### Course in ANSYS

Example0154



ANSYS Computational Mechanics, AAU, Esbjerg

# Example – Frame 2D

### **Objective:**

Compute the maximum deflection and the member forces **Tasks**:

Display the deflection figure? Display member forces?

#### **Topics:**

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

### Example - title



<b>Utility Menu &gt; Fil</b> /title, Frame 2D		Enter: F	rame 2D		
∧Change Title			×		
[/TITLE] Enter new title		•			
OK	Cancel	Help			

Example0154

# **Example - Keypoints**

#### Preprocessor > Modeling > Create > Keypoints > In Active CS



## Example - Numbering

Example0154

#### Utility Menu > PlotCtrls > Numbering

Pan Zoom Rotate ...

View Settings

Numbering .... Symbols ...

Font Controls Window Controls

Erase Options

Animate

Annotation

Hard Copy

Device Options ... Redirect Plots

Save Plot Ctrls ... Restore Plot Ctris ....

Reset Plot Ctrls

Capture Image ... Restore Image .... Write Metafile

Multi-Plot Controls ....

Best Quality Image

Style

#### ▲Plot Numbering Controls × [/PNUM] Plot Numbering Controls Keypoint numbers □ Off KP LINE Line numbers □ Off AREA Area numbers C Off VOLU Volume numbers □ Off NODE Node numbers □ Off Elem / Attrib numbering + No numbering TABN Table Names C Off SVAL Numeric contour values □ Off [/NUM] Numbering shown with Colors & numbers -[/REPLOT] Replot upon OK/Apply? Replot OK Apply Cancel Help Press OK Multi-Window Layout ...

Computational Mechanics, AAU, Esbjerg

Switch on Keypoint numbers

### **Example - Lines**

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

Create Straight Li... Select KP1 • Pick O Unpick and KP2 HINT: By clicking with the right-C Box Single C Polygon C Circle hand mouse button you shift Select KP2 C Loop between the Pick/Unpick and KP3 Count function. This is indicated by Maximum the direction of the cursor Select KP3 Minimum KeyP No. = arrow: and KP4 • List of Items Pick: upward arrow 🔘 Min, Max, Inc Select KP4 Unpick: downward arrow and KP5 OK. Apply Cancel Reset Press OK or Cancel Pick All Help to finish selection

## Example – Modeling



### Example – Element Type

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



ANSYS Computational Mechanics, AAU, Esbjerg

### **Example - Element Type**

### **Preprocessor > Element Type > Add/Edit/Delete**

Element Types		×			
Defined Element Types:					
Type 1 BEAM3					-1
					5
		Option	ns тог ведика, ететлен с туре кет. -	NO. 1	
		Member	force + moment output K6	Exclude output	
		Output at	t extra intermed pts K9	No intermed pts	]
		Load offs	et in terms of K10	Length units	
			OK Car	ncel Help -	
Add	Ontions		Press Options		
Auu		]			
Close	Help		Press Help to the element. the nex page	o learn more about Find the table on — ə	
ANSYS		Examp	le0154	10	

### **Example - Element Type**

Notice the key option

#### Table 3.7. BEAM3 Item and Sequence Numbers (KEYOPT(9) = 9)

#### number for later use

Γ	Output	ETABLE and ESOL Command I							mand I	BEAM3 element type options	×
	Quantity Name	Item	E	I	Ll	L2	IL3	IL4	IL5	Options for BEAM3, Element Type Ref. No. 1	
	SDIR	LS		1	4	7	10	13	16		
	SBYT	LS	-	2	5	8	11	14	17	Member force + moment output K6	
	SBYB	LS	-	3	6	9	12	15	18		- 1
	EPELDIR	LEPEL		1	4	7	10	13	16	Output at extra intermed pts K9	
	EPELBYT	LEPEL	-	2	5	8	11	14	17	intermed pts	<u> </u>
	EPELBYB	LEPEL		3	6	9	12	15	18	Load offert in terms of K10	
	EPTHDIR	LEPTH	-	1	4	7	10	13	16		
	EPTHBYT	LEPTH	-	2	5	8	11	14	17		
	EPTHBYB	LEPTH	-	3	6	9	12	15	18		
	EPINAXL	LEPTH	34	-	-	-	-	-	-		
	SMAX	NMISC	-	1	3	5	7	9	11		
	SMIN	NMISC		2	4	6	8	<b>•</b> • 10 • •	12-		
	MFORX	SMISC	-	1	7	13	19	25	31	37 43 49 55 61	
	MFORY =	• SMISO •		••)••	8	• • 1•4• •	- 28 -	<b>• 26</b> •	<b>• •</b> 32 •		
	MMOMZ	SMISC	-	6	12	18	24	30	36	42 48 54 60 66	
	P1	SMISC	-	67	-	-	-	-	-	68	
	OFFST1	SMISC	-	69	-	-	-				
	P2	SMISC		71	-	-	-	Re	em	ember MFORX. SMISC.1.7 —	
	OFFST2	SMISC	-	73	-	-	-				
	P3	SMISC		75	-	-	-	-	-		
	P4	SMISC	-	-	-	-	-	-	-	Press Help to launc	h
Γ										Pseudo Node	
							1				O
	TE	MP		LE	3FE		1			<sup>2</sup> <sup>3</sup> <sup>4</sup> this element type	
										this element type.	

### Example – Real Constants

#### Preprocessor > Real Constants > Add



#### ANSYS Computational Mechanics, AAU, Esbjerg

### **Example - Real Constants**

#### Preprocessor > Real Constants > Add

A Deal Constants for DEAM	21		Real Constants	×
MReal Constants for BEAM3	X		Defined Real Co	Instant Sets
Element Type Reference No. 1			Set <u>1</u>	
Real Constant Set No.	1			
Cross-sectional area AREA	•	Enter 1000		
Area moment of inertia IZZ		Enter		
Total beam height HEIGHT		208333		
Shear deflection constant SHEARZ				
Initial strain ISTRN		Enter 50		
Added mass/unit length ADDMAS				
OK Apply Cancel	Help			Edit Delete
	Pre	ess Close	Close	Help
	to f	inish		
Press OK				

ANSYS Computational Mechanics, AAU, Esbjerg

# **Example - Material Properties**

#### **Preprocessor > Material Props > Material Models**



# **Example - Material Properties**



Example0154

### **Example – Mesh Attributes**

### **Preprocessor > Meshing > Mesh Attributes > Picked Lines**

Line Attributes		Select All Lines	
• Pick • C Unpick		▲Line Attributes	×
• Single • Box		[LATT] Assign Attributes to Picked Lines	
C Polygon C Circle		MAT Material number	
C Loop		REAL Real constant set number	1 -
Count = 1		TYPE Element type number	
Maximum = 2 Minimum = 1			
Line No. = 1		SECT Element section	None defined
Clict of Itoms		Pick Orientation Keypoint(s)	I No
O Min May Inc			
OK • Apply			
Reset Cancel		OK 🌒 🛛 Apply	Cancel Help
Pick All Help			
			Press OK
ANSYS Pre	ess OK $-$	Example0154	
<b>Computational Mecha</b>	anics, AAU, Es	sbjerg	

### Example – Mesh size

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

Element Size on P	▲ Element Sizes on Picked Lines	
Pick     Olimpick	[LESIZE] Element sizes on picked lines	
	SIZE Element edge length	
• Single C Box	NDIV No. of element divisions	
O Polygon C Circle	(NDIV is used only if SIZE is blank or zero)	
	KYNDIV SIZE,NDIV can be changed Ves	
Count = 0	SPACE Spacing ratio	Enter 5
Minimum = 1		
Line No. =	ANGSIZ Division arc (degrees)	
	( use ANGSIZ only if number of divisions (NDIV) and	
• List of Items	element edge length (SIZE) are blank or zero)	
C Min, Max, Inc	Clear attached areas and volumes 📃 No	
OK Apply	OK Apply Cancel Help	
Paget Coursel		
Pick All Help	Press OK	
Salaat Diak All		
ANSYS	Example0154	17
Computational Mechanics	s, AAU, Esbjerg	17

### **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

# Example – Analysis Type

### Solution > Analysis Type > New Analysis



### Example – Define Loads

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



Example0154

### Example – Define Loads

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



ANSYS Computational Mechanics, AAU, Esbjerg

### **Example - Save**



Display of Analysis model



# Example – Write DB log file

### File > Write DB log file

Enter "example0154.lgw"

Write Database Log Write Database Log to	g         Directories:       OK         c:\\administrator       Cancel         Image: C:\       Image: Cancel         Image: Cookies       Image: Cookies         Image: Dokumenter       Image: Cookies           Image: Dokumenter
List Files of Type: Database Log (*.lgw)	Drives:
Write non-essential cmds as o	comments 💽

### Example – Solve LS

### Solution > Solve > Current LS



### **Example - Solution Status**



### **Example - PostProcessing**

#### **General Postproc > Plot Results > Deformed Shape**



### **Example - PostProcessing**



Read Maximum displacement: DMX

ANSYS Computational Mechanics, AAU, Esbjerg

ANSYS Main Menu	ANSYS Main Menu	Elem	í ent Table (	ata				X
🗟 Preferences	🗟 Preferences	0	irrently Defi	ned Data and	Status			
Preprocessor	Preprocessor	~~~	arrendy ben	nea bata ana	00000			
	🖽 Solution	La	abel	Item	Comp	Time Stamp	Status	
🗆 General Postproc	🖻 General Postproc	M		۲.				
📃 📰 Data & File Opt:	🔤 Data & File Opts							_
📰 Results Summa	🔤 Results Summary							
Read Results	Read Results							
🗉 Failure Criteria	🕀 Failure Criteria							
Plot Results	Dist Results							
🗉 List Results	🗄 List Results							
Description: De	Duery Results							
🔤 Options for Out	🖬 Options for Outp							
🔤 Results Viewer	Results Viewer							
🔤 Write PGR File	🖬 Write PGR File 🔄							
Description: Nodal Calcs	Description: Nodal Calcs							
🗆 Element Table	🗆 Element Table							
🔤 Define Table	🗰 Define Table 🗲							
📃 🔤 Plot Elem Tab	🧱 Plot Elem Table							
📃 🔤 List Elem Tab	🧱 List Elem Table							
🔤 Abs Value Op	🔤 Abs Value Option							
🔤 Sum of Each 🛙	🔤 Sum of Each Item							
🔤 Add Items	🔤 Add Items							
🔤 Multiply	🔤 Multiply			Add		Lindata	Delete	
🔤 Find Maximun	🥅 Find Maximum			Auu		Opuate	Delete	
🔤 Find Minimum	🥅 Find Minimum							
📃 🔤 Exponentiate	🧱 Exponentiate							
📃 🔤 Cross Produc	🔤 Cross Product			Class			Links	
🔤 Dot Product	🔤 Dot Product			Clos	;e		нер	
🔤 Erase Table	🔤 Erase Table							
E Path Operation:	Determine Path Operations							
	🗄 Load Case							
🗉 Check Elem Sha	🕀 Check Elem Shape			_				
🗰 Write Results	🔤 Write Results			∟ Pres	s Add	to add the firs	t data line	
E ROM Operations	ROM Operations	-						
A	4							

ANSYS Computational Mechanics, AAU, Esbjerg

▲ Define Additional Element Table Iten	15 🔀
[AVPRIN] Eff NU for EQV strain	
[ETABLE] Define Additional Element Table Items	
Lab User label for item	
Item,Comp Results data item	DOF solution       Image: Constraint of the solution o
(For "By sequence num", enter sec	uence
no. in Selection box. See Table 4	.хх-З
in Elements Manual for seq. numb	ers.)
OK Apply	Cancel Help

Scroll down in this menu to find the line "By sequence number" -



ANSYS Main Menu	ANSYS Main Menu	Elom	i ont Tablo I	ata					X
🔜 Preferences	Preferences		urrently Defi	ada Data and	Ctatuci				
Preprocessor	Preprocessor	CI.	unenuy ben	neu Data anu	Status.				
🗉 Solution	🗄 Solution	La	ahel	Item	Comp	Time Stamp	Status		
🗆 General Postproc	🗆 General Postproc			200111	comp	nine Stamp	00000		
🔤 Data & File Opt:	🔤 Data & File Opts	<u></u>	DIVE DEFINEI						
🔤 Results Summa	🔤 Results Summary								
Read Results	Read Results								
🗉 Failure Criteria	🗉 Failure Criteria								
	E Plot Results								
⊞ List Results	🗉 List Results								
Description: De	Duery Results								
🔤 Options for Out	🖬 Options for Outp								
🔤 Results Viewer	🔤 Results Viewer								
🔜 Write PGR File	🔜 Write PGR File 🔄	_							
	Description: Nodal Calcs								
🗆 Element Table	Element Table								
🔤 Define Table	🧰 Define Table 🔶								
📰 Plot Elem Tab	🧱 Plot Elem Table								
🧱 List Elem Tab	🧱 List Elem Table								
🔤 Abs Value Op	📰 Abs Value Option								
🔤 Sum of Each 🛛	📰 Sum of Each Item								
🔤 Add Items	📰 Add Items								_
🔤 Multiply	🗰 Multiply			bdd		Lindate		Delete	
🧱 Find Maximun	📰 Find Maximum			Addin		opada		DOIOTO	
🧱 Find Minimum	📰 Find Minimum								
🔤 Exponentiate	Exponentiate								
🔤 Cross Produc	Cross Product			Clor			Hole		
Dot Product	Dot Product				,e			<u> </u>	
🔤 Erase Table	🔤 Erase Table		11						
Determine	Path Operations								
E Load Case	Load Case								
Check Elem Sha	⊞ Check Elem Shape								
Write Results	Write Results			⊢ Pres	s Add 1	to add the se	cond data	aline	
ROM Operation:	ROM Operations	-						-	
•	•								

ANSYS Computational Mechanics, AAU, Esbjerg



	Element Tabl	e Data				×
	Currently D	efined Data and Sta	tus:			
	Label	Item C	omp	Time Stamp	Status	
	SMIS1	SMIS	1	Time= 1.0000	(Current)	
	SMIS7	SMIS	7	Time= 1.0000	(Current)	
				Lindata	Delet	- 1
		Auu		opuate		
		Close •	1		Help	
			]			
_						
Pr	ess Clo	se				
ANSYS				Example0154		
Computatio	nal Mecha	nics, AAU. Es	biera			

### Example – Plot Line-Element



### **Example – Plot Line-Element**



Example0154

### Example – Clear & Start New



Example0154

Computational Mechanics, AAU, Esbjerg

Exit ....

### Example – Comments/Questions

- The "example0154.lgw" can be edited in "Notepad"
- Change the position of force?
- Display the moment curves?
- Will the number of elements affect the solution?