



INSTITUTE OF AERONAUTICAL ENGINEERING

(Autonomous)

Dundigal - 500 043, Hyderabad, Telangana

COURSE CONTENT

STHERMO – FLUID MODELLING AND SIMULATION LABORATORY								
VI Semester: Mechanical Engineering								
Course Code	Category	Hours / Week			Credits	Maximum Marks		
AMEC27	Core	L	T	P	C	CIA	SEE	Total
		0	0	3	1.5	30	70	100
Contact Classes: Nil	Tutorial Classes: Nil	Practical Classes: 45			Total Classes: 45			
Prerequisite: Fluid Mechanics Lab, Heat Transfer Lab								

I. COURSE OVERVIEW

This laboratory course provides learners, the practical knowledge on Ansys Workbench and its different modules. The students will get the hands-on experience to validate their theoretical design ideas to provide solutions to real-world problems. The lab sessions focus on modeling, meshing, applying the physical conditions, solving and reviewing the results of fluid thermal systems. This helps the learners in interdisciplinary applications such as mechanical, aerospace, automobile, refrigeration systems etc.

II. COURSES OBJECTIVES

The students will try to learn

- I. The formulation of the problem including material properties and boundary conditions.
- II. The modelling of complex geometries in the best possible manner and identify the domain of interest.
- III. The discretization of the domain in a manner to capture the physics accurately.
- IV. The software to analyse results with different methods like contour plots, vector plot, streamlines, etc.
- V. The skills of using CFD packages to carry out research in interdisciplinary applications.

III. COURSE OUTCOMES

At the end of the course students should be able to:

- CO 1 Demonstrate the application of finite element method for analyzing 2D fluid flow problems.
- CO 2 Make use of Ansys CFX or Solid Works Flow Simulation for analyzing simple fluid flow problems.
- CO 3 Utilize Ansys or Solid Works Flow Simulation for analyzing simple heat transfer problems.
- CO 4 Make use of Ansys Fluent for analyzing conjugate heat transfer.
- CO 5 Utilize Ansys for analyzing thermal stress in Beam.
- CO 6 Develop the MATLAB code for analyzing 2D fluid flow problems.

IV. COURSE CONTENT

Safety

Safety is a vital issue in all workplaces. Before using any equipment and machines or attempt practical work in a laboratory everyone must understand basic safety rules. These rules will help keep all safe in the laboratory.

Safety Rules

1. Always listen carefully to the teacher and follow instructions.
2. When learning how to use a machine, listen very carefully to all the instructions given by the faculty / instructor. Ask questions, especially if you do not fully understand.
3. Bags should not be brought into the lab as people can trip over them.
4. Always be patient, never rush in the laboratory.
5. Report any damage to computer as this could cause an accident.
6. Keep your work area clean.

EXERCISES ON THERMO – FLUID MODELLING AND SIMULATION LABORATORY

1 Getting Started Exercises

1.1 Introduction

ANSYS is the software used to do analysis of structures and thermo-fluid systems. In this lab we use CFX, steady state thermal and Fluent tools. Go to the website <https://www.ansys.com/en-in/academic/students>. Register as a student for one year academic license. Download and Install latest version of ANSYS on your machine.

1.2 Exercises on sketching tools

Draw the 2D sketch shown in Figure 1.1

Hints:

- 1 Use draw tools line, rectangle, square, circle, pentagon, hexagon, ellipse, arc, and spline.
- 2 Use modify tools fillet, chamfer, trim, extend, offset, duplicate.
- 3 Use dimension tools horizontal, vertical, radius, diameter, general, and angle.

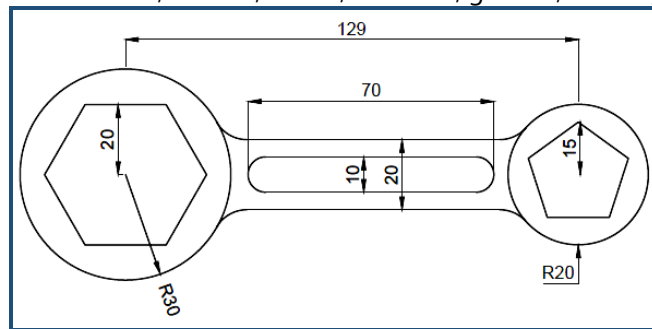


Figure 1.1: 2D sketch

1.3 Exercises on concept tools

Draw the 2D surface shown in Figure 1.2

Hints:

- 1 Use concept tools lines from points, lines from sketches, lines from edges, surfaces from sketches, surfaces from edges, cross sections.
- 2 Use body operation and body transformation commands.

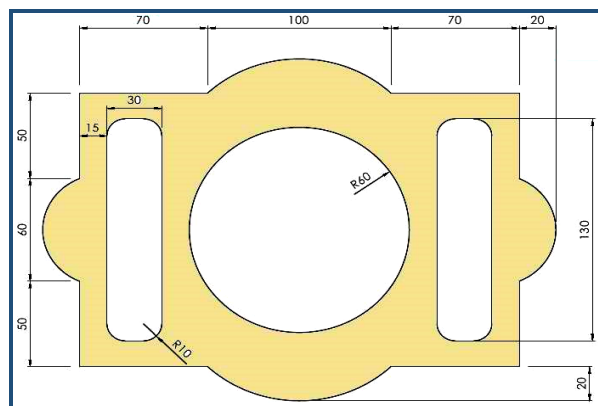


Figure 1.2: 2D Surface

1.4 Exercises on create tools

Draw the 3D model shown in Figure 1.3

Hints:

- 1 Use extrude, revolve, sweep, skin/loft.
- 2 Use Boolean operations, primitive operations, and pattern.

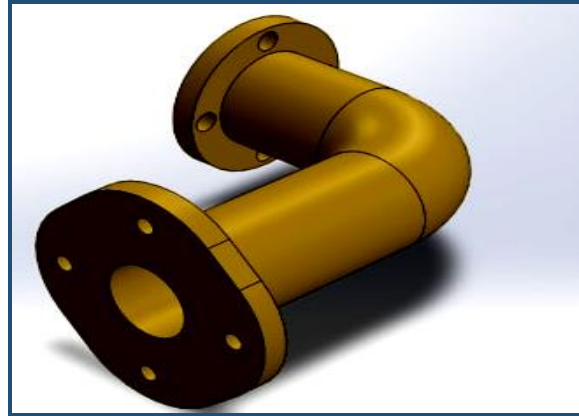


Figure 1.3: 3D model

Try:

1. Sketch the 3D model of piston head.
2. Sketch the 3D model of connecting rod.

2 Exercises on meshing (discretization)

In CFD discretization is a fundamental process that converts continuous governing equations into a set of discrete equations.

2.1 Exercises on line meshing

Discretize the 1D model shown in Figure 2.1

Hints:

1. Import the geometry from DesignMolder or any other modeling software into the work bench meshing.
2. Use the sizing command.
3. Set number of elements without bias, with bias and bias factor.

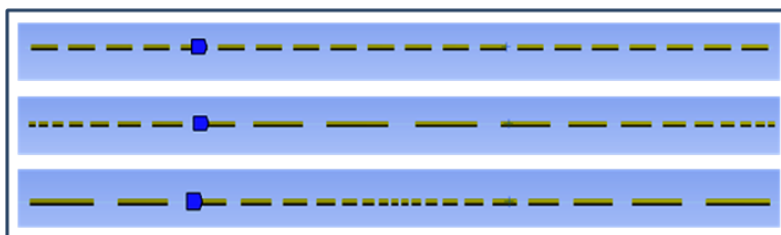


Figure 2.1: 1D meshing

2.2 Exercises on surface meshing

Discretize the 2D model shown in Figure 2.2

Hints:

1. Import the geometry from DesignMolder or any other modeling software into the work bench meshing.
2. Use the inflation command.
3. Set number of elements without bias, with bias and bias factor.

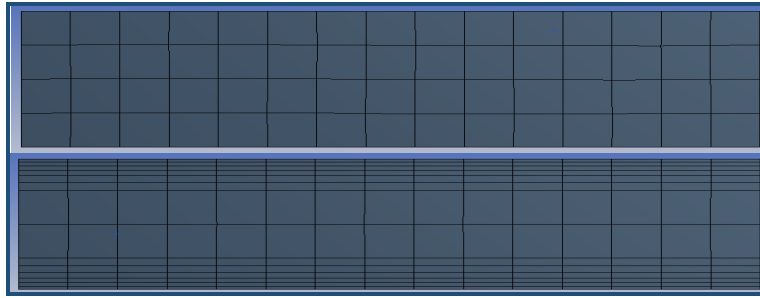


Figure 2.2: 2D meshing

2.3 Exercises on body meshing

Discretize the 3D model shown in Figure 2.3

Hints:

1. Import the geometry from DesignMolder or any other modeling software into the workbench meshing.
2. Divide the object into blocks.
3. Use the sizing command.
4. Set number of elements with bias and bias factor.

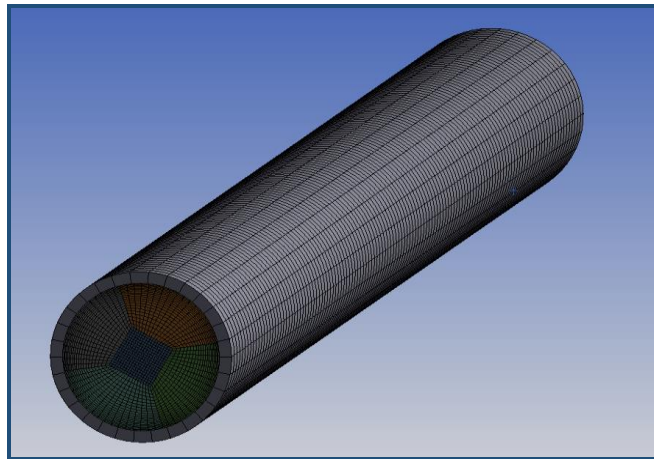


Figure 2.3: 2D meshing

Try:

1. Perform the meshing of ball valve.
2. Perform the meshing of piston head.

3 CFD analysis of flow through ducts

Ducts are conduits or passages used in heating, ventilation, and air conditioning (HVAC) to deliver and remove air.

3.1 Analyze the flow through a straight duct

Perform the CFD Simulation of flow of water through a circular duct for the given velocity.

Hints:

1. Make the model of the domain in Ansys Design-modeler or import domain form other modeling software.
2. Discretize of domain using the Ansys workbench meshing.
3. Setup the physics in Ansys CFX pre.
4. Run the Ansys CFX solver until the convergence attained.
5. Analyze the results using the Ansys CFD post to get the flow contour as shown in Figure 3.1.



Figure 3.1: Flow through the straight duct

3.2 Analyze the flow through a convergent divergent duct

Perform the CFD Simulation of flow of water through convergent divergent duct for the given velocity.

Hints:

1. Make the model of the domain in Ansys Design-modeler or import domain form other modeling software.
2. Discretize of domain using the Ansys workbench meshing.
3. Setup the physics in Ansys CFX pre.
4. Run the Ansys CFX solver until the convergence attained.
5. Analyze the results using the Ansys CFD post to get the flow contour as shown in Figure 3.2.

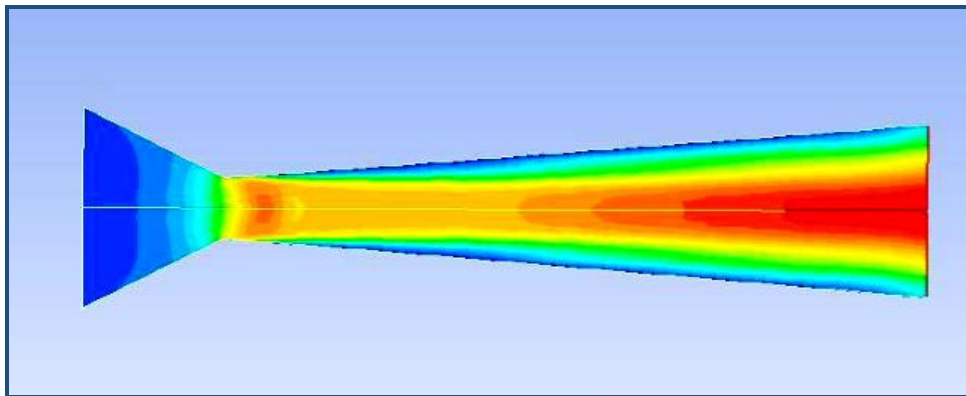


Figure 3.2: Flow through the convergent divergent duct

Try:

- 1 Analyze the fluid flowing through a rectangular duct.
- 2 Analyze the fluid flowing through a pipe bend.
- 3 Apply the FEM theory for analyzing the flow through the pipe.

4 CFD analysis of flow over blunt bodies

The non-streamlined or un-streamlined shapes with blunt front and/or rear faces called bluff bodies (or blunt bodies) have many engineering applications.

4.1 Analyze the flow over a cylinder

Perform the CFD Simulation of flow of air over a cylinder for the given velocity.

Hints:

1. Make the model of the domain in Ansys Design-modeler or import domain form other modeling software.
2. Discretize of domain using the Ansys workbench meshing.
3. Setup the physics in Ansys CFX pre.
4. Run the Ansys CFX solver until the convergence attained.

5. Analyze the results using the Ansys CFD post to get the flow contour as shown in Figure 4.1.

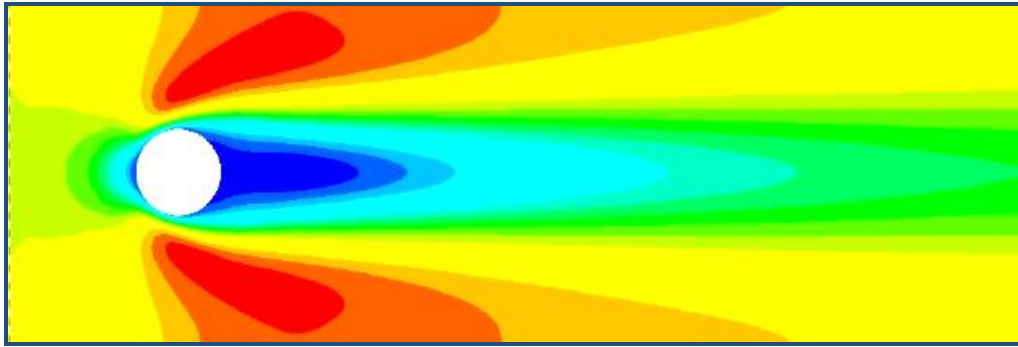


Figure 4.1: Flow over cylinder

4.2 Analyze the flow over a building

Perform the CFD Simulation of flow of air over a small building for the given velocity.

Hints:

1. Make the model of the domain in Ansys Design-modeler or import domain from other modeling software.
2. Discretize of domain using the Ansys workbench meshing.
3. Setup the physics in Ansys CFX pre.
4. Run the Ansys CFX solver until the convergence attained.
5. Analyze the results using the Ansys CFD post to get the streamlines as shown in Figure 4.2.

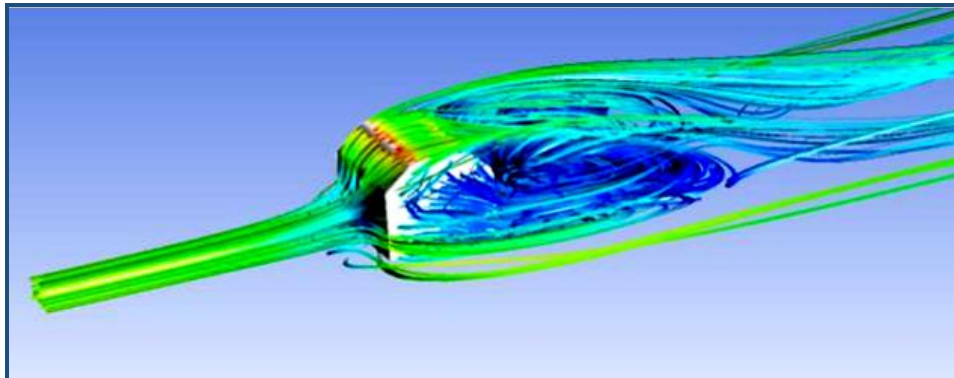


Figure 4.2: Flow over a building

Try:

1. Analyze the fluid flowing over a rectangle.
2. Analyze the fluid flowing over an automobile.

5 CFD analysis of flow over stream lined bodies

A streamlined body is a shape that lowers the friction drag between a fluid, like air and water, and an object moving through that fluid.

5.1 Analyze the flow over a symmetric aerofoil

Perform the CFD Simulation of flow of air over a symmetric aerofoil for the given velocity.

Hints:

1. Make the model of the domain in Ansys Design-modeler or import domain from other modeling software.
2. Discretize of domain using the Ansys workbench meshing.
3. Setup the physics in Ansys CFX pre.
4. Run the Ansys CFX solver until the convergence attained.

5. Analyze the results using the Ansys CFD post to get the flow contour as shown in Figure 5.1.

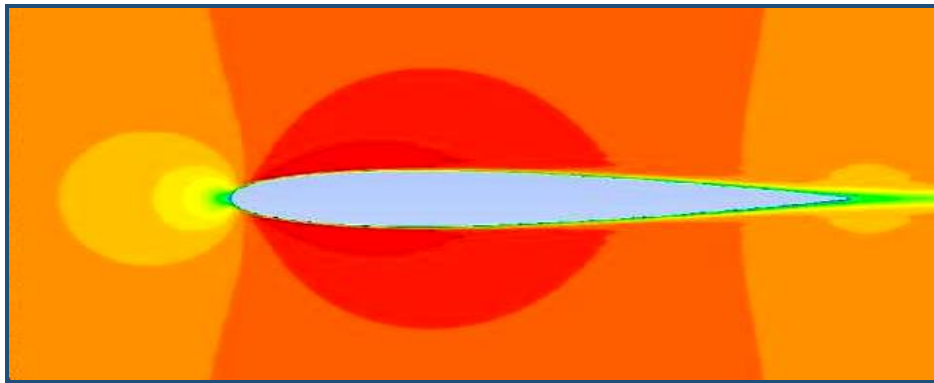


Figure 5.1: Flow over a symmetric aerofoil

5.2 Analyze the flow over a Cambered aerofoil

Perform the CFD Simulation of flow of air over a cambered aerofoil for the given velocity.

Hints:

1. Make the model of the domain in Ansys Design-modeler or import domain from other modeling software.
2. Discretize of domain using the Ansys workbench meshing.
3. Setup the physics in Ansys CFX pre.
4. Run the Ansys CFX solver until the convergence attained.
5. Analyze the results using the Ansys CFD post to get the flow contour as shown in Figure 5.2.

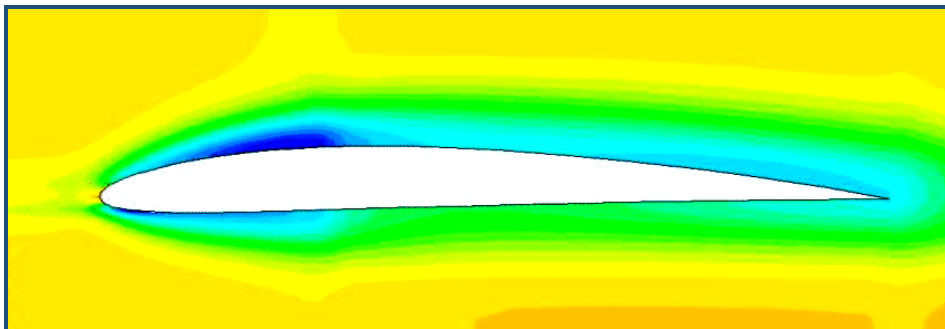


Figure 5.2: Flow over a Cambered aerofoil

Try:

1. Analyze the fluid flowing over an aerofoil with an angle of attack 10° .
2. Analyze the fluid flowing over an aerofoil with an angle of attack 15° .

6 CFD analysis of flow through valves

A valve is a device or natural object that regulates, directs or controls the flow of a fluid (gases, liquids, fluidized solids, or slurries) by opening, closing, or partially obstructing various passageways.

6.1 Analyze the flow through a ball valve

Perform the CFD Simulation of flow of water through a ball valve for the given velocity.

Hints:

1. Make the model of the domain in Ansys Design-modeler or import domain from other modeling software.
2. Discretize of domain using the Ansys workbench meshing.
3. Setup the physics in Ansys CFX pre.
4. Run the Ansys CFX solver until the convergence attained.

5. Analyze the results using the Ansys CFD post to get the flow contour as shown in Figure 6.1.

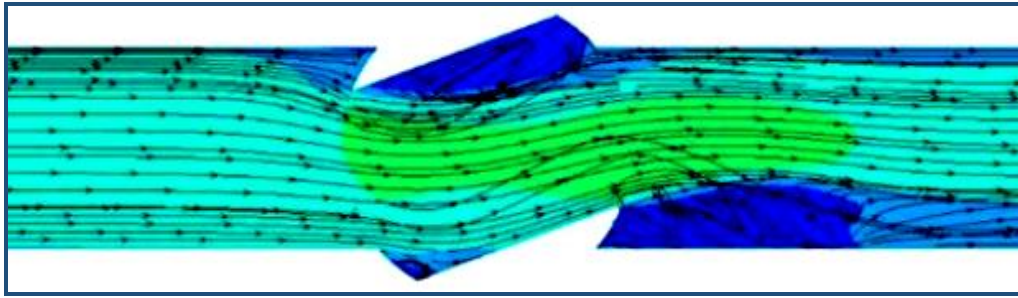


Figure 6.1: Stream lines through the ball valve

6.2 Analyze the flow through a gate valve

Perform the CFD Simulation of flow of water through a gate valve for the given velocity.

Hints:

1. Make the model of the domain in Ansys Design-modeler or import domain form other modeling software.
2. Discretize of domain using the Ansys workbench meshing.
3. Setup the physics in Ansys CFX pre.
4. Run the Ansys CFX solver until the convergence attained.
5. Analyze the results using the Ansys CFD post to get the flow contour as shown in Figure 6.2.

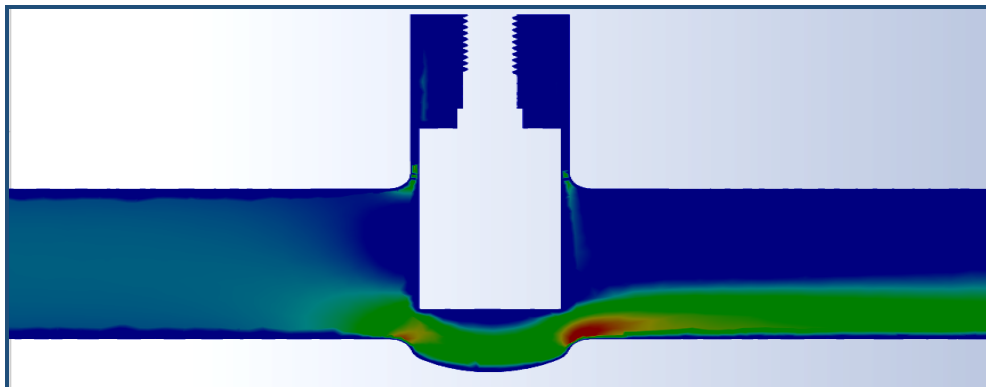


Figure 6.2: Velocity contour through gate valve

Try:

- 1 Analyze the fluid flowing through a Globe Valve.
- 2 Analyze the fluid flowing through a Needle Valve.

7 CFD Analysis of 2D steady state heat flow problems

Conduction is the process by which heat transfers from the hotter end to the colder end of an object.

7.1 Analyze the heat flow through a 2D wall

Perform the ANSYS Simulation of heat flow through a wall of given material.

Hints:

1. Make the model of the domain in Ansys Design-modeler or import domain form other modeling software.
2. Discretize of domain using the Ansys workbench meshing.
3. Setup the physics in Ansys steady state thermal pre.
4. Run the Ansys solver until the convergence attained.

5. Analyze the results using the Ansys CFD post to get the contour as shown in Figure 7.1.

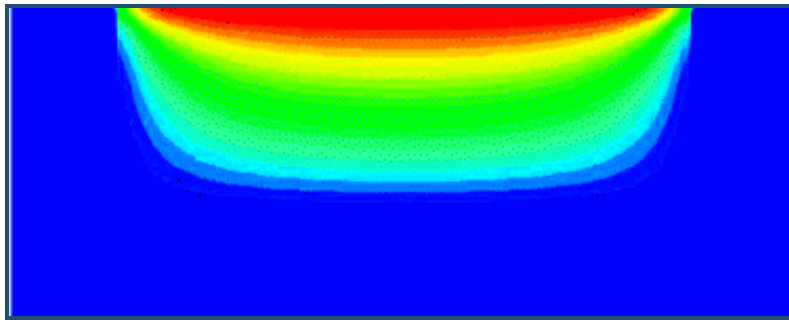


Figure 7.1: Temperature contour of single material

7.2 Analyze the heat flow through a 2D composite wall

Perform the ANSYS Simulation of heat flow through a composite wall of given materials.

Hints:

1. Make the model of the domain in Ansys Design-modeler or import domain from other modeling software.
2. Discretize of domain using the Ansys workbench meshing.
3. Setup the physics in Ansys steady state thermal pre.
4. Run the Ansys solver until the convergence attained.
5. Analyze the results using the Ansys CFD post to get the contour as shown in Figure 7.2.

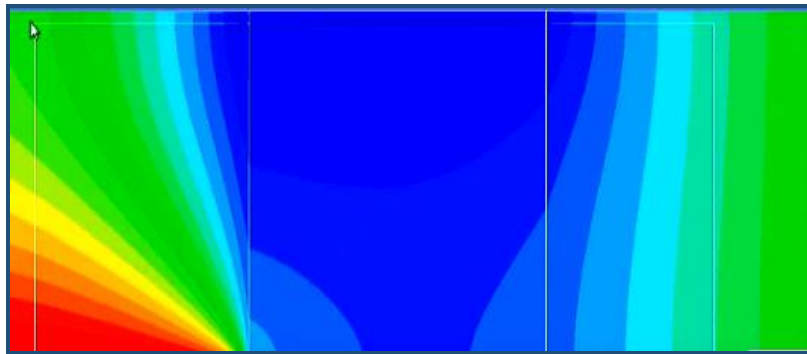


Figure 7.2: Temperature contour of composite material

Try:

1. Analysis of heat flowing through a composite wall subjected to constant end heat flux.
2. Analyze the heat flowing through a composite wall subjected to internal heat generation.
3. Analyze the heat flowing through a composite wall subjected to end convection and radiation.
4. Analyze the heat flowing through a wall using MATLAB.

8 CFD Analysis of 3D steady state heat flow problems

A composite wall is a structure made from two or more different materials joined together.

8.1 Analyze the heat flow through a 3D wall

Perform the ANSYS Simulation of heat flow through a wall of given material.

Hints:

1. Make the model of the domain in Ansys Design-modeler or import domain from other modeling software.
2. Discretize of domain using the Ansys workbench meshing.
3. Setup the physics in Ansys steady state thermal pre.
4. Run the Ansys solver until the convergence attained.

5. Analyze the results using the Ansys CFD post to get the contour as shown in Figure 8.1.

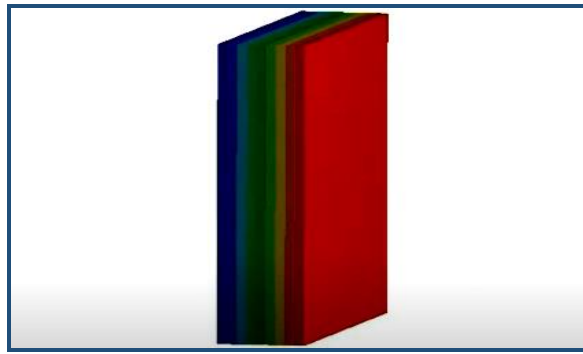


Figure 8.1: Temperature contour of a slab

8.2 Analyze the heat flow through a 3D wall

Perform the ANSYS Simulation of heat flow through a composite wall of given materials.

Hints:

1. Make the model of the domain in Ansys Design-modeler or import domain form other modeling software.
2. Discretize of domain using the Ansys workbench meshing.
3. Setup the physics in Ansys steady state thermal pre.
4. Run the Ansys solver until the convergence attained.
5. Analyze the results using the Ansys CFD post to get the contour as shown in Figure 8.2.

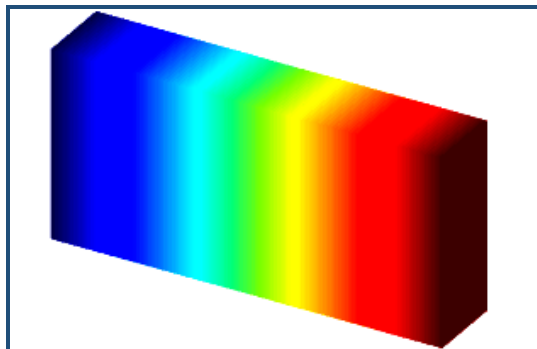


Figure 8.2: Temperature contour of composite wall

Try:

- 1 Analyze the heat flowing through a composite wall subjected to constant end heat flux.
- 2 Analyze the heat flowing through a composite wall subjected to internal heat generation.
- 3 Analyze the heat flowing through a composite wall subjected to end convection and radiation.

9 CFD analysis of natural convection problems

Natural convection, known also as free convection is a mechanism, or type of mass and heat transport, in which the fluid motion is only because of density differences in the fluid.

9.1 Analyze the heat flow from a hot rod

Perform the FLUENT Simulation of air flowing over the hot rod by natural convection.

Hints:

1. Make the model of the domain in Ansys Design-modeler or import domain form other modeling software.
2. Discretize of domain using the Ansys workbench meshing.
3. Setup the physics in Ansys Fluent.

4. Run the Ansys Fluent solver until the convergence attained.
5. Analyze the results using the Ansys CFD post to get the contour as shown in Figure 9.1.

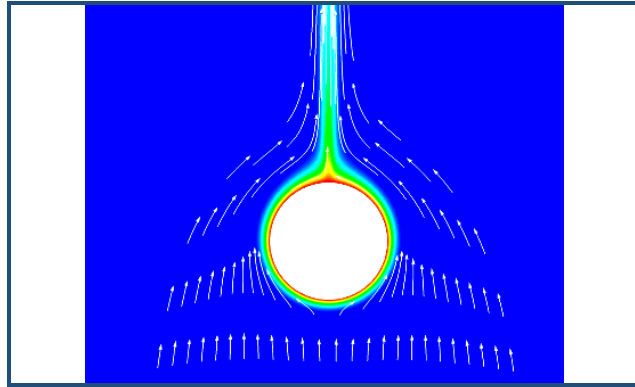


Figure 9.1: Temperature contour and velocity vectors from a rod

9.2 Analyze the heat flow from a circular fins

Perform the FLUENT Simulation of air flowing over the bank of hot rod by natural convection.

Hints:

1. Make the model of the domain in Ansys Design-modeler or import domain form other modeling software.
2. Discretize of domain using the Ansys workbench meshing.
3. Setup the physics in Ansys Fluent.
4. Run the Ansys Fluent solver until the convergence attained.
5. Analyze the results using the Ansys CFD post to get the contour as shown in Figure9.2.

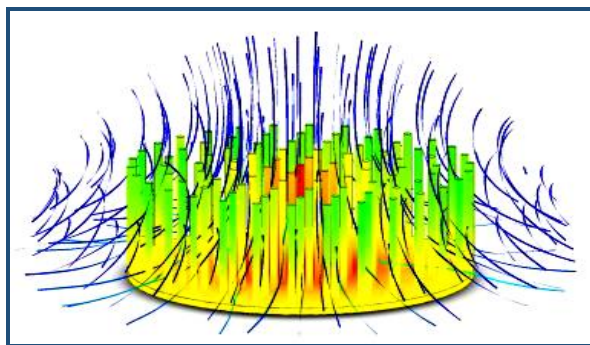


Figure 9.2: Temperature contour and velocity vectors from a set of circular fins

Try:

- 1 Analyze the heat transfer from a rod with longitudinal fins to the flowing air.
- 2 Analyze the heat transfer from a rot with circumferential fins to the flowing air.

10 CFD analysis of forced convection problems

Forced convection is a mechanism, or type of transport, in which fluid motion is by an external source.

10.1 Analyze the heat flow from a hot rod

Perform the FLUENT Simulation of air flowing over the hot rod by forced convection.

Hints:

1. Make the model of the domain in Ansys Design-modeler or import domain form other modeling software.
2. Discretize of domain using the Ansys workbench meshing.
3. Setup the physics in Ansys Fluent.
4. Run the Ansys Fluent solver until the convergence attained.

5. Analyze the results using the Ansys CFD post to get the contour as shown in Figure10.1.

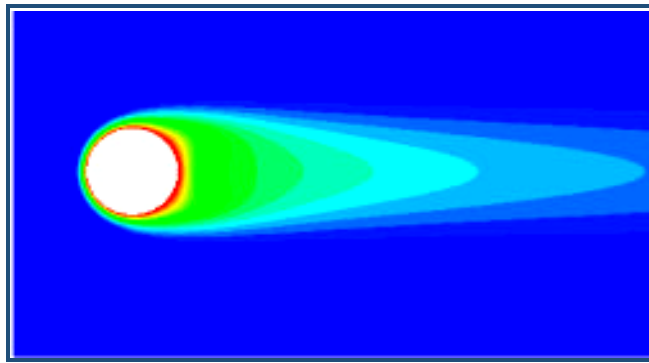


Figure 10.1: Temperature contour from a hot rod

10.2 Analyze the heat flow from a set of square fins

Perform the FLUENT Simulation of air flowing over the bank of hot rod by forced convection.

Hints:

1. Make the model of the domain in Ansys Design-modeler or import domain form other modeling software.
2. Discretize of domain using the Ansys workbench meshing.
3. Setup the physics in Ansys Fluent.
4. Run the Ansys Fluent solver until the convergence attained.
5. Analyze the results using the Ansys CFD post to get the contour as shown in Figure10.2.

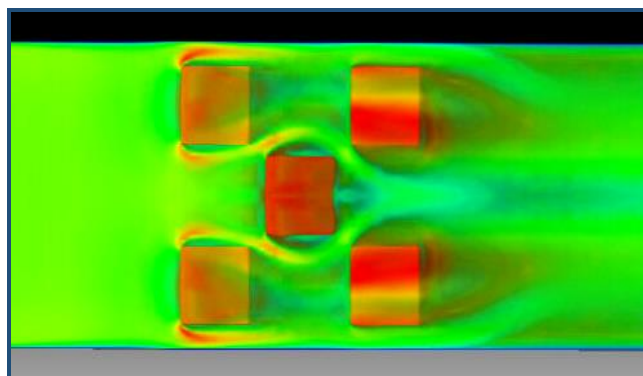


Figure 10.2: Temperature contour from a set of square fins

Try:

- 1 Analyze the heat transfer from a rod with longitudinal fins to the flowing air.
- 2 Analyze the heat transfer from a rot with circumferential fins to the flowing air.

11 CFD analysis of tube in tube heat exchanger problems

Heat exchangers play a vital role in various industries, facilitating the transfer of heat between two fluids while keeping them physically separate.

11.1 Analyze the heat flow through parallel flow heat exchanger

Perform the FLUENT Simulation of fluids flowing through tube in tube heat exchanger.

Hints:

1. Make the model of the domain in Ansys Design-modeler or import domain form other modeling software.
2. Discretize of domain using the Ansys workbench meshing.
3. Setup the physics in Ansys Fluent.
4. Run the Ansys Fluent solver until the convergence attained.

5. Analyze the results using the Ansys CFD post to get the contour as shown in Figure 11.1.

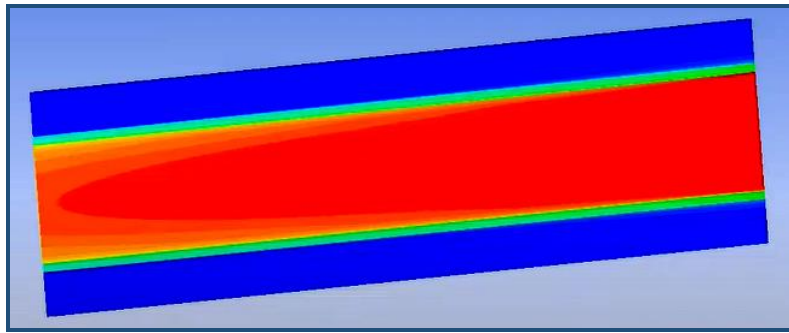


Figure 11.1: Temperature contour of parallel flow heat exchanger

11.2 Analyze the heat flow through counter flow heat exchanger

Perform the FLUENT Simulation of fluids flowing through tube in tube heat exchanger.

Hints:

1. Make the model of the domain in Ansys Design-modeler or import domain from other modeling software.
2. Discretize of domain using the Ansys workbench meshing.
3. Setup the physics in Ansys Fluent.
4. Run the Ansys Fluent solver until the convergence attained.
5. Analyze the results using the Ansys CFD post to get the contour as shown in Figure 11.2.

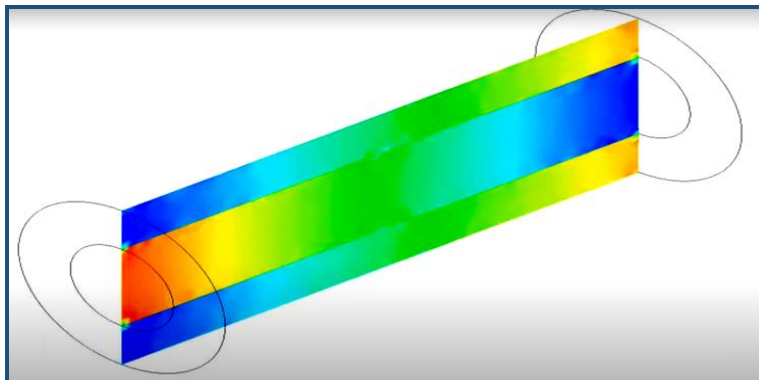


Figure 11.2: Temperature contour of counter flow heat exchanger

Try:

1. Analyze the heat flow through a two-pass tube in tube heat exchanger.
2. Analyze the heat flow through a multi pass tube in tube heat exchanger.

12 CFD analysis of shell and tube heat exchanger problems

A shell and tube exchanger consists of a number of tubes mounted inside a cylindrical shell.

12.1 Analyze the heat flow through parallel flow heat exchanger

Perform the FLUENT Simulation of fluids flowing through shell and tube heat exchanger.

Hints:

1. Make the model of the domain in Ansys Design-modeler or import domain from other modeling software.
2. Discretize of domain using the Ansys workbench meshing.
3. Setup the physics in Ansys Fluent.
4. Run the Ansys Fluent solver until the convergence attained.
5. Analyze the results using the Ansys CFD post to get the contour as shown in Figure 12.1.

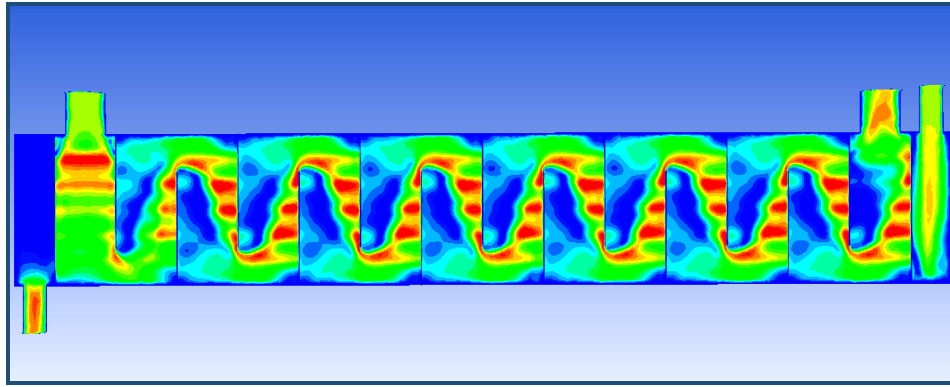


Figure 12.1: Temperature contour of parallel flow heat exchanger

12.2 Analyze the heat flow through counter flow heat exchanger

Perform the FLUENT Simulation of fluids flowing through shell and tube heat exchanger.

Hints:

1. Make the model of the domain in Ansys Design-modeler or import domain form other modeling software.
2. Discretize of domain using the Ansys workbench meshing.
3. Setup the physics in Ansys Fluent.
4. Run the Ansys Fluent solver until the convergence attained.
5. Analyze the results using the Ansys CFD post to get the contour as shown in Figure 12.2.

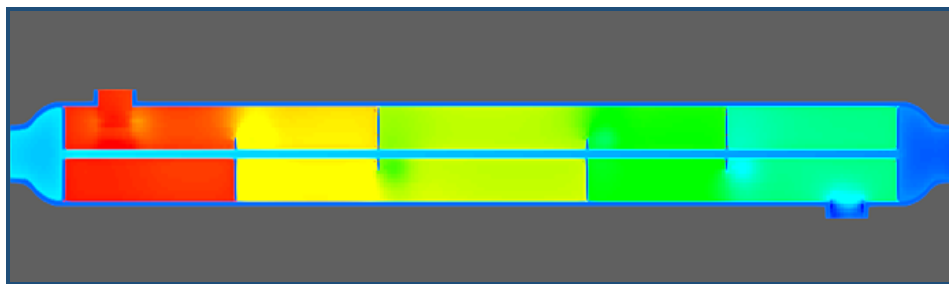


Figure 12.2: Temperature contour of counter flow heat exchanger

Try:

1. Analyze the heat flow through a two-pass shell and tube heat exchanger.
2. Analyze the heat flow through a multi pass shell and tube heat exchanger.

13 CFD analysis of flow through centrifugal pump

Centrifugal pumps are used to transport fluids by the conversion of rotational kinetic energy to the hydrodynamic energy of the fluid flow.

13.1 Analyze the flow through centrifugal pump with backward vanes

Perform the CFX Simulation of flow through centrifugal pump.

Hints:

1. Design the centrifugal pump in Ansys Vista CPD.
2. Make the model of the rotor domain in Ansys Blade Gen
3. Make the model of the casing in Ansys geometry.
4. Discretize the rotor domain using the Ansys Turbo Grid
5. Discretize the volute domain in workbench meshing.
6. Setup the physics in Ansys CFX pre.
7. Run the Ansys CFX solver until the convergence attained.

8. Analyze the results using the Ansys CFD post to get the contour as shown in Figure 13.1.

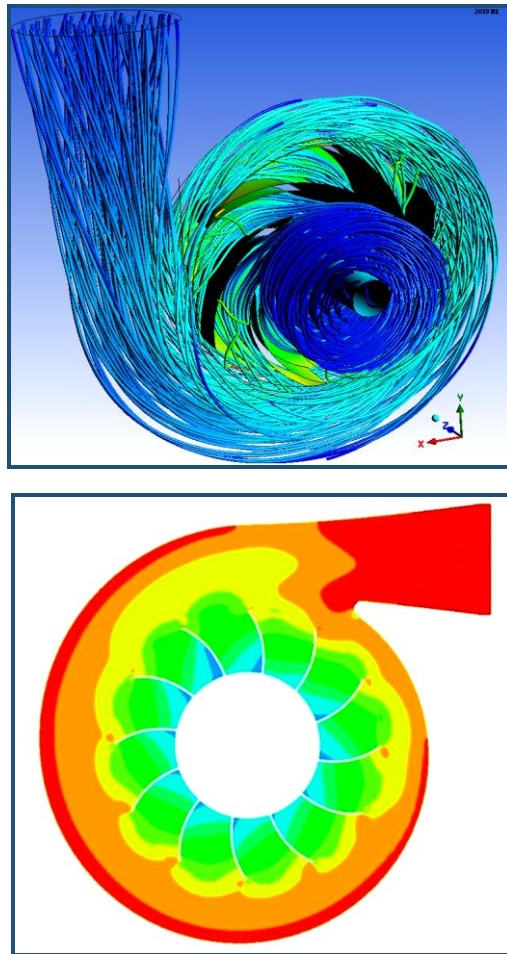


Figure 13.1: Velocity streamlines and pressure contour through centrifugal pump

Try:

- 1 Analyze the flow through a centrifugal pump with radial vanes.
- 2 Analyze the flow through a centrifugal pump with forward vanes.

14 CFD analysis of flow through radial turbines

A radial turbine is a turbine in which the flow of the working fluid is radial to the shaft.

14.1 Analyze the flow through radial turbine with backward vanes

Perform the CFX Simulation of flow through the radial turbines.

Hints:

1. Design the centrifugal pump in Ansys Vista RTD.
2. Make the model of the rotor domain in Ansys Blade Gen
3. Make the model of the casing in Ansys geometry.
4. Discretize the rotor domain using the Ansys Turbo Grid
5. Discretize the volute domain in workbench meshing.
6. Setup the physics in Ansys CFX pre.
7. Run the Ansys CFX solver until the convergence attained.
8. Analyze the results using the Ansys CFD post to get the contour as shown in Figure 14.1.

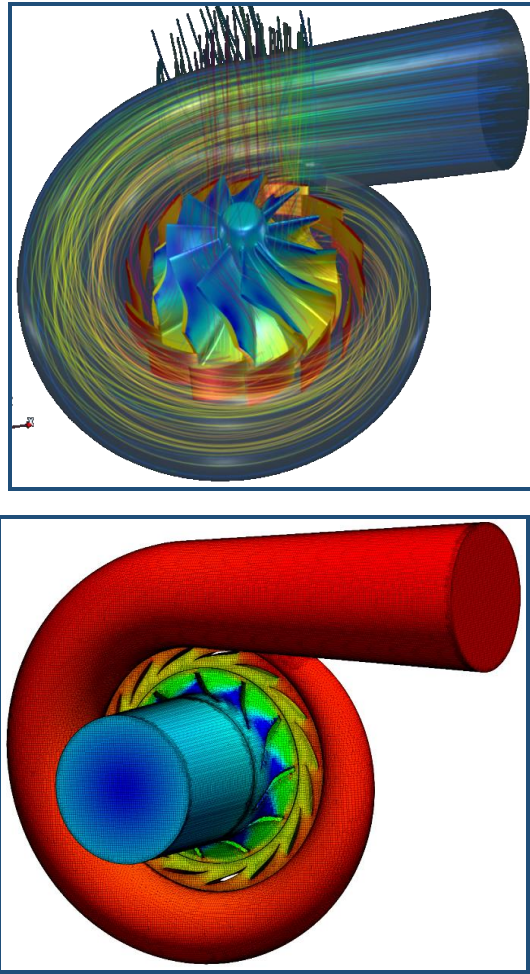


Figure 14.1: Velocity streamlines and pressure contour through radial turbine

Try:

- 1 Analyze the flow through a radial turbine with radial vanes.
- 2 Analyze the flow through a radial turbine with forward vanes.

V. TEXT BOOKS:

1. JiyanTu, Guan HengYeoh and Chaoqun Liu, “Computational Fluid Dynamics: A Practical Approach”, Elsevier Science, 2017
2. J. D. Anderson, (Jr), “Computational Fluid Dynamics”, McGraw-Hill Book Company, 1st edition, 2017.
3. K.A.Hoffman, and S. T. Chiang, “Computational Fluid Dynamics”, Vol. I, II and III, Engineering Education System, Kansas, USA, 2004.

VI. REFERENCE BOOKS:

1. Chung, T.J., “Computational Fluid Dynamics”, Cambridge University Press, 2003.
2. D. A. Anderson, J. C. Tannehill, and R. H. Pletcher, “Computational Fluid Mechanics and Heat Transfer”, CRC Press, 4th edition 2020.
3. Muralidhar K and Sundararajan. Computational Fluid Flow and Heat Transfer”, 2009.

VII. ELECTRONICS RESOURCES:

1. <https://courses.ansys.com/>
2. <https://www.udemy.com/course/ansys-tutorial/>

VIII. MATERIALS ONLINE:

1. Course Template
2. Lab manual