LAB 2. Finite element analysis of free vibrations (modal analysis).

CASE 1.

A cantilever beam with a C-section

1. Geometry of the beam (Fig. 1).



a) create a text file and save it as *keypoints.txt* in your working directory:

```
MACRO TO DEFINE KEYPOINTS IN C-SECTION
!
|*
                                       ! width along x-axis (mm)
w=60
h=40
                                       ! height along x-axis (mm)
t=5
                                       ! wall thickness (mm)
                                      ! y coordinate of the center of gravity (mm)
y=(h*h+(w-2*t)*(h-t/2))/(2*(h-t)+w)
/PREP7
                                       ! Main Menu/ Preprocessor
                   ! Main Menu/ Preprocessor/Modeling/Create/Keypoints/In Active CS
                                       ! 1- no. of keypoint, X=-w/2, Y=-y, Z=0
k,1,-w/2,-y,0,
k,2,-w/2+t,-y,0,
k,3,-w/2+t,0,0,
k,4,-w/2,0,0,
k,5,-w/2+t,h-y-t,0,
k,6,-w/2,h-y-t,0,
k,7,-w/2+t,h-y,0,
k,8,-w/2,h-y,0,
k,9,w/2,-y,0,
k,10,w/2-t,-y,0,
k,11,w/2-t,0,0,
k,12,w/2,0,0,
k,13,w/2-t,h-y-t,0,
k,14,w/2,h-y-t,0,
k,15,w/2-t,h-y,0,
k,16,w/2,h-y,0,
!* _____
                   END OF MACRO
```

b) Execute the macro (Utility Menu>File>Read Input from... -> keypoints.txt)

c) create 7 areas through the keypoints (Fig. 2): 1->2->3->4->OK, (3->5->6->4, 5->7->8->6, 5->13->15->7, 13->14->16->15, 11->12->14->13, 10->9->12->11



d) check geometric properties of the cross-sectional area (*Utility Menu>List>Picked Entities>Area>Areas> PickAll*) (Fig. 3)

Model Query Picker	ASUM Command	
@ Pick O Unpick	File	
© Single C Box C Polygon C Circle C Loop Query Item:	PRINT GEOHETRY ITEMS ASSOCIATED WITH THE CURRENTLY SELECTED AREAS TOTAL NUMBER OF AREAS SELECTED = 7 (OUT OF 36 DEFINED) TOTAL SURFACE AREA OF ALL SELECTED AREAS = 650.00 TOTAL VOLUME OF ALL SELECTED AREAS = 650.00	
Area On Entities: Areas List Option: NONE V	CENTER OF HASS: XC=-0.12598E-14 YC=-0.10186E-13 ZC= 0.0000 *** MOMENTS OF INERTIA **** (BASED ON A UNIT DENSITY AND A UNIT THICKNESS) ABOUT ORIGIN ABOUT ORIGIN ABOUT CENTER OF HASS IXX = 0.10097E+06 0.10097E+06 0.10097E+06 IXY = 0.35542E+06 0.35542E+06 0.35542E+06 0.35542E+06 IXY = 0.97122E-10 0.97122E-10 0.45639E+06 0.45639E+06 IXY = 0.0000 0.0000 0.0000 0.45639E+06 0.45639E+06	Fig. 3
Generate 3D Anno OK Apply Reset Cancel Pick All Help	Measurements of Area File Area 1 = 133.653846 Area 2 = 41.346154 Area 3 = 25.000000 Area 6 = 41.346154 Area 6 = 41.346154 Area 6 = 41.346154 Area 7 = 133.653846 AREA TOTAL = 650	

e) extrude the cross-section along z-axis by distance *l* (*Preprocessor>Modeling>Operate>Areas>By XYZ* Offset> Pick All)

2. Choose SOLID185 finite element: (brick 8-node, element technology: Enhanced strain) (*Preprocessor>Element Type>Add/Edit/Delete*) (Fig. 4)



3. Define material properties of a steel (linear elastic and isotropic) Young's modulus $E = 2 \cdot 10^5 MPa$, Poisson's ratio v = 0.3, density $\rho = 7.8 \cdot 10^{-9} Ns^2 / mm^4$: (*Preprocessor>Material Props>Material Models> Structural*) 4. Specify the density of discretization on lines (100 divisions along z-axis) and create a mapped hexahedral mesh (*Preprocessor>Meshing> MeshTool*) (Fig. 5)

5. Save the mesh as an image (Plot> Elements, PlotCtrls> Redirect Plots-> To JPEG File ...)

6. Save the database (*Utility Menu>File>Save As...*, for example: C_section_mesh.db)

7. Apply boundary conditions: Fixed support for the cross-section z = 0 (Solution> Apply> Structural> Displacement> On Areas> All DOF)

8. Choose modal analysis and define options (Fig. 6).

Main Menu	New Analysis	×	
 Preferences Preprocessor Solution Analysis Type 	[ANTYPE] Type of analysis	C Static C Modal	Fig. 6
Restart	Main Menu	Modal Analysis	×
 Restart Sol'n Controls Define Loads Load Step Opts SE Management Results Tracking Solve Manual Rezoning ADAMS Connect 	 Preferences Preprocessor Solution Analysis Type New Analysis ExpansionPass Analysis Options Define Loads Load Step Opts SE Management (Cl 	MODOPT Mode extraction method	 Block Lanczos PCG Lanczos Supernode Subspace Unsymmetric Damped
	■ Results Tracking ■ Solve	No. of modes to extract	8
	 Manual Rezoning ADAMS Connection Diagnostics Unabridged Menu General Postproc TimeHist Postpro Radiation Opt Session Editor Finish 	[MXPAND] Expand mode shapes NMODE No. of modes to expand Elcalc Calculate elem results?	Ves 8 Ves
		[LUMPM] Use lumped mass approx? [PSTRES] Incl prestress effects?	□ No □ No
		OK Cancel	Help

10. Read results of modal analysis (*Main Menu> General Postproc> Read Results> By Pick*) for the first set (Fig. 7)

ain Menu	Results File: fil	e3.rst			
Preferences	Available D	ata Sets:			
Preprocessor	Set	Frequency	Load Sten	Substen	Cumulative
Solution	1	25 206	1	1	1
eneral Postproc	2	53,300	4	2	2
Data & File Opts	2	33.750	1	2	2
Results Summary	3	124.11	1	3	3
Read Results	4	218.21	1	4	4
First Set	5	232.49	1	5	5
Next Set	6	477.64	1	6	6
Previous Set	7	597.02	1	7	7
Last Set	8	597.25	1	8	8
By Pick					
By Load Step					
By Time/Freq					
By Set Number			Fig 7		
ailure Criteria			1.8.7		
ot Results					
st Results					
uery Results					
Options for Outp					. [
Results Viewer		Read		Nex	d

11. Plot animation of displacement in y direction (Utility Menu>*PlotCtrls>Animate>Deformed Results >DOF Solution>UY*) (Fig. 8).

	Animate Nodal Solution Data		\times
Plot <u>C</u> trls <u>W</u> orkPlane Parame	Animation data eters 1 No. of frames to create Time delay (seconds)	10	
View Settings	[PLNSOL] Contour Nodal Solution	Data	
Numbering Symbols Style	Item,Comp Item to be contoured	DOF solution Tra Stress Strain-total Energy Strain ener dens Strain-elastic	UY UZ USUM
Font Controls	•		01
Window Controls	• ОК	Cancel	Help
Erase Options	•		
Animate Annotation Device Options Podiract Plats	 Mode Shape Cyc Traveling Wave Deformed Shape Deformed Results 	Fig. 8	
Hard Copy	Over Time Time-harmonic		
t=: 0.025T 0.05T 0.075T	0.1T 0.125T 0.15T 0.175	5T 0.2T 0.225T 0.25T	(T = 1/f = 1/35.3 s)
			max

12. Plot contour maps of von Mises stress for all frequencies (magnitudes are not realistic), select the option: Deformed shape with undeformed edge (Main Menu>General Postproc>Plot Results>Contour Plot>Nodal Solu> Stress>Von Mises). Save each contour map as an image (Fig. 9) and fill Table. 1.

(Results for l = 1000 mm)







1st mode = 1st bending mode in YZ plane at 35.3 Hz



4th mode = 2nd bending mode in YZ plane at 218.2 Hz



3rd mode = 2nd bending-torsional mode at 124.1 Hz



6th mode = 3nd bending-torsional mode at 477.6 Hz

NODAL SOLUTION STEP=1 SUB =5 FREQ=232.491 SEQV (AVG) PowerGraphics EFACET=1 AVRES=Mat DMX =43.0574 SMN =11.3224 SMX =3197.97

5th mode = 1st torsional mode at 232.5 Hz



NODAL SOLUTION STEP=1 SUB =7 FREQ=597.017 SEQV (AVG) PowerGraphics EFACET=1 AVRES=Mat DMX =28.1157 SMN =12.8319 =9373.61 12.8319 1052.92 SMX 2093 2093 3133.09 4173.18 5213.26 6253.35 7293.44 8333.52 9373.61

11.3224 365.395 719.467 1073.54 1427.61

1781.68 2135.76 2489.83

2843.9 3197.97



7th mode = 3rd bending mode in YZ plane at 579 Hz

8th mode = 4th bending-torsional mode at 579.2 Hz

Analytical solution: $c = \frac{1}{l^2} \sqrt{\frac{EJ}{\rho A}}$; $J = \begin{cases} IXX - \text{ for bending in YZ plane} \\ IYY - \text{ for bending in XZ plane} \end{cases}$

Type of support		f /o	f /a	f /c	
z = 0	z = l	JI/C	J_2/C	J_i/c	
fixed	fixed	3.560	9.815	0.393(2 <i>i</i> +1) ²	
fixed	pinned	2.454	7.951	0.098(4 <i>i</i> +1) ²	
fixed	free	0.560	3.506	0.393(2 <i>i</i> -1) ²	
pinned	pinned	1.571	6.283	1.571 <i>i</i> ²	

Example. Cantilever beam with length l = 1000mm:

1ST bending mode in YZ plane: $J = IXX = 100970mm^4, \ A = 650mm^2, \ E = 200000 \frac{N}{mm^2}, \ \rho = 7.810^{-9}Ns^2/mm^4$ $c = \frac{1}{1000^2} \sqrt{\frac{200000 \cdot 100970}{7.810^{-9} \cdot 650}} = 63.11 \frac{1}{s} \quad ; \quad f_1 = \frac{f_1}{c} \cdot c = 0.56 \cdot 63.11 \frac{1}{s} = 35.34 \text{Hz}$ relative error: $\Delta f_1 = \left| \frac{35.3056 - 35.34}{35.34} \right| = 0.1\%$ 2nd bending mode in YZ plane: $f_2 = \frac{f_2}{c} \cdot c = 3.506 \cdot 63.11 \frac{1}{s} = 221.27 \text{Hz}$ relative error: $\Delta f_2 = \left| \frac{218.2 - 221.27}{221.27} \right| = 1.4\%$

3rd bending mode in YZ plane: $f_3 = \frac{f_3}{c} \cdot c = 0.393 \cdot (2 \cdot 3 - 1)^2 \cdot 63.11 \frac{1}{s} = 620.07$ Hz relative error: $\Delta f_3 = \left| \frac{579.02 - 62 \cdot .07}{620.07} \right| = 6.7\%$

CASE 2.

Beam with fixed cross-section at z = 0 and pinned cross-section at z = l



Plot contour maps of von Mises stress for 8 frequencies (see point 12). Save contour maps as images and fill Table. 2.

CASE 3.

Beam with fixed cross-sections



Plot contour maps of von Mises stress for 8 frequencies (see point 12). Save contour maps as images and fill Table. 3.

CASE 4.

Rotating cantilever beam (with fixed cross-section at z = 0 and free cross-section at z = l)

The stress state in a structure changes its natural frequencies. To include this effect in a modal analysis we perform a static analysis with the prestress option.

- 13. Start a new analysis (static).
- 14. Change Unabrigded Menu to Abridged Menu (Main Menu>Solution)
- 15. Enable the prestress effect (Solution>Analysis Type>Analysis Options>Prestress>On) (Fig. 12).
- 16. Apply constraints (fixed cross-section at z = 0, the same as for CASE 1)
- 17. Apply an angular velocity ω_y (Fig. 13)
- 18. Solve (Solution-> Solve-> Current LS)

19. Read results (*Main Menu>General Postproc>Read Results>First Set*). Plot and save as image a contour map of stress in Z direction (*General Postproc>Plot Results>Contour Plot>Nodal Solu> Z- Component of Stress*) (Fig. 14).

20. Choose modal analysis and define options. Include prestress effects (Fig. 15).

21. Read results of modal analysis (Main Menu> General Postproc> Read Results> By Pick) and fill Table 4.

ANICYC Assistantis Teaching Intra-				
ANSYS Academic Teaching Introc	Static or Steady-State Analysis			
File Select List Plot P	Nonlinear Options			
D 🛩 🖬 🚳 🌆 🖉 🤋	[NLGEOM] Large deform effects	□ Off		
Toolbar	[NROPT] Newton-Raphson option	Program chosen -		
SAVE DR DESUM DR	Adaptive descent	ON if necessary		
SAVE_DB RESOM_DB	Linear Ontions			
	[LUMPM] Use lumped mass approx?			
Main Menu				
Preferences	[EQSLV] Equation solver	Program Chosen		
Preprocessor Solution	Tolerance/Level -			
Analysis Type	- valid for all except Sparse Solver			
New Analysis				
Restart	Multiplier -	0		
Sol'n Controls	- valid only for Precondition CG			
Analysis Options	Matrix files -	Delete upon FINISH -		
Define Loads	- valid only for Sparse			
Load Step Opts	[MSAVE] Memory Save -	□ Off		
SE Management (CM Desults Treaking	- valid only for Precondition CG			
Solve	[PCGOPT] Level of Difficulty -	Program Chosen 💌		
Manual Rezoning	- valid only for Precondition CG			
ADAMS Connection	[PCGOPT] Reduced I/O -	Program Chosen - Fig. 12		
Diagnostics Abridged Menu	- valid only for Precondition CG			
General Postproc	PCGOPTI Memory Mode -	Program Chosen 💌		
TimeHist Postpro	- valid only for Precondition CG			
Radiation Opt	IDN (CLIF CL/L Divists Chasts	= 0#		
Session Editor	IPIVCHECKI PIVOIS Check	I Off		
	- valid only for Sparse and PCG Solve			
	IFOTRESTINCIPIestiess effects?	res la		
	TOFFSTT lemperature difference-	0		
	 between absolute zero and zero of active temp scale 			
	OK	Cancol		
L		Help		





NODAL SOLUTION STEP=1 SUB =1 TIME=1 SZ (AVG) RSYS=0 PowerGraphics EFACET=1 AVRES=Mat DMX =.129793 SMN =.205256 SMX =54.5213 .205256 6.24037 12.2755 18.3106 24.3457 30.3808 36.416 42.4511 48.4862

54.5213

Fig. 14

Main Menu	Modal Analysis		\times
 ■ Preferences ■ Preprocessor 	[MODOPT] Mode extraction method		
 □ Solution □ Analysis Type □ New Analysis □ ExpansionPass □ Analysis Options □ Define Loads □ Load Step Opts □ SE Management (CMS) □ Results Tracking □ Solve □ Manual Rezoning □ ADAMS Connection □ Diagnostics □ Abridged Menu □ General Postproc □ TimeHist Postpro □ Radiation Opt □ Session Editor □ Finish 	Fig. 15	 PCG Lanczos Supernode Subspace Unsymmetric Damped QR Damped 	
	No. of modes to extract	8	
	[MXPAND] Expand mode shapes NMODE No. of modes to expand Elcalc Calculate elem results?	I⊽ Yes 8 I⊽ Yes	
	[LUMPM] Use lumped mass approx? [PSTRES] Incl prestress effects?	□ No I⊽ Yes	

Discuss results of cases 1-4 and write conclusions.