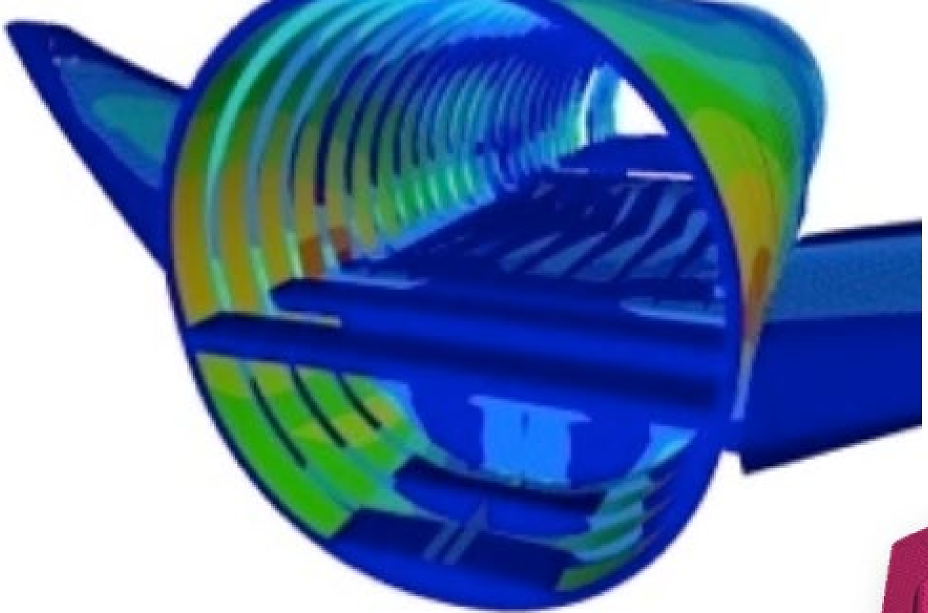
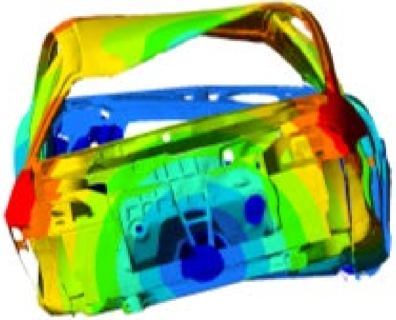
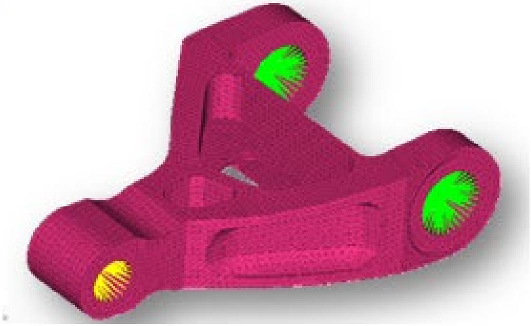






OPTISTRUCT FOR LINEAR ANALYSIS, V2019
CHAPTER 2: LINEAR STATIC ANALYSIS

Altair

AGENDA

CHAPTER 2: DETAILED AGENDA

Linear Static Analysis

- Linear static analysis defined
- Model definition: input decks, structure, outputs, case control and bulk data statements
- Run options and result outputs
- Stress evaluation and averaging

LINEAR STATIC ANALYSIS DEFINED

LINEAR STATIC ANALYSIS DEFINED

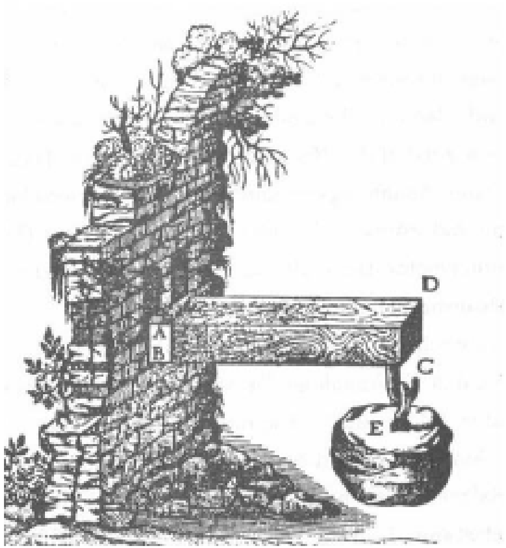
In mechanics we can define **static state** as the state of a system that is in equilibrium under an action of balanced forces and moments so that they remain at rest (no velocity).

System is subjected to loads and boundary conditions like:

- Forces, Moments, Temperature
- SPCs (Single point constraints), MPCs (Multi point constraints), …

Analysis has some assumptions like:

- Deformations are in the elastic range.
- Stresses are assumed to be linear functions of the strains.



LINEAR STATIC ANALYSIS FUNDAMENTALS

Linear Static solvers produce solutions from the basic equation:

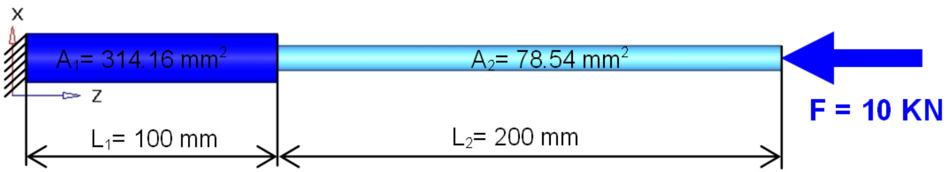
$$\mathbf{K}\mathbf{x} = \mathbf{f}$$

$\mathbf{K}$ global stiffness matrix

$\mathbf{x}$ displacement vector response to be determined

$\mathbf{f}$ external forces vector applied to the structure

Example: a loaded bar

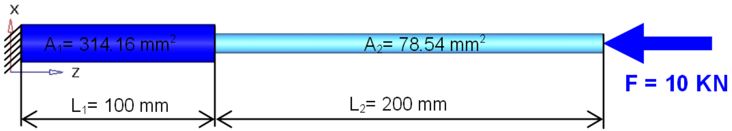


Material Steel:

E =210 GPa

LINEAR STATIC ANALYSIS EXAMPLE

We can break down the loaded bar setup computationally as follows:



Material Steel:

E =210 GPa

1. FE Model

2. Element Matrix

$$\mathbf{K} = \begin{bmatrix} \frac{EA}{L} & -\frac{EA}{L} \\ -\frac{EA}{L} & \frac{EA}{L} \end{bmatrix}$$

3. Global Matrix

$$\mathbf{K}_1 = \begin{bmatrix} \frac{210*314.16}{100} & -\frac{210*314.16}{100} \\ -\frac{210*314.16}{100} & \frac{210*314.16}{100} \end{bmatrix} = \begin{bmatrix} 659.74 & -659.74 \\ -659.74 & 659.74 \end{bmatrix}$$

$$\mathbf{K}_2 = \begin{bmatrix} \frac{210*78.54}{200} & -\frac{210*78.54}{200} \\ -\frac{210*78.54}{200} & \frac{210*78.54}{200} \end{bmatrix} = \begin{bmatrix} 82.47 & -82.47 \\ -82.47 & 82.47 \end{bmatrix}$$

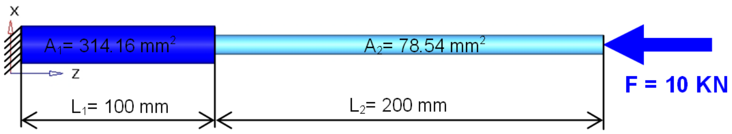
$$\mathbf{K}_G = \begin{array}{c|ccc|} & 1 & 2 & 3 \\ \hline 1 & 659.74 & -659.74 & 0 \\ 2 & -659.74 & 659.74+82.47 & -82.47 \\ 3 & 0 & -82.47 & 82.47 \\ \hline \end{array}$$

4. Forces and Displacements

$$\mathbf{f} = \begin{Bmatrix} 0 \\ 0 \\ -10 \end{Bmatrix} \qquad \mathbf{x} = \begin{Bmatrix} x_1 \\ x_2 \\ x_3 \end{Bmatrix}$$

LINEAR STATIC ANALYSIS EXAMPLE

By combining (5) the global system with (6) the prescribed DOF, (7) the system can be solved for strain, stress, and forces.



Material Steel:

E =210 GPa

5. Global System
$$\begin{bmatrix} 659.74 & -659.74 & 0 \\ -659.74 & 742.21 & -82.47 \\ 0 & -82.47 & 82.47 \end{bmatrix} \begin{Bmatrix} x_1 \\ x_2 \\ x_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \\ -10 \end{Bmatrix}$$

6. Eliminate the Prescribed DOF
$$\begin{bmatrix} 742.21 & -82.47 \\ -82.47 & 82.47 \end{bmatrix} \begin{Bmatrix} x_2 \\ x_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ -10 \end{Bmatrix}$$

7. Solving the system
$$\begin{Bmatrix} x_2 \\ x_3 \end{Bmatrix} = \begin{Bmatrix} -0.0152 \\ -0.1364 \end{Bmatrix}$$

Strain

$$\varepsilon_1 = \frac{\Delta L}{L} = \frac{x_2 - x_1}{L_1} = \frac{-0.0152 - 0}{100} = -1.52 * 10^{-4} \text{ mm/mm}$$

$$\varepsilon_2 = \frac{\Delta L}{L} = \frac{x_3 - x_2}{L_2} = \frac{-0.1364 - 0.0152}{200} = -6.06 * 10^{-4} \text{ mm/mm}$$

Stress

$$\sigma_1 = E\varepsilon_1 = -1.52 * 10^{-4} * 210 \text{ GPa} = -0.032 \text{ GPa}$$

$$\sigma_2 = E\varepsilon_2 = -6.06 * 10^{-4} * 210 \text{ GPa} = -0.127 \text{ GPa}$$

Forces

$$f_1 = \sigma_1 A_1 = -0.032 * 314.16 \text{ kN} = -10 \text{ kN}$$

$$f_2 = \sigma_2 A_2 = -0.127 * 78.54 \text{ kN} = -10 \text{ kN}$$

MODEL DEFINITION

MODEL DEFINITION: INPUT DECKS

OptiStruct uses a Nastran-style ASCII Input Format, e.g.

Elements

- 3D Shell: `TRIA3`, `TRIA6`, `QUAD4`, `QUAD8`
- 3D Solid: `TETRA4`, `TETRA10`, `PENTA6`, `PENTA15`, `PYRA5`, `PYRA13`, `HEXA8`, `HEX20`
- 1-D Elastic: `ROD`, `TUBE`, `BAR`, `BEAM`, `ELAS1`, `ELAS2`, `BUSH`
- 1-D Rigid: `RROD`, `RBAR`, `RBE2`, `RBE3`
- Concentrated Mass : `CONM`, `CMASS`
- 1-D Connections: `CWELD`, `CVISC`, `CDAMP`, `CGAP` (Non-Linear gap element), `CGAPG` (node-patch nonlinear gap element)

Properties

- Shells: `PSHELL`, `PCOMP(G)`, `PCOMPP`
- Solids: `PSOLID`
- …

Loads

- Force: `FORCE`
- …

MODEL DEFINITION: STRUCTURE

Input/Output Section is used to define general information about the model.

Subcase Information

- Define Load Cases
  - Subcases
  - Load Steps
- Define Objective
- Define Constraint Reference

Bulk Data Section

- Optimization Problem
  - Design Variables
  - Responses
  - Constraints
- Optimization parameters
- Finite Element Model

```
$$
$$   Template:  optistruct
$$
$$--------------------------------------------------------------------------------
$$                              Case Control Cards
$$--------------------------------------------------------------------------------
$
$HMNAME LOADSTEP              1"Load"      3
$
TITLE = Plate Model
ANALYSIS
FORMAT HM
FORMAT H3D
SCREEN NONE
SUBCASE       1
  LABEL Load
ANALYSIS MODES
  SPC =        1
  METHOD(STRUCTURE) =        3
$
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
```

Input/Output Section

Subcase Information

Bulk Data

MODEL DEFINITION: INPUT/OUTPUT SECTION

Defines the types and distribution of results requested

Types and functions include

- Run information (.out, .stat, .hist)
- Model-specific solutions (.sh, .desvar, .prop, .hgdatal, .grid, .oss)
- HyperMesh command files (.HM.comp.cmf, .HM.ent.cmf)
- HTML Reports (.html, _frames.html, _menu.html, .shuf.html)
- Model results (.res, .h3d, _des.h3d, _s#.h3d)
- HV session file (.mvw, _hist.mvw)

Result files may be ASCII or binary

```
$$
$$  Template:  optistruct
$$
$$----------------------------------------------------------------------
$$                    Case Control Cards
$$----------------------------------------------------------------------
$
$HMNAME LOADSTEP               1"Load"      3
$
TITLE = Plate Model
ANALYSIS
FORMAT HM
FORMAT H3D
SCREEN NONE
SUBCASE       1
  LABEL Load
ANALYSIS MODES
  SPC =        1
  METHOD(STRUCTURE) =        3
$
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
```

Input/Output Section

MODEL DEFINITION: INPUT/OUTPUT SECTION

The I/O Options section controls:

- location and names of the input, output and scratch files,
- type of run (analysis, check or restart)
- overall running of the analysis or optimization, and,
- type, format and frequency of the output.

Some Categories of I/O Options:

- Output Format Controls: `FORMAT`, `OUTPUT`
- Run Controls: `ANALYSIS`, `CHECK`, `CPU`, `NPROC`, `RESTART`, `SYSSSETTING`
- File Names, Headers and Locations: `EIGVNAME`, `INCLUDE`, `OUTFILE`, …
- Analysis Output: `CSTRAIN`, `CSTRESS`, `DISPLACEMENT`, `ELFORCE`, …
- Optimization Output: `DENSITY`, `DENSRES`, `DESHIS`, `PROPERTY`, `RESPRINT`, …
- Other Output Controllers: `ECHO`, `ECHOON`, `ECHOOFF`, `DMIGNAME`, `MODEL`, …

OptiStruct > Reference Guide > Output Data:

Files Created by OptiStruct

.amis_singularity.cmf file
.amses_singularity.cmf file
.bdst file
.cntf file
.contgap.fem file
.cstr file
.dens file
.desvar file
.dis.# file
.disp file
.dvgrid file
.echo file
.els.# file
.force file
.fsthick file
.gpf file
.grid file
.h3d file
.hgdata file
.hist file
.HM.auto.cmf file
.HM.comp.cmf file
.HM.conn.cmf file
.HM.elcheck.cmf file
.HM.elcheck.###.cmf file
.HM.ent.cmf file
.HM.gapstat.cmf file
.html file
.interface file
.k.op2 file
.load file
.m.op2 file
.mass file
.mnf file
.mpcf file
.op2 file
.oss file
.out file
.pch file
.pcomp file
.peak file
.pret file
.prop file

MODEL DEFINITION: CASE CONTROL SECTION

Defines subcases by specifying load collectors

- Subcase labels and other information may be listed within a subcase block
- Subcase type is specified within each subcase block
- Information specific to an individual subcase will be listed under that block
- Each subcase must have an unique Integer ID
- HM comments carry subcase name and identification information
- HM comments in the Case Control section may also include set and tag information

```
$$
$$   Template:  optistruct
$$
$$------------------------------------------------------------------
$$                    Case Control Cards
$$------------------------------------------------------------------
$
$HMNAME LOADSTEP               1"Load"       3
$
TITLE = Plate Model
ANALYSIS
FORMAT HM
FORMAT H3D
SCREEN NONE
SUBCASE       1
  LABEL Load
ANALYSIS MODES
  SPC =         1
  METHOD(STRUCTURE) =         3
$
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
```

Subcase Information

MODEL DEFINITION: CASE CONTROL SECTION

The Case Control section

- identifies which loads and boundary conditions are to be used in a subcase,
- controls output type and frequency, and,
- may contain objective and constraint information for optimization problems.

Some categories of Subcase Information:

- General: `LABEL`, `SUBCASE`
- FE Analysis: `SPC`, `LOAD`, `METHOD`, `EIGVRETRIEVE`, `EIGVSAVE`, …
- MBD Analysis: `INVEL`, `MBSIM`, `MLOAD`, `MOTION`, `SPC`
- Optimization: `DESGLB`, `DESOBJ`, `DESSUB`, …
- Component Mode Synthesis: `CMSMETH`, `MPC`

```
$$
$$   Template:   optistruct
$$
$$--------------------------------------------------------------------
$$                        Case Control Cards
$$--------------------------------------------------------------------
$
$HMNAME LOADSTEP              1"Load"       3
$
TITLE = Plate Model
ANALYSIS
FORMAT HM
FORMAT H3D
SCREEN NONE
SUBCASE        1
  LABEL Load
ANALYSIS MODES
  SPC =        1
  METHOD(STRUCTURE) =        3
$
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
```

Subcase Information

MODEL DEFINITION: BULK DATA SECTION

Comments and comment lines can be defined two ways

- All characters after $ until the end of the line will be considered comments
- A line beginning with two slashes "//" or a pound "#" will be considered as comments

Keywords are the first word in a card

- Must start from the first column
- They must be all caps and abbreviations are not allowed
- Example: GRID, CQUAD4, PSHELL, MAT, LOAD, FORCE, SPC, …

```
$$
$$   Template:  optistruct
$$
$$------------------------------------------------------------------------
$$                          Case Control Cards
$$------------------------------------------------------------------------
$
$HMNAME LOADSTEP               1"Load"       3
$
TITLE = Plate Model
ANALYSIS
FORMAT HM
FORMAT H3D
SCREEN NONE
SUBCASE       1
  LABEL Load
ANALYSIS MODES
  SPC =         1
  METHOD(STRUCTURE) =        3
$
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
```

Bulk Data

MODEL DEFINITION: BULK DATA SECTION

Continuation cards extend the length of cards to successive lines

- Must follow the parent entries
- If 1st character of any entry is either a blank, “+”, or “*”, it is treated as a continuation of the previous entry
- Content of 10th field in each card (with the exception of DTPG) and the 1st field in each continuation card is disregarded
- Do not have to be in the same format as the parent entries

Each entry can be placed anywhere within the field

- Blanks preceding and following an entry are ignored
- Keyword entry is the exception: must be left justified in its field

MODEL DEFINITION: BULK DATA SECTION

The Bulk Data section begins with the `BEGIN BULK` statement and ends with the `END DATA` statement.

Data lines can contain a maximum of 80 characters (characters after the 80th will be ignored).

Each line of data contains up to nine fields in one of the three accepted formats:

- Fixed Format

GRID----	-------1	--------	----24.0	----24.0	-----0.0	--------	--------	--------	--------

- Free Format

```
GRID, 1, , 24.0, 24.0, 0.0
```

- Large Field Format

GRID*---	---------------1	----------------	-----24.00000000	-----24.00000000	--------
*-------	------0.00000000	----------------	----------------	----------------	--------

MODEL DEFINITION: CARD ENTRY DETAILS

Character entries in OptiStruct cards have the following restrictions:

- Must start with a letter
- Can not contain blanks within the data
- Longer than 8 characters are truncated in large field and free field formats
  - Exception: file names on the `INCLUDE` card
- Case insensitive (except user-provided labels)

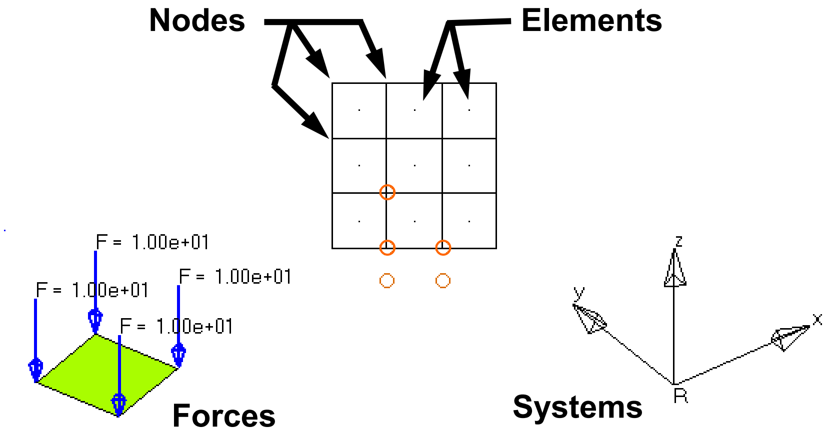
Numeric entries in OptiStruct cards have the following restrictions:

- Must start with a digit, '+' or '–'
- Integer entries may not contain a decimal point or an exponent part
- Integer data placed in the field reserved for real valued data is accepted and converted to a double precision
- Real must have a decimal can follow most formats within defined characters format

BULK DATA SECTION

This section starts with `BEGIN BULK` and ends with `END BULK`.

Categories of Bulk Data include




```
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
$
CBEAM        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: PARAM

`PARAM` cards define parameters used during the analysis

Some examples of parameter cards are:

- `ALMS`, `AUTOSPC`, `CHECKEL`, `CHECKMAT`, `INREL`, `PRGPST`, `WTMASS`

See also chapter 8.2 Parameters

(1)	(2)	(3)	(4)	(5)	(6)	(7)	(8)	(9)	(10)
PARAM	N	V							

N	Name of Parameter
V	Value of Parameter

```
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
$
CBEAM         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: PARAM, AUTOSPC

OptiStruct checks the global stiffness matrix for degrees-of-freedom (DOF) with no stiffness.

The default setting `PARAM,AUTOSPC,YES` automatically constrains these DOF and 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,YES`, OptiStruct would automatically fix this element and run the analysis.

The user should be aware of any DOF fixed by the AUTOSPC. Also, do not forget that in the end, if the run is made with activated AUTOSPC, to verify which DOF was fixed and if this has not affected the solution.

With `PARAM,AUTOSPC,NO`, the DOF with no stiffness are not automatically constrained.

BULK DATA CARDS: LOCAL COORDINATE SYSTEMS

Local Coordinate Systems differ in location and/or orientation from the global

Local coordinate systems can define:

- Location/Orientation of Nodes
- Elements, Materials, Properties
- Loads, Constraints, Results

Types of local systems include:

- Rectangular, Cylindrical & Spherical

Three non-collinear points define a local system

```
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
$
CBEAM        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: LOCAL COORDINATE SYSTEMS

Local Coordinate Systems differ in location and/or orientation from the global

(1)	(2)	(3)	(4)	(5)	(6)	(7)	(8)	(9)	(10)
CORD2R	CID	RID	A1	A2	A3	B1	B2	B3	
+	C1	C2	C3						

CID	System identification number. (Unique Integer > 0)
RID	ID of a system independently defined from this system (Optional, Default = 0, Integer)
A1,A2,A3 B1,B2,B3 C1,C2,C3	Coordinates of three points in the global system

```
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
$
CBEAM        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: GRID

`GRID` cards are used to represent discrete locations in the model space

Girds are also referred to as “nodes”

Grids define the structural model and its boundary conditions

(1)	(2)	(3)	(4)	(5)	(6)	(7)	(8)	(9)	(10)
GRID	ID	CP	X1	X2	X3	CD	PS		

ID	Unique grid point identification number. (Integer > 0)
CP	Identification number of coordinate system in which the location of the grid point is defined. (Integer > 0 or blank)
X1,X2,X3	Location of the grid point in coordinate system CP
CD	Identification number of coordinate system in which the displacements, degrees of freedom, constraints, and solution vectors are defined at grid point. (Integer > 0 or blank)
PS	Permanent single-point constraints associated with grid point. Up to six unique digits may be placed in the field with no embedded blanks. (Integer > 0 or blank)

```
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
$
CBEAM        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: ELEMENTS

The geometry of a model's structure is represented using elements

Elements can be 0-dimensional, 1-dimensional, 2-dimensional, or 3-dimensional

Available elements include:

0D	CONM2, …
1D	CBEAM, CELAS2, CGAP, PLOTEL, RBE2, RBE3, …
2D	CQUAD8, CQUAD4, CTRIA6, CTRIA3, CSHEAR, …
3D	CHEXA, CPENTA, CPYRA, CTETRA, …

```
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
$
CBEAM        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: ELEMENTS

The geometry of a model's structure is represented using elements

Models may contain elements of multiple dimensionality

(1)	(2)	(3)	(4)	(5)	(6)	(7)	(8)	(9)	(10)
CQUAD4	EID	PID	G1	G2	G3	G4			

EID	Unique element identification number
PID	Identification number of a PSHELL or PCOMP property entry
G1,G2,G3,G4	Grid ID' s of connection points (Integers > 0, all unique)

```
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
$
CBEAM        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: PROPERTIES

Properties fill in geometric information needed to represent elements as 3D

Properties are specific to a single element type

Available properties include:

1D: PBEAM, PELAS2, PGAP, PBAR, PBEAM, …

2D: PSHELL, …

3D: PSOLID

```
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
$
CBEAM        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: PROPERTIES

Property cards are used to define element attributes like

- Element thickness
- ID of the material being used
- Section properties for beams

Property cards are referenced by elements

- CHEXA, CPENTA, CPYRA, CTETRA → PSOLID
- CQUAD8, CQUAD4, CTRIA6, CTRIA3 → PSHELL or PCOMP
- CSHEAR → PSHEAR
- CBEAM → PBEAM

```
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
$
CBEAM        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: PROPERTIES

(1)	(2)	(3)	(4)	(5)	(6)	(7)	(8)	(9)	(10)
PSHELL	PID	MID1	T	MID2	12I/T3	MID3	TS/T	NSM	

PID	Unique shell element property identification number. (Integer > 0)
MID1	Material identification number for membrane. (Integer > 0)
T	Default value for the membrane thickness
MID2	Material identification number for bending
12I/T3	Bending stiffness parameter. (default = 1.0)
MID3	Material identification number for transverse shear
TS/T	Transverse shear thickness divided by the membrane thickness. (default = 0.833333)
NSM	Nonstructural mass per unit area

```
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
$
CBEAM         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: MATERIALS

Material cards are available for a range of directionality and thermal behaviors

Linear temperature independent

- `MAT1`: isotropic
- `MAT2`: anisotropic (for 2D elements)
- `MAT8`: orthotropic (for 2D elements)
- `MAT9`: anisotropic (for 3D elements)
- `MAT9ORT`: orthotropic (for 3D elements)

Temperature dependent

- defined with the respective `MAT` cards
- `MATT1`, `MATT2`, `MATT8` and `MATT9`.

```
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
$
CBEAM         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: MATERIALS

Materials are referenced by properties to be used by elements.

Material assignment relationships show up in the deck as MID references in other cards

(1)	(2)	(3)	(4)	(5)	(6)	(7)	(8)	(9)	(10)
MAT1	MID	E	G	NU	RHO	A	TREF	GE	

MID	Unique material identification number (Integer > 0)
E	Young's Modulus
G	Shear Modulus
NU	Poisson's Ratio
RHO	Mass density
A	Thermal expansion coefficient
TREF	Reference temperature for thermal loading
GE	Structural Element Damping coefficient

```
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
$
CBEAM        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: LOADS AND BOUNDARY CONDITIONS

Loads and boundary conditions are applied to FEM and geometric entities

The types of loads applied are dependent on several factors such as the desired solution type, elements available, and type or form of results desired

Available static load types include:

- point forces (FORCE, FORCE1)
- gravity loads (GRAV)
- moments (MOMENT, MOMENT1)
- pressures (PLOAD, PLOAD1, PLOAD2, PLOAD4)
- rotational forces (RFORCE)
- enforced displacements (SPCD)
- temperature gradients (TEMP, TEMPD)

```
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
$
CBEAM        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: LOADS AND BOUNDARY CONDITIONS

(1)	(2)	(3)	(4)	(5)	(6)	(7)	(8)	(9)	(10)
FORCE	SID	G	CID	F	N1	N2	N3		

SID	Load set identification number. (Integer > 0)
G	Grid point identification number
CID	Coordinate system identification number
F	Scale factor
N1, N2, N3	Components of vector measured in coordinate system CID.

```
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
$
CBEAM        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: LOADS AND BOUNDARY CONDITIONS

Boundary conditions are generally used to reduce or constrain FE structure

The types of loads applied are dependent on several factors such as the desired solution type, elements available, and type or form of results desired

OptiStruct allows the following boundary conditions to be applied at nodal locations on the structure:

single-point constraint (`SPC`, `SPC1`)

multi-point constraint (`MPC`)

fictitious support (`SUPORT`, `SUPORT1`)

```
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
$
CBEAM        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: LOADS AND BOUNDARY CONDITIONS

(1)	(2)	(3)	(4)	(5)	(6)	(7)	(8)	(9)	(10)
SPC	**SID**	**G**	**C**	**D**	**G**	**C**	**D**		

SID	Identification number of single-point constraint set. (Integer > 0)
G	Grid or scalar point identification number. (Integer > 0)
C	Component numbers. Scalar points: Integer, zero or blank Grid points: Up to six unique digits (0 thru 6) with no embedded blanks
D	Value of enforced displacement for all coordinates designated by G and C

```
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
$
CBEAM        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
```

CONSISTENT UNITS

All internal calculations in OptiStruct are unit-less

- It is the responsibility of the user to create the model using a consistent set of units.
- The equations that governs consistent units are:
  - Force = Mass × Acceleration
  - Mass = Density × Volume
  - Acceleration = Length / Time^2
- As an illustration:

Mass	Length	Young's Modulus	Density	Force	Stress	Time
kg	m	Pa	kg/m^3	N	Pa	s
kg	mm	GPa	kg/mm^3	kN	GPa	ms
t	mm	MPa	t/mm^3	N	MPa	s

RUN OPTIONS AND OUTPUT FILES

TYPE OF FILES OUTPUTTED BY OPTISTRUCT

File Extension	Files Format	File Generation	Details
OUT	ASCII	Automatically	The OptiStruct OUT file is a general and specific run information and output results file generated after every OptiStruct run.
H3D	Binary	By Request	The H3D file is a compressed binary file containing both model and output data
OP2	Binary	Request	The OP2 file is a binary file containing both model and output data.
RES	Binary	Request	The RES file is a HyperMesh results file containing both model and output data
PCH	ASCII	Request	The PCH file is a general output results file generated after an OptiStruct run
STAT	ASCII	Automatically	This file provides details on CPU and elapsed time for each solver module. Settings modified or updated in any configuration file will be written out in this file.
HTML	ASCII	Automatically	The .html file is a HyperText Markup Language file. This file contains a problem summary and results summary of the run
MVW	ASCII	Automatically	This file is a HyperView session file that is linked with the .h3d result file and can be open directly from HyperMesh using the HyperView button on the OptiStruct panel

OUTPUT RESULT TYPES

	H3D	HM	OP2	PCH	OPT	PAT	Controlling I/O Option
Nodal Displacements	●	●	●	●	●	●	DISPLACEMENT
Element Strain Energy	●	●	●	●	○	○	ESE
Element Stresses	●	●	●	●	●	●	STRESS / ELSTRESS
Element Strains	●	●	●	●	●	○	STRAIN
Ply Stresses	●	●	●	●	●	○	CSTRESS
Composite Failure Indices	●	●	●	○	●	○	CFAILURE
Ply Strains	●	●	●	●	●	○	CSTRAIN
Element Forces	●	●	●	●	●	○	FORCE / ELFORCE
Grid Point Stresses	●	●	●	●	○	○	GPSTRESS / GSTRESS
SPC Forces	●	●	●	●	●	○	SPCFORCE
MPC Forces	○	○	●	●	●	○	MPCFORCE
Grid Point Forces	●	○	●	●	●	○	GPFORCE
Applied Loads	●	○	●	●	●	○	OLOAD
Contact Force and Pressure	●	○	●	○	○	○	CONTF

OUTPUT: .OUT FILE DETAILS

The OptiStruct OUT ASCII file is a general and specific run information and output results file generated after every OptiStruct run. This file provides a process commentary and log of the solution. The contents of the OUT file depends on the output factors. Since this is an ASCII file, its contents can be viewed on any simple Text Editor.

There is only one type of OUT file generated regardless of the factors controlling output from OptiStruct. Only the contents of the OUT file varies depending on the model and run process. And here is the list of sections printed in the out file:

- General Analysis Information
- User Information (Messages/Warnings)
- Optional Diagnostic Information
- FE-Model Information
- Memory Estimation Information
- Disk Space Estimation Information
- Analysis Results or Optimization History Information

OUTPUT: *.*H3D FILE DETAILS

The H3D binary file is a compressed binary file containing both model and output data. It can be utilized to post-process results in HyperView, HyperView Player, and HyperGraph.

There are multiple types of H3D files output depending on the factors controlling output from OptiStruct, as shown in the table.

The availability of results sorted by Solution Sequence in H3D format is listed in the [Results Output by OptiStruct](#) page

H3D File	File Documentation
General/Default H3D file	.h3d file
BYITER and Optimization	.#.h3d file
Design Optimization H3D	_des.h3d file
Multi-body Dynamics H3D	_mbd.h3d file
Topology Sensitivity H3D	_topol.h3d file
Gauge Sensitivity H3D	_gauge.h3d file
Linear Static Optimization H3D	_s#.h3d file

OUTPUT: .H3D FILE CONTOUR PLOT AND ANIMATION

Result	Description
Displacement	Displacement results from static, frequency response, acoustic, transient response, and multi-body dynamics analyses. Output is controlled by the I/O Option `DISPLACEMENT`.
Eigenvector	Eigenvector results from normal modes and linear buckling analyses. Output is controlled by the I/O Option `DISPLACEMENT`.
Stress	Stress results from static, frequency response, acoustic, transient response, and multi-body dynamics analyses. Output is controlled by the I/O Option `STRESS` (or `ELSTRESS`).

RUN OPTIONS FOR OPTISTRUCT

Selected Run Options for OptiStruct (see Altair OptiStruct User's Guide for more)

`-analysis`	Submit an analysis run. This option will also check the optimization data; the job will be terminated if any errors exist.
`-optskip`	Submit an analysis run without performing check on optimization data (skip reading all optimization related cards).
`-check`	Submit a check job through the command line.
`-nt X`	Number of threads/cores (X) to be used for SMP solution.
`-np X`	Number of processors (X) to be used for SPMD analysis.
`-len X`	Preferred upper bound on dynamic memory allocation (with X in RAM MBytes)
`-maxlen X`	Hard limit on the upper bound of dynamic memory allocation (with X in RAM MBytes). OptiStruct will not exceed this limit.
`-core X`	The solver assigns the appropriate memory required. If there is not enough memory available, OptiStruct will error out. (in incore solution is forced, out out-of-core solution, min minimum core solution)
`-out`	Echoes the output file to the screen.

LINEAR STATIC ANALYSIS SETUP

Linear static analyses can be defined in 10 steps

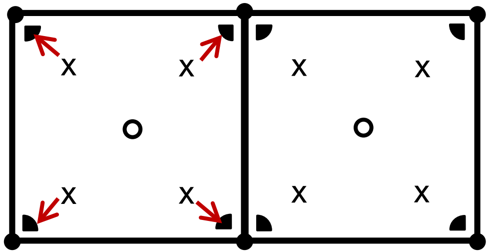
- Step 1 – Define the material
- Step 2 – Define the properties and associate it with the appropriate material
- Step 3 – Define the components and associate it with its property
- Step 4 – Create the Finite element mesh
- Step 5 – Define the constraint load collector and apply the model constraints
- Step 6 – Define the force load collector and apply the loads
- Step 7 – Define the load step
- Step 8 – Define the extra parameters to your analysis (optional)
- Step 9 – Run the analysis
- Step 10 – Post-process the results

STRESS EVALUATION & AVERAGING

STRESS EVALUATION: EXTRAPOLATION TO NODES

Stresses are always numerically **calculated** at the (Gauss) Integration Points (IP).

In OptiStruct by default, element stresses for shell and solid elements are **output** at the element centroid only.



- x Integration Points (IP)
- o Centroid
- ● Element nodes
- ◢ “Corner data” = nodal results (participation of element to a node)

Stress output at nodes is an extrapolation, the most simple option to archive this is bilinear extrapolation. This is equivalent to using shape functions of 1st-order shells.

This can be requested with:

`STRESS(H3D,CORNER)=YES` or `STRESS(H3D,BILIN)=YES`

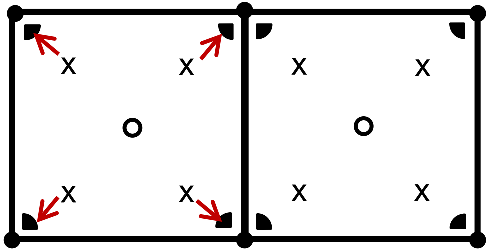
STRESS EVALUATION: EXTRAPOLATION TO NODES – SHELLS

Location	Method	Comment	Request example
Center	integration points stresses are averaged to the center	- “correct” stresses in term of the used FE-model - most stable regards to element quality	default
Gauss Integration Points	output integration point results to corner	- “correct” stresses in term of the used FE-model	STRESS(H3D,GAUSS)=YES
corner, bilin	bilinear extrapolation from integration points stresses to nodes	- good results for in-plane bending - can not capture out of plane bending effects - can be very oscillatory particular with bad element quality	STRESS(H3D,CORNER)=YES or STRESS(H3D, BILIN)=YES
cubic	strain gauge method with cubic bending correction	- good results for out of plane bending - less oscillatory then bilin on complex stress distributions - can overestimate stresses for in-plane bending	STRESS(H3D,CUBIC)=YES
sgage	strain gauge method	- kind of obsolete - cubic should provide more accurate results	STRESS(H3D,SGAGE)=YES

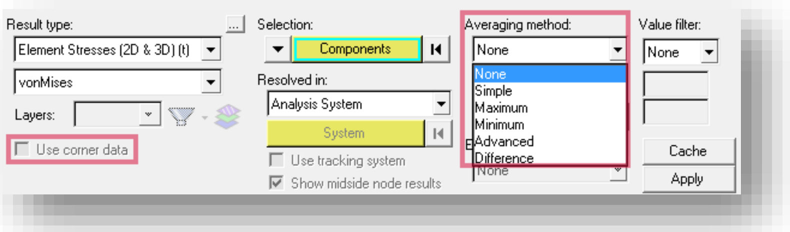
STRESS EVALUATION: EXTRAPOLATION TO NODES – SOLIDS

Location	Method	Comment	Request example
Center	integration points stresses are averaged to the center	- “correct” stresses in term of the used FE-Model - most stable regards to element quality	default
Gauss Integration Points	output integration point results to corner	- “correct” stresses in term of the used FE-Model	STRESS(H3D,GAUSS)=YES
corner, bilin, sage, cubic	bilinear extrapolation from integration points stresses to nodes	- extrapolation for solid-elements will always be bilinear - keywords sage and cubic are allowed, to support cubic stresses for shells in same model	STRESS(H3D,CORNER)=YES STRESS(H3D,BILIN)=YES STRESS(H3D,CUBIC)=YES STRESS(H3D,SGAGE)=YES
GPS – grid point stress	Bilinear extrapolation from integration points stresses to nodes and average at node	- 1 value per node, whereas the others give 1 nodal value per element - Global average or average by property can be requested	GPSTRESS(H3D)=YES
surface	membrane applied automatically for post processing only	- “correct” surface stresses in term of the used FE-model - can be used for fatigue	STRESS(H3D,SURF)=YES

STRESS AVERAGING AT NODES IN HYPERVIEW



- x   Integration Points (IP)
- o   Centroid
- ●   Element nodes
- ◢   “Corner data” = nodal results (participation of element to a node)

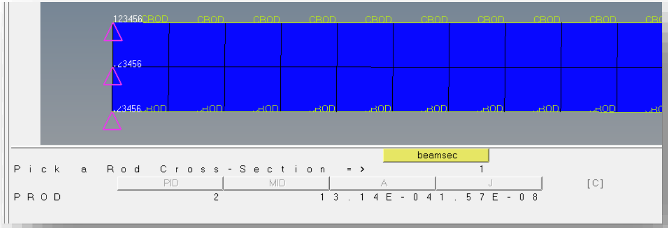


Location	Method	Comment
simple	Averaging of the stress invariants	Advanced should provide more accurate results
advanced	Averaging the stress tensor and calculate invariants from this	
difference	Max-min nodal value	Good to find areas with large with stress discontinuous → remeshing?
max/min	Max/min value	

ALTERNATIVE METHODS FOR STRESS CALCULATION

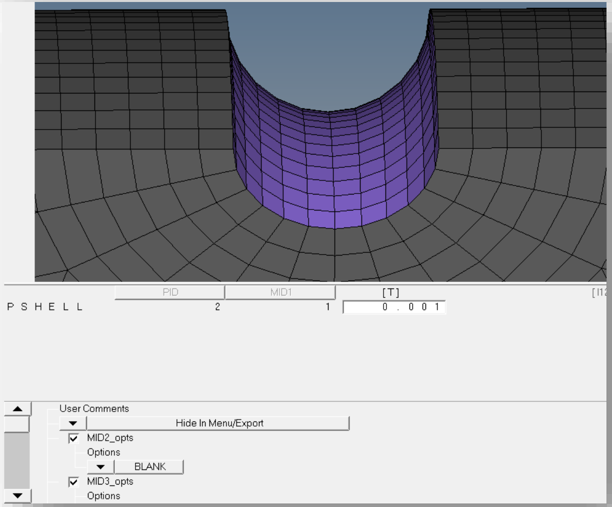
Rods on the edge of shells

- Use same Material as shell material
- Rod should be thin, but not to thin to jeopardies numerical stability
- Use HyperBeam to get reasonable A and J



Membrane on the surface of solids

- Thickness should be small, but not to small
- Remove transverse shear and bending stiffness by leave MID2 and MID 3 blank
- Use same Material as solid material
- `STRESS(H3D,SURF)=YES` creates that automatically



EXERCISES

EXERCISE 2A: STATIC ANALYSIS OF A SOLID BRACKET

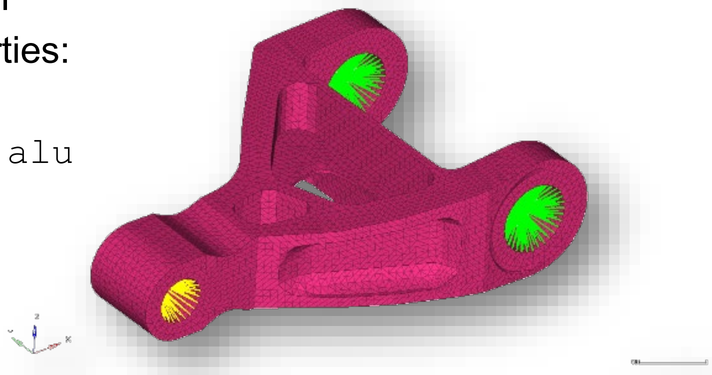
File Name and Location

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

Objectives (1/2)

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

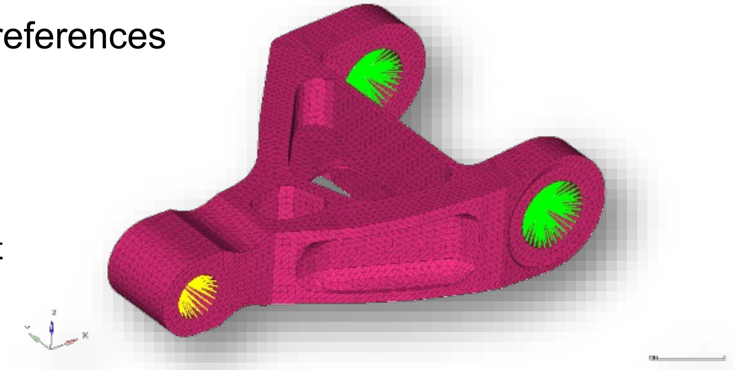
1. Open the model in HyperMesh Desktop with OptiStruct user profile selected
2. Review the model and check the dimensions of the model
3. Create a MAT1 material alu for aluminum with the properties: Young's modulus 70000 MPa, Poisson's ratio 0.33
4. Create a PSOLID property bracket referencing material alu and assign it to the torsion link component
5. Create a load collector Support (no card image) with the following SPC load type constraint
   - Node 4830: DOFs 1-6
   - Node 4831: DOFs 1, 3-6



EXERCISE 2A: STATIC ANALYSIS OF A SOLID BRACKET

Objectives (2/2)

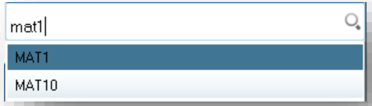
6. Create a load collector `force` (no card image) containing a force on node 1 with constant components {12000, 12000, -20000}
7. Create a load step `Load` of type Linear Static using `Support` as `SPC` and `force` as `LOAD` entry
8. Export the model as solver deck
9. Review the `.fem` file in a text editor and understand the references
10. Run the analysis with OptiStruct
11. Add output requests in order to
    - echo `.out` file on the screen
    - get results only in `H3D` format and suppress the `html` file output
12. Rerun the analysis with OptiStruct
13. Review the `.out` file wrt warnings, errors and Auto-SPC
14. Add parameter to deactivate automatic constraining
15. Rerun the analysis with OptiStruct and review the `.out` file again
16. Review the displacements and stresses in HyperView

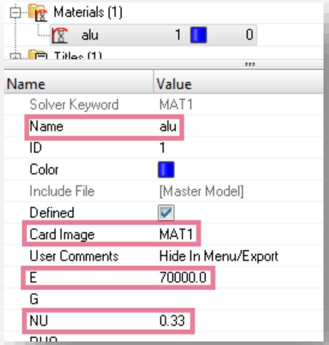


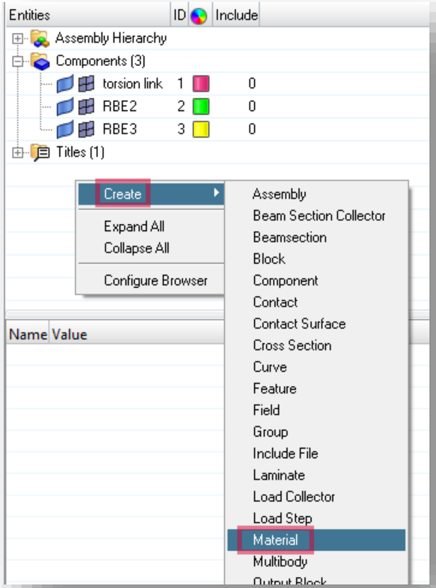
EXERCISE 2A: STATIC ANALYSIS OF A SOLID BRACKET

Hints (1/9)

2. 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.
     A `RBE2` would include a rigid condition that doesn't exist.
3. Click right mouse button in the model browser, select ***Create → Material***
   or
   use HyperMesh's Quick Access Tool (`Crtl+f`) to create according `MAT1` material card







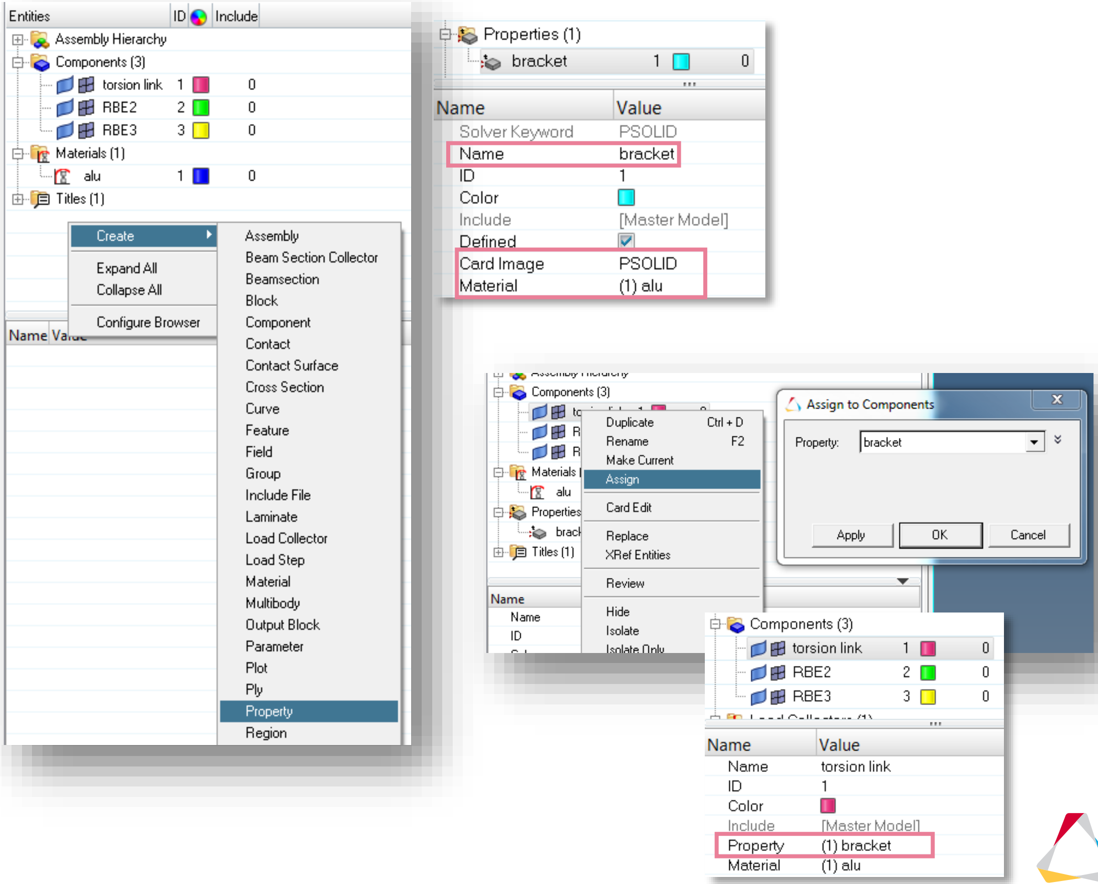
EXERCISE 2A: STATIC ANALYSIS OF A SOLID BRACKET

Hints (2/9)

4. 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

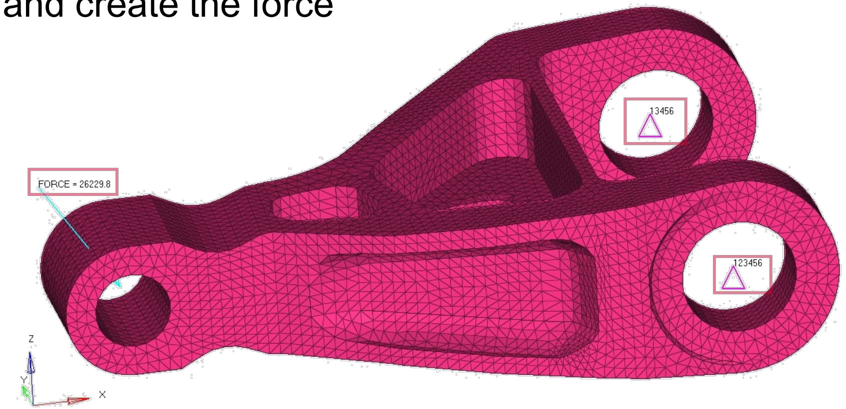
   assign property to `torsion link` component by right mouse click on the component, select ***Assign***

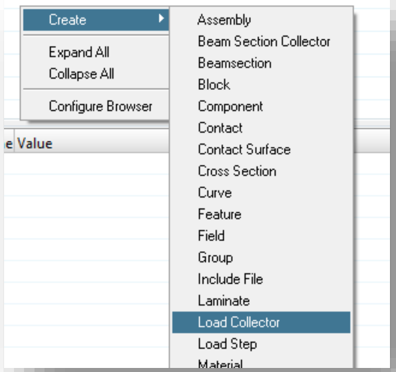


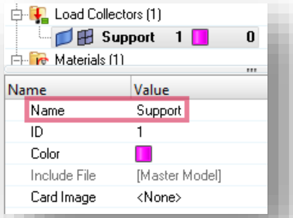
EXERCISE 2A: STATIC ANALYSIS OF A SOLID BRACKET

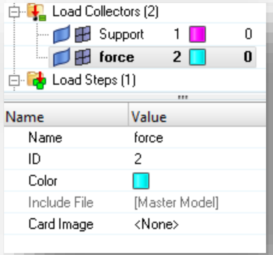
Hints (3/9)

5. Click right mouse button in the model browser, select ***Create*** → ***Load Collector*** and create the two SPCs
6. Click right mouse button in the model browser, select ***Create*** → ***Load Collector*** and create the force







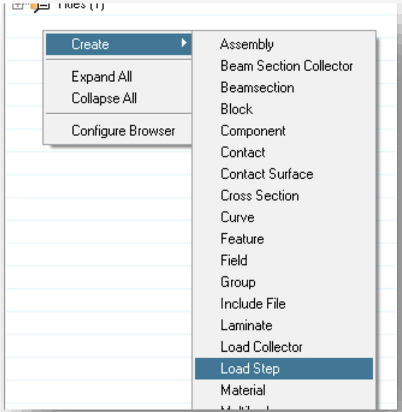


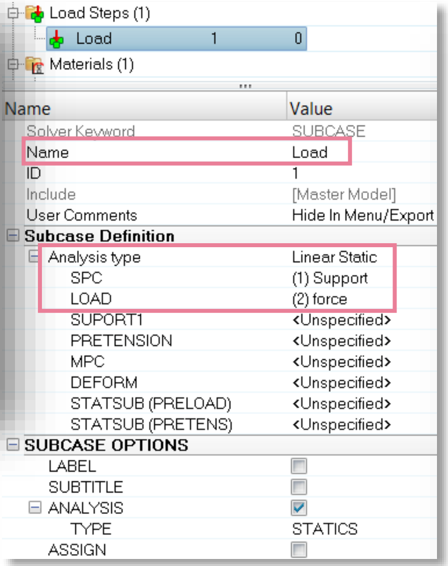
EXERCISE 2A: STATIC ANALYSIS OF A SOLID BRACKET

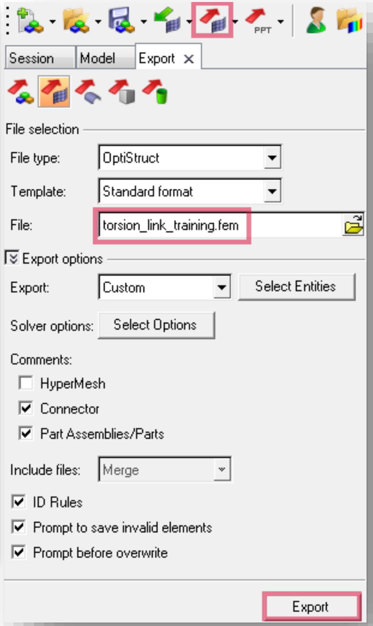
Hints (4/9)

7. Click right mouse button in the model browser, select ***Create*** → ***Load Step***

8. Click on button Solver Export Deck, choose file name and hit ***Export***







EXERCISE 2A: STATIC ANALYSIS OF A SOLID BRACKET

Hints (5/9)

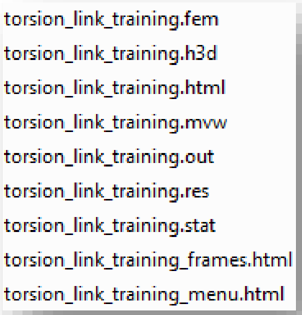
9. Exported `.fem` file without HyperMesh comments
    - Constraint reference
    - Load reference
    - Property reference
    - Material reference

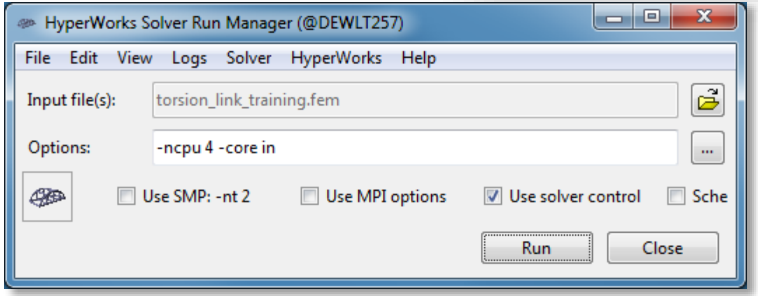
```
$$ optistruct
SUBCASE       1
  LABEL Load
ANALYSIS STATICS
  SPC =         1
  LOAD =         2
BEGIN BULK
GRID            1        -520.701-61.91252.0557-5
...
...
...
CTETRA     42822       1     5866    6434    10115    10208
PSOLID          1       1
MAT1            170000.0          0.33
SPC             1     4830  123456      0.0
SPC             1     4831   13456      0.0
FORCE           2       1        01.0     12000.0 12000.0 -20000.0
ENDDATA
```

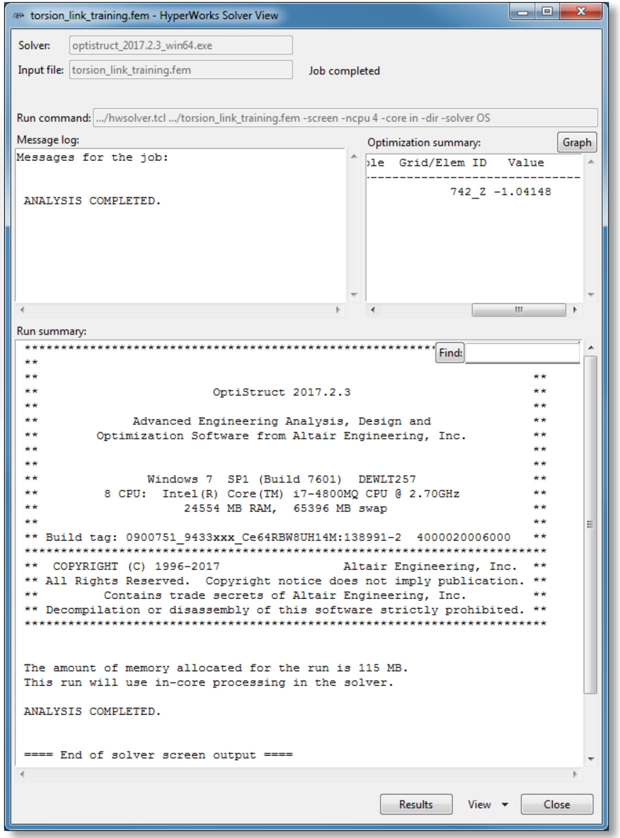
EXERCISE 2A: STATIC ANALYSIS OF A SOLID BRACKET

Hints (6/9)

10. Use the HyperWorks Solver Run Manager to run the exported `.fem` file, make sure that OptiStruct states “ANALYSIS COMPLETED” and review the created files.



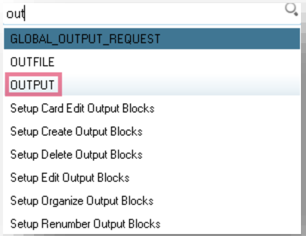


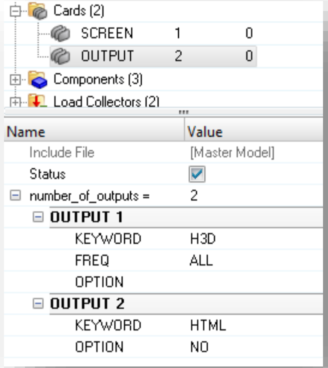


EXERCISE 2A: STATIC ANALYSIS OF A SOLID BRACKET

Hints (7/9)

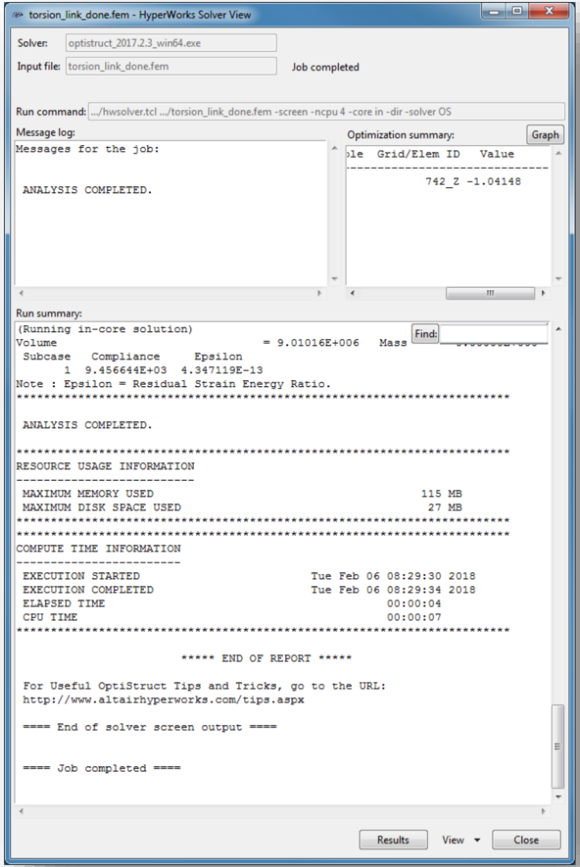
11. Use HyperMesh's Quick Access Tool (`Crtl+f`) to add control cards
    `SCREEN,OUT`
    `OUTPUT,H3D,ALL`
    `OUTPUT,HTML,,NO`





12. Review the created files and note the additional output in the HyperWorks Solver View

torsion_link_done.fem
torsion_link_done.h3d
torsion_link_done.mvw
torsion_link_done.out
torsion_link_done.stat



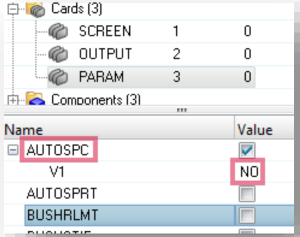
EXERCISE 2A: STATIC ANALYSIS OF A SOLID BRACKET

Hints (8/9)

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

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

14. 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.

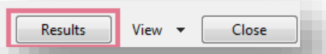



```
Element # 13814, element type TETRA.
WARNING - Outside of recommended range: Face Vertex Angle =    12.440
          lower limit =    15.000

NOTE : other similar error/warning messages were suppressed,
       use PARAM,CHECKEL,FULL to obtain full report


Element Quality Check Summary
-----------------------------
Total # of elements that exceeded recommended range (warning) =    19
Note: Only element with the highest violation of each check is listed below.

Recommended range violations:
-----------------------------------------------------------------------------
Element     Property            # of     Recommended Range    Max.   Viol.  Elem.
                                Viol.     Lower      Upper    Value  type    No.
------- ------------------- -------- --------- --------- --------- --- -------
TETRA   Face Vertex Angle         19     15.00    165.00     11.95  L    15597


List of Auto-SPC d.o.f.s for loadcase    1
Total number of Auto-SPC d.o.f.s = 1344

               ------------------------------
                 Grid No.          Component
               ------------------------------
                     3870             4 5 6
                     3871             4 5 6
                     3872             4 5 6
                     3873             4 5 6
                     3874             4 5 6
                     3875             4 5 6
                     3876             4 5 6
                     3877             4 5 6
                     3878             4 5 6
                     3879             4 5 6
                     3880             4 5 6
                     3881             4 5 6
```

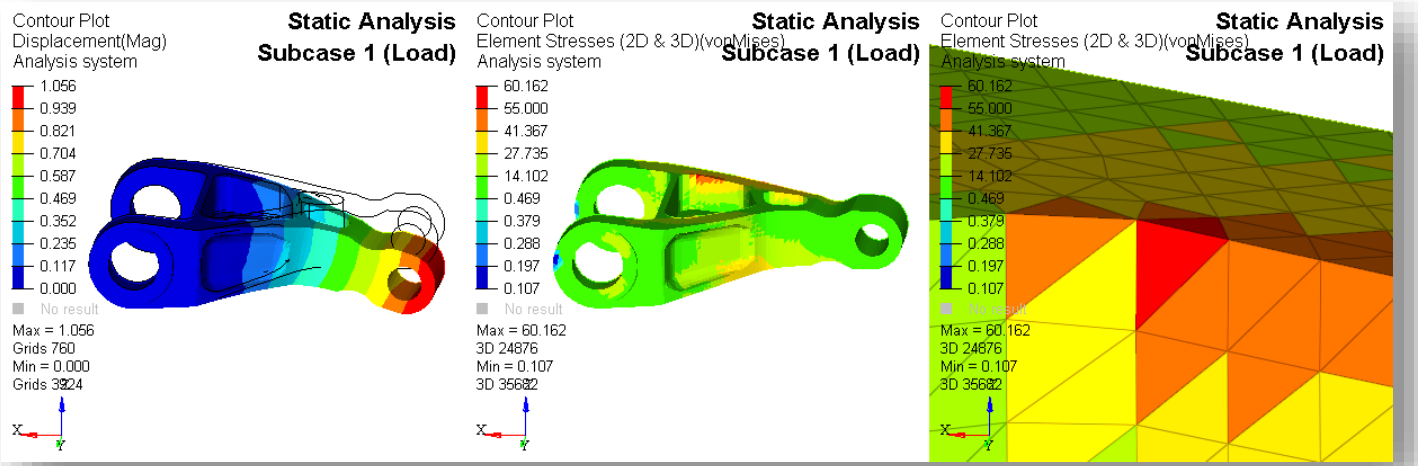
EXERCISE 2A: STATIC ANALYSIS OF A SOLID BRACKET

Hints (9/9)

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



17. 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.

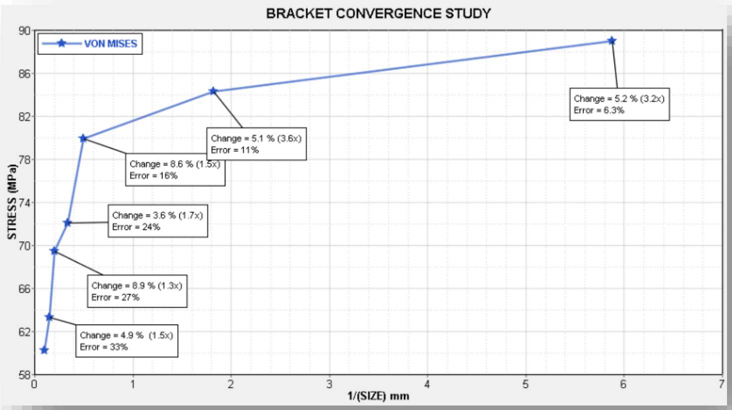


CONVERGENCE STUDY

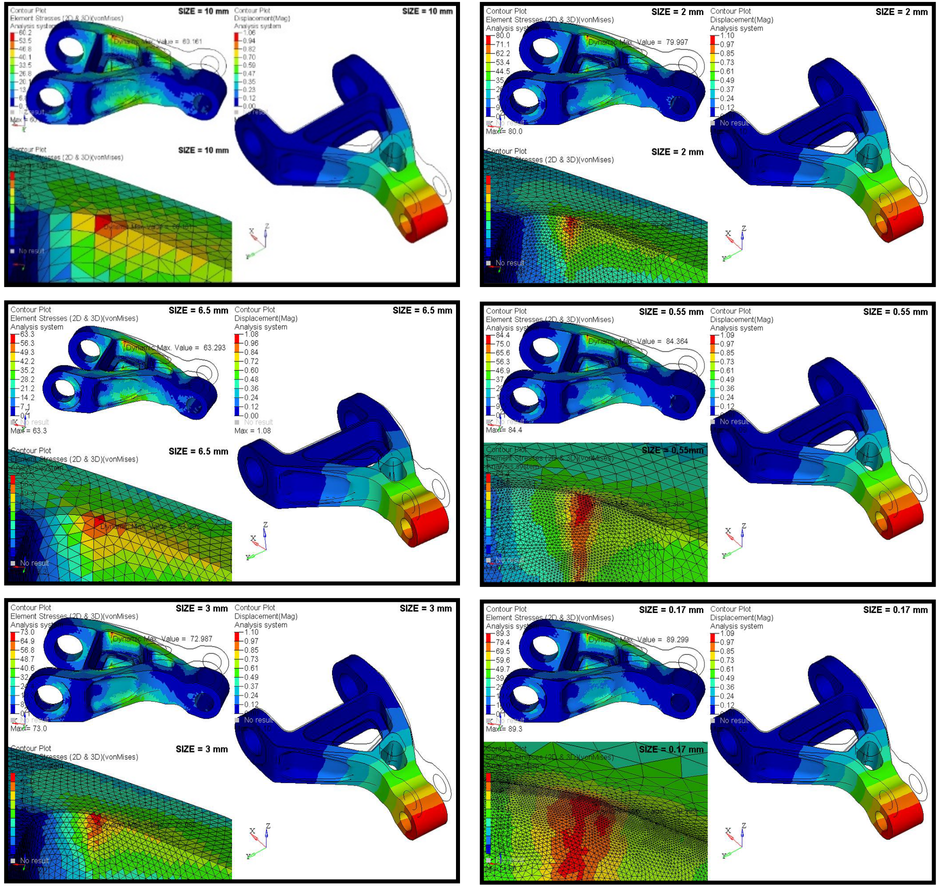
A finer mesh results on the one hand typically in a more accurate solution, one the other hand increases the computation time.

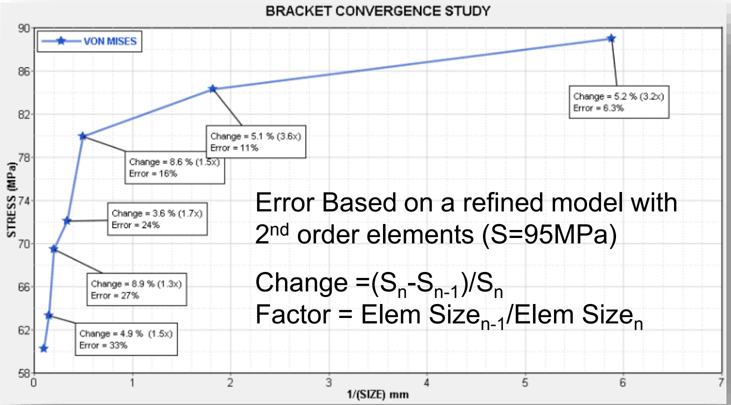
In order to get an idea of a finite element model that is good enough to predict an accurate solution for a problem with a reasonable model size, a convergence study can be performed:

- 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.



CONVERGENCE STUDY – EXAMPLE SOLID BRACKET



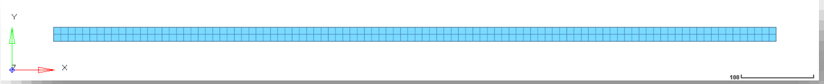


Element Size (mm)	Von Mises (MPa)	Displace-ment (mm)
10	60.2	1.06
6.5	63.3	1.08
5	69.5	1.09
3	73.0	1.10
2	80.0	1.10
0.55	84.4	1.09
0.17	89.3	1.09

EXERCISE 2B: STATIC ANALYSIS OF A SIMPLY SUPPORTED BEAM

File Name and Location

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



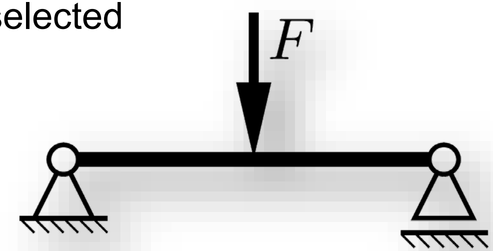
Objectives (1/3)

In this exercise, a structural analysis is performed on a simply supported beam:

- 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)
- Force of 1000 N in the center of the beam

The objective is to compare the results of the finite element model with the theoretical solution.

1. Open the model in HyperMesh Desktop with OptiStruct user profile selected
2. Review the model and check the dimensions of the model
3. Create a `MAT1` material `steel` for steel with the given properties
4. Create a `PSHELL` property `shell` with a thickness of 10 mm referencing material `steel` and assign it to the `beam` component



EXERCISE 2B: STATIC ANALYSIS OF A SIMPLY SUPPORTED BEAM

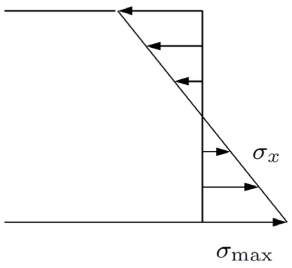
Objectives (2/3)

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`
6. Create a load collector `FORCE` (no card image) containing a force on node 53 with constant components {0, -1000, 0}
7. Create a load step `Load` of type Linear Static using `SPC` as `SPC` and `FORCE` as `LOAD` entry
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
9. Run the analysis with OptiStruct
10. Review the `.out` file wrt warnings, errors and Auto-SPC
11. Review the displacements and stresses in HyperView and check
    - max. displacement in y-direction
    - max. stress in (global) xx-direction

EXERCISE 2B: STATIC ANALYSIS OF A SIMPLY SUPPORTED BEAM

Objectives (3/3)

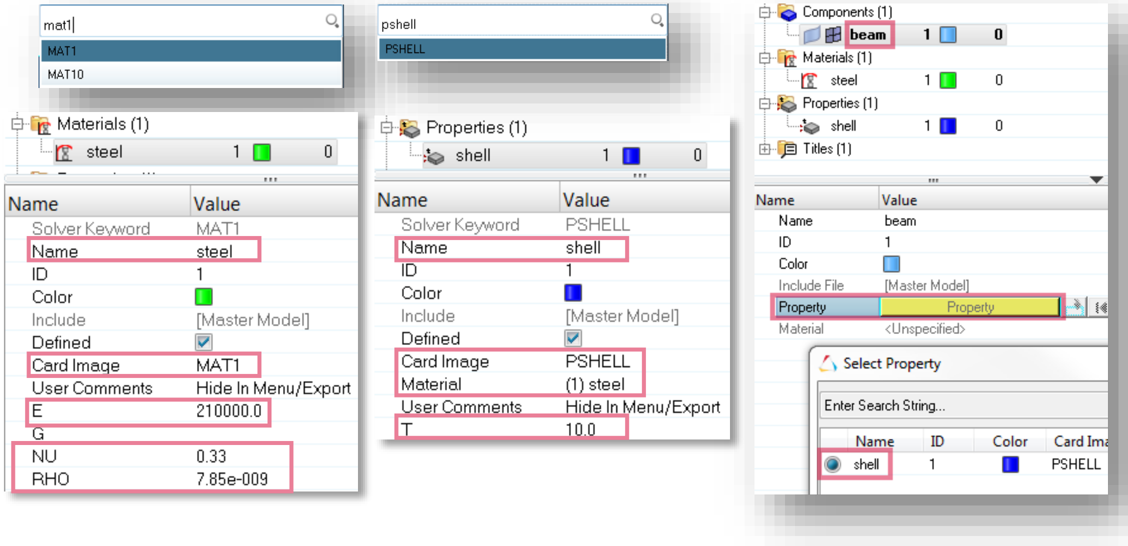
12. Calculate the theoretical results for
    - max. displacement in y-direction
    - max. stress in xx-direction
13. Add the global output request `STRESS(H3D,CORNER)=YES`
14. Rerun the analysis with OptiStruct
15. Review the stresses in HyperView and check
    - max. displacement in y-direction max. stress in (global) xx-direction



EXERCISE 2B: STATIC ANALYSIS OF A SIMPLY SUPPORTED BEAM

Hints (1/6)

2. 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
3. Use HyperMesh's Quick Access Tool (`Crtl+f`) to create according `MAT1` material card
4. Use HyperMesh's Quick Access Tool (`Crtl+f`) to create according `PSHELL` property card
   and
   assign property to `beam` component in the entity editor



EXERCISE 2B: STATIC ANALYSIS OF A SIMPLY SUPPORTED BEAM

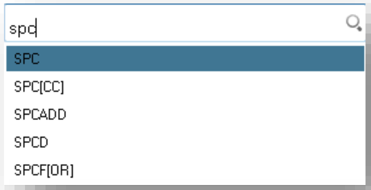
Hints (2/6)

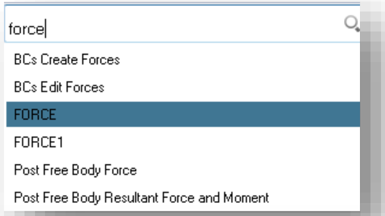
5. Use HyperMesh's Quick Access Tool (`Crtl+f`) with `SPC` to create the two SPCs

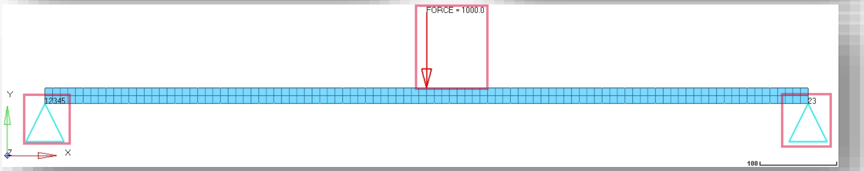
6. Use HyperMesh's Quick Access Tool (`Crtl+f`) with `LOAD` to create the load

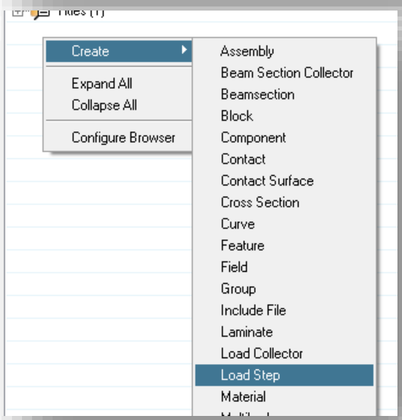
   Be aware, that loads and constraints can generate singularities. These can lead to a very high stress that is not physical, and appears only in the mathematical model

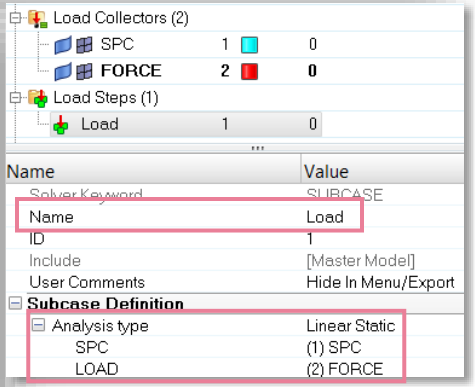
7. Click right mouse button in the model browser, select Create → Load Step and set Analysis type to Linear Static







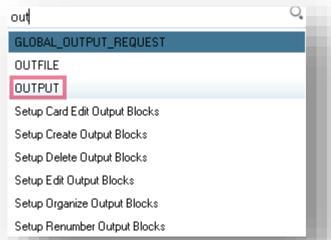


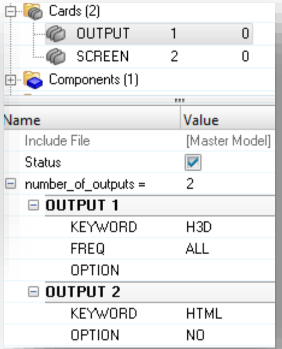


EXERCISE 2B: STATIC ANALYSIS OF A SIMPLY SUPPORTED BEAM

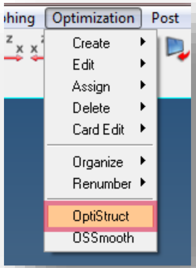
Hints (3/6)

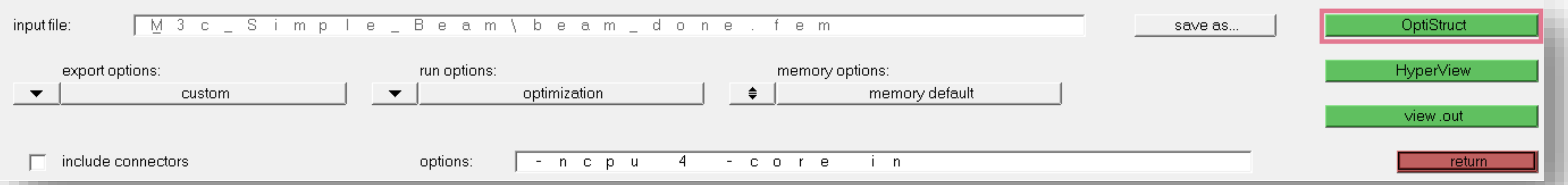
8. Use HyperMesh’s Quick Access Tool (`Crtl+f`) to add control cards
   `SCREEN OUT`
   `OUTPUT,H3D,ALL`
   `OUTPUT,HTML,,NO`





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

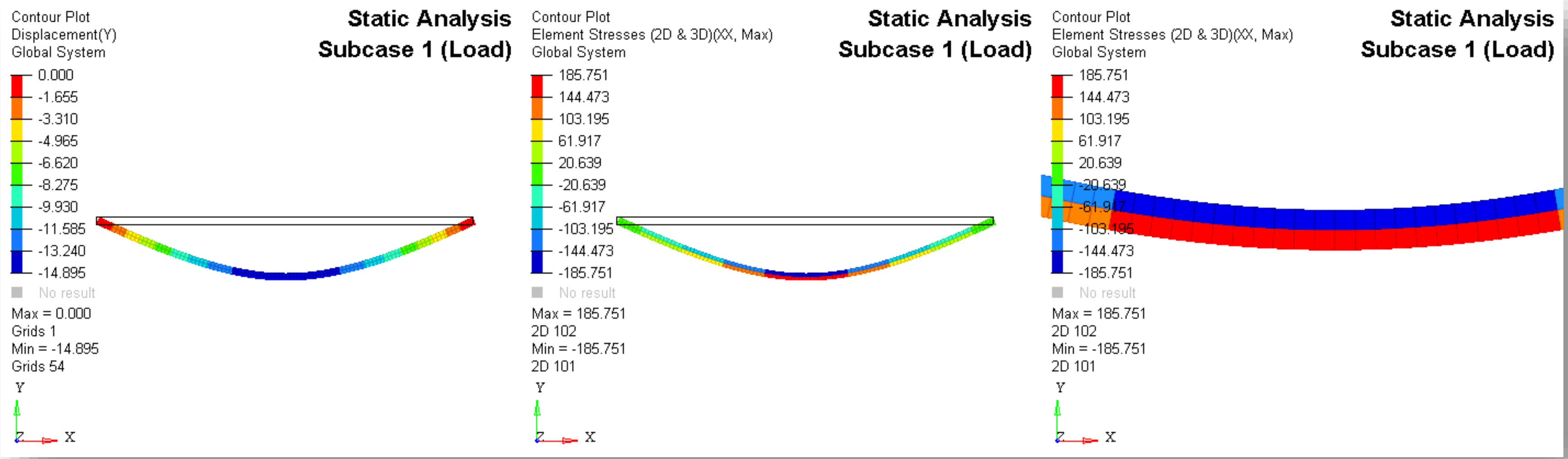




EXERCISE 2B: STATIC ANALYSIS OF A SIMPLY SUPPORTED BEAM

Hints (4/6)

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

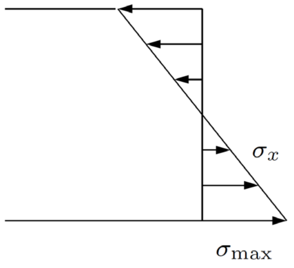


EXERCISE 2B: STATIC ANALYSIS OF A SIMPLY SUPPORTED BEAM

Hints (5/6)

12. $$u_{\max} = \frac{F\,l^3}{48\,EI} = \frac{F\,l^3}{48\,E\,\frac{t\,h^3}{12}} = \frac{F\,l^3}{4\,Eth^3} = \frac{-1000\cdot 1000^3}{4\cdot 210000\cdot 10\cdot 20^3} = 14.881\ \mathrm{mm}$$ versus 14.895 mm

$$\sigma_{\max} = \frac{M_{\max}\,z_{\max}}{I} = \frac{\frac{F\,l}{4}\left(\pm\frac{h}{2}\right)}{\frac{t\,h^3}{12}} = \pm\frac{3\,F\,l}{2\,th^2} = \pm\frac{3\cdot 1000\cdot 1000}{2\cdot 10\cdot 20^2} = \pm\,375\ \mathrm{MPa}$$ versus ±185.751 MPa



The displacement result of the analysis is very good with an error ~0.5%. However, the stress results look not good with an error superior to 50%.

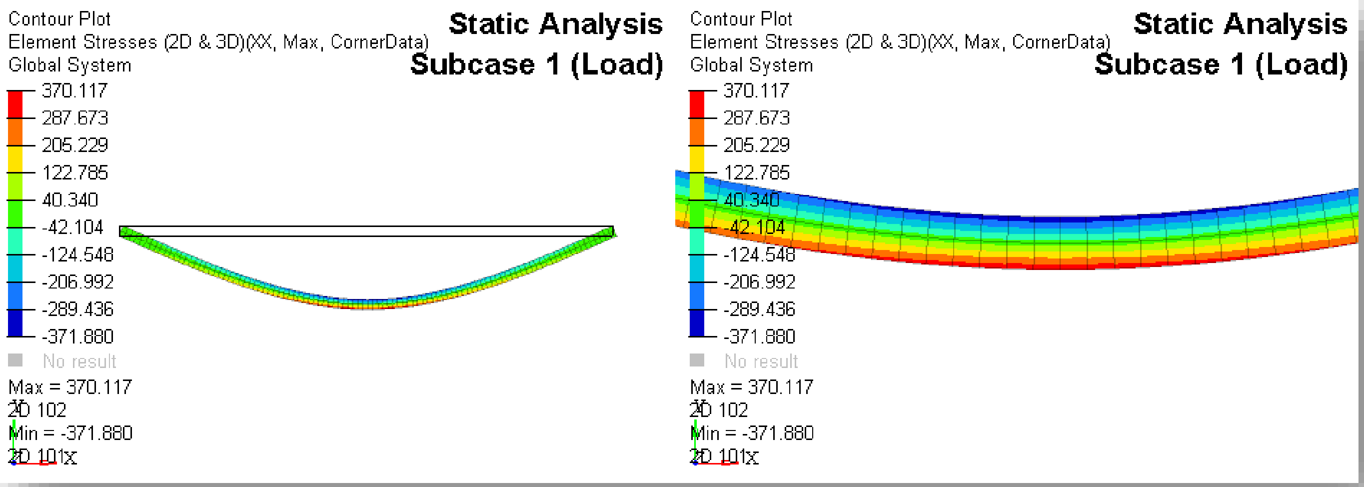
But as in OptiStruct element stresses for shell (and solid) elements are output at the element center only, you may not compare OptiStruct's stress result with σ at ± h/2, but ± h/4:

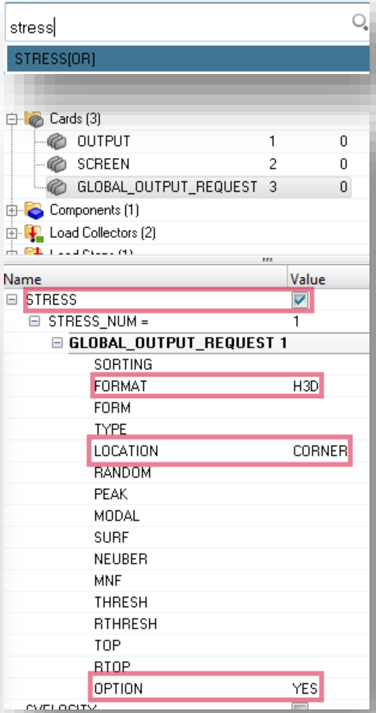
$$\sigma = \frac{M_{\max}\left(\pm\frac{h}{4}\right)}{I} = \pm\frac{3\,F\,l}{4\,th^2} = \pm\frac{3\cdot 1000\cdot 1000}{4\cdot 10\cdot 20^2} = \pm 187.5\ \mathrm{MPa}$$ versus ±185.751 MPa

EXERCISE 2B: STATIC ANALYSIS OF A SIMPLY SUPPORTED BEAM

Hints (6/6)

13. Use HyperMesh's Quick Access Tool (`Crtl+f`) with `STRESS` to add this control card/global output request `STRESS(H3D,CORNER)=YES`
15. The maximum stress is ±370 MPa versus ±375 MPa (theoretical result). Do not forget to set activate use corner data. ☑ Use corner data





QUESTIONS & ANSWERS

1. Which of the following is the most common reason for refining a linear static model's mesh?
   a) To lower the solution time to be able to get more runs within a project deadline
   b) To decrease the required memory to fit the solution cluster or analysis machine
   c) To more accurately capture part behavior at critical features or locations
   d) To reduce the size of the output files requested from the control cards
2. What will the results show as models are properly remeshed with finer elements?
   a) Lower and lower solution times down to less than ten seconds
   b) Smaller and smaller memory footprints per job leading to more jobs per machine
   c) Stresses, displacements, etc. will converge on the solution for that model setup
   d) The output files will have less data allowing you to archive whole projects on a flash drive

QUESTIONS & ANSWERS

3. Which property type would you expect to use in a model only containing solid elements?
   a) `PROD`
   b) `PSOLID`
   c) `PSHELL`
   d) a) and c)
   e) All of the above