UNIVERSITY OF OKLAHOMA

GRADUATE COLLEGE

VIBRATION REDUCTION OF A WATER PIPE T-JUNCTION USING CFD

A THESIS

SUBMITTED TO THE GRADUATE FACULTY

in partial fulfillment of the requirements for the

Degree of

MASTER OF SCIENCE

By

BLAKE ROBERT EISNER
Norman, Oklahoma
2013
VIBRATION REDUCTION OF A WATER PIPE T-JUNCTION USING CFD

A THESIS APPROVED FOR THE
SCHOOL OF AEROSPACE AND MECHANICAL ENGINEERING

BY

______________________________
Dr. Kurt Gramoll, Chair

______________________________
Dr. Wilson Merchan-Merchan

______________________________
Dr. Zahed Siddique
Acknowledgements

I would like to thank first and foremost my advisor Dr. Kurt Gramoll. He provided the computational resources necessary to conduct the large simulations in the allotted time. In addition, the weekly meetings he never missed helped me to come up with new ideas and meet the deadlines set forth. Both of these factors were vital to the success of the research.

The defense committee member also provided valuable feedback to the report, which I am grateful for. The FMC Technologies Fluid Control Division helped to come up with the topic for the research and define the scope. Without this assistance, the importance of the topic would have not been realized.

Finally, I would like to thank my parents, Grant and Julia Eisner, for sparking my interest in engineering and supporting me in my pursuit of higher education.
### Table of Contents

Acknowledgements ................................................................................................. iv  
List of Tables ........................................................................................................... viii  
List of Figures ......................................................................................................... ix  
Abstract ................................................................................................................. xiv  
Chapter 1: Introduction ......................................................................................... 1  
Chapter 2: Research Objectives ........................................................................ 14  
Chapter 3: Literature Review and Theory ....................................................... 16  
  Pipe Vibration Literature Review ................................................................. 16  
  CFD Background ............................................................................................ 19  
  Turbulent CFD Theory .................................................................................. 21  
  Boussinesq Eddy Viscosity Model Equations ........................................... 24  
  Turbulence Models ........................................................................................ 26  
    The $k-\omega$ Model .................................................................................... 26  
    Shear Stress Transport $k-\omega$ Model .................................................... 27  
    SAS Turbulent Model ............................................................................... 30  
Chapter 4: Geometry Setup and Meshing ....................................................... 32  
  Mesh Setup .................................................................................................... 34  
    Choke Part .................................................................................................. 42  
    Cone Part ................................................................................................... 45  
    Bulge Part .................................................................................................. 47  
    Thick Helix ................................................................................................. 48  
    Thin Helix ................................................................................................ 53
90 degree to 45 degree Comparison ................................................................. 124
Chapter 8: Conclusions .................................................................................. 125
References ........................................................................................................ 128
Appendix A: Velocity Input UDF .................................................................. 130
Appendix B: Force Output UDF ..................................................................... 132
Appendix C: Force Nature UDF .................................................................... 135
List of Tables

Table 1: Reynolds Numbers for Pipe Sections ................................................................. 62
Table 2: Choke Force Summary ..................................................................................... 93
Table 3: Helix Force Summary ...................................................................................... 100
Table 4: Z-Direction Force Summary ........................................................................... 104
Table 5: Y-Direction Force Summary .......................................................................... 107
Table 6: Out-of-Phase Flow Z-Direction Force Summary ............................................. 111
Table 7: OP Flow Y-Direction Force Summary .............................................................. 113
Table 8: In-Phase to Out-of-Phase Comparison Y-Direction .......................................... 115
Table 9: In-Phase to Out-of-Phase Comparison Z-Direction .......................................... 116
Table 10: 45 Degree Force Summary ............................................................................ 122
Table 11: Comparison of Highest Performing Trials ..................................................... 124
List of Figures

Figure 1: Hydraulic Fracturing Illustration ............................................................... 1
Figure 2: Typical North American Fracturing Fluid Composition ............................ 2
Figure 3: Hydraulic Fracturing Layout (GulfTex Operating Inc) .............................. 3
Figure 4: FMC AFAM High Pressure Flow ............................................................... 4
Figure 5: FMC Technologies Long Sweep Swivel Joint .......................................... 5
Figure 6: Fluid Volume Setup ..................................................................................... 7
Figure 7: Fluid Geometry Summary .......................................................................... 8
Figure 8: Positive Displacement Pump (Commonwealth of Australia, 2012) .......... 9
Figure 9: Flow vs. Crank Shaft Position for Two Triplex Pumps In-Phase ............... 10
Figure 10: Flow vs. Crank Shaft Position for Two Pumps Out-of-Phase ................. 11
Figure 12: Common Pulsation Dampers ................................................................. 17
Figure 13: Best Piping Practices (Taylor, 2006) ....................................................... 19
Figure 11: SST Model (ANSYS FLUENT, 2012) ..................................................... 28
Figure 14: T-Joint Geometry ...................................................................................... 33
Figure 15: T Junctions in Series ............................................................................... 34
Figure 16: Mesh Inflation in T-Part .......................................................................... 37
Figure 17: T-Part Mesh Cross-Section ..................................................................... 37
Figure 18: Straight Pipe Sections ............................................................................ 38
Figure 19: Tetrahedral to Swept Mesh Interface ..................................................... 39
Figure 20: End of Swept Mesh Part ......................................................................... 40
Figure 21: Final Meshed Sections ............................................................................ 40
Figure 45: T-Junction In-Phase Force Output ....................................................... 72
Figure 46: Force Output Inlet Small Sinusoidal Velocity ........................................ 74
Figure 47: Force Output Inlet Large Sinusoidal Velocity ........................................ 75
Figure 48: Total Force Output Inlet Large Sinusoidal Velocity ............................. 76
Figure 49: Sinusoidal Velocity Inlet Pressure ................................................... 77
Figure 50: V Velocity Profile Both Inlets ........................................................... 78
Figure 51: V-Flat Velocity Profile Both Inlets .................................................... 78
Figure 52: V-Parabola Velocity Profile Both Inlets ............................................. 79
Figure 53: V-Circle Velocity Profile ................................................................. 80
Figure 54: Discontinuous Jump Velocity Profile ............................................... 81
Figure 55: Inlet Fluid Velocity and Acceleration ................................................. 82
Figure 56: Highlighted Force Jump ................................................................. 83
Figure 57: T-Junction In-Phase Flow ............................................................... 84
Figure 58: T-Junction Static Pressure Contour ................................................... 87
Figure 59: T-Junction Total Pressure Contour .................................................... 87
Figure 60: Choke 1.5 Lengths ........................................................................ 88
Figure 61: Static Pressure Contour Choke 1.5 Lengths ........................................ 89
Figure 62: Total Pressure Contour Choke 1.5 Lengths ....................................... 90
Figure 63: Choke Trials Y-Direction Force ....................................................... 91
Figure 64: Choke Trial Z-Direction Force ....................................................... 92
Figure 65: Static Pressure Contour Cone .......................................................... 94
Figure 66: Total Pressure Contour Cone .......................................................... 95
Figure 67: Cone Force Data ............................................................................ 96
Figure 91: 45 Degree Z-Direction Force ................................................................. 121

Figure 92: 45 Degree Z-Direction Force Summary Chart ..................................... 123

Figure 93: 45 Degree Y-Direction Force Summary Chart ..................................... 123
Abstract

Pipe structure vibrations resulting from flow variations induced by displacement pumps can be problematic for hydraulic fracturing equipment. Multiple displacement pumps running in parallel are commonly combined in a single manifold to increase flow volume. This system creates a transient periodic flow velocity profile which is a function of the pump crankshaft phase. Specifically, a pipe T-Junction, commonly used in hydraulic fracturing manifolds, with two inlets and one outlet combines the flow of two or more pumps. This piping configuration suffers from vibrations. Research was conducted to study the coupling between flow variations from multiple displacement pumps and vibrations on a T-Junction. It had been observed in the field that adding a fixed orifice to the pipe arrangement reduced vibrations. The research studied the effects of changing inlet flow phase and T-Junction flow geometry with the goal of reducing the mean and variant forces on the pipe structure. Research was done using transient commercial CFD simulations with fine mesh and a recently developed turbulence model, the Scale Adaptive Solutions (SAS) model. Thousands of time steps were used to model 1-2 periods of the flow output from a displacement pump. The mesh elements used were the smallest size (highest mesh count) that the computational resources would allow, on the order of one million nodes. Inlet velocity and outlet static pressure boundary conditions were specified. The forces of interest were in the two directions along the axis of the orthogonal pipes and they were calculated by adding the total force exerted on the walls of the structure with the force caused by the static pressure at the inlets and outlets. Changes to the flow geometry included a fixed orifice,
helix causing rotating flow, an expansion chamber and a cone shape. A Modified T-Junctions was also studied with an inlet entering at a 45 degree angle. These studies were conducted for two inlet scenarios. First, both inlets had identical periodic flow input velocity profiles. Second, the inlets velocity profiles were modified so one inlet was completely Out-of-Phase from the other. Setting the flow inlet to be Out-of-Phase resulted in a reduction of the horizontal forces. The expansion chamber geometry study showed a significant reduction in the forces exerted in the transverse direction and the results showed it was the most favorable geometry for both flow input velocity profiles.
Chapter 1: Introduction

Hydraulic fracturing, henceforth known as fracturing, is a technique of oil well stimulation that has been around since the 1940’s (Halliburton, 2013). Fracturing involves pumping fluid into a rock formation at pressure and flow rates sufficient to cause fractures or fissures in the formation. Sand is suspended in the fracturing fluid which props open the fractures after pressure is relieved. The advent of directional drilling into shale formations has required larger hydraulic fracturing jobs. Figure 1 shows an illustration of a multi-stage horizontal well (left side circled red) vs. a single stage vertical well fracturing job (right side circled yellow) (ALL Consulting, 2012).

Figure 1: Hydraulic Fracturing Illustration
Around 95% of wells drilled in North America today are fractured. The fluid used for fracturing is comprised of mostly water and sand (Figure 2). A fracturing job may use over 120,000 barrels of water (5 million gallons) and may reach pressures as high as 15,000+ psi. Flow rates as high as 100+ Barrels per minute (4,200 gallons per minute) are not uncommon (ALL Consulting, 2012).

![Figure 2: Typical North American Fracturing Fluid Composition](image)

Before the fracturing slurry can be pumped into the formation, it must first be combined into one or more pipes in a manifold. Typically the manifold takes abuse by virtue of the unsteady flow rate exiting the pumps. A manifold typically consist of a semi-truck trailer with two to three large diameter pipes (diameter usually greater than 4 in) running down the length of the trailer. Five pumps could be connected on each side of the unit for a total of 10 pumps. It is not uncommon to have more pumps adding to the flow in a ground manifold upstream of the wellhead.
Water and sand travel from the water and sand trucks to the blender where it is mixed into slurry and pumped, generally with a centurial pump, to the low pressure side of the manifold as is illustrated in Figure 3. Typical output pressures from the blender are in the range of 50-200 psi. This low pressure slurry travels from the manifold to well service pumps (fracturing pumps) where the fluid pressure can be raised to as high as 15,000 psi. From the fracturing pump it travels back to the high pressure side of the manifold where the flows are combined into large pipes that run the length of the trailer. High pressure fluid exits the manifold trailer at the front and travels through pipes assembled on the ground to the wellhead.
An important area of concern is the manifold since it is exposed to vibrations resulting from the flow variations from the fracturing pumps. It is not uncommon for manifolds to experience high loads or vibrations that cause part failures. As a result of the recent power increase of fracturing pumps, vibrations and loads on the manifolds have also increased. Thus, the research in the field of vibration reduction of the manifold was conducted in this thesis.

A recent manifold, the FMC Technologies AFAM, is shown in Figure 4. The yellow line highlights the path the fracturing slurry takes coming from the high pressure output of a pump. Hard pipes are used to withstand the pressure of the slurry. A possible area of high vibration is the T-Joint where the flow is combined to one pipe (shown in red).

Figure 4: FMC AFAM High Pressure Flow

Forces causing vibrations in the manifold come from two different sources. First, some of the forces are transferred through the walls of the pipe structure from the
fracturing pumps. Swivel joints are used in the piping structure to help mitigate the force traveling through the wall of the structure. A swivel joint allows for movement of the pipe structure by allowing rotation of a ball bearing containing joint. This is depicted in the right image in Figure 5. Combining a number of rotational joints with pipe elbows allows for translational and rotational motion. This helps to reduce the vibrations transferred through the pipe walls. The second major source of vibration comes from the changing fluid flow interaction with the walls of the structure. Changing fluid flow causes forces on the walls in the form of both static and dynamic pressure. This study focused on the fluid interaction with the wall of the pipe as the vibration source and explores methods of reducing it. Specifically the pipe structure T-Junction highlighted in red previously was the focus of the research. This specific part of the manifold was chosen as it was thought to be a major contributor to the vibrations of the entire structure.

Figure 5: FMC Technologies Long Sweep Swivel Joint
The configuration of the fluid volume used for the study is seen in Figure 6. This corresponds to the T-Junction used on manifolds today. The interest lies in either finding a change to the flow geometry or the flow input that reduces vibrations. The flow input parameters will be discussed later on separately. Changing the flow geometry comes with restrictions. Any change made to the flow geometry must have a conceivable way of manufacturing the part. It also should not occupy a significant amount of space. Any part made would still need to fit onto a trailer that has both weight and space restrictions. Simply making all parts larger is not an option. Ideally, any change made would either take away a minimal amount of material or be an addition of a part.

It has been observed in the field that placing a fixed orifice (henceforth known as a choke), in the line just before the vertical line attached to the horizontal collection line, reduced vibration in the manifold. The flow direction is shown by the arrows. The choke is circled yellow in Figure 6. This is an example of a feasible change to the geometry because it could be achieved by inserting a choke into an existing geometry.
Figure 6: Fluid Volume Setup

One of the goals of the research was to try and verify the field observations that the addition of the choke reduced the vibration forces on the structure using CFD. If the simulations showed a reduction in vibrations the design was to be optimized. As is discussed in detail later, it was found that the largest vibration forces were found to be in the Y and Z-Directions. The forces were split into these two orthogonal components. The Z-Direction runs down the length of the trailer. All of the figures show the orientation of the parts if one was looking at the side of the trailer. Forces in this direction are aligned down the axis of the large diameter pipe and result in tension or compression of the pipe. Forces in the Y-Direction are transverse to the length of the trailer. As such they create bending loads on the unit which were thought to be more detrimental. For this reason, reducing the force in the Y-Direction was the main focus of
the study but the Z-Direction forces was still examined. Specific terminology, dimensions and mesh methodology had their own challenges and are outlined in the Geometry Setup and Meshing section. Other fluid geometries that were thought to be effective were explored and are shown summarized in Figure 7. Note that the figure just gives a summary of the geometries for reference and they are not to scale.

<table>
<thead>
<tr>
<th>Geometry</th>
<th>Image</th>
</tr>
</thead>
<tbody>
<tr>
<td>T-Junction</td>
<td><img src="image" alt="T-Junction" /></td>
</tr>
<tr>
<td>Choke</td>
<td><img src="image" alt="Choke" /></td>
</tr>
<tr>
<td>Cone</td>
<td><img src="image" alt="Cone" /></td>
</tr>
<tr>
<td>Bulge</td>
<td><img src="image" alt="Bulge" /></td>
</tr>
<tr>
<td>Helix</td>
<td><img src="image" alt="Helix" /></td>
</tr>
<tr>
<td>45 Degree Junction</td>
<td><img src="image" alt="45 Degree Junction" /></td>
</tr>
</tbody>
</table>

**Figure 7: Fluid Geometry Summary**

It was mentioned earlier that the energy needed to fracture rock comes from the pressure and flow rate created by the pumps. The transient output flow from the fracturing pumps is the cause of the fluctuating forces exerted on the pipe that this study examines. As such, the flow profile needs examining as it was an input of the study. Figure 8 shows an example of a piston displacement pump. In actuality fracturing pumps use plungers instead of pistons. For the purpose of this study they are the same.
Mechanically the pumps are similar to an internal combustion engine. They use a slider crank mechanism but unlike an engine, power is input to the pump instead of created as in an engine. Using the slider crank mechanism the flow output can be found with respect to the crankshaft position.

As a result of the high pressure and flow rate needed to fracture a rock formation, a number (10+) of reciprocating plunger pumps with power inputs as high as 3000 hp can be used in parallel to meet the flow parameters required for the formation. One drawback of using reciprocating pumps is that the discharge flow rate from the pump is not constant. Figure 9 shows the combined flow output of two triplex reciprocating pumps (three cylinders) running with their crankshaft positions In-Phase (IP). The example has an 8 inch plunger diameter and a stroke length of 12 in. The flow varies up to $\pm 9.4\%$ from the average flow rate. The pump speed used in the study was 180 rpm. When the flow of two similar pumps flow are combined In-Phase, the percentage of maximum flow variation from the average does not change, but the value
of the variation doubles to 28.3 gal/min. This was shown by the “Total Flow Both Pumps Curve.” If a pump with more cylinders such as a quintuplex pump is used, the flow variation decreases. Note Figure 9 gives an approximation of the pump discharge. It may be affected by valve efficiency and the ratio between the connecting rods to the crankshaft radius. These factors were neglected in order simplify the flow and try to remove the connection to specific products. Using the exact pump flow output would require choosing a specific pump model. The In-Phase flow serves as a worst case scenario for the flow. The section of the curve between the two yellow lines is referenced to as one flow cycle.

![Flow vs. Crank Shaft Position for Two Triplex Pumps In-Phase](image)

**Figure 9: Flow vs. Crank Shaft Position for Two Triplex Pumps In-Phase**

The same pump configuration with the crankshafts completely Out-of-Phase (OP) is shown in Figure 10. The second pump crankshaft is $\pi/6$ ahead of the first pump. This results in a flow with less output variation that could be seen in the total combined flow for both pumps. This OP flow created a total output flow variation of 2.2% from the mean flow, a variation of 6.7 gal/min. It is proposed that flow variation
is responsible for the vibrations and thus IP flows would induce more vibration on a
given pipe structure than OP flows.

![Figure 10: Flow vs. Crank Shaft Position for Two Pumps Out-of-Phase](image)

Changing the phase of the pump crankshafts is a possible way of decreasing
vibrations that was explored. Doing so would not require a change in the T-Junction or
the total flow rate. It is not common to see a device that controls the phase of the pump
crankshaft in comparison to other pumps running on the same parallel circuit. This
control system may not exist at all. It may have been possible for the pumps to go into
and Out-of-Phase freely during a fracturing job. The two crankshaft phases presented
(IP and OP) were only two of an infinite amount of different phase possibilities. They
gave the maximum and minimum output flow variations or what was assumed to be
worst and best case scenarios.

CFD simulations were used to analyze the problem for a few reasons. It made
easier to explore a number of different geometries. If this same study was done
experimentally, long lead times could be expected for manufacturing of parts. There would also be a high cost associated with both manufacturing and the other devices needed for the study (pumps, manifolds etc.). Manufacturing the parts would have severely limited the number of geometries that could be used. Using an experimental setup would make it not feasible to explore the effects of changing the phase of the flow inlets since it is not easy to control pump crankshaft phase. Perhaps one of the largest reasons to run this study using CFD is it would be difficult to isolate vibrations in the structure. As was discussed earlier, manifold vibrations result from both the forces through the wall of the pipes and the forces caused by the fluid. It is unlikely that the two forces could be differentiated in an experimental setup. Thus, direct effects of changing geometry likely could not be obtained experimentally. Additionally, ten or more of the T-Junctions may be on one trailer as well as other vibration causing parts such as pipe elbows. This would induce external vibrations on the T-Junction being studied further complicating the experiment. For these reasons, it was not feasible to study the T-Junction specifically without using CFD simulations.

This research is important to the Oil and Gas industry because excessive vibration forces can result in fatiguing and eventual failure of equipment. Part failure creates a substantial monetary cost resulting from replacing broken equipment and down time. In addition, workers may be close to equipment when it fails. Since the parts contain such high pressure, failure may lead to injury and possibly loss of life. Reducing the vibrations likely translates to reduced equipment failure. The research helps to understand the exact parameters that influence the vibration forces. This could help
define future design constraints. The vibrations had to first be understood before they could be combated. Thus there is a section which examines the forces on the structure resulting from the flow and the turbulent models needed to simulate the flow.
Chapter 2: Research Objectives

The objective of the research was to determine whether or not it was possible to reduce flow induced vibrations in a T-Junction by inserting an orifice into the flow using Computational Fluid Dynamics (CFD) simulations. Based upon field observations, the fixed orifice reduces vibrations on the T-Junction. The effectiveness of the fixed orifice was studied by varying its distance from the main line. Other geometries thought to reduce vibrations more effectively were also explored. Only minor changes in geometry were used because the part would have to be placed on a trailer if it were to go to production. Large changes to the geometry may violate the weight and space restrictions of the trailer and thus would not be feasible. Data resulting from this study may allow companies to make changes to part geometries that would be both feasible from a manufacturing standpoint and may create a competitive edge to their products. The CFD models used were the most accurate and complete ones available with the technology readily available at the time of the study. The mesh size in the areas though to have the highest velocity gradients was 0.0984 in. The highly detailed mesh resulted in a node count used was around one million and about one thousand time steps were used for each flow cycle. The high element count allowed the high end SAS turbulence model to be used. The goal was to try and reduce pipe vibrations as much as possible by either making small changes to the geometry or by changing the flow from In-Phase to Out-of-Phase. Changing the flow phase did not affect the time averaged flow rate, only the instantaneous flow velocity. Additionally, a
better understanding of the nature of the resulting forces and what parameters of the flow caused them was found.

The T-Junction shown in Figure 4 was the only part of the flow structure modeled. Studying the T-Junction only allowed direct cause and effect of a relatively simple geometry changes that could be analyzed. Furthermore, limiting the geometry to a small region allowed for a high ratio of the flow length scale to the mesh size. This also made it possible to use the most accurate but computationally expensive turbulence models possible with today’s technology. The CFD models output the force exerted on the walls. The force on the walls added to the pressure force on the inlets and outlets was taken as the total force output on the structure and was used to compare different geometries.

Successfully reducing the force on the T-Junction would reduce the vibrations of an entire manifold. This could lead to a reduction in equipment failure, which directly results in reduced monetary losses. Since personal is likely to be close to the manifold during operation, preventing equipment failure may also prevent injury.
Chapter 3: Literature Review and Theory

Pipe Vibration Literature Review

Pipe vibration has long been a problem for piping structures of reciprocating pump systems. Engineering Dynamics Incorporated has done a number of experiments studying the effects of varying flows on the vibrations of pipe structures. They have published a number of papers outlining the usefulness of dampers which are publically available on their website (Engineering Dynamics Incorporated, 2009). Figure 11 shows a number of different common pulsation dampers used in conjunction with reciprocating pumps. It is common to see dampers of this sort on the inlet of a well service pump (low pressure side), but not on the outlet (high pressure side). This is partly attributed to it being difficult to design and manufacture a damper that can withstand 15,000 psi. This is because a damper containing a rubber gas filled bladder as rubber typically has tensile strength that is only slightly higher (<20,000psi) than the working pressure.
It is common to see piping failure in cases where the flow pulsations coincided with the natural frequencies of the piping structure which can amplify the vibration on the system by up to 800 times. Dampers can be effective at damping out acoustic pulsations, which travel at the speed of sound in the fluid. Generally, well service pumps have output flow pulsations of less than 15-20 Hz. Reciprocating pumps generate pulsations at integer multiples of its operating frequency, but the higher the frequency
multiplier the lower the amplitude of the force (Engineering Dynamics Incorporated, 1988).

Placing a choke in the piping system acts as a resistive, or pressure drop device. However, the research showed placing a choke at the discharge of a displacement pump caused the static pressure to rise upstream of the choke. This raised the overall pressure of the system which may be problematic. This type of damper is said to be most effective at damping high frequency modes (Engineering Dynamics Incorporated, 1988). This study assumes that the higher frequency modes were of negligible magnitude compared forces induced at the pumping frequency and thus ignored higher frequency modes. Though it should not be ignored that higher frequency pulsations could make a significant contribution to the forces if they were close to the natural frequencies of the system. Unfortunately, the common designs for pulsation dampers used volumes to help attenuate the pulsations. The objective was to not severely influence the flow geometry, so these concepts were not feasible.

It has long been observed that there are preferred pipe junction arrangements to help prevent vibrations from occurring. These are laid out in handbooks and articles such as in Figure 12 (Taylor, 2006). The flow arrangement of the junction being examined matched with the Typical T. Ideally it would be angled. This is shown in the preferred arrangement, but other design constraints make this not realistic in many cases.
Due to the competitive nature of the oil industry, a large amount of the studies and research conducted around the field of pipe vibration is funded by companies. As such, research previously conducted may not have been made available to the public and was kept was proprietary. It was difficult to find information relating to pipe vibration in this context so no background studies or experiments are presented. There might have been more information and research that has been conducted in the private sector which is not public information.

**CFD Background**

Advances in computational resources made it possible to examine the effects of turbulence at high Reynolds Number flows. However, there are still limitations present with turbulent simulations. Many of these stem from the fact that turbulent flows are
inherently transient. A large number of time steps had to be used to resolve the characteristics of a turbulent flow.

The issue of mesh count is also critical in the case of turbulent flows. As a result of the vorticity equation,

\[
\frac{D\omega}{Dt} = \nu \nabla^2 \omega + \omega \cdot \nabla U
\]  

(1)

which is obtained by taking the curl of the Navier Stokes equations, comes a term known as vortex stretching. This term is inherently three dimensional, and thus mandates a three dimensional flow (Pope, Turbulent Flows, 2000). For turbulence modeling, even symmetric fluid geometries need to be completely modeled in three dimensions, without using symmetry planes to properly resolve the flow characteristics. For CFD, this means modeling the entire flow domain which increases the computational resources required.

Turbulent flows require a fine mesh scale to resolve the flow features. The Kolmogorov’s scales used to determine the number of three dimensional mesh elements necessary to directly resolve the Navier Stokes equations scaled as \( Re^{9/4} \) (Pope, Turbulent Flows, 2000). Pipe flow in the case of hydraulic fracturing commonly saw Reynolds number on the order of \( 10^6 \) which would require a mesh count on the order of \( 10^{13} \) to fully resolve the flow. As a reference, this was about 500 times as many grid points as the largest ever simulation completed by Japan’s Earth Simulator (Ishihara,
Gotoh, & Kaneda, 2009). Clearly it is currently not feasible to fully model all of the flow scales, so a turbulence model was used. As a result, some of the accuracy had to be sacrificed to obtain results in a timely manner. It was assumed that turbulence models would provide accurate results for the parameters of interest. Most if not all industrial turbulent flows use models to reduce the computational resources required. This is a statement to their accuracy with regards to most engineering problems.

**Turbulent CFD Theory**

All fluid motions are governed by the dynamical equations for the fluid. Written in Cartesian tensor notation, the Navier Stokes equations are

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho u_i}{\partial x_i} = 0 \quad (2)$$

$$\frac{D U_i}{Dt} = \mu \frac{\partial^2 U_i}{\partial x_i \partial x_j} - \frac{1}{\rho} \frac{\partial p}{\partial x_j} \quad (3)$$

Note that ANSYS FLUENT uses the integral form of the momentum equations but they are given in differential form in this document for simplicity.

This study used constant density which results in the conservation of mass equation to be simplified as
\[
\frac{\partial u_i}{\partial x_i} = 0
\]  

(4)

As a result of the equation above being a set of nonlinear second order partial differential equations, no analytical solution has been found. The equations may be solved numerically by means of Direct Numerical Simulation, but the computational cost for any realistic flows is too high to make this a viable option (Pope, Turbulent Flows, 2000). To try and combat the high computational cost, the Reynolds decomposition,

\[
u_i = \bar{u}_i + u'_i
\]

(5)
can be taken. This makes it possible to examine the statistics of the flow, as opposed to the instantaneous values.

\[
\frac{\overline{D\langle U_j \rangle}}{Dt} = \mu \frac{\partial^2 \langle U_j \rangle}{\partial x_i \partial x_j} - \frac{1}{\rho} \frac{\partial \langle p \rangle}{\partial x_j} - \frac{\partial \langle u_i u_j \rangle}{\partial x_j}
\]

(6)

which are known as the Reynolds equations or Reynolds Averaged Navier Stokes equations (RANS). They appear similar to the Navier Stokes equations except for the last term in equation (6). This term comes as a result of Reynolds Averaging.

In order to examine the final term in the Reynolds equation, (8), it can be re-written as
\[
\rho \frac{\partial \langle u_j \rangle}{\partial t} = \frac{\partial}{\partial x_i} \left[ \mu \left( \frac{\partial \langle u_i \rangle}{\partial x_j} - \frac{\partial \langle u_j \rangle}{\partial x_i} \right) - \langle p \rangle \delta_{ij} - \rho \langle u_i u_j \rangle \right] \tag{9}
\]

which is the general form of the conservation of momentum equation. \(\delta_{ij}\) is the Dirac Delta Function. Taking the time average of this equation makes the time component in the material derivative drop out as its average is defined to be zero. The term in the square brackets represents the sum of the three stresses; the viscous stress, the isotropic stress and the apparent stress results from fluctuation in the velocity field. The apparent stress, \(\langle u_i u_j \rangle\), is commonly known as the Reynolds Stress. This term results in what is referred to as the closure problem. Four equations were available, but more than four unknowns exist. Consequently the set of equations cannot be solved. This is referred to as an unclosed system of equations.

The Reynolds Stress term is explored as it is the problem term. \(\langle u_i u_j \rangle\) is a second order symmetric tensor which results in 6 unknowns within the term. The main diagonal components of the tensor are known as the normal stress, while the off diagonal components are known as the shear stresses. The normal stresses is defined as the turbulent kinetic energy,

\[
k \equiv \frac{1}{2} \langle \mathbf{u} \cdot \mathbf{u} \rangle = \frac{1}{2} \langle u_i u_i \rangle \tag{10}
\]

As the tensor was defined, the difference between a normal stress and a shear stress depended upon the coordinate system that is defined. A distinction could be made between the two stresses in the form of isotropic stress and anisotropic stress.

23
The anisotropic stress is

$$a_{ij} \equiv \langle u_i u_j \rangle - \frac{2}{3} k \delta_{ij} \quad (11)$$

Normalizing the tensor with respect to $k$ yields

$$b_{ij} = \frac{a_{ij}}{2k} = \frac{\langle u_i u_j \rangle}{\langle u_i u_i \rangle} - \frac{1}{3} \delta_{ij} \quad (12)$$

The Reynolds stress tensor can then be written as a sum of the isotropic and anisotropic parts

$$\langle u_i u_j \rangle = 2k \left( \frac{1}{3} \delta_{ij} + b_{ij} \right) \quad (13)$$

It can be found that the only part of the component effective for the transport of momentum is the anisotropic component, $a_{ij}$ (Pope, Turbulent Flows, 2000). The isotropic part is written as a part of the mean pressure to give

$$\rho \frac{\partial \langle u_i u_j \rangle}{\partial x_i} + \frac{\partial \langle p \rangle}{\partial x_j} = \rho \frac{\partial a_{ij}}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \langle p \rangle + \frac{2}{3} \rho k \right) \quad (14)$$

There are a number of ways to model the Reynolds stress term. Some of these will be explained in the next section.

**Boussinesq Eddy Viscosity Model Equations**

The Reynolds Stress tensor is denoted as $R_{ij} = -\rho(u_i u_j)$. It can be closed in two ways, using the Reynolds Stress Models (RSM) or using Eddy Viscosity Models.
(Boussinesq Models). Some of the Eddy or Turbulent Viscosity Models are explored here. This type of modeling relies on the turbulent viscosity hypothesis, which has two parts. First there is the intrinsic assumption that the Reynolds stress anisotropy tensor, given in $a_{ij}$, is determined by the mean velocity gradient, $\frac{\partial u_i}{\partial x_j}$. The previous assumption may not be adequate to resolve some complicated flows, but it may be sufficient for simple turbulent shear flow: boundary layers, round jets, mixing layers channel flows, etc. (ANSYS FLUENT, 2009).

On a historical note, it was interesting to see that Boussinesq published his relationship almost 20 years before the term “Reynolds Averages” was ever mentioned. He made his argument with regards to local averages of the flow, which possibly inspired Reynolds (Schmitt, 2007).

The second specific assumption is the relationship between $a_{ij}$ and $\frac{\partial u_i}{\partial x_j}$ given by the Boussinesq viscosity relation

$$\langle u_i u_j \rangle = -\rho \frac{2}{3} k \delta_{ij} + \mu_t \left( \frac{\partial \langle u_i \rangle}{\partial x_j} + \frac{\partial \langle u_j \rangle}{\partial x_i} \right) - 2 \frac{2}{3} \mu_t \frac{\partial \langle u_k \rangle}{\partial x_k} \delta_{ij}$$

(15)

Note that $\mu_t$ is a property of the flow, not of the fluid which will be discussed later. This assumes that the turbulent viscosity is an isotropic scalar quantity (Pope, Turbulent Flows, 2000).
Turbulence Models

The $k - \omega$ Model

The $k - \omega$ turbulence model was originally proposed by Wilcox (Wilcox D., 1988). This model is similar to the $k - \varepsilon$ model which can be found in the literature (Pope, Turbulent Flows, 2000). There are still two equations being solved including the same transport equation for the turbulent kinetic energy. The transport equation for the turbulent dissipation rate, $\varepsilon$, is replaced with a transport equation for the specific dissipation rate, $\omega$.

\[
\frac{\partial k}{\partial t} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] - \rho \beta^* f_\beta k \omega + \tau_{ij} \frac{\partial \bar{u}_i}{\partial x_j} \tag{16}
\]

\[
\frac{\partial \omega}{\partial t} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right] + \frac{\omega}{k} \tau_{ij} \frac{\partial \bar{u}_i}{\partial x_j} - \rho \beta f_\beta \omega^2 \tag{17}
\]

where $\mu_t \approx \frac{\varepsilon}{k}$, $\beta = \frac{3}{40}$, $\beta^* = \frac{9}{100}$, $\sigma_\omega = \sigma_k = \frac{1}{2}$ and $\varepsilon = \beta^* \omega k$. The turbulent viscosity for this model is computed from $\mu_t = \frac{\alpha^* \rho k}{\omega}$ where $\alpha^* = \alpha_\infty \left( \frac{\alpha_0^{*} + Re_T/R_k}{1 + Re_T/R_k} \right)$, $\alpha_0^* = \frac{\beta_l}{3}$, $\beta_l = \frac{9}{125}$, $R_k = 6$, $Re_T = \frac{\rho k}{\omega \mu}$ and $\alpha_\infty^* = 1.0$. As can be seen in the relationship for $\alpha^*$, as $Re_T$ approaches infinity, the turbulent viscosity approaches 1 which helps to recover the Chou relationship for $\mu_t$. For homogeneous turbulence, the equations turn out equivalent (Wilcox D. C., 1998). In general this model provides physically accurate near wall conditions, which results in it being applicable in wall bounded flows as well as free shear flows (ANSYS FLUENT, 2009).
Shear Stress Transport $k - \omega$ Model

The SST $k - \omega$ model is a popular model that has become one of the most popular models. One reason for this is it provides a compromise between the $k - \omega$ and the $k - \varepsilon$ model where the strengths of both models are utilized. It has been noted the $k - \omega$ model is overly sensitive to free stream flows, while the SKE is not (Menter F., 1994).

As is shown in Figure 13, the model was split into two separate parts depending upon what region of the flow the calculations were being carried out. In the outer layer of the flow the calculation uses a method similar to the standard $k - \omega$ model with some constants changed.

\[
\frac{\partial k}{\partial t} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} + \tau_{ij} \frac{\partial U_i}{\partial x_j} - \beta^* \rho \omega \right] \tag{18}
\]

\[
\frac{\partial \omega}{\partial t} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} + \frac{\gamma_1}{v_t} \frac{\partial \bar{U}_i}{\partial x_j} - \rho \beta_1 \omega^2 \right] \tag{19}
\]

where $\beta_1 = 0.075$, $\sigma_{k_1} = 1.176$, $\sigma_{\omega_1} = 2.0$, $\beta^* = 0.09$, $\gamma_1 = \frac{\beta_1}{\rho} - \kappa^2 / (\sqrt{\beta^* \sigma_{\omega_1}})$ and $\kappa = 0.41$. The difference in this model and the standard $k - \omega$ model is the change in the form of the turbulent viscosity,

\[
\mu_t = \rho \frac{a_1 k}{\max(a_1 \omega, \Omega F_2)} \tag{20}
\]
where $F_2 = \tanh(\arg_2)$ and $\arg_2 = \max \left( \frac{2\sqrt{k}}{0.09\omega}, \frac{500\nu}{y^*\omega} \right)$. The outer layer of the model equations are obtained from performing a change of variable on the Standard $k - \varepsilon$ (SKE) equation for $\varepsilon$, $(\omega \equiv \varepsilon/k)$ (Pope, Turbulent Flows, 2000).

$$
\rho \frac{D\omega}{Dt} = \frac{\partial}{\partial x_i} \left[ \left( \mu + \mu_t \right) \frac{\partial \omega}{\partial x_j} \right] + \frac{\gamma_2}{v_t} \tau_{ij} \frac{\partial \tilde{u}_i}{\partial x_j} - \rho \beta_2 \omega^2 + \frac{2\sigma_{\omega2} \rho_1}{\omega} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_j} \tag{21}
$$

where $\beta_2 = 0.0828$, $\sigma_{k2} = 1.0$, $\sigma_{\omega2} = 1.168$, $\beta^* = 0.09$, $\gamma_2 = \frac{\beta_2}{\beta^*} - \kappa^2 / (\sqrt{\beta^* \sigma_{\omega2}})$ and $\kappa = 0.41$. The equation for $k$ stays the same for the layers away from the walls.

![Figure 13: SST Model (ANSYS FLUENT, 2012)](image)

The outer layer computed the turbulent viscosity from

$$
\mu_t = \rho \frac{k}{\omega} \tag{22}
$$

There is the issue of blending the equations in the region common to both the inner wall region and the outer wall regions. This is accomplished by adding together the equations for the turbulent kinetic energy to create the form

$$
F_1 \left[ \rho \frac{Dk}{Dt} + \ldots \right]_{\text{inner}} + (1 - F_1) \left[ \rho \frac{Dk}{Dt} + \ldots \right]_{\text{outer}} \tag{23}
$$
where \( \phi = F_1 \phi_1 + (1 - F_1) \phi_2 \), \( F_1 = \tanh(arg_1^4) \),

\[
arg_1 = \min \left[ \max \left( \frac{k^{1/2}}{\beta^* \omega y'}, \frac{500 \nu}{\nu^*} \right), \frac{4 \rho \sigma_{\omega2} k}{C_{\nu \kappa} \nu' \omega} \right], \quad CD_{\kappa \omega} = \max \left( 2 \rho \sigma_{\omega2} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}, 10^{-20} \right).
\]

This implies that \( F_1 = 1 \) for the inner layer and \( F_1 \to 0 \) for the outer layer. Resulting from the blending equations, the equation for \( \omega \) takes the form

\[
\rho \frac{D \omega}{D t} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right] + \tau_{ij} \frac{\gamma}{\nu_t} \frac{\partial u_i}{\partial x_j} - \rho \beta_2 \omega^2 + 2 \cdot \rho (1 - F_1) \sigma_{\omega2} \left( \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \right) \tag{24}
\]

It is possible that a flow that contained a sufficiently large adverse pressure gradient, the production could grow much larger than the dissipation. This could lead to a large turbulent stress. It was found that one could use a limiter in the form

\[
P_k = \mu_t \frac{\partial u_i}{\partial x_j} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_j} \right) \quad \text{and} \quad \tilde{P}_k = \min (P_k, 10 \beta^* \rho k \omega) \tag{25}
\]

(26) allows a limit to be placed on the turbulent viscosity in stagnant regions (F.R. Menter, 2001). Using the \( k - \omega \) model provides performance superior to the SKE class of models in the region near a wall boundary. The previous formulation also does well with an adverse pressure gradient. However, it generally predicts flow separation early for transitional flows. Due to the extra calculations needed for the blending of the two equations, this model is more computationally expensive than the other RANS models.

The models discussed suffer from the same assumption of the Boussinesq hypothesis that allows the turbulent viscosity to be modeled as an isotropic scalar quantity. It is known that these assumptions are not exactly true, and this may be showed depending on what is to be gained from the simulation. Simple flows obtained...
good results with the RANS models and they have the advantage of being relatively computationally inexpensive (ANSYS FLUENT, 2009).

**SAS Turbulent Model**

The Scale Adaptive Solutions (SAS) turbulence model was created to resolve turbulence at smaller scales than RANS models could predict. The SAS model is based upon the SST $k - \omega$ model that is sensitive to unsteady flow fluctuations. This is done by adding a production term to the $\omega$ equation. It provides a midway point between the RANS models and a Large Eddy Simulations (LES). The SAS model has become relevant in recent years as the result computational technology improvement (Davidson, 2006).

LES directly models the turbulent kinetic energy of the large eddies at scales down to the grid size. If there are eddies that are relevant to the total flow at sizes smaller than the grid, the model may have not properly describe the flow. Fundamentally the LES created an artificial dissipation size at the size of the grid cutoff, which may have been orders of magnitude larger than the viscous dissipation scale. This is especially true for large Reynolds number flows (Menter F. R., 2009).

The SAS turbulence model provides corrections to the overly dissipative nature of the SST $k - \omega$ model by increasing the production of $\omega$. This effectively lowers the turbulent viscosity, $\mu_t$. The specific form of the SAS model used in ANSYS FLUENT version 14 has the form
\[
\frac{\rho D\omega}{Dt} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \mu_t \right) \frac{\partial \omega}{\partial x_j} \right] + \frac{\nu_t}{\nu} \tau_{ij} \frac{\partial u_i}{\partial x_j} - \rho \beta_t \omega^2 + (1 - F_\lambda) \frac{2\rho}{\sigma_{\omega,2}} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} + Q_{sas} \tag{27}
\]

The additional source term, \(Q_{sas}\), that appears in the equation is intended to negate the overly dissipate nature of the model. The new source term has the form

\[
Q_{sas} = \max \left[ \rho \eta_2 \kappa S^2 \left( \frac{L}{L_{vk}} \right)^2 - C \frac{2\rho k}{\sigma_o} \max \left( \frac{1}{\omega^2} \frac{\partial \omega}{\partial x_j} \frac{\partial \omega}{\partial x_j} , \frac{1}{k^2} \frac{\partial k}{\partial x_j} \frac{\partial k}{\partial x_j} \right) , 0 \right] \tag{28}
\]

where \(L_{vk}\) is the new von Karman Lengths Scale (ANSYS Fluent, 2011).

\[
L_{vk} = \kappa \frac{S}{|U''|} \tag{29}
\]

where

\[
|U''| = \sqrt{\sum_{(i)} \left( \frac{\partial^2 u_i}{\partial x_j \partial x_j} \right)^2} \tag{30}
\]

includes a second velocity gradient. In areas of the flow where the SAS production term is activated, (27) will increase which decreases the turbulent viscosity locally.
Chapter 4: Geometry Setup and Meshing

The initial T-Junction geometry used came from an existing hydraulic fracturing manifold. Initial choke experimentation was driven by rumors that placing the choke in the flow, close to where the pipes are combined, helps to reduce vibrations. The information of the exact spacing from the main lines was not available, so different spacing was tried. Figure 14 shows the general flow geometry used. Note that all geometry modeled is the flow geometry, not the solid pipe wall. This corresponds to empty space on the real parts. The horizontal pipe parallel with the Z-Axis has a diameter of 4 inches and will be known as the main or large line. The vertical pipe aligned with the Y-Axis had a diameter of 2.75 inches. The direction of the flow is indicated by the arrows. The inlets on the 2.75 and 4 inches pipes shall henceforth be known as the small and large inlets respectively. The portion shown in gray shall be known as the T-Part. The pipe between the large inlet and the T-Part was significantly shorter than the pipe downstream of the T-Part. This was because the flow was assumed to be fully developed turbulent flow coming from the inlets. It was assumed that the forces on the pipe walls were negligible in the regions of fully developed flow. By making this part of the geometry short, total node count was reduced.
The portion of the geometry downstream of the T-Part was made long in attempt to achieve fully developed flow at the outlet. The pipe between the small inlet and the T-Part was made long to accommodate for placement of chokes or other flow changing devices. The geometry was split up into multiple sections to allow for different mesh parameters to be used in regions of varying turbulent parameters. This was done in attempt to reduce the mesh node count to an acceptable level for the computational resources available.

To put the geometry into context, a number of these geometries could have been put in series when being used in the field. Figure 15 shows two of these parts put in series but there may well be five or more placed into series on a service manifold. The
reason for the T-Junctions is to combine the flow from multiple pumps into one pipe. Only one T-Junction was modeled to simplify the problem constraints and to reduce the total mesh count of the geometry.

![Figure 15: T Junctions in Series](image)

**Mesh Setup**

All parts were meshed in a similar manner. First the T-Part was meshed using tetrahedral elements. This was done because it was not easy to use a more desirable mesh on the part, such as a sweep element, because of its geometry. A sweep element is an extrusion of usually either a triangle or a rectangle to form a three dimensional shape. They are desirable because they may have low skewness and can easily be aligned to the direction of the flow for a pipe. It is usually preferable align the mesh in the direction of the flow. Since the T-Part mixed the two pipes, the flow was not moving in one general direction. This made it not feasible to align the mesh in the flow direction. The patch conforming algorithm was used which allowed for nodes to intersect on faces where one mesh section came in contact with another. The tetrahedral cells were meshed with an element size of 0.0984 in with a soft behavior setting. This
made the size be enforced as a guideline, not a hard setting. Setting this to soft allowed for slightly varying element sizes, which reduces skewness of some elements especially near boundaries. The growth rate was set to a strict 1.04. This implied the ratio between adjacent element sizes may have only been a great as the growth rate.

Since one of the desired outputs was the force on the walls of the structure, it was important to have a refined mesh near the walls. Wall inflation was used in attempt to make the near wall region more accurate. It was recommended by ANSYS FLUENT to use a $y^+$ value of $30 < y^+ < 300$ for models where viscous forces in the near wall region was not paramount (ANSYS FLUENT, 2012). An early assumption made was that pressure forces would dominate the vibrations of the structure. Exact modeling of the viscous region was not paramount. Moreover, resolving the near wall region would require the first layer height to be decreased by 1-2 orders of magnitude (resolving the near wall region required the $y^+$ value to be about 1). To avoid highly elongated cells, the other dimensions of the cells would have had to be reduced as well. This would greatly increase the mesh count near the wall. Keeping a reasonable growth rate, this would in turn greatly increase the overall mesh count to a size that the computational resources would not allow.

The first layer height was calculated using

$$y = \frac{y^+ \mu}{u_t \rho}$$  \hspace{1cm} (31)
where $y^+$ was the wall units or viscous sub length and $u_\tau$ was the friction velocity (Pope, Turbulent Flows, 2000). The friction velocity was calculated from

$$u_\tau = \sqrt{\frac{\tau_w}{\rho}}$$

(32)

$\tau_w$ is the shear stress at the wall and was found using

$$\tau_w = \frac{1}{2} \cdot C_f \rho \cdot U_{\infty}^2$$

(33)

where $C_f$ was known as the fanning friction factor (White F., 2006). For a round pipe the friction factor was calculated using

$$C_f \approx 0.0791 / Re_D^{0.25}$$

(34)

which was valid for fully turbulent flow in a round pipe (Blasius, 1913). Using the equations outlined above and a $y^+$ value of 140 it was found that the first layer height needed to be 0.015 in. The value of $y^+$ was chosen because it was in the acceptable range given by ANSYS and the assumption that the viscous sub layer would not substantially affect the results.

The inflation was controlled using the first layer height and the total number of inflation layers which was set to 12. The growth rate of the mesh layers was set to 1.03 to ensure a small change in the volume of the elements in the direction perpendicular to the walls. Figure 16 shows a cross-section of the mesh wall inflation region that was representative of the entire domain near the walls. Note the mesh was created first in the
near wall region and the tetrahedral mesh in the center volume of the pipe was created second. The nodes in the inflation region conformed to the tetrahedral volume nodes.

Figure 16: Mesh Inflation in T-Part

The final meshing of the T-Part contained 153,583 nodes and 586,363 elements with a maximum skewness of 0.82. Figure 17 showed a cross-section of the mesh at the T-Part. The T-Part was made small to reduce the number of tetrahedral cells in the overall geometry as they were not as desirable as other the swept element types.

Figure 17: T-Part Mesh Cross-Section

The next parts that were meshed were highlighted green and are shown in Figure 18. These were all straight pipe sections which allowed them to be meshed using a
swept mesh. They were meshed using the face shared with the T-Part as a source. In other words, the mesh on the respected face of the T-Part was extruded to form the swept mesh. This type of mesh required the T-Part to be meshed first as every subsequent mesh was determined by it. The part to the left of the T-Part was made shorter than the other two adjacent parts in the figure. It was assumed that little flow activity would take place upstream of the T-Part on the main line. This allowed the section to be made short to limit the node count.

Figure 18: Straight Pipe Sections

A section of the Tetrahedral to swept mesh interface is shown in Figure 19. The inflation region set by the T-Part carried over to the rest of the mesh. Thus all wall inflation regions were held constant across the domain without setting it in each region. Cells in the center volume of the swept parts were all pentahedrons.
The elements in the swept regions highlighted in green in Figure 18 contained elements with a maximum face length of 0.0984 in, the same face length as the T-Part. This helped to preserve the size of the elements in the transition between the different mesh types. However, the tetrahedral elements had less volume than a pentahedral element of the same face length. The number of nodes was not decreased by the same ratio. ANSYS FLUENT calculates at the nodal points even though it is a finite volume code. There was then an interpolation between the nodal points to fill the volume. This resulted in the most important factor for the calculation domain size being the number of nodes, not cells. Therefore, apparent volume of the cell did not directly relate to the accuracy of size of the computational domain of the study.

Cutting the part anywhere along the main line using a plane parallel with the face highlighted in green in the figure would result in the same image of the mesh, which implies extrusion. This is illustrated in Figure 20.
The sections farthest away from the T-Part, highlighted green in Figure 21, were meshed using a sweep with the same target settings. The only difference was the maximum face size was allowed to be twice that of the T-Part, 0.197 in. This was done because it was assumed that the flow had either been fully developed or was close enough that it would not significantly affect the force output on the structure.
The total node and element count for the entire mesh was 1,085,882 and 1,846,199 respectively. The maximum skewness was the same as that of the T-part, 0.82. This same meshing strategy was used on all of the different geometry configurations. The only differences came from the local geometry unique to the different configuration. The length of the element in the Z-Direction was twice as long as the elements shown in grey, which were in the section closest to the T-Part. This is shown in Figure 22.

![Figure 22: Final Meshed Section](image-url)
**Choke Part**

The geometry for the standard choke was taken from a production choke that was being used in the field. Figure 23 shows an image of the standard choke that was modeled. The image on the left shows a profile of the outside of the choke and a view of its outlet. An O-Ring type seal would have been placed in the grove near the top of the left image and the bottom of the part (not shown). The image on the right of Figure 23 gives a view looking down the flow bore of the choke in the direction of the fluid flow.

![Figure 23: Standard Choke](image)

Fluid entered the choke from the left and exited from the right side as it is oriented in Figure 24. The contraction part of the choke reduced the cross-sectional area at a smaller rate than that of the increase in cross-sectional area present in the expansion part. A round with a radius of 1 inch also was present in the transition to the expansion. The long length of the choke design may have been used to increase the erosion life of the part. The exact design parameters and usage of the choke was not known. The choke reduced the cross sectional area of the pipe by 5.43 times.
Interest laid in determining if the distance the choke was from the main line would affect the vibrations in the structure. The distance from the outlet of the choke to the center of the flow bore of the main line was varied. This was shown by the dimension line highlighted yellow in Figure 25. This distance was non-dimensionalized using

\[ D = \frac{l}{l_c} \]  \hspace{2cm} (35)

where \( l_c \) was the total length of the choke, 7.85 in. Figure 25 showed the choke 1 length away from the main line.
Figure 25: Choke Location

The geometry for the choke allowed it to be swept using the same settings that were used in the other straight pipe sections. The swept mesh profile was conserved through the choke section, which resulted in the cells being compressed. This caused smaller elements within the throat of the choke in both the inflation region and in the center volume. Smaller elements were created within the choke region, which was desirable. This region had higher velocity due resulting from its decrease in cross-sectional area. The smaller elements provided a more accurate flow description. Figure 26 shows elements reduced proportionately to the cross-sectional area of the choke.
It was desirable to design the cone in such a way that there would be a minimal change in the flow geometry which would prevent the pressure containing parts from undergoing a major design overhaul. The cone geometry was made in attempt to study the vibrations in the Y-direction by reducing the flow velocity coming from the small inlet. The cross-sectional area of the part increased from the inlet to the outlet by 39.6%, thus decreasing the bulk flow velocity by the same amount. This increased the diameter of the small inlet to the T-part to 3.25 inches. Figure 27 shows the Cone geometry highlighted in green. The length of the part was 6 inches. This resulted in an increase of the cross sectional area of the part at a rate of 0.041667 inches per inch length of the cone in the Y-Direction. This small rate of increase in cross-sectional area
was used to minimize wall separation of the flow which may have produced undesirable results.

Figure 27: Cone Part Geometry

The cone mesh was able to be swept in a similar way to the choke. The sweep allowed the elements to automatically be reduced in size in the pipe sections with the smaller cross-sectional area and increased velocity as was shown in Figure 28.

Figure 28: Cone Sweep Mesh
Bulge Part

The bulge part was made with the intent of enlarging the volume for the two pipes to mix. Doing this helped to prevent the stream of high velocity fluid from the small inlet from coming in contact with the bottom side of the T-Part. Figure 29 showed the geometry of the bulge part which replaced the T-Part. The diameter of the part was increased from 4 to 4.5 inches. This gave an overall increase in cross-sectional area of 21%.

Figure 29: Bulge Part Geometry

The part was meshed using a tetrahedral mesh and identical inflation settings as was used on the previous T-Parts for comparability. Figure 30 shows the mesh of the bulge part close to the inflation layer. The addition of the extra volume of the choke increased the mesh count of the bulge part to 163,591 nodes and 677,154 elements.
changes in the geometry raised the maximum skewness ratio to 0.91. This was substantially higher than the normal T-Part produced. It would be possible to refine the mesh in the areas with the highest skewness but this would likely involve increasing the mesh count. ANSYS FLUENT could still converge cases with much higher skewness than this therefore the mesh was not refined.

Figure 30: Bulge Mesh

Thick Helix

The thick helix part was first created using under the premise of both decreasing the cross-sectional area of the flow as well as adding a rotation to the flow in the hoop direction of the pipe (perpendicular to the direction of the flow). Originally the helix was designed to reduce the cross-sectional area of the flow by the same amount as the choke. This however was infeasible when creating the geometry. Figure 31 shows the flow geometry of the helix part. The solid part that would create this flow geometry
could be imagined as a flat plate that had been twisted into a helix. The helix had a pitch of 10 in.

Figure 31: Thick Helix Flow Geometry

A view looking down the flow bore of the part can be seen in Figure 32. Note the visible part represented the flow geometry, and empty space denoted solid material of the part.

Figure 32: Thick Helix Flow Bore
The helix part was created by extruding as a helix with a 10 in. pitch the geometry dimensioned in the left image of Figure 33. This reduced the cross-sectional flow area of the helix to 5.38 in², a decrease of 9.4% over the full bore pipe. The sharp edges of the part were rounded in attempt to decrease the amount of skewed cells in that region. This can be seen highlighted in Figure 33. All of the rounds had a radius of 0.1 in. for all of the helix parts. Both helix parts were 2 approximately choke lengths upstream of the center line of the main line.

![Figure 33: Helix Geometry (left) and Helix Round Surfaces (right)](image)

The helix geometry created a few challenges in the meshing process. As a result of the complex geometry it was not feasible to create an inflation region in the helix part. In attempt to still produce accurate results near the boundary layer, the entire mesh...
of the part was made of tetrahedral cells with a maximum edge length of 78.74 thousandths of an inch. Reducing the mesh to this size greatly increased the count of the entire part.

![Helix Part Mesh](image)

In attempt to decrease the drag force at the inlet and outlet of the thick helix, a splitter was added. The splitter shown highlighted in yellow in Figure 35 extends 1.2 inches in the flow direction and has an angle of 11.31 degrees. The base of the splitter, parallel to the Z-Axis had the same thickness as the helix plate, 0.2 in. The mesh of this part also had the inherent challenges relating to the boundary layer inflation that the helix part had. This part was meshed using the same tetrahedral mesh as the helix, but used a smaller maximum edge length of 74.8 thousandths of an inch. The total length of the part in the Y-Direction was 0.5 inches.
Figure 36 shows the mesh of helix splitter. The mesh interface between the helix and the splitter was set as patch independent to reduce the skewness of the parts. This resulted in the nodal points not being forced to match at the interface between mesh types. However they were matched if possible. This practice was not ideal and should have not been used unless elements with high skewness ratios needed to be avoided.
The entire structure contained 1,480,573 nodes and 3,818,254 elements which was an increase in the computational domain by over 40% compared to the other parts. The maximum skewness of 0.973 was found in the thick helix. This skewness was right on the edge of what was allowable using ANSYS FLUENT. The studies were still able to converge (ANSYS FLUENT, 2012).

**Thin Helix**

The thin helix was created as a theoretical case that added only a rotational element with a minimal decrease in the cross-sectional area. This was done in attempt to only study the addition of rotation to the flow without an increase in velocity. This part
was theoretical because the flat plate twisted to a helix used would have a thickness of only 0.02 in. The cross-sectional area in the direction perpendicular to the flow was 5.87 $in^2$ which was a reduction of 1.1% from the full bore pipe. This likely was not thick enough to exhibit the structural integrity or erosion resistance that it would require for service. Note the small gap between the two halves highlighted green in Figure 37. The flow splitter was dropped from this model as the helix plate did not block a significant part of the flow.

![Figure 37: Thin Helix Geometry](image)

The mesh was done with the same settings as the thick helix. The total volume of the thin helix was larger due to the thickness reduction of the helix plate. This increased the node and element count to 1,921,493 and 6,156,643 respectfully. Figure 38 showed the region where the helix plate ends and the flow enters the full bore pipe.
above the T-Part. The dark region at the center of the figure shows small elements near the interface of the three parts where the solution may have not be accurate. Some of these elements were highly skewed but they only represented a small portion of the domain.

Figure 38: Thin Helix Cross-section of Mesh

45 Degree Geometry and Mesh

A side study was also conducted to explore the effects of the small inlet entering the T-Part other than 90 degrees. It is not uncommon to see junctions that had one inlet coming in at 45 degrees to the main line in industry. This geometry is shown in Figure 39.
The choke was also implemented in the 45 degree geometry to give two test cases. These added studies which a manufacturer could use to make assessment of what design would best fit their application from a vibration and overall geometry standpoint. The choke was placed approximately one choke length from the center on the main line which can be seen in Figure 40. This distance was measured parallel to the center line of the small pipe at the 45 degree angle and is shown by the black line in the figure.
The mesh was created using the same parameters as the other setups to allow for comparability between the trials. Figure 41 shows the main difference between the mesh of 90 and 45 degree setups. There were different angles that had to be meshed using inflation in attempt to increase the accuracy near the walls. This was handled by the inflation auto mesh feature without any major problems or highly skewed cells. The mesh totaled 1,268,196 nodes and 2,265,533 elements for the geometry without the choke. It had a maximum skewness of 0.84. The mesh with the choke had about the same mesh statistics. Even though the volume of the flow domain with the choke was slightly smaller, it was meshed using a swept section. This resulted in the same number of elements, but the elements in the choke were slightly smaller.

Figure 41: 45 Degree Junction Inflation Mesh
In order to determine the force on the structure a few basic calculations were made. The outputs of FLUENT were the forces on the wall of the structure and the static pressure at the inlets. These were recorded for every time step. The wall force was calculated by FLUENT by summing the force across all nodes that laid on the walls. This was done separately for each of the three coordinate directions. The average of the static pressures at both of the inlets was area weighted. In order to get a complete picture of the forces on the entire structure, the forces resulting from the pressure at the inlets was added to the force on the walls (White F., 2011). This resulted in the total force on the structure being calculated using

\[ F_Y = F_{yw} + A_S p_s \]  

(36)

where \( F_{yw} \) was the force on the walls in the Y-Direction \( A_S \) was the area of the small inlet and \( p_s \) was the static pressure on the small inlet. Likewise,

\[ F_Z = F_{zw} + A_l (p_l - p_o) \]  

(37)

gave the total force on the structure for the Z-Direction. \( A_l \) was the area of the large inlet, which was the same area as the outlet, \( p_l \) was the static pressure of the large inlet and \( p_o \) was the static pressure at the outlet which was constant. Figure 42 shows a diagram of the force calculations. \( F_{yw} \) and \( F_{zw} \) were calculated on the walls of the pipe structure, shown orange in the figure, in their respected directions. FLUENT output the
sign wall force terms with the direction taken into account. The force in the X-Direction (into the page) did not require any inlet or outlet pressures to be taken into account because there were no inlets or outlets lying in this direction. The 45 degree trials were calculated using the same methodology but required the addition of direction sines and cosines to make the force directions comparable to the 90 degree trials.

![Figure 42: Force Calculations](image)

At this point, total force data was available for each of the Cartesian coordinate directions corresponding to a time step. In attempt to reduce the forces over an entire cycle to one value that could be easily compared, the mean was taken of the force over the time steps recorded. This was taken over two flow cycles for the 90 degree IP trials and over one cycle for the 90 degree OP and the 45 degree trials. The mean force was important for understanding the effect of a change in geometry, but did not correspond to vibration. Since reducing vibration was the main goal of the research, the maximum variance from the mean was calculated for each study. This produced a single force value for each study in each of the coordinate directions that was taken to be the most
important parameter in vibrations. This essentially gave the amplitude of the force profile and gave a result that could be a parameter for product design.

Since the study was done for a general case of a pipe structure, it was not applicable to do a modal analysis to determine the natural frequencies of the structure. Nonetheless, the natural frequency was still an important parameter that should be accounted for when attempting to reduce vibrations. This type of study would fall under the category of product design and not general research. Thus the only most important criterion for this research was the maximum force variance.
Chapter 5: CFD Setup

All studies were conducted using the same general settings in ANSYS FLUENT so each could be comparable. There were however differences between the solution methods and under-relaxation factors. This was because some geometries were harder to converge than others. Changing the solver settings did not affect the converged solution, just the number of iterations needed to reach convergence. Fortunately, the flows were relatively easy to converge and did not require any extreme solver settings. This was partly due to the fact that the fluid was incompressible and the geometries were relatively simple.

The solver used was a transient pressure based solver. This solver was most common for incompressible flows and could even solve most compressible flows (ANSYS FLUENT, 2012). The only output of interest was the force on the walls of the pipe structure. For this reason the only model used was a turbulence model. The fluid used for the study was water with a density of 62.31 $\text{lbm/ft}^3$ and a viscosity $6.739 \times 10^{-4} \text{ lbm/ft}$s.
### Table 1: Reynolds Numbers for Pipe Sections

<table>
<thead>
<tr>
<th>Pipe Section</th>
<th>Reynolds Number Range (10^5)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet small</td>
<td>6-7</td>
</tr>
<tr>
<td>Inlet Large</td>
<td>6.7-10.2</td>
</tr>
<tr>
<td>Outlet</td>
<td>12.9-15</td>
</tr>
</tbody>
</table>

All of the flow was fully turbulent for the entire domain as the transition from laminar to turbulent pipe flow is around 2,000. This is shown in Table 1. The transient flow velocity and the mixing of the two pipes also promoted turbulence. The SAS (Scale Adaptive Solution) turbulence model was used to model all of the flows. It was a two equation model which allowed the length scales to be fully modeled. The flow became fully transient when using this model type; no steady state solution existed. The curvature correction was not used and all model constants were left at their default values. Figure 43 showed two representative images of velocity magnitude for the T-Junction along the symmetry axis of the structure. The top image showed the k-e model time averaged out the flow throughout the domain. This resulted in a steady state solution. The SAS model showed the unsteady nature of the flow and did not yield a steady state solution.
The SAS turbulence model was the most comprehensive model that could be chosen for the computational resources available at the time of the simulations. It was intended to model the flow in the most accurate manner possible in attempt to capture the overall flow behavior at the smallest scales the resources allowed. Of course models only allow a portion of the spectrum to be represented but it was assumed that only the large scales (size of the mesh or larger) would contribute significantly to the mean flow. Using a higher level turbulence model such as Large Eddy Simulation (LES) or even using direct numerical simulation may have provided a more physically accurate flow. Though, it is not likely it would make a significant impact on the results of the study. The turbulence model was the only flow model used. It was assumed that other models (acoustics, energy etc.) would not contribute to the flow parameters of interest.
Boundary Conditions

Outlet

The outlet was set to a pressure outlet with a static pressure of 15,000 psi. This pressure was used as a reference to the highest hydraulic fracturing pressures generally used in industry. Using this high pressure gave results reflecting actual pressures that may be seen in the parts. This allowed the maximum probable pressures to be recovered. Knowing the maximum pressure might be helpful to know if the pressure containing body was near its yielding point. The additional pressure introduced to the system by a choke or other flow obstructions may have needed to be taken into account for the design. However, the outlet pressure for the simulation did not influence the results. The results from this study would have been unchanged if the outlet pressure was set to 0 or any other value because of the incompressible fluid.

The outlet turbulence settings were specified using backflow hydraulic diameter and turbulent viscosity. The turbulent intensity was taken to be 5%. This was the turbulent intensity recommended by FLUENT for standard pipe flows (ANSYS Inc.). It may have been possible to use iterative steps in attempt to fine tune the turbulent boundary conditions. This would involve choosing the turbulent boundary condition, running a simulation and calculating the turbulent intensity close to the inlet or outlet. This new value could then be put set as the boundary condition for a new study. However, it was originally assumed that pressure forces would dominate the total vibration of the part. From this it was assumed that the regions of the domain that only had straight pipe flow (next to inlets and outlets) would not significantly add to the
forces of the structure. Therefore the turbulent intensity of the inlets and outlet was
assumed to not affect the results and 5% was used for all studies.

The outlet turbulence setting would only be turned on if reversed flow occurred
at the outlet. This meant that essentially the pressure outlet would become a pressure
inlet boundary condition. Since both of the inlets were velocity inlets, which could not
contain reversed flow, this scenario did not occur.

**Inlets**

There were two inlets, the large and the small, which were both set to velocity
inlets. They could be thought of as mass flow inlets of constant density. The velocity
was specified as normal to the boundary. The supersonic gauge pressure was left at its
default value of 0 as it was not applicable. The turbulent parameters were specified
using the same settings as the outlet. They were not changed throughout the study as
they were assumed to not significantly affect the results.

The velocity input was specified using an interpreted UDF (User Defined
Function). This was code written in C that allows the user to inject code directly into
FLUENT source code allowing the user to change any parameter of interest. This
allowed the velocity profile to be a function of the flow time which translated to
crankshaft position and angular velocity. The input velocity is shown graphically in
Figure 9 in the Introduction as well as overlaid on a number of the figure in the results.
The UDF code used is given in Appendix A: Velocity Input UDF. The first block of
code gives the velocity used for the IP flow cases. Both inlets used the same velocity UDF. This physically simulated all pumps in the system running with their crankshaft In-Phase. The in-phase trails were conducted with both velocity inlets using the “velocity_ip” UDF.

The second UDF titled “velocity_op” gave the same velocity profile as velocity_ip except it was Out-of-Phase by $\pi/6$ rad. When an Out-of-Phase trial was conducted, the small inlet was set to the velocity_op UDF and the large inlet was left using the velocity inlet. The outlet settings were left the same for both the In-Phase and Out-of-Phase trials. Note that all calculations done using FLUENT were carried out using SI units within the source code. Since the UDF was directed directly into the source code, SI units must be used within the UDF since they were not converted. Likewise any value pulled from the source code using a UDF was in SI units.

Walls

The walls were set to stationary walls for the entire flow boundary. The only shear conditions used was the no slip boundary condition. The wall roughness cannot be specified when using the SAS turbulence model; the model ran solely off of the no slip condition.
**Outputs**

Three different types of outputs were used, pressure at inlets, force on the walls and velocity. The force and the pressure outputs were used to calculate the results of the study and the velocity was used to confirm the study was running properly.

The force on the walls of the structure was reported using a roundabout method as specified by a recommendation from ANSYS. FLUENT was set up to output drag as the coefficient of drag. The coefficient of drag was calculated using

$$C_d = \frac{2F_d}{\rho v^2 A}$$  \hspace{1cm} (38)

where $F_d$ was the desired output. The reference values of $\rho$, $v$ and $A$ were manually inputted in such a way that $F_d = C_d$. This allowed the coefficient of drag output to be used for force calculations. The coefficient of drag was set to output the sum of the force on all of nodes that laid on the wall of the structure.

The coefficient of drag was recorded for each time step of the calculation and was output in each of the three Cartesian coordinate directions. This resulted in three output files with a force value for each time step in the calculation. An UDF was needed to have FLUENT output the coefficient of drag output for each time step. The UDF was written and provided by ANSYS FLUENT and resides in Appendix B: Force Output UDF.
The static pressure at the inlets was needed to find the force being outputted on the structure. It was found using an area-weighted average of static pressure on the large and small inlet faces. The pressure was recorded for each time step and did not require an UDF to be recorded.

An area weighted velocity average was also recorded at the small and large inlets. This was done to confirm that the velocity input values were read into the software correctly and could be related to the force outputs. This output essentially gave a bulk velocity of the inlets.

**Solver and Convergence Settings**

Different solver settings were experimented with. The goal was to find settings that would converge solutions in the shortest amount of CPU time. This was not necessarily the same thing as the fewest number of iterations. The exact time it took conduct a study was not recorded, qualitative observations were used to optimize the solver settings. As discussed earlier, the solver settings did not influence the results obtained, only the rate at which they will converge.

Studies seemed to converge the quickest and were the most stable when a SIMPLE (Semi-Implicit Method of Pressure-Linked Equations) was used. This provided the fastest time per iteration but usually did not have the smallest number of
total iterations compared to other solver options such as the coupled solver. This solver also had the advantage of using less RAM than the coupled solver. The SIMPLEC (SIMPLE-Consistent) scheme was used for studies that had highly skewed elements (>0.94) because it allowed for skewness correction.

The spatial discretization schemes were not changed from study to study. The gradient was calculated using the Green-Gauss Cell Based Method. The pressure scheme used was the PRESTO! (PREssure Staggering Option). The momentum, turbulent kinetic energy and specific dissipation rate all used first order upwind schemes. The transient formulation used a bounded second order implicit scheme.

The solution control under-relaxation factors were generally reduced to low values (<0.5) until convergence was achieved for the first few time steps. The calculation was then stopped and the controls were returned to their default values. This was done to increase the performance of the solver and reduce the time it took to converge to a solution. The higher order terms were relaxed for all variables using a relaxation factor of 0.75. Double precision was used for all studies.

Convergence

Convergence was monitored by checking the residual values for the continuity, x-velocity, y-velocity, z-velocity, k and \( \omega \). The criteria for convergence used was the default value of 0.001. This meant all of the residuals must have been below this value before the solver would advance to the next time step. Figure 44 shows a representative
plot of the residuals that were taken from the Thick Helix study. When the residuals dropped below the criteria set, the flow time would advance which in turn changed the velocity inputs. This caused a jump up in the residuals meaning that each peak on the chart represented a time step. It was preferable for the residuals to converge monotonically which was a sign of a stable solution (ANSYS FLUENT, 2012). All residuals did converge monotonically.

Figure 44: Residual for Each Iteration

Unfortunately, the only parameter FLUENT would check for convergence before advancing the time step was the residuals. Ideally the parameter of interest, in this case the pressure at the inlets and the coefficient of drag, would be monitored each iteration and the solution would be advanced when it became constant. This was not an option for a transient simulation so the standard flow parameters discussed earlier were the only criteria checked for convergence. Initial studies used a convergence setting of
0.0001 and yielded the same solutions as using 0.001, so there was a degree of confidence in the convergence of the studies.

The time step was chosen to be 0.0001 seconds. FLUENT recommended estimating the time step using

\[
\Delta t = \frac{1}{3} \frac{L}{V}
\]

(39)

where L was the length of the cell and V was the velocity. Applying this formula resulted in \(8.33 \times 10^{-5}\) s time steps. This was simply rounded up to \(1 \times 10^{-4}\) s as the estimation given by FLUENT was said to be conservative (ANSYS FLUENT, 2012). It was also recommended that each simulation run for around 20 iterations before advancing the time step for solution stability.

It took anywhere from 4 days to two weeks to receive solutions for two cycles of the velocity input. Simulations were run on blade servers with 2 Intel Xeon X5650 processors (12 cores \(\times 2.66\) GHz) with 48 GB of RAM. The final simulations were run on a newly acquired server with 2 Intel Xeon E5-2640 processors (12 cores \(\times 2.5\) GHz) with 64 GB of ram and the AVX instruction set. The last studies would take around 2-4 days to run. The simulations were computationally intensive due to the large node count of the mesh. It was desired to run the simulations at the finest mesh settings possible for the technology available to try and capture any flow phenomenon that might be present in the flow.
Chapter 6: Force Examination

The results from the studies showed an interesting phenomenon. They appeared to have a discontinuous jump in the force of the structure in both the Y and the Z-Directions. It originally did not seem reasonable that this type of force behavior would be caused by a continuously changing velocity profile. A side study was done to understand the area around the discontinuous jump that occurred at around time step of 550 shown in Figure 45. This was the end of the first flow velocity cycle. The interest was in finding if the jump in the forces had a physical significance or was it caused by an instability in the numerical scheme. Note the figure showed two velocity cycles each lasting around 550 time steps.

![Figure 45: T-Junction In-Phase Force Output](image)

---

72
The force in the X-Direction was shown in the figure as reference only. The force values are so close to zero that the force was not discernible at this scale. The force had a mean value of 0 lbf and a max variance of 2 lbf. It was assumed that the force in the X-Direction did not make a significant contribution to the force on the entire structure. Thus, this force was neglected and calculations were not done for this direction in all subsequent trials.

A new coarse mesh was made to study the effects of using different velocity input curves on the force output of the structure. This allowed studies to be quickly run to gain an overall understanding of the results. The mesh was created in the same way as the other full size trials but the elements were allowed to be larger. The T-Junction geometry was used as it was the base line geometry. The tetrahedral elements in the T-Part were allowed to have a maximum length of 0.315 in. The elements in the swept part of the mesh used the same maximum element length as the T-Part. The inflation boundary layer had a first layer height of 0.0315 in and was comprised of 7 layers with a growth rate of 1.1. The entire mesh contained 35,993 nodes and 55,483 elements and had a maximum skewness of 0.768. The reduced mesh count greatly reduced the time it took to solve each iteration.

First, the effect of a continuous sine function input velocity through the small inlet only was studied. The large inlet had a constant input velocity of 32.8 ft/s. The pressure at the outlet for this group of simulations was 0 psi. This allowed for 6 significant digits throughout the calculations compared to 8 for the full size study.
calculations. Figure 46 shows the raw force output data from the T-Junction trial. The force caused by the pressure at the inlets was not subtracted for this figure. The force on the structure in the Y-Direction seemed to vary with the velocity input as was expected. The force in the Z direction also was influenced by the small inlet velocity but to a smaller degree. The forces did not exhibit the discontinuous jump seen in Figure 45.

The inlet velocities were then switched between the inlets. The large inlet took the sinusoidal velocity and the small inlet took the constant velocity of 32.8 ft/s. This was done to determine if changing the inlet velocity in one direction would change the force output in the orthogonal direction. Figure 47 shows the force in the Y-Direction being influenced by the changes in the large inlet velocity. It appeared that the for this pipe configuration, changing the inlet velocity only affected the force in the Y-direction. However, this was only the sum of the forces on the wall; the pressure at the inlets was not taken into account.
Subtracting the pressure at the inlets significantly changed the force results. Figure 48 shows the force in the Z-Direction varying as a function of the inlet velocity and the Y-Direction force staying mostly constant. This had two possible implications. First, the pressure at the inlet was a large contributor to the forces on the structure. The shape and sign of the force in both directions showed this. The graph was completely changed by taking into account the static pressure at the inlets. Next, the force on the structure in a direction was influenced significantly only by the velocity entering the structure in that same direction. The velocity that entered through the large inlet in the Z-Direction and only caused a force change in the Z-Direction.
To further examine the inlet pressures contribution to the overall force, Figure 49 gives the area weighted average of the pressure across the two inlets. Both of the pressures had about the same magnitude and shape and were Out-of-Phase with the velocity curve. When added to the force on the walls in the Y-Direction, the pressure from the small inlet appeared to flatten out the total force variance in the Y-Direction. This resulted in the total force in the Y-Direction being approximately the same as the change in momentum of the fluid in that direction, which can be calculated with

\[ F = \dot{m}v \]  

where \( v \) is the velocity in through the small inlet and \( \dot{m} \) is the mass flow rate. The momentum change in for the Y-Direction resulted in a force of approximately 86 lbf.
This was approximately the same value given by Figure 48. Thus the pressure at the inlets played an important role in determining the entire force on the structure.

![Graph showing pressure and velocity over time](image)

**Figure 49: Sinusoidal Velocity Inlet Pressure**

This helped understand what was responsible for the forces on the structure, but the jump still needed to be examined further. The flow velocity near the jump was molded using a simple V velocity profile through both inlets as shown in Figure 50. The figure shows the total force on the structure with the pressure forces at the inlets and outlets taken into account. The forces on the structure show the large jump in the forces on the structure in both orthogonal directions of interest. The value of the velocity hardly changed from one side of the jump to the other. The time derivative of the velocity at the inlets is the only input parameter that changed across the jump.
Next, it was tried to remove the jump by flattening out the sharp change in the velocity by introducing a plateau to the inlet velocity as shown in Figure 51. The force output for Flow Time <~85 and >~110 were approximately the same in Figure 50 and Figure 51. The introduction of the plateau introduced a plateau in the forces but the jump remained.
In attempt to remove the jump, a parabola was fitted to the point of the V velocity profile, which is shown in Figure 52. The addition of the parabola to the velocity profile removed the jump and replaced it with what appeared to be a linear function. This was interesting because the derivative of a parabolic function was a linear function.

![Figure 52: V-Parabola Velocity Profile Both Inlets](image)

The area around the jump was further examined by fitting a circle to the point of the V as is seen in Figure 53. It was difficult to match exactly at the point where the straight section from the V ended and the circle began, but the results were still useful. The resulting equation for a circular velocity profile took the form

\[ v = b \pm \sqrt{-x^2 + 2ax - a^2 + r^2} \]  

(41)

where \( f_t \) was the Flow Time, \( v \) was the velocity. \( a, b \) and \( r \) were constants to fit the circle to the velocity profile. The derivative of this circular velocity profile
\[ v = \frac{a - f_t}{\sqrt{-f_t^2 + 2af_t - a^2 + r^2}} \]  

(42)

had a shape similar to the force profile shown in both directions in the figure. This gave further evidence that the time derivative of the velocity profile was responsible for the force on the structure.

A discontinuous jump in the velocity was tried as a hypothetical test case to study its effect on the force. The velocity was dropped by ~0.3 ft/s from one time step to the next. It created two effects. First, it caused a spike in the forces that was an order of magnitude greater than the force before and after velocity jump. It was assumed that the magnitude of the spike was at least somewhat an artifact of numerical error caused by a discrete change in the velocity. Second, the force had approximately the same value
before and after the jump in velocity. This further strengthened the notion the force was largely caused by a change in time derivative of the velocity at the inlets.

![Figure 54: Discontinuous Jump Velocity Profile](image)

The time derivative of the velocity profile in Figure 45 was taken numerically to compare to the force on the structure. As shown in Figure 55, fluid acceleration appeared to be close to a linear function, implying that the region of the sine curve being used as an inlet velocity was close to a parabolic function. The shape of the fluid acceleration curve was similar to the shape of the forcing function on the structure. All of the results pointed to the fluid acceleration being a driving factor of the force on the structure.
Figure 55: Inlet Fluid Velocity and Acceleration

All of the previous flow trials resulted in about the same force on the structure in the flow times away from the discontinuity in the time derivative of the velocity profile. This showed that the jump in the force has some physical significance. It was possible that the jump did not happen instantaneously as Figure 45 shows, but the time steps were just not nearly small enough to have resolution of the jump. The true shape of the jump may be an area for future research.

This study assumed that the spike around the jump, as indicated by the yellow circle in Figure 56, was an anomaly caused by numerical instabilities. It was possible that these spikes could be eliminated by tightening the convergence criteria significantly around the spike. This however was not in the scope of this study and may be an area of future research. All data analysis was done with the spike manually truncated from the data along with the time steps on either side of it. This resulted in a total of 0.0009 s
being removed from each data set. Moreover, it was likely not possible to have such a sharp change in fluid acceleration in practice. The real flow output from a pump would likely have a more rounded velocity profile near the jump as a result of slip (temporary reversed flow through a check valve) in the valves of the pump.

Figure 56: Highlighted Force Jump
Chapter 7: Results and Discussion

90 Degree IP Trials

The first study was the T-Junction 90 Degree IP. The geometry was previously shown in Figure 14. This served as the base line and standard for what was commonly used in industry and gave the simplest case. Figure 57 shows the force on the wall of the structure over two periods. It was interesting to note, even though turbulence was inherently transient, the force output did not show it. Running the simulation for longer and gathering data for more periods did not result in a significant difference in the results. The cycle appeared to be truly periodic. This was likely because, even though the velocity at each point in space may not be the same for all flow cycles (resulting from the turbulent nature), but the average over the entire domain was. Thus, only two flow periods were analyzed for the In-Phase trials.

![Figure 57: T-Junction In-Phase Flow](image)

84
The magnitude of the force in the Z-Direction was larger than the magnitude in the Y-Direction. This seemed counterintuitive because fluid entering parallel to this direction did not have to undergo a change in direction. The larger change in force in the Z-Direction can be mostly attributed to the pressure differential between the large inlet and the outlet.

The small spike in the force in the Z-Direction around the flow time of 5 time steps was due to the stopping and starting of a simulation. This was done to either change a relaxation factor or confirm that the study was running correctly. This spike was seen in a number of the trials but was nothing more than fictitious data resulting from FLUENT recoding data at a non-converged time step. If a trial was stopped, FLUENT reported the data from the iteration number that it was calculating and advanced to the next time step even if the current one was not converged. These points were truncated from the data and not used in the analysis but were still reported in the figures.

Figure 57 shows the force at a maximum at the beginning of a cycle and reducing towards the end of the cycle. It was interesting to note that at the end of the cycle the force became negative indicating a switch of direction of the force. This was a result of the pressure at the large inlet actually being lower than at the outlet. The pressure at the inlet was as much as 10 psi lower at the inlet than at the outlet for this part of the cycle. The force on the walls of the structure was not significant for this
direction and was thought to be mostly caused by wall shear, which was assumed to be negligible.

The force in the Y-Direction exhibited a similar nature to the force in the Z-Direction. It began at a maximum and decreased monotonically throughout the cycle. The force was positive for around the first half of the cycle. This means that the structure had a force exerted on in the direction opposite the fluid flow. The pressure at the small inlet was as much as 11 psi lower than the pressure at the outlet. This further showed that the pressure at the inlets drove the force on the structure and the change in momentum of the flow in the Y-Direction was relatively small.

A pictorial representation of the pressure contours shown in Figure 58 gives a sense of the overall pressure distribution of the domain. The contour was set on the symmetry plane (x=0) of the domain. The figure showed that the pressure distribution was relatively uniform in the area close to inlets or outlet, far from the T-Part. The high pressure gradients were contained in the region just downstream of the T-Part.
The static pressure was only part of the equation though. To get a better sense of the forces on the structure, Figure 59 shows the sum of the static and dynamic pressures, or the total pressure. All of the pressure appeared to be mostly uniform except for just downstream of the T-Part on the upper half of the pipe. This area had a total pressure less than the rest of the domain and was likely responsible for the force at this time step.
In order to have a means to directly compare each of the different trials, the average of the force over two velocity periods was taken for each direction. The In-Phase T-Junction had a mean force of -1 and 200 lbf in the Y and Z-Directions respectively. The variance was 50 and 309 lbf in the Y and Z-Directions, respectively.

*Choke*

There were five different trials used for the In-Phase choke that featured the choke at different distances from the T-Part center line. Figure 60 shows a pronounced jet exiting the small pipe that remained partially intact through the T-Part. The magnitude of the flow velocity was as high as 178 ft/s. The jet flow even partially stuck to the wall exhibiting a mild Coandă Effect. It was seen in a video of the flow that the jet sticks to the upstream wall (opposite shown in picture) during part of the flow.

*Figure 60: Choke 1.5 Lengths*
A major difference that was seen in the choke trial was the increased static pressure at the small inlet. The pressure upstream of the choke was increased by as much as 170 psi above the outlet. The pressure at the small inlet never dropped below the outlet pressure. Figure 61 did not show the contour that contained the highest pressure, but rather a general pressure distribution. Though this increase in pressure at the small inlet was small compared to the outlet pressure, only about a 1% increase, it should not be ignored. The small pressure difference resulted in a significant increase of force on the structure.

![Figure 61: Static Pressure Contour Choke 1.5 Lengths](image)

The total pressure contour was examined to understand the location of the pressure differential on the structure responsible for the overall force. Figure 62 shows the largest pressure gradients in the domain were in the area around the choke exit. This means that it was possible that the largest contribution to the forces on the structure came from the area local to the choke, not the T-Part.
Figure 62: Total Pressure Contour Choke 1.5 Lengths

All of the choke trials were compiled into a single chart for each of the directions for easy comparison. Figure 63 shows that the location of the choke did not change the shape of its force curve or the time step for which the force was 0. The trial for 0.5 lengths resulted in data that appeared to be random. This was thought to be because the high velocity fluid jet exiting the choke was so close to the pipe wall of the T-Part that the grid size was not able to accurately resolve the velocity gradients. It was possible that reducing the mesh size in the T-Part would have yielded consistent results. The computational resources available would not allow for a simulation with a reduced mesh size to be run in a reasonable amount of time. This may be an area for future study.
The figure shows an overall trend, the further the choke was from the centerline of the T-Part the higher the force was. This was seen by first examining the time steps from 0 to around 225. The further away choke trials had higher force values in the Y-Direction for this time span. The forces all converged to around 0 force at time step 225. This same trend was seen for time steps 225 to around 550. The absolute value of the force was always higher for chokes further away from the T-Part. In the Y-Direction the results showed it was beneficial to have the choke as close as possible to the T-Part.

It seemed that for the second velocity cycle (time steps 550 to 1100), the force on the structure became more chaotic. This was possibly a result of the turbulent nature of the flow becoming more pronounced as the flow time advanced. Though the exact values of the forces for a given time step were slightly different from the first
cycle to the second cycle, the trend underlying force trend still remained. As technology is advanced, maybe it would be possible to run these trials for more time steps to confirm the trends do not change.

The forces on the structure in the Z-Direction did not seem to follow the same pattern as in the Y-Direction. Figure 64 shows there was not a substantial difference between the forces in the Z-Direction for different choke placements. All of these forces were taken to be about the same value. It was interesting to note that the magnitude of the forces in the Z-Direction was much larger than the forces in the Y-Direction. The Z-Direction also exhibited large force spikes just after the jump that occurred around time step 550. Since the wall shear on the pipes was neglected, the majority of the force was attributed to the pressure differential between the outlet and the large inlet. The pressure at the large inlet varied from 15046 to 14991 psi. This was almost not affected by the placement of the choke.

![Figure 64: Choke Trial Z-Direction Force](image-url)
The forces in the Z-Direction did reverse their direction for a portion of the cycle form time steps around 460 to 550. This corresponded to the period where the static pressure at the large inlet was lower than the outlet pressure, resulting in an adverse pressure gradient. It was interesting to note that the force for the 0.5 Lengths trial corresponded to the other trials and did not show the chaos present in the Y-Direction.

Table 2 shows that the mean force in the Y-Direction was about 0 lbf. The variance was the least for 1 length and highest for 1.5 lengths. The difference between the maximum and minimum force was 14.8 lbf, a significant amount. The mean and variance are all comparable in the Z-Direction. The difference in the mean was less than 1% and less than 2% for the variance. It is assumed that these differences could be attributed to error or uncertainty.

<table>
<thead>
<tr>
<th>Trial</th>
<th>Y-Direction</th>
<th>Z-Direction</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Mean (lbf)</td>
<td>Max Variance (lbf)</td>
</tr>
<tr>
<td>0.75 Lengths</td>
<td>0.9</td>
<td>59.7</td>
</tr>
<tr>
<td>1 Length</td>
<td>0.5</td>
<td>59.6</td>
</tr>
<tr>
<td>1.5 Lengths</td>
<td>1.1</td>
<td>69.6</td>
</tr>
<tr>
<td>2 Lengths</td>
<td>-0.2</td>
<td>74.4</td>
</tr>
<tr>
<td>3 Lengths</td>
<td>-0.3</td>
<td>73.1</td>
</tr>
</tbody>
</table>
Cone

The cone trial was unique for the fact that it was the only geometry that reduced the flow velocity in the Y-Direction before the fluid entered the T-Part. Figure 65 showed a static pressure distribution that was similar to the baseline T-Junction. The highest pressure gradient was around the outlet of the T-Part.

![Figure 65: Static Pressure Contour Cone](image)

The total pressure in Figure 66 shows similarities to the T-Junction as well. They both had a region just downstream of the T-Part on the upper half of the pipe that had the lowest pressure in the entire domain.
The cone was the easiest to converge out of all of the 90 degree trials and as such used the least amount of CPU time for the run. Figure 67 shows the force output on the structure has much the same shape as the other trials. There was a major difference in the Y-Direction force; it did not go to zero around time step 225 like the other trials. Instead it reached zero at around time step of 105 and spent the rest of the cycle at a negative force. This resulted in a mean and variance force of –22 and 48 lbf in the Y-Direction respectfully. This study had results similar to the other trials in the Z-Direction. The mean and variance was 200 and 309 lbf respectfully.
The helix proved to be an interesting trial as it was the only trial that added a rotation to the fluid. Since there was a direction that had substantial velocity unique to this trial, a simple contour could not be used to visualize the flow. Figure 68 shows pathline particles colored by velocity that were released from both of the inlets. The figure shows pathlines entering the T-Part with a rotational velocity component. The fluid rotation was still pronounced.
Another representation of the amount of rotation is seen in Figure 69. The contour shows velocity in the X-Direction (out of the page positive) alone the X-Z plane. The velocities in this direction were substantial in the small pipe exiting the helix all the way down to the T-Part. There were velocities in the X-Direction greater than the maximum velocity at the inlets.
The static pressure distribution shown in Figure 70 was similar to the T-Junction. The area of low pressure fluid circled in black was more pronounced and confined to a small region. Differences in static pressure are seen in the area close to the walls just downstream of the helix.

Figure 70: Thick Helix Static Pressure Distribution
Figure 71 shows a large spike in the forces following the jump compared to the other studies. It was of course truncated out for the final calculations but showed that the helix was sensitive to numerical errors. It was possible that the highly skewed cells present in the mesh were a cause of the large spike. It was also interesting to note the force curve became rough towards the end of the second velocity cycle. The forces for both of the geometries were less than zero for almost the entire flow cycle. The Thin Helix had lower force values than the thick helix for all time steps.

As with the other trials, the force in the Z-Direction did not change with the different geometry. The differences between the two curves shown in Figure 72 were not significant enough to make any conclusions from.
Figure 72: Helix Force Z-Direction

The Thick Helix had lower variance than the thin helix in the Y-direction. The Thick Helix variance force was almost the same as the Thin Helix. It also had more favorable mean values.

Table 3: Helix Force Summary

<table>
<thead>
<tr>
<th>Trial</th>
<th>Y-Direction</th>
<th>Z-Direction</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Mean (lb)</td>
<td>Max Variance (lb)</td>
</tr>
<tr>
<td>Thin Helix</td>
<td>-56</td>
<td>67</td>
</tr>
<tr>
<td>Thick Helix</td>
<td>-44</td>
<td>68</td>
</tr>
</tbody>
</table>

Bulge

The bulge was unique because it was the only trial that had a change in the geometry of the T-Part. Like the cone, it created an increase in volume to the flow domain. All of the other trials decreased the volume of the flow domain compared to the baseline T-Junction. There was a region at the bottom of the Bulge-Part where the
velocity magnitude approached zero as could be seen in Figure 73. The bulge is difficult to see but it is shown magnified in the next figure. The flow exiting the outlet also appeared to not be as close to fully developed turbulent flow as the other trials.

Figure 73: Bulge Velocity Magnitude Contour

There was a region at the bottom of the bulge that had lower total pressure than the surrounding area. This was a contrast to the other studies that had total pressure at least as high as the surrounding area at the bottom of the T-Part. The rest of the total pressure field appeared to the have the same overall features as the other trials.
Figure 74: Bulge Total Pressure Contour

Figure 75 shows the Z-Direction force was about the same as the other trials. It still had the pronounced spike right at the point of the jump. The force in the Y-Direction did not show a pronounced spike. The Y-Direction force became zero at about time step 178. Overall the Y-direction force variance was the lowest out of all of the trials. It was an average of 0 and a max of 38 lbf. The second flow cycle did appear to be slightly rougher than the first.
In-Phase Summary

Figure 76 shows that there was little difference between the force plots of the different trials in the Z-Direction. It was difficult to say if there was any difference between the different trials or it could all just be attributed to error and/or uncertainty. The velocity profile for both of the inlets was overlaid on the chart to help give context and understanding of the forces.

Figure 75: Bulge Force Data

Figure 76: Z-Direction Force Summary
Table 4 and Figure 77 gave two views of the force calculation results. The difference between the trial with the largest mean and the smallest was only 3 lbf. This was less than a 2% difference. It was assumed that any minute difference could be ignored or attributed to error. Even if the value was of physical significance, it would not be enough to make a real world difference. The same could be said about the maximum variance. The difference in variance was larger, 8 lbf from the highest to the lowest, a difference of 3.7%. This was taken to be an inconsequential amount. Thus the results showed that the changes made did not show any impact on the forces in the Z-Direction.

<table>
<thead>
<tr>
<th>Trial</th>
<th>Mean (lbf)</th>
<th>Max Variance (lbf)</th>
</tr>
</thead>
<tbody>
<tr>
<td>T-Junction</td>
<td>200</td>
<td>309</td>
</tr>
<tr>
<td>0.75 Lengths</td>
<td>201</td>
<td>310</td>
</tr>
<tr>
<td>1 Length</td>
<td>200</td>
<td>309</td>
</tr>
<tr>
<td>1.5 Lengths</td>
<td>201</td>
<td>311</td>
</tr>
<tr>
<td>2 Lengths</td>
<td>201</td>
<td>305</td>
</tr>
<tr>
<td>3 Lengths</td>
<td>202</td>
<td>311</td>
</tr>
<tr>
<td>Bulge</td>
<td>201</td>
<td>309</td>
</tr>
<tr>
<td>Cone</td>
<td>200</td>
<td>309</td>
</tr>
<tr>
<td>Thin Helix</td>
<td>202</td>
<td>313</td>
</tr>
<tr>
<td>Thick Helix</td>
<td>199</td>
<td>308</td>
</tr>
</tbody>
</table>
The forces in the Y-Direction were affected noticeably by the differences in flow geometry as can be seen in Figure 78. The choke trials had higher force magnitudes than the base line T-Junction for all flow times. It first appeared that the force approached ~10lbf around time step 275 for the choke, T-Junction and bulge trials. This was the location where the time derivative of the velocity inlets approached zero. The cone stood apart from the other trials. Its force was less the baseline for all time steps and less than the chokes that were closer than 1.5 lengths.
Table 5 and Figure 79 made it easy to quantitatively compare the different trials. The T-Junction, all of the choke trials and the bulge had mean values that were essentially 0 lbf. All of the choke trials had larger variance, a minimum of 20% higher, than the baseline. The Helix design did not show favorable results. They both had average absolute force values far larger than the base line and had large variances. The thin and thick helix had variance 126% and 36% higher than the baseline respectfully. The bulge and the cone both had encouraging results. The cone had a mean force magnitude 21 lbf higher than the base line, but the variance was 2 lbf less, a modest decrease of 4%. The bulge had the best results of all the trials with a mean force of 0 lbf and a variance 11 lbf less than the base line. This was a decrease of 22%. This was a physically significant amount.
Table 5: Y-Direction Force Summary

<table>
<thead>
<tr>
<th>Trial</th>
<th>Mean (lbf)</th>
<th>Max Variance (lbf)</th>
</tr>
</thead>
<tbody>
<tr>
<td>T-Junction</td>
<td>-1</td>
<td>50</td>
</tr>
<tr>
<td>0.75 Lengths</td>
<td>1</td>
<td>60</td>
</tr>
<tr>
<td>1 Length</td>
<td>0</td>
<td>60</td>
</tr>
<tr>
<td>1.5 Lengths</td>
<td>1</td>
<td>70</td>
</tr>
<tr>
<td>2 Lengths</td>
<td>0</td>
<td>74</td>
</tr>
<tr>
<td>3 Lengths</td>
<td>0</td>
<td>73</td>
</tr>
<tr>
<td>Bulge</td>
<td>0</td>
<td>39</td>
</tr>
<tr>
<td>Cone</td>
<td>-22</td>
<td>48</td>
</tr>
<tr>
<td>Thin Helix</td>
<td>-56</td>
<td>67</td>
</tr>
<tr>
<td>Thick Helix</td>
<td>-44</td>
<td>68</td>
</tr>
</tbody>
</table>

Figure 79: Y-Direction Force Summary Chart

90 Degree Out-of-Phase Results

The Out-of-Phase results provided similar contour plots to those discussed earlier in the In-Phase section. All of the shapes and defining features that could be seen
in the plots were essentially indistinguishable between the In-Phase and Out-of-Phase trials. It was useful to explain and point out features of the flow field, but it was all qualitative observations that did not necessarily translate into force data. For this reason the contours were not shown or discussed for the Out-of-Phase flow trials.

Since the force data appeared to be approximately periodic, the simulations were only run for one flow cycle. This cut the CPU time in half which allowed for more flow configurations to be tried. The 3 lengths choke study was not run for Out-of-Phase because it did not have favorable In-Phase results. It was possible to run only one cycle and avoid the jump because the force value after the jump was approximately the same as the value at the beginning of the simulation (time step 1). The assumption was made that the calculations for the mean and variance essentially would come out with the same value no matter the number of cycles were run. The trial with the choke 0.5 lengths from the center of the T-Part with Out-of-Phase flow resulted in the same data anomalies as was in the In-Phase results. Therefore it was not shown in the results.

All charts of the forces in the Z-Direction for different flow geometries were in essence the same chart. This further showed that other changes would have to be made to the system to affect the forces in the Z-Direction. Only one chart was shown of the forces in the Z-Direction and it can be seen in Figure 80. All of the different flow geometries were placed on the chart to illustrate that they are the same. There was a large change in the forces for the Out-of-Phase trial. At time step 225, there was another jump present in the force data. This was due to the time derivative of the small inlet
flow velocity changed signs. This created a force jump and spike of smaller magnitude than was seen for the In-Phase trials. The flow velocities at both of the inlets were overlaid on the figure for clarity. Figure 80 represented one flow cycle.

The maximum value of the force was at the beginning of the flow cycle and the minimum was still at the end. The small jump in the middle of the flow cycle had a magnitude of about 140 lbf. The large jump at the beginning or end of the cycle had a magnitude of about 400 lbf. This made it apparent that the large inlet velocity had a much larger effect on the force in the Z-Direction than the small inlet velocity. Even if the spike associated with the small jump was left in for the calculations, it would not have changed the maximum force variance. The force at the beginning of the cycle was still larger by almost to 100 lbf. It was also interesting to note that the force did not spend much of the cycle as a negative value. Only right at the end of the cycle did the force become negative and it had a minimum value of about -10 lbf.
Table 6 and Figure 81 show that there was little difference between the trials. There was only 11 (≈5.4%) lbf separating the trials with the highest (T-Junction) and lowest (1 Length) mean. Likewise there was only 10 lbf (4.9%) separating the trails with the highest and lowest max variance. These values were so close that no conclusion was made as to what geometry would perform best under these circumstances.
Table 6: Out-of-Phase Flow Z-Direction Force Summary

<table>
<thead>
<tr>
<th>Trial</th>
<th>Mean (lbf)</th>
<th>Max Variance (lbf)</th>
</tr>
</thead>
<tbody>
<tr>
<td>T-Junction OP</td>
<td>205</td>
<td>206</td>
</tr>
<tr>
<td>0.75 Lengths OP</td>
<td>197</td>
<td>201</td>
</tr>
<tr>
<td>1 Length OP</td>
<td>194</td>
<td>204</td>
</tr>
<tr>
<td>1.5 Lengths OP</td>
<td>196</td>
<td>204</td>
</tr>
<tr>
<td>2 lengths OP</td>
<td>197</td>
<td>202</td>
</tr>
<tr>
<td>Bulge OP</td>
<td>196</td>
<td>203</td>
</tr>
<tr>
<td>Cone OP</td>
<td>198</td>
<td>206</td>
</tr>
<tr>
<td>Thin Helix OP</td>
<td>196</td>
<td>201</td>
</tr>
<tr>
<td>Thick Helix</td>
<td>197</td>
<td>196</td>
</tr>
</tbody>
</table>

Figure 81: Out-of-Phase Flow Z-Direction Force Comparison Chart

There were however, quantifiable differences in the force in the Y-Direction that can be seen in Figure 82. For Out-of-Phase flow, all of the forces started with a negative value. All of the trials except for the cone and the helix had a value of about -10 lbf at time step 0. The forces stayed almost constant for all trials except for the two helix trials.
which decreased slightly. At time step 275 the only jump in this direction was present. All of the forces jump to their maximum values right at the point where the acceleration of the fluid changes its sign. The force in the Y-Direction did not jump when the time derivative of the velocity entering the large inlet changes its sign.

The different choke trials followed the same trend as In-Phase trials. The further the choke was from the main line the higher the forces were on the structure. The T-Junction had a lower absolute value of force than all of the choke trials. The force spike was present for all of the trials.

Table 7 shows there were significant differences between the different flow geometries. Both of the helix trials had mean values significantly less than the others.
The helix also had some of the highest variances out of all of the trials. The thin helix proved to be one of the worst from a vibration standpoint along with the thick helix and the choke furthest from the main line, 2 lengths. The closest choke (0.75 lengths) had a variance that was still 9 lbf higher than the baseline T-Junction.

The two trials that showed favorable results were the bulge and the cone. The cone had a mean force value that was significant. The cone essentially increased the force while decreasing the flow velocity in that direction. Despite this, the most important variable, the max variance, was less for the cone by 12%. The bulge was even more promising; its max variance was 24% less than the baseline. This was a large enough change that it was postulated that it was of physical significance.

<table>
<thead>
<tr>
<th>Trial</th>
<th>Mean (absolute value) (lbf)</th>
<th>Max Variance (lbf)</th>
</tr>
</thead>
<tbody>
<tr>
<td>T-Junction OP</td>
<td>2</td>
<td>49</td>
</tr>
<tr>
<td>0.75 Lengths OP</td>
<td>1</td>
<td>58</td>
</tr>
<tr>
<td>1 Length OP</td>
<td>0</td>
<td>57</td>
</tr>
<tr>
<td>1.5 Lengths OP</td>
<td>1</td>
<td>64</td>
</tr>
<tr>
<td>2 lengths OP</td>
<td>1</td>
<td>73</td>
</tr>
<tr>
<td>Bulge OP</td>
<td>0</td>
<td>37</td>
</tr>
<tr>
<td>Cone OP</td>
<td>21</td>
<td>43</td>
</tr>
<tr>
<td>Thin Helix OP</td>
<td>60</td>
<td>72</td>
</tr>
<tr>
<td>Thick Helix OP</td>
<td>43</td>
<td>68</td>
</tr>
</tbody>
</table>
Out-of-Phase to In-Phase comparison

\textit{Y-Direction}

The Out-of-Phase flow made for an interesting comparison to the In-Phase flow. It was originally thought that running pumps In-Phase would increase the forces on the structure. The thought behind this was that there would be more energy running through the system when two pumps were In-Phase and at the peak of the velocity cycle than any other time. This was the scenario that would have caused the most variance in the flow velocity and resulted in the highest outlet velocity during the flow cycle. The results showed that this was true, but the differences between the two flows was less than expected and certainly did not coincide with the differences in velocity discussed in the introduction.
The In-Phase T-junction was only 1 lbf higher than the Out-of-Phase in the Y-direction. This difference was small enough that it was difficult to say that there was an improvement going to Out-of-Phase. Table 8 showed differences between the trials found using the In-Phase less the Out-of-Phase values. A positive value denoted the Out-of-Phase force being less than the In-Phase. The only two variance changes that were worth noting are the choke for 1.5 lengths and the cone which both had a decrease of 5 lbf. This meant that the Out-of-Phase was 10.4% and 7.1% less force than the In-Phase. The thin helix even had higher forces for the Out-of-Phase trial.

Table 8: In-Phase to Out-of-Phase Comparison Y-Direction

<table>
<thead>
<tr>
<th>Trial</th>
<th>Mean Difference</th>
<th>Variance Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>T-Junction</td>
<td>-1</td>
<td>1</td>
</tr>
<tr>
<td>0.75 Lengths</td>
<td>0</td>
<td>2</td>
</tr>
<tr>
<td>1 Length</td>
<td>0</td>
<td>3</td>
</tr>
<tr>
<td>1.5 Lengths</td>
<td>1</td>
<td>5</td>
</tr>
<tr>
<td>2 lengths</td>
<td>-1</td>
<td>1</td>
</tr>
<tr>
<td>Bulge</td>
<td>0</td>
<td>2</td>
</tr>
<tr>
<td>Cone</td>
<td>0</td>
<td>5</td>
</tr>
<tr>
<td>Thin Helix</td>
<td>-3</td>
<td>-4</td>
</tr>
<tr>
<td>Thick Helix</td>
<td>1</td>
<td>0</td>
</tr>
</tbody>
</table>

**Z-Direction**

The forces in the Z-Direction showed forces completely different from the Y-Direction. There were vast improvements in variant force that were gained by using Out-of-Phase flow in the Z-Direction which were summarized in Table 9. For all different trials the mean force was rather small. All of the differences in the mean were less than 5% of the IP trial and thus were considered negligible differences. The
differences in the variance were noticeable. The baseline T-Junction had an Out-of-Phase force that was 33.3% lower than the In-Phase force. Since the forces in the Z-Direction did not change much between trials all of the improvements were around 30%. This was the largest improvement that was made to improve the forces out of all of the different parameters used. The statistics for this finding are perhaps the most substantial because almost identical results were seen for 8 different cases.

### Table 9: In-Phase to Out-of-Phase Comparison Z-Direction

<table>
<thead>
<tr>
<th>Trial</th>
<th>Mean Difference(lbf)</th>
<th>Variance Difference (lbf)</th>
</tr>
</thead>
<tbody>
<tr>
<td>T-Junction</td>
<td>-5</td>
<td>103</td>
</tr>
<tr>
<td>0.75 Lengths</td>
<td>4</td>
<td>109</td>
</tr>
<tr>
<td>1 Length</td>
<td>5</td>
<td>105</td>
</tr>
<tr>
<td>1.5 Lengths</td>
<td>5</td>
<td>107</td>
</tr>
<tr>
<td>2 Lengths</td>
<td>5</td>
<td>103</td>
</tr>
<tr>
<td>Bulge</td>
<td>4</td>
<td>106</td>
</tr>
<tr>
<td>Cone</td>
<td>2</td>
<td>103</td>
</tr>
<tr>
<td>Thin Helix</td>
<td>6</td>
<td>111</td>
</tr>
<tr>
<td>Thick Helix</td>
<td>2</td>
<td>111</td>
</tr>
</tbody>
</table>

### 45 Degree Trials

The 45 degree trials were done for both In-Phase and Out-of-Phase flows. Overall they were easier to converge and took less time to run than the 90 degree trials. This may have been due to the fluid not having to change direction as much from the small inlet to the outlet. Since all of the In-Phase 90 degree trials gave close to identical results from the first flow cycle to the second, only one flow cycle was run to save computational time. The velocity profile of the fluid downstream of the T-Part was
similar to the 90 degree studies. However, the area of low velocity fluid circled in Figure 84 appeared to be slightly smaller than the previous trials showed.

![Figure 84: 45 Degree Velocity Magnitude Contour](image)

The area of low total pressure circled in Figure 85 was also smaller than in the 90 degree trials. The pressure was uniform near two inlets. The total pressure field also appeared to be fully developed at the outlet.
The choke being introduced to the setup produced a similar result to the 45 degree trials with no choke. The area of low velocity seen downstream of the T-Part was almost non-existent as can be seen in Figure 86. It appeared that the jet of high velocity fluid leaving the choke was pushed up to partially fill the region circled black in the earlier figures.
The total pressure contour amplified what was seen in the velocity contour. The area of low total pressure is only slightly seen in Figure 87. The only area that the total pressure dips below the outlet static pressure was along the outlet of the choke. The entire large pipe had a pressure greater than the outlet pressure.

![Figure 87: 45 Degree Choke Total Pressure Contour](image)

It was interesting to see that the force in the Y-Direction and Z-directions, shown in Figure 88 and Figure 89, were close to the force profiles seen for the 90 degree trials. The force went to zero around the time when the fluid acceleration went to zero for the Y-Direction. Both of the forces followed almost the same profile. The results showed only slight differences between the two flow geometries. It was notable to see the Z-Direction force profiles not being affected by the flow geometries because a component of the force caused by the small inlet affects the Z-Direction forces for the 45-degree trial.
The results showed the same trends for the Out-of-Phase flows. The force data was almost identical for both flow geometries in each of the directions. It appeared that
the 45 degree small inlet reduced the effect on the forces that the changes in the geometry had. The force in the Y-Direction only experienced one jump, in the middle of the flow cycle. It was interesting to note that placing the small inlet at a 45 degree angle did not couple the jump to both coordinate directions.

Figure 90: 45 Degree OP Y-Direction Force

Figure 91: 45 Degree Z-Direction Force
Table 10, Figure 92 and Figure 93 agreed with the statements made about the force plots. The difference between the maximum force and the maximum variance between the two trials in the Y and Z directions was 2% and 0.5% respectfully from the baseline. Both of these differences were assumed to be negligible. The effects between the two geometries were slightly more substantial for the Out-of-Phase flows, 7.3% for the Y-Direction and 1.9% for the Z-Direction over the baseline trial. The average forces were approximately the same for all of the scenarios in both of the directions. The largest deviance from the mean forces was the choke In-Phase trial in the Z-Direction. It was 2% higher than the baseline force. The results showed that the forces in the Y-Direction were not impacted by moving to Out-of-Phase flow. The variance for the baseline case was decreased by 2.3% which was not substantial. Similar results were seen in for the choke. The Z-Direction also showed a different finding. The forces were substantially less for the Out-of-Phase flow than for the In-Phase flow. The baseline trial had a decrease of 36% and the choke had a decrease of 37%. The 45 degree showed similar results in the Z-Direction to the 90 degree trials.

<table>
<thead>
<tr>
<th>Trial</th>
<th>Y-Direction</th>
<th>Z-Direction</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Mean (lbf)</td>
<td>Max Variance (lbf)</td>
</tr>
<tr>
<td>45 Degree Choke 1 Length</td>
<td>1</td>
<td>42</td>
</tr>
<tr>
<td>45 Degree Junction</td>
<td>0</td>
<td>43</td>
</tr>
<tr>
<td>45 Degree Junction OP</td>
<td>0</td>
<td>42</td>
</tr>
<tr>
<td>45 Degree Choke 1 Length OP</td>
<td>1</td>
<td>42</td>
</tr>
</tbody>
</table>
Figure 92: 45 Degree Z-Direction Force Summary Chart

Figure 93: 45 Degree Y-Direction Force Summary Chart
90 degree to 45 degree Comparison

Table 11 shows a comparison of all of the best performing trials in each category along with the baseline geometries. The best performers in each category are shown in bold. The results show indications that the bulge geometry with Out-of-Phase flow would provide the lowest forces in the structure in all categories. The bulge geometry is also the best performer in the In-Phase flow category. Surprisingly the 45 degree trials did not outperform the 90 degree trials. It was originally thought that the 45 degree trials would outperform the 90 degree because the fluid would have a smaller change in momentum. It was only going through a direction change of 45 degrees as opposed to 90 degrees. The results did agree with this assertion. The second best performer was the base line T-Junction.

Table 11: Comparison of Highest Performing Trials

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Mean(lbf)</td>
<td></td>
<td>Mean(lbf)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>T-Junction(90 degree)</td>
<td>1</td>
<td>50</td>
<td>200</td>
<td>309</td>
<td>313</td>
</tr>
<tr>
<td>Bulge(90 degree)</td>
<td>0</td>
<td>39</td>
<td>201</td>
<td>309</td>
<td>311</td>
</tr>
<tr>
<td>45 Degree Junction</td>
<td>0</td>
<td>43</td>
<td>199</td>
<td>333</td>
<td>336</td>
</tr>
<tr>
<td>45 Degree Choke</td>
<td>1</td>
<td>42</td>
<td>203</td>
<td>332</td>
<td>335</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>T-Junction(90 degree)</td>
<td>2</td>
<td>49</td>
<td>205</td>
<td>206</td>
<td>212</td>
</tr>
<tr>
<td>Bulge(90 degree)</td>
<td>0</td>
<td>37</td>
<td>196</td>
<td>203</td>
<td>206</td>
</tr>
<tr>
<td>45 Degree Junction</td>
<td>0</td>
<td>42</td>
<td>199</td>
<td>211</td>
<td>215</td>
</tr>
<tr>
<td>45 Degree Choke</td>
<td>1</td>
<td>42</td>
<td>199</td>
<td>208</td>
<td>212</td>
</tr>
</tbody>
</table>
Chapter 8: Conclusions

It is interesting to note that for the 90 degree studies, favorable results are only obtained from the flow geometries that increased the flow volume domain (cone, bulge). The trials that took away from the fluid domain (choke, helix) increased the vibrations. The results also show the choke that is being used in the field is not the most effective, as it actually showed an increase in forces on the T-Junction. This observation was not pronounced in the 45 degree trials. It should be noted that it is possible the choke reduces vibrations on the manifold by changing some parameter of the system not in the scope of the study.

It appeared that the change in linear momentum of the fluid did not play a huge role on the vibration. The force caused by the changing of the momentum was much smaller than the force resulting from the pressure at the inlets.

The trials that show the best results had not been optimized in any way. The geometry and results shown for the cone and the bulge were the first design iterations of the concept. They were design under the premise of changing the geometry and removing the least amount of material as possible. It was possible that making minor geometry changes to the trials shown to be effective could make them more favorable. Optimizing these two designs would be a possible area of future research.
It may give interesting result to try and combat the forces in the Z-Direction by placing a choke or a cone type design in the main line. The results showed that there were some distinct advantages in having flow inputs that are Out-of-Phase. If these results could be verified experimentally, highest amount of vibration reduction may be achieved by controlling the phase of the pump crankshafts.

This study provided an interesting application where CFD may be the only way to get results for this application. Experimental results would be difficult to obtain because of the nature of a thick walled pipe structure. It would be difficult to isolate the forces induced on a T-Junction by the fluid from the forces traveling through the walls of the pipes from other areas in the system. In the field there would also be a number of T-Junctions attached to the same structure causing forces that may produce noise on the T-Junction of interest. For these reasons it may be difficult to record valid experimental data to validate a CFD model.

The research successfully accomplished the objective which was to find a possible means of reducing vibration forces. A greater understanding of specifically what flow characteristics induced the most damaging forces was found. The research showed the time derivative of the velocity was largely responsible for the vibrations. This knowledge gave the parameter that should first be addressed to reduce vibrations in this study as well as future studies. With that two realistic methods of reducing the vibrations were found. First the flow itself could be changed by controlling the phase of the pumps. This showed to make the largest decrease in the forces on the structure but
only in the Z-Direction. Second, a relatively minor change in geometry to the T-Junction, the bulge, was proposed that showed significant force reduction in the Y-Direction only. Likely, the effectiveness of this geometry change to curtail vibrations may be able to be optimized making the results more substantial. An additional advantage of the bulge could be protected by a patent. The research was able to come up designs verified by CFD to reduce vibrations. The minor changes in geometry make the designs reasonable to manufacture. They could be quickly implemented in the field where they could give a company a competitive edge.
References


Appendix A: Velocity Input UDF

#include "udf.h"
DEFINE_PROFILE(tm_pout2smallop, t, nv)
{
    real p_1, p_2, p_3, p_t, k, w;
    face_t f;
    real flow_time = RP_Get_Real("flow-time");
    k=10;
    w=19;
    begin_f_loop(f,t)
    {
        p_1=k*sin(flow_time*w+1.047197551/2);
        if(p_1<0){
            p_1=0;
        }
        p_2=k*sin(flow_time*w+2.094395102+1.047197551/2);
        if(p_2<0){
            p_2=0;
        }
        p_3=k*sin(w*flow_time+4.188790205+1.047197551/2);
        if(p_3<0){
            p_3=0;
        }
        p_t=p_1+p_2+p_3;
        F_PROFILE(f,t,nv) = p_t;
    }
    end_f_loop(f,t)
}

/* UDF for setting target mass flow rate in pressure-outlet */
#include "udf.h"
DEFINE_PROFILE(tm_pout2small, t, nv)
{
    real p_1, p_2, p_3, p_t, k, w;
    face_t f;
    real flow_time = RP_Get_Real("flow-time");
    k=10;
    w=19;
    begin_f_loop(f,t)
    {
        p_1=k*sin(flow_time*w+1.047197551/2);
        if(p_1<0){
            p_1=0;
        }
p_1=0;
}
p_2=k*sin(flow_time*w+2.094395102);
if(p_2<0){
   p_2=0;
}
p_3=k*sin(w*flow_time+4.188790205);
if(p_3<0){
   p_3=0;
}
p_t=p_1+p_2+p_3;
F_PROFILE(f,t,nv) = p_t;
}
end_f_loop(f,t)
Appendix B: Force Output UDF

#include "udf.h"

#define NUM_FACE_ZONES 3
int FaceZoneID[NUM_FACE_ZONES]={7,6,3};
char *BaseFileName = "monitor-surface-force-";

static int find_existing_file(char *filename)
{
    FILE *fp;

    if ((fp = fopen(filename,"r")))
    {
        fclose (fp);
        return 1;
    }
    return 0;
}

/* write monitor file */
static void svplt(real flow_time,
                   real pltdata[], char *base_filename, char *abscsnm, char *ttlnm)
{
    FILE *write_plt;
    int i;
    char filename[200];

    for(i=0; i<NUM_FACE_ZONES; i++)
    {
        sprintf(filename, "%s%03d", base_filename, FaceZoneID[i]);
        if(find_existing_file(filename) == 0)
        {
            write_plt= fopen(filename, "w");
            fprintf(write_plt, ""%s\"\n", ttlnm);
            fprintf(write_plt, ""%s" "%s\"\n", abscsnm, ttlnm);
            fprintf(write_plt, "0 0 0 0\n");
        } else
            write_plt= fopen(filename, "a");
        if (NULL == write_plt)
        {
            fprintf(stderr, "Error: unable to open input file '%s'
", filename);
            exit(1);

3
} /* for(i=0; i<NUM_MONITOR_ZONES; i++) */

DEFINE_EXECUTE_AT_END(execute_at_end)
{
Domain *MixtureDomain;
Thread *f_thread;
MixtureDomain = Get_Domain(1);
real origin[3]={0.0, 0.0, 0.0};
real axis[3]={0.0, 0.0, 1.0};

#if !RP_NODE /* either serial or host */
real force[ND_ND], moment[ND_ND];
real zone_force_x[NUM_FACE_ZONES];
real zone_force_y[NUM_FACE_ZONES];
#if RP_3D
real zone_force_z[NUM_FACE_ZONES];
#endif
#endif

int i;
char filename[200], base_filename[200];
char *abscsnm = "Flow Time";
char *ttlnm = "Surface Force";
char titlename[200];
char ttlnm_xforce[200], ttlnm_yforce[200], ttlnm_zforce[200];
real flow_time;

for(i=0; i<NUM_FACE_ZONES; i++)
{
    f_thread = Lookup_Thread(MixtureDomain, FaceZoneID[i]);
    Compute_Force_And_Moment (MixtureDomain, f_thread, origin, force, moment,1);

    zone_force_x[i] = force[0];
    zone_force_y[i] = force[1];
    #if RP_3D
    zone_force_z[i] = force[2];
    #endif
}
Appendix C: Force Nature UDF

/* UDF for setting target mass flow rate in pressure-outlet */
#include "udf.h"

DEFINE_PROFILE(tm_pout2small, t, nv)
{
    real p_1, p_2, p_3, p_t, k, w;
    face_t f;
    int i;
    real flow_time = RP_Get_Real("flow-time");
    k=10;
    w=19;
    begin_f_loop(f,t)
    {
        p_1=k*sin(flow_time*w);
        if(p_1<0) {
            p_1=0;
        }
        p_2=k*sin(flow_time*w+2.094395102);
        if(p_2<0) {
            p_2=0;
        }
        p_3=k*sin(w*flow_time+4.188790205);
        if(p_3<0) {
            p_3=0;
        }
        p_t=p_1+p_2+p_3;
        for (i=1; i<30; i++) {
            if(flow_time*w/(1.047197551*i)<=1.01 &&
            flow_time*w/(1.047197551*i)>=0.99) {
                p_t=8.712138;
            }
        }
        F_PROFILE(f,t,nv) = p_t;
    }
    end_f_loop(f,t)
}
/ * UDF for setting target mass flow rate in pressure-outlet */  
#include "udf.h"  
DEFINE_PROFILE(tm_pout2ZZZZ, t, nv)  
{  
    real p_1, p_2, p_3, p_t, k, w;  
    face_t f;  
    int i;  
    real flow_time = RP_Get_Real("flow-time");  
    k=10;  
    w=19;  
    begin_f_loop(f,t)  
    {  
        p_1=k*sin(flow_time*w);  
        if(p_1<0)  
            p_1=0;  
        }  
        p_2=k*sin(flow_time*w+2.094395102);  
        if(p_2<0)  
            p_2=0;  
        }  
        p_3=k*sin(w*flow_time+4.188790205);  
        if(p_3<0)  
            p_3=0;  
        }  
        p_t=p_1+p_2+p_3;  
        for (i=1; i<30; i++) {  
            if(flow_time*w/(1.047197551*i)<=1.02 &&  
                flow_time*w/(1.047197551*i)>=0.98) {  
                p_t=95*(flow_time*w-i*1.04719755)*(flow_time*w-i*1.04719755)+8.721395137;  
            }  
            }  
            F_PROFILE(f,t,nv) = p_t;  
        }  
        end_f_loop(f,t)  
    }  
/* UDF for setting target mass flow rate in pressure-outlet */  
#include "udf.h"
DEFINE_PROFILE(tm_pout_parabola, t, nv)
{
    real p_1, p_2, p_3, p_t, k, w;
    face_t f;
    int i;
    real flow_time = RP_Get_Real("flow-time");
    begin_f_loop(f,t)
    {
        p_t=95*(flow_time*20-0.1)*(flow_time*20-0.1)+8.721395137;
        F_PROFILE(f,t,nv) = p_t;
    }
    end_f_loop(f,t)
}

/* UDF for setting target mass flow rate in pressure-outlet */
#include "udf.h"
DEFINE_PROFILE(tm_v_parabola, t, nv)
{
    real p_1, p_2, p_3, p_t, k, w;
    face_t f;
    int i;
    real flow_time = RP_Get_Real("flow-time");
    k=10;
    i=1;
    w=19;
    begin_f_loop(f,t)
    {
        if(flow_time*w/1.047197551>1.02) {
            p_t=3.974*(flow_time*w-1.04719755*1.02)+8.7630668;
        }
        if(flow_time*w/1.04719551<0.98) {
            p_t=-3.974*(flow_time*w-1.04719755*0.98)+8.7630668;
        }

        if(flow_time*w/(1.047197551*i)<=1.02 &&
            flow_time*w/(1.047197551*i)>=0.98) {
            p_t=95*(flow_time*w-i*1.04719755)*(flow_time*w-
                i*1.04719755)+8.721395137;
        }
    }
}
F_PROFILE(f,t,nv) = p_t;
end_f_loop(f,t)

#define PROFILE(tm_v_circle, t, nv)
{
    real p_1, p_2, p_3, p_t, k, w;
    face_t f;
    int i;
    double x, y;
    real flow_time = RP_Get_Real("flow-time");
    k=10;
    i=1;
    w=19;
    begin_f_loop(f,t)
    {
        if(flow_time*w/1.047197551>=1.02) {
            p_t=3.974*(flow_time*w-1.047197551*1.0193)+8.7630668;
        }
        if(flow_time*w/1.047197551<=0.98) {
            p_t=-3.974*(flow_time*w-1.047197551*0.98)+8.7630668;
        }
        if(flow_time*w/(1.047197551*i)<1.02 &&
           flow_time*w/(1.047197551*i)>0.98) {
            x=pow(flow_time*w,2);
            y=-2.25*x+4.71238898*(flow_time*w)-2.466414;
            p_t=1.2*(-pow(y,0.5))+8.762789+0.008;
        }
        F_PROFILE(f,t,nv) = p_t;
    }
    end_f_loop(f,t)

#define PROFILE(tm_v_circle, t, nv)
{
    real p_1, p_2, p_3, p_t, k, w;
    face_t f;
    int i;
    double x, y;
    real flow_time = RP_Get_Real("flow-time");
    k=10;
    i=1;
    w=19;
    begin_f_loop(f,t)
    {
        if(flow_time*w/1.047197551>=1.02) {
            p_t=3.974*(flow_time*w-1.047197551*1.0193)+8.7630668;
        }
        if(flow_time*w/1.047197551<=0.98) {
            p_t=-3.974*(flow_time*w-1.047197551*0.98)+8.7630668;
        }
    }

    if(flow_time*w/(1.047197551*i)<1.02 &&
       flow_time*w/(1.047197551*i)>0.98) {
        x=pow(flow_time*w,2);
        y=-2.25*x+4.71238898*(flow_time*w)-2.466414;
        p_t=1.2*(-pow(y,0.5))+8.762789+0.008;
    }
    F_PROFILE(f,t,nv) = p_t;
    end_f_loop(f,t)
#include "udf.h"
DEFINE_PROFILE(tm_v_flat, t, nv)
{
    real p_1, p_2, p_3, p_t, k, w;
    face_t f;
    int i;
    double x, y;
    real flow_time = RP_Get_Real("flow-time");
k=10;
i=1;
w=19;
begin_f_loop(f,t)
{
    if(flow_time*w/1.047197551>=1.02) {
        p_t=3.974*(flow_time*w-1.04719755*1.0193)+8.7630668;
    }
    if(flow_time*w/1.047197551<=0.98) {
        p_t=-3.974*(flow_time*w-1.04719755*0.98)+8.7630668;
    }
    if(flow_time*w/(1.047197551*i)<1.02 &&
flow_time*w/(1.047197551*i)>0.98) {

        p_t=8.7630668;
    }
    F_PROFILE(f,t,nv) = p_t;
}
end_f_loop(f,t)
}

/* UDF for setting target mass flow rate in pressure-outlet */
#include "udf.h"
DEFINE_PROFILE(tm_v_only, t, nv)
{
    real p_1, p_2, p_3, p_t, k, w;
    face_t f;
    int i;
    double x, y;
    real flow_time = RP_Get_Real("flow-time");
k=10;
i=1;

w=19;
begin_f_loop(f,t)
{
    if(flow_time*w/1.047197551>=1) {
        p_t=3.974*(flow_time*w-1.04719755*1.0193)+8.7630668;
    }
    if(flow_time*w/1.04719551<1) {
        p_t=-3.974*(flow_time*w-1.04719755*0.98)+8.7630668;
    }
    F_PROFILE(f,t,nv) = p_t;
}
end_f_loop(f,t)

/* UDF for setting target mass flow rate in pressure-outlet */
#include "udf.h"
DEFINE_PROFILE(tm_sine, t, nv)
{
    real p_1, p_2, p_3, p_t, k, w;
    face_t f;
    int i;
    double x, y;
    real flow_time = RP_Get_Real("flow-time");
    k=10;
    i=1;
    w=19;
    begin_f_loop(f,t)
    {
        p_t=0.23975*sin(flow_time/0.002/3.1415927*3-5.82239-0.525)+8.96;
        F_PROFILE(f,t,nv) = p_t;
    }
    end_f_loop(f,t)
}

/* UDF for setting target mass flow rate in pressure-outlet */
#include "udf.h"
DEFINE_PROFILE(tm_jump, t, nv)
{
    real p_1, p_2, p_3, p_t, k, w;
}
face_t f;
int i;
double x, y;
real flow_time = RP_Get_Real("flow-time");
k=10;
i=1;
w=19;
begin_f_loop(f,t)
{
    if(flow_time*w/1.047197551>=1) {
        p_t=8.7;
    }
    if(flow_time*w/1.04719551<1) {
        p_t=8.8;
    }
    F_PROFILE(f,t,nv) = p_t;
}
end_f_loop(f,t)
}