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>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>Publication title</td>
<td>WASET</td>
</tr>
<tr>
<td>Type</td>
<td>Conference or Workshop Item</td>
</tr>
<tr>
<td>USIR 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: library-research@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

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 useless. 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 use a suitable bias factor is used to ensure the reliability of the results. Hence, this study has been attempted 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 669150 elements were created for Quad and Triangle mesh respectively. In case 3, at 0.03 mm cell size, 965200 and 1088920 elements were generated in the Quad and Triangle mesh respectively.

Validation

Ideally the cooling channel should be capable of transferring heat towards the opposite side of the wall, which will enable a faster diminution of heat, thus preventing the temperature drop across the channel (and the velocity) will ensure a swift transition of heat out of the system. The smallest aspect ratio appears to absorb the smallest amount of heat but achieves feeding a larger capacity of fluid in comparison to the other aspect ratios. It is also evident that thermal boundary layer thickness also shows a correlation with an increase in aspect ratio, where the layer becomes thicker. A thicker thermal boundary layer would indicate that there is a greater convective heat transfer properties. Also the maximum temperatures reached for the channels show that AR1 attains 341.8K, AR10 reaches 341.4K and AR20 peaks with a temperature of 339.5K which suggests that the maximum temperature reached is a function of aspect ratio (three plots in Fig. 6) i.e. better cooling is attained.

Methodology

ANSYS FLUENT CFD single-phase, two-dimensional turbulent forced convection simulations. We have used the data provided by Forrest[1] The fluid enters the rectangular mini-channel with a hydraulic diameter (Dhyd) of 3.79mm. Since the experiment considers turbulent flow, a Reynolds number of 50,443 and Prandtl number of 6.6 is used as, it corroborates with the experimental values Forrest[2] was able to collect. Fig. 1 shows the top section labelled as the isothermal 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 (28.5:1), and hence only 20 simulations are considered. Here we deploy the realistic k - ε 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 model takes the following form for the turbulent kinetic energy

\[ \frac{\partial (K)}{\partial t} + \nabla \cdot (K \nabla u) = \nabla \cdot (\tau_{ij}) + \nabla \cdot \left( \frac{\varepsilon}{\kappa} \nabla \right) + \nabla \cdot (K \nabla' \varepsilon) \]

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, MS9 8HF, UK.
Email: S.Kuharat@sis.ac.uk & O.A.Beg@sis.ac.uk

Teneqic Aerospace Design, Dakota House, Coventry Airport, Coventry, CV9 3AZ, UK.
Email: armghan@gmail.com

Department of Petroleum and Gas Engineering, University of Salford, UK.
Email: mbbabie@gmail.com

References


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 20.4 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 collected at a Reynold's 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 channel. Hence, the ‘28.1 aspect ratio provided the best characteristics and most 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, selection of the aspect ratio must play a crucial role in optimizing cooling performance.

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

Fig. 1

CFD SIMULATION OF TURBULENT CONVECTIVE HEAT TRANSFER IN RECTANGULAR MINI-CHANNELS FOR ROCKET COOLING APPLICATIONS

Fig. 2

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 669150 elements were created for Quad and Triangle mesh respectively. In case 3, at 0.03 mm cell size, 965200 and 1088920 elements were generated in the Quad and Triangle mesh respectively.

Fig. 3

To further validate the CFD computation, we compare the ANSYS FLUENT results with a graph from Forrest’s data as the trend line used provides an approximation to his results – see Fig. 4. Using the R-squared function in Excel, it provides an estimate of how accurate the trend line is in relation to Forrest’s results that were collected. The R-squared function was R² = 0.9242 and hence suggests that the trend line is an accurate representation with a small margin of error present. This implies that the current model can be used as a suitable baseline to design multiple Aspect Ratios.

Fig. 4

Fig. 5

Fig. 6

Fig. 7

Fig. 8

Velocity Streamline Plots

Forrest’s data collected at a Reynold’s 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 channel. Hence, the ‘28.1 aspect ratio provided the best characteristics and most effective cooling. However, the ‘28.1 aspect ratio provides a more suitable channel that will perform better cooling characteristics than the current channel.

Fig. 9

References