

altairhyperworks.com

Technical Support

Altair provides comprehensive software support via web FAQs, tutorials, training classes, telephone and e-mail.

Altair Support on the World Wide Web

The Altair web site is a valuable online companion to Altair software. Visit www.altairhyperworks.com for tips and tricks, training course schedules, training/tutorial videos, and other useful information.

Altair Training Classes

Altair training courses provide a hands-on introduction to our products, focusing on overall functionality. Courses are conducted at our main and regional offices or at your facility. If you are interested in training at your facility, please contact your account manager for more details. If you do not know who your account manager is, please send an e-mail to training@altair.com and your account manager will contact you.

Telephone and E-mail

When contacting Altair support, please specify the product and version number you are using along with a detailed description of the problem. Many times, it is very beneficial for the support engineer to know what type of workstation, operating system, RAM, and graphics board you have, so please have that information ready. If you send an e-mail, please specify the workstation type, operating system, RAM, and graphics board information in the e-mail.

To contact an Altair support representative, reference the following table or the information available on the HyperWorks website: www.altairhyperworks.com/ClientCenterHWSupportProduct.aspx.

Location	Telephone	E-mail
Australia	64.9.413.7981	anzsupport@altair.com
Brazil	55.11.3884.0414	br_support@altair.com
Canada	416.447.6463	support@altairengineering.ca
China	86.400.619.6186	support@altair.com.cn
France	33.1.4133.0992	francesupport@altair.com
Germany	49.7031.6208.22	hwsupport@altair.de
India	91.80.6629.4500 1.800.425.0234 (toll free)	support@india.altair.com
Israel		israelsupport@altair.com
Italy	39.800.905.595	support@altairengineering.it

Location	Telephone	E-mail
Japan	81.3.6225.5830	support@altairjp.co.jp
Malaysia		aseansupport@altair.com
Mexico	55.56.58.68.08	mx-support@altair.com
South Africa	27 21 8311500	support@altair.co.za
South Korea	82.70.4050.9200	support@altair.co.kr
Spain	34 910 810 080	support-spain@altair.com
Sweden	46.46.460.2828	support@altair.se
United Kingdom	01926.468.600	support@uk.altair.com
United States	248.614.2425	hwsupport@altair.com

For questions or comments about this help system, send an email to connect@altair.com.

In addition, the following countries have resellers for Altair Engineering: Colombia, Czech Republic, Ecuador, Israel, Russia, Netherlands, Turkey, Poland, Singapore, Vietnam, Indonesia

Official offices with resellers: Canada, China, France, Germany, India, Malaysia, Italy, Japan, Korea, Spain, Taiwan, United Kingdom, USA

See www.altair.com for complete contact information.

Table of Contents OptiStruct for Linear Analysis

Basic Linear and Dynamic Solutions

Table of Contents	3
Chapter 2: Linear Static Analysis	5
Exercise 2a: Static Analysis of a Solid Bracket	7
Exercise 2b: Static Analysis of a Simply Supported Beam	21
Chapter 3: Inertia Relief Analysis	31
Exercise 3a: Satellite Inertia Relief Analysis	
Chapter 4: Modal Analysis	
Exercise 4a: Compressor Bracket Modal Analysis	
Exercise 4b (optional): Simply Supported Beam Modal Analysis	49
Chapter 5: Linear Buckling Analysis	53
Chapter 5: Linear Buckling Analysis	
	55
Exercise 5a: Wing Linear Buckling Analysis	55 63
Exercise 5a: Wing Linear Buckling Analysis	55 63 65
Exercise 5a: Wing Linear Buckling Analysis Chapter 6: Thermal Stress Steady State Analysis Exercise 6a: Thermal Stress Analysis of a Beam	55 63 65 79
Exercise 5a: Wing Linear Buckling Analysis Chapter 6: Thermal Stress Steady State Analysis Exercise 6a: Thermal Stress Analysis of a Beam Chapter 7: Advanced Topics	55 63
Exercise 5a: Wing Linear Buckling Analysis Chapter 6: Thermal Stress Steady State Analysis Exercise 6a: Thermal Stress Analysis of a Beam Chapter 7: Advanced Topics Exercise 7a (optional): Static Analysis using Freeze Contact	

HyperWorks v2019

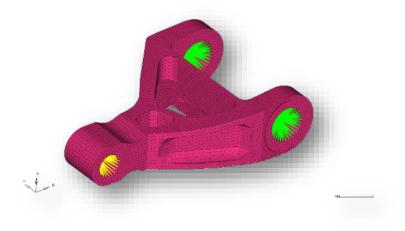
Chapter 2

Linear Static Analysis

Chapter 2: Linear Static Analysis

Exercise 2a: Static Analysis of a Solid Bracket

In this exercise, a structural analysis is performed on a bracket modeled with solid elements. The objective is to set up a linear static analysis from scratch starting just with the meshed model.



Model Information

- Force = (12000,12000, -20000) N
- Material Aluminium:
 - E =70000 MPa
 - Nu = 0.33
 - S₀ = 240 Mpa
 - S_{ADM} = 0.7*S₀
- o UNITS: N, mm, ton, s

File Name and Location

...\STUDENT-EXERCISE\2a_Torsion_Link\torsion_link.hm

Step 1: Open the model in HyperMesh Desktop with OptiStruct user profile selected

Step 2: Review the model and check the dimensions of the model

Tip: The length system is reasonable to be millimeter, the consistent units are mm, MPa, N. The mesh size is about 10 mm. There are three components:

- torsion link for tetra elements
- RBE2 for RBE2 elements for two supports
- RBE3 for a RBE3 element to apply the force. RBE2 would include a rigid condition that doesn't exist.

Step 3: Create a MAT1 material alu for aluminum with the properties: Young's modulus 70000 MPa, Poisson's ratio 0.33

Tip: Click right mouse button in the **Model Browser**, select **Create > Material** or use HyperMesh's Quick Access Tool (Ctrl+f) to create accordingly MAT1 material card.

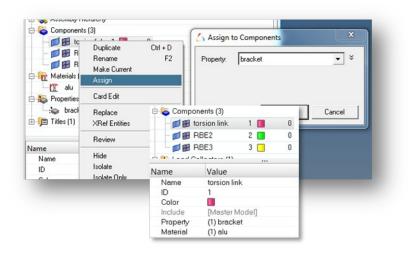
		🖨 🙀 Materials (1)	
		🔤 🈰 alu	1 🚺 🛛 0
		m. 🖻 Titles (1)	
		Name	Value
		Solver Keyword	MAT1
		Name	alu
		ID	1
		Color	
		Include File	[Master Model]
		Defined	
		Card Image	MAT1
		User Comments	Hide In Menu/Export
t1	Q	E	70000.0
.T1		G	
.T10		NU	0.33
		DUO	
	□ ■ RBE2 2 ■ □ ■ RBE3 3 □ ⊕□ Titles (1)	0 0	
	Expand All Collapse All Configure Browser Name Value	Assembly Beam Section Collector Beamsection Block Component Contact Contact Contact Surface Cross Section Curve Feature	
		Field Group Include File Laminate Load Collector Load Step	

Here we will use MAT1 which is a linear isotropic material that can represent the aluminum behavior well. For more details about this material or other material formulations, please refer to the HyperWorks Online Help

Step 4: Create a PSOLID property bracket referencing material alu and assign it to the torsion link component

Tip: Click right mouse button in the model browser, select *Create > Property* or use HyperMesh's Quick Access Tool (Crtl+f) to create according PSOLID property card and assign property to torsion link component by right mouse click on the component, select **Assign**. This will make all elements in this component use this property. Note that this assignment serves as a default for this collector and will not change any element from this component has another property directly associated with it in its element definition

Entities		ID 😧 In	clude
	embly Hierarchy		
	mponents (3)		
	🗭 torsion link		0
		2 📘	0
-	-	3 📃	0
3 🙀 Ma			
		1 📘	0
- 🧊 Titl	BS (I)		
-	Create	۱.	Assembly
	Expand All		Beam Section Collector
	Collapse All		Beamsection
-			Block
lame Van	Configure Bro	wser	Component Contact
			Contact Contact Surface
			Cross Section
		_	Curve
		_	Feature
		_	Field
			Group
			Include File
			Laminate
			Load Collector
			Load Step Material
			Multibody
			Output Block
			Parameter
		_	Plot
		_	Ply
			Property
			Region



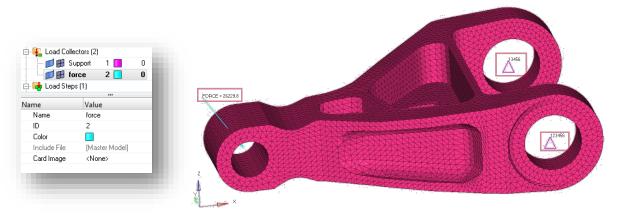
Step 5: Create a load collector Support (no card image) with the following SPC load type constraint, node 4830: DOFs 1-6 and node 4831: DOFs 1, 3-6

Tip: Click right mouse button in the **Model Browser**, select **Create > Load Collector** and create the two SPCs. The boundary conditions can be created from the menu **BCs > Create > Constraints**

Create 🕨 🕨	Assembly Beam Section Collector		
Expand All	Beamsection		
Collapse All	Block		
Configure Browser	Component		
	Contact		
alue	Contact Surface		
	Cross Section	🖨 👫 Load Colle	
	Curve	🚽 🗾 🔛 Su	upport 1 📘 🛛 0
	Feature	📥 🎰 Materials f	11
	Field	Name	Value
	Group	Name	Support
	Include File	ID	1
	Laminate	Color	
	Load Collector	Include File	[Master Model]
	Load Step	Card Image	<none></none>
	Material		

Step 6: Create a load collector force (no card image) containing a force on node 1 with constant components {12000, 12000, -20000}

Tip: Click right mouse button in the **Model Browser**, select **Create > Load Collector** and create the force using the top menu **BCs > Create > Forces**



Step 7: Create a load step Load of type Linear Static using Support as SPC and force as LOAD entry

Tip: Click right mouse button in the Model Browser, select Create > Load Step

		i⊖ 🝓 Load Steps (1) ↓ 🖕 Load 1 中 🙀 Materials (1)	0
U.		Name	Value
ate 🕨 🕨	Assembly	Solver Keyword	SUBCASE
	Beam Section Collector	Name	Load
	Beamsection	ID	1
se All	Block	Include	[Master Model]
Provincer	Component	User Comments	Hide In Menu/Expo
ure Browser	Contact	Subcase Definition	
	Contact Surface	🗆 Analysis type	Linear Static
		SPC	(1) Support
	Cross Section	LOAD	(2) force
	Curve	SUPORT1	<unspecified></unspecified>
	Feature	PRETENSION	<unspecified></unspecified>
	Field	MPC	<unspecified></unspecified>
	Group	DEFORM	<unspecified></unspecified>
	Include File	STATSUB (PRELOAD)	<unspecified></unspecified>
	Laminate	STATSUB (PRETENS)	<unspecified></unspecified>
	Load Collector	SUBCASE OPTIONS	
	Load Step	LABEL	
	Material	SUBTITLE	
	KJ 07 1	ANALYSIS	
		TYPE	STATICS
		ASSIGN	

Step 8: Export the model as solver deck

Tip: Click on button Solver Export Deck, choose file name and hit Export

12 - 🎉	- 🛃 - 省 - 🚰 -	🖍 - 🔉 🎁
Session M	1odel Export ×	
🍒 🌈 🎝	6 1 1	
File selection -		
File type:	OptiStruct	•
Template:	Standard format	•
File:	torsion_link_training.fem	Ē
Export optio	ons	
Export:	Custom 💌	Select Entities
Solver option:	s: Select Options	
Comments:		
HyperM		
Connect		
I Part Ass	emblies/Parts	
Include files:	Merge 💌	
🔽 ID Rules		
Prompt to	save invalid elements	
Prompt be	efore overwrite	
		Export

Step 9: Review the .fem file in a text editor and understand the references

Tip: Exported . fem file without HyperMesh comments

• Constraint reference

- Load reference
- Property reference
- Material reference

```
$$ optistruct
SUBCASE 1
 LABEL Load
ANALYSIS STATICS
SPC =
LOAD =
               2
BEGIN BULK
GRID
                       -520.701-61.91252.0557-5
              1
. . .
...
. . .
           42822
                            5866 6434 10115 10208
CTETRA
PSOLID
                      1
MAT1
                10000.0
                               0.33
                    4830 123456 0.0
4831 13456 0.0
SPC
              1
SPC
                    4831
                          13456
                                   0.0
                                       12000.0 12000.0 -20000.0
FORCE
               2
                       1
                               01.0
ENDDATA
```

Step 10: Run the analysis with OptiStruct

Tip: Use the HyperWorks Solver Run Manager to run the exported . fem file, make sure that OptiStruct states "ANALYSIS COMPLETED" and review the created files.

Chapter 2: Linear Static Analysis

# HyperWorks Solver Run Manager (@DEWLT257)	
File Edit View Logs Solver HyperWorks Help	
Input file(s): torsion_link_training.fem	
Options: -ncpu 4 -core in	
Use SMP: -nt 2 Use MPI options	☑ Use solver control
	Run Close
🖝 torsion_link_training.fem - HyperWorks Solver View	
Solver: optistruct_2017.2.3_win64.exe Input file: torsion_link_training.fem Job completed	
Run command:/hwsolver.tcl/torsion_link_training.fem -screen -ncpu 4 - core in -dir -solver OS Message log: Optimization summany: Graph	
Messages for the job:	
ANALYSIS COMPLETED.	
4	
Run summary:	
** ** OptiStruct 2017.2.3 ** ** Advanced Engineering Analysis, Design and ** ** Optimization Software from Altair Engineering, Inc. **	
** ** ** ** ** ** ** ** ** ** ** ** **	torsion_link_training.fem
Build tag: 0900751_9433xxx_Ce64RBW8UH14M:138991-2 4000020006000	torsion_link_training.h3d torsion_link_training.html
COPYRIGHT (C) 1996-2017 Altair Engineering, Inc. Altair Engineering, Inc. Contains trade secrets of Altair Engineering, Inc. Contains trade secrets of Altair Engineering, Inc. Decompilation of disassembly of this software structly prohibited.	torsion_link_training.mvw
	torsion_link_training.out torsion_link_training.res
The amount of memory allocated for the run is 115 MB. This run will use in-core processing in the solver. NALYSIS COMPLETED.	torsion_link_training.stat
	torsion_link_training_frames.html
<pre>==== End of solver screen output ===== </pre>	torsion_link_training_menu.html
Results View v Close	

Step 11: Add output requests in order to get desired

- 1. echo .out file on the screen
- 2. get results only in H3D format and suppress the html file output

Tip: Use HyperMesh's Quick Access Tool (Crtl+f) to add control cards

SCREEN,OUT OUTPUT,H3D,ALL OUTPUT,HTML,,NO

	😑 🛜 Cards (2)		
	- CREEN	1 0	
		2 0	
	🖽 💊 Components (3)		
	🙃 📭 📭 Load Collectors (2)		
Q	Name	Value	
	Include File	[Master Mod	lel]
UEST	Status		
	number_of_outputs =	2	
	🗆 OUTPUT 1		
ocks	KEYWORD	H3D	
5	FREQ	ALL	
ks	OPTION		
	DUTPUT 2		
ks	KEYWORD	HTML	
	OPTION	NO	

Step 12: Rerun the analysis with OptiStruct

Tip: Review the created files and note the additional output in the HyperWorks Solver View

olver: optistruct_2017.2.3_win64.exe		
nput file: torsion_link_done.fem	Job completed	
un command:/hwsolver.tcl/torsion_link_done	fem screen sponul score in sdir solver OS	
lessage log:		
essages for the job:		Graph
cooldgeb for one job.)le Grid/Elem ID Value	^
ANALYSIS COMPLETED.	742_Z -1.04148	
		*
	, ,	
un summary: Running in-core solution)		
<pre>kunning in-core solution) Olume Subcase Compliance Epsilon 1 9.456644E+03 4.347119E-1 fote : Epsilon = Residual Strain Er</pre>		-
ANALYSIS COMPLETED.		
ESOURCE USAGE INFORMATION		
MAXIMUM MEMORY USED	115 MB	
MAXIMUM DISK SPACE USED	27 MB	
***************************************	**************	
OMPUTE TIME INFORMATION		
EXECUTION STARTED EXECUTION COMPLETED	Tue Feb 06 08:29:30 2018 Tue Feb 06 08:29:34 2018	
ELAPSED TIME	00:00:04	
CPU TIME	00:00:07	
********	***********	
***** END (OF REPORT *****	
For Useful OptiStruct Tips and Tr: http://www.altairhyperworks.com/t;		
==== End of solver screen output =		torsion_link_done.fem
==== Job completed ====		torsion_link_done.h3d
		torsion_link_done.mvw
		torsion_link_done.out
	Results View v Clos	torsion_link_done.stat

Step 13: Review the .out file wrt warnings, errors and Auto-SPC

Chapter 2: Linear Static Analysis

Tip: There are 19 elements that exceeded recommended range (warning) for the element quality check.

OptiStruct auto-SPCed 1344 degrees-of-freedom (DOF).

	r similar error/w PARAM,CHECKEL,FUI				ed,	
450	I AIGAI, CILCILL, I OI					
				OIC		
Flomont Out	lity Check Summar					
Qua		- Y				
	elements that exc			-		
Note: Only	element with the	highest vi	lolation of	each che	ck is listed b	elow.
Recommended	range violations	:				
Liement	Property				Max. Viol. Value type	
TETRA Fac	e Vertex Angle	19	15.00	165.00	11.95 L	15597
	o-SPC d.o.f.s for					
LIST OF AUT						
Total numbe						
Total numbe	r of Auto-SPC d.c					
Total numbe	r of Auto-SPC d.c).f.s = 134	14			
Total numbe	r of Auto-SPC d.c	.f.s = 134	14			
Total numbe	r of Auto-SPC d.c Grid No.	.f.s = 134 	94 oonent			
Total numbe	r of Auto-SPC d.c	0.f.s = 134 Comp	14			
Total numbe	r of Auto-SPC d.c Grid No. 	0.f.s = 134 Comp	4 oonent 4 5 6			
Total numbe	r of Auto-SPC d.c Grid No. 3870 3871	0.f.s = 134 Comp	4 00nent 4 5 6 4 5 6			
Total numbe	r of Auto-SPC d.c Grid No. 3870 3871 3872	0.f.s = 134 Comp	44 0000ent 4 5 6 4 5 6 4 5 6			
Total numbe	r of Auto-SPC d.c Grid No. 	0.f.s = 134 Comp	44 000nent 4 5 6 4 5 6 4 5 6 4 5 6 4 5 6			
Total numbe	r of Auto-SPC d.c Grid No. 	0.f.s = 134 Comp	4 5 6 4 5 6 4 5 6 4 5 6 4 5 6 4 5 6 4 5 6 4 5 6			
Total numbe	r of Auto-SPC d.c Grid No. 	0.f.s = 134 Comp	4 5 6 4 5 6			
Total numbe	r of Auto-SPC d.c Grid No. 3870 3871 3872 3873 3874 3875 3876	0.f.s = 134 Comp	4 5 6 4 5 6			
Total numbe	r of Auto-SPC d.c Grid No. 	0.f.s = 134 Comp	4 5 6 4 5 6			

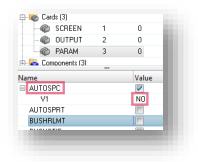
By default, OptiStruct uses AUTOSPC, ON as it helps to prevent undesired stops or failure runs. For example, if the model has an element unattached to the structure with no

constraint applied to it, the run would stop complaining about a rigid body movement. With AUTOSPC ON, OptiStruct would automatically fix this element and run the analysis.

The user should be aware of any DOF fixed by the AUTOSPC as it can lead to a wrong behavior. Also, do not forget that in the end, if the run is made with "AUTOSPC ON", to verify which DOF was fixed and if this has not affected the solution.

Step 14: Add parameter to deactivate automatic constraining

Tip: Use HyperMesh's Quick Access Tool (Crtl+f) to add control card PARAM, AUTOSPC, NO Note that this card will be added in the bulk section.



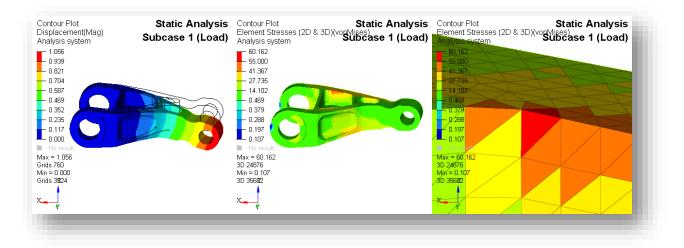
Step 15: Rerun the analysis with OptiStruct and review the .out file again

Step 16: Review the displacements and stresses in HyperView

Tip: Click on Results in the HyperWorks Solver View and HyperView will directly open the according .mvw session file created by OptiStruct.



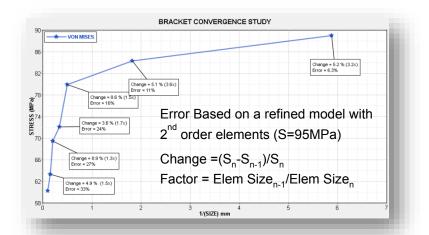
In the displacement contour plot the deformed shape is scaled by 100. In the stress contour plot it is easy to notice that the stress results are not ideal due to discontinuities in the mesh. The next step would be to rerun this model with a refined mesh.

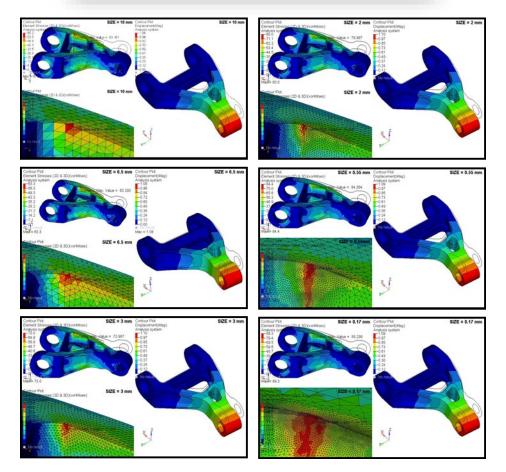


A finer mesh results on the one hand typically in a more accurate solution, on the other hand increases the computation time.

In order to get an idea of a finite element model that is good enough to predict an accurate solution for a problem with a reasonable model size, a convergence study can be performed:

- Create a mesh using low, but reasonable number of elements and do an analysis
- Refine the mesh, do a reanalysis and compare the results for the first mesh.
- Keep refining the mesh and reanalyzing until the results like max. stress and max displacement converge.





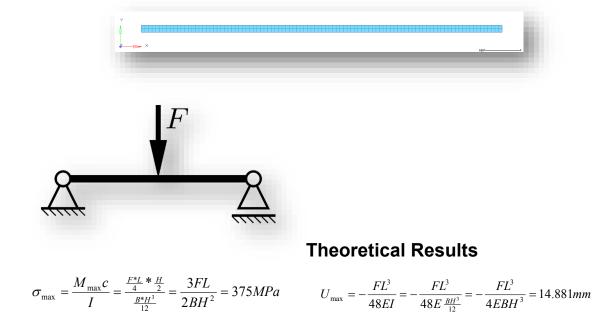
Chapter 2: Linear Static Analysis

Element Size (mm)	VonMises (MPa)	Displacement (mm)
10	60.2	1.06
6.5	63.3	1.08
5	69.5	1.09
3	73.0	1.10
2	80.0	1.10
0.55	84.4	1.09
0.17	89.3	1.09

Exercise 2b: Static Analysis of a Simply Supported Beam

In this exercise, a structural analysis is performed on a simply supported beam. The objective is to create a finite element model that is good enough to predict the theoretical solution for this model.

- Beam modelled by shell elements, length = 1000 mm, height = 20 mm, width = 10 mm
- Material steel (Young's modulus 210000 MPa, Poisson's ratio 0.33, density 7.85e-9 t/mm3)
- Force of 1000 N in the center of the beam.



File Name and Location

...\STUDENT-EXERCISE\2b_Simple_Beam\beam.hm.

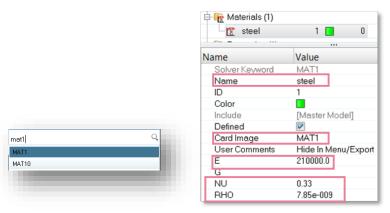
Step 1: Open the model in HyperMesh Desktop with OptiStruct user profile selected

Step 2: Review the model and check the dimensions of the model

Tip: The length system is reasonable to be millimeter, the consistent units are mm, MPa, N. The mesh size is 10 mm. There is one components beam containing elements and surfaces

Step 3: Create a MAT1 material steel for steel with the given properties

Tip: Use HyperMesh's Quick Access Tool (Crtl+f) to create according MAT1 material card



Step 4: Create a PSHELL property shell with a thickness of 10 mm referencing material steel and assign it to the beam component

Tip: Use HyperMesh's Quick Access Tool (Crtl+f) to create according PSHELL property card and assign property to beam component in the entity editor

	🖻 🗞 Properties (1)	
	😓 shell	1 🚺 0
	Name	Value
	Solver Keyword	PSHELL
	Name	shell
	ID	1
	Color	
	Include	[Master Model]
pshell	Defined	
PSHELL	Card Image	PSHELL
- Office	Material	(1) steel
	User Comments	Hide In Menu/Export
	Т	10.0

🗟 💊 Components				
🗆 🗾 🗭 bear	n	1 📃	0	
🛛 🙀 Materials (1)				
🔤 🍸 steel		1 🔲	0	
🕬 💫 Properties (1)				
🛁 😓 shell		1 📘	0	
🗄 🧊 Titles (1)				
				-
lame	Value			
Name	beam			
ID	1			
Color				
Include File	[Maste	er Model]		
Property		Prop	erty	
Material	<unsp< td=""><td>ecified></td><td></td><td></td></unsp<>	ecified>		
Select	t Prope	erty		
Enter Sea	arch Stri	ng		
Nar	ne	ID	Color	Card Ima
shell		1		PSHELL

Step 5: Create a load collector SPC (no card image) with the following SPC load type constraint

- Node 205: DOFs 1-5
- Node 1: DOFs 2-3

Tip: Use HyperMesh's Quick Access Tool (Crtl+f) with SPC to create the two SPCs

spd	୍
SPC	
SPC[CC]	
SPCADD	
SPCD	
SPCF[OR]	

Step 6: Create a load collector FORCE (no card image) containing a force on node 53 with constant components {0, -1000, 0}

Tip: Use HyperMesh's Quick Access Tool (Crtl+f) with FORCE to create the load

Be aware, that loads and constraints can generate singularities. These can lead to a very high stress that is not physical, and appears only in the mathematical model

BCs Create Forces	
ics create r urces	
3Cs Edit Forces	
FORCE	
FORCE1	
Post Free Body Force	
Post Free Body Resultant Force and Moment	

Step 7: Create a load step Load of type Linear Static using SPC as SPC and FORCE as LOAD entry

Tip: Click right mouse button in the Model Browser, select Create > Load Step and set Analysis type to Linear Static

Load Collectors (2 SPC SPC SPC Solver Kerword Name ID Include User Comments Subcase Definition Analysis type	1	0 0 Value SUBCASE Load 1 Master Mod
Sober Keyword Name D Include User Comments	1	0 Value SUBCASE Load 1 (Master Mod
Image: Forme Forme Image: Load Steps (1) Im	1	0 Value SUBCASE Load 1 [Master Mod
Coad Steps (1) Coad Name Sober Kerword Name ID Include User Comments Subcase Definition	1	Value SUBCASE Load 1 [Master Mod
Load Name D Include User Comments		Value SUBCASE Load 1 [Master Mod
Name Sober Keword ID Include User Comments		Value SUBCASE Load 1 [Master Mod
Solver Keyword Name ID Include User Comments		SUBCASE Load 1 [Master Mod
Solver Keyword Name ID Include User Comments		SUBCASE Load 1 [Master Mod
ID Include User Comments Subcase Definition		1 [Master Mod
Include User Comments		
User Comments		
Subcase Definition		
		Hide In Men
		Lincer Static
LOAD		
SPC		Linear Sta (1) SPC (2) FORCE
7		23

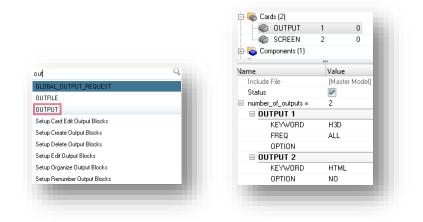
Step 8: Add output requests in order to

• echo .out file on the screen

• get results only in H3D format and suppress the html file output

Tip: Use HyperMesh's Quick Access Tool (Crtl+f) to add control cards

SCREEN OUT OUTPUT, H3D, ALL OUTPUT, HTML,, NO



Step 9: Run the analysis with OptiStruct

Tip: Run the model in OptiStruct using e.g. the OptiStruct panel via pull-down menu *Optimization > OptiStruct*

	Lining Optimization Post Create + Edit + Assign + Delete + Card Edit + Organize + Renumber + OptiStruct OSSmooth		
inputfile: M_3c_Simple_Beam \be export options: run options: ▼ custom V	memory options:	save as	OptiStruct HyperView
include connectors options: - n	ncpu 4 - core in		view.out

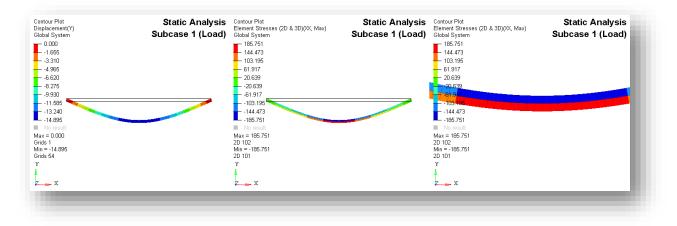
Step 10: Review the .out file wrt warnings, errors and Auto-SPC

Step 11: Review the displacements and stresses in HyperView and check

• max. displacement in y-direction

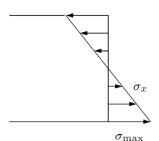
• max. stress in (global) xx-direction

Tip: In the contour plots the deformed shapes are scaled by 10. The maximum displacement is 14.895 mm, the maximum stress is \pm 185.751 MPa (both in the center of the beam).



Step 12: Calculate the theoretical results for

- max. displacement in y-direction
- max. stress in xx-direction



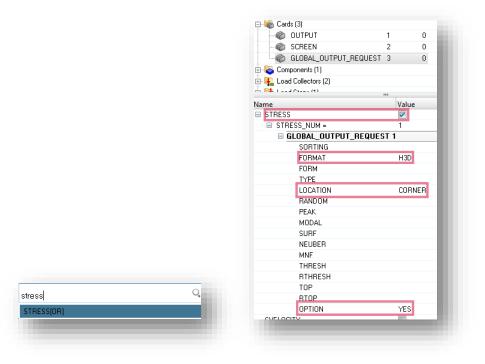
 $u_{\text{max}} = \frac{F l^3}{48 E l} = \frac{F l^3}{48 E \frac{t h^3}{12}} = \frac{F l^3}{4 E t h^3} = \frac{-1000 \cdot 1000^3}{4 \cdot 210000 \cdot 10 \cdot 20^3} = 14.881 \text{ mm} \text{ versus } 14.895 \text{ mm}$ $\sigma_{\text{max}} = \frac{M_{\text{max}} z_{\text{max}}}{l} = \frac{\frac{F l}{4} \left(\pm \frac{h}{2}\right)}{\frac{t h^3}{12}} = \pm \frac{3 F l}{2 t h^2} = \pm \frac{3 \cdot 1000 \cdot 1000}{2 \cdot 10 \cdot 20^2} = \pm 375 \text{ MPa} \text{ versus } \pm 185.751 \text{ MPa}$ The displacement result of the analysis is very good with an error $\sim 0.5\%$. However, the stress results look not good with an error superior to 50%.

But as in OptiStruct element stresses for shell (and solid) elements are output at the element center only, you may not compare OptiStruct's stress result with s at \pm h/2, but \pm h/4:

 $\sigma = \frac{M_{\max}(\pm \frac{h}{4})}{I} = \pm \frac{3 F l}{4 t h^2} = \pm \frac{3 \cdot 1000 \cdot 1000}{4 \cdot 10 \cdot 20^2} = \pm 187.5 \text{ MPa versus } \pm 185.751 \text{ MPa}$

Step 13: Add the global output request STRESS (H3D, CORNER) =YES

Tip: Use HyperMesh's Quick Access Tool (Crtl+f) with STRESS to add this control card/global output request STRESS (H3D, CORNER) =YES



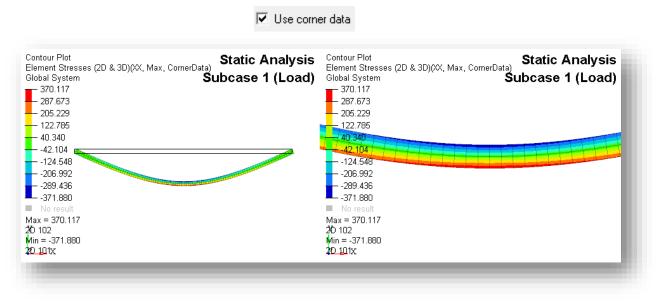
Chapter 2: Linear Static Analysis

Step 14: Rerun the analysis with OptiStruct

Step 15: Review the stresses in HyperView and check

• max. displacement in y-direction max. stress in (global) xx-direction

The maximum stress is ± 370 MPa versus ± 375 MPa (theoretical result). Do not forget to set activate use corner data.



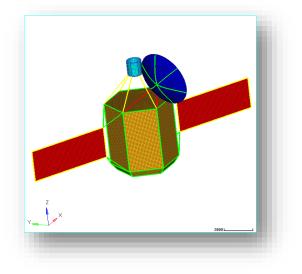
Chapter 3

Inertia Relief Analysis

Chapter 3: Inertia Relief Analysis

Exercise 3a: Satellite Inertia Relief Analysis

This exercise runs an inertia relief load case on a simple satellite. This is a test made with aerospace structures that will need to support inertia loads. The objective of this kind of test is to verify if the structure is strong enough to support these loads without a static failure.



File Name and Location

...\STUDENT-EXERCISE\3a_Satellite\satellite.hm

Chapter 3: Inertia Relief Analysis

Step 1: Open the model in HyperMesh Desktop

Step 2: Review the model and check total mass

Tip: Total mass: 3.090 t

a see based on			_	_				8
volume =	2	8	3	6	е	+	0	9
total mass =				3		0	9	0

Step 3: Set common OUTPUT requests

Use HyperMesh's Quick Access Tool (Crtl+f) to add control cards

SCREEN OUT

OUTPUT, H3D, ALL

OUTPUT, HTML, , NO

BAL_OUTPUT_REQUEST	Name	Value
FILE	Include File	[Master Model]
 TL	Status	
	number_of_outputs =	2
Card Edit Output Blocks	🗆 OUTPUT 1	
eate Output Blocks	KEYWORD	H3D
elete Output Blocks	FREQ	ALL
	OPTION	
Edit Output Blocks	OUTPUT 2	
Organize Output Blocks	KEYWORD	HTML
Renumber Output Blocks	OPTION	NO

Step 4: Set control card PARAM, INREL to -1

Tip: Use HyperMesh's Quick Access Tool (Crtl+f) to set control card PARAM, INREL to -1

Entities ⊕ 🔂 Assembly Hierarchy	ID 😵	Include	
	1)		
🗄 😪 Beam Section Collectors (.U		
🖻 🔞 Cards (3)			
- 🍘 SCREEN	1	0	
CUTPUT	2	0	
PARAM	3	0	
🗄 🛜 Components (14)			
🕀 🙀 Materials (4)			
🗄 💫 Properties (13)			
🗄 间 Titles (1)			
Name	Va	lue	
I64SLV			
INREL	~	1	
INREL_V1	-1		
INTRFACE			
IT A DE	1000		_

Step 5: Create a load collector Support (no card image) with the following SUPORT1 load type constraints representing the fictitious support

- Node 2: DOF 2
- Node 3: DOFs 1-3
- Node 4: DOFs 1-2

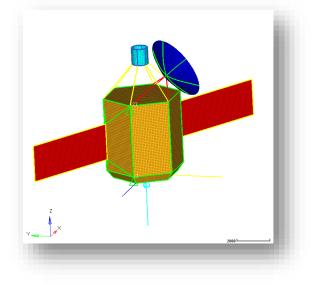
Tip: Switch the Load Type for constraint in HyperMesh from SPC to SUPORT1 before creating the constraints

moment = MOMENT constraint = SUPORT1 pressure = PLOAD4		force =	FORCE
		moment =	MOMENT
pressure = PLOAD4	pressure = PLOAD4	constraint =	SUPORT1
piccould i L o M D i		pressure =	PLOAD4

Step 6: Create the following load collectors (no card image) and forces on Node 1 of the model

• 3x Gx: force components {92700,0,0} (equivalent to three times gravity in x direction)

- 3x Gy: force components {0,92700,0} (equivalent to three times gravity in y direction)
- 2x Gz: force components {0,0,61800} (equivalent to two times gravity in z direction)



Step 7: Create a following load collector 4.7x G with card image LOADADD and select all three above created load collectors with scale factor 1.0

Name	Value	9	
Solver Keyword	LOAI	DADD	
Name	4.7x 0	G	
ID	5		
Color			
Include	[Mas	ter Model]	
Card Image	LOAI	DADD	
User Comments	Hide	In Menu/E>	(port
S	1.0	1.000	
LOAD_Num_Set =	3		D_Num_Set =
Data: S1,		S1	L1
	_		
		1 1.0	(2) 3x Gx
		2 1.0	(3) 3x Gy
		3 1.0	(4) 2× Gz

Step 8: Create four Linear Static load cases using Support as the SUPORT1 entry for each of the four load collectors created before

— 👍 Зх Сх	1	0	
— 👍 🛛 3x Gy	2	0	
	3	0	
📥 4.7x G	4	0	
· · · · · ·			
ame		Value	
Solver Keyword		SUBCASE	
Name		4.7x G	
ID		4	
Include File		[Master Model]	
User Comments		Hide In Menu/Export	
Subcase Definition			
🖃 Analysis type		Linear Static	
SPC		<unspecified></unspecified>	
LOAD		4.7x G (5)	
SUPORT1		Support (1)	
PRETENSION		/Unspecified\	_

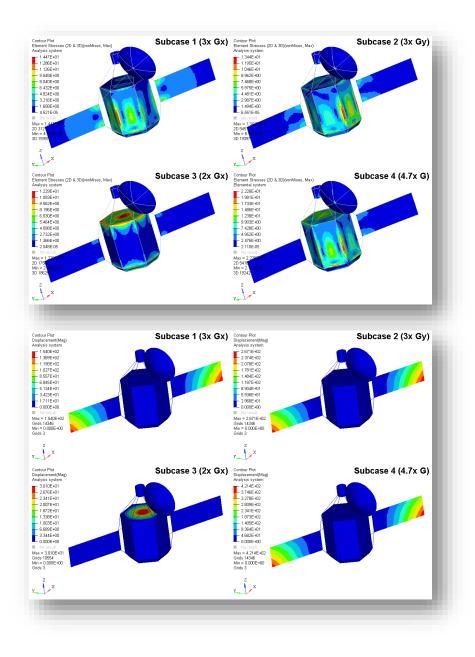
Step 9: Run the analysis with OptiStruct

Step 10: Review the .out file wrt warnings and errors

Step 11: Review the results in HyperView and check if

- Max. relative displacement < 500 mm
- Max. von Mises stress < 70 MPa for 2D modeled components

Tip: Max. relative displacement = 421.4 mm (load case 4.7x G) < 500 mm and Max. von Mises stress = 22.3 MPa (load case 4.7x G) < 70 MPa



Chapter 4 Modal Analysis

Chapter 4: Modal Analysis

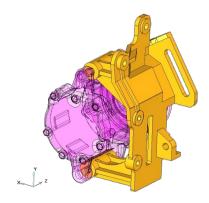
Exercise 4a: Compressor Bracket Modal Analysis

This exercise runs a modal analysis on a compressor system. This is very common problem for an engine designer, who needs to find the best way to link the compressor with the engine. To make this system viable the vibration produced by the engine can't have resonance with the compressor system, and then the key to the project is to develop a bracket that makes the frequencies higher than excitations. Suppose that our 4-cycle engine can work up to 8000 RPM, and then the excitations from the second order (2 explosions per cycle) are up to ~266 Hz.

Then the objective of this project is to have a Bracket with the first frequency higher than 350 Hz.

In this exercise, you will learn how to:

- Determine if a FEA model is well defined
- Understand how well the modal results represent the model



File Name and Location

...\STUDENT-EXERCISE\4a Bracket Compressor\bracket compressor 2nd.hm

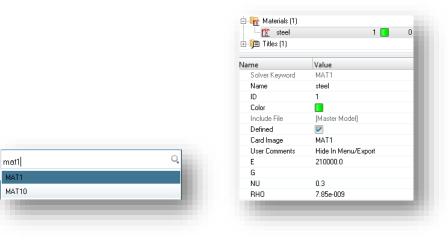
Step 1: Open the model in HyperMesh Desktop

Step 2: Review the model

Tip: The length system is reasonable to be millimeter. There is no representation for the bolts and the compressor. To do this kind of simplification the analyst needs to have knowhow about the system behavior, in general we can assume that the bolt is strong enough to not change the modal result. But the compressor geometry needs to be studied before any simplification. In this case we will add a mass element to represent the compressor.

Step 3: Create a MAT1 material steel for steel with the properties: Young's modulus 210000 MPa, density 7.85E-9 t/mm3, Poisson's ratio 0.3

Tip: Use HyperMesh's Quick Access Tool (Crtl+f) to add according MAT1 material card



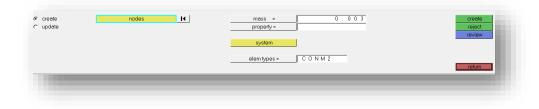
Step 4: Create a PSOLID property bracket referencing material steel and assign it to the bracket component

Tip: Use HyperMesh's Quick Access Tool (Crtl+f) to create property PSOLID and assign it to the bracket component with the right mouse menu

Components (4)	e F2 0 urrent 0 lit e		Assign t Property:	o Components X bracket V bracket ply OK Cancel
	→ Inc 11 → Inc 12 steel → Properties (1) → bracket → bracket		1 0	
	Name Solver Keyword Name ID Color Include File	Value PSOLID bracket 1 [Master Model]		Y
	Defined Card Image Material User Compete CORDM o ISOP	PSOLID	terial	
	PSOLIDX	lame ID eel 1	Color Card	I Image

Step 5: Create a mass element at node 6 (dependent node of the RBE3 element) with value 0.003

Tip: Check that mass is the active collector (marked in bold in the **Model Browser**), and create a CONM2 element (Concentrated Mass Element Connection, Rigid Body Form). You can reach the panel with HM's Quick Access Tool or with the pull-down menu *Mesh* > *Create* > *Masses*



Entities	ID	•	Include
🕀 💫 Assembly Hierarchy			
🖨 🔞 Cards (3)			
🍘 SCREEN	1		0
@ OUTPUT	2		0
GLOBAL_OUTPUT_REQUEST	3		0
🕂 😓 Components (4)			
📁 🖽 bracket	1		0
📁 🖽 RBE3	3		0
📁 🖽 RBE2	2		0
📁 🖽 mass	4		0
🖻 🙀 Materials (1)			
🔤 🍸 steel	1		0
🖻 💫 Properties (1)			

Note that a RBE3 element is used to link the mass element to the bracket. An RBE2 would include a rigid condition between the compressor links that doesn't exist. As optional exercise you can rerun the model with an RBE2 instead and compare the results.

Step 6: Create a load collector SPC (no card image) and with constraints to all five bolt locations RBE2 independent nodes (1-5) for DOF 1-3 each

Tip: Use HyperMesh's Quick Access Tool (Crtl+f) with SPC Note that with these five constraints (DOF 1-3) the engine all is considered to be rigid. It might be that the engine all is thin on the region where the bracket is fixed, and it can be very important on the modal behavior. Here the analyst needs to study the region to make the right assumption

spc	Q,
SPC	
SPC[CC]	
SPCADD	
SPCD	
SPCF[OR]	

Step 7: Create a load collector modal (card image EIGRL) and set the number of desired roots (ND) to 6

Tip: Use HyperMesh's Quick Access Tool (Crtl+f) with EIGRL and set ND to 6

	→ 🗗 🖽 SPC	I	1 📃 2 📃	0 0
	Name	Value		
	Solver Keyword	EIGRL		
	Name	modal		
	ID	2		
	Color			
	Include File	[Master Model]		
	Card Image	EIGRL		
	User Comments	Hide In Menu/Exp	ort	
	V1			
	V2			
	ND	6		
	MSGLVL			
Q	MAXSET			
	SHFSCL			
	NORM	MASS		

Step 8: Create a load step normal modes and reference the two local collectors accordingly

Tip: Set Analysis type to Normal modes in order to reduce the number of Subcase Information Entries

🕂 📬 Load Steps (1)	
🚽 📥 normal modes 🛛 1	0
1	
Name	Value
Solver Keyword	SUBCASE
Name	normal modes
ID	1
Include	[Master Model]
User Comments	Hide In Menu/Export
Subcase Definition	
Analysis type	Normal modes
SPC	(1) SPC
MPC	<unspecified></unspecified>
METHOD (STRUCT)	(2) modal
METHOD (FLUID)	<unspecified></unspecified>
STATSUB (PRELOAD)	<unspecified></unspecified>

Step 9: Set common control cards requests

Tip: Use HyperMesh's Quick Access Tool (Crtl+f) to add control cards SCREEN OUT OUTPUT, H3D, ALL OUTPUT, HTML, , NO Chapter 4: Modal Analysis

Step 10: Request the strain energy results using global output request ESE

Tip: Do the same for ESE

🕀 🏀 Caro	ts (3)		
~~	SCREEN	1	0
~ ©	OUTPUT	2	0
	GLOBAL_OUTPUT_REQUEST	3	0
🗄 🌄 Com	ponents (4)		
🗄 🚱 Loa	d Collectors (2)		
🗄 🔂 Loa	d Steps (1)		
Name			Value
E ESE			
	NULLA		4
🖃 ESE	_NUM =		1
	_NUM = GLOBAL_OUTPUT_REQUES	T 1	1
	-	ST 1	H3D
	- GLOBAL_OUTPUT_REQUES	it 1	H3D
	GLOBAL_OUTPUT_REQUES	it 1	H3D
	GLOBAL_OUTPUT_REQUES FORMAT TYPE	it 1	H3D
	GLOBAL_OUTPUT_REQUES FORMAT TYPE DMIG	it 1	H3D
	GLOBAL_OUTPUT_REQUES FORMAT TYPE DMIG PEAKOUT	ST 1	H3D

Step 11: Run the analysis with OptiStruct

Tip: Run the model in OptiStruct using e.g.the OptiStruct panel via pull-down menu *Optimization* \rightarrow *OptiStruct*

Ī	Optimization) Pi
	Create	F [
1	Edit	۲.
ł	Assign	۶IJ.
I	Delete	•
I	Card Edit	
I	Organize	F.
I	Renumber	۲.
I	OptiStruct	
l	OSSmooth	

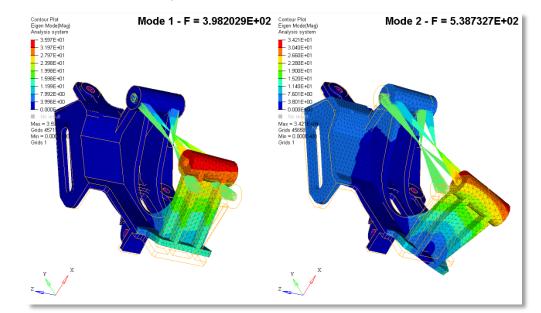
Step 12: Review the .out file wrt warnings and errors and check if f1 > 350 Hz

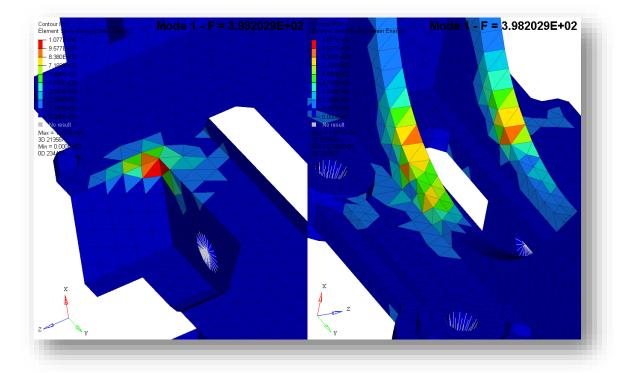
Tip: There are 22 elements that exceeded recommended range (warning) for the element quality check. f1 = 398 Hz > 350 Hz, so the constraint is fulfilled

				Generalized	Generalized
Subcase	Mode	Frequency	Eigenvalue	Stiffness	Mass
1	1	3.982029E+02	6.259916E+06	6.259916E+06	1.000000E+00
1	2	5.387327E+02	1.145794E+07	1.145794E+07	1.000000E+00
1	3	1.142351E+03	5.151795E+07	5.151795E+07	1.000000E+00
1	4	1.540108E+03	9.364018E+07	9.364018E+07	1.000000E+00
1	5	2.053619E+03	1.664943E+08	1.664943E+08	1.000000E+00
1	6	2.363966E+03	2.206186E+08	2.206186E+08	1.000000E+00

Step 13: Review contours of the mode shapes and strain energy in HyperView

Tip: Screenshot shows mode shapes 1 and 2. Strain energy can give to the analyst a very good indication if the mode is well refined or there is need for a mesh refinement. It works like the stress for a static analysis.





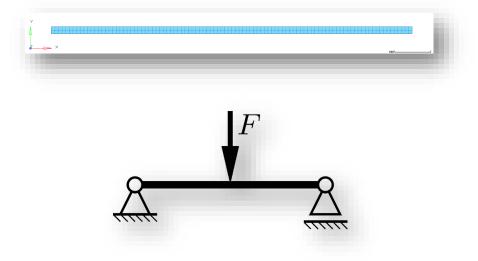
Chapter 4: Modal Analysis

Exercise 4b (optional): Simply Supported Beam Model Analysis

In this exercise, a modal analysis is performed on a simple supported beam from exercise 3c:

- Beam modelled by shell elements, length = 1000 mm, height = 20 mm, width = 10 mm
- Material steel (Young's modulus 210000 MPa, Poisson's ratio 0.33, density 7.85e-9 t/mm3)
- Force of 1000 N in the center of the beam.

The objective is to compare the results of the first three eigen frequencies of the finite element model with the theoretical solution.



File Name and Location

...\STUDENT-EXERCISE\4b_Simple_Beam\beam_modal.hm

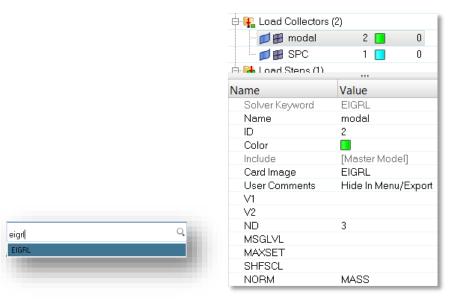
Step 1: Open the model in HyperMesh Desktop with OptiStruct user profile selected

Step 2: Review the model

Step 3: Constrain all nodes additionally in the z-direction in order to get only the shapes in xy-plane

Step 4: Create a load collector modal (card image EIGRL) and set the number of desired roots (ND) to 3}

Tip: Use HyperMesh's Quick Access Tool (Crtl+f) with EIGRL and set ND to 3



Step 5: Create a load step MODAL and reference the two load collectors accordingly

Tip: Set *Analysis type* to Normal modes in order to reduce the number of Subcase Information Entries

🕂 😝 Load Steps (1)	
🚽 🖕 normal modes 🛛 1	0
1 m	
Name	Value
Solver Keyword	SUBCASE
Name	normal modes
ID	1
Include	[Master Model]
User Comments	Hide In Menu/Export
Subcase Definition	
Analysis type	Normal modes
SPC	(1) SPC
MPC	<unspecified></unspecified>
METHOD (STRUCT)	(2) modal
METHOD (FLUID)	<unspecified></unspecified>
STATSUB (PRELOAD)	<unspecified></unspecified>

Step 6: Run the analysis with OptiStruct

Step 7: Review the .out file wrt warnings and errors and check f1 , f2 and f3

Tip: from the .out file:

Subcase	Mode	Frequency
1	1	4.682963E+01
1	2	1.863840E+02
1	3	4.155289E+02

Step 8: Calculate the theoretical results for f1 , f2 and f3

Tip: The equations for simply supported beam shown below

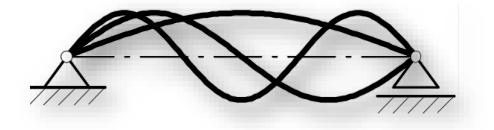
$$f_j = \frac{j^2 \pi}{2} \sqrt{\frac{El}{\rho t h l^4}} = \frac{j^2 \pi}{2} \sqrt{\frac{E h^2}{12 \rho l^4}} = \frac{j^2 h \pi}{4 l^2} \sqrt{\frac{E}{3 \rho}}$$
 for a simply supported beam

 $f_j = \frac{j^2 \cdot 20 \pi}{4 \cdot 10000^2} \sqrt{\frac{210000}{3 x 0.0000000785}}$ Hz = 46.9 j^2 Hz for this example

*f*₁=46.9 Hz versus 46.8 Hz

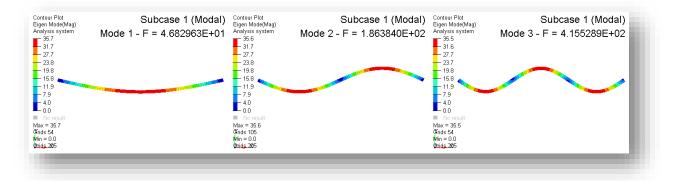
*f*₂=187.6 Hz versus 186.4 Hz

*f*₃=422.2 Hz versus 415.5 Hz



Step 9: Review contours of the three mode shapes HyperView

Tip: In the contour plots the deformed shapes are scaled by 2



Chapter 5

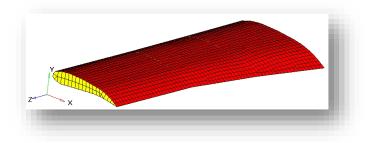
Linear Buckling Analysis

Chapter 5: Linear Buckling Analysis

Exercise 5a: Wing Linear Buckling Analysis

This exercise runs a linear buckling analysis on a simple aircraft wing. This is a typical problem in aerospace structures that need to be very light and consequently become slender. Because the structure has a high slenderness ratio, the buckling failure verification becomes necessary.

The objective of this project is to determine if the 3 static load cases applied to the wing will cause failure, the positive buckling factors should be higher than 1.5.



File Name and Location

...\STUDENT-EXERCISE\5a_Wing\wing.hm

Step 1: Open the model in HyperMesh Desktop

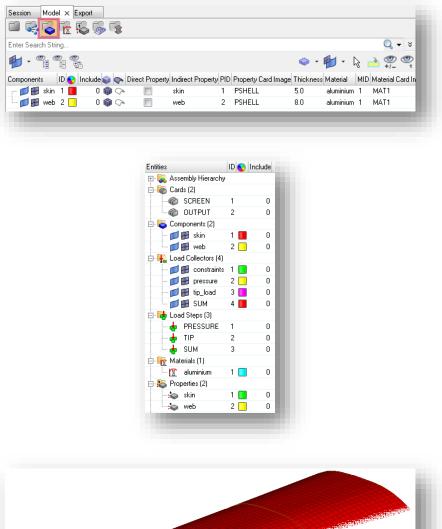
Step 2: Review the model (control cards, components, properties, materials, load collectors & steps)

Tip: To review the model wrt components, properties and materials the component view in the model browser is well suited.

Common control cards requests are set: SCREEN and OUTPUT

There are three static load steps defined:

- (1) pressure on skin
- (2) load on tip
- (3) combination of both using LOADADD card

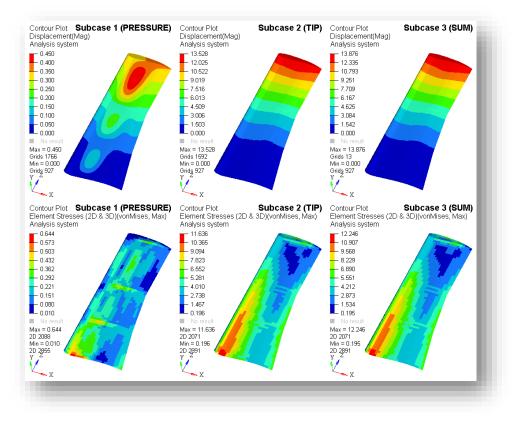




Step 3: Run the static analysis in OptiStruct, review the results in HyperView and check if

- Max. static displacement < 20 mm for all load cases.
- Max. von Mises stress < 70 MPa

Tip: Max. displacement = 13.9 mm (load case sum) < 20 mm Max. von Mises stress = 12.25 MPa (load case sum) < 70 Mpa



Step 4: Create a load collector buckling (card image EIGRL) and set the number of desired roots (ND) to 10 and V1 to 0.001

Tip: Use HyperMesh's Quick Access Tool (Crtl+f) to create an EIGRL card.

Note that with 0.001 as V1 negative buckling factors (i.e. load with opposite direction) are not considered.

		🖨 👫 Load Collectors	
		- 📁 🖽 constra	_
		- 📁 🎛 pressur	e 2 <mark>.</mark> 0
		- 💋 🖪 tip_load	H 3 📘 0
		- 💋 🖪 SUM	4 📕 0
		🚽 🔂 buckli	ng 5 🚺 0
		🗅 🔛 Load Steps (6)	
		Name	Value
		Solver Keyword	EIGRL
		Name	buckling
		ID	5
		Color	
		Include File	[Master Model]
		Card Image	EIGRL
		User Comments	Hide In Menu/Export
		V1	0.001
		V2	
		ND	10
		MSGLVL	
eigrl	Q	MAXSET	
		SHFSCL	
EIGRL		NORM	MASS

Step 5: Create the buckling load steps for each static load step

Tip: As the three new linear buckling load steps only differ by STATSUB entry, it is easier to create the first and use duplicate functionality for the others

Name	Value		
Solver Keyword	SUBCASE		
Name	BUCK_PRESSURE		
ID	4		
Include	[Master Model]		
User Comments	Hide In Menu/Export		
Subcase Definition			
🗏 Analysis type	Linear buckling		
SPC	constraints		
MPC	<unspecified></unspecified>		
STATSUB(BUCKLING)	(1) PRESSURE		
METHOD (STRUCT)	(5) buckling		
DEFORM	<unspecified></unspecified>		
STATSUB (PRELOAD)	<unspecified></unspecified>		
SUBCASE OPTIONS			
LABEL			
SUBTITLE		1.00	
ANALYSIS		Þ 😽	Load Steps (6)
TYPE	BUCK		👍 PRESSURE
EIGVRETRIEVE			👍 TIP
EIGVSAVE			👍 SUM
EXCLUDE			🖕 BUCK_PRESSI
POST			buck_tip
RADSND			BUCK_ALL
RESVEC		1	-
OUTPUT			
SUBCASE_UNSUPPORTED			

🕀 🛃 Loa	ad Steps	(6)			
	PRESS	URE	1	0	
📥	TIP		2	0	
	SUM		3	0	
💠	BUCK_	PRESSURE	4	0	
	BUCK_	TIP	5	0	
	BUC		^	0	
🖻 📴 Ma	toriale	Duplicate		Ctrl + D	
- <u>- 1</u> 2 Ma			Rename		
	nortio	Card Edit			

Name	Value
Solver Keyword	SUBCASE
Name	BUCK TIP
ID	5
Include	[Master Model]
User Comments	Hide In Menu/Export
Subcase Definition	· .
🖃 Analysis type	Linear buckling
SPC	(1) constraints
MPC	<unspecified></unspecified>
STATSUB(BUCKLING)	(2) TIP
METHOD (STRUCT)	(5) buckling
DEFORM	<unspecified></unspecified>
STATSUB (PRELOAD)	<unspecified></unspecified>
SUBCASE OPTIONS	
LABEL	
SUBTITLE	
ANALYSIS	
TYPE	BUCK
EIGVRETRIEVE	
EIGVSAVE	
EXCLUDE	
POST	
RADSND	
RESVEC	
OUTPUT	
SUBCASE_UNSUPPORTED	

Name	Value
Solver Keyword	SUBCASE
Name	BUCK_SUM
ID	6
Include	[Master Model]
User Comments	Hide In Menu/Export
Subcase Definition	
🗏 Analysis type	Linear buckling
SPC	(1) constraints
MPC	<unspecified></unspecified>
STATSUB(BUCKLING)	(3) SUM
METHOD (STRUCT)	(5) buckling
DEFORM	<unspecified></unspecified>
STATSUB (PRELOAD)	<unspecified></unspecified>
SUBCASE OPTIONS	
LABEL	
SUBTITLE	
ANALYSIS	
TYPE	BUCK
EIGVRETRIEVE	
EIGVSAVE	
EXCLUDE	
POST	
RADSND	
RESVEC	
OUTPUT	
SUBCASE_UNSUPPORTED	

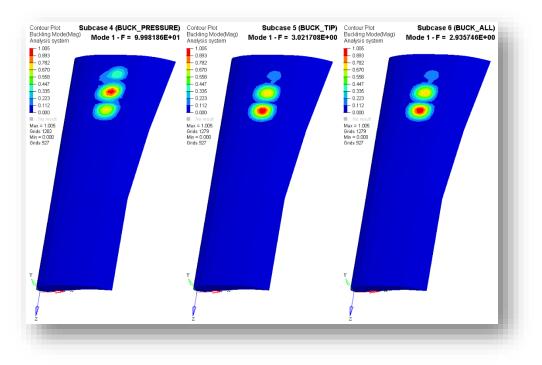
Step 6: Run the analysis with OptiStruct

Step 7: Review the .out file wrt warnings and errors and check if the lowest λ for each buckling subcase is > 1.5

Tip: buckling factors $\lambda_4 = 10.0$, $\lambda_5 = 3.0$, $\lambda_6 = 2.9$, so $\lambda_i > 1.5$

Step 8: Review contours of the buckling modes in HyperView

Tip: Buckling modes 1 for load steps 4-6 shown below



Chapter 5: Linear Buckling Analysis

Chapter 6

Thermal Stress Steady State Analysis

Chapter 6: Thermal Stress Steady State Analysis

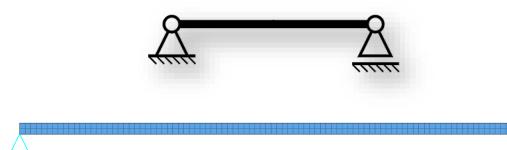
Exercise 6a: Thermal Stress Analysis of a Beam

This exercise runs a thermal stress analysis on a simple supported beam modeled with shell elements, known from exercises 3b and 5b.

The objective is to calculate the deformed shape and maximum displacement due to the SPCs and the following temperature load

Reference temperature of 20

Lower/middle/upper row of nodes has a temperature of 20/30/40



File Name and Location

...\STUDENT-EXERCISE\6a_Simple_Beam\beam.hm

Step 1: Open the model in HyperMesh Desktop

Step 2: Review the model

Step 3: Add the thermal expansion coefficient and a reference temperature to the material card

Tip: As the material is steel, take 1.2e-5 as A. TREF is given as 20

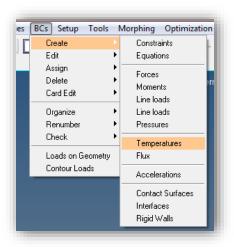
Eroperties (1)	1 🔳
me	Value
Solver Keyword	MAT1
Name	steel
ID	1
Color	
Include	[Master Model]
Defined	
Card Image	MAT1
User Comments	Do Not Export
E	210000.0
G	
NU	0.3
RHO	7.85e-009
A	1.2e-005
TREF	20.0

Step 4: Create a load collector TEMPERATURE (card image TEMPD) with T = 20 and create TEMP loads with a value of 30 and 40 for the middle and upper row of nodes respectively.

Tip: Use HyperMesh's Quick Access Tool (Crtl+f) to create a TEMPD card

	🕂 🧛 Load Collectors	; (3)
	- 💋 🖽 SPC	1 📘
	- 💋 📆 FORCE	2 📕
	- 💋 🔀 TEMP	ERATURE 3 📘
	🕀 🔂 Load Steps (2)	
	🚽 📥 Load	1
	🗏 🖕 Temp	2
	🕂 🙀 Materials (1)	
	🔤 🍸 steel	1 📘
	🕂 🙉 Properties (1)	
	Name	Value
	Solver Keyword	TEMPD*
	Name	TEMPERATURE
	ID	3
	Color	
	Include	[Master Model]
0	Card Image	TEMPD
Q	User Comments	Hide In Menu/Export
	T1	20.0

Due to TEMPD card with T = 20, there is no need to create TEMP loads for the lower row of nodes



j		
Image: state		create create/edit reject review
	Model 🛛 🖬 beam	TEMPERATURE

Ľ <u>Á</u>		Δ
Image: Solution of the second sec		create create/edit reject review
	Model a beam	return

Step 5: Create a load step Temp with SPC referencing SPC load collector and TEMP(LOADCOL) referencing new TEMPERATURE load collector

Tip: Use Generic as Analysis type.

Chapter 6: Thermal Stress Steady State Analysis

				_			
🗄 诸 Lo	ad Steps (2)						
	Load	1		0			
· 📥	Temp	2		0			
ф- 🙀 Ма	aterials (1)						
			_	•			
Name			Value				
Solver	Keyword		SUBCAS	E			
Name			Temp				
ID		2					
Include	e	[Master Model]					
User C	omments		Hide In M	fenu/Export			
🗏 Subca	ase Definition						
🖃 An	alysis type		Generic				
	SPC		(1) SPC				
	LOAD		<unspecified></unspecified>				
	MPC	<unspecified></unspecified>					
	FREQ		<unspecified></unspecified>				
	TEMP (LOADCOL)		(3) TEMF	PERATURE			
	TEMP (SUBCASE)		<unspec< td=""><td>ified></td></unspec<>	ified>			
	METHOD (STRUCT)		<unspec< td=""><td>ified></td></unspec<>	ified>			
	CUBODT4			70° 1.			

Step 6: Run the analysis with OptiStruct, review the results in HyperView and check the maximum displacement

Tip: Maximum displacement is 1.663 (deformed shape inflated by 100)



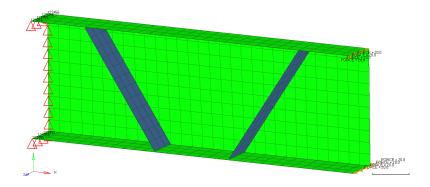
Chapter 7 Advanced Topics

Chapter 7: Advanced Topics

Exercise 7a (optional): Static Analysis using Freeze Contact

This exercise runs a linear static analysis on a simple C beam modeled with shell elements. Additional ribs will be added and connected to the beam using freeze contact.

The objective of this project is to determine without any mesh changes if the additional ribs will reduce the maximum total displacement to below 0.38.



File Name and Location

...\STUDENT-EXERCISE\7a_Beam_Rib\beam_rib.hm

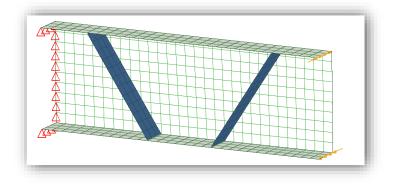
Step 1: Open the model in HyperMesh Desktop

Step 2: Review the model (control cards, components, properties, materials, sets, load collectors & step)

Tip: To review the model wrt components, properties and materials the component view in the **Model Browser** is well suited.

Common control cards requests are set:SCREEN and OUTPUT. There is one static load step defined: analysis There are two sets defined: slaves as grid set (ID 1) master as element set (ID 2)

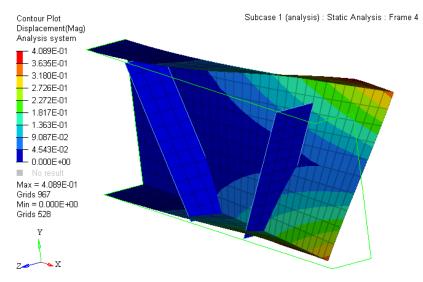
Model × Utility											
	S	r,									
Enter Search String											Q - ×
🚽 - 🖤 🖤	() ()						۲	•	I - 🗟		() +/- 1
Components I	D 💿 Inclu	de 🝙 💿	Direct Property	Indirect Propert	V PID	Property C					
🗾 🗭 beam		0 📦 🖓		p_beam	1	PSHELL		2.0	steel	1	MAT1
	2	0 📦 🔿		p_ribs	2	PSHELL		2.0	steel	1	MAT1
			Entities		1	o 💊 📗					
			🖽 즳 As	sembly Hierarch	ny .						
			🖨 🏀 Ca	ards (2)							
) OUTPUT		4					
) SCREEN	:	3					
				mponents (2)		- 11					
				Beam		1					
				ille iibs		2					
				ad Collectors (2 1 🖽 spc		1 🔳					
				force		2					
				ad Steps (1)							
				analysis		1					
			📄 🙀 Ма			- 1					
			- <u>(</u> 2	steel		1 🛄					
				operties (2)							
) p_beam		1					
				p_ribs		2					
			E 📴 Se								
				slaves master		1					
				Indstei		2					
					-					-	
<u>A</u>		11									
2	Δ				22						
Z	1						M.	- -	-		
4							1				
	$\vec{\mathbf{A}}$										
7	$\overline{\Delta}$					H					
4	\mathbf{A}					If a second seco					
	4					<u> </u>					
<u> </u>				the second	1						
					-	-					
						_			-		



Step 3: Run the static analysis in OptiStruct, review the results in HyperView and check maximum static displacement

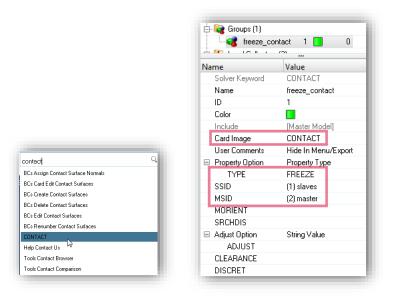
Tip: Max. displacement = 0.4089 (deformed shape inflated by 100)

The two ribs are not connected to the beam



Step 4: Create a group collector freeze_contact (card image CONTACT) with TYPE = FREEZE, SSID = slaves (grid) set (ID 1) and MSID master (element) set (ID 2)

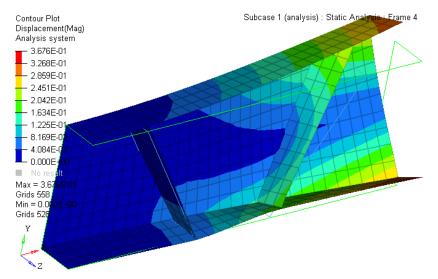
Tip: Use HyperMesh's Quick Access Tool (Crtl+f) to create a CONTACT card



Step 5: Rerun the analysis with OptiStruct, review the results in HyperView and check if maximum static displacement is below 0.38

Tip: Max. displacement = 0.3676 < 0.38 (deformed shape inflated by 100)

The two ribs are connect to the beam

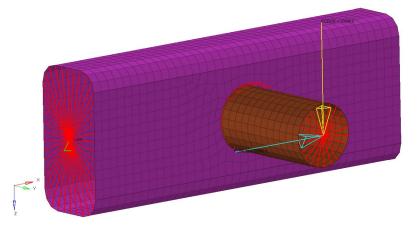


Chapter 8

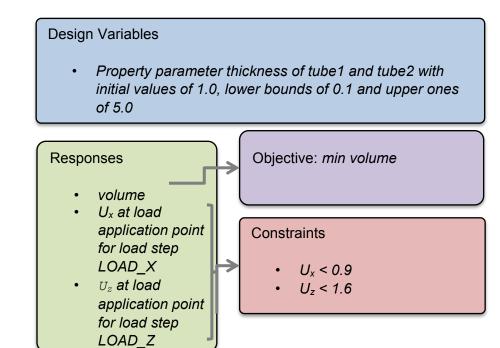
Optimization in Linear Analysis

Exercise 8a: Size Optimization of a Rail Joint

The purpose of this exercise is to set up a property parameter optimization on an automobile rail joint modeled with shell elements. The deflection at the end of the tubular cross-member should be limited. The optimal solution would use as little material as possible



Problem Definition



File Name and Location

...\STUDENT-EXERCISE\8a_Rail_Joint\joint_size.fem

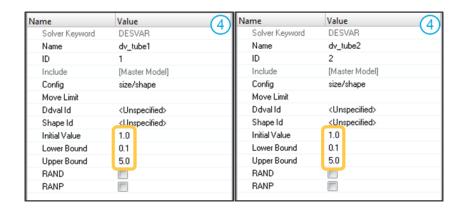
Step 1: Import the model in HyperMesh Desktop with OptiStruct user profile selected

Step 2: Review the model and check the constraints and load steps

Step 3: Run an analysis with OptiStruct and review displacements U_x and U_z according to the later constraints: $U_x = 1.273$ and $U_z = 2.144$

Step 4: Create two size/parameter design variables with initial values of 1.0, lower bounds of 0.1 and upper ones of 5.0, empty (default) move limit and no ddval.

Step 5: Create two generic relationships to link the property parameters thickness to the according design variables



Name	Value 5	Name	Value 5
Solver Keyword	DVPREL1	Solver Keyword	DVPREL1
Name	rel_t1	Name	rel_t2
Include	[Master Model]	Include	[Master Model]
Config	Generic	Config	Generic
Global Ply		Global Ply	
Property Type	PSHELL	Property Type	PSHELL
Property Id	(1) tube1	Property Id	(2) tube2
Property Name	Thickness T	Property Name	Thickness T
Constant	0.0	Constant	0.0
List of Design Variables	1 Designvars	List of Design Variables	1 Designvars
 Number of Design Variables 	1	Number of Design Variables	1
Desvar Coefficients	ÎN	Desvar Coefficients	İS

Step 6: Create a volume response

Name	Value	6
Solver Keyword	DRESP1	U
Name	r_volume	
ID	1	
Include	[Master Model]	
Response Type	volume	
Property	PROP_TOTAL	
Property Type	total	
Region Identifier		
DREPORT		

Step 7: Create a static displacement response in x-direction for the load application point (ID 5555), U_x

Ste 8: Create a static displacement response in z-direction for the load application point (ID 5555), U_y

Name	Value	$\overline{\mathbf{O}}$	Name	Value	0
Solver Keyword	DRESP1	\mathbf{U}	Solver Keyword	DRESP1	U
Name	r_ux		Name	r_uz	
ID	2		ID	3	
Include	[Master Model]		Include	[Master Model]	
Response Type	static displacement		Response Type	static displacement	
Property	PROP_TOTAL		Property	PROP_TOTAL	
List Of Nodes	1 Nodes		List Of Nodes	1 Nodes	
Region Identifier	_		Region Identifier	_	
	dof1			dof3	
COORD			COORD		
DREPORT			DREPORT		

Step 9: Create a constraint for U_x response to be lower than 0.9 for load step LOAD X

Step 10: Create a constraint for U_z response to be lower than 1.6 for load step LOAD_Z

Name	Value	9	Name	Value	(10)
Solver Keyword	DCONSTR	J	Solver Keyword	DCONSTR	U
Name	c_ux		Name	c_uz	
ID	1		ID	2	
Include	[Master Model]		Include	[Master Model]	
Lower Bound			Lower Bound		
Upper Bound	0.9		Upper Bound	1.6	
Response	(2) r_ux		Response	(3) r_uz	
List of Loadsteps	1 Loadsteps		List of Loadsteps	1 Loadsteps	
PROB			PROB		

Step 11: Define the objective function to minimize volume response

Name	Value	(11)
Solver Keyword	DESOBJ(MIN)	-U
Name	objective	
ID	1	
Include	[Master Model]	
Objective Type	Minimize	
Response Id	(1) r_volume	

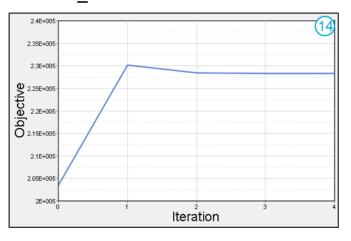
Step 12: Export and review .fem file wrt optimization cards

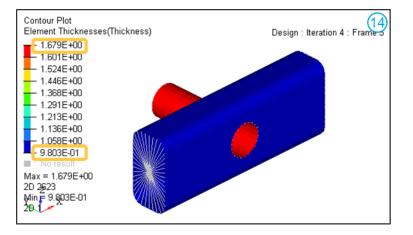
						\sim
DESOBJ(MIN)=1						(12)
SUBCASE	1					<u> </u>
LABEL Force	X					
SPC =	1					
LOAD =	2					
DESSUB =	3					
SUBCASE	2					
LABEL Force						
SPC =	1					
LOAD =	3					
DESSUB =	4					
Ş						
BEGIN BULK						
DESVAR	ldv_tuk	pe11.0	0.1	5.0		
DESVAR	2dv_tuk	pe21.0	0.1	5.0		
DVPREL1 1	PSHELI	J	1	4	0.0	
+ 1	1.0					
DVPREL1 2	PSHELI	J	2	4	0.0	
+ 2	1.0					
DRESP1 1	r_volu	IMEVOLUM	Ξ			
DRESP1 2	r_ux	DISP			TX	5555
DRESP1 3		DISP			ΤZ	5555
DCONSTR	1	2	0.9			
DCONSTR	2		1.6			
DCONADD	3	1				
DCONADD	4	2				
[]						
PSHELL	1	11.0		1	1	0.0
PSHELL	2	10.8		1	1	0.0

Step 13: Run optimization with OptiStruct, review .out file

ITERATION 4 the 2nd satisfied convergence ratio = 2.7998E-05
Objective Function (Minimize VOLUM) = 2.28294E+05 % change = -0.00 Maximum Constraint Violation % = 0.92458E-01
Volume = 2.28294E+005 Mass = 0.00000E+000
Subcase Compliance Epsilon 1 3.534265E+03 -1.106546E-14 2 8.007397E+03 1.515182E-13
Note : Epsilon = Residual Strain Energy Ratio.
RETAINED RESPONSES TABLE
Response Type Response Subcase Grid/ DOF/ Response Objective Viol. User-ID Label /RANDPS Element/ Comp Value Reference/ %
User-ID Label /RANDPS Element/ Comp Value Reference/ % /Model MID/PID/ /Reg Constraint
+Frqncy Mode No. Bound /Times
1 VOLUM r_volume TOTL 2.283E+05 MIN
1 VOLUM r_volume TOTL 2.283E+05 MIN 2 DISPL r_ux 1 5555 TX 7.069E-01 < 9.000E-01 0.0 3 DISPL r_uz 2 5555 TZ 1.601E+00 < 1.600E+00 0.1 A
3 DISPL r_uz 2 5555 TZ 1.601E+00 < 1.600E+00 0.1 A
Design Design Lower Design Upper Variable Variable Bound Variable Bound ID Label
1 dv_tube1 1.000E-01 9.803E-01 5.000E+00 2 dv_tube2 1.000E-01 1.679E+00 5.000E+00
DESIGNED PROPERTY ITEMS TABLE DVPREL1/2 USER-ID PROP-TYPE PROP-ID ITEM-CODE DVPREL1 1 PSHELL 1 T DVPREL1 2 PSHELL 2 T OPTIMIZATION HAS CONVERGED.
FEASIBLE DESIGN (ALL CONSTRAINTS SATISFIED).

Step 14: Review results of _hist.mvw and .mvw files





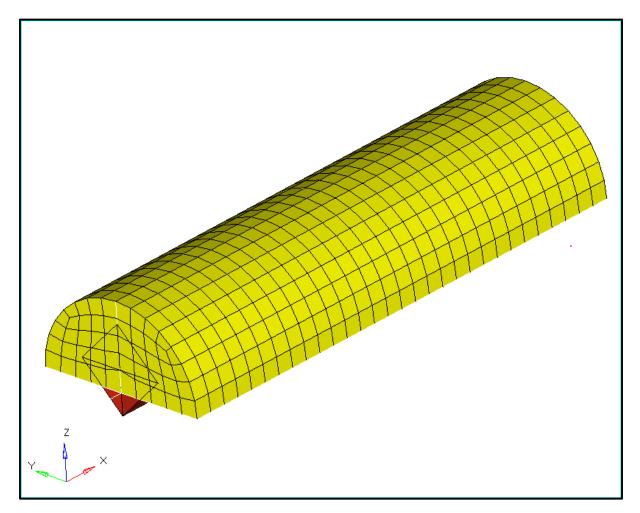
Step 15: Import .prop file in HyperMesh Desktop

Ş											TAR
Ş	PROPE	RTI	ES .	AND	MAT	ERIA	LS	AT I	ITERA:	TION	(1)
Ş											
PS	HELL,	1,	1,	.98	3030	8714	61	1,	1.0,	1,,	0.0
PS	HELL,	2,	1,	1.6	5789	6409	58	1,	1.0,	1,,	0.0

Exercise 8b: Size Optimization of a Shredder with Modal Loadcases

A size optimization involves the changing of the properties of either 1D or 2D elements. These properties include area, moments of inertia of the 1D elements, and the thickness of 2D elements and is performed when it is not necessary to remove materials, generate beads or change the shape of the structure. Size optimizations are highly flexible, able to modify cross-sectional properties of one-dimensional elements, material properties, thicknesses of shell and composite elements, and other selected card entities. Values for such entities are changed within given bounds to meet the necessary objective. Properties are linked with design variables (DESVAR) using DVPREL cards.

This exercise goes through the steps involved in defining a size optimization for a model comprised of shell and bar elements. You will update the PBARL property to simulate the properties of the bar elements and then link that to the design variable. The resulting design will have higher frequencies and updated element properties.



Problem Setup

You should copy this file: shredder.hm

Step 1: Open the file shredder.hm in HyperMesh Desktop and enter the Optimization View of the Model Browser

Name	Value
Solver Keyword	DESVAR
Name	cover_thk
ID	1
Include	[Master Model]
Config	size/shape
Move Limit	
Ddvalld	<unspecified></unspecified>
Shape Id	<unspecified></unspecified>
Initial Value	3.0
Lower Bound	0.5
Upper Bound	8.0
RAND	
RANP	

Step 2: Create the size design variables for the shredder components

Name	Value
Solver Keyword	DESVAR
Name	hat_height
ID	2
Include	[Master Model]
Config	size/shape
Move Limit	
Ddval Id	<unspecified></unspecified>
Shape Id	<unspecified></unspecified>
Initial Value	100.0
Lower Bound	75.0
Upper Bound	125.0
RAND	
RANP	

Name	Value
Solver Keyword	DESVAR
Name	hat_thick
ID	3
Include	[Master Model]
Config	size/shape
Move Limit	
Ddval Id	<unspecified></unspecified>
Shape Id	<unspecified></unspecified>
Initial Value	10.0
Lower Bound	3.0
Upper Bound	12.0
RAND	
BANP	

DESVAR hat_width 4 [Master Model] size/shape
4 [Master Model]
[Master Model]
size/shape
<unspecified></unspecified>
<unspecified></unspecified>
60.0
40.0
80.0

Name	Value
Solver Keyword	DESVAR
Name	hat_flange
ID	5
Include	[Master Model]
Config	size/shape
Move Limit	
Ddval Id	<unspecified></unspecified>
Shape Id	<unspecified></unspecified>
Initial Value	30.0
Lower Bound	1.0
Upper Bound	40.0
RAND	
RANP	

Step 3: Create relationships between the design variables and properties

Tip: Create > Design Variable Relationships > Generic

Name	Value	Select Designvars	×
Solver Keyword	DVPREL1	C) beleet besigning	~
Name	cover		Q, ~
Include	[Master Model]		-0
Config	Generic	Name	ID
Global Ply		🔽 cover_thk	1
Property Type	PSHELL	hat_height	2
Property Id	(2) cover	hat_thick	3
Property Name	Thickness T	hat_width	4
Constant	0.0	hat_flange	5
List of Design Variables	1 Designvars		
Number of Design Variables	1		1 selected.
Desvar Coefficients		ОК	Cancel
			Cancer
Name	Value	Select Designvars	×
Solver Keyword	DVPREL1	C) beleet besigning	~
Name	DIM_hat_height		Q, ~
Include	[Master Model]		-0
Config	Generic	Name	ID
Global Ply		🔲 cover_thk	1
Property Type	PBARL	🔽 hat_height	2
Property Id	(1) frame2	hat_thick	3
Property Name	Dimension 1	hat_width	4
Constant	0.0	hat_flange	5
List of Design Variables	1 Designvars		
Number of Design Variables	1		1 selected.
Desvar Coefficients		OK	Cancel
Name	Value	🛆 Select Designvars	×
Solver Keyword	DVPREL1		
Name	DIM_hat_thick		\bigcirc \sim
Include	[Master Model]		
Config	Generic	Name	ID
Global Ply		cover_thk	1
Property Type	PBARL	hat_height	2
Property Id	(1) frame2	hat_thick	3
Property Name	Dimension 2	🔲 hat_width	4
Constant	0.0	🔲 hat_flange	5
List of Design Variables	1 Designvars	HU HI UN	1.1.1.1
Number of Design Variables	1		1 selected.
Desvar Coefficients		ОК	Cancel

Name	Value	Select Designvars X
Solver Keyword	DVPREL1	
Name	DIM_hat_width	Q, ~
Include	[Master Model]	
Config	Generic	Name ID
Global Ply		cover_thk 1
Property Type	PBARL	hat_height 2
Property Id	(1) frame2	hat_thick 3
Property Name	Dimension 3	🔽 hat_width 4
Constant	0.0	nat_flange 5
List of Design Variables	1 Designvars	
Number of Design Variables	1	1 selected
Desvar Coefficients		OK Cancel
Name	Value	∧ Select Designvars X
Name Solver Keuword	Value DVPREL1	Select Designvars X
Name Solver Keyword Name	DVPREL1	
Solver Keyword Name	DVPREL1 DIM_hat_flange	Select Designvars X
Solver Keyword	DVPREL1	
Solver Keyword Name Include	DVPREL1 DIM_hat_flange [Master Model]	Q ~
Solver Keyword Name Include Config Global Ply	DVPREL1 DIM_hat_flange [Master Model]	Name ID
Solver Keyword Name Include Config Global Ply Property Type	DVPREL1 DIM_hat_flange [Master Model] Generic	Name ID cover_thk 1
Solver Keyword Name Include Config Global Ply Property Type Property Id	DVPREL1 DIM_hat_flange [Master Model] Generic PBARL	Name ID cover_thk 1 hat_height 2
Solver Keyword Name Include Config Global Ply Property Type	DVPREL1 DIM_hat_flange [Master Model] Generic PBARL (1) frame2	Name ID cover_thk 1 hat_height 2 hat_thick 3
Solver Keyword Name Include Config Global Ply Property Type Property Id Property Name Constant	DVPREL1 DIM_hat_flange [Master Model] Generic PBARL (1) frame2 Dimension 4 0.0	Name ID cover_thk 1 hat_height 2 hat_thick 3 hat_width 4 hat_flange 5
Solver Keyword Name Include Config Global Ply Property Type Property Id Property Name	DVPREL1 DIM_hat_flange [Master Model] Generic PBARL (1) frame2 Dimension 4	Name ID cover_thk 1 hat_height 2 hat_thick 3 hat_width 4

Step 4: Create three responses to track the mass and third and fourth modes

Name	Value	
Solver Keyword	DRESP1	
Name	mass	
ID	1	
Include	[Master Model]	
Response Type	mass	
Property	PROP_TOTAL	
Property Type	total	
Region Identifier		
DREPORT		

Name	Value
Solver Keyword	DRESP1
Name	freq_mode_3
ID	2
Include	[Master Model]
Response Type	frequency
Property	LOADSTEPS
Region Identifier	
Mode Number:	3
Subcase Mode Tracking	
DREPORT	
	¥ 1
Name	Value
Solver Keyword	DRESP1
Solver Keyword Name	DRESP1 freq_mode_4
Solver Keyword Name ID	DRESP1 freq_mode_4 3
Solver Keyword Name ID Include	DRESP1 freq_mode_4 3 [Master Model]
Solver Keyword Name ID Include Response Type	DRESP1 freq_mode_4 3 [Master Model] frequency
Solver Keyword Name ID Include Response Type Property	DRESP1 freq_mode_4 3 [Master Model]
Solver Keyword Name ID Include Response Type Property Region Identifier	DRESP1 freq_mode_4 3 [Master Model] frequency
Solver Keyword Name ID Include Response Type Property	DRESP1 freq_mode_4 3 [Master Model] frequency
Solver Keyword Name ID Include Response Type Property Region Identifier	DRESP1 freq_mode_4 3 [Master Model] frequency LOADSTEPS

Step 5: Create two constraints: a lower bound for the 4th mode of 6 Hz and an upper bound for the mass of 1.8

Name	Value				
Solver Keyword	DCONSTR				
Name	C_freq_mode_4				
ID	1				
Include	[Master Model]				
Response	(3) freq_mode_4				
List of Loadsteps	1 Loadsteps				
Lower Options					
Lower Options	Lower bound				
Lower Bound	6.0				
Upper Options	<off></off>				
PROB					

Name	Value				
Solver Keyword	DCONSTR				
Name	C_mass				
ID	2				
Include	[Master Model] (1) mass <off></off>				
Response					
Lower Options					
Upper Options					
Upper Options	Upper bound				
Upper Bound	1.8				
PROB					

Step 6: Create the objective to max the f3 response for the ld1 loadstep

Name	Value				
Solver Keyword	DESOBJ(MAX)				
Name	Objective				
ID	1				
Include	[Master Model]				
Objective Type	Maximize				
Response Id	(2) freq_mode_3				
Loadstep Id	(1) ld1				

Step 7: Run the optimization in OptiStruct

628	DVPREL1	/2 USE	R-ID	PROP-TYPE	PROP-ID	ITEM-COD	E PROP-VALUE	
629								
630	DVPREL1		1	PSHELL	-	т		
631	DVPREL1		2	PBAR	1	DIM1	1.250E+02	
632	DVPREL1		3	PBAR	1	DIM2	1.153E+01	
633	DVPREL1		4	PBAR	1	DIM3	8.000E+01	
634	DVPREL1		5	PBAR	1	DIM4	4.000E+01	
635								
636								
637	ITERATIO	V 7						
638	TRACKED	O OBJEC	TIVE F	UNCTION/CO	NSTRAINT	MODE # 4	GOES OUT OF	BOUND,
639	RETURN	ер то т	THE PRE	VIOUS GOOD	DESIGN.			
640								
641	Objective	e Funct	ion (M	laximize FR	EQ) = (5.34153E+0	0 % change	= 0.00
642	Maximum (Constra	int Vi	olation %	= (0.0000E+0	0	
643	Volume				= 2	.33075E+00	8 Mass	= 1.79468E+000
644								
645	Subcase	Mode	Order	• Weigh	t Fi	requency	Eigenvalue	Weight/Eigen
646	1	1	1	1.000E+	00 1.03	31023E+00	4.196590E+0	1 2.382887E-02
647	1	2	2	0.000E+	00 1.5	34846E+00	9.300133E+0	1 0.00000E+00
648	1	3	3	0.000E+	00 6.34	41527E+00	1.587623E+0	3 0.00000E+00
649	1	4	10	0.000E+	00 8.14	45025E+00	2.619055E+0	3 0.00000E+00
650	1	5	4	0.000E+	00 7.3	58200E+00	2.137484E+0	3 0.00000E+00
651	1	6	5	0.000E+	00 7.3	58540E+00	2.137682E+0	3 0.00000E+00
652	1	7	6	0.000E+	00 7.3	59955E+00	2.138504E+0	3 0.00000E+00
653	1	8	7	0.000E+	00 7.30	51666E+00	2.139498E+0	3 0.00000E+00
654	1	9	8	0.000E+	00 7.3	53115E+00	2.140341E+0	3 0.00000E+00
655	1	10	9	0.000E+	00 7.42	23498E+00	2.175590E+0	3 0.00000E+00
656								
657	~ implies	s mode	is amb	iguouslv t	racked	* implie	s mode cannot	be tracked.
658								

Step 8: Post-process the results by viewing the shredder.out and shredder.prop files

Tip: Some information about the design history can be reviewed visually by animating a contour of the thickness results in HyperView. Since most of the optimization in the model was performed on 1D element properties, the *.prop file is the best way to examine the final results.

```
3 $ PROPERTIES AND MATERIALS AT ITERATION 7
4 $ ------
5 $
6 $HMNAME PROP 1"frame2" 3
7 $HWCOLOR PROP 1 38
8 PBARL 1 1 HAT
9 +, 125.0, 11.525224906, 80.0, 40.0, 0.0
10 $ A=4459.681 I1=8799222. I2=7028952. I12= 0.0 J=197461.0 K1=.5269398 K2=.3539209
11 $ C1~F2 = + 65.87983 40.0-59.1202 80.0-59.1202 -80.065.87983 -40.0
12 $M1A~N2B= + 0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0
13 $HMNAME PROP 2"cover" 4
14 $HWCOLOR PROP 2 3
15 PSHELL, 2, 1, .63122871619, 1, 1.0, 1, .833333, 0.0
```

Since the *.prop file contains the optimization results in OptiStruct FEM format, it can be imported over an existing model to replace the property information in that file, allowing easy update of models to the latest optimization results.