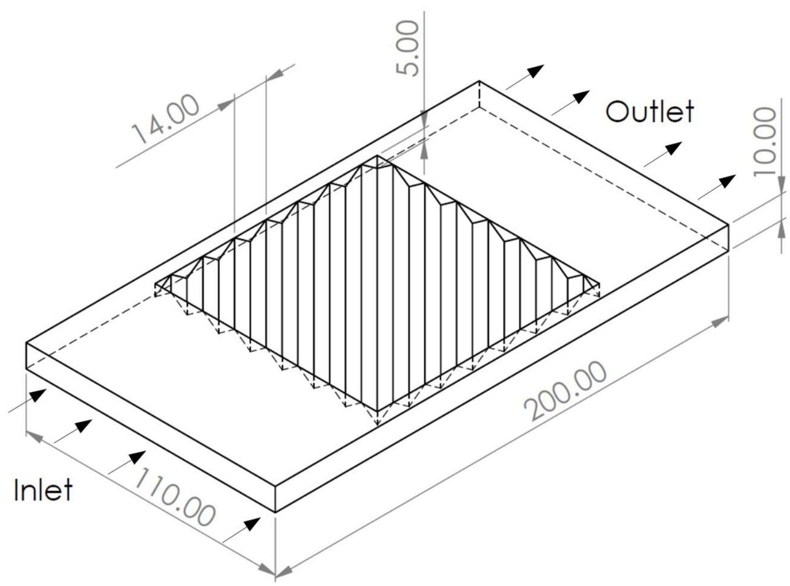
SIMULATION OF NARROW CHANNELS WITH CORRUGATED WALLS

Mohammed Danish

Synopsis

This research migration project aims to do numerical simulations of compact heat exchangers made of corrugated plates using OpenFOAM 7. The channel used for the simulation is formed by only one corrugated plate, while the other plate is flat. The mesh was defined using Ansys Meshing software. A steady-state, bouyantSimpleFoam solver was used in the simulation. A two-equation turbulence model $\kappa - \omega$ *SST* is used, in addition to isothermal flow, heat transfer simulations are conducted for a Reynolds number range (900–1400), for the case of hot water (60ºC) in contact with a constant-temperature wall (20ºC). The analysis executed by Kanaris et. al. using commercial CFD code, *CFX* was taken as a reference.



Geometry of the Heat Exchanger

There are 14 equal sized corrugations present with the distance between the plates at the conduit entrance is 10 mm. Flowing fluid (water) is entering from inlet with velocity (0.07 m/s – 0.1 m/s) corresponding to the Reynolds number and exiting from outlet.

References

Kanaris, Athanasios & Mouza, Aikaterini & Paras, Spiros. (2005). “Flow and Heat Transfer in Narrow Channels with Corrugated Walls”. Chemical Engineering Research & Design. Vol: 83. Page: 460-468. DOI: https://doi.org/10.1205/cherd.04162.

1 Introduction

In the reference paper, the flow and thermal parameters within the complex channels of a plate heat exchanger are investigated using CFD modelling. Corrugated plate heat exchangers, a type of small heat exchanger, have a number of advantages over traditional heat exchangers. Plate exchangers with corrugated walls have a high surface area-to-volume ratio, which improves heat transfer coefficients. A two-dimensional geometry is considered in the reference paper, with heat transfer taking place only on the corrugations for the study. All other walls are considered adiabatic. The paper started with reviewing various articles on the experiments that were conducted, which showed that there was a transitional flow. The results, expressed in terms of friction factor, wall shear stress, wall heat flux, and local Nusselt numbers, are consistent with other researchers' descriptions of fluid motion inside similar conduits. The computed mean heat transfer coefficients and friction factors are found to match rather well.

2 Governing Equations and Models

OpenFOAM 7 was used to replicate the paper's results. The flow simulation is governed by the Navier-Stokes equations for single-phase flows, which are then combined with turbulence models to capture turbulence in the flow. The energy conservation equation is also solved to evaluate the temperature of the field. The following are the governing continuity and momentum equations:

2.1 Governing Equations

$$\nabla \cdot u = 0 \qquad \nabla \cdot (u \otimes u) - \nabla \cdot R = -\nabla p + S_u$$

$$\frac{\partial \rho E}{\partial t} + \nabla \cdot (\rho \mathbf{U} E) + \nabla \cdot (\mathbf{U} p) = -\nabla \cdot \mathbf{q} + \nabla \cdot (\boldsymbol{\tau} \cdot \mathbf{U}) + \rho r + \rho \mathbf{g} \cdot \mathbf{U}$$

2.2 Turbulence Model

Both wall shear stress and wall heat flux are over predicted by the typical k–ε model employing wall functions, notably in the lower Reynolds number range. Over prediction of turbulent length scale in the region of flow reattachment, which is a distinctive phenomenon happening on the corrugated surfaces in these geometries, causes the over prediction. The k–ω model, which replaces turbulence dissipation ε with turbulence frequency ω, looks to be more robust, even in complicated situations, and does not require a fine grid near the wall.

The fundamental problem of the k– ω model is that it is sensitive to free stream turbulence frequency v beyond the boundary layer, which influences the solution. To prevent this, a combination of the two models, k– ε and k– ω, i.e. the SST (Shear-Stress Transport) model, is proposed. Using particular 'blending functions,' the SST model may automatically switch between the two aforementioned turbulence models, activating the k– ω model at the wall and the k– ε model for the rest of the flow. The equations are k– ω SST are–

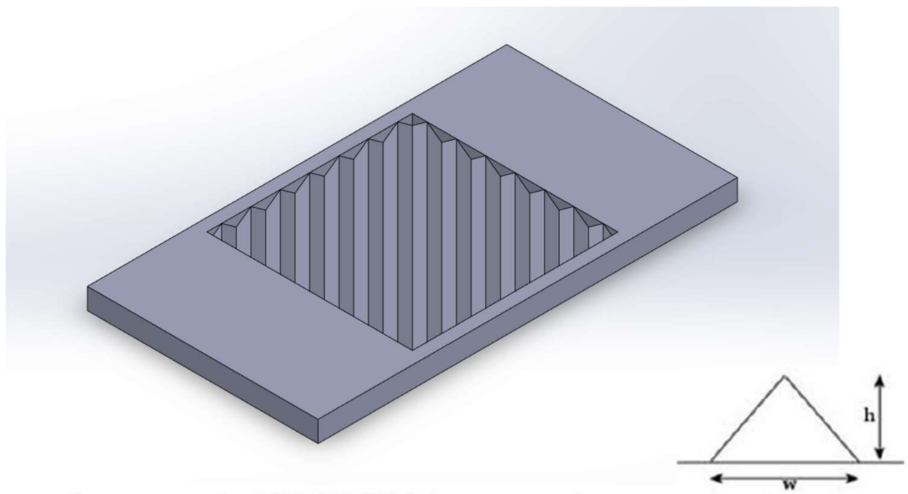
$$\frac{D}{Dt}(\rho\omega) = \nabla \cdot (\rho D_\omega \nabla \omega) + \frac{\rho \gamma G}{\nu} - \frac{2}{3}\rho\gamma\omega(\nabla \cdot \mathbf{u}) - \rho\beta\omega^2 - \rho(F_1 - 1)CD_{k\omega} + S_\omega$$

$$\frac{D}{Dt}(\rho k) = \nabla \cdot (\rho D_k \nabla k) + \rho G - \frac{2}{3}\rho k(\nabla \cdot \mathbf{u}) - \rho\beta^* \omega k + S_k \qquad \nu_t = a_1 \frac{k}{\max(a_1\omega, b_1 F_{23}\mathbf{S})}$$

3 Simulation Procedure

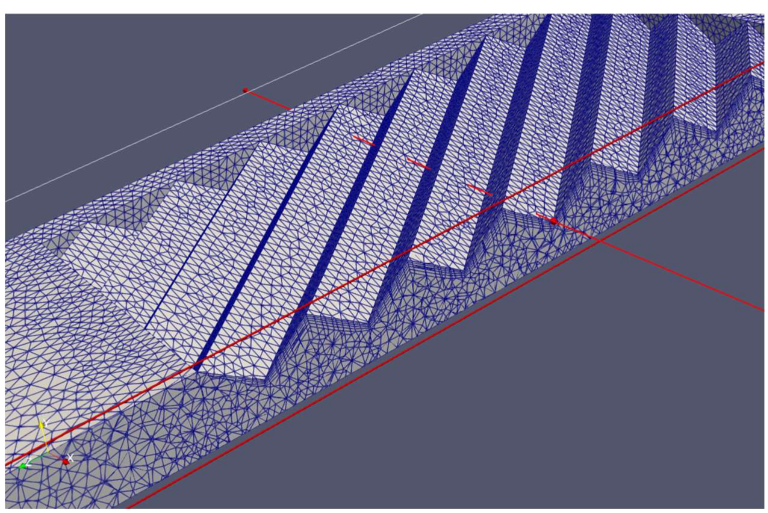
3.1 Geometry and Mesh

The geometry was created in SOLIDWORKS software, with the dimensions as given in the paper. The dimensions were matched to the experimental studies which the paper compares it to. The corrugations are built uniformly at angle 45º to the flow. The corrugation pitch *h* is set to 5mm and the corrugation width *w* is 14mm. Further, the geometry was imported into ANSYS where meshing was done.



Geometry viewed in SOLIDWORKS & the corrugation dimensions on the right

Unstructured tetrahedral mesh was performed with a 5 layer inflation present at the corrugations, so that the boundary layer is captured properly. Varying the mesh cell sizes, different meshes were generated. After analysing with 3 different meshes for the geometry, the one with 294,526 cells was chosen. A coarser mesh (200,000 cells) had high y+ values near the corrugations which was not necessary and the finer mesh (800,000 cells) was giving the same results but was taking a lot of computational resources.



Clipped mesh of the geometry

The final mesh had 2.94 Lakh elements, the max aspect ratio was 5.4. The mesh non-orthogonality had a maximum value of 71.9 and an average value 20.9, Further it was noted that orthogonality was above 50 only for 5 cells. The max skewness was 2.65. It was imported to OpenFOAM using the fluent3DMeshToFoam command.

The geometry had 6 sections, which were defined using named selections. The inlet and outlet are shown in fig. 1, the bottom plate is named Flat_Plate, and the side walls of the channel are named Side_walls. On the top there are Corrugations present at the centre and the rest of the top plate is named as Corrugated_Plate.

3.2 Initial and Boundary Conditions

There are six boundary conditions to be defined for the patches mentioned above. The fluid inlet velocity is determined by the Reynolds number of the flow, for Re =900 it is 0.07 m/s. Walls have no slip condition, and the outlet has a zero gradient velocity. The outlet acts as a pressure outlet and all other patches have been given zero gradient. The fluid is entering at a temperature of 333 K and the Corrugations have a fixed temperature of 293 K. All other walls are considered adiabatic for simplicity.

The k value was calculated for the initial conditions and velocity and it is of the order of e-5 initially, the turbulence is very low for this Reynolds number, kqRWallFunction has been used for the walls. The inlet is calculated turbulentIntensityKineticEnergyInlet with an intensity of 0.0683. Similarly, ω value was calculated = 15.28 and the inlet was calculated using TurbulentMixingLengthFrequencyInlet with a mixing length of 0.0007m.

The table below summarises all the conditions –

Field	**Velocity**	**Pressure**	**Temperature**	**k**	**ω**
Initial	0	0	293 K	3.434E-05 m2/s2	15.2841 1/s
Inlet	0.07 m/s	zeroGradient	333 K	turbulentIntensityKineticEnergyInlet	TurbulentMixingLengthFrequencyInlet
Outlet	zeroGradient	uniform	zeroGradient	zeroGradient	zeroGradient
Corrugations	noSlip	zeroGradient	293 K	kqRWallFunction	omegaWallFunction
Corrugated_Plate	noSlip	zeroGradient	zeroGradient	kqRWallFunction	omegaWallFunction
Side_walls	noSlip	zeroGradient	zeroGradient	kqRWallFunction	omegaWallFunction
Flat_Plate	noSlip	zeroGradient	zeroGradient	kqRWallFunction	omegaWallFunction

The properties of water was input into the thermophysical properties file. General thermo-physical model calculation based on enthalpy h (or internal energy) and density ρ was used. It was considered as a pure mixture with constant internal energy and Boussinesq approximation was taken. There is no gravity considered in this model. The properties are summarised in the table-

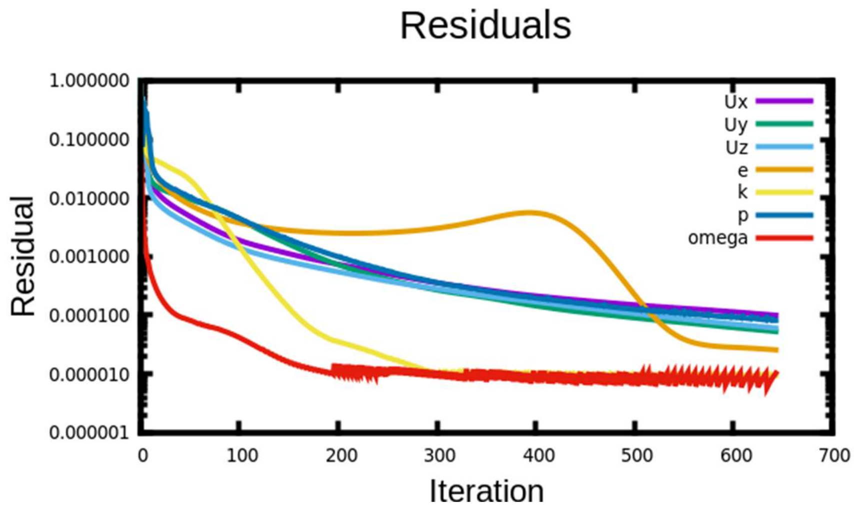
Molecular Weight	18 g
Density rho	1000 kg/m3
Initial Temperature	333K
Thermal Expansion, Beta	0.000522 1/K
Specific Heat Cp	4187 J/kg K
Specific Heat Cv	3980 J/kg K
Heat of Fusion	334 kJ/kg
Dynamic Viscosity	0.00077 N s/m2
Prandtl Number	5.4

3.3 Solver

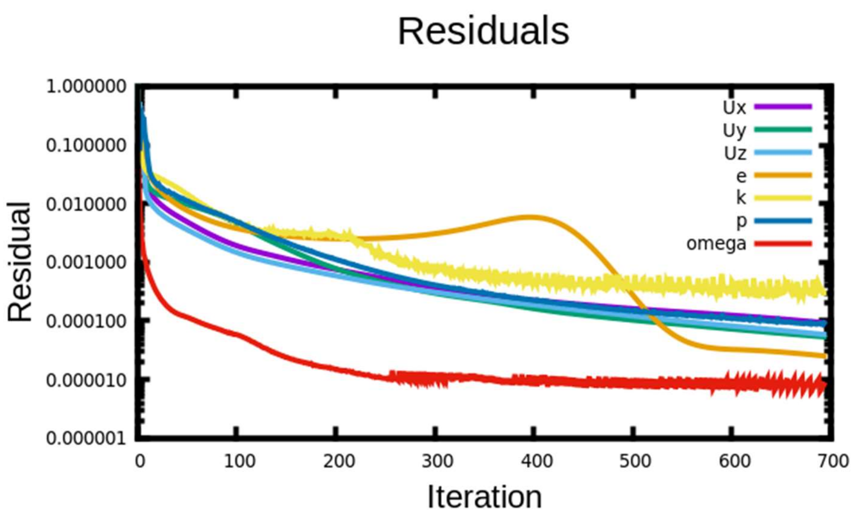
A steady-state incompressible turbulent flow-based bouyantSimpleFoam solver is used to run governing equations in the discretized domain. The solver uses SIMPLE (Semi-Implicit Method for Pressure Linked Equations) algorithm to evaluate NS equations. The solver follows a segregated solution strategy. This means that the equations for each variable characterizing the system (the velocity u, the pressure p, and the variables characterizing turbulence) are solved sequentially. For implementation in Open Foam, the ∇p and g terms are arranged to form p_rgh = p-$\rho g \boldsymbol{r}$ (r is the position vector).

For the convergence, conditional strategy used with 1000 maximum iterations or 10^{-4} convergence criteria. The residuals do drop quickly for all three Re values, 900, 1150 & 1400 in about 650, 700 & 800 iterations.

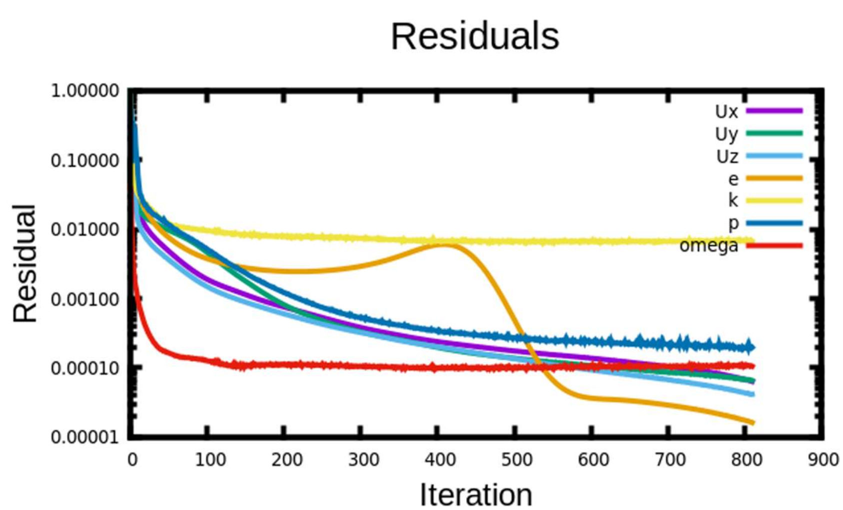
The y+ value is important in the simulations as it tells us how effectively the boundary layer has been captured. The yPlus post processing function in OpenFOAM yielded that y+ is under 20 for all the 3 cases of Reynolds Number.



Residuals for Re = 900



Residuals for Re = 1150



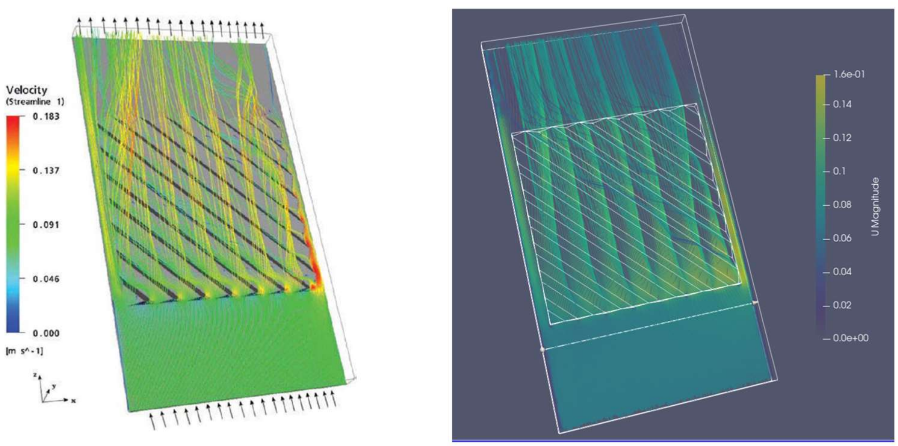
Residuals for Re = 1400

4 Results and Discussion

For the comparison of results, the plots were that were given in the paper have been replicated. The paper had different contour plots for different Reynolds number, so only those specific cases are compared.

Velocity-

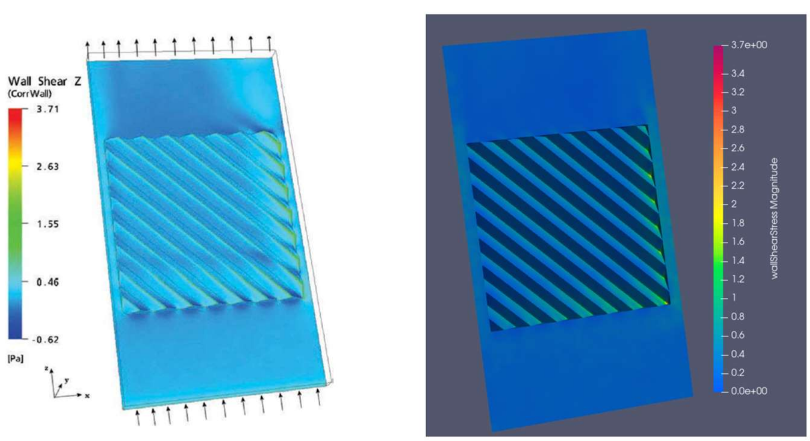
The velocity contours show the dominant role of the side-channels in flow distribution and suggest that fluid flow is mainly directed to the right side-channel of this model plate. Part of the fluid flows over the corrugation crests and after being 'reflected' on the right side wall, follows the furrows and reaches the opposite side-channel. The OpenFOAM simulation on the right also predicts the same.



Velocity Contours at Re -900 (CFX on the left & OpenFOAM on the right)

Wall Shear Stress

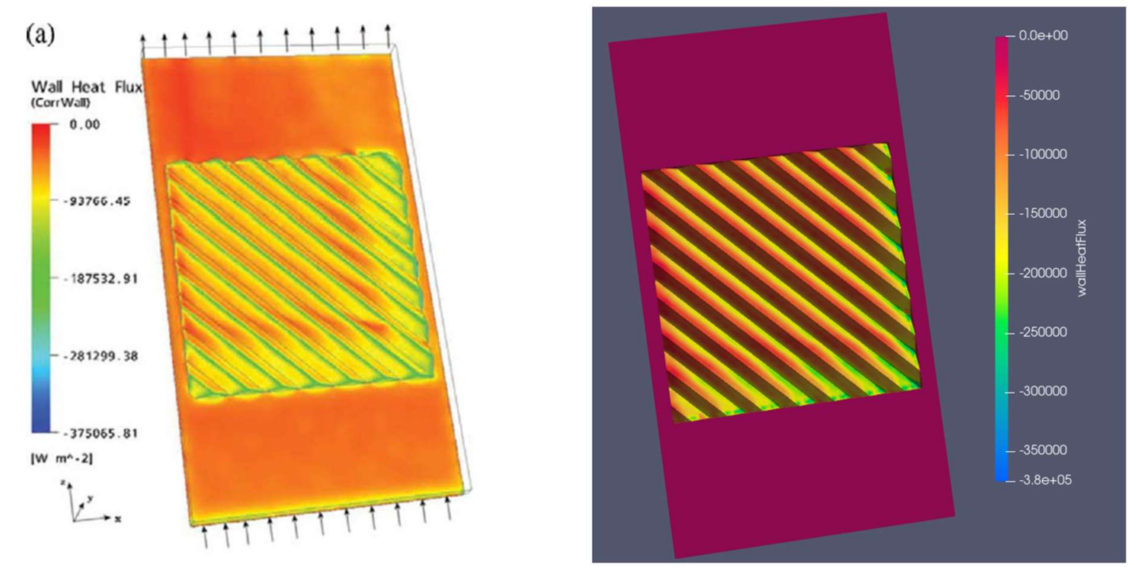
The Shear stress increases with Reynolds number, as expected, and it attains its maximum value at the crests of the corrugations.



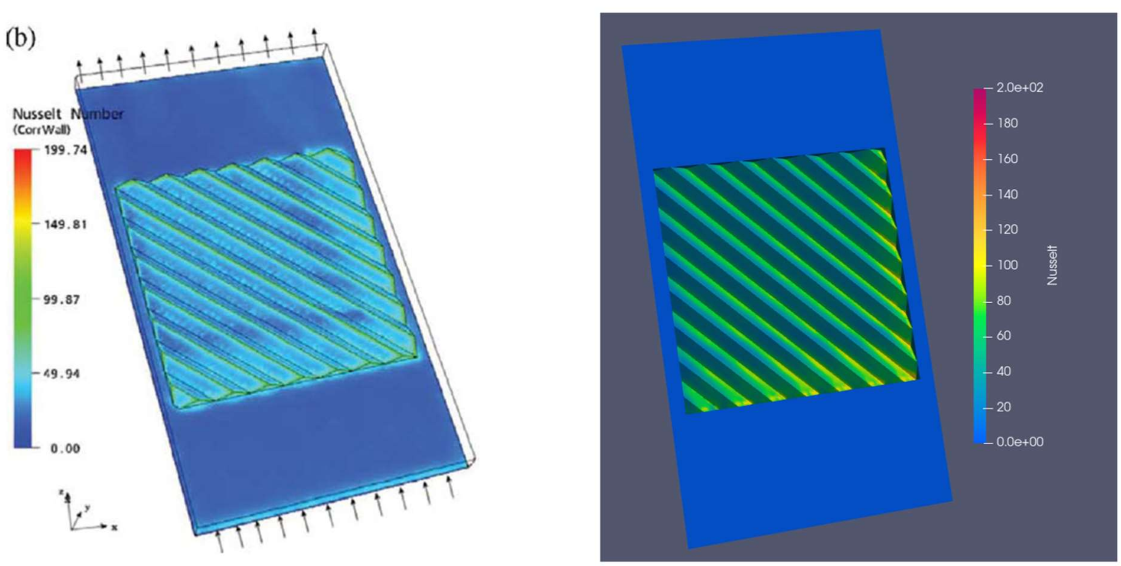
Wall Shear Stress at Re -1400 (CFX on the left & OpenFOAM on the right)

Heat Flux & Nusselt Number

Over the corrugated wall, the wall heat flux and local Nusselt number distributions are plotted for Re = 1400. For all Reynolds numbers investigated, the distributions of heat flow and Nu are comparable. The local Nusselt number reaches its highest value near the top of the corrugations, which is evident. On the crests of the corrugations, the mass and heat transfer performance peaks. This demonstrates that the corrugations have a significant impact on the flow distribution as well as the heat transfer results.



Heat Flux at Re = 1400 (CFX on the left & OpenFOAM on the right)



Nusselt Number at Re = 1400 (CFX on the left & OpenFOAM on the right)

Nusselt Number Comparison

The Nusselt number is given by -

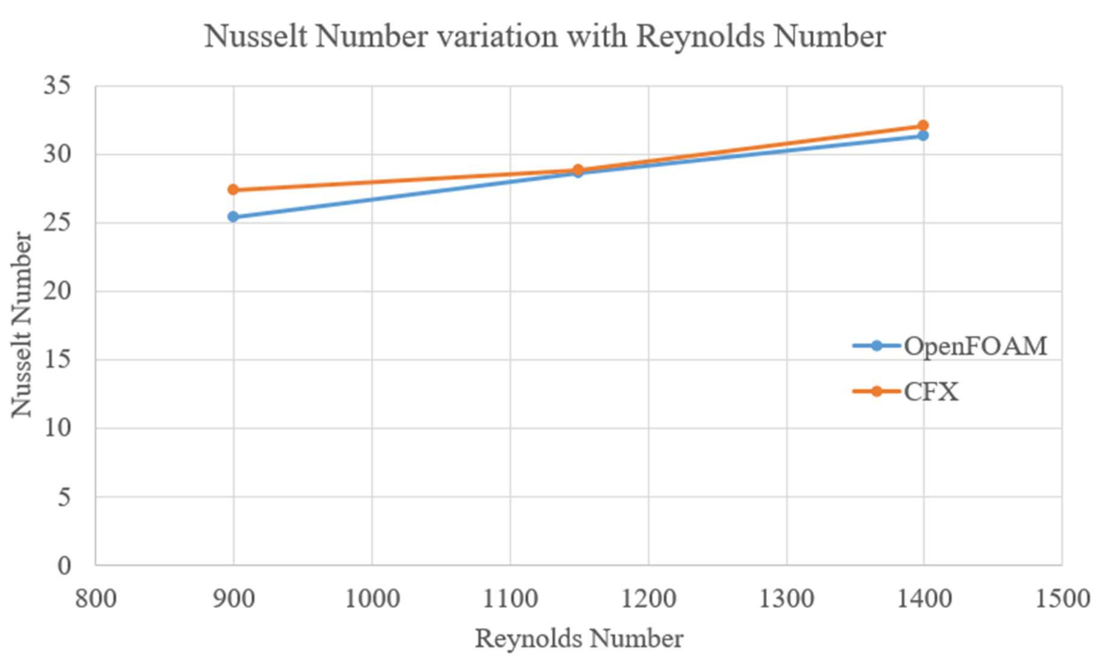
$$Nu_x = \frac{q \cdot d}{(T_b - T_w)k}$$

where q is wall heat flux, d is the distance between the plates (= 0.01m), T_w is the wall temperature, T_b is the local fluid temperature and k thermal conductivity (= 0.63 W/m K).

The Nusselt number is calculated using Integrator in ParaView which yielded these results. The average value of wall Heat Flux is taken for the entire top plate, the temperature near the corrugations is volume averaged after clipping the planes. The paper also has tabulated the overall average Nusselt number. Comparing the OpenFOAM and the CFX values we see that the error is small.

Re	Average Wall Heat Flux (W/m2)	Average Temperature at corrugations (K)	Nu – OpenFOAM	Nu - CFX	Error
900	50075.3	324.32	25.38	27.3	7.04%
1150	58024.4	325.21	28.60	28.8	0.71%
1400	64909.0	325.87	31.35	32	2.03%

Comparison of the Nusselt Numbers –



5 Conclusions

There is an increase in Heat transfer which is seen from the Nusselt number with increase in Reynolds number. The severity of turbulence increases as the inlet velocity rises, resulting in more mixing and a nearly uniform temperature at the output at steady state. The fluid qualities play a big part in determining how high the temperature rise will be.

The current study investigates OpenFOAM's capacity to estimate flow and heat transfer characteristics in a narrow channel with a corrugated wall of a specific corrugation angle, width, and height. The usage of a CFD code allows for the computation of numerous geometrical configurations in order to assess and investigate their impacts. An engineer can then optimise the efficiency (i.e., the ratio of heat transfer to friction losses) for a specific geometry. By the results and very less deviation of contour plots and engineering parameters, we can conclude that OpenFOAM indeed gives us accurate results.

References

Kanaris, Athanasios & Mouza, Aikaterini & Paras, Spiros. (2005). “Flow and Heat Transfer in Narrow Channels with Corrugated Walls”. Chemical Engineering Research & Design. Vol: 83. Page: 460-468. DOI: https://doi.org/10.1205/cherd.04162.