Part 29: Natural Frequencies of an unsupported hollow box with additional masses

The natural frequencies of an unsupported hollow box made of steel with a length of 2000 mm and a square profile of 150 mm x 150 mm x 10.7 mm are calculated with and without additional masses using the FEM system MEANS V12.



Two methods for additional masses

The additional masses can be specified in MEANS V12 either as an additional mesh extension with two solid bodies or with a weight force.

The latter method has the great advantage that only weight forces in N and no material data are required, which means that natural frequencies of FEM models with additional masses of cars and people can also be calculated.



Unsupported hollow box with two additional masses as a mesh extension



Unsupported hollow box with a weight force for the additional masses

Unsupported hollow box with additional masses as a mesh extension

The CAD model is expanded with 2 solid bodies 100x100x200 and 100x100x300 and meshed in STEP format with the 3D mesh generator GMSH



and exported in Abaqus INP format for MEANS V12:



The FEM mesh was imported into MEANS V12 with menu "INP-File Abaqus/Gmsh"



and consists of 37,938 TET10 volume elements and 73,299 nodes



Element group 2 with the additional masses is created with the "Create element group from several surfaces" menu

F Belastungen	1. Knotenbelast	tung	Elementg
EG= 2		፼ EG = 2	×
🗹 ON	EG= 1	Bementgruppen erzeugen	
Ø ON	EG= 2	Elementgruppe erzeugen: 2	
🗹 ON	EG=3	O EG mit aufgespannten Rechteck und einer Tiefe erzeugen	
🗹 ON	EG=4	O EG mit allen angezeigten Knoten erzeugen	
🗹 ON	EG=5	Elementgruppe aus mehreren Flächen erzeugen	
M ON	EG=6	O Elementgruppe mit Koordinatenbereich erzeugen	
🗹 ON	EG=7	O Elementgruppe mit Kreisbogen erzeugen	
Für neue Farbe au	f Farbrahmen klicken jitter sichtbar	O Z-Tiefe O Y-Tiefe X-Tiefe von: -100000 bis: 100000	
Refresh H Gruppen 1 - Neue Elementg	Hidden-Line 7 ~	Erzeuge Elementgruppe	

and with the "Material data" menu, the modulus of elasticity 72000 N/mm², the Poisson number 0.34 and the density 2.7E-6 kg/mm³ for aluminum are taken from the self-extensible material database.

FE	M-System MEANS V	12 - Strukturdatei C:\proj ;	ekte\massenschwinger\l	nohlkasten2\tet4m.fem		
C	Datei Ansic	ht Netzgenerierung	FEM-Projekt bearb	eiten FEM-Analyse Ergel	onisauswertung Traini	ng
Bel	F 1. Knote astungen ☑ Belast	enbelastung + ungen darstellen Rar	Dedingungen 2 Rai	ndbedingungen 🔹 Indbedingungen darstellen Ele	mentgruppen Material	daten Editor 6. Belastunge
	Materialdaten		- 🗆 🗙			
•	Bezeichnung E-Modul	Materialwerte 71000		💀 Material-Datenbank		
	Poisson-Zahl Dichte	.34 2.700003E-06		Werkstoff: Aluminii E-Modul (N/mm²): 71000	um	Dichte (kg/mm³): Wärmeausdehnungskoeffizier
	Waermekoeffizient	2.39E-05		Poisson-Zahl: 0.34		Wärmeleitfähigkeit (W/mmK):
				Add Delete Save	Material übernehmen	Datenbank einladen
				Werkstoff	E-Modul	Poisson-Zahl
E	ementgruppe: 2	Elementtyp: TET10 Anisotrop nk	с >	Auminium Beton Bronze Eis (4 °C) Eisen Glas Cuarz Glas technisches Glass - Window Glass Glimmer Gold Graphit Grauguss Gummi hart Gummi kat Gummi kat Kadmium Keamik	71000 / 20000 / 20000 / 20000 / 20000 / 20000 / 211000 / 75000 / 210000 / 210000 / 210000 / 78000 / 78000 / 78000 / 78000 / 78000 / 5000 / 5000 / 5000 / 5000 / 5000 / 5000 / 5000 / 51000 / 5	0.34 2 0.4 8 0.4 8 0.33 9 0.28 6 0.17 2 0.28 2 0.28 2 0.28 2 0.28 2 0.28 2 0.28 2 0.27 0.44 0.371 1 0.44 1 0.22 7 0.44 1 0.22 4 0.3 0 .3 2

Unsupported hollow box with a weight force

With this method, the weight forces are first calculated from the additional masses:

Weight 1 = 100 mm * 100 mm * 200 mm * density of aluminum = 0.1m * 0.1m * 0.2m * 2700kg/m³ * 10m/s² = 540N Weight 2 = 100 mm * 100 mm * 300 mm * density of aluminum = 0.1m * 0.1m * 0.3m * 2700kg/m³ * 10m/s = 810N

Create a Point Load

Select the tab "Edit FEM Project" and "Point Load" and create a load with the value = -1 in the Y-direction with the selection "Define coordinate range" and define the following area:

Date	i Ansicht Netzgenerierung	FEM-Projekt bearbeiten	FEM-Analyse	Ergebnisauswertung	Training	
Elastungen	1. Knotenbelastung Belastungen darstellen Rand	bedingungen 1. Randbedi	ngungen 🔹 Igungen darstellen	Elementgruppen	Materialdaten	Editor 8. 1
			mozen	2		
	🖳 Knotenlast erzeugen	- 0	× 💀	Goordinatenbereich		ı x
	Aktueller Lastfall: 1 Anzahl Lastwerte: 0	- + Neu	v	on X: 25	bis X: 125 bis Y: 150	
	Freiheitsgrad:	(Einheit z.B. in N)	vo	on Z: 200	bis Z: 400	
	Selektion:			Knotenlas	st erzeugen	
	 Flächenmodus einzelne Knoten anklicken Koordinatenbereich definieren 	Rechteck aufspannen alle angezeigten Knoten alle angezeigten Surfaces				
	Knotenlast-Symbole ände	em KL-Farbe:				
	Cancel Editor	Belastung erzeugen				
		Belastung löschen				
						1997 - 1997 - 1997 - 1997 - 1997 - 1997 - 1997 - 1997 - 1997 - 1997 - 1997 - 1997 - 1997 - 1997 - 1997 - 1997 -

Then select "Loads" and "Editor" and calculate the individual value from 540 N / 318 = 0.1698 N, which then has to be multiplied by -1 using the "Load Case Factor" menu.

72

ngen ⊠ B	Knotenbelastung elastungen darstelle	n Randbeding	1. Randbed	lingungen m ngungen dars Ir	tellen Elementgrup	pen Materialdate	Editor 6.	Belastungen	• Tempera	tur Ty	
stungen	14.14	510		×							
lr.	Knoten	FHG	Wert	- î							
	1112	2	1698								
	1112	2	1698								
	1147	2	- 1000								
	1149	2	. 1698								
	1140	2	- 1698					~			
	1150	2	. 1698		🖼 Lastfall			×			
	1151	2	. 1698								
	1152	2	. 1698		Aktueller Lastfall:	1 ~	< >				
0	1204	2	. 1698		Enktore	0 1698					
	1204	2	- 1698		Faktor=	0.1000					
12	1205	2	. 1699		mult	iplizieren 🔿 o	lividieren				
Neuer Lastfa Lastfall II Lastfall Flächenlast- Knotenlast-	ill erzeugen	Last Lastfälle (Temp Freih	fälle überlagem fälle überlagem eraturfast einlesen etsgrade ändem						V		
Ý											

Follow the same steps for weight force 2 with the following coordinate range:

X from	25 to	125
Y from	150 to	150
Z from	1000 to	1300



Results

Natural frequencies of the unsupported hollow box without additional masses

As a special case of multiple eigenvectors, the six rigid body modes (natural frequencies of 0.0 Hz) are usually not calculated in their pure form, but as a linear combination of the three translational and three rotational degrees of freedom:



In a static analysis, even a single rigid body mode would lead to the calculation being aborted (load cannot be absorbed) or at least to ambiguous solutions (no associated load). The number of rigid body media corresponds to the degree of static underdetermination, so that natural frequency analyzes can also be used to detect missing boundary conditions.

The determined eigenvectors 7 and 8 (first bending vibration) as well as 9 and 10 (second bending vibration) coincidentally correspond to the main axes:



Natural Frequency No. 7 = 251.48 Hz in the X-Z-Plane

Natural Frequency No. 8 = 251.48 Hz in the Y-Z-Plane





Natural Frequency No. 9 = 632.55 Hz in the X-Z-Plane

Natural Frequency No. 10 = 632.56 Hz in the Y-Z-Plane



The natural frequencies calculated with MEANS V12 agree very good with the literature "FEM-Formelsammlung Statik und Dynamik" ISBN 978-3-8348-0980-3 from Lutz Nasdala on Side 53.

Natural Frequencies of the hollow box with additional masses



Natural Frequency No. 7 = 240.1 Hz in the X-Z-Plane

Natural Frequency No. 8 = 244.52 Hz in the Y-Z-Plane





Natural Frequency No. 9 = 608.49 Hz in the Y-Z-Plane

Natural Frequency No. 10 = 612.47 Hz in the X-Z-Plane



Natural Frequencies of the hollow box with a weight force



Natural Frequency No. 7 = 236.5 Hz in the Y-Z-Plane

Natural Frequency No. 8 = 237.45 Hz in the X-Z-Plane





Natural Frequency No. 9 = 604.25 Hz in the Y-Z-Plane

Natural Frequency No. 10 = 610.29 Hz in the X-Z-Plane



Hollow Box generate with a structured Hexahedron mesh

To generate a structured hexahedron mesh as opposed to an unstructured tetrahedron mesh with Netgen or Gmsh, the following steps can be performed in the Line-Modus with the 3D mesh grid generator.

Select "New" and menu "2D/3D Beam Model with Line-Modus" and first enter the outer profile 150 x 150 and then the inner profile 139.3 x 139.3 of the hollow box.

U	Files	View	Mesh Ger	neration	Edit FEM-P	roject	FEM-Ar	nalysis
			6	FEM-Merg MPC-Cont	e Impor act Expor	t: STL + t: DXF +		
New	Γ ₂	Open 🕞	Save 🕞	Unit	C	AD	Path	Fa
	🖳 N	EW PROJEC	.т		-3		×	
	0	3D Mesh G	enerator MEA	NSMES V1 (S	TEP, IGES, S	STL)		
	0	3D Mesh G	enerator GMS	H (STEP)				
	0	Mesh Gene	rator MEANSI	MES V2 for co	omlex Structu	res Ins	stall	
	۲	2D/3D Bea	m Model with	Line-Modus				
	0	3D Shel Mo	del with a Cor	ntainer Mesh (Genenerator			
			N	EW PROJECT				

Outer profile:

Choose "New" to create node 1 with the coordinates 0,0,0 and menu "Create single node".

Choose "New" to create node 2 with the coordinates 150,0,0 and menu "Create single node".

Choose "New" to create node 3 with the coordinates 150,150,0 and menu "Create single node".

Choose "New" to create node 4 with the coordinates 0,150,0 and menu "Create single node".

Inner profile:

Choose "New" to create node 5 with the coordinates 10.7,10.7,0 and "Create single node" menu.

Choose "New" to create node 6 with the coordinates 139.3,10.7,0 and menu "Create single node".

Choose "New" to create node 7 with the coordinates 139.3,139.3,0 and menu "Create single node".

Choose "New" to create node 8 with the coordinates 10.7,139.3,0 and menu "Create single node".

You should see the following eight nodes:

	💀 – 🗆 X
	Surface Nodes Lines
■4 ●3 ●8 ●7	Node: 8 New X: 10.7
	Manipulate Nadar
	Copy Range of Nodes
	Unit Nodes
	Check Nodes
	Mesh Generators
	2D Mesh Generator
	3D Mesh Grid
	EG= 1 V New
	Load DXF-Lines UNDO / REDO
•5 •6 •1 •2	Quit Line-Modus

Right select "3D Mesh Grid" and give following 4 edges 1, 2, 6 and 5 as well

8	💀 3D Mesh G	irid			×	7
		Nadaa af Edaa 1	1	strife.	-	
		Nodes of Edge 2:	2			
		Nodes of Edge 3:	6		_	
		O Nodes of Edge 4:	5		_	
		Delete E Number of Nodes in X o Number of Nodes in Y o	idge lirection: 14 lirection: 5			
		Generate 3D	-Mesh Grid			
	1					
5				 <u></u>	<u> </u>	•

enter the mesh density in X-direction = 14 and in Y-direction = 5.

Select the View tab and click Rendering and click again "with mesh" then "Quad Mesh" appears and the Quad-Mesh can be seen.



Repeat the step with edges 2, 3, 7 and 6 and with the mesh density in X-direction = 14 and in Y-direction = 5.

a 4	🔛 Netzgitter		2752	×	73
		Knoten von Kante 1: 2]	
		O Knoten von Kante 2: 3]	
		O Knoten von Kante 3: 7]	
		O Knoten von Kante 4: 6]	
		Kante löschen			
		Anzahl Knoten in X-Richtung: 14			
		Anzahl Knoten in Y-Richtung: 5			+++
					-
		3D-Netzgitter generieren			+++
2	5///		1 1		
	1	///			

Repeat the step with the edges 8, 7, 3 and 4 and with the mesh density in X-direction = 14 and in Y-direction = 5.

netzgitter		5 <u>9461</u> 5	×
	Knoten von Kante 1:		
	O Knoten von Kante 2: 7		
	O Knoten von Kante 3: 3		
	O Knoten von Kante 4:		
	Kante löschen		
	Anzahl Knoten in X-Richtung:	14	
	Anzahl Knoten in Y-Richtung:	5	
	3D-Netzgitter generier	en	

Repeat the last step with edges 5, 8, 2 and 1 and with mesh density in x direction = 14 and in y direction = 5.

🖳 Netzgitter				×	1
	Knoten von Kante 1:	5			1
	O Knoten von Kante 2:	8	 		
	O Knoten von Kante 3:	2	 		
	O Knoten von Kante 4:	1			
	Kante löscl	hen			
	Anzahl Knoten in X-Richt	ung: 14			-
	Anzahl Knoten in Y-Richt	ung: 5			1 1
	3D-Netzgitter g	enerieren			/
			 	6	\mathbb{N}

Then select an extrusion with the "Mesh Generation" tab and the "Quad Meshes, Refine, Delete" menu and select the "Extrusion" tab in the new dialog box.

Files View	Mesh Generation	Edit FEM-Pr	roject FEM-Ana	lysis Po	ostproce	essing	Ti
3D Mesh Gen Local Refinen	erators nent		QUAD-Meshes,	Refine, Delet	:e	Check Jacob	c Node
D Mesh Generation	n 🕞 2D Me	sh Generation 5	Manipula	te Meshes	Г¥		Check
<u></u>							
🖳 Quad Meshir	ng / Refine / Conver	ter / <mark>E</mark> xtrusion			<u></u>		×
Quads Refine (Converter Extrusion	Rotation Delete	• Tum				
	(E. E. L.						
	1 HOLE HOLE 101010 10011 10						
	(FOI Extrusion you hi	eed a 20 mesh wit	h Z=0)				
	(For Exitosion you in Densit	eed a 2D mesh wit	201]			
	(Por Extrasion you m	eed a 2D mesn wit	201]			
	(For Exitusion you in Densit	eed a 20 mesn wit y in Z direction= Height=	201 2000]			
	(roi Exitusion you in Densit	y in Z direction= Height= Thickness from	201 2000 Material Datas]]			
	(For Exitusion yourn Densit	y in Z direction= Height= Thickness from	h Z=0) 201 2000 Material Datas]]			
DYF	OF Exitusion your Densit O Z	eed a 2D mesh wit y in Z direction= Height= Thickness from	201 2000 Material Datas]]]			

With "Density in Z direction = 201" and "Z-Height = 2000" a structured Hexahedron Mesh with 41 600 HEX8-Elements and 52 260 Nodes is created.

