

Turbulent Flow Modelling of a Jet in a Cross-Flow Stream: Model and App

M. Tabatabaian

Department of Mechanical Engineering, British Columbia Institute of Technology, School of Energy,
3700 Willingdon Ave., Burnaby, BC, Canada

Keywords: Turbulent Flow, Jet Flow, Cross Flow, CFD, $k - \varepsilon$ Model, Education-Aid App, COMSOL®.

Abstract: The flow of a jet stream entering a cross flow main stream is modelled along with introducing the related App. The turbulent flow details are computed and validated against similar computational and experimental results. For example, dimensionless streamwise and cross-streamwise velocity components, dimensionless turbulent kinetic energy, and dissipation are shown in colour-expression figures, along with streamlines near the jet entrance to the main stream flow. The results agree with those from the existing references. We used a custom designed structured mesh for all models. Also, for democratization of the model applications a COMSOL® model-based App is built and introduced^a.

1 INTRODUCTION

The topic of turbulent flow modelling of a crossflow jet is of interest, both for understanding of the flow detail and industrial applications. This type of flow occurs in many industrial and natural applications. For instance, during landing and take-off of airplanes, gas turbines, fuel injections, oil refinery piping, waste discharge into reservoirs or atmosphere. In any of these flow types turbulence occurs and enhances mixing or, in general, the exchange of mass and energy between the jet and the mainstream flows. With the advances of computational fluid dynamics (CFD) and practical availability of commercial CFD software tools more industries are using models of turbulent flows in their design process. COMSOL® is a Multiphysics software package including CFD facilities and tools for building license-independent App's. In this paper we model the flow of a jet stream which is injected into another main stream flow, using COMSOL. Among the volume of research work performed on this topic, we selected the work of Karvinen & Ahlstedt (Rodi, 2005) who modelled the experimental work of Özcan & Larsen (Report MEK-FM, 2001-02), done for a similar setting. Former authors used several turbulence models (i.e., $k-\varepsilon$,

versions of $k-\omega$, versions of low-Re $k-\varepsilon$, and Reynolds stress) for the purpose of comparison and validation with those experimental results of the latter. Their main conclusion was that the $k-\varepsilon$ model results matched the experimental results better. We use the above-mentioned references for validation of our modelling results, using the $k - \varepsilon$ model with a custom-designed structured mesh, after performing mesh sensitivity analysis. The model run on a workstation laptop with 16GB RAM, and a 2-core CPU. Step-by-step instructions for building the $k - \varepsilon$ model is given in (Tabatabaian, 2015).

2 FLOW DOMAIN GEOMETRY AND DATA

The flow domain consists of a main 2D channel and a perpendicular jet tube. The main channel and jet flow domains are schematically shown in Figure 1 below, with having the symmetry line in the middle of the main stream channel. The centre of coordinate system is located at the centre-line of the jet pipe with the $x = 0$ axis located at the bottom of the main stream channel. The main stream cross-flow enters

^a Materials in this article and the model/App are used with permission from Mercury Learning and Information, © 2015 and 2017. All rights reserved.

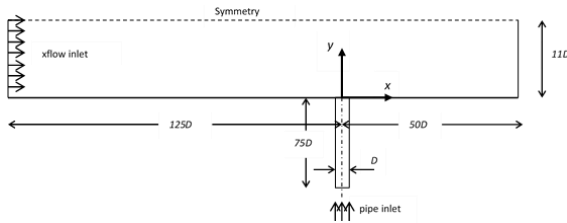


Figure 1: Schematic of cross-flow and jet flow domain geometry (not-to-scale).

left into the domain and jet flow enters through the pipe moving upwards in the positive y -direction. All dimensions are given as multiples of jet pipe diameter, $D = 24 \text{ mm}$. Hence; upstream inlet is at $125D$, downstream exit at $50D$, jet pipe length $75D$, and channel height $11D$.

The fluid properties (e.g. air) and model input data are given in Table 1. Two parameters R and $fact$ are introduced for having more options for parametric analysis when modelling the flow, COMSOL allows parametric analysis on any defined parameter in the model by sweeping through the range of assigned values and calculating the corresponding results. Therefore, it is not required to run the model separately for each value of a selected parameter. The turbulent intensity values are those of Karvinen & Ahlstedt (Rodi, 2005) and are defined as root-mean-square of turbulent velocity component scaled by its corresponding average. For example, $\sqrt{u_i'^2/\bar{u}_i}$ for component u_i' , ($i = 1, 2$).

Table 1: Fluid properties and model input data parameters.

Name/symbol	Value	Description
D	24 mm	Jet diameter
R	1.3 and 3.3	Ratio of jet over free-stream flow velocities
U_∞	1.5 m/s	Free stream velocity
U_j	$R * U_\infty$	Jet bulk inlet velocity
U_x	1.435 m/s	Bulk x-flow inlet velocity
ρ	1.225 kg/m ³	Air density
μ	$(1.7894 \times 10^{-5})fact$ kg/m.s	Air dynamic viscosity
$fact$	1	Viscosity multiplier
$linx$	0.025	x-flow turbulence intensity
$linp$	0.052	Jet flow turbulence intensity

Reynolds number based on free stream velocity and jet diameter is $Re = DU_\infty \left(\frac{\rho}{\mu}\right) = 2465$. Reynolds number based on the distance from the mainstream

entrance to the jet axis is $Re_x = 125Re = 308125$ and Reynolds number based on the jet bulk velocity and its diameter is $Re_j = R * Re$ which gives 3204 and 8134 for $R = 1.3$ and 3.3 , respectively. These Reynolds numbers are consistent with those given by the above-mentioned references.

3 TURBULENT FLOW MATHEMATICAL MODEL

The mathematical model used is the standard *realizable* k - ε model. This model is categorized as the two-equation RANS type with the Reynolds-stress closure method of Boussinesq's hypothesis (i.e. Reynolds stresses are proportional to the average turbulent velocity gradients). The turbulent variables are turbulent kinetic energy per unit mass, k (proportional to the trace of Reynolds stress tensor) and turbulent energy dissipation rate per unit mass, ε . The turbulent scales are then \sqrt{k} for velocity and $\sqrt{k^3}/\varepsilon$ for length which their product yields the turbulent kinematic viscosity ν_T proportional to k^2/ε . See (Tabatabaian, 2015) for more details.

The k - ε model consists of a system of six PDEs for average velocity \bar{u}_i , pressure \bar{p} , and two turbulent flow related quantities, k and ε as given by Equations (1)-(4).

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0 \quad (1)$$

$$\rho \left(\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} \right) = - \frac{\partial \bar{p}^*}{\partial x_i} + (\mu + \mu_T) \frac{\partial^2 \bar{u}_i}{\partial x_j \partial x_j} \quad (2)$$

$$\rho \left(\frac{\partial k}{\partial t} + \bar{u}_j \frac{\partial k}{\partial x_j} \right) = \frac{\partial}{\partial x_j} \left(\left(\mu + \frac{\mu_T}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right) + \bar{P}_k - \rho \varepsilon \quad (3)$$

$$\rho \left(\frac{\partial \varepsilon}{\partial t} + \bar{u}_j \frac{\partial \varepsilon}{\partial x_j} \right) = \frac{\partial}{\partial x_j} \left(\left(\mu + \frac{\mu_T}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right) + C_{\varepsilon 1} \frac{\varepsilon}{k} \bar{P}_k - C_{\varepsilon 2} \rho \frac{\varepsilon^2}{k} \quad (4)$$

Where $\bar{p}^* = \bar{p} + \frac{2}{3} \rho k$, $\bar{P}_k = \mu_T \frac{\partial \bar{u}_i}{\partial x_j} (\bar{u}_{i,j} + \bar{u}_{j,i})$, $\mu_T = \rho C_\mu \frac{k^2}{\varepsilon}$, $C_\mu = 0.09$, $C_{\varepsilon 1} = 1.44$, $C_{\varepsilon 2} = 1.92$, $\sigma_k = 1.0$, $\sigma_\varepsilon = 1.3$, $\bar{p}^* = \bar{p} + \frac{2}{3} \rho k$, and dynamic eddy viscosity $\mu_T = \rho \nu_T$.

For pressure reference point the outlet of the cross-flow is set to be at zero and the turbulent intensities for cross-flow and jet flow at their inlets are 2.5% and 5.2%, respectively. Also, the turbulent

length scales are set to be 5 mm for the cross-flow and $\frac{0.07D}{2} = 0.84 \text{ mm}$ for the jet flow.

4 RESULTS AND VALIDATION

We performed a mesh sensitivity analysis using four mesh settings, as shown in Table 2. Consequently, we then used a structured mesh consisting of 133120 elements for modelling. The results for dimensionless turbulent quantities; average velocity components (\bar{u}/U_{∞} , \bar{v}/U_{∞}), kinetic energy (k/U_{∞}^2), and energy dissipation ($\epsilon D/U_{\infty}^3$) are presented. These values are computed for two cases; $R = 1.3$ and 3.3 . For each case the variations of each quantity are presented at several locations along the main stream channel. These locations ($\frac{x}{D} = -0.67, 0.67, 1.5, 3, 3.67$) are selected such that the model results could be compared with those from the references for validation, see Table 3.

The results agree, with acceptable accuracy, with those of existing experimental and numerical data, see Figures 4-6 in (Rodi, 2005). In addition, the model produces maps of streamlines, velocity magnitude, and pressure contours as shown in Table 4, for $R = 3.3$. As shown, the recirculating zone located at downstream of the jet is captured with good accuracy. The length and depth of the recirculating zone is of interest for industrial and environmental

Table 2: Results used for mesh sensitivity analysis.

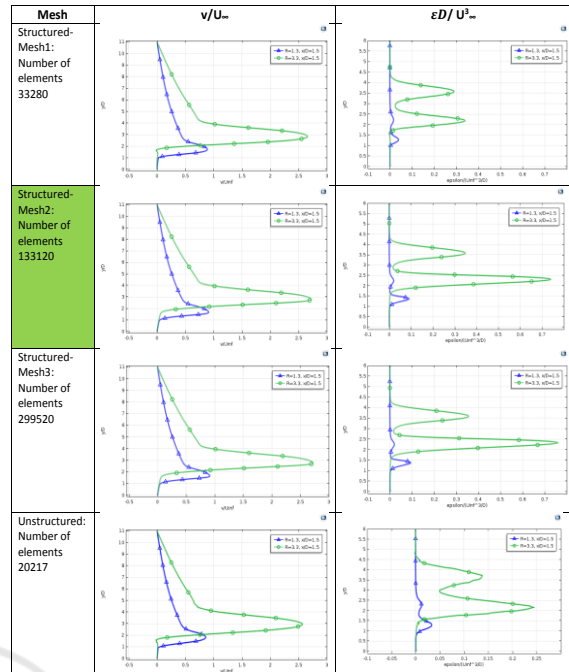
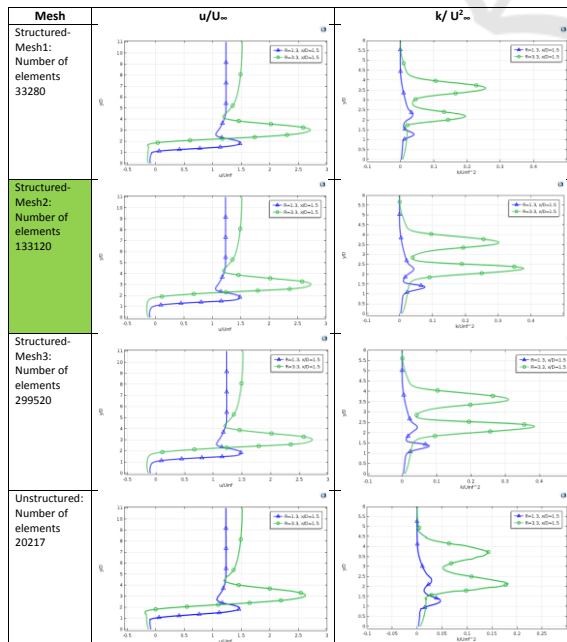
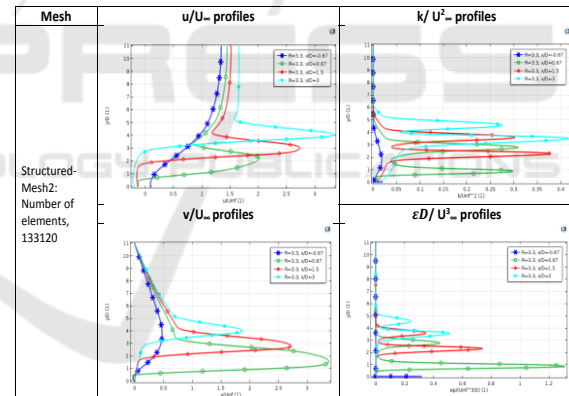


Table 3: Model results for dimensionless turbulent quantities.



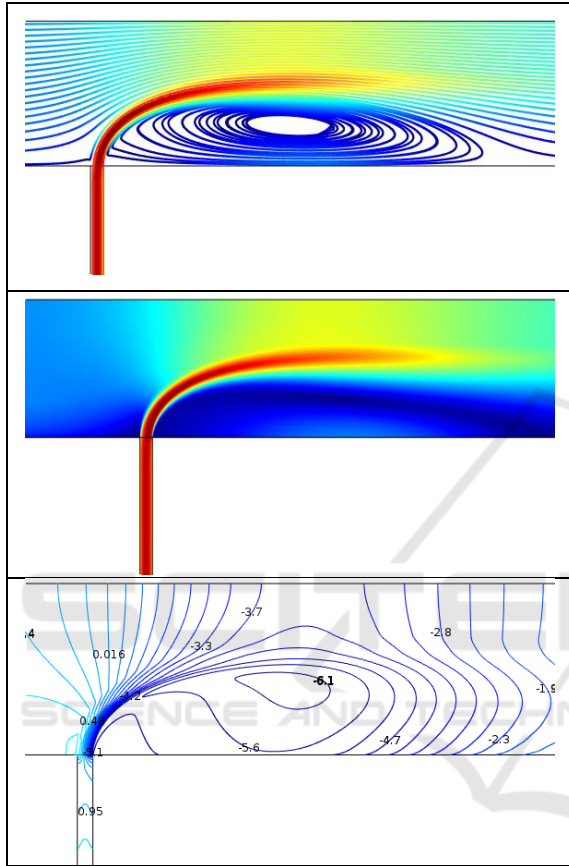
applications. If required, the model results can be used for generating the dimensions and characteristics of the velocity field map, pressure contours, and recirculating zone streamlines with higher resolution.

5 MODELLING APP

For practical applications, it is preferred having a tool besides CFD model which can be used by designers and engineers. For this purpose, we built an App using COMSOL tools. This App is based on the model developed and can run independent of the model. The latest version of the software (i.e., version

5.4) enables users building a license-independent App as *.exe type file using a COMSOL model. Hence, one doesn't require the software license for App execution.

Table 4: Model results; streamlines, velocity magnitude, and pressure contours maps close to the jet inflow.



The App presented here provides users modelling facilities to study and model a jet flow entering a main cross stream manipulating input data. A screen capture of the App's interface is shown in Figure 2.

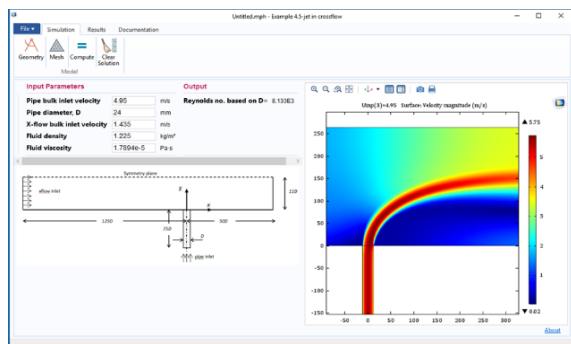


Figure 2: App user interface.

As shown, the interface has three tabs; *Simulation*, *Results*, and *Documentation*. The data can be entered under the *Input Parameters* section. The resulted Reynolds number appears in the *Output* section.

The Graphics window shows Geometry, Mesh, and a series of modelling results, when desired. A sketch of the modelling domain geometry is shown in the interface in order to help users for identifying input data and parameters. By clicking on the Simulation Tab, four buttons appear in the ribbon bar, as shown in Figure 3. By clicking on the *Geometry* button, the latest geometry of the model appears in the Graphics window. Similarly, the *Mesh* button would show the structured mesh build in the flow domain. The *Compute* button, when clicked on, would run the model. A green progressive bar appears on the bottom right-hand corner of the Interface window, indicating the calculations. Once the calculation is complete a default velocity plot appears in the Graphics window.

By clicking on the *Results* Tab, seven buttons appear in the ribbon bar, as shown in Figure 4. The label for each button describes the type of results plotted in the Graphics window, when that button is clicked on. By clicking on the *Documentation* Tab, two buttons appear in the ribbon bar, as shown in Figure 5. By clicking on these buttons, a report is generated. Two formats are provided; HTML and Word™. The App provides the option for saving the generated Report with the desired file name.

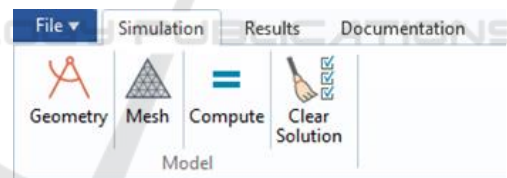


Figure 3: App Simulation Tab.

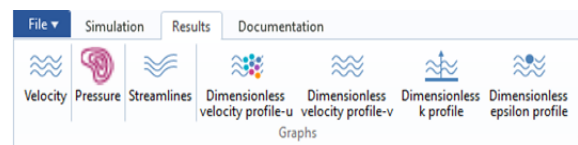


Figure 4: App Results Tab.

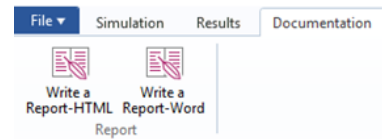


Figure 5: App Documentation Tab.

6 CONCLUSION

A model and an App are built for analysing and designing the turbulent flow of a jet entering a cross-flow main stream, using COMSOL. The model results are compatible within acceptable accuracy with the cited references existing results, considering that the model is built in a 2D domain and the existing experimental results are in 3D. The App is built to help with related practical applications of the model and facilitates parametric analysis of the flow behaviour.

REFERENCES

- Ozcan,O. and Larsen,P.S, Report MEK-FM 2001-02. *An Experimental study of a turbulent jet in cross-flow by using LDA*, s.l.: Technical University of Denmark.
- Rodi, W. (., 2005. Comparison of Turbulence Models in Case of Jet in Crossflow using Commercial CFD Code. In: *Engineering Turbulence Modelling and Experiments 6: ERCOFTAC International Symposium on Engineering Turbulence and Measurements - ETMM6*. s.l.:Elsevier, pp. 399-408.
- Tabatabaian, M., 2015. *CFD Module: Turbulent Flow Modeling and Turbulent Jet stream in a cross flow COMSOL model and App.* s.l.:Mercury Learning and Information.

