Course in ANSYS

Example0150

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

$$E = 210000 \text{N/mm}^2$$

 $\nu = 0.3$
 $L = 100 \text{mm}$
 $H = 120 \text{mm}$
 $a = b = 20 \text{mm}$
 $c = d = 10 \text{mm}$
 $F = 100 \text{N}$

Modeling considerations

- As you begin your model generation, you will (consciously or unconsciously) make a number of decisions that determine how you will mathematically simulate the physical system:
 - What are the objectives of your analysis?
 - Will you model all, or just a portion, of the physical system?
 - How much detail will you include in your model?
 - What kinds of elements will you use? How dense should your finite element mesh be?
- In general, you will attempt to balance computational expense (CPU time, etc.) against precision of results as you answer these questions.
- The decisions you make in the planning stage of your analysis will largely govern the success or failure of your analysis efforts.

Modeling considerations

- Linear or Higher Order Elements
- Take Advantage of Symmetry
 - The axis of symmetry *must* coincide with the global Cartesian Y-axis.
 - Negative nodal X-coordinates are not permitted.
 - The global Cartesian Y-direction represents the axial direction, the global Cartesian X-direction represents the radial direction, and the global Cartesian Z-direction corresponds to the circumferential direction.
 - Your model should be assembled using appropriate element types:
 - For axisymmetric models, use applicable 2-D solids with KEYOPT(3) = 1, and/or axisymmetric shells. In addition, various link, contact, combination, and surface elements can be included in a model that also contains axisymmetric solids or shells. (The program will not realize that these "other" elements are axisymmetric unless axisymmetric solids or shells are present.)
- How Much Detail to Include
- Appropriate Mesh Density

Example - title



Utility Menu > File /title, Truss 2D	e > Change T	itle	Ente	er: Truss 2D
∧Change Title			×	
[/TITLE] Enter new title		•		
ОК	Cancel	Help		

Example0150

ANSYS Computational Mechanics, AAU, Esbjerg 5

Example - Keypoints

Preprocessor > Modeling > Create > Keypoints > In Active CS

Note: An empty # result in automatic numbering.

/PREP7 K,,,, K,,100,, K,,,120,



General format: K,#,X,Y,Z # Keypoint number X Keypoint x-coordinate Y Keypoint y-coordinate Z Keypoint z-coordinate

> Press **Apply** for KP1 Enter 100 in the first field and Press **Apply** for KP2 Enter 120 in the second field and Press **Apply** for KP3

Example - Numbering

Utility Menu > PlotCtrls > Numbering

View Settings

Numbering Symbols ...

Font Controls

Erase Options

Animate

Annotation

Redirect Plots

Write Metafile

Best Quality Image

Hard Copy

Style

Pan Zoom Rotate ... Plot Numbering Controls \times [/PNUM] Plot Numbering Controls Keypoint numbers 🔽 On KP LINE Line numbers □ off Window Controls AREA Area numbers □ Off VOLU Volume numbers □ Off NODE Node numbers □ Off Elem / Attrib numbering -No numbering Device Options ... TABN Table Names □ Off SVAL Numeric contour values □ Off Save Plot Ctrls ... Restore Plot Ctris [/NUM] Numbering shown with Colors & numbers -Reset Plot Ctrls [/REPLOT] Replot upon OK/Apply? Replot Capture Image ... Restore Image Help OK Apply Cancel Multi-Plot Controls Press OK Multi-Window Layout ...

Example0150

Computational Mechanics, AAU, Esbjerg

Switch on Keypoint numbers

Example - Lines

Example0150

Preprocessor > Modeling > Create > Lines > Lines > Straight Line

Create a line between Keypoint 1 and Keypoint 2 and Create a line between Keypoint 2 and Keypoint 3.

L,1,2 L,2,3

• Pick	C Unpick	
🖲 Single	C Box	
C Polygon	C Circle	
C Loop		
Count =	0	
Maximum =	2	
Minimum =	2	
KeyP No. =		
 List of Items Min, Max, Inc 		
I		
OK 🗕	Apply	
Reset	Cancel	
Pick All	Help	

Create Straight Li...

HINT: By clicking with the righthand mouse button you shift between the Pick/Unpick function. This is indicated by the direction of the cursor arrow:

Pick: upward arrow

Unpick: downward arrow

Press OK or Cancel to finish selection

Example – Element Type

Preprocessor > Element Type > Add/Edit/Delete



ANSYS Computational Mechanics, AAU, Esbjerg

Example - Element Type

Preprocessor > Element Type > Add/Edit/Delete



Example – Real Constants

Preprocessor > Real Constants > Add



ANSYS Computational Mechanics, AAU, Esbjerg

Example - Real Constants

Preprocessor > Real Constants > Add

		Real Constants	X	
Real Constants for BEAM3	×	Defined Real Constant Sets	_	
Element Type Reference No. 1		Set 1		
Real Constant Set No.	1			
Cross-sectional area AREA	400			
Area moment of inertia IZZ	13333.3			Broce Cloce
Total beam height HEIGHT	20			to finish
Shear deflection constant SHEARZ				
Initial strain ISTRN				
Added mass/unit length ADDMAS				
OK Apply Cancel	Help	Add Edit Delete	1	
•			-	
		Close		
Press OK				

ANSYS Computational Mechanics, AAU, Esbjerg

Example - Real Constants

Preprocessor > Real Constants > Add

		Real Constants	×	
▲ Real Constants for BEAM3	×	Defined Real Constant Sets		
Element Type Reference No. 1		Set 1		
Real Constant Set No.	2	Set 2		
Cross-sectional area AREA	100			
Area moment of inertia IZZ	833.33			Press Close
Total beam height HEIGHT	10			to finish
Shear deflection constant SHEARZ				
Initial strain ISTRN				
Added mass/unit length ADDMAS				
			_	
OK Apply Cancel	Help	Add Edit Dele	te	
•				
Press OK				

ANSYS Computational Mechanics, AAU, Esbjerg

Example - Material Properties

Preprocessor > Material Props > Material Models



Example - Material Properties

Preprocessor > Material Props > Material Models



Example – Mesh Attributes

Preprocessor > Meshing > Mesh Attributes > Line Attributes > Picked Lines

Line Attributes	S	elect Line 1	
• Pick • C Unpick]		
• Single C Box		∧Line Attributes	×
C Polygon C Circle		[LATT] Assign Attributes to Picked L	ines
U Loop		MAT Material number	1
Count = 1		REAL Real constant set number	
Minimum = 1			
Line No. = 1		TYPE Element type number	1 BEAM3 💌
• List of Items		SECT Element section	None defined 💌
C Min, Max, Inc		Pick Orientation Keypoint(s)	
OK Apply			
Reset Cancel			
Dick Ml Help			
			CancelHelp
ANSYS Pre	ess OK —	Example0150	16
Computational Mech	anics, AAU, Esbjerg	•	

Example – Mesh Attributes

Preprocessor > Meshing > Mesh Attributes > Line Attributes > Picked Lines

Pick C Unpick	S	elect Line 2	
• Single C Box		∧Line Attributes	
C Polygon C Circle		[LATT] Assign Attributes to Picked Lines	
U Loop		MAT Material number	1 -
Count = 1 Maximum = 2		REAL Real constant set number	2
Minimum = 1 Line No. = 1		TYPE Element type number	1 BEAM3 💌
• List of Items		SECT Element section	None defined 🗨
C Min, Max, Inc		Pick Orientation Keypoint(s)	□ No
OK • Apply			
Reset Cancel			
Pick All Help		ОК Apply	Cancel Help
ANSYS - FI	JOD UN	Example0150	1
Computational Mech	anics, AAU, \Box sujerg		

Example - Meshing

Preprocessor > Meshing > Size Cntrls > ManualSize > Lines > Picked Lines

	Element Size on P	
•	• Pick C Unpick	
Select/Pick Lines to specify mesh size for	 Pick Ounpick Single O Box Polygon O Circle Loop Count = 0 Maximum = 1 Minimum = 1 Line No. = Ist of Items Min, Max, Inc OK Apply Reset Cancel 	
	Pick All Help	
		Pr

∧ Element Sizes on Picked Lines	×
[LESIZE] Element sizes on picked lines	
SIZE Element edge length	
NDIV No. of element divisions	•
(NDIV is used only if SIZE is blank or zero)	
KYNDIV SIZE,NDIV can be changed 🔽 Yes	
SPACE Spacing ratio	
ANGSIZ Division arc (degrees)	
(use ANGSIZ only if number of divisions (NDIV) and	
element edge length (SIZE) are blank or zero)	
Clear attached areas and volumes	
· · · · · · · · · · · · · · · · · · ·	
OK Apply Cancel	Help
ss OK when finish with selection	Enter 1 –
Example0150	18

Example - Meshing

Preprocessor > Meshing > Mesh > Lines

Mesh Lines
• Pick • C Unpick
• Single C Box
C Polygon C Circle C Loop
Count = 0
Maximum = 1
Minimum = 1
Line No. =
C List of Items Min, Max, Inc
10 <u></u>
OK Apply
Reset Cancel
Pick All Help

Select individual lines to be meshed by Picking

NB: It is often necessary to "Clear" the model for example if Element Type is to be changed

Select all lines defined to be meshed

ANSYS Computational Mechanics, AAU, Esbjerg

Example – Analysis Type Write Database Log File > Write DB log file Write Database Log to Directories: OK. Enter "example0150.lgw" c:\...\administrator *.lgw Cancel 🗁 c:\ 🗁 DOCUMENTS AN Help 👝 ADMINISTRATOL Cookies Dokumenter Foretrukne Solution > Analysis Type > New Analysis List Files of Type: Drives: Database Log (*.lgw) **C**: Network... • Ψ. New Analysis \times Write non-essential cmds as comments • [ANTYPE] Type of analysis Static C Modal C Harmonic O Transient C Spectrum C Eigen Buckling Press OK Substructuring OK (Cance Help

Example0150

Example – Define Loads

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



Example0150

Example – Define Loads

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



Example0150

Example – Define Loads

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



ANSYS Computational Mechanics, AAU, Esbjerg

Example - Save



Display of Analysis model



Example - Solve

Solution > Solve > Current LS



Example - Solve



Example - PostProcessing

General Postproc > Plot Results > Deformed Shape



Example - PostProcessing



Read Maximum displacement: DMX

ANSYS Computational Mechanics, AAU, Esbjerg

Example – Comments/Questions

- Try Link elements instead of beam elements?
- The "example0150.lgw" can be edited in "Notepad"
- Will the number of elements affect the solution?

File menu



Example0150

Computational Mechanics, AAU, Esbjerg

Exit