Brunel University London
College of Engineering, Design and Physical Sciences
Dept. of Mechanical, Aerospace and Civil Engineering

ME3621 CFD LAB REPORT

Student Number: 1911261

ABSTRACT

The aim of the project was to understand and simulate the heat transfer in a rectangular chamber with and without ribs. The software ANSYS Fluent was used to analyse the simulation. The heat transfer is an essential for energy system. The purpose of the study is the corrugated channel on has more effective heat transfer due to its increased surface area. In this study, the corrugated domain has three equisized ribs placed on one wall of the chamber. Fluent will be used to calculate the Nusselt number, Heat Flux and Total Heat Transfer Rate along with contour plots of TKE and temperature as well as a vector plot of the Velocity. The temperature of the inlet and outlet is provided which allows us to understand the flow of the air inside the chamber. Parameters like velocity and turbulence kinetic energy will be essential to understand the heat transfer in the chamber with and without ribs. The velocity will allow us to understand the mixing of air in the scenarios, and one of the major variables here will be the meshing of the domain. As suggested in the question, meshing will help us to see the differences in the results obtained by using either finer or courser mesh in the domain.

DESCRIPTION OF NUMERICAL SIMULATION

Introduction

Effective heat transfer is essential for energy systems, so corrugated channels with ribs on the walls have been used to enhance heat transfer by increasing the surface area. The corrugation configuration impacts on both thermal and hydraulic characteristics of the channel flow, which can be quantified by the Nusselt number Nu and the pressure drop ΔP, respectively. Different rib shapes have been used and effects on heat exchange investigated. In addition, in a smooth channel, heat transfer is achieved by forced convection. In this study, we will investigate the effects of ribs in rectangular shape on turbulent heat transfer in a corrugated channel.

Geometry

Figure 1 presents the two-dimensional cooling channel. The length and (half) height of the channel are L and H, respectively. For the corrugated channel, three equisized ribs are placed in the middle of the channel and are attached to the cooling wall. The distances of the rib set to the inlet and outlet are therefore equal and both D (see Fig. 1). The length and width of a rib are a and b, respectively. The space between two ribs is c. The parameters used are L = 50 mm, H = 5 mm, a = 4mm, b = 1mm, c = 13mm, D = 6mm. The velocity and temperature of the hot air flow entering the channel are U_0 = 50 m/s and T_0 = 400 K, respectively. The temperature of the cooling wall is T_w = 300 K. Symmetric conditions are to be used for the top boundary of the domain.

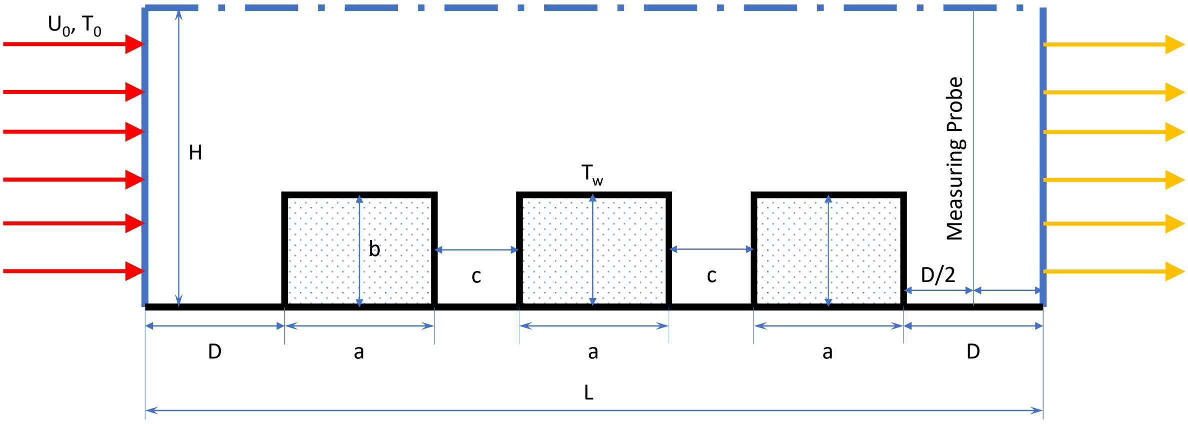

Figure 1) Schematic of two-dimensional corrugated cooling channel

The Nusselt number in this study is defined as,

$$Nu = \frac{hD_\mathrm{h}}{\kappa} = \frac{D_\mathrm{h}}{\kappa}\frac{q''}{(T_\mathrm{w} - \bar{T}_\mathrm{f})},$$

where h is the heat transfer coefficient (W m^{-2} K^{-1}); D_h is the hydraulic diameter (m); κ is the thermal conductivity of air at T_0. κ = 33.45×10^{-3} W m^{-1} K^{-1}. h is determined by $h = q''/(T_w - \bar{T}_\mathrm{f})$, where q'' is the (average) heat flux (W m^{-2}); $\bar{T}_\mathrm{f}$ is the volume-averaged fluid temperature (K). $D_\mathrm{h} = 4A/D_\mathrm{wp}$, where A is the cross-section area (m^2) and D_wp the wetted perimeter (m). In this study, $D_\mathrm{h} \approx 4H$ = 20 mm.

The Nusselt number is defined as the ratio of convection heat transfer to fluid conduction heat transfer under the same conditions. [1] The Nusselt number is an important parameter that can contribute to a better rate of heat exchange. It is basically a function of Reynolds and Prandtl number. If the Nusselt number is about 1, it represents that the heat transfer is conduction only, but if the value is between 1 and 10, then it shows laminar of slug flow. If the range is more, it is active convection with turbulence in the 100–1000 range. Nusselt number on the other hand is a non-dimensional heat transfer coefficient. It is used to determine whether the heat transfer is conduction or convection. [2]

Mesh Independence Study

Two meshes of the two-dimensional corrugated cooling channel were created in ANSYS ICEM CFD 2020 R2. Both fine and coarse shown in Figure 2 and 3, respectively. In this study, this will demonstrate that the coarse mesh can produce satisfying results, which are close the fine-mesh ones. In the pre-mesh parameters, the grid size of the fine and coarse mesh is 100x1000 and 25x250, respectively. Spacing 1 and 2 of the fine mesh is 1e-7 and is 0 for the coarse mesh. They both have a ratio of 1.2.

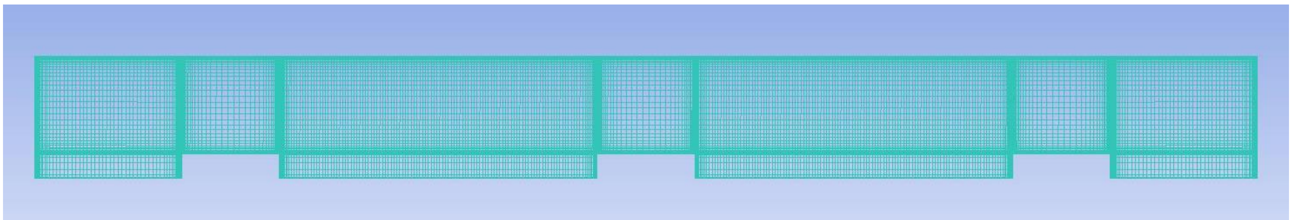

Figure 2) Fine mesh of corrugated channel created in ANSYS ICEM

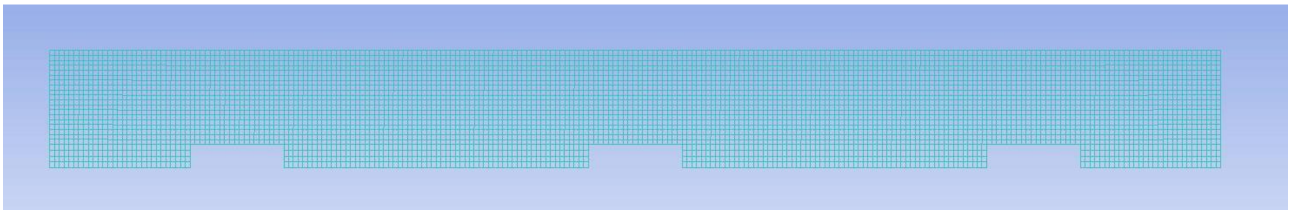

Figure 3) Coarse mesh of corrugated channel created in ANSYS ICEM

Setup & Boundary Conditions

The turbulent flow is numerically studied using ANSYS Fluent 2020 R2. The parameters Model, Boundary Conditions and Discretization were kept the same for both fine and coarse mesh.
The Model parameters used is Energy set to On and Viscous Model set to SST k-omega. The boundary conditions for Inlet are set to Velocity = 50m/s, Hydraulic Diameter = 20mm and Temperature = 400K as these are conditions of the hot airflow entering the channel. For Outlet, Hydraulic Diameter = 20mm and Temperature = 300K. For all the walls, leave the default values and set the Temperature to 300K due to the temperature of the cooling wall is T_w = 300K.

Solution

The parameters for Method are Scheme and under Spatial Discretization is Gradient, Pressure, Momentum, Turbulent Kinetic Energy, Specific Dissipation Rate and Energy. Set Scheme to Coupled, Gradient to Least Squares Cell Based and everything else set to Second Order/Second Order Upwind. Under Residuals both fine and coarse mesh have a convergence criteria of 0.001.

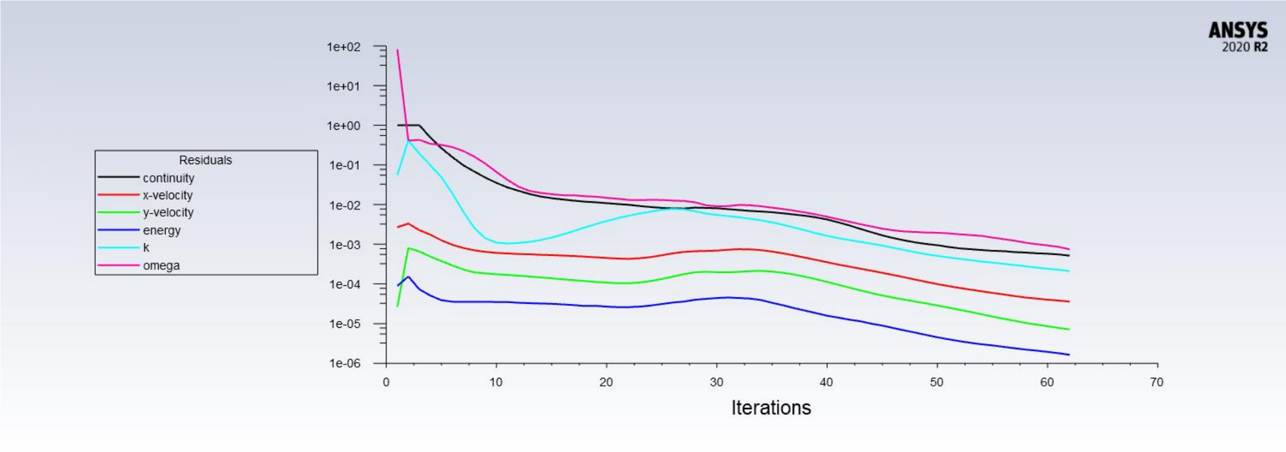

Figure 4) Convergence plot of Fine mesh converging at the 62nd iteration

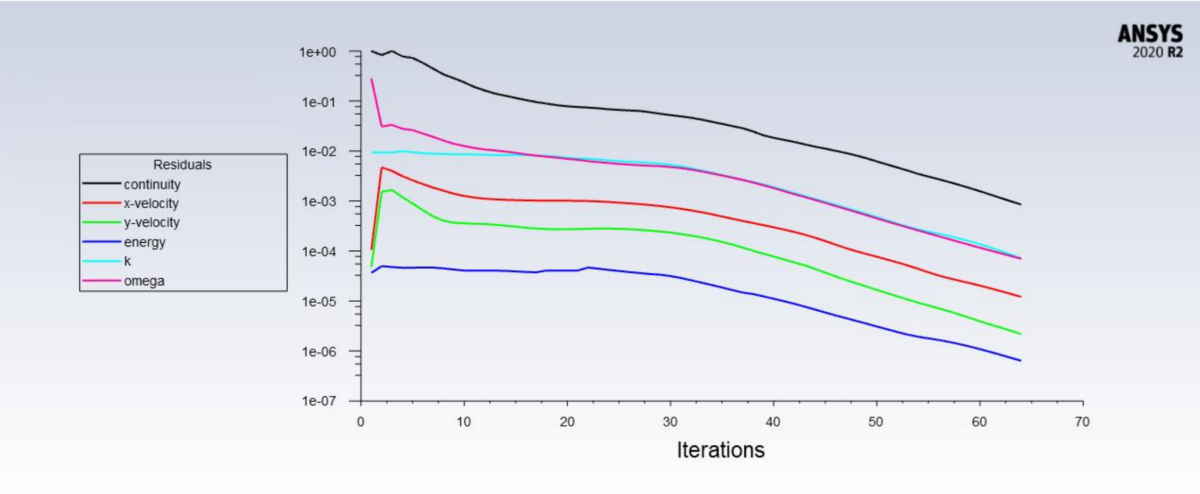

Figure 5) Convergence plot of Coarse Mesh converging at the 63rd iteration

Comparing Smooth and Corrugated Channel

This part of the study compares the Nusselt number and total pressure loss between the smooth and corrugated channels and the causes to the differences are analysed. The coarse mesh smooth channel is modelled in ANSYS ICEM CFD 2020 R2 with the same dimensions as the corrugated channel but without the ribs, 5mmx50mm. The pre-mesh parameters Spacing 1 and 2 is 0, the Ratio is 1.2 and the grid size is 25x250.

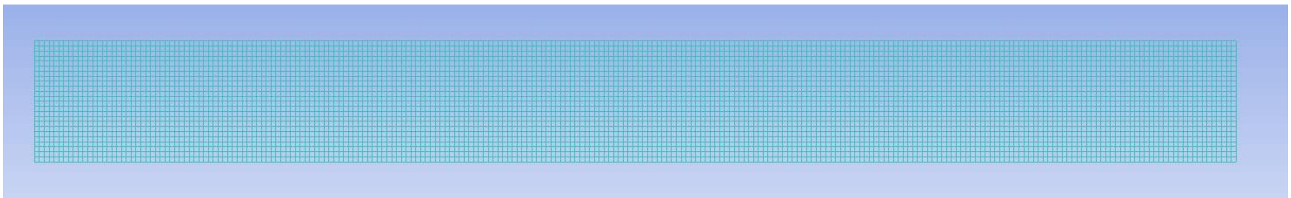

Figure 6) Coarse mesh of Smooth channel created in ANSYS ICEM

The mesh is numerically studied in ANSYS Fluent 2020 R2 using the same parameters used in the Setup and Solution for both corrugated channels, Fine and Coarse. With the same convergence criteria used in Residuals Figure 7 presents the convergence plot.

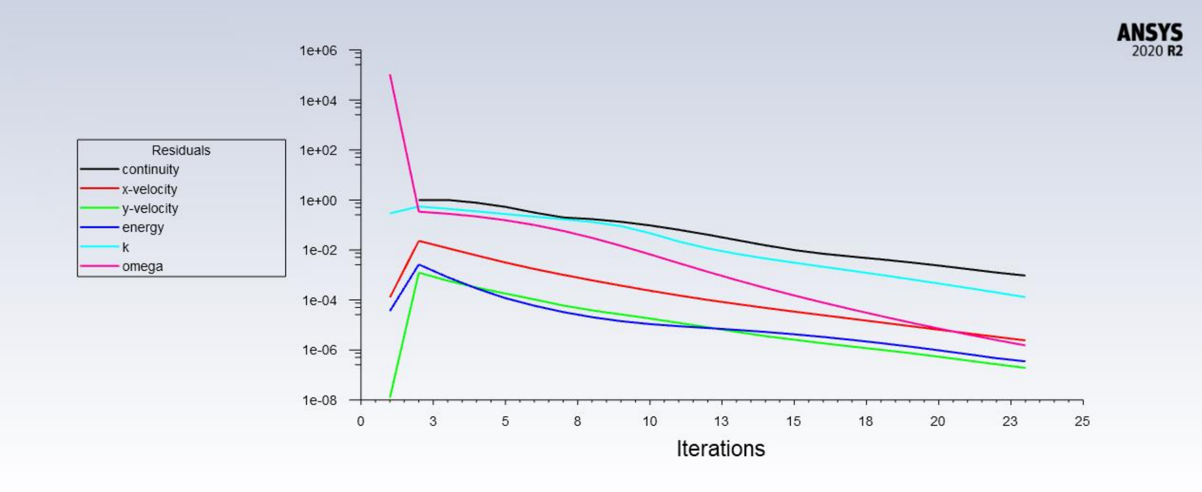

Figure 7) Convergence plot of Smooth channel coarse mesh converging at the 29th iterations

DISCUSSION OF RESULTS

Coarse vs Fine Mesh

Type of Mesh	X-Velocity (m/s)	Temperature (K)	Turbulence Kinetic Energy (m^2/s^2)
Fine	49.999506	387.30042	54.113987
Coarse	50.000165	388.49227	58.481592

Table 1) Results of the Corrugated channel Fine and Coarse meshing comparing the X-Velocity, Temperature and TKE

Table 1 presents the results obtained from the simulations in Fluent. Table 1 shows that the coarse mesh produces satisfying results as they are close to the Fine-mesh ones

Smooth vs Corrugated

Type of Channel	Nusselt Number	Total Pressure Loss (Pa)	Total heat transfer rate (W)
Smooth	8843.114	5278.824	-28.320
Corrugated	169.009	2410.249	-1468.311

Table 2) Nusselt number, total pressure loss and total heat transfer rate of the smooth and corrugated channels

The corrugated channel falls in the range of 100-1000, noting it is a convective heat transfer, but the smooth channel is larger than this range. In this case, more convection occurs. The different values are due to the area. As we know the equation for the Nusselt number is

$$Nu = \frac{hD_\mathrm{h}}{\kappa} = \frac{D_\mathrm{h}}{\kappa}\frac{q''}{(T_\mathrm{w} - \bar{T}_\mathrm{f})},$$

and that $D_\mathrm{h} = 4A/D_\mathrm{wp}$, because of the ribs, it increases the area of the corrugated channel.

In addition, the (average) heat flux, q'', plays a role too. For the smooth channel, $q'' = -566394.58\ W/m^2$ and for the corrugated channel $q'' = -26219.841\ W/m^2$. The smooth channel has a larger heat flux as more energy is required to push the fluid through the channel. In Figure 9, the velocity of the fluid near the cold wall ranges from 3.12 to 31.7 m/s and for fluid above ranges from 31.8 to 74.9 m/s. The fluid near the cold wall also extends itself from the inlet to the outlet. Both the low velocity and the distance it extends to, slows down the fluid above the cold wall, meaning more energy required to push the fluid and more energy means a higher heat flux.

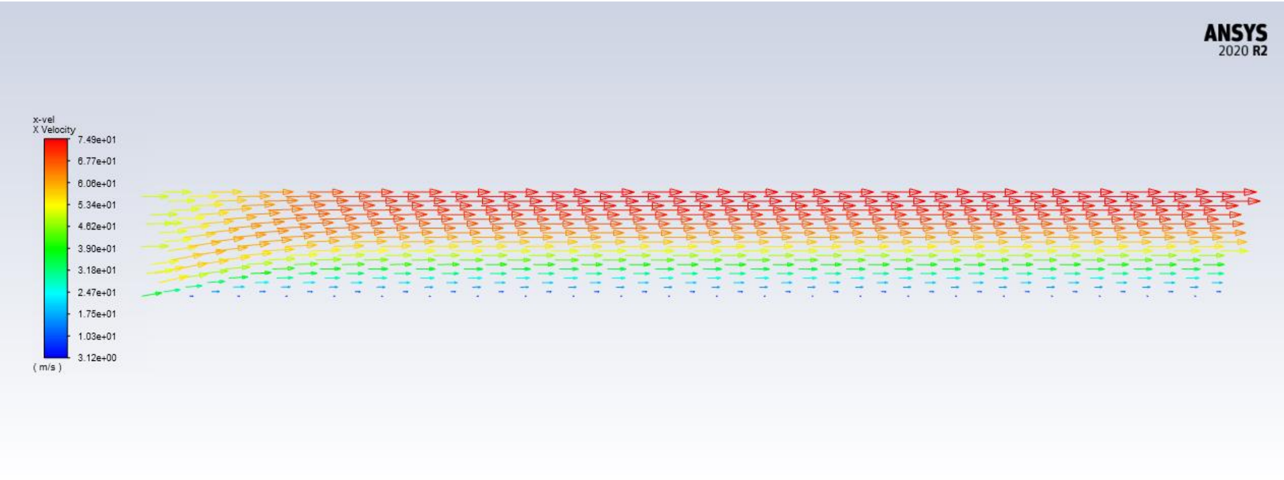

Figure 9) Vector plot of the Smooth coarse mesh

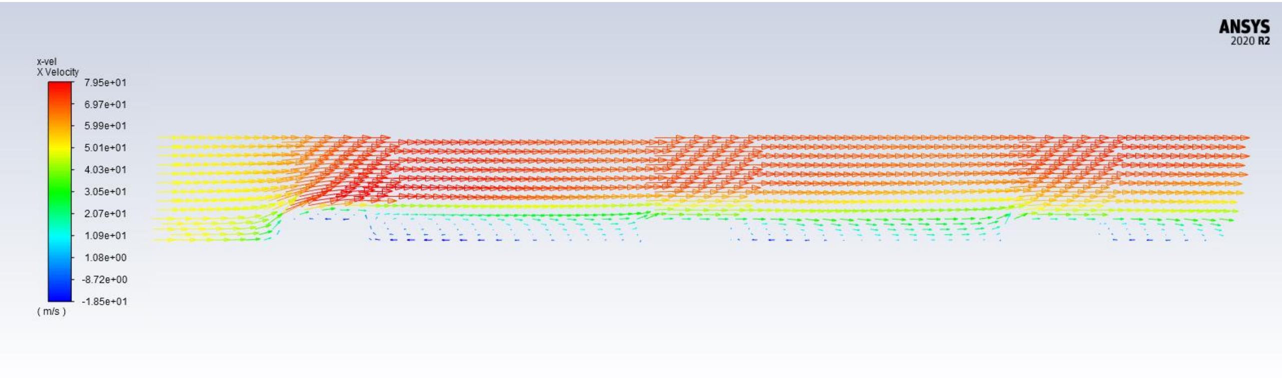

Figure 10) Vector plot of the Corrugated coarse mesh

Due to the presence of the ribs in the corrugated channel, the turbulence flow is increased. As seen in Figure 11. As a cause of the turbulence flow, energy is lost at the ribs and therefore the temperature also drops too, which changes the flow of the fluid as seen in Figure 13, but also suggests heat transfer is significant at the ribs too.

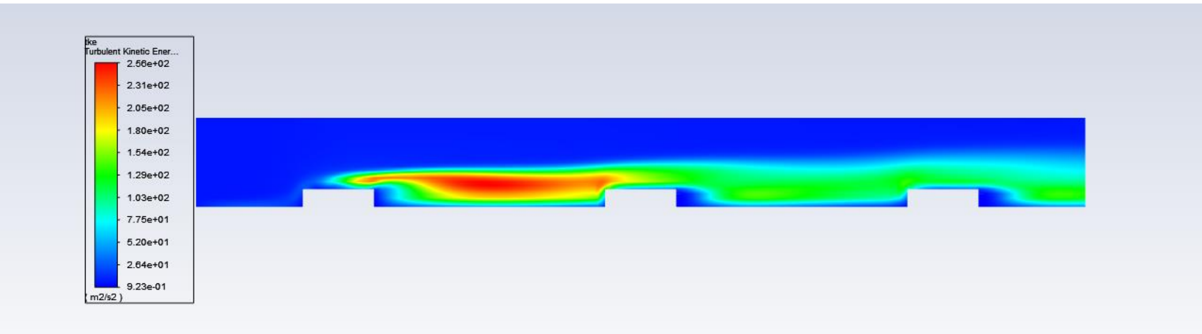

Figure 11) TKE Contour plot of Corrugated channel

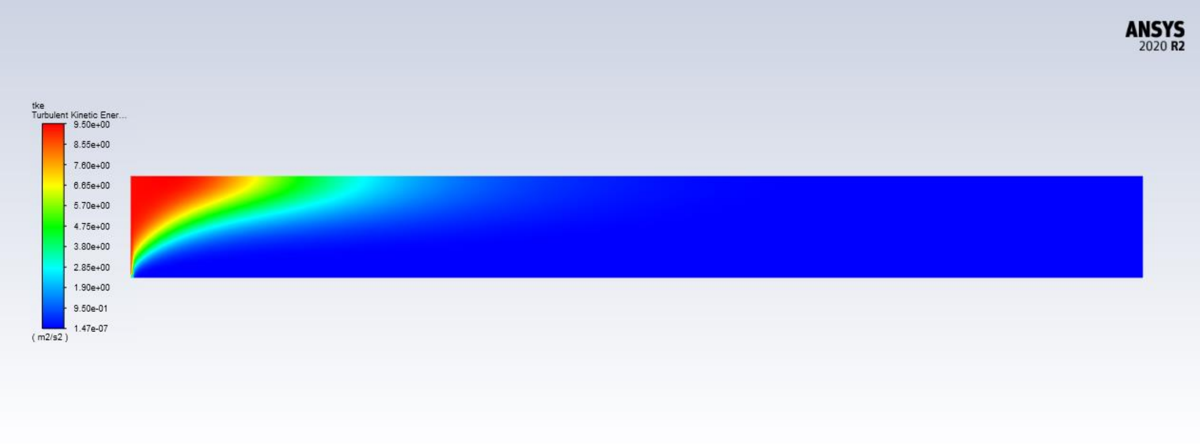

Figure 12) TKE Contour plot of Smooth channel

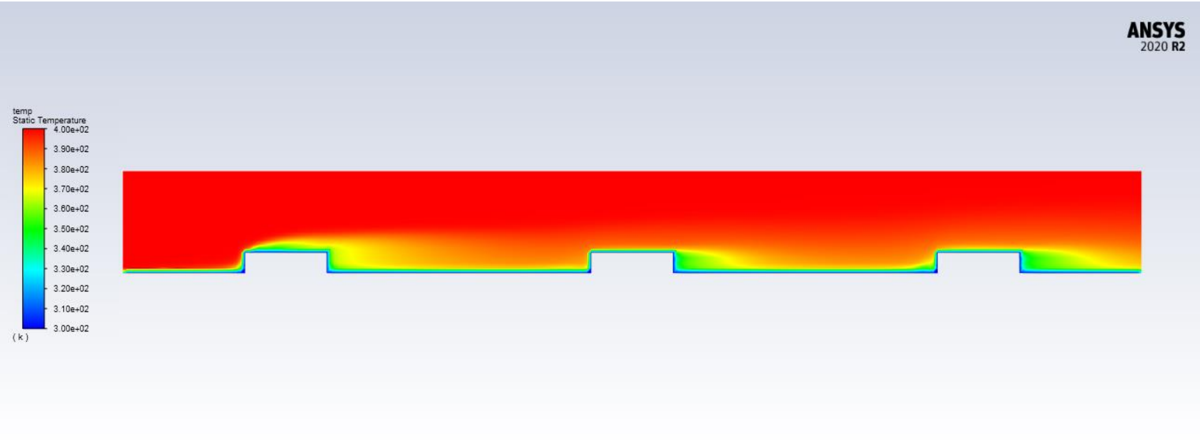

Figure 13) Temperature contour plot of Corrugated channel

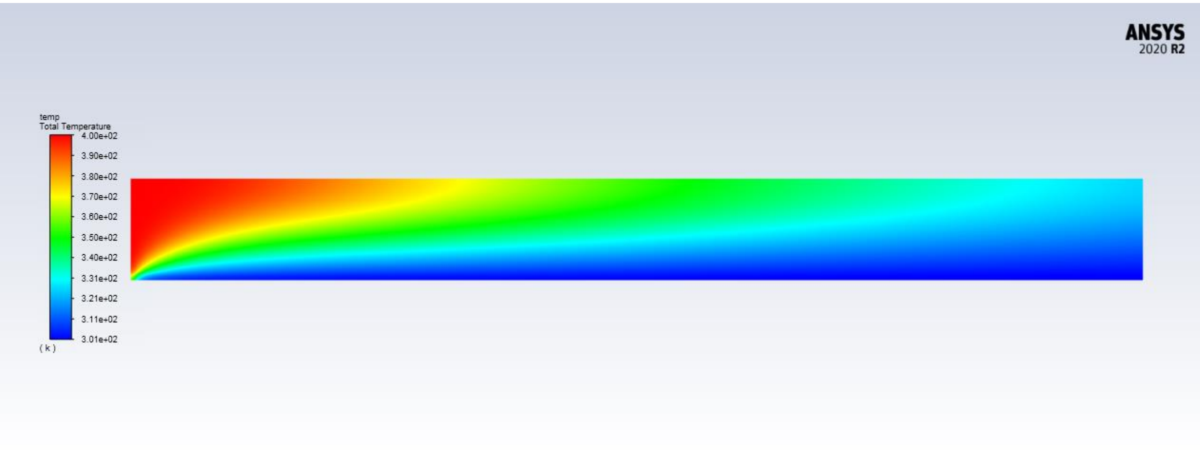

Figure 14) Temperature contour plot of Smooth channel

We have also studied total pressure loss in both cases. From Table 2 we see the Smooth channel has a larger pressure loss, 5278.824 Pa and the Corrugated flow has 2410.249 Pa. This is due to the flow of the fluid in the Smooth domain losing more energy. Which can be seen when you compare the temperature plots in Figure 13 and 14. In Figure 13, the fluid is able to retain its temperature throughout the channel meaning, the drop in pressure is quite low as the fluid still retains a lot of the energy it had at the inlet as shown in Figure 14, the fluid loses most of its heat as it gets to the outlet, so the pressure is significantly lower than at the inlet. These are reflected in Figures 15 and 16.

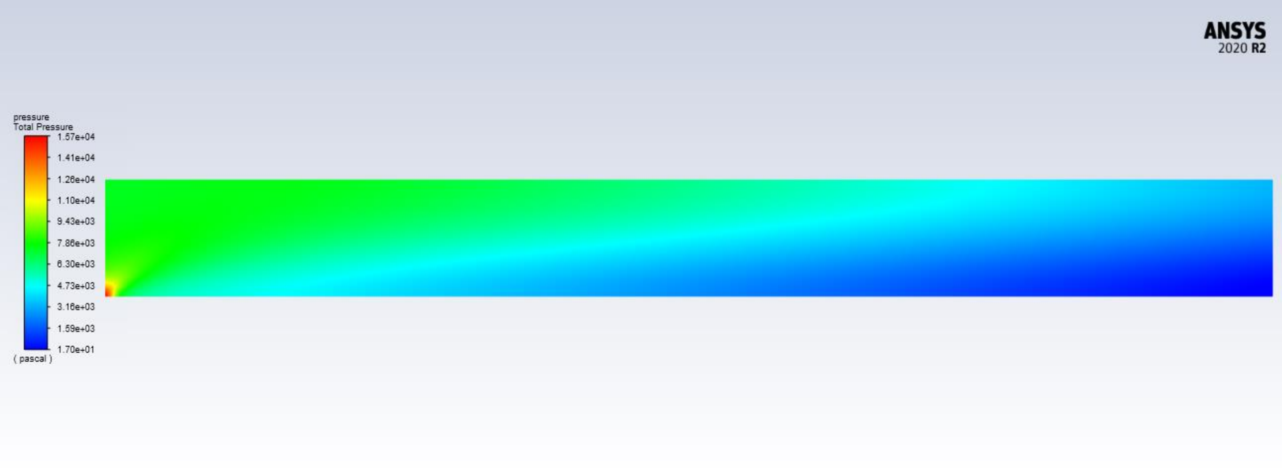

Figure 15) Pressure contour plot of Smooth Channel

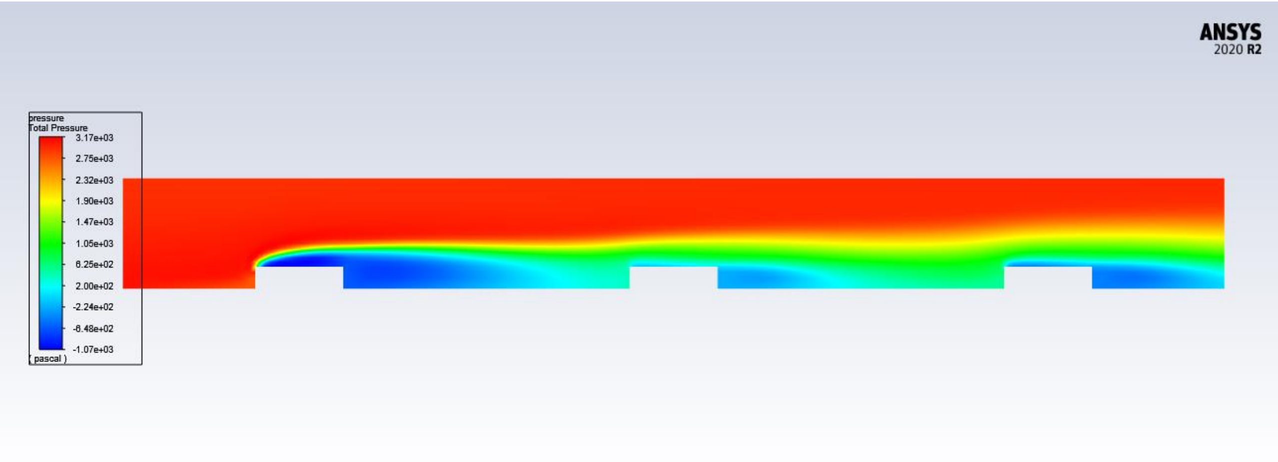

Figure 16) Pressure contour plot of Corrugated Channel

Lastly, we will be looking into the heat transfer performance among the three ribs. We'll be comparing the Total Heat Transfer Rate and Heat Flux at each rib and analysing the result. However, we already know the heat transfer rate for all the walls is -1468.311 W, which is significantly higher than the smooth domain suggesting that the corrugated channel is more efficient at transferring heat. The cause of the difference is what we'll find out.

Rib no.	**Total Heat Transfer Rate (W)**	**Heat Flux (W m^{-2})**
Rib 1 (Leftmost)	-121.18524	-20197.542
Rib 2 (Middle)	-199.75083	-33291.801
Rib 3 (Rightmost)	-158.5631	-26427.181

Table 3) Table of total heat transfer rate and heat flux for each rib in the corrugated channel

Figure 17) Surface Heat Transfer coefficient contour plot of Leftmost Rib

Figure 18) Surface Heat Transfer coefficient contour plot of Middle Rib

Figure 19) Surface Heat Transfer coefficient contour plot of Rightmost Rib

We have observed the middle rib has the highest heat transfer rate and heat flux. This is understood by looking at Figure 11 where its displays the TKE contour plot of the Corrugated channel. At the first rib, the TKE is at its highest when it passes the rib and flows to the middle, where most of the energy is lost. Therefore, most heat transfer occurs here. The rightmost rib has higher values compared to the leftmost rib due to TKE being higher overall. If you look at Figure 11, you can see the rightmost rib is mainly green and half of the rightmost rib is in the blue area, suggesting low TKE.

CONCLUSIONS

The purpose of this study was to compare the heat transfer in a smooth and corrugated channel, which allowed me to have a better understanding on how meshing works, as well as, conducting a mesh independence study, changing the convergence criteria to allow your solution to converge and that attention to detail is key.

We first started by demonstrating that a coarse mesh can give results like ones of a fine mesh.
From here we compared the Nusselt number and Pressure loss between a smooth and corrugated channel and understood the ribs affected the flow by increasing the energy lost at the ribs which induced more heat transfer to occur there but at the same time prevented heat loss everywhere else in the channel. Whereas in the smooth channel, more energy was lost away from the cold wall, resulting in a higher-pressure loss and a larger temperature drop from the inlet to the outlet. Learning this, you can conclude heat transfer in a corrugated channel is more effective than a smooth one. Also, the position of the ribs and its dimensions affect the heat transfer too.

REFERENCES

[1] V.P. Astakhov, 4 - Environmentally friendly near-dry machining of metals, Editor(s): V.P. Astakhov, S. Joksch, In Woodhead Publishing Series in Metals and Surface Engineering, Metalworking Fluids (MWFs) for Cutting and Grinding, Woodhead Publishing, 2012, Pages 135-200, ISBN 9780857090614, https://doi.org/10.1533/9780857095305.135.
(https://www.sciencedirect.com/science/article/pii/B9780857090614500049)

[2] Uttam Roy, Pranab Kanti Roy, Chapter 7 - Advances in heat intensification techniques in shell and tube heat exchanger, Editor(s): Libor Pekař, Advanced Analytic and Control Techniques for Thermal Systems with Heat Exchangers, Academic Press, 2020, Pages 197-207, ISBN 9780128194225, https://doi.org/10.1016/B978-0-12-819422-5.00007-4.
(https://www.sciencedirect.com/science/article/pii/B9780128194225000074)

FLUENT SUMMARY REPORT

Corrugated Coarse Mesh Summary

(Not all the information could fit, so I deleted everything after the second page)

Fluent
Version: 2d, pbns, sstkw (2d, pressure-based, SST k-omega)
Release: 20.2.0
Title:

Models

Model	Settings
Space	2D
Time	Steady
Viscous	SST k-omega turbulence model
Heat Transfer	Enabled
Solidification and Melting	Disabled
Radiation	None
Species	Disabled
Coupled Dispersed Phase	Disabled
NOx Pollutants	Disabled
SOx Pollutants	Disabled
Soot	Disabled
Mercury Pollutants	Disabled
Structure	Disabled
Acoustics	Disabled
Eulerian Wall Film	Disabled
Potential/Li-ion Battery	Disabled
Multiphase	Disabled

Material Properties

Material: air (fluid)

Property	Units	Method	Value(s)
Density	kg/m3	constant	1.225
Cp (Specific Heat)	j/kg-k	constant	1006.43
Thermal Conductivity	w/m-k	constant	0.0242
Viscosity	kg/m-s	constant	1.7894e-05
Molecular Weight	kg/kmol	constant	28.966
Thermal Expansion Coefficient	1/k	constant	0
Speed of Sound	m/s	none	#f

Material: aluminum (solid)

Property	Units	Method	Value(s)
Density	kg/m3	constant	2719
Cp (Specific Heat)	j/kg-k	constant	871
Thermal Conductivity	w/m-k	constant	202.4

Cell Zone Conditions

Zones

name id type

```
    fluid   21   fluid

  Setup Conditions

    fluid

      Condition       Value
      ---------------------
      Frame Motion?   no
      Mesh Motion?    no

Boundary Conditions
-------------------

  Zones

    name       id   type
    -------------------------------
    inlet      23   velocity-inlet
    outlet     24   pressure-outlet
    cwall_1    25   wall
    cwall_2    26   wall
    cwall_3    27   wall
    cwall_4    28   wall
    symmetry   29   symmetry
    rib1_l     30   wall
    rib1_t     31   wall
    rib1_r     32   wall
    rib2_l     33   wall
    rib2_t     34   wall
    rib2_r     35   wall
    rib3_l     36   wall
    rib3_t     37   wall
    rib3_r     38   wall

  Setup Conditions

    inlet

      Condition                       Value
      -----------------------------------------------------------------
      Velocity Magnitude (m/s)        50
      Temperature (k)                 400
      Turbulent Specification Method   Intensity and Hydraulic Diameter
      Hydraulic Diameter (mm)         20

    outlet

      Condition                         Value
      -------------------------------------------------------------------
      Turbulent Specification Method     Intensity and Hydraulic Diameter
      Backflow Hydraulic Diameter (mm)   20

    cwall_1

      Condition                  Value
      ------------------------------------------
      Thermal BC Type            Temperature
      Wall Motion                Stationary Wall
      Shear Boundary Condition   No Slip
      Wall Surface Roughness     0
```