# FEM-System *MEANS V12*

### Bending and torsional moments on a shaft



## (C) 2021 by Ing.Büro HTA-Software Germany-Rheinau

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

# Part 10: Bending and torsional moments an a shaft with MEANS V12

The shaft consists of 2 shaft sections with D = 50 mm and d = 25 mm and a shaft radius of R = 2.5 mm.



The shaft is loaded with a tensile load of 10 000 N, a bending moment of 10 000 Nmm and a torsional moment of 10 000 Nmm. How big are the deformations and stresses.



Tensile load = 10000 N Bending moment = 10000 Nmm Torsion moment = 10000 Nmm

#### **Create FEM model**

The shaft can be generated with MEANS V12 in line mode as in a CAD.

First of all, a half-shaft cross-section is used to generate a 2D mesh with triangles or quadrilaterals. From this, a pentahedron or hexahedral element mesh can be generated with the 3D rotation generator.



The lower line runs at Y = 0.1, which ensures that at Y = 0, the lower surfaces of the hexahedron or pentahedron do not overlap each other incorrectly. Choose tab "File" and "New" to create the model in line mode.

O P Datei	Ansicht Netz	generierung F	EM-Projekt bearbeite	en FEM	-Analyse	Ergebnisauswertur
Neu - Ein	laden 🗁	FEM-Zuladur MPC-Kontakt	g Import: STL + e Export: DXF +	MEANS- SHELL	Pfade F	1. C:\projekte
	Neues Projekt	-Netzgenerierung (S Projekt mit Balken-Lin Projekt mit Behälter-N Projekt mit Wälzlager NEUES PROJEKT	CL, STEP, IGES) ien-Modus erstellen etzgenerator Netzgenerator		Image: Constraint of the second sec	Keu     Inien     Neu     Neu     Neu     Rezeugen     ten erzeugen erzeugen erzeugen erzeugen erzeugen in prüfen n prüfen n prüfen isgenerator letzgitter Neu en einladen 0 / REDO

#### Create nodes

First, 7 nodes must be entered. On the Line Mode tab, click "New" for Node 1 with the coordinates X = 0, Y = 0.1, Z = 0. With menu "Create single node" create node 1.

Create the remaining 6 nodes in the same way:

Node 2 (110 / 0,1 / 0) Node 3 (110/25/0) Node 4 (50/25/0) Node 5 (50/15/0) Node 6 (0 / 12.5 / 0) Node 7 (47.5 / 12.5 / 0)

Now all nodes should be visible on the screen

ichtlinie\welle\neu.fem	
Projekt bearbeiten FEM-Analyse Ergebnisauswertung Training	
g: 10% - e neu 1. Hauptansicht - I. Flächen-Modus - Rotate X - Axis Cross	
	🖳 – 🗆 X
	Flächen Knoten Linien
	Knoten:         7         Neu           X:         47.5
	Knoten erzeugen
	Linian array gan
	Backtook / Kroin
	Nechieux / Weis
	Knoten manipulieren Knotenbereich kopieren
"4	Knoten vereinen
	Knoten prüfen
"5	2D-Netzgenerator
*6 *7	3D-Netzgitter
	EG= 1 V Neu
<b>*</b> 1	DXF-Linien einladen
	UNDO / REDO
	Linien-Modus beenden

#### Save node model

Save the model in the directory C:\ Projects\shaft\cad1.fem or any other directory so that the node model can be loaded again at any time.

#### Create arc

This is followed by the entry of the circular arc with the radius 2.5 mm, select the menu "Rectangle / circle" and enter the center point 47.5 / 15/0, the radius = 2.5, the screening = 4 and the start angle 270 and end angle 360 and select "Create arc" menu.

🖳 Rechteck oder Kreisbogen	- 🗆 X
Kreisbogen Rechteck	
Neu	•
Aktuelle Elementgruppe: 1	Punkte und Linien erzeugen
Kreisbogen-Mittelpunkt:	
X-Koordinate: 47.5	
Y-Koordinate: 15	n <mark>ur vorhandene Knoten</mark>
Z-Koordinate: 0.00	für Selektion anzeigen
Radius: 2.5	
Rasterung: 4	
Anfangswinkel: 270	
Endwinkel: 360	
Kreisbogen erzeugen	
Cancel	

There are now 10 nodes and 4 lines:

atei Ansicht Netzgenerierung FEM-Projekt bearbeiten FEM-Analyse Ergebnisauswertung Training	
FEM-Zuladung Import STL - MEANS.	
MPC-Kontakte Export: DXF - SHELL L	
Einladen 🖓 Sichem 🖓 Vereinen CAD Extern Pfade 🕫 Zuletzt geöffnete FEM-Projekte 🖓	
	🖷 – 🗆 🗙
	Rächen Knoten Linien
	Anzahl Eckknoten = 10
	Knoten ~
	von: 1
	bis: 10
	Knoten anzeigen
	Knotenbereich erzeugen
	Knotenbereich erzeugen
	Flächenknoten
=4	=3 Flächen-Randknoten
	Knotenbereich löschen
	Notenbereich löschen
	Knotenbereich andern Koordinaten-Faktor
<b>5</b>	
<b>-</b> 6 <b>-</b> 78 <sup>510</sup>	Knoten: 6 EDIT
	X: 0
	T: 12.5
	2: 0
	Rooten numereren
-	=2 Rementarippen numerieren
	Lastwerte anzeigen
	Knoten-Size editieren:
	Große= .01
×	Size= normal ~
Á COMPANY A	
•	
In line mede with means "One to line all your base to an at a Cline as	
In line mode with menu "Create lines" you have to create 6 lines:	
Line 1: connect node 1 to node 2	
Line 2: connect node 2 to node 3	
Line 3: Connect node 3 to node 4	
Line 4: Connect node 4 to node 5	
Line F. Connect node 6 to node 7	
Line 6: Connect node 1 to node 6	

#### Enter line 6

Activate the upper option "Point" and double-click on the node 6 on the model so that its coordinates are displayed in the Dialogbox, then activate the lower option "Point" and click on node 1. Finally, with the menu "Line generate" create line 1-6. It may also be useful to enter the node and element numbering.

	🖳 – 🗆 X
🖬 Linien 🛛 — 🗆 🗙	Flächen Knoten Linien
Punkt       6         X-Koordinate:       0         Y-Koordinate:       12.5         Z-Koordinate:       0         Image: Punkt       1         X-Koordinate:       0         Y-Koordinate:       0         Y-Koordinate:       0         Y-Koordinate:       0         Y-Koordinate:       0         Y-Koordinate:       0	Anzahl Eckknoten = 10 Knoten von: 1 bis: 10 Knoten anzeigen Knotenbereich erzeugen Knotenbereich erzeugen Flächenknoten Flächen-Randknoten Knotenbereich löschen Knotenbereich löschen Knotenbereich löschen
Anzahl Knoten pro Linie: 2	Koordinaten-Faktor           Knoten:         1         EDIT           X:         0            Y:         .1            Z:         0
Anzahl Elementgruppen: 1 OK Anzahl Knoten: 10 OK Anzahl Elemente: 9 OK Element löschen: 11 OK	Knoten numerieren     Elemente numerieren     Elementgruppen numerieren     Lastwerte anzeigen N
	Inien     Punkt     X-Koordinate:     Y-Koordinate:     Y-Koordinate: </td

The border of the half 2D shaft now consists of 10 nodes and 11 lines and a triangle mesh can be created.

	е – п х
	Rächen Knoten Linien
	Knoten: 10 Neu X: 49.8097 Y: 14.04329 Z: 0
	Knoten erzeugen
	Einzelknoten erzeugen
	Linien erzeugen
	Rechteck / Kreis
Ť4 Ť3	Knoten manipulieren
	Knotenbereich kopieren
	Knoten vereinen
±5	Knoten prufen
To 778 <sup>910</sup>	2D-Natzgeneratoren
	2D Nutrenter
	3D-ivetzgitter
	EG= 1 V Neu
±1	DXF-Linien einladen
	UNDO / REDO
Y	Linien-Modus beenden
<b>†</b>	
x x	

Select either the "Mesh Generation" tab or the "2D Mesh Generator" line mode menu and generate with a mesh density = 100 a not too fine triangle mesh of 1028 TRI3S elements and 574 nodes for the next step.

1111			<u></u>	
von Elemento	gruppe: 1 bis E	lementgruppe: 1		
Elementtyp:	TRI3S	~		
Netzdichte:	100	~		
Fangradius:	5E-06			
🗌 3D-Mode	ll extrudieren Knoten in Z-Richtung =	5		
	Knoten in Z-Richtung = Z-Objekthöhe =	5		
-	Knoten prüfen	Netzverfeineru	ng	
L				

161

#### Create 3D volume mesh

This 2D mesh now serves as a template to generate a rotation mesh of pentahedron elements. Select the "Mesh Generation" tab and the "Quad Mesh, Refine, Delete" menu and in the next dialog box the "Rotate" tab to generate a rotation mesh.

🖊 Da	tei Ansicht	Netzgenerie	rung FEM-Projekt k	pearbeiten FEM	I-Analyse	Ergebnisa	uswertung	Training	
	4			Quad-Netze, Verf	ieinern, Lösch	nen Jaco	oten-Überlagen obi-Determinar	ung nte testen	Netz aus Flächenmodell
D-Netzg	enerator mit STEP	STL, IGES 🕞	2D-Netzgenerator 🕞	Netze mar	nipulieren	Tu -	Netze prüf	fen 🕞	Netz aus Flächenmodell 🕞
		Vie	Quad-Netze, Verfeinerr errecke Verfeinerr Netzd Y-Wer Anfan Endwi	n, Löschen retter Extrudieren d ein Balken-, Dreiec ichte über Umfang = t der Rotationsachse gswinkel = nkel = UNDO	Rotieren Lös ks- oder Viered 32 = 0.11 0 360 3D-Rotatio	ichen Isksnetz mit Ze	=0 benötigt Lage der Overderar Vorderar © XY- O XZ- Sugen	r Achse izontal hsicht -Ebene Ebene Cancel	×

Select a mesh density of 32 and set the Y-value of the rotation axis to 0.0 to create a model of 32896 PEN6 elements and 18368 nodes. This is followed by a review with a new hidden line and surface model.

🚽 Bitte warten		<u>22</u>		×
Ohne Überprüfung:				
Struktumodell hat 986	88 Knotenpunkte und	32896 Elemente sow	ie 1 Eleme	entgruppe
Mit Überprüfung:				
Strukturmodell hat 183	68 Knotenpunkte und	32896 Elemente sow	ie <mark>1</mark> Elem	entgruppe
Fangradius:	0.000001			
Einzelne Knotenpu	nkte ohne eine Eleme	nt-Verbindung löschen	Î	
			_	
Cance	el	Numerierung prüfe	n	



Now enter a tensile force, a bending moment and a torsional moment using MPC beam elements (yellow lines) in the line mode.



Tensile Load 10000 N Bei

Bending moment 10 000 Nmm

Torsional moment 10000 Nmm

#### Create a Axial Load

Select the "Edit FEM project" and "Bending and torsional moment" tabs and define an Axial Point Load in the X direction of -10000 N that is not loaded to an FEM node but to an outside node point MP(-20, 0, 0).



Select the "Edit FEM Project" tab and select the "Bending and Torsional Moment" load type from the drop-down menu.



In the new dialog box, first select the "Step 1: Select Surfaces" menu and click on surface 4 to list all nodes of this surface.

💀 Bending and Torsional Moments				- 🗆	×
		Point Load	O Bending Moment	O Torsional Moment	
Step 1: Select Sufaces			F <sub>74</sub> = - 500 N	Hebelarm = 20 mm	
Triangle 361 - Knoten         7146 = X 0 Y -2.38511 Z 12.2703           Triangle 580 - Knoten         7154 = X 0 Y -12.2703 Z -2.38511           Triangle 988 - Knoten         7158 = X 0 Y 12.2703 Z -2.38511           Triangle 1038 - Knoten         7151 = X 0 Y 12.2703 Z -2.38511           Triangle 1038 - Knoten         7151 = X 0 Y 14.49061E-07 Z -12.5           Triangle 1048 - Knoten         7155 = X 0 Y -5.46392E-07 Z 12.5           Triangle 1048 - Knoten         7155 = X 0 Y -1.5988 Z -4.68258	All Edges All Nodes	MP Fz = 10 000 N	Hebelarm = 20 mm FY	=-500 N MP F <sub>Y2</sub> = 500 N	
Triangle 1570 - Knoten 7139 = X 0 Y 11.5898 Z 4.68258 Triangle 1586 - Knoten 7160 = X 0 Y -2.38511 Z 12.2703 Triangle 2596 - Knoten 7144 = X 0 Y 2.38511 Z 12.2703 Triangle 2004 - Knoten 7159 = X 0 Y -4.68258 Z -11.5898 Triangle 2008 - Knoten 7156 = X 0 Y -10.363 Z -6.39891 Triangle 2018 - Knoten 7156 = X 0 Y -10.363 Z -6.39898	Select All		Π		
Triangle 2020 - Knoten 7166 = X 0 Y 10.363 Z - 6.98991 Triangle 2022 - Knoten 7140 = X 0 Y 10.363 Z 6.98991 Triangle 2022 - Knoten 7150 - X 0 Y - 10.363 Z 6.98991 Triangle 2216 - Knoten 7158 = X 0 Y - 6.98991 2 - 10.363 Triangle 2216 - Knoten 7157 = X 0 Y - 6.98394 Z - 10.363			$\mathbf{U}$	$\underline{\mathbf{\mathbf{\mathcal{G}}}}$	
Inlangle 2228 - Knoten 7142 - X U Y 5 38991 2 10.363 Tinangle 223 - Knoten 7141 - X O Y 8 3883 2 7 8.3883 Tinangle 2496 - Knoten 7165 - X O Y 8 33883 2 8.3883 Tinangle 2508 - Knoten 7164 - X O Y 6 38832 2 8.3883 Tinangle 5239 - Knoten 7164 - X O Y 6 39891 2 -10.363 Tinangle 5239 - Knoten 7164 - X O Y 6 39891 2 -10.363 Tinangle 5239 - Knoten 7164 - X O Y 6 39891 2 -10.363	Loadcases = 1	$[\mathcal{D}]$	5 1 1	5	
Triangle 5253 - Noten 7148 = X 0 Y -6.98991 Z 10.363 Triangle 5251 - Knoten 7148 = X 0 Y -6.98991 Z 10.363 Triangle 5255 - Knoten 7151 = X 0 Y -11.5898 Z 4.68258 Triangle 5673 K - Knoten 7151 = X 0 Y 10.2707 Z 9.3953	Actual Loadcase= 1				
Triangle 6873 - Knoten 7162 = X 0 Y 1-12 2/03 Z 2.38511 Triangle 6873 - Knoten 7168 = X 0 Y 12 2/03 Z 2.38511 Triangle 6903 - Knoten 7163 = X 0 Y 4.68258 Z -11.5898	FX1 = -10000	FX2 = 0	MPx = -20	X Distance= -20	
Thangle 6317 - Knoten 7147 = X 0 Y -4.68258 Z 11.5898	FY1 = 0	FY2 = 0	MPy = -2.980233	2I Y Distance= 0	
Step 3: Create a Point Load without Lever Am	FZ1 = 0	FZ2 = 0	MPz = -3.57627	0 Z Distance= 0	
	X Lever Arm No. 1= 0	X Lever Am No. 2= 0	Step 2: Ca	alculate MP Coordinates	
REDO Cancel	Y Lever Am No. 1= 0	Y Lever Am No. 2= 0	]		
	Z Lever Arm No. 1= 0	Z Lever Am No. 2= 0	]		

#### **Define nodal load**

Activate "Point Load" and enter only FX1= -10000 N. Since neither a force couple nor a lever arm occur, all other fields must be zero.

#### Calculate MP node

Choose menu "Select All" to select all nodes and deactivate the two center points of the surface with the STRG key. Enter the MP distance in the X direction = -20 mm and calculate the MP coordinates using the "Step 2: Calculate MP with distance" menu.

#### Generate a Point Load without a lever arm

Now select "Step 3: Create a Point Load without lever arm" to create the Axial Load in the X-direction outside the FEM mesh.

#### **Create the Boundary Conditions**

First, the lower surface is clamped with Surface 5 in the x, y, and z directions.

To do this, select the "Edit FEM project" tab and "Boundary conditions" and choose the X-, Y- and Z Direction and "Select Surfaces" as well as "Create RBs" in the RB dialog box and click on the Surface 5. This is displayed in the select box, create the boundary conditions there with "Create".

Files View M	esh Generation Edit FEM-Project FEM-Analysis Postprocessing Training
Loads	Image: Boundary-Conditions     Image: Boundary-Conditions
	Sunace Modus is active - Sunace= .5
Surface 5 VIFAce 5 VIFAce 5 VIFAce 5 VIFAces VIFACES V	Boundary Conditions – – ×
	Value of Boundary       1E-10         Constraints displacement in
	Selection: <ul> <li>Select Surfaces</li> <li>Dragging a model region</li> <li>Select all showing nodes</li> <li>Define a coordinate range</li> <li>Select all showing surfaces</li> </ul> Edit BC-Symbols-Size <td< td=""></td<>
	Cancel Editor Create BCs Delete BCs

#### **FEM-Analysis**

Select the "FEM Analysis" tab and "Statics" to calculate the displacements and nodal stresses with the quick solver.

In the quick solver dialogbox, select "C3D8 (8-node linear isoparametric element)" for a quick FEM Analysis and the button "Start FEM solver with INP interface".

After closing MEANS V12, the FEM analysis is started in a Windows window and ends after a few seconds with a beep.

Then the button "Start Postprocessing MEANS V12 for DirectX11" can be clicked to display the FEM model again in MEANS V12.

Files View	Mesh Generation	Edit FEM-Proje	FEM-	Analysis	Postprocess
1. Statics	✓ Select	t FEM-Solver	Structure Mo Model Dime	odel nsions	
FEM-Analysi	s 🖓 S	Select FEM 🕞	Structure	Info 🕞	FEM-Assister
🖳 Finite E	lement Analysis		2		×
-		C1 (			
C:\pro	jekte \torsionsmoment \per	161.fem			
Selec	O MEANS-So	olver 💿 Qu	uick-Solver		
	Step 1:	Starting FEM Analy	sis	_	1
	Step 2: S	Starting Postprocess	sing		
	Step 3: Refine	ment of the Flemen	t Strassas		
	Select FEM Solvers		Define Result	s	
		Cancel			



#### Postprocessing

to evaluate the displacements Select the "Results evaluation" tab and the icon and stresses graphically.

Files View	Mesh Ger	neration	Edit Fl	EM-Project	FEM-A	nalysis	Postproce	ssing	Training	9
Show Result	s Displa Pick,	acement-F Search Val	actor ues	Legend 1 Diagram 1	*	DXF-Pos Value-Ar	tprocessing nimation 👻	FEN	A INP	FKM-Ri
Postprocessing	rs R	actor/Values 🕞 Legend/Di			Diagram 🖓 🛛 Animations 🗔			rs Lie	st Files 🗔	Fati
	Results Co	rocessing ntour of Dis dal Stress ( ement Stress cy: t Accuracy: 3 Result Comp von Mise Normal S Normal S Normal S	placemen Contour s Contour s Contour 4 s Stress Sign itress Sign itress Sign itress Sign	t ma x na y na z	Load Cas React Conto Disp Edit C Pick, S	e: 1 ion Forces ur of Forces blacement F. Colours for Li iearch, Save	actor egend e Values	×		
	5	Shear St Shear St 1st princ 2nd princ 3rd princ	ress Tau ) ress Tau ) ipal Stress cipal Stress ipal Stress	∧y yz zx s S1 s S2 s S3					]	

#### **Define neutral Stress Range**

Select "Element Stress Contour" to display the v.Mises Element Stresses. Since only the element stresses in the shaft shoulder are of interest, the higher stresses on the beam elements can be masked out either with a smaller maximum value or with a neutral stress area.

To do this, select "Displacement Factor" and define the following neutral stress range with an X range of -15 to +25 and a substitute value of 15:



Displacements	
Structure only with Displacements	
Structure with and without Displacements	$\mathcal{I} \to \mathcal{I} \to \mathcal{I}$
Wireframe     O Edge Model	
without Displ.: with Displace.:	
O Structure without Displacements	
•	
Displacement Factor	
~ ··· · - []	
Stress-Value for Range: 1 Accept	
without Calculate Turn it - +	
Setting a Range of max. and min. Values	
Max. Value: 512.4683 Default	
Min. Value: 0.53256 Default	
No Secondary Stresses on Loads and BCs	
Define a neutral Stress-Range:	
Range switch OFF     Range switch ON	
from X: -15.256 to X: 25	
trom Y: -25 to Y: 25	
from Z: -25 to Z: 25	
Range-Value: 15	
Created	
Lancel UK	

#### Exact result according to Roloff-Matek:



Quelle: Roloff/ Matek Maschinenelemente; 15. Auflage; August 2001; vieweg verlag

The maximum v.Mises Stresses is 40.77 N/mm<sup>2</sup>, this value also agrees well with the exact calculation according to Roloff-Matek with 40.74 N/mm<sup>2</sup>.



#### **Create a Bending Moment**

The bending moment of 10,000 Nmm is defined as a force couple  $F_{x1}$  = 500 N and  $F_{x2}$  = -500 N with a lever arm of 20 mm in Z direction.



Select the "Edit FEM Project" tab and select the "Bending and Torsional Moment" load type from the drop-down menu.

In the new dialog box, first select the "Step 1: Select Surfaces" menu and click on surface 4 to list all nodes of this surface.



#### **Define Bending Moment**

Activate "Bending Moment" and enter  $F_{X1}$ = -500 N and  $F_{X2}$  = 500 N with a Lever Arm in z direction No. 1= 10 mm and No. 2 = -10 mm, all other fields must be zero.

#### Calculate MP node

Choose menu "Select All" to select all nodes and deactivate the two center points of the surface with the STRG key. Enter the MP distance in the X direction = -20 mm and calculate the MP coordinates using the "Step 2: Calculate MP with distance" menu.

#### Generate Bending Moment with lever arm

Now select "Step 3: Create a Bending Moment with lever arm" to create the Bending Moment.

#### Exact result according to Roloff-Matek:



The v.Mises Stress is 11.99 N/mm<sup>2</sup>, this value also agrees well with the exact calculation according to Roloff-Matek with 11.47 N/mm<sup>2</sup>.



#### **Create a Torsional Moment**

The torsional moment of 10000 Nmm can also be replaced by a pair of forces with two equal forces FZ18370 = 500 N and FZ18371 = -500 N and a lever arm of 20 mm.



You can easily generate this load from the previous load by changing the degree of freedom in the X-direction FHG = 1 to the Y-direction FHG = 2 with "Editor".

ung	FEM-Projek	t bearbeiten 🔡 📕	:M-Analyse E	rgebnisauswertung	Training		
Randbe	≻ <b>-</b> ∳ dingungen	<ol> <li>Randbedingu</li> <li>✓ Randbedingun</li> </ol>	ngen 🔹 gen darstellen Knoten-Modus ak	Elementgruppen N tiviert	<b>D</b> Aaterialdaten	6. Belastungen -	Temperatur
B	elastungen			- 0	×		
	Ne	Knoten	EHG	Wert			
	1	18370	2	500	-		
	2	18371	2	-500		_	
10			2				
Aktue Anzah	iller Lastfall: nl Lasten/pro L Neuer Las Lastfal	1 < Lastfall: 2 Il löschen Il löschen	> Anzał Lasttyp: 1 Lastfal Lastfal	nl Lastfälle: 1 Knotenlast Lastfälle überlagem Ile addieren und kopieren mperaturlast einlesen			

#### Exact result according to Roloff-Matek:



The v.Mises Stresses is 5.08 N/mm<sup>2</sup>, this value also agrees well with the exact calculation according to Roloff-Matek with 4.79 N/mm<sup>2</sup>.

