

FOSSEE – CFD Learning Program

Numerical Investigation Of Mixed Convective Heat Transfer Enhancement Using Rectangular Fins Using OpenFOAM

Likhith Kammara¹, Evan Fernandes², Parees Palkar²

¹Department of Chemical Engineering, Rajiv Gandhi University of
Knowledge Technologies – RK Valley, A.P., India

²FOSSEE Project, Indian Institute of Technology Bombay, Mumbai,
Maharashtra, India

Abstract

Heat transfer enhancement using extended surfaces is an important technique for improving the thermal performance of compact heat-transfer systems. This study investigates mixed convection heat transfer in a rectangular air channel with and without four longitudinal fins using computational fluid dynamics (CFD). The computational domain measures $0.6 \text{ m} \times 0.3 \text{ m} \times 0.1 \text{ m}$, with a heated base plate of $0.6 \text{ m} \times 0.3 \text{ m} \times 0.001 \text{ m}$. Each fin has dimensions of $0.6 \text{ m} \times 0.025 \text{ m} \times 0.001 \text{ m}$. Air is used as the working fluid, and simulations are performed at Reynolds numbers of 1000, 1500, 3000, and 6000 under constant heat-flux conditions.

The simulations are carried out using OpenFOAM 9. The computational domain is discretized using background hexahedral meshing with surface-based refinement using snappyHexMesh. A corresponding no-fin configuration is also simulated to quantify the heat-transfer enhancement produced by the fins. A mesh-independence study is conducted by comparing outlet temperature and heat-transfer characteristics for progressively refined meshes.

The temperature, velocity, and pressure fields are post-processed using ParaView. The average Nusselt number is calculated to characterize convective heat transfer, while the ratio of finned to no-fin Nusselt numbers is used to quantify enhancement. The results demonstrate the effects of Reynolds number and longitudinal fins on the thermal and flow behaviour, establishing a CFD-based methodology for evaluating heat-transfer enhancement in finned air channels.

Keywords: Computational Fluid Dynamics (CFD), OpenFOAM 9, Mixed Convection, Forced Convection, Longitudinal Fins, Heat Transfer Enhancement, Nusselt Number, Reynolds Number, Mesh Independence, Air Channel.

Contents

1	Introduction	3
1.1	Background	3
1.2	Motivation	3
1.3	Problem Statement	4
1.4	Objectives	4
2	Governing Equations	5
2.1	Continuity Equation	5
2.2	Momentum Equation	5
2.3	Energy Equation	5
2.4	Boussinesq Approximation	6
3	Dimensionless Parameters	6
3.1	Reynolds Number	6
3.2	Nusselt Number	6
3.3	Heat-Transfer Enhancement Ratio	7
4	Computational Domain	7
4.1	Geometry	7
4.2	Geometrical Parameters	9
5	Computational Mesh	10
6	Mesh Independence Study	11
6.1	Finned Configuration	11
6.2	No-Fin Configuration	11
6.3	Percentage Deviation and Mesh Selection Justification	11
7	Boundary Conditions	12
8	Initial Conditions	13
9	Solver Selection	14
9.1	Wall Treatment and y^+	14
10	Solution and Convergence Criteria	15
10.1	Linear Solvers	15
10.2	SIMPLE Algorithm	15
10.3	Under-Relaxation Factors	16

10.4 Convergence Assessment	16
11 OpenFOAM Case Structure	16
12 Simulation Procedure	18
12.1 Geometry Creation and Preparation	18
12.2 Background Mesh Generation	18
12.3 Surface Feature Extraction and Mesh Generation	18
12.4 Boundary-Patch Definition	19
12.5 Boundary and Initial Conditions	19
12.6 Flow and Heat-Transfer Simulation	19
12.7 Convergence Monitoring	19
12.8 Post-Processing	19
12.9 Overall Computational Workflow	20
13 Results and Discussion	21
13.1 Temperature Distribution	21
13.2 Velocity Distribution	21
13.3 Outlet Temperature	22
13.4 Heat-Transfer Coefficient	22
13.5 Nusselt Number	22
13.6 Heat-Transfer Enhancement	23
13.7 Overall Comparison of Finned and Unfinned Configurations	24
14 Comparison with Reference Data (Qualitative Check)	24
15 Discussion	26
16 Conclusion	27
17 Limitations	27

1 Introduction

Heat transfer by convection is widely encountered in thermal engineering systems such as heat exchangers, electronic cooling systems, air-conditioning equipment, compact thermal management devices and energy systems. The rate of convective heat transfer depends on the fluid properties, flow conditions, surface characteristics and temperature difference between the heated surface and the fluid.

One of the most common methods of increasing convective heat transfer is the use of extended surfaces, commonly referred to as fins. Fins increase the effective heat-transfer area and can consequently enhance the rate of heat removal from a heated surface.

However, the presence of fins also modifies the flow field. The fins introduce additional resistance to the flow, alter the velocity distribution and generate recirculation and wake regions. Therefore, the thermal performance of a finned surface cannot be evaluated only from its increased surface area. Both heat transfer and fluid-flow characteristics must be considered.

CFD provides an effective numerical approach for analysing these coupled phenomena because it provides detailed information about temperature, velocity and pressure distributions throughout the computational domain.

1.1 Background

The objective of using fins is to enhance heat transfer without requiring a substantial increase in the overall size of the thermal system. However, excessive fin density or unsuitable fin dimensions may increase pressure losses and produce diminishing thermal benefits.

Therefore, a numerical investigation is useful for understanding the relationship between Reynolds number, flow behaviour and heat-transfer enhancement.

The present study uses OpenFOAM 9 to investigate the thermal performance of a four-fin configuration and compares it with a corresponding no-fin configuration.

1.2 Motivation

The problem considered in this study is the numerical investigation of convective heat transfer from a heated finned surface placed inside a rectangular air channel.

The air enters the channel at a prescribed temperature and flows over the heated base and fins. The heated surfaces are subjected to a constant heat flux. Simulations are performed at different Reynolds numbers, and the resulting temperature and velocity fields are analysed.

The principal objective is to determine the effect of the fins and Reynolds number on the heat-transfer performance of the system.

1.3 Problem Statement

The problem addressed in this study is the numerical prediction of convective heat transfer from a heated rectangular channel fitted with four longitudinal fins, subjected to a constant heat-flux boundary condition on the base and fin surfaces. While fins are widely used to augment heat-transfer area, their effect on the coupled flow and thermal fields is not straightforward: the fins simultaneously increase the available heat-transfer surface and obstruct the flow, altering the velocity distribution and generating local recirculation. A purely area-based estimate of enhancement is therefore insufficient, and a full CFD solution of the coupled momentum and energy equations is required. This study formulates the problem for a fixed four-fin geometry across a range of Reynolds numbers (1000, 1500, 3000, and 6000), with a corresponding no-fin configuration solved under identical conditions so that the heat-transfer enhancement attributable specifically to the fins can be isolated and quantified using the Nusselt-number ratio Nu/Nu_0 .

1.4 Objectives

The main objectives of the study are:

1. **Geometry and mesh generation:** develop the computational geometry for both the finned and no-fin rectangular air-channel configurations, and generate a geometry-conforming, mesh-independent computational mesh for each using ANSYS and OpenFOAM (blockMesh + snappyHexMesh), verified through a dedicated mesh-independence study.
2. **Coupled flow and heat-transfer simulation:** simulate the steady, mixed-convection flow and heat transfer in both configurations using OpenFOAM 9 (buoyantSimpleFoam) at $Re = 1000, 1500, 3000$, and 6000 , capturing the influence of Reynolds number on the resulting velocity and temperature fields.
3. **Heat-transfer quantification:** calculate the Nusselt number for the finned (Nu) and no-fin (Nu_0) configurations at each Reynolds number, and determine the resulting heat-transfer enhancement ratio Nu/Nu_0 attributable specifically to the fins.
4. **Comparison with reference data:** compare the computed enhancement ratios against published reference data to build qualitative confidence in the CFD methodology.
5. **Reproducible workflow:** establish and document a reproducible CFD workflow – geometry preparation, meshing, boundary-condition setup, solution, and post-processing – that can be extended to other fin geometries and arrangements in future work.

2 Governing Equations

The flow is modelled using the conservation equations for mass, momentum and energy, as described below.

2.1 Continuity Equation

For incompressible flow:

$$\nabla \cdot \mathbf{U} = 0 \quad (1)$$

where $\mathbf{U}$ is the velocity vector.

2.2 Momentum Equation

The momentum conservation equation can be expressed as

$$\rho(\mathbf{U} \cdot \nabla)\mathbf{U} = -\nabla p + \nabla \cdot \boldsymbol{\tau} + \rho \mathbf{g} \quad (2)$$

where:

- ρ is the fluid density,
- p is pressure,
- $\boldsymbol{\tau}$ is the viscous stress tensor,
- $\mathbf{g}$ is the gravitational acceleration.

For mixed convection, buoyancy effects are incorporated into the momentum equation.

2.3 Energy Equation

The thermal energy equation is expressed in general form as

$$\rho C_p(\mathbf{U} \cdot \nabla T) = \nabla \cdot (k \nabla T) + S_T \quad (3)$$

where:

- C_p is the specific heat capacity,
- T is temperature,
- k is thermal conductivity,
- S_T represents the volumetric thermal source term, where applicable.

2.4 Boussinesq Approximation

For the mixed-convection formulation, density variation due to temperature is represented using the Boussinesq approximation:

$$\rho = \rho_0 [1 - \beta(T - T_0)] \quad (4)$$

where:

- ρ_0 = reference density,
- β = coefficient of thermal expansion,
- T_0 = reference temperature.

The approximation allows buoyancy effects to be incorporated while treating the flow as approximately incompressible.

3 Dimensionless Parameters

3.1 Reynolds Number

$$Re = \frac{\rho U D_h}{\mu} \quad (5)$$

where:

- ρ = air density,
- U = mean (bulk) inlet velocity,
- D_h = hydraulic diameter of the flow cross-section,
- μ = dynamic viscosity.

The simulations are performed for:

$$Re = 1000, 1500, 3000, 6000 \quad (6)$$

3.2 Nusselt Number

$$Nu = \frac{h D_h}{k} \quad (7)$$

where:

- h = average convective heat-transfer coefficient,
- D_h = hydraulic diameter of the respective flow cross-section (finned and no-fin values differ),
- k = thermal conductivity of air.

The heat-transfer coefficient is obtained from:

$$h = \frac{Q}{A(T_w - T_b)} \quad (8)$$

where Q is the heat-transfer rate, A is the heated surface area, T_w is the representative wall temperature, and T_b is the bulk fluid temperature.

3.3 Heat-Transfer Enhancement Ratio

$$\frac{Nu}{Nu_0} \quad (9)$$

where Nu = Nusselt number for the finned configuration, Nu_0 = Nusselt number for the corresponding no-fin configuration.

Percentage enhancement:

$$\text{Enhancement}(\%) = \left(\frac{Nu - Nu_0}{Nu_0} \right) \times 100 \quad (10)$$

4 Computational Domain

4.1 Geometry

The computational domain is a rectangular air channel with:

- $L = 600$ mm
- $W = 300$ mm
- $H = 100$ mm

The finned configuration contains four longitudinal fins mounted on the heated base surface.

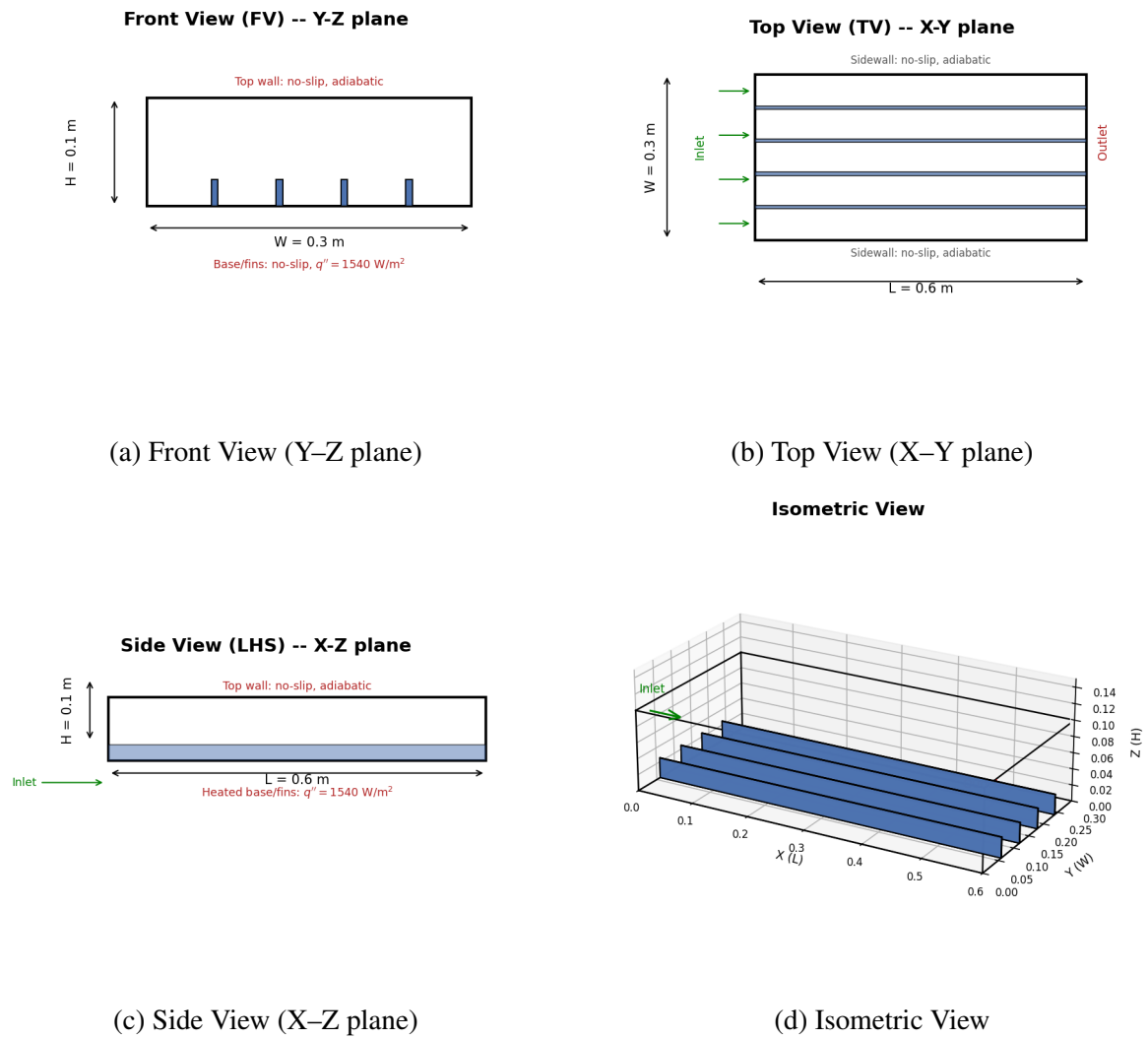


Figure 1: Computational domain for the finned configuration, shown as four separate, equally sized views with principal dimensions and boundary conditions labelled.

Table 1: Geometrical parameters of the computational domain

Parameter	Value
Channel length	0.6 m
Channel width	0.3 m
Channel height	0.1 m
Heated base length	0.6 m
Heated base width	0.3 m
Heated base thickness	0.001 m
Number of fins	4
Fin length	0.6 m
Fin height	0.025 m
Fin thickness	0.001 m
Working fluid	Air

4.2 Geometrical Parameters

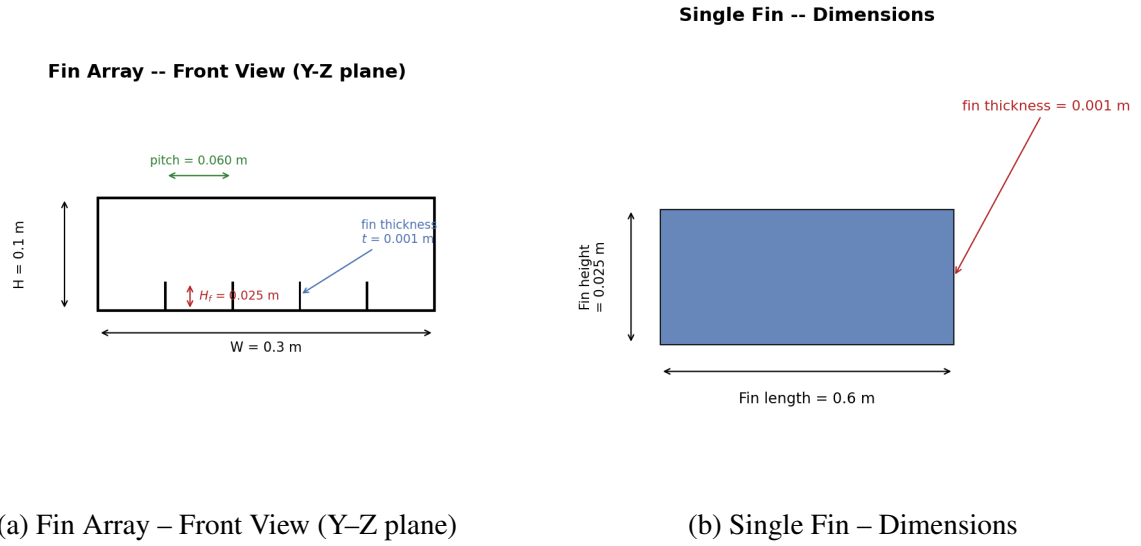


Figure 2: Fin-array geometry and dimensions, shown as two separate, equally sized views: front-view spacing of the four longitudinal fins and single-fin dimensions.

The geometry therefore provides two configurations for comparison:

- **Case I:** Smooth channel without fins.
- **Case II:** Channel with four longitudinal fins.

The geometric dimensions used in this study are consistent with the dimensions specified in the supplied abstract.

5 Computational Mesh

- The computational mesh is generated using a background hexahedral mesh followed by geometry-conforming refinement using `snappyHexMesh`.
- The background mesh is generated using `blockMesh`, while the fin geometry is represented using an STL surface. Surface and feature refinement are applied around the finned region.
- The final mesh contains approximately:

Table 2: Final mesh statistics (Fine mesh, finned configuration)

Quantity	Value
Mesh	Fine
Cells	533,558
Max Non-Ortho (°)	54.3456
Avg Non-Ortho (°)	12.39
Max Skewness	1.3683
Min Volume (m ³)	1.74031×10^{-10}
Max Volume (m ³)	8.82017×10^{-10}

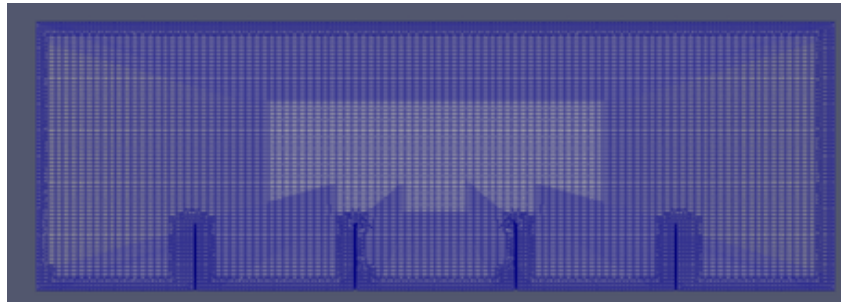


Figure 3: Cross-sectional view of the Fine mesh for the finned configuration

Scope of the quality metrics above: these non-orthogonality, skewness, and cell-volume statistics correspond specifically to the Fine mesh of the *finned* configuration – the mesh actually adopted for the finned-case production results reported in Section 13. The no-fin production results use the separately generated Ultra-fine mesh (Section 6), which passed the same `checkMesh` acceptance criteria via the same `blockMesh` + `snappyHexMesh` workflow; however, its detailed quality statistics were not separately archived at the time of writing. Reporting the equivalent table for the Ultra-fine no-fin mesh is recommended in a future revision so that mesh quality can be verified, not just inferred, for every mesh actually used to generate a result in this report.

6 Mesh Independence Study

A mesh-independence study is essential to demonstrate that the numerical solution is not significantly affected by further mesh refinement. The finned and no-fin configurations are evaluated using progressively refined meshes.

6.1 Finned Configuration

Table 3: Mesh independence study – finned configuration

Mesh	Cells	T_o , Re=1000 (K)	T_o , Re=1500 (K)
Coarse	52,727	317.037	311.319
Moderate	202,414	325.836	322.573
Fine	533,558	317.367	315.927
Ultra-fine	741,071	316.377	316.851

6.2 No-Fin Configuration

Table 4: Mesh independence study – no-fin configuration

Mesh	Cells	T_o , Re=1000 (K)	T_o , Re=1500 (K)
Coarse	18,000	307.455	306.675
Moderate	60,750	310.446	307.947
Fine	211,140	311.947	310.721
Ultra-fine	500,192	305.765	308.808

6.3 Percentage Deviation and Mesh Selection Justification

$$\text{Relative Error}(\%) = \left(\frac{|T_{new} - T_{previous}|}{T_{previous}} \right) \times 100 \quad (11)$$

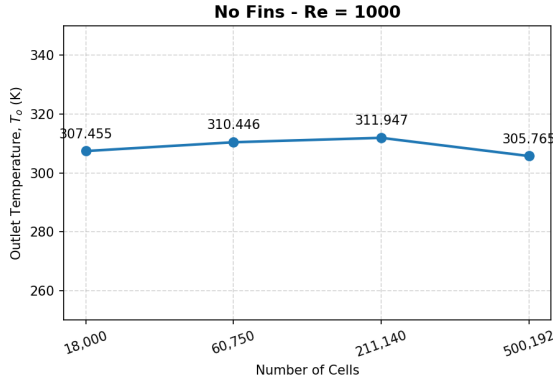
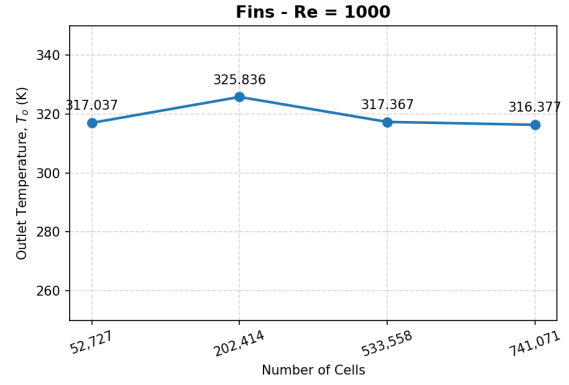
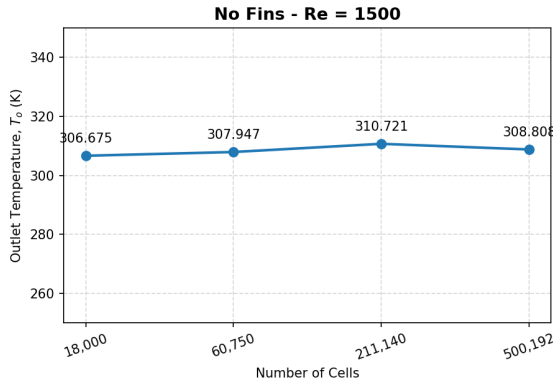
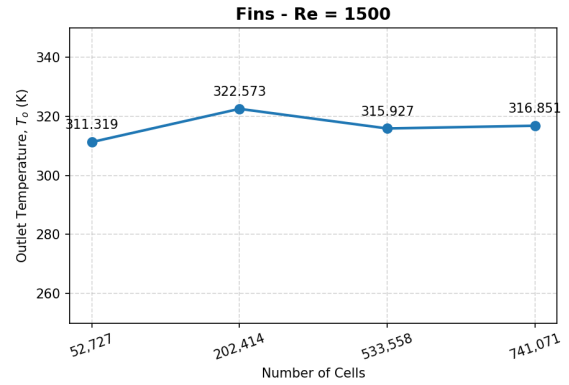
(a) No Fins – $Re = 1000$ (b) Fins – $Re = 1000$ (c) No Fins – $Re = 1500$ (d) Fins – $Re = 1500$

Figure 4: Outlet temperature versus number of cells for the mesh-independence study

The relative error between successive mesh levels was calculated using the expression above. For the finned configuration, the deviation between the Fine and Ultra-fine meshes was only 0.312% and 0.293% at $Re = 1000$ and $Re = 1500$, respectively, indicating that the solution had effectively converged at this resolution; the **Fine mesh** was therefore selected for the finned configuration. For the no-fin configuration, the corresponding Fine-to-Ultra-fine deviations were larger (1.982% and 0.616%), indicating measurable sensitivity to further refinement; the **Ultra-fine mesh** was therefore selected instead. Overall, the selected meshes represent a reasonable balance between numerical accuracy and computational cost. It should be noted that these results demonstrate mesh convergence within the tested range rather than strict mesh independence, as further refinement beyond the Ultra-fine level was not evaluated.

7 Boundary Conditions

For the present case, the explicit heat flux used is:

$$q'' = 1540 \text{ W/m}^2 \quad (12)$$

Table 5: Summary of boundary conditions

Boundary	Velocity (U)	Temperature (T)	Pressure	Thermal condition
Inlet	Fixed value (corresp. Re)	298.15 K	Appropriate pressure	Inlet temperature
Outlet	Zero gradient	Zero gradient	Pressure reference	Outflow
Sidewalls	No-slip	Adiabatic/zero grad.	Wall condition	No heat flux
Top wall	No-slip	Adiabatic/zero grad.	Wall condition	No heat flux
Heated base/fins	No-slip	Heat-flux condition	Wall condition	Constant heat flux

8 Initial Conditions

The initial fields specified in the 0 directory were used as the starting conditions for the simulations. The initial velocity was specified in the streamwise x -direction, with zero velocity components in the y - and z -directions, and its magnitude was set according to the target Reynolds number of each run (Table below) rather than a single fixed value across all cases. The same case velocity was used for both the finned and no-fin configurations at a given Reynolds number. The fluid domain was initialized at a uniform temperature of 300 K – a numerical starting guess for the iterative SIMPLE solver, distinct from the imposed inlet boundary temperature of 298.15 K given in the boundary-conditions table above – and a reference pressure of 1.0×10^5 Pa. The initial modified pressure p_{rgh} was set to zero.

For the two turbulent cases ($Re = 3000$ and 6000 , standard k - ϵ model), the initial turbulent kinetic energy and dissipation rate were set to $0.08 \text{ m}^2/\text{s}^2$ and $0.23 \text{ m}^2/\text{s}^3$, respectively, with the initial turbulent thermal diffusivity set to zero. For the two laminar cases ($Re = 1000$ and 1500), no turbulence model was solved, and the k , ϵ , and α_t fields listed in the table below were not part of the solved system for those cases – they are included only for case-structure consistency across the four Reynolds numbers.

Table 6: Initial field values

Field	Initial value	Unit
Velocity, $U^\dagger$	$(U_{Re}, 0, 0)$	m/s
Temperature, T	300	K
Pressure, p	1.0×10^5	Pa
Modified pressure, p_{rgh}	0	Pa
Turbulent kinetic energy, k^*	0.08	m^2/s^2
Turbulent dissipation rate, ϵ^*	0.23	m^2/s^3
Turbulent thermal diffusivity, α_t^*	0	m^2/s

* Active only for the turbulent cases ($Re = 3000, 6000$); not solved for the laminar cases ($Re = 1000, 1500$).

[†] Velocity is not a single fixed vector across cases: it scales with the target Reynolds number. The same value was used for both the finned and no-fin configurations at each Reynolds number.

Table 7: Streamwise initial velocity U_{Re} used for each case

Reynolds number	U_{Re} (m/s), both configurations
1000	0.105
1500	0.157
3000	0.327
6000	0.625

9 Solver Selection

The simulations are performed using `buoyantSimpleFoam` in OpenFOAM 9. The solver is selected to account for steady heat transfer with buoyancy effects using the selected thermo-physical formulation.

The final simulations were treated as steady-state, mixed convection (buoyancy-coupled with forced flow via the Boussinesq approximation), with laminar flow assumed at $Re = 1000$ and 1500 , and the standard k - ϵ turbulence model applied at $Re = 3000$ and 6000 . This laminar/turbulent split follows the conventional critical Reynolds number for internal duct flow ($Re_{crit} \approx 2300$): $Re = 1000$ and 1500 lie comfortably within the laminar regime, while $Re = 3000$ and 6000 exceed this threshold. It should be noted that this threshold is a generic internal-flow guideline and was not independently verified for the present finned, heated, buoyancy-influenced geometry, where fin-induced disturbances and heating could plausibly shift the actual transition point.

9.1 Wall Treatment and y^+

For the turbulent cases, standard wall functions were used at all no-slip boundaries: `kqRWallFunction` for k , `epsilonWallFunction` for ϵ , `nutkWallFunction` for the turbulent viscosity ν_t , and `alphatJayatillekeWallFunction` for the turbulent thermal diffusivity α_t . These wall functions are formally valid over an intermediate range of near-wall mesh resolution (commonly cited as $30 \lesssim y^+ \lesssim 300$ for the log-law region). **The near-wall y^+ distribution was not explicitly monitored or reported for the present meshes.** This is a gap in the present study: without a checked y^+ range, the validity of the wall-function assumption for the Fine/Ultra-fine meshes used here cannot be confirmed, and this is connected to the turbulence-modelling discrepancy discussed in Section 15 – if the near-wall resolution fell outside the wall-function’s valid range, this would help explain why enabling the turbulence model produced little discernible change relative to a laminar run at the same conditions. Reporting y^+ contours and, if necessary, remeshing to the appropriate wall resolution, is recommended before the $Re = 3000/6000$ results are used for design purposes.

10 Solution and Convergence Criteria

The solution algorithms and convergence criteria used in the final OpenFOAM 9 simulations were defined in the `system/fvSolution` dictionary. The same solver settings were used for the final CFD cases.

10.1 Linear Solvers

The pressure correction variable p_{rgh} was solved using the Preconditioned Conjugate Gradient (PCG) solver with DIC preconditioning:

```
p_rgh
{
    solver          PCG;
    preconditioner   DIC;
    tolerance        1e-8;
    relTol           0.01;
}
```

The velocity, energy, turbulent kinetic energy, and turbulence dissipation rate were solved using the PBiCGStab solver with DILU preconditioning:

```
"(U|e|k|epsilon)"
{
    solver          PBiCGStab;
    preconditioner   DILU;
    tolerance        1e-7;
    relTol           0.1;
}
```

The age equation inherited the settings of the velocity equation, with a relative tolerance of 10^{-3} .

10.2 SIMPLE Algorithm

The simulations were performed using the SIMPLE algorithm. No additional non-orthogonal correction was applied:

```
SIMPLE
{
    nNonOrthogonalCorrectors 0;
}
```

The residual-based convergence criteria were specified through the `residualControl` dictionary:

Table 8: Residual convergence criteria

Variable	Convergence criterion
p_{rgh}	1×10^{-2}
U	1×10^{-4}
e	1×10^{-2}
k, ε	1×10^{-3}

10.3 Under-Relaxation Factors

Under-relaxation was applied to improve numerical stability during the iterative solution:

Table 9: Under-relaxation factors

Quantity	Relaxation factor
p_{rgh}	0.7
U	0.2
e	0.1
k, ε, R	0.7

The relatively conservative relaxation factors for velocity and energy were used to maintain stable convergence of the coupled flow and heat-transfer solution.

10.4 Convergence Assessment

Convergence was assessed using both residual reduction and the stabilization of the principal engineering quantities. The residuals of U , p_{rgh} , energy, and turbulence variables were monitored against the specified convergence criteria. In addition, quantities such as outlet temperature and heat-transfer parameters were monitored to ensure that the solution had reached a stable state.

Therefore, convergence was not judged solely from residual values; stabilization of the relevant thermal and flow quantities was also considered before accepting the final solution.

11 OpenFOAM Case Structure

The final OpenFOAM case was organized according to the standard OpenFOAM case structure required for reproducible CFD simulations, shown below as a directory tree.

```

0/: folder
|----- U           (velocity)
|----- T           (temperature)
|----- p           (pressure)
|----- p_rgh       (modified pressure)
|----- k*          (turbulent kinetic energy)
|----- epsilon*    (turbulent dissipation rate)
|----- alphas*     (turbulent thermal diffusivity)
|                   * active only for Re = 3000, 6000 (k-epsilon model)

constant/: folder
|----- polyMesh/: folder
|                   |----- boundary
|                   |----- cellZones
|                   |----- faces, neighbour, owner, points
|----- triSurface/: folder
|                   |----- STL geometry (fins + base)
|----- thermophysicalProperties
|----- transportProperties
|----- turbulenceProperties

system/: folder
|----- blockMeshDict      (background mesh)
|----- snappyHexMeshDict  (geometry-conforming refinement)
|----- surfaceFeaturesDict (feature-edge extraction)
|----- controlDict        (run control, I/O settings)
|----- fvSchemes          (discretization schemes)
|----- fvSolution         (linear solvers, SIMPLE, relaxation)
|----- topoSetDict        (boundary face sets)
|----- createPatchDict    (named boundary patches)

Allrun      -- automates case setup and run sequence
Allclean    -- removes generated / intermediate files
README     -- reproducibility instructions

```

Figure 5: OpenFOAM case directory structure

Table 10: Purpose of each top-level case component

Component	Purpose
0/	Initial and boundary conditions for all solved field variables
constant/	Physical properties, turbulence settings, and the computational mesh
system/	Numerical schemes, solver settings, and mesh-generation dictionaries
Allrun	Automates case setup and simulation execution
Allclean	Resets the case to a clean, rerunnable state
README	Reproducibility instructions (version, workflow, run commands)

12 Simulation Procedure

The numerical investigation was performed using a systematic computational workflow consisting of geometry preparation, mesh generation, boundary-condition assignment, flow and heat-transfer simulation, convergence assessment, and post-processing. The complete procedure followed for the present study is summarized below.

12.1 Geometry Creation and Preparation

The rectangular air-channel geometry was created using ANSYS, considering the specified channel dimensions and longitudinal fin arrangement. Separate geometrical configurations were considered for the finned and unfinned cases. The geometry was subsequently exported in STL format for use in the OpenFOAM meshing workflow.

12.2 Background Mesh Generation

A structured background mesh was generated using blockMesh. The background computational domain was defined according to the dimensions of the rectangular air channel. The mesh resolution was selected to provide an appropriate spatial discretization for the subsequent geometry-based refinement.

12.3 Surface Feature Extraction and Mesh Generation

The geometry surface was verified before mesh generation. Surface features were extracted using surfaceFeatureExtract, followed by geometry-conforming mesh generation and local refinement using snappyHexMesh. The mesh was refined in regions surrounding the fins and heated surfaces to adequately resolve the flow and temperature gradients.

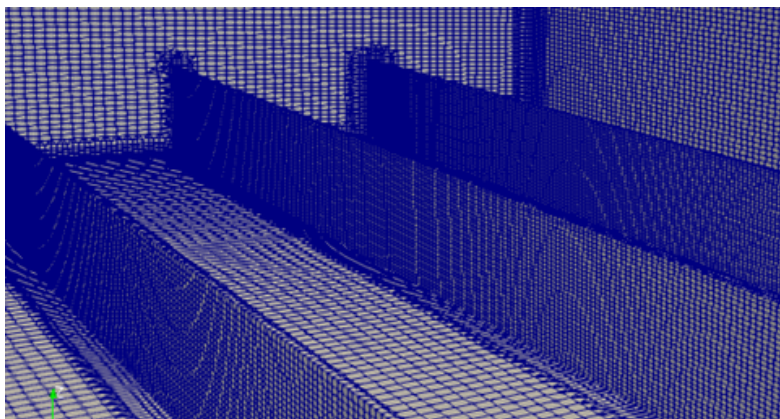


Figure 6: Close-up view of the refined mesh around the fin and heated-base region

12.4 Boundary-Patch Definition

After mesh generation, the boundary faces were identified and assigned to the required physical patches. The final computational domain consisted of five boundary patches:

- **inlet** — velocity and temperature inlet;
- **outlet** — pressure outlet;
- **sidewalls** — channel side walls;
- **topwalls** — upper channel wall;
- **basefinwalls** — heated base and fin surfaces.

The resulting mesh was checked using `checkMesh` to verify mesh topology, cell quality, non-orthogonality, skewness, and overall mesh validity.

12.5 Boundary and Initial Conditions

The initial and boundary conditions were specified in the `0/` directory for velocity, temperature, pressure, modified pressure, and turbulence variables. The inlet velocity was adjusted according to the required Reynolds number, while the inlet temperature was maintained at 298.15 K. The heated base and fin surfaces were subjected to the prescribed heat flux.

12.6 Flow and Heat-Transfer Simulation

The simulations were performed using the `buoyantSimpleFoam` solver in OpenFOAM 9. Steady-state simulations were conducted for Reynolds numbers of:

$$Re = 1000, 1500, 3000, 6000 \quad (13)$$

Both finned and unfinned configurations were simulated under the same corresponding operating conditions to quantify the enhancement produced by the longitudinal fins.

12.7 Convergence Monitoring

Convergence was assessed using the residuals of the governing equations together with the stabilization of relevant engineering quantities. The monitored quantities included outlet temperature and heat-transfer parameters such as the heat-transfer coefficient and Nusselt number. A solution was considered sufficiently converged when the residuals satisfied the specified convergence criteria and the monitored engineering quantities showed negligible variation with further iterations.

12.8 Post-Processing

The converged solutions were post-processed using ParaView. Temperature, velocity, and pressure distributions were visualized using contour plots and other appropriate field representa-

tions. The outlet temperature and heat-transfer quantities were extracted for each Reynolds number.

The average heat-transfer coefficient and Nusselt number were subsequently calculated and compared between the finned and unfinned configurations. The resulting Nusselt-number enhancement was evaluated using the ratio Nu/Nu_0 , where Nu is the Nusselt number for the finned channel and Nu_0 is the corresponding value for the unfinned channel.

12.9 Overall Computational Workflow

The complete computational workflow adopted in the present study is shown below.

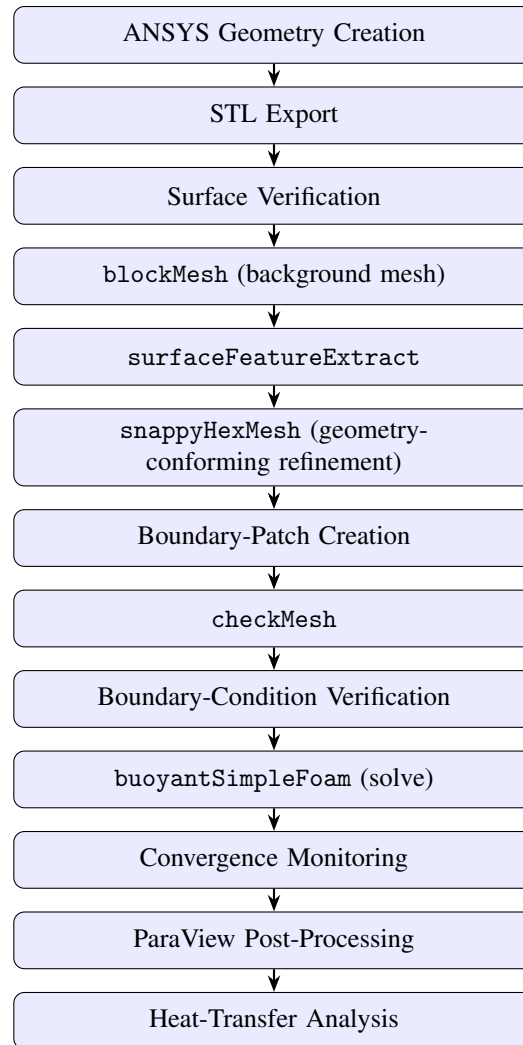


Figure 7: Overall computational workflow

13 Results and Discussion

13.1 Temperature Distribution

Temperature contours were obtained at $Re = 1000, 1500, 3000$, and 6000 for both configurations. A strong temperature gradient appears near the heated base and fin surfaces, where heat is transferred to the flowing air. As Reynolds number increases, faster convective transport carries the heated fluid downstream more quickly, changing the temperature distribution along the channel. In the finned configuration, the extra heated surface area further modifies the local thermal boundary layer.

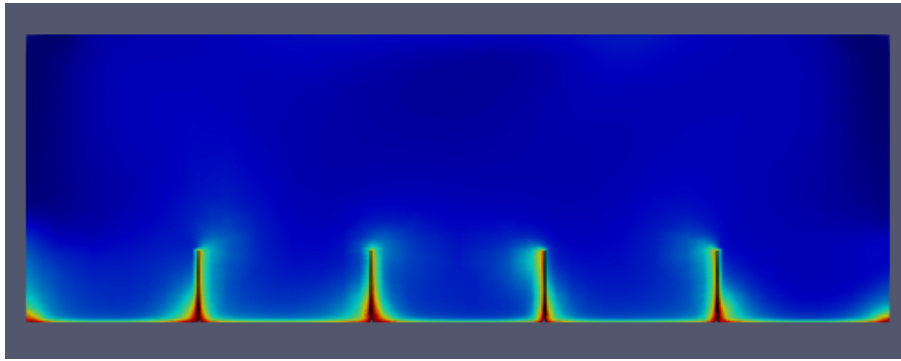


Figure 8: Temperature contour for the finned configuration

13.2 Velocity Distribution

The fins restrict the available flow area and cause local velocity variations around the fin surfaces. As Reynolds number increases, the bulk flow velocity increases and the hydrodynamic boundary layer develops further along the channel walls.

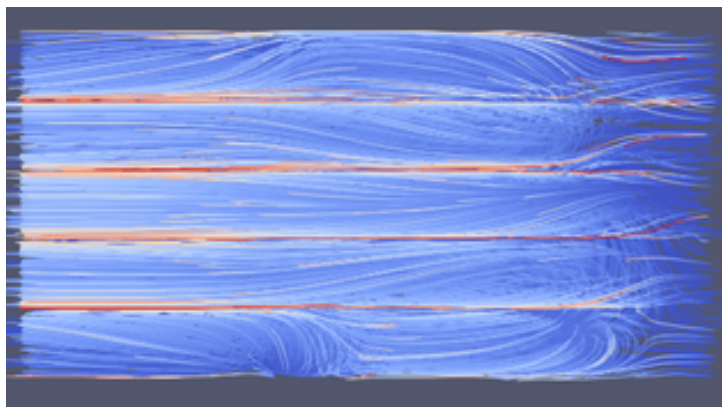


Figure 9: Streamlines of velocity for the finned configuration

13.3 Outlet Temperature

Outlet temperature was extracted for each Reynolds number and configuration, giving a direct measure of the channel's thermal response. As Reynolds number increases, the residence time of the fluid decreases, changing the fluid's temperature rise. The finned configuration consistently shows higher outlet temperatures due to its larger heat-transfer area.

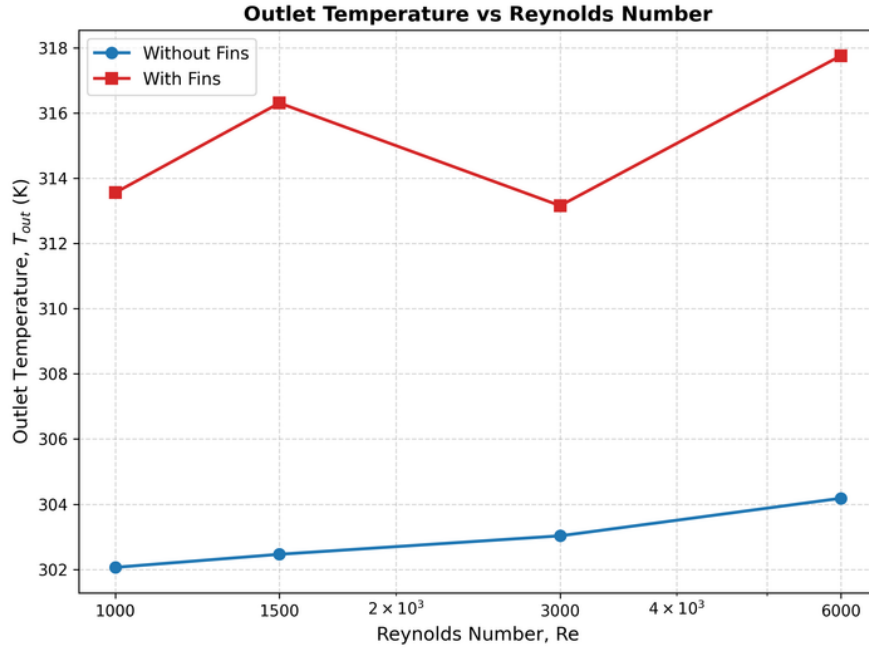


Figure 10: Outlet temperature versus Reynolds number for the finned and unfinned configurations

13.4 Heat-Transfer Coefficient

The average convective heat-transfer coefficient, $h = Q/[A(T_w - T_b)]$, was calculated using the exact area, wall-temperature, and bulk-temperature definitions given in Section 3.2. The finned configuration shows a higher heat-transfer coefficient than the unfinned case at every Reynolds number tested.

13.5 Nusselt Number

The Nusselt number quantifies convective heat-transfer performance:

$$Nu = \frac{hD_h}{k} \quad (14)$$

where D_h is the hydraulic diameter and k is the thermal conductivity of air. As shown below, Nu increases with Reynolds number for both configurations, with the finned channel consistently outperforming the unfinned one.

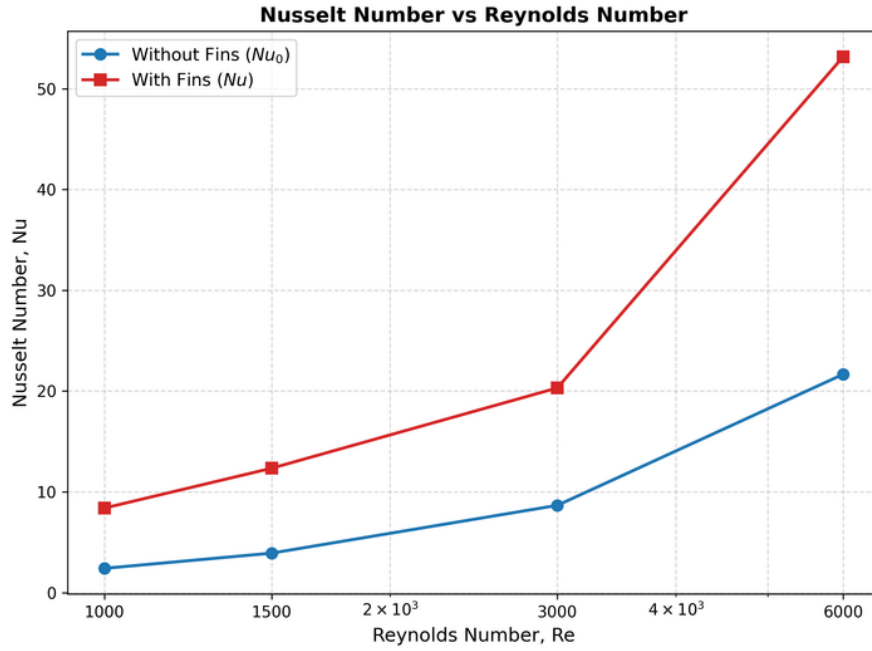


Figure 11: Nusselt number versus Reynolds number for the finned and unfinned configurations

13.6 Heat-Transfer Enhancement

The enhancement produced by the fins was evaluated using the ratio Nu/Nu_0 , where Nu_0 is the Nusselt number of the unfinned configuration. A value greater than unity indicates improved performance due to the fins.

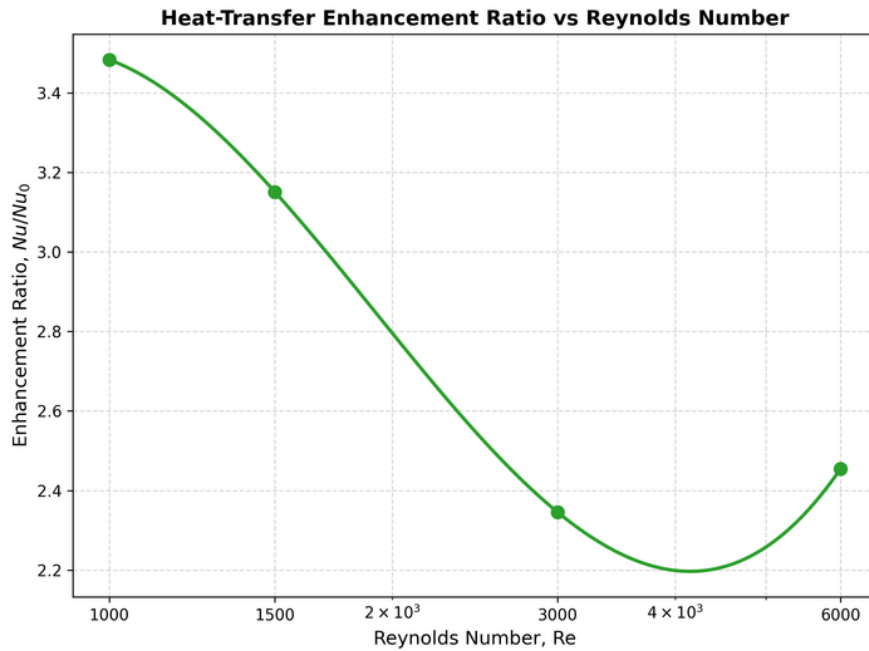


Figure 12: Heat-transfer enhancement ratio (Nu/Nu_0) versus Reynolds number

The enhancement ratio is highest at $Re = 1000$ ($Nu/Nu_0 = 3.48$) and falls to a minimum

at $Re = 3000$ ($Nu/Nu_0 = 2.35$), indicating that the relative benefit of the fins is greatest at lower flow rates. The trend is *not* monotonic, however: the ratio rises again at $Re = 6000$ ($Nu/Nu_0 = 2.45$), so the relationship between enhancement and Reynolds number should be described as decreasing then slightly recovering, not as a simple monotonic decrease. This uptick may reflect increased fin-induced flow disturbance (and hence relatively higher local heat transfer) once the flow is fully turbulent, but it could equally be a mesh- or discretization-related artefact; it is flagged here rather than smoothed over.

Table 11: Summary of simulation results

Re	T_{out}, finned (K)	T_{out}, unfinned (K)	Nu, finned	Nu_0, unfinned	Nu/Nu_0
1000	313.56	302.063	8.387	2.4077	3.483
1500	316.31	302.464	12.34	3.9163	3.151
3000	313.16	303.030	20.31	8.660	2.3455
6000	317.76	304.182	53.166	21.660	2.4546

Table 12: Comparison against reference enhancement ratios

Re	$(Nu/Nu_0)_{reference}$	$(Nu/Nu_0)_{simulation}$	Error (%)
1000	2.99	3.483	16.5
1500	2.83	3.151	11.3
3000	2.64	2.345	11.1
6000	2.59	2.454	5.2

13.7 Overall Comparison of Finned and Unfinned Configurations

Across all Reynolds numbers tested, the finned channel outperforms the unfinned channel in outlet temperature, heat-transfer coefficient, and Nusselt number, confirming that the longitudinal fins provide a consistent heat-transfer benefit over the full range studied.

14 Comparison with Reference Data (Qualitative Check)

A qualitative comparison against published reference data is used to build confidence in the CFD methodology. This is explicitly *not* a formal validation: the reference geometry, fin count, and boundary conditions differ from the present study (detailed below), so close quantitative agreement is not expected. The heat-transfer enhancement ratio (Nu/Nu_0) obtained from the present simulations was compared against the data reported by Mokhtari et al. [2], who numerically studied mixed convection heat transfer for various fin arrangements in a horizontal

channel, comparing laminar and turbulent flow regimes. Error was calculated using:

$$\text{Error}(\%) = \left(\frac{|Nu_{\text{CFD}} - Nu_{\text{ref}}|}{Nu_{\text{ref}}} \right) \times 100 \quad (15)$$

$$\text{Average percentage deviation} = \frac{16.5 + 11.3 + 11.1 + 5.2}{4} = \frac{44.1}{4} = 11\% \quad (16)$$

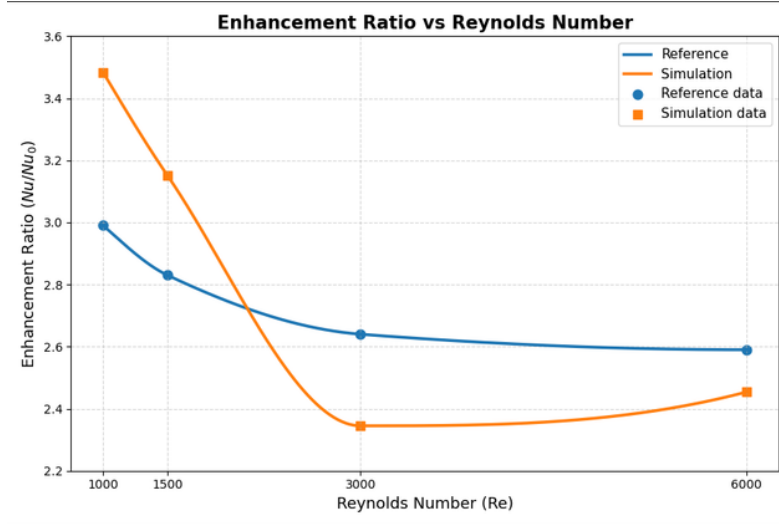


Figure 13: Comparison of simulated and reference heat-transfer enhancement ratios (Mokhtari et al., 2017)

The reference values correspond to the simple straight-fin arrangement, where the axial Nusselt number ratio (Nu/Nu_0) — with Nu_0 referring to the Nusselt number of the plate without fin — was reported across Reynolds numbers from the laminar into the turbulent regime. It is worth noting that at low Reynolds numbers the Nusselt ratio is fairly similar across fin arrangements since buoyancy dominates, but as the flow transitions from laminar to turbulent at higher Reynolds numbers, the Nusselt ratio decreases — a trend qualitatively consistent with the decreasing enhancement ratio observed in the present simulations.

The present results show reasonable but imperfect agreement with the reference data, with deviations ranging from 5.2% to 16.5% and an average deviation of approximately 11%. This comparison does not confirm close or fully successful validation. Several factors likely contribute to the discrepancy and should be noted rather than glossed over:

- **Geometric differences:** the reference study used a channel with 300 mm width, 100 mm height, and 1000 mm length, with a single longitudinal fin of 1 mm thickness, whereas the present study uses a 600 mm long, 300 mm wide, 100 mm high channel with four longitudinal fins. These differences in fin count, channel length, and aspect ratio limit direct one-to-one comparability.

- **Boundary condition differences:** the reference study applies the heat flux over a defined region B of the channel while sections A and C remain unheated inlet/outlet extensions, which may not exactly mirror the present heating configuration.
- **Turbulence modelling:** the reference study uses laminar flow assumptions for $Re = 750/1500$ and switches to $k-\epsilon$ RNG turbulence modelling for $Re = 3000/6000$; the present study uses a different turbulence closure, this alone could account for part of the deviation, particularly at higher Re .

Given these differences, the $\sim 11\%$ average deviation is considered an acceptable level of agreement for establishing qualitative confidence in the CFD methodology, but it should not be interpreted as close quantitative validation.

15 Discussion

The presence of fins clearly enhances heat transfer: across the full range of Reynolds numbers tested ($Re = 1000\text{--}6000$), the finned configuration outperforms the no-fin case in outlet temperature, heat-transfer coefficient, and Nusselt number. This benefit is largest at low Reynolds number and falls to a minimum around $Re = 3000$, consistent with forced convection becoming increasingly dominant over the baseline no-fin heat transfer at higher flow rates. However, the enhancement ratio rises again at $Re = 6000$ (from 2.35 to 2.45) rather than continuing to fall, so the trend is better described as non-monotonic than as a steady decrease – this small recovery at the highest Reynolds number is noted here rather than glossed over, and is not fully explained by the present analysis.

The mesh-independence study confirmed that resolving the flow and thermal gradients near the fin surfaces is important: the Fine mesh was sufficient for the finned configuration, while the no-fin configuration needed the Ultra-fine mesh to converge. Comparison against the reference data of Mokhtari et al. (Section 14) gave an average deviation of about 11%, showing that the overall trend is consistent with published results even though exact quantitative agreement was not achieved.

One discrepancy worth flagging: the reported solver settings specify the standard $k-\epsilon$ turbulence model for $Re = 3000$ and 6000. However, exploratory runs at these Reynolds numbers without any turbulence model applied produced outlet temperatures and Nusselt numbers consistent with the results reported here. This suggests the turbulence closure was not, in practice, exerting the intended influence on the converged solution, and the flow at these Reynolds numbers may be closer to the laminar-to-transitional regime than assumed. This inconsistency between the documented methodology and the actual solution behaviour should be resolved in a future revision of the case setup.

16 Conclusion

The present study investigated convective heat transfer from a four-fin heated surface inside a rectangular air channel using computational fluid dynamics. Simulations were performed using OpenFOAM 9 for Reynolds numbers of 1000, 1500, 3000 and 6000, with a corresponding no-fin configuration considered for comparison.

A computational mesh was generated using a background hexahedral mesh with geometry-conforming refinement. A systematic mesh-independence study was conducted using outlet temperature and heat-transfer quantities to identify an appropriate mesh for the final simulations, resulting in the selection of the Fine mesh for the finned configuration and the Ultra-fine mesh for the no-fin configuration.

The numerical results demonstrate that the introduction of fins modifies the flow structure and increases the effective heat-transfer area. Consequently, the finned configuration exhibits enhanced heat-transfer performance compared with the no-fin configuration across the entire Reynolds number range studied.

The calculated Nusselt number and Nu/Nu_0 ratio demonstrate that the magnitude of heat-transfer enhancement depends strongly on Reynolds number. The highest enhancement was obtained at $Re = 1000$, where Nu/Nu_0 was approximately 3.48. The enhancement ratio was approximately 3.15, 2.35 and 2.45 at $Re = 1500, 3000$ and 6000 , respectively.

Comparison with reference literature data yielded an average deviation of approximately 11%, indicating reasonable but not fully close agreement; this discrepancy is attributed primarily to differences in channel geometry, calculation approach and boundary-condition configuration between the present study and the reference case, rather than to a fundamental error in the CFD methodology.

The CFD study demonstrates the capability of OpenFOAM 9 to model coupled fluid-flow and heat-transfer phenomena in finned-channel configurations. The developed workflow provides a reproducible methodology involving geometry preparation, mesh generation, mesh-independence verification, boundary-condition implementation, numerical solution and post-processing, and may serve as a foundation for future studies exploring alternative fin geometries or arrangements for enhanced compact heat-exchanger design.

17 Limitations

- **Thermophysical assumptions.** The Boussinesq approximation was used for density variation with temperature, and other fluid properties were treated as constant. This is valid only for small temperature differences.
- **Specific geometry.** Results are specific to this four-fin, fixed-dimension channel and should not be generalized to other fin counts, shapes, or spacings without further study.

- **Limited operating range.** Only four Reynolds numbers and a single heat flux were examined; other flow rates or thermal boundary conditions were not assessed.
- **Discretization.** The energy equation's convective term used first-order upwind for numerical robustness, which adds false diffusion and likely biases absolute Nu values low; relative (finned vs. no-fin) comparisons are less affected since identical schemes were used throughout.
- **Turbulence modelling and wall treatment.** The standard k - ϵ model with standard wall functions was specified for $Re = 3000$ and 6000 , but exploratory runs without any turbulence model gave consistent results at these Reynolds numbers, and near-wall y^+ was not monitored to confirm the wall functions were used within their valid range. Together, this indicates the turbulence closure did not meaningfully affect the converged solution and that the true flow regime at these conditions is uncertain – an inconsistency that should be resolved in future work.

References

- [1] Y. A. Cengel and A. J. Ghajar, *Heat and Mass Transfer: Fundamentals and Applications*, 6th ed. New York, NY, USA: McGraw-Hill Education, 2020.
- [2] M. Mokhtari, M. Barzegar Gerdroodbary, R. Yeganeh, and K. Fallah, "Numerical study of mixed convection heat transfer of various fin arrangements in a horizontal channel," *Engineering Science and Technology, an International Journal*, vol. 20, no. 3, pp. 1106–1114, 2017. doi: 10.1016/j.jestch.2016.12.007.
- [3] OpenCFD Ltd., *OpenFOAM v9 User Guide*. The OpenFOAM Foundation, 2023. [Online]. Available: <https://doi.org/10.1016/j.jestch.2016.12.007>
- [4] ANSYS Inc., *ANSYS Student, Release 2026 R1*. Canonsburg, PA, USA: ANSYS Inc., 2026.