AnsysWorkbenchBasicsGuide

Mechanical Engineering

Techno India NJR Institute of Technology, Udaipur

For Techno India NJR Institute of Technology Gen T Cl 201 CM Dr. Pankaj Kumar Porwal (Principal)

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 the irjourney indiscovering this application.

The flow of information is structured that the reader gets an understanding of howimportant ANSYS is, and how it works and what type of machines are needed for the studentlevel research expected. Then the tutorial goes on with simple straight forward examples ofstructuralandfluidphysicssimulatedusingtheANSYSpackage.Eventually,thetutorialaddresses the most important problems generally faced by the students such as unsuccessfulmeshing, or divergent solutions.

Disclaimer

Itisextremelyimportanttonotetwopointswhilefollowingthistutorial:

- The knowledge contained in this paper is by no means, accepted as mainstream, or anindustry best practice. It is merely the product of the experience of senior engineeringstudents who explored the program and desired to share their experience with thepackage.
- 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 tryother configurations, commit to trial and error procedures, and develop their owntroubleshootingskills inordertocreateworkingmodels.

For Techno India NJR Institute of Technology पैकज परिवाल Dr. Pankaj Kumar Porwa Principal)

TableofContents

Abs	tract		2
Dise	laimer		2
1.	Introduct	ion	5
2.	Exercises		8
	2.1.	StaticStructural –CantileverBeam	8
	2.1.1.	ProblemsSpecifications:	8
	2.1.2.	Startingandassigningmaterial 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-AirdomainandBoundary	. 16
	2.2.2.	Geometry	. 17
	2.2.3.	Mesh	. 19
	2.2.4.	Setup	. 21
	2.2.5.	Changing theAngleofattack	. 27
	2.3.	Fluent–3D-FiniteWing	. 34
	2.3.1.	Geometry	34
	2.3.2.	Mesh	. 39
	2.3.3.	Setup	. 42
	2.3.4.	CFDPost	. 47
	2.3.5.	Tecplot	. 52
	2.4.	Fluent–Internal flowthroughpipesandducts	. 57
	2.4.1.	Geometry	. 57
	2.4.2.	Mesh	of Dechnology
	2.4.3.	Setup	18241 CM
3.	Common	Problems	j Kumar Perma rificipal)
	3.1.	AutodeskAutocadcompatibilitywithAnsys	. 67
	3.2.	Thesharptrailingedgesoftheairfoils	. 67
	3.3.	GeneralMeshingProblems	. 68

Ansy	/sWorkbenc	hBasicsGuide	
	3.4.	NamedSelectionProcess	9
	3.5.	SolutionDivergence	9
	3.6.	Temperaturesolution divergence while using Energy equation	9
	3.7.	Scaling6	9
	3.8.	Hugevaluesofliftanddrag7	2
4.	Recommen	dedTopics7	2
	4.1.	DynamicandSlidingmesh7	2
	4.2.	Meshingtechniques–Gambit	2
	4.3.	FluentModels	3
	4.4.	Combining the structural loads with the aerodynamic loads7	3
	4.5.	Cables7	3
	4.6.	Composite	3
5.	UsefulLinks		4



1. Introduction

ANSYS is a finite element analysis package used widely in industry to simulate theresponse 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 theassociatedproblem-specificboundaryconditions.

Unsaved Project - Workbench			(00m	
File View Tools Units Help				
🎦 New 🚰 Open 🛃 Save 🔣 Save As 🚮 Import.	. 🖓 Reconnect 🤃 Refresh Project 🎐	Update Project G Project	Compact Mode	
Toolbox 🔹 🐺 🗶 Project Sche	matic			
Analysis Systems				
Design Assessment				
Blectric				
Explicit Dynamics				
Fluid Flow-BlowMolding (POLYFLOW)				
Fluid Flow - Extrusion (POLYFLOW)				
🖸 Fluid Flow (CFX)				
G Fluid Flow (FLUENT)				
G Fluid Flow (POLYFLOW)				
Normanic Response				
🔯 Hydrodynamic Diffraction				
🙀 Hydrodynamic Time Response				
😥 Linear Buckling				
🔘 Magnetostatic				
🔢 Modal				
Random Vibration				
📶 Response Spectrum				
🚾 Rigid Dynamics				
Shape Optimization				
5 Static Structural				
👔 Steady-State Thermal				
1 Thermal-Electric				
📷 Transient Structural				
🔃 Transient Thermal				
Component Systems Progress				▼ ₽
AUTODYN	A		B	С
🔏 BladeGen 🕴 1	Status		Details	Progress
@ CFX				
Engineering Data				
View All / Customize				

Thismanualincludestheprocedureofsolvingthe(staticstructural,Fluent)problems.

For Techno India NJR Institute of Technology **ü**an J Dr. Pankaj Kumar Porwa (Principal)

Eachoneoftheanalysissystemshasitsownprocedure.However,therearesomecommonstagesi nall ofthesystems.

🔥 Unsaved Project - Workbench	territe States	- No No.	-										- 0 <mark>- X</mark>
File View Tools Units Help													
New Copen Save Save As	Import	connect 🛛 🤁 Refresh Pr	oiect 🍠 Up	date Project		pact Mode							
Toolbox • 무 X	Project Schematic												, ф
E Analysis Systems													89 - 28
M Design Assessment													
Electric	-	A	•	1	3	*		С		-		D	
Explicit Dynamics	1 w Static	Structural	1	Rigid Dyna	mics	1	0	Magnetostatic		1	0	Fluid Flow (FLUENT)	
Fluid Flow-BlowMolding (POLYFLOW)	2 Rencine	ering Data	2	Engineerin	n Data	2		Engineering Data	1	2		Geometry	2
Fluid Flow - Extrusion (POLYFLOW)			-	S Ligheen	g Data 🗢 🖌	-	~	Lingineening Data	<u> </u>	-		Geometry	· 4
Fluid Flow (CFX)	3 🎯 Geome	etry 📽 🖌	3 1	Geometry	F 4	3	9	Geometry	F A	3		Mesh	a 1
Fluid Flow (FLUENT)	4 🞯 Model	? 🖌	4	Model	P 🖌	4	9	Model	? 🖌	4		Setup	7 🖌
Fluid Flow (POLYFLOW)	5 🎡 Setup	2	5	Setup	P 🖌	5		Setup	2 4	5		Solution	2 🖌
Marmonic Response	6 🕥 Solutio	n 🌚	6 1	Solution	2	6	6	Solution	2	6	1	Results	2
W Hydrodynamic Diffraction	7 🔗 Result		7	Results		7		Results	0	1	-	luid Flow (FLUENT	and the later
Response Response	in the second			- results			-	- recourds	· · · · ·			ING FIOW (FLOENT	/
😥 Linear Buckling	Static	Structural		Rigid Dyr	amics			Magnetostatic					
🤟 Magnetostatic													
Modal													
Random Vibration													
Response Spectrum													
Rigid Dynamics													
Shape Optimization													
Static Structural													
Steady-State Thermal													
P Thermal-Electric													
Transient Structural	///·												
Transient Thermal	•					m	_						Ň
Component Systems	Progress												→ ₽
AUTODYN		A					в				T	C	
BladeGen	1	Status				r	Deta	ils		_		Progre	ss
(1) CFX											-		
🧑 Engineering Data 👻													
View All / Customize													

Foreachtypeofproblems,theprocedurecanbecompletedbygoingthroughthetreeone byoneuntilallthecellsgetmarked with

Itishighlyrecommended tosurfonlineandhaveagood ideaaboutthe "mesh" or the "grid"

- TheimportanceofthemeshforthecomputeraidedengineeringandsimulationsoftwarelikeANSYS.
- Typesofmesh
- Howtocontrolthemesh sizeandbasedonwhat the meshshouldbe modified
- Howdoesmeshsizeaffectthequalityandreliabilityof the results?

Moreover, it is recommended to use approximation must be a second second

- Processor:i5ori7
- Ram: 32 Gbs
- Harddisk:1TB
- Goodcoolingsystem(Important)

For Techno India NJR Institute of Technology धैकज परिवाल Dr. Pankaj Kumar PORM3 (Principal)

Ansys Work bench Basics Guide

Thegeometryshouldbemadeonexternalmodelingsoftware(Solidworks,CatiaorRhino)andsave dinanindividualgeometryfilewithrecommendedextensions(Solidpartfile .sldprt,IGESfile .igsorStepfile.stl).AutodeskAutocadis **not**compatiblewith Ansys.

NOTE: This manual provides a very brief idea and introduction the Ansys applications. Themanual is made for the beginners who are working on the application for the first time. It shouldguide the student to the basics of Ansys while he can develop himself with more advancedproblemsfromreal lifeandfromonlinesources.

For Techno India NJR Institute of Technology पैकाज पोरवाली Dr. Pankaj Kumar Porwa (Principal)

- 2. Exercises
- 2.1. StaticStructural-CantileverBeam



2.1.1. ProblemsSpecifications:

Findthestressandthe strainin thecantileverbeamwhere:

- L=1m
- H=0.2m
- Load=1KNDownwards,applied on thetop rightedge.

 The material properties are:Young's modulus E = 200 GPaPoissonratio=0.3

For Techno India NJR Institute of Technology Dr. Pankaj Kumar Porwa Unst (Principal)

2.1.2. Startingandassigningmaterialproperties

** Before starting, the geometryfile of the beam should be savedinanindividualfile

**InANSYSWorkbench window:

Drag (Static Structural) to theProjectSchematicinsidethere dsquare

**DoubleClickon(EngineeringD ata) to configure and add thematerials that would be used inthe analysis along with theirproperties.

**Theshownwindowwillappe ar where a new materialcanbeadded>>(click heretoadd a new material)>> add(materialforthebeam)





The new roos ones nep							
🎦 New 📸 Open 🛃 Save 🔣 Save As	Impe	ort 🗟 Reconnect 🥃 Refresh Project	🥖 U	pdate	Project GReturn to Pro	oject 🕜 Compact M	ode 🛛 🍸 🛍
Toolbox 👻 🖣 🗙	Outline o	of Schematic A2: Engineering Data				~ ₽ X	Table: No da
Physical Properties		A	в	С	D		
2 Density	1	Contents of Engineering Data 🗦	8	3	Descripti	ion	
Isotropic Secant Coefficient of Thermal Eq	2	🗢 Material					
Orthotropic Secant Coefficient of Thermal Isotropic Instantaneous Coefficient of The Orthotropic Instantaneous Coefficient of The	3	Structural Steel		8	Fatigue Data at zero me from 1998 ASME BPV Co 2, Table 5-110.1	an stress comes de, Section 8, Div	
😭 Constant Damping Coeffident	*	Click here to add a new material					
🔁 Demping Fector (β)		3					
Linear Elastic							
Isotropic Elastidy Orthotropic Elastidy Asis stropic Elastidy							
isotropic Elastichy Yorthotropic Elastichy Anisotropic Elastichy Bisperimental Stress Strain Data Hyperelastic							
Stofrapic Elastidy Stofrapic Elastidy Adiscincpic Elastidy Elastidy Experimental Stress Strain Data Hyperelastic Plasticity	Propertie	es of Outline Row 4:				+ ₽ X	Chart: No da
Cropp Elastory Orthomopic Elastory Orthomopic Elastory Andectropic Elastory Experimental Stress Strain Data Hyperelasto: Plasticby Croep Croep	Propertie	es of Outline Row 4: A			B	• # X c	Chart: No da
Stotropic Elastody Charlotopic Elastody Anasotopic Elastody Experimental Stress Strain Data Hyperelastic Plasticky Creep Life	Proper be	es of Outline Row 4: A Property			B	×₽X C Unit	Chart: No da
Isotropic Elastoly Grinktropic Elastoly Anisotropic Elastoly Anisotropic Elastoly Hoperimental Stress Strain Data Hyperelastic Prasticity Foren Creen	Propertie	es of Outline Row 4:				* # X	0

**IntheToolbox,thematerialpr operties can be added from"Density"or"Isotropic Elasticity". Double Clicking onthementionedoptionswillope nnew fields in the outline wherethe fields have to be filled withthevaluesoftheproperties.

Note: Try to find the desiredmaterial in the "EngineeringData Source" Library beforeaddinganewmaterial.Cli ckon theicon >> select the type

of the material and the materialswillappearinalist. If yo uwantto add a material to yourprojectlist, clickon.

** After you are done withaddingallthematerialsnee dedin the project, click on "ReturntoProject"

~

** The Engineering Data fieldshould be marked with aindicating that the process ofadding materials properties hasbeendone.

 Toolbox
 - 4 ×

 Physical Properties

 Ornsity

 Isotropic Secant Coefficient of Thermal Exp

 Orthotropic Secant Coefficient of Thermal

 Orthotropic Instantaneous Coefficient of The

 Onthotropic Instantaneous Coefficient of The

 Damping Factor (β)

 Linear Elastic

 Orthotropic Elastidty

 Orthotropic Elastidty

 Anisotropic Elastidty





2.1.3. Geometry

**RightClickon(Geometry)>>I mport Geometry >> Browse >>Locate the geometry file

Note: Simple geometry can beconstructedinAnsysGeometr ywindow itself. However,complex geometry should beimported from 3D modelingsoftware like Solidworks, as ithasbeendoneinthisexercise.

** Even though after locatingthe geometry file, the field willbemarkedwith , it is stillnecessary to do the followingstep.

**Doubleclickon"Geometry" >> Chose the units used whileconstructingthegeometry files

** On the Tree Outline on theleft side >> Right Click on"Import" >> Generate. Hence,thegeometrywillappeari nthegraphics window. After thisstep, close



Select desired length u	nit:	
Meter	C Foot	
Centimeter	C Inch	
C Millimeter		
C Micrometer		
Always use proje	ct unit	
Always use selec	ted unit	
0	ĸ	
		Taph
A Fluid Flow	(ELLIENIT)	e of leci
A: Fluid Flow	(FLUENT)	e of lear
✓ A: Fluid Flow	(FLUENT)	e of lech TIZAT
A: Fluid Flow インネ XYPlane スキ ZXPlane スキ YZPlane	(FLUENT)	e of lech aj Kumal Principal
A: Fluid Flow	(FLUENT)	e of lean ITZAT aj Kuma Principal
A: Fluid Flow	(FLUENT) Delete Generate	e of rech TIZAT aj Kumal Principal

2.1.4. Model

**Doubleclickon"Model"

** On the outline window, expandthe "Geometry" tree by clicking on "+", this tree should show you allthe parts in the project (will beclearwhen therearemultiplepartsintheproject).Moreover,thetree helps in assigning differentmaterial to different parts ormanaging the contact typebetween two parts (Frictional,Frictionless,etc).

**Ontheoutlinewindow,click on"Mesh". For generating the meshwiththe defaultsize, clickon Update from the top bars. Foradvancedmeshoptions,adjustt he

šětClogsfreoméDetaDowbleeshck vnindosskí.

Note:

Thedefaultmeshisusuallyaverybasi

caridwithnoattention, "Inthe MechanicalWindow", giventothedetailsofthe on the Outline part, Right clickgeometry.Advancedmeshdet ailson "Mesh" >> Insert >> Methodcanbeaddedby choosingthe >>Automatic.Thenclickonthe geometricaldetailandinsertingb vdywhichisrepresentingthe "sizing" asitisshowninthe figure.domain. Thenclick "apply".





AnsysWorkbenchBasicsGuide Thedetailscanbechosenusingthe

selectingicons.

For Techno India NJR Institute of Technology पैकर्ज परिवाली Dr. Pankaj Kumar Perwa (Principal)

[Type text]

2.1.5. Setup

**Aftersettingthematerialandge nerating the mesh, close the"Model" window. As it is clear,the first 3 stages have beenmarkedwith indicating thatthey are completed. Move to"Setup".

** In "Setup", the loads, thesupports and the desiredsolution parameters should bedefined.Bymarkingthelocatio non the geometry and adding aforce or a support, the "Setup"stage can be considered to bedone.

**Choosetheface where thecantileverbeamisfixed by using the "Faceselectiontool"

** Add the "Fixed Support" fromthe"SupportsList".Hence,ont he"Outline" tree, the fixed supportwillbedisplayedunderthe "StaticStructural"list.







**Similarly,selectthetoprighted ge ofthebeamusingthe "EdgeSelectingTool" 1



** Add the force from the "Loads" list. In the "Details ofForce" window, change "Defined By" to "Compo nents" and then set the "Y" directionforce to be " - 1000 N" as it isshowninthefigure.

Note: The negative sign of theforce is because the force isdownwards.Alwaysmakesure you check the coordinatesystem defaults directionsbefore settingthe forces.

**Fromthe sideview,the "Graphics window" should looklikethisafterclickingon"Stat icStructural" on the "Outline"window.

D	stails of "Force"	
-	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



** To define the desired solutionparameters, click on "Solutions" and define all the para meters needed to be found. The par ameters can be chosen from the lists shown in the figure.



** After defining theinvestigationparameters,cli cký Solve

togettheresults.Tos howtheresultsofthedifferentpar ameters, use the list under"solutions" in the "Outline"window.

Note: The previous procedurecan be considered one of thesimplest static structuralproblems. Practice more byfindingsolvedproblemsonlin eand comparing your results tothegiven results.



For Techno India NJR Institute of Technology J'an J Dr. Pankaj Kumar Porwa (Principal)

2.2. Fluent-2D-Airfoil

2.2.1. Methodology -Air domainandBoundary

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

Since computer resources management is a critical issue, the easiest and the leastresourceextensivemethodismentioned 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 thematerial of the boundary (air) is representing the wing. In other words, the wing has to besubtracted from the air boundary leaving the air moving inside the boundary avoiding the gap.Thenextfigure is showing the airboundary and the subtracted airfoil.



For the 2D cases, the air domain and the airfoil subtraction should be done from themodeling software. In the 3D cases, the wing has to be constructed in the 3D modeling softwarewhilethedomainconstructionandthesubtractionprocessshouldbedoneinAnsysworkbench.

Generally, the inlet should be away from the leading edge with a distance equal to Power twiceof the airfoil chord length while the outlet should be 8 – 10 times the chord length in the bottom of the boundary should be 4 – 6 times of the chord length away from the airfoil

2.2.2. Geometry

**Thegeometryfileshouldbesa vedinanindividualfile

**InANSYSWorkbench window:

Drag (Fluid Flow (Fluent)) totheProjectSchematicinsidethe redsquare

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





Ansys Work bench Basics Guide

** Chose the units used whileconstructingthegeometry files

**OntheTreeOutlineonthelef t side >> Right Click on"Import">> Generate

**Afterthegeometryappears,cl ose the geometry modelingwindow

elect desired length	unit:
Meter	C Foot
Centimeter	C Inch
Millimeter	
C Micrometer	
Always use pro	ject unit
Always use sele	ected unit
Enable large mod	del support
[OK

A: Fluid Flow XYPlane XZPlane XZPlane	(FLUENT)
	X Delete
	alp Rename

For Techno India NJR Institute of Technology J'an J Dr. Pankaj Kumar Porwa (Principal)

2.2.3. Mesh

**Doubleclickon"Model"

**Togeneratethemesh,click

Note: Thedefaultmesh isusuallyaverybasicgridwithno attentiongiventothedetailsofth egeometry. Advancedmesh details can be added as itisexplainedbellow.

**OntheOutlinepart,Leftclickon "Mesh". Then on the "Detailsof Mesh" window Change thefollowings:

Relevance>> controls thedensityofthemeshinregion scloser tothegeometry.

<u>*</u>

UGlosdþoædtsizeDjanbliocli€k**Om**P M**esh**″nityandCurvature

- RelevanceCenter>>Fine

- MinSize>>theminimumsize of the mesh elements in meters, "In the Mechanical Window", oMthefOctlSizeparthRightahchm oize MetherneshseletmenMathod methertomatic. Thenclickonthe bodywhichisrepresentingthe - MaxSize>>equalto "Maxdo main_____Then____click apply".faceSize"





ED	efaults		1
P	hysics Preference	CFD	-
s	olver Preference	Fluent	1
T	Relevance	0	-
- 5	izing		1
U	Ise Advanced Size Function	On: Proximity and Curvature 🔻	1
R	lelevance Center	Fine	1
Ir	nitial Size Seed	Active Assembly	1
S	moothing	Medium	
T	ransition	Slow	1
S	pan Angle Center	Fine	1
Ē	Curvature Normal Angle	Default (18.0 *)	1
Ē	Proximity Accuracy	0.5	1
Ē	Num Cells Across Gap	Default (3)	1
C	Min Size	8.e-004 m	-
Ē	Max Face Size	4.0 m	
Ē	Max Size	4.0 m	-
C	Growth Rate	Default (1.20)	
N	Ainimum Edge Length	1.6276e-004 m	1
E Ir	nflation		1
- 0	CutCellMeshing		1
A	Active	No	
A	dvanced		of Technolog
- 0	Defeaturing		le of ion
P	inch Tolerance	Default (7.2e-004 m)	TIZAICA
G	Senerate Pinch on Refresh	No	Por Por
A	utomatic Mesh Based Defeaturing	Off	kaj Kuman ret
+ 5	tatistics		(Principal)

**Afterthemeshisgenerated.Ch oosetheedgechoosingtool.

**Leftclick

oneachedgeoftheboundary>>R ight click >>Create Named Selection >>Name each edge according theorientation of the model. Makesure that the inlet is named"inlet", the outlet is named"outlet", and the other 2 sides'names start with "symmetry –(addname)".

** After selecting each edgeseparately and assigning anamedselection,changethe selectiontypeto"Boxselection"a sshown.

** Hence, select the airfoil as itis shown. Then right click>>create named selection >> callit any name (avoid calling itInlet, Outlet and Symmetry), inthisexampleitiscalled"airfoilf oreasierreference.

** After doing the namedselectionstep,thetreeo utlineshould look like the



In	sert	•				
Go	То	•				
Iso Iso Iso Re	ometric View t store Default					
	ursor Mode ew ok At	;	1.500		3.000 (m)	Z.
Pr Se	lect All ppress Body de Body		/60	2.250		
- ¥ !"				A	and the second	

File Edit	View Units Tools	Help
K K K	12 · 12 12 12	
P Show	Single Select	e
Mesh 孝	Box Select	🔍 Me



[Type text]

AnsysWorkbenchBasicsGuide shownfigure. Notice all the namedselectionsarelisted.

For Techno India NJR Institute of Technology पैकज परिवाली Dr. Pankaj Kumar Perwa (Principal)

2.2.4. <u>Setup</u>

** Close the "MechanicalWindow">>Ri ghtclickon"Mesh">> Update.

**DoubleClick on"Setup"







** Tick (Double Precision)>>Chose "Parallel" and chose thenumber of processors to be 4unless if more processors arelicensed.Inthecaseyourcomp uter has less than 4processors,selectthemaximum amountofprocessorsavailable.

**Chosethe"Type"tobe:

- "PressureBased"fori ncompressibleflow

- "DensityBased" for compressible flow

**Goto"Define">>OperatingC onditions.

** Define the Static Pressure inthe operationaltitude.

** In "Models" Section >>Doubleclickon"Viscous"an dchose:

- Model:K-epsilon
- K-epsilonmodel:Realizable

- Near-Wall Treatment: Non-EquilibriumWallFunctions



** In "Materials" Section >>DoubleClickon"air">>setthe density and the viscosityPressure intheoperationaltitude.

** In "Boundary Conditions"Section >> Double Click on"Inlet" >> Change "VelocitySpecification Method" to"Components" >> Insert thevalues of the flow velocity withrespect to the coordinatesystem(Noticeitis-50becausethe free stream is in thenegativeXdirection.

**In"ReferenceValues"section >>Chose"Computefrom"tobe"i nlet" >> Insert the flowconditions at the operatingaltitude. Moreover, insert:

- Area:thereferenceareaofthewi ng (the projection area fromthetopview)

Depth:thespanofthe2Dwing

- Length:MeanAerodynamic Chordlength







**In"Solution Methods"Section >>Chose"Scheme"tobe" Coupled".

**In"Solution Controls"Section >>Clickon"Limits">>setthe"M aximum Turb. ViscosityRatio"tobe1e+20.

** In "Monitors" section >>Double click on "Residuals" >>Tick on (Print, Plot) >> on theright side, remove the ticksfromalltheparametersexc eptcontinuity. Moreover, changethe absolute criteria of thecontinuity to be 1e-6 as showninthefigure.



Scale

Compute Local Scale

Plot Renormalize Cancel Help

[Type text]

OK

** In "Monitors" section >>Double click on "Drag" >> Tickon(Printtoconsole,Plot,Wri te) >> add (.txt) to the endof the file name >> Adjust theunitvectorwhichisrepresenti ngthedirectionofthe Drag force with respect tothecoordinatesystem(Noticeit is -1 in the X direction becausethe free stream is in thenegativeXdirection).

**Dothesameprocessfor"Lift"ke eping in mind that the X andYforcevectorswillbedifferen t.

**In"SolutionInitialization"s
ection >> Chose
"HybridInitialization".

**In"RunCalculations"Section >>Settherequirednumberofite rationsand"Calculate".

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





** The solution will completewhen the convergence (error)reachestothepredefinedlimit.The final C_l and C_dvalues aretheonesinthelastline.

**Toviewthegraphicalresults,In
"Results"chose"Graphicsand
Animations" >>
Doubleclickon"Contours"or"Vec
tors"
>> Chose
therequiredspecificationsofthef
igurefrom"Options">> Display.

**Moreresultscanbedisplayedus ingCFDPostandTecplotasitwill be demonstrated in the 3Dsection.





2.2.5. ChangingtheAngleofattack

In aerospace applications, the angle of attack is an important parameter where the testsusually includeastudyoftheliftandthedragunderdifferentanglesofattack.

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

2.2.5.1. Method1-Changingtheangleofattackusingthe3dmodellingsoftware

 $\label{eq:theta} The angle of attack an air foil can be changed using the 3D modelling software as it is shown.$



This method requires starting from the geometry modelling stage going through all thesteps of Ansys Fluent (Geometry – Mesh – Setup ... etc.). However, the shape of the containingduct can be rectangular as it is clear from the figure above. This method generates accurateresults. However, ittakes longersince the whole process has to be done.

2.2.5.2. Method2-ChangingtheangleofattackfromAnsysFluentsetup

The second method of changing the angle of attack is by changing the inlet velocityvectorswherethedefinedvelocitywillhavetherequiredmagnitudeanddirection. Theadvantag eofthismethodologyisthetimesavedwherethechangingprocesscanbedoneinthe"Setup" stepofAnsy sFluentwithoutre-doing thepreviousprocesses (GeometryandMesh).



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

- Ydirection: $v_y = V_{\infty} \times sina$
- XDirection: $v_x = V_{\infty} \times \cos a$

Hence, the velocity components can be entered to the "Boundary Conditions" whereansyswillautomaticallycalculatetheresultantvelocityandangle.

Forexample, if the free stream velocity is 80 m/sand the angle of attack is 15°:

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

For Techno India NJR Institute of Technology kai Kumar Principal)

Problem Setup	Boundary	/ Conditions	arricon.	AND .	A DOT OWNER	
General Models Materials	Zone airfoil inlet					
Cell Zone Conditions	outlet	Velocity Inlet				
Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Methods Solution Controls Monitors	symmetry-	Zone Name inlet Momentum Thermal Radiation Specie Velocity Specification Method	s DPM Multipha Components	se UDS		
Solution Initialization		Reference Frame	Absolute			AXXTHILLYSS
Run Calculation		Supersonic/Initial Gauge Pressure (pascal)	0	constant	\neg	AXXXXXXXXX
Results Graphics and Animations	Phase	X-Velocity (m/s)	-77.274	constant		
Plots Reports	mixture	Y-Velocity (m/s)	20.706	constant	-	A

Note: ThevelocityinXdirectioniswitha(-)sign.Thisisduetothefactthatthegeometryhasbeendesignedin suchorientation wherethefreestreamhastobeinthenegativeXdirection.

Note:Aftereachchangeintheangleofattack,the"ReferenceValues"shouldbeupdatedtocomputefrom"I nlet"asitisshown.

Problem Setup	Reference Values				
General Models Materials	A				
Phases	Reference Values				
Cell Zone Conditions Boundary Conditions	Area (m2)	0.05			
Mesh Interfaces Dynamic Mesh	Density (kg/m3)	1.225			
Solution	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			
Results	Temperature (k)	288, 16			
Graphics and Animations Plots Reports					
	Viscosity (kg/m-s)	1.7894e-05			
	Ratio of Specific Heats	1.4			

 $\label{eq:alpha} A fterup dating the ``Reference Values'' it can be noticed that the velocity has been automatically calculated to be the resultant velocity.$

For Techno India NJR Institute of Technology

Since the velocity has been defined using the components, the monitors of the lift and the draghast obeset to read the required force components.



Asit 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 berepresented by the following equations:

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

Hence, the coefficients of XandYhave to been tered to the "Monitors" section where:

- ForLift:(X:-sina,Y:cosa)
- ForDrag:(X:cosa,Y:sin a)

Forexample, for freestream velocity is 80 m/s and the angle of attack is 15°:

- For Lift:(X: -sin15=-0.259,Y:cos15=0.966)
- For Drag:(X:cos15=0.966,Y:sin15=0.259)

For Techno India NJR Institute of Technology kaj Kumar (Principal)

Problem Setup	MONITORS		J.
General	Residuals, Statistic and Force M	Ionitors	
Models Materials Phases	Residuals - Print, Plot Statistic - Off Drag - Print, Plot, Write	Lift Monitor	
Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh	Lift - Print, Plot, Write	Options Verint to Console Verint to Console	Wall Zones
Reference Values	Options	Window	
Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	Print to Console Plot Window Curves Window Winte File Name Cd-history.txt Per Zone Force Vector X Y	3 Curves VWrite File Name Id-history.txt Per Zone Force Vector X Y Z 1252 0.966	
	0.259	Plot Clear Cancel Help	Cancel

Note:All the coefficients in X direction havebeen inversed since the geometry has beendesigned in such orientation where the free stream has to be in the negative X direction, as ithasbeenmentionedpreviously.

For multiple angles of attack, the same process has to be repeated for each angle. It ispreferredtoconstructatablewiththerequiredvelocityvectorsandthemonitoringcoefficientsonMicros oftExcel asitisshownbelow.

				Monitors			
		BoundaryConditions>Inlet		D	rag	Lif	t
Velocitym/s	AOA	inletX	inletY	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	Tech 0.259
	20	75.175	27.362	0.342	or Teci0.940a	0.940	a1-0-842

Dr. Pankaj Kumar Porwal (Principal)

Theprocesswhichhastoberepeatedforeachanglecanbeconcluded infewsteps:

- 1. BoundaryConditions>>Inlet>>Edit>>Changingthevelocityvectors
- 2. ReferenceValues>>Computefrom>> Inlet
- 3. Monitors>>LiftandDragmonitors

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 approachingthewingwhichcausesinaccurateresults.



The best way to solve this problem is changing the enclosure duct from a rectangularshape 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 thewhole model is included inside the C shaped inlet. It is recommended to keep the model ascloseraspossibletotheleadingpartoftheC shapedinlet.



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. Comparisonbetweenthetwomethods

	Method1	Method2
Enclosureshape	Rectangular	Cduct
Anglechangeusing	3Dmodellingsoftware	FluentSetup
Timerequired	More	Less
Resultsaccuracy	More accurate	Lessaccurate(Acceptable)
DomainSize	Smaller	Bigger
Changingsteps	Thewholeprocedure	Boundaryconditions– ReferenceValues-Monitors
Calculating Velocitycompon ents	Notrequired	Required
LiftandDragMonitors	Lift: (X:0,Y: 1) Drag:(X:1,Y:0)	Lift: (X:-sina,Y:cosa)Drag:(X: of Technic cosa, Y:sina) o India NJR Institute of Technic
		Cich J Dr. Pankaj Kumar I (Principal)

2.3. Fluent-3D-FiniteWing

2.3.1. Geometry

**Thegeometryfileshouldbesa vedinanindividualfile



**InANSYSWorkbench window:

Drag (Fluid Flow (Fluent)) totheProjectSchematicinsidethe redsquare



**RightClickon(Geometry)>>I mport Geometry >> Browse >>Locate the geometry file


** Open Geometry by doubleclicking on "Geometry". Chosethe units used whileconstructingthegeometry files

**OntheTreeOutlineonthelef
t side >> Right Click
on"Import">> Generate

**Afterthegeometryappears,g oto: Tools>> Freeze

** Click on the axis whichreorientstheviewtothe sideview. InthiscaseitisXaxis.

elect desired length u	nit:
Meter	C Foot
Centimeter	C Inch
Millimeter	
C Micrometer	
Always use proje	ct unit
Always use selec	ted unit
Enable large mode	I support
C	Ж







** On the Tree Outline, click onthe side view plane. In this caseitisYZPlane

**OntheTreeOutlinewindow,Ch ose "Sketching">> Draw >>draw a rectangle whichrepresents the side view of thetestdomainorthespace.

** Fix the dimensions using"Dimensions" option on thesketchingtoolbaronthelef tside. The side view of thedomain should look like thefigure.

Note: In the C-Duct case, thecircle has to be drawn first,followed by the rectanglestartingexactlyfromth emiddleof the circle. The unwantedpartshavetobe trimmed.

**Afterfixingthedimensions,

clickon

tosetthe



L'IUT	A
 > Line ⊘ Tangent Line ⊘ Line by 2 Tangents ∧ Polyline ○ Polygon □ Rectangle ◇ Rectangle by 3 Points ⊘ Oval 	
Circle Circle by 3 Tangents Circle by 3 Tangents	
Circle Circle by 3 Tangents Arc by Tangent Modify	
Circle Circle by 3 Tangents Arc by Tangent Modify Dimensions	•
© Circle Circle by 3 Tangents Arc by Tangent Modify Dimensions Constraints	•



AnsysWorkbenchBasicsGuide *depth ofthedomain.*

For Techno India NJR Institute of Technology Cr. Pankaj Kumar Perwal (Principal)

**Afterthesesteps,theTreeO utline should look like theshownfigure.

**Aftergettingthepreviouso utline, go to Tools >>Enclosure.

**Changethe"Shape"to"userde fined"

	JUCZ
Extrude	Extrude2
Base Object	Sketch1
Operation	Add Material
Direction Vector	or None (Normal)
Direction	Normal
Extent Type	Fixed
FD1, Depth	(>0) 15
As Thin/Surface	e? No
Merge Topolog	gy? Yes







[Type text]

** For the cell "User DefinedBody",Chose"solid"fr omthe

**Click ^{Generate}. Theresulted geometryshouldlooklikethe shownfigure.

**Goto"Create">>Boolean.

**Onthedetailsviewontheleftbot tomcorner,change "Operation"from"unite"to "Subtract".



[Type text]

Tool Bodies Unite

Intersect

**Click Generate TheTree Outline shouldlookliketheshownfigure. Noticethatthereis only 1 Part, 1 Body while thegeometry "spiroid" has beensubtractedfromthe domain "solid".Moreover, "Solid"doesn otnecessarilymeanthatitissoli d body, it is still thesurroundingair.

However, Ansys calls thegenerateddomain"solid".T

2.3.2. Mesh

**CloseGeometry.Doubleclicko n"Mesh".

**Inthe"MechanicalWindow",o n the Outline part, Right clickon"Mesh">>Insert>>Metho d >>Automatic.Thenclickonthebo dy which is representing thedomain.Thenclick"apply".







**OntheOutlinepart,Leftclickon"Mesh".The n on the "Details of Mesh" windowChangethefollowings:

- Relevance>>controlsthe densityofthemeshinregionscloser tothegeometry.

- Useadvancedsizefunction>>OnPr oximityandCurvature

- RelevanceCenter>>Fine

- MinSize>>theminimumsizeofthemeshele mentsin meters

- MaxfaceSize>>themaximumsizeoftheme sh elementsinmeters

- MaxSize >>equalto"Maxface Size"

- AutoMeshBasedDefeaturing>>OffT

henclick

**Afterthemeshisgenerated.ChoosetheFace choosingtool.

**Leftclickoneach

faceinthegeometry>>Right click >> Create NamedSelection>>Name eachfaceaccordingtheorientation of the model. Make sure thatthe inlet is named "inlet", the outlet isnamed "outlet", and the other 4 sides'namesstartwith"symmetry– (name)".

Defaults	
Physics Preference	CFD
Solver Preference	Fluent
Relevance	0
Sizing	
Use Advanced Size Functio	On: Proximity and Curvature
Relevance Center	Fine
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Slow
Span Angle Center	Fine
Curvature Normal Angl	e 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	
CutCellMeshing	
Active	No
Advanced	
Defeaturing	
Pinch Tolerance	Default (7.2e-004 m)
Generate Pinch on Refresh	No No
Automatic Mesh Based De	featuring Off





** After selecting each faceseparately and assigning anamedselection,changethe selectiontypeto "Boxselection"a sshown.

** Hence, go to the side viewand select the model or thewing as it is shown. Then rightclick>>createnamedselec tion >> call it any name

(avoidcalling it Inlet, Outlet andSymmetry).

** After doing the namedselectionstep,thetreeo utlineshould look like the shownfigure. Notice all the namedselectionsarelisted.

** Close the "MechanicalWindow">>Ri ghtclickon"Mesh">> Update.

File Edit	View Units Tools	; Help	
E XXZ	12 - 12 E		4
P Show	Single Select	e	I
Mesh 👙	Box Select	0	Me





2.3.3. Setup

**DoubleClick on"Setup"

** Tick (Double Precision)>>Chose "Parallel" an dchoosethenumber of processors to be 4unless if more processors arelicensed.Inthe case yourcomputer doesnothave4processors, choose themaximum number of availableprocessors.

**Chosethe"Type"tobe:

- "PressureBased)fori ncompressibleflow

- "DensityBased" for compressible flow





**Goto"Define">>OperatingC onditions.

**DefinetheStaticPressureinth e operationaltitude.

** In "Models" Section >>Doubleclickon"Viscous"an dchose:

- Model:K-epsilon
- K-epsilonmodel:Realizable

- Near-Wall Treatment: Non-EquilibriumWallFunctions





** In "Materials" Section >>DoubleClickon"air">>setthe density and the viscosityPressure intheoperationaltitude.

** In "Boundary Conditions"Section >> Double Click on"Inlet" >> Change "VelocitySpecification Method" to"Components" >> Insert thevaluesoftheflowvelocitywit hrespect to the coordinatesystem.

**In"ReferenceValues"section >>Choose"Computefrom"tob e "inlet" >> Insert the flowconditions at the operatingaltitude.Moreover, insert:

- Area: the reference area of thewing(theprojectionarea)

- Length:MeanAerodynamic Chordlength







**In"Solution Methods"Section >>Chose"Scheme"tobe" Coupled".

** Change the "Momentum", "TurbulentKineti cEnergy"and "TurbulentDissipa tionRate"to "SecondOrderUpwi nd"

**In"Solution Controls"Section >>Clickon"Limits">>setthe"M aximum Turb. ViscosityRatio"tobe1e+20.

** In "Monitors" section >>Double click on "Residuals" >>Tick on (Print, Plot) >> on theright side, remove the ticksfromalltheparametersexc eptcontinuity. Moreover, changethe absolute criteria of thecontinuity to be 1e-6 as showninthefigure.





** In "Monitors" section >>Doubleclickon "Drag">>Tickon (Print to console, Plot,Write) >> add (.txt) to the endof the file name >> Adjust theunitvectorwhich isrepresenting the direction ofthe Drag force with respect tothe coordinatesystem.

**In"SolutionInitialization"s
ection >> Chose
"HybridInitialization".

**In"RunCalculations"Section >>Settherequirednumberofite rationsand"Calculate".

** The process can be paused,stoppedandsaved.Toco ntinuesolving the problem, the setupshould be started from"Solutions" in the main Ansyswindow.

**Theresultscanbefoundfromth esamewindowasitwasshown in the 2D case. Moreoptions can be found in CFDPost.

Pro	blem Setup		Monit	ors	
G	ieneral		Residua	ls, Statistic	and Force Monitors
M	lodels laterials		Residu Statist	als - Print, F ic - Off	Not
P	nases Drag N	Ionitor	Drag -	Print, Plot,	Write
	Options				Wall Zones
Sc	Print t Plot Windov 2 Write File Nar cd-his Per Zo	o Console	rves)	Axes	Wing
	Force Vecto	or			
i	x	Y	1	7	-



[Type text]

2.3.4. CFDPost

**Close"Setup".Doubleclickon"R esults".

** From the outline tree, theparts can be displayed orhidden. A
1 String Fluid Flow (FLUENT)
2 Second Seco







[Type text]

**Selecttheorientationofthepl anewhere:

- Method: select which planewill be parallel to the newconstructedplane(Inthisc aseitisYZplane).

- X:Thedistancefromtheoriginto the plane. If the origin set tobe at the wing root in the 3dmodelling geometry, then Xmeans the span wise distancefromthewingroot.

Afterfixingthesettings,click" Apply". The plane has beenconstructedasitisshown.

**Ontheupperbar,

1. Velocityvectors

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

3. Streamlines

**Todisplaytheproperties,thePl ane has to be selected as"Location".



** In the condition of the 3Dstreamlines,ithastobedefine dto start from "Inlet". In somecases, a custom plane has to beconstructed to define it as astarting plane of thestreamlines. This is useful whenit is needed to display thestreamlines over a specificregion.

** For example, it is noticed inthe first figure that thestreamlines have been startedfromtheinletwhichledtoc overupthewholedomainareawit h thestreamlines.

**Inordertodefinethestartingoft hestreamlinestobeexactly projected on the wingarea;

- Location>>UserSurface

- Method>>Transformed Surface



**Thesurfaceofthewingwillbe copied and transferredforward to use it as a startingplane ofthestreamlines:

- SurfaceName:Wing

- Activate"Transition">>Movet he plane to the forwarddirection (In this case it is Zdirection).

- Activate"Scale" and use factor of 1.02 (This step is to ensure that the starting plane is a littlewider than the original wingarea. Hence it will be ensured that the streamlines will becovering the whole model without gaps on thes ides.

** Create "Streamlines" usingthe "User Surface" for "Startfrom".Itisnoticedthattheli neshave been refined for a bettervisualization.





** A plot presenting the pressurecoefficientdistributionovert hewingsurfacecanbeplotted:

- Calculators>>MacroCalculators>>M acro:(CpPolarPlot)

- BoundaryList:theobjectwherethepr essure distributionhastobe investigated.Inthiscaseitis"wing".

- Slice Normal: to calculate thepressuredestitutionoveranairfoil,th ewing has to be sliced at a specific span.The axis going through the span is thespan wise axis which is normal to theslice.Inthiscase it isXaxis.

- SlicePosition:thedistanceoftheslicefro mtheorigin

- Plotaxis:thedirectioninwhichtheCpvar iation is investigated (the chordwise direction). In this case it is Zdirection.

**Chose"Calculate"then"ViewR eport".

Note: the Y axis is oriented in a waywhere it shows the upper surface at thetop which leads to the fact that the YaxishasnegativevaluesofCpatthetopan dthepositivevaluesatthebottom.



2.3.5. Tecplot

More results can be presented ** usingTecplot.Inordertoimportthesoluti ondatatotecplot:

- Open"Solutions"

- As it was explained in Ansys, planeshave to be constructed to show theresults. Hence, the plans have to beconstructed before exporting thesolutiondata becauseplanesconstructed using tecplot cannotrepresentthesolutiondataimp ortedfromAnys.

- To construct a new plane: Surface >>Plane

- In

"Options": Pointand Normal: allows theus ertoassignapointandanaxis as references for creating theplane.

- In"Points":entertheposition ofthepoint through which the plan will bepassing

- In"Normal":enter thedirectionvector for the axis which the plane hastobenormalto.

Note:

Theplanecanbecreatedwithanangle with respect to the axis by entering the direction vectors intomorethanonefield.





be text]

Solution

AnsysWorkbenchBasicsGuide - Click"Create"

For Techno India NJR Institute of Technology Dr. Pankaj Kumar Perwa (Principal)

** In order to display the createdplane:

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

Note: All the planes must becreatedbeforeexportingth esolutiondata

**Inordertoexportthesolutionda tatotecplot:

- File>>Export>>SolutionData





**InTecplot:

File>>LoadDataFile(s)>>T
 ecplotDataLoader

- Changethe"InitialPlotType"to:3 DCartesian

- The model gets imported in adifferentorientation.Hence,ithast o be rotated using thecoordinators controllers shown inthefigure.



**Afterreorientingthegeometryto therequired position:

- Chose"Streamtraces":

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

** Click on the sign showed in thefigure. This tool allows the user todrawalinewherethestreamlinesc overs all the area passed by thedrawn line. Hence, the user cancontrolthe densityofthe lines. Moreover,theconcentrationoftheli nes can be focused on a specificregion by drawing more than onelineat thatregion.







2.4. Fluent-Internalflowthroughpipesandducts

2.4.1. Geometry

**Thegeometryfileshouldbesa vedinanindividualfile



**InANSYSWorkbench window:

Drag (Fluid Flow (Fluent)) totheProjectSchematicinsidethe redsquare



**RightClickon(Geometry)>>I mport Geometry >> Browse >>Locate the geometry file



** Open Geometry by doubleclicking on "Geometry". Chosethe units used whileconstructingthegeometry files

**OntheTreeOutlineonthelef
t side >> Right Click
on"Import">> Generate

** The duct geometry willappear. The inlets and theoutletshavetobedefineda ssurfaces:

- Concepts>>SurfacesfromE dges

- Choosetheedgesoftheinletan dtheoutlet

- Click"Apply">>Generate



** After defining the surfaces,thewholeducthastobed efinedtobefilledwith material:

- Tools>>Fill

**In"DetailsofFill1":

- Extraction Type:ByCaps
- PreserveSolid:

* Choose "Yes" if the outers urface of the duct (the wall of the duct) is needed for the analysis. For example, if there isheat 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 ductwall is not needed. This savestheresourcesneededtopro cessthemesh ofthewall.

In this case, the wall is notrequired; hence, "No" will bechosenfor"PreserveSolids"> >Generate.

** It can be noticed that theinternal shape of the duct hasbeendefinedasthematerial



De			ą
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

**ClosetheGeometryDesignModular>>Do ubleClickon "Mesh".

OntheOutlinepart,Rightclickon "Mesh".Then on the "Details of Mesh" windowChangethefollowings:

- Relevance>>controlsthedensityofthem eshinregionsclosertothegeometry.

- Useadvancedsizefunction>>OnCu rvature

- Relevance Center >>

FineThenclickenerate Mesh

Note:Sinceboundarylayerisimportantin theinternalflowcases.Formoreaccuratestu dy of the boundary layer, whether it isinternalflowor externalflow(For example the boundary layer overthe Close Geometry. Double click surface of thewing), "Inflation" hastobe on Mesh createdwhicharrangesmorerefinedmesh fortheboundarylayerregion:

- Rightclickon"Mesh">>Inflation

**Chotse "Meshevisal Wiedne wholebody on Mesh">>Insert, Rightclick on "Mesh">>Insert>>Method -Choose all the faces except the inlet and >> Automatic. Then click on the outlet to be the "Boundary">>Apply body which is representing the domain the public fit poly of the option and Number of layers are upto the user)

•		A	
1	0	Fluid Flow (FLUENT)	
2	07)	Geometry	1
3		Mesh	1
4		Setup	P .
5	1	Solution	?.
6	1	Results	?





**Thedifferencecanbenoticedw here the mesh is refined afterusinginflation(Right).



Face

**Afterthemeshisgenerated.Ch oosetheFacechoosingtool.

**Leftclick oninletsurface >>Rightclick>>CreateNamedS election>>Type"Inlet"

-Dothesamefor"Outlet"

** After doing the namedselection step, the tree outlineshould look like the shownfigure.Noticetheinletan dtheoutletarelistedand marked.

** Close the "MechanicalWindow">>Ri ghtclickon"Mesh">> Update.



2.4.3. Setup

**DoubleClick on"Setup"

** Tick (Double Precision)>>Chose "Parallel" and chose thenumber of processors to be 4unless if more processors arelicensed.Inthecaseyourcomp uter does not have 4processors, then choose themaximumnumberofprocesso rsavailable.

**Chosethe"Type"tobe:

- "PressureBased)fori ncompressibleflow

- "DensityBased" for compressible flow





** In "Models" Section >>Doubleclickon"Viscous"an dchose:

- Model:K-epsilon

- K-epsilonmodel:Realizable

- Near-Wall Treatment:EnhancedWall Treatment

** More information aboutFluentModelscanbefoun don

<<<u>http://aerojet.engr.ucdavis.edu/fluent</u> help/html/ug/node1336.htm>>

** In "Materials" Section >>Double Click on "air" >> set thedensity and the viscosity. Morematerials can be added from"FluentDatabase".





** In "Boundary Conditions"Section>>DoubleClick on"Inlet" >> Change "Velocity SpecificationMethod"to"Magnitud e,NormaltoBoundary" >> Insert the inlet flowvelocity

** In the "Turbulence" section,enterthe"TurbulentIntensit y"and"Hydraulic"Diameter"ofthein let.

Note: Turbulent Intensity andHydraulic are well knownparametersinfluiddynamics. Bothof them can be calculated usingsimple formulas. The formulas caneasily foundonline.

<<<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 "tobe"Coupled".

** Change the "Momentum", "TurbulentKineti cEnergy"and "TurbulentDissipa tionRate" to "SecondOrderUpwi

ana Nama		
inlet		
Momentum Thermal Radiation Species DPM Multip	ohase UDS	
Velocity Specification Method Magnitude, Norr	mal to Boundary	•
Reference Frame Absolute		-
Velocity Magnitude (m/s) 10	constant	•
Supersonic/Initial Gauge Pressure (pascal)	constant	*
Turbulence		
Specification Method Intensity and Hyd	Iraulic Diameter	-
Turbulen	at Intensity (%) 10	
Hydrau	lic Diameter (m)	

Problem Setup	Solution Methods	
General	Pressure-Velocity Coupling	
Models Materials Phases	Scheme Coupled	•
Cell Zone Conditions Boundary Conditions	Spatial Discretization	
Mesh Interfaces	Gradient	
Dynamic Mesh Reference Values	Least Squares Cell Based	•
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 Plots Reports	Transient Formulation	

[Type text]

nd"

For Techno India NJR Institute of Technology Dr. Pankaj Kumar Perwa (Principal)

[Type text]

**In"Solution Controls"Section >>Clickon"Limits">>setthe"M aximum Turb. ViscosityRatio"tobe1e+20.

** In "Monitors" section >>Double click on "Residuals" >>Tick on (Print, Plot) >> on theright side, remove the ticksfromalltheparametersexc eptcontinuity. Moreover, changethe absolute criteria of thecontinuity to be 1e-6 as showninthefigure.

**In"SolutionInitialization"s
ection >> Chose
"HybridInitialization".

Problem Setup	Solution Controls			
General	Courant Number			
Models Materials	200			
Phases	Explicit Relaxation Factors			
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)			
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			
	Default			
	Equations Limits Advanced			

Problem Setup	Monitors			
General	Residuals, Statistic and Force Monitors			
Materials	Residuals - Print, Statistic - Off	Plot		
Monitors	2	100		
	Equations			
Console	Residual	Monitor C	heck Converge	ence Absolute Criteria
	continuity	7		1e-06
	x-velocity			0.001
s to Plot	y-velocity			0.001
	z-velocity			0.001
	Residual Value	s		Convergence Crit
Store	Normalize		Iterations 5	absolute
	Scale	Local Scale		
OK	Plot Renon	malize Ca	ancel	Help
		Solutio	n Initiali	zetion
Problem	Setup	Solutio	ii iiiuaii	20000
Gener	al	Initializat	ion Method	s





**In"RunCalculations"Section>>Settherequir ed number of iterations and"Calculate".

** The process can be paused, stopped andsaved.Tocontinuesolvingtheproblem,thes etup should be started from "Solutions" inthemain Ansyswindow.

** The results can be found from the samewindowasitwasshowninthe 2Dairfoilcase.More options can be found in CFD Post as itwasshownin3D – Finitewingcase.





3. CommonProblems

3.1. AutodeskAutocadcompatibilitywithAnsys

The 3d models constructed in Autodesk Autocad can be imported to Ansys if saved inIGES format. However, in the case of the multiple bodies, Ansys fails to define the contact typesontheboundaryelements.

This problem has been solved recently in the latest version of Ansys. However, if olderversions are used, it is better to construct the models usingSolidworks,Caita orRhinotoensurethattherewillbenogeometricalimportingproblems insomelaterstep.

Thesharptrailingedgesoftheairfoils 3.2.

The airfoils are usually constructed using the airfoil coordinates. The generated airfoilsusually haveverysharptrailingedgesasitis showninthefigurebelow.



The sharp trailing edge causes difficulties while meshing which might lead to the failureof generatinga good quality mesh. Since sharpedges cause sudden bending in the gridstructure, the quality has to be sacrificed to generate a grid which fits the airfoil. Moreover, this

Zall
problem can cause a failure ingenerating 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 orthebottomsurfaces.



3.3. GeneralMeshingProblems

Most of the meshing problems can be solved using custom sizing. When Ansys shows amesh failure due to a problematic geometry alert, the geometry can be displayed by (Right clickonthemessagefromthealertswindow>>ShowproblematicGeometry.Selectingtheproblematic component whether it is an edge or a face and assigning a cell size which is smallerenough to cover the details of the problematic geometry is the easiest way to solve the problemwithoutchangingthegeometry.

In some cases it is recommended to create a refinement for the mesh at certain points orlocations. For example, the leading edge of a wing if a type of micro vortex generators has beeninstalled 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:Bodyofinfluence>> Chosethesolidwhichis coveringthedetailedgeometry).

However, insome 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 of Technology solving process where the solution will not converge to the required margin of error.

3.4. NamedSelectionProcess

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

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 thewallasawall without "noslipcondition" or aboundary layer.

3.5. **SolutionDivergence**

Solutiondivergenceisadirectindicatorofthepoorqualityofthemesh.Whendivergenceisdetect ed, the mesh has to be refined or reconstructed with new setting. Mostly:

- Smaller(MinSize)
- Higher(Relevance)
- Customsizingforedgesandfaces
- Inflationlayerforbetterstudyof the boundarylayer
- Simplifiedgeometry
- Widerdomain

TemperaturesolutiondivergencewhileusingEnergyequation 3.6.

When energy equation is being used, an unrealistic exit temperature could force thesolution to accelerate \ decelerate the flow out of proportion leading to a "temperature solutiondivergence". Use common sense and experience when setting the initial guess for inlet and exittemperatures(only whenenergyequationis on, evenifthere is no combustion).

3.7. Scaling

(Principal) When the model size is too big and the calculation process is too time consuming, it isrecommended to scale down the model in order to reduce the needed resources. However, asper the flow similarity conditions, the boundary conditions have to be calculated to match thenewscaledmodel.

For Techno India NJR Institute of Technology

पैकाज पोरवाली Dr. Pankaj Kumar Porwa

Accordingtothetheoryofflowsimilarity, thetwoconditionswhich needtobesatisfiedare:

- Geometricsimilarity-Thegeometriesbodiesneedtobesimilar
- Dynamicsimilarity-

ThesimilarityparametersbasedonwhichotherflowparameterswillbecalculatedareReynolds numberandMachnumber.

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 sameMach number and Reynolds number for the two geometries, and also using the initial parametervaluesfortherealgeometry,theparametersthatwerere-calculatedare:

- Density
- Velocity
- ViscosityCoefficient
- Pressure
- Temperature

Thefollowingare the calculations that we redone to compute the new parameter values.

 $\label{eq:Forconvenience} For convenience, the temperature T_2 = 288.2 K. \ The other given parameters for the real case are:$

p ₁	0.28852kg/m3
V_1	237m/s
T ₁	217K
Τ2	288.2K
<u>C1</u> C2	3
μ1	4.7292×10 ⁻⁵ kgm ² /sec
	For Techno India non Tizal Cri

Dr. Pankaj Kumar Porwa Uanst (Principal)

EquatingMachnumber,

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

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

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

$$M_{2} = M_{1} = \frac{237}{\sqrt{(1.4x28x217)}} = 0.803$$

EquatingReynoldsnumber,

1

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

$$p_{1}V_{1}C_{1} = p_{2}V_{2}C_{2}$$

$$\mu_{1} \qquad \mu_{2}$$

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

$$\frac{p_{2}}{p_{1}} = \frac{V_{1}C_{1}}{V_{2}C_{2}} \times \sqrt{\frac{T_{2}}{T_{1}}} = \frac{237x3}{273.033} \sqrt{\frac{288.2}{217}} = 3$$

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

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

$$Re = \frac{p_{1}V_{1}C_{1}}{\mu_{1}} = \frac{(0.28852x237x2.61)}{0.000049272} = 3.662 \times 10^{6}$$
For Technol India NJR Institute of Technology
$$_{2} \mu = \frac{p_{2}V_{2}C_{2}}{Re_{2}} = 5.674 \times 10^{-5} \text{ kgm}^{2}/\text{sec}$$
For Technol India NJR Institute of Technology
$$CT_{2} = \frac{p_{2}V_{2}C_{2}}{Re_{2}} = 5.674 \times 10^{-5} \text{ kgm}^{2}/\text{sec}$$

[Type text]

3.8. Hugevaluesofliftanddrag

In some cases, the results show very huge or very small values for lift and drag eventhough the mesh has been refined and it can be considered as sufficient grid. Hence, theproblemcanbemostlyin theboundaryconditions, thereference values or the monitors.

In the reference values, the area and the length have to be defined accurately. The areaistheprojectionareaofthemodelwhilethelengthisthelengthofthemodel.Moreover,theinlettempe raturehastobedoublechecked.

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

Finally, the reference values have to be updated to be computing from the inlet aftereachchangeinanyoftheparameters.

4. Recommended Topics

The recommended topics are basically the topics or the problems which has not beenexplainedorcoveredinthemanual.Sincesometopicscanbeconsideredasadvancedtopics,a lot of research and troubleshooting will be needed to get the correct and reliable procedure ofsolvingsuchproblems.

4.1. DynamicandSlidingmesh

Dynamic andsliding meshes are types ofgrids where the geometry canchange itsshape or condition while running the calculations. For example, a wing flap changing its angle oracarspoilerchangingitsposition. For Techno India NJR Institute of Technology

4.2. Meshingtechniques-Gambit

Ansys uses ICEM meshing as a default meshing tool for all its products. However, it isrecommendedtocarryonastudyofcomparingthemeshingtechniquesandthequalitybetweenICEMa ndtheothermeshingtools likeGambit.

[Type text]

Cichat Citanat Contrology Dr. Pankaj Kumar Porwa

(Principal)

4.3. FluentModels

Viscous models are used mostly for the aerospace related studies. However, there areother 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. Combiningthestructuralloadswiththeaerodynamicloads

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 aerodynamicand the structural performance of an aircraft. However, the study will need very good computingresources.

4.5. Cables

Modellingcables in Ansys has to be investigated.Since creating an actual cable in the3d modelling software and generating the mesh for such cable is very resources consumingmethodology, an alternative way has to be found.For example, replacing the cable with aspring.

4.6. Composite

ModellingcompositematerialsinAnsysisawelldemandedtopic.Eventhoughthereisa 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 ACPlibraryisaviabletopicofresearch.

For Techno India NJR Institute of Technology पैकज परिवाल Dr. Pankaj Kumar Porwa (Principal)

5. UsefulLinks

- Brief aboutmesh andgridtypes

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

- AnsysModellingandMeshingGuide

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

<u>f</u>

- Fluent6.3userguide:

http://aerojet.engr.ucdavis.edu/fluenthelp/index.htm

- FluentModelsDetails http://aerojet.engr.ucdavis.edu/fluenthelp/html/ug/node1336.htm

- CFDanalysisof VehicleAerodynamics http://www.youtube.com/watch?v=dZR7Wi70Vec

- CFDanalysisofVehicleAerodynamics(CFXnotFluent) http://www.youtube.com/watch?v=6adO0mv-eWw

For Techno India NJR Institute of Technology पैकज परिवाली Dr. Pankaj Kumar Porwa (Principal)



[Type text]

[Type text]