COMPUTATIONAL AERODYNAMICS LAB MANUAL

Year	2019 - 2020
Course Code	AAE109
Regulations	R16
Class	VI Semester
Branch	Aeronautical Engineering

Prepared by

Ms. D.Anitha

Mr. G.Satya Dileep

Aeronautical Engineering

INSTITUTE OF AERONAUTICAL ENGINEERING

(AUTONOMOUS)

Dundigal – 500 043, Hyderabad

INSTITUTE OF AERONAUTICAL ENGINEERING

(AUTONOMOUS)

Dundigal, Hyderabad - 500 043

Certificate

This is to certify that it is a Bonafide record of practical work done by Sri/Kum. ____ bearing Roll No. ________________________ of _____________ class ____ branch in the Aerodynamics laboratory during the academic year _____________________under our supervision.

Head of the Department **Lecture In-Charge**

External Examiner **Internal Examiner**

Computational Aerodynamics Lab Syllabus

COMPUTATIONAL AERODYNAMICS LABORATORY

OBJECTIVE:

The objective of this lab complements the computational Aerodynamics course. Students will gain experience in computing aerodynamic problems and understanding flow physics over flat plate, pipe, cylinder, over a wedge and flow over an airfoil. They can gain knowledge in estimating flow analysis for different mach numbers in determining the pressure coefficients over different structural objects and can find lift and drag counters.

OUTCOMES:

After completing this course the student must demonstrate the knowledge and ability to:

1. Understand the behavior of flows around different structured objects.

2. Implement the computational fluid dynamic and computational aerodynamic fundamentals by using advanced solvers.

3. Explain the usage of modern tools like ICEM-CFD& FLUENT.

4. Understand the flow properties of flat plate to demonstrate Reynolds number.

5. Understand the Aerodynamic properties for flow through circular pipe.

6. Understand the Aerodynamic properties foe flow through cylinder.

7. Observe the properties at separation region and wake region of circular cylinder at different Reynolds numbers

8. Understand the aerodynamic properties of supersonic flow over a wedge.

9. Understand the aerodynamic properties of flow over an airfoil.

10. Differentiate the flow properties around symmetrical and cambered airfoil.

11. Analyze the errors and cause of errors in the computational analysis.

12. Analyze the coefficient of pressure, lift, drag and moment for different bodies for different flow conditions.

13. Determine the shock wave around cone and wedges for supersonic flow conditions.

14. Observe flow properties and compare the computation results with experimental results.

15. Observe the shock waves and 3D relieving effect around the cone at supersonic Mach number.

16. Solve the One dimensional wave equation using explicit method of lax equations using finite difference method

17. Solve the One dimensional heat conduction equation using explicit method

18. Generate the Algebraic and Elliptic grids for computational domains.

INSTITUTE OF AERONAUTICAL ENGINEERING

Dundigal, Hyderabad - 500 043

PROGRAM OUTCOMES

PO1	**Engineering knowledge**: Apply the knowledge of mathematics, science, engineering fundamentals, and an engineering specialization to the solution of complex engineering problems.
PO2	**Problem analysis**: Identify, formulate, review research literature, and analyze complex engineering problems reaching substantiated conclusions using first principles of mathematics, natural sciences, and engineering sciences.
PO3	**Design/development of solutions**: Design solutions for complex engineering problems and design system components or processes that meet the specified needs with appropriate consideration for the public health and safety, and the cultural, societal, and environmental considerations.
PO4	**Conduct investigations of complex problems**: Use research-based knowledge and research methods, including design of experiments, analysis and interpretation of data, and synthesis of the information to provide valid conclusions.
PO5	**Modern tool usage:** Create, select, and apply appropriate techniques, resources, and modern engineering and IT tools including prediction and modeling to complex engineering activities with an understanding of the limitations.

PROGRAM SPECIFIC OUTCOMES

AERONAUTICAL ENGINEERING	
PSO1	**Professional skills:** Able to utilize the knowledge of aeronautical/aerospace engineering in innovative, dynamic and challenging environment for design and development of new products.
PSO2	**Problem solving skills**: imparted through an understanding of experimental facilities and techniques, to solve practical, design and analysis problems of components to complete the challenge of airworthiness for flightvehicles.
PSO3	**Practical implementation and testing skills**: Providing different types of in house and training and industry practice to fabricate and test and develop the products with more innovative technologies.
PSO4	**Successful career and entrepreneurship**: To prepare the students with broad aerospace knowledge to design and develop systems and subsystems of aerospace and allied systems and becometechnocrats.

ATTAINMENT OF PROGRAM OUTCOMES & PROGRAM SPECIFIC OUTCOMES

Expt. No.	Program Outcomes Attained	Program Specific Outcomes Attained
1	PO1, PO2, PO3	PSO1, PSO2, PSO3
2	PO1, PO2, PO3	PSO1, PSO2, PSO3
3	PO1, PO2	PSO1, PSO2, PSO3
4	PO1, PO2, PO4	PSO1, PSO2, PSO3
5	PO1, PO2, PO3	PSO1, PSO2, PSO3
6	PO1, PO3, PO4,	PSO1, PSO2, PSO3
7	PO1, PO2	PSO1, PSO2, PSO3
8	PO1, PO2	PSO1, PSO2, PSO3
9	PO1, PO2, PO3	PSO1, PSO2, PSO3
10	PO1, PO3, PO2	PSO1, PSO2, PSO3
11	PO1, PO3, PO4	PSO1, PSO2, PSO3
12	PO1, PO3, PO2	PSO1, PSO2, PSO3

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 behaviour 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.

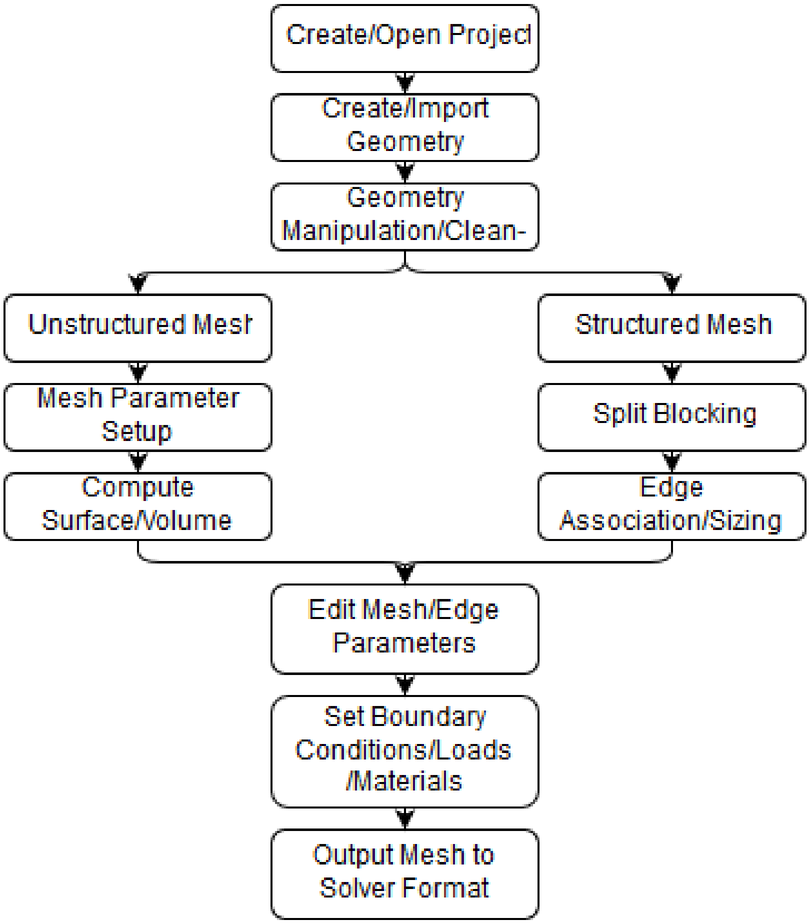
EXPERIMENT - 2

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 a 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 users requirements. It is a 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 maybe 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 surface
BLUE: curve shared by more than two surface.
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, visualisation 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. Initialise 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

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.

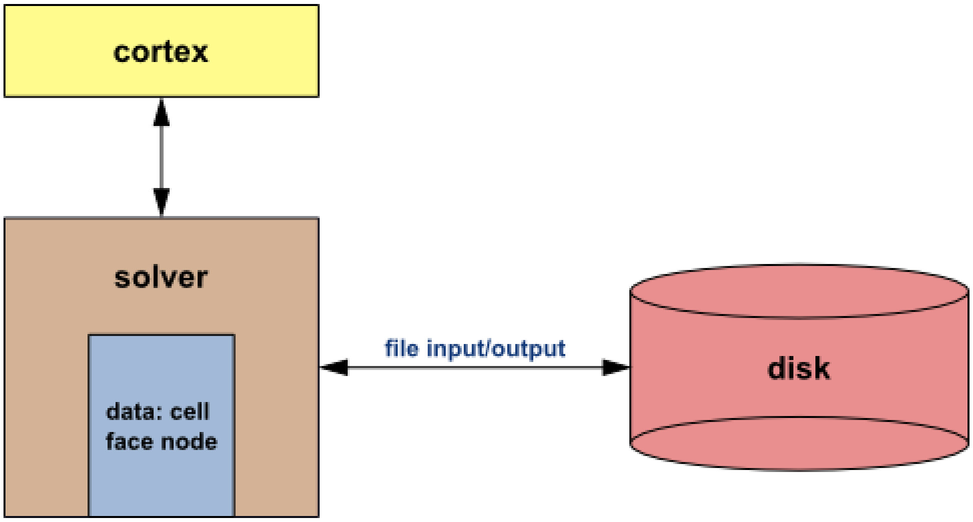


Figure 1.1: Parallel ANSYS Fluent Architecture

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.

EXPERIMENT - 4

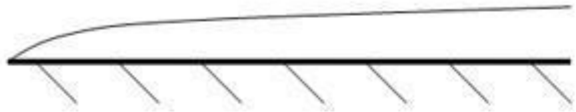
FLOW OVER A FLAT PLATE

Solve this problem in FLUENT. Validate the solution by plotting the y+ values at the plate. Also plot the velocity profile at x = 1m. Compare the accuracy of your results from FLUENT with empirical correlations.

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.

$$q_x'' = h_x(T_{plate} - T_\infty)$$



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.

Step 1: Create Geometry in ICEM CFD

Start ICEM CFD & Select Solver

Main Menu > Solver > FLUENT 5/6

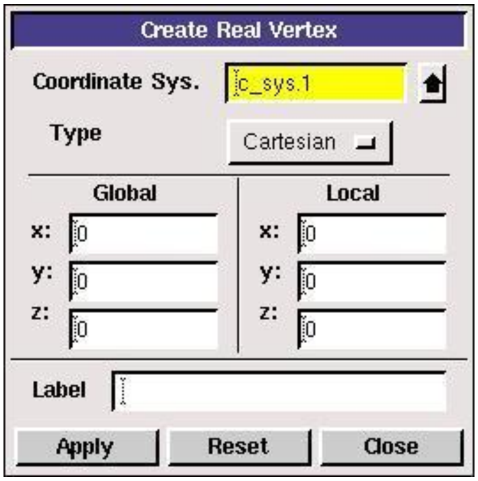
Verify this has been done by looking in the *Transcript Window* where you should see:

Create Vertices

We will treat this problem as a 2-dimensional problem by assuming that the plate is infinitely wide. Let's begin by creating the vertices that define our flow region.

Create the vertex at the lower-left corner of the rectangle:

Next to ***x:***, enter value 0. Next to ***y:***, enter value 0. Next to ***z:***, enter value 0 (these values should be defaults). Click ***Apply***.


Create Real Vertex

Coordinate Sys. c_sys.1

Type Cartesian

Global		Local	
x:	0	x:	0
y:	0	y:	0
z:	0	z:	0

Label

This creates the vertex (0,0,0) which is displayed in the graphics window.

In the *Transcript*window, ICEM CFD reports that it "Created vertex: vertex.1". The vertices are numbered vertex.1, vertex.2 etc. in the order in which they are created.

Repeat this process to create three more vertices:

Vertex 2: (1,0,0)
Vertex 3: (1,1,0)
Vertex 4: (0,1,0)

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

Operation Toolpad > Global Control > Fit to Window Button

This fits the four vertices of the rectangle we have created to the size of the *Graphics Window*.

Create Edges

An edge is created by selecting two vertices and creating a line between them.

Operation Toolpad > Geometry Command Button > Edge Command Button > Create Edge

Click the ***up arrow button*** next to the vertices box in the *Create Straight Edge* window.

This brings up a list of vertices, from which vertices 1 and 2 can be selected. Select ***Vertex.1*** and ***Vertex.2***. The push the ***right arrow button*** ---> to bring these vertices into the *Picked* column.

Click ***Close.*** Then click ***Apply*** in the *Create Straight Edge* window to create this edge.

Alternately, these vertices can be selected by holding down the ***Shift*** button and clicking on the corresponding vertices. As each vertex is picked, it will appear red in the *Graphics Window*. Then let go of the ***Shift*** button and click ***Apply*** in the *Create Straight Edge* window.

Repeat this process to create edges between vertices 2 & 3, vertices 3 & 4, and vertices 4 & 1.

Create Face

Operation Toolpad > Geometry Command Button > Face Command Button > Form Face

To form a face out of the area enclosed by the four lines, we need to select the four edges that enclose this area. This is done in much the same way as when we selected the vertices.

Click the ***up arrow button*** next to the vertices box in the *Create Face From Wireframe* window. Then push the ***All right arrow button*** All-> to bring these vertices into the *Picked* column.

Click ***Close***. Then click ***Apply*** in the *Create Face From Wireframe* window to create the face. The edges and vertices will become blue, indicating that they now form a face.

Save

Save your *ICEM CFD* file in your working directory.

Main Menu > File > Save As... > Browse...

Find your working directory and save your *ICEM CFD* file there. Make sure to enter the file name, plate.dbs, in the Selection box in addition to the path.

Step 2: Mesh Geometry in ICEM CFD

We'll now create a mesh on the rectangular face with 100 divisions in the vertical direction and 30 divisions in the horizontal direction. We'll first mesh the four edges and then the face. The desired grid spacing is specified through the edge mesh.

Mesh Edges

Operation Toolpad > Mesh Command Button > Edge Command Button > Mesh Edges

Mesh Strategy

In creating this mesh, it is desirable to have more cells near the plate (Edge 1) because we want to resolve the turbulent boundary layer, which is very thin compared to the height of the flow field.

Click the ***up arrow button*** ⬆ next to the *Edges* box in the *Mesh Edges* window. Select edge ***Edge.2***. Then push the ***right arrow button*** ---> to bring this vertex into the *Picked* column. Notice that the arrow on the selected edge should be pointing upwards. An upwards pointing arrow indicates the direction of closely spaced nodes to widely spaced nodes

The proper arrow direction is necessary to ensure a proper mesh. Select ***Edge.4*** in the *Mesh Edges* window. The arrow on this edge is pointing downwards, which needs to be changed. ***Shift + Middle-click*** on the selected edge to change the direction of the arrow to upward.

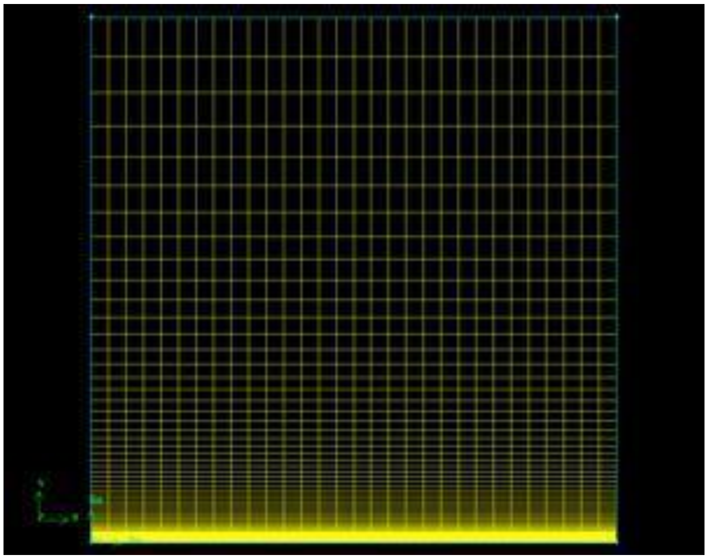
Under *Type*, select ***Successive Ratio***, if it is not already selected. Set *Ratio* to 1.08. Under *Spacing*, select ***Interval Count***. Set Interval Count to 100 and then click ***Apply***.

Select ***Edge.1*** and ***Edge.3*** in the *Mesh Edges* Window. The direction of the arrows on these edges is irrelevant because the divisions will be the same length. Leave the ***Successive Ratio*** set to 1 and set the ***Inteval Count*** to 30. Click ***Apply***.

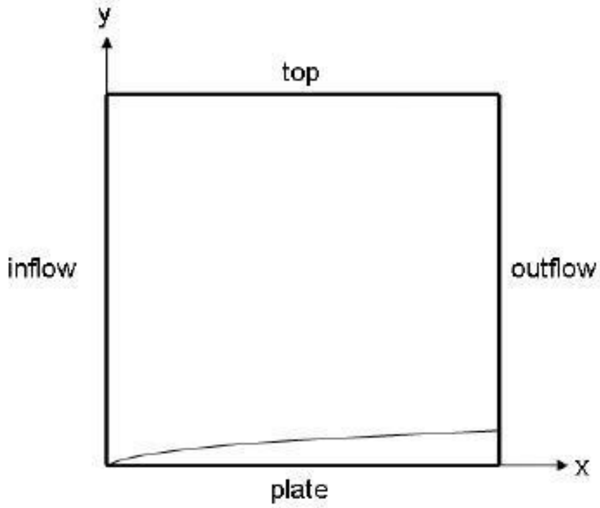
Mesh Face

Operation Toolpad > Mesh Command Button > Face Command Button > Mesh Faces

Shift left-click on the face or use the up arrow next to *Faces* to select the face. Click ***Apply***.



Step 3: Specify Boundary Types in ICEM CFD

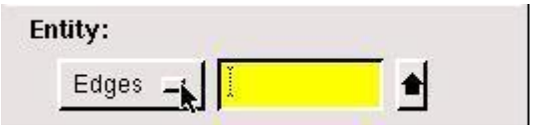


Create Boundary Types

We'll next set the boundary types in *ICEM CFD*. The left edge is the inflow of the flow field, the right edge the outflow, the top edge the open top of the flow field, and the bottom edge the plate.

Operation Toolpad > Zones Command Button > Specify Boundary Types Command Button

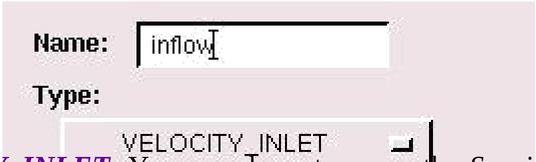
This will bring up the *Specify Boundary Types* window on the *Operation Panel*. We will first specify that the left edge is the inflow. Under *Entity:*, pick ***Edges*** so that *ICEM CFD* knows we want to pick an edge (face is default).


Entity:

Edges

Now select the left edge by ***Shift-clicking*** on it. The selected edge should appear in the yellow box next to the ***Edges*** box as well as the ***Label/Type*** list under the ***Edges*** box.

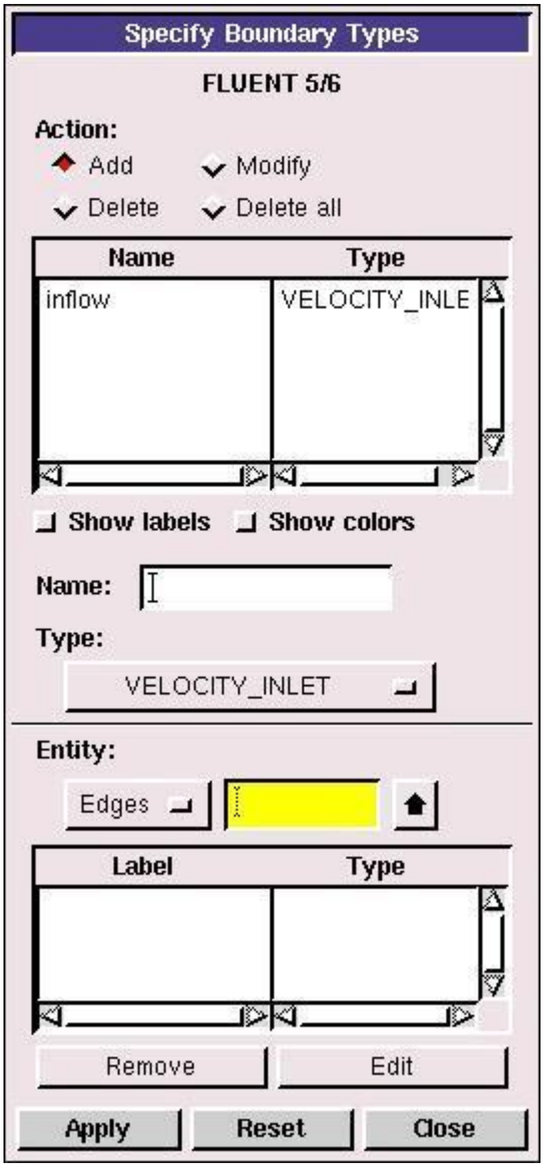
Next to ***Name:***, enter inflow.


Name: inflow

Type: VELOCITY_INLET

For *Type:*, select ***VELOCITY_INLET***. You may have to move the *Specify Boundary Types* box up in order to see the bottom of the list and select ***VELOCITY_INLET***.

Click ***Apply***. You should see the new entry appear under ***Name/Type*** box near the top of the window.


Specify Boundary Types

FLUENT 5/6

Action:
- ● Add
- ○ Modify
- ○ Delete
- ○ Delete all

Name	Type
inflow	VELOCITY_INLE

□ Show labels □ Show colors

Name:

Type: VELOCITY_INLET

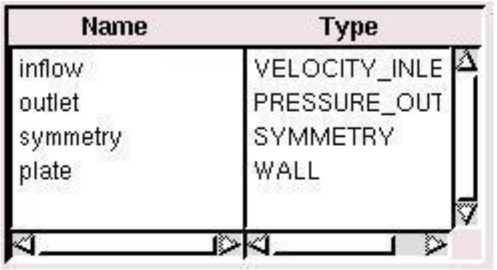
Entity: Edges

Label	Type

Repeat this process for the other three edges according to the following table:

Edge Position	Name	Type
Left	inflow	VELOCITY_INLET
Right	outflow	PRESSURE_OUTLET
Top	top	SYMMETRY
Bottom	plate	WALL

You should have the following edges in the *Name/Type* list when finished:



Name	Type
inflow	VELOCITY_INLE
outlet	PRESSURE_OUT
symmetry	SYMMETRY
plate	WALL

Save and Export

Main Menu > File > Save

Main Menu > File > Export > Mesh...

Type in plate.msh for the ***File Name***:. Select ***Export 2d Mesh*** because this is a 2 dimensional mesh. Click ***Accept***.

It is important to check that plate.msh has been created in your working directory. *ICEM CFD* may periodically fail to write the .msh file. If this should happen, simply try writing the .msh file to another directory and then coping it into your working directory.

Step 4: Set Up Problem in FLUENT

Launch Fluent 6.0

Start > Programs > Fluent Inc > FLUENT 6.0

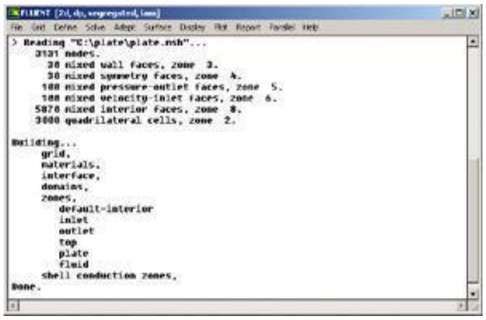
Select the 2ddp version and click Run.

The "2ddp" option is used to select the 2-dimensional, double-precision solver. In the double-precision solver, each floating point number is represented using 64 bits in contrast to the single-precision solver which uses 32 bits. The extra bits increase not only the precision but also the range of magnitudes that can be represented. The downside of using double precision is that it requires more memory.

Import Grid

Main Menu > File > Read > Case...

Navigate to the working directory and select the plate.msh file. This is the mesh file that was created using the preprocessor *ICEM CFD* in the previous step. FLUENT reports the mesh statistics as it reads in the mesh:


FLUENT [2d, dp, segregated, lam]
File Grid Define Solve Adapt Surface Display Plot Report Parallel Help

```
> Reading "C:\plate\plate.msh"...
     3131 nodes.
       30 mixed wall faces, zone  3.
       30 mixed symmetry faces, zone  4.
      100 mixed pressure-outlet faces, zone  5.
      100 mixed velocity-inlet faces, zone  6.
     5870 mixed interior faces, zone  8.
     3000 quadrilateral cells, zone  2.

Building...
      grid,
      materials,
      interface,
      domains,
      zones,
         default-interior
         inlet
         outlet
         top
         plate
         fluid
      shell conduction zones,
Done.
```

Check the number of nodes, faces (of different types) and cells. There are 3000 quadrilateral cells in this case. This is what we expect because we used 30 divisions in the horizontal direction and 100 divisions in the vertical direction while generating the grid. So the total number of cells is 30*100 = 3000.

Also, take a look under zones. We can see the four zones ***inflow***, ***outflow***, ***top***, and ***plate*** that we defined in *ICEM CFD*.

Check and Display Grid

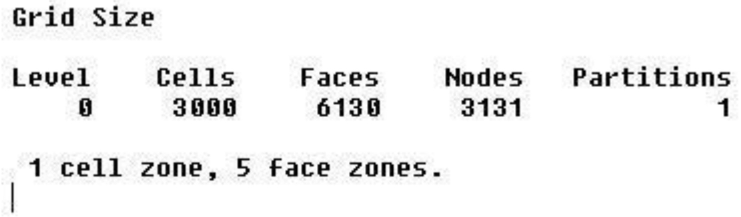
First, we check the grid to make sure that there are no errors.

Main Menu > Grid > Check

Any errors in the grid would be reported at this time. Check the output and make sure that there are no errors reported. Check the grid size:

Main Menu > Grid > Info > Size

The following statistics should appear:



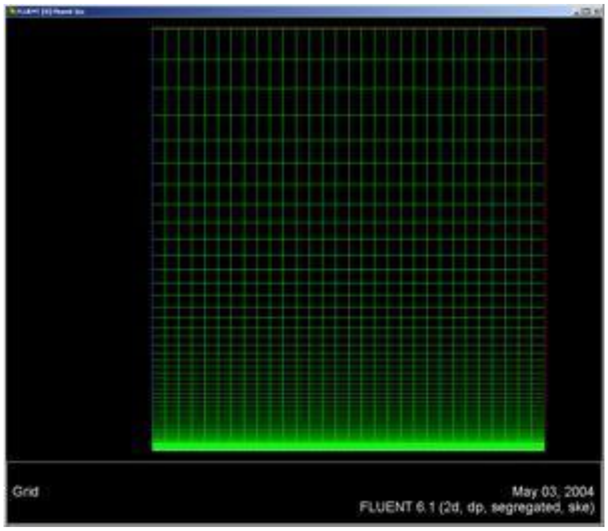
```
Grid Size

Level    Cells    Faces    Nodes    Partitions
    0     3000     6130     3131             1

 1 cell zone, 5 face zones.
```

Display the grid:

Main Menu > Display > Grid...

Make sure all 5 items under *Surfaces* are selected.

Then click ***Display***. The graphics window opens and the grid is displayed in it. Your grid should look like this:



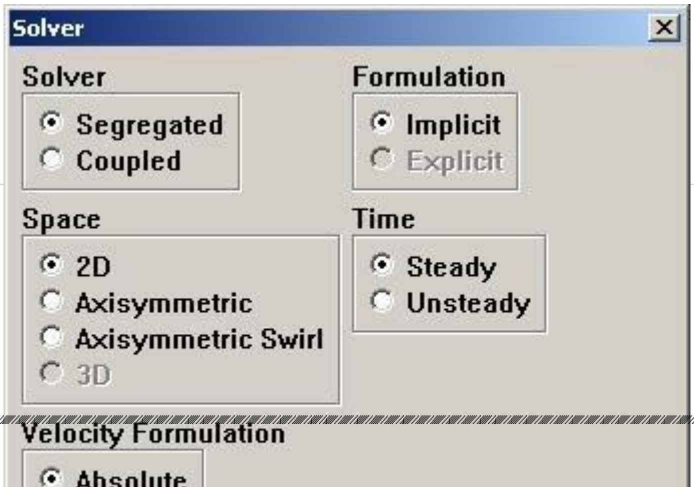
Grid

May 03, 2004
FLUENT 6.1 (2d, dp, segregated, ske)

Define Solver Properties

Main Menu > Define > Models > Solver

We'll use the defaults of 2D space, segregated solver, implicit formulation, steady flow and absolute velocity formulation. Click ***OK***.



Solver

Solver
● Segregated
○ Coupled

Formulation
● Implicit
○ Explicit

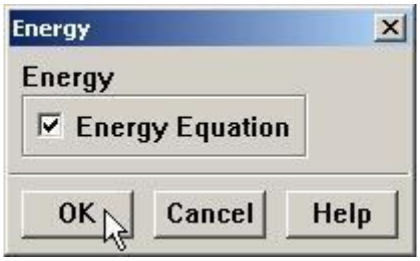
Space
● 2D
○ Axisymmetric
○ Axisymmetric Swirl
○ 3D

Time
● Steady
○ Unsteady

Velocity Formulation
● Absolute

Main Menu > Define > Models > Energy

We are interested in solving the temperature distribution, so we need to solve the energy equation. Select the ***Energy Equation*** and click ***OK*** to exit the menu.



Energy

Energy

☑ Energy Equation

Main Menu > Define > Models > Viscous

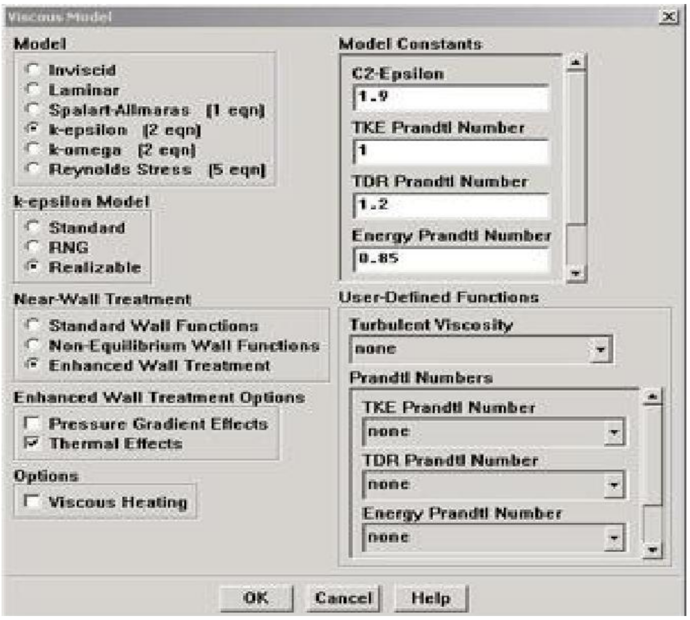
Under *Model*, select the ***k-epsilon*** turbulence model. We will use the ***Realizable*** model in the k-epsilon Model box. The Realizable k-epsilon model produces more accurate results for boundary layer flows than the Standard k-epsilon model. In the *Near-Wall Treatment* box, observe the ***Enhanced Wall Treatment*** option, which deals with the resolution of the boundar layer in our model. There are 3 regions in the boundary layer that we are concerned with, starting at the wall:

1. Laminar sublayer (y+ < 5)

2. Buffer region (5 < y+ < 30)

3. Turbulent region (y+ > 30)

4. Click ***OK***.



Viscous Model

Model
○ Inviscid
○ Laminar
○ Spalart-Allmaras (1 eqn)
● k-epsilon (2 eqn)
○ k-omega (2 eqn)
○ Reynolds Stress (5 eqn)

k-epsilon Model
○ Standard
○ RNG
● Realizable

Near-Wall Treatment
○ Standard Wall Functions
○ Non-Equilibrium Wall Functions
● Enhanced Wall Treatment

Enhanced Wall Treatment Options
□ Pressure Gradient Effects
☑ Thermal Effects

Options
□ Viscous Heating

Model Constants
C2-Epsilon: 1.9
TKE Prandtl Number: 1
TDR Prandtl Number: 1.2
Energy Prandtl Number: 0.85

User-Defined Functions
Turbulent Viscosity: none
Prandtl Numbers
TKE Prandtl Number: none
TDR Prandtl Number: none
Energy Prandtl Number: none

Define Material Properties

Main Menu > Define > Materials...

Change ***Density*** to ideal gas because we are treating the flow as compressible. FLUENT will calcualte the density of the flow at each point based on the pressure and temperature it calculates at that point. Leave ***Cp*** set as the default value of 1006.43. Change ***Thermal Conductivity*** to 9.4505 e-4. Change ***Viscosity*** to 6.667e-7. Scroll down to see ***Molecular Weight***. Leave ***Molecular Weight*** set to the default value of 28.966. These are the values that we specified under .

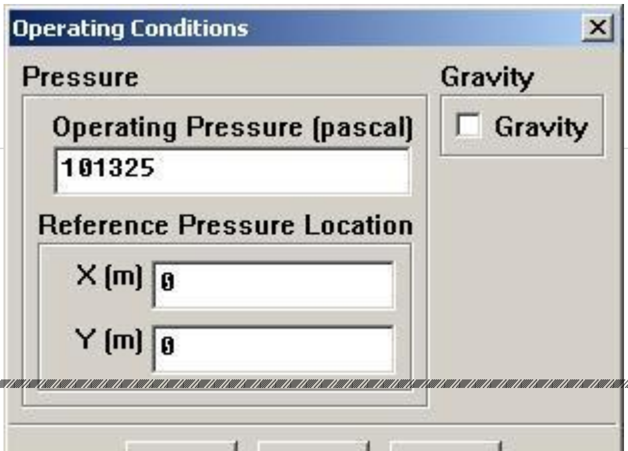
Click ***Change/Create***. Simply clicking close without clicking Change/Create will cause these properties to revert back to their default values.

Define Operating Conditions

Main Menu > Define > Operating Conditions...

For all flows, FLUENT uses gauge pressure internally. Any time an absolute pressure is needed, it is generated by adding the operating pressure to the gauge pressure. We'll use the default value of 1 atm (101,325 Pa) as the ***Operating Pressure***.

Click ***Cancel*** to leave the default value in place.



Operating Conditions

Pressure
Operating Pressure (pascal): 101325
Reference Pressure Location
X (m): 0
Y (m): 0

Gravity
□ Gravity

Define Boundary Conditions

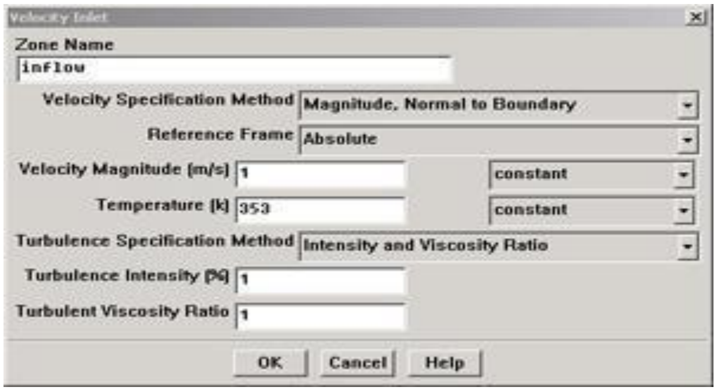
We'll now set the value of the velocity at the inflow and pressure at the outflow.

Main Menu > Define > Boundary Conditions...

We note here that the four types of boundaries we defined are specified as zones on the left side of the *Boundary Conditions Window*. There are also 2 zones default-interior fluid, used to define the interior of the flow field. We will not need to change any setting for these 2 zones.

Move down the list and select ***inflow*** under *Zone*. Note that FLUENT indicates that the *Type* of this boundary is ***velocity-inlet***. Recall that the boundary type for the inflow was set in ICEM CFD. If necessary, we can change the boundary type set previously in *ICEM CFD* in this menu by selecting a different type from the list on the right. Click ***Set....***

Enter 1 for ***Velocity Magnitude***. This sets the velocity of the fluid entering at the left boundary to a uniform velocity profile of 1m/s. Set *Temperature* to 353K. Change *Turbulence Specification Method* to ***Intensity and Viscosity Ratio***. Set *Turbulence Intensity* to 1 and *Turbulent Viscosity Ratio* to 1. Click ***OK***.



Zone Name: inflow

Velocity Specification Method: Magnitude, Normal to Boundary

Reference Frame: Absolute

Velocity Magnitude (m/s): 1 constant

Temperature (k): 353 constant

Turbulence Specification Method: Intensity and Viscosity Ratio

Turbulence Intensity (%): 1

Turbulent Viscosity Ratio: 1

Choose ***outflow*** under *Zone*. The *Type* of this boundary is ***pressure-outlet***. Click ***Set...***. The default value of the ***Gauge Pressure*** is 0. The (absolute) pressure at the outflow is 1 atm. Since the operating pressure is set to 1 atm, the outflow gauge pressure = outflow absolute pressure - operating pressure = 0. Because we do not expect any backflow, we do not need to set any backflow conditions. Click ***Cancel*** to leave the defaults in place.

Click on ***plate*** under *Zones* and make sure *Type* is set as ***wall***. Click ***Set...***. Because we have a heated isothermal plate, we need to set the temperature. On the ***Thermal*** tab, select ***Temperature*** under *Thermal Conditions*. Change *Temperature*to 413. The material selected is inconsequential because the plate has zero thickness in our model, thus the material properties of the plate do not affect the heat transfer properties of the plate. Click ***OK***.

The last boundary condition to set is for the ***top*** of the flow field. Click on top under *Zones* and make sure *Type* is set as ***symmetry***. Click ***Set...*** to see that there is nothing to set for this boundary. Click OK.

Click ***Close*** to close the *Boundary Conditions* menu.

Step 5: Solve!

We'll use a second-order discretization scheme.

Main Menu > Solve > Controls > Solution...

Change ***Density, Momentum, Turbulence Kinetic Energy, Turbulence Dissipation Rate,*** and

Energy all to ***Second Order Upwind***. Leave *Pressure* and *Pressure-Velocity Coupling* set to the default methods (***Standard*** and ***SIMPLE***, respectively). The other *Pressure* and *Pressure-Velocity Coupling* methods are useful for flows with particular characteristics not present in our problem.

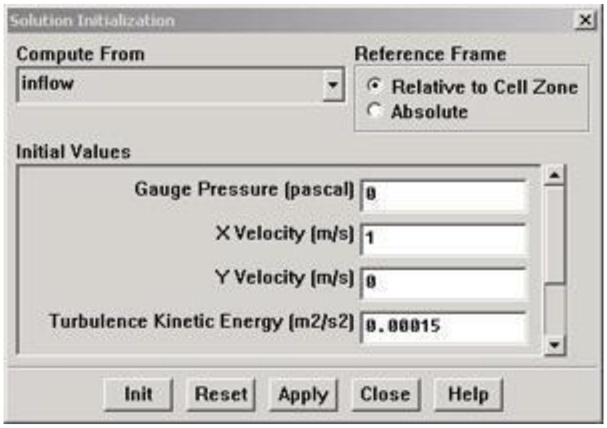
Click ***OK***.

Set Initial Guess

Initialize the flow field to the values at the inflow:

Main Menu > Solve > Initialize > Initialize...

In the *Solution Initialization* window that comes up, choose ***inflow*** under *Compute From*. The ***X Velocity*** for *all* cells will automatically be set to 1 m/s, the ***Y Velocity*** to 0 m/s and the ***Gauge Pressure*** to 0 Pa. These values have been taken from the inflow boundary condition.



Compute From: inflow

Reference Frame
● Relative to Cell Zone
○ Absolute

Initial Values
Gauge Pressure (pascal) 0
X Velocity (m/s) 1
Y Velocity (m/s) 0
Turbulence Kinetic Energy (m2/s2) 0.00015

Click ***Init***. This completes the initialization. Then click ***Close***.

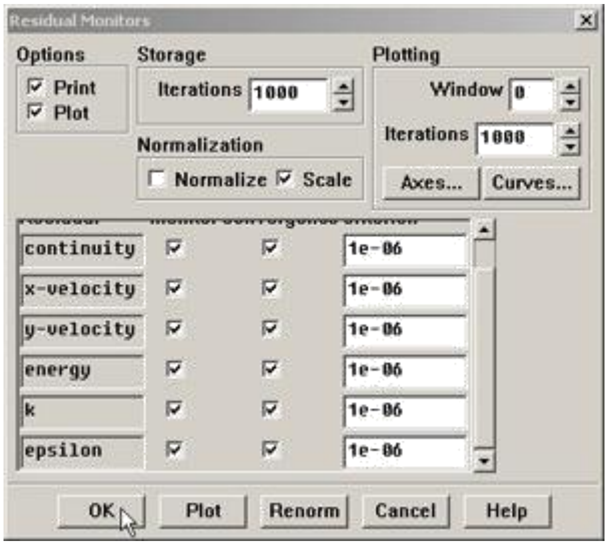
Set Convergence Criteria

FLUENT reports a residual for each governing equation being solved. The residual is a measure of how well the current solution satisfies the discrete form of each governing equation. We will iterate until the residual for each equation falls below 1e-6.

Main Menu > Solve > Monitors > Residual...

Change the residual under ***Convergence Criterion*** for ***continuity, x-velocity,*** and ***y-velocity, energy, k,*** and ***epsilon*** all to 1e-6.

Also, under ***Options***, select ***Print*** and ***Plot***. This will print the residuals in the main window and plot the residuals in the graphics window as they are calculated.



Residual Monitors

Options
☑ Print
☑ Plot

Storage
Iterations 1000

Normalization
□ Normalize ☑ Scale

Plotting
Window 0
Iterations 1000

Residual	Monitor	Convergence	Criterion
continuity	☑	☑	1e-06
x-velocity	☑	☑	1e-06
y-velocity	☑	☑	1e-06
energy	☑	☑	1e-06
k	☑	☑	1e-06
epsilon	☑	☑	1e-06

Click ***OK***.

This completes the problem specification. Save your work:

Main Menu > File > Write > Case...

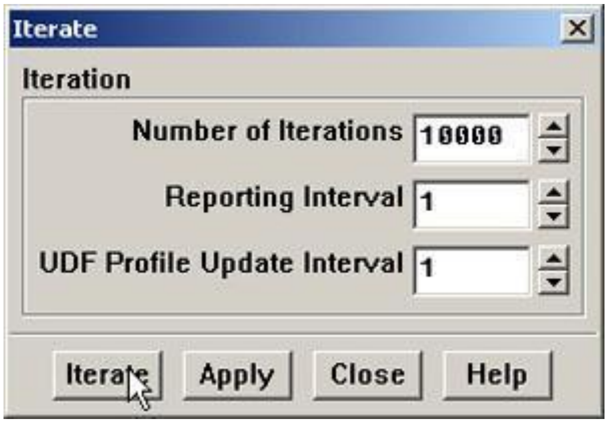
Type in plate.cas for ***Case File***. Click ***OK***. Check that the file has been created in your working directory. If you exit FLUENT now, you can retrieve all your work at any time by reading in this case file.

Iterate Until Convergence

Start the calculation by running 10,000 iterations. The solution will converge before 10,000 iterations are performed, which will stop the iteration process.

Main Menu > Solve > Iterate...

In the *Iterate* Window, change the ***Number of Iterations*** to 10000. Click ***Iterate***.



Iterate

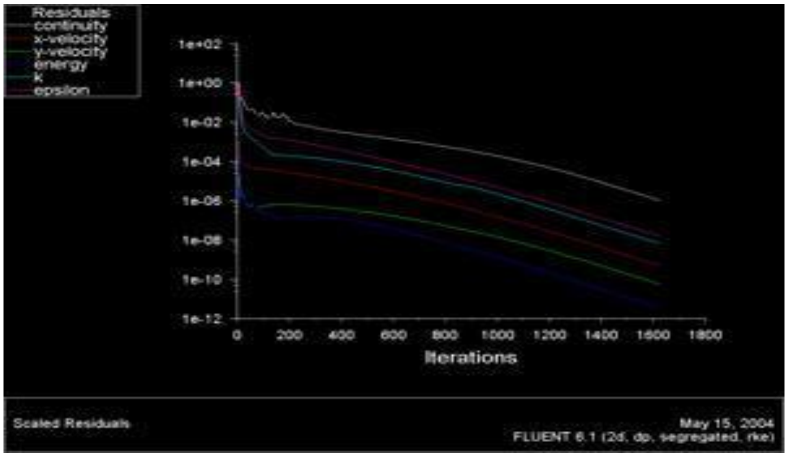
Iteration

Number of Iterations 10000

Reporting Interval 1

UDF Profile Update Interval 1

The residuals for each iteration are printed out as well as plotted in the graphics window as they are calculated.



The residuals fall below the specified convergence criterion of 1e-6 in approximately 1623 iterations.

Main Menu > File > Write > Data...

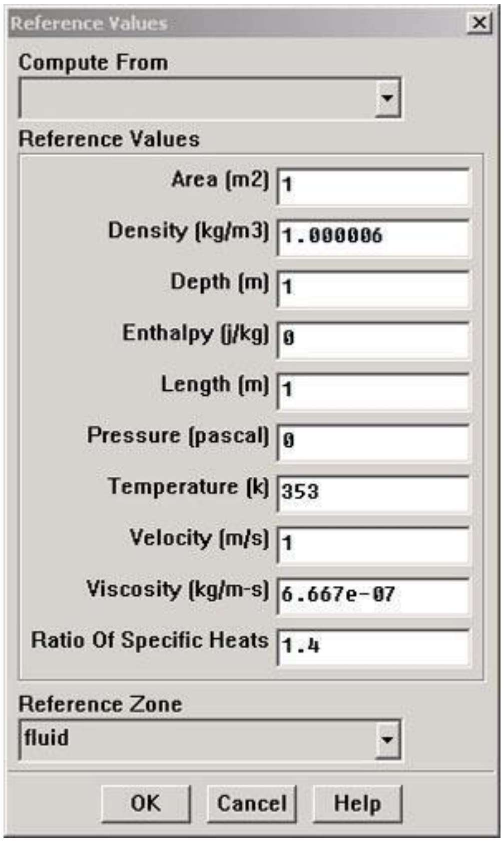
Enter plate.dat for ***Data File*** and click ***OK***. Check that the file has been created in your working directory. You can retrieve the current solution from this data file at any time.

Step 6: Analyze Results

First, we need to set the reference values needed to calculate *y+*.

Main Menu > Report > Reference Values...

Select ***inflow*** under ***Compute From*** to tell FLUENT to use values at the inflow for the reference values. Check that the reference value for velocity is *1 m/s*, temperature is *353 K*, and coefficient of viscosity is *6.667e-7 kg/m-s* Click ***OK***.



Reference Values

Compute From:

Reference Values

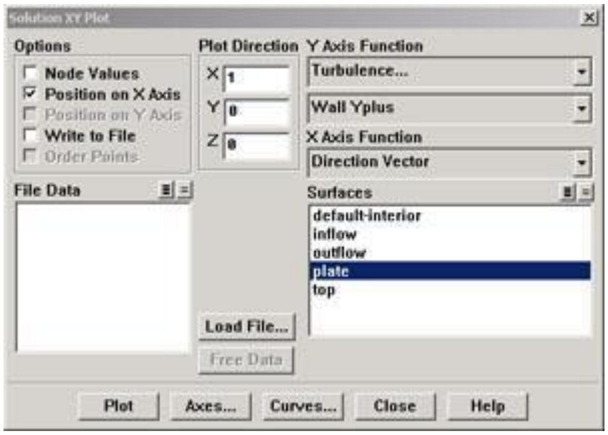
Area (m2)	1
Density (kg/m3)	1.000006
Depth (m)	1
Enthalpy (j/kg)	0
Length (m)	1
Pressure (pascal)	0
Temperature (k)	353
Velocity (m/s)	1
Viscosity (kg/m-s)	6.667e-07
Ratio Of Specific Heats	1.4

Reference Zone: fluid

By using the following method, plot *y+* values for wall-adjacent cells to check how they compare with the recommendation mentioned above.

Main Menu > Plot > XY Plot...

Make sure that ***Position on X Axis*** is set under ***Options***, that 1 is the value next to ***X***, and 0 is the value next to ***Y*** under ***Plot Direction***. Recall that this tells FLUENT to plot the x-coordinate value on the abscissa of the graph. Select ***Turbulence...*** under ***Y Axis Function*** and select ***Wall Yplus*** from the drop down list under that. Since we want the *y+* value for cells adjacent to the wall of the pipe, choose ***plate*** under ***Surfaces***.



Solution XY Plot

Options
- ☐ Node Values
- ☑ Position on X Axis
- ☐ Position on Y Axis
- ☐ Write to File
- ☐ Order Points

Plot Direction
- X 1
- Y 0
- Z 0

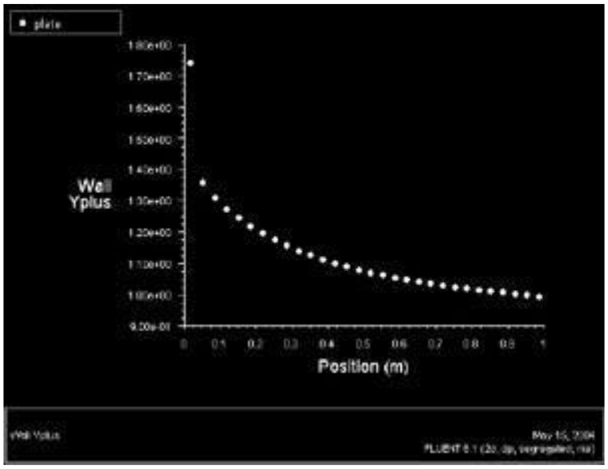
Y Axis Function: Turbulence... / Wall Yplus

X Axis Function: Direction Vector

File Data

Surfaces
- default-interior
- inflow
- outflow
- plate
- top

Click ***Plot***.



As we can see, the wall *y+* value is between 1.0 and 1.4 (ignoring the anamolous at the inflow). Because these values are less than 5, the near-wall mesh resolution is in the laminar sublayer, which is the most accurate region to which we can resolve the boundary layer.

Save Plot

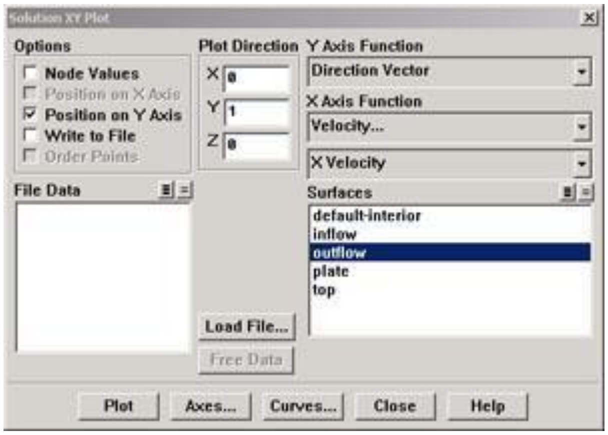
In the *Solution XY Plot Window*, check the ***Write to File*** box under ***Options***. The ***Plot*** button should have changed to the ***Write...*** button. Click on ***Write....*** Enter yplus.xy as the filename and click ***OK***. Check that this file has been created in your FLUENT working directory.

Velocity at x = 1m

Main Menu > Plot > XY Plot...

Under *Options*, unselect ***Position on X Axis*** and select ***Position on Y Axis***. Under *Plot Direction*, enter 0 in the ***X*** box and 1 in the ***Y*** box. This tells FLUENT to plot a vertical rather than horizontal profile.

Under ***X Axis Function***, pick ***Velocity...*** and then in the box under that, pick ***X Velocity***. Finally, select ***outflow*** under ***Surfaces*** since we are plotting the velocity profile at the outflow. De-select ***plate*** under ***Surfaces***.



Solution XY Plot

Options
☐ Node Values
☐ Position on X Axis
☑ Position on Y Axis
☐ Write to File
☐ Order Points

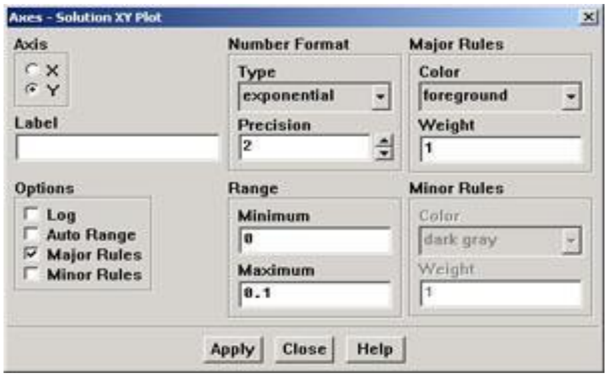
Plot Direction
X 0
Y 1
Z 0

Y Axis Function: Direction Vector
X Axis Function: Velocity...
X Velocity

File Data

Surfaces
default-interior
inflow
outflow
plate
top

Click on ***Axes...*** in the *Solution XY Plot* window. Select ***X*** in the *Axis* box. In the *Options* box select ***Major Rules*** to turn on the grid lines in the plot. Click ***Apply***. Then select the ***Y*** in the *Axis* box, select ***Major Rules*** again, and turn off ***Auto Range***. In the *Range* box enter 0.1 for the *Maximum* so that we may view the velocity profile in the boundary layer region more closely. Click ***Apply*** and ***Close***.



Axes - Solution XY Plot

Axis
○ X
● Y

Label

Options
☐ Log
☐ Auto Range
☑ Major Rules
☐ Minor Rules

Number Format
Type: exponential
Precision: 2

Range
Minimum: 0
Maximum: 0.1

Major Rules
Color: foreground
Weight: 1

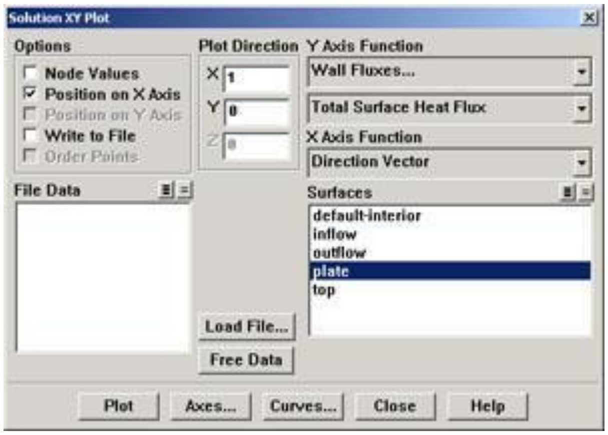
Minor Rules
Color: dark gray
Weight: 1

Uncheck ***Write to File***. Click ***Plot***.

Select ***Write to File*** and save the data for this plot as outflow_profile.xy.

Main Menu > Plot > XY Plot...

In the *Options* box, change back to ***Position on X Axis***. In the *Plot Direction* box, enter the default values of 1 in the ***X*** box and 0 in the ***Y*** box. Under *Y-Axis Function* choose ***Wall Fluxes***. In the box below, chose ***Total Surface Heat Flux***. Select ***Plate*** under *Surfaces*. Before plotting, be sure to turn on ***Auto Range*** for the Y axis under ***Axes...***.


Solution XY Plot

Options
- ☐ Node Values
- ☑ Position on X Axis
- ☐ Position on Y Axis
- ☐ Write to File
- ☐ Order Points

Plot Direction
- X 1
- Y 0
- Z 0

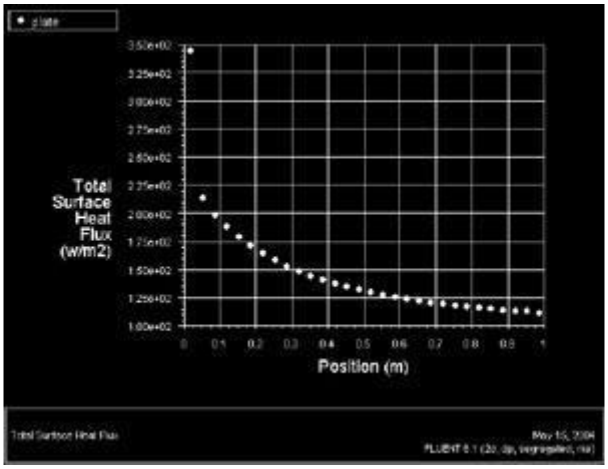
Y Axis Function: Wall Fluxes... / Total Surface Heat Flux

X Axis Function: Direction Vector

File Data

Surfaces
- default-interior
- inflow
- outflow
- plate
- top

Click ***Plot***.



Now Select ***Write to File***. Save the data for this plot as heatflux.xy. Click ***Write...***.

Open the file heatflux.xy using Wordpad or a similar application. You can simply copy and paste the data into Excel.

EXPERIMENT - 5
FLOW THROUGH PIPE

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.

Step 1: Create Geometry in ICEM CFD

Start ICEM CFD & Select Solver

Main Menu > Solver > FLUENT 5/6

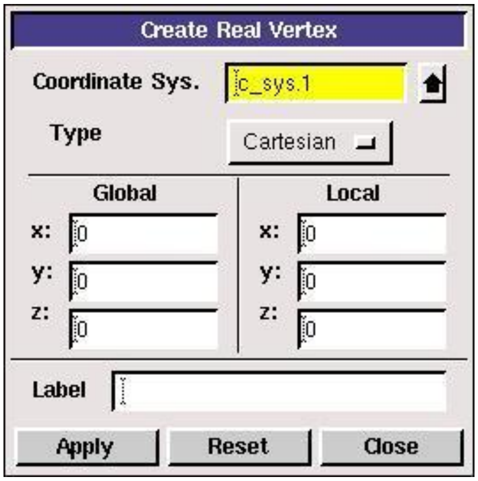
Verify this has been done by looking in the *Transcript Window* where you should see:

Create Vertices

We will treat this problem as a 2-dimensional problem by assuming that the pipe is infinitely wide. Let's begin by creating the vertices that define our flow region.

Create the vertex at the lower-left corner of the rectangle:

Next to ***x:***, enter value 0. Next to ***y:***, enter value 0. Next to ***z:***, enter value 0 (these values should be defaults). Click ***Apply***.



Create Real Vertex

Coordinate Sys. c_sys.1

Type Cartesian

Global		Local	
x:	0	x:	0
y:	0	y:	0
z:	0	z:	0

Label

This creates the vertex (0,0,0) which is displayed in the graphics window.

In the *Transcript*window, ICEM CFD reports that it "Created vertex: vertex.1". The vertices are numbered vertex.1, vertex.2 etc. in the order in which they are created.

Repeat this process to create three more vertices:

Vertex 2: (1,0,0)

Vertex 3: (1,1,0)
Vertex 4: (0,1,0)

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

Operation Toolpad > Global Control > Fit to Window Button

This fits the four vertices of the rectangle we have created to the size of the *Graphics Window*.

Create Edges

An edge is created by selecting two vertices and creating a line between them.

Operation Toolpad > Geometry Command Button > Edge Command Button > Create Edge

Click the ***up arrow button*** next to the vertices box in the *Create Straight Edge* window.

This brings up a list of vertices, from which vertices 1 and 2 can be selected. Select ***Vertex.1*** and ***Vertex.2***. The push the ***right arrow button*** ---> to bring these vertices into the *Picked* column.

Click ***Close.*** Then click ***Apply*** in the *Create Straight Edge* window to create this edge.

Alternately, these vertices can be selected by holding down the ***Shift*** button and clicking on the corresponding vertices. As each vertex is picked, it will appear red in the *Graphics Window*. Then let go of the ***Shift*** button and click ***Apply*** in the *Create Straight Edge* window.

Repeat this process to create edges between vertices 2 & 3, vertices 3 & 4, and vertices 4 & 1.

Create Face

Operation Toolpad > Geometry Command Button > Face Command Button > Form Face

To form a face out of the area enclosed by the four lines, we need to select the four edges that enclose this area. This is done in much the same way as when we selected the vertices.

Click the ***up arrow button*** next to the vertices box in the *Create Face From Wireframe* window. Then push the ***All right arrow button*** All-> to bring these vertices into the *Picked* column.

Click ***Close***. Then click ***Apply*** in the *Create Face From Wireframe* window to create the face. The edges and vertices will become blue, indicating that they now form a face.

Save

Save your *ICEM CFD* file in your working directory.

Main Menu > File > Save As... > Browse...

Find your working directory and save your *ICEM CFD* file there. Make sure to enter the file name, plate.dbs, in the Selection box in addition to the path.

Step 2: Mesh Geometry in ICEM CFD

We'll now create a mesh on the rectangular face with 100 divisions in the vertical direction and 30 divisions in the horizontal direction. We'll first mesh the four edges and then the face. The desired grid spacing is specified through the edge mesh.

Mesh Edges

Operation Toolpad > Mesh Command Button > Edge Command Button > Mesh Edges

Mesh Strategy

In creating this mesh, it is desirable to have more cells near the plate (Edge 1) because we want to resolve the turbulent boundary layer, which is very thin compared to the height of the flow field.

Click the ***up arrow button*** next to the *Edges* box in the *Mesh Edges* window. Select edge ***Edge.2***. Then push the ***right arrow button*** ---> to bring this vertex into the *Picked* column. Notice that the arrow on the selected edge should be pointing upwards. An upwards pointing arrow indicates the direction of closely spaced nodes to widely spaced nodes

The proper arrow direction is necessary to ensure a proper mesh. Select ***Edge.4*** in the *Mesh Edges* window. The arrow on this edge is pointing downwards, which needs to be changed. ***Shift + Middle-click*** on the selected edge to change the direction of the arrow to upward.

Under *Type*, select ***Successive Ratio***, if it is not already selected. Set *Ratio* to 1.08. Under *Spacing*, select ***Interval Count***. Set Interval Count to 100 and then click ***Apply***.

Select ***Edge.1*** and ***Edge.3*** in the *Mesh Edges* Window. The direction of the arrows on these edges is irrelevant because the divisions will be the same length. Leave the ***Successive Ratio*** set to 1 and set the ***Inteval Count*** to 30. Click ***Apply***.

Mesh Face

Operation Toolpad > Mesh Command Button > Face Command Button > Mesh Faces

Shift left-click on the face or use the up arrow next to *Faces* to select the face. Click ***Apply***.

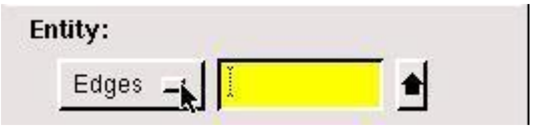
Step 3: Specify Boundary Types in ICEM CFD

Create Boundary Types

We'll next set the boundary types in *ICEM CFD*. The left edge is the inflow of the flow field, the right edge the outflow, the top edge the open top of the flow field, and the bottom edge the plate.

Operation Toolpad > Zones Command Button > Specify Boundary Types Command Button

This will bring up the *Specify Boundary Types* window on the *Operation Panel*. We will first specify that the left edge is the inflow. Under *Entity:*, pick ***Edges*** so that *ICEM CFD* knows we want to pick an edge (face is default).



Entity:

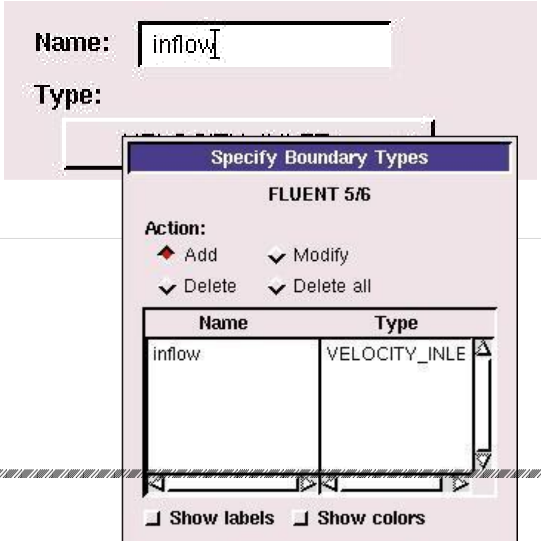
Edges

Now select the left edge by ***Shift-clicking*** on it. The selected edge should appear in the yellow box next to the ***Edges*** box as well as the ***Label/Type*** list under the ***Edges*** box.

Next to ***Name:***, enter inflow.

For *Type:*, select ***VELOCITY_INLET***. You may have to move the *Specify Boundary Types* box up in order to see the bottom of the list and select ***VELOCITY_INLET***.

Click ***Apply***. You should see the new entry appear under ***Name/Type*** box near the top of the window.



Name: inflow

Type:

Specify Boundary Types

FLUENT 5/6

Action:

● Add ○ Modify

○ Delete ○ Delete all

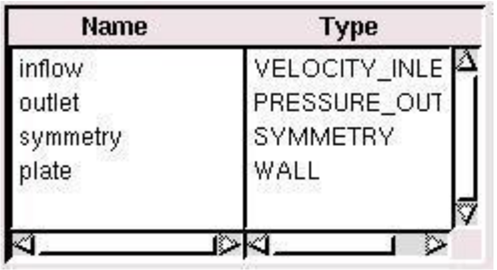
Name	Type
inflow	VELOCITY_INLE

□ Show labels □ Show colors

Repeat this process for the other three edges according to the following table:

Edge Position	Name	Type
Left	inflow	VELOCITY_INLET
Right	outflow	PRESSURE_OUTLET
Top	top	SYMMETRY
Bottom	plate	WALL

You should have the following edges in the *Name/Type* list when finished:



Name	Type
inflow	VELOCITY_INLE
outlet	PRESSURE_OUT
symmetry	SYMMETRY
plate	WALL

Save and Export

Main Menu > File > Save

Main Menu > File > Export > Mesh...

Type in plate.msh for the ***File Name:***. Select ***Export 2d Mesh*** because this is a 2 dimensional mesh. Click ***Accept***.

It is important to check that plate.msh has been created in your working directory. *ICEM CFD* may periodically fail to write the .msh file. If this should happen, simply try writing the .msh file to another directory and then coping it into your working directory.

Step 4: Set Up Problem in FLUENT

Launch Fluent 6.0

Start > Programs > Fluent Inc > FLUENT 6.0

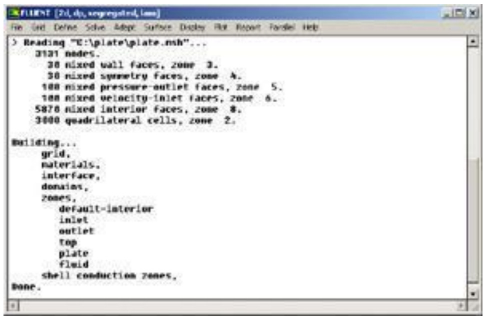
Select the 2ddp version and click Run.

The "2ddp" option is used to select the 2-dimensional, double-precision solver. In the double-precision solver, each floating point number is represented using 64 bits in contrast to the single-precision solver which uses 32 bits. The extra bits increase not only the precision but also the range of magnitudes that can be represented. The downside of using double precision is that it requires more memory.

Import Grid

Main Menu > File > Read > Case...

Navigate to the working directory and select the plate.msh file. This is the mesh file that was created using the preprocessor *ICEM CFD* in the previous step. FLUENT reports the mesh statistics as it reads in the mesh:


FLUENT [2d, dp, segregated, lam]
File Grid Define Solve Adapt Surface Display Plot Report Parallel Help

```
> Reading "C:\plate\plate.msh"...
     3131 nodes.
       30 mixed wall faces, zone  3.
       30 mixed symmetry faces, zone  4.
      100 mixed pressure-outlet faces, zone  5.
      100 mixed velocity-inlet faces, zone  6.
     5870 mixed interior faces, zone  8.
     3000 quadrilateral cells, zone  2.

Building...
      grid,
      materials,
      interface,
      domains,
      zones,
         default-interior
         inlet
         outlet
         top
         plate
         fluid
      shell conduction zones,
Done.
```

Check the number of nodes, faces (of different types) and cells. There are 3000 quadrilateral cells in this case. This is what we expect because we used 30 divisions in the horizontal direction and 100 divisions in the vertical direction while generating the grid. So the total number of cells is 30*100 = 3000.

Also, take a look under zones. We can see the four zones ***inflow***, ***outflow***, ***top***, and ***plate*** that we defined in *ICEM CFD*.

Check and Display Grid

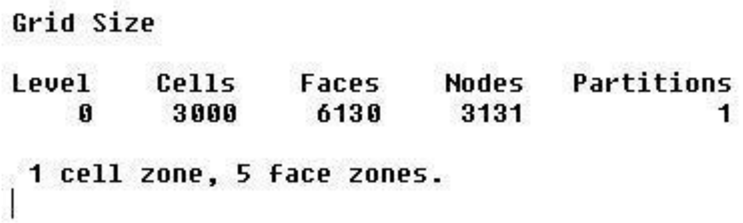
First, we check the grid to make sure that there are no errors.

Main Menu > Grid > Check

Any errors in the grid would be reported at this time. Check the output and make sure that there are no errors reported. Check the grid size:

Main Menu > Grid > Info > Size

The following statistics should appear:



```
Grid Size

Level    Cells    Faces    Nodes   Partitions
    0     3000     6130     3131            1

 1 cell zone, 5 face zones.
```

Display the grid:

Main Menu > Display > Grid...

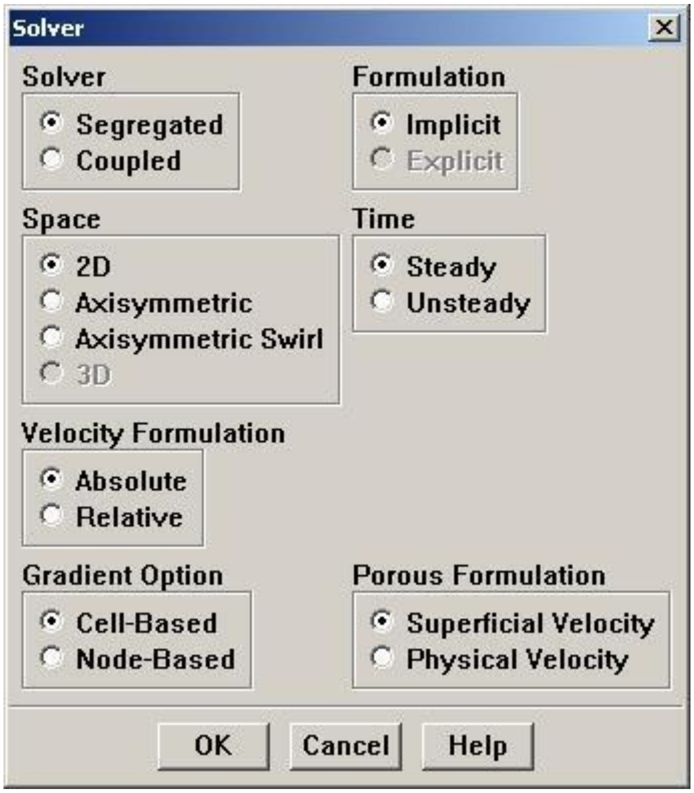
Make sure all 5 items under *Surfaces* are selected.

Then click ***Display***. The graphics window opens and the grid is displayed in it. Your grid should look like this:

Define Solver Properties

Main Menu > Define > Models > Solver

We'll use the defaults of 2D space, segregated solver, implicit formulation, steady flow and absolute velocity formulation. Click ***OK***.



Solver

Solver
- ● Segregated
- ○ Coupled

Formulation
- ● Implicit
- ○ Explicit

Space
- ● 2D
- ○ Axisymmetric
- ○ Axisymmetric Swirl
- ○ 3D

Time
- ● Steady
- ○ Unsteady

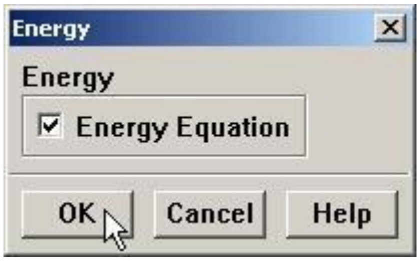
Velocity Formulation
- ● Absolute
- ○ Relative

Gradient Option
- ● Cell-Based
- ○ Node-Based

Porous Formulation
- ● Superficial Velocity
- ○ Physical Velocity

Main Menu > Define > Models > Energy

We are interested in solving the temperature distribution, so we need to solve the energy equation. Select the ***Energy Equation*** and click ***OK*** to exit the menu.



Energy

Energy
- ☑ Energy Equation

Main Menu > Define > Models > Viscous

Under *Model*, select the ***k-epsilon*** turbulence model. We will use the ***Realizable*** model in the k-epsilon Model box. The Realizable k-epsilon model produces more accurate results for boundary layer flows than the Standard k-epsilon model. In the *Near-Wall Treatment* box, observe the ***Enhanced Wall Treatment*** option, which deals with the resolution of the boundar layer in our model. There are 3 regions in the boundary layer that we are concerned with, starting at the wall:

5. Laminar sublayer (y+ < 5)

6. Buffer region (5 < y+ < 30)

7. Turbulent region (y+ > 30)

Click ***OK***.

Define Material Properties

Main Menu > Define > Materials...

Change ***Density*** to ideal gas because we are treating the flow as compressible. FLUENT will calcualte the density of the flow at each point based on the pressure and temperature it calculates at that point. Leave ***Cp*** set as the default value of 1006.43. Change ***Thermal Conductivity*** to 9.4505 e-4. Change ***Viscosity*** to 6.667e-7. Scroll down to see ***Molecular Weight***. Leave ***Molecular Weight*** set to the default value of 28.966. These are the values that we specified under .

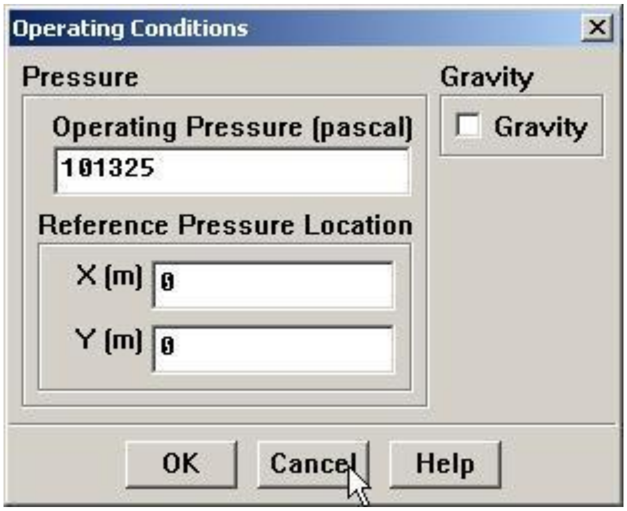
Click ***Change/Create***. Simply clicking close without clicking Change/Create will cause these properties to revert back to their default values.

Define Operating Conditions

Main Menu > Define > Operating Conditions...

For all flows, FLUENT uses gauge pressure internally. Any time an absolute pressure is needed, it is generated by adding the operating pressure to the gauge pressure. We'll use the default value of 1 atm (101,325 Pa) as the ***Operating Pressure***.

Click ***Cancel*** to leave the default value in place.



Operating Pressure (pascal) 101325

X (m) 0

Y (m) 0

□ Gravity

Define Boundary Conditions

We'll now set the value of the velocity at the inflow and pressure at the outflow.

Main Menu > Define > Boundary Conditions...

We note here that the four types of boundaries we defined are specified as zones on the left side of the *Boundary Conditions Window*. There are also 2 zones default-interior fluid, used to define the interior of the flow field. We will not need to change any setting for these 2 zones.

Move down the list and select ***inflow*** under *Zone*. Note that FLUENT indicates that the *Type* of this boundary is ***velocity-inlet***. Recall that the boundary type for the inflow was set in ICEM CFD. If necessary, we can change the boundary type set previously in *ICEM CFD* in this menu by selecting a different type from the list on the right. Click ***Set....***

Enter 1 for ***Velocity Magnitude***. This sets the velocity of the fluid entering at the left boundary to a uniform velocity profile of 1m/s. Set *Temperature* to 353K. Change *Turbulence Specification Method* to ***Intensity and Viscosity Ratio***. Set *Turbulence Intensity* to 1 and *Turbulent Viscosity Ratio* to 1. Click ***OK***.

Choose ***outflow*** under *Zone*. The *Type* of this boundary is ***pressure-outlet***. Click ***Set...***. The default value of the ***Gauge Pressure*** is 0. The (absolute) pressure at the outflow is 1 atm. Since the operating pressure is set to 1 atm, the outflow gauge pressure = outflow absolute pressure - operating pressure = 0. Because we do not expect any backflow, we do not need to set any backflow conditions. Click ***Cancel*** to leave the defaults in place.

Click on ***plate*** under *Zones* and make sure *Type* is set as ***wall***. Click ***Set...***. Because we have a heated isothermal plate, we need to set the temperature. On the ***Thermal*** tab, select ***Temperature*** under *Thermal Conditions*. Change *Temperature*to 413. The material selected is inconsequential because the plate has zero thickness in our model, thus the material properties of the plate do not affect the heat transfer properties of the plate. Click ***OK***.

The last boundary condition to set is for the ***top*** of the flow field. Click on top under *Zones* and make sure *Type* is set as ***symmetry***. Click ***Set...*** to see that there is nothing to set for this boundary. Click OK.

Click ***Close*** to close the *Boundary Conditions* menu.

Step 5: Solve!

We'll use a second-order discretization scheme.

Main Menu > Solve > Controls > Solution...

Change ***Density, Momentum, Turbulence Kinetic Energy, Turbulence Dissipation Rate,*** and

Energy all to ***Second Order Upwind***. Leave *Pressure* and *Pressure-Velocity Coupling* set to the default methods (***Standard*** and ***SIMPLE***, respectively). The other *Pressure* and *Pressure-Velocity Coupling* methods are useful for flows with particular characteristics not present in our problem.

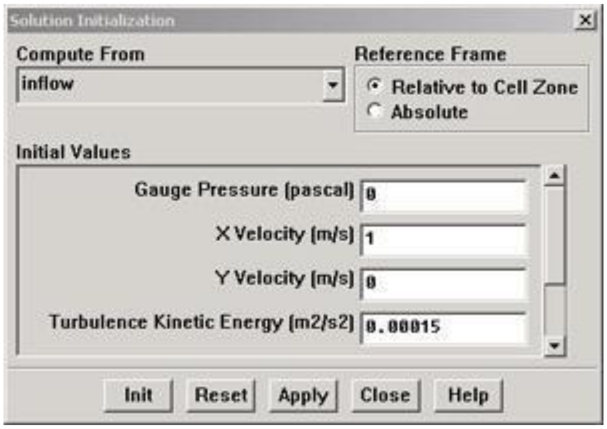
Click ***OK***.

Set Initial Guess

Initialize the flow field to the values at the inflow:

Main Menu > Solve > Initialize > Initialize...

In the *Solution Initialization* window that comes up, choose ***inflow*** under *Compute From*. The ***X Velocity*** for *all* cells will automatically be set to 1 m/s, the ***Y Velocity*** to 0 m/s and the ***Gauge Pressure*** to 0 Pa. These values have been taken from the inflow boundary condition.


Solution Initialization

Compute From: inflow

Reference Frame
- ● Relative to Cell Zone
- ○ Absolute

Initial Values

Gauge Pressure (pascal) 0

X Velocity (m/s) 1

Y Velocity (m/s) 0

Turbulence Kinetic Energy (m2/s2) 0.00015

Click ***Init***. This completes the initialization. Then click ***Close***.

Set Convergence Criteria

FLUENT reports a residual for each governing equation being solved. The residual is a measure of how well the current solution satisfies the discrete form of each governing equation. We will iterate until the residual for each equation falls below 1e-6.

Main Menu > Solve > Monitors > Residual...

Change the residual under ***Convergence Criterion*** for ***continuity, x-velocity,*** and ***y-velocity, energy, k,*** and ***epsilon*** all to 1e-6.

Also, under ***Options***, select ***Print*** and ***Plot***. This will print the residuals in the main window and plot the residuals in the graphics window as they are calculated.

Click ***OK***.

This completes the problem specification. Save your work:

Main Menu > File > Write > Case...

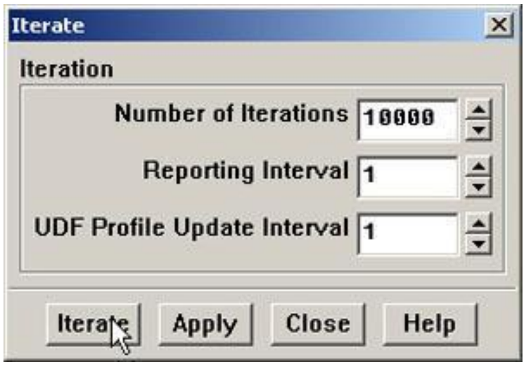
Type in plate.cas for ***Case File***. Click ***OK***. Check that the file has been created in your working directory. If you exit FLUENT now, you can retrieve all your work at any time by reading in this case file.

Iterate Until Convergence

Start the calculation by running 10,000 iterations. The solution will converge before 10,000 iterations are performed, which will stop the iteration process.

Main Menu > Solve > Iterate...

In the *Iterate* Window, change the ***Number of Iterations*** to 10000. Click ***Iterate***.



Iterate

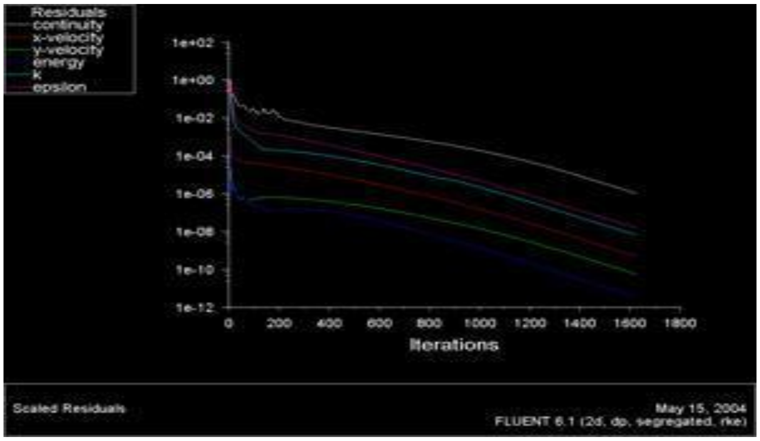
Iteration

Number of Iterations: 10000

Reporting Interval: 1

UDF Profile Update Interval: 1

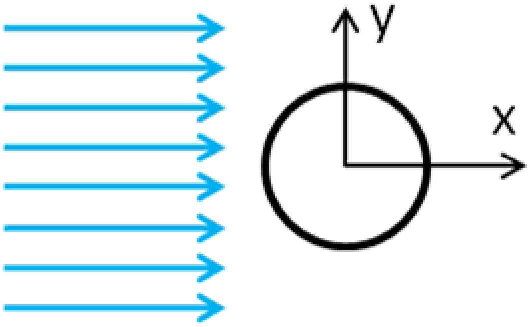
The residuals for each iteration are printed out as well as plotted in the graphics window as they are calculated.



The residuals fall below the specified convergence criterion of 1e-6 in approximately 1623 iterations.

EXPERIMENT - 6
FLOW OVER A CIRCULAR CYLINDER

Problem Specification



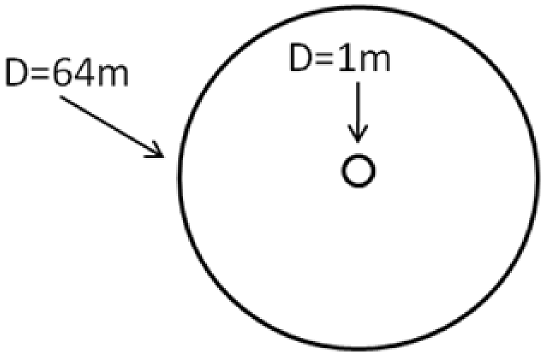
Consider the steady state case of a fluid flowing past a cylinder, as illustrated above. Obtain the velocity and pressure distributions when the Reynolds number is chosen to be 20. In order to simplify the computation, the diameter of the cylinder is set to 1 m, the x component of the velocity is set to 1 m/s and the density of the fluid is set to 1 kg/m^3. 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.

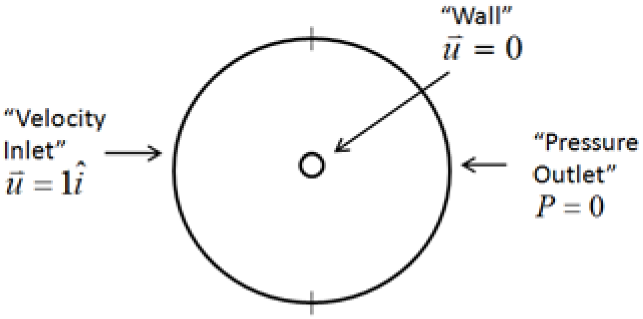
Solution Domain

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 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 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**

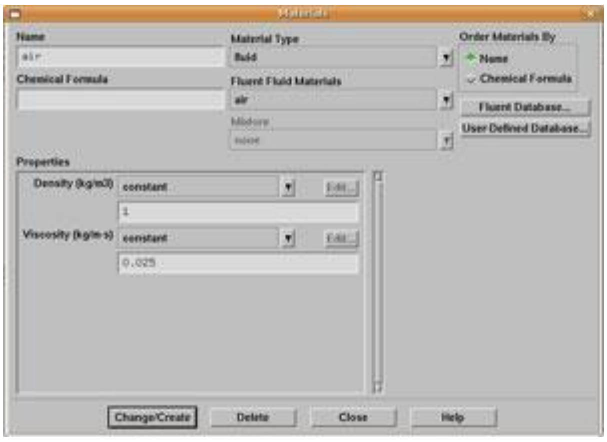
Click **OK**.

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 kg/m ^{3} and **Viscosity** to 0.025 kg/m-s. We choose these numbers so that Re = 40.



Name: air

Chemical Formula:

Material Type: fluid

Fluent Fluid Materials: air

Mixture: none

Order Materials By: ● Name ○ Chemical Formula

Properties

Density (kg/m3): constant

1

Viscosity (kg/m-s): constant

0.025

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

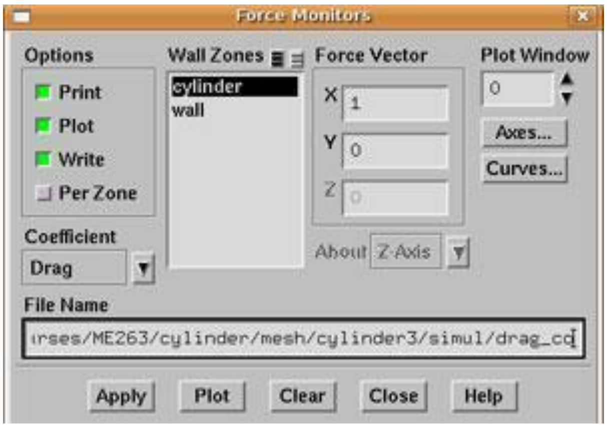
OK.

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.



Options
☑ Print
☑ Plot
☑ Write
□ Per Zone

Wall Zones: cylinder (selected), wall

Force Vector
X 1
Y 0
Z 0

Plot Window 0

Coefficient: Drag

About Z-Axis

File Name: ırses/ME263/cylinder/mesh/cylinder3/simul/drag_cd

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 freestream. 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?

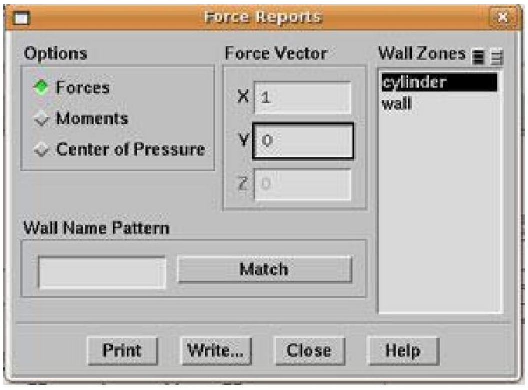
Main Menu > File > Write > Case & Data...

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.



Force Reports

Options
● Forces
○ Moments
○ Center of Pressure

Force Vector
X 1
Y 0
Z 0

Wall Zones
cylinder
wall

Wall Name Pattern

```
Force vector: (1 0 0)
                    pressure     viscous      total        pressure     viscous      total
zone name           force        force        force        coefficient  coefficient  coefficient
                    n            n            n
------------------  -----------  -----------  -----------  -----------  -----------  -----------
cylinder            [illegible]  [illegible]  [illegible]  [illegible]  [illegible]  [illegible]
------------------  -----------  -----------  -----------  -----------  -----------  -----------
net                 [illegible]  [illegible]  [illegible]  [illegible]  [illegible]  [illegible]
```

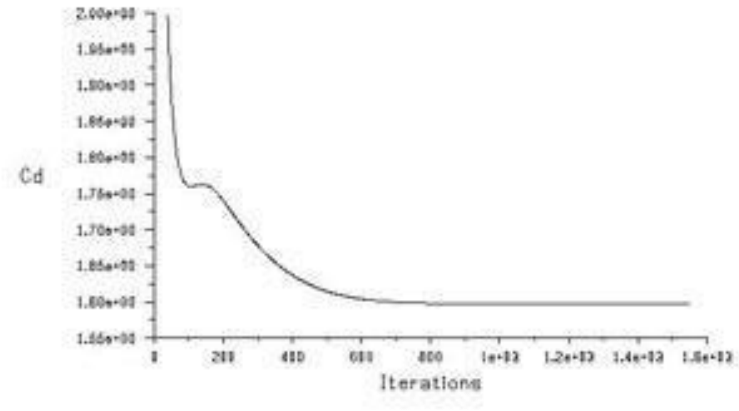
Plot convergence of the drag coefficient versus the number of iterations. Report the drag coefficient and compare it with the result in literature as shown in Table 1.

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.

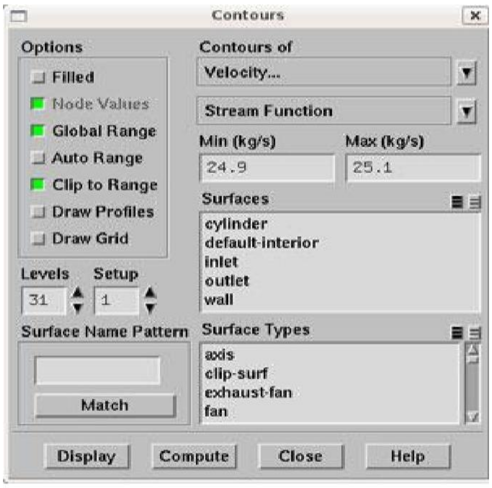
Similarly, you can plot the lift coefficient, which should be zero for the symmetric flow. As you can see, the drag coefficient is around 2.1, which is significantly higher than the result in the table. Next, we will try to increase the domain size and repeat the simulation.



Plot streamlines. Plot pressure, velocity, vorticity contours.

Streamline

Display > Contours >



Options
☐ Filled
☑ Node Values
☑ Global Range
☐ Auto Range
☑ Clip to Range
☐ Draw Profiles
☐ Draw Grid

Contours of: Velocity...
Stream Function

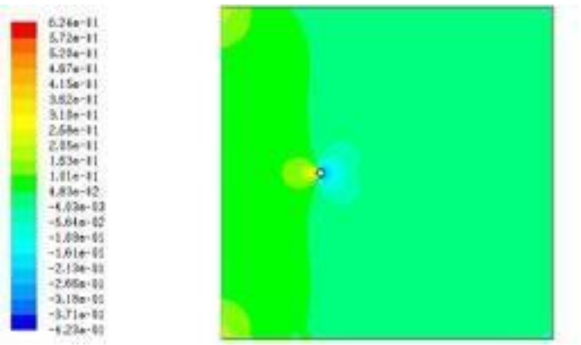
Min (kg/s): 24.9
Max (kg/s): 25.1

Levels: 31
Setup: 1

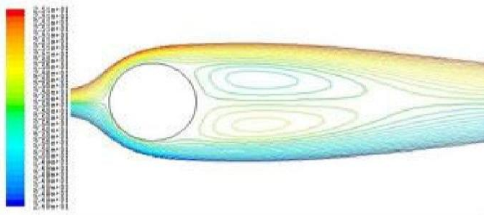
Surfaces
cylinder
default-interior
inlet
outlet
wall

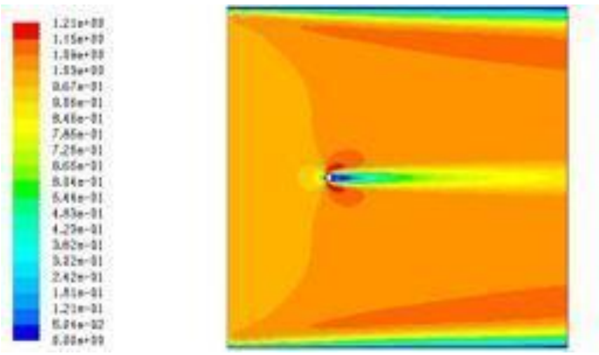
Surface Name Pattern

Surface Types
axis
clip-surf
exhaust-fan
fan



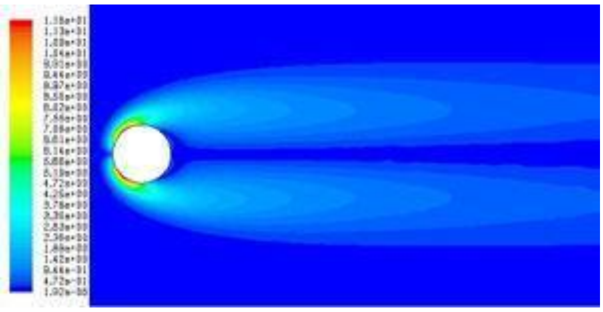
Contour of the dimensionless pressure





Contour of the velocity magnitude (normalized by the freestream velocity)

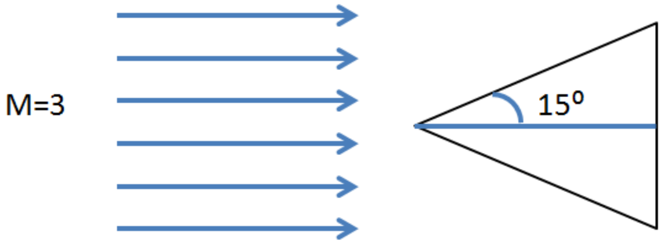
Vorticity Magnitude (normalized by U/a)



EXPERIMENT - 7
FLOW OVER A WEDGE

Problem Specification

A uniform supersonic stream encounters a wedge with a half-angle of 15 degrees as shown in the figure below.



The stream is at the following conditions:

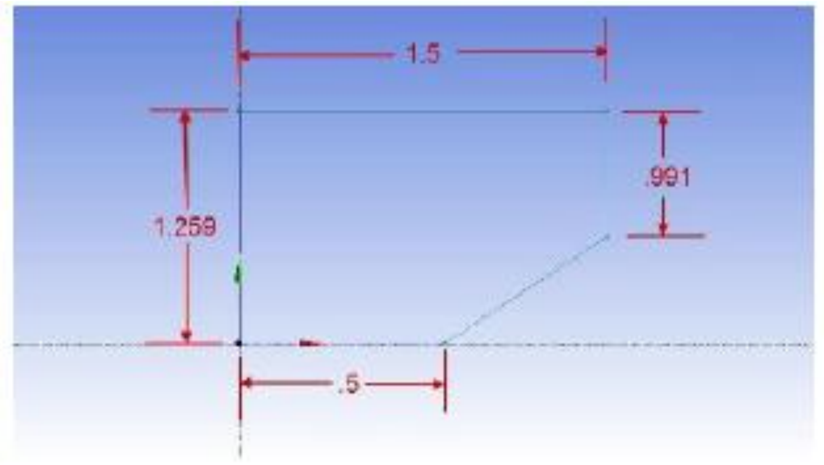
$$\text{Mach Number } M_1 = 3$$

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

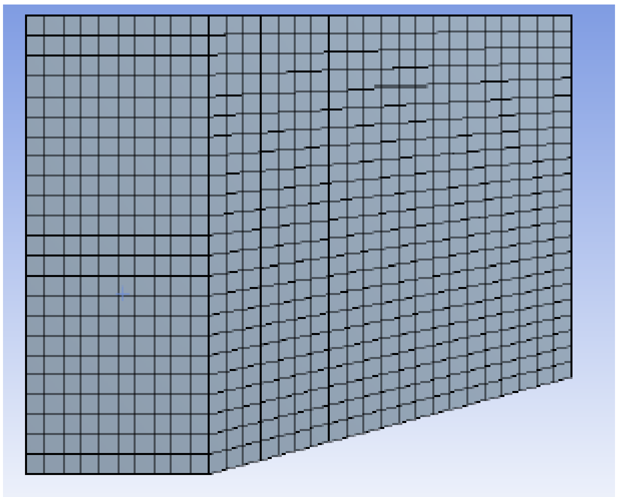
$$\text{Static Temperature } T_1 = 300\ k$$

Using FLUENT, calculate the Mach Number, static and total pressure behind the oblique shock that will be formed. Also, calculate the shock angle, pressure coefficient along the wedge and drag coefficient. Compare the FLUENT results with the corresponding analytical results.

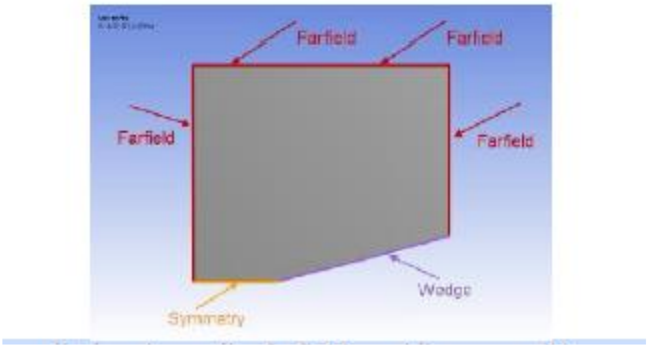
Design considerations



Mesh



Boundary conditions



Physics Setup

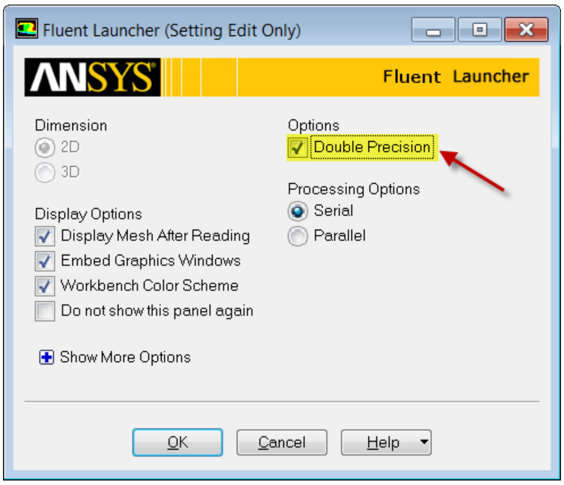
Update the Project and Open FLUENT

Before we open FLUENT, we need to update the project so we can import the mesh into FLUENT. To do this, click Update Project . When the project updates, double click Setup to open FLUENT.

Initial Settings

(Double Click) Setup in the ***Workbench Project Page***.

When the ***FLUENT Launcher*** appears change options to "Double Precision", and then click ***OK*** as shown below.The Double Precision option is used to select the double-precision solver. In the double-precision solver, each floating point number is represented using 64 bits in contrast to the single-precision solver which uses 32 bits. The extra bits increase not only the precision, but also the range of magnitudes that can be represented. The downside of using double precision is that it requires more memory.



Fluent Launcher (Setting Edit Only)

Dimension
● 2D
○ 3D

Options
☑ Double Precision

Processing Options
● Serial
○ Parallel

Display Options
☑ Display Mesh After Reading
☑ Embed Graphics Windows
☑ Workbench Color Scheme
□ Do not show this panel again

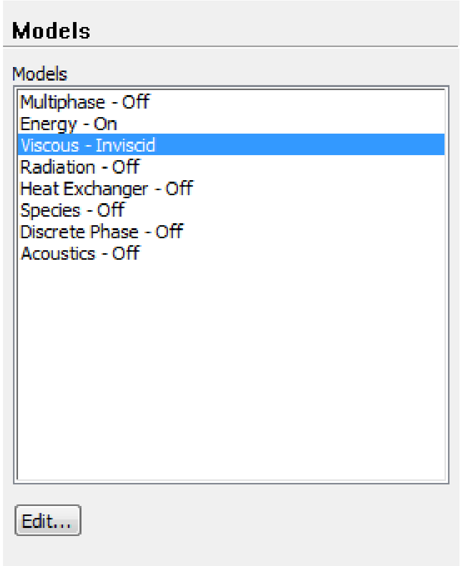
Problem Setup - General

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***.



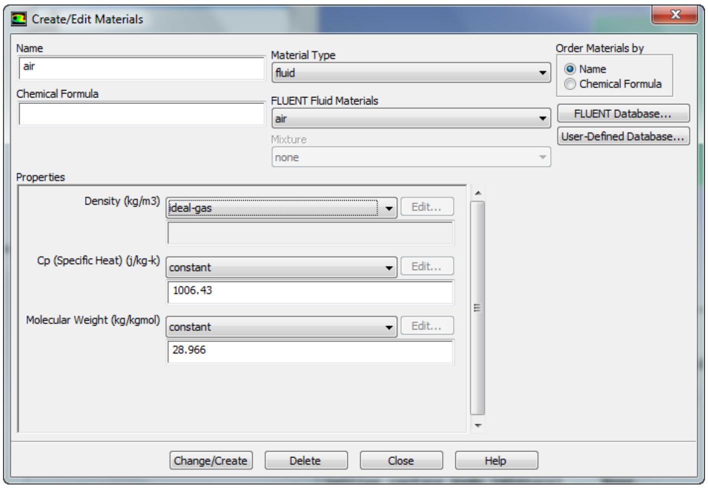
Models

Models

Multiphase - Off
Energy - On
Viscous - Inviscid
Radiation - Off
Heat Exchanger - Off
Species - Off
Discrete Phase - Off
Acoustics - Off

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***.



Create/Edit Materials

Name: air

Chemical Formula:

Material Type: fluid

FLUENT Fluid Materials: air

Mixture: none

Order Materials by: ● Name ○ Chemical Formula

Properties

Density (kg/m3): ideal-gas

Cp (Specific Heat) (j/kg-k): constant — 1006.43

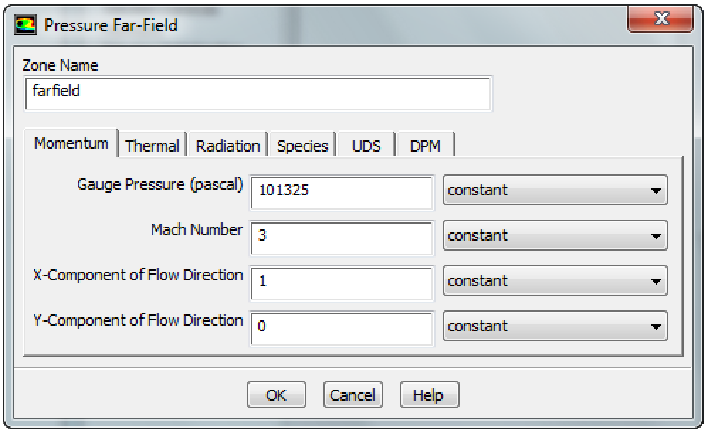
Molecular Weight (kg/kgmol): constant — 28.966

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***.



Zone Name: farfield

Gauge Pressure (pascal): 101325 — constant

Mach Number: 3 — constant

X-Component of Flow Direction: 1 — constant

Y-Component of Flow Direction: 0 — constant

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

Wedge

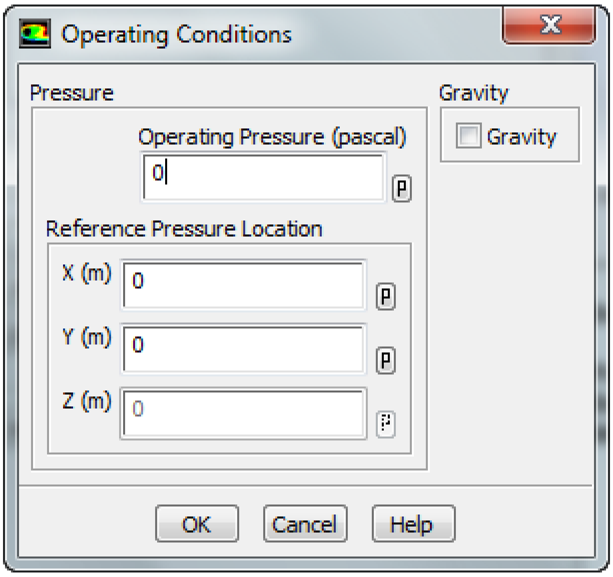
In the *Boundary Conditions* window, select ***wedge***. Use the drop-down menu to change the ***Type*** to ***wall***.

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***



Operating Pressure (pascal): 0

□ Gravity

Reference Pressure Location

X (m): 0

Y (m): 0

Z (m): 0

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.

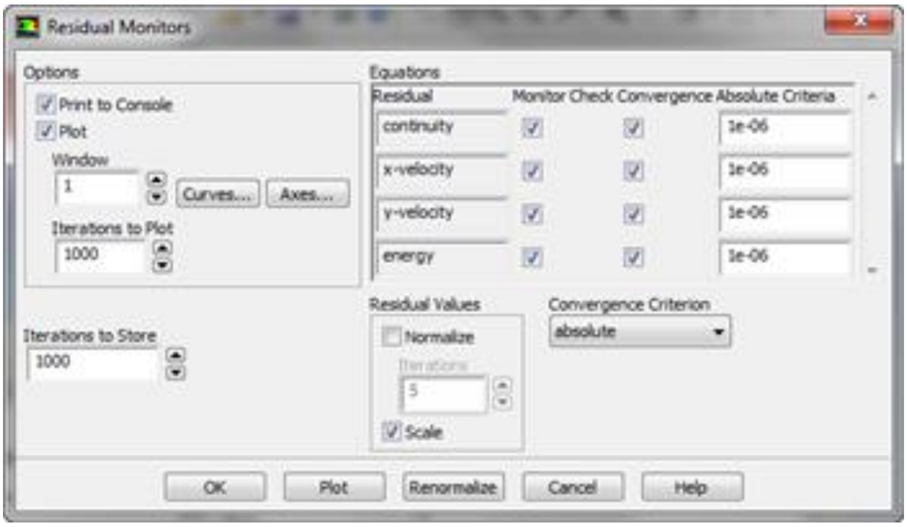
Solution Controls

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 timestep. 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***.



Residual Monitors

Options

☑ Print to Console

☑ Plot

Window: 1

Iterations to Plot: 1000

Iterations to Store: 1000

Equations

Residual	Monitor	Check Convergence	Absolute Criteria
continuity	☑	☑	1e-06
x-velocity	☑	☑	1e-06
y-velocity	☑	☑	1e-06
energy	☑	☑	1e-06

Residual Values

□ Normalize

Iterations: 5

☑ Scale

Convergence Criterion: absolute

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.

Mach Number Contours

1. Some calculated parameters are not by default carried over into CFD-post. We are interested in such quantities (i.e. Mach Number). To manually transfer a customized selection of quantities
   a. Select File > Data File Quantities
   b. Under Additional Quantities, Select Static Pressure, Total Pressure, Mach Number, and Total Temperature
2. Post-processing will be done in CFD-post > Double Click Results in Workbench
3. We are interested in viewing contours of Mach Number in CFD-post
   a. Select Insert > Contour > Name > Mach No.
   b. Under Details of Mach No, select Locations > symmetry 1.
   c. Variable > Mach Number> No Contours = 101
4. Turn on the mesh in the graphics pane
   a. Check the box next to symmetry 1 in the Outline tree > Click Render > Show Mesh Lines.
5. Turn off the mesh by deselecting symmetry 1 in the outline tree
6. Save a copy of the figure in the graphics pane

Pressure Contours

The following video shows how to make a plot of the pressure contours.

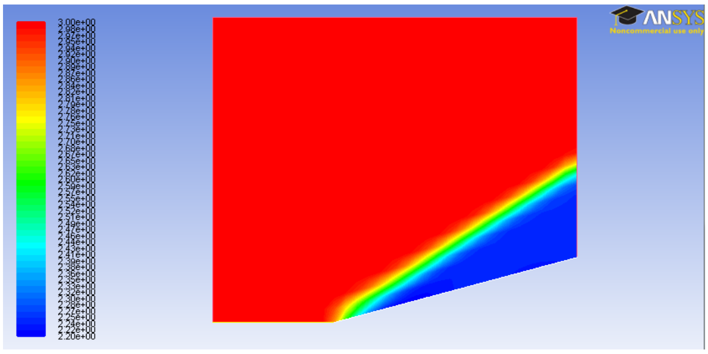
1. Turn off the Mach Number contours in the graphics window
2. We are interested in viewing contours of pressure in CFD-post
   a. Select Insert > Contour
   b. Name > Pressure Contours
   c. Under Details of Pressure contours, select Locations > symmetry 1.Variable > Pressure.
3. To increase the number of contours to 101
4. Under Details of Pressure contours, scroll down to # of Contours. Type “101”.

Velocity Vectors

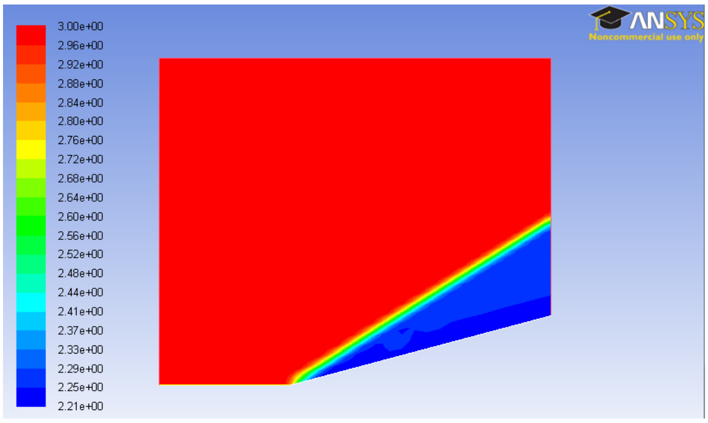
1. We are interested in viewing velocity vectors in CFD-post.
   a. Select Insert > Vector
   b. Type “Velocity vectors” under Name in the Insert Vector dialogue box that appears.
   c. Under Details of Velocity vectors, select Locations > symmetry 1.
2. To make the velocity vectors more visible, turn off the pressure contours

a. Uncheck the box next to Pressure contours in the Outline tree

Now, once again, plot the contours of the mach number. Below is a comparison of the mach number results from the original mesh and the refined mesh.



Original Mesh



Refined Mesh

The most striking difference between the two results is the thickness of the shock. Notice that for the refined mesh, the shock is less thick that for the original mesh. This shows that the refined mesh is converging towards the real case.

Comparison to Analytical Solution

In order to verify our simulation, we need to compare our results to either an analytical solution or an experiment. Below is a table comparing the values from the simulation with the calculations from the pre-analysis.

	Mach Number	**Static Pressure (atm)**	**Shock Angle (degrees)**
Theory Value	2.254	2.824	32.22
FLUENT Solution	2.243	2.803	34.99
Percent Difference	0.8%	0.7%	8.2%

As we can see from the table, we are getting fairly good matching between the computation and analytical approaches. From this we can build our trust in our simulation.

Save Project

Save the project using *File > Save*. Copy *wedge.wbp*j and the associated *wedge_files folder* to a flash drive. You will need both entities to resume the project.

Alternately, you can select *File > Archive* and save the project as one file called wege.wbpz. When prompted, select the option to save *Result/Solution* also. You will then need to save only this file. This is also convenient to e-mail the project. Double-clicking on the wedge.wbpz file will resume the project.

EXPERIMENT - 8

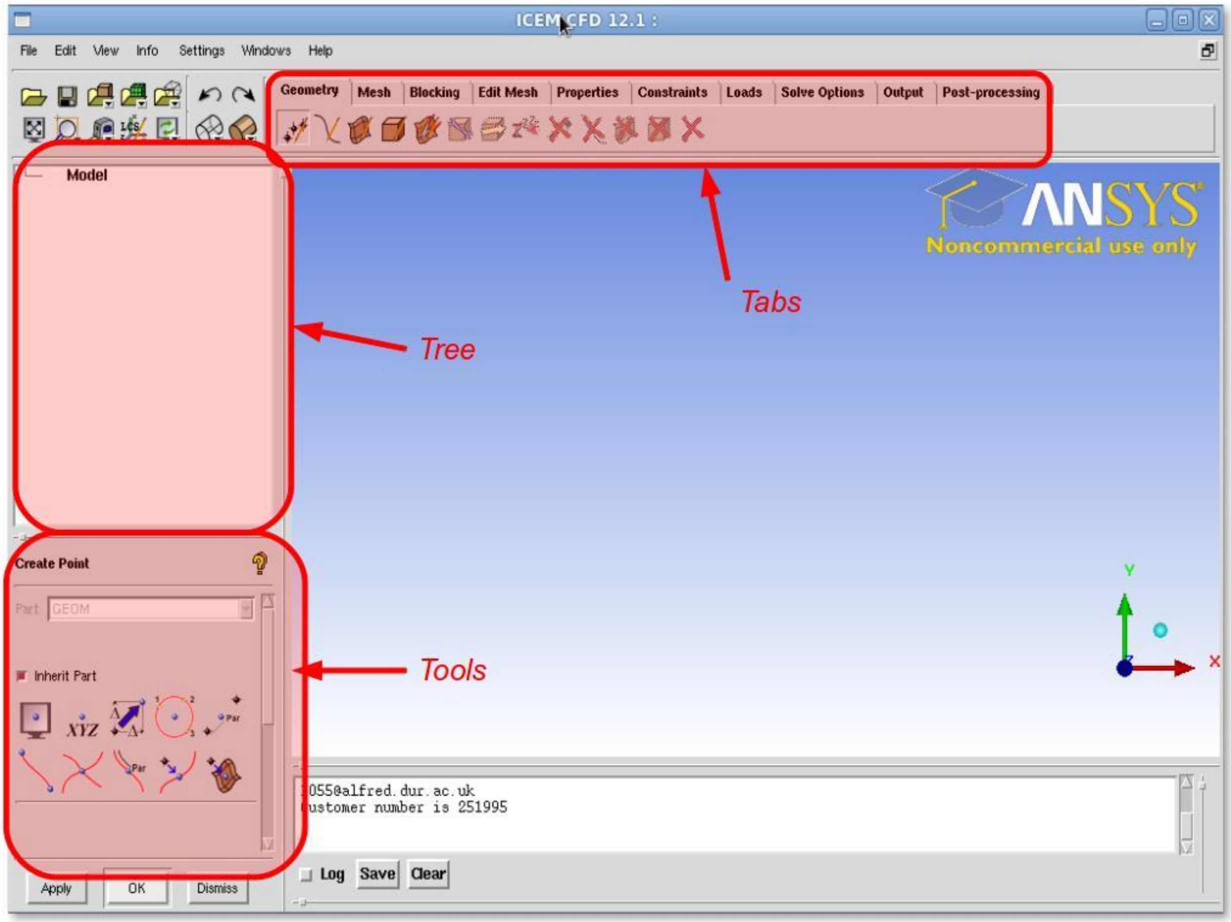
FLOW OVER A CAMBERED AIRFOIL

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.***

The initial screen looks something like:



```
055@alfred.dur.ac.uk
ustomer number is 251995
```

Figure 1: The icemcfd screen.

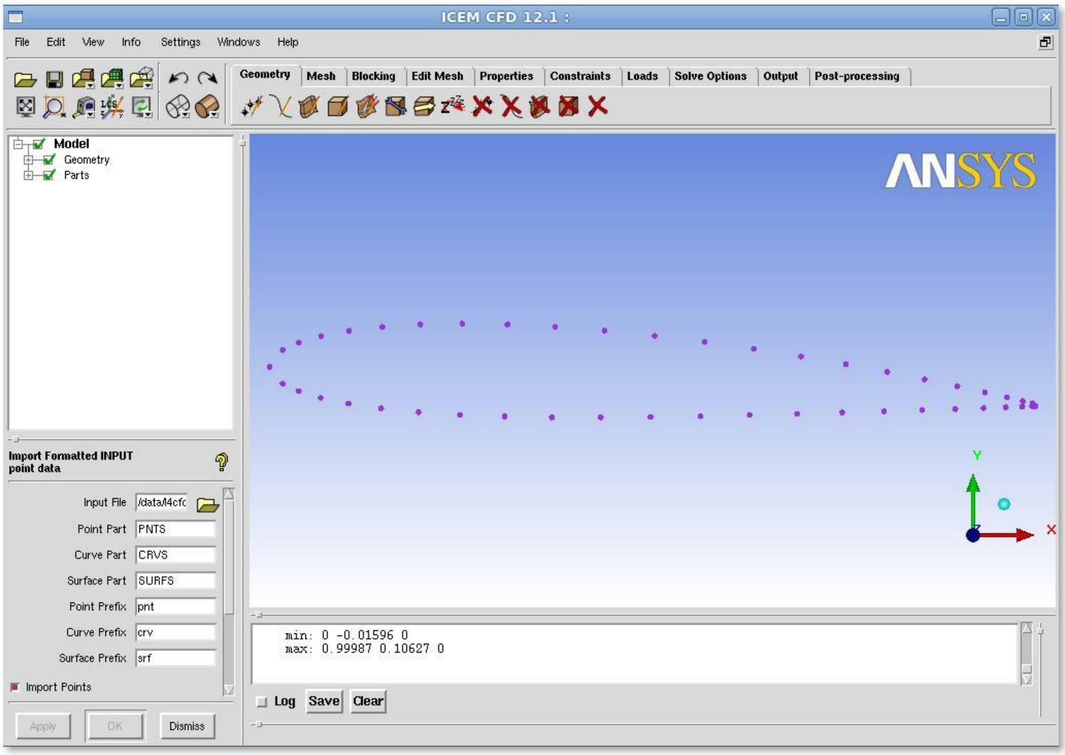
As well as the conventional display area and menus there are three areas of note:

- Tabs: These provide access to broad categories of tools which enable the meshing to take place.
- Tools: This area in the bottom left is where the tools are actually operated from
- Tree: In this area a hierarchical list of points, lines and so on will be built up as you work with the software.

icemcfd is a powerful package but does have something of a learning curve, so if something doesn't work the first time review the steps you took carefully and try again.

Importing the Geometry

- Start ***icemcfd***
- Obtain the NACA 4412 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
- You should have a screen that looks like Figure 2:



Import Formatted INPUT point data

Input File /data/l4cfc
Point Part PNTS
Curve Part CRVS
Surface Part SURFS
Point Prefix pnt
Curve Prefix crv
Surface Prefix srf
☒ Import Points

```
min: 0 -0.01596 0
max: 0.99987 0.10627 0
```

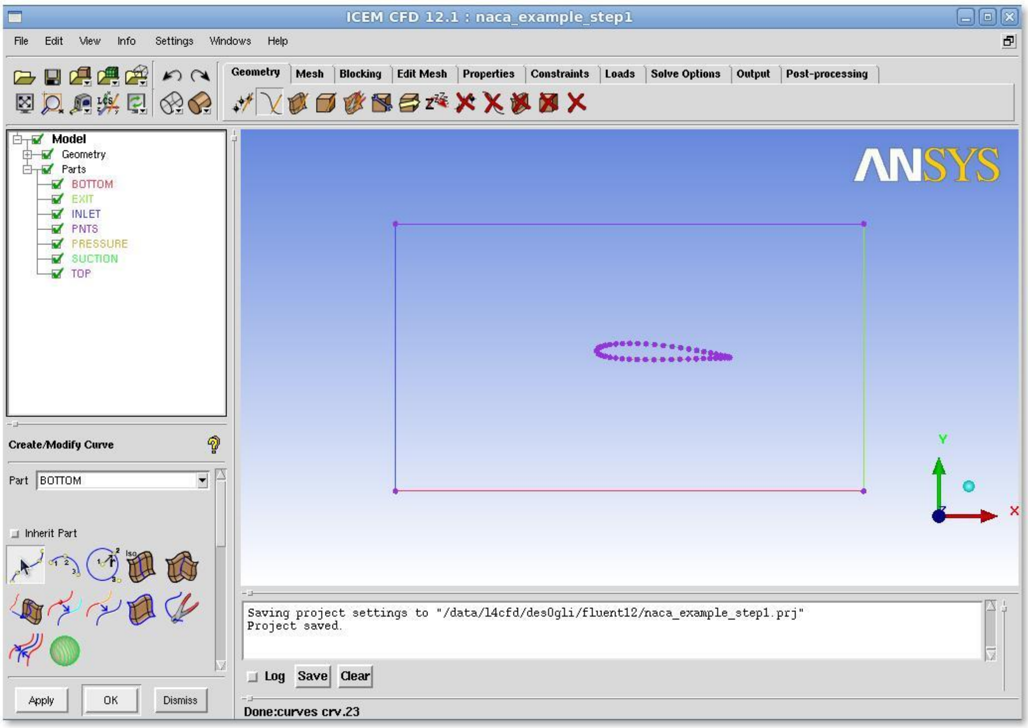
Figure 2: Point Input to icemcfd

- 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

Hint: Pressing F9 suspends the Selection Mode and allows you to use the mouse to control the view, hitting F9 again puts you back into Selection Mode.

- 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. Your screen should look like Figure 3.



```
Saving project settings to "/data/l4cfd/des0gli/fluent12/naca_example_step1.prj"
Project saved.
```

Done:curves crv.23

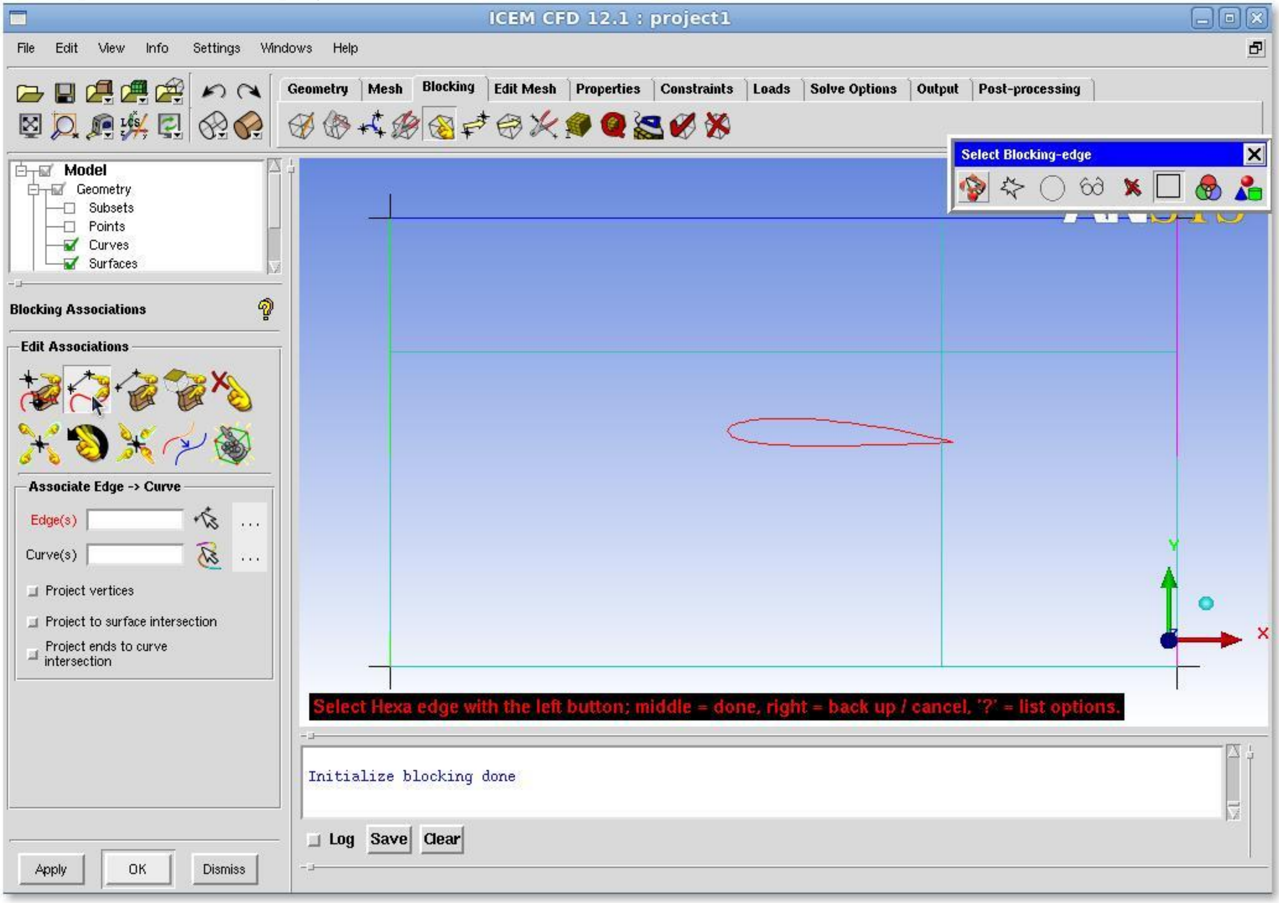
Figure 3: Aerofoil and Calculation Domain.

- 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 “Initalize 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.



Select Hexa edge with the left button; middle = done, right = back up / cancel, '?' = list options.

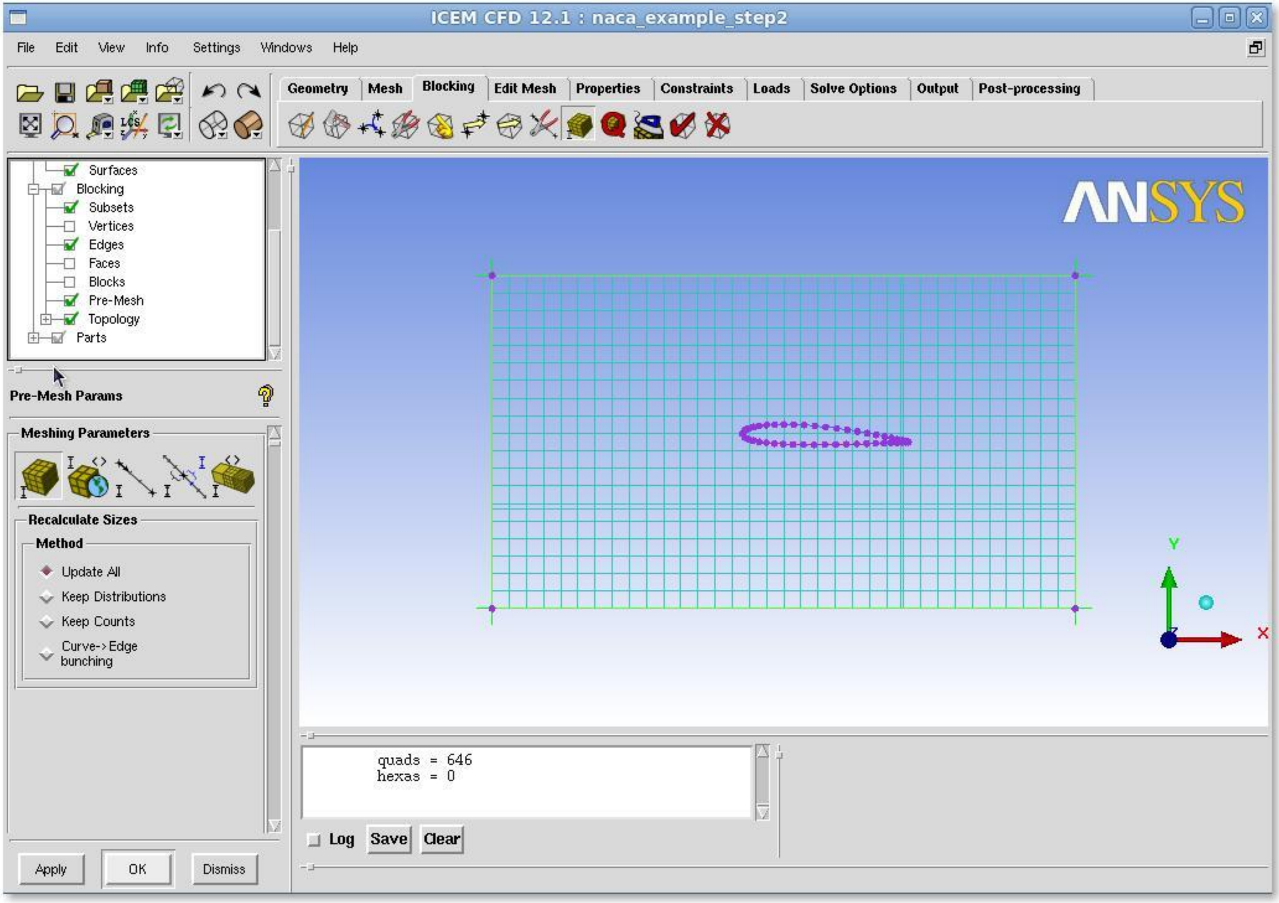
```
Initialize blocking done
```

Figure 4: Setting up the Blocking

- 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.

Hint: This is a part of the tutorial that confuses people a lot. The edges (part of the blocking) and the curves (geometry) are actually on top of each other on the screen. Although they are physically coincident on the screen there is no logical connection between them until you make one using the associate tool.

- 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 (9th 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.



Recalculate Sizes

Method

● Update All
○ Keep Distributions
○ Keep Counts
○ Curve->Edge bunching

```
        quads = 646
        hexas = 0
```

Figure 5: The Pre-Mesh View turned on.

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.

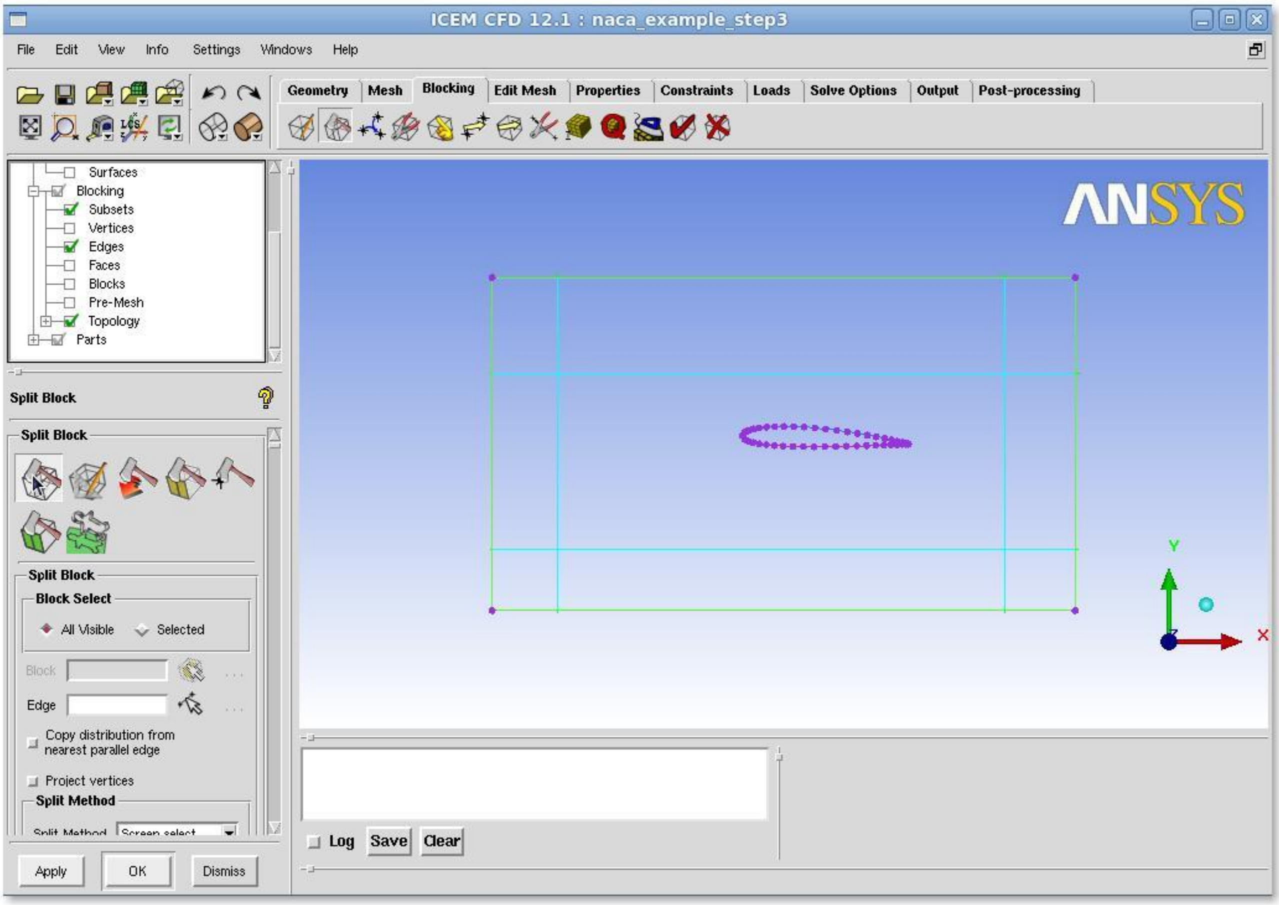
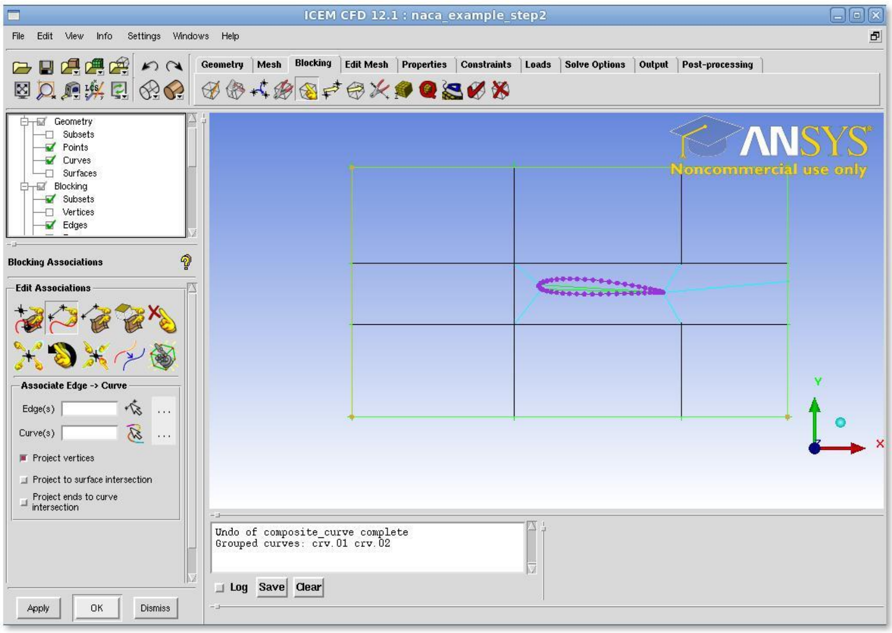


Figure 6: Splitting the Original Block

- 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 “Ogrid” around the aerofoil. Go to the Blocking Tab → Split Block (Second from left) and then select the Ogrid 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. Your screen should look something like Figure 7 whilst you are doing this.


ICEM CFD 12.1 : naca_example_step2
File Edit View Info Settings Windows Help
Geometry Mesh Blocking Edit Mesh Properties Constraints Loads Solve Options Output Post-processing
Geometry
Subsets
Points
Curves
Surfaces
Blocking
Subsets
Vertices
Edges
Blocking Associations
Edit Associations
ANSYS
Noncommercial use only
Y
X
Log Save Clear
Apply OK Dismiss

Associate Edge -> Curve

Edge(s)

Curve(s)

☒ Project vertices

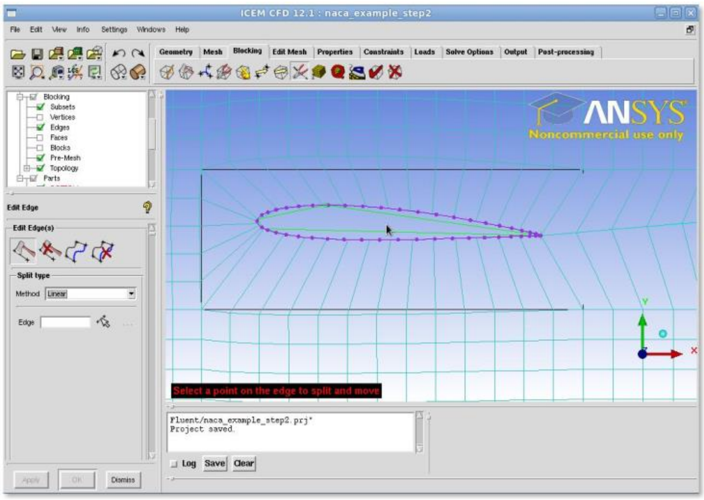
□ Project to surface intersection

□ Project ends to curve intersection

```
Undo of composite_curve complete
Grouped curves: crv.01 crv.02
```

Figure 8: Trailing Edge Block Eliminated

- 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.
- This should look like Figure 9.


ICEM CFD 12.1 : naca_example_step2
File Edit View Info Settings Windows Help
Geometry Mesh Blocking Edit Mesh Properties Constraints Loads Solve Options Output Post-processing
Blocking
Subsets
Vertices
Edges
Faces
Blocks
Pre-Mesh
Topology
Parts
Edit Edge
Edit Edge(s)
ANSYS
Noncommercial use only
Y
X
Log Save Clear
Apply OK Dismiss

Split type

Method Linear

Edge

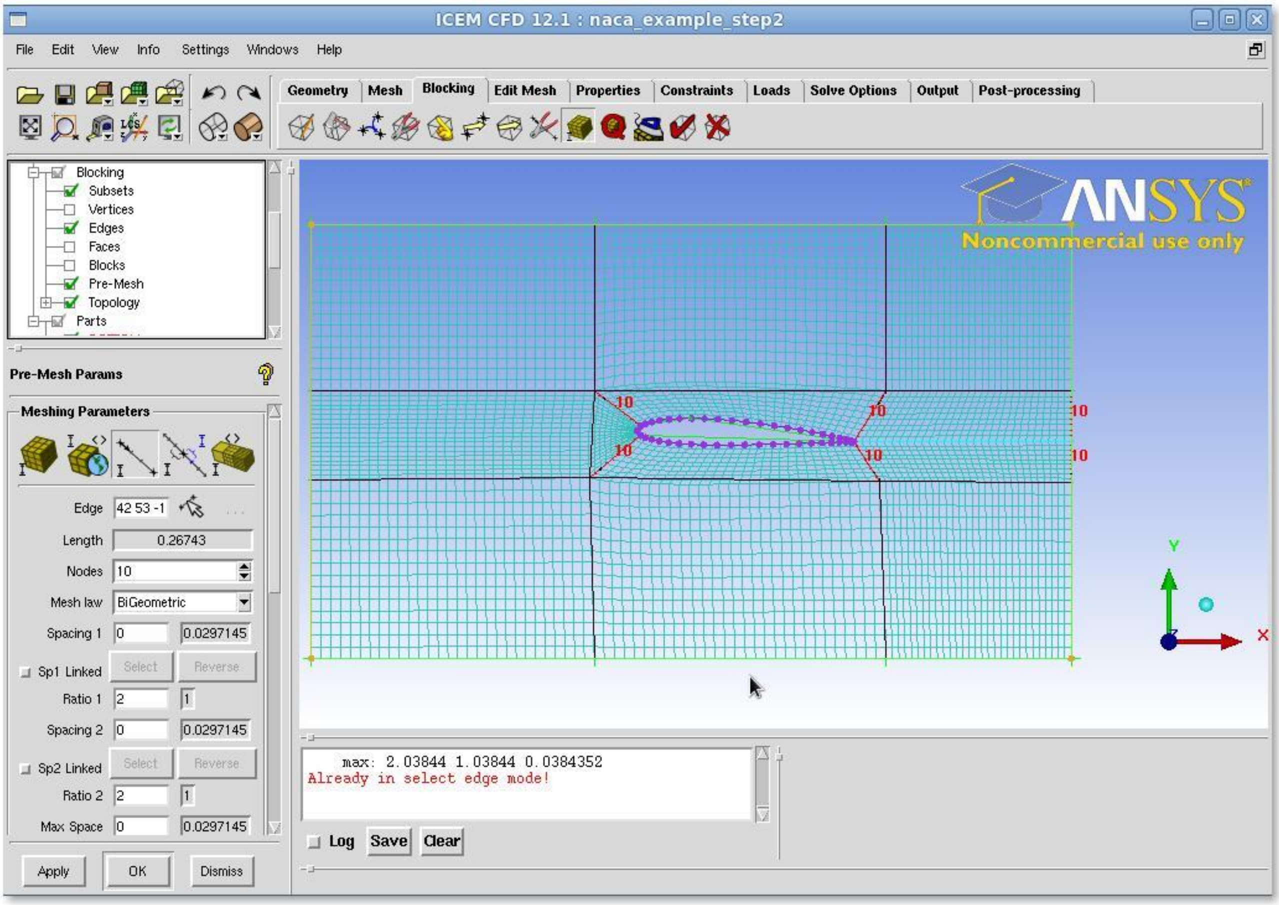
Select a point on the edge to split and move

```
Fluent/naca_example_step2.prj"
Project saved.
```

Figure 9: Setting up the Linear Point to nap the Mesh to the Outside of the Aerofoil

- 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.

After some work and moving points around, along with changing the number of points, the mesh is shown in Figure 10.



Edge 42 53 -1

Length 0.26743

Nodes 10

Mesh law BiGeometric

Spacing 1 0 0.0297145

□ Sp1 Linked

Ratio 1 2 1

Spacing 2 0 0.0297145

□ Sp2 Linked

Ratio 2 2 1

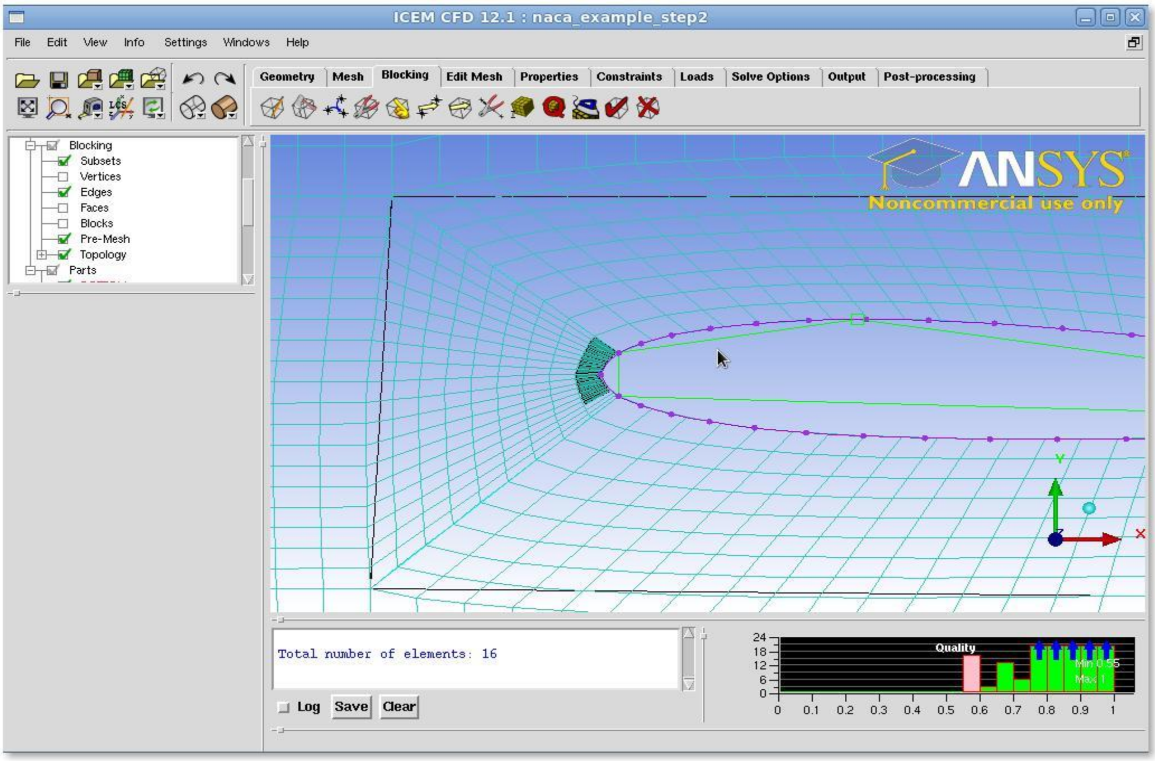
Max Space 0 0.0297145

```
   max: 2.03844 1.03844 0.0384352
Already in select edge mode!
```

Figure 10: A First Attempt at the Mesh

- 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.
- For our mesh this looks like Figure 11.

This isn't a great mesh and if you wanted to actually trust the results from it you'd need to do more work.



```
Total number of elements: 16
```

Figure 11: *Finding the Lowest Quality Elements*

Step 4: Set Up Problem in FLUENT

Launch FLUENT

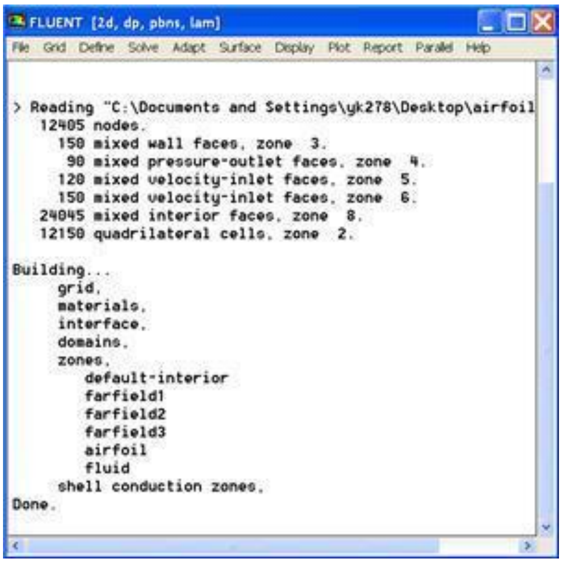
Start > Programs > Fluent Inc > FLUENT 6.3.26

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***. The following should appear in the FLUENT window:


FLUENT [2d, dp, pbns, lam]
File Grid Define Solve Adapt Surface Display Plot Report Parallel Help

```
> Reading "C:\Documents and Settings\yk278\Desktop\airfoil
   12405 nodes.
     150 mixed wall faces, zone  3.
      90 mixed pressure-outlet faces, zone  4.
     120 mixed velocity-inlet faces, zone  5.
     150 mixed velocity-inlet faces, zone  6.
   24045 mixed interior faces, zone  8.
   12150 quadrilateral cells, zone  2.

Building...
     grid,
     materials,
     interface,
     domains,
     zones,
        default-interior
        farfield1
        farfield2
        farfield3
        airfoil
        fluid
     shell conduction zones,
Done.
```

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.

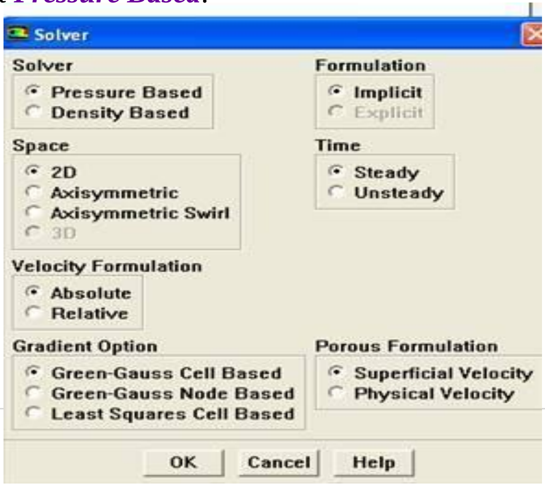
Zoom into the airfoil.

Where are the nodes clustered? Why?

Define Properties

Define > Models > Solver...

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



Solver
● Pressure Based
○ Density Based

Formulation
● Implicit
○ Explicit

Space
● 2D
○ Axisymmetric
○ Axisymmetric Swirl
○ 3D

Time
● Steady
○ Unsteady

Velocity Formulation
● Absolute
○ Relative

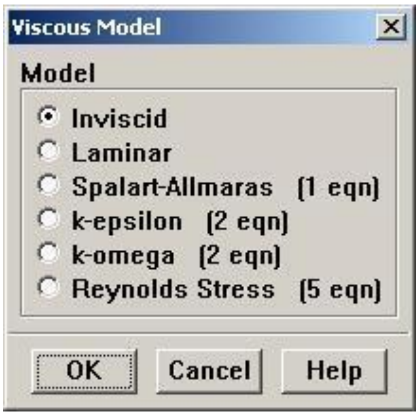
Gradient Option
● Green-Gauss Cell Based
○ Green-Gauss Node Based
○ Least Squares Cell Based

Porous Formulation
● Superficial Velocity
○ Physical Velocity

Click ***OK***.

Define > Models > Viscous

Select ***Inviscid*** under ***Model***



Viscous Model

Model

- ● Inviscid
- ○ Laminar
- ○ Spalart-Allmaras (1 eqn)
- ○ k-epsilon (2 eqn)
- ○ k-omega (2 eqn)
- ○ Reynolds Stress (5 eqn)

Click ***OK***.

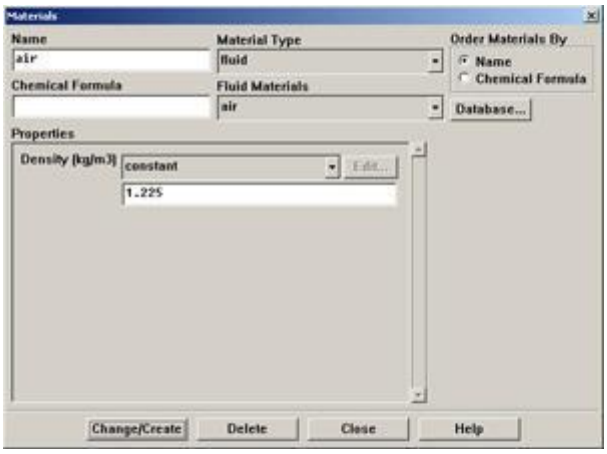
Define > Models > Energy

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^3.



Materials

Name: air

Material Type: fluid

Order Materials By: ● Name ○ Chemical Formula

Chemical Formula:

Fluid Materials: air

Properties

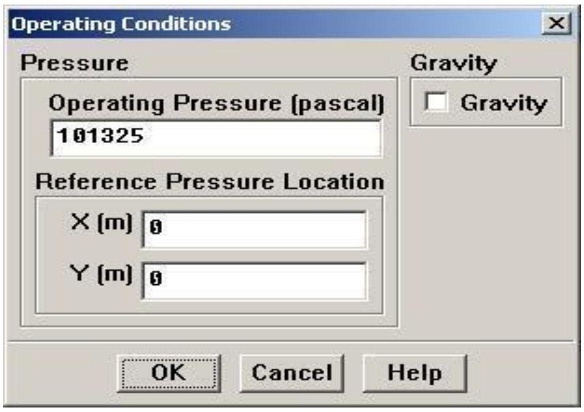
Density (kg/m3): constant

1.225

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.



Operating Conditions

Pressure

Operating Pressure (pascal) 101325

Reference Pressure Location

X (m) 0

Y (m) 0

Gravity

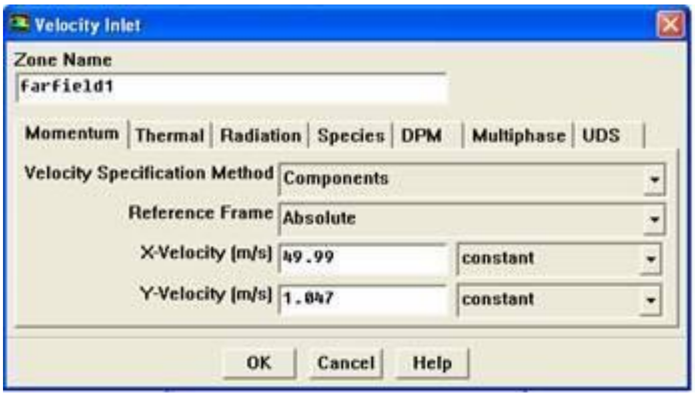
□ Gravity

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*cos(1.2)=49.99. (Note that 1.2° is used as our angle of attack instead of 2° to adjust for the error caused by assuming the airfoil to be 2D instead of 3D.)



Velocity Inlet

Zone Name farfield1

Velocity Specification Method: Components

Reference Frame: Absolute

X-Velocity (m/s) 49.99 constant

Y-Velocity (m/s) 1.047 constant

Click ***OK***.

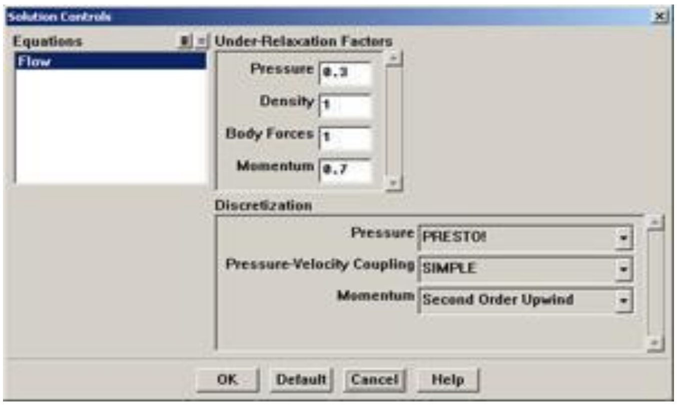
Set ***farfield3*** to ***pressure-outlet*** boundary type, click ***Set...*** and set the ***Gauge Pressure*** at this boundary to 0. Click ***OK***.

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***.



Equations

Flow

Under-Relaxation Factors

Pressure	0.3
Density	1
Body Forces	1
Momentum	0.7

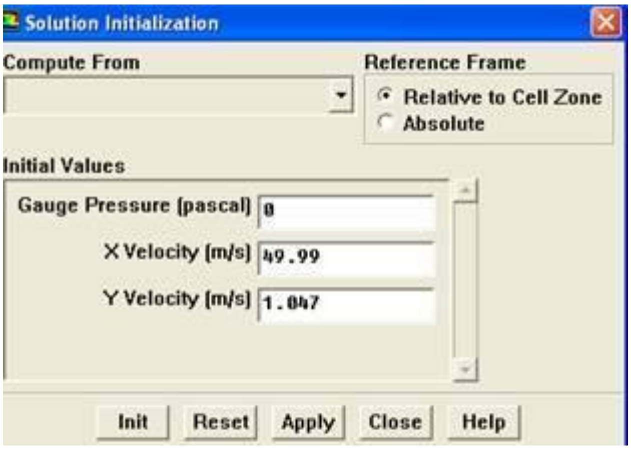
Discretization

Pressure	PRESTO!
Pressure-Velocity Coupling	SIMPLE
Momentum	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***.



Compute From

Reference Frame

● Relative to Cell Zone

○ Absolute

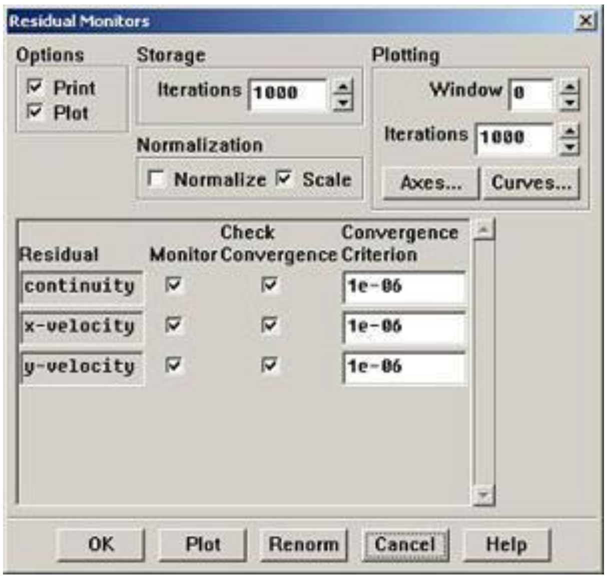
Initial Values

Gauge Pressure (pascal)	0
X Velocity (m/s)	49.99
Y Velocity (m/s)	1.047

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.



Options: ☑ Print ☑ Plot

Storage: Iterations 1000

Normalization: □ Normalize ☑ Scale

Plotting: Window 0, Iterations 1000

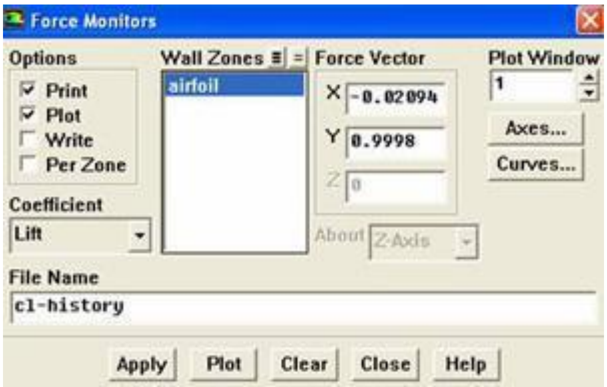
Residual	Monitor	Check Convergence	Convergence Criterion
continuity	☑	☑	1e-06
x-velocity	☑	☑	1e-06
y-velocity	☑	☑	1e-06

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°)=-020942 and ***Y*** to cos(1.2°)=0.9998.



Options: ☑ Print ☑ Plot □ Write □ Per Zone

Coefficient: Lift

Wall Zones: airfoil

Force Vector: X -0.02094, Y 0.9998, Z 0

About Z-Axis

Plot Window 1

File Name: cl-history

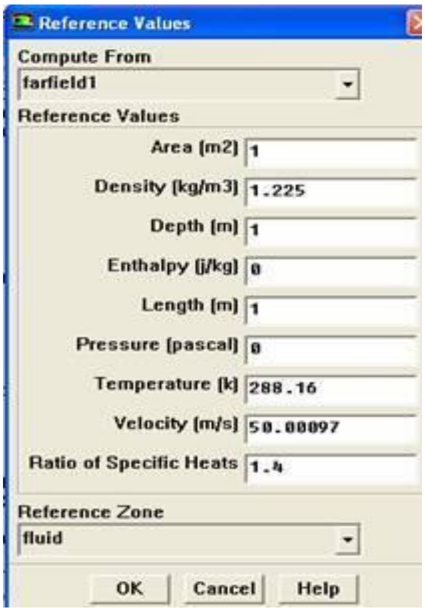
Click ***Apply*** for these changes to take effect.

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°)=0.9998

and ***Y*** to sin(1.2°)=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***.



Reference Values

Compute From

farfield1

Reference Values

Area (m2) 1

Density (kg/m3) 1.225

Depth (m) 1

Enthalpy (j/kg) 0

Length (m) 1

Pressure (pascal) 0

Temperature (k) 288.16

Velocity (m/s) 50.00097

Ratio of Specific Heats 1.4

Reference Zone

fluid

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 compared 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
```

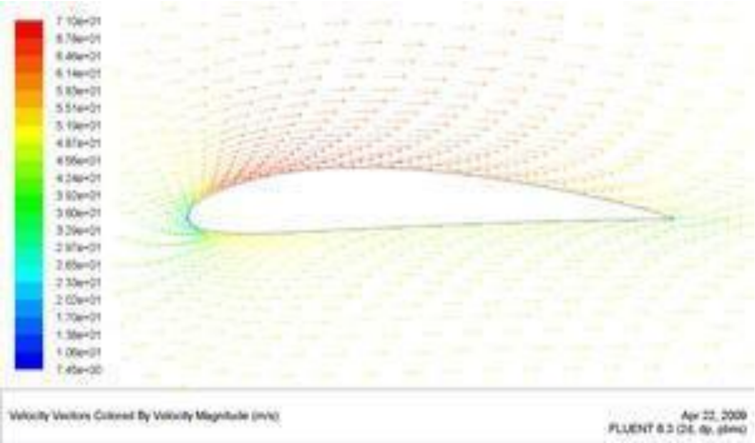
From FLUENT main window, we see that the lift coefficient is 0.647.

Plot Velocity Vectors

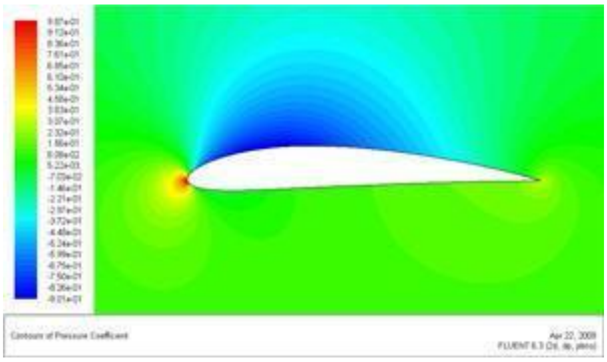
Let's see the velocity vectors along the airfoil.

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.



EXPERIMENT – 9

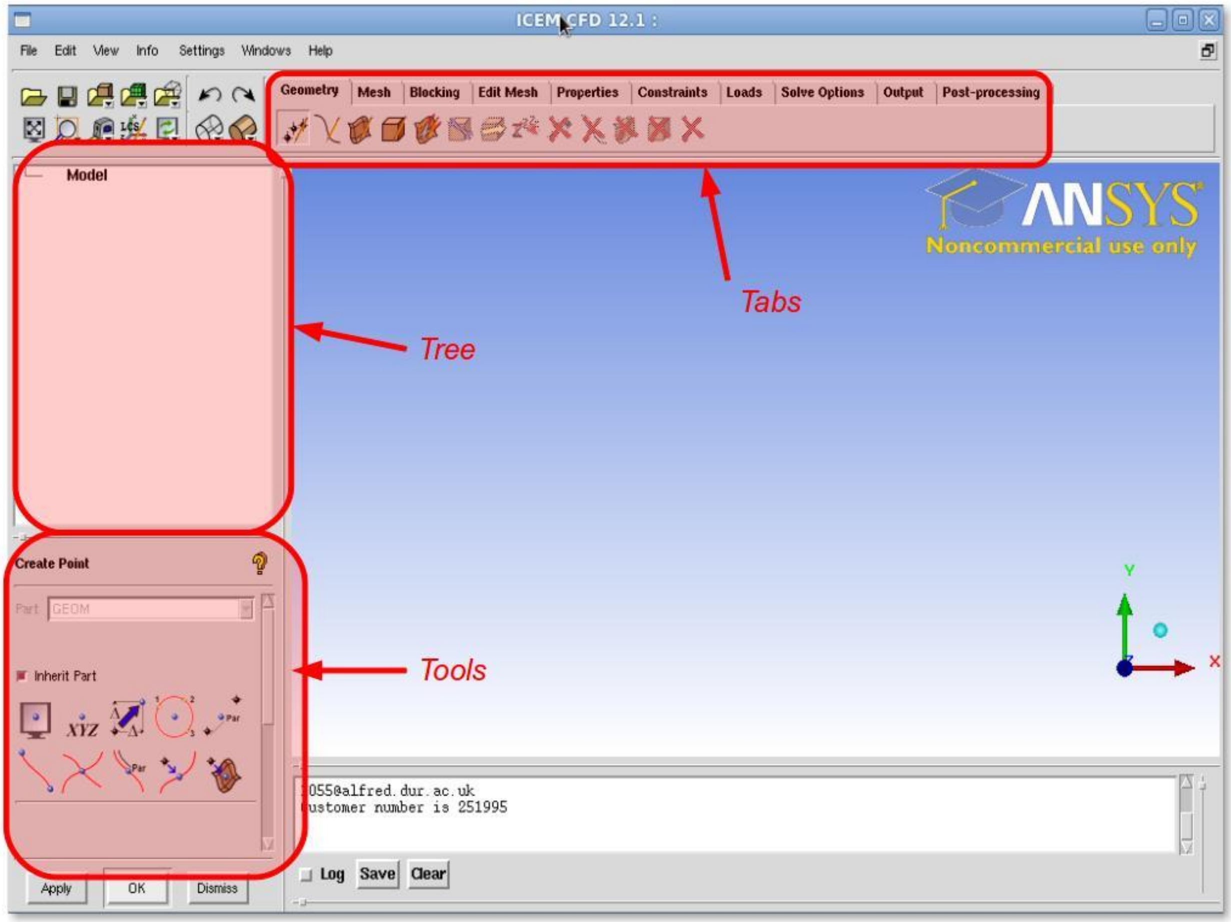
FLOW OVER A SYMMETRIC AEROFOIL

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.***

The initial screen looks something like:


ICEM CFD 12.1 :
File Edit View Info Settings Windows Help
Geometry Mesh Blocking Edit Mesh Properties Constraints Loads Solve Options Output Post-processing
Model
ANSYS
Noncommercial use only
Tabs
Tree
Tools
Create Point
Part GEOM
Inherit Part
XYZ
Y
X
Apply OK Dismiss
Log Save Clear

```
0055@alfred.dur.ac.uk
ustomer number is 251995
```

Figure 1: The icemcfd screen.

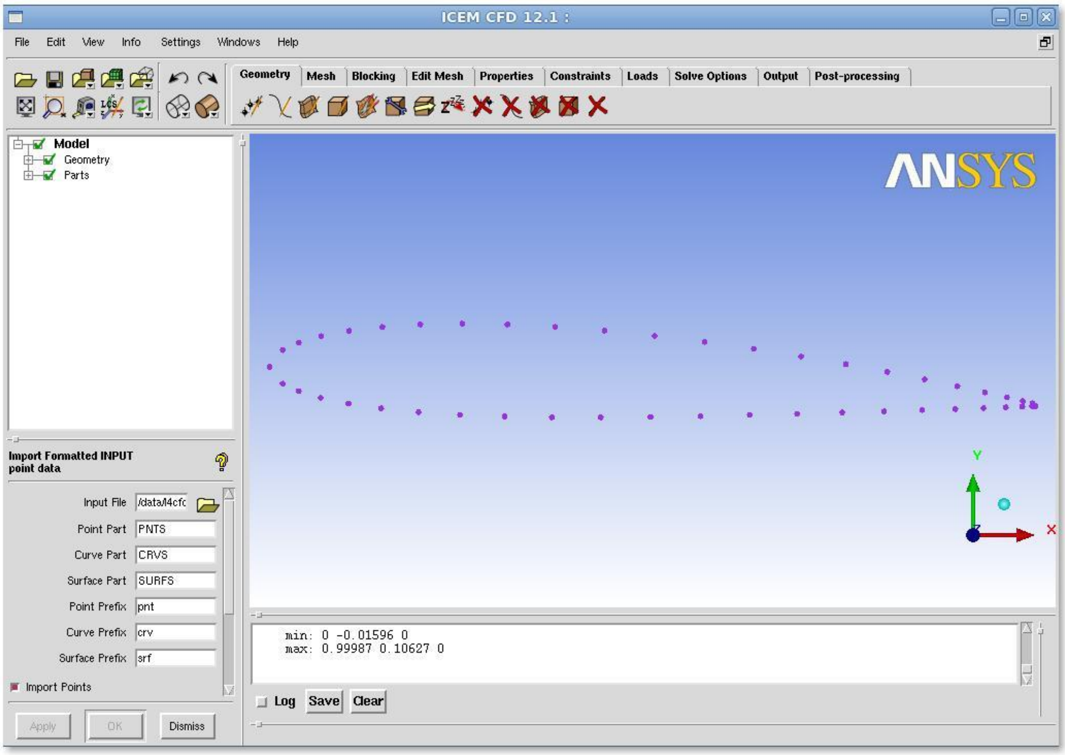
As well as the conventional display area and menus there are three areas of note:

- Tabs: These provide access to broad categories of tools which enable the meshing to take place.
- Tools: This area in the bottom left is where the tools are actually operated from
- Tree: In this area a hierarchical list of points, lines and so on will be built up as you work with the software.

icemcfd is a powerful package but does have something of a learning curve, so if something doesn't work the first time review the steps you took carefully and try again.

Importing the Geometry

- Start ***icemcfd***
- Obtain the NACA 2412 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
- You should have a screen that looks like Figure 2:



Import Formatted INPUT point data

Input File /data/l4cfc

Point Part PNTS

Curve Part CRVS

Surface Part SURFS

Point Prefix pnt

Curve Prefix crv

Surface Prefix srf

■ Import Points

```
min: 0 -0.01596 0
max: 0.99987 0.10627 0
```

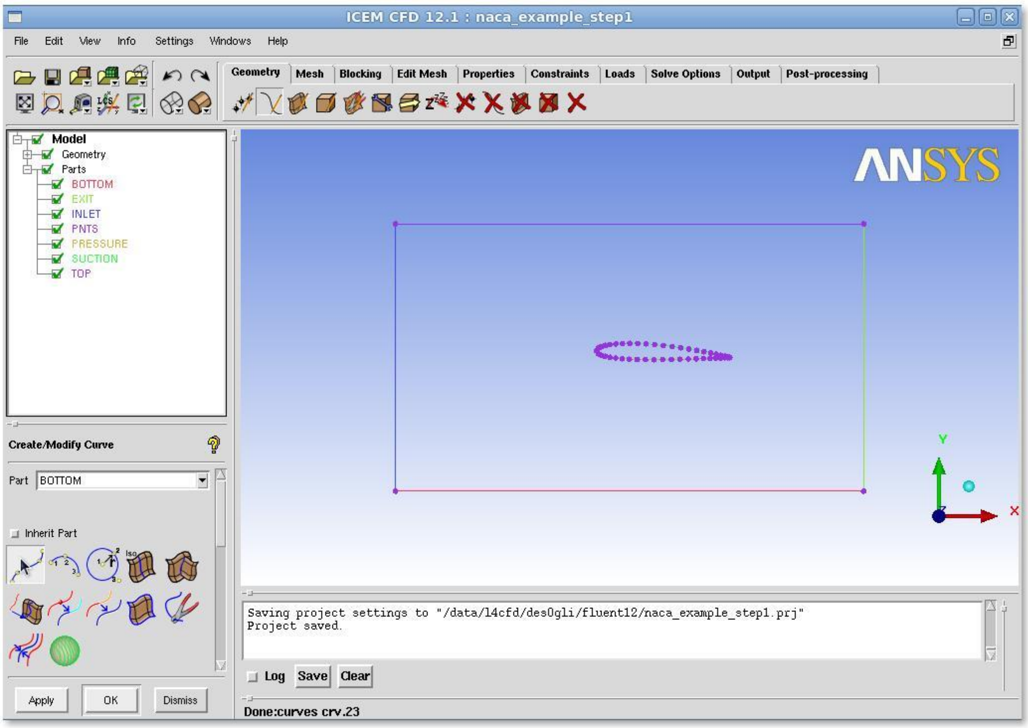
Figure 2: Point Input to icemcfd

- 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

Hint: Pressing F9 suspends the Selection Mode and allows you to use the mouse to control the view, hitting F9 again puts you back into Selection Mode.

- 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. Your screen should look like Figure 3.



```
Saving project settings to "/data/l4cfd/des0gli/fluent12/naca_example_step1.prj"
Project saved.
```

```
Done:curves crv.23
```

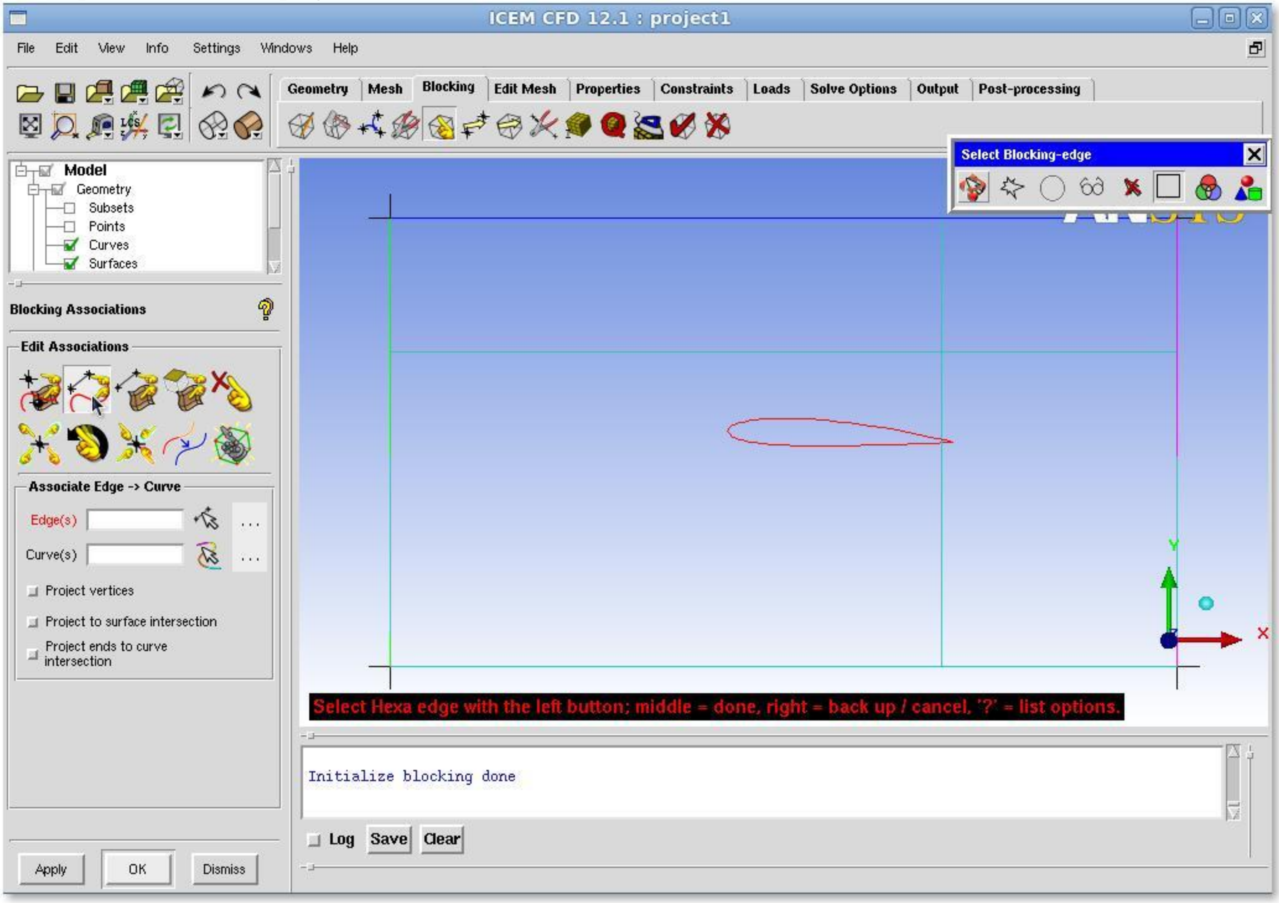
Figure 3: Aerofoil and Calculation Domain.

- 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 “Initalize 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.



Select Hexa edge with the left button; middle = done, right = back up / cancel, '?' = list options.

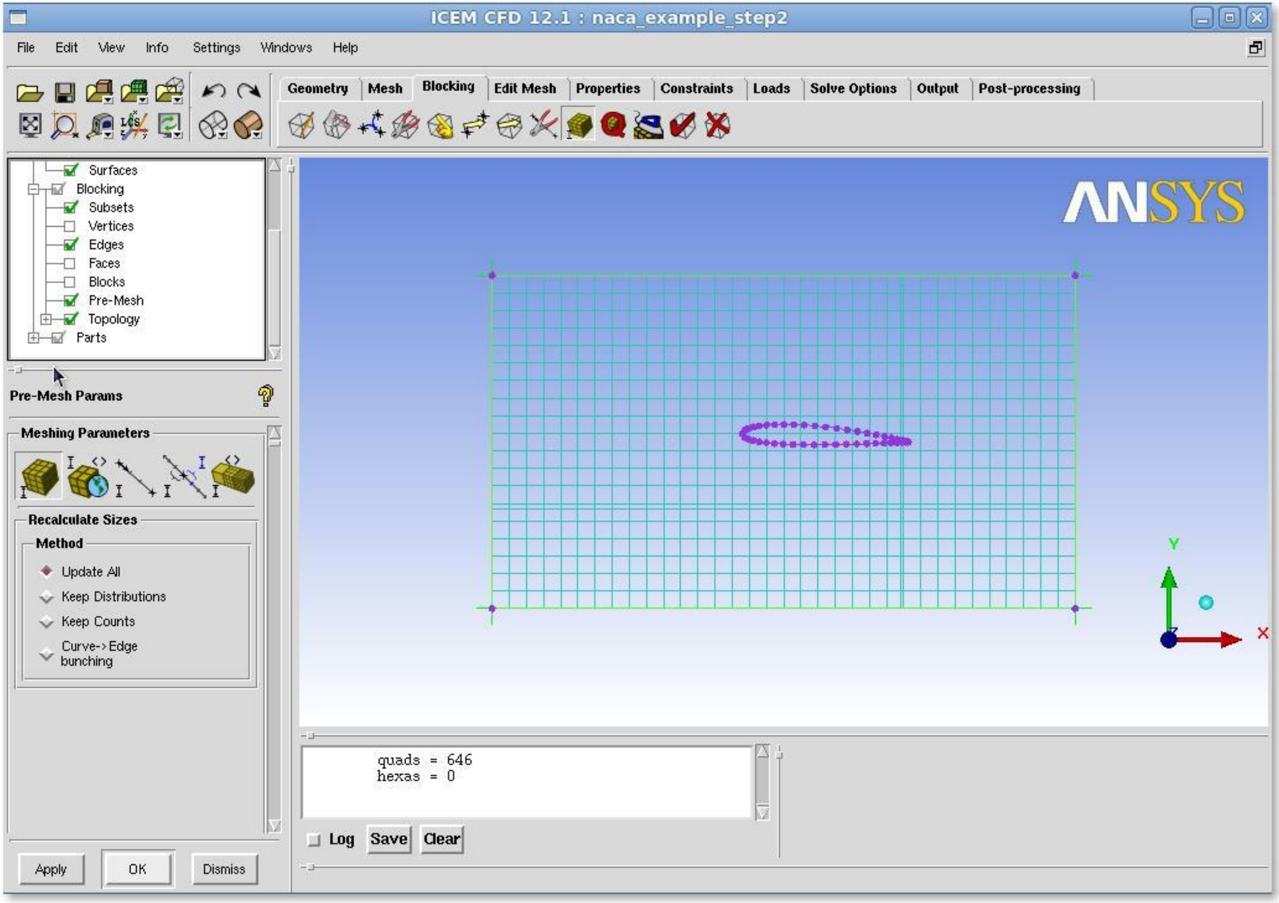
```
Initialize blocking done
```

Figure 4: Setting up the Blocking

- 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.

Hint: This is a part of the tutorial that confuses people a lot. The edges (part of the blocking) and the curves (geometry) are actually on top of each other on the screen. Although they are physically coincident on the screen there is no logical connection between them until you make one using the associate tool.

- 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 (9th 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.



Method

● Update All
○ Keep Distributions
○ Keep Counts
○ Curve->Edge bunching

```
        quads = 646
        hexas = 0
```

Figure 5: The Pre-Mesh View turned on.

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.

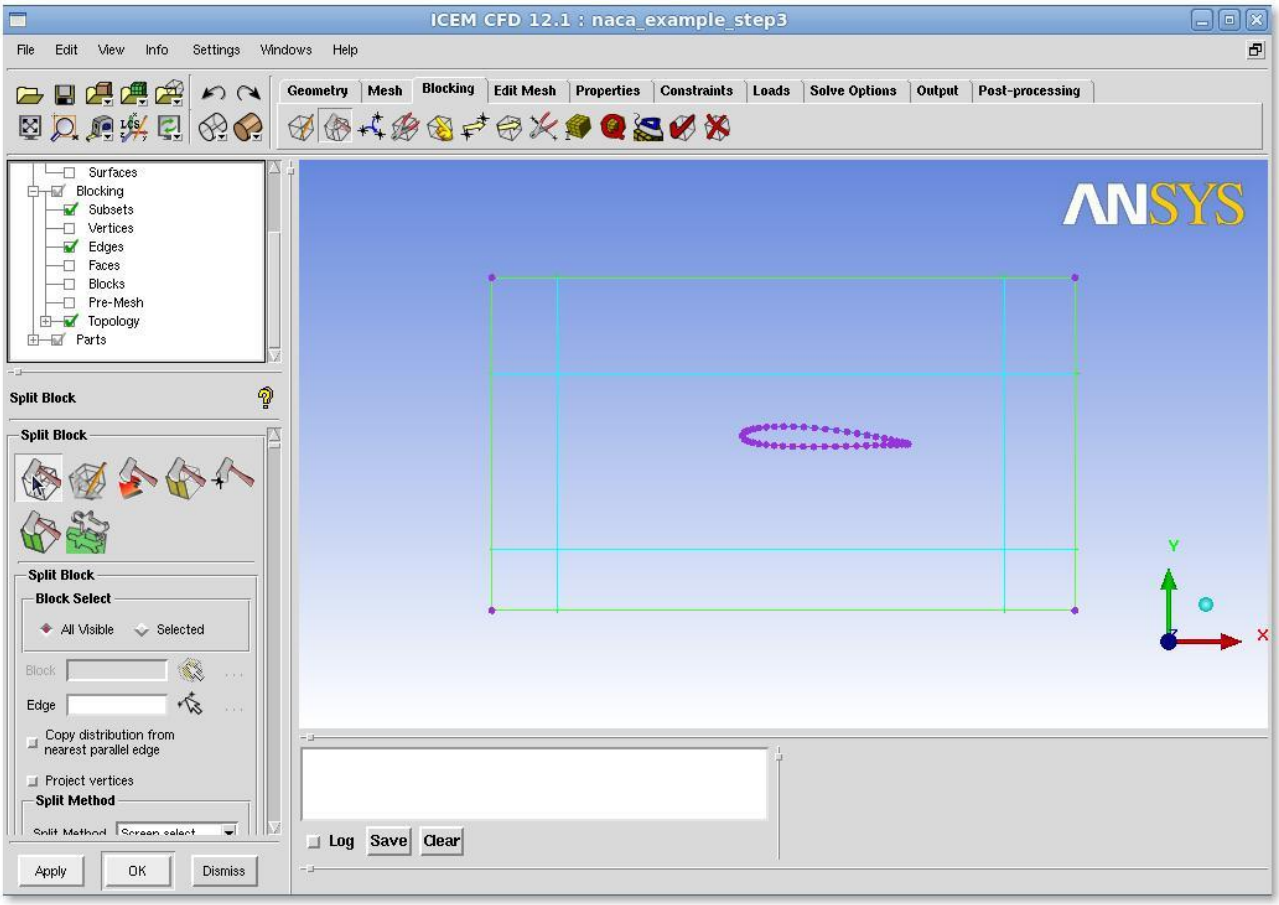
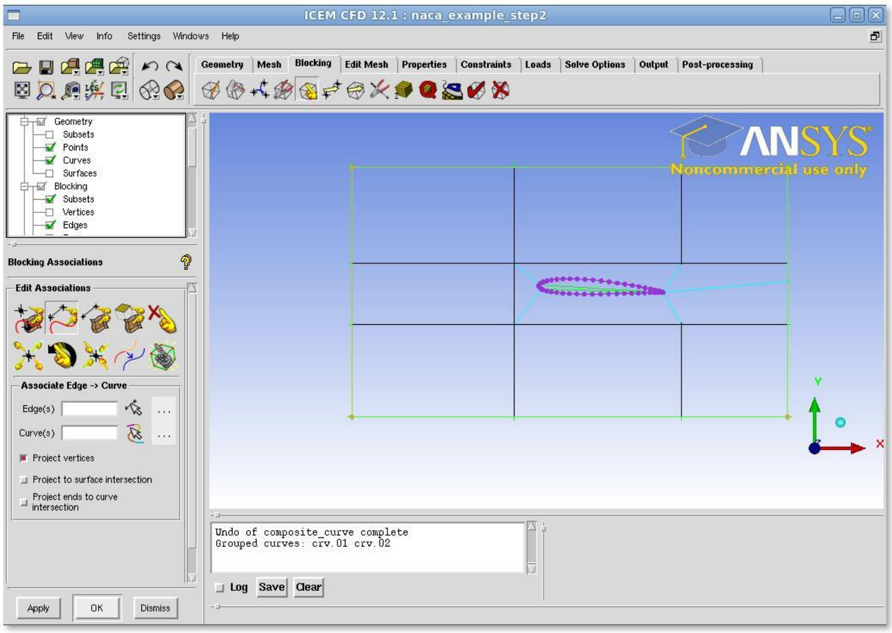


Figure 6: Splitting the Original Block

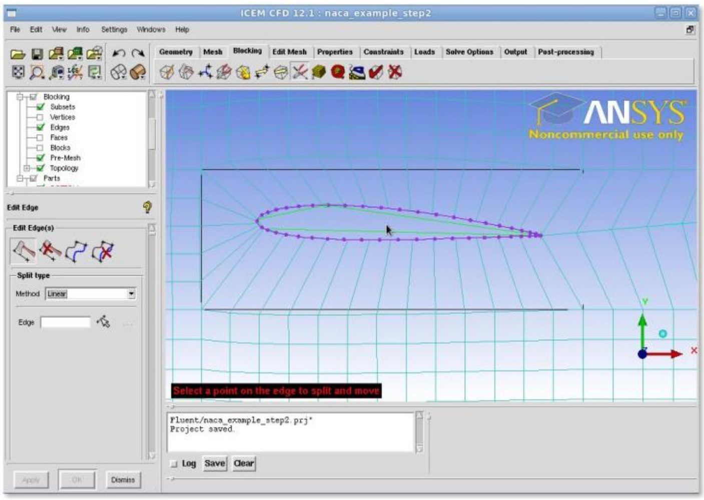
- First further divide up the Blocks. Blocking Tab → Split Block and then select the Split Block tool. Use this tool to split the block upstream and downstream of the aerofoil and above and below the trailing edge. Again left and middle clicking is the most efficient method. Your screen should look something like Figure 6
- Now we want to add a special type of block called an “Ogrid” around the aerofoil. Go to the Blocking Tab → Split Block (Second from left) and then select the Ogrid 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. Your screen should look something like Figure 7 whilst you are doing this.



```
Undo of composite_curve complete
Grouped curves: crv.01 crv.02
```

Figure 8: Trailing Edge Block Eliminated

- 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.
- This should look like Figure 9.



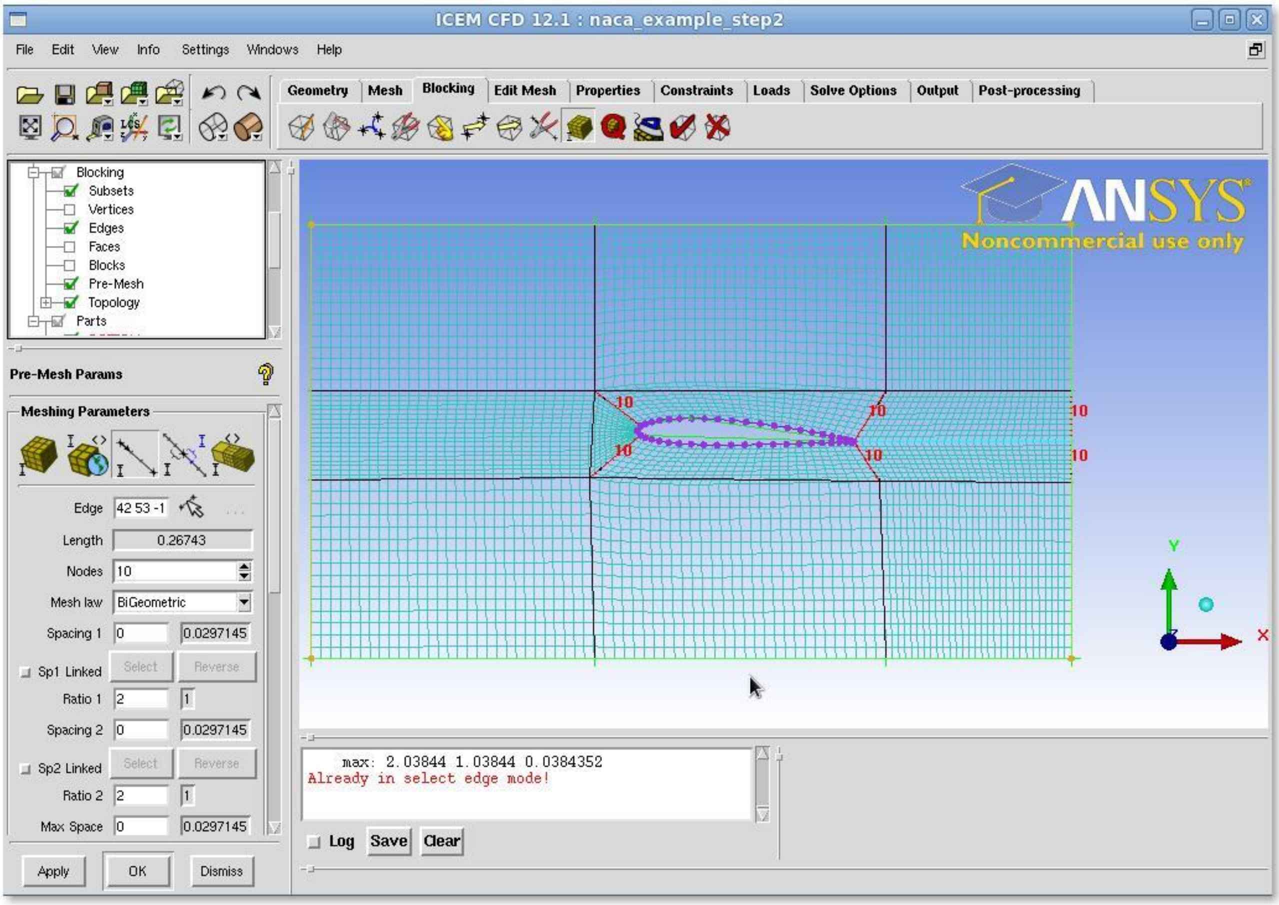
Select a point on the edge to split and move

```
Fluent/naca_example_step2.prj"
Project saved.
```

Figure 9: Setting up the Linear Point to nap the Mesh to the Outside of the Aerofoil

- 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.

After some work and moving points around, along with changing the number of points, the mesh is shown in Figure 10.



Edge 42 53 -1

Length 0.26743

Nodes 10

Mesh law BiGeometric

Spacing 1 0 0.0297145

□ Sp1 Linked

Ratio 1 2 1

Spacing 2 0 0.0297145

□ Sp2 Linked

Ratio 2 2 1

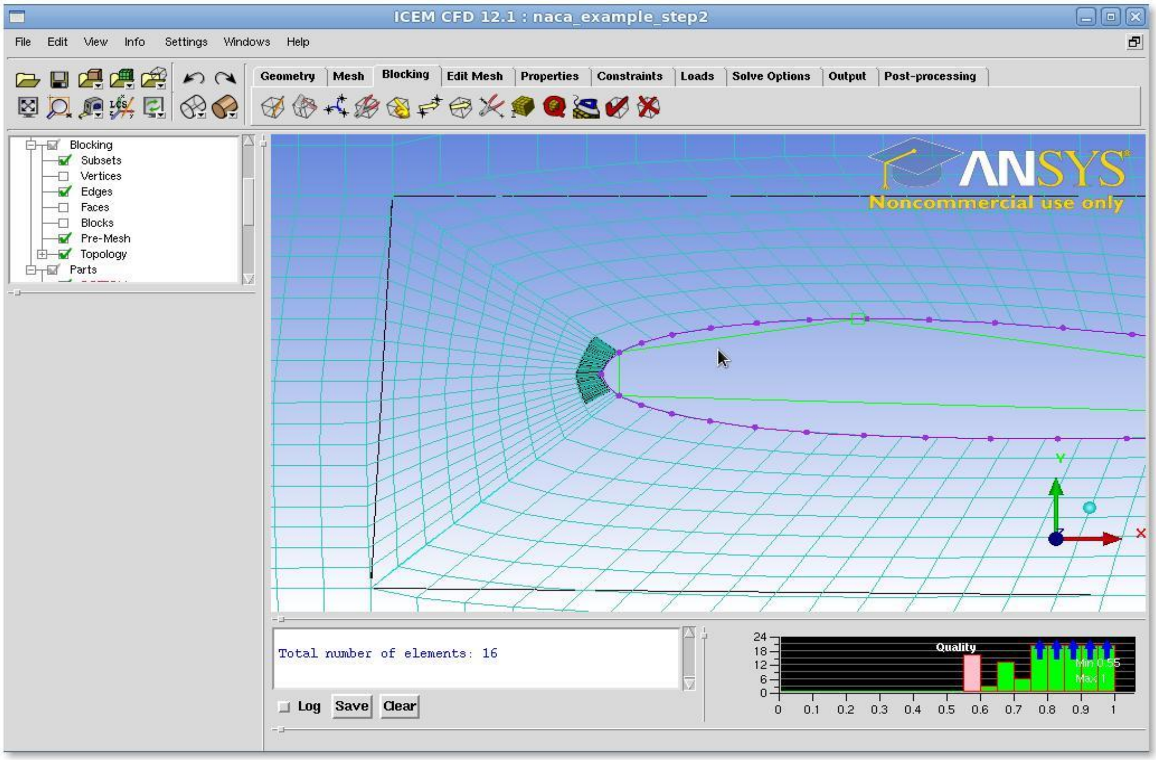
Max Space 0 0.0297145

```
    max: 2.03844 1.03844 0.0384352
Already in select edge mode!
```

Figure 10: A First Attempt at the Mesh

- 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.
- For our mesh this looks like Figure 11.

This isn't a great mesh and if you wanted to actually trust the results from it you'd need to do more work.



```
Total number of elements: 16
```

Figure 11: Finding the Lowest Quality Elements

Step 4: Set Up Problem in FLUENT

Launch FLUENT

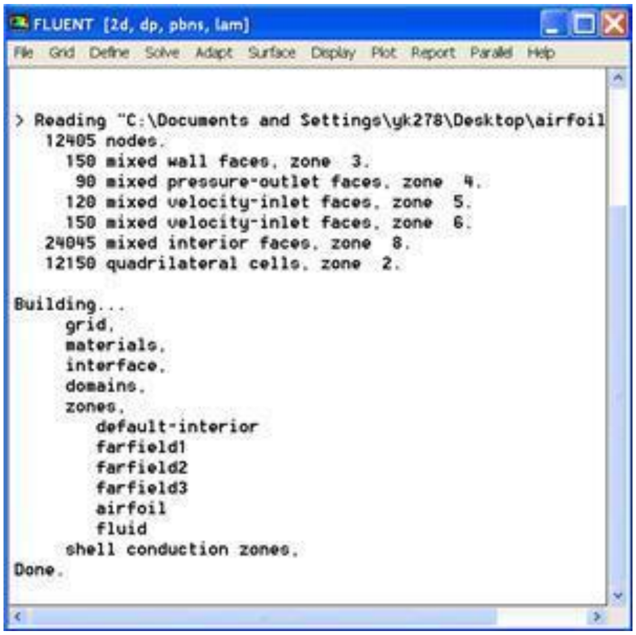
Start > Programs > Fluent Inc > FLUENT 6.3.26

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***. The following should appear in the FLUENT window:


FLUENT [2d, dp, pbns, lam]
File Grid Define Solve Adapt Surface Display Plot Report Parallel Help

```
> Reading "C:\Documents and Settings\yk278\Desktop\airfoil
   12405 nodes.
     150 mixed wall faces, zone  3.
      90 mixed pressure-outlet faces, zone  4.
     120 mixed velocity-inlet faces, zone  5.
     150 mixed velocity-inlet faces, zone  6.
   24045 mixed interior faces, zone  8.
   12150 quadrilateral cells, zone  2.

Building...
     grid,
     materials,
     interface,
     domains,
     zones,
        default-interior
        farfield1
        farfield2
        farfield3
        airfoil
        fluid
     shell conduction zones,
Done.
```

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.

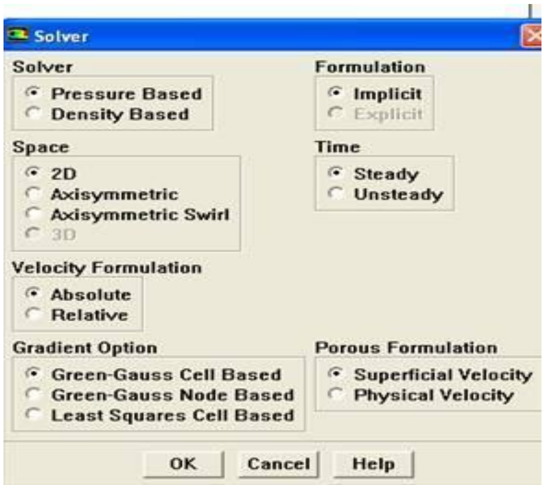
Zoom into the airfoil.

Where are the nodes clustered? Why?

Define Properties

Define > Models > Solver...

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



Solver

Solver
● Pressure Based
○ Density Based

Formulation
● Implicit
○ Explicit

Space
● 2D
○ Axisymmetric
○ Axisymmetric Swirl
○ 3D

Time
● Steady
○ Unsteady

Velocity Formulation
● Absolute
○ Relative

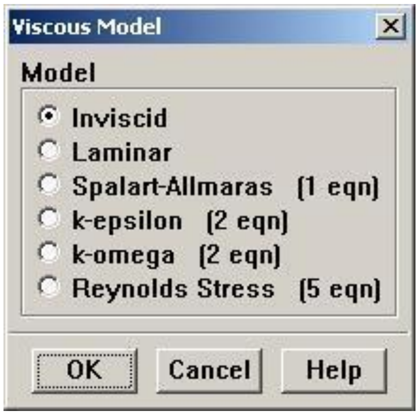
Gradient Option
● Green-Gauss Cell Based
○ Green-Gauss Node Based
○ Least Squares Cell Based

Porous Formulation
● Superficial Velocity
○ Physical Velocity

Click ***OK***.

Define > **Models** > **Viscous**

Select ***Inviscid*** under ***Model***



Viscous Model

Model
● Inviscid
○ Laminar
○ Spalart-Allmaras (1 eqn)
○ k-epsilon (2 eqn)
○ k-omega (2 eqn)
○ Reynolds Stress (5 eqn)

Click ***OK***.

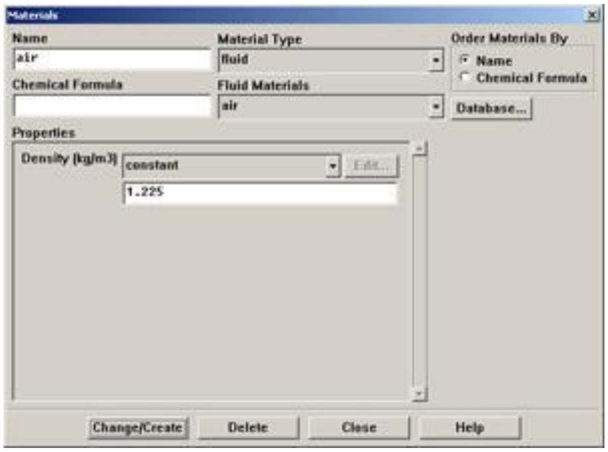
Define > **Models** > **Energy**

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^3.



Name: air

Material Type: fluid

Order Materials By: ● Name ○ Chemical Formula

Chemical Formula:

Fluid Materials: air

Properties

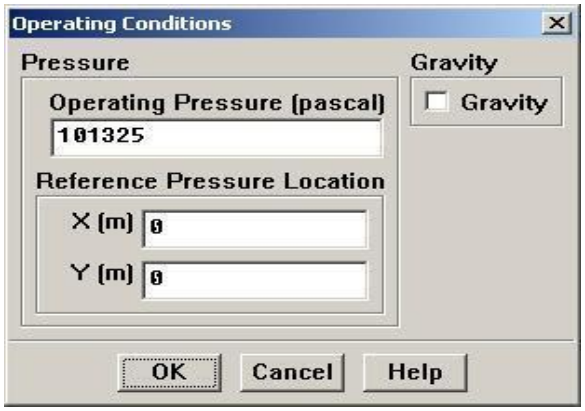
Density (kg/m3): constant

1.225

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.



Operating Pressure (pascal): 101325

Reference Pressure Location

X (m): 0

Y (m): 0

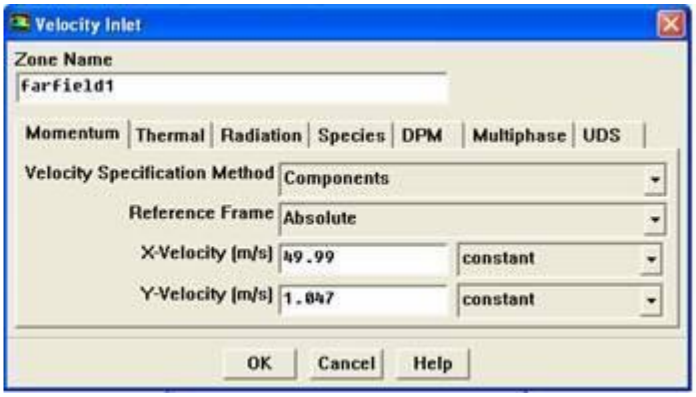
□ Gravity

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*cos(1.2)=49.99. (Note that 1.2° is used as our angle of attack instead of 2° to adjust for the error caused by assuming the airfoil to be 2D instead of 3D.)



Zone Name: farfield1

Velocity Specification Method: Components

Reference Frame: Absolute

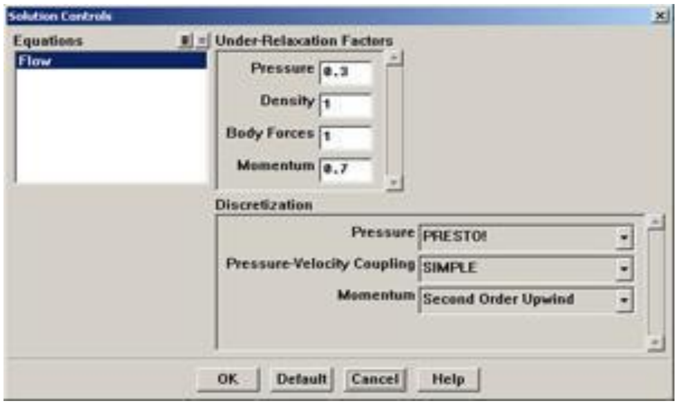
X-Velocity [m/s]: 49.99 constant

Y-Velocity [m/s]: 1.047 constant

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***.



Equations: Flow

Under-Relaxation Factors

Pressure: 0.3

Density: 1

Body Forces: 1

Momentum: 0.7

Discretization

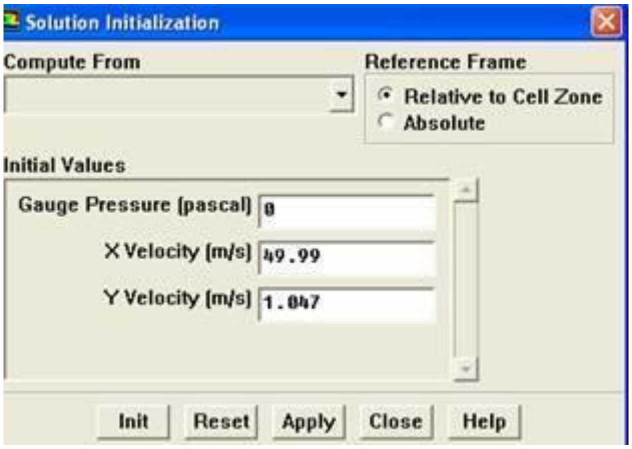
Pressure: PRESTO!

Pressure-Velocity Coupling: SIMPLE

Momentum: 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***.



Compute From

Reference Frame

● Relative to Cell Zone

○ Absolute

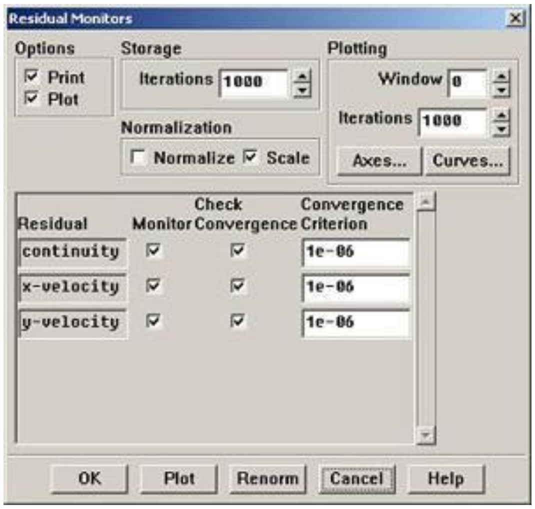
Initial Values

Gauge Pressure (pascal)	0
X Velocity (m/s)	49.99
Y Velocity (m/s)	1.047

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.



Options: ☑ Print ☑ Plot

Storage: Iterations 1000

Normalization: □ Normalize ☑ Scale

Plotting: Window 0, Iterations 1000

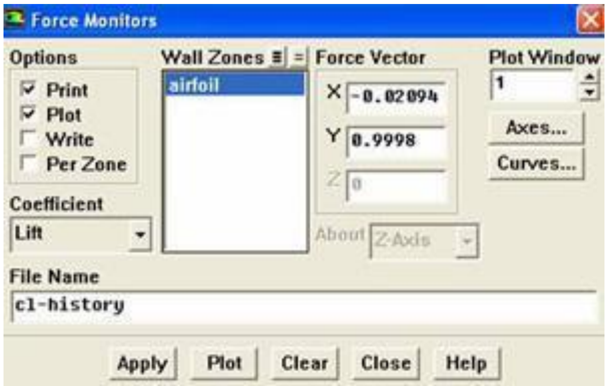
Residual	Monitor	Check Convergence	Convergence Criterion
continuity	☑	☑	1e-06
x-velocity	☑	☑	1e-06
y-velocity	☑	☑	1e-06

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°)=-020942 and ***Y*** to cos(1.2°)=0.9998.



Options

- ☑ Print
- ☑ Plot
- □ Write
- □ Per Zone

Coefficient: Lift

Wall Zones: airfoil

Force Vector: X -0.02094, Y 0.9998, Z 0

About: Z-Axis

Plot Window: 1

File Name: cl-history

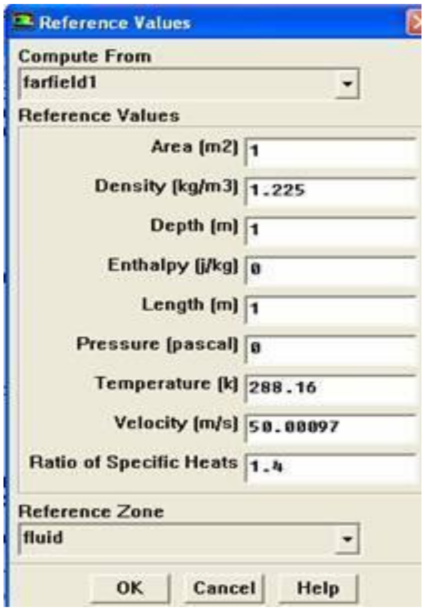
Click *Apply* for these changes to take effect.

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°)=0.9998

and *Y* to sin(1.2°)=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*.



Compute From: farfield1

Reference Values

Area (m2)	1
Density (kg/m3)	1.225
Depth (m)	1
Enthalpy (j/kg)	0
Length (m)	1
Pressure (pascal)	0
Temperature (k)	288.16
Velocity (m/s)	50.00097
Ratio of Specific Heats	1.4

Reference Zone: fluid

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 compared 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
```

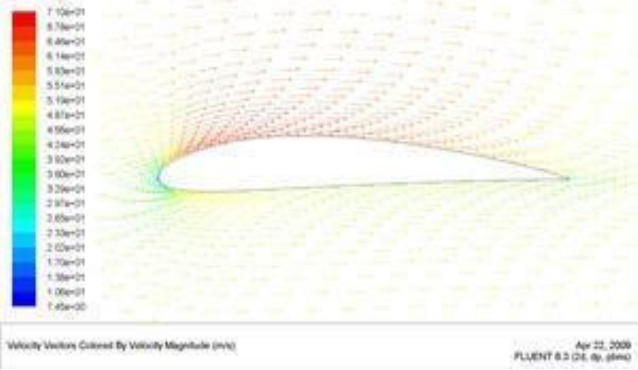
From FLUENT main window, we see that the lift coefficient is 0.647.

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.

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.

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 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.

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...**

Hint: *If you are unsure of which inlet zone corresponds to the cold inlet, you can probe the mesh display using the right mouse button or the probe toolbar button (✎) as described in a previous step. The information will be displayed in the **ANSYS FLUENT** console, and the zone you probed will be automatically selected from the **Zone** selection list in the **Boundary Conditions** task page.*

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.

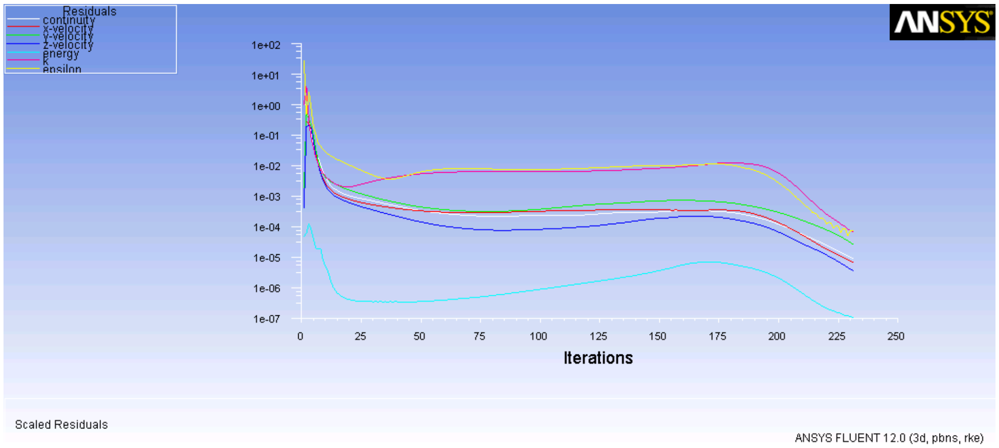
(a) Start the calculation by requesting 250 iterations.

✥ **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**.

EXPERIMENT - 11

ONE DIMENSIONAL WAVE EQUATION USING EXPLICIT METHOD OF LAX

The wave equation considered here is an extremely simplified model of the physics of waves. Many facts about waves are not modeled by this simple system, including that wave motion in water can depend on the depth of the medium, that waves tend to disperse, and that waves of different frequency may travel at different speeds. However, as a first model of wave motion, the equation is useful because it captures a most interesting feature of waves, that is, their usefulness in transmitting signals.

This program solves the 1D wave equation of the form:

```
    Utt = c^2 Uxx
% matlab script waveeq1dfd.m
% finite difference scheme for the 1D wave equation
% fixed boundary conditions
% raised cosine initial conditions
%%%%%% begin global parameters

SR = 44100;                    % sample rate (Hz)
f0 = 441;                     % fundamental frequency (Hz)
TF = 1;                       % duration of simulation (s)
ctr = 0.7; wid = 0.1;            % center location/width of excitation
u0 = 1; v0 = 0;                % maximum initial displacement/velocity
rp = 0.3;                     % position of readout (0-1)
lambda = 1;                    % Courant number
gamma = 2*f0;                   % wave equation free parameter
k = 1/SR;                      % time step
NF = floor(SR*TF);                % duration of simulation (samples)
% stability condition/scheme parameters

h = gamma*k/lambda; N = floor(1/h); h = 1/N; lambda = gamma*k/h;
s0 = 2*(1-lambda^2); s1 = lambda^2;
rp_int = 1+floor(N*rp);           % rounded grid index for readout
rp_frac = 1+rp/h-rp_int;          % fractional part of readout location
xax = [0:N]'*h;
ind = sign(max(-(xax-ctr-wid/2).*(xax-ctr+wid/2),0));
rc = 0.5*ind.*(1+cos(2*pi*(xax-ctr)/wid));

% initialize grid functions and output

u2 = u0*rc; u1 = (u0+k*v0)*rc; u = zeros(N+1,1); out = zeros(NF,1);
for n=3:NF
   u(2:N) = -u2(2:N)+s0*u1(2:N)+s1*(u1(1:N-1)+u1(3:N+1));  % scheme calculation
   out(n) = (1-rp_frac)*u(rp_int)+rp_frac*u(rp_int+1);     % readout
   u2 = u1; u1 = u;                                  % update of grid variables
end
```

EXPERIMENT - 12

ONE DIMENSIONAL HEAT CONDUCTION EQUATION USING EXPLICIT METHOD

```
L=1;
t=1;
k=.001;
n=11;
nt=500;
dx=L/n;
dt=.002;
alpha=k*dt/dx^2;
T0(1)=400;
for j=1:nt
    for i=2:n
        T1(i)=T0(i)+alpha*(T0(i+1)-2*T0(i)+T0(i-1));
    end
    T0=T1;
end
plot(x,T1)
```