

T.C.  
MARMARA UNIVERSITY  
ENGINEERING FACULTY  
MECHANICAL ENGINEERING DEPARTMENT

**DESIGN AND ANALYSIS OF JET ENGINE  
COMBUSTION CHAMBER**

Submitted by

Mesut Çimen	150410054
Ömer Oğuz Tanrıverdi	150409024

Advisor

Asst. Prof. Dr. Mustafa Yılmaz  
Dr. Hasan Köten

Head of Department

Prof. Dr. Paşa Yayla

Istanbul, Turkey, 2015

## **Design and Analysis of Jet Engine Combustion Chamber**

### **ABSTRACT**

Doing this investigations, it can be realized initially, we obtained that solution which was done in geometry included in idealized combustion house defined in three dimensional cylindrical polar coordinate system. In the model combustion house we assumed that a turbulent jet mixing occur in a limited region swirling flow environment. Having high turbulence intensity, the isothermal flow was visually simulated injection hydrogen / air as a jet fluid that localized center in that flow process. Generalizing in three dimensions and taking into consideration effects of swirl searched in this flow condition.

Turbulence models have to be used for numerical calculations of turbulent flows. In our project, miscellaneous turbulence performance was also examined in detail. We had made numerous numerical experiments for finding effects of swirl. Making numerical computation ANSYS CFD programme was used with three-dimensional elliptic finite elements code in suitable coordinate system. Eventually results were compared to each other with respect to experiments and another susceptible predictions and they stated better correlations.

**Keywords:** Swirling flow, combustion chamber, finite element method, ANSYS CFD

## CONTENTS

Engineering Project Report.....	1
Design and Analysis of Jet Engine Combustion Chamber .....	1
Design and Analysis of Jet Engine Combustion Chamber .....	2
1.INTRODUCTION .....	4
2.PRINCIPLES OF JET PROPULSION.....	4
3.COMBUSTION CHAMBER .....	4
3.1.MAIN REQUIREMENTS FOR COMBUSTION CHAMBER .....	5
4.SWIRL FLOW .....	5
5.COMPUTATIONAL FLUID DYNAMICS .....	5
6.AIM OF THE PROJECT .....	5
7.MODELING USING SOLIDWORKS.....	6
7.1.NORMAL SOLID PART OF COMBUSTION HOUSE.....	6
7.2.INTERIOR VOLUME PART OF COMBUSTION HOUSE .....	7
8.MESH MODEL .....	7
8.1.MESH MODELS .....	7
9.BOUNDARY CONDITIONS .....	9
10.ANSYS ANALYSIS PROCESS .....	10
10.1. INITIAL CONDITIONS of CFD FLUENT .....	10
10.2.SOLUTION EQUATIONS .....	10
10.2.1. CONSERVATION EQUATIONS.....	10
10.2.2. MOMENTUM EQUATIONS.....	11
10.3.CASES OF CENTERLINE JET VELOCITY .....	12
10.3.1.CASE 1 (Air + Fuel) .....	13
10.3.2.CASE 2 (Air) .....	22
10.3.3.CASE 3 (Air) .....	31
11. RESULTS AND CONCLUSION .....	41
12.REFERENCES .....	412

## 1. INTRODUCTION

In recent days aircraft industry improved highly level. Because of this jet engines has to be improved in efficiently. Therefore fuel and flow analysis has to be greater importance of timely. The most important part of airplane and aircrafts is the jet engines. In our project we analyzed the swirl flow caused by the turbine blade entrance, laminar flow occurs by the injection of air and fuel and pressure and velocity analysis of different kind of cases. Among them, importance of designing of combustion chambers that located between the gas turbine and high-pressure compressor is considered. Flow fields analysis of accurate predictions in the combustion house is needed to a better design and high efficiency of our jet engines. Most of flows that we encountered in industry, the flow in combustion chamber mostly turbulent. Reason researching turbulence flow is having an important role in fluid mechanics of jet engines.

Combustion is also the most important topic that has to be analysis before a jet engine design period. Combustion practices has been practiced for many years to develop a good fluid modelling in jet aircrafts. There are several objectives are to be fully filled within a standard combustion analysis. Obviously the combustion house design is of prime importance in any combustion analyze of gas-turbine engines. Turbine producers have made excessive analytical design methods using computers. Most of the designers have to be considered in their designs in the mathematical presentation of the complicated flow patterns in a combustion house.

Finally we start a design of jet engine combustion house before design process swirling flow processes searched.

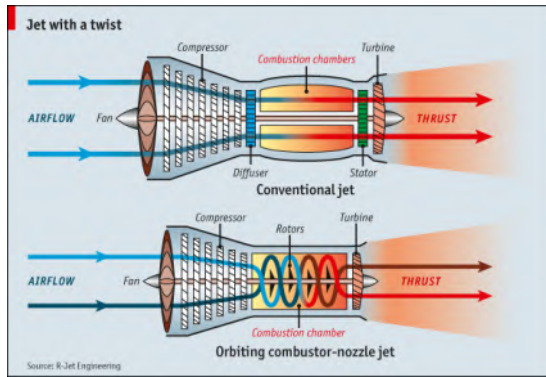
Experimental and theoretical study of axisymmetric, turbulent jet mixing confined swirling flow required. The principal objective of the present study is describing the flow of multiphase and non-turbulent jets in combustion chamber.[1]

## 2.PRINCIPLES OF JET PROPULSION

Jet propulsion is a practical application of Sir Isaac Newton's third law of motion which states that, 'for every force acting on a body there is an opposite and equal reaction'. [2]

## 3.COMBUSTION CHAMBER

- It is also called the combustor or combustion house.
- The most heavily thermal load part in a jet engine.
- Ignition and combustion occur inside it.
- It converts the chemical potential energy of the fuel to mechanical energy.
- It increases both the temperature and the specific volume of the air.



### 3.1.MAIN REQUIREMENTS FOR COMBUSTION CHAMBER

1. High Power Output
2. High Thermal Efficiency
3. Low Specific Fuel Consumption
4. Smooth Engine Operation
5. Reduced Exhaust Pollutant

### 4. SWIRL FLOW

Swirling flows have been commonly used for a number of years for the stabilization of high-intensity combustion processes. In general these swirling flows are poorly understood because of their complexity.

Two main types of swirl combustor can be identified:

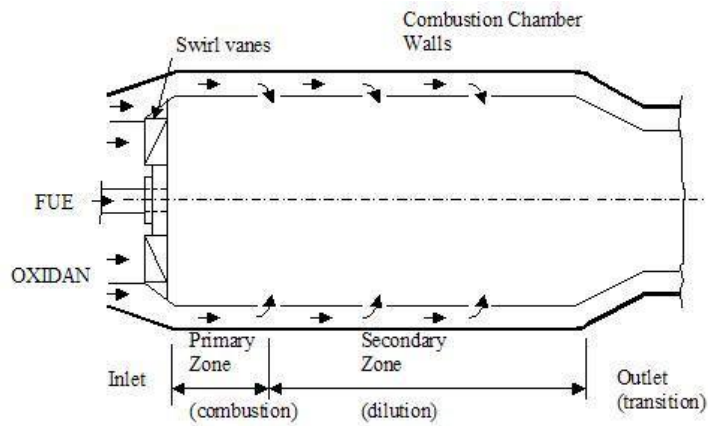
1. *The Swirl Burner*
2. *The Cyclone Combustion Chamber* [3][6]

### 5.COMPUTATIONAL FLUID DYNAMICS

The fundamental basis of almost all CFD problems are the Navier–Stokes equations, which define any single-phase (gas or liquid, but not both) fluid flow. These equations can be simplified by removing terms describing viscous actions to yield the Euler equations. Further simplification, by removing terms describing vorticity yields the full potential equations. Finally, for small perturbations in subsonic and supersonic flows (not transonic or hypersonic) these equations can be linearized to yield the linearized potential equations.[4]

### 6.AIM OF THE PROJECT

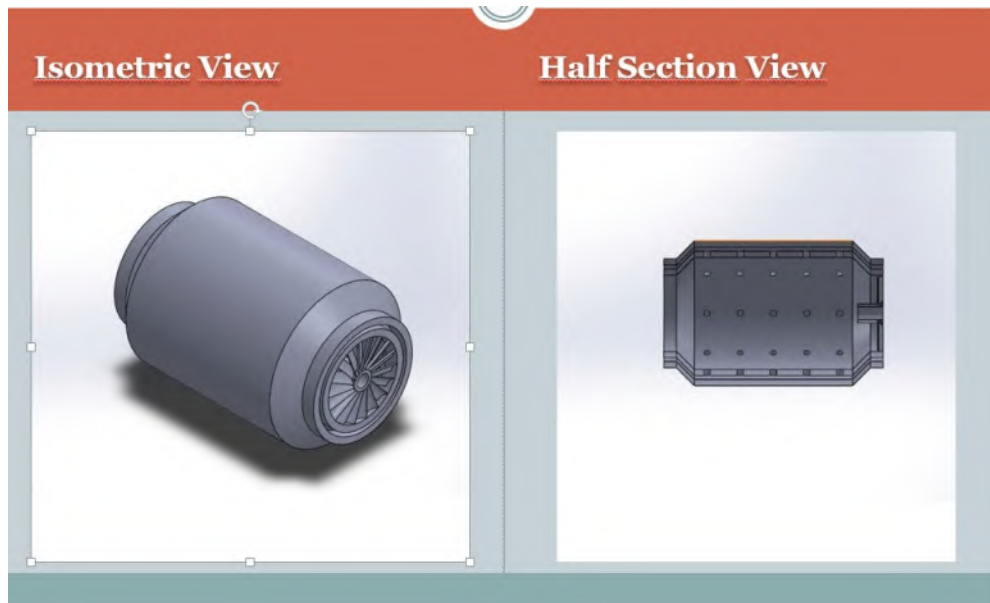
The main purpose of this project is designing and analyzing a jet engine with respect to velocity and pressure for different cases. The aim of this study is to find out flow structure inside combustion chamber without combustion process.



## 7.MODELING USING SOLIDWORKS

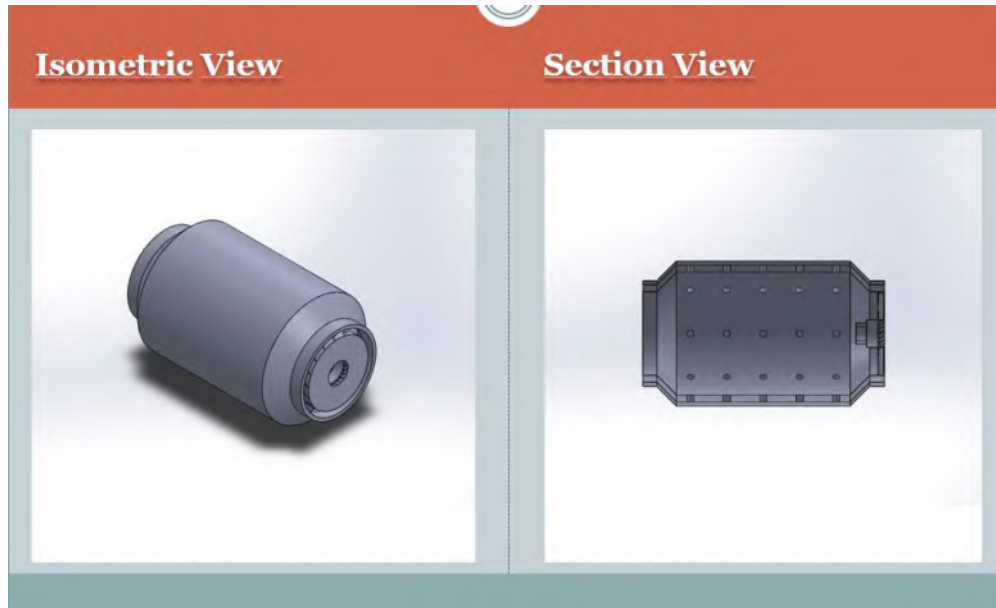
### 7.1.NORMAL SOLID PART OF COMBUSTION HOUSE

First of all we draw all parts of our combustion house in Solidworks 2013. It consists of two air inlet one fuel inlet and a exit occurs in our solid part. Cross section of air entrance is 60mm diameter. Fuel inlet is also having 10mm diameter. In addition to that 45° degree cone angle occurs in entrance to the combustion house. There is also a cooling air entrance occurs in 5 mm thickness. 18 unit turbine blades located in air entrance also caused swirl flow in entrance part.



## 7.2.INTERIOR VOLUME PART OF COMBUSTION HOUSE

Using combine command allows us to extract the interior volume of our solid combustion house parts that remains from the solid parts. This interior volume required to estimate velocity and pressure analysis of some different flow cases.



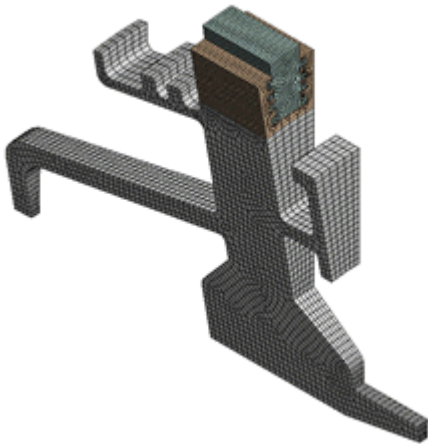
## 8.MESH MODEL

### 8.1.MESH MODELS

- Cut cell Cartesian meshing
  1. This mesh method generates a high percentage of hexahedral cells in a Cartesian layout in the far field, to deliver accurate fluid flow results.
  2. Local to the surface, mixed element types are used that allow the mesh to conform to sharp features.
  3. The surface cells can be inflated to generate hexahedral and prismatic layers to capture near-wall physics effects.
  4. Rapid mesh generation of hexahedral cells with minimal user setup make this mesh method ideal for complex geometry for computational fluid dynamics (CFD) simulation.
- Automated sweep meshing
  1. Sweepable bodies are automatically detected and meshed with hex mesh when possible..
  2. Edge increment assignment and side matching/mapping are done automatically.
  3. Sweep paths are found automatically for all regions/bodies in a multibody part.
  4. Defined inflation is swept through connected swept bodies.
  5. Sizing controls and mapped controls can be added, and source faces selected to modify and take control of the automated sweeping.

6. Adding/modifying geometry slices/decomposition to the model greatly aids in the automation to obtain a pure hex mesh.
- Thin solid sweep meshing
    1. This mesh method quickly generates a hex mesh for thin solid parts that have multiple faces as source and target.
    2. This method can be used in conjunction with other mesh methods.
    3. Sizing controls and mapped controls can be added, and source faces selected to modify and take control of the automated sweeping.
  - Multi Zone sweep meshing
    1. This advanced sweeping approach uses automated topology decomposition behind the scenes to attempt to automatically create a pure hex or mostly hex mesh on complicated geometries.
    2. Decomposed topology is meshed with a mapped mesh or a swept mesh if possible. The option to allow for free mesh in sub-topologies that can't be mapped or swept is available.
    3. This method supports multiple source/target selection.
    4. Defined inflation is swept through connected swept bodies.
    5. Sizing controls and mapped controls can be added, and source faces selected to modify and take control of the automated sweeping.
  - Hex-dominant meshing
    1. This mesh method uses an unstructured meshing approach to generate a quad-dominant surface mesh and then fill it with a hex-dominant mesh.
    2. This approach generally gives nice hex elements on the boundary of a chunky part with a hybrid hex, prism, pyramid, tet-mesh used internally.

Mesh generation is one of the most critical aspects of engineering simulation. Too many cells may result in long solver runs, and too few may lead to inaccurate results. ANSYS Meshing technology provides a means to balance these requirements and obtain the right mesh for each simulation in the most automated way possible. ANSYS Meshing technology has been built on the strengths of stand-alone, class-leading meshing tools. The strongest aspects of these separate tools have been brought together in a single environment to produce some of the most powerful meshing available.



Mesh parameter changes



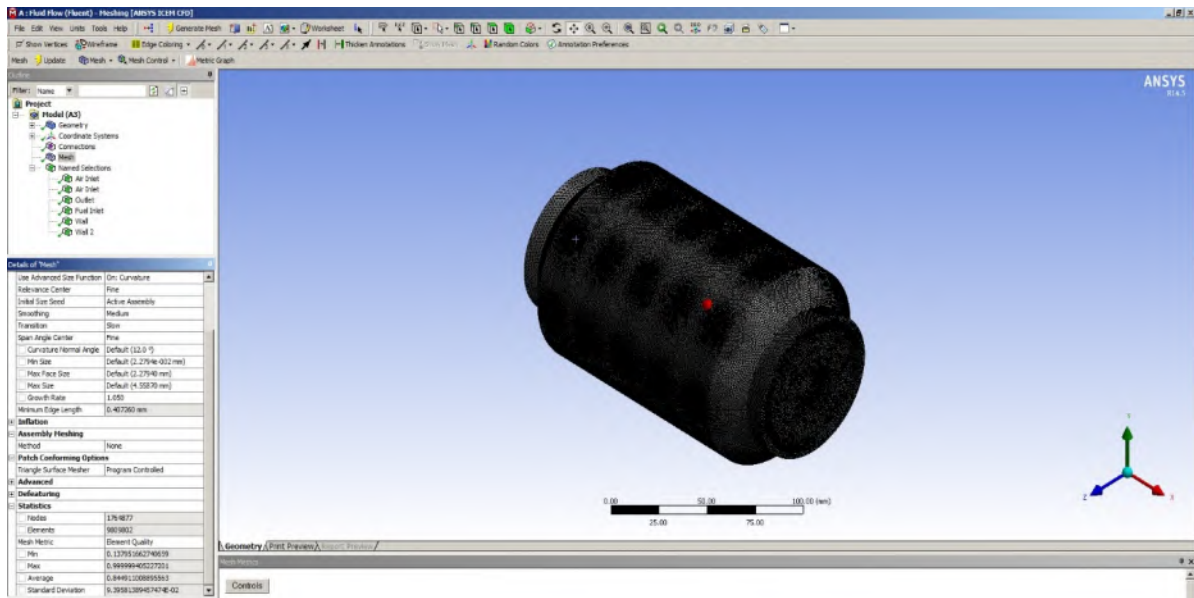
The highly automated meshing environment makes it simple to generate the following mesh types:

- Tetrahedral
- Hexahedral
- Prismatic inflation layer
- Hexahedral inflation layer
- Hexahedral core
- Body fitted Cartesian
- Cut cell Cartesian

Consistent user controls make switching methods very straight forward and multiple methods can be used within the same model. Mesh connectivity is maintained automatically.

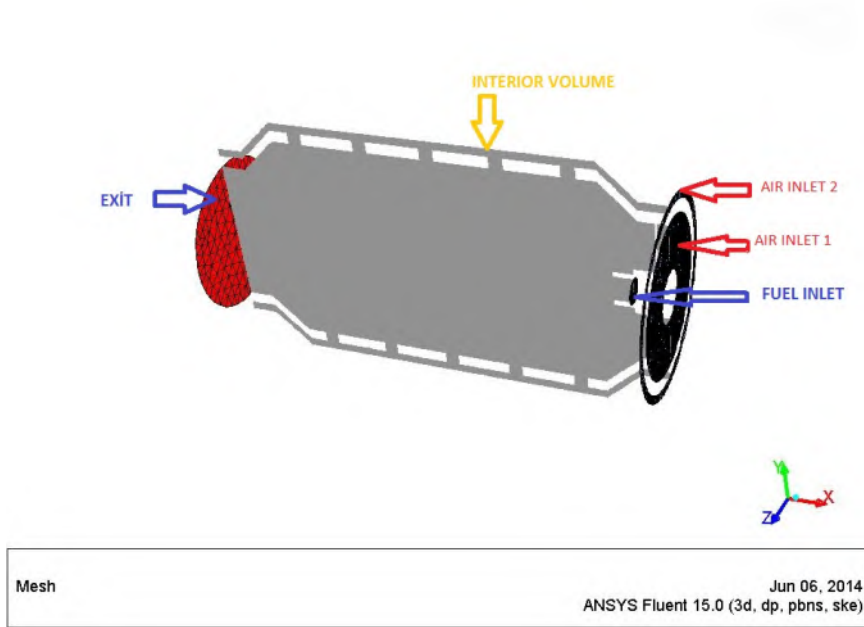
Different physics requires different meshing approaches. Fluid dynamics simulations require very high-quality meshes in both element shape and smoothness of sizes changes. Structural mechanics simulations need to use the mesh efficiently as run times can be impaired with high element counts. ANSYS Meshing has a physics preference setting ensuring the right mesh for each simulation.

The first step of analysis period is having a better mesh to obtain better results from analysis of the flow regions. It is limited into some standards in meshing process. Mesh processing was made tetrahedron mesh cells. Node number is **1764877**. Mesh Element Number is **9809802**. Skewness number is **0.232**. We changed relevance center is **fine**. Growth rate is decreased to **1.050** for getting better mesh to obtain better analysis process on our combustion house.



## 9. BOUNDARY CONDITIONS

Boundary conditions for the fluid interior volume is air inlet that has 30 mm radius. Cooling air inler also having a 5mm thickness. Fuel injection tube inlet has 15mm diameter outside 10 mm diameter inside. All analysis done in ANSYS CFD Fluent. The fluid is incompressible therefore flow is **incompressible flow** type. All the walls that influenced from flow are also treated **turbulent law-of the-wall**. Walls are also **adiabatic**. Prescribed boundary condition at the inlet of the intake port is pressure inflow, while that at the outlet of exhaust port pressure of outflow.



## 10.ANSYS ANALYSIS PROCESS

### 10.1. INITIAL CONDITIONS of CFD FLUENT

Some initial conditions are taken considered of previous analysis process. This conditions satisfies for solution; Double Precision. For processing we chose Serial Processing. Solver type had chosen Density-Based. Time is steady. Solid material of combustion house is aluminum. Ratio of Specific Heats is **1.4** for this situations. For obtaining better convergence in results demonstrations **500** iterations were made.

### 10.2.SOLUTION EQUATIONS

A set of related equations of motion for viscous, axisymmetric, recirculating flows was derived from the Navier Stokes equation, the species conservation equation, and the first law of thermodynamics. The coordinate system, which was used in the derivation of the differential equations based on the laws of conservation, is shown in figure below. The governing equations in solving level used are: [4] Turbulence flow has to be analyzed therefore **k-epsilon (2 eqn)** species transport mode also off for analyze start setup in ANSYS.

#### 10.2.1. CONSERVATION EQUATIONS

##### 1.a. Conservation of Mass

$$\frac{\partial}{\partial z}(r\rho Vz) + \frac{\partial}{\partial r}(r\rho Vr) = 0 \quad (10.2.1)$$

##### 1.b. Conservation of Chemical Species

$$\begin{aligned} \rho V_z \frac{\partial m_j}{\partial z} + \rho V_r \frac{\partial m_j}{\partial r} = \\ \frac{1}{r} \frac{\partial}{\partial r} (\rho D_T r \frac{\partial m_j}{\partial r}) + \frac{\partial}{\partial z} (\rho D_T \frac{\partial m_j}{\partial z}) + R_j \end{aligned} \quad (10.2.2)$$

## 10.2.2. MOMENTUM EQUATIONS

### 2.a. Conservation of Radial Momentum

$$\rho V_z \frac{\partial V_r}{\partial z} + \rho V_r \frac{\partial V_r}{\partial r} - \rho \frac{V^2 \theta}{r} = -\frac{\partial p}{\partial r} + \frac{1}{r} \frac{\partial}{\partial r} (r \tau_{rr}) + \frac{\partial}{\partial z} \tau_{rz} - \frac{\tau_{\theta\theta}}{r} \quad (10.2.3)$$

$$\tau_{rr} = \mu_{eff} \left[ 2 \frac{\partial V_r}{\partial r} - \frac{2}{3} \left( \frac{1}{r} \frac{\partial}{\partial r} (r V_r) + \frac{\partial V_z}{\partial z} \right) \right] \quad (10.2.3a)$$

$$\tau_{rz} = \mu_{eff} \left[ \frac{\partial V_z}{\partial r} + \frac{\partial V_r}{\partial z} \right] \quad (10.2.3b)$$

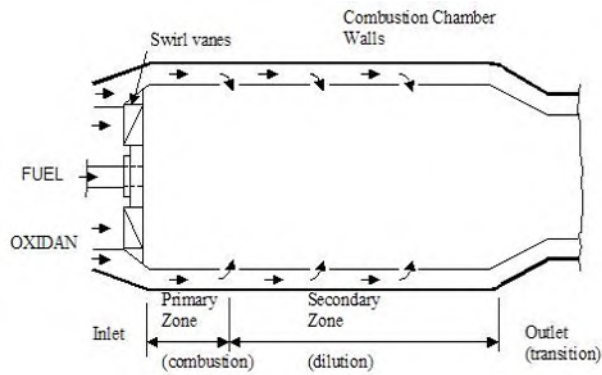
$$\tau_{\theta\theta} = \mu_{eff} \left[ \frac{2V_r}{r} - \frac{2}{3} \left( \frac{1}{r} \frac{\partial}{\partial r} (r V_r) + \frac{\partial V_z}{\partial z} \right) \right] \quad (10.2.3c)$$

### 2.b. Conservation of Axial Momentum

$$\rho V_z \frac{\partial V_z}{\partial z} + \rho V_r \frac{\partial V_z}{\partial r} = -\frac{\partial p}{\partial z} + \frac{1}{r} \frac{\partial}{\partial r} (r \tau_{zr}) + \frac{\partial}{\partial z} \tau_{zz} \quad (10.2.4)$$

$$\tau_{zr} = \tau_{rz} \quad (10.2.4a)$$

$$\tau_{zz} = \mu_{eff} \left[ 2 \frac{\partial V_z}{\partial z} - \frac{2}{3} \left( \frac{1}{r} \frac{\partial}{\partial r} (r V_r) + \frac{\partial V_z}{\partial z} \right) \right] \quad (10.2.4b)$$



**Figure 2-** Schematic Combustor Design

## 2.c. Conservation of Angular Momentum

$$\rho V_z \frac{\partial V_\theta}{\partial z} + \rho V_r \frac{\partial V_\theta}{\partial r} + \rho \frac{V_r V_\theta}{r} = \frac{1}{r^2} \frac{\partial}{\partial r} (r^2 \tau_{r\theta}) + \frac{\partial}{\partial z} \tau_{z\theta} \quad (10.2.5)$$

$$\tau_{r\theta} = \mu_{eff} \left[ r \frac{\partial}{\partial r} \frac{V_\theta}{r} \right] \quad (10.2.5a)$$

$$\tau_{z\theta} = \mu_{eff} \left[ r \frac{\partial}{\partial z} \frac{V_\theta}{r} \right]$$

The LES solver is a fully unstructured compressible code, including species transport and variable heat capacities [7]. It can work with both structured and unstructured grids, which makes it easily applicable to complex geometries [8]. Centered spatial schemes and explicit time advancement are used to control numerical dissipation and capture acoustics [9]. For the present case, a three-step Runge–Kutta method is employed, with a time step controlled by the speed of sound. Subgrid-scale viscosity is defined by the WALE model, derived from the classic Smagorinsky model [10]. Subgrid thermal and molecular fluxes are modeled using an eddy diffusion assumption with constant subgrid Prandtl and Schmidt numbers, respectively. As required by the WALE model, a no-slip condition is applied at the wall. Characteristic boundary conditions NSCBC [11,12] are used for all inlets and for the outlet. Many recent Large Eddy Simulations (LES) have demonstrated the potential of LES in reasonably simple academic combustion chambers. For such configurations, LES provide excellent evaluations of mean and root mean square (RMS) fields of temperature, species and velocities [13–22]. It also allows to investigate the unsteady structures in such flows [13–16] which can lead to instabilities known to be critical in many industrial programmes.

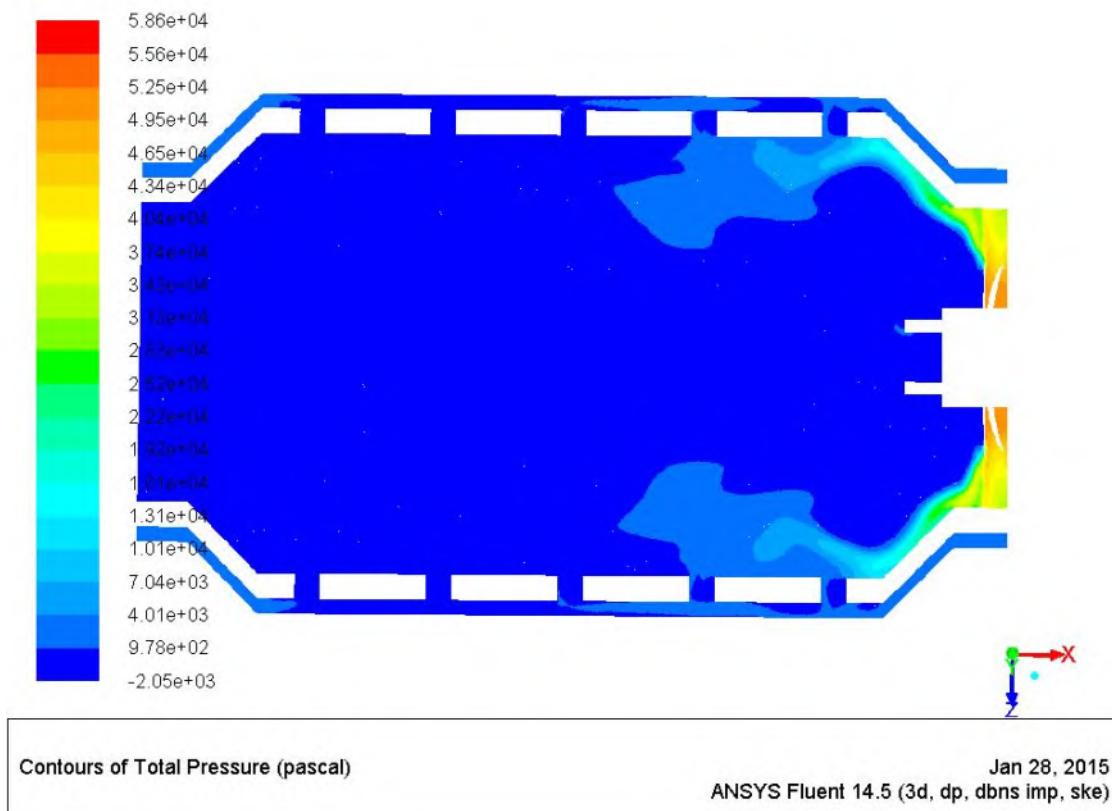
## 10.3.CASES OF CENTERLINE JET VELOCITY

The solution of the centerline velocity values on each region can be found by the using of conservation of momentum principle in ANSYS CFD solution. Energy conservations for viscous fluid flow and energy in fluid region, otherwise energy equation solved in the non-fluid region. Pressure values are also calculated from the conservations of mass principle. If the temperature aim to be estimated in combustion house it has to be used the law of conservation of energy. Finite element discretization of governing equations for every degree of freedom is solved separately were derived from matrix system by the way of using segregated solver algorithms.

### 10.3.1.CASE 1 (Air + Fuel)

- Case 1 includes air and fuel entering into the combustor blades which are combined into air for oxidizer and hydrogen for fuel.
- Air enter at two different inlet into the combustor that has a velocity of **50 m/s**. First inlet for general air entrance which has turbine blades, other one is for cooling into the combustion house.
- Fuel is hydrogen enters one inlet into the combustor which has velocity of **60 m/s**.

#### 10.3.1.1.TOTAL PRESSURE



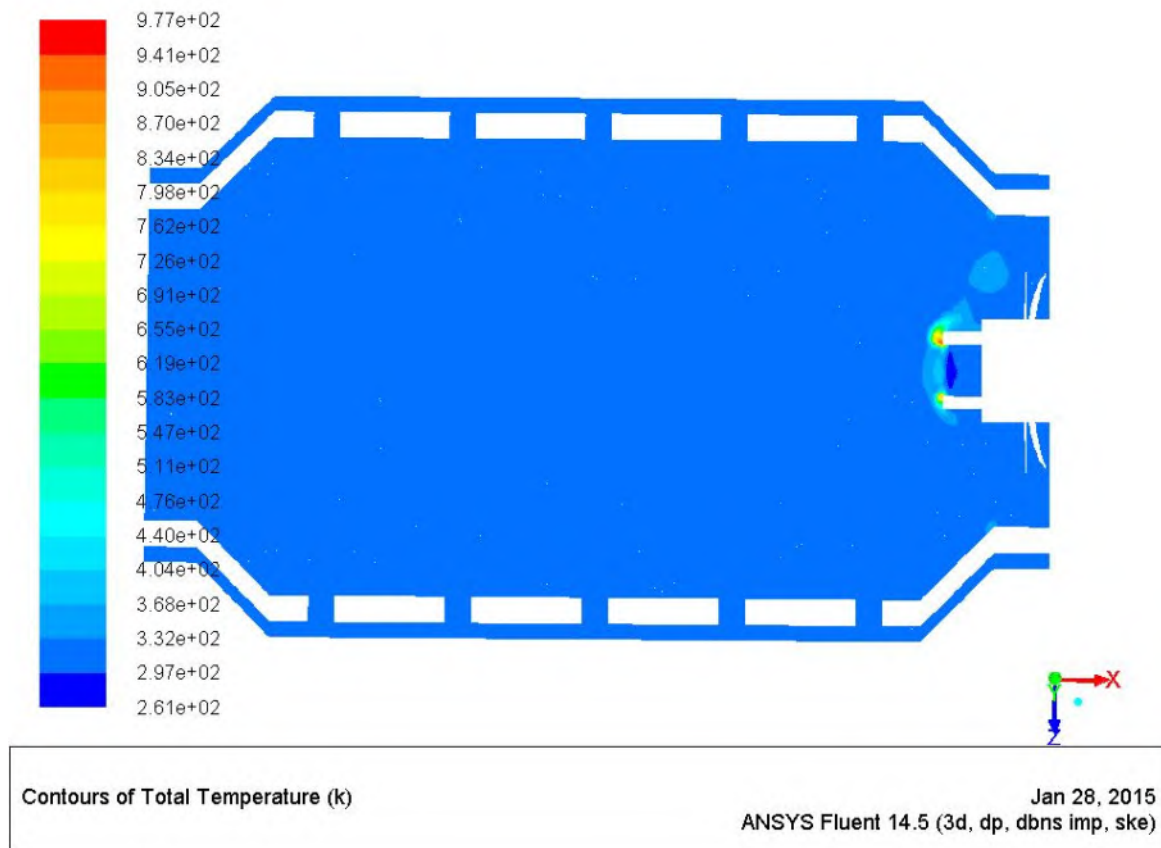
Pressure contours are related with the force of air that enters the fan blades (inlet) part of combustion house.  
Bottom side of blade edges were affected greater pressure than upper edges.

*Maximum pressure value in inlet at about = 586000 Pa*

*Minimum pressure value is negatives because of turbulence = -2050 Pa*

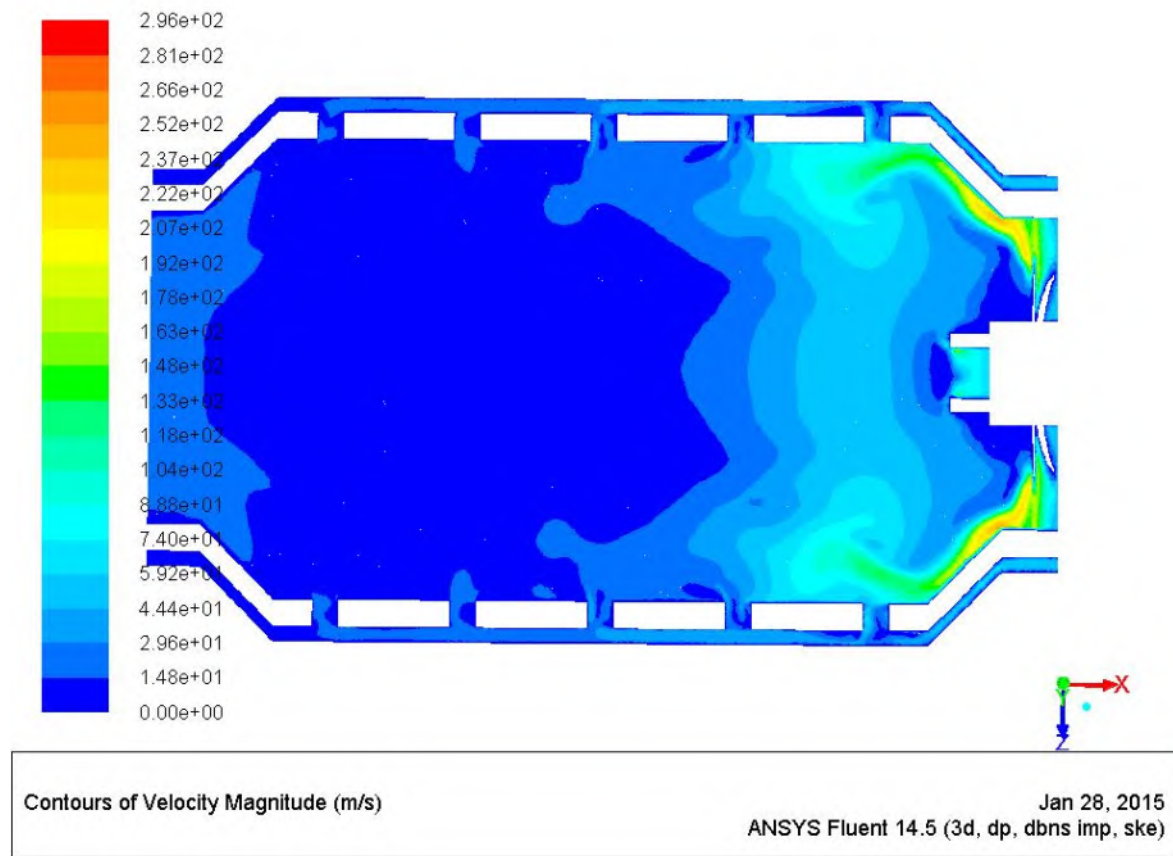
### 10.3.1.2.TOTAL TEMPERATURE

Temperature values are not so much change in turbulence flow it only changes in fuel entrance 977 K. This caused by turbulence flow and mixing of air and hydrogen (fuel). Temperature of all combustion house couldnt so much changes only 15 °C because of friction.

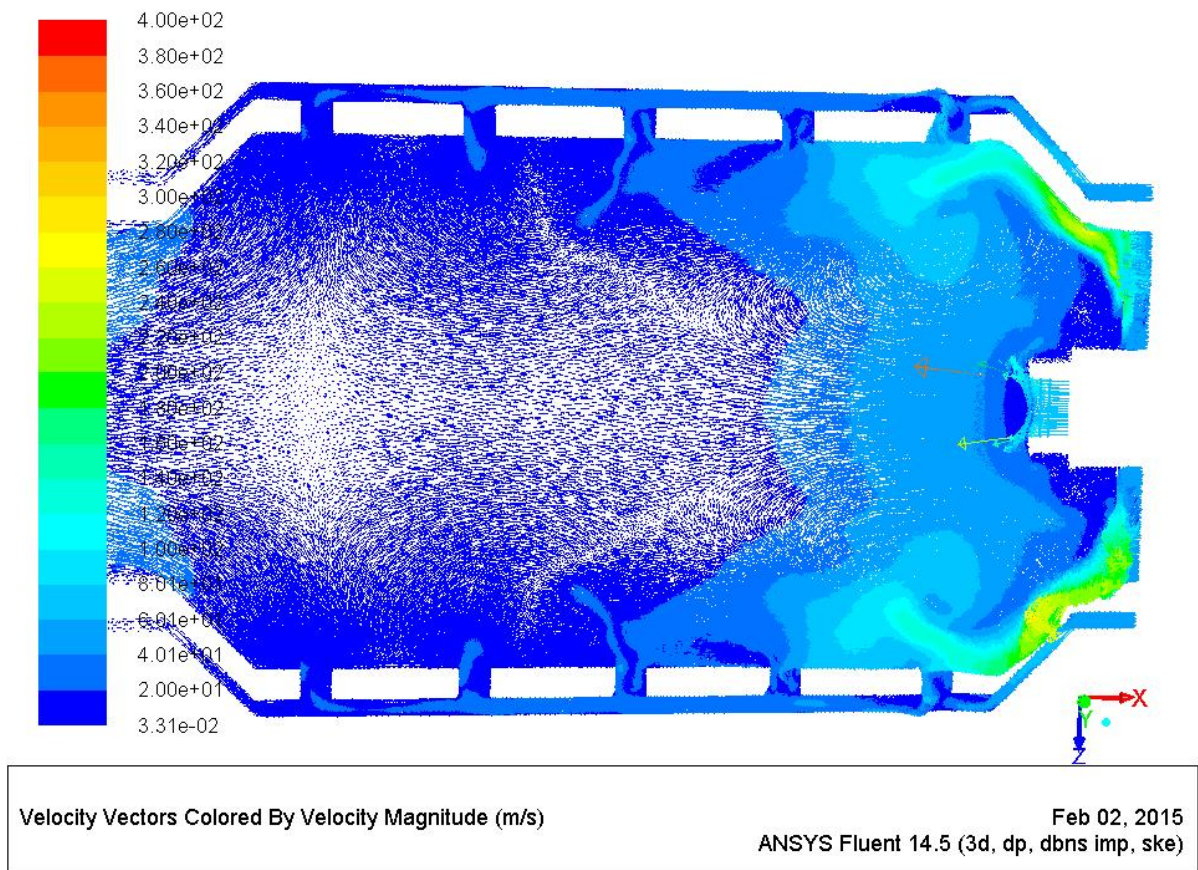


Temperature analysis shows us there is no much change in temperature value in general process. Because of non-combustion process considered in flow analysis in our jet engine combustor. Therefore energy equation could not be solved by the ANSYS CFD Fluent because of only flow not combustion occurs in process. Temperature remains at 300 K. A small changes at the 4 % occurs in this analysis. It can be appear from the mesh structure that used in analysis process.

### 10.3.1.3.VELOCITY MAGNITUDE







Air entrance velocity is the greatest value all regions of combustor. Nearly 296 m/s.

Fuel entrance values cannot be changed because no blades occurs fuel inlet part. (60m/s)

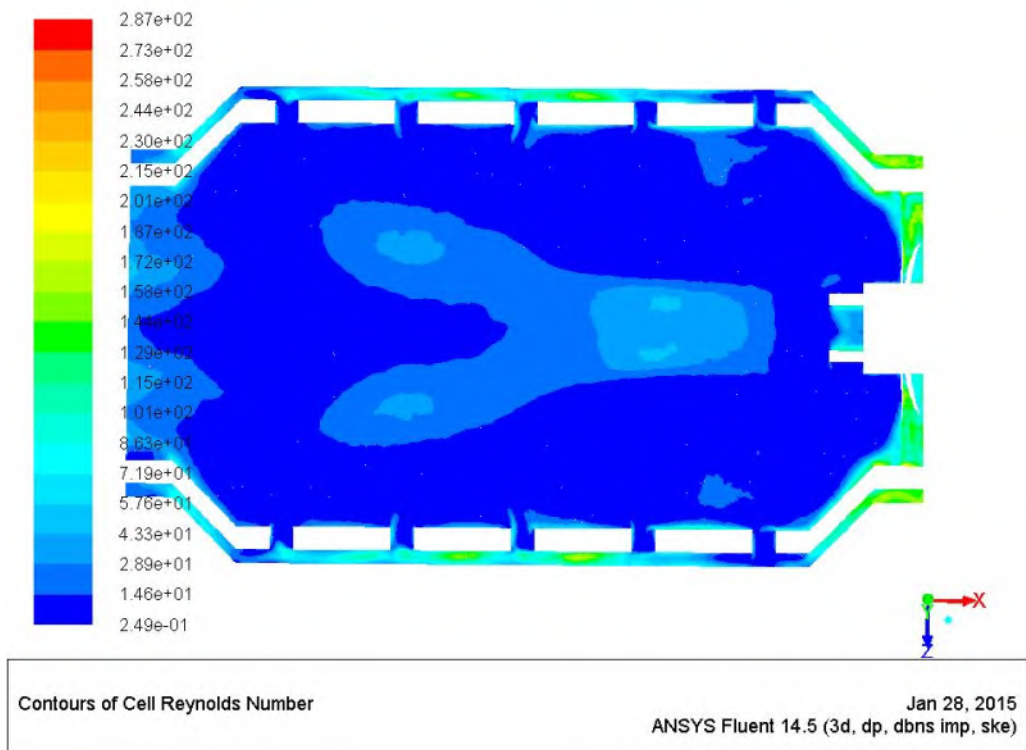
Exit (Exhaust) part velocity reduces at about 37 m/s. Reason of this combustion is not being in this process.

**Mach Number**= $v/V_{\text{sound}}$ = 0.86=> **Transonic**

In two situations the flow is **turbulent** with respect to evaluated Reynolds numbers.



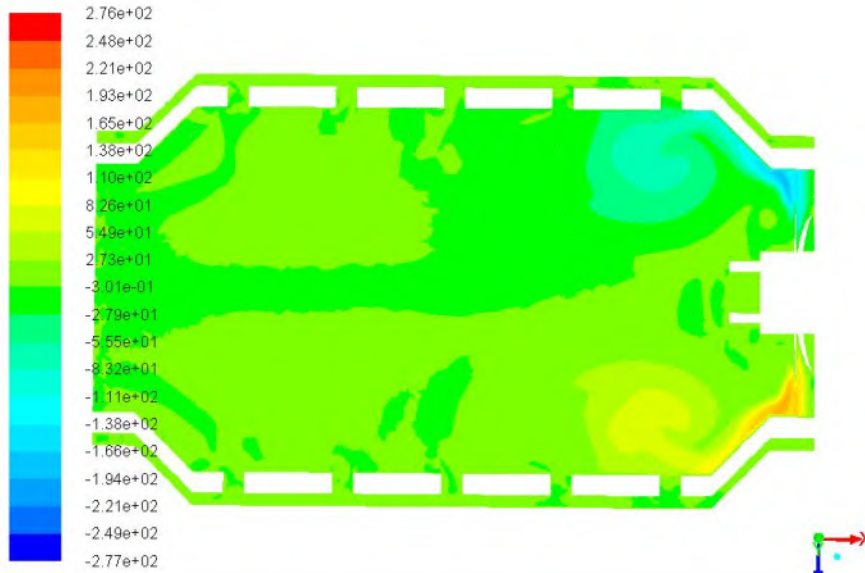
#### 10.3.1.4. Cell Reynolds Number



Maximum cell Reynolds number = 287

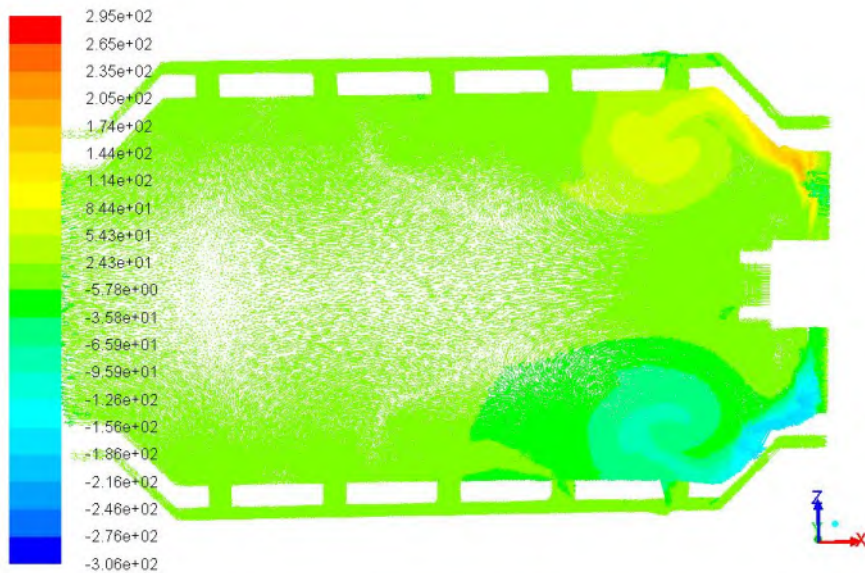
Minimum cell Reynolds number = -249

### 10.3.1.5.TANGENTIAL VELOCITY



Contours of Tangential Velocity (m/s)

Jan 28, 2015  
ANSYS Fluent 14.5 (3d, dp, dbns imp, ske)



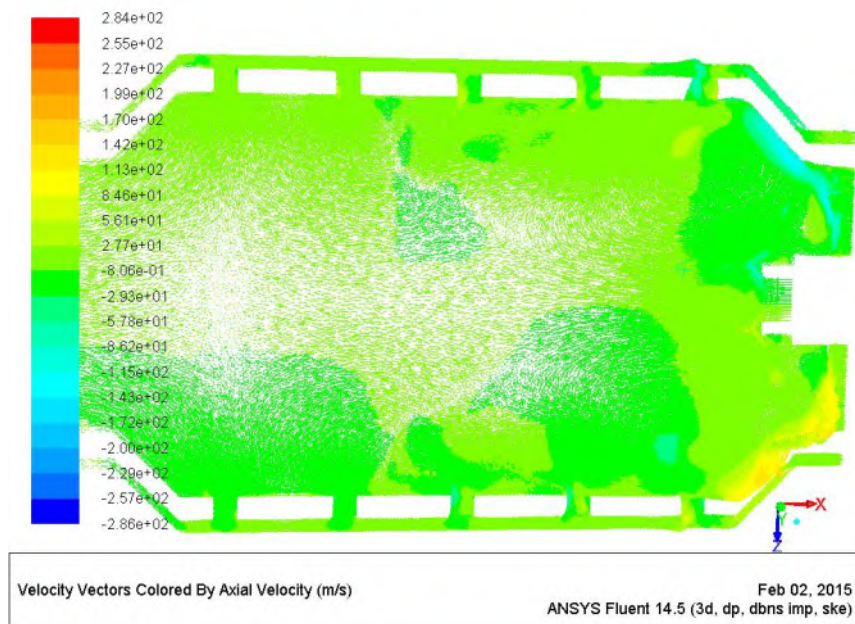
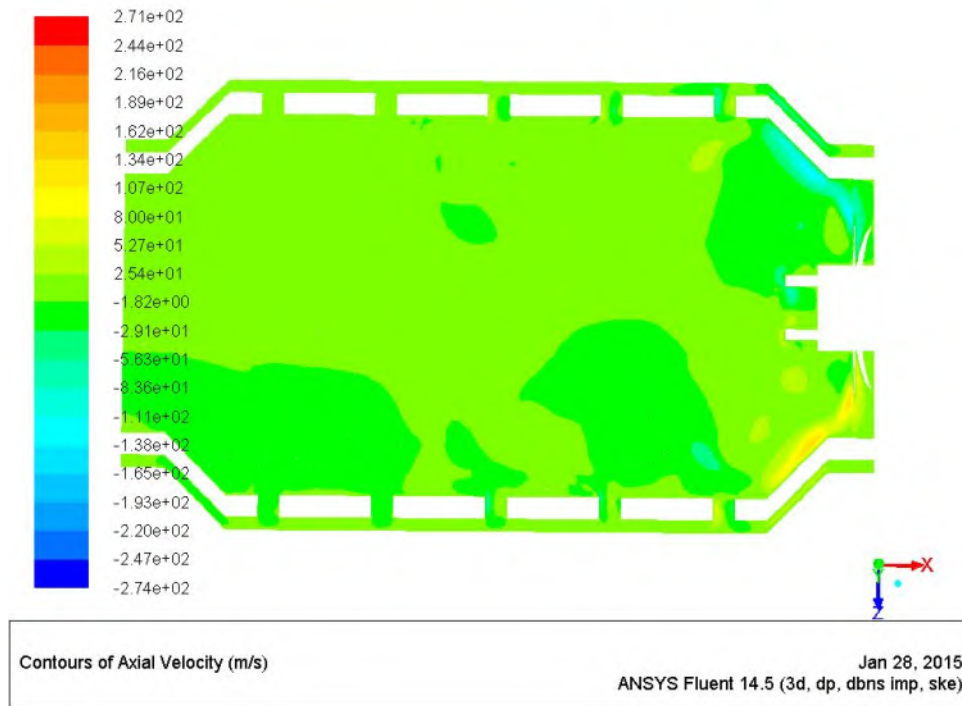
Velocity Vectors Colored By Tangential Velocity (m/s)

Feb 02, 2015  
ANSYS Fluent 14.5 (3d, dp, dbns imp, ske)

*Maximum tangential velocity is = 276 m/s*

*Minimum tangential velocity is = -277 m/s. (Opposite direction)*

### 10.3.1.6.AXIAL VELOCITY

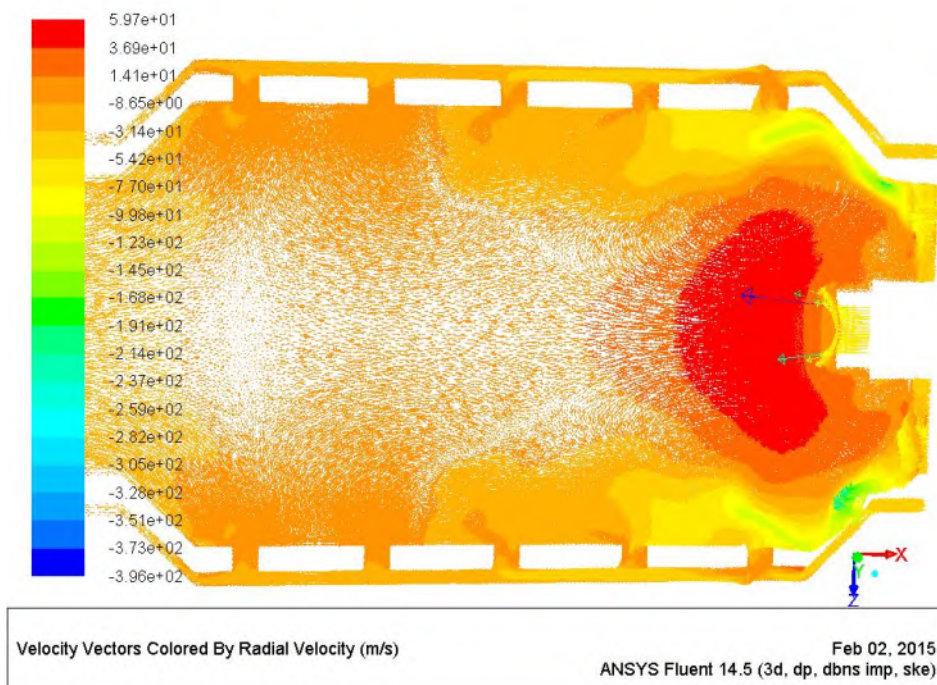
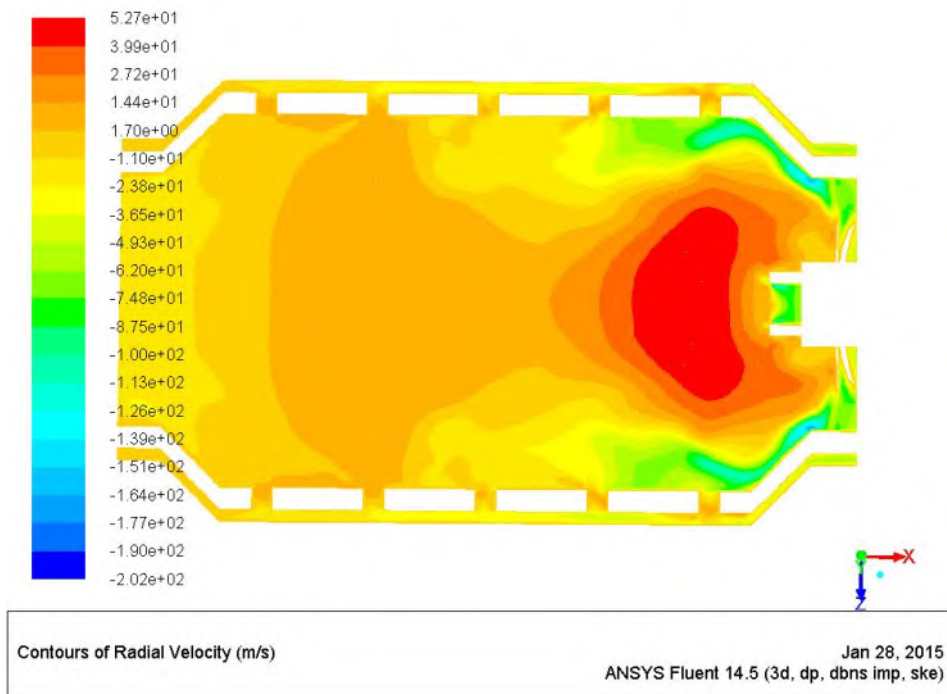


Positive y-axis direction of entrance region of fuel inlet centerline of air axial velocity values are lower otherwise negative y-axis direction it is also gets higher values.

*Maximum axial velocity is = 271 m/s*

*Minimum axial velocity is = -274 m/s*

### 10.3.1.7.RADIAL VELOCITY

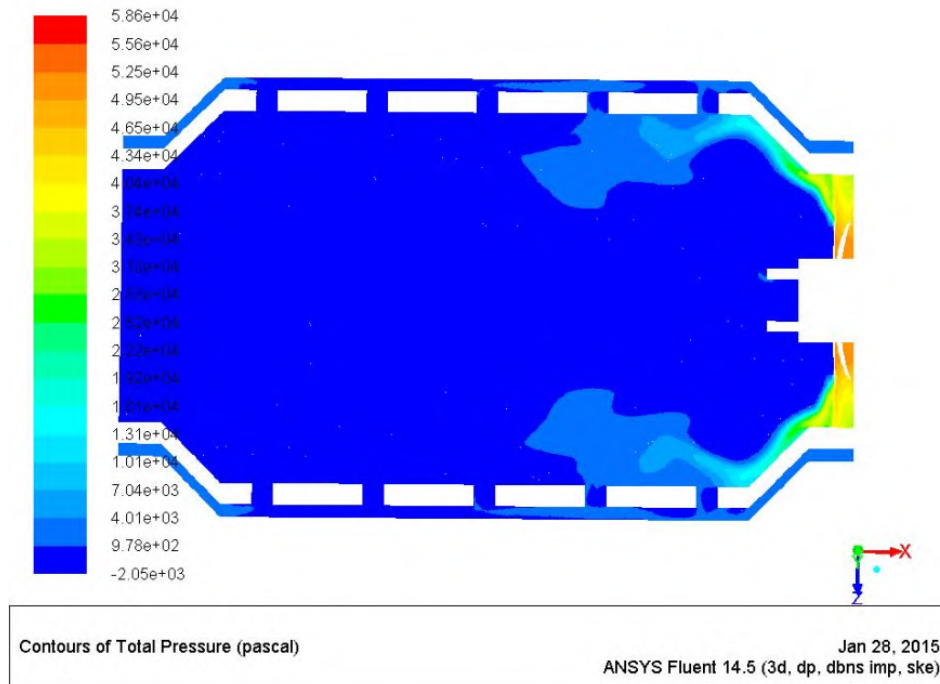
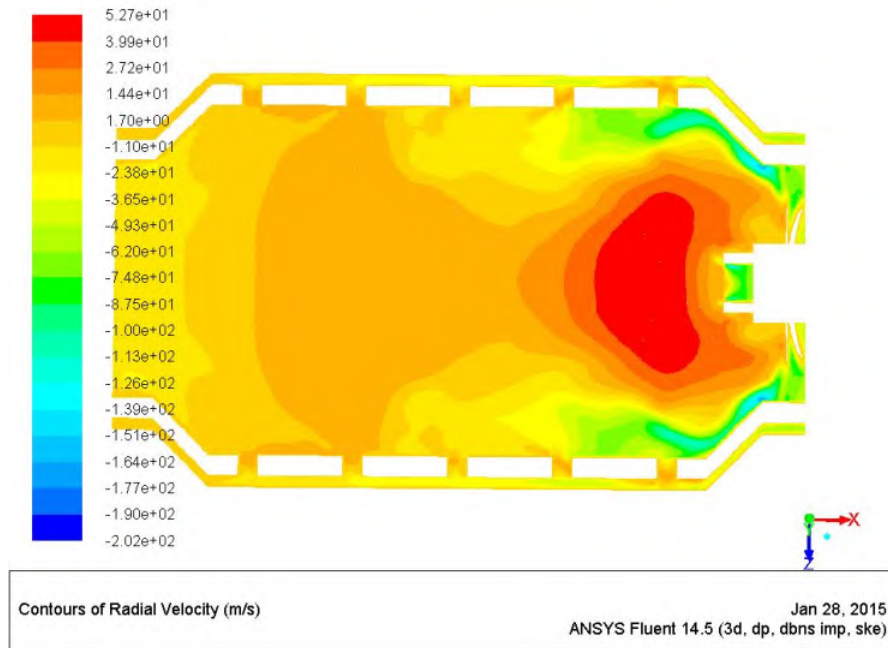


Maximum Radial Velocity = **52.7 m/s**

Minimum Radial Velocity = **-20 m/s**



### 10.3.1.8.RADIAL VELOCITY & TOTAL PRESSURE



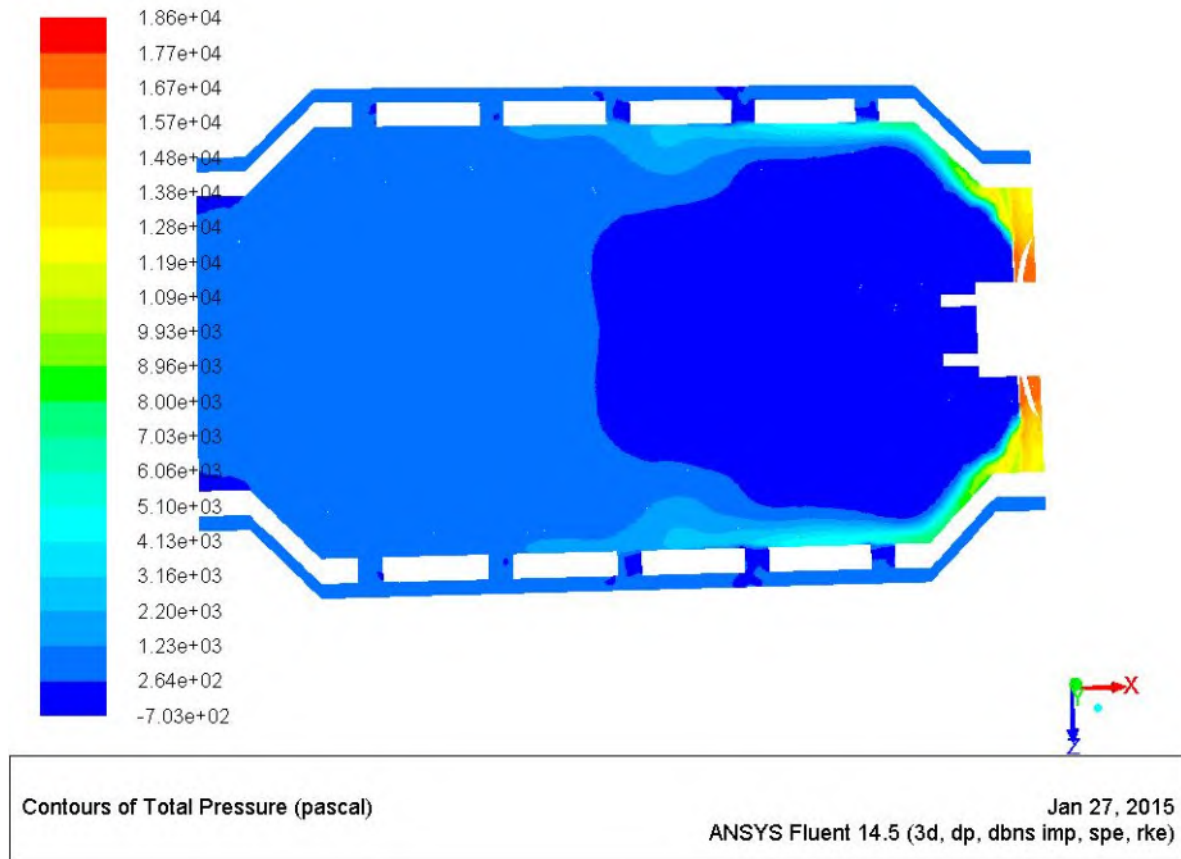
Radial velocity directly influences from total pressure.

It is obvious that two of them symmetrical to the centerline of fuel injection tube.

### 10.3.2.CASE 2 (Air)

- In case 2 only air entrance occurs inside to the combustor.
- Air has 25.4 m/s velocity in entrance regions that is two inlet occurs in.

#### 10.3.2.1.TOTAL PRESSURE



Boundary conditions are same for case 1.

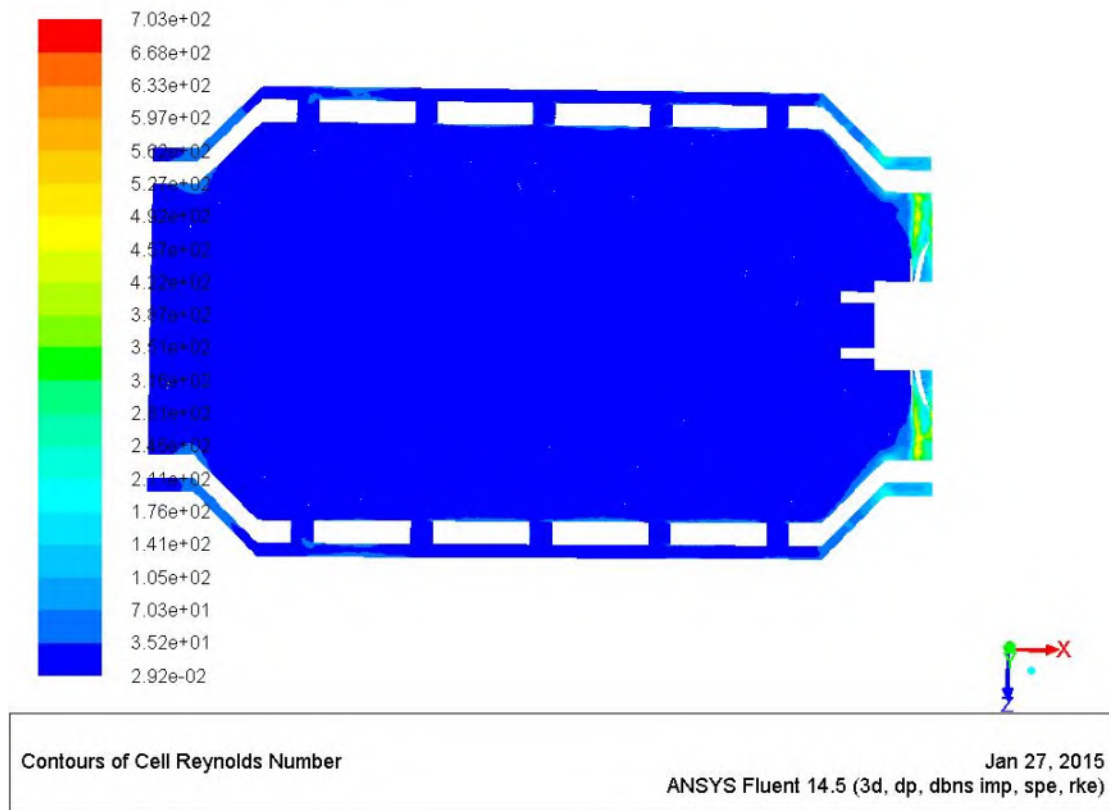
No-slip Wall, Adiabatic Wall etc...

**Maximum pressure value occurs at intake air part is: 18600 Pa**

**Minimum pressure is: -703 Pa.**

Also higher at blade bottom edges.

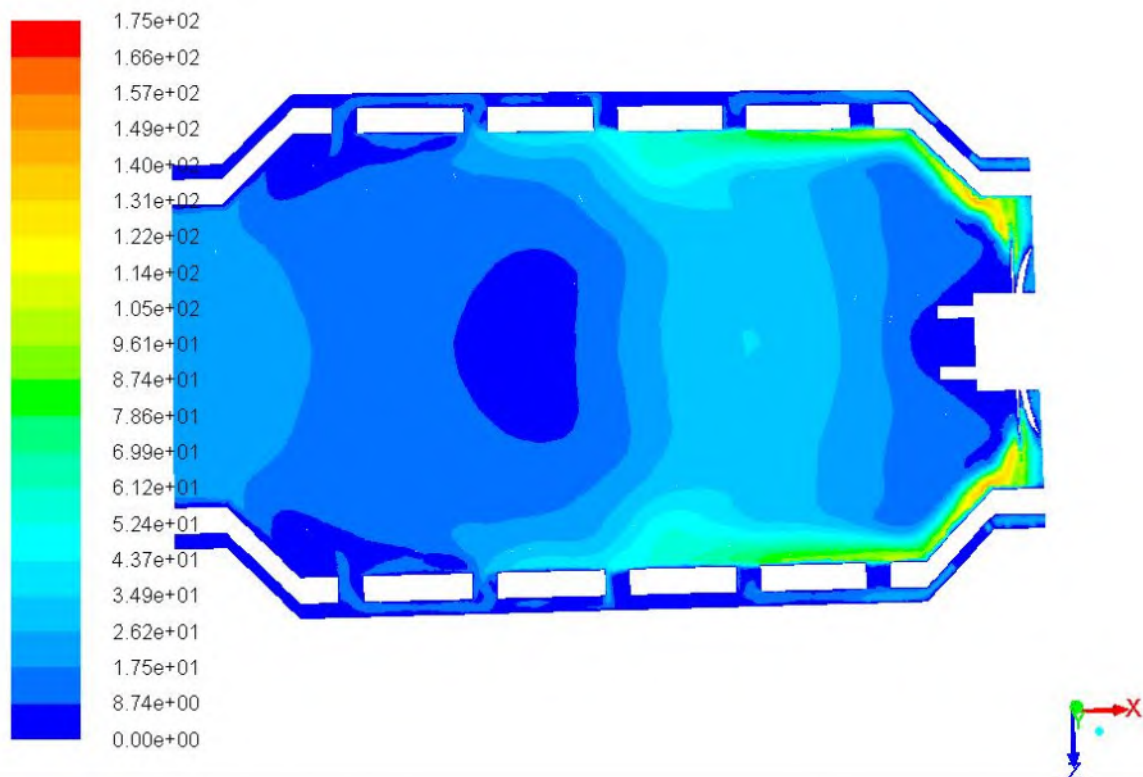
### 10.3.2.2. CELL REYNOLDS NUMBER



Maximum cell Reynolds number = 703

Minimum cell Reynolds Number = -292

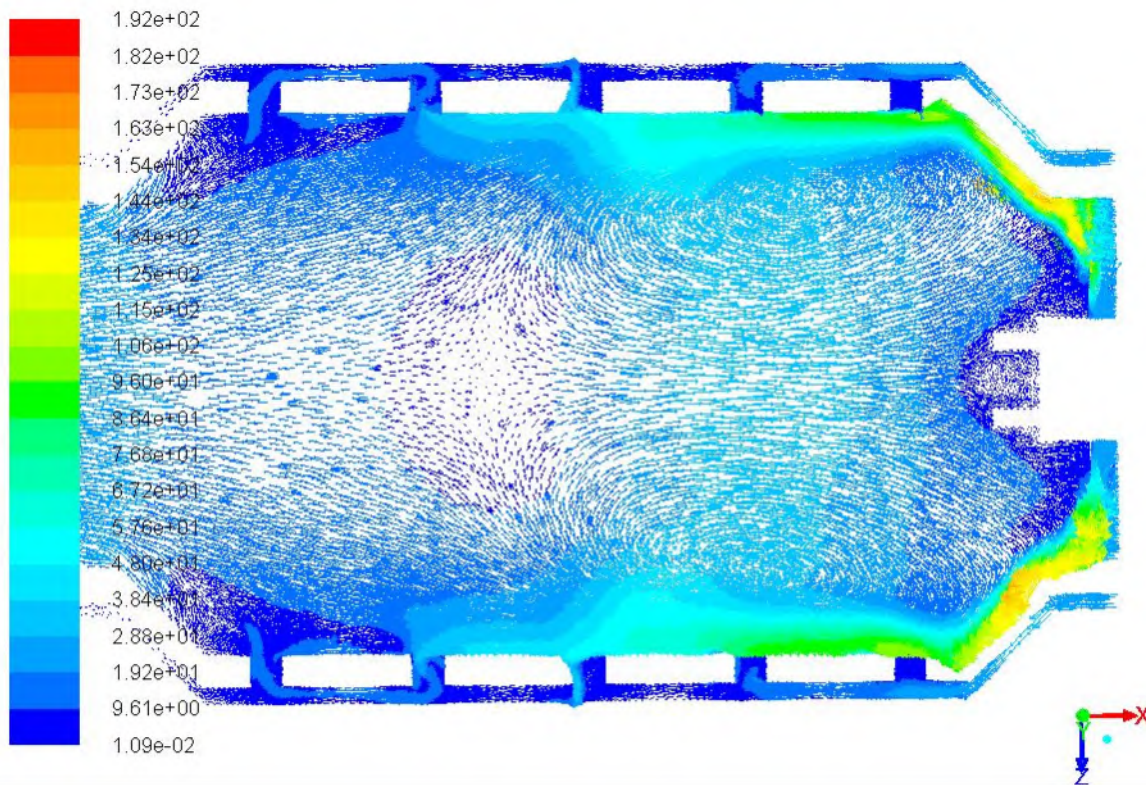
### 10.3.2.3. VELOCITY MAGNITUDE



Contours of Velocity Magnitude (m/s)

Jan 27, 2015  
ANSYS Fluent 14.5 (3d, dp, dbns imp, spe, rke)





Velocity Vectors Colored By Velocity Magnitude (m/s)

Feb 02, 2015  
ANSYS Fluent 14.5 (3d, dp, dbns imp, spe, rke)

Velocity graphics are very similar to the case 1. Velocity decreases effects of friction effects only.

The maximum velocity is find on the free regions located between the blades at entrance.

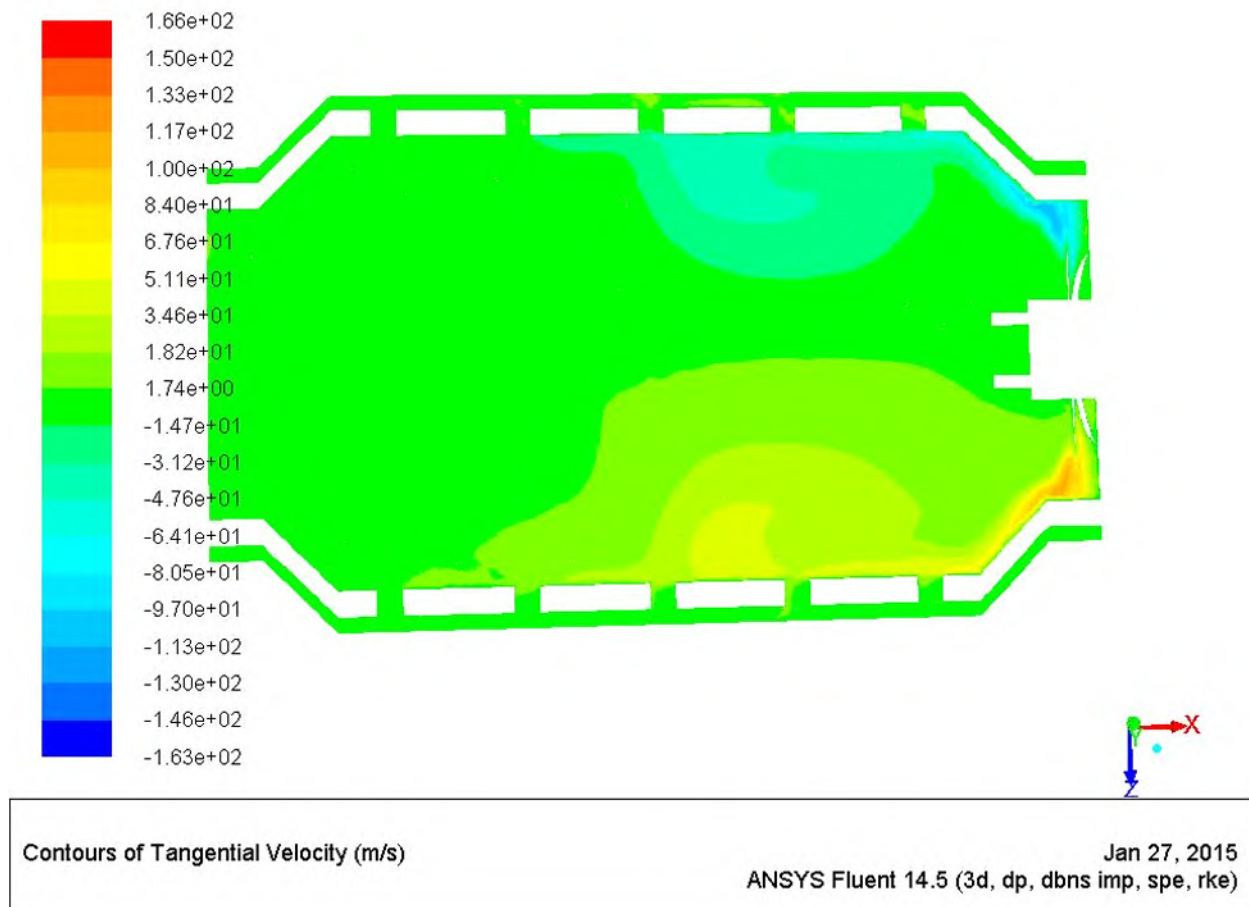
*Maximum velocity* = **175 m/s**

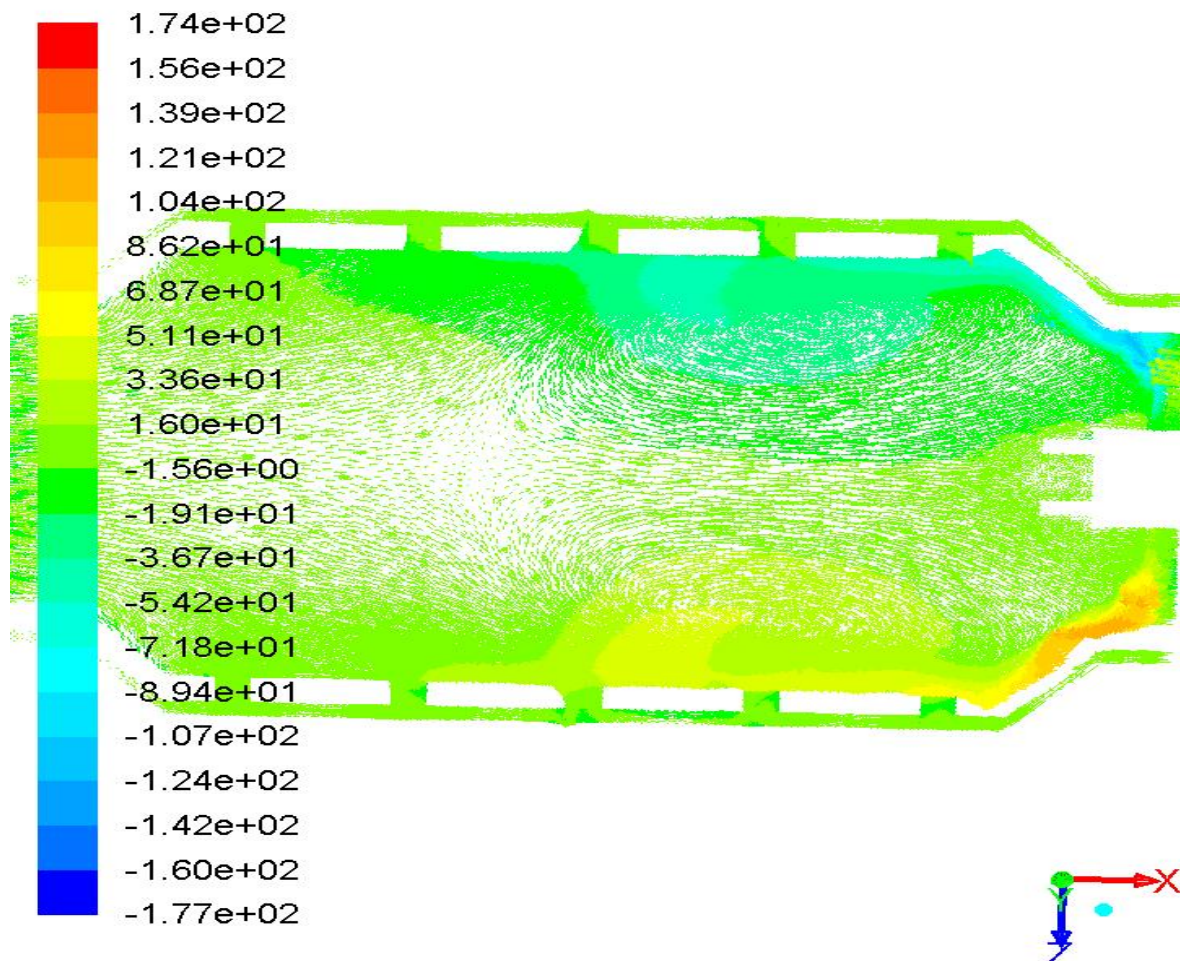
*Minimum velocity* = **0 m/s**

Mach Number= $v/V_{\text{sound}}$ = **0.51** =>**Subsonic**

The flow is **turbulent** with respect to evaluated Reynolds number.

#### 10.3.2.4. TANGENTIAL VELOCITY





Velocity Vectors Colored By Tangential Velocity (m/s)  
ANSYS Fluent 14.5 (3d, dp, dbns imp, spe, rke)

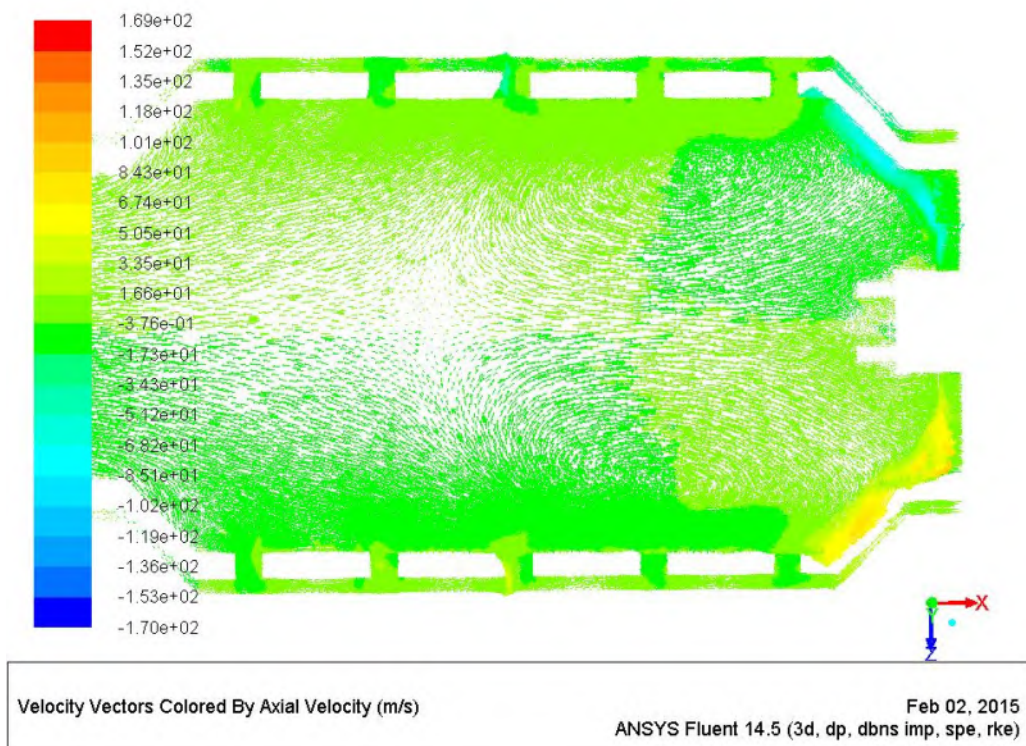
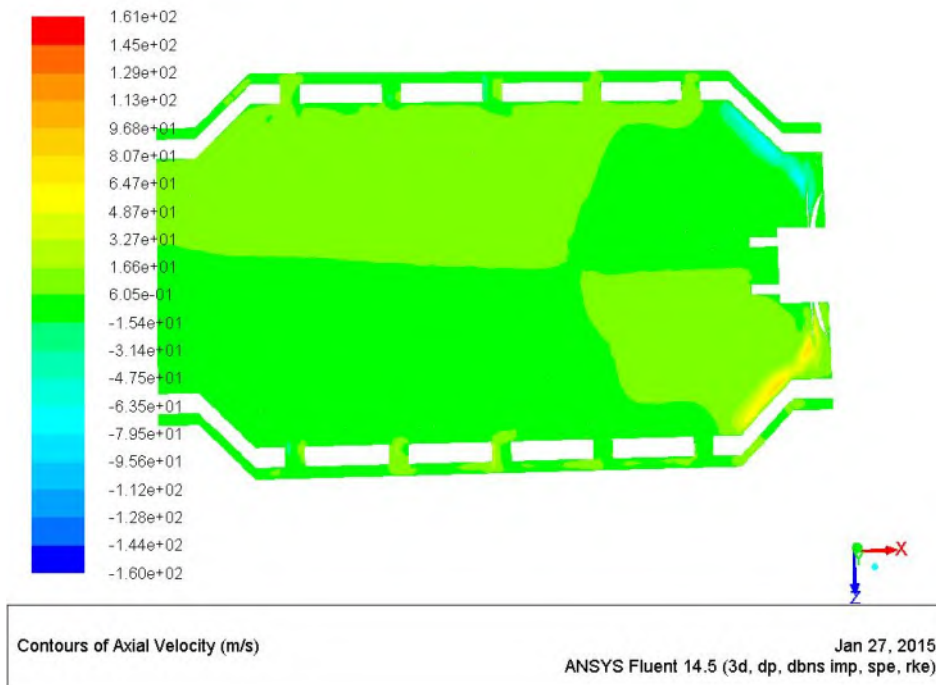
Tangential velocity values are increasing in positive y-axis direction from air inlet and also symmetric at entrance port with respect to the fuel injection tube.

Maximum tangential velocity= **174 m/s**

Minimum tangential velocity= **-177 m/s**



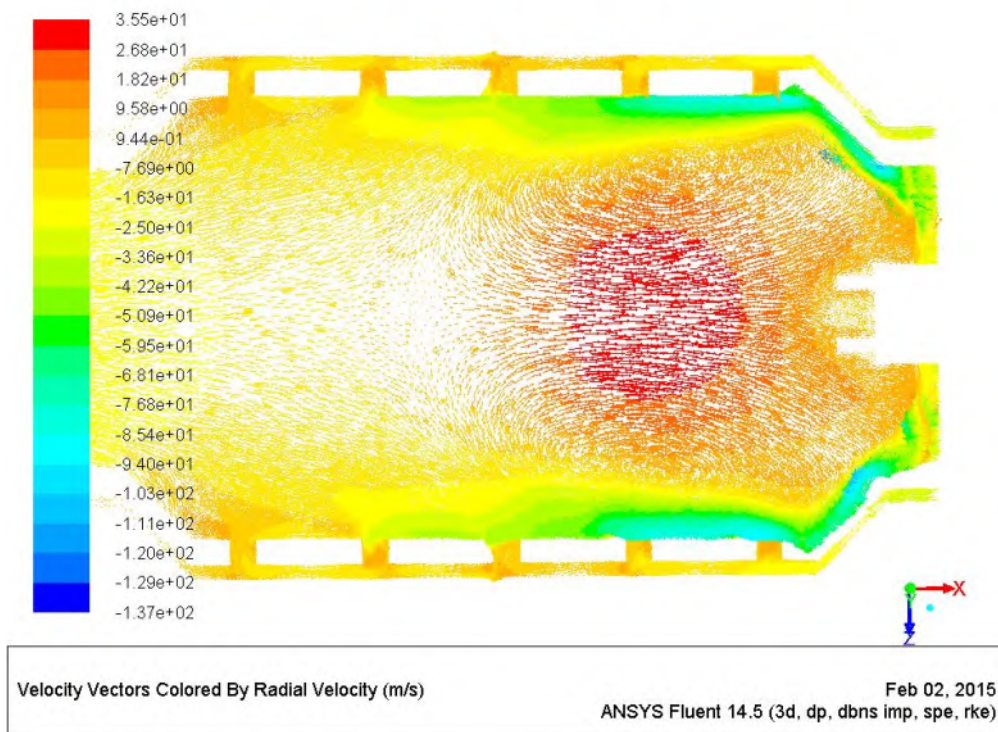
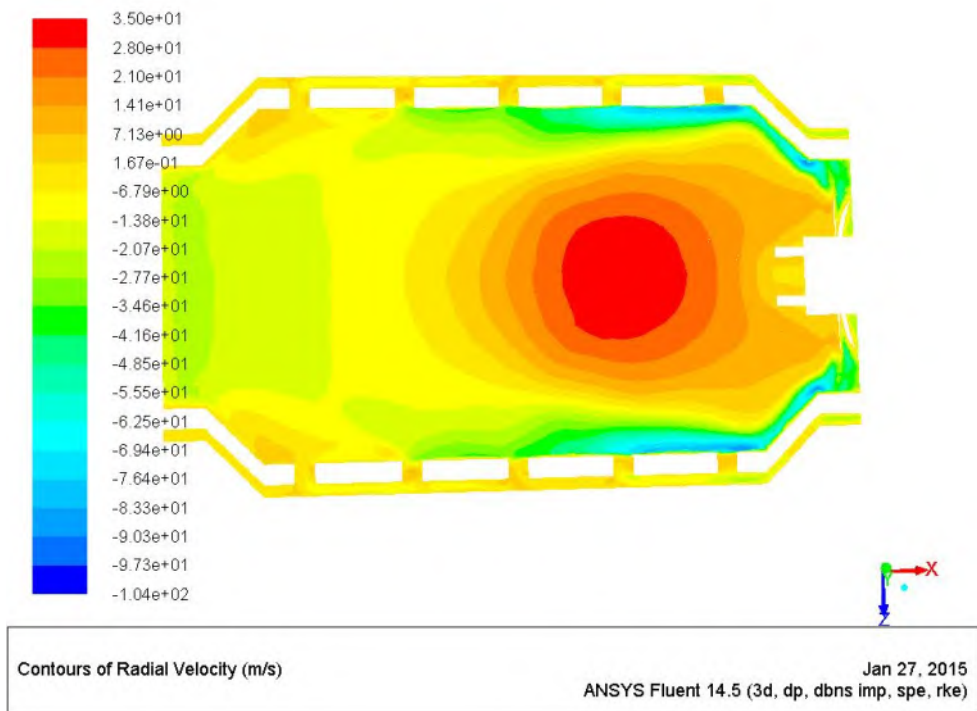
### 10.3.2.5. AXIAL VELOCITY



Maximum value of Axial Velocity = **169 m/s**

Minimum value of Axial Velocity = **-170 m/s**

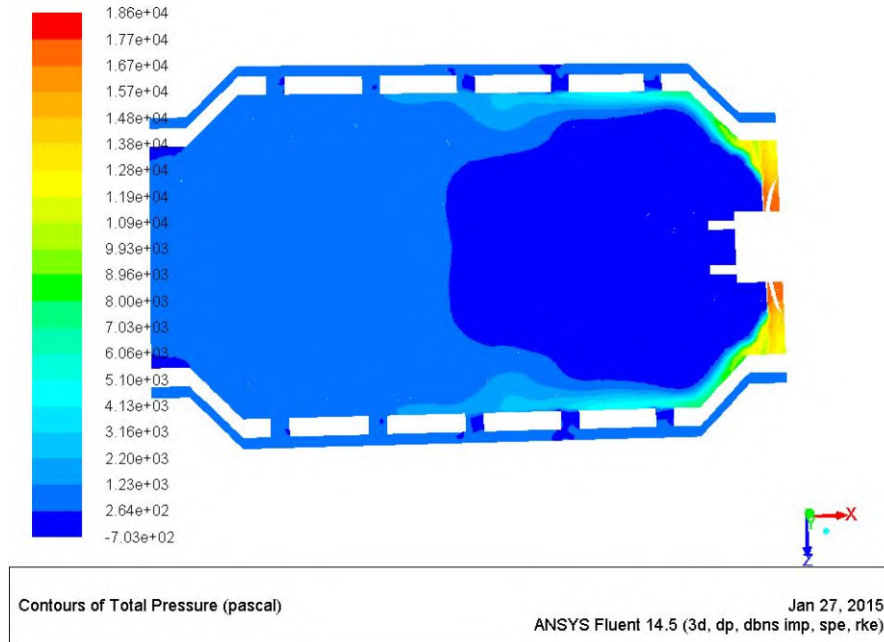
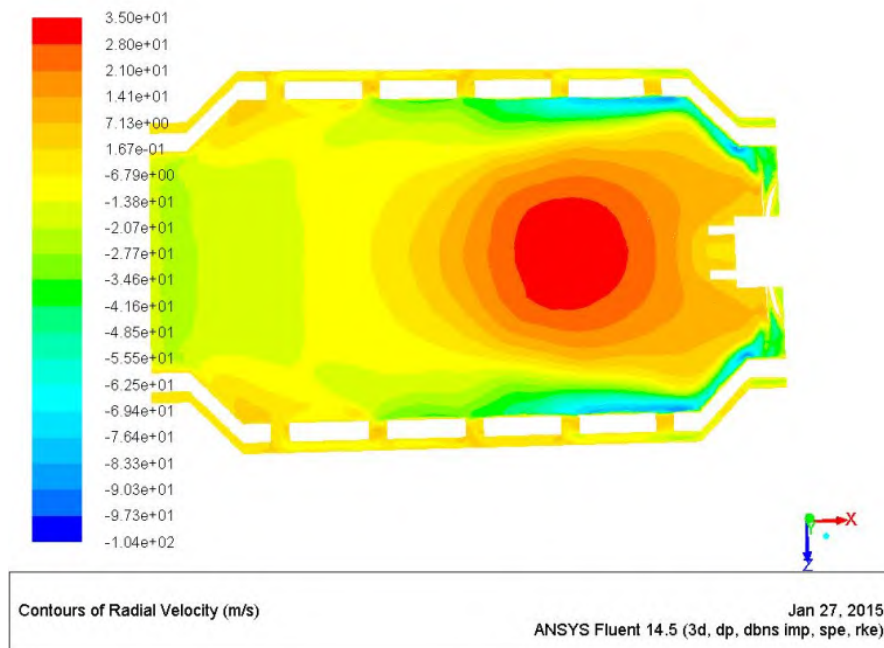
### 10.3.2.6. RADIAL VELOCITY



*Maximum Radial Velocity : 35 m/s*

*Minimum Radial Velocity : -104 m/s*

### 10.3.2.7. RADIAL VELOCITY & TOTAL PRESSURE

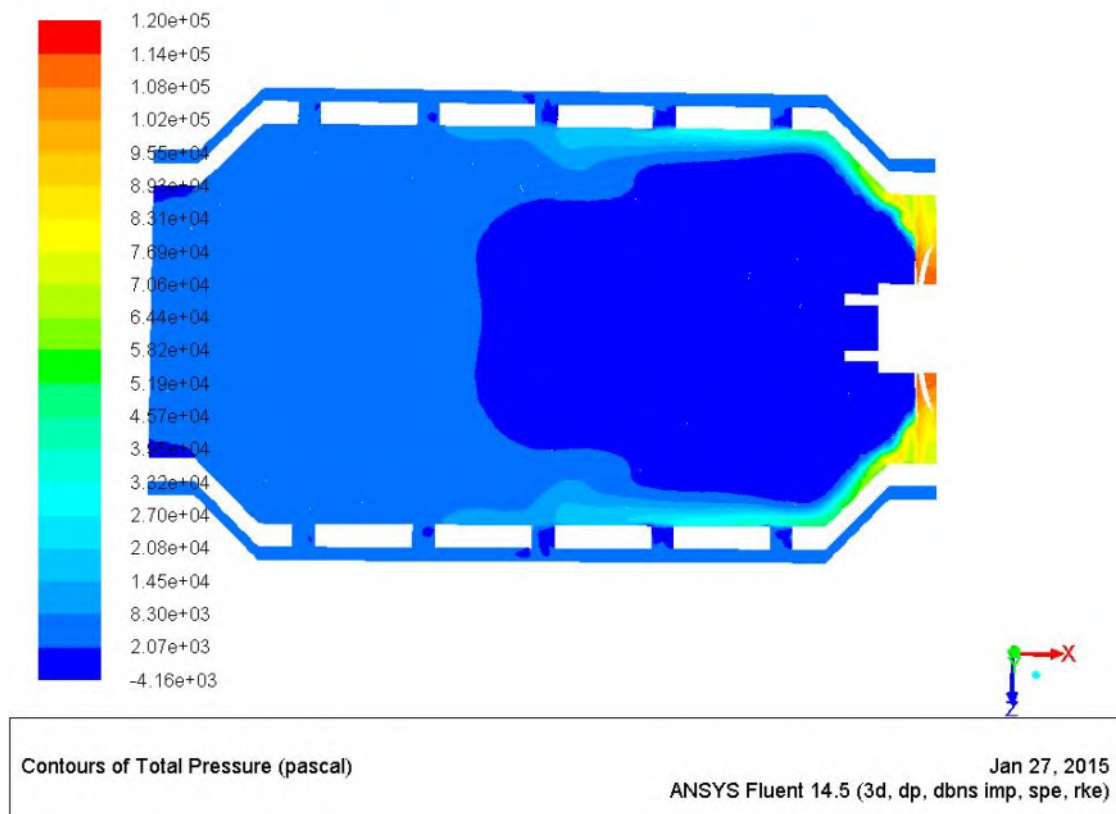


Maximum radial velocity values occurs at minimum pressure values on regional stage.

### 10.3.3.CASE 3 (Air)

- In case 3 also air entrance occurs inside to the combustor.
- Air flows has much more speed 66.8 m/s velocity in entrance regions that is two inlet occurs .
- Same initial conditions used like case 2.

#### 10.3.3.1.TOTAL PRESSURE



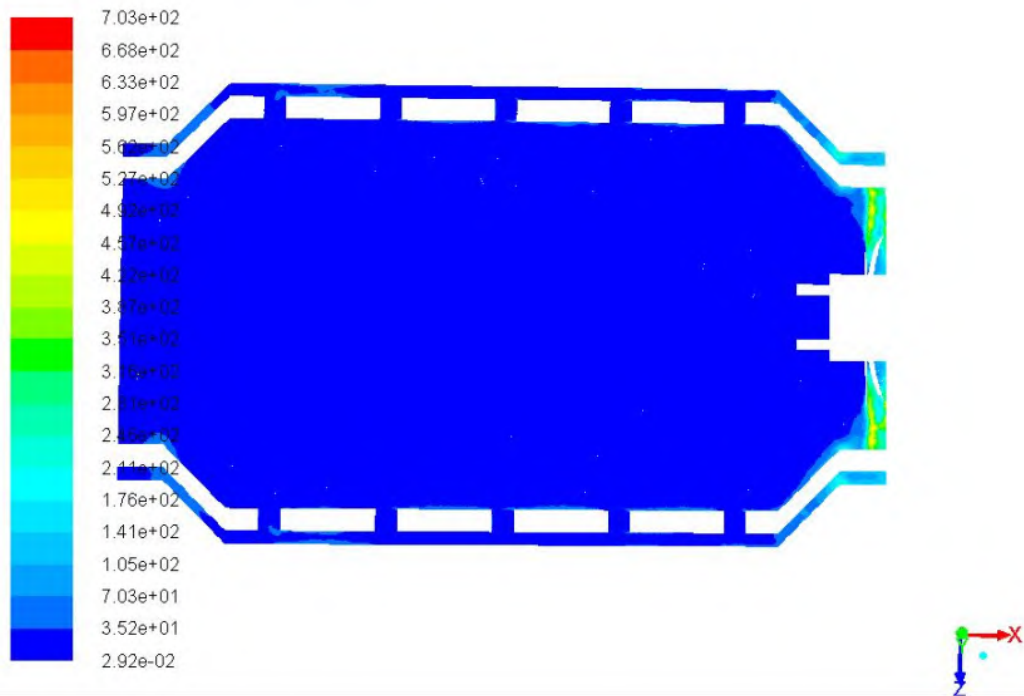
Centerline Jet Velocity of Air is more higher than case 2.

Maximum Total Pressure : **120000 Pa**

Minimum Total Pressure : **-4160 Pa**



### 10.3.3.2.CELL REYNOLDS NUMBER



Contours of Cell Reynolds Number

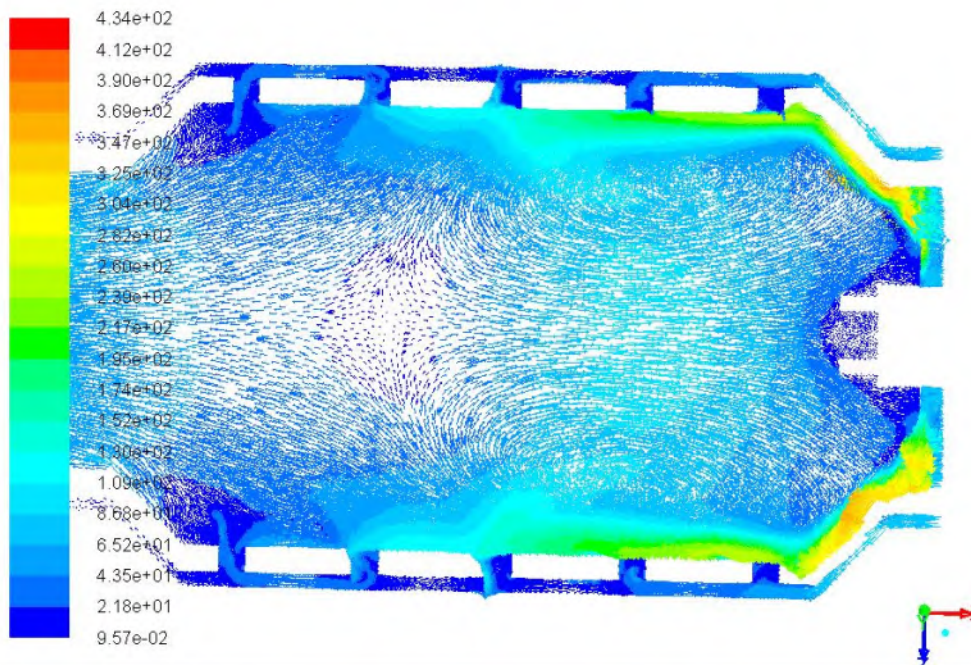
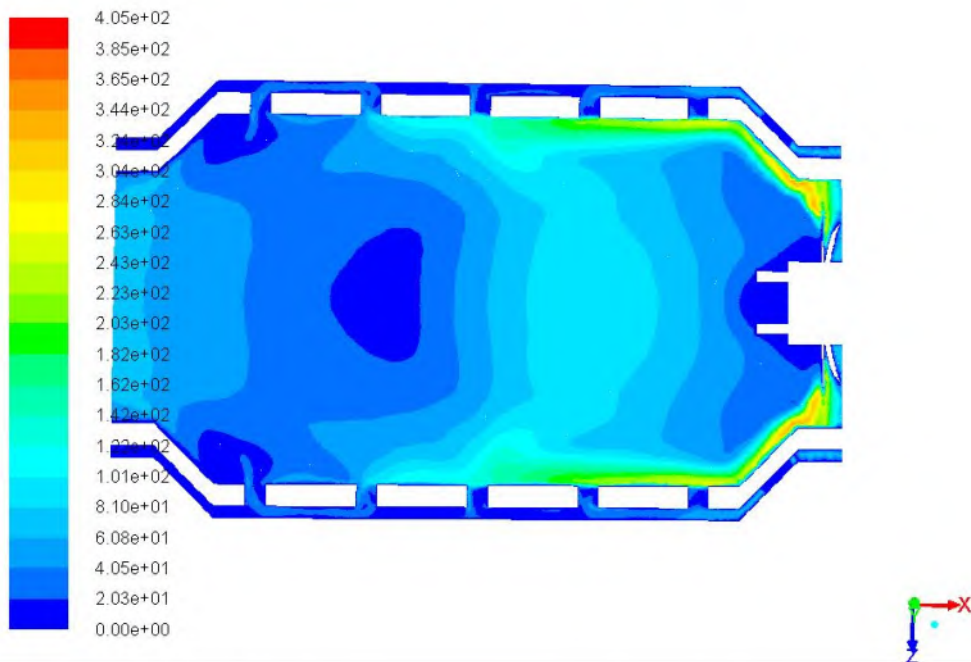
Jan 27, 2015  
ANSYS Fluent 14.5 (3d, dp, dbns imp, spe, rke)

Maximum cell Reynolds number = 703

Minimum cell Reynolds number = -292



### 10.3.3.3. VELOCITY MAGNITUDE



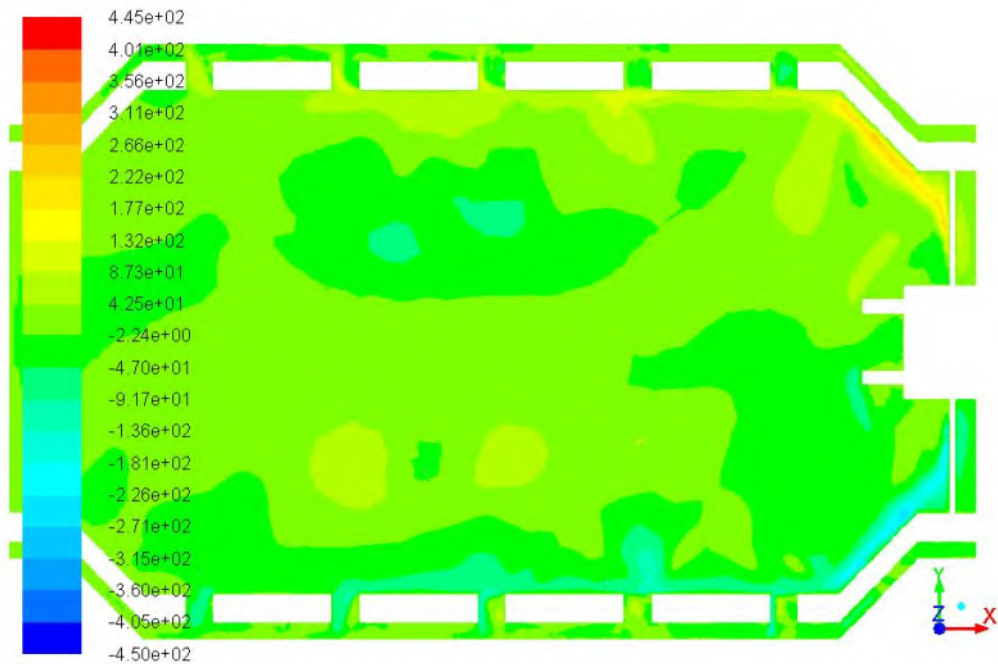
Maximum Velocity Magnitude : 405 m/s

Minimum Velocity Magnitude : 0 m/s

Mach Number= $v/V_{\text{sound}}$ = 1.18=>Transonic

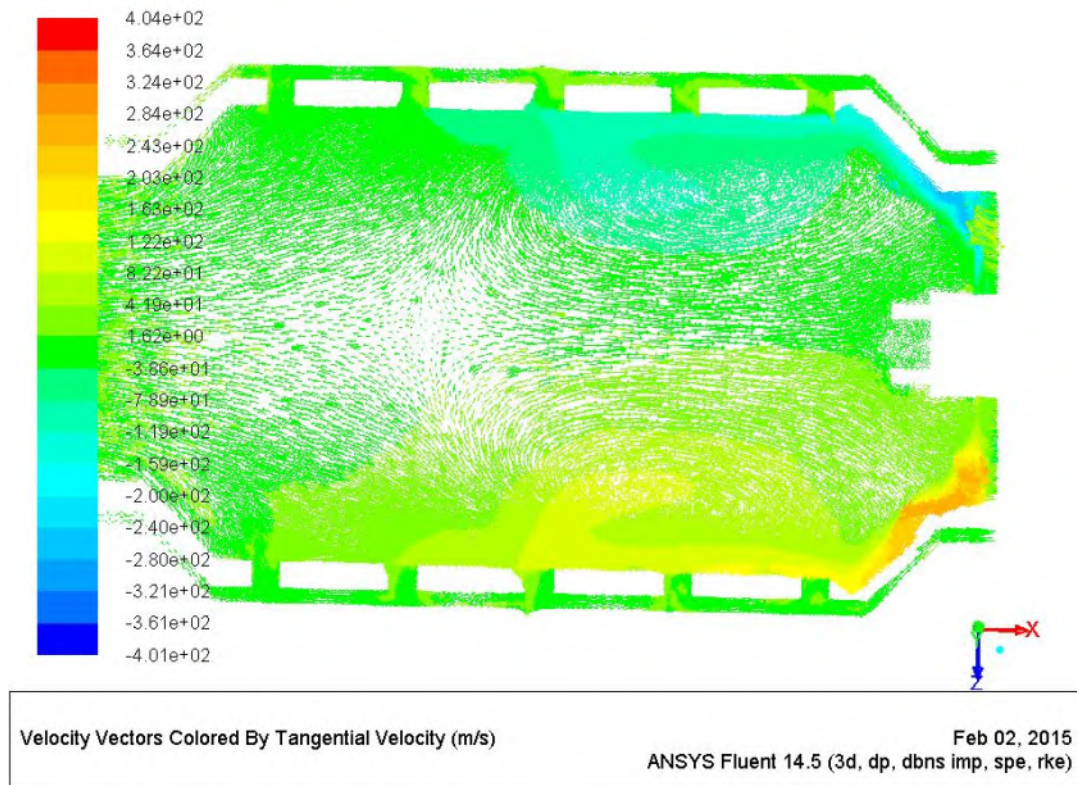
The flow is **turbulent** with respect to evaluated Reynolds number.

#### 10.3.3.4. TANGENTIAL VELOCITY



Contours of Tangential Velocity (m/s)

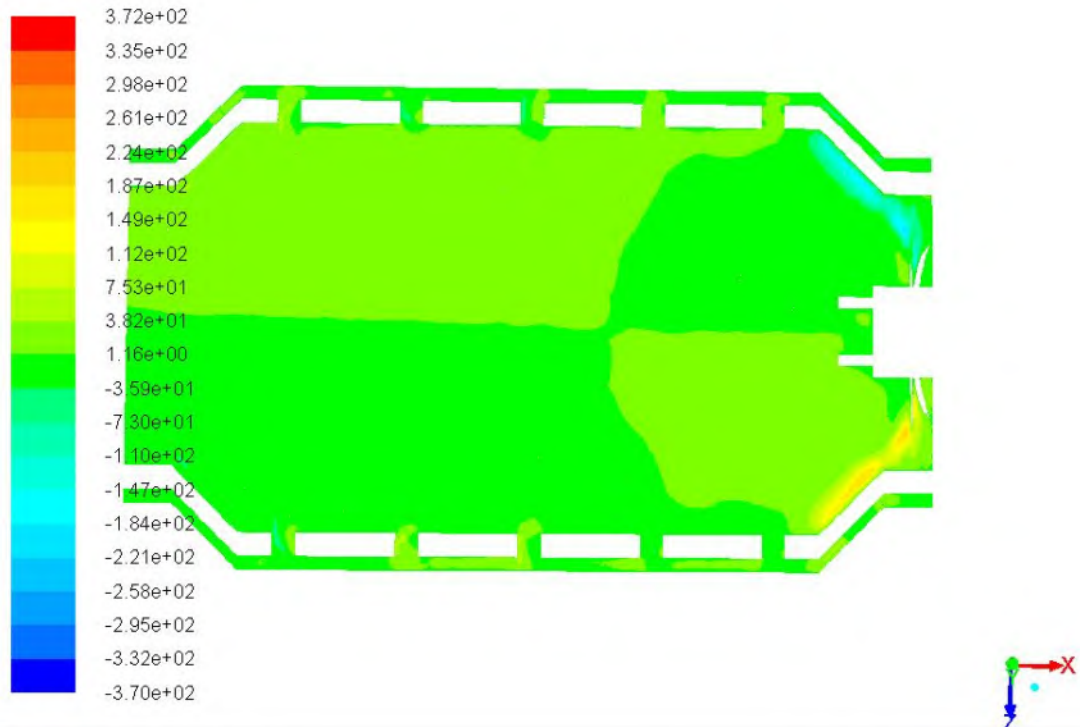
Jun 08, 2014  
ANSYS Fluent 14.5 (3d, dp, pbns, lam)



Maximum Tangential Velocity: **445 m/s**

Minimum Tangential Velocity: **-450 m/s**

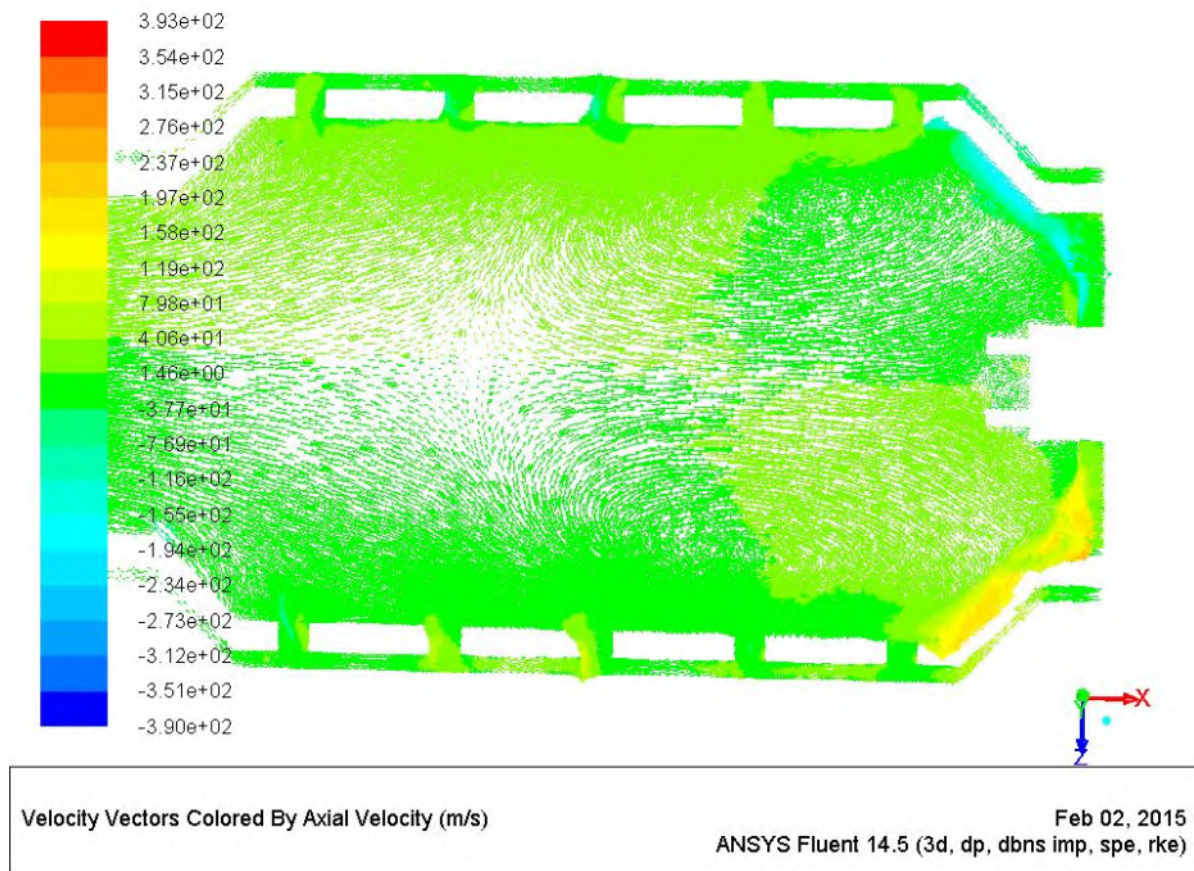
### 10.3.3.5. AXIAL VELOCITY



Contours of Axial Velocity (m/s)

ANSYS Fluent 14.5 (3d, dp, dbns imp, spe, rke)  
Jan 27, 2015

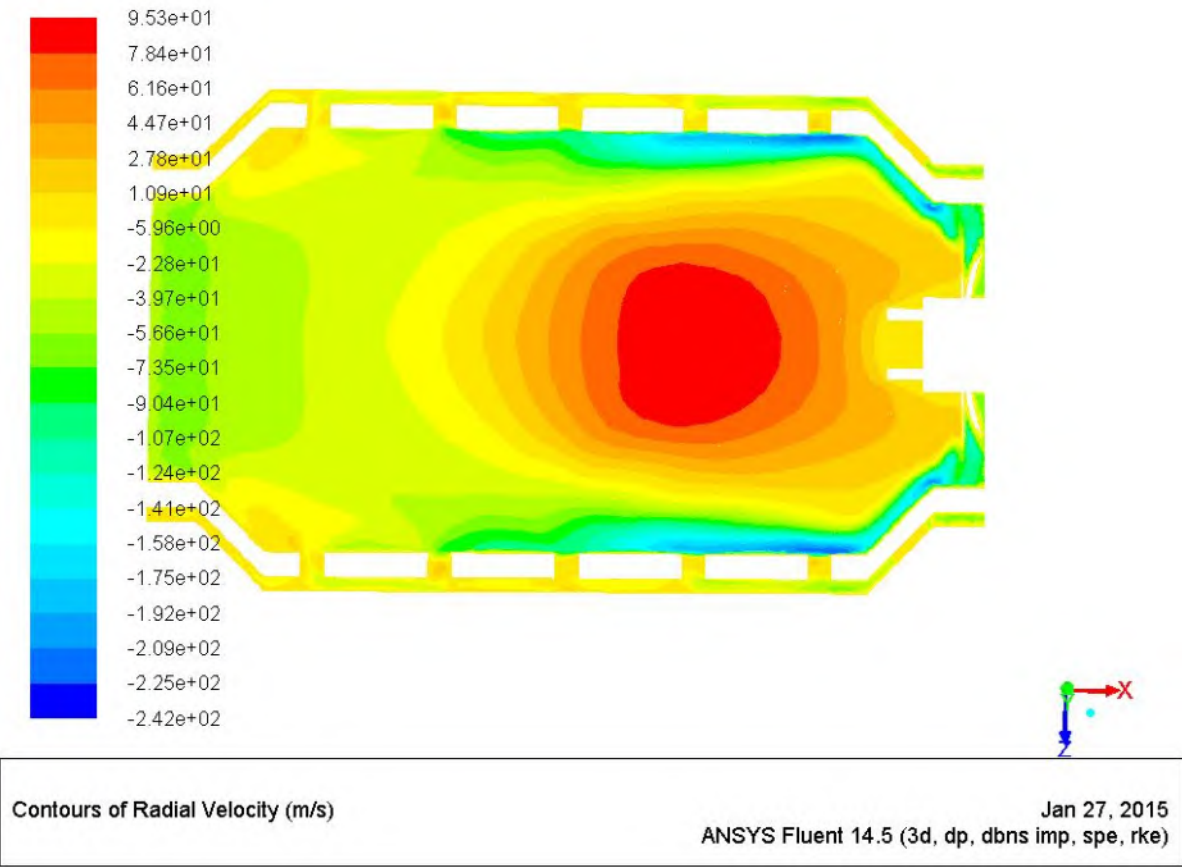


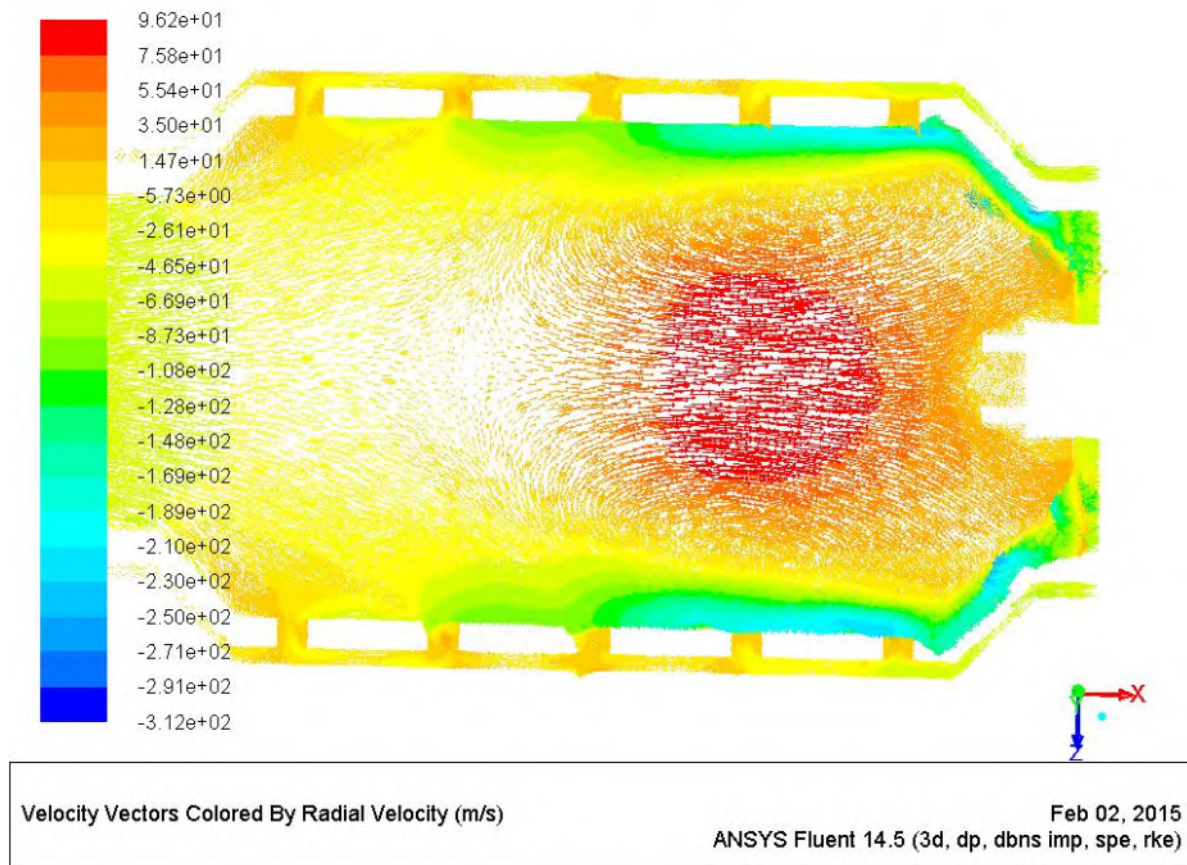


Maximum Axial Velocity = **462 m/s**

Minimum Axial Velocity = **-456 m/s**

### 10.3.3.6. RADIAL VELOCITY



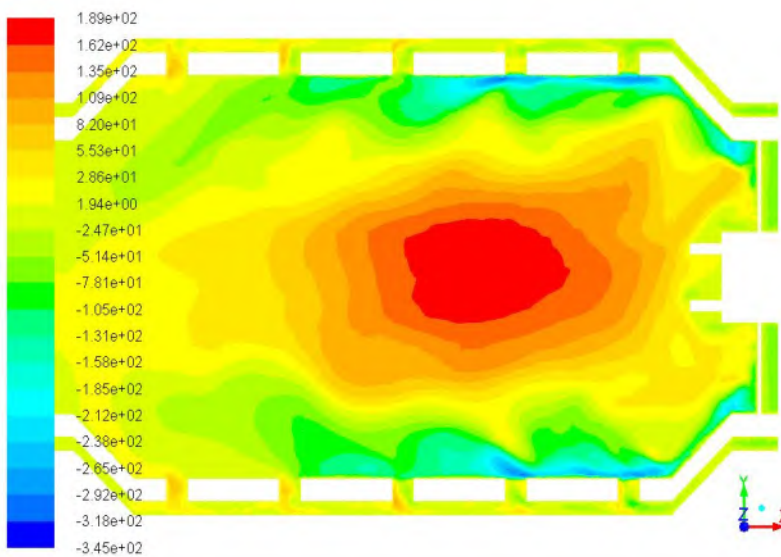


Red regions are the maximum radial velocity regions. The speed increases from 25.4 m/s to 66.8 m/s. The regions can have greater values than Case 2.

***Maximum Radial Velocity: 189 m/s***

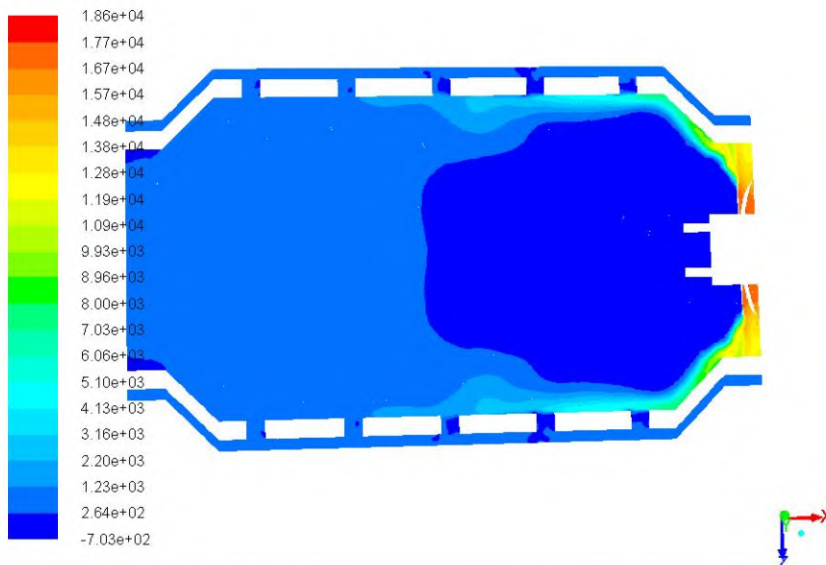
***Minimum Radial Velocity: -345 m/s***

### 10.3.3.7. RADIAL VELOCITY & TOTAL PRESSURE



Contours of Radial Velocity (m/s)

Jun 08, 2014  
ANSYS Fluent 14.5 (3d, dp, pbns, lam)



Contours of Total Pressure (pascal)

Jan 27, 2015  
ANSYS Fluent 14.5 (3d, dp, dbns imp, spe, rke)

Eventually comparison between radial velocity and total pressure. Pressure values in that case also greater than case 2. Therefore radial velocity is influenced for this condition.



## 11. RESULTS AND CONCLUSION

- CFD analysis can explore the the major performance of the fluid flow in the gas turbine combustor.
- An investigation was conducted to understand the characteristics of swirl flow.
- The reliability of the simulation was demonstrated by comparing different cases.
- The predicted results indicate that the swirling flow has a slight decay, and this decay increases towards the exit boundary.
- Swirl flow effects caused by blade in entrance has to be considered in cold-flow simulations in gas turbine engines.
- Velocity magnitudes directly effects flow quality, and design conditions.
- Energy equations has to be solved for combustion process, however it is not needed for cold-flow analysis
- Radial velocity increasing shows us turbulence phenomena.

## 12. REFERENCES

1. Gül M.Z., “SIMULATION OF COLD FLOW IN A MODEL GAS TURBINE COMBUSTION CHAMBER”
2. Combustion and Flame Volume 23, Issue 2, October 1974, Pages 143–201 N.Syred\*, J.M.Beer\*.
3. So, R.M.C.; Ahmed, S.A.; Mongia, H.C.: “Jet Characteristics in Confined Swirling Flow”, Experiments in Fluids, Mechanical and Aerospace Engineering Department, Arizona State University, Tempe, USA, (1995) 3, 221-230.
4. Milne-Thomson, L.M. (1973). *Theoretical Aerodynamics*. Dover Publications.
5. Gül M.Z., SIMULATION OF COLD FLOW IN A MODEL GAS TURBINE COMBUSTION CHAMBER
6. Hogg, S.; Leschziner, M.A.: “Computation of Highly Swirling Confined Flow with a Reynolds Stress Turbulence Model”, J. AIAA, University of Manchester, Manchester, England, U.K. (1999) 27, 57-63.
7. V. Moureau, G. Lartigue, Y. Sommerer, C. Angelberger, O. Colin, T. Poinso, J. Comput. Phys. 202 (2) (2005) 710–736.
8. T. Schönfeld, M. Rudgyard, A cell-vertex approach to local mesh refinement for the 3-D Euler equations, in: 32nd AIAA Aerospace Sciences Meeting and Exhibit, AIAA, Reno, NV, USA, 1994.
9. O. Colin, M. Rudgyard, J. Comput. Phys. 162 (2) (2000) 338–371.
10. F. Nicoud, F. Ducros, Flow Turb. Combust. 62 (3) (1999) 183–200.
11. V. Moureau, G. Lartigue, Y. Sommerer, C. Angelberger, O. Colin, T. Poinso, J. Comput. Phys. 202 (2) (2005) 710–736.
12. T. Poinso, S. Lele, J. Comput. Phys. 101 (1) (1992) 104–129.
13. S. Roux, G. Lartigue, T. Poinso, Combust. Flame 141 (1/2) (2005) 40–54.
14. L. Selle, L. Benoit, T. Poinso, F. Nicoud, W. Krebs, Combust. Flame 145 (1/2) (2006) 194–205.
15. A. Giauque, L. Selle, L. Gicquel, et al., J. Turbomach. 6 (21) (2005) 1–20.
16. C. Prie`re, L. Gicquel, P. Gajan, A. Strzelecki, T. Poinso, C. Be´rat, AIAA J. 43 (8) (2005) 1753–1766
17. H. Forkel, J. Janicka, Flow Turbul. Combust. 65 (2) (2000) 163–175.
18. H. Pitsch, H. Steiner, Phys. Fluids 12 (2000) 2541–2554.
19. H. Pitsch, L. Duchamp de la Geneste, Proc. Combust. Inst. 29 (2002) 2001–2008.
20. H. Pitsch, Ann. Rev. Fluid Mech. 38 (2006) 453–482.
21. V. Sankaran, S. Menon, J. Turbomach. 3 (2002) 011
22. V. Chakravarthy, S. Menon, Flow Turbul. Combust. 65 (2000) 133–161.

