FEM-System *MEANS V12*

BEAM-Analysis for

Calculation of Displacements and Stresses



Ing.Büro HTA-Software

<u>www.femcad.de</u> www.fem-infos.com

Part 6: BEAM-Analysis with MEANS V12

Example 1: Cantilever Beam



Create bars in line mode

Start MEANS V12 via the Windows desktop icon and select the tab "File" and menu "New" and click on the project menu "Create a new Model with Beam-Line-Modus" The dialog box for the line mode appears on the right side.

In the Line-Modus here you will find menus containing e.g. Nodes, Rectangles or Circles can be created and connected.

System MEA	NS V12 - FEM Struct	ure File C:\proj	jekte\windschu	tzscheibe\new.	fem		
Files V	2 √iew Mesh Gen	eration Ed	lit FEM-Project	FEM-Analy	ysis	Postprocessing	Training
		FEM-Merge MPC-Contact	Import: STL Export: DXF	-	1.	C:\projekte\wind	lschutzscheil
Га Оре	n 🗐 Save 🗐	Unit	CAD	Path	Fa	L	ast opened f
							×
					Surface	e Nodes Lines	
🛃 NEW	PROJECT	-	ЦХ		Node:		ew _
() 3D	Mesh Generator NET	GEN (STEP, IGE	ES, STL)		X: Y.	0	
O 3D	Mesh Generator GMS	H (STEP)			Z:	0	
0					12	Create Nodes	-
	/3D Beam Model with	Line-Modus				Create Nodes	
() 3D	Shel Model with a Cor	ntainer Mesh Ger	nenerator			Create Lines	
0 20	Diana Madalumba D	and a March Car				Circle / Rectangle	
0 20	Flane Model With a b	eanng Mesn Ger	lerator		м	anipulate Nodes	
	axialsym. Model with a	a Screw Mesh G	enerator		Ca	opy Range of Nodes	
						Unit Nodes	
	NEW PRO	JECT				Check Nodes	
	1					Mesh Generators	
					2	2D Mesh Generator	
						3D Mesh Grid	
					FC		
					EG=		ew
						Load DXF-Lines	
						UNDO / REDO	
						Quit Line-Modus	

Input of the nodes

Input node 1

Click on "New" and enter X = 0, Y = 0, Z = 0Click on "Create Nodes" to create node 1.

Input node 2

Click on "New" and enter X = 2000, Y = 0, Z = 0Click on "Create Nodes" to create node 2.



Now both nodes should be numbered on the screen.

Create beam elements

Select the menu "Create Line" and create 24 beam elements.

With this dialog box, two nodes clicked on the screen can connected to a beam element.

First activate the upper point option and click on the left node so that the node number 1 with the coordinates is displayed.

Then activate the lower point option and click on the right node so that the node number 2 with the coordinates is also displayed.

		🖷 — 🗆 X
		Surface Nodes Lines
		Number of Nodes = 2
	Image: Create Lines X O Node 1 X direction: 0 Y direction: 0 Z direction: 0 Image: O Node 2 X direction: 2000	Nodes from: 1 until: 2 Show Nodes <i>Knotenbereich erzeugen</i> Create a Range of Nodes Surface Nodes Edge Nodes Surface <i>Knotenbereich löschen</i> Delete Range of Nodes
=1	Y direction: 0 Z direction: 0 O all showing Nodes Number of Nodes per Line: 25 Create Lines	Knotenbereich ändern Coordinate-Factor Node: 2 EDIT X: 2000 Y: 0 Y: 0 Z: 0 One Coordinate-Factor Node: Coordinate-Factor Node: 2 EDIT X: 2000 Y: 0 Coordinate-Factor Coordinate-Factor V: 0 Coordinate-Factor Coordinate-Factor Y: 0 Coordinate-Factor Coordinate-Factor Y: 0 Coordinate-Factor Coordinate-Factor V: 0 Coordinate-Factor Coordinate-Factor Y: 0 Coordinate-Factor Coordinate-Factor Q: 0 Coordinate-Factor Coordinate-Factor Q: 0 Coordinate-Factor Coordinate-Factor Y: 0 Coordinate-Factor Coordinate-Factor Y: 0 Coordinate-Factor Coordinate-Factor Coordinate-Factor Coordinate-Factor Coordinate-Factor Coordinate-Factor Coordinate-Factor Coordinate-Factor
	Cancel	☐ Blement groups ☐ Lastwerte anzeigen N Node-Size: Value= .02 Size= large ∨

Refinement

Enter the Number of Nodes per Line = 25 so that 24 beam elements are generated.

Create beam structure

Select menu "Create Lines" to generate a beam structure of 25 nodes and 24 BEAM2 elements.



Input of the Material and Profile Datas

Select the "Edit FEM Project" tab and menu "Material-Datas" menu. There, select the menu "Create Profile Sections" to enter the Material and Profile Datas for BEAM2 or B32 elements.



In the dialog box for BEAM2 Profiles, select "Profile Section 3: Rectangle" and enter a Width of 50 mm and a Height of 200 mm, then select the bottom "Create Element group 1" to generate the material and profile datas.

Material Datas for **BEAM2** Elements with Circle-, Ring-, Rectangle- or Square Profile and a own Profil Data Base in the ASCII-Format for special profile as IPE or HEB which can be edited or extended even with Notepad:

0.0000	Material Datas									
Yaupaa madulua E	210000	-	O Profile Sectio	n 1: Circle Pro	ofile	0	Profile Section	2: Ring Profile		
Poisson Batio P	3	-	Diameter D=			0	utside-D=			
Cross Section Area A	10000	-								
Moment of Inertia IV	2083333	-				In	iside-D=			
Moment of Inertia IZ	3 333333E±07	-	Profile Section	n 3: Rectand	-	0	Dusfile Continu	A. Causan Darf	1-	
Moment of Inertia IZ	7023194	-	I Tome Secto	n o. neotany	c	0	Profile Section	4: Square From	le	
Density PO	7023134	-	Width =	50		Wie	dth = Height =			
Groups tem GK	2	-	Heigh=	200						
Fiber Distance B7	25	-		200						
Fiber Distance RY	100	-		ons from the P	rofile Data Bas	e				
W/K	0	-							_	
WIX	0	-	Nr. Kurzzeicher	n A	IY	IZ	IT	RZ	RY	
ment Group: 1 El	lement BEAM2	< >	2 IPE100 3 IPE120 4 IPE140	10.3 13.2 16.4	171 318 541	15.9 27.7 144.9	1.21 1.74 2.45	27.5 32 36.5	5 6 7	
ment Group: 1 El	lement BEAM2 O B32-Element (cc Copy M	< > > ombinable) laterial Datas	2 IPE100 3 IPE120 4 IPE140 5 IPE160 6 IPE180 7 IPE200 9 IPE220 9 IPE240 10 IPE270	10.3 13.2 16.4 20.1 23.9 28.5 33.4 39.1 45.9	171 318 541 869 1320 1940 2770 3890 5790	15.9 27.7 144.9 68.3 101 142 205 284 42	1.21 1.74 2.45 3.62 4.8 7.02 9.1 12.9 16	27.5 32 36.5 41 45.5 5 55 60 67.5	5 6 7 8 9 100 11 12 135	
ment Group: 1 E BEAM2-Element Material Data Base Create Profil Sections	lement BEAM2 O B32-Element (cc Copy M	< > ombinable) Iaterial Datas OK	2 IPE100 3 IPE120 4 IPE140 5 IPE160 6 IPE180 7 IPE2200 9 IPE220 9 IPE220 10 IPE270 1	10.3 13.2 16.4 20.1 23.9 28.5 33.4 39.1 45.9	171 318 541 869 1320 1940 2770 3890 5790	15.9 27.7 144.9 68.3 101 142 205 284 42	1.21 1.74 2.45 3.62 4.8 7.02 9.1 12.9 16	27.5 32 36.5 41 45.5 55 60 67.5 er IY <	5 6 7 8 9 100 11 12 135 =>IZ	
ment Group: 1 E BEAM2-Element Material Data Base Create Profil Sections	lement BEAM2 O B32-Element (cc	ombinable) Iaterial Datas	2 IPET00 3 IPET40 4 IPET40 5 IPET60 6 IPET80 7 IPE200 8 IPE220 9 IPE240 10 IPE270 Next Refresh	10.3 13.2 16.4 20.1 23.9 28.5 33.4 39.1 45.9	171 318 541 869 1320 1940 2770 3890 5790 H-Profil	15.9 27.7 144.9 68.3 101 142 205 284 42	1.21 1.74 2.45 3.62 4.8 7.02 9.1 12.9 16 rofil Oth	27.5 32 36.5 41 45.5 5 5 60 67.5 er IY <	5 6 7 8 9 100 11 12 135 =>IZ	
ment Group: 1 E BEAM2-Element Material Data Base Create Profil Sections	lement BEAM2 O B32-Element (cc	ombinable) Iaterial Datas	2 IPE100 3 IPE140 4 IPE140 5 IPE160 6 IPE180 7 IPE200 8 IPE220 9 IPE240 10 IPE270 Next Refresh	10.3 13.2 16.4 20.1 23.9 28.5 33.4 39.1 45.9 I-Profil Edit Profil w	171 318 541 869 1320 1940 2770 3890 5790 H-Profil ith Notepad	15.9 27.7 144.9 68.3 101 142 205 284 42 L-/U-Pi	1.21 1.74 2.45 3.62 4.8 7.02 9.1 12.9 16 rofil Oth	27.5 32 36.5 41 45.5 55 60 67.5 er IY <	5 6 7 8 9 100 11 12 135 =>IZ	

Material Datas for B32 Elements only with a Circle- and a Rectangle-Profil

T COLLING	Material Datas	^				
Youngs modulus E	210000		Profile Section 1: Circle Profile	 Profil 	le Section 2: Ring Profile	
Poisson Ratio P	.3		Diameter D=	Outside	e-D=	
Density RO	7.8E-06			Inside-	D=	
Groups Item GK	3					
Profil section H	50		Profile Section 3: Rectangle	O Profile	e Section 4: Square Prof	le
Profil section W	200		Width -	Web -	Height -	
Vector VX	0		Widen = 50		riogra -	
Vector VY	0		Heigh= 200	1		
Vector VZ	1					
WLX	0		 Profile Sections from the Profil 	e Data Base		
WLY	0		Nr. Kurzzeichen A	IY IZ	IT RZ	RY
						-
ment Group: 1 E	lement B32	< >				
WLZ ment Group: 1 E	0 Element B32 () B32-Element (c)	c >				
WLZ ment Group: 1 E O BEAM2-Element Material Data Base	0 Bernent B32 B32-Element (c Copy 1					
WLZ ment Group: 1 E BEAM2-Element Material Data Base Create Profil Sections	0 Bement B32 B32-Element (c Copy M	A sombinable) Interial Datas OK				
WLZ ment Group: 1 E Ø BEAM2-Element Material Data Base Create Profil Sections	0 Barrier B32 B32-Element (c Copy I	 > > >	Next I-Profil	H-Profil L-yU-Profil	Other IV <	=>IZ
WLZ ment Group: 1 E O BEAM2-Element Material Data Base Create Profil Sections	0 Banent B32 B32-Element (c Copy N	ombinable)	Next I-Profi Refreah Edit Profil with 1	H-Profil L-7/J-Profil Notepad (Other IY of mm	=>1Z () m
WLZ ment Group: 1 E O BEAM2-Element Material Data Base Create Profil Sections	0 Barrier B32 B32-Element (c Copy N	ombinable) laterial Datas	Next I-Profil Refresh Edit Profil with 1 Calculate Elit	H-Profit L-/U-Profit Notepad (a	Other IY < o. cm O. mm	=>1Z () m

Consider Units of Beam elements

Especially with the beam structures, you have to consider the correct units:

Coordinates are in millimeters -> then profile and material data also in millimeters

X = 2000 mm, Y = 0 mm -> E-Modulus = 210 000 N / mm², H = 50 mm, B = 200 mm

Coordinates are in meters -> then profile and material data also in meters

X = 2 m, Y = 0 m -> E-Modulus = 210 000 000 000 N / m², H = 0.05 m, B = 0.20 m

Bending stress

With beam elements, almost all structures can be calculated with sufficient accuracy with relatively little mesh generation effort, provided that the cross-sectional area and the moments of inertia can be easily determined.

However, if the profile has an irregular cross-sectional shape, it is very difficult to determine, in particular, the torsional moment of inertia, and then the structure can only be calculated accurately with either area or volume elements.

Since the beam elements on the screen can only be represented as a line, the orientation of the cross section - very important in bending stress - can not always be optically controlled, so that the transverse and vertical axes can be easily reversed.



Create Boundary Conditions

The cantilever beam is fixed clamped at the left end. Create the BCs in the following steps:

- Select the "Edit FEM Project" tab and click on the BC icon to enter the boundary conditions.
- In the BC dialog box, select "Clamped fixed" and "Create BCs" and click on node 1.
- Node 1 is displayed in the Selectbox, there you select "Create" to create the RBs.



Create a Point Load

The cantilever beam is loaded at the right end with a single load of -10 000 N in Y direction. Create the Point Load in the following steps:

- Select the "Edit FEM Project" tab and menu "1. Point Load" to enter the node load.
- Enter Loadcase = 1, Number of loads = 0 and Value of Load = -10000
- Select the load direction "Y direction" and "Create Point Load" and click on node 2
- Node 2 is displayed in the Selectbox, there you choose "Create" to generate the load
- Select the "View" tab and menu "Node-Modus" and switch on the node numbering and load values.

🛃 FEM System MEANS V11 - FEM Structure File C	:\projekte\beams\beam2.fem	
010-		
Files View Mesh Generation	Edit FEM-Project FEM-Analysis Postprocessing Training	
F 1. Point Load -	1. Boundary-Condition:	
Loads Show Loads Boundary	-Conditions Show Boundary-Conditions Element-Groups Material-Datas Editor Temperature	
	current Node 2 - X-Coord.= 2000; Y-Coord.= 0; Z-Coord.= 0	
🖷 – 🗆 X		
Node 2		🖷 – 🗆 X
CLEAR		Surface Nodes Lines
EDIT		Number of Nodes = 25
Nodes Surfaces		Nodes ~
Elements Edges		from: 1
CANCEL CREATE	🗑 Loads — 🗆 🗙	until: 25
	Cirmant Lasdrages 1	Show Nodes
	Number of leads	Create a Range of Nodes
	Value of Load 10000 Kerneneles M	Surface Nodes
		Edge Nodes Surface
		Delete a Range of Nodes
	Degrees of O in X Direction O in Z Direction	Delete Range of Nodes
	freedom: (in Y Direction	Edit a Range of Nodes
	(Colour of Axis: BLACK: X-Axis; BLUE: Y-Axis; RED: Z-Axis)	Coordinate-Factor
	Selection:	Node: 1 EDIT
18 17 16 15 1 4	O Select Surfaces O Dragging a model region 6 5 4 3 2	X: 0
	Select Nodes Select all showing nodes	Y: 0
	U berne a coordinate range U Select all snowing surraces	Z: 0
	Show the Londer 0005 parent in	Node numbering
		Element numbering
	Cancel Editor Create Point Loads	Element groups
	Delete Loads	Show Load Values N
		Node-Size:
		Value= .01
Y		Size= normal ~
A		
Т		
X		

FEM Analysis

Select the "FEM-Analysis" tab and "1. Statics" to start the FEM solver and to calculate the displacements and stresses.



Postprocessing

Select the "Postprocessing" tab and click on the icon Displacments, Stresses and Forces.

Files	View N	lesh Generation	Edit FEM-Project	FEM-Analysis	Postproces	sing Trainin	ng
💦 🖂 Sł	now Results	Displacement-Fa Pick, Search Valu	ctor List Result es Legende: I	Values Legende 1 🕞	Intern Extern	List FEM-File List STA-File	MEANS V11
Postproce	ssing 🖓	Skalieren/Anzei	gen 🗔 🛛 Legend	le/Tabelle 🕞	Animations 🖓	List Files	MEANS V
	Post Results O No O No Co O No Co Co No Co Co No Co Co Co Co Co Co Co Co Co Co Co Co Co	processing : ontour of Displacemen odal Stress Contour ement Stress Contour cy:	t Load C O Re O Co	Case: 1 action Forces ntour of Forces			
			Ec	lit Colours for Legen	d		
	1	3 4	Pick	t, Search, Save Val	Jes		
	Select	Result Component: Displacement in Y Cancel	direction Start Po:	stprocessing			

to show the Contours of



Max. Displacements in Y Direction with BEAM2 Elements = - 0.3725 mm



Max. Displacements in Y Direction with B32 Elements = - 3.8076 mm



Max. v.Mises-Stress with BEAM2 Elements = 58.42 N/mm²



Max. v.Mises-Stress with B32 Elements = 58.42 N/mm²

Example 2: IPE beam with trapezoidal line load

This is followed by a beam calculation with a steel beam IPE-240 with a length of 2000 mm that is loaded with a trapezoidal line load.

88

Exact results according to the beam theory

With the BEAM-Calculator <u>https://calcresource.com/statics-cantilever-beam.html</u> the exact results are calculated according to the beam theory:



 $\sigma_{\rm B} = M_u / W_v = 204.68 \, \text{N/mm}^2$

Material Datas:

Create BEAM model in Line Mode

To create a beam model with 21 nodes and 20 BEAM elements, select the "File" tab and "New" as well as "2D/3D Beam Model with Line-Modus" to display the side menu of the line mode.



Input of the nodes

Input node 1

Click on "New" and enter X = 0, Y = 0, Z = 0Click on "Create Nodes" to create node 1.

Input node 2

Click on "New" and enter X = 2000, Y = 0, Z = 0Click on "Create Nodes" to create node 2.

•	_		×
Surface	e Nodes	Lines	
Node	2	1	New
X:	2000		
Y:	0		
Z:	0		
	Create No	odes	
	Create N	lodes	
	Create I	Lines	
	Circle / Re	ectangle	

Now both nodes should be numbered on the screen.

1	Image: Constraint of the sector of the se
	Mesh Generators 2D Mesh Generator 3D Mesh Grid EG= New Load DXF-Lines UNDO / REDO

Create beam elements

With the menu "Create lines" nodes 1 and 2 are connected with "Number of nodes = 21", thus creating 20 BEAM2 bars, each with a node distance of 100 mm.

🛃 Create Lines		-		×
Node 2				
X direction:	2000]	
Y direction:	0]	
Z direction:	0]	
O Node 1				
X direction:	0]	
Y direction:	0]	
Z direction:	0	ļ]	
 all showing Node Number of Nodes per 	es er Line: 21			
(Create Lines			

Check with menu "Check Nodes" whether there are still overlapping nodes.

	🖳 Please wait		-	1 × 1	
	without Check:				
	Structure model has 21 nodal poin	nts and 20 elements and	1 element group	s	
	with Check:				
	Structure model has 21 nodal poir	nts and 20 elements and	1 element group	os	
	Tolerance: 0.000001				
	Delete nodes which are not co	onnected to an element			
	Cancel only Hic	dden-Line Cł	neck nodal point	3	
*1 =21 =20 =19 =					
18 17	16 E 15 E 14 E 13 E 12				
	12	*11 * 10 * 9	-8		
			=7	* 6 *	5 \$4 \$3 \$2
					2

Create Boundary Conditions

Select the register "Edit FEM Project" and "Boundary Conditions" as well as the degree of freedom "Restraint" and the selection "Click individual nodes" and click on node 1 after the "Create BCs" menu to clamp this in the select box with the "Create" menu .

Node 1 DELETE	📲 Boundary Conditions — 🗆 🗙]
EDIT Nodes Surfaces Elements Edges CANCEL CREATE	Number of Boundary Conditions 6 New Value of Boundary 1E-10 Constraints displacement in In X Direction In Z Direction In X Direction In Z Direction In X Direction In Z Direction	
1 21 20 19 18 17 11	Select Surfaces Oragging a model region Select Nodes Oselect all showing nodes Obefine a coordinate range Select all showing surfaces	¶5 ¶4 ¶3 ¶2
Y	Edit BC-Symbols-Size	
x	Cancel Editor Create BCs Delete BCs	

Create a Range of Nodes

In order to generate a trapezoidal line load, a Node of Range with a defined coordinate range of X = 500 mm to X = 1800 mm must first be generated in Node-Modus.

-4
from X: 500 until X: 1800
from Y: 0 until Y: 0
until Z: 0 from Z: 0
Create a Range of Nodes

Create a trapezoidal line load

Select the "Edit FEM Project" tab and the "Line load" menu and select the drop-down menu "Trapezoidal load positive" in the dialog box to apply a nodal load in the Y direction with load case 1 and the load values L1 = -30 N / mm and L2 = -50 N / mm at the indicated node area.

	Edit Loads			- 🗆 X	🖳 Create a Line Load —		×			
	Nr.	Node	FHG	Value						
•	1	17	2	-1500	Line Load with a Range of Nodes Point Load -> Line Load Hilfe					
	2	16	2	-3153.846						
	3	15	2	-3307.692	Loadcases: 0					
	4	14	2	-3461.539	New Londonna 1					
	5	13	2	-3615.385						
	6	12	2	-3769.231	Transzoidal Load positi v L1 (N/mm) ·					
	7	11	2	-3923.077						
	8	10	2	-4076.923						
	9	9	2	-4230.77	U in X-Direction In Y-Direction U in Z-Direction					
	10	8	2	-4384.615						
	11	7 2 -4538.461		-4538.461	Number of Nodes from a Range of Nodes with Node-Modus: 14					
	12	6	2	-4692.308						
	Load Case: 1 Load Cases: 1 Number of Loads/Load Case: 1 Point Load New Load Case Combine Load Cases Delete Load Case Copy Load Case Load Factor Convert Temperature to a Load Case Pressure->Point Load Change FHG Point Load Change FHG			Point Load bine Load Cases opy Load Case operature to a Load Case Change FHG	X-Length 0 Z-Length 0 Create a Line Load					
OK					Line Load for Arcs and Circles Cancel					

A Point Load with 14 load nodes in the Y direction is now generated. In the Node Modus, select "Show load values" to display the load values in N.



Circular line loads

For circular or round line loads, select the "Line Load for Arcs and Circles" menu, but here the node distance is always determined by the length of the element edges.

Material Datas

Select the register "Edit FEM Project", "Material data" and "Generate bar profiles" and use the menu "Create bar profile for element group 1" to transfer the profile data from "IPE240" from the profile database with Notebad "can be changed or expanded with any text editor.

dit Material Datas			🖳 Create F	Profile Sections				-		×	
Name	O Purfile Section 1: Circle Purfile). Ring Profile					
Youngs modulus E 210000			O FIGINE 3	,	onie .						
Poisson Ratio P	.3	Diameter	D=	Outside-D=							
Cross Section Area A	3910					Insi	de-D=	8			
Moment of Inertia IY	2840000							1			
Moment of Inertia IZ	3.89E+07	O Profile S	ection 3: Rectang	O Profile Section 4: Square Profile							
Moment of Inertia IT	129000	Width =		Width = Height =							
Density RO							1.77				
Groups Item GK 0				Heigh=							
Fiber Distance RZ	60										
Fiber Distance RY	120		Profile S	Sections from the I	Profile Data Base	•					
VK	0		Nr. Kurzze	ichen A	IY	IZ	IT	RZ	RY		
ent Group: 1 E BEAM2-Element Material Data Base Create Profil Sections	Bement BEAM2 O B32-Element (combin Copy Materia OK	< > > al Datas	3 IPE12 4 IPE14 5 IPE16 6 IPE18 7 IPE20 8 IPE22 9 IPE24 10 IPE2	20 1320 40 1640 50 2010 30 2390 20 3340 10 3910 70 4590	277000 1449000 683000 1010000 1420000 2050000 2840000 420000	3180000 5410000 8690000 1.32E+07 1.94E+07 2.77E+07 3.89E+07 5.79E+07	17400 24500 36200 48000 70200 91000 129000 160000	32 36.5 41 45.5 50 55 60 67.5	60 70 80 90 100 110 120 135		
			Next I-Profil H-Profil L-/U-Profil Other IY <=>IZ Refresh Edit Profil with Notepad ○ cm ● mm ○ m								
			Calculate Elementgroup:								

Profile data in the XY plane

If, on the other hand, the vertical axis of the profile does not run in the Z but in the Y direction, the profile must also be reversed accordingly, select "IY <=> IZ" to rotate the profile data.

FEM analysis

Select the "FEM Analysis" tab and use the static solver to calculate the displacements, reaction forces and stresses. Before doing this, back up on the hard drive under any name.

Evaluate Displacements and Bending Stresses

Select the "Postprocessing" tab to evaluate the displacements and bending stresses.

Max. Displacements = -8.21 mm (exactly = 8.19 mm)



Max. v.Mises Stress = 204 N/mm² (exactly = 204 MPa)

