

Computational Fluid Dynamics Modelling and Thermal Performance Analysis of an Aluminium Microchannel

Engr. Imran Hussain

Convener Mechanical Engineers Committee
Pakistan Engineers Association, Pakistan

Abstract - This study investigates flow behaviour and heat transfer in an open rectangular aluminium microchannel conveying deionized water using computational fluid dynamics. The aluminium microchannel was analysed for mass flow rates of 20, 40, and 80 g/s and heating powers of 250, 500, and 1000 W applied at the bottom wall. The study evaluates temperature distribution, velocity, pressure drop, wall heat flux, and thermal performance under the different operating conditions. A three dimensional model was developed using aluminium 2024-T4 as the solid material and deionized water as the fluid domain. Mesh independence was assessed using skewness and aspect ratio, and the CFD simulations were conducted for the selected operating conditions. The results show that increasing mass flow rate improves convective heat transfer and cooling effectiveness but also increases pressure losses. The results indicate balanced thermal and hydraulic performance at the intermediate mass flow rate, while the highest mass flow rate produces higher velocity and pressure penalties with comparatively small additional thermal improvement.

Keywords - computational fluid dynamics; aluminium microchannel; deionized water; heat transfer; mass flow rate; pressure drop; wall heat flux

1. INTRODUCTION

Enhancing thermal management in small systems like heat exchangers, microreactors, and electronic components requires research into single-phase heat transport in microchannels that are the flow properties of deionized water through an open rectangular aluminium microchannel (Aluminium 2024-T4), where controlled thermal gradients are produced by selective heating of the bottom wall, are the main focus of this study. The study intends to simulate and optimize heat transfer performance by altering flow rates and thermal inputs, which is essential for enhancing dependability and efficiency in contemporary engineering applications. In-depth examination of the heat transfer and fluid dynamics in this configuration offers important information for improving the microchannel design for increased cooling capacity and operational stability.

2. OBJECTIVE

- To analyse temperature distribution and flow behaviour at varying flow rates (20, 40, 80 g/s) and heating powers (250, 500, 1000 W) using CFD.
- To evaluate the thermal efficiency and heat transfer performance of the system under different operating conditions.

3. LITERATURE REVIEW

According to Sahar et al., ignoring wall conduction in microscale heat exchangers can lead to significant inaccuracies in heat transfer prediction—up to 50%. Additionally, the study noted a non-uniform distribution of Nusselt numbers along the channel, highlighting how important it is to appropriately simulate solid wall conduction in CFD simulations (Sahar, Madi et al., 2025). Yu and associates (2018): Solid wall conduction significantly pre-heats the fluid upstream, which lowers the local heat transfer rate, according to research on the impact of axial conduction in low Peclet number regimes. In microchannels with a high solid-to-fluid thermal conductivity ratio, this is especially important (Li & Tu, 2009).

In order to provide a conservative standard for heat transfer research, Bergman et al. (2011) offered analytical solutions for external laminar flow over a homogeneous flat plate. Even though they are idealized, these solutions might be used as a starting point, particularly when the model is first being validated. However, actual performance in this project is projected to be higher due to 3D effects and side wall conduction In 2024, Anjaneya et al. CFD results for a microchannel layout akin to Design 0 in this research

were presented by Alnaimat et al. (2024). Pressure drops, wall temperature gradients, and Nusselt number profiles were all analysed in the study. Additionally, it covered non-uniform grid space and mesh refinement methodologies, both of which are crucial for guaranteeing numerical correctness in the current investigation. offered a conservative standard for heat transfer research by providing analytical approaches for external laminar circulation over a homogeneous flat plate. Even though they are idealized, these solutions might be used as a starting point, particularly when the model is first being validated. However, because of side wall conduction and 3D effects, actual efficiency in this project should be higher.

CFD results for the microchannel layout akin to Design 0 in this research were presented by Alnaimat et al. (2024). Pressure drops, temperature at the wall variations, and Nusselt coefficient profiles were all analysed in the study. Additionally, it covered non-uniform grid spacing and mesh refinement techniques, both of which are crucial for guaranteeing numerical correctness in the current investigation. investigated methods for improving heat transfer in microchannel heat sinks and determined the impact of flows maldistribution and surface roughness. Their research supports the incorporation of geometry-specific thermal resistance modelling and sheds light on the effects of manufacturing in the real world (Ekkad & Singh, 2023). Minea (2012): Examined several CFD techniques used for transitioning and laminar flows in microchannels. According to the study, simulation reliability is significantly impacted by mesh quality, thermal boundary parameters, and conjugate heat transfer modelling (Zhang et al., 2023).

4. COMPUTATIONAL FLUID DYNAMICS MODELLING

An aluminium microchannel was subjected to CFD simulations in order to examine flow behavior and heat transfer. Three distinct mass flow rates—20, 40, and 80 g/s—were taken into account in the investigation. At the bottom wall, heated power levels of 250 W, 500 W, and 1000 W were applied. Under each circumstance, the Nusselt number, pressure drop, and temperature distribution were assessed. The findings support the evaluation of the system's thermal efficiency and design optimization for a range of operating circumstances.

4.1 Geometry Model

Using the draw, edit, and extrusion commands in Ansys Design Modeler, the geometry of a microchannel conveying deionized water in an open channel was modelled using aluminium 2024-T4 as a solid material. After that, the components were designated as solid and fluid parts for additional CFD processing. The channel's dimensions and shape are as follows

4.1.1 Solid part Aluminium

Width 12mm

Hight 06mm

Depth 60mm

4.1.2 Fluid Domain deionised water

Width 6mm

Hight 2mm

Depth 60mm

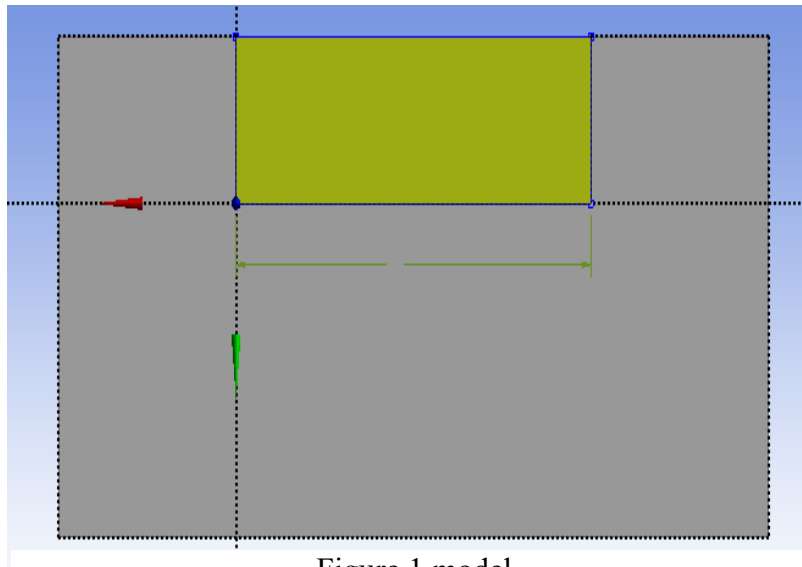


Figure 1 model

4.2 Mesh Metrics

ANSYS meshing tools were used to create a 3D mesh for the current CFD study, with a definite preference for fluid flow analysis. With an element size of 0.0004, 0.0003, and 0.00025 m, important flow areas and temperature barriers may be finely resolved. With roughly 50778, 76599, 142746 nodes and 245765, 377754, 714509 elements, the final mesh ensured a fair trade-off between computational cost and precision. The technique known as Sweep meshing, which works well with structured, extrudable geometries like microchannels, was used. This mesh configuration offers enough information to faithfully represent wall interactions, heat profiles, and velocity gradients in the simulation.

Table 1 Course Meshing Details

Mesh type	3D
Level	Course
Preference	CFD
Element size	0.0005m
Nodes	50778
Elements	245765

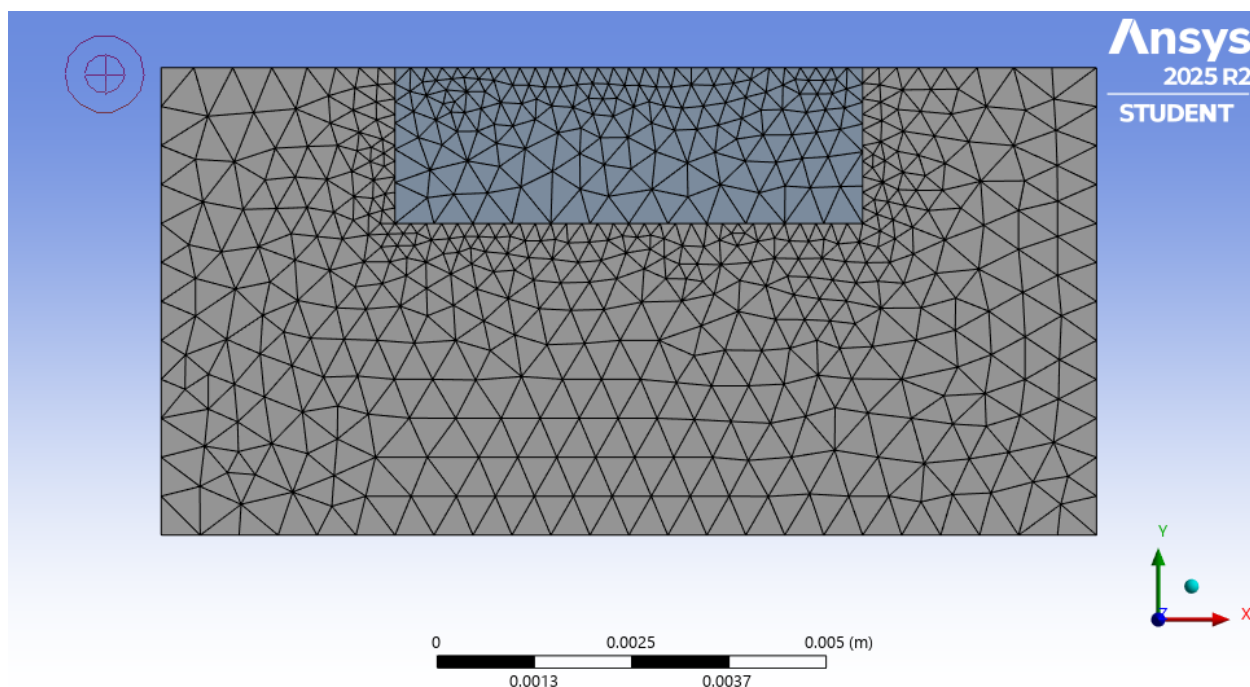


Figure 2 Front view coarse mesh

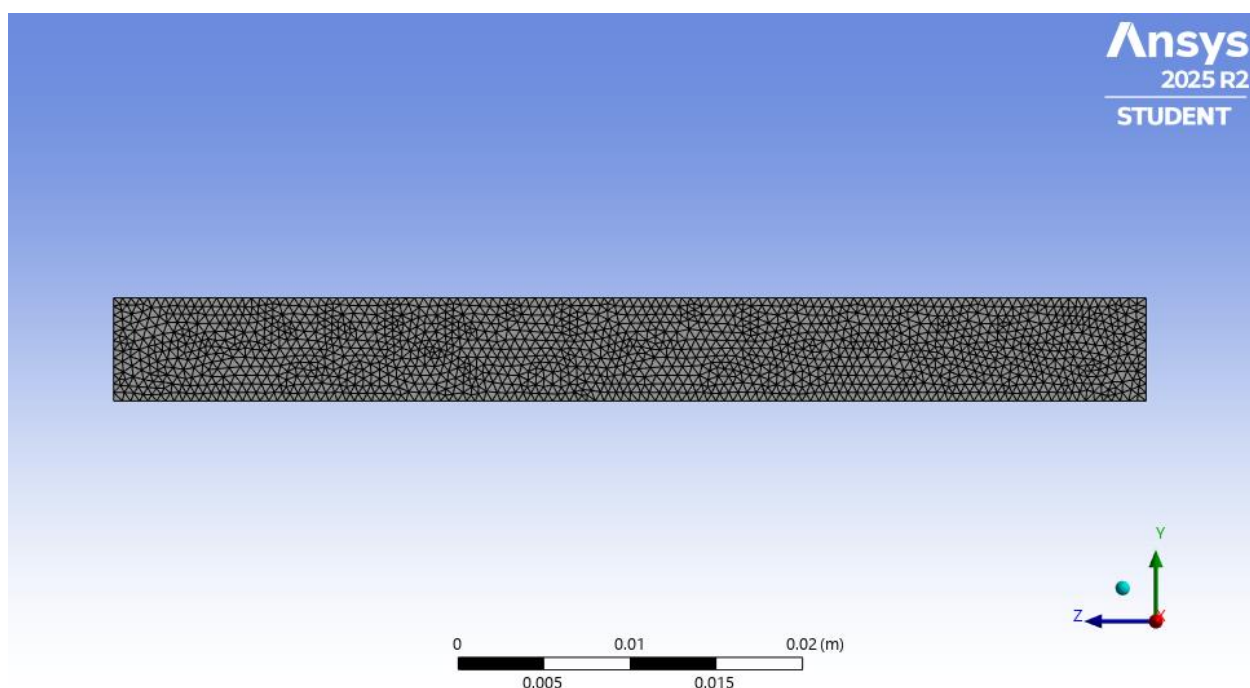


Figure 3 Side view coarse mesh

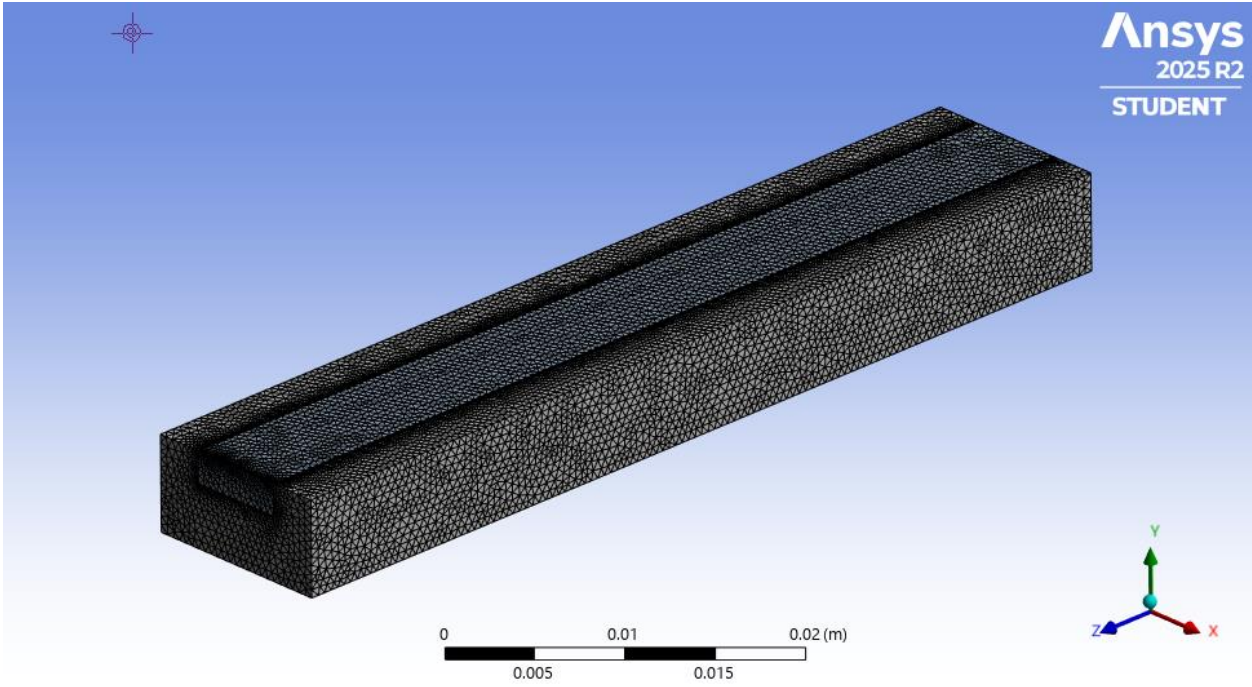


Figure 4 Isometric view Coarse mesh

Table 2 Medium Meshing Details

Mesh type	3D
Level	medium
Preference	CFD
Element size	0.0004m
Nodes	76599
Elements	377754

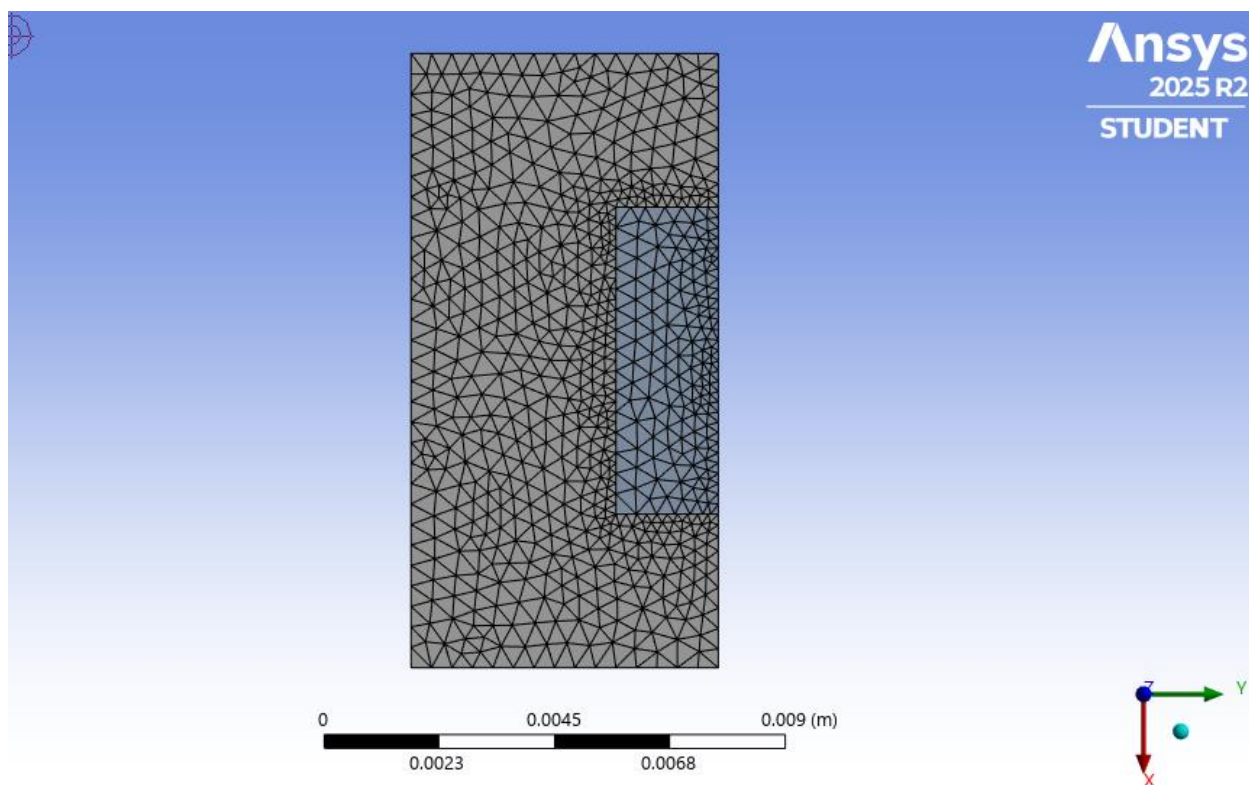


Figure 5 Medium mesh Front view

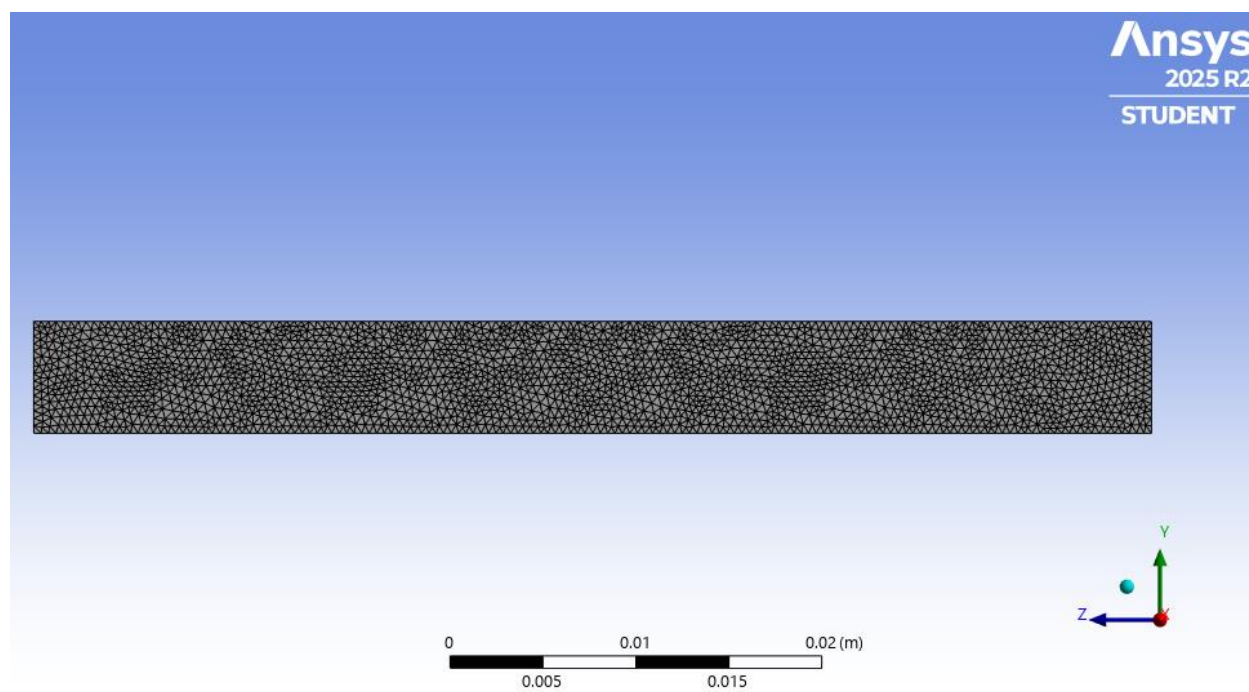


Figure 6 Medium mesh side view

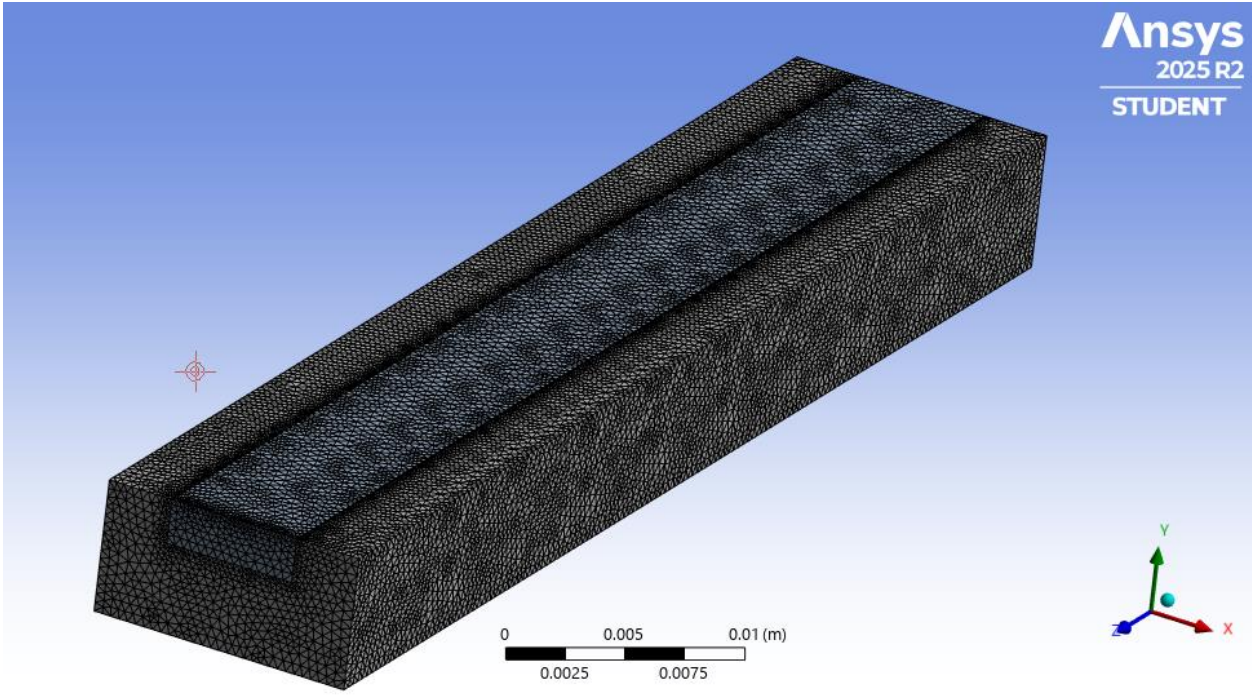


Figure 7 Medium mesh isometric view

Table 3 Fine mesh details

Mesh type	3D
Level	Fine
Preference	CFD
Element size	0.00025m
Nodes	142746
Elements	714509

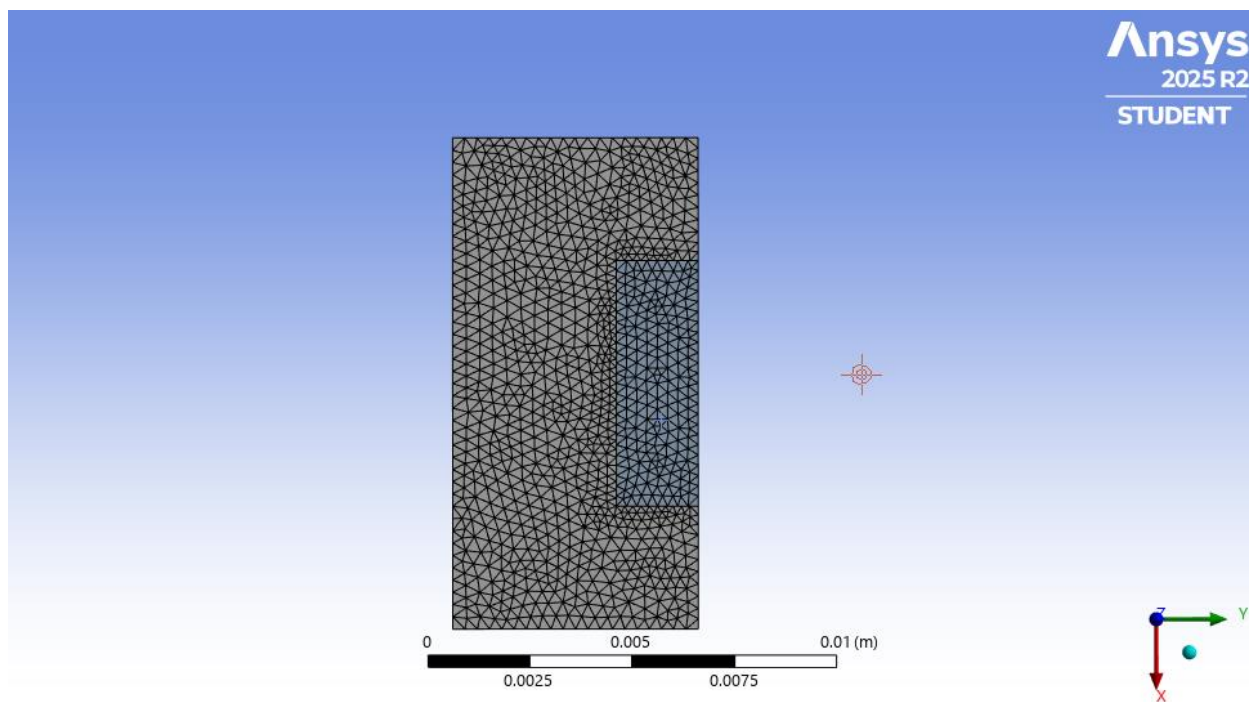


Figure 8 Fine mesh front view

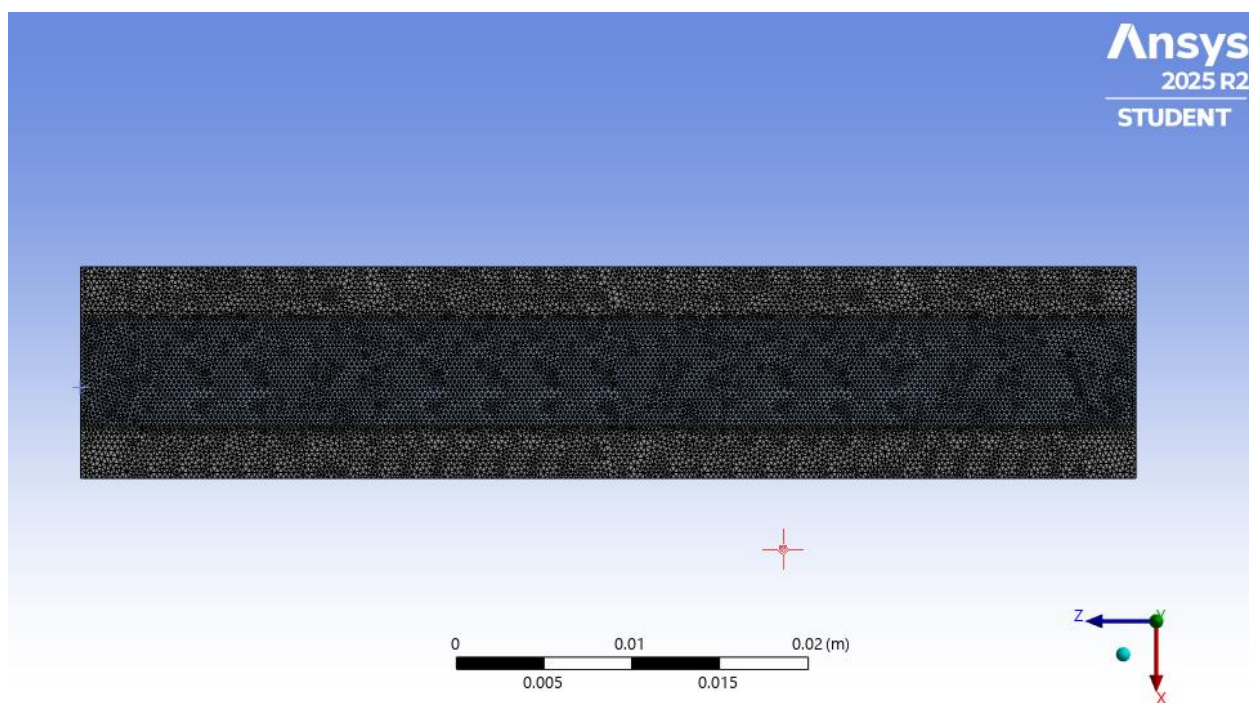


Figure 9 Fine mesh topside view

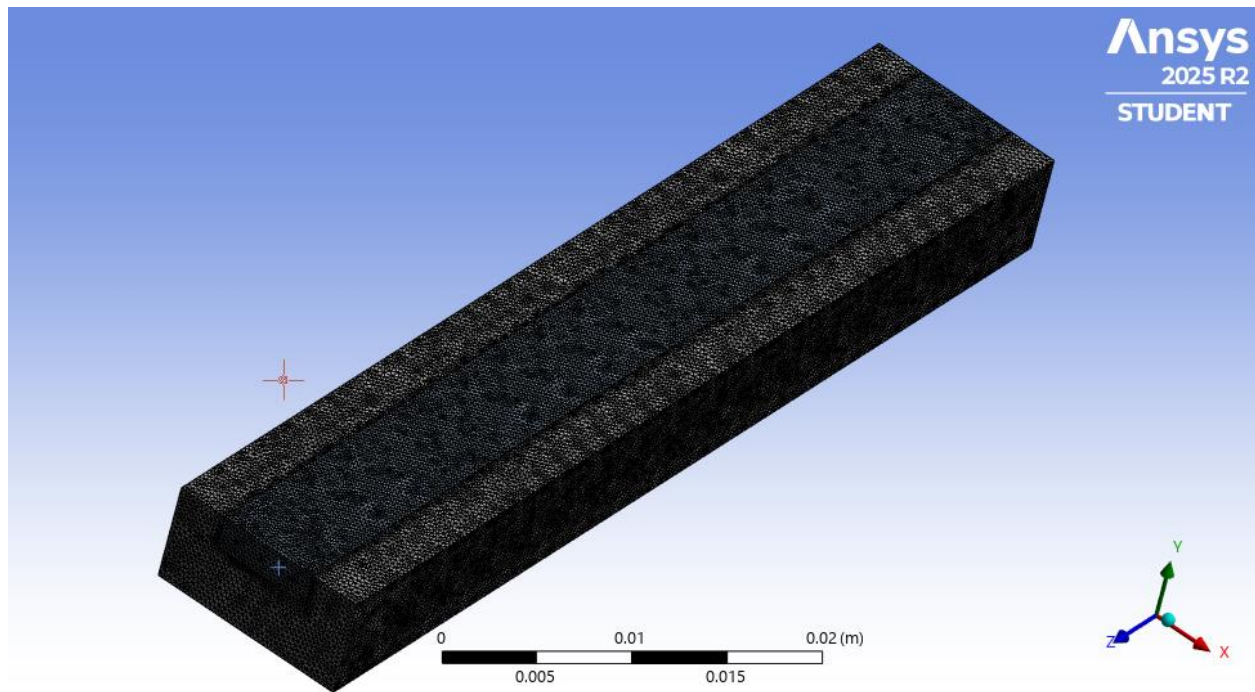


Figure 10 Fine mesh top view

4.3 Mesh independence test based on skewness and aspect ratio

The skewness histogram chart shows average skewness of 0.2314 and most of the elements are below 0.4 level of skewness element size was taken as 0.5mm for this mesh type. At edge sizing bias factor was take as 2 and no of divisions as 30.

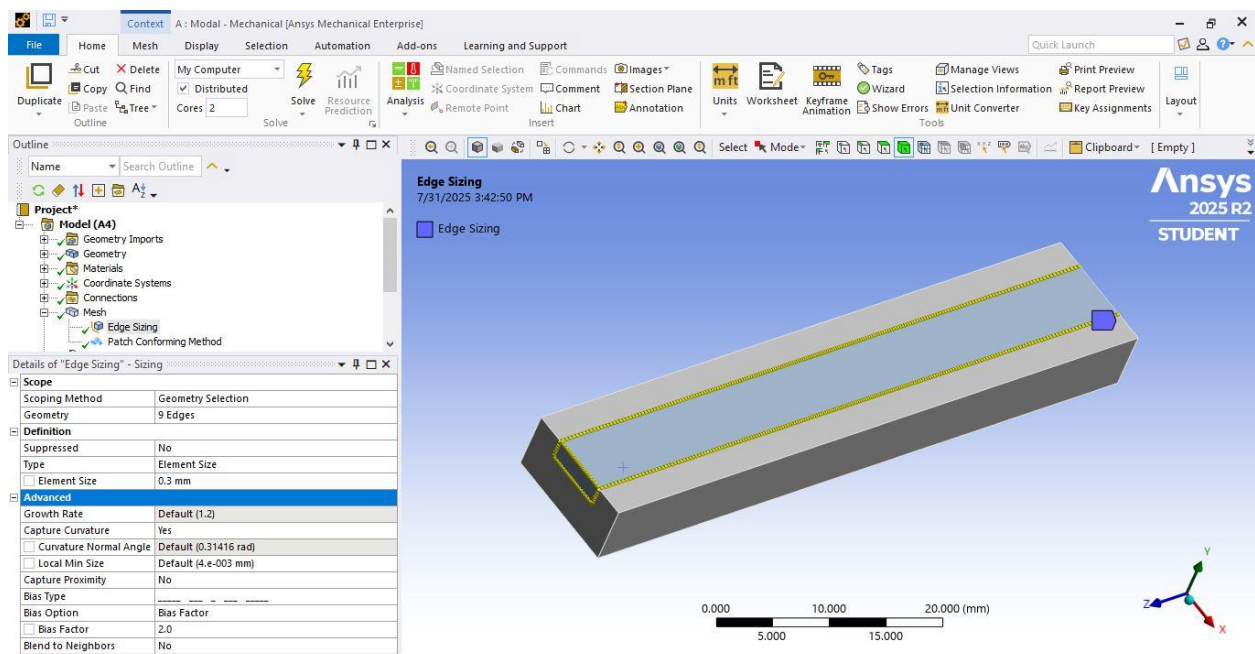


Figure 11 edge sizing details

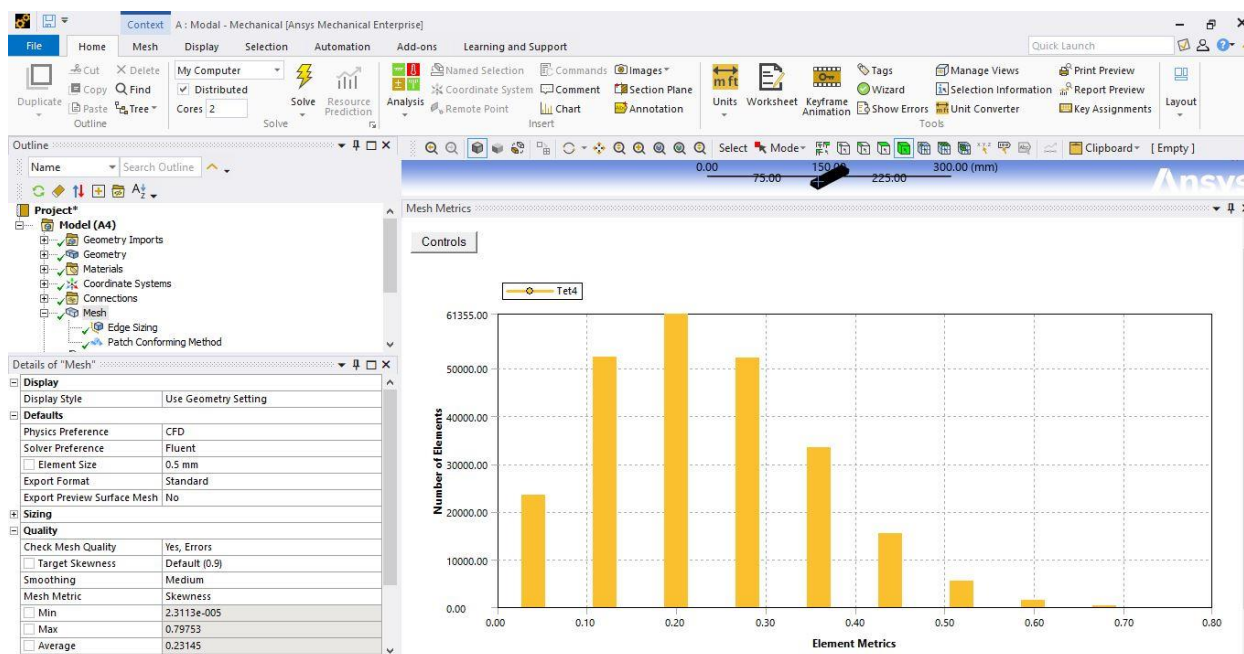


Figure 12 Skewness histogram plots Coarse Mesh

Medium mesh was created with element size 0.4mm and skewness was found as 0.2367 as average bias factor was again taken as 2 with 30 divisions

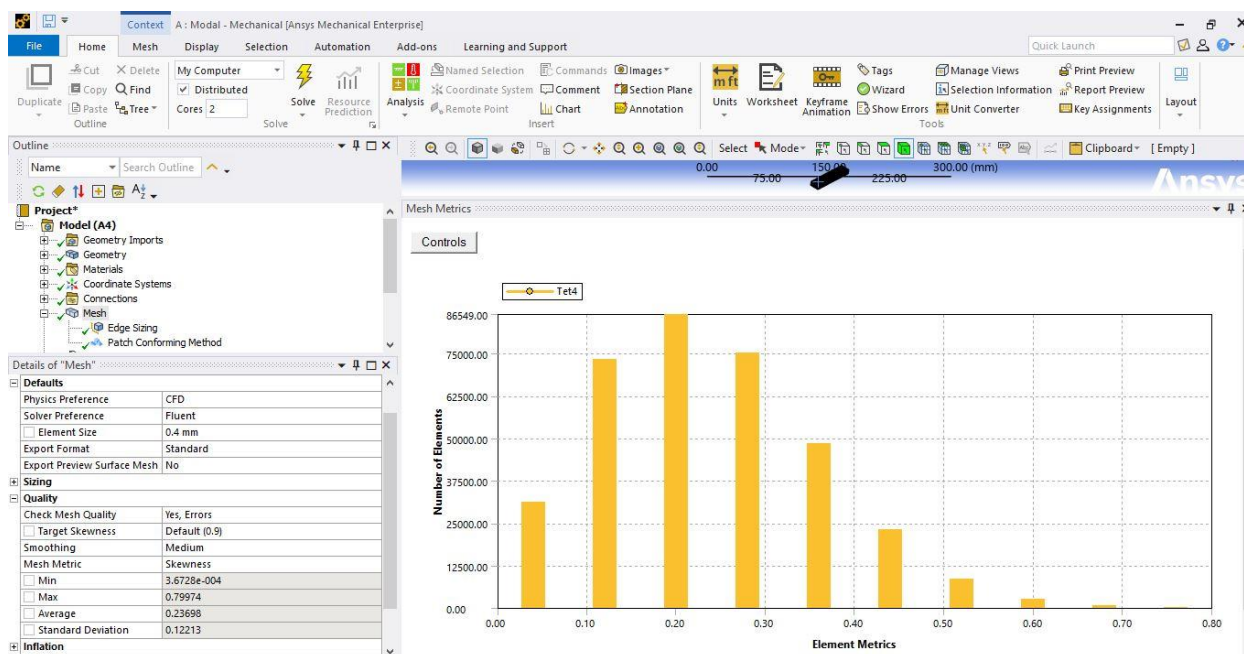
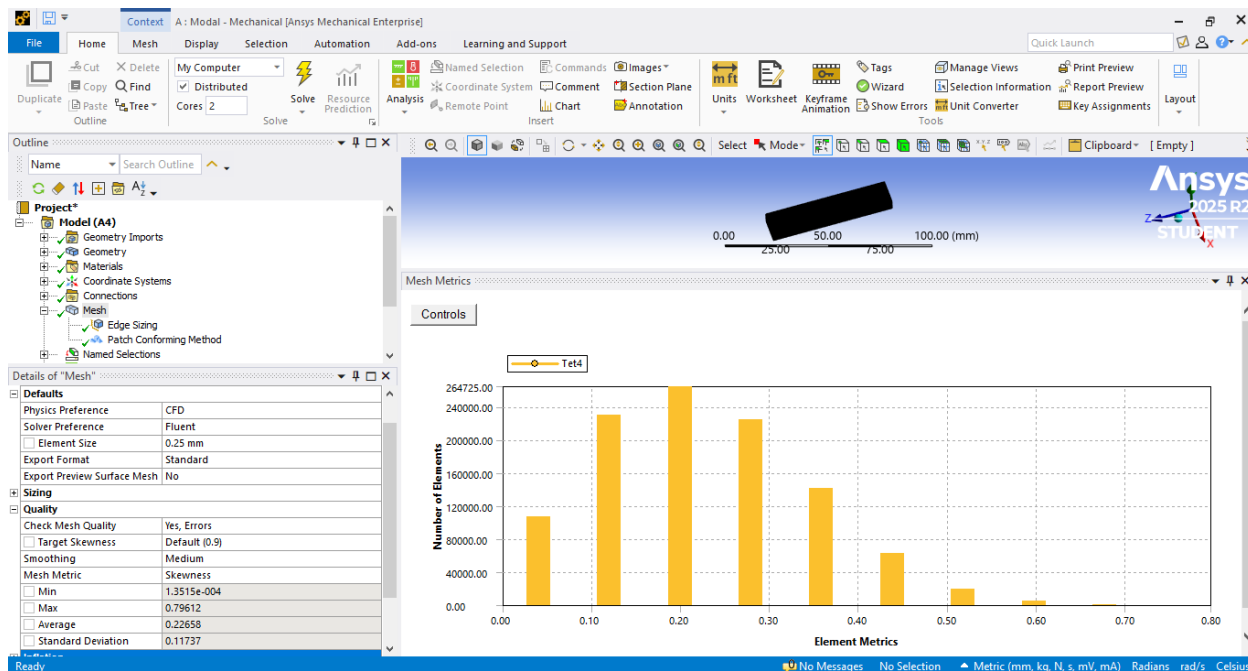


Figure 13 Skewness histogram plots medium mesh

Fine mesh was created with element size of 0.25 mm. the edge sizing was taken as 0.2mm and bias factor again as 2 with same number of divisions as 30. The average skewness was 0.225 which is good mesh quality.



Skewness histogram plots fine mesh

It is evident from the skewness histograms for coarse medium and fine mesh that more than 99% of the elements have skewness index less than 0.02 skewness value and the meshing successfully passes the mesh test to conduct flow and heat transfer simulations in ANSYS WORKBENCH 25R2

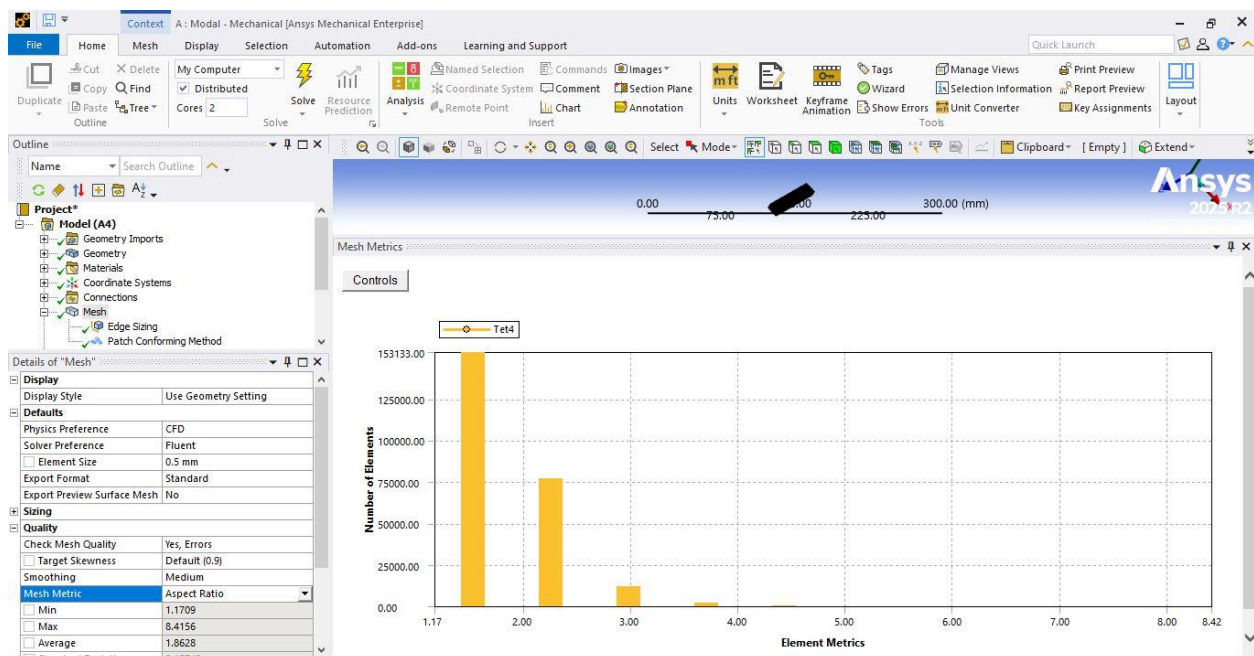


Figure 15 Aspect ratio Histogram Coarse Mesh

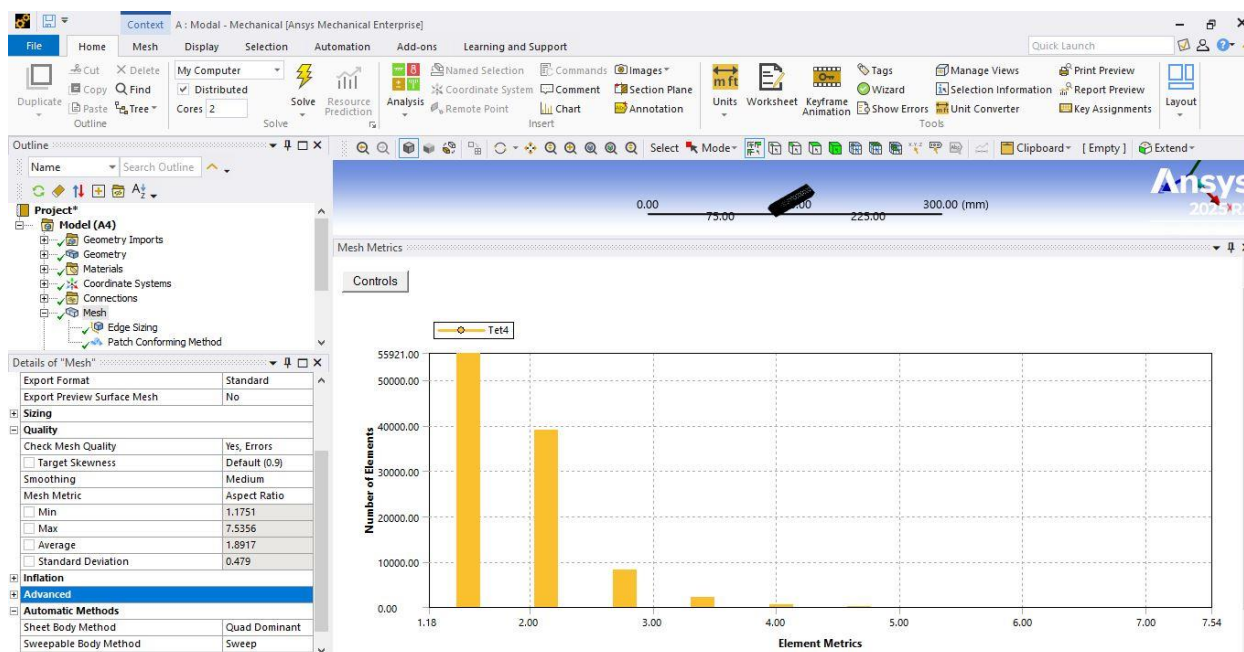


Figure 16 Aspect ratio Histogram Medium mesh

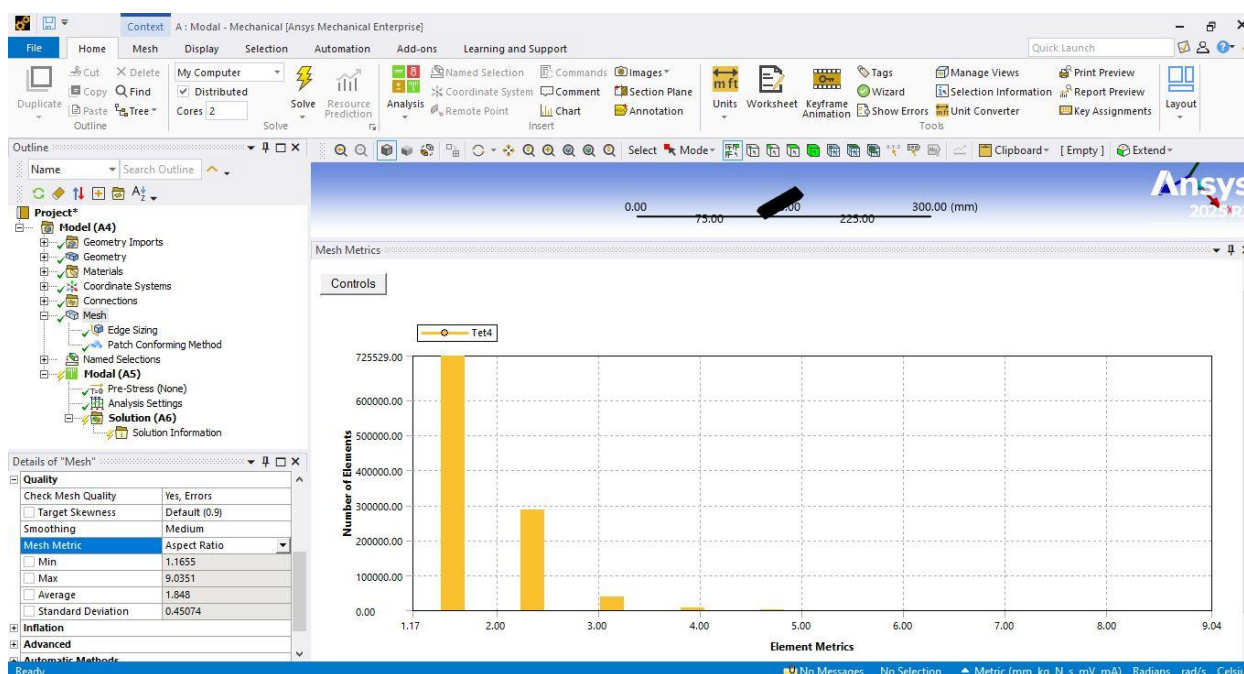


Figure 17 Aspect ratio Histogram plot fine mesh

The histogram charts in figure 14, 15 and 16 indicate the aspect ratio where all the element are below skewness index of 5 in coarse, medium and fine mesh which provides the basis to conduct CFD simulations on fine non uniform mesh.

Table 4 Zone wise mesh details

Sr No	Cell zone	No of nodes	Behaviour
1	Bottom Wall	2880	hard
2	Side Wall-1	1440	hard
3	Side Wall-2	1440	hard
5	Front Wall	240	hard
6	Back Wall	240	hard
7	Top Wall-1	720	hard
8	Top Wall-2	720	hard
9	Convection Wall	1440	soft
10	Inlet	48	soft
11	Outlet	48	soft

Table 5 Comparative Mesh table

Mesh type	Elements	Nodes
Coarse	245765	50778
Medium	377754	76599
Fine	714509	142746

Table 6 Nusselt number table

Mesh type	Nusselt number	%error
Coarse	4.974856322	0.5054%
Medium	4.962883142	0.2654%
Fine	4.949712644	0

Table 7 Velocity changes with Meshing

Mesh type	V _{min}	Error	V _{max}	Error
Coarse	0	0	1.6696721	0.599%
Medium	0	0	1.6796721	1.191%
Fine	0	0	1.6596721	0

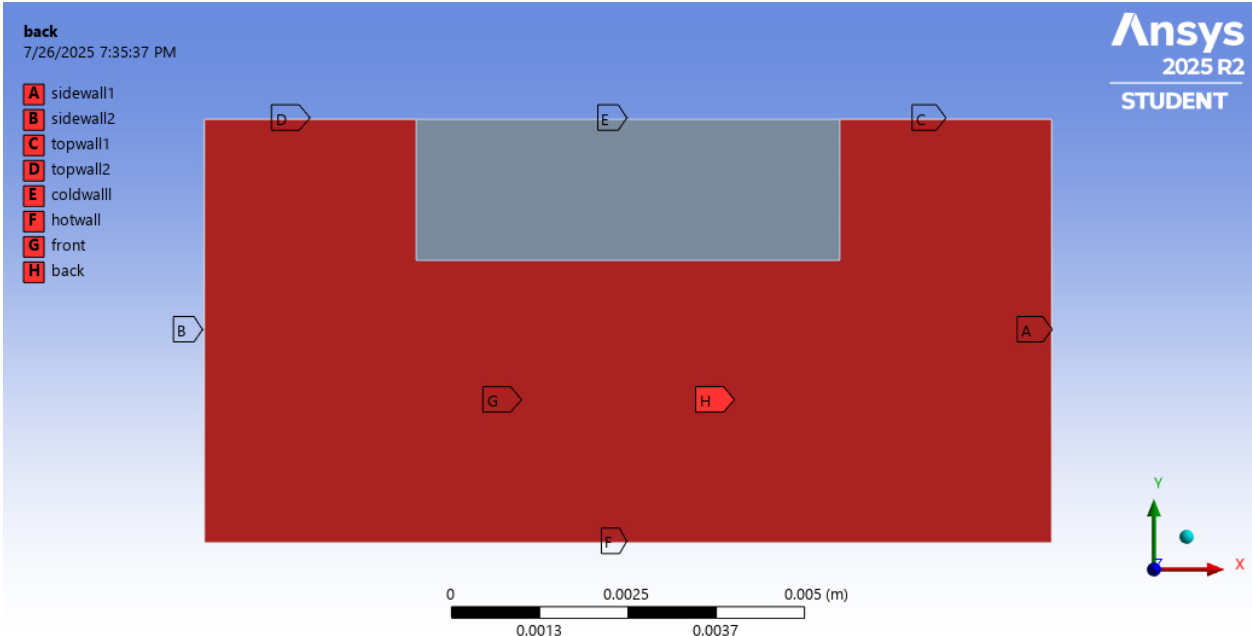


Figure 18 Name Selection for Boundary Conditions

4.4 Material selection

The properties of Aluminium 2024-T4 and deionised water such as thermal conductivity, heat capacity density and thermal coefficients were provided to create these materials in CFD design modeler materials the cell zones were then assigned to solid as well as fluid materials.

4.4.1 Aluminium Properties

Density	$2750 \frac{kg}{m^3}$
Heat capacity	$0.875 \frac{W}{MK}$
Thermal conductivity	$121 \frac{W}{MK}$
Expansion coefficient	$23.2 \cdot 10^{-6} \frac{1}{K}$

4.4.2 Denoised water properties

Density	$1000 \frac{kg}{m^3}$
Heat capacity	$4180 \frac{W}{MK}$
Thermal conductivity	$58 \frac{W}{MK}$
Expansion coefficient	$210 \times 10^{-6} \frac{1}{K}$

4.5 Boundary conditions

Table 8 Boundary conditions at cell zones

SR No	Cell zone	Boundary Condition
1	Bottom Wall	Heat Power: 250, 500, 1000 W Temperature 295K
2	Side Wall-1	Insulated Temperature 295K Momentum: Wall Motion: Stationary Wall
3	Side Wall-2	Insulated Temperature 295K Momentum: Wall Motion: Stationary Wall
5	Front Wall	Insulated Temperature 295K Momentum: Wall Motion: Stationary Wall
6	Back Wall	Insulated Temperature 295K Momentum: Wall Motion: Stationary Wall
7	Top Wall-1	Insulated Temperature 295K Momentum: Wall Motion: Stationary Wall Shear condition: No slip at the fluid wall interface
8	Top Wall-2	Insulated Temperature 295K Momentum: Wall Motion: Stationary Wall Shear condition: No slip at the fluid wall interface
9	Convection Wall	Heat transfer coefficient 200 W/MK Wall Motion: Stationary Wall Temperature 295
10	Inlet	Mass flow: 20 g/s, 40 g/s, 80 g/s Temperature 291K
11	Outlet	Pressure Ambient Temperature 295
12	Symmetry	Type: symmetry

4.6 Reasons and justifications for mesh and boundary condition selection

Based on the project requirements and relevant research articles, the selections for mesh size and boundary conditions were carefully chosen to ensure accurate simulation results in the CFD analysis of microchannel heat transfer. Two criteria were applied on all the three-mesh metrics for final selection of mesh. First Criteria was based on skewness level if 96% of the elements are in one group of sizing the mesh is taken as appropriate mesh sizing for accurate results. This criterion was fulfilled in fine meshing used for current study. 2nd criteria were based on aspect ratio as in the fine mesh the element size is smaller as compared to coarse and medium so minimum aspect ratio with a value of 0.1-0.2 was observed for 95% of the elements. At the fulfilment of both criteria a 3D mesh type was selected with an element size of 0.00025m, resulting in 305106 nodes and 276480 elements. This finer mesh resolution is essential for capturing detailed flow characteristics and thermal gradients within the microchannel, aligning with best practices recommended in similar studies ([1], [2]).

Boundary conditions were set to simulate realistic operating conditions of the microchannel. Distilled water was chosen as the working fluid with mass flow rates of 20, 40, and 80 g/s to investigate varying flow regimes. The fluid inlet temperature was fixed at 18°C (291K), reflecting typical laboratory conditions for experimental validation and ensuring consistency with previous studies (Reference: [3]). Heat flux values of 250, 500, and 1000 W applied to the bottom wall simulate different heat transfer intensities encountered in practical applications, facilitating a comprehensive analysis of thermal behaviour (Reference: [4]). Additionally, ambient temperature was set to 22°C to simulate standard laboratory environments, and a convective heat transfer coefficient of 200 W/m²·K was applied to model heat exchange with the surroundings. The no-slip condition at the walls ensures that fluid velocity is zero relative to the wall, which is typical for microscale flows. These selections were made to optimize the accuracy and reliability of the CFD simulations, ensuring that the results effectively characterize the heat transfer performance and fluid dynamics within the microchannel under varying operational parameters.

Table 9 Solution methods

1	Formulation Type	Pressure based
2	Gravity	9.81m/s ²
3	Velocity Formulation	absolute
4	Time	Steady
5	Planner	2D
6	Viscous model	Laminar
7	Energy	on
8	Gradient	Least Squares Cell Based
9	Scheme	Coupled
10	Pressure	2nd order upwind
11	Momentum	2nd order upwind
12	Energy	2nd order upwind

4.7 Convergence Residual Criteria

Continuity	0.001
x-velocity	0.001
y-velocity	0.001
z-velocity	0.001
Energy	1e-06

Convergence of residuals

iter	continuity	x-velocity	y-velocity	z-velocity	energy	time/iter
177	1.1220e-03	4.2088e-07	4.1721e-07	1.7255e-06	1.6861e-08	0:01:19 23
178	1.1020e-03	4.1592e-07	4.1166e-07	1.6928e-06	1.6888e-08	0:01:16 22
179	1.0813e-03	4.1033e-07	4.0465e-07	1.6670e-06	1.6865e-08	0:01:12 21
180	1.0630e-03	4.0476e-07	3.9823e-07	1.6415e-06	1.6859e-08	0:01:09 20
181	1.0437e-03	3.9970e-07	3.9243e-07	1.6205e-06	1.6815e-08	0:01:05 19
182	1.0239e-03	3.9444e-07	3.8722e-07	1.5985e-06	1.6885e-08	0:01:02 18
183	1.0044e-03	3.8845e-07	3.8136e-07	1.5767e-06	1.6898e-08	0:01:00 17
184	9.8598e-04	3.8003e-07	3.7545e-07	1.5543e-06	1.6927e-08	0:00:57 16

! 184 solution is converged

Writing "| gzip -2cf > SolutionMonitor.gz"...

5. CFD Simulation results at 20 g/s

Using an aluminium microchannel with water entering at 291 K, ambient temperature at 295 K, and a heat transfer coefficient of 200 W/m-K, a CFD analysis was performed to assess thermal performance under various mass flow rates (20, 40, and 80 g/s) and heat fluxes (250, 500, and 1000 W). The findings demonstrated that convective heat transfer was much improved by raising the mass flow rate, which decreased wall temperatures and increased cooling effectiveness. Similar to this, larger heat fluxes produced larger thermal gradients, which accelerated the rate of heat dissipation because of the stronger temperature differentials but also raised the demands for heat removal. In order to maintain stable temperature profiles, the heat transfer coefficient was essential in controlling the interaction between the microchannel and its surroundings. All things considered, the study shows that heat flux and mass flow rate are important determinants of microchannel thermal behaviour, providing insightful information for improving microscale cooling systems.

Table 10 Console computed results at 20g/s

			250 W		500W		1000W	
Zone	Total Pressure Pa	Velocity m/s	Temperature K	Heat flux W/M^2 K	Temperature K	Heat flux W/M^2 K	Temperature K	Heat flux W/M^2 K
Back	0	0	295	15570.351	294.99611	15291.925	294.76563	2.6890354e-05
Cold wall	802.66352	0	291.06543	0	291.07767	0	291.07761	0
contact_region1	0	0	294.6996	24246.627	294.70863	24268.004	294.68868	24144.262
contact_region2	767.50579	0	294.69907	24246.619	292.30927	24267.995	292.3017	24144.252
Front	0	0	295	9592.9914	294.99764	9304.314	294.99755	9617.4965
Hot wall	0	0	294.84561	31.776515	294.85291	500	294.83002	8.3317453e-05
inlet	2647.1528	1.6696721	291	0	290.99998	0	290.99991	0
outlet	1522.3583	1.6713782	291.27217	0	291.27239	0	291.27125	0
sidewall1	0	0	295	10014.377	294.99736	9664.5347	294.99702	10905.741
sidewall2	0	0	295	9988.4004	294.99732	9638.4841	294.99697	10877.567
topwall1	0	0	295	16136.124	294.99611	16029.611	294.9959	16839.947
topwall2	0	0	295	16166.444	294.99605	16059.865	294.99584	16869.717
Net	228.1668	0.011441953	294.43461	4151.8656	294.02864	4155.5257	294.01517	4134.3533

5.1 Discussion on simulation results at 20 g/s and 250, 500, 1000W

Across all heat flux levels, the maximum velocity stayed comparatively consistent at around 2.23 at the mass flow rate of 20 g/s. Pressure was observed as 1.359kPa by maximum for mass flow rate of 20 g/s. This suggests that at this ow flow rate, the additional thermal input had no impact on the flow resistance. It is interesting to note that as the applied heat flux increased, the maximum wall heat flux declined somewhat from 9.35×10^5 to 9.750×10^4 W/m²K. This is probably because of localized temperature saturation and diminished thermal gradients at the wall. Constrained convective enhancement at lower flow rates was indicated by this flow regime's overall reduced ability to respond dynamically to increasing thermal loads.

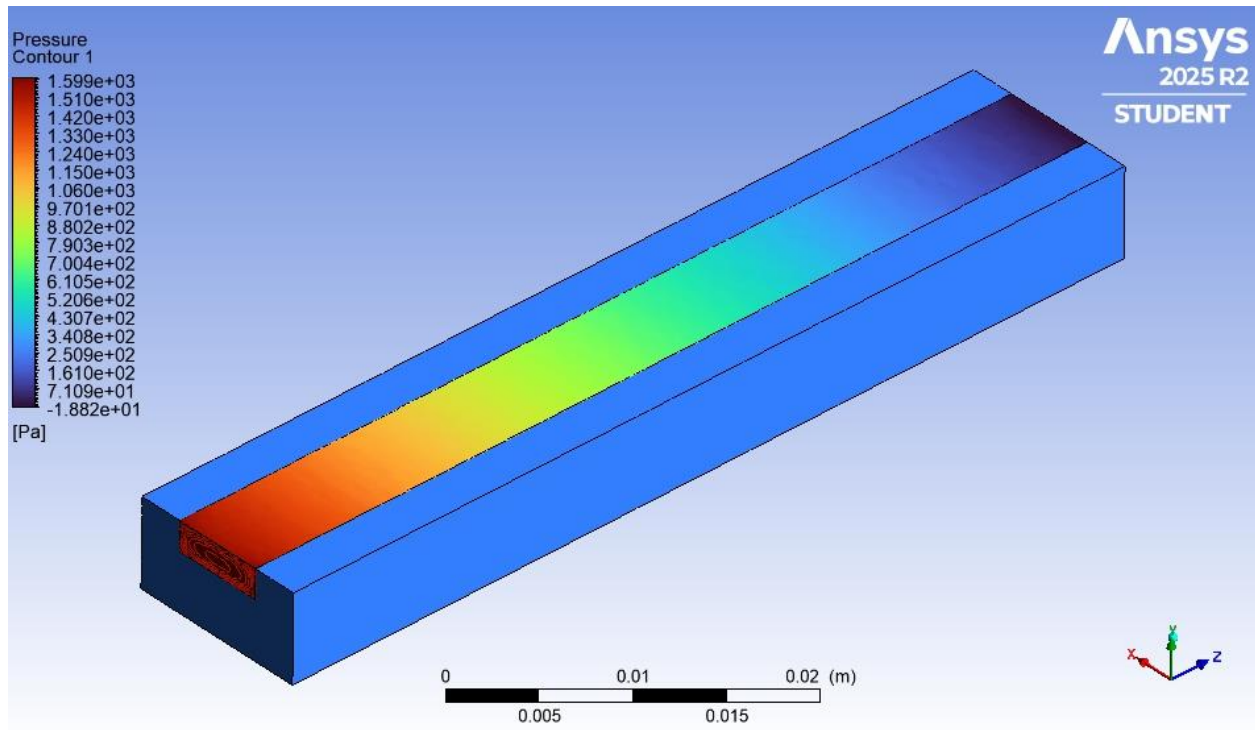


Figure 19 Pressure At 20 g/s, 250W

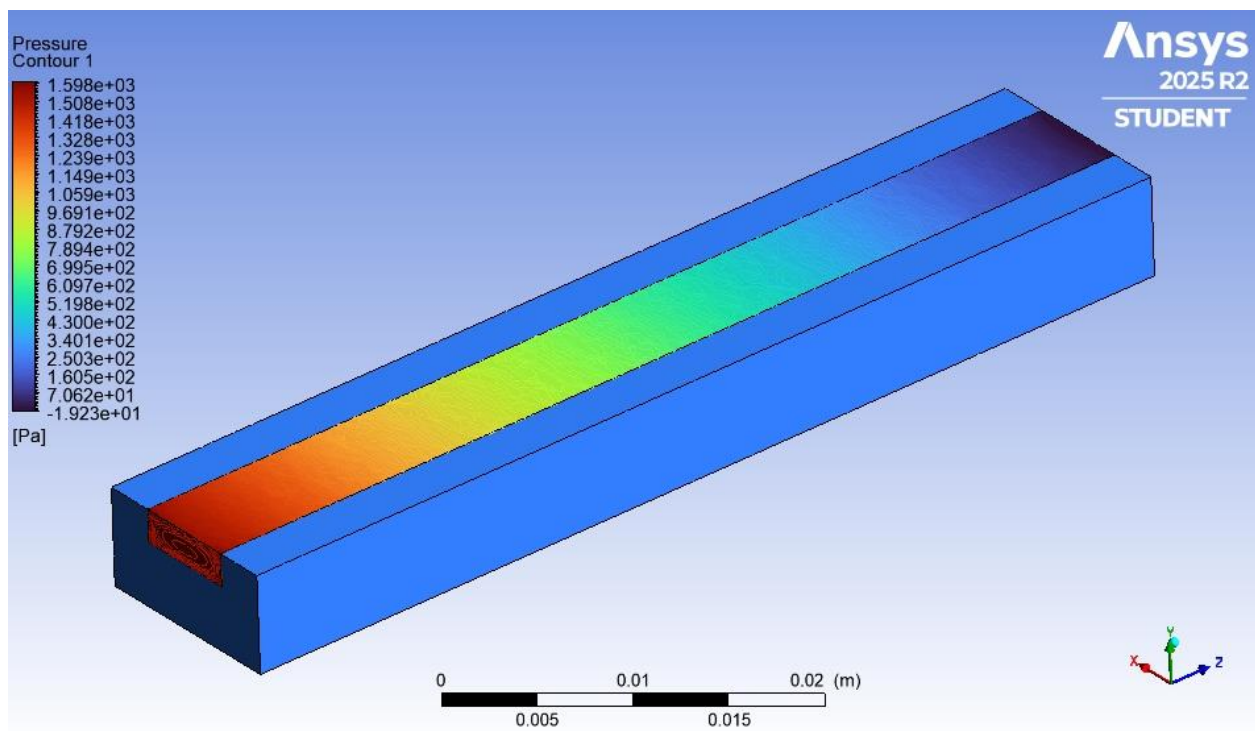


Figure 20 Pressure At 20 g/s, 500W

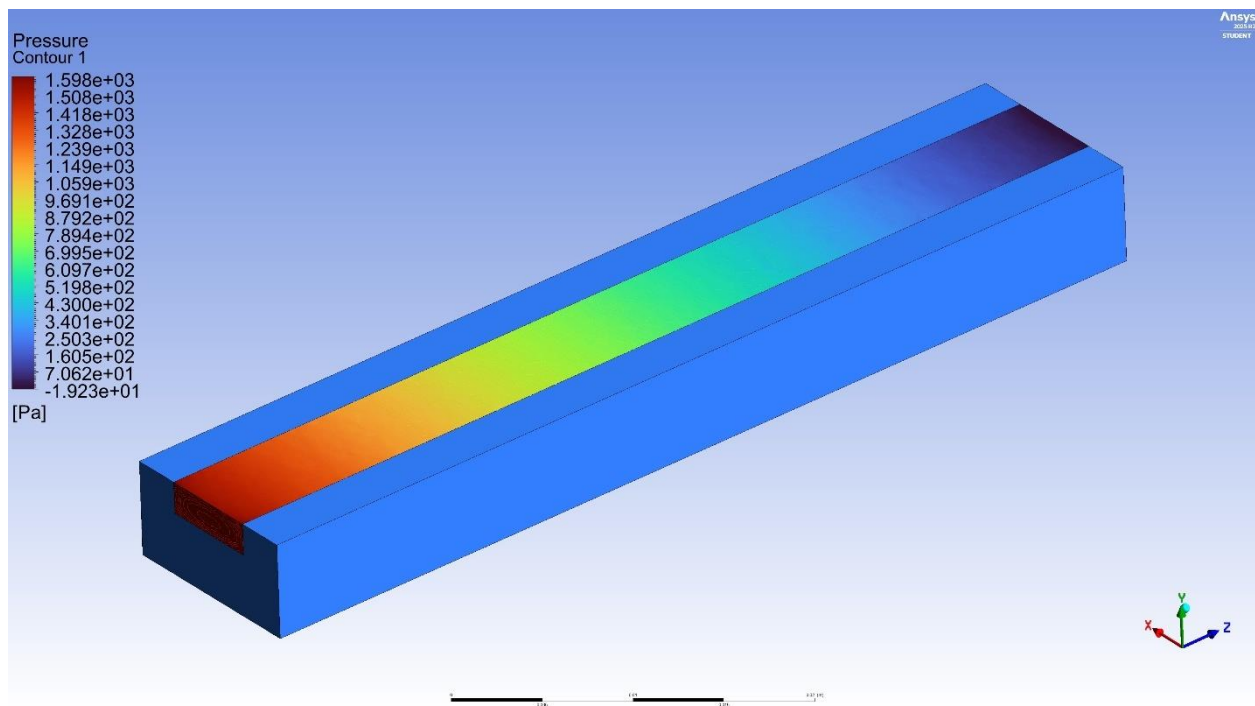


Figure 21 Pressure at 20 g/s 1000W

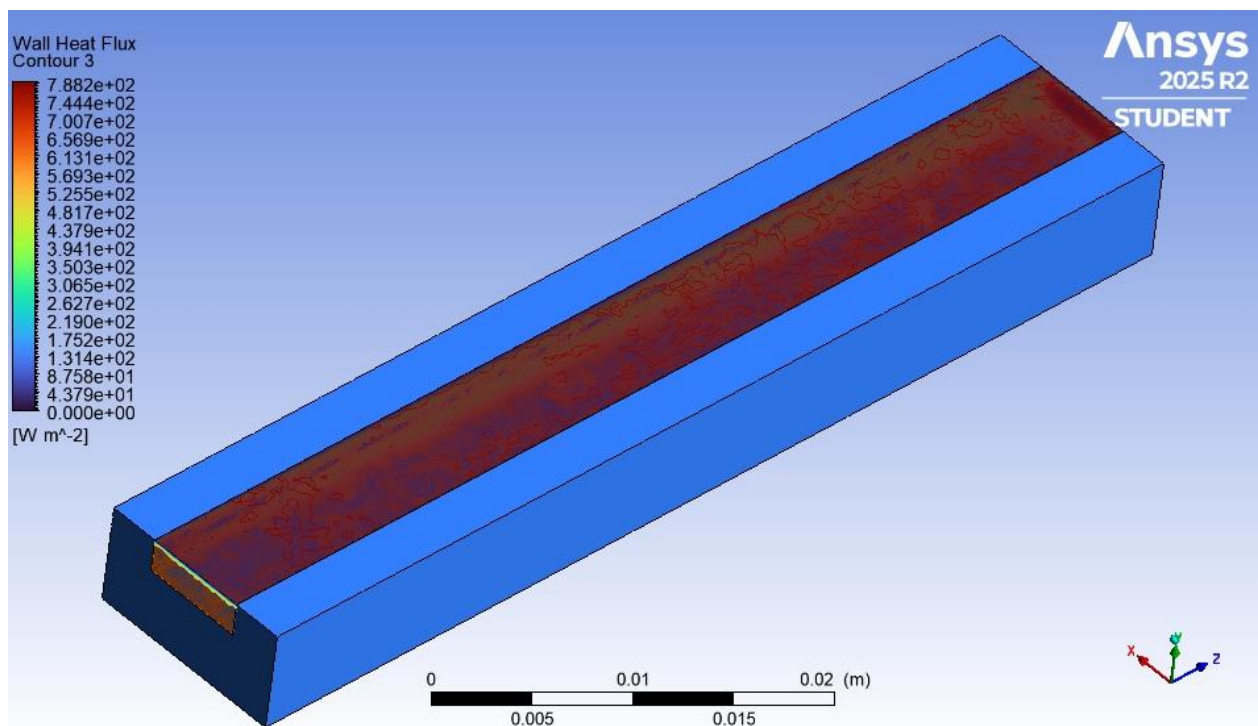


Figure 22 Heat flux at 20 g/s, 250W

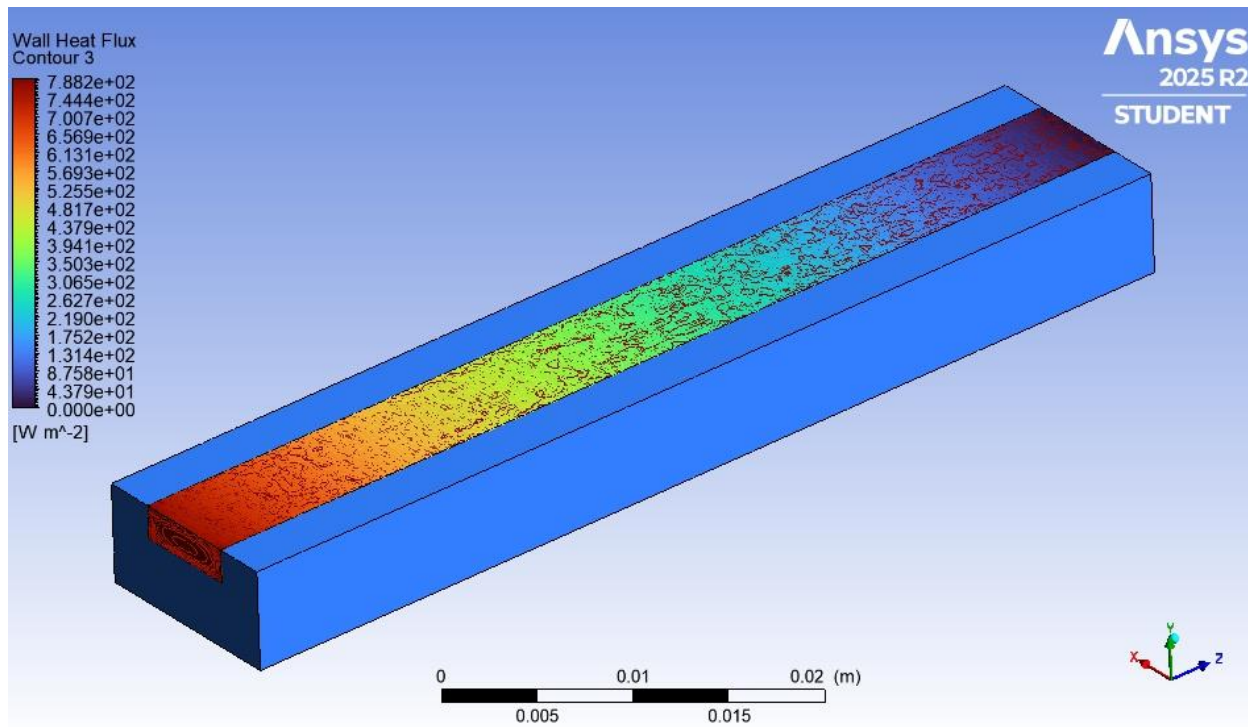


Figure 23 Heat flux at 20 g/s, 500W

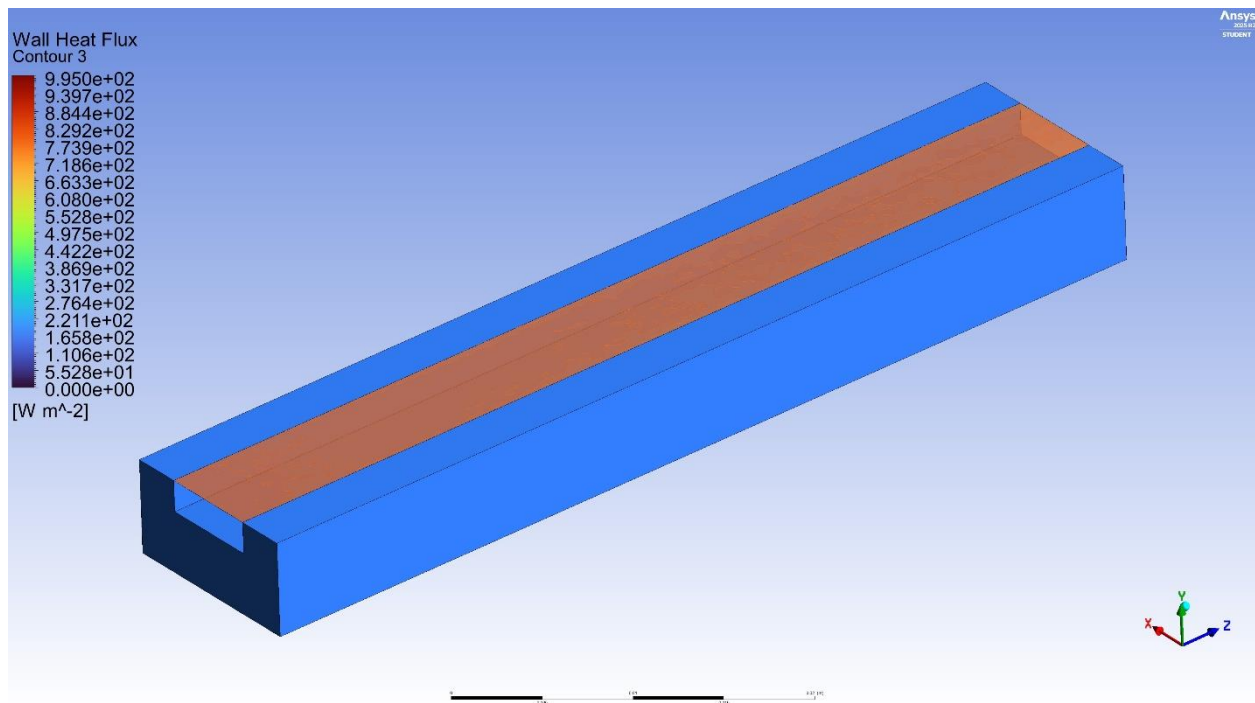


Figure 24 Heat flux at 20 g/s, 1000W

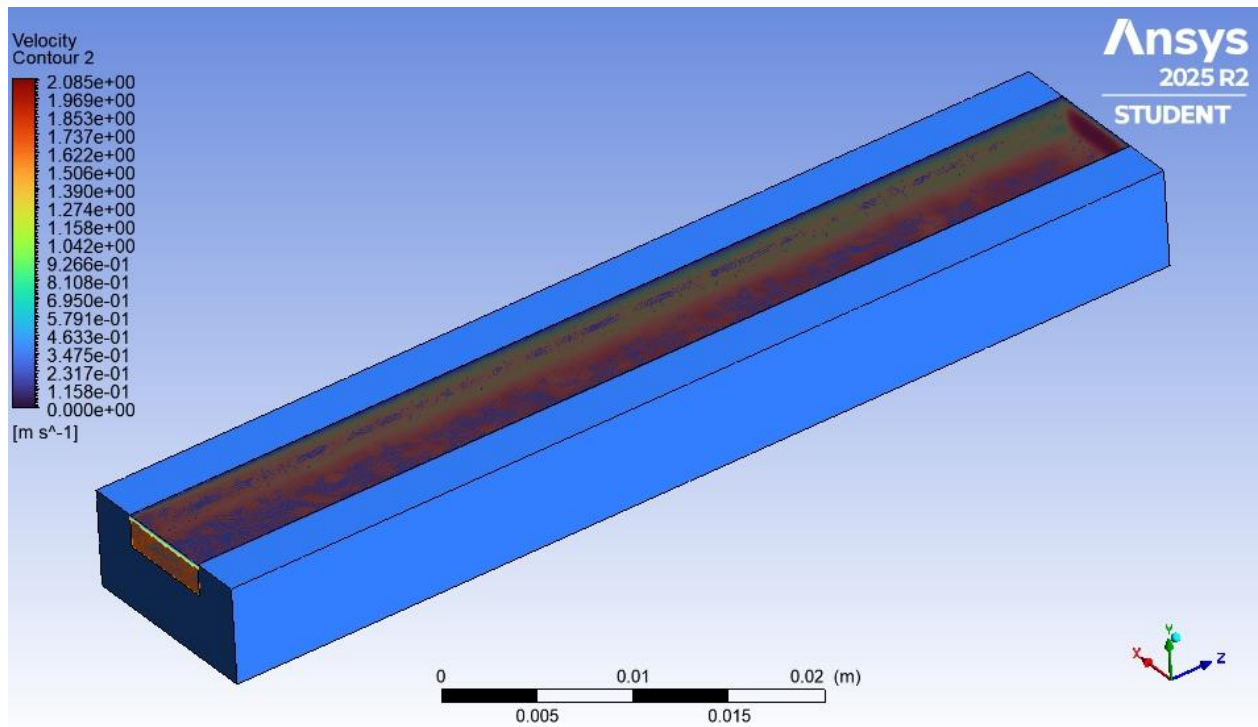


Figure 6 Velocity Streams-1 at 20 g/s, 250w

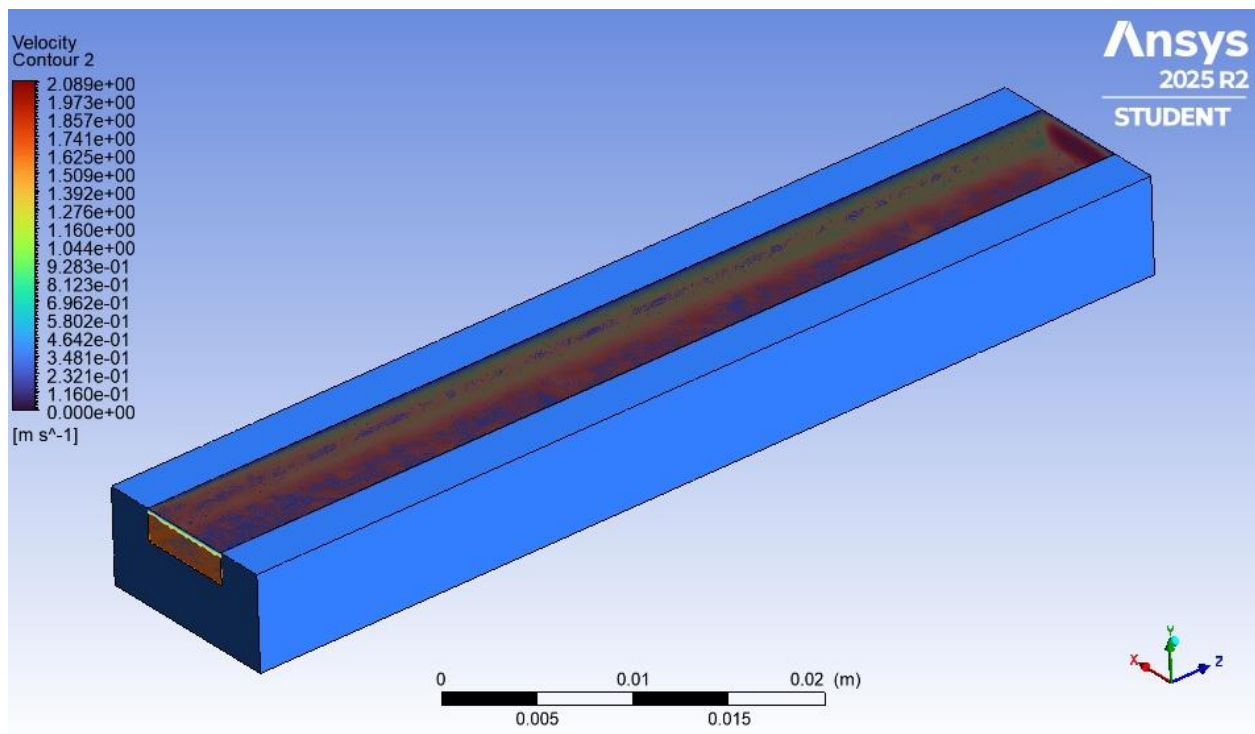
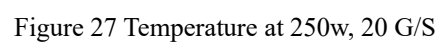
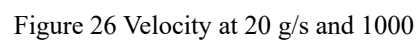


Figure 25 Velocity at 20 g/s and 500W



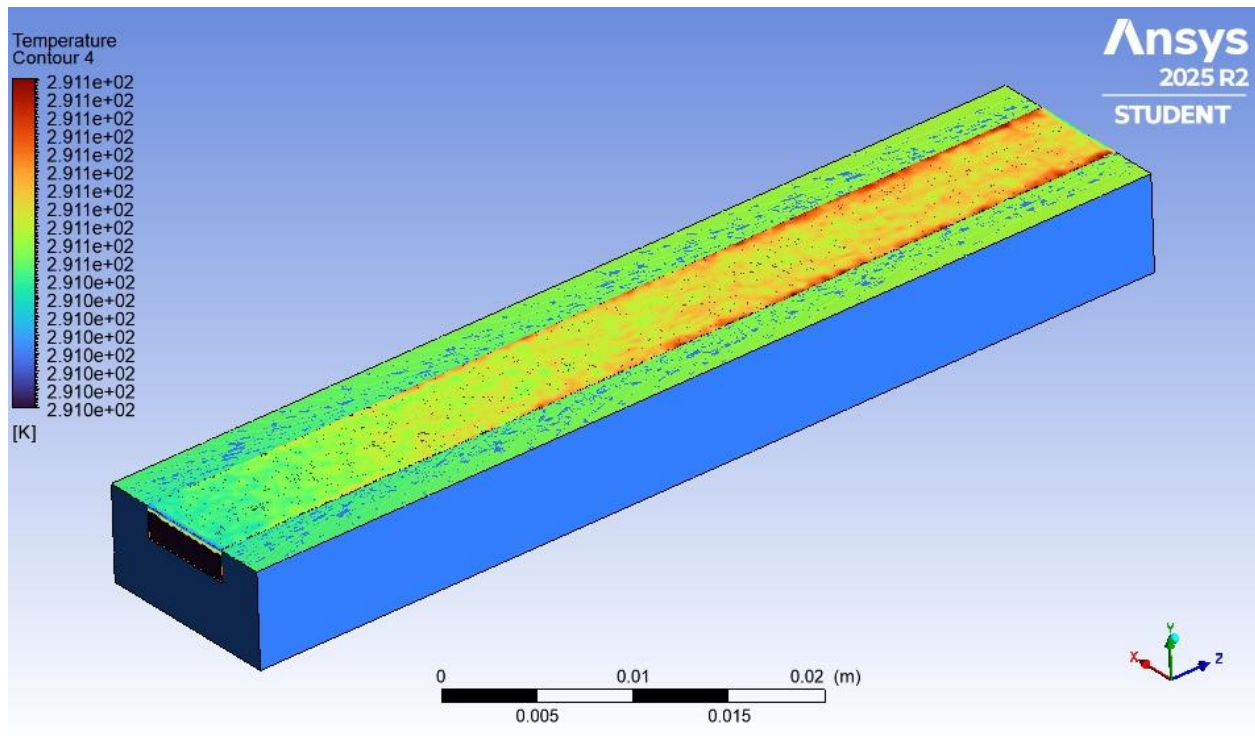


Figure 28 Temperature at 20 g/s and 500W

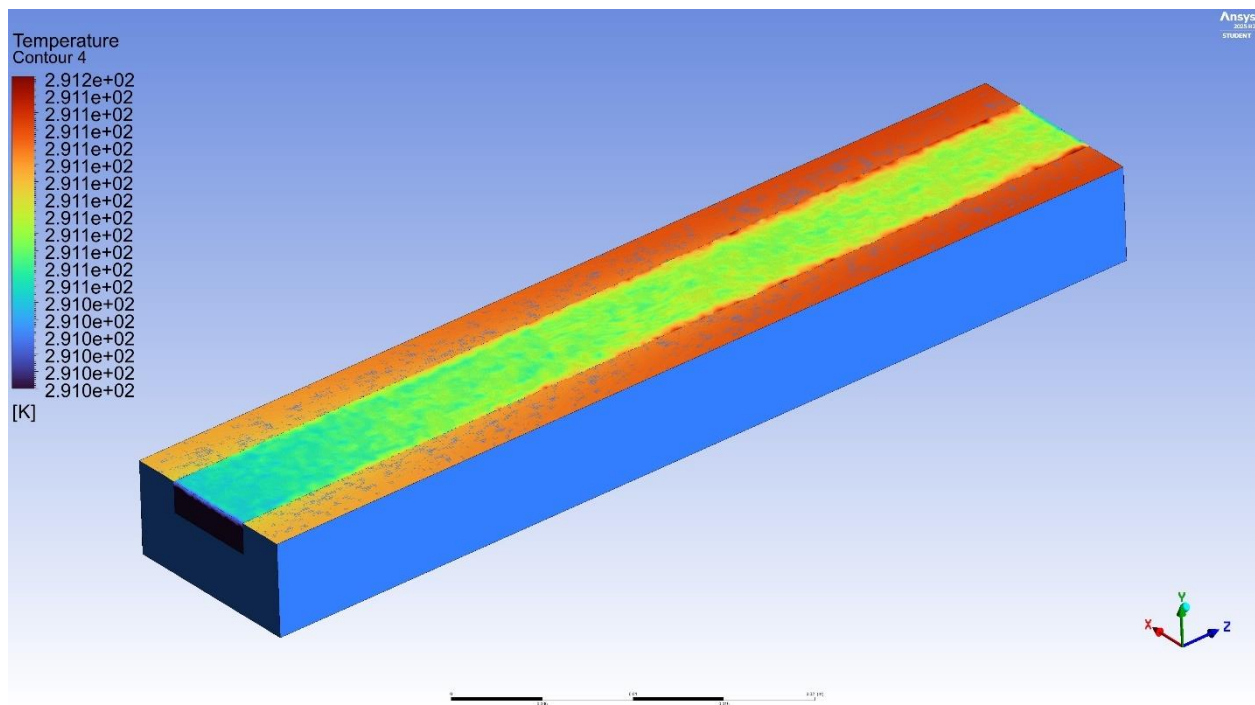


Figure 29 Temperature at 20 g/s and 1000 W

5.2 Discussion on simulation results at 40 g/s and 250, 500, 1000W

The maximum velocity rose dramatically to roughly 4.14 m/s at a medium mass flow rate of 40 g/s, demonstrating that doubling the flow rate from 20 g/s resulted in an essentially proportionate increase in velocity. Additionally, the pressure drops increased significantly to 3.5KPa, as compared to pressure at 20 g/s indicating stronger flow resistance brought on by increased turbulence and velocity. The maximum wall heat flux, in contrast to the 20 g/s instance, reduced marginally at 1000 was $10.13 \times 10^4 \text{ W/m}^2\text{K}$ but increased slightly with higher thermal loads, from $9.57 \times 10^4 \text{ W/m}^2\text{K}$ at 250 W to $9.551 \times 10^4 \text{ W/m}^2\text{K}$ at 500 W. This behaviour points

to more effective convective heat transfer at this flow rate, with the system better able to handle higher heat inputs without sacrificing performance. As a result, 40 g/s is the ideal balance between hydrodynamic effectiveness and heat evacuation.

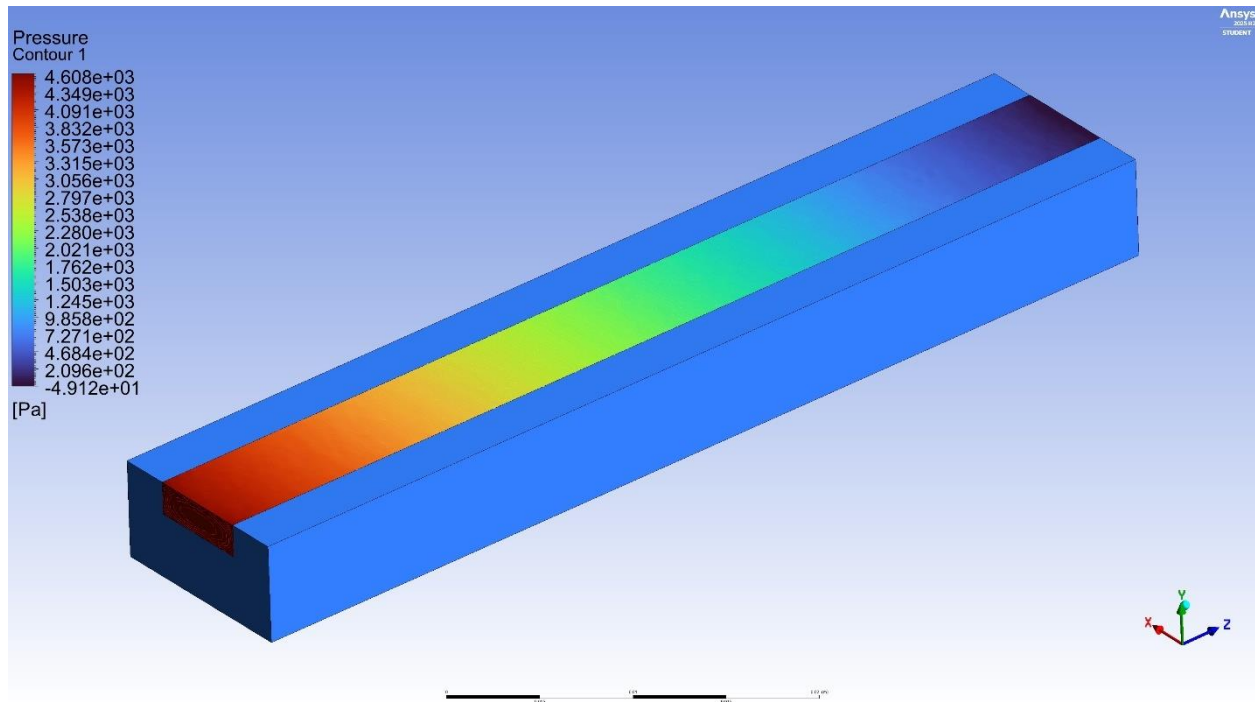


Figure 30 Pressure at 40 G/S and 250W

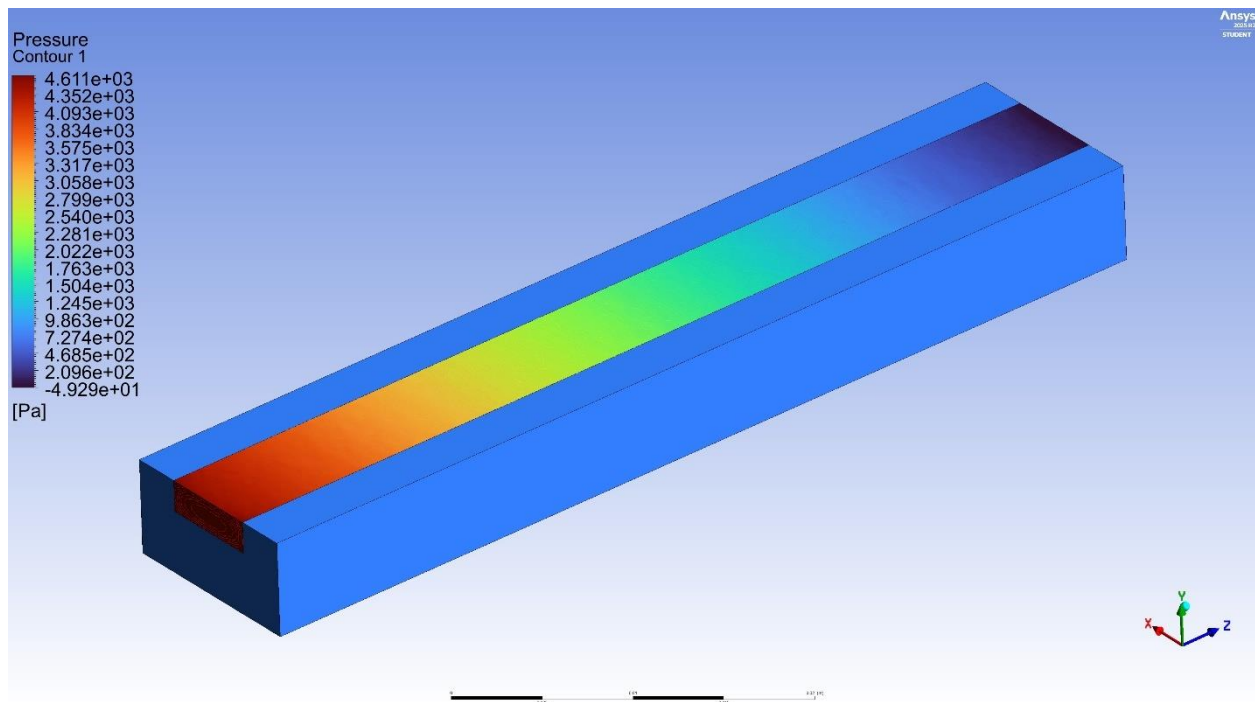


Figure 31 Pressure 40 g/s 500W

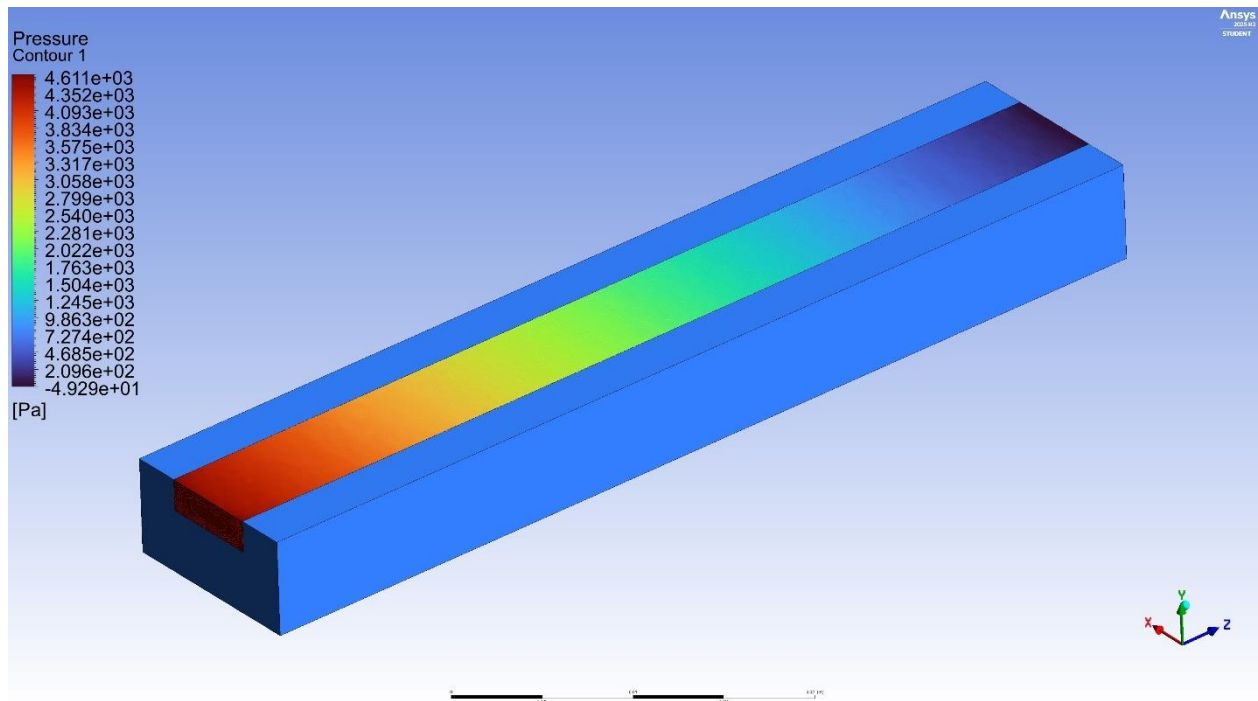


Figure 32 Pressure 40 g/s 1000W

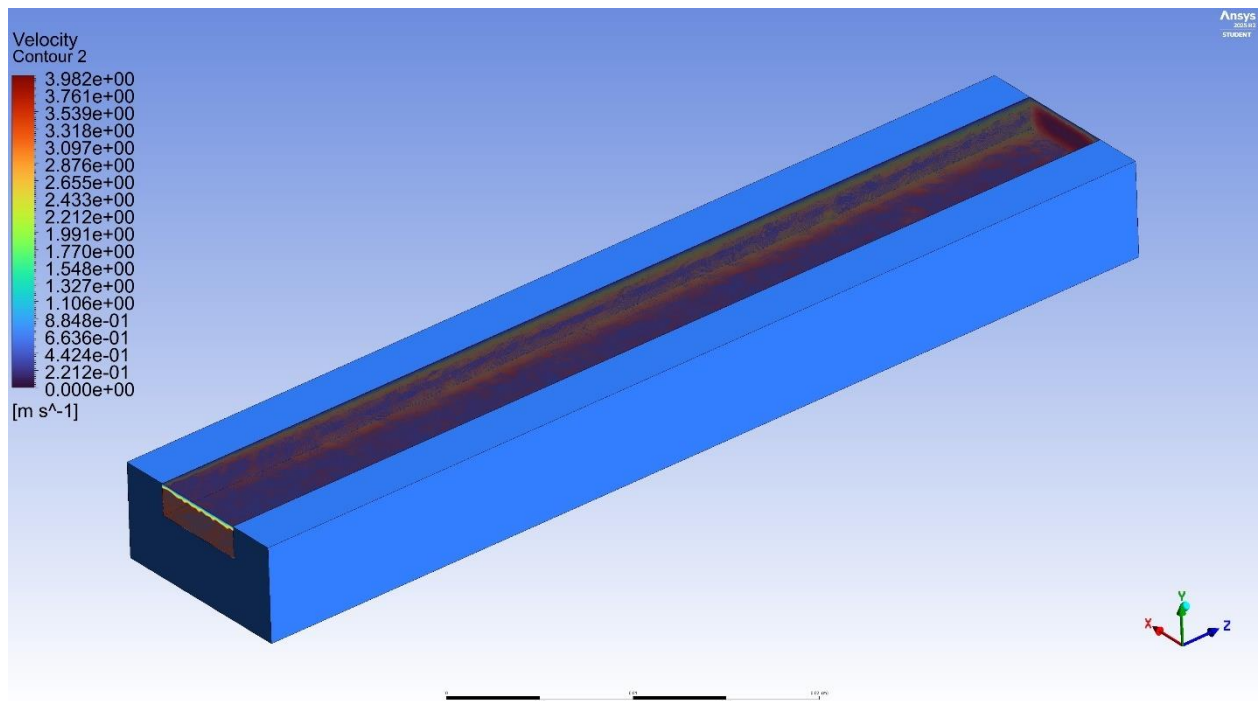


Figure 33 velocity at 40 g/s 1000W

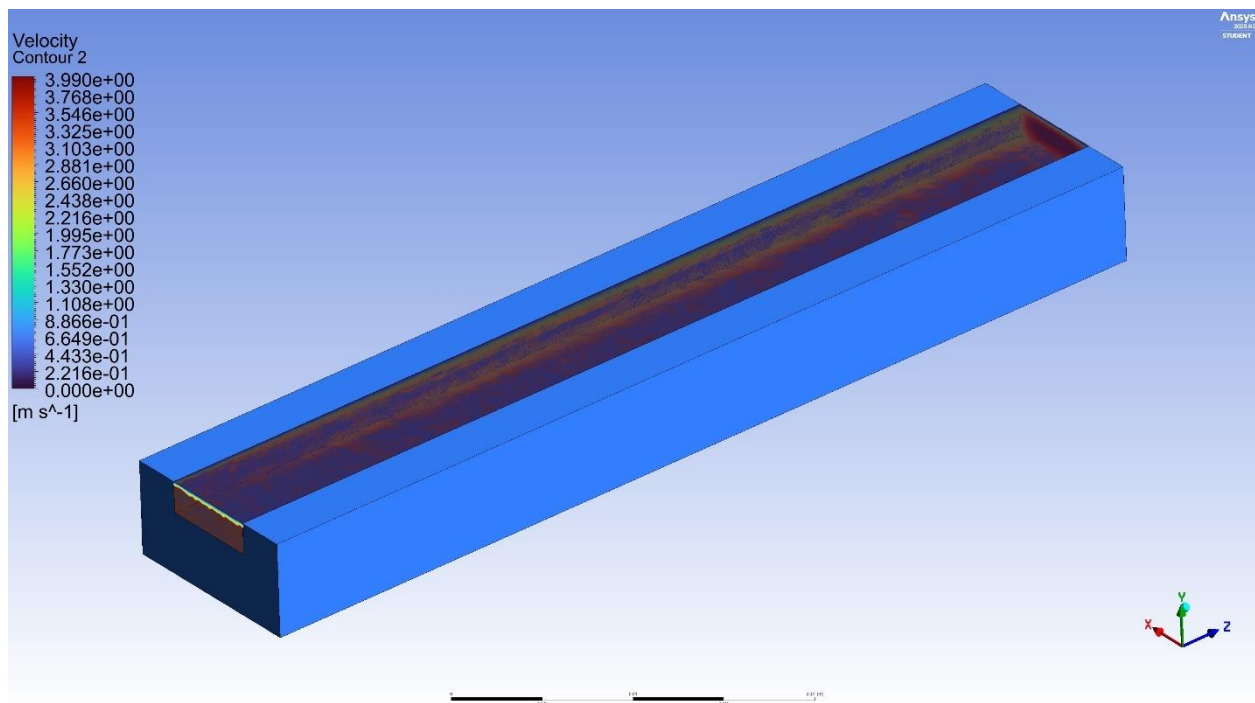


Figure 34 Velocity at 40 g/s

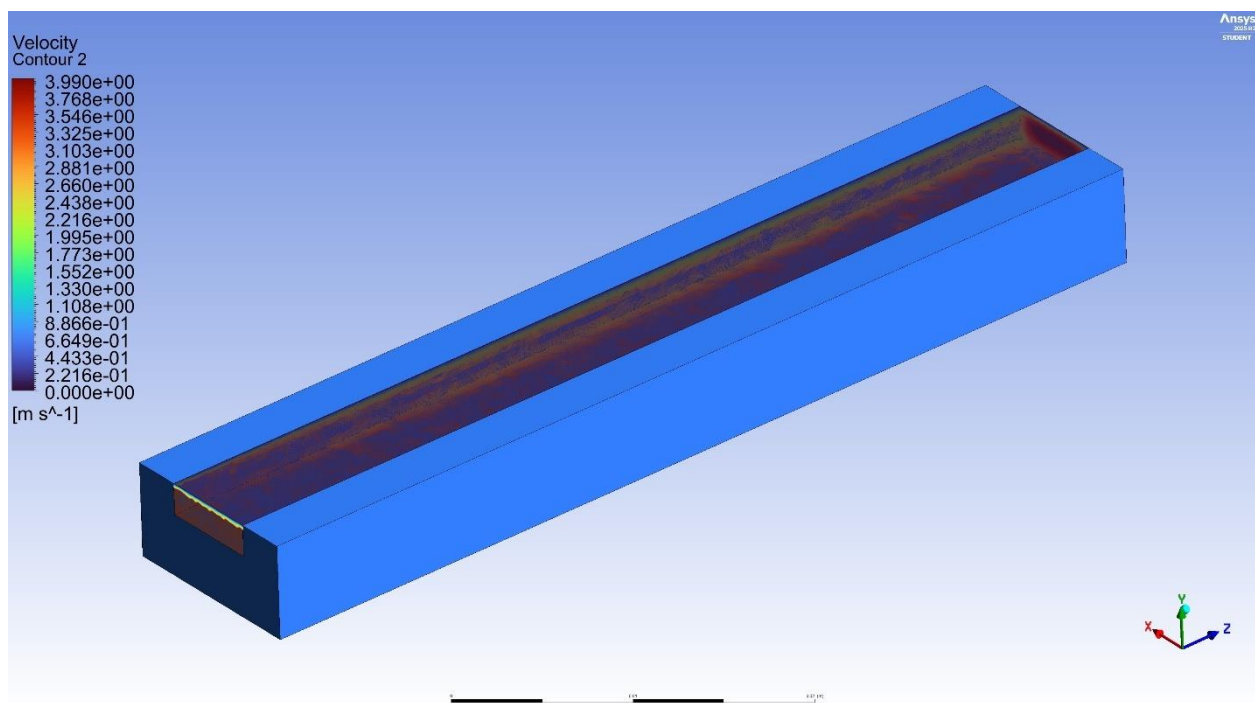


Figure 35 Velocity-3 AT 40 g/s 1000W

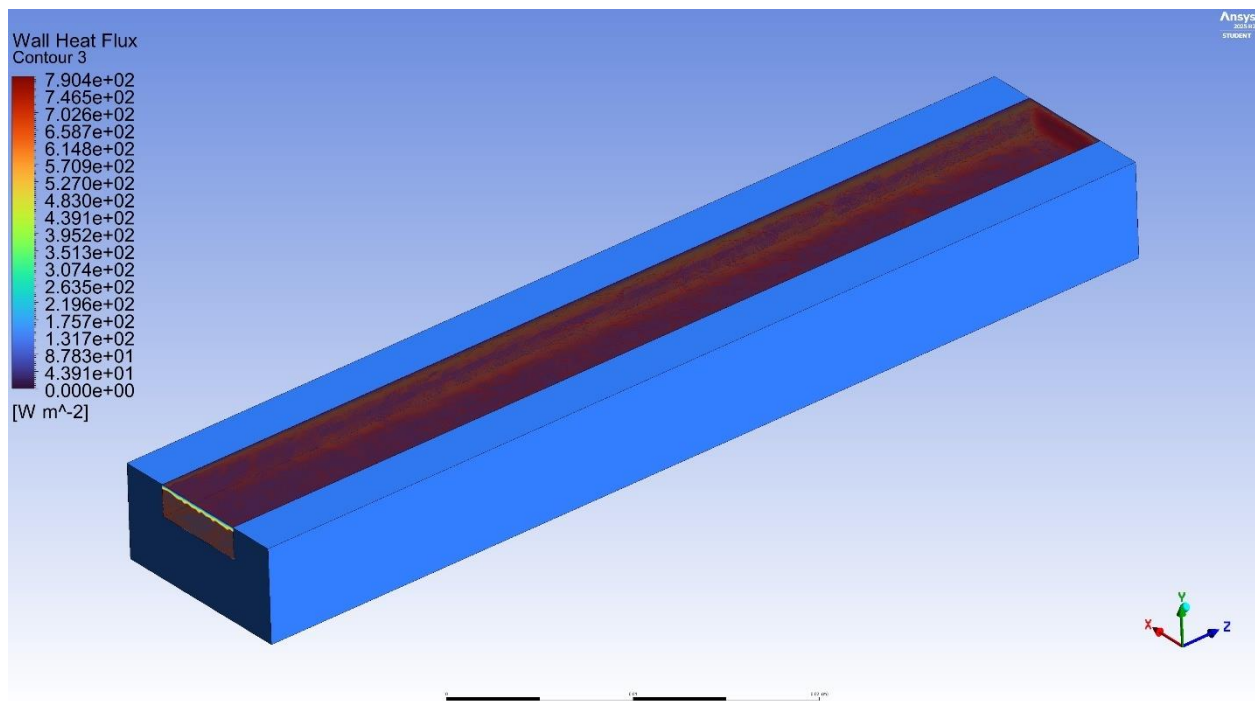


Figure 36 Heat Flux-1 At 40 G/S

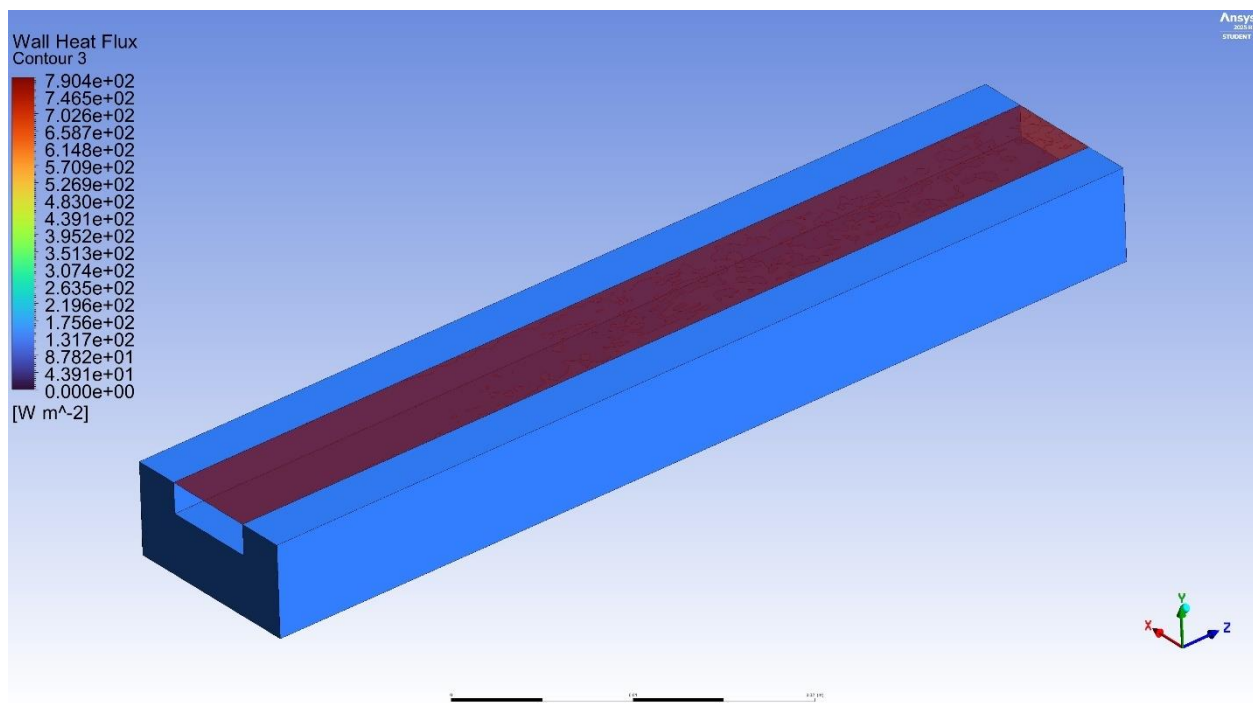


Figure 37 Figure 23 Heat Flux-2 At 40 G/S

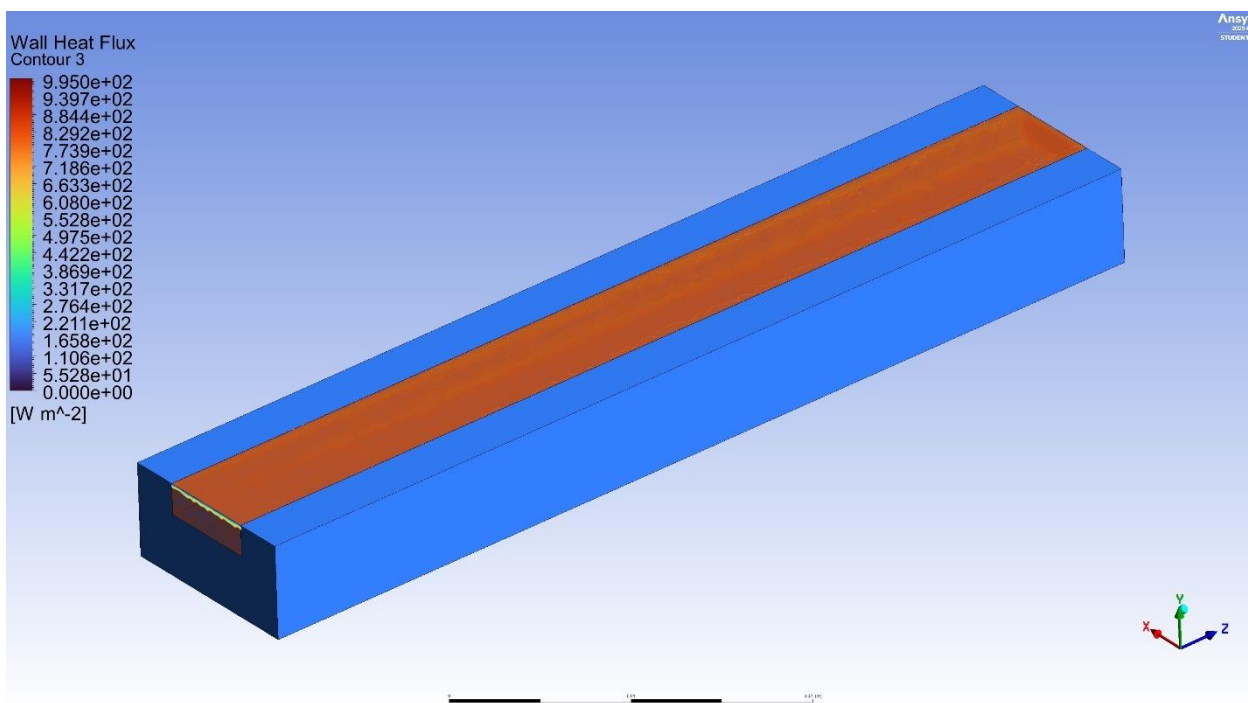


Figure 38 Heat Flux-3 At 40 G/S

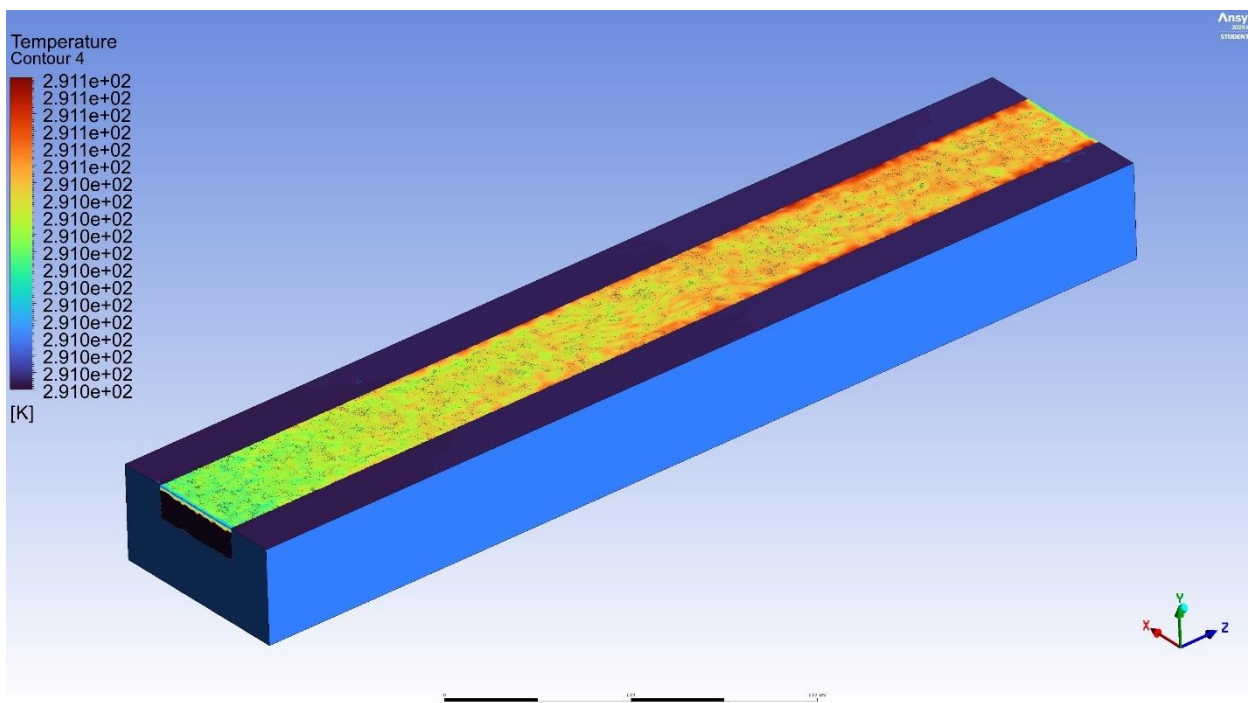


Figure 39 Temperature at 40 g/s and 500W

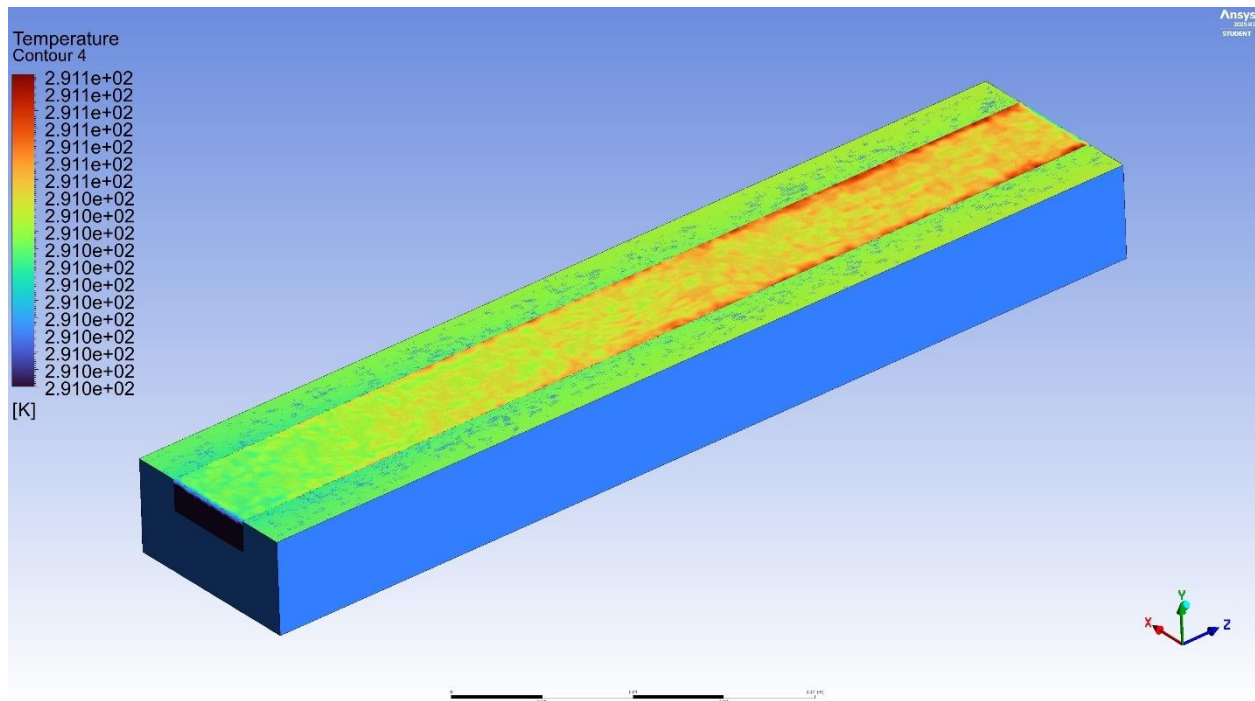


Figure 40 Temperature at 40 g/s and 500W

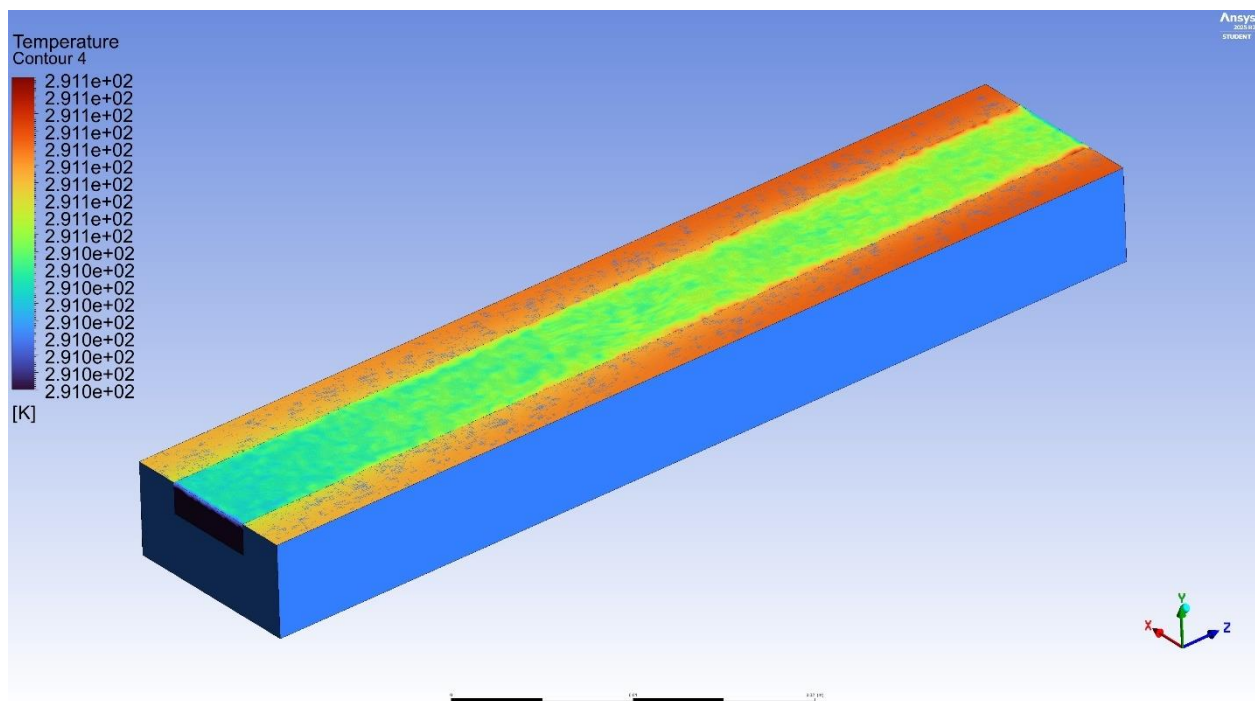


Figure 41 Temperature at 40 g/s and 1000W

5.3 Discussion on simulation Results at 80 g/s and 250, 500, 1000W

The system showed a peak flow velocity of 71.6 m/s at the maximum mass flow rate of 80 g/s, which was constant at all heat flux levels, suggesting the flow was probably close to its saturation point. A considerable hydraulic cost for operating at such a high flow rate was shown by the pressure drop, which doubled from the 40 g/s instance to 7.69 kPa. The maximum wall heat flux, on the other hand, stayed comparatively constant, increasing marginally from $10.31 \times 10^4 \text{ W/m}^2\text{K}$ at 250 W to $10.86 \times 10^4 \text{ W/m}^2\text{K}$ at 500 W before significantly declining to $10.06 \times 10^4 \text{ W/m}^2\text{K}$ at 1000 W. These findings indicate declining returns in thermal efficiency at very high

flow rates, even though thermal performance improved marginally. The benefits in heat transfer do not offset the sharp increase in pressure losses.

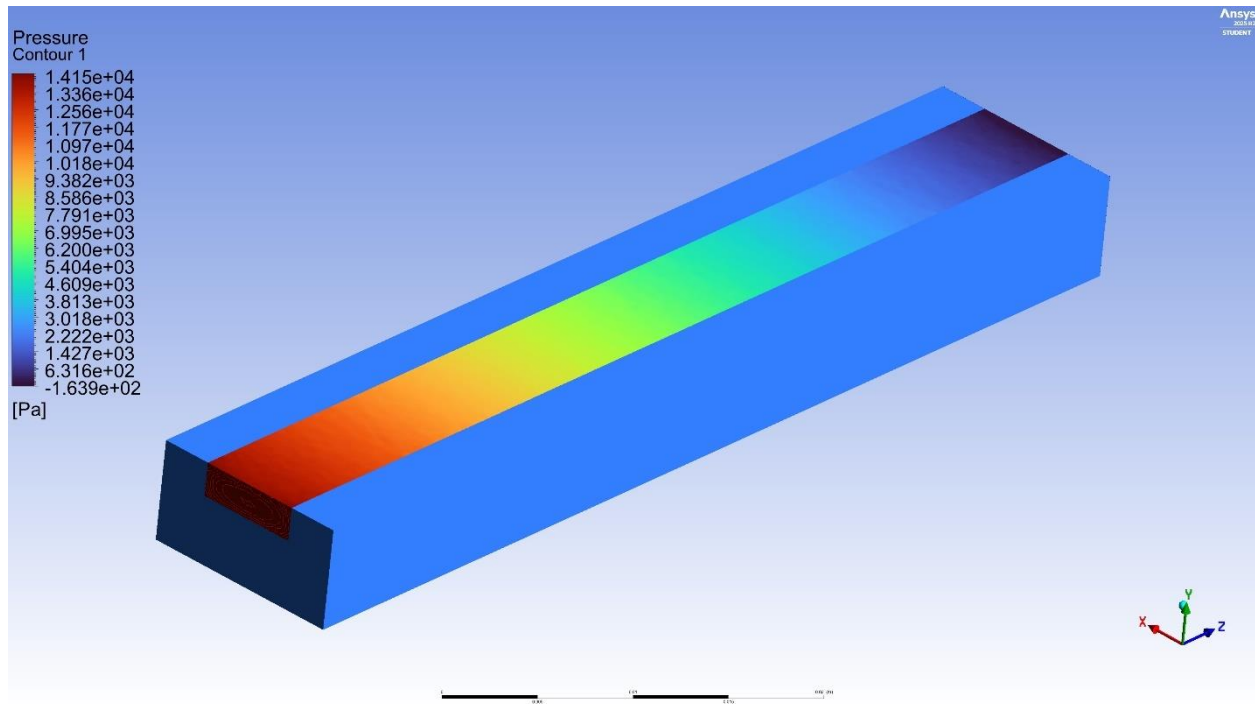


Figure 42 PRESSURE-1 AT 80 G/S and 250W

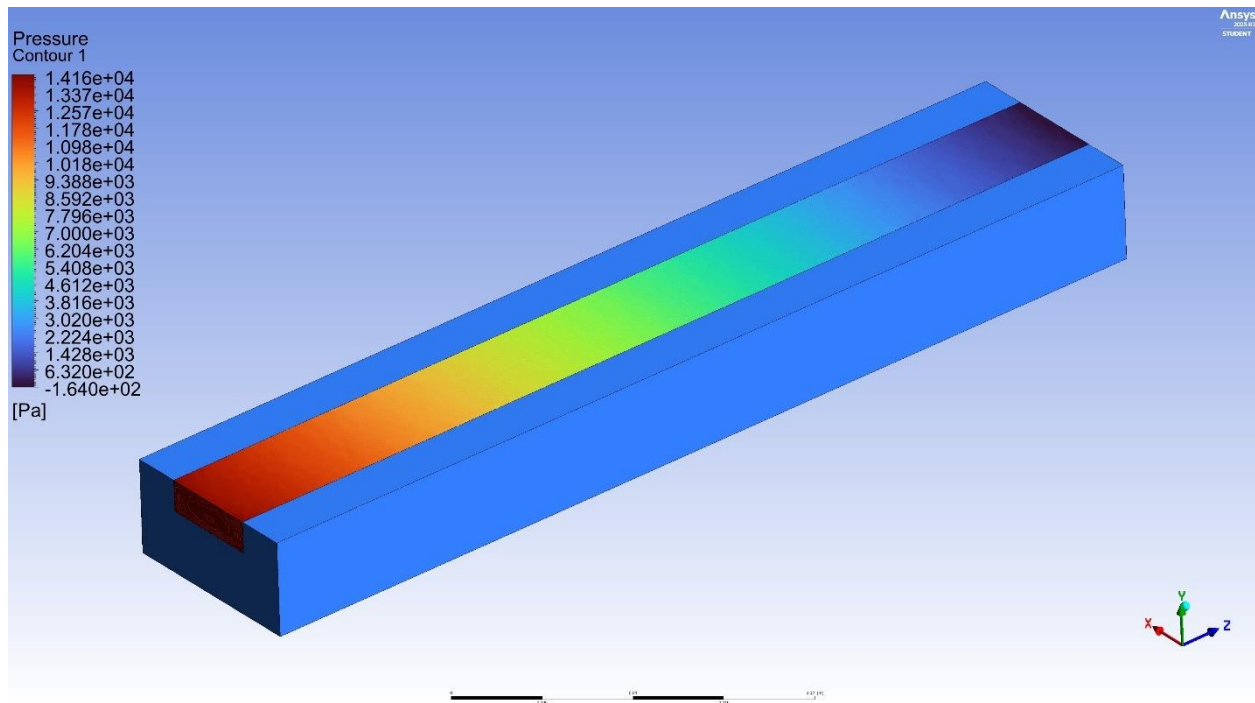


Figure 43 PRESSURE-2 AT 80 G/S and 500W

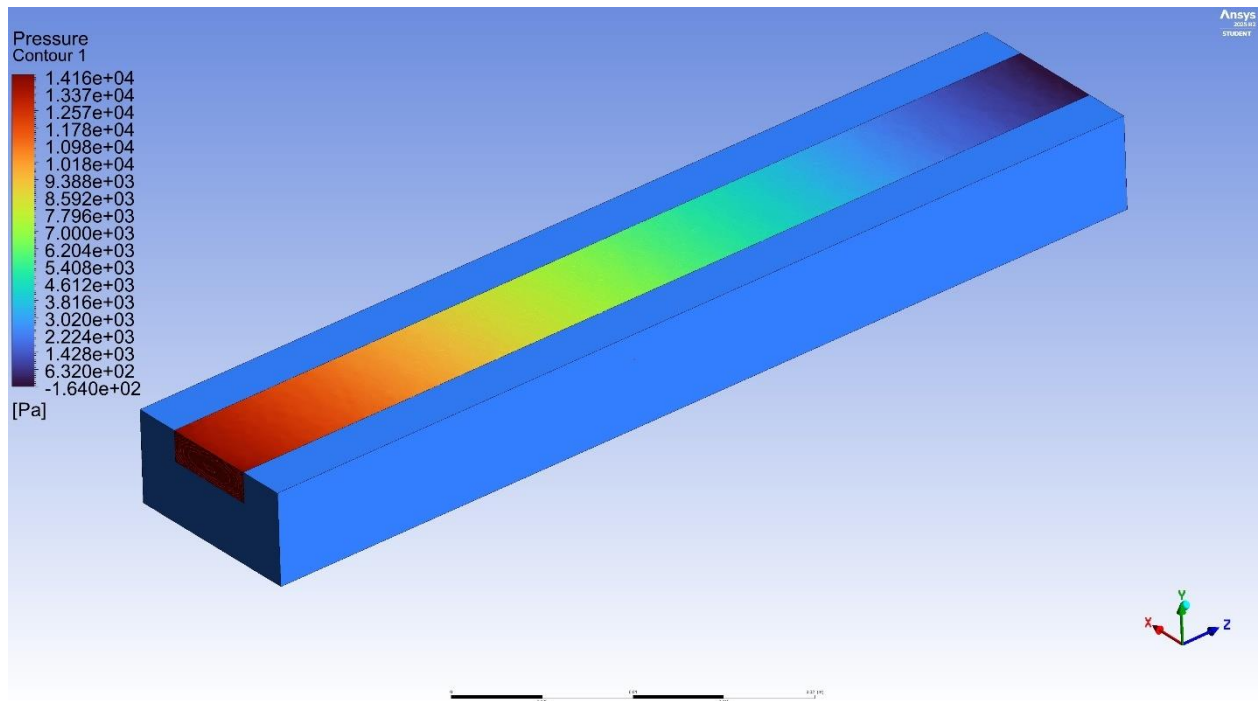


Figure 44 PRESSURE-3 AT 80 G/S and 1000W

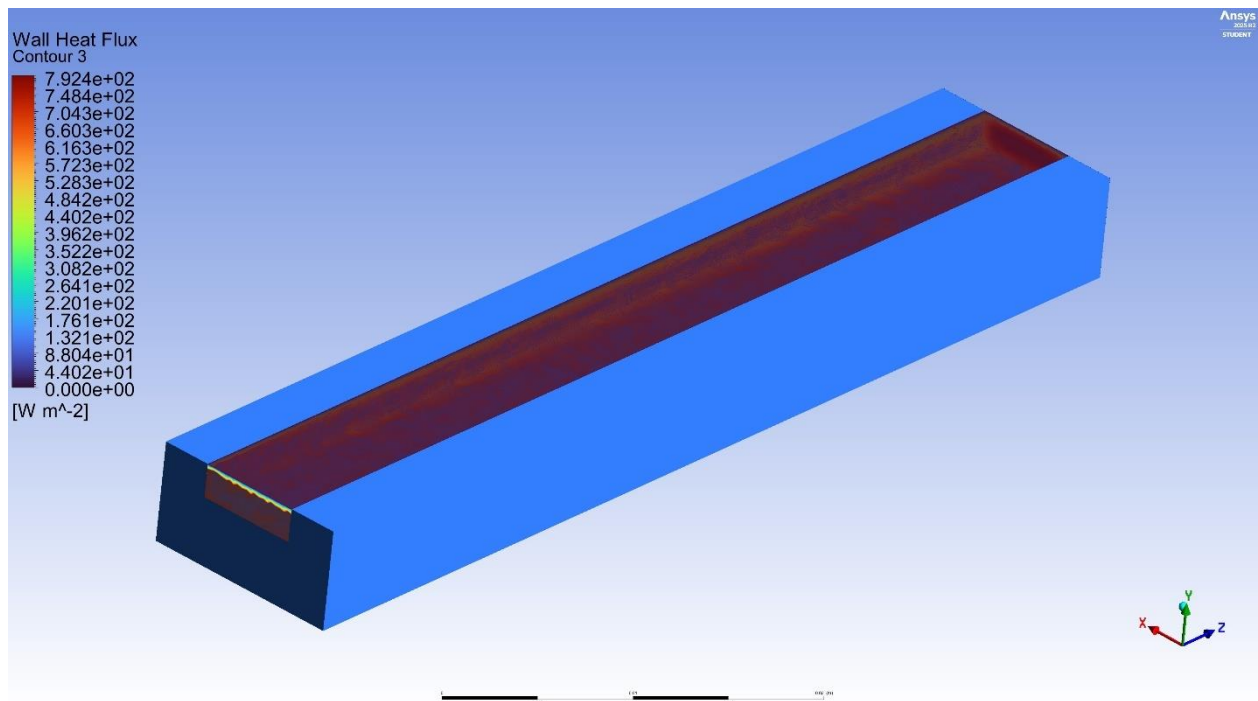


Figure 45 Wall Heat Flux At 80 G/S and 250w

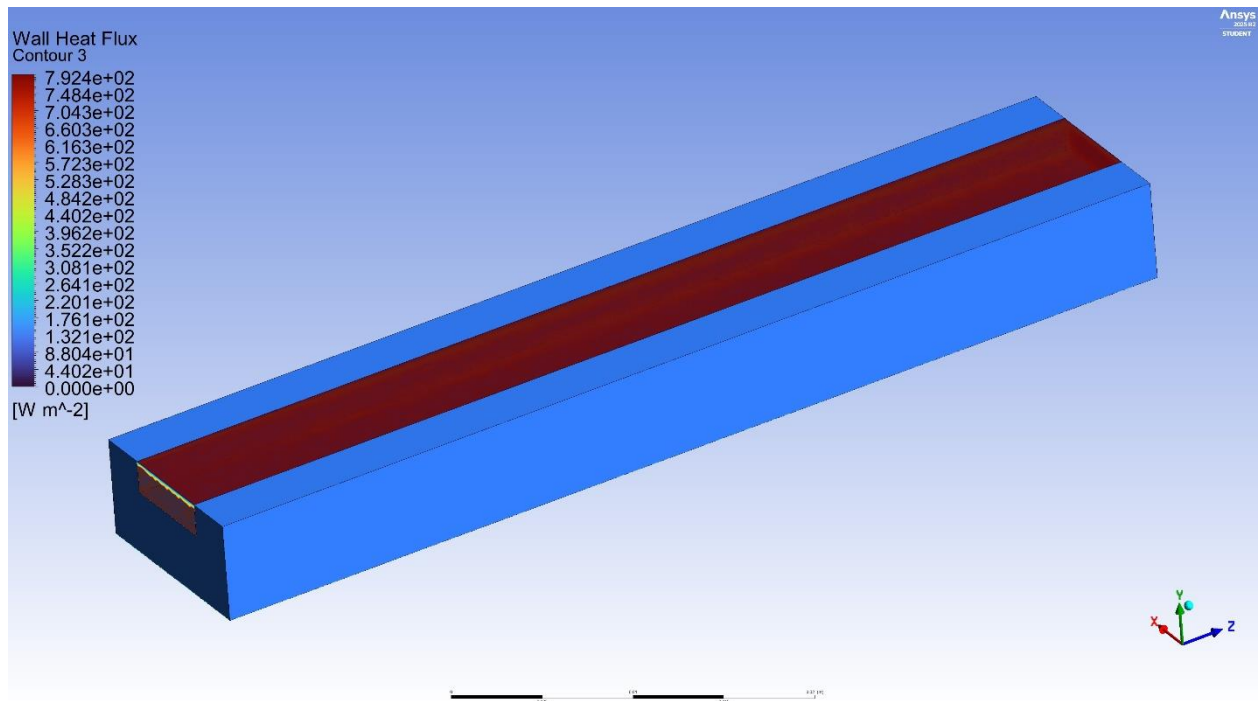


Figure 46 Wall Heat Flux At 80 G/S and 5000w

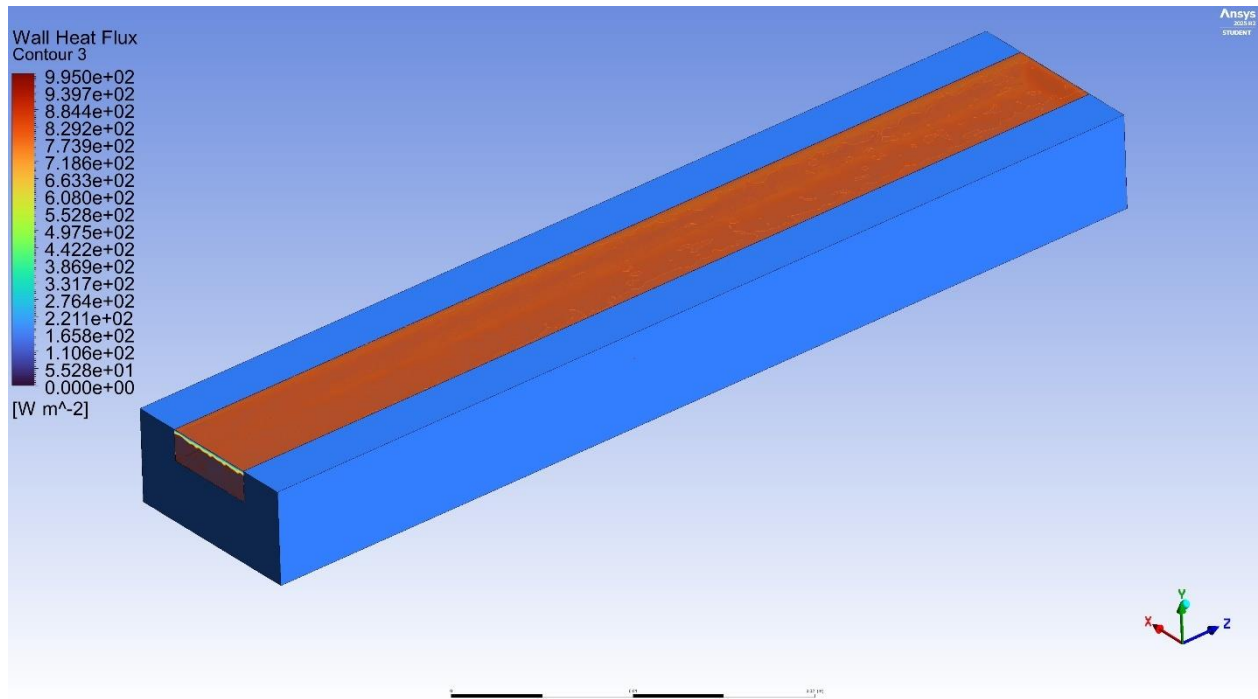


Figure 47 Wall Heat Flux At 80 G/S and 10000w

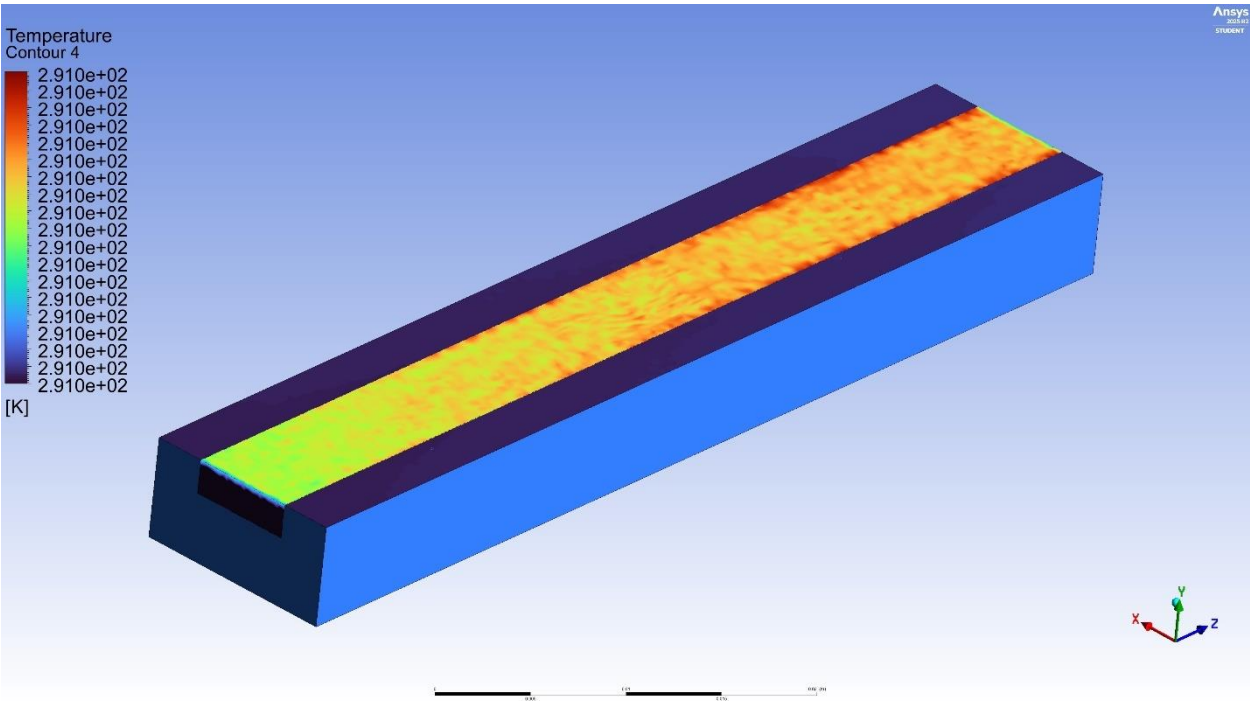


Figure 48 Temperature at 80 g/s and 250W

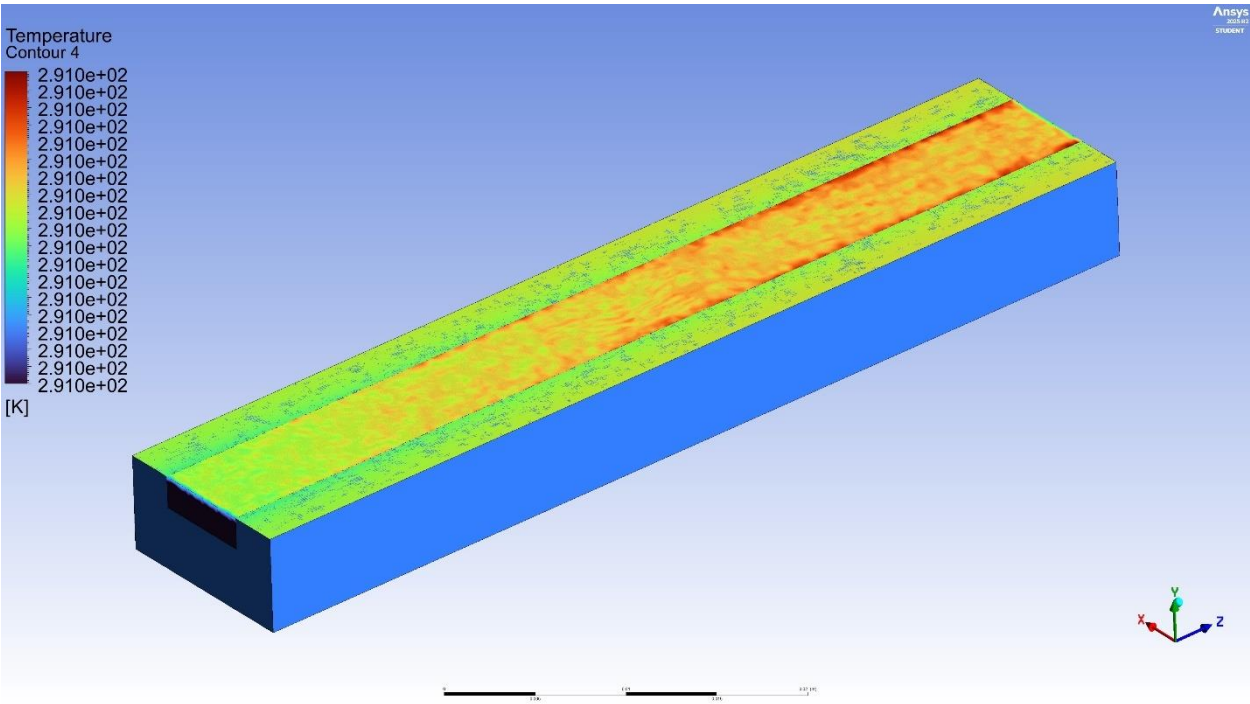


Figure 49 Temperature at 80 g/s and 500W

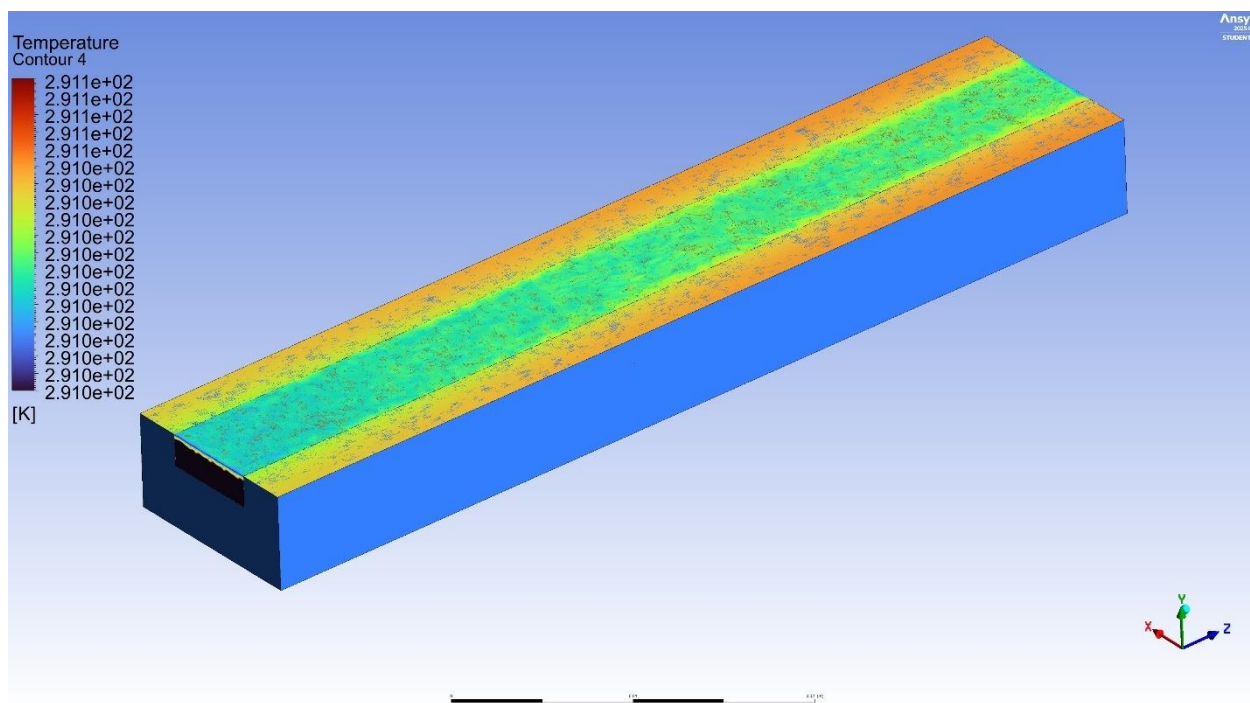


Figure 50 Temperature at 80 g/s and 1000W

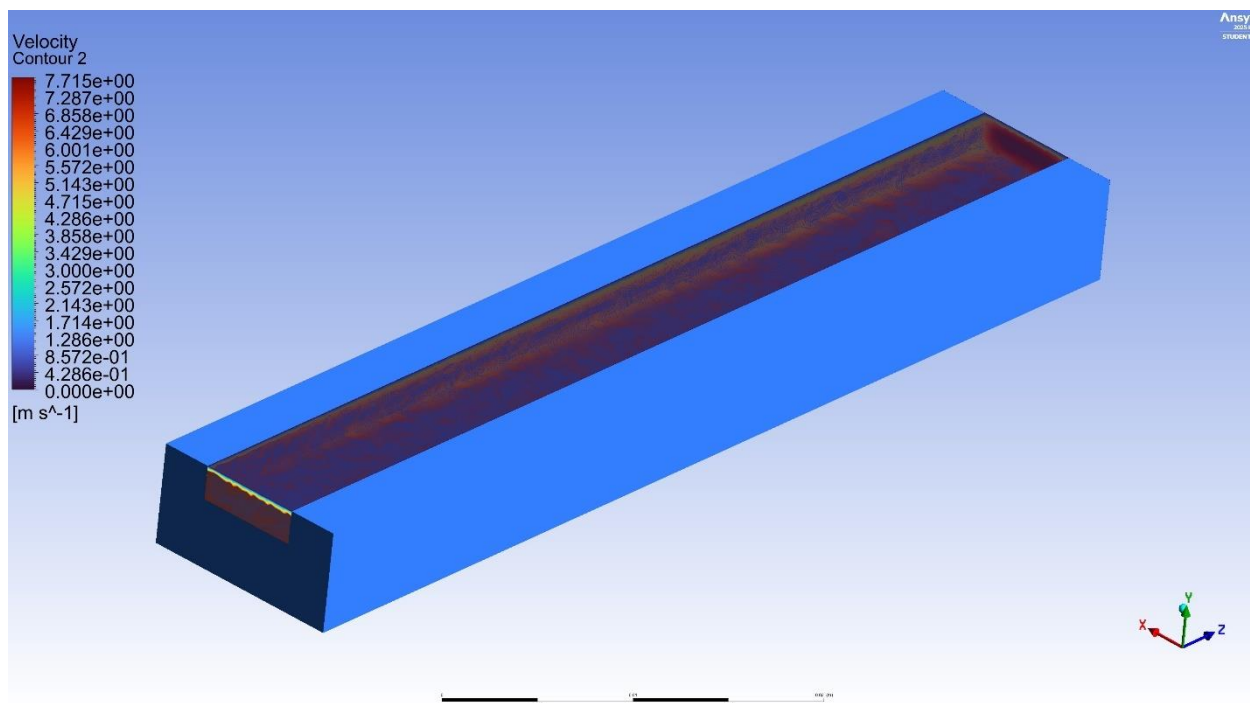


Figure 51 velocity at 80 g/s and 250W

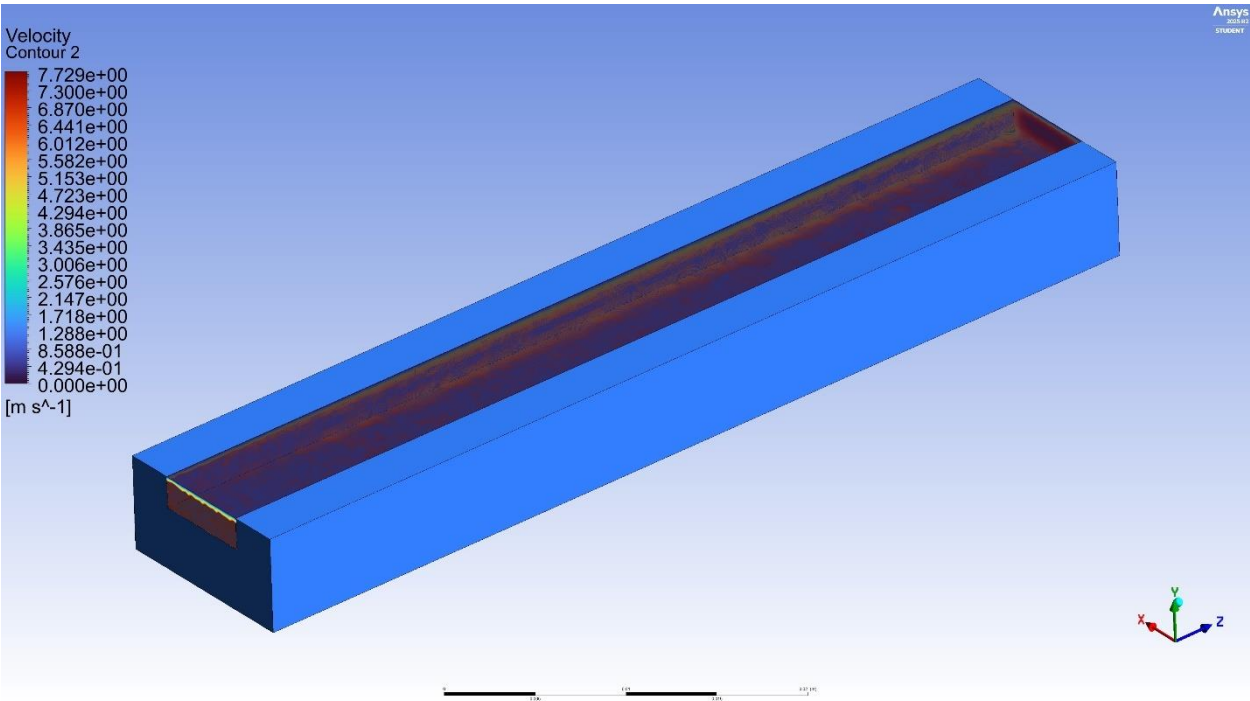


Figure 52 velocity at 80 g/s and 500W

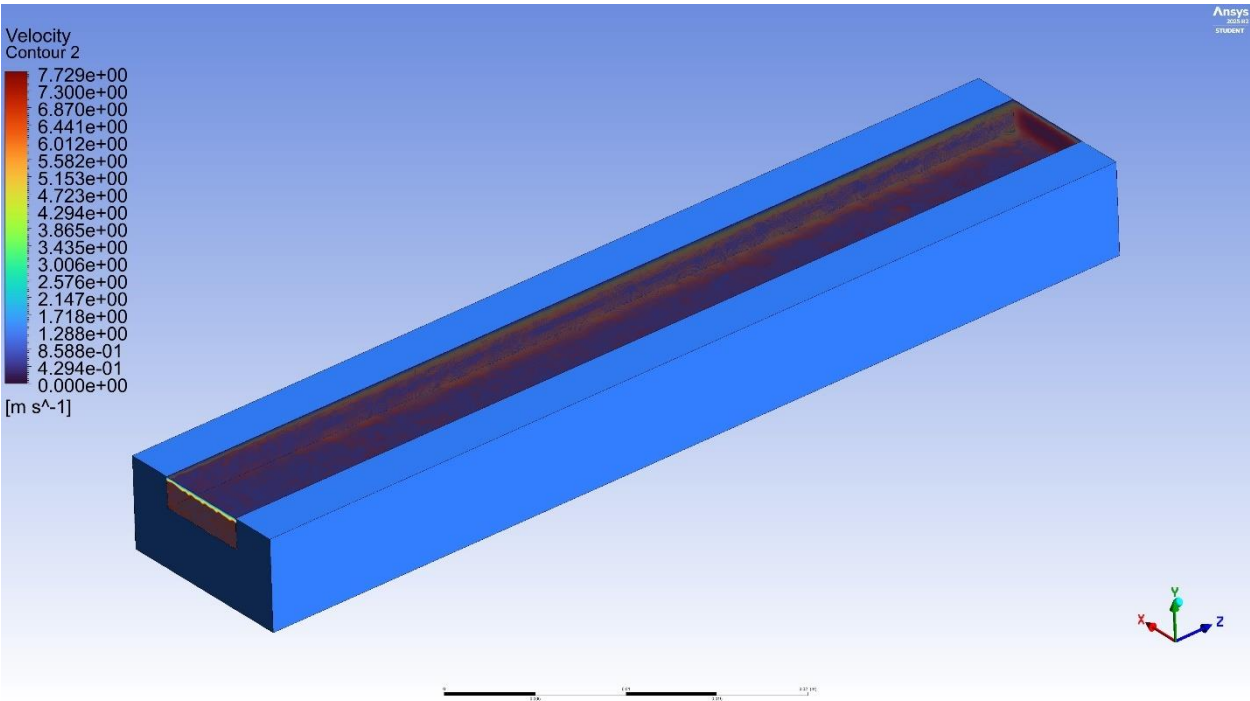


Figure 53 velocity at 80 g/s and 1000W

Table 11 minimum and maximum values

Sr No	Pressure Pa		Velocity		Temperature		Wall Heat flux	
	maximum	minimum	maximum	minimum	maximum	minimum	maximum	minimum
1	1599	71	2.089	0.0115	291.1	291	788.2	43.78
2	1598	70.6	2.089	0.0116	291.1	291	788.2	43.79
3	1598	70.6	2.089	0.0116	291.2	291	995	55.28
4	4608	209.6	3.98	0.2216	291.12	291	790.4	43.91
5	4611	209.6	3.99	0.2216	291.12	291	790.4	43.91
6	4611	209.6	3.99	0.2216	291.12	291	995	55.28
7	14150	631.6	7.71	0.4286	291.15	291	792.4	44.01
8	14160	632	7.72	0.4294	291.15	291	792.4	44.02
9	14160	632	7.72	0.4294	291.15	291	995	55.28

6 CONCLUSION

Important details about the aluminium microchannel's thermal and hydraulic performance were uncovered by the CFD simulation conducted under various heat fluxes (250 W, 500 W, and 1000 W) and mass flow rates (20 g/s, 40 g/s, and 80 g/s). The maximum velocity and pressure drop across the channel increased dramatically as the mass flow rate increased, suggesting both higher hydraulic resistance and improved convective transport. With little change in flow characteristics and a slight drop in wall heat flux at 20 g/s, the system demonstrated limited reactivity to greater heat flux, indicating decreased heat removal efficiency at low flow rates. The channel's ideal working point was determined by its balanced performance at 40 g/s, which included enhanced heat dissipation and a modest pressure drop. On the other side, the 80 g/s condition resulted in severe pressure losses and just small thermal improvements, although offering the highest velocities and marginally improved heat flux handling. In general, heat transmission is improved by increasing mass flow rate up to a certain point, after which the advantages are diminished by increasing pressure penalties. These results are useful for microchannel cooling system optimization, where pumping power and thermal efficiency must be taken into account.

7 REFERENCES

- [1] Anjaneya, G., Sunil, S., K, M., Santhosh, A., Patil, S., Prasad C, D., Aden, A., & G, G. (2024). Study of Impact of Nano Fluids on Performance of Microchannel Heat Exchangers Using CFD. *International Journal of Thermofluids*, 24, 100836. <https://doi.org/10.1016/j.ijft.2024.100836>
- [2] Ekkad, S. V, & Singh, P. (2023). Chapter Six - Recent advancements in single-phase liquid-based heat transfer in microchannels (J. P. Abraham, J. M. Gorman, & W. J. Minkowycz (eds.); Vol. 55, pp. 239–293). Elsevier. <https://doi.org/https://doi.org/10.1016/bs.aiht.2022.12.001>
- [3] Li, J.-X., & Tu, S.-T. (2009). Boundary analysis of fluid laminar forced flow over an isothermal plate in porous medium considering the local non-equilibrium. *Huadong Ligong Daxue Xuebao /Journal of East China University of Science and Technology*, 35, 481–485.
- [4] Madi, K., Raafat, A., & Al Nuaimi, S. (2025). Enhanced thermal management in microelectronic cooling: A study on pairing

multiple pin-fin shapes in microchannel heat sinks. International Journal of Thermofluids, 28, 101283. <https://doi.org/https://doi.org/10.1016/j.ijft.2025.101283>

- [5] Zhang, J., Zou, Z., & Fu, C. (2023). A Review of the Complex Flow and Heat Transfer Characteristics in Microchannels. Micromachines, 14(7). <https://doi.org/10.3390/mi14071451>