Part 23 - Eccentric Bolt with a local Refinement and a Torsional Moment



(C) 2021 Ing.Büro HTA-Software Maiwaldstraße 24 77866 Rheinau-Germany <u>www.femcad.de</u> <u>www.fem-infos.com</u>

1.0 Part 23 - Eccentric Bolt with a local Refinement and a Torsional Moment

A hole with D = 50 mm is inserted into the eccentric bolt from Part 3. The hole is created by re-meshing the FEM mesh with an additional node list. With this new mesh generation, areas can be specifically refined or new element groups can be created.

1.1 Exact Results

The axial stress can be calculated exactly using the axial force and the cross section of the circular ring and the torsional stress using the torsional moment and the polar moment of inertia.



Axial Stress = Axial Force / cross section of circular ring

= 1 000 000 N * 4 / 3.14159 * (60² - 50²) mm² = 1157.49 N/mm²

Torsional Stress = Torsional Moment / polar moment of inertia

= 10000 N * 200 mm * 16 / 3.14154 * (60³ - 50³) mm³

= 111.93 N/mm²

1.2 Generate the Eccentric Bolt

Select "New" and "3D Tetrahedral Mesh Generator MEANS V12" and generate an FEM mesh from 2454 tetrahedra with the STEP file "Exzenterbolzen.step" from Part 3 with the mesh density "1.6". Then the new dialog box for the local mesh refinement is called up with the menu "Refining" in order to generate an 8x finer FEM mesh with 19632 tetrahedra with the menu "Refine all Elements to 8x TET Mesh". Please save the FEM file "tet4x8.fem" for later.

Mesh Generator V12	😸 3D-Mesh-Generation — 🗆 🔿	×
GENERATE	Mesh Parameters Refining Jacobi-Test Infos	
1.6 ~	Step 1: Create a new Point-List and Surface-List of the actual FEM-Mesh	
Nodes: 4337 With Mesh Meshing Options Create Surfaces Refining	Refine with a Range of Nodes Image: Center-Points of TET-Edges Step 2: Create a Range of Nodes Image: Center-Points of TET-Surfaces Step 4: Generate with refined Nodes Image: Center-Point of TET-Volume Create or load an existing Point-List and create an additional Point-List Center-Point of TET-Volume	
Coarsing	Create an additional Point-List Load a Point-List from a FEM File Load a Point-List from a Node-File	
Cancel	Repair and Refine Tools	
	Load a FEM mesh from ELE-File Refine all Elements to 8x TET Mesh Refine TET Mesh with V11 Cancel OK	
Ľ	×	

1.3 Create a 3D Circular Ring

Create 2D Circular Ring

Select the register "File" and "New" as well as "2D/3D Beam Model with Line-Modus" to create a 2D circular ring with Ra = 30 mm and Ri = 25 mm.





Create a 3D circular ring

Select "2D Mesh Generator" in the menu on the right and generate a 3D pentahedron model with a mesh density in the X and Y direction of "50" and the "3D-Extrusion" option.

🖳 2D Mesh Generation	-	×
from Element Group: 1 until Element Group: 1		
Element Typ: TRI3S ~		
Mesh Density: 50 🗸		
Snap Radius: 5E-06		
QUAD-Meshing		
☑ 3D-Extrusion		
Nodes in Z Direction: 80		
Z-Depth: 240		
Check Nodes Refine Mesh	1	
Cancel Help MESH GENERATI	ON	

The new FEM model with a Z-Depth of "240" mm and "Nodes in Z Direction = 80" consists of 36972 PEN6 elements and 25920 nodes and is saved under the name "Circular Ring.fem".



1.4 Generate FEM mesh with an additional Node-List

Reload the refined Eccentric Bolt "tet4x8.fem" and select the dialogbox for Local Refinement with the "Mesh Generation" and "Local Refinement" tab.

Select "Step 1: Create a new Point-List and Surface-List of the actual FEM-Mesh" to save the Node List of the Eccentric Bolt for the mesh generator.

C ? C ?	Mach Generation Edit FEM-Project FEM-Analysis Postorocessing Training
3D Mesh Generation	ators ht IIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIII
Mesh Generator V12 🔹	
GENERATE	■ 3D-Mesh-Generation – □ ×
moderate ~	Mesh Parameters Refining Jacobi-Test Infos
Elements: 0 Nodes: 0	Step 1: Create a new Point-List and Surface-List of the actual FEM-Mesh
with Mesh	Refine with a Range of Nodes
Meshing Options Create Surfaces	Step 2: Create a Range of Nodes Step 3: Refine the Range of Nodes Step 4: Generate with refined Nodes Center-Points of TET-Surfaces
Refining	
Coarsing	Ureate or load an existing Point-List and create an additional Point-List
New	Create an additional Point-List Load a Point-List from a FEM File Load a Point-List from a Node-File
Cancel	Generate with the additional Point-List from Node: 4338 until Node: 30257
	Repair and Refine Tools Load a FEM mesh from ELE-File Refine all Elements to 8x TET Mesh Refine TET Mesh with V11
	Cancel OK

Then select the menu "Load a Point-List from a FEM File" and insert the FEM file "Circular Ring.fem". The nodes of the Circular Ring are now displayed in the rendering or wire frame model.

In addition, the additional nodes from 4338 to 30257 are entered in the two node fields, which can also be edited.



Generate with the additional Node-List

Select the menu "Generate with the additional Node-List" to re-mesh the FEM mesh of the Eccentric Bolt with the Circular Ring as an additional Node-List.

After about 10 minutes of computing time, an FEM model with 140 199 TET4 elements and 24 630 nodes generated and displayed.

C:\Program Files\FEM-System_MEANS_V12\Debug\meanstetxp.exe

```
Opening file.1.smesh.
Opening file.1.node.
Opening file.1.a.node.
Constructing Delaunay tetrahedralization.
Delaunay seconds: 1.43
Creating surface mesh.
Perturbing vertices.
Delaunizing segments.
Constraining facets.
Segment and facet seconds: 1.173
Removing unwanted tetrahedra.
Hole seconds: 0.041
Repairing mesh.
Repair seconds:
                0.03
Insert additional points into mesh.
```



The circle hole only goes through the entire eccentric bolt if the circular ring is exactly 240 mm long. Annular nodes that lie outside the eccentric bolt are not generated.



1.5 Create circular element group 2

Select the register "Edit FEM-Project" and "Element-Groups" and the menu "Create Element-Groups". In the next dialogbox, select the option "Create element group with a Circular Arc" and the button "Create Element Group" to create circular element group 2 with an outer radius of "25.505" and a very high grid of "1500".

C 0 1 0 =		
Files View M	esh Generation Edit FEM-Project FEM-Analysis Postprocessing Training	
Image: Straight of the straig		-
EG= 1 ×		
✓ ON EG= 1	Element Groups = 2 ×	
EZ ON EG-2	Create Element Group	
	Create Element Group = 2	
V ON EG=3	O Dragging a Model Region New REDO	
⊡ ON EG=4	Select all showing Nodes Middle-Point	
☑ ON EG=5	O Select several surfaces X direction: 0.00	
☑ ON EG=6	Create Element Group with a Coordinate Range Y direction: 0.00	
☑ ON EG=7	Create Element Group with a Circular Arc	
To edit colours click on to the frames	X-Denth Y-Denth 7-Denth Outer Radius: 25.505	
🗹 as Wireframe visible	from: [-100000 until: 100000]	
Refresh Hidden-Line		
Groups 1 - 7	Create Element Group	
	Start Angel: 0	
Create Element Groups	Change Element Group End Angel: 360	
	Number of Groups= 2	
	Edit Number of Bernent Group new: Create Arc	
	Edit Number of Elements: 140199	
	Cancel	
	Cancel	
	WAXIXNAW Y	



1.6 Simulate Drilling

Element group 1 is already preset for "Steel". In order to simulate the drilling, the material data of element group 2 are simply set to a very small value of "0.1". To do this, select the "Edit FEM Project" tab and "Material Data".

	dit Material Datas		-		×		Edit Material Datas		_		\times
	Name	Material Datas					Name	Material Datas			
•	Youngs modulus	210000				•	Youngs modulus	.1			
	Poisson Ratio	.3					Poisson Ratio	.3			
	Density	7.8E-06					Density	0			
	Heat Coefficient	1.2E-05					Heat Coefficient	0			
E	ement Group: 1	Element TET4	_	<	>		Element Group: 2	Element TET4	_	<	>
	 Isotrop 	Anisotrop	p				 Isotrop 	O Anisotrop)		
	Material Data Ba	se	OK				Material Data Ba	ise	ОК		
	Copy Material Da	ita					Copy Material Da	ata			

1.7 Delete element group 2

To create the hole, element group 2 must be deleted. Select the "Mesh Generation" tab and "Quad Meshes, Refine, Delete" and then "Delete" again to delete element group 2 with a subsequent mesh check.

Files View Mesh Generatio	ion Edit FEM-Project FEM-Analysis Postproce	ssing Training
3D Mesh Generators Local Refinement 3D Mesh Generation 🕞 2D M	QUAD-Meshes, Refine, Delete Mesh Generation 17 Manipulate Meshes 17	Check Node Numbering FEMM Mesh Bearing Generator Jacobi-Determinante Vessel Generator Screw Generator Check Meshes Image: Special Mesh Generators Image: Special Mesh Generators
🔛 Quad Meshing / Refine / Conver	rter / Extrusion — C	X 💀 Delete nodes and elements — 🗆 🗙
Quads Refine Converter Extrusion	Rotation Delete Tum Delete Nodes and Elements	 Delete element groups Delete elements Delete nodes Delete all showing nodes Delete with mouse and a depth Delete with several surfaces X-Depth Y-Depth Z-Depth from: 2 until: 2



1.8 Boundary Conditions

With the "Edit FEM-Project" and "Boundary Condition" tab, the front surface of the quader is clamped fix in the X, Y and Z directions.

U	Files	View	Mesh Ge	eneration	Edit FEM	-Project	FEM-Analysis	Pos	stprocessing	Trai	ning				
F_	3. Su	rface Load	÷	D	,,	1. Bo	undary-Conditior	11 -				1			
Loads	Sho	w Loads		Boundary	y-Condition	s 🗹 Sho	w Boundary-Con	ditions	Element-G	roups	Material-Datas	Editor	6. Loads	•	Ter
								Infol	Line						
	🖳 Bo	undary Cor	nditions			-									
	Nur	mber of Boun	idary Condit	ions 7	29	New									
	Val	ue of Bounda	ary	1E-10											
	Cor	nstraints displ	acement in	_	_	_									
			Ĺ	in X Direc	tion	_ in Z Dire ☑ Clamped	ction fixed								
	Sel	ection:													
	0	Select Surfa	ces	۲) Dragging a	model region	n				-				
		Select Node	is		Select all sh	iowing node	is			I MI	للببلية ليرايي	•			~
	0	Define a coo	ordinate ran	ge O	Select all sh	iowing surfa	ces		-	1.	الراليراني للم	لمنسرا	·		
	Edit	t BC-Symbols	Size							·=1 ·		1	17.7	<u> </u>	
				_					7	'न	متعضيه المتعلم ومل	(T)		***	
							-			7				100	
			Tum	BC-Symbols	Color	'S:							A Providence		
														1, 1,	
		Cancel	Ed	itor		Create	BCs							1. <u>1</u>	
						Delete	BCs	1						ן קיק ק	
														الم الم الم	
												A	-1- -1-		
												4 <u>1</u> 3.	1-1-1 - i	1	
				0								ار رز از	<u>ار ارج</u>	77	/
													P	·	
														- 1	
		Y													
		6													
		z		-											
		E-a	×				/								

1.9 Create Surface Load

To create the surface load, first hide EG 2 and refresh the Hidden-Line so that Surface 3 is selected for the Surface Load with 100t or "1 000 000" N. Select "Edit FEM-Project" and "Surface load" to create the load with surface 3.



1.10 Create the Torsional Moment

The torsional moment is a bit more complex because a beam model for the tetrahedron mesh has to be generated. In the Line-Mode enter the node 24631 (0/0 / -50) with "New".



Then use the "Surface Edge Node" menu to create a node range of surface 3.



Now create the nodes 24632 (-100 / 0 / -50) and 26633 (100/0 / -50) in the Line-Mode and connect the two lines 24631/24632 and 24 631/24633 with "Create Lines".



Select the "Edit FEM-Project" tab and "Editor" to create load case 1 with a nodal load. Edit a "2" in the editor with "Number of loads/Load Case" and enter nodes 24632 and 24633 in the Y direction with "FHG = 2" and a load value of "10000" and "-10000".

Nr.	Node	FHG	Value	
1	24633	2	10000	
2	24632	2	-10000	
ad Case:	1 <	> Load C	ases: 1	
umber of Load	ls/Load Case: 2	Load Type: 1	Point Load	
Ne	w Load Case	Co	mbine Load Cases	
Dele	ete Load Case		Copy Load Case	
L	load Factor	Convert T	emperature to a Load Cas	se
Press	ure->Point Load		Change FHG	
		01/		
		OK		
	-10000.00 N			
*				
	\mathbf{i}			
× 🤸	Z Z			
	4		4	
	V		10000 00 N	

1.11 Results Evaluation

After the FEM analysis, the "Result evaluation" tab and the icon menu can be used to evaluate the axial and torsional stresses on the cylinder.

Load case 1: Axial Stress on the cylinder = 1159 MPa (exact = 1157 MPa)



Load Case 2: Torsional Stress of the cylinder = 127 MPa (exact 111 MPa)

