R-22

MALLA REDDY COLLEGE OF ENGINEERING & TECHNOLOGY

MRCETCAMPUS

(AUTONOMOUSINSTITUTION–UGC, GOVT. OF INDIA)

Department of AERONAUTICAL ENGINEERING

COMPUTATIONAL AERODYNAMICS LAB

Prepared by:

E Dinesh Guptha/M Pramod

Assistant Professor
Department of ANE

COMPUTATIONAL AERODYNAMICS

LABORATORY MANUAL

B.TECH (R-22 Regulation)

(III YEAR – I SEM)
(2025-26)

DEPARTMENT AERONAUTCAL ENGINEERING

MALLA REDDY COLLEGE OF ENGINEERING & TECHNOLOGY

(Autonomous Institution–UGC, Govt. of India)
Recognized under2 (f) and 12(B) of UGC ACT 1956
(Affiliated to JNTU H, Hyderabad, Approved by AICTE -Accredited by NBA & NAAC–'A' Grade- ISO9001:2015Certified) Maisammaguda, Dhulapally (Post Via. Hakimpet), Secunderabad – 500100, Telangana State, India

DEPARTMENT OF AERONAUTICAL ENGINEERING

VISION

Department of Aeronautical Engineering aims to be indispensable source in Aeronautical Engineering which has a zeal to provide the value driven platform for the students to acquire knowledge and empower themselves to shoulder higher responsibility in building a strong nation.

MISSION

(a) The primary mission of the department is to promote engineering education and research.

(b) To strive consistently to provide quality education, keeping in pace with time and technology.

(c) Department passions to integrate the intellectual, spiritual, ethical and social development of the students for shaping them into dynamic engineers.

QUALITY POLICY

Impart up-to date knowledge to the students in Aeronautical area to make them quality engineers. Make the students experience the applications on quality equipment and tools. Provide systems, resources, and training opportunities to achieve continuous improvement. Maintain global standards in education, training, and services.

PROGRAMME EDUCATIONAL OBJECTIVES (PEO'S)

PEO1: PROFESSIONALISM & CITIZENSHIP To create and sustain a community of learning in which students acquire knowledge and learn to apply it professionally with due consideration for ethical, ecological and economic issues.
PEO2: TECHNICAL ACCOMPLISHMENTS To provide knowledge based services to satisfy the needs of society and the industry by providing hands on experience in various technologies in core field.
PEO3: INVENTION, INNOVATION AND CREATIVITY To make the students to design, experiment, analyze, interpret in the core field with the help of other multi disciplinary concepts wherever applicable.
PEO4: PROFESSIONAL DEVELOPMENT To educate the students to disseminate research findings with good soft skills and become a successful entrepreneur.
PEO5: HUMAN RESOURCE DEVELOPMENT To graduate the students in building national capabilities in technology, education and research.

PROGRAM SPECIFIC OBJECTIVES (PSO's)

1. To mould students to become a professional with all necessary skills, personality and sound knowledge in basic and advance technological areas.
2. To promote understanding of concepts and develop ability in design manufacture and maintenance of aircraft, aerospace vehicles and associated equipment and develop application capability of the concepts sciences to engineering design and processes.
3. Understanding the current scenario in the field of aeronautics and acquire ability to apply knowledge of engineering, science and mathematics to design and conduct experiments in the field of Aeronautical Engineering.
4. To develop leadership skills in our students necessary to shape the social, intellectual, business and technical worlds.

PROGRAM OBJECTIVES (PO'S)

Engineering Graduates will be able to:

1. **Engineering knowledge**: Apply the knowledge of mathematics, science, engineering fundamentals, and an engineering specialization to the solution of complex engineering problems.
2. **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.
3. **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.
4. **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.
5. **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.
6. **The engineer and society**: Apply reasoning informed by the contextual knowledge to assess societal, health, safety, legal and cultural issues and the consequent responsibilities relevant to the professional engineering practice.
7. **Environment and sustainability**: Understand the impact of the professional engineering solutions in societal and environmental contexts, and demonstrate the knowledge of, and need for sustainable development.
8. **Ethics**: Apply ethical principles and commit to professional ethics and responsibilities and norms of the engineering practice.
9. **Individual and team work**: Function effectively as an individual, and as a member or leader in diverse teams, and in multidisciplinary settings.
10. **Communication**: Communicate effectively on complex engineering activities with the engineering community and with society at large, such as, being able to comprehend and write effective reports and design documentation, make effective presentations, and give and receive clear instructions.
11. **Project management and finance**: Demonstrate knowledge and understanding of the engineering and management principles and apply these to one's own work, as a member and leader in a team, to manage projects and in multi disciplinary environments.
12. **Life- long learning**: Recognize the need for, and have the preparation and ability to engage in independent and life-long learning in the broadest context of technological change.

MALLA REDDY COLLEGE OF ENGINEERING & TECHNOLOGY

IV Year B. Tech, ANE-I Sem

L	T/P/D	C
0	**-/3/-**	**1.5**

(R18A2188)COMPUTATIONAL AERODYNAMICS LAB

Objectives:

- To develop an understanding for the major theories, approaches and methodologies used in CFD.
- To build up the skills in the actual implementation of CFD methods (e.g. boundary conditions, turbulence modeling etc.) in using commercial CFD codes.
- To gain experience in the application of CFD analysis to real engineering designs.

LIST OF EXPERIMENTS:

1. Numerical simulation of Flow over an airfoil using commercial software
2. Numerical simulation of Supersonic flow over a wedge using commercial Software
3. Numerical simulation of Flat plate boundary layer using commercial software
4. Numerical simulation of Laminar flow through pipe using commercial software
5. Numerical simulation of Flow past cylinder using commercial software
6. Numerical simulation of flow through nozzle using commercial software
7. Numerical simulation of flow over wing using commercial software
8. Numerical simulation of combustion using commercial software
9. Solution for the one dimensional wave equations using explicit method of lax using finite
    a. Difference method (code development)
10. Solution for the one dimensional heat conduction equation using explicit method using finite difference method (code development)
11. Generation of the Algebraic Grid (code development)
12. Generation of the Elliptic Grids (code development)

Note: Any 10 Experiments can be conducted.

Equipment Needed:

Computers: Core 2 duo processor with 1 GB RAM
Softwares: Matlab or scilab and Ansys or equivalent softwares

Reference Books:

- ANSYS FLUENT and CFX Tutorials
- MATLAB an Introduction with Applications Fifth Edition AMOS GILAT by WILEY Publications
- Programming in SCI lab by VINU V DAS New Age International Publications

Outcomes:

- Students will develop a better intuition of Aerodynamics more quickly than is possible with traditional analytical approaches.
- Ability to undertake problem identification, formulation and solution and apply knowledge of basic science and engineering fundamentals.
- Developing a geometrical model of the flow, applying appropriate boundary conditions, specifying solution parameters, and visualizing and analyzing the results.

CONTENTS

CODE OF CONDUCT FOR THE LABORATORIES

- All students must observe the Dress Code while in the laboratory.
- Sandals or open-toed shoes are NOT allowed.
- Foods, drinks and smoking are NOT allowed.
- All bags must be left at the indicated place.
- The lab timetable must be strictly followed.
- Be PUNCTUAL for your laboratory session.
- Program must be executed within the given time.
- Noise must be kept to a minimum.
- Workspace must be kept clean and tidy at all time.
- Handle the systems and interfacing kits with care.
- All students are liable for any damage to the accessories due to their own negligence.
- All interfacing kits connecting cables must be RETURNED if you taken from the lab supervisor.
- Students are strictly PROHIBITED from taking out any items from the laboratory.
- Students are NOT allowed to work alone in the laboratory without the Lab Supervisor
- USB Ports have been disabled if you want to use USB drive consult lab supervisor.
- Report immediately to the Lab Supervisor if any malfunction of the accessories, is there.

Before leaving the lab

- Place the chairs properly.
- Turn off the system properly
- Turn off the monitor.
- Please check the laboratory notice board regularly for updates.

1. FLOW OVER AN AEROFOIL **AIM: To simulate flow over NACA 0012 airfoil** Problem description:

Consider air flowing over NACA 0012 airfoil. The free stream velocity is 40 m/s . Assume standard sea-level values for the free stream properties:

Pressure = 101,325 Pa

Density = 1.2250 kg/m^3

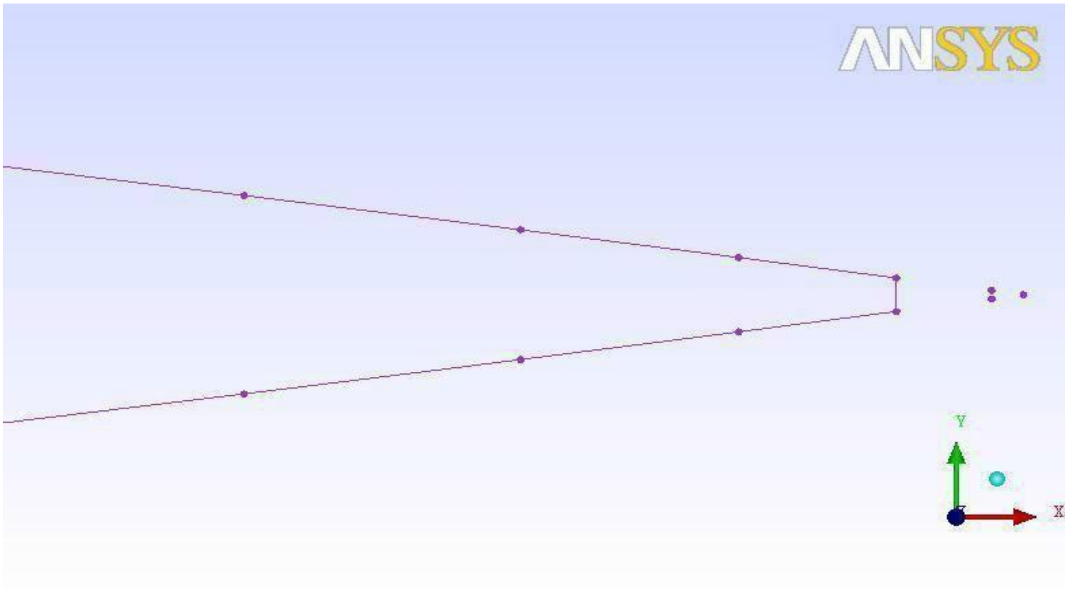
Temperature = 288.16 K

Kinematic viscosity v = 1.4607e-5 m^2/s

Steps Involved In ICEM

CFD: Creation of Geometry in ICEM CFD:

- Importing the Aerofoil coordinates File→Import Geometry→Formatted point data→Select the file of aerofoil coordinates which is in DAT format→ok. Now the coordinates will be displayed.
- Geometry→Create/modify curve→From points→Select above points and leave last 2 points→middle click
- Similarly on bottom side
- Join the end points of the curves



1) Creation of parts:

- Parts in the tree→Right click→Create part→ Select Upper curve: Suction
  Select Lower curve: Pressure 3rd Line: TE

2) Creation of Domain:

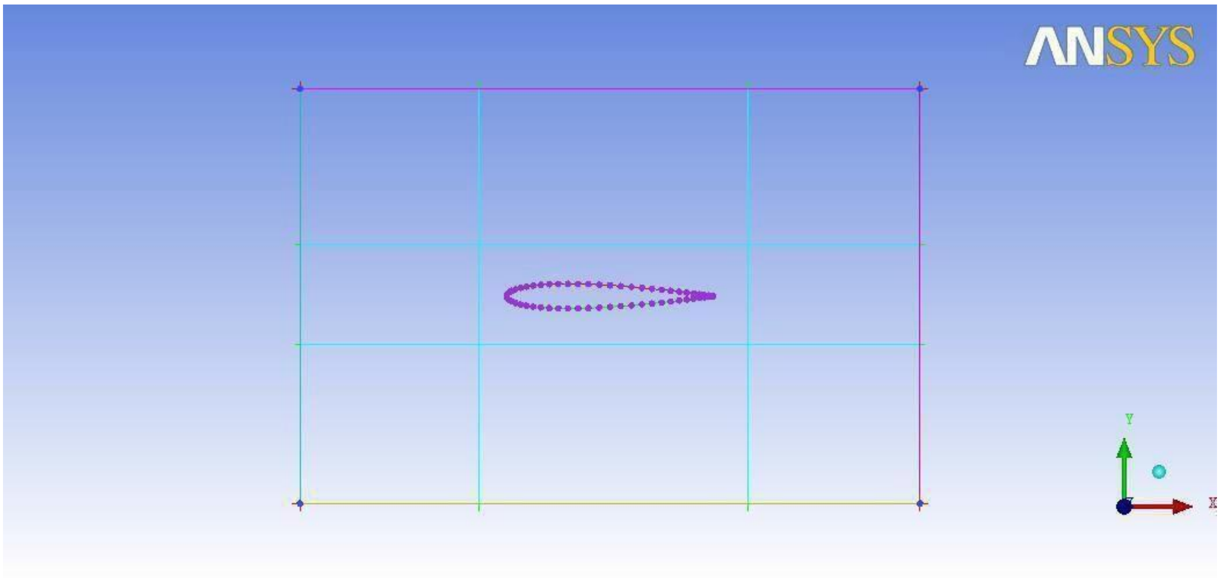
- Create points (-1,1),(-1,-1),(2,1),(2,-1)
- Join these points
- Create parts as Inlet, Outlet, Top & Bottom
- Geometry→Create/Modify surface→Simple surface→Select all the lines of domain→ok
- Create the new part as: Surface

3) Saving the Geometry:

- File→Change working directory→Choose the folder
- File→Geometry→Save Geometry as→Give the name.

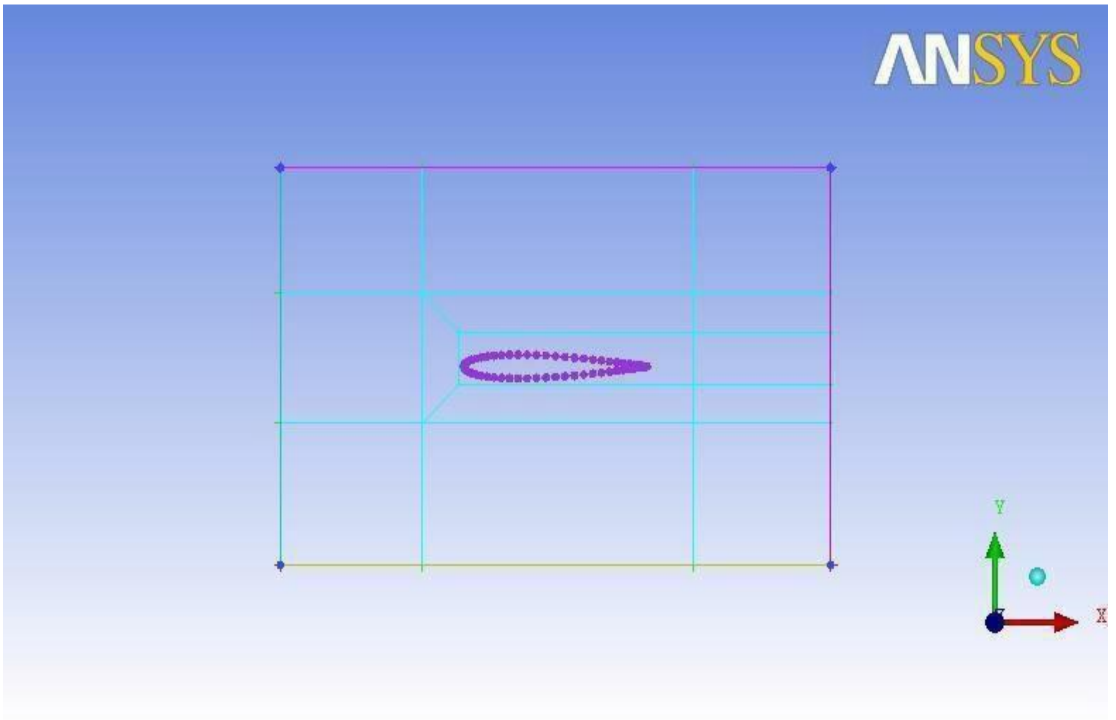
4) Creation of Blocking and Association:

- Blocking→Create block→Initialize blocks→Type as:2D Planar→ok
- Associate→Associate vertex to point→Select a vertex and a point→Apply→ Similarly associate remaining 3 vertices to points
- Associate→Associate edge to curve→Select a edge and a curve →Apply→ Similarly associate remaining 3 edges to curves
- Split block→Select the edges→Create the blocks as shown in figure

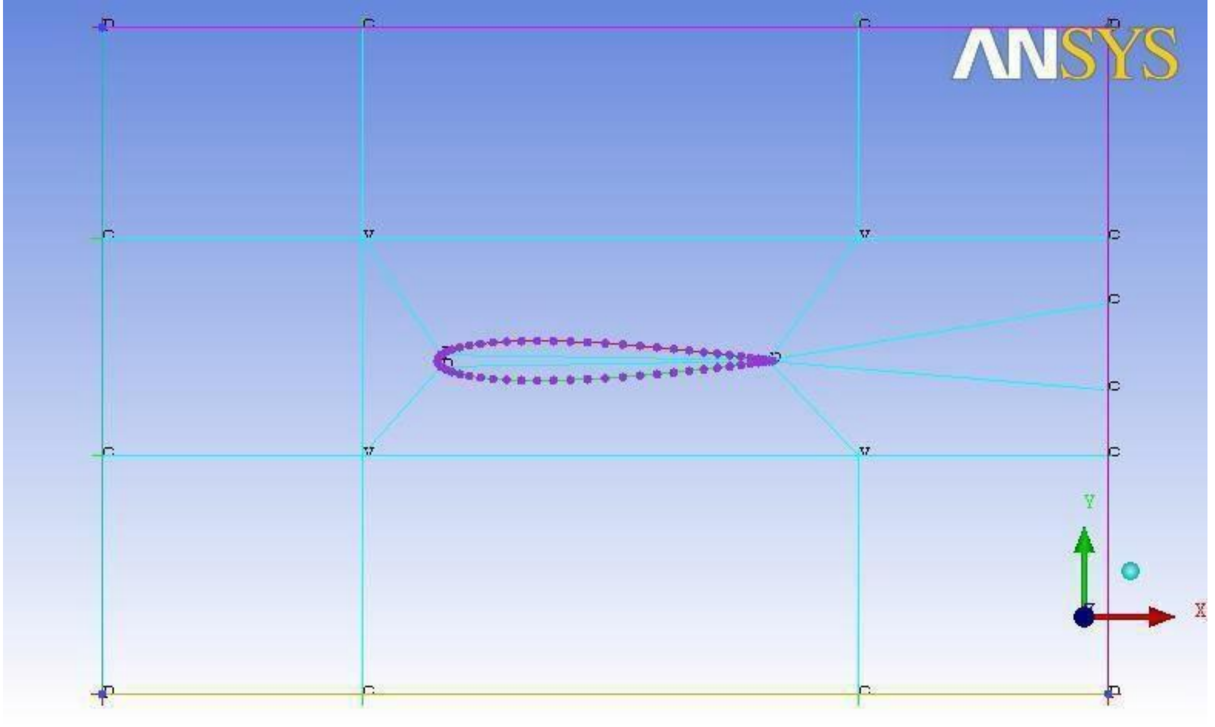


- Split block→O grid →Select edges→Select last 2 edges in the middle row→ok→Select blocks→Select last 2 blocks in the middle row→ok

Thus the O grid has been generated as shown in below fig.



- Associate →Associate vertex to point→Select the vertex of the O grid and the 2nd point on the upper curve(suction) →ok
- Similarly associate remaining 3 vertices of the O grid to the points on the aerofoil as shown in the below fig.

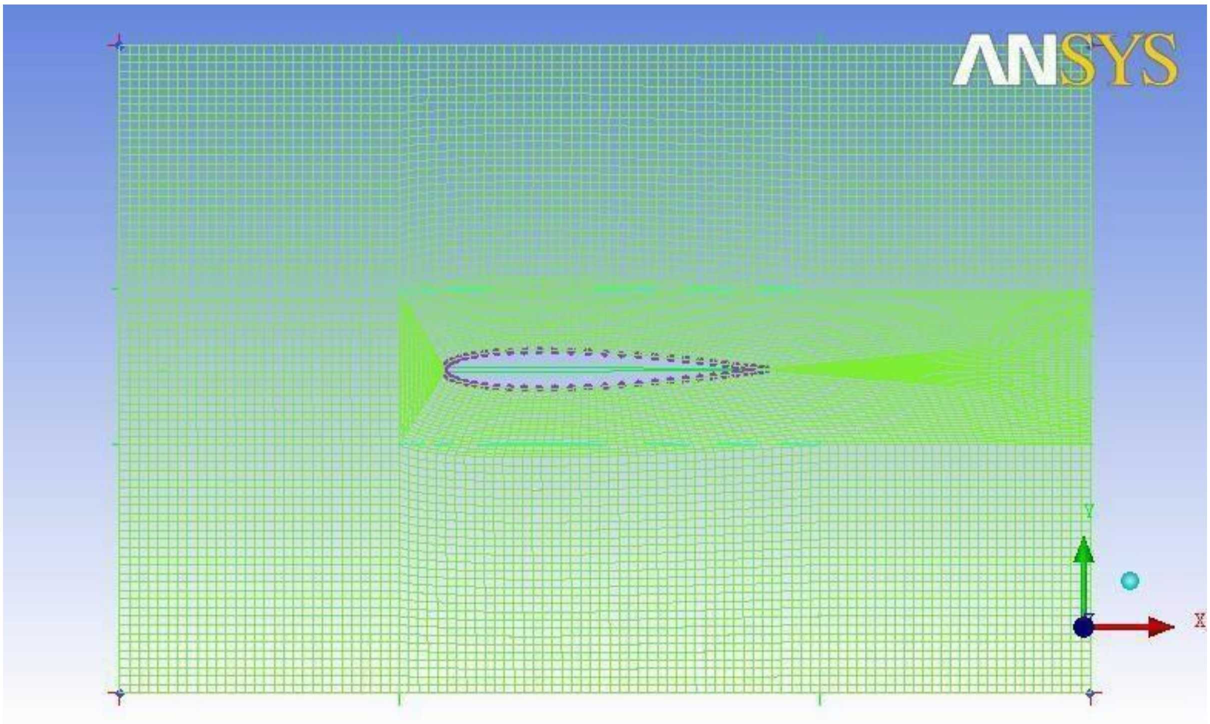


- Associate→Associate edge to curve→Select the 3 edges of block which is inside of the aerofoil and select the suction & pressure curves→ok Similarly associate the TE edge to TE curve.
- Delete block→Select the block inside the aerofoil→ok

5) Generation of Mesh:

- Pre-mesh parameters→Edge parameters→Switch ON the Copy Parameters→ Select the edges and give desired no. of nodes→ok
- Switch ON Pre-mesh in the tree→click yes to compute the meshing
- Pre-mesh→Right click→Convert to unstructured mesh

Now the required mesh has been generated as shown in below fig.



6) Saving the Project:

- File→Save Project as→Give the name.

7) Writing output file:

- Output→Select solver→Output solver as: Fluent_V6→Common Structural solver as: ANSYS→ok
- Write input→Click NO→Open the file→Click 2D→ok

Steps Involved in Fluent:

8) Importing the mesh file:

- File→Read→mesh→Choose the output file written in ICEM CFD Now the mesh has imported into the fluent solver.

9) Problem setup:

- General→Type as: Pressure based
- Models→Energy ON→ inviscid
- Materials→Air
- Cell zone conditions→Type as: fluid→ok
- Boundary conditions→Select inlet→Edit→Give velocity magnitude as: 40 m/s.
- Boundary conditions→Select outlet→Edit→Give gauge pressure as: 0 Pa
- Monitors →Drag and Lift → select suction, pressure, TE parts→plot.

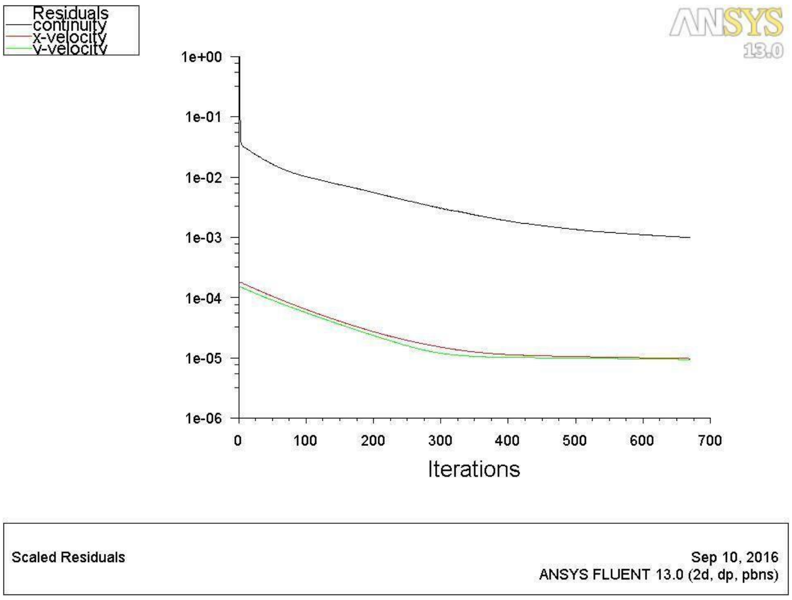
10) Solution:

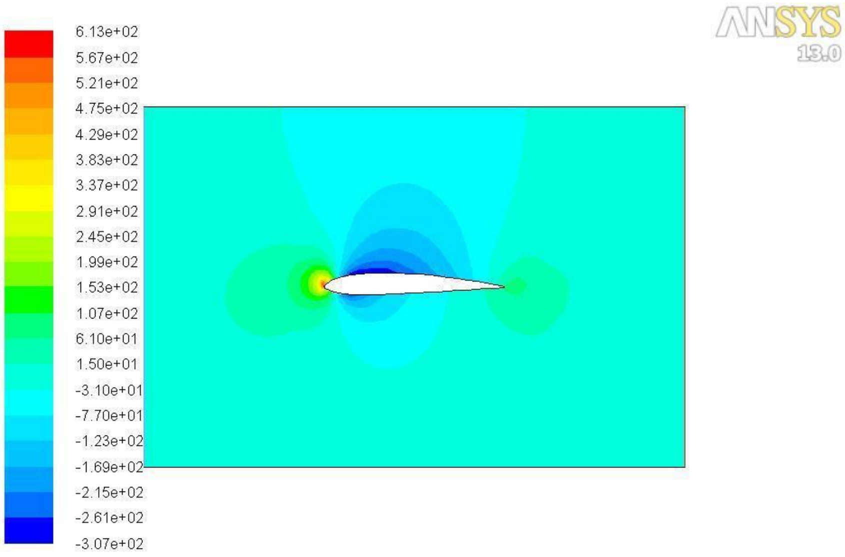
- Select the required monitors
- Solution initialization→Compute from: inlet→Initialize
- Run calculations→Enter the no. of iterations as: 1000→Calculate

11) Results:

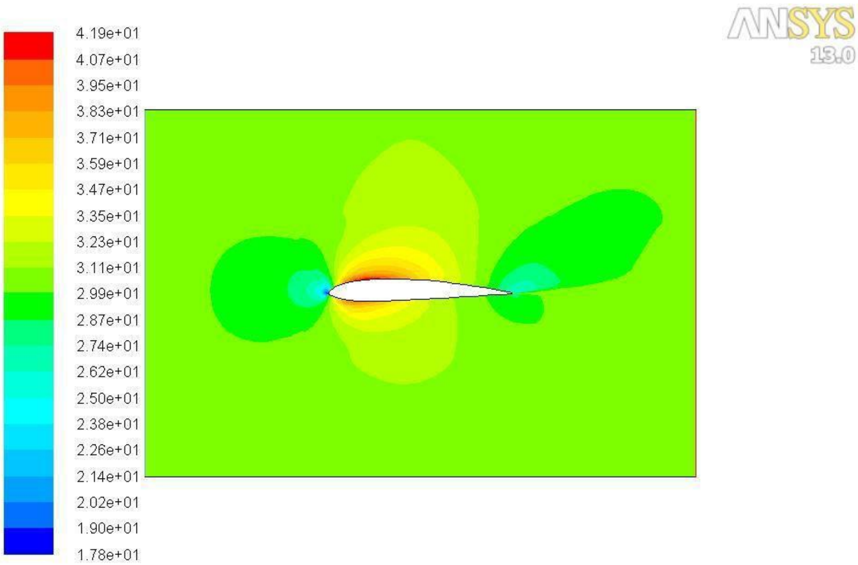
- Graphics and animations→select the required flow parameters in the contours and vectors.

The results are shown below as:

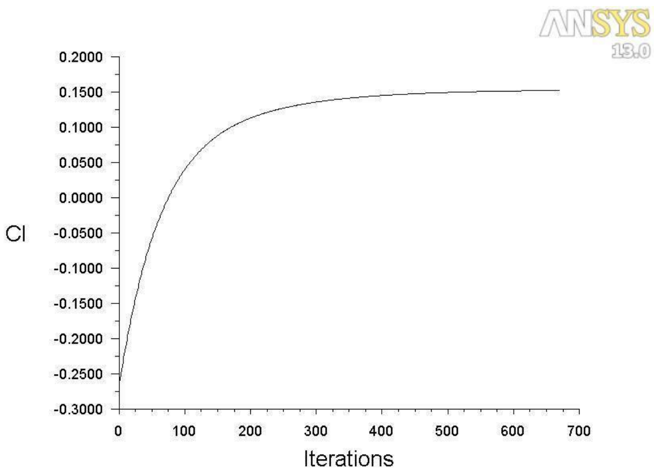




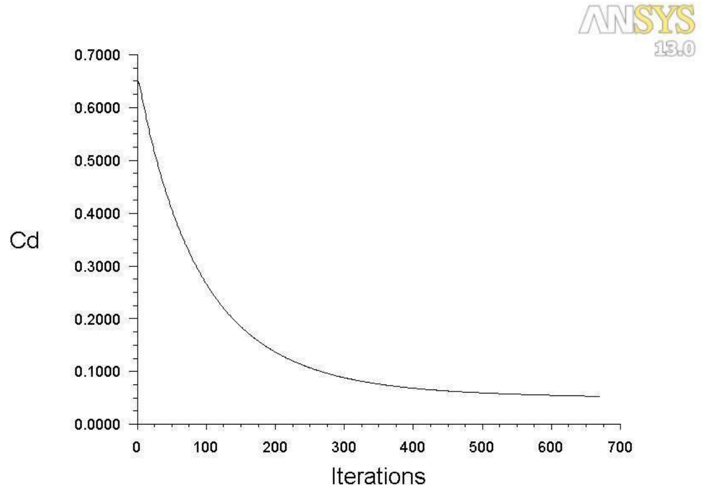
Contours of Static Pressure (pascal) Sep 10, 2016
ANSYS FLUENT 13.0 (2d, dp, pbns)



Contours of Velocity Magnitude (m/s) Sep 10, 2016
ANSYS FLUENT 13.0 (2d, dp, pbns)



Lift Convergence History Sep 10, 2016
ANSYS FLUENT 13.0 (2d, dp, pbns)



Drag Convergence History Sep 10, 2016
ANSYS FLUENT 13.0 (2d, dp, pbns)

```
  660 1.0102e-03 9.8172e-06 9.4580e-06  0:01:39  440
 iter continuity x-velocity y-velocity     time/iter
  661 1.0089e-03 9.8135e-06 9.4543e-06  0:01:19  439
  662 1.0076e-03 9.8097e-06 9.4505e-06  0:01:03  438
  663 1.0063e-03 9.8059e-06 9.4467e-06  0:00:50  437
  664 1.0049e-03 9.8022e-06 9.4429e-06  0:00:40  436
  665 1.0040e-03 9.7984e-06 9.4390e-06  0:00:32  435
  666 1.0027e-03 9.7946e-06 9.4352e-06  0:00:25  434
  667 1.0014e-03 9.7907e-06 9.4314e-06  0:00:20  433
  668 1.0001e-03 9.7868e-06 9.4276e-06  0:00:16  432
!  669 solution is converged
  669 9.9879e-04 9.7829e-06 9.4238e-06  0:00:13  431
Writing "C:\Documents and Settings\student\Desktop\airfoil\AEROFOIL.cas"...
   4693 quadrilateral cells, zone 18, binary.
   9215 2D interior faces, zone 19, binary.
     19 2D wall faces, zone 20, binary.
```

Editor - Untitled

Exercise Problems:

1.1 Evaluate aerodynamic characteristics of aerofoil at 15 degrees angle of attack.
1.2 Perform the grid independence study over airfoil.

2. SUPERSONIC FLOW OVER A WEDGE

Problem description:

Consider air flowing over wedge. The free stream Mach number is 3 and the angle of attack is 5°. Assume standard sea-level values for the free stream properties:

Pressure =101,325Pa

Density =1.2250kg/m^3

Temperature =288.16K

Kinematic viscosity v = 1.4607e-5 m^2/s

Steps Involved In ICEM CFD:

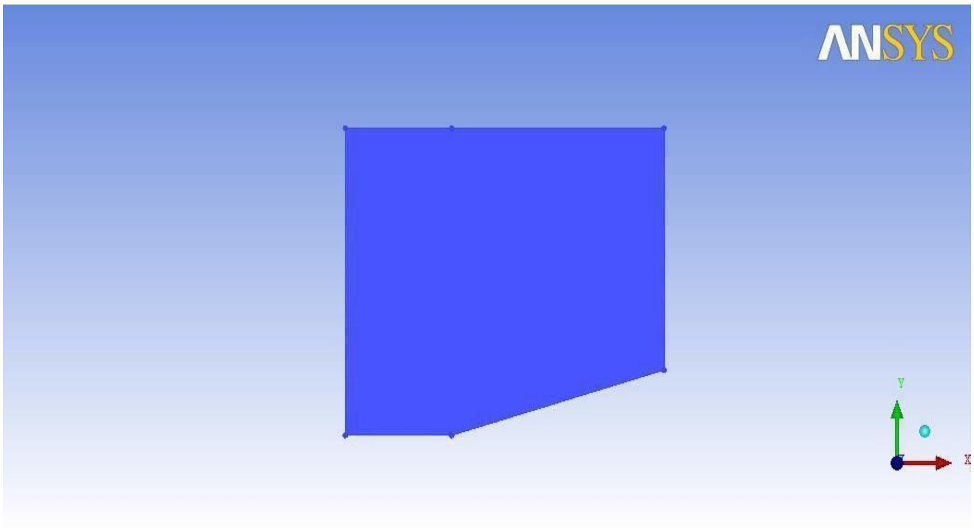
1) Creation of Geometry in ICEM CFD:

a. Geometry→Create point→Explicit coordinates→Enter the coordinates as given in table shown:

X	0	0	0.5	1.5	1.5	0.5
Y	0	1.259	1.259	1.259	0268	0
Z	0	0	0	0	0	0

Geometry→Create/modify curve→From points→Select any 2 points→ok→Similarly create the curves to all points

b. Geometry→Create/Modify surface→Simple surface→Select all the lines of domain→ok



2) Creation of parts:

a. Parts in the tree→Right click→Create part→

Select Left curve: **Inlet**

Select Right curve: **Outlet**

Select Top curve: **Top**

Select inclined curve: **Wedge**

Select bottom curve: **Front_wedge**

3) Saving the Geometry:

a. File→Change working directory→Choose the folder

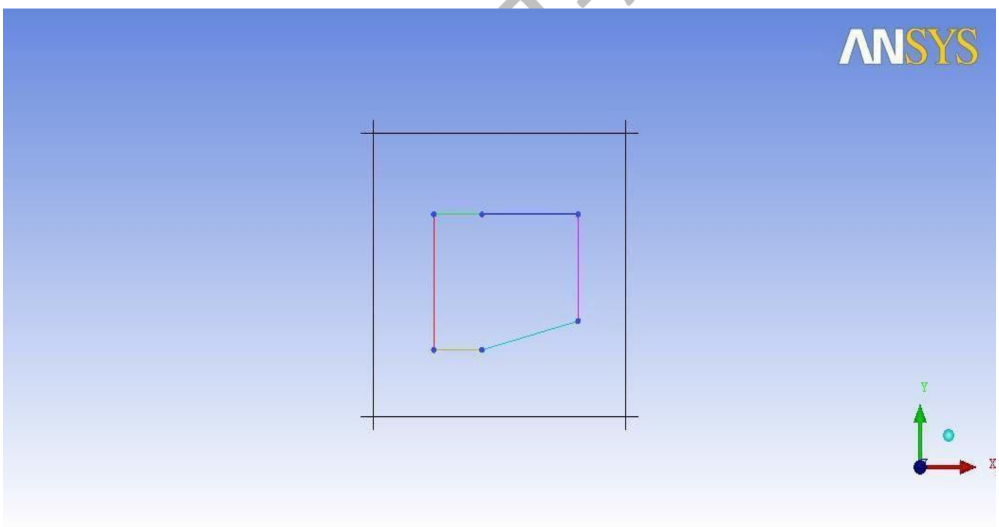
b. File→Geometry→Save Geometry as→Give the name.

4) Creation of Blocking and Association:

a. Blocking→Create block→Initialize blocks→Type as:2D Planar→ok

b. Associate→Associate vertex to point→Select a vertex and a point→Apply→ Similarly associate remaining 3 vertices to points

c. Associate→Associate edge to curve→Select a edge and a curve →Apply→ Similarly associate remaining 3 edges to 5 curves

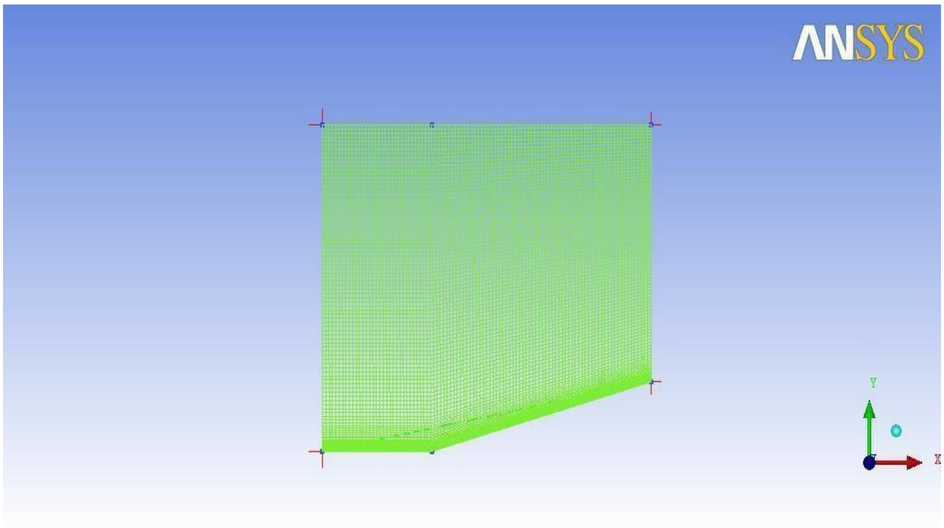


5) Generation of Mesh:

a. Pre-mesh parameters→Edge parameters→Switch ON the Copy Parameters→ Select the Horizontal edge and give no. of nodes as: 100→ok

b. Pre-mesh parameters→Edge parameters→Switch ON the Copy Parameters→ Select the Vertical edge and give no. of nodes as: 100→Spacing as: 0.001→Ratio as: 1.1→ok

c. Switch ON Pre-mesh in the tree→click yes to compute the meshing
d. Pre-mesh→Right click→Convert to unstructured mesh
Now the required mesh has been generated as shown in below fig:



6) Saving the Project:

a. File→Save Project as→Give the name.

7) Writing output file:

a. Output→Select solver→Output solver as: **Fluent_V6**→Common Structural solver as: ANSYS→ok
b. Write input→Click NO→Open the file→**Click 2D**→ok

Steps Involved in Fluent:

8) Importing the mesh file:

a. File→Read→mesh→Choose the output file written in ICEM CFD
Now the mesh has imported into the fluent solver.

9) Problem setup:

a. General→Type as: Density based
b. Models→Energy ON→Select Viscous-laminar→Edit→Set model as: k-omega(2 equ)
c. Materials→Air→Create/Edit→Set density as: Ideal-gas→Set viscosity as: Sutherland→Change
d. Cell zone conditions→Type as: fluid→Set operating conditions→Set operating pressure as: 0Pa

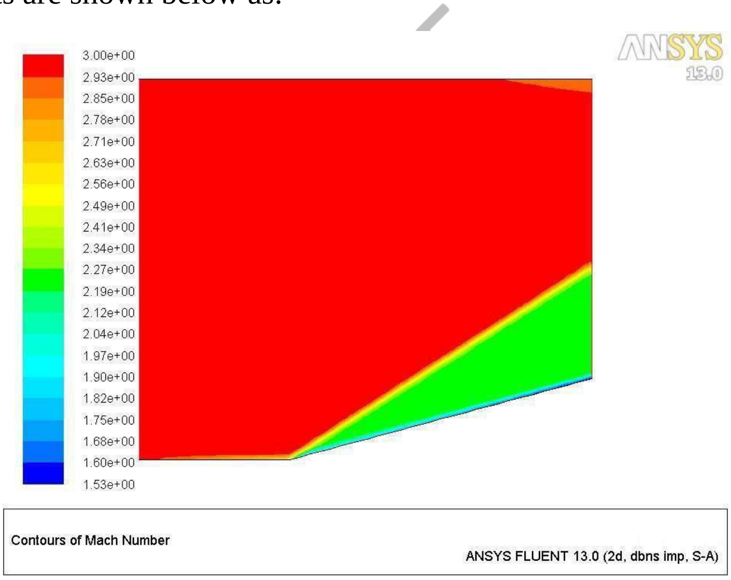
e. Boundary conditions→Select inlet→Give type as: pressure-far-field→Edit→Give Gauge pressure as: 101325Pa→Set Mach as: 3→ok
f. Boundary conditions→Select outlet→Edit→Give gauge pressure as: 0Pa

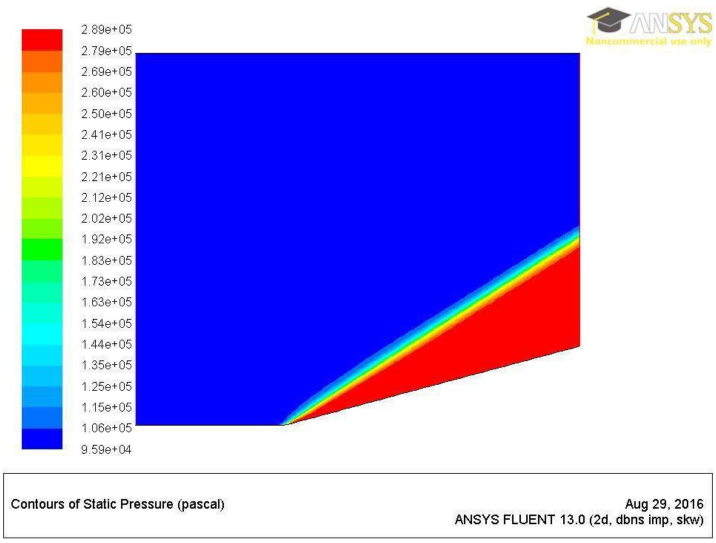
10) Solution:

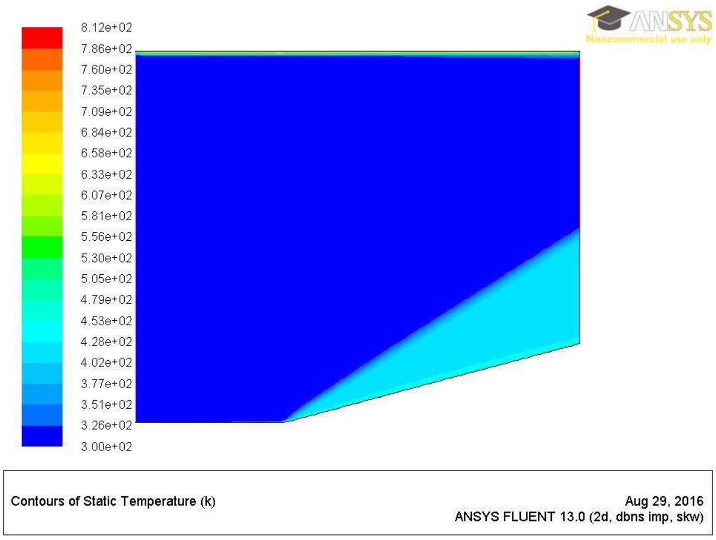
a. Select Solution Controls→Set Courant number as: 1
b. Select the required monitors
c. Solution initialization→Compute from: inlet→Initialize
d. Run calculations→Enter the no. of iterations as: 1000→Calculate

11) Results:

a. Graphics and animations→Select the required flow parameters in the contours. The results are shown below as:







Exercise Problems:

2.1 Evaluate flow properties across oblique shock of wedge at 15 degrees angle of attack.
2.2 Compare the results with analytical formulas.

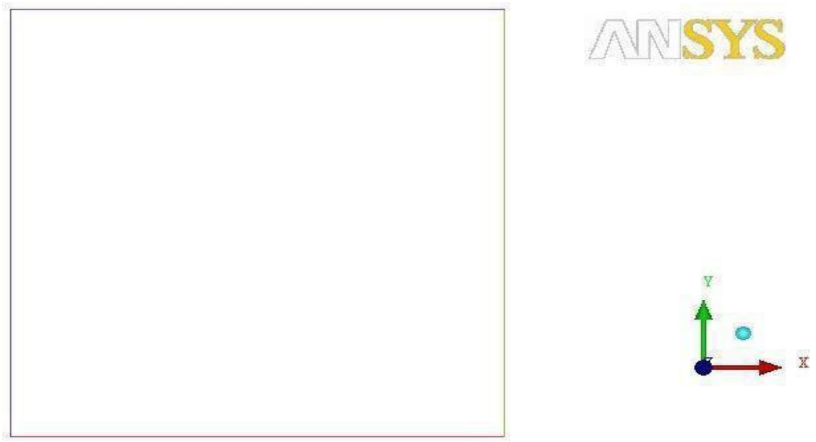
3. FLOW OVER A FLAT PLATE

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

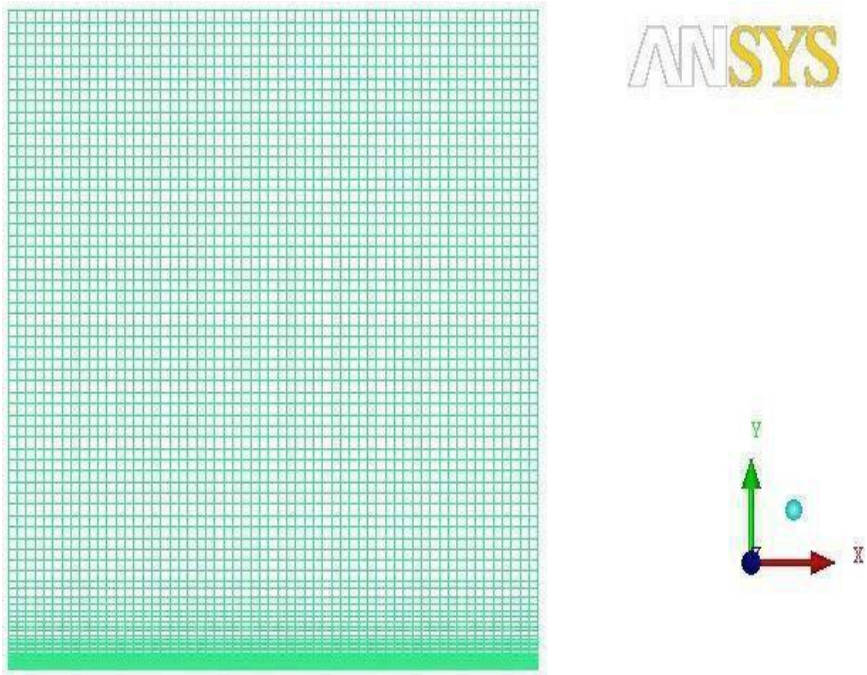
Description: Consider a plate of 1m and the flow of air is 0.00133 m/s. The plate is an stationary solid wall having no slip as its boundary condition.

Procedure:

- Geometry→ create point→ explicit coordinates→ 1(0,0,0), 2(1,0,0), 3(1,1,0) and 4(0,1,0) → ok
- Create/modify curve→ select 2 points→ middle click
- Select all points to make a rectangle



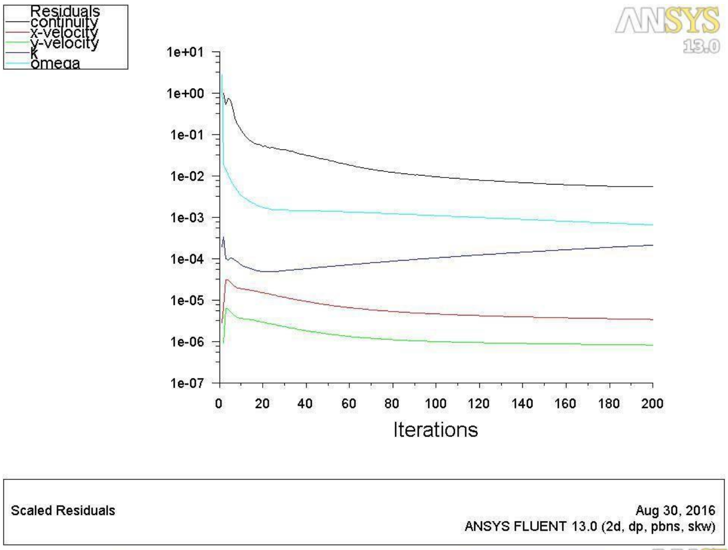
- Create/modify surface→ select all the lines→ surface is created
- Create part→ name inlet→ select the left edge→ middle click
- Similarly create outlet, top and bottom
- Switch off points and curves→ create part→ name surf→ click on surface→ ok
- Blocking→ create block→ select entities→ click spectacles→ middle click→ switch on 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 copy parameters→ nodes-60, spacing-0.01, ratio-1.1→ ok
- Blocking tree→ premesh→ right click→ convert structured to unstructured mesh

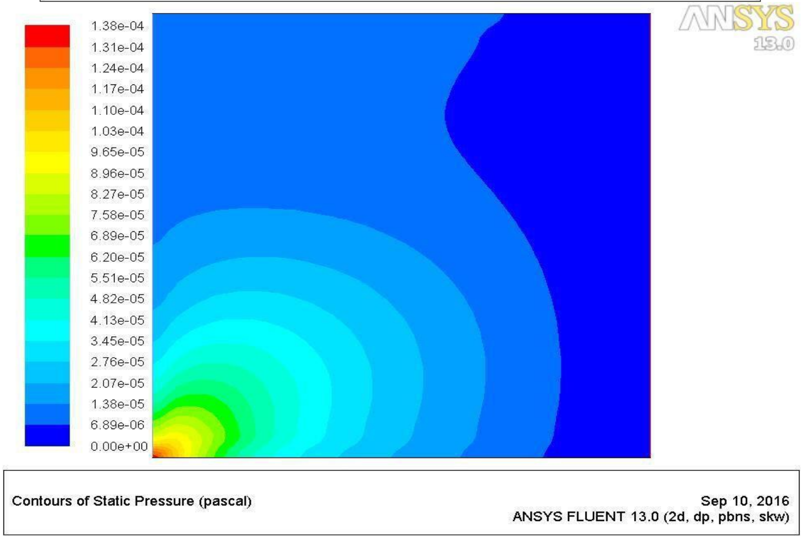


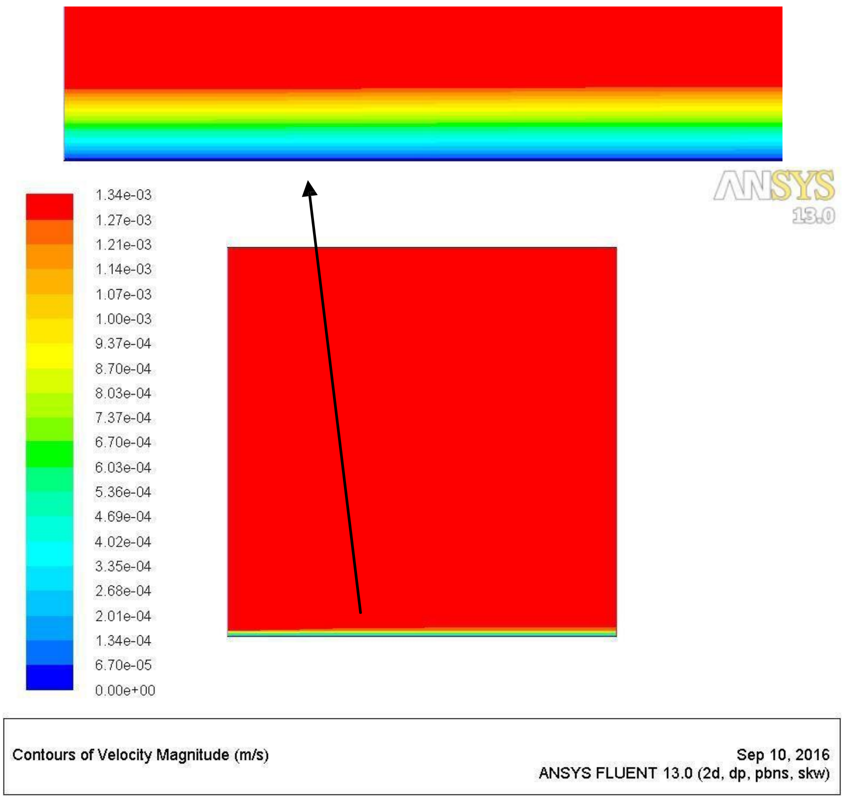
- Change the working directory
- output→ output solver→ fluent V6→ common-ansys→ ok

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 on calculate→ ok
- Results→ graphics and animations→ contour→ set up→ display options→ filled→ display
  Contour→ velocity→ display







8.1 Find out the effect of viscosity of water on flat plate.
8.2 Find the material of the plate on velocity profile.

4. LAMINAR FLOW THROUGH PIPE

AIM: To study characteristics of laminar flow through a pipe.

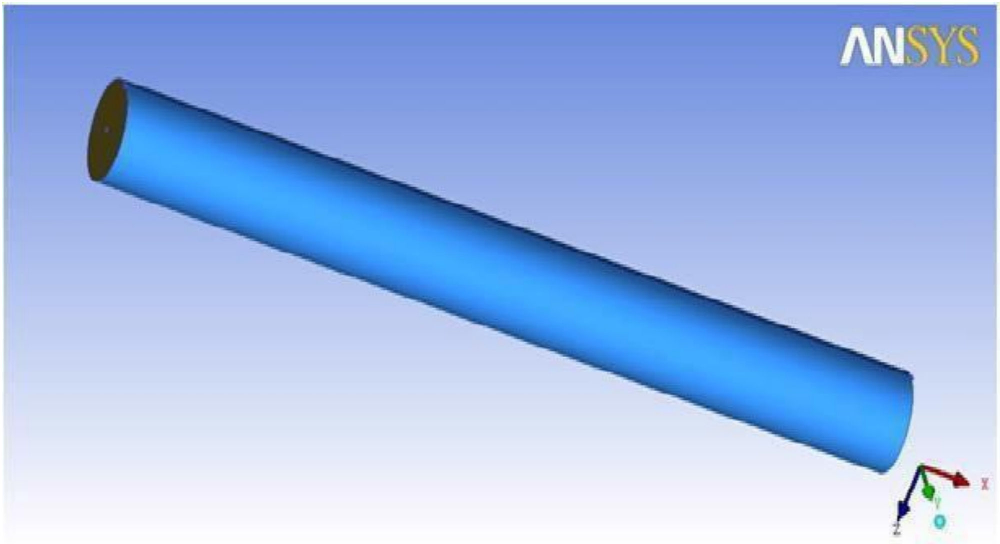
DESCRIPTION: Consider a pipe of radius 0.05 and 1 mt length. The freestream velocity considered is 40m/s.

STEPS INVOLVED:

1) Create A Geometry:

a) **Create a point:** Geometry →create point →explicit coordinates→(X, Y, Z) = (0, 0, 0) →apply→(X, Y, Z) = (1, 0, 0) →ok.

b) **Create a pipe:** Geometry→Create/modify surfaces→Standard shapes→ cylinder→radius1=radius2 = 0.05→select the 2 points →ok



2) Generation of parts:

- Part →create part→inlet→select inlet→ok.
- Part →create part →outlet →select outlet →ok.
- Part →create part →pipe →select pipe without inlet and outlet →ok.

3) Generation of blocking:

- Blocking →create block →solid →select pipe element with inlet and outlet→ apply→ok.
- Blocking →associate →edge to curve →select the 4 edges of the blocking at inlet →apply →select the inlet curve→ok.

- Blocking→associate →edge to curve→select the 4 edges of the blocking at outlet → apply→select the outlet curve→ok.

- Associate→faces to surface→select inlet face → apply →select as inlet→accept →ok.

- Associate →faces to surface →select outlet face →apply →select as outlet →accept →ok.

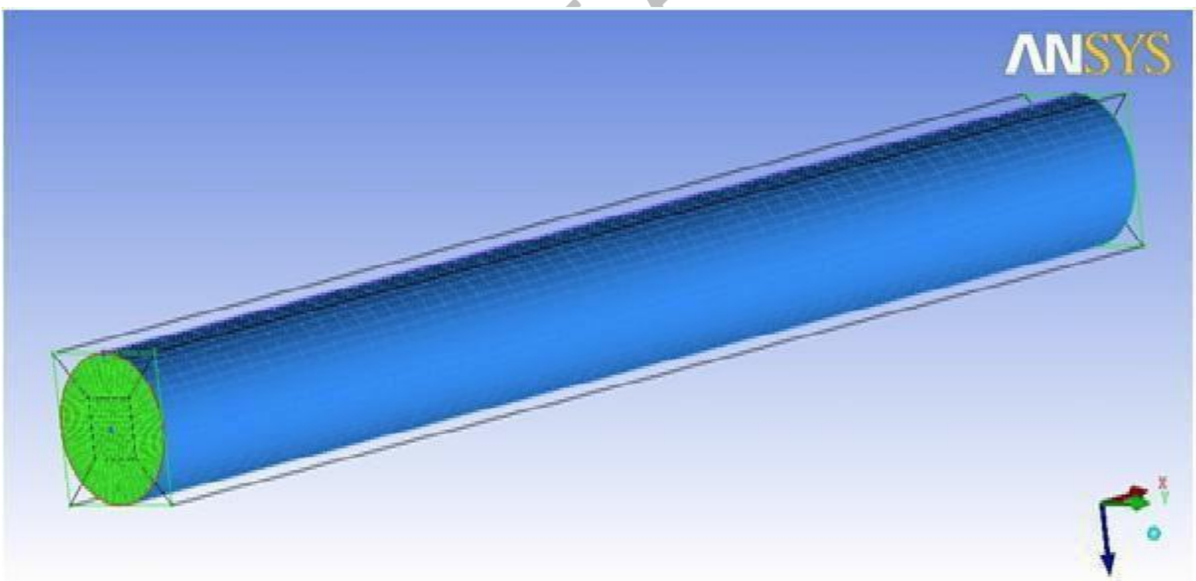
- Associate →face to surface →select pipe faces →apply →select as pipe → accept →
- Blocking →split block →O grid block →select the 2 faces (inlet & outlet) →apply →ok.

4) Generation of Meshing:

- Blocking →pre-mesh parameters →edge parameters →switch on the copy parameters →select 1 edge →give no. of nodes =20→ok.

- Repeat the above steps to the remaining edges also and then apply.

- Blocking →pre-mesh →compute.



- Blocking →pre-mesh →convert to unstructured mesh.

5) Generation of Solver file :

- Output→select solver→ansys.cfx→ANSYS→ok.

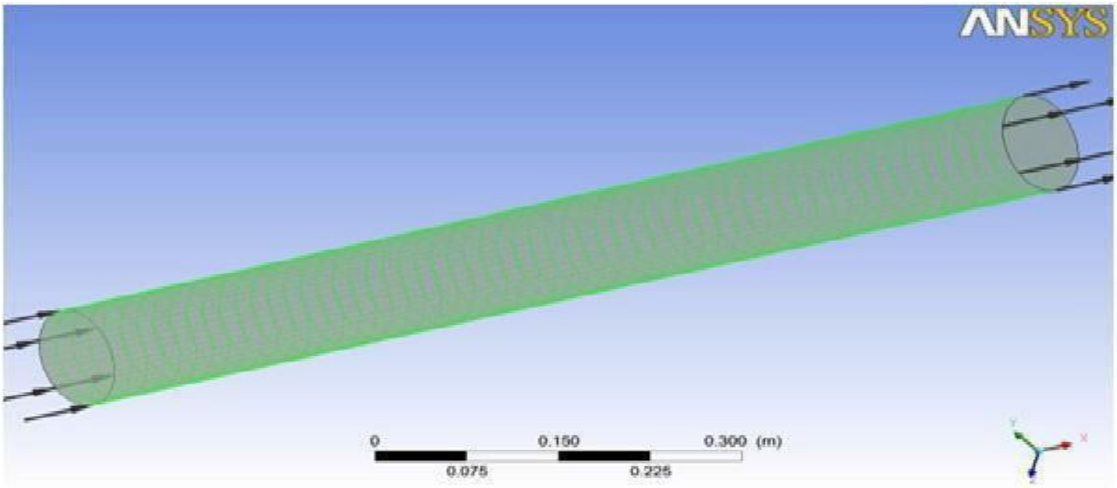
- Output →write input →done →check the file is saved folder →ok.

6) Solution in CFX Solver:

- Start →programs →ANSYS →fluid dynamics →CFX →ok.
- Change the working directory (where ICEM CFD mesh file was saved)→Click on CFX-PRE.

7) CFX-PRE:

- File→new case→general→ok.
- File→import→mesh→select the meshed filed→ok.
- Boundary→Boundary1→Boundary type as: inlet→location as: inlet→boundary conditions as: velocity=40m/s→ok.
- Boundary→Boundary2→boundary type as: outlet→location as: outlet→boundary conditions as: static pressure=0→ok.
- Boundary→Boundary3→boundary type as: wall→location as: pipe→boundary conditions as: no slip condition, smooth wall→ok.
- Domain→basic settings→location as: solid→domain type as: fluid domain→material as: air→ok.
- Solver control→basic settings→ max.itterationsas:1000→residual target as: 0.000000001→ok.
- Write solver input file→give the name of the file→ok.



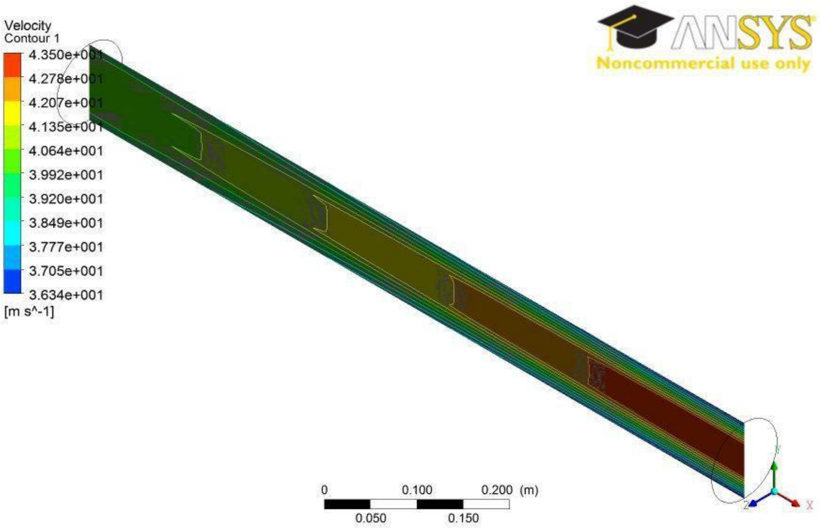
8) CFX-Solver Manager:

- File→Define run→Solver input file→Select the file→Start run.

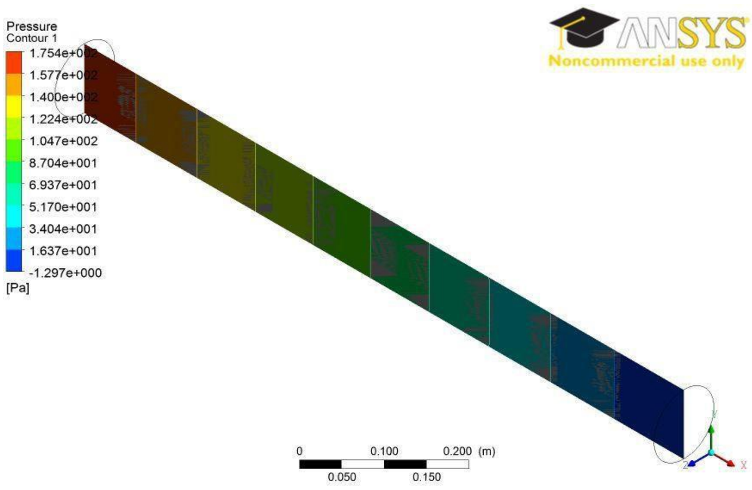
9) CFD POST

- Select →Location →Plane →XY Plane → Apply
- Contours →Location → plane→Variable→ Velocityand pressure →Apply.

Incoming flow through inlet



pressure contour



Exercise problems

9.1 Find out the effect convergent section on velocity.

9.2 Find the minor losses in a bend pipe.

5. FLOW PAST OVER A CYLINDER

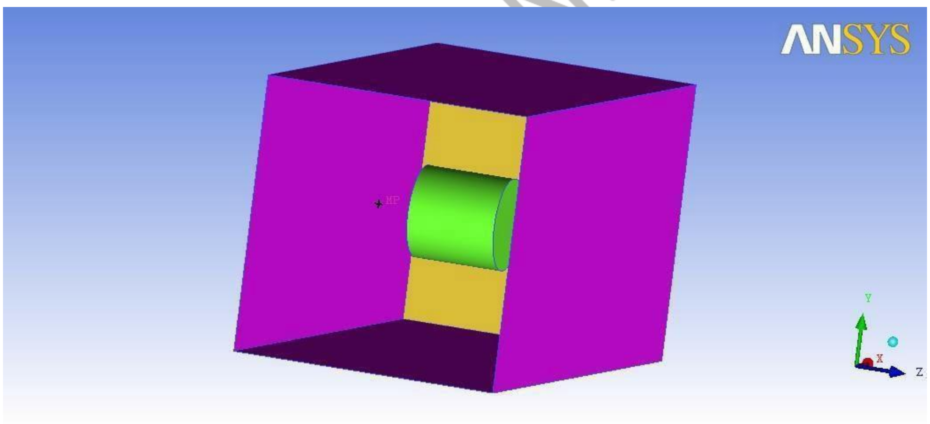
AIM : To study the characteristics of flow over a cylinder.

DESCRIPTION: Consider a cyclinder of 3m radius and 6m height. The free stream velocity considered 20m/s. the properties of air is ρ=1.18kg/m^3.

PROCEDURE:

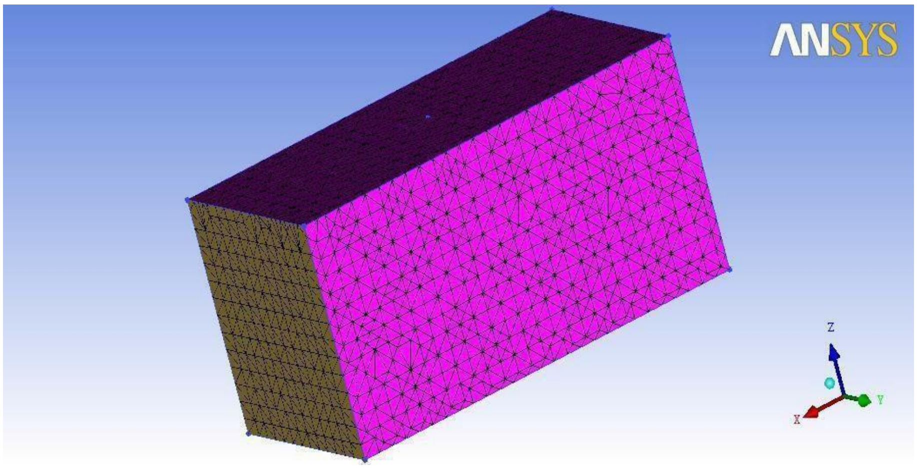
CREATION OF GEOMETRY:

- 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 of cylinder → apply



CREATION OF PARTS AND MESH GENERATION:

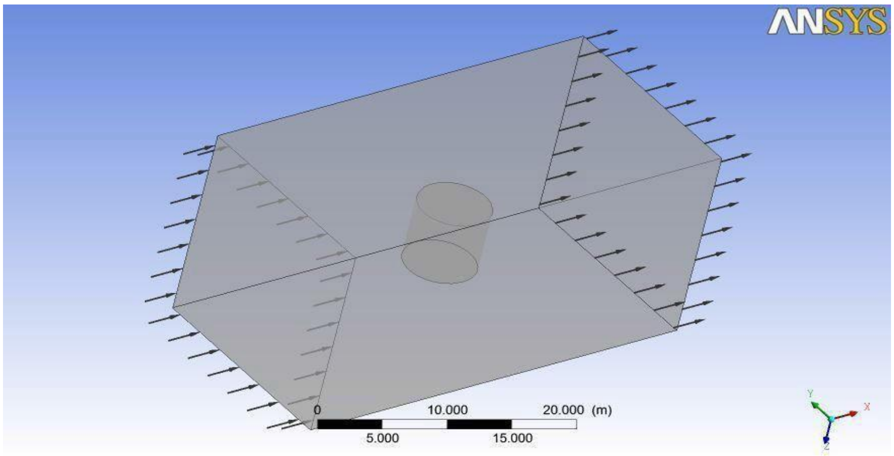
- parts→ create parts → (part name) → select entities → middle click (create parts according to the problem i.e. inlet, outlet, cylinder & free slip wall)
- geometry → solid → part(mp) → select two points lying outside the cylinder → apply.
- Mesh → mesh parameters → cylinder -1.5, inlet-2.5, outlet-2.5, 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

PROBLEM DEFINITION IN CFX-PRE:

- CFX → change the working directory → cfx-pre File → new case
- → general → apply.
- Mesh → import mesh → ICEM CFD → OK domain → fluid
- domain → air at 25c
- 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



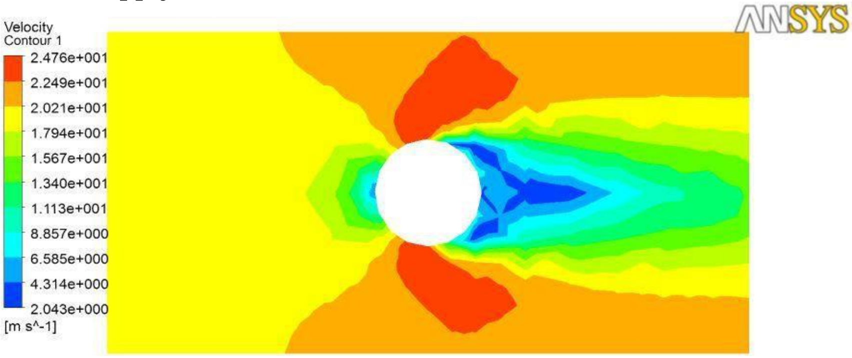
Solve:

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

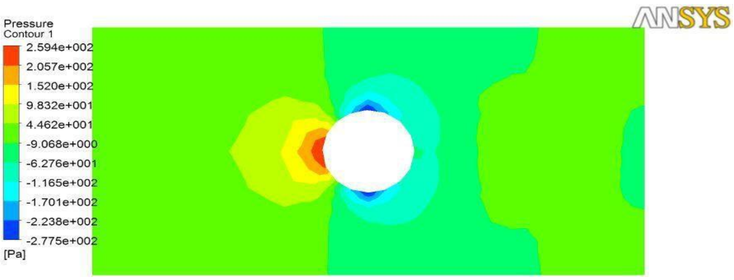
Post processing:

- CFD post → load result →
- select .res file Location →
- plane → Z=9 apply

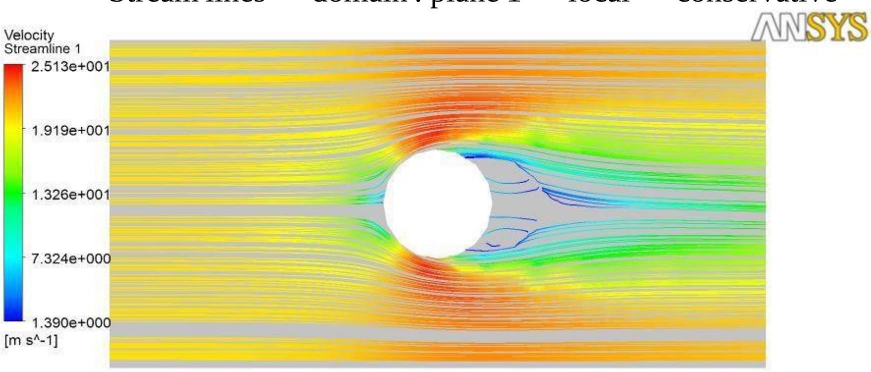
Contours → domain: plane1 → velocity → local → conservative → apply.



- Contours → domain: plane1 → pressure→ local → conservative → apply.



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



Exercise problems

5.1 Find out the effect of supersonic flow over cylinder .

5.2 Find the minor losses in a bend pipe

6. *Numerical simulation of flow through CD Nozzle*

Aim: Consider the nozzle having a cross sectional area A varies with axial distance from the throat, according to the formula $A = 0.1+X^2$; where X varies from $-0.5<X<0.5$. Stagnation pressure $P_o = 101325$ pa; stagnation temperature $T_o = 300K$;

Software: ICEM and CFX

Procedure:

Pre processing:

1. **Create geometry in ICEM**
   - Create the vertex data of Nozzle contour variation along axis line with vertices (-0.5, 0) and (0.5, 0) and $A = 0.1+X^2$ in excel sheet.
   - Import vertex data into ICEM by creating .dat file.
   - Create edges by create/modify curves and face by create/modify faces of nozzle using geometry function tab.
   - Create body point.
   - Create parts inlet, outlet and wall.

2. Mesh geometry:

- Click Mesh Function tab
- Give global mesh parameters in global mesh setup like element scale factor and element max size.
- Create partmesh setup for inlet, outlet and wall.
- Compute mesh – volume mesh – mesh type – select geometry – compute.
- Edit mesh – check mesh – quality mesh – smooth mesh.

3. Export mesh

- Output – output to cfx – save project – output type – output scale factor - .msh file created.

4. CFX:

- CFX Pre – New file – general.
- Mesh - Import mesh – ICEM CFD – open .msh file.

Domain - Default Domain Modified

Type	Fluid		
Location	FLUID		
Materials			
Air Ideal Gas			
Fluid Definition	Material Library		
Morphology	Continuous Fluid		
Settings			
Buoyancy Model	Non Buoyant		
Domain Motion	Stationary		
Reference Pressure	1.0000e+00 [atm]		
Heat Transfer Model	Isothermal		
Fluid Temperature	2.5000e+01 [C]		
Turbulence Model	k epsilon		
Turbulent Wall Functions	Scalable		
Domain	**Boundaries**		
Default Domain Modified	**Boundary - inlet**		
	Type	INLET	
	Location	IN	
	Settings		
	Flow Regime	Subsonic	
	Mass And Momentum	Normal Speed	
	Normal Speed	2.8000e+02 [m s^-1]	
	Turbulence	Medium Intensity and Eddy Viscosity Ratio	
	Boundary - outlet		
	Type	OUTLET	
	Location	OUT	
	Settings		
	Flow Regime	Subsonic	
	Mass And Momentum	Average Static Pressure	
	Pressure Profile Blend	5.0000e-02	

	Relative Pressure	1.0132e+05 [Pa]
	Pressure Averaging	Average Over Whole Outlet
	Boundary - WALL	
	Type	WALL
	Location	Wall
	Settings	
	Mass And Momentum	No Slip Wall
	Wall Roughness	Smooth Wall

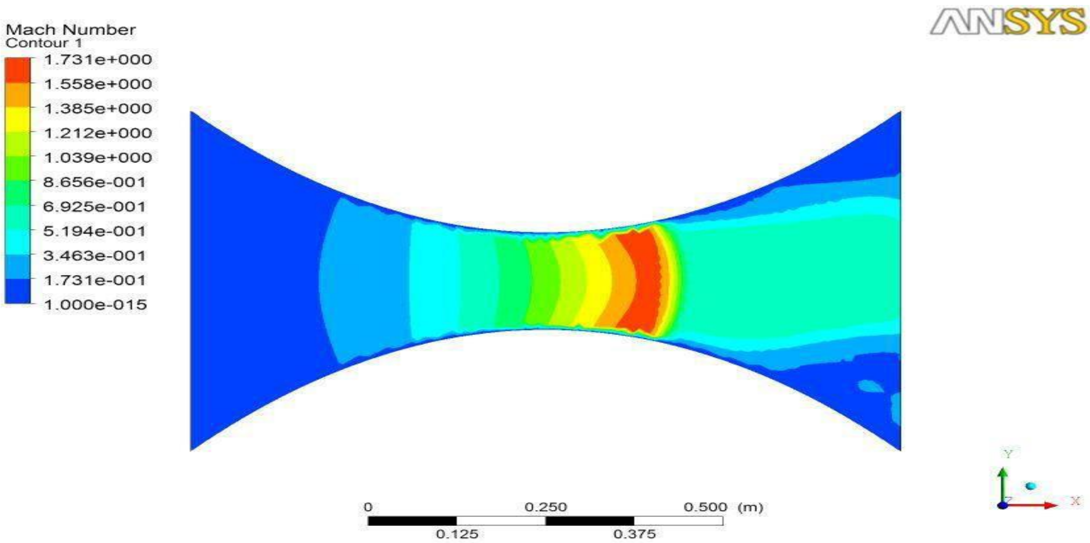
5. Solve the problem:

- Solver Control – Basic setting
- Advection scheme
- Turbulence Numerics
- Convergence controls
- Fluid timescale control
- Convergence criteria

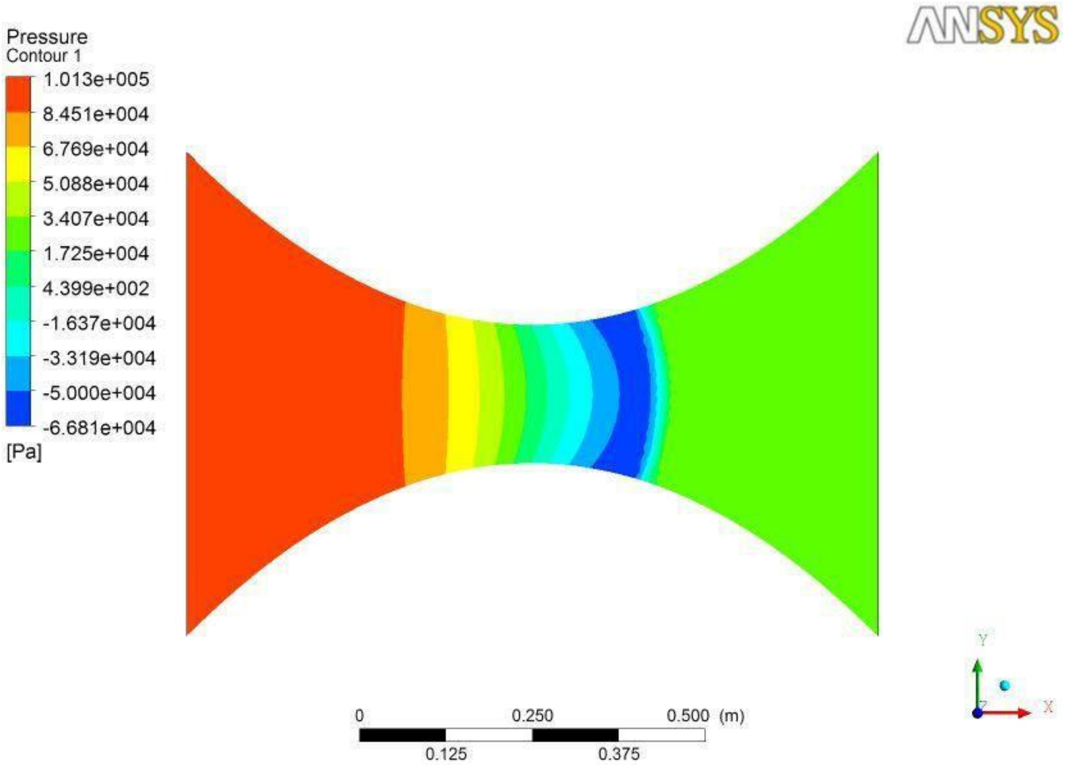
POST PROCESSING

6. Analyze results:

- Create plane
- Create contour – pressure, mach number
- Create streamlines
- Create chart for temperature along nozzle axis.



Mach number contour



Pressure contour

7. Numerical simulation of flow over wing

Aim: To obtain flow field over a finite rectangular wing with incoming flow conditions velocity 120 m/s, Pressure 1 atm and air at 25^0c.

Software: ICEM and CFX

Procedure: **Pre processing:**

1. Create geometry in ICEM

- **Import vertex data of airfoil coordinates from File** → Import Geometry → Formatted point data.
- Join the points by selecting geomtry tab, Create curve from points.
- Create a line in Z direction to extrude airfoil.
- Geometry → Create surface → Curve driven → select driving curve and then the remaining lines under the driven curve → Apply.
- Geometry → Create surface → Simple surface → for upper and lower surface.
- Create domain for the flow analysis.
- Delete unwanted surfaces, lines and points.
- From the model tree → right click on parts → create part → Name

  Inlet – in_1

  Outlet – out_1

  Top – top_1

  Side – sidewall_1

  Bottom – Bot _ 1

  Airfoil – Airfoil_1

  Apply

- Geometry → Create body → Location → centroid of 2 points → select the two diagonally opposite points on the model such that point should not be within airfoil→ Apply.

2. Mesh geometry:

- Click Mesh Function tab
- Give global mesh parameters in global mesh setup like element scale factor and element max size = 0.1 → display on → apply.
- Create partmesh setup for inlet, outlet and wall.

- Compute mesh → volume mesh → mesh type → tetra/mixed → create prism layers → create hexa-core → mesh method → Robust[octree] → select geometry – compute.
- Edit mesh – check mesh – quality mesh – smooth mesh.

3. Export mesh

- Output – output to cfx – save project – output type – output scale factor - .msh file created.

4. CFX:

- CFX Pre – New file – general.
- Mesh - Import mesh – ICEM CFD – open .msh file.
- Default Domain Basic Settings Materials Air at 25^0c, Reference Pressure 1 atm

Domain - Default Domain Modified	
Type	Fluid
Location	FLUID
Materials	
Air Ideal Gas	
Fluid Definition	Material Library
Morphology	Continuous Fluid
Settings	
Buoyancy Model	Non Buoyant
Domain Motion	Stationary
Reference Pressure	1.0000e+00 [atm]
Heat Transfer Model	Isothermal
Fluid Temperature	2.5000e+01 [C]
Turbulence Model	SST
Turbulent Wall Functions	Scalable

<table>
<tr><th>Domain</th><th colspan="2">Boundaries</th></tr>
<tr><td rowspan="8">Default Domain Modified</td><td colspan="2">Boundary - inlet</td></tr>
<tr><td>Type</td><td>INLET</td></tr>
<tr><td>Location</td><td>IN_1</td></tr>
<tr><td colspan="2">Settings</td></tr>
<tr><td>Flow Regime</td><td>Subsonic</td></tr>
<tr><td>Mass And Momentum</td><td>Normal Speed</td></tr>
<tr><td>Normal Speed</td><td>1.3000e+02 [m s^-1]</td></tr>
</table>

	Turbulence	low Intensity = 1%
	Boundary - outlet	
	Type	OUTLET
	Location	OUT_1
	Settings	
	Flow Regime	Subsonic
	Mass And Momentum	Average Static Pressure
	Pressure Profile Blend	5.0000e-02
	Relative Pressure	1.0132e+05 [Pa]
	Pressure Averaging	Average Over Whole Outlet
	Boundary - WALL	
	Type	WALL
	Location	Airfoil_1, Top_1, Bot _1
	Settings	
	Mass And Momentum	No Slip Wall
	Wall Roughness	Smooth Wall

- Click ok and define run to start solution.

POST PROCESSING

5. Analyze results in CFX POST:

- Create plane
- Create contour – pressure, mach number
- Create streamlines
- Create chart for velocity along plate.

RESULTS:

8. Numerical simulation of combustion

Aim: The problem to be considered is shown schematically in below figure. The combustion fuel enters into combustion chamber through the fuel nozzle. Through the air inlets air enters from the primary and secondary zones. Both the air and fuel mix in the combustion chamber.

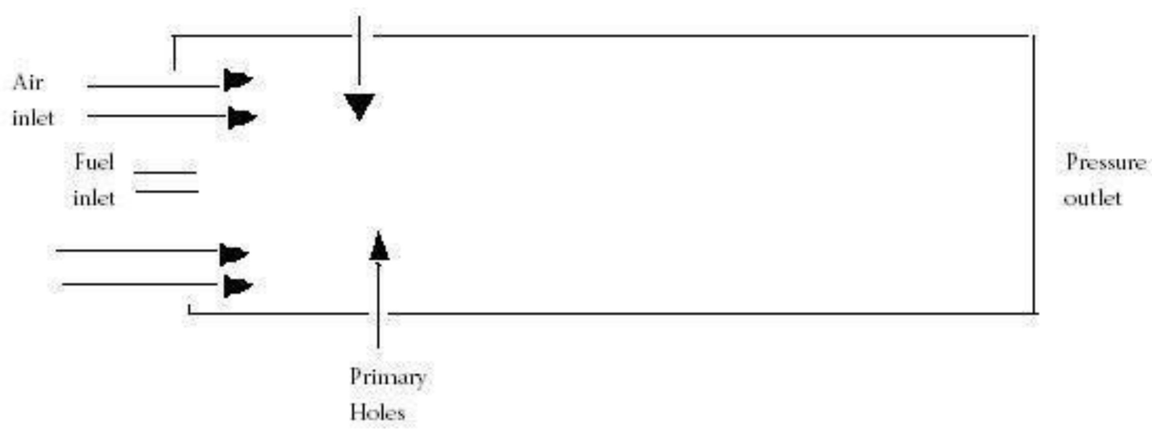


Fig 8.1 Problem Description

Procedure:

- Geometry→ create point→ explicit coordinates→ 1(0,0,0), 2(6,0,0), 3(6,1,0) and
  4(0,1,0) → ok
- Create/modify curve→ select 2
- points→ middle click Select all
  points to make a rectangle
- Geometry→ create point→ explicit coordinates→ 1(0,1,0), 2(0,1.25,0), 3(0,2.75,0) and
  4(0,3,0), 5(0,4.75,0), 6(0,5,0) → ok
- Select trim using above points to create air and fuel inlets as shown in figure.
- Create/modify surface→ select the entire lines→ surface is
- created Create part→ name air and fuel 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→ switch on 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 copy parameters→ nodes-60→ ok

- Blocking tree→ premesh→ right click→ convert structured to unstructured mesh
- Change the working directory
- output→ output solver→ fluent V6→ common-ansys→ ok

FLUENT:

Grid

- Start FLUENT 2d mode
- Read→ Case and select the mesh file combustionchamber.msh
- Check the grid using grid→check

Models

- ## Defining the domain

  a) Select 2D from the Space list.
  b) Retain the default settings for the remaining pressure-based solver parameters.
  c) Click OK to close the Solver panel.

- Enable heat transfer by enabling the energy equation.
- Select the standard k-epsilon turbulence model.

  a) Select k-epsilon from the Model list. The Viscous Model panel will expand to provide further options for the k-epsilon model.
  b) Retain the default settings for the k-epsilon model.
  c) Click OK to close the Viscous Model panel.

- ## Enabling chemical species transport and reaction

  a) Select Species Transport from the Model list. The Species Model panel will expand to provide further options for the Species transport model.
  b) Enable Volumetric in the Reactions group box.
  c) Select n-butane-air from the Mixture Material drop-down list.
  d) Select Eddy-Dissipation from the Turbulence-Chemistry Interaction list.
  e) The eddy-dissipation model computes the rate of reaction under the assumption that chemical kinetics are fast compared to the rate at which reactants are mixed by turbulent fluctuations (eddies).

f) Click OK to close the Species Model panel.

Materials

- Revise the properties for the mixture materials.
- The Materials panel will display the mixture material (n-butane-air) that was selected in the Species Model panel.
- The properties for this mixture material have been copied from the FLUENT database and will be modified in the following steps.
  a) Retain the default selection of mixture in the Material Type drop-down list.
  b) Click the Edit... button to the right of the Mixture Species drop-down list to open the Species panel.
  c) Select constant from the Cp drop-down list and enter 1000 J/kg K for the specific heat value.
  d) Scroll down to find the Cp drop-down list and number-entry box.
  e) Click Change/Create to accept the material property settings.
  f) Close the Materials panel.

Boundary Conditions

- **Set the boundary conditions for the air inlet**

  (a) Enter air-inlet for Zone Name. This name is more descriptive for the zone than velocity-inlet-8.
  (b) Enter 5 m/s for Velocity Magnitude.
  (c) Select Intensity and Hydraulic Diameter from the Specification Method dropdown list in the Turbulence group box.
  (d) Retain the default value of 10% for Turbulent Intensity.
  (e) Enter 1 m for Hydraulic Diameter.
  (f) Click the Thermal tab and type the value of 850 K for Temperature.
  (g) Click the Species tab and enter 0.23 for o2 in the Species Mass Fractions group box.
  (h) Click OK to close the Velocity Inlet panel.

- **Set the boundary conditions for the fuel inlet**

  (a) Enter fuel-inlet for Zone Name. This name is more descriptive for the zone than velocity-inlet-6.
  (b) Enter 80 m/s for the Velocity Magnitude.

(c) Select Intensity and Hydraulic Diameter from the Specification Method dropdown list in the Turbulence group box.

(d) Retain the default value of 10% for Turbulent Intensity.

(e) Enter 0.01 m for Hydraulic Diameter.

(f) Click the Thermal tab and retain the default value of 300 K for Temperature.

(g) Click the Species tab and enter 1 for ch4 in the Species Mass Fractions group box.

(h) Click OK to close the Velocity Inlet panel.

- **Set the boundary conditions for the exit boundary**

a) Retain the default value of 0 Pa for Gauge Pressure.

b) Select Intensity and Hydraulic Diameter from the Specification Method dropdown list in the Turbulence group box.

c) Retain the default value of 10% for Turbulent Intensity.

d) Enter 0.45 m for Backflow Hydraulic Diameter.

e) Click the Thermal tab and retain the default value of 300 K for Backflow Total Temperature.

f) Click the Species tab and enter 0.23 for O_2 in the Species Mass Fractions group box.

g) Click OK to close the Pressure Outlet panel.

- **Set the boundary conditions for the outer wall**

a) Enter outer-wall for Zone Name.

b) Click the Thermal tab.

i. Select Temperature from the Thermal Conditions list.

ii. Retain the default value of 300 K for Temperature.

c) Click OK to close the Wall panel.

- **Set the boundary conditions for the primary holes**

a) Select the primary holes zone.

b) Enter 5 m/s for Velocity Magnitude.

c) Select Intensity and Hydraulic Diameter from the Specification Method dropdown list in the Turbulence group box.

d) Retain the default value of 10% for Turbulent Intensity.

e) Enter 1 m for Hydraulic Diameter.

f) Click the Thermal tab and type the value of 850 K for Temperature.

g) Click the Species tab and enter 0.23 for o2 in the Species Mass Fractions group box.

h) Click OK to close the Velocity Inlet panel.

Initial Solution

- Initialize the field variables.
  a) Select all-zones from the Compute From drop-down list.
  b) Click Init to initialize the variables.
  c) Close the Solution Initialization panel.
- **Set the under-relaxation factors for the species.**

  The default under-relaxation parameters in FLUENT are set to high values. For a combustion model, it may be necessary to reduce the under-relaxation to stabilize the solution. Some experimentation is typically necessary to establish the optimal under-relaxation. For this analysis, it is sufficient to reduce the species under- relaxation to 0.95.

  a) Enter 0.95 for each of the species (ch_4, O_2, co_2, and h_2o) in the Under-Relaxation Factors group box.
  b) Scroll down the Under-Relaxation Factors group box to find the species.
  c) Click OK to close the Solution Controls panel.
- Save the case file
- Start the calculation by requesting 500 iterations.
- Save the case and data files.

Postprocessing

- **Display Filled contours of mass fraction for C_4H_{10}**

  a) Select Species... and Mass fraction of ch4 from the Contours of drop-down lists.
  b) Click Display.

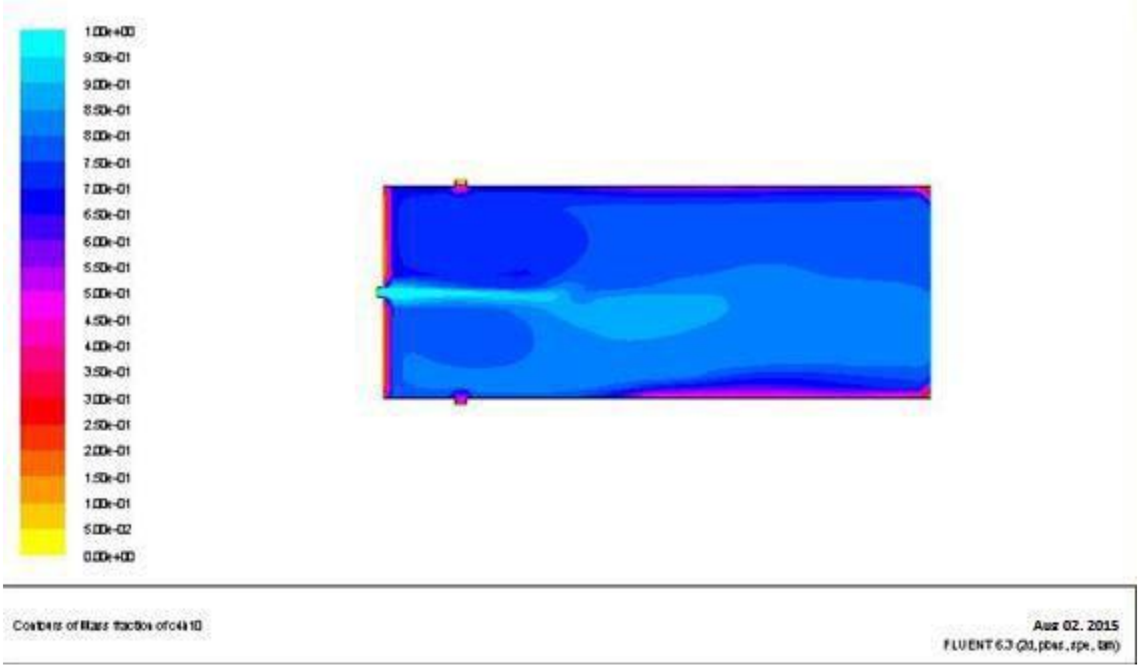


Fig 8.3. Mass Fraction of Butane (C4H10)

- In a similar manner, display the contours of mass fraction for the specie O_2.

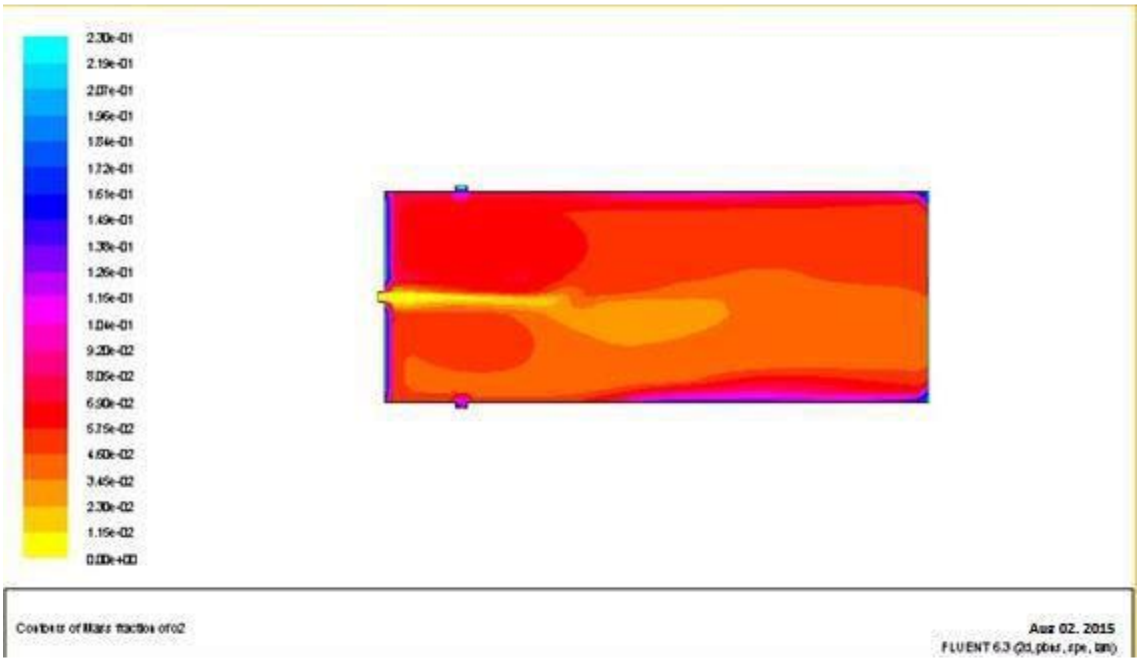


Fig 8.4 Mass fraction of O_2

9. SOLUTION FOR THE ONE DIMENSIONAL WAVE EQUATION USING EXPLICIT METHOD OF LAX (CODE DEVELOPMENT).

The one dimensional scalar wave equation is given as

$$\frac{\partial u}{\partial t} + c\frac{\partial u}{\partial x} = 0$$

This equation represents a linear advection process with wave speed c = constant, which is the speed of the travelling wave or the speed of propagation. u(x,t) is the signal or wave information. The wave propagates at constant speed to the right if c > 0 and to the left if c <1. The spatial domain can vary from -∞ to ∞. Suppose the initial conditions are

$$u(x,0) = u_0(x)$$

where $u_0(x)$ is any function. The exact solution to the wave equation then is

$$u = u_o(x - ct)$$

$u_0(x)$ is called the wave shape of wave form. Travelling or propagation here means that the shape of the signal function with respect to x stays constant, however the function is translated left or right with time at the speed c.

Numerical Solution

Method of descretisation – finite difference form

Replace the spatial partial derivative with a central difference expression

$$\frac{\partial u}{\partial x} = \frac{u_{j+1}^n - u_{j-1}^n}{2\Delta x}$$

Where n is the temporal index and j is the spatial index. Replace the time derivative with a forward difference formula

$$\frac{\partial u}{\partial t} = \frac{u_j^{n+1} - u_j^n}{\Delta t}$$

We then have

$$\frac{u_j^{n+1} - u_j^n}{\Delta t} = -c\frac{u_{j+1}^n - u_{j-1}^n}{2\Delta x} \qquad (1)$$

Now let us replace u_j^n by an average value between grid points j+1 and j-1 as

$$u_j^n = \frac{u_{j+1}^n + u_{j-1}^n}{2}$$

Substituting this in equation (1) we get the explicit method of Lax for the 1D scalar wave equations as,

$$u_j^{n+1} = \frac{u_{j+1}^n + u_{j-1}^n}{2} - c\frac{\Delta t}{\Delta x}\frac{u_{j+1}^n - u_{j-1}^n}{2}$$

Test Case for the numerical solution

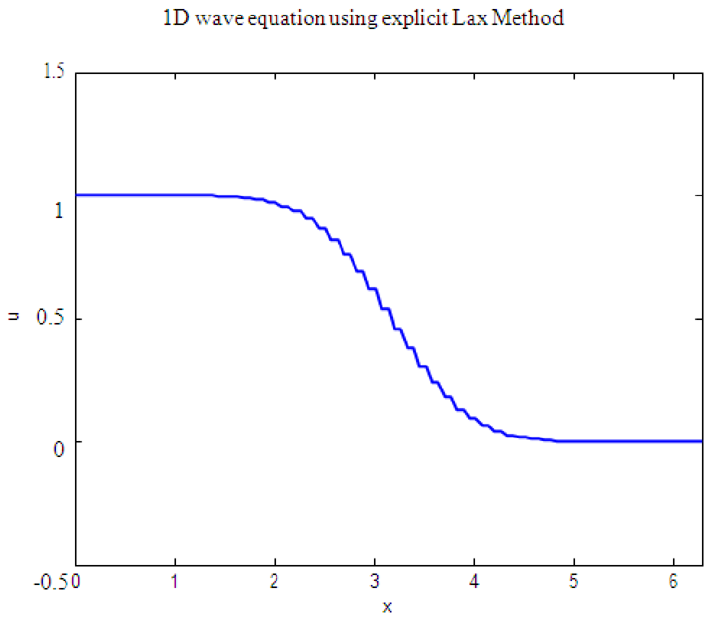
Solve the one dimensional wave equation in the spatial domain of [0, 2*pi] with an initial
step function
condition given by
$U_0(x,0) = 1$ for $x \leq$
pi-1
= 0 otherwise
Choose 100 grid points and find the wave form at t = 0.2 s.

Matlab code for the one dimensional wave equation

```
% Solves the one dimensional scalar wave equation du/dt + du/dx = 0
[0,2*pi]
% Using LAX METHOD
clc;
clear all;
t0 = 0;
tf = 1;
M = 100; % number of points in x
direction
N = 100; % number of points in y
direction

% define the mesh in
space
dx =2*pi/M;
x = 0:dx:2*pi;

% define the mesh in
time
dt = (tf-t0)/N;
t = t0:dt:tf;

% calculate value for
lamda
c = 1;
lambda = c*dt/dx
display('lambda should be less than 1 for stability:')

% choose the wave number of the initial data and give its decay rate
 u0 = x<=(pi-1);
u = zeros(M+1,N+1);
u(:,1) = u0;
```

```
% Implement the time marching Lax scheme:
for n=1:N
 for i=2:M
u(i,n+1)=(u(i+1,n)+u(i-1,n))/2-(lambda/2)*(u(i+1,n)-u(i-1,n));
         end
% Introduce exact values at the endpoints.
         u(1,n+1)=1;
        u(M+1,n+1)=0;
end
% plot the result in 21 intervals
for  j=0:20
plot(x,u(:,1+5*j),'LineWidth',2);
axis([0,2*pi,-0.5,1.5]);
title('1D wave equation using explicit Lax Method','FontSize',12)
 xlabel('x');
ylabel('u');
pause(1)
end
%plot(x,u(:,101));
```

Results:



Exercise problems:

9.1 Write a Matlab Program to solve one dimensional wave equation using FTCS method.

9.2 Write a Matlab program to find wave propagation at t=0.5 secs

10. SOLUTION FOR THE ONE DIMENSIONAL TRANSIENT HEAT CONDUCTION EQUATION USING EXPLICIT METHOD (CODE DEVELOPMENT)

The one dimensional transient (unsteady) heat conduction equation is given as

$$\frac{\partial T}{\partial t} = \alpha \frac{\partial^2 T}{\partial x^2}$$

Where α is the thermal diffusivity

This equation represents the conduction of heat energy in time and space. Transient nature of this equation is represented in the dependence of temperature with time as opposed to a steady state condition.

Numerical Solution

Method of discretization – finite difference form

Replace the time derivative with a forward difference expression

$$\frac{\partial T}{\partial t} = \frac{T_j^{n+1} - T_j^n}{\Delta t}$$

Where n is the temporal index and j is the spatial index.
Replace the second order spatial derivative on the RHS with a central difference formula

$$\frac{\partial^2 T}{\partial x^2} = \frac{T_{j+1}^n + T_{j-1}^n - 2T_j^n}{\Delta x^2}$$

We then have

$$\frac{T_j^{n+1} - T_j^n}{\Delta t} = \alpha \frac{T_{j+1}^n + T_{j-1}^n - 2T_j^n}{\Delta x^2} \qquad (1)$$

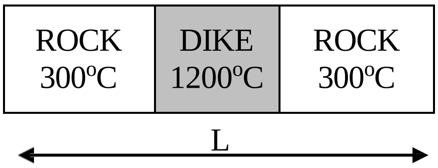
i.e. $$\boldsymbol{T_j^{n+1} = (1 - 2A)T_j^n + AT_{j+1}^n + AT_{j-1}^n} \qquad (2)$$

where $$A = \alpha \frac{\Delta t}{\Delta x^2}$$

Equation (2) is the final explicit update equation for the one dimensional transient heat conduction equation.

Test Case for the numerical solution

A country rock has a temperature of 300^{o}C and the dike a width of 5m, with a magma temperature of 1200^{o}C. Total length of the rock formation is 100m. Initial conditions are temperatures of 300^{o}C and 1200^{o}C for the rock and dike respectively. Boundary conditions at x = -L/2 and x = L/2 are at 300^{o}C (see figure). Find the temperature distribution after 100 days. Use 200 grid points in the x direction with a 1 day time interval.

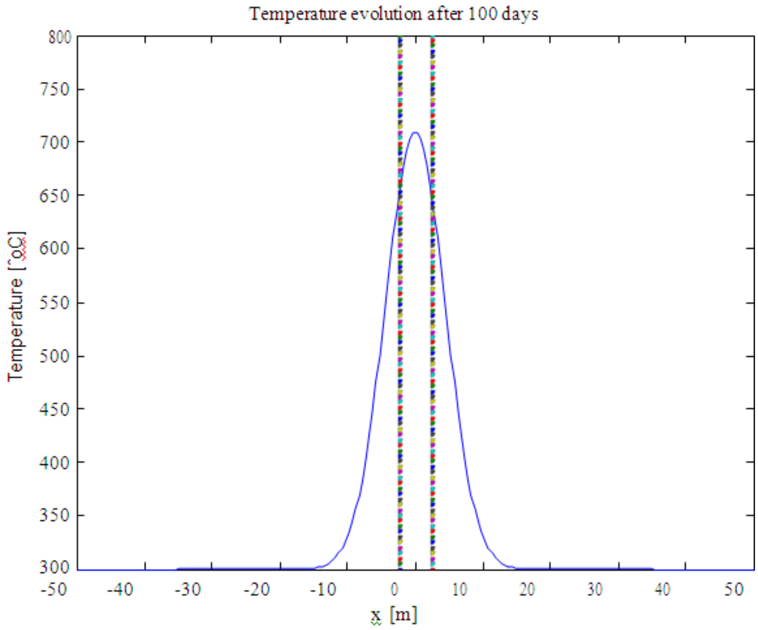


Matlab code for the one dimensional transient heat conduction equation

```
% Solves the 1D heat equation with an explicit finite difference scheme
clear all
clc
%Physical parameters
L = 100; % Length of modeled domain [m]
Td = 1200; % Temperature of magma [0C]
Tr = 300; % Temperature of country rock [0C]
kappa = 1e-6; % Thermal diffusivity of rock [m2/s]
W = 5; % Width of dike [m]
day = 3600*24; % # seconds per day
dt = 1*day; % Timestep [s]
% Numerical parameters
nx = 200; % Number of gridpoints in x-direction
nt = 100; % Number of timesteps to compute
dx = L/(nx-1); % Spacing of grid
 x = -L/2:dx:L/2;% Grid

% Setup initial temperature profile
T = ones(size(x))*Tr;
T(abs(x)<=W/2) = Td;
time = 0;
for n=1:nt % Timestep loop
% Compute new temperature
Tnew = zeros(1,nx);
for i=2:nx-1
Tnew(i) = T(i) + (kappa*dt/(dx)^2)*(T(i+1)-(2*T(i))+T(i-1));
end
% Set boundary conditions
Tnew(1) = T(1);
Tnew(nx) = T(nx);
% Update temperature and time
T = Tnew;
time = time+dt;
end
% Plot solution
plot(x,Tnew);
xlabel('x [m]')
ylabel('Temperature[^oC]')
title(['Temperature evolution after ',num2str(time/day),' days'])

% draw the dike boundaries
x1 = -2.5;
x2 = 2.5;
```

```
y = linspace(300,800);
% Plot the dike
boundaries
hold on
plot(x1,y, x2, y);
```

Results:



Exercise problems:

10.1 Write a Matlab Program to solve one dimensional heat conduction equation using Lax method.

10.2 Write a Matlab program to one dimensional heat conduction equation after 200 days.

11. GENERATION OF THE ALGEBRAIC GRIDS

Problem

Generate an algebraic grid about the upper surface of the airfoil. Points are clustered in j direction near the lower surface (using β=1.05 in algebraic grid). Make sure the number of points in i and j are flexible.

Introduction, Theory, & Formulations

A key component of grid generation is the conversion from the physical domain to the computational domain, in order to allow for equidistant grid lines in rectangular form. In considering a simple two dimensional case, physical coordinates x and y must be converted to computational coordinates ξ and η. These computational coordinates are furthermore known via the rectangular grid relations. As a result, they must be converted back into physical coordinates in order to be of use. For the particular case concerning an airfoil placed on the x axis, the following relationships exist:

$$x = \xi \qquad (1)$$

$$y = H.\frac{(\beta+1) - (\beta-1).(\frac{\beta+1}{\beta-1})^{1-\eta}}{(\frac{\beta+1}{\beta-1})^{1-\eta}+1} \qquad (2)$$

As can be seen, Eq. (1) simply states that the x coordinate is the ξ coordinate, as there exists no irregularities to alter that axis. The precise relationship in Eq. (2) is due to a required clustering near the bottom surface. Here, β represents the clustering parameter, which is given, and H represents the total height along the y axis. However, this does not account for the geometry of the airfoil, wherein its top surface coordinate is a function of the distance along the x axis. The exact equation is:

$$y = \frac{t}{0.2}.\left(0.2969x^{\frac{1}{2}} - 0.126x - 0.3516x^2 + 0.2843x^3 - 0.1015x^4\right) \qquad (3)$$

Here, y represents the max height of the airfoil, which would thus be the correspond to y=0 in Eq.(2). Height is determined by subtracting this value from maximum height. This allows a total expression for the grid y coordinative can be obtained. Note that the x used in Eq. (3) assumes 0 at the nose of the airfoil and 1 at the tail. The previous equations effectively define all that is needed to generate an algebraic grid. However, this grid will simply be used as a starting point for the generation of an elliptic grid. Thus, once x and y are obtained algebraically, they will be set as initial conditions for the x and y values used in order to perform iterations of the developed finite difference equations.

MATLAB code for Algebraic Grid Generation

```
%Algebraic Grid Generation
clear all;
clc;
%Assign values for t and beta
t=0.15;
beta=1.05;
%Prompt user for number of grid points
n=input('Enter the number of grid points in the i direction: ');
m=input('Enter the number of grid points in the j direction: ');
%Create zeroes matrix for surface
plots z=zeros(n,m);
%Assign lengths and values for eta and
xi
L=3;
eta=linspace(0,1,m);
xi=linspace(0,L,n);
%x is equal to xi
X=xi;
%Find height
ytop=2;
for  i=1:n
if X(i) < 1
ybottom(i)=0;
elseif X(i) > 2
ybottom(i)=0;
else
x2(i)=X(i)-1;
  ybottom(i)=(t/.2)*(0.2969*x2(i)^.5-0.126*x2(i)-0.3516*x2(i)^2+0.2843*x2(i)^3-0.1015*x2(i)^4);
end
  H(i)=ytop-ybottom(i);
  end
%Loop to calculate coordinates
zeta=beta+1;
gamma=beta-1;
alpha=zeta/gamma;
for i=1:n
for
j=1:m
chi=1-
eta(j);
y(i,j)=H(i)*(zeta-gamma*alpha^chi)/(alpha^chi+1)+ybottom(i);
x(i,j)=X(i);
end
end
surface(x,y,z);
xlabel ('x');
ylabel ('y');
title ('Algerbraic Grid');
```

Discussion of Results

Enter the number of grid points in i direction: 50
Enter the number of grid points in the j direction: 50

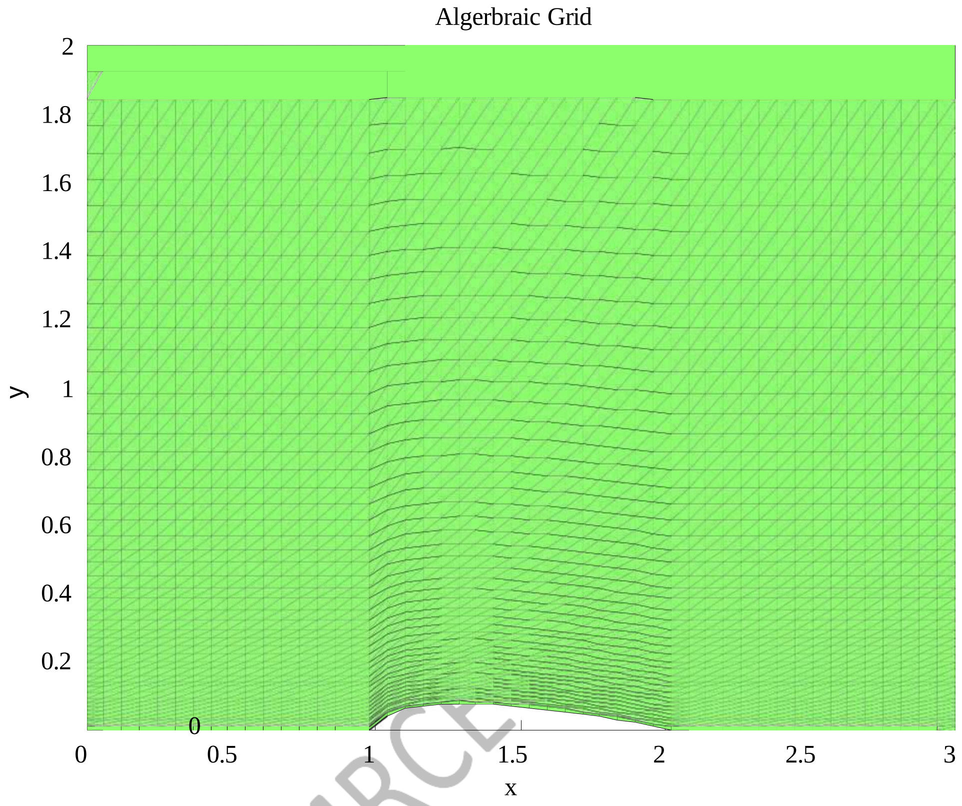


Figure shows the algebraic grid generation with the growth rate β=1.05 the grids are very fine at y=0 and it gets coarser as the y increases.
The value of growth rate β can be varied and you can see the difference in the growth rate of the grid.

Exercise problems:

11.1 Write a Matlab Program to generate algebraic grid over flat plate.

11.2 Write a Matlab program to generate algebraic grid over circle.

12. GENERATION OF THE ELLIPTIC GRIDS

Problem

Starting with an algebraic grid, generate an elliptic grid about the upper surface of the airfoil. Points are clustered in j direction near the lower surface (using β=1.05 in algebraic grid). Make sure the number of points in i and j are flexible.

Using a predetermined algebraic grid, an elliptic grid can be generated in order to fine tune the results for airfoil flow. Coding an algebraic grid necessitates an accounting for the geometry of the airfoil, as well as clustering via appropriate equations. Once these issues are addressed, partial differential equations can be utilized in order to generate an elliptic grid.

Introduction, Theory, & Formulations

A key component of grid generation is the conversion from the physical domain to the computational domain, in order to allow for equidistant grid lines in rectangular form. In considering a simple two dimensional case, physical coordinates x and y must be converted to computational coordinates ξ and η. These computational coordinates are furthermore known via the rectangular grid relations. As a result, they must be converted back into physical coordinates in order to be of use. For the particular case concerning an airfoil placed on the x axis, the following relationships exist:

$$x = \xi \qquad (1)$$

$$y = H.\frac{(\beta+1)-(\beta-1).(\frac{\beta+1}{\beta-1})^{1-\eta}}{(\frac{\beta+1}{\beta-1})^{1-\eta}+1} \qquad (2)$$

As can be seen, Eq. (1) simply states that the x coordinate is the ξ coordinate, as there exists no irregularities to alter that axis. The precise relationship in Eq. (2) is due to a required clustering near the bottom surface. Here, β represents the clustering parameter, which is given, and H represents the total height along the y axis. However, this does not account for the geometry of the airfoil, wherein its top surface coordinate is a function of the distance along the x axis. The exact equation is:

$$y = \frac{t}{0.2}.\left(0.2969x^{\frac{1}{2}} - 0.126x - 0.3516x^2 + 0.2843x^3 - 0.1015x^4\right) \qquad (3)$$

Here, y represents the max height of the airfoil, which would thus be the correspond to y=0 in Eq.(2). Height is determined by subtracting this value from maximum height. This allows a total expression for the grid y coordinative can be obtained. Note that the x used in Eq. (3) assumes 0 at the nose of the airfoil and 1 at the tail. The previous equations effectively define all that is needed to generate an algebraic grid. However, this grid will simply be used as a starting point for the generation of an elliptic grid. Thus, once x and y are obtained algebraically, they will be set as initial conditions for the x and y values used in order to perform iterations of the developed finite difference equations.

Two elliptic partial differential equations must be solved in order to fully define the desired grid. In doing this, boundary conditions are required. For this case, x and y values along the edges of the defined physical domain will be left in place. These being predefined allows all interior coordinates to be developed. The following system of elliptic partial differential equations can be used to define the domain:

$$\frac{\partial^2 \xi}{\partial x^2} + \frac{\partial^2 \xi}{\partial y^2} = 0 \qquad (4)$$

$$\frac{\partial^2 \eta}{\partial x^2} + \frac{\partial^2 \eta}{\partial y^2} = 0 \qquad (5)$$

Here, the subscripts denote second order derivative of that variable. Notice that these equations do not express x and y as dependent variables. Rather, they are treated as the independent variables, requiring a transformation. When such a mathematical transformation is preformed Eqs. (4) And (5) become, respectively:

$$a.\frac{\partial^2 x}{\partial \xi^2} - 2.b.\frac{\partial^2 x}{\partial \xi \partial \eta} + c.\frac{\partial^2 x}{\partial \eta^2} = 0 \qquad (6)$$

$$a.\frac{\partial^2 y}{\partial \xi^2} - 2.b.\frac{\partial^2 y}{\partial \xi \partial \eta} + c.\frac{\partial^2 y}{\partial \eta^2} = 0 \qquad (7)$$

Where,

$$a = (\partial x/\partial \eta)^2 + (\partial y/\partial \eta)^2 \qquad (8)$$

$$b = \frac{\partial x}{\partial \xi}.\frac{\partial x}{\partial \eta} + \frac{\partial y}{\partial \xi}.\frac{\partial y}{\partial \eta} \qquad (9)$$

$$c = (\partial x/\partial \xi)^2 + (\partial y/\partial \xi)^2 \qquad (10)$$

The previously stated equations must all be expressed in terms of finite differences. Once this is done, x and y at each grid point can be found through iterations. Expanding Equation (8) through (10) explicitly in central space yields:

$$a = \left[\frac{x_{i,j+1}^{n} - x_{i,j-1}^{n+1}}{2.\Delta\eta}\right]^2 + \left[\frac{y_{i,j+1}^{n} - y_{i,j-1}^{n+1}}{2.\Delta\eta}\right]^2 \qquad (11)$$

$$b = \left[\frac{x_{i+1,j}^{n} - x_{i-1,j}^{n+1}}{2.\Delta\xi}\right].\left[\frac{x_{i,j+1}^{n} - x_{i,j-1}^{n+1}}{2.\Delta\eta}\right] + \left[\frac{y_{i+1,j}^{n} - y_{i-1,j}^{n+1}}{2.\Delta\xi}\right].\left[\frac{y_{i,j+1}^{n} - y_{i,j-1}^{n+1}}{2.\Delta\eta}\right] \qquad (12)$$

$$c = \left[\frac{x_{i+1,j}^{n} - x_{i-1,j}^{n+1}}{2.\Delta\xi}\right]^2 + \left[\frac{y_{i,j+1}^{n} - y_{i,j-1}^{n+1}}{2.\Delta\xi}\right]^2 \qquad (13)$$

Here, the superscript, n, indexes the iteration, where n is the current iteration and n+1 is the following iteration. These equations are written this way due to the fact that points above and to the right of the point being evaluated are unknown, and, thus, old values must be used. The same procedure of finite differencing can be applied to Eqs. (6) and (7). However, results from these will be of the same form; that is, only the terms x and y will be different. Considering the expansion of Eq. (6) yields:

$$a.\left[\frac{x_{i+1,j}^{n} - 2.x_{i,j}^{n+1} + x_{i-1,j}^{n+1}}{(\Delta\xi)^2}\right] - 2.b\left[\frac{x_{i+1,j+1}^{n} - x_{i+1,j-1}^{n} - x_{i-1,j+1}^{n} - x_{i-1,j-1}^{n+1}}{4.\Delta\xi.\Delta\eta}\right] + c.\left[\frac{x_{i,j+1}^{n} - 2.x_{i,j}^{n+1} + x_{i,j-1}^{n+1}}{\Delta\eta^2}\right] = 0 \qquad (14)$$

Considering,

$$\alpha = \frac{a}{(\Delta\xi)^2};\ \beta = \frac{b}{2.\Delta\xi.\Delta\eta};\ \gamma = \frac{c}{\Delta\eta^2}$$

This equation can then be explicitly solved for the value $x_{i,j}^{n+1}$ which is the coordinate of interest. Doing so yields:

$$x_{i,j}^{n+1} = \frac{\alpha.(x_{i+1,j}^{n} + x_{i-1,j}^{n+1}) + \beta.(x_{i+1,j+1}^{n} - x_{i+1,j-1}^{n} - x_{i-1,j+1}^{n} - x_{i-1,j-1}^{n+1}) + \gamma.(x_{i,j+1}^{n} + x_{i,j-1}^{n+1})}{2.(\alpha+\gamma)}$$

Similarly, Considering the expansion of Eq.(7) and solving it for value of $y_{i,j}^{n+1}$:

$$y_{i,j}^{n+1} = \frac{\alpha.(y_{i,j+1}^{n} + y_{i,j-1}^{n+1}) + \beta.(y_{i+1,j+1}^{n} - y_{i+1,j-1}^{n} - y_{i-1,j+1}^{n} - y_{i-1,j-1}^{n+1}) + \gamma.(y_{i+1,j}^{n} + y_{i-1,j}^{n+1})}{2.(\alpha+\gamma)}$$

This formula can then be implemented through coding in order to find all values of x. The formulation is exactly the same for the y value. Through code, multiple iterations will occur until convergence is reached; that is, the desired x values will be found once the difference between $x_{i,j}^{n+1}$ and $x_{i,j}^{n}$ is below tolerance and the desired y values will be found once difference between $y_{i,j}^{n}$ and $y_{i,j}^{n}$ falls below said tolerance. These values, when plotted, should produce an elliptic grid that can be utilized to determine flow within the domain containing the airfoil.

MATLAB code for Elliptic Grid Generation

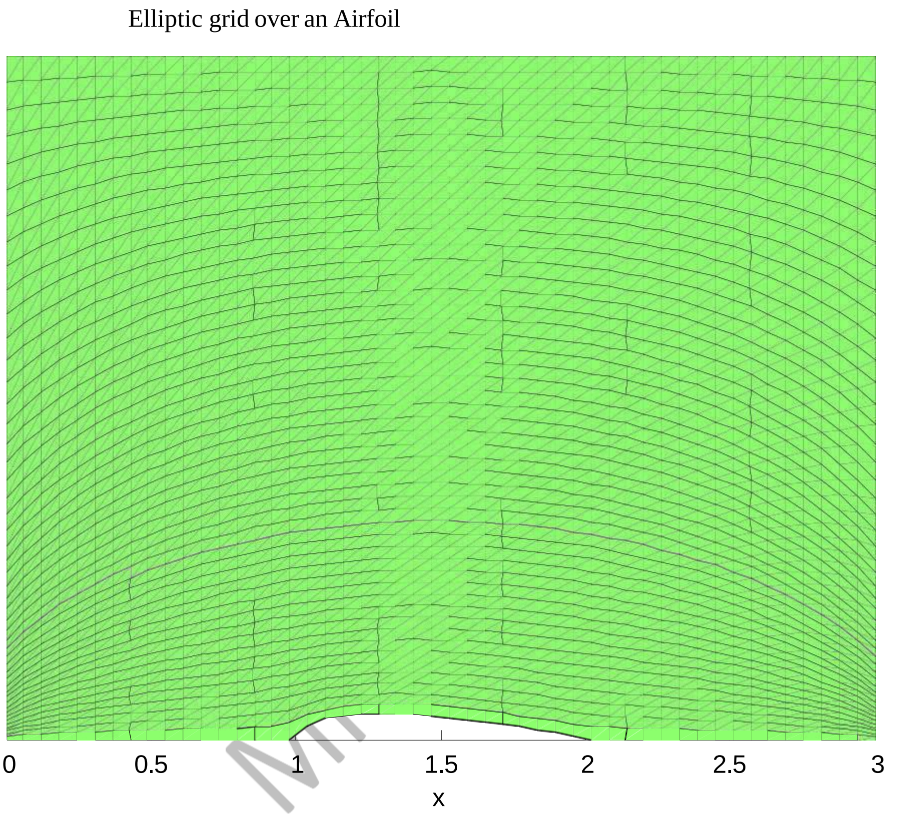
```
%Elliptic Grid Generation
Clear all;
clc;
%Assign values for t and beta
t=0.15;
beta=1.05;
%Prompt user for number of grid points
n=input('Enter the number of grid points in the i direction: ');
m=input('Enter the number of grid points in the j direction: ');
%Create zeroes matrix for surface
plots z=zeros(n,m);
%Assign lengths and values for eta and xi
L=3;
eta=linspace(0,1,m);
xi=linspace(0,L,n);
%x is equal to xi
X=xi;
%Find height
ytop=2;
for  i=1:n
if X(i) < 1
ybottom(i)=0;
elseif X(i) > 2
ybottom(i)=0;
else
x2(i)=X(i)-1;
 ybottom(i)=(t/.2)*(0.2969*x2(i)^.5-0.126*x2(i)-0.3516*x2(i)^2+0.2843*x2(i)^3-0.1015*x2(i)^4);
end
 H(i)=ytop-ybottom(i);
 end
%Loop to calculate coordinates
zeta=beta+1;
gamma=beta-1;
alpha=zeta/gamma;
for i=1:n
for
j=1:m
chi=1-
eta(j);
y(i,j)=H(i)*(zeta-gamma*alpha^chi)/(alpha^chi+1)+ybottom(i);
x(i,j)=X(i);
end
end

%Elliptic initial conditions
xold=x;
yold=y;
```

```
%Calculate computational step sizes
delta_eta=1/(m-1);
delta_xi=L/(n-1);
dx=1; %Conditions to start loop
dy=1; %Conditions to start loop
%Assign tolerance value
tol=.0001;
%Nested loop to determine elliptic grid
xdiff=0;
ydiff=0;
count=0;
while dy > tol || dx > tol
for i=2:n-1
for j=2:m-1
a1=(xold(i,j+1)-x(i,j-1))/(2*delta_eta);
a2=(yold(i,j+1)-y(i,j-1))/(2*delta_eta); a=a1^2+a2^2;
c1=(xold(i+1,j)-x(i-1,j))/(2*delta_xi);
c2=(yold(i+1,j)-y(i-1,j))/(2*delta_xi); c=c1^2+c2^2;
b=a1*c1+a2*c2;
alpha=a/delta_xi^2;
beta=-2*b/(4*delta_xi*delta_eta); gamma=c/delta_eta^2; theta=1/(2*alpha+2*gamma);
phi_1=beta*(xold(i+1,j+1)-xold(i+1,j-1)-xold(i-1,j+1)+x(i-1,j-1));
x(i,j)=theta*(alpha*(xold(i+1,j)+x(i-1,j))+gamma*(xold(i,j+1)+x(i,j-1))+phi_1);
xdiff=x(i,j)-xold(i,j)+xdiff;
phi_2=beta*(yold(i+1,j+1)-yold(i+1,j-1)-yold(i-1,j+1)+y(i-1,j-1));
y(i,j)=theta*(alpha*(yold(i+1,j)+y(i-1,j))+gamma*(yold(i,j+1)+y(i,j-1))+phi_2);
  ydiff=y(i,j)-yold(i,j)+ydiff;
  end
end
dx=xdiff; dy=ydiff; xdiff=0;
ydiff=0;
xold=x;
yold=y;
count=c
ount+1;
end
fprintf('The solution took %i iterations to converge. \n \n', count);
surface(x,y,z);
xlabel ('x');
ylabel ('y');
title ('Elliptic grid over an Airfoil');
```

Result and Discussion

In the plot of an Elliptical, the Grid lines have been smoothed out due to the elliptic equations, eliminating extreme jaggedness resulting from the algebraic grid. This would ensure a more accurate flow model.

Enter the number of grid points in the i direction: 50
Enter the number of grid points in the j direction: 50
The solution took 2434 iterations to converge.



Overall, an elliptic grid was shown to provide desired results for discretization. It succeeded in smoothing out otherwise rough edges created through algebraic grid generation. At the same time, the algebraic grid provided a suitable starting point for the generation of the elliptic grid.

VIVA QUESTIONS:

1. Define CFD?
2. What are the three major steps of CFD?
3. What are the governing equations of CFD?
4. What is meant by Discretization?
5. Which type of Discretization is used in CFD?
6. Difference between forward and backward differencing scheme?
7. What is Explicit method and Implicit method?
8. What is LAX method?
9. What is a stability criterion?
10. What is thermal diffusivity?
11. Define Grid?
12. Difference between Structured and Unstructured grid?
13. What is meant by Grid Independence study?
14. What is linspace command in MATLAB?
15. How to give titles to X and Y axis of a graph?
16. How to create Hybrid mesh in ICEM?
17. How to create structured grid in ICEM?
18. What is the importance of Body point in ICEM?
19. How to define material properties in CFX or FLUENT?
20. What is meant by convergence criteria?
21. How to define supersonic inlet conditions in CFX?
22. What is Grid adaption technique?
23. What is meant by parallel and serial processing?
24. In how many ways CFD results can be presented?
25. How to define formulas in CFD Post?
26. What are the causes for reverse flow or diverged flow during CFD iterations?
27. What are the relaxations factors in FLUENT?
28. What is courant number and how does it affects the solution?
29. What are the different types of turbulence models in CFD?
30. Difference between free slip and no-slip conditions?