

Introduction to ANSYS Workbench



Suhail Mahmud Mohamad Wissam



Abstract

With the emerging importance of CFD and finite element analyses, it is of great necessity that engineering students get a good base of knowledge on one of the most used software packages in the industry of simulation, ANSYS. This brief tutorial states a few simple examples of the main applications of the software package ANSYS and highlights some of the possible problems students may face during their journey in discovering this application.

The flow of information is structured that the reader gets an understanding of how important ANSYS is, and how it works and what type of machines are needed for the student level research expected. Then the tutorial goes on with simple straight forward examples of structural and fluid physics simulated using the ANSYS package. Eventually, the tutorial addresses the most important problems generally faced by the students such as unsuccessful meshing, or divergent solutions.

Disclaimer

It is extremely important to note two points while following this tutorial:

- The knowledge contained in this paper is by no means, accepted as mainstream, or an industry best practice. It is merely the product of the experience of senior engineering students who explored the program and desired to share their experience with the package.
- The choices and configurations in every example given are not to be considered as a one size fits all – template. As the student grows in experience they are expected to try other configurations, commit to trial and error procedures, and develop their own troubleshooting skills in order to create working models.

Table of Contents

Abs	stract		2
Dis	claimer		2
1.	Introduc	tion	5
2.	Exercises	5	8
	2.1.	. Static Structural – Cantilever Beam	8
	2.1.1.	Problems Specifications:	8
	2.1.2.	Starting and assigning material properties	9
	2.1.3.	Geometry	. 11
	2.1.4.	Model	. 12
	2.1.5.	Setup	. 13
	2.2.	. Fluent – 2D - Airfoil	. 16
	2.2.1.	Methodology - Air domain and Boundary	. 16
	2.2.2.	Geometry	. 17
	2.2.3.	Mesh	. 19
	2.2.4.	Setup	. 21
	2.2.5.	Changing the Angle of attack	. 27
	2.3.	. Fluent – 3D - Finite Wing	. 34
	2.3.1.	Geometry	. 34
	2.3.2.	Mesh	. 39
	2.3.3.	Setup	. 42
	2.3.4.	CFD Post	. 47
	2.3.5.	Tecplot	. 52
	2.4.	. Fluent – Internal flow through pipes and ducts	. 57
	2.4.1.	Geometry	. 57
	2.4.2.	Mesh	. 60
	2.4.3.	Setup	. 62
3.	Commor	n Problems	. 67
	3.1.	. Autodesk Autocad compatibility with Ansys	. 67
	3.2.	. The sharp trailing edges of the airfoils	. 67
	3.3.	. General Meshing Problems	. 68

Ansys Workbench Basics Guide

	3.4.	Named Selection Process
	3.5.	Solution Divergence
	3.6.	Temperature solution divergence while using Energy equation
	3.7.	Scaling
	3.8.	Huge values of lift and drag72
4.	Recommen	ded Topics72
	4.1.	Dynamic and Sliding mesh72
	4.2.	Meshing techniques – Gambit72
	4.3.	Fluent Models73
	4.4.	Combining the structural loads with the aerodynamic loads73
	4.5.	Cables73
	4.6.	Composite
5.	Useful Link	s74

1. Introduction

ANSYS is a finite element analysis package used widely in industry to simulate the response of a physical system to structural loading, and thermal and electromagnetic effects. ANSYS uses the finite-element method to solve the underlying governing equations and the associated problem-specific boundary conditions.

File View Tools Units Help	
🖹 New 😂 Open 🗟 Save 🗟 Save As 🕼 Import 🗧 Reconnect 🚔 Refresh Project 🍼 Update Project 🎧 Project 🎧 Compact Mode	
	. П. У
La Analysis Systems	
Design Assessment	
Explicit Dynamics	
Fluid Flow - Biowronaing (POLTFLOW)	
E Huid How (FLUENI)	
Harmonic Kesponse	
Pydrodynamic Diffradon	
Pydrodynamic lime kesponse	
magnetostad	
Contraction of the second seco	
Jeedy Jake Hennik	
Transient Structural	
Transient Thermal	
Component Systems Progress	▼ 7 X
A AUTODYN B	С
BladeGen 1 Status Detaile	Progress
CFX CFX	
🖉 Engineering Data 🔹	
View All / Customize	
🔋 Ready 💷 Hide Progress 📮	Show 4 Messages

This manual includes the procedure of solving the (static structural, Fluent) problems.

Each one of the analysis systems has its own procedure. However, there are some common stages in all of the systems.

1 Unsaved Project - Workbench	these here here here	
File View Tools Units Help		
New 😂 Open 🗏 Save 🐼 Save As	Compact And Reference And Reference Project And Compact Mode	
		- 0 Y
Pasian Assessment		
Electric		▼ D
Explicit Dynamics	1 🐷 Static Structural 1 🐷 Binid Dynamics 1 🔟 Magnetostatic	1 R Eluid Flow (FLUENT)
R Fluid Flow-BlowMolding (POLYFLOW)		
Fluid Flow - Extrusion (POLYFLOW)		
Fluid Flow (CFX)	3 Geometry F 3 Geometry F 3 Geometry F 3	3 Wesh Y
Fluid Flow (FLUENT)	4 💓 Model 😨 🖌 4 💓 Model 😨 🖌	4 💓 Setup 😨 🖌
S Fluid Flow (POLYFLOW)	5 🙀 Setup 😨 🖌 5 🙀 Setup 😨 🖌	5 📢 Solution 🔗 🖌
🔁 Harmonic Response	6 🗌 Solution 😨 🖌 6 🗌 Solution 😨 🖌 6 🗌 Solution 😨 🖌	6 😥 Results 🛛 😨 🖌
Mydrodynamic Diffraction	7 🐼 Results 👕 7 🐼 Results 👕 7	Fluid Flow (FLUENT)
Hydrodynamic Time Response	Static Structural Rigid Dynamics Magnetostatic	
Linear Buckling	Sector Regio Sharing Progratistic	
Magnetostatic		
Random Vibration		
Response Spectrum		
El Shape Optimization		
Static Structural		
Steady-State Thermal		
Thermal-Electric		
Transient Structural		
Transient Thermal	٠ (III) III / IIII / III / IIII / III /	
Component Systems	Progress	→ 中 ×
AUTODYN	AB	с
BladeGen	1 Status Details	Progress
CFX CFX		rieg.cov
🥏 Engineering Data 🔻		
View All / Customize		
🖁 Ready		Hide Progress Show 4 Messages

For each type of problems, the procedure can be completed by going through the tree one by one until all the cells get marked with \checkmark .

It is highly recommended to surf online and have a good idea about the "mesh" or the "grid"

- The importance of the mesh for the computer-aided engineering and simulation software like ANSYS.
- Types of mesh
- How to control the mesh size and based on what the mesh should be modified
- How does mesh size affect the quality and reliability of the results?

Moreover, it is recommended to use a pc with minimum specifications of:

- Processor: i5 or i7
- Ram: 32 Gbs
- Hard disk: 1 TB
- Good cooling system (Important)

The geometry should be made on external modeling software (Solidworks, Catia or Rhino) and saved in an individual geometry file with recommended extensions (Solid part file .sldprt, IGES file .igs or Step file .stl). Autodesk Autocad is **not** compatible with Ansys.

NOTE: This manual provides a very brief idea and introduction the Ansys applications. The manual is made for the beginners who are working on the application for the first time. It should guide the student to the basics of Ansys while he can develop himself with more advanced problems from real life and from online sources.

2. Exercises

2.1. Static Structural – Cantilever Beam



2.1.1. Problems Specifications:

Find the stress and the strain in the cantilever beam where:

- L = 1 m
- H = 0.2 m
- Load = 1 KN Downwards, applied on the top right edge.
- The material properties are: Young's modulus E = 200 GPa Poisson ratio = 0.3

2.1.2. Starting and assigning material properties

** Before starting, the geometry file of the beam should be saved in an individual file



Drag (Static Structural) to the Project Schematic inside the red square

** Double Click on (Engineering Data) to configure and add the materials that would be used in the analysis along with their properties.

** The shown window will appear where a new material can be added >> (click here to add a new material)>> add (material for the beam)



File View Tools Units Help										
🎦 New 对 Open 🛃 Save 📓 Save As	. 👔 Imp	oort 🖗 Reconnect 义	Refresh Project	🗲 Up	date F	Project	CReturn to Pr	oject 🕜 Cor	npact M	ode 🛛 🍸 🎒
oolbox 🝷 🕈 🗙	Outline	of Schematic A2: Engineering	Data					•	Ψ×	Table: No data
Physical Properties		A		в	С		D			
🔁 Density	1	Contents of Engineeri	ng Data 🛛 📮	8	s		Descript	ion		
Isotropic Secant Coefficient of Thermal Eq	2	Material								
Orthotropic Secant Coefficient of Thermal			-1		æ	Fatigue	e Data at zero me	an stress com	es	
Orthotropic Instantaneous Coefficient of 1	3	Structural Ste	ei		=	2, Tabl	998 ASME BPV Co e 5-110.1	de, Section 8,		
Constant Damping Coefficient		Click here to add a r	new material							
🔁 Damping Factor (β)										
Linear Elastic										
Isotropic Elastidty										
Orthotropic Elastidty										
Anisotropic Elastidty										
Hyperelastic										
	Properti	ies of Outline Row 4:						•	Ψ×	Chart: No data
Creep			A				В	С		
∃ Life	1	Property					Value	Unit		
I Strength										
_ brengen										
Z Tensile Yield Strength										
Tensile Yield Strength Compressive Yield Strength										

** In the Toolbox, the material properties can be added from "Density" or "Isotropic Elasticity". Double Clicking on the mentioned options will open new fields in the outline where the fields have to be filled with the values of the properties.

Note: Try to find the desired material in the "Engineering Data Source" Library before adding a new material. Click on the icon Select the type of the material and the materials will appear in a list. If you want to add a material to your project list, click on

** After you are done with adding all the materials needed in the project, click on "Return to Project"

** The Engineering Data field should be marked with a indicating that the process of adding materials properties has been done.







2.1.3. Geometry

** Right Click on (Geometry) >> Import Geometry >> Browse >> Locate the geometry file

Note: Simple geometry can be constructed in Ansys Geometry window itself. However, complex geometry should be imported from 3D modeling software like Solidworks, as it has been done in this exercise.

** Even though after locating the geometry file, the field will be marked with , it is still necessary to do the following step.

** Double click on "Geometry" >> Chose the units used while constructing the geometry files

** On the Tree Outline on the left side >> Right Click on "Import" >> Generate. Hence, the geometry will appear in the graphics window. After this step, close the geometry window.



ANSYS Workbench	X	
Select desired length unit	:	
Meter	C Foot	
C Centimeter	C Inch	
C Millimeter		
C Micrometer		
Always use project of Always use selected Always use selected Enable large model s	unit d unit upport	
ОК		



2.1.4. Model

** Double click on "Model"

** On the outline window, expand the "Geometry" tree by clicking on "+", this tree should show you all the parts in the project (will be clear when there are multiple parts in the project). Moreover, the tree helps in assigning different material to different parts or managing the contact type between two parts (Frictional, Frictionless, etc).

** On the outline window, click on "Mesh". For generating the mesh with the default size, click on Update from the top bars. For advanced mesh options, adjust the settings from "Details of Mesh" window.

Note: The default mesh is usually a very basic grid with no attention given to the details of the geometry. Advanced mesh details can be added by choosing the geometrical detail and inserting "sizing" as it is shown in the figure. The details can be chosen using the selecting icons.





2.1.5. Setup

** After setting the material and generating the mesh, close the "Model" window. As it is clear, the first 3 stages have been marked with ✓ indicating that they are completed. Move to "Setup".

** In "Setup", the loads, the supports and the desired solution parameters should be defined. By marking the location on the geometry and adding a force or a support, the "Setup" stage can be considered to be done.

** Choose the face where the cantilever beam is fixed by using the "Face selection tool" .

** Add the "Fixed Support" from the "Supports List". Hence, on the "Outline" tree, the fixed support will be displayed under the "Static Structural" list.







** Similarly, select the top right edge of the beam using the "Edge Selecting Tool" 💽.

** Add the force from the "Loads" list. In the "Details of Force" window, change "Defined By" to "Components" and then set the "Y" direction force to be " - 1000 N" as it is shown in the figure.

Note: The negative sign of the force is because the force is downwards. Always make sure you check the coordinate system defaults directions before setting the forces.

** From the side view, the "Graphics window" should look like this after clicking on "Static Structural" on the "Outline" window.



De	Details of "Force"							
Ξ	Scope							
	Scoping Method	Geometry Selection						
	Geometry	1 Edge						
	Definition							
	Туре	Force						
	Define By	Components						
	Coordinate System	Global Coordinate System						
	X Component	0. N (ramped)						
	🔲 Y Component	-1000						
	Z Component	0. N (ramped)						
	Suppressed	No						
I '								



Ansys Workbench Basics Guide

** To define the desired solution parameters, click on "Solutions" and define all the parameters needed to be found. The parameters can be chosen from the lists shown in the figure.



** After defining the investigation parameters, click Solve to get the results. To show the results of the different parameters, use the list under "solutions" in the "Outline" window.

Note: The previous procedure can be considered one of the simplest static structural problems. Practice more by finding solved problems online and comparing your results to the given results.



2.2. Fluent – 2D - Airfoil

2.2.1. Methodology - Air domain and Boundary

In aerospace applications, fluent is usually used to calculate the lift and the drag, present the pressure distribution, vorticity, velocity vectors, streamlines.. etc.

Since computer resources management is a critical issue, the easiest and the least resource extensive method is mentioned in the manual where the properties are calculated using only one material (air) without going through the details of the wing material or the internal structure of the wing.

Hence, a boundary of air has to be defined where it covers the wing while the gap in the material of the boundary (air) is representing the wing. In other words, the wing has to be subtracted from the air boundary leaving the air moving inside the boundary avoiding the gap. The next figure is showing the air boundary and the subtracted airfoil.



For the 2D cases, the air domain and the airfoil subtraction should be done from the modeling software. In the 3D cases, the wing has to be constructed in the 3D modeling software while the domain construction and the subtraction process should be done in Ansys workbench.

Generally, the inlet should be away from the leading edge with a distance equal to twice of the airfoil chord length while the outlet should be 8 - 10 times the chord length. Moreover, the top and the bottom of the boundary should be 4 - 6 times of the chord length away from the airfoil

2.2.2. Geometry

** The geometry file should be saved in an individual file

** In ANSYS Workbench window:

Drag (Fluid Flow (Fluent)) to the Project Schematic inside the red square

** Right Click on (Geometry) >> Import Geometry >> Browse >> Locate the geometry file







Suhail Mahmud and Mohamad Wissam

** Chose the units used while constructing the geometry files

** On the Tree Outline on the left side >> Right Click on "Import" >> Generate

ANSYS Workbench		×
Select desired length unit:		
Meter	C Foot	
C Centimeter	C Inch	
C Millimeter		
C Micrometer		
Always use project uni Always use selected un Enable large model supp	t nit port	
ОК		



** After the geometry appears, close the geometry modeling window 2.2.3. Mesh

** Double click on "Model"

** To generate the mesh, click

Note: The default mesh is usually a very basic grid with no attention given to the details of the geometry. Advanced mesh details can be added as it is explained bellow.

** On the Outline part, Left click on "Mesh". Then on the "Details of Mesh" window Change the followings:

Relevance>> controls the density of the mesh in regions closer to the geometry.

- Use advanced size function >> On Proximity and Curvature

- Relevance Center >> Fine

- *Min Size>> the minimum size of the mesh elements in meters*

- Max face Size>> the maximum size of the mesh elements in meters

- Max Size >> equal to "Max face Size"

•	A	
1	S Fluid Flow (FLUENT)	
2	🕅 Geometry	× .
3	🎯 Mesh	2
4	🍓 Setup	P .
5	Solution	7
6	🥩 Results	? 🖌
	Fluid Flow (FLUENT)	



De	etails of "Mesh"	4					
	Defaults						
	Physics Preference	CFD					
	Solver Preference	Fluent					
	Relevance	0					
Ξ	Sizing						
		On: Proximity and Curvature 🔫					
	Relevance Center	Fine					
	Initial Size Seed	Active Assembly					
	Smoothing	Medium					
	Transition	Slow					
	Span Angle Center	Fine					
	Curvature Normal Angle	Default (18.0 °)					
	Proximity Accuracy	0.5					
	Num Cells Across Gap	Default (3)					
	Min Size	8.e-004 m					
	Max Face Size	4.0 m					
	Max Size	4.0 m					
	Growth Rate	Default (1.20)					
	Minimum Edge Length	1.6276e-004 m					
÷	Inflation						
Ξ	CutCellMeshing						
	Active	No					
+	Advanced						
Ξ	Defeaturing						
	Pinch Tolerance	Default (7.2e-004 m)					
	Generate Pinch on Refresh	No					
	Automatic Mesh Based Defeaturing	Off					
+	Statistics						

** After the mesh is generated. Choose the edge choosing tool.

** Left click on each edge of the boundary>>Right click >> Create Named Selection >> Name each edge according the orientation of the model. Make sure that the inlet is named "inlet", the outlet is named "outlet", and the other 2 sides' names start with "symmetry – (add name)".

** After selecting each edge separately and assigning a named selection, change the selection type to "Box selection" as shown.

** Hence, select the airfoil as it is shown. Then right click>> create named selection >> call it any name (avoid calling it Inlet, Outlet and Symmetry), in this example it is called "airfoil for easier reference.

** After doing the named selection step, the tree outline should look like the shown figure. Notice all the named selections are listed.



Insert Go To Jometric View Set Set Set Cursor Mode View View N Select All Select All Se			Ľ
Insert Go To Isometric View Set Quarter Mode View 1.500 3.000 (m) 1.500 3.000 (m) 1.500 3.000 (m) 1.500 3.000 (m) 1.500 3.000 (m) Create Coordinate System Create Named Selection Create Named Selection			
Insert Go To Sometric View Sometric View Sometr			
Insert Go To Go To Sometric View S Set S S			
Sometric View Set Restore Default © Zoom To Fit Cursor Mode View 1.500 3.000 (m) 1.500 3.000 (m) 1.	Go To	-	
Set Set Set Set Set Set Set Set Set Set Set Set Set Set	🔵 Isometric View		
• Action De Defailt • Q Zoom To Fit Cursor Mode View • 1.500 3.000 (m) ① Z 250 □ □ Z 250 □ □ Z 250 □ □ □	ISO Set		
Cursor Mode X View 1.500 I Look At 0.750 Q Select All Image: Select All <td>Coom To Fit</td> <td></td> <td>l</td>	Coom To Fit		l
View 1.500 3.000 (m) Int Pr Select All Image: Suppress Body Image: Provide Body Image: Suppress Body	Cursor Mode	•	
Int Pr 0.750 2.250 Int Pr Suppress Body	View	1.500 3.000 (m)	ſ
nt Pr	R Look At	0.750 2.250	
Image: Suppress body Image: Suppress body Image: Suppress body Association Image: Suppress body Association	nt Pr		-
Association Create Coordinate System Create Named Selection	Hide Body		
Create Named Selection		Association	
	Create Named Selection		
1¢1 Refresh Geometry	Refresh Geometry		

File	Edit	View	Units	Tools	Help	
line R	X,Y,Z	₽3 ₹	R (1		é
🖵 Show 📘 Single Select						
Mesh 🔰 🔩 Box Select					Ø, N	1e





Ansys Workbench Basics Guide

2.2.4. <u>Setup</u>

** Close the "Mechanical Window" >> Right click on "Mesh" >> Update.



•		А	
1		Fluid Flow (FLUENT)	
2	œ	Geometry	× .
3	۲	Mesh	× .
4	3	Setup	R •
5		Solution	? ,
6	6	Results	2 🖌
		Fluid Flow (FLUENT)	

○ 2D ⊚ 3D	Double Precision
⊚ 3D	Use Job Scheduler
Display Options	📄 Use Remote Linux Nodes
📝 Display Mesh After Reading	Processing Options
📝 Embed Graphics Windows	🔘 Serial
📝 Workbench Color Scheme	Parallel (Local Machine)
Do not show this panel again	Number of Processes
手 Show More Options	

** Double Click on "Setup"

** Tick (Double Precision)>> Chose "Parallel" and chose the number of processors to be 4 unless if more processors are licensed. In the case your computer has less than 4 processors, select the maximum amount of processors available. ** Chose the "Type" to be:

- "Pressure Based" for incompressible flow

- "Density Based" for compressible flow

** Go to "Define">> Operating Conditions.

** Define the Static Pressure in the operation altitude.

** In "Models" Section >> Double click on "Viscous" and chose:

- Model: K-epsilon
- K-epsilon model: Realizable

- Near-Wall Treatment: Non-Equilibrium Wall Functions



** In "Materials" Section >> Double Click on "air" >> set the density and the viscosity Pressure in the operation altitude.

** In "Boundary Conditions" Section >> Double Click on "Inlet" >> Change "Velocity Specification Method" to "Components" >> Insert the values of the flow velocity with respect to the coordinate system (Notice it is -50 because the free stream is in the negative X direction.

** In "Reference Values" section >> Chose "Compute from" to be "inlet" >> Insert the flow conditions at the operating altitude. Moreover, insert:

- Area: the reference area of the wing (the projection area from the top view)

Depth: the span of the 2D wing

- Length: Mean Aerodynamic Chord length

Suhail Mahmud and Mohamad Wissam





roblem Setup	Reference Values	
General Models	Compute from inlet	•
Phases	Reference Values	
Cell Zone Conditions Boundary Conditions	Area (m2)	0.05
Mesh Interfaces Dynamic Mesh	Density (kg/m3)	1.225
Reference Values colution	Depth (m)	0.5
Solution Methods Solution Controls	Enthalpy (j/kg)	0
Monitors Solution Initialization	Length (m)	0.2
Calculation Activities Run Calculation	Pressure (pascal)	0
esults Graphics and Animations	Temperature (k)	288.16
Plots Reports	Velocity (m/s)	50
	Viscosity (kg/m-s)	1.7894e-05
	Ratio of Specific Heats	1.4

** In "Solution Methods" Section >> Chose "Scheme" to be "Coupled".

** In "Solution Controls" Section >> Click on "Limits" >> set the "Maximum Turb. Viscosity Ratio" to be 1e+20.

** In "Monitors" section >> Double click on "Residuals" >> Tick on (Print, Plot) >> on the right side, remove the ticks from all the parameters except continuity. Moreover, change the absolute criteria of the continuity to be 1e-6 as shown in the figure.

Problem Setup	Solution Methods		
General	Pressure-Velocity Coupling		
Models	Scheme		
Materials	Scheme		
Phases	Coupled		
Boundary Conditions	Spatial Discretization		
Mesh Interfaces	Cradient		
Dynamic Mesh	Gradient		
Reference Values	Deserves		
Solution	Pressure		
Solution Methods	Standard		
Solution Controls	Momentum		
Monitors	First Order Upwind		
Solution Initialization	Turbulent Kinetic Energy		
Calculation Activities	First Order Upwind		
Run Calculation	Turbulent Dissipation Rate		
Results	First Order Upwind		
Graphics and Animations	Transient Formulation		
Pious			
Problem Setup	Solution Controls		
Problem Setup General	Solution Controls		
Problem Setup General Models	Solution Controls Courant Number		
Problem Setup General Models Materials	Solution Controls Courant Number 200		
Problem Setup General Models Materials Phases Cell Zone Conditions	Solution Controls Courant Number 200 Explicit Relaxation Factors		
Problem Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions	Solution Controls Courant Number 200 Explicit Relaxation Factors Momentum 0.75		
Problem Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces	Solution Controls Courant Number 200 Explicit Relaxation Factors Momentum 0.75		
Problem Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Defenses Values	Solution Controls Courant Number 200 Explicit Relaxation Factors Momentum 0.75 Pressure 0.75		
Problem Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution	Solution Controls Courant Number 200 Explicit Relaxation Factors Momentum 0.75 Pressure 0.75		
Problem Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Methods	Solution Controls Courant Number 200 Explicit Relaxation Factors Momentum 0.75 Pressure 0.75 Under-Relaxation Factors		
Problem Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Methods Solution Controls	Solution Controls Courant Number 200 Explicit Relaxation Factors Momentum 0.75 Pressure 0.75 Under Relaxation Factors Solution Limits		
Problem Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Methods Solution Controls Monitors	Solution Controls Courant Number 200 Explicit Relaxation Factors Momentum 0.75 Pressure 0.75 Under-Relaxation Factors Solution Limits Minimum Absolute Pressure (pascal)		
Problem Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Methods Solution Controls Monitors Solution Tribalization Calculation Activities	Solution Controls Courant Number 200 Explicit Relaxation Factors Momentum 0.75 Pressure 0.75 Under-Relaxation Factors Solution Limits Minimum Absolute Pressure (pascal) 1		
Problem Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Methods Solution Methods Solution Controls Monitors Solution Controls Monitors Solution Activities Run Calculation	Solution Controls Courant Number 200 Explicit Relaxation Factors Momentum 0,75 Pressure 0,75 Under-Relaxation Factors Solution Limits Minimum Absolute Pressure (pascal) 1 Maximum Absolute Pressure (pascal) 5e+10		
Problem Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Methods Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results	Solution Controls Courant Number 200 Explicit Relaxation Factors Momentum 0.75 Pressure 0.75 Under-Relaxation Factors Solution Limits Minimum Absolute Pressure (pascal) 1 Maximum Absolute Pressure (pascal) 5e+10 Minimum Turb, Kinatic Energy (m2/c2)		
Problem Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Methods Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations.	Solution Controls Courant Number 200 Explicit Relaxation Factors Momentum 0.75 Pressure 0.75 Under-Relaxation Factors Under-Relaxation Factors Solution Limits Minimum Absolute Pressure (pascal) 1 Maximum Absolute Pressure (pascal) 5e+10 Minimum Turb. Kinetic Energy (m2/s2) 1e-14		
Problem Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Methods Solution Methods Solution Centrols Monitors Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots	Solution Controls Courant Number 200 Explicit Relaxation Factors Momentum 0.75 Pressure 0.75 Under-Relaxation Factors Under-Relaxation Factors Solution Limits Minimum Absolute Pressure (pascal) 1 Maximum Absolute Pressure (pascal) 5e+10 Minimum Turb. Kinetic Energy (m2/s2) 1e-14 Minimum Turb. Dissipation Rate (m2/s3) 1e-20		
Problem Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Methods Solution Methods Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	Solution Controls Courant Number 200 Explicit Relaxation Factors Momentum 0.75 Pressure 0.75 Under-Relaxation Factors Solution Limits Minimum Absolute Pressure (pascal) 1 Maximum Absolute Pressure (pascal) 5e+10 Minimum Turb. Kinetic Energy (m2/s2) 1e-14 Minimum Turb. Dissipation Rate (m2/s3) 1e-20		



Default

OK Default Cancel Help

Equations... Limits... Advanced...

** In "Monitors" section >> Double click on "Drag" >> Tick on (Print to console, Plot, Write) >> add (.txt) to the end of the file name >> Adjust the unit vector which is representing the direction of the Drag force with respect to the coordinate system (Notice it is -1 in the X direction because the free stream is in the negative X direction).

** Do the same process for "Lift" keeping in mind that the X and Y force vectors will be different.

** In "Solution Initialization" section >> Chose "Hybrid Initialization".

** In "Run Calculations" Section >> Set the required number of iterations and "Calculate".

Phases	Drag - Print, Plot, Write			
Cell Zone Conditions Boundary Conditions	Drag Monitor			
Mesh Interfaces Dynamic Mesh Reference Values	Options	Wall Zones		
olution	V Print to Console V Plot	airtoil		
Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation esults	Window 2 Curves Axes Write File Name cd-history.txt			
Graphics and Animations Plots Reports	Per Zone Force Vector			
	X Y Z -1 0 0			





** The solution will complete when the convergence (error) reaches to the pre-defined limit. The final C_1 and C_d values are the ones in the last line.

** To view the graphical results, In "Results" chose "Graphics and Animations" >> Double click on "Contours" or "Vectors" >> Chose the required specifications of the figure from "Options" >> Display.

** More results can be displayed using CFD Post and Tecplot as it will be demonstrated in the 3D section.





2.2.5. Changing the Angle of attack

In aerospace applications, the angle of attack is an important parameter where the tests usually include a study of the lift and the drag under different angles of attack.

There are 2 basic methods of changing the angle of attack where one of them is more accurate and time consuming while the other one is less accurate and less time consuming. The most significant difference between the two methods is the shape of the enclosure duct.

2.2.5.1. Method 1- Changing the angle of attack using the 3d modelling software

The angle of attack an airfoil can be changed using the 3D modelling software as it is shown.



This method requires starting from the geometry modelling stage going through all the steps of Ansys Fluent (Geometry – Mesh – Setup ... etc.). However, the shape of the containing duct can be rectangular as it is clear from the figure above. This method generates accurate results. However, it takes longer since the whole process has to be done.

2.2.5.2. Method 2- Changing the angle of attack from Ansys Fluent setup

The second method of changing the angle of attack is by changing the inlet velocity vectors where the defined velocity will have the required magnitude and direction. The advantage of this methodology is the time saved where the changing process can be done in the "Setup" step of Ansys Fluent without re-doing the previous processes (Geometry and Mesh).



As it is shown in the figure, the velocity with an angle of attack can be resolved to two components:

- Y direction: $v_v = V_{\infty} \times \sin \propto$
- X Direction: $v_x = V_{\infty} \times \cos \propto$

Hence, the velocity components can be entered to the "Boundary Conditions" where ansys will automatically calculate the resultant velocity and angle.

For example, if the free stream velocity is 80 m/s and the angle of attack is 15°:

- $v_y = 80 \times sin 15 = 20.706$
- $v_x = 80 \times cos \, 15 = 77.274$

Problem Setup	Boundar	y Conditions	
General Models Materials Phases Cell Zone Conditions	Zone airfoil inlet interior-nat outlet	Velocity Inlet X	
Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values	symmetry- symmetry-	Zone Name Inlet	
Solution Solution Methods Solution Controls Monitors		Momentum Thermal Radiation Species DPM Multiphase UDS Velocity Specification Method Components	
Solution Initialization Calculation Activities Run Calculation		Supersonic/Initial Gauge Pressure (pascal)	
Results Graphics and Animations Plots Reports	Phase mixture	X-Velocity (m/s) 277.274 constant Y-Velocity (m/s) 20.706 constant	Ai

Note: The velocity in X direction is with a (-) sign. This is due to the fact that the geometry has been designed in such orientation where the free stream has to be in the negative X direction.

Note: After each change in the angle of attack, the "Reference Values" should be updated to compute from "Inlet" as it is shown.

Problem Setup	Reference Values		
General Models Materials	Compute from inlet	•	
Phases Cell Zone Conditions Boundary Conditions	Reference Values Area (m2)	0.05	
Mesh Interfaces Dynamic Mesh Reference Values	Density (kg/m3)	1.225	
Solution	Depth (m)	0.5	
Solution Methods Solution Controls	Enthalpy (j/kg)	0	
Solution Initialization	Length (m)	0.2	
Run Calculation	Pressure (pascal)	0	
Results Graphics and Animations	Temperature (k)	288.16	
Plots Reports	Velocity (m/s)	80.00006	
	Viscosity (kg/m-s)	1.7894e-05	
	Ratio of Specific Heats	1.4	

After updating the "Reference Values" it can be noticed that the velocity has been automatically calculated to be the resultant velocity.

Since the velocity has been defined using the components, the monitors of the lift and the drag has to be set to read the required force components.



As it is clear from the graph, with the existence of the angle of attack, the lift and the drag are not exactly the pure forces on one of the Y or X axis. The lift and the drag can be represented by the following equations:

- $L = (Y \times \cos \alpha) (X \times \sin \alpha)$
- $D = (Y \times \sin \alpha) + (X \times \cos \alpha)$

Hence, the coefficients of X and Y have to be entered to the "Monitors" section where:

- For Lift: $(X : \sin \propto, Y : \cos \propto)$
- For Drag: $(X : \cos \propto, Y : \sin \propto)$

For example, for free stream velocity is 80 m/s and the angle of attack is 15°:

- For Lift: $(X : -\sin 15 = -0.259, Y : \cos 15 = 0.966)$
- For Drag: $(X : \cos 15 = 0.966, Y : \sin 15 = 0.259)$

Problem Setup	Monitors		
General	Residuals, Statistic and Force Mo	onitors	
Models	Residuals - Print, Plot		1-1
Materials	Statistic - Off	Lift Monitor	
Phases	Drag - Print, Plot, Write		
Cell Zone Conditions	Lift - Print, Plot, Write	Options	Wall Zones
Boundary Conditions		Print to Concelo	airfoil
Mesh Interfaces	Drag Monitor	Print to Console	chi ton
Dynamic Mesh		V Plot	
Reference Values	Options	Window	
Solution	Print to Console	3 Curves Aves	
Solution Methods	V Plot		
Solution Controls	Window	Write Write	
Monitors		File Name	
Solution Initialization	Curves	d-history.txt	
Calculation Activities	Write		
Run Calculation	File Name	Per Zone	
Results	cd-history.txt	Force Vector	
Graphics and Animations		v v 7	
Plots	Per Zone		
Reports	Force Vector	0.966 0	
	Y		
	-0.966 0.259		
		OK Plot Clear	Cancel
		L	- Ileci
	ОК	Plot Clear Cancel Help	ter
		Interrunting	

Note: All the coefficients in X direction have been inversed since the geometry has been designed in such orientation where the free stream has to be in the negative X direction, as it has been mentioned previously.

For multiple angles of attack, the same process has to be repeated for each angle. It is preferred to construct a table with the required velocity vectors and the monitoring coefficients on Microsoft Excel as it is shown below.

			Monitors					
		Bound	dary Conditions > Inlet		D	rag	Lif	t
Velocity m/s	AOA	inlet X	inlet Y		Υ	Х	Y	Х
80	-5	79.696	-6.972		-0.087	0.996	0.996	0.087
	0	80.000	0.000		0.000	1.000	1.000	0.000
	5	79.696	6.972		0.087	0.996	0.996	-0.087
	10	78.785	13.892		0.174	0.985	0.985	-0.174
	15	77.274	20.706		0.259	0.966	0.966	-0.259
	20	75.175	27.362		0.342	0.940	0.940	-0.342

The process which has to be repeated for each angle can be concluded in few steps:

- 1. Boundary Conditions >> Inlet >> Edit >> Changing the velocity vectors
- 2. Reference Values >> Compute from >> Inlet
- 3. Monitors >> Lift and Drag monitors

The method saves a lot of time in the case of testing many angles of attack. However, it is noticed from the figure below that the angles of attack of the flow changes before approaching the wing which causes inaccurate results.



The best way to solve this problem is changing the enclosure duct from a rectangular shape to a C-duct shape. A circular inlet covering the whole model will insure that the angle of attack is maintained to cover the whole wing with the required flow angle of attack.

Note: The most important step while constructing the C shaped inlet is making sure that the whole model is included inside the C shaped inlet. It is recommended to keep the model as closer as possible to the leading part of the C shaped inlet.



Hence, it is noticeable that the whole model is being covered with the flow approaching with the required angle of attack.



2.2.5.3. Comparison between the two methods

	Method 1	Method 2
Enclosure shape	Rectangular	C duct
Angle change using	3D modelling software	Fluent Setup
Time required	More	Less
Results accuracy	More accurate	Less accurate (Acceptable)
Domain Size	Smaller	Bigger
Changing steps	The whole procedure	Boundary conditions –
		Reference Values - Monitors
Calculating Velocity	Not required	Required
components		
Lift and Drag Monitors	Lift: (X : 0, Y : 1)	Lift: $(X : - \sin \propto, Y : \cos \propto)$
	Drag: (X : 1, Y : 0)	Drag: (X : $\cos \propto$, Y: $\sin \propto$)

2.3. Fluent – 3D - Finite Wing

2.3.1. Geometry

** The geometry file should be saved in an individual file



** In ANSYS Workbench window:

Drag (Fluid Flow (Fluent)) to the Project Schematic inside the red square



** Right Click on (Geometry) >> Import Geometry >> Browse >> Locate the geometry file



Suhail Mahmud and Mohamad Wissam

** Open Geometry by double clicking on "Geometry". Chose the units used while constructing the geometry files

** On the Tree Outline on the left side >> Right Click on "Import" >> Generate

** After the geometry appears, go to: Tools >> Freeze

** Click on the axis which reorients the view to the side view. In this case it is X axis.

ANSYS Workbench		×
Select desired length unit:		
Meter	C Foot	
C Centimeter	C Inch	
C Millimeter		
C Micrometer		
Always use project u Always use selected Enable large model su	unit 9 unit upport	







** On the Tree Outline, click on the side view plane. In this case it is YZ Plane

** On the Tree Outline window, Chose "Sketching">> Draw >> draw a rectangle which represents the side view of the test domain or the space.

** Fix the dimensions using "Dimensions" option on the sketching tool bar on the left side. The side view of the domain should look like the figure.

Note: In the C-Duct case, the circle has to be drawn first, followed by the rectangle starting exactly from the middle of the circle. The unwanted parts have to be trimmed.

** After fixing the dimensions, click on to set the depth of the domain.




** Set the directions and the depth of the domain then click

誟 Generate

** After these steps, the Tree Outline should look like the shown figure.

** After getting the previous outline, go to Tools >> Enclosure.

** Change the "Shape" to "user defined"

De	etails View	4		
-	Details of Extrude2			
	Extrude	Extrude2		
	Base Object	Sketch1		
	Operation	Add Material		
	Direction Vector	None (Normal)		
	Direction	Normal		
	Extent Type	Fixed		
	FD1, Depth (>0)	15		
	As Thin/Surface?	No		
	Merge Topology?	Yes		



File Create Concept	Tools View Help			
2 📙 📕 🖉	🕞 Freeze			
YZPlane 🔹 🔸	🙇 Unfreeze			
誟 Generate 🛛 😚 Share	🍘 Named Selection			
ree Outline	Attribute			
_ → A: Fluid Flow (F	🕼 Mid-Surface			
ZPlane Image: Constraint of the sector of				
🗸 🛧 ZXPlane	Enclosure			
🗄 🗸 🖈 YZPlane	📕 Face Split			

S	ketching Modeling		
De	etails View	д	
Ξ	Details of Enclosure1		
	Enclosure	Enclosure1	
	Shape	Box 🔻	
	Number of Planes	Box	
	Cushion	Cylinder	
	FD1, Cushion +X value (>0)	User Defined	
	FD2, Cushion +Y value (>0)	1 m	
	FD3, Cushion +Z value (>0)	1 m	
	FD4, Cushion -X value (>0)	1 m	
	FD5, Cushion -Y value (>0)	1 m	
	FD6, Cushion -Z value (>0)	1 m	
	Target Bodies	All Bodies	
	Merge Parts?	No	

** For the cell "User Defined Body", Chose "solid" from the Tree outline and click "Apply".

** Click ^{Generate}. The resulted geometry should look like the shown figure.

** Go to "Create" >> Boolean.

** On the details view on the left bottom corner, change "Operation" from "unite" to "Subtract".

Suhail Mahmud and Mohamad Wissam







De	etails View		ą
Ξ	Details of Bo	olean1	
	Boolean	Boolean1	
	Operation	Unite	Ŧ
	Tool Bodies	Unite	
		Subtract	
		Intersect	

** Click ^{Generate}. The Tree Outline should look like the shown figure. Notice that there is only 1 Part, 1 Body while the geometry "spiroid" has been subtracted from the domain "solid". Moreover, "Solid" does not necessarily mean that it is solid body, it is still the surrounding air.

However, Ansys calls the generated domain "solid". The object can be renamed if required.

2.3.2. Mesh

** Close Geometry. Double click on "Mesh".

** In the "Mechanical Window", on the Outline part, Right click on "Mesh" >> Insert >> Method >> Automatic. Then click on the body which is representing the domain. Then click "apply".







** On the Outline part, Left click on "Mesh". Then on the "Details of Mesh" window Change the followings:

- Relevance>> controls the density of the mesh in regions closer to the geometry.

- Use advanced size function >> On Proximity and Curvature

- Relevance Center >> Fine

- *Min Size>> the minimum size of the mesh elements in meters*

- Max face Size>> the maximum size of the mesh elements in meters

- Max Size >> equal to "Max face Size"

- Auto Mesh Based Defeaturing >> Off

Then click ³ Generate Mesh

** After the mesh is generated. Choose the Face choosing tool.

** Left click on each face in the geometry>>Right click >> Create Named Selection >> Name each face according the orientation of the model. Make sure that the inlet is named "inlet", the outlet is named "outlet", and the other 4 sides' names start with "symmetry – (name)".

De	etails of "Mesh"	φ.
	Defaults	
	Physics Preference	CFD
	Solver Preference	Fluent
	Relevance	0
Ξ	Sizing	
	Use Advanced Size Function	On: Proximity and Curvature 💌
	Relevance Center	Fine
	Initial Size Seed	Active Assembly
	Smoothing	Medium
	Transition	Slow
	Span Angle Center	Fine
	Curvature Normal Angle	Default (18.0 °)
	Proximity Accuracy	0.5
	Num Cells Across Gap	Default (3)
	Min Size	8.e-004 m
	Max Face Size	4.0 m
	Max Size	4.0 m
	Growth Rate	Default (1.20)
	Minimum Edge Length	1.6276e-004 m
÷	Inflation	
Ξ	CutCellMeshing	
	Active	No
÷	Advanced	
Ξ	Defeaturing	
	Pinch Tolerance	Default (7.2e-004 m)
	Generate Pinch on Refresh	No
	Automatic Mesh Based Defeaturing	Off
+	Statistics	





** After selecting each face separately and assigning a named selection, change the selection type to "Box selection" as shown.

** Hence, go to the side view and select the model or the wing as it is shown. Then right click>> create named selection >> call it any name (avoid calling it Inlet, Outlet and Symmetry).

** After doing the named selection step, the tree outline should look like the shown figure. Notice all the named selections are listed.

** Close the "Mechanical Window" >> Right click on "Mesh" >> Update.

File Edit	View Unit	s Tools Help
	्रिन् 🕟 (b 🖪 🖌
」	🞝 Single Se	elect e 📕
Mesh 乡	🖧 Box Sele	ct 🕅 🕅 🕅 Me







2.3.3. Setup

** Double Click on "Setup"

** Tick (Double Precision)>> Chose "Parallel" and choose the number of processors to be 4 unless if more processors are licensed. In the case your computer does not have 4 processors, choose the maximum number of available processors.

** Chose the "Type" to be:

- "Pressure Based) for incompressible flow

- "Density Based" for compressible flow



** Go to "Define">> Operating Conditions.

** Define the Static Pressure in the operation altitude.

** In "Models" Section >> Double click on "Viscous" and chose:

- Model: K-epsilon
- K-epsilon model: Realizable

- Near-Wall Treatment: Non-Equilibrium Wall Functions

File Mesh	Define	Solve	Adapt	Surface	Display
i 💕 🕶 🛃	G	eneral			
	M	odels			=
Problem Setup	M	aterials			_
General Models	Pł	nases			-
Materials	C	ell Zone (Conditio	ns	
Phases Cell Zone Co	В	oundary	Conditio	ns	
Boundary C	0	perating	Conditio	ns	-

Operating Co	nditions
Pressure Operating Reference Pressure X (m) 0 Y (m) 0 Z (m) 0 OK	Gravity Gravity Gravity Gravity Gravity Gravity Gravity Gravity Help
Problem Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results Graphics and Animation Plots Reports	Models Models Multiphase - Off Energy - Off Viscous - Realizable k-e, Non-Eq Wall F Off Viscous - Realizable k-e, Non-Eq Wall F Off Viscous Model Model Model Inviscid Laminar Spalart-Allmaras (1 eqn) k-omega (2 eqn) Transition k-kl-omega (3 eqn) Transition SST (4 eqn) Reynolds Stress (7 eqn) Scale-Adaptive Simulation (DES) Large Eddy Simulation (DES) Large Eddy Simulation (LES) k-epsilon Model Standard RNG Realizable Near-Wall Treatment
	 Standard Wall Functions Non-Equilibrium Wall Functions Enhanced Wall Treatment User-Defined Wall Functions

Ansys Workbench Basics Guide

** In "Materials" Section >> Double Click on "air" >> set the density and the viscosity Pressure in the operation altitude.

** In "Boundary Conditions" Section >> Double Click on "Inlet" >> Change "Velocity Specification Method" to "Components" >> Insert the values of the flow velocity with respect to the coordinate system.

** In "Reference Values" section >> Choose "Compute from" to be "inlet" >> Insert the flow conditions at the operating altitude. Moreover, insert:

- Area: the reference area of the wing (the projection area)

- Length: Mean Aerodynamic Chord length

Suhail Mahmud and Mohamad Wissam

Problem Se	tup	M	aterials
General		Ma	terials
Models		F	uid
Materials			air
Phases		S	olid
Cell Zone	Conditions		aluminum
Bound			
Mesh 1	Create/Edit M	Aate	erials
Dynam [
Refere	Name		
Solution	air		
Solutio	Chemical Formula		
Solutio			
Monito			
Solutio			
Calcula			
Run Ci			
Results	Properties		
Creaki	Density (ka/m	2) (
Graphi	Density (kg/iii	"	constant
Piots		Ì	0.6601
Repor			0.0001
	Viscosity Acalm	a) (
	viscosity (kg/iii-	ຶ	constant
			1.595e-05



** In "Solution Methods" Section >> Chose "Scheme" to be "Coupled".

** Change the "Momentum", "Turbulent Kinetic Energy" and "Turbulent Dissipation Rate" to "Second Order Upwind"

** In "Solution Controls" Section >> Click on "Limits" >> set the "Maximum Turb. Viscosity Ratio" to be 1e+20.

** In "Monitors" section >> Double click on "Residuals" >> Tick on (Print, Plot) >> on the right side, remove the ticks from all the parameters except continuity. Moreover, change the absolute criteria of the continuity to be 1e-6 as shown in the figure.



Ansys Workbench Basics Guide

** In "Monitors" section >> Double click on "Drag" >> Tick on (Print to console, Plot, Write) >> add (.txt) to the end of the file name >> Adjust the unit vector which is representing the direction of the Drag force with respect to the coordinate system.

** In "Solution Initialization" section >> Chose "Hybrid Initialization".

** In "Run Calculations" Section >> Set the required number of iterations and "Calculate".

** The process can be paused, stopped and saved. To continue solving the problem, the setup should be started from "Solutions" in the main Ansys window.

** The results can be found from the same window as it was shown in the 2D case. More options can be found in CFD Post.

blem Setup	Monitors
General	Residuals, Statistic and Force Monit
1odels	Residuals - Print. Plot
laterials	Statistic - Off
hases	Drag - Print, Plot, Write
Drag Monitor	Person of
Options	Wall Zones
Print to Console	wing
V Plot	
Window	
Curv	es Axes
Write	
File Name	
cd-history.txt	
Cu notor prote	
Per Zone	
Force Vector	
X Y	Z
	-1
General	Initialization Methods
Models	O Hybrid Initialization
Phases	Standard Initialization
Cell Zone Conditions	
Boundary Conditions	More Settings Initialize
Mesh Interfaces	Patch
Dynamic Mesh	[ddini
Reference Values	
Solution	
Joid don	Help
Solution Methods	Help
Solution Methods Solution Controls	Help
Solution Methods Solution Controls Monitors	Help
Solution Methods Solution Controls Monitors Solution Initialization	Help
Solution Methods Solution Controls Monitors Solution Initialization	Help
Solution Methods Solution Controls Monitors Solution Initialization	Help ne Solve Adapt Surface D
Solution Methods Solution Controls Monitors Solution Initialization	Help ne Solve Adapt Surface D @
Solution Methods Solution Controls Monitors Solution Initialization	Help ne Solve Adapt Surface D Run Calculation
Solution Methods Solution Controls Monitors Solution Initialization	Help Te Solve Adapt Surface D Run Calculation
Solution Methods Solution Controls Monitors Solution Initialization	Help The Solve Adapt Surface D
Solution Methods Solution Controls Monitors Solution Initialization File Mesh Defin Problem Setup General Models Materials	Help The Solve Adapt Surface D
Solution Methods Solution Controls Monitors Solution Initialization File Mesh Defin Problem Setup General Models Materials Phases	Help The Solve Adapt Surface D Run Calculation Check Case Number of Iterations
Solution Methods Solution Controls Monitors Solution Initialization File Mesh Defin Problem Setup General Models Materials Phases Cell Zone Condition	Help The Solve Adapt Surface D The Solve Adapt
Solution Methods Solution Controls Monitors Solution Initialization File Mesh Defin Problem Setup General Models Materials Phases Cell Zone Condition Boundary Condition	Help The Solve Adapt Surface D Profile Undate Interval
Solution Methods Solution Controls Monitors Solution Initialization File Mesh Defin Problem Setup General Models Materials Phases Cell Zone Condition Boundary Condition Mesh Interfaces	Help Help Ne Solve Adapt Surface D Run Calculation Check Case Number of Iterations Profile Update Interval
Solution Methods Solution Controls Monitors Solution Initialization File Mesh Defin Problem Setup General Models Materials Phases Cell Zone Condition Boundary Condition Mesh Interfaces Dynamic Mesh	Help The Solve Adapt Surface D The Solve Adapt

Data File Quantities...

Calculate

Help

Solution

Monitors

Solution Methods

Solution Controls

Solution Initialization Calculation Activities

Calculation

2.3.4. CFD Post

** Close "Setup". Double click on "Results".

** From the outline tree, the parts can be displayed or hidden.

** In order to display pressure distribution or velocity vectors, a plane has to be constructed at the section as it is shown.



** Select the orientation of the plane where:

- Method: select which plane will be parallel to the new constructed plane (In this case it is YZ plane).

- X: The distance from the origin to the plane. If the origin set to be at the wing root in the 3d modelling geometry, then X means the span wise distance from the wing root.

After fixing the settings, click "Apply". The plane has been constructed as it is shown.

** On the upper bar,

1. Velocity vectors

2. Contours (Pressure, Vortectiy, Turbulence... etc.)

3. Streamlines

** To display the properties, the Plane has to be selected as "Location".



Suhail Mahmud and Mohamad Wissam

** In the condition of the 3D streamlines, it has to be defined to start from "Inlet". In some cases, a custom plane has to be constructed to define it as a starting plane of the streamlines. This is useful when it is needed to display the streamlines over a specific region.

** For example, it is noticed in the first figure that the streamlines have been started from the inlet which led to cover up the whole domain area with the streamlines.

** In order to define the starting of the streamlines to be exactly projected on the wing area;

- Location >> User Surface

- Method >> Transformed Surface



** The surface of the wing will be copied and transferred forward to use it as a starting plane of the streamlines:

- Surface Name: Wing

- Activate "Transition" >> Move the plane to the forward direction (In this case it is Z direction).

- Activate "Scale" and use factor of 1.02 (This step is to ensure that the starting plane is a little wider than the original wing area. Hence it will be ensured that the streamlines will be covering the whole model without gaps on the sides.

** Create "Streamlines" using the "User Surface" for "Start from". It is noticed that the lines have been refined for a better visualization.







** A plot presenting the pressure coefficient distribution over the wing surface can be plotted:

- Calculators >> Macro Calculators >> Macro: (Cp Polar Plot)

- Boundary List: the object where the pressure distribution has to be investigated. In this case it is "wing".

- Slice Normal: to calculate the pressure destitution over an airfoil, the wing has to be sliced at a specific span. The axis going through the span is the span wise axis which is normal to the slice. In this case it is X axis.

- Slice Position: the distance of the slice from the origin

- Plot axis: the direction in which the Cp variation is investigated (the chord wise direction). In this case it is Z direction.

** Chose "Calculate" then "View Report".

Note: the Y axis is oriented in a way where it shows the upper surface at the top which leads to the fact that the Y axis has negative values of Cp at the top and the positive values at the bottom.



2.3.5. Tecplot

** More results can be presented using Tecplot. In order to import the solution data to tecplot:

- Open "Solutions"

- As it was explained in Ansys, planes have to be constructed to show the results. Hence, the plans have to be constructed before exporting the solution data because planes constructed using tecplot cannot represent the solution data imported from Anys.

- To construct a new plane: Surface >> Plane

- In "Options": Point and Normal: allows the user to assign a point and an axis as references for creating the plane.

- In "Points": enter the position of the point through which the plan will be passing

- In "Normal": enter the direction vector for the axis which the plane has to be normal to.

Note: The plane can be created with an angle with respect to the axis by entering the direction vectors into more than one field.

- Click "Create"



Fluid Flow (FLUENT) Pa	rallel FL	UENT@Si	ıhail-PC	[3d, dp, pbn
File Mesh Define Solve	Adapt	Surface	Display	Report Para
) 🖄 • 🛃 • 🔟 🞯 📗	G ↔	Zone. Partiti	 on	に開・
Problem Setup	Run C	Point.		
General		Line/R	ake	nuiouu Mode
Models		Plane.		eview mesi
Materials	Number	Quadr	ic	orting Inte
Cell Zone Conditions	200	Iso-Su	irface	
Boundary Conditions	Profile I	Iso-Cl	ip	
Mesh Interfaces Dynamic Mesh	1	Transf	form	
Reference Values	Data	Manag	je	Acoustic Si
Solution	Data		_	

ptions		Sample Density	Surfaces	
Aligned wit Aligned wit Point and N Bounded Sample Point Plane Tool	h Surface h View Plane Jormal nts	Edge 1 1 A	inlet interior-solid outiet symmetry-bottom symmetry-top symmetry-top	
		Reset Points		
oints			Normal	
x0 (m)	×1 (m) 0.15	×2 (m)	ix (m)	
y0 (m) 0	y1 (m)	y2 (m) 0.15	iy (m) 0	
z0 (m)	z1 (m) 0.03627	Edge 1 Image: constant symmetry-solid outlet symmetry-solid symme		
lew Surface Na	me			
plane-8				

** In order to display the created plane:

- Display >> Mesh
- Highlight (Plane #) >> Display

Note: All the planes must be created before exporting the solution data

** In order to export the solution data to tecplot:

- File >> Export >> Solution Data







Suhail Mahmud and Mohamad Wissam

** In "Export":

- File Tpye: Tecplot

- Surfaces: Chose the "Wing" and all the plans needed to be transferred to tecplot.

- In "Quantities": all the needed parameters should be highlighted. Generally:

- Static Pressure
- Pressure Coefficient
- Dynamic Pressure
- Absolute Pressure
- Total Pressure
- Relative Total Pressure
- Density
- Density All
- Velocity Magnitude
- X Velocity
- Y Velocity
- Z Velocity
- Vorticity Magnitude
- Helicity
- X-Vorticity
- Y-Vorticity
- Z-Vorticity
- X Coordinate
- Y Coordinate
- Z Coordinate

- Click "Write". Close Ansys after the import is done.



Ansys Workbench Basics Guide

Suhail Mahmud and Mohamad Wissam



- File >> Load Data File(s)>> Tecplot Data Loader

- Change the "Initial Plot Type" to: 3D Cartesian

- The model gets imported in a different orientation. Hence, it has to be rotated using the coordinators controllers shown in the figure.



** After reorienting the geometry to the required position:

- Chose "Stream traces":

- U: X-velocity
- V: Y-velocity
- W: Z-velocity

** Click on the sign showed in the figure. This tool allows the user to draw a line where the stream lines covers all the area passed by the drawn line. Hence, the user can control the density of the lines. Moreover, the concentration of the lines can be focused on a specific region by drawing more than one line at that region.

3D Cartesian 🔹	
Zone Surfaces Layers: Mesh Contour Vector Scatter Shade	
Effects:	
Translucency	
Zone Style	Select Variables
Derived Objects: Iso-Surfaces	The vector variables are not defined. Choose the vector variables:
Streamtraces 🛄 🛰	U: x-velocity 🗸
Time 0	V: y-velocity
	W: z-velocity 🔹
4 − + 	OK Cancel Help
I¶ - ▶ + ▶I	OK Cancel Help

Add a single or rake of streamtraces



ne O

2.4. Fluent - Internal flow through pipes and ducts

2.4.1. Geometry

** The geometry file should be saved in an individual file



** In ANSYS Workbench window:

Drag (Fluid Flow (Fluent)) to the Project Schematic inside the red square



** Right Click on (Geometry) >> Import Geometry >> Browse >> Locate the geometry file



** Open Geometry by double clicking on "Geometry". Chose the units used while constructing the geometry files

** On the Tree Outline on the left side >> Right Click on "Import" >> Generate

** The duct geometry will appear. The inlets and the outlets have to be defined as surfaces:

- Concepts >> Surfaces from Edges

- Choose the edges of the inlet and the outlet

- Click "Apply" >> Generate





** After defining the surfaces, the whole duct has to be defined to be filled with material:

- Tools >> Fill

- ** In "Details of Fill1":
- Extraction Type: By Caps
- Preserve Solid:

* Choose "Yes" if the outer surface of the duct (the wall of the duct) is needed for the analysis. For example, if there is heat transfer between the fluid and the wall and then between the wall and the environment like the case in the heat exchanger.

* Chose "No" if the duct wall is not needed. This saves the resources needed to process the mesh of the wall.

In this case, the wall is not required; hence, "No" will be chosen for "Preserve Solids" >> Generate.

** It can be noticed that the internal shape of the duct has been defined as the material.



S	Sketching Modeling		
De	Details View 🛛		
Ξ	Details of Fill1		
	Fill	Fill 1	
	Extraction Type	By Caps 🔹	
	Target Bodies	By Cavity	
	Preserve Capping Bodies	By Caps	
	Preserve Solids	Yes	



2.4.2. Mesh

** Close the Geometry Design Modular >> Double Click on "Mesh".

On the Outline part, Right click on "Mesh". Then on the "Details of Mesh" window Change the followings:

- Relevance>> controls the density of the mesh in regions closer to the geometry.

- Use advanced size function >> On Curvature

- Relevance Center >> Fine

Then click ^{\$ Generate Mesh}

Note: Since boundary layer is important in the internal flow cases. For more accurate study of the boundary layer, whether it is internal flow or external flow (For example, the boundary layer over the surface of the wing), "Inflation" has to be created which arranges more refined mesh for the boundary layer region:

- Right click on "Mesh" >> Inflation

- Choose "Geometry" to be the whole body >> Apply

- Choose all the faces except the inlet and the outlet to be the "Boundary" >> Apply

- The other options like (Inflation Option and Number of layers are up to the user)

▼	А	
1	G Fluid Flow (FLUENT)	
2	🔞 Geometry	× .
3	🎯 Mesh	- 🔁 🔒
4	🍓 Setup	° .
5	Solution	° ? 🔒
6	😡 Results	° ? 🔒
Fluid Flow (FLUENT)		





** The difference can be noticed where the mesh is refined after using inflation (Right).

** After the mesh is generated. Choose the Face choosing tool.

** Left click on inlet surface >>Right click >> Create Named Selection >> Type "Inlet"

- Do the same for "Outlet"

** After doing the named selection step, the tree outline should look like the shown figure. Notice the inlet and the outlet are listed and marked.

** Close the "Mechanical Window" >> Right click on "Mesh" >> Update. Suhail Mahmud and Mohamad Wissam







2.4.3. Setup

** Double Click on "Setup"

** Tick (Double Precision)>> Chose "Parallel" and chose the number of processors to be 4 unless if more processors are licensed. In the case your computer does not have 4 processors, then choose the maximum number of processors available.

** Chose the "Type" to be:

- "Pressure Based) for incompressible flow

- "Density Based" for compressible flow



	A Fluid Flow (FLUENT) Geometry ✓ Mesh ✓ Setup Solution Results Fluid Flow (FLUENT)
	FLUENT Launcher
Dimension 2D 3D Display Options Display Mesh After Reading Embed Graphics Windows Workbench Color Scheme Do not show this panel aga Show More Options	Options Double Precision Use Job Scheduler Use Remote Linux Nodes Processing Options Serial Parallel (Local Machine) Number of Processes 4
<u>0</u> K	<u>C</u> ancel <u>H</u> elp ▼
File Mesh Define Sol	ve Adapt Surface Display Report Parallel 了夺 ④ ① / ◎ ① 川 · □ ·
Problem Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	Mesh Scale Check Report Quality Display Display Solver Yelocity Formulation Image: Pressure-Based Image: Absolute Density-Based Relative Time Steady Transient Image: Cravity Help Help

Suhail Mahmud and Mohamad Wissam

** In "Models" Section >> Double click on "Viscous" and chose:

- Model: K-epsilon

- K-epsilon model: Realizable

- Near-Wall Treatment: Enhanced Wall Treatment

** More information about Fluent Models can be found on

<<<u>http://aerojet.engr.ucdavis.edu/flu</u> enthelp/html/ug/node1336.htm>>

** In "Materials" Section >> Double Click on "air" >> set the density and the viscosity. More materials can be added from "Fluent Database".

Problem Setup	Models
General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces	Models Multiphase - Off Energy - Off <u>Viscous - Realizable k-e, Non-Eq Wall F</u> Viscous Model
Dynamic Mesh Reference Values Solution Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results Graphics and Animation	Model r Inviscid Laminar Spalart-Allmaras (1 eqn) k-epsilon (2 eqn) k-omega (2 eqn) Transition k-kl-omega (3 eqn) Transition SST (4 eqn) Reynolds Stress (7 eqn) Scale-Adaptive Simulation (SAS) Detached Eddy Simulation (DES) Large Eddy Simulation (LES) Large Eddy Simulation (LES)
Reports	k-epsilon Model Standard RNG Realizable Near-Wall Treatment Standard Wall Functions Non-Equilibrium Wall Functions Enhanced Wall Treatment User-Defined Wall Functions

Create/Edit Ma	iterials			×
Name		Material Type	Order Materials by	
air		fluid	Name O	
Chemical Formula		FLUENT Fluid Materials Jair Mixture	FLUENT Database	a ase
Properties		none		
Density (kg/m3)	constant 1.225	Edit		
viscosity (kg/m-s)	constant 1.7894e-05	Edit		
		_		
	Change/Create	Delete Close Help		

Ansys Workbench Basics Guide

Suhail Mahmud and Mohamad Wissam

** In "Boundary Conditions" Section >> Double Click on "Inlet" >> Change "Velocity Specification Method" to "Magnitude, Normal to Boundary" >> Insert the inlet flow velocity

** In the "Turbulence" section, enter the "Turbulent Intensity" and "Hydraulic" Diameter" of the inlet.

Note: Turbulent Intensity and Hydraulic are well known parameters in fluid dynamics. Both of them can be calculated using simple formulas. The formulas can easily found online.

<<<u>http://www.cfd-</u> online.com/Wiki/Turbulence_intensity>>

<<<u>http://en.wikipedia.org/wiki/Hydraulic</u> <u>diameter</u>>>

** In "Solution Methods" Section >> Choose "Scheme" to be "Coupled".

** Change the "Momentum", "Turbulent Kinetic Energy" and "Turbulent Dissipation Rate" to "Second Order Upwind"

Velocity Inlet	×
Zone Name	
inlet	
· · · · · · · · · · · · · · · · · · ·	
Momentum Thermal Radiation Species DPM Multiphase UDS	
Velocity Specification Method Magnitude, Normal to Boundary	
Reference Frame Absolute	
Velocity Magnitude (m/s) 10 constant	
Supersonic/Initial Gauge Pressure (pascal) 0 constant	
Turbulence	
Specification Method Intensity and Hydraulic Diameter	
Turbulent Intensity (%) 10	
Hydraulic Diameter (m) 1	
OK Cancel Help	

	8 I J I J I
oblem Setup	Solution Methods
General	Pressure-Velocity Coupling
Models Materials	Scheme
Phases	Coupled 👻
Cell Zone Conditions Boundary Conditions	Spatial Discretization
Mesh Interfaces	Gradient
Dynamic Mesh Reference Values	Least Squares Cell Based 👻
olution	Pressure
Solution Methods	Standard 👻
Solution Controls	Momentum
Monitors	First Order Upwind 👻
Solution Initialization	Turbulent Kinetic Energy
Calculation Activities	First Order Upwind 👻
Run Calculation	Turbulent Dissipation Rate
esults	First Order Upwind 🗸
Graphics and Animations Plots	Transient Formulation
Reports	▼

** In "Solution Controls" Section >> Click on "Limits" >> set the "Maximum Turb. Viscosity Ratio" to be 1e+20.

** In "Monitors" section >> Double click on "Residuals" >> Tick on (Print, Plot) >> on the right side, remove the ticks from all the parameters except continuity. Moreover, change the absolute criteria of the continuity to be 1e-6 as shown in the figure.

** In "Solution Initialization" section >> Chose "Hybrid Initialization".

Problem Setup	Solution Controls
General Models Materials	Courant Number 200
Phases	Explicit Relaxation Factors
Cell Zone Conditions Boundary Conditions Mesh Interfaces	Momentum 0.75
Dynamic Mesh Reference Values	Pressure 0.75
Solution	Under-Relaxation Factors
Solution Methods Solution Controls	Solution Limits
Monitors Solution Initialization	Minimum Absolute Pressure (pascal)
Calculation Activities Run Calculation	Maximum Absolute Pressure (pascal) 5e+10
Results Graphics and Animations	Minimum Turb. Kinetic Energy (m2/s2)
Plots Reports	Minimum Turb. Dissipation Rate (m2/s3)
	Maximum Turb. Viscosity Ratio
	OK Default Cancel Help
	[D-6-th]
	Equations Limits Advanced

Monitors			
Residuals, Statistic and Force Monitors			
Residuals - Print, Plot			
Statistic - Off			
Dog Asso, No. 199	2		
Equations			
Residual	Monitor C	Check Converge	nce Absolute Criteria
continuity	V		1e-06
x-velocity			0.001
y-velocity			0.001
z-velocity			0.001
Residual Values			Convergence C
Normalize		Iterations	absolute
Scale	al Scale		
	Residuals, Statistic and Residuals, Statistic and Residuals - Print, Plot Statistic - Off Residual continuity x-velocity y-velocity z-velocity Residual Values Normalize Compute Loc	Monitors Residuals, Statistic and Force Mo Residuals - Print, Plot Statistic - Off Equations Residual Monitor (Continuity) y-velocity y-velocity z-velocity Residual Values Normalize Y Scale Compute Local Scale	MOINTOPS Residuals, Statistic and Force Monitors Residuals - Print, Plot Statistic - Off Equations Residual Monitor Check Convergent continuity V x-velocity V y-velocity V z-velocity V Residual Values V Normalize Trerations Scale Compute Local Scale

Problem Setup	Solution Initialization
General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values	Initialization Methods Hybrid Initialization Standard Initialization More Settings Initialize Patch
Solution Solution Methods Solution Controls Monitors Solution Initialization	Help

Suhail Mahmud and Mohamad Wissam



3. Common Problems

3.1. Autodesk Autocad compatibility with Ansys

The 3d models constructed in Autodesk Autocad can be imported to Ansys if saved in IGES format. However, in the case of the multiple bodies, Ansys fails to define the contact types on the boundary elements.

This problem has been solved recently in the latest version of Ansys. However, if older versions are used, it is better to construct the models using Solidworks, Caita or Rhino to ensure that there will be no geometrical importing problems in some later step.

3.2. The sharp trailing edges of the airfoils

The airfoils are usually constructed using the airfoil coordinates. The generated airfoils usually have very sharp trailing edges as it is shown in the figure below.



The sharp trailing edge causes difficulties while meshing which might lead to the failure of generating a good quality mesh. Since sharp edges cause sudden bending in the grid structure, the quality has to be sacrificed to generate a grid which fits the airfoil. Moreover, this problem can cause a failure in generating the inflation layers constructed to study the boundary layer.

This problem can be solved by trimming a small part of the trailing edge (few millimeters) and closing the gap with a curve which has a starting portion parallel to at least one of the top or the bottom surfaces.



3.3. General Meshing Problems

Most of the meshing problems can be solved using custom sizing. When Ansys shows a mesh failure due to a problematic geometry alert, the geometry can be displayed by (Right click on the message from the alerts window >> Show problematic Geometry. Selecting the problematic component whether it is an edge or a face and assigning a cell size which is smaller enough to cover the details of the problematic geometry is the easiest way to solve the problem without changing the geometry.

In some cases it is recommended to create a refinement for the mesh at certain points or locations. For example, the leading edge of a wing if a type of micro vortex generators has been installed on it. The refinement can be constructed by creating a solid part in the geometry stage (while drawing the domain). However, it shouldn't be included in the "Boolean" process. Later, in the meshing process, the solid part can be chosen (Right click on mesh >> Insert >> Sizing >> Type : Body of influence >> Chose the solid which is covering the detailed geometry).

However, in some cases, when the geometry contains a high order of nurbs, the smoothing has to be reduced in order to generate a mesh with acceptable quality. Although simplifying the geometry will be a better option since a mesh with low quality might cause problems in the solving process where the solution will not converge to the required margin of error.

3.4. Named Selection Process

The named selection process (assigning names to the surfaces) is an important stage where the spellings of some words have to be maintained carefully. For example, inlet, outlet and symmetry. These words are keywords where Ansys can define the surface as inlet if it has been named inlet.

The walls which are not supposed to have any friction or boundary layers (like the wall of a domain for external flow) should be called "Symmetry". This will direct Ansys to consider the wall as a wall without "no slip condition" or a boundary layer.

3.5. Solution Divergence

Solution divergence is a direct indicator of the poor quality of the mesh. When divergence is detected, the mesh has to be refined or reconstructed with new setting. Mostly:

- Smaller (Min Size)
- Higher (Relevance)
- Custom sizing for edges and faces
- Inflation layer for better study of the boundary layer
- Simplified geometry
- Wider domain

3.6. Temperature solution divergence while using Energy equation

When energy equation is being used, an unrealistic exit temperature could force the solution to accelerate \ decelerate the flow out of proportion leading to a "temperature solution divergence". Use common sense and experience when setting the initial guess for inlet and exit temperatures (only when energy equation is on, even if there is no combustion).

3.7. Scaling

When the model size is too big and the calculation process is too time consuming, it is recommended to scale down the model in order to reduce the needed resources. However, as per the flow similarity conditions, the boundary conditions have to be calculated to match the new scaled model.

According to the theory of flow similarity, the two conditions which need to be satisfied are:

- Geometric similarity The geometries bodies need to be similar
- Dynamic similarity The similarity parameters based on which other flow parameters will be calculated are Reynolds number and Mach number.

The C_L and C_D values will remain the same for both the geometries.

This is an example if scaling a wing the 1/3rd of its original size. Maintaining the same Mach number and Reynolds number for the two geometries, and also using the initial parameter values for the real geometry, the parameters that were re- calculated are:

- Density
- Velocity
- Viscosity Coefficient
- Pressure
- Temperature

The following are the calculations that were done to compute the new parameter values.

For convenience, the temperature T_2 =288.2K. The other given parameters for the real case are:

ρ_1	0.28852 kg/m3
V1	237 m/s
T ₁	217 K
T ₂	288.2 K
$\frac{C_1}{C_2}$	3
μ ₁	$4.7292 \times 10^{-5} \text{ kgm}^2/\text{sec}$

Equating Mach number,

$$M_{1} = M_{2}$$

$$\frac{V_{1}}{\sqrt{T_{1}}} = \frac{V_{2}}{\sqrt{T_{2}}}$$

$$V_{2} = V_{1} \times \sqrt{\frac{T_{2}}{T_{1}}} = 237 \sqrt{\frac{288.2}{217}} = 273.033 \text{ m/sec}$$

$$M_{2} = M_{1} = \frac{237}{\sqrt{(1.4 \times 28 \times 217)}} = 0.803$$

Equating Reynolds number,

$$Re_{1} = Re_{2}$$

$$\frac{\rho_{1}V_{1}C_{1}}{\mu_{1}} = \frac{\rho_{2}V_{2}C_{2}}{\mu_{2}}$$

$$\frac{C_{1}}{C_{2}} = 3$$

$$\frac{\rho_{2}}{\rho_{1}} = \frac{V_{1}C_{1}}{V_{2}C_{2}} \times \sqrt{\frac{T_{2}}{T_{1}}} = \frac{237 \times 3}{273.033} \sqrt{\frac{288.2}{217}} = 3$$

$$\rho_{2} = (0.28852)3 = 0.866 \text{ kg/m}^{3}$$

$$P_{2} = \rho_{2}RT_{2} = 71,717.97 \text{ KPa}$$

$$Re_{1} = \frac{\rho_{1}V_{1}C_{1}}{\mu_{1}} = \frac{(0.28852 \times 237 \times 2.61)}{0.000049272} = 3.662 \times 10^{6}$$

$$\mu_{2} = \frac{\rho_{2}V_{2}C_{2}}{Re_{2}} = 5.674 \times 10^{-5} \text{ kgm}^{2}/\text{sec}$$

3.8. Huge values of lift and drag

In some cases, the results show very huge or very small values for lift and drag even though the mesh has been refined and it can be considered as sufficient grid. Hence, the problem can be mostly in the boundary conditions, the reference values or the monitors.

In the reference values, the area and the length have to be defined accurately. The area is the projection area of the model while the length is the length of the model. Moreover, the inlet temperature has to be double checked.

Furthermore, for an altitude different than the sea level, the density and the viscosity has to be defined from the "materials" list and the pressure has to be defined in the "Operating conditions" (Define >> Operating Conditions).

Finally, the reference values have to be updated to be computing from the inlet after each change in any of the parameters.

4. Recommended Topics

The recommended topics are basically the topics or the problems which has not been explained or covered in the manual. Since some topics can be considered as advanced topics, a lot of research and troubleshooting will be needed to get the correct and reliable procedure of solving such problems.

4.1. Dynamic and Sliding mesh

Dynamic and sliding meshes are types of grids where the geometry can change its shape or condition while running the calculations. For example, a wing flap changing its angle or a car spoiler changing its position.

4.2. Meshing techniques – Gambit

Ansys uses ICEM meshing as a default meshing tool for all its products. However, it is recommended to carry on a study of comparing the meshing techniques and the quality between ICEM and the other meshing tools like Gambit.
4.3. Fluent Models

Viscous models are used mostly for the aerospace related studies. However, there are other models which can be useful like (Multiphase, Energy, Acoustics... etc) which are related to the engineering applications. For example, modelling heat exchangers, combustion chambers, mixing chambers, turbines and compressors.

4.4. Combining the structural loads with the aerodynamic loads

The aerodynamic loads can be calculated using fluent then transferred to the structural analysis to analyze the structural behavior. This study can be used to optimize the aerodynamic and the structural performance of an aircraft. However, the study will need very good computing resources.

4.5. Cables

Modelling cables in Ansys has to be investigated. Since creating an actual cable in the 3d modelling software and generating the mesh for such cable is very resources consuming methodology, an alternative way has to be found. For example, replacing the cable with a spring.

4.6. Composite

Modelling composite materials in Ansys is a well demanded topic. Even though there is a special library in Ansys for composite materials (ACP), it is not available for all Ansys licenses. Hence, finding a methodology to model the composite materials in Ansys without using the ACP library is a viable topic of research.

5. Useful Links

- Brief about mesh and grid types

http://www.innovative-cfd.com/cfd-grid.html

- Ansys Modelling and Meshing Guide

http://www.ewp.rpi.edu/hartford/users/papers/engr/ernesto/hillb2/MEP/Other/Articles/MeshingGuid

<u>e.pdf</u>

- Fluent 6.3 user guide: http://aerojet.engr.ucdavis.edu/fluenthelp/index.htm

- Fluent Models Details http://aerojet.engr.ucdavis.edu/fluenthelp/html/ug/node1336.htm

- CFD analysis of Vehicle Aerodynamics http://www.youtube.com/watch?v=dZR7Wi70Vec

- CFD analysis of Vehicle Aerodynamics (CFX not Fluent) http://www.youtube.com/watch?v=6adO0mv-eWw

The End

Good Luck =)

