Course in Ansys

Example 0150

Example – Truss 2D



Objective:

Compute the maximum deflection Tasks: Display the deflection figure? Topics: Topics: Start of analysis, Element type, Real constants, Material, modeling, element size for beam models, saving/restoring

E - 210000N/mm²



Step by Step guide for Ansys 13

1. Preprocessor > Modeling > Create > Keypoints In Active CS



3 Points (click apply for 1 and 2 and ok for 3)

Х	Y	Ζ
0	0	0
100	0	0
0	120	0

Eile Select List Plot PlotCtris WorkPlane Parameters Macro MeguCtris Help				
D 📽 🖬 🗗 🖨 🖉 🕈 📓			- 💽 👯 💷	
ANSYS Toolbar				۲
CAVE DR DECUM DR OUT D	OWDCDDH			
SAVE_DB RESUM_DB QUIT P	OWRORPH			
ANSYS Main Menu	8		~	1. 2
Preferences	- POINTS			n 3 -
Preprocessor	TYPE NUM		Nancommercial use only	
Element Type Real Constants			Noncommercial use only	
Material Props			FEB 16 2012	
Sections			05.55110	6
Modeling				
□ Create				
				9
B Lines				
P Straight Line	Create Straight Line			ø
P In Active Coord				
P Overlaid on Area	@ Pick C Unpick			0
Tan to 2 Lines	6 Single C Box			
P Normal to Line	C Polygon C Circle			<u> </u>
P Norm to 2 Lines	C Loop			0
P At angle to line	Count = 0			
Angle to 2 Lines	Maximum = 2			40
B Splines	Minimum = 2			
A Line Fillet	KeyP No. =			
Areas	<u></u>			6
Volumes	(* List of Items			\$
Nodes Elemente	C Min, Max, Inc			8
Contact Pair				8
Circuit		x x	2	-0
Racetrack Coil				Q.
Transducers	OK Apply			9)
Operate Movie / Medify	Reset Cancel			0
E Copy	Pick All Help			_
Reflect				
Check Geom				

2. Preprocessor > Modeling > Create > Lines > Lines > Straight Line

Click on point 1, ok, point 2, ok

Create lines between points 1-2; points 2-3;

Alternatively type the point numbers into the popup box and press enter to create lines



3. Preprocessor>Sections>Beam>Common Sections

Section 1: B=20 H=20 Section 2: B=10 H=10 4. Preprocessor>Materials>Material Props>Material Model Inside the resulting window

Structural>Linear>Elastic>Isotropic





5. Close popup window

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

Click Add Select Beam> 2 node 188 Click OK, Click Close



7. Preprocessor>Meshing>Mesh Attributes>Picked Lines

Eile Select List Plot PlotCtrls WorkPlane Parameters Macro MenuCtrls Help



Select the line between points 1 and 2 (horizontal line) > Click OK

Under Element Type select 1

	▲ Line Attributes	×
	[LATT] Assign Attributes to Picked Lines	
	MAT Material number	1
	REAL Real constant set number	None defined 🗨
	TYPE Element type number	1 BEAM188 -
\triangleleft	SECT Element section	1
	Pick Orientation Keypoint(s)	□ No
	ОК Арріу	Cancel Help

Select the Line between points 2 and 3 (diagonal line) > click OK

Under Element Type change to 2

8. Preprocessor>Meshing>Mesh>lines

a. Pick All



9. Solution > Analysis Type> New Analysis

a. Select static > click OK



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

a. Select keypoint 3 (or type 3 into the box) > click ok



b. Select All DOF and type in 0 for Value > click OK

[Apply U,ROT on KPs	×
	[DK] Apply Displacements (U,ROT) on Keypoints	
	Lab2 DOFs to be constrained	All DOF UX UY UZ ROTX
	Apply as If Constant value then:	Constant value 🗨
\triangleleft	VALUE Displacement value KEXPND Expand disp to nodes?	0
	OK Apply	Cancel Help

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

a. Select Keypoint 1 > click OK



b. In the next box select UX and UZ and for Value select 0 (this creates a roller fixture which allows movement in the Y direction ONLY) >Click OK

Apply U,ROT on KPs	X
[DK] Apply Displacements (U,ROT) on Keypoints	
Lab2 DOFs to be constrained	All DOF UX UY UY ROTX
Apply as	Constant value 💌
If Constant value then:	
VALUE Displacement value	0
KEXPND Expand disp to nodes?	
OK Apply	Cancel Help

12. Solution > Define Loads > Apply> Structural> Force/Moment>On Keypoints

a. Select keypoint 2 (or type the number 2) > OK



b. Select FY for direction, Constant Value, and enter -100 for the VALUE box (the force is directed down, hence the negative)> click OK

	Apply F/M on KPs	x
	[FK] Apply Force/Moment on Keypoints	
	Lab Direction of force/mom	
	Apply as Constant value	•
	If Constant value then:	
\bigcirc	VALUE Force/moment value -100	>
	OK Apply Cancel Help	

13. Solution > Solve >Current LS



- 14. General Postproc>Plot Results>Deformed Shape
 - a. Select Def + undeformed> OK



Pick a menu item of enter an ANSTS Command (POSTT)

- 15. General Postproc>Plot Results> Contour plot>Nodal Solu
 - a. In Popup window Nodal Solution>Stress>con Mises stress
 - b. Click OK

ANSYS Main Menu		1.0
Preferences		
Preprocessor	A Contour Notal Solution Data	
Solution	citien to be contoured cial use or	🗤 😌 😁
General Postproc	16 16 2010 CONDUCT	
Data & File Opts	Modal Solution	
Results Summary	B DOF Solution	1
Read Results	of Stress	
Failure Criteria	X-Component of stress	<u> </u>
Plot Results	X-Component of stress	3
Deformed Shape	7-Component of stress	a 1
Contour Plot	XY Shear stress	
Nodal Solu	Y Char drugs	a
Element Solu	VI Share trace	
Elem Table		
Line Elem Res	Principal Stress Principal Stress	Q.
Vector Plot	Principal suess	82
Plot Path Item	3rd Principal stress	
Concrete Plot	Stress intensity	
ThinFilm	And Mises stress	Q
List Results		
Query Results		40
Options for Outp	Undisplaced shape key	••
Results Viewer		
Write PGR File	Undisplaced shape key Deformed shape only	-
Nodal Calcs	Scale Factor Auto Calculated • 4459.35768406	\$
Element Table		
Path Operations	Additional Options	8.
Surface Operations	Additional Options	18
Load Case	OK Acely Cancel Help	100
Check Elem Shape		<u><u></u><u></u><u></u><u></u><u></u></u>
Write Results		19
ROM Operations		0
Submodeling		<u>4</u>