CFD5 First Assignment

Charlie Seviour

November 5, 2010

Abstract

This is the first assignment in the Edinburgh University CFD 5 course, in which a 225mm 45 Degree Polysewer Equal Junction is simulated using Starccm+.

Contents

1 Interpretation of the Assignment Question 1

2 Methodology 1

2.1 Initial Design of the Grid 1

2.2 Grid Optimisation 2

2.2.1 Critical Analysis 3

3 Results 4

4 Analysis of Results and Conclusion 5

A Supporting Calculations 6

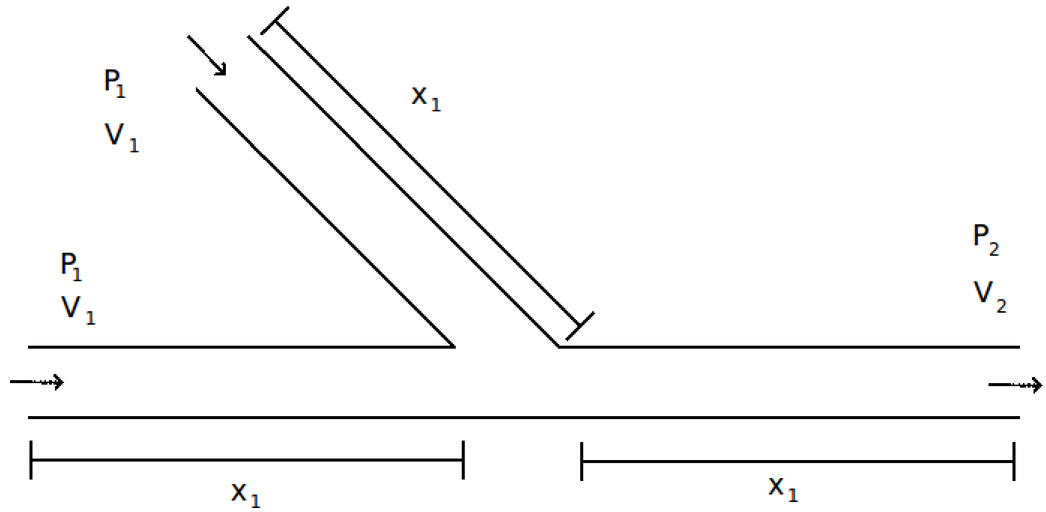

Figure 1: Plan view of the pipe geometry.

1 Interpretation of the Assignment Question

The aim of the assignment was to find the pressure loss in a 45 degree pipe junction, and the point where fully developed flow is restored. A number of definitions could be used for fully developed flow. One possible definition would be the point at which the flow reattaches to the wall after the junction. Alternatively one could assume that fully developed flow is at the point a Nikuradse velocity profile is obtained.

Knowing whether swirl is present is of particular interest in the cases of flow meters such as turbine of vortex meters. Therefore for the purposes of this assignment fully developed flow has been defined as the point where a Nikuradse profile is obtained.

The pipe arrangement given in the assignment has been sketched in figure 1. The fluid was assumed to be water with a dynamic viscosity, ν of $10^{-6} m^2/s$ and a density of $1000 kg/m^3$.

2 Methodology

Since it is easy to produce inaccurate solutions in CFD by using inappropriate solver settings or crude meshes, to begin with simple analytical calculations were made. Then an appropriate mesh was produced. A simulation was run using carefully selected settings. These settings are surmised in the appendix.

2.1 Initial Design of the Grid

The geometry of the pipe junction is simple. Since this was the case and the junction is completely symmetric along the centre plane the flow should be mostly 2D (2 dimensional). Using a 2D grid has the benefit that it requires far less computational time. However it is conceivable that vortex structures should form as shown in figure 2. As a matter of interest the final simulation was examined and it was found that the tangential velocity was negligible (see figure 3).

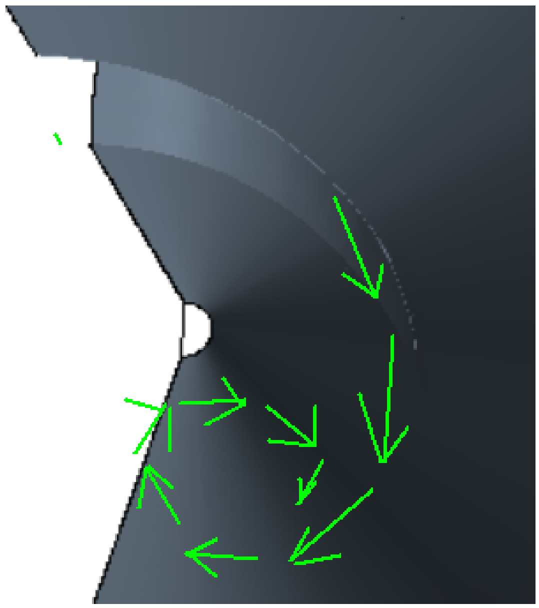

Figure 2: Possible 3D flow at the pipe junction.

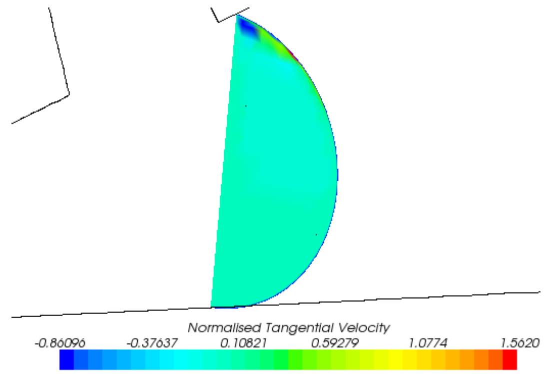

Figure 3: The tangential velocity was examined in Starccm+ using a custom scaler function, defined as the tangential velocity divided by the axial velocity. It can be seen that the tangential velocity is negligible.

In a 2D grid the walls would be represented by 2D lines rather than 3D surfaces. As a result care would have to be taken so that the 2D lines produce a frictional effect equivalent to what the 3D surfaces would produce.

Since it would be less accurate to describe the flow only in 2D and computational time was not a great constraint in this case, the author opted for a 3D grid.

2.2 Grid Optimisation

According to A. Bakker (2008) a high quality grid is needed to produce accurate solutions and fast convergence. In Starccm+ it is possible to examine various grid properties, such as skewness angle and aspect ratio. The various meshing functions in the grid generation part of the program could be assessed to quantitatively highlight which meshing model was most suitable. An example of one comparison made can be seen in figures 4 to 7.

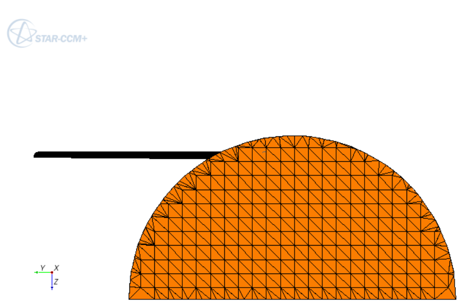

Figure 4: The mesh generated using the *extruder*, *prism layer mesher*, *surface wrapper* and *trimmer* model

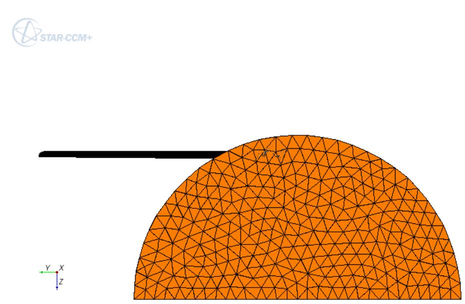

Figure 5: The mesh generated using the *generlized cylinder*, *polyhedral mesher*, *prism layer mesher* and *surface remesher* model

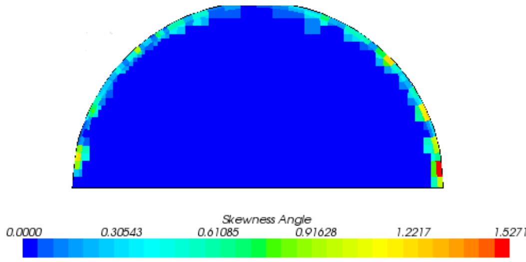

Figure 6: A scalar scene of the mesh skewness. It can be seen that the skewness angle is better in this case than in figure 7

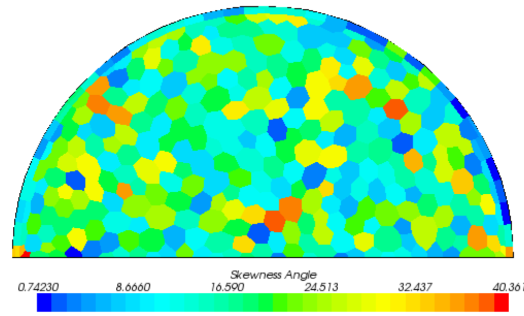

Figure 7: A scalar scene of the mesh skewness

Grid convergence was assessed using the Richardson extrapolation method (John W. Slater, 2008). The calculations can be found in the appendix.

2.2.1 Critical Analysis

Only necessary grid refinements were made. To reduce computational time the grid could have been coarsened in areas that are of little interest such as the inlets and outlet. Starccm+ is good at automated grid generation, however using dedicated grid generation software it is possible to produce more bespoke meshes. An example in this context is an H-grid grid produced in gridgen. In the H-grid the skewness angle is kept to a minimum. See figure 8 for an illustration of this grid.

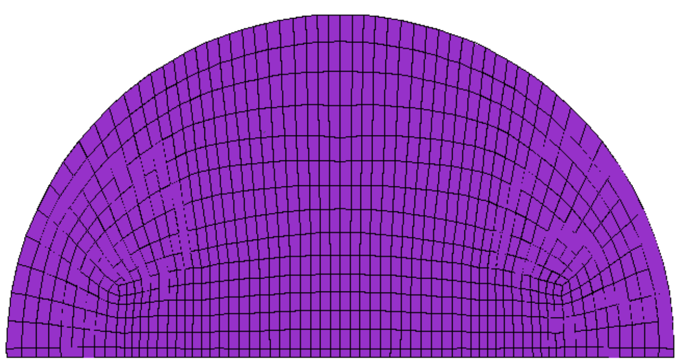

Figure 8: An H-grid produced using gridgen.

3 Results

The pressure loss and the distance till fully developed flow were found. The pressure loss has been defined as the difference between the inlet pressure and the pressure at the point in question. The pressure loss is presented in figure 9.

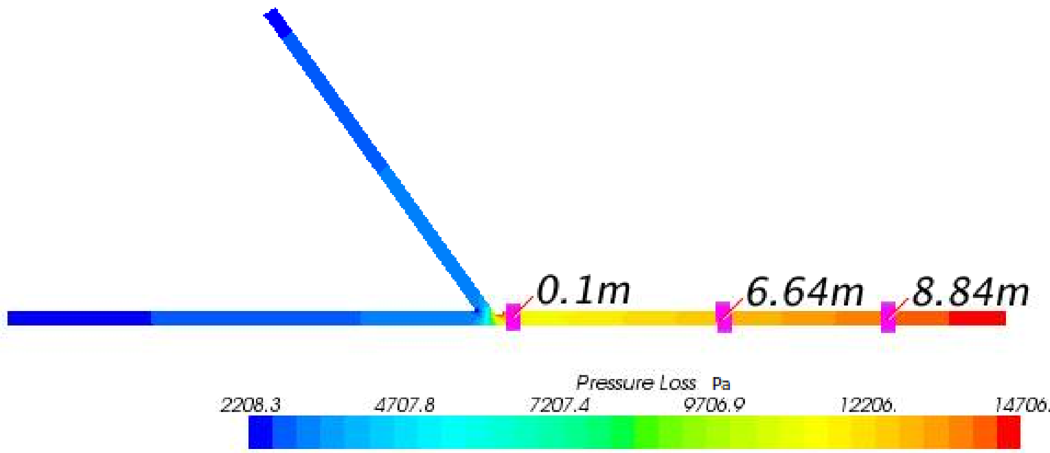

Figure 9: Pressure loss

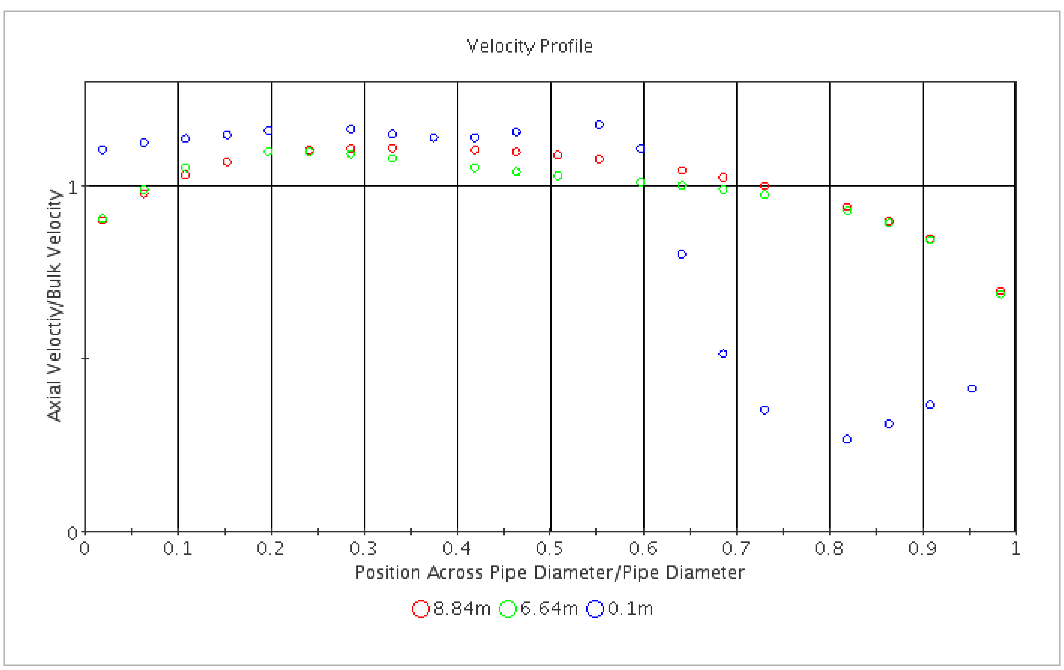

Figure 10: Velocity profiles for the probe lines as shown in figure 9. At 0.1m the velocity profile is greatly disturbed. At 6.64m the profile is almost symmetrical, after which there is little change. At 8.84m the flow can be generally assumed to be fully developed.

4 Analysis of Results and Conclusion

The flow has been determined as fully developed at 8.84m. In light of this it shall not be possible to accurately measure the flow rate at the intended site 6m from the junction. However one could, use correction factors to deal with this. Some flow meters can be corrected based on flow properties which can be calculated in Starccm (Mickan, 1996). In the final simulation at 6m downstream of the junction the pressure loss is approximately $10kPa$. However the pressure loss predicted by the analytical solution was $2kPa$. The reason for this is likely to be that the equivalent length method did not assume a sufficiently high loss for the junction for the Reynolds number in this assignment. Also the simulation may have been too numerically dissipative.

References

A. Bakker. The colorful fluid mixing gallery, 2008. URL `http://www.bakker.org/cfm`. Accessed 011110.

Engineering Tool Box. URL `http://www.engineeringtoolbox.com/equivalent-pipe-length-method-d_804.html`. Accessed 011110.

John W. Slater. Examining spatial (grid) convergence, 2008. URL `http://www.grc.nasa.gov/WWW/wind/valid/tutorial/spatconv.html`. Accessed 011110.

Bodo Mickan. Systematische analyse von installationseffekten sowie der effizienz von stromungsgleichrichtern in der grossgasmengenmessung. 1996.

List of Figures

1 Plan view of the pipe geometry 1

2 Possible 3D flow at the pipe junction. 2

3 Tangential velocity at pipe junction 2

4 The mesh generated using the *extruder*, *prism layer mesher*, *surface wrapper* and *trimmer* model 2

5 The mesh generated using the *generlized cylinder*, *polyhedral mesher*, *prism layer mesher* and *surface remesher* model 2

6 A scalar scene of the mesh skewness 3

7 A scalar scene of the mesh skewness 3

8 An H-grid produced using gridgen. 3

9 Pressure loss 4

10 Velocity profiles 4

A Supporting Calculations

Analytical Pressure Loss Calculation

Using standard notation for pressure, P, velocity, V density, ρ, and assuming the pipe arrangement is flat Bernoulli's equation simplifies to:

$$V_1^2 - V_2^2 = \frac{2(\Delta P)}{\rho} \tag{1}$$

where $\Delta P = P_2 - P_1$

The DarcyWeisbach equation is:

$$\Delta P = \rho g h_f \tag{2}$$

where $h_f = f\frac{LV^2}{2g}$

g is the gravitational acceleration and can be cancelled out. L is the pipe length, referring to figure 1 it is $3x_1$. However to account for pressure losses due to the junction an additional length of 0.43m is added, using the equivalent length for a 45 degree bend in Engineering Tool Box. Equation (2) requires an estimation of the friction factor, f, which can be obtained from the Blasuis smooth pipe approximation if the Reynolds number, Re, is known.

$$f = \frac{0.0791}{Re^{0.25}} \tag{3}$$

This approximation only applies for turbulent flow. To check that the flow is turbulent a bulk velocity, V, of 2m/s is assumed. Using the pipe diameter, ϕ, the Reynolds number can be calculated:

$$Re = \frac{V\phi}{\nu} \approx 4.5 * 10^6 \tag{4}$$

Which is clearly turbulent. Equations (1) and (2) can be solved simultaneously by assuming the bulk velocity is $(V_1 + V_2)/2$ for the system pressure loss:
Substitute

$$V_1 = \sqrt{\frac{2\Delta P}{\rho} + V_2^2} \tag{5}$$

Into

$$\Delta P = \frac{\rho f L (V_1 + V_2)^2}{8\phi} \tag{6}$$

Results in

$$\Delta P = \frac{\rho f L}{8\phi}\left(\sqrt{\frac{2\Delta P}{\rho} + V_2^2} + V_2\right)^2 \tag{7}$$

Where f is obtained from the Blasuis approximation in terms of V_2 only:

$$f = \frac{0.0791}{\left(\frac{\left(\sqrt{\frac{2\Delta P}{\rho} + V_2^2} + V_2\right)\phi}{2\nu}\right)^{0.25}} \tag{8}$$

In the simulation a ΔP of $14706Pa$ corresponding to a head of 1.5m was used. To check the simulation, the value of V_2 in the simulation was used in equation (8) and ΔP calculated. The equation is recursive so Matlab was used to solve it.

Pluging in the numbers $x_1 = 6$, => $L = 18.43$, Initial assumption $\Delta P = 14706Pa$, $\phi = 0.225m$, $\rho = 1000kg/m^3$, $\nu = 10^{-6}St$, $V_2 = 4.2m/s$: => $V_1 = 4.65m/s, \Delta P = 2kPa$

Grid Convergence Calculation

Grid convergence works by extrapolation of properties such a pressure or velocity (given as f) for a series of grid refinements:

$$f_{h=0} \approx f_1 + \frac{f_1 - f_2}{r^2 - 1} \tag{9}$$

where, r, is the grid refinement ratio, $f_{h=0}$, is the converged value, f_1, refers to the coarse grid and f_2 refers to the fine grid. A grid refinement ratio of 10 was used. The property f chosen was the mean velocity across the prob-line 8.84 meters from the junction. The mean axial velocity for the fine grid was 4.26m/s and 3.96m/s for the coarse grid. For these values $f_{h=0} = 4.26$.

$$A_1 = \frac{f_1 - f_{h=0}}{f_{h=0}} \tag{10}$$

$A_1 = -7.11 * 10^{-4}$

$$E_1 = \frac{\epsilon}{r^p - 1} \tag{11}$$

p is the order of grid convergence. With three girds this can be found simply using logarithmic ratios of the f values and the grid refinement ratio. With two grids the exact solution is required. Unfortunately the exact solution shall not be known until after the flow measurements have been made. where $\epsilon = \frac{f_2 - f_1}{f_1}$

Summary of Settings Used

Setting	Action/Justification
Continua/Regions	
Fluid, water	
Constant density	Water is incompressible.
Segregated flow	Alternative would be coupled–not necessary for low Mach number flows.
Steady	Only steady flow was expected as there were no great obstructions that could for example cause a von Kármán vortex street.
k-ε turbulence model	Flow in pipe bends and vortex meters is turbulent. The alternatives were k-ω, which is suitable for higher Re and Spalart Almar which is only very useful for wings.
Two layer all y+ wall treatment	This is the only wall treatment available in Starccm+, for the k-ε turbulence model.
Initial pressure	A pressure of $14706Pa$ was assumed at the inlets, $0Pa$ at the exit and the reference pressure was set to $0Pa$.
Turbulence specification	Intensity + viscosity ratio of 0.01 and 10 used respectively, these values are simply estimates.
Other	For any of the other settings not mentioned the default program settings were used.

Table 1: Summary table of the settings I used in my simulations