FOSSEE, IIT Bombay
OpenFOAM Research Migration Project
April 22, 2022

Effect of amplitude of walls on thermal and hydrodynamic characteristics of laminar flow through an asymmetric wavy channel

Abhishek Raghuvanshi
Department of Mechanical Engineering, IIT Bombay

Synopsis

This research migration project aims to do numerical simulations of the forced convection of a laminar flow in asymmetric wavy channels under different flow conditions using `OpenFOAM-v2012`. The geometry and mesh were made using third-party meshing software due to being complex and then imported into OpenFOAM using the `fluentMeshtoFoam` utility. A steady-state, incompressible, laminar flow was validated with `buoyantBousinessqSimpleFoam` solver being used in the simulation. Various flow parameters (namely, Re, wave amplitudes, type of channel- *Linearly Incr. Ampl. Channel(LIAC), Linearly Decr. Ampl. (LDAC), Constant Ampl. (CAC)*) were studied and compared to find out the optimum wavy channel for heat transfer. The analysis executed by Sumit et. al. [1] using commercial CFD code Fluent was taken as a reference.

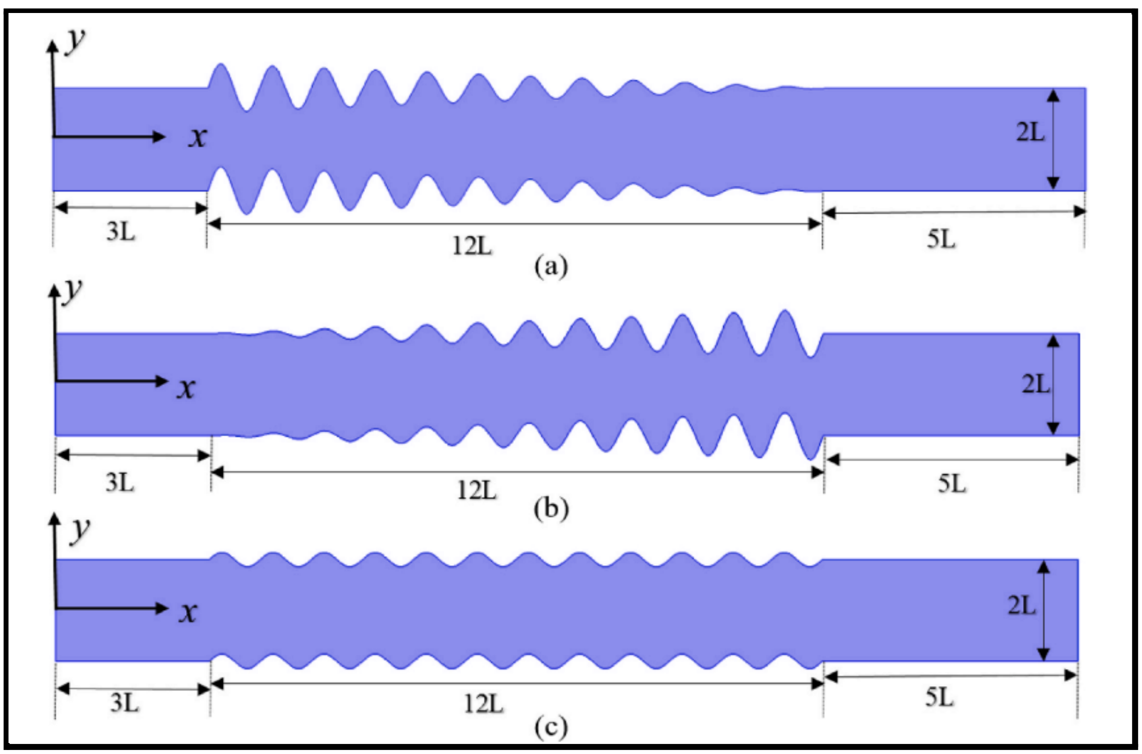


Figure 1: Geometry and Dimension

References

[1] Mehta, Sumit & Pati, Sukumar & Baranyi, Laszlo. (2022). Effect of amplitude of walls on thermal and hydrodynamic characteristics of laminar flow through an asymmetric wavy channel. Case Studies in Thermal Engineering. 31. 101796. 10.1016/j.csite.2022.101796.

URL: https://www.sciencedirect.com/science/article/pii/S2214157X22000429?via%3Dihub

1 Introduction

In the reference paper [1], an asymmetric wavy channel test case was analyzed to compare numer ical results generated by Ansys against experimental data. A two-dimensional geometry is considered in the reference paper from data given in the test cases. The channel geometry given in fig. 1 can be divided into three sections: an inflow channel, the asymmetric wavy heated section, and an outflow channel. The inflow channel portion is sufficiently long to obtain fully developed laminar channel flow. In order to capture flow separation in the wavy section and outflow channel, results of six different Reynold's number for 3 types of channels for 3 different amplitudes were generated using commercial CFD code Fluent and compared against experimental data. The flow is incompressible laminar and neglects buoyancy effects. The reference study also did a grid conversion study with LDAC of A=0.04. The pressuredrop ratio, Nusselt number, some speciaal parameters such as percent effectiveness and performance factor, velocity stream lines and temperature contours values were compared to conclude.

2 Governing Equations and Models

To reproduce results generated by Sumit et al [1], OpenFOAM-v2012 software was used. The Navier-Stokes equations for single-phase incompressible flows govern the simulation and are later compiled with energy equation to capture temperature and heat transfer in the flow. The governing continuity and momentum equations and energy equations are given by:

Continuity Equation : $$\nabla \cdot \left(\rho \vec{U}\right) = 0$$

Navier-Stokes Equation : $$\frac{\partial U}{\partial t} + \nabla \cdot (\vec{v}\vec{v}) - \nabla \cdot (\nu \nabla \vec{v}) = -\nabla p$$

Energy Equation : $$\frac{\partial T}{\partial t} + (u \cdot \nabla)T = \alpha \nabla^2 T$$

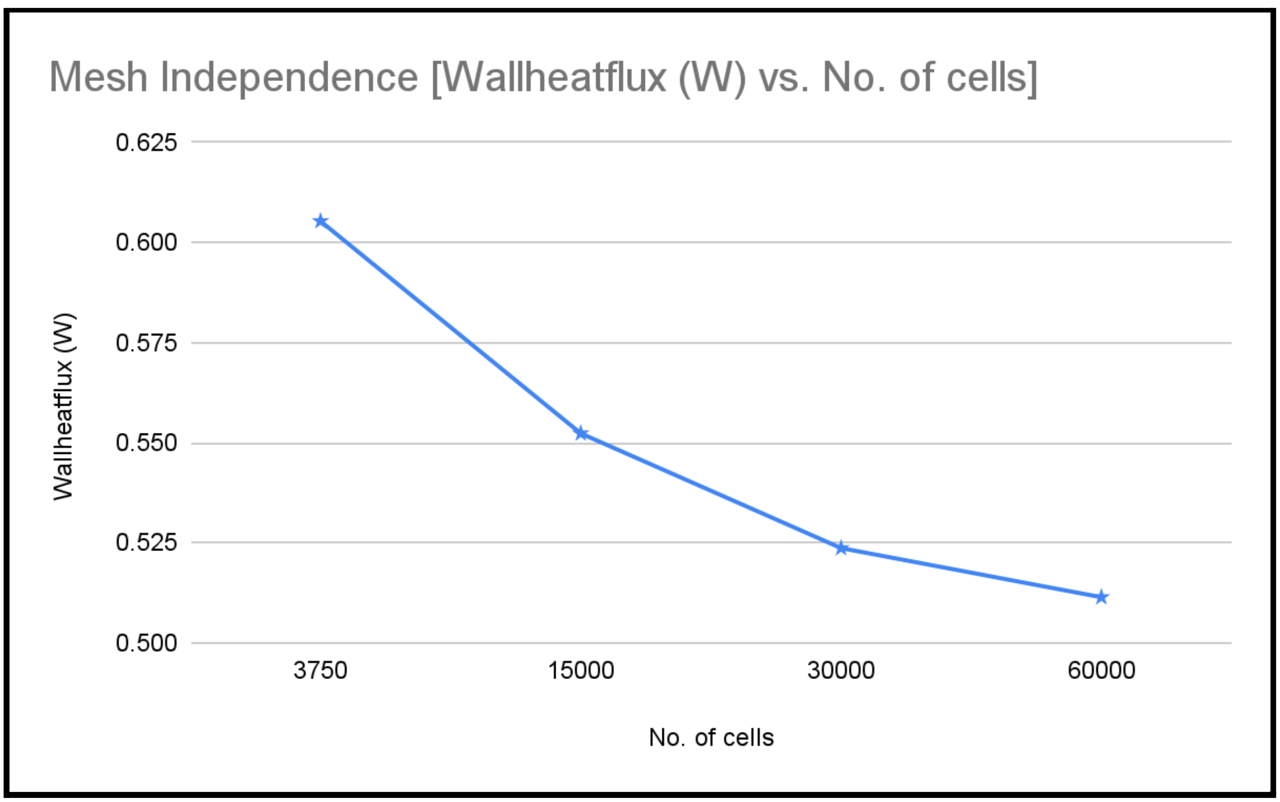
3 Simulation Procedure

3.1 Geometry and Mesh

The geometry presented in fig. 1 has three sections: an inlet, asymmetric wavy portion and an outlet channel. The inlet channel is sufficiently long to generate a fully developed flow profile at the beginning of the wavy channel. The asymmetric diffuser angle is. A 2-dimensional geometry has been considered in the study, created using Ansys spaceclaim and Meshing. And then imported into openfoam using fluentMeshtoFoam utility. Mesh/Grid refinement/independence study us done for the most complex case, viz., LDAC (A=0.04, Re=200). It is observed that there is less than 5% change in wallHeatFlux, so the grid is fine enough to not affect the results, as some accuracy is sacrificed for saving computational time, as one simulation took around 20 mins. As we have converge steady state simulation for 60 cases (3x3x6 Geometry x amplitude x Re + 6 plane wall cases).

For a4ldac Re=200

S.no	No. of cells	Wallheatflux (W)
1	3750	0.605343
2	15000	0.55233
3	30000	0.52369
4	60000	0.51142



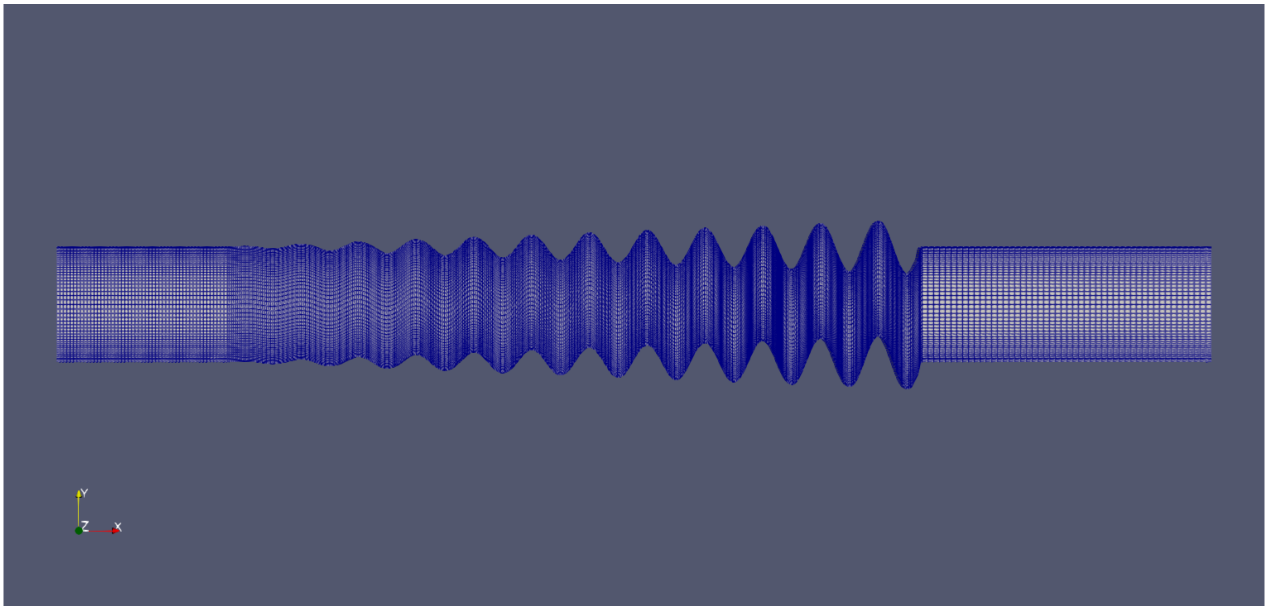


Figure 2: Computational grid

A structured grid consisting of 50×50 cells for inlet and outlet region and 500×50 for the wavy region in the stream-wise and wall-normal direction respectively has been used. Thus making up for a total of 30000 cells. For refinement near the wall, expansion ratios 10 and 0.1 were used for the top and bottom portions, as shown in fig. 2.

3.2 Initial and Boundary Conditions

There are four boundaries that required boundary conditions: An inlet, outlet, wall(inlet and outlet walls) and wave (wavy region wall). The other two boundaries are front and back; that set as empty boundaries because our analysis is 2D. At the inlet velocity, p and T are specified, as shown in table 1. At the walls, velocity, pressure and Temperature conditions are specified.

Boundary	**Pressure (Value)**	**Velocity(Value)**	**Temperature(Value)**
Inlet	ZeroGrad	fixedValue(0.1m/s)	fixedValue(300K)
Outlet	fixedValue(0)	ZeroGrad	ZeroGrad
Wall	ZeroGrad	noSlip	ZeroGrad
Wave	ZeroGrad	noSlip	fixedValue(310K)
Front&Back	Empty	Empty	Empty

3.3 Solver

A steady-state for incompressible, turbulent flow-based buoyantBoussinesqSimpleFoam solver is used to run governing equations in the discretized domain. The buoyantBoussinesqSimpleFoam solver uses SIMPLE (Semi-Implicit Method for Pressure Linked Equations) algorithm to evaluate NS equations and for density it uses the Buossinesq approximation (given below) and hence the name of the solver.. 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. The solution of the preceding equations is inserted in the following equation. We used only the laminar and constant density so the solver was good enough.

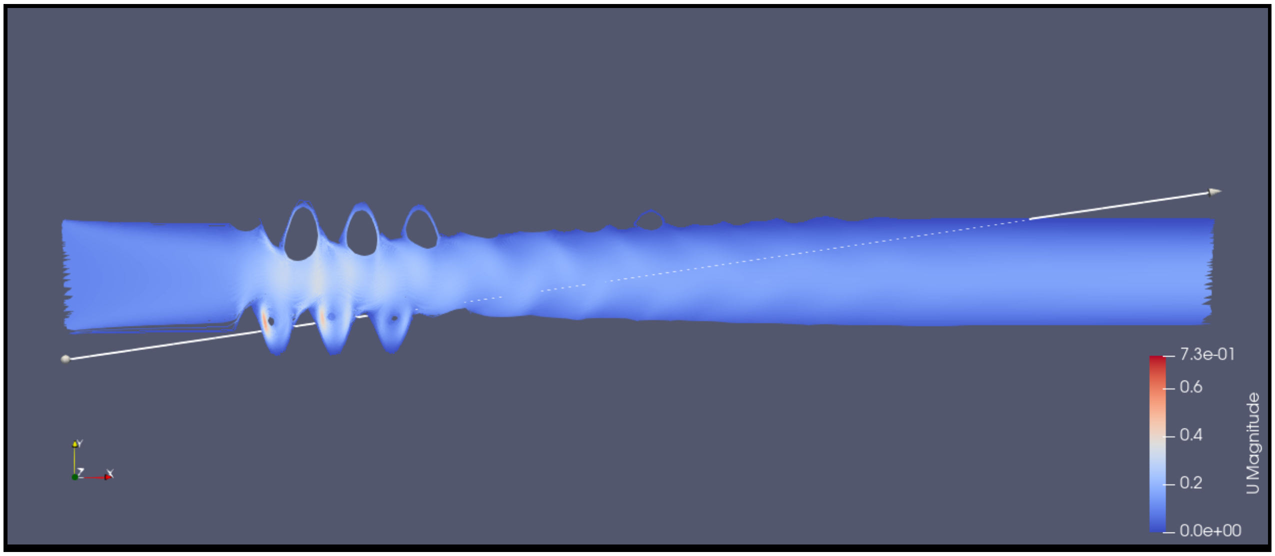
The Buossinesq approximation is that the changes in density due to pressure are negligible as compared to temperature density changes.

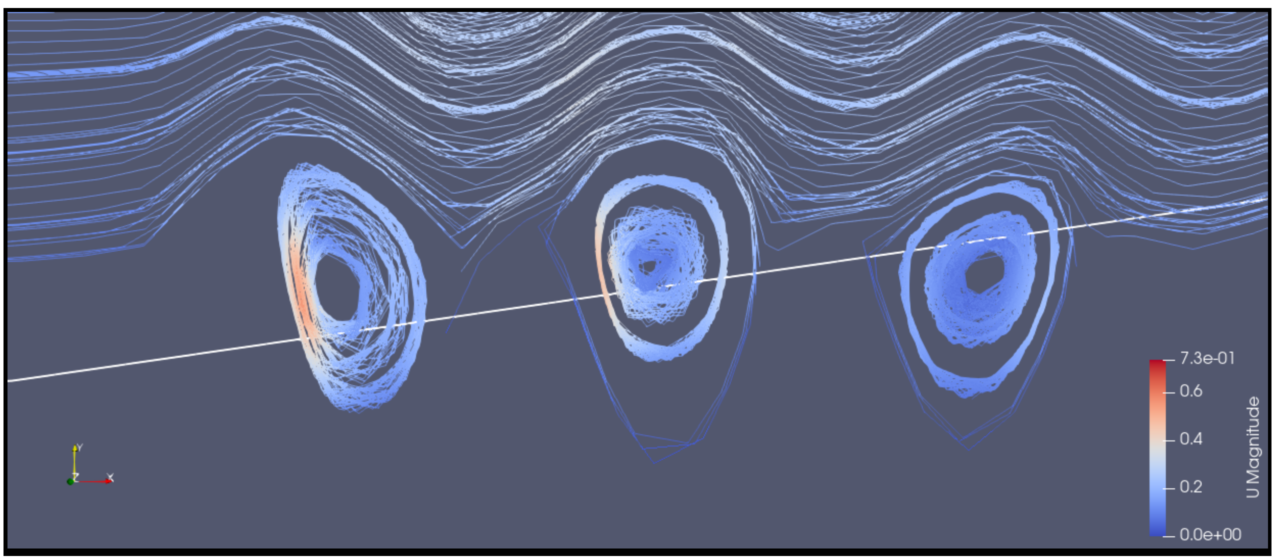
$$rhok = 1 - beta(T - Tref), \text{ valid when } beta(T - Tref)/rhoref << 1$$

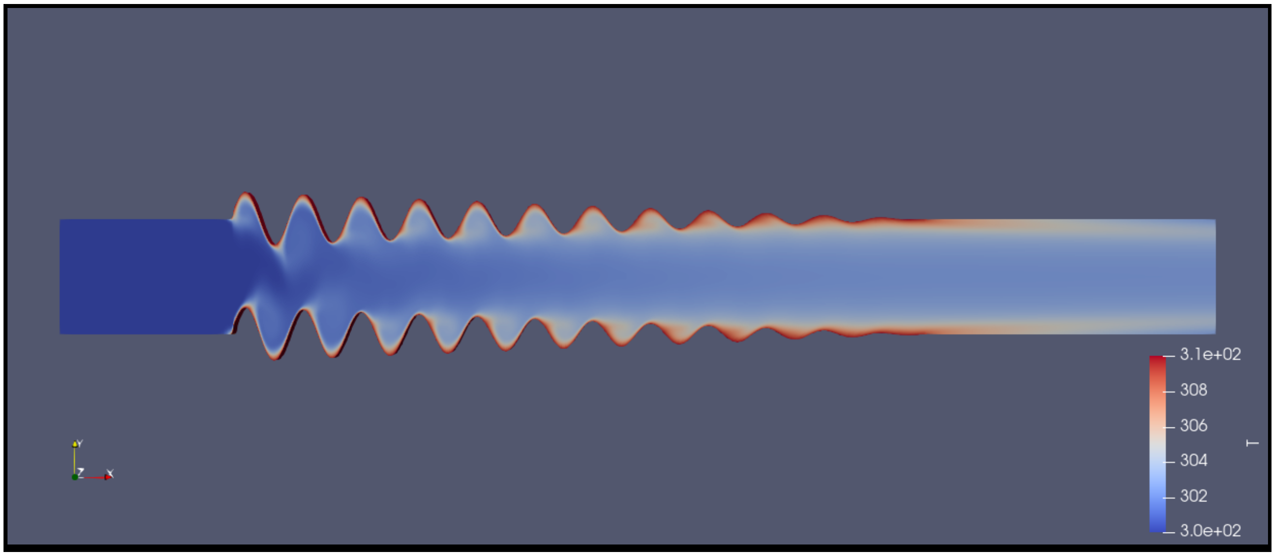
For the convergence, conditional strategy used with 3000 maximum iterations or 10^5convergence criteria. The standard LDAC and LIAC cases converged relatively earlier than CAC, for higher Reynolds number we needed more iterations. Whereas the CAC for Re=200,A=0.04 was the only case out of the 60 cases which did not converge at given criteria untill 2000 iterations and stopped running.

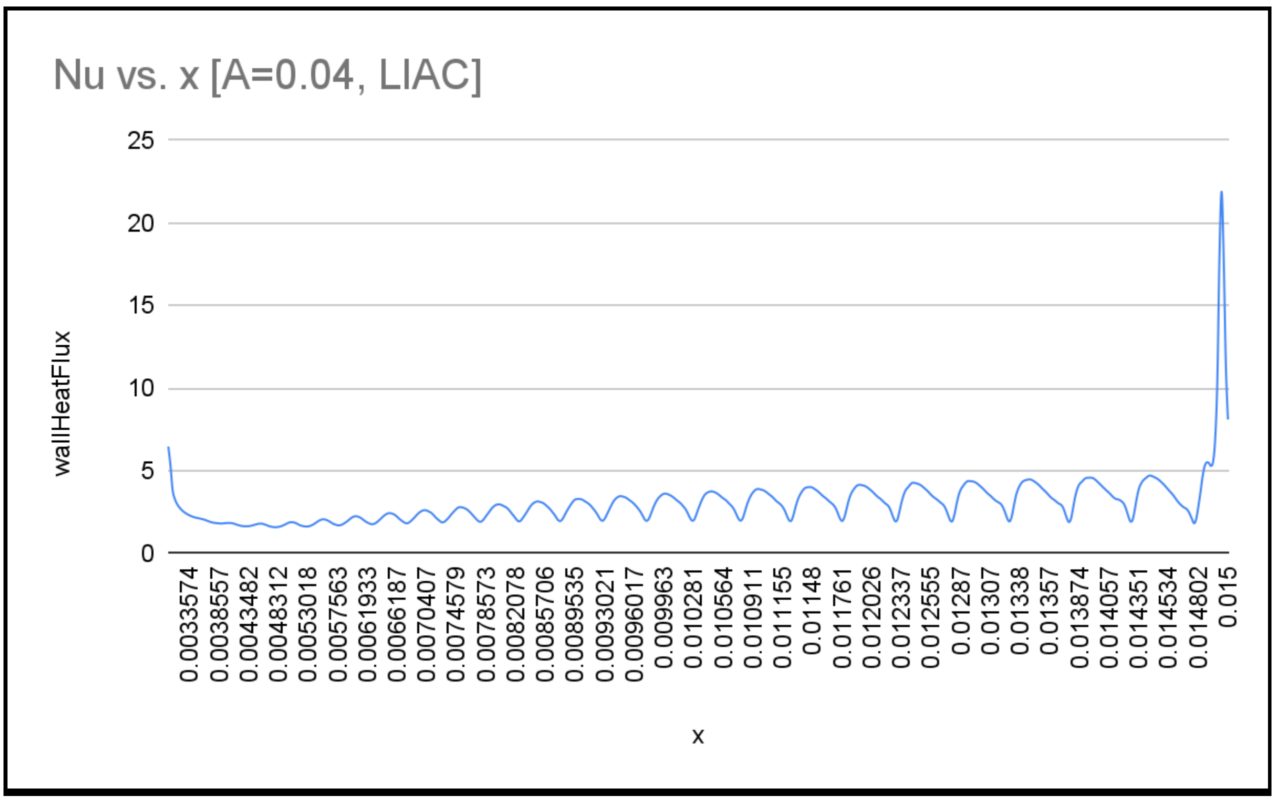
```
transportModel Newtonian;

// Laminar viscosity
nu              1e-06;

// Thermal expansion coefficient
beta            0.21e-03;

// Reference temperature
TRef            300;

// Laminar Prandtl number
Pr              Pr [0 0 0 0 0 0 0] 7.56;

// Turbulent Prandtl number
Prt             0.85;

    // Heat capacity
Cp0  Cp0 [0 2 -2 -1 0 0 0] 1864;

    // Fluid density
rho0 rho0 [1 -3 0 0 0 0 0] 998;

    // Thermal Diffusivity
alpha     alpha [0 2 -1 0 0 0 0] 0.1456e-6;
```

4 Results and Discussions

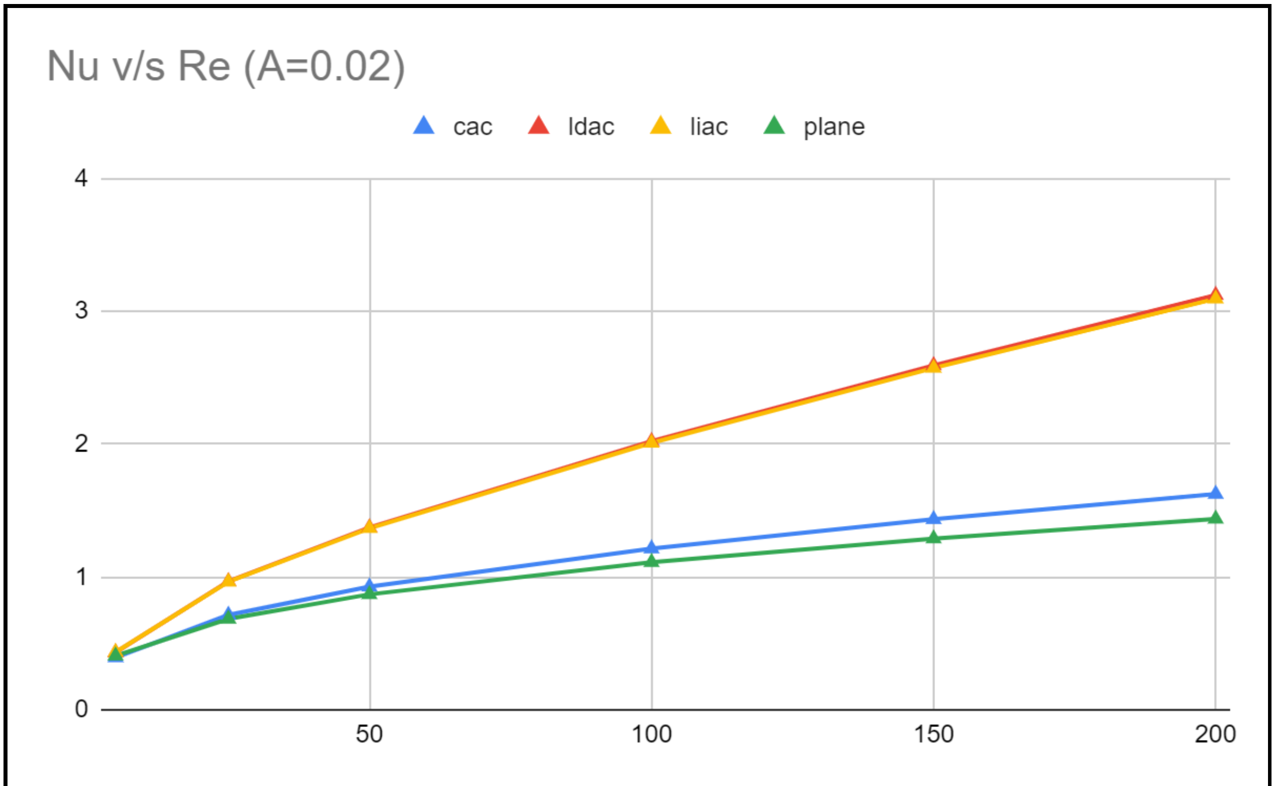


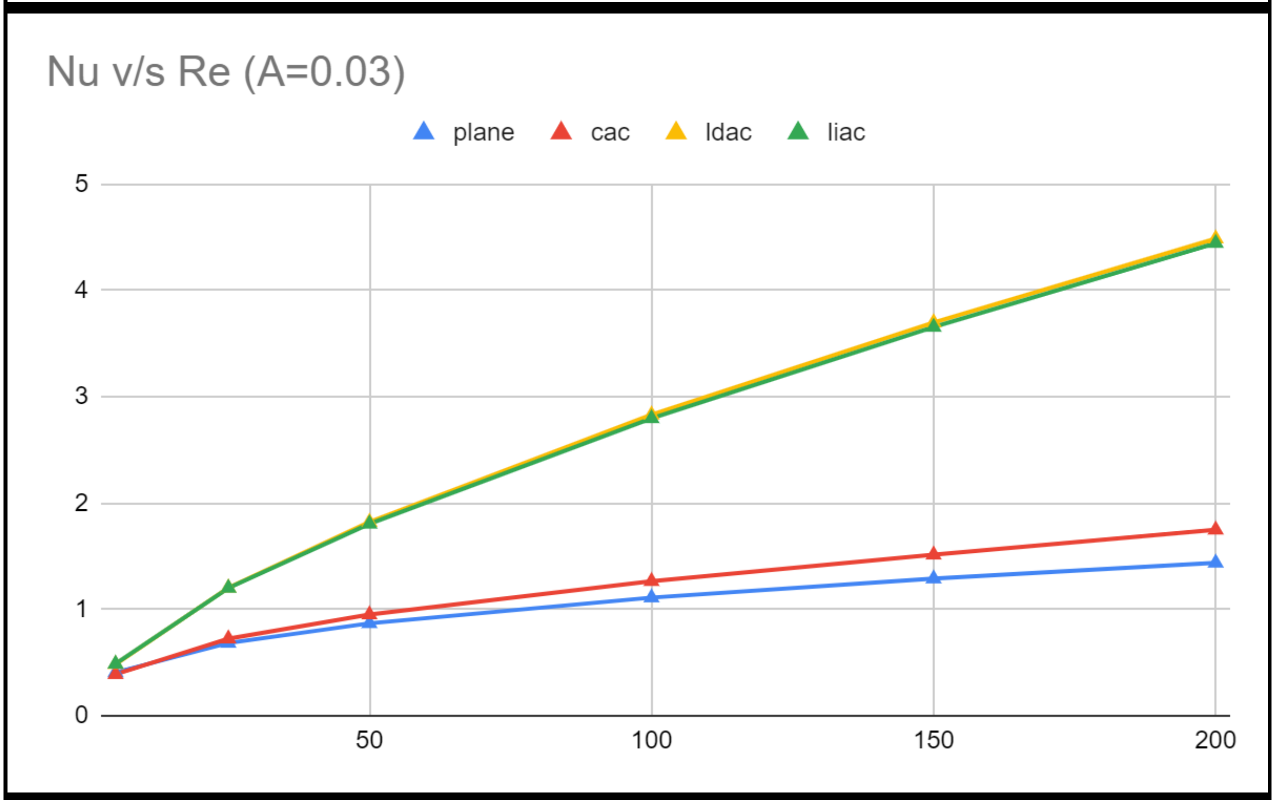


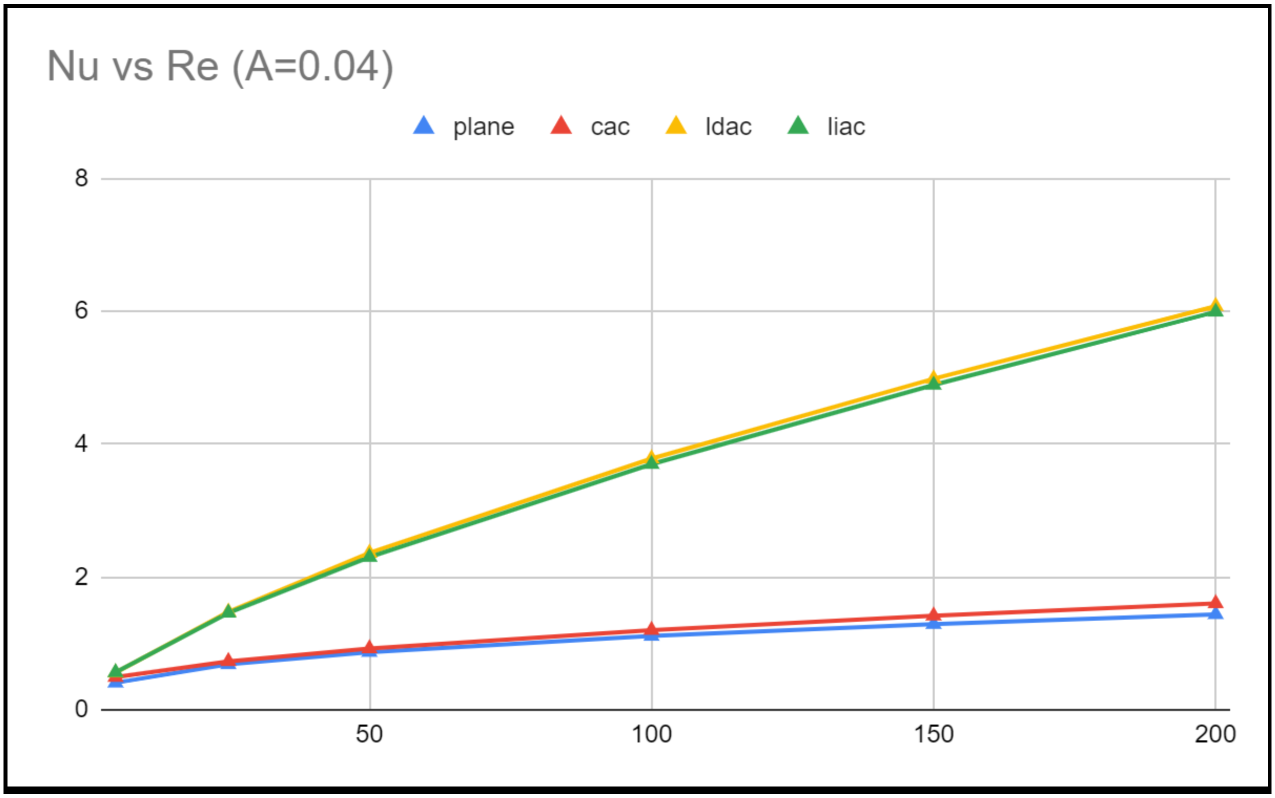




The above graph shows the local nusselt number v/s the length(only wavy portion) for LIAC, Re=200,A=0.04, and we can see a result similar to the reference clearly indicating more mixing towards the end.


Nu v/s Re (A=0.02)
cac
ldac
liac
plane
4
3
2
1
0
50
100
150
200


Nu v/s Re (A=0.03)
plane
cac
ldac
liac
5
4
3
2
1
0
50
100
150
200



Performance Characteristics

Several performance characteristics are required to judge the better use case of the wavy-structure. Since there is a trade-off between high htc and pressure drop. So we need to optimize the results by using appropriate parameters as given in reference.

PR-Pressure drop Ratio: Ratio of pressure drop between inlet and outlet od current case to plane wall case.

ER- Effectiveness Ratio: Ratio of average Nu of current to plane wall case.

PE(%) - Percentage effectiveness: Percentage increase in htc w.r.t plane wall case

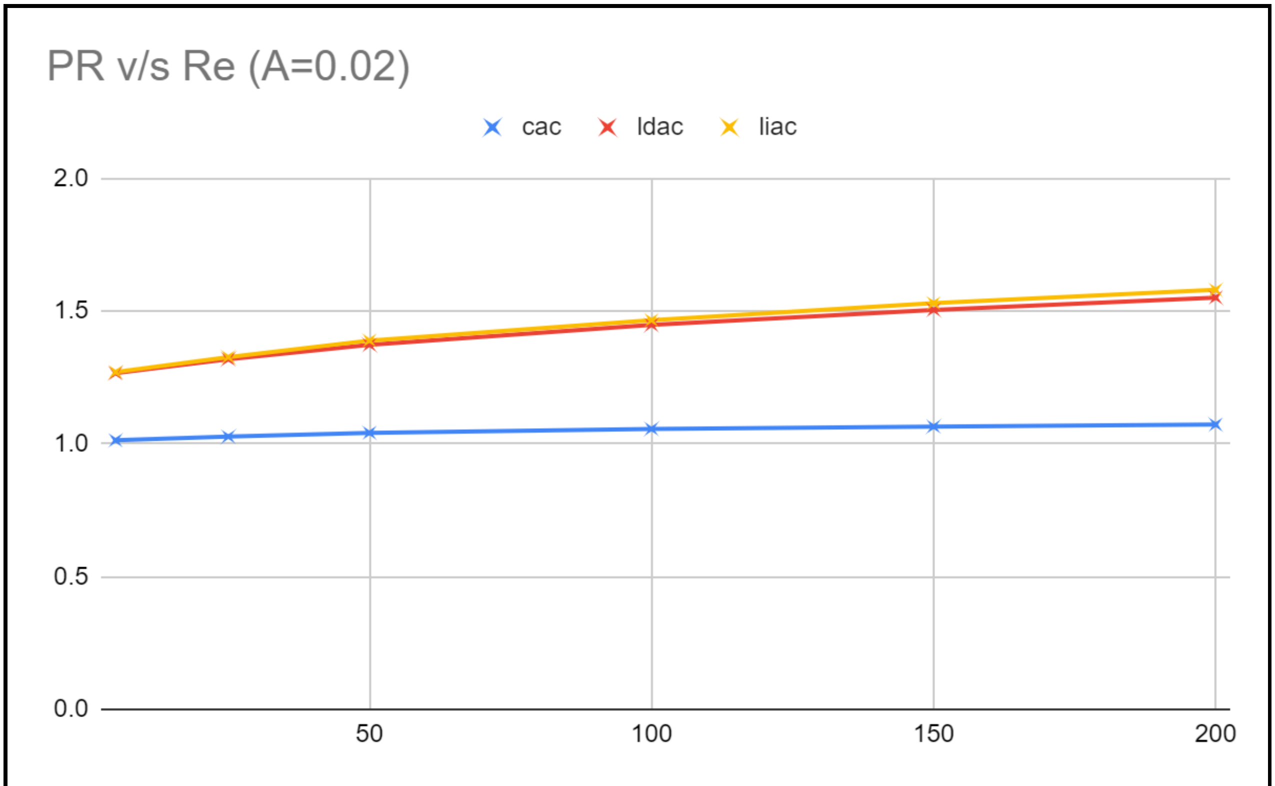
PF- Performance factor : Optimization parameter for htc to pressure drop trade-off considreations

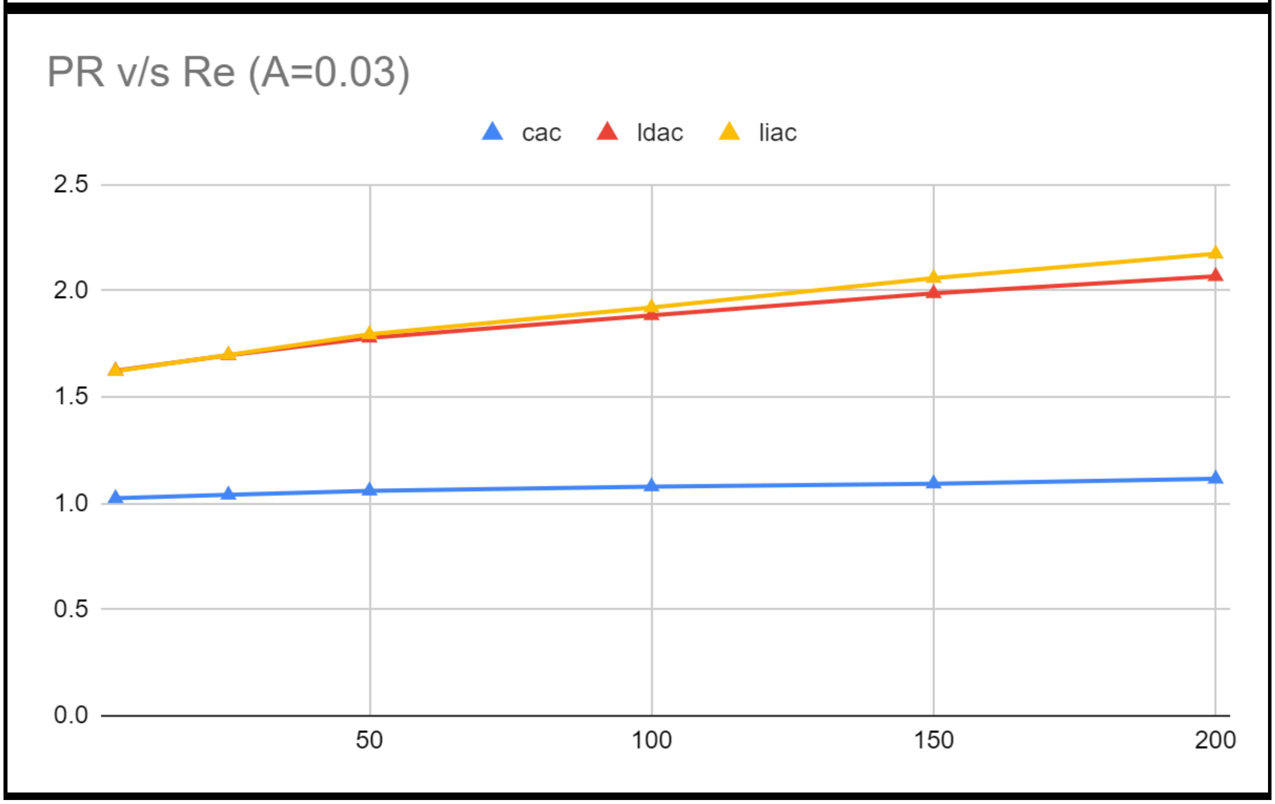
$$ER = Nu_{wavy} / Nu_{plane}$$

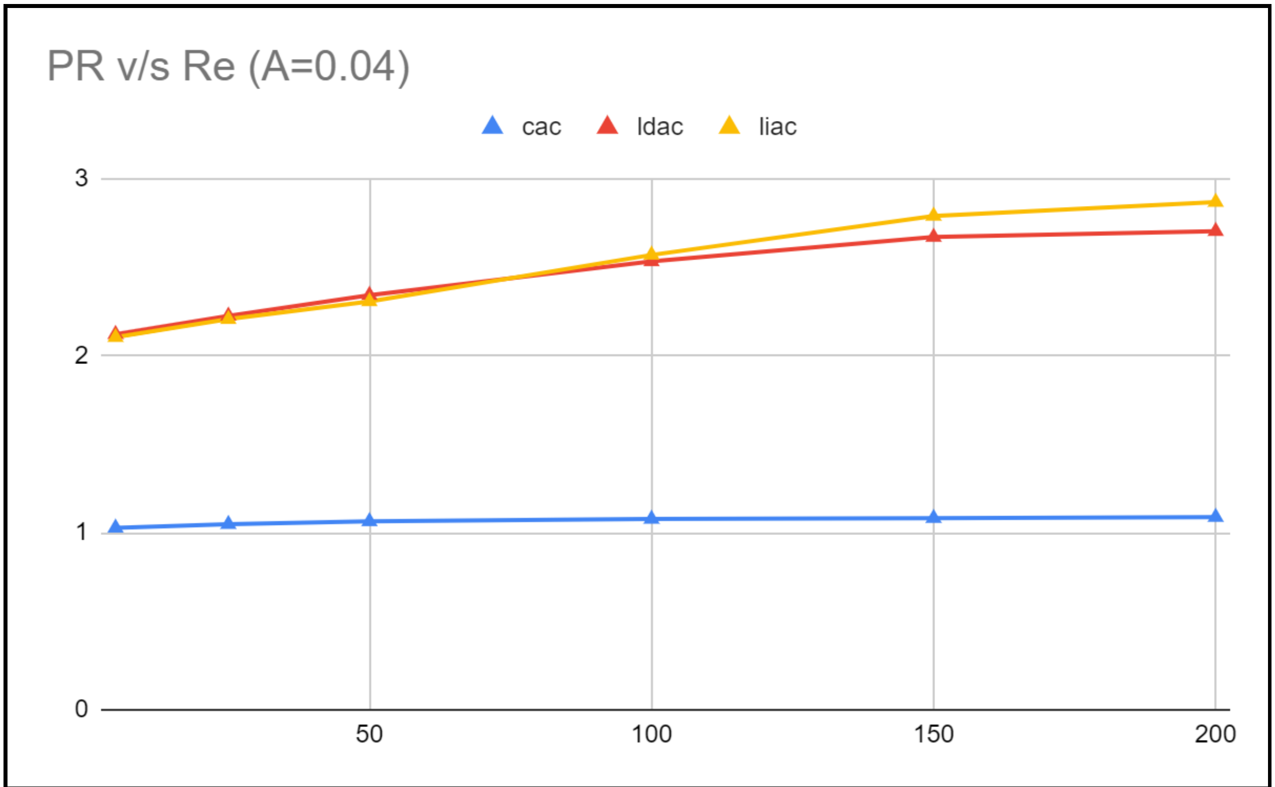
$$PR = \Delta p_{wavy} / \Delta p_{plane}$$

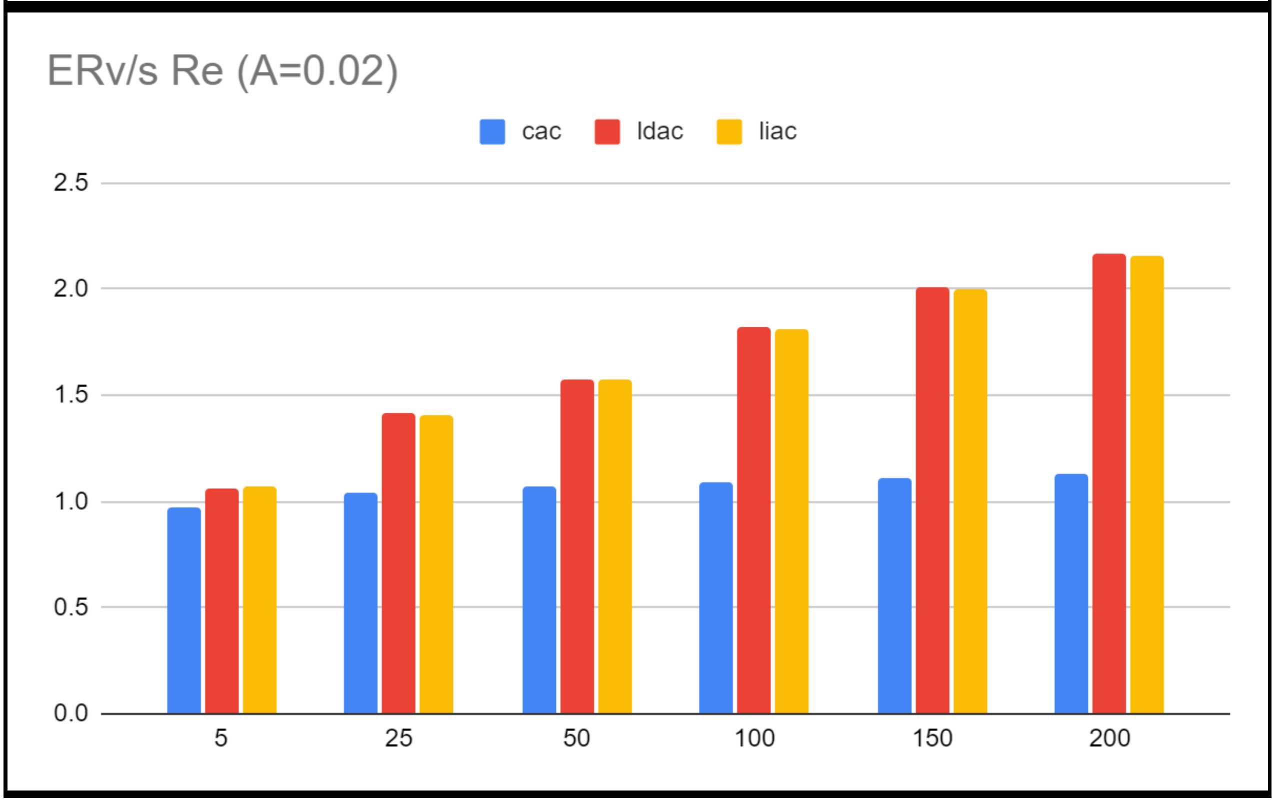
$$PE = (Nu_{wavy} - Nu_{plane})/Nu_{plane} \times 100$$

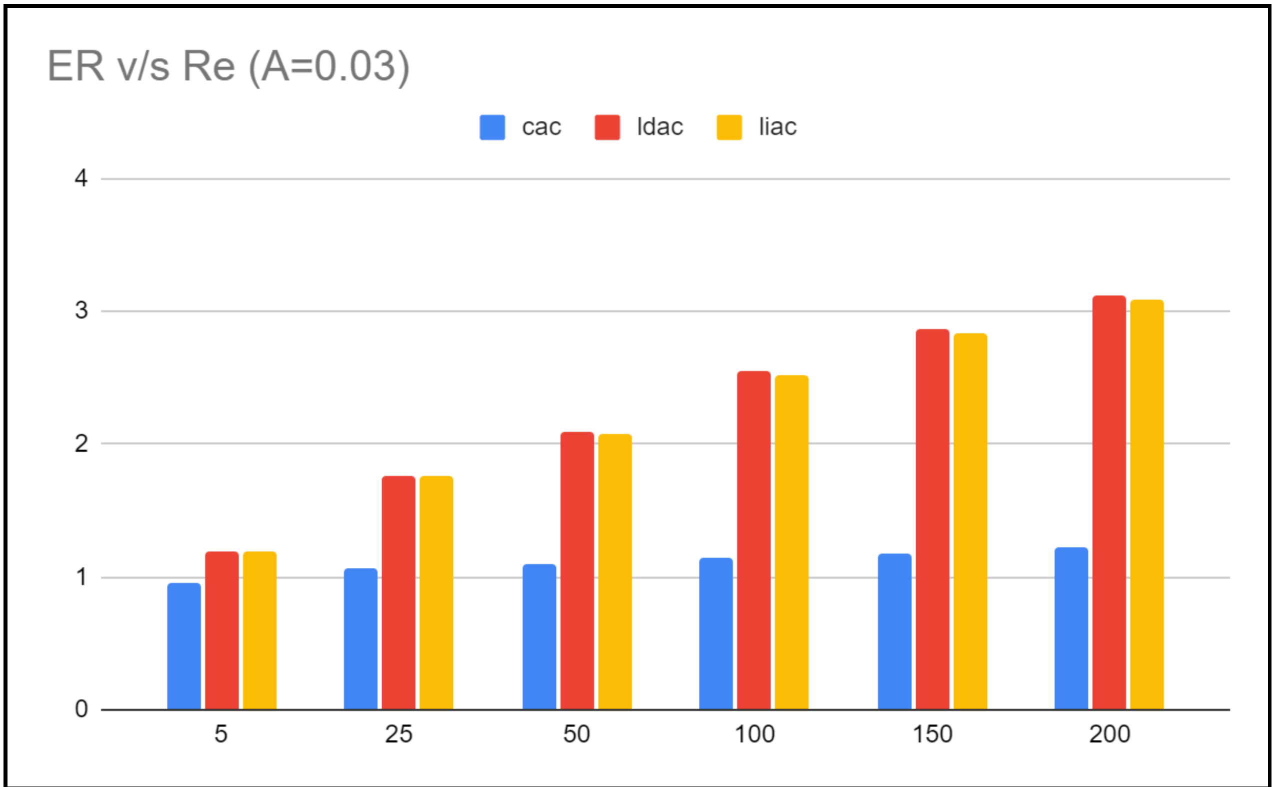
$$PF = \frac{ER}{(PR)^{1/3}}$$

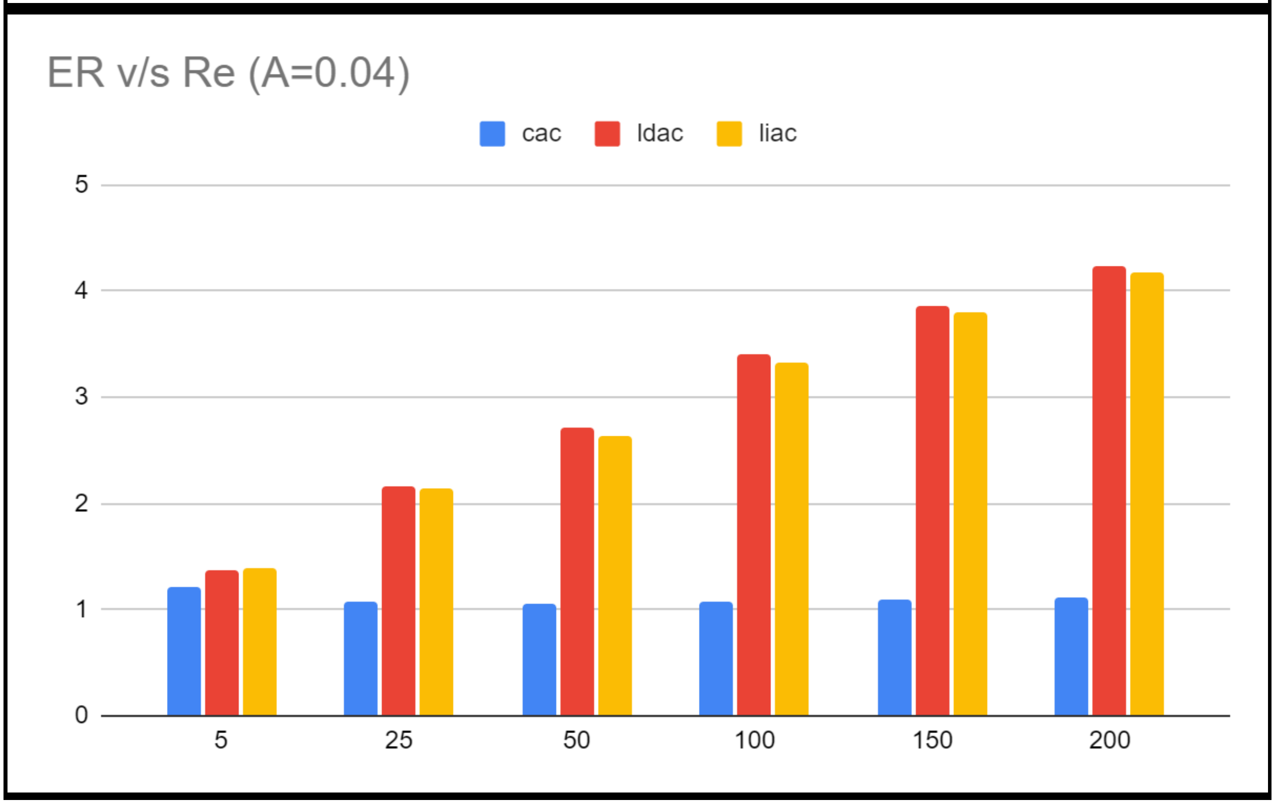

PR v/s Re (A=0.02)
cac
ldac
liac
2.0
1.5
1.0
0.5
0.0
50
100
150
200

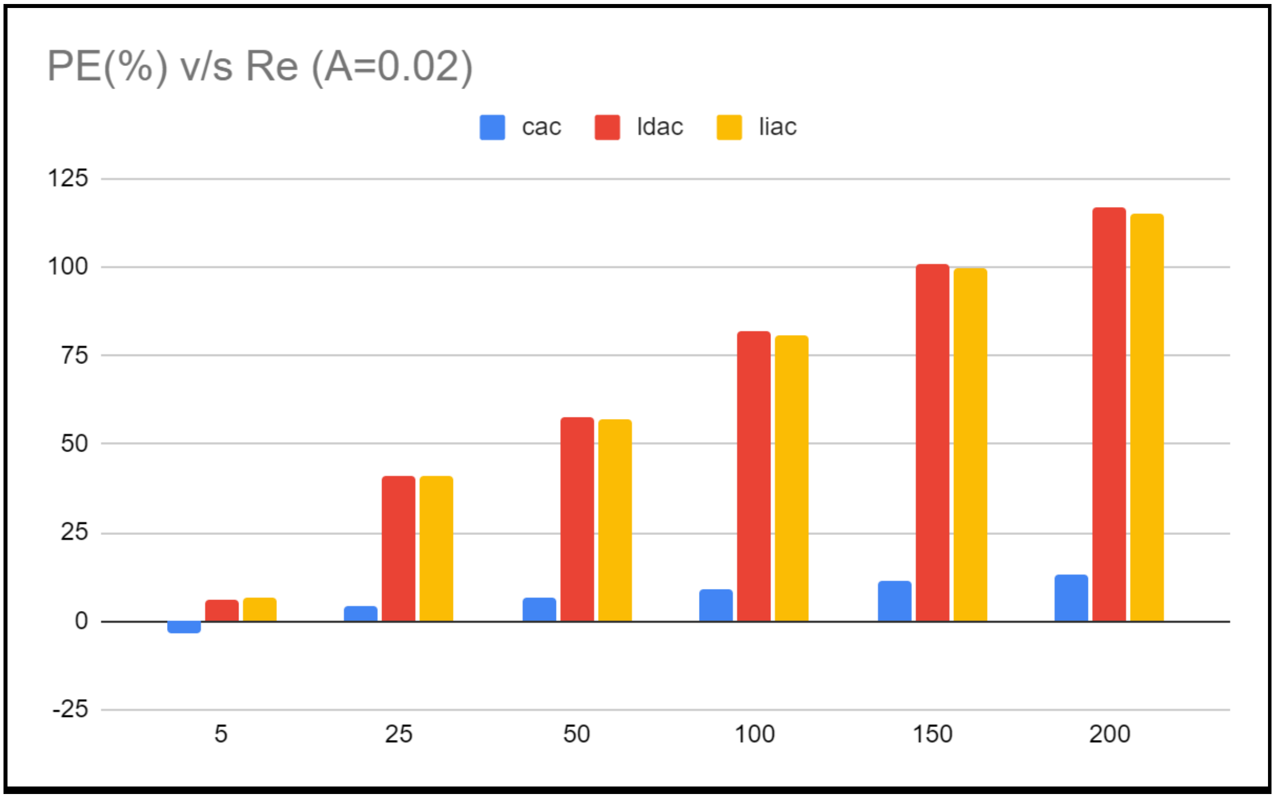

PR v/s Re (A=0.03)
cac
ldac
liac
2.5
2.0
1.5
1.0
0.5
0.0
50
100
150
200

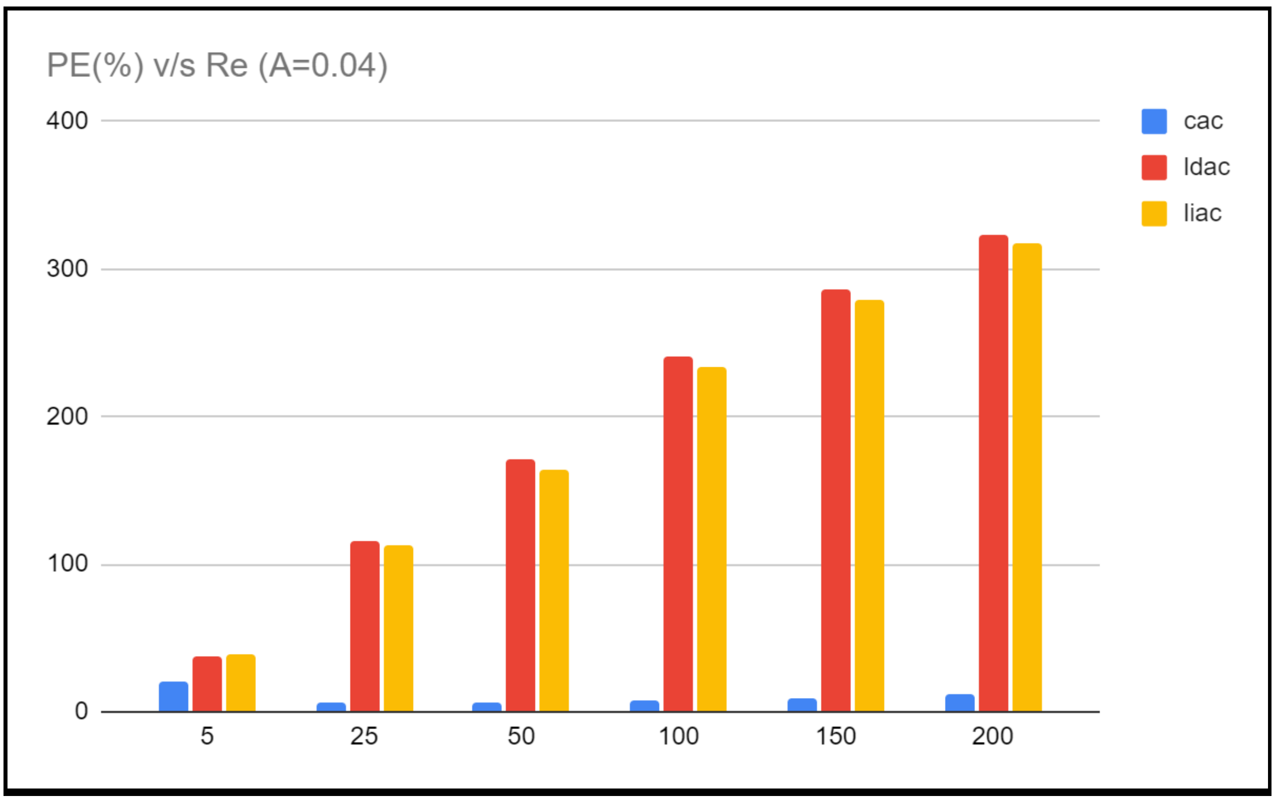

PR v/s Re (A=0.04)
cac
ldac
liac
3
2
1
0
50
100
150
200

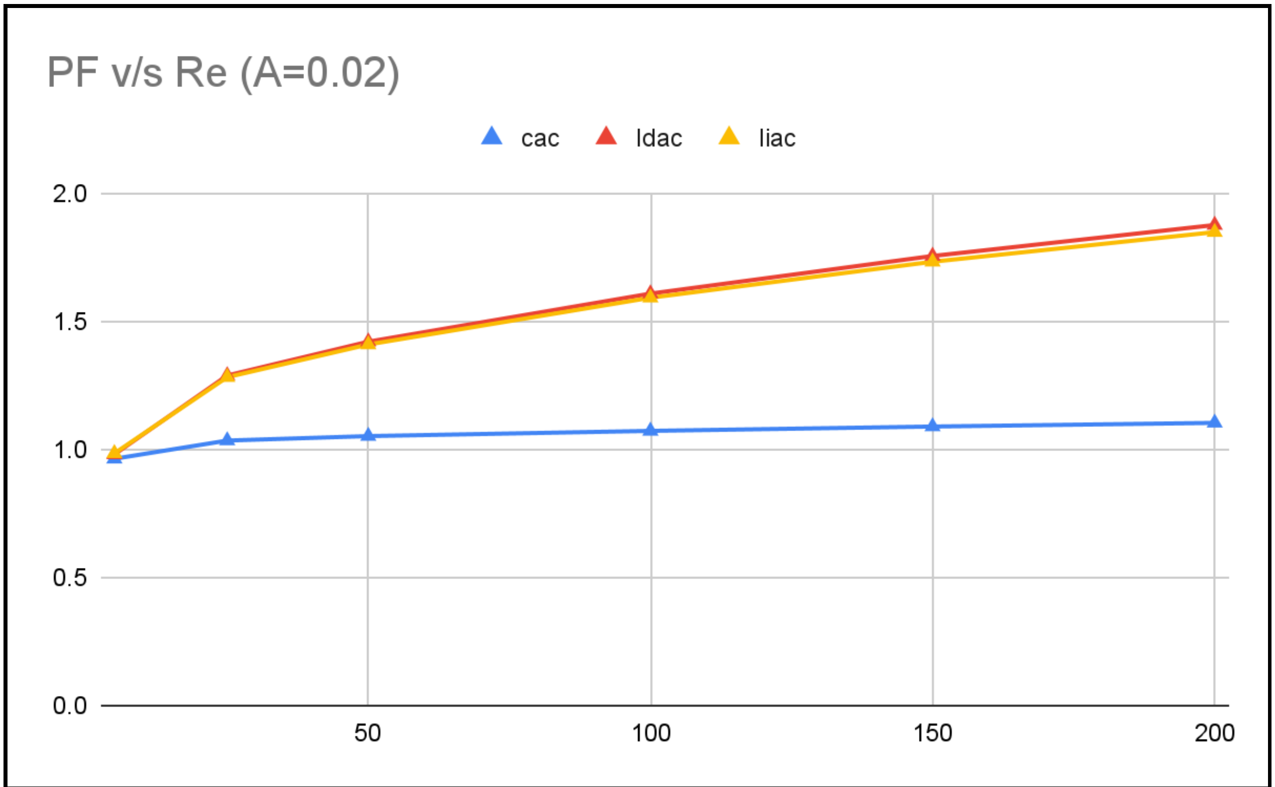

ERv/s Re (A=0.02)
cac
ldac
liac
2.5
2.0
1.5
1.0
0.5
0.0
5
25
50
100
150
200

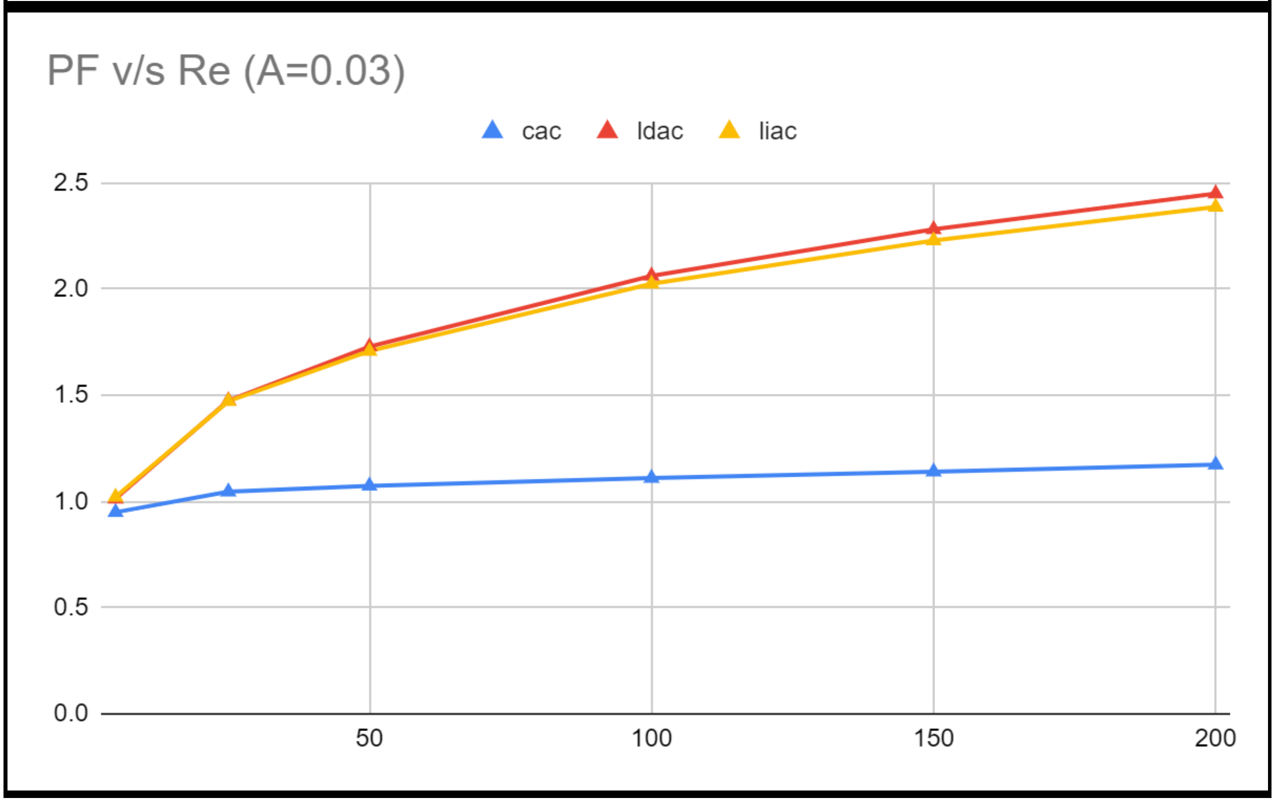

ER v/s Re (A=0.03)
cac
ldac
liac
0
1
2
3
4
5
25
50
100
150
200

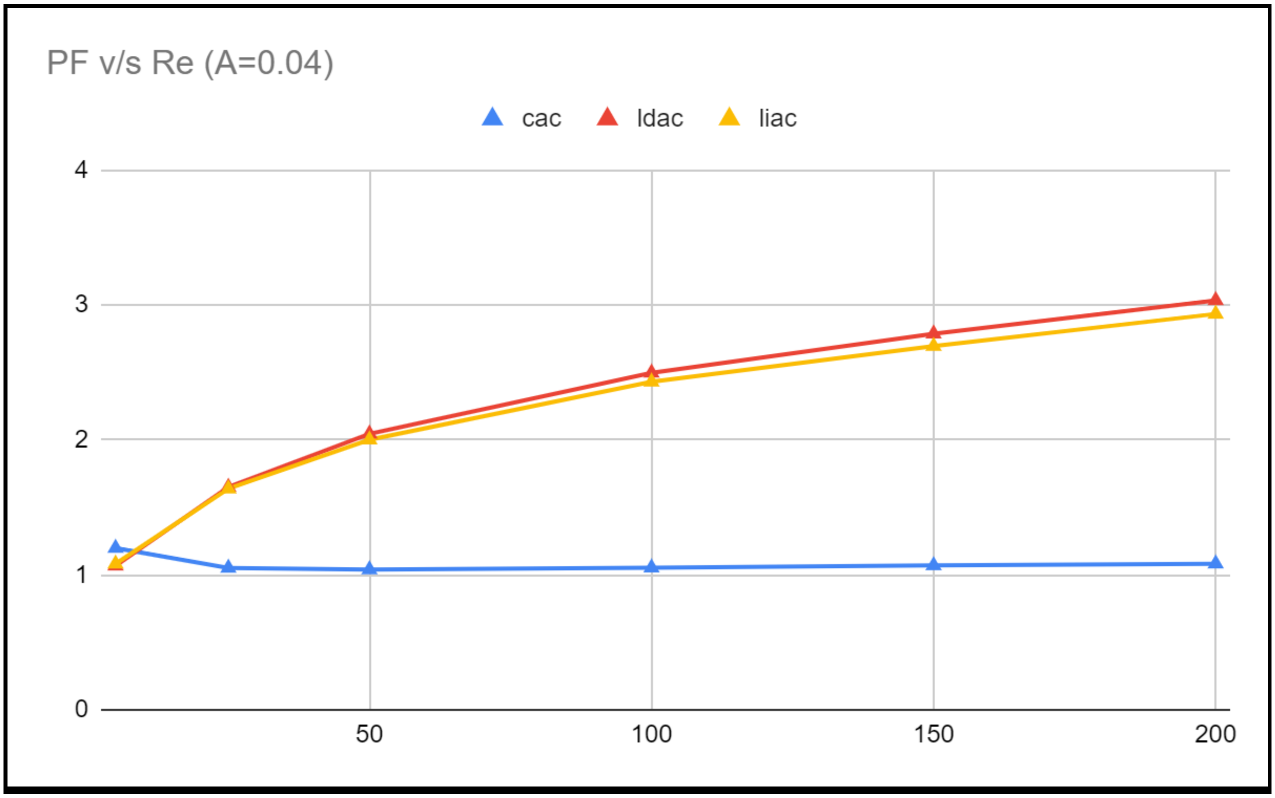

ER v/s Re (A=0.04)
cac
ldac
liac
0
1
2
3
4
5
5
25
50
100
150
200


PE(%) v/s Re (A=0.02)
cac
ldac
liac
125
100
75
50
25
0
-25
5
25
50
100
150
200


PE(%) v/s Re (A=0.04)
400
300
200
100
0
cac
ldac
liac
5
25
50
100
150
200


PF v/s Re (A=0.02)
cac
ldac
liac
2.0
1.5
1.0
0.5
0.0
50
100
150
200


PF v/s Re (A=0.03)
cac
ldac
liac
2.5
2.0
1.5
1.0
0.5
0.0
50
100
150
200



LDAC came out to be almost same as LIAC and both were well above CAC and Plane wall cases in terms of performance parameters.
Thus, we can conclude that CAC is better for lower reynolds numbers whereas LDAC is for higher ones, among which LIAC is better for higher amplitudes for getting lower pressure drops and also we get eddie in the outlet region leading to better htc. We didn’t get the last mentioned effect due to the flow being laminar in our simulatio

References

[1] Mehta, Sumit & Pati, Sukumar & Baranyi, Laszlo. (2022). *Effect of amplitude of walls on thermal and hydrodynamic characteristics of laminar flow through an asymmetric wavy channel*. Case Studies in Thermal Engineering. 31. 101796. 10.1016/j.csite.2022.101796. URL: https://www.sciencedirect.com/science/article/pii/S2214157X22000429?via%3Dihub

[2] Mehta, S.K., Pati, S. Numerical study of thermo-hydraulic characteristics for forced convective flow through wavy channel at different Prandtl numbers | SpringerLink *J Therm Anal Calorim* 141, 2429–2451 (2020)

[3] Gerrit Nils Werner Master's thesis, June 2016, *CFD analysis to determine the local Nusselt number over a heated micro wire in multicomponent flow*, Thermal Energy and Process Engineering

[4] C.-C. Wang, C.-K. Chen, *Forced convection in a wavy-wall channel* * Department of Mechanical Engineering, National Cheng-Kung University, Tainan, Taiwan, International Journal of Heat and Mass Transfer 45 (2002) 2587–2595

[5] Johan Magnusson et. al., *conjugateHeatFoam with explanational tutorial together with a buoyancy driven flow tutorial and a convective conductive tutorial,* Magnusson's Technology Center Chalmers University of Technology CFD with OpenSource software, Project

[6] How to check Heat Balance in heat transfer like mass balance for flow -- CFD Online Discussion Forums

[7] https://github.com/wyldckat/wallHeatFluxIncompressible
[8] www.openfoam.com
[9] www.openfoamwifi.net

DISCLAIMER: This project reproduces the results from an existing work, which has been acknowledged in the report. Any query related to the original work should not be directed to the contributor of this project.