REDUCTION OF HEEL-TOE AFFECT IN A LONG HORIZONTAL WELLBORE BY VARYING INLET SIZE ALONG THE WELL

by

NAZMUL HOSSAIN

Student ID: 1591454

COURSE PROJECT

MECE 6353

Intro to Computational Fluid Dynamics

Presented to Dr. Ralph Metcalfe

Mechanical Engineering Department

UNIVERSITY OF HOUSTON

March, 2017

Abstract

Analysis of heel-toe affect due to pressure difference between heel and toe part of a long horizontal well is to be done using numerically established turbulent flow model. Firstly, to choose a numerical physics model, a simple T-junction pipe flow model was used to analyze different turbulence model present in ANSYS Fluent and their accuracy were compared to previous seminal experimental and numerical work. Then single stage of horizontal well was modeled and analyzed. Later, multiple stage of horizontal well were modeled and analyzed for varying inlet size along the horizontal length. The objective is study of varying inlet sizes from heel to toe for reduced heel-toe affect without compromising production flow rate.

Table of Contents

List of Figures

Chapter 1: Introduction

Directional drilling brought a new era to wellbore technology. Rather than just focusing on one location with a conventional vertical well, production from large radius of shale area is possible from a single location, which is also extremely important in off-shore hydrocarbon extraction. A very high angle (usually >85°) wellbore obtained from directional drilling, also known as horizontal wellbore are well known and very commonly used in present day petroleum industries. It has several advantages over conventional vertical wells. Optimizing petroleum production from horizontal well will have a big impact on underground hydrocarbon extraction process and their price. Fractures along the horizontal part of wellbore are divided into multiple stages. Optimizing performance of all stages confirms increased productivity. However, multiple challenging aspects are needed to overcome for an optimized performance. One of the biggest challenge in horizontal well is pressure difference formed between the near heel fracture inlets and near toe fracture inlets which is also known as heel-toe effect. At the beginning of production from a horizontal wellbore, a high flow rate from the near heel inlets are obtained compared to the near toe inlets, which leads to fully and partially blocked flow of near toe inlets. Although satisfactory overall production is achieved at the starting period, petroleum is not collected uniformly around the horizontal wellbore due to fully and partially blocked flow near toe. Petroleum from region near heel is nearly depleted, while the region near toe is still operational at later period of production. Average volumetric pressure of reservoir near heel region is dropped and slugging occurs due to pressure below bubble point at near heel inlets. It leads to discontinuous production of petroleum which is highly undesirable. It is extremely challenging and one of the major problem confronted in horizontal well. Artificial lifting is installed in the system to overcome this discontinuous production which is expensive, complicated and slow process with few drawbacks. In this paper, a numerical approach is attempted to investigate heel-toe effect for inlet diameter increasing from heel to toe at an optimized ratio to obtain stable effective wellbore pressure and nearly equal volumetric flow rate from all inlets without compromising productivity. This may

reduce time, expenditure and complexity in horizontal wellbore. Initially a single fracture inlet of horizontal wellbore was analyzed and for our final model, five stages with varying inlet diameters were used to analyze developing pressure difference and their effect on fluid flow. Different mesh and turbulence models were analyzed for our model to obtain result with less computational cost without compromising accuracy. Different flow regimes developed over time for the proposed model is also discussed.

Chapter 2: Background and Literature Review

In oil and gas industry, study of pressure drop and flow regimes in horizontal well is an extremely important discussion. Effects of average pressure drop in reservoir over time, heel-toe effect in horizontal wellbore and change of flow with time have been the key research area in horizontal well technology.

2.1 TYPES OF WELLBORE

Depending on situation, different types of wellbore is chosen to optimize production from a field. Conventional vertical wellbore has been used from early days in shale oil and gas extraction. With the most recent directional drilling technique, wellbore can be curved at maximum 15^o (usually 6^o-8^o) angle at any instantaneous point. After directional drilling was perfected, high angle wellbore has become more popular. With these high angle, horizontal wellbore, hydrocarbon production from a single wellbore has increased. It also enabled hydrocarbon extraction from places that are impossible to do with conventional wellbore. Different cost effective techniques like designer wells, multilateral wells are also adopted using high angle horizontal wellbore. Thus, performance of horizontal well is extremely important in present day oil and gas technology.

2.2 HORIZONTAL WELLBORE

Horizontal well is a high inclination angle ($>85^o$) well, exposed to large shale area for higher production in a stratified region. There are multiple completion stages in the horizontal wellbore. Each stage has fracture inlets, collecting petroleum inflow through hydraulic fractures. Inlets are at a minimum distance so that there is minimum interaction between inlets. A schematic of a horizontal wellbore is shown in figure 1. When horizontal section is at an angle (usually 0.5^o) with horizontal axis, it is called toe up, and the opposite is called toe down.

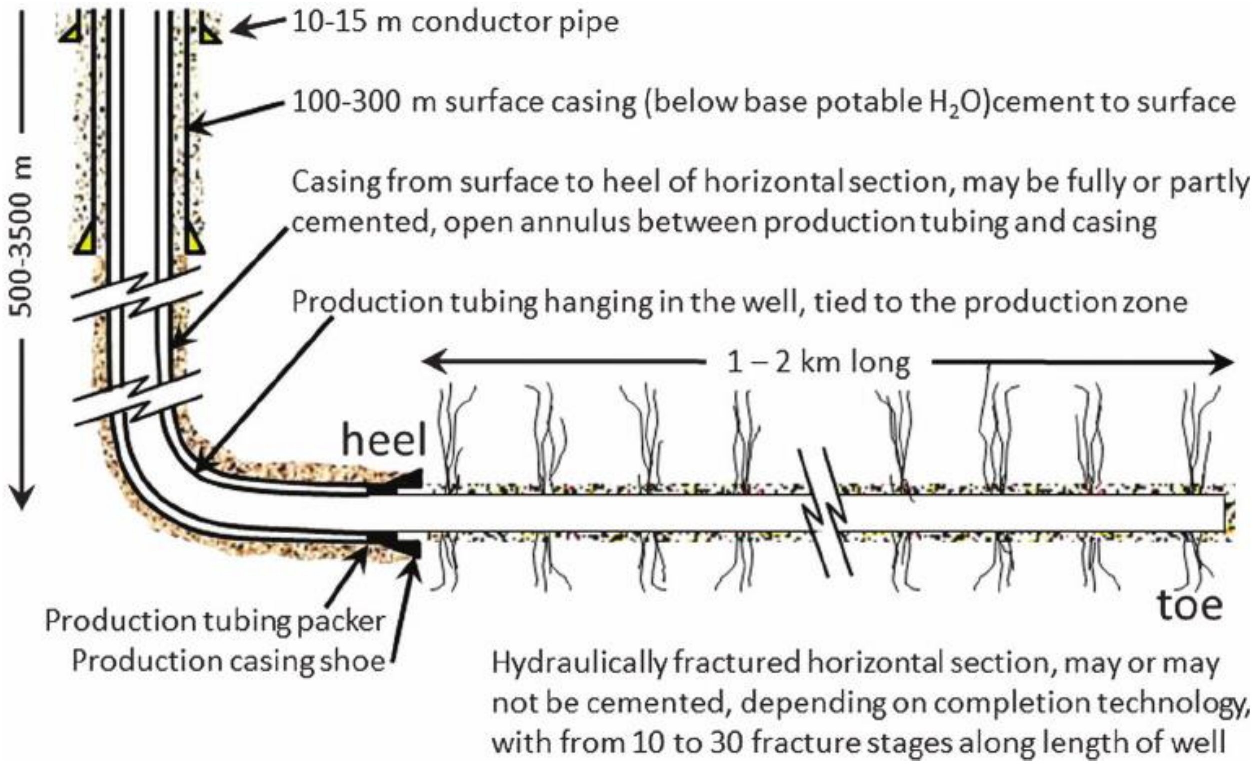


Figure 2.1: Schematic of a horizontal wellbore [3]

Production from horizontal well depends on the length of horizontal section of the wellbore. However, from analytical study by Dikken [1], horizontal well production performance is limited for horizontal length due to pressure drop along the length. Dikken [1] showed that after a certain optimum length, very little additional production results from extending well horizontal length. As we move from heel to toe region, average pressure inside the wellbore increases. This leads to lower pressure difference between shale reservoir pressure and wellbore inner pressure. This process is completely a pressure driven flow from shale reservoir to wellbore and due to lower pressure gradient near toe, the upstream region may face fully or partially blocked flow at the beginning of the production cycle. It limits the number of operational stages in horizontal wellbore. After certain period of production, overall volumetric pressure near heel region reduces because of the volumetric discharge. Then, previously fully and partially blocked inlets near toe, starts producing. However, due to low pressure in near heel volumetric reservoir, slugging occurs below bubble point pressure and discontinuous supply from well takes place. To overcome this problem, several artificial techniques have been adopted. However, these techniques are expensive, complicated and time consuming. Most popular artificial lifting is using screw type positive displacement pump. Also, with toe up horizontal wellbore, more stable intermittent slugging can be achieved, which is preferable over unstable long slugs from tow down wellbore at later period of production. In summary, heel-toe effect is one the most challenging problem in optimizing production from horizontal well. To improve performance of horizontal wells, more study on the heel-toe effect needs to be done for achieving stable continuous production from shale reservoirs.

2.3 NUMERICAL MODELS

For our study, we are expecting internal pipe flow with high Reynolds number. From literature, we know that generally for internal flow of $Re \geq 2000$ is turbulent. Turbulent flow is unstable, irregular and hard to analyze. There is no turbulence model yet, that can capture all types of turbulent flows with reliable accuracy. However, there are RANS, LES and DNS modeling approach dealing with turbulent flow.

RANS based models available in Fluent are One equation model, Spalart-Allmaras model, Two equation model, Standard k-ε model, RNG k-ε model, Realizable k-ε model, Standard k-ω model, SST k-ω model and Reynolds stress model. Among them, Spalart-Allmaras model is a low computational cost model and has shown good result with adverse pressure gradient. It solves the transport equation for a modified eddy viscosity. Also, Realizable k-ε models are recommended for near wall viscous effect consideration. It also has a very good performance with adverse pressure gradient, separation and recirculation of flows.

LES model is recommended when RANS model fails. It is better suited for combustion, mixing and external aerodynamic problems. There are SGS and DES turbulence model available in Fluent.

Chapter 3: Problem Statement & Methodology

With high average pressure at shale reservoir, turbulence flow in the pipe is expected. However, as time goes by and reservoir pressure reduces, flow slowly becomes laminar. In this paper, we are trying to obtain stable pressure along horizontal length at the beginning of production, and prevent partially or fully blocked flow. Thus, we are considering turbulent flow with Reynolds number 10000 to 40000 at the beginning of production, for our problem. Initially pipe flow with one inlet was analyzed and verified. The descriptions of the model are as follows.

Length = 3 m   Flow rate at pipe inlet = Q_{in}

Diameter = 0.0254 m   Flow rate at fracture inlet = $0.1Q_{in}$

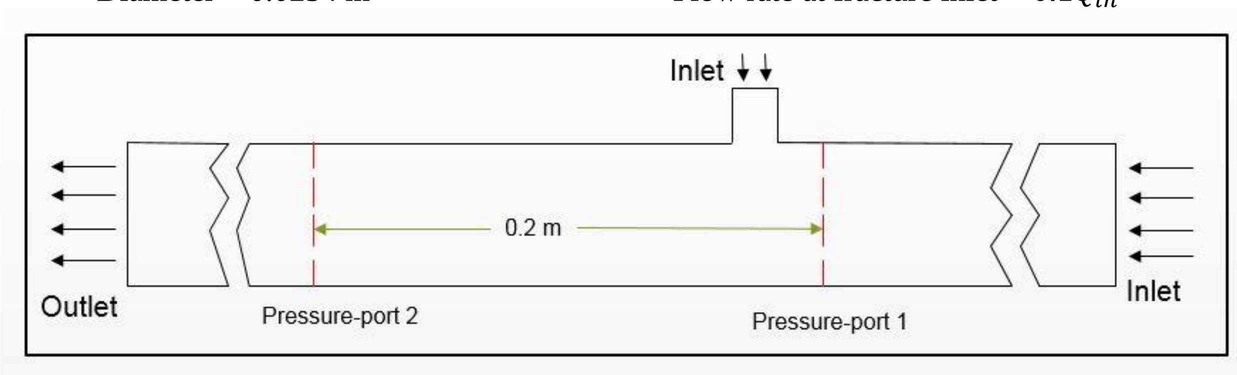


Figure 3.1: Schematic of single-inlet cross flow

Frictional factor and Reynolds number was calculated using following equations.

$$f_T = \frac{\frac{\Delta P}{\Delta x}}{\frac{\rho \bar{u}_2^2}{2D}} \qquad Re = \frac{\rho \overline{u_2} D}{\mu}$$

Here, ΔP = average pressure difference between press port 1 and press port 2

Δx = Distance between two press ports

$\bar{u}_2$ = average velocity at pressure port 2

More practical approach would be to analyze multiple fracture inlets with pressure driven flow. So, total 5 inlets along 0.125 m diameter pipe, each 5 m apart was analyzed to study heel-toe effect. The schematic diagram is as follows.

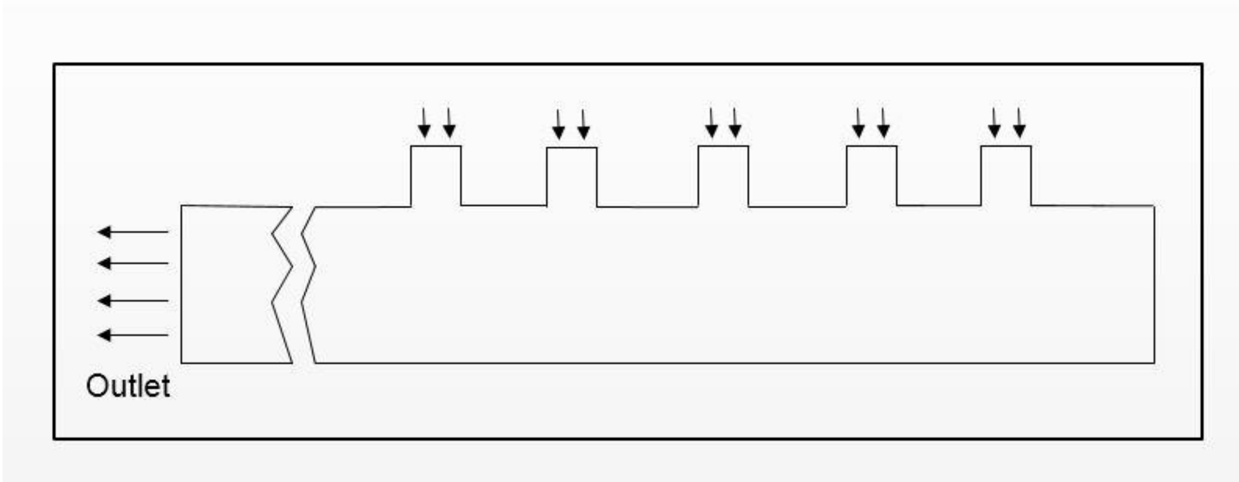


Figure 3.2: Schematic of multiple-inlet cross flow

Inlets were considered pressure inlets at 50 mPa. Volume flow rate at outlet, and fracture inlets were observed and discussed for different inlet sizes to analyze the performance. Different inlet size was represented considering normalized inlet size.

$$h' = \frac{h}{H}$$ Here, h = inlet size, H = pipe diameter

3.1 Design

A 2D non-axisymmetric geometry was made for initial analysis using the dimension of the experimental setup from yuan et al [2]. Then a 3D model was generated using the same dimension for a single inlet cross flow. Then we developed a 2D model including five fracture inlets of horizontal wellbore with same inlet diameters to observe heel-toe effect and flow analysis at steady-state. Then the model with varying inlet diameters are analyzed and compared with equal diameters to observe the performance.

3.2 Meshing

Different meshing techniques for structured and unstructured mesh are adopted. For the 2D single inlet model, we noticed that unstructured mesh near cross inlet at sharp corners shows better performance that structured mesh. If we were to use structured mesh at that region, mesh near boundary have to be very fine with smooth transition. The mesh element number for 2D analysis was low enough to ignore the rise in computational cost due to much finer mesh at cross-section. However, foe our 3D model there was a significant rise in number of elements. So, we made a combination of structured mesh at horizontal part and unstructured mesh at cross-section to find the optimum mesh. The goal is to achieve a mesh independent model with minimum number of element. We used a symmetric model to reduce computational cost.

Figure 3.2.1 shows the mesh at the pipe wall. Mesh at the sharp edge of cross-section was made finer for better accuracy.

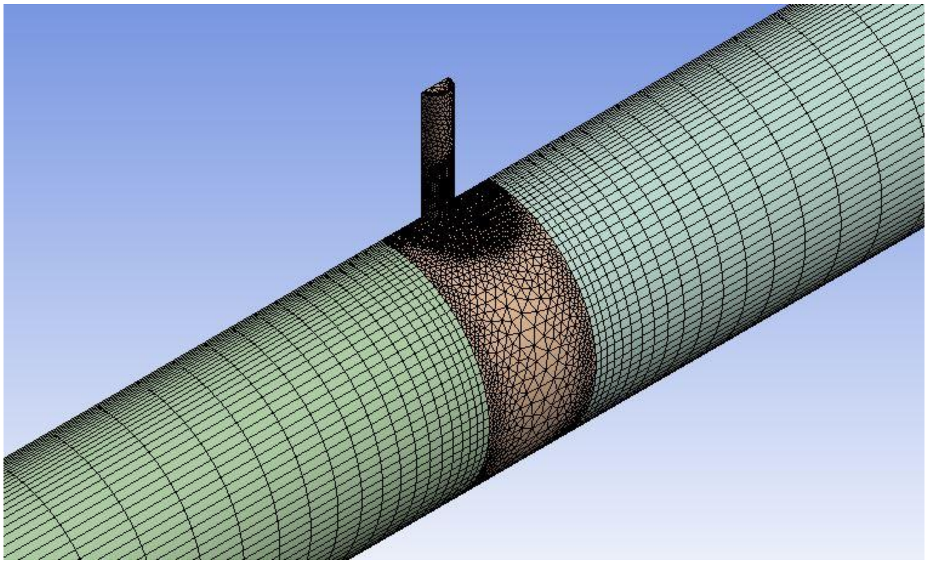


Figure 3.2.1: Mesh – Isometric view

Figure 3.2.2 and figure 3.2.3 shows the mesh at cross-section. As we can see from figure 3.2.3, 5 layers of inflation was applied at the boundary. Several mesh control techniques were applied to obtain this mesh.

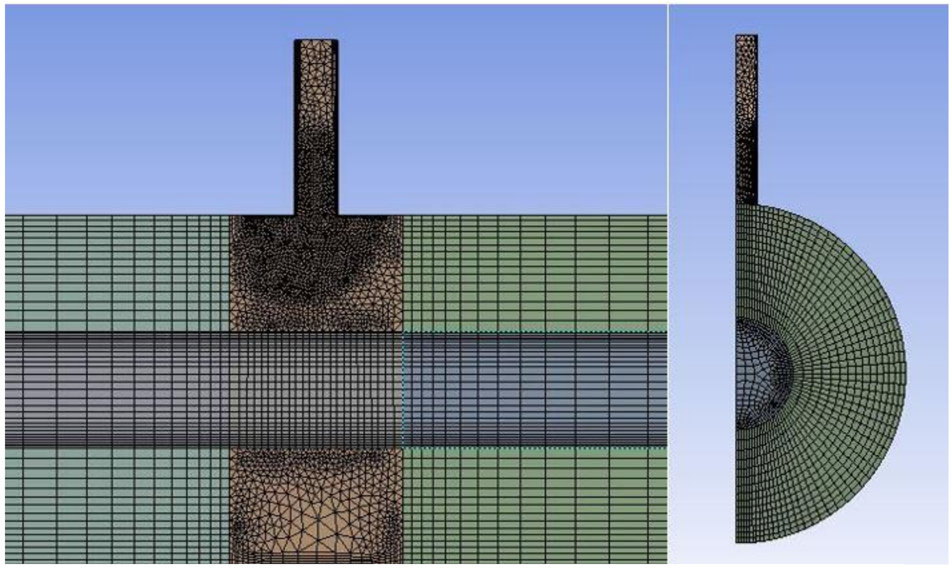


Figure 3.2.2: Mesh – cross section

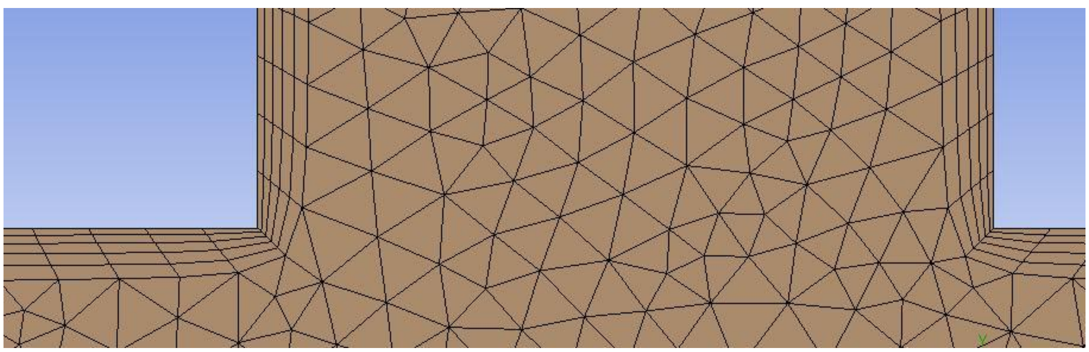


Figure 3.2.3: Mesh – near t-joint

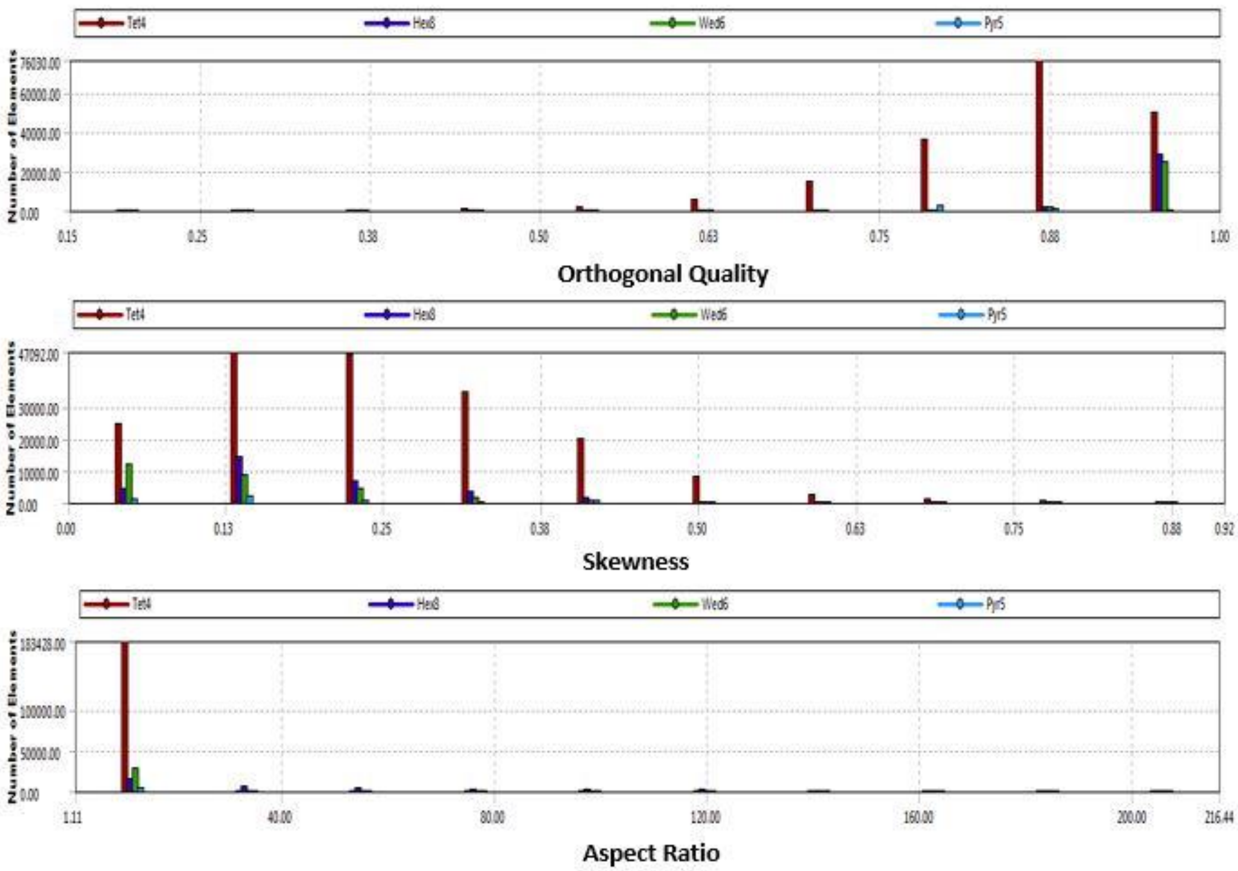


Figure 3.2.2: Mesh Quality

In figure 3.2.2, selected mesh quality is checked. From observation, most of the elements has orthogonal quality near 1, low skewness and low aspect ratio. These are all quality of a good mesh.

3.3 Computational Modeling

Observing the high Reynolds number, Standard k-ε, Realizable k-ε and Spalart-Allmaras model was compared to find the best model for our problem in ANSYS Fluent. Comparing our numerical results with yuan et al [3], we can understand accuracy and limitations of our used numerical model for a similar scenario as a single stage of horizontal wellbore. Analyzing that with different established turbulent models, meshing methods and solvers, we have selected our model to be used for efficient simulation by reducing computational cost with optimum accuracy. Residuals had six decimal place accuracy for our cases. For the transient problem, fluent udf was used to apply hydrostatic pressure loss over time.

3.3.1 Turbulence Model

For our study Spalart-Allmaras model was selected after comparative analysis. It is a RANS model with modified eddy viscosity. It works great on aerodynamic problems, mild flow separation and flow past an airfoil. It is a very low cost turbulence model.

Chapter 4: Result

4.1 Numerical Results

To understand the scope of numerical simulation for t-section pipe flow, initially a 2D geometry of t-section using dimensions from the experimental setup of Yuan et al[2] was done. As the 2D geometry of t-section is not axisymmetric, the result in figure 4.1 represents flow analysis using Spalart-Allmaras (SA) and Realizable k-eps model with enhanced wall treatment.

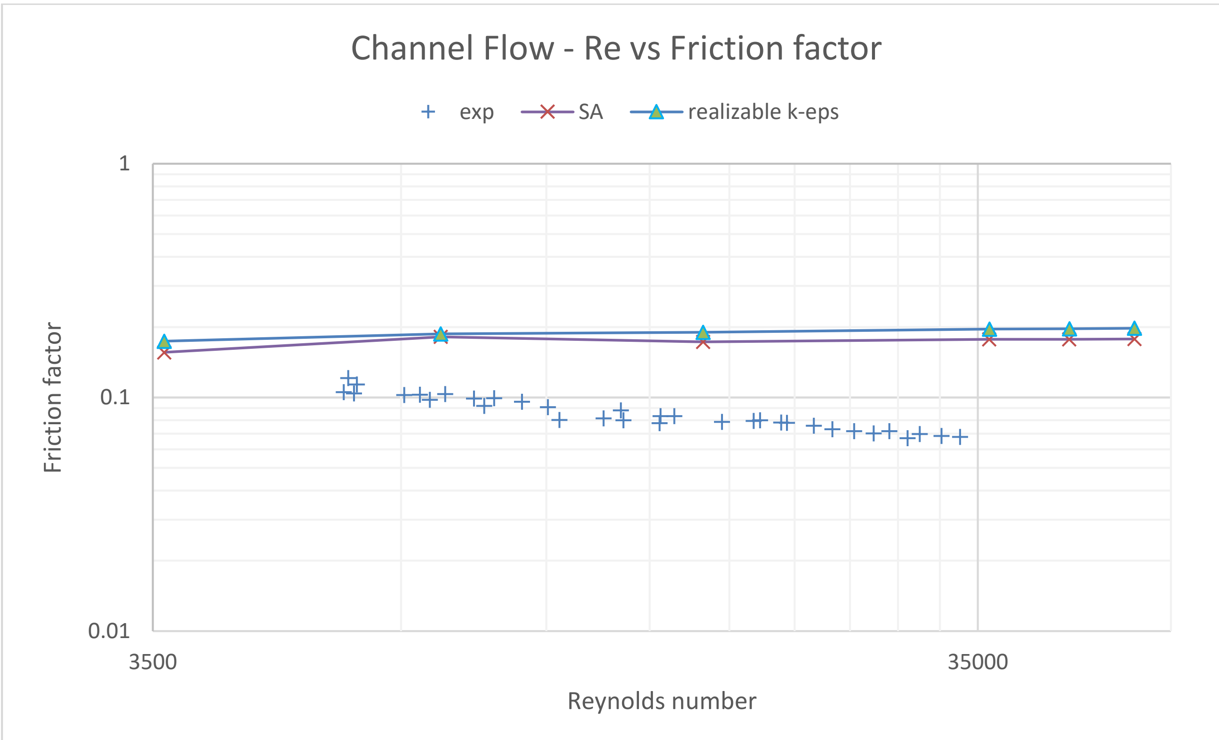


Figure 4.1: Channel Flow -Reynolds Number (Re) vs Friction factor (f_T)

In figure 4.1, there is noticeable deviation from the experimental data of Yuan et al[2]. Considering both viscous models showing a very similar result, we conclude that it may be due to non-axisymmetric geometry analysis comparing to pipe flow.

An analysis of a 3D symmetric geometry using Spalart-Allmaras model was done. In figure 4.2, comparison between that numerical and experimental result from Yuan et al[2] is presented. From that figure, we notice that the numerical and experimental results are very close. We get a good approximation of pipe flow using 3D geometry.

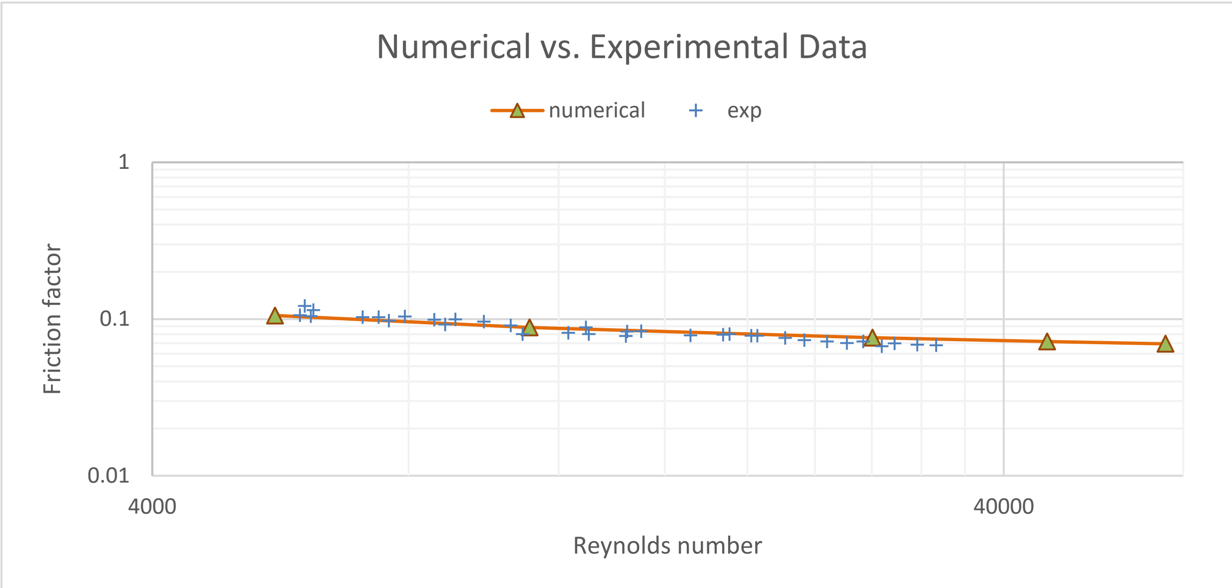


Figure 4.2: Pipe flow - Numerical vs Experimental data

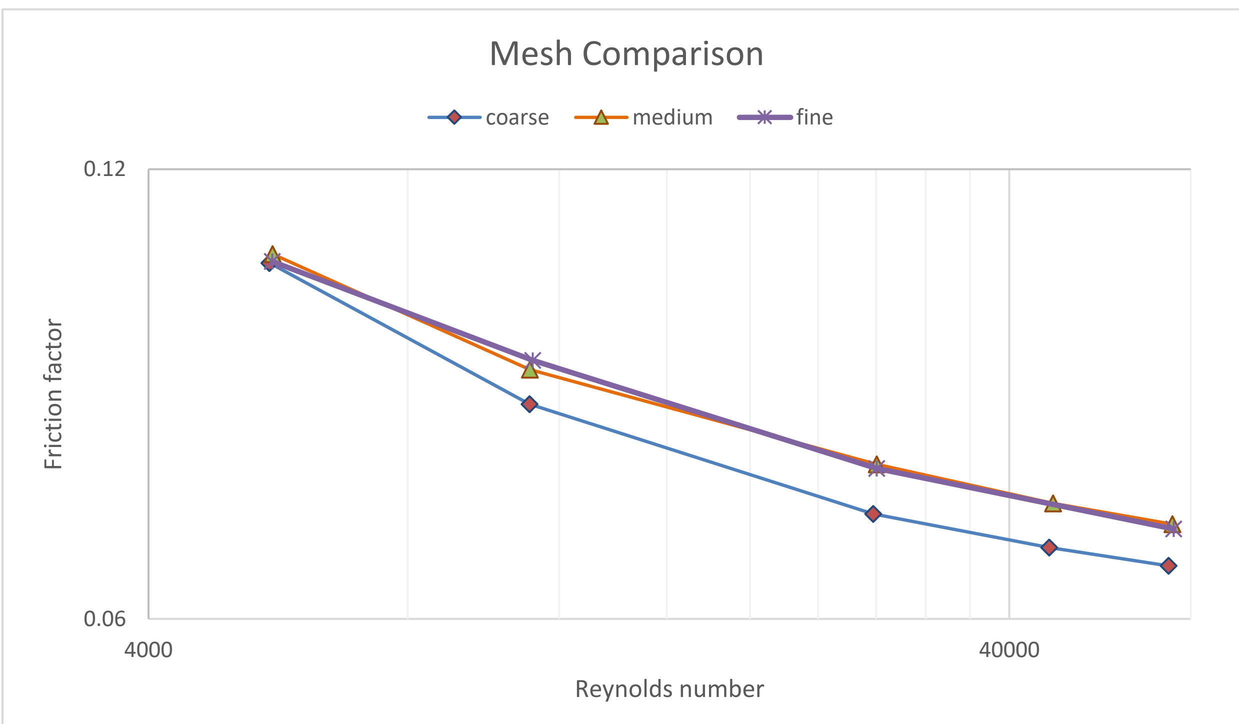


Figure 4.3: Pipe flow – Result with different mesh

In figure 4.3, result using different number of mesh element is shown. Coarse mesh contains about 72k cells, medium mesh has about 200k cells and fine mesh has about 350k cells.

All of them include structured mesh at horizontal part and unstructured mesh at cross-section. From the figure, it is observed that there is little or no change in result for medium and fine mesh, however there is a slight deviation at higher Reynolds number with coarse mesh. To obtain mesh independent simulation and efficient computation, medium mesh was selected for further study.

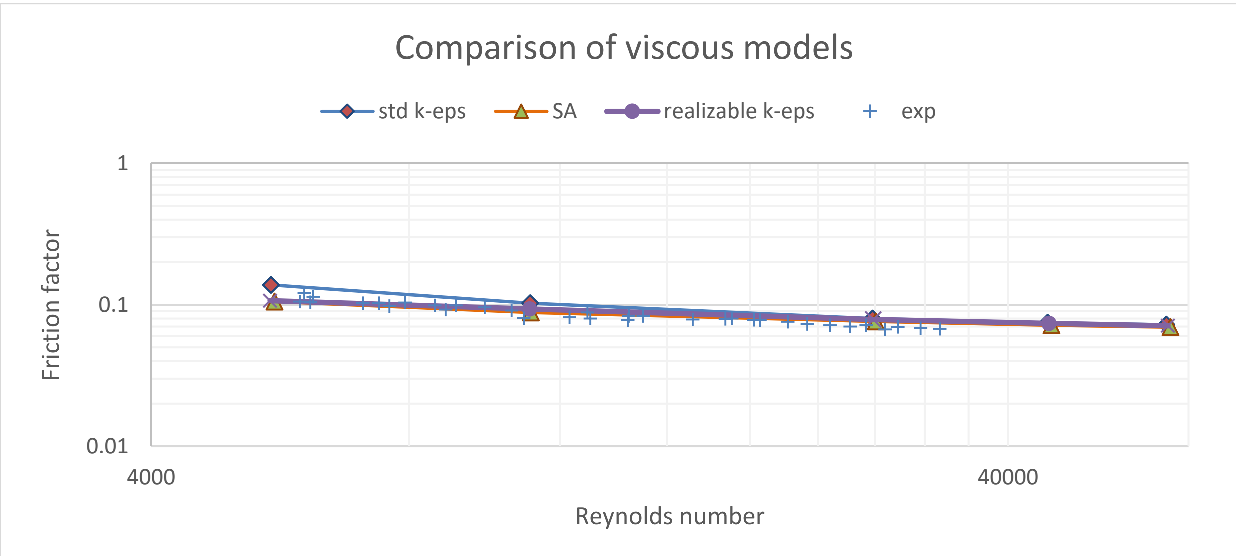


Figure 4.4: Pipe flow – Viscous model comparison at increasing Reynolds number

In figure 4.4, three different viscous model is compared. Standard k-eps and SA model has nearly same computational time. However, Realizable k-eps turbulence model with enhanced wall treatment takes nearly twice computational time. From observation, we notice that Realizable k-eps turbulence model and SA has nearly same result. However, standard k-eps model shows a slight deviation at low Reynolds number. So, Spalart-Allmaras model was chosen for further study. The experimental data was extracted using plot digitizer. So, the plot was not scaled in the middle for risk of losing graphical accuracy.

Figure 4.5 and figure 4.6 shows, velocity contour and pressure contour at pipe cross section for Reynolds number 10000. Reynolds number from 4000 to 60000 for pipe flow was analyzed.

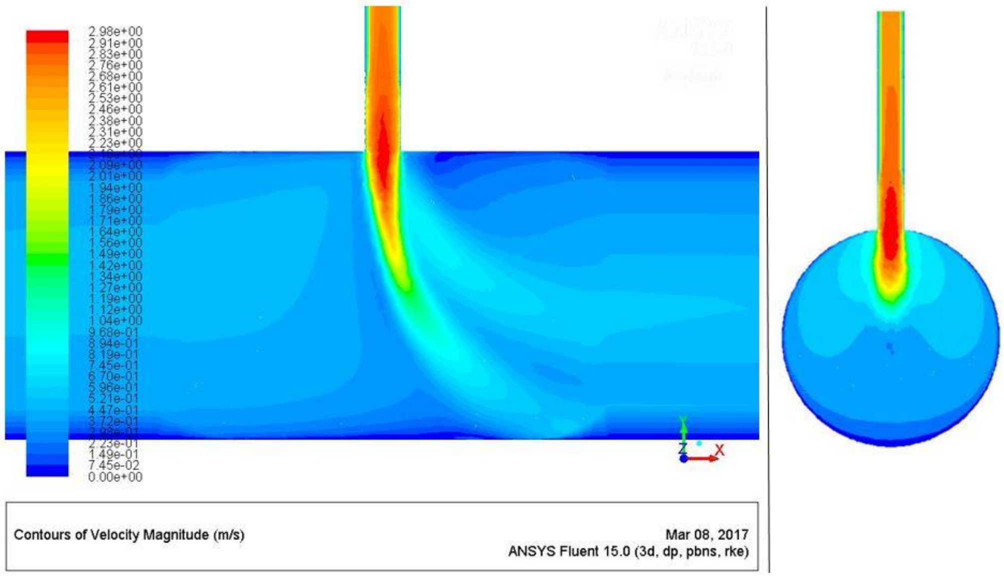


Figure 4.5: Velocity Contour at pipe cross-section for “Re=10000”

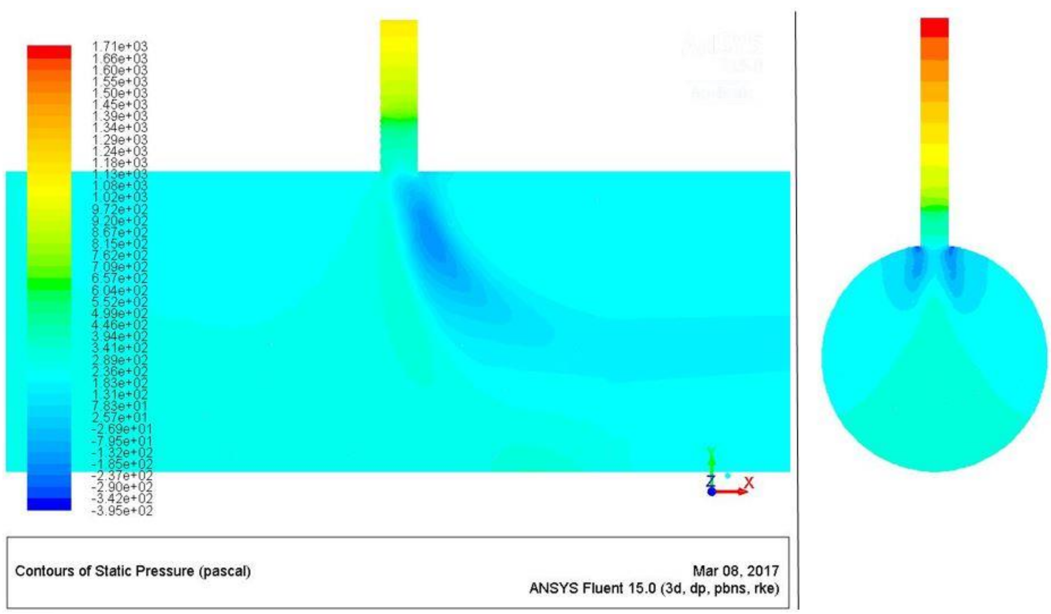


Figure 4.6: Pressure Contour at pipe cross-section for “Re=10000”

Then we moved on to 5 inlet steady-state 2D problem to analyze the characteristic flow behavior at different inlets. Figure 4.7 shows the velocity contour of 5 inlets at the beginning of production.

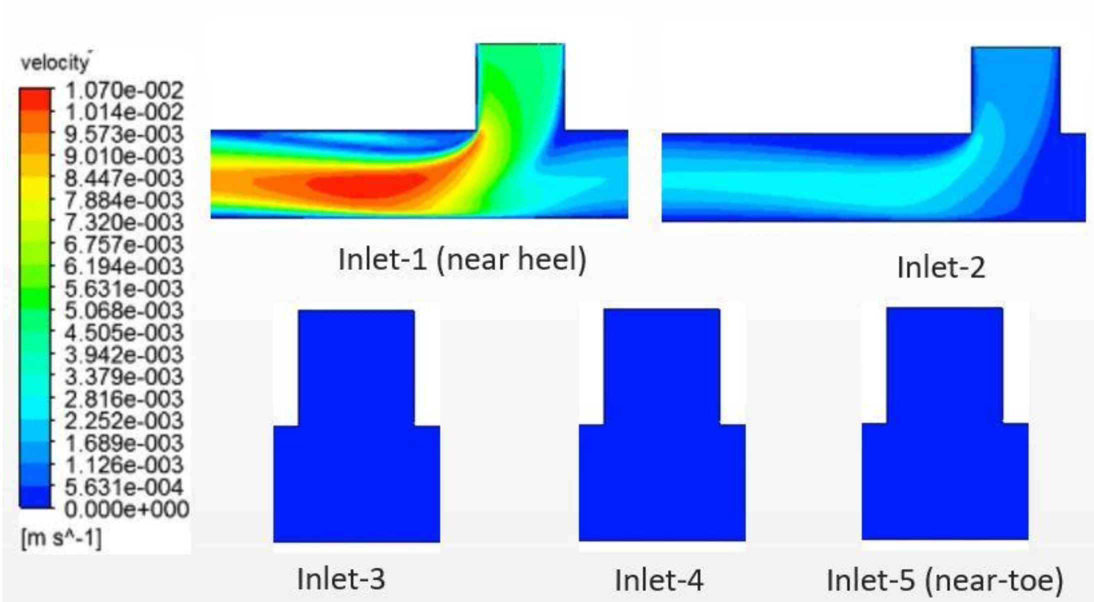


Figure 4.7: Velocity Contour at 5-inlets for h'=1

As we can see, there is almost no production from inlet 3,4 and 5 at the beginning of production.

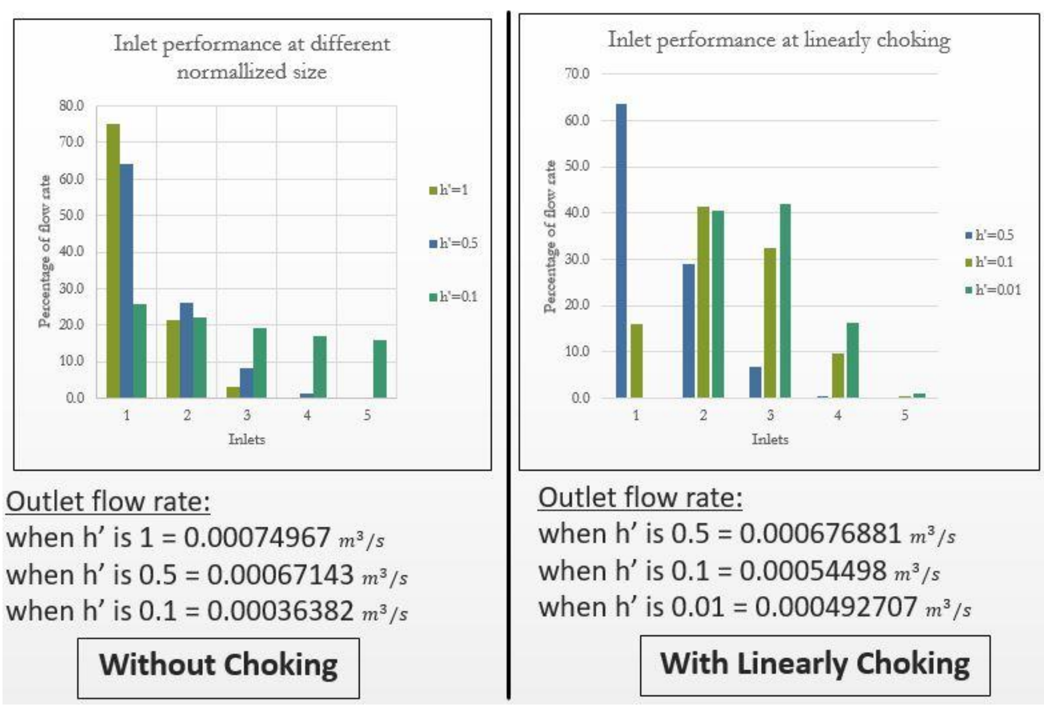
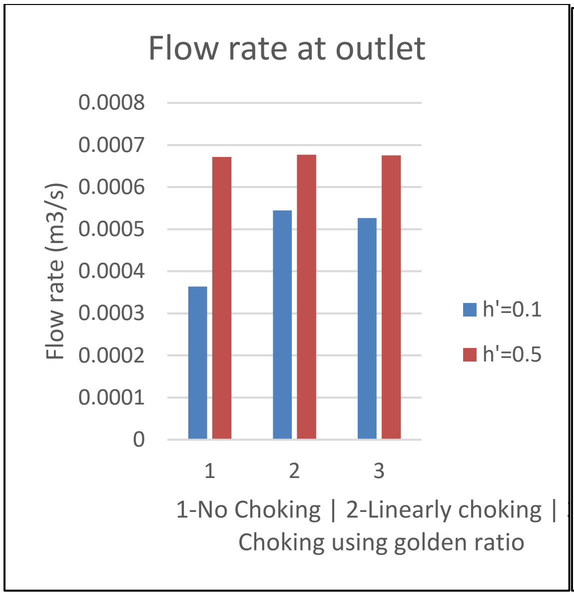


Figure 4.8: 5-inlet performance at different normalized inlet size

Here, with linear choking, mid inlets flow rate increased and also for same h', we have a higher outflow rate than without choking.



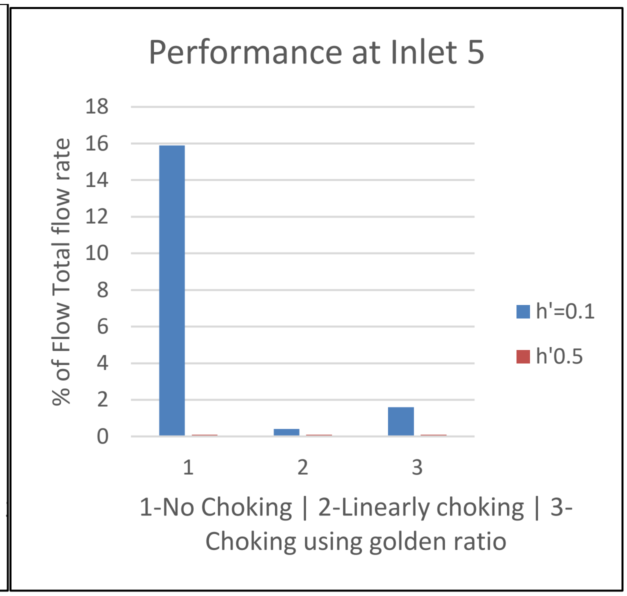


Figure 4.9: 5-inlet flow rate at outlet and inlet 5

For transient analysis, a udf was prepared. There were 5 udf for 5 inlets to implement hydraulic pressure loss with time depending on discharge. However segmentation fatal error was occurring. The udf code is as follows.

```
#include "udf.h"
/* Defining constants */
# define RHO 998.2
# define G 9.8
# define P0 0.05
# define A 0.125 /* cross sec area */
/* Change Zone IDs for each inlet */
# define ID 6 /* Thread ID */
DEFINE_PROFILE(in1,thread,index)
{
Thread *t;
face_t f;
real last_vol=0, new_vol=0, P=0;
real h;
real mass_flow=0;
Domain *d;
real present_time = CURRENT_TIME;  /* getting real current flow time */
real time_step = CURRENT_TIMESTEP; /* getting real current physical timestep size */
t = Lookup_Thread(d,ID); /*inlet thread id*/
if(present_time<= time_step){
```

```
/*at time=0*/
begin_f_loop(f,t)
{
F_PROFILE(f,t,index) = P0; /* initial pressure */
F_UDMI(f,t,0) = A * P0/(RHO*G); /* initial volume */
}end_f_loop(f,t)
}
else{
begin_f_loop(f,t){
mass_flow += F_FLUX(f,t);
}end_f_loop(f,t)
printf("Mass flow rate at i1= \%f\n", mass_flow);
begin_f_loop(f,t){
        last_vol = F_UDMI(f,t,0);}end_f_loop(f,t)
        new_vol = last_vol + (mass_flow * time_step / RHO);
/*as mass flux is negative at inlet */
        h = new_vol/(A);
P = RHO * G * h; /*initial volume */
begin_f_loop(f,t){
F_PROFILE(f,t,index) = P;
F_UDMI(f,t,0) = new_vol; /*assign new volume to memory */
}end_f_loop(f,t)
}
}
```

Chapter 5: Conclusion

Heel-toe effect in horizontal wellbore was understood and analyzed with the help of CFD study. Different meshing techniques was learned and applied. Understood advantages of different turbulence models. Learnt a great deal about UDF coding using fluent macros. Different inlet sizes were analyzed and studied in horizontal wellbore to understand their effects in heel-toe effect.

Future Work

Porous volumetric pressure loss consideration would be more accurate. Needto run transient analysis in 3D pipe geometry.

References

[1] B. J. Dikken, "Pressure Drop in Horizontal Wells and Its Effect on Production Performance," *J. Pet. Technol.*, vol. 42, no. 11, pp. 1426–1433, 1990.

[2] H. Yuan, C. Sarica, S. Miska, and J. P. Brill, "An Experimental and Analytical Study of Single-Phase Liquid Flow in a Horizontal Well," *J. Energy Resour. Technol.*, vol. 119, no. 1, p. 20, 1997.

[3] M. Dusseault and R. Jackson, "Seepage pathway assessment for natural gas to shallow groundwater during well stimulation, in production, and after abandonment," *Environ. Geosci.*, vol. 21, no. 3, pp. 107–126, 2014.