Part 4: Buckling-Analysis with FEM-System MEANS V11

Euler Cases

For 4 typical cases, the critical load and can be calculated using the Euler Load Cases.Typical buckling problems occur with pressure loaded bars or buckling of thin sheets, plates or shells.



- Case 1: clamped/free $\beta = 2$: $s_1 = 2 \cdot L$:
- Case 2: articulated/articulated eta = 1: $s_2 = 1 \cdot L$
- Case 3: clamped/articulated eta=0.699 : $s_3=0.699\cdot L$
- Case 4: clamped/clamped eta=0.5: $s_4=0.5\cdot L$

$$F_{K} = \frac{\pi^{2} EI}{{l_{K}}^{2}}$$

 $\mathbf{F}_{\mathbf{k}}$ is the Critical Buckling Load und $\mathbf{I}_{\mathbf{k}}$ is the effective length of the column.

Euler Case 1:

The same cantilevered IPB beam with a HE-B300 profile according to DIN 1025 with a length of 5600 mm from the previous chapter 3 is used again. The axial force is 500 kN. How big is the Critical Buckling Load and the Safety Factor for Buckling?





1000.00

1000

300.00

19.00

36.00

		Ab	messungen	i		Flächen		Biegung um die y-Achse			Biegu	ng um die z	-Achse	
Profil	h	b	ts	tg	r	A	Asteg	G	ly	iy	Wy	Iz	iz	Wz
HE-B	mm	mm	mm	mm	mm	cm ²	cm ²	kg/m	cm ⁴	mm	cm ³	cm ⁴	mm	cm ³
100	100.00	100.00	6.00	10.00	12.00	26.00	4.80	20.41	450.00	41.60	89.90	167.00	25.30	33.50
120	120.00	120.00	6.50	11.00	12.00	34.00	6.37	26.69	864.00	50.40	144.00	318.00	30.60	52.90
140	140.00	140.00	7.00	12.00	12.00	43.00	8.12	33.76	1510.00	59.30	216.00	550.00	35.80	78.50
160	160.00	160.00	8.00	13.00	15.00	54.30	10.70	42.63	2490.00	67.80	311.00	889.00	40.50	111.00
180	180.00	180.00	8.50	14.00	15.00	65.30	12.90	51.26	3830.00	76.60	426.00	1360.00	45.70	151.00
200	200.00	200.00	9.00	15.00	18.00	78.10	15.30	61.31	5700.00	85.40	570.00	2000.00	50.70	200.00
220	220.00	220.00	9.50	16.00	18.00	91.00	17.90	71.44	8090.00	94.30	736.00	2840.00	55.90	258.00
240	240.00	240.00	10.00	17.00	21.00	106.00	20.60	83.21	11260.00	103.00	938.00	3920.00	60.80	327.00
260	260.00	260.00	10.00	17.50	24.00	118.00	22.50	92.63	14920.00	112.00	1150.00	5130.00	65.80	395.00
280	280.00	280.00	10.50	18.00	24.00	131.00	25.60	102.84	19270.00	121.00	1380.00	6590.00	70.90	471.00
300	300.00	300.00	11.00	19.00	27.00	149.00	28.80	116.97	25170.00	130.00	1680.00	8560.00	75.80	571.00
320	320.00	300.00	11.50	20.50	27.00	161.00	32.10	126.39	30820.00	138.00	1930.00	9240.00	75.70	616.00
340	340.00	300.00	12.00	21.50	27.00	171.00	35.60	134.24	36660.00	146.00	2160.00	9690.00	75.30	646.00
360	360.00	300.00	12.50	22.50	27.00	181.00	39.40	142.09	43190.00	155.00	2400.00	10140.00	74.90	676.00
400	400.00	300.00	13.50	24.00	27.00	198.00	47.50	155.43	57680.00	171.00	2880.00	10820.00	74.00	721.00
450	450.00	300.00	14.00	26.00	27.00	218.00	55.70	171.13	79890.00	191.00	3550.00	11720.00	73.30	781.00
500	500.00	300.00	14.50	28.00	27.00	239.00	64.40	187.62	107200.00	212.00	4290.00	12620.00	72.70	842.00
550	550.00	300.00	15.00	29.00	27.00	254.00	73.80	199.39	136700.00	232.00	4970.00	13080.00	71.70	872.00
600	600.00	300.00	15.50	30.00	27.00	270.00	83.70	211.95	171000.00	252.00	5700.00	13530.00	70.80	902.00
650	650.00	300.00	16.00	31.00	27.00	286.00	94.10	224.51	210600.00	271.00	6480.00	13980.00	69.90	932.00
700	700.00	300.00	17.00	32.00	27.00	306.00	108.00	240.21	256900.00	290.00	7340.00	14440.00	68.70	963.00
800	800.00	300.00	17.50	33.00	30.00	334.00	128.00	262.19	359100.00	328.00	8980.00	14900.00	66.80	994.00
900	900.00	200.00	19 50	25.00	20.00	271.00	154.00	201 24	494100.00	265.00	10090.00	15820.00	65 20	1050.00

644700.00

12890.00

16280.00

63.80

1090.00

401.00

30.00

400.00

176.00

314.00

Calculaion of the Critical Buckling Load and Safety Factor according Euler Case 1:

 $\begin{array}{l} \mathsf{E} = 210\ 000\ \mathsf{N/mm^2} \\ \mathsf{I_z} = 8\ 560\ \mathsf{cm^4} \\ \mathsf{I_k} = 2\ ^*\ 5\ 600\ \mathsf{mm} \\ \mathsf{tg} = 19\ \mathsf{mm} \ \mathsf{and} \ \mathsf{ts} = 17\ \mathsf{mm} \end{array}$

Critical Buckling Load

Fk = 3.141 * 3.141 * 210 000 N/mm² * 8 560 * 10 000 mm⁴ / 4 * 5600 * 5600 mm²

Fk = 1 413 816 N

Safety Factor for Buckling

S_k = F_k / F = 1 413 816 N / 500 000 N = **2.823**

Safety Factor for Buckling calculate with FEM-System MEANS V11



Buckling Factor S_k = 2.79

Euler Case 3:

This IPE Profile is additionally clamped in X and Y directions according to Euler Case 3. How big is the Critical Buckling Load and the Safety Factor?



Calculation of the Critical Buckling Load and Safety Factor according Euler Case 3:

 $\begin{array}{l} \mathsf{E} = 210 \ 000 \ \mathsf{N/mm^2} \\ \mathsf{I}_z = 8560 \ \mathsf{cm^4} \\ \mathsf{I}_k = 0.6992 \ ^* \ 5600 \ \mathsf{mm} \end{array}$

Critical Buckling Load

 $Fk = 9.869 * 210 \ 000 \ N/mm^2 * 856000000 \ mm^4 \ / \ (0.6992 * 0.6992 * 5600 \ ^2 m^2)$

Fk = 11 571 437.5 N

Safety Factor for Buckling

 $S_k = F_k / F = 11571437.5 N / 500000 N = 23.14$

Safety Factor for Buckling calculate with FEM-System MEANS V11 and a Tetrahedral Model:



Buckling Factor S_k = 22.41

Load Case 1

Now rotate the model with the left and right mouse button (only for DX9) around the Z axis into the following view so that the front Surface 13 can be loaded in the Z direction and the rear Surface 10 can be clamped.



Select the "Edit FEM Project" tab and the drop-down menu "1. Point Load " to enter Load Case 1.



In the next dialog box, enter Load Case 1 with the pro forma value "-1" because the number of nodes in area 13 is not yet known as well as the degree of freedom "in Y direction" with the selection "Surface " and click on the button "Create Point Load".

Create Point Loads	>
Actual Load Case 1 Number of Loads 0 Value of Load -1 ungleich entlang Z-Achse V	New (for example: N)
Degrees of O in X Dire freedom: (Colour of Axis: BLACK: X-Axis; BLUE	ction O in Z Direction ction E: Y-Axis; RED: Z-Axis)
Selection:	
Select Surfaces	O Dragging a model region
Select Nodes	Select all showing nodes
0 00000 110000	Crocical anothing nodes
O Define a coordinate range	O Select all showing surfaces
O Define a coordinate range Show the Loads:	Select all showing surfaces
O Define a coordinate range Show the Loads: .003 Cancel Editor	O Select all showing surfaces Image: Image state showing surfaces Image state showing state showing surfaces Image state showing state sho

Now create the Load with a Double-click on the Surface 13. This is displayed in the 'Select box and must be created there with "Create" as a Point Load with 86 nodes.



Enter the load value

A point load with 86 node forces and the load value -1 has now been generated. Now multiply with menu "8. Editor " and "Load Factor" the loads with the value $500\ 000\ /\ 86 = 5813.95$.

17	😧 💟 🗧				FEM	System MEANS	V11 - FEM Structur	e File C:\Program Files\FEM-Sy
File	es View M	lesh Generation	Edit FEM-Project	FEM-Analysis	Postpro	cessing Tra	ining	
8. s 1	Editor Point Load	Boundary	-Conditions	oundary-Condition: Iow Boundary-Cond	• itions E	lement-Groups	Material-Datas	6. Loads
2	Line Load			Surface Moo	lus is active	- Surface= 1		
3	Surface Load							
5	Gravitation Load							
6	. Temperature Loa	d						
7	. Edit Load Case							
8	. Editor	-						
e E	dit Loads	_		- 🗆	×			
_	Nr	Node	EHG	Value				
F	1	9	2	-5813.95				
	2	10	2	-5813.95		🖳 Edit Load	Case	- 0
	3	13	2	-5813.95				
	4	15	2	-5813.95				
	5	17	2	-5813.95		Actual	Load Case: 1	× < >
	6	19	2	-5813.95			Factor: 5813	.95
	7	21	2	-5813.95				
	8	23	2	-5813.95			multiply	🔘 divide
	9	25	2	-5813.95			🔾 add	🔘 replace
	10	27	2	-5813.95				
	11	29	2	-5813.95			CANCEL	ОК
	12	31	2	-5813.95	~			100 ET
Load	d Case: 1 her of Loads: New Load C Delete Load (Load Fact	< 86 Lo ase Case or t load	Load Cases: Load Cases: Combine Copy I Convert Tempere	1 t Load t Load Cases Load Case ature to a Load Case			177	· MART
-	rressure->Point	LUad						
		0	()					
		Or	×					

Create Boundary Conditions

Rotate to the rear view with the left and right mouse button (only for DX9) around the Z axis to clamped Surface 10. To clamp the model, select the "Edit FEM Project" tab and click on "Boundary- Conditions". Select "Clamped fixed" and the Selection "Select Surfaces" and choose the "Create RBs" button and double-click on the Surface 10 and enter in the Selectbox with "Create".



Create Material Datas



Select the "Edit FEM Project" tab and the icon ^{Material-Datas} to enter the material data such as Young's modulus and Poisson Ratio where steel is always preset.

	Name	Material Datas			
	Youngs modulus	210000			
	Poisson Ratio	.3			
	Density	7.8E-06			
	Heat Coefficient	0			
*					
El	ement Group: 1	Bement TET4		<	>
Đ	ement Group: 1	Element TET4	ı p	<	>
E	ement Group: 1	Element TET4 O Anisotro se	p Oł	<	>

Buckling-Analysis

Select "FEM-Analysis" tab and menu "4. Buckling" to calculate the Critical Buckling Load and the the Safety Factor for Buckling with the Quick-Solver for 2 Buckling-Modes.

ŢŢŢ	4.	Buckling Construction Statics Dynamics	pose Results Select FEM			
	3. 4.	Temperature Buckling				
	5.	Geometric Nonlinear		💀 Buckling Analysis		×
	6.	Contact-Analysis				
	7.	Material Nonlinear			101	
	8.	Formoptimization		Number of Buckling Modes:	2	
	9.	Fatique (AD-Merkblatt SD2)		Tolerance:	.1	
	10.	Casting-Analysis-Tool				
				Start Buc	kling Analysis	
				Start Po	storocessina	

Select the Quick-Solver menu "convert C3D4->C3D10 …" to calculate with the high precision tetrahedral elements TET10.

🖳 INP-Interface for FE-	Solvers	024		×
High Precision	 show and solve with C3D4 (4-node linear tetrahedral elements) show C3D4 and solve intern with a refining mesh of 8 x C3D4 convert C3D4 -> C3D10 and show and solve with C3D10 	2		*
Path for INP-Solver: Path for INP Files:	C:\Program Files\FEM-System_MEANS_V11\Debug\inpsolver\inpsolver64bit.c C:\projekte\300x300_IPE-Profil\tet4.INP]	Browser	
	Select Solver In-Core-Solver Out-of-Core-Solver Start FEM-Solver with INP-Interface Settings Help + Infos Cancel			



After the FEM Analysis you can start the Postprocessor with menu "Start Postprocessing MEANS V11" for the result evaluation.

Postprocessing

Select the tab "Postprocessing" and click on the Icon Postprocessor for the result evaluation automatically.



to start the

			FEM Syst	em MEANS V11 -	FEM Structure File C:\projekte\3
Files View	Mesh Generation Edit	FEM-Project FEM-Analysis	Postprocessi	ng Training	
Show Results	Displacement-Factor Pick, Search Values	List Result Values Legende: Legende 1	Intern Extern	List FEM-File List STA-File	Install DirectX11 Start FAST-Postprocessor
Postprocessing	ra Settings ra	Legende/Tabelle 🕞	Animations 🕞	List Files 🕞	New Beta-Version DX11 🕞

Results							
Buckling Analys	is	Load Case 🗾 🗸					
Nodal Stress Co	ntour	Reaction Forces					
O Element Stress	Contour	O Contour of Forces					
Accuracy							
Edit Accuracy:		Displacement Factor					
		Edit Colours for Legend					
1 3	4	Pick, Search, Save Values					
Select Result Compo	nent						
Displacem	ent in y directio	n v					
	_	Ond Backson and					

Now 3 Load Cases available for buckling result evaluation:

Load Case 1 with linear Static according Euler Case 1 Load Case 2 with Buckling-Factor 1 according Euler Case 1

Load Case 3 with Buckling Factor 2 according Euler Case 1

Displacements in X-, Y- and Z-Direction with Buckling Factor No. 1 = 2.79



v.Mises-Stresses

