Part 31: Temperature Distribution and Heat Flux of a Windows Profile

The following window profile from "Porenbeton-Wärmebrückenkatalog" Side 908 is recalculated with the following element groups, thermal conductivities and convections:





Now start the FEM system MEANS V12 via the desktop icon

Select the "File" tab and the "New" menu and select "2D mesh generator FEMM for complex element groups" in the project menu to create a 2D FEM model with the 2D mesh generator FEMM.

🔜 NEW PROJECT			×
O 3D Mesh Generator NETGEN V1 (STEP,	IGES, STL)		
O 3D Mesh Generator GMSH (STEP)			
3D-Mesh Generator NETGEN V2 for com	lex Structur	es In	stall
2D-Mesh Generator FEMM for comlex Eler	ment <mark>Gro</mark> up	s	
O 2D/3D Beam Model with Line-Modus			
○ 3D Shel Model with a Container Mesh Ger	nenerator		
NEW PROJECT			

The FEMM user interface now appears. Choose menu "File" and "New" and create a new project with "Magnetics Problem".

fe fe	:mm - [Unb	enan	nt]														
LED .	File E	dit	Viev	v P	roble	em	Gric	I Op	perati	on	Prop	erti	es l	Mesh	A	nalys	is V	Vindo
<u> </u>	6		• /	1		0		1	8	12		0		\bigcirc	Ð	Ð		⇔ r
	: 12 ⁰		\$	<u>(</u> 2	15)	Ť.	28	125	觐	25	53	12	155	3	85	125	22	25
브		ß		15		65		8		13		15		10		38		2
2	12	Cr	eate a	a new	/ pro	blem	1					×		8	2	32	52	20
Ð		1												2		15		82
Ø	8		Magn	etics	Prob	em						_		3		13	83	83
								ОК		Ca	ncel			32		33		
0]	15		- V	- 2	•		- 5	÷.	-22	2	v			1	15	34	25	3
î		2		121		3		37 8.		12		1		3		72		10
t	-	15	\$ 0	(c).	3)	2	1		22	25		(2)	5	3	88	3	3 2	28
\$	-	8		15				24				5		18		24		12
	-		8	21		8		32	22		8	21	12	*		32	52	12

TAB key

With the Tab key and the icon, all 23 corner nodes can now be entered one after the other in a dialog box. The coordinates are entered in millimeters but must later be converted to meters in MEANS to match the thermal conductivities.

	File	Edit	Vie	ΞW	Pi	roble	em	Grid	Ope	ation	Pro	operti	es	Mesh	An	alysis	Wi	ndow	Hel	p	
: <u>D</u>) 🗃		•	7	2	0	0		E	123	R	5	=	\bigcirc	<u>جا</u>	B] €		×	0	
 Ed																					
€																					
2																					
<u>a</u>						п															
						0															
T L																					
4																					
4																				•	
×																					
grid size														-	_						
														Ent	ter Poi	int				×	
														×	(-coord	4	13	33			
														У	/-coord	1	-5	23			
																	ОК		Canc	el	
																3				0	

DXF import

The nodes can also be created first with AutoCAD or MEANS and imported into FEMM via the DXF interface. Here, however, care must be taken that no overlapping or double nodes or lines are created.

Edit FEM file with editor

If you load the FEM file into any text editor, e.g. Notepad, you can quickly edit or delete the node coordinates or lines.

For a correction with the editor you should save the FEM file and exit FEMM. Then edit the FEM file with the editor and start FEMM again and reload the FEM file.

[NumPoints] = 23 57 -1800 0 0 305 -1800 0 0 305 -721 0 0 305 -800 0 0 1435 -800 0 0

1435	-721	0	0
1435	-523	0	0
1435	-442	0	0
1435	-494	0	0
215	-523	0	0
57	-721	0	0
0	-721	0	0
0	-485	0	0
133	-485	0	0
133	-523	0	0
133	-513	0	0
203	-513	0	0
215	-442	0	0
215	-494	0	0
203	-442	0	0
215	-721	0	0
203	552	0	0
133	552	0	0
[Num	Segme	ents]	= 0
[Num/	ArcSeg	gmer	nts] = 0
[Numl	-loles]	= 0	
[Num]	BlockL	abel	s] = 0

Definition of the 7 element groups

After all 23 nodes have been entered, switch to the icon **1** to define the lines for the 7 element groups by clicking on the following pairs of nodes one after the other.

Elementgruppe 1 1 - 2, 2 - 3, 3 - 6, 6 - 7, 7 - 10, 10 - 21, 21 - 11, 11 - 1Elementgruppe 2 4 - 5, 5 - 6, 6 - 3, 3 - 4Elementgruppe 3 10 - 7, 7 - 9, 9 - 19, 19 - 10Elementgruppe 4 19 - 9, 9 - 8, 8 - 18, 18 - 19Elementgruppe 5 12 - 21, 21 - 10, 10 - 15, 15 - 14, 14 - 13, 13 - 12Elementgruppe 6 16 - 17, 17 - 22, 22 - 23, 23 - 16Elementgruppe 7 15 - 10, 10 - 18, 18 - 20, 20 - 17, 17 - 16, 16 - 15



(DOD)	File	Edit	View	Problem	Grid	Opera	ation	Proper	ties N	Aesh	An	alysis	Window
: <u>C</u>	נ 🖻		• /	<u>n @ (</u>) <u>r</u>		हा ह			\bigcirc	日 日	50] ⇔ r
	2									•20	2	•	
2			» •		-52		12		1	52	10		
Ð	1		8 9		83	÷		89	3		\overline{e}		3
P	120	S7	s. 1	1 A 14	25	2	15	a a	11		¥2	9	8 8
0	1		<i>.</i> :		10	8		39 -	12		25		8
	8		0 X	8 8 X	-53	• •	13	8 8	3	83	×.		а в
<u>Î</u> л	- 22		34 		25	12			3		1		14
4	20	12	10. I			0 10	84	88 <mark>-</mark> 6	1	172		12	
₽			*		22			2	*		5		
		•	 G		85					-	*: 	2	
⊾ grid													
size						10 AN		**	2				
	-	12											
	2		10 1	NST .	255	12		20	13		2		14
	8		82 X		52	e 18	22		8	52	38	8	
	ϵ		8	8	-83	12		20	3		\overline{e}		3
	- 22	9	8 . 7	1 21 1 2	26	v 9	15	3 1 3	11	1	92	9	8 8
			1	(s).	-82	13		25			\$2		1
	8		18 X		63		8	8 B	3	-	8		
	1		33-	88 d	46	14			3 2		22		3
	1 °	12	407. ¹ 2	8 97 9 <u>5</u>	20	6 12	19	98 - S	1 22	201	23	12	
			38	5	**			8	10		<u>.</u>		
	×.	•	· d	d		8 - S	12	x 3	3 - S	- 20	E.		x 8

Define Materials

Only the FEM mesh is required to be saved to an ANS file, but the FEM mesh must first be generated and an analysis performed. To do this, simply select the material "Air" for analysis from the "Properties/Material Library" menu.

Library Materials Air Hard Magnetic Materials Content Magnetic Materials Content Magnetic Materials Conductors	Model Materials	
Copper AWG Magnet Wire Copper SWG Magnet Wire Copper SWG Magnet Wire Metals Handbook DC Magnetization Curves Source 15% Copper Clad Aluminum Magnet Wire 10% Copper Clad Aluminum Magnet Wire		

Move Materials

The material must be moved in each element group with the icon by first placing the material with the left mouse button and activating it red with the right

mouse button. The material "Air" can now be selected with the icon



Analysis in FEMM

Finally, the FEM mesh is generated with the icon and the analysis is carried out with the icon. If no error message is displayed, there is now a new file with the extension .ANS in the project directory, which can be imported and processed with MEANS. FEMM can be closed and is no longer needed.

File Edit View Problem Grid Operation Properties Mesh Analysis Window Help Image: Im	
Î 🖌	
	a s
	12
	14 IZ
	-
	28
	12
	N 12
	1
	10
	19 12
	a 2
	14 P

FEMM-Interface

In the MEANS user interface, select the "Mesh generation" tab and the right button "FEMM" to import the ANS file.

Files	View Mesh G	eneration Edit FEM-Proje	ect FEM-Analysis Postproc	essing Training			
3D M Local	1esh Generators I Refinement		QUAD-Meshes, Refine, Delete	Check Node Numbe Jacobi-Determinante	ring :	SOLIDGET FEMM	N SILO GMSH
3D Mesh Ge	eneration 🕞	2D Mesh Generation 🕞	Manipulate Meshes 🕞	Check Meshes	Fa	Mesh G	enerators 🕞
janisieren 👻 N	leuer Ordner	. A. C. C. C.	- <u>.</u>				(ja 1
→ ↓ ↓ ↓	Dieser PC > B	oot (C:) > projekte > fens	terprofil	~	õ	fensterp	rofil durchs
PerfLogs	^	Name	Datum	Typ	Grö	Be	Markierum
ProgramData	r -	femm1	02.04.2023 15:19	ANS-Datei		340 KB	
Programme		<pre>fenster_femm</pre>	14.04.2023 16:58	ANS-Datei		119 KB	
Programme ((x86)	📃 fensterprofil-eckknoter	n 24.04.2023 10:34	ANS-Datei		200 KE	
🧧 projekte		grundriss	05.04.2023 20:33	ANS-Datei		487 KB	

A 2D model consisting of 4046 TRI3S, 2232 nodes and 7 element groups is created.

Rendering mit Netz	Ansicht Netzgenerien O Drahtgitter Schattien mit Kanten + Hidden-L	ing FEM-Projekt bearbeiten ing: 10% - Image:	FEM-Analyse	Ergebnisauswertung ansicht v Surfaces	Training 2. Knoten-Modu Nodes	Hintergrund Axis Cross	
		Y X					

Enter the Thermal Conductivities

The Thermal Conductivities in W/mK for all 7 element groups are entered with the "Edit FEM-Project" tab and the "Material-Datas" menu:

Element group 1 - Reinforced Concrete WL = 2.3 W/mK Element group 2 - Isulation-WL = 0.032 W/mK Element group 3 - Floor-Isulation-WL = 0.04 W/mK Element group 4 - Estrich WL = 2.3 W/mK Element group 5 - Window Frame WL = 0.13 W/mK Element group 6 - Estrich Strip WL = 0.04 W/mK Element group 7 - Isulation-WL = 0.032 W/mK

on D ndarv	Edit FEM-Project	FEM-Analysis oundary-Condition: + ow Boundary-Conditi	Postprocessing	Training Training Material-Datas Editor 6. Loads
		1920	Info Line	
	1	Edit Material Datas		– 🗆 X
		Name	Material Datas	
	•	Youngs modulus	1	
		Poisson Ratio	0	
		Density	0	
		WK	0	
		WL	2.3	
		CV	0	
		RV	0	
		QD	0	
		DAMP	0	
		Thickness	1	
		5		
			_	
		Element Group: 1	Element TRI	6S < >
		<u></u>	e -	
				eratur
		Material Data Ba	ise	OK
		Copy Material Da	ata	

Entering the Convections

The window profile is loaded with an outside and inside convection.

Outside Convection

Convection = 20 W/m²K Temperature = -5°C

Inside Convection

Convection = 5.89 W/m²K Temperature = 20° C



Create a Range of Nodes

Select the "View" and "Node-Modus" tabs and first create all edge nodes with "Edges" and "Show nodes".

Select "Delete Range of Nodes" and delete all nodes of the inner area for the Outside Convection with a rectangle and all nodes of the outer area for the Inside Convection.





Use the temperature icon Temperatur from the "Edit FEM project" tab to generate the two convections with the "Convection, load type 8 (W/m²K)" menu with the selection "Select all showing nodes" in load case 1.

Nodal Temperatures (°C)	Stationary O Transient O Statics
Point Heat Source, Load Type 2 (W)	Material Datas
	Material Data Base
Surface Heat Source, Load Type 7 (W/m ²)	Convection Data Base
	controlion data data
Convection, Load Type 8 (W/m ² K)	Internet-Links:
	Wärmeleitfähigkeit für unter. Materialien
Radiation, Load Type 9	Spez. Wärmekapazität für unter. Materialien

Cancel	Editor	Create Conve	ction
) Define a coord	linate range		
) Select Nodes	۲) Select all showing no	des
) Select Surface	is C) Dragging a model reg	jion
election:			
	() in y direction	Vertical to Surf	ace
	O in x direction) in z direction	
Degree of Freedo	m		
Temperatur:	-5	(°C)	
/alue of Load	20	(W/m²K)	
Number of Loads	0	New	
uneni Luaucase			

Change node coordinates from millimeters to meters

WLs and convection were entered in meters but the coordinates in millimeters, so the coordinates must now be divided by "1000" using the "Edit FEM project" and "Editor" and "Node coordinates" tabs.

Datei	Ansicht Netzgeneri	erung FEM-Pr	ojekt bearbeiten	FEM-Analyse Ergebnisauswertung Training			
ngen ⊻ E	Knotenbelastung 🔹 👻	Randbedingung	1. Randbed en Randbedi	dingungen - Jahren Koordinater ingungen darstellen Infozeile Jahren Koordinater Aterialdaten Editor 1. Elementgruppen 1. Elementgruppen			
tenkoordina	iten		- 0	X Koordinaten Esktor			
Nr.	X-Koordinaten	Y-Koordinaten	Z-Koordinaten	4. Materialdaten			
1	.057	-1.8	0	Faktor setzen : 5. Randbedingunger			
2	.057	-1.79918	0	O multiplizieren			
3	.0578221	-1.8	0	O addieren O ersetzen 7. Formoptimierung			
4	.057	-1.79707	0	8. Löschen			
5	.0587294	-1.79827	0	Achsen vertauschen			
6	.0597468 -1.6		0	X-Werte mit Y-Werte vertauschen			
7	.057 -1.75		0	X-Werte mit Z-Werte vertauschen			
8	.0597184	-1.79565	-1.79565 0 Y-Werte mit Z-Werte vertauschen				
9	.0607091	-1.79827	0				
10	.0616715	-1.8	0	Koordinaten mit Faktor verändern			
11	.057	-1.79076	0	X-Koordinaten VY-Koordinaten Z-Koordinaten			
12	.05897	-1.79286	0				
13	.0635961	-1.7978	0	nur die angezeigten Knoten im Knotenmodus verwenden			
14	.0636668	-1.79309	0	von Kostannunkt			
15	.0655208	-1.8	0				
16	.057	-1.78234	0	bis Knotenpunkt: 2232			
17	.0613593	-1.78865	0	Koordinaterfaktor 1000			
18	.0680915	-1.79581	0				
19	.0696342	-1.78717	0	Nullpunktsverschiebung durch Knotenpunkt: 1			
19	0696342	-1.78717 Koordinate	0 n-Faktor	Welformungen mit Faktor zu den Koordinaten addieren:			
				Veformungs-Faktor: 1 CANCEL			

Temperature Analysis

Save the FEM model under any name in the project directory and select the "FEM Analysis" tab to carry out a temperature analysis with the triangles TRI6S or CPS6.

	ht Netzgeneri	erung FEM-Projel	t bearbeiten	FEM-Analyse	Ergebnisau	swert
3. Tempera	tur 👻	FEM-Solver wählen	Infos zum FEI Modell-Abm	M-Modell essungen		
FEM-Analys	e 🖓	FEM-Ablauf 🕞	Infos Strukt	urmodell 🕞 🛛 FB	M-Assistent 🖓	
🖳 Quick-Solver				1	- 🗆	Х
Normal Precision	2D-Scheihe	n CPS6 or CPS8 (quadric	nlane stress eler	nent)		
				ineric)		
Sec. A.	O 3D-Schalen	1 56 or 58 (6- or 8-hode q	uadric shell eleme	nt)		
	C3D20 (20)	node quadric isoparametr	ic element)			
Path for INP-Solver	C:\Program Files\F	EM-System MEANS V12		r\inpsolver64bit e	Browser	ĩ
		()(()))				
Path for INP Files:	C: projekte venste	rprofil verim2.llvP				
	Select Solver	In-Core-Solver (Out-of-Core-So	lver		
		Start FEM-Solver with I	NP-Interface			
	Show a	Start FEM-Solver with I and edit last INP	NP-Interface Resolve last INF	,		
	Show a	Start FEM-Solver with I	NP-Interface Resolve last INF	· · · · · · · · · · · · · · · · · · ·		

Postprocessing

The temperatures and the heat flow density are displayed with the icon the "Postprocessing" tab.



loouro.	Load Case: 1					
Steady State Temperature						
Heat Flux Density	 Reaction Forces 					
 Element Stress Contour 	O Contour of Forces					
vccuracy:						
Edit Accuracy:	Displacement Factor					
_	Edit Colours for Legend					
1 3 4 8	Pick, Search, Save Values					
Select Result Component:						
Steady State Temperat	ture 🗸 🗸					

Steady State Temperature Distribution



Heat Flow Density

The heat flow density must be set to the value "20" W/m² using the "Deformation factor" menu be scaled, otherwise too high values will be seen.

	📍 🞑 🔍							
Da'	ei Ansicht Net	zgenerierung FE	M-Projekt bearbeiten	FEM-Analyse	Ergebnisauswertung	Training		
Erg	Ergebnisse auswerten	Knotenwerte pio Knotenwerte ed	itieren Legende Diagrami	3 * m 1 * Diagramme 17	Verformungsfaktor Value-Animation	FEM INP STA FRD Dateien listen Fa	Bemessungen für max 🔹 Bemessung/Nachweise 🕫	
🖳 Legend	e von C:\projekte\fenste	rprofil\femm2.ERD					2	– n >
Wärmest	romdichte gesamt							
				-		_		-
0.0001	2.857	2	5.7144	8.5715	11.429	14.2	86 17.143	3 20.000
	Verformungen da Verformungen da Verforme S Verforme S Verforme S Verforme V Drat ahne V O unverformt Verformungen mit Verformungen mit Ver	aktor rstellen Struktur alleine darstelle ind unverfomte Struktu rigitter ingitter i kant efomung: i i i i i i i i i i i i i i i i i i i	n rzusammen enmodell nt Verfomsung: llen darstellen darstellen ehen • + he hervorzuheben) Zurücksetzen Zurücksetzen zurücksetzen ehen • + i ter internet in					
	•	Cancel	ОК					