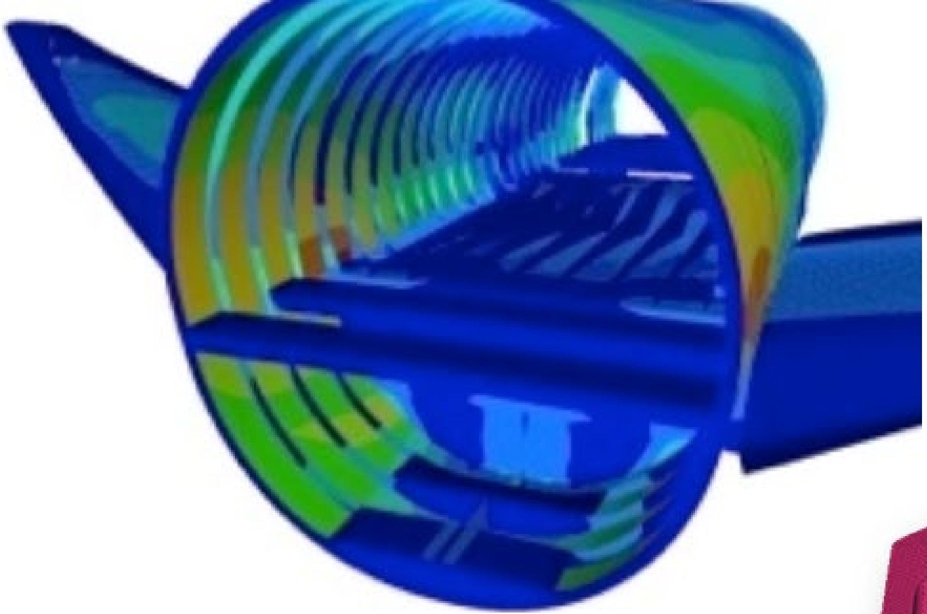
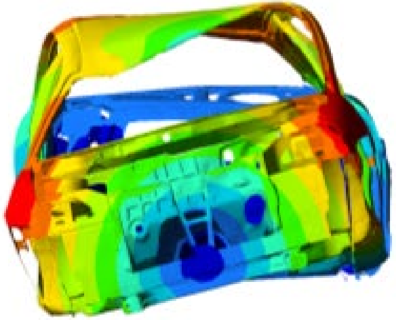
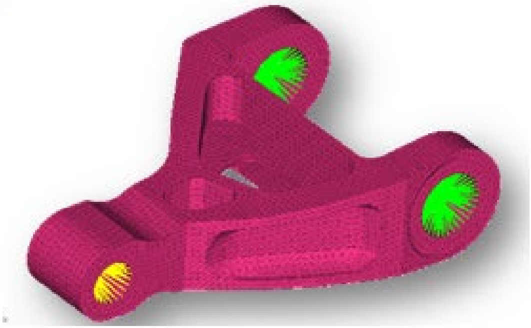






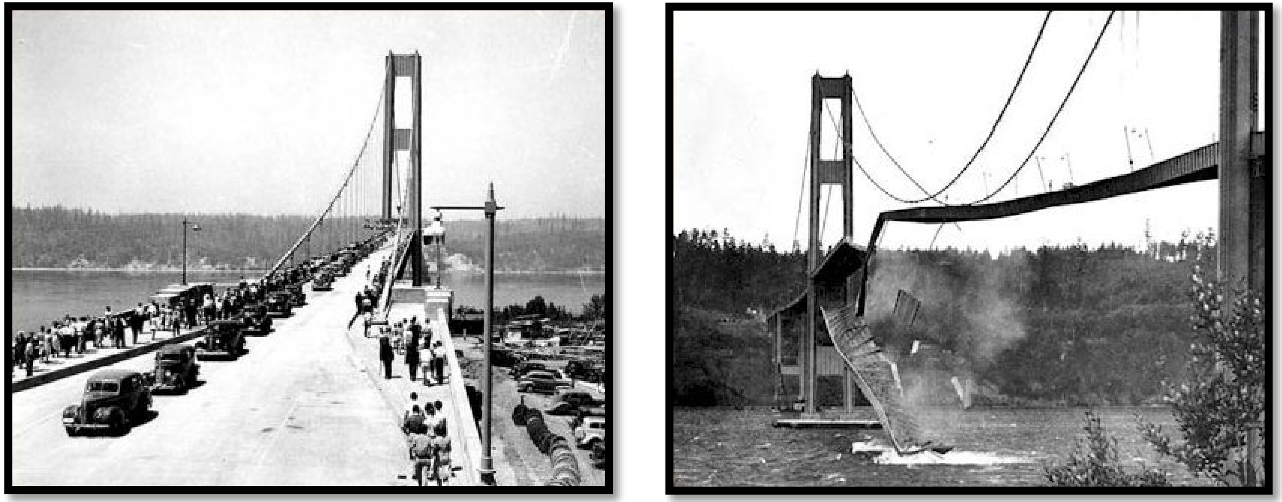
OPTISTRUCT FOR LINEAR ANALYSIS, V2019
CHAPTER 4: MODAL ANALYSIS

AGENDA

1. Introduction to Linear Analysis
   - Type of Analysis
   - Type of Elements and Materials
   - Type of Loads & Boundary Conditions
2. Linear Static Analysis
3. Inertia Relief Analysis
4. **Modal Analysis**
5. Linear Buckling Analysis
6. Thermal Stress Steady State Analysis
7. Advanced Topics
   - Debugging Guide
   - Parameters
   - Transitioning Elements
   - Introduction to Parallelization
   - Run Options
   - Output Management
8. Optimization in Linear Analysis
   - OptiStruct Optimization
   - DRCO Approach
   - Setting up Optimization
   - Optimization Responses for Linear Analysis

NORMAL MODES ANALYSIS

Normal Modes analysis is performed when you are interested in the natural frequencies and the mode shapes of the structure.



Tacoma Narrows Bridge in Washington

DEFINITIONS

Modal analysis calculates the **frequency modes** or **natural frequencies** of a given system, but not necessarily its full time history response to a given input.

The natural frequency of a system is dependent only on the stiffness of the structure, and the mass which participates with the structure (including self-weight) and the boundary conditions.

$$\mathbf{M}\ddot{\mathbf{x}} + \mathbf{K}\mathbf{x} = 0$$

$$\left(\mathbf{K} - \omega^2 \mathbf{M}\right)\Phi = 0$$

with $\mathbf{x} = \Phi \sin(\omega t)$

Φ: Mode shape or Eigenvector

ω: Circular natural frequency

WHY MODAL ANALYSIS IS IMPORTANT

For any kind of structural simulation, a modal analysis will help the engineer to understand the global behavior of the system.

An initial modal analysis helps analysts to

- identify the natural frequencies and modal shapes of the system
- verify if there are rigid modes on the system, and the link between components
- understand if the BCs applied to the system are correct
- determine where the part should be reworked to improve the performance (if strain energy results are requested/available)
- help predict the dynamic responses that this system will have, then all the other dynamic simulations should be done only after a modal analysis

WHY MODAL ANALYSIS IS IMPORTANT

Questions that illustrate the importance of modal analysis:

- Is there resonance in the structure?
- Is the issue a static or a dynamic problem?
- If it is dynamic, which type of analysis should I do?
- How can I calibrate my CAE model with a test?

The modal analysis is designed to determine the normal modes and normal shapes, but can also help with characterizing the system and guide other dynamic analysis.

EIGENVALUE SOLUTION METHODS

There are several mathematical methods for extracting eigenvalues:

- Vector Iteration Methods
- Transformation Methods
- Polynomial Iterations Methods
- Lanczos Iteration Method
- Subspace Iteration Method

OptiStruct offers two options to solve eigenvalue problems

- Lanczos Iteration Method → `EIGRL` card
- Automatic Multi-level Sub-structuring Eigensolver Solution (AMSES) → `EIGRA` card

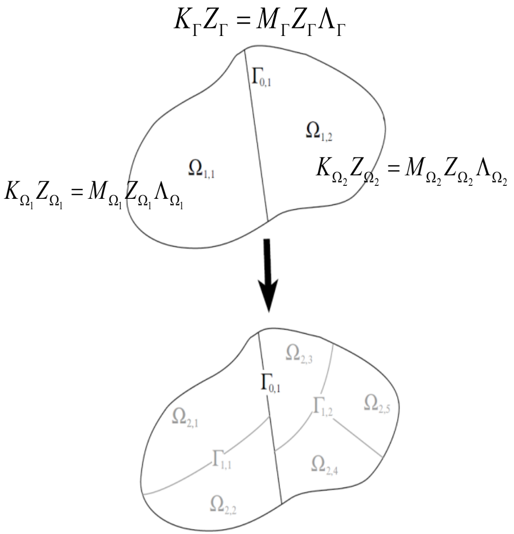
EIGENVALUE SOLUTION METHODS

`EIGRL` (Lanczos)

- No approximation of the eigenvectors
- Full eigenvector calculation
- Leads to large run times for large problems
- Limited to about 2000 modes

`EIGRA` (AMSES)

- Approximate solution
- Accurate enough for NVH
- Partial eigenvector calculation
- Efficient reduction to modal space
- No additional cost over `EIGRL` (both are 25 HyperWorks Units)



BULK DATA CARDS: MODAL ANALYSIS

The `EIGRL` card is the main card used in solving normal modes analyses.

For normal modes analysis, a number of frequencies or a frequency range is required

For linear buckling analysis, a number of eigenvalues or an eigenvalue range is required

Both requirements are met through the use of the real eigenvalue extraction (`EIGRL`) card

```
BEGIN BULK
PARAM,AUTOSPC,YES
PARAM,CHECKEL,YES
$  SYSTEM Data
CORD2R         1       20.0    -20.0   0.0     20.0    -20.0   -100.0
+       -80.0   -20.0   0.0
$
GRID           1       50.0    -50.0   0.0
GRID           2       50.0    -40.0   0.0
$
CBAR         101       1       94      550.0   1.0     0.0
$
CQUAD4         1       1       55      58      59      54
CQUAD4         2       1       54      59      50      51
CQUAD4         3       1       58      43      44      59
$
PSHELL         1      11.0             1               1       0.0
$
PBEAM          1      178.53975490.8734490.87340.0     981.7469
$
MAT1           1210000.0       0.3     7.90E-09
$
EIGRL          3                       20                              MASS
$
FORCE          2      11       01.0    0.0     0.0     -100.0
$
SPC            1      31    1234560.0
ENDDATA
```

BULK DATA CARDS: MODAL ANALYSIS

(1)	(2)	(3)	(4)	(5)	(6)	(7)	(8)	(9)	(10)
EIGRL	**SID**	**V1**	**V2**	**ND**	**MSGLVL**	**MAXSET**	**SHFSCL**	**NORM**	

SID	Unique set identification number. No default (Integer > 0)
V1,V2	For vibration analysis: Frequency range of interest For buckling analysis: Eigenvalue range of interest See comments 3, 4, and 10. Default = blank (V1 < V2, Real, or blank)
ND	Number of roots desired. See comments 3 and 4. No default (Integer > 0 or blank)
MSGLVL	Diagnostic level. Default = 0 (Integer 0 through 4 or blank)
MAXSET	Number of vectors in block or set. Default = 8 (Integer 1 through 16 or blank)
SHFSCL	Vibration: Estimate of the frequency of the first flexible mode. Buckling: Estimate of the first eigenvalue. Default = blank (Real or blank)
NORM	Method used for eigenvector normalization. See OptiStruct Reference Guide for more information

```
BEGIN BULK
PARAM,AUTOSPC,YES
PARAM,CHECKEL,YES
$  SYSTEM Data
CORD2R         1        20.0    -20.0   0.0     20.0    -20.0   -100.0
+       -80.0   -20.0   0.0
$
GRID           1        50.0    -50.0   0.0
GRID           2        50.0    -40.0   0.0
$
CBAR         101       1      94      550.0     1.0     0.0
$
CQUAD4         1       1      55      58      59      54
CQUAD4         2       1      54      59      50      51
CQUAD4         3       1      58      43      44      59
$
PSHELL         1       11.0            1               1        0.0
$
PBEAM          1       178.53975490.8734490.87340.0     981.7469
$
MAT1           1210000.0        0.3     7.90E-09
$
EIGRL          3                      20                             MASS
$
FORCE          2      11       01.0     0.0     0.0     -100.0
$
SPC            1      31  1234560.0
ENDDATA
```

HOW TO SETUP A MODAL ANALYSIS

Normal modes analyses can be defined in six steps:

Step 1 – Generate a FEM model

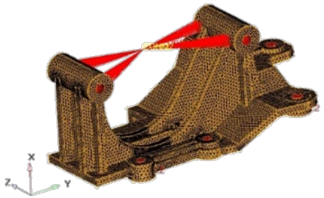
Step 2 – Set up material and properties

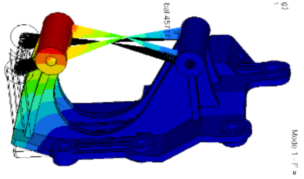
Step 3 – Define the constraint load collector and the boundary conditions.

Step 4 – Define the `EIGRL` card

Step 5 – Define the normal modes load step

Step 6 – Run and post process the results





Name	Value
Name	SPC
ID	1
Color	
Include File	[Master Model]
Card Image	<None>

Name	Value
Solver Keyword	EIGRL
Name	EIGRL
ID	2
Color	
Include File	[Master Model]
Card Image	EIGRL
User Comments	Hide In Menu/Export
V1	
V2	
ND	12
MSGLVL	
MAXSET	
SHFSCL	
NORM	MASS

Name	Value
Solver Keyword	SUBCASE
Name	Normal Modes
ID	1
Include File	[Master Model]
User Comments	Hide In Menu/Export
Subcase Definition	
Analysis type	Normal modes
SPC	SPC (1)
MPC	<Unspecified>
METHOD (STRUCT)	EIGRL (2)
METHOD (FLUID)	<Unspecified>
STATSUB (PRELOAD)	<Unspecified>
Subcase Options	
LABEL	☐
SUBTITLE	☐
ANALYSIS	☑
TYPE	MODES
EIGVRETRIEVE	☐
EIGVSAVE	☐
EXCLUDE	☐
RADSND	☐
RESVEC	☐
OUTPUT	☐
SUBCASE_UNSUPPORTED	☐

EXERCISE 4A: COMPRESSOR BRACKET MODAL ANALYSIS

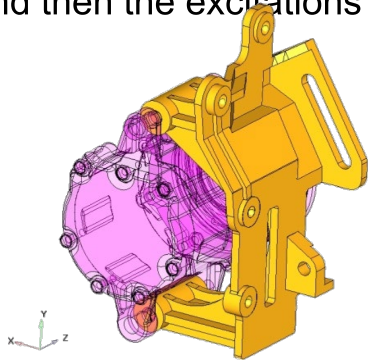
File Name and Location

`…\STUDENT-EXERCISE\4a_Bracket_Compressor\bracket_compressor_2nd.hm`

Objectives (1/2)

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.

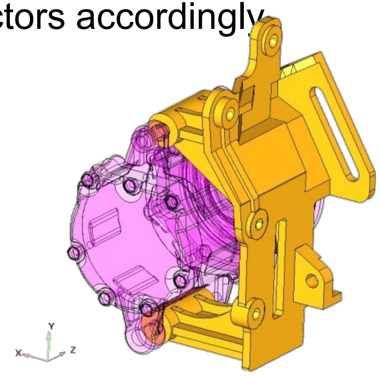


1. Open the model in HyperMesh Desktop
2. Review the model
3. Create a `MAT1` material `steel` for steel with the properties:
   Young's modulus 210000 MPa,
   density 7.85E^{-9} t/mm^3, Poisson's ratio 0.3

EXERCISE 4A: COMPRESSOR BRACKET MODAL ANALYSIS

Objectives (2/2)

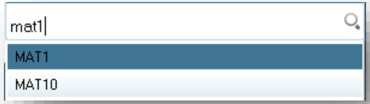
4. Create a `PSOLID` property `bracket` referencing material `steel` and assign it to the `bracket` component
5. Create a mass element at node 6 (dependent node of the RBE3 element) with value `0.003`
6. Create a load collector `SPC` (no card image) and with constraints to all five bolt locations RBE2 independent nodes (1-5) for DOF 1-3 each
7. Create a load collector `modal` (card image `EIGRL`) and set the number of desired roots (ND) to `6`
8. Create a load step `normal modes` and reference the two local collectors accordingly
9. Set common control cards requests
10. Request the strain energy results using global output request `ESE`
11. Run the analysis with OptiStruct
12. Review the `.out` file wrt warnings and errors and check if
    - $f_1 > 350$ Hz
13. Review contours of the mode shapes and strain energy in HyperView

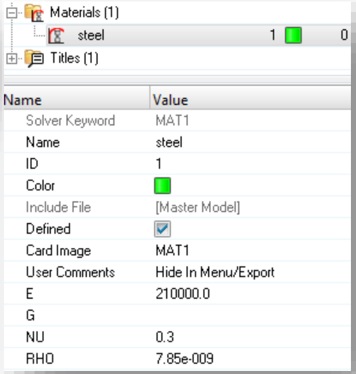


EXERCISE 4A: COMPRESSOR BRACKET MODAL ANALYSIS

Hints (1/6)

2. 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 know-how 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.
3. Use HyperMesh’s Quick Access Tool (`Crtl+f`) to add according `MAT1` material card

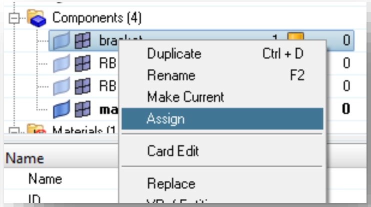


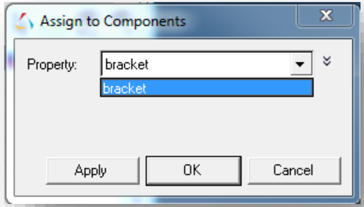


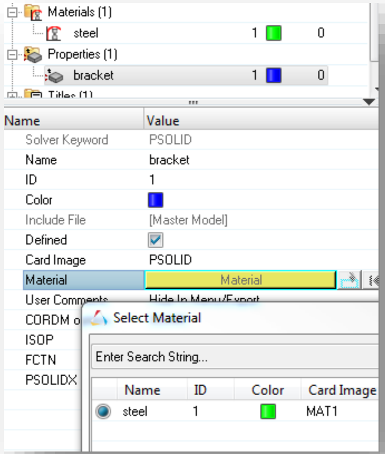
EXERCISE 4A: COMPRESSOR BRACKET MODAL ANALYSIS

Hints (2/6)

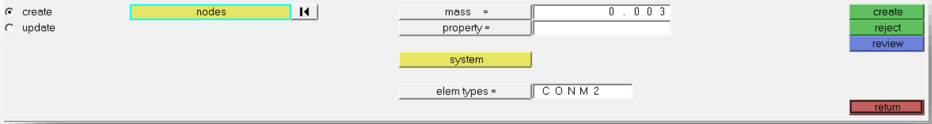
4. Use HyperMesh's Quick Access Tool (`Crtl+f`) to create property `PSOLID` and assign it to the bracket component with the right mouse menu

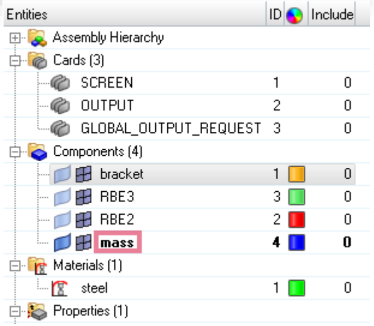






5. Check that `mass` is the active collector (marked in bold in the model browser), and create a `CONM2` element (Concentrated Mass Element Connection, Rigid Body Form). You can reach the panel with HM's Quick Access Tool or with the pull-down menu Mesh → Create → Masses





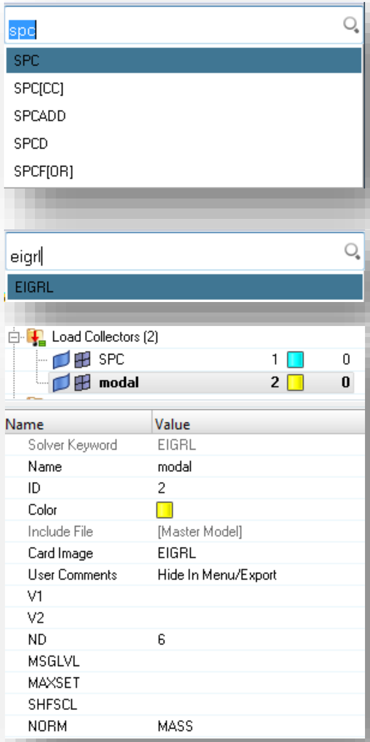
EXERCISE 4A: COMPRESSOR BRACKET MODAL ANALYSIS

Hints (3/6)

5. Note that a `RBE3` element is used to link the mass element to the bracket.
   A `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.

6. 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.

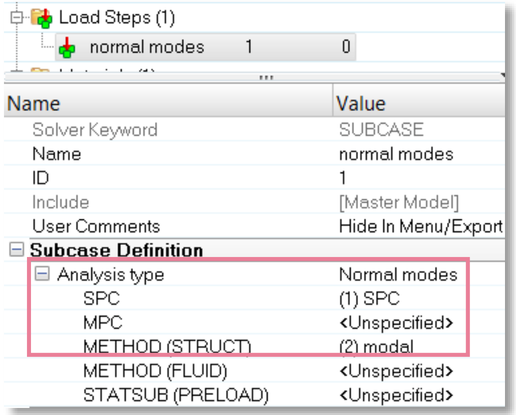
7. Use HyperMesh's Quick Access Tool (`Crtl+f`) with `EIGRL` and set ND to `6`



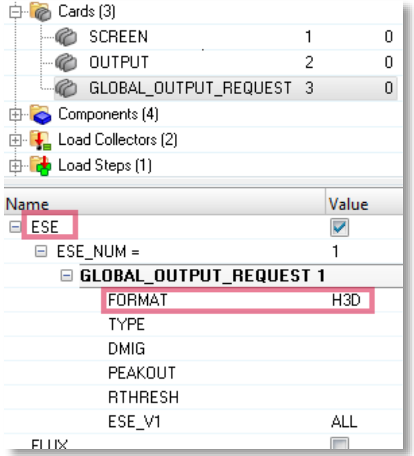
EXERCISE 4A: COMPRESSOR BRACKET MODAL ANALYSIS

Hints (4/6)

8. Set Analysis type to Normal modes in order to reduce the number of Subcase Information Entries

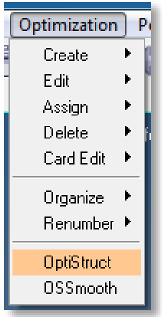


9. Use HyperMesh's Quick Access Tool (`Crtl+f`) to add control cards
   `SCREEN OUT`
   `OUTPUT,H3D,ALL`
   `OUTPUT,HTML,,NO`



10. Do the same for `ESE`

11. Run the model in OptiStruct using e.g. the OptiStruct panel via pull-down menu Optimization → OptiStruct



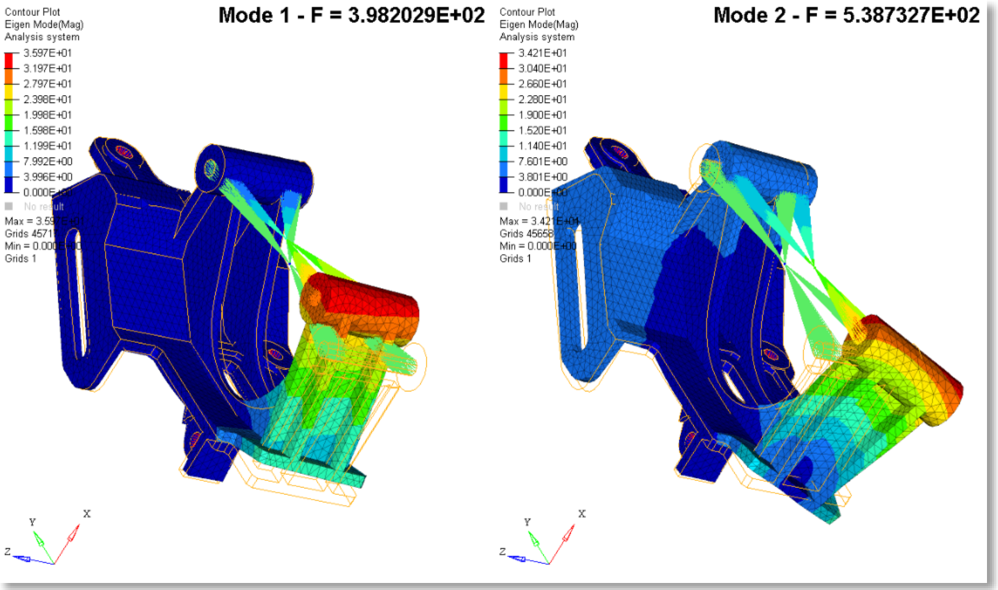
EXERCISE 4A: COMPRESSOR BRACKET MODAL ANALYSIS

Hints (5/6)

12. There are 22 elements that exceeded recommended range (warning) for the element quality check.
    f_1 = 398 Hz > 350 Hz, so the constraint is fulfilled.

Subcase	Mode	Frequency	Eigenvalue	Generalized Stiffness	Generalized 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

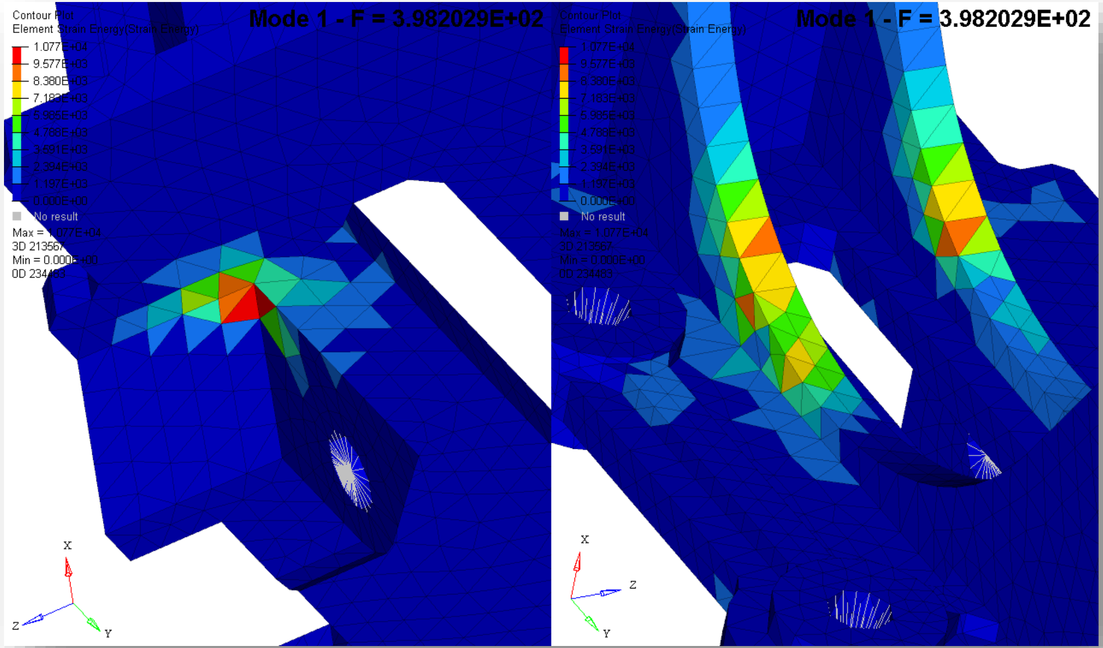
13. Screenshot shows mode shapes 1 and 2



EXERCISE 4A: COMPRESSOR BRACKET MODAL ANALYSIS

Hints (6/6)

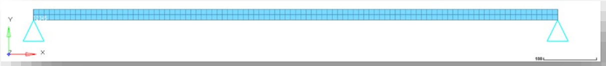
13. Strain energy can give to the analyst a very good indication if the mode is well refined or there is need for a mesh refinement. It works like the stress for a static analysis.



EXERCISE 4B(OPT.): SIMPLY SUPPORTED BEAM MODEL ANALYSIS

File Name and Location

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



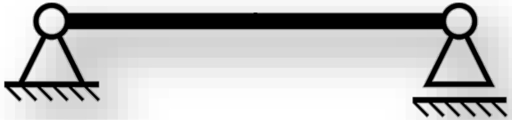
Objectives (1/2)

In this exercise, a modal analysis is performed on the simple supported beam from example 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 0.3, density 7.85e-9 t/mm^3)

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

1. Open the model in HyperMesh Desktop
2. Review the model
3. Constrain all nodes additionally in the z-direction in order to get only the shapes in xy-plane.



EXERCISE 4B (OPT.): SIMPLY SUPPORTED BEAM MODEL ANALYSIS

Objectives (2/2)

4. Create a load collector `modal` (card image `EIGRL`) and set the number of desired roots (ND) to `3`
5. Create a load step `Modal` and reference the two local collectors accordingly
6. Run the analysis with OptiStruct
7. Review the `.out` file wrt warnings and errors and check f_1 , f_2 and f_3
8. Calculate the theoretical results for f_1 , f_2 and f_3
9. Review contours of the three mode shapes HyperView

EXERCISE 4B (OPT.): SIMPLY SUPPORTED BEAM MODEL ANALYSIS

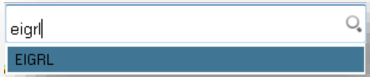
Hints (1/3)

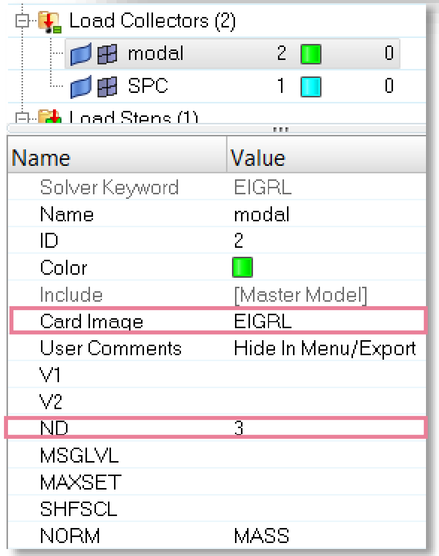
4. Use HyperMesh's Quick Access Tool (`Crtl+f`) with `EIGRL` and set ND to `3`

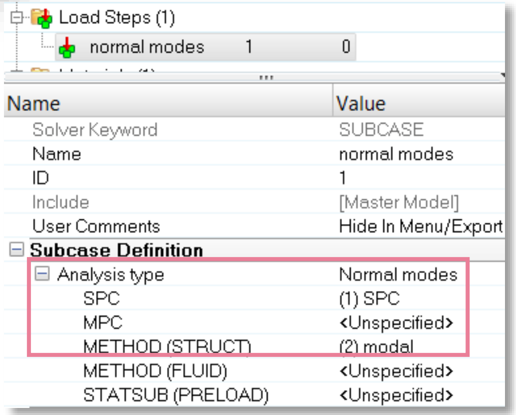
5. Set Analysis type to Normal modes in order to reduce the number of Subcase Information Entries

7. from the `.out` file:

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







EXERCISE 4B (OPT.): SIMPLY SUPPORTED BEAM MODEL ANALYSIS

Hints (2/3)

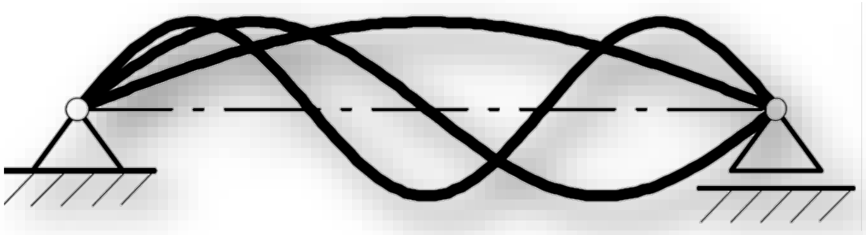
8. $f_j = \frac{j^2\,\pi}{2}\sqrt{\frac{EI}{\rho\,t\,h\,l^4}} = \frac{j^2\,\pi}{2}\sqrt{\frac{E\,h^2}{12\,\rho\,l^4}} = \frac{j^2\,h\,\pi}{4\,l^2}\sqrt{\frac{E}{3\,\rho}}$ for a simply supported beam

$f_j = \frac{j^2\cdot 20\,\pi}{4\cdot 10000^2}\sqrt{\frac{210000}{3\cdot 0{,}00000000785}}\ \mathrm{Hz} = 46{,}9\,j^2\ \mathrm{Hz}$ for this example

$f_1 = 46{,}9\ \mathrm{Hz}$ versus 46.8 Hz

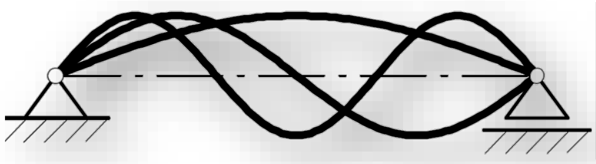
$f_2 = 187{,}6\ \mathrm{Hz}$ versus 186.4 Hz

$f_3 = 422{,}2\ \mathrm{Hz}$ versus 415.5 Hz

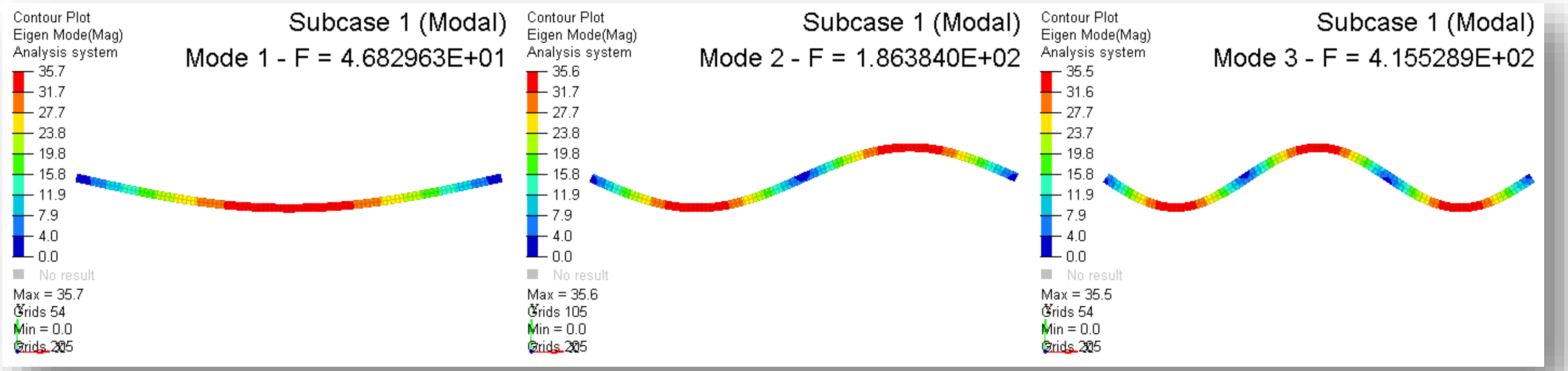


EXERCISE 4B (OPT.): SIMPLY SUPPORTED BEAM MODEL ANALYSIS

Hints (3/3)



9. In the contour plots the deformed shapes are scaled by 2.



QUESTIONS & ANSWERS

1. Which of the following describes the expected result of a modal analysis?
   a) Determining the natural frequency values of a structure undergoing vibration.
   b) Determining the shape of a structure under long-term point loads.
   c) Determining the mode shapes of a structure undergoing vibration.
   d) Determining the operational lifetime (total number of load cycles) of a part undergoing cyclic loading.
   e) a) and c)
2. What role does element length play in modal analysis?
   a) Lower element lengths allow more exact calculation of frequencies and mode shapes.
   b) Lower element lengths allow higher frequencies and mode shapes to be captured
   c) Element length plays no role in modal analysis
   d) a) and b)

QUESTIONS & ANSWERS

3. Which load collector card type was used in this chapter to specify natural frequency analysis information?
   a) `EIGRL`
   b) `MAT1`
   c) `PSHELL`
   d) `PSOLID`
   e) a) and c)
   f) All of the above