CFD Based Analysis of a Single-Phase Flow inside Coiled Tubing String

Mohamed Ben Abdullah*, Samah Alghoul*,
Abdulmaged Shati (*), Elhadi Dekam***
***Mechanical Engineering department, Faculty of Engineering, University of Tripoli**
****Mechanical Engineering department, Faculty of Engineering, University of Zawia**

Abstract

Coiled Tubing (CT) is well known in well intervention and servicing oil and gas wells. Experiments and testing models have been conducted for a better understanding of the Coiled Tubing flow. The use of Computational Fluid Dynamic (CFD) software has assisted greatly in studying the CT flow patterns, obtaining velocity profiles, pressure and drag histories, and discovering the secondary flow regimes. This paper

(*) Email: a.shati@zu.edu.ly

has established the procedures and applied the appropriate CFD technique to evaluate an isothermal Newtonian laminar fluid flow model inside the Coiled Tubing Unit (CTU) utilized in the oil and gas wells services. This CT strings are analyzed by using the free "OpenFOAM" software to investigate a single-phase flow inside the CT string and to get a quantitative behavior estimate of the fluid flows in the three different segments of the CT string.

The CFD results show that while the flow in the CT straight section is a parabolic, the flow inside the Gooseneck and the Reel shows a rotational flow nature with a maximum velocity distributes in the outer side of the CT, where the velocity profile peaked at the top side of Tube Guide section of the Gooseneck and reaches a value of almost 25% higher than the maximum velocity at the middle of the Reel. Secondary flow profile is observed clearly in Gooseneck and Reel. The frictional pressure-loss gradient in the CT straight section is 1.1 Pa/m, where it increases substantially in the Reel with a pressure loss gradient of 2.1 Pa/m, higher by 101% than the CT straight section followed by the Gooseneck with a pressure loss gradient of 1.34 Pa/m higher by 25% compared to the straight CT section pressure loss. The results of the simulation are in agreement with the published results; that is obtained by using ANSYS-Fluent commercial simulation software.

Keywords: *CFD; Coiled Tubing; Oil and Gas Wells Services; OpenFOAM software, Laminar Flow, Pressure gradient.*

Introduction

Coiled Tubing (CT) is a circular string used in the Coiled Tubing Unit (CTU) utilized in the oil and gas service industry and it is well known as Coiled Tubing Services (CTS). Figure 1 displays a well Coiled Tubing Unit (CTU). The CTS involves forcing a CT string into an oil or gas well using an injector head to perform various activities. The CT string can conduct fluid, so this allows it to provide many different functions in servicing oil and gas wells, as introduced by Job Execution Training (JET) Manual 16 [1]. This manual briefly discusses the history of coiled tubing (CT) and presents different types of CT applications carried out by "Schlumberger Company" CT Well services. The manual also clarifies the advantages of employing CT over other involvement methods. The applications of the CT include but are not limited to cleaning-up of debris and fill, drilling, cleaning-out sand production from a wellbore, prop pant fracturing, acidizing, scale removal, activating and placement of downhole tools [1].

Figure 1 Coiled tubing unit (CTU), JET Manual 16 [1].

Referring to the excellent reputation of the CT applications, a large volume of related research work has been published. Zhou et al. [2,3] presented a literature review of the research studies of the steady, fully-developed flow of Newtonian and non-Newtonian fluids related to coiled tubing. They briefly described the recent development in the experimental frictional pressure losses of various fluids in coiled tubing.

Zhou and Shah [4] simulated the flow of both Newtonian and Non-Newtonian fluids using different fluids and the ANSYS-Fluent software package. The unique flow patterns revealed by the simulations showed the maximum velocity shifted to the outer side of the tubing due to the centrifugal forces. The frictional pressure gradients estimated by the CFD simulations were verified by comparing them with previously done experimental studies. Blanco et al. [5] modeled an actual field data using a fluid slurry that is used in fracturing services, where this eliminates the need for experiential testing. The CT geometry studied in their paper is the CT on the Reel and straight CT pipe of 2 7/8” OD, where the goal of the analyses was to compare the actual recorded CT frictional pressures to the frictional pressure computed by the CFD simulation. The outcome of this study is that the CFD can be facilitated to estimate the CT frictional pressure with reasonable accuracy.

In a different study, Zhu [6] worked on simulating the flow of Newtonian fluid flow in helical Coiled Tubing using the CFD commercial software (ANSYS-Fluent). Laminar and turbulent flows were investigated with different inlet velocities. The CFD simulation results were in close agreement with the published correlations. Jayakumar et al. [7] conducted CFD simulations for vertically oriented helical coils, where they traced the movement of the fluid particles in the considered helical pipe. They determined the variation of the local Nusselt number along the length and circumference at the wall of the helical pipe. Rosine et al. [8] studied the flow phenomenon of the CT with special focus on the tubing guide in the Gooseneck using the CFD and ANSYS-Fluent software packages to study the fluid flow, particularly over the Gooseneck. The Gooseneck flow-velocity profile and flow pattern show that the maximum velocity distributes is on the outer side of the Gooseneck.

The Open Source Field Operation and Manipulation (OpenFOAM) computer package has numerous pre-built applications designed to solve problems in fluid mechanics and perform tasks that involve data manipulation [9]. One of the concerns of the OpenFOAM software [10] is related to computational fluid dynamics (CFD) for real, 3-dimensional problems in engineering, where a number of solver applications correspond to the combinations of transport equations are solved. Geveci et al. [11] and Ayachit [12] provided tools for extremely large scientific data analysis and visualization.

This present paper establishes the procedures in order to create and validate a laminar Newtonian fluid flow model for the case of the CT string by using the OpenFOAM software package. That is to analyze the single-phase flow inside the CT string in order to achieve a quantitative estimation of the behavior of the fluid flows related to the three segments of the CT String. The three segments of the Coiled Tubing Unit are the straight CT pipe section, the Gooseneck (tubing guide arch) and the CT reel section are investigated.

Methodology

Here, the tube geometry and its specifications, fluid characteristics and properties, and flow operating conditions with the appropriate boundary conditions are to be described in detail. The OpenFoam

Software is to be introduced. The considered geometry would be modeled, where three segments are considered; the straight CT section, Gooseneck CT pipe section, and the CT section on the Reel, each of which has its particular mesh generation. The solver is to be selected and run the developed code, leading to post-processing and result presentation.

The Physical Model and the Boundary Conditions

The standard configuration of the CTU shown in Figure 1 is divided into three segments, as shown in Figure 2. The first is the straight CT pipe section that will be inside the oil and gas wells, the second is the Gooseneck CT pipe section that will bend to be able to inject CT inside the well, and finally, the CT section on the reel is used for storage of the CT and transportation of the access pipe when moving from one well to another or to the operational base.

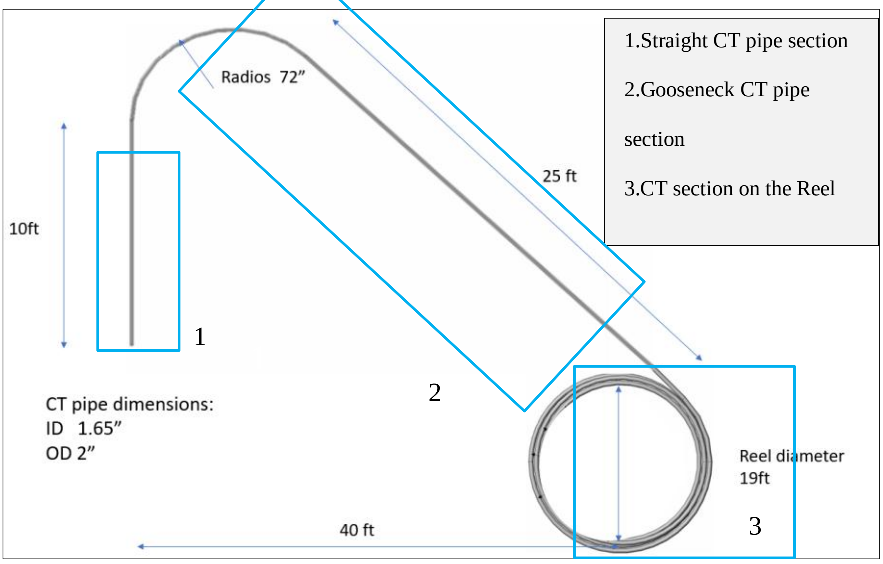

Figure 2 Geometries of CT string case studies.

The first section is the straight CT pipe, which is 10 ft length. The second section is the Gooseneck (tubing guide arch) with another 25 ft straight section. The tubing guide radius is 72 in and the angle of the tubing coming off the reel is 45°. The third section is the CT reel, which has a diameter of 19 ft and a width of 82 in. The CT string wraps consisted of one layer with four wraps on the reel and a 25 ft straight section coming off the reel.

The CT string under study has an outer diameter (OD) of 0.0508 m and an inner diameter (ID) of 0.042 m (2 inches nominal CT size). An inlet volumetric flow rate of water is $Q = 4.16\times10^{-3}\ m^3/min$ with an outlet pressure of $P_{outlet} = 0$, i.e., discharging to the atmosphere.

CFD Modeling

In this section, the OpenFOAM software is introduced in brief. The geometry and meshing process are explained in general and then they are discussed for the three different cases of the CT.

OpenFOAM Software

The Open Source Field Operation and Manipulation (OpenFOAM) version 17.02 is used to simulate different models in this work. OpenFOAM is an open source code written in C++ libraries that are used to create executable named applications, Greenshields [9]. These applications fall under two categories: solvers and utilities. The OpenFOAM contains many solvers and utilities that are able to solve many problems. It has a wide range of functionality with various pre-processing, solving, and post-processing tools built into the program. The most valuable feature in OpenFOAM is that users can write new solvers and utilities to be used in the program. The solver employed in this paper is a *simpleFoam* solver that is built for steady-state, incompressible and laminar fluid flow. The interface to the pre and post-processing are themselves OpenFOAM utilities consequently, this makes the software more cohesive and user friendly, Greenshields [9]. The main structure of the OpenFOAM, is shown in Figure 3.

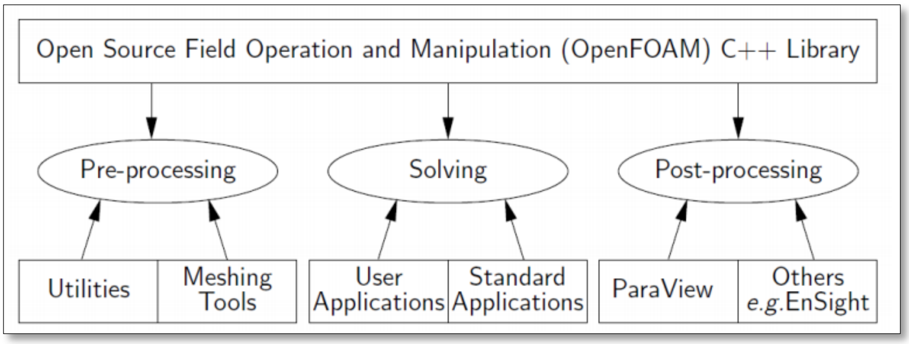

Figure 3 Review of OpenFOAM software structure, User Guide [10].

The OpenFOAM directory uses three main folders: 0, constant, and system. in the 0 folder, which is a time folder, all the initial conditions defined by the user are saved. in the constant folder, information about the mesh generation as well as the physical properties of the flow are saved. All the information regarding the solvers is stored in the system folder.

Modeling Geometry

Three different cases are modelled and run separately to simulate the CT pipe during a CT pumping operation; the straight CT pipe section, the Gooseneck CT pipe section, and the CT section on the Reel.

Straight CT Pipe Section

The straight CT pipe section is created using the *blockMeshDict* utility in the OpenFOAM software. The file is stored in the dictionary under the system folder. In this dictionary, it is possible to code different but not complex geometries consisting of blocks of hexahedral cells. In the *blockMeshDict*, it is also necessary to write the boundary conditions. In this case, the inlet and the outlet of the flow are defined as *patches* and are represented in the dictionary with these names: *inlet* and *outlet*, respectively. For the other boundaries, the type used as a *wall* and was names *pipe*. The CT pipe section is shown in Figure 4.

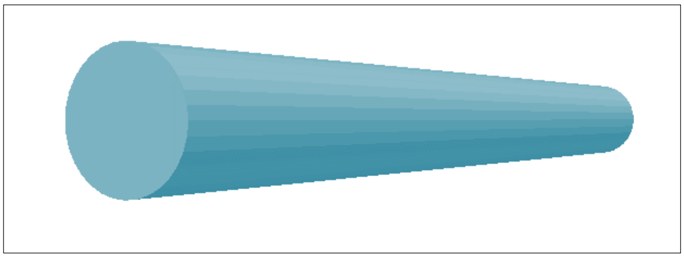

Figure 4 Straight CT Pipe Section.

Gooseneck CT pipe section

ANSYS R19.2 software is used to draw and create Gooseneck CT pipe section geometry. This was done by creating a new project schematic, then dragging and dropping Mesh from the list of component systems to the drawing space, creating New Design Modeler Geometry. Finally, the Gooseneck CT pipe section is fully drawn as per the actual Coiled Tubing Unit (CTU). The Gooseneck case study is shown in Figure5.

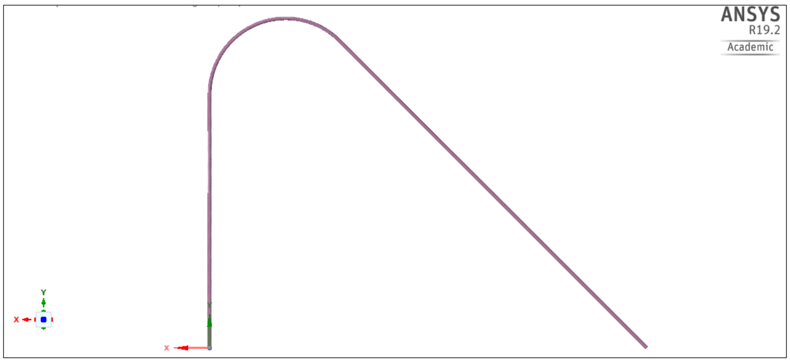

Figure 5 Gooseneck CT pipe Geometry created by ANSYS R19.2.

CT section on the Reel

The same procedures have been followed to generate the CT section on the Reel using the ANSYS R19.2 software, as shown in Figure 6. Modeling the minimum number of wraps reduces the required computer capacity and also dramatically reduces the simulation time. Once the pressure gradient is established per wrap of tubing, the CT reel pressure could be calculated based on the number of wraps per layer and the number of layers, based on the tubing reel capacity [5].

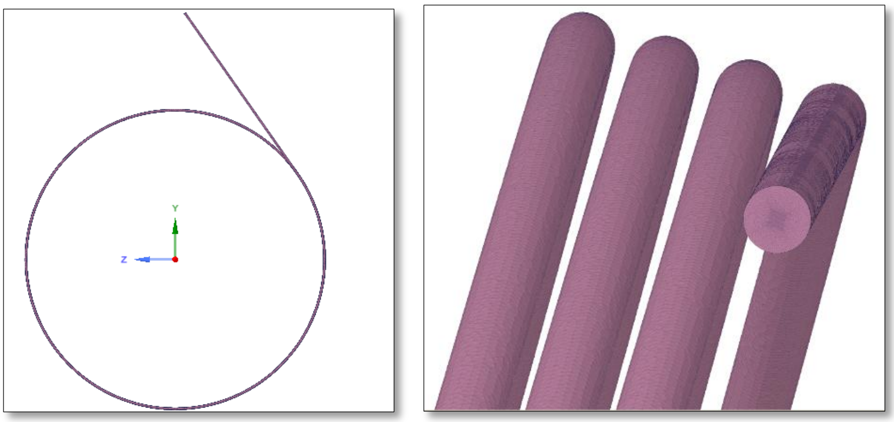

Figure 6 CT section on the Reel Geometry created by ANSYS R19.2.

Mesh Generation

The workflow used in blockMesh to generate a multi-block Mesh is shown in Figure 7. The blockMeshDict files in the system directory contain all the coordinates of the vertices, edges, blocks and patches in the blockMeshDict. To create the simple meshes, blockMesh command is used to create a polyMesh directory that contains a full description of the case mesh. After running the blockMesh command in OpenFOAM it's possible to visually verify it in ParaView in the geometry as shown in Figure 8.

In the second method, due to the complexity of the geometries, ANSYS software is used to generate the models for the Gooseneck CT pipe section and CT section on the Reel. Based on the length of the section and the size of the geometry, the number of the cells will be determined by the OpenFOAM meshing tool and it can be changed for more refinement.

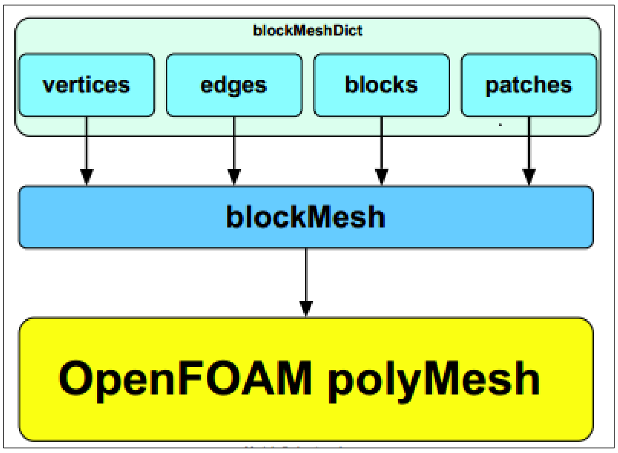

Figure 7 Mesh generation using blockMesh (blockMesh workflow).

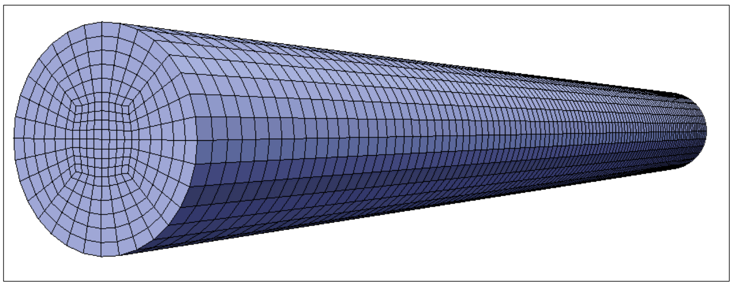

Figure 8 The mesh of Straight CT Pipe in ParaView.

CFD Simulation

OpenFOAM operates in 3-D Cartesian coordinate systems, and all geometries are generated in 3-D models. Generally, it solves the cases in 3 dimensions by default. However, the 3-D problem can be created and solved as 2-D one by using some empty boundary conditions in OpenFoam.[9]

While many schemes exist for solving fluid flow models, the selected scheme to solve these models, described in this work, is the Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) scheme. The solver chosen to solve this problem is simpleFoam. After choosing the solver, it is necessary to select the corresponding preconditioned conjugate gradient solver. OpenFOAM recognizes some well-known pre-conditioners. However, the one selected for this work was the GAMG (Generalized Geometric-algebraic Multi-grid).

The mesh that is generated and used in the present simulations is structured hexahedral. Increasing the number of cells will likely improve the solution accuracy, but at a higher computational cost (Grid independence has been done during the course of the work). The most common mesh quality metrics are: orthogonality, skewness, aspect ratio and smoothness. Utility *checkMesh* produces a full report with mesh quality metrics and element count statistics.

Time Step and Data Output Control

The SimpleFoam is significantly more sensitive to the Courant number than other models that calculate simple fluid flows. In the region of the interface, it is preferable to have a Courant number not exceeding 0.5, User Guide [10]. A fixed time step is set in *controlDict*. It is applied by setting the keyword *writeControl timeStep* to allow results to be written at a fixed time intervals. Results are allowed to be written at fixed times *writeInterval* of 50 iterations. In the OpenFOAM, the Courant number C_0 is defined as:

$$C_0 = \frac{\delta t|U|}{\delta x} \qquad (1)$$

Where δt is the time step, U is the magnitude of the velocity through that cell and δx is the cell size in the direction of the velocity

The first data inputs must be read from a directory named *0*. The time required for the solution to converge differs for different CT case studies, mesh refinement and layouts. The optimum time will be obtained after several trials. In these simulations, to optimize the CPU usage, a maximum endTime of 2000 seconds is applied.

Running the code

To run the simulation using *simpleFoam* Steady-state, incompressible, laminar flow, using the SIMPLE algorithm, the code is run by typing *simpleFoam* into the software shell. The sub-directory *controlDict* has a function specified under the application name *simpleFoam*. Residuals are logged during a separate run of the case, where functions in *controlDict* are converted to text.

Post-processing

According to Geveci [11], ParaView is used to post-process results and produce figures and plots for all output results. Any data saved in VTK format can be read in ParaView. It is simply launched by typing paraFOAM in the terminal window.

Results and Discussions

It is very helpful to employ the considered OpenFOAM software for simulation and ParaView software for post-processing and visualizing the results for different case studies with different geometries and divide them into segments to observe and analyze the CT flow phenomena.

The Straight CT Pipe Section

The pressure drop per fluid density along the straight pipe is shown in Figure 10. From the figure, the total pressure drop through the CT section is 0.0108 m^2/sec^2 which equals to 10.76 Pa along the 10 m pipe. Thus, the frictional pressure loss gradient along the straight CT pipe section is 1.076 Pa/m.

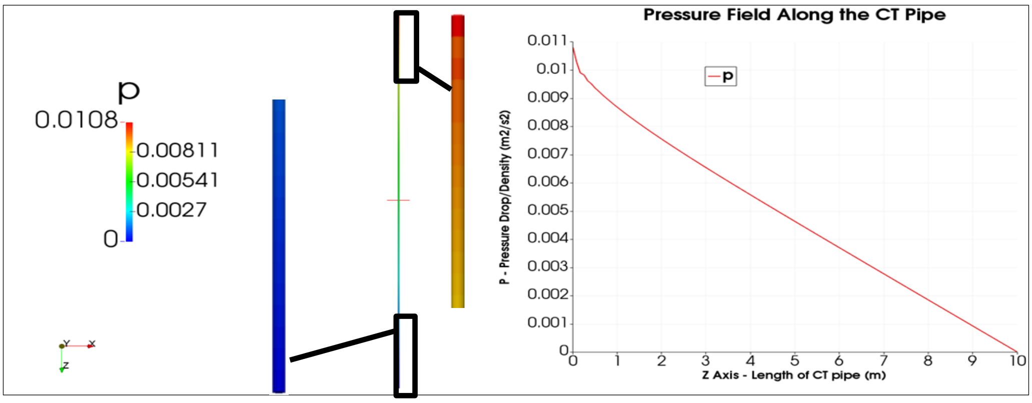

Figure 9 Pressure field and pressure gradient (m^2/sec^2)

Figure 11 shows the velocity profile and contours at the segment exit cross-section, 10 m from the CT. The velocity profile has a laminar parabolic shape with a velocity peak at the center of the pipe with a value of 0.1 m/sec. The velocity field at the outlet of the CT segment, Z= 10 m, represents a fully-developed laminar pipe flow.

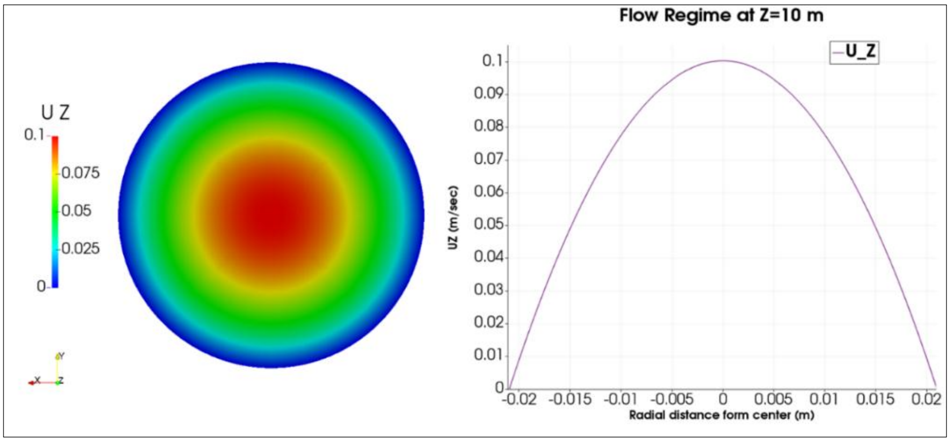

Figure 10 Velocity profile and contours of velocity at the outlet of the CT pipe (m/sec).

Gooseneck CT Pipe Section

Figures 12-14 show the simulation results of water flow in the 0.0508 m diameter CT with a flow rate of 4.16×10^{-3} m^3/min. Referring to Figure 12, the static pressure is assumed to be zero (gage) in the outlet of the Gooseneck segment, where the OpenFOAM computed the static pressure profile shown in the Figure. It is noticed that total static pressure drop per density through the Gooseneck section is 0.0143 m^2/sec^2, which is equal to 14.25 Pa, while the frictional pressure-loss gradient is 1.34 Pa/m. This is higher by 25% when compared to the straight CT section pressure loss gradient.

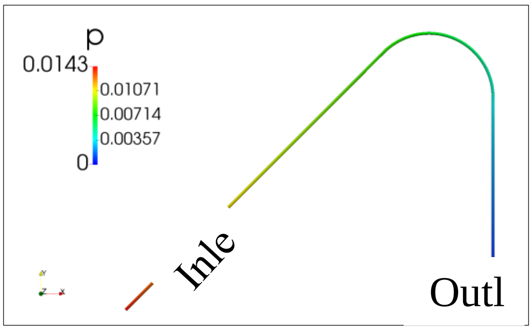

Figure 11 Pressure Gradient in the Goozneck (m^2/sec^2).

The velocity field at the tube guide section of the Gooseneck is shown in Figure 13. The velocity profile is displaced from being a parabola shape with a peak at the top side of the tube guide, due to the action of the centrifugal force on the fluid flow. The maximum velocity reached a value of 0.091 m/sec.

The high velocity region is shifted toward the upper side of the tubing due to the centrifugal forces. As a result, a pressure differential is generated across the tube, leading to the secondary flow to occur within the tube cross section, where it flows towards the high-speed region. Figure 14 shows the secondary flow streamlines form a pair of symmetrical vortices, which have been called Dean vortices, Zhou [2].

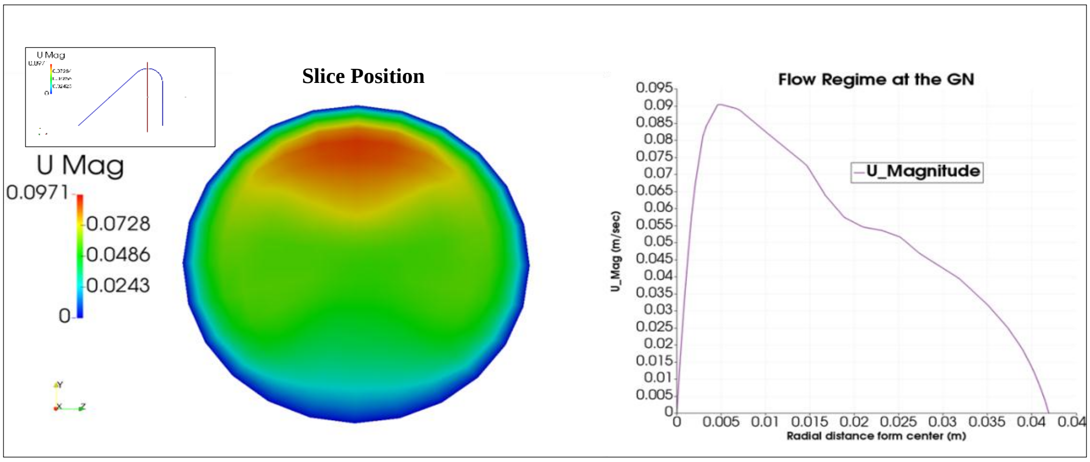

Figure 13 Velocity profile and contours at Tube Guide of the Gooseneck (m/sec).

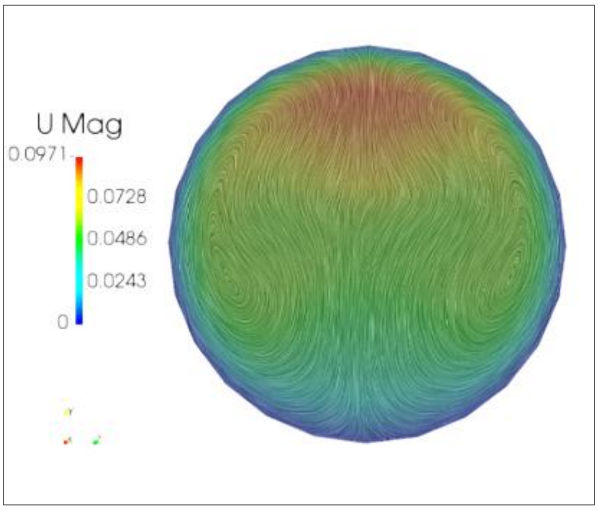

Figure 14 Secondary flow at the for the case shown in figure 13.

CT Section on the Reel

Figure 15 shows the pressure gradient through the Reel of the CT. From the resulted pressure profile, it is noticed that the total pressure drop per density through the CT Reel section is 0.139 m^2/sec^2. This is equal to 138.6 Pa, while the frictional pressure loss gradient is 2.1 Pa/m. This is higher by 101% when compared to the straight CT section pressure loss.

Figure 16 illustrates that the maximum velocity occurs on the outer side of the CT reel due to the centrifugal forces. Figures 17 and 18, respectively, show the velocity contours of the upper and lower sections of the reel in four wraps of the CT pipe, indicating the inlet and outlet of the reel.

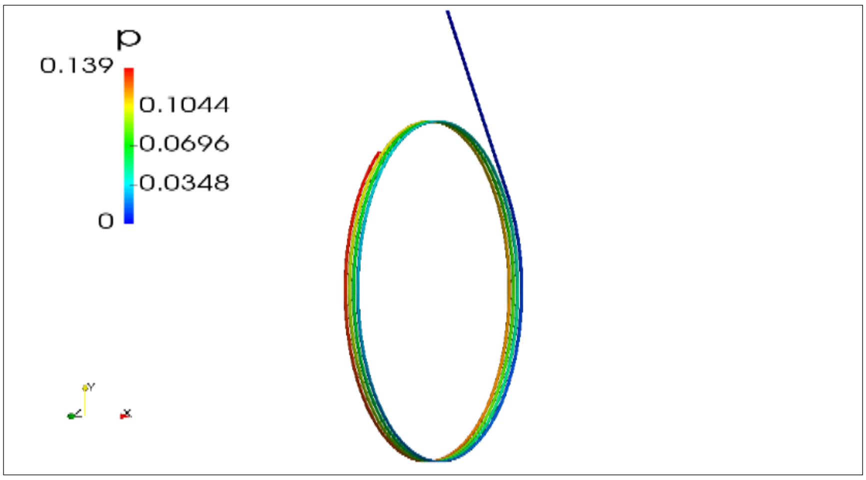

Figure 15 Pressure Gradient in the Reel (m2/sec2).

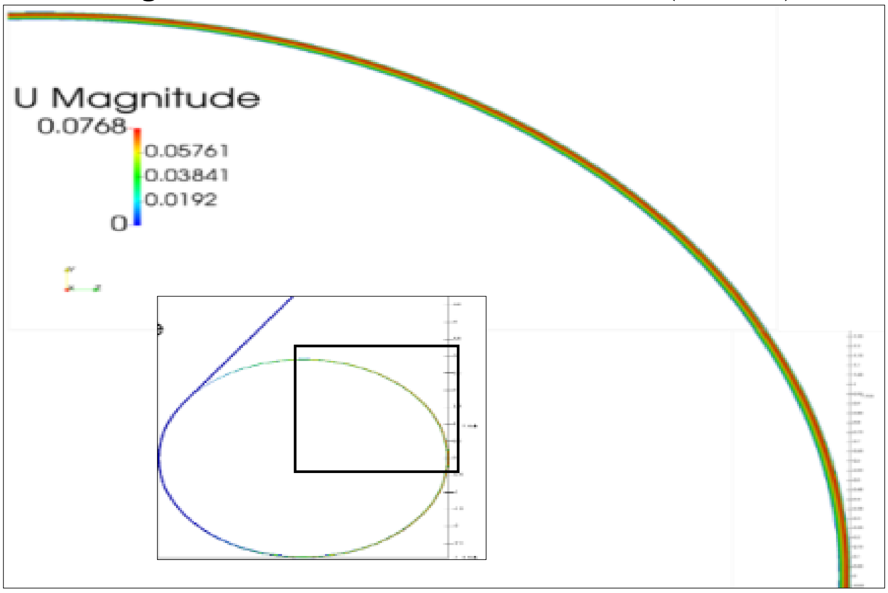

Figure 16 Velocity Contours of the Coiled Tubing Reel (m/sec).

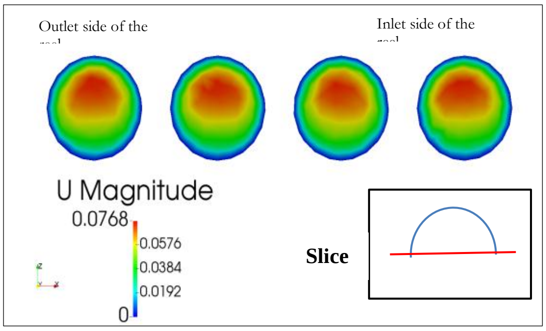

Figure 17 Velocity Contours of the upper section of the Reel.

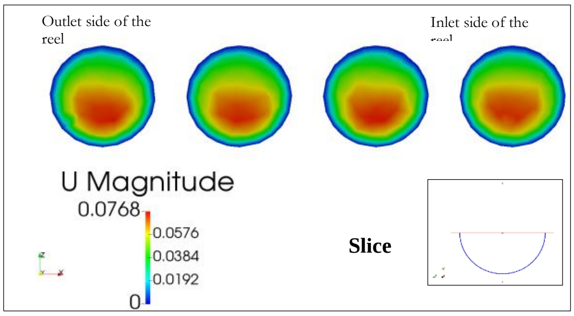

Figure 18 Velocity Contours of the lower section of the Reel.

Figures 19 and 20 show the secondary flow profiles at the upper and lower sections of the reel, respectively.

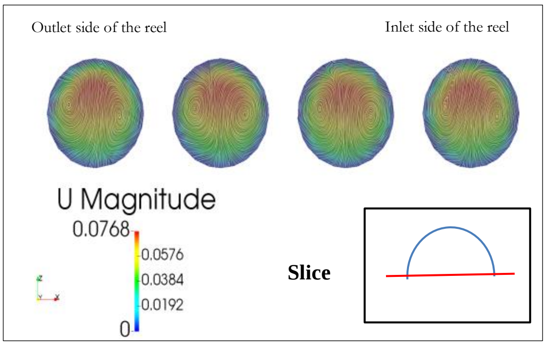

Figure 19 Velocity profile secondary flow at the upper section of the Reel.

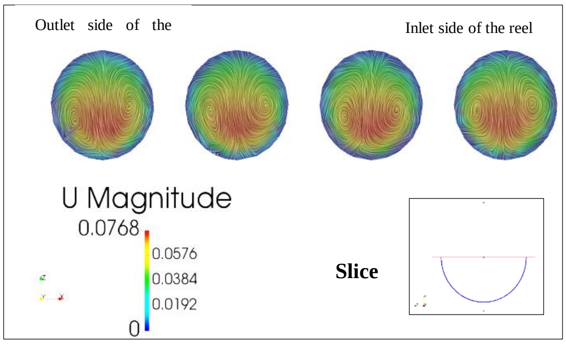

Figure 12 Velocity profile secondary flow at the lower section of the Reel.

The velocity field at the middle of the Reel section is shown in Figure 21, where the velocity profiles are no longer parabolic. The peak velocity is at the outer side of the CT reel due to the action of the centrifugal force on the fluid flow, where the maximum velocity reached a value of 0.073 m/sec.

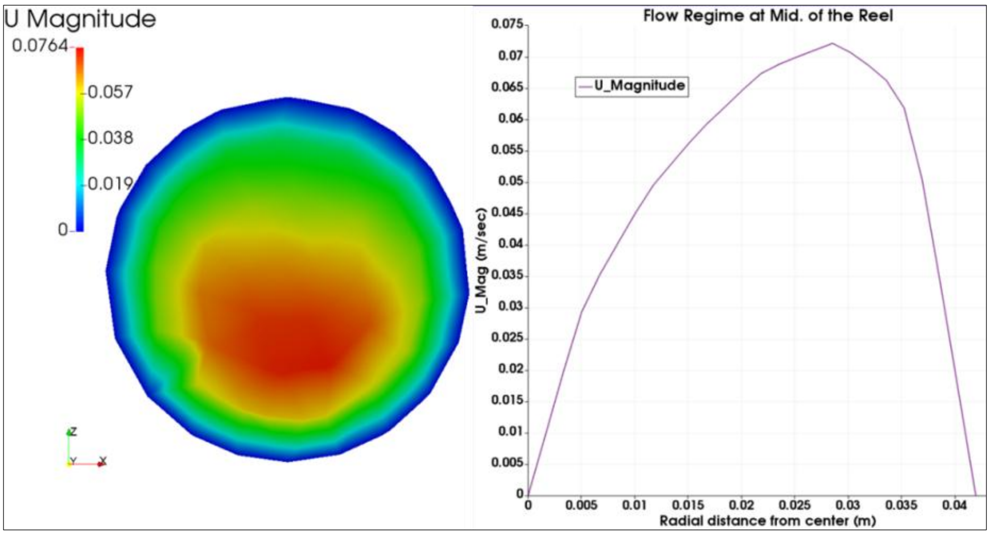

Figure 13 Velocity profile and contours of velocity at middle of the Reel section.

Verification of the Obtained Results

Referring to the introduction, the cases studied with the operating conditions considered in the present study are the first research work done on the CT using the OpenFOAM. So, a verification of the results seems to be required. The specifications of the verification model were obtained from Zhu [6].

According to the laminar flow, Srinivasan et al. [13] obtained a correlation equation to calculate the pressure drop. Thus, the pressure drops obtained using Srinivasan et al. [13] correlation equation, and CFD simulations using OpenFOAM with FLUENT-ANSYS software will be compared to validate the simulation accuracy. The compared results are shown in Table 1.

Table 1 The pressure gradients by the CFD simulations and correlation.

Simulation	Dimension of CT	Number of Grid hexahedra cells	Type of Meshing	CPU time (hours)	CFD simulations		Srinivasan correlation Pressure drop (Pa/m)	OpenFOam vs Fluent Error %	OpenFoam vs Srinivasan Error %
					OpenFoam Pressure drop (Pa/m)	Fluent Pressure drop (Pa/m)			
Full 3D Pipe	Reel	200,000	Coarse	2.26	0.399	0.396	0.376	1%	6%

From the table, the OpenFOAM CFD results were in close agreement with the ANSYS and Srinivasan et al. [13] correlation.

Figures 22-24 show the comparison between OpenFOAM and FLUENT ANSYS commercial software for velocity contours and velocity profiles, respectively, on the outlet face for an inlet velocity of 0.02 m/s. From the figures, the results are almost identical for OpenFOAM and FLUENT ANSYS commercial software.

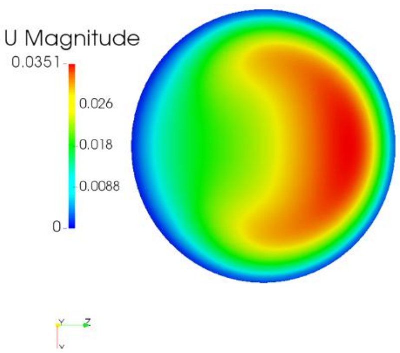

Figure 14 OpenFOAM results Contours of velocity (inlet velocity v = 0.02 m/s).

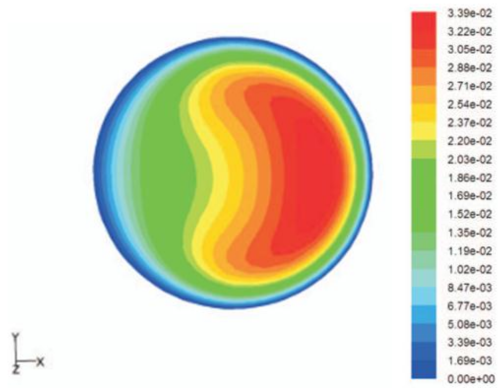

Figure 15 ANSYS results Contours of velocity (inlet velocity v = 0.02 m/s), Zhu [6].

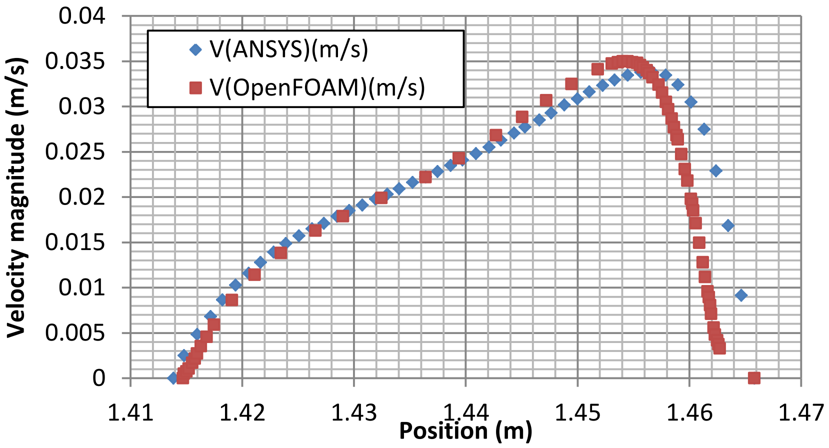

Figure 24 Comparison between OpenFOAM (from this work) and FLUENT ANSYS [6] velocity profile

Conclusions

Engagement of the Coiled Tubing (CT) in well intervention and servicing oil and gas wells has grown substantially over the past few decades. Consequently, the flow through the Coiled Tubing has been extensively studied by researchers to reveal the category, characteristics, and behavior of the flow through the Coiled Tubing. Experiments and testing models have been adapted for a better understanding of the Coiled Tubing flow, where the use of Computational Fluid Dynamic software has aided greatly in reviewing the CT flow patterns, visualizing velocity profiles and discovering secondary flow regimes.

The present work concerning an isothermal Newtonian laminar fluid flow model inside the Coiled Tubing Unit (CTU) utilized in the oil and gas wells services. These CT strings are analyzed by using the free "OpenFOAM" software to investigate a single-phase flow inside the CT string and to get a quantitative behavior estimate of the fluid flows in the three different segments of the CT string. Three segments of the Coiled Tubing Unit; the straight CT pipe, the Gooseneck (tubing guide arch), and the CT reel segments are considered and investigated, where an appropriate mesh is generated for each one. The study shows the following results:

- The CFD results show that while the flow in the CT straight section is laminar with parabolic velocity profiles, the flow inside the Gooseneck and the Reel shows a rotational flow nature with the maximum velocity distribution in the outer side of the CT.
- The velocity profile peaked at the top side of the tube guide section of the Gooseneck and reaches a value of 0.091 m/sec,. which is almost 25% higher than the maximum velocity at the center of the Reel 0.073 m/sec.
- Secondary flow profile is observed clearly in Gooseneck and Reel.
- The frictional pressure-loss gradient in the CT straight section is 1.07 Pa/m, where it increases substantially in the Reel with pressure loss gradient of 2.1 Pa/m, higher by 101% than the CT straight section.
- The pressure loss gradient in the Gooseneck is 1.34 Pa/m higher by 25% compared to the straight CT section pressure loss.
- The comparison with previous recognized published results show that, the Srinivasan correlation and OpenFOAM CFD results are in full agreement with the published results obtained by the ANSYS-Fluent commercial software simulation.

References

JET Manual 16, Introduction to Coiled Tubing, Version: 1.0, Well Services Training and Development, IPC Schlumberger private, February 22, 2007.

Y. Zhou and S. Shah, “Fluid Flow in Coiled Tubing: A Critical Review and Experimental Investigation TM,” Proc. Can. Int. Pet. Conf., no. 2, pp. 1–15, 2002.

Y. Zhou, Subhash N. Shah, Fluid Flow in Coiled Tubing: A Literature Review and Experimental Investigation, Journal of Canadian Petroleum Technology 43(6), DOI: 10.2118/04-06-03, , June 2004.

Y. Zhou and S. N. Shah, Fluid Flow in Coiled Tubing : CFD Simulation, The Canadian International Petroleum Conference, Calgary, Alberta, Paper Number: PETSOC-2003-212, June 2003.

I. Blanco, M. Bailey, and R. Rosine, “Comparison of computation fluid dynamics of slurry flow in coiled tubing to field data,” Soc. Pet. Eng. Coiled Tubing Well Interv. Conf. Exhib. 2007, pp. 338–343, 2007.

Z. Y. Zhu, “CFD simulation in helical coiled tubing,” J. Appl. Sci. Eng., vol. 19, no. 3, pp. 267–272, 2016.

J. S. Jayakumar, Sanjay Madhusudan Mahajani, J. C. Mandal, Kannan Iyer, Vijayan Pallippattu Krishnan; CFD analysis of single-phase flows inside helically coiled tubes, Journal of Computers and Chemical Engineering 34(4):430-446, April 2010.

R. Rosine, M. Bailey, and I. Blanco, “SPE 94057 Fluid-Flow Phenomena in CT Using CFD,” 2005.

Christopher J. Greenshields, The OpenFOAM Foundation Ltd., User Guide version 9, p-1-241, https://openfoam.org, 14th July 2021.

Open∇FOAM, The Open Source CFD Toolbox, User Guide, Version v1812, 7th December 2018.

B. Geveci et al., “The ParaView Guide Community Edition,” 2016.

Utkarsh Ayachit, The ParaView Guide Community Edition Updated for ParaView version 5.8, Published by Kitware Inc., February 18, 2020.

Srinivasan, P. S., Nandapurkar, S. S. and Holland, F. A., “Friction Factors for Coil,” Trans Instn Chem Eng, Vol. 48, pp. 156-161, 1970.