Course in ANSYS

Example0504



Objective:

Compute the buckling load

Tasks:

Display the deflection figure?

Topics:

Topics: Start of analysis, Element type, Real constants, Material, modeling, element size for beam models, saving/restoring, orientation keypoints

ANSYS Computational Mechanics, AAU, Esbjerg

Example0504

a $E = 210000 \text{N/mm}^2$ $\nu = 0.3$ L = 5000 mm a = 250 mm b = 450 mm c = 10 mm d = 20 mm e = 15 mm f = 350 mmF = ?

Example - title



Utility Menu > File > Change Title Enter: Cantilever beam /title, Cantilever beam [/TITLE] Enter new title OK Cancel Help

ANSYS Computational Mechanics, AAU, Esbjerg

Example - Keypoints

Note: An empty #



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 ...

Example0504

Computational Mechanics, AAU, Esbjerg

Switch on Keypoint numbers

Example - Lines

Example0504

Preprocessor > Modeling > Create > Lines > Lines > Straight Line Create a line between Keypoint 1 and Keypoint 2.

L,1,2

Create Str	aight Li					
• Pick	C Unpick					
🕑 Single	C Box					
\mathbb{C} Polygon	C Circle					
C Loop						
Count =	0					
Maximum =	2					
Minimum =	2					
KeyP No. =						
 List of Items Min, Max, Inc 						
OK ●	Apply					
Reset Cancel						
Pick All	Heln					

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

Ele	emen	t Types				×	1	
	Defi Type	ned Elemer = 1	nt Types: BEAM189					Press Options
		Add Clo	JISE	Options	Delete Help			

ANSYS Computational Mechanics, AAU, Esbjerg

Example - Element Type

Preprocessor > Element Type > Add/Edit/Delete

▲ BEAM189 element type options	×		
Options for BEAM189, Element Type Ref. No. 1			
Warping degree of freedom K1	Unrestrained 💌		
Cross section scaling is K2	Func of stretch 🔹	Press Help to lear	rn more about the
Shear stress output K4	Torsional only	element.	
Section force/strain output K6	At intgr points		
Stress / Strain (sect points) K7	NONE		
Stress / Strain (sect nodes) K8	NONE		
Stress/Strain (elmt/sect nds) K9	NONE		
User defined initial stress K10	No USTRES routh		
Section integration K11	Automatic 💌		
OK Cancel	Help 🗕		
ANSTS	Example0504	1	9

Example - Section





Example0504



Example – Line Attributes

Preprocessor > Meshing > Mesh Attributes > Picked Lines



Example - Material Properties

Preprocessor > Material Props > Material Models



Example - Material Properties

Preprocessor > Material Props > Material Models



Example - Meshing

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

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

Centric Content 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	Ves
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	🗖 No
OK Apply C	Cancel Help
no OK when finish with calest	ion Entor 5
SS OK WHEN IINISH WITH SELECT	
Example0504	14

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. =						
 List of Items Min, Max, Inc 						
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 - PlotCtrls Menu

Pan Zoom Rotate … View Settings					
Numbering Symbols					
Style 🕨 🕨	Hidden Line Options				
Font Controls	Size and Shape Edge Options				
Erase Options 🔹 🕨	Contours •				
Animate • Annotation •	Graphs Colors				
Device Options Redirect Plots Hard Copy	Light Source Translucency Texturing				
Save Plot Ctrls Restore Plot Ctrls Reset Plot Ctrls	Background Multilegend Options Floating Point Format				
Capture Image Restore Image	Displacement Scaling Vector Arrow Scaling				
Write Metafile 🔹 🕨	Shell Normals				
Multi-Plot Controls Multi-Window Layout	Solid Model Facets Symmetry Expansion 🕨				
Best Quality Image 💦 🕨					

∧Size and Shape	×
[/SHRINK] Shrink entities by	0 percent 🔽
[/ESHAPE] Display of element	□ Off
shapes based on real	
constant descriptions	
SCALE Real constant multiplier	0
[/EFACET] Facets/element edge	1 facet/edge
[/RATIO] Distortion of Geometry	
WN Window number	Window 1
RATOX X distortion ratio	1
RATOY Y distortion ratio	1
[/CFORMAT] Component/Parameter Format	
NFIRST, NLAST Widths	32 0
[/REPLOT] Replot upon OK/Apply?	Replot 🔹
OK Apply	Cancel Help

Example0504

Example – Display of Element



Example – Analysis Type Write Database Log File > Write DB log file Write Database Log to Directories: OK. Enter "example0504.lgw" c:\...\administrator *.lgw Cancel 🗁 c:\ 🗁 DOCUMENTS AN Help 👝 ADMINISTRATOL Cookies C Dokumenter Foretrukne Solution > Analysis Type > New Analysis List Files of Type: Drives: Database Log (*.lgw) **C**: Network... -• 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

Example0504

Static solution – Analysis Options

ANSYS Main Menu	ANSYS Main Menu	۲	Λ	Static or Steady-State Analysis				×
📰 Preferences	🗐 Preferences		No	nlinear Options				
Preprocessor	Preprocessor		[N	LGEOM] Large deform effects	□ off			
🗆 Solution	🗆 Solution		- EN	POPT1 Newton-Rankson ontion				
Analysis Type	Analysis Type				Program chos	en 🔽		
New Analysis	New Analysis			Adaptive descent	ON if necessar	ry 💌		
🔤 Restart	📰 Restart		Lin	ear Ontions				
🔤 Sol'n Controls	Sol'n Controls		Г	UMPM1Use lumned mass annrox?				
⊕ Define Loads	ExpansionPass							
	Analysis Options		[E	QSLV] Equation solver	Program Chos	sen 🔹		
Solve	Define Loads			Tolerance/Level -				
FSI Set Up	E Load Step Opts				1			
🕘 🔟 Unabridged Menu	Physics			- valid for all except Frontal and Sparse Solver	rs			
🗈 General Postproc	⊞ Solve							
🖽 TimeHist Postpro	FLOTRAN Set Up			Multiplier -	0			
E lopological Upt				- valid only for Precondition CG				
	⊞ FSI Set Up		[PF	RECISION] Single Precision -	□ Off		_	
E Design Upt	🔤 Abridged Menu			- valid only for Precondition CG		Select I	Prestr	ess ON
E Prob Design	🖽 General Postproc		ENA	SAVEL Memory Save -	Foff			
E Raulation Opt	I Topological Opt		[14]					
E Run-Time Stats	BOM Tool			- valid only for Precondition CG				
Enich	E Rolfi Tool		-					
	🖾 Design Opt		[PI	VCHECK] Pivots Check	🔽 On			
	Prob Design Rediation Ont			- valid only for Frontal, Sparse and PCG Solver	rs			
	I Run-Time Stats		[59	STIE][PSTRES]				
	Session Editor		Str	ress stiffness or prestress	None			
	🖬 Finish				None			
			NO:	te: If NLGEOM, ON then set SSTIF, ON.	Stress stiff ON	J		
II '	1		LIC.	DEESI ji remperature difference-	Prestress Of	N O		
Activ	vate the			- between absolute zero and zero of active temp :	scale			
	bridged means			011				
Unabridged menu				UK	Cancel		неір	
				Everale 0501				10
				Example0504				19

Example – Define Loads

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



Example0504

Example – Define Loads

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



ANSYS Computational Mechanics, AAU, Esbjerg

Example - Save



Example - Solve

Solution > Solve > Current LS



Example - Finish



Computational Mechanics, AAU, Esbjerg

24

Example - PostProcessing

General Postproc > Plot Results > Deformed Shape



Example - PostProcessing



Read Maximum displacement: DMX

ANSYS Computational Mechanics, AAU, Esbjerg

Eigen Buckling - New Analysis

Main Menu> Solution> Analysis Type> New Analysis



Eigen Buckling – Analysis Options

Main Menu> Solution> Analysis Type> Analysis Options



Example - Shifting

- In some cases it is desirable to shift the values of eigenvalues either up or down. These fall in two categories:
 - Shifting down, so that the solution of problems with rigid body modes does not require working with a singular matrix.
 - Shifting up, so that the bottom range of eigenvalues will not be computed, because they had effectively been converted to negative eigenvalues. This will, in general, result in better accuracy for the higher modes.

∧Subspace Eigenvalue Buckling	×
[SUBOPT] Options for Subspace Eigenvalue Buckling	
SUBSIZ Subspace working size	0
NPAD No. of extra vectors	0
NPERBK No of modes/memory block	0
Strmck Sturm sequence check	At shift+end pts 🔹
Number of subspace iterations	
NUMSSI Maximum number	0
NSHIFT Min, before shift	0
OK Cancel	Help

Subspace working size. Defaults to *NMODE* + 4 (where *NMODE* is input on the <u>MODOPT</u> or <u>BUCOPT</u> command). Minimum is 8. Maximum is *NMODE* +*NPAD*. The larger the value, the smaller the number of iterations (but more time per iteration).

Number of extra vectors used in the iterations. Defaults to 4. The total number of vectors used is *NMODE* +*NPAD*.

∧Subspace Eigenvalue Buckling	×
[SUBOPT] Options for Subspace Eigenvalue Buckling	
SUBSIZ Subspace working size	0
NPAD No. of extra vectors	0
NPERBK No of modes/memory block	0
Strmck Sturm sequence check	At shift+end pts 🔹
Number of subspace iterations	
NUMSSI Maximum number	0
NSHIFT Min, before shift	0
OK Cancel	Help

Number of modes per memory block. If 0 (or blank), perform data management in-memory for all modes (no disk I/O). If greater than zero, use some disk I/O (slower for decreasing *NPERBK* values, but may be needed for large problems). The minimum nonzero value is the number of degrees of freedom per node for the model.

∧Subspace Eigenvalue Buckling	×
[SUBOPT] Options for Subspace Eigenvalue Buckling	
SUBSIZ Subspace working size	0
NPAD No. of extra vectors	0
NPERBK No of modes/memory block	0
Strmck Sturm sequence check	At shift+end pts 🔹
Number of subspace iterations	
NUMSSI Maximum number	0
NSHIFT Min, before shift	0
OK Cancel	Help

Minimum number of subspace iterations completed before a shift is performed. The default is 5 and the minimum is 2. Use *FREQB* on the **MODOPT** command or *SHIFT* on the **BUCOPT** command to define the initial shift point. Maximum number of subspace iterations (defaults to 100). Fewer iterations will be done if convergence occurs before the 100th iteration. Convergence occurs whenever the normalized change in the eigenvalue calculations between successive iterations for the first *NMODE* eigenvalues is less than 1.0E-5.

ANSYS Computational Mechanics, AAU, Esbjerg

∧Subspace Eigenvalue Buckling	×
[SUBOPT] Options for Subspace Eigenvalue Buckling	
SUBSIZ Subspace working size	0
NPAD No. of extra vectors	0
NPERBK No of modes/memory block	0
Strimck Sturm sequence check	At shift+end pts
Number of subspace iterations	
NUMSSI Maximum number	0
NSHIFT Min, before shift	0
OK Cancel	Help

Number of Jacobi iterations used per subspace iteration (used only with the JCG and PCG options on the <u>EQSLV</u> command). Defaults to the number of degrees of freedom divided by the maximum wave front for the model. The minimum is 5.

ANSYS Computational Mechanics, AAU, Esbjerg

Example0504

Sturm sequence check key: ALL --Perform check at all shift points as well as at the end point (default). PART --

Perform check only at all shift points.

NONE --

Do not perform Sturm sequence check.

Eigen Buckling – Expanding Modes

Main Menu> Solution> Load Step Opts > ExpansionPass > Single Expand > Expand Modes

ANSYS Main Menu 🛞		
Preferences	Expand Modes	×
Preprocessor	[MXPAND] Expand Modes	
Solution		
Analysis Type	NMODE No. of modes to expand	
🖬 Fast Sol'n Optn	EREOR EREOE Erequency range	
	TREQUIREQE Trequency range	
Load Step Opts	Elcalc Calculate elem results?	
Output Ctris		
Solution Ctri	SIGNIF Significant Threshold	
H Nonlinear	-only valid for SPRS and DDAM	0.001
		0.001
I Other		
Reset Options		
Read LS File	OK Cancel	Help
Write LS File		
Initial Stress		
Physics		
Solve		
FLOTRAN Set Up		Entor 2
Run FLOTRAN		Enter 3 —
ADAMS Connection		
Abridged Menu		
General Postproc Timellist Destance		
Innehist Postpro Topological Opt		Shift to Yes to Calculate
I Design Opt		element results
Prob Design		
Radiation Opt		
Run-Time Stats		<u>.</u>
🔤 Session Editor 📃 🚽	Example0504	34
	s, AAU, Esbjerg	

Example - Solve

Solution > Solve > Current LS

Solve Current Load Step	Note
[SOLVE] Begin Solution of Current Load Step	Solution is done!
Review the summary information in the lister window (entitled "/STATUS Command"), then press OK to start the solution.	
OK Cancel Help	Close
Prose OK	
F1633 UN	

Example - Finish



Example – Results Summary

ANSYS Main M	nu 🛞	
 Preference Preproces Solution General Po Data & Results Read Results 	tproc e Opts ummary ults	
	LIST Command	×
File File		
	IDEX OF DATA SETS ON RESULTS FILE *****	
E SET E 10 E 20 E 30 E 30 E 20 E 30 E 10 E 10	HHE/FREQ LOAD STEP SUBSTEP CUHULATIVE 1268E+06 1 2 2 12848E+07 1 3 3	
11010	Example0504	37

Example – Read Results



Example – First set



Example0504

ANSYS Computational Mechanics, AAU, Esbjerg

39

Example – Next set



ANSYS Computational Mechanics, AAU, Esbjerg

Example – Next set

