Part 1: FEM-System MEANS V12 with a Ribbon-Interface

Hardware requirements for MEANS V12

- Windows 10 with 64-Bit is preloaded
- DirectX11 for Windows 10 is installed
- Working memory with 16 GB
- Hard disk with over 500 GB and more

Licence agreement

Before MEANS can be installed, the licence arrangements must be accepted by HTA software. If you do not agree with it, you may not use MEANS V12 as a full version install only as a limited demo version on your PC.



Installation in different directories

If the installation is started normally, MEANS V12 will be in the current directory

C: \ Programs \ FEM-System MEANS V12

installed. However, MEANS must always be started there with "Administrator Rights".

However, if this directory is write-protected, it can also be installed in the non-write-protected User directory using "Browse ...":

C: \ Users \ FEM-System MEANS V12

However, the system-relevant DirectX11 must also be installed in the write-protected program directory.



Check DirectX11 installation

In Run, enter the command "DXDIAG" to check the DirectX11. A prerequisite for DirectX11 is that the current DirectX11-capable graphics card driver is installed.

Jispidy	Sound	Input	36				
reports	detailed	informat	tion about the Direct	X components and dri	ivers installe	d on your system.	
ow wha I page ir	t area is i sequen	causing ce.	the problem, click th	e appropriate tab abo	ove. Otherv	vise, you can use the "Next Page" b	utton below to
Informa	ation						
			Current Date/Time:	Mittwoch, 3. Januar	2018, 15:33	: 16	
			Computer Name:	USER-PC2			
			Operating System:	Windows 7 Profession	nal 64-bit (6	. 1, Build 7601)	
			Language:	Deutsch (Regional Se	tting: Deuts	sch)	
		Sys	stem Manufacturer:	MEDIONPC			
			System Model:	MS-7255			
			BIOS:	Phoenix - AwardBIOS	S v6.00PG		
			Processor:	Intel(R) Core(TM)2 C	PU 43	00 @ 1.80GHz (2 CPUs), ~1.8GHz	
			Memory:	4096MB RAM			
			Page file:	1205MB used, 1899M	1B available		
			DirectX Version:	DirectX 11			
ck for V	VHQL dig	ital signa	atures				
		D	xDiag 6.01.7601.17	514 32-bit Unicode Co	opyright ©	1998-2006 Microsoft Corporation.	All rights reserved.
	reports ow wha page ir Informa	reports detailed ow what area is page in sequen Information Information	reports detailed informat ow what area is causing page in sequence. Information Sys ck for WHQL digital signa	reports detailed information about the Direct ow what area is causing the problem, dick th page in sequence. Information Current Date/Time: Computer Name: Operating System: Language: System Manufacturer: System Model: BIOS: Processor: Memory: Page file: DirectX Version: sck for WHQL digital signatures	reports detailed information about the DirectX components and dri ow what area is causing the problem, click the appropriate tab about page in sequence. Information Current Date/Time: Mittwoch, 3. Januar Computer Name: USER-PC2 Operating System: Windows 7 Profession Language: Deutsch (Regional Se System Manufacturer: MEDIONPC System Model: MS-7255 BIOS: Phoenix - AwardBIOS Processor: Intel(R) Core(TM)2 C Memory: 4096MB RAM Page file: 1205MB used, 1899N DirectX Version: DirectX 11 sck for WHQL digital signatures DxDiag 6.01.7601.17514 32-bit Unicode C	reports detailed information about the DirectX components and drivers installe ow what area is causing the problem, dick the appropriate tab above. Otherv page in sequence. Information Current Date/Time: Mittwoch, 3. Januar 2018, 15:33 Computer Name: USER-PC2 Operating System: Windows 7 Professional 64-bit (6 Language: Deutsch (Regional Setting: Deuts System Manufacturer: MEDIONPC System Model: MS-7255 BIOS: Phoenix - AwardBIOS v6.00PG Processor: Intel(R) Core(TM)2 CPU 43 Memory: 4096MB RAM Page file: 1205MB used, 1899MB available DirectX Version: DirectX 11 sck for WHQL digital signatures DxDiag 6.01.7601.17514 32-bit Unicode Copyright ©	reports detailed information about the DirectX components and drivers installed on your system. ow what area is causing the problem, dick the appropriate tab above. Otherwise, you can use the "Next Page" b page in sequence. Information Current Date/Time: Mittwoch, 3. Januar 2018, 15:33:16 Computer Name: USER-PC2 Operating System: Windows 7 Professional 64-bit (6. 1, Build 7601) Language: Deutsch (Regional Setting: Deutsch) System Manufacturer: MEDIONPC System Model: MS-7255 BIOS: Phoenix - AwardBIOS v6.00PG Processor: Intel(R) Core(TM)2 CPU 4300 @ 1.80GHz (2 CPUs), ~1.8GHz Memory: 4096MB RAM Page file: 1205MB used, 1899MB available DirectX Version: DirectX 11 cdc for WHQL digital signatures DxDiag 6.01.7601.17514 32-bit Unicode Copyright © 1998-2006 Microsoft Corporation. /

Ribbon user interface of MEANS V12

The new ribbon interface of MEANS V12 consists of a ribbon or Ribbon that links the menu control, toolbars, and dialogs elements. Instead of using a menu item, you now call up the complete ribbon via a tab, for example "File".

Thus, the new MEANS V12 user interface is much more clearly arranged and faster to operate than the old interface of MEANS V10 with the many small icon bars and dialog boxes.



Above, the FEM model of the Ball Valve with Hidden-Line in the Surface-Modus with the surface load and the boundary conditions are shown in different colors.

MEANS V12 has the following 7 Ribbon Tabs:

- File
- Views
- Mesh Generation
- Edit FEM Project
- FEM-Analysis
- Postprocessing
- Training

Files

	? [€] ≠ es View	Mesh Ge	neration Edit	FEM-Project	FEM-Analysis	Postprocessing Training		
		7	FEM-Merge MPC-Contact	Import: STL +		1. C:\projekte\kugelventil\kugelventil1.fem		 German English
New 🕞	Open 🕞	Save 🕞	Unit	CAD	Path 🕞	Last opened FEM-Projects	Fa.	Language 🕞

View

C	Piles	View	Mesh Generation	Edit FEM-Proje	ect FEM	-Analysis	Postprocess	ing	Training			
● Re ○ wi	ndering th Mesh	O Wirefr with Edge	ame Light 10% s • Hidden-Line	e new View	3D Preview Z	.oom 1.	Main View	•	Surfaces	Nodes	5. Switch Surfaces C +	Background
					Sur	face Modu	is is active					Ta:

Mesh Generation

C	0 ? 🖸 🔻							
U	Files View	Mesh Generation	Edit FEM-Project	FEM-Analysis Po	stproc	essing Training		
	3D Mesh Gen Local Refinen	erators nent	QU	AD-Meshes, Refine, Delet	e	Check Node Numbering Jacobi-Determinante	FEMM Mesh Vessel Generator	Bearing Generator Screw Generator
	3D Mesh Generation	n 🗔 2D Mes	h Generation 🕞	Manipulate Meshes	۲ <u>،</u>	Check Meshes r	😨 Special Me	sh Generators 🛛 🖓

Edit FEM-Project

6		0 1	() =										
4	2	Files	View	Mesh Gen	eration Edit FEM-F	Project	FEM-Analysis F	Postprocessing	Training	I.			
	F_	3. Su	rface Load	+	\succ	1. Bo	undary-Condition: +			<u>J</u>	1	6 Loads	Ö
l	oads	Sho	w Loads		Boundary-Conditions	Sho	w Boundary-Condition	15 Element-Gro	ups M	laterial-Datas	Editor	U. LUGUS	Temperature
							Surface M	lodus is active					100

FEM-Analysis

0	7 💟 📼					
Files	View	Mesh Generat	ion Edit FEM	-Project	FEM-Analysis	Postprocessing
	1. Statics		Select FEM-Solv	er St M	ructure Model odel Dimensions	
	FEM-Analysis	Г¥	Select FEM	- Fa	Structure Info 🛛 🖓	FEM-Assistent 🕞

Postprocessing

0	0 1	ء 🔘										
U	Files	View	Mesh Generation	Edit FEN	1-Project	FEM-A	nalysis	Postproc	essing	Training		
		our Docult	Displacement-Fact	tor	Legend 1	+	DXF-Pos	stprocessin	9	List FEM-File	EVM Pichtlinia forma	121
U	[™] a	IOW RESULT	Pick, Search Value	s	Diagram 1	*	Value-A	nimation	-	List STA-File	PRIM-Richame for we	9 Y
	Postproce	ssing	Factor/Values	Fa.	Legend/Diag	gram 🖓	Ani	mations	T ₂	List Files 🖓	Fatique-Analysis	E.

Training

0	0 1	() =						
9	Files	View	Mesh Generation	Edit FEN	M-Project	FEM-Analysis	Postprocessing	Training
1. Ri	bbon-Inte	rface 👻	1. Ball Valve	-	~			son
	Tutorials	5 lī	FEM-Projects	rs.	Video 100t	Press 🖓 🛛 Forn	n Optimization 🕞	MPC-Structure

Mesh Generation

Based on the simple FEM example Ball Valve, the Ribbon Interface is now described.



Select the "Mesh Generation" tab and "New" to create a new FEM project.

ROJECT			×
③ 3D Mesh Generator NETG	SEN (STEP,	IGES, ST	L)
O 3D Mesh Generator GMSH	H (STEP)		
○ 2D/3D Beam Model with L	line-Modus		
O 3D Shel Model with a Cont	tainer Mesh	Genenera	tor
○ 2D Plane Model with a Be	aring Mesh (Generator	
○ 2D axialsym. Model with a	Screw Mesh	n Generati	or
NEW PRO	JECT		

Select menu "3D Mesh Generator NETGEN (STEP, IGES, STL)" to appear

a dialog box, with following CAD formats:

STL consists of a triangular outer shell for 3D mesh generation

STEP consists of solid elements and is the most suitable 3D format

IGES like STEP format but not so common anymore

as well as other CAD formats:

DXF is now in the Line mode for reading circles, lines or polygons3DS can converted with the external tool MEANS-SHELL or with the Visual Basic CAD Viewer 3DVBVIEW.

Select the STL file "ball valve.stl" with "Browser" and click on the button "Start Mesh generator No. 2 with CAD File" to display it in the mesh generator.

Hesh Generation			<u></u>)		×
)irectory: C:\Program	n Files \FEM-System_MEANS	_V11\CAD\STL\		Browser	
⊖ STEP		STL / AST (ASCII)		Default	
100 tons heave metal	press.STL				
eccentric bolt.stl eaf spring.stl nachine casing.stl sheet metal.STL					
STL Optimization and	l Repair ile before starting Mesh Gene ation with all tests	eration			
STL Optimization and Optimize STL F Optimiza Mesh Generator No.	I Repair ile before starting Mesh Gen tion with all tests 2 with OpenGL-Interface	eration			
STL Optimization and Optimize STL F Optimiza Mesh Generator No. Start Mesh Gene	I Repair ile before starting Mesh Gen ation with all tests 2 with OpenGL-Interface erator No.2 with CAD File	eration with error messages		Help	
STL Optimization and Optimize STL F Optimize Mesh Generator No. Start Mesh Gene Mesh Generator No.	I Repair ile before starting Mesh Gene ation with all tests 2 with OpenGL-Interface rator No.2 with CAD File 3 with automatic repair funct	eration with error messages		Help	

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



Select the menu "Mesh" and "Meshing Options" and generate with the mesh density "Very fine" and the main menu "Generate Mesh" a FEM Model with tetrahedral elements.

74 Meshing Options		—		×
General Mesh Size STL Charts	<u>O</u> ptimizer <u> </u>	nsider	Debug	1
Mesh granularity :	moderate			
First Step :	very coar	se		
Last Step :	coarse moderate	e	_	
Print Messages :	fine verv fine			
🔽 Parallel me	user defi	ned		
C Second or	der elements		57 	
🗖 Quad d	ominated			
🗌 Invert volu	me elements			
🗌 Invert surfa	ace elements			
🗌 Automatic	Z-refinement			
Eleme	ent order: 1	•		

The generated FEM Mesh now consists of 18106 nodes and 83911 tetrahedral elements.



Coarsing

With the option mesh size "STL / IGES / STEP - close edges" and the following setting, "coarse" tetrahedral meshes can be generated, for example, to be able to reduce the number of elements for MEANS-LITE or is often use for complex or thin Structures is the only setting to obtain a useable FEM Mesh.



With the above setting, the generated FEM Mesh now only consists of 6 610 nodes and 29 249 elements.



.

After Generation, the FEM Mesh named "test.fem" must be exported to MEANS V12. Select the "File" and "Export Mesh" menu and save the mesh "test.fem" into the default Debug / Mesh directory.

the set of		iemeni	
Load Geometry <i><g> Save Geometry Recent Files</g></i>	,		
Load Mesh <i><m> Recent Meshes Save Mesh <s><m> Merge Mesh</m></s></m></i>			export here with the
Import Mesh Export Mesh			name "test.fem" into th debug / mesh directory

After exporting "test.fem", MEANS V12 will be started automatically and will first create the surface model so that surfaces, edges and nodes for loads, boundary conditions or element groups can be selected.

O very fine O very much
Help
C model region

Show and hide surfaces

For the pressure load on the inner tube, first the outer surfaces 3, 4 and 8 must be hidden. Select the "View" tab and the drop-down menu "5. Switch Surfaces ON/OFF" to hide these surfaces by clicking on the "ON / OFF" column.



Create Load Case 1

Select the "Edit FEM Project" tab and the drop-down menu "3. Surface load " to enter load case 1 with a pressure load of 5 bar loaded on the inner tube with Surface 2.



You can enter 6 different load types:

- Load type 1: node loads for all element types Value input e.g. 10 000 N or 10 kN
- Load type 2: Line loads for all element types Value input e.g. 1000 N / mm or 1 N / m
- Load type 3: Surface or pressure load for plates, shells and volume elements Value input in N / mm² or N / m² (1 bar -> 0.1 N / mm²)
- Load type 4: Temperature load for all element types Value input node temperature in degrees Celsius and Heat transfer coefficients in material data
- Load type 5: Centrifugal load for all element types Value input in 1/s or 1/min and density in material data
- Load type 6: Gravity load for all element types Value input e.g. 9.81 m / s² and density in material data

In the next dialog box, enter Load case 1 with the value 0.5 N / mm² (= 5 bar) and with the degree of freedom "Vertical to Surface" and with the Selection "Select Surfaces" and click on the button "Create Surface Loads".

🖳 Create Surfac	Create Surface Loads				×
Actual Load Car Number of Load Value of Load	se 1 s 0 .5 ntlang Z-Achse W2=	New (for	example	e: MPA)	
Degrees of freedom: (Colour of Axis: BL/	 in X Direction in Y Direction ACK: X-Axis; BLUE: Y-Ax 	◯ in Z ● Verti Is; RED: Z-Axis	Directio cal to S	in iurface	
Selection:					
Select Surfa	aces	 Dragging 	a mode	el region	
O Select Node	es	 Select all 	showin	ig nodes	
O Define a co	ordinate range	Select all	showin	ig suiface	S
Show the Loads:	.00036	nom	al	~	
Cancel	Editor	Create S	Surface	e Loads	
		Dele	ete Lo	ads	

With a double-click on Surface 2 and choose menu "CREATE" in the Selectbox, the Surface Load is created.



Create Load Case 2

For the point load, a selectable Range of Nodes must first be created. To do this, switch in the "View" tab from the drop-down-menu "1. Surface-Modus" to the "2. Node-Modus".

FEM System MEANS V11 - FEM Structure File C:\projekte\Ball_Valve\ballvalve.fem



In the right menu field of Node-Modus, enter Surface = 19 and select "Edge Node Surface" to display the edge nodes of Surface 19 with the Node-Size = 0.142



Select the "Edit FEM Project" tab and the "1. Point Load" drop-down menu to create Load Case 2 with a point load in Z-direction.

In the next dialog box, enter Load Case 2 with the value -1500 N and with the degree of freedom "in Z direction" and with the Selection "Select all showing nodes" and select "Create Point Load" to create a point load with 33 nodes.

|--|

				FEM Syster	m MEANS V	/11 - FEM Structure File C:\projekte\Ball_Valve\ballvalve.fem
Files View	Mesh Generation	Edit FEM-Project FEI	/I-Analysis	Postprocessing	Trainin	g
without Hidden-Line O with Me	Mesh 🔿 Wireframe Ish 🗹 Edges	✓ Lighting Hidden-Line new	1 . M	lain View 👻	Ø	2. Node-Modus
		Su	face Modus	is active - Surface=	23	Γ ₂
	₩₩ A A N V E D fr fr	Create Point Loads Ctual Load Case 2 umber of Loads 33 alue of Load -1500 ungleich entlang Z-Achse 1 egrees of In X Dire eedom: In Y Dire blour of Axis: BLACK: X-Axis; BLUE lection	Ne V2= V ction © ction	w (for example: N) in Z Direction Z-Ax(s)	×	
) Select Surfaces) Select Nodes) Define a coordinate range	◯ Dra ● Sel ◯ Sel	gging a model region ect all showing nodes ect all showing surface	9	
	Sh	ow the Loads: .0003	e]	nomal ~		
		Cancel Editor	Cr	eate Point Loads		
				Delete Loads		
rt-x				U	5	

Set load cases

With the dropdown menu "7. Select load case ", load case 1 and load case 2 can be set. Also, the loads can be edited with menu "8.Editor" to delete the loads, copy them or change them with a load case factor.

F	7	Edit Load Case		1 Br	oundary-Condition	-		
Loads	1.	Point Load	Boundan	-Conditions Sho	ow Boundary-Cond	litions	Element-	Groups
	2.	Line Load			Surface Mor	dus is ac	tive - Surfa	ce= 9
	3.	Surface Load						
	4.	Centrifugal Load				~		
	5.	Gravitation Load						
	6.	Temperature Load	-					
	7.	Edit Load Case		Edit Load Case		9 — 6		×
	8.	Editor						
				Actual Load Case	. 🔽 🗸		>	
				A State Load Gab]		
				Factor:				
				(e) mult	iply C) divide		
				🔘 add) replace		
				CAN	CEL	ОК		

Create Boundary Conditions

To clamp the model, select the "Edit FEM Project" tab and click on "boundary conditions".

	FEM System MEANS V11 - FEM Structure File C:\projekte\Ball_Valve\ballvalve
Files View Mesh Generation	dit FEM-Project FEM-Analysis Postprocessing Training
Evads Show Loads ■ Boundary-Co	1. Boundary Conditions → Inditions I Show Boundary-Conditions Element-Groups Surface Modus is active - Surface= 9
	🖷 Create Boundary Conditions — 🗆 🗙
	Number of Boundary Conditions 0 New Value of Boundary 1E-10 Constraints displacement in in X Direction in Y Direction in Z Direction in Y Direction Clamped fixed (Colour of Axis: BLACK: X-Axis: BLUE: Y-Axis: RED: Z-Axis) Selection: Oragging a model region Select Nodes Select all showing nodes O Define a coordinate range Select all showing surfaces Show the Boundary Conditions: .0002 Cancel Editor Cancel Editor Cancel Editor Challet BCs

The boundary conditions are defined by the node and the degree of freedom. An additional value specification indicates how large the displacement or the rotation of this boundary condition is. This value is almost always zero or very small, since in practice solid bearings or clamps predominate. An exception are the spring constants and the elastic bedding.

Degrees of freedom

- FHG = 1 Constrain the displacement in X direction
- FHG = 2 Constrain the displacement in Y direction
- FHG = 3 Constrain the displacement in Z direction

Additional degrees of freedom for BEAM2 and Shell elements:

- FHG = 4 Constrain the rotation about X axis
- FHG = 5 Constrain the rotation about Y axis
- FHG = 6 Constrain the rotation about Z axis

Select in the next dialog box "Clamped fixed" and the Selection "Select Surfaces" and click on the button "Create RBs" and double click on the surfaces 16 and 13 and confirm in the Selectbox the input with "Create".



Select the "View" tab and the menu to select the color for model, intersection, edges, background, loads or boundary conditions by clicking on the color frame.

The following dialogue box for the settings appears in the DirectX9 interface:



💀 Show Loads, Boundary Con	ditions and No	<u></u>)		×
Colour Gradient Background				
Colours for background				
Colours for model				
Show Boundary Conditions	.0002	nomal	~	
Show Loads	.0005	nomal	~	
Show Nodes	.07105027	nomal	~	
Show Boundary Conditions				
Show Loads	Loads values	. N		
Node numbering	Element numb	pering		
Element groups				
CANCEL		OK		

Create Material Datas



Select the "Edit FEM Project" tab and the icon Material-Datas to enter the material data such as Young's modulus and Poisson Ratio where steel is always preset.

10.10	C. STORY	The sectors we see as sectors of the		1	
	Name	Material Datas			
	Youngs modulus	210000			
	Poisson Ratio	.3			
	Density	7.8E-06			
	Heat Coefficient	0			
€					
Ele	ement Group: 1	Element TET4	_	<	>
Ele	ement Group: 1	Element TET4		<	>
E	ement Group: 1 Isotrop Material Data Ba	Element TET4 O Anisotro		<	>

To call up an extensible material database, use the "Material Database" menu.

es					
ial:		Dens	ity (kg/mm³):		
n Modulus (MPa)		Heat	Coefficient:		
g Hoddids (Hird).					
on Ratio:		Heat	Conductivity (W / mm K):		
amand Buttons					
dd Edit Delete	Cancel Save	Take Up Material	el SI-Einheiten:	Millimeter	Meter
erial Data Base			-		
Material	Young Modulus	Poisson Ratio	Density	Heat Coefficient	Heat Conduct.
Glass technic	75000	0.17	0.0000022	0.0000059	0.009
Gold	192900	0.42	0.00001929	0.0000143	0.312
Gray Iron	108000	0.22	0.0000072	0.0000118	0.05
Ice	9600	0.33	0.00000917	0.00000502	0.0022
Iron	211000	0.28	0.000078	0.000012	0.074
Lead	160000	0.44	0.00001134	0.000029	0.34
Magnesium	44000	0.28	0.000001740	0.000026	0.171
Marble	72000	0.3	0.0000026	0.000011	0.028
Nickel Silver	110000	0.37	0.000083	0.000018	0.025
Platinic	170000	0.39	0.000021400	0.000009	0.07
Plexiglass	3200	0.35	0.0000012	0.000077	0.00017
Polyamid (Nylon)	3300	0.41	0.0000033	0.00009	0.25
Polycarbonat	2350	0.37	0.0000012	0.00007	0.2
Polyester	4000	0.4	0.0000014	0.00008	0.3
Polyethylen	1200	0.46	0.0000094	0.0002	0.45
Polymethylmethacrylat (PMM	3300	0.38	0.00000119	0.00008	0.18
	2100	0.35	0.0000014	0.00011	0.31
Polyoxymethylen (POM)	5100	0.00	010000011		1212.7

FEM Analysis

Following is a FEM Analysis, select the FEM Analysis tab. Here, the various FEM solvers for Statics, Dynamics, Temperature, Buckling, Nonlinear and Fatigue Analysis are called.

Files	View Mesh Generatio	n Edit FEM-Projec	t FEM-Ana
1. 1. 1. 1. 1. 1. 1. 1. 1. 1. 1. 1. 1. 1	Statics	Choose FEM-Solver	Structure Moo Model Dimen
1. 2. 3. 4. 5. 6. 7. 8. 9. 10.	Statics Dynamics Temperature Buckling Geometric Nonlinear Contact-Analysis Material Nonlinear Formoptimization Fatique (AD-Merkblatt SD2) Casting-Analysis-Tool	Select FEM	Structure Ir

Select "1. Statics" for either the MEANS-Solver from HTA-Software or

C:\projekte\B	all_Valve\ballvalve.fem			
Select Solver	O MEANS-Solver	Quick-Solver		
	Step 1: Starting	FEM Analysis		
	Step 2: Starting F	ostprocessing		
	Step 3: Refinement of t	he Element Stresses		
Select	FEM Solvers	Define Res	ults	

or the quick "Quick-Solver" to calculate the deformations and stresses.

Quick-Solver		(222)		×
Normal Precision	 show and solve with C3D4 (4-node linear tetrahedral elements) show C3D4 and solve intern with a refining mesh of 8 x C3D4 		\bigwedge	
Path for INP-Solver	C convert C3D4 -> C3D10 and show and solve with C3D10	Bro	wser	
Path for INP Files:	C:\projekte\kugelventil\Kugelventil.INP			
	Select Solver In-Core-Solver Out-of-Core-Solver			
	Start FEM-Solver with INP-Interface			
	Settings Help + Infos Cancel			

FEM-Project: C Please wait: FE	C:∖projekte∖Ball_Val EM-Analysis of 8391	re∖ballvalve 1 Elements and 18106 Nodes						
	Start Posprocessing MEANS V11							
		Runtime:	0:0:0:11:486	Cancel				
STEP	1	octod						
Decascading	the MPC's	re re u						
Determining number of eq 53523	the structur quations	re of the matrix:						
number of no 1025082	onzero lower	triangular matrix ele	ments					
Using up to 1 cpu(s) for the stress calculation.								
Using up to	1 cpu(s) for	spooles.	Summeric Spoores Sorver					

After the FEM analysis a short tone signal is heard, now the menu "Start Postprocessing MEANS V12" is active again and you can start the postprocessor for the result evaluation.

Postprocessing

The postprocessor for the result evaluation is started automatically, select the tab "Postprocessing".

				FEM Sys	stem MEANS V11	- FEM Structure File C:\projekte\E
Files View M	Aesh Generation Edit F	EM-Project FEM	I-Analysis	Postprocessi	ng Training	
Show Results	Displacement-Factor Pick, Search Values	List Result Values Legende: Legende	e 1 🔻	Intern Extern	List FEM-File List STA-File	Install DirectX11 Start FAST-Postprocessor
Postprocessing 🕞	Settings 🗔	Legende/Tabe	lle 🗔	Animations 🕞	List Files 🕞	New Beta-Version DX11 🕞
	R Postproces	sing		-	X	
	Results © Contour o O Nodal Str O Element :	Results Loa Contour of Displacements Nodal Stress Contour Element Stress Contour 		ad Case 1		
	Accuracy Edit Accu	Accuracy Edit Accuracy:		Displacement Factor		
	1	1 3 4 Pic		Edit Colours for Legend ick, Search, Save Values		
	Select Result (Select Result Component Displacement in x, y and z direction			~	
	Cancel	Cancel Start Postprocessing				

The following Result Evaluations are available:

- Contour of Displacments
- Nodal Stress Contour
- Element Stress Contour
- Reaction Forces
- Contour of Forces



Load Case 1: Displacments in X-, Y- and Z-Direction

Load Case 1: v.Mises-Stress



The Displacements are shown with a Displacement Factor of -3178872