# CFD simulation of turbulent convective heat transfer in rectangular mini-channels for rocket cooling applications

**Beg, OA, Zubair, A, Kuharat, S and Babaie, M**

<table>
<thead>
<tr>
<th><strong>Title</strong></th>
<th>CFD simulation of turbulent convective heat transfer in rectangular mini-channels for rocket cooling applications</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Authors</strong></td>
<td>Beg, OA, Zubair, A, Kuharat, S and Babaie, M</td>
</tr>
<tr>
<td><strong>Type</strong></td>
<td>Conference or Workshop Item</td>
</tr>
<tr>
<td><strong>URL</strong></td>
<td>This version is available at: <a href="http://usir.salford.ac.uk/id/eprint/47889/">http://usir.salford.ac.uk/id/eprint/47889/</a></td>
</tr>
<tr>
<td><strong>Published Date</strong></td>
<td>2018</td>
</tr>
</tbody>
</table>

USIR is a digital collection of the research output of the University of Salford. Where copyright permits, full text material held in the repository is made freely available online and can be read, downloaded and copied for non-commercial private study or research purposes. Please check the manuscript for any further copyright restrictions.

For more information, including our policy and submission procedure, please contact the Repository Team at: usir@salford.ac.uk.
CFD SIMULATION OF TURBULENT CONVECTIVE HEAT TRANSFER IN RECTANGULAR MINI-CHANNELS FOR ROCKET COOLING APPLICATIONS

Dr. O. Anwar Bég, Mr. Armghan Zubair, Miss Sireetorn Kuharat & Dr. Meisam Babie

Abstract

Heat transfer is one of the most critical aspects of the rocket propulsion design process. According to released heat, thermal loads are extremely large, and thermal insulation is frequently necessary in the motor combustion chambers and nozzles. In high temperature conditions, large thermal dilatation may present, and also the motor’s parts mechanical characteristics decreases. These occurrences are very important in the motor design process, and they are directly dependent from them temperature. This is the reason why precise heat transfer calculation is necessary. Non-eroding metallic throat inserts made with pure tungsten, tungsten-rhenium alloys, and tungsten-rhenium alloys, deoped with hafnium carbide are now common. Combustion gas temperatures can rise up to 3000 Celsius. Very high heat transfer rates from hot gases to the chamber wall must be designed for important research areas related to heat transfer of rocket nozzle include the internal and external heat transfer-coefficient predictions, metal temperature distribution, wall cooling methods, and ceramic coatings among others. Life extension of the nozzle, which consists of an expensive super alloy, is very effective for reduction of the running costs of a power generation plant. Accordingly, it is very important for the life assessment of the nozzle to predict the operating conditions and to establish a basis for the criteria of repair. In order to assess the life of the nozzle accurately, it is necessary to estimate its temperature distribution by prediction of the thermal environment. A cooling system is essential therefore in order to maintain engine integrity.

Validation

A mesh convergence study is given later to assess the optimum mesh density to collect accurate results. Hence, for this study an element size of 0.05mm was used to generate 575,120 number of elements to generate a turbulent flow model problem. Deploying a greater bias factor would increase the mesh density to the furthest edges of the channel which would prove to be useful. If the focus of the rectangular channel was just on a single side of the wall. Since a bulk temperature is involved in the calculations, it is essential to ensure a suitable bias factor is used to ensure the reliability of the results. Hence, this study we have opted to use a bias factor of 5 to allow greater mesh density at both edges of the channel – see below in Fig.2.3

Methodology

ANSYS FLUENT CFD single-phase, two-dimensional turbulent forced convection simulations. We have used the data provided by Forrest. The fluid enters the rectangular channel with a hydraulic diameter (Dhyd) of 3.79mm. Since the experiment considers turbulent flow, a Reynolds number of 50,433 and Prandtl number of 0.7 is used, as it corroborates with the experimental values Forrest (2014) was able to collect. Fig 1 shows the top section labelled as the isothermal length of 88.9mm and the channel height is 33.5mm. After the 88.9mm location from the datum (bottom of the channel) to 33.7mm, the section is labelled as the heated wall with a constant heat flux of 241.66 kW/m2. From 33.7mm to the 428mm section is considered isothermal. The aspect ratio of the channel is very high (28.5:1), and hence only 20 simulations are considered. Here we deploy the realizable k - e model available in ANSYS FLUENT. This turbulence model is one of the most popular used in the aerospace industry since it does not impact too heavily on computational power and can accommodate quite complex geometries and also heat transfer. The purpose of using k – ε is to develop a suitable eddy viscosity formulation and eddy dissipation equation. The Reynolds averaging model is used to be able to determine the governing RANS equations and the two model equations to solve the kinetic energy ‘K’ and the dissipation ‘ε’. Hence, the models take the following form for the turbulent kinetic energy:

\[ \frac{1}{2} \rho u'^2 = \frac{1}{2} \rho u'^2 \]

The CFD analysis is used to generate representative pressure, velocity and thermal fields. This grid refinement is conducted in the solver phase of the simulation to confirm adequate accuracy. The solver is set to include the double precision option to allow a higher accuracy and the parallel processing option is enabled to utilize the power of the multi-core system and the double GPU feature.

Contact

Dr. O. Anwar Bég & Miss Sireetorn Kuharat
Department of Aeronautical and Mechanical Engineering, University of Salford, Newton Building, Manchester, MS 6FW, UK.
Email: I. alahir@elev.info.salford.ac.uk & O.A.Beg@fluent.co.uk

Tenenile Aerospace Design, Dakota House, Coventry Airport, Coventry, CV9 2AZ, UK.
Email: armghan@gmail.com

Department of Petroleum and Gas Engineering, University of Salford, UK.
Email: meisamb@elev.info.salford.ac.uk

References