# FEM-System MEANS V11

# **FEM-Contact-Analysis**

# of a Polyurethane Wheel



(C) 2020 by Ing.Büro HTA-Software Germany-Rheinau

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

# Part 19 - Contact-Analysis of a hot cast Polyurethane Wheel with MEANS V11

The model is of a small 95 Shore A hot cast polyurethane wheel with 56 MPa Young's Modulus, 0.498 Poisson's Ratio and 96 MPa Tensile Yield Strength. To keep the FEM model as small as possible we analyzing 1/4 of the wheel with a symmetry surface.

#### Full Model



The inner diameter is fixed, it would be bonded to ball bearings. The wheel is loaded with a steel road at 2225N.

#### Half Model:



#### **Quarter Model**





## Mesh Generation

First an FEM mesh is generated from the CAD Step model assembly.



Select the "Files" tab and select "New" to create a new FEM project.

Select "3D Tetrahedral Meshing with STL, STEP or IGES for the following formats:

- STL this 3D model consists of a triangular outer shell for the 3D Mesh generation, this flexibel format can also be imported and exported. STL Files with holes or gaps, it can also be repaired before meshing.
- STEP today it is the Standard format, note that no CAD assemblies but only single parts can be meshed. Assemblies can be combined via the "Boolean operations" or with "Screw Models" into one part.
- IGES like STEP format but is not as common anymore

Use the "Browser" button to select the "wheel\_road.step" STEP file and click on menu "Start Mesh Generator No. 2 with CAD File" to display it in the mesh generator.

		vileei_loau (		Browse
۲	STEP	O IGES	O STL / AST (ASCII)	Default
				Help
ter W ter W	heel and Road heel and Road	d for FEA_0.0084mm.step d for FEA_assembly_8.4m	icrometer.step	
. Opti	mization and F	Repair		
Opt	timize STL File	before starting Mesh Ger	heration	
	Optimizatio	on with all tests		
sh Ge	nerator No. 2	with OpenGL-Interface		
~ .				11.1
Start	Mesh Genera	tor No.2 with CAD File	with error messages	Help
			AT COM	
sh Ge	nerator No. 3	with automatic repair func	2001	
Opt Ge tart	timize STL File Doptimization Optimization Optization Optimization Optimization Optimization Optimization Opt	before starting Mesh Ger on with all tests with OpenGL-Interface tor No.2 with CAD File	with error messages	Hel

The model can now be seen in the mesh generator and can be rotated as required.



First select the "Mesh" menu and "Mesh-Size" tab and generate with the following setting and a Refinement an FEM Mesh with 174 176 tetrahedral elements with 42 824 nodes. The contact surfaces is also very fine meshed by a contact surface distance of 0.0084 mm or 8.4 Micrometer (could be too much).



#### Export Mesh

After generating the mesh, the FEM mesh with the name "test.fem" must be exported to MEANS V11. Select the menu "File" and "Export Mesh" and save the mesh "test.fem" in the specified debug mesh path to start MEANS V11 automatically with the FEM-Mesh and the Surface Model for selecting Nodes, Edges and Surfaces.



# **Create Element Group 2**

At first we must created Element Group 2 because we are calculated with the two different material datas:

Element Group 1 with the PUR-Wheel and Element Group 2 with the Steel-Road.

Select the "Edit FEM-Project" and "Element-Groups" tab and choose in the new element group menu "Create Element groups" and select in the new dialogbox the option "Select several surfaces" to create EG 2.

Choose "Create EG" and create Element Group 2 with a click on Surface 4 and 8. At least click on the color-box for a blue EG 2.

FEM System MEANS V11 - FE Files View M Surface Load Loads Show Loads	M Structure File C:\projekte\wheel_road\mesh.fem  Aesh Generation Edit FEM-Project FEM-Analysis Postproc	essing Training	Loads • Temperature
	Surface Modus is active	- Surface= 4	r <sub>a</sub>
EG= 2	₩ EG = 2 — □ ×		
☑ ON EG=1	Create EG with mouse and a depth		
☑ ON EG= 2	Create Element group = 2		
⊡ ON EG=3	O Dragging a Model Region		
🗹 ON 🛛 EG=4	<ul> <li>Select all showing Nodes</li> </ul>		
☑ ON EG=5	Select several surfaces		
🗹 ON 🛛 EG=6	O X-Depth O Y-Depth O Z-Depth		
☑ ON EG=7	von: -100000 bis: 100000		
To edit colours click on to the frames	Create EG		
Draw hidden EGs and Surfaces as Wireframe     Weight per EG     Groups 1 - 7     Create Element groups	All elements get the element group 1         Create EG with a define range         Calculate Dimension         from X:       100000         until X:       100000         from Y:       -100000         until Y:       100000         from Z:       -100000         UNDO Element group         Number of Groups=       2         Edit Number of Elements:       174176         Edit       Edit		

#### **Check the Contact-Distance**

The Contact-Distance is very important for the Contact-Analysis. Now we check it easy with the FEM-Analysis and Model Dimension tabs for every Element Group.



Now we are able to read the Contact-Distance 38.10681mm - 38.1mm = 0.00681mm

# **Edit the Contact-Distance**

But it is just as important that you know how to do it to reduce the distance. If the contact distance is a little bit too much the Contact-Solver does not convergence and canceled the calculation.

```
C:\Program Files\FEM-System_MEANS_V11\Debug\cmd.exe - C:\PROGRA~1\FEM-SY~1\Debug\INPSOL~1\INPSOL~2.EXE C:\projekte
```



At first you must show all Nodes of Element Group 2 with the View and "Node-Modus" tab. Then choose menu "Coordinte Factor" to add a Coordinate-Faktor of -0.0068mm.



🛃 Coordinate Facto	r	-		×
Input Factor				
O Multiply	O Divide			
Add	O Replace			
Change Axis				
Change x-value	s with y-values			
Change x-value	s with z-values			
O Change y-value	s with z-values			
Change Coordinates				
X Coordinates	Y Coordinates	🗌 Z Co	ordinates	
Using the select	ed nodes			
from node:	1			
to node:	42824			
Coordinate Factor:	-0.0068			
Moving point 0 /	0 / 0 through the node:	[1	]	
Cha	ange Coordinates with a I	Factor		
Add the	Displacements to the Co	pordinates		
Displacme	nt-Factor: 1			
	CANCEL			

Now the Contact-Distance is 0.000mm.

# **Create Boundary Conditions**

#### **Clamped fixed**

The inner diameter of the wheel is fixed, it would be bonded to ball bearings. Select the tab "Edit FEM-Project" and "Boundary-Conditions" and create the fixed BCs on Surface 9, 11, 12 and 14 with "Create" in the selectbox.

U	Files	View	Mesh G	eneration	Edit FEM-	Project	FEM-Analysi	s Post	processing	Trainir	ng				
F	3. Su	irface Load			$\succ$	1. Bo	undary-Conditio	on: •				1	6 Loads		Ö
Loads	Shc	ow Loads		Boundar	y-Condition	s 🗹 Sho	w Boundary-Co	nditions	Element-Gr	oups	Material-Datas	Editor	o. Lodus		Temperature
							Surface M	lodus is ac	tive - Surface:	= 9					Г
Rufees	-		-							14					
Surface	12	CLEAR							1						
Surface	9	EDIT	Ē.						-//	1					
☑ Nod	les 🖂	Surfaces													
⊡ Ber	nents 🖂	Edges	_					1		[					
CANC	EL	CREATE							/				X		
🛃 Bo	undary C	onditions			6-	- 🗆	×		1						
						_									
Nur	nber of Bo	oundary Cond	litions	684	New	-0.			1						
Vali	ue of Bour	ndary	1E-10				- 8								
Cor	nstraints di	splacement ir	n 🔲 in X Di	rection	🔲 in Z Din	ection	- 8							1	
			in Y Di	rection	Clampe	d fixed	- 8		~h						
Sel	ection:								T						
0	Select Su	ifaces		O Draggin	g a model regio	n	- 11	A	CT.	T					
۲	Select No	odes		Select a	ll showing nod	es	- 8	*~	TYT	1.	YX				/
0	Define a c	coordinate ra	nge	<ul> <li>Select a</li> </ul>	Il showing surf	aces			1,	TT	~				$\setminus$
Edit	BC-Symb	ols-Size						×-		· ~	т <sup>•</sup>	2			
	-	μοκοιοι	mounomoo			-	T.		N.	2	•	<u> </u>			
		Tur	n BC-Symb	ols C	olors:		7.	in in		K	Ţ	Ι	$\backslash$		
-							لير	i Ti	ĴĴ -	Γ,	•		4		
	Cancel	Б	ditor		Create	BCs		T			Г /	- 7	1		
					Delete	BCs	=		ゴゴ		Ť.	-T-	T		
				L					イイ		TT	· •	TT		
									ŢŢ	L'T	4	r 1 ⊷	T		
		Y A							* <u>7</u> 1	- 	T	TT			<u></u>
		9							-571		TT HT		PLF		1 77
													430° 4		T T
x			z							Ċ.	•	- 1	· [		
~															

#### Symmetric-Boundary-Conditions

The nodes of Surface 3 and 4 must be constrained in z direction in order to take advantage of the symmetry of the quartered model.



#### Road clamped

The road is only moved in y-direction so we must constrained the displacements in the x- and z-direction at the top and bottom side.



# **Create the Surface Load**

The roller is loaded with 1.236 Mpa at the top side of the road in y direction.

Select the tab "Edit FEM-Project" and "Surface Load" to create Load Case 1 with a Surface Load of 1.236 Mpa and with a mouseclick on Surface 8 and "Create" in the selectbox.



## **Create the Master-Contact-Surface**

Now we must create the Master-Contact-Surface with Load Case 2. Select the tab "Edit FEM-Project" and "Surface Load" to create Load Case 2 with a Surface Load and with a mouseclick on Surface 8 and "Create" in the selectbox. A value is not necessary.

🧡 Files View Mesh 🛛	Generation Edit FEM-Project	FEM-Analysis	Postprocessing T	raining		
Image: Second secon	Boundary-Conditions	oundary-Condition: + w Boundary-Conditi Surface Modu	ions Element-Group	s Material-Datas	6. Loads	*
Image: Surface 1       DELETE         CLEAR       EDIT         Image: Surfaces       Edges         CANCEL       CREATE         Image: Surfaces       2         Number of Loads       13936         Value of Loads       0         Image: unequal along Z Axis W2=         Degrees of or in X Direction         Selection:         Image: Select Surfaces         Select Nodes         Define a coordinate range	<ul> <li>- ×</li> <li>• ×</li> <li>• in MPa orin N</li> <li>• in Z Direction</li> <li>• Vettical to Surface</li> <li>• Dragging a model region</li> <li>• Select all showing nodes</li> <li>• Select all showing surfaces</li> </ul>					
Calculate Surface Load Value (MPA)	from Load SL-Color:					
Cancel Editor	Create Surface Load Delete Loads		1.			
Y X Z						

# **Create the Slave-Contact-Surface**

Now we must create the Slave-Contact-Surface with Load Case 3. Select the tab "Edit FEM-Project" and "Surface Load" to create Load Case 3 with a Surface Load and with a mouseclick on Surface 2 and "Create" in the selectbox. A value is not necessary.



# **Material Datas**

Now we must input the Material Datas with the tab "Edit FEM-Projects" and "Material-Datas". Input for EG 1 with the Polyurethane Wheel the Youngs modulus of 56MPa and Poisson Ratio of 0.498 and for EG 2 with the Steel-Road the Youngs modulus of 210000MPa and Poisson Ratio of 0.3.

Files View I	Mesh Generation	Edit FEM-Project	FEM-Analysis	Postprocessing	Training	
F3. Surface Load	- 0	→ 1. Bo	undary-Condition: ·	-		1
Loads 🗹 Show Loads	Boundary	-Conditions Sho	w Boundary-Condit	tions Element-Gr	oups Material-Datas	Editor
				Info Line		
EG= 2						
⊠ ON EG=1						
				-		
EG=3						
☑ ON EG=4						
ON EG=5						
🗹 ON 🛛 EG=6						
⊡ ON EG=7						
To adit colours click on to the frames		and the second sec		Y		
Draw bidden ECs and	🖳 Edit Ma	aterial Datas		X		
Surfaces as Wireframe	New York Charles	Maria	Deter			
Weight per EG	Na You	nas modulus 56	I Datas			
	Pois	son Ratio 0.498				
Groups 1 - 7	Den	sity 7.8E-0	6			
Create Element groups	Hea	t Coefficient 1.2E-0	5			
	•					
				-		
	Element (	Group: 1 Eleme	ent TET4			
			O Aniastron			
		souop				
Y	M	aterial Data Base		OK		
4	C	opy Material Data				
7						
L.						
	Z					

# **FEM-Contact-Analysis**

The contact model is ready for the FEM-Analysis. Select the tab "FEM-Analysis" and "Contact-Analysis" and start the calculation to solve the displacements and stresses with the TET4 tetrahedral elements in a few minutes.



# Postprocessing

After the calculation the results can be evulated. Select the tab "Postprocessing" and choose the "Contour of Displacement" or "Nodal Stress Contour".

Files	View	Mesh Generatio	on Edit F	EM-Project	FEM-A	Analysis	Postpro	cessi	ng
	Show Resu	Its Displaceme Pick, Search	nt-Factor Values	Legend 1 Diagram 1	*	Value-Ar	nimation	•	List FEN
Postpre	ocessing	Factor/	Values 🖓	Legend/Dia	gram 🕞	Anir	nations	Fa	List F
Postpri	Post Results C CC No Ed Accura Ed 1 1 Select	rs Factor/ processing a: ontour of Displacement odal Stress Contour ement Stress Contour ement Stress Contour cy: it Accuracy: 3 4 Result Component: 3 4 Result Component: CPRESS CSHEAR1 CSHEAR2 CPRESS CSHEAR1 CSHEAR2 CPRESS CSHEAR1 CSHEAR2 CSHEAR2 Normal Stress Sig Normal Stress Sig Normal Stress Sig Shear Stress Tau Shear Stress Tau	ma x ma y ma z xy yz	Legend/Dia	gram IS — 1 → Forces f Forces ement Fac urs for Leg ch, Save N	Anir Anir Anir Anir Anir Anir Anir Anir Anir Anir	x		List F
		1st principal Stres 2nd principal Stre 3rd principal Stres	s S1 ss S2 ss S3						

# Displacements

Max. Displacement in y direction is -1.458 mm



# **1st principal Stresses S1 of the Wheel**

Maximale 1st principal Stresses S1 of the Wheel is 21.9 MPa



# **3rd principal Stress S3 of the Wheel**

Minimale 3rd principal Stresses S3 of the Wheel is -40.04MPa

ன FEM System MEANS V11 - F	EM Structure File C:\projekte\wheel_road\roller1.f	em		
Files View	Mesh Generation Edit FEM-Project FEM-A	analysis Postprocessing	Training	
Show Results	Displacement-Factor Legend 1 +	Value-Animation -	EM-File	FEM-Modell
	Pick, Search Values Diagram 1 -	List S	TA-File	STA-/FRD-Modell
Postprocessing	Factor/ values	Animations I <sub>M</sub> List	t Files 1 <sub>2</sub> Fatique-Analysis 1 <sub>2</sub>	MEANS-Android 12
LOAD CASE= 1				
\$3				
15.894				
7.9026				
-0.0883				
-8.0792				
-16.070				
-24.061				
-32.052				
-40.043				
40.045				
Edit · +	ALC: NO			
		A start in		
			and the second s	
		-45.043 MPa	•	
		the free	$\sim$	
			$\sim$	
			$\langle \rangle$	
Y				
<b>A</b>				
X 7				

# v.Mises Stresses of the Road

Maximale v.Mises Stress of the Road is 98.6 MPa

