20th Australasian Fluid Mechanics Conference
Perth, Australia
5-8 December 2016

Internal Flow: Pipe Losses Steady State and Transient Analysis

N. Findanis

Department of Aerospace Engineering
UNSW Australia, New South Wales 2052, Australia

Abstract

An important area of industrial fluid mechanics is internal pipe flows. This ubiquitous application of internal fluid flow is applied in many fields of engineering requiring the transport and transfer of fluids and so is extremely wide spread and an ongoing area of research interest. A key area of industrial pipe flows is the associated pressure drop in the line pressure between two points caused by the major loss of pipe friction. The added effect of "minor" internal pipe losses is the application of the losses from pipe fittings, entry/exit losses, sudden expansion/contraction. Some pipe fittings include the use of bends, reducers, exit and entry types, pipe junctions such as wyes and tees apart from the major loss of pipe friction. Currently, the majority of industrial fluid flow is concerned with the steady state regime, especially when considering a Computational Fluid Dynamics (CFD) analysis of the flow field simulation. The application of new and immerging technology is increasingly concerned with flows that require the CFD analysis to be performed as a transient flow to observe performance parameters as an evolution of time. One area that utilises highly transient flow is reverse pulse-jet cleaning systems. The present work is concerned with using passive flow control to reduce the minor pipe losses. By using a CFD simulation that is conducted as both a steady state and transient flow analysis, different pipe fittings were analysed in the two systems of analysis to show the effectiveness of the passive flow control device with reference to the plain pipe flow. It was shown that using passive flow control that induces a swirling motion upstream of the pipe fitting has the effect that it reduces the downstream diameters that are needed to produce fully developed and smooth pipe flow.

Introduction

The flow fluids through internally fully closed conduits is a ubiquitous area of industrial plants; including power generation, powder and bulk solids handling, water and waste handling, fluid distribution networks, minerals processing and refineries and other manufacturing process and many other application [1]. There is a broad theoretical basis to describe the fundamentals of fluid flow in pipes, however closed analytical solutions are relatively few and exist in only simple or simplified cases [1]. An example of analytical solutions, would be in fluid flow cases involving laminar or fully developed flow. However, the majority of the many types of different industrial flows are not in the laminar flow regime, but rather these flows exceed the turbulent flow transition criterion for internal flows involving the transfer of fluid through circular conduits or round pipes. Drawing direction from the Moody Chart indicates that at a Reynold's number between 2300 and 4000, the flow inside a completely full pipe will transition from laminar to the turbulent flow regime [1]. The transition can be affected by various factors that can induce a disturbance to the internal pipe flow to cause transition to turbulence. Therefore, for flows that are initially laminar they can quite easily become turbulent if not strictly controlled when the Reynolds number approaches the transition criterion. This further emphasises the omnipresent nature of turbulent flows in industry and in nature. Aside from an increase in turbulence caused by surface inconsistencies found in commercially manufactured pipe, piping components or pipe fittings are a major source of disturbance in the internal pipe fluid flow [2]. In the modelling of a fluid flow piping network it is key to determine the performance parameters of a piping network and to optimise the design in terms of these parameters including the flow quality for optimum operating conditions to generate the highest capable efficiency. This is key in industrial applications. A core parameter to the performance of a piping network is the pressure drop between the driving source of the fluid flow where the total pressure is at a maximum, and the delivery point of the fluid, where the total pressure has decreased. The drop in pressure along the fluid path is caused by the major loss attributed to irreversible viscous losses or fluid friction between the fluid and the internal supporting pipe surface. The interest of this current work has a focus on the turbulent losses induced by pipe components and fittings termed minor losses. Depending on the configuration of the piping system the minor losses could exceed the major losses. This is the case in reverse pulse-jet filter cleaning systems. A reverse pulse-jet filter cleaning system is used in dust collectors to perform the periodic cleaning of filter bags.

A typical RPJ cleaning system consists of three main components: compressed air supply, valve and blowtube. The valve is also more explicitly known as pulse-jet valve. The blowtube is the name given to the pipe connected to the valve and configured with the same number of outlets as the number of filters to which it delivers the reverse flow. In the present work, improvements are sought from the pipe fittings or components between the valve and the blowtube for a more efficient operation of a well-designed cleaning system. The RPJ cleaning system is energised from a store of compressed gas. The work a compressor must do to compress the gas comes at a cost. For a large dust collector comprised of over a thousand pulse-jet valves, pulsing a number of valves simultaneously every few seconds with each valve possibly using 300 L of air is a large operating cost. Thus, there is a need to increase the efficiency of the cleaning system so as to decrease the overall air consumption. The lower power consumption of the compressor goes towards decreasing the carbon footprint of the dust collector and the overall process whether that be power generation, cement plant, minerals processing etc. Operators of industrial plants are keenly interested in making improvements wherever they can be obtained provided it can be shown that there is a net operating cost benefit. Government regulatory agencies, such as the Environmental Protection Agency (EPA) in the United States and Australia with its own Environmental Protection Authority (EPA), are tightening the standards on pollutants released into our environment.

The flow physics of an RPJ cleaning system falls into the category of gas dynamics. The stored energy in terms of the compressed gas and the quick release of the high potential energy from the gas results in a compressible flow. The high speed phenomena associated with the system need to be further investigated to gauge whether there exist possibilities to enhance any inherent flow characteristics to improve the cleaning system. This deficiency in understanding of the system gives the impetus needed to further investigate and gauge whether there exist possibilities to enhance any inherent flow characteristics to improve the cleaning system including pipe fittings.

Piping systems are not usually directed in a linear path from the starting point to the end point but rather change direction many times along the pathway. The present paper focuses on improving the flow through pipe fittings. A frequently used pipe fitting is the 90^o bend [3], which is also commonly used in reverse pulse-jet filter cleaning systems. Other fittings are also used between the pulse-jet valve and the blowtube to distribute the flow according to the configuration of the cleaning system design. For example, the use of a wye or tee fitting is used to split the flow from a single pulse-jet valve to two blowtubes simultaneously.

The operation of a single event or an actuation of the pulse-jet valve, is extremely rapid; typically approximately 300 ms. The valve is actuated and the diaphragm moves and allows the compressed air to travel from the pressure vessel or header tank through the valve past the valve seat into the blowtube which could include pipe fittings of some type.

The present paper will discuss and compare the losses of pipe fittings as analysed using both the steady state and transient analysis flow regimes. This will enable the determination of a methodology for the efficient design analysis of this type of industrial flow system. Further, it will be shown that using passive control devices can reduce the losses of pipe fittings.

Other technologies could be added to the system to extract fluid dynamic losses and improve the flow through the system. Advancements made by the author in subsonic flows [4, 5] could have an application to modify the gas dynamics of an RPJ cleaning system which could provide not only improvements to flow but further insight to the physics of high speed flows, in particular around pipe fittings.

Method and Procedure

The method used to analyse and assess the various configurations of the pipe fittings was carried out using Computational Fluid Dynamics (CFD). The CFD analysis process will be described in more detail in this section.

a. Modelling, Setup and Governing Equations

This computational investigation has been conducted in a three-dimensional space domain. The solid models were constructed using commercially available CAD software, this was then modified using the same software to produce the fluid volume.

The flow through the system can be analysed in either one of two possible analysis options, as a steady state (snap shot) flow or transient flow (time evolution) of the flow field. For the steady state system the flow does not change over time but rather as was mentioned previously, is a snap shot in time of the fluid flow of the system. Whereas for a transient analysis, it accounts for how the flow develops, forms and fluctuates over a specified time period. In the present case, the steady state analysis provides the means to be able to determine if there is a difference in the performance between the pipe fittings. The transient analysis can be useful in showing if the evolution of the flow over time in the pipe fitting will show any behaviour that could be used critically in the selection process between two pipe fittings. In the present case, a transient analysis was also performed to determine if the steady state analysis was adequate to perform the assessment between the different pipe fittings.

The three-dimensional governing equations for the flow of an incompressible fluid are solved using the unsteady Reynolds Averaged Navier-Stokes equations. The conservation of mass and momentum equations are shown as equations (1) and (2).

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

$$\rho \frac{\partial \overline{\mathbf{u}}}{\partial t} + \rho \overline{\mathbf{u}} \cdot \nabla \overline{\mathbf{u}} = -\nabla \overline{p} + (\mu + \mu_t) \nabla^2 \overline{\mathbf{u}} \qquad (2)$$

The simulation is firstly conducted as a steady state analysis and then followed with a transient flow analysis for certain cases with and without passive flow control.

The simulation is conducted as a steady state analysis for all cases with and without the passive flow control device applied. High resolution turbulence numerics based on the advection and transient scheme is used in the turbulence modelling [6]. The turbulence model used to calculate the averaged turbulent stresses is based on Menter Shear Stress Transport (SST) two-equation model [6] being the most suitable when flow separation occurs in the flow field.

The transient scheme used is second order backward Euler for the discretisation of the transient term. A high resolution upwind advection scheme with third order spatial accuracy is used to calculate the advection terms.

The simulation was performed for two different flow regimes through the pipe fitting for the 90^o bend. This was conducted to determine if there is a difference in the performance between the plain 90^o bend and then with the modification of the passive flow control device, between a subsonic incompressible flow and a compressible flow regime through the pipe fitting. The subsonic flow had an inlet condition set at 100 ms^{-1} and the compressible flow regime used an inlet flow condition of 500 kPa. The 500 kPa represents an initial header pressure that would be used in the reverse pulse-jet cleaning system. The physics setup is shown below for the plain bend in Figure 1.

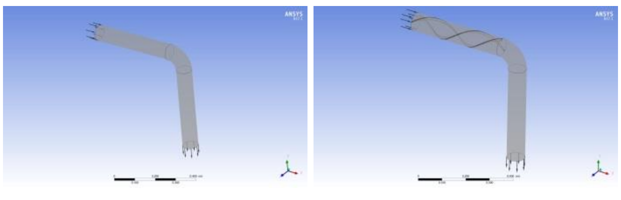

Figure 1. 3in Plain 90^o Pipe Bend. R/D = 1 (left), 3in 90^o Pipe Bend with passive flow control with swirl C(right).

The 90^o pipe bend was based on 3” SCH40 commercial pipe dimensions, which is commonly used in 3” reverse pulse-jet systems. The ratio of R/D = 1.

The 90^o pipe bend also included modifications in the form of passive flow control to improve the flow through the pipe bend and mitigate the occurrence of flow separation. In addition to mitigating flow separation, the quality of the flow downstream of the pipe bend is normally non-uniform, and requires a number of diameters for homogeneous correction and allow the flow to be useful in industrial applications.

b. Grids-independence study and validation

The grid was generated using the ANSYS CFX mesher based on an unstructured approach. The mesh is based on a patch conforming tetrahedral method with inflation on the relevant surfaces. The mesh was created with particular attention to sizing on the pipe passive flow control surfaces to capture the flow physics. Further, many trails were performed with the configuration of the inflation layers to ensure the separation of flow is captured with the SST turbulence model.

Figure 2 shows the meshed fluid domain volumes for the plain 3in pipe and the passive flow control model with type swirl C.

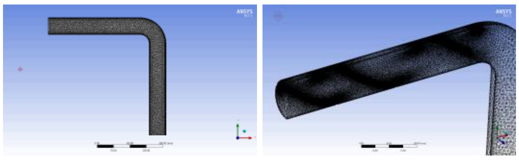

Figure 2. Mesh on 3in Plain 90° Pipe Bend (left), Mesh on 3in 90° Pipe Bend with passive flow control with swirl C (right).

A grid independent study was conducted to ensure an accurate solution to the CFD analysis. This was conducted using the three different mesh densities. The results show that a minimum deviation occured between the medium and fine grid densities. Thus, it was decided that to maintain the integrity of the solution, the fine mesh was used in the present work.

Results and Discussion

The steady state simulations where performed first with the inlet flow into the 90° pipe bend types set at 100 ms^{-1}, a subsonic flow regime.

The reference or base case with which to compare the passive flow control cases was the 90° pipe bend with a ratio of R/D =1. The flow into the pipe bend was uniform and the same length on the pipe bend was used post of the the pipe bend.

For the cases where the 90° pipe bend was modified, three different types of passive flow control devices were used on the 90° pipe bend for the steady state subsonic flow regime. These were labelled as:

- Swirl A
- Swirl B
- Swirl C

For each type of passive flow control device, the method that was used to affect the behaviour of the upstream flow to the bend was to induce a swirl motion to the flow as it approached the pipe bend. It was surmised that inducing a degree of swirl to the motion of the fluid would impart another component of velocity to the flow and possibly affect the flow of the fluid around the 90° bend and mitigate flow separation. The swirling motion would encourage the fluid to break apart the separation and recirculation zone that could form just post the 90° bend. An additional advantage of the swirling motion in the 90° bend upstream would be to produce a more uniform flow in the section following the 90° bend and reduce the length required or the diameters downstream required to have once again uniform flow in the pipe.

To generate the swirl in the fluid, air, a type of blade was used with a radius curve that would scoop and hold a portion of the air flow from the inlet and impart a crossflow component to turn the flow through the pipe section. In the first swirler, A, a helical curved path of the blade is placed from the inlet to the start of the bend with a single turn or revolution. In swirler B, the helical path has 1.2 revolutions and shorter pitch. Then in swirler C, the helical path is as was constructed in swirler B, but two helical paths were created that do not intersect. For swirler C, the two paths would generate or would produce an additional amount or degree of swirl to the fluid motion in the pipe upstream of the 90° bend.

The simulation results were in ANSYS CFX post processing for the base case and the three different passive flow control devices or swirlers (A, B & C).

In Figure 3, the pressure contour is plotted for each case. In the top left is shown the base case of the plain 3in 90° pipe bend. The region of very low pressure indicating recirculation is indicated in the inside corner of the pipe bend. The low region of recirculation is reduced in the other three pipes with passive flow control accompanied with an increase in the pressure developed around the outside of the bend.

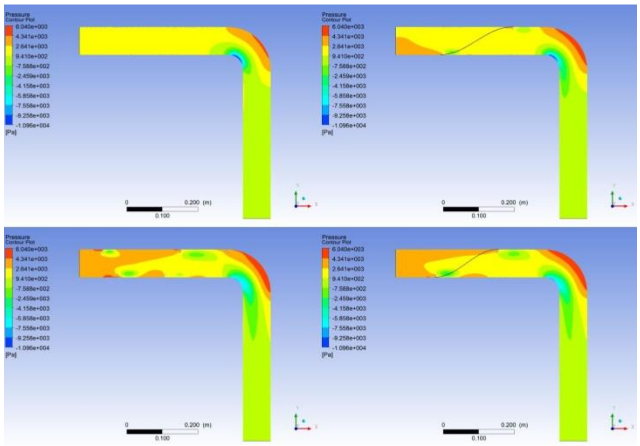

Figure 3. Pressure Contour, steady state, 100 ms^{-1} inlet flow, clockwise from top left; Base, swirl A, swirl B & swirl C.

In Figure 4, the velocity contour plot indicates the total velocity around the bend is most uniform for swirl C as well as the downstream flow in the pipe section. However, there is a significant increase of uniformity of swirl upstream of the bend owing to the double helical passive flow control device of swirl C. This would inhibit the mass flow rate through the system by generating a larger resistance and pressure drop in the upstream flow.

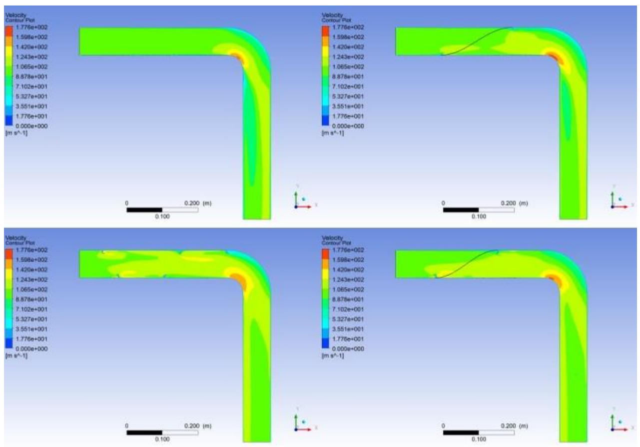

Figure 4. Velocity Contour, steady state, 100 ms^{-1} inlet flow, clockwise from top left; Base, swirl A, swirl B & swirl C.

In Figure 5, the turbulent kinetic energy (TKE) shows the separation region in the plain pipe on the inside corner bend and on the outer bend the smaller amount of recirculation. The swirl devices increase the TKE especially the double helical swirl in C.

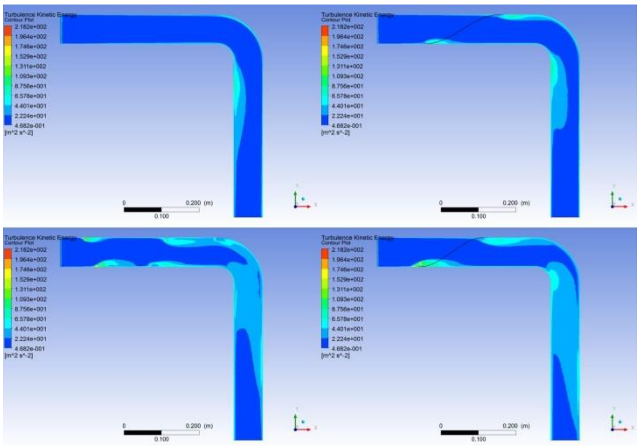

Figure 5. Turbulent Kinetic Energy Contour, steady state, 100 ms^{-1} inlet flow, clockwise from top left; Base, swirl A, swirl B & swirl C.

The flow regime was then changed to compressible flow by changing the inlet condition to 500 kPa total pressure. The comparison was made between the base 3in pipe section, swirl B and a different swirler type D. Swirl D was also a double helical path, however the direction of the blade was altered to lower pressure drop and direct the flow to trouble regions of slower flow.

In Figure 6, a more dramatic increase in the flow separation is indicated on the inside of the bend for the base case. The flow separation is reduced when using the flow control passive devices in swirl B and swirl D.

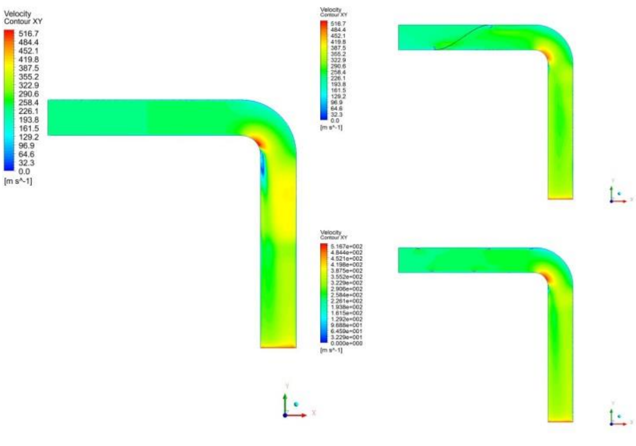

Figure 6. Velocity Contour, steady state, 100 ms^{-1} inlet flow, clockwise from left; Base, swirl C, & swirl D.

In Figure 7, the TKE is plotted to show the effect on the mixing or swirl of the two passive flow control devices and the base case. The flow separation region of the base case can be clearly seen in this contrasted plot by the increased activity around the inside of the 90^{o} plain 3in bend. With the device swirl B the same increased swirl activity is shown around the blade upstream of the bend including the separation region on the inside of the bend. However relative to the base case the amount of TKE is significantly reduced. Consider swirl D, this has a relatively lower amount of swirl in the separation region on the inside bend of the bend in the form of a lower TKE in this location. Swirl D has the effect of generating a homogeneous TKE downstream of the pipe bend.

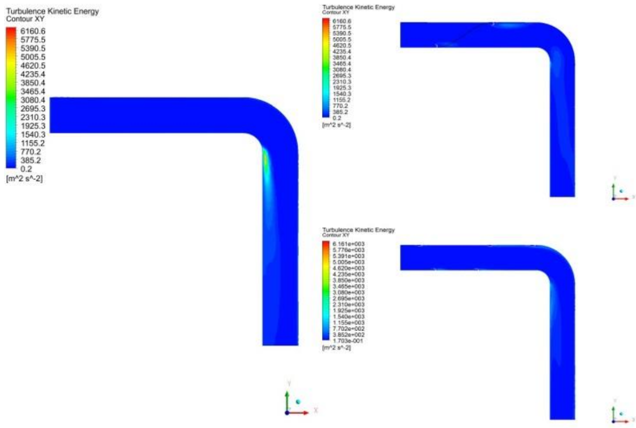

Figure 7. Velocity Contour, steady state, 100 ms^{-1} inlet flow, clockwise from left; Base, swirl C, & swirl D.

Conclusions

By using a CFD simulation that is conducted as both a steady state and transient flow analysis, different pipe fittings were analysed in the two systems of analysis to show the effectiveness of passive flow control device swirlers in reference to the plain pipe flow. It was shown that using passive flow control that induces a swirling motion upstream of the pipe fitting has the effect that it could reduce the downstream diameters that are needed to produce fully developed and smooth pipe flow.

Acknowledgments

I would like to acknowledge the support of UNSW in my research support.

References

[1] Cengel, Y.A. & Cimbala, J.M., *Fluid Mechanics: Fundamentals and Applications*, 2nd Edition, McGraw Hill, 2010.

[2] Miller, D.S., *Internal Flow: A Guide to Losses in Pipe and Duct Systems*, BHRA, 1971.

[3] Beij, K.H., *Pressure Losses for Fluid Flow in 90^{o} Pipe Bends*, Journal of Research of the National Bureau of Standards, Vol. 21, 1938, 1-18.

[4] Findanis, N., Ahmed, N.A., *The interaction of an asymmetrical localised synthetic jet on a side supported sphere,* Journal of Fluids and Structures 2008; 24: 1006-1020.

[5] Findanis, N., Ahmed, N.A., *Three-dimensional flow reversal and wake characterisation of a sphere modified with active flow control using synthetic jet*, Advances and Applications in Fluid Mechanics 2011; 9 (1): 17-76.

[6] Menter, F.R., *Zonal two equation k-ω turbulence models for aerodynamic flows*, AIAA paper 93-2906.