MVJ COLLEGE OF ENGINEERING

An Autonomous Institute

NEAR ITPB, CHANNASANDRA, BENGALURU – 560 067
Affiliated to VTU, Belagavi
Approved by AICTE, New Delhi
Recognized by UGC under 2(f) & 12(B)
Accredited by NAAC

DEPARTMENT OF AERONAUTICAL ENGINEERING

VII SEMESTER

MVJ22AE72 – COMPUTATIONAL FLUID DYNAMICS LAB

ACADEMIC YEAR 2025 – 2026

LABORATORY MANUAL

NAME OF THE STUDENT   : _____

BRANCH   : _____

UNIVERSITY SEAT NO.   : _____

SEMESTER & SECTION   : _____

BATCH   : _____

VISION AND MISSION OF THE INSTITUTION

Institute Vision
To become an institute of academic excellence with international standards.

Institute Mission

1. Impart quality education along with industry exposure.

2. Provide world-class facilities to undertake research activities relevant to industrial and professional needs.

3. Promote entrepreneurship and value-added education that is socially relevant with economic benefits

Department Vision

To have an international standing for imparting quality technical education in the field of Aeronautical engineering and technology, to a more sustainable and socially responsible future.

Department Mission

- **Knowledge & Innovation:** The department aims in dissemination of knowledge to develop innovative solutions to the various problems in Aeronautical Engineering and related fields.

- **Professional Skills:** To mold students into successful Aeronautical Engineers by maintaining best teaching and learning environment in which faculty grow professionally and students receive unsurpassed knowledge, skills, insights and tools for lifelong learning.

- **Research Inculcation:** To inculcate the state-of-the-art technologies and R & D to design the next generation of high performance, efficient air & space transportation.

- **Socio-Ethical responsibility:** To nurture Aeronautical Engineers to be sensitive to ethical, societal, and environmental issues while conducting their professional work

Program Outcomes (PO):

1. **Engineering Knowledge:** Apply knowledge of mathematics, natural science, computing, engineering fundamentals and an engineering specialization as specified in respectively to develop to the solution of complex engineering problems.

2. **Problem Analysis:** Identify, formulate, review research literature and analyze complex engineering problems reaching substantiated conclusions with consideration for sustainable development.

3. **Design/Development of Solutions:** Design creative solutions for complex engineering problems and design/develop systems/components/processes to meet identified needs with consideration for the public health and safety, whole-life cost, net zero carbon, culture, society and environment as required.

4. **Conduct Investigations of Complex Problems:** Conduct investigations of complex engineering problems using research-based knowledge including design of experiments, modelling, analysis & interpretation of data to provide valid conclusions.

5. **Engineering Tool Usage:** Create, select and apply appropriate techniques, resources and modern engineering & IT tools, including prediction and modelling recognizing their limitations to solve complex engineering problems.

6. **The Engineer and The World:** Analyze and evaluate societal and environmental aspects while solving complex engineering problems for its impact on sustainability with reference to economy, health, safety, legal framework, culture and environment.

7. **Ethics:** Apply ethical principles and commit to professional ethics, human values, diversity and inclusion; adhere to national & international laws.

8. **Individual and Collaborative Team work:** Function effectively as an individual, and as a member or leader in diverse/multi-disciplinary teams.

9. **Communication:** Communicate effectively and inclusively within the engineering community and society at large, such as being able to comprehend and write effective reports and design documentation, make effective presentations considering cultural, language, and learning

differences

10. **Project Management and Finance:** Apply knowledge and understanding of engineering management principles and economic decision-making and apply these to one's own work, as a member and leader in a team, and to manage projects and in multidisciplinary environments.

11. **Life-Long Learning:** Recognize the need for, and have the preparation and ability for i) independent and life-long learning ii) adaptability to new and emerging technologies and iii) critical thinking in the broadest context of technological change.

Program Educational Objectives (PEOs):

- **PEO-01: Employability & Skills**: Graduates will be successful Aeronautical Engineers in Industry, Research and Academic sectors by applying the basic principles of Mathematics, Science and Engineering, with high quality communication and interpersonal skills to work effectively in multidisciplinary teams, both as team members and as leaders.

- **PEO-02: Professional Development**: Graduates will be able to synthesize data & derive technical specifications and design and develop innovative solutions to the various problems in Aeronautical Engineering by engaging in lifelong learning and professional development.

- **PEO-03: Social & Ethical values**: Graduates will use modern engineering techniques, skill and tools with high degree of professional ethics and standards, to fulfill the societal and personal needs

Program Specific Outcomes (PSO):

Program Specific Outcomes: PSOs are statements that describe what the graduates of a specific engineering program should be able to do.

PSO-1: DGCA/FAR/FAA/MIL/JAR/DEFSTD-regulations: Use the standard government regulations/specifications for design, manufacturing and testing purposes of civil and military

aircrafts and perform various maintenance related works and synthesize information / data from various sources of Aircraft operations.

PSO-2: Design, Development & Manufacturing of aircraft and related systems: Carry out the preliminary design and development of aircraft and manufacturing of various systems involved and Predict performance characteristics along with the stability analysis.

COURSE OUTCOMES:

Course Code	Outcomes
CO1	Apply knowledge of CFD ideas, and Flow Equations
CO2	Assimilate Mathematical behaviour of PDEs vis a vis nature of flow
CO3	Utilize finite difference techniques.
CO4	Generate & Utilize grids
CO5	Apply finite volume techniques

LIST OF EXPERIMENTS

GENERAL PROCEDURE OF CFD ANALYSIS

The codes provide a complete CFD analysis, consisting of three main elements:
1. Pre-processor
2.Solver
3.Post-processor
Figure (i) presents a framework that illustrates the interconnectivity of the three elements within the CFD analysis.

PROBLEM SETUP—PRE-PROCESS
Creation of Geometry—Step 1

The first step in any CFD analysis is the definition and creation of the geometry of the flow region, i.e., the computational domain for the CFD calculations. It is important that the reader should always acknowledge the real physical flow representation of the problem that is to be solved, as demonstrated by the respective physical domains in Figures (II). One important aspect that the reader should always note in the creation of the geometry for CFD calculations is to allow the flow dynamics to be sufficiently developed across the length L of the computational domains.

Mesh Generation—Step 2

The second step, mesh generation, is one of the most important steps in the preprocess stage after the definition of the domain geometry. CFD requires the subdivision of the domain into a number of smaller, non-overlapping subdomains in order to solve the flow physics within the domain geometry that has been created; this results in the generation of a mesh (or grid) of cells (elements or control volumes) overlying the whole domain geometry. The essential fluid flows that are described in each of these cells are usually solved numerically, so that the discrete values of the flow properties, such as the velocity, pressure, temperature, and other transport parameters of interest, are determined. This yields the CFD solution to the flow problem that is being solved. The accuracy of a CFD solution is strongly influenced by the number of cells in the mesh within the computational domain.
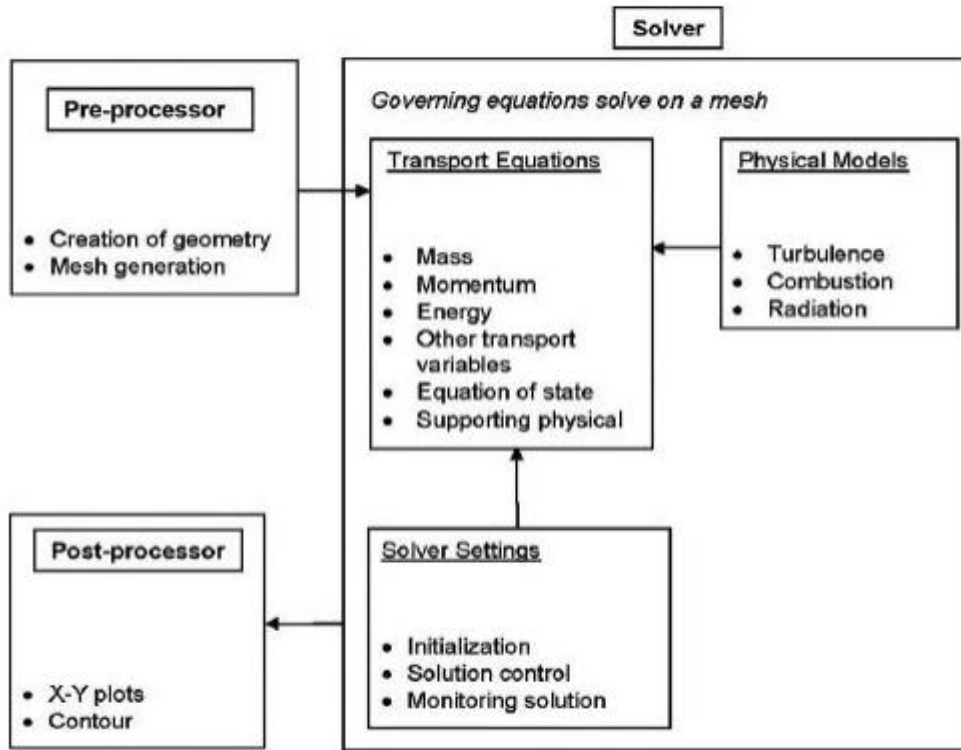
Figure (I) The interconnectivity functions of the three main elements within a CFD analysis framework.
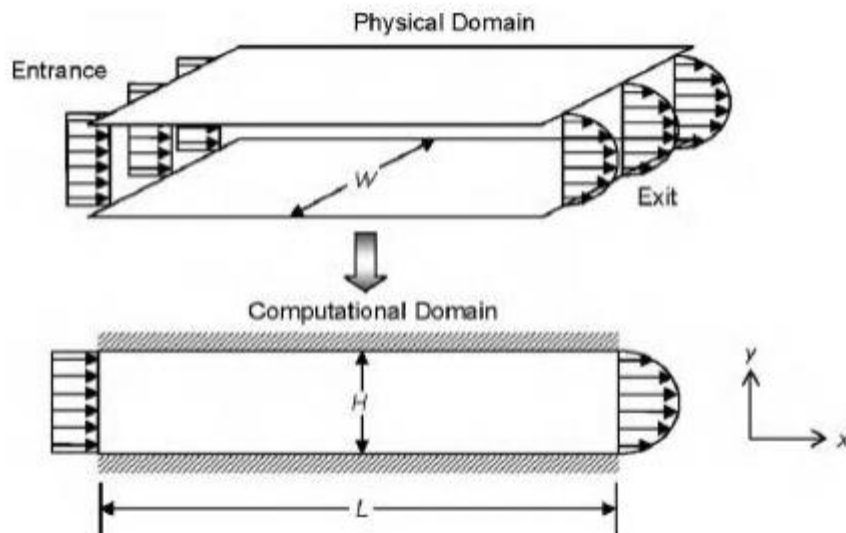

Figure (II) Fluid flowing between two stationary parallel plates.

Figure (III) shows a mesh of 20 (L) x 20 (H) cells, resulting in a total of 400 cells allocated for the Case of CFD problem between two stationary parallel plates. For more complex geometries, meshing by triangular cells allows flexibility in mesh generation for geometries having complicated shape boundaries. Figure (IV) illustrates a typical distribution of triangular cells within the computational domain for the Case of problem of fluid passing over two cylinders, with a mesh totaling 16,637 cells mapping the whole flow domain.
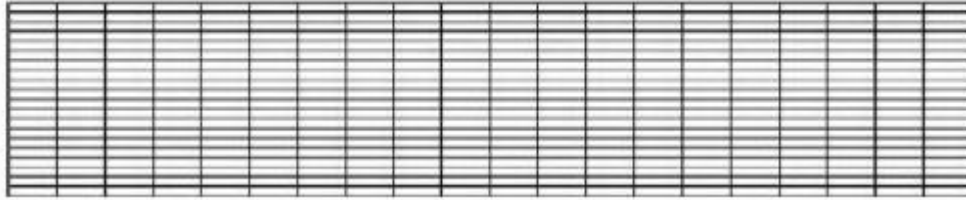
Figure (III) Structured meshing for fluid flowing between two stationary parallel plates.
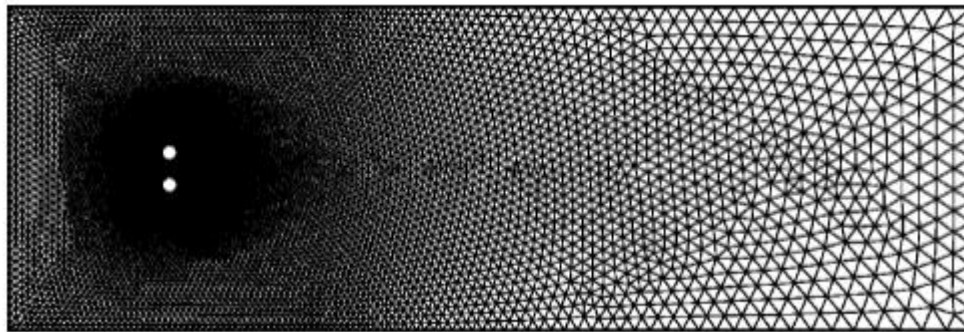

Figure (IV) Unstructured meshing for fluid passing over two cylinders in an open surrounding

Selection of Physics and Fluid Properties—Step 3

Many industrial CFD flow problems may require solutions to very complex physical flow processes, such as the accommodation of complicated chemical reactions in combusting fluid flows. The inclusion of combustion and possibly radiation models in the CFD calculations is generally a prerequisite for successful modeling of these types of flows.

Combustion and radiation processes have the tendency to strongly influence the local and global heat transport, which consequently affects the overall fluid dynamics within the flow domain. It is therefore imperative that the CFD user carefully identify the underlying flow physics unique to the particular fluid-flow system. For clarity and ease of reference, a flowchart highlighting the

various flow physics that may be encountered within the framework of CFD and heat transfer processes is presented in Figure (V). Under the main banner "Computational Fluid Dynamics & Heat Transfer," a CFD user declares initially whether simulations of the fluid-flow system are to be attained for transient/unsteady or steady solutions. He/she subsequently defines which class of fluids that the flows belong to: inviscid or viscous.
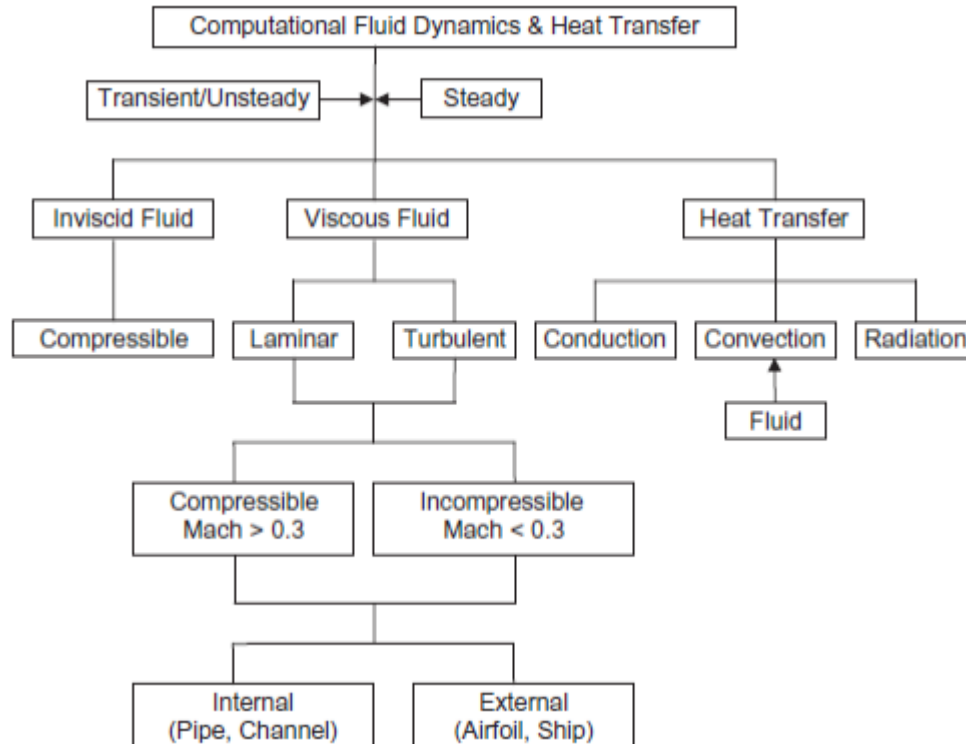
Figure (V) A flowchart encapsulating the various flow physics in CFD

Specification of Boundary Conditions—Step 4

The complex nature of many fluid-flow behaviors has important implications for which boundary conditions are prescribed for the flow problem. A CFD user needs to define appropriate conditions that mimic the real physical representation of the fluid flow in a solvable CFD problem. The fourth step in the pre-process stage deals with the specification of permissible boundary conditions that are available for impending simulations. Evidently, where inflow and outflow boundaries exist within the flow domain, suitable fluid-flow boundary conditions are required to accommodate the fluid behavior upon entering and leaving the flow domain. Schematic descriptions of the boundary

conditions are demonstrated in Figure (VI) for Case of CFD problem between two stationary parallel plates.

Numerical solution— CFD solver

The appropriate use of either an in-house or a commercial CFD code requires a core understanding of the underlying numerical aspects of the CFD solver. This section focuses on the solver element. A CFD solver can usually be described and envisaged by the solution procedure presented in Figure (VII)
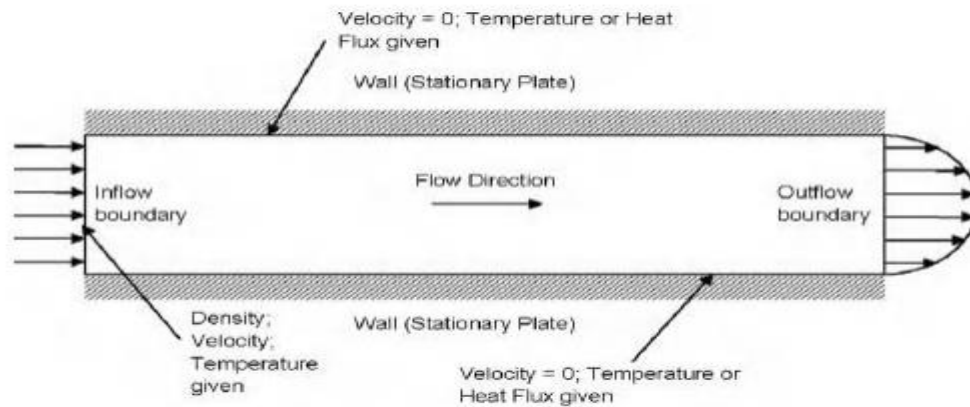

Figure (VI) Boundary conditions for an internal flow problem: CFD problem of flow between two stationary parallel plates.
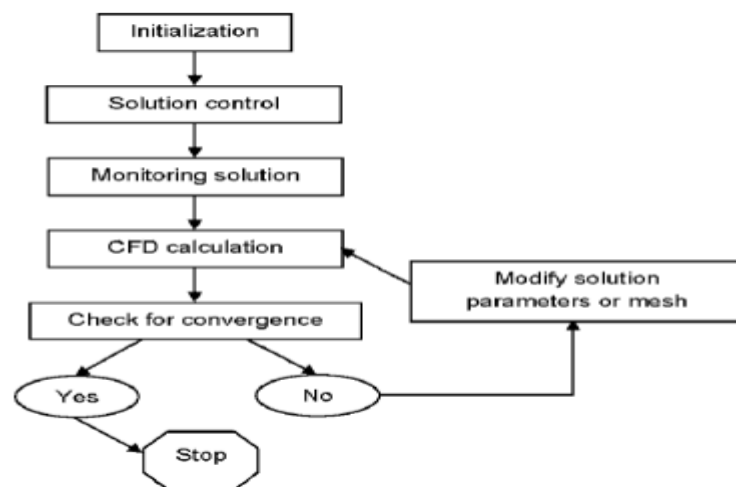

Figure (VII) An overview of the solution procedure

Initialization and Solution Control—Step 5

The fifth step of the CFD analysis encompasses two prerequisite processes within the CFD solver: initialization and solution control. Two reasons that a CFD user should undertake the appropriate selection of initial conditions are

• If the initial conditions are close to the final steady-state solution, the quicker the iterative procedure will converge and yield results in a shorter computational time.

• If the initial conditions are far away from reality, the computations will require longer computational efforts to reach the desired convergence. Also, improper initial conditions may lead to the iterative procedure's misbehaving and possibly "blowing up" or diverging. Second, setting up appropriate parameters in the solution control usually entails the specification of appropriate discretization (interpolation) schemes and selection of suitable iterative solvers.

Monitoring Convergence—Step 6

The sixth step of the CFD solver involves the interlinking operations of three prerequisite processes: monitoring solution, CFD calculation, and checking for convergence. Two aspects that characterize a successful CFD computational solution are convergence of the iterative process and grid independence.

RESULT REPORT AND VISUALIZATION—POST-PROCESS

CFD has a reputation for generating vivid graphic images and, while some of the images are promotional and are usually displayed in stunning and superb colorful output, the ability to present the computational results effectively is an invaluable design tool. X-Y Plots X-Y plots are mainly two-dimensional graphs that represent the variation of one dependent transport variable as compared with another, independent variable. They can usually be drawn by hand or more conveniently by many plotting packages. Such plots are the most precise and quantitative way to present the numerical data. Often, laboratory data are gathered by straight-line traverses. An X-Y plot of a laminar velocity profile at the fully developed region for the Case of flow between two stationary parallel plates is shown in Figure(VIII).

Vector Plots

A vector plot provides the means whereby a vector quantity (usually velocity) is displayed at discrete points by an arrow, whose orientation indicates direction and whose size indicates

magnitude. A vector plot generally presents a perspective view of the flow field in two dimensions. In a three-dimensional flow field, different slices of two-dimensional planes containing the vector quantities can be generated in different orientations to better scrutinize the global flow phenomena
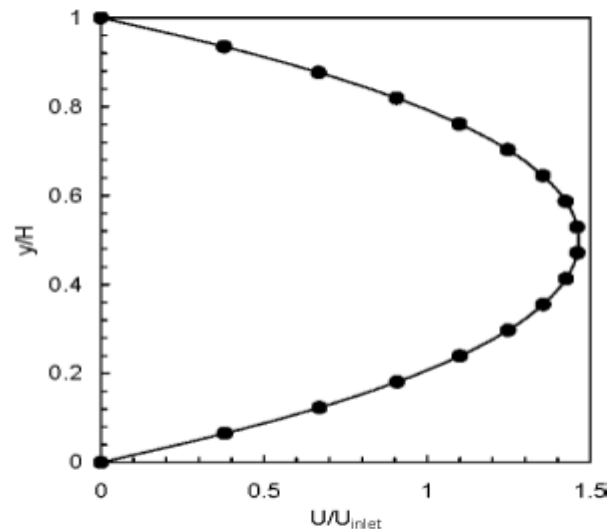

Figure (VIII) X-Y plot of a parabolic laminar velocity profile at the fully developed region for the case of flow between two stationery parallel plate

Figure (IX) Velocity vectors showing the flow development along the parallel-plate channel for Case of flow between two stationery parallel plates

Contour Plots Contour plotting is another useful and effective graphic technique that is frequently utilized in viewing CFD results. The proliferation of contour plots ever since the advent of the computer is not surprising. In CFD, contour plots are one of the most commonly found graphic representations of data. A contour line (also known as an isoline) can be described as a line indicative of some property that is constant in space. The equivalent representation in three dimensions is an isosurface. In contrast to X-Y plots, contour plots, like vector plots, provide a global description of the fluid flow encapsulated in one view.

Experiment 1

Modelling of Symmetric Airfoil Geometry and Generation of Body-Fitted Adaptive Mesh

Aim: In this experiment, we will model symmetrical aerofoil (NACA 0012) and generate body fitted adaptive mesh Using Ansys workbench

**Apparatus:** A computer hardware, software (ANSYS) with a graphical user interface.

Procedure:

Open ANSYS Workbench

Now that we have the pre-calculations, we are ready to do a simulation in ANSYS Workbench! Open ANSYS Workbench by going to Start > ANSYS > Workbench. This will open the start up screen as seen below
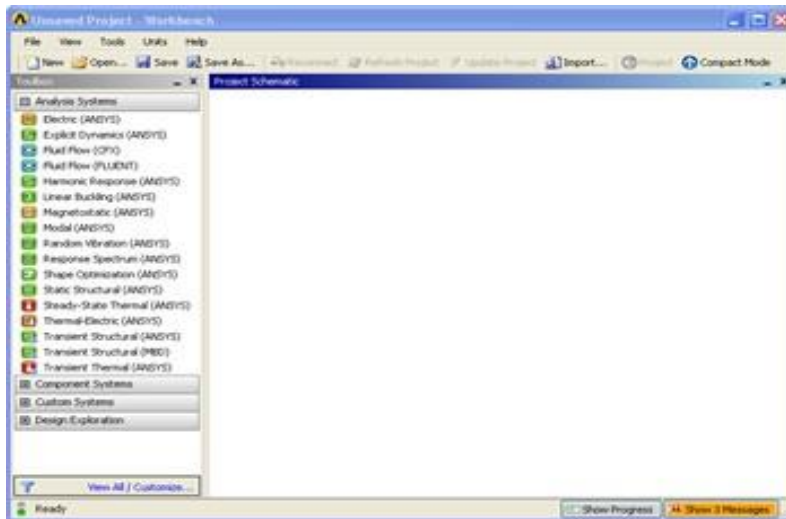
To begin, we need to tell ANSYS what kind of simulation we are doing. If you look to the left of the start up window, you will see the Toolbox Window. Take a look through the different selections. We will be using FLUENT to complete the simulation. Load the *Fluid Flow (FLUENT)* box by dragging and dropping it into the Project Schematic.
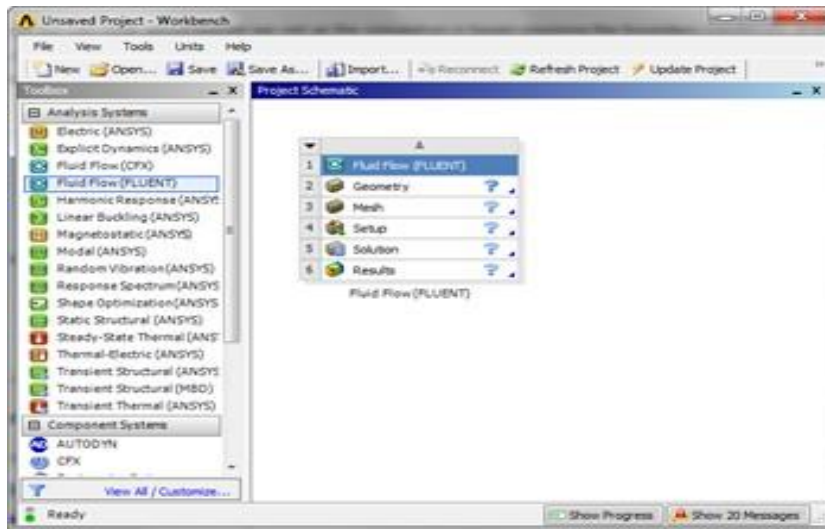
Once you have loaded FLUENT into the project schematic, you are ready to create the geometry for the simulation.

Geometry

Download the Airfoil Coordinates

In this step, we will import the coordinates of the airfoil and create the geometry we will use for the simulation. Begin by downloading this file NACA 0012 airfoil coordinates and saving it somewhere convenient. This file contains the points of a NACA 0012 airfoil.

Launch Design Modeler

Before we launch the design modeler, we need to specify the problem as a 2D problem. Right click ⬚ 2 📦 Geometry   ❓ ◢ and select **_Properties_** . In the _Properties of Schematic A2: Geometry_ Window, select **_Analysis Type > 2D_** . Now, double click ⬚ 2 📦 Geometry   ❓ ◢ to launch the Design Modeler. When prompted, select **_Meters_** as the unit of measurement.

Airfoil

First, we will create the geometry of the airfoil. In the menu bar, go to **_Concept > 3D Curve_**. In the _Details View_ window, click **_Coordinates File_** and select the ellipsis ▣ to browse to a file. Browse to and select the geometry file you downloaded earlier. Once you have selected the desired geometry file, click ⚡ Generate to create the curve. Click ▣ to get a better look at the curve. (Fig. 1.1)
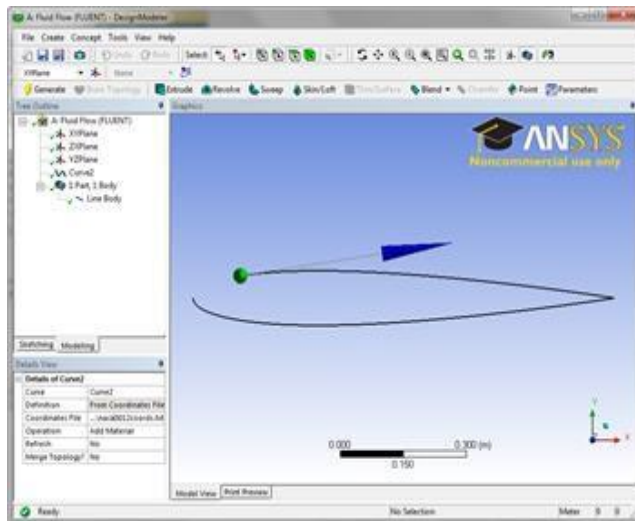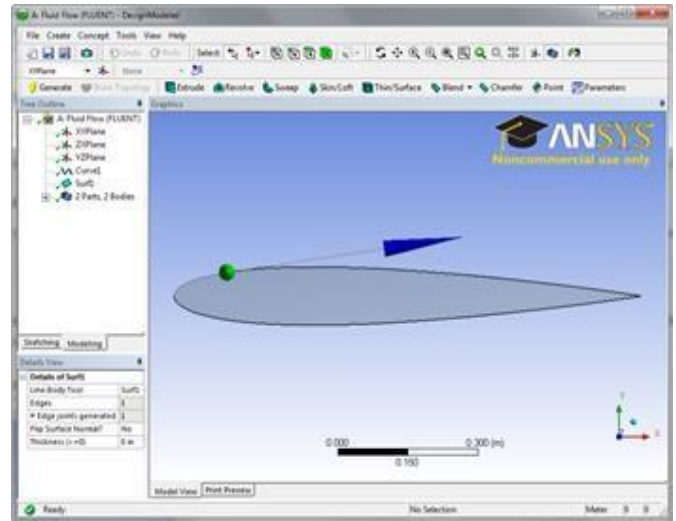
Fig. 1.1

Fig. 1.2

Next, we need to create a surface from the curve we just generated. Go to ***Concepts > Surfaces from Edges***. Click anywhere on the curve you just created, and select ***Edges > Apply*** in the *Details View* Window. Click ⚡ **Generate** to create the surface

Create C-Mesh Domain

Now that the airfoil has been generated, we need to create the meshable surface we will use once we begin to specify boundary conditions. We will begin by creating a coordinate system at the tail of the airfoil - this will help us create the geometry for the C-mesh domain. Click ⚹ to create a new coordinate system. In the *Details View* window, select ***Type > From Coordinates*** . For ***FD11, Point X*** , enter 1. (Fig. 1.3)

Click ⚡ **Generate** to generate the new coordinate system. In the *Tree Outline* Window, select the new coordinate system you created (defaulted to ***Plane 4***), then click 📐 to create a new sketch. This will create a sketching plane on the XY plane with the tail of the airfoil as the origin (Fig.1.4).

Details View	
Details of Plane4	
Plane	Plane4
Sketches	0
Type	From Coordinates
☐ FD11, Point X	1 m
☐ FD12, Point Y	0 m
☐ FD13, Point Z	0 m
☐ FD14, Normal X	0 m
☐ FD15, Normal Y	0 m
☐ FD16, Normal Z	1 m
Transform 1 (RMB)	None
Reverse Normal/Z-Axis?	No
Flip XY-Axes?	No
Export Coordinate System?	No

Fig. 1.3
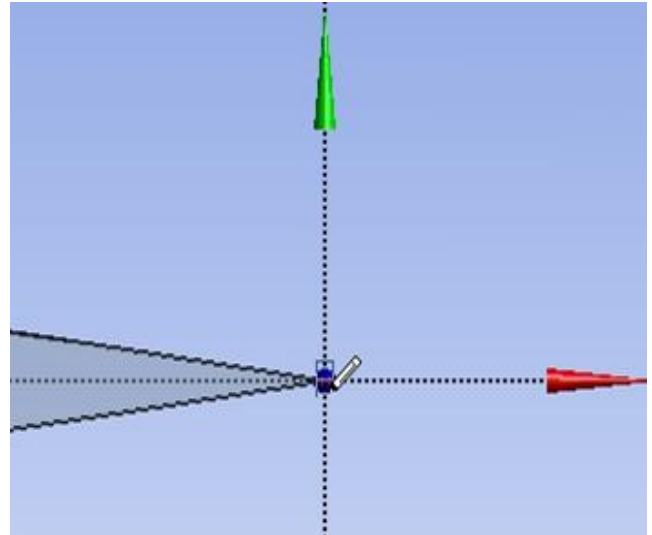
Fig. 1.4

At the bottom of the *Tree Outline* Window, click the ***Sketching*** tab to bring up the sketching window. The first action we will take is create the arc of the C-Mesh domain. Click Arc by Center . The first click selects the center of the arc, and the next two clicks determine the end points of the arc. We want the center of the arc to be at the tail of the airfoil. Click on the origin of the sketch, making sure the P symbol is showing

For the end points of the arc, first select a point on the vertical axis above the origin (a C symbol will show), then select a point on the vertical axis below the origin. You should end up with the following (Fig. 1.5)

To create the right side of the C-Mesh domain, click Rectangle by 3 Points . Click the following points to create the rectangle in this order - where the arc meets the positive vertical axis, where the arc meets the negative vertical axis, then anywhere in the right half plane. The final result should look like this (Fig.1.6) .Now, we need to get rid of necessary lines created by the rectangle.
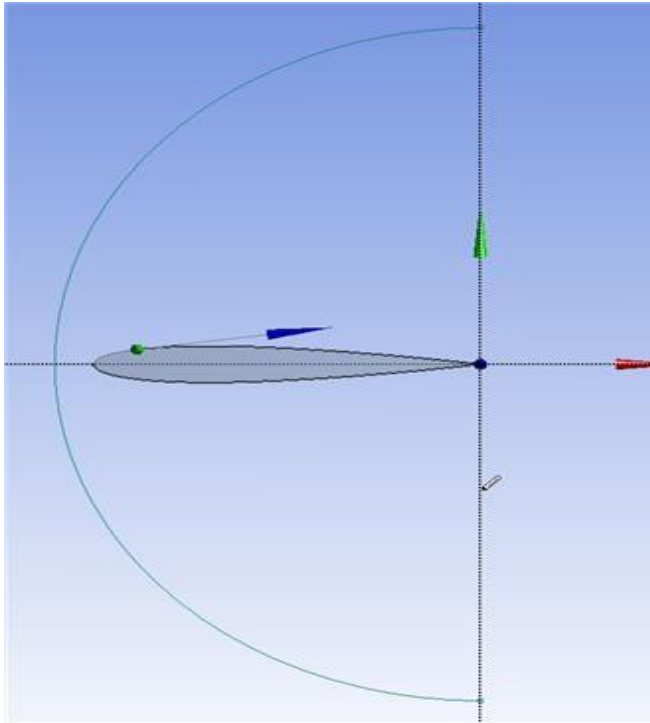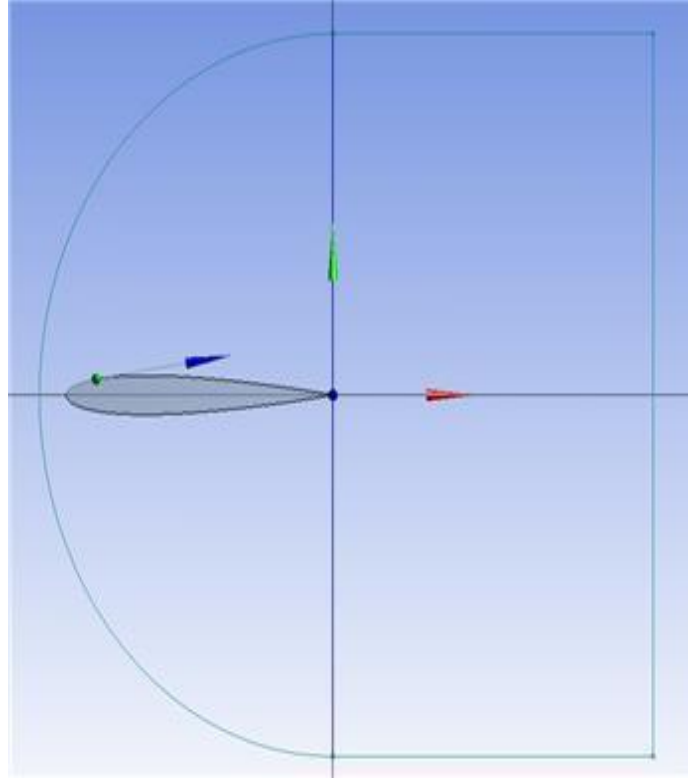
Figure 1.5   Figure 1.6

Select ***Modify*** in the *Sketching Toolboxes* window, then select ⊤ Trim . Click the lines of the rectangle the are collinear with the positive and negative vertical axis. Now, select the ***Dimensions*** toolbox to dimension the C-Mesh domain. Click ⌒ Radius , followed by the arc to dimension the arc. Assign the arc a value of 12.5. Next, select ⊢⊣ Horizontal . Click the vertical axis and the vertical portion of the rectangle in the right half plane. Also assign the horizontal dimension a value of 12.5.

Next, we need to create a surface from this sketch. To accomplish this, go to ***Concept > Surface From Sketches***. Click anywhere on the sketch, and select ***Base Objects > Apply*** in the *Details View* Window. Also, select ***Operation > Add Frozen*** . Once you have the correct settings, click ⚡ Generate . The final step of creating the C-Mesh is creating a surface between the boundary and the airfoil. To do this, go to ***Create > Boolean***. In the *Details View* window, select ***Operation > Subtract*** . Next, select ***Target Bodies > Not selected*** , select the large C-Mesh domain surface, then click ***Apply*** . Repeat the same process to select the airfoil as the ***Tool Body*** . When you have selected the bodies, click ⚡ Generate .
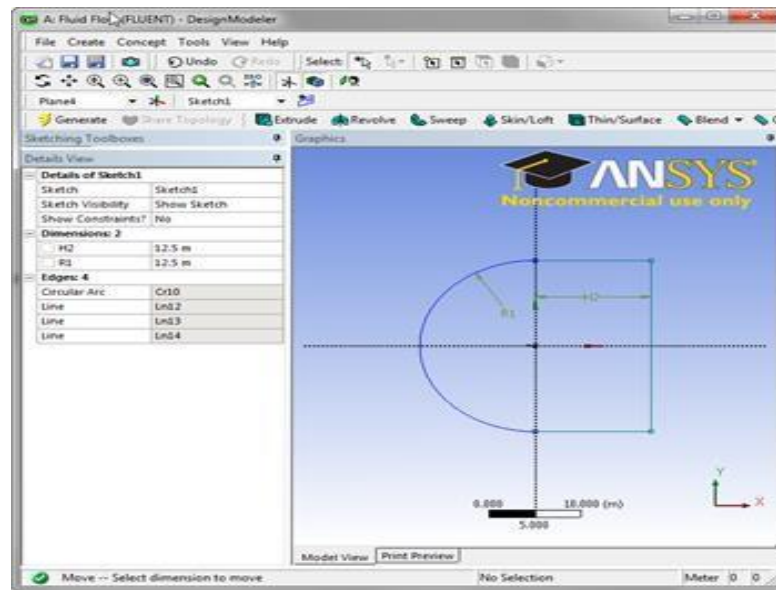
Figure 1.7

Create Quadrants

In the final step of creating the geometry, we will break up the new surface into 4 quadrants; this will be useful for when we want to mesh the geometry. To begin, select **Plane 4** in the *Tree Outline* Window, and click 🔲. Open the sketching menu, and select ⟍ Line . Draw a line on the vertical axis that intersects the entire C mesh. Trim away the lines that are beyond the C-Mesh, and you should be left with this: (Fig. 1.9)
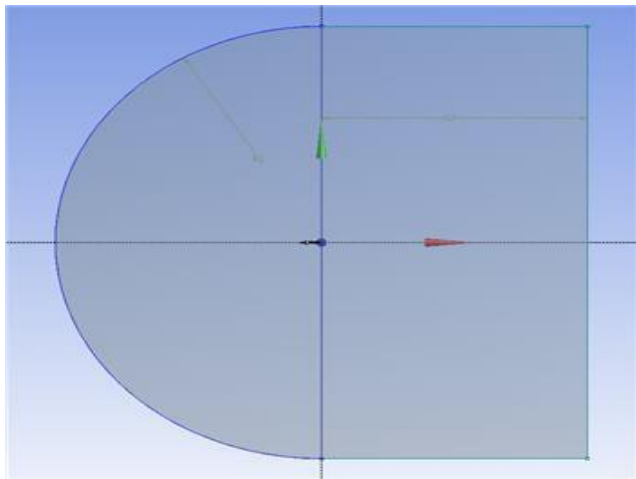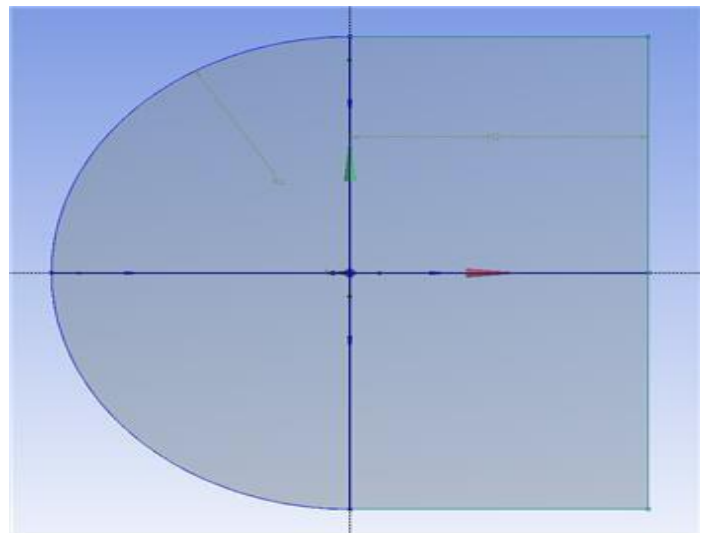

Figure 1.8

Figure 1.9

Next, go to *Concepts > Lines from Sketches*. Select the line you just drew and click *Base Objects > Apply* , followed by ⚡ Generate . Now that you have created a vertical line, create a new sketch and repeat the process for a horizontal line that is collinear to horizontal axis and bisects the geometry. Now, we need to project the lines we just created onto the surface. Go to *Tools > Projection* . Select *Edges* press Ctrl and select on the vertical line we drew (you'll have to select both parts of it), then press *Apply* . Next, select *Target* and select the C-Mesh surface, then click *Apply* .

Once you click ⚡ Generate , you'll notice that the geometry is now composed of two surfaces split by the line we selected. Repeat this process to create 2 more projections: one projection the line left of the origin onto the left surface, and one projecting the right line on the right surface (Fig. 1.10). When you're finished, the geometry should be split into 4 parts.
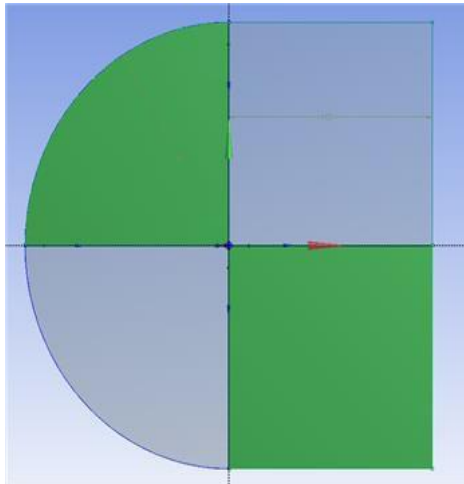

Figure 1.10

Suppress the line bodies by right clicking in the tree. You only need the surface body to be transferred to the Mesher. The geometry is finished. Save the project and close the design modeler, as we are now we are ready to create the mesh for the simulation.

Create a Structured Mesh
Mapped Face Meshing

First, we will apply a mapped face meshing control to the geometry. In the *Outline* window, click on *Mesh* to bring up the Meshing Toolbar. In the Meshing Toolbar, select *Mesh Control > Mapped Face Meshing*. Making sure the face selection filter is selected ⬚ , select all four faces by holding down the right mouse button and dragging the mouse of all of the quadrants of the geometry. When all of the faces are highlighted green, in the *Details view* Window select *Geometry > Apply*. Next, select
Edge Sizing

Next, we will apply edge sizing controls to all of the edges of the mesh. To begin, go to *Mesh Control > Sizing*. Next, click the edge selection filter 🔲. Select the following 4 edges buy holding Ctrl and using the left mouse button:
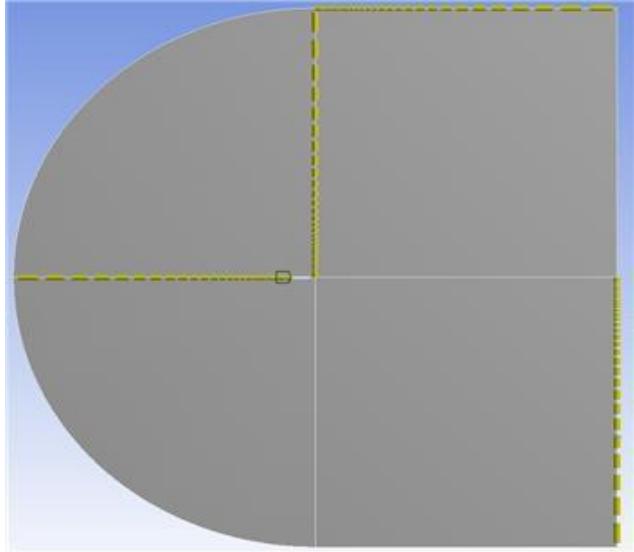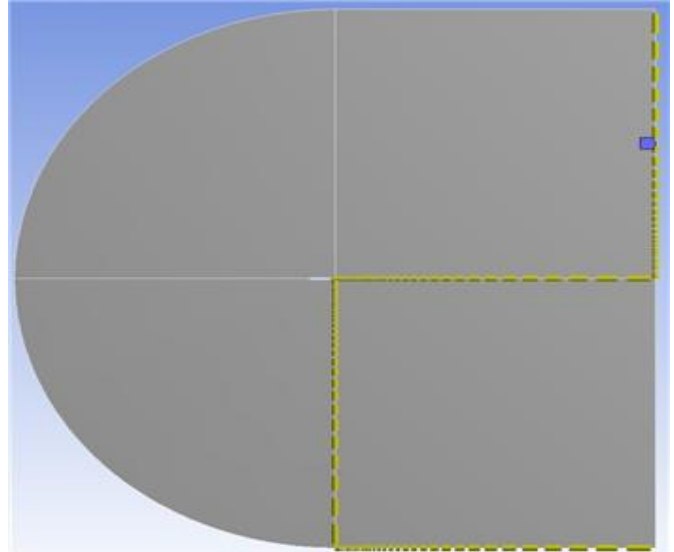

Fig. 1.11

Fig. 1.12

Once the edges are selected, in the *Details View* Window select *Geometry > Apply*. Next, select *Type > Number of Divisions*. Change the *Number of Divisions* to 50. Select *Behavior > Hard*. (Fig. 1.11).We also want the mesh to have a bias, so select the first bias type: *Bais > --* ,and give the edge sizing a *Bias Factor* of 150. The Edge sizing should now look like this: (Fig. 1.11)

Notice that the element sizes get smaller towards the airfoil. This will give us a better resolution around the airfoil where the flow gets more complicated. Create a new edge sizing with the same parameters, but choose the 4 remaining straight edges (see fig. 1.12). The number of divisions will still be 50, but now will be selecting a different biasing type by selecting the second Bias option: *Bias > - - — -----*. Again, set the *Bias Factor* to 150

Edge Bias

It is important to make sure that the edge divisions to this point are *biased towards the center of the mesh*: otherwise you may run into some problems later. If your mesh does not match the pictures in the tutorial, make sure to change the parameters of the mesh to make sure that they do: this might mean choosing different edges for the different biasing types than those outlined in this tutorial.
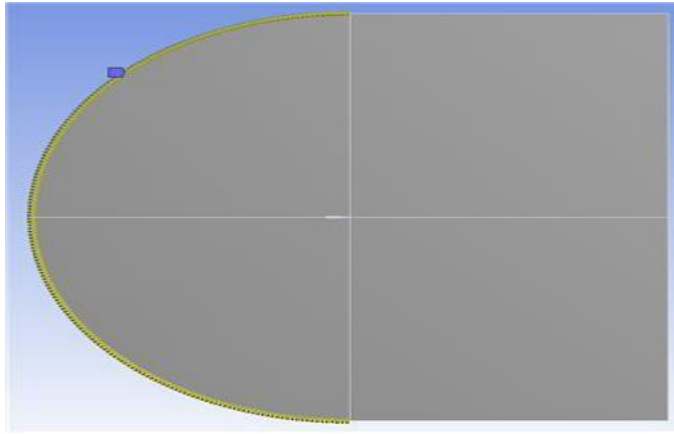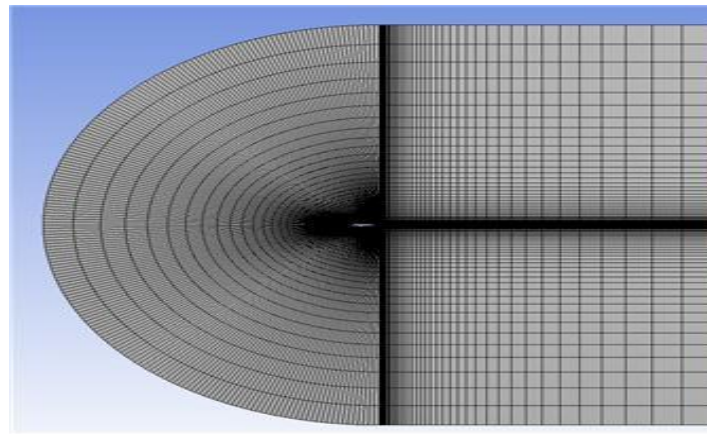
Fig. 1.13   Fig. 1.14

Finally, create a third edge sizing, and select the rounded edges as the geometry. Again, select *Type > Number of Divisions*, and change *Number of Divisions* to 100. Select *Behavior > Hard*. This time, we will not bias the edges. (Fig. 1.13).Now, select *Mesh > Generate* to generate the mesh. It should look like this. (Fig. 1.14)

Now will assign names to some of the edges to make creating boundary conditions for the mesh easier. Let's recall the boundary conditions we planned in the Pre-Analysis Step:
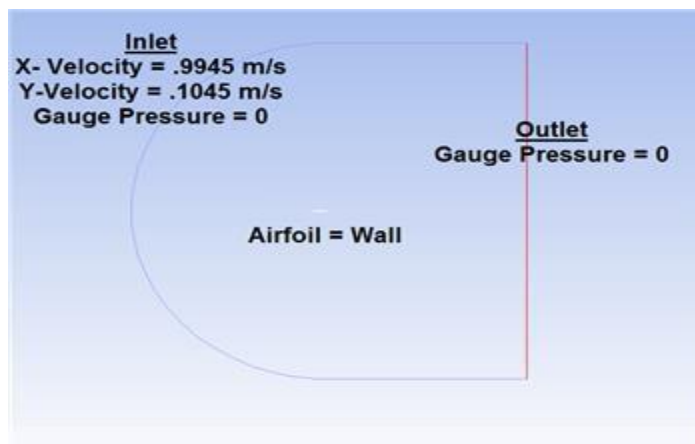

Fig. 1.15

The edges highlighted blue are the inlet, the edges highlighted red are the outlet (Fig.1.15), and the airfoil is highlighted white in the middle. Now we are ready to name the sections. In the *Outline* window, select geometry - this will make seeing the edges a little easier.

Again make sure the edge selection tool is selected. Now, select the two vertical edges on the far right side of the mesh. Right click, and select *Create Named Selections*. Name the edges outlet. Next, select the edges that correspond to the inlet of the flow as defined by the picture above. Again, right click and select *Create Named Selections* and this time name the selection inlet. Finally, select the two edges making up the airfoil, and name the selection airfoil.

Select File>>Export>>Mesh>>FLUENT Input File>>Export from the menu. Enter airfoil-flow-mesh as file name and Save as type: FLUENT Input Files (*.msh). Close the meshing window. Right click on Mesh and select Update in Ansys Workbench.

VIVA QUESTIONS

1. What is a symmetric airfoil and how is it identified?
2. How do you generate the geometry of a symmetric airfoil in ANSYS
3. What is the significance of choosing a C-type or O-type domain around an airfoil?
4. Why is it necessary to keep the far-field boundary at a sufficient distance from the airfoil?
5. What is meant by a body-fitted mesh, and why is it important in CFD simulations of airfoils?
6. What are the advantages of using adaptive mesh refinement in CFD simulations?
7. Which mesh quality parameters should be checked before starting the simulation?
8. What boundary conditions are applied at the airfoil surface, inlet, outlet, and far-field?
9. What is the importance of symmetry in modeling
10. How does the angle of attack affect the flow behavior over a symmetric airfoil?

<div align="center">

Experiment 2

Modelling of cambered Airfoil Geometry and Generation of Body-Fitted Adaptive Mesh

</div>

Aim: In this experiment, we will model a cambered aerofoil (NACA 2412) and generate body fitted adaptive mesh Using Ansys workbench

Apparatus: A computer hardware, software (ANSYS) with a graphical user interface.

Procedure:

Launching Ansys Workbench and Selecting Fluent

Start by launching Ansys Workbench. Double click on Fluid Flow (Fluent) that is located under Analysis Systems in Toolbox.

Right click Geometry and select Properties. In Properties of Schematic A2: Geometry, select Analysis Type 2D under Advanced Geometry Options. Right-click on Geometry in the Project Schematic window and select New Design Modeler Geometry. Select Units>>Millimeter from the menu as the desired length unit in the new Design Modeler window.

Select Create >>Point from the menu. We will use the coordinates file NACA 2412 Airfoil coordinates**.** Open the Coordinates File from Details View and click on Generate. Select Look at Face Select Concept>>Lines From Points in the menu. Right click in the graphics window and select Point Chain. Right click in the graphics window and select All. Apply the Point Segments in Details View. Click on Generate. Select Concept->Surfaces From Edges from the menu (Fig. 2.1). Right click in the graphics window and Select All. Apply Edges in Details View. Click on Generate.

Next, we will be sketching the mesh region around the airfoil. Create a new coordinate system by first clicking on New Plane Select Type>>From Coordinates in the Details View (Fig. 2.2). Enter 230 mm for FD11, Point X. Click on Generate. Select the new Plane4 in the tree outline and click on to generate a new sketch. Click on the Sketching tab and select Arc by Center from the Draw tab.
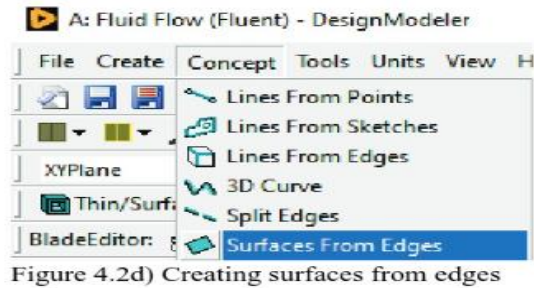
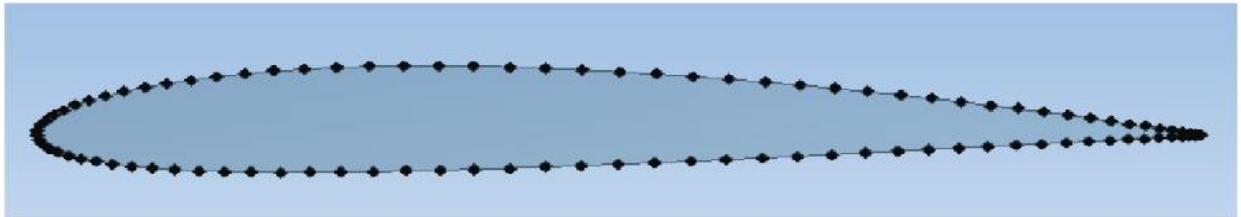Figure 4.2d) Creating surfaces from edges

Figure 4.2e) NACA 2412 Airfoil in DesignModeler

Fig. 2.1

Click on Look At Face/Plane/Sketch Click on the origin of the coordinate system at the trailing edge of the airfoil. Make sure that you see the letter P when you move the cursor over the origin of the coordinate system. Next, click on the vertical axis above the origin (make sure you have a C) and finally on the vertical axis below the origin (make sure you have a C once again), see Fig. 2.3).
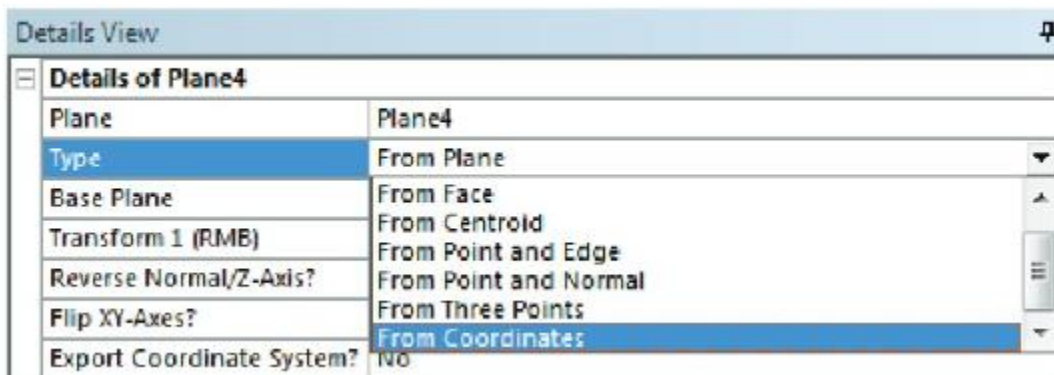

Fig. 2.2

Select Rectangle by 3 Points from the Draw tab. Click at the intersection of the arc and the positive vertical axis, at the intersection between the arc and the negative vertical axis, and finally click in the right-hand side plane. Select Modify and Trim from Sketching Toolboxes. Click on the lines of the rectangle that are aligned with and on top of the vertical axis (Fig. 2.4).
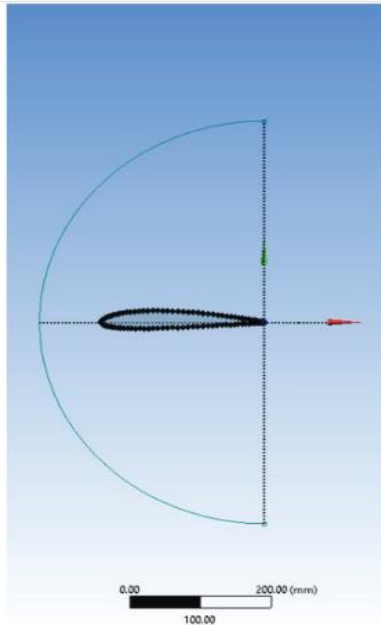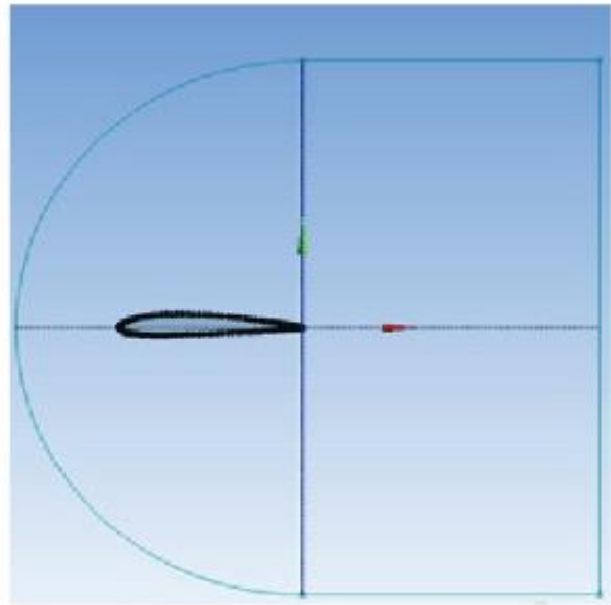
Fig. 2.3

Fig. 2.4

Select Dimensions under Sketching Toolboxes, select Radius under Dimensions and select the arc in the graphics window. Set the radius of the arc to 2875 mm which gives a ratio of the radius of the arc to the chord length of 12.5. Next, select Horizontal under Dimensions from the Sketching Toolboxes and click on the vertical axis and the vertical edge of the rectangle on the right-hand side. Enter 5750 mm as the length (Fig. 2.5)
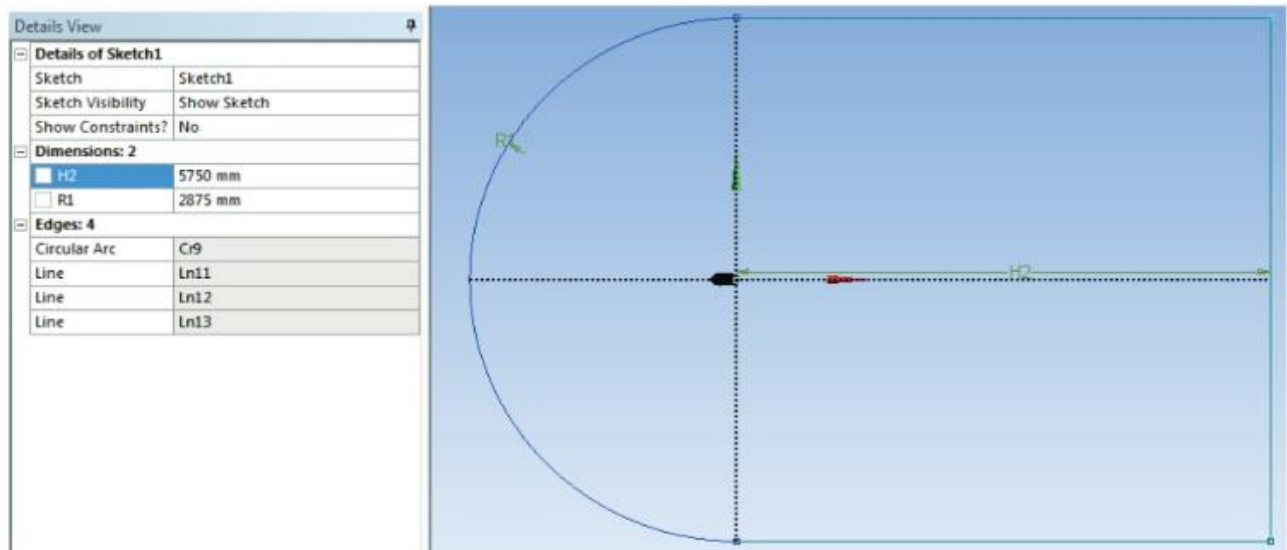

Fig. 2.5

Select Concept>>Surface from Sketches. Select Sketch1 under Plane4 in the Tree Outline and Apply as Base Objects in Details View. Select Operation>>Add Frozen in Details View followed by Generate. (Fig. 2.6)
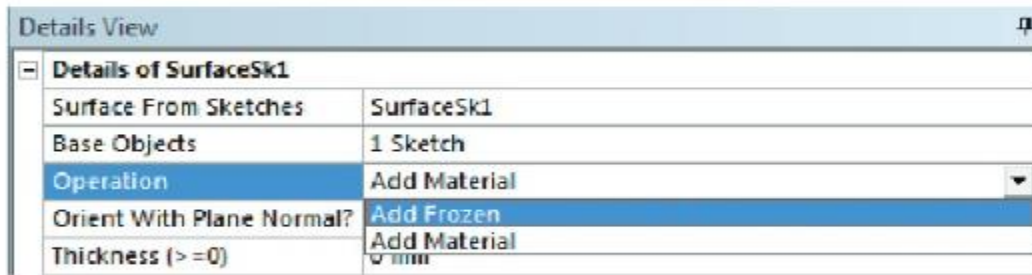

Fig. 2.6

Select Create>>Boolean from the menu. Select Subtract Operation from the Details View. Click on the mesh region in the graphics window and click on Apply as Target Bodies in Details View. Zoom in on the airfoil, select the Airfoil as Tool Bodies in Details View and click on Generate, see Fig. 2.7 a-b

Select Plane 4 in the tree outline and create a new sketch Select the Sketching tab and the Line tool under Draw. Draw a line on the vertical axis from top to bottom of the mesh that intersects the entire mesh region. Select Concepts>>Lines from Sketches from the menu. Select the newly created vertical line and Apply it as a Base Object in the Details View. Click on Generate.
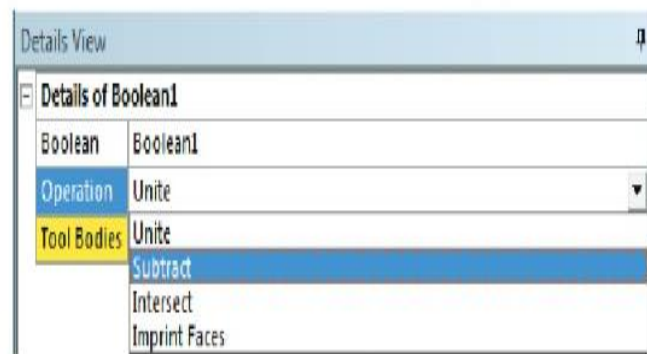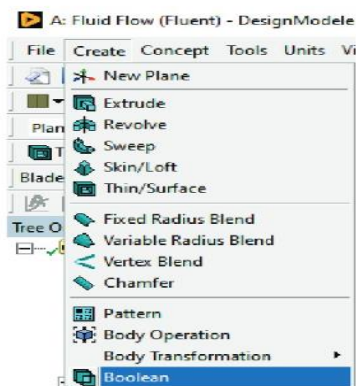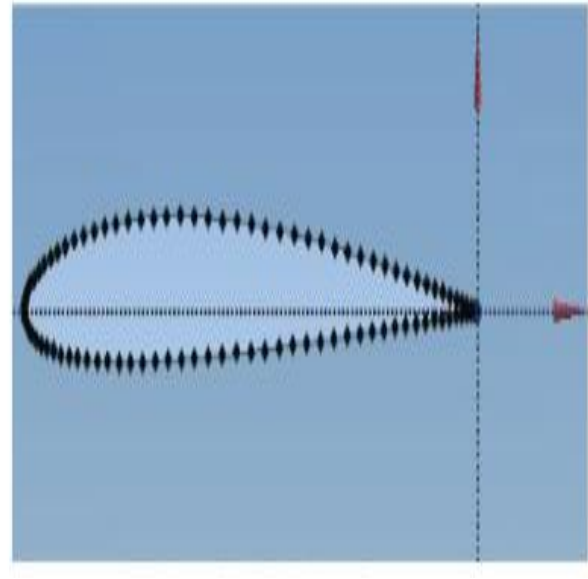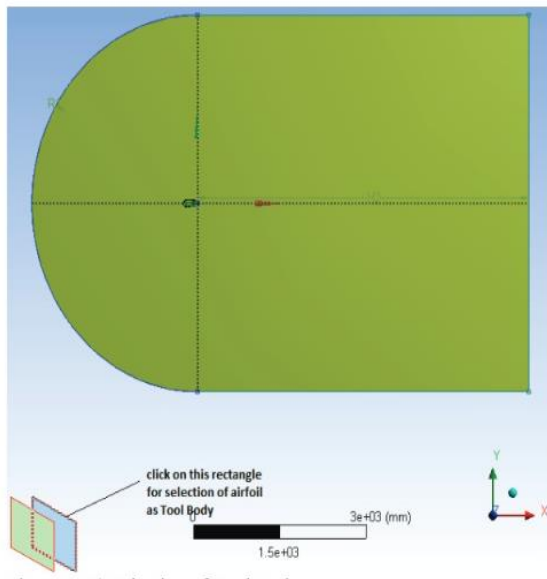

Fig. 2.7 a

Fig. 2.7 b

Repeat this step as described above and create a new sketch with a line on the horizontal axis in Plane4 that is drawn from left to right and goes through the whole mesh region including the leading and trailing edges of the airfoil. You may need to create a Coincident Constraint using the Sketching Toolbox and insert the constraint between the new line and the horizontal axis. This is completed by selecting the Coincident Constraint and clicking on both the new line and the horizontal axis. It may be helpful but not necessary to draw the horizontal line a little bit below the horizontal axis and use the constraint to fix the line with the horizontal axis. Make sure you have the letter C show up when you start and end the horizontal line. Select Concepts>>Lines from Sketches from the menu. Select the newly created horizontal line and Apply it as a Base Object in the Details View. Click on Generate.

Select Tools Projection from the menu.(Fig. 2.9). Select Edges on Face as Type in Details View. Control- select both parts of the newly created vertical line and the two parts of the newly created horizontal line and Apply the four Edges in the Details View. Click in the yellow region next to Target in Details View. Select the mesh region around the airfoil and Apply it as the Target under Details View. Click Generate. You should now have four different mesh regions (Fig. 2.10)
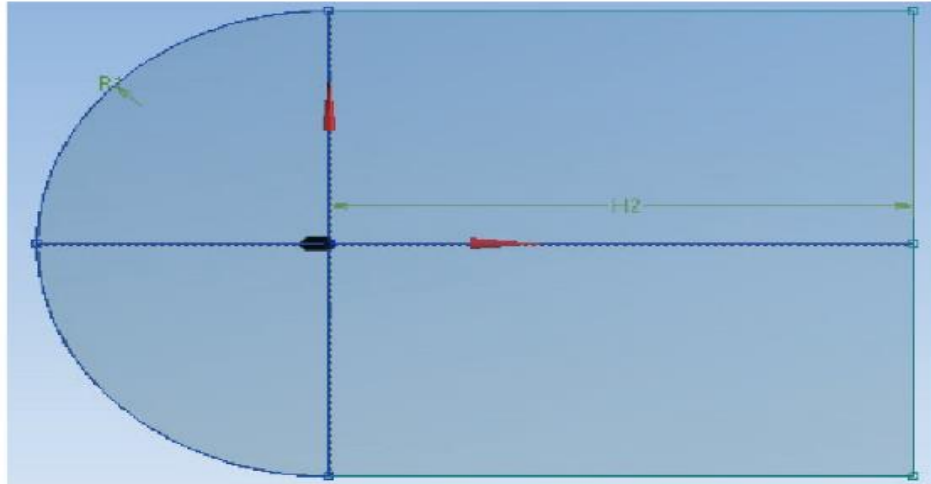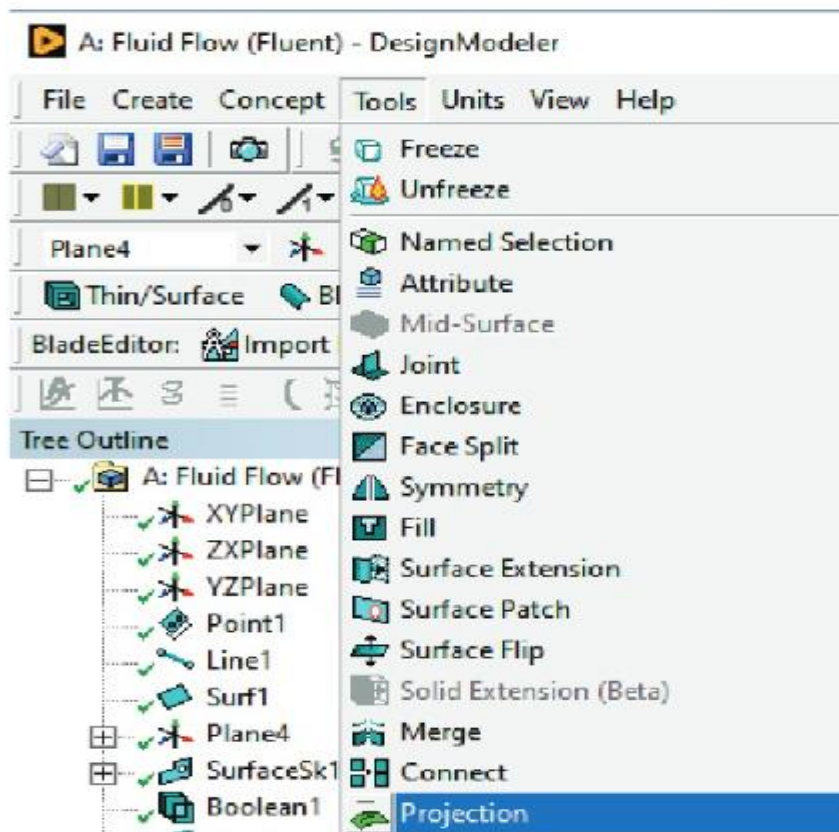
Fig. 2.8

Fig. 2.9

Click on the plus sign next to 2 Parts, 2 Bodies in the Tree Outline. Right click the Line Body and select Suppress Body. Select File>>Save Project from the menu and save the project with the name NACA2412 Airfoil Flow Study. Close Design Modeler
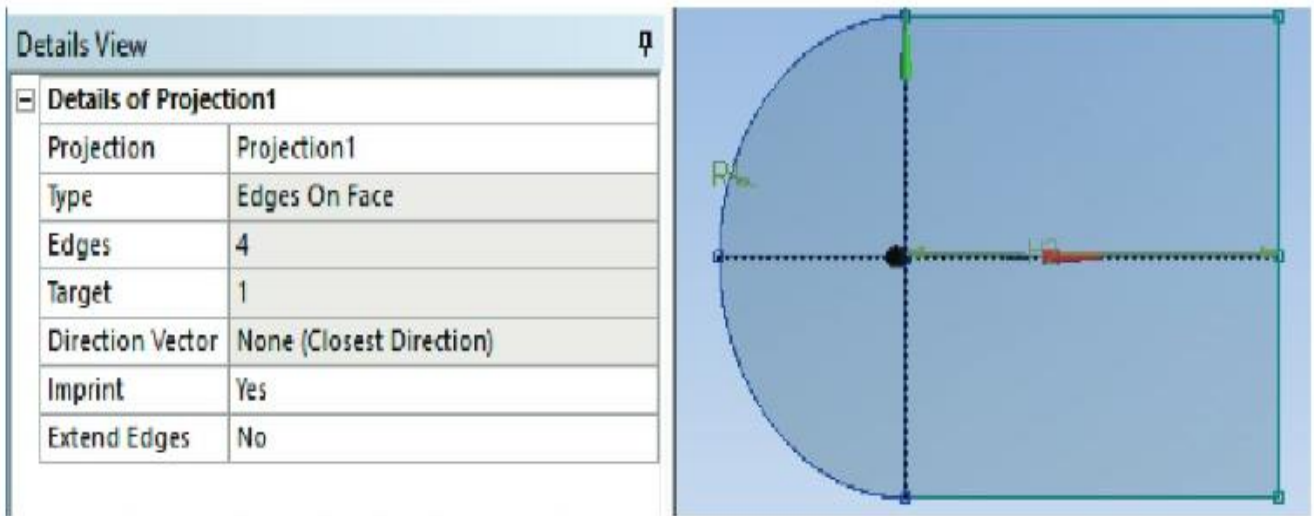
Fig. 2.10

Launching Ansys Meshing

Double click on Mesh under Project Schematic in Ansys Workbench. Select Mesh in the Outline of the Meshing window and select Mesh>>Controls Face Meshing from the menu. Control- select all four faces of the mesh. Apply the Geometry under Details of "Face Meshing". (Fig. 2.11)
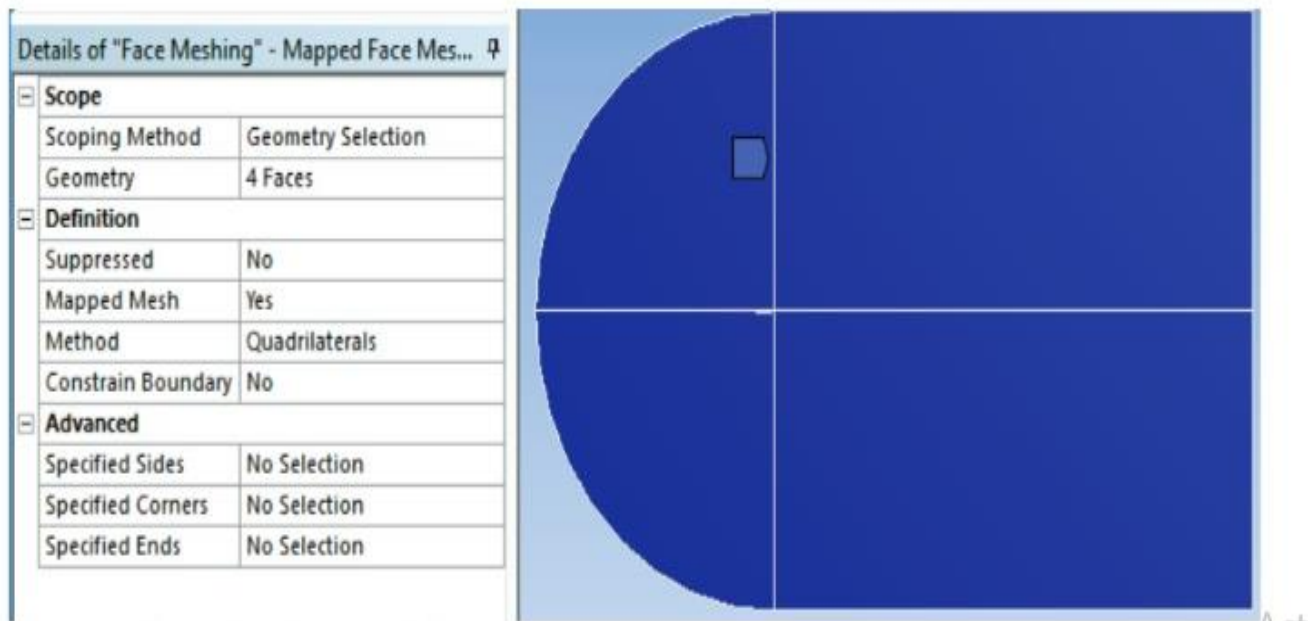

Fig. 2.11

Select Mesh Control>>Sizing from the menu and click on the Edge Selection Filter Control-select the eight edges as shown in Fig.2.12). Apply the Geometry under Details of "Edge Sizing". Select Number of Divisions as Type under Details of "Edge Sizing". Set the number of Divisions to 50. Set Capture Curvature to No and set the Behavior as Hard. Select the first Bias Type. Enter Bias Factor 150. Control select the horizontal edge behind the airfoil, the horizontal edge at the bottom of the mesh domain, the upper vertical edge at the outlet and the lower vertical edge under the airfoil. Apply the four edges as Reverse Bias.
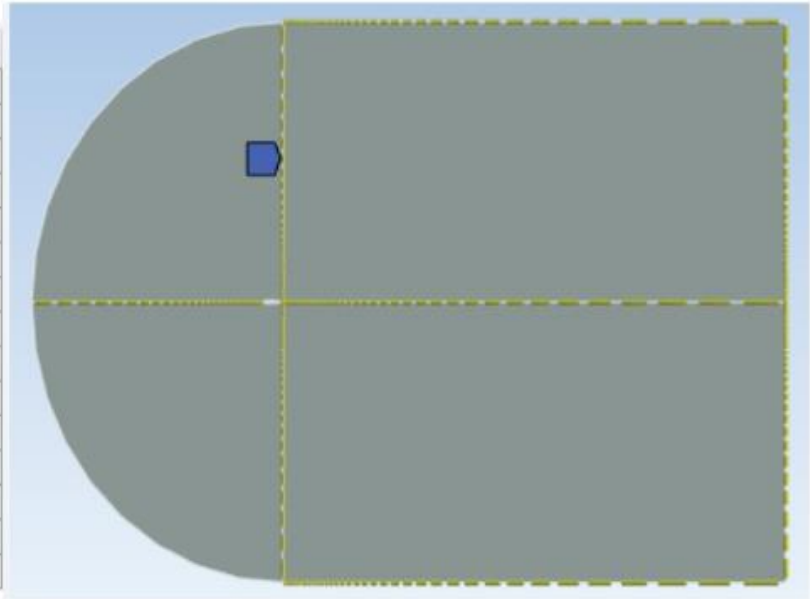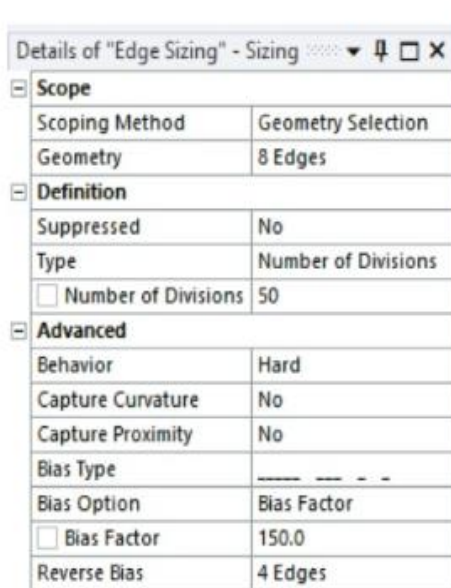

Fig. 2.12

The second edge sizing will be for the two round edges. Set the Number of Divisions to 100, Capture Curvature to No and Hard behavior. No bias will be used for the round edges. Right click on Mesh in the tree outline and select Generate Mesh. (Fig. 2.13)

Select Geometry in the Outline. Select the Edge Selection filter Control-select the two vertical edges on the right-hand side of the mesh region, right click and select Create Named Selection. Name the edges "outlet" and click OK. Name the two round edges "inlet". Name the horizontal upper and lower edges "symmetry". (Fig. 2.14). Save the project.
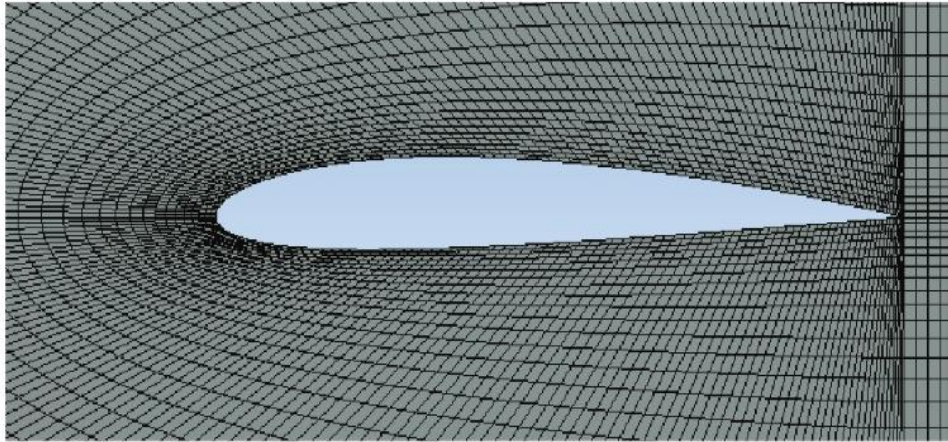
Fig. 2.13

Select File>>Export>>Mesh>>FLUENT Input File>>Export from the menu. Enter airfoil-flow-mesh as file name and Save as type: FLUENT Input Files (*.msh). Close the meshing window. Right click on Mesh and select Update in Ansys Workbench.
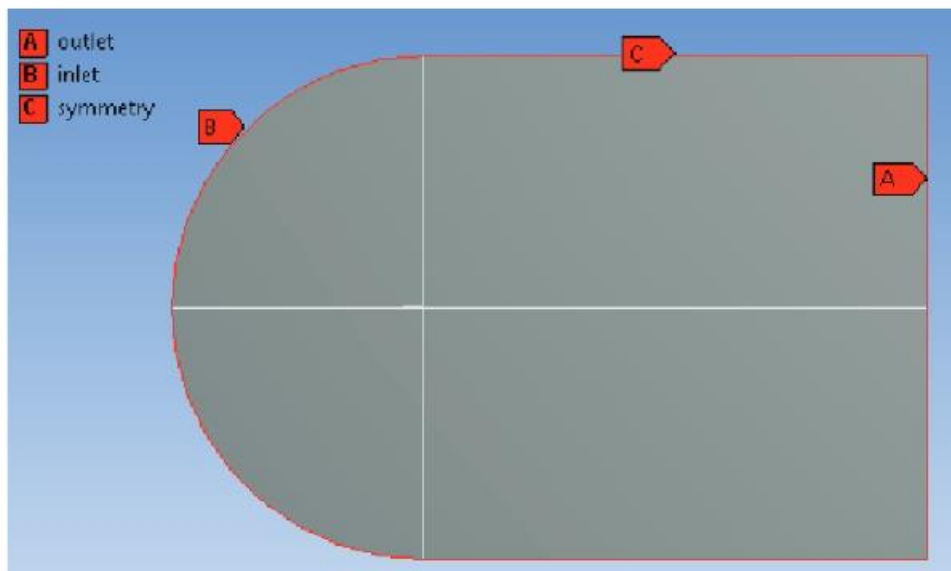
Fig. 2.14

VIVA QUESTIONS

1. What is a cambered airfoil?
2. How does a cambered airfoil differ from a symmetric one?
3. Which airfoil did you model in this experiment?
4. What software did you use to create the airfoil geometry?
5. What is a body-fitted mesh?
6. Why is a body-fitted mesh important for airfoil simulations?
7. What is adaptive meshing in CFD?
8. How did you define the far-field boundary in your model?
9. What type of boundary conditions did you apply?
10. How do you check mesh quality in ANSYS?

Experiment 3

Modelling of 2D incompressible and inviscid flow over symmetric airfoil and plotting of pressure distribution and velocity vectors for subsonic mach nos

Aim: In this experiment, we will model 2D incompressible and inviscid flow over symmetric airfoil (NACA 0012) and plotting of pressure distribution and velocity vectors for subsonic mach nos
Apparatus: A computer hardware, software (ANSYS) with a graphical user interface.
Procedure:

In this step, we will open fluent and define the boundary conditions of the problem. We will be use the model and meshed NACA 0012 airfoil for this experiment. Open Ansys Fluent and   load the mesh of NACA 0012 created in experiment 1 into FLUENT. Now, double click *Setup*. The *Fluent Launcher* Window should open. Check the box marked *Double Precision*. To make the solver run a little quicker, under *Processing Options* we will select *Parallel* and change the *Number of Processes* to 2. This will allow users with a double core processor to utilize both. (Fig. 3.1)

Fig. 3.1

Click *OK* to launch Fluent. The first thing we will do once Fluent launches is define the solver we are going to use. Select *Problem Setup > General*. Under *Solver*, select *Pressure-Based*.

Next, we will define the model we are going to use. We do this by going *Problem Setup > Models > Viscous-Laminar*. Then press *Edit...* This will open the *Viscous Model* Menu Window. Select *Inviscid* and press *OK*. Now, we will specify characteristics of the fluid. Because we specified the fluid as inviscid, we will only have to define the density of the fluid.
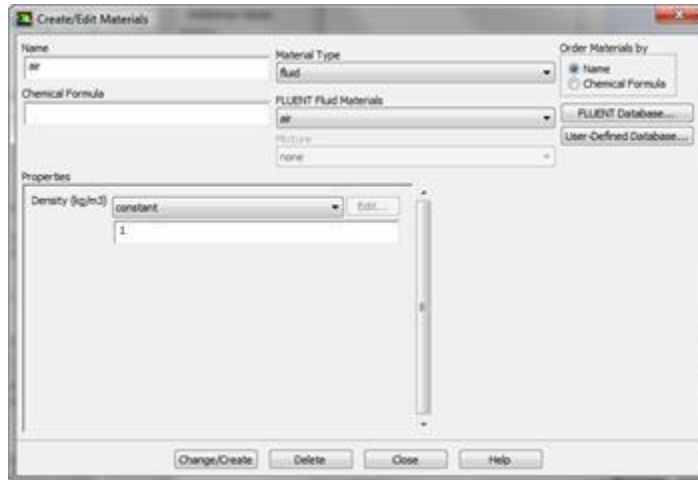
Fig. 3.2

To make matters even simpler, we are only looking for non-dimensionalized values like pressure coefficient, so we will define the density of our fluid to be 1 kg/m^3. To define the density, click *Problem Setup > Materials > (double click) Air*. This will launch the *Create/Edit Materials* window.

Under *Properties*, ensure that density is set to *Constant* and enter 1 kg/m^{3}as the density. Click *Change/Create* to set the density. (Fig. 3.2). Now that the fluid has been described, we are ready to set the boundary conditions of the simulation. Bring up the boundary conditions menu by selecting *Problem Setup > Boundary Conditions*. In the *Boundary Conditions* window, look under *Zones*. First, let's set the boundary conditions for the inlet. Select *Inlet* to see the details of the boundary condition. The boundary condition type should have defaulted to *velocity-inlet*: If it didn't, select it. Now, click *Edit* to bring up the *Velocity-Inlet* Window. We need to specify the magnitude and direction of the velocity. Select *Velocity Specification Method > Components*.
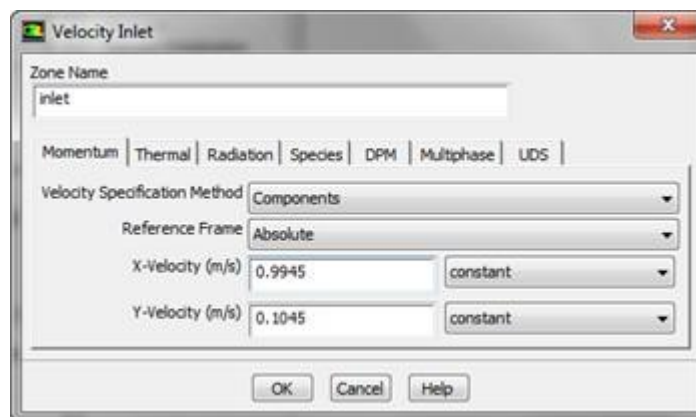

Fig 3.3

Remember, we want the flow to enter the inlet at an angle of 6 degrees since the angle of attack of the airfoil is 6 degrees; thus, the x velocity will be cosθ, and the y velocity will be sinθ. Specify *X-Velocity* as 0.9945 m/s and *Y-Velocity* as 0.1045 m/s. When you have finished specifying the velocity as entering the inlet at 6 degrees (the same thing as having an angle of attack of 6 degrees), press *OK (Fig. 3.3)*

In the *Boundary Conditions* window, look under *Zones*. Select *Outlet* to see the details of the boundary condition. The boundary condition type should have defaulted to *pressure-outlet*: if it didn't, select it. Click *Edit*, and ensure that the *Gauge Pressure* is defaulted to 0. If it is, you may close this window. In the *Boundary Conditions* window, look under *Zones* and select *airfoil*. Select *Type > Wall* if it hasn't been defaulted.

The final thing to do before we move on to solution is to acknowledge the reference values. Go to *Problem Setup > Reference Values*. In the *Reference Values* Window, select *Compute From > Inlet*. Check the reference values that appear to make sure they are as we have already set them.

First, go to *Solution > Solution Methods*. Everything in this section should have defaulted to what we want, but let's make sure that under *Flow* the selection is *Second Order Upwind*. If this is the selection, we may move on. Now we are ready to begin solving the simulation. Before we hit solve though, we need to set up some parameters for how Fluent will solve the simulation.

Let's begin by going to *Solution > Monitors*. In the *Monitors* Window, look under *Residuals, Statistic, and Force Monitors*. Select *Residuals - Print, Plot* and press *Edit*. In the *Residual Monitors* Window, we want to change all of the *Absolute Criteria* to 1e-6. This will give us some further trust in our solution. (Fig. 3.4)
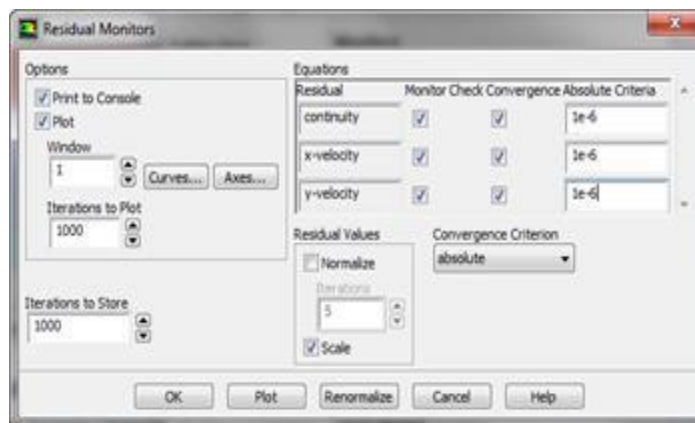

Fig. 3.4

Now, we need to initialize the solution. Go to *Solution > Solution Initialization*. In the *Solution Initialization* Window, select *Compute From > Inlet*. Ensure the values that appear are the same values we inputted in Step 5. If they are, initialize the solution by clicking *Initialize*.

Once the solution has been initialized, we are ready to solve the simulation. Go to *Solution > Run Calculation*. Change *Number of Iterations* to 3000, then double click *Calculate*. Sit back and twiddle your thumbs until Fluent spits out a converged solution.

To view the pressure contours over the entire mesh, go to *Results > Graphics and Animations* again, and in the *Graphics and Animations* Window, select *Contours*. Click *Set Up...* to bring up the *Contours* Menu. Check the box next to **Filled**. Under *Contours Of*, ensure that the two boxes that are selected are *Pressure...* and *Static Pressure*. Once these parameters are set, press *Display* to see the pressure contours. (Fig. 3.5)
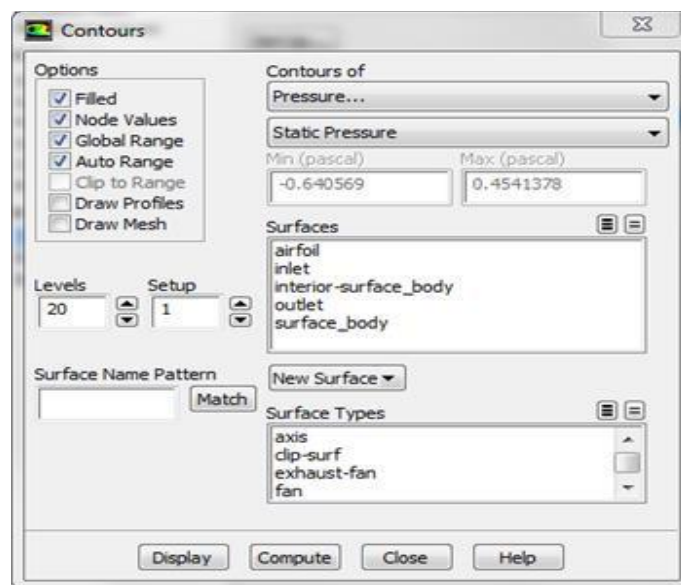
Fig. 3.5

Next, we will look at the velocity vectors of the solution to see if they make intuitive sense. To plot the velocity vectors, go to *Results > Graphics and Animations*. In the *Graphics and Animations* Window, select *Vectors* and click *Set Up.* This will bring up the *Vectors* Menu. (Fig. 3.6)
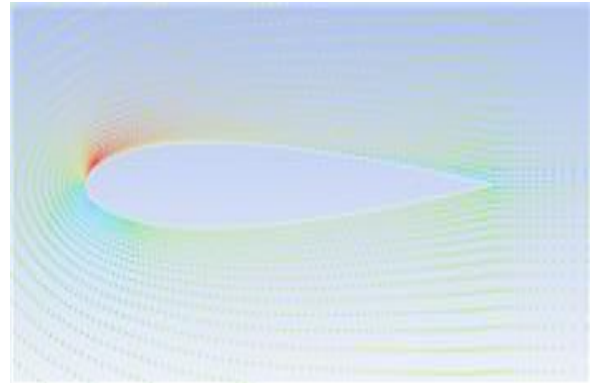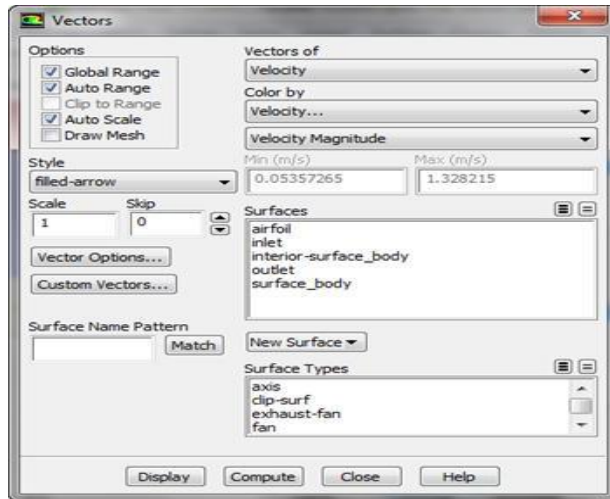
Fig. 3.6

Make sure the settings of the menu match the figure above: namely *Vectors of > Velocity*, *Color by > Velocity*, and set the second box as *Velocity Magnitude*. To see the velocity vectors, press *Display*. (Fig. 3.6)

VIVA QUESTIONS

1. What assumptions define incompressible and inviscid flow?
2. Why is a symmetric airfoil used in this experiment?
3. What is the significance of subsonic Mach numbers in this simulation?
4. Which solver settings are used for incompressible inviscid flow in ANSYS Fluent?
5. How is the airfoil geometry imported or created in the software?
6. What boundary conditions are applied in this setup?
7. How do you visualize pressure distribution over the airfoil?
8. What does the velocity vector plot indicate in flow over an airfoil?
9. Why is no lift generated at zero angle of attack for a symmetric airfoil?
10. What is the effect of increasing Mach number within subsonic range on pressure distribution?

Experiment 4

Modelling of 2D Incompressible and Inviscid flow over a cambered aerofoil and plotting of velocity vectors and pressure distribution for subsonic mach nos.

Aim: In this experiment, we will model 2D Incompressible and Inviscid flow over cambered airfoil (NACA 2412) and plotting of pressure distribution and velocity vectors for subsonic mach nos

Procedure: In this step, we will open fluent and define the boundary conditions of the problem. We will be using the model and meshed NACA 2412 airfoil for this experiment. Open Ansys Fluent and load the mesh of NACA 2412 created in experiment 2 into FLUENT. Now, double click *Setup*. The *Fluent Launcher* Window should open. Check the box marked *Double Precision*. To make the solver run a little quicker, under *Processing Options* we will select *Parallel* and change the *Number of Processes* to 2. This will allow users with a double core processor to utilize both. (Fig. 4.1)
.

Fig. 4.1

Select the pressure-Based Solver under General on the Task Page. Double-click on Models under Setup in Outline View and double click on the Viscous on the Task Page. we are solving for inviscid flows. So, select the inviscid and click on OK. Then fluids, that's a material, so selecting the material is fluid. Air is a fluid we are passing through the domain and give density is 1.225.

So now applying the boundary conditions, selecting the inlet and choosing the direction and selecting it, and giving the velocity as 150 meter per second. Then the outlet is, the outlet here. For the wall, the type is wall here and surface body, which is fluid here, and material is air here. click on OK. Now we

need to initialize the solution. Select solution initialization, click standard initialization, computing it from the inlet (Fig. 4.2). Then Initialize and calculate, run calculations. The number of iterations is 500.
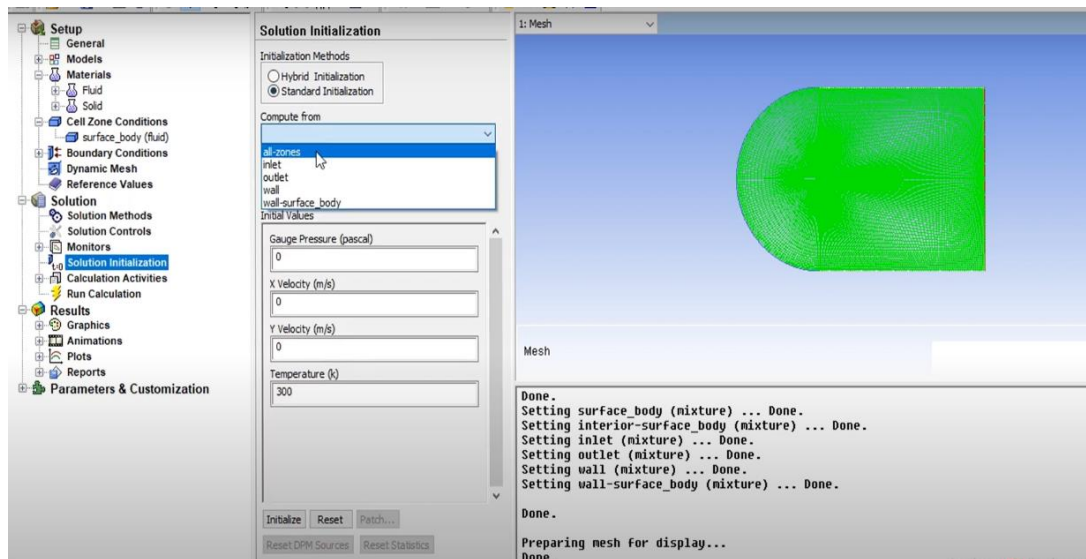

Fig. 4.2

Now go to the graphics contours. Then select as filled and check all these boxes and display. So now you can see the airflow around an airfoil here. You can zoom and zoom near to the airfoil. Now, this is how the air is distributed around an airfoil. The static pressure contours are shown here. These are the pressure contours around an airfoil (Fig. 4.3). Also, the velocity magnitude and velocity vectors can be obtained a shown in fig. 4.4 (a-b)
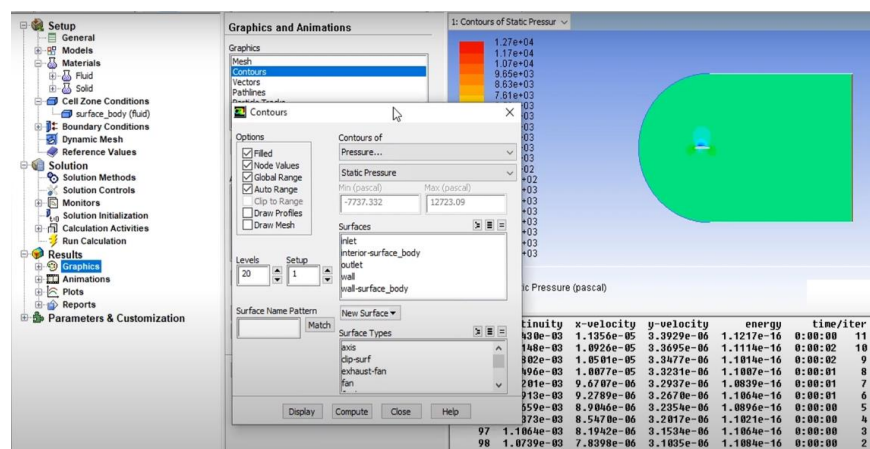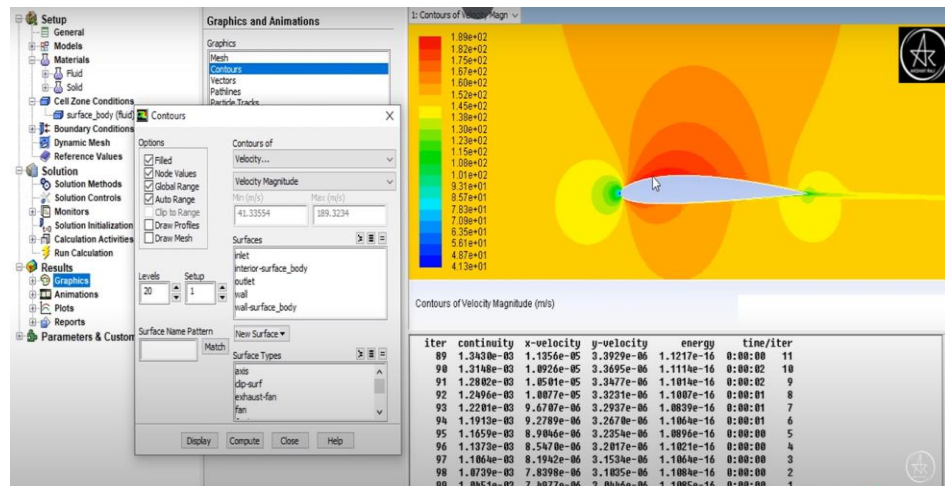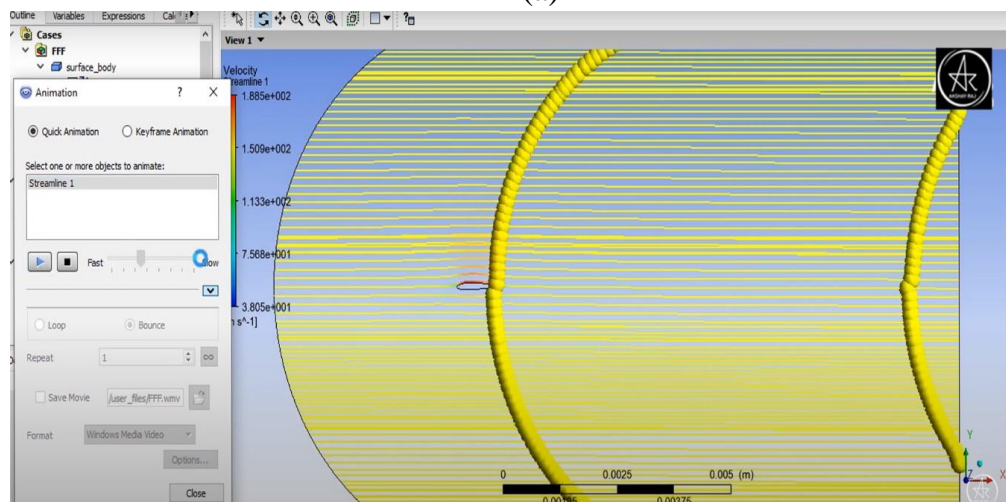

Fig. 4.3

(a)

Fig. 4.4 (b)

VIVA QUESTIONS

1. What is a cambered airfoil and how is it different from a symmetric airfoil?
2. Why is incompressible and inviscid flow assumed in this simulation?
3. What are the typical applications of cambered airfoils in subsonic flows?
4. Which boundary conditions are used in this experiment?
5. How is the angle of attack defined in the simulation?
6. What software tools are used for modeling and simulation?
7. How is the pressure distribution plotted over the airfoil surface?
8. What does the velocity vector plot reveal about flow behavior over a cambered airfoil?
9. Why does a cambered airfoil produce lift at zero angle of attack?
10. How does increasing Mach number within the subsonic range affect the pressure distribution?

Experiment 5

Modeling of 2D viscous flow over symmetrical airfoil and plotting of pressure distribution and velocity vectors for subsonic Mach nos.

Aim: In this experiment, we will model 2D vi**scous** flow over symmetric airfoil (NACA 0012) and plotting of pressure distribution and velocity vectors for subsonic mach nos

Apparatus: A computer hardware, software (ANSYS) with a graphical user interface.

Procedure: In this step, we will open fluent and define the boundary conditions of the problem. We will be use the model and meshed NACA 0012 airfoil for this experiment. Open Ansys Fluent and load the mesh of NACA 0012 created in experiment 1 into FLUENT. Now, double click *Setup*. The *Fluent Launcher* Window should open. Check the box marked *Double Precision*. To make the solver run a little quicker, under *Processing Options* we will select *Parallel* and change the *Number of Processes* to 2. This will allow users with a double core processor to utilize both. (Fig. 5.1)

Fig. 5.1

Click *OK* to launch Fluent. The first thing we will do once Fluent launches is define the solver we are going to use. Select *Problem Setup > General*. Under *Solver*, select *Pressure-Based*.

Double-click on Models under Setup in Outline View and double click on the Viscous on the Task Page. Select the k-omega (2 eqn) turbulence model. Select OK to close the Viscous Model window.

To define the density, click *Problem Setup > Materials > (double click) Air*. This will launch the *Create/Edit Materials* window. Click *Change/Create* to set the density.

Now that the fluid has been described, we are ready to set the boundary conditions of the simulation. Bring up the boundary conditions menu by selecting *Problem Setup > Boundary Conditions*. In the *Boundary Conditions* window, look under *Zones*. First, let's set the boundary conditions for the inlet. Select *Inlet* to see the details of the boundary condition. The boundary condition type should have defaulted to *velocity-inlet*: If it didn't, select it. Now, click *Edit* to bring up the *Velocity-Inlet* Window. We need to specify the magnitude and direction of the velocity. Select *Velocity Specification Method > Components*. Specify *X-Velocity* as 8 m/s, press *OK.*

In the *Boundary Conditions* window, look under *Zones*. Select *Outlet* to see the details of the boundary condition. The boundary condition type should have defaulted to *pressure-outlet*: if it didn't, select it. Click *Edit*, and ensure that the *Gauge Pressure* is defaulted to 0. If it is, you may close this window. In the *Boundary Conditions* window, look under *Zones* and select *airfoil*. Select *Type > Wall* if it hasn't been defaulted.

The final thing to do before we move on to solution is to acknowledge the reference values. Go to *Problem Setup > Reference Values*. In the *Reference Values* Window, select *Compute From > Inlet*. Check the reference values that appear to make sure they are as we have already set them.

First, go to *Solution > Solution Methods*. Everything in this section should have defaulted to what we want, but let's make sure that under *Flow* the selection is *Second Order Upwind*. If this is the selection, we may move on. Now we are ready to begin solving the simulation. Before we hit solve though, we need to set up some parameters for how Fluent will solve the simulation.

Let's begin by going to *Solution > Monitors*. In the *Monitors* Window, look under *Residuals, Statistic, and Force Monitors*. Select *Residuals - Print, Plot* and press *Edit*. In the *Residual Monitors* Window, we want to change all the *Absolute Criteria* to 1e-6. This will give us some further trust in our solution. (Fig. 5.2)

Now, we need to initialize the solution. Go to *Solution > Solution Initialization*. In the *Solution Initialization* Window, select *Compute From > Inlet*. Ensure the values that appear are the same values we inputted. If they are, initialize the solution by clicking *Initialize*.

Once the solution has been initialized, we are ready to solve the simulation. Go to *Solution > Run Calculation*. Change *Number of Iterations* to 1000, then double click *Calculate*. Sit back and twiddle your thumbs until Fluent spits out a converged solution.

To view the pressure contours over the entire mesh, go to *Results > Graphics and Animations* again, and in the *Graphics and Animations* Window, select *Contours*. Click *Set Up...* to bring up

the *Contours* Menu. Check the box next to **Filled**. Under *Contours Of*, ensure that the two boxes that are selected are ***Pressure...*** and ***Static Pressure***. Once these parameters are set, press ***Display*** to see the pressure contours. (Fig. 5.2)
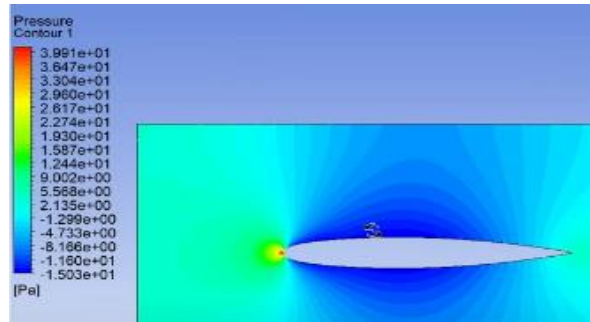

Fig. 5.2

Next, we will look at the velocity vectors of the solution to see if they make intuitive sense. To plot the velocity vectors, go to ***Results > Graphics and Animations***. In the *Graphics and Animations* Window, select ***Vectors*** and click ***Set Up.*** This will bring up the *Vectors* Menu.
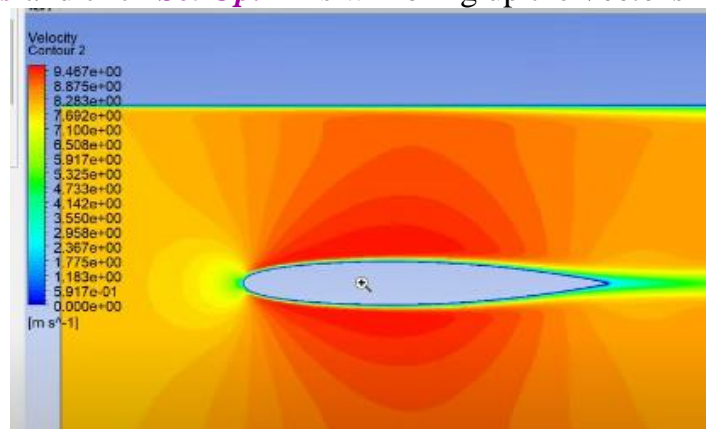

Fig. 5.3

Make sure the settings of the menu match the figure above: namely ***Vectors of > Velocity***, ***Color by > Velocity***, and set the second box as ***Velocity Magnitude***. To see the velocity vectors, press ***Display***. (Fig. 5.3)

VIVA QUESTIONS

1. What is the difference between viscous and inviscid flow?
2. Why do we study flow over a symmetrical airfoil?
3. What does subsonic Mach number mean?
4. Which turbulence model is used for viscous flow in this simulation?
5. What boundary conditions are applied in this experiment?
6. How is the no-slip condition applied at the airfoil surface?
7. How do you plot pressure distribution in the post-processing step?
8. What does the velocity vector plot show around the airfoil?
9. How does viscosity affect the flow over the airfoil compared to inviscid flow?
10. What is the role of Reynolds number in viscous flow simulations?

Experiment- 6

Modelling of 2D viscous flow over cambered airfoil and plotting of pressure distribution and velocity vectors for subsonic mach nos

Aim: In this experiment, we will model 2D viscous flow over cambered airfoil (NACA 2412) and plotting of pressure distribution and velocity vectors for subsonic mach nos

Apparatus: A computer hardware, software (ANSYS) with a graphical user interface.

Procedure: In this step, we will open fluent and define the boundary conditions of the problem. We will be use the model and meshed NACA 2412 airfoil for this experiment. Open Ansys Fluent and load the mesh of NACA 2412 created in experiment 2 into FLUENT. Now, double click *Setup*. The *Fluent Launcher* Window should open. Check the box marked *Double Precision*. To make the solver run a little quicker, under *Processing Options* we will select *Parallel* and change the *Number of Processes* to 2. This will allow users with a double core processor to utilize both. (Fig. 6.1)
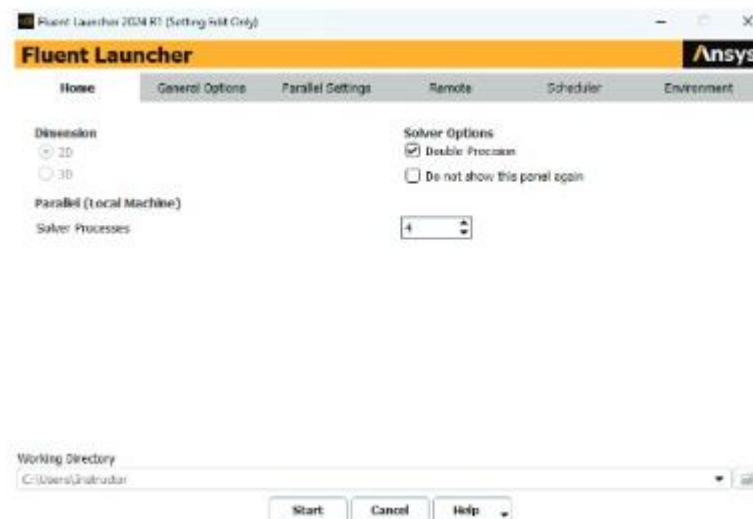
.

Fig. 6.1

Select the pressure-Based Solver under General on the Task Page. Double-click on Models under Setup in Outline View and double click on the Viscous on the Task Page. Select the Spalart-Allmaras (1 eqn) turbulence model. Select OK to close the Viscous Model window. (Fig. 6.2)

Double click on Boundary Conditions under Setup in Outline View. Double click the inlet Zone on the Task Page. Select Components as Velocity Specification Method. Enter 12.7 as X-Velocity [m/s]. Click on Apply and Close the Velocity Inlet window. (Fig. 6.3).
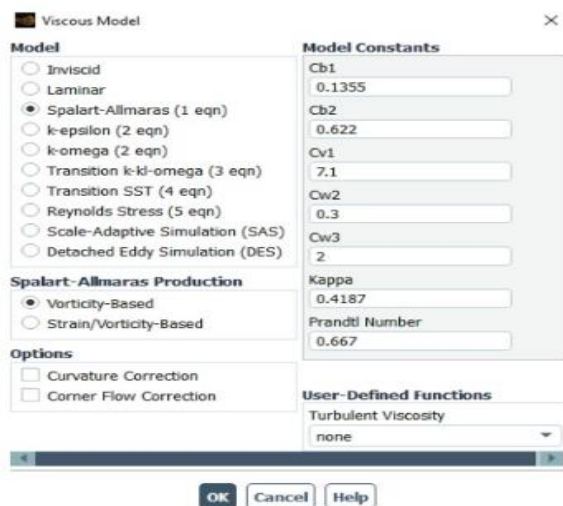
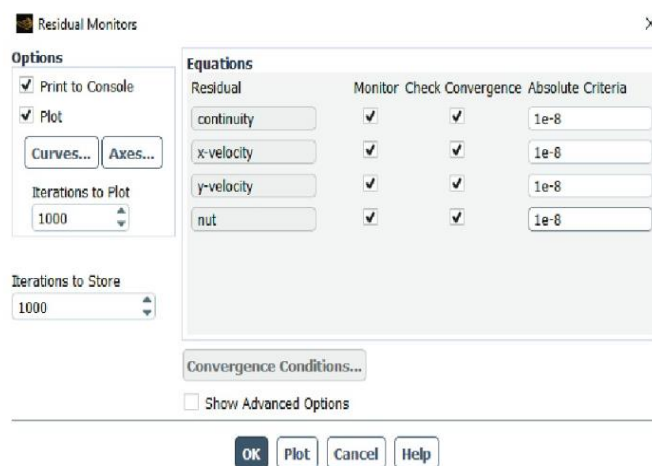Fig. 6.2   Fig. 6.3

Double click on Monitors under Solution in the Tree. Double click on Residual under Monitor. Change the Convergence Criterion for all four residuals to 1e-8. Click on OK to exit the Residual Monitors window.

Double click on Reference Values under Setup in the Outline View. Select Compute from inlet. Enter the value 0.23 for Length [m] which is the chord length and enter the value 12.7 for Velocity [m/s]. Enter the value 0.3048 for Depth [m] and 0.070104 for Area [m2]. The Area is equal to the Depth times the Length. (Fig. 6.4)
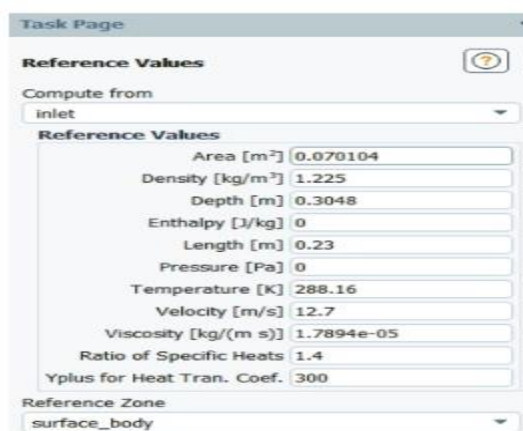

Fig. 6.4

Double click on Initialization under Solution under the Outline View. Select Standard Initialization as Initialization Method, select Compute from inlet and click on Initialize. Select File>>Save Project from the menu. Select File>>Export>>Case & Data... from the menu. Save the Case/Data File with the name Flow Past an Airfoil.cas.h5. Double click on Run Calculation under Solution in the Outline View. Set number of Iterations to 4000. Click on Calculate. Click OK when the calculations are complete. (Fig. 6.5)
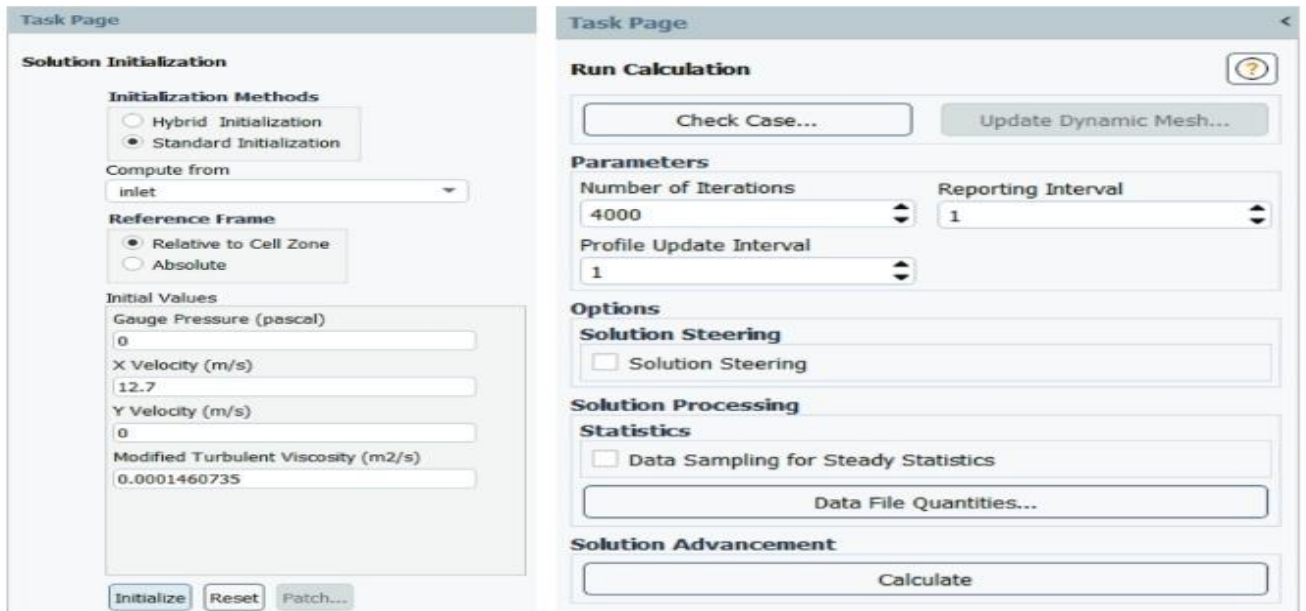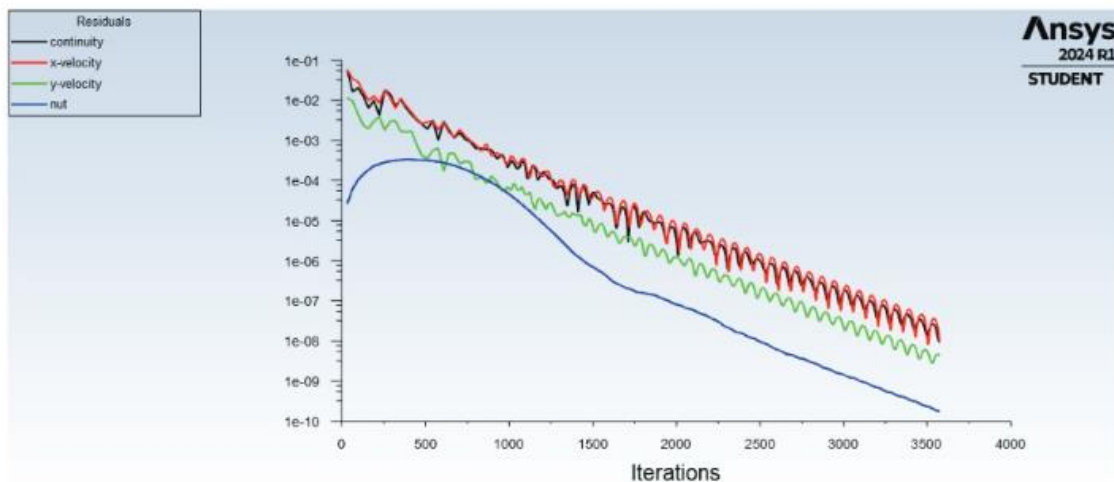

Fig. 6.5

Fig. 6.6

Click on Copy Screenshot of Active Window to Clipboard, see Figure 6.6). The Scaled Residuals can be pasted into a Word document.

Double click on Graphics under Results in the Outline View. Display a plot of pressure contours by double clicking on Contours under Graphics, check the box for Filled under Options, select all Surfaces and select Pressure and Static Pressure under Contours of Click on Colormap Options and set Colormap Alignment to Top. Set the Font Behavior to Fixed and Font Size under Font to 16. Set Type to general and Precision to 2 under Number Format. Select Apply followed by Close for the Colormap window. Click on Save/Display in the Contours window. (Fig. 6.7)
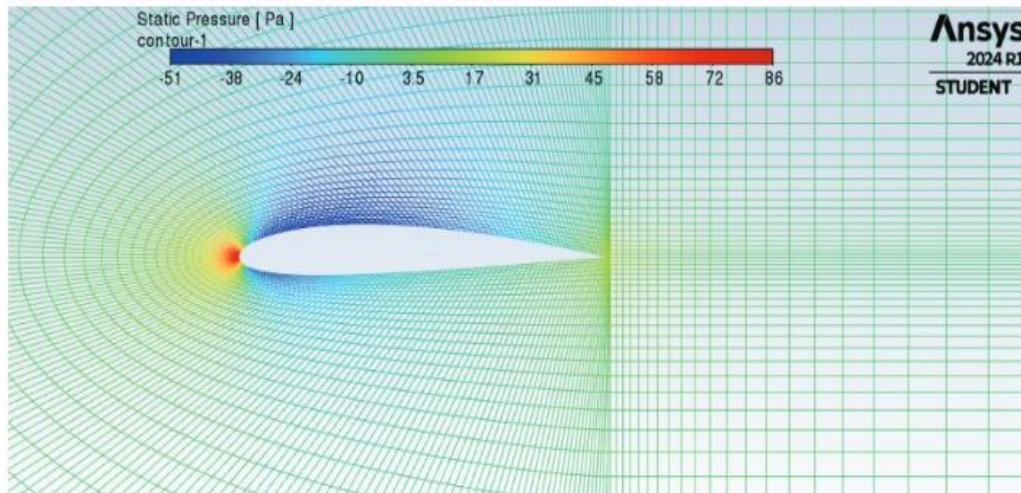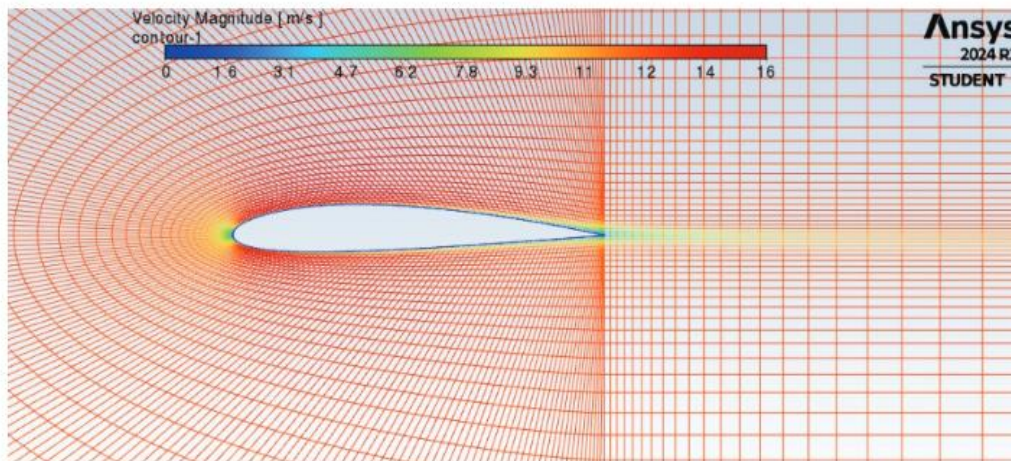

Fig. 6.7

Fig. 6.8

Create another contour plot for Velocity and Velocity Magnitude. (Fig. 6.8)

VIVA QUESTIONS

1. What is a cambered airfoil?
2. How does a cambered airfoil generate lift at zero angle of attack?
3. What is the significance of modeling viscous flow in this experiment?
4. Why is subsonic Mach number considered in this analysis?
5. What boundary conditions are used in this simulation?
6. What turbulence model did you use for viscous flow and why?
7. How does viscosity affect the flow near the airfoil surface?
8. What does the pressure distribution reveal about lift generation?
9. What do the velocity vectors indicate in a viscous flow over a cambered airfoil?
10. How is the Reynolds number related to viscous flow behavior?

Experiment- 7

Modelling of 2D compressible and Inviscid flow over a symmetrical aerofoil and plotting of velocity vectors and pressure distribution for supersonic mach nos.

Aim: To Model 2D compressible and Inviscid flow over a symmetrical aerofoil and plotting of velocity vectors and pressure distribution for supersonic mach. nos.

Apparatus: A computer hardware, software (ANSYS) with a graphical user interface.

Procedure: start the Ansys Fluid Flow Fluent. Right-click on geometry, go to properties, and select analysis type as 2D. right-click on geometry and select new design modular geometry. Go to units tab and select units as meter.
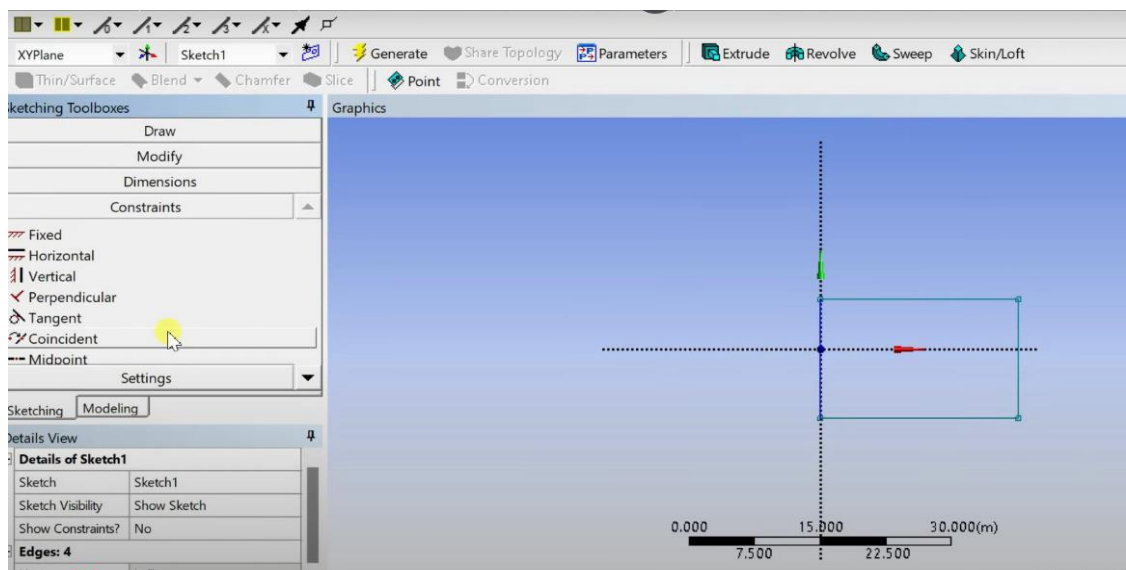

Fig. 7.1

Select the XY plane. Click on the Z-axis. Go to sketch. Write it, draw a rectangle. (Fig. 7.1) `And go to constraints and select symmetry. Select the axis first, then select the two lines of the rectangle. Put the dimensions of H1 2m and V1 as 0.3m We're going to create the wedge shape first.

Use the move tool to move the dimension nearest to the sketch so that it would be, better to visible. And click on generate. Go to draw panel, draw a polyline, starting from the middle of this rectangle, then the sidewise. Now go to constraints. Use the concise coincidence option.

To merge the two lines into single point. Then select the midpoint constraints. Select the rectangle-line of the rectangle first, then the, then the vertex point of this rectangle, so that it will be a coincidence. Now click on generate. Now go to modify tool and trim out the unnecessary lines. Here is a point remaining, so we are going to remove this using the point tool. Go to select option

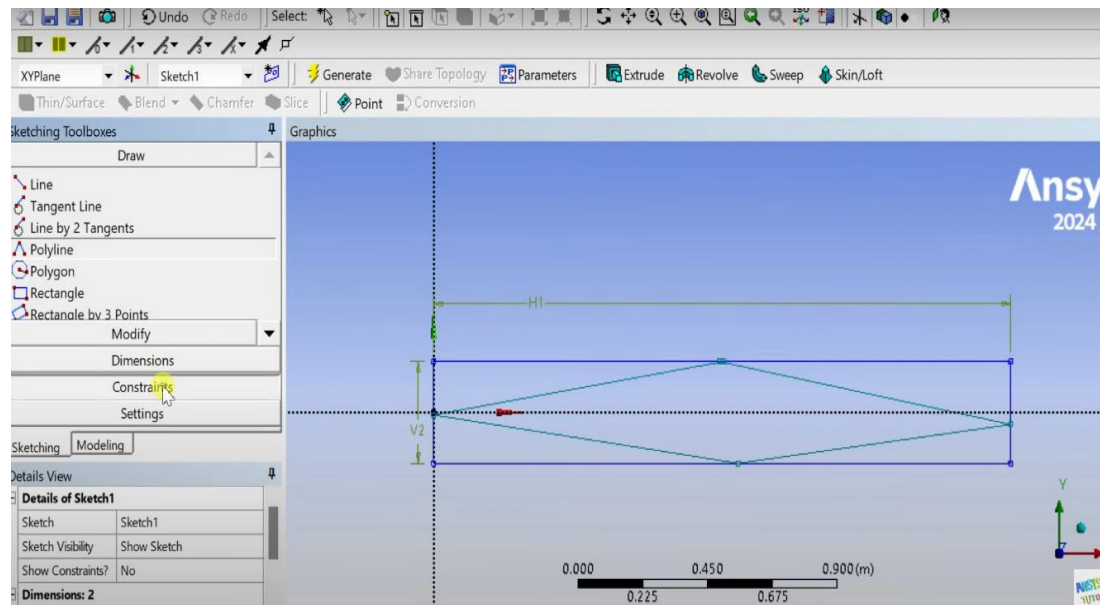and select it as point selection tool. Select this point and click on delete. It will be deleted
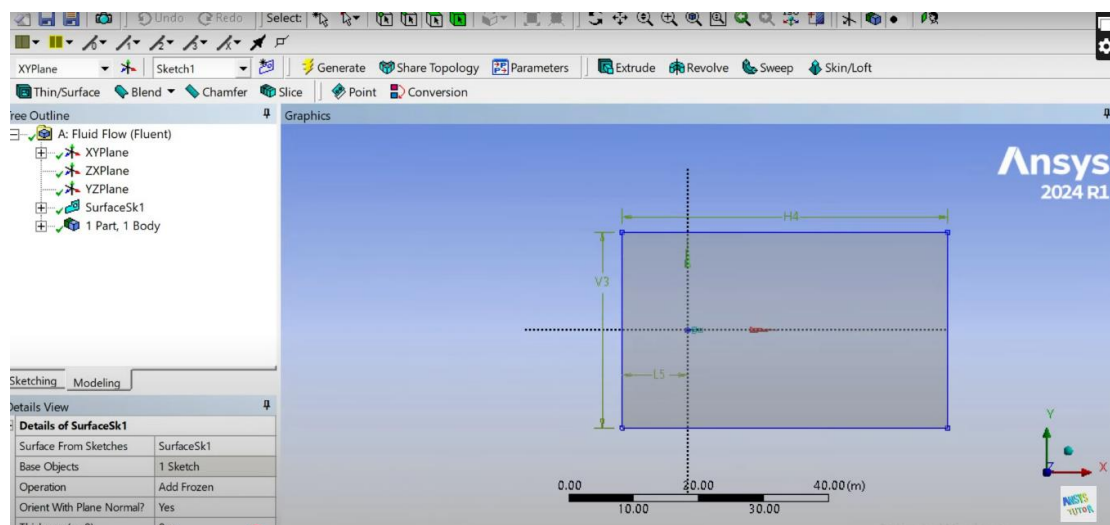

Fig. 7.2

.

Fig. 7.3

 Now we are going to draw a outer fluid domain around this aerofoil. Draw a larger rectangle. Put the symmetry constraints. Now put the dimensions as horizontal side H4 as 50m, distance between leading edge of outer rectangle and leading edge of inner wedge as 10m and vertical side V3 as 30m. And click on generate. Go to concept menu and select surface from sketches. Click on this

sketch and then click on apply. (Fig. 7.3). Set the operation as add frozen and then click on generate. This is our fluid domain which is around this wedge-shaped aerofoil. And geometry is complete.
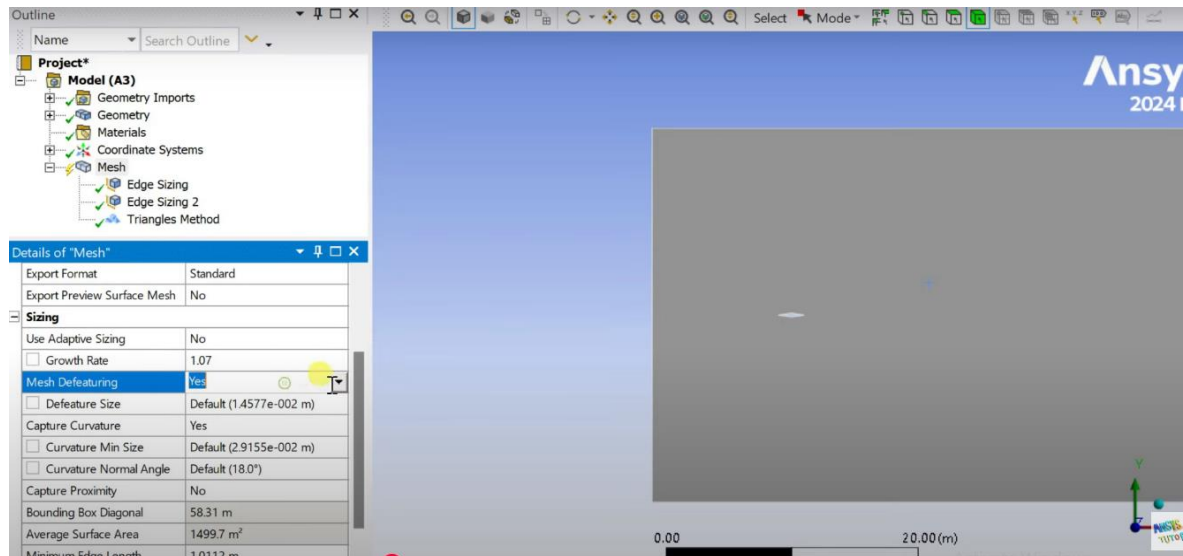

Fig. 7.4

We can now proceed for meshing. Right-click on mesh, select edit. The meshing workbench window will open. And right-click on the mesh and select sizing option. Select the edges of this fluid domain, outer fluid domain. Hold the control key to select all these four sides, then click on apply. Enter the element size as one meter. And insert another sizing for the sides of this aerofoil. Select all the four sides, then click on apply. Enter the element size as 0.01 meter. Then insert the meshing method.

Select the fluid domain and select triangular methods. Go to default mesh setting. Go to sizing. We are going to change the growth rate to 1.07. (Fig. 7.4). Then the depicturing size. Now, next depicturing, we are disabling the mesh depicturing. Capture curvature option as no. Capture proximity as yes. Then now you are going to generate this mesh. The mesh is currently generated. You can see there is smaller cells around this wedge, aerofoil wedge, and gradually larger cells towards outer surfaces (Fig. 7.5).
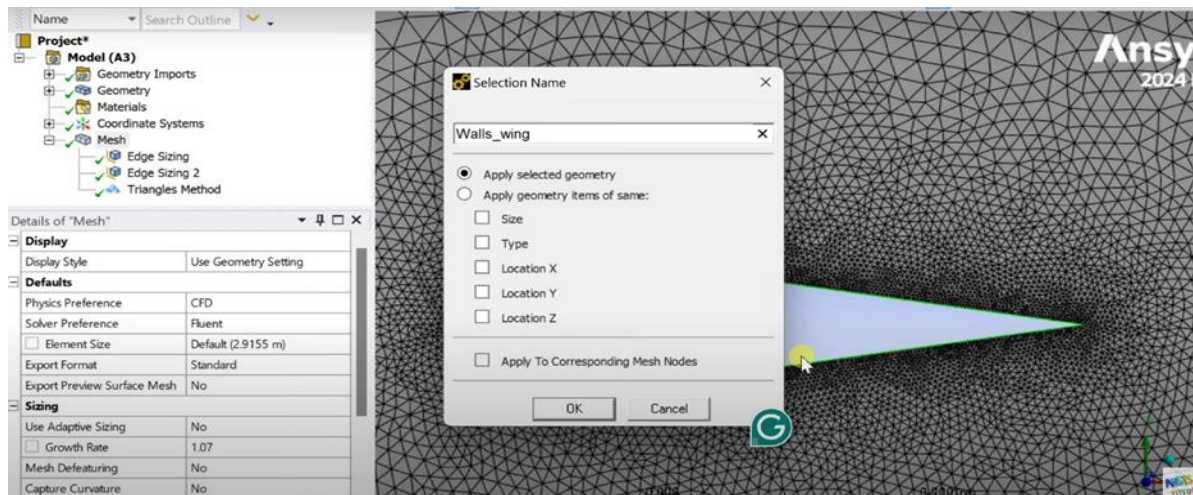
Fig. 7.5

Now we are going to name these boundary surfaces. Select all the four sides and name it as wall swing. Click on okay. Select this outer walls and select- name it as pressure far field. Now we are going to update this mesh. You can see that mesh is now updated. This mesh is linked with the Fluent solver. Close this meshing and we are going to open this Fluent solver.

Select this double precision option. Then click on start., this is the Fluent's, setup window. we have selected the dark theme, so it is black in color. So, start with analysis. We are going to turn on the energy equation. And select the fluid flow model as inviscid. Go to general tab and select it as a density-based solver type.
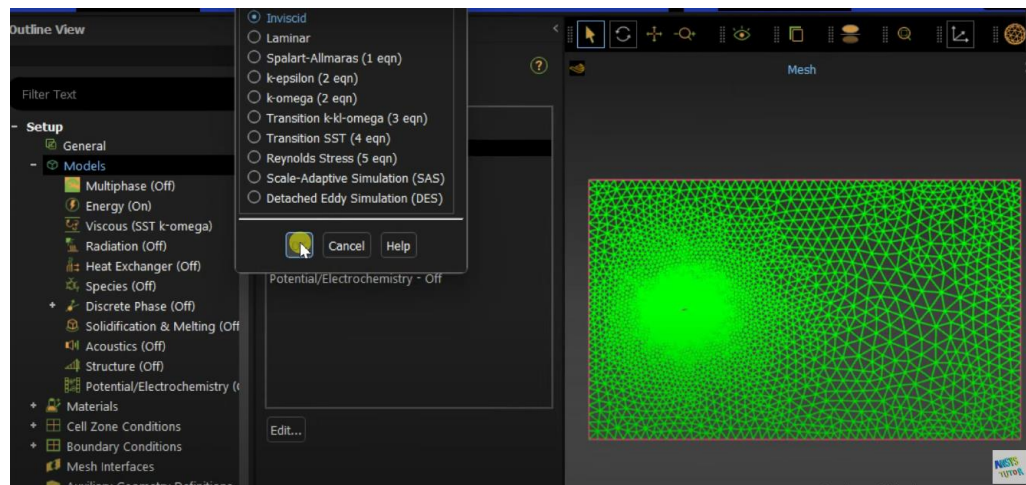

Fig. 7.6

Then go to materials. Go to fluid. Select air. Change the density of the air. Change the density to ideal gas. Keep the default value of specific heat and molecular weight over here and click on change create.
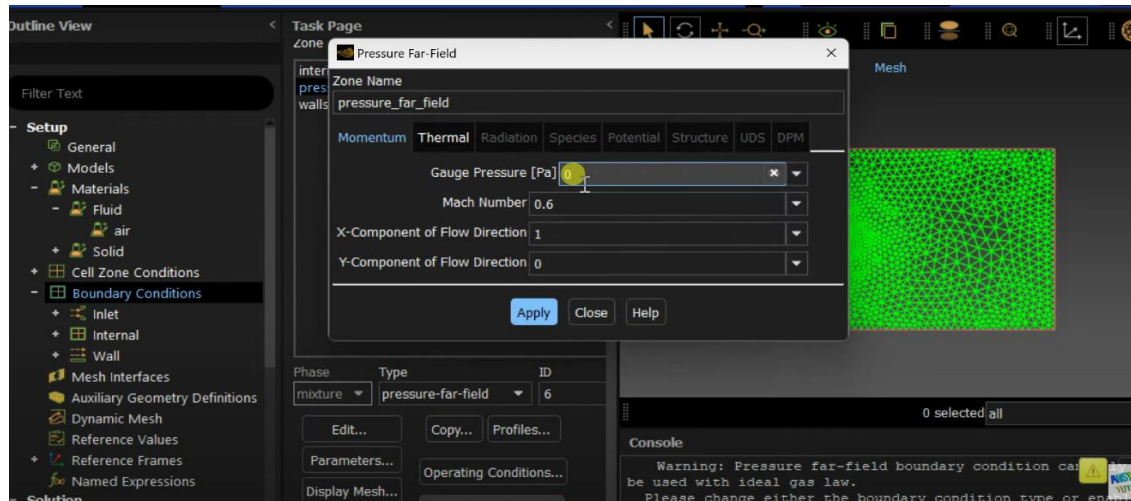
Fig. 7.7

Now go to boundary conditions. Select the pressure far field. You can see that according to the names, now Fluent already assigned the boundary surface type as pressure far field. Click on edit. Gauge pressure as 101325 pascal. Make number as 3. Keep the default temperature up there, then click on apply. Click on close. Now go to operating conditions and make the operating pressure as zero. Then click on okay. And the reference value, you can see... The reference value, we can see the pressure as zero. So while we are selecting compute from inlet or compute from far field, it would show the ascent pressure at that point, at that location. we have selected pressure far field, now you can see pressure is 101325.

 Now go to solution methods. Select formulation as implicit. Select gradient as Green-Gauss cell-based gradient. Keep the flow as second-order upwind. Now go to controls. Go to monitors. Select the residuals. Decrease the continuity residuals up to 10 to the power minus 16 so that the solution would be more accurate.

Now go to initialization tab. Select the standard initialization. Select compute from pressure far field. Then click on initialize. Now we are going to save this case file. Select the location where you want to save this case file. Name the case file and click on OK. The case file will be saved in the desired location. So if you are losing something or you are, we want to change something, you can assess that case file later.
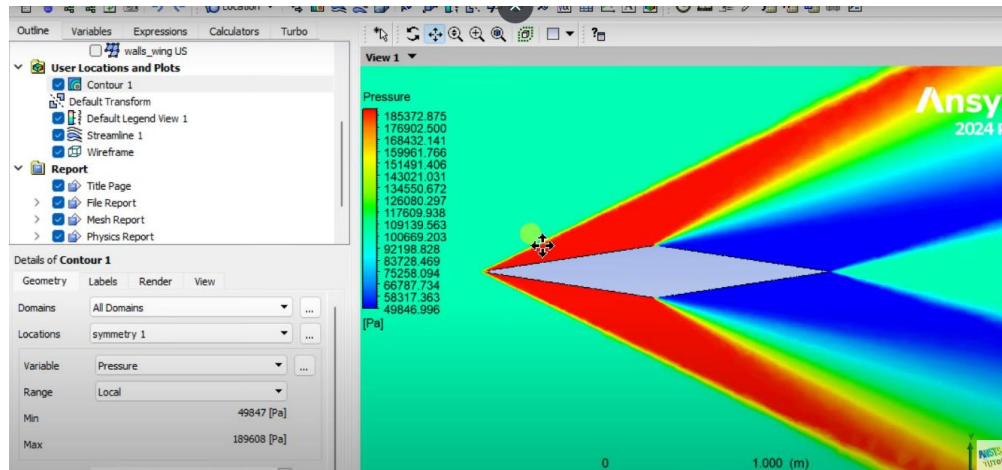
Fig. 7.8

We do not require to be set up all these things again. You can just import that case file so that it will be loaded from that position. Now, run this calculation. Calculation is complete. Now we are going to analyze these results. Go to contours. Select the contour of velocity and pressure. Click on save or display.

VIVA QUESTIONS

1. What is compressible flow and when is it considered in CFD?
2. What defines supersonic flow in terms of Mach number?
3. Why is inviscid flow assumed in this supersonic flow simulation?
4. What is the role of shock waves in supersonic flow over an airfoil?
5. Why is a symmetrical airfoil chosen for this experiment?
6. What solver settings are required for compressible inviscid flow?
7. How are pressure distribution plots useful in supersonic flow analysis?
8. What information do velocity vectors provide in supersonic flow?
9. What boundary conditions are used for this simulation setup?
10. How does flow behavior differ between subsonic and supersonic regimes?

Experiment 8

Modelling of 2D compressible and Inviscid flow over a cambered aerofoil and plotting of velocity vectors and pressure distribution for supersonic mach nos.

Aim: Modelling of 2D compressible and Inviscid flow over a cambered aerofoil and plotting of velocity vectors and pressure distribution for supersonic mach nos.

Apparatus: A computer hardware, software (ANSYS) with a graphical user interface.

Procedure:

In this step, we will open fluent and define the boundary conditions of the problem. We will be use the model and meshed NACA 2412 airfoil for this experiment. Open Ansys Fluent and load the mesh of NACA 2412 created in experiment 2 into FLUENT. Now, double click *Setup*. The *Fluent Launcher* Window should open. Check the box marked *Double Precision*. To make the solver run a little quicker, under *Processing Options* we will select *Parallel* and change the *Number of Processes* to 4. This will allow users with a double core processor to utilize both. (Fig. 8.1)
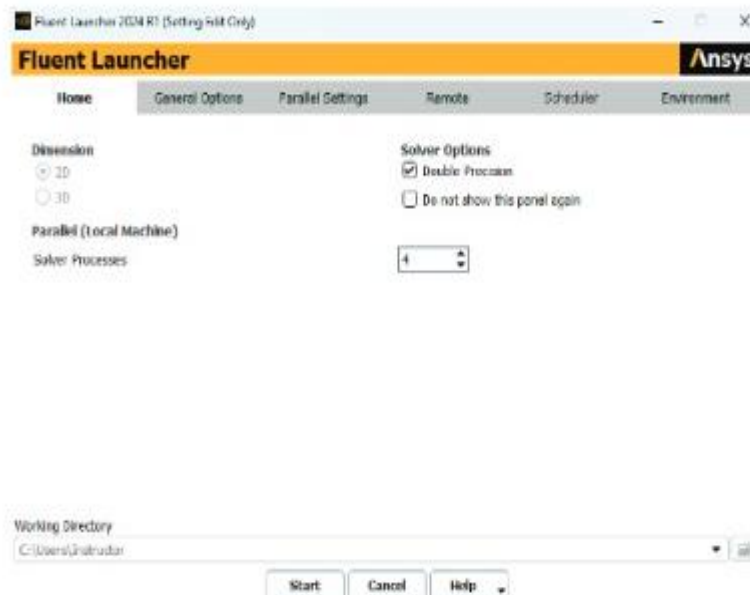

Fig. 8.1

Select the density-Based Solver under General on the Task Page. Double-click on Models under Setup in Outline View and double click on the Viscous on the Task Page. we are solving for inviscid flows. So, select the inviscid and click on OK.
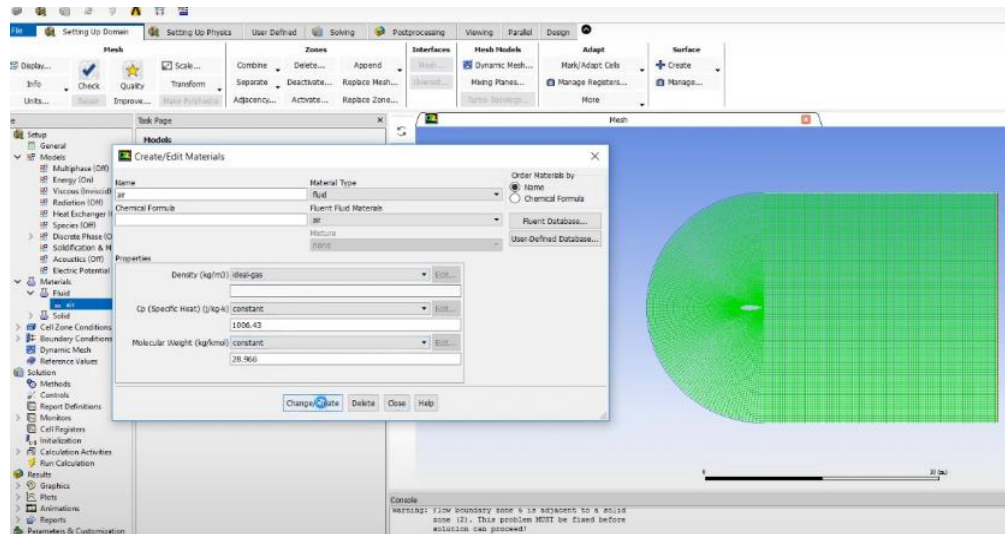
Fig. 8.2

Then fluids, that's a material, so selecting the material is fluid. Air is a fluid we are passing through the domain. Then go to materials. Go to fluid. Select air. Change the density of the air. Change the density to ideal gas. Keep the default value of specific heat and molecular weight over here and click on change create. (Fig. 8.2)

So now applying the boundary conditions, selecting the inlet and right click and change it to pressure far field type. Now make gauge pressure to 1e5, Mach number 1.5, Now in outlet boundary condition, change the gauge pressure to 80000 pascals at the outlet. Set the airfoil wall surface to wall boundary condition. Then click on OK. Now we need to initialize the solution. Select solution initialization and choose standard initialization, computing it from the inlet. Initialize. So, calculate, run calculations. Now set the number of iterations is 500.
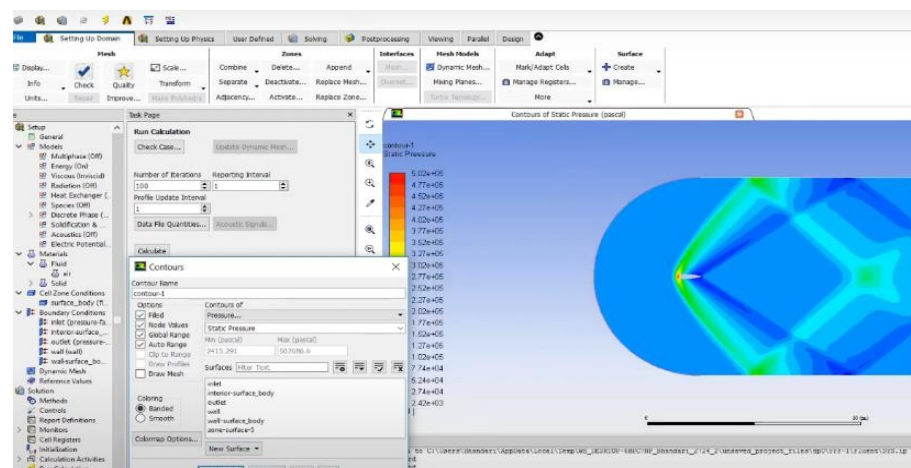

Fig. 8.3

Now go to the graphics contours. So select as filled and check all these boxes and display. So now you can see the airflow around an airfoil here. You can zoom and zoom near to the airfoil. So this is how the air is distributed around an airfoil.
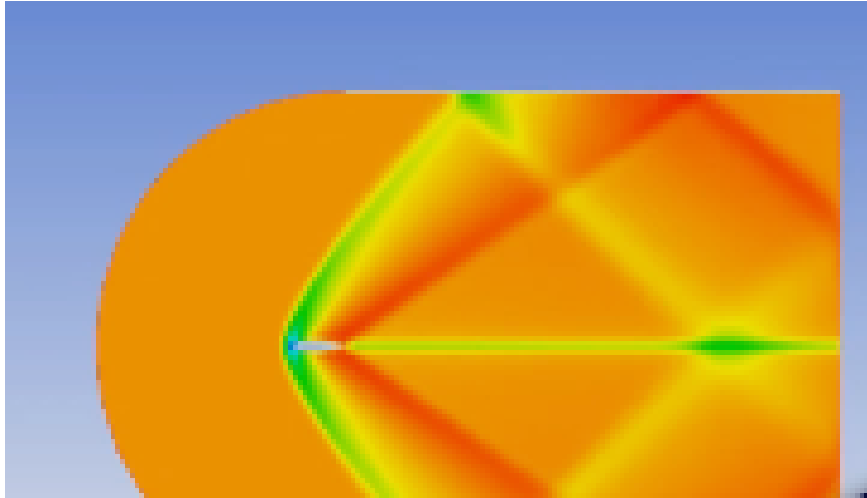

Fig. 8.4

The static pressure contours is shown here. These are the pressure contours around an airfoil (Fig. 8.3). Also the velocity magnitude and velocity vectors can also be obtained as shown in figure 8.4

VIVA QUESTIONS

1. What is a cambered aerofoil and how does it generate lift?
2. What is meant by compressible flow and why is it important at supersonic speeds?
3. Why is the flow assumed inviscid in this supersonic simulation?
4. What is the significance of Mach number in supersonic flow analysis?
5. What types of shock waves can form over a cambered aerofoil in supersonic flow?
6. What solver settings are used for compressible inviscid flow in ANSYS Fluent?
7. How are velocity vectors helpful in understanding supersonic flow behavior?
8. How is pressure distribution affected by camber in supersonic flow?
9. What boundary conditions are applied for supersonic flow simulations?
10. How does camber influence shock formation and lift in supersonic flow?

Experiment 9
Isentropic Flow Analysis in a 2-D Subsonic Diffuser and a Subsonic Nozzle

Aim: In this experiment, we will perform the isentropic flow analysis in a 2-D subsonic diffuser and a subsonic nozzle using ANSYS Workbench

Apparatus: A computer hardware, software (ANSYS) with a graphical user interface.

Procedure:

The main steps to be involved are:

Open Ansys Work Bench – Analysis System – Fluid Flow (Fluent). Then Geometry - New Geometry - Units-m- Select XY Plane

Sketching- Use line and draw the shape of C D Nozzle, with the extent of diffuser entry (V1) at 25m and exit height (V2) at 45m and the horizontal length (h3) is 50m (Fig. 9.1). Dimensions- Length or Distance –Give Dimensions as Specified
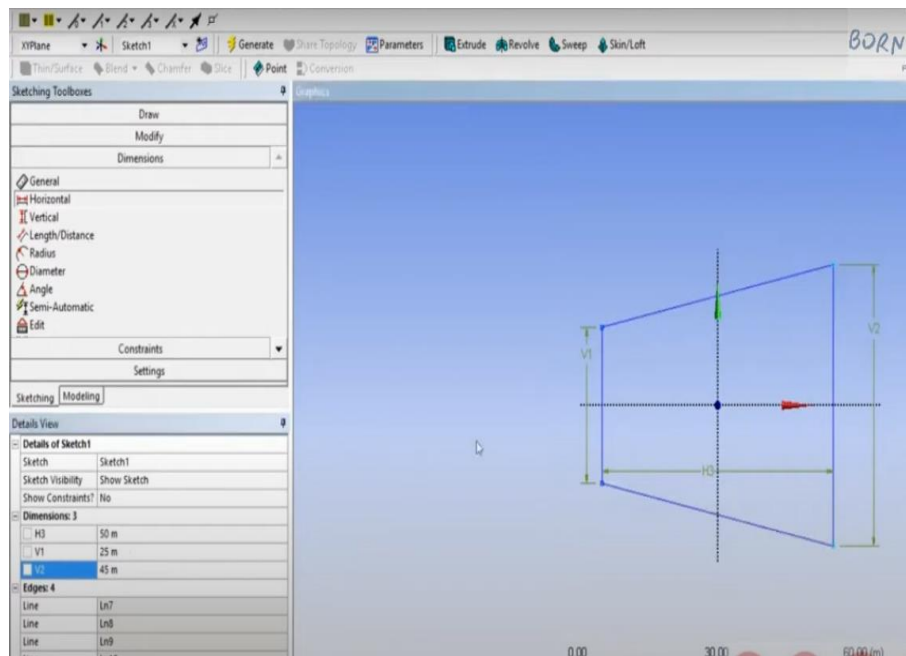

Fig. 9.1

Then go to concept menu and select surface from line to convert the lines of subsonic diffuser into a Surface. Mesh – Insert Method – Select Body –All Quads – Ok. Select - Mesh -Face meshing. Insert – Sizing –Vertical Edges –No: Of Divisions -50 for both vertical edges and no. of divisions of 100 for both horizontal edges. Insert –Mapped Face Meshing – Select All –Update. (Fig. 9.2). Select Edge – Create Named Selection - Inlet- Outlet Walls.
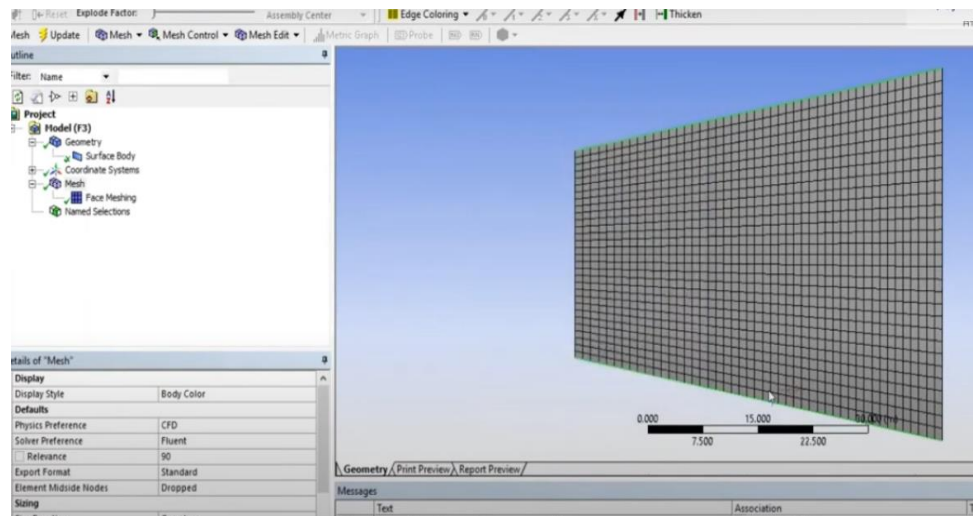
Fig. 9.2

Select Solution – General- density Based Models- Energy Equations –On. Then Materials- Air – Ideal Gas – Viscosity Sutherland. Then Boundary Conditions Inlet – Pressure Inlet 1e5 pascal and Outlet – Pressure Outlet -gauge pressure 1.5e5 (Fig. 9.3)
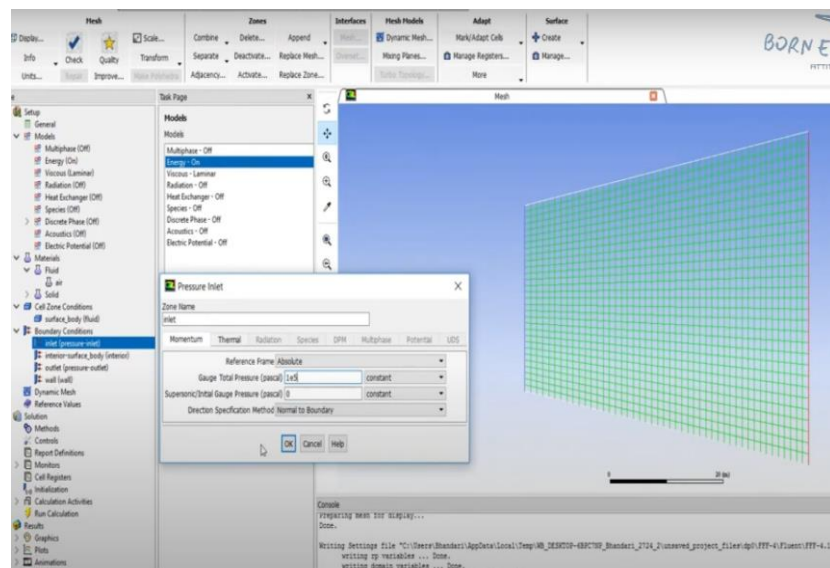

Fig. 9.3

Monitors Create –Drag - Print, Plot – Select Walls. Solutions Initialization – Calculate From – outlet. Run Calculation – No: Of Iterations -500 –Ok. Graphics –Contour Pressure –Static pressure and click on compute and then save/display. (Fig. 9.4)
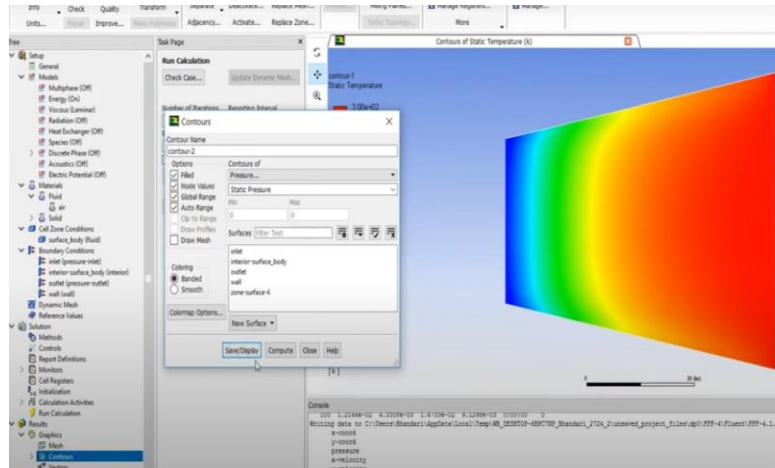
Fig. 9.4

For modelling of Isentropic Flow Analysis in a 2-D Subsonic Nozzle:

Open Ansys Work Bench – Analysis System – Fluid Flow (Fluent). Then Geometry - New Geometry - Units-m- Select XY Plane
Sketching- Use line and draw the shape of C D Nozzle, with the height of nozzle entry (V1) at 45m and exit height (V2) at 25m and the horizontal length (h3) is 50m (Fig. 9.5). Dimensions-Length or Distance –Give Dimensions as Specified
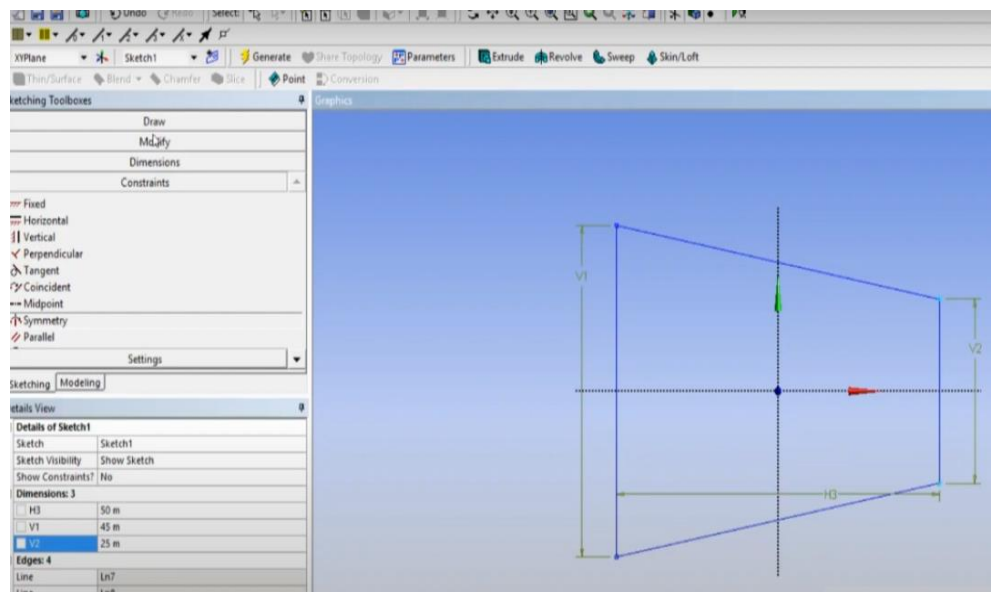

Fig. 9.5

Then go to concept menu and select surface from line to convert the lines of subsonic diffuser into

a Surface. Mesh – Insert Method – Select Body –All Quads – Ok. Select - Mesh -Face meshing. Insert – Sizing –Vertical Edges –No: Of Divisions -50 for both vertical edges and no. of divisions of 100 for both horizontal edges. Insert –Mapped Face Meshing – Select All –Update. (Fig. 9.6). Select Edge – Create Named Selection - Inlet- Outlet Walls.
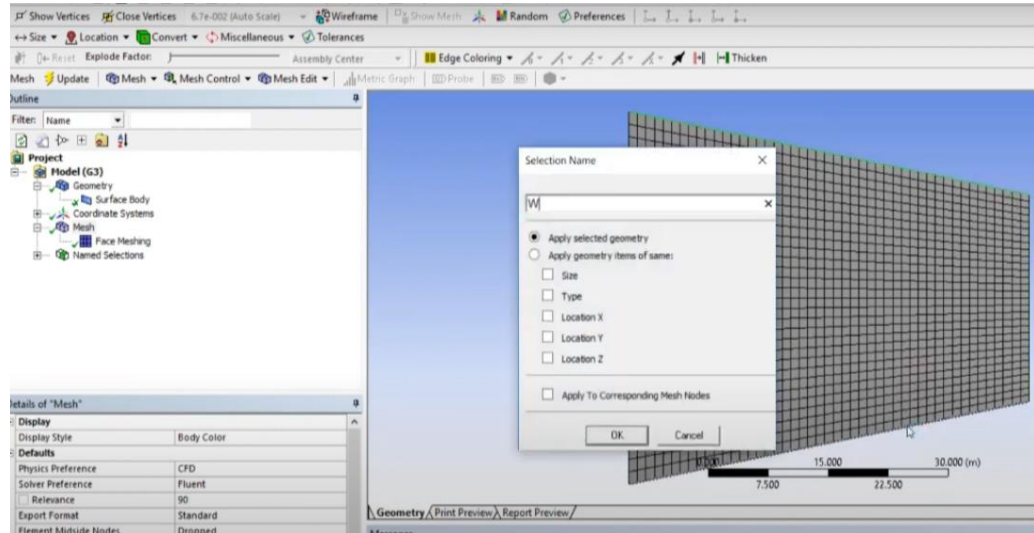

Fig. 9.6

Select Solution – General-Pressure Based Models- Energy Equations –On. Then Materials- Air – density as ideal gas, Cp as constant. Then Boundary Conditions Inlet – velocity inlet and velocity specification method as magnitude and direction and velocity magnitude as 50 m/s and Outlet – Pressure Outlet -gauge pressure as 0.

Monitors Create –Drag - Print, Plot – Select Walls. Solutions Initialization – Calculate From – Inlet, Run Calculation – No: Of Iterations -500 –Ok. Graphics –Contour Pressure –Dynamic pressure and click on compute and then save/display. (Fig. 9.7) and the total pressure is also be displayed as shown in Fig. 9.8.
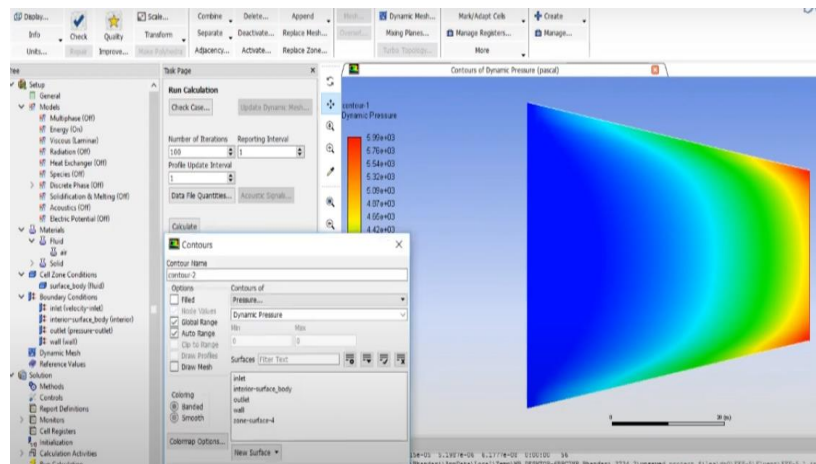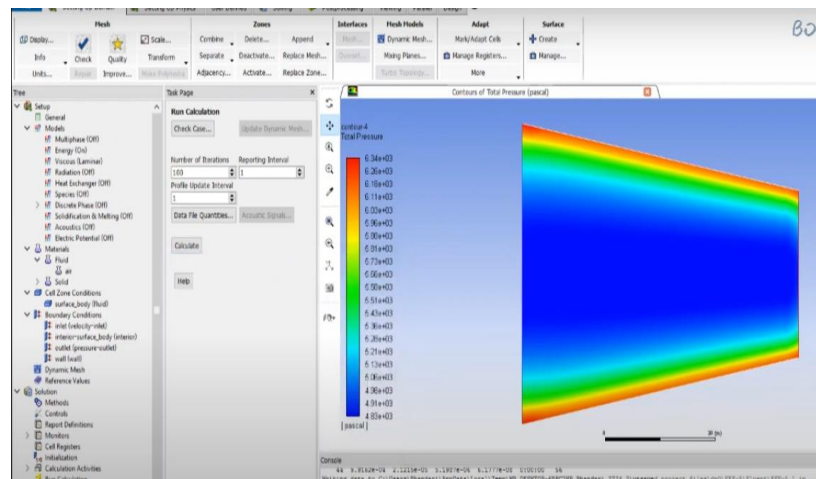
Fig. 9.7

Fig. 9.8

VIVA QUESTIONS

1. What is isentropic flow?
2. What are the assumptions made in isentropic flow analysis?
3. How does a subsonic diffuser affect velocity and pressure?
4. What happens to flow properties in a subsonic nozzle?
5. What is the function of a diffuser in a subsonic flow system?
6. How does the area change affect flow in a nozzle and diffuser?
7. Why is Mach number important in nozzle and diffuser analysis?
8. What boundary conditions are applied for isentropic subsonic flow?
9. What software module did you use for modeling the nozzle and diffuser?
10. How do you interpret pressure and velocity contour plots in this experiment?

Experiment 10

Isentropic Flow Analysis in a 2-D Supersonic Diffuser and a Supersonic Nozzle

Aim: Isentropic Flow Analysis in a 2-D Supersonic Diffuser and a Supersonic Nozzle using ANSYS Workbench

Apparatus: A computer hardware, software (ANSYS) with a graphical user interface.

Procedure:

The main steps to be involved are

Open Ansys Work Bench – Analysis System – Fluid Flow (Fluent). Then Geometry - New Geometry - Units-m- Select XY Plane

Sketching- Use line and draw the shape of C D Nozzle, with the height of nozzle entry (V1) at 45m and exit height (V2) at 25m and the horizontal length (h3) is 50m (Fig. 10.1). Dimensions- Length or Distance –Give Dimensions as Specified.
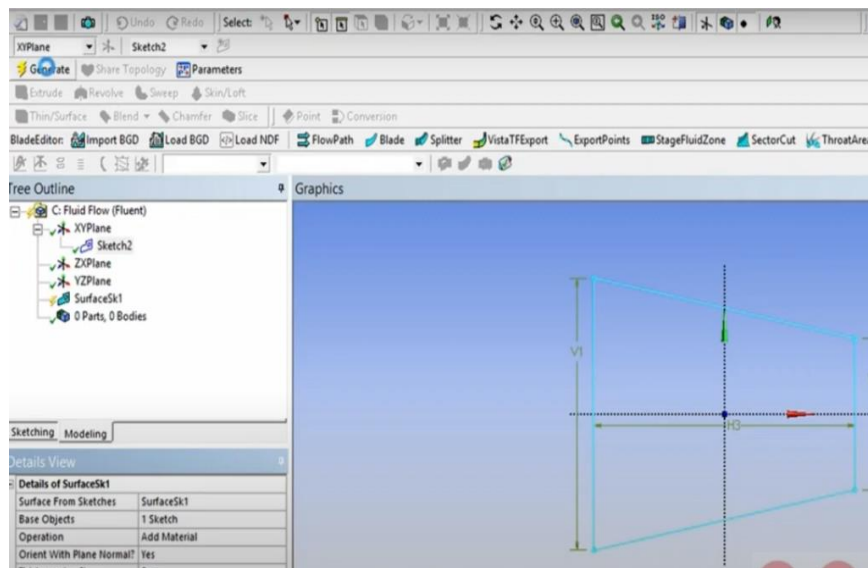

Fig. 10.1

Then go to concept menu and select surface from line to convert the lines of subsonic diffuser into a Surface. Mesh – Insert Method – Select Body –All Quads – Ok. Select - Mesh -Face meshing. Insert – Sizing –Vertical Edges –No: Of Divisions -60 for both vertical edges and no. of divisions of 150 for both horizontal edges. Insert –Mapped Face Meshing – Select All –Update. (Fig. 10.2). Select Edge – Create Named Selection - Inlet- Outlet Walls.
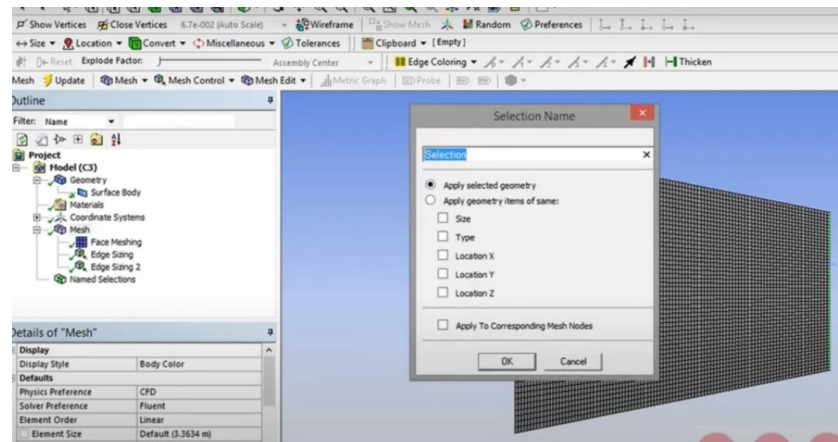
Fig. 10.2

Select Solution – General-Density Based Models- Energy Equations –On. Then Materials- Air – Ideal Gas – Viscosity Sutherland. Click on change/create. Then Boundary Conditions Inlet – pressure inlet with gauge total pressure .5e5 and Outlet – Pressure Outlet -gauge pressure as 8e5.

Monitors Create –Drag - Print, Plot – Select Walls. Solutions Initialization – Calculate From – Inlet Run Calculation – No: Of Iterations -500 –Ok. Graphics –Contour Pressure –Static pressure and click On compute and then save/display. (Fig. 10.3)
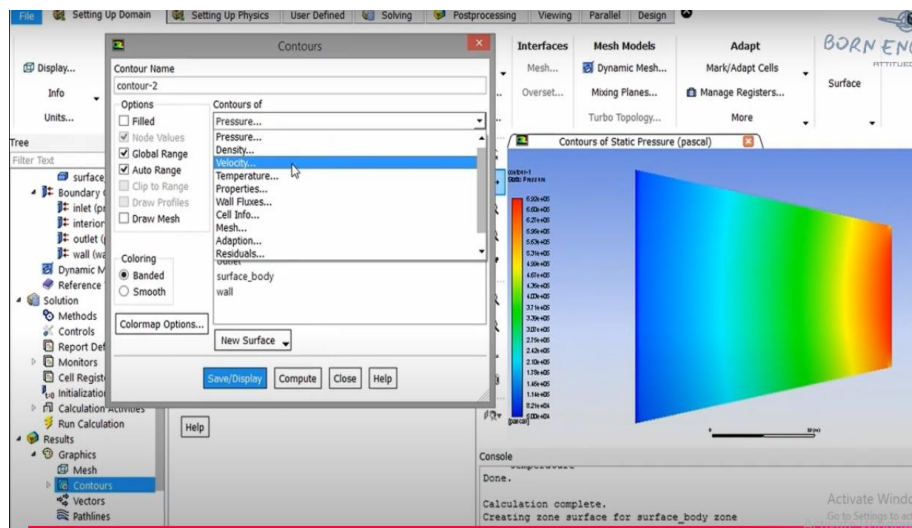

Fig. 10.3

To model isentropic flow in a 2-D supersonic nozzle:

Open Ansys Work Bench – Analysis System – Fluid Flow (Fluent). Then Geometry - New Geometry - Units-m- Select XY Plane

Sketching- Use line and draw the shape of C D Nozzle, with the extent of diffuser entry (V1) at 25m and exit height (V2) at 45m and the horizontal length (h3) is 50m (Fig. 10.4). Dimensions- Length or Distance –Give Dimensions as Specified
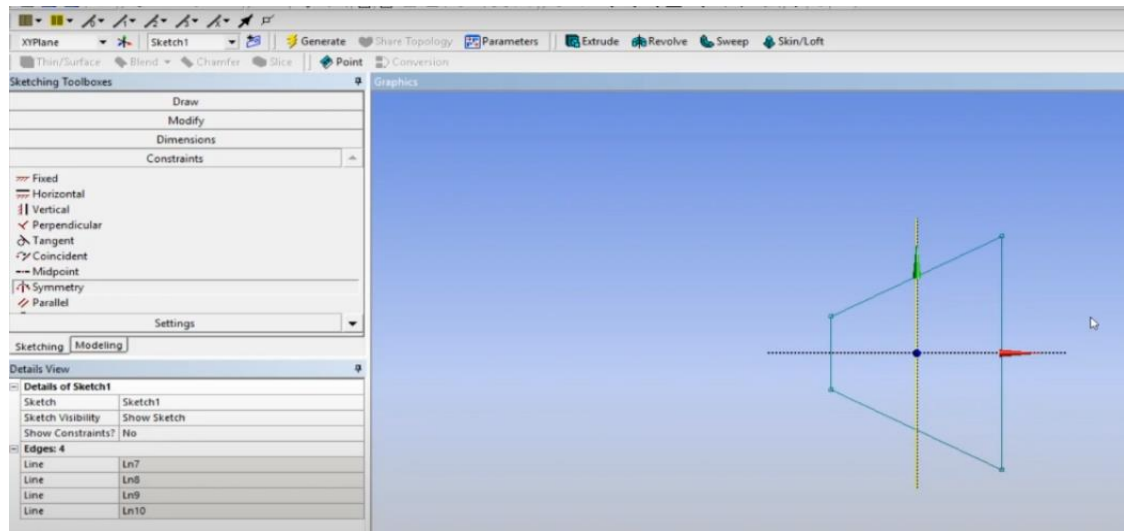

Fig. 10.4

Then go to concept menu and select surface from line to convert the lines of subsonic diffuser into a Surface. Mesh – Insert Method – Select Body –All Quads – Ok. Select - Mesh -Face meshing. Insert – Sizing –Vertical Edges –No. Of Divisions -50 for both vertical edges and no. of divisions of 100 for both horizontal edges. Insert –Mapped Face Meshing – Select All –Update. Select Edge– Create Named Selection - Inlet- Outlet Walls.

Select Solution – General-Density Based Models- Energy Equations –On. (Fig. 10.5). Then Materials- Air –density as ideal gas, Cp as constant, viscosity as Sutherland law . Then Boundary Conditions Inlet – velocity Inlet as 360 m/s and Outlet – Pressure Outlet -0 (Fig. 9.3)
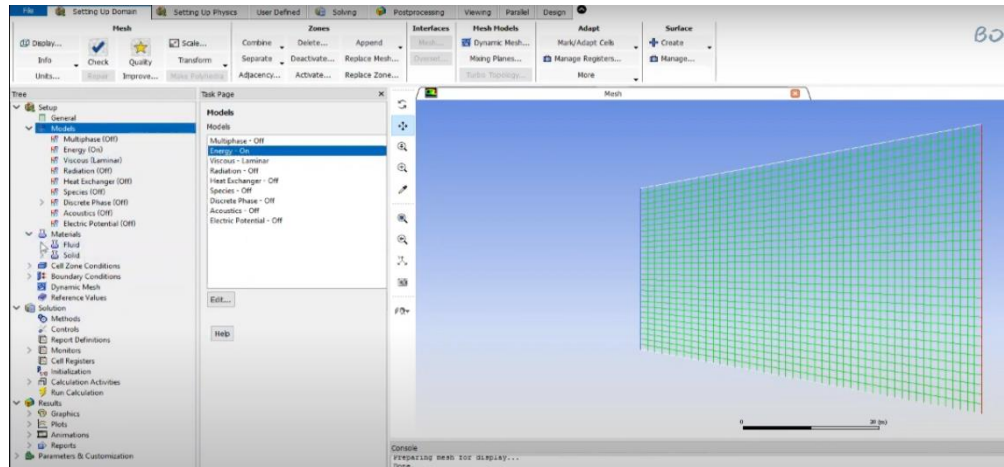
Fig. 10.5

Monitors Create –Drag - Print, Plot – Select Walls. Solutions Initialization – Calculate From – Inlet

Run Calculation – No: Of Iterations -500 –Ok. Graphics –Contour Pressure –Static pressure and click on compute and then save/display. Also, we can check the static temperature by selecting static temperature form the drop-down menu and click on compute and then display (Fig. 10.6)
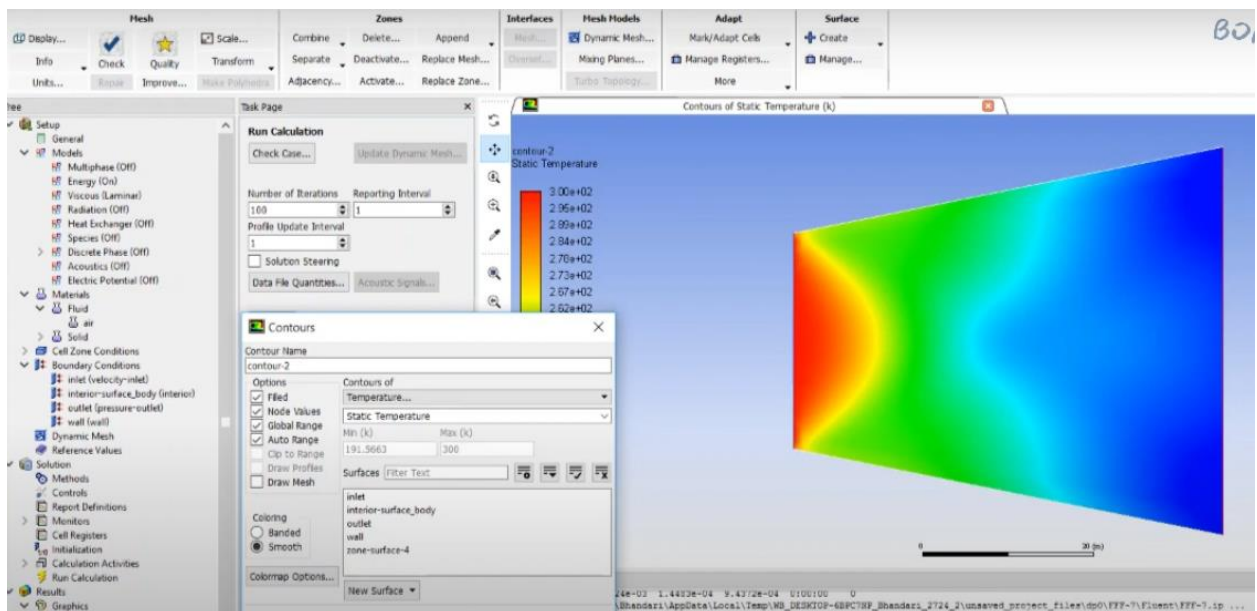

Fig. 10.6

VIVA QUESTIONS

1. what is isentropic flow and what conditions must be met for it?
2. How does flow behave in a supersonic nozzle?
3. What does the effect of area change in a supersonic diffuser?
4. Why does velocity decrease in a supersonic diffuser?
5. How are pressure and temperature affected in a supersonic nozzle?
6. What is the significance of Mach number greater than 1 in this experiment?
7. What assumptions are made for isentropic supersonic flow analysis?
8. What boundary conditions are set for simulating a supersonic nozzle and diffuser?
9. What solver and models are used in ANSYS for isentropic supersonic flow?
10. How do you identify shock formation or uniform expansion in your simulation results?