Lab Manual

ET-102L

Basic Aerodynamics – Lab

Institute of Aviation Studies
University of Management and Technology Lahore

Institute of Aviation Studies
University of Management and Technology

Course Outline

Course code: ET-102L   Course title: Basic Aerodynamics - Lab

Program	BSc AMET
Credit Hours	0.5
Duration	1 semester
Learning Methodology:	Lab instructions and experiment

Course Learning Outcomes (CLOs) and their Mapping to Program Learning Outcomes (PLOs):

Semester	Course Code	Title	Course Learning Outcomes	PLO 1 Engg .Tech. Knowledge	PLO 2 Problem Analysis	PLO 3 Solution Design	PLO 4 Investigation	PLO 5 Mod. Tool Usage	PLO 6 Engr. & Society	PLO 7 Env. &Sust.	PLO 8 Ethics	PLO 9 Team Work	PLO 10 Communication	PLO 11 Proj. Mgmt.	PLO 12 Lifelong Learning
1	ET-102L	Basic Aerodynamics	CLO 3: Analyze performance variables of aerodynamic bodies and airflow properties.					P 3							
			CLO 4: Effectively communicate experiment results through both written reports and oral presentation.										P 3		

Grade Evaluation Criteria

Components	Marks
Class Participation (Team work)	5%
Assignment/Project	15%
Viva	5%
Lab Report	15%
Final evaluation	60%
Total	100

List of resources:

- ANSYS Workbench 19.0
- High performance computers

List of Experiments:

Sr. No.	Objective	Experiment Number	CLOs
1	Introduction to Computational Fluid Dynamics (CFD) and software	1	01 to 06
2	To identify and create different aerofoil sections	2	
3	Studying the nature of airflow over NACA 0012 aerofoil	3	
4	Studying the nature of airflow over cylinder	4	
5	Studying the nature of airflow over flat plate	5	
6	Study the lift and drag characteristics of rectangular plate	6	
7	Study the lift and drag characteristics of NACA 0012 aerofoil.	7	

Experiment 1: Introduction to Computational Fluid Dynamics (CFD) and software

ANSYS ICEM CFD meshing software starts with advanced CAD/geometry readers and repair tools toallow the user to quickly progress to a variety of geometry-tolerant meshers and produce high-qualityvolume or surface meshes with minimal effort. Advanced mesh diagnostics, interactive and automatedmesh editing, output to a wide variety of computational fluid dynamics (CFD) and finite element analysis(FEA) solvers and multiphysics post-processing tools make ANSYS ICEM CFD a complete meshingsolution. ANSYS endeavors to provide a variety of flexible tools that can take the model from anygeometry to any solver in one modern and fully scriptable environment.

- Mesh from dirty CAD and/or faceted geometry such as STL
- Efficiently mesh large, complex models
- Hexa mesh (structured or unstructured) with advanced control
- Extended mesh diagnostics and advanced interactive mesh editing
- Output to a wide variety of CFD and FEA solvers as well as neutral formats

ANSYS ICEM CFD is a popular proprietary software package used for CAD and mesh generation. Someopen source software includes OpenFOAM, FeatFlow, Open FVM etc. Present discussion is applicableto ANSYS ICEM CFD software.It can create structured, unstructured, multi-block, and hybrid grids with different cell geometries.

Geometry modelling:

ANSYS ICEM CFD is meant to mesh a geometry already created using other dedicated CAD packages.Therefore, the geometry modelling features are primarily meant to 'clean-up' an imported CAD model.Nevertheless, there are some very powerful geometry creation, editing and repair (manual andautomated) tools available in ANSYS ICEM CFD which assist in arriving at the meshing stage quickly.Unlike the concept of volume in tools like GAMBIT, ICEM CFD rather treats a collection of surfaceswhich encompass a closed region as BODY. Therefore, the typical topological issues encountered inGAMBIT (e.g. face cannot be deleted since it is referenced by higher topology) don't show up here. Theemphasis in ICEM CFD to create a mesh is to have a 'water-tight' geometry. It means if there is a sourceof 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, ANSYS ICEM CFD especiallyhas the capability to do BUILD TOPOLOGY which removes unwanted surfaces and then you can viewif there are any 'holes' in the region of interest for meshing. Existence of holes would mean that thealgorithm which generates the mesh would cause the mesh to 'leak out' of the domain. Holes are typicallyidentified through the colour of the curves. The following is the colour coding in ANSYS ICEM CFD,after the BUILD TOPOLOGY option has been implemented:

- YELLOW: curve attached to a single surface - possibly a hole exists. In some cases this might be
- desirable for e.g., thin internal walls require at least one curve with single surface attached to it.
- RED: curve shared by two surface - the usual case.
- BLUE: curve shared by more than two surface.
- Green: Unattached Curves - not attached to any surface

Meshing approach and mesh

There are often some misunderstandings regarding structured/unstructured mesh, meshing algorithm andsolver. A mesh may look like a structured mesh but may/may not have been created using a structuredalgorithm based tool. For e.g., GAMBIT is an unstructured meshing tool. Therefore, even if it creates amesh that looks like a structured (single or multi-block) mesh through pain-staking efforts in geometrydecomposition, the algorithm employed was still an unstructured one. On top of it, most of the popularCFD tools like, ANSYS FLUENT, ANSYS CFX, Star CCM+, OpenFOAM, etc. are unstructured solverswhich can only work on an unstructured mesh even if we provide it with a structured looking meshcreated using structured/unstructured algorithm based meshing tools. ANSYS ICEM CFD can generate
both structured and unstructured meshes using structured or unstructured algorithms which can be givenas inputs to structured as well as unstructured solvers, respectively.

<u>Structured meshing strategy</u>

While simple ducts can be modelled using a single block, majority of the geometries encountered in reallife have to be modelled using multi-block strategies if at all it is possible.

The following are the different multi-block strategies available which can be implemented using ANSYSICEM CFD.

- O-grid
- C-grid
- Quarter O-grid
- H-grid

<u>Unstructured meshing strategy</u>

Unlike the structured approach for meshing, the unstructured meshing algorithm is more or less anoptimization problem, wherein, it is required to fill-in a given space (with curvilinear boundaries) withstandard shapes (e.g., triangle, quadrilaterals - 2D; tetrahedrals, hexahedrals, polyhedrals, prisms,pyramids - 3D) which have constraints on their size. The basic algorithms employed for doingunstructured meshing are:

 Octree (easiest from the user's perspective; robust but least control over the final cell count which isusually the highest)

Delaunay (better control over the final cell count but may have sudden jumps in the size of the elements)

 Advancing front (performs very smooth transition of the element sizes and may result in quite accuratebut high cell count)

<u>Best practices</u>

If using Octree -

- Perform volume meshing
- Improve the quality of the volume mesh using Edit Mesh options
- Create prism layers for boundary layer near the walls
- Improve the total mesh quality using Edit Mesh options

If using Delaunay or Advancing Front -

- Perform surface meshing
- Improve the quality of the surface mesh using Edit Mesh options
- Perform volume meshing
- Improve the quality of the volume mesh using Edit Mesh options
- Create prism layers for boundary layer near the walls
- Improve the total mesh quality using Edit Mesh options

basic viewport interaction

- use the **left** mouse button and drag to *rotate* the view
- use the **middle** mouse button to *pan* the viewimporting data

Creating a structured grid

The first thing to do when creating a structured grid is to create the geometry or a .tin file in ICEM. Youcan do this by manually creating it in ICEM or importing data into ICEM, for example 3-dimensionalpoint data from a .txt file.

The tools available are specified under the **geometry** tab. There are quite a number of tools and they canbe quite useful. However, it is suggested that some planning is done before beginning to make ageometry. There are tools specifically for curves.

- curves can be split or joined to other curves.
- Points can be created at cross-sections of curves.
- Surfaces can be created from curves.

All of this gives extra flexibility in the methods of designing a grid.

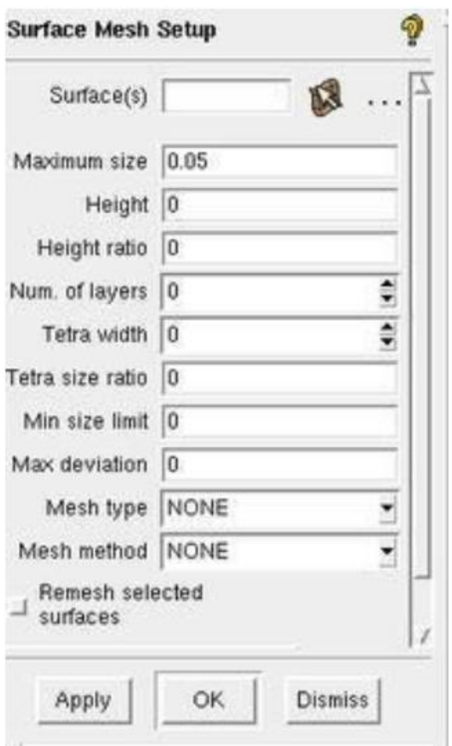
Tip

A tip that is quite useful is the use ofthe F9 key to "pause" the tool beingused so the grid can be moved orzoomed in to.

Also, different parts of the grid can be saved under a *partname* which can be switched off or on if you want certain thingsto be invisible like points or curves or certain surfaces. You canalso copy an entire set of geometry by selecting the parts youwant and translating it to a specified point using the'**translation'** tool. This is useful, especially when creating a symmetrical object such as a wing, wherethe aerofoil can be copied to another location and then joined up to the original aerofoil with curves.Once the geometry is created, the next step is to create the actual grid. Note that the tolerances of thegeometry plays an important role in the accuracy of the grid. So make sure that depending on what youwant, the tolerances are high enough. Using the **blocking** tab, a block can be created around the entiregeometry and then split up into sections. The mesh is created by specifying the distribution of pointsalong the edges of the blocks. Therefore the more blocks you have, the more flexibility you have inchanging the distribution of points along the edges. The edges and vertices of the blocks must beassosciated with the geomery curves and points.Once the blocks have been created and all the required points and curves assosciated, the number ofpoints and the distribution can be set along each edge. In somecases, you want the density of cells to behigh, for example at the boundary layer of an object, whereas to save time, you may want the cellsfurther away to be large. There are various types of distribution such as linear, geometrical andexponential variation that can be used. The ***premesh*** tool can then be used to view the meshing. There isalso a ***quality check*** tool, where one can specify how you want to check the quality of the blocking. Forexample, one can check the variation in volume size to see if it varies smoothly, or if there are anynegative volumes, which would suggest that the grid crosses into solid surfaces.The blocking is saved as a .blk file. When all is done, the mesh can be made readable by a solver byspecifying what type of solver is to be used in the "output tab".

Creating an unstructured grid

Once the curves and surfaces have been created, click the mesh tab ->***surface mesh*** and define the meshdensity on the surfaces.



The surface menu is shown on the right, and to select surfaces, click the button next to it and startselecting surfaces, using middle-click when done. Then select a mesh density (0.05 in this case, but willvary with each case) and check**remesh selected surfaces** if needed, and click **ok**.
Then, click ***volume mesh***, and select the method (tetra for tetragonal unstructured meshes) to generate theunstructured grid, press 'ok' and wait for the grid to be generated and review the result.
ANSYS computational fluid dynamics (CFD) simulation software allows you to predict, with confidence, the impact of fluid flows on products — throughout design and manufacturing as well as during end use. The software's unparalleled fluid flow analysis capabilities can be used to design and optimize new equipment and to troubleshoot already existing installations.Whatever phenomena you are studying — single- or multi-phase, isothermal or reacting,compressible or not — ANSYS fluid dynamics solutions give you valuable insight into

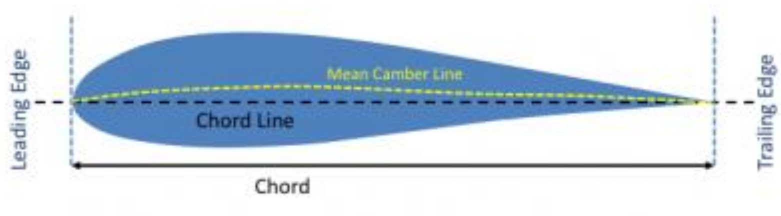
yourproduct's. ANSYS CFD analysis tools include the widely used and well-validated ANSYSFluent and ANSYS CFX, available separately or together in the ANSYS CFD bundle. Becauseof solver robustness and speed, development team knowledge and experience, and advancedmodeling capabilities, ANSYS fluid dynamics solutions provide results you can trust. Thetechnology is highly scalable, providing efficient parallel calculations from a few to thousandsof processing cores. Combining Fluent or CFX with the full-featured ANSYS CFD-Post postprocessingtool allows you to perform advanced quantitative analysis or create high-qualityVisualizations and animations.

As a result of these tight connections, ANSYS CFX delivers benefits that include the ability TO:

- Quickly prepare product/process geometry for flow analysis without tedious rework.
- Avoid duplication through a common data model that is persistently shared across physics —beyond basic fluid flow.
- Easily define a series of parametric variations in geometry, mesh, physics and post-processing,
- enabling automatic new CFD results for that series with a single mouse click
- Improve product/process quality by increasing the understanding of variability and design
- sensitivity.
- Easily set up and perform multiphysics simulations

Experiment 2:To identify and create different aerofoil sections

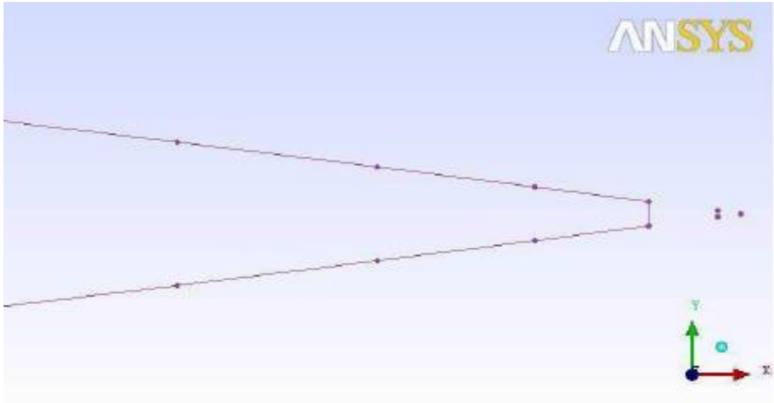
Introduction:
An Aerofoil is a shape capable of producing lift with relatively high efficiency as it passes through the air.



An aerofoil can have many cross sectional shapes. Different aerofoils are used to construct the aircraft wings. The designers choose the shape that has the best aerodynamic characteristics to suit the purpose, weight and speed of the aircraft.

Procedure:

1. Visit the following website
   http://airfoiltools.com/
2. Familiarize yourself with the website and explore its different sections (airfoil search, airfoil plotter, NACA 4 digit airfoil generator etc.)
3. Using the website, download the .dat file of the following 3 aerofoils
   - NACA 2412
   - NACA 4412
   - B737a-il
4. Using the next steps create geometries of the above 3 aerofoils
5. Importing the Aerofoil coordinates
   File→Import Geometry→Formatted point data→Select the file of aerofoilcoordinates which is in DAT format→ok. Now the coordinates will bedisplayed.
6. Geometry→Create/modify curve→From points→Select above points and leave last 2 points→middle click
7. Similarly on bottom side
8. Join the end points of the curves



Comments:

Experiment 3:Studying the nature of airflow over NACA 0012 aerofoil

Theory:

An aerofoil is constructed in such a way that its shape takes advantage of the air's response to certain physical laws. This develops two actions from the air mass: a positive pressure lifting action from the air mass below the wing, and a negative pressure lifting action from lowered pressure above the wing.Different aerofoils have different flight characteristics. The weight, speed, and purpose of each aircraft dictate the shape of its aerofoil.

Procedure

1. NACA 0012 airfoil section has a chord of 1 meter, a span of 1 meter, and a thickness of 0.01 meter. The wing is made of Aluminum 6061-T6.
2. If air moves at 987.84 km/hour around the airfoil, find the velocity vectors of compressible flow over the airfoil.
3. Use the procedure specified in the document titled "Experiment 3: NACA 0012 aerofoil" to study the flow over an aerofoil
4. Use the provided geometry file named "Exp 3 3dAirfoilSurface.igs" for this experiment.

Observations:

Provide the mesh and result plots.

Assignment:

Repeat the above experiment with the mesh refinement as described on pages 20 and 21 of the document titled "Experiment 3: NACA 0012 aerofoil"

Experiment 4:Studying the nature of airflow over cylinder

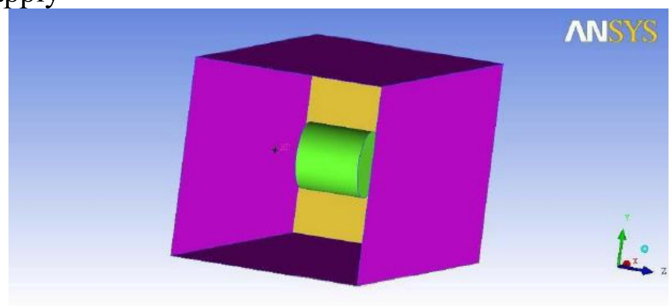
Aim: To study the characteristics of flow over a cylinder.

Description: Consider a cylinder of 3m radius and 6m height. The free stream velocityconsidered is 20m/s. The properties of air is ρ=1.18kg/m3.

Procedure:

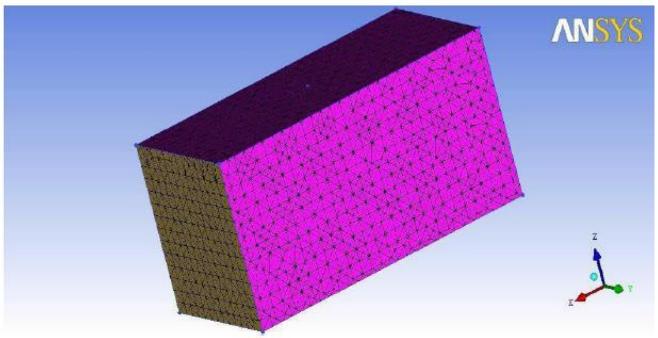
Creation of geometry:

- Geometry → create point → explicit coordinates → (0,0,0)
- Geometry → create surface → standard shapes → box → (36 18 18) → apply →solid simple display
- Geometry → create point → based on 2 locations → select 2 diagonal points of face
- Geometry → transform geometry → copy → select point → Z-offset =6 → apply →z-offset=12→ ok.
- Geometry → surfaces → standard shapes → cylinder r1=3.r2=3 → select 2 points ofcylinder → apply



Creation of parts and mesh generation:

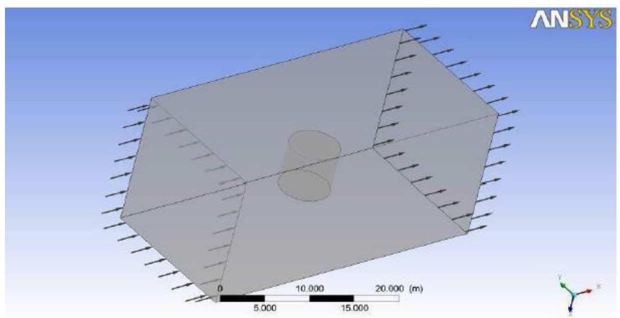
- Parts→ create parts → (part name) → select entities → middle click (createparts according to the problem i.e. inlet, outlet, cylinder & free slip wall)
- Geometry → solid → part(mp) → select two points lying outside the cylinder→ apply.
- Mesh → mesh parameters → cylinder -1.5, inlet-2.5, outlet-2.5, slipfree-0.7
- Mesh → global mesh setup → global mesh size → max element size (3) →apply.
- Mesh → compute mesh → compute.



- Output → outpur solver- ANSYS CFX → common solver → ANSYS →
- APPLY
- WRITE INPUT → OK

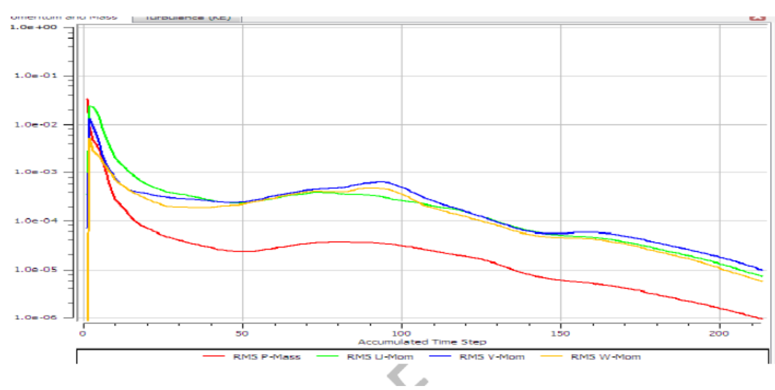
<u>Problem definition in cfx-pre:</u>

- CFX → change the working directory → cfx-pre
- File→ new case → general → apply.
- Mesh → import mesh → ICEM CFD → OK
- Domain → fluid domain → air at 25°C
- Boundary → inlet → domain: inlet → velocity=40m/s.
- Boundary → outlet → domain outlet → static pressure=0 Pa → apply
- Boundary → freeslip → domain free slip → free slip → ok.
- Solver settings → 1000 iterations → apply. Define
- solver → solver input file → ok



<u>Solve:</u>

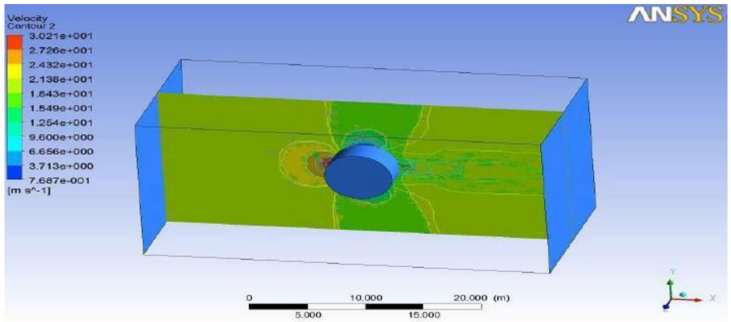
- CFD solver → open cfx file → define run → ok



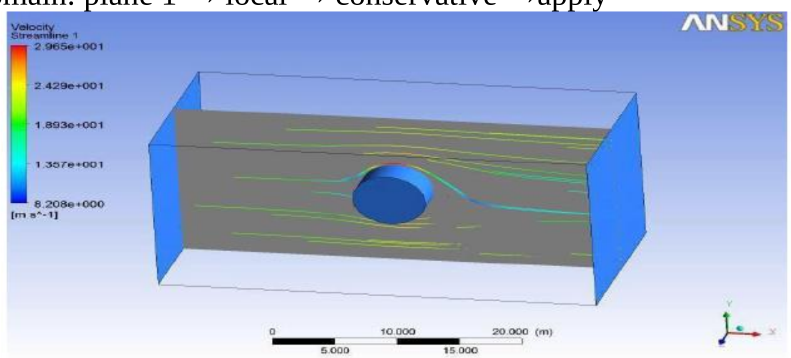
<u>Post processing:</u>

- CFD post → load result → select .res file

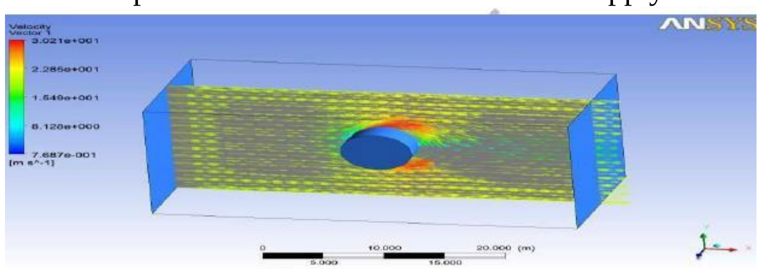
- Location → plane → Z=9 apply
- Contours → domain: plane1 → velocity → local → conservative →apply.

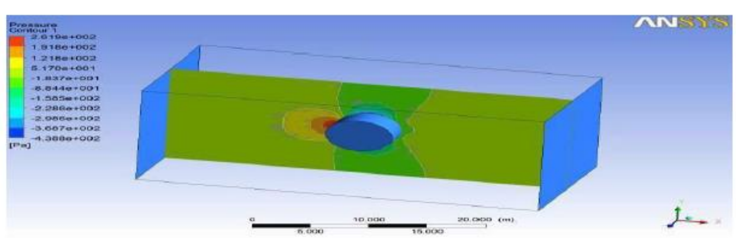


- Contours → domain: plane1 → pressure→ local → conservative → apply.
- Vectors → domain: plane 1 → local → conservative →apply



- Stream lines → domain : plane 1 → local → conservative → apply.





Observation:
Provide the mesh and result plots.

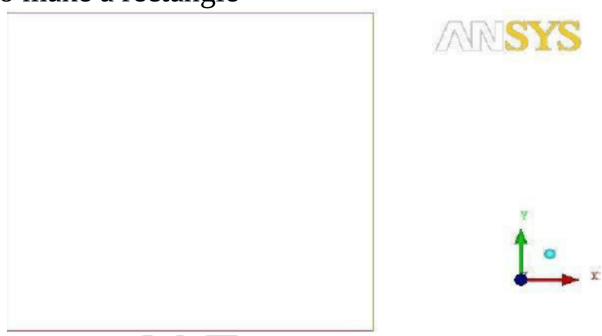
Experiment 5:Studying the nature of airflow over flat plate

Aim: To study the characteristics of flow over a flat plate

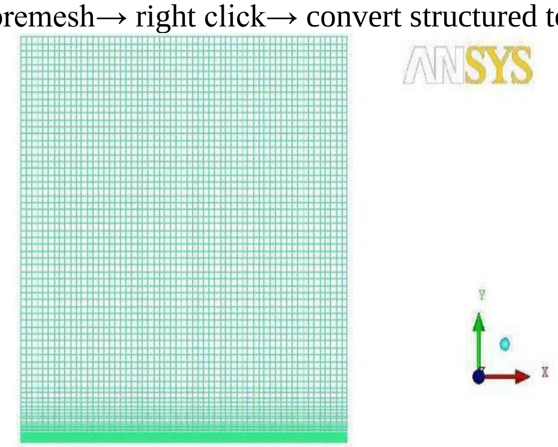
Description: Consider a plate of 1m and the flow of air is 0.00133 m/s. The plate is astationary solid wall having no slip as its boundary condition.

Procedure:

- Geometry→ create point→ explicit coordinates→ 1(0,0,0), 2(1,0,0), 3(1,1,0) and4(0,1,0) → ok
- Create/modify curve→ select 2 points→ middle click
- Select all points to make a rectangle



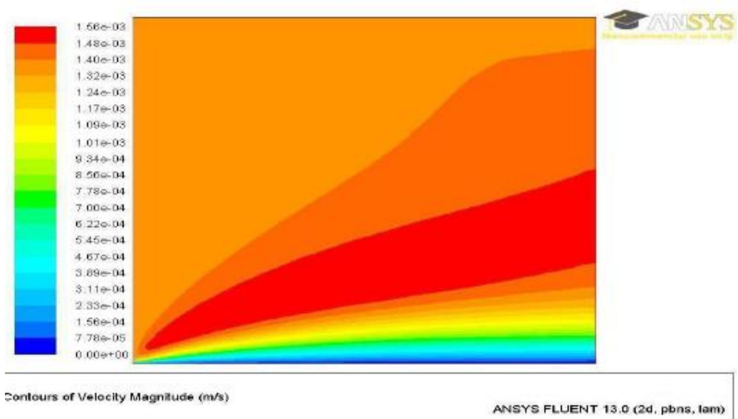
- Create/modify surface→ select the entire lines→ surface is created
- Create part→ name inlet→ select the left edge→ middle click
- Similarly create outlet, top and bottom
- Switch off points and curves→ create part→ name surf→ click on surface→ ok
- Blocking→ create block→ select entities→ click spectacles→ middle click→ switchon points and curves
- Go to association→ associate vertex→ select the point→ double click on the point
- Associate→ edge to curve→ select the edge→ ok→ again select the edge→ ok
- Similarly for the remaining edges
- Premesh parameters→ edge parameters→ select any edge→ click on copyparameters→ nodes-60, spacing-0.01, ratio-1.1→ ok
- Blocking tree→ premesh→ right click→ convert structured to unstructured mesh



- Change the working directory
- output→ output solver→ fluent V6→ common-ansys→ ok

FLUENT:

- Folder→ general→ mesh→ fluent mesh→ ok
- Click on check→ done
- Models→ viscous laminar→ materials→ air
- Cell zone conditions→ solid→ ok
- Boundary conditions→ bottom→ edit→ stationary wall→ ok, inlet→ velocity-0.00133→ ok, outlet→ guage pressure-0→ ok, top→ edit→ moving wall→ ok
- Dynamic mesh→ solution→ solution method-simple, solution controls-0.3,1,0.3→ ok
- Monitor initializer→ compute from inlet→ x=0.00133→ initialize
- Calculation activities→ no of iterations-200→ run calculations→ click oncalculate→ ok
- Results→ graphics and animations→ contour→ set up→ display options→filled→ display
- Contour→ velocity→ display
- Vector→ velocity→ display
- For residue→ contour→ residue→ display



Observations:
Provide the mesh and result plots.

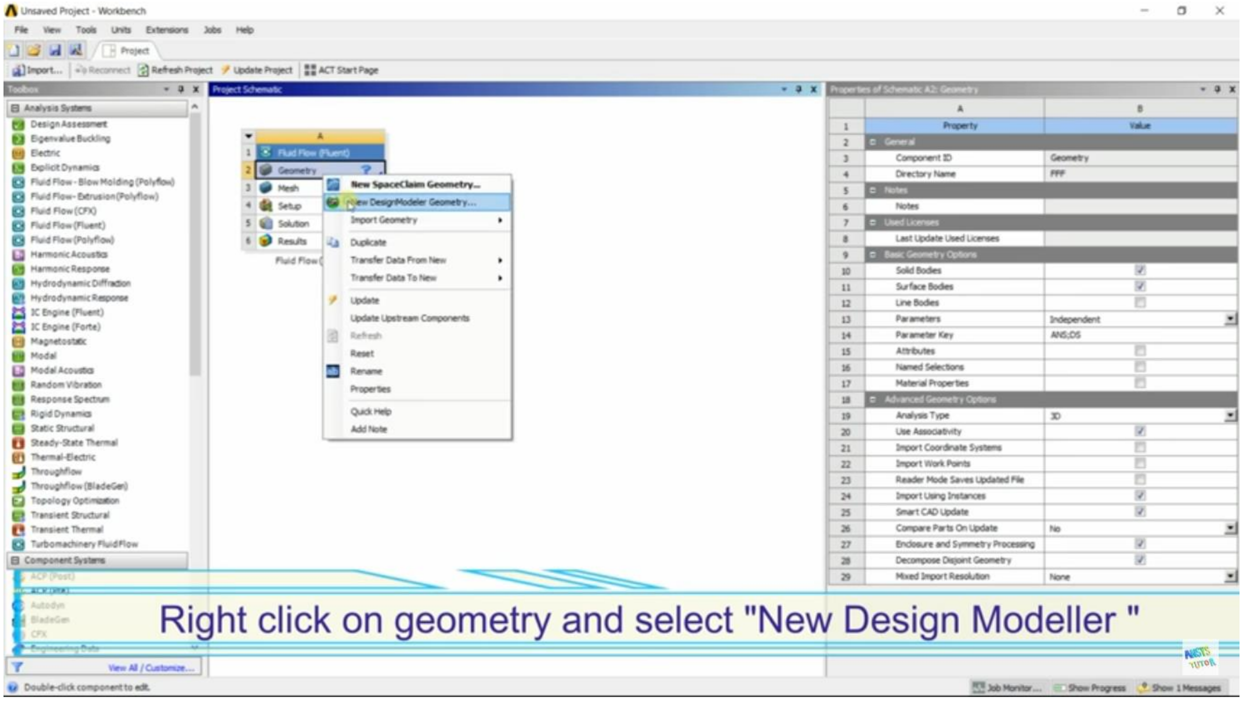
Experiment 6: Study the lift and drag characteristics of rectangular plate

Aim:
In this lab, it has been shown how you can calculate drag and lift forces and coefficients. A rectangular plate has been taken as a specimen and placed perpendicular to flow direction. The air at high velocity is blowing over it. Due to blow of air, the drag and lift forces got developed on this specimen. In the current tutorial, it has been shown how you can calculate the drag and lift forces.

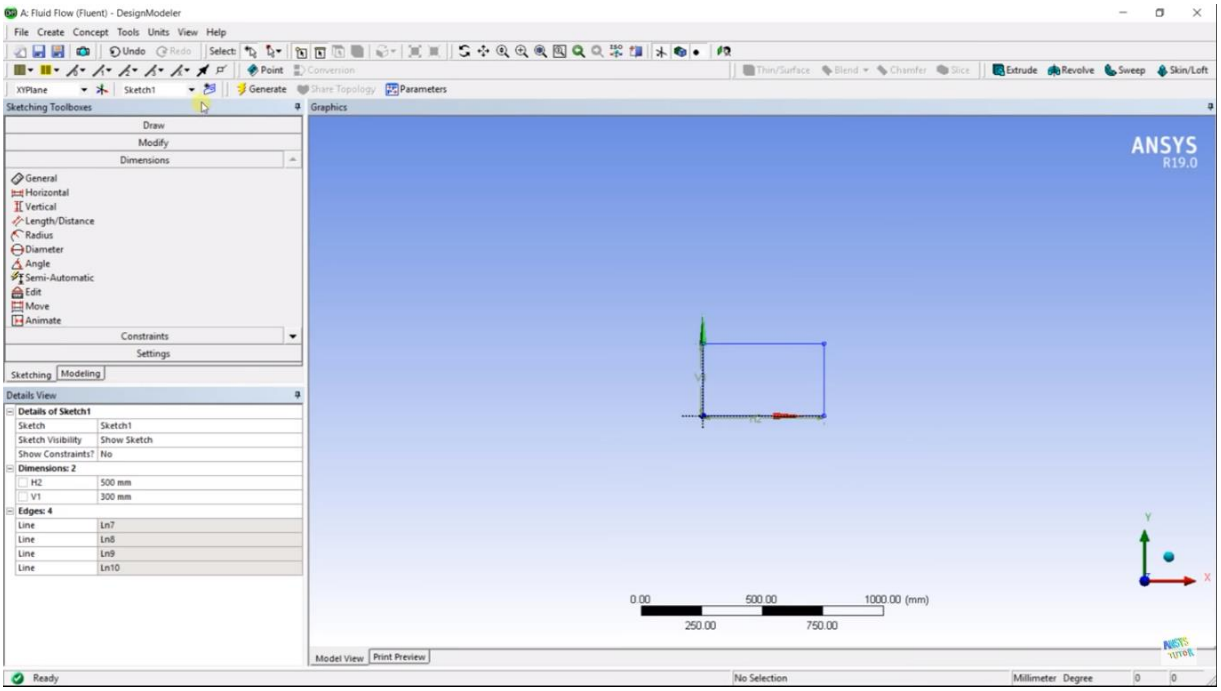
What will you learn from this?
- Creating the flow domain in ANSYS Design modeler
- Structured Mesh Creation
- Solver setup
- Drag and Lift calculations:

Procedure:

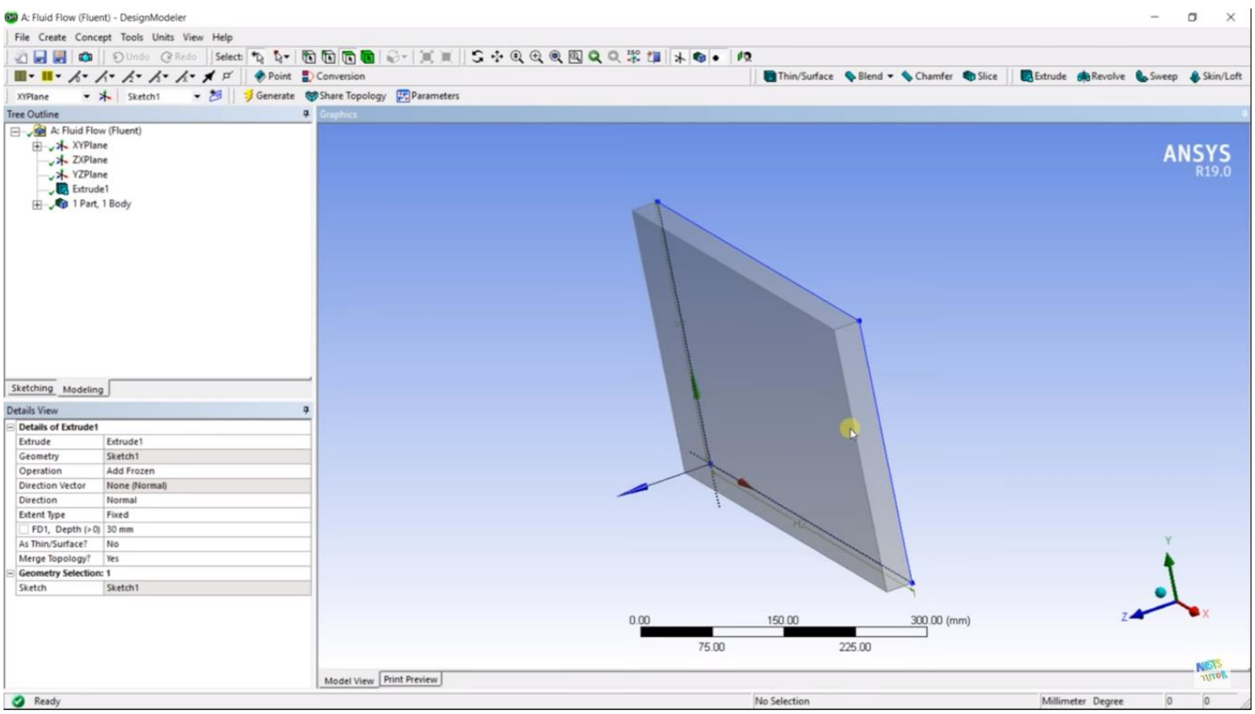
- Drag the fluid flow (fluent) into the project schematic window
- Right click on geometry and select "New Design Modeller"



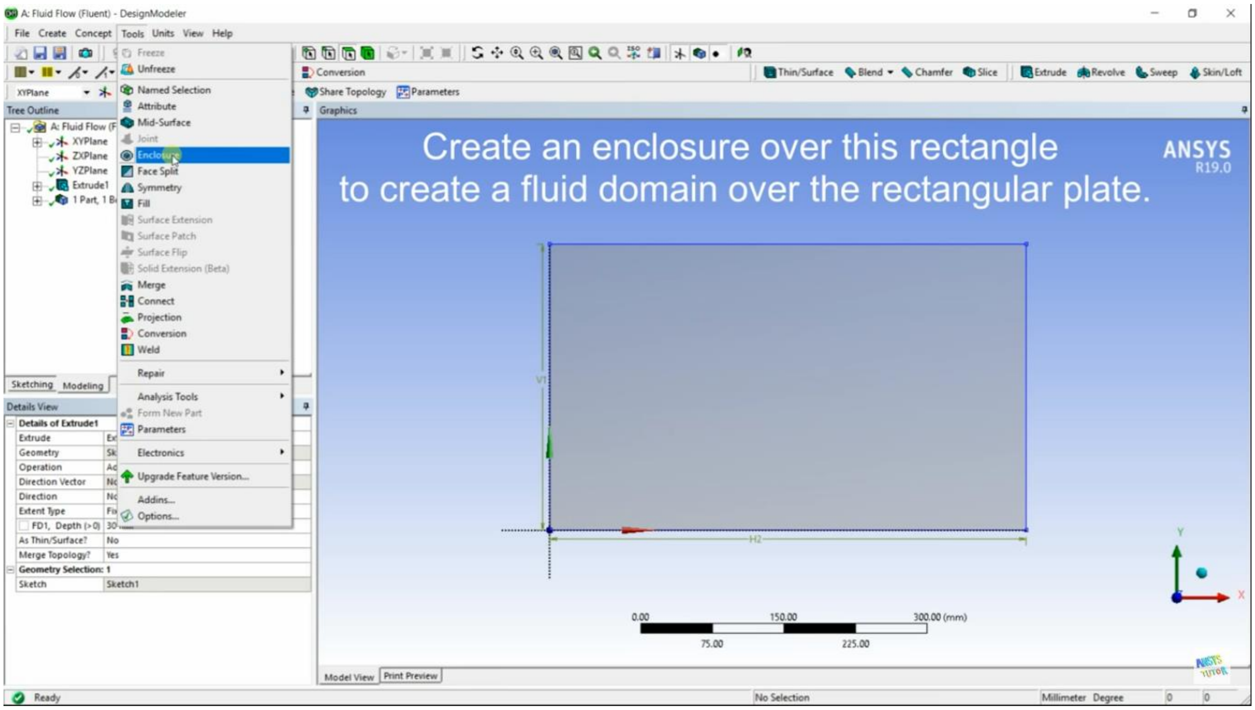
- Change default unit to "mm"
- Select any plane and draw a rectangle of 500 by 300.



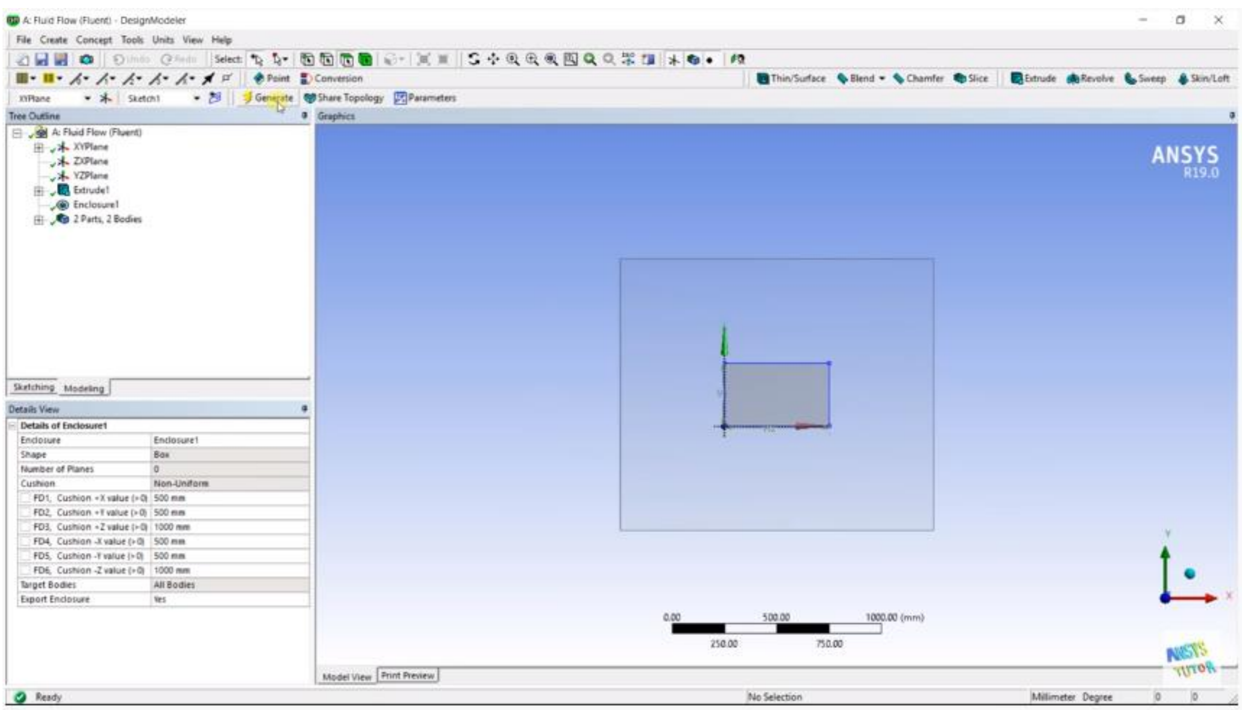
- Extrude the sketch



- Create an enclosure over this rectangle to create a fluid domain over the rectangular plate.



- Edit the “enclosure 1” bar and generate it.



- For details about the above steps and further steps, follow the tutorial in the below mentioned link or file.
    - Link: https://www.youtube.com/watch?v=u0-WgxMAvOsOR
    - File: Aero Exp6 lift-drag.mp4

- Submit your results of plots

Observations:
Lift (N):

Drag (N):

Lift coefficient:

Drag coefficients:

Assignment:

As an extension to the steps followed in this lab, follow the below tutorial (CFD Post processing) and submit your results

https://www.youtube.com/watch?v=IRPMwcMJY10

Experiment 7: Study the lift and drag characteristics of NACA 0012 aerofoil.

Procedure:

Using the procedures specified in Experiment 3 and Experiment 6, calculate the lift and drag characteristics of NACA 0012 aerofoil.

Observations:

Lift (N):

Drag (N):

Lift coefficient:

Drag coefficients:

Assignment:

Using the tutorial below, submit plots for angle of attack of 4 degrees.

https://www.youtube.com/watch?v=gB05xw8Q8YE

Note: The tutorial has 5 parts and you need to go through all parts to complete the assignment

Flow over sphere

https://www.youtube.com/watch?v=5wWPy5ErwuI