Part 6 - Temperature-Analysis with FEM-System MEANS V11

Chip-Assembly:

Element Group 1 **Board:** X = 100 mm; Y = 100 mm; Z = 1 mm; Λ = 1 W/m*K Element Group 2 **Leiter:** X = 100 mm; Y = 100 mm; Z = 0.1 mm; Λ = 30 W/m*K Element Group 3 **Chipkleber:** X = 0.5 mm;Y = 0.5 mm; Z = 0.1 mm; Λ = 40 W/m*K Element Group 4 **Chip:** X = 0.5 mm; Y = 0.5 mm; Z = 0.3 mm; Λ = 300 W/m*K



How big is the temperature distribution at a source of heat = 1.5W in Element Group 4 (Chip) when the board is cooled with a convection of $5.5W / m^2 * K$ and an ambient temperature of 20 ° C. Since the model is symmetrical, it is sufficient to meshes only a quarter.



3D-Model

Additional module TEMPERATURE

The additional module MEANS-TEMPERATURE contains a module for stationary (steady state) and unsteady (transient) temperature field calculation.

All finite elements used can process the following boundary conditions:

- Node Temperatures [K]
- Convection on all surfaces [Watt / (m² * K)]
- Point Source on Nodes [Watt]
- Heat Source on Surfaces [Watt / m²]
- Radiation with Emissions



For a Temperature-Analysis select the "Edit FEM-Project" tab and

0	101	() =										FEM-System MEA	NS V11	
U	Files	View	Mesh Genera	tion Edit FEM-Pr	oject	FEM-Analysis	Postpro	ocessing	Training					
F Loads	3. Surfa	ice Load Loads	* Bo	bundary-Conditions	1. Bo	undary-Condition: • w Boundary-Conditi	ions E	ilement-Grou	ups Mate	rial-Datas	Editor	6. Loads	•	Temperature
							Infozeile							la la

Node Temperatures (K)	Radiation on Surfaces with Emmision Load Case 9
Heat Sources	Setting Temperature
Point Source on Nodes Load Case 2 (Watt)	stationary transient Statics
Heat Source on Surfaces Load Case 7 (Watt/m³)	Enter Material Datas for transient Heat
Convection	Start Temperatures for transient Heat
Convection on Surfaces Load Case 8 (Watt/(m²K))	
Load Case 8 (Watt/(m²*K))	

Create 2D template with Line-Modus

First, create a 2D template for hexahedral extrusion by selecting the "View" tab and drop-down menu "3. Line Modus" and enter the following 4 nodes with the new side menu.

0	1 6	• 🙆 •									FEM-S
U	Files	View	Mesh	Generation	Edit FEM-Project	FEM-Ana	lysis	Postproces	sing Trai	ning	
Hidder	D-Line	 without with Me 	: Mesh esh	 O Wireframe ✓ Edges 	☑ Lighting Hidden-Line new	30	1. Mai Line Mo	n View dus is active	- 🗗	3. 1. 2. 3. 4. 5.	Line-Modus Surface-Modus Node-Modus Line-Modus Create Surface Model Switch Surfaces ON/OFF

Node 1:

Select "New" and enter node 1 with X = 0, Y = 0 and Z = 0 and select "Create Nodes".

Node 2:

Select "New" and enter as above node 2 with X = 0.25, Y = 0 and Z = 0

Node 3:

Select "New" and enter, as above, Node 3 with X = 0.25, Y = 0.25 and Z = 0

Node 4:

Select "New" and enter, as above, Node 4 with X = 0, Y = 0.25 and Z = 0



Select the menu "3D Mesh Grid" and enter Node 1, 2, 3, 4 as Edge 1,2, 3, 4, then define the Number of Nodes in X direction = 15 and in Y direction = 15 and select "Generate 3D-Mesh Grid".

Nodes of Edge 1:	1		
	[
Nodes of Edge 2:	2		
Nodes of Edge 3:	3		
O Nodes of Edge 4:	4		
Delete E	dge		
Number of Nodes in X di	irection: 15		
Number of Nodes in Y di	irection: 15		
Generate 3D-	Mesh Grid		

A FEM model of QUA4S-Elements und 225 Nodes will be generated.

2 7 0 🗹 -			FEM	System M	EANS V11	- FEM Str	ucture File	C:\Progr	am Files\	FEM-Syste	m_MEANS	S_V11\FEM	-Projects	New\net;	1.fem		0
Files View M	Mesh Generation Edit FEM-Project	FEM-Analysis Post	processin	g Tra	ning												
F3. Surface Load	- 1. Bo	oundary-Condition: *				C	1	6 Lord									
Loads Show Loads	Boundary-Conditions Sho	ow Boundary-Conditions	Element	t-Groups	Materia	al-Datas	Editor	01 2000.			Tempera	ture					
3= 2		Line Modus	is active									15					
																Surface Nodes	Lines
Z ON EG=1																Number of Node	m = 225
ON EG+2																Nodes	~
ON EG=3																from: 1	
ON EG=4																unti: 225	
ON EG=5																Show	lodes
		4	213	214	215	218	217	218	210	220	221	222	223	224	225	3 Create a Ran	ge of Nodee
C ON CORD		108	100	200	201	202	203	204	205	206	207	208	200	210	211	212 Delete Rang	e of Nodes
CON EG=7		183	184	185	198	197	188	189	190	191	192	193	194	195	198	197 Surface= 1	
edit colours click on to the frames																Surface	Noder
Draw hidden EGs and Surfaces as Wireframe		168	169	170	171	172	173	174	175	176	177	178	179	180	181	182	o Suface
Weight per EG		163	164	166	158	167	169	169	160	161	162	163	184	165	166	167 Contract	e-Factor
Groups 1 · 7		120	120	140	141	142	1.42	100	1.05	105	147	140	140	150	161	162 No.do: 1	
Create Element oroune							140			1		1	1		101	Node: 1	
and an and a state		123	124	125	120	127	128	120	130	131	132	133	134	135	135	137 Y: 0	
		108	109	110	111	112	113	114	115	110	117	118	119	120	121	122 Z: 0	
											100				100	☑ Node numb	ering
		85	94	80	90	97	**	~	100	101	102	105	104	105	105	Bement nur	bering
		78	79	80	81	82	60	84	65	86	87	88	89	90	91	92 Element gro	.ps
		63	64	65	88	67	60	69	70	71	72	73	74	75	78	77 Node-Size:	0877193
		48	~	00	01	02	03	04	00	00	57	08	09	00	01	02 gma	al
		33	34	36	36	37	38	39	40	41	42	43	44	45	40	ar Gut are	
		18	19	20	21	22	23	24	25	28	27	28	29	30	31	32	
		- 1	5	6	7	8	-9	-10	-11	-12	-13	14	15	-16	-17	72	
*																	
ZXX																	

First, switch on the Node-Modus with the "Nodes" tab and show all nodes with a Node numbering, then with "Lines" tab expand the mesh with two new nodes:

Node 226:

Select "New" and enter node 226 with X = 50, Y = 0 and Z = 0 and select "Create Nodes".

Node 227:

Select "New" and enter node 227 with X = 50, Y = 50 and Z = 0 as above



Select the menu "3D Mesh Grid" and enter Node 2, 226, 227, 3 as Edge 1,2, 3, 4

🛃 3D Mesh G	rid		220	×
	 Nodes of Edge 1: Nodes of Edge 2: Nodes of Edge 3: 	2		
	Nodes of Edge 4:	3		
	Number of Nodes in X di Number of Nodes in Y di	rection: 45		
	Generate 3D-	Mesh Grid		

then define the Number of Nodes in X direction = 45 and in Y direction = 15 and select "Generate 3D-Mesh Grid" and generate a FEM model with 812 QUA4S elements, 887 nodes and 2 element groups.

Select the "Edit FEM Project" and "Element Groups" tabs and give element group 2 the color "white" by clicking on the color frame.



Zoom and position the FEM model in the middle and check the nodes between element group 1 to 2 with "Hidden Line with mesh" and without lighting.



Node 888:

Select "New" and enter node 888 with X=0, Y=50 and select "Create Nodes".



Select "3D-Mesh Grid" and enter the Nodes 4, 3, 887, 888 as the Edges 1, 2, 3, 4 and define the Number of Nodes in X direction = 15 and in Y direction = 45 and and generate a mesh with 1428 QUA4S elements, 1501 nodes and 3 element groups.

🖳 3D Mesh G	rid			×
	• Nodes of Edge 1:	4	 	
	O Nodes of Edge 2:	3	 	
	O Nodes of Edge 3:	887	 	
	O Nodes of Edge 4:	888	 	
	Delete E	dge		
	Number of Nodes in X di	rection: 15		
	Number of Nodes in Y di	rection: 45		
	Generate 3D-	Mesh Grid		

Select "Create element groups" and change element group 3 to 2 and the number of element groups from 3 to 2.

Extrude a Hexahedral FEM-Model

The 2D quadrilateral mesh will now be used as a template for 3D extrusion. Select the "Mesh Generation" tab and the "QUAD Meshes, Refine, Delete" menu and the tab in the new "Extrude" dialog box.



and extrude with the following settings "Density in Z direction = 16" and with "Elevation in Z direction = 1.5 mm" so that every 0.1 mm a row of nodes in the Z direction is created.



Select "Create a 3D-Mesh" and a FEM-Model with 18700 HEX8 elements, 21618 nodes and 2 element groups is created.



Adjust the view above by rotating the z-axis with the right mouse button (only with DX9) and zooming the model with the mouse wheel. Set the Node Modus and create a range of nodes and check the Z coordinates of the 16 rows of nodes. Select the "Edit FEM Project" and "Element Groups" tab and click on the color frames to assign any color to the two element groups. You can also show and hide both element groups for practice later.

Generation of element group 3

Select the Element Group menu "Create Element Groups" and enter the Element Group 3 first and select "Calculate Dimensions" and enter a Z-Range from 0.0 to 0.105 to create the Element Group 3. Give the element group the color "red".

	10	() =				FEM-Syste	m MEANS V1	l - Strukturdatei C	:\Progran	n Files\FEM-System	_MEANS_V11\F
	Datei	Ansichten	Netzgenerierung	FEM-Projekt bearbeiten	FEM-Analyse	Ergebnisi	auswertung	Training			
Hidder	n-Line () ohne Netz) mit Netz	O Drahtgitter ☑ m ☑ Kanten Hidd	it Beleuchtung en-Line neu	1. Hauptansicht		2. Kno	ten-Modus 👻		 XYZ-Drehung XZY-Drehung 	
EG= 3				(action	inous unifiere i	idente i					
🛛 ом		EG= 1					🖳 EG = 3			- C	ı ×
ON 🗹		EG= 2					Elementgrup	open erzeugen			
		FG= 3					Elemento	gruppe erzeugen:	3		
							● EG m	t aufgespannten Re	chteck un	id einer Tiefe erzeuge	n
MON		El3=4					⊖ EG m	t allen angezeigten	Knoten erz	eugen	
⊡ on	9 0 2	EG=5					O Eleme	ntgruppe aus mehre	eren Fläche	en erzeugen	
✓ ON		EG=6					0	Z-Tiefe O Y	-Tiefe	🔿 X-Tiefe	
🗹 ON	8	EG=7					von:	-100000	bis	: 100000	
Für neue F	Farbe auf Fa	irbrahmen klicken						Erzeuge	e Elementa	nuppe	
🗹 au	sgeblende	te EGs als		/							
Ge	wicht für i	ede EG					EG mit eine	n definierten Bereich	n erzeugen	1	
								Abmessu	ungen bere	echnen	
Gruppe	an I-7	~					von X	-25	bis	X: 25	
Elen	nentgrupper	n erzeugen					von Y	-25	bis	Y: 25	
						_	von Z	0	bis	Z: 0.105	
								Element	gruppe erz	eugen	
								Elementarupo	en wieder	nickgängig	
							-			5.6.00	
							Elementgrup	open ändem			
							Flow ante	enigruppen andem:	3		
							Anzahl Elem	ente ändem:	16500		
									10000	Åndem	

Part 6 - Temperature-Analysis with FEM-System MEANS V11

Generation of element group 4

Select the element group menu "Create element groups" and enter the element group 4 first and select "Calculate dimensions" and enter a z-range from 0.1 to 1.105 to create the element group 4. Give the element group the color "green".

Date Anichten Netzgeneierung EM-Projekt bastelten FEM-Analyse Egebnisuuwertung Taining Image: Image		• 💽) =		FEM-Syster	m MEANS V11 - Strukturdatei C:\Program Files\FEM-System_MEANS_V11\FEM-Proj
Image: Bige of the Netz Dahigiter Hidden-Line Imit Belsuchtung Imit Belsuchtung <t< td=""><td>Datei</td><td>Ansichten</td><th>Netzgenerierung FEM-Projekt bearbeiten</th><td>FEM-Analyse Ergebnisa</td><td>uswertung Training</td></t<>	Datei	Ansichten	Netzgenerierung FEM-Projekt bearbeiten	FEM-Analyse Ergebnisa	uswertung Training
EGS-4. Image: Comparison of the compar	Hidden-Line	◯ ohne Netz ◉ mit Netz	O Drahtgitter ☑ mit Beleuchtung ☑ ☑ Kanten Hidden-Line neu	1. Hauptansicht - Elächen 1	2. Knoten-Modus
Image: Constraint of the second of the s	EG= 4	1.1		r modus aktivitit i fideric= 1	14
○ ON ● EG-2 ○ ON ● EG-3 ○ ON ● EG-3 ○ ON ● EG-5 ○ ON ● EG-5 ○ ON ● EG-5 ○ ON ● EG-7 Firtnese Fates of Futchmen Kicken ○ Bernertgruppe aus mehreren Rächen erzeugen ○ ausgebiendete EGs als Dahtgitzer dantien ○ Y-Tiefe ○ X-Tiefe ○ Gewicht für jede EGS ○ Y-Tiefe ○ X-Tiefe ○ Gruppen 1-7 ○ ● EG mt einem definieten Bereich erzeugen ● EG mt inem definieten Bereich erzeugen ● Bereitgruppe erzeugen ● EG mt inem definieten Bereich erzeugen ● Bereitgruppe erzeugen ● Bereitgruppe micken ● Context ● Sign tiefe on the Sign tiefe ○ Gruppen 1-7 ○ ● Sign tiefe ● X-Tiefe ○ Gruppen 1-7 ● ● Sign tiefe ● Sign tiefe ○ Gruppen 1-7 ● ● Sign tiefe ● Sign tiefe ○ Gruppen 1-7 ● ● ● ● ○ Gruppen 1-7 ● ● ● ● ○ Gruppen 1-7 ● ● ● ● ○ Sign tiefe ●	Ø ON	EG= 1			⊯ EG = 4 - □ ×
I ON EG=3 I ON EG=4 I ON EG=5 I ON EG=6 I ON EG=6 I ON EG=7 Fir neve Face auf Furchmen Kickee I augebiendere EGs als I Gewicht für jede EGS Gewicht für jede EGS Gewicht für jede EGS Guppen 1-7 I Elementgruppen erzeugen Abmessungen berechnen Von Y: 25 Dis X: 25 Dis Dis X: 26 27 Dis Dis X: 26 27 28 <td></td> <td>FG= 2</td> <th></th> <td></td> <td>Elementgruppen erzeugen</td>		FG= 2			Elementgruppen erzeugen
 ○ ON ■ EG=4 ○ ON ■ EG=5 ○ ON ■ EG=6 ○ ON ■ EG=7 For reve Fatte auf Fattrahmen kilden ○ ausgebiendete EGs als ○ Outputer dantelen ○ Gewicht für jede EG ○ Gruppen 1: 7 ··· ■ Etementgruppe mizzeugen Elementgruppe mizzeugen Elementgruppe mizzeugen Elementgruppe erzeugen Bementgruppe erzeugen Bementgruppe ausgebierdet rückgängig Elementgruppe mizzeugen 		EG= 3			Bementgruppe erzeugen: 4
 N ■ Lu-s N ■ EG-5 O N ■ EG-5 O N ■ EG-6 C I Treke Fate auf Fatorahmen Kloken G ausgebiendete EGs als Drahtgitter darstellen G ewicht für jede EG Gruppen 1.7 ▼ Elementgruppe erzeugen EG mit einem definierten Bereich erzeugen Abmeasungen berechnen von X: 25 bis X: 25 von X: 25 bis Y: 25 von Y: 25 bis Y: 25 von Z: 0.1 bis Z: 1.105 Elementgruppe erzeugen 		5G- 4			EG mit aufgespannten Rechteck und einer Tiefe erzeugen
Image: Normal Eg-5 Image: Seg-5 Image: Normal Eg-5 Image: Seg-7 Furneue Face auf Partnamen kiloken Image: Seg-7 Image: Seg-7 Im		Lury			O EG mit allen angezeigten Knoten erzeugen
○ ON EG=6 ○ ON EG=7 Für neue Fate auf Fatramen klicken	⊠ ON	EG=5			O Elementgruppe aus mehreren Flächen erzeugen
N EG=7 Fur neve Facts av/ Factnamen kilcken Busgebiendete EGs als Drighter danstelen Gruppen 1-7 Elementgruppen erzsugen Von: 10000 bis: 10000 EG mit einem definierten Bereich erzeugen Abmessungen berechnen von X: 25 bis X: 25 von Y: 25 bis Y: 25 von Y: 25 bis Y: 25 von Y: 25 bis Z: 1105 Elementgruppen wieder rückgängig Bementgruppen wieder rückgängig Bementgruppen andem Attraction provider rückgängig Elementgruppen ändem Attraction provider rückgängig Bementgruppen ändem Attraction provider rückgängig	☑ ON	EG=6			◯ Z-Tiefe ◯ Y-Tiefe ◯ X-Tiefe
Für neue Patre auf Patre	I ON	EG=7			von: -100000 bis: 100000
^A susgeblendete EGs als Drahtgiter dantellen ^C Crouge Zumon gruppe ^G Gruppen 1 - 7 ^C	Für neue Farbe auf F	arbrahmen klicken			Erzeuge Elementarippe
Uparigner darstelen Gewicht für jede EG Gruppen 1 - 7 Image: Comparison of the state of the	ausgeblend	ete EGs als			
Gruppen 1-7 Elementgruppen erzeugen Abmessungen berechnen von X: 25 bis X: 25 von Y: 25 bis Y: 25 von Z: 0.1 bis Z: 1.105 Bementgruppen wieder rückgängig Bementgruppen wieder rückgängig Bementgruppen andem Azzahl Elementgruppen ändem	Gewicht für	iede FG			EG mit einem definierten Bereich erzeugen
Grappen 1: 7 von X: -25 bis X: 25 Biementgruppen erzeugen von X: -25 bis X: 25 von Z: 0.1 bis Z: 1.105 Bementgruppen wieder nückgängig Bementgruppen ändem Anzehl Eenertgruppen ändem Anzehl Eenertgruppen ändem					Abmessungen berechnen
Extended uppen exactlyer von Y: 25 bis Y: 25 von Z: 0.1 bis Z: 1.105 Elementgruppen wieder rückgängig Bementgruppen ändem Azzahl Elementgruppen ändem	Gruppen T - 7	~			von X: -25 bis X: 25
von Z: 0.1 bis Z: 1.105 Bementgruppe erzeugen Bementgruppen wieder rückgängig Bementgruppen ändem Azzehl Bemertgruppen ändem	Elementgrupp	en eizeugen			von Y: -25 bis Y: 25
Bementgruppe erzeugen Bementgruppen wieder rückgängig Bementgruppen ändem Anzehl Bementgruppen ändem					von Z: 0.1 bis Z: 1.105
Elementgruppen wieder rückgängig Elementgruppen ändem Anzehl Elementgruppen ändem					Elementgruppe erzeugen
Elementgruppen ändem					Elementgruppen wieder rückgängig
Anzahl Elementoningen ändern: 4					Elementgruppen ändem
Ander Contrargopper ander. 4					Anzahl Elementgruppen ändem: 4
Bementgruppe ändem von: auf:					Elementgruppe ändem von: auf:
Anzahl Elemente ändem: 16500					Anzahl Elemente ändem: 16500 Ändem

Part 6 - Temperature-Analysis with FEM-System MEANS V11

Generation of element group 5

It remains the element group for the Chipkleber, show element group 2 and hide all the others 1, 3 and 4 and enter the element group 5 and select again "Calculate dimensions" and enter a Z-range from 1.1 to 1.205 to create the element group 5. Give the element group the color "gray" and show all element groups again.



Part 6 - Temperature-Analysis with FEM-System MEANS V11

Delete element group 1

Finally, element group 1 is left which still has to be deleted. Select the "Mesh Generation" tab and "QUAD Meshes, Refine ..." and delete the Element Group 1 with the next dialog box.

Alternatively, you can hide the element group 2 and 5 and delete the element group 1 with a defined range of Z = 1.1 to Z = 1.5.

Then a model check is performed automatically and the hidden line is recreated.



Change element groups

Select "Create Element Group" and change the element groups as follows:

Element group 4 -> Element group 1 Element group 2 -> element group 4 Element group 3 -> Element group 2 Element group 5 -> Element group 4

Finally, enter the number of element groups = 4.

The model is now ready and can be saved under the name "Chip-Simulation.fem. Then you have to enter the heat conductivities, the convections and the point source.



Create Surface Model

Select the "View" tab and "Create Surface Model" to create a surface model with 9 surfaces, the surface turn in red as soon as you move the mouse over the surface.



Create Convection

Ö

Select the "Edit FEM project" tab and mean and choose in the next dialogbox menu "Convection on Surfaces Load Case 8 (Watt/(m²K)" and enter the Value of Convection of 5.5 W/(m²K) and the Temperature of 20 Grad C on the top side of the board with surface 3 and on the underside with surface 2.

	Node Temperature (I/)	Radiation on Surfaces with
	Node Temperatures (K)	Emmision Load Case 9
leat Sour	ces	Setting Temperature
	Point Source on Nodes Load Case 2 (Watt)	● stationary ○ transient ○ Statics
	Heat Source on Surfaces Load Case 7 (Watt/m³)	Enter Material Datas for transient Heat
onvectio	n	Start Temperatures for transient Heat
	Convection on Surfaces	
Convectio	n Convection on Surfaces	Start Temperatures for transient Heat

Select "Create Convection" and double-click on Surface 3 and Surface 2.

Create Convect	ion			222		×
Actual Load Case	: 1					
Number of Loads:	0		New			
Value of Convecti	on: 5.5		0	W/m²*K)		
Temperature:	20		(Grad C)		
Degrees of freedo	m					
	O in X Direc	tion	🔘 in Z	Directio	n	
) in Y Direc	tion	Vert	tical to S	iurface	
(Achsen-Farben: SCI	O in Y Direc	tion e; BLAU	Vert Y-Achse;	tical to S ROT: Z-A	iuiface .chse)	
(Achsen-Farben: SC) Selection:	O in Y Direc	tion a; BLAU	Vert Y-Achse;	tical to S ROT: Z-A	iurface .chse)	
(Achsen-Farben: SCH Selection: Select Surface	O in Y Direc HWARZ: X-Achse s	tion ; BLAU	Vert Y-Achse; Dragging	tical to S ROT: Z-A a model	iurface .chse) region	
(Achsen-Farben: SCH Selection: Select Surface Select Nodes	O in Y Direc IWARZ: X-Achse s	tion BLAU	Vert Y-Achse; Dragging - Select all	tical to S ROT: Z-A a model showing	iurface chse) region Nodes	
(Achsen-Farben: SCH Selection: Select Surface Select Nodes Define a coord) in Y Direc HWARZ: X-Achse s inate range	tion a; BLAU	Vert Y-Achse; Dragging Select all s	iical to S ROT: Z-A a model showing	iurface ichse) region Nodes	
(Achsen-Farben: SCH Selection: Select Surface Select Nodes Define a coord Cancel) in Y Direc HWARZ: X-Achse s inate range Editor	tion a; BLAU 0 0	Vert V-Achse; Dragging Select all Creat	iical to S ROT: Z-A a model showing t e Com	iuface .chse) region Nodes	

The surfaces are listed in the selectbox, there select "Create" to generate the convection.



Create a Point Source

Hide the convection with "Edit FEM Project" tab and "Show Loads". Create with the "View" tab and "Node-Modus" a Range of Nodes of the Chip Surface 7. If the nodes are displayed too large, they can be zoomed down with node size and "small".

Select the "Edit FEM project" tab and shows and choose in the next dialogbox menu "Point Source on Nodes Load Case 2 (Watt)" and enter the Load Case 2 and enter the value of point source 1.5.

Than select "Create a Point Source" and double-click on Node 10672. The Node is listed in the selectbox, there select "Create" to generate the point source.



Enter Heat Conductivities

Select "Edit FEM-Project" tab and "Material-Datas" and enter the heat conductivities for the element groups 1 - 4:

k 1 =	1 W / m*K	(Board)
1 ₂ =	30 W / m*K	(Leiter)
Аз =	40 W / m*K	(Chipkleber)
1 4 =	300 W / m*K	(Chip)

C	1 0	• 🚺 =						FEM System MEA
0	Files	View	Mesh Generation	Edit	EM-Project	FEM-Analysis	Postprocessing	Training
F.	3. Sui ds Sho	rface Load w Loads	* Bounda	▶ → ıry-Condi	1. Bo tions ☑ Sho	undary-Condition w Boundary-Con Surface Mo	ditions Element-	Groups Material-Datas
	Edit Materia	al Datas		<i></i>		(
	Name		Material Datas			A8		
+	Youngs r	nodulus	210000					
	Poisson I	Ratio	.3					
	Density		0					
	WK		1					
	WL		300					
	CV		0					
	RV		0					
	QD		0					
	DAMP		0					
*								
E	lement Group O Isotroj Materia	p: 4 p al Data Bas	Element HEX2(Tempera) tur O	< >>			
	Materialo	laten kopie	ren					

Change Nodal Coordinates

Finally, the coordinates have to be changed from millimeters to meters by dividing the coordinates by a coordinate factor of 1000. Select "Edit FEM Project" tab with menu "3. Nodal Coordinates" and select in the next dialogbox "Coordinate-Factor" and set "Divide" and enter a factor of "1000" and select "Change Coordinates with a Factor".



FEM-Analysis

Select "FEM-Analysis" tab and "Quick-Solver" to start the FEM-Solver.

	les	View	Mesh Generatio	n Edit FEM-Projec	t FEM-Analysis	Postprocessing	Training
	3. 1	Temperati	ure 🔻 (Choose FEM-Solver Choose Results	Structure Model Model Dimensions		
	FEM	l-Analysis	Γ ₂	Select FEM 🕞	Structure Info 🕞	FEM-Assistent	ž.
		🖳 Finit	e Element Analysis		- 0	×	
		C:\p	rojekte\chip-simulatio	on \chip 1b.fem			
		Sel	ect Solver O ME	ANS-Solver 🔘 🤇	Quick-Solver		
				Step 1: Starting FEM Anal	ysis		
			9	itep 2: Starting Postproce	ssing		
/			Step 3:	Refinement of the Eleme	nt Stresses		
	$\overline{\ }$	-				_	
			Select FEM Solv	are	D.C. D. H.		
			Deleter Elit Delv	013	Define Results		
			SCICCUTEN SON		Define Results	_	
			Select En Selv	Cancel	Define Results		
				Cancel			
INP-	Interfa	ace for FE	Solvers	Cancel	Define Results		- ×
INP-I	Interfa	ice for FE	Solvers	Cancel D8 (6-/8-node linear isopa	arametric element)		□ ×
INP-	Interfa gh Pres	cision	Solvers O C3D6/C3 O show C3E	Cancel D8 (6-/8-node linear isopa	arametric element)	- 1	- ×
INP-	Interfa	cision	Solvers C3D6/C3 Solvers Solvers C3D6/C3 Solvers C3D15/C	Cancel D8 (6-/8-node linear isopa)4 and solve intern with a 3D20 (15-/20-node quadr	arametric element) refining mesh of 8 x C3D4 ic isoparametric element)	- 1 	×
INP-	Interfa gh Pres	cision	Solvers C C3D6/C3 Show C3E C3D15/C	Cancel D8 (6-/8-node linear isopa)4 and solve intern with a 3D20 (15-/20-node quadr	arametric element) refining mesh of 8 x C3D4 ic isoparametric element)		×
INP-I	Interfa Jh Pre d	cision	Solvers C C3D6/C3 Show C3E C C3D15/C C:\Program Files	Cancel D8 (6-/8-node linear isopa)4 and solve intern with a 3D20 (15-/20-node quadr FEM-System_MEANS_V	arametric element) refining mesh of 8 x C3D4 ic isoparametric element) 11\Debug\inpsolver\inpsc	- [- [-] -] -] -] -] -] -] -] -] -]	X X X X X X X X X X X X X X X X X X X
INP-	Interfa Jh Pred	cision -Solver: Files:	Solvers C C3D6/C3 Solvers C:\Program Files C:\projekte\chip-	Cancel D8 (6-/8-node linear isopa)4 and solve intern with a 3D20 (15-/20-node quadr .FEM-System_MEANS_V simulation \chip 1b.INP	arametric element) refining mesh of 8 x C3D4 ic isoparametric element) 11\Debug\inpsolver\inpsc	olver64bit.e	x x x x x x x x x x x x x x x x x x x
INP-	Interfa Jh Pre for INP	cision •Solver: Files:	Solvers C C3D6/C3 Show C3E C:\Program Files\ C:\projekte\chip- Select Solver	Cancel Ca	arametric element) refining mesh of 8 x C3D4 ic isoparametric element) 11\Debug\inpsolver\inpso	- [- [x x x x x x x x x x
INP-	Interfa Jh Pre	-Solver: Files:	Solvers C C3D6/C3 Solvers C C3D6/C3 Solver C:\Program Files C:\projekte \chip- Select Solver	Cancel Ca	arametric element) refining mesh of 8 x C3D4 ic isoparametric element) 11\Debug\inpsolver\inpso	olver64bit.e	X X X X X X X X X X X X X X X X X X X
INP- Hig Path f Path f	Interfa Jh Pre d	-Solver: Files:	Solvers C C3D6/C3 Solvers C C3D6/C3 Solver C3E C:\Program Files\ C:\projekte\chip- Select Solver	Cancel D8 (6-/8-node linear isopa 04 and solve intern with a 3D20 (15-/20-node quadr FEM-System_MEANS_V simulation\chip1b.INP (In-Core-Solver Start FEM-Solver with	arametric element) refining mesh of 8 x C3D4 ic isoparametric element) 11\Debug\inpsolver\inpsc O Out-of-Core-Solver	olver64bit.e	X X X X X X X X X X X X X X X X X X X
INP-	Interfa Jh Pred	-Solver: Files:	Solvers C3D6/C3 Solvers C:\Program Files\ C:\projekte\chip Select Solver	Cancel D8 (6-/8-node linear isopa)4 and solve intern with a 3D20 (15-/20-node quadr FEM-System_MEANS_V simulation\chip1b.INP () In-Core-Solver Start FEM-Solver with	arametric element) refining mesh of 8 x C3D4 ic isoparametric element) 11\Debug\inpsolver\inpsc O Out-of-Core-Solver INP-Interface	olver64bit.e	X X X X X X X X X X X X X X X X X X X

Select "C3D15/C3D20 (15-/20-node quadratic....) to calculate the node temperature with HEX20-Solid elements.

Postprocessing

Select "Postprocessing" and the Icon



to evaluate the node temperature.

C 1 0 0 F				FEN		
🧡 Files View N	1esh Generation Edit F	EM-Project FEM-Analysis	Postprocessir	g Training		
Postprocessing	Displacement-Factor Pick, Search Values Settings 12	List Result Values Legende: Legende 1 Legende/Tabelle	Intern Extern Animations 🛱	List FEM-File List STA-File List Files		
	🖷 Postprocessing		- 0	×		
	Results Load Case 1 Image: Steady State Temperature Image: Steady State Temperature Image: Steady State Temperature Image: Heat Rux Density Image: Steady State Temperature Image: Steady State Temperature Image: Heat Rux Density Image: Steady State Temperature Image: Steady State Temperature Image: Heat Rux Density Image: Steady State Temperature Image: Steady State Temperature Image: Heat Rux Density Image: Steady State Temperature Image: Steady State Temperature Image: Heat Rux Density Image: Steady State Temperature Image: Steady State Temperature Image: Heat Rux Density Image: Steady State Temperature Image: Steady State Temperature Image: Heat Rux Density Image: Steady State Temperature Image: Steady State Temperature Image: Heat Rux Density Image: Steady State Temperature Image: Steady State Temperature Image: Heat Rux Density Image: Steady State Temperature Image: Steady State Temperature Image: Heat Rux Density Image: Steady State Temperature Image: Steady State Temperature Image: Heat Rux Density Image: Steady State Temperature Image: Steady State Temperature Image: Heat Rux Density Image: Steady State Temperature Image: Steady State Temperature Image: Heat Rux Density Image: Steady State Temperature Image: Steady State					
	Accuracy Edit Accuracy:	Displace	ment Factor			
	1 3	4 Pick, Searc	h, Save Values			
	Select Result Componen Steady State	it Temperature	~			
	Cancel	Start Postproces	ssing]		

