

# **ADVANCED COMPUTATIONAL AERODYNAMICS LABORATORY**

## **LAB MANUAL**

**Course Code : BAEB09**  
**Regulation : R18**  
**Semester : I**  
**Branch : AE**  
**Academic Year : 2019-2020**

### **PREPARED BY**

**Ms.D.Anitha**  
**Assistant Professor**



**Department of Aeronautical Engineering**

**INSTITUTE OF AERONAUTICAL ENGINEERING**

**(AUTONOMOUS)**

**Dundigal, Hyderabad-500043**



# **INSTITUTE OF AERONAUTICAL ENGINEERING**

**(AUTONOMOUS)**

**Dundigal, Hyderabad-500043**

## **DEPARTMENT OF AERONAUTICAL ENGINEERING**

### **VISION AND MISSION OF THE DEPARTMENT**

#### **VISION**

To build a strong community of dedicated graduates with expertise in the field of aeronautical science and engineering suitable for industrial needs having a sense of responsibility, ethics and ready to participate in aerospace activities of national and global interest.

#### **MISSION**

To actively participate in the technological, economic and social development of the nation through academic and professional contributions to aerospace and aviation areas, fostering academic excellence and scholarly learning among students of aeronautical engineering.



# **INSTITUTE OF AERONAUTICAL ENGINEERING**

(AUTONOMOUS)

Dundigal, Hyderabad - 500 043

## *Certificate*

*This is to certify that it is a bonafied record of practical work done by Sri/Kum. \_\_\_\_\_ bearing the RollNo. \_\_\_\_\_ of \_\_\_\_\_ class \_\_\_\_\_ Master in the Advanced Computational Aerodynamics laboratory during the academic year \_\_\_\_\_ under our supervision.*

**Head of the Department**

**Lecture In-Charge**

**External Examiner**

**Internal Examiner**



# INSTITUTE OF AERONAUTICAL ENGINEERING

(AUTONOMOUS)  
Dundigal, Hyderabad-500043

## DEPARTMENT OF AERONAUTICAL ENGINEERING

PROGRAM OUTCOMES	
<b>PO1</b>	Identify, formulate, and solve complex aerospace engineering problems by applying advanced principles of engineering.
<b>PO2</b>	Apply aerospace engineering design to produce solutions that meet specified needs with frontier technologies
<b>PO3</b>	Formulate and solve complex engineering problems related to aerospace materials, propulsion, aerodynamics, structures, avionics, stability and control.
<b>PO4</b>	Write and present a substantial technical report / document.
<b>PO5</b>	Independently carry out research / investigation and development work to solve practical problems
<b>PO6</b>	Function effectively on a team whose members together provide leadership, create a collaborative and inclusive environment, establish goals, plan tasks, and meet objectives in aerospace engineering.
<b>PO7</b>	Recognize ethical and professional responsibilities in aerospace engineering situations and make informed judgments, which must consider the impact of engineering solutions in global, economic, environmental, and societal contexts.

**ADVANCED COMPUTATIONAL AERODYNAMICS LABORATORY**  
**SYLLABUS**

<b>S.NO</b>	<b>LIST OF EXPERIMENTS</b>	<b>PAGE NO</b>
1	Introduction	
2	Introduction to ICEM CFD	
3	Introduction to fluent	
4	Shock wave boundary layer intersection over a flat plate	
5	Subsonic flow in a convergent divergent nozzle	
6	Circulation of the lift over a circular cylinder	
7	Pressure distribution over a symmetric aerofoil	
8	Pressure distribution over a cambered aerofoil	
9	Shock wave supersonic flow over wedge	
10	Shock wave around a cone	
11	Flow through diffuser	
12	Flow through supersonic intake	

## **ADVANCED COMPUTATIONAL AERODYNAMICS LABORATORY**

### **OBJECTIVE:**

The objective of this lab is to teach students and give knowledge about the computational techniques of aerodynamic problems with the flow visualization and usage of the flow properties for different geometries with different boundary conditions. This lab also enables the students to learn the tools of ICEM CFD and Fluent This laboratory also enhances experimental skills to the students to assess the simulation and visualization of flow around the aerodynamic bodies under different flow streams .

### **OUTCOMES:**

**The course should enable the students to:**

- I. Implement the computational fluid dynamic and computational aerodynamic fundamentals by using advanced solvers.
- II. Understand the flow properties of flat plate, nozzle and cylinder to demonstrate Reynolds number.
- III. Differentiate the flow properties around symmetrical and cambered airfoil.
- IV. Analyse the coefficient of pressure, lift, drag and moment for different bodies for different flow conditions.
- V. Visualize the flow around the different bodies under supersonic conditions.

### ATTAINMENT OF PROGRAM OUTCOMES

<b>Experiment No</b>	<b>Program Outcomes Attained</b>
1	PO1
2	PO1,PO2
3	PO1,PO2,PO5
4	PO2, PO5
5	PO2, PO5
6	PO2, PO5
7	PO2, PO5
8	PO2, PO5
9	PO2, PO5
10	PO2, PO5
11	PO2, PO5
12	PO2, PO5

# EXPERIMENT-1

## INTRODUCTION

**Computational fluid dynamics (CFD)** is a branch of fluid mechanics that uses numerical analysis and data structures to analyze and solve problems that involve fluid flows. Computers are used to perform the calculations required to simulate the free-stream flow of the fluid, and the interaction of the fluid (liquids and gases) with surfaces defined by boundary conditions. With high-speed supercomputers, better solutions can be achieved, and are often required to solve the largest and most complex problems. Ongoing research yields software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. Initial validation of such software is typically performed using experimental apparatus such as wind tunnels.

In addition, previously performed analytical or empirical analysis of a particular problem can be used for comparison. A final validation is often performed using full-scale testing, such as flight tests.

CFD is applied to a wide range of research and engineering problems in many fields of study and industries, including aerodynamics and aerospace analysis, weather simulation, natural science and environmental engineering, industrial system design and analysis, biological engineering and fluid flows, and engine and combustion analysis.

### Methodology

In all of these approaches the same basic procedure is followed.

During preprocessing

- The geometry and physical bounds of the problem can be defined using computer aided design (CAD). From there, data can be suitably processed (cleaned-up) and the fluid volume (or fluid domain) is extracted.
- The volume occupied by the fluid is divided into discrete cells (the mesh). The mesh may be uniform or non-uniform, structured or unstructured, consisting of a combination of hexahedral, tetrahedral, prismatic, pyramidal or polyhedral elements.
- The physical modeling is defined – for example, the equations of fluid motion + enthalpy + radiation + species conservation
- Boundary conditions are defined. This involves specifying the fluid behavior and properties at all bounding surfaces of the fluid domain. For transient problems, the initial conditions are also defined.
- The simulation is started and the equations are solved iteratively as a steady-state or transient.
- Finally a postprocessor is used for the analysis and visualization of the resulting solution.

### Concept of Computational Fluid Dynamics

Computational Fluid Dynamics (CFD) is the simulation of fluids engineering systems using modeling (mathematical physical problem formulation) and numerical methods (discretization methods, solvers, numerical parameters, and grid generations, etc.).

### Process of Computational Fluid Dynamics

Firstly, we have a fluid problem. To solve this problem, we should know the physical properties of fluid by using Fluid Mechanics. Then we can use mathematical equations to describe these physical properties. This is Navier-



Stokes Equation and it is the governing equation of CFD. As the Navier-Stokes Equation is analytical, human can understand it and solve them on a piece of paper. But if we want to solve this equation by computer, we have to translate it to the discretized form. The translators are numerical discretization methods, such as Finite Difference, Finite Element, Finite Volume methods. Consequently, we also need to divide our whole problem domain into many small parts because our discretization is based on them. Then, we can write programs to solve them.

The typical languages are Fortran and C. Normally the programs are run on workstations or supercomputers. At the end, we can get our simulation results. We can compare and analyze the simulation results with experiments and the real problem. If the results are not sufficient to solve the problem, we have to repeat the process until find satisfied solution. This is the process of CFD.

### **Importance of Computational Fluid Dynamics**

There are three methods in study of Fluid: theory analysis, experiment and simulation (CFD). As a new method, CFD has many advantages compared to experiments.

- Simulation (CFD) Experiment
- Cost Cheap Expensive
- Time Short Long
- Scale Any Small/Middle
- Information All Measured Point
- Repeatable Yes Some
- Safety Yes Some Dangerous

### **Application of Computational Fluid Dynamics**

As CFD has so many advantages, it is already generally used in industry such as

- Aerospace,
- Automotive,
- Biomedicine,
- Chemical processing,
- Heat ventilation air condition,
- Hydraulics,
- Power generation,
- Sports and
- Marine etc.

### **Viva questions:**

1. What is computational Aerodynamics?
2. What is topology?
3. List the properties of fluid dynamics.
4. Define discretization.
5. What are governing equations?

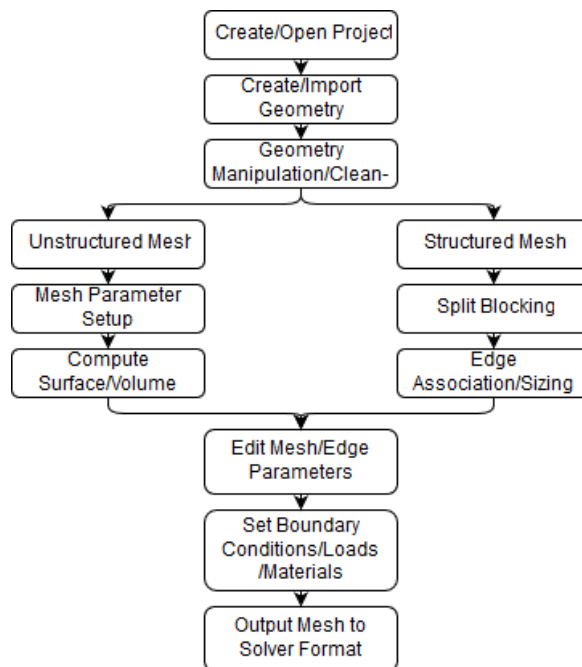
## EXPERIMENT - 2

### INTRODUCTION TO ICEM CFD

ANSYS ICEM CFD is a popular proprietary software package which provides advanced geometry/mesh generation as well as mesh diagnostics and repair functions useful for in-depth analysis. Its design is centered around aerospace, automotive and electrical engineering applications with a specific focus on computational fluid dynamics and structural analysis. The ability to accurately create computational grids about geometrically complex configurations is becoming increasingly important in the analysis world.

ANSYS ICEM CFD offers mesh generation with the capacity to compute meshes with various different structures depending on the user's requirements. It is powerful and highly manipulative software which allows the user to generate grids of high resolution. This is a requirement as mesh generation is an inherently geometry dependent problem meaning there is no singular meshing method which can be used for every problem. ICEM CFD allows the following different types of grid structures to be created

Multi-block structured meshes, Unstructured meshes, Hybrid meshes. A simplified overview of the meshing methodology provides an indication of how to begin the meshing procedure with the different stages shown.



#### Geometry Modelling

ICEM CFD allows for its geometry to either be made using its own geometry topology package or to import geometry via external CAD software. For simple geometries the former tends to be used and for more complex geometries the latter is often used. Regardless of this, the geometry should be checked using the geometry analysis to ensure the model contains a closed volume, meaning there are no holes or gaps within the geometry, so that further down the line, no negative volume elements are present.

Negative volume elements are not permitted in external solvers. These may be present due to differences in geometrical and meshing tolerances. There is hence an emphasis in ICEM CFD to create a mesh that has a

'water-tight' geometry. It means if there is a source of water inside a region, the water should be contained and not leak out of the BODY.

Apart from the regular points, curves, surface creation and editing tools, ICEM CFD especially has the capability to do BUILD TOPOLOGY which removes unwanted surfaces and then you can view if there are any 'holes' in the region of interest for meshing. Existence of holes would mean that the algorithm which generates the mesh would cause the mesh to 'leak out' of the domain. Holes are typically identified through the colour of the curves. The following is the colour coding in ICEM CFD, after the BUILD TOPOLOGY option has been implemented:

YELLOW: curve attached to a single surface

RED: curve shared by two surfaces

BLUE: curve shared by more than two surfaces.

Green: curve not attached to any surface

An analysis post building the topology is then required, whereby if it is clear a curve is supposed to share two surfaces yet is being displayed as yellow, a hole exists and it is likely there will be meshing problems.

Geometrical entities which may include points, curves and surfaces must also be associated to a given part. Each part can then be controlled for meshing, visualization or various other purposes and is stored within the aforementioned tetin file.

The unstructured mesh generation procedure will now be considered:

1. Create/Import geometry
2. Repair geometry ensuring a closed volume
3. Determine global meshing parameter
4. Specify part mesh setup
5. Specify curves and surface mesh size
6. Compute mesh

The structured mesh generation procedure will now be considered:

1. Create/Import geometry.
2. Initialize blocking with respect to geometry dimension
3. Generate block structure using the split, merge, Ogrid definition.
4. Associate vertices to points, edges to curve and block faces to geometry face.
5. Check block structure quality to ensure the block model meets specified quality threshold.
6. Determine edge meshing parameters and using spacing 1 or spacing 2 for increasing mesh density in specific zone.
7. Using pre-mesh to update mesh
8. Check the cell quality of the mesh once its generated.
9. Convert structure mesh to substructure mesh by right click on the re-compute mesh
10. Write output files to desired solver like ANSYS fluent or Star CCM

### **Exporting Mesh**

ICEM CFD allows the user to export their mesh into various different formats for compatibility with other external solvers. The meshing topology, associated parts, boundary conditions and loads should all be predefined before this stage. Some of the possible supported output solvers are:

- ANSYS CFX
- ANSYS Fluent
- CGNS
- Plot3D
- STAR-CCM

### **Viva questions:**

1. What is topology?
2. What is geometry?
3. Differentiate the structured and unstructured mesh.
4. What is full form ICEM CFD?

## EXPERIMENT – 3

### INTRODUCTION TO ANSYS FLUENT

ANSYS Fluent is a state-of-the-art computer program for modeling fluid flow, heat transfer, and chemical reactions in complex geometries. ANSYS Fluent is written in the C computer language and makes full use of the flexibility and power offered by the language. Consequently, true dynamic memory allocation, efficient data structures, and flexible solver control are all possible. In addition, ANSYS Fluent uses a client/server architecture, which enables it to run as separate simultaneous processes on client desktop workstations and powerful computer servers. This architecture allows for efficient execution, interactive control, and complete flexibility between different types of machines or operating systems.

ANSYS Fluent provides complete mesh flexibility, including the ability to solve your flow problems using unstructured meshes that can be generated about complex geometries with relative ease. Supported mesh types include 2D triangular/quadrilateral, 3D tetrahedral/hexahedral/pyramid/wedge/polyhedral, and mixed (hybrid) meshes. ANSYS Fluent also enables you to refine or coarsen your mesh based on the flow solution.

You can read your mesh into ANSYS Fluent, or, for 3D geometries, create your mesh using the meshing mode of Fluent (see the Fluent Meshing User's Guide for further details). All remaining operations are performed within the solution mode of Fluent, including setting boundary conditions, defining fluid properties, executing the solution, refining the mesh, and post processing and viewing the results.

The ANSYS Fluent serial solver manages file input and output, data storage, and flow field calculations using a single solver process on a single computer. ANSYS Fluent also uses a utility called cortex that manages ANSYS Fluent's user interface and basic graphical functions. ANSYS Fluent's parallel solver enables you to compute a solution using multiple processes that may be executing on the same computer, or on different computers in a network.

Parallel processing in ANSYS Fluent involves an interaction between ANSYS Fluent, a host process, and a set of compute-node processes. ANSYS Fluent interacts with the host process and the collection of compute nodes using the cortex user interface utility.

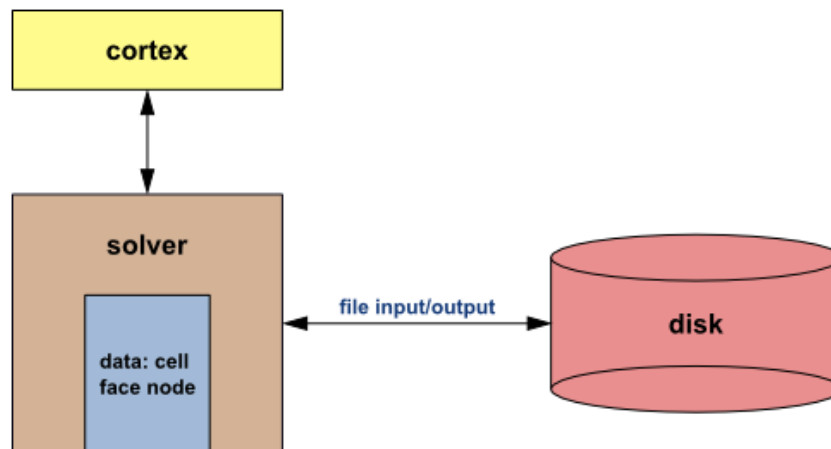


Figure 3.1: Parallel ANSYS Fluent Architecture

For more information about ANSYS Fluent's parallel processing capabilities, message passing interfaces (MPI), and so on, refer to Parallel Processing in the User's Guide.

All functions required to compute a solution and display the results are accessible in ANSYS Fluent through an interactive interface.

**Viva questions:**

1. What are the solver software's?
2. Differentiate the triangular and tetrahedral elements
3. Define cell and node.

## EXPERIMENT – 4

### SHOCK WAVE BOUNDARY LAYER INTERSECTION OVER A FLAT PLATE

**Aim:** To visualize the shock wave boundary layer intersection over a flat plate and plot the velocity profile at  $x = 1\text{m}$ . Compare the accuracy of your results from FLUENT with empirical correlations.

**Software Used:** ICEM CFD and FLUENT

#### Preliminary Analysis

We expect the turbulent boundary layer to grow along the plate. As the boundary layer grows in thickness, the rate of heat transfer ( $q''$ ) and thus the heat transfer coefficient ( $h$ ) will decrease.

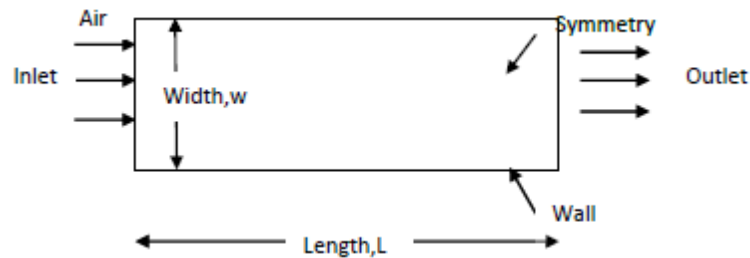


Fig 4.1: Boundary layer

We will compare the numerical results with experimentally-derived heat transfer correlations. We will create the geometry and mesh in ICEM CFD, read the mesh into FLUENT, and solve the flow problem.

- Create or import Geometry
- Block the geometry
- Associate entities to the geometry.
- Apply mesh parameters.
- Generate the initial mesh.
- Export mesh

#### Start ICEM CFD

##### Create Geometry in ICEM CFD

- 1) Close any previous files, have blank screen in ICEM.
- 2) Create new project file for the pipe in ICEM:
  - a) File, New Project.
  - b) Select folders in turn: Desktop, CFX Training Files – Copy, Flat plate.
  - c) Save project with name “Flat plate” in folder “Flat plate”.

3) We are going to create some points now:

- a) Select Geometry tab at top of screen.
- b) Select “Create Point” option.
- c) In the bottom left are various options for creating the point. Select the second “XYZ” option.
- d) Input the numbers X=0, Y=0, Z=0.
- e) Press “Apply” to create a point on screen

### Create points

x	y
0	0
1	0
1	1
0	1

Explicit location > specify x and y location and click ok  
Similarly create 4 points

### Create curves

- a) Under “Geometry” tab, select “Create and modify curve”.
  - b) Under “Create and modify curve” option panel on bottom left, select 3rd option “
- Select two points on the screen and select ok Create remaining curves

### Create surface

Simple surface – from 2-4 curves, to check surface click on solid frame

### Label the curves

Give names to the INLET, OUTLET and WALL surfaces:

- a) Under “Geometry” tree, highlight only “Surfaces”.
- b) Right click “Parts”, Create Part.
- c) Part = “INLET”.
- d) Entities: click arrow icon, left click surface (near surface, around origin) to highlight it, middle click to save selection, right click to de-select mode.
- e) Note that the INLET part has been created, and can be turned on and off independently using the tree view.
- f) Part = “OUTLET”, repeat step to identify an outlet at the far end.
- g) Part = “WALL”, repeat step to identify the wall.
- h) Notes: “GEOM” now contains only 0D and 1D elements; points and lines.

### Block the Geometry

The geometry and part information has already been defined for this tutorial. You will create the initial block in this step.

1. Create the initial block.

Blocking > Create Block > Initialize Blocks

- a. Enter FLUID in the Part field.
- b. Select 2D Planar in the Type drop-down list and click apply.



## Associate Entities to the Geometry

You will associate the edges of the blocking to the curves of the CAD geometry in this step. First select the edges and then the curves to which you want to associate them.

1. Associate the inlet, the left-most end of the large flat plate.
  - a. Ensure that Project Vertices is disabled.
  - b. Click (Select edge(s)) and select edge 13–41. Click the middle-mouse button to accept the selection.
  - c. Click (Select compcurve(s)) and select CURVES/1. Click the middle-mouse button to accept the selection. Click Apply.

The associated edge will be colored green.

## Generate the Initial Mesh

Blocking > Pre-Mesh Params > Update Sizes

Retain the selection of Update All in the Method list.

## Blocking Pre-Mesh

The Mesh dialog will appear, asking if you want to recompute the mesh. Click Yes in the Mesh dialog to compute the initial mesh. Disable Vertices and Edges.

## Verify and Save the Mesh and Blocking

1. Convert the mesh to unstructured format.

Blocking Pre-Mesh Convert to Unstruct Mesh
  2. Save the blocking file (2D-Flat plate -geometry-final.blk).

File > Blocking > Save Blocking As...

This block file can be loaded in a future session (File > Blocking > Open Blocking...) for additional modification or to mesh a similar geometry. Save each blocking to a separate file instead of overwriting a previous one. In more complex models, you may have to back track and load a previous blocking.
  3. Save the project file (2D-flat plate-geometry-final.prj).

File > Save Project As...

This will save all the files—tetin, blocking, and unstructured mesh.
  4. Exit the current session.

File > Exit
- Save your ICEM CFD file in your working directory.

## Setting Up the CFD Simulation in ANSYS FLUENT

Now that you have created a computational mesh for the elbow geometry, you can proceed to setting up a CFD analysis using **ANSYS FLUENT**.

1. Start **ANSYS FLUENT**.

In the **ANSYS Workbench Project Schematic**, double-click the **Setup** cell in the elbow fluid flow analysis system. You can also right-click on the **Setup** cell to display the context menu where you can select the **Edit** option.

When **ANSYS FLUENT** is first started, **FLUENT Launcher** is displayed, allowing you to view and/or set certain **ANSYS FLUENT** start-up options.

*ANSYS FLUENT Launcher allows you to decide which version of ANSYS FLUENT you will use, based on your geometry and on your processing capabilities.*

Ensure that the proper options are enabled.

Note that the **Dimension** setting is already filled in and cannot be changed, since **ANSYS**

**FLUENT** automatically sets it based on the mesh or geometry for the current system.

- i. Make sure that **Serial** from the **Processing Options** list is enabled.
- ii. Make sure that the **Display Mesh After Reading**, **Embed Graphics Windows**, and **Workbench Color Scheme** options are enabled.
- iii. Make sure that the **Double-Precision** option is disabled.

(b) Click **OK** to launch **ANSYS FLUENT**.

2. Set some general settings for the CFD analysis.

*Select **General** in the navigation pane to perform the mesh-related activities and to choose a solver.*

### **General**

(a) Change the units for length.

Since we want to specify and view values based on a unit of length in inches from within **ANSYS FLUENT**, change the units of length within **ANSYS FLUENT** from meters (the default) to inches.

#### **General → Units...**

This displays the **Set Units** dialog box.

- i. Select **length** in the **Quantities** list.
- ii. Select **in** in the **Units** list.
- iii. Close the dialog box.

*Now, all subsequent inputs that require a value based on a unit of length can be specified in inches rather than meters.*

(b) Check the mesh.

#### **General → Check**

*ANSYS FLUENT will report the results of the mesh check in the console.*

3. Set up your models for the CFD simulation.

### **Models**

(a) Enable heat transfer by activating the energy equation.

#### **Models → Energy → Edit...**

*You can also double-click a list item in order to open the corresponding dialog box.*

- i. Enable the **Energy Equation** option.
  - ii. Click **OK** to close the **Energy** dialog box.
- (b) Enable the **k-ε** turbulence model.

#### **Models → Viscous → Edit...**

- i. Select **k-epsilon** from the **Model** list.

*The Viscous Model dialog box will expand.*

- ii. Select **Realizable** from the **k-epsilon Model** list.
  - iii. Click **OK** to accept the model and close the **Viscous Model** dialog box.
4. Set up your materials for the CFD simulation.

#### **Materials**

- (a) Create a new material called **air** using the **Create/Edit Materials** dialog box.
  - (b) **Materials** → **Fluid** → **Create/Edit...**
    - i. Enter **air** for **Name**.
    - ii. specify the properties
    - iii. Click **Change/Create**.
5. Set up the cell zone conditions for the CFD simulation.

#### **Cell Zone Conditions**

- (a) Set the cell zone conditions for the fluid zone.
  - i. Select **fluid** in the **Zones** list in the **Cell Zone Conditions** task page, then click the **Edit...** button to open the **Fluid** dialog box.

*You can also double-click a list item in order to open the corresponding dialog box.*

- ii. In the **Fluid** dialog box, select **water** from the **Material Name** drop-down list.
  - iii. Click **OK** to close the **Fluid** dialog box.
6. Set up the boundary conditions for the CFD analysis.

#### **Boundary Conditions**

- (a) Set the boundary conditions at the cold inlet (**velocity-inlet-large**).

#### **Boundary Conditions** → **velocity-inlet-large** → **Edit...**

- i. Select **Components** from the **Velocity Specification Method** drop-down list.

*The Velocity Inlet dialog box will expand.*
- ii. Enter 0.4 m/s for **X-Velocity**.
- iii. Retain the default value of 0 m/s for both **Y-Velocity** and **Z-Velocity**.

7. Set up solution parameters for the CFD simulation.

*In the steps that follow, you will set up and run the calculation using the task pages listed under the **Solution** heading in the navigation pane.*

Change the convergence criteria for the continuity equation residual.

#### **Monitors** → **Residuals** → **Edit...**

- i. Make sure that **Plot** is enabled in the **Options** group box.
  - ii. Enter 1e-05 for the **Absolute Criteria** of **continuity**, as shown in the **Residual Monitor** dialog box.
  - iii. Click **OK** to close the **Residual Monitors** dialog box.
- Initialize the flow field, using the boundary conditions settings at the cold inlet (**velocity-inlet-large**) as a starting point.

### **Solution Initialization**

- i. Select **velocity-inlet-large** from the **Compute From** drop-down list.
- ii. Enter 1.2 m/s for **Y Velocity** in the **Initial Values** group box.

*Note: While an initial X Velocity is an appropriate guess for the horizontal section, the addition of a Y Velocity component will give rise to a better initial guess throughout the entire elbow.*

- iii. Click **Initialize**.

Check to see if the case conforms to best practices.

**Run Calculation** → **Check Case**

8. Calculate a solution.  
Start the calculation by requesting 250 iterations.

### **Run Calculation**

- i. Enter 250 for **Number of Iterations**.
- ii. Click **Calculate**.

*As the calculation progresses, the residuals will be plotted in the graphics window*

9. View the files generated by **ANSYS Workbench**.

### **Result:**

### **Viva questions:**

1. What is boundary layer thickness?
2. Calculate the displacement thickness and momentum thickness at a given set of points?
3. Was the transition point identified?

## **EXPERIMENT – 5**

### **SUBSONIC FLOW IN A CONVERGENT DIVERGENT NOZZLE**

**AIM:** To visualize the flow through convergent divergent nozzle and calculate the flow properties at different velocities.

**Software Used:**ICEM CFD and FLUENT

We will create the geometry and mesh in ICEM CFD, read the mesh into FLUENT, and solve the flow problem

- Create or import Geometry
- Block the geometry
- Associate entities to the geometry.
- Apply mesh parameters.
- Generate the initial mesh.
- Export mesh

#### **Start ICEM CFD**

##### **Create Geometry in ICEM CFD**

- 1) Close any previous files, have blank screen in ICEM.
- 2) Create new project file for the pipe in ICEM:
  - a) File, New Project.
  - b) Select folders in turn: Desktop, CFX Training Files – Copy, Flat plate.
  - c) Save project with name “Flat plate” in folder “Flat plate”.
- 3) We are going to create some points now:
  - a) Select Geometry tab at top of screen.
  - b) Select “Create Point” option.
  - c) In the bottom left are various options for creating the point. Select the second “XYZ” option.
  - d) Input the numbers  $X=0$ ,  $Y=0$ ,  $Z=0$ .
  - e) Press “Apply” to create a point on screen
  - f) Note - If the point does not appear, then check that points are displayed in the tree, and use the “Fit Window” option in the top left to zoom the screen correctly.

##### **Create points**

Explicit location > specify x and y location and click ok  
Similarly create 4 points

##### **Create curves**

- a) Under “Geometry” tab, select “Create and modify curve”.
- b) Under “Create and modify curve” option panel on bottom left, select 3rd option “

Select two points on the screen and select ok Create remaining curves

### **Create surface**

Simple surface – from 2-4 curves  
To check surface click on solid frame

### **Label the curves**

Give names to the INLET, OUTLET and WALL surfaces:

- a) Under “Geometry” tree, highlight only “Surfaces”.
- b) Right click “Parts”, Create Part.
- c) Part = “INLET”.
- d) Entities: click arrow icon, left click surface (near surface, around origin) to highlight it, middle click to save selection, right click to de-select mode.
- e) Note that the INLET part has been created, and can be turned on and off independently using the tree view.
- f) Part = “OUTLET”, repeat step to identify an outlet at the far end.
- g) Part = “WALL”, repeat step to identify the wall.
- h) Notes: “GEOM” now contains only 0D and 1D elements; points and lines.

### **Block the Geometry**

The geometry and part information has already been defined for this tutorial. You will create the initial blockin this step.

1. Create the initial block.
  - Blocking > Create Block > Initialize Blocks
  - a. Enter FLUID in the Part field.
  - b. Select 2D Planar in the Type drop-down list.
  - c. Click Apply.

### **Associate Entities to the Geometry**

You will associate the edges of the blocking to the curves of the CAD geometry in this step. First select the edges and then the curves to which you want to associate them.

1. Associate the inlet, the left-most end of the large flat plate.
  - a. Ensure that Project Vertices is disabled.
  - b. Click (Select edge(s)) and select edge 13–41. Click the middle-mouse button to accept the selection.
  - c. Click (Select compcurve(s)) and select CURVES/1. Click the middle-mouse button to accept the selection.
  - d. Click Apply.

The associated edge will be colored green.

### **Generate the Initial Mesh**

Blocking > Pre-Mesh Params > Update Sizes

Retain the selection of Update All in the Method list.

### **Blocking Pre-Mesh**

The Mesh dialog will appear, asking if you want to recompute the mesh.

Click Yes in the Mesh dialog to compute the initial mesh.

Disable Vertices and Edges.

### **Verify and Save the Mesh and Blocking**

1. Convert the mesh to unstructured format.

Blocking Pre-Mesh Convert to Unstruct Mesh

2. Save the blocking file (2D-Flat plate -geometry-final.blk).

File > Blocking > Save Blocking As...

This block file can be loaded in a future session (File > Blocking > Open Blocking...) for additional modification or to mesh a similar geometry. Save each blocking to a separate file instead of overwriting a previous one. In more complex models, you may have to back track and load a previous blocking.

3. Save the project file (2D-flat plate-geometry-final.prj).

File > Save Project As...

This will save all the files—tetin, blocking, and unstructured mesh.

4. Exit the current session.

File > Exit

Save your ICEM CFD file in your working directory.

### **Setting Up the CFD Simulation in ANSYS FLUENT**

Now that you have created a computational mesh for the elbow geometry, you can proceed to setting up a CFD analysis using **ANSYS FLUENT**.

1. Start **ANSYS FLUENT**.

In the **ANSYS Workbench Project Schematic**, double-click the **Setup** cell in the elbow fluid flow analysis system. You can also right-click on the **Setup** cell to display the context menu where you can select the **Edit** option.

When **ANSYS FLUENT** is first started, **FLUENT Launcher** is displayed, allowing you to view and/or set certain **ANSYS FLUENT** start-up options.

Ensure that the proper options are enabled.

Note that the **Dimension** setting is already filled in and cannot be changed, since **ANSYS FLUENT** automatically sets it based on the mesh or geometry for the current system.

- i. Make sure that **Serial** from the **Processing Options** list is enabled.
  - ii. Make sure that the **Display Mesh After Reading**, **Embed Graphics Windows**, and **Workbench Color Scheme** options are enabled.
  - iii. Make sure that the **Double-Precision** option is disabled.
- (b) Click **OK** to launch **ANSYS FLUENT**.

2. Set some general settings for the CFD analysis.

Select **General** in the navigation pane to perform the mesh-related activities and to choose a solver.

## General

- (a) Change the units for length.

Since we want to specify and view values based on a unit of length in inches from within **ANSYS FLUENT**, change the units of length within **ANSYS FLUENT** from meters (the default) to inches.

### General → Units...

This displays the **Set Units** dialog box.

- i. Select **length** in the **Quantities** list.
- ii. Select **in** in the **Units** list.
- iii. Close the dialog box.

*Now, all subsequent inputs that require a value based on a unit of length can be specified in inches rather than meters.*

- (b) Check the mesh.

### General → Check

*ANSYS FLUENT will report the results of the mesh check in the console.*

3. Set up your models for the CFD simulation.

## Models

- (a) Enable heat transfer by activating the energy equation.

### Models → Energy → Edit...

*You can also double-click a list item in order to open the corresponding dialog box.*

- i. Enable the **Energy Equation** option.
- ii. Click **OK** to close the **Energy** dialog box.

- (b) Enable the **k- $\epsilon$**  turbulence model.

### Models → Viscous → Edit...

- i. Select **k-epsilon** from the **Model** list.

*The **Viscous Model** dialog box will expand.*

- ii. Select **Realizable** from the **k-epsilon Model** list.
- iii. Click **OK** to accept the model and close the **Viscous Model** dialog box.



4. Set up your materials for the CFD simulation.

### Materials

(c) Create a new material called **air** using the **Create/Edit Materials** dialog box.

(d) **Materials** → **Fluid** → **Create/Edit...**

i. Enter **air** for **Name**.

ii. specify the properties

iii. Click **Change/Create**.

5. Set up the cell zone conditions for the CFD simulation.

### Cell Zone Conditions

(a) Set the cell zone conditions for the fluid zone.

i. Select **fluid** in the **Zones** list in the **Cell Zone Conditions** task page, then click the **Edit...** button to open the **Fluid** dialog box.

*You can also double-click a list item in order to open the corresponding dialog box.*

ii. In the **Fluid** dialog box, select **water** from the **Material Name** drop-down list.

iii. Click **OK** to close the **Fluid** dialog box.

6. Set up the boundary conditions for the CFD analysis.

### Boundary Conditions

(a) Set the boundary conditions at the cold inlet (**velocity-inlet-large**).

**Boundary Conditions** → **velocity-inlet-large** → **Edit...**

i. Select **Components** from the **Velocity Specification Method** drop-down list.

*The **Velocity Inlet** dialog box will expand.*

ii. Enter 0.4 m/s for **X-Velocity**.

iii. Retain the default value of 0 m/s for both **Y-Velocity** and **Z-Velocity**.

7. Set up solution parameters for the CFD simulation.

*In the steps that follow, you will set up and run the calculation using the task pages listed under the **Solution** heading in the navigation pane.*

Change the convergence criteria for the continuity equation residual.

## Monitors → Residuals → Edit...

- i. Make sure that **Plot** is enabled in the **Options** group box.
- ii. Enter 1e-05 for the **Absolute Criteria** of **continuity**, as shown in the **Residual Monitor** dialog box.
- iii. Click **OK** to close the **Residual Monitors** dialog box.

*Note:* By default, all variables will be monitored and checked by **ANSYS FLUENT** as a means to determine the convergence of the solution.

Initialize the flow field, using the boundary conditions settings at the cold inlet ( **velocity-inlet-large**) as a starting point.

### Solution Initialization

- i. Select **velocity-inlet-large** from the **Compute From** drop-down list.
- ii. Enter 1.2 m/s for **Y Velocity** in the **Initial Values** group box.

*Note:* While an initial **X Velocity** is an appropriate guess for the horizontal section, the addition of a **Y Velocity** component will give rise to a better initial guess throughout the entire elbow.

- iii. Click **Initialize**.

Check to see if the case conforms to best practices.

### Run Calculation → Check Case

8. Calculate a solution.

Start the calculation by requesting 250 iterations.

### Run Calculation

- i. Enter 250 for **Number of Iterations**.
- ii. Click **Calculate**.

*As the calculation progresses, the residuals will be plotted in the graphics window*

9. View the files generated by **ANSYS Workbench**.

### Result:

#### Viva questions:

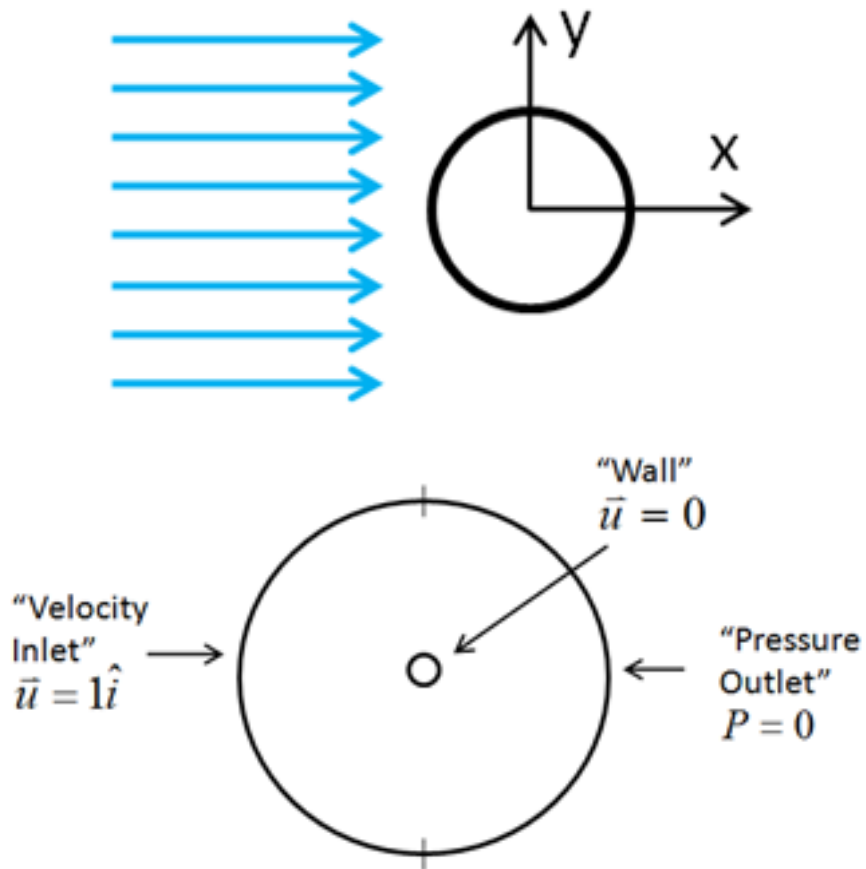
1. Define the pressure contour?
2. What is normal shock wave?
3. What is gauge pressure?

## EXPERIMENT – 6

### CIRCULATION OF THE LIFT OVER A CIRCULAR CYLINDER

Aim: To visualize the circulation of the lift over a circular cylinder.

Software Used: ICEM CFD and FLUENT



We will create the geometry and mesh in ICEM CFD, read the mesh into FLUENT, and solve the flow problem.

- Create or import Geometry
- Block the geometry
- Associate entities to the geometry.
- Apply mesh parameters.
- Generate the initial mesh.
- Export mesh

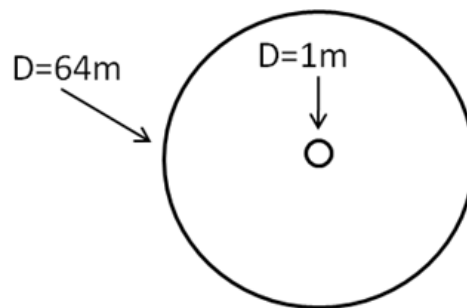
Consider the steady state case of a fluid flowing past a cylinder, as illustrated above. Obtain the velocity and pressure distributions when the Reynolds number is chosen to be 20. In order to simplify the computation, the diameter of the cylinder is set to 1 m, the x component of the velocity is set to 1 m/s and the density of the fluid is set to 1 kg/m<sup>3</sup>. Thus, the dynamic viscosity must be set to 0.05 kg/m\*s in order to obtain the desired Reynolds number.

### Pre-Analysis & Start-Up

Prior to opening FLUENT, we must answer a couple of questions. We must determine what our solution domain is and what the boundary conditions are.

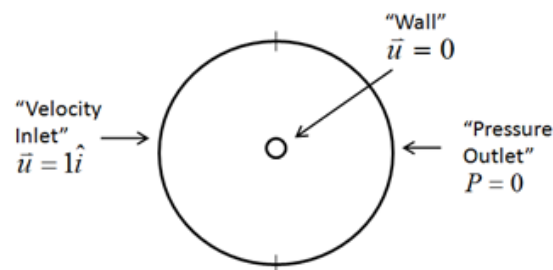
#### Solution Domain

For an external flow problem like this, one needs to determine where to place the outer boundary. A circular domain will be used for this simulation. The effects that the cylinder has on the flow extend far. Thus, the outer boundary will be set to be 64 times as large as the diameter of the cylinder. That is, the outer boundary will be a circle with a diameter of 64 m. The solution domain discussed here is illustrated below.



#### Boundary Conditions

First, we will specify a velocity inlet boundary condition. We will set the left half of the outer boundary as a velocity inlet with a velocity of 1 m/s in the x direction. Next, we will use a pressure outlet boundary condition for the right half of the outer boundary with a gauge pressure of 0 Pa. Lastly, we will apply a no slip boundary condition to the cylinder wall. The aforementioned boundary conditions are illustrated below.



## Set Up Problem in FLUENT

### Launch FLUENT

#### Start > Programs > Fluent> FLUENT 6.3.26

Select 2ddp from the list of options and click Run.

### Import File

#### Main Menu > File > Read > Case...

Navigate to your working directory and select the cylinder.msh file. Click OK.

### Analyze Grid

#### Grid > Info > Size

Check how many cells and nodes the mesh has.

#### Display > Grid

Display the grid information.

### Define Properties

#### Define > Models > Solver...

Under the Solver box, select **Pressure Based**.

Click **OK**.

#### Define > Models > Viscous

Select **Laminar** under **Model**

Click **OK**.

#### Define > Models > Energy

Do not select Energy Equation.

#### Define > Materials

Make sure air is selected under **Fluent Fluid Materials**.

Set **Density** to constant and equal to  $1 \text{ kg/m}^3$  and **Viscosity** to  $0.025 \text{ kg/m-s}$ . We choose these numbers so that  $Re = 40$ .

Click **Change/Create**.

#### Define > Operating Conditions

We'll work in terms of gauge pressures in this example. So set *Operating Pressure* to the ambient value of 101,325 Pa.

Click **OK**.

#### Define > Boundary Conditions

Set inlet, click **Set...** and set the **Velocity Magnitude** to 1 m/s. Click **OK**.

Set outlet, click **Set...** and set the **Gauge Pressure** at this boundary to 0. Click **OK**.

## **Solve**

### **Solve > Control > Solution**

Under **Discretization**, set **Momentum** to Second-Order Upwind.

### **Solve > Initialize > Initialize...**

Select inlet under **Compute From**. We'll set these values to be equal to those at the inlet.

### **Solve > Monitors > Residual...**

Now we will set the residual values (the criteria for a good enough solution). Once again, we'll set this value to 1e-06.

Select **Print** and **Plot** under Options. Click

**OK**

### **Solve > Monitors > Force...**

Under **Coefficient**, choose **Drag**. Under **Options**, select **Print** and **Plot**. Then, Choose cylinder under **Wall Zones**.

Set the **Force Vector** components for the drag. The drag is the force in the direction of the freestream. So to get the drag coefficient, set **X** to 1 and **Y** to 0.

Record the histories of Cd. Under **Options**, select **Write**. Fill in the name in the box under **File Name**, then the text file containing drag coefficients at each iteration will be stored in the file.

Click **Apply** for these changes to take effect.

Similarly, set the Force Monitor options for the Lift force. The lift is defined as the force component perpendicular to the direction of the free stream. So under **Force Vector**, set **X** to 0 and **Y** to 1.

Click **Apply**.

### **Report > Reference Values**

Now, set the reference values to set the base cases for our iteration. Select inlet under **Compute From**.

### **Main Menu > File > Write > Case...**

Save the case file before you start the iterations.

### **Solve > Iterate**

Make note of your findings, make sure you include data such as:

What does the convergence plot look like?

How many iterations does it take to converge?

### **Main Menu > File > Write > Case & Data...**

Save case and data after you have obtained a converged solution.

## **Analyze Results**

### **Drag / Lift coefficients**

#### **Report > Forces >**

Under **Force Vector**, we set **X** = 1 and **Y** = 0 to identify the direction of drag force. Click **Print** to

see what's displayed in the main window.

Plot convergence of the drag coefficient versus the number of iterations. Report the drag coefficient and compare it with the result in literature as shown in Table 1.

**Plot>File**

Click **Add...** choose the file with drag or lift coefficient.

Before you plot, you can adjust the **Axes** and **Curves** to get a better view.

Similarly, you can plot the lift coefficient, which should be zero for the symmetric flow. As you can see, the drag coefficient is around 2.1, which is significantly higher than the result in the table. Next, we will try to increase the domain size and repeat the simulation.

**Plot streamlines. Plot pressure, velocity, vorticity contours.**

**Display > Contours >**

**Results:**

**Viva questions:**

1. Define boundary conditions around the cylinder?
2. What is iteration?
3. What is center of pressure?

## EXPERIMENT – 7

### PRESSURE DISTRIBUTION OVER A SYMMETRIC AEROFOIL

Aim: To visualize the pressure distribution over a symmetric aerofoil at different velocities.

We will create the geometry and mesh in ICEM CFD, read the mesh into FLUENT, and solve the flow problem.

- Create or import Geometry
- Block the geometry
- Associate entities to the geometry.
- Apply mesh parameters.
- Generate the initial mesh.
- Export mesh

#### ***Introduction***

This provides a short step by step guide to meshing an aerofoil using the *icemcfd* package. The aim being to introduce you the important features of the program in the shortest possible space of time. The mesh that this tutorial produces will require further work to be used as a basis for serious calculations but this tutorial contains all the information that you need to do this.

*icemcfd* is available on the University GNU/Linux service called vega. Once you have logged into vega you can start it from the command line using the command *icemcfd*.

#### ***Importing the Geometry***

- Start *icemcfd*
- Obtain the NACA5012 coordinates file (this should be on the same page/folder where you obtained this document)
- File → Import Geometry → Formatted point data (then select the file you just obtained) A good tip at this point is to go to File → Change Working Directory and change the directory to where you have stored the NACA data – this saves a lot of navigation through file dialogues later.
- Then click apply in the bottom left corner of the screen
- We then want to create a curve made of those points to describe the surface of aerofoil
  - Under the Geometry tab in the top centre of the screen click on the Create/Modify Curve button
  - This pops a new menu in the bottom left. Click on the top left icon underneath “Inherit Part”, This is the “From Points” tool. Click on “Inherit Part” to deselect it and name the part SUCTION.
  - You should then select each point on the top of the curve one after the other with the left mouse button. (There ought to be a better way of doing this but I can't seem to find it!) You might



need to zoom in to get all the points in the trailing edge. The middle mouse button accepts your final selection

➤ Repeat the process for the points on the bottom of the curve. Name this PRESSURE.

- Then we want to create the surface on which we will calculate the fluid flow. To do this we will create a domain around 1 chord lengths above and below the aerofoil and 1.5 chord lengths before and after. This is slightly arbitrary and we may want to change this after we look at the results!
- Add points at (-1.5,1),(-1.5,-1),(2,1) and (2,-1) Click on the Create Point tool under the geometry tab and use the Explicit Locations tool to do this.

Join these points together with straight lines. Label the left hand one INLET, the right hand line EXIT and the upper and lower parts TOP and BOTTOM.

- We are going to construct an outer surface called FLUID which will then mesh later.
- Make a surface from the outer edges into surface (Geometry Tab → Create/Modify Surface → Simple Surface)
  - Click on the “Inherit Part” button so that it is no longer checked
  - Rename the Surface as FLUID
  - Select the four curves
  - Click on OK or Apply
  - This surface might not actually be visible. On the left hand side of the screen is tree menu that controls what is and is not visible. If you want to see the surface simply go through the menu and ensure that under “Geometry” the “Surfaces” node is ticked.

### ***Setting up the Initial Blocking***

To do this we need to apply what is known as a “blocking strategy”. In *icemcfd* for structured meshes the geometry is joined to a series of blocks which are then filled with the cells that make up the actual volumes on which the calculation will take place on. The process of meshing is therefore somewhat different from the CAD programs you will have used up to now as you are not trying to draw the part but the surrounding area which the fluid surrounds.

- Go to the Blocking Tab
- Click on “Create Block” and ensure in the “Initialize Blocks” drop down the type is 2D Planar and the part selected is “FLUID”. Click OK, A blue message saying “Initialize blocking done” should appear in the message window
- This creates a Block which we then need to associate to various edges
- In the Blocking Tab click on “Associate” (fifth tool from the left) then in the left hand menu that appears select “Associate Edge to Curve”. This should look a bit like Figure 4 , note that in Figure 4 the points are not displayed.
- You now need to associate each side of the block one of the parts we made earlier. (INLET, OUTLET, TOP and BOTTOM). **This must be done separately for each part.** To do this: Click on the first edge (which is part of the blocking) and then the first curve (which is geometry). This is most easily done by left clicking and middle clicking - an on screen prompt guides as to which mouse button to press when.

- We can now get *icemcfd* to do a crude initial first mesh. Go to the Mesh Tab → Part Mesh Setup and set the max size to be 0.1 on all the parts. Then go to the Blocking Tab → Pre-Mesh Params (9<sup>th</sup> from left) select the “Recalculate Sizes” tool and click ok. It may look like nothing has happened but a first mesh has been created!
- To see it you need to make the mesh viewable. On the left hand side of the screen there is a tree menu with a white background. Expand the Blocking Menu and click on “Pre-Mesh” a square mesh that completely ignores the aerofoil should now appear. Clearly there is work to be done! You should now turn the Pre-Mesh view off to make manipulations easier. We now need to associate the face (the mesh) with the surface (the FLUID part we created earlier.
- On the Blocking Tab select the Associate Vertices icon (fifth from left). In the tool that pops on the bottom left select Associate Face → Surface (fourth from left). Then select the face and the FLUID part in the pop up window.

### ***Meshing the aerofoil***

Clearly we need some way to tell *icemcfd* that we would like the mesh associated with the aerofoil shape in the middle of the screen and to organise this mesh so that we end up with the right sort of mesh.

- First further divide up the Blocks. Blocking Tab → Split Block and then select the Split Block tool. Use this tool to split the block upstream and downstream of the aerofoil and above and below the trailing edge. Again left and middle clicking is the most efficient method. Your screen should look something like Figure 6
- Now we want to add a special type of block called an “O grid” around the aerofoil. Go to the Blocking Tab → Split Block (Second from left) and then select the O grid Block tool (Second item from left in the bottom left panel). After you have selected the tool you need to Click on Select Blocks.
- Select the central block and the block on the middle right by using the left mouse then the middle mouse and two turquoise squares should appear in the centre of your screen. Now click on the Select Edge(s) option within the tool and then select the right hand most edge. A yellow square should appear on the right hand side. Click ok or apply to define a new O grid.
- Now we should delete the central block as this is the aerofoil itself and not something that we actually want to calculate. Blocking Tab → Delete Block and left click on the central square and use middle mouse to get rid of it.
- On the left hand side of the screen there is that tree menu with a white background, if you click on Pre-Mesh you should be able to view the mesh again.
- The mesh most likely still occupies the centre of the aerofoil, so we add a point to guide the program about where we would like the mesh to go. Blocking Tab → Edit Edge, then select the Split Edge tool and select Linear from the drop down menu.
- At this point you will want to adjust the position of the vertices on the aerofoil so that it gives a more reasonable mesh. Blocking Tab → Move Vertex tool will accomplish this.
- The Pre-Mesh Parameters tool (Blocking Tab → Pre-Mesh Parameters (Ninth from left) then the Edge Params Tool (Third from Left) should be used to add more points in various places. Until the mesh looks better.
- A number of tools allow you to assess the mesh quality. Blocking Tab → Pre-Mesh Quality Histograms allow you to look at things like the worst element. A graph appears in the bottom right

which you can click on to examine the the worst element then adjust the mesh spacing, the position of the vertices on the blocks and so on.

## **Set Up Problem in FLUENT**

### **Launch FLUENT**

**Start > Programs > Fluent Inc > FLUENT 6.3.26**

Select *2ddp* from the list of options and click *Run*.

#### **Import File**

**Main Menu > File > Read > Case...**

Navigate to your working directory and select the airfoil.msh file. Click *OK*. Check that the displayed information is consistent with our expectations of the airfoil grid.

#### **Analyze Grid**

**Grid > Info > Size**

How many cells and nodes does the grid have?

**Display > Grid**

Note what the surfaces *farfield1*, *farfield2*, etc. correspond to by selecting and plotting them in turn.

Zoom into the airfoil.

Where are the nodes clustered? Why?

#### **Define Properties**

**Define > Models > Solver...**

Under the *Solver* box, select *Pressure Based*.

Click *OK*.

**Define > Models > Viscous**

Select *Inviscid* under *Model*

Click *OK*.

**Define > Models > Energy**

The speed of sound under SSL conditions is 340 m/s so that our freestream Mach number is around 0.15. This is low enough that we'll assume that the flow is incompressible. So the energy equation can be turned off.

Make sure there is no check in the box next to *Energy Equation* and click *OK*.

## Define > Materials

Make sure *air* is selected under *Fluid Materials*. Set *Density* to *constant* and equal to 1.225 kg/m<sup>3</sup>.

Click *Change/Create*.

## Define > Operating Conditions

We'll work in terms of gauge pressures in this example. So set *Operating Pressure* to the ambient value of 101,325 Pa.

Click *OK*.

## Define > Boundary Conditions

Set *farfield1* and *farfield2* to the *velocity-inlet* boundary type.

For each, click Set.... Then, choose *Components* under *Velocity Specification Method* and set the x- and y-components to that for the freestream. For instance, the x-component is  $50 \cdot \cos(1.2) = 49.99$ . Click *OK*.

Set *farfield3* to *pressure-outlet* boundary type, click *Set...* and set the *Gauge Pressure* at this boundary to 0. Click *OK*.

## Step 5: Solve!

### Solve > Control > Solution

Take a look at the options available.

Under *Discretization*, set *Pressure* to *PRESTO!* and *Momentum* to *Second-Order Upwind*.

Click *OK*.

### Solve > Initialize > Initialize...

As you may recall from the previous tutorials, this is where we set the initial guess values (the base case) for the iterative solution. Once again, we'll set these values to be equal to those at the inlet. Select *farfield1* under *Compute From*.

Click *Init*.

### Solve > Monitors > Residual...

Now we will set the residual values (the criteria for a good enough solution). Once again, we'll set this value to 1e-06. Click *OK*.

### Solve > Monitors > Force...

Under *Coefficient*, choose *Lift*. Under *Options*, select *Print* and *Plot*. Then, Choose *airfoil* under *Wall Zones*.

Lastly, set the **Force Vector** components for the lift. The lift is the force perpendicular to the direction of the freestream. So to get the lift coefficient, set **X** to  $-\sin(1.2^\circ)=-0.020942$  and **Y** to  $\cos(1.2^\circ)=0.9998$ . Click **Apply** for these changes to take effect.

Similarly, set the **Force Monitor** options for the **Drag** force. The drag is defined as the force component in the direction of the freestream. So under **Force Vector**, set **X** to  $\cos(1.2^\circ)=0.9998$  and **Y** to  $\sin(1.2^\circ)=0.020942$ . Turn on only **Print** for it.

### **Report > Reference Values**

Now, set the reference values to set the base cases for our iteration. Select **farfield1** under **Compute From**.

Click **OK**. Note that the reference pressure is zero, indicating that we are measuring gage pressure.

### **Main Menu > File > Write > Case...**

Save the case file before you start the iterations.

### **Solve > Iterate**

Make note of your findings, make sure you include data such as; What does the convergence plot look like?

How many iterations does it take to converge?

How does the Lift coefficient compare with the experimental data?

### **Main Menu > File > Write > Case & Data...**

Save case and data after you have obtained a converged solution.

## **Step 6: Analyze Results**

### **Lift Coefficient**

The solution converged after about 480 iterations.

```
476 1.0131e-06 4.3049e-09 1.5504e-09 6.4674e-01 2.4911e-03 0:00:48 524
```

- 477 solution is converged

```
477 9.9334e-07 4.2226e-09 1.5039e-09 6.4674e-01 2.4910e-03 0:00:38 523
```

From FLUENT main window, we see that the lift coefficient is 0.647.

### **Plot Velocity Vectors**

Let's see the velocity vectors along the airfoil.

### **Display > Vectors**

Enter 4 next to *Scale*. Enter 3 next to *Skip*. Click *Display*.

As can be seen, the velocity of the upper surface is faster than the velocity on the lower surface.

### **Main Menu > File > Hardcopy**

Make sure that *Reverse Foreground/Background* is checked and select *Color* in

*Coloring* section. Click *Preview*. Click *No* when prompted "*Reset graphics window?*"

On the leading edge, we see a stagnation point where the velocity of the flow is nearly zero. The fluid accelerates on the upper surface as can be seen from the change in colors of the vectors.

On the trailing edge, the flow on the upper surface decelerates and converge with the flow on the lower surface.

### **Plot Pressure Coefficient**

**Pressure Coefficient** is a dimensionless parameter defined by the equation where  $p$  is the static pressure,

$P_{ref}$  is the reference pressure, and

$q_{ref}$  is the reference dynamic pressure defined by

### **Plot > XY Plot...**

Change the *Y Axis Function* to *Pressure...*, followed by *Pressure Coefficient*. Then, select *airfoil* under *Surfaces*.

Click *Plot*.

The lower curve is the upper surface of the airfoil and has a negative pressure coefficient as the pressure is lower than the reference pressure.

### **Plot Pressure Contours**

Plot static pressure contours.

### **Display > Contours...**

Select *Pressure...* and *Pressure Coefficient* from under *Contours Of*. Check the *Filled* and *Draw Grid* under *Options* menu. Set Levels to 50. Click *Display*.

From the contour of pressure coefficient, we see that there is a region of high pressure at the leading edge (stagnation point) and region of low pressure on the upper surface of airfoil. This is of what we expected from analysis of velocity vector plot. From Bernoulli equation, we know that whenever there is high velocity, we have low pressure and vice versa.

**Result:**

**Viva Questions:**

1. What is vega in terms of ICEM CFD?
2. What is presto under the discretization?
3. List the types of airfoils?
4. What is nomenclature of Airfoil?

## EXPERIMENT –8

### PRESSURE DISTRIBUTION OVER A CAMBEREDAEROFOIL

**Aim:** To visualize the pressure distribution over a cambered aerofoil at different velocities.

**Software Used:** ICEM CFD and FLUENT

We will create the geometry and mesh in ICEM CFD, read the mesh into FLUENT, and solve the flow problem.

- Create or import Geometry
- Block the geometry
- Associate entities to the geometry.
- Apply mesh parameters.
- Generate the initial mesh.
- Export mesh

#### *Introduction*

This provides a short step by step guide to meshing an aerofoil using the *icemcfd* package. The aim being to introduce you the important features of the program in the shortest possible space of time. The mesh that this tutorial produces will require further work to be used as a basis for serious calculations but this tutorial contains all the information that you need to do this.

*icemcfd* is available on the University GNU/Linux service called vega. Once you have logged into vega you can start it from the command line using the command *icemcfd*.

#### *Importing the Geometry*

- Start *icemcfd*
- Obtain the NACA5012 coordinates file (this should be on the same page/folder where you obtained this document)
- File → Import Geometry → Formatted point data (then select the file you just obtained) A good tip at this point is to go to File → Change Working Directory and change the directory to where you have stored the NACA data – this saves a lot of navigation through file dialogues later.
- Then click apply in the bottom left corner of the screen
- We then want to create a curve made of those points to describe the surface of aerofoil
  - Under the Geometry tab in the top centre of the screen click on the Create/Modify Curve button
  - This pops a new menu in the bottom left. Click on the top left icon underneath “Inherit Part”, This is the “From Points” tool. Click on “Inherit Part” to deselect it and name the part SUCTION.
  - You should then select each point on the top of the curve one after the other with the left mouse button. (There ought to be a better way of doing this but I can't seem to find it!) You might need



to zoom in to get all the points in the trailing edge. The middle mouse button accepts your final selection

- Repeat the process for the points on the bottom of the curve. Name this PRESSURE.

- Then we want to create the surface on which we will calculate the fluid flow. To do this we will create a domain around 1 chord lengths above and below the aerofoil and 1.5 chord lengths before and after. This is slightly arbitrary and we may want to change this after we look at the results!
- Add points at (-1.5,1),(-1.5,-1),(2,1) and (2,-1) Click on the Create Point tool under the geometry tab and use the Explicit Locations tool to do this.

Join these points together with straight lines. Label the left hand one INLET, the right hand line EXIT and the upper and lower parts TOP and BOTTOM.

- We are going to construct an outer surface called FLUID which will then mesh later.
- Make a surface from the outer edges into surface (Geometry Tab → Create/Modify Surface → Simple Surface)
  - Click on the “Inherit Part” button so that it is no longer checked
  - Rename the Surface as FLUID
  - Select the four curves
  - Click on OK or Apply
  - This surface might not actually be visible. On the left hand side of the screen is tree menu that controls what is and is not visible. If you want to see the surface simply go through the menu and ensure that under “Geometry” the “Surfaces” node is ticked.

### ***Setting up the Initial Blocking***

To do this we need to apply what is known as a “blocking strategy”. In *icemcfd* for structured meshes the geometry is joined to a series of blocks which are then filled with the cells that make up the actual volumes on which the calculation will take place on. The process of meshing is therefore somewhat different from the CAD programs you will have used up to now as you are not trying to draw the part but the surrounding area which the fluid surrounds.

- Go to the Blocking Tab
- Click on “Create Block” and ensure in the “Initialize Blocks” drop down the type is 2D Planar and the part selected is “FLUID”. Click OK, A blue message saying “Initialize blocking done” should appear in the message window
- This creates a Block which we then need to associate to various edges
- In the Blocking Tab click on “Associate” (fifth tool from the left) then in the left hand menu that appears select “Associate Edge to Curve”. This should look a bit like Figure 4 , note that in Figure 4 the points are not displayed.
- You now need to associate each side of the block one of the parts we made earlier. (INLET, OUTLET, TOP and BOTTOM). **This must be done separately for each part.** To do this: Click on the first edge (which is part of the blocking) and then the first curve (which is geometry). This is most easily done by left clicking and middle clicking - an on screen prompt guides as to which mouse button to press when.

- We can now get *icemcfd* to do a crude initial first mesh. Go to the Mesh Tab → Part Mesh Setup and set the max size to be 0.1 on all the parts. Then go to the Blocking Tab → Pre-Mesh Params (9<sup>th</sup> from left) select the “Recalculate Sizes” tool and click ok. It may look like nothing has happened but a first mesh has been created!
- To see it you need to make the mesh viewable. On the left hand side of the screen there is a tree menu with a white background. Expand the Blocking Menu and click on “Pre-Mesh” a square mesh that completely ignores the aerofoil should now appear. Clearly there is work to be done! You should now turn the Pre-Mesh view off to make manipulations easier. We now need to associate the face (the mesh) with the surface (the FLUID part we created earlier).
- On the Blocking Tab select the Associate Vertices icon (fifth from left). In the tool that pops on the bottom left select Associate Face → Surface (fourth from left). Then select the face and the FLUID part in the pop up window.

### ***Meshing the aerofoil***

Clearly we need some way to tell *icemcfd* that we would like the mesh associated with the aerofoil shape in the middle of the screen and to organize this mesh so that we end up with the right sort of mesh.

- First further divide up the Blocks. Blocking Tab → Split Block and then select the Split Block tool. Use this tool to split the block upstream and downstream of the aerofoil and above and below the trailing edge. Again left and middle clicking is the most efficient method. Your screen should look something like Figure 6
- Now we want to add a special type of block called an “O grid” around the aerofoil. Go to the Blocking Tab → Split Block (Second from left) and then select the O grid Block tool (Second item from left in the bottom left panel). After you have selected the tool you need to Click on Select Blocks.
- Select the central block and the block on the middle right by using the left mouse then the middle mouse and two turquoise squares should appear in the centre of your screen. Now click on the Select Edge(s) option within the tool and then select the right hand most edge. A yellow square should appear on the right hand side. Click ok or apply to define a new O grid.
- Now we should delete the central block as this is the aerofoil itself and not something that we actually want to calculate. Blocking Tab → Delete Block and left click on the central square and use middle mouse to get rid of it.
- On the left hand side of the screen there is that tree menu with a white background, if you click on Pre-Mesh you should be able to view the mesh again.
- The mesh most likely still occupies the centre of the aerofoil, so we add a point to guide the program about where we would like the mesh to go. Blocking Tab → Edit Edge, then select the Split Edge tool and select Linear from the drop down menu.
- At this point you will want to adjust the position of the vertices on the aerofoil so that it gives a more reasonable mesh. Blocking Tab → Move Vertex tool will accomplish this.
- The Pre-Mesh Parameters tool (Blocking Tab → Pre-Mesh Parameters (Ninth from left) then the Edge Params Tool (Third from Left) should be used to add more points in various places. Until the mesh looks better.
- A number of tools allow you to assess the mesh quality. Blocking Tab → Pre-Mesh Quality Histograms allow you to look at things like the worst element. A graph appears in the bottom right

which you can click on to examine the the worst element then adjust the mesh spacing, the position of the vertices on the blocks and so on.

## **Set Up Problem in FLUENT**

### **Launch FLUENT**

**Start > Programs > Fluent Inc > FLUENT 6.3.26**

Select *2ddp* from the list of options and click *Run*.

#### **Import File**

**Main Menu > File > Read > Case...**

Navigate to your working directory and select the airfoil.msh file. Click *OK*. Check that the displayed information is consistent with our expectations of the airfoil grid.

#### **Analyze Grid**

**Grid > Info > Size**

How many cells and nodes does the grid have?

**Display > Grid**

Note what the surfaces *farfield1*, *farfield2*, etc. correspond to by selecting and plotting them in turn.

Zoom into the airfoil.

Where are the nodes clustered? Why?

#### **Define Properties**

**Define > Models > Solver...**

Under the *Solver* box, select *Pressure Based*.

Click *OK*.

**Define > Models > Viscous**

Select *Inviscid* under *Model*

Click *OK*.

**Define > Models > Energy**

The speed of sound under SSL conditions is 340 m/s so that our freestream Mach number is around 0.15. This is low enough that we'll assume that the flow is incompressible. So the energy equation can be turned off.

Make sure there is no check in the box next to *Energy Equation* and click *OK*.

## Define > Materials

Make sure *air* is selected under *Fluid Materials*. Set *Density* to *constant* and equal to  $1.225 \text{ kg/m}^3$ .

Click *Change/Create*.

## Define > Operating Conditions

We'll work in terms of gauge pressures in this example. So set *Operating Pressure* to the ambient value of 101,325 Pa.

Click *OK*.

## Define > Boundary Conditions

Set *farfield1* and *farfield2* to the *velocity-inlet* boundary type.

For each, click *Set...* Then, choose *Components* under *Velocity Specification Method* and set the x- and y-components to that for the freestream. For instance, the x-component is  $50 \cdot \cos(1.2) = 49.99$ . (Note that  $1.2^\circ$  is used as our angle of attack instead of  $2^\circ$  to adjust for the error caused by assuming the airfoil to be 2D instead of 3D.)

Click *OK*.

Set *farfield3* to *pressure-outlet* boundary type, click *Set...* and set the *Gauge Pressure* at this boundary to 0. Click *OK*.

## Solve!

### Solve > Control > Solution

Take a look at the options available.

Under *Discretization*, set *Pressure* to *PRESTO!* and *Momentum* to *Second-OrderUpwind*.

Click *OK*.

### Solve > Initialize > Initialize...

Click *Init*.

### Solve > Monitors > Residual...

Now we will set the residual values (the criteria for a good enough solution). Once again, we'll set this value to  $1e-06$ .

Click *OK*.

### Solve > Monitors > Force...

Under *Coefficient*, choose *Lift*. Under *Options*, select *Print* and *Plot*. Then, Choose *airfoil* under *Wall Zones*.

Lastly, set the **Force Vector** components for the lift. The lift is the force perpendicular to the direction of the freestream. So to get the lift coefficient, set **X** to  $-\sin(1.2^\circ)=-0.020942$  and **Y** to  $\cos(1.2^\circ)=0.9998$ . Click **Apply** for these changes to take effect.

Similarly, set the **Force Monitor** options for the **Drag** force. The drag is defined as the force component in the direction of the freestream. So under **Force Vector**, set **X** to  $\cos(1.2^\circ)=0.9998$  and **Y** to  $\sin(1.2^\circ)=0.020942$ . Turn on only **Print** for it.

### **Report > Reference Values**

Now, set the reference values to set the base cases for our iteration. Select **farfield1** under **Compute From**.

Click **OK**. Note that the reference pressure is zero, indicating that we are measuring gage pressure.

### **Main Menu > File > Write > Case...**

Save the case file before you start the iterations.

### **Solve > Iterate**

Make note of your findings, make sure you include data such as; What does the convergence plot look like?

How many iterations does it take to converge?

How does the Lift coefficient compare with the experimental data?

### **Main Menu > File > Write > Case & Data...**

Save case and data after you have obtained a converged solution.

## **Step 6: Analyze Results**

### **Lift Coefficient**

The solution converged after about 480 iterations.

```
476 1.0131e-06 4.3049e-09 1.5504e-09 6.4674e-01 2.4911e-03 0:00:48 524
```

- 477 solution is converged

```
477 9.9334e-07 4.2226e-09 1.5039e-09 6.4674e-01 2.4910e-03 0:00:38 523
```

From FLUENT main window, we see that the lift coefficient is 0.647.

### **Plot Velocity Vectors**

Let's see the velocity vectors along the airfoil.

### **Display > Vectors**

Enter 4 next to *Scale*. Enter 3 next to *Skip*. Click *Display*.

As can be seen, the velocity of the upper surface is faster than the velocity on the lower surface.

### **Plot Pressure Coefficient**

**Pressure Coefficient** is a dimensionless parameter defined by the equation where  $p$  is the static pressure,

$P_{ref}$  is the reference pressure, and

$q_{ref}$  is the reference dynamic pressure defined by

### **Plot > XY Plot...**

Change the *Y Axis Function* to *Pressure...*, followed by *Pressure Coefficient*. Then, select *airfoil* under *Surfaces*.

Click *Plot*.

The lower curve is the upper surface of the airfoil and has a negative pressure coefficient as the pressure is lower than the reference pressure.

### **Plot Pressure Contours**

Plot static pressure contours.

### **Display > Contours...**

Select *Pressure...* and *Pressure Coefficient* from under *Contours Of*. Check the *Filled* and *Draw Grid* under *Options* menu. Set Levels to 50.

Click *Display*.

### **Result:**

### **Viva Questions:**

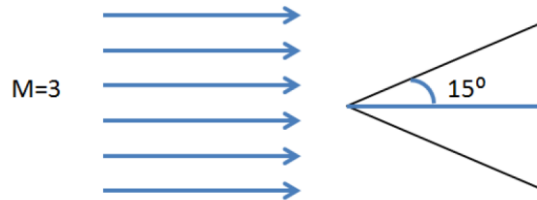
1. Differentiate the symmetrical and cambered airfoil
2. What is Lift coefficient and Drag coefficient?
3. What is camber?

## EXPERIMENT –9

### SHOCK WAVE SUPERSONIC FLOW OVER WEDGE

**Aim:** To observe the shock wave phenomena around a wedge at supersonic flow.

**Software Used:** ICEM CFD and FLUENT.

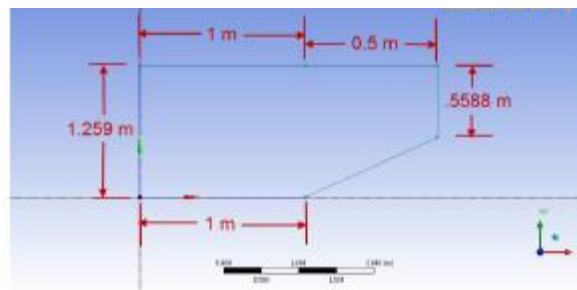


The stream is at the following conditions:

Mach Number  $M_1 = 3$

Static Pressure  $p_1 = 1 \text{ atm}$

Static Temperature  $T_1 = 300 \text{ k}$



We will create the geometry and mesh in ICEM CFD, read the mesh into FLUENT, and solve the flow problem.

- Create or import Geometry
- Block the geometry
- Associate entities to the geometry.
- Apply mesh parameters.
- Generate the initial mesh.
- Export mesh

## **Start ICEM CFD**

### **Create Geometry in ICEM CFD**

- 1) Close any previous files, have blank screen in ICEM.
- 2) Create new project file for the pipe in ICEM:
  - a) File, New Project.
  - b) Select folders in turn: Desktop, CFX Training Files – Copy, Flat plate.
  - c) Save project with name “Flat plate” in folder “Flat plate”.
- 3) We are going to create some points now:
  - a) Select Geometry tab at top of screen.
  - b) Select “Create Point” option.
  - c) In the bottom left are various options for creating the point. Select the second “XYZ” option.
  - d) Input the numbers  $X=0$ ,  $Y=0$ ,  $Z=0$ .
  - e) Press “Apply” to create a point on screen
  - f) Note - If the point does not appear, then check that points are displayed in the tree, and use the “Fit Window” option in the top left to zoom the screen correctly.

### **Create points**

Explicit location > specify x and y location and click ok

Similarly create required points

Note that for a 2D problem, the z-coordinate can always be left to the default value of 0.

### **Create curves**

Under “Geometry” tab, select “Create and modify curve”.

- b) Under “Create and modify curve” option panel on bottom left, select 3rd option “  
Select two points on the screen and select ok Create remaining curves

### **Create surface**

Simple surface – from 2-4 curves

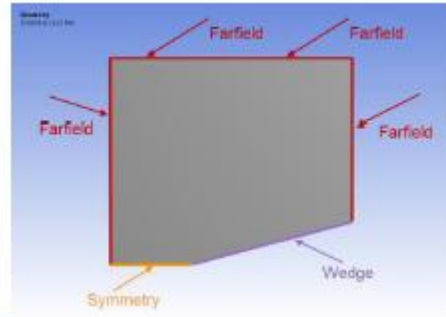
To check surface click on solid frame

### **Label the curves**

Give names to the INLET, OUTLET and WALL surfaces:

- a) Under “Geometry” tree, highlight only “Surfaces”.
- b) Right click “Parts”, Create Part.
- c) Part = “INLET”.
- d) Entities: click arrow icon, left click surface (near surface, around origin) to highlight it, middle click to save selection, right click to de-select mode.
- e) Note that the INLET part has been created, and can be turned on and off independently using the tree view.
- f) Part = “OUTLET”, repeat step to identify an outlet at the far end.
- g) Part = “WALL”, repeat step to identify the wall.
- h) Notes: “GEOM” now contains only 0D and 1D elements; points and lines.





## Block the Geometry

The geometry and part information has already been defined for this tutorial. You will create the initial block in this step.

1. Create the initial block.

Blocking > Create Block > Initialize Blocks

- a. Enter FLUID in the Part field.
- b. Select 2D Planar in the Type drop-down list.
- c. Click Apply.

## Associate Entities to the Geometry

You will associate the edges of the blocking to the curves of the CAD geometry in this step. First select the edges and then the curves to which you want to associate them.

1. Associate the inlet, the left-most end of the large flat plate.
  - a. Ensure that Project Vertices is disabled.
  - b. Click (Select edge(s)) and select edge 13–41. Click the middle-mouse button to accept the selection.
  - c. Click (Select compcurve(s)) and select CURVES/1 . Click the middle-mouse button to accept the selection.
  - d. Click Apply.

The associated edge will be colored green.

## Generate the Initial Mesh

Blocking > Pre-Mesh Params > Update Sizes

1. Retain the selection of Update All in the Method list.
2. Select Run Check/Fix Blocks to check for mesh problems automatically and repair them if possible.
3. Click Apply.
4. Enable Pre-Mesh.

## Blocking Pre-Mesh

The Mesh dialog will appear, asking if you want to recompute the mesh.

5. Click Yes in the Mesh dialog to compute the initial mesh.
6. Disable Vertices and Edges.

## Verify and Save the Mesh and Blocking

1. Convert the mesh to unstructured format.  
Blocking Pre-Mesh Convert to Unstruct Mesh
2. Save the blocking file (2D-Flat plate -geometry-final.blk).

### File > Blocking > Save Blocking As...

This block file can be loaded in a future session (File > Blocking > Open Blocking...) for additional modification or to mesh a similar geometry. Save each blocking to a separate file instead of overwriting previous one. In more complex models, you may have to back track and load a previous blocking.

3. Save the project file (2D-flat plate-geometry-final.prj).  
File > Save Project As...

This will save all the files—tetin, blocking, and unstructured mesh.

4. Exit the current session.

File > Exit

Save your ICEM CFD file in your working directory.

## Setting Up the CFD Simulation in ANSYS FLUENT

### Problem Setup - General

Now, FLUENT should open. We will begin setting up some options for the solver. In the left hand window (in what I will call the *Outline* window), under **Problem Setup**, select **General**. The only option we need to change here is the type of solver. In the *Solver* window, select **Density-Based**.

### Models

In the outline window, click **Models**. We will need to utilize the energy equation in order to solve this simulation. Under **Models** highlight **Energy - Off** and click **Edit...** Now, the *Energy* window will launch. Check the box next to **Energy Equation** and hit OK. Doing this turns on the energy equation.

We also need to change the type of viscosity model. Select **Viscous - Laminar** and click **Edit...** Choose the **Inviscid** option and press **OK**.

### Materials

In the *Outline* window, highlight **Materials**. In the *Materials* window, highlight **Fluid**, and click **Create/Edit...** this will launch the *Create/Edit Materials* window; here we can specify the properties of the fluid. Set the **Density** to **Ideal Gas**, the **Specific Heat** to 1006.43, the **Molecular Weight** to 28.966. When you have updated these fields, press **Change/Create**.

### Boundary Conditions

In the *Outline* window, select **Boundary Conditions**. We will now specify each boundary condition for the simulation.

### Farfield

In the *Boundary Conditions* window, select **farfield**. Use the drop-down menu to change the **Type** to **pressure-far-field**. You will be asked to confirm the change, and do so by pressing **OK**. Next, a

dialogue box will open with some parameters we need to specify. Change the *Gauge Pressure (Pascal)* to **101325**, and *Mach Number* to **3**.

Also, select the *Thermal* tab, and ensure that the temperature correctly defaulted to **300 K**. When you are finished, press **OK**.

### **Wedge**

In the *Boundary Conditions* window, select **wedge**. Use the drop-down menu to change the *Type* to **wall**.

### **Symmetry**

In the *Boundary Conditions* window, select **symmetry**. Use the drop-down menu to change the *Type* to **symmetry**.

### **Operating Conditions**

In the *Boundary Conditions* window, select the **Operating Conditions** button. Change the *Gauge Pressure* to **0**. Then press **OK**

It is important to check the operating conditions. When setting the density in materials to ideal gas, FLUENT calculates the density using the absolute pressure. However, the pressure we specify is the gauge pressure, not the absolute pressure. FLUENT will use the absolute pressure to compute the density therefore if we do not set the operating pressure to 0 our density will be incorrect for the flow field.

### **Numerical Solution**

#### **Solution Methods**

In the *Outline* window, select **Solution Methods** to open the *Solution Methods* window. Under *Spatial Discretization*, ensure that the option under *Flow Second Order Upwind* is selected.

#### **Solution Controls**

In the *Outline* window, select **Solution Controls** to open the *Solution Controls* window. Ensure that the *Courant Number* is set to **5.0**.

The Courant number can be considered a non dimensionalized time step. The density-based solver obtains the steady-state solution by starting with the initial guess and marching in pseudo-time until convergence is obtained. The Courant number controls the time step the solver uses. The larger it is, the faster the solution will converge but it will not be very stable and can diverge. The smaller it is, the slower it is to reach convergence but the solution is much more stable.

#### **Monitors**

In the *Outline* window, click **Monitors** to open the *Monitors* window. In the *Monitors* window, select **Residuals - Print,Plot** and press **Edit...** This will open the *Residual Monitors* window. We want to change the convergence criteria for our solution. Under *Equation* and to the right of **Continuity**, change the **Absolute Criteria** to 1e-6. Repeat for **x-velocity**, **y-velocity**, and **energy**, then press **OK**.

#### **Solution Initialization**

In the *Outline* window, select **Solution Initialization**. We need to make an "Initial Guess" to the solution so FLUENT can iterate to find the final solution. In the *Solution Initialization* window, select **Standard**

**Initialization**, then under **Compute from**, select **farfield** from the drop down box. Check to see that the values that generate match our inputted values, then press **Initialize**

### **Run Calculation**

In the *Outline* window, select **Run Calculation**. Change the **Number of Iterations** to 4000. Double click **Calculate** to run the calculation. It should a few minutes to solve. After the calculation is complete, save the project. Do not close FLUENT.

### **Result:**

### **Viva Questions:**

1. List the types of flow field under Mach number.
2. Define the Mach number and Reynolds number.
3. What do you understand by expansion waves?

## **EXPERIMENT – 10**

### **SHOCK WAVE AROUND A CONE**

**Aim:** To observe the shock wave phenomena and change of properties around a cone at supersonic Mach number.

**Software Used :**ICEM CFD and FLUENT

We will create the geometry and mesh in ICEM CFD, read the mesh into FLUENT, and solve the flow problem.

- Create or import Geometry
- Block the geometry
- Associate entities to the geometry.
- Apply mesh parameters.
- Generate the initial mesh.
- Export mesh

#### **Start ICEM CFD**

#### **Create Geometry in ICEM CFD**

- 1) Close any previous files, have blank screen in ICEM.
- 2) Create new project file for the pipe in ICEM:
  - a) File, New Project.
  - b) Select folders in turn: Desktop, CFX Training Files – Copy, Flat plate.
  - c) Save project with name “Flat plate” in folder “Flat plate”.
- 3) We are going to create some points now:
  - a) Select Geometry tab at top of screen.
  - b) Select “Create Point” option.
  - c) In the bottom left are various options for creating the point. Select the second “XYZ” option.
  - d) Input the numbers  $X=0$ ,  $Y=0$ ,  $Z=0$ .
  - e) Press “Apply” to create a point on screen
  - f) Note - If the point does not appear, then check that points are displayed in the tree, and use the “Fit Window” option in the top left to zoom the screen correctly.

#### **Create points**

Explicit location > specify x and y location and click ok

Similarly create required points

Note that for a 2D problem, the z-coordinate can always be left to the default value of 0.

#### **Create curves**

Under “Geometry” tab, select “Create and modify curve”.

- b) Under “Create and modify curve” option panel on bottom left, select 3rd option “  
Select two points on the screen and select ok Create remaining curves

## **Create surface**

Simple surface – from 2-4 curves  
To check surface click on solid frame

## **Label the curves**

Give names to the INLET, OUTLET and WALL surfaces:

- a) Under “Geometry” tree, highlight only “Surfaces”.
- b) Right click “Parts”, Create Part.
- c) Part = “INLET”.
- d) Entities: click arrow icon, left click surface (near surface, around origin) to highlight it, middle click to save selection, right click to de-select mode.
- e) Note that the INLET part has been created, and can be turned on and off independently using the tree view.
- f) Part = “OUTLET”, repeat step to identify an outlet at the far end.
- g) Part = “WALL”, repeat step to identify the wall.
- h) Notes: “GEOM” now contains only 0D and 1D elements; points and lines.

## **Block the Geometry**

The geometry and part information has already been defined for this tutorial. You will create the initial block in this step.

1. Create the initial block.  
Blocking > Create Block > Initialize Blocks
  - a. Enter FLUID in the Part field.
  - b. Select 2D Planar in the Type drop-down list.
  - c. Click Apply.

## **Associate Entities to the Geometry**

You will associate the edges of the blocking to the curves of the CAD geometry in this step. First select the edges and then the curves to which you want to associate them.

1. Associate the inlet, the left-most end of the large flat plate.
  - a. Ensure that Project Vertices is disabled.
  - b. Click (Select edge(s)) and select edge 13–41. Click the middle-mouse button to accept the selection.
  - c. Click (Select compcurve(s)) and select CURVES/1 . Click the middle-mouse button to accept the selection.
  - d. Click Apply.

The associated edge will be colored green.

## **Generate the Initial Mesh**

Blocking > Pre-Mesh Params > Update Sizes

1. Retain the selection of Update All in the Method list.
2. Select Run Check/Fix Blocks to check for mesh problems automatically and repair them if possible.
3. Click Apply.
4. Enable Pre-Mesh.

## **Blocking Pre-Mesh**

- The Mesh dialog will appear, asking if you want to recompute the mesh.
5. Click Yes in the Mesh dialog to compute the initial mesh.
  6. Disable Vertices and Edges.

### **Verify and Save the Mesh and Blocking**

1. Convert the mesh to unstructured format.  
Blocking Pre-Mesh Convert to Unstruct Mesh
2. Save the blocking file (2D-Flat plate -geometry-final.blk).

File > Blocking > Save Blocking As...

This block file can be loaded in a future session (File > Blocking > Open Blocking...) for additional modification or to mesh a similar geometry. Save each blocking to a separate file instead of overwriting a previous one. In more complex models, you may have to back track and load a previous blocking.

3. Save the project file (2D-flat plate-geometry-final.prj).  
File > Save Project As...  
This will save all the files—tetin, blocking, and unstructured mesh.
4. Exit the current session.  
File > Exit  
Save your ICEM CFD file in your working directory.

### **Setting Up the CFD Simulation in ANSYS FLUENT**

Now that you have created a computational mesh for the elbow geometry, you can proceed to setting up a CFD analysis using **ANSYS FLUENT**.

1. Start **ANSYS FLUENT**.

In the **ANSYS Workbench Project Schematic**, double-click the **Setup** cell in the elbow fluid flow analysis system. You can also right-click on the **Setup** cell to display the context menu where you can select the **Edit** option.

When **ANSYS FLUENT** is first started, **FLUENT Launcher** is displayed, allowing you to view and/or set certain **ANSYS FLUENT** start-up options.

*ANSYS FLUENT Launcher allows you to decide which version of ANSYS FLUENT you will use, based on your geometry and on your processing capabilities.*

- (a) Ensure that the proper options are enabled.

Note that the **Dimension** setting is already filled in and cannot be changed, since **ANSYS FLUENT** automatically sets it based on the mesh or geometry for the current system.

- i. Make sure that **Serial** from the **Processing Options** list is enabled.
- ii. Make sure that the **Display Mesh After Reading**, **Embed Graphics Windows**, and **Workbench Color Scheme** options are enabled.

(b) Click **OK** to launch **ANSYS FLUENT**.

2. Set some general settings for the CFD analysis.

*Select **General** in the navigation pane to perform the mesh-related activities and to choose a solver.*

### **General**

(a) Change the units for length.

Since we want to specify and view values based on a unit of length in inches from within **ANSYS FLUENT**, change the units of length within **ANSYS FLUENT** from meters (the default) to inches.

### **General** → **Units...**

This displays the **Set Units** dialog box.

i. Select **length** in the **Quantities** list.

ii. Select **in** in the **Units** list.

iii. Close the dialog box.

*Now, all subsequent inputs that require a value based on a unit of length can be specified in inches rather than meters.*

(b) Check the mesh.

### **General** → **Check**

***ANSYS FLUENT** will report the results of the mesh check in the console.*

3. Set up your models for the CFD simulation.

### **Models**

(a) Enable heat transfer by activating the energy equation.

### **Models** → **Energy** → **Edit...**

*You can also double-click a list item in order to open the corresponding dialog box.*

i. Enable the **Energy Equation** option.

ii. Click **OK** to close the **Energy** dialog box.

(b) Enable the **k- $\epsilon$**  turbulence model.

### **Models** → **Viscous** → **Edit...**

i. Select **k-epsilon** from the **Model** list.

*The **Viscous Model** dialog box will expand.*



- ii. Select **Realizable** from the **k-epsilon Model** list.
  - iii. Click **OK** to accept the model and close the **Viscous Model** dialog box.
4. Set up your materials for the CFD simulation.

### Materials

- (e) Create a new material called **air** using the **Create/Edit Materials** dialog box.
  - (f) **Materials** → **Fluid** → **Create/Edit...**
    - i. Enter **air** for **Name**.
    - ii. specify the properties
    - iii. Click **Change/Create**.
5. Set up the cell zone conditions for the CFD simulation.

### Cell Zone Conditions

- (a) Set the cell zone conditions for the fluid zone.
    - i. Select **fluid** in the **Zones** list in the **Cell Zone Conditions** task page, then click the **Edit...** button to open the **Fluid** dialog box.
 

*You can also double-click a list item in order to open the corresponding dialog box.*
    - ii. In the **Fluid** dialog box, select **water** from the **Material Name** drop-down list.
    - iii. Click **OK** to close the **Fluid** dialog box.
6. Set up the boundary conditions for the CFD analysis.

### Boundary Conditions

- (a) Set the boundary conditions at the cold inlet (**velocity-inlet-large**).
 

**Boundary Conditions** → **velocity-inlet-large** → **Edit...**

    - i. Select **Components** from the **Velocity Specification Method** drop-down list.
 

*The **Velocity Inlet** dialog box will expand.*
    - ii. Enter 0.4 m/s for **X-Velocity**.
    - iii. Retain the default value of 0 m/s for both **Y-Velocity** and **Z-Velocity**.
7. Set up solution parameters for the CFD simulation.

*In the steps that follow, you will set up and run the calculation using the task pages listed under the **Solution** heading in the navigation pane.*

- (a) Change the convergence criteria for the continuity equation residual.

**Monitors** → **Residuals** → **Edit...**

- i. Make sure that **Plot** is enabled in the **Options** group box.
  - ii. Enter 1e-05 for the **Absolute Criteria** of **continuity**, as shown in the **Residual Monitor** dialog box.
  - iii. Click **OK** to close the **Residual Monitors** dialog box.
- (b) Initialize the flow field, using the boundary conditions settings at the cold inlet (velocity-**inlet-large**) as a starting point.

**Solution Initialization**

- i. Select **velocity-inlet-large** from the **Compute From** drop-down list.
  - ii. Enter 1.2 m/s for **Y Velocity** in the **Initial Values** group box.
  - iii. Click **Initialize**.
- (c) Check to see if the case conforms to best practices.

**Run Calculation** → **Check Case**

8. Calculate a solution.
- (a) Start the calculation by requesting 250 iterations.

**Run Calculation**

- i. Enter 250 for **Number of Iterations**.
- ii. Click **Calculate**.

*As the calculation progresses, the residuals will be plotted in the graphics window*

9. View the files generated by **ANSYS Workbench**.

**Result:**

**Viva Questions:**

1. Differentiate the cone and wedge?
2. What is the Mach reflection?
3. How the pressure and velocity changes in cone?

# **EXPERIMENT – 11**

## **FLOW THROUGH DIFFUSER**

**Aim:** To visualize the flow through diffuser at different velocities.

**Software used:** ICEM CFD AND FLUENT

We will create the geometry and mesh in ICEM CFD, read the mesh into FLUENT, and solve the flow problem.

- Create or import Geometry
- Block the geometry
- Associate entities to the geometry.
- Apply mesh parameters.
- Generate the initial mesh.
- Export mesh

### **Start ICEM CFD**

#### **Create Geometry in ICEM CFD**

- 1) Close any previous files, have blank screen in ICEM.
- 2) Create new project file for the pipe in ICEM:
  - a) File, New Project.
  - b) Select folders in turn: Desktop, CFX Training Files – Copy, Flat plate.
  - c) Save project with name “Flat plate” in folder “Flat plate”.
- 3) We are going to create some points now:
  - a) Select Geometry tab at top of screen.
  - b) Select “Create Point” option.
  - c) In the bottom left are various options for creating the point. Select the second “XYZ” option.
  - d) Input the numbers  $X=0$ ,  $Y=0$ ,  $Z=0$ .
  - e) Press “Apply” to create a point on screen
  - f) Note - If the point does not appear, then check that points are displayed in the tree, and use the “Fit Window” option in the top left to zoom the screen correctly.

#### **Create points**

Explicit location > specify x and y location and click ok

Similarly create required points

Note that for a 2D problem, the z-coordinate can always be left to the default value of 0.

#### **Create curves**

Under “Geometry” tab, select “Create and modify curve”.

- b) Under “Create and modify curve” option panel on bottom left, select 3rd option “  
Select two points on the screen and select ok Create remaining curves

## **Create surface**

Simple surface – from 2-4 curves  
To check surface click on solid frame

## **Label the curves**

Give names to the INLET, OUTLET and WALL surfaces:

- a) Under “Geometry” tree, highlight only “Surfaces”.
- b) Right click “Parts”, Create Part.
- c) Part = “INLET”.
- d) Entities: click arrow icon, left click surface (near surface, around origin) to highlight it, middle click to save selection, right click to de-select mode.
- e) Note that the INLET part has been created, and can be turned on and off independently using the tree view.
- f) Part = “OUTLET”, repeat step to identify an outlet at the far end.
- g) Part = “WALL”, repeat step to identify the wall.
- h) Notes: “GEOM” now contains only 0D and 1D elements; points and lines.

## **Block the Geometry**

The geometry and part information has already been defined for this tutorial. You will create the initial block in this step.

1. Create the initial block.

Blocking > Create Block > Initialize Blocks

- a. Enter FLUID in the Part field.
- b. Select 2D Planar in the Type drop-down list.
- c. Click Apply.

## **Associate Entities to the Geometry**

You will associate the edges of the blocking to the curves of the CAD geometry in this step. First select the edges and then the curves to which you want to associate them.

1. Associate the inlet, the left-most end of the large flat plate.
  - a. Ensure that Project Vertices is disabled.
  - b. Click (Select edge(s)) and select edge 13–41. Click the middle-mouse button to accept the selection.
  - c. Click (Select compcurve(s)) and select CURVES/1 . Click the middle-mouse button to accept the selection.
  - d. Click Apply.

The associated edge will be colored green.

## **Generate the Initial Mesh**

Blocking > Pre-Mesh Params > Update Sizes

1. Retain the selection of Update All in the Method list.

Note

This will automatically determine the number of nodes on the edges from the mesh sizes set on the curves.

2. Select Run Check/Fix Blocks to check for mesh problems automatically and repair them if possible.
3. Click Apply.
4. Enable Pre-Mesh.

## Blocking Pre-Mesh

The Mesh dialog will appear, asking if you want to recompute the mesh.

5. Click Yes in the Mesh dialog to compute the initial mesh.
6. Disable Vertices and Edges.

## Verify and Save the Mesh and Blocking

1. Convert the mesh to unstructured format.  
Blocking Pre-Mesh Convert to Unstruct Mesh
2. Save the blocking file (2D-Flat plate -geometry-final.blk).

File > Blocking > Save Blocking As...

This block file can be loaded in a future session (File > Blocking > Open Blocking...) for additional modification or to mesh a similar geometry. Save each blocking to a separate file instead of overwriting previous one. In more complex models, you may have to back track and load a previous blocking.

3. Save the project file (2D-flat plate-geometry-final.prj).

File > Save Project As...

This will save all the files—tetin, blocking, and unstructured mesh.

4. Exit the current session.

File > Exit

Save your ICEM CFD file in your working directory.

## Setting Up the CFD Simulation in ANSYS FLUENT

Now that you have created a computational mesh for the elbow geometry, you can proceed to setting up a CFD analysis using **ANSYS FLUENT**.

1. Start **ANSYS FLUENT**.

In the **ANSYS Workbench Project Schematic**, double-click the **Setup** cell in the elbow fluid flow analysis system. You can also right-click on the **Setup** cell to display the context menu where you can select the **Edit** option.

When **ANSYS FLUENT** is first started, **FLUENT Launcher** is displayed, allowing you to view and/or set certain **ANSYS FLUENT** start-up options.

*ANSYS FLUENT Launcher allows you to decide which version of ANSYS FLUENT you will use, based on your geometry and on your processing capabilities.*

Ensure that the proper options are enabled.

Note that the **Dimension** setting is already filled in and cannot be changed, since **ANSYS FLUENT** automatically sets it based on the mesh or geometry for the current system

Make sure that serial from the processing options list is enabled it. Make sure that the display Mesh after reading, Embed Graphics windows, and work bench color Scheme options are enabled .

Click **OK** to launch **ANSYS FLUENT**.

2. Set some general settings for the CFD analysis.

Select **General** in the navigation pane to perform the mesh-related activities and to choose a solver.

## General

- (a) Change the units for length.

Since we want to specify and view values based on a unit of length in inches from within **ANSYS FLUENT**, change the units of length within **ANSYS FLUENT** from meters (the default) to inches.

### General → Units...

This displays the **Set Units** dialog box.

- i. Select **length** in the **Quantities** list.
- ii. Select **in** in the **Units** list.
- iii. Close the dialog box.

- (b) Check the mesh.

### General → Check

*ANSYS FLUENT* will report the results of the mesh check in the console.

3. Set up your models for the CFD simulation.

## Models

- (a) Enable heat transfer by activating the energy equation.

### Models → Energy → Edit...

*You can also double-click a list item in order to open the corresponding dialog box.*

- i. Enable the **Energy Equation** option.
- ii. Click **OK** to close the **Energy** dialog box.

- (b) Enable the **k- $\epsilon$**  turbulence model.

### Models → Viscous → Edit...

- i. Select **k-epsilon** from the **Model** list.

*The **Viscous Model** dialog box will expand.*

- ii. Select **Realizable** from the **k-epsilon Model** list.
- iii. Click **OK** to accept the model and close the **Viscous Model** dialog box.

4. Set up your materials for the CFD simulation.

## Materials

(g) Create a new material called **air** using the **Create/Edit Materials** dialog box.

(h) **Materials** → **Fluid** → **Create/Edit...**

i. Enter **air** for **Name**.

ii. specify the properties

iii. Click **Change/Create**.

5. Set up the cell zone conditions for the CFD simulation.

## Cell Zone Conditions

(a) Set the cell zone conditions for the fluid zone.

i. Select **fluid** in the **Zones** list in the **Cell Zone Conditions** task page, then click the **Edit...** button to open the **Fluid** dialog box.

*You can also double-click a list item in order to open the corresponding dialog box.*

ii. In the **Fluid** dialog box, select **water** from the **Material Name** drop-down list.

iii. Click **OK** to close the **Fluid** dialog box.

6. Set up the boundary conditions for the CFD analysis.

## Boundary Conditions

(a) Set the boundary conditions at the cold inlet (**velocity-inlet-large**).

**Boundary Conditions** → **velocity-inlet-large** → **Edit...**

i. Select **Components** from the **Velocity Specification Method** drop-down list.

*The **Velocity Inlet** dialog box will expand.*

ii. Enter 0.4 m/s for **X-Velocity**.

iii. Retain the default value of 0 m/s for both **Y-Velocity** and **Z-Velocity**.

7. Set up solution parameters for the CFD simulation.

*In the steps that follow, you will set up and run the calculation using the task pages listed under the **Solution** heading in the navigation pane.*

(a) Change the convergence criteria for the continuity equation residual.

**Monitors** → **Residuals** → **Edit...**

i. Make sure that **Plot** is enabled in the **Options** group box.

ii. Enter 1e-05 for the **Absolute Criteria** of **continuity**, as shown in the **Residual Monitor** dialog box.

iii. Click **OK** to close the **Residual Monitors** dialog box.

***Note:** By default, all variables will be monitored and checked by ANSYS FLUENT as a means to determine the convergence of the solution.*

(b) Initialize the flow field, using the boundary conditions settings at the cold inlet (velocity-**inlet-large**) as a starting point.

#### **Solution Initialization**

i. Select **velocity-inlet-large** from the **Compute From** drop-down list.

ii. Enter 1.2 m/s for **Y Velocity** in the **Initial Values** group box.

***Note:** While an initial **X Velocity** is an appropriate guess for the horizontal section, the addition of a **Y Velocity** component will give rise to a better initial guess throughout the entire elbow.*

iii. Click **Initialize**.

(c) Check to see if the case conforms to best practices.

#### **Run Calculation → Check Case**

8. Calculate a solution.

(a) Start the calculation by requesting 250 iterations.

#### **Run Calculation**

i. Enter 250 for **Number of Iterations**.

ii. Click **Calculate**.

*As the calculation progresses, the residuals will be plotted in the graphics window*

9. View the files generated by **ANSYS Workbench**.

#### **Results:**

#### **Viva Questions:**

1. Differentiate the nozzle and diffuser?
2. List the types of diffuser.
3. How the pressure and velocity changes in diffuser?



## EXPERIMENT – 12

### FLOW THROUGH SUPERSONIC INTAKE

**Aim:** To visualize the flow through Supersonic intake at different velocities and observe the velocity changes for laminar and turbulent flows.

**Software used:** ICEM CFD AND FLUENT

We will create the geometry and mesh in ICEM CFD, read the mesh into FLUENT, and solve the flow problem.

- Create or import Geometry
- Block the geometry
- Associate entities to the geometry.
- Apply mesh parameters.
- Generate the initial mesh.
- Export mesh

#### Start ICEM CFD

##### Create Geometry in ICEM CFD

- 1) Close any previous files, have blank screen in ICEM.
- 2) Create new project file for the pipe in ICEM:
  - a) File, New Project.
  - b) Select folders in turn: Desktop, CFX Training Files – Copy, Flat plate.
  - c) Save project with name “Flat plate” in folder “Flat plate”.
- 3) We are going to create some points now:
  - a) Select Geometry tab at top of screen.
  - b) Select “Create Point” option.
  - c) In the bottom left are various options for creating the point. Select the second “XYZ” option.
  - d) Input the numbers  $X=0$ ,  $Y=0$ ,  $Z=0$ .
  - e) Press “Apply” to create a point on screen
  - f) Note - If the point does not appear, then check that points are displayed in the tree, and use the “Fit Window” option in the top left to zoom the screen correctly.

##### Create points

Explicit location > specify x and y location and click ok

Similarly create required points

Note that for a 2D problem, the z-coordinate can always be left to the default value of 0.

##### Create curves

Under “Geometry” tab, select “Create and modify curve”.

- b) Under “Create and modify curve” option panel on bottom left, select 3rd option “ Select two points on the screen and select ok Create remaining curves

## **Create surface**

Simple surface – from 2-4 curves  
To check surface click on solid frame

## **Label the curves**

Give names to the INLET, OUTLET and WALL surfaces:

- a) Under “Geometry” tree, highlight only “Surfaces”.
- b) Right click “Parts”, Create Part.
- c) Part = “INLET”.
- d) Entities: click arrow icon, left click surface (near surface, around origin) to highlight it, middle click to save selection, right click to de-select mode.
- e) Note that the INLET part has been created, and can be turned on and off independently using the tree view.
- f) Part = “OUTLET”, repeat step to identify an outlet at the far end.
- g) Part = “WALL”, repeat step to identify the wall.
- h) Notes: “GEOM” now contains only 0D and 1D elements; points and lines.

## **Block the Geometry**

The geometry and part information has already been defined for this tutorial. You will create the initial block in this step.

1. Create the initial block.

Blocking > Create Block > Initialize Blocks

- a. Enter FLUID in the Part field.
- b. Select 2D Planar in the Type drop-down list.
- c. Click Apply.

## **Associate Entities to the Geometry**

You will associate the edges of the blocking to the curves of the CAD geometry in this step. First select the edges and then the curves to which you want to associate them.

1. Associate the inlet, the left-most end of the large flat plate.
  - a. Ensure that Project Vertices is disabled.
  - b. Click (Select edge(s)) and select edge 13–41. Click the middle-mouse button to accept the selection.
  - c. Click (Select compcurve(s)) and select CURVES/1 . Click the middle-mouse button to accept the selection.
  - d. Click Apply.

The associated edge will be colored green.

## **Generate the Initial Mesh**

Blocking > Pre-Mesh Params > Update Sizes

1. Retain the selection of Update All in the Method list.
2. Select Run Check/Fix Blocks to check for mesh problems automatically and repair them if possible.
3. Click Apply.
4. Enable Pre-Mesh.

## Blocking Pre-Mesh

The Mesh dialog will appear, asking if you want to recompute the mesh.

5. Click Yes in the Mesh dialog to compute the initial mesh.
6. Disable Vertices and Edges.

## Verify and Save the Mesh and Blocking

1. Convert the mesh to unstructured format.  
Blocking Pre-Mesh Convert to Unstruct Mesh
2. Save the blocking file (2D-Flat plate -geometry-final.blk).

File > Blocking > Save Blocking As...

3. Save the project file (2D-flat plate-geometry-final.prj).  
File > Save Project As...  
This will save all the files—tetin, blocking, and unstructured mesh.

4. Exit the current session.  
File > Exit  
Save your ICEM CFD file in your working directory.

## Setting Up the CFD Simulation in ANSYS FLUENT

Now that you have created a computational mesh for the elbow geometry, you can proceed to setting up a CFD analysis using **ANSYS FLUENT**.

1. Start **ANSYS FLUENT**.

In the **ANSYS Workbench Project Schematic**, double-click the **Setup** cell in the elbow fluid flow analysis system. You can also right-click on the **Setup** cell to display the context menu where you can select the **Edit** option.

When **ANSYS FLUENT** is first started, **FLUENT Launcher** is displayed, allowing you to view and/or set certain **ANSYS FLUENT** start-up options.

**FLUENT** automatically sets it based on the mesh or geometry for the current system.

- i. Make sure that **Serial** from the **Processing Options** list is enabled.

- ii. Make sure that the **Display Mesh After Reading**, **Embed Graphics Windows**, and **Workbench Color Scheme** options are enabled.

- (b) Click **OK** to launch **ANSYS FLUENT**.

2. Set some general settings for the CFD analysis.

Select **General** in the navigation pane to perform the mesh-related activities and to choose a solver.

## General

- (a) Change the units for length.

Since we want to specify and view values based on a unit of length in inches from within **ANSYS FLUENT**, change the units of length within **ANSYS FLUENT** from meters (the default) to inches.

### General → Units...

This displays the **Set Units** dialog box.

- i. Select **length** in the **Quantities** list.
- ii. Select **in** in the **Units** list.
- iii. Close the dialog box.

*Now, all subsequent inputs that require a value based on a unit of length can be specified in inches rather than meters.*

- (b) Check the mesh.

### General → Check

*ANSYS FLUENT will report the results of the mesh check in the console.*

3. Set up your models for the CFD simulation.

## Models

- (a) Enable heat transfer by activating the energy equation.

### Models → Energy → Edit...

*You can also double-click a list item in order to open the corresponding dialog box.*

- i. Enable the **Energy Equation** option.
- ii. Click **OK** to close the **Energy** dialog box.

- (b) Enable the **k- $\epsilon$**  turbulence model.

### Models → Viscous → Edit...

- i. Select **k-epsilon** from the **Model** list.

*The **Viscous Model** dialog box will expand.*

- ii. Select **Realizable** from the **k-epsilon Model** list.
- iii. Click **OK** to accept the model and close the **Viscous Model** dialog box.

4. Set up your materials for the CFD simulation.

## Materials

- (i) Create a new material called **air** using the **Create/Edit Materials** dialog box.
- (j) **Materials** → **Fluid** → **Create/Edit...**

- i. Enter **air** for **Name**.
- ii. Specify the properties
- iii. Click **Change/Create**.

5. Set up the cell zone conditions for the CFD simulation.

## Cell Zone Conditions

- (a) Set the cell zone conditions for the fluid zone.
  - i. Select **fluid** in the **Zones** list in the **Cell Zone Conditions** task page, then click the **Edit...** button to open the **Fluid** dialog box.

*You can also double-click a list item in order to open the corresponding dialog box.*

- ii. In the **Fluid** dialog box, select **water** from the **Material Name** drop-down list.
- iii. Click **OK** to close the **Fluid** dialog box.

6. Set up the boundary conditions for the CFD analysis.

## Boundary Conditions

- (a) Set the boundary conditions at the cold inlet (**velocity-inlet-large**).

## Boundary Conditions

 → **velocity-inlet-large** → **Edit...**

- i. Select **Components** from the **Velocity Specification Method** drop-down list.

*The **Velocity Inlet** dialog box will expand.*

- ii. Enter 0.4 m/s for **X-Velocity**.
- iii. Retain the default value of 0 m/s for both **Y-Velocity** and **Z-Velocity**.

7. Set up solution parameters for the CFD simulation.

- (a) Change the convergence criteria for the continuity equation residual.

## Monitors

 → **Residuals** → **Edit...**

- i. Make sure that **Plot** is enabled in the **Options** group box.
- ii. Enter 1e-05 for the **Absolute Criteria** of **continuity**, as shown in the **Residual Monitor** dialog box.
- iii. Click **OK** to close the **Residual Monitors** dialog box.

- (b) Initialize the flow field, using the boundary conditions settings at the cold inlet (velocity-**inlet-large**) as a starting point.

**Solution Initialization**

- i. Select **velocity-inlet-large** from the **Compute From** drop-down list.
- ii. Enter 1.2 m/s for **Y Velocity** in the **Initial Values** group box.
- iii. Click **Initialize**.

- (c) Check to see if the case conforms to best practices.

**Run Calculation → Check Case**

8. Calculate a solution.

- (a) Start the calculation by requesting 250 iterations.

**Run Calculation**

- i. Enter 250 for **Number of Iterations**.
  - ii. Click **Calculate**.
9. View the files generated by **ANSYS Workbench**.

**Results:**

**Viva Questions:**

1. What is supersonic intake?
2. List the types of intake.
3. How the pressure and velocity changes in supersonic intake?