for the 304L stainless steel was 0.4754 while the maximum of equivalent plastic strain PEEQ for the AA5049 and AISI1015 were 0.3842 and 0.411 in order. This means that the hard material (304 stainless steel) gives the largest deformation compared to AISI1015 and AA5049. The plastic deformations were increased at the contact zone between the material and the rollers because of the friction while at the center of the thickness of bar the plastic deformations were the least.
Fig. 6. PEEQ distribution for (a) AA5049, (b) AISI1015, and (c) 304L stainless steel

Fig. 7 shows the relationship between maximum of equivalent plastic strain PEEQ and friction coefficient for different materials at the reduction of area of 19%, roller speed of 10 rad/sec, and the step time of 0.075 sec. It clearly observed that the equivalent plastic strain increases with increasing the friction coefficient. Because of the contact between the surface of the rolls and the material surface, the friction forces try to prevent the movement of material at rolling direction that leads to the increase of plastic deformation.

4.3 von Mises Stress with Different Materials and Roller Speed

The Fig. 8 indicates the relationship between maximum von Mises stress and roller speed for different materials when reduction of area was 31%, friction coefficient of 0.3, and the step time of 0.12 sec. Generally, the maximum von Mises stress increases with increasing roller speed due to the increase of the roller work on the material [21]. Form the figure, the maximum of von Mises stress for AA5049 were 351, 380, and 390 MPa at roller speeds of 10, 20, and 30 rad/sec respectively. On the other hand, the maximum von Mises stress for the AA5049 was the least compared to the two other materials. For both the AISI1015 and 304L stainless steel, the maximum of von Mises stresses were close to each other compared to the AA5049 due to the difference of yield limit for the material. The maximum of von Mises stress for AISI1015 was calculated 750 MPa at roller speed of 30 rad/sec. For the 304L stainless steel, the maximum von Mises stress were almost constant and determined as 760, 760.4, and 760.5 MPa at roller speeds of 10, 20, and 30 rad/sec respectively. This means that the roller speed has minimum effect on the von Mises stress when the material is hard.
5. CONCLUSIONS

In this study, the finite element method was successfully utilized for the analyzing the cold rolling process. Different materials of AA5049, AISI1015, and 304L stainless steel and different process parameters of the reduction of area, the friction coefficient, and the roller speed were considered for the simulations of models. Following conclusions were drawn:

1. The roll force increases with increasing reduction of areas.
2. The material type has a great influence on the selection of cold rolling parameters to determine maximum von Mises stress and maximum equivalent plastic strain through the process.
3. Reduction of areas for the AA5049 has a great effect on the maximum von Mises stress while it is almost constant for the 304L stainless steel at all reduction of areas.
4. Equivalent plastic strain PEEQ increases with increasing the friction coefficient for all materials.
5. The most interesting point is that the effect of roller speed has the different effects on maximum von Mises stress. For the 304L stainless steel, there were little differences for the roller speed of 10, 20, and 30 rad/sec.

The recommendations for future works can be summarized as follows:

1. The effect of rolling parameters (roller diameter, reduction of area, roller speed, and friction coefficient) on axial and radial residual stresses of the bar using experimental and finite element method can be studied.
2. Analysis of chatter in cold strip rolling can be investigated both numerically and experimentally to determine the optimal parameters of process.
3. Comparison between cold and hot rolling process according to the rolling force and the temperature that give the least of residual stresses can be made.

REFERENCES


INVESTIGATION OF THE EFFECT ON AERODYNAMIC PERFORMANCE OF STEPPED DIFFUSER IN A LAND VEHICLE

Ismail ÖZEN, Yasin VAROL

1Kardeş Technical University, Engineering Faculty, Department of Mechanical Engineering, Trabzon, Turkey
2Fırat University, Technology Faculty, Department of Automotive Engineering, Elazığ, Turkey
3Correponding Author: Ismail ÖZEN, ismailozen@ktu.edu.tr

ABSTRACT

Vehicle aerodynamics is an important study field in terms of fuel consumption, road holding, maneuverability and driving stability of vehicles. Recently, with the rapid advancements in computer technology, automotive designers have made use of Computational Fluid Dynamics (CFD) to determine aerodynamic performance of the vehicles. In this study, flow around a land vehicle was experimentally and numerically investigated. In addition, the effect on aerodynamic performance of a stepped diffuser designed to the rear side of the vehicle was numerically determined. As a result, it was determined that aerodynamic drag coefficient decreases and road holding significantly increases depending on the angle of the stepped diffuser.

Keywords: Aerodynamics, Computational Fluid Dynamics, Fuel Consumption, Road Holding

1. INTRODUCTION

Wind tunnel and road tests are conducted to determine the main aerodynamic characteristics of vehicles. Large model sizes used for wind tunnel tests are resulted in a significant increase in test costs. However, in road tests, equipment and device costs used to simulate real road conditions are more expensive than wind tunnel tests. Recently, with developing computer technology, the interest in numerical studies has been continuously increased (Hucho et al., 1993). As flow around a land vehicle is generally turbulent, three-dimensional and time-dependent, it is not possible to model analytically by computers. Computational Fluid Dynamics (CFD) has been used to obtain the approximate solution of many problems that solution is not possible in fluid Dynamics. CFD provides visual graphics and animations to be determined the main aerodynamic characteristics by modelling the flow around the vehicle before the production stage. In addition, aerodynamic forces and moments can be calculated by CFD (Katz, 2006).

Vehicle aerodynamics is an important study field in terms of fuel economy, road holding and maneuverability of vehicles (Heisler, 2002). Tabacu et al. (2010) investigated the air flow around Sedan and Hatchback type vehicles by using a commercial CFD software and they determined that the flow behind the Hatchback type vehicle was more turbulent than the Sedan type vehicle. Prasad et al. (2014) investigated the air flow around a TATA Nano type vehicle. They made four different changes in the exterior body design of the vehicle and stated that the drag coefficient can be reduced by 14.3% thanks to these design changes. Barbut and Negrus (2011) stated that aerodynamic drag coefficient can be reduced 12.7% by a new design on the bottom side of a Sedan type vehicle. Moreover, they proved that numerical and experimental results were in good agreement with each other. Ince (2007) investigated the air flow around a vehicle model scaled of 1:4 and used k-epsilonon turbulence models, Reynolds stress model, k-omega turbulence models in numerical simulations. According to experimental and numerical results, he confirmed that the highest pressure acting on the vehicle model is in front nose side, whereas the lowest pressure is in where the windscreen and the ceiling meet. Selenbaş (2011) conducted numerical and experimental studies to examine the air flow around a heavy vehicle model scaled of 1:15. He determined that the drag coefficient can be decreased down to a certain level by design improvements.

Although laminar flows are solved by CFD, it is impossible to solve turbulent flows without any turbulence model. The sensitivity of any CFD solution for turbulent flow depends on the turbulence model chosen for the analyzes. The aerodynamic forces acting on the vehicle model depend on many factors such as its outer body design, road and
ambient conditions. Therefore, dimensionless numbers are used to compare aerodynamic performance of the vehicles with each other (Cengel and Cimbala, 2006). The drag and lift coefficients are important dimensionless numbers and they are respectively given in Eqs. (1) and (2).

\[ C_D = \frac{F_D}{\frac{1}{2} \rho V^2 A} \]  
\[ C_L = \frac{F_L}{\frac{1}{2} \rho V^2 A} \]  

C_D and C_L in the above equations represent the drag and lift coefficients, respectively. Furthermore, F_D and F_L are respectively the drag force and lift force acting on the vehicle. ρ is fluid density, V is vehicle velocity, A is projection area.

In this study, the air flow around a land vehicle was investigated experimentally and numerically. In addition, the effect on aerodynamic performance of a stepped diffuser designed on its rear side was numerically determined.

2. METHOD

2.1. Experimental Model

Wind tunnel tests are conducted to determine aerodynamic forces and moments affecting on the vehicles. As the cost of wind tunnel depends on sizes of the model to be used in the experiments, scaled models are used in the experimental studies. In this study, a vehicle model scaled of 1:24 was used as a verification model for the numerical simulations. Reynolds number similarity was implemented to determine the main aerodynamic characteristics related to the vehicle model scaled of 1:1 in real road conditions (Özen, 2015). The details of the wind tunnel used in the experiments are shown in Fig. 1.

Fig. 1. Wind tunnel test system

2.2. Numerical Model

Technical details of the vehicle model used for CFD analyzes were taken from the reference URL 1. Table 1 shows basic sizes and properties of the vehicle model used in this study.

<table>
<thead>
<tr>
<th>Height</th>
<th>Width</th>
<th>Length</th>
<th>Projection area</th>
<th>Model year</th>
<th>C_D</th>
</tr>
</thead>
<tbody>
<tr>
<td>1427 mm</td>
<td>1842 mm</td>
<td>4726 mm</td>
<td>2.2x10^6 mm^2</td>
<td>2008</td>
<td>0.27</td>
</tr>
</tbody>
</table>

2.2.1. Mathematical Modelling

Eqs. (3) and (4) show respectively mass and momentum conservation (Navier-Stokes) equations for turbulent flow case of a Newtonian fluid which is viscous, incompressible and without free surface effects.
2.2.2. Computational Fluid Dynamics

There are two basic approaches to design and examine engineering systems with flow interaction. Wind tunnel tests are required for the experiments, whereas CFD solves numerically differential equations regarding the flow field. The data related to the flow field can be obtained in detail by CFD.

2.2.2.1. Turbulent Flow Modelling

Modelling the turbulent flow are much more time consuming as it is irregular and time dependent. Computers having high capacity must be used to reduce solution time of CFD analyzes. There isn’t any turbulence model fully solving turbulent flow. However, the most accurate solutions can be obtained by using the most suitable turbulence model. When the literature on flow around vehicles is examined, it has been understood that the most widely used turbulence model is the standard k-epsilon turbulence model as it provides the best convergence for the turbulent flow. Recently, Realizable k-epsilon turbulence model has been become widespread as it gives more precise results. Therefore, Realizable k-epsilon turbulence model was used to precisely solve the turbulent flow around the vehicle model in this study. The standard k-epsilon turbulence model assumes that flow is full turbulent, and it is only valid for the turbulent flows, whereas the Realizable k-epsilon turbulence model takes flows in transition and turbulent zone into account (Fluent, 2006). The relation between kinetic energy, diffusion rate and turbulence viscosity for the turbulent flows is given in Eq. (5).

\[ \mu_t = C_\mu \frac{k^2}{\varepsilon} \]  

(5)

Generalized transport equations for k and \( \varepsilon \) in the Realizable k-epsilon turbulence model are given by tensor notation in Eqs. (6) and (7), respectively.

\[ \frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_j} (\rho k u_j) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon - Y_M + S_k \]  

(6)

\[ \frac{\partial}{\partial t} (\rho \varepsilon) + \frac{\partial}{\partial x_j} (\rho \varepsilon u_j) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \rho C_1 S_k - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{\varepsilon}} + C_3 \frac{\varepsilon}{k} C_3 \varepsilon G_b + S_\varepsilon \]  

(7)

Where \( k \) is the turbulence kinetic energy and \( \varepsilon \) is diffusion rate. \( G_k \) represents the kinetic energy generated due to the average velocity gradients. \( G_b \) is the turbulence kinetic energy generated due to lift forces. \( Y_M \) represents the contribution of the wave expansion in compressible flow to the overall diffusion rate. \( C_2 \) and \( C_{1\varepsilon} \) are the model constants. \( \sigma_k \) and \( \sigma_\varepsilon \) represent Prandtl numbers for \( k \) and \( \varepsilon \), respectively. \( S_k \) and \( S_\varepsilon \) are user-origin source terms.

2.2.3. ANSYS_Fluent Analysis

In this study, a computer having cache memory of 16 GB with the Intel 3.5 GHZ i7 processor was used for CFD analyzes.

2.2.3.1. Geometry

The vehicle model used for CFD analyzes was created in Solidworks program and then it was transferred to ANSYS_Fluent 15.0 software. In order to determine the effect on aerodynamic performance of a stepped diffuser designed to the rear side of the vehicle model, a new design was made. The geometric details of the stepped diffuser are given in Fig. 2.
2.2.3.2. Domain & Boundary Conditions

Domain sizes were determined to be proportional to the sizes of the vehicle model with a negligible blocking ratio as shown in Fig. 3. Velocity inlet at the inlet side of the domain and pressure outlet at the outlet side of the domain were chosen as boundary conditions.

In order to shorten analysis time, the domain and vehicle model were divided into two parts along the longitudinal axis and the symmetry boundary condition was chosen for mid-plane. The wall boundary condition type was chosen for other surfaces of the domain and all surfaces of the vehicle model. The vehicle model scaled of 1:24 was used to validate the wind tunnel test results. For this reason, wind tunnel tests and numerical analyzes were performed with increments of 2 m/s from 5 m/s to the maximum inlet velocity of 27 m/s. In addition, the effect on the aerodynamic performance of the stepped diffuser for flows having high Reynolds numbers was numerically investigated by using vehicle model scaled of 1:1. In the numerical analyzes using 1:1 scale, it was assumed that inlet velocity of air to the domain is 45 m/s, its density is 1.225 m$^3$/kg, its dynamic viscosity is 1.7894x10$^{-5}$ kg (ms), its relative pressure at the outlet side is 0 Pa. Length of the vehicle model was chosen as the characteristic length. It is understood that the flow around the vehicle model is turbulent as the value obtained from Eq. (8) for the Reynolds number is much higher than the value of 10$^5$, known as the turbulence limit for the flow on the plate.

$$Re = \frac{gx VX L}{\mu} = \frac{1.225 \times 45 \times 4.7}{1.7894 \times 10^{-5}} = 14479015$$  (8)

2.2.3.3. Mesh

Fig. 4 shows mesh structure generated for CFD model. Fine elements were used to precisely solve the flow around the vehicle model. Since the flow at the rear side of the model is highly turbulent, and there is flow restriction between the
bottom side of the model and the road; the fine elements at the rear and bottom side of the model were generated. It was ensured that CFD analysis results are independent of mesh structure.

![Mesh structure generated for CFD model](image)

**Fig. 4.** Mesh structure generated for CFD model

2.2.3.4. Wall Function

The non-equilibrium wall function was used to precisely solve velocity boundary layers formed on the surfaces of the model. The non-equilibrium wall function provides more precise solutions for model surfaces having complex geometry and flow separations (Fluent, 2006).

2.2.3.5. Fluent Solver

As the flow around the model was accepted as incompressible and steady, pressure based solver was chosen for the numerical analyzes. The pressure based solver linearizes equations for each control volume and integrates it with an algebraic equation for each iteration. Coupled method was used as a solution method for the analyzes. This method combines equations depending on the algorithm used during the analysis and provides better convergence when compared to other solution methods (Fluent, 2006). Second order upwind method was chosen as solution method to obtain more accurate results.

3. RESULTS & DISCUSSION

3.1. Model Verification &Validation

Fig. 5 shows variation versus Reynolds number of the drag coefficient which is directly related to the fuel consumption of the vehicle model. Drag coefficient obtained from experimental and numerical studies conducted for low Reynolds numbers is unstable. However, it exhibits a more decisive trend for high Reynolds numbers such as 350000.

![Variation of drag coefficient vs. Reynolds number](image)

**Fig. 5.** Variation of drag coefficient vs. Reynolds number

Fig. 6 shows variation depending on Reynolds number of the lift coefficient, an important parameter about road holding of the vehicle model. As can be seen from the figure, there is an incompatibility between lift coefficients obtained from experimental and numerical studies conducted for low Reynolds numbers. A reason for this could be the turbulence model used for the numerical study. Because, local Reynolds number over the vehicle model is low (Reynolds number $< 10^5$) and flow regime is laminar. As the turbulence model used in the numerical study can solve effectively the flow in the regions close to the transition zone from the laminar to the turbulence; the lift coefficients obtained from the
experimental and numerical studies for the Reynolds numbers which are close to the transition zone \( (5 \times 10^5) \) are in a good agreement with each other.

![Graph](image)

**Fig. 6.** Variation of lift coefficient vs. Reynolds number

Figs. 7a and 7b show velocity contours along the symmetry axes of the vehicle models without the diffuser and with the diffuser, respectively. Front sides of their ceilings have maximum flow velocity. An air flow of low velocity at the end of windshield leads to a recombined flow separation. Since the flow separation occurred at the rear side of the their ceilings, it causes to a flow zone having much lower velocity than the other sides. The flow restriction occurring on their undersides leads to a flow zone of high velocity. There is an air flow of lower velocity at the rear side of the vehicle model with the diffuser.

![Velocity Contours](image)

**Fig. 7.** Velocity contours around the vehicle models: a) without the diffuser and b) with the diffuser \( (\theta=3.5^\circ) \)

Figs. 8a and 8b show static pressure contours along the symmetry axes of the vehicle models without the diffuser and with the diffuser, respectively. From Figs. 7a and 7b, flow zones having higher velocity have lower static pressure as shown in Figs. 8a and 8b. This case indicates that Bernoulli Equation is provided.

![Pressure Contours](image)

**Fig. 8.** Pressure contours around the vehicle models: a) without the diffuser and b) with the diffuser \( (\theta=3.5^\circ) \)

Static pressure contours over the vehicle models are shown as 3D in Figs. 9a and 9b. As can be seen from the figures, high pressure zones are formed on their nose sides due to flow stopping. There is a rapid pressure increase over the windshields. Therefore, a downward thrust effect is occurred on windshields. Since the flow velocity increases in the region between the end of windscreen and the beginning of ceiling, the flow pressure decreases down to the negative level. This case results in a suction effect to pull up the vehicle models. In the upper side of their luggage compartments, the static pressure increases up to the positive level and it causes to a downward thrust effect at the rear side of the vehicle models.
Fig. 9. Pressure contours over the vehicle models: a) without the diffuser and b) with the diffuser (θ=3.5°)

Figs. 10a and 10b show velocity vectors along the symmetry axes on the rear sides of the vehicle models without the diffuser and with the diffuser, respectively. Vortex movements occur due to the flow separation on their rear sides. This vortex movement decrease the contact between surface and air, and they lead to a decrease in the surface friction. There is less vortex zone at the rear side of the vehicle model with the stepped diffuser.

Fig. 10. Velocity vectors behind the vehicle models: a) without the diffuser and b) with the diffuser (θ=3.5°)

Figs. 11a and 11b show streamlines along the symmetry axes of the vehicle models without the diffuser and with the diffuser, respectively. Since a high pressure zone occur at their nose regions, streamlines are separated from the surfaces. Due to the surface geometry at their back sides, the air molecules could not be attached to the surface and this case causes streamlines not to follow the surface geometry.

Fig. 11. Streamlines around the vehicle models: a) without the diffuser and b) with the diffuser (θ=3.5°)

Aerodynamic drag and lift coefficients obtained from CFD simulations for different diffuser angles (2.5°, 3°, 3.5°, 4°, 4.5°, and 5°) are given in Table 2. Air intake to the engine, roughness on the underside of the vehicle, windscreen wipers, door borders, wheel rims etc. were neglected for solid drawing of the vehicle model without the diffuser scaled of 1:1 in numerical simulations. Therefore, its aerodynamic drag coefficient was obtained as 0.2398 with an error of 11.2% according to to its actual value. When the diffuser angle is 3.5°, the drag coefficient is decreased down to 2%, and the lift coefficient is continuously increased in negative level depending on increment in the diffuser angle.
Table 2. Variations depending on diffuser angle of aerodynamic drag and lift coefficients

<table>
<thead>
<tr>
<th>Vehicle type</th>
<th>θ (°)</th>
<th>(C_D)</th>
<th>(C_L)</th>
<th>Δ(C_D)</th>
<th>Δ(C_L)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Without diffuser</td>
<td>-</td>
<td>0.2398</td>
<td>-0.0020</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>With diffuser</td>
<td>2.5</td>
<td>0.2378</td>
<td>-0.0023</td>
<td>-0.002</td>
<td>-0.0002</td>
</tr>
<tr>
<td>With diffuser</td>
<td>3</td>
<td>0.2367</td>
<td>-0.0179</td>
<td>-0.0034</td>
<td>-0.0158</td>
</tr>
<tr>
<td>With diffuser</td>
<td>3.5</td>
<td>0.2350</td>
<td>-0.0306</td>
<td>-0.0048</td>
<td>-0.0286</td>
</tr>
<tr>
<td>With diffuser</td>
<td>4</td>
<td>0.2367</td>
<td>-0.0321</td>
<td>-0.0031</td>
<td>-0.0301</td>
</tr>
<tr>
<td>With diffuser</td>
<td>4.5</td>
<td>0.2394</td>
<td>-0.0355</td>
<td>-0.0004</td>
<td>-0.0335</td>
</tr>
<tr>
<td>With diffuser</td>
<td>5</td>
<td>0.2394</td>
<td>-0.0509</td>
<td>-0.0004</td>
<td>-0.0489</td>
</tr>
</tbody>
</table>

4. CONCLUSION

In this study, the air flow around a land vehicle was experimentally and numerically investigated. In addition, the effect on aerodynamic performance of a stepped diffuser designed at the rear side of the vehicle model was determined numerically. The results obtained from the study are given in below:

1. Flow around a land vehicle model can be effectively modeled by making use of CFD.
2. Road holding can be increased significantly by a stepped diffuser designed at the rear side of the vehicle.
3. More realistic solutions related to the flow around the vehicle model can be obtained by using different mesh strategies and more advanced turbulence models with a computer having higher hardware and functional features.

REFERENCES

SHEET METAL FORMING PROCESSES FOR VARIOUS MATERIALS USING
FINITE ELEMENT ANALYSIS

Jabbar Gattmah¹, Muzher Taha Mohamed², Suha K. Shihab³

¹,²,³ Department of Material Engineering, College of Engineering, University of Diyala, Diyala, Iraq

Corresponding Author: Jabbar Gattmah, E-mail: msc_jgi_katma7@yahoo.com

ABSTRACT

Sheet metal forming process is one of the traditional manufacturing operations when the metal formed by a punch and die to convert a flat sheet of material into a part of the desired shape without fracture. In this present study, sheet metal forming based on V-bending die is selected to form the copper alloy, aluminum alloy, and steel alloy. Three-dimensional models based on finite element analysis were constructed using commercial ABAQUS/Standard to simulate sheet metal bending models. Also, the model of finite element analysis is successfully validated with experimental model obtained from the previous study. Effects of punch radius, plate thickness, and friction coefficient on tension and compression stress are shown in detail. The main results revealed that the change in sheet plate thickness has a strong influence on the tension and compression stress while friction coefficient slightly effects on the stresses.

Keywords: Sheet metal forming, finite element analysis, V-bending die, tension and compression stress

1. INTRODUCTION

Sheet metal forming process is known as a metal forming into thin and flat pieces. It is considered one of the fundamental forms in metalworking due to obtaining different shapes of products. The main tools of sheet metal forming are die and punch as well as stamping press where the final shape of the product can be determined according to the geometry of die and punch. The major types of sheet metal forming are shearing (cutting), bending, drawing, and squizing [1]. All these types produce high plastic deformation and stresses in forming zone due to the compression force that applies to form the metal. Generally, experimental and finite element method are most important techniques applied in metal forming processes such as sheet metal forming, extrusion, and drawing processes [2-3]. The plastic deformation, the elastic deformation or spring back is one of the major defects that are generated at end of the sheet metal bending process and thereby affected the dimensional accuracy of a finished part. Selecting appropriate sheet metal bending parameters, such as tool design i.e. punch radius, speed of punch, angle of bend, and friction value, is significant due to affecting these parameters on plastic deformation and induced stresses during the process [4]. Some investigations were presented to study springback in sheet metal bending process using experimental and finite element methods. Lagrangian formulation using finite element method was developed to determine the effect of bend angle on camber press during V-bending process [5]. Also, Finite element analysis based on a total-elastic-incremental-plastic (TEIP) algorithm was utilized to show the influence of parameters of sheet metal bending process on springback [6]. Experimental and finite element analysis was achieved to predicate the springback that produced in U-bending models of sheet stainless steel [7]. V-bending of stainless steel 304 sheet was achieved using both the experimental and finite element method to study the influence of the punch radius, die radius, die gap, punch travel, and punch velocity on the springback behavior [8]. Finite element analysis based on ANSYS™ LS – DYNA™ was modelled to investigate the influence of sheet metal thickness, friction value, tool radius, and tool shape on spring back on different materials during sheet metal bending process [9]. Taguchi method based on DOE was utilized to analyze various parameters i.e. punch angle, die opening, and sheet width on spring back of sheet metal bending of CR2 grade steel sheet of IS 513-2008 material [10]. Another study was carried out to estimate the stresses and deformation for different sheet metal and parameters of the bending process using
Finite element analysis based on ANSYS software [11]. Experimental procedure and finite element analysis based on the LS-DYNA were performed to analysis of sheet mild steel bending at different thickness of sheet and different die angle [12]. An incremental sheet metal hammering process was done to analyze the effect of diameter and frequency of punch on the hardness, tensile strength, and grain size during bending od Al-100-O aluminum alloy [13]. Simulation of the incremental sheet aluminum was investigated using finite element method based on ABAQUS software. The model of the finite element method was validated with the experimental model and there was a good agreement [14]. Sheet metal process of stainless steel also was carried out by laser under temperature gradient mechanism. The temperature field and deformation of sheet metal at various laser powers were determined using finite element method. The experimental and finite element results were in a good correlation according to the different angle bending [15]. Neural network based on feedforward was applied to identify the final bend angle of sheet metal process that was described by two stages which are bent and drawn [16]. Experimental and numerical method were used to analyze the damage of sheet DC04 cup that was formed according to the gas detonation forming to produce complex geometries including sharp angles and undercuts, in a very short time. The finite element method was also applied to simulate 3D computational models with explicit dynamic analysis [17].

The most of previous studies deal with the effect of process parameters on springback, however, still sheet stresses produced by elastic-plastic deformations which take place through the metal bending under high load need to more understanding. In this study, three-dimensional finite element analysis using ABAQUS/ Standard software was performed to study the effect of punch radius, plate thickness, and friction coefficient on tension and compression stress through the process.

2. MODELLING OF THE FINITE ELEMENT ANALYSIS

In this study, sheet metal V-bending was modelled using three-dimensional finite element analysis based on ABAQUS/ Standard software. The modeling contains three parts: die and punch were considered as a rigid body and sheet plate was defined as a deformable body. Fig.1 reveals to the schematic of sheet metal bending type V-die before and after bending. The die geometry includes: bend angle ($\alpha = 90^\circ$), opening die ($w_d = 100$ mm), length ($L_d = 250$ mm), and thickness ($H_d = 70$ mm) as well as die radius that is similar to punch radius. The dimensions of the punch are depth of the bending ($w_p = 50$ mm), width ($L_p = 98$ mm) and height ($H_p = 50$ mm). Different punch radiusses ($R = 5, 10, 15, \text{and} 20$ mm) were applied in the modeling. The sheet plate was simulated with 130 x 20 mm cross section at various of sheet plate thicknesses ($t = 1, 2, 3, 4$ mm).

![Figure 1. Schematic sheet metal V bending process (a) before bending (b) after bending](image)

Three types of alloys viz Cu alloy, AA5086 alloy, and AISI1008 alloy were used to simulate sheet plate bending models using three-dimensional finite element analysis. The mechanical properties needed in simulation for used alloys and the materials are assumed to be isotropic elastic-plastic behavior. These magnitudes of mechanical properties as well as the true stress and true plastic strain were determined by Refs. [18-20]. The solid homogenous type was utilized to consider the assignment section of the material. The die, punch, and sheet plate...
were assembled using the “assembly option” then these parts are mated together by “translate instance option” to obtain a correct position for the parts.

The sheet bending process includes a non-linear static analysis with applying structural low-speed dynamic. This type of problem can be solved using ABAQUS/Standard by selecting static/general analysis to simulate sheet metal bending models [21]. Surface to surface contact type is required during the bending process. First is contact between V-die surface and the lower surface of the sheet plate and second is contact between V-punch surface and the upper surface of sheet plate. Also, master and slave in the region of contacts constraint were defined using the surface to surface discretization. Tangential behavior with the magnitude of the friction coefficients of 0.1, 0.15, 0.2, and 0.25 were considered.

Boundary conditions for both the die and punch were set to fix the die in the directions of U1, U2, U3, UR1, UR2, and UR3 while the punch moved in the direction U2≠0. The displacement of punch represents the depth of sheet plate bending and it was calculated according to the radius of punch and the thickness of sheet plate. Table 1 reports the punch displacement (S) with the different radius of punch (R) and sheet plate thicknesses (t) that used in this study.

<table>
<thead>
<tr>
<th>R (mm)</th>
<th>S (mm)</th>
<th>R (mm)</th>
<th>S (mm)</th>
<th>R (mm)</th>
<th>S (mm)</th>
<th>R (mm)</th>
<th>S (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>48.8</td>
<td>5</td>
<td>47.8</td>
<td>5</td>
<td>46.8</td>
<td>5</td>
<td>45.8</td>
</tr>
<tr>
<td>10</td>
<td>46.3</td>
<td>10</td>
<td>45.3</td>
<td>10</td>
<td>44.3</td>
<td>10</td>
<td>43.3</td>
</tr>
<tr>
<td>15</td>
<td>43.8</td>
<td>15</td>
<td>42.8</td>
<td>15</td>
<td>41.8</td>
<td>15</td>
<td>40.8</td>
</tr>
<tr>
<td>20</td>
<td>41.3</td>
<td>20</td>
<td>40.3</td>
<td>20</td>
<td>39.3</td>
<td>20</td>
<td>38.3</td>
</tr>
</tbody>
</table>

The type of element that was used for meshing is C3D8R (8-node linear brick, reduced integration hourglass control). The number of elements was selected according to the geometry for each part. Fig. 2 shows details of meshing for sheet metal bending where the element number of 2786, 1484, and 10400 were used for the die, punch, and sheet plate respectively.

**Figure 2. Mesh of sheet metal bending**

### 3. VALIDATION OF NUMERICAL MODEL

For the reliability with the real practical bending process, the validation plot of predicted results that obtained from 3D models of bending forces vs different values of punch displacement was performed through the comparison with the experimental results of the previous study [22] as represented in Fig. 3. It can be seen from figure 3 that there is a good convergence between the predicted and experimental results that means the initial boundary condition was truthy applied. The small observed difference may be due to experiments setup or noise.
factors. For the validation, the bend angle, die radius, and punch radius are 89.58°, 5.37 mm, and 2.89 mm in order. The dimensions of $100 \times 58 \, mm$ and thickness of 1 mm were applied. The sheet plate material that used for validation is FeP01 (Germany system) that is equivalent to SAE1008 (USA System) [23]. The yield stress is 285 MPa, ultimate stress represents 340 MPa, and density equals 7.87 g/cm$^3$ [24].

For the validation, the bend angle, die radius, and punch radius are 89.58°, 5.37 mm, and 2.89 mm in order. The dimensions of $100 \times 58 \, mm$ and thickness of 1 mm were applied. The sheet plate material that used for validation is FeP01 (Germany system) that is equivalent to SAE1008 (USA System) [23]. The yield stress is 285 MPa, ultimate stress represents 340 MPa, and density equals 7.87 g/cm$^3$ [24].

4. RESULTS OF FINITE ELEMENT SIMULATION

Different materials (Cu alloys, AA5086, and AISI1008) were used in finite element modeling based on ABAQUS/Standard to study the effect of punch radius, sheet plate thickness, and friction coefficient on upper and lower peak stresses at V- bending metal during the process.

4.1 Stress Distribution

Fig. 4 shows the stresses distribution in bend zone with the complete intersection of sheet plate. The punch radius ($R$) is 5 mm, sheet plate thickness ($t$) equals 3 mm, and the friction coefficient ($f$) represents 0.1. It can be seen that the surface of the sheet plate which formed by punch surface i.e. upper surface of V peak has compression stress (negative value) while the surface of sheet plate which calibrated by die surface viz lower surface of V peak has tension stress (positive value). Moreover, it was observed that the AISI1008 alloy records higher stresses than both the Cu alloy and AA5086 alloy. This attributed to material properties which have impact effect on the value of compression and tension stresses.

![Figure 4. Stress distribution for three different material (a) Cu alloy, (b) AA5086 alloy, and (c) AISI1008 alloy](image)
4.2 Effect of Punch Displacement

The relationship between internal stress (at upper and lower surface of V peak) and punch displacement is displayed in Fig. 5 at constant punch radius \((R = 10 \text{ mm})\), sheet plate thickness \((t = 3 \text{ mm})\), and friction coefficient \((f = 0.1)\). It can be detected from the figure that the stresses of tension and compression increase gradually with the increase the punch displacement then almost constant. The states of internal stresses are so complex because of the contact between tools surfaces and sheet plate surfaces where the material is subjected to the elastic and plastic deformation during the process due to the effect of applied force bending. Aluminum alloy (AA5086) recorded less stresses than the copper alloy (Cu) and steel alloy (AISI1008). For the copper alloy and steel alloy, the behavior of internal stresses is the same at both the punch side (upper contact between bunch surface and sheet plate surface) and die side (lower contact between die surface and sheet plate surface) while the behavior of internal stresses for the aluminum alloy was more increased with the increase punch displacement. It can be argued that the maximum internal stresses (compression and tension) depend on the punch displacement [22].

![Figure 5. Stress vs punch displacement plots at different materials](image)

4.3 Effect of Punch Radius

The relationship between internal stress (at upper and lower of V peak) and punch radius is exhibited in Fig. 6 with constant values of sheet metal thickness at \((t = 3 \text{ mm})\) mm, friction coefficient at \((f = 0.1)\) and punch displacement at \((S = 35 \text{ mm})\). The figure shows that the punch radius has a strong effect on the compression and tension stress. It can be observed at the punch radiiuses of 5 mm and 10 mm, the stresses are almost constant while the stresses decrease at the punch radiiuses 15 mm and 20 mm. Also, it can be noticed from the figure that the alloy steel records higher stress than the copper and aluminum alloy.

![Figure 6. Stress vs punch radius plots at different materials](image)
4.4 Effect of Sheet Plate Thickness

Fig. 7 indicates the relationship between internal stress (at upper and lower of V peak) and sheet plate thickness at punch radius \((R = 20 \text{ mm})\), friction coefficient \((f = 0.2)\), and punch displacement \((S = 30 \text{ mm})\). It was found that for the three materials, the tension and compression stress increase with increasing sheet plate thickness. Thus due to increasing the bending force with increase the sheet plate thickness which resulted in an increase in the internal stress. So that the internal stresses mainly depend on the bending force.

![Figure 7. Stress vs sheet plate thickness plots at different materials](image)

4.5 Effect of Friction Coefficient

Fig. 8 refers to the relationship between internal stress (at upper and lower of V peak) and friction coefficient at punch radius \((R = 15 \text{ mm})\), sheet plate thickness \((t = 3 \text{ mm})\), and punch displacement \((S = 40 \text{ mm})\). The figure demonstrates that there is a very little effect of friction coefficient value on both the tension and compression stress where the stresses are almost constant for the three materials.

![Figure 8 Stress vs friction coefficient plots at different materials](image)
5. CONCLUSIONS

In current study, copper alloy, aluminum alloy, and steel alloy were utilized in sheet metal forming process based on V-bending die using three-dimensional finite element analysis. Effect of different punch radius, sheet plate thickness, and friction coefficient on internal stresses at upper contact surface (punch and sheet plate contact) and lower contact surface (die and sheet plate contact) were studied. The obtained simulation results were summarized as follows:

1. The bending force gradually increases with increasing the punch displacement then sharply increases when the sheet plate completely is formed.
2. The tension and compression stresses during the sheet bending process depend on punch displacement and force bending.
3. The sheet plate thickness has a great significant effect on the pattern of the tension and compression stress. These stresses increase with increasing the sheet plate thickness hence needs to the increase in the bending force.
4. The tension and compression stress at the punch radius of 15 mm and 20 mm are less compared to the punch radius of 5 mm and 10 mm.
5. Friction coefficient has a little influence on the state of tension and compression stress.

REFERENCES


THE INVESTIGATION OF STRESS DISTRIBUTION ON THE TRACTOR PTO WET CLUTCH COVER BY USING FINITE ELEMENT METHOD

Özgür Erdoğan

Valeo Automotive Systems, Bursa, Turkey
Corresponding Author: Özgür Erdoğan, ozgur.erdogan@valeo.com

ABSTRACT

Wet clutches are able to transfer high torque values by means of hydraulic pressure and multi-frictional surfaces. As a consequence of this capability, they have very common application areas including agricultural vehicles such as tractors. Parallel to technological development, tractors have engines with higher power and torque levels compared to previous decades. Therefore, the tractor manufacturers prefer to use wet clutch packs in drivetrain systems. In this study, stress distribution of the tractor PTO (Power take off) wet clutch cover is investigated by using finite element method based on real working conditions. The PTO wet clutch cover design is validated for the over-torque situation. The results show that the PTO wet clutch cover can withstand 3.6 times of the engine torque.

Keywords: Tractor, Wet clutch, Power take off, Finite Element Analysis

1. INTRODUCTION

In powertrain systems, torque is transferred from the engine to the gearbox. Wet clutches are quite compact compared to the conventional dry clutches, and they are capable of transferring high torque values. One of the application areas of the wet clutches is in agricultural activities. Tractors are exposed to severe working conditions in the field depending on the land characteristics and agricultural equipment that is why high performance is demanded from tractors. Wet clutch usage on the tractors may be for traction side, PTO (Power Take-Off) or both of them. PTO wet clutch position on the tractor is shown in figure 1.

![Figure 1. PTO wet](image)
Studies on the wet clutch in terms of durability are quite limited. Previous studies are focused on the wear of the wet the clutch facing. Parameters affecting the wear on the facings were investigated by using bench tests.

Ost et al. (2001) in their study of wet clutches used in wet clutches SAE II and pin-on testing mechanisms and examined the changes in wear and friction characteristics. As a result of the study, although the result of the pin-on test system cannot be obtained according to the results of SAE II wear on the lining of the bridges and the important parameters affecting the performance of the wet clutch are interpreted. Li et al. (2015) prepared a test mechanism in order to measure instant wear on the facings and made the repetitive engagement. As a consequence of this study, the theoretical model regarding two stage wear was improved. Ost et al. (2001) made a test with 18590, 11140 and 30455 cycles with aiming the determine the wear characteristics of the wet clutch. Two of that cycles ended up due to high vibration and the other finished because of the accident. The thickness of the facings was measured before and after the test. As a result of this study, it was observed that abrasion was faster at the beginning. When facing surface become more stable, abrasion getting slower. Lloyd et al. (1998) investigated that working conditions of wet clutch and parameters which affect wear on the facing. After 100 and 500 cycles, wear amounts were measured. Result of this study claimed that abrasion is proportional with inertia, rotational velocity and energy. Erdoğan and Solmaz (2017) conducted field test with tractor and investigated dissipated energy during engagement of PTO wet clutch for three different conditions. In this study, the rotational velocity of engine and PTO shaft were recorded. Resistive torque was applied to PTO shaft with a dynamometer. Facing thicknesses were measured before and after the test. The lifespan of the wet clutch was estimated based on facings wear capacity. Güneş et al. (2012) worked on wet clutch system design and calculation for the farming tractor. Testing of new design under different conditions were done. As a consequence of this study, optimization of system components were done and adaptation of this wet clutch design to new generation tractors was explained. Anderson (1972) one of the first researcher for wet clutches, divided engagement phenomenon to three phases. The first phase was named as compression and modeled with Reynolds equations. The second phase was named as crushing and during contact of facings, surface crushings were obtained. The third phase was named as adhesive wear and rough wears due to a mechanical contact between facings were observed. As a result of this study, the importance of hydraulic quality was proven in terms of torque transmissibility capacity. Karan (2012) performed a design study of wet clutch for a tractor with power shuttle transmission. Calculations were done and dynamic analyses were performed. Assembly and leakage resistance were taken into account during this study.

Different than the literature studies, the objective of the present study is that the mechanical strength of the wet clutch cover investigation by using the finite element method.
2. MATERIAL AND METHOD

Producing a real part and testing it are very costly steps. Eliminating this cost and time saving have a critical role for companies. In order to save time and spend less money, computer-aided design (CAD) and finite element analysis (FEA) is well-known and commonly applied technic. New geometries and new designs can be transferred to 3D data easily with CAD programs and their endurance limits, fatigue performance and maximum stress locations can be investigated with FEA software. Thereby, concept design studies may be low cost.

As the first step of the study, a solid model of the wet clutch cover is created in CATIA V5 software program as shown in figure 3. The wet clutch cover is made of cast iron GJS 500 and material mechanical properties are given in table 1.

![Figure 3. Solid model of the wet clutch cover](image)

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Material</td>
<td>GJS 500</td>
</tr>
<tr>
<td>Poisson’s Ratio</td>
<td>0.275</td>
</tr>
<tr>
<td>Modulus of Elasticity</td>
<td>169 GPa</td>
</tr>
<tr>
<td>Yield Strength</td>
<td>320 MPa</td>
</tr>
<tr>
<td>Tensile Strength</td>
<td>500 MPa</td>
</tr>
</tbody>
</table>

Following of solid model creation, geometry is imported into Ansys Workbench 17 for the creation of finite element model. In the mesh model, tetrahedral elements are used to create mesh construction that is consisting of approximately 302000 elements and 480000 nodes (Figure 4).
Loads and boundary conditions, which are applied to wet clutch cover during analysis are defined in Ansys. The analysis is divided into different steps for convergence and to observe stress levels at different hydraulic pressure. In order to investigate the durability of the wet clutch cover, different torque and hydraulic pressure are applied to the solid model (Table 2).

**Table 2.** Time dependent load values

<table>
<thead>
<tr>
<th>Steps</th>
<th>Time[s]</th>
<th>Torque [Nm]</th>
<th>Hydraulic Pressure [MPa]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>500</td>
<td>0.1</td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>1000</td>
<td>0.5</td>
</tr>
<tr>
<td>3</td>
<td>3</td>
<td>1500</td>
<td>0.8</td>
</tr>
<tr>
<td>4</td>
<td>4</td>
<td>2000</td>
<td>1.5</td>
</tr>
<tr>
<td>5</td>
<td>5</td>
<td>2500</td>
<td>1.8</td>
</tr>
<tr>
<td>6</td>
<td>6</td>
<td>3000</td>
<td>2.3</td>
</tr>
</tbody>
</table>

Based on geometric constraints, two way directional movement is restricted by defining displacement value as “0 mm” to the hub surface. The second displacement is defined to the input shaft contact surfaces as opposite direction of the torque. Cylindrical support is defined to hydraulic entrance ring contact surface. All loads and boundary conditions are shown in figure 5. In order to decide whether a wet clutch cover is validated or not, maximum principal stress is investigated.
3. RESULTS AND DISCUSSION

The linear static finite element analysis is performed by Ansys static structural model. Maximum stress location and stress distribution throughout wet clutch cover are observed. Additional to these results, maximum displacement value and location are identified. The highest stress value is 140 MPa at 500Nm and occurred at bottom surface of the input shaft spline (Figure 6). Maximum displacement is observed at the same point as shown in figure 7 and the value is 0.00096mm.

In Turkey, most of the farm tractors which use PTO wet clutch have engine up to 500Nm torque capacity. Since the maximum principle stress is below the material limit, which is 500MPa, the PTO wet clutch cover is validated for the
maximum engine torque condition with 3.6 safety factor. On the other hand, for the maximum hydraulic pressure condition safety factor is 1.2. Torque vs. stress graph is given at figure 8.

![Figure 8. Torque versus stress graph.](image)

4. CONCLUSION

In this paper, mechanical durability of the PTO wet clutch cover is investigated in terms of the over-torque situation by using finite element method. Simulation model is created and boundary conditions are defined based on the real working conditions. The results showed that the PTO wet clutch cover is overdesigned in terms of overtorque conditions. Hence, if other design criterias are also met lower grade material can be used for lower cost. For the future studies, design validation can be investigated with bench tests and the correlation between FEA results and test results can be investigated.

REFERENCES


AN ARTIFICIAL NEURAL NETWORK (ANN) APPROACH FOR SOLUTION OF THE TRANSCENDENTAL EQUATION OF LONGITUDINAL VIBRATION

Gözde Rabia Aktaş¹, Abdullah Emül², Sadettin Orhan²

¹Faculty of Engineering, Department of Mechanical Engineering, Bursa Uludağ University, Bursa/Turkey
²Faculty of Engineering and Natural Sciences, Department of Mechanical Engineering, Ankara Yıldırım Beyazıt University, Ankara/Turkey

agozderabia@uludag.edu.tr,b aemul16@gmail.com,csadettinorhan@ybu.edu.tr

ABSTRACT

Study of mechanical vibration is one of the major issues in engineering applications. Especially, during the design and test stages of a mechanical component or system, vibration must be considered. When a vibration issue is studied theoretically, a differential equation called characteristics equation or equation of motion (EOM) is obtained. A solution of EOM gives vibrational behavior of an object considered. When vibration of a continuous system is studied, a transcendental equation is finally obtained, whose solution by classical methods is not possible. In this study, the solution of the transcendental equation derived from the longitudinal vibration of a bar with one end fixed and a mass at the other end was studied. For this purpose, an ANN model was constructed. The datasets were created for the ANN model. The effects of the number of neurons, input data, and training function on the model were examined. In addition, multiple regression models were developed using the ANN data and natural frequency formulation was obtained by ANN analysis for each mode. A finite element modal analysis was performed by ANSYS software. The results obtained by ANN and ANSYS were compared with analytic calculation and it was shown that they were in enough agreement.

Keywords: Artificial Neural Network (ANN), Longitudinal Vibration, Finite Element Method (FEM)

1. INTRODUCTION

Today's engineering systems are becoming increasingly complex in terms of design and materials. As a result, new methods are being developed to determine more accurately and practically the dynamic behavior of a multiplicity of degrees of freedom or large structures. One of the most basic features of engineering constructions is the vibrations in these structures. Actually, all structures in nature have an infinite number of vibration frequencies and mode shapes. Calculation of frequencies of these structures and their mode shapes are significant to solve the vibration induced engineering problems [1-4]. Vibration analysis of structural systems has been performed using different methods [5-13]. Some researchers were used approximate and exact solution method to calculate natural frequencies. William F. Stokey [14] studied the natural frequencies of the longitudinal vibration of a uniform rod having a rigid mass attached to it. He obtained approximate natural frequencies by Rayleigh’s Method. These methods contain differential equations that dynamic behavior of a structure is defined by these equations, and solved with the aid of them. There are various difficulties encountered while solving with differential equations. The reason is that the differential equations are interdependent [15]. Natural frequencies of longitudinal vibration of a beam that attached a mass can be also solved by using commercial finite element packages such as ANSYS, ABAQUS etc. As for the solution process with the help of computer software increases the time loss when compared to the solution process with the help of artificial neural networks (ANN). Because of these difficulties, in this study, it was tried to be found the Natural Frequencies for a beam that carry a mass with the aid of ANN. Artificial neural networks have been used to solve a wide variety of
problems in many areas of engineering, and they continue to be used increasingly today. Gates et al. presented a method of using artificial neural networks to stabilize large flexible space structures. In this study, they showed the neural controller learns the dynamics of the structure to be controlled and constructs a control signal to stabilize structural vibrations[16]. Ding et al. used artificial neural networks with the aim of structural dynamics-guided for locating and quantifying damage in beam-type structures[17]. Bağdatlı et al. investigated nonlinear vibration of stepped beams having different boundary conditions with ANN[18]. Lazarevska et al. presented a study that used some of the positive aspects of the neural network’s model that used for determination of fire resistance of construction elements[19]. Flood and Christopilos studied construction processes using artificial neural networks (ANN). Their goal was to evaluate a neural network approach to modeling the dynamics of construction processes that exhibit both discrete and stochastic behavior, providing an to the more conventional method of discrete-event simulation[20]. Jeng et al. In their study an artificial neural network (ANN) were applied to several civil engineering problems, which had difficulty to solve or interrupt through conventional approaches of engineering mechanics[21]. Furthermore, in a series of research on the artificial neural network by Toktas et al. [22-24] were presented. They used ANN for solving some problems. Such as, experimental investigation of the effects of uncoated, PVD- and CVD-coated cemented carbide inserts and cutting parameters on surface roughness in CNC turning and its prediction, contemporary analyses in assessing residual stress topographic images enclosing a cold expanded hole and chain gear design. They showed that artificial neural network (ANN) is an emerging research field. In addition, artificial neural networks (ANN) techniques have been used to solve the complex problems of various engineering branches, such as mechanical, electrical, computer and other engineering fields. This method is increasingly used in the engineering field as it facilitates solving complex engineering problems.

2. STATEMENT OF THE PROBLEM

When the vibrations of continuous systems, especially bars and beams have been studying, a characteristic equation in the type of a transcendental equation is obtained. These characteristic equations have no exact analytical solution. In this study, an artificial neural network approach was proposed for the solution of the transcendental equation derived from the longitudinal vibration of a bar with one end fixed and a mass at the other end, as shown in Fig.1. If the solution procedures of a continuous system are applied to the longitudinal vibration of a bar with one end fixed and a mass at the other end, the following transcendental characteristic equation is obtained [25].

\[ \alpha_n \tan \alpha_n = \beta, \quad n = 1, 2, ..., \quad (1) \]

with \( \alpha_n = \frac{\omega_n}{c} \) or \( \omega_n = \frac{\alpha_n c}{l} \) \quad (2)

Where \( \beta = \frac{m}{M} \); and \( m \) is the mass of the bar.

\[ \beta = \frac{\alpha l}{M} = \frac{m}{M} \quad (3) \]

As it is seen, the transcendental equation depends on the mass ratio (\( \beta \)). \( \beta \) is known, the neural frequency is unknown.
The main goal is to find the natural frequencies of the bar. To achieve this, an artificial neural network approach was applied to this problem. Since the bar is a continuous system, it has an infinite number of natural frequencies. In this study, only the first six natural frequency is considered for simplicity. Fifty mass ratio ($\beta$) is considered as $\beta=0.5; 0.55; 0.6; 0.65; 0.7; 0.75; 2.75; 2.8; 2.85; 2.9; 2.95; 3$ and the corresponding natural frequencies were obtained. This solution will be obtained by an artificial neural network (ANN) method in the subsequent parts.

3. ANALYSIS OF SYSTEM

3.1. FINITE ELEMENT ANALYSIS

There is a need to get natural frequencies and corresponding mode shapes of structures for a convenient design and avoiding resonance. Modal analysis is a process in which natural frequencies and corresponding mode shapes of a considered structure are determined. In other words, the process is a free vibration analysis. For simple systems, a theoretical modal analysis is easy but for complex structures or continuous systems, it is difficult compared to simple systems. Nowadays, finite element programs are very useful to analyze any kind of engineering problems. ANSYS program is widely used in engineering problems. In this study, modal analysis of the bar with one end fixed and carrying a mass at the other end was carried out by ANSYS. The goal of modal analysis in this section is to confirm whether ANN and analytically calculated natural frequencies are correct or not. The result that we found As a first, 3D model of the bar was created then a Finite Element(FE) model was obtained. Secondly, boundary conditions were applied, analysis options were defined as it is shown Fig.2 through Fig.4. Later, the first six natural frequencies and the corresponding mode shapes were determined for longitudinal vibration as shown Fig.5.

![Fig. 2 3D model](image1)
![Fig. 3 FE model](image2)
![Fig. 4 Boundary condition](image3)

![Fig. 5 Mode shapes of longitudinal vibration of a bar carrying a mass](image4)

In this study, the main goal is to obtain the natural frequencies easily using equations developed with the ANN approach. For this purpose, the steps given below were followed.

- The transcendental equation was solved by a numerical method (Wolfram).
- Longitudinal vibration of a bar with one end fixed and a mass at the other end was analyzed by Finite Element software ANSYS and the first six natural frequencies were obtained.
➢ The first six natural frequencies are obtained by using ANN prediction and modeling system for a bar with one end fixed and a mass at the other end.
➢ The obtained values by using the numerical method, the finite element method using Ansys software and ANN estimation method were compared with each other.
➢ The values obtained were very close to each other. From $\beta (m/M)$ versus longitudinal natural frequency graph, a function was created for each mode by using the values of ANN. In this way; longitudinal natural frequency can be found easily by using the rod mass and an end mass.
➢ Numerical, Finite Element and ANN results were compared. And the average errors were given in Table 1. and Table 2.

<table>
<thead>
<tr>
<th>Mode Number</th>
<th>Ansys</th>
<th>Analytical</th>
<th>%Error</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1042.4</td>
<td>1012.198</td>
<td>2.983</td>
</tr>
<tr>
<td>2</td>
<td>5191.6</td>
<td>5101.202</td>
<td>1.772</td>
</tr>
<tr>
<td>3</td>
<td>10011</td>
<td>9856.881</td>
<td>1.563</td>
</tr>
<tr>
<td>4</td>
<td>14876</td>
<td>14684.694</td>
<td>1.302</td>
</tr>
<tr>
<td>5</td>
<td>19666</td>
<td>19532.130</td>
<td>0.685</td>
</tr>
<tr>
<td>6</td>
<td>23734</td>
<td>24387.580</td>
<td>2.679</td>
</tr>
</tbody>
</table>

Table 1 Comparison of Ansys and Analytical values

![Comparison of Ansys and Analytical values](image)

3.2. ARTIFICIAL NEURAL NETWORK(ANN)

With the development of technology; analysis programs, simulation software, engineering simulation programs etc. have been used frequently. One of the main reasons why these programs are frequently preferred is the fact that the analyzes and the predictions are quite realistic. In this study, artificial neural networks called estimation program used. The term ANN stand for an artificial neural network. ANN is a system which is modeling of a human biological brain with the simplest description. Also, they use defined rules to achieve appropriate results for a problem[23]. Similarly; ANN is a way that to make a smart program with using the model of the human brain that simulate the working network of the neurons [4]. To explain briefly, biological neural networks consist of many neurons. A neuron has different shape and size depend on its function. It is important to examine the way neurons work and their activities to construct artificial neural networks. Ann is a set of neuron clusters that work for a specific purpose and can be seen as a black box. A set of neuron clusters work as processing elements. The process of collecting data, processing the collected data and sending the results to the relevant element is made through each processing element[23]. There are three main neural network structure elements which are the input element, the output element, and a hidden layer. The hidden layer can be at least one or more. Firstly, input signals are accepted at the input layer. It passes through the hidden layer. Finally, it reaches the output layer of the artificial neural network model. Also, the backpropagation algorithm is the most useable algorithm for the multi-layer network because the mathematical program has complex, nonlinear relations. MSE stands for Mean Square Error. Mse is a performance index for Backpropagation algorithm. In this algorithm, the error is a difference between a target value and network value. Mse should be minimum [26]. Shortly, the ANN algorithm has some steps to predict the data. Artificial neural network method and working principle can be explained as follows. These steps are written below respectively.

- Data collection
- Training and testing data separation
- Select Network architecture
- Parameter tuning and weight initialization
- Data transformation
The artificial neural network method works with the MATLAB software program. In the ANNs, input values, test function and training function are used to obtain output values. In this study, the analytical calculation is made firstly. Education and test data are prepared for learning and good estimation. Hidden layer, the number of repetitions, and the number of neurons are determined before running the program and these values are entered when running the code in the MATLAB program. Training ends in two ways. First is an error level has been reached the target. Second, iteration is repeated until the end. Also, the purpose of the test data is to understand whether the artificial neural network method makes a good estimate. Test data is data that has never been used in education. The results are compared and checked with the test data. At the same time, some statistical methods are used to make comparisons like R2, RMSE, MEP [23]. In figure 6, the ANN model is shown basically. In this study, three hidden layers are used. As the table illustrates the network with three hidden layers of [2+9+11+1] neurons at each layer has provided the best results (Fig. 7).

![Fig. 7 The basic artificial neural network model](image)

In fig. (8, 9, 10, 11) the multiple-correlation coefficients and comparison between linear regression and ANN for training, validation, and testing were shown.

![Fig. 8 Training Plot](image)  ![Fig. 9 Validation Plot](image)  ![Fig. 10 Testing Plot](image)  ![Fig. 11 Over All Plot](image)

Fig. 12, the error histogram in the complete training process is shown. As for the fig. 13, the performance plot shows that MSE becomes small while the number of epochs is increased.

![Fig. 12 Error histogram plot for training data](image)  ![Fig. 13 Best validation performance plot of ANN](image)
Artificial neural network (ANN) modeling was used for developing and then train the simulation of a solution of the transcendental equation of longitudinal vibration. Results obtained from, ANSYS, Analytical and ANN prediction was compared by the use of statistical error analysis methods. Here, mean error percentages were significantly small for training and testing. These different approaches were in agreement.

Table 2 Comparison of Ansys-Analytical, ANN-Analytical and Ansys-ANN values

<table>
<thead>
<tr>
<th>Mode</th>
<th>Ansys</th>
<th>Analytical</th>
<th>%Error</th>
<th>ANN</th>
<th>Analytical</th>
<th>%Error</th>
<th>Ansys</th>
<th>ANN</th>
<th>%Error</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ø1</td>
<td>1042.4</td>
<td>1012.198</td>
<td>2.983</td>
<td>1012.3</td>
<td>1012.198</td>
<td>0.010</td>
<td>1042.4</td>
<td>1012.3</td>
<td>2.887</td>
</tr>
<tr>
<td>Ø2</td>
<td>5191.6</td>
<td>5101.202</td>
<td>1.772</td>
<td>5101.3</td>
<td>5101.201</td>
<td>0.001</td>
<td>5191.6</td>
<td>5101.3</td>
<td>1.739</td>
</tr>
<tr>
<td>Ø3</td>
<td>10011</td>
<td>9856.881</td>
<td>1.563</td>
<td>9866</td>
<td>9856.881</td>
<td>0.092</td>
<td>10011</td>
<td>9866</td>
<td>1.448</td>
</tr>
<tr>
<td>Ø4</td>
<td>14876</td>
<td>14684.694</td>
<td>1.302</td>
<td>14736</td>
<td>14684.694</td>
<td>0.349</td>
<td>14876</td>
<td>14736</td>
<td>0.941</td>
</tr>
<tr>
<td>Ø5</td>
<td>19666</td>
<td>19532.130</td>
<td>0.685</td>
<td>19532</td>
<td>19532.13</td>
<td>0.0006</td>
<td>19666</td>
<td>19532</td>
<td>0.681</td>
</tr>
<tr>
<td>Ø6</td>
<td>23734</td>
<td>24387.580</td>
<td>2.679</td>
<td>24388</td>
<td>24387.58</td>
<td>0.001</td>
<td>23734</td>
<td>24388</td>
<td>2.755</td>
</tr>
</tbody>
</table>

The following figure and equations indicate beta versus longitudinal natural frequency. Beta is the ratio of the mass of the bar to the attached mass. Natural frequencies are the values obtained as a result of artificial neural network method. When the beta values are substituted into the equations, the natural frequencies can be obtained without a need for solving the transcendental equation. The natural frequency values obtained by the artificial neural network method were very close to the results obtained from Ansys and analytical solutions. If the mass of the bar and the value of the attached mass are known, then the natural frequencies of the longitudinal vibration of the structure can be easily calculated by using equations shown in the Fig.16.
Fig. 16 Equation of frequencies for longitudinal vibration
4. CONCLUSION

Artificial neural network method is one of the most suitable learning methods for nonlinear, complex and dynamic tasks. ANN is frequently used and preferred as one of the modeling and prediction methods. One of the main reasons for its preference is due to its quite accurate results. The artificial neural network is widely used to get quick results to suitable problems with obtained data. In this study, an ANN approach has been applied to, the solution of the transcendental equation of longitudinal vibration of a bar for the first six modes. The natural frequency predictions are obtained as equations for the first six natural frequency. ANNs prediction gave a good result with the minimum error. It is seen that ANN results are good agreement with Ansys and analytical, results. Using ANN data, longitudinal natural frequencies were calculated for each mode. If the mass of rod (m) and the end mass (the attached mass) are known, natural frequencies can be found easily from the obtained function for the transcendental equation. (β=0.5; 0.55; 0.6; 0.65; 0.7; 0.75—2.75; 2.8; 2.85; 2.9; 2.95; 3). It is seen that the application of ANN approach to the solution of the transcendental equation is possible and gives accurate results. The method can be applied for more modes than six. It can also be applied to the solution of the similar transcendental equations in different fields.

REFERENCES


[24] İ. Toktaş, H. Başak “Chain Gear Design Using Artificial Neural Networks”


Optimum Design of Rubber Damper on Clutch Disc Using Response Surface Methodology

Mehmet Onur Genç¹, Süleyman Konakçı¹, Necmettin Kaya²
¹ Valeo Automotive Systems, Bursa, Turkey
² Bursa Uludag University, Mechanical Engineering Department, Bursa, Turkey

1 Corresponding Author: Mehmet Onur Genç, mehmet-onur.genc@valeo.com

ABSTRACT

Clutch disc has a mission of torque and vibration damping coming from the vehicle engine during the internal combustions. Proper damping is one of the important tasks for clutch disc that safely provides torque transmission during all driving conditions. Elastomer rubbers are widely used on vehicle industry because of their high damping ability and low price. In this study, behavior of rubber damper in the clutch disc is firstly investigated with finite element method. Rubber are modeled using hyperelastic material model in finite element model. Then, in order to obtain the desired stiffness in terms of NVH (noise, vibration, harshness) performance, design of experiment table was created according to the selected geometric parameters. Furthermore, curve fitting study was implemented in order to have an equation that replaces the FEM model. This equation, which is called the response surface, will be used for the purpose of optimization in obtaining the desired stiffness in future study.

Keywords: Rubber damper, Automobile Clutch damper, Response surface methodology, Hyperelastic modelling, Clutch disc design, Vehicle powertrain system, Design of experiment

1. INTRODUCTION

Clutch that allows gear shifting during driving has the importance for torque transmission in powertrain systems. One of the tasks of clutch is damping vibration generating by engine. At conventional clutch disc, metallic damper springs are widely used for many applications. Recently elastomer type polymers such as rubbers’ usage have been increasing and preferred in many areas. In clutch technology, rubber dampers need to be well designed and optimized in parallel to vehicle dynamics. One of the most important parameters is stiffness for the vehicle comfort due to its effect on torsional filtration related to noise, vibration and harshness (NVH) characteristics. Figure 1 shows a clutch assembly and rubber damper.

Figure 1. Clutch System (1-Flywheel, 2-Disc, 3-Cover assembly) and rubber damper
Although there are some research about modeling of rubber materials, there is no study about rubber damper on clutch disc. Genc et al. (2018) studied hyperelastic and viscoelastic modelling for the clutch rubber dampers. They made time-dependent viscoelastic finite element analysis (FEA) by taking into consideration dynamic forces in vehicle. Genc et al. (2017) investigated hyperelastic modelling of rubber and made correlation between experimental test data and FEA. Kaya (2014) performed rubber bushing optimization by using differential evaluation (DE) algorithm. In the study, material testing and mathematical basis of DE algorithm studied and related constraints for rubber bushing stiffness were explained. Wu et al. (2016) studied hyperelastic and viscoelastic modelling parameters.

The optimization of the rubber damper is necessary for the clutch to have certain vibration characteristics. In order to have target stiffness curve of rubber damper, design of experiment and response surface methodologies have been used. Damper rubber has been modeled with finite element model. Hyperelastic material model is defined according to test data.

2. MATERIAL and METHODS

Hyperelastic Modelling

Rubber damper was modeled with finite element method. Material model parameters are defined using test data. Ogden N=3 material model was chosen as hyperelastic material model. Figure 2 shows the rubber damper finite element model compressed between two rigid plates as the boundary conditions. Total 27 analyses were performed for three parameters (depth, diameter, length) at three different levels.

![Figure 2. Hyperelastic FEA – compression analysis](image)

Design of Experiment (DOE) Studies

If the model used in design optimization is a finite element model, the optimization solution time can be very time consuming. Therefore, the approximate model (response surface) that replaces the finite element model is used. In this approach, analyzes are performed at the sampled points with DOE, the curve is fitted to the results obtained and the new function is created. This function will be used for optimization.

In this study, depth, diameter and length were chosen as shape parameters for the DOE studies (Figure 3). Three levels have been defined for each parameters. $3^3=9$ cases were listed and chi-square method has been performed to have target stiffness of rubber damper.
Figure 3. Rubber damper shape parameters

Figure 4 explains the chi-square calculation which represents the deviation of statistical case. In order to have target stiffness curve, chi-square value should be minimum. $U_{\text{target}_i}$ represents the target stiffness value at specified displacement, also $U_{\text{calculated}_i}$ is the calculated FEA result at the specified displacement. Chi-square calculation is shown as below (eq.1).

$$\text{Chi-square} = \sum_{i=1}^{N} \frac{(U_{\text{calculated}_i} - U_{\text{target}_i})^2}{U_{\text{target}_i}}$$  (1)

Figure 4. Chi-square calculation template
3. ANALYSIS and DISCUSSIONS

In this section the shape parameters for the rubber damper have been determined and finite element analyses has been performed. Upper and lower limits of shape parameters have been defined as:

Shape parameters:

Length: 18.8 < x < 19.0
Diameter: 16.8 < y < 17.2
Depth: 3 < z < 7

At equal intervals, each parameters were divided into sub-values called levels. For the length, 18.8, 18.9, 19.0; for diameter, 16.8, 17.0, 17.2; and for depth, 3, 5, 7 values were taken into consideration during finite element analysis. Following the finite element analysis, chi-square study have been performed and statistical deviation were calculated between FEA results and target stiffness values.

Table 1 shows the full factorial design table which includes the total 27 cases at each parameters. Calculated chi-square values are used in order to provide stiffness equation by using data fitting method. After curve fitting, response surface was created and stiffness equation was obtained (eq.2)

Table 1. Factorial design table for rubber damper

<table>
<thead>
<tr>
<th>Case No</th>
<th>Length (x)</th>
<th>Diameter (y)</th>
<th>Depth (z)</th>
<th>Chi-Square</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>18.8</td>
<td>16.8</td>
<td>3</td>
<td>29,4013</td>
</tr>
<tr>
<td>2</td>
<td>18.8</td>
<td>16.8</td>
<td>5</td>
<td>29,278</td>
</tr>
<tr>
<td>3</td>
<td>18.8</td>
<td>16.8</td>
<td>7</td>
<td>35,4322</td>
</tr>
<tr>
<td>4</td>
<td>18.8</td>
<td>17</td>
<td>3</td>
<td>7,10299</td>
</tr>
<tr>
<td>5</td>
<td>18.8</td>
<td>17</td>
<td>5</td>
<td>10,7618</td>
</tr>
<tr>
<td>6</td>
<td>18.8</td>
<td>17</td>
<td>7</td>
<td>13,8343</td>
</tr>
<tr>
<td>7</td>
<td>18.8</td>
<td>17.2</td>
<td>3</td>
<td>2,15036</td>
</tr>
<tr>
<td>8</td>
<td>18.8</td>
<td>17.2</td>
<td>5</td>
<td>3,89284</td>
</tr>
<tr>
<td>9</td>
<td>18.8</td>
<td>17.2</td>
<td>7</td>
<td>4,22423</td>
</tr>
<tr>
<td>10</td>
<td>18.9</td>
<td>16.8</td>
<td>3</td>
<td>39,9167</td>
</tr>
<tr>
<td>11</td>
<td>18.9</td>
<td>16.8</td>
<td>5</td>
<td>42,7512</td>
</tr>
<tr>
<td>12</td>
<td>18.9</td>
<td>16.8</td>
<td>7</td>
<td>45,6809</td>
</tr>
<tr>
<td>13</td>
<td>18.9</td>
<td>17</td>
<td>3</td>
<td>22,8438</td>
</tr>
<tr>
<td>14</td>
<td>18.9</td>
<td>17</td>
<td>5</td>
<td>23,9842</td>
</tr>
<tr>
<td>15</td>
<td>18.9</td>
<td>17</td>
<td>7</td>
<td>26,3617</td>
</tr>
<tr>
<td>16</td>
<td>18.9</td>
<td>17.2</td>
<td>3</td>
<td>15,4967</td>
</tr>
<tr>
<td>17</td>
<td>18.9</td>
<td>17.2</td>
<td>5</td>
<td>15,1622</td>
</tr>
<tr>
<td>18</td>
<td>18.9</td>
<td>17.2</td>
<td>7</td>
<td>20,1936</td>
</tr>
<tr>
<td>19</td>
<td>19.0</td>
<td>16.8</td>
<td>3</td>
<td>62,2803</td>
</tr>
</tbody>
</table>
Objective function is defined as min chi-square function \( f(x) \). This function is given as:

\[
f(x) = -1662042.03 + 160316.8119x + 8077.896835x^2 - 401.83237x^3 + 134857.05164yz + 637.6758865xz - 17.13613276x^2z + 444.3375812yz - 12.7536694y^2z + 11549.35821z^2 + 1.107137723xz^2 - 1.333743022yz^2 - 769.8227843z^3
\]

Figure 5 shows the comparison between FEA and calculated data with stiffness equation found by data fitting method. According to results the biggest deviation was calculated as 28% at 18\textsuperscript{th} row, in addition the best approach has been found at 5\textsuperscript{th} row with 0.33%. Therefore this function can be used to find shape parameters to have target stiffness curve.

In the continuation of this study, this equation will be solved by a global optimization algorithm and optimum shape parameters will be found.

4. CONCLUSION

In this study rubber damper’s stiffness curve has been modelled by response surface methodology. Rubber damper design was firstly created at different level of shape parameters such as length, diameter and depth. Then design of experiments and the full factorial sampling method have been performed. Chi-square values were calculated and the stiffness equation
were obtained as objective function for optimization. This study enables to model of rubber spring prior to production phase with desired stiffness value in automotive clutch systems. This function will be optimized by using one of the selected new generation optimization algorithm at future study.

References

1- Genc, M. O., Konakci, S., Kaya, N., ‘Experimental and numerical approach on viscoelastic modelling of rubber clutch damper spring’, 9th International automotive congress (OTEKON), pp. 101-109, May 2018
2- Genc, M. O., Konakci, S., Kaya, N., ‘Elastomer spring modelling for automobile clutch disc’, 1st International conference of advanced materials and manufacturing technologies (ICAMT), pp. 95-98, October 2017
3- Kaya, N., ‘Shape optimization of rubber bushing using differential evolution algorithm’, The scientific world journal, Volume 2014, ID 379196
ANALYSIS OF DENTAL IMPLANT ACCORDING TO NUMBER OF COMPONENTS IN TERMS OF APPLICABILITY AND MECHANICAL BEHAVIOR

Hojjat Ghahramanzadeh Asl1*, Salim Çam1, Osman Orhan1, Özgü Bayrak1

1 Erzincan Binali Yıldırım University, Mechanical Engineering Department, 24100, Erzincan, Turkey
* Corresponding Author: Hojjat Ghahramanzadeh Asl, hojjat.g@erzincan.edu.tr

ABSTRACT

Dental implants have been used among dentists to provide a concrete base for teeth construction and long use of advantage. Even though implants have been used by dentists, they have to be evaluated by engineering approaches. In this study, three different dental implants, which have same geometry, designed with SolidWorks 2018 software as single-part, double-part and three-part model. Models were placed to simulated mandibular. Finite element analysis with commercial software ANSYS 18.0 have been performed with this model to compare bone deformations and stress. From these results, stresses on mandible (cortical bone) decreased by around ~49% from three-part to double-part. Stresses on implants decreased by around ~26% from three-part to double-part. With regards to deformations at mandible, significant fluctuation has not been observed. Whether ease of application or stress levels, double-part implant has been found suitable for teeth replacement.

Keywords: Dental Implant, Finite Element Analysis, Cp-Ti, Cortical, Trabecular

1. INTRODUCTION

Dental health is essential for humans since it even affects life expectancy [1]. According to World Health Organization, the main reason of the tooth loss is caries and periodontal diseases [2]. Implant dentistry is a convenient way for restoring lost teeth and it has been widely used in recent years. Size and geometry of a dental implant differs depending on the location of the tooth. One of the most important factors of dental implant is the number of components. It could be produced as single or multi-part. Single-part dental implants are placed in jawbone as a whole while placement of three-part dental implants comprises multi-steps. There are various studies in the scientific literature and each one of them considers different geometries and designs. In a study conducted by Aumnakmanee et al. [3], different commercially available threads which have a thread size of 4 mm (Buttress, Trapezoidal and Reverse Buttress) were compared to their model. They have carried out finite element analysis for each thread type and recorded stresses on the bone and implant. They indicated that maximum stress value on their implant model was reduced when compared to other commercial thread types. Significance of the osseointegration of the dental implant was investigated by Marcian et al. [4]. They have conducted finite element analyses considering different bone densities. The study revealed that partial osseointegration was a potential threat to the reliability of the dental implant. They have also noted that biomechanical conditions in the oral cavity can be affected by undesirable modifications due to remodeling of the trabecular bone. NarendraKumar et al. [5] have designed different geometries by changing thread size and angle. It was stated that the dental implant which has a 60° of thread angle showed distinctly varying stress distributions when compared to other types (20°, 30° and 45°). They have also concluded the lowest von Mises stress value was achieved when the thread angle was 45°. Dhatrak et al. [6] were designed three dental implants which have different thread profiles and conducted finite element analyses for 6 different zones in the bone to obtain critical regions. They have concluded that the area of the maximum stress at the cancellous bone around implant thread has been measured as 0.5-1 mm, 0.7-1 mm and 0.8-1.5 mm for square profile, V profile and reverse buttress profile respectively.

This study aims to investigate effects of part number of the dental implant with respect to the stresses in the bone and implant. For this purpose, single-part, double-part and three-part dental implants were designed by using SolidWorks® software. Then, finite element analyses were carried out by using commercial finite element analysis software ANSYS®. Finally, stresses and deformations which were occurred at bone and implant were investigated.
2. MAIN BODY

2.1. Material and Method

The geometry which represents mandible characteristics, was modelled in 3D considering outcomes of previous studies in scientific literature. Figure 1 shows geometries of dental implants used in this study. All models have same geometry with lower diameter 1.5 mm, upper diameter 3 mm, implant length 10.5 mm, abutment length 4.5 mm and total length 15 mm.

![Figure 1. 3D Models of 1) single-part, 2) double-part and 3) three-part dental implants](image)

Firstly, dental implants were placed to the bone and the bone which was filled by implant was extracted. Then models were transferred to the finite element analysis software. Figure 2 shows dental implants which are placed to the bone.

![Figure 2. Dental implants which are placed the simulated mandible bone [7] 1) single-2)double-3) three-part.](image)

Finite element models contain 844708, 861913 and 912942 elements for single-part, double-part and three-part dental implants respectively. In order to obtain as precise results as possible, mesh metrics such as mesh quality and skewness were considered. Average mesh quality and skewness values were obtained as 0.83 and 0.23 respectively. Table 1 shows mechanical properties of the materials used in the finite element analysis.

<table>
<thead>
<tr>
<th>Material</th>
<th>Modulus of Elasticity (MPa)</th>
<th>Poisson Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>CP-Ti (Grade 4)</td>
<td>110000</td>
<td>0.37</td>
</tr>
<tr>
<td>Cortical Bone</td>
<td>13700</td>
<td>0.3</td>
</tr>
<tr>
<td>Trabecular Bone</td>
<td>1370</td>
<td>0.3</td>
</tr>
</tbody>
</table>
Finite element analyses were performed by considering the axial forces acting on the mandible via implant. The bottom surface of the 3D model was fully constrained and a compression force of 300 N was applied to the dental implant (Figure 3). Also, connections in the finite element model were customized to obtain reasonable results.

2.2. Results and Discussion

There are some factors that need to be considered in analysis of dental implants such as deformations and stresses which occur in cortical and trabecular bone as well as implant. Figure 4-5 shows maximum von Mises stresses and maximum deformations along y-direction for implants and bones for single-, double- and three- part implants. The 1,2 and 3 numbers at x-axis represent single- part, double- part and three- part dental implants respectively.

Figure 3. Finite element model details a) boundary conditions b) applied force c) mesh structure.

Figure 4. Maximum von Mises stresses

Figure 5. Maximum y-directional deformations
Deformations and stresses occurred at cortical bone by implant models are given in Figure 6. When comparing all models according to finite element analysis results, the lowest von Mises stress at cortical bone was obtained from the model which used double-part dental implant.

Finite element analysis results showed that the highest maximum von Mises stress (65.7 MPa) and y-directional deformation (0.0086 mm) at cortical bone was obtained from the model which used the three-part dental implant. The best result was achieved when double-part dental implant was used. Maximum value of the von Mises stress was measured as 33.67 MPa and cortical bone deformed 0.0078 mm along y axis for double-part implant. From Figure 6, it can be seen that the location of the maximum von Mises stress at cortical bone for the single-part dental implant is different from others. This stress occurred at the top surface of the cortical bone where was in contact with the single-part dental implant. The single-part implant showed limited flexibility because of its rigidity and stresses concentrated at the top surface of the cortical bone. However, maximum stress located at the contact region of the trabecular and cortical bones for double-part and three-part dental implants. This indicates, stresses were transmitted to the cortical and trabecular bones via implants. The location of the maximum von Mises stresses and y-directional deformations at trabecular bone are also given in Figure 7.

The highest maximum stress at trabecular bone (9.73 MPa) was obtained from the model which uses the double-part dental implant. Also, highest value maximum deformation along y axis (0.0072 mm) at trabecular bone was obtained from the model which uses the three-part dental implant. The lowest value of the maximum stress at trabecular bone was
obtained from the three-part dental implant. Because the area corresponding to transmitted force by implant is wider when compared to other models. The stresses and deformations occurred at implants are also given in Figure 8.

![Figure 8](image)

**Figure 8.** Location of maximum stresses and deformations at implants a) stresses b) y-directional deformations by implant type.

The maximum (297.58 MPa) and the minimum (219.1 MPa) stresses at implants were obtained from three-part and double-part dental implants respectively (Figure 8a). Also, the same fashion was observed for y-directional displacements. The three-part dental implant deformed 0.0137 mm while the double-part dental implant deformed 0.0124 mm. Figure 8(b) shows the amount of y-directional deformations for each type of model. The lowest deformation at implant’s bottom region was observed at the single-part dental implant while double-part and three-part implants showed relatively higher deformations. This situation is also related to rigidity of the single-part dental implant while three-part implants transmitted the load to the bottom of the implants via connection zones.

3. CONCLUSIONS AND RECOMMENDATIONS

3.1. Conclusions

The finite element analysis results revealed that the best result was obtained from the double-part dental implant with respect to stresses and deformations occurred at both implant and bone. The double-part dental implant neither shows rigid behavior alike the single-part dental implant nor transmits the load excessively alike the three-part dental implant. The double-part implant offers ease of application and advantage of integration between the bone and the implant. Also, in case of a damage it is easier to replace abutment and does not require surgical operations when compared to other implant types.
References


WIND ENERGY POTENTIAL IN ECEABAT, TURKEY

1Abdulah Düzcan, 2Yusuf Ali Kara

1,2 Bursa Technical University, Mechanical Engineering Department, Bursa, Turkey
1 Corresponding Author: Abdullah Düzcan, abdullah.duzcan@btu.edu.tr

ABSTRACT

Wind energy usage and investments have been rapidly increasing all over the world owing to the detrimental effects of limited fossil fuels. Evaluating of wind energy potential is a crucial step to determine to build wind power plant in all over the world. Gathering hourly wind direction and speed data from Turkish State Meteorological Service enables us to determine the wind potential in Eceabat. The wind energy potential assessment is conducted with the help of Windsim, a commercial CFD software that combines the roughness of the ground and wind data. The results show that establishing a wind power plant in Eceabat is strongly suggested.

Keywords: Wind energy potential, Eceabat, Windsim, Turkey

1. INTRODUCTION

Energy demand has been rapidly rising due to industrialization all over the globe. In order to meet the high energy need, conventional power plants are not built because of the harmful effects to human life and the environment. Instead of having fossil fuel based power plants, constructing renewable energy based power plants such as wind, solar, hydro, geothermal, wave and biomass is the focus of the countries. [1]

The shape of Turkey is like a peninsula and the mountains in the Aegean Region are orthogonal to the Aegean Sea. That is why the wind potential is sufficiently high when compared to the Black Sea Region and Mediterranean Sea Region where the mountains are parallel to the adjacent Sea.

The total installed electric generation capacity of Turkey is 87138.7 MW [2]. 6620.6 MW is gathered from wind power plants, which is equivalent to %7.6 of total installed electric generation capacity. According to Wind Energy Potential Atlas, the wind potential of Turkey is calculated as 48000 MW [3].

Wind energy potential has been studied for decades in many regions [4-8]. Vogiatriz et al. studied the wind potential in northern Greece, very close to the coastline. Shabbaneh et al. studied wind energy potential at Palestine both coastlines and onshore sites [7]. Adeleja et.al. have analysed the wind potential on the Michigan Lake to indicate the applicability of wind power plants to offshore [9]. In addition, Becerra et al. performed the wind potential assessment on the residential areas [10].

The purpose of the study is to determine the wind potential at the cost of Eceabat open to Aegean Sea located in Canakkale using the software of Windsim. Hourly measured wind data is provided by Turkish State Meteorological Service [11] to carry out this study.

2. MATERIAL METHOD

Windsim is a computational fluid dynamics (CFD) premised wind energy software that solves Reynolds averaged Navier-Stokes (RANS). Windsim can achieve it by combining wind data with the roughness of the terrain. At complex terrain, Windsim is more efficient than other commercial softwares [12-13]. Windsim allows to use different roughness databases ranging from 100 meters resolution to 500 meter resolution. CORINE (Coordination
of Information on the Environment) Land Cover Europe 2006 (100 m) roughness database is used for this study to get much detailed resolution of the specified terrain. For the elevation, among many of databases, ASTER (Advanced Spaceborne Thermal Emission and Reflection Radiometer) GDEM global worldwide elevation data has been used.

![Elevation and roughness of the Eceabat region respectively](image1)

Figure 1. Eceabat Region

Hourly measured wind speed and direction data were gathered from Turkish Meteorological Service at 10 meters for the Eceabat Region shown on Figure 1. The height of the wind turbine hub is at 80 m. Wind power law is normally used to find the wind speed at the given hub elevation. However, conventional wind power law causes some error when compared to real wind speed data. In order to get rid of those errors, Windsim use RANS equations to transfer wind speed at 10 meters to the location closer to windfarm with 80 meters height because wind measurement mast is not on the wind turbine area. Then, Four Vestas V90 commercial wind turbines are located on the Aegean Sea coast side of Eceabat shown on Figure 3 due to easy access to the wind turbine building area by sea.

![Elevation and roughness of the Eceabat region respectively](image2)

Figure 2. Elevation and roughness of the Eceabat region respectively
3. RESULTS AND DISCUSSION

As shown on Figure 4, Weibull distribution and the frequency of the wind speed data for the specified region are gathered for the 10 meters elevation. The transferred climatology data for the 80 meters elevation is calculated by solving RANS equations. According to the Weibull distribution, Weibull parameters, shape factor (k) and scale factor (c), are calculated as 2.07 and 5.97 respectively for the 10 meters elevation. At the 80 meters height, the Weibull parameters are computed as 2.00 and 11.08 respectively. By using four Vestas V90 wind turbines, capacity factor is calculated as %63.0. Capacity factor has to be above %35 for economically efficient investment [14]. Annual energy production (AEP) for installed wind power plant is 44.1 GWh/y. Installed wind turbines generate electricity 5528.4 hours yearly.

Average wind speed of the Eceabat region at 10 meters is 5.21 m/s. Windsim solves Navier-Stokes equations to find the wind speed for the specified region at 80 meters elevation. Figure 5 illustrates that the average wind speed at the 80 meters is 9.83 m/s.
As shown on Figure 6, the Eceabat region is divided into 1190280 cells totally. It is partitioned 182x218 cells horizontally and 30 cells vertically for the converged CFD analysis. Table 1 indicates that the CFD result of wind potential study is independent of the grid size. The relative error at the last two-cell size for capacity factor and AEP is 0.009 and 0.006 respectively.
Table 1. Results of the analysis

<table>
<thead>
<tr>
<th>Total Number of Cells</th>
<th>Capacity Factor (%)</th>
<th>AEP (GWh/y)</th>
</tr>
</thead>
<tbody>
<tr>
<td>86400</td>
<td>60.5</td>
<td>42.5</td>
</tr>
<tr>
<td>198380</td>
<td>60.3</td>
<td>42.4</td>
</tr>
<tr>
<td>350900</td>
<td>61.3</td>
<td>43.1</td>
</tr>
<tr>
<td>793520</td>
<td>62.4</td>
<td>43.8</td>
</tr>
<tr>
<td>1190280</td>
<td>63.0</td>
<td>44.1</td>
</tr>
</tbody>
</table>

4. CONCLUSION

Since Turkey is a developing country, the energy demand augments as the time passes by. In order to meet the increasing energy demand and diminish foreign dependence, new power plant investments are implemented onto renewable energy. These renewable energy investments will diminish the release of greenhouse gases to the atmosphere. Current statistics indicates that wind power plant just cover the %7.6 of total installed power generation capacity, equivalent to 6620.6 MW. That much power is just %13.8 of the Turkey’s wind potential according to the Wind Energy Potential Atlas.

In this study, wake effects of the wind turbines are neglected. Even neglecting it, by establishing four Vestas V90 on the Aegean Sea coast side of the Eceabat region, annual energy production and the capacity factor found as 44.1 GWh/y and %63.0 respectively.

ACKNOWLEDGEMENT

The authors thanks to the support of measured wind data given by Turkish State Meteorological Service.

REFERENCES


AUXETIC STRUCTURES - A REVIEW

Bahattin CEMİLOĞLU, Madiyar ALKHOZHAYEV, Suat KUYUMCU, Dr. Mostafa RANJBAR

Department of Mechanical Engineering, Ankara Yıldırım Beyazıt University, Ankara, TURKEY
Corresponding Author Email: bahattin.cemiloglu@gmail.com
*All authors have contributed in a same level to this work.

ABSTRACT

Some materials exhibit “auxetic” properties; that is, they have a negative Poisson’s ratio and for maintaining it, a specific microscopic structure in the auxetic materials is important. Thus, auxetic and non-auxetic materials exhibit different deformation mechanisms. In this review, we are basically spotting lights on auxetic materials, their structures, modeling, properties and applications in several fields. Showing you how they differ on the basis of structure, scale and deformation mechanism. There are many types of auxetic materials such as auxetic cellular solids, microscope auxetic polymers, molecular auxetic materials and auxetic composites. As we will go through each of them briefly determining and deciding the various aspects of auxetic nature and their products. The work also describes the finite element analysis of anti-tetrachiral structures using “ANSYS” software with performing tensile test to the ABS plastic produced by anti-tetrachiral structures that is produced by 3D FDM printer. As the main purpose of this thesis is to compare the deformation and stress values that are obtained from both of analytical and experimental data.

Keywords: Poisson’s ratio, Auxetic cellular solids, Microscope auxetic polymers, Molecular auxetic materials, Auxetic composites, Auxetics

1. Introduction about Auxetics

Before we start, from our daily life experience, when a material is stretched, the material does not only become longer in the direction of stretch, but also becomes thinner in cross-section. The behavior of the material in this case is under a deformation that is governed by one of the fundamental mechanical properties of material that we have been talking about “Poisson’s ratio”. But in case of auxetic material, counterintuitive behavior is occurred as it undergoes lateral expansion when stretched longitudinally and becomes thinner when compressed! So, to ease up, “Auxetics” are structures or materials that have a negative Poisson's ratio. When stretched, they become thicker and perpendicular to the applied force. As this occurs due to their particular internal structure and the way is this how a sample deforms when uniaxially loaded. [1,2]

1.2. History of Auxetics

Auxetics actually have a long old history in our life as their history dates back to first decades of the great French mathematician, physician and astronomer Simeon Denis Poisson (1781-1840). As he was the one who wrote the formula which has defined that negative ratio of transverse to axial strain. That was later named Poisson’s Ratio of 1800 and based on the mathematical theory of elasticity, Saint Venant had also his role in Poisson Ratio, as he was the first one to suggest that it could be negative or greater than 0.5. But the thermodynamic restrictions imposed on the constitutive law of elastic solids gave limits of the Poisson’s Ration as -1 < 0 < 0.5 for isotropic solids.

Also, in 1944 some naturally occurring materials have been discovered to have auxetic effects as negative Poisson’s Ratio materials were already conceptually known during that period. Such as iron pyrites, pyrolytic graphite, rock with micro cracks, arsenic, cadmium, cancellous bone “Also known as spongy or trabecular bone”, cow teat skin and cat skin.

However, auxetic materials had not drawn much attention to people that period until 1987 when Science - Rodene Lakes (1987) article was published by the University of Iowa, Department of Biomedical Engineering. Which has revealed to people how isotropic auxetic foams could be easily manufactured from conventional open-cell foam. As since then, extensive works have been done to gain deep insight into what makes materials auxetic and how these materials behave if compared with conventional non auxetic normal materials. Nevertheless, knowledge about auxetics was really shallow and still at its beginnings as they were actually called materials with negative Poisson’s Ratio. As Auxetics name was finally revealed and given in 1991 by the British researcher Ken Evans. While the term “Auxetic” is actually derived from the Greek word “Auxetikos” which means “that which tends to increase and has its root in the word
auxesis, meaning “increase” (noun). As for auxetic structures, they are also called Dilatational materials cause of their capacity and ability to exhibit substantial volumetric changes when loaded.

Last decades, many reviews have been written concerning auxetic materials such as Q. Liu (2006), Alderson and Alderson (2006-2007) and Miretal (2014)-edited by Teik-Cheng Lim- in 2015. As nowadays, auxetic structures are well-known by researchers and scientists. But however, the knowledge about them is still not deep enough and it still needs further investigations [2-8].

2. Poisson’s Ratio

Poisson's ratio ($\nu$), which is one of the important mechanical properties of materials, denoted by the Greek letter 'nu', and named after Siméon Poisson, is the negative of the ratio of (signed) transverse strain to (signed) axial strain. To ease up, it is the ratio of the lateral contractile strain to the longitudinal tensile strain for a material undergoing tension in the longitudinal direction. And since auxetic materials become thinner when stretched, they have negative Poisson’s ratio it means [1,20].

3. Applications of Auxetic Structures

Auxetics can be single molecules, crystals, or a particular structure of macroscopic matter. Such materials and structures are expected to have mechanical and thermal properties in comparison with solid materials such as high energy absorption, fracture resistance, low density, high acoustic isolation and damping, better thermal management (for use in heat exchangers, flame arresters, heat shields), high energy absorption capabilities (for crash absorbers), durability at dynamic loadings and fatigue, filters etc.

Auxetics may be useful in applications such as body armor, packaging material, knee and elbow pads, robust shock absorbing material, and sponge mops [6,8-12].

4. Modeling of Auxetic structures

The plastic failure of the 3D auxetic structure weave was investigated under one-axis and transverse loads to support an ongoing research study on miniaturized tensile-based sandwich cores. The beams are composed of struts whose curvatures gradually decrease from zero. The plastic tensile strength of an auxiliary tensile core under uniaxial and shear loads was theoretically determined by two parameters, namely the packaging parameter and the relative density. The results were obtained by using the numerical data obtained from the finite element analysis by ABAQUS / standard and by the lateral contractions and stresses of the negative Poisson ratio.

Finite element simulations are used in another study, where the results will provide a high sensitivity of the mechanical properties for certain ranges of auxiliary geometric cell parameters. Mechanical property values are combined with finite element simulations of a straight-stressed axis-assisted honeycomb models [13].

Finite element method models have been developed to evaluate the in-plane properties of reentrant cell combs. The models consist of 2508 two-node beam elements with a total of 2131 nodes. Displacements were applied to the nodes and the reaction forces were calculated on these nodes and divided by the starting area to determine average normal stresses. For correct straight stress loading condition, rotations in the plane of the displacement nodes applied are limited.

Two-dimensional beam elements are used to model a rectangular cross-section with a unit depth, and a three-dimensional beam is used to model a circular cross-section from C-C andâ€‰C connections. Boundary constraints have been used to provide movement along the free edges and to provide transverse shrinkage or expansion under load bearing boundary conditions along the axes.

The goal of finite element modeling is to replace complex molecular structures with simpler subunits to simplify calculations, and for this purpose different finite element models were used to identify the most sensitive subunit with the closest results from the molecular model. Two-dimensional representations of the different finite element models for the structure of the structure are for the reflex structure. A mathematical technique known as topology optimization has been used to design structures that can meet specific requirements. A two-dimensional metallic layer with low porosity was designed and the Poisson's ratio has been proven to be manipulated by the change in the aspect ratio of the voids. The Poisson ratio was found to decrease with increasing aspect ratios. Even setting one of the parameters in the structure causes large negative Poisson Ratio values [14-17].
Conclusion

The conclusion of this study is spotting the lights on the Auxetic materials and their structures, focusing on the importance of them, their applications, benefits and their uses in our life. Clarifying how much in need we are of such unique materials in several fields, showing where and how they can be affectively used and what differs them of any other normal familiar material. Even though we are actually in shortage of them in our world. Nevertheless, due to the shortage of auxetics in nature and the difficulty in attributing them a specific application, there were too much efforts to synthesize them. As this was possible by working out the material internal structure, considering the way that it deforms when subjected to a load.

References

INVESTIGATION OF LEADING EDGE TUBERCLES ON NACA 0015 AIRFOIL

Mehmet Seyhan\textsuperscript{1*}, Mustafa Sarıoğlu\textsuperscript{1}, and Yahya Erkan Akansu\textsuperscript{2}

\textsuperscript{1}Karedeniz Technical University, Mechanical Engineering Department, Trabzon, TURKEY
\textsuperscript{2}Niğde Ömer Halisdemir University, Mechanical Engineering Department, Niğde, TURKEY
*Corresponding Author: Mehmet SEYHAN, mehmetseyan@ktu.edu.tr

ABSTRACT

An experimental study is performed to investigate the effects of leading edge tubercles on the NACA 0015 airfoil at an open type wind tunnel. The objective of this study is to carried out lift and drag force measurements on the flow around NACA 0015 airfoil having amplitude modulation (tubercles) at the leading edge for the Reynolds number of 1.2x10\textsuperscript{5}. This airfoil having a chord of 150 mm and a spanlength of 450 mm is manufactured from PLA material with the help of the 3D printer. Force measurements are carried out with the help of a load cell at different angles of attack between 0\textdegree- 16\textdegree. A little increase in drag, and a little reduction in lift has been found for this airfoil having amplitude modulation as compared to baseline foil. The airfoil with the amplitude modulation showed better performance as compared to the baseline airfoil at the poststall region.

Keywords: NACA 0015, Tubercle, Amplitude modulation, Leading edge, Lift coefficient.

1. INTRODUCTION

From past to present, flow control around bodies such as bluff and aerodynamic bodies is needed to suppress vibration, improve drag and lift performance, delay flow separation and reduce noise. Therefore, flow control is extensively studied by researchers. Flow control methods can be classified as active and passive. Passive flow control (PFC) methods are more favorable than active flow control (AFC) methods for unmanned aerial vehicles (UAVs). Therefore, passive flow control methods are important in improving maneuverability and aerodynamic performance of the UAVs.

Bio-inspired design for the flow control play a vital important role in the solution of aerodynamic deficiency. Hence, researchers see nature as a temple for device or new technology development(Fish et al., 2011). Tubercles are bio-inspired device that inspired from the humpback whale (Megaptera novaeangliae) having enormous size and extreme maneuverability performance (Lohry et al., 2012). The humpback whale flipper has tubercles at the leading edge of the flipper. For the first time, it has been suggested by Fish and Battle (1995) that the flipper can be used in the aerodynamic research. In the following years, Miklosovic et al. (2004) examined the aerodynamic properties of NACA 0020 airfoil having sinusoidal shape (known as tubercles) at the leading edge in the wind tunnel. As a result, their results showed that while wavy airfoil model decreases the lift and drag force as compared to the baseline airfoil, it significantly delayed the stall angle.

Johari (2015) investigated experimentally the cavitation characteristics of NACA 63\textsubscript{1}-021 airfoil having sinusoidal leading edge at a water tunnel having 0.3mx0.3m test section. They performed a flow visualization experiment at Re = 7.2x10\textsuperscript{5}. Their results implied that the cavitation at the baseline airfoil always is seen at an attack angle bigger than that at the airfoil having sinusoidal leading edge. Johari et al. (2007) examined the aerodynamic characteristics such as drag coefficient, lift coefficient and pitching moment coefficient for NACA 63\textsubscript{1}-021 airfoil having sinusoidal leading edge at a water tunnel. Flow visualization experiment using tufts methods and force measurement experiment using load cell was carried out for the Reynolds number of 1.83x10\textsuperscript{5} at the angle of attack between -6\textdegree and 30\textdegree. While modified airfoil caused a reduction in lift coefficient, it also showed higher lift coefficient at poststall region. In the study of Hansen et al. (2011), they showed the effect of leading edge tubercles for NACA 0021 and NACA 63\textsubscript{1}-021 airfoil with the help of force measurement and hydrogen bubble flow visualization. These airfoils having tubercles indicated that when it compared with baseline airfoils (not having tubercles), the baseline airfoils had no effect over the lift coefficient but it was effective for lift coefficient at poststall regime.
The objective of this study is to carry out lift and drag force measurements on the flow around NACA 0015 airfoil having amplitude modulation (tubercles) at the leading edge for the Reynolds number of $1.2 \times 10^5$. Force measurements are measured by using six axis load cell at the attack angles from $0^\circ$ to $16^\circ$.

2. EXPERIMENTAL SETUP

Experiments are carried out with an open type wind tunnel having 57 cm square test section. This tunnel has turbulent intensity less than 1%. NACA 0015 airfoil has a mean chord ($c$) of 150 mm and a span length of 450 mm. This airfoil is produced by using a 3D printer. In the Reynolds number calculation, the mean chord is used due to wavy shaped leading edge. Reynolds number is $1.2 \times 10^5$. Figure 1 indicated the modified and baseline NACA 0015 airfoil. Experimental setup consists of a NACA 0015 airfoil, and a plate, a connection rod, a load cell and a rotary unite. End plates are used in order to eliminate the end effects. Force measurement such as lift and drag were performed using a six-component load cell. Force data were collected with the help of NI PCIe-6323 DAQ card. Load cell was fixed by the rotary unit.

![Figure 1](image1.jpg)

**Figure 1.** (a) NACA 0015 airfoil profile having amplitude modulation at the leading edge, (b) baseline NACA 0015 airfoil

As shown in Figure 1 (a), the geometry of amplitude modulation of NACA 0015 airfoil at leading edge is defined as Equation (1). Here, $a_1$ is the first amplitude of the wave, $a_2$ is the second amplitude of the wave, $x$ is the spanwise location, $\lambda_1$ is the first wavelength of the wave and $\lambda_2$ is the second wavelength of the wave. In this study, these amplitude and wavelength values are $a_1 = 0.05c$, $a_2 = 0.33c$, $\lambda_1 = 0.75c$ and $\lambda_2 = 0.253c$.

$$F(x) = (a_1 \cos((2\pi x)/\lambda_1)) + (a_2 \cos((2\pi x)/\lambda_2))$$

(1)

3. RESULT

Effects of amplitude modulation of NACA 0015 at the leading edge over the aerodynamic characteristics such as drag and lift are investigated for Re = $1.2 \times 10^5$. Figure 2 indicates the variation of the drag coefficient for wavy and baseline airfoil at the attack angle between $\alpha = 0^\circ$ and $\alpha = 16^\circ$. Wavy and baseline airfoil lift coefficient shows reasonably similar variation with airfoil theory represented by $2\pi \alpha$ but there is a difference between wavy and baseline airfoil up to the angle of attack of $7^\circ$. In the range of this angle of attack, lift augmentation decrease with increasing attack angle. These differences could be attributed to the amplitude modulation at the leading edge of the airfoil. For baseline and wavy airfoil, stall occurs at $\alpha = 12^\circ$ and $10^\circ$, respectively. Although $C_L$ for baseline suddenly decreases at $13^\circ \leq \alpha \leq 16^\circ$, there is a little decrease in lift coefficient for wavy airfoil at the poststall region. It can be deduced from here that wavy airfoil significantly change the traditional stall characteristic known as a sudden decrease in lift coefficient. Drag coefficient ($C_D$) variation as a function of attack angle is given in Figure 3. Wavy airfoil shows higher drag coefficient than baseline airfoil for all angles of attack. As shown in Figure 2, there is a significant increase in $C_D$ for wavy airfoil after the stall angle whereas the wavy airfoil indicates the drag penalty for all angle of attack.
4. CONCLUSION

The effects of NACA 0015 airfoil having wavy shape at the leading edge is experimentally examined. Forces acting on the wavy and baseline airfoil are measured at $Re = 1.2 \times 10^5$ in the wind tunnel. Amplitude modulation at leading edge of the airfoil are $a_1 = 0.05c$, $a_2 = 0.33c$, $\lambda_1 = 0.75c$ and $\lambda_2 = 0.253c$. Even if leading edge modification lead to lift reduction, it does not indicate similar stall characteristic with baseline airfoil. In the poststall regime, wavy airfoil has higher lift coefficient than baseline airfoil. This wavy airfoil model presented the drag penalty as compared to baseline airfoil.

ACKNOWLEDGEMENTS

The authors would like to acknowledge the financial support of this work by the Scientific and Technological Research Council of Turkey (TUBITAK) under the Contract Number of 118M592.
REFERENCES


A REVIEW OF FLOW ACOUSTIC EFFECTS ON A COMMERCIAL AUTOMOTIVE EXHAUST SYSTEM

Barhm Mohamad

Faculty of Mechanical Engineering and Informatics, University of Miskolc, 3515 Miskolc, Hungary
pywand@gmail.com

ABSTRACT

Acoustic simulation methods are being increasingly used for practical exhaust system design of automotive. In many practical applications, the sound source emits, partly, a low frequency sound spectrum comprised of superposed discrete tones and partly, a higher frequency broadband spectrum. The turbulent vortices that develop in the boundary layer between the duct wall and the flowing medium are said to generate a self-excited noise, that noise is broadband character. The self-excitation is enhanced when the flow is disturbed by irregularities in the duct wall. Unsteady compressible fluid flow through a duct is often encountered in many engineering applications and has been investigated by many researchers. When a pressure wave generated inside a duct is discharged from an open end of the duct, an impulsive wave that is usually characterized by high sound pressure level of short duration forms at the vicinity of the exit of the duct. Acoustic simulations solve the equations for motion, mass, momentum, and energy and can be divided into two methods, linear and non-linear. Through those literature review, we can analyze the methods and the latest development done on exhaust systems with regard to acoustic performance. The basic theory behind both approaches is explained as well as a source characterization technique that can be used to link the two methods. Some acoustic software tool has been applied to a variety of exhaust systems.

Keywords: Computational fluid dynamics (CFD), Aeroacoustics, Exhaust system, Flow effect, Transmission loss, Muffler modeling

1. INTRODUCTION

Noise is therefore studied, regulated and monitored by many countries, authorities, and establishments due to the negative effects. Noise from the transportation sector, and more specifically road vehicles with internal combustion engines is something people interact within a day-to-day basis making it an important area for noise control. Manufacturers of all kinds of road vehicles strive to mitigate as much noise as possible to produce silent vehicles both due to legislation and competition. Knowledge of the acoustic source characteristics of internal combustion engines (IC-engines) is of great importance when designing the exhaust duct system and its components to withstand the resulting dynamic loads and to reduce the exhaust noise emission. The goal of the present review is to show numerically and experimentally investigate the variety speed IC-engine acoustic source characteristics, not only in the plane wave range but also in the high frequency range define the wave equation one must first look at one - dimensional the linear conservation equation of continuity which relates
density and particle velocity up in the medium. The decomposed definition of density have been inserted and higher-order terms are neglected [1].

2. MATHEMATICAL MODELS

The sound usually generated because of the coupling between the turbulent average flow field and the acoustic field is said to be self-excited. The solution to the wave equation; the general solution for free, plane and one-dimensional wave propagation:

\[ P(x, t) = f(t-x/c) + g(t+x/c) \]  

(1)

Where f and g are arbitrary functions. f(t-x/c) implies wave propagation in the positive direction along the x-axis, with the speed c and P(x, t) represent sound pressure. We can write the linear conservation equation of continuity:

\[ \frac{\partial \rho}{\partial t} + \rho \frac{\partial u_p}{\partial x} = 0 \]  

(2)

Where:

\( \rho \): Acoustic density disturbance kg/m³
\( \rho_0 \): Density in undisturbed medium kg/m³

The linear inviscid conservation equations of momentum are also needed. Assuming that viscous effects can be neglected, the equation system relates velocity of the sound wave with acoustic pressure. Here, higher-order terms are also neglected.

Linear inviscid of motion:

\[ \rho_0 \frac{\partial u_p}{\partial t} + \frac{\partial p}{\partial x} = 0 \]  

(3)

Equation (1) and (2) include five unknown variables to solve but only four equations, similarly, as in fluid dynamics, the equation of state must be used to complete the equation system. Thermodynamic equation of state:

\[ (p_0 + p) = (\rho_0 + \rho)RT/M \]  

(4)

Where:

\( p_0 \): Pressure in undisturbed medium Pa
\( p_{ac} \): Acoustic pressure Pa
\( R \): Ideal gas constant \( (R = 8.315) \) J/(mol. K)

The gas law for adiabatic changes of state:

\[ \frac{(p_0 + p)}{\rho_0} = \left( \frac{\rho_0 + \rho}{\rho_0} \right)^\gamma \]  

(5)

The acoustic wave equation can then be defined by subtracting the time derivative of continuity (2) from the spatial derivative of momentum (3) and eliminating \( \rho \) by inserting the equation of state (4). The governing acoustic wave equation is defined as:

Source-free linearized acoustic wave equation:

\[ \frac{\partial^2 p}{\partial x^2} - \frac{1}{c^2} \frac{\partial^2 p}{\partial t^2} = 0 \]  

(6)

The speed of sound expressed as:

\[ C = \sqrt{\gamma p_0/\rho_0} \]  

(7)

\( \gamma \): Turbulence intermittency defining the propagation speed of an acoustic wave in the medium. The temperature dependence of the speed of sound:

\[ C = C_0 \sqrt{T/273} \]  

(8)

Where \( C_0 \) is the speed of sound at 0 °C.

There is important part in mufflers called Helmholtz Resonance chamber formed by a part of the inlet pipe and chamber is to reduce the specific frequency noise. It acting as an acoustic filter is designed to attenuate low frequency noise from engine [3]. The effects of mean flow on the acoustic properties of Helmholtz resonator and frequency domain-based Computer Aided Engineering (CAE) can describe the acoustic properties without mean flow. But to be able to design effective resonators with consideration for flow effects, a computational technique able to capture the fluid physics and
Acoustic behavior in the time-domain accurately is necessary. A simplified way of describing the interaction between fluid flow and acoustic wave propagation is applying the linearized Navier-Stokes equations (LNSE) [4].

\[
fr = \frac{c}{2\pi} \sqrt{\frac{A_n}{V_c(\Delta + \delta_n)}}
\]  

(9)

Where:
fr: Cut-off frequency
C: The speed of sound
An: The neck cross-sectional area
Vc: The volume of the resonator house
L\(n\): The length of the neck
\(\delta_n\): The end correction

2.1 Transmission loss

There are several acoustic quantities to define the noise mitigating performance of a silencer in a duct system. The most common quantities are noise reduction (NR), insertion loss (IL) and transmission loss (TL). The NR is defined as the difference in sound pressure level (SPL) before and after the muffler. The most accepted approach today is the approach developed by Munjal and Doige who proposed a two-source method for measuring the four-pole parameters of an acoustic element or combination of elements. The method can also be used in the presence of a mean flow.

2.1. Decomposition method

This method used for calculation transmission loss. TL with decomposition method can be expressed as:

\[
TL = 10\log_{10} \frac{W_i}{W_t}
\]

(10)

Where \(W_i\), \(W_t\) denote incident and transmitted sound power level of the acoustic wave present in the exhaust-duct system [4]. The equation can be written in other form to express the transmission isolation of an expansion chamber with a circular cross section and eccentrically placed inlet and outlet point:

\[
D_{TL} = 10. \log(1 + \frac{S_1^2}{S_2^2} - \frac{S_2^2}{S_1^2})^2 \sin^2(kL)
\]

(11)

Where:
\(S_1\): Cross section area of incoming of channel
\(S_2\): Cross section area of outgoing of channel
K: Wave number
L: The length of chamber

2.2. Two-Source Method

The two-source method is based on the transfer matrix approach that represents the acoustic behaviour of the muffler [5].

![Acoustical element](image)

Figure 1. The Four-Pole

The transfer matrix is

\[
\begin{bmatrix} P_1 \\ V_1 \end{bmatrix} = \begin{bmatrix} A & B \\ C & D \end{bmatrix} \begin{bmatrix} P_2 \\ V_2 \end{bmatrix}
\]

Where \(P_1\) and \(P_2\) are the sound pressure amplitudes at the inlet and outlet, respectively; \(V_1\) and \(V_2\) are the particle velocity amplitudes at the inlet and outlet, respectively; and \(A\), \(B\), \(C\) and \(D\) are the four-pole parameters of the system [5].

2.3. Two-Load Method

The transmission loss measurement setup schematically as shows in figure 2. A speaker is placed at the end of the impedance tube. Two microphones are mounted upstream and the other two microphones are mounted downstream. Two different termination loads are applied, and four transfer functions are measured for each load [6].
Tao and Seybert, studied and compared the results from two source method and two-load method calculation of transmission loss in mufflers which do not require an anechoic termination.

3. SOFTWARE ANALYSIS

3.1 Software Description and Calculation:
This chapter aims to present the numerical methods used in the further simulations to describe the mathematical models presented in literature. Only numerical methods related to CFD and how the methods are implemented in Ansys - Fluent are of interest. This means that numerical methods related to the acoustic. R. Glav et al. study acoustic using simulation software based on the linear and nonlinear approach and the solutions to the basic fundamental equations for mass, momentum, and energy. The results was implied that the linear acoustic simulation can then be used to predict radiated sound and take advantage of the faster calculation times. Increasing use of after treatment devices in exhaust systems requires specific models of such components. This method is also able to utilize optimization techniques which are particularly well suited to the fast calculation speed of the linear acoustics. Hynninen et al. Investigate numerically and experimentally at the medium speed of internal combustion engine acoustic source characteristics and at high frequency range. One-dimensional process simulation code was used for this study. The design of exhaust system was done according to engine standard parameter.

Munjal et al. discuss the transverse plane wave analysis of short elliptical end chambers, acoustical source characterization of the exhaust systems of reciprocating internal combustion engines, analysis of multiply-connected element mufflers, breakout noise of non-circular muffler shells, and analysis of porous inside the muffler.

Recently, Elnady et al. have presented a two-port method for flow and pressure drop calculation as well as acoustical analysis of complex perforated-element automotive mufflers. The study proposed a new segmentation approach based on two-port analysis techniques in order to model perforated pipes using general two-port codes, which are widely available.

3.2 Flow distribution
When flow is introduced through the muffler, its transmission properties are affected in three ways. The first is through the convective effects, which affect the propagation inside straight pipes. This effect was accounted for in the formulation of the transfer matrices for different pipe elements. The second is the introduction of extra losses at the area expansion, which takes place at the end of the inlet pipe. The third and most important effect of flow is introduced by the change of the perforate impedance. The flow can be either grazing to the perforate, through the perforate, or both. The bigger effect comes from the flow through the perforate, which increases the resistance considerably.

A. Mimani et al. considered the problem of wave propagation in short elliptical chambers having ports located along the major axis of the elliptical section. The wave propagation was considered along with the transverse direction (along with the major axis), wherein. Matrizant method was used to obtain transfer matrices relating the upstream and downstream variables. It was pointed out that such short chamber muffler characterized by dominant transverse plane wave propagation is acoustically analogous to an extended inlet and outlet chamber. In fact, a short chamber shows a striking resemblance with a side inlet and side outlet chamber (long in the axial direction). A prerequisite for this investigation is to have realistic values of the pressure-time history. These were computed using the commercial software AVL-BOOST for different acoustical loads. This finite-volume CFD model is used in conjunction with the two-load method to evaluate the source characteristics at a point in the exhaust pipe just downstream of the exhaust manifold.

Y. Gerges et al. study muffler performance experimentally and a test rig was designed in order to measure the Transmission Loss of a set of muffler configurations in the stationary medium. The experimental set up is based in a
combination of the decomposition method. It shows the geometry and comparison between the experimental and Transfer Matrix Method (TMM) modeling numerical results of TL for a high porosity perforated concentric tube inside an elliptic expansion chamber. A reasonable similarity in the transmission loss curves can be verified.

Y. Sathyanarayana et al. present a new hybrid approach or prediction of noise radiation from engine exhaust systems by making use of the time domain modeling of the cylinder and cavity pipe junction, and the linear frequency domain analysis of the muffler.

Denia et al. had analyzed an elliptical expansion-chamber muffler as well as an end chamber muffler using a 3-D analytical method based on the modal superposition technique and the point-source method. The shows 1-D model based on the transverse plane-wave method is used to predict the transmission loss characteristics of short axial-length mufflers, taking the same dimensions as considered.

B. Ouédraogo, et al. presented in their technical paper, the performance of circular duct with non-locally lining by numerically and experimentally. The liner concept is based on perforated screens backed by air cavities. Dimensions of the cavity are chosen to be bigger than the wavelength so acoustic waves within the liner can propagate parallel to the duct surface. The aim of this research was to identify the best multi-cavity muffler configuration for reduction of exhaust noise from the engine. The result shows that the cavity configuration achieving the maximum overall acoustic Transmission Loss. The study also illustrates how the acoustic performances are dependent on the nature of the incident field.

Sagar and Munjel, analyzed the limitation of the net insertion loss of the muffler, by using three-pass double reversal muffler in automotive exhaust systems. This muffler is characterized by a fairly wideband transmission loss [TL] curve as well as relatively low back pressure.

S. Talegaonkar et al. prove experimentally, the ratio of the reduction in the back pressure at 11000 engine rpm is around 40% with open valve, as compared closed valve condition. The increase in sound transmission loss, with closed valve, is around 15–20 dB higher when compared to open valve condition.

M. Dixit et al. studied the effects of back pressure on muffler effectiveness and the simulation is carried out using GT-POWER® tool. The discretization of muffler and resonator shell and pipes for element generation had played an important role in the proper prediction of back pressure and thereby reducing valuable design cycle time and cost. S.J. Liu et al. presented in their technical paper optimization of intake exhaust system of a single-cylinder water-cooled swirl chamber diesel engine, by using computational fluid dynamics [CFD] and steady flow test method. The configurations and performances of intake and exhaust port, air filter and muffler were optimized for reducing flow resistance, increasing charge amount and lowering residual exhaust gas, leading to the improvement of engine performances and emissions. The result of this research show great achievement in engine exhaust gas properties. Zhenlin and Yiliang presented in their research paper the acoustic characteristics of duct muffling systems. The program was based on the plane wave theory and uses the Visual Basic 6.0 to build and modify the duct muffling systems quickly with proper the geometrical and physical parameters, to examine the effects of design changes on the acoustic attenuation characteristics and finally to get an acceptable solution.

X. hua et al. was contributed to determine acoustic performance of muffler and can be reviewed for design and development of muffler. In his research paper two-load method was conducted for measuring muffler transmission loss also algorithm for computing the transmission loss was involved. It is demonstrated that the effect of adding conical adapters is significant at low frequencies especially if the adapter is short in length. It was found that measurements are improved by selecting a downstream microphone as a reference instead of an upstream microphone with good agreement for both methods.

Jin Woo Lee use an acoustical topology optimization for a partition volume minimization problem achieving high value transmission loss through muffler.

Singh et al. Used Computational Fluid Dynamics (CFD) methods for simulation of acoustic pulse in muffler and develop a full compressible Navier-Stokes solution algorithm for acoustic propagation problems. A new hybrid low Mach number pressure based compressible solver was developed to simulate propagation of pulses of random shape, demonstrated by application through a simple expansion muffler.

C.J. Wu et al. Compared model meshing approach result with numerical analysis in various cases of length–diameter ratio to predict transmission loss of a muffler. Takashi Yasuda et al. studied the automotive muffler experimentally and numerically. 3D CFD was used to evaluate both mean flow and acoustic performance of an expansion chamber muffler, with various modifications including baffles an extended inlet or out let pipes.

S. Allam et al. studied acoustic plane wave properties of a complex geometry, and some parameter were experimentally calculated such as reflection coefficient using the TMM on the upstream and downstream side of the test object. In this research Presented and tested a method for measuring the two-port data in the form of a scattering-matrix, describing the relationship between the traveling wave amplitudes of the pressure on either side of the test object.
Linear and passive two-port in the frequency domain, be written:

\[
X = TY
\]

Where, \(X/Y\) are the state vectors at the input/output and \(T\) is a [2×2]-matrix, which is independent of \(Y\). To determine \(T\), from measurements four unknown must be determined. The transfer-matrix form uses the acoustic pressure \(P\) and the volume velocity \(V\).

\[
\begin{bmatrix}
P_a \\
V_a
\end{bmatrix} = \begin{bmatrix}
T_{aa} & T_{ab} \\
T_{ba} & T_{bb}
\end{bmatrix}\begin{bmatrix}
P_b \\
V_b
\end{bmatrix}
\]

(13)

\[
P_a = P_a\exp(-ikL_a)+P_b\exp(ikL_a)
\]

(14)

\[
V_a = \frac{A_b}{\rho c^2}[(P_a\exp(-ikL_a)-P_b\exp(ikL_a))
\]

and

\[
P_b = P_b\exp(-ikL_b)+P_a\exp(ikL_b)
\]

(15)

\[
V_b = \frac{A_a}{\rho c^2}[(P_b\exp(-ikL_b)-P_a\exp(ikL_b))
\]

The result shows, the acoustic two port, of a single diaphragm orifice and then validated with the theoretical result has been calculated using 3D FEM software FEMLAB from other literatures.

Munjal present different approaches for measurement and evaluation of the source characteristics of an engine exhaust system have been briefly reviewed, with particular emphasis on their relative implications and limitations. These approaches combine the advantages of the frequency-domain analysis of mufflers with those of the time-domain analysis of the exhaust manifold source. Because in some method does not require prior knowledge of the source characteristics, like piston motion, exhaust valve/port opening and high blow-down pressure in the cylinder.

4. CONCLUSIONS

The main contribution of the present review, as well as the development of an expression for the non-dimensional frequency for obtaining the resonance peak in the TL graph for the short chambers with end ports, is another noteworthy contribution of this present review. This enables a muffler designer to have a quick estimate of the positions of the peak and trough in the TL spectra for short chamber mufflers and to get a qualitative understanding of the basic nature of such short chamber mufflers.

REFERENCES


A REVIEW OF FLOW ACOUSTIC EFFECTS ON A COMMERCIAL AUTOMOTIVE EXHAUST SYSTEM- METHODS & MATERIALS

Barhm Mohamad

Faculty of Mechanical Engineering and Informatics, University of Miskolc, 3515 Miskolc, Hungary
pywand@gmail.com

ABSTRACT

Literature review on flow acoustic methods and materials of an automotive muffler. A car is judged comfortable also depending on the acoustic level transmitted inside, and a thorough knowledge of acoustics of ducts and mufflers is needed for the design of efficient muffler configurations. Unstable exhaust gas at high temperature flowing from internal combustion engine manifold may cause of noise and vibrations conflicting with the high standard of acoustic comfort requested by this kind of vehicle. The basic goals are to define most important methods to identify noise occur from the motion of fluid in case of turbulent model. Materials properties like velocity, temperature, thermal conductivity and density have been technical presented in this work.

Keywords: Internal combustion engine, Exhaust gas system, Two-source method, Two-load method, Acoustic analysis, Computational Fluid Dynamic

1. INTRODUCTION

Though the prime function of muffler is to reduce engine noise, it is designed to control the back pressure of the exhaust system as well. Performance of an engine greatly depends on the effective exhaust of the combustion gases and its properties to atmosphere through muffler. The variation back pressure has the direct effect on muffler temperature distribution that regulates fuel economy, wave propagation and energy loss. Prakash Chandra Mishra et al [1] studied the effect of muffler geometry modification on exhaust parameters of an engine. Such engine is powered with blend of methanol and gasoline by different volume percentage blended with 95% petrol, 5% methanol; 90% petrol, 10% methanol and 85% petrol, 15% methanol blends respectively. Based on heat release from combustion of different type of fuels, has direct effects on exhaust gas properties. In this research four type of geometry chambered straight (CS), chambered elliptic (CE), turbo straight (TS) and turbo elliptic (TE) was used to identify acoustic level. The author also shows the relation between emission and gas cooling through a muffler. The result from CFD analysis was show the exhaust gases from inlet pipe passes through the baffle or perforated pipe inside the shell, the gases get scattered in different directions. The variation of pressure, velocity, temperature distribution, and flow were calculated. After reflection from the wall, the cancellation of sound waves occurs. Such process occurs multiple times resulted in reduction of sound with in muffler. K. Ashok Reddy [2] made a survey on optimization of automotive muffler. This review depicts about transmission loss characteristics, different methods used in the design, calculation and construction of reactive and reflective mufflers both experimentally and in practical.

Prakash Chandra Mishra at al [3] studied the effect of perforated and non-perforated muffler on outlet temperature. A computational fluid dynamic method was used to develop muffler performance. Based on the solver, the effect of back pressure, temperature, density and velocity streamline of exhaust gas on the muffler performance was studied for different blends (5%, 10%, 15%) of gasoline and methanol by volume percentage. Result show clear variations of these parameters are observed between non-perforated and perforated type turbo pipe mufflers. For different design parameter and fuels, the results of CFD analysis shows great difference of gas properties during damping process.

M.L. Munjal et al [4] presented in their research paper the effects of reverse flow in three duct, open-end perforated element muffler, by using transfer matrix method to predict noise reduction through system. The result was validated with experimental data using two-load method.

M.L. Munjal [5] studied transmission loss [TL] for a muffler system, transfer matrix method has been derived for side inlet and side outlet elements for typically small mean flow Mach numbers. The result shows clear effect of mean flow on TL of the side inlet/ outlet chambers has been studied.

S. Allam et al [6] studied acoustic plane wave properties of a complex geometry, and some parameter were experimentally calculated such as reflection coefficient using the TMM on the upstream and downstream side of the test object. In this
research presented and tested a method for measuring the two-port data in the form of a scattering-matrix, describing the relationship between the traveling wave amplitudes of the pressure on either side of the test object.

Linear and passive two-port in the frequency domain, be written:

\[ X = TY \]

Where, \( X/Y \) are the state vectors at the input/output and \( T \) is a \([2\times2]\)-matrix, which is independent of \( Y \). To determine \( T \), from measurements four unknown must be determined. The transfer-matrix form uses the acoustic pressure \( P \) and the volume velocity \( V \).

\[ X = \begin{bmatrix} P_a \\ V_a \end{bmatrix} \quad Y = \begin{bmatrix} P_b \\ V_b \end{bmatrix} \]

The transfer-matrix could be written in the following form:

\[
\begin{bmatrix} P_a \\ V_a \end{bmatrix} = \begin{bmatrix} T_{aa} & T_{ab} \\ T_{pa} & T_{bb} \end{bmatrix} \begin{bmatrix} P_b \\ V_b \end{bmatrix}
\] (2)

\[ P_a = P_a e^{-ikL_a} + P_b e^{ikL_a} \]

\[ V_a = \frac{A_a}{\rho c} \{ (P_a e^{-ikL_a} - P_b e^{ikL_a}) \} \] (3)

and

\[ P_b = P_b e^{-ikL_b} + P_a e^{ikL_b} \]

\[ V_b = \frac{A_b}{\rho c} \{ (P_b e^{-ikL_b} - P_a e^{ikL_b}) \} \] (4)

The result shows, the acoustic two port, of a single diaphragm orifice and then validated with the theoretical result has been calculated using 3D FEM software FEMLAB from other literatures. El-Sharkawy and El-Chazly [7] present a critical survey of latest technique to determine the transmission loss, reflection coefficient and attenuation of sound in regarding the basic theories used for muffler design and analysis. The exact and approximate methods for solution are reviewed as well. The Actual measurements of sound pressure levels in the exhaust pipes of large engines showed that the maximum levels are in the order of 165dB at 300 Hz and about 135 dB at 1000 Hz. The non-linear effects play an important role in the attenuation of sound. The non-linear effect scan be classified into two types: the non-linearity of the gas itself which is significant only at sound pressure levels exceeding 160 dB, and the material non-linearity (especially perforated plates) which may be significant at pressure levels above 30dB.

Sathyanarayana and Munjal [8] predicting radiated noise requires a model of the acoustic behavior of the intake/exhaust system and a model of the engine cycle source characteristics. In their research the frequency domain analysis of mufflers is done by means of the transfer matrix method. The result shows the effect of duct length and expansion chamber on muffler performance and transmission loss. Desmons and Kergomard [9] studied the noise radiated from the exhaust system of a four-cylinder engine, analytically and numerically, and validated to experiment. The calculation includes the effect of the exact shape of the volume velocity signal produced by a cylinder during opening two valves. The pertinence of the resonances of the whole system including exhaust and manifold is demonstrated. The results show the overlap duration plays a key role in the amplitude of the harmonics. Heymann [10] present a research paper consist of two part. The first part reviews, in non-mathematical terms, the analysis techniques which can be used for acoustic circuits. Slightly more mathematical appendices were used to express the acoustic impedance and acoustic velocity in fluid-filled ducts. The second part was deal more directly with practical considerations in the design of acoustic test loops for liquid piping system components, and important property of a noise source is the distribution of acoustic pressures over the cross-section, since this will determine to what degree the various modes of propagation will be excited.

In another technical paper of Heymann [11] studied the measurements and characterization of the acoustic performance of fluid system components, for ducts and acoustic filters. The first part reviewed, in non-mathematical terms, the analysis techniques which can be used for acoustic circuits and pointed out that the effect of any component cannot be predicted accurately without making a complete circuit analysis. Approaches toward acoustic characterization of noise sources were discussed. Second part deals more directly with practical considerations in the design of acoustic test loops for duct system components.

H. Bailliet et al [12] studied multi-modal acoustic propagation in circular duct with turbulent flow. An acoustic source has been especially designed to generate high level acoustic power. The basic design a laboratory facility that share common acoustic and mean flow features with real engines. Apart from classical aerodynamic study, a Laser Doppler Velocimetry (LDV) has been adapted to the purpose of measuring acoustic velocity in a turbulent mean flow either by
using techniques developed in the context of fluid mechanics measurement or by using least mean square detection in which turbulence is viewed as a noise.

Flow profiles have been performed for different locations in the duct and at different Mach numbers from 0.1 to 0.3. The axial velocity was measured across the section with a 0.1 mm step near the wall and with gradually growing steps up to 5 mm in the centre of the duct. These results show that LDV is an appropriate method for estimating acoustic velocity with a mean flow. An experiment was conducted by Fenech and Ganz [13] to measure the characteristics of flow noise in a bounded system with forced circulation. The flow noise facility is described in research paper with location of vortex shedding in the conducting rectangular ducting was describe. The experimental was done by the insertion of cylinders of various diameters and pitch perpendicular to the flow. The flow noise experiment provides many information about the nature of the acoustical noise in the test section and about the response of the cover plate to flow noise excitation. The pressure fluctuations spectrum results, it is seen that, at low flow velocities, the acoustical noise in the cavity consists mainly of blower noise peaks at the blade passage frequency with some lower broad band noise due to air turbulence. The flow dynamic head is considered the most important parameter in flow noise production. Furthermore, the Reynolds number, which is a measure of turbulence in the flow, is also of significance in relating flow noise level, flow velocity and conduit size.

Bilawchuk and Fyfe [14] used a proper examines of three different methods used in calculation of TL values; SYSNOISE is an FEM/BEM based computational acoustics program, the 4-pole transfer matrix method with the 3-point method and (traditional) laboratory method. To set up this experimental facility, a high-level acoustic source was developed to generate higher propagating modes in the presence of mean flow. A comparison of these methods based on such criteria as accuracy in terms of computation time. The result predicts the performance of a silencer system, acoustic propagation inside a cylindrical duct in presence of a turbulent mean flow. It can be seen the results for both the traditional and 3-point methods overlay each other exactly. The 4-pole FEM results match those of the other methods, and the 4-pole BEM results are also very close to the other BEM results.

F. Payri et al [15] presented in their research paper exact solution using linear acoustic theory and non-linear calculations. The pressure and mass velocity distributions along the exhaust system were used to evaluate the results from linear acoustic theory. The mass velocity distribution presents the additional interest that it may be related to the insertion loss of the system.

A full non-linear model was used to estimate the pressure and mass velocity distributions along the exhaust line. The model was validated by comparison with experiments at selected points in the exhaust system. Bies and Hansen [16] studied flow resistance information for the calculation of the acoustical properties of porous materials. Some definition and expression have been done on non-linear flow resistance. Experimental investigation has shown that, for a wide range of materials the differential pressure, and the induced normal velocity, of the gas at the surface of the material are linearly related provided that the normal velocity is small.

Yasser Elnemr [17] investigate and performed on three of the most nowadays commonly used absorbing materials in exhaust systems (PowerTex, Biosil, and HakoTherm), with perforated through flow chamber silencers, and a perforated multi chamber silencers the aim was to show the influence of adding the absorbing material on these silencers acoustic resonances. The simulation was there using FEM approaches; COMSOL MULTIPHYSICS as a finite element tool analyzing the problems in 3-D AVL BOOST-SID as a 1-D tool, and show a perfect match with the measurement results, so both of them can be used in the design phase of the commercial mufflers.

Part of simulation done by 1D AVL-Boost. The model consists of system boundary, ducts, and junctions. Linear acoustic calculation was done to the muffler to verify the performance, measure transmission loss and noise reduction. B. Mohamad et al [18] Used 1D AVL-Boost software to describe the effect of using different kind blend fuels on engine performance and exhaust properties. The result show variation of outlet temperature, and emission gas characteristics for CO, NOx, and CO2 by using different volume percentage of alcohol-gasoline blends. B. Mohamad et al [19] studied the effect of Ethanol-Gasoline blend fuel on engine power output and emissions, the literatures results show great improvement in combustion process and exhaust gas characteristics. B. Mohamad et al [20] Presented in their technical paper review of muffler used in industry, and this review depicts flow and temperature distribution along the muffler ducts. The techniques for different methods used in the design, calculation and construction of muffler both experimentally, practically and transmission loss characteristics were described. 1D calculations are much faster, and still give a good overview of the system under investigation.

Nakra et al [21] Experimental work was carried out on the design and analysing performance of the reactive type of muffler, fitted on internal combustion engines to reduce the exhaust noise, the relationships used were those derived from plane wave or acoustic filter theory. Experimentally it has been reported that the mean back pressure of the exhaust system is an important parameter so far as engine performance is concerned. The absorption material was recommended to reduce
a noise level at diesel combustion high frequency. Overall sound pressure level was 108 dB without muffler, with muffler was
97 dB, and with resonator muffler was 101 dB.
Lamancusa [22] studied the transmission loss of a double expansion chamber muffler with unequal chambers has been
derived and used to study the performance. It is also shown that significant tuning benefits result with chambers of unequal
size.
Howard and Richard [23] describe the acoustic performance of mufflers that include, Transmission Loss (TL), and Noise
Reduction (NR). Numerical predictions of the expected transmission loss of a quarter-wave tube were made using Finite
Element Analysis (FEA). One benefit of using FEA is that the transmission loss can be predicted for the three side-branch
generes considered.
Experimental testing was conducted using three configurations of side-branch geometries with an adaptive quarter-wave
tube attached to the exhaust of a large diesel engine. It was shown that the side-branch with the bell-mouth opening
provided the greatest noise reduction and hence was the least affected by the gas flow.
Singh and Rubini [24] studied experimentally and numerically the simple expansion muffler both with and without flow
are obtained to compare attenuation in forced pulsation for various mean-flow velocities. A three-dimensional Large Eddy
Simulation was carried out on the same muffler geometry to ensure good agreement for detailed analysis of the muffler.
As shown in results a mean flow of 10 m/s (Mach number of 0.03) was considered. Using Unsteady Reynolds Averaged
Navier-Stokes (URANS) simulation of a muffler conducted.
The results are presented to demonstrate inherent limitations associated with this approach and validated for compressible
Large Eddy Simulation (LES) of channel flow. The result show improvement in the prediction of vortex shedding inside
the chamber is highlighted in comparison to the URANS method. Further, the effect of forced pulsation on flow-acoustic
is observed. Munjal [25] present different approaches for measurement and evaluation of the source characteristics of an
engine exhaust system have been briefly reviewed, with particular emphasis on their relative implications and limitations.
These approaches combine the advantages of the frequency-domain analysis of mufflers with those of the time-domain
analysis of the exhaust manifold source. Because in some method does not require prior knowledge of the source
characteristics, like piston motion, exhaust valve/port opening and high blow-down pressure in the cylinder.
Guhan et al [26] present in their research work, the pressure drop, noise level by optimizing muffler volume with the help
of 3D design tool CATIA V5 and computational fluid dynamics commercial tool ANSYS CFX, based on FVM. Result
of this study, existing muffler volume has been reduced by 15 % and weight reduced by 2 %, this reduction gives cost
benefits as well as fuel economy benefits. Existing muffler has been analyzed and then compared with vehicle level real
test.

2. CONCLUSIONS
The acoustic performances of proposed muffler were studied experimentally and theoretically in the present review paper.
The simulation models for flow and regeneration in internal combustion engine. The models have been implemented into
the 1D code BOOST and into the 3D code FIRE and other CFD codes. A workflow is designed which makes it possible
to automatically determine the necessary model parameters and to use them in both 1D and 3D simulations. It was
concluded that studying on noise eliminations by computational fluid dynamic and in experimental approach becomes a
new area of study.

NOMENCLATURES

\[ a \] Inlet zone
\[ b \] Outlet zone
\[ P \] Pressure, Pa
\[ V \] Velocity, m/s
\[ L \] Length of duct, m
\[ K \] Wave number, m\(^{-1}\)
\[ A \] Cross section area, m\(^2\)
\[ \rho c \] Density coefficient, kg/m\(^3\)
\[ TL \] Transmission loss, dB

ABBREVIATIONS

CFD Computational Fluid Dynamic
FVM Volume Method Finite
BEM Boundary Element Method
LES Large Eddy Simulation
LDV Laser Doppler Velocimetry
URANS Unsteady Reynolds Averaged Navier Stokes
FEA Finite Element Analysis
REFERENCES


PRODUCTION OF CORN AND HAZELNUT OIL METHYL ESTERS USING SODIUM HYDROXIDE AND METHYL ALCOHOL

Mert Gülüm and Atilla Bilgin
Karadeniz Technical University, Mechanical Engineering Department, Trabzon, Turkey
Corresponding Author: Mert Gülüm, gulum@ktu.edu.tr

ABSTRACT

In this study, transesterification reaction parameters of corn and hazelnut oils were determined and compared. For this, the effects of main reaction parameters (NaOH amount (0.25%–1.50%), reaction temperature (40–70°C) and methyl alcohol/oil molar ratio (3:1–12:1)) on the kinematic viscosities of produced corn and hazelnut oil biodiesels were investigated. According to results, the reaction parameters were determined as: NaOH amounts of 0.90% (for corn oil biodiesel) and 1.00% (for hazelnut oil biodiesel), reaction temperature of 50°C and alcohol/oil molar ratio of 9:1. The corn and hazelnut oil biodiesels produced according to the determined reaction parameters have the lowest viscosities of 4.095 mm²/s and 4.128 mm²/s, respectively.

Keywords: Green energy, Biodiesel, Viscosity, Transesterification, Corn oil, Hazelnut oil

1. INTRODUCTION

Biodiesel is receiving serious attention globally as a potential alternative fuel for replacing diesel fuel, partially or fully [1]. It is a biodegradable, renewable and non-toxic fuel, and it contributes no net carbon dioxide or sulfur to the atmosphere [2]. On the other hand, higher viscosity and pour point temperature, and lower heating value and volatility are the disadvantages of biodiesel. Further, their oxidation stability is lower, they are hygroscopic, and may cause corrosion in various engine components. For all the above reasons, it is generally accepted that blends of diesel fuel, with up to 20% bio-diesel and vegetable oils, can be used in existing diesel engines without any modifications [3,4]. The most common way to produce biodiesel is transesterification as the biodiesel from transesterification can be used directly or as blends with diesel fuel in diesel engine [5]. In transesterification reaction, vegetable oils or animal fats, largely comprised of triacylglycerols, react with monohydric alcohols, most commonly methanol and ethanol, in the presence of a suitable catalyst to give the corresponding mono-alkyl esters. Transesterification reaction can be alkali-catalyzed, acid-catalyzed or enzyme-catalyzed [6]. In the existing literature, many researchers investigate biodiesel production via alkali-catalyzed transesterification using different feedstocks and alcohols such as: Samuel et al. produced coconut oil biodiesel through base-catalyzed transesterification using potassium hydroxide (KOH) as catalyst and ethanol. The reaction variables (KOH amount (0.5–2.25 wt.%), reaction temperature (30–80 °C), reaction duration (30–70 min) and ethanol to oil molar ratio (3–12)) were optimized. According to results, the optimum reaction variables giving a biodiesel yield of 97.20% were obtained as: KOH amount of 1.00%, ethanol/oil molar ratio of 6:1, reaction temperature of 70 °C and reaction duration of 60 min., and the fuel properties of CNNOE at the optimum conditions conformed to both ASTM D6751 and EU 14214 standards [7]. In the study performed by Sanli and Canakci, sunflower, corn, soybean, rapeseed, hazelnut, and cottonseed oil biodiesels were produced by means of transesterification reactions using different alcohols (methanol, ethanol, 2-propanol, and 1-butanol) and catalysts (potassium hydroxide, sodium hydroxide, and sulfuric acid). It was concluded that the reaction conditions of 6:1 molar ratio of methanol to oil, 1.0% KOH amount (w/w), room temperature and 1 h are adequate in order to produce biodiesel fuel within the fuel qualifications [8]. Farooq et al. investigated heterogeneous catalysts derived from waste chicken bones were employed in the transesterification reaction of waste cooking oil for biodiesel production. The heterogeneous catalyst calcined at 900 °C exhibited good catalytic activity in the transesterification of waste chicken bones, providing maximum biodiesel yield of 89.33% at 5.0 g of catalyst amount, 15:1 methanol to oil molar ratio at temperature of 65 °C in reaction duration of 4 h [9]. When considering the related literature, it can be seen that there is still the lack of studies on the investigation and comparison of transesterification reaction variables of the lowest viscosity corn and hazelnut...
oil biodiesels produced by using sodium hydroxide and methanol. Therefore, in this study, (1) the effects of sodium hydroxide amount, reaction temperature, reaction duration and methyl alcohol/oil molar ratio on viscosities of produced corn and hazelnut oil biodiesels were investigated, and (2) the reaction variables giving the lowest viscosity were determined and compared. Density and viscosity measurements were carried out according to ISO 4787 and DIN 53015 standards. Reaction duration of 60 minutes was kept constant during all reactions.

2. MATERIALS AND METHODS

Biodiesel production

Methanol, sodium hydroxide and anhydrous sodium sulfate used in the transesterification were purchased from Merck. Refined corn and hazelnut oils which are agricultural products at Black Sea region of Turkey were used to produce biodiesels. Transesterification reaction was carried out in a 1 L flat-bottomed reaction flask, equipped with a magnetic stirrer heater, thermometer and spiral reflux condenser. Before starting the reaction, NaOH was dissolved in methanol by stirring in a small flask, and this alcohol/catalyst mixture was added into the corn or hazelnut oil that was formerly warmed. Then, the final mixture was mixed with stirring by means of the magnetic stirrer heater. After the transesterification, lower phase (glycerol) was removed by a separating funnel, while upper one (biodiesel) was washed with warm distilled water. Then, it was dried over anhydrous sodium sulfate (left over night) and finally filtered using qualitative filter papers [10]

Density and viscosity measurements

The densities of produced biodiesels were determined by means of pycnometer in accordance with ISO 4787 standard. In addition, the dynamic viscosities of produced biodiesels were measured according to DIN 53015 standard. Viscosity measurements were made by using universal Haake Falling Ball Viscometer, Haake Water Bath and stopwatch. Details of the measurements were given in [10]. The kinematic viscosity was determined by dividing dynamic viscosity to density at the same temperature, as well known.

3. RESULTS AND DISCUSSION

Effect of catalyst amount

In order to investigate the effects of catalyst (NaOH) amount on kinematic viscosities of produced corn and hazelnut oil biodiesels, reaction temperature of 60°C and alcohol/oil molar ratio of 6:1 were kept constant throughout for this set of the experiments and catalyst concentration was changed as 0.25%, 0.50%, 0.75%, 0.90%, 1.00%, 1.10%, 1.25% and 1.50%, as shown in Fig. 1. According to this figure, as catalyst concentrations are increased, kinematic viscosities of produced corn and hazelnut oil biodiesels gradually decrease until the catalyst concentrations of 0.90% and 1.00%, respectively. At these points, the kinematic viscosities take minimum values of 4.219 mm²/s and 4.346 mm²/s, respectively. Then, when the catalyst concentrations are continued to increase, the viscosities gradually increase. This variation can be attributed to the yield of the transesterification reaction. It is known that the viscosity of the produced biodiesel decreases with increasing reaction yield [10]. Since there is not enough amount of the catalyst in reaction medium for low catalyst concentrations (e.g., 0.50 of sodium hydroxide), much of the triglycerides in the oil cannot be converted sufficiently to ethyl esters throughout the reaction period (60 minutes). This situation reduces the yield of the transesterification reaction and increases kinematic viscosity of produced biodiesel. If higher catalyst concentration is used, the yield of the transesterification reaction improves and thus the viscosity of the produced biodiesel decreases. However, when excess catalyst concentration is used, the yield of the transesterification reaction decreases [11] and the viscosity of the produced biodiesel increases on account of formation of fatty acid salts (soap), decrease in activity of the catalyst and difficulty in separation of glycerol. In the next stage of the study, the catalyst concentrations (0.90% and 1.00%) giving the lowest viscosity were kept constant, and the other parameters were changed.
Effect of reaction temperature

To determine the effects of reaction temperature on kinematic viscosities of produced biodiesels, the catalyst concentrations of 0.90% (for corn oil) and 1.00% (for hazelnut oil), and alcohol/oil molar ratio of 6:1 were kept constant throughout this set of the experiments and the reaction temperature was changed as 40°C, 50°C, 60°C and 70°C. Fig. 2 shows the changes of kinematic viscosities versus reaction temperature. According to this figure, when reaction temperatures are increased, the kinematic viscosities of produced biodiesels gradually decrease until the reaction temperature of 50°C. At this point, the viscosities take minimum values of 4.137 mm²/s and 4.184 mm²/s for corn and hazelnut oil biodiesels. Then, as the reaction temperatures are continued to increase, the viscosities gradually increase. Kinematic viscosity of biodiesel produced at low reaction temperatures (e.g., 40°C) is higher since transesterification reaction cannot be effectively completed. As the reaction temperature is increased, the yield of the transesterification reaction improves due to higher energy input [10], and the viscosity of produced biodiesel decreases. However, in case of increasing reaction temperature higher than the boiling point of methyl alcohol, viscosity of produced biodiesel increases due to diminishing of alcohol concentration by evaporating from the reaction medium. Moreover, the saponification and decomposition of esters in biodiesel at higher temperatures may also be contributed to increase in viscosity of produced biodiesel. According to results, the reaction temperature of 50°C giving the lowest viscosity was regarded to be optimal value.

![Fig. 2. Changes of viscosities of biodiesels with respect to reaction temperature](image-url)
Effect of alcohol/oil molar ratio

To analyze the effects of methyl alcohol/oil molar ratio on kinematic viscosities of produced biodiesels, the catalyst concentrations of 0.90% and 1.00%, and reaction temperature of 50°C were kept constant throughout this set of the experiments and the molar ratio was changed as 3:1, 6:1, 9:1 and 12:1. Fig. 3 shows the changes of kinematic viscosities of biodiesels versus molar ratio. Kinematic viscosities of produced biodiesels decrease until the values between 6:1 and 9:1 when molar ratios are increased. Considering the measurement values, the kinematic viscosities take the minimum values of 4.095 mm²/s and 4.128 mm²/s at 9:1 molar ratio for corn and hazelnut oil biodiesels. Then, as the molar ratios are continued to increase, the viscosities tend gradually to increase. According to the experimental results, the biodiesels have the maximum viscosities with 4.940 mm²/s and 5.256 mm²/s when molar ratio of 3:1 is used. If more alcohol/oil molar ratio is used (e.g., 6:1 or 9:1), because the transesterification reaction shifts toward products [10], the yield of the transesterification reaction increases and viscosity of biodiesel decreases. Molar ratio of 12:1 gives higher viscosity than 6:1 and 9:1 molar ratios because use of excess alcohol could be attributed to deactivation of the catalyst and increase in the solubility of the glycerol in the methyl ester phase. Therefore, alcohol/oil molar ratio of 9:1 giving the lowest viscosity was taken to be optimal value.

![Fig. 3. Changes of viscosities of biodiesels with respect to alcohol/oil molar ratio](image)

CONCLUSIONS

In the present study, the effects of catalyst amount, reaction temperature and methyl alcohol/oil molar ratio on kinematic viscosities of produced corn and hazelnut oil biodiesels were investigated parametrically to determine the reaction parameters giving the lowest viscosity by using NaOH as catalyst. It was determined that NaOH amounts of 0.90% and 1.00%, reaction temperature of 50°C and alcohol/oil molar ratio of 9:1 provide the lowest kinematic viscosities of 4.095 mm²/s and 4.128 mm²/s for corn and hazelnut oil biodiesels.

REFERENCES


DENSITY and VISCOSITY CHANGES of CORN OIL BIODIESEL-DIESEL FUEL BINARY BLENDS

Mert Gülüm, Atilla Bilgin
Karadeniz Technical University, Mechanical Engineering Department, Trabzon, Turkey
Corresponding Author: Mert Gülüm, gulum@ktu.edu.tr

ABSTRACT

In this study, corn oil biodiesel was produced by means of transesterification using potassium hydroxide and methanol. The produced biodiesel was mixed with commercially available diesel fuel on different volume ratios (5%, 10%, 15% and 20%). The densities and viscosities of biodiesel-diesel fuel blends were measured at different temperatures (10°C, 20°C, 30°C and 40°C) according to the related international standards. Based on the changes of the fuel properties with respect to temperature, the exponential and quadratic equations were proposed to estimate viscosities and densities of blends, respectively. According to results, the equations were found to be suitable ones in predicting the fuel properties with respect to temperature.

Keywords: Corn oil biodiesel, density, viscosity, transesterification.

1. INTRODUCTION

Biodiesel is a mixture of mono-alkyl esters of saturated and unsaturated long chain-fatty acids obtained by a trans-esterification of oils and fats from plant and animal sources [1]. Biodiesel is receiving increasing attention as an alternative fuel for diesel engines because of many benefits such as [2]: It has enhanced biodegradability, reduced toxicity, improved lubricity and higher cetane number and flash point temperature in comparison with diesel fuel [3]. In addition, this biofuel can be completely miscible with diesel fuel in any proportion. Further, biodiesel can be used in diesel engine without major modifications [3], and it significantly reduces particulate matter, hydrocarbon, carbon monoxide and life cycle net carbon dioxide emissions from combustion sources [4]. However, engine manufacturers are concerned about biodiesel’s higher viscosity, iodine number, (generally) NOX emissions and price, and lower energy content and volatility [5].

Viscosity is one of the most important fuel properties affecting fuel atomization in-cylinder. High viscosity (1) alters the injection spray characteristics, resulting in fuel impingement on the piston and combustion chamber surfaces [6], and (2) causes excessive fuel injection pressures during engine warm-up [4], poor fuel atomization and incomplete combustion. One possible method to overcome the high viscosity problem of biodiesel is blending with diesel fuel or alcohols in proper proportion. However, it is not practical to measure viscosities of the biodiesel-diesel fuel or biodiesel-alcohol blends at every turn for each blending ratio or temperature. Therefore, regression equations have been generally derived to estimate viscosity for the thermodynamic models of combustion process in the engine. However, some regression equations are available in the existing literature for predicting fuel properties of biodiesel-diesel fuel blends, there will be always need for reliable equations having higher accuracy. Therefore, in this study, (1) corn oil biodiesel was synthesized by means of basic transesterification, (2) the produced biodiesel (B100) was mixed with commercially available diesel fuel (D) on volume ratios of 5 (B5), 10 (B10), 15 (B15) and 20% (B20), (3) the densities and viscosities of biodiesel-diesel fuel blends were measured at different temperatures (10°C, 20°C, 30°C and 40°C) according to DIN 53015 standard and (4) the exponential and quadratic equations depending on temperature of blend were proposed and the equations were fitted to the experimental data.

2. MATERIALS AND METHODS

Biodiesel production
Corn oil biodiesel was produced in this study. Methanol, potassium hydroxide and anhydrous sodium sulphate were of analytical grades. Transesterification reaction parameters were determined according to the detailed parametric investigation in the authors’ previous study as: 1.10% catalyst concentration, 60°C reaction temperature, 60 min reaction time and 9:1 alcohol/oil molar ratio [7]. More details regarding biodiesel production can be found in Ref. [5]. Some important fuel properties (density at 15°C, kinematic viscosity at 40°C, flash point temperature, cold filter plugging point and heating value) of produced biodiesel were determined as: 877.94 kg/m³, 4.005 mm²/s, 171°C, -4°C and 39947 kJ/kg, respectively.

**Density and viscosity measurements**

The densities of all biodiesel-diesel fuel blends were measured at different temperatures by means of pycnometer accordingly ISO 4787 standard using a top loading balance with an accuracy of ±0.01 g. Dynamic viscosities of them were also measured using universal Haake Falling Ball Viscometer, Haake Water Bath, a stopwatch (±0.001 s) and a thermometer (±0.5 °C) according to DIN 53015. All measurements were repeated three times, and then average of them was taken to minimize the measurement errors. The kinematic viscosity was computed by dividing dynamic viscosity to density at the same temperature, as well known. Details about the measurements were also given in Refs. [5,7].

### 3. RESULTS AND DISCUSSION

**Viscosity variation**

The effects of temperature (T) on kinematic viscosities of pure fuels (biodiesel and diesel fuel) and biodiesel-diesel fuel blends (B5, B10, B15 and B20) were shown in Figure 1. In this figure, the points correspond to the measured viscosity values for studied temperatures and biodiesel fractions, while the lines are plots of the curve fit equation. Viscosities of all the blends decrease with increasing temperature, as expected. For the changes of viscosities vs. temperature, the exponential model (Eq. (1)) is recommended:

$$v = v(T) = a + b \cdot e^{-c \cdot T}$$

Table 1 lists the measured viscosities, relative errors between measured and calculated values at the measurement points, regression constants and correlation coefficients (R). The maximum error and the lowest R value were computed as 4.1425% and 0.9980, respectively. According to regression results and Fig. 1, the measured viscosity data were qualitatively and quantitatively properly described by the exponential model.

![Fig. 1. Change of viscosity vs. temperature](image-url)
Table 1. For different biodiesel fractions, the measured viscosities of biodiesel-diesel fuel blends, relative errors, regression constants and correlation coefficients.

<table>
<thead>
<tr>
<th>Biodiesel fraction X (%)</th>
<th>Temperature, T (°C)</th>
<th>Measured, v (mm²/s)</th>
<th>Regression constant</th>
<th>Relative error (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>10</td>
<td>20</td>
<td>30</td>
<td>40</td>
</tr>
<tr>
<td>0</td>
<td>4.218</td>
<td>3.298</td>
<td>2.863</td>
<td>2.700</td>
</tr>
<tr>
<td></td>
<td>2.5450</td>
<td>3.7630</td>
<td>0.0810</td>
<td>0.9999</td>
</tr>
<tr>
<td></td>
<td>0.0237</td>
<td>0.2517</td>
<td>0.4645</td>
<td>0.2815</td>
</tr>
<tr>
<td>5</td>
<td>4.406</td>
<td>3.400</td>
<td>3.051</td>
<td>2.805</td>
</tr>
<tr>
<td></td>
<td>2.6596</td>
<td>3.9929</td>
<td>0.0850</td>
<td>0.9992</td>
</tr>
<tr>
<td></td>
<td>0.0862</td>
<td>0.7353</td>
<td>1.4290</td>
<td>0.8517</td>
</tr>
<tr>
<td>10</td>
<td>4.564</td>
<td>3.582</td>
<td>3.113</td>
<td>3.060</td>
</tr>
<tr>
<td></td>
<td>2.9406</td>
<td>4.3310</td>
<td>0.0978</td>
<td>0.9880</td>
</tr>
<tr>
<td></td>
<td>0.1161</td>
<td>0.8068</td>
<td>1.8599</td>
<td>1.0719</td>
</tr>
<tr>
<td>15</td>
<td>4.645</td>
<td>3.755</td>
<td>3.408</td>
<td>3.293</td>
</tr>
<tr>
<td></td>
<td>2.9845</td>
<td>3.4218</td>
<td>0.0705</td>
<td>0.9992</td>
</tr>
<tr>
<td></td>
<td>0.1249</td>
<td>0.7696</td>
<td>1.3703</td>
<td>0.7510</td>
</tr>
<tr>
<td>20</td>
<td>4.835</td>
<td>3.889</td>
<td>3.566</td>
<td>3.293</td>
</tr>
<tr>
<td></td>
<td>3.1665</td>
<td>3.6856</td>
<td>0.0789</td>
<td>0.9988</td>
</tr>
<tr>
<td></td>
<td>0.1324</td>
<td>0.8383</td>
<td>1.5844</td>
<td>0.8898</td>
</tr>
<tr>
<td>100</td>
<td>7.750</td>
<td>5.636</td>
<td>4.898</td>
<td>4.005</td>
</tr>
<tr>
<td></td>
<td>3.3686</td>
<td>7.8119</td>
<td>0.0594</td>
<td>0.9981</td>
</tr>
<tr>
<td></td>
<td>0.3832</td>
<td>2.3989</td>
<td>4.1425</td>
<td>2.3970</td>
</tr>
</tbody>
</table>

Density variation

Fig. 2 demonstrates the variations of densities of pure fuels and biodiesel-diesel fuel blends with respect to temperature. Densities of all fuels decrease with increasing temperature, as expected. In order to characterize the variation, the quadratic model (Eq. (2)) was proposed:

\[ \rho = \rho(T) = a + b \cdot T + c \cdot T^2 \]  

(2)

The measured densities, relative errors between measured and calculated values at the measurement points, regression constants and correlation coefficients were listed in Table 2. The maximum error and the lowest R values were obtained as 0.009400% and 0.9998, respectively. According to these values, the quadratic model can be said to be given quantitatively well accurate results for the changes of densities of the all fuels with respect to temperature.

Table 2. For different biodiesel fractions, the measured densities of biodiesel-diesel fuel blends, relative errors, regression constants and correlation coefficients.

<table>
<thead>
<tr>
<th>Biodiesel fraction X (%)</th>
<th>Temperature, T (°C)</th>
<th>Measured, ( \rho ) (kg/m³)</th>
<th>Regression constant</th>
<th>Relative error (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>10</td>
<td>20</td>
<td>30</td>
<td>40</td>
</tr>
<tr>
<td>0</td>
<td>833.12</td>
<td>831.87</td>
<td>829.74</td>
<td>826.95</td>
</tr>
<tr>
<td></td>
<td>833.700</td>
<td>-1.390</td>
<td>-3.850</td>
<td>0.9998</td>
</tr>
<tr>
<td></td>
<td>0.00672</td>
<td>0.001440</td>
<td>0.009400</td>
<td>0.004110</td>
</tr>
<tr>
<td>5</td>
<td>835.36</td>
<td>834.10</td>
<td>831.97</td>
<td>829.17</td>
</tr>
<tr>
<td></td>
<td>835.900</td>
<td>-1.450</td>
<td>-3.850</td>
<td>0.9999</td>
</tr>
<tr>
<td></td>
<td>0.001197</td>
<td>0.005397</td>
<td>0.003606</td>
<td>0.001200</td>
</tr>
<tr>
<td>10</td>
<td>838.00</td>
<td>836.75</td>
<td>834.61</td>
<td>831.80</td>
</tr>
<tr>
<td></td>
<td>838.500</td>
<td>-1.240</td>
<td>-3.900</td>
<td>0.9998</td>
</tr>
<tr>
<td></td>
<td>0.001671</td>
<td>0.006932</td>
<td>0.000959</td>
<td>0.004328</td>
</tr>
<tr>
<td>15</td>
<td>839.43</td>
<td>838.17</td>
<td>836.03</td>
<td>833.22</td>
</tr>
<tr>
<td></td>
<td>840.000</td>
<td>-1.395</td>
<td>-3.875</td>
<td>0.9999</td>
</tr>
<tr>
<td></td>
<td>0.005123</td>
<td>0.000139</td>
<td>0.007655</td>
<td>0.002640</td>
</tr>
<tr>
<td>20</td>
<td>842.28</td>
<td>841.02</td>
<td>838.87</td>
<td>836.05</td>
</tr>
<tr>
<td></td>
<td>842.800</td>
<td>-1.340</td>
<td>-3.900</td>
<td>0.9998</td>
</tr>
<tr>
<td></td>
<td>0.000475</td>
<td>0.005707</td>
<td>0.002146</td>
<td>0.003110</td>
</tr>
<tr>
<td>100</td>
<td>877.72</td>
<td>876.40</td>
<td>874.16</td>
<td>871.22</td>
</tr>
<tr>
<td></td>
<td>878.300</td>
<td>-1.490</td>
<td>-4.050</td>
<td>0.9999</td>
</tr>
<tr>
<td></td>
<td>0.002962</td>
<td>0.002054</td>
<td>0.005491</td>
<td>0.000459</td>
</tr>
</tbody>
</table>
4. CONCLUSIONS

In this study, corn oil biodiesel was produced, the densities and viscosities of corn oil biodiesel-diesel fuel blends were measured at different temperatures, and the exponential and quadratic models were proposed to estimate these fuel properties. The following conclusions can be drawn from the study:

1) The maximum relative error and correlation coefficient (R) coming from the exponential model for the viscosity-temperature variation were determined as 4.1425% and 0.9980, respectively.
2) The proposed quadratic model with the maximum relative error of 0.009400% and the minimum correlation coefficient of 0.9998 is well suitable to describe the variation of density vs. temperature for the biodiesel-diesel fuel blends.

5. REFERENCES

TEMPERATURE AND BIODIESEL FRACTION DEPENDENCE OF DENSITY OF BIODIESEL-DIESEL FUEL BLENDS

Mert Gülüm, Atilla Bilgin
Karadeniz Technical University, Mechanical Engineering Department, Trabzon, Turkey
Corresponding Author: Mert Gülüm, gulum@ktu.edu.tr

ABSTRACT

In the presented study, biodiesel was produced from hazelnut oil, which is an agricultural product at the Black Sea region of Turkey, using methanol and sodium hydroxide via the transesterification reaction. The produced biodiesel was mixed with commercially available diesel fuel at the volume ratios of 5%, 10%, 15%, and 20%. The densities of each blend were measured at average climate conditions such as 10°C, 20°C, 30°C, and 40°C by following international ISO 4787 standard. A two-dimensional surface model was derived through the least squares regression method for the variations in densities of blends with respect to temperature and biodiesel fraction, simultaneously. Finally, densities of hazelnut oil biodiesel-diesel fuel binary blends were estimated using the surface model.

Keywords: Hazelnut oil biodiesel, density, fuel blends, two-dimensional model, estimation.

1. INTRODUCTION

There is an increasing worldwide concern for the environmental protection and the conservation of non-renewable natural resources. For this reason the possibility of developing alternative energy sources to replace traditional fossil fuels has been receiving a large interest in the last few decades. Fatty acid methyl esters (FAME) show large potential applications as diesel substitutes, and they are known as biodiesel [1]. Biodiesel has several advantages over conventional diesel. It is safe, renewable, non-toxic, and biodegradable in water. It contains less sulphur compounds, and it has a higher flash point (>130°C). Furthermore, it is almost neutral with regard to carbon dioxide emissions, and it emits 80% fewer hydrocarbons and ~50% less particles. Finally, biodiesel production enjoys a positive social impact, by enhancing rural revitalization [2,3]. However, the main problem associated with the use of biodiesel is its poor low-temperature flow properties, measured in terms of cloud point, pour point and cold filter plugging point. The low-temperature properties can be improved by blending with biodiesel produced from unsaturated feedstocks. Other major disadvantages of biodiesel are its higher viscosity, prices, engine wear and nitrogen-oxides (NOx) emissions, and lower energy content.

Density is one of the most important fuel properties since cetane number and calorific value are related to it. In addition, the change of density directly affects the engine output power and fuel consumption since the amount of fuel injected to combustion chamber is measured volumetrically for diesel engines. Further, density influences the start of injection, injection pressure and fuel spray characteristics affecting the combustion and exhaust emissions [4]. Although density measurement is not very complex, it is not practical to measure densities of biodiesel-diesel fuel blends for each biodiesel ratio in blend and temperature. For this reason, researchers have shown a strong interest in regression equations for rapid estimation of density and optimization of process equipment such as heat exchangers, reactors, mixing vessels and process piping. In fact, although many studies on measurement and prediction of densities of biodiesel-diesel fuel blends have been performed in the existing literature, there is a scarcity of two-dimensional surface equations depending on blending ratio and temperature. Therefore, in this study, (1) hazelnut oil biodiesel was produced by means of transesterification reaction, (2) the biodiesel was mixed with commercially available diesel fuel on different volume ratios (5%, 10%, 15% and 20%), (3) the densities of biodiesel-diesel fuel blends were measured at different temperatures (10°C, 20°C, 30°C and 40°C) according to ISO 4787 standard, and (4) a surface equation including quadratic and linear terms was fitted to the measurements.
2. MATERIALS AND METHODS

Biodiesel production

In this study, all reagents (methanol, sodium hydroxide and anhydrous sodium sulfate) were purchased from Merck. Refined hazelnut oil which is agricultural products at Black Sea region of Turkey was used to produce biodiesel. The reaction parameters for transesterification of hazelnut oil were determined as: catalyst concentration of 1.00%, reaction temperature of 50°C, reaction time of 60 minutes and alcohol/oil molar ratio of 9:1. More details on biodiesel production can be also found in the authors’ previous study [5].

Density measurements

The densities of the produced biodiesel and its blends were determined by means of a pycnometer and top loading balance, and measurements in accordance with ISO 4787 standard. In order to minimize measurement errors, all the measurements were conducted three times for each sample and the results were averaged. More details on density measurements can be also found in the authors’ previous study [5].

3. RESULTS AND DISCUSSION

In this section, a two-dimensional surface model (Eq. 1) was derived to make quick estimates of densities for a given blending ration \((X)\) and a specific temperature \((T)\) simultaneously. As well-known, the relationship between density and biodiesel percentage is linear while linear and quadratic models were tried to represent changes of densities vs. temperature, which may have nonlinear characteristics. In light of this knowledge, the two-dimensional surface model (Eq. 1) including quadratic term of temperature and linear term of blending ratio was fitted to the experimental density data. The tried surface model is given in the following:

\[
\rho = \rho(T, X) = a + bT + cX + dT^2 + eTX
\]  

(1)

Table 1 presents the regression parameters, measured densities, and % errors between density data and calculated values. The maximum error and R value of Eq. (1) are 0.0484% and 0.9983, respectively. According to these results, it can be said the variations of densities with X and T simultaneously is well correlated by Eq. (1). The plot of changes of constant density lines for fuel blends as functions of T and X calculated from Eq. (1) is given in Fig. 1. There are two characteristic regions where constant density lines have different gradients, as shown in Fig. 1. At the higher gradient regions, corresponding to the higher temperature and lower biodiesel percentage area, less temperature changes are required for a unit change of density for a given blending percentage, while higher temperature changes are needed at lower gradient regions, corresponding to the lower temperature and higher biodiesel percentage area. Finally, Eq. (1) has quite well accuracy with correlation coefficient of 0.9983 for representing change of density with respect to temperature and biodiesel percentage simultaneously.

<table>
<thead>
<tr>
<th>Temp. T (°C)</th>
<th>Blending ratio X (%)</th>
<th>Measured (\rho) (kg/m³)</th>
<th>Relative error rate (%)</th>
<th>Regression constants and correlation coefficient</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0</td>
<td>833.12</td>
<td>0.0155</td>
<td>a = 833.5000</td>
</tr>
<tr>
<td>5</td>
<td>5</td>
<td>834.95</td>
<td>0.0458</td>
<td>b = -1.2070x10⁻²</td>
</tr>
<tr>
<td>10</td>
<td>15</td>
<td>840.04</td>
<td>0.0027</td>
<td>c = 4.6950x10⁻¹</td>
</tr>
<tr>
<td>20</td>
<td>20</td>
<td>842.28</td>
<td>0.0094</td>
<td>d = -3.8850x10⁻³</td>
</tr>
<tr>
<td>30</td>
<td>0</td>
<td>831.87</td>
<td>0.0199</td>
<td>e = -1.0600x10⁻⁴</td>
</tr>
<tr>
<td></td>
<td>5</td>
<td>833.70</td>
<td>0.0409</td>
<td>R = 0.9983</td>
</tr>
<tr>
<td></td>
<td>10</td>
<td>836.75</td>
<td>0.0444</td>
<td></td>
</tr>
<tr>
<td></td>
<td>15</td>
<td>838.78</td>
<td>0.0077</td>
<td></td>
</tr>
<tr>
<td></td>
<td>20</td>
<td>841.02</td>
<td>0.0038</td>
<td></td>
</tr>
<tr>
<td></td>
<td>0</td>
<td>829.74</td>
<td>0.0119</td>
<td></td>
</tr>
<tr>
<td></td>
<td>5</td>
<td>831.57</td>
<td>0.0484</td>
<td></td>
</tr>
<tr>
<td></td>
<td>10</td>
<td>834.61</td>
<td>0.0366</td>
<td></td>
</tr>
<tr>
<td></td>
<td>15</td>
<td>836.64</td>
<td>0.0005</td>
<td></td>
</tr>
<tr>
<td></td>
<td>20</td>
<td>838.87</td>
<td>0.0117</td>
<td></td>
</tr>
</tbody>
</table>
CONCLUSIONS

In this study, hazelnut oil biodiesel (methyl ester) was produced via the basic transesterification, the biodiesel was mixed with diesel fuel on different volume basis to prepare binary blend, densities of binary blends were measured at different temperatures according to ISO 4787 test method, the effects of biodiesel percentage and temperature on the densities of binary blends were investigated, and the two-dimensional regression model was developed for predicting densities at various temperatures and biodiesel percentages. According to the results, the following conclusions can be drawn from this study:

1) The change of density with respect to temperature and biodiesel percentage was found to be well expressed by the two-dimensional Eq. (1) with correlation coefficient of 0.9983. The maximum error between the measured and calculated density values was computed as 0.0484% for this model.

2) In the two-dimensional constant density line plot (Fig. 1) obtained by Eq. (1), there exist two characteristic regions. At the higher gradient regions, less temperature changes are required for a unit change of density for a given blending percentage, while higher temperature changes are needed at lower gradient regions.

REFERENCES


DENSITY VARIATION OF ETHYL ESTER-DIESEL-BUTANOL TERNARY BLENDS

Mert Gülüm, Atilla Bilgin
Karadeniz Technical University, Mechanical Engineering Department, Trabzon, Turkey
gulum@ktu.edu.tr, bilgin@ktu.edu.tr

ABSTRACT
Waste cooking oil ethyl ester (biodiesel) having the lowest viscosity was produced via the basic transesterification. The ethyl ester was blended with commercially available fossil diesel fuel and n-butanol to prepare ternary blends. The density measurements of ethyl-ester-diesel-n-butanol ternary blends was performed at different temperatures (278.15 K-368.15 K, in steps of 5 K) according to the related international standard. According to the variations in densities of biodiesel-diesel-fuel-n-butanol ternary blends with respect to temperature, the exponential model, previously recommended by the authors, was fitted to predict densities of ternary blends. According to results, the exponential model is found to be suitable one in predicting densities.

Keywords: Waste cooking oil, ethyl ester, density, transesterification, butanol, fuel property.

1. INTRODUCTION
Renewable fuels have gained attention during the last few decades because of the decreasing oil supply and increasing environmental consciousness. Among these fuels, biodiesel used as substitutes for conventional petroleum fuel in diesel engines has also recently received increased interest. This interest is based on a number of properties of biodiesel including its biodegradability and the fact that it is produced from a renewable resource. These features of biodiesel lead to its greatest advantage, which is its potential for emission reduction [1]. Chemically, biodiesel is composed of mono-alkyl esters of long-chain fatty acids derived from renewable feed stocks (vegetable oil or animal fats). It is generally synthesized by transesterification in which the triglycerides in vegetable oils react with alcohols of low molecular weights in the presence of suitable catalysts [2].

The knowledge of thermodynamic properties of biodiesel plays an important role in the understanding of molecular interactions, affecting their thermodynamic properties [3]. Density directly affects the engine performance characteristics, and some fuel properties (cetane number, heating value, etc.) are related to density. Therefore, density data of fuel as a function of temperature are needed to combustion models and other applications [4]. In addition, it is very critically important to know whether the densities of biodiesel-diesel binary or biodiesel-diesel-alcohol ternary blends ensure the related international standards, or not. Therefore, this study purposes to (1) measure densities of biodiesel-diesel-n-butanol ternary blends at different temperatures, (2) derive a regression model as a function of temperature which can be used in combustion models for predicting density, and finally (3) investigate whether the density data of ternary blends are within the related international biodiesel standards, or not. For these aims, (i) waste cooking oil biodiesel (ethyl ester) was produced by means of the basic-transesterification reaction, (ii) the biodiesel was mixed with commercially available diesel fuel on a 20% volume basis, (iii) the biodiesel-diesel binary blend (B20) was blended with n-butanol on 4% (Bu4), 6% (Bu6) and 8% (Bu8) volume ratios to prepare ternary blends, (iv) the densities of ternary blends were measured at different temperatures (from 278.15 K to 368.15 K) according to ISO 4787 standard, (v) the effect of temperature on change of density of ternary blends was investigated, (vi) the exponential model, previously suggested by the authors for predicting densities of different biodiesel-diesel binary blends [9], was fitted to the density data of ternary blends, and (vii) the proposed model was compared to well-known linear model previously recommended in the existing literature.

2. MATERIALS AND METHODS
Biodiesel production
In this study, waste cooking oil ethyl ester was produced. Waste cooking oil was obtained from a canteen in Karadeniz Technical University. Ethanol, sodium hydroxide and anhydrous sodium sulphate used in transesterification were of
analytical grades. Transesterification reaction parameters were determined according to the detailed parametric investigation performed by the authors: 1.25% catalyst concentration, 70°C reaction temperature, 120 min reaction time and 12:1 alcohol/oil molar ratio [5]. More details regarding biodiesel production can be found in [4].

Density measurements

The densities of ternary blends were measured at different temperatures by means of pycnometer accordingly ISO 4787 standard using a top loading balance with an accuracy of ±0.01 g. More details of the measurements were given in Ref. [4].

3. RESULTS AND DISCUSSION

The effects of temperature on densities of waste cooking oil-diesel fuel-n-butanol ternary blends (Bu4, Bu6 and Bu8) measured at different temperatures (from 278.15 K to 368.15 K) were shown in Figs. 1-3 where densities decrease with increasing temperature, as expected. Symbols, and red and green lines correspond the experimental data, and computed values from the exponential and linear models, respectively. The changes in densities versus temperature were correlated by exponential and linear models given in Eqs. (1) and (2):

\[
\rho = \rho(T) = a \cdot e^{b \cdot T} + c \cdot e^{d \cdot T} \quad (1)
\]

\[
\rho = \rho(T) = a + b \cdot T \quad (2)
\]

where \( \rho \) is density in \( \text{kg/m}^3 \), \( T \) is temperature of blend (K), and \( a, b, c \) and \( d \) are regression constants. Density values of ternary blends measured by the authors, percent relative errors between measured and calculated density values at the measurement points from Eqs. (1) and (2), and regression parameters of models (regression constants and correlation coefficients (\( R \))) were listed in Table 1. The maximum errors and lowest \( R \) values were computed as 0.0685%, 0.9999 (all of them) and 0.5060%, 0.9808 for the exponential and linear models, respectively. These results and Fig. 1 indicate that the exponential model quite well represents the experimental value than the linear model.

![Fig. 1. Density variation vs. temperature for Bu4.](image-url)
Table 1. Density measurements of ternary blends performed by the authors, errors between measured and estimated densities by the exponential and linear models, and regression parameters of models.

<table>
<thead>
<tr>
<th>Fuel</th>
<th>Measured, $\rho$ (kg/m$^3$)</th>
<th>Temp., T (K)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>278.15</td>
</tr>
<tr>
<td>Bu4</td>
<td>838.90</td>
<td>838.69</td>
</tr>
<tr>
<td>Bu6</td>
<td>838.30</td>
<td>838.09</td>
</tr>
<tr>
<td>Bu8</td>
<td>837.90</td>
<td>837.69</td>
</tr>
</tbody>
</table>

Table 1. (Continued)

<table>
<thead>
<tr>
<th>Fuel</th>
<th>Measured, $\rho$ (kg/m$^3$)</th>
<th>Temp., T (K)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>323.15</td>
</tr>
<tr>
<td>Bu4</td>
<td>828.95</td>
<td>826.50</td>
</tr>
<tr>
<td>Bu6</td>
<td>828.35</td>
<td>825.91</td>
</tr>
<tr>
<td>Bu8</td>
<td>827.96</td>
<td>825.51</td>
</tr>
</tbody>
</table>
Table 1. (Continued)

<table>
<thead>
<tr>
<th>Fuel</th>
<th>Eqs.</th>
<th>Regression constants</th>
<th>R</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bu4</td>
<td>Eq. (1)</td>
<td>a: 1270, b: -0.001125, c: -2089, d: -0.01132</td>
<td>0.9999</td>
</tr>
<tr>
<td>Bu6</td>
<td>Eq. (1)</td>
<td>a: 1266, b: -0.001121, c: -2102, d: -0.01139</td>
<td>0.9999</td>
</tr>
<tr>
<td>Bu8</td>
<td>Eq. (1)</td>
<td>a: 1270, b: -0.001128, c: -2071, d: -0.01127</td>
<td>0.9999</td>
</tr>
<tr>
<td>Bu4</td>
<td>Eq. (2)</td>
<td>a: 9.450e2, b: -3.665e-1, c: -1, d: -</td>
<td>0.9808</td>
</tr>
<tr>
<td>Bu6</td>
<td>Eq. (2)</td>
<td>a: 9.444e2, b: -3.662e-1, c: -1, d: -</td>
<td>0.9807</td>
</tr>
<tr>
<td>Bu8</td>
<td>Eq. (2)</td>
<td>a: 9.439e2, b: -3.660e-1, c: -1, d: -</td>
<td>0.9808</td>
</tr>
</tbody>
</table>

Table 1. (Continued)

<table>
<thead>
<tr>
<th>Fuel</th>
<th>Eqs.</th>
<th>Relative errors (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>Temp., T (K)</td>
</tr>
<tr>
<td>Bu4</td>
<td>Eq. (1)</td>
<td>278.15  0.0268  0.0188  0.0320  0.0432  0.0277  0.0118  0.0255  0.0419</td>
</tr>
<tr>
<td>Bu6</td>
<td>Eq. (1)</td>
<td>298.15  0.0133  0.0045  0.0169  0.0273  0.0110  0.0292  0.0074  0.0231</td>
</tr>
<tr>
<td>Bu8</td>
<td>Eq. (1)</td>
<td>308.15  0.0027  0.0112  0.0015  0.0124  0.0034  0.0432  0.0059  0.0103</td>
</tr>
<tr>
<td>Bu4</td>
<td>Eq. (2)</td>
<td>293.15  0.4957  0.3023  0.1591  0.0407  0.0791  0.1992  0.2203  0.2415</td>
</tr>
<tr>
<td>Bu6</td>
<td>Eq. (2)</td>
<td>298.15  0.5060  0.3127  0.1695  0.0512  0.0685  0.1885  0.2094  0.2303</td>
</tr>
<tr>
<td>Bu8</td>
<td>Eq. (2)</td>
<td>303.15  0.5009  0.3076  0.1645  0.0463  0.0734  0.1933  0.2141  0.2349</td>
</tr>
</tbody>
</table>

Table 1. (Continued)

<table>
<thead>
<tr>
<th>Fuel</th>
<th>Eqs.</th>
<th>Relative errors (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>Temp., T (K)</td>
</tr>
<tr>
<td>Bu4</td>
<td>Eq. (1)</td>
<td>318.15  0.0372  0.0126  0.0685  0.0088  0.0294  0.0345  0.0250  0.0003</td>
</tr>
<tr>
<td>Bu6</td>
<td>Eq. (1)</td>
<td>338.15  0.0166  0.0073  0.0468  0.0134  0.0068  0.0115  0.0017  0.0233</td>
</tr>
<tr>
<td>Bu8</td>
<td>Eq. (1)</td>
<td>348.15  0.0044  0.0201  0.0359  0.0251  0.0043  0.0010  0.0084  0.0342</td>
</tr>
<tr>
<td>Bu4</td>
<td>Eq. (2)</td>
<td>323.15  0.2639  0.2877  0.2138  0.2388  0.1680  0.0981  0.0290  0.0381</td>
</tr>
<tr>
<td>Bu6</td>
<td>Eq. (2)</td>
<td>333.15  0.2538  0.2762  0.2032  0.2280  0.1570  0.0868  0.0174  0.0499</td>
</tr>
<tr>
<td>Bu8</td>
<td>Eq. (2)</td>
<td>343.15  0.2583  0.2818  0.2075  0.2334  0.1623  0.0919  0.0224  0.0439</td>
</tr>
</tbody>
</table>

4. CONCLUSION

In this study, densities of waste cooking oil ethyl ester-diesel fuel-n-butanol ternary blends (Bu4, Bu6 and Bu8) were measured for different temperatures (from 278.15 K to 368.15 K) according to international ISO 4787 standard. Measured data showing the distributions of densities with respect to temperature were fitted with the exponential and linear models by means of regression analysis. The following conclusions can be deduced from this study:

1) Densities of ternary blends non-linearly decrease with increasing temperature.
2) The maximum relative error and the minimum correlation coefficient (R) for the linear model recommended for the variation of density with respect to temperature were determined as 0.5060% and 0.9808, respectively.
3) The exponential model having the maximum relative error of 0.0685% and the correlation coefficients of 0.9999 is well suitable to describe the variations of densities of the ternary blends.

5. REFERENCES


MODELING AND ANALYSIS OF ANISOGRID LATTICE STRUCTURES USING AN INTEGRATED ALGORITHMIC MODELLING FRAMEWORK

Reza Mohammadzadeh Gheslaghi1, Hamit Akbulut2

1Architectutre Dept., Architecture and Design Faculty, Ataturk University, Erzurum, Turkey
2Mechanical Engineering Dept., Engineering Faculty, Ataturk University, Erzurum, Turkey
1reza.mohammadzadeh@gmail.com

ABSTRACT

Lightweight structures have become very important in many technological applications such as aerospace, automotive, shipbuilding, military industries. This paper studies the composite lattice structures (Anisogrid Lattice Structures) as one of the main solutions. The term Anisogrid lattice Structure is predicated to Anisotropic Grid Structures [1]. In this study, Anisogrid lattice structures are modeled using a 3D algorithmic modeling technique. 3D models are transferred into finite element analysis environment integrated into the algorithmic modeling environment and then the finite element analysis is carried out on it. The results obtained from finite element analysis are compared with those obtained from experimental tests. The results of numerical analysis obtained from this study show that the acceptable conformance with the results of experimental tests was carried out in past literature. The results show that algorithmic modeling can be used as a flexible modeling method for modeling of Anisogrid lattice structures.

Keywords: Anisogrid, Lattice Structures, Composite Isogrid, Composite Structures, Composite, Algorithmic Modelling

1. INTRODUCTION

Considering the fact that triangular trusses are very efficient, in 1964, Dr. Robert R. Meyer under a NASA-MSFC contract began to find an optimum pattern for stiffening under pressure loads. It produced immense results and expanded into cylinders as an independent research and development program. The new structure was called the "Isogrid", because it behaved like an isotropic material [2].

The isogrid is a name used to refer to continuous reinforced structures and shells that grids form an equilateral triangular pattern (Figure 1). This pattern allows the structure exhibit rigidity to weight and strength to weight characteristics which is expected in many applications. Also, the repetitive pattern is a major factor in reducing the cost of construction due to the efficient use of capital equipment and rapid construction methods[3].

Figure 1. Isogrid Machined Aluminum Panel

Composite materials are widely used in many applications particularly aerospace, defense, automotive and sports applications, due to their good mechanical, flexible and easy manufacturing characteristics. The term Anisogrid lattice
structure is predicated to Anisotropic Grid Structures according to [1]. Contrary to the Isogrid structures, the Anisogrid structures can have different patterns and are not limited to the equilateral triangle. In other words, they are non-isotropic in terms of both material and structural patterns. Anisogrid structures can be made with or without skin depends on design requirements. Advanced Anisogrid lattice structures formula, weight and bending stiffness and suggested fabrication procedure are defined in a NASA technical memorandum at 1975 [4].

An HM-S/X-30 graphite/epoxy composite curved Isogrid panel, maid using thermal pressure forming method, is introduced in [3] which it shows superior material strength and thermal stability properties with minimal tooling complexity. Although advances in Anisogrid structures started lease than 50 years ago [5], on of ancient evidence of using of such structures can be seen in Turkmens traditional homes called Yurt (Figure 2). They have been used a wooden lattice structure as the core of their homes since long time ago. The structure comprises an angled assembly or latticework of pieces of wood or bamboo for walls, a door frame, ribs (poles, rafters), and a wheel (crown, compression ring) possibly steam-bent[6].

Several studies have been carried out on the analysis and optimization of anisogrid lattice structures. Thomas D. Kim (1999) explained the composite isogrid rigid cylinder manufacturing and test. The axial compression test objective identifies various fault patterns in structures, such as ribs, skin collapse, and general instability. Rib failure is find as critical failure mode for anisogrid cylinder[7].

Samuel Kidanea et al. (2003) study generally determines the overall buckling load for the cross and horizontal grid-hardened composite cylinder. In this assay, the matrices associated with stiffeners were identified by coupling and bending matrices (respectively, based on A, B and D). Using the energy method, the buckling load was solved for a specific stiffener configuration. The buckling test was also performed on the hardened composite cylinder and compared with the analytical results[8].

G. Totaro and Z. Gurdal (2009) proposed an optimization method for the composite lattice shell structures under axial compressive loads, aimed at preliminary design. The proposed method allows the designer to implement numerical minimization to easily process the optimal configurations located at least near the Mass solution [9].

Morozov et al. (2011) examined the buckling behavior of cylindrical shells of anisogrid composite cage under axial pressure, transverse bending, pure bending and torsion. Three-dimensional frame structures are modeled in lattice shells, consisting of curvilinear ribs subjected to stretching / clamping, in two planes and torsion bending. The length-changing effect of the shells, the number of helical beams, and the orientation angles on the buckling behavior of the cage structures were analyzed using parameter analysis. Cage crusts with cuts are also analyzed for buckling [10].

M. Buragohain and R. Velmurugan (2011) Three different non-rigidized shell of circular cylindrical structures, cage cylinder (rib only) and grid hardened shell (with shell and ribs) have been evaluated for experimental study and adapted to a series of structures simplified and cheap production process they had. Axial pressure tests were performed and the results were compared with finite element analysis [11].

Tom Mathew and colleagues (2013) investigated the strength of composite grid lattice structures and tried to obtain the most heavily efficient design with the highest compressive load efficiency. In addition, finite element analysis using MSC Patran parametric studies on the structure analyzed. Cylindrical grided model and different loading conditions are

![Figure 2. Turkmens traditional homes (Yurt)](image)
evaluated with conical grided model. It has been found that the ratio in width is possible to withstand a load of 400kN up to 2.5 and 30° helix angle structures. It provides very high weight strength ratio and cost advantages [12].

In general, anisogrid structures are automatically produced by continuous filament winding. Made of fiber reinforced composite and thin ribs, these structures are lightweight and have high resistance to loads. In the literature, 3D shell or beam elements have been proposed as different analysis elements. The analyzes are performed with some commercial programs such as ABAQUS, ANSYS and SAP.

Finite Element modeling of the anisogrid lattice structures is time-consuming and in some cases requires the development of a special program. Today, with the development of parametric modeling programs, this problem has been somewhat resolved, but the flexibility of these programs is not as large as algorithmic modeling programs. In this paper, an environment for the algorithmic modeling of anisogrid structures is designed. More precisely, in this environment, instead of being modeled, it is programmed. This is very important when we need to make extensive changes to the various parameters of finite element models and repeated analyzes in the optimization process.

2. ALGORITHMIC MODELING

Anisogrid (anisotropic grid) composite lattice structures are usually made in the form of a cylindrical or conical Shell consisting of helical and circumferential (hoop) unidirectional composite ribs formed by continuous winding (Figure 3). Anisogrid carbon-epoxy lattice structures are normally designed for axial compression as the main loading case. The ribs are the principal load-bearing elements of the structure, whereas the skin, the presence of which can be justified by design requirements, is not considered as a load-bearing element in the design of lattice structures[13].

Grasshopper is a platform in Rhino to deal with the Algorithms modelling. Rhinoceros (Rhino) 3D is a CAD / CAM design software used in 3D modeling and prototyping, especially in industrial applications. It is more suitable for obtaining 3D prototypes as it uses NURBS modeling instead of mesh modeling. Grasshopper is an add-on program that allows algorithmic modeling using Rhino History.

In 3D Grasshopper, geometric models of anisogrid structures were formed with the help of developed algorithmic modeling framework. Algorithmic modeling framework consists of various command blocks and the connections of these commands are provided by means of data wires. The variables of each command are defined in the command blocks in the form of inputs and outputs. Figure 4 shows an overview of the modeling framework.
ANSYS finite element program is used for FEM analysis. Anisogrid structures are produced from continuous fibers, it is appropriate to use beam element in the analysis. The selected analysis element is 3D beam 188 which supports orthotropic materials and is based on the Tymoshenko beam theory. It also has 3D and 6 degrees of freedom. The framework constitutes a text file in accordance with the ANSYS APDL program in four steps.

In the first step; The shape of the pattern is formed on the plane (Figure 5).

In the second step; the determined pattern is projected onto the cylindrical shell and at the same time divided into elements according to the FEM analysis program (Figure 6).

In the third step; The model is placed into a suitable shape for analysis and also material properties, cross-sectional area, shape, loads and boundary conditions are applied. Figures 7,8 show FEM modeling block and three-dimensional geometries respectively.
In fifth step: The model is recorded as an ANSYS APDL file. This file carries the analysis model to the ANSYS finite element analysis program automatically. The ANSYS program starts analyzing after reading the APDL file. In this analysis, the critical buckling loads of each model are determined according to the first mode. Figures 9, 10 and 11 show the finite element model and the displacements respectively.
The upper flange of the cylindrical structure has a distributed unit load and the lower part is completely fixed.

3. ANALYSIS RESULTS

In order to prove the correct operation of the program, modeling and analysis were done by taking a few samples from the literature. Table 1 provides some results from the Buragohain et al.[11] and their comparison with the results of this paper.

<table>
<thead>
<tr>
<th>Model Dimensions (mm)</th>
<th>Material Properties</th>
<th>Buckling Load (KN)</th>
<th>Differences(FEM Analysis) %</th>
</tr>
</thead>
<tbody>
<tr>
<td>Diameter</td>
<td>Carbon fiber(T300), Epoxy (LY55/HY5200)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Diameter</td>
<td>140</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Height</td>
<td>204</td>
<td>E₁₁</td>
<td>44.2 GPa</td>
</tr>
<tr>
<td>Ribs Dimensions</td>
<td>3.2x6</td>
<td>E₁₂</td>
<td>5.0 GPa</td>
</tr>
<tr>
<td>Ribs Dimensions</td>
<td>4.0x6</td>
<td>G₁₂</td>
<td>2.4 GPa</td>
</tr>
<tr>
<td>Ribs Dimensions</td>
<td>4.8x6</td>
<td>V₁₂</td>
<td>0.194</td>
</tr>
</tbody>
</table>

There are 5.6% differences between the results according to numerical FEM analysis. The reason for this is that the models formed by a different method in this study may be. Due to the use of curved lines instead of linear lines in the models used in this study, it is considered that more realistic results are obtained than studies in the literature [14].
4. CONCLUSION

An algorithmic modeling environment was developed for modeling anisogride structures. Models created in the algorithmic framework transferred to ANSYS FEM analysis program and buckling analysis is carried out. To verify the accuracy of the results, these results were compared with experimental and numerical studies. The results are consistent with the results of previous studies. The maximum difference between the results of this analysis and previous studies is 6%, which seems to be due to the difference in the modeling and analysis environment. Regarding the results, it can be said that the algorithmic modeling environment is a flexible and suitable environment for anisotropic structures modeling, especially when there is a need for continuous changes in design parameters and frequent analyzes.

REFERENCES

REVIEW ON AUXETIC MATERIALS

Onur GUNEL¹, Mostafa RANJBAR¹

¹. Ankara Yıldırım Beyazıt University, Department of Mechanical Engineering, Ankara, Turkey

Corresponding Author: Onur Gunel, ogunel@ybu.edu.tr

ABSTRACT

Mechanical properties of materials are important for engineering applications. In addition, poisson’s ratio is one of the important mechanical properties in terms of shear strength and modulus of the material. However, auxetic materials have a negative poisson’s ratio value, on contrary of conventional materials. Therefore, auxetic materials show different deformation mechanism from conventional materials. In this paper, the different type of auxetic materials are defined by giving information to the readers about history and structure of them. Moreover, in-plane and out-of-plane linear elastic mechanical properties of hexagonal and the anti-tetrachiral honeycomb are described based on homogenization approach. Also, the studies on vibration and energy absorption of auxetic materials are reviewed and found that the auxetic materials can be used as a passive noise controller. Investigated papers show that auxetic materials have significant advantages over conventional materials for their high shear strength and modulus. Thus, auxetic materials are important candidate for the biomedical, aerospace, smart filters, sensor and actuator and electromagnetic launcher application.

Keywords: negative poisson’s ratio, auxetic materials, cellular solid structures, honeycomb structures

1. INTRODUCTION

High shear strength and modulus, high impact strength, high energy absorption capability and fatigue resistance are important material properties to design advanced structures for engineering applications. Composite materials have some combination of these properties, except impact damage, and energy absorption [1]. It is thought that auxetic materials may get an important role in engineering structure. Therefore, the studies on auxetic materials have become more popular, recently.

Honeycomb structures have been used as sandwich panel cores in engineering applications such as aerospace, since the maiden flight in November 1940 of the De Havilland Mosquito aircraft [2] and also, naval and packaging applications [3]. Due to that he needs of low mass and cost components is important in transport applications, honeycomb structures are preferred [4]. Actually, honeycomb structures are widely used example of cellular core configurations and the general example of cellular solid as a sandwich core is hexagonal honeycomb and unit cell of hexagonal structure has ribs with same length and internal cell angle of 30° [5]. Cellular materials have some superiority mechanical and thermal properties when compared with solid materials [5], such as low density, high acoustic, isolation and damping, better thermal management, high energy absorption capability, durability at dynamic loadings. Due to that, honeycomb sandwich structures have superior out-of-plane mechanical properties which can change with their shape and topology [2].

Negative poisson ratio materials are used as a core in honeycomb sandwich structure [6-9]. This type of materials is called as “Auxetic” material. “Auxetic” materials under uniaxial loading show different behavior from conventional materials. Conventional materials expand in load direction but, “Auxetic” materials can expand in all direction when uniaxial load is applied [10].

Structures with negative Poisson's ratio show synclastic curvature or dome-shape curvature when bending load is applied. Therefore, they are good candidate for structure which is used under bending load [11-14]. However, a conventional material shows anticlastic curvature, or saddle shape, when it is subjected to an out-of-plane bending moment.

The manufacturing of auxetic cellular cores needs complicated mold and heat compression processes, but, development on 3D printing technology allows to manufacture auxetic cores with complex geometry [15]. It is important for the use of auxetic materials become widespread.

Sandwich panels are used in aerospace and automotive applications as an exterior frame due to the protection from the low impact which is caused by small objects hitting. Sandwich structures with auxetic cellular cores show unique mechanical properties such as increased indentation resistance [16], shear resistance [13], fracture toughness [17-19], and
energy absorption capacity [1,20-22] Mechanical properties of sandwich structures such as shear modules could be developed by using auxetic cellular material as a core material [13]. In addition, the amount of deflection under bending force could be reduced [20]. The different studies on sandwich panels with auxetic cores have been carried out to determine the structural behavior such as small deformation conditions [23-25], large deformation situations [26,27], bending responses [28-30], and impact resistance [31,32].

Gradient honeycombs with wall thickness which linearly increase, have been modeled and tested by Lira and Scarpa and they improved transverse shear modulus when considering to conventional hexagonal structure [33]. Also, the cellular structures with gradient thickness increase specific shear stiffness compared to conventional configurations and The flexural properties and failure behavior of sandwich structures with changing internal cell angle have been tested. They determine that global and local mechanical response could be tuned and controlled with geometry and core material [28]. The studies of Scarpa F et. al. [34] and Lira C. et. Al [33] exhibits that auxetic cellular cores have better shear properties than conventional lightweight cores.

Also, a few studies have been performed on architected sandwich panels to determine flexural behavior [35-37], out-of-plane compressive strength [38,39].

Several studies have been performed on architected 3D printed cores. These studies show that it is possible to optimize energy absorption of sandwich structure with auxetic core, thus, their response under impact load can be improved than conventional sandwich structures [40-42]. Sarvestani, H. Y., et. al. have determined the energy absorption of sandwich panel with different architectured 3D printed core. They have obtained that when relative density of core could be selected properly, level of energy absorption capability could be increased up to 33% compared to the rectangular and hexagonal sandwich panels [43].

There are several papers which investigate the vibrational and acoustic behavior of sandwich structures with normal or gradient cellular core. Scarpa F et. al have been studied on re-entrant auxetic honeycomb to determine their vibrational characteristics and it is possible to decrease the natural frequencies of honeycomb structure by means of increasing density of honeycomb [34].

This paper gives a short review of the types, manufacturing methods, mechanical properties, applications and vibration and energy absorption capabilities of cellular auxetic materials.

2. What is the Poisson’s ratio?

Poisson’s Ratio was invented by Simeon Dennis Poisson [44]. It is defined as the ratio of transverse contraction strain to longitudinal extension strain with respect to the direction of stretching force applied. Normal materials have a positive poisson’s ratio. Poisson’s ratio is 0.3 for most materials and slightly less than 0.5 for rubbery materials [45,46].

It is known that there is a correlation between the Poisson’s ratio and the atomic packing density. The poisson’s can change between -1 to 0.5 according to atomically dense of material. For instance, gold which has its poisson’s ratio 0.42 poisson’s ratio, but, atomically less dense materials such as steels which have their Poisson’s ratio 0.3.

Also, the typical value of Poisson’s ratio for ceramics, glass, and semi-conductors changes between 0.25–0.42 [47-50]. The poisson’s can change between -1 to 0.5 according to isotropic linear elasticity theory. When the microstructure of a material is change to make poisson’s ratio lower and young modulus is not change, its shear modulus and bulk modulus can be changed. Therefore, most materials show resistance in volume change than in shape change. It means that bulk modulus of materials is more than shear modulus. [50,51]:

The relations between shear modulus, young modulus, bulk modulus and poisson’s ratio are given in Eq.1 and Eq.2

\[ G = \frac{E}{2(1 + \nu)} \quad (1) \]

and

\[ K = \frac{E}{3(1 - 2\nu)} \quad (2) \]
3. Auxetic Materials

The term “auxetic materials” was first used by Evans et al. in 1991 [52]. The word “auxetic” is derived from the word αυχητικοξ (read: auxetikos). It means that it tends to increase. Known natural auxetic materials have been found in single crystals of arsenic [53] and cadmium [54], α-cristobalite [55], iron pyrites [56], and many cubic elemental metals [57], certain forms of skin (e.g. cat skin [58], salamander skin [59], and cow teat skin [60], and load-bearing cancellous bone from human shins [61].

As it is known, most materials shrink in the transverse direction when pulled in the longitudinal direction or expand in the transverse direction when compressed in the longitudinal direction, it means that they have a positive Poisson’s ratio (PR) [62]. However, the materials with negative poisson’s ratio can be define as it expands in the transverse direction when loaded in the longitudinal direction [63].

Owing to this unusual characteristic behavior, the materials with negative poisson’s ratio have several advantages such as high shear modulus, synclastic curvature, high damping resistance, high fracture toughness, enhanced crack growth resistance and high energy absorption capability [64].

Auxetic materials can be divided in to four main group as the following:

- auxetic honeycombs [65]
- special subsets of foams [13]
- auxetic microporous polymers [66]
- long fibre composites [67]

3.1. Auxetic Honeycombs

Conventional and auxetic honeycomb structures have been investigated widely for few decades. Auxetic honeycomb structures can be investigated under four main types as the following:

- Re-entrant Structures
- Rotating Units
- Missing Rib
- Chiral Structures

3.1.1. Re-entrant Structures

The 2D re-entrant honeycombs structures were suggested firstly by Almgren in 1985 [68]. The 2D re-entrant honeycombs was developed by using analytical calculations of various deformation mechanisms [68]. The 2D re-entrant auxetic structure has orthotropic mechanical properties.

Also, there are some studies on the 3D re-entrant honeycombs [69,70]. The 3D re-entrant honeycomb structures were suggested firstly by Evans et. al in 1994 [71]. The 3D re-entrant honeycomb structures have negative Poisson’s ratios in all three principal directions.

3.1.2. Rotating Units

Grima and Evans have reviewed auxetic structures with rotating unit [72]. The poison’s ratio of these types of structures equals to -1 when carried out analytical analysis for rotating rectangles and triangles [72-74]. Also, Scarpa, F. et. al. have developed semi-rigid rotating units and they have carried out experiment and numerical simulation [75,76].

3.1.3. Missing Rib

When remove some ribs in conventional honeycomb, structure exhibits the auxetic behavior [77]. A missing rib model was suggested by Smith et. al. in 2000. They have obtained mechanical properties of missing rib model [77].

3.1.4. Chiral Structures

The first time, auxetic chiral configuration has been suggested by Wojcichowski. His suggestion has been based on rotating disks and nearest neighbor inverse nth power interactions [78]. Chiral structures are consisted by connecting straight ligaments (ribs) to central nodes. Central nodes can be circles or other geometrical forms. Jackmans, R. et. al.
have determined poisson’s ratio of chiral structures theoretically and experimentally as \(-1\) [79]. Chiral cellular solids have superiority over conventional hexagonal honeycombs such as the compressive and shear strengths [3].

### 3.2. Auxetic Composites

It is possible to transform a non-auxetic material into auxetic materials by means of use different stacking sequences of individual lamina and also, lamina material should be highly anisotropic [1]. Specially designed software are used to determine the stacking sequence and the software also can determine mechanical properties such as maximum stiffness, bending strength. Several different optimization approaches could be used to design stacking sequence, but, general approach for the optimization of stacking sequence was provided by Wenchao and Evans [80].

Evans and Alderson [81] have suggested that auxetic reinforcements are used to enhance the fracture toughness of a composite. When an auxetic fibre are pulled out of the matrix, it will expand and not pulling out of the matrix easily. Auxetic polypropylene (PP) fibres have been produced [82] and embedded auxetic fibres are used to produce a single fibre composite [83] Posisitive poisson’s ratio and auxetic fibers of pull-out resistance and energy absorption were tested and the auxetic fibre can carry more than twice the maximum load than positive poisson’s ratio fibre. Also, energy absorption capability of auxetic fibre is more than three times than positive poisson’s ratio fibre.

### 4. Mechanical Properties of Auxetic Sandwich Panel

#### 4.1. Mechanical Properties of Auxetic Hexagonal Sandwich Panel

Gibson et. al. have used to an analytical model to determine the mechanical properties [5]. Lira et al. and Ranjbar et al. have used the same analytical method to determine the mechanical properties of auxetic hexagonal core. Their model depends on three non-dimensional parameters \(\alpha = h/l, \beta = t/l, \gamma = b/l\) and the angle \(\theta\) [84,85]. The auxetic hexagonal core are modeled as orthotropic material and the engineering constants \(E_x, E_y, E_z, G_{xy}, G_{xz}\) can be determined as in the literature [5,84]. The compliance matrix \([S]\) of orthotropic materials is given in Eq. 3. The out-of-plane Poisson’s ratios \(\nu_{xz}\) and \(\nu_{yz}\) can be used as about zero when Cellular Material Theory are considered [5]. However, the transverse Poisson’s ratios \(\nu_{yx}, \nu_{xy}\) can be used as Poisson’s ratio of the core material, \(\nu_c\) [5,84].

\[
[S] = \begin{bmatrix}
\frac{1}{E_x} & -\frac{\gamma}{E_y} & -\frac{\gamma}{E_z} & 0 & 0 & 0 \\
-\frac{\gamma}{E_x} & \frac{1}{E_y} & -\frac{\gamma}{E_z} & 0 & 0 & 0 \\
-\frac{\gamma}{E_x} & -\frac{\gamma}{E_y} & \frac{1}{E_z} & 0 & 0 & 0 \\
0 & 0 & 0 & \frac{1}{G_{xy}} & 0 & 0 \\
0 & 0 & 0 & 0 & \frac{1}{G_{xz}} & 0 \\
0 & 0 & 0 & 0 & 0 & \frac{1}{G_{yz}}
\end{bmatrix}
\]

(3)

Lira C. et. al. have calculated the mechanical properties of hexagonal honeycombs made of at least 12×12 unit cells by means of Eq. 4-12 [84].

The in-plane and out of plane mechanical properties of the hexagonal honeycomb are defined as the following equations;

\[
E_x = E_c \beta^3 \left[ \frac{\alpha \sin \theta}{\cos^3 \theta \left[ 1 + (2.4 + 1.5 \nu_c \tan^2 \theta + \frac{2\alpha}{\cos^2 \theta}) \beta^2 \right]} \right]
\]

(4)

\[
E_y = E_c \beta^3 \left[ \frac{\cos \theta}{\sin^2 \theta (\alpha + \sin \theta) \left[ 1 + (2.4 + 1.5 \nu_c + \cot^2 \theta) \beta^2 \right]} \right]
\]

(5)

\[
\gamma_{xy} = \left[ \frac{\cos^2 \theta}{\sin \theta (\alpha + \sin \theta)} \right] \left[ \frac{1 + (1.4 + 1.5 \nu_c) \beta^2}{1 + (2.4 + 1.5 \nu_c + \cot^2 \theta) \beta^2} \right]
\]

(6)
where
\[ F = 1 + 2\alpha + \beta^2 \left[ \frac{2.4 + 1.5\beta}{\alpha} \right] (2 + \alpha + \sin \theta) + \left( \frac{\alpha + \sin^2 \theta}{\alpha^2} \right) [(\alpha + \sin \theta) \tan^2 \theta \sin \theta] \]

\[ E_x = E_c \beta \left( \frac{\alpha + 2}{2(\alpha + \sin \theta) \cos \theta} \right) \]

\[ G_{yx} = G_c \beta \left( \frac{\cos \theta}{(\alpha + \sin \theta)} \right) \]

Byz modulus has a unique value [5] when honeycombs are center symmetric hexagonal, however, the value of G_{xz} takes between the upper (Voigt) and the lower (Reuss) bound. The upper (Voigt) and the lower (Reuss) bound are defined as the following equations:

\[ G_{xz}^{\text{upper}} = G_c \beta \left( \frac{\alpha + 2 \sin \theta}{2(\alpha + \sin \theta) \cos \theta} \right) \]

\[ G_{xz}^{\text{lower}} = G_c \beta \left( \frac{\alpha + \sin \theta}{(1 + 2\alpha) \cos \theta} \right) \]

It may be considered that G_{xz} depends on the width, b, of honeycomb when width ratio to edge length is between 1 to 10. G_{xz} can be obtained as a unique value [86, 87].

\[ G_{xz} = G_{xz}^{\text{lower}} + \frac{K}{y} (G_{xz}^{\text{upper}} - G_{xz}^{\text{lower}}) \]

where
\[ K = \begin{cases} 0.787 & \theta \geq 0 \\ 1.342 & \theta < 0 \end{cases} \]

4.1. MECHANICAL PROPERTIES OF ANTI-TETRACHIRAL LATTICES SANDWICH PANEL

Chen et al. [88] and Ranjbar et al. [85] have used to calculate the mechanical properties of anti-tetrachiral lattices. The anti-tetrachiral Lattices core are modeled as orthotropic material and the engineering constants \( E_x, E_y, G_{xy}, \) and \( G_{xz} \) can be determined as in the literature [5, 84]. The out-of-plane Poisson’s ratios \( \nu_{xz} \) and \( \nu_{yz} \) can be used as about zero when cellular Material Theory are considered [5]. However, the transverse Poisson’s ratios \( \nu_{zx}, \nu_{zy} \) can be used as Poisson’s ratio of the core material, \( \nu_c \) [5, 84].

The model is based on strain energy methods and depends on four non-dimensional geometrical parameters as \( \alpha_x = L_x/r, \beta_x = L_y/r, \gamma = b/r. \)

The in-plane mechanical properties can be derived as follows:

\[ \theta_{xy} = -\frac{L_x}{L_y} \]

\[ E_x = \frac{E_c \beta^3 \alpha_x}{12 \left( 1 - \frac{\beta}{2} \right)^2} \left( \frac{1}{\alpha_x - 2\sqrt{2\beta - \beta^2}} + \frac{1}{\alpha_x - 2\sqrt{2\beta - \beta^2}} \right) \]

\[ E_y = \frac{E_c \beta^3 \alpha_y}{12 \left( 1 - \frac{\beta}{2} \right)^2} \left( \frac{1}{\alpha_y - 2\sqrt{2\beta - \beta^2}} + \frac{1}{\alpha_y - 2\sqrt{2\beta - \beta^2}} \right) \]
When ligament lengths along the x and y direction are equal, on the other word, α_x equals to α_y, E_x equals to E_y. The elastic modulus along the z-direction is represented by the following equation:

\[ E_z = E_x \beta \left[ \alpha_x + \alpha_y + \pi (2 - \beta) \right] - 2[\varphi - (1 - \beta) \sin \varphi] \alpha_x \alpha_y \]  \hspace{1cm} (16)

where \( \varphi = \cos^{-1}(1 - \beta) \)

The transverse shear modulus (\( G_{xy} \)) of the anti-tetrachiral cell can be found by the following equation:

\[ G_{xy} = \frac{E_x}{2(1 + \nu_{xy})} \]  \hspace{1cm} (17)

The Eq.17 becomes undefined for \( \nu_{xy} = -1 \), therefore, for anti-tetrachiral systems \([12,89]\), the in-plane Poisson’s ratio is considered as -0.98.

The value of the transverse shear modulus of general honeycomb structures takes between the upper (Voigt) and the lower (Reuss) bound. The upper (Voigt) bounds are defined as the following:

\[ G_{xz} \leq \alpha_x + \pi \]  \hspace{1cm} (18)

\[ G_{yz} \leq \alpha_y + \pi \]  \hspace{1cm} (19)

The upper (Voigt) and the lower (Reuss) bound can be calculated by using the theorems of the minimum potential and minimum complementary energies \([89]\). When ligament lengths along the x and y direction are equal on the other word, \( \alpha_x \) equals to \( \alpha_y \), the value of \( G_{xy} \) equals to \( G_{yz} \) and the the upper (Voigt) can be calculated by the following equation:

\[ G_{xz} = G_{yz} \leq \alpha + \pi \beta G_c \]  \hspace{1cm} (20)

the lower (Reuss) bound can be calculated by the following equation:

\[ G_{xz} = G_{yz} \geq \alpha \frac{k_1}{1 + \alpha k_2} \beta G_c \]  \hspace{1cm} (21)

where \( k_1 = 0.045 \) and \( k_2 = 0.67 \) as suggested by Lorato et al. \([90]\).

The transverse shear modulus can also be calculated by the calculation:

\[ G_{xz} = G_{xz}^{lower} + \frac{K}{\gamma} (G_{xz}^{upper} - G_{xz}^{lower}) \]  \hspace{1cm} (22)

where \( K \) for the anti-tetrachiral lattice =1.57 as suggested by Lorato \([90]\).

5. APPLICATION OF AUXETIC MATERIAL

Cellular solids with honeycomb structure can be used as a sandwich core material in different engineering applications, such as automotive lightweight structures and biomedical \([11,89,91]\). Auxetic materials have been used for various engineering applications. For example, they can be used to prototype morphing wings \([92,93]\). Also, auxetic chiral structures are used in composite aero structures designs by Airoldi et al. \([94]\).

As mentioned above, auxetic composites shows higher shear resistance, that makes them one of the important material for aerospace industry. Also, the metals are quite heavy than the auxetic composites and the auxetic composites have a higher strength-to-weight ratio than metals. It is known that the impact of some objects such as birds causes tragic accidents. impact resistance needs to be increased to prevent these kinds of accidents \([95]\). The engines are mounted to wings in the commercial aircraft. Generated high noises may effect passengers. Auxetic composite sandwich structures may be used to reduce the noise. Also, auxetic materials may used for thermal protection in aerospace applications such as vanes for aircraft turbine engines \([11]\).
In the biomedical application, auxetic materials can be used to open the cavity of an artery in coronary angioplasty because of that a flexible auxetic polytetrafluoroethylene expands in lateral direction when tension force applied [11.91]. Another application of auxetic materials is smart filters. Pores size of filter made from auxetic materials can be controled. Pores size become enlarge or narrow when it is pulled or compressed. Therefore, filter can be adjusted for different range [96]. Recently, auxetic polymer matrix materials are prefered than conventional materials. As mentioned above, auxetic materials have high shear modulus relative to bulk modulus, so that, incident stresses on the polymer can be efficiently converted to lateral stresses acting on the ceramic rods and the acoustic-to-electrical energy conversion can be improved.

6. ADVANTAGES AND DISADVANTAGES

Auxetic composites have many advantages such as high specific stiffness, high specific strength and light weight. The auxetic composites have higher shear modulus, enhanced indentation resistance, synclastic curvature, better crack resistance, higher damping resistance than conventional composite materials. These advantages make auxetic composites very applicable for engineering applications, such as automotive and aerospace. Most parts of the aircraft are subjected to shear force, therefore, it is need to use the materials which have high shear modulus. This requirement makes auxetic composites very suitable for aerospace engineering

Auxetic composites have the synclastic curvature property. The synclastic curvature property gives material to have better formability, therefore, auxetic composites may be used in complex shape needs to be formed

Also, auxetic composites have a high strength-to-weight ratio and it is known that energy consumption and pollution reduce and performance is improved, when reduce the weight. Another advantage of auxetic composites is that mechanical properties can be change by changing component’s proportions. However, auxetic composites are difficult to manufacture [62].

7. VIBRATION AND ENERGY ABSORPTION

Lim [97] has investigated the elastic stability and the vibration characteristics of auxetic circular plates. He has carried out buckling and vibration analysis for various boundary condition. He has determined that the critical buckling load gradually reduces when poisson ratio’s becomes negative. Also, fundamental frequency decreases when poisson’s ratio decrease.

Shiyin et al. [98] have investigated the vibration transmission and isolation performance of the trichiral structures with uniform and gradient geometry parameters. They have been carried out FE analysis. They have been determined that the distribution of the bandgap could be controlled by means of changing the unit cell.

Lira et al. [84] have studied on potential cores for fan blade and they have aimed to optimized to reduce the dynamic response for the first three fundamental frequencies. Therefore, they used auxetic re-entrant cellular as core material

Marburg [99] has been studied on applications and methods of structural acoustic optimization to control noise. He has optimized structure versus various acousto-structural properties (such as root mean square level of the structural velocity, radiated sound power) to design passive noise control structures

Mostafa Ranjbar et. al. have studied on vibroacoustics response of sandwich panels with auxetic anti-tetrachiral and hexagonal cores for different gradient configurations. They have minimized the radiated sound power level of these auxetic sandwich panels by changing core geometries [85].

8. CONCLUSION

This paper includes types, mechanical properties, applications and advantages of auxetic material. Auxetic materials may be used in wide range of engineering applications in terms of high shear stiffness, energy absorption capability. Also, it is known that auxetic materials show synclastic curvature behavior under bending loading and this property makes auxetic materials a good candicate for complex shaped parts. Auxetic materials have orthogonal characteristics and their mechanical properties may be determined by means of homogenization approach. The mechanical properties of cellular auxetic honeycomb sandwich directly depend on geometrical parameters of unit cell of cellular solids. Therefore, their mechanical properties may be improved by using optimization methods according to the aim of their use. Recently, the studies on their energy absorption capability and vibration transmission performance have increased. Also, development in additive manufaturing technologies supports more investigation on auxetic materials.
REFERENCES

[24] Assidi M, Ganghoffer J-F. Composites with auxetic inclusions showing both an auxetic behavior and enhancement of their mechanical properties. Compos Struct 2012;94(8):2373–82
[37] Li T, Wang L. Bending behavior of sandwich composite structures with tunable 3D printed core materials. Compos Struct 2017;175:46–57
[38] Dikshit V et al. Out of plane compressive strength of 3D printed vertical pillared corrugated core structure. 2016-
[39] Turner AJ. Low-velocity impact behavior of sandwich panels with 3D printed polymer core structures. Wright State University; 2017
[64] Rana S., Fangueiro R., Advanced composites in aerospace engineering


CONTACT FORCE CALCULATION OF AN ELECTRICALLY ACTUATED MICRO CANTILEVER BEAM SWITCH

M. Cihat YILMAZ1, Sadettin ORHAN1

1Ankara Yıldırım Beyazıt University, Mechanical Engineering Department, Ankara, TURKEY
Corresponding Author: M. Cihat YILMAZ, mcyilmaz@ybu.edu.tr

ABSTRACT

Minimizing the sizes is quite important since micro structures are being used more and more with the developing science. RF MEMS switches are one of those that main duty is to transmit the signal and the most important effect on signal quality is the contact force which occurs between the contact surfaces. In order to investigate the contact force concept, a relatively smaller metal to metal contact, cantilever beam type micro switch is designed. A rectangular hole is used in design to reduce the stiffness. Because of the physical modelling of the system, a statically indeterminate problem is needed to be solved. Moment-area method is used to solve this complexity and it is demonstrated with some graphics. FEM simulation is utilized to compare the results. Moment-area method showed very good performance therefore, this study may give primitive idea to designers about the behavior of micro cantilevers under electrical actuation and help them to analyze the contact force.

Keywords: RF MEMS, contact force, cantilever beam, stiffness

1. INTRODUCTION

Nanotechnology is a very fashionable horizon line for almost all branches of science. Down-sizing the vehicles, structures, machines and devices etc. is sometimes necessity for current applications and sometimes a plunge point for future works. As well, RF MEMS (Radio Frequency Micro Electro Mechanical Systems) technology that is used in wireless communication, defense systems, test and instrumentation areas, was born as a result of this minimize movement. RF MEMS switch can be basically defined as a micro sized electro-mechanical device that allows or not to an RF signal to pass through a signal transmission line. They are used to operate at radio frequency to millimeter wave frequency. They are assigned in electronic systems like signal filters, tunable antennas and phase shifters etc.

From mechanical perspective, RF MEMS switches structures generally divides into two type; fixed-fixed beam and cantilever beam. Apart from these, special designs can be seen for RF MEMS switches like beams anchored from more than two place or radial disk design etc.

In a study about membrane switches is presented by C. Goldsmith and T. H. Lin. The authors mentioned that, micro sized cantilever beam structures were used for low frequency electrical signal switching for the first time in 1979. The handicaps of these early works are being not able to create suitable contact force and requiring much more actuation voltage. Even so, revision of dimensions and materials have been opened a road for microwave switches. Unambiguously, they studied low-loss, low-cost, electrically actuated, thin metal membrane switches and build some variety of them. They frequently compared their results with GaAs and CMOS technology. One of the sizes of their switch es are 700 x 800 µm, pull-in voltage of nearly 40 V. In addition, they revealed that the switching time is about 10 to a couple hundred micro seconds [1].

Another study of Rebeiz shows the latest level of RF MEMS switch. In this paper, a new switch version is developed to enhance stiffness. This design also let them to reduce residual stress effects. They also presented the mathematical model of the beam and demonstrated basic equations for stiffness and contact force. Contact force depends on physical properties and the space between the beam and the underlying electrode that is needed for electrostatic actuation. The beam is made of Ti/Au material. Results show that that the stress gradient causes a huge effect on contact force [2].

A different metal contact switch model that leads to variable stiffness was designed by Pisheh and Rebeiz in 2010. The difference is placing a dimple before the free end of the cantilever and make the signal pass through it with a high contact
force, when still the free end of the cantilever touches the dielectric layer. The contact force is obtained between 0.6-1.1 μN for averagely 80 V applied voltage. After the voltage applied to the beam, first the free end touches the dielectric part and then the dimple touches the contact pad. Because of this two touching period, two stiffness level occurs and the dielectric layer leads a decrease in contact resistance. Geometry of the gold switch is 7x150x170 μm and SiN is used as dielectric layer [3].

2. THEORY

Because the beam type is cantilever and there is no dielectric layer is used in this design, RF MEMS switch type is metal to metal contact switch and the free end of the cantilever touches the signal line metal. The amplitude of this touch is called contact force. This touching action mostly done with a dimple which is a bulge created on the bottom face of the free end of the cantilever.

Contact force is important for the reliability. A weak contact may not be sufficient for enough safety for signal. For better contact force, more actuation voltage is needed, even so, there is a tender balance between the applied voltage and the geometry. More voltage means bigger geometry hence slower switching. As it is seen, an optimum contact force that ensures the reliability is enough.

Just like spring constant calculations, the shape of the distributed load is needed for contact force calculations, but this time, shape of the contact situation must be used.

Obtaining the total electrostatic force $F_c$ for contact situation – i.e. hold down position - is a little different from pull_in situation because of the changing shape of the beam resulted from the decreasing gap between the actuation pad and the beam. For pull_in situation, the shape of the beam was easily considered as linear. Although the form of the beam is still linear in the first contact instant, the shape of the beam starts to change after this time. New $q$ and $q''$ in 80 V contact situation are found just like pull_in condition with different gap values.

The actuation voltage is 60% bigger than the pull_in voltage, hence bigger $F_c$ consists and leads the beam to bend more, after the free end of the cantilever contacts to contact pad (transmission line). The shape of the beam and the distributed load at contact situation are shown together in Figure 1.

Here, the distributed load can be considered as linear again but in a different way. The linearization may start and end from the actuation pad’s alignment. In detail, a short part that is closest to the free end can be considered as parallel. The gap of the point B equals to dimple high.

In order to find the contact force, moment-area method is used [4]. Since there are four unknown variables and only three equations, the system is statically indeterminate. Moment-area method reduces the unknown variables and rest of them is solved with total moment and total force equations. The shape of the distributed load under 80 V contact situation, the reaction forces and the contact force are shown in Figure 2. It is mentioned before that the new beam is in fixed-simply supported type after contact.
The distributed load is divided into three parts and moment diagrams are obtained from them. After that, a unit load that has a magnitude of 1 N is applied to the free end of the beam in opposite direction with distributed load and also its moment diagram is obtained.
The reaction force at point C that equals to contact force is obtained by summation of equivalent reaction forces due to the three divided load parts,

\[ F_c = C_1 + C_2 + C_3 \]  

\[ \delta_1 C_1 + \Delta_{1p} = 0 \]  

where \( \delta_1 \) equals to \((\bar{M} \text{ area}) \times (\bar{M} \text{ center of gravity}) / EI\), \(\bar{M}\) is the moment area of the unit load and \(\Delta_{1p}\) equals to \((\bar{M} \text{ area}) \times (\bar{M} \text{ equivalent height}) / EI\), where \( M \) is the moment area of the load [4].

\[ \delta_1 = \frac{(l + l_z)l_z}{2EI_{OA}} + \frac{l_1 l_2}{2EI_{OA}} + \frac{(l + l_z)(l + l_z)}{2EI_{AB}} \]  

is found for the first part that includes \( q_1 \).

\[ \Delta_{1p} = \frac{q_1(l - l_z)l_1 l_z}{2EI_{OA}} + \frac{(l - l_z)l_1 (l + l_z)}{2EI_{OA}} + \frac{q_1(l - l_z)}{2EI_{AB}} + \frac{q_1(l - l_z)(l - l_z)}{3EI_{AB}} \]  

Same solution is done for second part of the distributed load,

\[ \delta_2 C_2 + \Delta_{2p} = 0 \]  

\[ F_c = 34.67 \mu N \]  

There is a closed form equation for contact force calculation and it can be used to compare the results.
where $\Sigma q$ is the total distributed load under 80 V at contact situation and $\Delta g$ is the total distance that beam passes. Substituting the variables into Eq.7 and

$$F_c = 25.8 \text{ } \mu N$$

Daniel Hyman says that a contact force of a cantilevered gold metal to metal contact beam type switch must be between 50-200 $\mu N$. Even so, this result is good enough for such a small design of beam. It is important to reduce the contact resistance for a better signal transmission, hence, for low contact resistance, to increase the contact force or contact area is needed [5]. Figure 4 shows a closer drawing that describes the contact surfaces better.

**Release Force**

Release force, in other words restoring force, is the force that tries to get the beam to the equilibrium position at the beginning. Rebeiz acclaims the restoring force as the concentrated load that occurs at the free end of the cantilever. It is acceptable until the contact position because during the motion beam’s shape remains linear till the contact happens.

**3. SIMULATION RESULTS**

It is mentioned that the contact force is important for reliable signal transmission. That is why 1.6 times the pull_in voltage is used for actuation. Figure 6 shows that the contact force is determined 35.55 $\mu N$. It can be seen from simulation that, when the applied voltage decreases, contact force decreases too. It would be zero after hold-down voltage of course.
Reaction forces at the fixed end can also be obtained from same simulation. Again, only the vertical axis can be taken into consideration. The reaction force in vertical direction is determined 46.6 μN as is seen in Figure 7.

<table>
<thead>
<tr>
<th>Step</th>
<th>Actuation Electrode_trajectory</th>
<th>Fix_Fx</th>
<th>Fix_Fy</th>
<th>Fix_Fz</th>
</tr>
</thead>
<tbody>
<tr>
<td>Step_1</td>
<td>80</td>
<td>1.245299E00</td>
<td>-8.870017E-04</td>
<td>4.661531E01</td>
</tr>
<tr>
<td>Step_2</td>
<td>70</td>
<td>8.260004E-01</td>
<td>-4.713608E-04</td>
<td>3.362937E01</td>
</tr>
</tbody>
</table>

**Figure 7** Reaction forces of fixed end of the beam

**Table 1** Comparison of switch characteristics results for beam model

<table>
<thead>
<tr>
<th></th>
<th>Simulation</th>
<th>Calculation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Contact force</td>
<td>35.55 µN</td>
<td>34.7 µN</td>
</tr>
<tr>
<td>Reaction force</td>
<td>46.6 µN</td>
<td>44.2 µN</td>
</tr>
</tbody>
</table>

4. **CONCLUSION**

In this thesis study, a basic cantilever beam type switch, which is used in RF MEMS, is designed and its mechanical parameters are calculated.

Another inference is about using dimple at the end of the beam. Lower hold down voltage can be reached without dimple. But this time, a good contact force may not be derived from beam without dimple. Decision must be done according to the aim of the system.

The release force is important to get pull_in voltage and the restoring time. It is described in the literature as the ratio of concentrated load at the end to the deflection. It is useful while obtaining the pull_in voltage but to obtain release time it is not enough. To solve this problem two approaches are derived and compared with each other. If the comparison is done by looking at release time results, it is seen that the restoring force model that has two restoring force parts gives better outcome. This approach is logical because the shape of the beam is changing after contact.

For future works, new beam types can be designed using these mechanical parameters. Also, size optimization may be performed for this model to find the best or optimum pull_in voltage and contact force values.

**ACKNOWLEDGEMENT**

This work and especially FEM simulation could be done under a huge support of research group of Dr. Mehmet Unlu from TOBB University (Ankara/Turkey).

**REFERENCES**


EXPERIMENTAL INVESTIGATION OF FLOW CHARACTERISTICS BETWEEN THE PLATES WITH SQUARE CROSS-SECTIONAL RIBS VIA PARTICLE IMAGE VELOCIMETRY

Ilker Goktepeli*, Ulas Atmaca, Sercan Yagmur

Konya Technical University, Faculty of Engineering and Natural Sciences, Department of Mechanical Engineering, Konya, Turkey
*Corresponding Author: ilkergoktepeli@selcuk.edu.tr

ABSTRACT

Heat transfer is substantially affected by flow stagnation, separation and reattachment regions owing to the ribs on the plates. Placing the ribs that trigger the flow separation to disturb the thermal and hydrodynamic development lengths, augments the heat transfer inside the duct by increasing the turbulence intensity. However, the ribs also lead to the increment of the required pumping power due to the increasing pressure loss in such systems. Since the aforementioned method is used for various systems such as the heat exchangers, the cooling of electronic devices; the investigation of flow characteristics is very important to understand the heat transfer mechanism in the ducts. In the present study, the flow characteristics between the parallel plates with square ribs have been experimentally visualized. Particle Image Velocimetry system in the open water channel of Selcuk University Advanced Technology Research and Application Center has been used. The ribs having the constant dimensions of \( h' = 0.1 \) and \( w' = 0.1 \) have been symmetrically placed on the internal surfaces of the plates via various spacing values of \( 0.5 \leq S' \leq 1 \) at \( Re = 1000 \). Flow characteristics have been presented in terms of velocity vector field, streamwise velocity component and vorticity contours. The forward-facing step flow in the upstream of the first ribs, the cavity flow between the two ribs and the backward-facing step flow in the wake of the last ribs have been observed. For all cases, the distance between the successive ribs has less effect on the flow characteristics.

Keywords: Flow separation, Parallel plate, PIV, Reynolds number, Square cross-sectional rib, Turbulent flow.

NOMENCLATURE

- CMOS: Complementary Metal Oxide Semiconductor
- \( D_H \): Hydraulic diameter [m]
- \( h \): Rib height [m]
- \( h' \): Dimensionless rib height (= \( h / H \))
- \( h_w \): Water height [m]
- \( H \): Distance between the plates [m]
- \( L \): Length [m]
- PIV: Particle Image Velocimetry
- \( Re \): Reynolds number (= \( U_\infty D_H \nu^{-1} \))
- \( S \): Spacing between the ribs [m]
- \( S' \): Dimensionless spacing between the ribs (= \( S / H \))
- \( u \): Streamwise velocity component [m s\(^{-1}\)]
- \( U_\infty \): Free stream velocity [m s\(^{-1}\)]
- \( V \): Velocity vector field
- \( w \): Rib width [m]
- \( w' \): Dimensionless rib width (= \( w / H \))
- \( W \): Plate width [m]
- \( W' \): Dimensionless plate width (= \( W / H \))
- \( \nu \): Kinematic viscosity [m\(^2\) s\(^{-1}\)]
- \( \omega \): Vorticity [s\(^{-1}\)]
1. INTRODUCTION

Artificial surface elements are used for the increment of the heat transfer inside the channels. This application is seen in practice such as heat exchangers, solar collectors, cooling parts of electronic devices, internal cooling passages of gas turbines and also chemical processes. The reason is that heat transfer is significantly influenced by flow stagnation, separation and reattachment regions between the plates having ribs. Mounting these ribs on the plates in order to trigger the flow separation and increase the heat transfer surface area, is relatively efficient technique in terms of the heat transfer augmentation. As done in the present study, the ribs placed between the parallel plates cause the flow separation by disturbing the thermal and hydrodynamic development lengths and rotational flows in front of them. This type of flow is explained in the literature by the step flow which is shear flow where the flow that separates and reattaches. In general, the step flow is categorized into forward-facing step flow (Sherry et al., 2009), cavity flow (Timuralp and Altac, 2017) and backward-facing step flow (Aung, 1983). Each of these flows can be observed separately and there are also some circumstances where two or all of them can be for the considered problem. It is probable to consider both the forward-facing step flow and the backward-facing step flow at the same time in case of the flow acting on a single rib. As in this study, all of them are seen together when the flow past the ribs mounted periodically is the case.

Various rib designs have been experimentally and/or numerically applied in the previous studies. When these studies are considered, the rectangular cross-sectional ribs are more common. However, the arrangement of the ribs also influences the overall heat transfer performance depending on the conditions. For instance, the effects of the rectangular ribs on heat transfer characteristics have been studied for the staggered arrangement (Mayle, 1991; Wongcharee et al., 2011; Desrues et al., 2012; Marocco and Franco, 2017). On the other hand, the rectangular ribs have also been examined for the symmetrical arrangement (Hwang and Liou, 1995; Lopez et al., 1996; Tafti, 2005; Tokgoz et al., 2017). In some studies, staggered and symmetrical arrangements have been compared for the rectangular cross-sectional ribs (Promvonge and Thianpong, 2008; Skullong et al., 2015; Vanaki and Mohammed, 2015; Yang et al., 2017). In the literature, second common shape is defined as the triangular ribs. Different orientations and arrangements have been found in the studies related with the triangular ribs. The staggered arrangement (Kilicaslan and Sarac, 1998) and the symmetrical arrangement (Pehlivan et al., 2013) for this kind of the ribs have been encountered. Furthermore, the specific geometrical shapes for the ribs have also been exemplified as convex-concave (Yemenici and Umur, 2016), semi-circular cross-sectional (Nine et al., 2014), sinusoidal (Aslan et al., 2016), diamond-shaped (Sripattanapipat and Promvonge, 2009), trapezoidal (Ahmed et al., 2013) in the literature. However, the ribs cause increase in the pumping power in the meantime because of the ascending pressure loss in such systems. Thus, the investigation of the flow characteristics is very crucial to understand the heat transfer mechanism between the parallel plates.

In this study, the ribs with the constant dimensions of \( h' = h/H = 0.1 \) and \( w' = w/H = 0.1 \) have been symmetrically mounted on the internal surfaces of the parallel plates by considering the spacing values of \( 0.5 \leq S' = S/H \leq 1 \) at \( Re = 10000 \). As a result, the flow characteristics have been given in terms of velocity vector field, streamwise velocity component and vorticity contours via Particle Image Velocimetry (PIV) technique.

2. MATERIAL AND METHOD

All experiments have been performed in a large-scale open water channel of Selcuk University Advanced Technology Research and Application Center as shown in Figure 1. Flow characteristics have been experimentally investigated for \( Re = U_\infty \, D_H / v = 10000 \) by considering the system capacity. In the study, \( U_\infty \) is the free-stream velocity, the hydraulic diameter is expressed as \( D_H = 2H \) (Dean, 1978) depending on the distance between the plates and \( v \) is the kinematic viscosity.

![Figure 1. The experiments in the water channel](image)
The dimensions of the channel have the rectangular cross-section are 6 x 0.77 x 0.6 m for length, width, and height, respectively. The glasses with low light refraction with the thickness of 15 mm have been utilized in the water channel to make the laser beam passing easier. The channel has been filled with water up to the level of \( h_w = 0.47 \) m. Water is circulated in the channel by the centrifugal pump in the closed loop and the speed value of the centrifugal pump is controlled by using a frequency converter. The water recirculation is provided between two tanks which are large enough and the honeycomb filter is used for flow regulation. Nd:YAG laser source with a maximum value of 15 Hz has been operated to obtain the laser sheet with the thickness of 1 mm. The silver-coated hollow particles with the spherical diameter of 10 μm have been used since there is no density difference when compared with water. All images have been captured via a high-speed Complementary Metal Oxide Semiconductor (CMOS) camera with the resolution of 1632 x 1200 pixels. Using the high-speed digital camera, two digital photos in a row of the particles in the region illuminated by the laser beam, have been taken for the time interval (Δt) with the level of microsecond. Throughout the experiments, 1024 digital images have been attained after getting 1025 pairs of digital photos. In the software of the experimental setup, all images have been divided into small interrogation areas with 32 x 32 pixels. Approximately 20–30 particles have been contained in each interrogation areas to provide the high-image density criterion. The raw vector maps obtained as a result of the calculation have been provided for each time step. However, erroneous velocity vectors can occur in the flow field owing to the laser reflections observed during the image capture. In terms of the post processing, the Dantec DynamicStudio has been employed to attain the raw displacement vector fields from the experimental data by applying the appropriate filters embedded in the software.

The plates and the ribs have been made of plexiglass, a transparent material, in order to allow the laser beam to pass smoothly through the model during the experiments. The model has been positioned at the uniform flow conditions for the minimization of the effects of the free surface and the boundary layer at the channel base. In order to keep the model at the desired position in the water channel and to be exposed to uniform flow conditions, the experimental setup has been designed. The flow characteristics have not been affected by the connectors belonging to the setup. As given in Figure 2, the duct height is \( H = 0.05 \) m for the experimental system. All dimensions have been normalized with the height as done, \( L' = L/H = 20 \), for the length of the channel. Also, the width of the plates has been given as \( W' = W/H = 6 \). The thickness of each plate was 0.008 m, which is the lowest value to facilitate the laser transmission.

![Figure 2. The schematic of the model used in the experiments](image)

Six bolt-nut connections have been preferred to obtain the distance between the parallel plates. The distance between the center of each bolt-nut connections and the nearest edge was 0.03 m and determined by taking into consideration that the connection apparatuses do not affect the flow conditions. The ribs have been bonded to the plates with the silicone adhesive after the point of \( L' = 10 \) that corresponds the fully developed flow region. The dimensions are shown by \( w' = w/H = 0.1 \) and \( h' = h/H = 0.1 \) for the width and the height of the rib, respectively. Three models have been formed with respect to change in the spacing values between two successive ribs as \( 0.5 \leq S' = S/H \leq 1 \). All images have been captured for \( 9.5 \leq L' \leq 18.5 \) in two stages by scanning this interval with the camera and laser system together.

### 3. RESULTS AND DISCUSSION

Flow characteristics between the parallel plates in the presence of the ribs have been presented as a result of the experimental study via PIV method. The comparison of the square-ribbed plates has been done at \( Re = 10000 \) in terms of time-averaged results for velocity vector field \(<V>\) in Figure 3, streamwise velocity component \(<u>\) in Figure 4 and vorticity \(<\omega>\) in Figure 5. All images for \( 9.5 \leq L' \leq 18.5 \) have been given with respect to the formation below as \( S' = 0.5, 0.75 \) and 1, respectively.
Velocity vector fields \(<V>\) have been given in Figure 3. The velocity vectors are more intense in the region of decreasing cross-sectional area due to the first ribs. In this case, there are the effects of the forward-facing step flow in the upstream direction of the first rib, the cavity flow between the two ribs and the backward-facing step flow in the wake region of the last ribs. Therefore, the reverse flows have been observed in the wake region of the ribs as a result of the flow separations. The rotational flow between the first two ribs is stronger. Subsequently, this effect has been relatively reduced by the other ribs. In the formation of secondary flows between the ribs, the cavity flow is dominant. In addition, when the spacing between the ribs has been enlarged, the region of the velocity vectors localization has extended along the distance towards the exit.

Streamwise velocity components \(<u>\) have been presented for Re = 10000 in Figure 4. The values of the streamwise velocity components tend to increase by the decreasing cross-sectional area depending on the first ribs. As observed, the maximum values of the streamwise velocity components have been attained between the first two ribs. The effect of this situation has been observed by the wake region of the last ribs. With the effect of flow separation, reverse flows have
been obtained in the vicinity of the ribs. In particular, the secondary flows were stronger in the region between the first two ribs. This is explained by the forward-facing step flow in the upstream direction of the first rib, the cavity flow between the two ribs and the backward-facing step flow in the wake of the last rib. This kind of flow is relatively ineffective between the other ribs when compared to the first two ones. After that, periodical flow has been seen up to the last ribs. This situation has been observed depending on the increasing fluctuations and laser reflections, respectively. Nevertheless, nearly symmetrical flow structure has been encountered despite the aforementioned circumstances. The change in the spacing between the ribs has less effect since the distributions of the streamwise velocity components on the contour graphics are nearly similar.

Vorticity contours $\omega$ have been given for $Re = 10000$ in Figure 5. Almost symmetrical distribution has been seen in all cases. The upstream corners of the first ribs have greatly increased the flow separation. In all models where the ribs mounted, the maximum or the minimum values have been obtained at that corner of the first rib according to the position of the plate. The maximum values have been seen on the upper plate while the minimum values have been attained on the opposite plate. This situation has been explained by the direction of the vorticity rotation observed as a result of the flow separation. Although this situation is more effective between the first two ribs, the regional distribution of the vortices has shrunk between the other ribs in a row due to the periodical flow. However, the vortices occurring as a consequence of both the flow separation and the effect of the short-term periodical flow have elongated for the streamwise and shrunk for the cross-stream direction after the last rib. For all cases investigated, this effect has been seen just after the last rib for the distance of $L'$ or $1.5L'$ depending on the circumstances. On the other hand, the distance between two successive ribs has less effect on the vorticity contours as seen from the contour graphics.

4. CONCLUSIONS

The fluid flow characteristics between the horizontal parallel plates having the square cross-sectional ribs have been visualized by using PIV technique. Under the thumb of different distances between the ribs at Reynolds number of $Re = 10000$, the results obtained from the experiments can be outlined as follows:

- As expected, the flow structure has been disturbed with the ribs mounted on the plates.
- The forward-facing step flow in the upstream direction of the first ribs, the cavity flow between the two ribs and the backward-facing step flow in the wake region of the last ribs have been observed.
- The rotational flow between the first two ribs is stronger, however, its effect has been relatively reduced by the other ribs.
- In the formation of secondary flows between the ribs, the cavity flow is dominant.
- The upstream corner of the first rib has triggered the flow separation.
- In terms of vorticity contours, the maximum values have been seen on the upper plate while the minimum values have been attained on the opposite plate depending on the direction of the rotation.
- For all cases, the distance between the successive ribs has less effect on the flow characteristics.
ACKNOWLEDGEMENTS

This study has been supported by Selcuk University Academic Staff Training Program (OYP) with the project number of 2015-OYP-007. The authors are thankful for Particle Image Velocimetry setup established by Ministry of Development of the Republic of Turkey via the 2009K12180 coded project of Selcuk University Advanced Technology Research and Application Center.

REFERENCES

HEAT TRANSFER CHARACTERISTICS OF A SYNTHETIC JET ACTUATOR WITH VARIED DRIVEN SIGNAL TYPE

Okan Kocabiyik¹, Hürrem Akbıyık²*, Mehmet Seyhan³, Mustafa Sarıoğlu¹ and Yahya Erkan Akansu¹

¹Nigde Ömer Halisdemir University, Mechanical Engineering Department, Niğde, TURKEY
²Çukurova University, Mechanical Engineering Department, Adana, TURKEY
³Karedeniz Technical University, Mechanical Engineering Department, Trabzon, TURKEY
¹Corresponding Author: Hürrem Akbıyık, hakbiyik@cu.edu.tr

ABSTRACT

Since synthetic jets have significant advantages in terms of application area such as the need for an external mass transfer, integration into the field of use and fast response time, the studies on this subject have been increasing in recent years. When the studies in the literature are analyzed, the importance of zero net mass flow synthetic jet which is advantageous compared to natural convection and continuous synthetic jet is seen. In this study, an experimental investigation of the heat transfer characteristics of circular synthetic impinging jet is presented. The effect of the thermal jet on the plate was visualized by the thermal camera. The highest heat transfer distribution for all h/d distances in the impinging synthetic jet heat transfer experiments were obtained with a triangular signal type. It is observed that increase in the frequency of synthetic jet velocity increases the heat transfer on the plate. The effect of the frequency increase on the heat transfer according to the signal structure differs in the measurements taken in the 4, 10 and 15 Hz conditions of the different frequencies. As a result, the heat transfer distributions on the plate, the synthetic jet signal type and frequency have a significant effect depending on the distance between the nozzle and the plate.

Keywords: Impinging synthetic jet, actuator, signal type, velocity distribution, heat transfer.

1. INTRODUCTION

Synthetic jet actuators have many applications such as cooling turbine blades, solution to heating problems of electronic devices, and use in flow control around airfoil. Synthetic jet actuator does not need an external mass transfer, easily integrated into the system and fast response time. Therefore, studies on this actuator have been increased in recent years. Ghaffari et al. (2016) examined the effect of a synthetic jet on the heat transfer with the vertically placed heater. They stated that the maximum cooling performance obtained jet to surface gap between 5 ≤ H/Dn ≤ 10. In the study of Greco et al. (2018), They examined the effects of nozzle to plate distance and stroke length on synthetic jet cooling performance. They showed that heat transfer behavior at high stroke length is similar to the continuous jet. They stated that as the dimensionless stroke length decreased, the distribution of heat transfer decreased. Bhapkar et al. (2014) examined the effects of different jet outlet geometries such as rectangular, square and circular over the average heat transfer. They found that the maximum increase in heat transfer was obtained at the resonance frequency of the elliptical shaped orifice with an aspect ratio of 1.4 and jet-plate distance of 3. They stated that the elliptical orifice at the values of the jet-plate distance of less than 6 showed better performance than the other geometries. Jeng and Hsu (2016) examined the heat transfer by blowing jet over plate placed vertical. In their experimental study, they changed the Reynolds number, Grashof number and the variable distance. They showed that the Nusselt numbers of the various models in the specific Reynolds number increased with jet distance. Mangate and Chaudhari (2016) investigated the effects of multiple orifice structure on cooling performance. They tried for the different configurations of multiple circular holes. They found that the maximum heat transfer coefficient obtained with the multi-orifice synthetic jet was 12% higher than the conventional single-hole synthetic jet. They stated that the multi-hole synthetic jet is 10 times higher than the natural convection synthetic jet. Chaudhari et al. (2011) revealed that the jet formed with the traditional single-hole actuator was single, but multi-hole had two peaks.

The aim of this study is to investigate heat transfer characteristics of circular impinging jet driving by different signal types such as ramp, sinus, pulses and square.
2. EXPERIMENTAL SETUP

In this study, heat transfer characteristic of impinging jet is investigated for four different signal types including sinusoidal, square, ramp and pulse. Experimental setup consists of a loudspeaker, a thermal camera, a computer, a two axis traverse mechanism, homogeneous heated plate, a multimeter, a thermocouple and a power supply. As shown in figure 1, the two ends of the stainless steel thin film plate heater wrapped up in 2 copper rods. Thin film heater having 0.03mm thickness purchase from Repco Technology Inc.. The surface of the film heater is painted black mat in order to eliminate the reflection. Emissivity for upper (painted black) and lower side of heater is 0.95 and 0.27, respectively. Sorenson Amatek XG30-50 power supply is used to supply the heater. While the power supply was at a constant current of 50 A, the test was carried out at variable voltage values between 2.17-2.25 V. The synthetic jet actuator is placed the bottom surface of film heater having dimension of 375 mm x 305 mm. Environment and jet temperature was measured using the thermocouple. A Testo 885-2 thermal camera with a resolution of 320x240 pixels was used to measure the surface temperature distribution of the homogeneously heated plate.

![Figure 1. Stainless steel film heater setup](image)

3. RESULTS

Heat transfer performance of impinging jet produced with the loudspeaker synthetic jet actuator is examined by using thermal camera for different jet to plate distance (h/d) and driving frequency. In this study, heat transfer characteristic of impinging jet is investigated for four different signal types including sinusoidal, Time averaged Nusselt number distribution for different signal types, that are square, puls, ramp and sinusoidal, at H / D = 1, 2, 4, 6, 8 and 10 is given in Figure 2. Time averaged Nusselt number reached the maximum value at stagnation point located at r/d = 0 for all cases. The Nusselt number distribution increase with increasing jet to plate distance up to h/d = 6. This increase attributed to conservation of jet core region between these distances. In the puls driving signal, averaged Nusselt number increase and close the other driving signals, when jet to plate distance increase from h/d =1 to h/d =10. For all driving signal, the averaged Nusselt number distribution state generally similar trend with each other for all plate to jet distances. Ramp driving signal state the maximum Nusselt number distribution at H / D = 1, 2, 4, 6, 8 and 10. At h/d = 1 and 2, averaged Nusselt number is spread at the radial direction between r/d= -2 and r/d= +2 due to wall jet formation in these ranges.
Figure 2. Time averaged Nusselt number distribution for Square, puls, ramp and sinusoidal signal structure at $H/d = 1, 2, 4, 6, 8$ and 10

Time averaged Nusselt number distributions versus $r/d$ for square, sinusoidal, ramp and puls driving signals at $f = 4$ Hz, 10 Hz, and 15 Hz are presented in Figure 3. When the driving frequency increase from $f = 4$ Hz to 15 Hz, the averaged heat transfer distributions augment for square, sinusoidal, ramp and puls driving signals. At $f = 4$ Hz, Nusselt number distribution of the square signal is higher than that of the ramp, puls and sinusoidal signals. At $f = 10$ Hz, sinusoidal signal indicated the better heat transfer performance than the other driving signals. Driving frequency of 10 Hz showed almost similar averaged Nusselt number distribution with that of 15 Hz, while the actuator is driven by sinusoidal signal.
Figure 3. Time averaged Nusselt number distribution versus r/d for (a) square, (b) sinusoidal, (c) ramp and (d) puls signal structure at f = 4 Hz, 10 Hz, and 15 Hz

4. CONCUTION

Heat transfer distribution of impinging jet produced by synthetic jet actuator is researched for different driving signals that are square, ramp, puls and sinusoidal. Jet to plate distance varied from 1 to 10 and driving frequencies that are f = 4Hz, 10Hz and 15Hz are used as experimental parameters. Heat transfer distribution is observed with thermal camera. Impinging jet produced with ramp signal indicated best performance than the other driving signal types.

ACKNOWLEDGEMENTS

The authors would like to acknowledge the financial support of this work by the Scientific and Technological Research Council of Turkey (TUBITAK) under the Contract Number of 215M920.

REFERENCES

INVESTIGATION OF THE JET FLOW CHARACTERISTICS OF A SYNTHETIC JET ACTUATOR INTERMS OF SIGNAL TYPE

Yahya Erkan Akansu¹, Mehmet Seyhan ², Hürrem Akbıyık ³*, and Okan Kocabıyık ¹

¹Nigde Ömer Halisdemir University, Mechanical Engineering Department, Niğde, TURKEY
²Karedeniz Technical University, Mechanical Engineering Department, Trabzon, TURKEY
³Çukurova University, Mechanical Engineering Department, Adana, TURKEY
*Correponding Author: Hürrem Akbıyık, hakbıyik@cu.edu.tr

ABSTRACT

Actuators are the elements that convert electrical signals into desired physical properties. In engineering systems, fluid mechanics, aerodynamics, electronic systems etc. have many applications. The studies involving the variety of actuators, the importance of actuators and the increasing number of researches on increasing their effectiveness in the field of the application are proof of the popularity of the subject. In this study, the flow characteristics of the synthetic jet obtained in four different signal types including sine, square, ramp (triangle), pulse (25% duty cycle version of the square signal) using the speaker type actuator were investigated. In addition, changes in the instantaneous velocity distribution using the hot wire anemometer at different values of the y / D ratio were examined. As a result, when the jet structures formed by the different signal types are examined, it is understood that the sinusoidal signal is suitable when the long jet is desired to be formed and the square or pulse wave structure can be used for the high speed jet. y / D = average speed distribution for the square version signal at lower version frequencies at 1 and 5 is higher than sinus and ramp. In addition, for the same version signal, a significant change in the average velocity distribution in the range of 6 to 10 Hz was not observed.

Keywords: Synthetic jet, actuator, signal type, velocity distribution.

1. INTRODUCTION

The development of technology and the shrinkage of the devices but the increase in their functions cause the heat flux to increase. For this reason, different studies have been carried out for many years in order to improve heat transfer applications and synthetic jets have become more prominent than continuous jet and natural convection. The reason for this is the lack of a complex structure of synthetic jets, low power consumption, a compact design and high reliability. Speaker-type synthetic jet is used and is one of the most widely used actuators. They are suitable for feedback control, fast response times, safe and low power consumption, etc. as they have advantages, they can be used in many applications. These actuators are Zero Net Mass Flux (ZNMF) type actuators that give an momentum to the flow through an orifice or nozzle to the fluid and to the flow through the suction cycle.

In the literature, there are many studies in order to increase the performance of these actuators, increasing the efficiency of the system, improving the heat transfer, determining different jet characteristics, examining the geometry of the orifice / nozzle structure and the suction / blowing volume affecting the jet formation. Pavlova and Amitay (2006) studied continuous cooling of the flow surface by means of synthetic jet. In their experimental study, they made a comparison of continuous blow cooling and synthetic jet application. They examined the effects of synthetic jet excitation frequency and Reynolds number on the distance between the nozzle and surface (H / D). Synthetic jets have a better cooling effect than low excitation frequencies (f = 420 Hz) at the smallest H / D ratio (f = 1200 Hz). They also stated that jet characteristics are more effective at larger H / d ratios for the low excitation frequencies of the actuator. They also reported that the impinging synthetic jets were more effective than the continuous synthetic jets for the same Reynolds number. The experimental study of Gil and Strzelczyk (2016) investigated the momentum rate, Reynolds number, applied energy to the actuator and different gap configurations related to the actuator excitation frequency. The speaker used in the study also stated that the efficiency of the synthetic jet actuator was 5% and that the result would change when a different speaker was used. The experimental studies were carried out with the Reynolds number between 0 and 22,600 and Stokes number between 13 and 308. They found that the maximum efficiency was close to the speaker resonance frequency or the Helmholtz frequency. They showed that maximum efficiency depends on the
geometry of the actuator. Gaffari et al. (2016) examined the effect of a synthetic jet on the heat transfer affecting the vertically positioned heater. They investigated that the maximum cooling performance is taken at 5≤H/Dh≤10 blowing-surface distance that related with the vortex structure. At H/Dh=2, jet-to-surface distance is not shown full consistency in heat transfer. Greco et al. (2018) examined the combined effects of nozzle-plate distance and stroke length on multiplier synthetic jet cooling performance. In their experimental studies, the Reynolds number is constant at 5250 and the stroke length is changed (L0 / D) to 5, 10, 15, and the nozzle-plate distance (H / D) was changed between 2 and 10. They reported that the heat transfer ratio for high stroke length is similar to the continuous jet. They characterized the synthetic jet as ring shape while the nozzle-to-surface distance is between 4 to 6 and lower values at the maximum Nusselt number. For the higher values of the nozzle-to-plate distance, the synthetic jet is characterized as bellshape. They stated that the distribution of the heat transfer is decreased as the dimensionless stroke length is decreased. They also reported that the distribution of heat transfer is increased as the distance of the nozzle-to-plate distance is increased. Bhapkar et al. (2014) examined the effects of different type jet outlet geometries such as rectangular, square and circular on the change of the average heat transfer. They found that the maximum heat transfer increase was obtained at the resonance frequency of the elliptical shaped orifice with an aspect ratio of 1.4 and jet-to-plate distance of 3. They also concluded that the elliptical orifice at the values of the jet-plate distance of less than 6 showed better performance than the other geometries. However, they showed that circular and square orifice structures were more effective in heat transfer than elliptical and rectangular structures at the values of jet plate distance 6 and higher. In the study of Valiorgue et al. (2009), The jet-to-surface distance was determined as H / D = 2 and the dimensionless stroke length was determined between 1 and 22 and the Reynolds number was between 1000 and 4300. In their study, the relationship between convective heat transfer and synthetic jet flow structure is revealed and the critical stroke length is obtained as L0/H=2.5.

The aim of this study is to produce synthetic jet with the help of a loudspeaker, to determine the flow rate of the synthetic jet at different excitation frequencies and to determine the flow area characteristics of the synthetic jet. Also, the synthetic jet flow rate will be obtained and the effects of H / D ratios on synthetic jet in experimental environment will be examined.

2. EXPERIMENTAL SETUP

In this study, the flow field properties of synthetic jet obtained in four different signal types including sine, square, ramp (climbing triangle), pulses (25% duty cycle version of the square signal) using the speaker type actuator were investigated. The velocity measurements are taken at different excitation frequency between 2 and 20Hz by using a hot-wire anemometry. The experiments for the impinging synthetic jet are made at y/D = 1 and 5. The experimental setup of the synthetic jet mechanism is shown in Figure 1. The desired signal, frequency and amplitude is generated by using signal generator. Next, the produced signal is amplified in the amplifier before sending it to the loudspeaker. With the amplified signal, the speaker's diaphragm movement is provided by feeding the speaker for forming the synthetic jet actuator. The loudspeaker outlet was covered by a circular acrylic plate and the nozzle of 20mm diameter is centered to the circular acrylic plate. Then, the uniform synthetic jet flow is generated.

Figure 1. The experimental setup of the synthetic jet system
In Figure 2, shows the synthetic jet actuator that is used in generation of the synthetic jet. The Jameson brand JW-36 loudspeaker is used for generating the synthetic jet. The synthetic jet generation system consist of a 1000W loudspeaker, a volume-forming chamber with a closed volume of acrylic top cover, an acrylic tube and a 20 mm diameter nozzle. The top cover is used to make a specific volume. The plate and tube on the plate were produced using ISEL brand 3 axis CNC Router.

![Figure 2. The structure of the synthetic jet actuator](image)

The nozzle (20mm) is produced by a Zortrax brand M200 model 3D printer. As it seen in Figure 3, the model with 20mm diameter is located on the production table of the 3D printer.

![Figure 3. 20mm nozzle production in Zortrax 3D printer](image)

In this study, 5 V output voltage, 2-20 Hz frequency range and sine, square, ramp, pulse (25% duty cycle) signal types were used in order to produce the synthetic jet. As it seen in Fig. 4, the devices were used to drive synthetic jet actuator. The signal types were generated by a AA Tech brand AWG-1010 model signal generator. The 2-channel Tektronix model TDS2022 model oscilloscope was used to observe the transmitted signal. The signal from the signal generator is controlled from a channel of this oscilloscope. In the other channel, the output signal was measured by the current probe of the Fluke 80i-110s type. The Boss brand CX750 sound amplifier was used to amplify the signal. A 1200 W DC power supply was also used to drive this amplifier.

![Figure 4. Signal generator, BNC DAQ, and Oscilloscope](image)
3. RESULTS

Signal forms were used at different frequencies to obtain velocity distributions. As seen in Figure 5, the signal forms used for 10 Hz. In this study, The experimental parameters were obtained as excitation frequencies between 2 Hz and 20 Hz and y/D values between 0 and 10. The average velocity distributions in the axial direction, the average velocity distributions in the radial direction and the instantaneous velocity distributions are presented in three headings.

![Figure 5](image.png)

**Figure 5.** Variation of sinus, square, triangle and pulse signal types used in the study at apiliec voltage of 5 Vpp and excitation frequency of 10 Hz

Figure 6 shows the change in the time average of the axial velocity values along the synthetic jet axis. The high velocity values at the initial values of y / D are high due to the velocity readings of the synthetic jet in the suction mode where the proximity of the hot-wire probe to the nozzle. However, for values beyond the value of y/D = 1, only the average velocity values are read in blowing mode. It is seen that different signal types have similar distributions in radial direction. The velocity values of the synthetic jet are increasing while the excitation values are increased. The velocity values for the sinusoidal and triangular waveforms are nearly the same. However, in the square signal type, much larger average velocity value was obtained at lower frequencies than other signal types.
Secondary jet structures are formed at 2 Hz and 4 Hz, away from the main jet flow. However, in the case of 6 Hz, it is combined with the jet in the blowing mode. In 8 Hz and 10 Hz cases, these jets formed during the negative waiting period which are observed as a small turbulence on the main jet. The effect of these secondary jets on the average velocities in the square signal structure is also evident in the average velocity distributions in the radial direction taken from the y / D = 1 and 5 positions given in Figure 7. At a frequency of 2 Hz, the jet velocities of the sine and triangular waves are less than 1 m / s and around 3 m / s in the case of a square signal. While the velocities obtained at 15 and 20 Hz overlapped in the sine and triangular signal structures, in the case of a square signal there was still some increase in speed between these two frequencies.

4. CONCLUSION

In this study, the flow field characteristics of the jet flow produced using the loudspeaker type synthetic jet actuator were investigated. Square signal, sinusoidal signal, triangular signal (ramp) and square signal (pulse) types have been used as the version signal structures used to create the motion of the actuator diaphragm in the formation of the synthetic jet. In this study where the diameter of the nozzle is kept as 20 mm, frequency values of 2, 4, 6, 8, 10, 15 and 20 Hz are studied in the flow area velocity measurements depending on the actuator structure.
In the flow area, the velocity measurements were made along the axis of the outlet of the nozzle and the mean velocity distributions in radial direction were given along the axis at the stations $y / D = 1$ and $5$. Significant differences were observed in jet formation depending on signal structures. The velocity of the synthetic jet increases for the increase of the frequency in all signal types. However, the value of the velocity level varies in each signal structure depending on the change in the frequency values. In particular, when the actuator is driven by a square signal, the value of the velocity corresponding to each frequency is characteristically altered. The version signal type plays an important role in the formation of synthetic jet.

ACKNOWLEDGEMENTS

The authors would like to acknowledge the financial support of this work by the Scientific and Technological Research Council of Turkey (TUBITAK) under the Contract Number of 215M920.

REFERENCES

NiSO₄.6H₂O / CoSO₄.7H₂O ORANININ Ni-Co KAPLAMALARIN YAPISAL ÖZELLİKLERINE ETKİLERİ

Ramazan KARSLIOĞLU¹, Hatem AKBULUT²

¹Ankara Yıldırım Beyazıt Üniversitesi, Metalurji ve Malzeme Mühendisliği Bölümü, Ankara / TÜRKİYE
²Sakarya Üniversitesi, Metalurji ve Malzeme Mühendisliği Bölümü, Sakarya / TÜRKİYE
¹rkarslioglu@ybu.edu.tr, ²akbulut@sakarya.edu.tr

ÖZET
Bu çalışmada Ni-Co alışımı farklı banyo bileşimi ve elektrodepozisyon yöntemi kullanılarak Cu altlıklar üzerine kaplanmıştır. Banyo bileşiminde nikel kaynağı olarak ilave edilen NiSO₄.6H₂O ve kobalt kaynağı olarak ilave edilen CoSO₄.7H₂O banyo bileşimindeki oranlarının Ni-Co kaplamaların yapısallık etkilerine incelenmiştir. Kaplamalar 5 A/dm² akım şiddetinde ve modifiye edilmiş Watt banyosu kullanılarak üretilmiştir. Üretilen kaplamalar Taramalı elektron mikroskobu (SEM) ve X-ışınları difraksiyonu (XRD) metodu kullanılarak karakterize edilmiştir. Banyo bileşimindeki CoSO₄.7H₂O oranın artmasıyla tane yapısı polyhedrondan fleyk yapısına dönüştürmüştür. Banyo bileşiminin mekanik etkisini etkisi mikrosertlik yöntemi kullanılarak incelenmiştir. En yüksek sertlik 200/100 g/L NiSO₄.6H₂O / CoSO₄.7H₂O oranında elde edilmiştir.

Anahtar Kelimeler: Nikel, Kobalt, elektrodepozisyon

1.GİRİŞ
Nikel-kobalt alaşmları; yüksek mukavemet, iyi aşınma direnci, yüksek termal ve iletkenlik, elektro katalitik aktivite gibi özelliklerinden dolayı endüstride yaygın olarak kullanılan önemli mühendislik malzemeleridir [1]. Son yıllarda Nikel-Kobalt manyetik filmler mükemmel manyetik özelliklerinin yanı sıra adhezyon, yüksek korozyon direnci, düşük iç gerilimler, yüksek termal stabilite gibi özelliklerinden dolayı Mikro Elektro Mekanik Sistemlerde (MEMS) uygulama amaçına bağlı olarak nanometre seviyesinde milimetre seviyelerine kadar farklı kalınlıklarda kullanılmaya başlanmıştır [2].

Ni-Co filmleri üretmek için çeşitli üretim yöntemleri arasında elektrolitik kaplamanın, yüksek sıcaklık ve basınç gerektirmesi, basit ve ekonomik bir yöntem olduğu kabul edilmiştir [3-5].

Bu çalışmada son yıllarda önemli artan mikro elektromekanik sistemler (MEMS) ve nano elektromekanik sistemler (NEMS) için mücadele manyetik güçlerini alan, ağırlığının direnci gibi mekanik özellikleri ve iyi elektriksel özelliklere Ni-Co kablalar için kaplama banyo bileşimindeki NiSO₄.6H₂O /CoSO₄.7H₂O oranını mikro yapı üzerindeki etkilerinin incelenmesi ve optimize edilmesidir.

2.DENEYSEL ÇALIŞMALAR
Bakır altlıklar üzerine elektrodepozisyon yöntemi ve farklı banyo bileşimleri kullanılarak Nikel-Kobalt kaplanmıştır. Banyo bileşiminin kaplama mikro yapısı, kompozisyonu ve mekanik özelliklere etkileri incelenmiştir. Altlık (katot) malzemesi olarak 20 x 60 x 2 mm ebatlarında yüksek safı şafakta bakır levhalar kullanılmıştır. Kaplama öncesi Cu katot yüzeyleri 60, 120, 400 ve 600’lük zimpara ile zimparalanarak mekanik temizleme yapılmış yüzey yüzey pürüzlüğü düşürülmüştür. Zimparalama işlemi sonrasında bakır levhalar hacimce % 25 HCl + %75 H₂O çözeltisine daldırılmıştır. 2 sn bekletilmiş sonrasında saf su ile kıyısal temizleme işlemine tabi tutunulmuştur.

Önişlem yapılan altlık malzemeleri hazırlanan elektrolitik kaplama banyosunun içeresine uygun bir şekilde katot olarak yerleştirilmiştir. Kaplama için kullanılan banyo bileşimi ve şartları Tablo 4.1’de verilmiştir. Kaplama banyosunda
NiSO₄.6H₂O nikel kaynağı olarak CoSO₄.7H₂O ise kobalt kaynağı olarak kullanılmıştır. Borik asit (H₃BO₃) ise pH dengeleyici olarak kullanılmıştır.


Üretilen Ni-Co kaplama tabakalarının kristal yapı karakterizasyonu Rigaku marka D/MAX/2200/PC model XRD cihazı ve CuKα ışını kullanılarak 1.54059Å 30-80º 2θ aralıkları arasında yapılmıştır. Farklı banyo bileşimlerinden elde edilen kaplamaların mekanik özellikleri mekanik özelliklerine incelemek için mikrosertlik çalışmaları Leica VMHT marka cihaz ile 50 gr yük altında 15 saniye süreyle uygulanmıştır. Elde edilen izlerden cihazda bulunan yazılım kullanılarak değerlendirilmiştir. Her bir numuneden en az beş adet ölçüm alınmış ve ortalamaları grafik olarak sunulmuştur.

<table>
<thead>
<tr>
<th>Banyo Kompozisyonu</th>
<th>250/50, 50/250, 100/200 ve 200/100 g/l</th>
</tr>
</thead>
<tbody>
<tr>
<td>NiSO₄.6H₂O/CoSO₄.7H₂O</td>
<td>250/50, 50/250, 100/200 ve 200/100 g/l</td>
</tr>
<tr>
<td>H₃BO₃</td>
<td>40 g/l¹</td>
</tr>
</tbody>
</table>

3. SONUÇLAR VE TARTIŞMA

Şekil 1 Kaplama tabakasına NiSO₄.6H₂O / CoSO₄.7H₂O (g/l) oranlarının etkisi a) 250 / 50, b) 200 / 100, c) 100 /200 ve d) 5 / 25 yüksek çözünürlüklü SEM fotoğrafı

Şekil 2 Kaplama tabakasına NiSO₄.6H₂O / CoSO₄.7H₂O (g/l) oranlarının etkisi a) 250 / 50, b) 200 / 100, c) 100 /200 ve d) 5 / 25 yüksek çözünürlüklü SEM fotoğrafı.

Kaplama tabakasında Ni-Co oranının ve NiSO₄.6H₂O /CoSO₄.7H₂O değişiminin kaplama bileşimine etkileri EDS ile analiz edilmiştir. Kaplama tabakalarının EDS analiz sonuçları ve NiSO₄.6H₂O /CoSO₄.7H₂O oranının etkileri Şekil 2 de verilmektedir. Şekil 2 incelendiğinde banyo bileşimindeki CoSO₄.7H₂O oranının artış kaplama tabakasının içine giren Co oranının artmıştır. Grafik incelendiğinde yapı içerisine giren Co oranının banyo bileşimine ilave edilen CoSO₄.7H₂O çok daha yüksek oladığı görülmektedir. Dolayısı ile nikelin kobaltla göre daha soy olmasına rağmen elektrodepozisyon esnasında yapı içerisine anormal olarak daha fazla Co yerleşmektedir. Bu durumun nedeni; elektrot etrafındaki pH ve metal hidroksil değişimi ve onların adsorbsiyon rekabeti Co lehine arttırmaktadır [3].
1. International Conference on Advances in Mechanical and Mechatronics Engineering
ICAMMEN 2018

Şekil 3. Banyo bileşimindeki \( \text{CoSO}_4 \cdot 7\text{H}_2\text{O} \) miktarının büyüme yönlerine etkileri

X-ışınları ile Ni-Co kaplama tabakaları karakterize edilmişdir sonuçlar Şekil 3’te verilmiştir. XRD analizinde kullanılan X-ışının dalga boyu 1.54059 Å dur. Çözümlemede kullanılan JPDS kart no: 00-004-0850 dur. Bu sonuçlara göre Ni-Co’nun ağırlıklı olarak (111), (200) ve (220) düzlemlerinde büyüdüğü görülmüştür. XRD analizleri ayrıntılı olarak incelemişte, banyo bileşimindeki \( \text{CoSO}_4 \cdot 7\text{H}_2\text{O} \) oranın artması ile birlikte önce (220) düzleminde büyümenin azaldığı, \( \text{NiSO}_4 \cdot 6\text{H}_2\text{O} /\text{CoSO}_4 \cdot 7\text{H}_2\text{O} \) oranının 200/100 gr/L üzerinde banyo bileşiminde \( \text{CoSO}_4 \cdot 7\text{H}_2\text{O} \) oranını artması ile birlikte tekrar artan bir eğilim gösterdiği Şekil 3’ten görülmektedir. (111) ve (200) düzlemlerindeki büyümenin banyo bileşimindeki \( \text{CoSO}_4 \cdot 7\text{H}_2\text{O} \) oranını artması ile birlikte arttığı görülmektedir. Düşük \( \text{CoSO}_4 \cdot 7\text{H}_2\text{O} \) oranlarında ayrı pikler görülmemektedir bunun nedeni belli bir orana kadar Co atomlarının yüzey merkezi kübik Nikel kafesinde yer almasıdır. Ancak kaplama tabakasında Co oranını artması ile birlikte XRD analizlerinde Hegzagonal Co yapısına ait piklerin büyüdüğü ve bu büyümenin kaplama tabakasındaki Co miktarını artması ile birlikte arttığı gözlenmektedir.

Şekil 4. Banyo bileşimindeki \( \text{CoSO}_4 \cdot 7\text{H}_2\text{O} \) miktarının kaplama sertliğine etkileri

Üretilen kaplamaların sertlikleri Vickers sertlik metodu kullanılarak ölçülmüştür. Nikel-kobalt kaplamaların sonuçları Şekil 4’te grafik olarak verilmiştir. Banyo bileşimindeki \( \text{NiSO}_4 \cdot 6\text{H}_2\text{O} /\text{CoSO}_4 \cdot 7\text{H}_2\text{O} \) oranının 250/50 gr/L den 200/100 gr/L olması ile birlikte sertlik 290 HV den 316 HV çıktığı Şekil 4’ten görülmektedir. Bu değerin üzerinde banyo bileşimindeki \( \text{CoSO}_4 \cdot 7\text{H}_2\text{O} \) oranlarda, \( \text{CoSO}_4 \cdot 7\text{H}_2\text{O} \) miktarının artması ile birlikte sertlik değeri düşüş eğilimi göstermiştir. Setlik sonuçları EDS ve XRD sonuçları ile birlikte değerlendirdiğinde yüzey merkezi kübik(YMK) yapıdaki Ni kafesine Co elementinin girmesi ile sertliği arttırdığı, yapı içerisinde hegzagonal Co yapısına ait piklerin büyüdüğü ve bu büyümenin kaplama tabakasındaki Co miktarını artması ile birlikte arttığı gözlenmektedir.
4. GENEL SONUÇLAR

Farklı NiSO₄.6H₂O /CoSO₄.7H₂O oranlarına sahip banyo bileşimleri kullanılarak bakır altlıklar üzerine Ni-Co başarı bir şekilde kaplanmıştır.

Kaplama banyosundaki CoSO₄.7H₂O oranın artması ile birlikte plihedron şeklindeki tane yapısı fleky şekline dönmuştur ve kaplama tabakası içerisindeki Co miktarını artırmıştır.

XRD analizleri sonucunda, banyo bileşimindeki CoSO₄.7H₂O oranın artması ile birlikte önce (220) düzleminde Büyümeyi azaldığı, NiSO₄.6H₂O /CoSO₄.7H₂O oranının 200/100 gr/L üzerinde banyo bileşiminde CoSO₄.7H₂O oranın artması ile birlikte tekrar artan bir eğilim göstermiştir.

En yüksek sertlik 200/100 gr/L banyo bileşimindeki NiSO₄.6H₂O /CoSO₄.7H₂O oranında elde edilmiştir.

REFERENCES


NUMERICAL INVESTIGATIONS OF LAYERED AND FUNCTIONALLY GRADED PIEZOELECTRIC MATERIALS WITH CRACKS UNDER IMPACT LOADINGS

M. Wünsche¹, Ch. Zhang² and F. García-Sánchez³

¹Institute of Construction and Architecture, Slovak Academy of Sciences, 84503 Bratislava, Slovakia
²School of Science and Technology, Chair of Structural Mechanics, University of Siegen, D-57068 Siegen, Germany
³Departamento de Ingeniería Civil, de Materiales y Fabricación, Universidad de Málaga, 29071 Málaga, Spain

Corresponding Author: M. Wünsche, wuensche@bauwesen.uni-siegen.de

ABSTRACT

In this paper, the transient dynamic analysis of layered as well as functionally graded piezoelectric composites with cracks is presented. For this purpose, a time-domain boundary element method (BEM) is developed. The present BEM uses the collocation method for the spatial discretization of the time-domain boundary integral equations. The convolution quadrature method is applied to the temporal discretization. Since fundamental solutions for functionally graded piezoelectric materials are not available, a boundary-domain integral formulation is derived. The Laplace transformed fundamental solutions for homogeneous piezoelectric materials are applied. The radial integration method is used to compute the resulting domain integrals. An explicit time-stepping scheme is obtained to compute the unknown boundary data. To consider the non-linear semi-permeable electric crack-face boundary conditions an iterative solution algorithm is implemented. Numerical examples will be presented to show the influences of the functional gradation in comparison to the layered configuration and the homogeneous material, the electric crack-face boundary conditions and the transient dynamic loading on the dynamic field intensity factors.

Keywords: piezoelectric materials, functional gradation, smart laminate composites, non-linear crack-face boundary conditions, time-domain BEM, impact loading

1. INTRODUCTION

Piezoelectric materials offer many possibilities in advanced engineering structures due to their ability to convert electric energy into mechanical energy and vice versa. They are widely applied in smart devices and structures like transducers, actuators and sensors [5]. Important applications of piezoelectric materials are layered or laminated composites because they can be optimized to satisfy the high-performance requirements according to different in-service conditions. Piezoelectric composites are very brittle and have low fracture toughness. Beside cracks inside homogeneous domains, one of the most dominant failure mechanisms in layered piezoelectric composites are interface cracks. They may be induced by the mismatch of the mechanical, electrical and thermal properties of the material constituents during the manufacturing process and the in-service loading conditions. Recently, functionally graded materials are receiving increasing attention in advanced engineering applications. An important advantage over conventional laminates is that interfaces and stress discontinuities are avoided. The analysis of define layered as well as functionally graded materials is mathematically complex and analytical solutions are possible only for very simple geometry and loading conditions. Therefore, accurate and efficient numerical methods are required for the simulation and safety analysis of smart piezoelectric composites.

Although the dynamic crack problems in homogenous piezoelectric solids have been considered by several authors (e.g., [3,5,8]) the corresponding analysis of interface cracks in layered piezoelectric solids as well as of cracks in functionally graded piezoelectric solids is rather limited due to the problem complexity. This paper presents such an analysis by using a boundary element method (BEM) for crack problems in two-dimensional (2D) and linear piezoelectric solids.
2. PROBLEM FORMULATION

Let us consider a non-homogeneous and linear piezoelectric cracked solid. In the absence of body forces, free electrical charges and by applying the generalized notation, the cracked solid is prescribed by the generalized equations of motion

\[ \sigma_{ij}(x,t) = \rho(x) \delta_{jk} \ddot{u}_k(x,t), \quad \delta_{jk} = \begin{cases} \delta_{jk}, & J,K = 1,2 \\ 0, & \text{otherwise} \end{cases}, \]  

(1)

the constitutive equations

\[ \sigma_{ij}(x,t) = c_{ijkl}(x) u_{kj}(x,t), \]  

(2)

where the generalized displacements, stresses and the elasticity matrix are defined by

\[ u_i = \begin{cases} u_i, & I = 1,2 \\ \phi, & I = 4 \end{cases}, \quad \sigma_{ij} = \begin{cases} \sigma_{ij}, & J = 1,2 \\ D_{ij}, & J = 4 \end{cases}, \]  

(3)

\[ c_{ijkl}(x) = \Theta(x) c_{ijkl}^0. \]  

(4)

\( \Theta(x) \) prescribes the spatial variation of the material properties and the generalized elasticity tensor \( c_{ijkl}^0 \) for the homogeneous material is given by

\[ c_{ijkl}^0 = \begin{cases} c_{ijkl}, & J,K = 1,2 \\ c_{ijkl}, & J = 1,2, K = 4 \\ c_{ijkl}, & J = 4, K = 1,2 \\ -\kappa_{ij}, & J,K = 4 \end{cases}. \]  

(5)

Further, the initial conditions

\[ u_i(x,t = 0) = \bar{u}_i(x,t = 0) = 0, \]  

(6)

the boundary conditions

\[ t_i(x,t) = \bar{t}_i(x,t), \quad x \in \Gamma_t, \]  

(7)

\[ u_i(x,t) = \bar{u}_i(x,t), \quad x \in \Gamma_u, \]  

(8)

with \( t_i \) being the generalized traction vector defined by

\[ t_i(x,t) = \sigma_{ij}(x,t) e_j(x), \]  

(9)

and the continuity as well as the equilibrium conditions on the interface between domains with distinct material properties

\[ u_i^t(x,t) = u_i^l(x,t), \quad x \in \Gamma_{il}, \]  

(10)

\[ t_i^t(x,t) = -t_i^l(x,t), \quad x \in \Gamma_{il}, \]  

(11)

are satisfied. In Eqs. (1)-(11), \( u_i, \sigma_{ij}, \phi \) and \( D_{ij} \) represent the mechanical displacements, the stresses, the electrical potential and the electrical displacements, \( \rho, c_{ijkl}, e_{ijkl} \) and \( \kappa_{ij} \) are the mass density, the elasticity tensor, the piezoelectric tensor and the dielectric permittivity tensor, respectively. \( \Gamma_t \) and \( \Gamma_u \) are the external boundaries where the generalized tractions \( t_i \) and the generalized displacements \( u_i \) are prescribed and \( \Gamma_{il} \) is the interface between domains with distinct material properties.

On the upper and the lower crack-face \( \Gamma_{c+} \) and \( \Gamma_{c-} \) self-equilibrated generalized tractions are considered and three different electrical conditions are applied. The electrical impermeable crack-face condition

\[ D_{ij}(x \in \Gamma_{c+}, t) = D_{ij}(x \in \Gamma_{c-}, t) = 0 \]  

(12)

denotes in a physically sense that both crack-faces are free of electrical displacements. In contrast, the electrical permeable crack-face condition

\[ D_{ij}(x \in \Gamma_{c+}, t) = D_{ij}(x \in \Gamma_{c-}, t), \quad \phi(x \in \Gamma_{c+}, t) - \phi(x \in \Gamma_{c-}, t) = 0 \]  

(13)
implies identical potentials at the upper and the lower crack-face. Cracks are in generally limited electrical permeable. Therefore, a more realistic semi-permeable crack-face boundary condition has been introduced by [4]

\[ D_n(x \in \Gamma_c, t) = D_n(x \in \Gamma_c, t) = -\kappa_c \frac{\varphi(x \in \Gamma_c, t) - \varphi(x \in \Gamma_c, t)}{u_n(x \in \Gamma_c, t) - u_n(x \in \Gamma_c, t)}, \]  

where \( \kappa_c = \kappa_r \kappa_0 \) is the product of the relative permittivity of the considered crack medium \( \kappa_r \) and the permittivity of vacuum \( \kappa_0 = 8.85 \times 10^{-12} \text{C/(Vm)} \). \( D_n \) and \( u_n \) are the normal components of the electrical displacements and the crack-opening-displacements (CODs)

\[ \Delta u_l(x,t) = u_l(x \in \Gamma_c, t) - u_l(x \in \Gamma_c, t). \]  

A comma after a quantity represents spatial derivatives while a dot over the quantity denotes time differentiation. Lower case Latin indices take the values 1 and 2 (elastic), while capital Latin indices take the values 1, 2 (elastic) and 4 (electric). Unless otherwise stated, the conventional summation rule over repeated indices is implied.

### 3. BOUNDARY INTEGRAL EQUATIONS

To solve the corresponding initial boundary value problem with a time-domain boundary element method, it is formulated as boundary integral equations (BIEs). Since the fundamental solutions for functionally graded piezoelectric materials are not available a boundary-domain integral formulation is derived. By applying the procedure shown in Gao et al. [2] the BIEs have the form

\[ c_{ij}[\tilde{u}_i(x,t) = \int_G u^0_{ij}(x,y,t) + t^0_j(y,t) \tilde{u}_j(y,t) \, \text{d}y + \int_{\Omega} h^0_{ij}(x,y,t) \tilde{u}_j(y,t) \, \text{d}\Omega_y, \]  

where \( u^0_{ij}(x,y,t) \), \( t^0_j(y,t) \) and \( h^0_{ij}(x,y,t) \) are the fundamental solutions

\[ t^0_j(y,t) = c_{qK} \Theta_q(y) u_{KL}^0(x,y,t), \]  

\[ h^0_{ij}(x,y,t) = c_{qK} \Theta_q(y) u_{KL}^0(x,y,t), \]

\( \tilde{u}_j \) are the generalized normalized displacements and an asterisk "*" denotes the Riemann convolution defined by

\[ g(x,t) * h(x,t) = \int_0^t g(x,t - \tau) h(x,\tau) \, d\tau. \]

The free term \( c_{ij} \) is defined for a source point \( x \) on the smooth boundary \( \Gamma \) with \( c_{ij} = 0.5 \delta_{ij} \) and for \( x \) inside the domain \( \Omega \) with \( c_{ij} = \delta_{ij} \), with \( \delta_{ij} \) being the Kronecker delta. The Riemann convolution integral is approximated by the convolution quadrature of Lubich which requires Laplace-domain fundamental solutions. For homogeneous linear piezoelectric materials the Laplace-domain fundamental solutions are not available in explicit form [7]. In 2D they can be expressed by a line integral over the unit-circle by

\[ u^0_{ip}(x,y,p) = \frac{1}{8\pi} \sum_{\nu=0}^{m} \frac{P^m_{\nu}}{\sum_{\nu=0}^{m} \Psi^{m}_{\nu}(p \cdot n - x)} \, \text{d}n, \]

where \( n \), \( c_m \) and \( P^m_{\nu} \) and \( \Psi^{m}_{\nu}(p \cdot n - x) \) denote the wave propagation vector, the phase velocities of the elastic waves, the projector and the kernel function defined in [7]. The Laplace-domain fundamental solutions can be divided into a singular static and a regular dynamic part as

\[ u^s_{ip}(x,y,p) = u^s_{ip}(x,y) + u^d_{ip}(x,y,p). \]

It should be mentioned that for homogeneous materials the domain integral in the BIEs (16) vanishes.

### 4. NUMERICAL SOLUTION ALGORITHM

To solve the strongly singular displacement BIEs (16) the external boundary and the crack-faces are discretized by E elements

\[ \Gamma = \Gamma_b + \Gamma_c^+ + \Gamma_c^- = \sum_{e=1}^{E} \Gamma_e, \]
and the time is divided into \( K \) constant time-steps

\[
t = \sum_{k=1}^{K} k \Delta t. \tag{23}
\]

For the spatial discretization, the boundary \( \Gamma \) is divided into quadratic elements and quarter-point elements are used to describe the local square-root behavior of the crack-opening displacements at crack-tips. In the case of a functionally graded material, the BIEs (16) involve a domain integral. To avoid an additional meshing of the domain, the radial integration method [1, 2] is applied to transform the domain integrals into equivalent boundary integrals. This requires only internal nodes inside the domain. A fourth order spline-type radial basis function is used.

In order to get a solvable system of linear algebraic equations, the BIEs (16) are written for all boundary and internal nodes. After spatial and temporal discretization, the following explicit time-stepping scheme is obtained for the unknown field quantities

\[
x^K = (C^l)^{-1} \left[ D^l y^K + \sum_{k=1}^{K} (B^{k-k+1} t^k - A^{k-k+1} u^k) \right], \tag{24}
\]

where \( y^K \) is the vector of the prescribed boundary data. The vector \( x^K \) contains the unknown boundary data and in the case of a functionally graded material additionally the internal displacements.

Since special crack-tip shape functions are implemented in the present time-domain BEM to describe the local \( r^{1/2} \)-behavior of the generalized CODs at the crack-tips properly, the dynamic intensity factors are obtained in a direct and accurate manner without special techniques from the numerically computed generalized CODs at the closest nodes to the crack-tips [3] by using

\[
\begin{bmatrix}
K_{II}(t) \\
K_{I}(t) \\
K_{IV}(t)
\end{bmatrix} = \sqrt{\frac{2\pi}{l}} \mathbf{H}(x) \begin{bmatrix}
\Delta u_1(t) \\
\Delta u_2(t) \\
\Delta \phi(t)
\end{bmatrix}. \tag{25}
\]

Here, \( l \) is the distance between the closest node to the crack-tip, \( K_I \) and \( K_{II} \) are the mode-I and mode-II stress intensity factors respectively, and \( K_{IV} \) is the electrical displacement intensity factor. The matrix \( \mathbf{H} \) can be computed as shown in [5] by using the material properties at the crack-tip.

For an interface crack between two dissimilar linear piezoelectric materials, the dynamic intensity factors can be computed from the generalized crack-opening displacements by

\[
\begin{bmatrix}
K_1(t) \\
K_2(t) \\
K_4(t)
\end{bmatrix} = \sqrt{\frac{2\pi}{l}} \mathbf{B} \begin{bmatrix}
\Delta u_1(t) \\
\Delta u_2(t) \\
\Delta \phi(t)
\end{bmatrix}, \tag{26}
\]

with \( K_1 \) and \( K_2 \) being the real and the imaginary part of the complex stress intensity factor, and \( K_4 \) is the electrical displacement intensity factor and the matrix \( \mathbf{B} \) can be obtained from [6].

5. NUMERICAL RESULTS

In this section, numerical examples are presented and discussed. The loading parameter \( \chi \) is introduced to measure the intensity of the electrical loading

\[
\chi = \frac{\varepsilon_{22} D_2}{\varepsilon_{22} \sigma_0}, \tag{27}
\]

where \( \sigma_0 \) and \( D_0 \) are the mechanical and electrical loading amplitude. For convenience, the mode-I, the mode-II and the mode-IV dynamic intensity factors for crack-tips inside a homogeneous or functionally graded domain are normalized by

\[
K'_1(t) = \frac{K_{II}(t)}{K_0}, \quad K'_0(t) = \frac{K_{II}(t)}{K_0}, \quad K'_I(t) = \frac{\varepsilon_{22}}{\varepsilon_{22}} K_{IV}(t) \frac{K_{II}(t)}{K_0}. \tag{28}
\]

The real part \( K_I \) and the imaginary part \( K_2 \) of the complex dynamic stress intensity factors and the electrical displacement intensity factors \( K_4 \) for interface cracks are normalized by

\[
K'_I(t) = \frac{K_{II}(t)}{K_0}, \quad K'_2(t) = \frac{K_{II}(t)}{K_0}, \quad K'_4(t) = \frac{\varepsilon_{22}}{\varepsilon_{22}} K_{IV}(t) \frac{K_{II}(t)}{K_0}. \tag{29}
\]
with $K = \sigma \sqrt{\pi a}$ and $a$ is the half length of an internal crack.

As piezoelectric material Zirconate Titanate (PZT-5H) is chosen in all examples, which has the following material parameters

$$c_{11}^0 = 126.0 \text{ GPa}, \quad c_{12}^0 = 84.1 \text{ GPa}, \quad c_{22}^0 = 117.0 \text{ GPa}, \quad c_{33}^0 = 23.0 \text{ GPa}, \quad e_{11}^0 = -6.5 \text{ C/m}^2, \quad e_{22}^0 = 23.3 \text{ C/m}^2, \quad e_{16}^0 = 17.0 \text{ C/m}^2, \quad \kappa_{11}^0 = 15.04 \text{ C/(GVm)}, \quad \kappa_{22}^0 = 13.0 \text{ C/(GVm)}$$

and the mass density $\rho = 7500 \text{ kg/m}^3$.

5.1. A rectangular plate with a central crack and functionally gradation in the $x_2$-direction

In the first example, as shown in Fig. 1, a central crack of length $2a$ in a functionally graded rectangular piezoelectric plate is considered. The geometry of the cracked plate is determined by $h=20.0\text{mm}, w=10.0\text{mm}$ and $2a=4.8\text{mm}$.

![Figure 1](image)

**Figure 1.** A rectangular plate with a central crack and functionally gradation in the $x_2$-direction

On the left and the right boundary a pure impact tensile loading $\sigma(t)=\sigma_0 H(t)$ is applied, where $H(t)$ denotes the Heaviside step function. The spatial discretization of the boundary is performed by an element-length of $2.0\text{mm}$. Each crack-face is approximated by 6 elements and a normalized time-step of $c_L\Delta t/h=0.05$ is chosen, where $c_L$ is the longitudinal wave velocity. The gradation of the material in the $x_1$-direction is defined by the exponential law

$$c_{ij} = c_{ij}^0 e^{\beta x_1}, \quad \beta = \frac{1}{2h} \ln(\alpha), \quad \alpha = \frac{c_{ij}(x_2=2h)}{c_{ij}(x_2=0)}.$$  \hspace{1cm} (31)

The numerical results of the developed time-domain BEM obtained for an impermeable crack and different material gradations $\alpha$ are shown in Fig. 2.

![Figure 2](image)

**Figure 2.** Normalized dynamic intensity factors for different gradations
Fig. 2 indicates a significant influence of the functionally gradation on the normalized dynamic mode-I and mode-IV intensity factors. It can be seen that the normalized dynamic intensity factors are zero until the elastic waves reach the crack. Immediately after the waves reach the crack-faces, the normalized dynamic intensity factors increase rapidly with the time. After reaching a peak value they decrease. The mode-II intensity factors vanish for the applied tensile loading normal to the crack-faces and the transversally isotropic material behavior. The electrical displacement intensity factors of the functionally graded material are higher than for the homogeneous plate of PZT-5H.

5.2. A rectangular layered plate with a central interface crack

In the next example, as shown in Fig. 3, we consider a central interface crack of length $2a$ in a rectangular layered piezoelectric plate with the geometrical dimensions $h=20.0\text{ mm}$, $w=10.0\text{ mm}$ and $2a=4.8\text{ mm}$.

![Figure 3. An interface crack in a layered rectangular piezoelectric plate](image)

The layered piezoelectric plate is subjected to a combined tensile impact loading of the form $\sigma(t)=\sigma_0H(t)$ and an electrical impact loading $D(t)=D_0H(t)$. The spatial and temporal discretization are chosen as in the first example. Barium Titanate (BaTiO$_3$) [8] is used for the domain II. The computed results are presented and compared in Fig. 4.

![Figure 4. Normalized dynamic intensity factors for the interface crack](image)

According to the quasi-static assumptions of the electrical field, which implies that the cracked plate is immediately subjected to an electric impact, the dynamic intensity factors start from a non-zero value. The elastic waves need some time to reach the crack-faces. After that the dynamic intensity factors increase rapidly until their maximum values. The real part and the imaginary part of the normalized complex intensity factors for the interface crack show a significant influence of the electrical loading. The peak values of the normalized dynamic intensity factors decrease with increasing electrical loading. Since the crack opening modes I and II are coupled each other for the interface crack the imaginary
part of the complex intensity factor is unequal zero. As expected the electrical displacement intensity factors show a
strong dependence on the applied electrical loading.

5.3. A rectangular plate with a central crack and functionally gradation in the $x_1$-direction

As last example let us consider a rectangular plate with a central crack of length $2a$ subjected to a combined impact
tensile loading $\sigma(t)=\sigma_0 H(t)$ and impact electrical loading $D(t)=D_0 H(t)$ on the left and the right boundary, as shown in the
Fig. 5.

![Figure 5. A rectangular plate with a central crack and functionally gradation in the $x_2$-direction](image)

The geometrical data are $h=20.0\text{mm}$, $w=10.0\text{mm}$ and $2a=4.8\text{mm}$. The gradation of the material in the $x_1$-direction is
prescribed by

$$c_u = c_u^0 e^{\beta t}, \quad \beta = \frac{1}{2w} \ln(\alpha), \quad \alpha = \frac{c_u(x_1 = 2w)}{c_u(x_1 = 0)}.$$  \hspace{1cm} (32)

The spatial and temporal discretization are identically to the first example. The normalized dynamic intensity factors of the
impermeable (ip.), permeable (p.) and semi-permeable (sp.) crack-face boundary conditions are shown in Fig. 6. For the
computations using the non-linear semi-permeable crack-face boundary condition vacuum is assumed inside the
internal crack. Only the results of the Tip A are shown for the sake of brevity.

![Figure 6. Normalized dynamic intensity factors of Tip A for the electromechanical loading $\chi=0.5$ and different crack-face boundary conditions](image)

It can be seen, that the electric permittivity has a strong influence on the dynamic mode-IV intensity factors. For the
impermeable crack-face boundary condition the mode-IV intensity factor depends only weakly on the time. In contrast,
by applying the permeable crack-face boundary condition the crack does not exist for the electric field. As a
consequence, the curve of the mode-IV intensity factor has a similar behavior as that for the mode-I. It should be
mentioned that the impermeable crack-face boundary condition leads to the strongest possible electrical crack-tip field in the case of a static loading. As observed from Fig. 6 an opposite tendency can be obtained by a dynamic loading. In generally, the results of the semi-permeable crack are between the bounds of the impermeable and the permeable crack-face boundary conditions.

ACKNOWLEDGEMENTS

This work is supported by the Slovak Academy of Sciences Project 0106/01/01. The financial support is gratefully acknowledged.

REFERENCES

INVESTIGATION OF HALF NACA 0018 AIRFOIL ON TRUCK-TRAILER

Mustafa Sarıoğlu¹, Mehmet Seyhan¹,* and Yahya Erkan Akansu²

¹Kadıneğiz Technical University, Mechanical Engineering Department, Trabzon, TURKEY
²Nigde Ömer Halisdemir University, Mechanical Engineering Department, Nigde, TURKEY
*Corresponding Author: Mehmet SEYHAN, mehmetseyhan@ktu.edu.tr

ABSTRACT

An experimental study is performed to investigate the effects of a roof fairing and a half airfoil are used as passive flow control methods in order to reduce the drag coefficient acting on a truck-trailer. Aerodynamic characteristics of the half of NACA 0018 airfoil and roof fairing having roof fairing positions of -4.5mm, -2.5mm, 0 mm, +2.5mm and +4.5mm are evaluated by performing the force measurements, surface oil flow visualization and smoke wire flow visualization. Force measurement experiments were executed in the range 6.6x10⁴ ≤ Re ≤ 4.9x10⁵. Optimum combination for maximum drag reduction is found to be half of NACA 0018 airfoil at zero roof fairing position. Oil flow visualization results elucidated that separated shear layer around the surface of the half airfoil for the case of truck-trailer with the half airfoil and around the surface of the roof fairing for the case of truck-trailer with the half airfoil and roof fairing.

Keywords: NACA 0018 airfoil, drag coefficient, truck-trailer.

1. INTRODUCTION

It is possible to use active and passive flow control methods to solve flow induced problems such as flow separation, vortex shedding, vibration and so forth. Examples of active flow control methods include blowing and suction (Fransson et al., 2004), DBD plasma actuators (Feng et al., 2012; Little et al., 2010) and synthetic jets (Belinger et al., 2014; Lin and Hsiao, 2012; Rylatt and O’Donovan, 2013). Examples of passive flow control methods include the control rod (Firat et al., 2015; Sarioglu et al., 2005), the splitter plate (Sarioglu, 2016; Shukla et al., 2013) and the deflector (Ozono, 2003; Raina et al., 2017). It is important to solve or improve the flow induced problems in road vehicles in terms of fuel economy and passenger comfort. The majority of fuel consumption in road vehicles is spent to overcome the drag force due to the aerodynamic structure of the vehicle. For this reason, nowadays aerodynamic design of road vehicles such as cars and trucks is designed by taking into account the aerodynamic features. In the literature, studies related to reducing drag force acting on road vehicles are summarized below in recent years.

Researchers try to reduce drag force with the help of a roof fairing (Kim et al., 2017c, 2017b, 2017a), a front spoiler (Hyams et al., 2011), boat tail (Altaf et al., 2014; Khalighi et al., 2001) and gap fairing (Khosravi et al., 2015; Salati et al., 2017; Kim et al., 2017a). Within these drag reduction devices, roof fairing is most known and used as a drag reduction device. Khosravi et al. (2015) investigated the effects of addition devices such as deflector and cab vane corner on drift reduction in heavy commercial vehicles with numerical method. It was revealed that the cab vane corner reduced the drag coefficient by up to 20% using the appropriate deflector angle. Kim et al. (2017c) used roof fairing in order to control the flow around the truck trailer. This device provides drag reduction up to 19%. Unlike the study of Kim et al. (2017c), Kim et al. (2017a) developed bio-inspired roof fairing device to obtain drag reduction. Flow structure around the heavy vehicle with bio-inspired roof fairing is investigated by using particle image velocimetry. They obtained drag reduction up to 22.4% via bio-inspired roof fairing. In the study of Salati et al. (2017), they numerically and experimentally examined the effects of front and rear trailer device for 1/10 scale truck model on the wind tunnel. These devices are provided a maximum of 9.5% reduction in drag coefficient. Landman et al. (2010) experimentally investigated seven different drag reduction devices on the heavy truck in order to understand the practical limits on the heavy modern truck-trailer. Drag reduction devices include side skirt with varied length, gap seal and tapered rear panel. The results showed that the wind speed of 65 m/s for the modern conventional cabin tractor can achieve a maximum drag reduction of 31%.

The main aim of the study is to investigate the effect of half NACA 0018 airfoil over aerodynamic structure in terms of drag reduction. Optimum position of roof fairing is researched with the combination of the half airfoil. Experimental studies such as force measurement, smoke wire flow visualization and surface oil flow visualization are carried out to evaluate the drag reduction performance of these devices.
2. EXPERIMENTAL SETUP

An Open type wind tunnel having a cross section of 570mm x 570mm and a length of 1000 mm is used for smoke flow visualization and oil flow visualization in Mechanical engineering department of Karadeniz Technical University. The desired velocity varied from 0 to 50 m/s is ensured by frequency inverter in the test section. Another wind tunnel having a test section of 570 mm x 570 is used for force measurements located in Mechanical engineering department of Niğde Ömer Halisdemir University. For force measurement setup, detail information could be found in the study of Akansu et al. (2016). Force measurement experiments were performed at Re = 6.6x10⁴, 1.2x10⁵, 1.6x10⁵, 2x10⁵, 2.4x10⁵, 2.9x10⁵, 3.2x10⁵, 3.6x10⁵, 4x10⁵, 4.5x10⁵, and 4.9x10⁵. Roof fairing and half NACA 0018 airfoil are produced using a 3D printer. Different roof fairing positions that are -4.5, -2.5, 0, +2.5 and +4.5 used with half NACA 0018 airfoil. Zero roof fairing position is indicated the same level of the edge of the trailer and the edge of the spoiler. Figure 1(a) and (b) show wind tunnels for flow visualization and force measurement, respectively.

![Wind tunnel for (a) flow visualization and (b) force measurements](image)

In the surface oil flow visualization method, surface of the half airfoil and roof fairing is painted mat black to obtain better results. This method consists of titanium dioxide, oleic acid and kerosene and provides better insights into flow separation points. Surface oil flow and smoke wire flow visualization are carried out at Re = 4.5x10⁵ and Re = 1.6x10⁵, respectively.

3. RESULT

In M_0_NACA0018, “M”, “0” and “NACA0018” denote alone truck trailer, roof fairing position and half airfoil, respectively. Effects of the half airfoil with roof fairing placed on truck-trailer model over drag coefficient are given in Figure 2. Drag coefficient for roof fairing of +4.5 position is higher than drag force for the base model while the Reynolds number is in the range of 6.6x10⁴ to 2x10⁵. However, more detailed measurements are needed to demonstrate the physical mechanism behind drag reduction from 1.12 to 0.52 for Reynolds number from 6.6x10⁴ to 4.9x10⁵. Drag
coefficient of \( M_{\text{NACA0018}} \) is 9% lower than that of alone truck-trailer model. While the roof fairing was in the -2.5 position, the drag coefficient was reduced by an average of 19.7% compared to the simple model. The drag coefficient for \( M_{+2.5}\text{NACA0018} \) slightly decreased up to \( Re = 3.2\times10^5 \) and reached the value of \( M_{R0}\text{NACA0018} \). The maximum drag reduction was 27.5% in the case of \( M_{0}\text{NACA0018} \) as compared to simple model.

![Graph showing drag coefficient versus Reynolds number](image)

**Figure 2.** Drag coefficient versus Reynolds number for truck-trailer with/without NACA 0018 and different roof fairing position.

The alone truck trailer, roof fairing at zero position, half NACA 0018 airfoil at zero position of the roof fairing and the half airfoil for the smoke wire flow visualization are shown in Figure 3 (a), (b), (c) and (d), respectively. Flow separation, indicated by blue arrow, around the truck trailer with roof fairing at zero position is shown in Figure 3 (c). The flow follows the surface of roof fairing and the half airfoil, consecutively. This event brings about smaller wake region and also smallest drag coefficient seen in Figure 2.

![Images of truck trailer configurations](images)

**Figure 3.** Smoke wire flow visualization for (a) the alone truck trailer, (b) roof fairing at zero position, (c) the half airfoil with roof fairing at zero position and (d) the half airfoil at \( Re= 1.6\times10^5 \).
In Figure 3 (a) and (d), it is clearly seen that separated flow from the edge of the truck reattached at the edges of the trailer. Reattachment point is indicated with yellow arrow. Drag coefficient of Figure 3 (d) is 9% less than that of Fig. 3 (a) due to narrow wake length. Figure 4(a) indicates the flow separation curve with the yellow curve due to accumulation of oil pigments. In this point, flow is forced to separate owing to growing of adverse pressure gradient. Flow reattachment is swept the oil pigments on the surface of the half airfoil. Figure 4(b) showed the flow reattachment and again separation. Incoming flow reattach at the edge of trailer and start to separate after yellow line boundary.

![Image](a) M_R0_NACA0018  
(b) M_NACA0018  

Figure 4. Oil flow visualization for (a) half airfoil without spoiler and (b) half airfoil with spoiler at zero position

4. CONCLUSION

The effects of half NACA 0018 airfoil and roof fairing at different positons is experementally examined. Roof fairing positions are -4.5, -2.5, 0, +2.5 and +4.5mm. Force measurements at $6.6\times10^4 \leq \text{Re} \leq 4.9\times10^5$, smoke wire flow visualization at $\text{Re} = 1.6\times10^5$, and surface oil flow visualization at $\text{Re} = 4.5\times10^5$ are carried out in the wind tunnel. Maximum drag reduction is obtained as 27.5% for the case of M_R0_NACA0018. Flow visualization results supported the maximum reduction case indicating a narrower the wake and better aerodynamic structure.

ACKNOWLEDGEMENTS

The authors would like to acknowledge the financial support of this work by the Scientific Research Projects Unit of Karadeniz Technical University with the contract number of 2007.112.003.2.

REFERENCES

ACTUATOR SATURATED STATE FEEDBACK H∞ CONTROL OF CREEP TORQUE OF A DUAL CLUTCH TRANSMISSION

Kaan Ozgokhan, Hakan Yazici

1AVL Research and Engineering Turkey, Sancaktepe, İSTANBUL
2Yildiz Technical University, Faculty of Mechanical Engineering, Beşiktaş, İSTANBUL
akaan.ozgokhan@avl.com, hyazici@yildiz.edu.tr

ABSTRACT
This paper proposes an actuator saturated H∞ state feedback controller for creep torque control of a dual clutch transmission. Creep is a functionality which highly affects the driving comfort, especially in conditions such as traffic jams where the driver has to perform stop and go maneuvers frequently. This comfort necessity which is desired by automotive industry motivates the engineers to research high-performance and reliable control techniques. In this work, the strongly mathematical grounded state feedback H∞ is designed for creep torque control of dual clutch transmission clutches. In the design, actuator saturation dynamics are also included in the controller dynamics to obtain a practically applicable controller. Proposed controller is applied on a powertrain model and results are presented. The results show that the designed controller yields effective results while guaranteeing the stability of the closed loop system.

Keywords: dual clutch transmission, creep control, clutch torque control, H∞ control, saturated H∞, powertrain control, transmission control

1. INTRODUCTION
Creep is a functional state where vehicle moves without the activation of gas and brake pedals. In creep state, vehicle moves with a constant velocity while the engine speed is kept at the idle speed. Since the creep quality directly affects the driving comfort, it is very important to perform a smooth vehicle motion during creep. This comfort necessity is desired by the automotive industry motivates the engineers to research high-performance controllers.

Dual clutch transmission (DCT) is a type of automatic transmission which includes two shafts, one is mounted inside of an outer hollow shaft. One clutch is attached to each shaft, so that two different power paths can be possessed in a compact volume. In DCTs, odd numbered gears and even numbered gears are separated to the shafts, e.g. shaft 1 runs with 1st, 3rd, 5th gears and shaft 2 runs with 2nd, 4th, 6th gears. During normal drive, one gear is engaged to the each of the shafts and this makes possible to perform shifts without torque interruption by reducing the driving clutch pressure and increasing the desired clutch pressure during torque handover. Dual clutch transmissions makes possible to combine efficiency, sportiness and comfort. (Fischer et. al., 2015)

During creep, the objective is to keep the vehicle speed at a constant speed and the engine controller preserves a constant rotational speed. The speed reference is tracked by transmitting the correct amount of torque from engine to the wheels through clutches. This paper focuses the control of clutch torques during creep state.

Although there are many patents that is written regarding clutch control, there are not many research papers in the literature. Zhou et al., in their work, proposed a nonlinear robust controller to control vehicle creep for dual clutch transmission. (Zhou et al., 2017) Liu et al., focused on dynamic modelling and analysis of DCTs during launch and shifts. They also tested the proposed model on a test vehicle. (Fischer et al., 2009) Kulkarni et al., developed a model and a control logic for shift dynamics for a DCT. (Kulkarni et. al., 2007) Hu et. al., designed a model based gearshift controller to improve the quality of shift of dual clutch transmissions and presented the performance of the controller with hardware in the loop tests. (Hu et. al., 2014)

This paper is organized as follows: Section 2 presents the mathematical powertrain model for simulation purposes and
proposes a control model for the controller design. In Section 3 actuator saturated state feedback $H_\infty$ controller is derived and in Section 4, obtained controller is simulated in the powertrain model and control performance is discussed. In Section 5, conclusions are made.

2. MATHEMATICAL MODEL

2.1 Dual Clutch Transmission Model

For the validation and simulation purposes, a mathematical powertrain model depicted in Figure 1 consisting dual clutch transmission is employed which is proposed by (Zhou et. al., 2017).

The dual clutch transmission has two clutches each is mounted on a shaft. During drive, two gears are engaged at the same time, one is the drive gear on the driving shaft and the other is desired gear which is on the desired shaft. Therefore, it is possible to perform a gearshift without any torque interruption thanks to two different power paths that DCT employs. The odd gears (1st, 3rd, 5th and Reverse gears) run with clutch 1 and the even gears (2nd, 4th and 6th gears) run with the clutch 2.

To reduce the complexity, some simplifications are made in the modelling given as follows (Zhou et. al., 2017):

1. Drive shaft and wheel properties are exactly the same for the left and right sides.
2. Wheel slip and vehicle cornering are ignored and the torque is distributed equally to the wheels by differential.
3. The input shafts are exposed to explicit rotational motion and output shaft compliances are reserved.

With these assumptions and the powertrain shown in Figure 1, equations of motion of the powertrain are derived as follows (Zhou et. al., 2017),

\begin{align}
J_e \ddot{\omega}_e &= T_e - T_d \\
J_d \ddot{\omega}_d &= T_d - T_{c1} - T_{c2} \\
J_{c1} \ddot{\omega}_{c1} &= T_{c1} - \frac{\tau_{c1}}{i_{g1}} \\
J_{c2} \ddot{\omega}_{c2} &= T_{c2} - \frac{\tau_{c2}}{i_{g2}} \\
J_o \ddot{\omega}_o &= i_{f1} T_{t1} + i_{f2} T_{t2} - T_o \\
J_v \ddot{\omega}_v &= T_o - T_v \\
T_d &= k_d(\theta_e - \theta_d) + c_d(\omega_e - \omega_d) \\
T_{t1} &= k_{t1} \left( \frac{\theta_{c1}}{i_{g1}} - i_{f1} \theta_o \right) + c_{t1} \left( \frac{\omega_{c1}}{i_{g1}} - i_{f1} \omega_o \right)
\end{align}
\[ T_{t2} = k_{t2} \left( \frac{\theta_{t2}}{i_{t2}} - i_{t2} \dot{\theta}_o \right) + c_{t2} \left( \frac{\omega_{t2}}{i_{t2}} - i_{t2} \omega_o \right) \]  
(9)

\[ T_o = k_o (\theta_o - \theta_e) + c_o (\omega_o - \omega_v) \]  
(10)

where the rotational velocity is stated with \( \omega \), \( \theta \) is the rotational displacement, \( f \) is the inertia, \( i \) is the gear ratio, \( c \) is the damping coefficient, \( k \) is the coefficient of stiffness and \( T \) represents torque. \( e \) subscript means engine, \( d \) represents the clutch drum, \( c1 \) and \( c2 \) represent clutch 1 and clutch 2, \( t1 \) and \( t2 \) are input shafts, \( o \) stands for the drive shaft, \( v \) is for vehicle, \( g1 \) and \( g2 \) are the gears engaged to shaft 1 and shaft 2, \( f1 \) and \( f2 \) represent the final gears.

Resistance torque on the vehicle is given as,

\[ T_v = (mgsina + fmgcosα + 0.5C_aρu^2)R_w \]  
(11)

Here, \( m \) is the vehicle mass, \( g \) is the gravity, \( α \) is road gradient, \( f \) is the tyre rolling coefficient, \( A \) is frontal area of the vehicle, \( C_a \) is the aerodynamic resistance coefficient, \( ρ \) is the air density, \( u \) is vehicle velocity and \( R_w \) is the wheel radius.

The powertrain and DCT model (Zhou et. al., 2017) parameters are used as follows: the inertias \( J_e = 0.2 \text{ kgm}^2 \), \( J_d = 0.086 \text{ kgm}^2 \), \( J_{c1} = 0.043 \text{ kgm}^2 \), \( J_{c2} = 0.047 \text{ kgm}^2 \), \( J_o = 0.04 \text{ kgm}^2 \), \( J_p = 149.9363 \text{ kgm}^2 \); the stiffness coefficients \( k_d = 1000 \text{ Nm/rad} \), \( k_{c1} = 314200 \text{ Nm/rad} \), \( k_{c2} = 301200 \text{ Nm/rad} \), \( k_o = 10000 \text{ Nm/rad} \); the damping ratios \( c_d = 10 \text{ Nms/rad} \), \( c_{c1} = 50 \text{ Nms/rad} \), \( c_{c2} = 50 \text{ Nms/rad} \), \( c_o = 100 \text{ Nms/rad} \); mass of the vehicle \( m = 1530 \) kg; the rolling coefficient \( f = 0.02 \); aerodynamic resistance coefficient \( C_a = 0.3 \); frontal area of the vehicle \( A = 2.33 \text{ m}^2 \); rolling radius \( R_w = 0.312 \) m; gear ratios of 1/3/5th gears 3.69/1.47/0.92 and gear ratios of 2/4/6th gears 2.24/1.06/0.74, respectively.

### 2.2. Control Model

In the creep state, vehicle speed is maintained at a predefined constant speed and the transmission input shaft has a constant rotation speed. Hence, the engine is responsible for controlling the input shaft speed and the transmission control unit is responsible for controlling the vehicle speed.

Transmission control unit calculates the torque amount which is need to be transmitted through each clutch and then the required clutch pressure and the hydraulic valve currents are calculated to have desired pressures in the clutches. However, in this work, only the desired clutch torque is calculated and it is assumed that the desired clutch torque is realized by transmission control software and hardware at the same instant when it is requested.

In DCT applications it is possible to measure the input shaft speed, output shaft speed, shaft speeds and clutch pressures with the help of hardwired sensors. Therefore states have to be chosen among those to have a applicable controller design and here the states are selected as \( x_1 = \omega_e - \omega_o \) and \( x_2 = T_{c1} \) whilst the control input is \( u = T_{c1} \).

During creep, since the engine speed is constant at idling speed, engine torque is equal to the clutch torque as it can be seen equation (1).

\[ J_{en} \dot{\omega}_e = T_e - T_{c1} = 0 \]  
(12)

\[ J_{ec} \dot{\omega}_{c1} = T_{c1} - \frac{T_o}{i_{g1}i_{f1}} \]  
(13)

Assuming the shafts are rigid, following relation between clutch acceleration and output shaft acceleration,

\[ \dot{\omega}_{c1} = i_{g1}i_{f1} \dot{\omega}_o \]  
(14)

Substituting the equation (14) into equation (13), one can get,

\[ J_{ec} \ddot{\omega}_o = \frac{T_{c1}}{i_{g1}i_{f1}} - \frac{T_o}{(i_{g1}i_{f1})^2} \]  
(15)

Hence, the state equations are,

\[ \dot{x}_1 = \omega_e - \omega_o = -\dot{\omega}_o = \frac{T_p}{(i_{g1}i_{f1})^2} - \frac{T_{c1}}{i_{g1}i_{f1}i_{ec}} \]  
(16)
\[ \dot{x}_2 = u. \]  

Considering the state space representation of the system as,

\[ \dot{x} = Ax + B_1 u(t) + B_2 \omega(t) \tag{18} \]

where \( x \) is the state vector, \( u \) is the control input, \( \omega \) is the disturbance input, \( A \) is the system matrix, \( B_1 \) is the control matrix and \( B_2 \) is the disturbance matrix.

Considering \( T_\omega \) as the disturbance, one can obtain,

\[ \begin{bmatrix} \omega_e - \omega_o \\ \dot{T}_{c1} \end{bmatrix} = \begin{bmatrix} 0 & -1 \\ 0 & 0 \end{bmatrix} \begin{bmatrix} \omega_e - \omega_o \\ \omega_e \end{bmatrix} + \begin{bmatrix} 0 \\ 1 \end{bmatrix} \dot{T}_{c1} + \begin{bmatrix} 1 \\ \frac{(ig_1 f_1)^2}{j}ce \end{bmatrix} T_\omega \]  

\[ A = \begin{bmatrix} 0 & -1 \\ 0 & 0 \end{bmatrix}, \quad B_1 = \begin{bmatrix} 0 \\ 1 \end{bmatrix}, \quad B_2 = \begin{bmatrix} \frac{(ig_1 f_1)^2}{j}ce \\ 0 \end{bmatrix} \tag{19} \]

### 3. CONTROLLER DESIGN

The state space representation of the system is given by,

\[ \dot{x} = Ax + B_1 u(t) + B_2 \omega(t) \tag{21} \]

\[ z(t) = Cx(t) \tag{22} \]

Where \( x(t) \) is the state vector, \( u(t) \) is the control input vector, \( \omega(t) \) is the disturbance input, \( z(t) \) is the controlled output of the system.

Selecting a state feedback control law given by,

\[ u(t) = Kx(t). \tag{23} \]

And, substituting the control law into the closed loop system (18) one can obtain,

\[ \dot{x} = (A + B_1 K)x(t) + B_2 \omega(t) \tag{24} \]

\[ z(t) = Cx(t) \tag{25} \]

\( H_\infty \) norm of a system is defined as,

\[ \|G\|_\infty = \sup_{\|w\|_2} \frac{\|z\|_2}{\|w\|_2} \tag{26} \]

Supposing \( T_{zo} \) is the transfer function of the system from disturbance to its output, the objective of the state feedback \( H_\infty \) controller is to minimize \( \gamma \) such that,

\[ \|T_{zo}\|_\infty < \gamma \tag{27} \]

Hence, one can obtain the \( H_\infty \) performance criteria as,

\[ z^T z - \gamma^2 \omega^T \omega < 0 \tag{28} \]

There exist a quadratic Lyapunov function \( V = x(t)^T P x(t) \) such that \( P > 0 \) and its derivative is written as,

\[ \dot{V}(x(t)) = \dot{x}(t)^T P x(t) + x(t)^T P \dot{x}(t) \tag{29} \]

Hence one can write,

\[ \dot{V}(x(t)) + z^T z - \gamma \omega^T \omega < 0 \tag{30} \]

Let us substitute (24), (25) and (29) into (30),
\[(A + B_1K)x(t) + B_2\omega(t)\] 
\[Px + x^TP[(A + B_1K)x(t) + B_2\omega(t)] + x(t)^TC^T\] 
\[-\omega^T\omega < 0 \quad (31)\]

Arranging the inequality (21) and taking the parenthesis of \([x(t) \quad \omega(t)]^T\) from left and \([x(t) \quad \omega(t)]\) from right,

\[\begin{bmatrix} x(t) \\ \omega(t) \end{bmatrix}^T \begin{bmatrix} (A + B_1K)P + P(A + B_1K) + C^T \omegaI \\ (PB_2)^T \end{bmatrix} \begin{bmatrix} x(t) \\ \omega(t) \end{bmatrix} < 0 \quad (32)\]

Using the Schur complement,

\[(A + B_1K)P + P(A + B_1K) + C^T \omegaI - (PB_2)(-\omegaI)^{-1}(PB_2)^T < 0 \quad (33)\]

Pre and post multiplying with \(P^{-1}\), using the Schur complement again and applying \(X = P^{-1}\) variable transformation,

\[\begin{bmatrix} (A + B_1K)X + X(A + B_1K)^T + XC^TCX \\ B_1^T \\ CX \end{bmatrix} < 0 \quad (34)\]

With the help of the KYP lemma (Boyd et al., 1994),

\[\begin{bmatrix} (A + B_1K)X + X(A + B_1K)^T & B_2 & XC^T \\ B_1^T & -\omegaI & 0 \\ CX & 0 & -\omegaI \end{bmatrix} < 0 \quad (35)\]

Inequality (35) still includes some BMI terms, applying \(W = KX\) transformation,

\[\begin{bmatrix} AX + XA^T + B_2W + W^TB_1^T & B_2 & XC_1^T \\ B_1^T & -\omegaI & 0 \\ C_1X & 0 & -\omegaI \end{bmatrix} < 0 \quad (36)\]

Here, the controller gain \(K\) can be obtained as,

\[K = WX^{-1} \quad (37)\]

The constraint for control input \(\|u(t)\|_2 \leq u_{\text{max}}\) can be expressed as,

\[\sqrt{u(t)^Tu(t)} \leq u_{\text{max}} \quad (38)\]

Substituting the state feedback control law \(u(t) = Kx(t)\) into (28) together with (37)

\[(WX^{-1}x(t))^TWX^{-1}x(t) \leq u_{\text{max}}^2 \quad (39)\]

Multiplying by \(\frac{1}{u_{\text{max}}}\) from left and right,

\[\frac{1}{u_{\text{max}}}x(t)^TX^{-1}W^TWX^{-1}x(t) \leq 1 \quad (40)\]

Inequality (40) states the \(x(t) \in \mathbb{E}_{\frac{1}{u_{\text{max}}}W^TWX^{-1}}\) ellipsoid and it is contained by the ellipsoid \(\mathbb{E}_X^{-1}\). Hence,

\[X^{-1} \geq \frac{1}{u_{\text{max}}}W^TWX^{-1} \quad (41)\]

Multiplying (41) by \(X\) from left and right,

\[X \geq \frac{W^TW}{u_{\text{max}}} \quad (42)\]

Using Schur complement, one can obtain,

\[\begin{bmatrix} X & W^T \\ W & u_{\text{max}}^2I \end{bmatrix} \geq 0 \quad (43)\]
**Theorem 1.** Consider an actuator saturated closed loop system given in (21)-(22), the closed loop system is asymptotically stable for a given positive scalar constant $\gamma$, if there exist matrix $W$ and positive definite matrix $X$ with appropriate dimensions such that,

$$\begin{bmatrix}
AX + XA^T + B_2W + W^TB_2^T & B_2 & XC_1^T \\
B_1^T & -\gamma I & 0 \\
C_1X & 0 & -\gamma I
\end{bmatrix} < 0 \quad (44)$$

$$\begin{bmatrix}
X & W^T \\
W & u_{\text{max}}^2 I
\end{bmatrix} \geq 0 \quad (45)$$

where $u_{\text{max}}$ is the control input saturation upper limit. Then, disturbance attenuation level $\gamma$ is an $H_\infty$ upper bound of the resulting closed loop system from $\omega(t)$ to $z(t)$ for all $t \geq 0$ and there exists the state feedback $H_\infty$ controller $u(t) = WX^{-1}x(t)$, associated with $\gamma$.

**4. SIMULATION AND RESULTS**

In this section, the proposed controller is simulated in a powertrain model in order to present the validity and the effectiveness of the controller. Once the linear matrix inequalities which are given in Theorem 1 are solved with the help of YALMIP (Löfberg, 2004) and SeDuMi (Sturm, 1999), the feedback control gain is obtained as,

$$K = \begin{bmatrix} 0.3275 & -0.2359 \end{bmatrix}.$$  

Figure 2 shows that the closed loop poles are stable whilst the open loop poles are marginally stable with the proposed state feedback control gain $K$.

![Figure 2](image)

**Figure 2.** Pole placement

Calculated controller performance is simulated in the model (1)-(10) and results are shown in Figure 3. In the simulations, vehicle starts creeping with the first gear from standstill and during the simulation clutch 2 torque is zero as the vehicle moves with the first gear. Creep speed is set to 5.75 km/h and the engine idling speed is set to 750 rpm.
As it can be seen in Figure 3, the vehicle speed starts oscillating around the creep speed at 2.5s and the vehicle speed settles at around 3s. Slip speed becomes zero in 2s and the input shaft speed remains nearly constant at creep speed. Also, vehicle speed is changing very smoothly.

Since the controller input is chosen as the change in the clutch torque, controller saturation upper limit is chosen as $u_{\text{max}} = 1000 \, N/s$ and the state feedback control gain optimization problem is solved using this value. As it can be seen
in Figure 4, the clutch torque change remains in the limits and the clutch torque values are very reasonable. Those results show that the controller performance is very satisfactory and quite applicable.

5. CONCLUSIONS

Creep is an important functionality of DCTs and it affects the driving comfort directly. In this work, a state feedback $H_\infty$ controller with actuator saturation is proposed to control clutch torque during creep. The proposed controller is simulated with a powertrain model which employs a DCT to transmit the torque between the engine and the wheels. It has been shown that the controller provides the closed loop stability and yields very applicable and smooth vehicle control. Moreover, proposed controller annihilates the necessity of time consuming calibration work thanks to optimization process which takes place in computer environment.

REFERENCES

POLİMER HİBRİD RULMANLARDA SICAKLİĞIN YÜK TAŞIMA KAPASİTESİTESİNE ETKİSİ

1Burcu KÜÇÜKOĞLU DOĞAN, 2Tuncay KARAÇAY

1,2Gazi Üniversitesi, Mühendislik Fakültesi, Makine Mühendisliği Bölümü, MALTEPE/ANKARA/TÜRKİYE

1Muş Alparslan Üniversitesi, Mühendislik Fakültesi, Makine Mühendisliği Bölümü,
MUŞ/TÜRKİYE

Sorumlu Yazar: 1Burcu KÜÇÜKOĞLU DOĞAN, burcukucukoglu@gazi.edu.tr

ÖZET


Anahtar Kelimeler: Polimer Hibrid Rulman, Sıcaklık, Elastisite Modülü

ABSTRACT

Polymer hybrid ball bearings have polymer inner and outer ring and steel, polymer or glass balls. Polymers elastic modulus is decreased with increasing the temperature. The glass transition temperature is very important for polymers working condition. This condition valid of polymer hybrid ball bearings. Temperature rise have been affected the polymer ball bearings static and dynamic behavior. So, temperature effect must be added to contact mechanic. In this work, the mathematical model has been used to determine hybrid ball bearings, which is with inner and outer rings made from polymer balls made from steel, elastic modulus change with temperature. In addition, the hybrid ball bearing load capacities has been investigated. The study results have been demonstrated hybrid ball bearings elastic modulus to decrease with increasing the temperature. However, increasing the critical deformation and load capacity.

Keywords: Polymer Hybrid Ball Bearing, Temperature, Young Modulus

1. GİRİŞ


2. SICAKLIK- ELASTİSİTE MODÜLÜ ARASINDAKİ İLİŞKİ


Eşitlik 1 de sıcaklık değişimle kayma modülü elde edilebilmekte, kayma modülü elde edildikten sonra eşitlik 2 de verilen elastisite modülü, kayma modülü ve poisson oranı arasındaki ilişkiyi belirtmektedir. Burada G kayma modülü, E elastisite modülü, V ise poisson oranıdır.

$$G(T) = \frac{T_g}{T} + 2 \quad T < T_g$$

(1)

$$E = 2G(1 + \nu)$$

(2)

Eşitlik 1 de sıcaklık değişimi ile kayma modülü edilebilirken, kayma modülü edil dikten sonra eşitlik 2 de verilen elastisite modülü, kayma modülü ve poisson oranı arasındaki ilişkiden yararlanılarak malzemelerin değişen sıcaklıkla birlikte elastisite modülleri bulunabilmektedir.

3. KRİTİK DEFORMASYON


$$\delta_c = \left( \frac{\pi C S_x}{2E'} \right)^2 R$$

(3)

$$\frac{1}{E'} = \frac{1 - \nu_1^2}{E_1} + \frac{1 - \nu_2^2}{E_2}$$

(4)
4. SICAKLIĞIN KRİTİK DEFORMASYONA ve YÜK TAŞIMA KAPASİTESİNE ETKİSİ


\[ \delta = \delta_i + \delta_o \]  

\[ F = K\delta^{3/2} \]

Şıcaklık artış ile rulmanın elstisite modülü düşmekte ve Eşitlik 3 ile gösterilen kritik deformasyon oranını artmaktadır. Kritik deformasyonun artmasıyla birlikte rulmanın yük taşıma kapasitesi artış göstermektedir.

5. BULGULAR ve TARTIŞMA

Tablo 1 de verilen rulman özellikleri görece benzetim yapılarak sıcaklık artışıyla birlikte rulmanın elstisite modülü düşmekte ve Eşitlik 3 ile gösterilen kritik deformasyon oranı artmaktadır. Kritik deformasyonun artmasıyla birlikte rulmanın yük taşıma kapasitesi artış göstermektedir.

Tablo 1. Rulman Özellikleri

<table>
<thead>
<tr>
<th>Tanım</th>
<th>Boyut</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tip</td>
<td>radyal sabit bilyalı rulman</td>
</tr>
<tr>
<td>İç bilezik yuvarlanma çapı</td>
<td>0.046038 m</td>
</tr>
<tr>
<td>Ortalama Çap</td>
<td>0.0539855 m</td>
</tr>
<tr>
<td>Dış bilezik yuvarlanma çapı</td>
<td>0.061933 m</td>
</tr>
<tr>
<td>Bilya Çap</td>
<td>0.0079375 m</td>
</tr>
<tr>
<td>İç bilezik eğrilik yarıçapı</td>
<td>0.004082 m</td>
</tr>
<tr>
<td>Dış bilezik eğrilik yarıçapı</td>
<td>0.004161 m</td>
</tr>
<tr>
<td>Elastisite Modülü (çelik+HDPE)</td>
<td>3.1259 Gpa</td>
</tr>
<tr>
<td>Poisson ratio (çelik)</td>
<td>0.3</td>
</tr>
<tr>
<td>Poisson ratio (HDPE)</td>
<td>0.46</td>
</tr>
</tbody>
</table>

Şekil 1. Elastisite Modülü-Sıcaklık Değişim Grafiği
Şekil 2 de verilen grafik artan sıcaklık değerleriyle birlikte rulmandaki yük taşıma kapasitesinin artışını göstermektedir. Sıcaklık artışı rulmanın iç ve dış bileziğinin elastik durumunun artmasına, dolaysıyla bilyaların bileziklere daha çok temas etmesine ve bilezikler üzerindeki ezilmenin daha çok olması yol açmaktadır. Aynı zamanda sıcaklık artışı Esitlik 3 de verilen matematiksel formülde görüldüğü gibi elastisite modülü ile ters orantılı olarak değişmektedir. Rulmanların elastik bölge sınırlarındaki yük taşıma kapasitesi 

\[ \frac{\delta}{\delta_c} \leq 1 \]

durumu ile belirlendiğinden toplam ezilmenin ve kritik deformasyon oranının artışından dolayı sıcaklık artışı ile rulmanların yük taşıma kapasitelerinde artış görülmüştür.

**Şekil 2. Sıcaklık-Yük Taşıma Kapasitesi Değişim Grafiği**

Tablo 2 de ise sıcaklık etkisiyle değişen kritik deformasyon parametresini belirtmektedir. Tablo 2 de görüldüğü gibi sıcaklık artışıyla kritik deformasyon parametresinin artışı görülündü. Esitlik 3 ile gösterilen kritik deformasyon oranı belirlenen kullanılan eşdeğer elastisite modülü sıcaklık artışıyla birlikte düşmesi kritik deformasyon parametresinin artmasına neden olmaktadır.

**Tablo 2. Rulmanın Sıcaklık Değişiminin Kritik Deformasyona Etkisi**

<table>
<thead>
<tr>
<th>Sıcaklık (°C)</th>
<th>Kritik Deformasyon (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>24</td>
<td>2.9407e-05</td>
</tr>
<tr>
<td>30</td>
<td>3.4913e-05</td>
</tr>
<tr>
<td>40</td>
<td>4.5142e-05</td>
</tr>
<tr>
<td>50</td>
<td>5.6680e-05</td>
</tr>
<tr>
<td>60</td>
<td>6.9538e-05</td>
</tr>
<tr>
<td>70</td>
<td>8.3699e-05</td>
</tr>
<tr>
<td>80</td>
<td>9.9168e-05</td>
</tr>
<tr>
<td>83</td>
<td>1.0407e-04</td>
</tr>
</tbody>
</table>

**6. SONUÇ**


**KAYNAKLAR**

CHARACTERIZATION OF OSCILLATIONS DURING FLOW BOILING OF WATER IN PARALLEL MICROCHANNELS

Anže Sitar1,* , Andrej Lebar1,2 , Michele Crivellari3 , Alvise Bagolini3 , Iztok Golobič1

1 Faculty of Mechanical Engineering, Ljubljana, Slovenia.
2 Faculty of Health Sciences, Ljubljana, Slovenia.
3 Fondazione Bruno Kessler, Trento, Italy.
*Corresponding Author: Anže Sitar, anze.sitar@fs.uni-lj.si

ABSTRACT

Several challenges coexist in the field of flow boiling in microchannels, ranging from high superheat required for boiling incipience to boiling instabilities and early dryouts. The aim of this study is to hinder or overcome some of the challenges and develop an image processing algorithm for analysis of the boiling oscillations in multiple parallel channels. The experimental results were measured in an array of 64 parallel 25 × 25 μm microchannels with a synchronized high-speed visualization and measuring system. The small cross section of the microchannels allowed only the formation of annular two-phase flow and a computer algorithm was developed for tracking the meniscus oscillations during boiling. The applied image analysis focuses on reliability with the simultaneous use of brightness variation and brightness derivative along with an image subtraction. Moreover, the images were preprocessed to determine the number of microchannels and their orientation with applying different filtering and Radon transformation. The data extracted from the visualization was used to determine the peak to peak amplitudes and fundamental frequencies of the oscillating meniscus. The results exhibit lower amplitudes and higher fundamental frequencies with increasing heat flux. The mass flux was kept constant at 83 kg/m²s, whereas the heat flux varied from 150 kW/m² to 250 kW/m². The amplitudes and the fundamental frequencies of the meniscus oscillations are determining the length and duration of the microchannel with periodically alternating liquid and vapor phase.

Keywords: microchannels, flow boiling oscillations, high-speed visualization, image analysis.

1. INTRODUCTION

The next step in microchannel heat transfer is the implementation of flow boiling, which has the potential of higher heat transfer coefficients at lower pumping powers and constant fluid temperature, due to the utilization of the working fluid’s latent heat. However, there are several challenges, which inhibit a straightforward transfer of boiling from macro- to microchannels: intabilities and early dryouts, represented in (Kandlikar et al., 2013); high temperatures of the onset of boiling (ONB), which was covered in (Ghiaasiaan and Chedester, 2002) and (Kandlikar, 2010); and oscillations, which are emphasized during boiling in microchannels, due to the small channel cross sections and highly superheated walls.

(Kandlikar et al., 2013) reviewed the current status and future research needs of heat transfer in microchannels. The prime concerns of the reviewed literature on flow boiling in microchannels were the instabilities, which accompany the boiling process: temperature and pressure oscillations, thermal stress cycling, vibration and premature transition to critical heat flux (CHF). Therefore, it is crucial to focus on detection and reduction of these instabilities in pursuit of heat transfer enhancement during flow boiling in microchannels. (Wu and Cheng, 2004) investigated boiling instabilities of water flowing in an array of trapezoidal microchannels (Xu et al., 2009) thoroughly eliminated pressure oscillations of acetone boiling in an array of triangular microchannels. However, the same technic is prone to less successful results if water is used as the working fluid due to the large latent heat of vaporization. (Xu and Xu, 2012) replaced water with nanofluid (water and Al2O3) and reported heat transfer enhancement.

There are generally two possible solutions for flow stabilization available in the literature. The first possible improvement is the fabrication of inlet restrictors, which provide a higher upstream pressure drop and suppress the potential vapor backflows (Kandlikar et al., 2005, Kuo and Peles, 2007, Wang et al., 2008, Sitar et al., 2012) (Szczukiewicz et al., 2013). However, flow restrictors also introduce an additional, inevitable and undesired
downstream pressure drop and the added flow resistance must be compensated with a higher pumping power and energy consumption.

The second already used improvement to stabilize the flow boiling in microchannels is to include nucleation sites, which will become active at the lowest possible temperature. This measure lowers the temperatures of the walls during boiling and hence lowers the additional heat available for bubble growth after the initial onset of boiling. Most commonly, the nucleation sites are fabricated in the form of a small cavity at the microchannel bottom or the side wall. The most important parameter of the nucleation cavities is their size, which has a significant effect on their activation during boiling in microchannels. The advantages of implementing the potential nucleation cavities were already covered in (Sitar et al., 2012, Sitar and Golobic, 2016).

In single microchannel experiments the boiling front instabilities are distinctly manifested in the pressure measurements as well as in high-speed visualization (Sitar et al., in press 2019), however the same is not valid in multiple channel flow boiling. Namely, the meniscus oscillations during boiling have different phase offsets from one microchannel to another and since the pressure sensor is located in the manifold, the measured pressure is the sum of all the microchannel pressure oscillations. Consequently, the boiling front oscillations in the manufactured microchannel array were analyzed from the results of visualization, as the pressure measurements are insufficient.

The aim of the current study was the mitigation of the flow boiling instabilities in parallel microchannels with flow restrictors and potential nucleation cavities. Moreover, the instabilities detected with high speed visualization were analyzed in order to evaluate the magnitude and the frequencies of the oscillations and the effect of the working conditions on flow boiling.

2. EXPERIMENTAL SETUP AND MICROCHANNEL TEST SECTION

The boiling heat transfer experiments were performed with the measurement setup presented in a block diagram in Fig. 1. The experimental equipment is comprised from visualization and measuring components. Pressure, temperature and heat flux are measured and analyzed with the data acquisition and analysis system on the left hand side of the diagram, whereas the high-speed visualization is performed and analyzed with the components on the right hand side of the block diagram in Fig. 1. The emphasis of this study is to visually capture the oscillations during flow boiling in microchannels and develop a reliable image analysis, which will adequately recognize the two-phase flow structures and oscillations in microchannels.

![Fig. 1. Experimental setup.](image-url)
All experiments were conducted in an array of 64 parallel microchannels with a cross section of $25 \times 25$ µm as shown in Fig. 2. Degassed double-distilled water was pumped through the microchannels at a constant mass flux of 83 kg/m²s from the left to the right hand side of the microchannel test section. The distinctive construction features of the tested microchannel array are: airgaps between active and inactive microchannel array; inlet and outlet manifolds; fabricated potential nucleation cavities in the microchannel walls, which are all thoroughly presented in Sitar et al. [10]. In order to suppress vapor backflows during flow boiling in microchannels, inlet restrictors with a cross-section of $7.5 \times 25$ µm (width × depth) were manufactured. The mass flux of the working fluid was kept constant and ONB was achieved at approximately 120 °C, afterwards the temperature was gradually raised up to 200 °C, thus allowing an assessment of heat flux effect on oscillations of the boiling front in microchannels.

![Fig. 2. Array of 64 parallel microchannels 25 × 25 µm – (a) complete microchannel array, (b) inlet manifold and restrictors, (c) potential nucleation cavities.](image)

The array of microchannels has 64 parallel microchannels of which every 16th channel has smooth walls without any potential nucleation sites fabricated, as presented in Fig. 2 (c). These four microchannels serve as a reference for comparison and evaluation of etched cavities effect on the boiling process, the excess temperature of the ONB and on the instabilities during boiling.
3. DIGITAL IMAGE SEQUENCE ANALYSIS

The analysis of flow boiling visualizations was based on the acquired sets of nearly 3000 images. Later on, the specific computer procedure was applied to each image in order to determine the meniscus location. Consecutive analysis yielded information on dynamics of meniscus oscillations. In Fig. 3 the digital image analysis workflow is given in the form of a functional diagram, which is to be explained in the following paragraphs in more detail. Image analysis was performed in the Matlab environment.

Due to manipulation of the microchannel array specimen and the camera during experiments a small undesired rotation of the coordinate system local to the specimen towards the coordinate system of the camera occurred. The edge filter and the Radon transform was applied to the image in order to determine angular difference between coordinate systems. Radon transform was applied in the range of one degree with the incremental steps of 0.01 degree. Angle of rotation between both coordinate systems was determined from the angle at which maximal value of Radon transform occurred. Once the angle of rotation misalignment was found, the image matrix was enlarged to four times of the original size, rotated and scaled back to the original size. This way the artifacts due to the aliasing were avoided or diminished to acceptable level.

In order to determine the region of interest (ROI) in which meniscus location was searched for, the center location for every microchannel in the array had to be found. Locations were determined by locating the dark sections on the left side of the image matrix. Resulting center locations are marked with green dots in Fig. 4. Reflections can be observed in the middle of microchannels in Fig. 4, which disturbed determination of dark segments to some extent, therefore a smoothing filter was applied to observed column. Center locations were afterwards precisely determined. After the microchannel centers were known, each channel was identified as a region of interest (ROI) determined by the center of the microchannel and channel width.

![Fig. 3. Functional diagram of the digital image sequence analysis.](image)

![Fig. 4. Labeled microchannels and their center](image)
Computer procedure searched for meniscus location in each ROI. Initially it was based on the straightforward fact that the image brightness changes significantly at the border between the liquid and vapor phase, as it is seen in Fig. 5 (a). In order to decrease brightness variations or noise in the image along the microchannel, a sum of pixel values in the columns of ROI was calculated. Resulting graph can be observed in the Fig. 5 (b). Location of the meniscus can be determined by applying a threshold and the first point which exceeds the threshold is the location of the meniscus.

Fig. 5. Identification of the meniscus location – a) ROI of channel #1 cropped from the complete image (Fig. 4); b) brightness variation; c) derivative of the brightness variation.

An improvement of the discussed method is to compute the derivative of the brightness variation, as it can be observed in Fig. 5 (c). The highest peak is expected to correspond to the meniscus location. Several partially unavoidable challenges have been found in locating the meniscus location: bubbles nucleating upstream of meniscus, microchannel surface features, potential dust and stains etc. In order to overcome this challenge, an analysis was performed on the differences or subtraction of the consecutive images instead of on just one single image. Brightness variation and derivative of image brightness variation was observed. The result of subtraction of consecutive images from the example presented in Fig. 5 is shown in Fig. 6. Analysis of the meniscus location on the differences of images yielded more reliable results.

Fig. 6. Subtraction of consecutive images.

4. RESULTS AND DISCUSSION

Flow boiling heat transfer in channels and especially in microchannels is usually closely followed by instabilities, vapor backflows, high superheats needed for the onset of boiling and overall unpredictable behavior. A simple solution to suppress vapor backflows and ensure a more stable boiling in microchannels is the incorporation of inlet restrictors, which increase the pressure drop in the upstream direction and therefore promote the downstream flow of both vapor and liquid phase of the working fluid. An additional design features often incorporated are the potential nucleation cavities, which lower the temperature of the ONB and hence hinder the vapor backflows due to a lower excess heat available at the ONB.

Inlet restrictors with a 30 % cross section compared to the original channel size were fabricated in all microchannels to ensure stable boiling of water, which is presented in Fig. 7 at the heat and mass flux of 230 kW/m² and 83 kg/m²s, respectively. Although, the maximum measured temperature in the microchannels was 196 °C we have not achieved boiling in 4 out of 64 microchannels, which are full with liquid water and marked with asterisks on Fig. 7 and 8. These
microchannels were all etched without the potential nucleation cavities in the channel walls to verify the benefits of etching cavities in the microchannel walls. The presented frames in Fig. 7 demonstrate stable boiling and the necessity of etched nucleation sites, as boiling was achieved only in microchannels with the fabricated nucleation sites, which considerably lowered the superheat required for ONB. The boiling front was not oscillating vigorously, which could lead to vapor flow entering the inlet flow manifold and to unstable flow. Moreover, the boiling front is located at approximately the same axial location in all microchannels, which demonstrates a similar temperature distribution and conditions for heat transfer irrespective of the microchannel location in an array.

besides characterizing flow boiling as stable there is also possible to assess the degree of boiling stability from the visual meniscus oscillations. To estimate the stability and compare different microchannels and working conditions a manual analysis of the meniscus location was made. A single analyzed frame is given in Fig. 8 (a) where the liquid phase is depicted darker as the vapor phase.

The locations of the meniscuses during boiling were analyzed and marked with crosses in all 64 parallel microchannels in a hundred frames, which were taken at 528 fps. Fig. 8 (b) shows the results of the minimum, average and maximum location of the meniscus during the observed time period. The average value of the meniscus location is between 0.74 mm and 1.65 mm for all microchannels (with flow boiling) in the analyzed consecutive frames. The airgaps between active and inactive arrays of microchannels lowered heat loss and ensured similar working conditions in all microchannels, which is confirmed with an almost vertical boiling front. Alternating liquid and vapor water was observed only in the length between maximum (downstream) and minimum (upstream) locations of the meniscus, which can be characterized as the effective length for heat transfer during flow boiling in microchannels. The small cross section of a microchannel contributed to a very short microchannel length with two-phase annular flow of water, which was almost instantly replaced with a single-phase vapor flow. A rapid dryout of the channel during boiling results in a considerably lower heat transfer coefficient at the dry walls. Therefore, high heat transfer coefficients in microchannels with small hydraulic diameters mainly originate from the alternating liquid-vapor flow. The average length of alternating liquid-vapor flow of all 60 boiling microchannels in the analyzed frames is 1.19 mm, which refers to the average peak to peak amplitude of the meniscus oscillations. The average location of meniscus during boiling is
closer to its minimum value compared to the maximum value which implies that the meniscus is located at the downstream (maximum) rarely and for a shorter time period.

A rigorous principle for determination of the meniscus locations was required to eliminate the influences of upstream emerging bubbles, irregularities in etched features and possible dust and stains in the visualization equipment. Furthermore, the maximum number of consecutive frames is depended on the frame size, due to the video equipment restrictions. Therefore a frame size reduction was required to allow longer times of observation, which were crucial for an adequate statistical analysis of the oscillating meniscus locations. The video samples analyzed with the designed computer algorithm were 5.7 seconds long and filmed at 525 fps with the frame size reduced to visualization of 16 microchannels. The analysis presented in Fig. 8 (b) was made for oscillations in all 64 microchannels in an array and exhibits statistically insignificant deviations among the microchannels, therefore the reduced frame size is not introducing a distinctive error.

The oscillations of meniscus during boiling in microchannels were observed in 16 of 64 parallel channels to allow longer times of analysis, as the number of frames is inversely proportional to their size. The events in microchannels were visualized at different heat fluxes to evaluate its’ effect on the fluctuations. The results of the analysis algorithm for one of the microchannels are presented in Fig. 9 with the heat flux varying from 158 kW/m$^2$ to 209 kW/m$^2$.

![Meniscus oscillations of the analyzed microchannel #9 at different heat fluxes.](image)

The oscillations presented in Fig. 9 and also in general are more periodic at lower heat fluxes and become more chaotic at higher heat fluxes. There are several factors, which contribute to this behavior: (i) higher heat flux promotes a higher fundamental frequency of oscillations, due to the higher energy available; (ii) the neighboring channels have a larger effect on the observed channel at higher heat fluxes; (iii) the high-speed visualization was not fast enough for the oscillation at high heat fluxes. The Fast Fourier Transformation (FFT) conducted on the meniscus location in microchannel #9 at 158 kW/m$^2$ is presented in Fig. 10. The fundamental frequency of 35.6 Hz is distinctly emphasized from its’ harmonics. The fundamental frequencies are evident in more periodical oscillations, whereas it was difficult to assess the frequency at higher heat fluxes in some cases.

The fluctuations of the boiling front have higher fundamental frequencies and lower amplitudes with increasing the heat flux. Moreover, the observed oscillations are becoming more chaotic and less periodic at higher heat fluxes, where the determination of the fundamental frequencies also becomes more uncertain, although the amplitudes of the oscillating meniscus are smaller. The results presented were gathered from experimental work on 25 × 25 µm microchannels and are demonstrating the impact of heat flux on boiling instabilities, which should be considered in two-phase micro heat exchangers. Namely, the heat flux is considerably affecting the length of the microchannel with alternating liquid and vapor flow, which is proportional to the surface of effective heat transfer.
Preliminary experiments were conducted also in an array of 50 × 50 µm microchannels and showed larger amplitudes of oscillations during boiling compared to the 25 × 25 µm microchannels. However, the employed visualization equipment was adequate only for observation of oscillations during boiling in smaller microchannels, due to the large amplitudes of oscillations occurring in 50 × 50 µm microchannels, which were often out of the visual field of the microscope.

4. CONCLUSIONS

Flow boiling of degassed water was visualized in square microchannels with hydraulic diameters of 25 µm. The visualization and analysis of the high speed images during flow boiling of water in 64 parallel 25 × 25 µm microchannels enhanced with flow restrictors and potential nucleation cavities are confirming stable boiling. A characterization of oscillations during flow boiling of water in 25 × 25 µm microchannels was made and average peak to peak amplitude of 1.19 mm was found at 230 kW/m². The amplitude of oscillation is an indicator of the microchannel length effective for heat transfer, as it depicts the alternating liquid-vapor length of the microchannel with a substantially higher heat transfer coefficient compared to single-phase liquid or vapor flow. Moreover, the detailed analysis revealed a very periodic behavior of the boiling front oscillations and the effect of heat flux on the fluctuations. The analysis includes determination of fundamental frequencies with FFT and peak to peak amplitudes of the boiling front oscillations. The average peak to peak amplitude of the microchannel has lowered from 1.9 mm at 158 kW/m² to only 0.6 mm at 209 kW/m², which is an apparent indicator of a shorter length of alternating liquid and vapor phase in microchannels at higher heat fluxes. The fundamental frequencies are increasing with the heat flux, whereas the amplitudes are demonstrating a decrease at higher heat fluxes.

The effect of the heat flux is evident, which is crucial for developing and designing future two-phase heat transfer devices. Namely, the heat flux along with other working conditions is affecting the alternating liquid-vapor length of the microchannels with a high heat transfer coefficient. Therefore, the amplitudes and frequencies of the oscillating meniscus are an important parameter of the performance of two-phase heat transfer devices. The statistical analysis of the boiling front presents a novel method of determining the alternating liquid-vapor length of the microchannel, in which the liquid and vapor phase are constantly alternating. Although experimental studies already coped with instabilities during boiling in microchannels, their method of assessing the fluctuations was in most cases supported by pressure measurements, which has two main deficiencies: (i) it is suitable for experimenting in single microchannels, whereas in arrays of microchannels it is not the proper indicator of the oscillations during boiling and (ii) although it is adequate for estimation of the frequency of the fluctuations, it is not appropriate for analyzing the amplitudes of the boiling front oscillations, due to their indistinct effect on pressure in microchannels. The amplitudes of oscillating boiling front are crucial for determining the surface with high heat transfer coefficient, which was found to be directly dependent on the applied heat flux. Higher heat fluxes demand shorter lengths of microchannels for efficient heat transfer, which could be achieved with a larger number of inlets and outlets.
4. REFERENCES

Effect of Production Temperature on Vibration Behavior of Composite Materials

Aydın GÜNEŞ1*, Abdullah ASLAN2, Emin SALUR3, Ömer Sinan ŞAHİN1

1Konya Technical University, Mech. Eng. Dept., Konya, Turkey
2Selçuk University, Mech. Eng. Dept., Konya, Turkey
3Selçuk University, Met. and Mat. Eng. Dept., Konya, Turkey
1Corresponding author: Aydın Güneş, aydingns@hotmail.com

ABSTRACT

In this study, the vibration behavior of metal matrix composite materials obtained by recycling of waste metal chips with isostatic hot pressing method is investigated. Bronze chips were used as a matrix component and spheroidal cast iron chips were used as reinforcement component. 80% bronze and 20% spheroidal graphite cast iron were mixed in the production of composite materials. The production of the determined materials was carried out at three different temperatures and at a pressure of 170 MPa. As a result of modal rigidity and natural frequency values obtained by the application of hammer hammer tests, the effect of different temperatures on the vibration behavior of composite materials was observed.

Keywords: metal matrix composites, CuSn10, GGG40, isostatic hot pressing, vibration

1. INTRODUCTION

Many studies have been carried out to evaluate and/or recover waste metal chips. The majority of the studies are about making the chips useful with the help of melting processes. It is observed that the studies focused on bronze, brass and aluminum chips. Nowadays, many machine elements can be produced by powder metallurgy method. Powder metallurgy method used in the preparation and supply of dust is difficult and costly. In today’s industry, the evaluation of sawdust, which is the production waste, is mostly applied in the form of recycling by melting process. This process is both efficient and low cost. The production of the chips obtained at a very low cost with the proposed method and the usability of the porosity of the porous structure as a self-lubricating plain bearing will be investigated [1-4].

The melting of the chips requires a lot of energy. The energy consumed for this process creates negative effects both for our national economy and for our environment [5]. The melting process and the subsequent casting process can be considered as an inefficient process considering the harmful gases released into the environment and the workforce required for this process [6]. In this respect, it will have positive results in every aspect of the evaluation of the waste chip with a process that requires less energy and does not harm our environment. In this study, more efficient production will be realized by using double acting hot pressing method instead of melting process for this purpose. The
works will be carried out at three different temperatures under a constant pressure and the effect of the production temperature on the composite material will be examined.

2.MATERIALS AND METHOD

Metal-matrix composite materials were produced by pressing the waste bronze (CuSn10) and cast iron (GGG40) chips at different temperatures using hot pressing method. The mixing ratios were selected from the matrix material bronze 80% by weight and 20% by weight of the reinforcing material cast iron. Composite materials were produced at a constant pressure of 170 MPa and three different temperatures. As seen in Figure 1, the heat loss is minimized by wrapping around the female mold with glass wool. Double and top pressing at the same time with double-acting pressing.

![Fig.1 Production Unit](image)

In order to determine the vibration behaviors after the production of composite materials, hammer hammer tests were carried out at Sabancı University. Figure 2 shows the application of the experiments with the production of the metal matrix composite materials to the appropriate geometries for the impact hammer test. In order to obtain the instantaneous data on the reverse side of the material which is connected vertically, the acceleration meter is applied and the impact loads are carried out with the help of hammer.
3. RESULTS AND DISCUSSION

Vibration behavior of metal matrix composite materials which are produced at different temperatures in double acting hot press with pulsing hammer tests have been investigated. Table 1 shows the results. Changes in the data obtained with the change in production temperature has occurred. As the temperature increases, the natural frequency and modal stiffness values increase and the damping rate decreases.

Table. 1 Impact hammer test results

<table>
<thead>
<tr>
<th>Specimen</th>
<th>Temp. (°C)</th>
<th>Pressure (MPa)</th>
<th>Natural Frequency (Hz)</th>
<th>Damping Ratio (%)</th>
<th>Modal Stiffness (N/m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>80B20D</td>
<td>350</td>
<td>170</td>
<td>1301,69</td>
<td>0,775</td>
<td>8,20E+05</td>
</tr>
<tr>
<td></td>
<td>400</td>
<td></td>
<td>1583,26</td>
<td>0,65</td>
<td>1,23E+06</td>
</tr>
<tr>
<td></td>
<td>450</td>
<td></td>
<td>1975,25</td>
<td>0,509</td>
<td>2,38E+06</td>
</tr>
</tbody>
</table>

ACKNOWLEDGEMENT

This work has been funded by Tubitak (The scientific and technological research council of Turkey) under grant number 113M141.

REFERENCES


Vibration Behavior of Metal Matrix Composite Materials Produced at Different Pressures

Aydın GÜNEŞ¹, Emin SALUR², Abdullah ASLAN³, Ömer Sinan ŞAHİN¹

¹Konya Technical University, Mech. Eng. Dept., Konya, Türkiye
²Selçuk University, Met. and Mat. Eng. Dept., Konya, Türkiye
³Selçuk University, Mech. Eng. Dept., Konya, Türkiye
¹Corresponding author: Aydın Güneş, aydingns@hotmail.com

ABSTRACT

In this study, the effect of production pressure on the vibration behavior of metal matrix composite materials produced at different pressures was investigated. CuSn10 (bronze) was used as matrix material and GGG40 (cast iron) was used as reinforcement material. In the production of metal matrix composite materials, the reinforcement material is 10% by weight. The materials were produced at three different pressures (100 MPa, 133 MPa and 170 MPa) and at a temperature of 450 °C. The effect of different pressures on the vibration behavior of composite materials were investigated as a result of the modal rigidity and natural frequency values found after the impact hammer tests.

Keywords: metal matrix composites, bronze, cast iron, hot pressing, vibration behavior

1. INTRODUCTION

The world population is growing very rapidly. For this reason, our rapidly declining natural resources have caused material costs as well as cost and corrosion. As a result of the natural balance deteriorating due to excessive consumption, the issue of recycling has become one of the most important topics in the world of science and industry. Since recycling is more cost-effective than remanufacturing, it is becoming increasingly important worldwide. In many industrial applications with recycling processes, the materials left in the waste state are recycled. Metals also come at the beginning of these materials [1-5]. Thus, the amount of waste is reduced, natural resources are protected, large energy savings are provided and all these play an important role on the material costs [6].

The wastes recovered from the factories are re-used according to their usage areas and according to the desired properties and re-used in various furnaces. This process requires very large energy and is an...
extremely inefficient process. In addition, the air pockets inside the material make the heat conduction very difficult. This makes it difficult to melt the material. In the new method developed as an alternative to these methods, there are significant differences both in terms of environment and applicability [7-8].

In this study, metal shavings in waste are brought to certain dimensions with appropriate mechanical processes. Metal shavings were pressed at 450 °C at three different pressures by double acting press and metal matrix composite materials were obtained. Vibration behaviors of metal matrix composite materials were investigated by using impact hammer test equipment.

2. MATERIALS AND METHODS

Vibration behavior of metal matrix composite materials which are investigated under temperature is produced by double acting press. The matrix material was made of bronze (CuSn10) and spheroidal graphite cast iron (GGG40) was used as reinforcement material. 90% bronze and 10% spheroidal graphite cast iron material were used as the mixing ratio. The production of the materials was carried out at a constant temperature of 450 °C and under three different pressures. 100 MPa, 133MP and 170 MPa were chosen as production pressures. The production of prismatic materials is then increased to 13mmx13mmx70mm. Impact hammer tests were carried out at Sabanci University Manufacturing Laboratory. The operation is shown in figure 1. As a result of the modal rigidity and natural frequency values obtained after the vibration tests, the effect of pressure on the vibration behavior of the materials was determined.

![Figure 1 Vibration test unit](image-url)
3. RESULTS AND DISCUSSIONS

Pulsed hammer tests were performed to determine the vibration behaviors of the composite materials. Impact hammer tests were performed in room conditions and in five replicates. The data obtained after the vibration tests are shown in Table 1. When the data in the table are analyzed, the natural frequency and damping ratio values increase with the increase in the production pressure of the composite material. In addition, the modal stiffness value increases as seen at the far right of the table.

<table>
<thead>
<tr>
<th>Mixing ratio (specimen)</th>
<th>Temp. (°C)</th>
<th>Pressure (MPa)</th>
<th>Natural Frequency (Hz)</th>
<th>Damping Ratio (%)</th>
<th>Modal Stiffness (N/m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>90B10D</td>
<td>450</td>
<td>100</td>
<td>1628.72</td>
<td>0.403</td>
<td>1.43E+06</td>
</tr>
<tr>
<td></td>
<td></td>
<td>133</td>
<td>1770.22</td>
<td>0.475</td>
<td>1.70E+06</td>
</tr>
<tr>
<td></td>
<td></td>
<td>170</td>
<td>1970.22</td>
<td>0.634</td>
<td>2.12E+06</td>
</tr>
</tbody>
</table>

4. ACKNOWLEDGEMENT

This work has been funded by Tubitak (The scientific and technological research council of Turkey) under grant number 113M141.

REFERENCES


MULTI OBJECTIVE OPTIMIZATION OF CUTTING PARAMETERS IN A SINGLE PASS TURNING OPERATION USING THE BEES ALGORITHM

Osman Öztürk¹, Mete Kalyoncu², Ali Ünüvar³

¹,²,³ Konya Technical University, Mechanical Engineering Department, Konya, Turkey
¹ Corresponding Author: Osman Öztürk, osmanozturk@selcuk.edu.tr

ABSTRACT

The aim of this study is to solve a multi-objective optimization problem in single pass turning operation by using a heuristic optimization method. Tool life is one of the most important objectives of a productive manufacturing. In order to achieve a longer tool life, cutting parameters must be determined carefully. Although these parameters can be chosen according to tool manufacturers’ recommendations, optimization of these parameters with the aim of a higher tool life and lower cutting force is a more cost-effective solution. In this study, parameters of single pass turning operation such as cutting speed, feed rate and cutting depth are taken into consideration. These three parameters are optimized using the Bees Algorithm. Tool life and cutting force are objective functions, which are maximized and minimized respectively. Mathematical models of tool life and cutting force is obtained by using polynomial regression. These mathematical models are reduced to one multi-objective function. The Bees Algorithm is used to optimize the cutting parameters. Population size and initial size of neighborhood are significant parameters for the performance of the method. The primary advantage of the Bees Algorithm is the ability to reach the global optimum point. The convergence performance of the optimization method is also investigated by choosing different values of population size and initial size of neighborhood.

Keywords: The Bees Algorithm, regression model, optimization, tool life, cutting force, turning

1. INTRODUCTION

Functional properties of a product are generally affected by its surface quality. Some important features such as good wear and corrosion resistance, high fatigue strength, decent ability of distributing and holding lubricant can be obtained, when required surface roughness is achieved (Neşeli, 2011). The surface quality of a machined product directly depends on tool geometry and cutting parameters. As tool geometry is a design parameter, process parameters in a turning operation must be determined by experienced staff. However, great variety of materials is used in today’s industry, so optimizing cutting parameters properly plays an important role in an efficient machining process. Because the cutting tool inevitably wears during turning operation, cutting parameters must be chosen carefully to delay the depreciation.

Measurement of the cutting forces in a turning operation gives useful information about the condition of the tool. If the cutting forces acting on a tool reaches high values, the tool which is expensive for a manufacturer wears quickly. Therefore, measurement of the cutting forces in a turning operation gives useful information about the condition of the tool. Objectives of a productive turning process can be selected as minimum cutting forces and maximum tool life. Maximum cutting speed is also very significant for the cost of the process, but it will not be the objective factor in this paper.

In literature, several researchers used heuristic optimization methods for material removal processes. Chen and Tsai (1996) focused on simulated annealing algorithm (SA) which is a heuristic method. A hybrid optimization algorithm based on SA algorithm and Hooke-Jeeves pattern search is developed to find optimum parameters of multi pass turning, Their intention is to find the best solution of parameters in a vast solution space. This hybrid optimization approach helps user to avoid local minimums in a reasonable computation time. By using two different approaches, speed advantage of Hooke-Jeeves search and ability to find the global optimum in SA are combined (Chen and Tsai, 1996).

Comparison of recent optimization methods in machining processes is also studied by several researches. Wang et al. (2005) proposed a hybrid algorithm based on genetic algorithm (GA) and simulated annealing (SA) for a multi pass milling operation problem. Proposed method is named as parallel genetic simulated annealing (PGSA) and is also compared with three different methods. It is concluded that PGSA is more efficient for optimization of cutting parameter than geometric programming, dynamic programming and parallel GA (Wang, 2005). Ali R. Yıldız (2012) studied a hybrid
artificial bee colony (HABC) algorithm to solve cutting parameter problem in multi pass turning operations. HABC algorithm is compared with other evolutionary algorithms in literature. Cost function of the multi pass turning operation is minimized by HABC and introduced better performance among other methods when compared (Yildiz, 2012). Das et al. (2014) studied the optimization of process parameter of electro discharge machining by using artificial bee colony algorithm. Material removal rate and surface roughness is considered as objective functions. Instead of a separate evaluation of these objectives, multi objective optimization approach is proposed for electro discharge machining (Das, 2014).

In this study, multi objective optimization of cutting parameters in single pass turning is developed by using the bees algorithm. Objective functions are generated from empirical data using polynomial regression model. AISI 4140 steel with cylindrical geometry is machined with WNMG rough turning cutting tool.

2. MATERIALS AND METHOD

This study contains an experimental procedure and calculation section for optimum values. In experimental section, heat treated AISI 4140 steel with cylindrical geometry is used. Single pass turning operation is performed using Iscar WNMG P0804 cutting tool with a tool radius of 0.8 mm. Cutting parameters are chosen as cutting speed, feed rate and cutting depth. Range of the parameters are given in Table 1.

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Cutting Speed, v (m/min)</th>
<th>Feed Rate, f (mm/rev)</th>
<th>Cutting Depth, d (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Range</td>
<td>150 ≤ v ≤ 250</td>
<td>0.05 ≤ f ≤ 0.2</td>
<td>0.5 ≤ d ≤ 1.5</td>
</tr>
</tbody>
</table>

Instead of possible values in a range of parameters, specific levels of parameters are also needed for polynomial regression. Therefore, three levels for each parameter are chosen for experiments. Levels of parameters for cutting speed, feed rate and cutting depth are; 150-190-250, 0.05-0.1-0.2, 0.5-0.87-1.5 respectively. Cutting force is measured during experiments, while tool wear is also measured after each experiment. An example of tool wear in this study is given in Figure 1.

Cutting parameters are considered as inputs, while tool wear and cutting forces as outputs. Full factorial experimental design is established to minimize the total number of experiments. Experimental procedure contains main cutting forces and tool wear measurement. Cutting force data are collected from Kistler 9257B dynamometer and tool wear is measured by using tool manufacturer’s microscope which is given in Figure 2. Experimental plan is given in Table 2.
Table 2. Experimental design for single pass turning operation.

<table>
<thead>
<tr>
<th>Exp. No.</th>
<th>Cutting Speed, ( v ) (m/dk)</th>
<th>Feed Rate, ( f ) (mm/dev)</th>
<th>Cutting Depth, ( d ) (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>250</td>
<td>0.05</td>
<td>0.5</td>
</tr>
<tr>
<td>2</td>
<td>150</td>
<td>0.2</td>
<td>0.5</td>
</tr>
<tr>
<td>3</td>
<td>150</td>
<td>0.05</td>
<td>1.5</td>
</tr>
<tr>
<td>4</td>
<td>250</td>
<td>0.2</td>
<td>1.5</td>
</tr>
<tr>
<td>5</td>
<td>190</td>
<td>0.1</td>
<td>0.87</td>
</tr>
<tr>
<td>6</td>
<td>150</td>
<td>0.1</td>
<td>0.87</td>
</tr>
<tr>
<td>7</td>
<td>190</td>
<td>0.05</td>
<td>0.87</td>
</tr>
<tr>
<td>8</td>
<td>190</td>
<td>0.1</td>
<td>1.5</td>
</tr>
<tr>
<td>9</td>
<td>190</td>
<td>0.2</td>
<td>0.5</td>
</tr>
<tr>
<td>10</td>
<td>250</td>
<td>0.1</td>
<td>0.5</td>
</tr>
<tr>
<td>11</td>
<td>250</td>
<td>0.2</td>
<td>0.87</td>
</tr>
<tr>
<td>12</td>
<td>250</td>
<td>0.1</td>
<td>0.87</td>
</tr>
<tr>
<td>13</td>
<td>190</td>
<td>0.2</td>
<td>0.87</td>
</tr>
<tr>
<td>14</td>
<td>190</td>
<td>0.1</td>
<td>0.5</td>
</tr>
</tbody>
</table>

3. RESULTS AND DISCUSSION

In order to develop a multi objective function, two different polynomial regression models are developed. The general form of polynomial regression model is given in equation (1). In this model; \( \beta_0 \) is intercept, \( \beta_1, \beta_2, \beta_3, \beta_4, \beta_5, \beta_6, \beta_7, \beta_8 \) and \( \beta_9 \) are coefficients of the empirical model (Can, 2017). \( Y \) is output of the regression model, \( v \) is cutting speed, \( f \) is feed rate and \( d \) is cutting depth.

\[
Y = \beta_0 + \beta_1 v + \beta_2 v^2 + \beta_3 f + \beta_4 f^2 + \beta_5 d + \beta_6 d^2 + \beta_7 v f + \beta_8 v d + \beta_9 f d
\]  

(1)

3.1. Polynomial Regression Model

Using experimental data, two objective functions are generated from polynomial regression model. Output of the first one is cutting force and second one is tool life. Cutting force is given in equation (2) and demonstrated as \( F_c \). Tool life is given in equation (3) and demonstrated as \( T \). In addition to cutting parameters defined above, tool wear influences cutting forces as well. Therefore, a fourth parameter named tool wear (\( t_w \)) is used in equation (2).

\[
F_c = 272.09 - 8.64v + 145.54f + 142.87d + 35.27t_w + 38.6v^2 + 53f^2 - 9.2d^2 - 4.3t_w^2 + 3.7vf + 2.41vd - 6.25vt_w + 69.37f d - 6.41ft_w + 9.69dt_w
\]  

(2)

\[
T = 4.393347 - 0.89789v - 0.85021f - 0.40039d - 0.10808v^2 - 0.57828f^2 - 0.011584d^2 - 0.05711vf + 0.439612vd - 0.15914fd
\]  

(3)

Multi objective function is developed according to modelling approach which is proposed by Das et. al. (2014) and given in equation (4). Main objective function is aimed to be maximized, so tool life function must be maximized. Cutting force is minimized at the same time because of the negative sign. As a result, cutting forces are minimized and tool life is maximized at the same optimization procedure. Impact of each output is considered as equal. For this reason, both terms are multiplied by 0.5 as seen in equation (4).

Main Objective Function = 0.5 \( T/T_{max} \) - 0.5 \( F_c/F_{c,min} \)

(4)

3.2. Optimization Results

The bees algorithm (BA) is an optimization method based on behavior of bees in nature while searching the best food source. The best food source refers to best solution of the objective function that outputs either maximum or minimum. Each bee in a certain population size refers to an iterative solution of the function. Scout bees look for food sources in a random way. Number of scout bees are one of the parameters in BA. Thus, random solutions are generated, and best
solution is located by algorithm. Initial size of neighborhood is another significant parameter in BA and refers to searching radius of the last optimum solution. Second set of bees are sent to this searching radius looking for a better solution. This loop continues until the solution converges to an optimum point (Pham, 2009). Algorithm is developed by using MATLAB software. The best solution of the multi objective function which is given earlier in equation (4) is found for three different population sizes. Each optimization is performed and finished for three repeats. Each repeat is performed with 300 iterations. To investigate the effect of number of bees; population size of 20, 60 and 120 is used. Population size is indicated by letter n. Optimum cutting parameters found by BA is given in Table 3.

Table 3. Optimum cutting parameters generated by bees algorithm. Result are given for population size of 20, 60 and 120. Number of iterations is 300 for each repeat.

<table>
<thead>
<tr>
<th>No. Of Repeat</th>
<th>Population Size</th>
<th>Cutting Speed, v (m/dk)</th>
<th>Feed Rate, f (mm/dev)</th>
<th>Cutting Depth, d (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 n=20</td>
<td>211.045</td>
<td>0.094</td>
<td>0.534</td>
<td></td>
</tr>
<tr>
<td>2 n=20</td>
<td>208.324</td>
<td>0.059</td>
<td>0.888</td>
<td></td>
</tr>
<tr>
<td>3 n=20</td>
<td>202.880</td>
<td>0.144</td>
<td>0.515</td>
<td></td>
</tr>
<tr>
<td>1 n=60</td>
<td>159.392</td>
<td>0.081</td>
<td>0.569</td>
<td></td>
</tr>
<tr>
<td>2 n=60</td>
<td>155.457</td>
<td>0.071</td>
<td>0.548</td>
<td></td>
</tr>
<tr>
<td>3 n=60</td>
<td>152.268</td>
<td>0.089</td>
<td>0.518</td>
<td></td>
</tr>
<tr>
<td>1 n=120</td>
<td>164.516</td>
<td>0.059</td>
<td>0.555</td>
<td></td>
</tr>
<tr>
<td>2 n=120</td>
<td>158.624</td>
<td>0.065</td>
<td>0.671</td>
<td></td>
</tr>
<tr>
<td>3 n=120</td>
<td>165.625</td>
<td>0.063</td>
<td>0.587</td>
<td></td>
</tr>
</tbody>
</table>

Optimum cutting speed found by bees algorithm is graphically represented in Figure 3 with error bars. Population size or number of scout bees is the control parameter. Feed rate and cutting depth is also given in Figure 4 and Figure 5 respectively.

Figure 3. Optimum cutting speed for population sizes of 20, 60 and 120.

Figure 4. Optimum feed rate for population sizes of 20, 60 and 120.
3.2. Performance Evaluation

Convergence performance is the key factor for evaluating effectiveness of any optimization method. Convergence is the ability to reach a constant value for fitness (objective) function after a certain number of iterations. Convergence is generally taken into consideration when comparing optimization methods as well. In this case, convergence performance of the bees algorithm is investigated in terms of population size and initial size of neighborhood. Effect of population size on main objective function is given in Figure 6. Initial size of neighborhood also affects the algorithm when reaching an optimum value. Initial size of neighborhood is indicated as ngh and given in Figure 7.

Figure 5. Optimum cutting depth for population sizes of 20, 60 and 120.

Figure 6. Convergence performance of the bees algorithm for population sizes of 20, 60 and 120.

Figure 7. Convergence performance of the bees algorithm for initial neighborhood sizes of 0.0001, 0.001, 0.01, 0.1.
4. CONCLUSION

In this study, optimization of cutting parameters in single pass turning operation is performed. Outputs of the objective function in this case are tool life and cutting force. Multi objective approach is preferred so as to optimize parameters simultaneously. Empirical model of the operation is developed and the Bees Algorithm is used as optimization method. Based on the results and calculations, following conclusion can be drawn:

- As can be seen in Figure 4 and Figure 5, maximum standard error is observed for population size of 20. As population size is increased, standard error decreased. As opposed to feed rate and cutting depth, standard error in cutting speed is approximate for all populations and is given in Figure 3.

- In Figure 6, it can be seen that convergence takes less time when population size increased. It is thought that, cause of this improvement can be great number of solutions for higher population. Global optimum point is reached much faster when number of random solutions is increased.

- In Figure 7, small sizes of initial neighborhood (ngh) resulted in poor convergence performance and divergence of global optimum point. Reason for this may be the insufficient number of iterations in a small search radius. On the other hand, if the radius is chosen too big, it has been seen that optimum solutions diverges from the actual point. An optimum size of neighborhood must be selected according to the nature of the process parameters. In this case, ngh can be selected as 0.01.

For further study; particle swarm optimization, ant colony optimization and genetic algorithm method can be used to compare performance of the different heuristic methods. Effect of other parameters and outputs in a turning operation such as tool geometry, type of cutting fluid and surface roughness of the workpiece can be included in optimization. Moreover, a hybrid optimization algorithm can be developed to improve the performance of the given method.

ACKNOWLEDGEMENTS

The research is supported and financed by Academics Training Program (ÖYP) with the project number 2014-ÖYP-103.

REFERENCES


A REVIEW OF AUXETIC STRUCTURES WITH CHIRAL CORES FOR MORPHING WING APPLICATIONS

Murat Ünal\textsuperscript{a†}, Dilan Kutmaral\textsuperscript{b†}, Metehan Altay\textsuperscript{c†}, Fatih Göncü\textsuperscript{d}

Department of Mechanical Engineering, Ankara Yıldırım Beyazıt University, Ankara, TURKEY
\textsuperscript{a}muratunal9696@gmail.com, \textsuperscript{b}dilankutmaral@gmail.com, \textsuperscript{c}mete.altay.1996@gmail.com, \textsuperscript{d}fgoncu@ybu.edu.tr

ABSTRACT
Morphing wings offer an attractive alternative to traditional high-lift flap mechanisms due to their advantage in adapting to different flight conditions and reducing turbulence, noise and vibrations \cite{1}. One way to obtain a morphing geometry is by replacing conventional spars with auxetic structures. In this work we review several studies, focusing on auxetic structures with chiral cores. In particular, the effect of dimensional parameters such as the ligament length and core radius on the trailing-edge deflection, lift coefficient and maximum stress are reviewed. Furthermore, the difference between circular and elliptical cores are highlighted. Finally, we present a new chiral elliptical core geometry and strategies to improve the performance of morphing wings.

Keywords: Morphing wing, auxetics, chiral, negative Poisson's ratio, elliptical core

† These authors contributed equally

1. INTRODUCTION
An airplane has to operate in different flight conditions during take-off, high altitude cruise and landing \cite{1,2}. Currently, aircraft wings are optimized for a limited number of flight conditions which reduces their efficiency. Birds, in contrast, can adapt the shape of their wings to different flight conditions to achieve the maximum performance. This observation has led researchers to devise various methods and materials such as shape memory alloys and piezoelectric composites to achieve morphing wings similar to birds \cite{1,2}. In this presentation, we will outline the geometries based chiral auxetic structures.

In industries where mobility is the main objective, reducing weight while maintaining superior structural properties is a primary concern of engineers and designers. In particular, this has led to the development and use of composite materials in aerospace structures where weight reduction is pushed to the extreme. Composite materials, in addition to lower weight have the advantage that they can be tailored for a specific application e.g. by reinforcing the structure with fibers in the direction of loading \cite{3}. However, current composites do not provide the best solution in every application. For example, they cannot achieve high shear strength and energy absorption and ease of repair \cite{3}. Auxetics, i.e. structures with negative Poisson’s ratio, can remedy to this problem as they can theoretically achieve infinite shear resistance. This can be easily illustrated by the equation of the shear modulus $G$ for an isotropic material:

$$G = \frac{E}{2(1+v)}$$

(1)

where $E$ is the elastic modulus and $v$ is the Poisson’s ratio. As the Poisson’s ratio approaches –1 the shear modulus tends to infinity. Consequently, other properties such as indentation resistance, and fracture resistance dependent on the negative Poisson's ratio tend to increase. Due to their aforementioned advantages, auxetics have attracted considerable interest for aerospace structures including morphing wing applications \cite{1,2,3,7}.

2. CHIRAL STRUCTURE
A specific class of auxetics are chiral auxetic structures which accomplish negative Poisson's ratio effect by the rotation of the chiral cores leading to unwrapping and wrapping of ligaments connecting neighbor cores. There are many advantages of using chiral structures in airfoils which can be grouped into aerodynamic advantages and structural advantages. Flow conditions could be improved by producing higher coefficient of lift with respect to symmetric rigid airfoil. Beside this advantage, drag could be minimized for greater Mach numbers. Chiral structures show morphing
airfoil property, which can be characterized as continuous change in shape without the need of traditional wing mechanisms. By virtue of this property, the need for flap mechanism can be eliminated and the handling and control of the aircraft could be improved. In terms of structural advantages; the compliant behavior of the chiral structure could be used in order to obtain cord wise bending. Auxetic structures can deform continuously and significantly without causing plastic deformations in the structure. This is advantageous as it allows reversible morphing which can be repeated any number of times. On the other hand, continuous deformations allow a smooth surface which improves aerodynamic efficiency. In addition to these properties, auxetics have structural advantages related to their shear resistance briefly mentioned in the previous section.

2.1. Applications

Researchers at Georgia Institute of Technology investigated the influence of the number of cells and the ligament length to core radius ratio \( L/R \) on the trailing edge displacement \([5,6]\). They found that greater lift coefficient and smaller trailing edge displacement can be obtained for larger \( L/R \) ratios and number of cells.

Budarapu et al. compared the response chiral structures with elliptical and cylindrical cores under various loading conditions \([7]\). They found that chiral structures with elliptical cores show better results in terms of trailing edge displacement.

3. NEW CHIRAL GEOMETRY

Inspired by previous research, we developed a new chiral geometry with elliptical cores as shown in Figure 1. In the future, we plan to characterize mechanical properties of structures with the new geometry and expect to observe negative Poisson’s ratio. Subsequently, we will explore the effect of several parameters such as the ligament length and core dimensions and the number of cores on morphing properties of wings. We also plan to experimentally test the performance of the new structure.

REFERENCES


Comparison of electricity generated by a new photovoltaic system based on fixed and single axis sun tracking system is concerned in this study. As it is known, studies on photovoltaic systems focus generally on increasing the efficiency and reducing the cost of the system. Therefore, a comparative experiment was done through the evaluation of a photovoltaic system for the cases of with and without a developed single axis sun tracker in this study. Measurement results obtained during a period of summer in Ankara show that the electricity generation of the photovoltaic system increases significantly when the single axis sun tracking system is utilized. As a result, when the efficiency of the system is taken into account, it can be seen that more electrical energy will be obtained from the solar panels per unit surface by means of the sun tracking system.

Keywords: Photovoltaic cell, sun tracking system, single axis sun tracking, efficiency.

1. INTRODUCTION

Sun tracking systems are used for maximizing efficiency of PV (photovoltaic) systems. The main aim of these systems is to catch the coming sun light at near to 90° with the PV cell plane. The amount of the direct sun light reaching to the PV solar panel is inversely proportional to obliquity factor which is cosine of the angle between the coming sun light and the solar panel plane [1].

The first tracker was built by Finster in 1962 and it was a mechanical system [2]. After him, Saavedra have developed a system which was controlled electronically and utilized Eppley pyrheliometer [3]. Starting from these dates, sun tracking systems have been widely studied because the energy gain provided by sun tracking might reach important percentages. Energy gain varies from 20% to 80% according to type of tracking system and some other factors like geographical location, season, and shading effect [4-7].

When solar panels convert light energy to electrical energy, one of the main factors affecting the efficiency is “angle of incidence”. It is the angle between coming sun light and the normal of PV panel plane and the efficiency increases when it closes to zero degree thus the best incident angle is 0° for the maximum efficiency. In other words, the best angle between the coming sun light and the PV plane is the right angle (90°) for the maximum efficiency. During a day, the angle of coming sunlight to the earth changes and a second change of incident angle on south-north direction shows up during the seasons due to the axial tilt of the earth. Over a year, two times about 23.5° which is the axial tilt value of the earth, totally about 47° of variation in incident angle occurs along south-north direction [8]. Therefore, the places which have higher latitude values cannot reach zero incident angle to the sunlight without sun trackers. However, sun tracking systems make it possible to catch the nearly zero incident angle on any latitude.

Sun tracking systems can be divided into two groups with respect to the degree of freedom as single axis and double axis tracking systems. Double axis tracking systems provide more freedom while single axis tracking systems can only rotate on a single direction which is generally east-west direction. As an example of how much beneficial a sun tracker system can be, in a study, measurements done on 30° north latitude showed that even one axis tracking systems can gain more energy from 15% up to 35% compared to the fixed PV system with the same characteristics [9].

Also, from another perspective, the sun tracking systems can be divided as passive and active sun trackers. While research studies about passive trackers remained limited with small numbers [10, 11], active sun tracking systems have been studied by many [12-19]. These active sun tracking systems are based on either real time (electro-optic) control by following the incident sun light or pre-saved astronomical information about the angles of a location on the earth with respect to the sun.

It was seen that the generated electrical energy of a PV system is increased when an electro-optic based one axis sun tracking system is utilized. The experimental procedure and results of the study are explained as follows.
2. EXPERIMENTAL PROCEDURE AND RESULTS

A prototype PV cell with the maximum power of 2 Watts was used in the experiment. An electrooptic based single axis sun tracking system was added to the system. After the PV system was set up, the output power measurements were done for two different cases which are fixed system and single axis sun tracking system to have comparable results. The measurements were done in Ankara which is at about 40° N latitude. Also, the PV system was tilted with two different angles which are 30° and 45° to have more comprehensible information about the effects of solar trackers. The effect of sun tracker is seen based on the measurements done in Figures 1 and 2.

![Real Time 30 Degrees Front Panel Tracking vs. Fixed System](image1)

**Figure 1.** Power-Hour graphs of tracking and fixed systems for 30°-tilted case [20]

While the maximum output power values are similar, the tracking systems have larger areas thereby higher total energies than the fixed ones on both graphs. Thus, sun tracking systems provide more energy than the fixed system cases.

![45 Degrees Front Panel Tracking vs. Fixed System](image2)

**Figure 2.** Power-Hour graphs of tracking and fixed systems for 45°-tilted case [20]

As known that the energy calculation is done by multiplying power and hour values in a graph, while the energy gain was about 35% in 30°-tilted case, it was about 50% in 45°-tilted case. Therefore, high amounts of energy gains were achieved through utilizing the electrooptical based sun tracking system.
In Figure 3, the system is with sun tracking and tilted with two different angles for 30° and 45°. There is not much difference between the measured power values for these two angles in Figure 3. However, output power values when the system is tilted with 45° were smaller than 30° case at noon hours. The reason is that the sunlight has nearly 0° of incident angle to the earth; thus, 45° of tilt angle in the system results some loss on output power at noon hours.

In Figure 4, 30°-tilted and 45°-tilted fixed system case results are given. 30°-tilted system has higher power values because it has smaller incident angles toward the sunlight in August which was the time of the experiment. Considering the declination of the sun is nearly 15° N in August [21], 30°-tilted PV system has about 5° of incident angle to the sunlight while 45°-tilted system has 20° of incident angle in Ankara which is at 40° N latitude. Therefore, 30°-tilted case has about 8% of higher output energy than that of 45°-tilted system according to the measured results.

3. CONCLUSION

In this study, a comparative experiment was done through the evaluation of a photovoltaic system for the cases of with and without single axis sun tracking. Measurement results of the study show that the electricity generation of the photovoltaic system increases significantly when the single axis sun tracking system is utilized. When the system is tilted with 30°, the energy gain becomes about 35% in Ankara and it even reaches to 50% if the system is tilted with 45°.
single axis sun tracking system in comparison to the fixed system. As a result, utilizing sun tracking systems increases significantly the efficiency of photovoltaic systems.

REFERENCES