# INSTITUTE OF AERONAUTICAL ENGINEERING

**(Autonomous)**

Dundigal, Hyderabad – 500043

## COURSE CONTENT

| COMPUTATIONAL AERODYNAMICS LABORATORY | | | | | | | | |
|---|---|---|---|---|---|---|---|---|
| **VI Semester:** AE | | | | | | | | |
| **Course Code** | **Category** | **Hours / Week** | | | **Credits** | **Maximum Marks** | | |
| AAEC32 | Core | **L** | **T** | **P** | **C** | **CIA** | **SEE** | **Total** |
| | | - | - | 3 | 1.5 | 30 | 70 | 100 |
| **Contact Classes: Nil** | **Tutorial Classes: Nil** | **Practical Classes: 45** | | | | **Total Classes: 45** | | |
| **Prerequisite:** Mechanics of Solids | | | | | | | | |

## I. COURSE OVERVIEW:

Computational aerodynamics laboratory focus on the creation of geometry, meshing (Discretization) and the physics applied to aerodynamics in order to visualize fluid flow and temperature distribution, and estimating the flow parameters around the aerodynamic body. It also covers the usage of finite difference methods and necessary coding techniques. In this lab course, the students are trained on conducting simulations using the numerical methods analysis tool of CAD systems. The simulations include fluid, structural, thermal systems in the emerging technologies of interdisciplinary applications such as mechanical, aerospace, and refrigeration systems.

## II. COURSES OBJECTIVES:

**The students will try to learn:**

I. The concepts of grid generation techniques for simple and complex domains to model.
II. The aspects of numerical discretization techniques such as finite volume and finite difference methods.
III. The mathematical modelling of different classes of partial differential equations to show their impact on computational fluid dynamics.
IV. The characteristics of different turbulence models and numerical schemes for estimating the criteria of stability, convergence, and error of fluid flow problem.

## III. COURSE OUTCOMES:

**After successful completion of the course, students should be able to:**

CO1 Choose the finite difference method at grid points of the domain for understanding discretization technique in solving fluid flow problem.
CO2 Classify the nature of fluid flow problems for solving the governing equations using computational methods.
CO3 Make use of the computational methods and algorithms for obtaining solutions of fluid flow problems using Ansys
CO4 Modify the parameters of thermo-fluid systems using simulation methods for validating numerical and experimental results.
CO5 Calculate the aerodynamic forces on the slender and bluff bodies for calculating the lift and drag coefficients.
CO6 Interpret the numerical solution of fluid flow problems using discretization methods and convergence criteria for better results and minimize the errors.

# EXERCISES FOR COMPUTATIONAL AERODYNAMICS LABORATORY

**Note:** Students are encouraged to bring their own laptops for laboratory practice sessions.

## 1. Getting Started with Ansys Student Version

### 1.1 Ansys Student Version Installation procedure

**System requirement**

**Supported Platforms and Operating Systems:**

Microsoft Windows 10, 64-bit

**Minimum Hardware Requirements for Ansys Student Product:**

Processor(s): Workstation class

4 GB RAM

25 GB hard drive space

Computer must have a physical C:/" drive present

Graphics card and driver: Professional workstation class 3-D

OpenGL-capable

**Installation Procedure**

1. Extract (unzip) the downloaded installation files.
2. Right-click on setup.exe and select Run as Administrator. (This will run setup.exe from the extracted files.)
3. Read and accept the clickwrap to continue.
4. Click the right arrow button to accept the default values throughout the installation.
5. Click the exit button to close the installer.
6. The Ansys Student software is now installed.
7. Reboot your machine and then run the Ansys Student product from your Start menu by selecting Workbench.

**Problem size limits**

- No Geometry Export

Limits for Ansys Student and Discovery (Refine Mode)

- Structural Physics: 128K nodes/elements
- Fluid physics: 1 Million cells/nodes

## 1.2 Getting Started with Ansys CFX/ICEM CFD

Open Ansys ICEM CFD: Windows Start Menu button → Ansys → ICEM CFD

Ansys ICEM CFD provides a direct link between geometry and analysis. In Ansys ICEM CFD, you can input geometry in almost any format, whether a commercial CAD design package, third-party universal database, scan data, or point data available. The overall process has shown in Fig 1.1.

**The general meshing workflow is:**

1. Create a working directory.
2. Create a new project (or open an existing project).
3. Create or import the geometry.
4. If necessary, repair the geometry before meshing.
5. If necessary, block the geometry.
6. Compute the mesh
7. Check the mesh for errors and edit as required.
8. Describe the physics.
9. Generate the input for the solver

The ICEM CFD graphical user interface was shown in Fig 1.1.

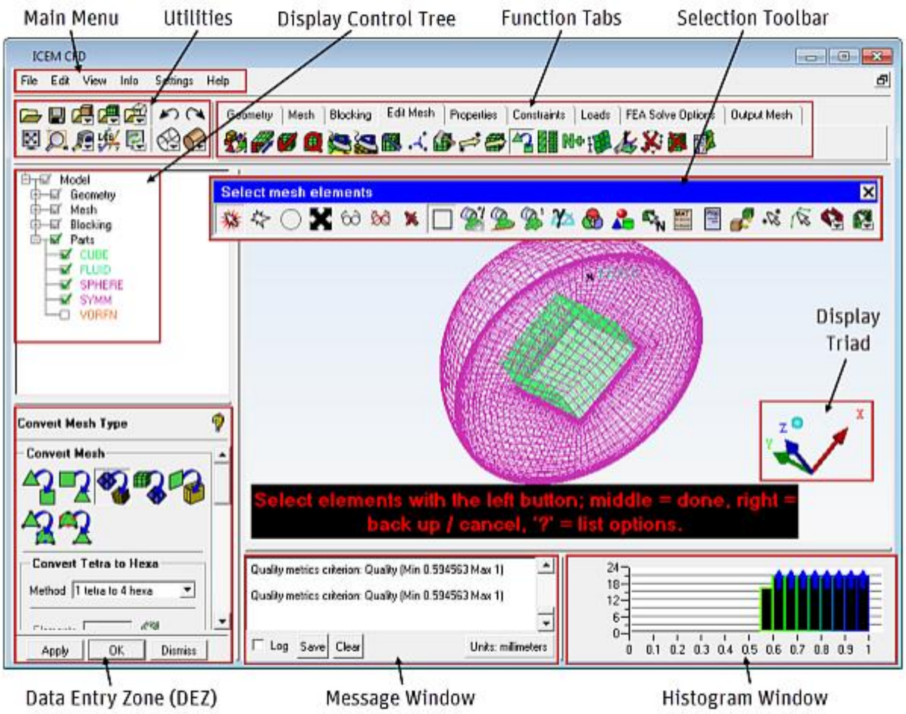



**Fig 1.1: ICEM CFD GUI**

**Geometry Preparation**

Geometry → Creating or Importing the Geometry→ Open

Describing the Physics

**Parts**

All geometry entities—including surfaces, curves and points—are tagged or associated into a grouping called a part.

**Blocking the Geometry**

The blocking step is used when a structured, hexa-mesh is desired in one or more parts

**Computing the Mesh**

To proceed from geometry and parts to a completed mesh requires two steps:

1. Set up Mesh Parameters-You should specify mesh size, type, and method along with several type- and method-specific controls.
2. Compute the Mesh

**Checking/Editing the Mesh**

The mesh editing tools in Ansys ICEM CFD enable you to diagnose and fix problems in the mesh.

# 2. Getting Started with FLUENT

Ansys Fluent to set up and solve the CFD problem, then visualize the results in both Ansys Fluent and in the CFD-Post postprocessing tool. The FLUENT GUI is arranged such that the tasks are generally arranged from top to bottom in the project setup tree. Selecting an item in the tree opens the relevant input items in the center pane.

## 2.1 Fluent GUI

**Set up the CFD simulation in Ansys Fluent, which includes:**

-Governing equations

-Turbulence models

– Setting material properties and boundary conditions for a turbulent forced-convection problem.

– Initiating the calculation with residual plotting.

– Calculating a solution using the pressure-based solver.

– Examining the flow and temperature fields using Ansys Fluent and CFD-Post.

The fluent graphical user interface was shown in Fig 2.1.

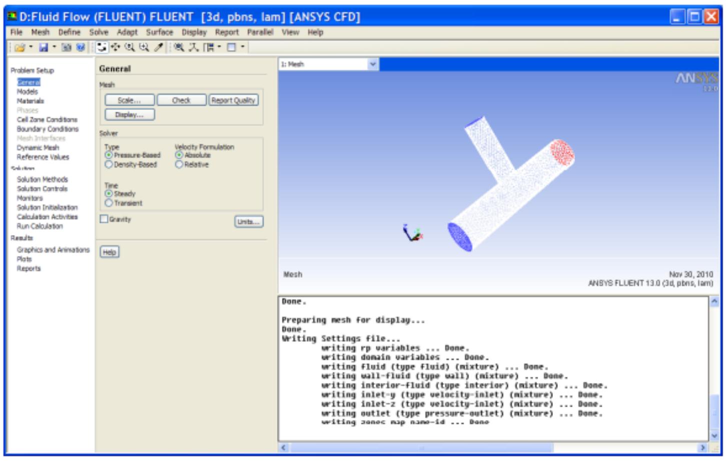



Fig. 2.1: Fluent GUI

## 2.2 Material selection

Material properties need to be defined for all fluids and solids to be simulated.
The parameters asked for will depend on the models selected for the simulation.
Many common materials are already defined in the 'FLUENT Database' and can easily be copied over to the model are shown in Fig 2.2.

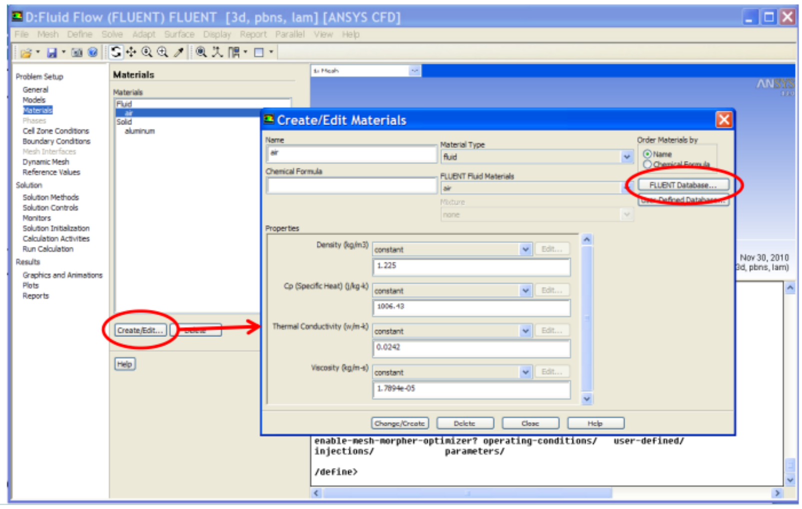



Fig. 2.2: Material properties

# 3. Exercises on FLOW OVER A FLAT PLATE

## 3.1 Flow over a flat plate – Laminar flow

Simulate the two-dimensional laminar flow on a horizontal flat plate. The size of the plate is considered being infinite in the spanwise direction and therefore the flow is 2D instead of 3D. The inlet velocity for the 1 m long plate is 5 m/s and we will be using air as the fluid for laminar simulations. Determine the velocity profiles and plot the profiles for flow over shown in Fig 3.1.

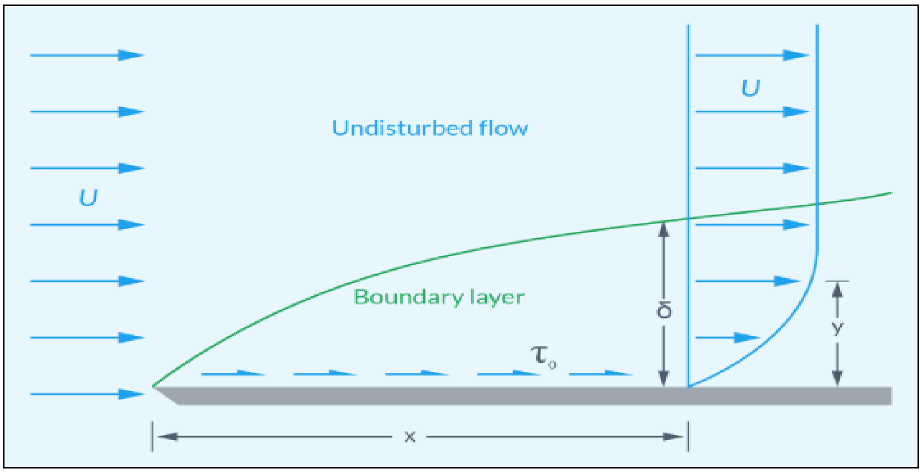



Fig. 3.1: LAMINAR FLOW OVER FLAT PLATE

**Hints**

```
1. Pre-processor
```

```
• Create new project file - File → New Project → Select folders in turn →
Save project with name
• Geometry → Create Point → specify values → Apply.
• Geometry → Create and modify curve → Select points → OK.
• Geometry → Right click "Parts" → Part = "Specify name" → OK.
• Blocking → OK.
• Mesh → Select geometry → Global Mesh Size → Compute Mesh icon→ OK.
• Output → Output Solver: Fluent V6 → apply
• Write Input → Save project → save as filename ".msh" → Done
• Main Menu → File → Export → Mesh...
```

```
2. Solution
```

```
• Start → Programs → Fluent Inc → FLUENT 6.0 →
• Main Menu > File > Read > Case...
• Main Menu > Display > Grid...
• Main Menu > Define > Models > Solver
• Main Menu > Define > Materials... Click Change/Create.
• Main Menu > Define > Operating Conditions... Operating Pressure.
• Main Menu > Define > Boundary Conditions...
• Main Menu > Solve > Controls > Solution...
• Main Menu > Solve > Initialize > Initialize... Set Initial Guess
• Main Menu > Solve > Monitors > Residual... Set Convergence Criteria
• Main Menu > File > Write > Case...
• Main Menu > Solve > Iterate... Iterate Until Convergence
• Main Menu > File > Write > Data...
```

```
3. General Post Processing:
```

```
• General Post Processing
• Main Menu > Report > Reference Values...
• Main Menu > Plot > XY Plot... Save Plot
• Main Menu > Plot > contours... Save Plot
```

## 3.2 Flow over a flat plate – Turbulent flow

Simulate the two-dimensional turbulent flow on a horizontal flat plate. The size of the plate is considered being infinite in the spanwise direction and therefore the flow is 2D instead of 3D. The inlet velocity for the 1 m long plate is 100 m/s and we will be using air as the fluid for laminar simulations. Determine the velocity profiles and plot the profiles.

**Hints**

```
1. Pre-processor
```

```
• Create new project file - File → New Project → Select folders in turn →
Save project with name
• Geometry → Create Point → specify values → Apply.
• Geometry → Create and modify curve → Select points → OK.
• Geometry → Right click "Parts" → Part = "Specify name" → OK.
• Blocking → OK.
• Mesh → Select geometry → Global Mesh Size → Compute Mesh icon→ OK.
• Output → Output Solver: Fluent V6 → apply
```

• Write Input → Save project → save as filename ".msh" → Done
• Main Menu → File → Export → Mesh...

2. Solution

• Start → Programs → Fluent Inc → FLUENT 6.0 →
• Main Menu > File > Read > Case...
• Main Menu > Display > Grid...
• Main Menu > Define > Models > Solver
• Main Menu > Define > Materials... Click Change/Create.
• Main Menu > Define > Operating Conditions... Operating Pressure.
• Main Menu > Define > Boundary Conditions...
• Main Menu > Solve > Controls > Solution...
• Main Menu > Solve > Initialize > Initialize... Set Initial Guess
• Main Menu > Solve > Monitors > Residual... Set Convergence Criteria
• Main Menu > File > Write > Case...
• Main Menu > Solve > Iterate... Iterate Until Convergence
• Main Menu > File > Write > Data...

3. General Post Processing:

• General Post Processing
• Main Menu > Report > Reference Values...
• Main Menu > Plot > XY Plot... Save Plot
• Main Menu > Plot > contours... Save Plot

### 3.3 Flow over a flat plate – Mach number

Model two-dimensional laminar flow on a horizontal flat plate. The size of the plate is considered being infinite in the spanwise direction and therefore the flow is 2D instead of 3D. The inlet Mach number for the 1 m long plate is 0.5 and we will be using air as the fluid for laminar simulations. Determine the velocity profiles and plot the profiles.

#### Hints

1. Pre-processor

• Create new project file - File → New Project → Select folders in turn → Save project with name
• Geometry → Create Point → specify values → Apply.
• Geometry → Create and modify curve → Select points → OK.
• Geometry → Right click "Parts" → Part = "Specify name" → OK.
• Blocking → OK.
• Mesh → Select geometry → Global Mesh Size → Compute Mesh icon→ OK.
• Output → Output Solver: Fluent V6 → apply
• Write Input → Save project → save as filename ".msh" → Done
• Main Menu → File → Export → Mesh...

2. Solution

• Start → Programs → Fluent Inc → FLUENT 6.0 →
• Main Menu > File > Read > Case...
• Main Menu > Display > Grid...
• Main Menu > Define > Models > Solver
• Main Menu > Define > Materials... Click Change/Create.
• Main Menu > Define > Operating Conditions... Operating Pressure.
• Main Menu > Define > Boundary Conditions...

```
• Main Menu > Solve > Controls > Solution...
• Main Menu > Solve > Initialize > Initialize... Set Initial Guess
• Main Menu > Solve > Monitors > Residual... Set Convergence Criteria
• Main Menu > File > Write > Case...
• Main Menu > Solve > Iterate... Iterate Until Convergence
• Main Menu > File > Write > Data...
```

```
3. General Post Processing:
```

```
• General Post Processing
• Main Menu > Report > Reference Values...
• Main Menu > Plot > XY Plot... Save Plot
• Main Menu > Plot > contours... Save Plot
```

**Try**

1. Model and analyze the flow over the flat plate with vortex generator of triangular shape as per the dimensions and plot the variation at pressure over the flat plate and XY plot using the available tools at a velocity of 0.5 m/s by considering the flow to be viscous. Use the flow parameters as Length =10 M and Breadth =5M.
2. Model and analyze the flow over the flat plate as per the dimensions and plot the variation at pressure over the flat plate and XY plot using the available tools at a velocity of 1 m/s by considering the flow to be viscous. Use the flow parameters as Length =5 M and Breadth =3M.

# 4. Exercises on flow through pipe

## 4.1 Flow through pipe - Laminar

Study characteristics of laminar flow through a pipe. Consider a pipe of breadth 0.05 and 1 m length as shown in Fig. 4.1. The freestream velocity considered is 0.5 m/s. Determine the flow development inside the pipe at laminar and turbulent zone.

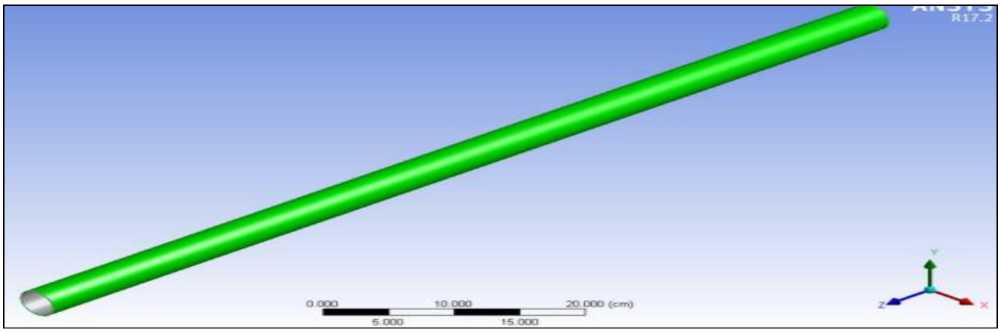



**Fig. 4.1: Flow through pipe**

**Hints**

```
1. Pre-processor
```

```
• Create new project file - File → New Project → Select folders in turn → Save project with name
• Geometry → Create Point → specify values → Apply.
• Geometry → Create and modify curve → Select points → OK.
• Geometry → Right click “Parts” → Part = “Specify name” → OK.
• Blocking → OK.
```

- Mesh → Select geometry → Global Mesh Size → Compute Mesh icon→ OK.
- Output → Output Solver: Fluent V6 → apply
- Write Input → Save project → save as filename ".msh" → Done
- Main Menu → File → Export → Mesh...

2. Solution

- Start → Programs → Fluent Inc → FLUENT 6.0 →
- Main Menu > File > Read > Case...
- Main Menu > Display > Grid...
- Main Menu > Define > Models > Solver
- Main Menu > Define > Models > laminar
- Main Menu > Define > Materials... Click Change/Create.
- Main Menu > Define > Operating Conditions... Operating Pressure.
- Main Menu > Define > Boundary Conditions...
- Main Menu > Solve > Controls > Solution...
- Main Menu > Solve > Initialize > Initialize... Set Initial Guess
- Main Menu > Solve > Monitors > Residual... Set Convergence Criteria
- Main Menu > File > Write > Case...
- Main Menu > Solve > Iterate... Iterate Until Convergence
- Main Menu > File > Write > Data...

3. General Post Processing:

- General Post Processing
- Main Menu > Report > Reference Values...
- Main Menu > Plot > XY Plot... Save Plot
- Main Menu > Plot > contours... Save Plot

## 4.2 Flow through pipe -Turbulent

Study characteristics of turbulent flow through a rectangular pipe. Consider a pipe of breadth 0.05m, h=0.1 and 1 m length. The freestream velocity considered is 50 m/s. Determine the flow development inside the pipe at laminar and turbulent zone.

### Hints

1. Pre-processor

- Create new project file - File → New Project → Select folders in turn → Save project with name
- Geometry → Create Point → specify values → Apply.
- Geometry → Create and modify curve → Select points → OK.
- Geometry → Right click "Parts" → Part = "Specify name" → OK.
- Blocking → OK.
- Mesh → Select geometry → Global Mesh Size → Compute Mesh icon→ OK.
- Output → Output Solver: Fluent V6 → apply
- Write Input → Save project → save as filename ".msh" → Done
- Main Menu → File → Export → Mesh...

2. Solution

- Start → Programs → Fluent Inc → FLUENT 6.0 →
- Main Menu > File > Read > Case...
- Main Menu > Display > Grid...
- Main Menu > Define > Models > Solver

```
• Main Menu > Define > Models > Viscous
• Main Menu > Define > Materials... Click Change/Create.
• Main Menu > Define > Operating Conditions... Operating Pressure.
• Main Menu > Define > Boundary Conditions...
• Main Menu > Solve > Controls > Solution...
• Main Menu > Solve > Initialize > Initialize... Set Initial Guess
• Main Menu > Solve > Monitors > Residual... Set Convergence Criteria
• Main Menu > File > Write > Case...
• Main Menu > Solve > Iterate... Iterate Until Convergence
• Main Menu > File > Write > Data...
```

```
3. General Post Processing:
```

```
• General Post Processing
• Main Menu > Report > Reference Values...
• Main Menu > Plot > XY Plot... Save Plot
• Main Menu > Plot > contours... Save Plot
```

### Try

1. Change the velocity, dimensions of the pipe and plot the velocity and pressure contours.
2. Modify the Reynolds number, dimensions of the pipe and plot the velocity magnitude and pressure contours.

# 5. Exercises on flow over a circular cylinder

## 5.1 Flow over a circular cylinder - Laminar

Model and analyze the flow over the cylinder as per the dimensions shown in Fig 5.1 and plot the variation at pressure over the cylinder and evaluate the $C_L$ and $C_D$ values using the available tools at a velocity of 0 .5 m/s by considering the flow to be viscous. Diameter of the cylinder is 0.5 m.

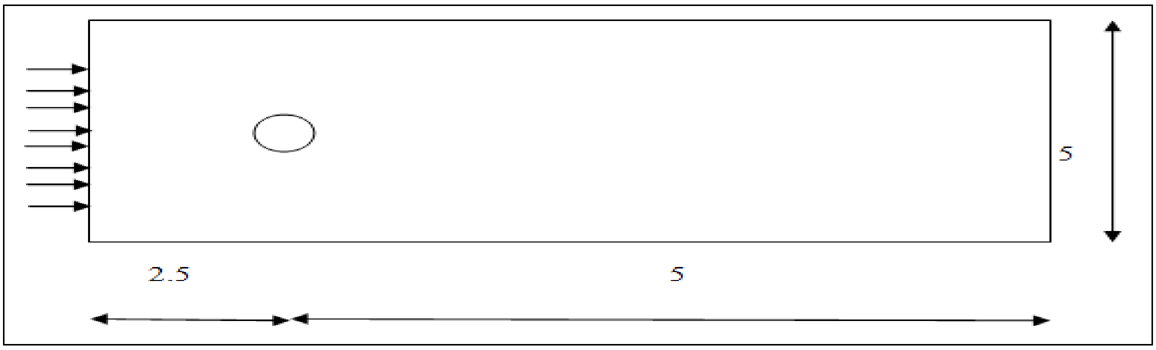



**Fig. 5.1: Flow over the cylinder**

### Hints

```
1. Pre-processor
```

```
• Create new project file - File → New Project → Select folders in turn →
Save project with name
• Geometry → Create Point → specify values → Apply.
• Geometry → Create and modify curve → Select points → OK.
• Geometry → Right click "Parts" → Part = "Specify name" → OK.
• Blocking → OK.
• Mesh → Select geometry → Global Mesh Size → Compute Mesh icon→ OK.
• Output → Output Solver: Fluent V6 → apply
• Write Input → Save project → save as filename ".msh" → Done
```

- Main Menu → File → Export → Mesh...

2. Solution

- Start → Programs → Fluent Inc → FLUENT 6.0 →
- Main Menu > File > Read > Case...
- Main Menu > Display > Grid...
- Main Menu > Define > Models > Solver
- Main Menu > Define > Models > Viscous
- Main Menu > Define > Materials... Click Change/Create.
- Main Menu > Define > Operating Conditions... Operating Pressure.
- Main Menu > Define > Boundary Conditions...
- Main Menu > Solve > Controls > Solution...
- Main Menu > Solve > Initialize > Initialize... Set Initial Guess
- Main Menu > Solve > Monitors > Residual... Set Convergence Criteria
- Main Menu > File > Write > Case...
- Main Menu > Solve > Iterate... Iterate Until Convergence
- Main Menu > File > Write > Data...

3. General Post Processing:

- General Post Processing
- Main Menu > Report > Reference Values...
- Main Menu > Plot > XY Plot... Save Plot
- Main Menu > Plot > contours... Save Plot

## 5.2 Flow over a circular cylinder – Turbulent

Model and analyze the flow over the cylinder as per the dimensions shown in Fig 5.2 and plot the variation of pressure distribution over the cylinder and evaluate the $C_L$ and $C_D$ values using the available tools at a velocity of 50 m/s by considering the flow to be viscous. Diameter of the cylinder = 2m.

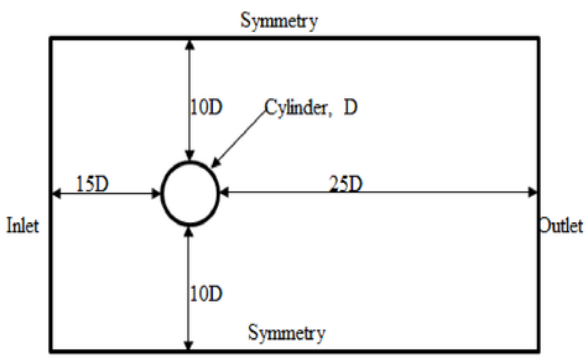



**Fig. 5.2: Flow over the cylinder**

### Hints

1. Pre-processor

- Create new project file - File → New Project → Select folders in turn → Save project with name
- Geometry → Create Point → specify values → Apply.
- Geometry → Create and modify curve → Select points → OK.
- Geometry → Right click "Parts" → Part = "Specify name" → OK.
- Blocking → OK.
- Mesh → Select geometry → Global Mesh Size → Compute Mesh icon→ OK.

```
• Output → Output Solver: Fluent V6 → apply
• Write Input → Save project → save as filename ".msh" → Done
• Main Menu → File → Export → Mesh..
```

```
2.  Solution
```

```
• Start → Programs → Fluent Inc → FLUENT 6.0 →
• Main Menu > File > Read > Case...
• Main Menu > Display > Grid...
• Main Menu > Define > Models > Solver
• Main Menu > Define > Materials... Click Change/Create.
• Main Menu > Define > Operating Conditions... Operating Pressure.
• Main Menu > Define > Boundary Conditions...
• Main Menu > Solve > Controls > Solution...
• Main Menu > Solve > Initialize > Initialize... Set Initial Guess
• Main Menu > Solve > Monitors > Residual... Set Convergence Criteria
• Main Menu > File > Write > Case...
• Main Menu > Solve > Iterate... Iterate Until Convergence
• Main Menu > File > Write > Data...
```

```
3.  General Post Processing:
```

```
• General Post Processing
• Main Menu > Report > Reference Values...
• Main Menu > Plot > XY Plot... Save Plot
• Main Menu > Plot > contours... Save Plot
```

**Try**

1. Change the diameter, location of circle and plot the variation of pressure over the cylinder.
2. Change the velocity and dimensions plot the velocity and pressure contours.

# 6. Exercises on flow over a cambered aerofoil

## 6.1 Flow over a cambered aerofoil - NACA 2412

Consider air flowing over NACA 2412 airfoil as shown in Fig 6.1 at the free stream Mach number is 1.2. Assume standard sea-level values for the free stream properties:

Pressure = 101,325 Pa
Density = 1.2250 kg/m3
Temperature = 288.16 K
Kinematic viscosity v = 1.4607e-5 m2/s

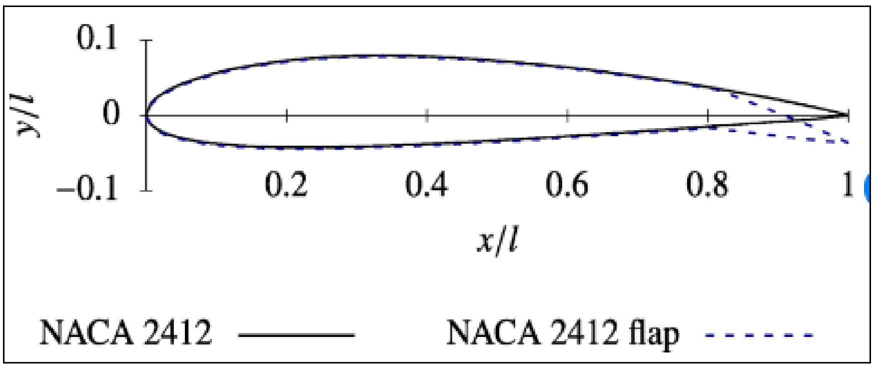



**Fig. 6.1: Flow over a airfoil – NACA2412**

**Hints**

```
1. Pre-processor
```

```
• Create new project file - File → New Project → Select folders in turn →
Save project with name
• Geometry → Create Point → specify values → Apply.
• Geometry → Create and modify curve → Select points → OK.
• Geometry → Right click "Parts" → Part = "Specify name" → OK.
• Blocking → OK.
• Mesh → Select geometry → Global Mesh Size → Compute Mesh icon→ OK.
• Output → Output Solver: Fluent V6 → apply
• Write Input → Save project → save as filename ".msh" → Done
• Main Menu → File → Export → Mesh..
```

```
2. Solution
```

```
• Start → Programs → Fluent Inc → FLUENT 6.0 →
• Main Menu > File > Read > Case...
• Main Menu > Display > Grid...
• Main Menu > Define > Models > Solver
• Main Menu > Define > Materials... Click Change/Create.
• Main Menu > Define > Operating Conditions... Operating Pressure.
• Main Menu > Define > Boundary Conditions...
• Main Menu > Solve > Controls > Solution...
• Main Menu > Solve > Initialize > Initialize... Set Initial Guess
• Main Menu > Solve > Monitors > Residual... Set Convergence Criteria
• Main Menu > File > Write > Case...
• Main Menu > Solve > Iterate... Iterate Until Convergence
• Main Menu > File > Write > Data...
```

```
3. General Post Processing:
```

```
• General Post Processing
• Main Menu > Report > Reference Values...
• Main Menu > Plot > XY Plot... Save Plot
• Main Menu > Plot > contours... Save Plot
```

**Try**

1. Consider the different Reynolds number and evaluate the $C_L$ and Cd and Cp distribution.
2. Consider the different angle of attack and evaluate the $C_L$ and Cd and Cp distribution.

# 7. Exercises on flow over a symmetric aerofoil

## 7.1. Flow over a symmetric aerofoil - NACA 0012

Consider air flowing over NACA 0012 airfoil as shown in Fig 7.1 at the free stream Mach number is 1.2. Assume standard sea-level values for the free stream properties:

Pressure = 101,325 Pa
Density = 1.2250 kg/m3
Temperature = 288.16 K
Kinematic viscosity v = 1.4607e-5 m2/s

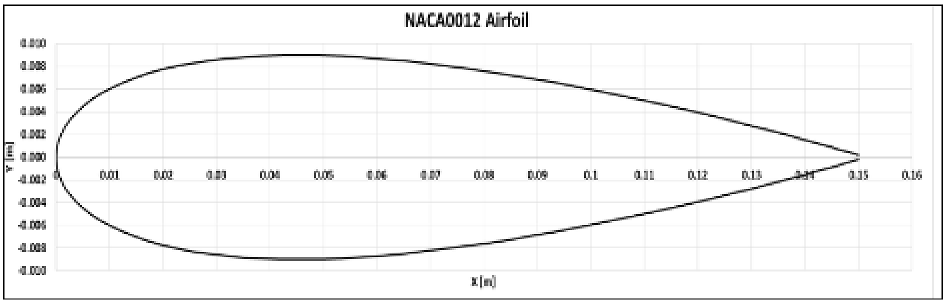



**Fig. 6.2: Flow over a airfoil – NACA0012**

## Hints

1. Pre-processor

- Create new project file - File → New Project → Select folders in turn → Save project with name
- Geometry → Create Point → specify values → Apply.
- Geometry → Create and modify curve → Select points → OK.
- Geometry → Right click “Parts” → Part = “Specify name” → OK.
- Blocking → OK.
- Mesh → Select geometry → Global Mesh Size → Compute Mesh icon→ OK.
- Output → Output Solver: Fluent V6 → apply
- Write Input → Save project → save as filename “.msh” → Done
- Main Menu → File → Export → Mesh...

2. Solution

- Start → Programs → Fluent Inc → FLUENT 6.0 →
- Main Menu > File > Read > Case...
- Main Menu > Display > Grid...
- Main Menu > Define > Models > Solver
- Main Menu > Define > Materials... Click Change/Create.
- Main Menu > Define > Operating Conditions... Operating Pressure.
- Main Menu > Define > Boundary Conditions...
- Main Menu > Solve > Controls > Solution...
- Main Menu > Solve > Initialize > Initialize... Set Initial Guess
- Main Menu > Solve > Monitors > Residual... Set Convergence Criteria
- Main Menu > File > Write > Case...
- Main Menu > Solve > Iterate... Iterate Until Convergence
- Main Menu > File > Write > Data...

3. General Post Processing:

- General Post Processing
- Main Menu > Report > Reference Values...
- Main Menu > Plot > XY Plot... Save Plot
- Main Menu > Plot > contours... Save Plot

## Try

1. Consider the different Reynolds number and evaluate the $C_L$ and Cd and Cp distribution.
2. Consider the different angle of attack and evaluate the $C_L$ and Cd and Cp distribution.

# 8. Exercises on flow over wedge

## 8.1 Flow over wedge - zero angle of attack

Consider a 15° angle wedge at zero angle of attack shown in Fig 8.1. The incoming flow conditions are: Mach no= 3, $p_1$=1 atm, $T_1$=273 K. Use FLUENT to obtain the flow field over the wedge. Evaluate the results of pressure, boundary layer profile variation by contours and plots.

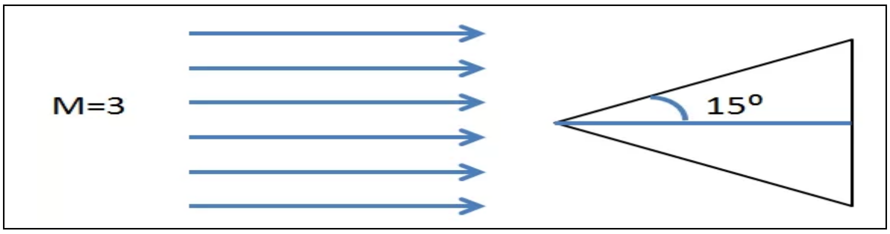



**Fig. 8.1: Flow over a wedge - Mach number 3**

### Hints

1. Pre-processor

- Create new project file - File → New Project → Select folders in turn → Save project with name
- Geometry → Create Point → specify values → Apply.
- Geometry → Create and modify curve → Select points → OK.
- Geometry → Right click "Parts" → Part = "Specify name" → OK.
- Blocking → OK.
- Mesh → Select geometry → Global Mesh Size → Compute Mesh icon→ OK.
- Output → Output Solver: Fluent V6 → apply
- Write Input → Save project → save as filename ".msh" → Done
- Main Menu → File → Export → Mesh...

2. Solution

- Start → Programs → Fluent Inc → FLUENT 6.0 →
- Main Menu > File > Read > Case...
- Main Menu > Display > Grid...
- Main Menu > Define > Models > Solver
- Main Menu > Define > Materials... Click Change/Create.
- Main Menu > Define > Operating Conditions... Operating Pressure.
- Main Menu > Define > Boundary Conditions...
- Main Menu > Solve > Controls > Solution...
- Main Menu > Solve > Initialize > Initialize... Set Initial Guess
- Main Menu > Solve > Monitors > Residual... Set Convergence Criteria
- Main Menu > File > Write > Case...
- Main Menu > Solve > Iterate... Iterate Until Convergence
- Main Menu > File > Write > Data...

3. General Post Processing:

- General Post Processing
- Main Menu > Report > Reference Values...
- Main Menu > Plot > XY Plot... Save Plot
- Main Menu > Plot > contours... Save Plot

## 8.2 Flow over wedge – Mach number

Consider a 15° angle wedge at zero angle of attack shown in Fig.8.2. The incoming flow conditions are: Mach no= 4, p1=1 atm, T1=273 K. Use FLUENT to obtain the flow field over the wedge. Evaluate the results of pressure, boundary layer profile variation by contours and plots.

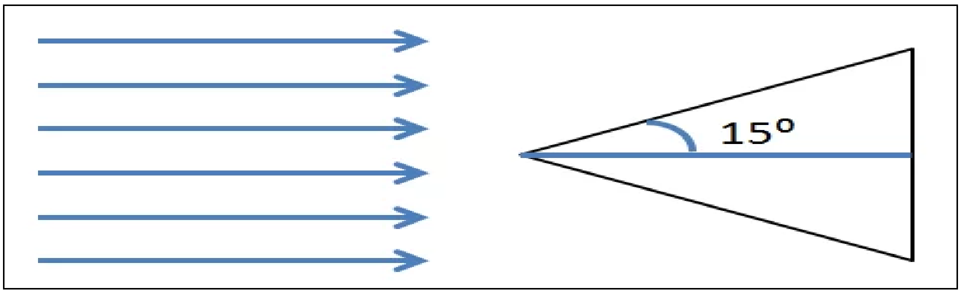



**Fig. 8.2: Flow over a wedge – Mach number 4**

### Hints

1. Pre-processor

- Create new project file - File → New Project → Select folders in turn → Save project with name
- Geometry → Create Point → specify values → Apply.
- Geometry → Create and modify curve → Select points → OK.
- Geometry → Right click "Parts" → Part = "Specify name" → OK.
- Blocking → OK.
- Mesh → Select geometry → Global Mesh Size → Compute Mesh icon→ OK.
- Output → Output Solver: Fluent V6 → apply
- Write Input → Save project → save as filename ".msh" → Done
- Main Menu → File → Export → Mesh...

2. Solution

- Start → Programs → Fluent Inc → FLUENT 6.0 →
- Main Menu > File > Read > Case...
- Main Menu > Display > Grid...
- Main Menu > Define > Models > Solver
- Main Menu > Define > Models > Viscous
- Main Menu > Define > Materials... Click Change/Create.
- Main Menu > Define > Operating Conditions... Operating Pressure.
- Main Menu > Define > Boundary Conditions...
- Main Menu > Solve > Controls > Solution...
- Main Menu > Solve > Initialize > Initialize... Set Initial Guess
- Main Menu > Solve > Monitors > Residual... Set Convergence Criteria
- Main Menu > File > Write > Case...
- Main Menu > Solve > Iterate... Iterate Until Convergence
- Main Menu > File > Write > Data...

3. General Post Processing:

- General Post Processing
- Main Menu > Report > Reference Values...
- Main Menu > Plot > XY Plot... Save Plot
- Main Menu > Plot > contours... Save Plot

**Try**

1. Consider different angle wedge at zero angle of attack. Evaluate the results of pressure, boundary layer profile variation by contours and plots.
2. Consider different flow over a wedge at zero angle of attack. Evaluate the results of pressure, boundary layer profile variation by contours and plots.

# 9. Exercises on flow over a cone

## 9.1 Flow over a cone

Study supersonic flow past a 3D Cone at an angle of attack as shown in Fig 9.1. Analyze the simulation and understand the physics of supersonic flows over cone.

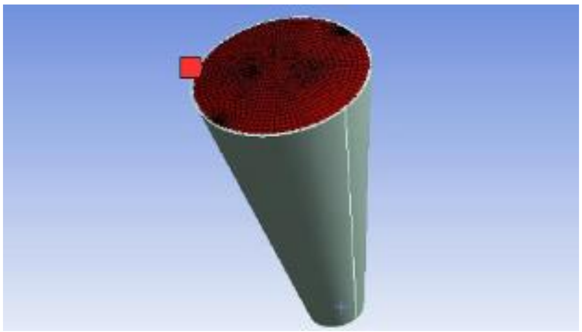


**Fig 9.1. 3D Cone at an angle of attack**

### Hints

```
1. Pre-processor
```

```
• Create new project file - File → New Project → Select folders in turn → Save project with name
• Geometry → Create Point → specify values → Apply.
• Geometry → Create and modify curve → Select points → OK.
• Geometry → Right click "Parts" → Part = "Specify name" → OK.
• Blocking → OK.
• Mesh → Select geometry → Global Mesh Size → Compute Mesh icon→ OK.
• Output → Output Solver: Fluent V6 → apply
• Write Input → Save project → save as filename ".msh" → Done
• Main Menu → File → Export → Mesh...
```

```
2. Solution
```

```
• Start → Programs → Fluent Inc → FLUENT 6.0 →
• Main Menu > File > Read > Case...
• Main Menu > Display > Grid...
• Main Menu > Define > Models > Solver
• Main Menu > Define > Models > Viscous
• Main Menu > Define > Materials... Click Change/Create.
• Main Menu > Define > Operating Conditions... Operating Pressure.
• Main Menu > Define > Boundary Conditions...
• Main Menu > Solve > Controls > Solution...
• Main Menu > Solve > Initialize > Initialize... Set Initial Guess
• Main Menu > Solve > Monitors > Residual... Set Convergence Criteria
• Main Menu > File > Write > Case...
• Main Menu > Solve > Iterate... Iterate Until Convergence
```

```
• Main Menu > File > Write > Data...
```

```
3. General Post Processing:
```

```
• General Post Processing
• Main Menu > Report > Reference Values...
• Main Menu > Plot > XY Plot... Save Plot
• Main Menu > Plot > contours... Save Plot
```

## 9.2 Flow over a wedge and cone – results comparison

Study supersonic flow past a 3D Cone and wedge at an angle of attack. Analyze the simulation and understand the physics of supersonic flows over cones and wedge for different contours in Fig 9.2.

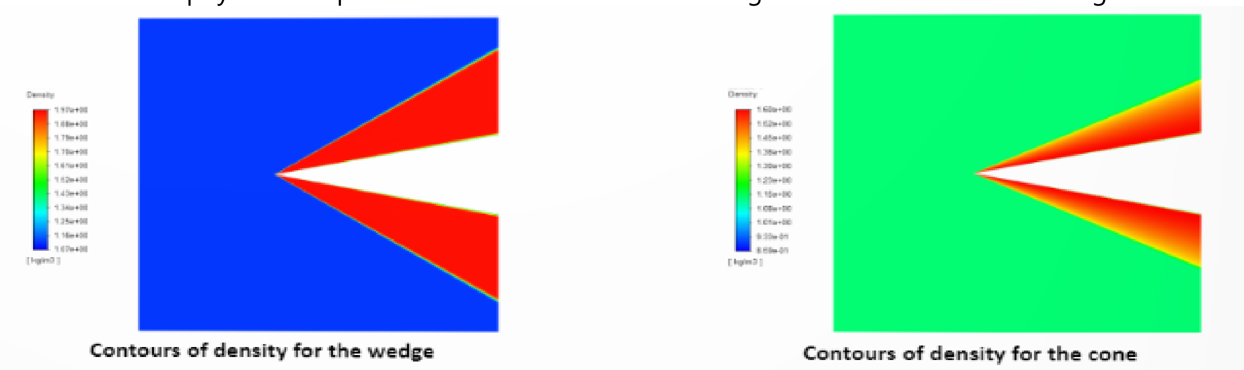



**Fig 9.2. Results comparison between wedge and cone**

### Hints

```
1. Results and discussion
```

```
• General Post Processing:
• Main Menu > Report > Reference Values...
• Main Menu > Plot > XY Plot... Save Plot
• Main Menu > Plot > contours... Save Plot
```

**Try**

1. Repeat the above analysis at different angle of attack and visualize the shock.
2. Change the Mach number and observes the shock waves and 3D relieving effect.

# 10. Exercises on flow analysis of aircraft structure: Wing

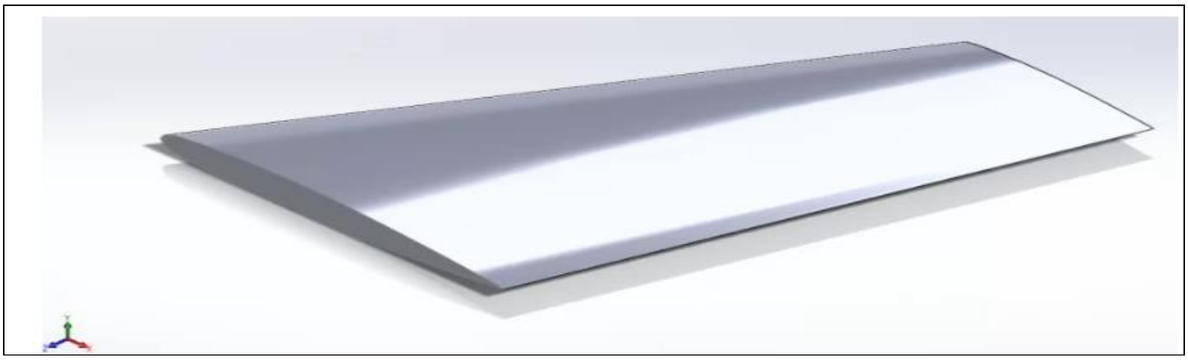
## 10.1 Flow analysis of aircraft structure: Wing (Onera M6 wing)

Flow over the Onera M6 wing is transonic and compressible as shown in Fig 10.1. The wing flow experiences supersonic conditions, a shock, and boundary layer separation. There is no wing twist, with all chords being on the same plane. Therefore, the angle of attack is simply the angle between the free stream and the chord line. There is no side-slip in the simulation. The flow conditions are given below

| Mach | Reynolds Number | Angle of Attack (degrees) | Angle of Side-slip (degrees) |
|---|---|---|---|
| .8395 | 11.72E6 | 3.06 | 0 |

The half span dimension is 1 ft and from there we were able to calculate a scaling factor for the entire wing. More about the geometry creation can be found on the exercises page. The table below describes some key geometries, the leading and trailing edge angles are measured from the vertical.

| Span (ft) | Taper Ratio | Mean Aerodynamic Chord (ft) | Leading Edge Angle (degrees) | Trailing Edge Angle (degrees) |
|---|---|---|---|---|
| 1 | .562 | .5400 | 30 | 15.8 |



**10.1 Onera M6 wing**

## Hints

1. Pre-processor

```
• Create new project file - File → New Project → Select folders in turn →
Save project with name
• Geometry → Create Point → specify values → Apply.
• Geometry → Create and modify curve → Select points → OK.
• Geometry → Right click “Parts” → Part = “Specify name” → OK.
• Blocking → OK.
• Mesh → Select geometry → Global Mesh Size → Compute Mesh icon→ OK.
• Output → Output Solver: Fluent V6 → apply
• Write Input → Save project → save as filename “.msh” → Done
• Main Menu → File → Export → Mesh...
```

2. Solution

```
• Start → Programs → Fluent Inc → FLUENT 6.0 →
• Main Menu > File > Read > Case...
• Main Menu > Display > Grid...
• Main Menu > Define > Models > Solver
• Main Menu > Define > Models > Viscous
• Main Menu > Define > Materials... Click Change/Create.
• Main Menu > Define > Operating Conditions... Operating Pressure.
• Main Menu > Define > Boundary Conditions...
• Main Menu > Solve > Controls > Solution...
• Main Menu > Solve > Initialize > Initialize... Set Initial Guess
• Main Menu > Solve > Monitors > Residual... Set Convergence Criteria
• Main Menu > File > Write > Case...
• Main Menu > Solve > Iterate... Iterate Until Convergence
• Main Menu > File > Write > Data...
```

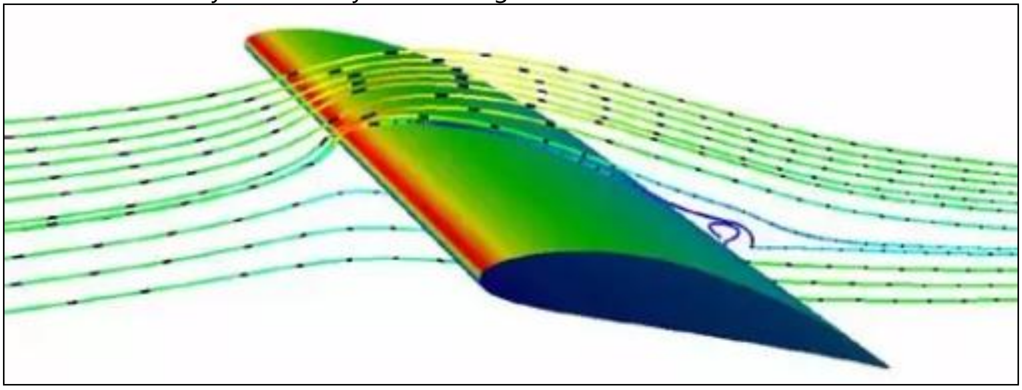
3. General Post Processing:

```
• General Post Processing
• Main Menu > Report > Reference Values...
```

```
• Main Menu > Plot > XY Plot... Save Plot
• Main Menu > Plot > contours... Save Plot
```

## 10.2 Flow analysis (Rectangular Wing)

Perform flow analysis of an aircraft rectangular wing of a span 100cm and the chord length of wing is 10 cm and cross-sectional area is 100 $cm^2$ as shown in Fig 10.2. Evaluate the $C_L$ and $C_D$ values using the available tools at a velocity of 5 m/s by considering the flow to be viscous.



**10.2 Rectangular wing**

**Hints**

```
1. Pre-processor
```

```
• Create new project file - File → New Project → Select folders in turn →
Save project with name
• Geometry → Create Point → specify values → Apply.
• Geometry → Create and modify curve → Select points → OK.
• Geometry → Right click "Parts" → Part = "Specify name" → OK.
• Blocking → OK.
• Mesh → Select geometry → Global Mesh Size → Compute Mesh icon→ OK.
• Output → Output Solver: Fluent V6 → apply
• Write Input → Save project → save as filename ".msh" → Done
• Main Menu → File → Export → Mesh...
```

```
2. Solution
```

```
• Start → Programs → Fluent Inc → FLUENT 6.0 →
• Main Menu > File > Read > Case...
• Main Menu > Display > Grid...
• Main Menu > Define > Models > Solver
• Main Menu > Define > Models > Viscous
• Main Menu > Define > Materials... Click Change/Create.
• Main Menu > Define > Operating Conditions... Operating Pressure.
• Main Menu > Define > Boundary Conditions...
• Main Menu > Solve > Controls > Solution...
• Main Menu > Solve > Initialize > Initialize... Set Initial Guess
• Main Menu > Solve > Monitors > Residual... Set Convergence Criteria
• Main Menu > File > Write > Case...
• Main Menu > Solve > Iterate... Iterate Until Convergence
• Main Menu > File > Write > Data...
```

```
3. General Post Processing:
```

```
• General Post Processing
• Main Menu > Report > Reference Values...
```

```
• Main Menu > Plot > XY Plot... Save Plot
• Main Menu > Plot > contours... Save Plot
```

**Try**

1. Calculate lift and drag coefficient of an elliptical wing.
2. Calculate lift and drag coefficient of tapered wing with different Mach number and angle of attack.

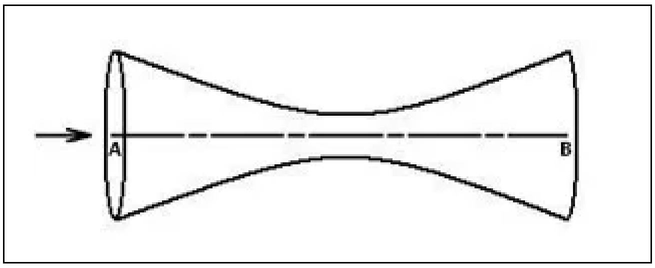
# 11. Exercises on Compressible flow in a nozzle

## 11.1 Compressible flow in a nozzle – Correctly expanded case

Consider air flowing at high-speed through a convergent-divergent nozzle in Fig 11.1 having a circular cross-sectional area, A, that varies with axial distance from the throat, x, according to the formula

$$A = 0.1 + x^2;\ -0.5 < x < 0.5$$

where A is in square meters and x is in meters. The stagnation pressure po at the inlet is 101,325 Pa. The stagnation temperature To at the inlet is 300 K. The static pressure p at the exit is 3,738.9 Pa. Calculate the Mach number, pressure and temperature distribution in the nozzle using Ansys Fluent.




**11.1 convergent-divergent nozzle**

### Hints

```
1. Pre-processor
```

```
• Create new project file - File → New Project → Select folders in turn →
Save project with name
• Geometry → Create Point → specify values → Apply.
• Geometry → Create and modify curve → Select points → OK.
• Geometry → Right click “Parts” → Part = “Specify name” → OK.
• Blocking → OK.
• Mesh → Select geometry → Global Mesh Size → Compute Mesh icon→ OK.
• Output → Output Solver: Fluent V6 → apply
• Write Input → Save project → save as filename “.msh” → Done
• Main Menu → File → Export → Mesh...
```

```
2. Solution
```

```
• Start → Programs → Fluent Inc → FLUENT 6.0 →
• Main Menu > File > Read > Case...
• Main Menu > Display > Grid...
• Main Menu > Define > Models > Solver
• Main Menu > Define > Materials... Click Change/Create.
• Main Menu > Define > Operating Conditions... Operating Pressure.
• Main Menu > Define > Boundary Conditions...
```

```
• Main Menu > Solve > Controls > Solution...
• Main Menu > Solve > Initialize > Initialize... Set Initial Guess
• Main Menu > Solve > Monitors > Residual... Set Convergence Criteria
• Main Menu > File > Write > Case...
• Main Menu > Solve > Iterate... Iterate Until Convergence
• Main Menu > File > Write > Data...
```

```
3. General Post Processing:
```

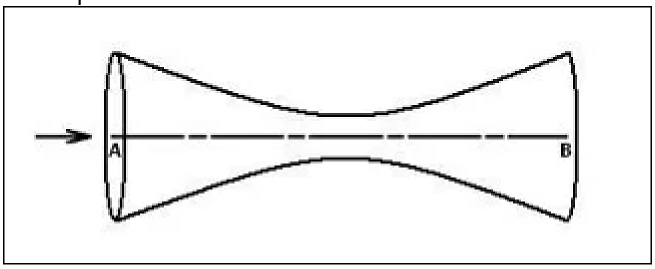
```
• General Post Processing
• Main Menu > Report > Reference Values...
• Main Menu > Plot > XY Plot... Save Plot
• Main Menu > Plot > contours... Save Plot
```

## 11.2 Compressible flow in a nozzle – under expanded case

Consider air flowing at high-speed through a convergent-divergent nozzle in Fig 11.2 having a circular cross-sectional area, A, that varies with axial distance from the throat, x, according to the formula

$$A = 0.1 + x^2; -0.5 < x < 0.5$$

where A is in square meters and x is in meters. The stagnation pressure $p_o$ at the inlet is 101,325 Pa. The stagnation temperature $T_o$ at the inlet is 400 K. The static pressure p at the exit is 3,738.9 Pa. Calculate the Mach number, pressure and temperature distribution in the nozzle using Ansys Fluent and compare the solution to quasi-1D nozzle flow results.




**11.2 convergent-divergent nozzle**

### Hints

```
1. Pre-processor
```

```
• Create new project file - File → New Project → Select folders in turn → Save project with name
• Geometry → Create Point → specify values → Apply.
• Geometry → Create and modify curve → Select points → OK.
• Geometry → Right click “Parts” → Part = “Specify name” → OK.
• Blocking → OK.
• Mesh → Select geometry → Global Mesh Size → Compute Mesh icon→ OK.
• Output → Output Solver: Fluent V6 → apply
• Write Input → Save project → save as filename “.msh” → Done
• Main Menu → File → Export → Mesh...
```

```
2. Solution
```

```
• Start → Programs → Fluent Inc → FLUENT 6.0 →
• Main Menu > File > Read > Case...
• Main Menu > Display > Grid...
• Main Menu > Define > Models > Solver
• Main Menu > Define > Models > Viscous
```

```
• Main Menu > Define > Materials... Click Change/Create.
• Main Menu > Define > Operating Conditions... Operating Pressure.
• Main Menu > Define > Boundary Conditions...
• Main Menu > Solve > Controls > Solution...
• Main Menu > Solve > Initialize > Initialize... Set Initial Guess
• Main Menu > Solve > Monitors > Residual... Set Convergence Criteria
• Main Menu > File > Write > Case...
• Main Menu > Solve > Iterate... Iterate Until Convergence
• Main Menu > File > Write > Data...
```

```
3. General Post Processing:
```

```
• General Post Processing
• Main Menu > Report > Reference Values...
• Main Menu > Plot > XY Plot... Save Plot
• Main Menu > Plot > contours... Save Plot
```

**Try**

1. Change the inlet Mach number to supersonic and analysis its behavior as diffuser.
2. Change the dimensions and repeat the above analysis for a diffuser.

# 12. Exercises on one dimensional wave equation using explicit method of lax

## 12.1 One dimensional wave equation using explicit method of lax Method

Write a program to solve one dimensional wave equation using explicit method of lax method - MATLAB.

**Hints**

```
1. Lax Method
```

```
• N=40; % No. of grid points
• Tmax=1; % time period
• alpha=1;  %given
• h=1; %given value of delX
```

```
2. Solution
```

```
delt=1;
maxt=Tmax/delt; %Number of time steps
c=alpha*delt/h;
u=zeros(N+1,maxt+1);
x=zeros(N+1,1);
% Initial condition
for i=1:N+1
  x(i)=(i-1)*h;
  u(i,1)=sin((2*pi*x(i)/40));
end
for k=1:maxt
  u0 = u(N,k);
  u(1,k+1)=(1-c)/2*(u(2,k)-u0)+(1+c)/2*(u(2,k));
  for i=2:N
    u(i,k+1) =(1-c)/2*(u(i+1,k)+(1+c))/2*(u(i+1,k));
  end
  uNp2 = u(2,k);
```

```
  u(N+1,k+1) =(1-c)/2*(uNp2-u(N,k))+(1+c)/2*(uNp2-2*u(N+1,k));
end
plot(x,u)
```

## 12.2 One dimensional heat conduction equation using explicit method.

Write a program to solve one dimensional heat conduction equation using explicit method – MATLAB.

### Hints

```
1.  Lax Method
```

```
% Parameters
L = 1;          % Length of the rod
T = 1;          % Total simulation time
alpha = 0.01;   % Thermal diffusivity
Nx = 50;        % Number of spatial grid points
Nt = 1000;      % Number of time steps

% Discretization
dx = L / (Nx - 1);
dt = T / Nt;
% Initial conditions
u = zeros(Nx, 1);
x = linspace(0, L, Nx)';
u(:, 1) = sin(pi * x / L);  % Example: Initial temperature distribution
```

```
2. Solution
```

```
% Explicit method time-stepping loop
for n = 1:Nt
for i = 2:Nx-1
u(i, n+1) = u(i, n) + alpha * dt / dx^2 * (u(i+1, n) - 2*u(i, n) + u(i-1,
n));
    end
end

% Plot the results
figure;
surf(linspace(0, T, Nt), x, u', 'EdgeColor', 'none');
title('One-dimensional Heat Conduction');
xlabel('Time');
ylabel('Position');
zlabel('Temperature');
```

### Try

1. Solve for one-dimensional heat conduction equation using explicit method with fixed boundary conditions.
2. Solve for one-dimensional heat conduction equation using implicit method with fixed boundary conditions.

# 13. Exercises on generating algebraic grids using MATLAB for CFD applications:

## 13.1 Generating algebraic grids using MATLAB for CFD applications:

Write a program to generating algebraic grids using MATLAB for Computational fluid dynamics applications.

### Hints

```
% Parameters
Lx = 1;          % Length of the grid in x-direction
Ly = 1;          % Length of the grid in y-direction
Nx = 50;         % Number of grid points in x-direction
Ny = 50;         % Number of grid points in y-direction

% Generate Cartesian grid
x_cartesian = linspace(0, Lx, Nx);
y_cartesian = linspace(0, Ly, Ny);
[x, y] = meshgrid(x_cartesian, y_cartesian);

% Plot
figure;
plot(x(:), y(:), 'o', 'LineWidth', 2);
title('Regular Cartesian Grid');
xlabel('X');
ylabel('Y');
axis equal;
grid on;
```

## 13.2 Generating Elliptic grids using MATLAB for CFD applications:

Write a program to generating elliptic grids using MATLAB for Computational fluid dynamics applications.

### Hints

```
% Parameters
Lx = 1;          % Length of the grid in x-direction
Ly = 1;          % Length of the grid in y-direction
Nx = 50;         % Number of grid points in x-direction
Ny = 50;         % Number of grid points in y-direction

% Generate elliptic grid
x_elliptic = linspace(0, Lx, Nx);
y_elliptic = linspace(0, Ly, Ny);

% Create meshgrid and transform to elliptic coordinates
[x, y] = meshgrid(x_elliptic, y_elliptic);
alpha = 1;  % Ellipticity factor
theta = linspace(0, 2*pi, Ny);
x_elliptic = x .* cos(alpha * theta);
y_elliptic = y .* sin(alpha * theta);

% Plot
figure;
plot(x_elliptic(:), y_elliptic(:), 'o', 'LineWidth', 2);
```

```
title('Elliptic Grid');
xlabel('X');
ylabel('Y');
axis equal;
grid on;
```

**Try**

1. Write a program to generating algebraic grids – Stretched grid using MATLAB for Computational fluid dynamics applications.
2. Write a program to generating elliptic grids – stretch grid using MATLAB for Computational fluid dynamics applications.

# 14. Exercises on linear convection equation (wave equation) by Lax Wendroff Scheme for CFD applications:

## 14.1 linear convection equation (wave equation) by Lax Wendroff Scheme:

Write a program to solve linear convection equation (wave equation) by Lax Wendroff Scheme using MATLAB for Computational fluid dynamics applications

**Hints**

```
% Parameters
L = 1;          % Length of the domain
T = 1;          % Total simulation time
Nx = 100;       % Number of spatial grid points
Nt = 200;       % Number of time steps
c = 1;          % Wave speed

% Discretization
dx = L / Nx;    % Spatial step
dt = T / Nt;    % Time step
x = linspace(0, L, Nx+1);   % Spatial grid
t = linspace(0, T, Nt+1);   % Time grid

% Initial condition
u0 = sin(2*pi*x);

% Lax-Wendroff scheme
u = zeros(Nx+1, Nt+1);
u(:,1) = u0;

for n = 1:Nt
    % Lax-Wendroff update
    u(2:end-1, n+1) = u(2:end-1, n) - 0.5*c*dt/dx*(u(3:end, n) - u(1:end-2,
n)) + ...
   0.5*(c*dt/dx)^2 * (u(3:end, n) - 2*u(2:end-1, n) + u(1:end-2, n));

    % Periodic boundary conditions
    u(1, n+1) = u(end-1, n+1);
    u(end, n+1) = u(2, n+1);
end

% Plotting
figure;
```

```
surf(x, t, u');
xlabel('Spatial Domain (x)');
ylabel('Time (t)');
zlabel('u(x, t)');
title('Lax-Wendroff Scheme for Linear Convection Equation');
```

## 14.2 Upwind scheme for solving the linear convection equation:

Write a program to solve linear convection equation (wave equation) by Upwind scheme Scheme using MATLAB for Computational fluid dynamics applications.

### Hints

```
% Parameters
L = 1;          % Length of the domain
T = 1;          % Total simulation time
Nx = 100;       % Number of spatial grid points
Nt = 200;       % Number of time steps
c = 1;          % Wave speed

% Discretization
dx = L / Nx;    % Spatial step
dt = T / Nt;    % Time step
x = linspace(0, L, Nx+1);   % Spatial grid
t = linspace(0, T, Nt+1);   % Time grid

% Initial condition
u0 = sin(2*pi*x);

% Upwind scheme
u = zeros(Nx+1, Nt+1);
u(:,1) = u0;

for n = 1:Nt
    % Upwind update
    u(2:end, n+1) = u(2:end, n) - c*dt/dx*(u(2:end, n) - u(1:end-1, n));

    % Periodic boundary conditions
    u(1, n+1) = u(end, n+1);
end

% Plotting
figure;
surf(x, t, u');
xlabel('Spatial Domain (x)');
ylabel('Time (t)');
zlabel('u(x, t)');
title('Upwind Scheme for Linear Convection Equation');
```

### Try

1. Write a program to solve Crank Nicholson technique using MATLAB for Computational fluid dynamics applications.
2. Write a program to solve Relaxation technique using MATLAB for Computational fluid dynamics applications.

# 15. Exercises on 1D advection equation using the forward-time finite difference method for CFD applications:

## 15.1 1D advection equation using the forward-time finite difference method:

Write a program to solve the 1D advection equation using the forward-time finite difference method for Computational fluid dynamics applications.

### Hints

```
% Parameters
L = 1;          % Length of the domain
T = 0.2;        % Total simulation time
Nx = 50;        % Number of spatial grid points
Nt = 100;       % Number of time steps
c = 1;          % Advection velocity

% Discretization
dx = L / Nx;    % Spatial step
dt = T / Nt;    % Time step
x = linspace(0, L, Nx+1);   % Spatial grid
t = linspace(0, T, Nt+1);   % Time grid

% Initial condition
u0 = sin(2*pi*x);

% Forward-time finite difference method for advection
u = zeros(Nx+1, Nt+1);
u(:,1) = u0;

for n = 1:Nt
    for i = 2:Nx
        u(i, n+1) = u(i, n) - c * dt / dx * (u(i, n) - u(i-1, n));
    end

    % Periodic boundary conditions
    u(1, n+1) = u(end-1, n+1);
end

% Plotting
figure;
surf(x, t, u');
xlabel('Spatial Domain (x)');
ylabel('Time (t)');
zlabel('u(x, t)');
title('Advection using Forward-Time Finite Difference Method');
```

## 15.2 1D advection equation using the backward-time finite difference method:

Write a program to solve the 1D advection equation using the backward -time finite difference method for Computational fluid dynamics applications

### Hints

```
%% Parameters
L = 1;          % Length of the domain
T = 0.2;        % Total simulation time
Nx = 50;        % Number of spatial grid points
Nt = 100;       % Number of time steps
c = 1;          % Advection velocity

% Discretization
dx = L / Nx;    % Spatial step
dt = T / Nt;    % Time step
x = linspace(0, L, Nx+1);   % Spatial grid
t = linspace(0, T, Nt+1);   % Time grid

% Initial condition
u0 = sin(2*pi*x);

% Backward-time finite difference method for advection
u = zeros(Nx+1, Nt+1);
u(:,1) = u0;

% Coefficient matrix for backward-time scheme
A = spdiags(ones(Nx-1,1)*[-c*dt/dx, 1 + c*dt/dx], [-1, 0], Nx-1, Nx-1);

for n = 1:Nt
    % Update using implicit backward-time scheme
    u(2:end, n+1) = A \ u(2:end, n);

    % Periodic boundary conditions
    u(1, n+1) = u(end-1, n+1);
end

% Plotting
figure;
surf(x, t, u');
xlabel('Spatial Domain (x)');
ylabel('Time (t)');
zlabel('u(x, t)');
title('Advection using Backward-Time Finite Difference Method');
```

### Try

1. Write a program to solve the 1D advection equation using the central finite difference method for Computational fluid dynamics applications.
2. Write a program to solve the 1D advection equation using forward finite difference method for Computational fluid dynamics applications.

## V. TEXTBOOKS:


1. R.K Bansal, Strength of Materials., Laxmi publications, 5th edition, 2012.
2. T. H. G. Megson, Aircraft Structures for Engineering Students., Butterworth-Heinemann Ltd, 5th edition, 2012.
3. Gere, Timoshenko, Mechanics of Materials., McGraw Hill, 3rd edition,1993.


## VI. REFERENCE BOOKS:


1. Peery, D.J. and Azar, J.J., “Aircraft Structures”, McGraw-Hill, 2nd edition,1982, ISBN0-07-049196-8.
2. Bruhn.E.H, “Analysis and Design of Flight Vehicles Structures”, Tri-state Off-set Company, USA, 1965.

3. Lakshmi Narasaiah, G., “Aircraft Structures”, BS Publications, 2010.

## VII. ELECTRONICS RESOURCES:

1. https://akanksha.iare.ac.in/index?route=course/details&course_id=1645.

## VIII. MATERIALS ONLINE

1. Course Template
2. Laboratory Manual