Natural Convection in a Vertical Channel with fins using conjugate heat transfer solver in OpenFOAM

Chaitanya Kishore

MS Scholar, IIT Madras

Synopsis

This research migration project aims to simulate a natural convection flow in a vertical channel . A chtMultiRegionSimpleFoam solver is used for numerical simulation of natural convection in a vertical channel with fins, a steady-state solver in OpenFOAM. Square fin, triangle fin, and semi-circle fins are roughness elements in the flow domain. K-epsilon turbulence model is chosen as the turbulence model. For radiation modeling, the fvDOM radiation model is used. This research migration project uses OpenFOAMv2012 software and validated the results from a published paper.
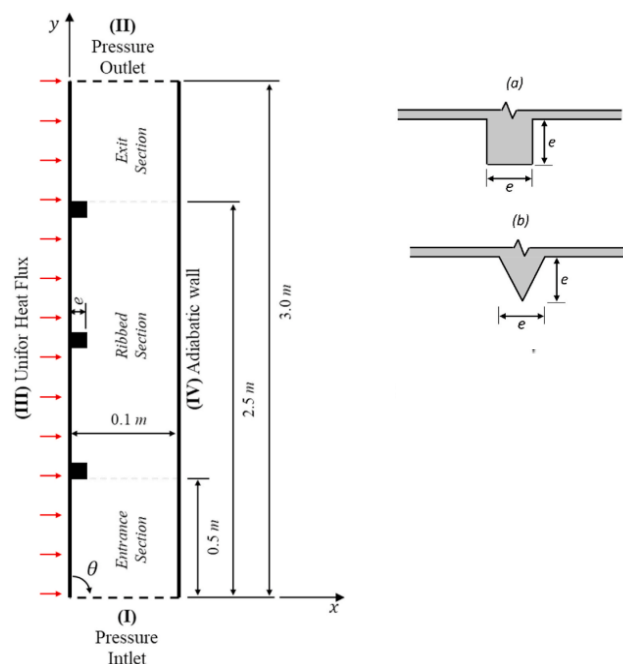

Figure 1: Vertical Channel with different ribs/fins (a) square fins (b) triangle fins

1   Introduction

Building-integrated Photovoltaic and Thermal (BIPV/T) systems enhance the economic feasibility of solar panels. They are installed on the building envelope with an air gap for thermal regulation. This air gap has to be cooled using active cooling or passive cooling. Passive cooling is preferred since it is cost-effective and simple also. For passive cooling, fins/ribs are used. Nghana et al. [1] studied the effect of ribs on the natural convection in a BIPV air channel. This study considers a vertical channel with three different fins/ribs for simulation. The variation of the Nusselt number with the modified Rayleigh number is plotted and compared with the results from the paper.

This project aims to demonstrate the use of OpenFOAMv2102 software to study the natural convection in vertical channels with conjugate heat transfer and reproduces the results in the paper by Nghana et al. [1]. This simulation uses the finite volume method for incompressible, steady-state, and turbulent flow.

A straight vertical channel with a height of 2 m and channel depth (hydraulic diameter, D) of 0.1 m is considered for simulation. A 0.5 m entrance length and 0.5 m exit length are provided to allow the flow to develop properly. The thickness of the solid plate is 0.635 mm. On this solid plate, fins/ribs are attached from 0.5 m to 2.5 m from the entrance point. The relative roughness height e/D is 0.3, and the relative roughness pitch p/e is 7.5. These details are enough to construct the geometry for simulation. The illustration of the vertical channel with fins or ribs is shown in fig-1. Square fin and triangle fin are validated from the paper.

2   Governing Equations and Models

For every region with fluid and solid, various equations are being solved for the fluid and the solid region, which are listed below. Thermal boundary conditions at the interfaces couple these different regions.

Partial differential equation for the solid Region

In the solid domain, the energy equation needs to be solved to determine the temperature field T. The steady-state energy equation is shown in the equation-1.

$$\nabla.(k_s \nabla T) = Q_s \tag{1}$$

The specific heat capacity, Cp, the density, ρ, and thermal conductivity, ks, are constant properties for the domain. The term, Qs, is the energy source within the domain.

Partial differential equation for the fluid region

Equation of Mass conservation

The steady-state continuity equation for variable density is defined in the equation-2, which was implemented in OpenFOAM and can be found in the rhoEqn.H file.

$$\nabla.(\rho \vec{U}) = 0 \tag{2}$$

The working fluid in this study is air which is considered a perfect gas. So, density is calculated using the perfect gas law shown in the equation-3.

$$\rho = \frac{pM}{RT} \tag{3}$$

Equation of Momentum conservation

The steady-state momentum equation is shown in the equation-4. The LHS term in equation-4 represents the variation of momentum inside the domain. The term, $\nabla.\tau$, represents the work done by viscous forces and the expanded equation is shown in the equation-5. The term, $\rho\vec{g}$ represents the work done by body forces, and the term $-\nabla p$, represents the work done by pressure forces.

$$\nabla.\left(\rho\vec{U}\vec{U}\right) = \nabla.\tau + \rho\vec{g} - \nabla p \tag{4}$$

$$\tau = \mu\left[-\frac{2}{3}\nabla.\vec{U}I + \nabla\vec{U} - \left(\nabla.\vec{U}\right)^T\right] \tag{5}$$

In this case, the fluid's dynamic viscosity obeys Sutherland's law, as shown in the equation-6.

$$\mu = \frac{A_s T^{3/2}}{T + T_s} \tag{6}$$

Equation of Energy conservation

The steady-state energy equation to be solved is given in the equation-7.

$$\nabla.\left(\rho\vec{U}H\right) = \nabla.\left(\tau\vec{U}\right) + \rho\vec{g}\vec{U} - \nabla.\vec{q} + Q_s \tag{7}$$

Total enthalpy, H, is a sum of enthalpy and the kinetic energy shown in the equation-8.

$$H = C_p T + \frac{\left|\vec{U}\right|^2}{2} \tag{8}$$

The thermal conductivity, k, in the energy balance is calculated using the modified Euken thermal conductivity model shown in the equation-9. This model considers the variation of thermal conductivity with temperature.

$$k = \mu\left(C_p - \frac{R}{M}\right)\left(1.32 + \frac{1.77\frac{R}{M}}{C_p - \frac{R}{M}}\right) \tag{9}$$

Radiation: FvDOM Model

In this case study, it is necessary to include a radiation model to account for the radiative heat transfer (RTE) in addition to the heat transfer by conduction and convection. Equation-10 is a radiative heat transfer equation that describes the energy balance of a radiation beam as it travels through an absorbing, emitting, and scattering media.

$$\nabla \cdot \left((\vec{x}, \vec{s}) \vec{s} \right) = -aI(\vec{x}, \vec{s}) - \sigma_s I(\vec{x}, \vec{s}) + an^2 \frac{\sigma T^4}{\pi} + \frac{\sigma_s}{4\pi} \int_0^{4\pi} I(\vec{x}, \vec{s}) \Phi(\vec{s}.\vec{s}) d\Omega' \qquad (10)$$

'I' is Intensity, 'a' is absorption coefficient, 'n' is the refractive index, 'σ_s' is the scattering coefficient, 'σ' is Stefan-Boltzmann constant, 'Φ' is the scattering phase function, and 'Ω' is the directional vector of radiative intensity. The RHS first term represents the attenuation by absorption, the second term the attenuation by scattering, the third term the augmentation by emission, and the fourth term the augmentation by scattering. The working fluid air was considered a non-absorbing, non-scattering, and non-emitting media, so RHS terms of RTE became zero.

The finite volume discrete ordinates (FvDOM) model solves the RTE for a finite number of discrete solid angles. This model solves as many equations as the number of considered discrete solid angles. This model's boundary condition for diffuse grey surfaces is shown in the equation-11, which requires emissivity, ε, for every given surface. The transport equation for the incident radiation is solved, and the radiative flux q_r is included in the equation-7 as a source term as shown in the equation-12.

$$(q_r)_{bf} = (1 - \varepsilon_{bf}) \int_{\vec{s}\vec{n}>0} I(\vec{x}, \vec{s}) \vec{s}\vec{n} d\Omega + n^2 \varepsilon \sigma_s T_{bf}^4 \qquad (11)$$

$$Q_s = a \left(4\sigma T^4 - \int_{4\pi} I(\vec{x}, \vec{s}) \vec{s} d\Omega \right) \qquad (12)$$

Boundary Condition at Fluid and Solid Interface

The interface between the solid and fluid domain must be at the same temperature as shown in the equation-13. The second boundary condition comes from the energy balance at the interface surface as shown in the equation-14.

$$(T_{solid}) = (T_{fluid}) \qquad (13)$$

$$(k_{solid} \nabla T_{solid}) = (k_{fluid} \nabla T_{fluid}) + q_r \qquad (14)$$

Thermo-Physical Properties for air

For viscosity modelling, Sutherland equation is used as shown in equation-6. The Sutherland coefficients for air are As = 1.4584e-6 and Ts = 110.4.

For specific heat of air, JANAF model is used.

3 Simulation Procedure

3.1 Geometry and Mesh

The geometries are created in Salome software and blocked to get the hexahedral-dominated mesh. For some geometries, tetrahedral meshing is allowed in the solid domain since it is challenging to block those geometrical entities. The blocked geometries are shown in fig-2.
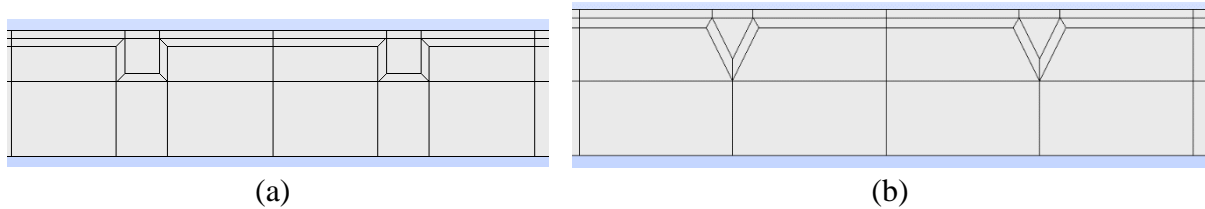

(a)

(b)

Figure 2: Blocked Geometries of (a) square fin (b) triangle fin

The meshing of all geometries is performed in Salome software only. The mesh at the single fin for all three geometries is shown in fig-3.

(a)

(b)
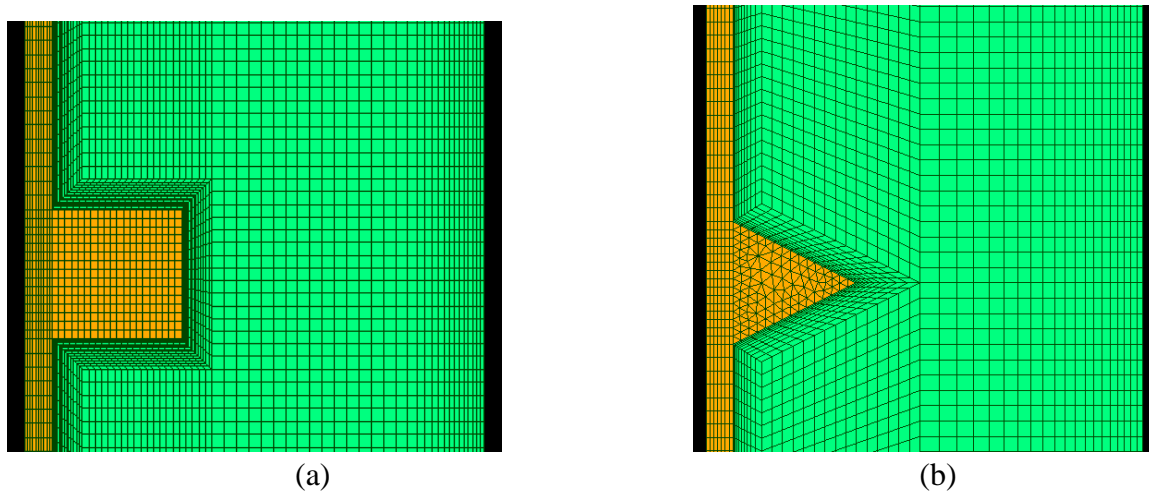
Figure 3: 2D meshes of (a) square fin (b) triangle fin

3.2   Initial and Boundary Conditions

The details of the boundary conditions are listed in table-1.

Table 1: Boundary Conditions

	U	T	P_rgh	K	Epsilon
SolidWall1	noSlip	Uniform heatflux	fixedFluxPressure	epsilonWall Function	kqRWallFunctio n
SolidWall2	noSlip	zeroGradient	fixedFluxPressure	epsilonWall Function	kqRWallFunctin
inlet	Pressure InletVelocity	totalTemperature uniform value T0 = 296 K	prghTotalPressure value p0 = 101325 Pa	zeroGradient	zeroGradient
outlet	inletOutlet	inletOutlet	zeroGradient	zeroGradient	zeroGradient
solidBottom	-	zeroGradient	-	-	-
solidTop	-	zeroGradient	-	-	-
frontAndBack	empty	empty	empty	empty	Empty
fluid_to_solid/ solid_to_fluid	-	Compressible: turbulent TemperatureRadC oupledMixed	-		

3.3   Solver

The chtMultiRegionSimpleFoam is a steady-state fluid flow and solid heat conduction solver with conjugate heat transfer between domains. The effect of buoyancy, turbulence, variable thermophysical properties, and radiation can also be included in this solver based on user requirements. This solver's velocity-Pressure coupling is a SIMPLE algorithm used for steady-state simulations.

3.4 Postprocessing

ParaView software is used for postprocessing the results from OpenFOAM. Wall shear stress and wall heat flux are calculated in OpenFOAM using post-process utility. For simplicity, python code is written in paraview to automate the results for each case.

4 Results and Discussions

In this project, the heat flux is varied from 1000 W/m² to 100 W/m² to vary the modified Rayleigh number. The modified Rayleigh number is calculated using the equation-

$$Ra_s^* = \frac{g\beta q'' S^4}{k\alpha v} \tag{15}$$

In the above equation, g is the acceleration due to gravity, β is the thermal expansion coefficient, S is the channel depth or hydraulic diameter, k is the fluid thermal conductivity, and v is the kinematic viscosity. β is approximated as 1/(film temperature Tf). g is 9.81 m/s². The fluid properties in this equation are evaluated at film temperature. Film temperature is calculated as average of reference temperature and average surface temperature.

Nusselt number in this study is calculated using the formula shown in equation-.

$$Nu = D \int_{0.5}^{2.5} \frac{q''(y)}{k(y) \times (T_w(y) - T_{ref})} \, dy \tag{16}$$

The residuals for all the simulations are converged up to less than 10^{-5}.

4.1 StreamLines

The streamlines from paper and from OpenFOAM result for two different fins are shown in fig-4, fig-5.

(a)

(b)
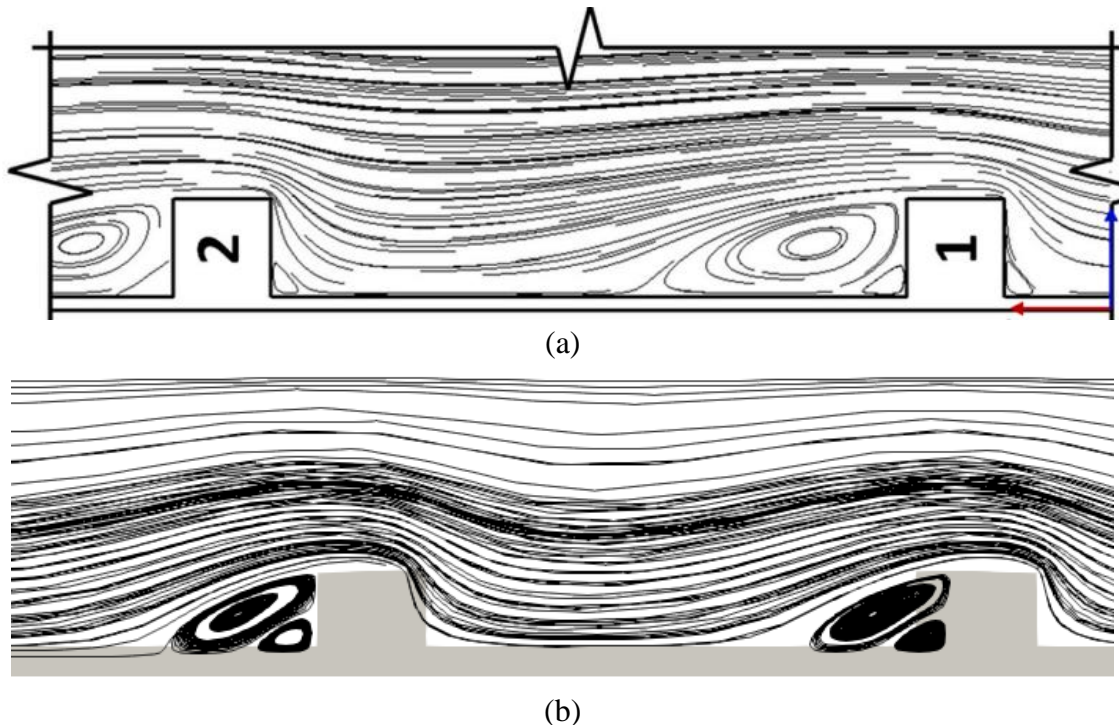
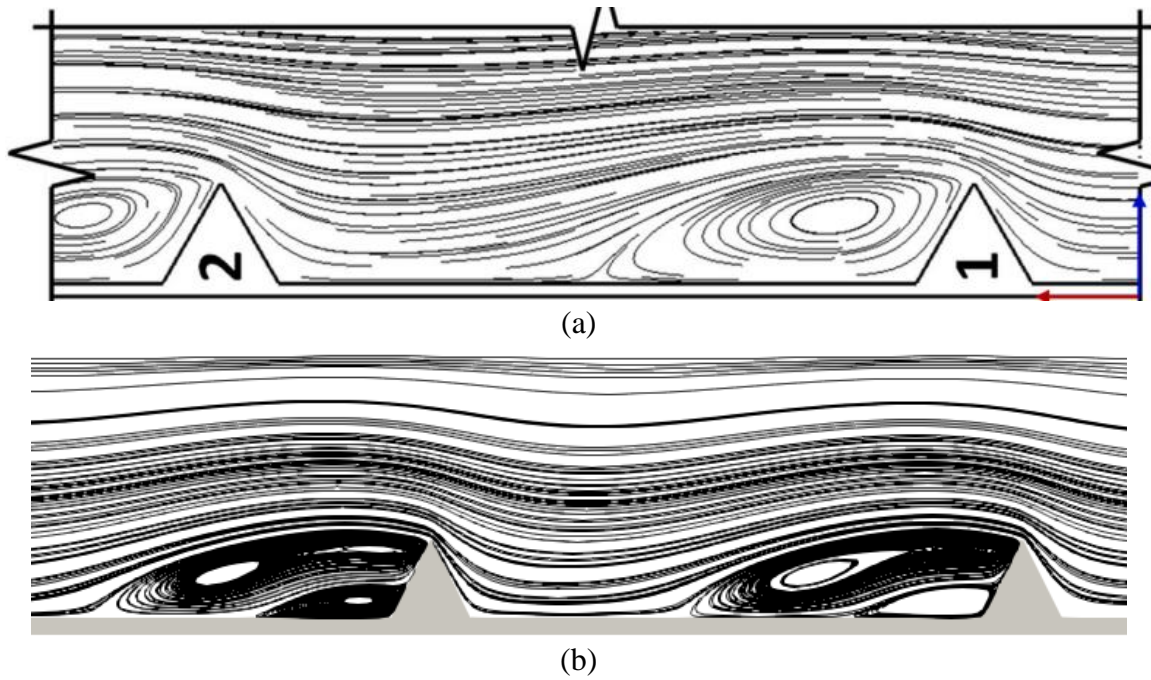Figure 4: Stream for square fins (a) Paper (b) OpenFOAM

(a)

(b)

Figure 5: Streamlines for triangle fins (a) Paper (b) OpenFOAM

4.2 Temperature Contours

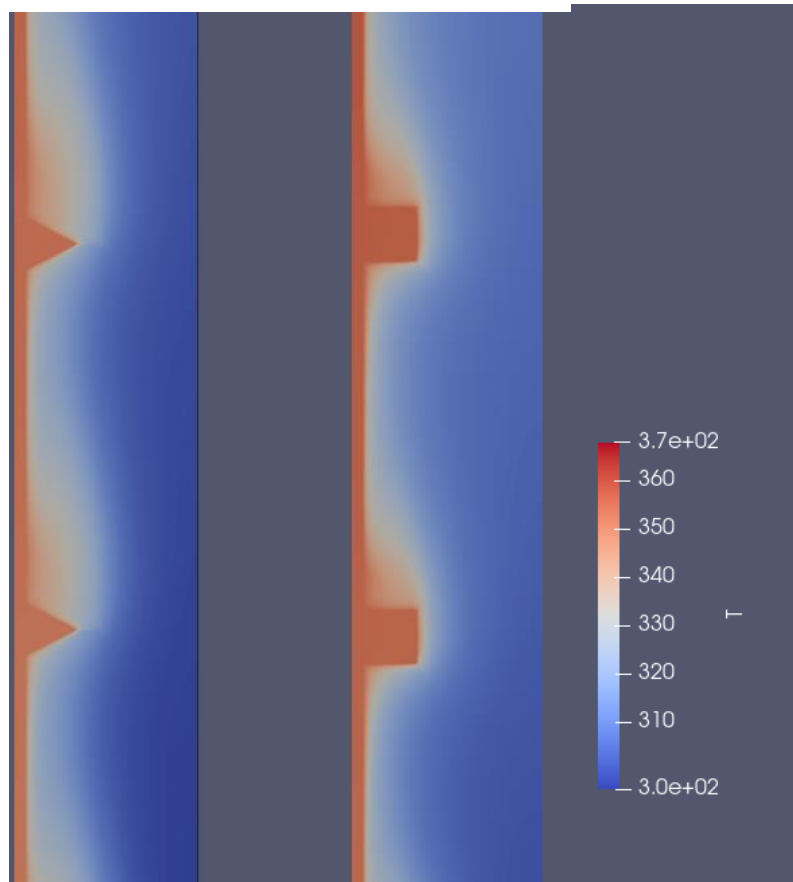The temperature contours for square fin and triangle fin are shown in fig-6.

Figure 6: Temperature contours for triangle fin and square fin

4.3 Nusselt number vs Ra_s^*

The Nusselt number vs Ra_s^* graph from the paper is compared with the results from OpenFOAM simulations for two different fins. The are shown in fig-7 and 8 respectively. From figure, we can observe that the results from OpenFOAM are very near to results of that of the paper.
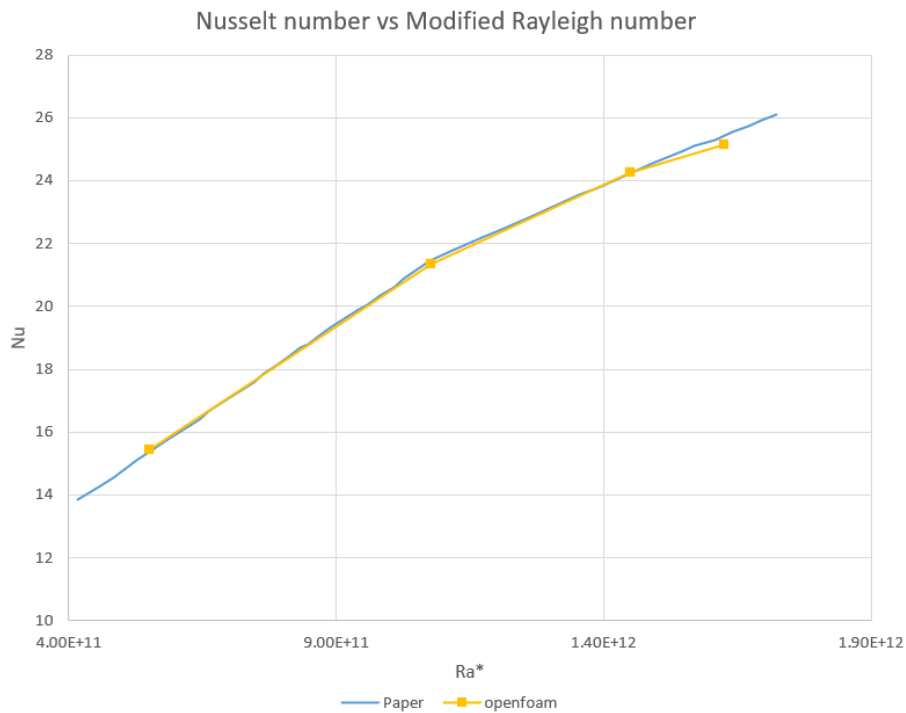
Figure 7: Nu v Ra* for square fins
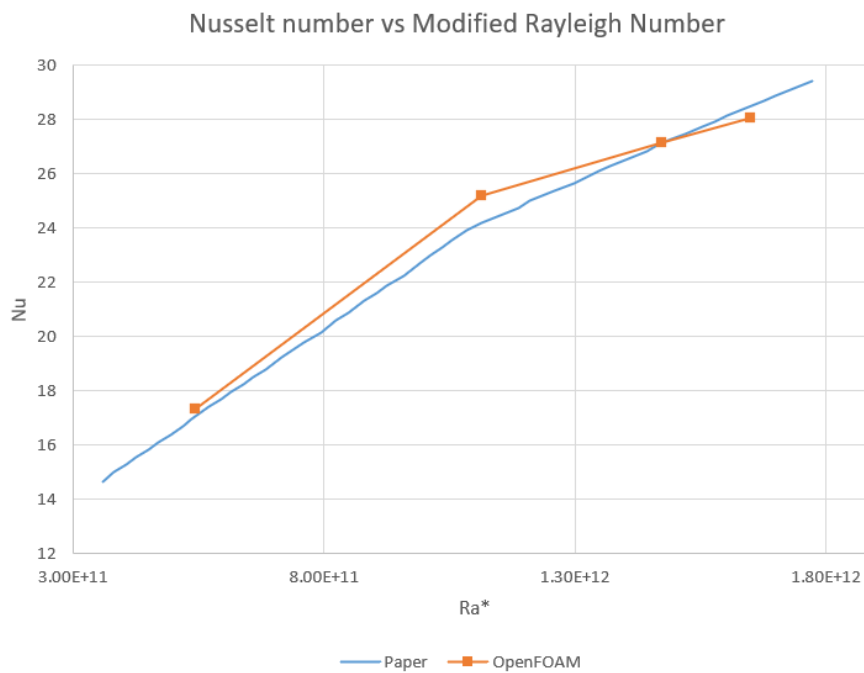
Figure 8: Nu vs Ra* for triangle fins

4.4 Wall Shear Stress

The variation of wall shear stress between adjacent fins are plotted vs non-dimensional length y/H for square fin in fig-9 and triangle fin in fig-10. The wall shear stress data from the paper is also compared with obtained results in the same figures.
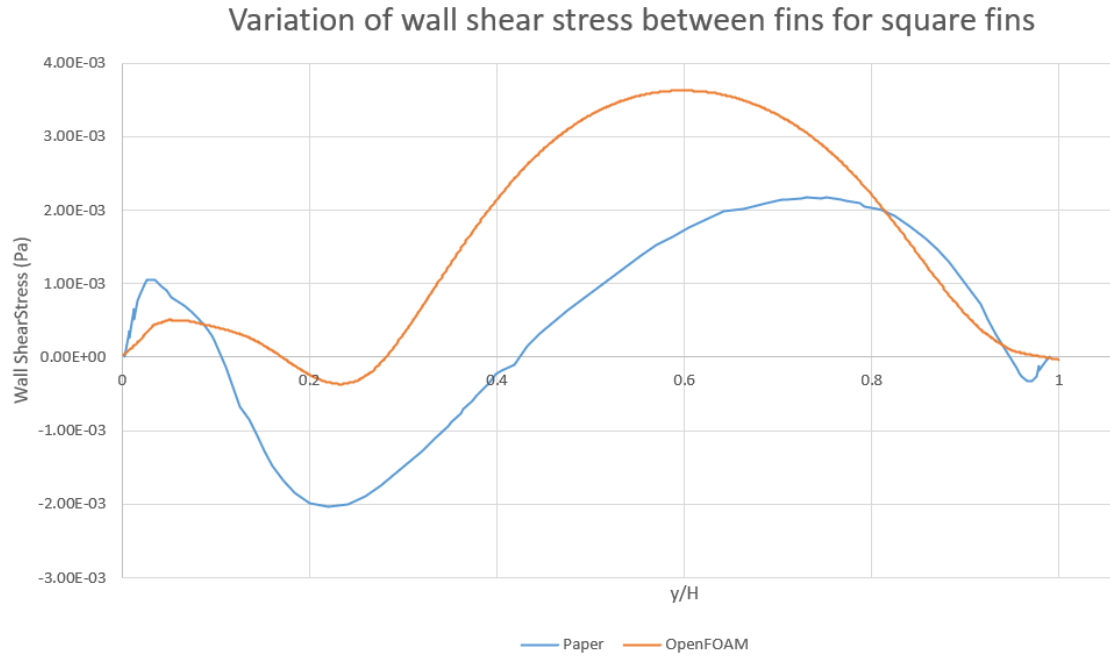

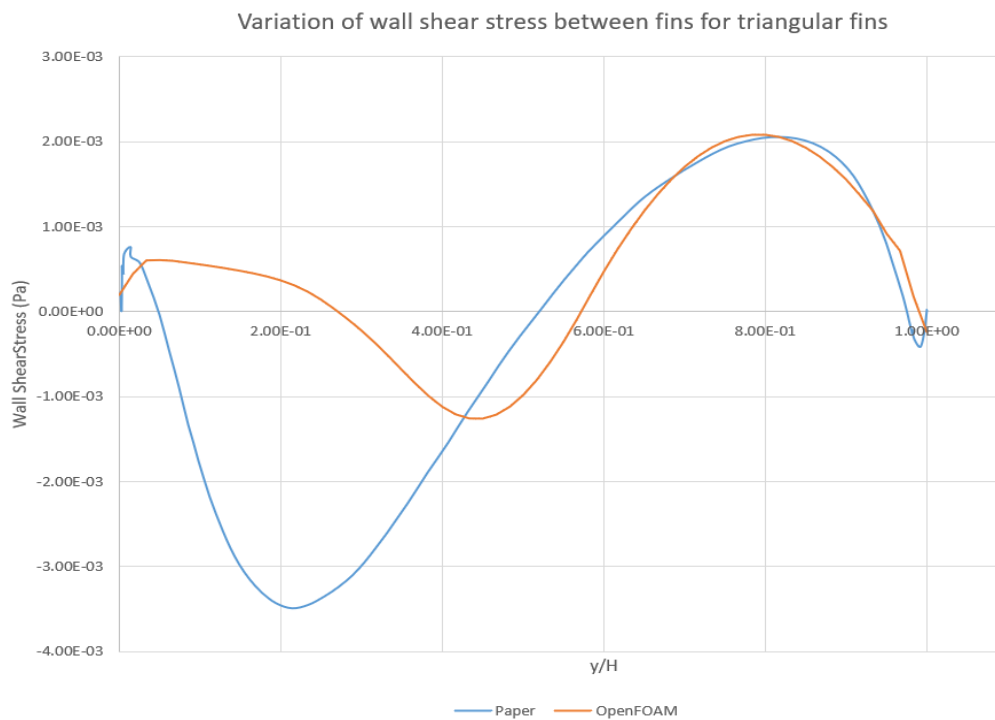Figure 9: Wall shear stress vs y/H for square fins

Figure 10: Wall shear stress vs y/H for triangle fins

Acknowledgement

References

[1]   B. Nghana, F. Tariku, and G. Bitsuamlak, "Numerical assessment of the impact of transverse roughness ribs on the turbulent natural convection in a BIPV air channel," *Build. Environ.*, vol. 217, no. March, p. 109093, 2022, doi: 10.1016/j.buildenv.2022.109093.

[2]   P. S. Pupo , "Validation of a mathematical model for the simulation of loss of coolant accidents in nuclear power plants Mechanical Engineering Examination Committee," no. November, 2016.