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

U20ASCJ07 - Computational Methods in Aerospace Engineering

Program: Bachelor of Technology (B. Tech.)

Branch: Aerospace Engineering

U20ASCJ07
Computational Methods in Aerospace Engineering

TABLE OF CONTENTS

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

Program Outcomes (PO's)

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, 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.

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.

U20ASCJ07 Computational Methods in Aerospace Engineering

Part A — **Introduction** of **the** Course

This course introduces students to the concepts of computational mechanics and application of commercial software to solve aerospace engineering problems.

Course Code	Course Category	Course Title	L	T	P	C
			1	0	2	2
U20ASCJ07	C	Computational Methods in Aerospace Engineering	Pre-requisite: U20MEEJ01 Engineering Graphics and Design U20CSEJ01 — Programming and Problem Solving U20MABT04 Numerical Methods for Engineers U20ASCJ03- Low and High speed Aerodynamics U20ASCJ04- Advanced Aerospace Structures U20ASCJ05- Aerospace Propulsion			
Name of the Course Coordinator		Mr. M.K. Karthik	Contact Hrs: 45			
Course offering Dept./Sc1ioo1		Department of Aeronautical Engg.	Total Marks: 100			

Course Objective and Summary

This course will make students

1. To introduce students about theoretical basics and practical application of Computational techniques.
2. To design and solve practical engine ring problems related to solid mechanics, heat transfer and aerodynamics.

Course Outcomes (CO's)

CO1	Explain the concepts of f1ui5 structure interaction, aeroelastic phenomenon and numerical solutions of aerodynamics problems. **(Understand)**
CO2	**Describe** the process of performing finite element analysis on structural elements to obtain stress and strain. (**Understand)**
CO3	Discuss the basics of heat transfer problems and its numerical solutions. **(Understand)**
CO4	**Carry out** modelling, meshing and pre-processing as per the problem statement **(Imitation)**
COS	**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)**

Flapping/Alignment of COs with PO & PSO

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

(Tick mark or level of correlation: 3-High, 2-Mediuin,1-Low)

Part B — Content of the Course

Sl. No.	SUAI6IARY OF COURSE CONTENT	Hrs	Alignment to COs
1	**Topic 1: Basics of CFD and Fluid Structure Interaction** Review of Aerodynamics and introduction to CFD; Coupling fluid and solid mechanics; Dimensional analysis of interaction Dimensionless coupled equation; Aeroelastic approximation — Flow induced dynamic instability.	5	CO1
	Lab experiment: Flow over an airfoil; Flow over a flexible 3D wing and its deformation – one-way FSI.	10	CO4, CO5, CO6
2	**Topic 2: Structural Analysis** Review of structural analysis; Introduction to FEA- preprocessing-mathematical modeling, Geometry and mesh creation, solver-Discretization method (Basics) and post processing- Contours, vectors, plots, streamlines, Residuals.	5	CO2
	Lab experiment: Structural analysis using 1D element - Truss problem, Structural analysis using 2D element - Plate with hole problem, Stress analysis of plate/beam with hole using 3D element	10	CO4, CO5, CO6
3	**Topic 3: Thermal Analysis** Basic approach in Numerical Heat conduction, one dimensional steady state problem, Introduction to Convection diffusion, Thermal boundary layer flow, transient free convection.	5	CO3
	Lab experiment: ID transient heat conduction - thermal analysis of a rod, 2D steady state heat transfer analysis of a plate	10	CO4, CO5, CO6

Part C- Assessment and Evaluation

1. Assessment Strategy (Bloom's Taxonomy and Rubric based: CLA-1, CLA-2, CLA-3, Assignment, Semester Final Exam)
2. Evaluation Policy (Grading System)

Part D - Learning Resources

Text Book:

1. Reddy. JN, An Introduction to finite element method, Third Edition, 2005, Tata Mc Graw Hill.
2. Versteeg H, Malalasekara W, An Introduction to Computational Fluid Dynamics: The Finite Volume Method, Second Edition, 2007, Pearson India.
3. Blevins RD. Flow-induced vibration. New York. 1977.
4. Suhas V Patankar, Numerical Heat Transfer and Fluid Flow, 2009, Hemisphere Publishing Corporation.

Reference Books:

1. Muralidhar K and Sundararajan T, Computational Fluid flow and Heat transfer, second edition, 2008, Narosa Publishing House, Newdelhi.
2. Ghoshdasdidar PS, Computer Simulation of fluid flow and heat transfer, 1998, Tata McGraw Hill Publishing Company Ltd, India.
3. Tirupathi R Chandrupatla, Ashok D Belegundu, Introduction to Finite Elements in Engineering, Fourth Edition, 2012, Pearson India.
4. Mc Cornick B. W, “Aerodynarnics, Aeronautics & Flight Mechanics”, John Wiley, NY, 1995.

Other Resources (Online Resources or others)

- https://mitocw.ups.edu.ec/courses/aeronautics-and-astronautics/16-90-coinputationa1-methods-in-aerospace-engineering-spring-2014/#

Experiment No: 1 DATE:	**Flow over an Airfoil**

Aim: -

To determine the pressure and velocity distribution around an airfoil geometry using ANSYS Fluent software.

Computer System Configuration:

- Processor: 64-bit operating System Intel Multi Core
- RAM: 8 GB
- Operation System: Windows 11
- Software: ANSYS Fluent
- Software: MS Excel

Procedure:

Airfoil Coordinates for geometry – MS Excel

- Download any Airfoil coordinates file from airfoil tool Database.
- Arrange the coordinates and add Z-Coordinates with value 0 for every point and remove the last point
- Add point numbers and a reference point value of 1 for each point
- Save the file as a text format
- Open ANSYS workbench → Drag and drop Fluent Workflow in Workspace
- Right click on Geometry → Select New Design Modular

Geometry:

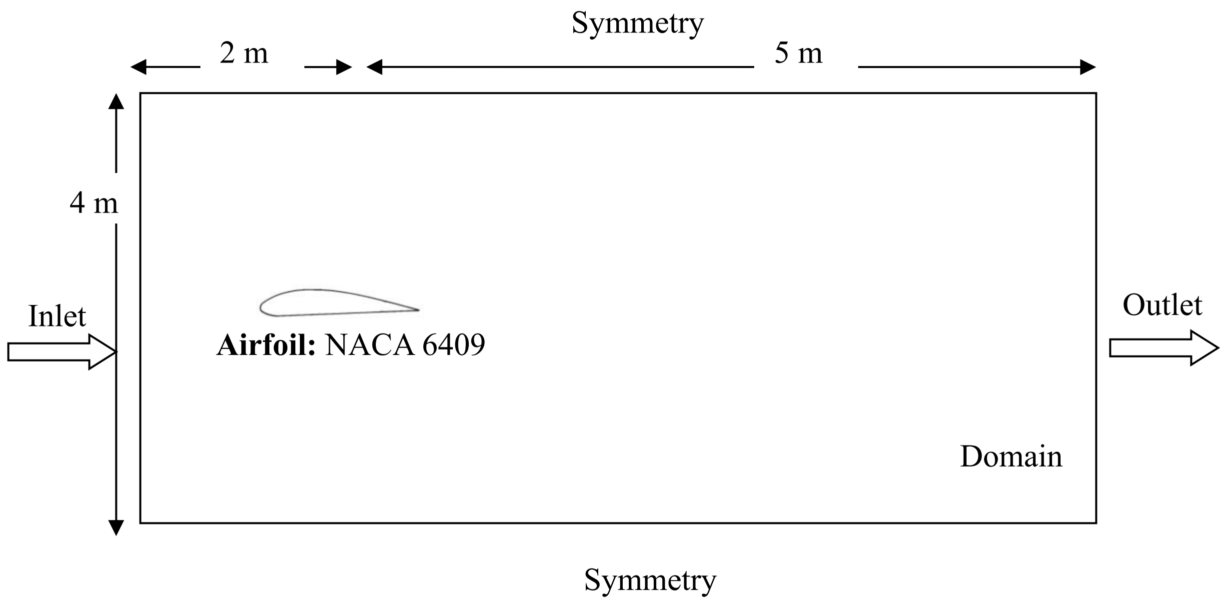

Fig 1.1 Schematic representation of Airfoil – Domain Geometry

- Concept → 3D Curve → Coordinate file → Select the previously edited airfoil coordinate file → Generate

- Concept → Lines from Points → join the edges at airfoil trailing edge → Generate

- Concept → Surface from Edges → Select 3D curve and line at same time by holding Ctrl button→ Generate

- XY plane → Sketching → Rectangle → Draw the Rectangle → Dimensions → horizontal → select reference Y-axis and horizontal sides for two separate horizontal dimensions→ Dimensions → General → Apply dimension on vertical side → Generate

- Concept → Surfaces from Sketches → Select Sketch1→ Generate
- Tools → Boolean → Operation → Subtract → Target Body = Surface1 → Tool Body = Surface2 → Generate

- Close the Design Modular and right click on Mesh option → Edit

Mesh:

- Mesh (Right Click on mouse) → Insert → Method → All Triangles → Select the Complete Geometry using Body selection → Apply

- Mesh → Insert → Sizing → select airfoil using Edge Selection → Definition → Type → Change from element size to Number of Divisions → enter a value of 60 → Enter

- Mesh → Insert → Inflation → Geometry = select entire geometry using body selection → Boundary = select airfoil using edge selection → Number of Layers = 30
- Mesh → Defaults → Element size = Enter the value of 0.09
- Mesh (Right Click on mouse) → Update
- Named Selection → Select the boundaries and create
- Check the Final Mesh and Close the window
- Right Click on Setup → Edit.

Solution Setup (Fluent):

- Tick the Double Precision Option → Solver progress → Enter the value of 8 → start
- Boundary condition → Inlet → Velocity magnitude → Enter a value of 30 m/s
- Reference Values → Compute from → Inlet
- Initialization → Standard initialization → Compute from → Inlet → Initialize
- Run calculation → No of iterations → Enter a Value of 500 → Calculate

Results:

- Insert → Location → Plane → Select XY → Apply
- Right Click on Result → Select Contour → Location = Select Plane → Variable = Pressure → Apply
- Select Contour (for velocity distribution) Location = Select Plane → Variable = Velocity → Apply
- The Pressure and Velocity fields across the airfoil geometry are represented in fig1.2, 1.3.

Observation:

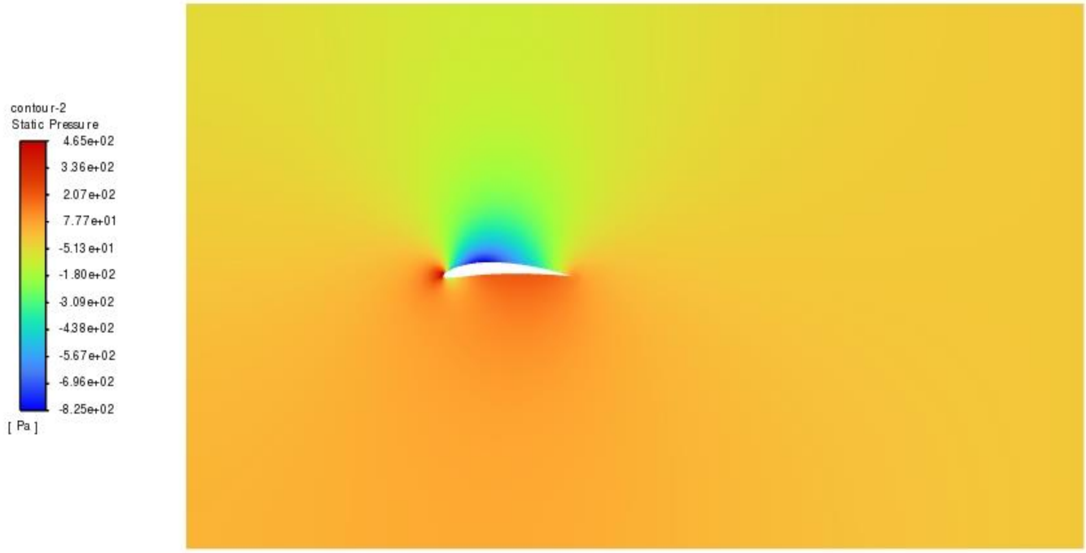

Fig 1.2 Pressure Distribution across aerofoil geometry

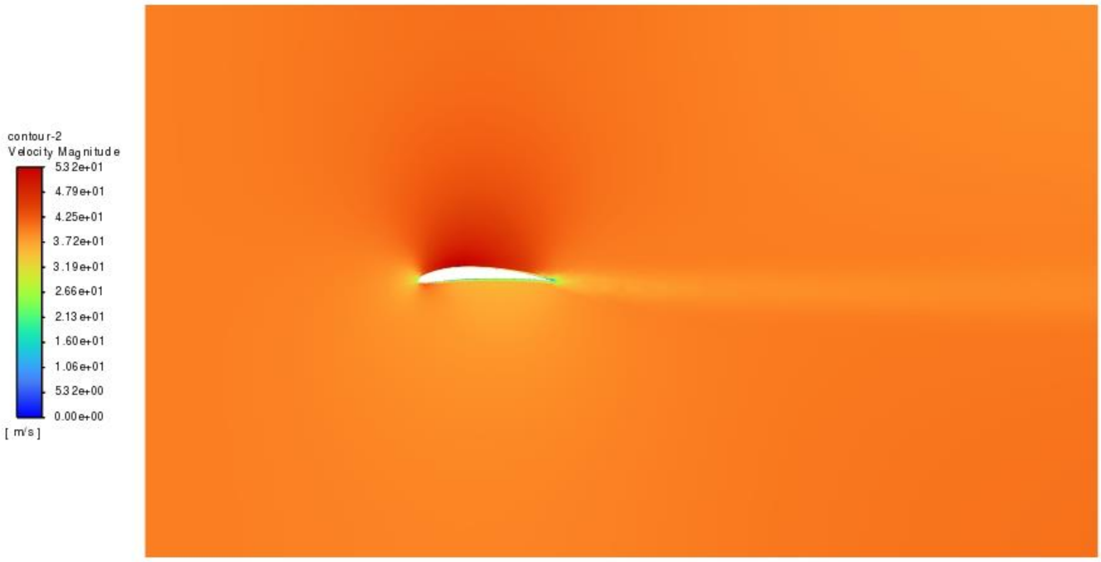

Fig 1.3 Velocity Distribution across aerofoil geometry

Results:

Thus, velocity and pressure distribution across a given airfoil geometry is obtained from numerical model generated using ANSYS Fluent software

Experiment No: 2 DATE:	**Flow over a flexible 3D Wing and its Deformation – one-way FSI**

Aim:

To analyse the flow over a flexible 3D wing and its deformation due to the pressure difference using one way Fluid Structural Interaction method

Computer System Configuration:

- Processor: 64-bit operating System Intel Multi Core
- RAM: 8 GB
- Operation System: Windows 11
- Software: ANSYS Fluent and Static Structural

Procedure:

Geometry

- Concept → 3D curve → import airfoil co-ordinates files → Generate
- Concept → lines from points → intersect the end points of airfoil at trailing edge
- Create Surface from edges → Extrude → 1 m
- Create a Domian across the wing, similar to experiment no.1 → Extrude → 1m → Apply Boolean as performed in experiment no.1
- Open ANSYS Fluent and Static Structural Workflow
- Connect the designed geometry to Fluent and Static Structural Geometry.

Mesh (Fluent):

- Select Mesh in fluent → Right Click → Edit → Insert → Method → All Triangles → Select the Complete Geometry using Body selection → Apply
- Mesh → Insert → Sizing → select airfoil using Edge Selection → Definition → Type → Change from element size to Number of Divisions → enter a value of 60 → Enter
- Mesh → Insert → Inflation → Geometry = select entire geometry using body selection

→ Boundary = select airfoil using edge selection → Number of Layers = 30

- Mesh → Defaults → Element size = Enter the value of 0.08
- Mesh (Right Click on mouse) → Update
- Named Selection → Select the boundaries and create named sections as mentioned in fig 1.1
- Check the Final Mesh and Close the window
- Right Click on Setup → Edit.

Setup (fluent):

- Select Double Precision Option → Solver progress → Enter the value of 8 → start
- Boundary condition → Inlet → Velocity magnitude → Enter a value of 40 m/s → Outlet → Pressure Inlet
- Reference Values → Compute from → Inlet
- Initialization → Standard initialization → Compute from → Inlet → Initialize
- Run calculation → No of iterations → Enter a Value of 800 → Calculate

Results:

- Locations → Plane → XY → Z = 0
- Contour → From → Plane → Variable → Pressure → Apply
- Contour → From → Wing → Variable → Pressure → Apply
- Contour → From → Plane → Variable → Velocity → Apply

Model:

- Select Model in static structural workflow → Right Click → Edit
- Select Wing with Surface Selection with Box Type Selector → Right Click → Suppress all other bodies
- Mesh → Insert→ Edge Sizing → Edge Selection → No. of divisions → Enter a value of 80

- Generate Mesh → Update

Setup (Static Structural):

- Static Structural → Insert → Fixed Support → Select one cross sectional face of wing → Apply
- Connect Fluent solution data to Static Structural solution by simple drag and drop technique → Imported Load → Interpolation type → select CFD results interpolator
- Imported load → Right Click → Insert → Pressure → Geometry → select airfoil surface → CFD surface → select airfoil
- Solution → Solve

<u>Observation:</u>

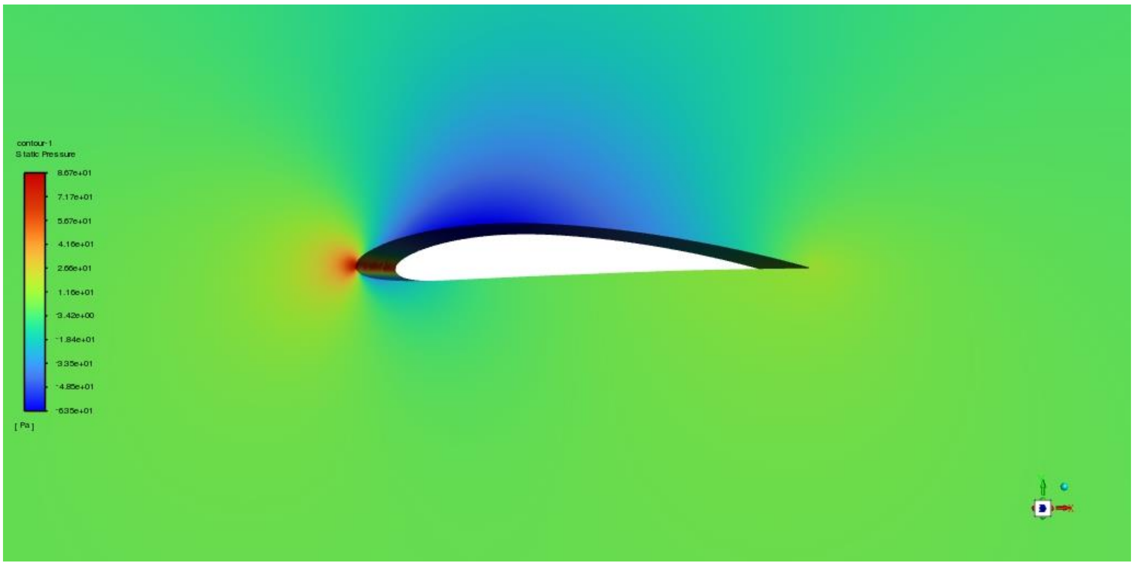

Fig 2.2 Pressure profile of wing

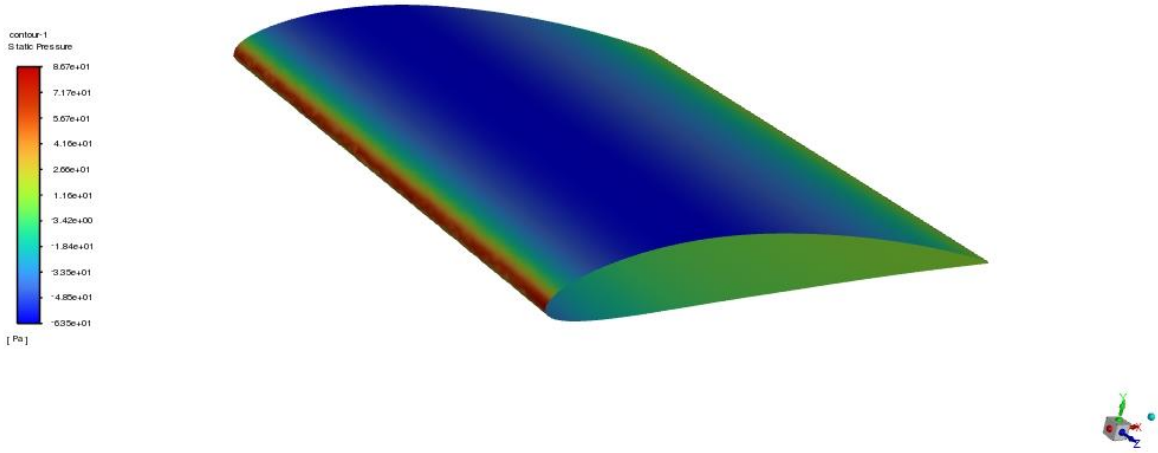

Fig 2.3 Pressure profile of geometry wing

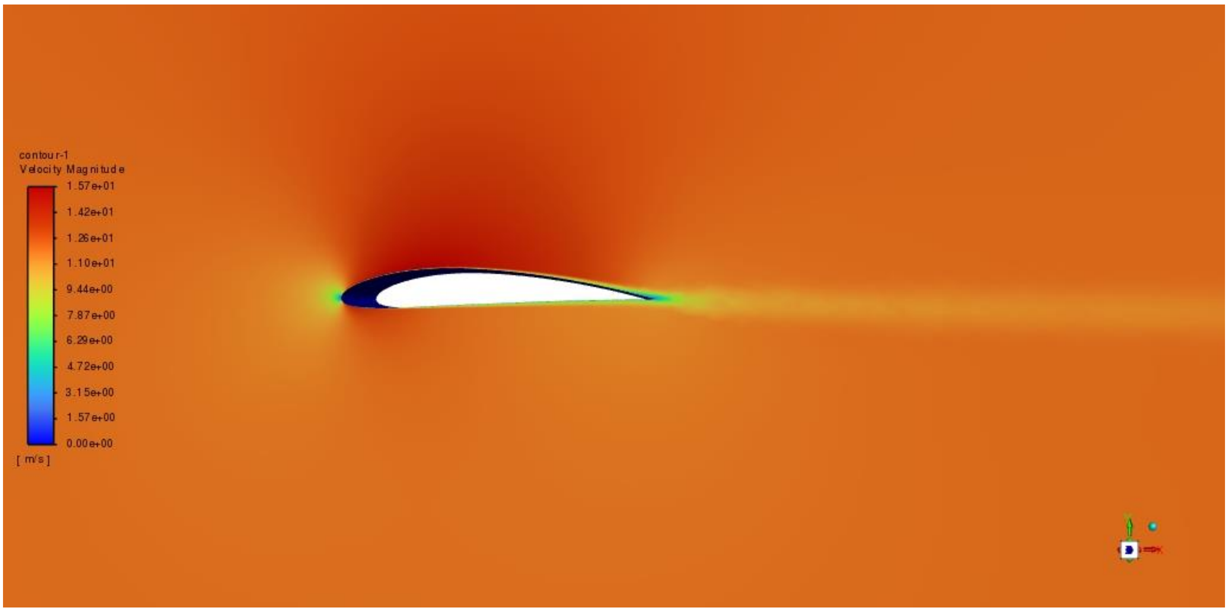

Fig 2.4 Velocity profile of wing

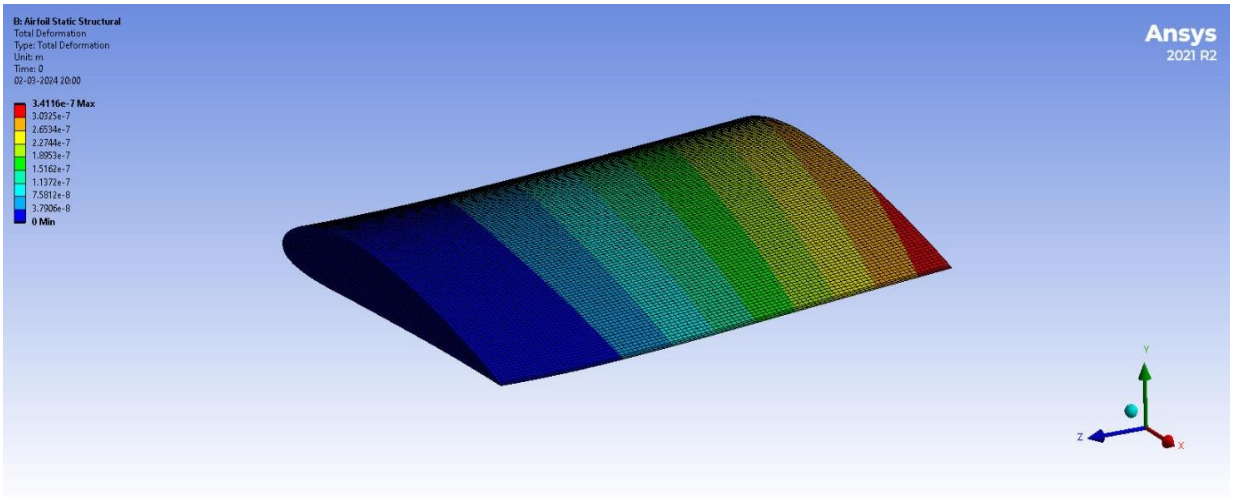

Fig 2.5 Deformation across the Wing Geometry

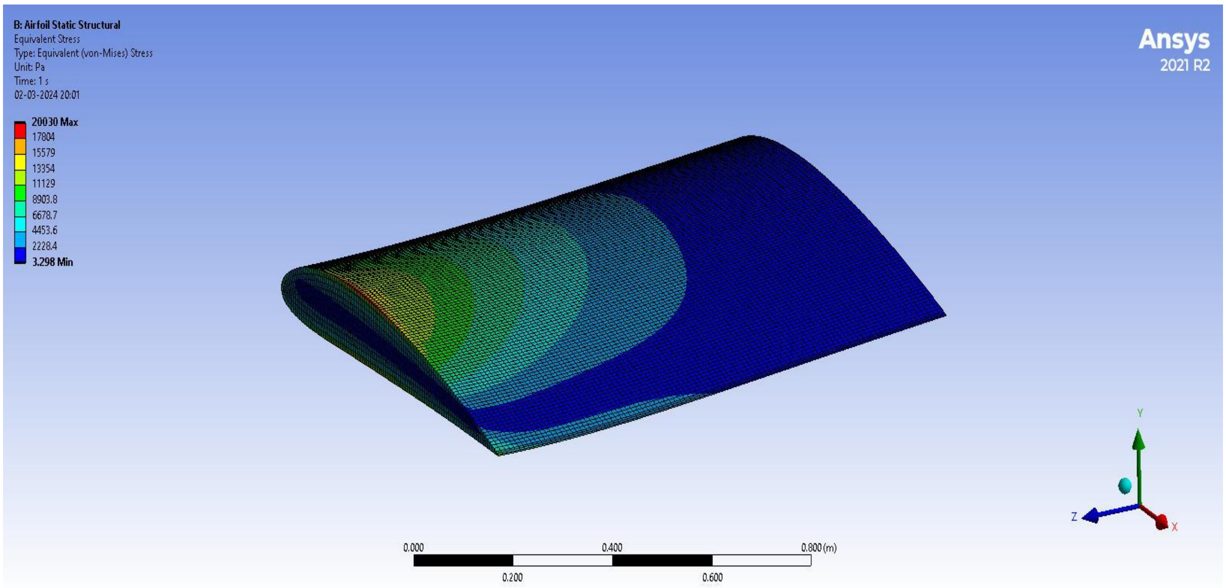

Fig 2.6 Equivalent Stress across the Wing Geometry

Results:

Thus, the air flow and deformation across a wing due to its pressure difference is estimated using one way Fluid Structural Interaction (FSI).

Experiment No: 3 DATE:	**Structural analysis using 1D element – Truss problem**

AIM:

To analyze the deflection and stress on the members of given truss subjected to tensile loads using ANSYS Static Structural Workflow

Computer System Configuration:

- Processor: 64-bit operating System Intel Multi Core
- RAM: 16 GB
- Operation System: Windows 11
- Software: ANSYS Static Structural

Procedure:

Static Structural

- Engineering Data → Material → Structural Steel (Isotropic Elasticity).
- The Young's modulus is 13.1Gpa and Poisson's Ratio is 0.29

Geometry – New Design Modeler Geometry

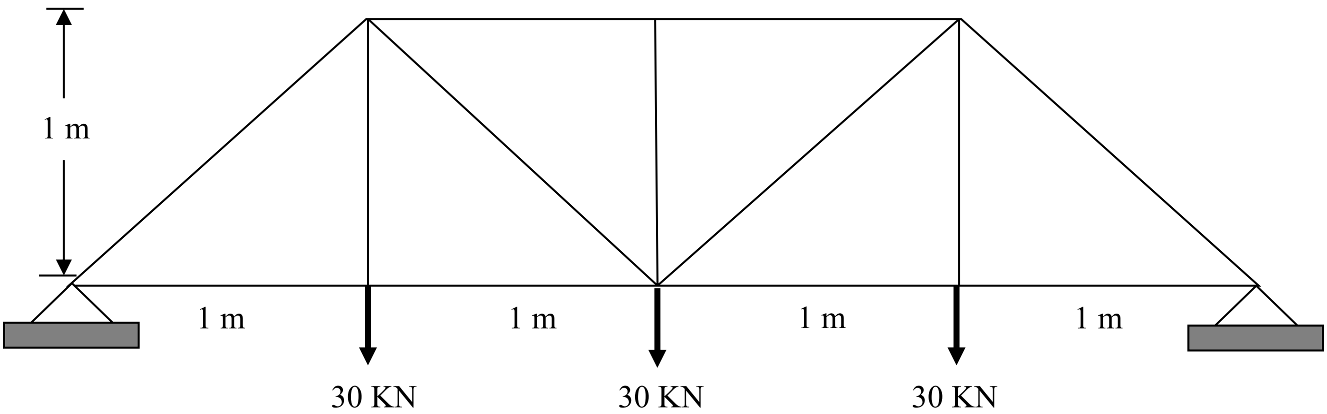

Fig 3.1 Schematic Representation of Truss

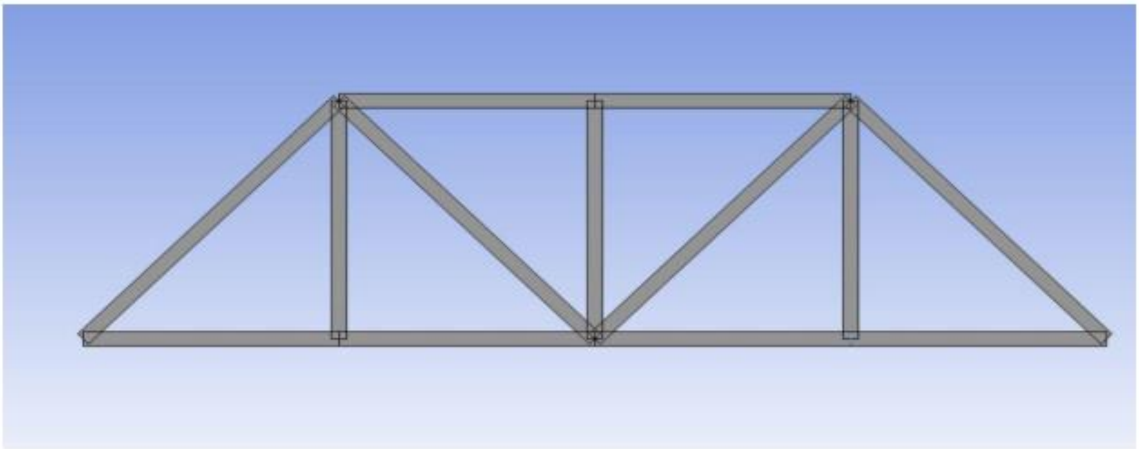

Fig 3.2 Design Modular Geometry of Truss

- Select XY-Plane and sketch shown in figure
- Concept → Lines from Sketches → Select the sketch → Generate.
- Concept → Cross section → Rectangular (0.6m x 0.6m)
- In the line body add cross section as Rectangle → generate → View → Cross section Solids
- Close the design modeler → Right Click on Model → Edit

Model:

- Geometry → Line Body → Material → Assignment → Steel
- Static Structure → Insert → Fixed Support →Select the bottom to edges & Apply
- Static Structure → Insert → Force → Select the Three Vertices shown in fig 3.2 of load 30KN → Apply
- Mesh → Element size (2m) → Generate Mesh
- Solution → Insert → Deformation → Total Deformation
- Solution → Insert → Stress
- Solve

Observation:

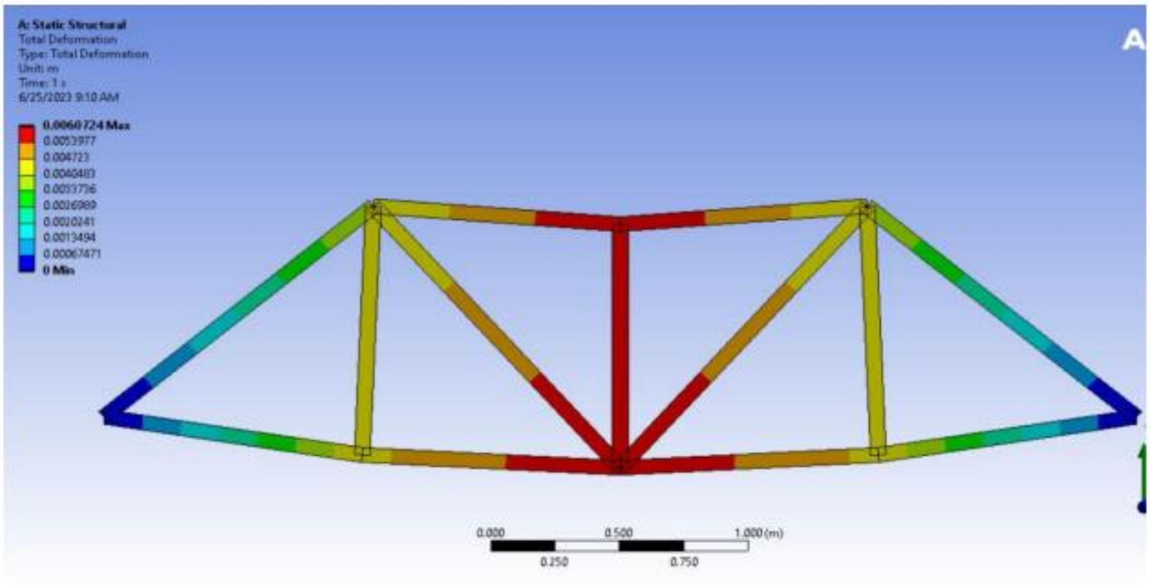

Fig 3.3 Total Deformation across Truss Geometry

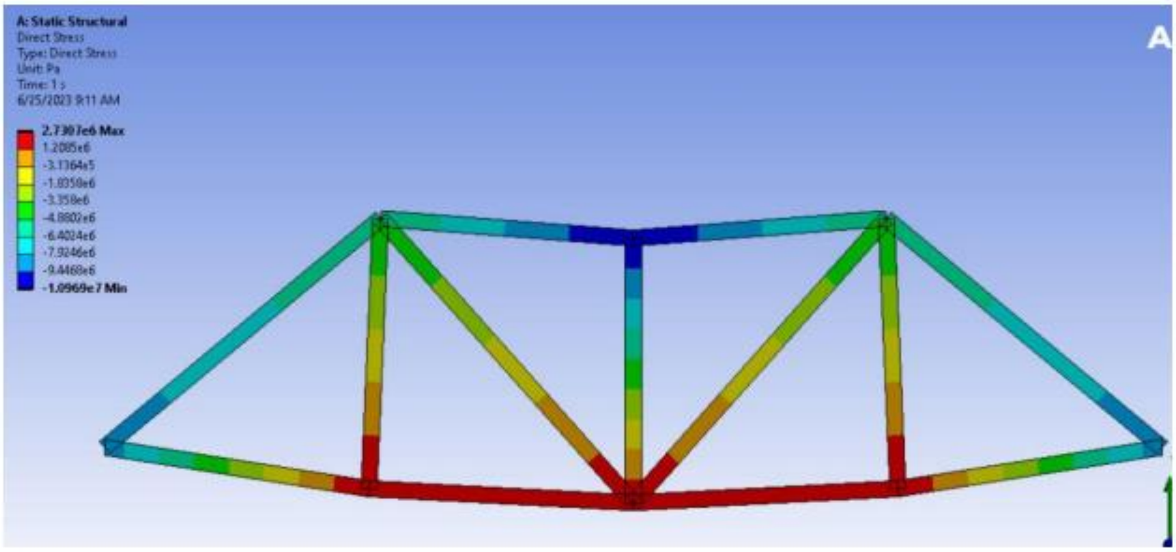

Fig 3.4 Direct Stress across Truss Geometry

<u>Results:</u>

Thus, the deflection and Stresses on the members of a given truss problem subjected to load is analyzed using ANSYS Static Structural Workflow

Experiment No: 4 DATE:	**Structural analysis using 2D element – Plate with hole problem**

AIM:

To analyze the stress and deformation across a plate with a hole using ANSYS Static Structural Workflow.

Computer System Configuration:

- Processor: 64-bit operating System Intel Multi Core
- RAM: 8 GB
- Operation System: Windows 11
- Software: ANSYS Static Structural

Procedure:

Static Structural

- Engineering data → Add Steel → Isotropic elasticity → Young's modulus (210 GPa) → Poisson's ratio (0.3)

Geometry– New Design Modeler Geometry

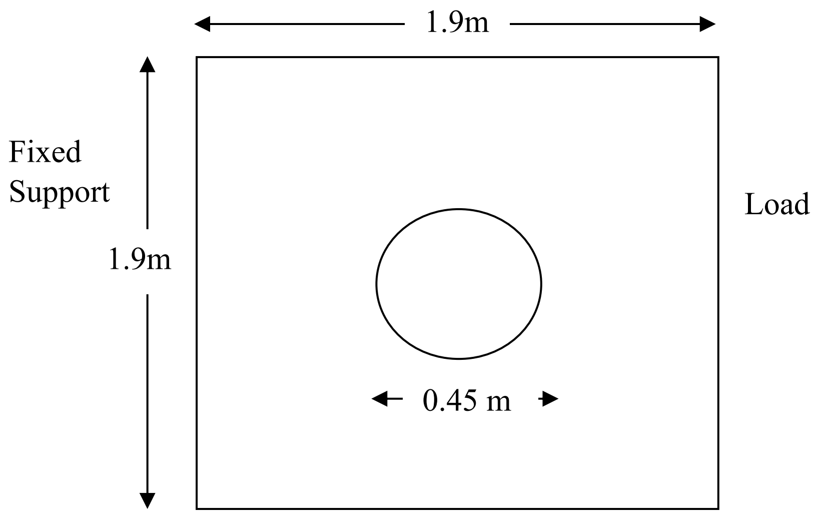

Fig 4.1 Schematic Representation of plate with hole

- Select XY-plane → Sketching → Rectangle → Dimensions → General → 1.9m (Horizontal), 1.9m (Vertical)

- Circle → Dimensions → 0.45 m
- Close the Design Modeler → Right click on Model → Edit

Model

- Geometry → Solid → Assignment → Nonlinear Effects = No → Thermal Strain Effects = No
- Mesh → Generate → Element size = 0.2 m → Generate mesh
- Static structural → insert → Fixed Support → Select left side face → Click Enter
- Static structural → insert → Force → Select Right side edge → Apply → Define by→ Component = -10 KN
- Solution → insert → Deformation → Total
- Solution → insert → Stress → Equivalent Von Mises Stress → Solve

Observation:

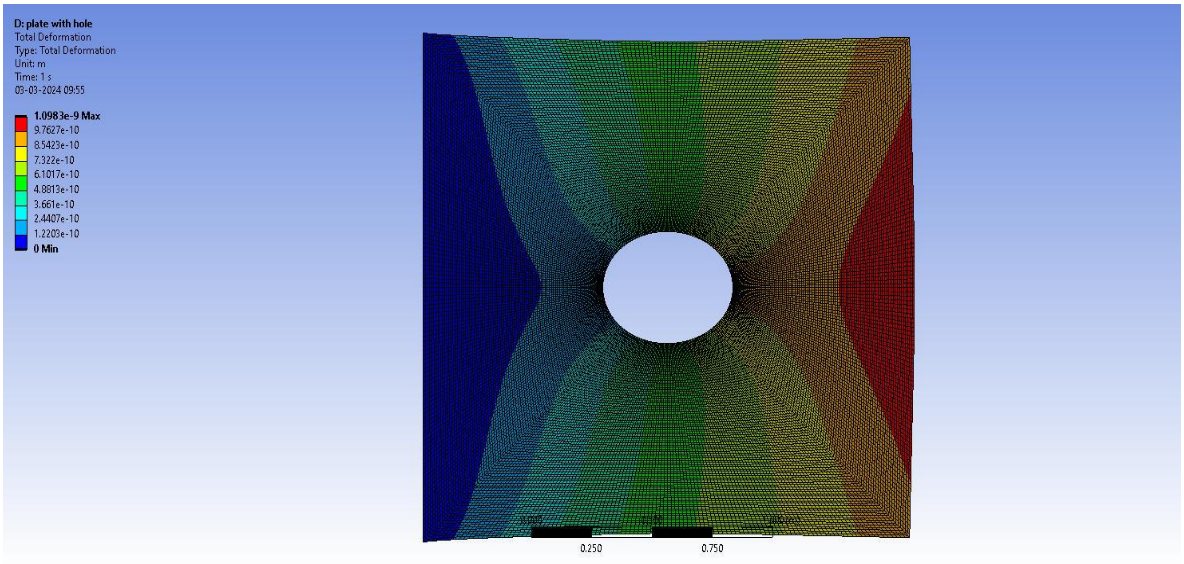

Fig 4.2 Deformation across the plate geometry

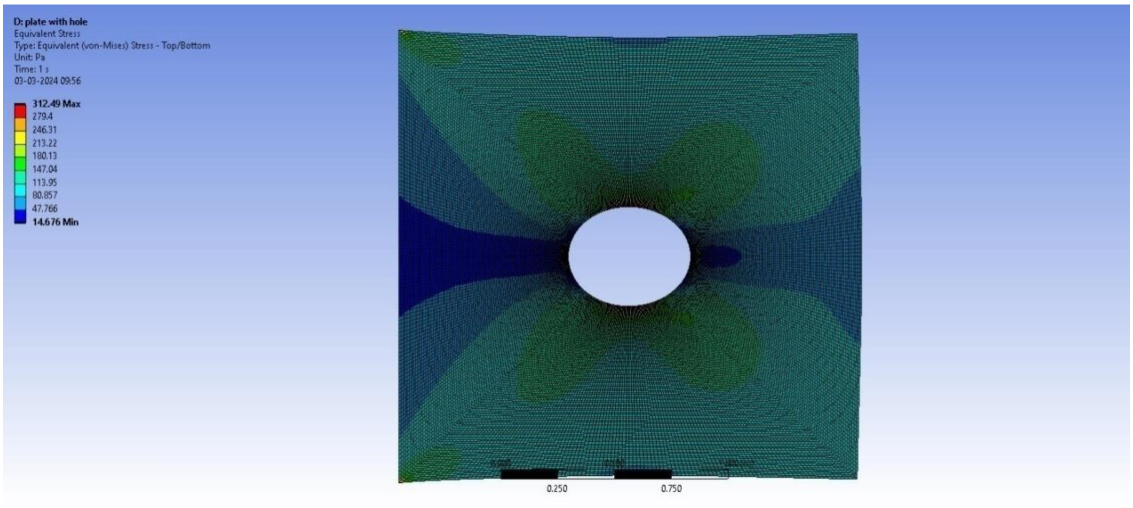

Fig 4.3 Equivalent Stress across the plate geometry

<u>Result:</u>

Thus, the stress and deformation of given rectangular plate with circular hole subjected to force at the center is analyzed using ANSYS Static Structural Workflow

Experiment No: 5 DATE:	**Stress analysis of plate/beam with hole using 3D element**

AIM:

To determine the Stress and deformation across rectangular beam geometry with circular hole subjected to Tensile Force using ANSYS Static Structural Workflow.

Computer System Configuration:

- Processor: 64-bit operating System Intel Multi Core
- RAM: 8 GB
- Operation System: Windows 11
- Software: ANSYS Static Structural

Procedure:

Static Structural:

- Engineering data → Add Steel → Isotropic elasticity → Young's modulus (210 GPa) → Poisson's ratio (0.3)

Geometry – New Design Modeler Geometry:

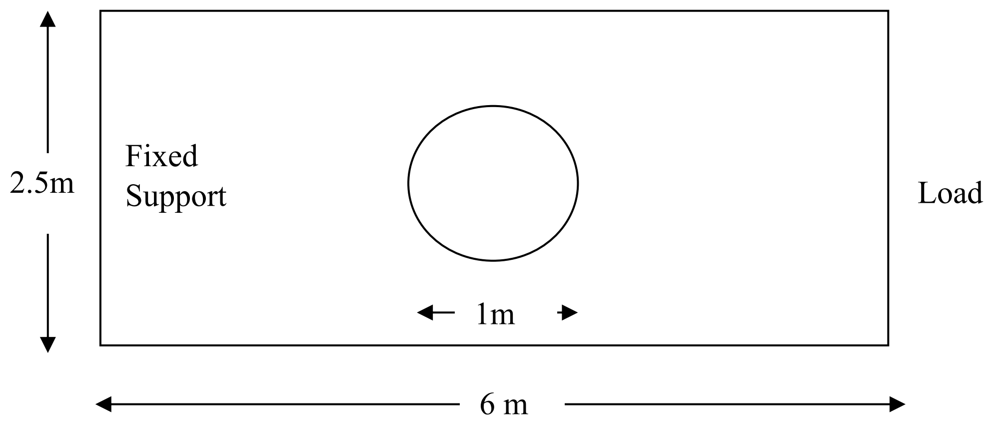

Fig 5.1 Schematic Representation of Plate Geometry

- Geometry → properties → Analysis type → 3D
- Select XY-plane → Sketching → Rectangle → Dimensions → General → 6m (Horizontal), 2.5m (Vertical)
- Concept → Surfaces from sketches → Select Sketch 1 → Apply → Generate
- Extrude → Geometry → Select Sketch1 → Direction = Normal → FD1, Depth = 0.3 m → Generate

Model:

- Geometry → Surface body → Assignment → Steel → Nonlinear Effects (No) → Thermal Strain Effects (No).
- Mesh → Generate → Element order → linear → Element size (0.001) → Generate mesh.
- Static structural → insert → Force → Select Right Wall → Apply → Defined by → Components → Y-components = -200 N
- Static structural → insert → Fixed support → Left wall → Apply
- Solution → insert →Deformation → Total Deformation
- Solution → insert → Stress → Equivalent → Solve

<u>Observation:</u>

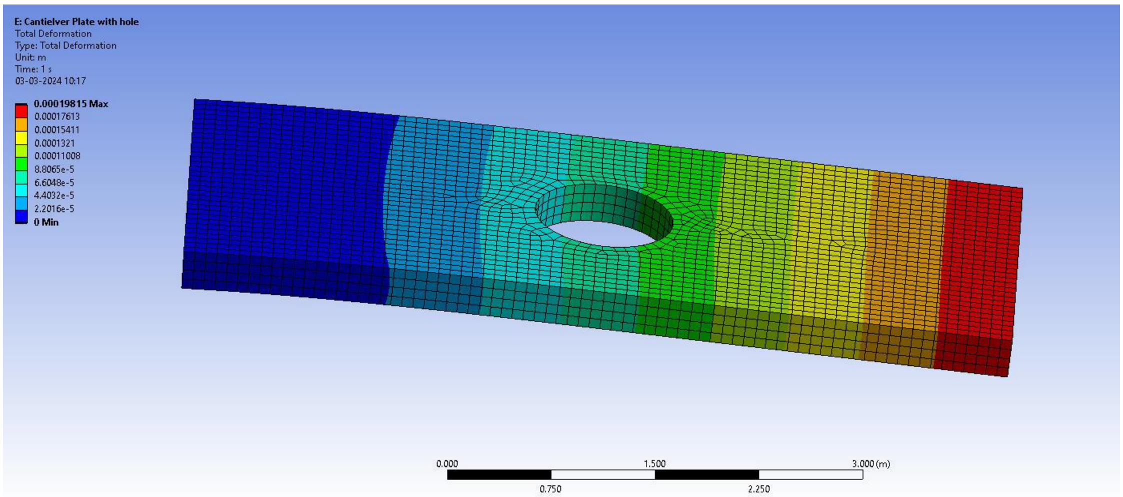

Fig 5.2 Deformation across the cantilever plate with hole

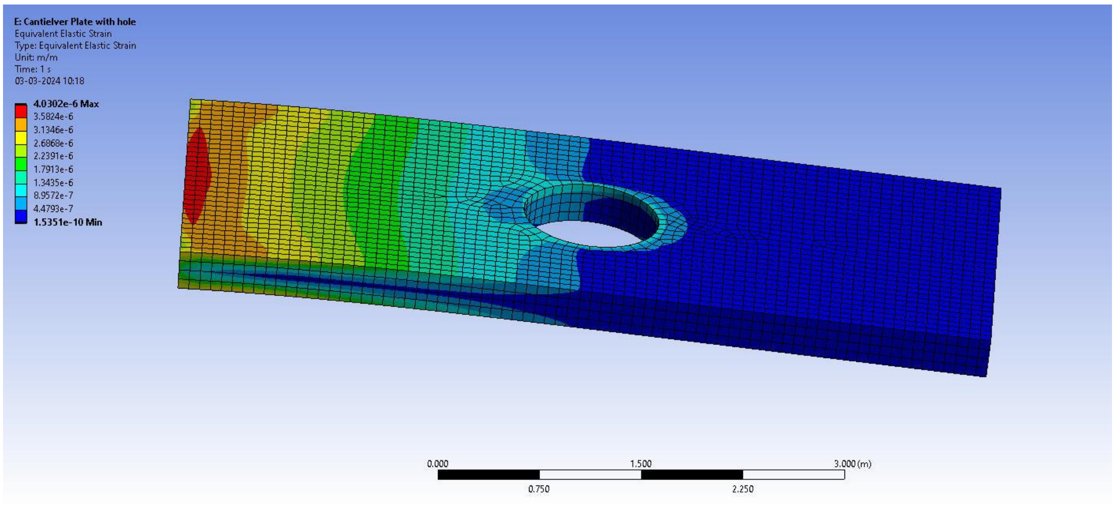

Fig 5.3 Equivalent Stress across the cantilever plate with hole

Result:

Thus, the stress and Deformation of a given rectangular plate with circular hole subjected to Tensile Force is analyzed using ANSYS software.

Experiment No: 6 **DATE:**	**1D transient heat conduction – thermal analysis of a rod**

Aim:

To study the transient thermal analysis over a cylindrical rod using ANSYS Transient Thermal workflow

Computer System Configuration:

- Processor: 64-bit operating System Intel Multi Core
- RAM: 8 GB
- Operation System: Windows 11
- Software: ANSYS Transient Thermal

Procedure:

- Open ANSYS Workbench → drag and drop the transient thermal workflow to workspace
- Right Click on Engineering Data → Select Structural Steel

Geometry–New Design Modeler Geometry:

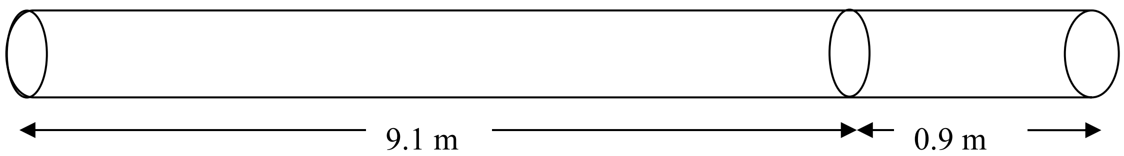

Fig.6.1 Schematic Representation of Rod Geometry

- Select XY Plane → Sketching → Circle → Draw a Circle → Dimensions → General → click on circle and drag to display the dimension → enter a value of 0.05m on below left side dimensions column
- Extrude → Geometry → Select Sketch1 → Direction = Normal → FD1, Depth = 10 m → Generate
- Select Plane → Base Plane = XY → Transform 1 (RMB) = Off set Z → FD1, Value 1 = 0.9 m → Generate

- Slice → Plane1 → apply → Generate

Model:

- Mesh → Generate mesh
- Mesh → Element size → 0.01 → Generate mesh
- Transient Thermal → Insert → Temperature → Solid-1 → Apply → Magnitude → 50
- ANSYS settings → Stop End Time → Enter 800
- Solution → Insert → Thermal → Temperature → Solve

Setup:

- Transient Thermal (Right Click) → Insert → Temperature → Solid-1 → Apply → Magnitude → 80 → Solve
- ANSYS settings → Stop End Time → Enter a value of 800
- Solution → Insert → Thermal → Temperature → Solve

Observation:

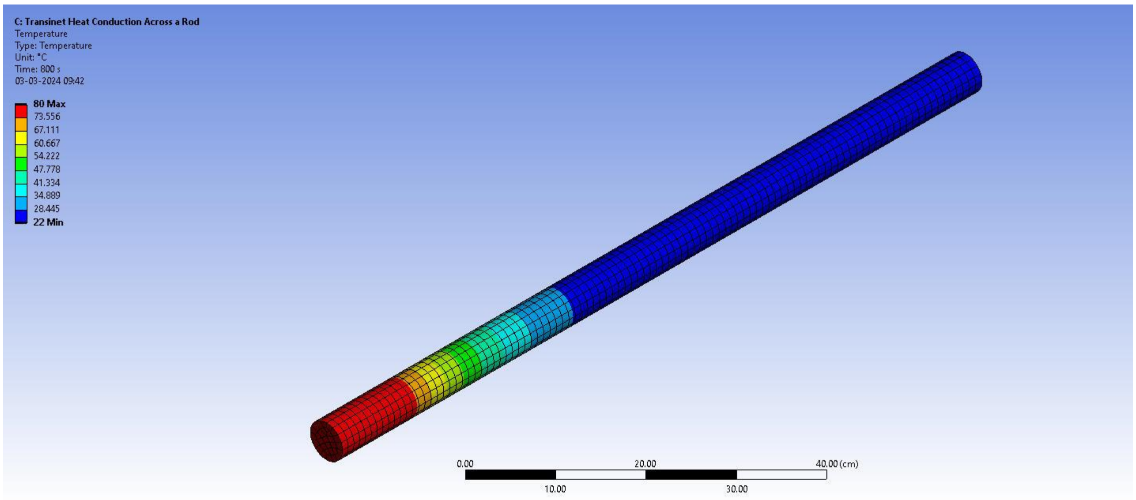

Fig 6.2 Transient Heat Transfer across a rod

Result:

Transient thermal flow analysis over a cylinder rod has been analyzed using ANSYS Transient Thermal workflow.

Experiment No: 7 DATE:	**2D steady state heat transfer analysis of a plate**

AIM:

To study the 2-D Study State Heat Thermal Analysis over a flat plate by using ANSYS Steady state Thermal.

Computer System Configuration:

- Processor: 64-bit operating System Intel Multi Core
- RAM: 8 GB
- Operation System: Windows 11
- Software: ANSYS Steady State Thermal

Procedure:

- Open ANSYS Workbench → drag and drop Fluent Workflow to workspace
- Right click on Geometry → Select New Design Modular

Geometry – New Design Modeler Geometry:

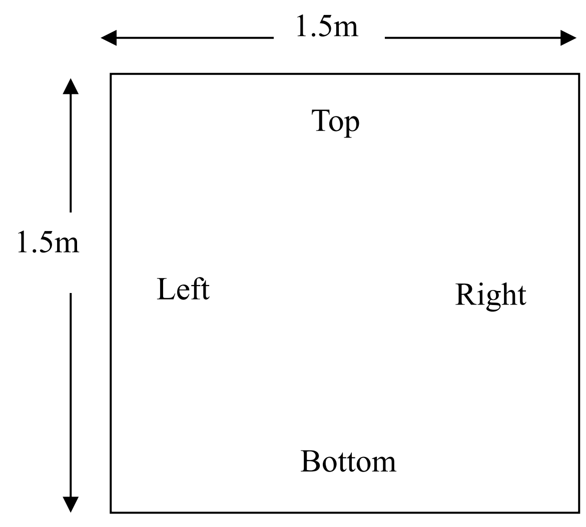

Fig 7.1 Schematic Representation of Flat Plate Geometry

- Sketch → Rectangle (1.5m x 1.5m)

- Concept → Surface from Sketches → Select the sketch1 → Apply
- Close the Design Modeler → Right Click on Model → Edit

Mesh:

- Geometry → Assignment → Material → steel
- Mesh → Element size 0.05 → Generate
- Named Selection → Specify the boundaries as given in fig.7.1

Setup (Fluent):

- Select Energy → Tick
- Material → Solid → Fluent Database → Select Steel → Copy
- Boundary Conditions → left → Thermal → select Temperature → enter the value of 293 K → Material → Steel
- Boundary Conditions → right → Thermal → select Temperature → enter the value of 293 K → Material → Steel
- Boundary Conditions → top → Thermal → select Temperature → enter the value of 293 K → Material → Steel
- Boundary Conditions → bottom → Thermal → select Temperature → enter the value of 393 K → Material → Steel
- Reference Values → Compute from → Select Bottom
- Initialization → Standard Initialization → Compute from → select Top
- Run Calculation → No of iterations → Enter a value of 200 → Calculate

Results:

- Right Click on result → Edit
- Insert → Location → Plane → Select XY → Apply

- Select Contour → Location = Select Plane → Variable = Temperature → Apply
- Named Selection → Specify the boundaries as given in fig.7.2

<u>Observation:</u>

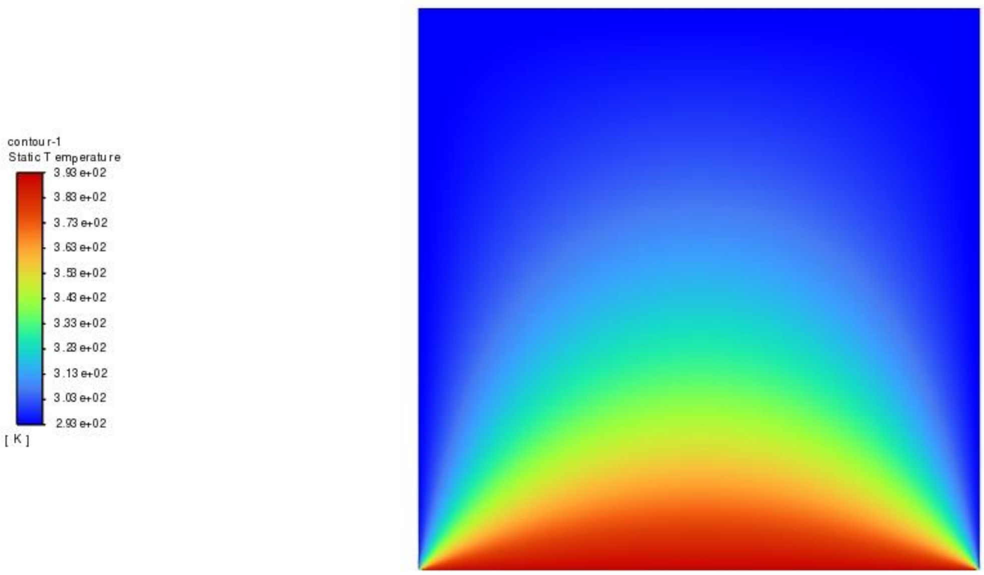

Fig 7.2 Steady State Heat Transfer across a 2D flat plate

<u>Result:</u>

Thus, the 2-D Study State Heat Thermal Analysis flow over a flat plate has been analyzed