ENERGY ENGINEERING

Spring 2014

ME 536 – COMPUTATIONAL FLUID DYNAMICS

TERM PROJECT FINAL

CFD Analysis for 2D Diffuser Pipe Flow into the Storage Tank

Bekir Kağan Yavuz*

202007003

ARTICLE INFO
Presented 5 June 2014

Keywords
CFD
Diffuser Pipe
API Storage Tanks
Laminar Flow
ANSYS Fluent
Velocity Profile

ABSTRACT
This study is tried to investigate the flow characteristics and velocity imaging of a diesel fuel through a diffuser pipe in a storage tank. The fuel in the tank shall be spread into a tank in a range of as wide as possible. The FLUENT software is used to get data and plots to understand situation and solve the problem.

Email: kaganyavuz87@gmail.com
bekiryavuz@iyte.edu.tr

Contents

1. INTRODUCTION

1.1. Fluid & Flow

Any liquid or gas or generally any material that cannot sustain a force when at rest and that undergoes a continuous change in shape when subjected to such a stress is called as fluid. [1]

The continuous and irrecoverable change of position of one part of the material relative to another part when under stress constitutes flow which is also a characteristic property of fluids. In contrast, the shearing forces within an elastic solid, held in a twisted or flexed position, are maintained and the solid undergoes no flow and can spring back to its original shape. [1]

1.2. Computational Fluid Dynamics

Computational fluid dynamics, usually abbreviated as CFD, is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows.

2. AIM OF THE STUDY

2.1. Description

In this study, perfomance of a diffuser pipe in a diesel oil storage tank is tried to be analyzed regarding its capability of diffusing the fluid into the tank. It is desired that the flow (extracted from slots) are spreading in the tank as much as possible. The analyze will be carried out with the assumptions below:

- In 2D system
- Symmetric modelling
- Avoiding turbulence taking Re number into consideration
- Steady-state (The tank is full of fluid and there is flow inward and outward)
- Flow is newtonian [2]
- Viscous dissipation is neglected.
- Fluid is incompressible.
- No Turbulence, No Heat Generation, No Body Force

The system is tried to be examined is as shown below:

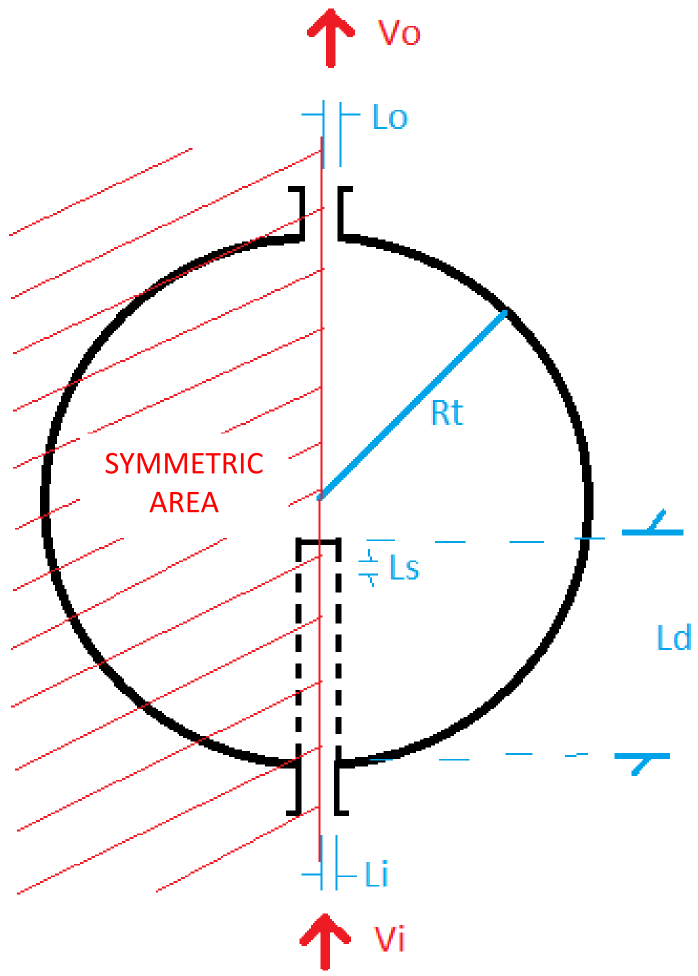

Figure 1: Schematic diagram of the model

Inlet Velocity	: Vi = 1.5 m/s
Outlet Velocity	: Vo = 1.5 m/s
Inlet Pipe Radius	: Li = 150 mm
Outlet Pipe Radius	: Lo = 150 mm
Length of Diffuser	: Ld = 21000 mm
Length of Slots	: Ls = 150 mm
Spacing of Slots	: Lss = 850 mm
Number of Slots	: 20
Radius of Tank	: Rt = 30000 mm

The properties of fluid in this study which is diesel oil are given below [3]:

Temperature	: 50
Density	: 0.89 g/cm3 (@ 55°C)
Viscosity	: 76.2 cP (@ 55°C)

2.2. Governing Equations

In order to find velocity, mass and pressure values, the governing equations include the following conservation laws of physics:

2.2.1. Conservation of mass: (Continuity Eqn.)

$\partial u/\partial x + \partial v/\partial y = 0$

2.2.2. Newton's second law: The change of momentum is equal to the sum of forces on a fluid particle. (Momentum Eqn.)

a. $u(\partial u/\partial x) + v(\partial u/\partial y) = (-1/\rho)\,(\partial p/\partial x) + \gamma\,(\partial^2 u/\partial x^2 + \partial^2 u/\partial y^2)$

b. $u(\partial v/\partial x) + v(\partial v/\partial y) = (-1/\rho)\,(\partial p/\partial y) + \gamma\,(\partial^2 v/\partial x^2 + \partial^2 v/\partial y^2)$

2.2.3. Energy Equation:

$u(\partial T/\partial x) + v(\partial T/\partial y) = \alpha\,(\partial^2 T/\partial x^2) + (\partial^2 T/\partial y^2)$

2.3. Laminar Fluid Flow

The flow from inlet is checked whether it is laminar or not. This is done by calculating Re number for inlet as follows:

Re = Velocity . Diameter / Dynamic Viscosity

Re inlet = (2m/s x 0,15m) / 0.076 = **3513**

For slots, it is obvious that the velocity will be much more lower regarding to the velocity of inlet. We can assume it as around one of ten of inlet's velocity which is 0.2 m/s.

Re slots = (0,2m/s x 0,15m) / 0.076 = **351**

It is known that the flow is laminar if Re number is less than 4000 [4]. Hence both flow at inlet and slots are laminar.

3. MESH

3.1. Structure of Mesh

Configuring mesh is basicly the second step of a fluent analysis which is just after geometry drawing and just before fluent solution.

In this Project, the diffuser pipeline in the tank firstly thought as a part of integrity of tank. Then it is understood that the diffuser line can be appraised and evaluated as single part whether it reaches our goal or not. Our goal, which also stated above, is monitoring how diffuser spreads the diesel oil through the tank and maximizing the spreaded area.

Firstly, the diffuser line is drawn in 2D with 20 slots as below:

inlet; velocity=2m/s, width=150mm

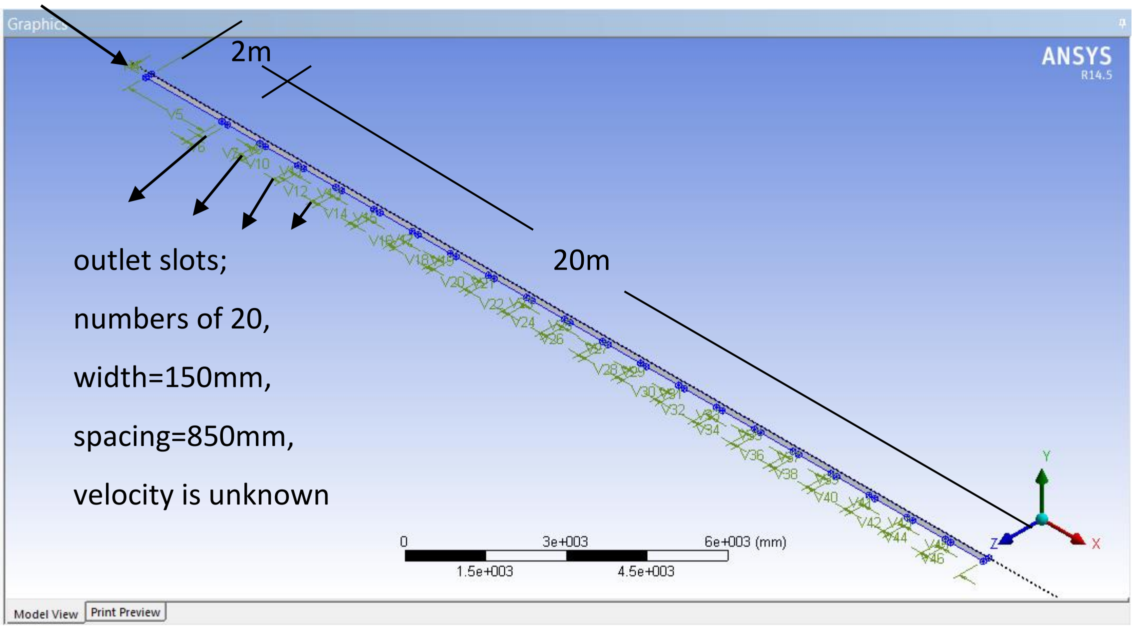

Figure 2: Geometry of Diffuser Pipeline

The upper edge of the diffuser is defined as symmetry wall on x- axis. Except inlet, outlet and symmetry, rest of the edges of diffuser are defined as wall without slip.

The temperature of fluids and solids all around the system is assumed in ambient condidtions as 20°C.

After forming the geometry, the mesh is generated with hexa elements and conditioning the max size as 5mm and relevance center as fine. The mesh is done with 132014 elements as the following:

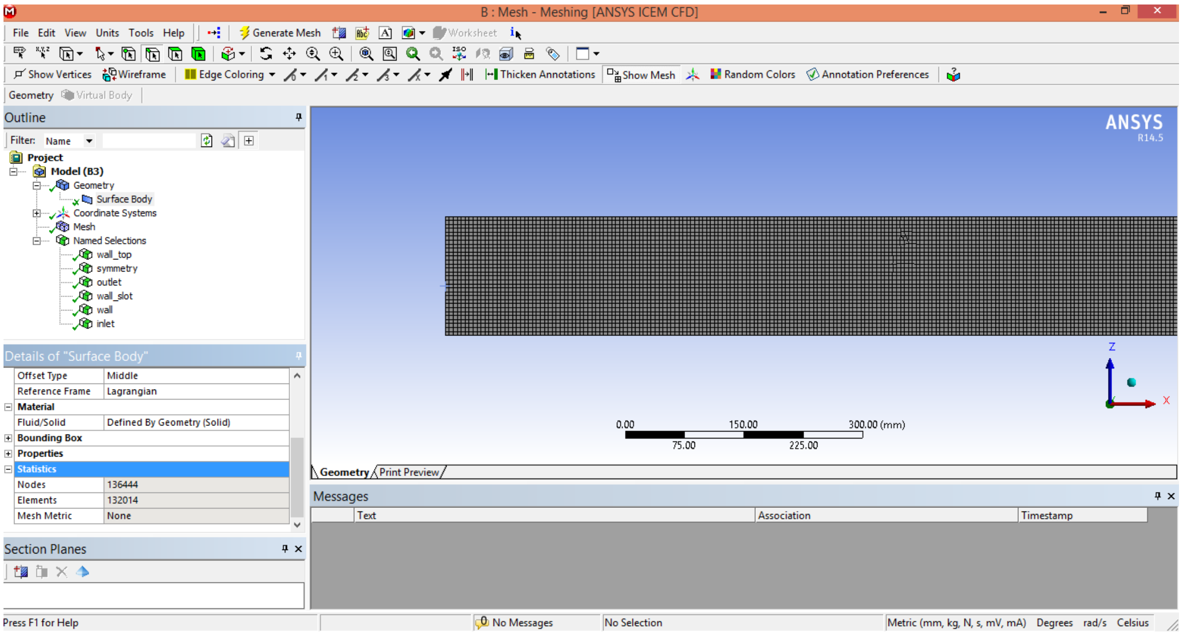

Figure 3: The Mesh of Diffuser Line

3.2. Quality of Mesh

In order to check the quality of mesh, some numbers are controlled:

- Skewness max: 0.55 (under 0.95 is acceptable)
- Orthogonal quality average: 0.99 (which is perfect)

In addition to these, the 6 named selections are done for fluent solution. These are:

- Wall_top: The opposite of the inlet edge of diffuser
- Symmetry: The upper edge of diffuser on x- axis in Figure 2
- Outlet: There are 20 slots for pressure outlet
- Wall_slot: These are the walls next to the slots
- Wall: The edge between inlet and first slot
- Inlet

4. RESULTS

For analyzing the system, the software of ANSYS FLUENT will be run implementing the following steps mentioned below. Then the diffuser is tried to be developed in order to spread the diesel oil all over the tank. Therefore, the geometry of diffuser pipe and the sizes of slots can be revamped.

Before starting the Fluent, the mesh is developed by adding "Inflation" for the edge with slots. This situation increase the amount of calculation around the slots where are quite sensitive for this Project.

4.1. In Fluent, as initial step the governing equations are selected which are energy equations and k-epsilon (2 eqn) for viscous equations in Figure 4 as below.

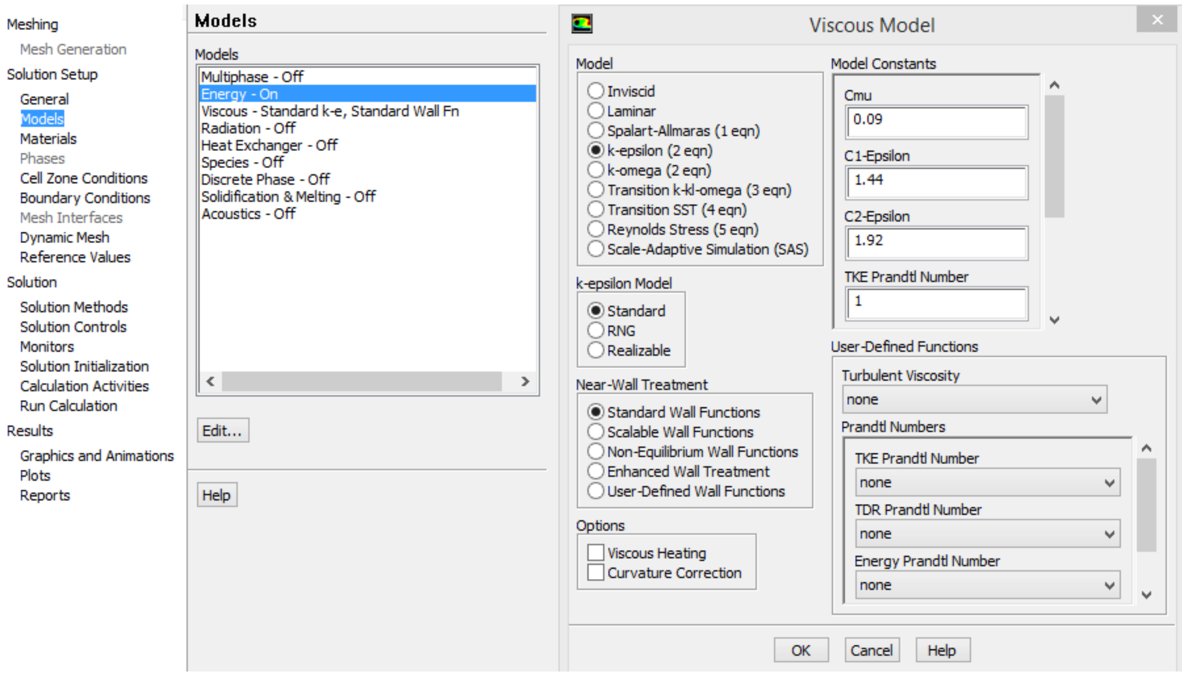

Figure 4: Models in Fluent

4.2. The liquid material is selected as diesel-vapor and aluminum for solid from fluent own database.

4.3. As Boundary Conditions:

Velocity-inlet: 1,5m/s ; 330°K ; 150mm of hydraulic diameter

Pressure outlet: 150.000 Pa for each 22 slots

Symmetry for one side of the diffuser and the rest of the edges are identified as Wall with no slip.

4.4. For "Solution Method", the following default equations are selected:

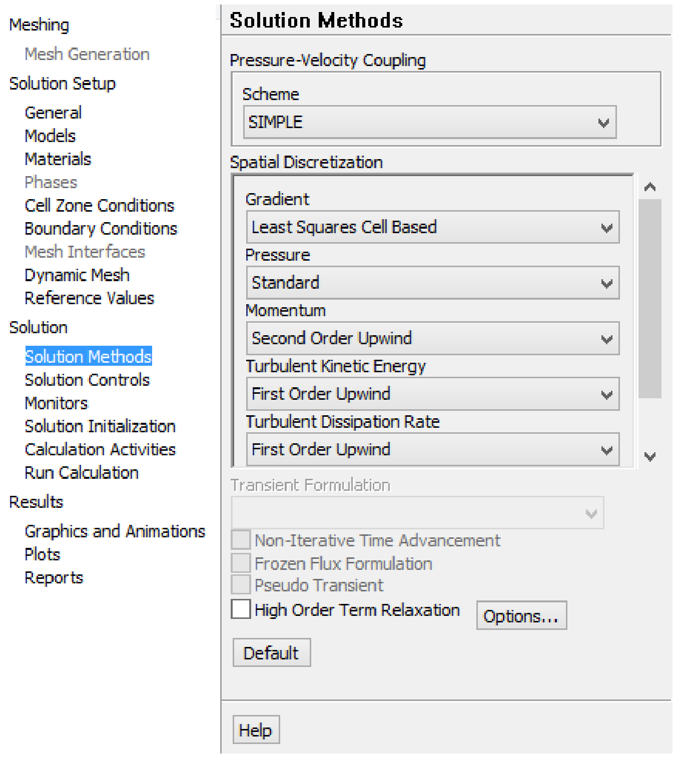

Figure 5: Solution Methods in Fluent

4.5. Convergence criteria is chosen as 10^-5 from Residuals in Monitors option. After around 500 iterations, the convergence is obtained.

After the run calculation steps above, the result is plotted as filled contour as below:

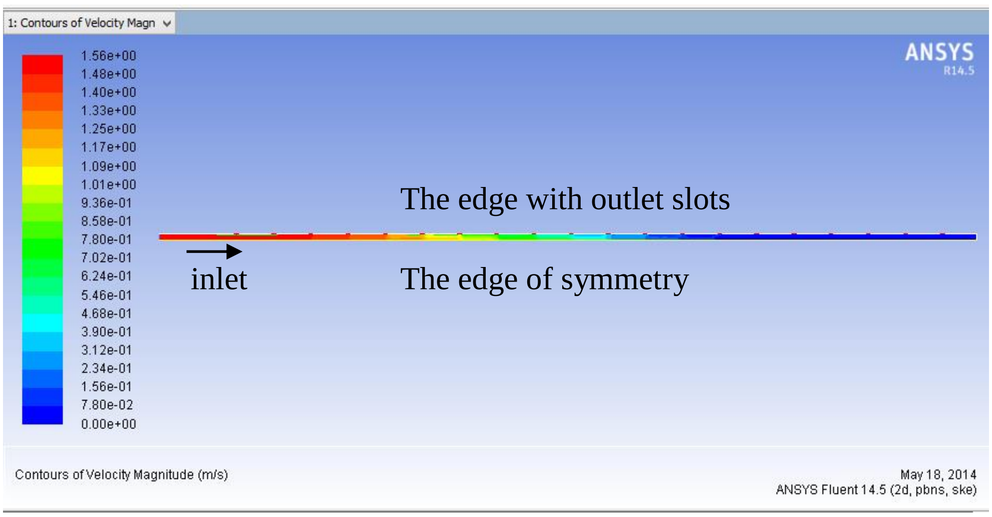

Figure 6: Velocity Magnitude of the diffuser with filled contour

According to the result in Figure6, it seems that the first 6 slots have almost same velocity magnitude with inlet which is 1,5m/s. In addition to this, there is remarkable motion and diesel exit via the successive 5 slots with around 0,5m/s. On the other hand, ort he last 9 slots, there seems almost no motion with 0m/s velocity magnitude which points that these slots can be estimated as unneccessary.

The recalculations done with 0.5m/s and 5m/s show us that there is not a remarkable change in velocity distribution and the results of those inlet velocities are quite similar with the result of study with the inlet velocity of 1.5m/s.

The velocity profile of inlet of the diffuser pipe is plotted using XYPlots module of the FLUENT. As assumed, the flow is uniform and the velocity is 1.5 m/s and same all over the inlet edge. The plot directions are selected as X=0 and Y=1 as below:

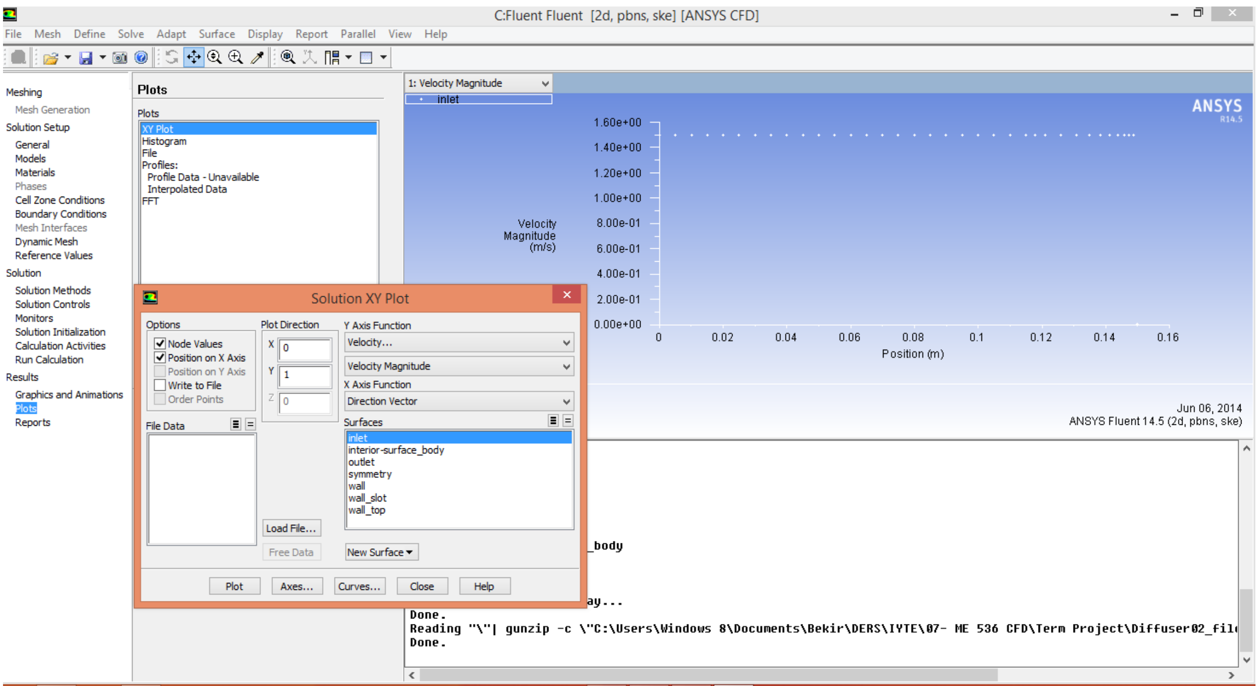

Figure 7: Velocity Profile at Inlet Edge

After plotting the velocity distrubiton for inlet, also outlet slots are investigated to figure velocity-outet values. As it is seen in next page, when the velocity is 1.5 m/s first two slots have around 0.5 m/s and next 7 slots have remarkable outlet zones for the diesel and higher than 1 m/s with a peak point of 1.2 m/s at the slot 5.

Between the slot 9th and 15th, there is still a velocity value and motion for the fluid but after that there is almost 0 m/s velocity for the last 5 slots. The figure of the outlet velocity profile is as below:

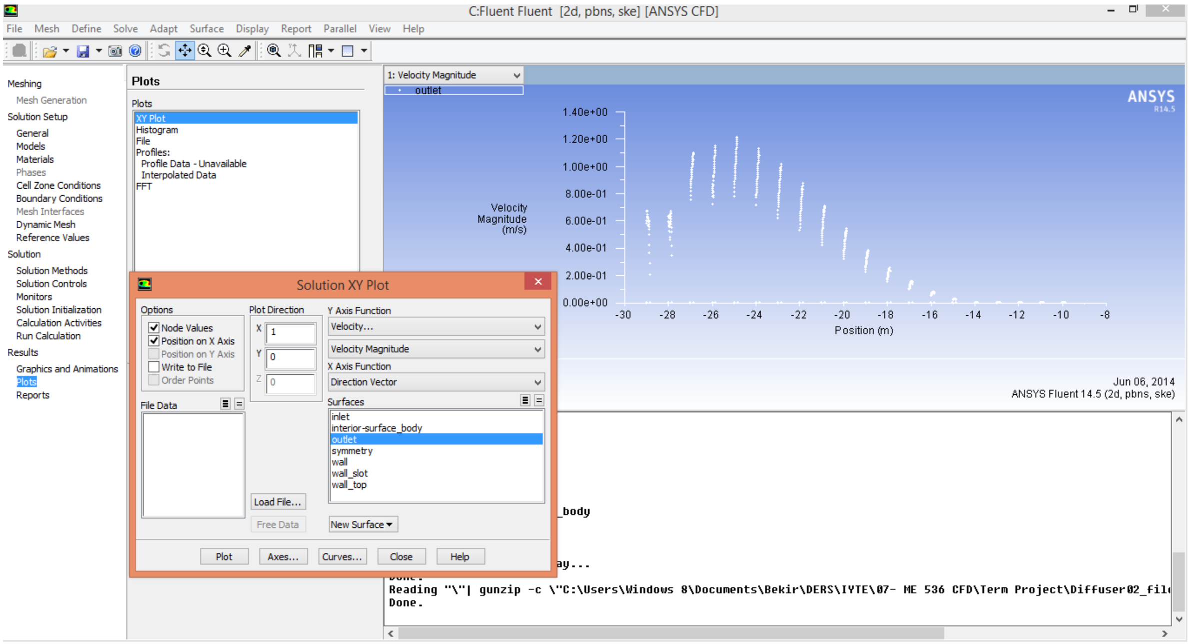

Figure 8: Velocity Profile at Oulet Slots

The temperature distribution is tried to analyzed using energy equations. According to the results, the inlet temperature provides the same temperature in the diffuser around 330°K except for the last 4m of the diffuser pipe (which is 300°K) as shown below. This result is proportional to the previous velocity results since this 4m long end is the part of the region with no motion (9m) which can reduce the heat transfer.

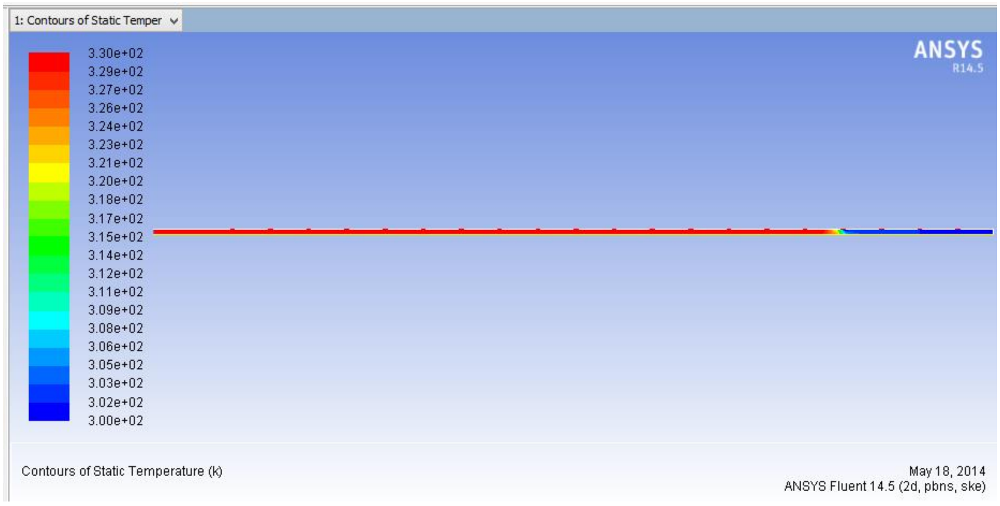

Figure 9: Contour of Static Temperature

5. DISCUSSIONS

This study is executed in order to see diffuser performance in a tank. After having results from FLUENT, it is observed that the velocity magnitude is not causing remarkable changes in flow type and distribution but the number of slots have an obvious effect on flow and velocity values. In such a case, the industrial diffuser pipes are seen that they are manufactured unnecessarily long and the last 33% of the body of these pipes are having 0 velocities which means no motion.

To have optimized diffuser pipes in such magnitudes and industries, the length of the pipes can be shortened. In addition to this, the slots in mid pipe and end of the pipe can be widened hence the fluid can be forwarded to the end of the pipe thanks to the relatively higher resistance at the entrance of the pipe.

6. REFERENCES

1. http://www.globalspec.com/reference/73229/203279/chapter-3-fluid-flow
2. http://www.pump-zone.com/topics/pumps/centrifugal-pumps/newtonian-and-non-newtonian-fluids-0
3. http://www.gates.com/brochure.cfm?brochure=2625&location_id=3046
4. http://ocw.mit.edu/courses/mechanical-engineering/2-000-how-and-why-machines-work-spring-2002/study-materials/TurbulentFlow.pdf