Bharath

INSTITUTE OF HIGHER EDUCATION AND RESEARCH

Declared as **Deemed-to-be-University** u/s 3 of the UGC Act, 1956

B.Tech – Aerospace Engineering

U20ASSJ10 – Numerical Heat Transfer

Lab Manual

Vision of the Institute

"Bharath Institute of Higher Education & Research (BIHER) envisions and constantly strives to provide an excellent academic and research ambience for students and members of the faculties to inherit professional competence along with human dignity and transformation of community to keep pace with the global challenges so as to achieve holistic development."

Mission of the Institute

- To develop as a Premier University for Teaching, Learning, Research, and Innovation on par with leading global universities.
- To impart education and training to students for creating a better society with ethics and morals.
- To foster an interdisciplinary approach in education, research and innovation by supporting lifelong professional development, enriching knowledge banks through scientific research, promoting best practices and innovation, industry driven and institute-oriented cooperation, globalization and international initiatives.
- To develop as a multi-dimensional institution contributing immensely to the cause of societal advancement through spread of literacy, an ambience that provides the best of international exposures, provide health care, enrich rural development and most importantly impart value based education.
- To establish benchmark standards in professional practice in the fields of innovative and emerging areas in engineering, management, medicine, dentistry, nursing, physiotherapy and allied sciences.
- To imbibe human dignity and values through personality development and social service activities.

B.Tech- Aerospace Engineering

Vision of the Department

Department of Aeronautical Engineering will endeavor to accomplish worldwide recognition with a focal point of Excellence in the field of Aeronautics by providing quality Education through world class facilities, enabling graduates turning out to be Professional Experts with specific knowledge in Aeronautical & Aerospace engineering.

Mission of the Department

- To be the state of art Teaching and Learning center with excellent infrastructure and empowered Faculties in Aeronautical & Aerospace Engineering.
- To foster a culture of innovation among students in the field of Aeronautics and Aerospace with updated professional skills to enhance research potential for sponsored research and innovative projects.
- To Nurture young individuals to be knowledgeable, skillful, and ethical professionals in their pursuit of Aeronautical & Aerospace Engineering.

B.Tech- Aerospace Engineering

Program Educational Objectives Statements (PEO)

PEO 1: Demonstrate a solid grasp of fundamental concepts in Mathematics, Science, and Engineering, essential for effectively addressing engineering challenges within the Aerospace industry.

PEO 2: Involve in process of designing, simulating, fabricating, testing, and evaluating in the field of Aerospace.

PEO 3: Obtain advanced skills to actively engage in research and development endeavors within emerging domains, while also pursuing further education opportunities.

PEO 4: Demonstrate efficient performance both as independent contributors and as valuable team members in diverse multidisciplinary projects.

PEO 5: Embrace lifelong learning and career advancement while adapting to the evolving social demands and needs.

B.Tech- Aerospace Engineering

Programme Outcomes (PO's)

PO1: Engineering knowledge: Apply the knowledge of mathematics, science, engineering fundamentals, and Engg. Specialization to the solution of complex engineering problems.

PO2: Problem analysis: Identify, formulate, research literature, and analyze engineering problems to arrive at substantiated conclusions using first principles of mathematics, natural, and engineering sciences.

PO3: Design/development of solutions: Design solutions for complex engineering problems and design system components, processes to meet the specifications with consideration for the public health and safety, and the cultural, societal, and environmental considerations.

PO4: Conduct investigations of complex problems: Use research-based knowledge 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 assess societal, 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 solutions in 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 teamwork: Function effectively as an individual, and as a member or leader in teams, and in multidisciplinary settings.

PO10: Communication: Communicate effectively with the engineering community and with society at large. Be able to comprehend and write effective reports documentation. Make effective presentations and give and receive clear instructions.

PO11: Project management and finance: Demonstrate knowledge and understanding of engineering and management principles and apply these to one's own work, as a member and leader in a team. Manage projects 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.

B.Tech- Aerospace Engineering

Program Specific Outcomes (PSO) - R2020

PSO1: Design and analyze aerospace components/systems for aerospace industries.

PSO2: Acquire the concepts of spacecraft attitude dynamics for the prediction of spacecraft motion.

Course Outcomes (COs)

CO1	Identify different CFD techniques available for relevant partial differential equations to **predict** analytical solutions for fluid flow. **(Apply)**
CO2	**Select** the type of flow from the finite control volume and infinitesimal small fluid element for the fluid flow analysis. **(Analyze)**
CO3	**Develop** the governing equations for computational fluid dynamics CFD analysis by setting appropriate boundary conditions. **(Apply)**
CO4	**Carry out** modelling, meshing and pre-processing as per the problem statement**. (Imitation)**
CO5	**Acquire** data using the simulation tools from software**. (Manipulation)**
CO6	**Perform** basic post-processing techniques and represent the results in form of graph and table **(Precision)**

Mapping/Alignment of Cos with PO & PSO

	PO1	PO2	PO3	PO4	PO5	PO6	PO7	PO8	PO9	PO10	PO11	PO12	PSO1	PSO2
CO1	3	-	-	-	-	-	-	1	-	2	-	2	3	-
CO2	3	3	-	-	-	-	-	1	-	-	-	2	3	-
CO3	3	-	-	-	-	-	-	1	-	-	-	2	3	-
CO4	3	-	-	-	3	-	-	1	3	3	-	3	3	-
CO5	3	-	-	-	3	-	-	1	3	3	-	3	3	-
CO6	3	-	-	-	3	-	-	1	3	3	-	3	3	-

List of Experiments

1. Model the laminar flow over a flat plate. 2. Model the turbulent flow over a flat plate. 3. Model the velocity boundary layer for internal flow. 4. Model the convective boundary layer for internal flow. 5. Model the flow over an airfoil moving at Mach 0.3, through air, at tropopause. 6. Model the flow over a wedge/cone moving at Mach 1.5, through air, at an altitude of 15 km from the sea level.	**CO4, CO5, CO6**

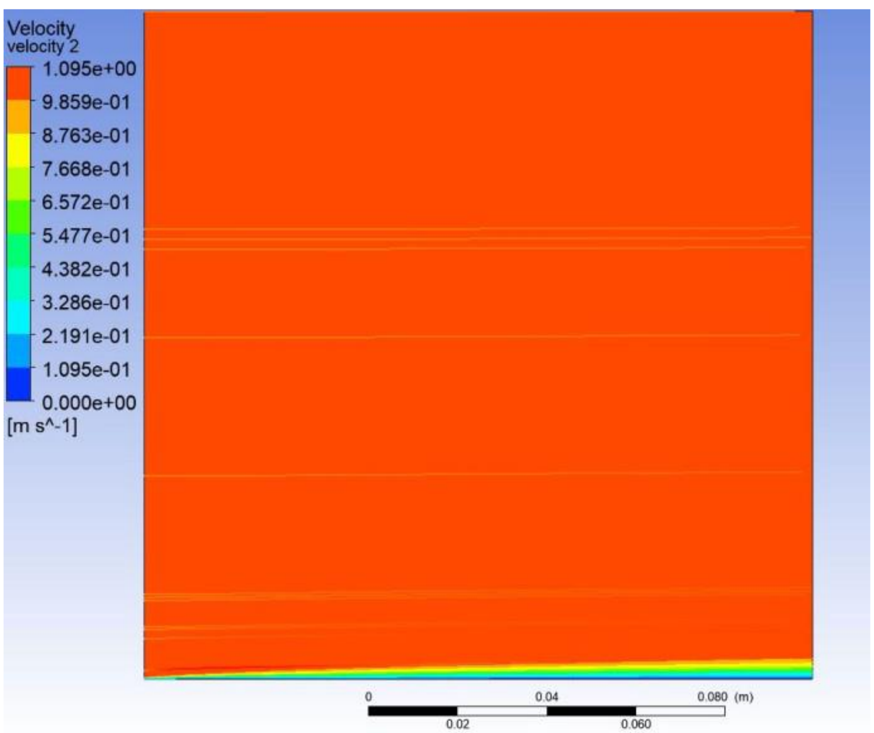
Exp:1	**Model the laminar flow over a flat plate**

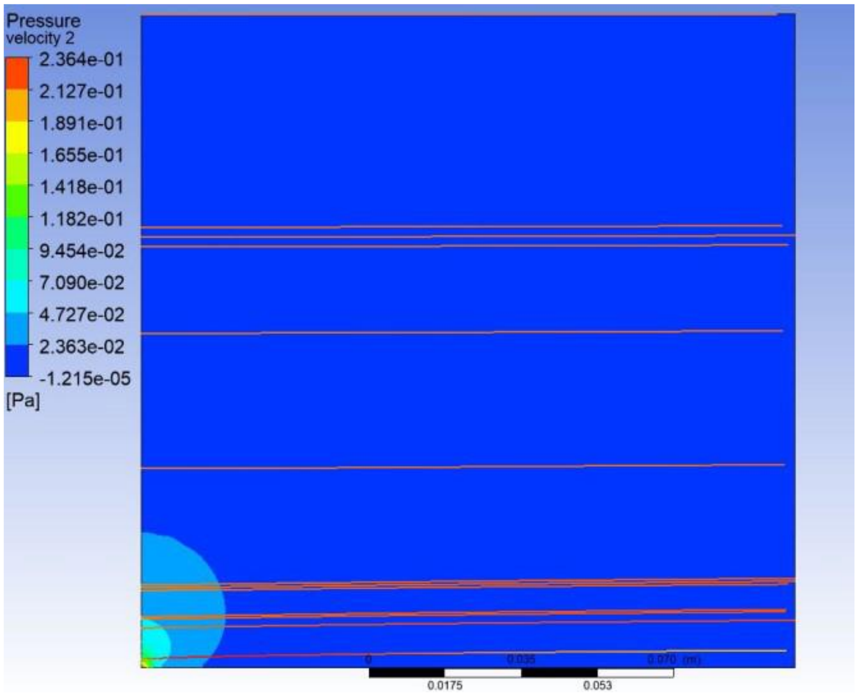
Aim: To Model the laminar flow over a flat plate using ansys Workbench

SOFTWARE USED: ANSYS FLUENT

Steps Involved:

1) **Geometry design of the flat plate**
2) **Constraining of the obtained geometry**
3) **Selection of method in mesh- no of divisions-element size**
4) **Providing Inflation with appropriate no of divisions and bias**
5) **Providing the cell zone conditions and boundary conditions**
6) **Initialization –Hybrid type**
7) **Proving No of iterations and run calculation**
8) **Exploring the results in the form of contours and plots.**
9) **Pressure ,velocity can be calculated from the results obtained**





Results:

Thus the flow over the laminar flat plate has been done using Ansys Fluent and the velocity and pressure have been calculated

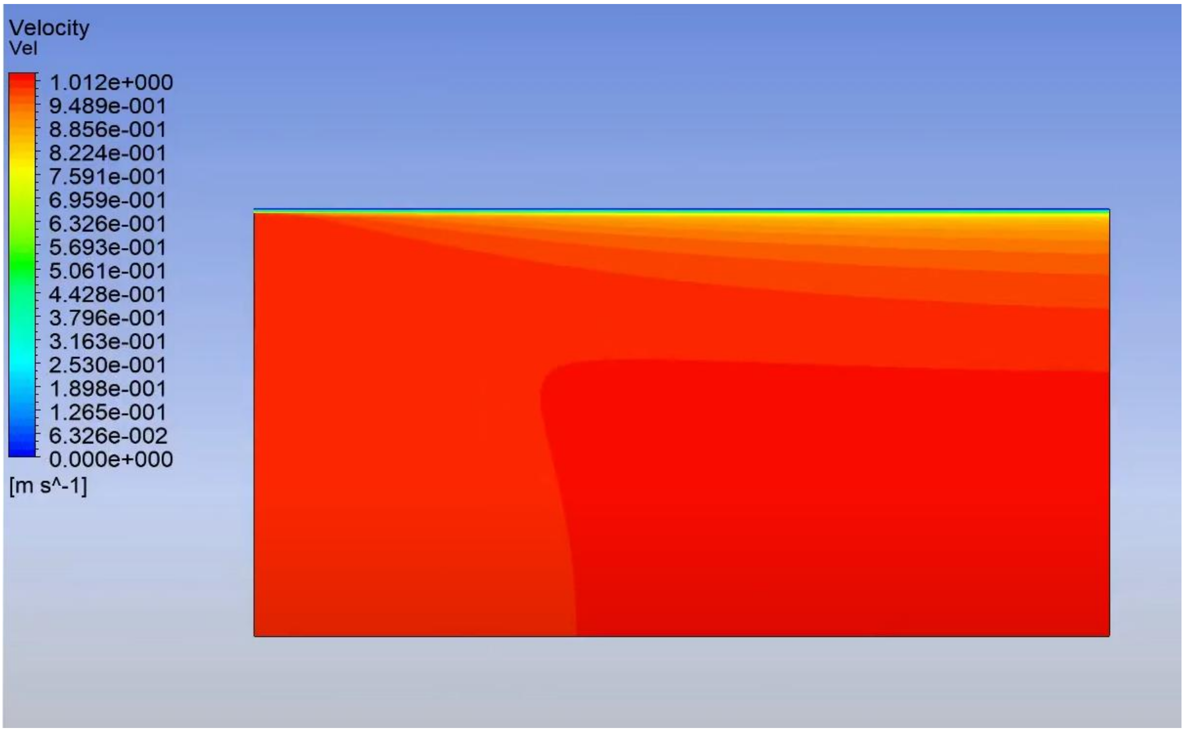
Exp:2 Model the turbulent flow over a flat plate.

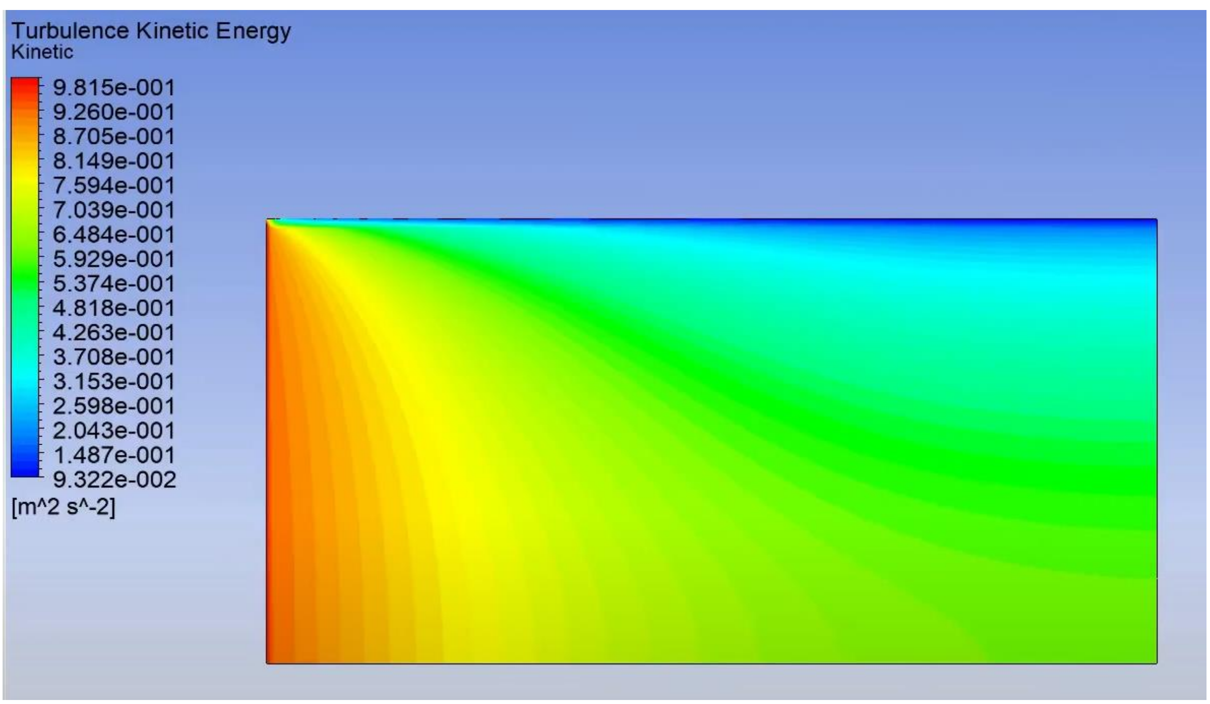
Aim: To Model the turbulent flow over a flat plate using ansys Workbench

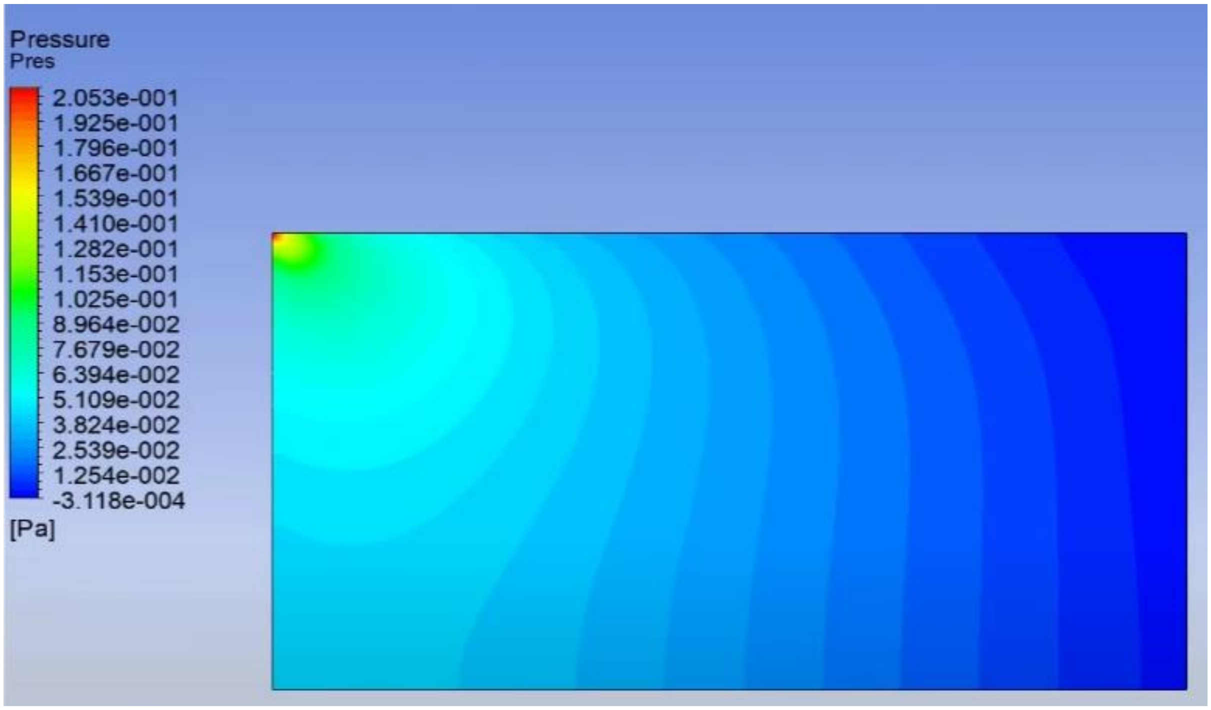
SOFTWARE USED: ANSYS FLUENT

Steps Involved:

1) **Geometry design of the flat plate**
2) **Constraining of the obtained geometry**
3) **Selection of method in mesh- no of divisions-element size**
4) **Providing Inflation with appropriate no of divisions and bias**
5) **Turn on Energy Equation and select k-epsilon turbulence model.**
6) **Providing the cell zone conditions and boundary conditions**
7) **Initialization –Hybrid type**
8) **Proving No of iterations and run calculation**
9) **Exploring the results in the form of contours and plots.**
10) **Pressure ,velocity can be calculated from the results obtained**
11) **The turbulent kinetic energy is calculated from the results**


Velocity
Vel
1.012e+000
9.489e-001
8.856e-001
8.224e-001
7.591e-001
6.959e-001
6.326e-001
5.693e-001
5.061e-001
4.428e-001
3.796e-001
3.163e-001
2.530e-001
1.898e-001
1.265e-001
6.326e-002
0.000e+000
[m s^-1]


Turbulence Kinetic Energy
Kinetic
9.815e-001
9.260e-001
8.705e-001
8.149e-001
7.594e-001
7.039e-001
6.484e-001
5.929e-001
5.374e-001
4.818e-001
4.263e-001
3.708e-001
3.153e-001
2.598e-001
2.043e-001
1.487e-001
9.322e-002
[m^2 s^-2]



Results:

Thus the turbulent flow over the flat plate has been done using Ansys Fluent and the velocity ,pressure ,turbulence kinetic energy have been calculated.

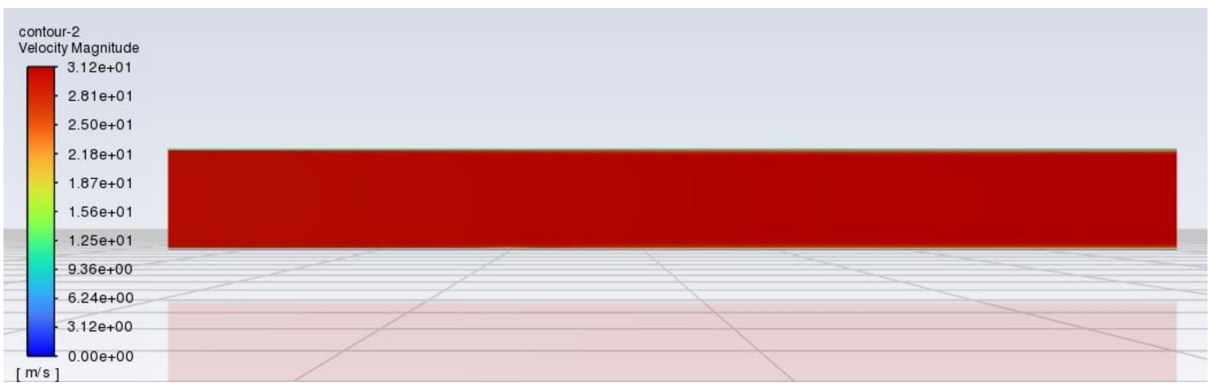
Exp:3 Model the velocity boundary layer for internal flow

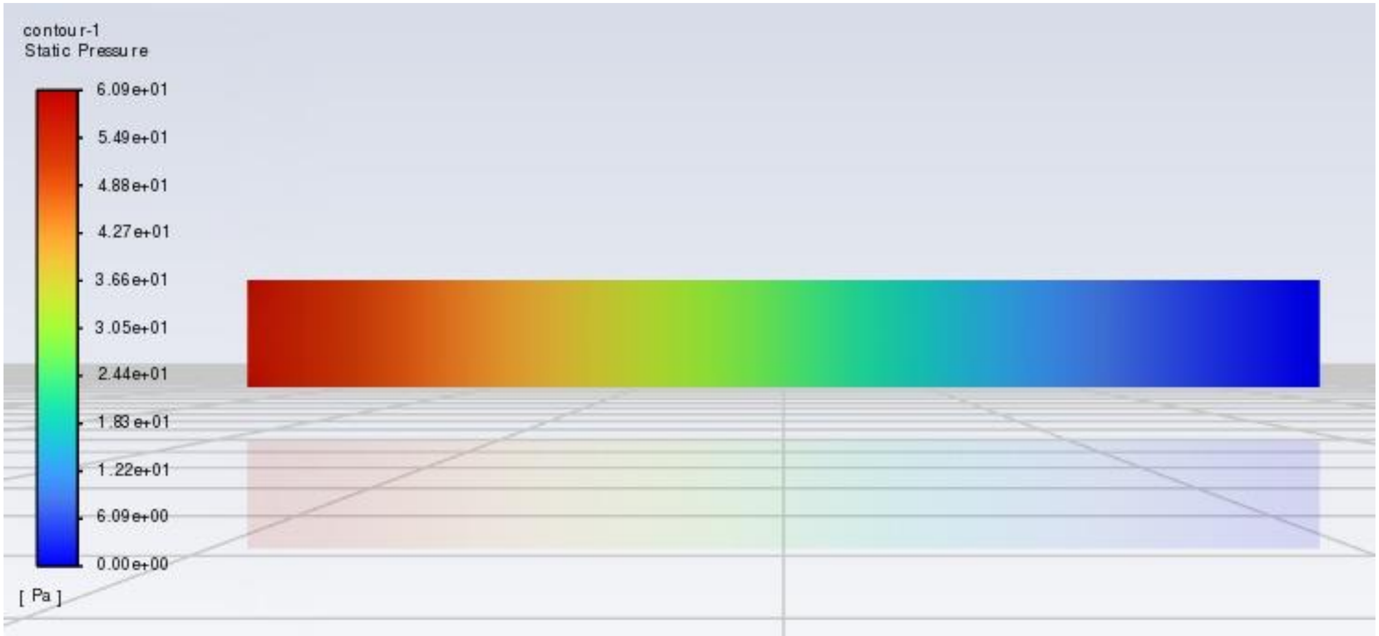
Aim: To determine the flow in a cylindrical pipe or any cross section using ansys Workbench

SOFTWARE USED: ANSYS FLUENT

Steps Involved:

1) **Geometry design of the cylindrical pipe**
2) **Constraining of the obtained geometry**
3) **Selection of method in mesh- no of divisions-element size**
4) **Turn on Energy Equation and select k-epsilon turbulence model.**
5) **Providing the cell zone conditions and boundary conditions**
6) **Initialization –Hybrid type**
7) **Proving No of iterations and run calculation**
8) **Exploring the results in the form of contours and plots.**
9) **Pressure ,velocity can be calculated from the results obtained**





Results:

Thus the internal flow over the cylinder has been done using Ansys Fluent and the velocity ,pressure have been calculated

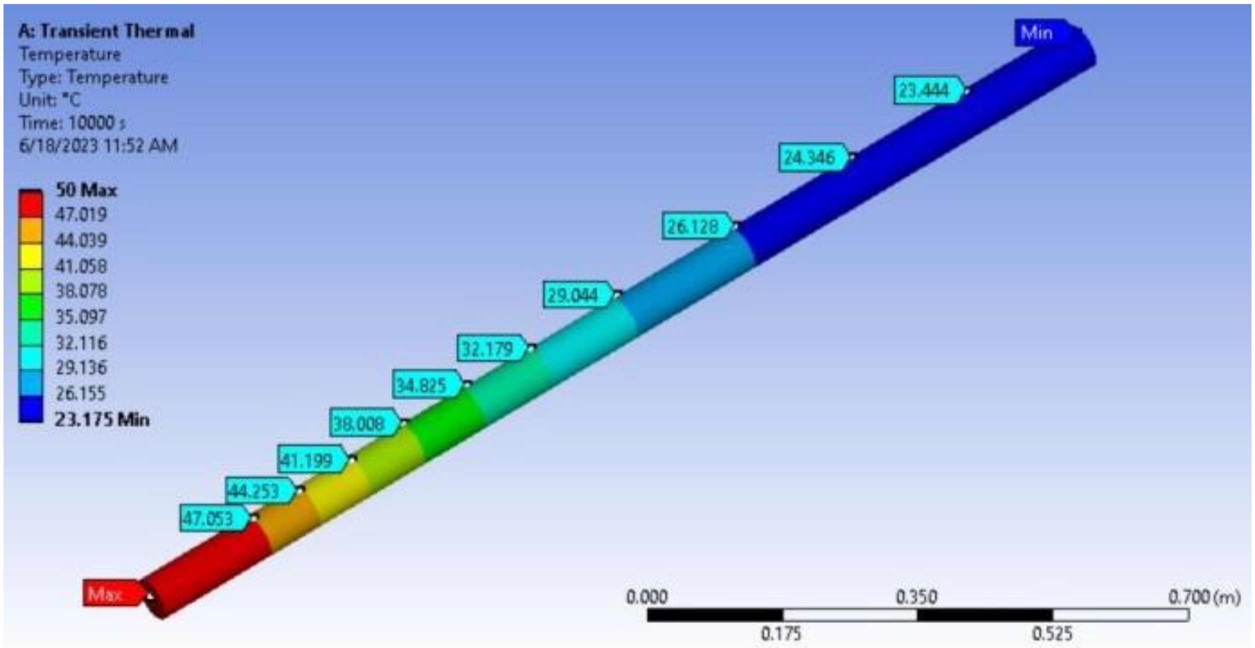
Exp : 4 Model the convective boundary layer for internal flow.

Aim: To determine the flow in a cylindrical pipe or any cross section using ansys Workbench

SOFTWARE USED: ANSYS FLUENT

Steps Involved:

1) **Geometry design of the cylindrical pipe**
2) **Constraining of the obtained geometry**
3) **Selection of method in mesh- no of divisions-element size**
4) **Turn on Energy Equation and select k-epsilon turbulence model.**
5) **Providing the cell zone conditions and boundary conditions(convective heat transfer)-thermal**
6) **Initialization –Hybrid type**
7) **Proving No of iterations and run calculation**
8) **Exploring the results in the form of contours and plots.**
9) **Pressure ,velocity can be calculated from the results obtained**



Results:

Thus the internal flow over the cylinder with convective heat transfer has been done using Ansys Fluent , the results of temperature distribution is calculated

Exp:5 Flow over an airfoil moving at Mach 0.3, through air, at tropopause.

AIM: To simulate flow over NACA 0012 airfoil

SOFTWARE USED: ANSYS WORKBENCH

Problem description:

Consider air flowing over NACA 0012 airfoil. The free stream Mach number is 1.2. Assume standard sea-level values for the free stream properties: Pressure = 101,325 Pa Density = 1.2250 kg/m3

Temperature = 288.16 K

Kinematic viscosity v = 1.4607e-5 m^2/s

Steps Involved In CFD:

Creation of Geometry : Importing the Aerofoil coordinates File→Import Geometry→Formatted point data→Select the file of aerofoil coordinates which is in DAT format→ok. Now the coordinates will be displayed.

Geometry→Create/modify curve→From points→Select above points and leave last 2 points→middle click

Similarly on bottom side

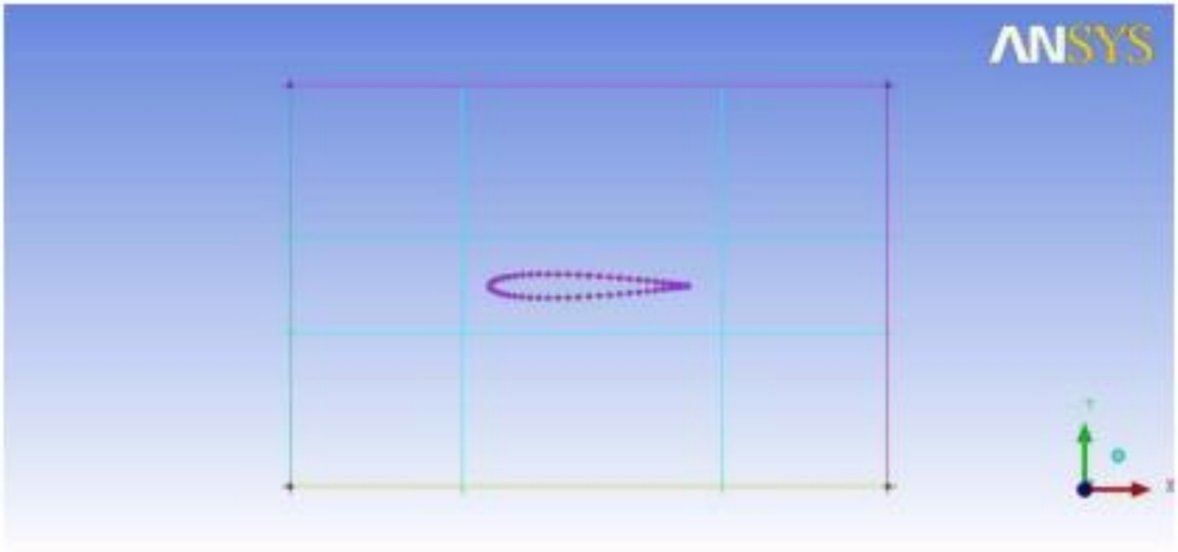
Join the end points of the curves

1) **Creation of parts:**
   - Parts in the tree→Right click→Create part→

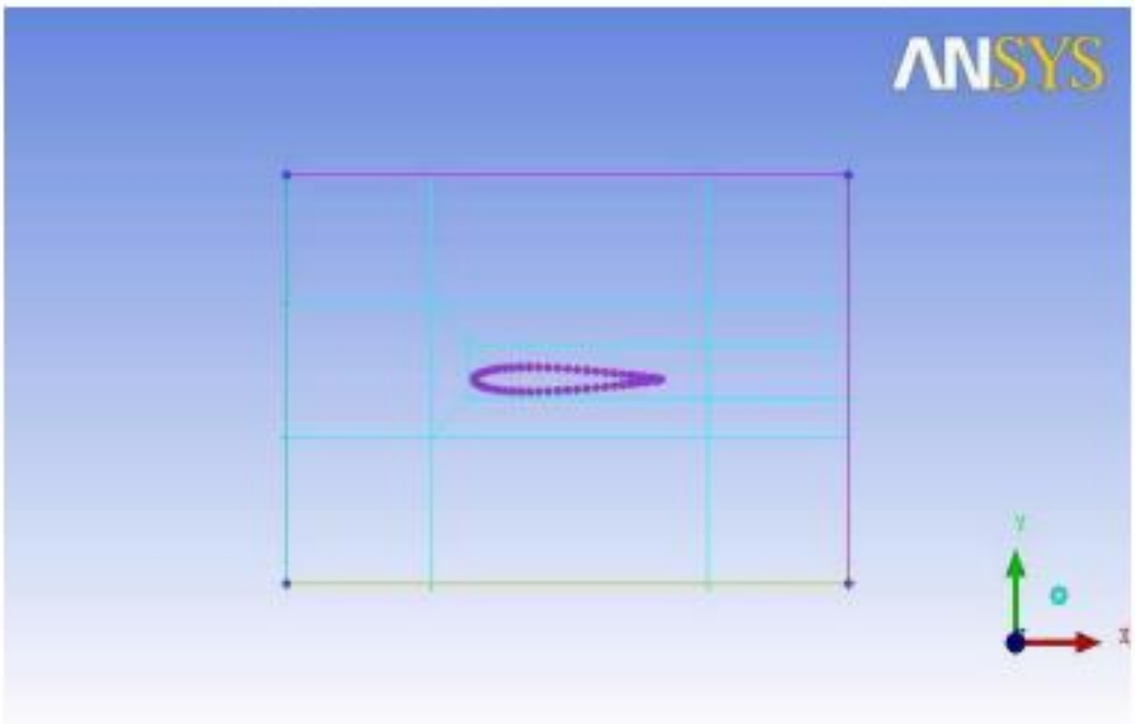
     Select Upper curve: Suction

     Select Lower curve: Pressure

     3rd Line: TE
2) **Creation of Domain:**
   - Create points (-1,1),(-1,-1),(2,1),(2,-1)
   - Join these points
   - Create parts as Inlet, Outlet, Top & Bottom
   - Geometry→Create/Modify surface→Simple surface→Select all the lines of domain→ok
   - Create the new part as: Surface
3) **Saving the Geometry:**
   - File→Change working directory→Choose the folder
   - File→Geometry→Save Geometry as→Give the name.



Thus the O grid has been generated as shown in below fig.



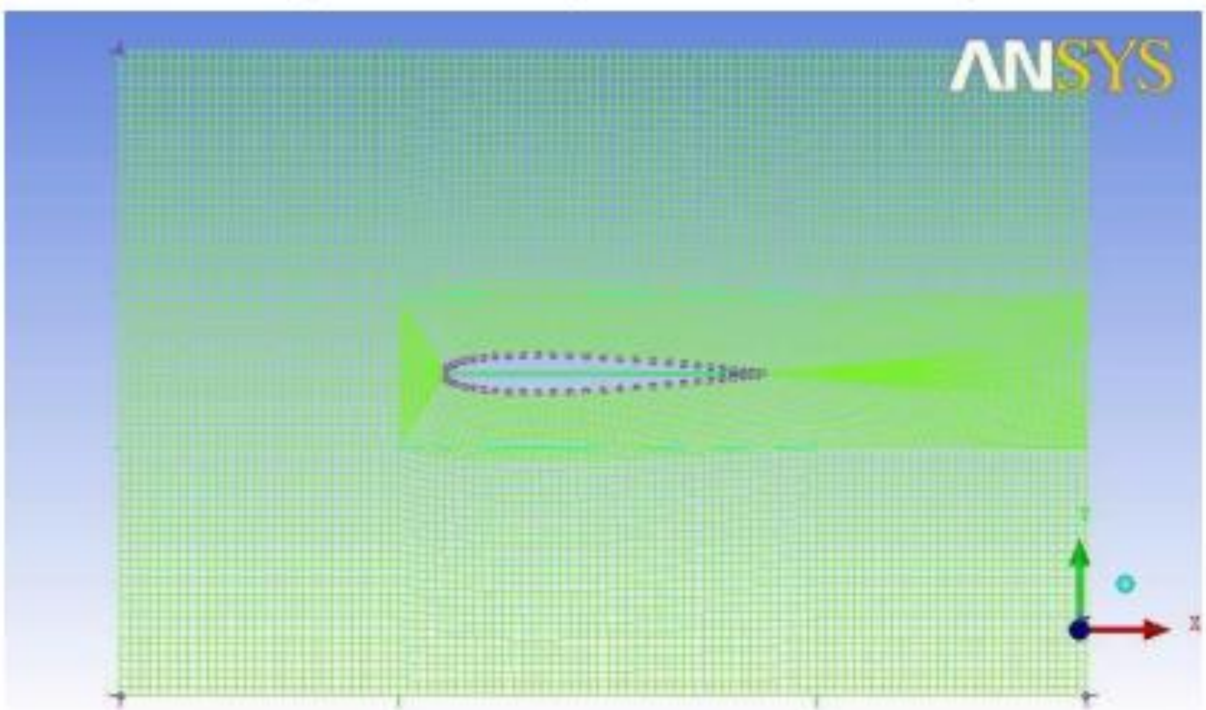
4) Creation of Blocking and Association:

- Blocking→Create block→Initialize blocks→Type as:2D Planar→ok
- Associate→Associate vertex to point→Select a vertex and a point→Apply→
  Similarly associate remaining 3 vertices to points
- Associate→Associate edge to curve→Select a edge and a curve →Apply→
  Similarly associate remaining 3 edges to curves
- Split block→Select the edges→Create the blocks as shown in figure

5) Generation of Mesh:

- Pre-mesh parameters→Edge parameters→Switch ON the Copy Parameters→
  Select the edges and give desired no. of nodes→ok
- Switch ON Pre-mesh in the tree→click yes to compute the meshing
- Pre-mesh→Right click→Convert to unstructured mesh

  Now the required mesh has been generated as shown in below fig.



Problem setup:

General→Type as: Pressure based

Models→Energy ON→Viscous-laminar ¬Materials→Air

Cell zone conditions→Type as: fluid→ok

Boundary conditions→Select inlet→Edit→Give velocity magnitude as: 400m/s.

Boundary conditions→Select outlet→Edit→Give gauge pressure as: 0 Pa

Results:

In Graphics and animations→select the required flow parameters in the contours and vectors for obtaining the required results.

Exp : 5

Model the flow over a wedge/cone moving at Mach 1.5, through air, at an altitude of 15 km from the sea level.

AIM: To model the flow over a wedge/cone moving at Mach 1.5, through air, at an altitude of 15 km from the sea level.

SOFTWARE USED: ANSYS WORKBENCH

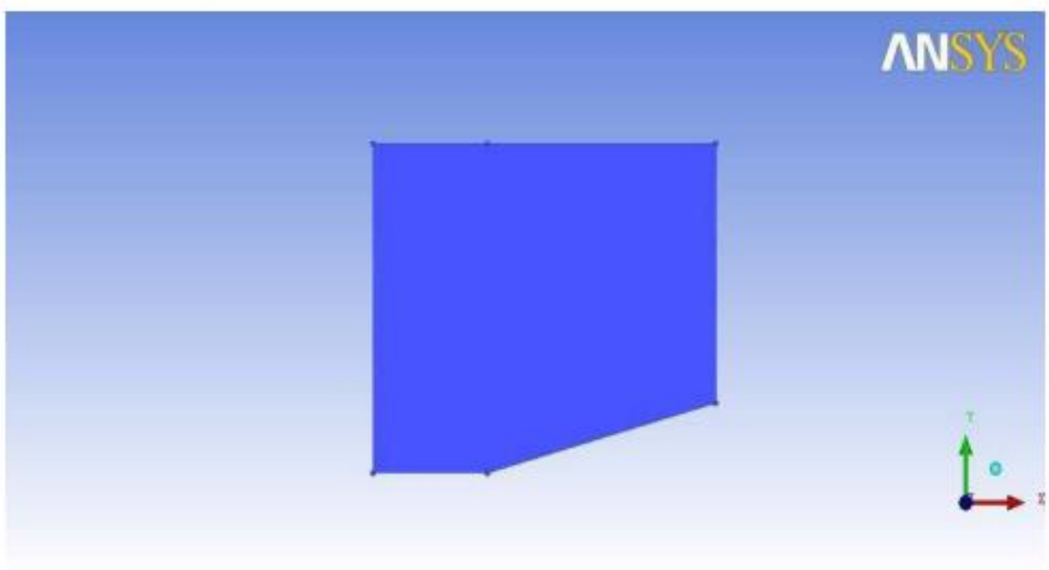
Creation of Geometry

- Geometry→Create point→Explicit coordinates→Enter the coordinates as given in table shown:

X	0	0	0.5	1.5	1.5	0.5
Y	0	1.259	1.259	1.259	0268	0
Z	0	0	0	0	0	0

Geometry→Create/modify curve→From points→Select any 2 points→ok→Similarly create the curves to all points

- Geometry→Create/Modify surface→Simple surface→Select all the lines of domain→ok



Creation of parts:

Parts in the tree→Right click→Create part→

Select Left curve: Inlet

Select Right curve: Outlet

Select Top curve: Top

Select inclined curve: Wedge

Select bottom curve: Front_wedge

Saving the Geometry:

File→Change working directory→Choose the folder

File→Geometry→Save Geometry as→Give the name.

Creation of Blocking and Association:

Blocking→Create block→Initialize blocks→Type as:2D Planar→ok

Associate→Associate vertex to point→Select a vertex and a point→Apply→

Similarly associate remaining 3 vertices to points

Associate→Associate edge to curve→Select a edge and a curve →Apply→

Similarly associate remaining 3 edges to 5 curves

Generation of Mesh:

Pre-mesh parameters→Edge parameters→Switch ON the Copy Parameters→

Select the Horizontal edge and give no. of nodes as: 100→ok

Pre-mesh parameters→Edge parameters→Switch ON the Copy Parameters→

Select the Vertical edge and give no. of nodes as: 100→Spacing as:

0.001→Ratio as: 1.1→ok

Model the laminar flow over a flat plate

AIM:

To model the laminar flow over a flat plate.