FLUID THERMAL MODELLING AND SIMULATION LABORATORY

LAB MANUAL

Course Code	AME113
Regulations	IARE - R18
Class	B.Tech VI Semester
Branch	MECH
Academic Year	2019-2020

Prepared By

Ms.NSanthiSree

Assistant Professor

Department of Mechanical Engineering

INSTITUTE OF AERONAUTICAL ENGINEERING

(Autonomous)

Dundigal- Hyderabad - 500 043

INSTITUTE OF AERONAUTICAL ENGINEERING
(Autonomous)
Dundigal, Hyderabad - 500 043

Department of Mechanical Engineering

Vision
The Department of Mechanical Engineering envisions value based education, research and development in the areas of Manufacturing and Computer Aided Engineering as an advanced centre for Mechanical Engineering, producing graduates of world-class competence to face the challenges of global market with confidence, creating effective interface with various organizations.

Mission
The mission of the Mechanical Engineering Department is to prepare effective and responsible engineers for global requirements by providing quality education & to improve pedagogical methods employed in delivering the academic programs to the needs of the industry and changing world by conducting basic and applied research and to generate intellectual property.

INSTITUTE OF AERONAUTICAL ENGINEERING
(Autonomous)
Dundigal, Hyderabad - 500 043

Department of Mechanical Engineering

Program Outcomes	
PO1	**Engineering knowledge**: Apply the knowledge of mathematics, science, engineering fundamentals, and an engineering specialization to the solution of complex engineering problems.
PO2	**Problem analysis**: Identify, formulate, review research literature, and analyze complex engineering problems reaching substantiated conclusions using first principles of mathematics, natural sciences, and engineering sciences.
PO3	**Design/development of solutions**: Design solutions for complex engineering problems and design system components or processes that meet the specified needs with appropriate consideration for the public health and safety, and the cultural, societal, and environmental considerations.
PO4	**Conduct investigations of complex problems**: Use research-based knowledge and research methods including design of experiments, analysis and interpretation of data, and synthesis of the information to provide valid conclusions.
PO5	**Modern tool usage**: Create, select, and apply appropriate techniques, resources, and modern engineering and IT tools including prediction and modeling to complex engineering activities with an understanding of the limitations.
PO6	**The engineer and society**: Apply reasoning informed by the contextual knowledge to assesssocietal, health, safety, legal and cultural issues and the consequent responsibilities relevant to the professional engineering practice.
PO7	**Environment and sustainability**: Understand the impact of the professional engineering solutionsin societal and environmental contexts, and demonstrate the knowledge of, and need for sustainable development.
PO8	**Ethics**: Apply ethical principles and commit to professional ethics and responsibilities and norms of the engineering practice.
PO9	**Individual and team work**: Function effectively as an individual, and as a member or leader indiverse teams, and in multidisciplinary settings.
PO10	**Communication**: Communicate effectively on complex engineering activities with the engineering community and with society at large, such as, being able to comprehend and write effective reports and design documentation, make effective presentations, and give and receive clear instructions.
PO11	**Project management and finance**: Demonstrate knowledge and understanding of the engineering and management principles and apply these to one's own work, as a member and leader in a team, to manage projects and in multidisciplinary environments.
PO12	**Life-long learning**: Recognize the need for, and have the preparation and ability to engage in independent and life-long learning in the broadest context of technological change.
Program Specific Outcomes	
PSO1	**Professional Skills:** To produce engineering professional capable of synthesizing and analyzing mechanical systems including allied engineering streams.
PSO2	**Modelling and Simulation Practices:** An ability to adopt and integrate current technologies in the design and manufacturing domain to enhance the employability.
PSO3	**Successful Career and Entrepreneurship:** To build the nation, by imparting technological inputs and managerial skills to become Technocrats.

ATTAINMENT OF PROGRAM OUTCOMES & PROGRAM SPECIFIC OUTCOMES		
Expt. No.	ME	
	Program Outcomes Attained	Program Specific Outcomes Attained
I	PO1,PO3	PSO2
II	PO1, PO3, PO4	PSO2
III	PO2	PSO2
IV	PO2, PO3	PSO2
V	PO1, PO3, PO4	PSO2
VI	PO1, PO3	PSO2
VII	PO1, PO3	PSO2
VIII	PO2, PO3, PO4	PSO2
IX	PO2	PSO2
X	PO3, PO4	PSO2
XI	PO3, PO4	PSO2
XII	PO3, PO4	PSO2

INSTITUTE OF AERONAUTICAL ENGINEERING

(AUTONOMOUS)

Dundigal, Hyderabad - 500 043

Certificate

This is to certify that it is a bonafied record of Practical work done by Sri/Kum. ______________________________________ bearing the Roll No. _______________________ Of ____________ class ____ branch in the ____________________________ laboratory during the academic year ___________________ under our supervision.

Head of the Department **Lecture In-Charge**

External Examiner **Internal Examiner**

LIST OF EXPERIMENTS

S. NO	NAME OF EXPERIMENT	PAGE NO
1	Internal pipe fluid flow – fem	
2	Internal pipe fluid flow – Ansys	
3	Internal pipe fluid flow-MATLAB	
4	External fluid flow	
5	Flow through ball valve	
6	Heat conduction	
7	Temperature distribution	
8	3d heat conduction	
9	Counter flow heat exchanger	
10	Conjugate heat transfer	
11	3d thermal analysis	
12	Thermal stress analysis	

EXPERIMENT-I

INTERNAL PIPE FLUID FLOW – FEM

AIM: To find Internal Pipe flow Using theoretical FEM.

Introduction

1.0 Computational Methods
- 1.1 Idealization
- 1.2 Discretization
- 1.3 Solution

2.0 The Finite Elements Method
- 2.1 FEM Notation
- 2.2 Element Types

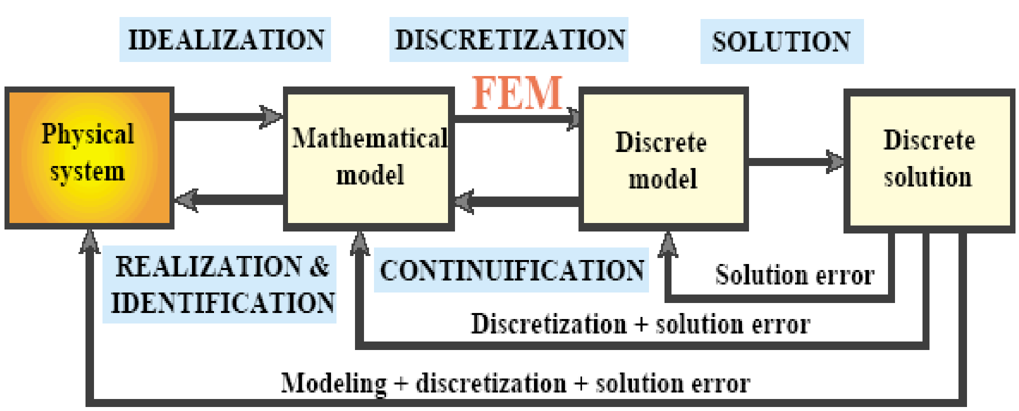
3.0 Mechanichal Approach
- 3.1 The Problem Setup
- 3.2 Strain Energy
- 3.3 External Energy
- 3.4 The Potential Energy Functional

Mathematical Formulation

The Mathematics Behind the Method

1. Weighted Residual Methods
2. Approxiamting Functions
3. 4he Residual
4. Galerkin's Method
5. The Weak Form

FLOW CHART



	Property [K]	Behavior {u}	Action {F}
Elastic	Stiffness	Displacement	Force
Thermal	Conductivity	Temperature	Heat Source
Fluid	Viscosity	Velocity	Body Force
Electrostatic	Dielectric permittivity	Electric Potential	Charge

Fluid Dynamics:

- Fluid flow is laminar at low velocities but turns turbulent as the velocity increases beyond the critical value. Transition from laminar to turbulent occurs within a range of velocity where the flow fluctuates between laminar and turbulent flows before becoming fully turbulent.
- The Reynolds Number (Re) defines whether the fluid is laminar, transitional or turbulent in the following way

$$Re < 2300 \text{ laminar flow}$$
$$2300 \leq Re \leq 10\,000 \text{ transitional flow}$$
$$Re > 10\,000 \text{ turbulent flow}$$

Steps:

- Defining the flow problem
- Building the mathematical model
- Discretization
- Iteration
- Simulation process

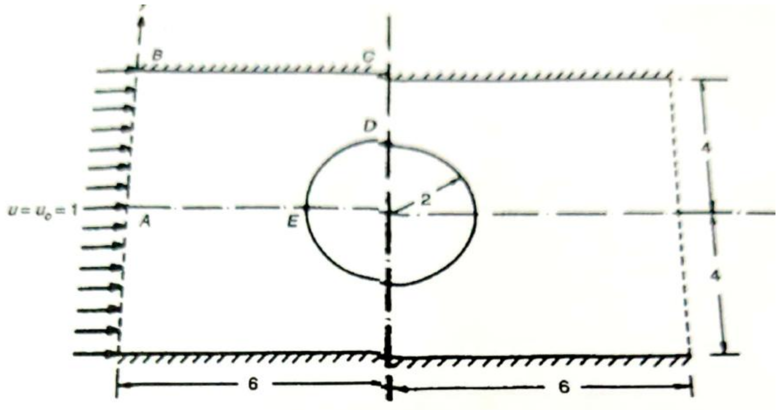
Governing Equation

Both fluid flow and temperature field can be described by the general equation that governs the two-dimensional problems. However, the coefficients of the governing equation differ from the fluid flow to the temperature field such that they satisfy the governing equations given in equations and for fluid flow and temperature flow, respectively. Since the flow analysis is characterized by the Poisson's expression; it is recalled here for convenience.

$$\frac{\partial^2 \Psi}{\partial x^2} + \frac{\partial^2 \Psi}{\partial y^2} = 0$$

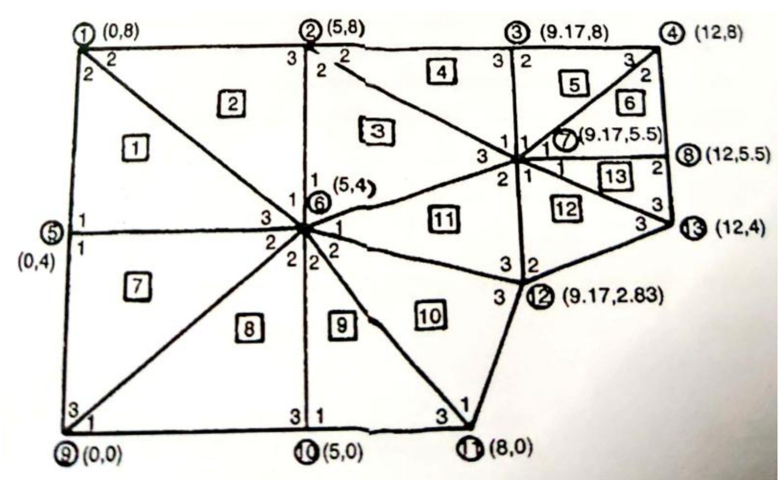
The boundary conditions for the fluid flow analysis are illustrated in Fig. The x-component of the velocity at the entrance and the exit is constant. ux is constant specific value for the velocity and y is the y-coordinate of the nodes in that boundary. Since the flow is non-viscous and incompressible, the x-component of the velocity remains constant at the upper and lower boundaries.

Problem Description:



(a) Confined flow around a cylinder

Descritization:



(b) Finite element idealizaion

OUTCOME:

Understand basic units of measurement, convertUnits, and appreciate their magnitudes. Understand the basic principles of FEMUtilize the governing equation for solving thefluid flow through the pipe and determine velocity .

ViVA Questions:

1. What is the equation in FEM .2.What is Reynolds number 3.What is the necessity of Discretization.

EXPERIMENT-2

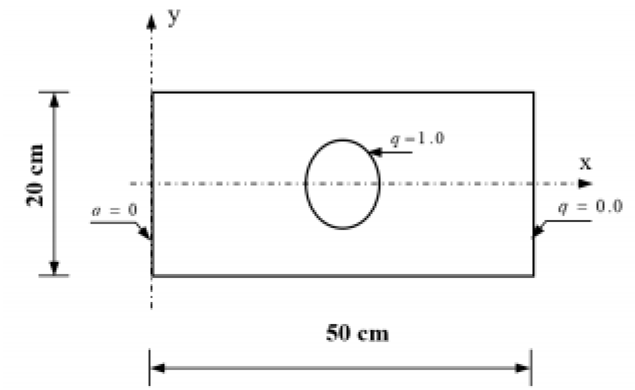
INTERNAL PIPE FLUID FLOW – ANSYS

AIM: To determine the internal pipe fluid flow using ANSYS.

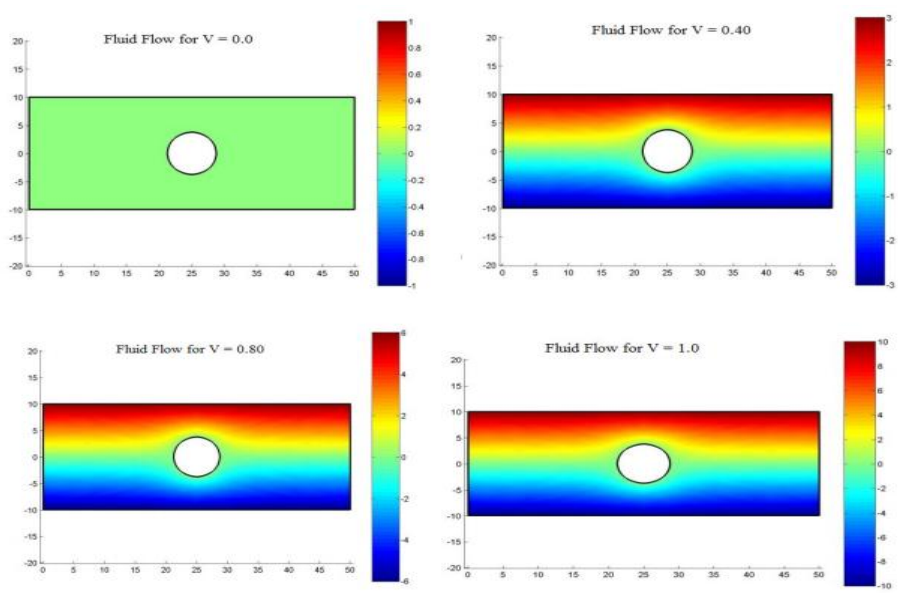
Boundary Conditions:

The boundary conditions for the temperature field analysis are illustrated in Fig.. The small pipe in the middle is assumed to produce q units of energy per surface area of the pipe per unit time. The energy q is considered equal to unity at the small pipe and equal to zero at exit, lower boundary and upper boundary. The temperature flow φ is considered equal to zero at the entrance. Once the boundary conditions and the coefficients are defined, the analysis of the temperature field is ascertained.

Problem Description:

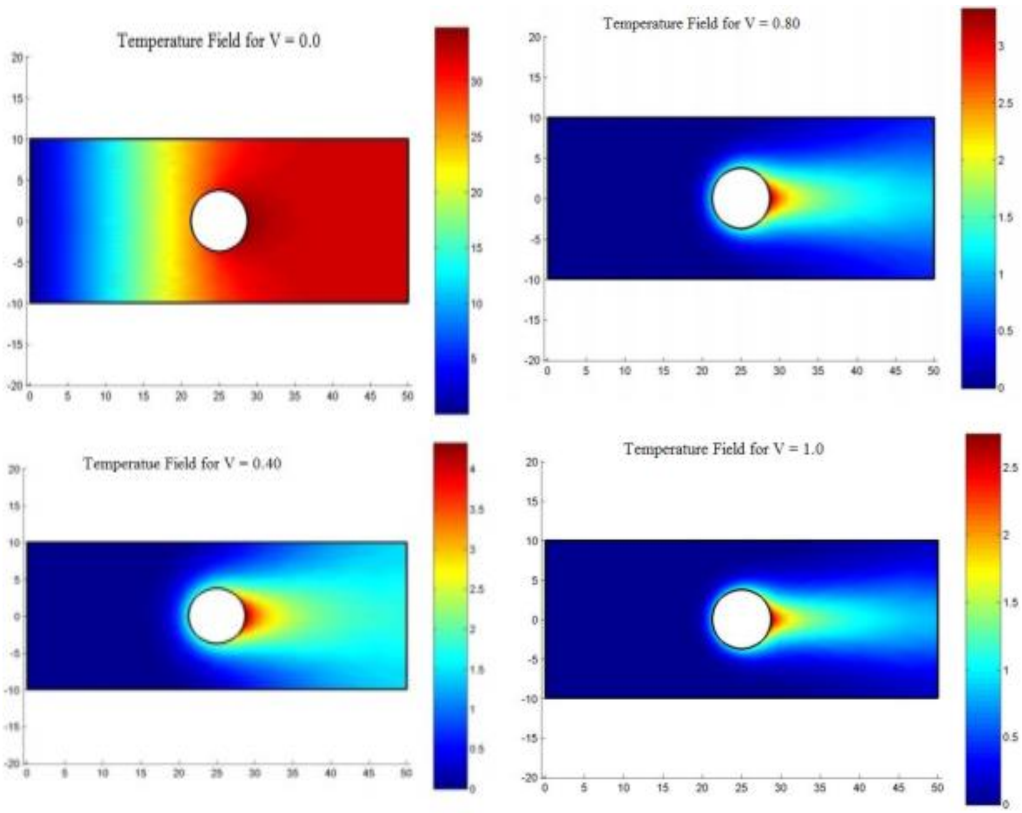


Fluid flow for different values of entrance velocity:



Temperature field for different values of entrance velocity:

In the present study, the flow is considered to be twodimensional with no difference in the span-wise direction. A partial differential equation is used to represent the flow domain. A series of numerical calculations have been conducted, and the results are presented in order to show the effects of fluid flow and temperature field on temperature distribution in the pipe channel. It should be noted that since the pipe size in the flow direction is relatively coarse, the local heat transfer is not as accurate or detailed as is the case of the x and y directions. However, the resolution is sufficient to aid in the design of pipe for industrial applications and also to provide information and insight into the fluid flow characteristics in the flow direction. In reality, since it is difficult to achieve an adiabatic boundary at the inlet and outlet of the pipe as assumed in the numerical model, a significant portion of the heat loss is transferred to the ambient environment, especially for low fluid flow conditions. Thus, when evaluating the heat transfer in pipe with low fluid flow rates, particular attention should be paid to the effects of this heat loss.



OUTCOME:

Understand basic units of measurement, convert Units, and appreciate their magnitudes. Understand the basic principles of FEM Utilize the governing equation for solving the fluid flow through the pipe.

VIVA Questions:

1. Why Ansys fluent is used.2.What is steady state.3.How do you define mesh quality?

EXPERIMENT- 3

INTERNAL PIPE FLUID FLOW – MATLAB

AIM:Internal Pipe flow problem using MAT LAB.

Objective of the project :

- To simulate for internal Laminar flow through the pipe and compute the velocity profiles & Shear stresses at various cross cestions .
- To Set up the sufficient length of the pipe so that the flow is fully developed (i.e) axial component of the velocity remains unchanged
- To correlate the velocity proifle obtained from the simulations with the Hagen poiseuille equation **(Equation for u(r))**

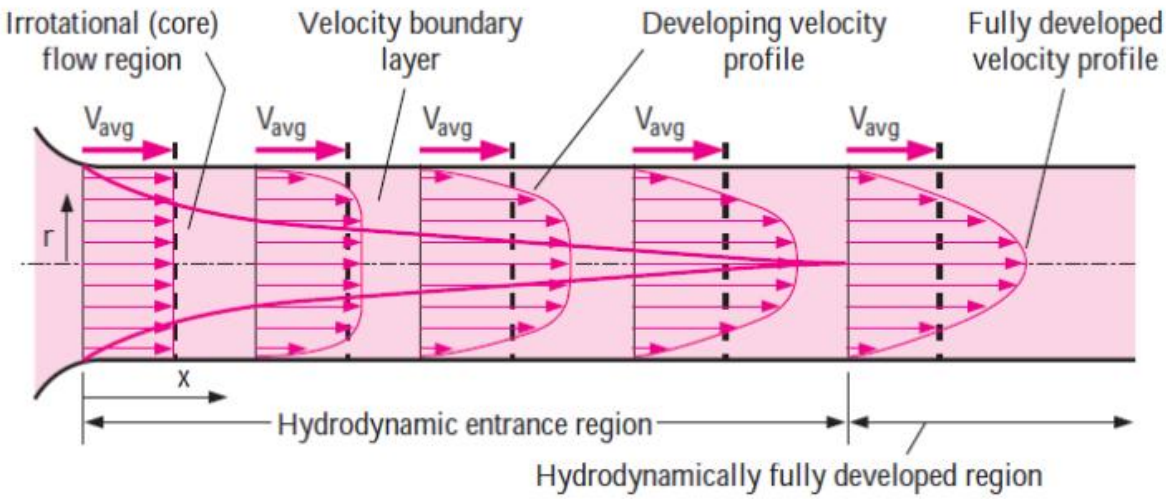
Assumptions :

- **Steady flow , incompressible and laminar flow**
- **Flow is two dimenional (Flow WRT theta is ignored)**
- **Constant fluid properties (Viscosity , Density etc)**

Introduction - Laminar flow through pipes :

As we all aware that the fluid in contact with the surface , it experience a momentum disturbances such fluid particles in the layer in contact with the surface of the pipe come to a complete stop. This layer also causes the fluid particles in the adjacent layers to slow down gradually as a result of friction. To make up for this velocity reduction, the velocity of the fluid at the midsection of the pipe has to increase to keep the mass flow rate through the

pipe constant. As a result, a velocity gradient develops along the pipe.The region of the flow in which the effects of the viscous shearing forcescaused by fluid viscosity are felt is called the **velocity boundary layer** or just the boundary layer. The hypothetical boundary surface divides the flow in a pipe into two regions: the **boundary layer** region, in which the viscous effects and the velocity changes are significant, and the **irrotational (core)** flow region, in which the frictional effects are negligible and the velocity remains essentially constant in the radial direction. The thickness of this boundary layer increases in the flow direction untilthe boundary layer reaches the pipe center and thus fills the entire pipe The region from the pipe inlet to the point at which the boundary layer merges at the centerline is called the hydrodynamic entrance region, and the length of this region is called the **hydrodynamic entry length** Lh. Flow in the entrance region is called hydrodynamically developing flow since this is the region where the velocity profile



It is also to be noted that the velpocity proifle for a fully devloped flow i case of a laminar flow will be parabolic as shown above whereas , for turbulent flows the profile will be fuller and also flatter

Modelling Inputs :

- Re = **2100** (essentially laminar)
- Working fluid water : Kinematic viscosity = **1e-06 m^2/s** (at 20 deg)
- Dynamic viscosity of the water : `mu = **0.89*10^-3** Pa.S`
- Density of water = **997** Kg/m^3
- Diameter of the pipe D = **0.02 m** (Assuming)
- The average velocity for the corresponding values of Re and D is found out from the expressions

$$V_{avg} = \frac{Re \cdot \mu}{\rho \cdot D} = 0.09373ms^{-1}$$

- The maximum velocity is computed from the expression

$$V_{\max} = 2 \cdot V_{avg} = 0.1875ms^{-1}$$

- The pressure drop across the pipe is computed from the Hagen poiseuille equations as follows

$$\Delta P = \frac{32\mu L V_{avg}}{D^2} = 18.2Pa$$

- Since openFoam deals with the kinematic quantites , the kinematic pressure drop is calculated as follows

$$\Delta P_{k \in ematic} = \frac{\Delta P}{\rho} = 0.018m$$

OUTCOME:

Mesh the 3D pipe flow domain using ANSYS Validate the results of analytical models introduced in lecture to the actual behavior of real fluid flows.

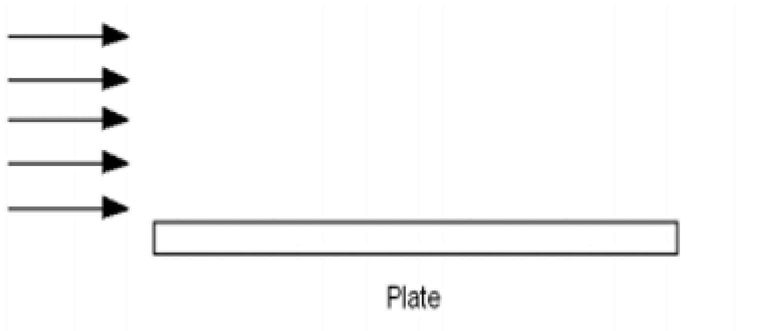
```
%PIPE_FLOW_1.M Plot Laminar and Turbulent Velocity Profiles in a pipe
%This illustrates function evaluation and plotting in Mat lab.
%The goal here is simply to evaluation and plot the velocity profiles for
% laminar and turbulent flow in a circular pipe geometry.
%For laminar flow, the velocity profile is parabolic, where u(r) is given
%by  u(r) = u max * [1 - (r / R)^2]
%For turbulent flow the velocity profile is given by
% u(r) = u max * [1 - (r / R)]^ m
%Where m = 1/7 for fully developed turbulent flow.
%In evaluating these functions, we will again illustrate Mat lab' s vector processing capabilities %and some
simple 2-D plotting function available in MATLAB.
%getting started
clc
 clear all, close all, nfig=0;
%
%define problem parameters
 Q1 = 64 .25  %volumetric flow rate
 Q2 = Q1 / 15850  ;                          %Convert Q to
 D = 2 * 0.0254                              %Pipe diameter (inches converted to meter)
 R = D / 2;                                  %Pipe radius (m)
 m = 1/7 ;                                   %exponent in turbulent flow equations
%
%Compute some derived quantities
A= pi*R^2;                                   %pipe flow area
uave= Q2/A;                                  %ave velocity
uLmax= 2*uave;                               %max velocity for turbulent flow
uTmax= (m+1)*(m+2)*uave/2;                          %max velocity for turbulent flow
%
%Evaluate the laminar and turbulent velocity profiles
Nr=101;            r=linspace (0, R, Nr);  %discrete r vector
uL=uLmax*(1-(r/R) ^2);  %laminar velocity profile
uT=uTmax*(1-(r/R)^m;    %turbulent velocity profile
%
%Now plot the results(both profiles on the same plot)
nfig=nfig+1; figure (nfig)
Plot (r/R,uL,' r-' , r/R, uT, ' g- -' , ' Line Width' , 2), grid
title (' Pipe\_Flow\_1: Laminar vs. Turbulent Velocity Profiles' );
xlabel(' Normalized Radial Position- (r/R)' ), ylabel (' Fluid Velocity (m/s)' )
legend(" Laminar Flow' ,' Turbulent Flow' )
%
%Finally, let' s print out a few summary results
fprintf(1,' \n   Summary Results for the Pipe Flow Problem \n\n' )
fprintf(1,'  Volumetric flow rate (gal/min):               %10.3f \n' , Q1)
fprintf(1,' Volumetric flow rate (m^3/s):                  %10.3e \n' , Q2)

fprintf(1,' Pipediameter (inches):              %10.3f \n' ,D/0.0254)
fprintf(1,' Average velocity (m/s):                                 %10.3f \n' , uave)
fprintf(1,' LAMINAR flow peak velocity (m/s):  %10.3f   \n' , uLmax)
fprintf(1,' TURBULENT flow peak velocity  (m/s):  %10.3f \n' , uTmax)
%
%end program
```

EXPERIMENT -4

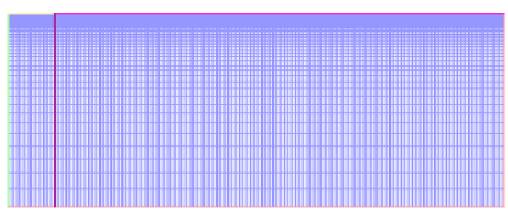
EXTERNAL FLUID FLOW

AIM: Determination of the skin friction coefficient of a rectangular plate when fluid is flowing over the surface of plate usingANSYS Flow Simulation.

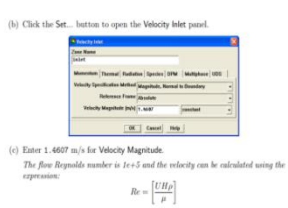
Problem description:a flat plate with a length L = 1 m (Figure 3.1). The flow of air is at a velocity of 1.4607 m/sec, such that the Reynolds number based on plate length is 1e+05. The plate is maintained at a temperature of 400 K.



MESH:



Physics of the Problem

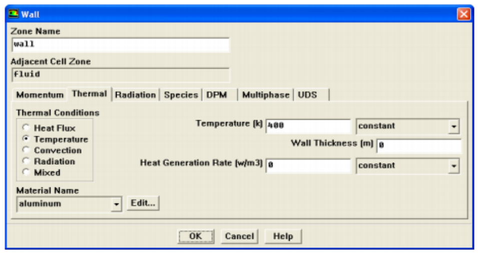


(b) Click the Set... button to open the Velocity Inlet panel.

(c) Enter 1.4607 m/s for Velocity Magnitude.

The flow Reynolds number is 1e+5 and the velocity can be calculated using the expression:

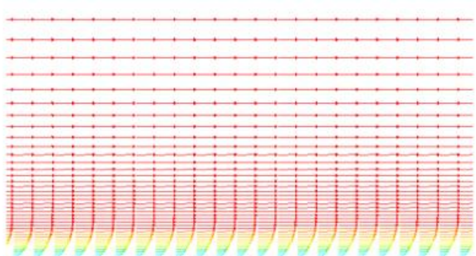
$$Re = \left[\frac{UH\rho}{\mu}\right]$$



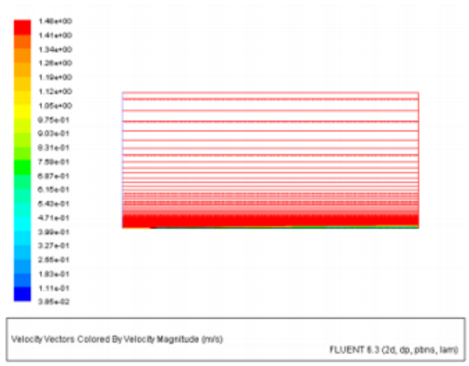
Result:

Temperature Profile

Velocity Profile:



Velocity Contours:



VIVA Questions:

1. What is external flow compared to internal flow.

2. What is convection?

3. Write Nusselt Number?

EXPERIMENT-5

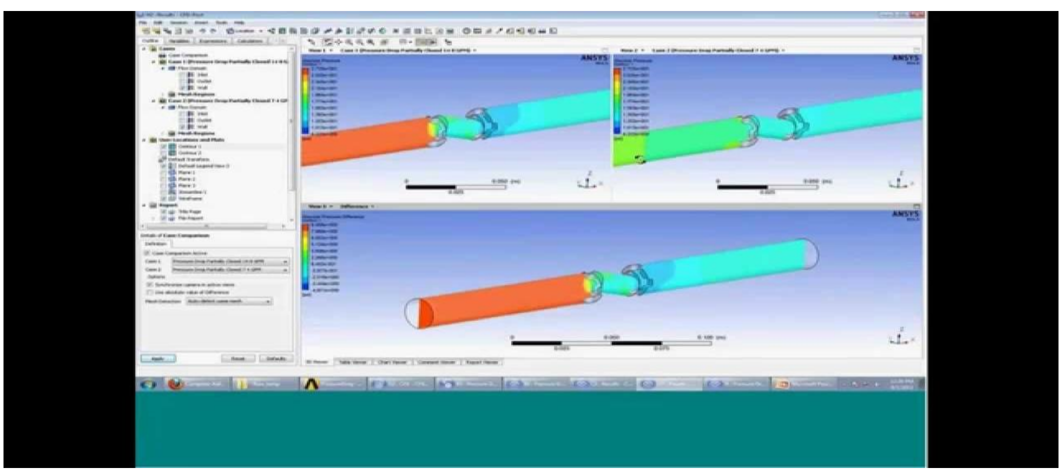
FLOW THROUGH BALL VALVE

AIM:Flow of water through a ball valve assembly using ANSYS/ Solid Works Flow Simulation.

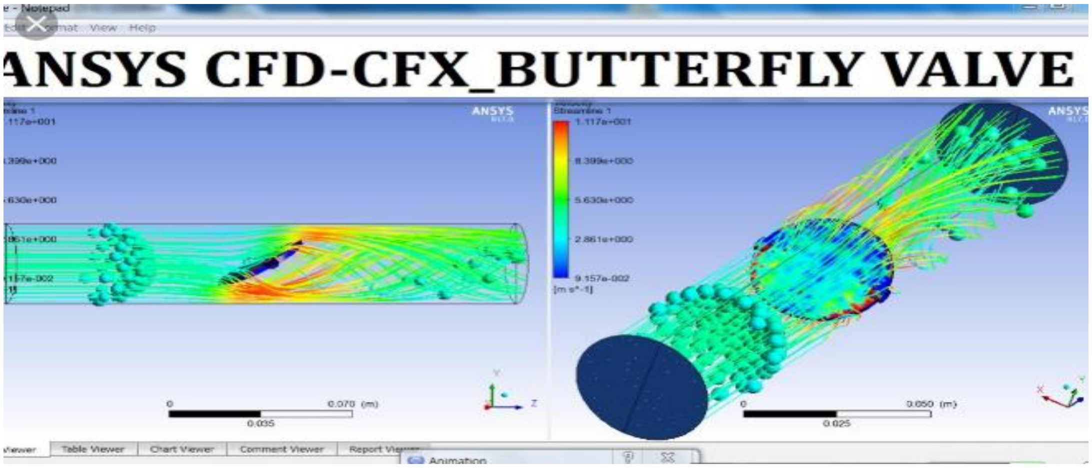
Introduction:

A **ball valve** is a device with a spherical closure unit that provides **on**/off control of **flow**. The sphere has a port, also known as a bore, **through** the center. When the **valve** is positioned such that the bore is aligned **in the** same direction as the pipeline, it is in open position and fluid can **flow through** it.

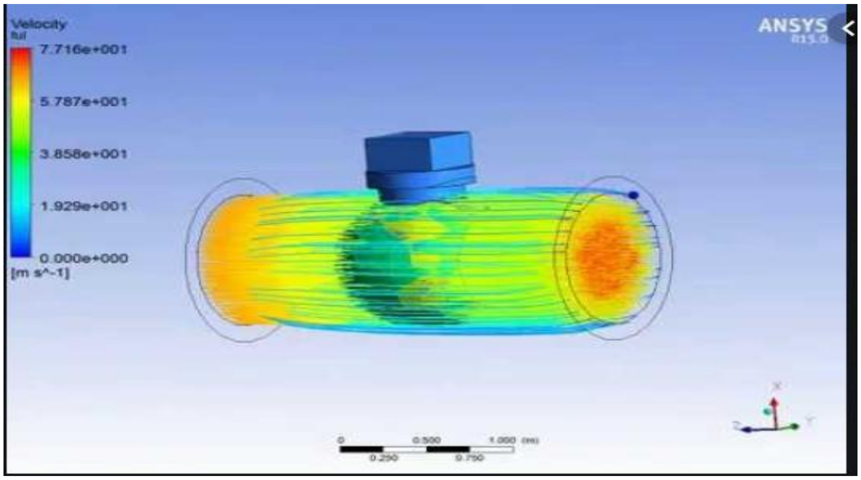
Result:



Flow Distributuin:



Velocity contours:



Outcome:

Modeling and assemble of ball valve .Simulation of flow through ball valve.

EXPERIMENT -6

HEAT CONDUCTION

AIM:

To conduct the thermal stress analysis of a 2D component by using ANSYS software

System Configuration:

Ram	:	8 GB
Processor	:	Core 2 Quad / Core 2 Duo
Operating system	:	Windows 7
Software	:	ANSYS (Version12.0/12.1)

Procedure:

The three main steps to be involved are

1. Pre Processing
2. Solution
3. Post Processing

Start - All Programs – ANSYS 12.0/12.1 - Mechanical APDL Product Launcher – Set the Working Directory as E Drive, User - Job Name as Roll No., Ex. No. – Click Run

Preprocessing:

1. Preference – Thermal - h-Method - ok

2. Preprocessor - Element type - Add/Edit/Delete – Add – Solid, Quad 4 node 42 – ok – Options – plane strs w/thk – ok – Close

3. Real constants - Add/Edit/Delete – Add – ok – THK 100 – ok – Close

4. Material props - Material Models –Structural – Linear – Elastic - Isotropic – EX 2e5, PRXY 0.3 – ok –Thermal expansion – Secant coefficient – Isotropic – ALPX 12e-6 – ok

5.Modeling – Create – Areas - Rectangle – by 2 corners – Enter the coordinate values, height, width - ok

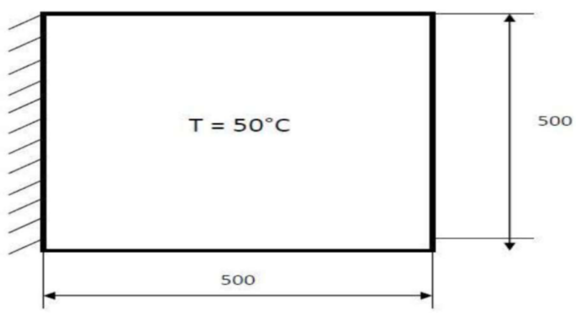
6. Meshing – Mesh tool – Areas, set – select the object – ok – Element edge length 10 - ok – Mesh tool- Tri, free - mesh – Select the object

Solution:

7. Solution – Define Loads – Apply – Structural – Displacement - On lines – Select the boundary on the object –ok – Temperature – Uniform Temp – Enter the temp. Value 50 –ok.
8. Solve – Current LS – ok – Solution is done – close

Post Processing:

9. General post proc – Plot results – Contour plot – Nodal solution – Stress – 1st principal stress – ok – Nodal solution – DOF Solution – Displacement vector sum - ok
10. File – Report Generator – Choose Append – ok – Image Capture – ok - close



Young's Modulus	= 200 GPa
Poisson's ratio	= 0.3
Thermal expansion coefficient	= 12×10^{-6} /^{0}C

Result:

Thus the thermal stress analysis of a 2D component is done by using the ANSYS Software.

Outcome:

Able to analyse the thermal stress of a 2D component using the ANSYS software.

EXPERIMENT-7

TEMPERATURE DISTRIBUTION

AIM: To determine the temperature distribution in solid body.

System Configuration:

Ram	:	8 GB
Processor	:	Core 2 Quad / Core 2 Duo
Operating system	:	Windows 7
Software	:	ANSYS (Version12.0/12.1)

Procedure:

The three main steps to be involved are

1. Pre Processing
2. Solution
3. Post Processing

Start - All Programs – ANSYS 12.0/12.1 - Mechanical APDL Product Launcher – Set the Working Directory as E Drive, User - Job Name as Roll No., Ex. No. – Click Run

Preprocessing:

1. Preference – Thermal - h-Method - ok
2. Preprocessor - Element type - Add/Edit/Delete – Add – Solid, Quad 4 node 55 – ok – close – options – plane thickness – ok
3. Real constants - Add/Edit/Delete – Add – ok – THK 0.5 – ok – Close
4. Material props - Material Models –Thermal – Conductivity – Isotropic – KXX 10 – ok
5. Modeling – Create – Areas - Rectangle – by 2 corners – Enter the coordinate values, width - ok
6. Meshing – Mesh tool – Areas, set – select the object – ok – Element edge length 0.05 - ok – Mesh tool- Tri, free - mesh – Select the object –ok

Solution:

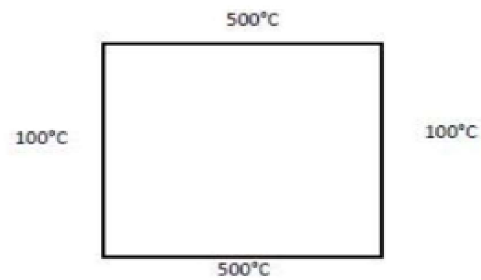
7. Solution – Define Loads – Apply – Thermal – Temperature - On lines – Select the right and left side of the object –ok – Temp. Value 100 – On lines – select the top and bottom of the object – ok –Temp 500 – ok

|

8. Solve – Current LS – ok – Solution is done – Close

Post Processing:

9. General post proc – Plot results – Contour plot – Nodal solution – DOF solution – Nodal Temperature – ok
10. File–Report Generator–Choose Append–ok–Image Capture–ok - close



Thermal Conductivity of the material = 10 W/m ^{0}C

Dimension of the object = 2m × 2m

Result:

Thus the conductive heat transfer analysis of a 2D component is done by using ANSYS software.

VIVA Questions:

1. Define Conduction
2. Units of Thermal conductivity
3. Basic law for conduction.
4. Define thermal resistance

EXPERIMENT -8

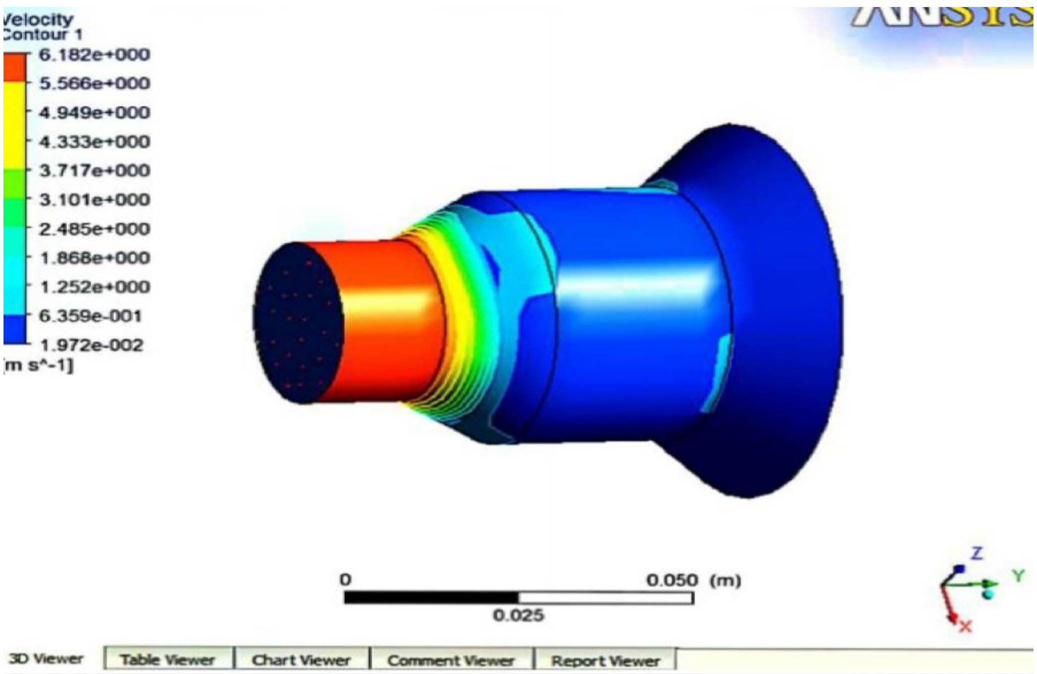
3D HEAT CONDUCTION

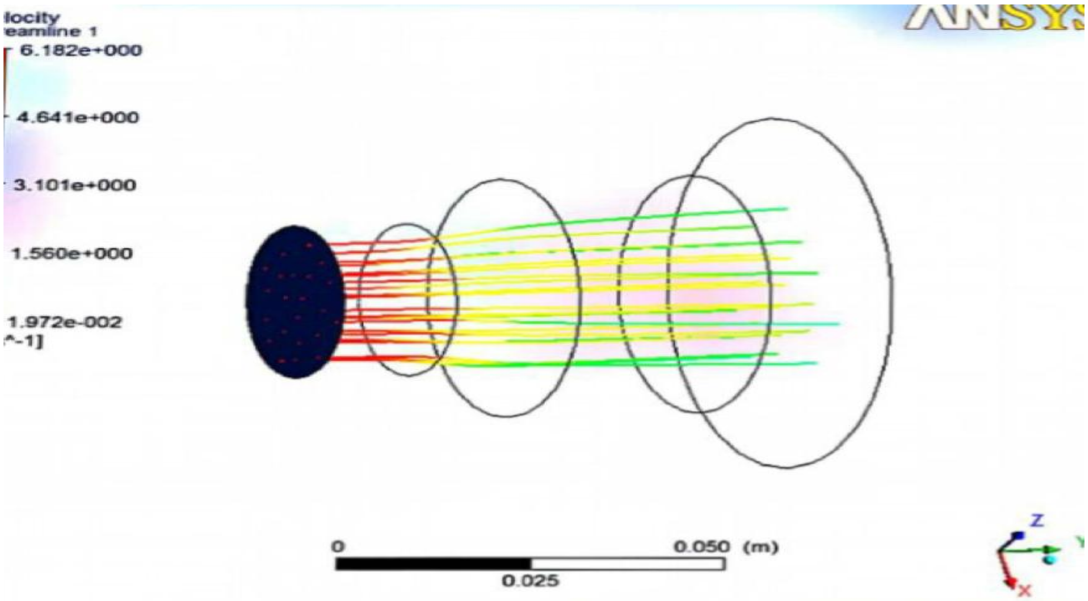
AIM:3D Heat Conduction within a Solid-Cell Phone using ANSYS.

Requirements:

Ansys software.

RESULT:





VIVA:

1. What is Conduction?

2. Write expression for conduction equation

3. Units of heat flux

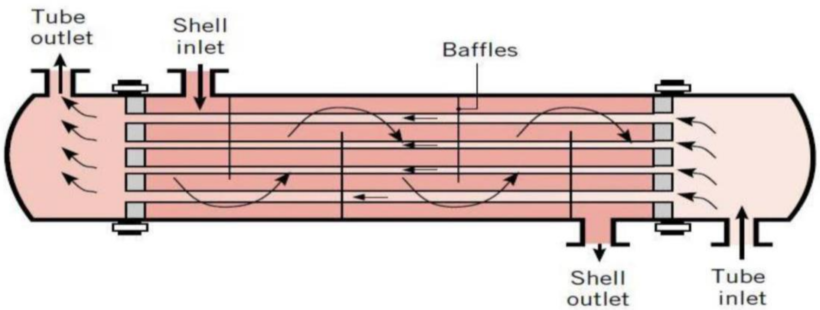
EXPERIMENT -9

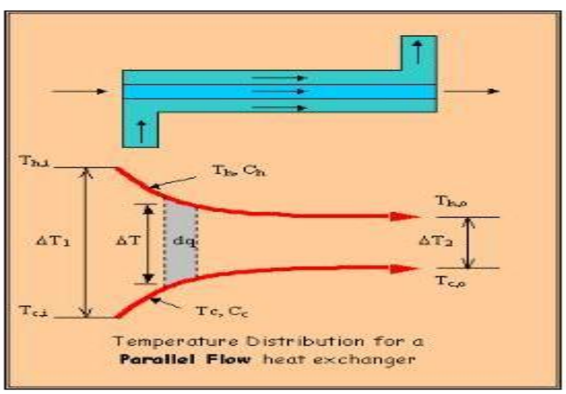
COUNTER FLOW HEAT EXCHANGER

AIM: TO determine temperature distribution in heat exchanger.

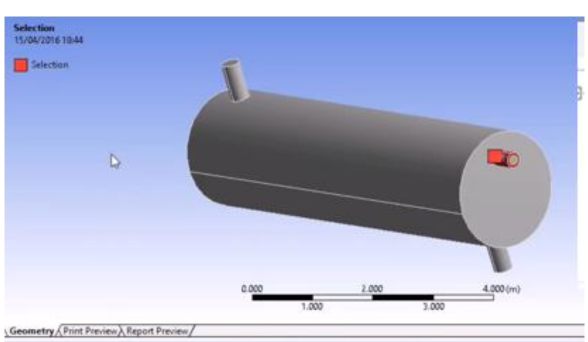
Introduction:

This exchanger is built of a bundle of round tubes mounted in a large cylindrical shell with the tube axis parallel to the shell to transfer the heat between the two fluids. The fluid flows inside the tubes and other fluid flows across and along the tubes.

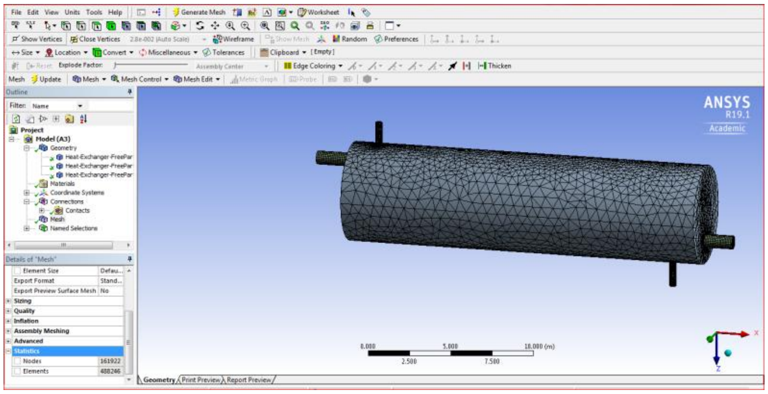




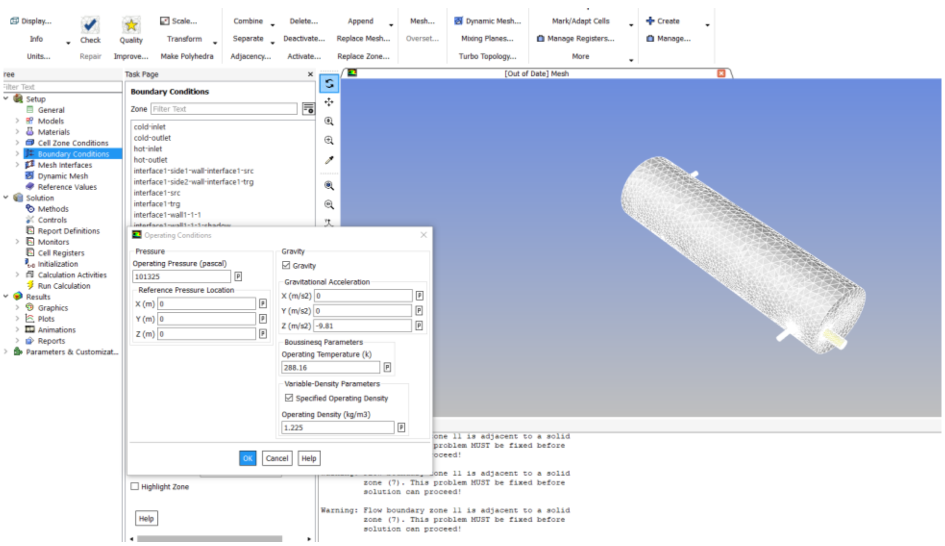
Geometry:



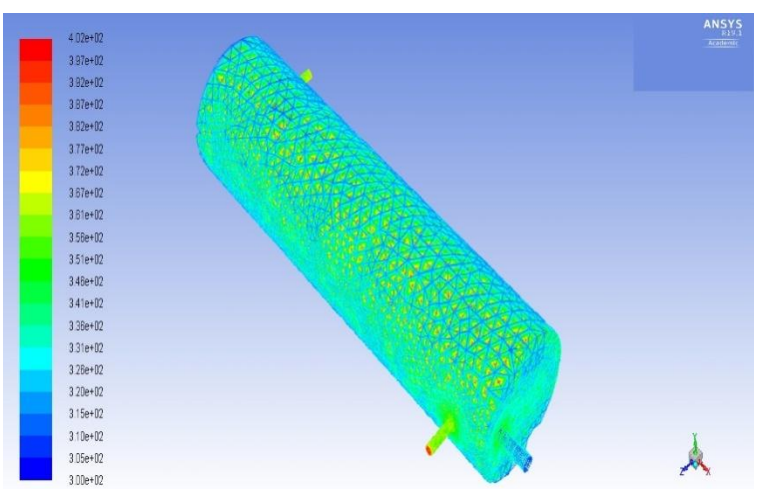
Meshing:



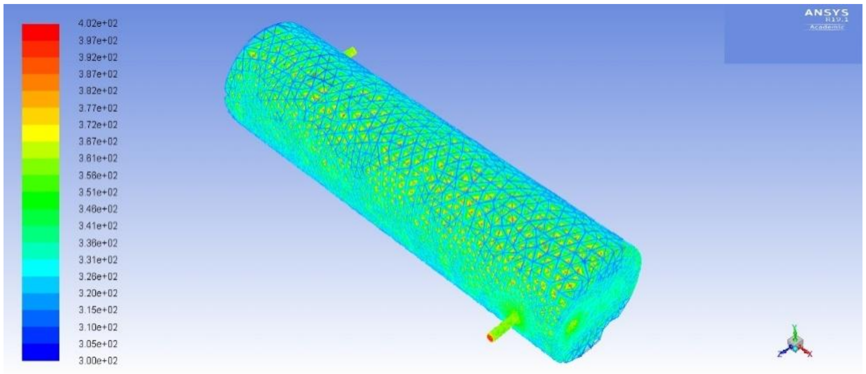
Boundary Conditions:



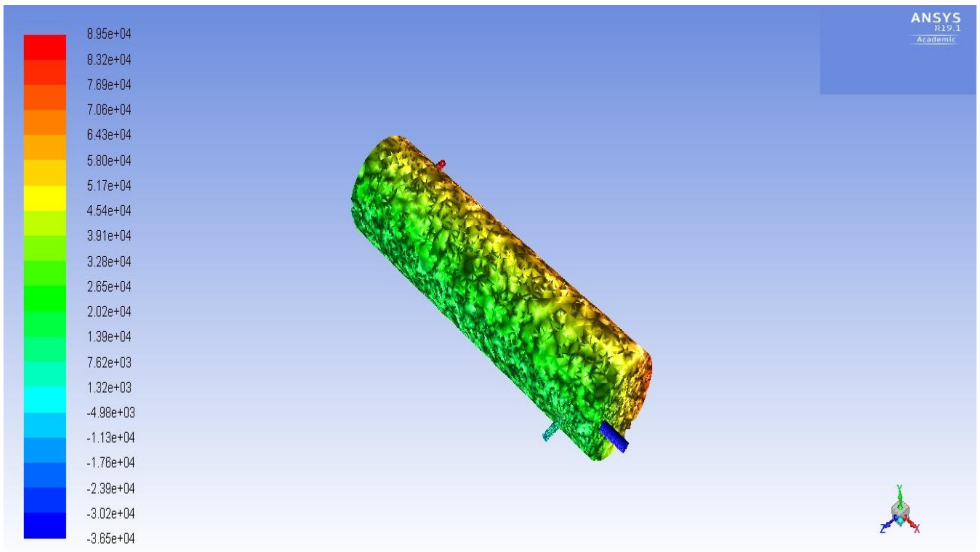
Result:



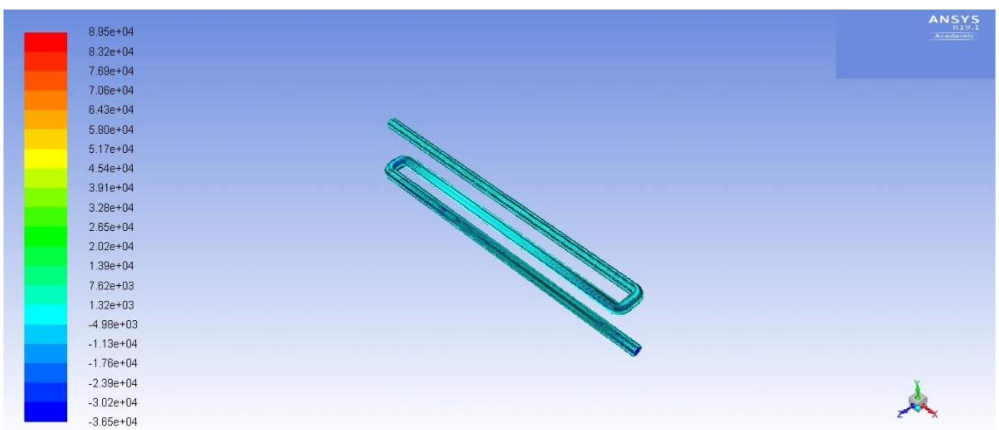
Static temperature of tube:



Static temperature of shell



Static pressure of tube



VIVA Questions:

1. What is parallel flow and counter flow heat exchanger.

2. Why do we use baffle plates.

3. What is effectiveness of heat exchanger?

EXPERIMENT -10

CONJUGATE HEAT TRANSFER

AIM: Conjugate heat transfer problem using ANSYS/ Solid Works Flow Simulation.

Introduction

The physics of conjugate heat transfer is common in many engineering applications, including heat exchangers, HVAC, and electronic component design. The geometry and flow domain consists of a flat circuit board with a heat generating electronic chip mounted on it. Heat is conducted through the source (chip) and the board on which it is mounted. A laminar stream of air flows over the board and the chip, causing simultaneous cooling of the solid components and heating of the air stream due to convection. Thermal energy is also transported due to the complex flow field.

ANSYS FLUENT

•Set up appropriate boundary conditions for a conjugate heat transfer simulation in Enable source terms for specified zones.

• Perform flow and energy calculations using various materials (both solid and fluid).

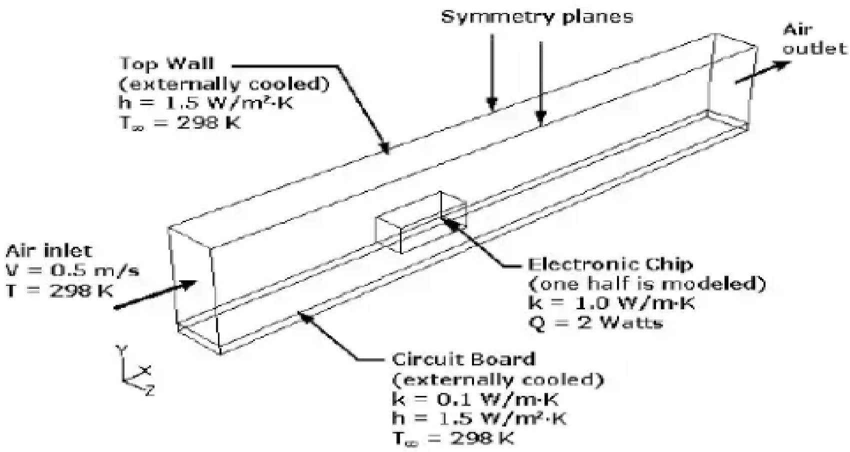
• Manipulate mesh adaption registers and perform Boolean operations on them.

• Perform mesh adaption and verify that the solution is mesh independent.

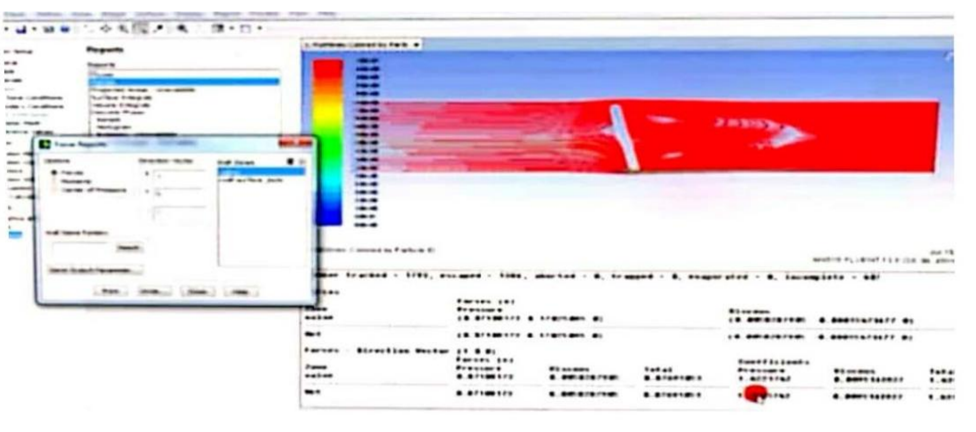
Problem Description

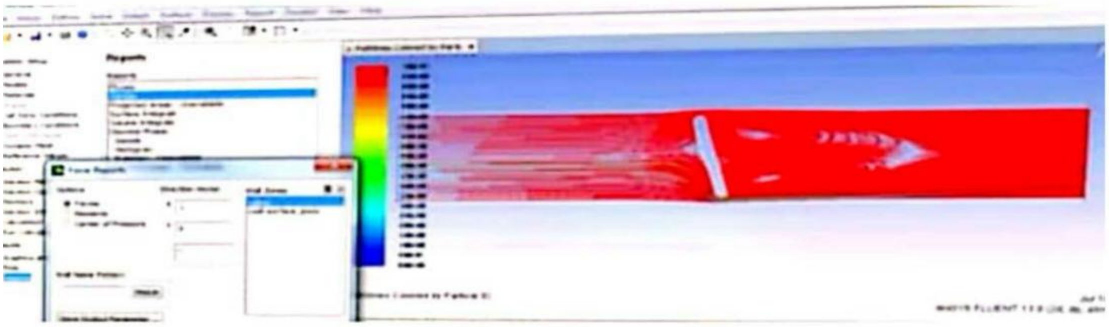
The problem considered is shown schematically in Figure 1. The configuration consists of a series of heat-generating electronic chips mounted on a circuit board. Air flow, confined between the circuit board and an upper wall, cools the chips and the board. As the air flows over the chips and the board, its temperature rises. Taking the symmetry of the configuration into consideration, the model extends from the middle of one chip to the plane of symmetry between it and the next chip.

As shown in Figure 1, each half chip is assumed to generate 1 Watt and have a thermal conductivity of 1.0 W/m-K. The circuit board conductivity is assumed to be one order of magnitude lower, at 0.1 W/m-K. Air enters the system at 298 K with a velocity of 0.5 m/s. The inlet Reynolds number (based on the spacing between the upper and lower walls) is approximately 870 and thus, the flow is treated as laminar.



Result:





VIVA Questions:

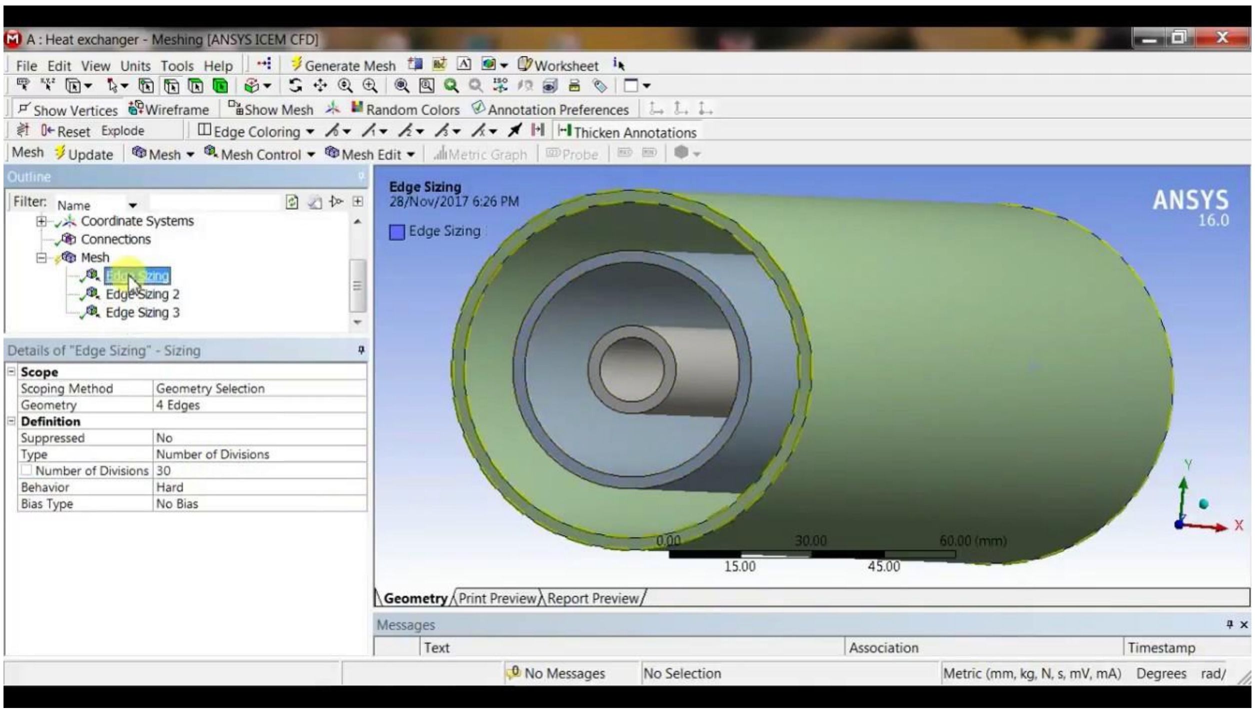
1. Explain conjugate heat transfer.
2. What is lamina flow?
3. What is fully developed turbulent flow?

EXPERIMENT -11

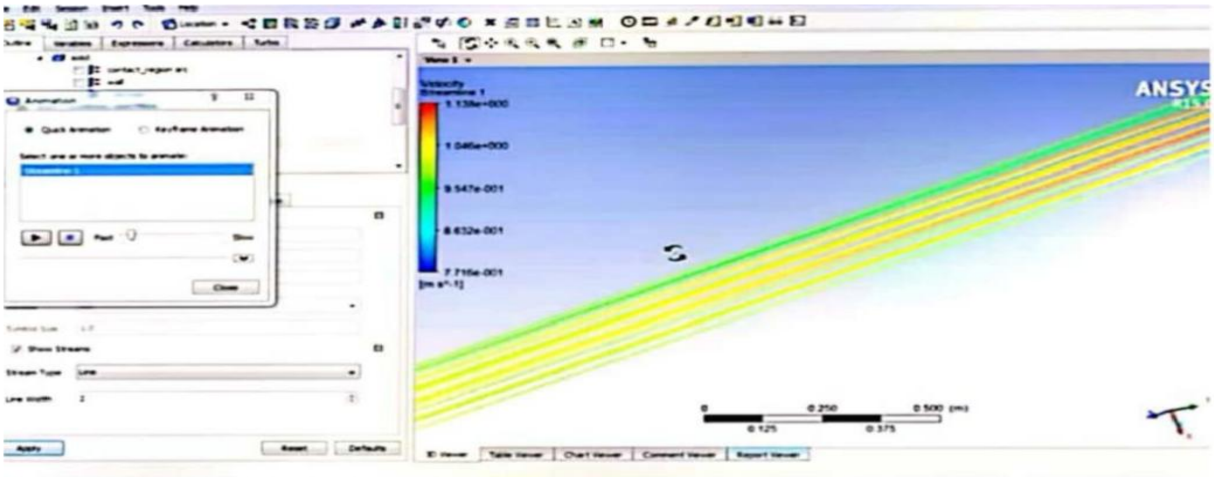
3D THERMAL ANALYSIS

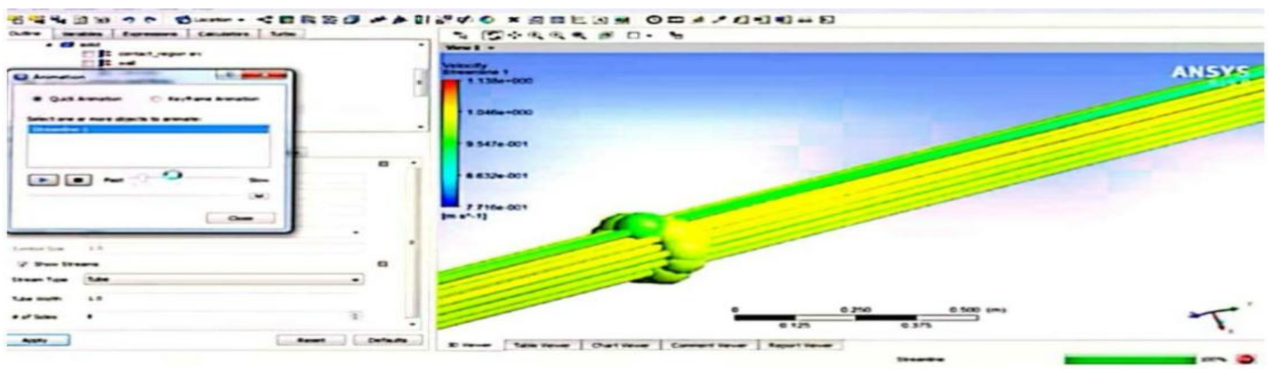
AIM: To do the 3D Thermal Analysis for a Finned Pipe using ANSYS.

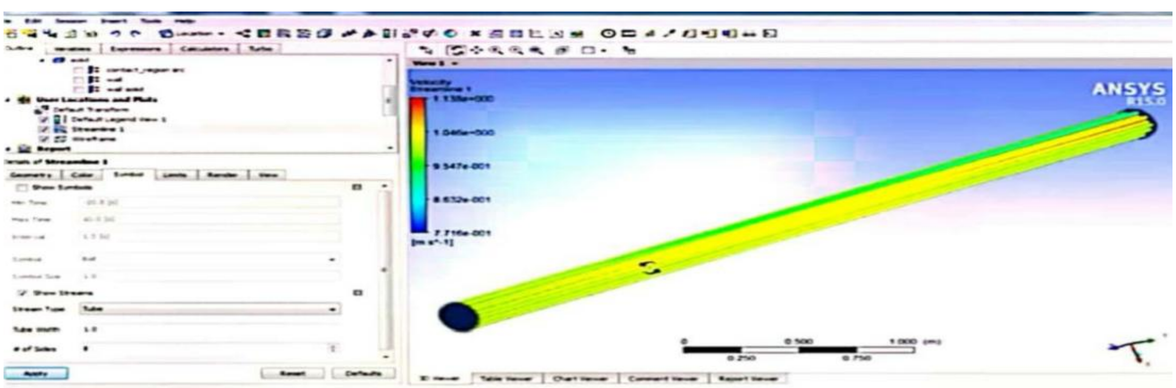
Problem Description:



RESULT







VIVA Questions:

1. What are the assumptions in the analysis?

2. What is 3D PROBLEM?

3. What is steady state?

EXPERIMENT - 12

THERMAL STRESS ANALYSIS

AIM: Thermal stress analysis of piston.

Materials:

The most commonly used materials for pistons of I.C. engines are cast iron, aluminum alloy (cast aluminum, forged aluminum), cast steel and forged steel.

a)For a cast iron piston, the temperature at the Centre of the piston head (TC) is about 425°C to 450°C under full load conditions and the temperature at the edges of the piston head (TE) is about 200°C to 225°C.

b)For aluminum alloy pistons, TC is about 260°C to 290°C and TE is about 185°C to 215°C.

ANALYTICAL CALCULATION FOR PISTON Analytical calculations are done for cast iron piston. For doing analytical calculation material properties and dimensional information should be known and so all the parameters consider for design of piston are calculated by using one analytical problem. Design Considerations for a Piston In designing a piston for an engine, the following points should be taken into consideration:

• It should have enormous strength to withstand the high pressure.

• It should have minimum weight to withstand the inertia forces.

• It should form effective oil sealing in the cylinder. • It should provide sufficient bearing area to prevent undue wear.

• It should have high speed reciprocation without noise. • It should be of sufficient rigid construction to withstand thermal and mechanical distortions.

• It should have sufficient support for the piston pin. Procedure for Piston Design The procedure for piston designs consists of the following steps:

• Thickness of piston head (tH)

• Heat flows through the piston head (H)

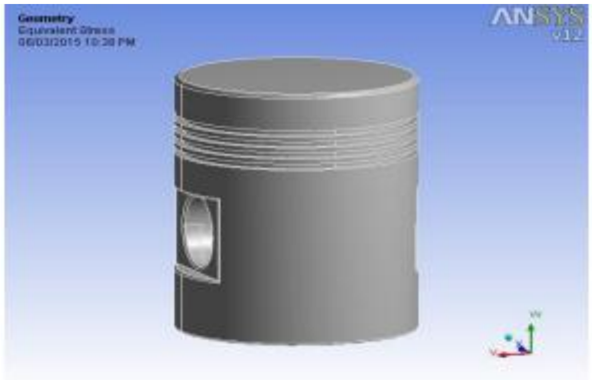
• Radial thickness of the ring (t1)

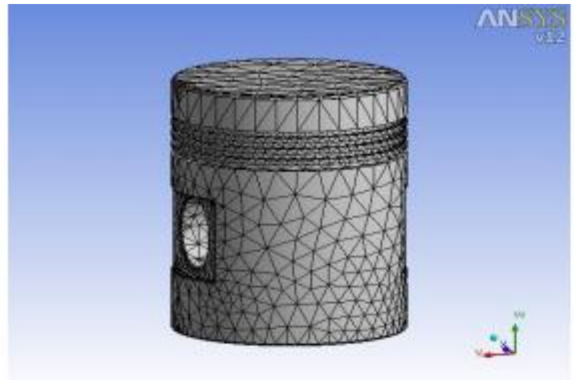
• Axial thickness of the ring (t2)

• Width of the top land (b1)

• Width of other ring lands (b2)

Design and Meshing of a Piston:





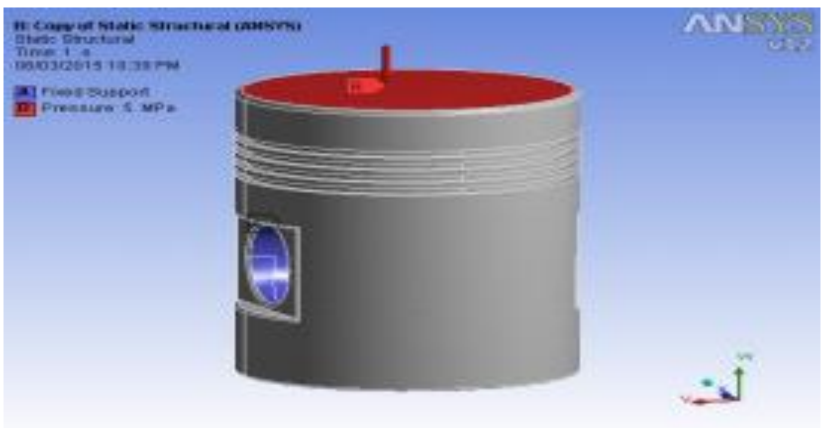
Analysis of Piston The piston is analyzed by giving the constraints they are

1. Pressure or structural analysis

2.Boundary condition

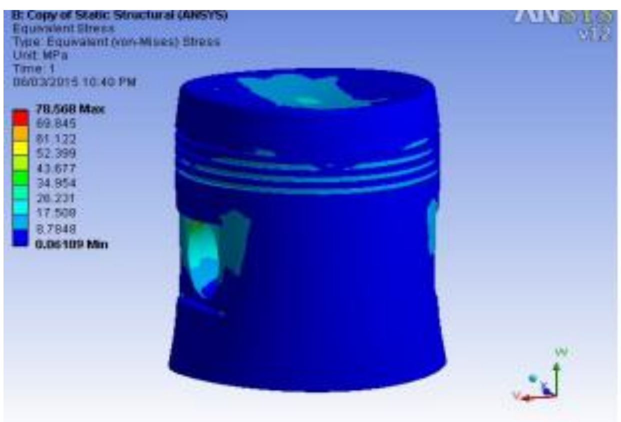
3.Thermal analysis

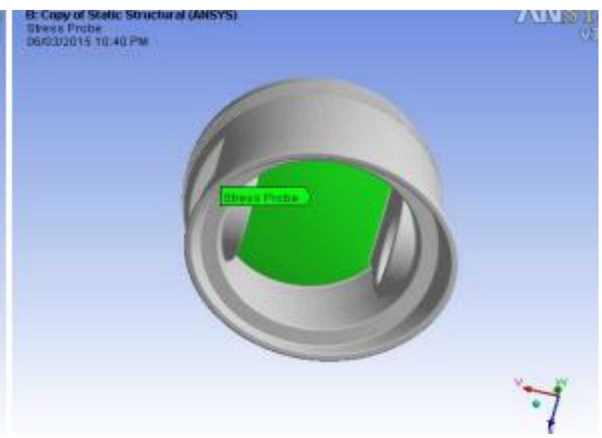
4. Thermo-mechanical analysis



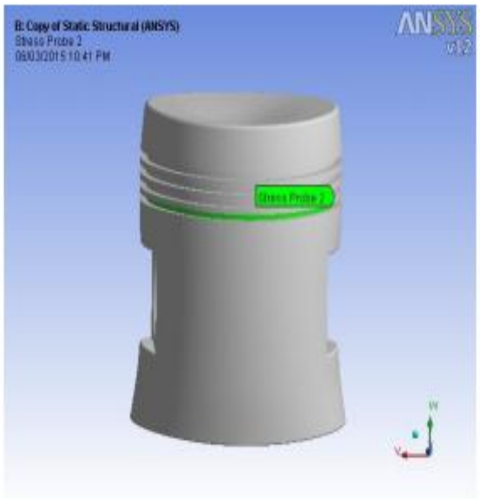
Pressure applied

RESULTS:

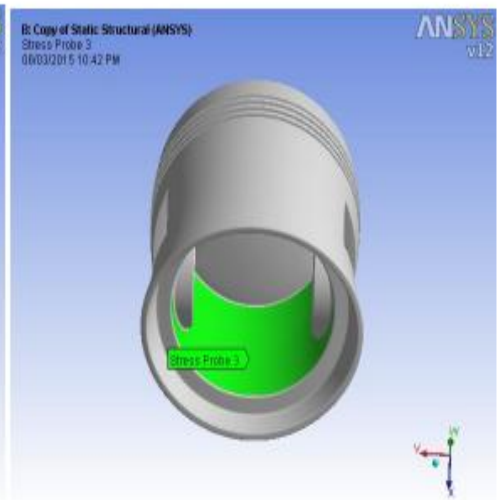


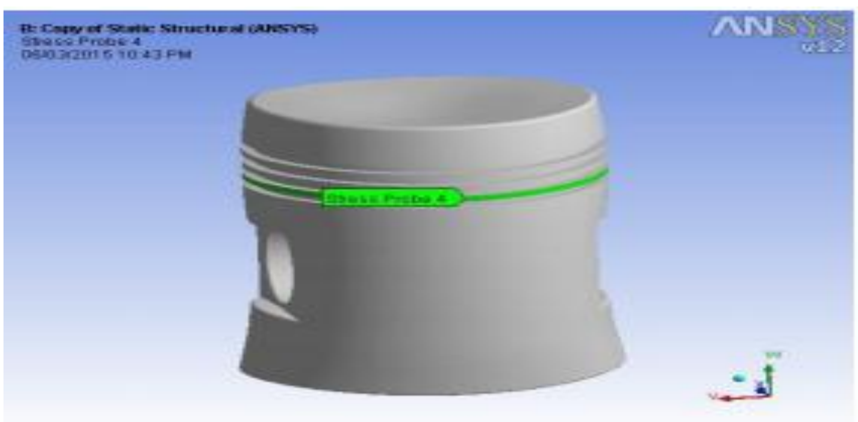


Maximum Equivalent Stress -78.56 MPa

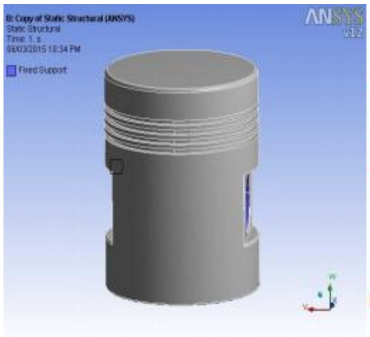


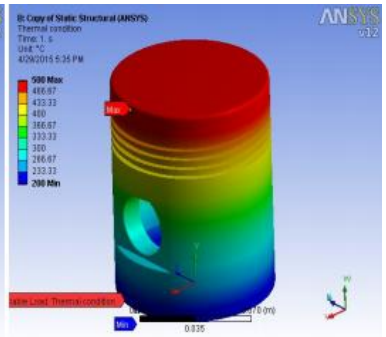
Bending Stress – 38.25 MPa

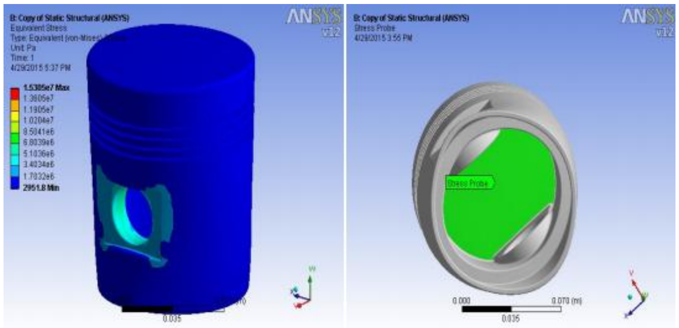




Thermal Analysis







VIVA Questions:

1. What is stress?

2. What is heat flux?

3. Define thermal stress analysis