



## OpenFOAM Simulation of Turbulent Flow Through T-Junction

Prathamesh Joshi  
*M.Tech IIT Bombay*

### 1.Introduction

This report presents a numerical investigation of turbulent flow interactions within a T-junction featuring rectangular cross-sections, with particular emphasis on the merging behavior between a turbulent cross-flow and an incoming turbulent jet. Such flow configurations are commonly encountered in various industrial applications, including heating, ventilation, and air conditioning (HVAC) systems, nuclear reactor cooling circuits, and exhaust gas recirculation systems in automotive engines. Due to their practical significance, T-junction flows have been the subject of extensive experimental and numerical research. In the present study, a subset of results from a referenced journal article is replicated using OpenFOAM which is a open source cfd software. The accuracy of our numerical model is validated by comparing our simulation outcomes with those reported in the literature, thereby ensuring fidelity in capturing the relevant flow phenomena. For this migration report accuracy is validated by comparing the variation of mean non dimensional streamline velocity with respect to the non dimensional vertical distance at specified axial location. Given flow is simulated using RANS (Reynold's Averaged Turbulence Model). Where as in paper it is simulated using LES (Large Eddy Simulation).

### 2.Governing Equations and Turbulence Modeling

In the original study by Georgiou and Papalexandris, a wall-resolved Large Eddy Simulation (LES) model was employed to capture the interaction between a turbulent crossflow and an incoming jet in a rectangular T-junction. The LES model solves the spatially filtered, incompressible Navier-Stokes

equations. Non-dimensionalized using the crossflow bulk velocity  $U$  and the main channel half-width  $\delta$ , the governing equations used in their study are:

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

$$\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} = -\frac{1}{\rho} \nabla p + \frac{1}{Re} \nabla \cdot [(\nu + \nu_t) \nabla \mathbf{u}] \quad (2)$$

Here,  $\mathbf{u} = (u, v, w)$  is the spatially filtered velocity field,  $Re = 7500$  is the Reynolds number based on  $U$  and  $\delta$ , and  $\nu_t$  is the modeled subgrid-scale (SGS) eddy viscosity.

## 2.1 Reynolds-Averaged Navier-Stokes (RANS) Model

In the present work, the flow is modeled using the Reynolds-Averaged Navier-Stokes (RANS) approach, which solves for the mean flow quantities while modeling the effects of turbulence through additional equations. The incompressible RANS equations are:

$$\nabla \cdot \mathbf{u} = 0 \quad (3)$$

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u} = -\frac{1}{\rho} \nabla p + \nabla \cdot [(\nu + \nu_t)(\nabla \mathbf{u} + \nabla \mathbf{u}^T)] \quad (4)$$

Here,  $\mathbf{u}$  is the velocity vector averaged in time,  $p$  is pressure,  $\nu$  is the molecular kinematic viscosity and  $\nu_t$  is the turbulent viscosity (eddy).

## 2.2 Turbulence Model: Standard $k$ - $\varepsilon$

To close the RANS equations, the standard  $k$ - $\varepsilon$  turbulence model is used. It introduces two additional transport equations—one for the turbulent kinetic energy  $k$ , and one for its dissipation rate  $\varepsilon$ :

$$\frac{\partial k}{\partial t} + \mathbf{u} \cdot \nabla k = \nabla \cdot \left[ \left( \nu + \frac{\nu_t}{\sigma_k} \right) \nabla k \right] + P_k - \varepsilon \quad (5)$$

$$\frac{\partial \varepsilon}{\partial t} + \mathbf{u} \cdot \nabla \varepsilon = \nabla \cdot \left[ \left( \nu + \frac{\nu_t}{\sigma_\varepsilon} \right) \nabla \varepsilon \right] + C_{1\varepsilon} \frac{\varepsilon}{k} P_k - C_{2\varepsilon} \frac{\varepsilon^2}{k} \quad (6)$$

The eddy viscosity  $\nu_t$  is given by:

$$\nu_t = C_\mu \frac{k^2}{\varepsilon} \quad (7)$$

The standard constants used are:

$$C_\mu = 0.09, \quad C_{1\varepsilon} = 1.44, \quad C_{2\varepsilon} = 1.92, \quad \sigma_k = 1.0, \quad \sigma_\varepsilon = 1.3$$

## 2.3 Estimation of Turbulence Quantities at Inlet

For the calculation of the the turbulence intensity, turbulent kinetic energy and turbulent dissipation rate following empirical formulae were used :

$$I = 0.16 \cdot Re^{-1/8} \quad (8)$$

**Turbulent Kinetic Energy:**

$$k = \frac{3}{2}(U \cdot I)^2 \quad (9)$$

**Turbulent Dissipation Rate:**

$$\varepsilon = 0.164 \cdot \frac{k^{3/2}}{0.07L} \quad (10)$$

Here,  $U$  is the bulk velocity at the inlet, and  $Re$  is the Reynolds number.

## 3.Simulation Procedure

### 3.1 Geometry and Mesh Generation

The computational domain consists of a rectangular T-junction geometry that represents the interaction between a cross-flow and a incoming jet flow. It was constructed using OpenFOAM's `blockMesh` utility, employing quadrilateral blocks with structured meshing. All coordinates were defined in decimeters with a scaling factor of 0.1 via `convertToMeters`.

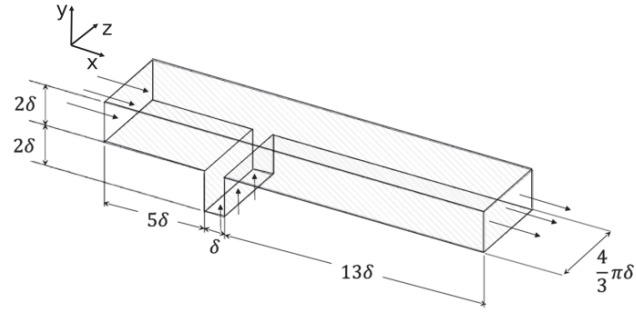


Figure 1: 3D geometry of the given paper T-junction used in the simulation.

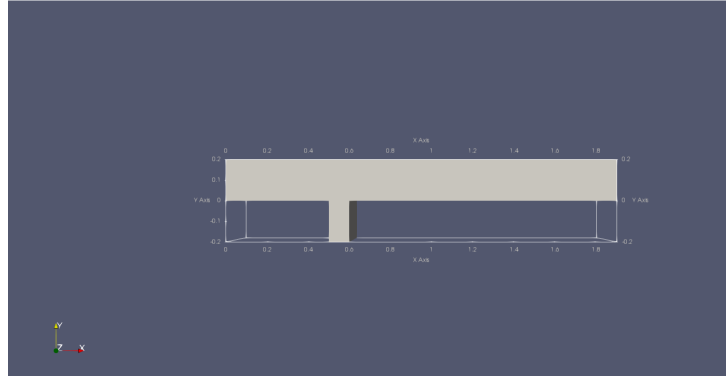


Figure 2: Geometry in paraFoam.

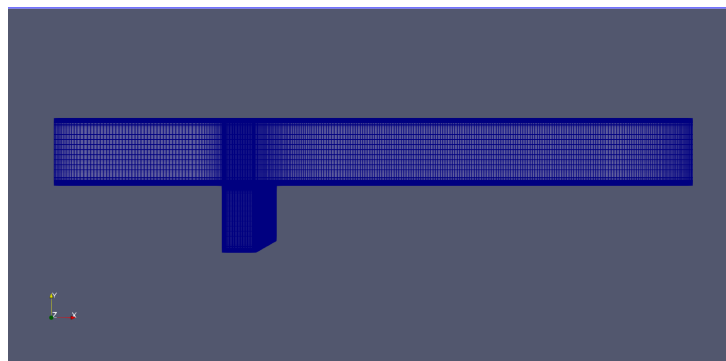


Figure 3: Mesh visualization: overall domain with graded cells.

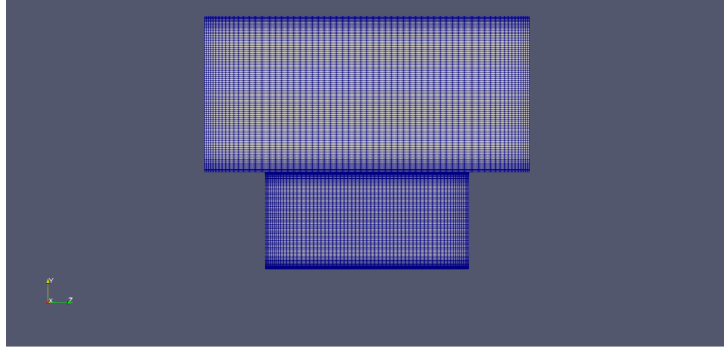


Figure 4: Zoomed-in view near the jet entrance.

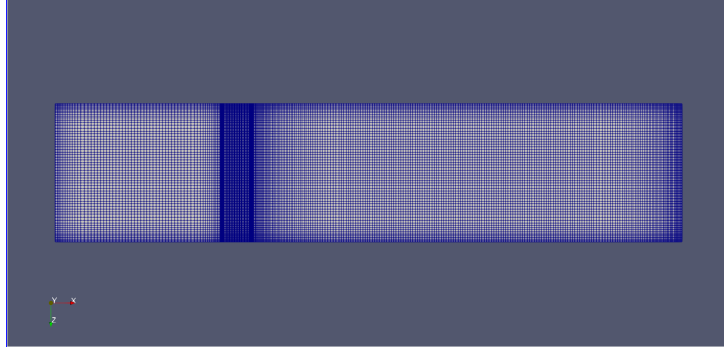


Figure 5: Cross-sectional mesh view showing structured layout.

### 3.1.1 Domain Decomposition and Blocks

The domain was split into four hexahedral blocks:

- Block 1: Main channel upstream section
- Block 2: Side jet inlet region
- Block 3: Main channel downstream section
- Block 4: Jet interaction zone (center region)

The blocks were defined using 20 vertices and meshed using `simpleGrading` to cluster cells near walls and jet interfaces.

### 3.1.2 Mesh Resolution and Grading Strategy

Each block was meshed with a structured grid. The grading in all three directions ( $x$ ,  $y$ ,  $z$ ) follows a three-zone pattern:

- 20% domain length with 25% of cells (refined)
- 60% domain length with 50% of cells (uniform)
- 20% domain length with 25% of cells (refined)

## 3.2 Boundary Conditions

The boundaries were defined in the `blockMeshDict` as follows:

- **inlet\_\_1**: Crossflow channel inlet
- **inlet\_\_2**: Side jet inlet
- **outlet**: Downstream outlet
- **fixedWalls**: All solid boundaries (walls, top, and bottom)

### 3.2.1 Boundary Conditions for SIMPLEFoam Simulation

The work in this paper was carried out for the bulk Reynolds number 7500 for both cross-flow and incoming jet.

Patch	Variable	BC Type	Value / Description
inlet__1	$k$	<i>fixedValue</i>	$0.00522 \text{ m}^2/\text{s}^2$
	$\varepsilon$	<i>fixedValue</i>	0.008842
	$p$	<i>zeroGradient</i>	-
	$\vec{U}$	<i>fixedValue</i>	$1.125 \text{ m/s}$
inlet__2	$k$	<i>fixedValue</i>	$0.02089 \text{ m}^2/\text{s}^2$
	$\varepsilon$	<i>fixedValue</i>	0.14147
	$p$	<i>zeroGradient</i>	-
	$\vec{U}$	<i>fixedValue</i>	$2.25 \text{ m/s}$
outlet	$k$	<i>zeroGradient</i>	-
	$\varepsilon$	<i>zeroGradient</i>	-
	$p$	<i>fixedValue</i>	0
	$\vec{U}$	<i>pressureInletOutletVelocity</i>	0
fixedWalls	$k$	<i>kqRWallFunction</i>	0
	$\varepsilon$	<i>epsilonWallFunction</i>	0.14147
	$p$	<i>zeroGradient</i>	-
	$\vec{U}$	<i>noSlip</i>	-

Table 1: Boundary conditions for turbulence, pressure, and velocity fields

### 3.3 Solver Configuration

The steady-state incompressible RANS solver `simpleFoam` was selected for this T-junction flow simulation, implementing the standard  $k-\varepsilon$  turbulence model. SIMPLE algorithm stands for Semi-Implicit Method for Pressure Linked Equations and it is a steady state turbulence solver.

Parameter	Value
<code>application</code>	<code>simpleFoam</code>
<code>startFrom</code>	<code>startTime</code>
<code>endTime</code>	2000
<code>deltaT</code>	1
<code>writeInterval</code>	100
<code>purgeWrite</code>	0
<code>runTimeModifiable</code>	yes

Table 2: Key control parameters for the steady-state simulation

## 4. Results and Discussions

The present study investigates turbulent flow in a rectangular T-junction using a Reynolds-Averaged Navier-Stokes (RANS) approach with the  $k-\varepsilon$  turbulence model in OpenFOAM. For comparison, a reference Large Eddy Simulation (LES) study is used to assess the accuracy of RANS in capturing complex flow features. The accuracy of simulation has been validated by comparing the variation of non dimensional axial velocity with non dimensional vertical distance. At the entrance location velocity has been non dimensionalized with respect to the inlet 1 cross-flow velocity. At the 'T' junction it has been non dimensionalized with square mean velocity and rest at other locations velocity has non dimensionalized with respected to the average velocity. At some locations the comparison results are matching to reasonable extent but at some locations there has been slight deviations due to different models. But at these locations also the trend is showing reasonable agreement with the paper results. Simulations are performed at MR(momentum ratio) 2.

### 4.1 Simulation Results

#### 4.1.2 Velocity Profile

Solution was converged in 572 iterations. Following are velocity profile for the converged simulation.

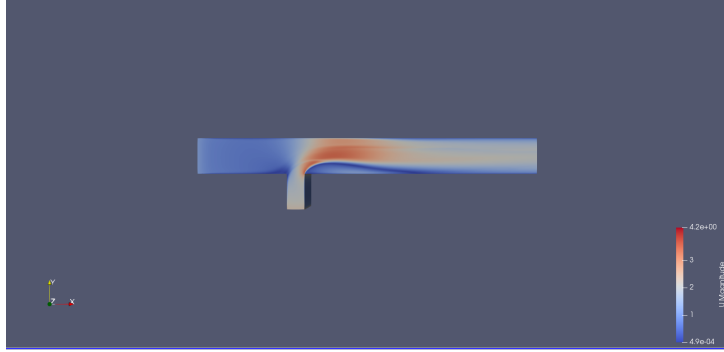


Figure 6: Velocity contours

Velocity streamline plot is reasonable matching with the paper.

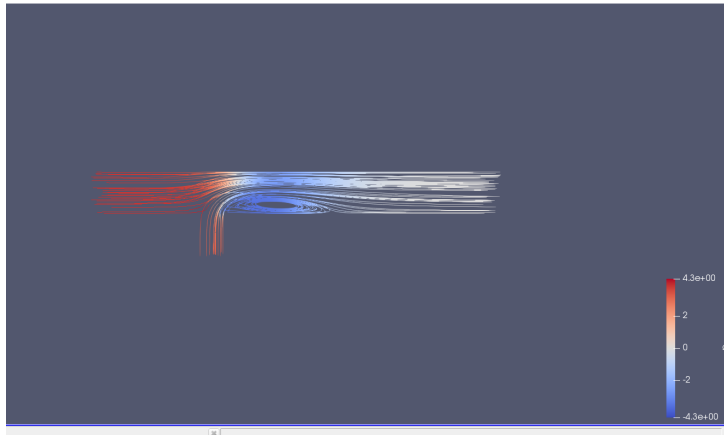


Figure 7: Velocity contours(simulation)

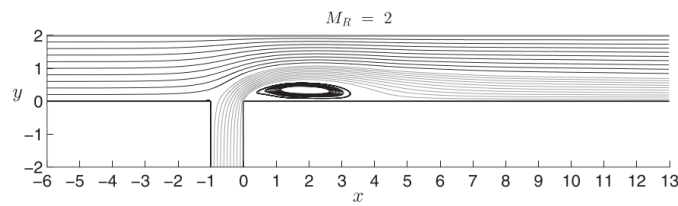


Figure 8: Velocity contours(paper)

## 4.2 Velocity Profile Analysis

Following plots shows the comparison between simulation and paper:



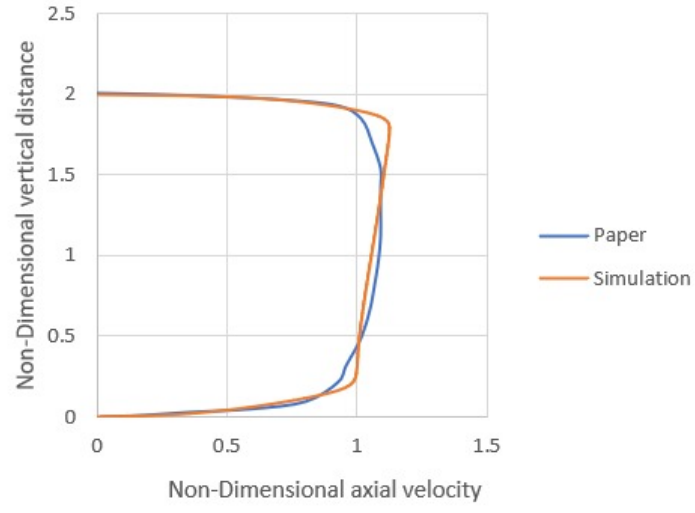


Figure 9: Downstream profile at  $\bar{x} = 0.371$

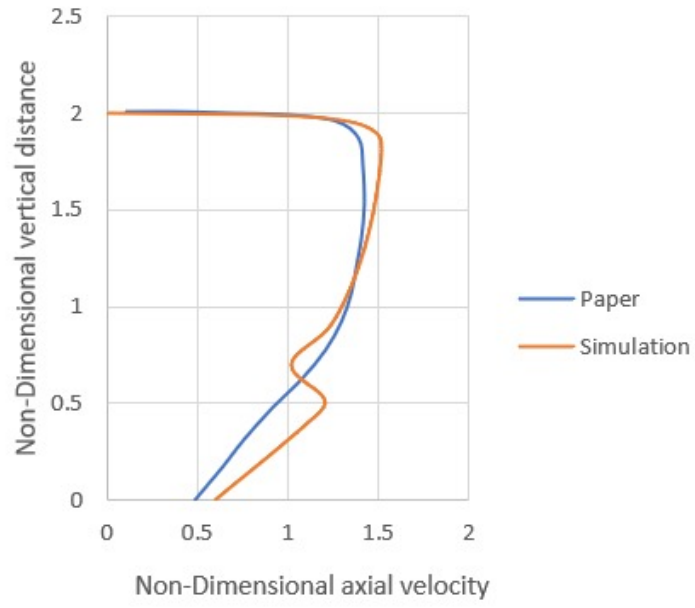


Figure 10: Downstream profile at  $\bar{x} = 0.55$

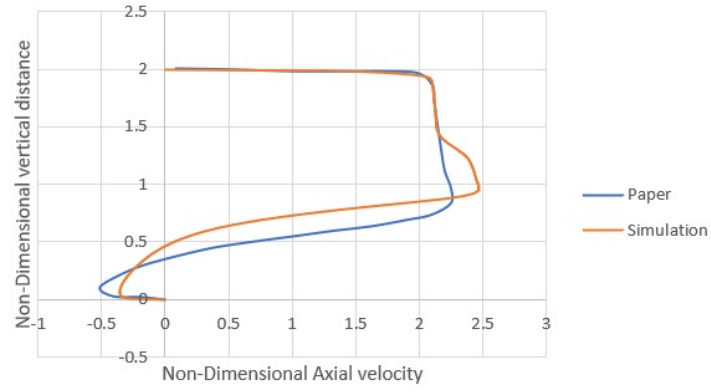


Figure 11: Downstream profile at  $\bar{x} = 0.761$

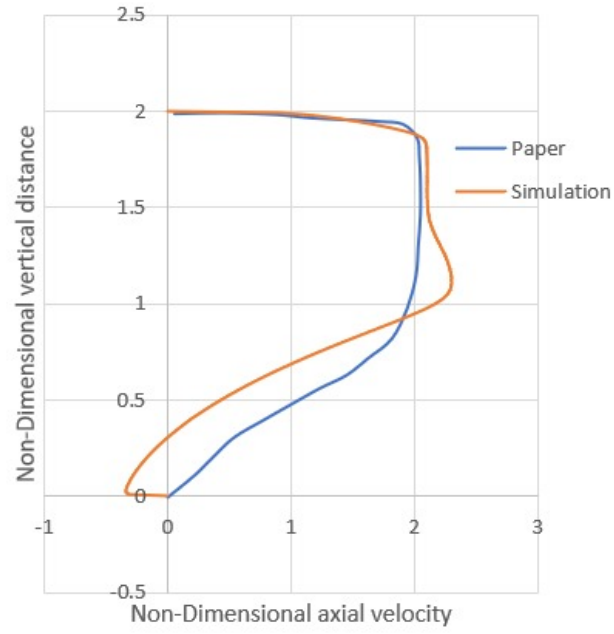


Figure 12: Downstream profile at  $\bar{x} = 1.012$

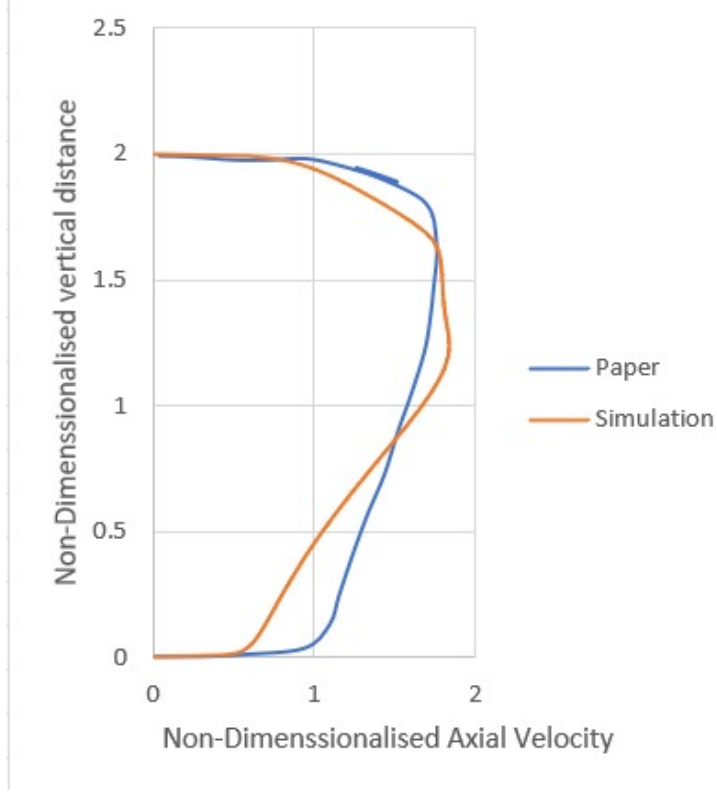


Figure 13: Downstream profile at  $\bar{x} = 1.486$

## 4.3 Main Findings

### Relative Error

The relative percentage error (RPE) between simulation and paper results is calculated as:

$$\text{RPE} = \left( \frac{\text{sim} - \text{paper}}{\text{paper}} \right) \times 100$$

All values are taken at the mid-point of the cross-section at various non-dimensionalized axial positions  $\bar{x} = \frac{x}{L}$ .

The results which are compared at momentum ratio of 2. Momentum ratio is the ratio of momentum flux of crossflow and incoming jet. From the above plots it is clear that the results are reasonably following the same trend as that of the paper's. The results are also showing an acceptable match with the paper except at some locations but the trend is almost following the paper.

Case	$\bar{x}$	Simulation (sim)	Paper	RPE (%)
1	0.371	1.0585	1.0966	-3.47%
2	0.550	1.3250	1.2740	+4.00%
3	0.761	2.4540	2.2360	+9.77%
4	1.012	2.1620	1.9490	+10.94%
5	1.486	1.6810	1.5740	+6.80%

Table 3: Relative Percentage Error at Mid-Point of Cross-Section for Various Non-Dimensionalized Axial Positions

The main reason behind these small deviations are the different turbulence models used in paper and in simulation. From the velocity contour plot it can be observed that the flow characteristics are perfectly captured.

## References

- [1] Liu, H., & Pletcher, R. H. (2017). Numerical study of turbulent flow in a rectangular T-junction. *Physics of Fluids*, **29**(6), 065106. <https://pubs.aip.org/aip/pof/article/29/6/065106/908323/Numerical-study-of-turbulent-flow-in-a-rectangular>
- [2] Spoken Tutorial Project, IIT Bombay. (Year). Simulation of a 2D turbulent flow in a channel using OpenFOAM [Tutorial]. <https://spoken-tutorial.org/media/videos/141/1514/resources/Simulation-a-2D-Turbulent-Flow-in-a-Channel-using-OpenFOAM-Additionalmaterial.pdf>

**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.