CFD Modelling of Erosion in Slurry Tee-Junction

Edward Yap

Jeremy Leggoe
Department of Chemical Engineering
The University of Western Australia

David Whyte
CEED Client: Alumina Centre of Excellence, Alcoa of Australia

Abstract

The project CFD Modelling of Erosion in Slurry Tee-Junctions aims to determine the feasibility and workflow for integrating OpenFOAM into Alcoa's existing CFD practices. OpenFOAM is a licensed free computational fluid dynamics (CFD) software giving the user freedom to freely run simulations with the limiting factor being the number of cores available in the user's machine. The successful implementation of OpenFOAM will decrease the operational costs requried to produce refinery solutions through CFD modelling. For the purposes of modelling, a blanked pipe tee-junction has been chosen. Blanked pipe tee-junctions are occassionally subject to a case of erosion which results from a vortex driling a hole through blanks during operation. The tee-junction has first been modeled in ANSYS CFX using different meshes and run using both the $k-\epsilon$ and $k-\omega$ SST turbulence model using the high-order advection scheme and coupled solving algorithm. With an initial workflow laid out, OpenFOAM has the potential to be a powerful tool to run multiple simulations simultaneously all with slight variations to see their effects without incurring high operational costs.

1. Introduction

Computaional fluid dynamics (CFD) utilises numerical analysis to solve and analyse problems involving fluid flows (Versteeg 2007). Typically CFD codes consist of a pre-processor, a solver and a post-processor (Versteeg 2007). Pre-processors generally define the inputs of the problem at hand, these inputs include; defining the geometry, grid generation, selection of the physical and chemical phenomena being modelled, fluid properties and appropriate boundary conditions (Versteeg 2007). Solvers utilise numerical methods to resolve the flow field and finally, post-processors provide graphical representations of the resolved flow field (Versteeg 2007).

The commericial ANSYS package houses tools for pre-processing, solving and post-processing. Within ANSYS the CFD solvers CFX and Fluent are available for annual license subsriptions. On the other hand, OpenFOAM is a free CFD code developed by OpenCFD and OpenFOAM Foundation supplying pre-processing, solving and post-processing utilities (OpenCFD 2017). It provides the user no limitation for parallel computing and results in no licensing fees.

1.1 Problem Identification and Project Objective

Erosion is an issue in Alcoa's refinery operations. It contributes to higher operating costs in the form of maintenace and production impacts. To better understand the phenomena causing erosion in equipment, Alcoa are making use of CFD simulations to model equipment subject to erosion. Presently, CFD modelling is simulated through the commerical ANSYS package.

A particular case of erosion that Alcoa is interested in, is vortex erosion in blanked pipe tee-junctions. Tee-junctions are used throughout refineries to redirect flow between potential pathways. A schematic diagram of a blanked pipe tee-junction indicated the point of vortex erosion may be seen in Figure 1. While erosion occurs in other sections of the tee-junction, erosion on the steel blank is highly unusual.

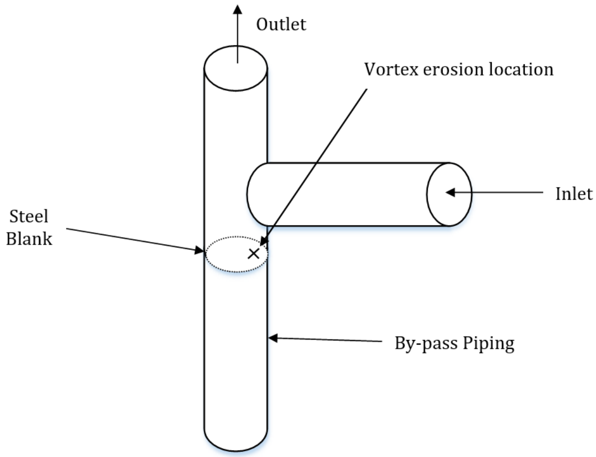

Figure 1 Blanked pipe tee-junction schematic diagram indicating location of vortex erosion

In order to better understand vortex erosion, previous experiments exploring the issue have been conducted by the CSIRO on behalf of Alcoa. This has provided Alcoa with laser doppler velocimetry (LDV) measurments, flow visualisations and paint tests showing vortex erosion.

The project CFD Modelling of Erosion in Slurry Tee-Junctions will recreate a blanked pipe tee-junction using ANSYS and simulate the flow field using CFX-19.0. The outputs will then be validated using CSIRO's LDV data. The effect of different mesh parameters, turbulence models and geometry configurations will also be explored.

Upon completion of the validation, the models will be replicated in OpenFOAM to be resimulated. The OpenFOAM outputs will then be compared to those outputted by ANSYS CFX to determine the feasibility of using OpenFOAM in a commerical environment. Determining the feasibility of OpenFOAM and developing a workflow to integrate the software into Alcoa's existing practices will decrease the operational costs for CFD modelling.

1.2 Past work

Brown's (2002) article "Erosion Prediction in Slurry Pipeline Tee-Junctions" discusses the issues of erosion occuring on steel blanks within tee-junctions in Alcoa's refinery. Brown (2002) utilises CFD modelling through ANSYS CFX 4.2 to model the flow field causing erosion. The model was run using a Eulerian-Eulerian continuum approach with the $k - \epsilon$ turbulence model and an enforced swirl velocity profile at the inlet. According to Brown (2002) a vortex is generated due to the inlet conditions of the tee-junction which is heavily influenced by the pipe's upstream conditions. This led to the conclusion that abrasive particles become entrained within the centre of the vortex which impact with the blank causing erosion (Brown 2002).

A substantial portion of the work exploring erosion in pipe tee-junctions is concerned with sand erosion in the oil and gas industry for multi-phase flow investigating gas as the primary phase. While there are common erosion locations such as around the bend of a tee-junction, vortex erosion does not appear to be present.

2. Methodology

The geometry of the blanked pipe tee-junction will be imported from previous CAD work created by the CSIRO with the orientation altered to reflect CSIRO's LDV measurements orientation. A schematic diagram of the the pipe tee-junction in the x-y axis showing orientation, dimensions, coordinates of the measured points (1, 2, 3, 4) and boundary conditions can be seen in **Figure**.

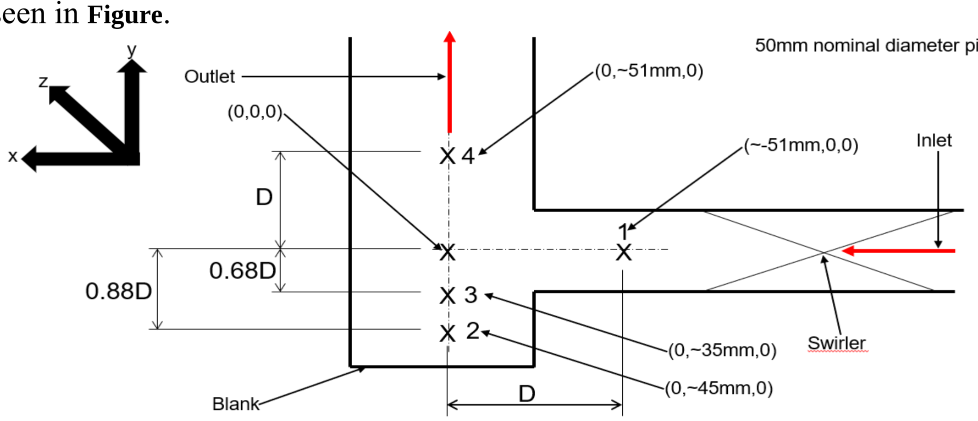

Figure 2 Schematic diagram of blanked pipe tee-junction indication boundary conditions, dimensions and positions of interest (1, 2, 3, 4)

The Tee-Junction has been extended sufficiently far downstream to ensure the flow reaches a fully developed state where no change occurs in the flow direction (Versteeg 2007). On the other hand, both extended and non extended upstream geometries have been used for testing. Varying CFX meshes will then be created to run simulations in ANSYS CFX 19.0 to determine the mesh conditions required to capture accurate trends. Each mesh will be run using the $k - \epsilon$ turbulence model as it has been found to work well with most industrial flows (Versteeg 2007). Each configuration will also be run using the $k - \omega$ SST with curvature correction turbulence model, as traditionally $k - \epsilon$ is not able to capture swirling flow. All combinations of mesh and geometries will first be run in steady state, the steady state outputs will then be used to initalise transient simulations run over 3 seconds. A run time of 3 seconds has been chosen as it allows for 7-8 residence times to pass during the run time, thus giving sufficient time for any phenomena to occur. Each simulation has been run at an input velocity of $3ms^{-1}$ as experimental data exists at this velocity. The simulations will be run in a single-phase using water at 25℃ at 1atm reference pressure with gravity switched off. CFX's high-resolution advection scheme and coupled solving algorithm has been chosen. A numerical convergence criteria of 1e-04 RMS will be used along with monitors observing variables such as imbalance, velocities at points of interest, and turbulence will be used to determine convergence. The transient average velocity values at lines 1, 2, 3 and 4 as shown in Figure 5 will then be exported and compared to CSIRO experimental data.

For OpenFOAM verification purposes the geometries and meshes built in ANSYS 19.0 will be imported into OpenFOAM for a direct comparison of the solvers. Both $k-\epsilon$ and $k-\omega$ SST turbuelnce models will be considered while advection schemes and solving algorithms will be experimented with as the coupled solving algorithm does not exist within OpenFOAM. In addition to a comparison of the solvers, a user guide will be created for the client outlining workflow, trustworthy sources, available guides and manuals.

3. Results and Discussion

3.1 CFX Results

Mesh A has an extended upstream fetch to replicate that used in the CSIRO experiments, an extended downstream to ensure that the flow is fully developed as it exits the model, and is comprised of 489446 elemements. Figure 3 shows the axial and swirl velocity profiles for mesh A at line 1 found from CFX simulations. As can be seen from the profiles both the $k-\epsilon$ and $k-\omega$ sst turbulence model accurately capture the velocity measurements found by CSIRO's experiments.

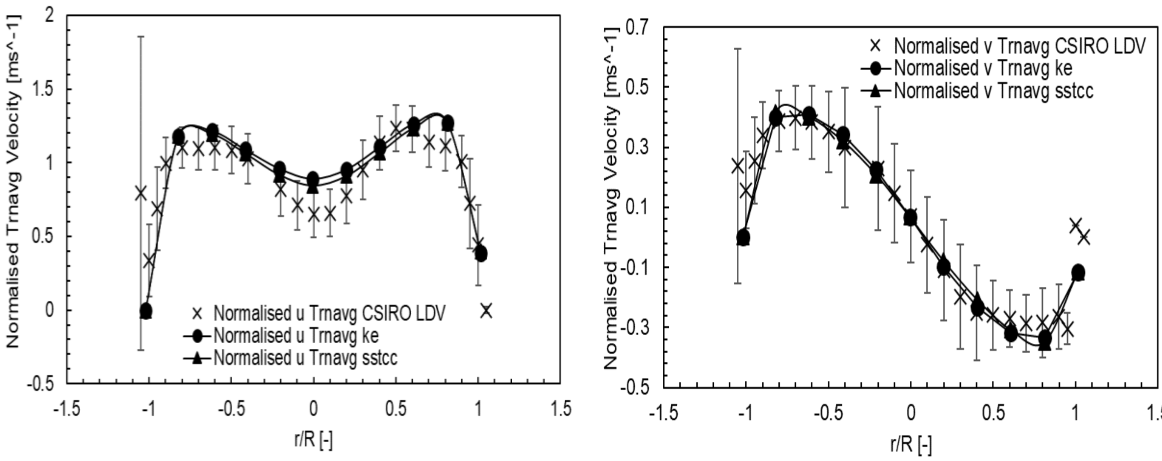

Figure 3 Axial (u) and swirl (v) velocity profile for mesh A at line 1

Figure 4 shows the swirl velocity profile for mesh A at line 2. As can be seen by the velocity profile neither turbulence model tested results in the swirling pattern seen in CSIRO's flow visualisation and velocity profile at line 2.

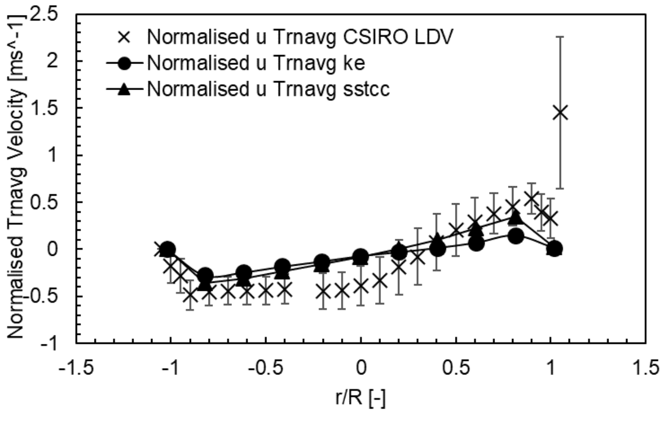

Figure 4 Swirl (u) velocity profile for mesh A at line 2

Figure 5 shows the axial and swirl velocity profile for mesh B at line 1. Mesh B does not have an extended upstream fetch and is comprised of 324222 elements. Variations of mesh B have been created, however, little difference has been found in the velocity profiles from the original mesh B. As can be seen in Figure 5 the trends captured are not as accurate as those shown in Figure 3.

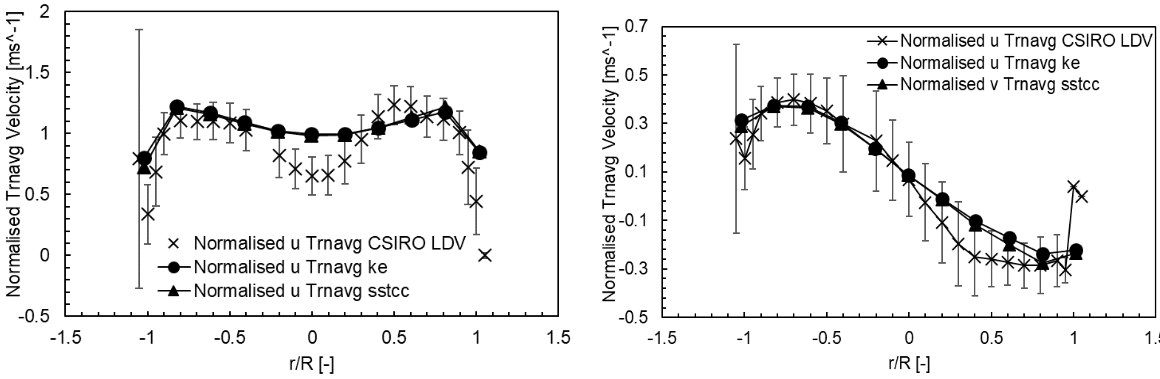

Figure 5 Axial (u) and swirl (v) velocity profile for mesh B at line 1

Figure 6 shows the swirl velocity profile for mesh B. Despite having larger discrepancies in the velocity profile when compared to those found by mesh A at line 1, mesh B indicates a swirl right above the blank at line 2. However, as can be seen in figure 6, it is swirling in the opposite direction to that found in the experiments.

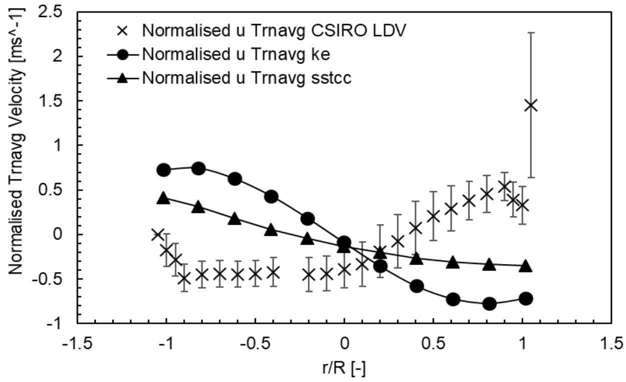

Figure 6 Swirl (u) velocity profile for mesh B at line 2

These results indicate that the model is highly sensitive to upstream conditions as noted by Brown (2002) in section 1.2. As can be seen from the simulations run with mesh B, despite $k-\epsilon$ being traditionally thought to be incapable of capturing swirling flow, with appropriate mesh selection, $k-\epsilon$ can predict a swirling profile.

3.2 OpenFOAM Workflow

The ANSYS DesignModeler and Mesh tool may be used to create geometries and mesh to export. Within the ANSYS meshing tool, mesh files may be exported into a single file. It is important that the mesh file be expored in ASCII format (which is not the default in Fluent) as OpenFOAM is incapable of reading binary text files. Using OpenFOAM's inbuilt mesh conversion script 'fluentMeshToFoam', Fluent meshes may be imported into OpenFOAM for simulation purposes. As the meshes were originally built for CFX, the meshes have been directly converted to Fluent meshes without any changes to their parameters.

3.2.1 Set up

OpenFOAM run directories are comprised of three main folders, these are '0', 'constant' and 'system'. The '0' folder contains the intial pressure and velocity conditions stored in text files.

In the case of turbulent simulations, transported turbulence properties such as turbulence kinetic energy and turbulence dissipation rate are stored in this file. In the case of the 'fluentMeshToFoam' script being utilised a 'polyMesh' folder is created within the 'constant' folder. This folder stores the imported mesh information. Opening the 'boundary' text file contained within outlines the boundary conditions imported from ANSYS. It is important that the initial properties stored in the '0' folder are altered to reflect the same boundary conditions as outlined in the 'boundary' text file. A 'transportProperties' text file contained within the 'constant' folder stores the viscosity of the fluid flowing through the domain. Finally the turbulence model may be specified within the 'turbulenceProperties' script file.The last folder stored in the run directory is the 'system' folder. This stores three main text files called 'controlDict', 'fvSchemes' and 'fvSolution'. Within the 'controlDict' file, variables such as the exact OpenFOAM solver, run times and time steps for tranisent simulations and number of iterations may be chosen. The 'fvSchemes' text file specicfies the advection scheme used for each term and finally the 'fvSolution' text file specifies the solving agorithm used, convergence criterion and relaxation factors.

3.2.2 Post-Processing

ParaView is OpenFOAM's post-processing program which may be directly compared with CFD-Post contained within ANSYS. Like CFD-Post and OpenFOAM, ParaView utilises a graphical user interface to allow for contours, streamlines, etc. to be examined. Velocity profiles may be exported as Comma Separated Variable (.csv) files to be used for direct comparison between the two solver's outputs.

4. Conclusions and Future Work

Despite the $k - \epsilon$ turbulence model being traditionally incapable of capturing swirling flow, it has captured a swirl velocity profile in mesh B. However, it is evident that the case is extremely sensitive to the upstream boundary conditions. Despite being difficult to initially setup OpenFOAM has the potential to be an extremely powerful tool. Once single case files have been created, multiple small variations may be made and simultaneously run without restrictions. With the workflow setup, future work will be focused on importing tee-junction models to be simulated and then exporting results to be compared to the ouputs produced by CFX.

5. Acknowledgements

The author would like to acknowledge the support and guidance provided by client mentor David Whyte and academic supervisor Jeremy Leggoe. The continued feedback and support has ensured that this project is continuing to grow and progress.

6. References

Brown, GJ 2002, 'Erosion Prediction in Slurry Pipeline Tee-Junctions', Applied Mathematical Modelling, vol. 26, no.2, pp. 155-170.
OpenCFD 2017, OpenFOAM User Guide v5.0, OpenFOAM Foundation
Versteeg, HK 2007, An Introduction to Computaional Fluid Dyanmics : The Finite Volume Method, 2nd ed, Pearson Education LTd., Harlow, England;.