<table>
<thead>
<tr>
<th>Title</th>
<th>CFD simulation of turbulent convective heat transfer in rectangular mini-channels for rocket cooling applications</th>
</tr>
</thead>
<tbody>
<tr>
<td>Authors</td>
<td>Beg, OA, Zubair, A, Kuharat, S and Babaie, M</td>
</tr>
<tr>
<td>Type</td>
<td>Conference or Workshop Item</td>
</tr>
<tr>
<td>URL</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>Published Date</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 dilatations are 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 distribution. This is the reason why precise heat transfer calculation is necessary. Non-eroding metallic thrust inserts made with pure tungsten, tungsten-tungsten alloys, and tungsten-rhenium alloys, doped 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 channel 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.

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 mini-channel with a hydraulic diameter (Dywall) of 3.79mm. Since the experiment considers turbulent flow, a Reynolds of number of 50433 and Prandtl number of 0.69 is used, as it corroborates with the experimental values Forrest (2014) was able to collect. Fig. 1 shows the top section labelled as the square length of 88.9mm along the channel, held at 333.5K. After the 88.9mm location from the datum (bottom of the channel) to 339.7mm, the section is labelled as the heated wall with a constant heat flux of 241.66 kW/m². From 339.7mm to the 428.6mm section is considered isothermal. The aspect ratio of the channel is very high (28:1), and hence only 2D simulations are considered. Here we deploy the realistic 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-e 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 model takes the following form for the turbulent kinetic energy.

\[ \frac{\partial (\rho k)}{\partial t} + \nabla \cdot (\rho u_i k) = \nabla \cdot (\Gamma_k \nabla k) + \dot{W}_k - \rho \varepsilon \]

\[ \frac{\partial (\rho \varepsilon)}{\partial t} + \nabla \cdot (\rho u_i \varepsilon) = \nabla \cdot (\Gamma_\varepsilon \nabla \varepsilon) + \dot{W}_\varepsilon - C_{\varepsilon1} \frac{\varepsilon}{k} \dot{W}_k \]

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.

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 579,120 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 mini-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.

For case 1 of Quad mesh at 0.07 mm cell size, 206850 elements were produced whereas in Triangle mesh 344750 elements were produced. For the cell size 0.05 mm, in case 2, 579120 and 696950 elements were created for Quad and triangle mesh, respectively. In case 3, at 0.03 mm cell size, 965200 and 1008890 elements were generated in the Quad and Triangle mesh, respectively.

Results

A clear trend can be seen in Fig. 5 where a larger aspect ratio appears to provide greater Nußelt number characteristics across the same channel length. Both AR 20 and AR 28 follow the gradient however aspect ratio 20.1 provides a more suitable channel that will perform better cooling characteristics than the current channel.

Conclusions

This research sought to conduct an investigation into the turbulent flow in mini-channels to observe in a reliable manner Forrest's data, in order to evaluate the cooling performance at different aspect ratios. A successful model developed in ANSYS FLUENT with a mean average error of 5.97% relative to Forrest’s data collected at a Reynolds number 50,443 with a Prandtl number of 3.01. This suggests that the simulation model created for turbulent flow was suitable to set as a foundation for the study of different aspect ratios in the channel.

Multiple aspect ratios were also considered to understand the influence of high aspect ratios to analyse the best performing cooling channel, which was determined to be the highest aspect ratio channels. Hence, the ‘28:1’ aspect ratio provided the best characteristics and allow effective cooling. However, the limitations on mesh density and hardware have curtailed the sophistication achievable for the turbulence characteristics. LES and DNS could not be used, nor could the RANS FLUENT turbulence model. Also only linear rectangular channels were considered, i.e. curvature was ignored. Furthermore we only considered conventional water coolant.

From this CFD study the variation of aspect ratio provided a deeper appreciation of the effect of small to high aspect ratios with regard to cooling channels. Hence, when considering an application for the channel, the suitability of the aspect ratio must play a crucial role in optimizing cooling performance.

Further extensions to this study could include the use of nanoparticle doping to achieve better cooling efficiency by modifying the coolant thermal conductivity, viscosity etc. This constitutes a good pathway for future MCs and perhaps PhD students on the work reported here.

References


Contact

Dr. O. Anwar Bég & Miss Sireetorn Kuharat
Department of Aeronautical and Mechanical Engineering, University of Salford, Newton Building, Manchester, MS9 5HT, UK.
Email: O.A.Beg@salford.ac.uk & S.K.Kuharat@salford.ac.uk

Tenenau Aerospace Design, Dukeshouse, Coventry Airport, Coventry, CV8 3AZ, UK.
Email: armghan@uwe.ac.uk

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

ICHFTM 2018 : 20th International Conference on Heat Transfer and Fluid Mechanics