

Computation Fluid Dynamics over the Micro Combustion Chamber with a Varying Pressure & Velocity

D.Brahmaiah¹, K.Chendrasekhar²

M.Tech Student, QIS college of Engineering and Technology, Ongole, A. P, India¹

Associate Professor, Department of Mechanical Engineering, QIS College of Engineering and Technology, Ongole, A.P, India²

ABSTRACT: In this study, temperature distribution in a CFD coated turbine used in an micro combustion chamber engine was investigated and was guided by existing experimental results. A model based on experimental observations obtained from the experiments at Shell research center was developed by using finite element technique to investigate 3D thermal analyses for both clean and deposited turbine. Effect of the CFD thickness on the temperature distribution was analyzed by comparing the results from the clean turbine and deposited one. It was observed that the deposited surface temperature increases with transient step or rotation of blades in a time step rate. Aluminum turbine compared with each other in respect to the maximum temperature for the same CFD thicknesses. Compared to the clean turbine, maximum temperature on the top surface of the aluminum and steel metals diminish 800 °C and 1500 °C respectively, because of CFD's relatively low heat transfer coefficient.

KEYWORDS: Temperature distribution on the deposit CFD is a function of coating thickness

I. INTRODUCTION

A combustor is a component or area of a gas turbine, ramjet, or scramjet engine where combustion takes place. It is also known as a burner, combustion chamber or flame holder. In a gas turbine engine, the combustor or combustion chamber is fed high pressure air by the compression system. The combustor then heats this air at constant pressure. After heating, air passes from the combustor through the nozzle guide vanes to the turbine. In the case of a ramjet or scramjet engines, the air is directly fed to the nozzle. A combustor must contain and maintain stable combustion despite very high air flow rates. To do so combustors are carefully designed to first mix and ignite the air and fuel, and then mix in more air to complete the combustion process. Early gas turbine engines used a single chamber known as a can type combustor. Today three main configurations exist: can, annular and cannular (also referred to as can-annular turbo-annular). Afterburners are often considered another type of combustor. Combustors play a crucial role in determining many of an engine's operating characteristics, such as fuel efficiency, levels of emissions and transient response (the response to changing conditions such as fuel flow and air speed).

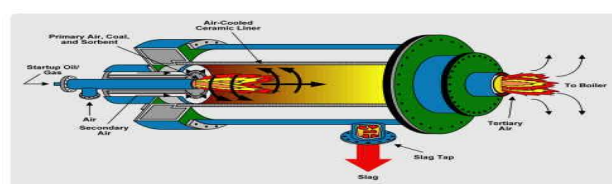


Figure 1.1 Concept of turbine

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

1.1. Internal combustion engine

ICEs typically comprise reciprocating turbine engines, rotary engines, gas turbines and jet turbines. The combustion process increases the internal energy of a gas, which translates into an increase in temperature, pressure, or volume depending on the configuration. In an enclosure, for example the cylinder of a reciprocating engine, the volume is controlled and the combustion creates an increase in pressure. In a continuous flow system, for example a jet engine combustor, the pressure is controlled and the combustion creates an increase in volume. This increase in pressure or volume can be used to do work, for example, to move a turbine on a crankshaft or a turbine disc in a gas turbine. If the gas velocity changes, thrust is produced, such as in the nozzle of a rocket engine.

Fundamentals

The objective of the combustor in a gas turbine is to add energy to the system to power the turbines, and produce a high velocity gas to exhaust through the nozzle in aircraft applications. As with any engineering challenge, accomplishing this requires balancing many design considerations, such as the following:

Completely combust the fuel. Otherwise, the engine wastes the unburnt fuel and creates unwanted emissions of unburnt hydrocarbons, carbon monoxide (CO) and soot. Low pressure loss across the combustor. The turbine which the combustor feeds needs high pressure flow to operate efficiently. The flame (combustion) must be held (contained) inside of the combustor. If combustion happens further back in the engine, the turbine stages can easily be overheated and damaged. Additionally, as turbine blades continue to grow more advanced and are able to withstand higher temperatures, the combustors are being designed to burn at higher temperatures and the parts of the combustor need to be designed to withstand those higher temperatures. It should be capable of relighting at high altitude in an event of engine flame-out.

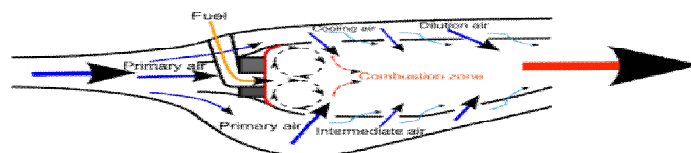


Figure 1.2 Air flow paths

Uniform exit temperature profile. If there are hot spots in the exit flow, the turbine may be subjected to thermal stress or other types of damage. Similarly, the temperature profile within the combustor should avoid hot spots, as those can damage or destroy a combustor from the inside.

Small physical size and weight. Space and weight is at a premium in aircraft applications, so a well designed combustor strives to be compact. Non-aircraft applications, like power generating gas turbines, are not as constrained by this factor.

II. LITERATURE REVIEW

2.1. Converting Potential Energy

Humans have been inventing engines, or machines designed to convert potential energy into useful mechanical energy, for thousands of years. The invention of the oar, one of the earliest known forms of the lever, dates back to the Neolithic period. As innovations in science and technology improved over the millennia, sophistication of engines improved as well. In the 13th century, the rocket motor was invented in China, and though it was not able to supply sustained power, it used gunpowder (invented by the Chinese in the ninth century) to propel projectiles for weaponry and fireworks. Nearly a millennium later, the industrial revolution of the 18th century led to the increased use of steam power. The atmospheric engine (or Newcomen engine) was invented in the early 18th century by Thomas Newcomen

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

and was the first machine to practically harness the power of steam to generate mechanical work. Over the years, many different people made improvements on Newcomen's work and eventually the steam turbine was developed.

2.2. The Turbine

A turbine is a machine that utilizes a wheel or rotor to produce continuous sustainable power, generated by the motion of water, steam, gas, air, or any other fluid. The flow of a gas or liquid is strategically placed to drive rotating parts of the turbine to generate power. A turbine will always incorporate at least one moving part known as a rotor assembly, which is a long shaft with attached blades. The flow of fluids over these blades is designed to rotate the turbine, thus generating power. Turbines have evolved much over history from their origins in early windmills and waterwheels. There are many different types of turbines, but the most common turbines are wind, water, steam, and gas.

2.3.1. Wind Turbines

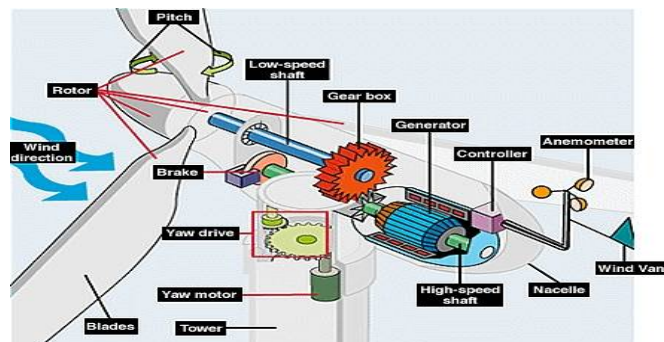


Figure 2.1 Wind Turbine.

Wind turbines are a bit different in that they utilize kinetic energy of wind to generate electricity, as opposed to steam or the combustion of fossil fuels. Obviously, there are inherent limitations to the power available, which are completely dependent on the natural forces and weather of the area. Thus, wind turbines are strategically located in places known to experience large amounts of consistent wind. The benefits of wind turbines are that wind is a renewable resource and has no attached inherent cost. Like all sources of power, wind turbines do have limitations on their potential efficiency. Due to conservation of mass principles, Beltz' law shows the maximum efficiency of a wind turbine to be 59%.

2.3.2. Water Turbines

Water turbines date as far back as the early Roman Empire, but were significantly developed in the 1800s and supplied industrial power before electrical grids became prevalent. They commonly utilize some form of water barrier such as an embankment dam to limit a large flow of water through a narrowed channel. This routed flow of water is then engineered to rotate a turbine to generate electricity. Modern water turbines are known to operate at greater than 90% mechanical efficiencies, but are still limited by the flow and volume of water available in terms of maximum potential power output. One of the main advantages of water turbines is that water is considered to be a completely renewable resource. Unlike steam turbines, which heat water to create steam, the water used in a water turbine is largely unaltered, and more importantly, unheated. Steam turbine engineers face the dilemma of properly disposing of large amounts of very hot water. The downside of water turbines is the potential for negatively affecting the bodies of water they depend on. Water turbines rely heavily on the construction of dams, which can disrupt the natural ecology of

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

the water supply, negatively affecting both wildlife and the livelihood of industries and people who may depend on it. Even so, water turbines still produce a significant amount of electricity consumed in the world today.

2.3.3. Steam Turbine Engines : Steam engines employ the thermal energy from steam under pressure to mechanically drive its rotating shaft. Hero of Alexandria described the earliest device resembling a steam engine in the first century in Roman Egypt. Sir Charles Parsons invented the first modern steam turbine resembling steam turbines of today in 1884, and revolutionized the world by providing affordable and plentiful electricity. In fact, most of the electricity generated in the United States today is still produced by some form of steam turbine. Today, steam turbines use coal, natural gas, oil, or nuclear fission to create steam. The steam is then routed through carefully designed turbine parts that drive a power plant's generator.

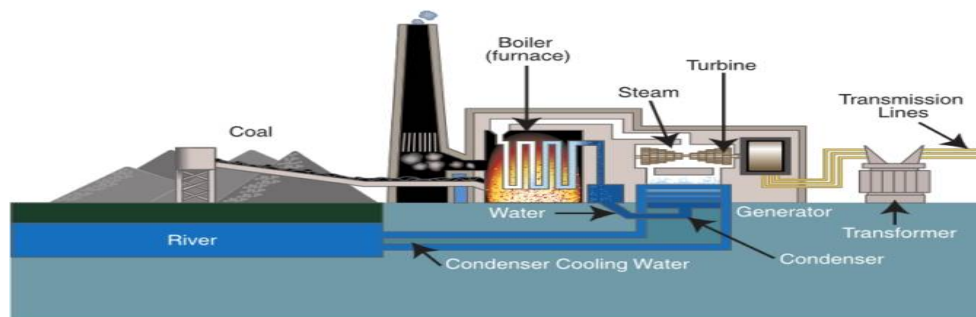


Figure 2.2 Coal Powered Steam Turbine

2.3.4. Gas Turbine Engines

Gas turbine engines utilize an internal combustion to generate the force needed to rotate the turbine. The most well known example of a gas turbine engine is the jet engine. Jet engines use accelerated exhaust to provide thrust by jet propulsion, following Isaac Newton's laws of motion. Obviously, jet engines are now used commonly in both commercial and military airplanes to provide the thrust required to fly. The most significant leap in gas turbine engines was made in 1941, when Frank Whittle, British Royal Air Force engineer, singlehandedly invented the jet engine. General Electric, who had been a leading force in the steam turbine industry, arranged for a license to use Whittle's engine. Expanding from their proprietary GE turbine parts and knowledge, they were able to quickly produce and improve upon Whittle's work. By 1944, they had created the I-40 engine, which would be used in military aircraft like the P-80 Shooting Star. It wouldn't be long before this technology would expand from military to civilian applications. To this day, General Electric turbines still provide the jet thrust needed to fly many of the aircraft in the world today.

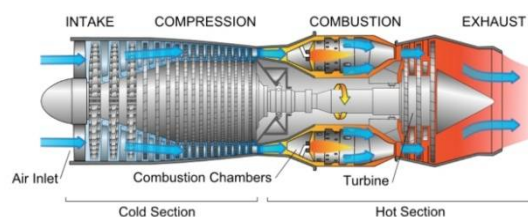


Figure 2.3 Jet Engine With Turbine

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

III. INTRODUCTION CATIAV5R20

CATIA is the leading solution for product success. It addresses all manufacturing organizations. CATIA can be applied to a wide variety of industries- from aerospace- automotive- and industrial machinery- to electronics- shipbuilding- plant design- and consumer goods. Today- CATIA is used to design anything from an airplane to jewelry and clothing. With the power and functional range to address the complete product development process- CATIA supports product engineering- from initial specification to product-in-service- in a fully-integrated manner. It facilitates reuse of product design knowledge and shortens development cycles- helping enterprises to accelerate their response to market needs.

. Modeling Of Slap And Slider For Friction Surface

. A Part 1: Slap

To create go to catia software and click on start – mechanical design and select part design module. In part design module then go to sketcher tool and select sketcher tool and select required plane. Then we enter into sketcher module and go to profile and create a profile structure as shown in fig. After completing sketcher go to workbench and select exit work bench. Then we enter into part module again. In part module, go to sketcher based feature and select pad tool. In pad definition, select above sketch as selection profile and specify length as shown in fig.

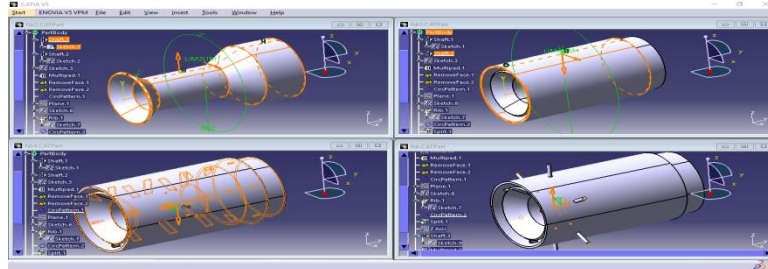


Figure 3.1 Pad Tool , shaft and rib

To create go to catia software and click on start – mechanical design and select part design module. In part design module then go to sketcher tool and select sketcher tool and select required plane. Then we enter into sketcher module and go to profile and create a profile structure as shown in fig.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

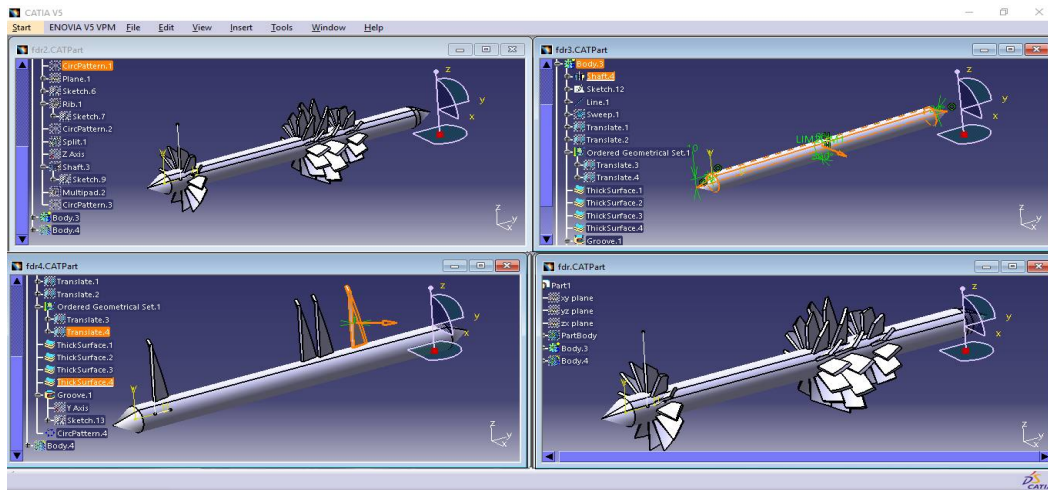


Figure 3.2 Shaft Blade

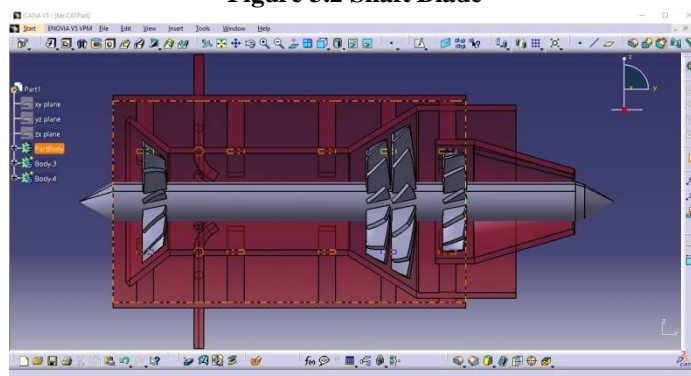


Figure 3.3 combustion chamber and its section views

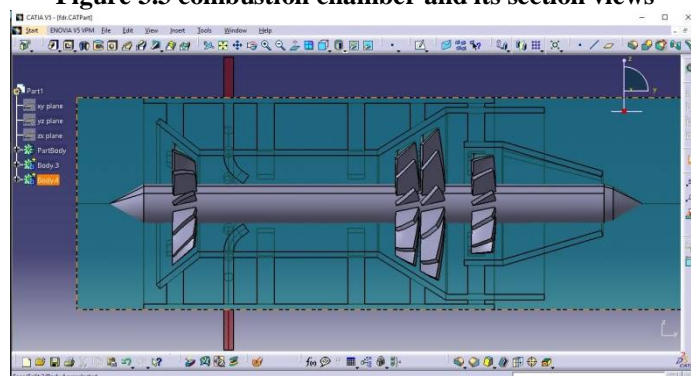


Figure 3.4 combustion chamber with zone creation of cfd

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

IV. INTRODUCTION OF ANSYS

ANSYS a product of SASI (System Analysis Simulation And Integrating). It is a world's leading software which run on CAE (computer aided engineering) and FEM (finite element methods). It is widely used by designers/analysis in industries such as aerospace, automotive, manufacturing, nuclear, electronics, biomedical, and much more. ANSYS provides simulation solution that enables designers to simulate design performance directly on the desktop. In this way, it provides fast efficient and cost efficient product development from design concept stage to performance validation stage of product development cycle.

In ANSYS, the basics of FEA concepts, modelling and the analysing of engineering problem using ANSYS workbench. In addition, describe of importance tools and concepts given whenever required .this following simulation streams of ANSYS.

Ansyes software is divide into 3 characterizes analysis

1. Structural Analysis
2. Thermal Analysis
3. Computational fluid dynamic.
- 4.1. Introduction to FEA

4.1.1. Project objectives

After complete of this chapter,

- The basic concept and general working of FEA
- Understand the advantage and limitations of FEA
- Understanding the analysis type
- Understanding important terms and definition of FEA

4.2.. Elements type

- Real constants for elements (cross-sectional area, moments of inertia, shell thickness and soon)
- Material properties (young's modulus, Poisson's ratio, spring constant, thermal conductivity, coefficient of thermal expansion, and soon)
- Geometry model (either created in FEA software or imported from cad package)
- FEA model (discretizing the geometric model into small elements)
- Loading and boundary conditions (defining loads, pressures, moments, temperature, conductivity, convection, constraints (fixed, pinned or frictionless/ symmetrical)), and so on.
- The input data are preprocessed for the output data and the preprocessor generates the data files automatically with help of uses.

4.2.3. Solution

The solution phase is completely automatic. The FEA software generates elements matrices, compute nodal value and derivatives, and stores the result data in files. These files are further used in the subsequent phase (postprocessor) to review and analyse the results through the graphics display and tabular listing.

4.2.4. Postprocessor

The output from the solution phase (results in data files) is in the numerical form and consist of nodal values of the field variable and its derivative. For example, in structural analysis, the output of the post processor is nodal displacements and stress in elements. The postprocessor processes the results data and displays them in graphical form to check or analysis the results. The graphical output gives the detailed information about the required results data. The postprocessor phase is automatic and generates graphical output in the specified form.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

V. ADVANTAGE AND LIMITATIONS OF FEA SOFTWARE

Following are some of the advantage and laminations of FEA software.

5.1. Advantage

- It reduces the amount of prototype testing, thereby saving the cost and time.
- It gives the graphical representation of the results of analysis
- The finite elements modelling and analysis are performed in the preprocessor and solution phases which if done manually would consume a lot of time and in some cases, might be impossible to perform.
- Variables such as stress and temperature can be measured at any desired point of the model
- It helps optimise a design
- It is used to simulate the designs that are not suitable for prototype testing.
- Its helps you create more reliable, high quality, and competitive designs.

5.2. Limitations

- It does not provide exact solutions
- FEA package are costly
- An inexperienced user can deliver incorrect answer, upon which expensive decision will be based
- Results give solutions but not remedies
- Feature such as bolts, welded joints, and so on cannot be accommodated to model. This may lead to approximation and errors in the results.
- For more accurate results, more hard disk space, ram, and time are required

VI. INTRODUCTION TO ANSYS WORKBENCH

6.1. Project objective

In this chapter, we will be able to define:

- Understand the types of system
- Understand different types of cells
- Understand the graphic user interface of the workbench window
- Start a new project in ansys workbench windows
- Add the first and subsequent analysis system to a project
- Set units for the project

6.2. Introduction to ansys workbench

In ansys workbench this are the list of analysis can be determined:

- Modal analysis
- Static structural analysis
- Transient structural analysis
- Steady state thermal analysis
- Transient thermal analysis
- Fluid flow (CFD) etc

6.2.1. Starting ansys workbench

To start ansys workbench, choose start- programs/ all programs- ansys - workbench from the taskbar. Alternatively, we can start ansys workbench by double click on the workbench 16/17/18/19.0.

The workbenches windows helpstreamline an entire project to be carried out in ansys workbench. In this window, one can create, manage, and view the workflow of the entire project creates by using standard analysis system. The workbench windows mainly consist of the menu bar, standard toolbar, the toolbar windows, project schematic windows, and the status bar.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

6.2.2. Toolbox windows

The toolbox windows are located on the left in the workbench windows. The toolbox windows list the standard and customised templates or the individual analysis components that are used to create a project. To create a project, drag a particular analysis or component system from the toolbox window and drop into the project schematic windows or double click on GUI table it will add it into project schematic windows and to create the project

6.2.3. Table of analysis and definitions

Name Of Analysis	Application Of Loads	Solution Determines
Fluid Flow (CFX)	Compressible Or Incompressible Of Air Or Gases	Heat Transfer Or Flow Of Air
Fluid Flow (CFD)	Compressible Or Incompressible Of Fluid, Air Or Gases	Heat Transfer Fluid and flow

Table of analysis and definitions

6.3. Project schematic windows

The project schematics windows help manage an entire project. It displays the workflow of entire analysis project. To add an analysis system to the project schematic windows, drag the analysis system from toolbox windows and drop into the green coloured box displayed in the project schematic windows.

Step 1: importing and modification of geometry

- Go to import option on top and select
- Then geometry will display
- Later, go fluent flow fluid on analysis system on left side and double over it
- After the fff, go to the geometry on geometry select it and drag it and over geometry on fff as shown in below.

6.3.1. Custom system analysis.

By default, the custom system toolbox is also displayed in collapsed state in the toolbox. To expand this node, click on + on a custom system. The system in the customs system toolbox is used to carry out the standard coupled analysis like static and thermal analysis (the combination of more than single or multiple of GUI tab). In every GUI tabs, we can drag more GUI tabs makes the links analysis.

6.3.2. Component of the system:

An item that is added from the toolbox window to the project schematic windows is known as a system and the constituent elements of the system are known as cells. Each cell of a system plays an important role in carrying out a project and are discussed next

6.3.2.1. Engineering data cell

The engineering data cell is used to define the material to be used in the analysis. To define the materials, double click on the engineering data cell, the workbench corresponding to this the engineering data cell will be displayed.

Engineering cell-double click-click on the shell system (engineering data book)-select general materials in the outline of the engineering data sources- select materials in the outline of general materials

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

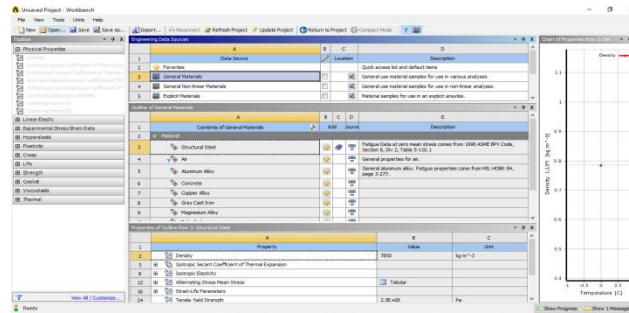


Figure 6.1 The engineering data workspace

6.3.2.2. Geometry cell

The geometry cell is used to create, edit or import the geometry that is used for analysis. To create geometry for analysis, double click on geometry cell, the design modeller windows will be displayed. The new geometry option in the menu is used to get into design modeller windows, where you can create geometry or import the geometry from the existing geometry file create in another CAD software packages.

6.3.2.3. Model cell

The model cell will be displayed for mechanical analysis system and is used to discredited geometry into small elements, apply boundary and load conditions, solve the analysis, and so on. 6.3.2.4. Mesh cell The mesh cell will be displayed for fluid flow analysis and is used to mesh the geometry, on double clicking on this cell, the meshing windows will be displayed. In other words, this cell is associated with the meshing windows.

6.3.2.5. Setup cell

The setup cell is used to define the boundary conditions of an analysis system, such as loads and constraints. This cell is also associated with the mechanical workspace.

6.3.2.6. Solution cell

The solution cell is used to solve the analysis problem based on the conditions defined in the cells above the solution cell. The cell is also associated with the mechanical workspace.

6.3.2.7. Results cell

The results cell is used to display the results of the analysis in the user specified formats; this cell is also associated with the mechanical workspace.

VII. COMPUTATIONAL FLUID DYNAMICS (CFD)

Computational Fluid Dynamics (CFD) is a computer-based tool for simulating the behavior of systems involving fluid flow, heat transfer, and other related physical processes. It works by solving the equations of fluid flow (in a special form) over a region of interest, with specified (known) conditions on the boundary of that region. Computers have been used to solve fluid flow problems for many years. Numerous programs have been written to solve either specific problems, or specific classes of problems. From the mid-1970s, the complex mathematics required to generalize the algorithms began to be understood, and general purpose CFD solvers were developed. These began to appear in the early 1980s and required what were then very powerful computers, as well as an in-depth knowledge of fluid dynamics, and large amounts of time to set up simulations. Consequently, CFD was a tool used almost exclusively in research. Recent advances in computing power, together with powerful graphics and interactive 3D manipulation of models, have made the process of creating a CFD model and analyzing results much less labor intensive, reducing time and, hence, cost. Advanced solvers contain algorithms that enable robust solutions of the flow field in a reasonable

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

time. As a result of these factors, Computational Fluid Dynamics is now an established industrial design tool, helping to reduce design time scales and improve processes throughout the engineering world. CFD provides a cost-effective and accurate alternative to scale model testing, with variations on the simulation being performed quickly, offering obvious advantages.

7.1. Uses of CFD

CFD is used by engineers and scientists in a wide range of fields. Typical applications include:

- Process industry: **Mixing vessels, chemical reactors**
- Building services: **Ventilation of buildings, such as atriums**
- Health and safety: **Investigating the effects of fire and smoke**
- Motor industry: **Combustion modeling, car aerodynamics**
- Electronics: **Heat transfer within and around circuit boards**
- Environmental: **Dispersion of pollutants in air or water**
- Power and energy: **Optimization of combustion processes**
- Medical: **Blood flow through grafted blood vessels**

7.2 Project objective

ANSYS fluent is a general purpose Computational Fluid Dynamics (CFD) software suite that combines an advanced solver with powerful preprocessing and post processing capabilities. It includes the following features:

- An advanced coupled solver that is both reliable and robust.
- Full integration of problem definition, analysis, and results presentation.
- An intuitive and interactive setup process, using menus and advanced graphics.

7.2 .1 ANSYS fluent is capable of modeling:

- Steady-state and transient flows
- Laminar and turbulent flows
- Subsonic, transonic and supersonic flows
- Heat transfer and thermal radiation
- Buoyancy
- Non-Newtonian flows
- Transport of non-reacting scalar components
- Multiphase flows
- Combustion
- Flows in multiple frames of reference
- Particle tracking.

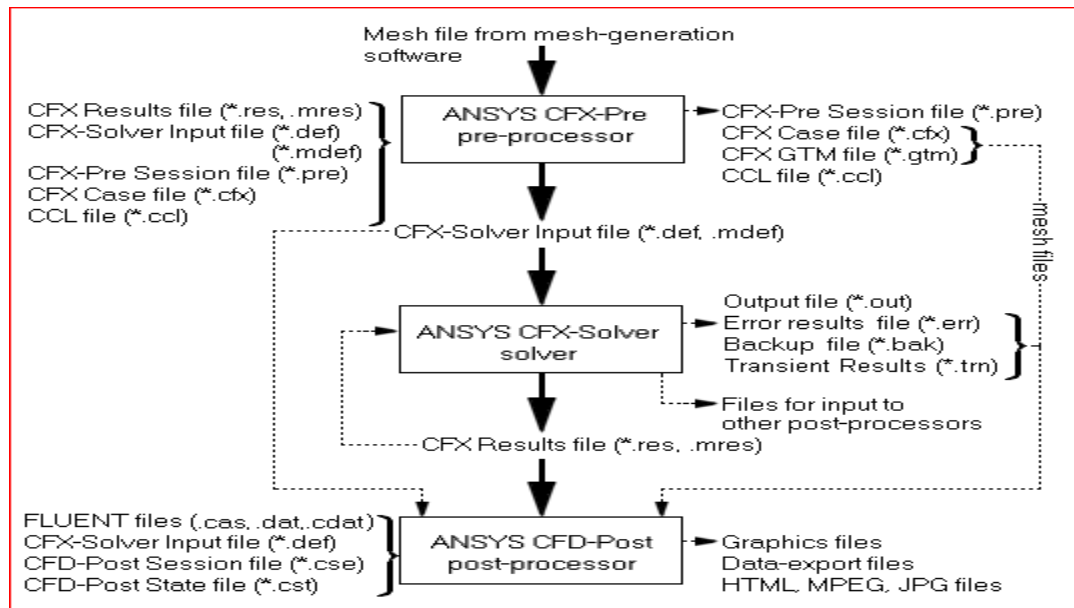
During the process of creating the model, running the CFX-Solver and analyzing the results, a number of different files are created by the various modules of the software. This section describes some of these files and their purposes. The use of these files with their default extension is shown in the following flowchart. The standard files used and produced are indicated with solid black lines; other possible uses are indicated with dotted lines

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019



7.3. Project Object

- To find the flow transfer over the cfd

After opening the software, we can see a white screen called it as a project schematic (ps) and on left we can see the analysis system tool box. Now double click on the fluent fluid flow analysis, then it will be added ps. Now go to the import and select it the browse to the step file (standard exchange programming file). Then click ok or open it, it will come on ps.

Later select the geometry on a tab, drag it and leave it over the geometry on b tab, it makes the sharing of the geometry in between the analysis system to another analysis system.

- Model imported
- Meshing properties
- Boundary conditions

Now select the parts and renames as shown in the figure below, and surrounding as air and turbine shaft and blades as material = al alloy. Go to the details and specify the reference temperature by body as 22 c. After that go the geometry and select the material assigned under the detail of geometry.

- Analysis settings

7.1.1. Modal Imported

- Go to ansys software and select import from top left corner and browser to step file
- After that go to fluent fluid flow, and select the meshing cell so that we can see the modal in fluid flow fluent.

Step 1: importing and modification of geometry

- Go to import option on top and select
- Then geometry will display
- Later, go fluent flow fluid on analysis system on left side and double over it
- After the fff, go to the geometry on geometry select it and drag it and over geometry on fff as shown in below.

Step 2: modification:

- Go to geometry, and right click over it and select designer module
- In design module on left we can see the tree outline, in tree outline select, import and click generate. After the geometry generate, go to tree outline and select and rename if necessary.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

- After the naming selection id complete adjust the geometry as per the fluent module by using Boolean operations on creations. As we can see in tree outline life side of the figure below

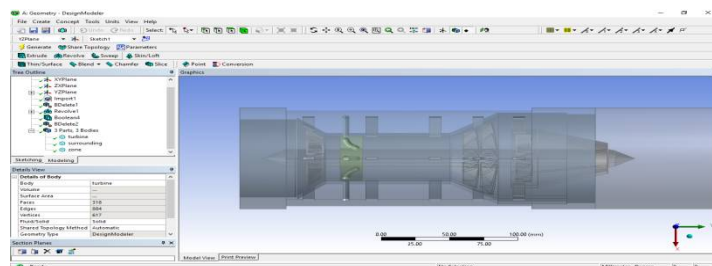


Figure 7.1 Modal In Fluid Flow Fluent

7.1.2. Meshing

Step 3: meshing

Go to ps (project schematic) and double click over the mesh and we will enter into meshing GUI (graphical user interface).

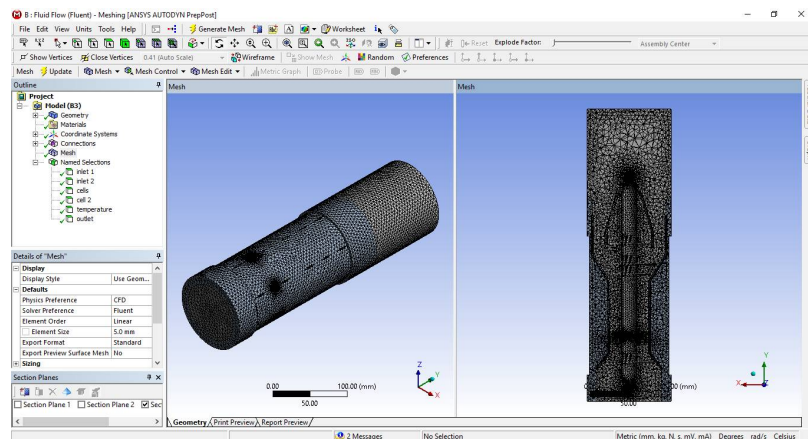


Figure 7.2 Modal In Fluid Flow Fluent

Now go to the meshing under the tree outline. Then specify the mesh size should be fewer than 32 k elements. After the mesh was generated, go to the selection, then the front and back of the turbine and right over it then select named selection. Under the named selection name it as outlet 1 and 2. Again go to the selection, and then the face of the fuel combustion chamber of the turbine with is located in middle of the turbine and right over it then select named selection. Under the named selected it as “temperature”. Again go to the selection, and then the mid faces or turbine of the turbine solid and right over it then select named selection. Under the name it as “turbine walls” as shown in figure.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

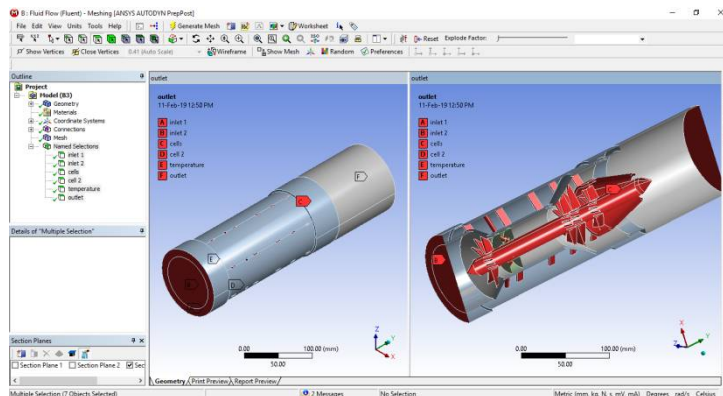


Figure 7.3 After The Name Selecting Go To File And Select The Close Meshing.

7.2. Fluent Fluid Flow Process:

After the meshing, go to file and select meshing close. Then we will come back to the project schematic. In project schematics under the fluid flow fluent double click on the setup. Then we get fluent launcher (setting editing only). Then select parallel under it –click ok

Step 1: under the fluent flow fluent go to the tree and select or double click on the general. In middle general properties will be modify and select the steady. Then click on unit to change the temperature k to c. Then click ok.

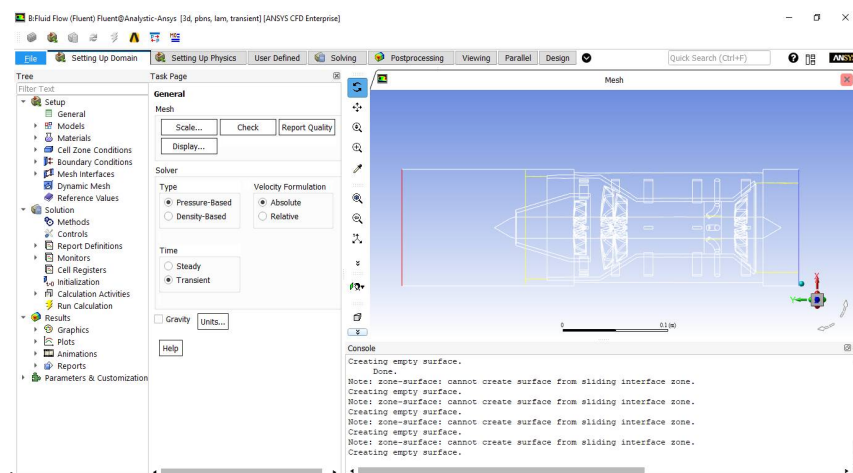


Figure 7.4 General Setting Under Fff Analysis

Step 2: Model Type

After the general setting, double click on the models. In model settings, make energy –on, so that temperature can be applied over the combustion chamber. After that, double click on the viscous model. In viscous model go to the model and select the k-epsilon (2 equations.). In k- epsilon model, select the rng. In rng, select swirls dominated flow and under near wall treatment select scalable wall function. Then select ok to confirm.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

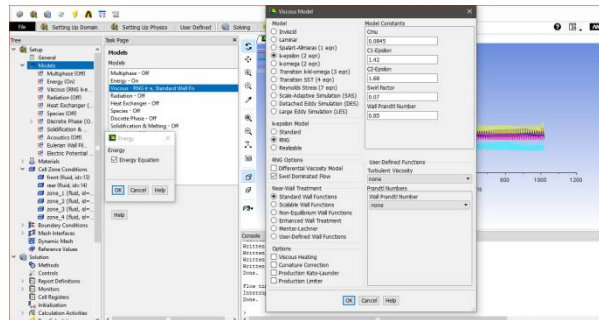


Figure 7.5 Model- Energy And Viscous

Step 3: Material Selection.

After that, model setting. Double click on the materials, under the material task page, double click on the fluid. In fluid, select create / edit, select the order material by name and select the fluent data base. In fluid database materials, select the air as a material for the domain, and at combustion chamber the diesel vapours been selected. And similarly select material under the create or edit, change the fluid to solid and select the material as aluminum alloy.

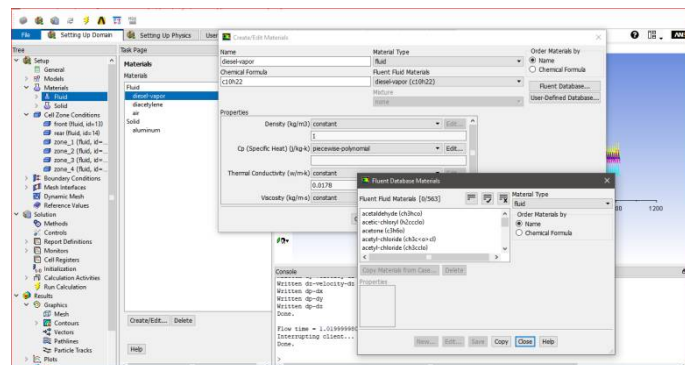


Figure 7.6 material – Fluid And Solid.

Step 4. Cell Zone Conditions

It is the type of the setting which specific the body and its material type. Go to the tree outline and double click on the cell zone conditions, then specify the zone and as a fluid- and select material or fluid as air and turbine as solid and select material as aluminum alloy as shown in figure. For the zone rotation go to cell zone condition and select the front turbine and rear turbine and specify the mesh motion as y axis rotation axis direction and speed as 360 degree per sec for zone1 of turbine and zone 2 and zone 4 720 degree per sec.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

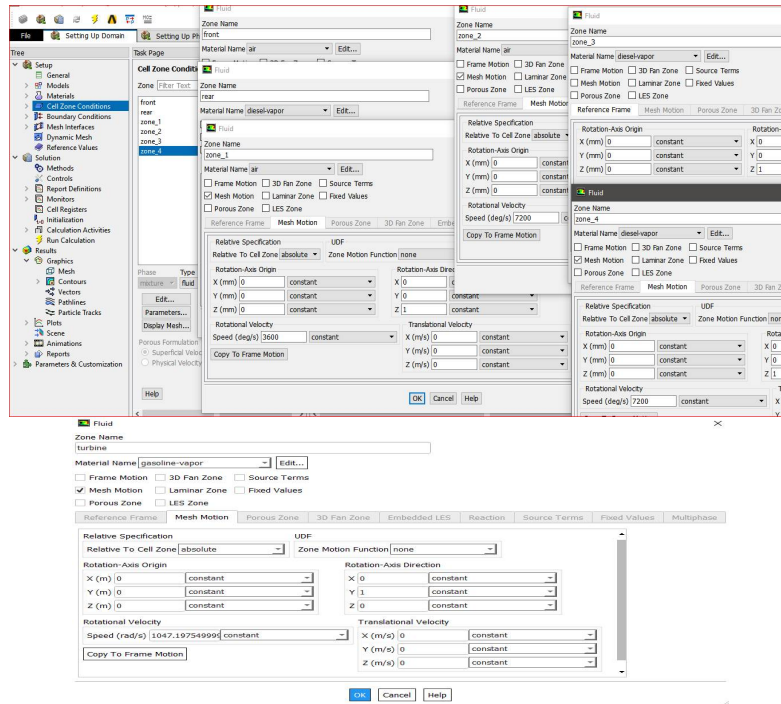


Figure 7.7fff- Cell Zone Conditions.

Step 5: Mesh Interface.

The interface helps to transfer the temperature between the body and liquid as same as contact region in steady thermal analysis. Go to mesh interface and double click over it. Under the mesh interface – select edit- and select interface 1 and 2 is coupled wall then select ok to confirm.

Step 6: Boundary Conditions.

Now go to the boundary condition and double click over it. In boundary condition select the outlet 3 and specify the boundary conditions are pressure is default and the temperature out must be 800 C and select temperature (thermal) of the turbulent at combustion will be 1500 C

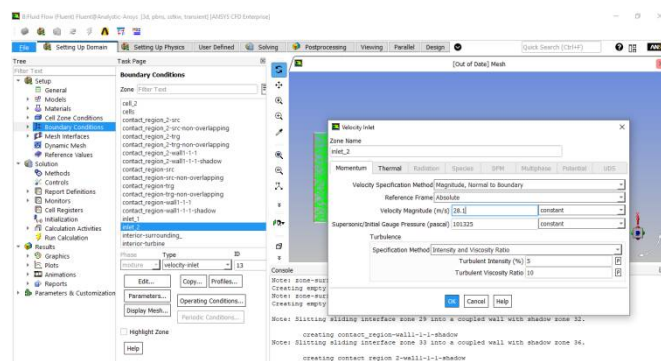


Figure 7.8 Boundary Condition – Velocity.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

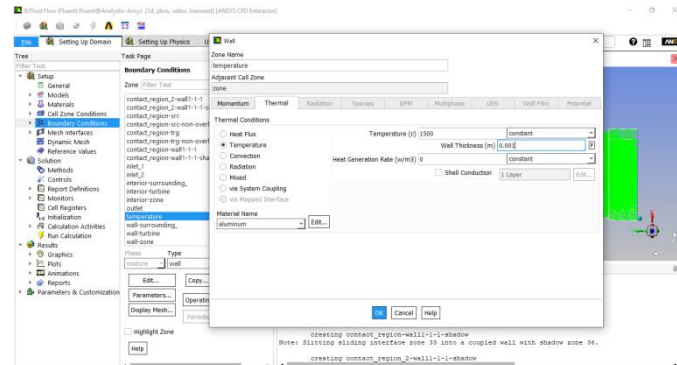


Figure 7.9 Boundary Condition – temperature.

Step 7: Solution- Calculation Activities.

Go to tree outline double on the calculation activities under solution. In calculation activities select automatic export- create-file type – cfd-post –select all quantities then click ok.

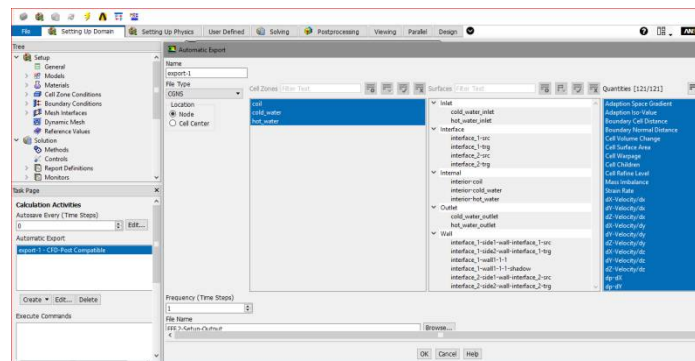


Figure 7.10 Calculation Activities

Step 8: Results Calculation

Double click on the run calculation- then specify the no of iterations as 100 and time size step 0.01= 1 sec

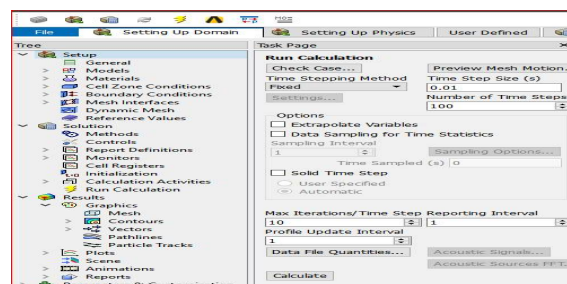


Figure 7.11 Run Calculation

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

Go to the run calculation and double click over it. Then click on calculation to run simulation process. The process will complete upon 100 iterations at time step 0.01.

7.3. Results

Go to results and double click on the contours. In contours, select pressure in contour of, then select the entire surfaces in surfaces then click ok

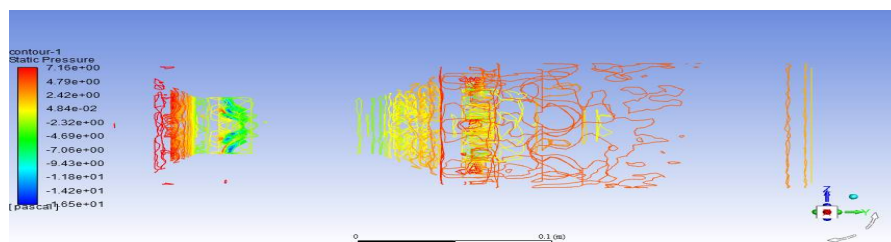


Figure 7.12 Contours- Pressure

Go to results and double click on the contours. In contours, select velocity in contour of, then select the entire surfaces in surfaces then click ok.

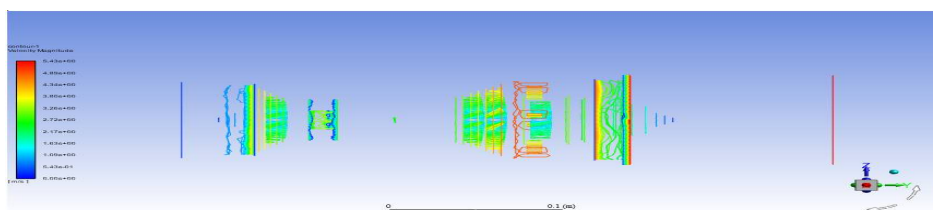


Figure 7.13 Contours – Velocity

Go to results and double click on the contours. In contours, select temperature in contour of, then select the entire surfaces in surfaces then click ok.

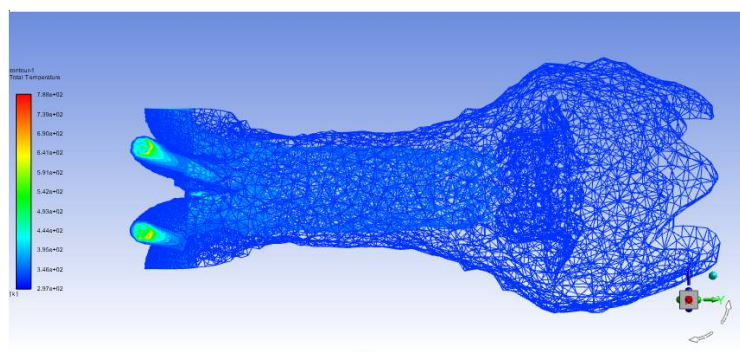


Figure 7.14 Contours – temperature

Go to results and double click on the contours. In contours, select turbulence in contour of, then select the entire surfaces in surfaces then click ok.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

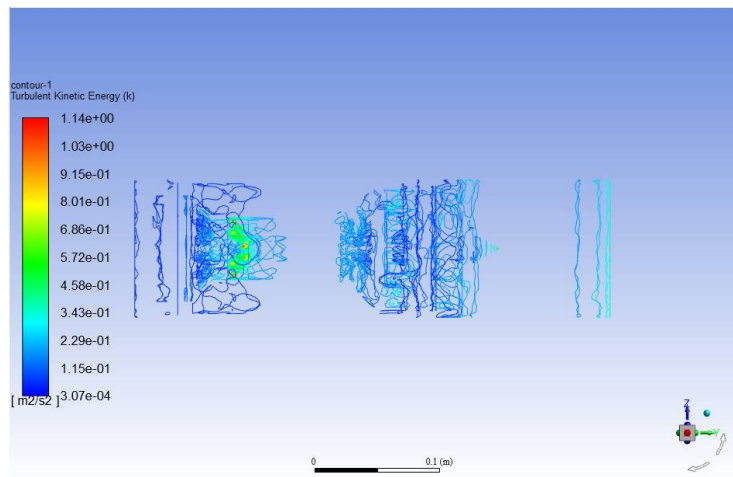


Figure 7.15Contours – turbulence

Go to results and double click on the vector. In vector, select temperature in vector of, then select the entire surfaces in surfaces then click ok.

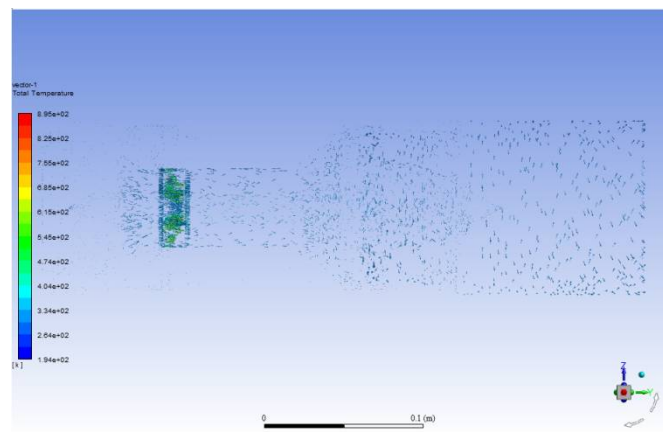


Figure 7.16vector – temperature

Results

After, the results in fluent fluid flow will be done. Go to file and select close fluent simulation

In ansys workbench to go the results cell, and double click over it. Then we will enter into the results cfd post processor. On top of the window we can see the select box to select streamline, vector, plane location etc.

Now go to file load and select the .cdat file from the heat exchange file- file- fff-dp0- fluent-fff- 100.cdat file then click ok.

- In resultant on flow fluid go to user location and plots. Right click on it and select plan and specify the details of plane and select xy plane
- Now go user location and plots and right click on it select contour and select xy pane as references so we can see the layout of temperature different in shell fluid and tube fluid.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

- Similarly select vector plots also.
Go to streamline- and specify the start from the cold water outlet and specify the max no of particles 1000. Again go to streamline- and specify the start from the hot water outlet and specify the max no of particles 1000. Then we can see the steam lines in the view in graphical windows.

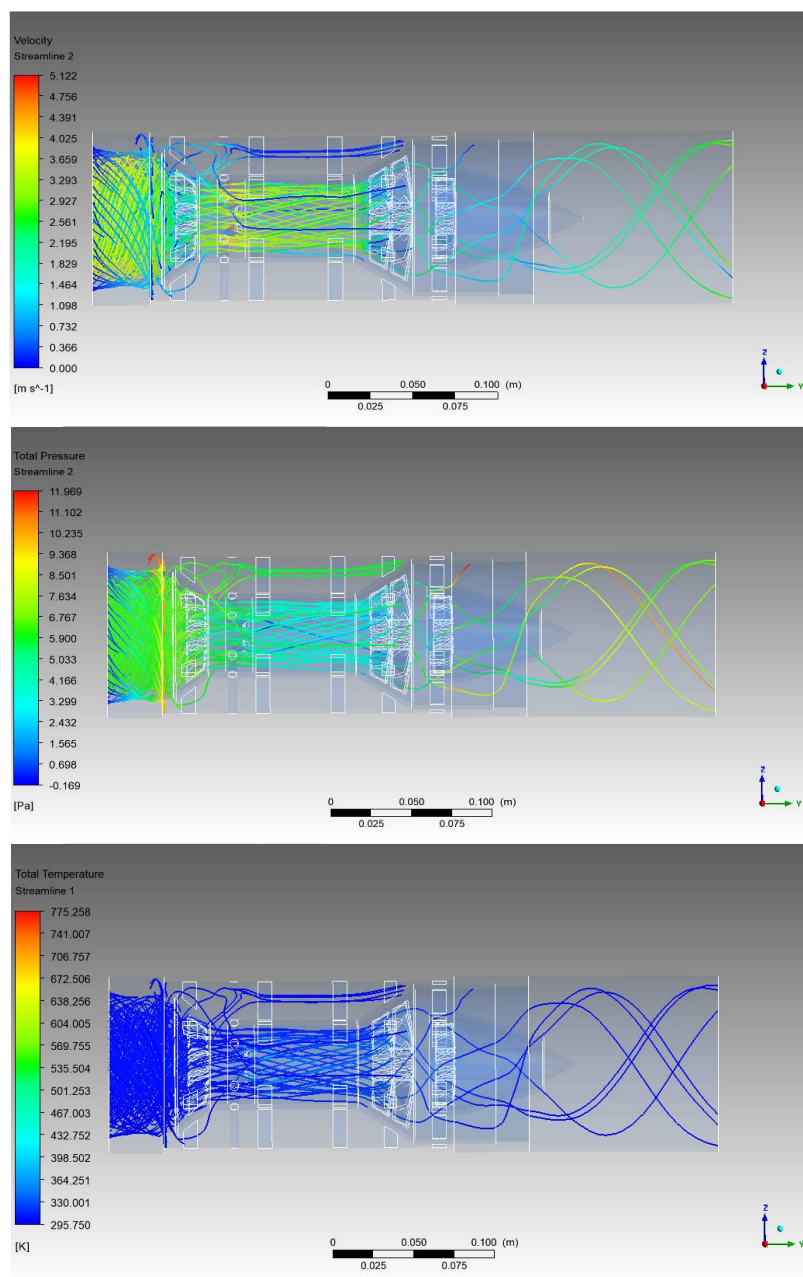


Figure 7.17 Stream Line- Flow Streamline.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

Now go to the location, and select the plane- then the select the xy plane and then click ok. Again, go to the location, and select the plane- then the select the zx plane and then click ok. Again, go to the location, and select the plane- then the select the yz plane and offset 10 mm then click ok. Again, go to the location, and select the plane- then the select the yz plane and offset -10 mm and then click ok

Go to contours- and location then selects the plane 1 as created at the above. Similarly go to contours- and locations then select the plane 1 as created at the above. Similarly go to contours- and locations then select the plane 1 as created at the above. Then specify the pressure, velocity as the type of the variation, so that we can see the change of the pressure, velocity.

Pressure:

Go to user location and plot- right click over it and select insert and select contour results. In contour results select location as above planes and variable as pressure then click apply.

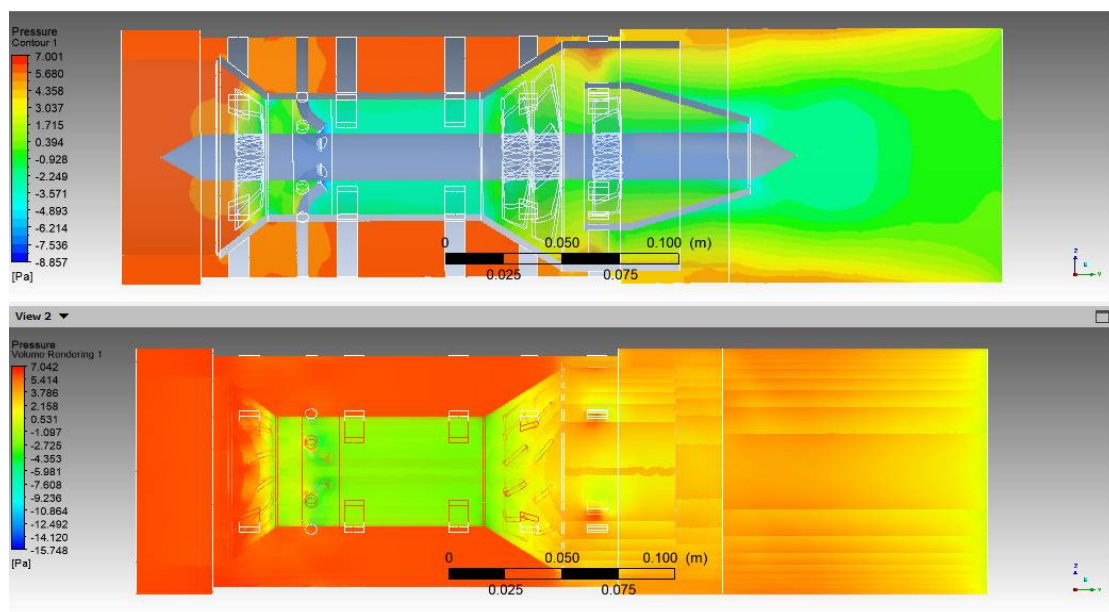


Figure 7.18 Contour and volume rendering- pressure.

Velocity:

Go to user location and plot- right click over it and select insert and select contour results. In contour results select location as above planes and variable as velocity then click apply.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

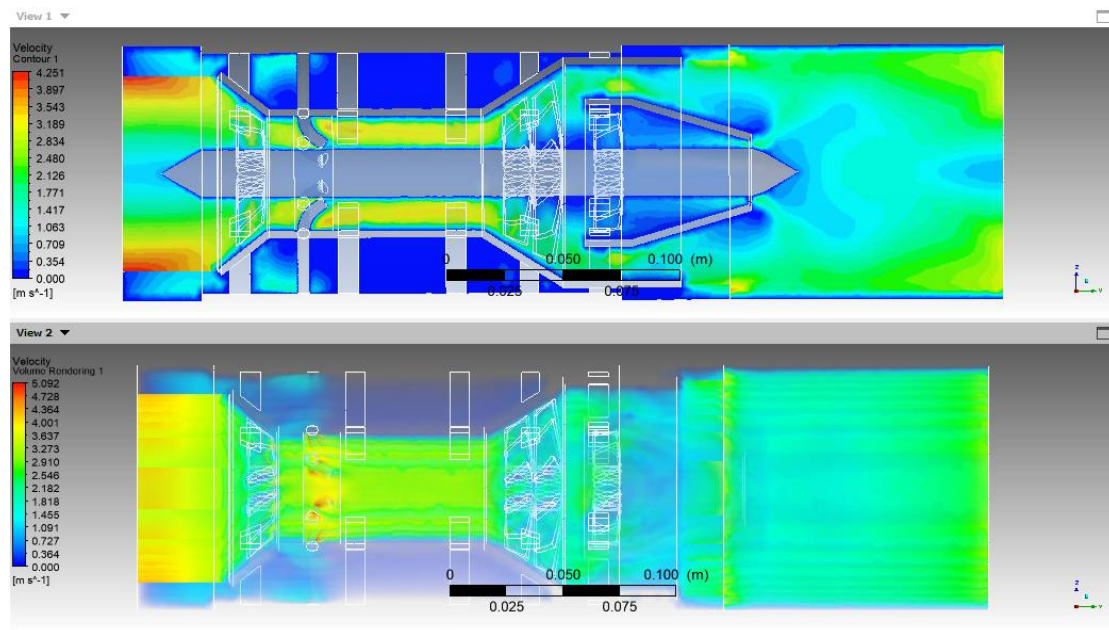


Figure 7.19 Contour and volume rendering - velocity.

Temperature:

Go to user location and plot- right click over it and select insert and select contour results. In contour results select location as above planes and variable as temperature then click apply.

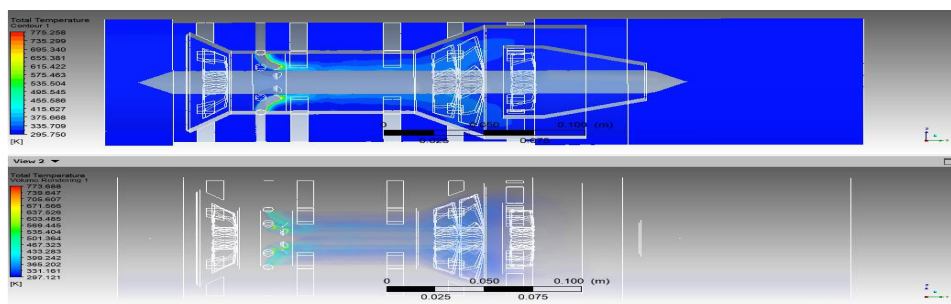


Figure 7.20 Contour and volume rendering - temperature.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

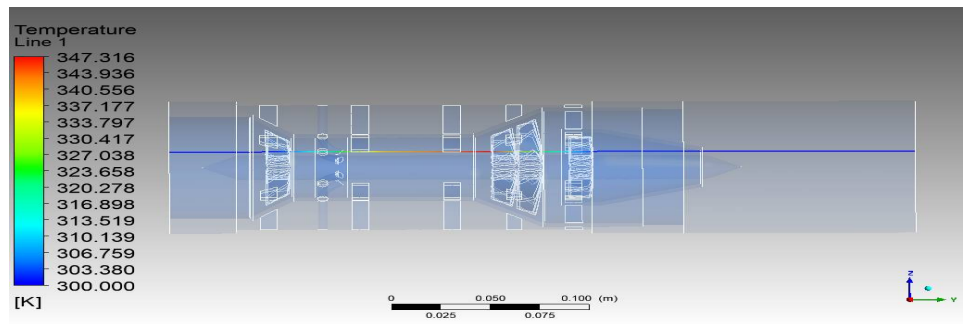
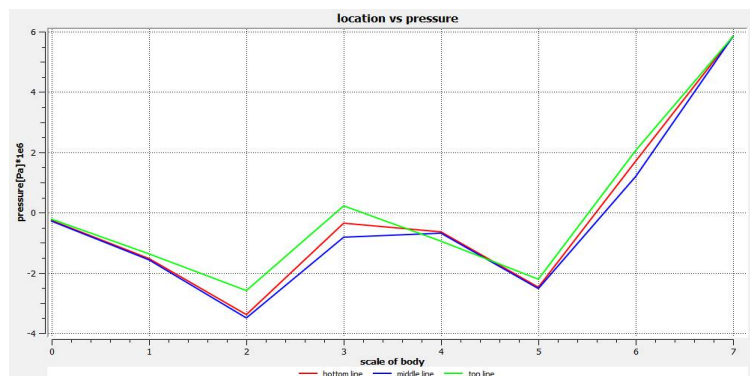


Figure 7.21 Contour path - temperature.

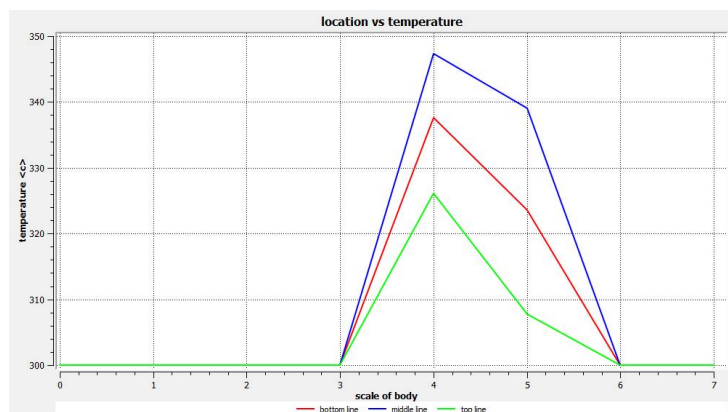
Go to the results on cfd results- and select chart, then select histogram in options. In the chart, select the pressure at the type of the charts.



- Pressure is increase as per the flow from inlet to outlet by streamline path

Chart – Absolute Pressure Mpa

Go to the results on cfd results- and select chart, then select histogram in options. In the chart, select the temperature at the type of the charts.



International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

Chart – Absolute temperature K

- **temperature is changing as per the flow from inlet to outlet by streamline path**

Go to the results on cfd results- and select chart, then select histogram in options. In the chart, select the velocity at the type of the charts.

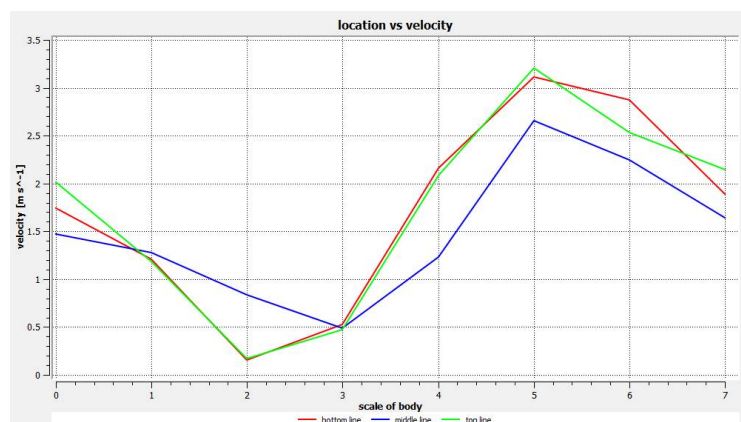


Chart – Absolute Velocity

- **velocity is changing as per the flow from inlet to outlet by streamline path**

VIII. TRANSIENT THERMAL ANALYSIS

After completing of this chapter, we can able to

Understand the types of thermal analysis

- Understand the mechanical interface used for thermal analysis.
- Understand different terms used in thermal analysis.
- Perform steady-state thermal analysis.
- Perform transient thermal "analysis."
- Understand temperature distribution.
- Run problem

Before a model is set for production, it passes through several stages. Thermal analysis is one of them and plays an important role in product development. Various products such as engines, refrigerators, heat exchangers, and so on are designed based on the results of this analysis. Thermal analysis is used to determine the temperature distribution and related thermal quantities in the model. In this analysis, all heat transfer modes, namely conduction, radiation, and radiation are analyzed. The output from a thermal analysis can be the following:

- Temperature distribution.
- Amount of heat loss or gain.
- Thermal gradients.
- Thermal fluxes.

This analysis is used in many engineering industries such as automobile, piping, electronic, power generation, and so on. In Ansys workbench, two types of thermal analysis can be carried out, namely steady-state and transient thermal analysis.

The following are the basic steps required to perform the thermal analysis:

- Set the analysis preference.
- Create or import solid model.
- Define element attributes (element types, real constants, and material properties).
- Mesh the solid model.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

- Specify the analysis type, analysis options, and the loads to be applied.
- Solve the analysis problem.
- Post-process results.

8.1. Types Of Thermal Analysis

In Ansys workbench, two types of thermal analysis can be carried out, namely steady-state thermal analysis and transient thermal analysis.

8.1.1. Steady-State Thermal Analysis

In the steady-state thermal analysis, the thermal load does not vary with time and remains constant throughout the period of application. This analysis considers only steady loads and does not consider any thermal load that varies with time. In the steady-state thermal analysis, the system is studied under steady thermal loads with respect to time. These thermal loads include radiation, radiation, heat flow rates, heat fluxes (heat flow per unit area), heat generation rates (heat flow per unit volume), and constant temperature boundaries. The steady-state thermal analysis may be either linear or nonlinear, with respect to material properties that depend on temperature. The thermal properties of most of the materials do vary with temperature, therefore the analysis usually is nonlinear. Including radiation effects or temperature dependent radiation in a model also makes the analysis nonlinear. The steps to solve a problem related to the thermal analysis are the same as that of the structural analysis, except a few steps such as selecting the element type, applying the load, and post processing results.

8.1.2. Transient Thermal Analysis

Transient thermal analysis, the application of thermal loads is time dependent. Most of the engineering applications need transient thermal analysis, such as engine blocks, pressure vessels, nozzles, piping systems, and so on. The process of solving the transient thermal analysis problem is the same as that of the steady-state thermal analysis. The only difference between these two analyses is that in transient thermal analysis, the load applied on a body is the function of time. In the transient thermal analysis, the system is studied under varying thermal loads with respect to time. You can get the temperatures varying with time, thermal gradients, and thermal fluxes in a transient thermal analysis. The transient thermal analysis takes more time compared to other analyses types. It is necessary to understand the basic mechanism of the problem to reduce the time involved in getting its solution. For example, if the problems contain nonlinearity, then you first need to understand how they affect the response of structures by doing the steady-state thermal analysis.

8.2. Project Objective

In this project we will open the existing above project. Next, your project with a different name and run the steady-state thermal analysis on the model shown in figure to find out the effect of temperature on the whole body of the model. Also, we will evaluate the total heat flux and the directional heat flux with respect to the direction heat flux with respect to axis. The parameters required to run the analysis are given next.

As discussed in previous chapters, analysis can be carried out in three major steps: pre-processing, solution, and post-processing. The tools required to carry out these steps are discussed next.

8.1.1. Pre-Processing

The pre-processing of an analysis system involves specifying the material, generating a mesh, and defining boundary conditions.

In Ansys workbench, the various tools related to boundary conditions are available in the environment contextual toolbar, which is displayed when you select the static structural node in the tree outline

In order to provide a support to the model, you need to choose the required tool from the supports drop-down. Similarly, to add a load, choose the desired tool from the loads drop-down in the environment contextual toolbar. Also, when you choose any tool from the environment contextual toolbar; the corresponding entity is placed under the static structural node in the tree outline.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

The main purpose of an analysis is to evaluate the results. After the boundary condition is set and loads are applied, you need to specify the desired outcomes of the analysis. In Ansys workbench, you can analyze various parameters such as deformation, stresses, strains, and so on. To do so, you need to specify the results required and then evaluate them. You can use the tools available in the solution contextual toolbar to specify results, refer to figure 9-4. Alternatively, right-click on the solution node in the tree outline and then use the desired option from the shortcut menu displayed.

In order to evaluate deformations, stresses, strains, and so on, choose the desired options from the drop-downs available in the solution contextual toolbar.

8.1.2. Solution

In an analysis, after pre-processing (meshing, specifying material, and specifying boundary condition) is done, the next step is to solve the analysis. In Ansys workbench, you will use the solve tool from the standard toolbar to run the solver. The solver runs in the background of a software and acquires results of an analysis, based on the specified boundary conditions.

8.1.3. Post-Processing

After the analysis is complete, you need to generate the report in the mechanical window. To do so, choose the report preview tab from the bottom of the graphics screen; the Ansys report generation in progress message is displayed on the screen. After sometime, this message vanishes and the report is generated.

8.2.1. Boundary Conditions:

Apply the temperature load of 1500 °C on the front flat face of the model as shown in figure. Apply the radiation load on the inner faces of the model, as shown in figure. The inner faces are exposed to air. The ambient temperature is 800 °C.

Material: structural steel (default)

The following steps are required to complete this tutorial

1. Open an existing project and save it with a different name
2. Change the unit and generate the mesh.
3. Apply thermal boundary conditions
4. Analyze the results.
5. Save the project and exit Ansys workbench.

8.2.2. Applying Thermal Boundary Conditions

After the model is meshed, you now need to apply the boundary condition for the analysis.

1. Select the transient thermal analysis node in the tree outline to display the environment contextual toolbar.
2. From the environment contextual toolbar, choose the temperature tool; temperature is added under the steady-state thermal node. Also, the details of “temperature” window is displayed.
3. In the details of “temperature” window, select the geometry selection box to display the apply and cancel
4. Select the front face of the model
5. Choose the apply button from the geometry selection box in the details of “temperature” window; the face is selected for applying the temperature boundary condition.
6. In the details of “temperature” window, specify tabular data in the magnitude edit box.
7. Select temperature from the steady-state thermal node in the tree outline; the graph and tabular data windows are displayed, as shown in figures.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

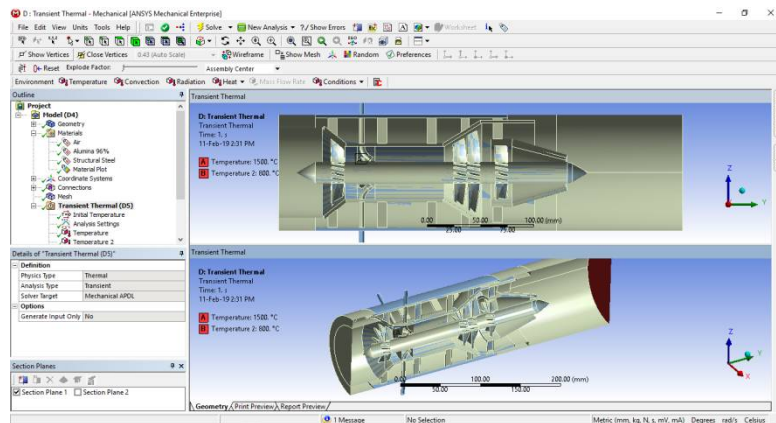


Figure 8.1 Temperature And Radiation The Steady-State Thermal

8.3. Analyzing The Results

After the thermal boundary conditions are applied, it is now important to analyze the behavior of the model with respect to the boundary conditions applied.

1. Right-click on the solution node in the tree outline to display a shortcut menu.
2. From this shortcut menu, choose insert > thermal > temperature; temperature is added under the solution node in the tree outline. Also, the details of “temperature” window is displayed.
3. In this window, select the geometry selection box to display the apply and cancel buttons.
4. Choose the body tool from the select toolbar and then select the model, as shown in figure.
5. Choose the apply button from the geometry selection box in the details of “temperature” window; 1 body is displayed in the geometry selection box.
6. Choose thermal > temperature from the solution contextual toolbar; temperature 2 is added under the solution node in the tree outline. Also, the details of “temperature 2” window is displayed.
7. In the details of “temperature 2” window, select the geometry selection box to display the apply and cancel buttons.
8. Choose the face tool from the select toolbar and then select the cylindrical face of the model
9. Choose the apply button from the details of temperature 2 window, 1 face is displayed in the geometry selection box in the details of temperature 2 window.
10. Choose the solve tool from the standard toolbar; the Ansys workbench solution status message box is displayed and the temperature distribution for the temperature 2 boundary condition is displayed in the graphics screen.
11. Select temperature under the solution node in the tree outline; temperature distribution in the model is displayed in the graphics screen, as shown in figure.
12. Similarly, select temperature 2 under the solution node in the ‘tree outline; the temperature distribution in the model is displayed in the graphics screen,
13. Select the solution node in the tree outline to display the solution contextual toolbar.
14. Choose total temperature from the thermal drop-down in the solution contextual toolbar;

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

8.3.1. For Aluminum Alloy

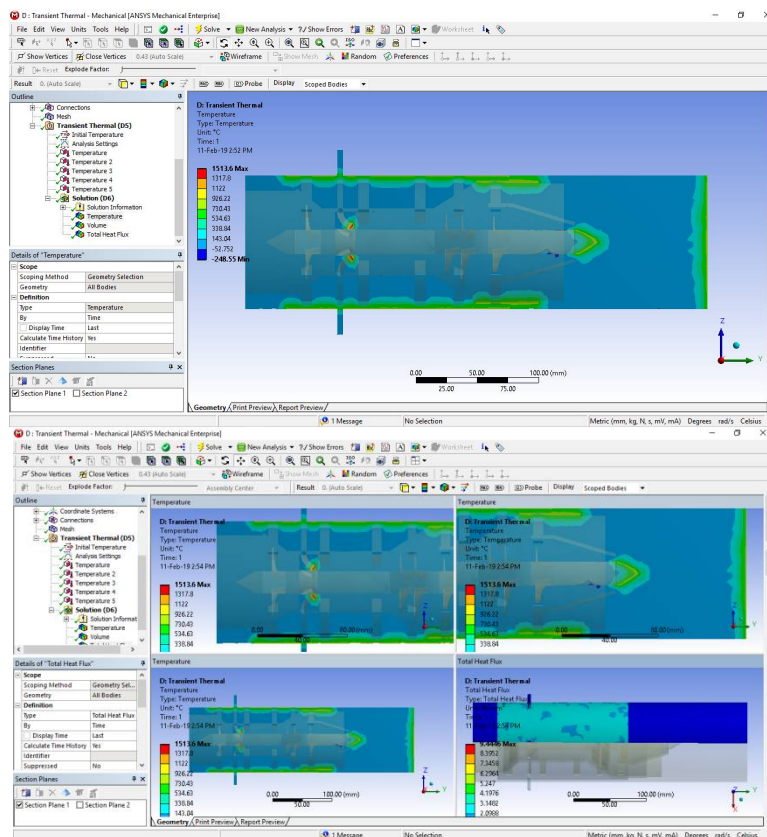


Figure 8.2 Total Temperature Thunderbolt Is Added Under Solution Node

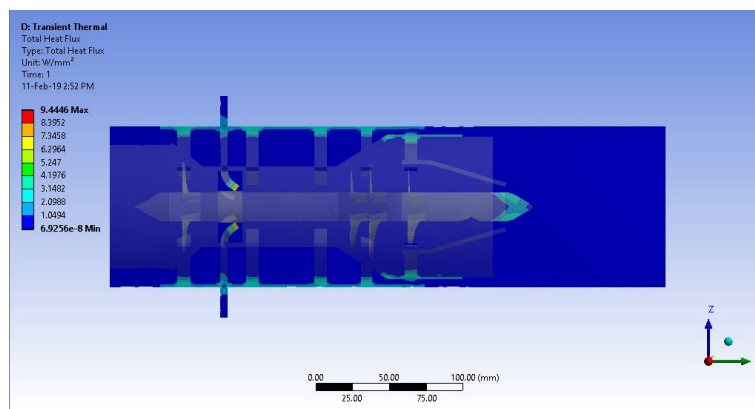


Figure 8.3 Total Heat Flux Thunderbolt Is Added Under Solution

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

15. In the details of “total heat flux” window, select the geometry selection box to display the apply and cancel buttons.
 16. Choose the body tool from the select toolbar and then select the model,
 17. Choose the apply button from the geometry selection box in the details of “total heat flux” window; 1 body is displayed in the geometry selection box.
 18. Similarly, choose heat flux from the thermal fly out in the solution contextual toolbar;
- It is obvious from the above results that the total heat flux is maximum in the region where the model is marked red and minimum where it is marked blue. Each color contour depicts a value and can be seen in the legend in the graphics screen.
- Exit the mechanical window to display the workbench window.

IX. CONCLUSION

For cfd process, we requires a geometry structure was design in catia software. Later the geometry is modified as per fluent flow fluid in ansys which consists of the rotational zone and turbine chamber as shown in above figure. Modal in fluent flow fluid (fff).

After that complex geometry will be mesh to find the no of element and nodes as shown in figure. in mesh step2. While come to the fff, we used the modal as k epsilon- RNG standard function and boundary conditions a flow inlet as 2m/s. with the ambient temperature 22 and temperature of the combustion 1500 C

On scenario, we observer that mpa pressure, m/s velocity and $(m/s)^2$ turbulence and temperature k effect because of turbine zone rotation at speed of 100m/s. for time interval of 0.01 time steps and 100 time period means 1 second so that the I can define the temperature variation of combustion chamber

After the simulation is completed the pressure in the zone 1,2 and 4 is increasing from default pressure to high as we can see from above figure. Similary the velocity of inlet or outlet is less when it is compare to the pressure around the turbine (fan). Where it comes to the temperature, the temperature inside the turbine is increasing due to the pressure, velocity.

REFERENCES

1. Flack, Ronald D. (2005). "Chapter 9: Combustors and Afterburners". Fundamentals of Jet Propulsion with Applications. Cambridge Aerospace Series. New York, NY: Cambridge University Press. ISBN 978-0-521-81983-1.
2. Henderson, Robert E.; Blazowski, William S. (1989). "Chapter 2: Turbopropulsion Combustion Technology". In Oates, Gordon C. Aircraft Propulsion Systems Technology and Design. AIAA Education Series. Washington, DC: American Institute of Aeronautics and Astronautics. ISBN 0-930403-24-X.
3. Mattingly, Jack D.; Heiser, William H.; Pratt, David T. (2002). "Chapter 9: Engine Component Design: Combustion Systems". Aircraft Engine Design. AIAA Education Series (2nd ed.). Reston, VA: American Institute of Aeronautics and Astronautics. ISBN 1-56347-538-3.
4. Mattingly, Jack D. (2006). "Chapter 10: Inlets, Nozzles, and Combustion Systems". Elements of Propulsion: Gas Turbines and Rockets. AIAA Education Series. Reston, VA: American Institute of Aeronautics and Astronautics. ISBN 1-56347-779-3.