CFD MODELLING AND VALIDATION OF HEAD LOSSES IN PIPE BIFURCATIONS

Kasturi Sukhapure*[a], Alan Burns[a], Tariq Mahmud[a], Jake Spooner[b].

[a] School of Chemical and Process Engineering, University of Leeds, Leeds LS2 9JT, UK
[b] Mott MacDonald, Brighton BN1 4FY, UK
*Author for Correspondance- Email: kasturi.sre@gmail.com

ABSTRACT

Manifold arrangements with bifurcations/trifurcations lead to head losses in a fluid system. It is necessary to predict the steady state head loss coefficients to characterise the losses arising due to these junctions. Miller [1] provides one of the most comprehensive sources for the junction loss coefficients data. However, these data were compiled from isolated research programmes without cross validation. This paper provides detailed quantitative data of sharp-edged bifurcation loss coefficients obtained using Computational Fluid Dynamics (CFD). ANSYS Design Modeller, ICEM and CFX software are used in conjunction to model and simulate flow of water through the pipe junctions. Loss coefficients at high Reynolds number (4.5 x 10^6) flows are investigated. These are expressed as a function of mass flow ratio, cross sectional area ratio and branch angle. In this study, 60 scenarios of pipe junction flows are investigated. The loss coefficients calculated using CFD are compared with measured values from experiments and empirical expressions reported in literature.

Keywords: head loss coefficient, pipe bifurcation, pipe junction loss coefficient, dividing flow, CFD, turbulent flow

NOMENCLATURE

A	[m²]	Cross sectional area of pipe
D	[m]	Pipe diameter
g	[m/s²]	Acceleration due to gravity
H	[m]	Total head
K	[-]	Head loss coefficient
L	[m]	Pipe length
$\dot{M}$	[kg/s]	Mass flow rate
P	[Pa]	Pressure
Q	[m³/s]	Flow rate
Re	[-]	Reynolds number
U	[m/s]	Cross-sectional average velocity
z	[m]	Elevation
Special characters		
f	[-]	Moody friction factor
ε	[mm]	Pipe wall roughness
θ	degree	Angle of junction
μ	kg/ms	Dynamic viscosity
ρ	[kg/m³]	Density of water

Subscripts

1	Concerning Leg 1
2	Concerning Leg 2
3	Concerning Leg 3
31	Across junction between legs 3 and 1
32	Across junction between legs 3 and 2
f	Frictional
in	Inlet
m	Minor
T	Total

INTRODUCTION

Pipe networks with multiple junctions (for example, bifurcations and trifurcations) are widely used to transport liquids or gases in engineering applications, such as in combustion engines, compressors, hydropower turbines, etc. In such flow systems, it is crucial to have knowledge of fluid pressures at various points. For straight pipes, the head loss is a function of the pipe length, hydraulic diameter, wall friction factor and the square of the flow velocity. For pipe junctions, the losses are dependent on additional parameters such as the area ratio, angle between the legs, edge radius and chamfer at the junction. The head loss coefficients at pipe junctions have classically been established experimentally or using mathematical models by simplifying assumptions such as uniform pressure in the recirculation zone or by ignoring incremental frictional losses in the branch. Determining loss coefficients experimentally is a laborious process; it requires a significant time, laboratory space to be available and the physical set up to be built and tested. Hence, it is not necessarily suitable as a standard or regular measurement tool. On the other hand, CFD is an effective tool to investigate and understand the fluid flow behaviour at the junction and predict the head losses in a cost-effective way, which takes complex physics into account.

Various analytical relationships have been developed by combining the momentum equation and Bernoulli's equation to calculate these loss coefficients as a function of the branch angle, area and mass flow ratios [1-4]. Head loss coefficients have also been quantified using the second law of thermodynamics by combining the entropy change due to the junction and the energy transfer in the two legs [5].

The present authors have identified a lack in literature on CFD modelling studies carried out specifically for bifurcations and for a range of area ratios as also echoed by Daniels et al [6]. The study published by Daniels et al. [6], investigating cavitation and head loss coefficients, focusses on penstock bifurcations in hydropower schemes with high Reynolds number (4×10^6 to 17×10^6) flows which are of similar magnitude to the flow rates used in this work.

The main objective of this study is to investigate flow and head loss coefficients for the pipe bifurcations independent of the branch pipe length using CFD. The loss coefficients are calculated as a function of angle of the branch, area ratio and flow rate ratio for a high Reynolds number (4.5×10^6) flow in the main pipe upstream of the junction. The calculated loss coefficient values from CFD are compared and validated against those given in Miller [1].

THEORETICAL BACKGROUND

In viscous fluid flow systems, the total head loss between two locations due to frictional forces is given by the Bernoulli energy equation as:

$$\Delta H = h_f + h_m \qquad (1)$$

Where the total head (or energy per unit weight) can be expressed as:

$$H = P/\rho g + U^2/2g + z \qquad (2)$$

The total head loss in a pipe network includes major losses by pipe friction (h_f) and minor losses (h_m) by components such as bends, junctions, valves, etc. The major head loss in a straight pipe can be expressed in terms of the Moody friction factor (f) as:

$$h_f = f(L/D)(U^2/2g) \qquad (3)$$

The minor loss, for example, for a junction is expressed using a dimensionless head loss coefficient:

$$h_m = K\, U_{in}^2/2g \qquad (4)$$

Where, U_{in} is defined as the upstream cross-sectional average velocity.

MODEL FLOW SYSTEMS

The pipe bifurcation system, with 45° and 90° branch angles, investigated in this study is illustrated in Figure 1. The working fluid is water at 20°C (ρ = 998.20 kg/m^3 and μ = 1.002 kg/ms). The pipe junction designs are sharp-edged to facilitate direct comparison and to be consistent with previous literature. The inlet pipe (Leg 3) extends 20 m upstream from the junction, and both outlet branches (Leg 1 and 2) extend 50 m downstream of the junction. The diameters of Leg 3 carrying the combined flow and Leg 2 carrying the through flow are fixed at 1.5 m. The diameter of Leg 1 is varied according to the area ratio. Six geometries are used in this study: three area ratios (A_1/A_3 = 0.5, 0.33 and 0.25) for 90° and 45° bifurcation angles. ANSYS Design Modeller was used to create the geometry for each angle and area ratio as depicted in Figure 2.

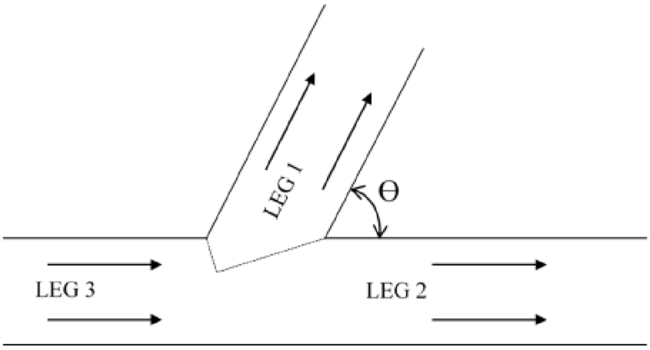

Figure 1 Schematic of the pipe bifurcation system with θ = 45° and 90°

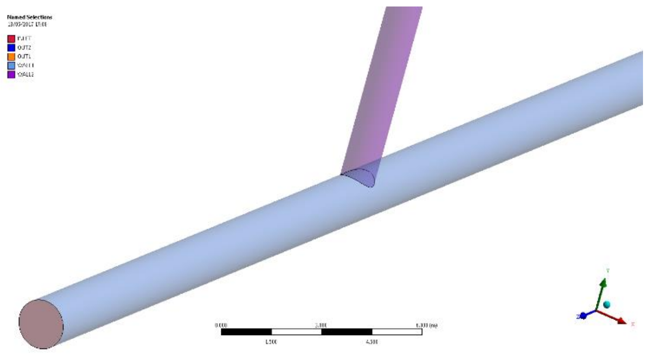

Figure 2 A 90° pipe bifurcation with $A_1/A_3 = 0.5$ created by ANSYS Design Modeller

A hexahedral mesh was constructed using ANSYS ICEM (see Figure 3). Hexahedral elements are more sensitive to quantify head losses than other mesh types such as tetrahedral [7]. One million nodes are used for the mesh after undertaking mesh independence studies on a straight pipe. The straight pipe model was calibrated by comparing the friction factor obtained using CFD with values obtained from the literature. The cells at and around the junction are set to be smaller than the rest of geometry with the cell spacing being exponentially finer closer to the junction from the pipe openings.

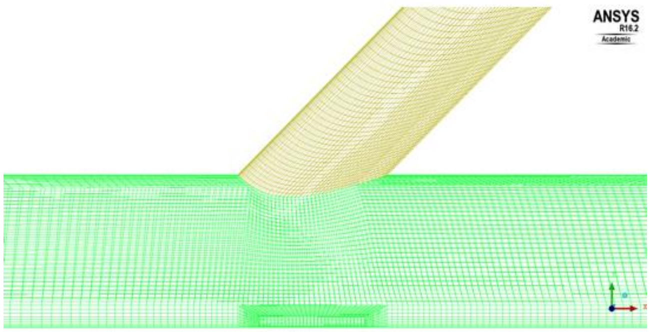

Figure 3 45° Junction mesh created by ANSYS ICEM

CFD METHODOLOGY

ANSYS CFX (version 16.2) software is used to solve the fluid dynamics model using steady, impressible, Reynolds averaged Navier-Stokes equations. The base case flow simulation (90° pipe bifurcation, $A_1/A_3 = 0.5$ and $Q_1/Q_3 = 0.5$) was carried out using the k-ε and the Shear Stress Transport (SST) turbulence models for the purposes of comparison. Both turbulence models predicted the values of head loss coefficient within 3% of each other. The loss coefficient values obtained using the k-ε turbulence model were closer to the literature values. Therefore, the k-ε model was used for the simulations of the rest of the flow cases. The high resolution/second order upwind scheme was used for the discretisation of the convection term of the governing equations. Normalised residuals were set to 10^{-6}.

The flow at the inlet of Leg 3 is set as fully developed flow with a Reynolds number of 4.5×10^6. The inlet boundary condition is set as 0 Pa relative static pressure. Mass flow boundary conditions are used at the outlet of Legs 1 and 2. Flow ratios between Leg 1 and Leg 3 (Q_1/Q_3) are varied between 0 and 1 at 0.1 intervals. The pipe wall roughness is set as 0.025 mm for all the model flow systems. This is a typical roughness value for new steel pipes [1].

Boundary Conditions

The total mass flow rate through the system was defined in the CFD software as $\dot{M}$. The pressure specified boundary condition at the inlet results in zero velocity gradients at the inlet thus giving a fully developed flow if the pipe is long enough to accommodate the entrance effects.

Table 1 Boundary Conditions

Part	Quantity	Boundary Condition	Unit
Inlet	Static pressure	0	Pa
Out 1	Mass flow ratio	e.g. 0.1, 0.2,	[-]
Out 2	Mass flow ratio	e.g. 0.9, 0.8,	[-]
Wall	No Slip Wall Roughness	 0.025	 mm

Table 2 Computational Details

Parameter	Specification
Domain	Water at 20°C
Heat transfer	Isothermal
Turbulence model	k-ε /SST
Density of water (ρ)	998.20 kg/m^3
Dynamic viscosity of water (μ)	1.002 kg/ms
Residuals	E-06
Solver order	High resolution/ Second order upwind

PROCESSING OF CFD RESULTS

The methodology for estimating the head loss coefficient, K, in equation (4) using CFD simulation results is based on previous CFD and experimental procedures [1, 4, 6]. In previous studies [1, 6], a pressure probe was inserted 2-3 pipe diameters upstream of the junction and another probe was inserted around 6-10 pipe diameters downstream. In this study, the cross-sectional average pressure is determined at 3 pipe diameters upstream of the junction and at multiple locations (ranging from 1 to 40 pipe diameters) downstream of the junction. These results show how the junction loss coefficient becomes steady after about 5-7 pipe diameters by illustrating the trend of the head loss along Leg 1 (see Figure 11).

Calculation of Friction Factor

Moody friction factor is used in this paper for consistency. The Colebrook-White equation is widely used to calculate the friction factor for turbulent flow in rough pipes [1, 8]

$$1/\sqrt{f} = -2\log\left(2.51/\left(Re\sqrt{f}\right) + \varepsilon/(3.7D)\right) \quad (5)$$

Where Reynolds number is the ratio of viscous forces to the inertial forces; and is characterised as $Re = \rho UD/\mu$.The equation requires an iterative solution, which is unfavourable for quick estimations of friction factors. Previous studies [9, 10] have established explicit approximations of equation (5). In this study, an approximation put forth by Miller [1], which is applicable for low relative roughness (defined as the ratio of pipe wall roughness/pipe diameter) values is used. For example, for Leg 3, relative roughness (ε/D) is 1.67×10^{-5}.

$$f = 0.25 \,/\, [\log(\varepsilon/(3.7D) + 5.74/Re^{0.9})]^2 \quad (6)$$

The friction factor for fully developed turbulent flow in a straight pipe is computed using the CFD predicted pressure drop from equation (7):

$$f = \Delta P/[(L/D)(\rho U^2/2)] \quad (7)$$

The CFD predicted friction factor values are validated against both the Colebrook-White equation (5) and its approximation, (Equation 6).

Calculation of Loss Coefficient

The total head loss coefficient for any two legs of the flow system is calculated in terms of the pressure drop obtained from CFD. This includes the losses due to pipe friction and at the junction:

$$K_T = \Delta P/\left(\rho {U_3}^2/2\right) \quad (8)$$

As the total head loss includes the losses due to pipe friction, these losses need to be subtracted in order to obtain the absolute value of the loss coefficient at the junction. The head loss coefficient at the junction of Legs 3 and 1 is defined as K_{31} and for Legs 3 and 2 as K_{32}. For example, for determining the loss coefficient at the junction of Leg 3 and Leg 1, the following equation is used:

$$K_{31} = h_{m,31}/(U_3^2/2g) = \left(\Delta H_{31} - h_{f,31}\right)/(U_3^2/2g) \quad (9)$$

The loss coefficient (K_{31}), based on incremental pipe length, downstream of the junction is calculated using equation (9). Once the trend of the loss coefficient is established, pressure planes are inserted at 20 pipe diameters downstream for Leg 1 and Leg 2 for the rest of the geometries, except for the 45º Leg 2 where the plane is inserted at 30 pipe diameters due to geometry restrictions.

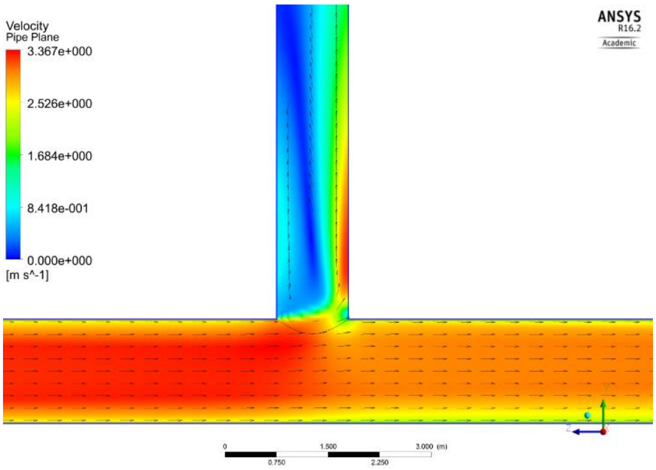

Figure 4 Velocity distribution and velocity vectors for the 90° junction (Q_1/Q_3 = 0.1)

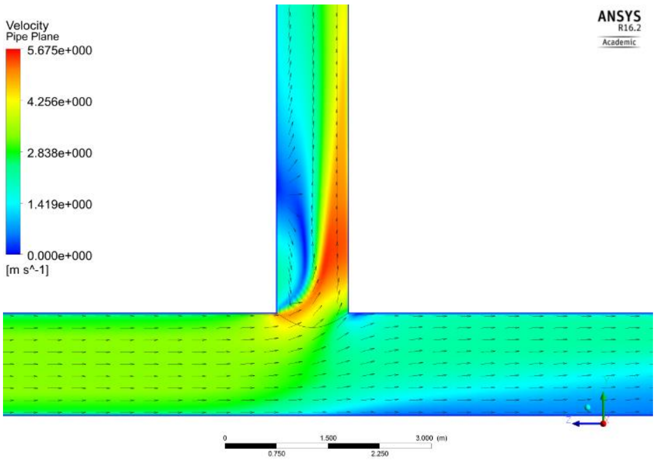

Figure 5 Velocity distribution and velocity vectors for the 90° junction (Q_1/Q_3 = 0.5)

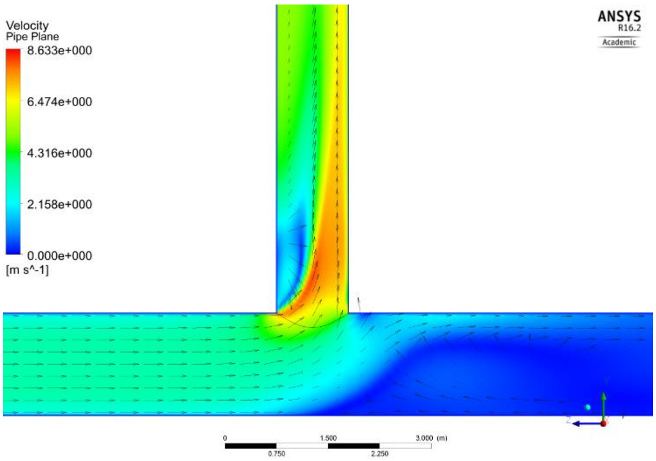

Figure 6 Velocity distribution and velocity vectors for the 90° junction (Q_1/Q_3 = 0.9)

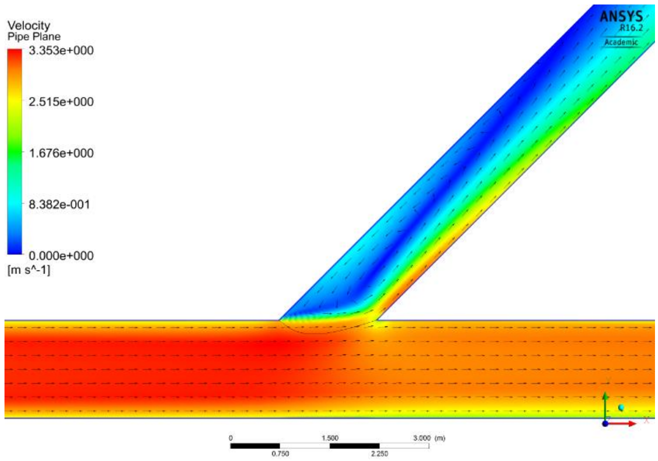

Figure 7 Velocity distribution and velocity vectors for the 45° junction (Q_1/Q_3 = 0.1)

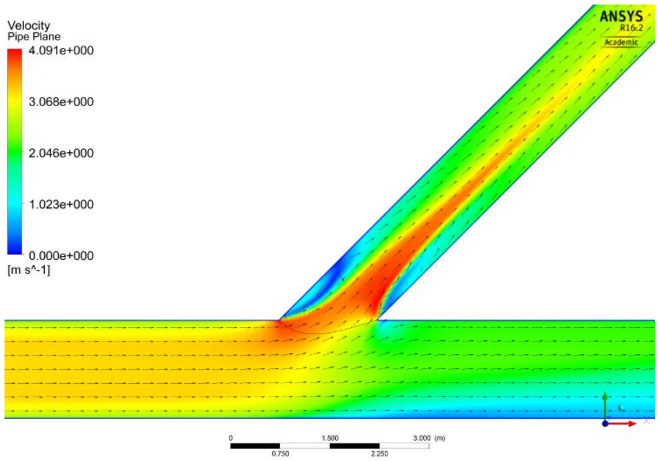

Figure 8 Velocity distribution and velocity vectors for the 45° junction (Q_1/Q_3 = 0.5)

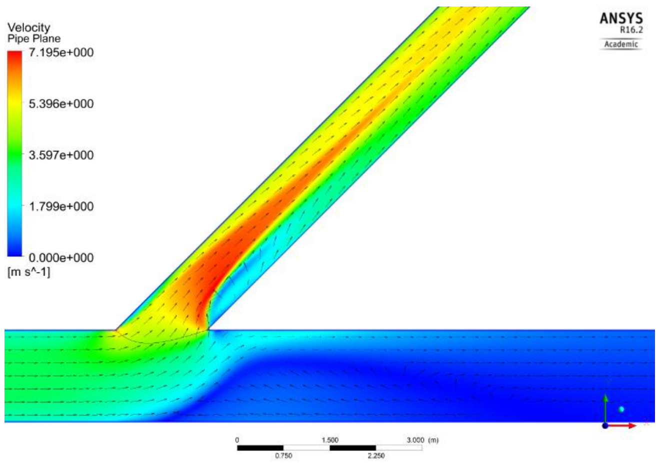

Figure 9 Velocity distribution and velocity vectors for the 45° junction (Q_1/Q_3 = 0.9)

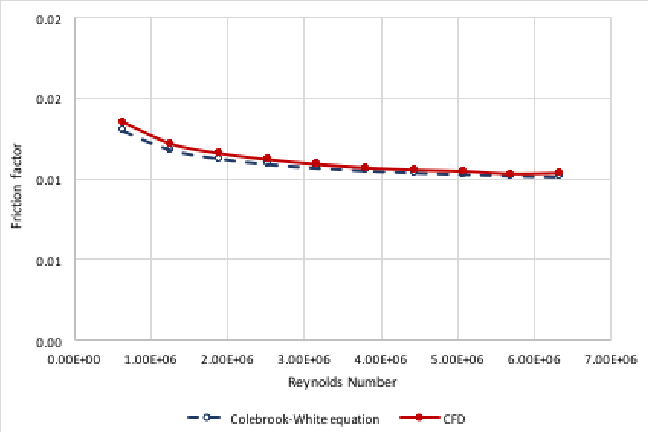

Figure 10 Comparison of friction factors for 90° angle ($A_1/A_3 = 0.5$)

RESULTS & DISCUSSION

Flow Pattern

A symmetry plane along the pipe cross section is used to plot the predicted flow pattern. The velocity distribution and velocity vectors are plotted on this symmetry plane for three flow rate splits (Q_1/Q_3 = 0.1, 0.5 and 0.9) for the 90° and 45° angle junction ($A_1/A_3 = 0.5$) as shown in Figure 4 to Figure 9. The velocity vectors indicate that there is a significant recirculation zone in Leg 1. The recirculation is more distinct with lower Reynolds number flows through Leg 1. The entrance effect can also be observed in Leg 3 as the flow approaches the junction where the fully developed flow branches off. The pressure plane to determine the loss coefficient in Leg 3 is inserted before this effect.

Friction Factor

Simulations of flow in a straight pipe were first performed in order to assess the mesh resolution based on the accuracy of the predicted friction factor. The flows in the legs of the bifurcation model are not fully developed hence the friction factor in the legs can vary. A comparison of the friction factors for the 90° junction ($A_1/A_3 = 0.5$) obtained from Colebrook-White equation and the CFD model is shown in Figure 10. The CFD results are similar to the Colebrook-White equation results. The friction factor decreases with an increase in Reynolds number.

Loss Coefficient

Figure 11 best illustrates the methodology used in calculating the head loss coefficients for the bifurcation. The initial increase in the head loss and loss coefficient is due to flow recirculation and flow stagnation. It is observed that around 5-7 pipe diameters downstream of the junction the total head loss and total loss coefficient reach the minimum value and then increase steadily with incremental length due to the frictional losses in the pipe. This steady increase is indicated by the slope of the frictional head loss curve. As discussed previously, these frictional losses are subtracted to calculate the head loss and loss coefficient solely due to the junction. Any point downstream from 5-7 pipe diameters onwards yields a consistent head loss coefficient for the junction. Loss coefficients, K_{31}, for the 90° junction for all three geometries are compiled together in Figure 12. The loss coefficient K_{31} increases with an increase in the flow ratio i.e. with Reynolds number in Leg 1. The CFD results for the three A_1/A_3 cases are similar to the values given in Miller [1]. The Miller [1] values for loss coefficient above 0.8 flow splits are not available for $A_1/A_3 = 0.25$ and hence the comparison at these high flow rates is difficult. CFD results address this gap in data. The loss coefficients generally increase with a decrease in area ratio especially at higher flow splits.

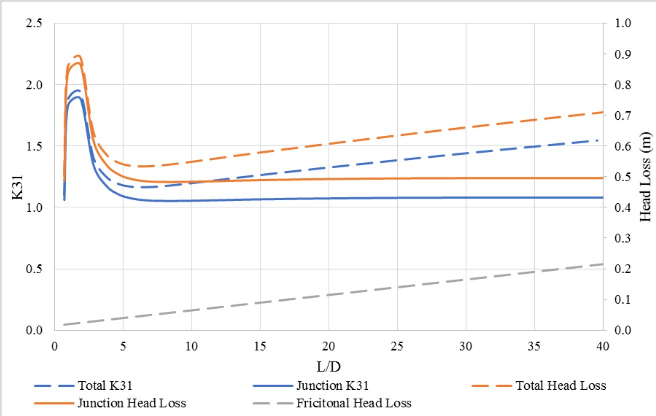

Figure 11 Loss Coefficient K_{31} for 90° angle ($A_1/A_3 = 0.5$ and $Q_1/Q_3 = 0.5$)

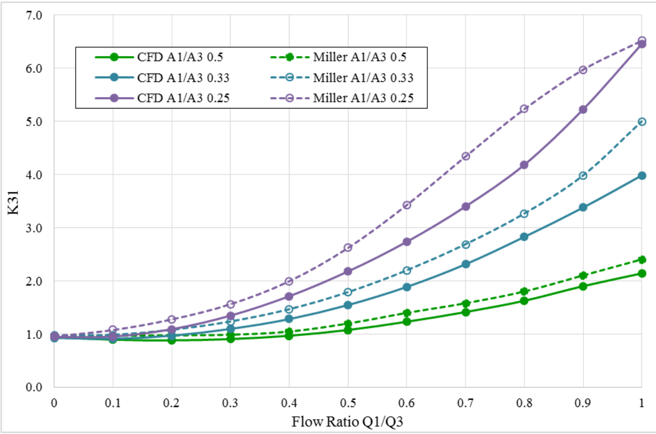

Figure 12 K_{31} loss coefficients for 90° junction

For an area ratio of $A_1/A_3 = 0.5$ the flow splits are more likely to be around $Q_1/Q_3 = 0.5$. Similarly, the loss coefficients for an area ratio $A_1/A_3 = 0.33$ the flow splits are more likely to be around $Q_1/Q_3 = 0.33$ and so on. At these typical ratios the difference between the CFD results and Miller values reduces. For example, for the 90° case ($A_1/A_3 = 0.33$), the difference at $Q_1/Q_3 = 0.33$ is around 10% which increases to 20% above $Q_1/Q_3 = 0.6$ between the CFD and Miller values. Loss coefficients, K_{31}, for 45° bifurcation angle are compiled together in Figure 13. The CFD results are similar to the reference values [1]]. The loss coefficients for $A_1/A_3 = 0.25$ show a rapid rise in the loss coefficient at higher flow ratios through Leg 1. Overall, for all cases, the CFD predicted loss coefficients, K_{31} values, were lower than the corresponding values from Miller [1]]. The largest difference was 50% for the 45° junction (A_1/A_3 = 0.5 at Q_1/Q_3 = 0.5) and the smallest difference was 1% for the 45° junction (A_1/A_3 = 0.33 at Q_1/Q_3 = 0.1).

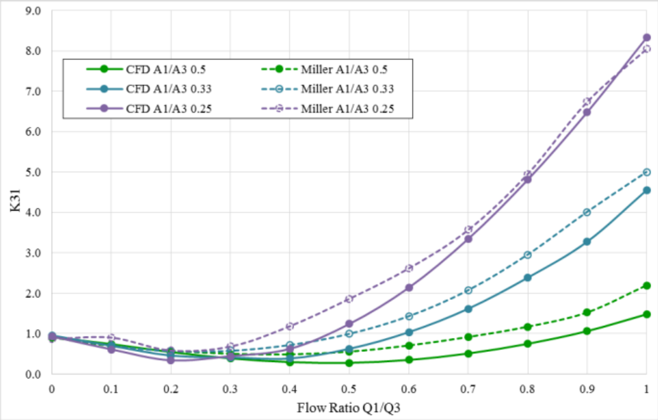

Figure 13 K_{31} loss coefficients for 45° junction

The K_{32} loss coefficient values for all the cases (90° and 45° branch angle and A_1/A_3 = 0.5, 0.33 ad 0.25) are presented in Figure 14. The K_{32} loss coefficient does not vary greatly with the angle or area ratio. Collectively the CFD results display an altered trend when compared with Miller [1]]. The K_{32} value is the lowest for flow splits between 0.1-0.3 (Q_1/Q_3 = 0.1-0.3). It then increases with a decrease in flow ratio through Leg 2.

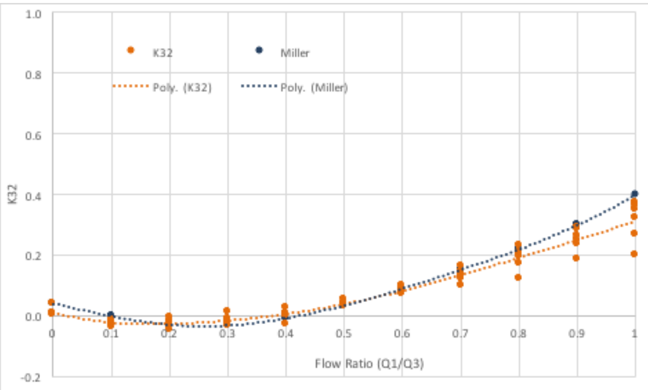

Figure 14 K_{32} (leg 2) loss coefficients for 90° and 45° junction

CONCLUSION

Loss coefficient, K, values are dependent on the angle, area ratio and the flow split ratio. A range of configurations have been considered and compared to existing literature. Flow phenomenon like recirculation zones and entrance effects can be visualised using CFD. These results indicate that CFD results for loss coefficients K_{31} are lower than those presented in Miller [1]. It is assumed that the difference is at least in part due to idealised geometries in CFD compared to physical modelling; the result of which may be impacted by any imperfections in the geometry and uncertainties in the measuring instruments. The results presented in this paper suggests that Miller's [1] results can be considered conservative for design purposes. For smaller pipes and higher Reynolds numbers, the impact of such minor imperfections is likely to increase. As such, suitable allowances and factors of safety may need to be considered when applying results based on CFD for real world applications.

The ANSYS ICEM software used for this study was the Academic version, which restricts the number of nodes permits and prevents an elaborate mesh independence exercise. The flow in the legs was not fully developed. Hence applying the same friction factor throughout the legs could yield in minor errors. CFD can be used to fill the gap in existing data, for example for the 90° junction (A_1/A_3= 0.25) where the values for higher flow splits are not available in Miller [1]. Future opportunities for this study include exploring further bifurcation angles for various area ratios.

REFERENCES

1. Miller, D.S., *Internal Flow Systems* 2nd ed. 1990, Surrey: Ulwin Brothers Ltd.
2. Fried, E. and I. Idelchick, E, *Flow Resistance: A design guide for engineers*. 1989, Philadelphia: Hemisphere Publishing Corporation.
3. Štigler, J., R. Klas, and O. Šperka, *Characteristics of the T-junction with the equal diameters of all branches for the variable angle of the adjacent branch.* EPJ Web of Conferences, 2014. **67**: p. 02110.
4. Bassett, M.D., D.E. Winterbone, and R.J. Pearson, *Calculation of steady flow pressure loss coefficients for pipe junctions.* Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science, 2001. **215**(8): p. 861-881.
5. Schmandt, B., V. Iyer, and H. Herwig, *Determination of head change coefficients for dividing and combining junctions: A method based on the second law of thermodynamics.* Chemical Engineering Science, 2014. **111**: p. 191-202.
6. Daniels, S., E , M. Johnson, C, and R. Spall, E, *Cavitation and head loss charateristics in nonsymmetric bifurcations.* American Water Works Association 2014. **106**(6): p. E280-E294.
7. Aguirre, C., A and R.R. Camacho, *Head losses analysis in symmetrical trifurcations of penstocks-high pressure pipeline systems cfd.*
8. Moody, L.F., *Friction factors for pipe flow.* Trans. Asme, 1944. **66**(8): p. 671-684.
9. Shaikh, M.M., S.-u.-R. Massan, and A.I. Wagan, *A new explicit approximation to Colebrook's friction factor in rough pipes under highly turbulent cases.* International Journal of Heat and Mass Transfer, 2015. **88**: p. 538-543.
10. Brkić, D., *A note on explicit approximations to Colebrook's friction factor in rough pipes under highly turbulent cases.* International Journal of Heat and Mass Transfer, 2016. **93**: p. 513-515