16ME77 – COMPUTER AIDED SIMULATION AND ANALYSIS LABORATORY MANUAL

(FOR SEVENTH SEMESTER B.E MECHANICAL ENGINEERING STUDENTS)



DEPARTMENT OF MECHANICAL ENGINEERING

REV : 02 JUNE: 2021



P.S.R ENGINEERING COLLEGE, SIVAKASI – 626 140. (An Autonomous Institution) DEPARTMENT OF MECHANICAL ENGINEERING (2021 - 2022)



1 | P a g e

P.S.R. ENGINEERING COLLEGE, SIVAKASI – 626 140. (An Autonomous Institution) DEPARTMENT OF MECHANICAL ENGINEERING



161ME77-COMPUTER AIDED SIMULATION & ANALYSIS LABORATORY MANUAL

For Seventh Semester B. F. Mechanical Engineering Students (2021-2022)

REV: 02 JUNE: 2021

7/2021 Prepared by

Dr.P. Shenbaga Velu ASP/MECH Dr. Srinivasagam Ramesh ASP/MECH

minimi

Approved by HOD/MECH

Head Of the Department Department of Mechanical Engineering P.S.R. Engineering Collage Sevalpatti, Sivakasi - 626 140

2 | P a g e

Ex.No	Date	Name of the Exercise	Page.No	Staff Initial

INDEX

P.S.R. Engineering College Vision & Mission Statement

<u>Vision</u>

• To contribute to the society through excellence in technical education with societal values and thus a valuable resource for industry and the humanity

Mission

- To create an ambience for quality learning experience by providing sustained care and facilities
- To offer higher level training encompassing both theory and practices with human and social values
- To provide knowledge-based services and professional skills to adapt tomorrow's technology and embedded global changes

Department of Mechanical Engineering Vision & Mission Statement

<u>Vision</u>

• To provide broad-based education and training in mechanical engineering and its applications to enable the graduates to meet the demands in a rapidly changing needs in industry, academia and society

<u>Mission</u>

- To impart high quality technical education and training that encompasses both theory and practices with human and social values
- To equip the students to face tomorrow's technology embedded global changes
- To create, explore, and develop innovations in mechanical engineering research

Department of Mechanical Engineering Programme Specific Outcomes

- PSO 1 Apply the concepts of mathematics and science in mechanical systems
- PSO 2 Design and analyze components and systems for mechanical engineering applications
- PSO 3 Synthesis data and technical concepts for application to mechanical engineering software
- PSO 4 Apply manufacturing and management practices in industries

Programme Outcomes of Mechanical Engineering

- 1. **Engineering Knowledge:** Apply the knowledge of mathematics, science, engineering fundamentals, and an engineering specialization to the solution of complex engineering problems.
- 2. **Problem Analysis:** Identify, formulate, review research literature, and analyze complex engineering problems reaching substantiated conclusions using first principles of mathematics, natural sciences, and engineering sciences.
- 3. **Design/ Development of Solutions:** Design solutions for complex engineering problems and design system components or processes that meet the specified needs with appropriate consideration for the public health and safety, and the cultural, societal, and environmental considerations.
- 4. **Conduct Investigations of Complex Problems:** Use research-based knowledge and research methods including design of experiments, analysis and interpretation of data, and synthesis of the information to provide valid conclusions.
- 5. **Modern Tool Usage:** Create, select, and apply appropriate techniques, resources, and modern engineering and IT tools including prediction and modeling to complex engineering activities with an understanding of the limitations.
- 6. **The Engineer and Society:** Apply reasoning informed by the contextual knowledge to assess societal, health, safety, legal and cultural issues and the consequent responsibilities relevant to the professional engineering practice.
- 7. **Environment and sustainability:** Understand the impact of the professional engineering solutions in societal and environmental contexts, and demonstrate the knowledge of, and need for sustainable development.
- 8. **Ethics:** Apply ethical principles and commit to professional ethics and responsibilities and norms of the engineering practice.
- 9. **Individual and teamwork:** Function effectively as an individual, and as a member or leader in diverse teams, and in multi-disciplinary settings.
- 10. **Communication:** Communicate effectively on complex engineering activities with the engineering community and with society at large, such as, being able to comprehend and write effective reports and design documentation, make effective presentations, and give and receive clear instructions.
- 11. **Project management and finance:** Demonstrate knowledge and understanding of the engineering and management principles and apply these to one's own work, as a member and leader in a team, to manage projects and in multidisciplinary environments.
- 12. Lifelong learning: Recognize the need for, and have the preparation and ability to engage in independent and life-long learning in the broadest context of technological change.

16ME77COMPUTER AIDED SIMULATION AND ANALYSISL T P CLABOROTARY0 0 3 2

SYLLABUS

ANALYSIS (SIMPLE TREATMENT ONLY)

- 1. Introduction of ANSYS 14.5
- 2. Stress Analysis of Cantilever Beam
- 3. Stress Analysis of Fixed End with Point Load Beam
- 4. Stress Analysis of Simply Supported with UDL Load Beam
- 5. Stress Analysis of Plate with hole
- 6. Stress Analysis of an Axis Symmetric Component
- 7. Modal Analysis of Cantilever 2d Plate
- 8. Modal Analysis of Cantilever Beam
- 9. Modal Analysis of Simply supported Beam
- 10. Modal Analysis of Fixed End Beam
- 11. Harmonic Analysis of Cantilever Beam
- 12. Thermal Mixed Boundary (Conduction/Convection/Insulation)

SYSTEM REQUIREMENTS (for a batch of 30 students)

Description of Equipment

HARDWARE

Computer Server Computer System 17" VGA Colour Monitor Pentium IV processor 40 GB HDD 512 MB RAM Colour Desk Jet Printer

SOFTWARE Suitable Analysis Software C / MATLAB **Quantity Required**

1 NOS

30 NOS 1 NOS

30 Licenses 5 Licenses

37

Course				I	Progr	am O	utcor	nes (l	POs)				Pr Ou	ogram itcome	Specif s (PSC	fic)s)
Outcomes	PO1	PO2	PO3	PO4	PO5	PO6	PO7	PO8	PO9	PO10	PO11	PO12	PSO1	PSO2	PSO3	PSO4
CO1	2	3	3						2	3		2	2			2
CO2	2	1	1		2				2	2		3	3	3	1	2
CO3	2	3	3		2					2		2	3	3	2	2
CO4	2	3	3		2					2		2	3	3	2	2
CO5	2	3	3		2				2	2		2	3	3	2	2
CO6	2	3	3		2				2	1		3	2	3	3	3

1: Slight (Low) 2: Moderate (Medium) 3: Substantial (High)





P.S.R. ENGINEERING COLLEGE, SIVAKASI – 626140. (An Autonomous Institution) DEPARTMENT OF MECHANICAL ENGINEERING COURSE PLAN

SI.NO	LIST OF EXERCISE	PERIODS	CUMULATIVE PERIODS
1.	Introduction to ANSYS -14.5	3	3
2.	Stress Analysis of Cantilever beam	3	6
3.	Stress Analysis of Fixed end with point load beam	2	8
4.	Stress Analysis of Simply supported with UDL load beam	2	10
5.	Stress Analysis of plate with hole	3	13
6.	Stress Analysis of Axis-Symmetric component	3	16
7.	Modal Analysis of Cantilever 2D plate	3	19
8.	Modal Analysis of Cantilever beam	2	21
9.	Modal Analysis of simply supported beam	2	23
10.	Modal Analysis of Fixed end beam	2	25
11.	Harmonic Analysis of Cantilever beam	3	28
12.	Thermal mixed boundary (Conduction /Convection/Insulation)	3	31

EXTRA EXERCISE (Beyond the Syllabus)

1.	Stress Analysis on 2D Truss	2	33
2.	Stress Analysis in 2D Fixed beam with UDL	2	35
3.	Modal Analysis	2	37
4.	Stress Analysis on Cantilever beam-pipe element	2	39
5.	Effect of Self Weight on a Cantilever beam	2	41
6.	Application of Distributed Loads	2	43
7.	Simple Conduction	2	45
	TOTAL	PERIODS	45

Introduction to ANSYS

ANSYS is a general-purpose finite element modeling package for numerically solving a wide variety of mechanical problems. These problems include: static/dynamic structural analysis (both linear and non-linear), heat transfer and fluid problems, as well as acoustic and electromagnetic problems.

In general, a finite element solution may be broken into the following three stages. This is a general guideline that can be used for setting up any finite element analysis.

- 1. **Preprocessing: defining the problem**; the major steps in preprocessing are given below:
 - a. Define keypoints/lines/areas/volumes
 - b. Define element type and material/geometric properties
 - c. Mesh lines/areas/volumes as required
 - d. The amount of detail required will depend on the dimensionality of the analysis (i.e. 1D, 2D, axisymmetric, 3D).
- 2. Solution: assigning loads, constraints and solving; here we specify the loads (point or pressure), constraints (translational and rotational) and finally solve the resulting set of equations.
- 3. **Postprocessing: further processing and viewing of the results;** in this stage one may wish to see:
 - a. Lists of nodal displacements
 - b. Element forces and moments
 - c. Deflection plots
 - d. Stress contour diagrams

ANSYS 14.5 Environment

The ANSYS Environment for ANSYS 14.5 contains 2 windows: the Main Window and an Output Window. Note that this is somewhat different from the previous version of ANSYS which made use of 6 different windows.

1. Main Window

thity Menu Wolfland	Pagameters Bacro MeguCtris Help	
	Input Line	
A ANEYS Toobar		
ON TOOLOATON LOFF	UPLOT A_ON A_OFF APLOT V_ON V_OFF VPLOT N_ON	N_OFF NPLOT E_ON E_OFF EPLOT LGO DTE UPONT EPONT REPLOT CLR LOG
Proferences *	1	ANSVS
Preprocessor	NODES	74515
Anthe Menu		
Topological Opt		
Design Opt		
Radiation Opt		
Run-Time Stats Session Editor	Grapi	nics window
Finish		
		<u>k</u>

Within the Main Window are 5 divisions:

a. Utility Menu

The Utility Menu contains functions that are available throughout the ANSYS session, such as file controls, selections, graphic controls and parameters.

b. Input Window

The Input Line shows program prompt messages and allows you to type in commands directly.

c. Toolbar

The Toolbar contains push buttons that execute commonly used ANSYS commands. More push buttons can be added if desired.

d. Main Menu

The Main Menu contains the primary ANSYS functions, organized by preprocessor, solution, general postprocessor, design optimizer. It is from this menu that the vast majority of modeling commands are issued. This is where you will note the greatest change between previous versions of ANSYS and version 14.5. However, while the versions appear different, the menu structure has not changed.

e. Graphic Window

The Graphic Window is where graphics are shown and graphical picking can be made. It is here where you will graphically view the model in its various stages of construction and the ensuing results from the analysis.

2. Output Window



The Output Window shows text output from the program, such as listing of data etc. It is usually positioned behind the main window and can depute to the front if necessary.

<u>ANSYS Interface</u> <u>Graphical Interface vs. Command File Coding</u>

There are two methods to use ANSYS. The first is by means of the graphical user interface or GUI. This method follows the conventions of popular Windows and X-Windows based programs.

The second is by means of command files. The command file approach has a steeper learning curve for many, but it has the advantage that an entire analysis can be described in a small text file, typically in less than 50 lines of commands. This approach enables easy model modifications and minimal file space requirements.

The tutorials in this website are designed to teach both the GUI and the command file approach, however, many of you will find the command file simple and more efficient to use once you have invested a small amount of time into learning the code.

For information and details on the full ANSYS command language, consult:

Help > Table of Contents > Commands Manual.

FEM Convergence Testing

Introduction

A fundamental premise of using the finite element procedure is that the body is sub-divided up into small discrete regions known as finite elements. These elements defined by nodes and interpolation functions. Governing equations are written for each element and these elements are assembled into a global matrix. Loads and constraints are applied and the solution is then determined.

The Consequences

Finer meshes come with a cost however: more calculational time and large memory requirements (both disk and RAM)! It is desired to find the minimum number of elements that give you a converged solution.

Beam Models

For beam models, we actually only need to define a single element per line unless we are applying a distributed load on a given frame member. When point loads are used, specifying more than one element per line will not change the solution, it will only slow the calculations down. For simple models it is of no concern, but for a larger model, it is desired to minimize the number of elements, and thus calculation time and still obtain the desired accuracy.

General Models

In general however, it is necessary to conduct convergence tests on your finite element model to confirm that a fine enough element discretization has been used. In a solid mechanics problem, this would be done by creating several models with different mesh sizes and comparing the resulting deflections and stresses, for example. In general, the stresses will converge more slowly than the displacement, so it is not sufficient to examine the displacement convergence.

ANSYS: Saving and Restoring Jobs

Saving Your Job

It is good practice to save your model at various points during its creation. Very often you will get to a point in the modeling where things have gone well and you like to save it at the point. In that way, if you make some mistakes later on, you will at least be able to come back to this point.

To save your model, select Utility Menu Bar -> File -> Save AsJobname.db. Your model will be saved in a file called jobname.db, where jobname is the name that you specified in the Launcher when you first started ANSYS.

It is a good idea to save your job at different times throughout the building and analysis of the model to backup your work incase of a system crash or other unforeseen problems.

Recalling or *Resuming* **a Previously Saved Job**

Frequently you want to start up ANSYS and recall and continue a previous job. There are two methods to do this:

- 1. Using the Launcher...
 - In the ANSYS Launcher, select Interactive... and specify the previously defined jobname.
 - Then when you get ANSYS started, select Utility Menu -> File -> Resume Jobname.db.
 - This will restore as much of your database (geometry, loads, solution, etc) that you previously saved.
- 2. Or, start ANSYS and select Utility Menu -> File -> Resume from... and select your job from the list that appears.

Ex. No. : 1 Date:

STRESS ANALYSIS OF PLATE WITH HOLE

Problem Specification:

Consider the square plate of uniform thickness with a circular hole with dimensions shown in the figure below. The thickness of the plate is 1 mm. The Young's modulus E = 10 e7 MPa and the Poisson ratio is 0.3. A uniform pressure p=1 MPa acts on the boundary of the hole. Assume that plane stress conditions prevail. The stress and displacement fields are to be determined usingANSYS.



Step 1: Start-up and preliminary set-up

Create a folder

Create a folder called plate at a convenient location. We'll use this folder to store filescreated during the ANSYS session.

Start ANSYS

Start > Programs > ANSYS > ANSYS Product launcher

In the window that comes up, enter the location of the folder you just created as your**Working directory** by browsing to it. All files generated during the ANSYS run willbe stored in this directory.

Specify plate as your **Initial job name**. The job name is the prefix used for all filesgenerated during the ANSYS session. For example, when you perform a save operation in ANSYS, it'll store your work in a file called plate.db in your working directory.

For this tutorial, we'll use the default values for the other fields. Click **Run**. Forthis tutorial, we'll use the default values for the other fields. Click **Run**. Thisbrings up the ANSYS interface. To make best use of screen real estate, move thewindows around and resize them so that you approximate this screen arrangement. This way you can read instructions in the browser window and implement them inANSYS.

Set Preferences

As before, we'll more or less work our way down the Main Menu. **Main Menu > Preferences**

In the Preferences for GUI Filtering dialog box, click on the box next to **Structural** Soa tick mark appears in the box. Click **OK**.

Preferences for GUI Filtering		×
[KEYW] Preferences for GUI Filtering		
Individual discipline(s) to show in the GUI		
	✓ Structural	
	Thermal	
	ANSYS Fluid	
	FLOTRAN CFD	
Electromagnetic:		
	Magnetic-Nodal	
	Magnetic-Edge	
	High Frequency	
	Electric	
Note: If no individual disciplines are selected they will all sho	w.	
Discipline options		
	• h-Method	
ОК	Cancel Help	

Enter Parameters

For convenience, we'll create scalar parameters corresponding to the plate half-widtha, hole radius r, pressure p, and material properties E and v.

Utility Menu > Parameters > Scalar Parameters

Enter the parameter value for a: a=10e-3 Click **Accept**.

Similarly, enter the other parameter values and click **Accept** after each.

r=7e-3 p=1e6 E=1e13 Nu=0.3

а –	= 1.00000000E+02
E	= 1.00000000E+13
NU P	= 0.3 = 1000000
R	= 7.00000000E-03
Sele	ction
	1 000000005 00

Close the Scalar Parameters window.

Step 2: Specify element type and constants

Specify Element Type

Main Menu > Preprocessor> Element Type > Add/Edit/Delete > Add...

Pick **Structural Solid** in the left field and **Quad 4 node 182** in the right field. Click**OK** to select this element.

▲ Library of Element Types			x
Only structural element types are shown			
Library of Element Types	Structural Mass Link Beam Pipe Solid Shell Solid-Shell	Quad 4 node 182 8 node 183 Brick 8 node 185 20node 186 concret 65 Quad 4 node 182	•
Element type reference number	1		
OK Apply	Cancel	Help	

You'll now see the Element Types menu with PLANE182 as the only defined elementtype.

		s.		
	Element Types			×
ſ				
	Defined Element Ty	Pes:		
	Type I PLA	NETOZ		
		0-1	Delete	
	Add	Options	Delete	
	Class	1	Liele	
Ň Ì	Close		нер	

Let's take a look at the online help pages to learn about the properties of this element.

Utility Menu > Help > Help Topics

Select the **Search** tab, type in pictorial summary as the keyword and click **ListTopics**. You should see **Pictorial Summary** as one of the topics listed; double-clickon this. This brings up the Pictorial Summary of Element Types help page. Scroll down to Plane182 under Structural 2-D Solid. Note that the PLANE182 element isdefined by four nodes with two degrees of freedom at each node: translations UXand UY in the (nodal) x and y-directions.

Click on the PLANE182 box to bring up the help page for this element. Read theElement Description and take a look at the figure of the element. Think about whythis element is appropriate for the problem at hand. Minimize the help window.

If you actually read the Element Description for PLANE182, you'd have noticed that this element can also be used for axisymmetric problems also. In the axi symmetric case, you would choose **Options** for the element in the Element Types menu. Note that in the PLANE182 element type options menu that comes up, under **Elementbehavior**, you have the option of **Axisymmetric**. For the current problem, we'll of course use the default of **Plane stress**. Click **Cancel** to exit the PLANE182 element type options menu retaining the defaults.

Close the Element Types menu.

Specify Element Constants

Main Menu > Preprocessor> Real Constants > Add/Edit/Delete > Add

This brings up the Element Type for Real Constants menu with a list of the element types defined in the previous step. We have only one element type and it is automatically selected.

<i>(</i>					
T Ele	ement Typ	e for Real (Constant	5	L X
	hoose e	element	type:		
	Tupe 1	DI ANE 192))		
	rype i	FLANE TO2			
	OK			Cancel	
				Guncer	

Click OK.

You should get a note saying "Please check and change key point setting for elementPLANE182 before proceeding." Close the yellow warning window and the Real Constants menu. To see what this message implies, let's again take a look at the help pages for PLANE182.



Under PLANE182 Input Summary, the documentation says that there are no real constants for this element when KEYOPT (3) = 0, 1, 2.

To see what the value of KEYOPT (3) is, bring up the Element Type menu again:

Main Menu > Preprocessor> Element Type > Add/Edit/Delete > Options

K3 i.e. KEYOPT (3) is set to **Plane stress**. In the help page, under PLANE182 Input Summary, you can check that plane stress corresponds to KEYOPT (3) =0. Thus, there are no real constants to be specified. That's why we got the "Please check and change key point settings..." warning message. Of course, the ANSYS warning could have been less cryptic but what fun would that be.

Cancel the PLANE182 element type options menu, **Close** the Element Types menu and close the Element Type sticky menu.

Save your work

Toolbar > SAVE_DB

Step 3: Specify material properties.

Main Menu > Preprocessor > Material Props > Material Models....

In the Define Material Model Behavior menu, double-click on **Structural**, **Linear**, **Elastic**, and **Isotropic**.



17 | P a g e

We'll use the previously defined parameter names while specifying the materialproperties. Enter E for Young's modulus **EX**, nu for Poisson's Ratio **PRXY**. Click**OK**.

Temperatures		
EX E		
DD YV		
	u and and a second	

To double-check the material property values, double-click on **Linear Isotropic**under **Material Model Number 1** in the Define Material Model Behavior menu. This will show you the current values for EX and PRXY. **Cancel** the LinearIsotropic Properties window.

besteril orGoWasset	Energy Delivery Properties for Halterial Sustains 1	
B Transferingener	silen accordination migerales for manufacture (
	Imperature 11	
	Re (1411) mar (23	
	Atil forgenature Delate Temperature Graph	
	06 Dancel Hep	10

When you enter parameter names, ANSYS substitutes the corresponding parameter values as soon as you click OK or Apply.

This completes the specification of Material Model Number 1. When we mesh thegeometry later on, we'll use the reference no. 1 to assign this material model. Closethe Define Material Model Behavior menu.

Save your work

Toolbar > SAVE_DB

Step 4: Specify geometry.

Since the geometry, material properties and loading are all symmetric with respect to the horizontal and vertical centerlines, we need to model only a quarter of the plate. We will take the origin of the coordinate system to be at the center of the hole andmodel only the top right quadrant. We'll create the geometry by creating a square areaof side a and subtracting the circular sector of radius r from it.

Create the Square

Main Menu > Preprocessor > Modeling >Create > Areas > Rectangle > ByDimensions X1 and X2 are the x-coordinates of the left and right edges of the square, respectively.

Enter 0 for **X1**,a for**X2**.

Y1 and **Y2** are the y-coordinates of the bottom and top edges of the squarerespectively. Enter 0 for **Y1**, a for**Y2**.

[RECTNG] Create Rectangle	by Dimensions		
X1,X2 X-coordinates		0	a
Y1,Y2 Y-coordinates		0	[a
ок	Apply	Cancel	Help

Click **OK**. You should see a square appear in the graphics window. **Create the Circular Sector**

Main Menu > Preprocessor > Modeling > Create > Areas > Circle > PartialAnnulus

WP X and **WP Y** are the x- and y-coordinates of the center of the circular arc. Soenter 0 for both **WP X** and **WP Y**. (WP refers to the Working Plane which by defaultcoincides with the global Cartesian coordinate system. We won't have to worry about the working plane in this friendly example.)

Rad-1 is the radius of the inner circular arc. We want to create a solid rather than anannular arc. Enter 0 for **Rad-1** to create a solid arc.

Rad-2 is the (outer) radius of the arc. Since we had defined the whole radius asParameter r earlier, enter r for **Rad-2**.

Theta-1 and **Theta-2** are the starting and ending angles of the arc, respectively. These angles need to be specified in degrees. Enter 0 for **Theta-1** and 90 for **Theta-2**. Click **OK**.

• Pick		C Unpick
WP X	=	
Y	=	
Global X	=	
¥	-	
z	=	
WP X		0
WP Y		0
Rad-1		0
Theta-1		0
Rad-2		r
Theta-2		90
ок	j	Apply
Reset		Cancel
Help	- 1	1

This will create and draw the circular sector. You'll see a white line denoting the circular sector.

Subtract Circular Sector from Square

Main Menu > Preprocessor > Modeling > Operate > Booleans > Subtract > Areas

In the Input window, ANSYS tells you to "pick or enter base areas from which to subtract". So we pick the square area as follows: Hold down the left mouse button, move the cursor over the areas until the square is selected (it will change color) and release the left mouse button. Click **OK**.



In the Input window, ANSYS now tells you to "pick or enter areas to be subtracted".So, select the circular sector by holding down and releasing the left mouse button.Click **OK**.



If you did this correctly, you will see that the circular sector has been subtracted outfrom the square area.



You can also select areas during the Boolean subtract operation by simply clicking on them but it becomes difficult to select areas (and other components) in this fashion in more complicated geometries. That's why I made you use the "holding-down-the-mouse-and-releasing" technique.

If you picked an area incorrectly, you can unpick it by clicking the right mouse button and selecting the area. The cursor changes to a downward arrow during an unpick operation. Right-click to return pick mode.

Save Your Work

Toolbar > SAVE_DB

Step 5: Mesh geometry.

Bring up the MeshTool:

Main Menu > Preprocessor > MeshTool

The MeshTool is used to control and generate the mesh.

Set Meshing Parameters

We'll now specify the element type, real constant set and material property set to beused in the meshing. Since we have only one of each, we can assign them to theentire geometry using the **Global** option under **Element Attributes**.

Make sure Global is selected under Element Attributes and click on Set.

Global Set Fine 6 Coarse Size Controls: Global Set Clear Areas Set Clear Lines Set Clear Layer Set Clear Keypts Set Clear Keypts Set Clear Mesh: Areas Shape: Tri Quad Free Mapped Sweep 3 or 4 sided Mesh Clear Refine at: Elements	
Smart Size Fine Fine 6 Coarse Size Controls: Global Set Clear Layer Set Clear Layer Set Clear Keypts Set Clear Keypts Set Clear Mesh: Areas Shape: Tri Quad Free Mapped Sweep 3 or 4 sided Mesh Clear Befine at: Elements	
Fine 6 Coarse Size Controls: Glabal Set Clear Areas Set Clear Layer Set Clear Keypts Set Clear Mesh: Areas Shape: Tri Quad Free Mapped Sweep 3 or 4 sided Refine at: Elements	
Fine 6 Coarse Size Controls: Global Set Clear Areas Set Clear Lines Set Clear Layer Set Clear Keypts Set Clear Mesh: Areas Sweep 3 or 4 sided Sweep Befine at: Elements	
Size Controls: Global Set Clear Areas Set Clear Lines Set Clear Copy Flip Layer Set Clear Keypts Set Clear Mesh: Areas Shape: Tri © Quad Free © Mapped © Sweep 3 or 4 sided Mesh Clear Refine at: Elements	
Global Set Clear Areas Set Clear Lines Set Clear Layer Set Clear Keypts Set Clear Mesh: Areas Shape: Tri © Quad Free © Mapped © Sweep 3 or 4 sided Mesh Clear Mesh Clear	
Areas Set Clear Lines Set Clear Copy Flip Layer Set Clear Keypts Set Clear Mesh: Areas Shape: Tri © Quad © Free © Mapped © Sweep 3 or 4 sided Mesh Clear Mesh Clear	
Lines Set Clear Copy Flip Layer Set Clear Keypts Set Clear Mesh: Areas Shape: Tri © Quad © Free © Mapped © Sweep 3 or 4 sided Mesh Clear Mesh Clear Refine at: Elements	
Copy Flip Layer Set Clear Keypts Set Clear Mesh: Areas Image: Clear Shape: Tri Quad • Free Mapped Sweep 3 or 4 sided Image: Clear Mesh Clear Befine at: Elements	
Layer Set Clear Keypts Set Clear Mesh: Areas Shape: Tri © Quad © Free © Mapped © Sweep 3 or 4 sided Mesh Clear Refine at: Elements	
Lager Set Clear Keypts Set Clear Mesh: Areas Image: Clear Shape: Tri © Quad © Free Mapped Sweep 3 or 4 sided Image: Clear Mesh Clear	
Keypts Set Clear Mesh: Areas Image: Constraint of the second secon	
Mesh: Areas Shape: Tri Ouad • Free Mapped Sweep 3 or 4 sided Mesh Clear Refine at: Elements	
Mesh: Areas Shape: Tri Quad Free Mapped Sweep 3 or 4 sided Mesh Clear Refine at: Elements	
Shape: Tri Ouad Free Mapped Sweep 3 or 4 sided Mesh Clear Refine at: Elements	
 Free Mapped Sweep 3 or 4 sided Mesh Clear Refine at: Elements 	
3 or 4 sided Mesh Clear Refine at: Elements	
3 or 4 sided Mesh Clear Refine at: Elements	
Mesh Clear Refine at: Elements	
Mesh Clear Refine at: Elements	
Refine at: Elements	
Refine at: Elements	
Refine at: Elements	
Refine	

This brings up the Meshing Attributes menu. You will see that the correct elementtype and material number are already selected since we have only one of each. Recall that no real constants need to be defined for PLANE182 element type with the planestress key option.

▲ Element Attributes		l	×
Define attributes for elements			
[TYPE] Element type number		1 PLANE182	ŀ
[MAT] Material number		1 💌	
[REAL] Real constant set number		1 💌	
[ESYS] Element coordinate sys		0 💌	
[SECNUM] Section number		None defined	•
[TSHAP] Target element shape		Straight line	-
ок	Cancel	Help	

Click **OK**. ANSYS now knows what element type and material type to use for themesh.

Set Mesh Size

Instead of setting the mesh size at each boundary, we'll use the Smart Size optionwhich enables automatic element sizing. Click on the SmartSize checkbox so that atickmark appears in it.

Global	1	- Set
Smart 9	Size	
•		•

The only input necessary for the SmartSize option is the overall element size level formeshing. The element size level determines the fineness of the mesh. Its value iscontrolled by the slider shown in the above picture. Change the setting for the overallelement size level to **5** by moving the slider under **SmartSize**from the left.

Mesh Areas

In the MeshTool, make sure **Areas** is selected in the drop-down list next to **Mesh**. This means the geometry components to be meshed are areas (as opposed to lines orvolumes). We'll use quadrilateral elements. So make sure the default option of **Quad** is selected under **Shape**. We'll also use the default of **Free**meshing.

Click on the **Mesh** button. This brings up the pick menu.

• Pick	() Unpick
Single	C Box
C Polygon C Loop	C Circle
Count =	0
Maximum =	l
Minimum =	ı
♠ List of♠ Min, Ma	Items x, Inc
© List of C Min, Ma	Items x, Inc Apply
© List of C Min, Ma OK Reset	Items x, Inc Apply Cancel

In the Input window, ANSYS tells you to "pick or enter areas to be meshed". Sincewe have only one area to be meshed, click on **Pick All**. The geometry has beenmeshed and the elements are plotted in the Graphics window. **Close** the MeshTool.

Save Your Work

Toolbar > SAVE_DB

Step 6: Specify boundary conditions.

Next, we step up to the plate to define the displacement constraints and loads. Recallthat in ANSYS terminology, the displacement constraints are also "loads". As in the truss tutorial, we'll apply the loads to the geometry rather than the mesh. That way wewon't have to reapply the loads on changing the mesh.

Apply Symmetry Boundary Conditions

ANSYS provides the option of applying a "symmetry boundary condition" along lines of symmetry.

Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural >Displacement > Symmetry B.C. > On Lines

Select the straight lines corresponding to the left and bottom edges (which are the lines of symmetry for this problem) by clicking on them. Click **OK** in the pick menu. The symbol s appears along these lines indicating that the symmetry B.C. is applied along these lines.



Apply Pressure

Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural >Pressure > On Lines

Select the circular arc and click **OK**. This brings up the Apply Pressure on Linesmenu. Enter p for **Value** and click **OK**. A single red arrow denotes the pressure and the direction in which it is acting.



Check Loads

Let's check that the displacement constraints have been applied correctly.

Utility Menu > List > Loads > DOF constraints > On All Lines

ile		
LIST CONST	RAINTS ON ALL SELEC	CTED LINES
LINE	LOAD LABEL	VALUE(S)
8	SYMM	0.0000
9	SVMM	0 0000

Symmetry BCs are applied on lines 8 and 9. Turn on line numbering:

Utility Menu >PlotCtrls> Numbering

Turn on **Line numbers** and click **OK**. Are lines L8 and L9 the ones on which you want the symmetry BCs?

Similarly, check that the pressure is applied correctly using **Utility Menu > List >Loads > Surface Loads > On All Lines**. Note that **VALI** and **VALJ** would be different if the applied pressure were linearly varying along the line.

Turn off line numbering: **Utility Menu >PlotCtrls> Numbering**. Turn off **Line numbers**and click **OK**.

Save Your Work

Toolbar > SAVE_DB

Step 7: Solve.

Enter solution module:

Main Menu > Solution

Enter check in the Input window. If the problem has been set up correctly, there will be no errors or warnings reported. If you look in the Output window, you should see the message: The analysis data was checked and no warnings or errors were found.

Main Menu > Solution > Solve > Current LS

Recall from the truss tutorial that this solves the current load step (LS) i.e. the current loading conditions. In this problem also, there is only one load step.

Review the information in the /STATUS Command window. Close this window.

Click **OK** in Solve Current Load Step menu.



ANSYS performs the solution and a yellow window should pop up saying "Solution is done!"

Verify that ANSYS has created a file called plate.rst in your working directory. This file contains the results of the (previous) solve.

Step 8: Postprocessor the Results

Enter the Postprocessing module to analyze the solution.

Main Menu > General Postproc Plot Deformed Shape

Main Menu > General Postproc> Plot Results > Deformed Shape

Select **Def + undeformed**and click **OK**.

This plots the deformed and undeformed shapes in the Graphics window. The maximum deformation DMX is 0.232E-08m as reported in the Graphics window. Note that the deformation is magnified in the plot so as to be visible.

The deformation would be better visible if the foreground and background were not of the same color. Turn off the background:

Utility Menu >PlotCtrls> Style > Background > Display Picture Background



Animate the deformation:

Utility Menu >PlotCtrls> Animate > Deformed Shape...

Select **Def + undeformed**and click **OK**. Select **Forward Only** in the AnimationController.

The left and bottom edges move parallel to them which means that the full deformed plate is also symmetric about these edges. This shows that the symmetry boundary condition at these edges is imposed correctly. The circular edge of the holemoves outward which is what one would expect from the outward pressure acting along it. Thus, the deformation of the structure agrees with the applied boundary conditions and matches with what one would expect from intuition.

Close the Animation Controller.

Plot Nodal Solution of von Mises Stress

To display the von Mises stress distribution as continuous contours, select

Main Menu > General Postproc> Plot results > Contour Plot > Nodal Solution

Select Stress from the left list, von Mises SEQV from the right list and click OK.



The contour plot will show you the locations of the maximum and minimum values with the labels MX and MN, respectively. Are these locations where you expect them? SMX and SMN values reported in the Graphics window are the corresponding maximum and minimum stress values.

The diagonal is an additional line of symmetry. How symmetric is your result about the diagonal?

Save this plot to a file:

Utility Menu >PlotCtrls> Hard Copy > To File

Select the file format you want and type in a filename of your choice under **saveto:** and click **OK**. Check that the file has been created in your working directory.

When you plot the "Nodal Solution", ANSYS obtains a continuous distribution as follows:

1. It determines the average at each node of the values of all elements connected to the node.

2. Within each element, it linearly interpolates the average nodal value obtained in the previous step.

Plot Element Solution of von Mises Stress

To obtain results without nodal averaging, select

Main Menu > General Postproc> Plot results > Contour Plot > Element Solution

Select **Stress** from the left list, **von Mises SEQV** from the right list and click **OK**. This displays the von Mises stress results as discontinuous element contours.



Save this plot to a file: **Utility Menu >PlotCtrls> Hard Copy > To File**

Element solution contours are determined by linear interpolation within each elementbut no nodal averaging is performed. The discontinuity between contours of adjacent elements is an indication of the gradient across elements. The inter-element discontinuities in our solution are relatively small compared to the stress levels. This indicates that the mesh resolution is reasonably good.

Query Results

To determine the value of the first principal stress sigma1 at a selected location, select

Main Menu > General Postproc> Query Results > SubgridSolu

This brings up the Query Subgrid Solution Data menu. Select **Stress** from the left list, **1st principal S1** from the right list and click **OK**.

This brings up the pick menu. You can click on any location in the geometry andANSYS will print the sigma1 value at that location. Try querying the values at a fewlocations. Note that the coordinates of the picked location and the corresponding solution value are reported in the pick menu.

Cancel the pick menu.

Step 9: Validate the results

It is **very important** that you take the time to check the validity of your solution. This section leads you through some of the steps you can take to validate yoursolution.

Simple Checks

Does the deformed shape look reasonable and agree with the applied boundaryconditions? We checked this in step 8.

Do the reactions at the supports balance the applied forces for static equilibrium? Tocheck this, select

Main Menu > General Postproc> List Results > Reaction Solution

Select allstrucforc F for Item to be listed and click OK.

The total reaction force in the x-direction is -7000 N.

Applied force = (pressure) x (projected distance in x-direction of the line along which the constant pressure acts) = (p) (r) = 7000 N in positive x-direction.

So the reaction cancels out the applied force in the x-direction. Similarly, you cancheck that this is true in the y-direction also.

Refine Mesh

Let's repeat the calculations on a mesh with overall element size level underSmartSize set to 4 instead of 5 and compare the results on the two meshes. Delete thecurrent mesh:

Main Menu > Preprocessor > Mesh Tool

Select **Clear** under **Mesh:** and **Pick all**in the pick menu. The mesh is deleted. Set the overall element size level under SmartSize to 4 by dragging the slider to theleft. Click on **Mesh** and **Pick All**.

In the Output window, check how many elements are contained in this mesh? Yournew mesh should have 276 quadrilateral elements.

Obtain a new solution: Main Menu > Solution > Solve > Current LS

Plot nodal solution of the von Mises stress: Main Menu > General Postproc> Plot results > Contour Plot > Nodal Solution

Select Stress from the left list, von Mises SEQV from the right list and click OK.



Compare this with the von Mises contours for the previous mesh:



The two results compare well with the finer mesh contours being smoother asexpected. Compare the maximum stress and displacement values:

		Coarser Mesh	Finer Mesh
,	DMX	0.232e-8m	0.234e-8m
, 	SMX	3.64MPa	3.74MPa

The maximum displacement value changes by less than 1% and the maximum vonMises stress value by less than 3%. This indicates that the meshes used provideadequate resolution.

Exit ANSYS

Utility Menu > File > Exit

Select Save Everything and click OK.

Ex. No. : 2 Date:

STRESS ANALYSIS OF A BRACKET

The problem to be modeled in this example is a simple bracket shown in the following figure. This bracket is to be built from a 20 mm thick steel plate. A figure of the plate is shown below.



This plate will be fixed at the two small holes on the left and have a load applied to the larger hole on the right.

Preprocessing: Defining the Problem

- 1. Give the Bracket example a Title
 - Utility Menu > File > Change Title

2. Form Geometry

Boolean operations will be used to create the basic geometry of the Bracket.

- a. Create the main rectangular shape The main rectangular shape has a width of 80 mm, a height of 100mm and the bottom left corner is located at coordinates (0,0)
- b. Create the circular end on the right hand side
- The center of the circle is located at (80,50) and has a radius of 50 mm c. Now create a second and third circle for the left hand side using the following dimensions:

parameter	circle 2	circle 3
XCENTER	0	0
YCENTER	20	80
RADIUS	20	20

d. Create a rectangle on the left hand end to fill the gap between the two small circles.

XCORNER	-20
YCORNER	20
WIDTH	20
HEIGHT	60

e. Screen should now look like the following



- f. Boolean Operations Addition Now add these five discrete areas together to form one area.
 - To perform the Boolean operation, from the Preprocessor menu select:



- g. Create the Bolt Holes from this plate.
 - Create the three circles with the parameters given below:

parameter	circle 1	circle 2	circle 3
WP X	80	0	0
WP Y	50	20	80
radius	30	10	10

- Select
 - Preprocessor > Modeling > Operate > Booleans > Subtract > Areas
- Select the base areas from which to subract (the large plate that was created)
- Next select the three circles that we just created. Click on the three circles that you just created and click 'OK'.



3. **Define the Type of Element**

As in the verification model, PLANE82 will be used for this example

- Preprocessor > Element Type > Add/Edit/Delete
- \circ $\;$ Use the 'Options...' button to get a plane stress element with thickness

Define Geometric Constraints

- Preprocessor > Real Constants > Add/Edit/Delete
- Enter a thickness of 20mm.

Element Material Properties

Preprocessor > Material Props > Material Library > Structural > Linear > Elastic > Isotropic

Properties of Steel. EX 200000 PRXY 0.3

Mesh Size

- Preprocessor > Meshing > Size Cntrls> Manual Size > Areas > All Areas
- \circ Select an element edge length of 5. Again, we will need to make sure the model has converged.

Mesh

 Preprocessor > Meshing > Mesh > Areas > Free and select the area when prompted

(Alternatively, the command line code for the above step is **AMESH,ALL**)



Saving Your Job Utility Menu > File > Save as

Solution Phase: Assigning Loads and Solving

You have now defined your model. It is now time to apply the load(s) and constraint(s) and solve the the resulting system of equations.

1. **Define Analysis Type**

• 'Solution' > 'New Analysis' and select 'Static'.

2. Apply Constraints

As illustrated, the plate is fixed at both of the smaller holes on the left hand side.

- Solution > Define Loads > Apply > Structural > Displacement > On Nodes
- Instead of selecting one node at a time, you have the option of creating a box, polygon, or circle of which all the nodes in that area will be selected. For this case, select 'circle' as shown in the window below. Click at the center of the bolt hole and drag the circle out so that it touches all of the nodes on the border of the hole.



- Click on 'Apply' in the 'Apply U,ROT on Lines' window and constrain all DOF's in the 'Apply U,ROT on Nodes' window.
- Repeat for the second bolt hole.

3. Apply Loads

As shown in the diagram, there is a single vertical load of 1000N, at the bottom of the large bolt hole. Apply this force to the respective keypoint (**Solution > Define Loads > Apply > Structural > Force/Moment > On Keypoints** Select a force in the y direction of -1000) The applied loads and constraints should now appear as shown below.



4. Solving the System

Solution > Solve > Current LS Post-Processing: Viewing the Results

1. Convergence using ANSYS

As shown previously, it is necessary to prove that the solution has converged. Reduce the mesh size until there is no longer a sizeable change in your convergence criteria.

2. Deformation

- **General Postproc> Plot Results >Def + undeformed** to view both the deformed and the undeformed object.
- The graphic should be similar to the following



 \circ $\,$ Observe the locations of deflection. Ensure that the deflection at the bolt hole is indeed 0.

3. Deflection

 To plot the nodal deflections use General Postproc> Plot Results > Contour Plot > Nodal Solution then select DOF Solution - USUM in the window.



- Alternatively, obtain these results as a list. (General Postproc> List Results > Nodal Solution...)
- Are these results what you expected? Note that all translational degrees of freedom were constrained to zero at the bolt holes.

4. Stresses

 General Postproc> Plot Results > Nodal Solution... Then select von Mises Stress in the window.



List the von Mises stresses to verify the results at certain nodes
 General Postproc> List Results. Select Stress, Principals SPRIN
Ex. No. : 3 Date:

STRESS ANALYSIS OF AN AXI-SYMMETRIC COMPONENT

Problem Description:

The model will be that of a closed tube made from steel. Point loads will be applied at the center of the top and bottom plate to make an analytical verification simple to calculate. A 3/4 cross section view of the tube is shown below.

As a warning, point loads will create discontinuities in the model near the point of application. If you chose to use these types of loads in your own modeling, bevery careful and be sure to understand the theory of how the FEA package is applyingthe load and the assumption it is making. In this case, we will only be concernedabout the stress distribution far from the point of application, so the discontinuities will have a negligible effect.



Preprocessing: Defining the Problem

1. Give example a Title

Utility Menu > File > Change Title .../title, Axisymmetric Tube

2. Open preprocessor menu

ANSYS Main Menu > Preprocessor/PREP7

3. Create Areas

Preprocessor > Modeling > Create > Areas > Rectangle > By DimensionsRECTNG, X1, X2, Y1, Y2. For an axisymmetric problem, ANSYS will rotate the area around the y-axis at x=0. Therefore, to create the geometry mentioned above, we must define a U-shape. We are going to define 3 overlapping rectangles as defined in the following table:

Rectangle	X1	X2	Y1	Y2
1	0	20	0	5
2	15	20	0	100
3	0	20	95	100

4. Add Areas Together

Preprocessor > Modeling > Operate > Booleans > Add > Areas ADD, ALL

Click the Pick All button to create a single area.

5. Define the Type of Element

Preprocessor > Element Type > Add/Edit/Delete...

For this problem we will use the PLANE182 (Structural, Solid, Triangle 6node) element. This element has 2 degrees of freedom (translation along the X and Y axes).

Many elements support axisymmetry, however if the Ansys Elements Reference (which can be found in the help file) does not discuss axisymmetric applications for aparticular element type, axisymmetry is not supported.

6. Turn on Axisymmetry

While the Element Types window is still open, click the **Options...** button.Under Element behavior K3 select **Axisymmetric**.

Options for PLANE	2. Element Type Ref. No. 1	
Element behavior	K3	Axisymmetric 💌
Extra stress output	К5	No extra output
Element output	К6	Basic element
ОК	Cancel	Help

7. Define Element Material Properties.

Preprocessor > Material Props > Material Models > Structural > Linear > Elastic > Isotropic In the window that appears, enter the following geometric properties for steel:

- 1. Young's modulus EX: 200000
- 2. Poisson's Ratio PRXY: 0.3

8. Define Mesh Size

Preprocessor > Meshing > Size Cntrls>ManualSize> Areas > All Areas For this example we will use an element edge length of 2mm.

9. Mesh the frame

Preprocessor > Meshing > Mesh > Areas > Free > click 'Pick All' Your model should know look like this:

Solution Phase: Assigning Loads and Solving

1. Define Analysis Type

• Solution > Analysis Type > New Analysis > StaticANTYPE,0

2. Apply Constraints

 Solution > Define Loads > Apply > Structural > Displacement >Symmetry B.C. > On Lines

17. 2000

Pick the two edges on the left, at x=0, as shown below. By using the symmetry B.C.command, ANSYS automatically calculates which DOF's should be constrained for the line of symmetry. Since the element we are using only has 2 DOF's per node, we could have constrained the lines in the x-direction to create the symmetric boundaryconditions.

• Utility Menu > Select > Entities

Select **Nodes** and **By Location** from the scroll down menus. Click **Y coordinates** and type **50** into the input box as shown below, then click OK.



• Solution > Define Loads > Apply > Structural > Displacement > On Nodes > PickAll

Constrain the nodes in the y-direction (UY). This is required to constrain the model in space; otherwise it would be free to float up or down. The location to constrain the model in the y direction (y=50) was chosen because it is along a symmetry plane. Therefore, these nodes won't move in the y-direction according to theory.

3. Utility Menu > Select > Entities

In the select entities window, click **Select All** to reselect all nodes. It is important to always reselect all entities once you've finished ensuring future commands are applied to the whole model and not just a few entities. Once you've clicked Select All, click on **Cancel** to close the window.

4. Apply Loads:

• Solution > Define Loads > Apply > Structural > Force/Moment > On Key points Pick the top left corner of the area and click OK. Apply a load of 100 in the FY direction.

• Solution > Define Loads > Apply > Structural > Force/Moment >On Key points Pick the bottom left corner of the area and click OK. Apply a load of -100 in the FY direction.

• The applied loads and constraints should now appear as shown in thefigure.

PTPMPNTe	200	ANSY
BDDAG(15		JUL 31 2003
	The second second	13.00120
Axisymmetric Tube		

5. Solve the System:

• Solution > Solve > Current LS SOLVE

Determine the Stress Through the Thickness of the Tube.

• Utility Menu > Select > Entities...

Select **Nodes > By Location > Y coordinates** and type **45,55**in the Min,Max box, as shown below and click OK.

Select En	tities 🔀
Nodes	-
By Loca	tion 💌
C X coord	linates
· Y coord	linates
C Z coord	linates
Min, Max	
45,55	
• From F	ull
C Resele	ct
C Also Se	lect
C Unsele	ct
Sele All	Invert
Sele None	Sele Belo
ок	Apply
Plot	Replot
Cancel	Help

• General Postproc> List Results > Nodal Solution > Stress > Components SCOMP

The following list should pop up.

X,Y,Z UALUES AR SV 99E-05 0.17893 20E-05 0.17873 38E-05 0.17866 02E-05 0.17876 65E-05 0.17873 84E-04 0.18521 78E-04 0.18554 72E-04 0.18554	E IN GLOBAL COORDINATES SZ SXV 0.18640E-04-0.13372E-04 -0.52933E-03-0.32806E-06 -0.79076E-03-0.38045E-06 -0.73294E-03-0.38945E-06 0.17546E-04 0.13278E-04 0.23216E-04-0.23292E-04 0.16776E-02-0.2392E-04 0.16776E-02-0.35547E-05	SYZ 0.0000 0.0000 0.0000 0.0000 0.0000 0.0000 0.0000	5X2 0.0000 0.0000 0.0000 0.0000 0.0000 0.0000 0.0000	
SV 99E-05 0.17893 20E-05 0.17873 38E-05 0.17866 02E-05 0.17866 02E-05 0.17893 84E-04 0.18521 78E-04 0.18554 72E-04 0.18554	SZ SXV 0.18640E-04-0.13372E-04 -0.52933E-03-0.32006E-06 -0.79076E-03-0.3004E-06 -0.73234E-03-0.50904E-06 -0.53104E-03-0.383495E-06 0.17546E-04-0.13278E-04 0.23216E-04-0.23292E-04 0.16776E-02-0.35547E-05	SV2 0.0000 0.0000 0.0000 0.0000 0.0000 0.0000	5X2 0.0000 0.0000 0.0000 0.0000 0.0000 0.0000 0.0000	
99E-05 0.17893 20E-05 0.17873 38E-05 0.17867 70E-05 0.17866 02E-05 0.17872 65E-05 0.17893 84E-04 0.18551 72E-04 0.18551	0.18640E-04-0.13372E-04 -0.52935:03+0.3206E-06 -0.79076E-03 0.17733E-05 -0.79294E-03-0.50904E-0 -0.53104E-03 0.38945E-66 0.17546E-04 0.13276E-04 0.23216E-02-0.2292E-04 0.16776E-02-0.35547E-05	0.0000 0.0000 0.0000 0.0000 0.0000 0.0000 0.0000	0.000 0.000 0.000 0.000 0.000 0.000 0.000 0.000 0.000	
20E-05 0.17873 38E-05 0.17867 70E-05 0.17866 02E-05 0.17872 65E-05 0.17893 84E-04 0.18521 78E-04 0.18551	-0.52932-03-0.32806E-06 -0.79076E-03.0.17733E-05 -0.79294E-03-0.50904E-06 -0.53104E-03.0.38945E-06 0.17546E-04.0.13278E-04 0.23216E-02-0.22292E-04 0.16776E-02-0.35547E-05	0.0000 0.0000 0.0000 0.0000 0.0000 0.0000	0.0000 0.0000 0.0000 0.0000 0.0000 0.0000	
38E-05 0.17867 70E-05 0.17866 02E-05 0.17872 65E-05 0.17893 84E-04 0.18521 78E-04 0.18544 72E-04 0.18551	-0.79076E-03 0.17733E-05 -0.79294E-03-0.59904E-06 -0.53104E-03 0.38945E-06 0.17546E-04 0.13278E-04 0.23216E-02-0.22292E-04 0.16776E-02-0.35547E-05	0.0000 0.0000 0.0000 0.0000 0.0000	8.0000 8.0000 8.0000 8.0000 8.0000 8.0000	
70E-05 0.17866 02E-05 0.17872 65E-05 0.17893 84E-04 0.18521 78E-04 0.18544 72E-04 0.18551	-0.79294E-03-0.50904E-06 -0.53104E-03 0.38945E-06 0.17546E-04 0.1278E-04 0.23216E-02-0.22292E-04 0.16776E-02-0.35547E-05	0.0000 0.0000 0.0000 0.0000	0.0000 0.0000 0.0000 0.0000	
02E-05 0.17872 65E-05 0.17893 84E-04 0.18521 78E-04 0.18544 72E-04 0.18551	-0.53104E-03 0.38945E-06 0.17546E-04 0.13278E-04 0.23216E-02-0.22292E-04 0.16776E-02-0.35547E-05	0.0000 0.0000 0.0000	0.0000 0.0000 0.0000	
65E-05 0.17893 84E-04 0.18521 78E-04 0.18544 72E-04 0.18551	0.17546E-04 0.13278E-04 0.23216E-02-0.22292E-04 0.16776E-02-0.35547E-05	0.0000	0.0000	
84E-04 0.18521 78E-04 0.18544 72E-04 0.18551	0.23216E-02-0.22292E-04 0.16776E-02-0.35547E-05	0.0000	0.0000	
78E-04 0.18544 72E-04 0.18551	0.16776E-02-0.35547E-05	0 0000		
72E-04 0.18551		0.0000	0.0000	
	0.13437E-02 0.11754E-05	0.0000	0.0000	
68E-04 0.18552	0.13462E-02 0.33412E-06	0.0000	0.0000	
73E-04 0.18545	0.16803E-02 0.36055E-05	0.0000	0.0000	
64E-04 0.18522	0.23258E-02 0.22180E-04	0.0000	0.0000	
10E-04 0.18205	0.74230E-03 0.16929E-03	0.0000	0.0000	
10E-04 0.18206	0.30056E-03 0.54607E-04	0.0000	0.0000	
27E-04 0.18206	0.15398E-03-0.14236E-05	0.0000	0.0000	
70E-04 0.18206	0.30047E-03-0.55975E-04	0.0000	0.0000	
14E-04 0.18205	0.74157E-03-0.16999E-03	0.0000	0.0000	
	273E-04 0.18545 164E-04 0.18522 110E-04 0.18205 110E-04 0.18206 27E-04 0.18206 170E-04 0.18206 14E-04 0.18205	773E-04 0.18545 0.16803E-02 0.36055E-05 1064E-04 0.18522 0.23258E-02 0.2005E-05 106-04 0.18205 0.74230E-03 0.16929E-03 110E-04 0.18205 0.30056E-03 0.54607E-04 27E-04 0.18206 0.30047E-03-0.14236E-05 0.55975E-04 114E-04 0.18205 0.74157E-03-0.16399E-03 0.5999E-03	173E-04 0.1683E-02 0.36055E-05 0.0000 164E-04 0.18522 0.23258E-02 0.22180E-04 0.0000 110E-04 0.18525 0.74230E-03 0.16929E-03 0.0000 110E-04 0.18205 0.74230E-03 0.16929E-03 0.0000 110E-04 0.18206 0.3005E-03 0.5607E-04 0.0000 127E-04 0.18206 0.3005E-03 0.5607E-04 0.0000 170E-04 0.18206 0.30047E-03-0.14238E-05 0.0000 170E-04 0.18206 0.30047E-03-0.155975E-04 0.0000 114E-04 0.18205 0.74157E-03-0.16299E-03 0.0000	773E-04 0.18545 0.16603E-02 0.36055E-05 0.0000 0.0000 164E-04 0.18522 0.23258E-02 0.22180E-04 0.0000 0.0000 106E-04 0.18205 0.74230E-03 0.16923E-03 0.0000 0.0000 110E-04 0.18205 0.74230E-03 0.16923E-03 0.0000 0.0000 110E-04 0.18206 0.30056E-03 0.54607E-04 0.0000 0.0000 27E-04 0.18206 0.15382E-03-0.14236E-05 0.0000 0.0000 70E-04 0.18206 0.30047E-03-0.15537E-04 0.0000 0.0000 70E-04 0.18205 0.74157E-83-0.16939E-03 0.0000 0.0000

If you take the average of the stress in the y-direction over the thickness of the tube, (0.18552 + 0.17866)/2, the stress in the tube is 0.182 MPa, matching the analytical solution. The average is used because in the analytical case, it is assumed the stress is evenly distributed across the thickness. This is only true when the location is far from any stress concentrators, such as corners. Thus, to approximate the analytical solution, we must average the stress over the thickness.

Ex. No. : 4 Date:

MODAL ANALYSIS OF A CANTILEVER 2D-PLATE

Step 1: Set preferences.

1. Main Menu>Preferences

- **2.** Turn on structural filtering. The options may differ from what is shown here since they depend on theANSYS product you are using.
- **3.** OK to apply filtering and close the dialog box.

Step 2: Define element types and options.

1. Main Menu> Preprocessor> Element Type> Add/Edit/Delete

- 2. Add an element type.
- 3. Structural solid family of elements.
- 4. Choose the 8-node quad (PLANE183).
- 5. OK to apply the element type and close the dialog box.
- 6. Options for PLANE183 are to be defined.
- 7. Choose plane stress with thickness option for element behavior.
- 8. OK to specify options and close the options dialog box.
- **9.** Close the element type dialog box.

Step 3: Define material properties.

- 1. Main Menu> Preprocessor> Material Props> Material Models
- 2. Double-click on Structural, Linear, Elastic, Isotropic.
- 3. Enter 30e6 for EX
- 4. Enter 0.27 for PRXY.
- 5. OK to define material property set and close the dialog box.
- 6. Density = 7.8e6
- 7. Material > Exit

Step 4: Build Geometry.

- 1. Main Menu> Preprocessor> Modeling> Create> Areas> Rectangle> By Dimensions
- 2. Enter the following:
 - X1 = 0 (Note: Press the Tab key between entries)
 - X2 = 10
 - Y1 = 0

Y2 = 2.5

- 3. Apply to create the first rectangle.
- 4. OK to create the second rectangle and close the dialog box.

Step 5: Generate Mesh.

Main Menu> Preprocessor> Meshing> Mesh Tool

 Set area element edge length to 1 and select mapped mesh option and mesh the area.

Step 6: Solution: Assigning Loads and Solving.

- 1. Define Analysis Type
 - Solution > Analysis Type > New Analysis > Modal ANTYPE,2
- 2. Set options for analysis type:
 - Select: Solution > Analysis Type > Analysis Options...
 - The following window will appear.

▲ Modal Analysis	
[MODOPT] Mode extraction method	
	Block Lanczos
	C PCG Lanczos
	C Reduced
	C Unsymmetric
	C Damped
	O QR Damped
	C Supernode
No. of modes to extract	5
(must be specified for all methods except the Re	luced method)
[MXPAND]	
Expand mode shapes	✓ Yes
NMODE No. of modes to expand	5
Elcalc Calculate elem results?	□ No
[LUMPM] Use lumped mass approx?	□ No
[PSTRES] Incl prestress effects?	□ No
OK	Cancel Help
	Tiep

- As shown, select the Block Lanczos method and enter 5 in the 'No. of modes to extract'
- Check the box beside 'Expand mode shapes' and enter 5 in the 'No. of modes to expand'
- Click 'OK'.

The following window will then appear.

ĺ	N Block Lanczos Method	J
	[MODOPT] Options for Block Lanczos Modal Analysis	
	FREQB Start Freq (initial shift) 0	
	FREQE End Frequency 0	
	Nrmkey Normalize mode shapes To mass matrix	
	OK Cancel Help	
		_

- For a better understanding of these options see the Commands manual.
- For this problem, we will use the default options so click on OK.

3. Apply Constraints

- Solution > Define Loads > Apply > Structural > Displacement > On Keypoints
- Fix Keypoint 1 (i.e. all DOFs constrained).
- 4. Solve the System
 - Solution > Solve > Current LS

Step 7: Postprocessing: Viewing the Results.

- 1. Verify extracted modes against theoretical predictions
 - Select: General Postproc> Results Summary
- 2. View Mode Shapes
 - Select: General Postproc> Read Results > First Set
 - This selects the results for the first mode shape
 - Select General Postproc> Plot Results > Deformed shape. Select 'Def + undefedge'
 - The first mode shape will now appear in the graphics window.
 - To view the next mode shape, select General Postproc>ReadResults> Next Set. As above choose General Postproc>PlotResults> Deformed shape. Select 'Def + undef edge'.

3. Animate Mode Shapes.

- Select Utility Menu (Menu at the top) > Plot Ctrls> Animate > Mode Shape
- The following window will appear

Animation data		
No. of frames to create	10	
Time delay (seconds)	0.5	
Acceleration Type		
	💽 Linear	
	🔿 Sinusoidal	
Nodal Solution Data		
Display Type	DOF solution Stress Strain-total Energy Strain-elastic Strain-thermal Strain-plastic Strain-creep	Deformed Shape Def + undeformed Def + undef edge Translation UX UZ UZ USUM
	Jacrain Other	
07	Cases 1	Hele
UR	Cancel	нетр

Keep the **default setting** and click 'OK'. Then animated mode shapes

MODAL ANALYSIS OF A CANTILEVER BEAM

44 | P a g e



Modulus of Elasticity (E) = $206800(10^6)$ N/m²

 $Density = 7830 \text{ kg/m}^3$

Step 1: Preprocessing: Defining the Problem.

1. Main Menu>Preferences

- 2. Turn on structural filtering. The options may differ from what is shown here since they depend on the ANSYS product you are using.
- 3. OK to apply filtering and close the dialog box.

Step 2: Define element types and options.

- 1. Main Menu> Preprocessor> Element Type> Add/Edit/Delete
- 2. Add an element type.
- 3. Structural beam family of elements.
- 4. Choose the BEAM189.
- 5. OK to apply the element type and close the dialog box.

Main Menu> Preprocessor> Preprocessor > Sections > Beam > Common Sections

- 1. Enter B=0.01
- 2. Enter H=0.01
- 3. Ok

Step 3: Define material properties.

1. Main Menu> Preprocessor> Material Props> Material Models

- 2. Double-click on Structural, Linear, Elastic, Isotropic.
- Enter 206800e6 for EX.
 Enter 0.27 for PRXY.
- 5. Ok

Date:

- 6. Enter 7830 for density
- 7. OK to define material property set and close the dialog box.
- 8. Material > Exit

Step 4: Modeling.

- 1. Main Menu> Preprocessor> Modeling > Create >keypoints> in active cs Createkeypoint 1 at 0,0,0 and create keypoint 2 at 1,0,0
- 2. Main Menu> Preprocessor> Modeling> Create>line>straight line Create line throughkeypoint 1 to 2

Step 5: Meshing.

- 1. Main Menu> Preprocessor>meshing> mesh tool
 - Set the no of element division to 100 in line set controls and mesh the line

Step 6: Solution: Assigning Loads and Solving

1. Define Analysis Type

- Solution > Analysis Type > New Analysis > Modal ANTYPE,2
- 2. Set options for analysis type
 - Select: Solution > Analysis Type > Analysis Options
 - The following window will appear

Modal Analysis	
[MODOPT] Mode extraction method	
	Block Lanczos
	O PCG Lanczos
	O Reduced
	O Unsymmetric
	O Damped
	O QR Damped
	O Supernode
No. of modes to extract	5
(must be specified for all methods except the Reduced meth	nod)
[MXPAND]	
Expand mode shapes	✓ Yes
NMODE No. of modes to expand	5
Elcalc Calculate elem results?	□ No
[LUMPM] Use lumped mass approx?	∏ No
[PSTRES] Incl prestress effects?	□ No
OK Can	Icel Help

- As shown, select the **Block Lanczos**method and enter 5 in the 'No. of modes to extract'
- Check the box beside 'Expand mode shapes' and enter 5 in the 'No. of modes to expand'
- Click 'OK'
- The following window will then appear

A Block Lanczos Method	×	J
[MODOPT] Options for Block Lanczos Modal Analysis		
FREQB Start Freq (initial shift) FREQE End Frequency	0	
Nrmkey Normalize mode shapes	To mass matrix	
OK Cancel	Help	Ş

- For a better understanding of these options see the Commands manual.
- For this problem, we will use the default options so click on OK.

3. Apply Constraints

- Solution > Define Loads > Apply > Structural > Displacement > On Keypoints
- Fix Keypoint 1 (i.e. all DOFs constrained).

4. Solve the System

• Solution > Solve > Current LS SOLVE

Step 7: Postprocessing: Viewing the Results.

- 1. Verify extracted modes against theoretical predictions
 - Select: General Postproc> Results Summary...
 - The following window will appear

MHRHH INDEX OF DATA SETS ON RESULTS FILE MHRHH SET TIME/FRED LOAD STEP SUBSTEP CUMULATIUE 1 8.3000 1 1 1 2 52.011 1 2 2 3 145.64 1 3 3 4 245.51 1 4 4 5 472.54 1 5 5	
SET TIME/FREQ LOAD STEP SUBSTEP CUMULATIVE 1 8 3000 1 1 1 2 52.01 1 2 2 3 145.64 1 3 3 4 285.51 1 4 4 5 472.54 1 5 5	
1 8,3000 1 1 1 2 52,011 1 2 2 3 145,64 1 3 3 4 285,51 1 4 4 5 472,54 1 5 5	
2 52.011 1 2 2 3 145.64 1 3 3 4 285.51 1 4 4 5 472.54 1 5 5	
3 145.64 1 3 3 4 285.51 1 4 4 5 472.54 1 5 5	
4 285.51 1 4 4 5 472.54 1 5 5	
5 472.54 1 5 5	

• The following table compares the mode frequencies in Hz predicted by theory and ANSYS.

Note: To obtain accurate higher mode frequencies, this mesh would have to be refined even more (i.e. instead of 10 elements, we would have to model the cantilever using 15 or more elements depending upon the highest mode frequency of interest).

2. View Mode Shapes

- Select: General Postproc> Read Results > First Set
 > This selects the results for the first mode shape
- Select General Postproc> Plot Results > Deformed shape. Select 'Def + undef edge'
- The first mode shape will now appear in the graphics window.
- To view the next mode shape, select General Postproc> Read Results > Next Set . As above choose General Postproc> Plot Results > Deformed shape. Select 'Def + undef edge'.
- The first four mode shapes should look like the following:

		2	
DISPLACEMENT	ANSYS 5.7.1	DISPLACEMENT	ANSYS 5.7.1
STEP=1		STEP=1	
SUB =1		SUB =2	
FREQ=8.3		FREQ=52.011	
DMIX =2.25		DMX =2.25	
Y		Y	
e x		k X	
		4	
DISPLACEMENT	ANSYS 5.7.1	DISPLACEMENT	ANSYS 5.7.1
STEP=1		STEP=1	
3DB =3		SUB =4	
FREQ=145.538		FREQ=285.513	
DMIX =2.25		DMX =2.253	
Y		Y	
e x		z x	

3. Animate Mode Shapes

• Select **Utility Menu (Menu at the top) > Plot Ctrls> Animate > Mode Shape** The following window will appear

Animation data		
No. of frames to create	10	
Time delay (seconds)	0.5	
Acceleration Type		
	💽 Linear	
	🔿 Sinusoidal	
Nodal Solution Data		
Display Type	DOF solution Stress Strain-total Energy Strain-elastic Strain-thermal Strain-creep Strain-creep Strain-other	Deformed Shape Def + undeformed Def + undef edge Translation UX UZ UZ USUM Deformed Shape
OK	Cance 1	Help

- Keep the default setting and click 'OK'
- The animated mode shapes are shown below.

Ex. No. : 6 Date:

MODAL ANALYSIS OF A SIMPLY SUPPORTED BEAM



Modulus of Elasticity (E) = $206800(10^6)$ N/m²

Density = 7830 kg/m^3

Step 1: Preprocessing: Defining the Problem.

- 1. Main Menu>Preferences
- 2. Turn on structural filtering. The options may differ from what is shown here since they depend on the ANSYS product you are using.
- 3. OK to apply filtering and close the dialog box.

Step 2: Define element types and options.

- 1. Main Menu> Preprocessor> Element Type> Add/Edit/Delete
- 2. Add an element type.
- 3. Structural beam family of elements.
- 4. Choose BEAM189.
- 5. OK to apply the element type and close the dialog box.

Main Menu> Preprocessor> Preprocessor> Sections> Beam> Common Sections

- 1. Enter B=0.01
- 2. Enter H=0.01
- 3. Ok

Step 3: Define material properties.

- 1. Main Menu> Preprocessor>Material Props> Material Models
- 2. Double-click on Structural, Linear, Elastic, Isotropic.
- 3. Enter 206800e6 for EX.
- 4. Enter 0.27 for PRXY.
- 5. Enter 7830 for density.
- 6. OK to define material property set and close the dialog box.
- 7. Material > Exit

Step 4: Modeling.

- 1. Main Menu> Preprocessor> Modeling> Create>key points>in active cs Create key point 1 at 0,0,0 and create key point 2 at 1,0,0
- 2. Main Menu> Preprocessor> Modeling> Create>line>straight line Create line through key point 1 to 2

Step 5: Meshing.

- 1. Main Menu> Preprocessor>meshing>mesh tool
 - Set the no of element division to 100 in line set controls and mesh the line

Step6: Solution: Assigning Loads and Solving.

1. Define Analysis Type

• Solution > Analysis Type > New Analysis > Modal ANTYPE,2

2. Set options for analysis type:

•	Select:	Solution	>	Analy	/sisˈ	Туре	2	Anal	ysis	Op	otions
---	---------	----------	---	-------	-------	------	---	------	------	----	--------

The following window will appear

▲ Modal Analysis		
[MODOPT] Mode extraction method		
	 Block Lanczos 	
	C PCG Lanczos	
	C Reduced	
	O Unsymmetric	
	C Damped	
	C QR Damped	
	O Supernode	
No. of modes to extract	5	
(must be specified for all methods except the R	Reduced method)	
[MXPAND]		
Expand mode shapes	🔽 Yes	
NMODE No. of modes to expand	5	
Elcalc Calculate elem results?		
[LUMPM] Use lumped mass approx?	∏ No	
[PSTRES] Incl prestress effects?	∏ No	
ок	Cancel Help	
<u> </u>	Cancel Help	

- As shown, select the Block Lanczos method and enter 5 in the 'No. of modes to extract'
- Check the box beside 'Expand mode shapes' and enter 5 in the 'No. of modes to expand'
- Click 'OK'

The following window will then appear.

A Block Lanczos Method	×
[MODOPT] Options for Block Lanczos Modal Analysis	
FREQB Start Freq (initial shift) FREQE End Frequency	0
Nrmkey Normalize mode shapes	To mass matrix 💌
OK Cancel	Help

- For a better understanding of these options see the Commands manual.
- For this problem, we will use the default options so click on OK.

3. Apply Constraints

 Solution > Define Loads > Apply > Structural > Displacement >On Keypoints

fix Key point 1 and Key point 2 (i.e.**ux,uy constrained).**

4. Solve the System

- Solution > Solve > Current LS
- SOLVE

Step7: Postprocessing: Viewing the Results.

- **1.** Verify extracted modes against theoretical predictions
 - Select: General Postproc> Results Summary
- 2. View Mode Shapes

- Select: General Postproc> Read Results > First Set This selects the results for the first mode shape
- Select General Postproc> Plot Results > Deformed shape. Select 'Def + undef edge'
- The first mode shape will now appear in the graphics window.
- To view the next mode shape, select General Postproc> Read Results > Next Set.
- As above choose General Postproc> Plot Results > Deformed shape. Select 'Def + undef edge'.

3. Animate Mode Shapes

• Select **Utility Menu (Menu at the top) > Plot Ctrls> Animate > Mode Shape** The following window will appear

No. of frames to create	10		
Time delay (seconds)	0.5		
Acceleration Type			
	💽 Linear		
	🔘 Sinusoidal		
Nodal Solution Data			
Display Type	DOF solution Stress Strain-total Energy Strain-elastic Strain-thermal Strain-plastic Strain-creep Strain-coher	Deformed Shape Def + undeformed Def + undef edge Translation UX UZ UZ USUM Deformed Shape	

- Keep the default setting and click 'OK'
- The animated mode shapes

Ex. No.: 7

MODAL ANALYSIS OF A FIXED BEAM

51 | P a g e

Date:



Modulus of Elasticity (E) = 206800(106) N/m2

Density = 7830 kg/m³

Step 1: Preprocessing: Defining the Problem.

1. Main Menu>Preferences

- 2. Turn on structural filtering. The options may differ from what is shown here since they depend on the ANSYS product you are using.
- 3. OK to apply filtering and close the dialog box.

Step 2: Define element types and options.

- 1. Main Menu> Preprocessor> Element Type> Add/Edit/Delete
- 2. Add an element type.
- 3. Structural beam family of elements.
- 4. Choose BEAM189.
- 5. OK.

Main Menu > Preprocessor > Preprocessor > Sections > Beam > Common Sections

- 1. Enter B=0.01
- 2. Enter H=0.01
- 3. Ok

Step 3: Define material properties.

- 1. Main Menu> Preprocessor> Material Props> Material Models
- 2. Double-click on Structural, Linear, Elastic, Isotropic.
- 3. Enter 206800e6 for EX.
- 4. Enter 0.27 for PRXY.
- 5. Enter 7830 for density.
- 6. OK to define material property set and close the dialog box.
- 7. Material > Exit

Step 4: Modeling.

- Main Menu> Preprocessor> Modeling> Create>key points>in active cs Create key point 1 at 0,0,0 and create key point 2 at 1,0,0
- Main Menu> Preprocessor > Modeling > Create > line > straight line Create line through key point 1 to 2

Step 5: Meshing.

 Main Menu> Preprocessor>meshing>mesh tool Set the no of element division to 100 in line set controls and mesh the line

Step6: Solution: Assigning Loads and Solving.

- 1. Define Analysis Type
 - Solution > Analysis Type > New Analysis > Modal ANTYPE,2
- 2. Set options for analysis type:
 - Select: Solution > Analysis Type > Analysis Options.

🔨 Modal Analysis		×
[MODOPT] Mode extraction method		
	Block Lanczos	
	O PCG Lanczos	
	C Reduced	
	O Unsymmetric	
	O Damped	
	C QR Damped	
	C Supernode	
No. of modes to extract	5	
(must be specified for all methods except the Red	duced method)	K
[MXPAND]		
Expand mode shapes	Ves	
NMODE No. of modes to expand	5	
Elcalc Calculate elem results?	□ No	
[LUMPM] Use lumped mass approx?	∏ No	
[PSTRES] Incl prestress effects?	□ No	
ок	Cancel Help	

- As shown, select the Block Lanczos method and enter 5 in the 'No. of modes to extract'
- Check the box beside 'Expand mode shapes' and enter 5 in the 'No. of modes to expand'
- Click 'OK'
- The following window will then appear

RIock Lanczos Method	×
[MODOPT] Options for Block Lanczos Modal Analysis	
FREQB Start Freq (initial shift) FREQE End Frequency Nrmkey Normalize mode shapes	0 0 To mass matrix
OK	Help

- For a better understanding of these options see the Commands manual.
- For this problem, we will use the default options so click on OK.

3. Apply Constraints.

• Solution > Define Loads > Apply > Structural > Displacement > On Keypoints fix Key point 1 and Key point 2.

4. Solve the System

• Solution > Solve > Current LS

SOLVE

Step7: Postprocessing: Viewing the Results.

- 1. Verify extracted modes against theoretical predictions
 - Select: General Postproc> Results Summary
- 2. View Mode Shapes
 - Select: General Postproc> Read Results > First Set
 - This selects the results for the first mode shape
 - Select General Postproc> Plot Results > Deformed shape. Select 'Def + undef edge'
 - The first mode shape will now appear in the graphics window.
 - To view the next mode shape, select General Postproc> Read Results > Next Set. As above choose General Postproc> Plot Results > Deformed shape. Select 'Def + undef edge'.

3. Animate Mode Shapes

• Select Utility Menu (Menu at the top) > Plot Ctrls> Animate > Mode Shape

The following window will appear

Animation data		
No. of frames to create	10	
Time delay (seconds)	0.5	
Acceleration Type		
	Linear	
	🔿 Sinusoidal	
Nodal Solution Data		
Display Type	DOF colution Stress Strain-total Energy Strain-thernal Strain-thernal Strain-thernal	Deferred Shape Def + undeferred Def + undeferred Translation UX UV UZ USUM 2
	Strain-other	 Deformed Shape

- Keep the default setting and click 'OK'
- The animated mode shapes

Ex. No.: 8

THERMAL-MIXED BOUNDARY

54 | P a g e

Date:

(Conduction/Convection/Insulation)





Preprocessing: Defining the Problem

- 1. Give example a Title
- 2. Open preprocessor menu

ANSYS Main Menu > Preprocessor /PREP7

3. Create geometry

 Preprocessor > Modeling > Create > Areas > Rectangle > By 2 Corners X=0, Y=0, Width=1, Height=1 BLC4,0,0,1,1

4. Define the Type of Element

 Preprocessor > Element Type > Add/Edit/Delete... > click 'Add' Select Thermal Mass Solid, Quad 4Node 55 ET, 1, PLANE55

As in the conduction example, we will use PLANE55 (Thermal Solid, Quad 4node 55). This element has 4 nodes and a single DOF (temperature) at each node. PLANE55 can only be used for 2 dimensional steady-state or transient thermal analysis.

5. Element Material Properties

- Preprocessor > Material Props > Material Models > Thermal >Conductivity > Isotropic > KXX = 10
- MP,KXX,1,10
- This will specify a thermal conductivity of 10 W/m*C.

6. Mesh Size

 Preprocessor > Meshing > Size Cntrls>Manual Size> Areas > All Areas > 0.05 AESIZE,ALL,0.05

7. Mesh

 Preprocessor > Meshing > Mesh > Areas > Free > Pick All AMESH,ALL

Solution Phase: Assigning Loads and Solving

1. Define Analysis Type

- Solution > Analysis Type > New Analysis > Steady-State ANTYPE,0
- 2. Apply Conduction Constraints
 - In this example, all 2 sides of the block have fixed temperatures, while convection occurs on the other 2 sides.
 - Solution > Define Loads > Apply > Thermal > Temperature > On Lines
 - Select the top line of the block and constrain it to a constant value of 500°C Using the same method, constrain the left side of the block to a constant value of 100°C

3. Apply Convection Boundary Conditions

- Solution > Define Loads > Apply > Thermal > Convection > On Lines
- Select the right side of the block.

The following window will appear:

R Apply CONV on lines	×
[SFL] Apply Film Coef on lines	Constant value 💌
If Constant value then:	
VALI Film coefficient	10
[SFL] Apply Bulk Temp on lines	Constant value 💌
If Constant value then:	
VAL2I Bulk temperature	100
If Constant value then:	
Optional CONV values at end J of	line
(leave blank for uniform CONV)	
VALJ Film coefficient	
VAL2J Bulk temperature	
OK Apply Cancel	l Help

- Fill in the window as shown. This will specify a convection of 10 W/m2*C and an ambient temperature of 100 degrees Celsius. Note that VALJ and VAL2J have been left blank. This is because we have uniform convection across the line.
- 4. Apply Insulated Boundary Conditions
 - Solution > Define Loads > Apply > Thermal > Convection > On Lines
 - Select the bottom of the block.
 - Enter a constant Film coefficient (VALI) of 0. This will eliminate convection through the side, thereby modeling an insulated wall. Note: you do not need to enter a Bulk (or ambient) temperature



5. Solve the System

- Solution > Solve > Current LS
- SOLVE

Postprocessing: Viewing the Results

1. Results Using ANSYS

• Plot Temperature

General Postproc> Plot Results > Contour Plot > Nodal Solution> DOF solution, Temperature TEMP



57 | P a g e

Ex. No. : 9 Date:

Problem Description

Determine the nodal deflections, reaction forces, and stress for the truss system shown below (E = 200 GPa, A = 3250 mm^2).



Ex. No. : 10 Date:

Modal Analysis

Problem Description

ANSYS has an option that will allow for the masses to be modeled as point masses. Using this option, no dimensions or material properties would be necessary for the masses. However, so that the mode shapes are easier to understand when they are animated, in this case, the masses will be modeled as blocks, 1 m x 1 m, with unit thickness. So, they have a volume of 1 m^3 . The densities will be specified to produce the correct masses. Also, the free lengths of the springs are irrelevant in this analysis, as only the stiffness matter. But, they will both be assumed to have a length of 5 m.



Ex. No.: 11 **Stress Analysis on a Cantilever beam with Pipe Element Date:**

Problem Description

EX = 70000, PRXY=0.33,



Problem Description

Date:

Find the temperature distribution for the given figure. Thermal conductivity (k) of the material is5W/mK and the block is assumed to be infinitely long. Also, the density of the material is 920 kg/m³ and the specific heat capacity (c) is 2.040 kJ/kgK



59 | P a g e

Ex. No. : 13 Date:

Problem Description

Consider a rectangle plate of uniform thickness with a circular hole of radius 20mm as shown in figure. The thickness of the plate is 1 mm. The Young's modulus E =10e7 MPa and the Poisson ratio is 0.33. Assume plane stress condition.



Ex. No. : 14 Date:

Stress Analysis in a bicycle frame

Problem Description

Find the stress distribution in the bicycle frame with outer diameter as 30mm and wall thickness as 3mm. Take $E = 70000 \text{ N/mm}^2$, u=0.33





P.S.R. ENGINEERING COLLEGE SEVALPATTI - 626 140 SIVAKASI

Virudhunagar District.

BONAFIDE CERTIFICATE

Certified that this is a Bonafide Record of work done

by M: BASILAR

Roll No. 18ME009 in the ISIMETT - Computer Aided Analysis

Laboratory of this College during the academic year 2021 - 2022.

P.8 8 2/11/21 Staff - in - Charge

Head Of the Department Department of Mechanical Engineering Held of the giberting College Sevalpatti, Sivakasi - 626 140

Register No. 1806009

Internal Examiner

R1.111.115 External Examiner

I 15/9/2011 Stress analysis of cantilevon boam	Page No.	Marks	Signatura
15/9/2021 introduction & analys 1 15/9/2021 Stress analysis & cantilition boam	Page No.	Marks	Stomanieal
15/9/2021 Introduction & analys 1 15/9/2021 Stress analysis & cantilevon boam	1		Signature
1 15/9/2011 Stress analysis of cantilevon beam			1
	9	21	PAC Fee
2 15191202) Stress analysis & Fixed boram	13,	21	Ptol \$221
5, 22/9121 sticss analysis & simple supported boarn with udy	17	21	PALEDA
4, 22191222) Stress analysis & Jruss elemone	81	26	PALESA
5 2919121 Stress analysis & plate with hole	85	24	P&LFis/
6 2919121 Analysis of AXI- Symmetric doman	29	24	P.S.Fislu
7, 13/10/21 madel analysis & Fixes boam	33.	28	Polypol
8 13/10121 modal analysis & canviliever boarn	32	24	P817201
9 20/10/21 Mornal mixed boundary	57,	25	RSlg2
10, 2010/21 hormonic analysis of vanities of boam	431	26	Polpon/
11, 2711000 model analysis & simple supporter bocon	45	24	Rele
12 2710121 modal analysis 8 20 place with cantilitiest support	49,	24	Pal 30

Exp. No.: Page No.:) Date · INPRODUCTION OF ANSYS ANSYS is a arcronal purpose Anal domant monalling Pocking for numerically solving & mide volocity of majoria problem. more problem include staric dynames structural analysis for born linier and non - silinter has transfer & Fluid Pro bloms in accord a finite channel solution many be broken into tollowing three signals mis 15 9 array availabling resultants entities and printer rate day ad new tam processing dorinding position!-The major step in pre - processing are minon bolam is define han point linge (and volume is define element type and material (heresmatric proper iii) most ling laves (volume are required ive manuel of details required depends on dimensionally & mo analysis (10:20:30) Solution Assing Land Construct: have use specify the Loope (points) pressure constrain finally solve the resulting set & amorrion post projessing, in this stage and may with to 1) List & Look displayment. is stamon of force and moment. Ilis dollaction.

Exp. No. 3 Page No ... ? Date : ivo stress combon diagram. ANSYS 14'S EDVIJOD DENIL. MOLAND "2" UI TELUR " TOF MOM MOTION ZEERD ON 8 mindanne rea main mindanne and an oursput mindanne Note mat is some what differs from the previous varian 5 answs which make use ditt window. main midanus! The main many 's' division as wrom kiliture it contrain Tunction such as avuilable throwshow no anys session such as fine analysis control selamon. Carophic Contains. binspeur mindami-Las output line show prostam orastor and allows you to type is commande directily. citod loof co it contains push builton may exercute commonly BNISHS' commandes can be added it datindat. do main monuis it contains the primary MNSYJ Software abrammas eavorants noinunios lozzesione ere porsinaero an stor you way and it is an source an introped change bin prentions session prosos 145

Date · our put mindami. it show the output From program such as listaning & data acc. it will would positional bahira me main mindom and can desporte to Front. ANSYS INTOTADU . GREPHICAL INTERFACE WSS COMMAND TILE COEDING! There are two mornado to use ANTYS returned a strating con advident of anomy interface connection to popular window and y- direction programs teau both recommanded file many you will file command file. have - trable to content -> command manual. FEM CONVOISOND Techenique! A Findamanda Promise or using to Files stan pobision dure 21 probot and tant 21 arub and mando small districtive region and more elements and assembled kirkam Ladom ofni Constantios :-Tibon mostos comos with a cost allawance more calcularing time and large momenty requirment Beam models: To boom module was alticulary mades to define a single domaner for line unlaw use for applying a distributor loved. mis until only flow the columnon down for simple moras an converse but the larson meral.

Exp. No. 7 Page No.: 7 Date : monoral models: In moronal the small will conversione more simuly thank the displayment so it not sufficient to Cramina. solving and resolving Jobs !. solving your Job: it will be crood prairile to are your moder at various point during its marions vory often you may like to salve it all the point in this you make some mistalles after on you allow to able to Common base to This point Resuming a pretimizing solved Job! Recalling Frequently you want to Start up ANISYI and revell and continue a previous Job Miles are tuo morrodos are usins no banchar. in the answer to the interactive and spectre no dotind Tab. man ma that stands schow whility money -> File -> Resume Jobs. on store Anusys and solow willity The - > relune From and selace your Job From lift total approxime. Jist9/2021 Result'. Thus the develop stundy & "ANSYS" was studies.



FALL DIMENSION FIRE IN MM.

E = 8x105 NIMm

V=0.3

1=100mm

a=10mm

b=10mm.

F= - 1000

xp. No.:		Page No.:9
ST	ress analysis of cantilever be	กัก
Aim'r		
to analy	sis no stress indunal on the m	iven boom with
me univer Love	2 Condition wint ANSYS Soft were	•
soft mare un	£!.	
	0272 17.2	
PROCEDURE !.	The same	a man and
ບມານ	y moon -> The - change hitle -> 1	Enter how tritte.
ບມາແທ	A worn -) charge for wow -> eur	or Job Name
stop 8 ! PRETE	ienie !-	
P	<u>RETERENCE -) STRUCTURED -) OX.</u>	
Steps: Pren	DTO (CSSON !.	
F	ic processor -> domony type -> Agg	1 Edit 1 dolore 13 com
בזסזק בזק	tommos (- mass (- nomess (- not	
	ENTO Delo	
	Entor H=10	
PICPIOLE	501 -> modeling -> cruat -> vas paint) Inclute
pre prous	12 (- anil (- ban (- muldom (- roi	aist line
preprotezza	-> moshing -> size Content -> mo	ruma sizo lino->
Pre ploi es pr	-> LODA -> doring LODA -> OPPIY	-> Tuultera ->
	and branch ou thank boling	1
	OTEDIOIOUTOUT OU LIGIT DOLLA	•

ANALYTICAL METHODE !.

and an and the in the state of the state of the

Shoon FOTU SE:-N

ST= -LOON

Conding moment = WL

= -100×100

=-10000 N-mm

momore g induita $I = bd^{3}$ 12 12 12 $I = 8333 \cdot 33 \text{ mm}^{3}$

> displacement n = w2 = 100 ×10³ 3 ×2×10⁵ ×833.83 N = 0.20mm.

STRESS ANALYSIS OF CANTILIVER BEAM





e: 15/9/12021		P	'age No.:
PTE PTOLESSOY ->)	- Socal britad c- Bas) FIDDIN -) CANULL	nifto -)
	Jor u momont	hanneles.	
		trad hottor .	
Stork :- Solution!			
innia	ions -) SOIV (-) CULITON	15-2016.	
Steps: Otenanal p	ost pictoronu !.		
CTOR	mal post proc -> plot	mrozob c- wazan	shape.
Chengra	post pro poste -	ACT with > Contou	un plate ->
	3 Plois	line -lomon.	Mansen .
CREMTERS	post pro - , domans t	mble -2 doring .	toble - dire
			odd 3,6,16119.
	A Martin	A Company of the second	
		1 12	;
	State 1 67%	12	
	Star Star	,	
Domith		1	
dataining	displaymont (mm)	Shoop Toylo Law	ben diere mome
	USPA OTTALY OTTAL		
Throwstiene and up			
noonna youro			
		1	
Timing 0 0			
Explored voluo:		Jan a	
Trust	10 Strezz analysis u	ups along for	ma
ouven contilivor	boam in Ansyls 1	114.2 Software.	
		PRACTICAL	VALUATION
		Observation (10)	
		Calculation / Resu	
		Viva (10)	0.8
		Total Marks (30)	Del P
		Signature of Facu	ity motols

stress analysis & fixed boom :. 11001 *0 50 50 -16-11 VO PROSTA ALL DIMENSIONS ARE TO MM E= 2 x10 5 NIMM 220.3 aslomm 1 b=10mm. anad the set with stand and standing the aginiendo
No.: 8			Page No.:13
3	MRESS ANALYSIS O	TIXED BEAM.	
Bim!-			
to analui	is no shoan ind	in an no som	iven boam with
no univen con	dirig using analys	· SOFT WORD ·	
soft ware une	se : -		
P	ANSYT 14-5	-6124	
PROIOQUITO :-			
Step1: wi	ility monu:-		
سانان	ty monu -> Tile -)	tionse litre ->	Enter now trice.
whit	14 monu -) Chanse	Job namo ->	ons or Job name.
Stop 0: PREFOR	<u>conos</u> !,		
	PICTOTORIO -> Struit	-> 0K.	
Steps: piep	270165501 !~		
	PIC PIOLETION -> de	mons type -) (2001 chet 1 daleu ->
	Ba	am 188-close.	
	\bigcap		
		A second second second second	
prc	moreson -> securion	boam -> Com	amon seurion.
Prc	Processon -> securion B=1	boom -> Com	amon seunar.
Prc	Processon -> securion B=1 HII	boom -> Com	amon seunar
Dic Pie p	PROCESSON -> SCURTON B=1 H=1 H=1 MOLESSON -> MOLEMAN	boom -> Com	utoring mades -)
prc Pre pr	PROCESSON -> SCURTON B=1 H=1 H=1 H=1 H=1 H=1 Ex = 2.6	eli	amon securar.
prc Pie pr	Ex = 2.6 Prug = 0'	boom -> Com x props -> mo cl) 3.	amon seunan. Arochan (madale -)
Pre pre	Production -> securitor B=1 H=1 H=1 TOLESSON -> MONDAR Ex =2.6 Prod = 0. 250 -> Modoliling	boom -> Com 2 prop 5 -> mc cl) 3. -> URCOS -> DO	HEDDAL -2 inclusive.





STRESS ANALYSIS OF FIXED BEAM



and the second second				Page No.	and the second
Exp. Date	No. Account of the second	All		and the second sec	
	keypoint	104 (01010)			
	Loss poine	01 (101010)			
	Im. Onior C				
	Tally Point				
		molouine	lices point -> line	o-jstiino	
	preprocessor -3	alot drom c-zoid zon			
	Preporezzov - 7	Tax longround OD W	DOIN -> PODA	- 44 -100-di.	
ac a tanàna amin'ny fisiana	bie biorerran	tore (monorm or p	and built		
				1.0.1	
	steph: solution	1,1~			
	Solu	union -> solve -> cu	MUTON 12-JOK.	arrangie d.	
	Stepsin Orenau	o post processory.		1 1 2	
	UTCH UTCH	c- lord trog baron	proje result -	2 DOM TO JOB C	none
		determos - , undo	Jormon - Joursof		
			Wylar All		
	(nerout	0 DOZT DIDIE -> CON	tals pay - mod	armunica Par	
		DOT COlumion -	NK		
	1 hus	DEL - CODIOUE - 2 M	admo solution	IS STICKS CON	prot
	areading his				
	Result:-			N. ~~~~	
	doscription	gezblarowork (mm)	S hoar Frace (N)	Bandins mome	in n-in
	Transical value.				6
	Exprimonanevalue				1
	Thus The	w TRuban TTAN	a dans Tar	autras	
	HIXED born	In ANSUT MUST ON	Fluente for t		
			1	PRACTICAL EVAL	ATION
		1		Observation (10)	7
				Calculation / Result (10)	7
			h	/iva (10)	7
				fotal Marks (30)	21
				Cianature of Faculty	mel





E= 87105 NIMM

2=0.3

ALL DIMENSIONS ARE IN MMI.

1.E

- allor patronal

(11 (17) × 19) (1

1207

Contractor

a= 10mm

p=10mm.

Exp. N	io. 5	Page No
Date :.	22(4)(0)1 (10) (10) (10) (10) (10) (10) (10)	0.0
	Stierss builders of 220 Brunn OUTH ON CO	ην
	Aim:	
	to analysis no stress indural on no unive	n boam with
	mo criven condition using analysis software.	
	Sottware use!.	100
	4N2A2 14.2	
	- 10 X CON -	
P	rocoluro:	
	stepl: whility menu:	
	writity monu -> Tile-> showe title -> E	אסו הסוט האפ
	writity monu -, change Job Name -, Enter	Job Name.
	S. Hits The second	
S	terd, pietoinu:.	
	PICTOTONO-) STILL+UROA-JOK.	
S	epz: pieprolessi:	
	preprocessor -> domand type -> Add I Edit	: 1 dolar - 20 00m.
	preprocessor -> s clouton -> Beam -> Common	scioudi
	Enter B= 1	
1	Emor H=1	
P	eprocessor -) motoclas pops -) maintains mo	dou lines
d	ask zes and orston pros	
	preprojetto -> modauine -> incas -> loupoin	-> Inaute (3
	less point (projo) and ever lipupoint (210,0)	
	preprotection the double - > thend line - and	anne line
	,	
		the same is the same is a same

ANALYTICAL METHODE :.

 $z \log x \log z = \frac{2\omega z}{2} = c = z$

SE = 30000 N

Bonding momany (mn) = W22 3 = 1000 × 1002

mor = 0.152 × 107 mm

displacement n= sully 324Et

moment q insure I: bd3

= lox103

12

12

11 Filment 1. 1 5 4 5 1 1 5

I = 813.33 mm1.

21: 3×1000 ×1004 324 ×2 ×105 ×833.0 n - a + 2 9 mm

Sheet and the second

STRESS ANALYSIS OF SIMPLY SUPPORTED BEAM WITH UDL

*

.

*

- 95



LINE STRESS				ANSYS R14.5
STEP=1				Academic
SUB =1			SEP	23 2021
SMIS3 SMIS16				10:55:40
MIN =33333.0				
MAX =111111				
Eliza-2				
2				
			•	
55555.6 67901.2	80246.	9 92592.	6 104938	
61728.4	74074.1	86419.8	98765.4	111111

No.:		is and at 2	Page No.:19
Step 4 soluurion!			
Solu	union Suloz (- minu	ena la	
Stepy: CTENDED	POZA prol:		
Unc	19 c-ord trad larg	are result ->	PO1701102-)
Acountation		a state of the sta	
100000000000000000000000000000000000000	tothe and a second to	ale - 2 dofine .	tolale -> ar.
	bland returns - 2 G	antoun Plat-2	ITNO
Incronad Probs	PIQ - TEXUILLA - 3 O		1
doman result			
	¥	PRACTICA	AL EVALUATION
	A TOBELOR	Observation (10) 7
		Calculation / F	Result (10) 7
	SA AN BEN	Viva (10)	20) 21
		Total Marks (aculty P&LP
	and the second second	Signature of t	22
Docusti	A STATE SOURCE	June -	
Kesour:	L'annormoration)	STOOTIFDING (14)	Bendinsmona
002(111100	USP ICC I I G Q I I I I I		
0			
moornical volu;			
analytical value.			
mus mo	stress analysis u	not anab cau	mo criven
LOU MELL SZZ	beam uning Anal	14.5 2.71 2.15M	Timalo.

STRESS ANDLUSIS IN THUSS COMONY.

- Incharge Marie

KTA ITAL REMATTERS & MARTERS



Exp. No.: H	
Date: 221912021	Page No.:
STRESS ANALYSIS IN TRUSS ELEMENT	
Bimi	
Lo Oppluste the state is	
novin In sound in the criven	11435
comon for arriver loading condition wins ansys	
SOTTWORD UNDE!	
WNZAZ Iń.Z	
piococuro :-	
Step1: chanse directory -> creat Toldor -> chan	NIC MONU.
Step 2: - Main monu -> PICTORONO STULITURE -> el	omone type
Edit 30 link.	
Step 31 - material property	
Scient read constrant -> Add 1)	
command Arca = 3250 mm	
Scient matering -, now matering propo	AL.
I JONTYOPIL -JEXZES P=0.17	13
Stop 42 - modalling	
sclau modeling -> liey point -> in outive	
1 (010:0) 2(1800) 30010) 36360.00	
5 (720010) 619000131010) 7110000101	1) H(SHOB (3)000)
The wins straight line.	•,
Step 5 1- mostiling	
Sclock moshing -) mesh ton -2 set al	
-line-Jox.	scliue

STRESS ANALYSIS OF TRUSS ELEMENT





Exp. No.: Date :	and caller and the second	Page No23
Steph: Loopl.		
ec- Anol anisab c-Anal	HYLLMULLI, displaymo	M.
Torus on hodo :1 23	0000	
Jorio on nodo :2 20	0000	
TONIO ON NOLO : 3 28	0000	
OC 4: OLAN NO ETOT	0000	
Stept:-		
Result		
Plot resulut -> Control Pl	aturior actor c- 10.	-Jeolve.
avvorris dans	7	
PIOL CHIL -2 Contour PI	noinnos bapar - to	
Stop 8:-	1. A AB	
GIERONA DOSE PROJETSO.		
משחשת אל	· onit ab c - mutor - tor	sticat
All deformentop.		
Schuld Rudthoo Highs	ZNY22- MIOZ BON	
Result:		
Thus the stress analysi	r E vivon mass	clamaly
was danc using AMEYS 14.5	soft water.	
	PRACTICAL EVAL	HATION
	Observation (10)	9
	Calculation / Result (10	1 9
	Viva (10)	8
	Signature of Second	26
	Logistical of Faculty	- ala

()

stress analysis & prove with how.



11111

MM.

100

ALL DIMENSTONS FIRE N= 30 mm U= 50 mm

1:05 mm

de ryonero

N120 M22100

51:0 32=100

EX: BES NIMM

Px07 = 0'3

STRESS ANALYSIS OF PLATE WITH HOLE



DISPLACEMENT VECTOR SUM



VON MISES STRESS



DEFORMED SHAPE

Exp. No.:	Page No.:2.5
STRESS ANALYSIS OF PLATE WITH HOLE.	
Aim:-	
ord working out of a soundary starts out zistered of	se when here
For viven loading conducion using answer costman	
Software used!	
"ANSYS 14.5"	
proladure:	
Step 1: chanse direttory -> creat Foldor -> chanse	Job Namo
Stop 2: main monu -> pretore -> structure -> ch	created anoma
Agg 200111 A vogo 1215	
Steps :- materia property.	
Choose material property and enter	olio o Ex
ster 4: modellins	
Sobul and -> conbit and -> ne itansue	
(WIIN2) -> 0.100	
191, 42) -> 0.100	
steps: ofse	
au ezito (- stald straw tuals	
MIN 02102 DZFPO NIM	
Step 6:. creat	
crons -) area -) circo -> by dimension -:	aws
2 c- Marind c- exportance - 2 prillaborn +1003	UP23451
meshina	
Step 1:- setous no mosting -> most tool -> size	(1+1-) motoder
chio ano-isizomon -> Arca Forus mash -> (512 120 OR
Arca -> 85.	

****************					Page No2.7
Step 8: -	londs.		and the second second second second second	mite an entered	stif the the
	doFing	Londo->	Structure	annanacih (a	inford in a
	Solunion	- time D	DT.	- J Cu Jerden ruge	- HANNA
Stop 9:	- Forio				
	SHOLIN -	Foria -> c	infrom an		andres
Stop 10 !.	solve.			- JOIC-J QUOL	VOLUM - 3 2 00
	Solve		•		
	Tion cho		EDO DAN	PRO - 1 DIOL T	er
	datormo	ci 20 -1	2 Octor	Olivera.	- Cutta
GED II '-	Desmit	3100-1			
	Hanty	retund	-) 50/0111	Dial manual	
	line show			Par Kruuu	-> 0/(100)
	Timelu	dan on			
and a second					
			hour ou	e la resource.	
			CONTR DO	r y isiaur.	
			POUR DU	r y isidur.	
			POUR DU		
			bolla oa		
			bolla ca		
				PRACTICAL EV	/ALUATION
				PRACTICAL EV Observation (10)	/ALUATION
				PRACTICAL EV Observation (10) Calculation / Result	/ALUATION 8 (10) 8
				PRACTICAL EV Observation (10) Calculation / Result Viva (10)	/ALUATION 8 (10) 8 8
				PRACTICAL EV Observation (10) Calculation / Result Viva (10) Total Marks (30)	/ALUATION 8 (10) 8 24
pesuut:.				PRACTICAL EV Observation (10) Calculation / Result Viva (10) Total Marks (30) Ignature of Facury	ALUATION 8 (10) 8 24 Palgi,
pesuut:.		s3 analys		PRACTICAL EV Observation (10) Calculation / Result Viva (10) Total Marks (30) gnature of Facury	ALUATION 8 (10) 8 24 Palei 13 lo
pesuut:.		23 analys		PRACTICAL EV Observation (10) Calculation / Result Viva (10) Total Marks (30) gnature of Facury	ALUATION 8 (10) 8 24 Pst Fisto WDR
pesuut:. Mu		sz analys		PRACTICAL EV Observation (10) Calculation / Result Viva (10) Total Marks (30) "gnature of Facury Calvern Prose ATS Softwork.	VALUATION 8 (10) 8 24 Patristo WAR



Analysis of Axi symmetry clamons.

80

ALL DIMENISIO

5

HRF

MM

PRXY =0.3

EX : DES NIME.

518

STRESS ANALYSIS OF AXIS SYMMETRIC COMPONENTS



/

xp. No	Page No
ANALYSIS OF AXI - SYMMETRIC ELEMENT:	
Aim !-	
to antizzy and the state of zorth of antipart of	uic element
to deal and a second mosts sollower.	
Software une?.	
NN24217.2	
Provodure!	
step 1:- chanse directory -> crost Foldor -> chanse	NDMO.
step 8!. main models -> programmer-settuitule.	
Sig 3!- motoring property.	
domone type -> Add -> Edit -> Add -> sclout.	
Qualt Hand 185 Nois-> options - scalt Axi	-SAMOMUT.
Step 4: modelling	
matorial motal -> yours modulas -> Ex -	SE 2 650.3
Steps: meshins	.0.
meshing - 2 conta - 2 mica - 27 change - 2 dimensi	(C31)(0)
(10120) 213120) 3(0120) 4(013) 3(01(00) 4	
OUPPLUS - 2 bolios - 2 ACC.	
Size Control -> manutas size -> Arca -> All	Mice - STRASS
Arca -) Free mesh -oli-	
Step 6 :- solution :	
· viomaisidzeo (- sais gritable- drinuuloz	· 3 MINGLIN ·
boundary (111-2 oiling 481 Scort Cruas	
by . warding t v direnin - 250 doring U	000)
displacement - 200 1000 - 2 Diu au coron o.	

- Result. Solue -> curanny Ls -> cloge. - aronand post prov. bost practor -> prol result cirronaid -> stress ron miss s	L- anitador - 1 Total - 20011 - 20011 - 20011	P11
Solve - > curanny LS -> Cloge. - acoronal post prov. bost practor -> Pibl resum dimonsion -> stress ron misso s	c- entrob c- 1	9 11
L- GRONDIDU POST PINY. DOST PHOLOSION -J PHOL TOSUUL dimonsion -JSHOSS NON missons Seein nor ZZDHESC- NOIGNOMIS	e-source antrabe-	A 11
Dost Diacton -> Pipi Tesuu dimonsion -> 20142c- noisenamis	c- earls antrabe-	P 11
dimonsion -setters no misses	3NCT7	
	A Charles of D	008-002-3
75 Barry 6		200-2003
		t
	<u> </u>	121 022 - 7
	and the second s	
		TION
	PRACTICAL EVALUA	8
	Observation (10)	8
	Viva (10)	8
	Total Marks (30)	200
	Signature of Faculty	PSIL
H.		13/00
mus ma stress analysis 7	by axisymmarvic	clement
he wint Henry WT COFFUE	1/0	
and more thanks with the second		



PRXY = 0.27

P: 7.830 lus[m³

· TALLO TRU

MODAL ANALYSIS OF FIXED BEAM



13/10/21			(mored of	allana a	Page No.:
M	todal ana	wsis I tixed	baam		
Alm!					
10	nalusis r	or Nochorn on	sound in the	orluen FI	x of barm
using	NNZYZ 20	And the second the			
autroz	is une?:				
	ANTA	14.2			
		13.11			
procodu) <u>'</u> -		1.		man and and
Step	:- will'the	morans -> cha	me anchos	8-201001	TOTOLI CABINE
	Job No	ma - s Tix ct	midor.	Turren 2 13 13	- <u></u>
Step 8	. main	nonu -> pictor	Unresc- and	WAR -JOK.	E an will
STED 3	. pre pro	tome tozza	and burd	-JESTULIOU	run linia -)
	Eloza	JIGAYAAJT	- potor tas	POD FX P	0287 : P= 7800
Chank.			a home al		uprint-)
51004	- previor	SUP- SOULIN			1
	main	JEDICI)	1000 00 1101	ore) titore	John Street
Stops	:- moun	org side- unon	162202 -> (11)	107 -25110	LUSTU LITE
	plot poir	19 -			
Step 6	- PIC PIO	(23M(- 023)	ning -> size	CH+ -) mos	tome-of 12 Cane
Stept	main n	onu - preprole	prov - was	JANALes	r type-,
	BODIL	or - only prove	homorpiazet	how point .	anourforc-
9460 8	. (mennano)	bost protect	-> Brodus	I Options	-> honge
Stre 0	. main h		bosi brou	reer -> Ree	
- Pille 4	1 33 94441 11	and subsult			Sammer
0.00					
- resu	Our To I	adal and	0	There 1	
-	I ON QUIT	cont anaust	U Orluen	tixes por	un unes
analy	iter by	where present	rs Softung.	PRACTI	CAL EVALUATION
				Observation	n (10)
				Viva	/ Result (10) 8
				Total Marks	(30)
				Circo Piarks	



MODAL ANALYSIS OF CANTILEVER BEAM









131 101 21	Page No
modal analysis of cannilivor boam.	maland some une
Aim! -	
to perform nodro anduir on a miven	continer boar
using "ANALYSIS 14'S " SOTHWARD	
Software unon.	
ANZAZ IA.2	
proceduro !-	
step 1: change the directory -) creat tolda	-> change name.
Step 8 !- FICTODALD -) SAVU HILLO -> DIC.	
Stop 3: PTDIETS -2 Hdd -2 bonn -2 Anodo 19	0
Step 11 - Manarina Dig -) saturaula -) of the	- imax phu
Ex 120066 0:017 0: 7900 lin	
The own of the man of the state of the state of the state	
Check modellan a more the paint secure	U-0.011 M.001
	TINE (2 (1010)3) 3(1)
STOTI- IIIO-> STIOLEN IINO -> DICIC POINTE	
STEP 3. MESTURS -) STOP CHI - 2 CONTUR PIOP	- I manual Sr 2
Step 9! LODG analysis type - 1 how analysis	modercua siz
Normalde Maillo -> solos -> cilian solution	QMO
Step 11: - Lond define Londs - JAPPIY - JSH	PRACTICAL EVALUATION
OD HOY -> HIL.	Observation (10)
Step 12 1/ UTEROLDO POST PROJECTON NETWORK	Calculation / Result (10) 8
	Viva (10)
RETURN 1.	Total Marks (30) 24
Thus the regard analysis of univer con	Signature of Faculty P&





NODAL TEMPERATURE



W.

No.: 9 20 [10]2]	Page No.: 37
Thormal mined 1 1	
molinua mixez boundarcy:	
Bim:	
to portorm the mormal analysis of	The chiven retranscular ame
min ditt boundary tempteraturo	USING ANAYT UW'S SOTTUDE
Software usod!	
ANZYZ 14.2	
Procodurro !	
step 1 :- utility man	
writty monu -> change Job N	ame -) enters Job name
whity monu-, chance while ->	Enter now hite.
Stop 8 !. PICECTONIO	
pretorno - momuno - one	
Step 3 !- Preprocessor '	
preprocessor -> clomont tipe -:	Add least Idologe - 2011 -
Qu	ac nabe
Proproverson -> materiale prop ->	Tia Minutonoo Opmail
preprotezz -> modalume -> ente	DIC -) FICO -) TECCONSIE-)
	pg carriers
<u> </u>	
	BIOD -1 DUDAL
preprotest -> merning -> merning	man
preproterso nodo - > Anoussi a	U Down D - Hennes HM
MCPTOICTSO - 2 COUNTY STATE	
CAMID -7 CAS-3 TOTAL	100-19h

tepu : Solution !-		
anona por prod c- and mod Caronan	e eapme- ruomas	anua
clar -> non bay -> plac -> nod -> ac		
unerouse post piel - map on the	Solutor.	
inenaras post plas - 1 pan operanto	n place parts	stem
on waph tomp -) onc.		
	and the second second	
A Manuelle		
	M2	
and the state of t		
The second se	-	
	Theme and the second se	
The second secon		
	PRACTICAL EVALU	JATION
	Observation (10)	8
NALAZIZ NAINE:-	Viva (10)	9
mininum tempressule =100°C	Total Marks (30)	8
monunum temprenturo = 500'e	Signature of Facult	2
Csuut:		1 Port
Thus the thormas analysis & thormas		MOL

hormonic analysis & canniver boxom.



MALESSA NOTEMENTE THE

E: 206200E6 NIM

PR74:0.3

P: 7830 15103

HARMONIC ANALYSIS OF CANTILEVER BEAM





Exp. No.: 19 Date: 25 1 101 2 1			Page No			
hormonic analysis of convision boam.						
	1					
aim :.	Succession Porror	Com B				
to POLIDIT	m nonmonic anoussis on a countin					
aiven Lorabing	Condition mine "Huzzes isofunce					
Sont anolitics						
^u Al	NZYJ 14.2"		/			
ProloAuro!						
Stepl: Prep	-xoc- autimite (- azojorgorg -119912					
Step 8: prep	Step A: pre projector -> clonone type -> bdd chi -> beam ->none					
5100 3:- mat	יו כ- ובלמות מהנשמות כ- עותסקסוק נחנים	inia - sel	ante			
-Jiz	285 - 3 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2	io ks m ³				
Ctapu' CCT	0-3 whus common c- mod c- a on	oro:H p.	1			
	duine -) unconte - > you poir · > in au	me-> Ile	1010) 2(11010)			
Stort 1 mark	- of 12 unknow c- lovena asize - ani	s linal ->	all lines.			
	aproprie diversion -> 05	34 92.87	- 1			
Class: Mark	Line -> Diur all - Jok					
STOPT: Mer		ci.				
Steps:, Solu						
Step 9: 500	A LOOK A BODILL & CAMILLAND - JTORD	Comme				
do Tim	The loss along - Saling and a loss	PRACTI	CAL EVALUATION			
[igg]		Observation	n (10) 🗳			
Stedio: Solv	e -> curdon ->ac solution (1 date	Calculation	/ Result (10)			
	/	Total Marks	(20) 24			
ROSUUUS !.	1	Signature	Faculty PSLP			
Thus	the hormone: analysis on a and	ARVOL	27/			
eusa pretos	by win any 14.5 softwar					


MODAL ANALYSIS OF SIMPLY SUPPORTED BEAM







SUBSYLACEDEENT STREET S

Exp. No. Page No. 45 Date: 27/10121 model analysis of simply supported boars. Aim :to analysis no model Frequery in the criver simply supported boom wing Annus softwork 1 4000 ONDUITO ANSYS IL'S Procoduro !. Step1 : Whility many -> change deretry -> and Toldor -> change Job name -> Fixed model Stop A: main mony -> pretoconto -> scoularize -> ole. Step 3 :- Pretoning -> material prop -> structured -> slastic-> 1501110011 -> Enter Ex:206800 Cb-> P=027 -> P=7830 Stopy: PICPDICESON -) Servich - , boom -) (roale -) lies point -) in any -> enervalue Lororo) (1100) Stops :- moun monu -> proprocessor -> line straight like-> pilic point. stopp : propossor ") moshing -> size control -> manual size. moth Ster :- main monu -> preprolessor -> Land -> Analyse -> hopy - strumunal -> downo pron -> second point -> CLILL UX, UZ, RUX, RUY, RUZ Step 8: Chemanal port projector -> Minalysis option -> swert

STOPIO :- DEXLO	and - 2 pla result	-> datamos shape -:	all determo
		- 3 9 4	
	Start and "		
		The second second	
		anti- and and	re trat
			4.177.19
		1 221 0222	
		PRACTIC	ALEVALUATION
		Calculation /	10) <u>8</u>
		Viva (10)	Court (20) 8
		Total Marked	101 010
0		Signature of L	andre Rela
THE TAXABLE		L'endinaria de la	
KCSUU.*			



MODAL ANALYSIS OF 2D CANTILEVER PLATE





Exp. No.:...................... Page No .: 49 Date : 27 110121 model analysis of 20 place with cantiliver support Frequency. Aim! to analyst mo model Frequency in mo 20 place with contrilium with contrilium boom wing ANSYS 14.5 Software was: AN545 14.5 protodure: stop: - white monu -> change -> more Toldon -> change Tob name-> Tixez moder. Stor 8 :- main monu -> preteron 10 -> structure -> oic. Step 3:- Pretorono -> materia prop -> Structure ELOSHL -> ELOSHL -1501 10011 -> ENTER -> EX = 30 C b PNDX : 0.87 0:1830 Stepy . PICPIOLOTION -) SCIOLA - > PICO -> DECLANSE (MIY2) (1012-5) Quiyi) Low Stops: - moshing -> size control -> manual size ->Enter. Step 6: mosh -> Area -> Free moth. ston: solution -) Analysis type -> moder -> Analysis oporation -) Enton Stop 8: dotting land -> Apply -> displacement -> on lieu point-> Avoust to the upter day. Step q: Solution-) solue -> selection is done. Step 10 :- Crience polly prepriesso -> Road Rosum -> Finish sex - , plot result - , devotines shapes - Rewe

0	Page No
	and the second s
	*
A A	CE CE D
	The Part Ma
	PRACTICAL EVALUATION
	Observation (10)
	Calculation / Result (10)
	Viva (10) 218
	Total Marks (30)
	Signature of Faculty
	3
2 mm 11141	
Kosuman model anal	Lysis & The OD PRUE WITH
Inus ine inter	analysis by using arrive rograms
unnilium unni	

Completed



P.S.R. ENGINEERING COLLEGE

(Autonomous Institution, Affiliated to Anna University, Chennai) Sevalpatti, Sivakasi – 626140 Department of Mechanical Engineering



ATTAINMENT VALUE OF LABORATORY OUTCOMES

161ME77 - ANALYSIS LABORATORY

Course Outcomes:

The students will be able to

- CO1. Demonstrate the features of ANSYS software
- CO2. Validate the stress analysis in beam problems with empirical formulas
- CO3. Explicit the stress analysis of a plate with a circular hole and axi-symmetric component
- CO4. Identify the need of mode frequency analysis in 2D component
- CO5. Realize the Thermal analysis of a 2D component
- CO6. Import any solid model to ANSYS for contact analysis

Course Outcomes					Pro	gram O	outcom	es (POs	5)				Prog	am Spec (PS	cific Out Os)	comes
Outcomes	PO1	PO2	PO3	PO4	PO5	PO6	PO7	PO8	PO9	PO10	PO11	PO12	PSO1	PSO2	PSO3	PSO4
CO1	2	3	3						2	3		2	2			2
CO2	2	1	1		2				2	2		3	3	3	1	2
CO3	2	3	3		2					2		2	3	3	2	2
CO4	2	3	3		2					2		2	3	3	2	2
CO5	2	3	3		2				2	2		2	3	3	2	2
CO6	2	3	3		2				2	1		3	2	3	3	3

1: Slight (Low) 2: Moderate (Medium) 3: Substantial (High)

COURSE OUTCOMES ATTAINMENT – PRACTICAL COURSES



[Reference from Evaluation Manual]

			Evaluation of	Course Outcomes	
Course Code & Name	: 161ME7	7 & Con	nputer Aided Simu	llation & Analysis Laboratory	
Course Teacher	: Dr. P. SI	henbaga	Velu ASP/Mech &	Mr.D.Sundarrajan AP/Mech	
Year / Semester	: IV/VII/]	I & II			
Academic Year	:2020-21	Odd	Batch	2017-2021	

Course End Survey

Course	Marks	s obtained f	for Cours	e Outcor	ne	Total No of Students		Score
Outcomes	5	4	3	2	1	Total No of Students	Net	100
C01	90	30	14	0	0	134	612	91.34
CO2	95	25	14	0	0	134	617	92.09
CO3	95	19	10	0	0	134	581	86.72
CO4	85	35	14	0	0	134	607	90.60
CO5	100	24	10	0	0	134	626	93.43
CO6	80	40	14	0	0	134	602	89.85

Particulars	C01	CO2	CO3	CO4	CO5	CO6
Internal	76.67	79.67	78.33	78.67	80.00	79.00
End Semester Exam	71.07	71.07	71.07	71.07	71.07	71.07
Course End Survey	91.34	92.09	86.72	90.60	93.43	89.85
Attainment (0.65 of Internal+0.25 of ESE + 0.1 of CES)	76.46	78.33	76.99	77.58	78.66	77.71

Course Outcomes	C01	C02	C03	C04	C05	CO6
Average Score Out of 5	3.82	3.92	3.85	3.88	3.93	3.89
Average Score Out of 3	2.29	2.35	2.31	2.33	2.36	2.33



P. A. Fel2ler Signature of the Course Tutor P.S

Signature of the Course Co-ordinator/Moderator

Head of the Department

161ME77- Computer Aided Simulation and Analysis Laboratory CO- PO Mapping

Course					P	rogram Ou	tcomes (PC)s)		SIG			Progra	m Specific	Outcomes	(PSOs)
Outcomes	PO1	PO2	PO3	PO4	PO5	PO6	PO7	PO8	PO9	PO10	PO11	PO12	PSO1	PSO2	PSO3	PSO4
CO1	2	3	3						2	3		2	2			2
CO2	2	1	1		2				2	2		3	3	3	1	2
CO3	2	3	3		2					2		2	3	3	2	2
CO4	2	3	3		2					2		2	3	3	2	2
CO5	2	3	3		2				2	2		2	3	3	2	2
CO6	2	3	3		2				2	1		3	2	3	3	3

	CO	PO1	PO2	PO3	PO4	PO5	PO6	PO7	PO8	PO9	PO10	PO11	PO12	PSO1	PSO2	PSO3	PSO4
Į	CO1	76.67	76.67	76.67				1 ALALINE Y		76.67	76.67		76.67	76.67			76.67
ĺ	CO2	79.67	79.67	79.67		79.67				79.67	79.67		79.67	79.67	79.67	79.67	79.67
	CO3	78.33	78.33	78.33		78.33					78.33		78.33	78.33	78.33	78.33	78.33
	CO4	78.67	78.67	78.67		78.67					78.67		78.67	78.67	78.67	78.67	78.67
	CO5	80.00	80.00	80.00		80.00		Section 7		80	80.00	1000	80.00	80.00	80.00	80.00	80.00
	CO6	79.00	79.00	79.00		79.00	and the second second			79	79.00	-	79.00	79.00	79.00	79.00	79.00

Internal CO-PO Mapping

External CO-PO Mapping

CO	PO1	PO2	PO3	PO4	PO5	PO6	PO7	PO8	PO9	PO10	PO11	PO12	PSO1	PSO2	PSO3	PSO4
CO1	71.07	71.07	71.07						71.07	71.07		71.07	71.07			71.07
CO2	71.07	71.07	71.07		71.07				71.07	71.07		71.07	71.07	71.07	71.07	71.07
CO3	71.07	71.07	71.07		71.07					71.07		71.07	71.07	71.07	71.07	71.07
CO4	71.07	71.07	71.07		71.07					71.07		71.07	71.07	71.07	71.07	71.07
CO5	71.07	71.07	71.07		71.07				71.07	71.07		71.07	71.07	71.07	71.07	71.07
CO6	71.07	71.07	71.07		71.07				71.07	71.07	The second	71.07	71.07	71.07	71.07	71.07

	PO1	PO2	PO3	PO4	PO5	PO6	PO7	PO8	PO9	PO10	PO11	PO12	PSO1	PSO2	PSO3	PSO4
Internal	78.7	78.6	78.6		79.1				78.8	78.5		78.8	78.8	79.1	79.1	78.7
External	71.1	71.1	71.1		71.1				71.1	71.1		71.1	71.1	71.1	71.1	71.1

R. A. F. 5/2/21 Signature of the Course Tutor

Signature of the Course Co-ordinator/Moderator

R. & Stan Programme Co-ordinator

Head of the Department

						Eval	uation	of PO	& PSO	O								1100.10
	Co	urse Code &	Name	: 161N	1E77 8	& Com	puter A	ided S	Simula	tion &	Analy	sis Labo	oratory			1		
		Year / Se	mester	: IV /	VII													
		Dire	ct Tool	: Prog	ram O	utcom	es (POs	s) & Pr	ogran	Speci	fic Out	comes	(PSOs)					
	Table 3. Average	attainment s	core of	Cours	e Outco	omes b	ased of	n Prog	ram O	utcom	es (PO	s) & Pr	ogram	Specific	Outcor	nes (PSC	Ds)	
Course End S	uniou from COL	A	ttainme	ent of P	Os & 1	PSOs f	rom a	Course	e consi	dering	all the	Direct	tools					
Atta	inment	CO-P	O - PSC) Map	oing									5-18-1-1				
Course Outcomes	Survey Score	со	PO1	PO2	PO3	PO4	PO5	PO6	PO7	PO8	PO9	PO10	PO11	PO12	PSO1	PSO2	PSO3	PSO4
C01	91.34	C01	91.34	91.34	91.34						92.00	91.34		91.34	91 34			91 34
C02	92.09	C02	92.09	92.09	92.09		92.09				96.00	92.09		92.09	92.09	96.00	92.09	92.09
C03	86.72	C03	86.72	86.72	86.72		86.72					86.72		86.72	86.72	88.00	86.72	86.72
C04	90.60	C04	90.60	90.60	90.60		90.60	100	121 121			90.60		90.60	90.60	88.00	90.60	90.60
C05	93.43	C05	93.43	93.43	93.43		93.43				93.43	93.43		93.43	93.43	96.00	93.43	93.43
CO6	89.85	CO6	89.85	89.85	89.85		89.85			31-11-12	89.85	89.85		89.85	89.85	100.00	89.85	89.85
		Score	4.53	4.53	4.53		4.53	1. Notes	1 classical	C. G. Martin	4.64	4.53		4.53	4.53	4.68	4.53	4.53

Particulars	PO1	PO2	PO3	PO4	PO5	PO6	PO7	PO8	PO9	PO10	PO11	PO12	PSO1	PSO2	PSO3	PSO4
Internal	78.70	78.60	78.60		79.10				78.80	78.50		78.80	78.80	79.10	79.10	78.70
Endsemester	71.10	71.10	71.10		71.10		1.23		71.10	71.10		71.10	71.10	71.10	71.10	71.10
Course End Survey	4.53	4.53	4.53		4.53				4.64	4.53		4.53	4.53	4.68	4.53	4.53
Attainment (0.65 of Internal+0.25 of ESE + 0.1 of CES) Out of 5	3.88	3.88	3.88		3.89				3.89	3.87		3.88	3.88	3.91	3.89	3.88
Attainment out of 3	2.329	2.327	2.327		2.335			attended to	2.337	2.325		2.330	2.330	2.345	2.335	2.329



p.81 75/2/21 Signature of the Course Tutor

Signature of the Course Co-ordinator/Moderator

A-Ry 512/2 Programme Co-ordinator

Head of the Department