

OPTISTRUCT FOR LINEAR ANALYSIS, V2019 CHAPTER 2: LINEAR STATIC 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

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

© 2019 Altair Engineering, Inc. Proprietary and Confidential. All rights reserved.

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:

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

- K global stiffness matrix
- x displacement vector response to be determined
- $f \quad \mbox{external forces vector applied to the structure}$

Example: a loaded bar





LINEAR STATIC ANALYSIS EXAMPLE

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



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

$$\mathbf{K}_{1} = \begin{bmatrix} \frac{210 * 314.16}{100} & -\frac{210 * 314.16}{100} \\ -\frac{210 * 314.16}{100} & \frac{210 * 314.16}{100} \end{bmatrix} = \begin{bmatrix} 659.74 & -659.74 \\ -659.74 & 659.74 \end{bmatrix} \qquad \mathbf{K}_{2} = \begin{bmatrix} \frac{210 * 78.54}{200} & -\frac{210 * 78.54}{200} \\ -\frac{210 * 78.54}{200} & \frac{210 * 78.54}{200} \end{bmatrix} = \begin{bmatrix} 82.47 & -82.47 \\ -82.47 & 82.47 \end{bmatrix} \qquad \mathbf{K}_{G} = \begin{bmatrix} 1 & 2 & 3 \\ 659.74 & -659.74 & 0 \\ -659.74 & 659.74 + 82.47 & -82.47 \\ 0 & -82.47 & 82.47 \end{bmatrix}$$

(...)

4. Forces and Displacements

$$\mathbf{f} = \begin{cases} \mathbf{0} \\ \mathbf{0} \\ -1\mathbf{0} \end{cases} \qquad \mathbf{x} = \begin{cases} x_1 \\ x_2 \\ x_3 \end{cases}$$



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.

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

$$\begin{bmatrix} 742.21 & -82.47 \\ -82.47 & 82.47 \end{bmatrix} \begin{cases} x_2 \\ x_3 \end{cases} = \begin{cases} 0 \\ -10 \end{cases}$$

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

364





© 2019 Altair Engineering, Inc. Proprietary and Confidential. All rights reserved.

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

<pre>\$\$ Template: optist \$\$</pre>	truct					
\$\$ \$\$ \$\$	Case Co	ontrol Ca	ards			
\$ \$HMNAME LOADSTEP \$ TITLE = Plate Model ANALYSIS FORMAT HM FORMAT H3D SCREEN NONE		1"Load"		3 Input/0	Output \$	Section
SUBCASE 1 LABEL Load ANALYSIS MODES SPC = 1 METHOD(STRUCTURE) =	3			Subca	se Infor	mation
<pre>S BEGIN BULK PARAM,AUTOSPC,YES PARAM,CHECKEL,YES \$ SYSTEM Data CORD2R 1 + -80.0 -20.0</pre>	20.0 0.0	-20.0	0.0	20.0	Bu -20.0	Ik Data -100.0

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

<pre>\$\$ Case Control Cards \$\$</pre>	<pre>\$\$ \$\$ Template: optist \$\$ \$\$</pre>	ruct					
<pre>\$\$</pre>	\$\$	Case Co	ontrol Ca	inds			
<pre>\$HMNAME LOADSTEP 1"Load" 3 \$ TITLE = Plate Model ANALYSIS Input/Output Section 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</pre>	\$\$ ¢						
TITLE = Plate Model ANALYSIS FORMAT HM 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	\$HMNAME LOADSTEP		1"Load"		3		
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	> TITLE = Plate Model ANALYSIS FORMAT HM FORMAT H3D SCREEN NONE				Input/C	Output	Section
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	SUBCASE 1 LABEL Load ANALYSIS MODES SPC = 1 METHOD(STRUCTURE) = \$ BEGIN BULK	3					
CORD2R 1 20.0 -20.0 0.0 20.0 -20.0 -100.0 + -80.0 -20.0 0.0	PARAM,AUTOSPC,YES PARAM,CHECKEL,YES \$ SYSTEM Data						
\$	CORD2R 1 + -80.0 -20.0 \$	20.0 0.0	-20.0	0.0	20.0	-20.0	-100.0

 \bigtriangleup

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.cm nie .amses_singularity.cmf file .bdst file .cntf file .contgap.fem file .cstr file .dens file .desvar file .dis.# file .disp file .dvarid file .echo file .els.# file .force file fsthick file .gpf file .arid 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

TT						
\$\$ Template: optist	ruct					
\$\$						
\$\$						
\$\$	Case C	ontrol Ca	rds			
\$\$						
\$						
\$HMNAME LOADSTEP		1"Load"		3		
\$						
TITLE = Plate Model						
ANALYSIS						
FORMAT HM						
FORMAT H3D						
SCREEN NONE						
SUBCASE 1						
SUBCASE 1 LABEL Load				Subca	ase Infor	mation
SUBCASE 1 LABEL Load ANALYSIS MODES				Subca	ase Infor	mation
SUBCASE 1 LABEL Load ANALYSIS MODES SPC = 1				Subca	ase Infor	mation
SUBCASE 1 LABEL Load ANALYSIS MODES SPC = 1 METHOD(STRUCTURE) =	3			Subca	ase Infor	mation
SUBCASE 1 LABEL Load ANALYSIS MODES SPC = 1 METHOD(STRUCTURE) = \$	3			Subca	ase Infor	mation
SUBCASE 1 LABEL Load ANALYSIS MODES SPC = 1 METHOD(STRUCTURE) = \$ BEGIN BULK	3			Subca	ase Infor	mation
SUBCASE 1 LABEL Load ANALYSIS MODES SPC = 1 METHOD(STRUCTURE) = \$ BEGIN BULK PARAM,AUTOSPC,YES	3			Subca	ase Infor	mation
SUBCASE 1 LABEL Load ANALYSIS MODES SPC = 1 METHOD(STRUCTURE) = \$ BEGIN BULK PARAM,AUTOSPC,YES PARAM,CHECKEL,YES	3			Subca	ase Infor	mation
SUBCASE 1 LABEL Load ANALYSIS MODES SPC = 1 METHOD(STRUCTURE) = \$ BEGIN BULK PARAM,AUTOSPC,YES PARAM,CHECKEL,YES \$ SYSTEM Data	3			Subca	ase Infor	mation
SUBCASE 1 LABEL Load ANALYSIS MODES SPC = 1 METHOD(STRUCTURE) = \$ BEGIN BULK PARAM,AUTOSPC,YES PARAM,CHECKEL,YES \$ SYSTEM Data CORD2R 1	20.0	-20.0	0.0	Subca 20.0	ase Infor	-100.0
SUBCASE 1 LABEL Load ANALYSIS MODES SPC = 1 METHOD(STRUCTURE) = \$ BEGIN BULK PARAM,AUTOSPC,YES PARAM,CHECKEL,YES \$ SYSTEM Data CORD2R 1 + -80.0 -20.0	3 20.0 0.0	-20.0	0.0	Subca 20.0	-20.0	mation -100.0
SUBCASE 1 LABEL Load ANALYSIS MODES SPC = 1 METHOD(STRUCTURE) = \$ BEGIN BULK PARAM,AUTOSPC,YES PARAM,CHECKEL,YES \$ SYSTEM Data CORD2R 1 + -80.0 -20.0 \$	3 20.0 0.0	-20.0	0.0	Subca 20.0	-20.0	mation -100.0

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

77						
\$\$ Template: optistr	uct					
\$\$						
\$\$						
\$\$	Case C	ontrol Ca	ards			
\$\$						
\$						
\$HMNAME LOADSTEP		1"Load"		3		
\$						
TITLE = Plate Model						
ANALYSIS						
FORMAT HM						
FORMAT H3D						
SCREEN NONE						
SUBCASE 1						
SUBCASE 1 LABEL Load				Subca	ise Infor	mation
SUBCASE 1 LABEL Load ANALYSIS MODES				Subca	ise Infor	mation
SUBCASE 1 LABEL Load ANALYSIS MODES SPC = 1				Subca	ise Infor	mation
SUBCASE 1 LABEL Load ANALYSIS MODES SPC = 1 METHOD(STRUCTURE) =	3			Subca	ise Infor	mation
SUBCASE 1 LABEL Load ANALYSIS MODES SPC = 1 METHOD(STRUCTURE) = \$	3			Subca	ise Infor	mation
SUBCASE 1 LABEL Load ANALYSIS MODES SPC = 1 METHOD(STRUCTURE) = \$ BEGIN BULK	3			Subca	ise Infor	mation
SUBCASE 1 LABEL Load ANALYSIS MODES SPC = 1 METHOD(STRUCTURE) = \$ BEGIN BULK PARAM,AUTOSPC,YES	3			Subca	ise Infor	mation
SUBCASE 1 LABEL Load ANALYSIS MODES SPC = 1 METHOD(STRUCTURE) = \$ BEGIN BULK PARAM,AUTOSPC,YES PARAM,CHECKEL,YES	3			Subca	ise Infor	mation
SUBCASE 1 LABEL Load ANALYSIS MODES SPC = 1 METHOD(STRUCTURE) = \$ BEGIN BULK PARAM,AUTOSPC,YES PARAM,CHECKEL,YES \$ SYSTEM Data	3			Subca	ise Infor	mation
SUBCASE 1 LABEL Load ANALYSIS MODES SPC = 1 METHOD(STRUCTURE) = \$ BEGIN BULK PARAM,AUTOSPC,YES PARAM,CHECKEL,YES \$ SYSTEM Data CORD2R 1	3	-20.0	0.0	Subca	-20.0	-100.0
SUBCASE 1 LABEL Load ANALYSIS MODES SPC = 1 METHOD(STRUCTURE) = \$ BEGIN BULK PARAM,AUTOSPC,YES PARAM,CHECKEL,YES \$ SYSTEM Data CORD2R 1 + -80.0 -20.0	3 20.0 0.0	-20.0	0.0	Subca	-20.0	mation -100.0
SUBCASE 1 LABEL Load ANALYSIS MODES SPC = 1 METHOD(STRUCTURE) = \$ BEGIN BULK PARAM,AUTOSPC,YES PARAM,CHECKEL,YES \$ SYSTEM Data CORD2R 1 + -80.0 -20.0 \$	3 20.0 0.0	-20.0	0.0	Subca	-20.0	mation -100.0

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, ...

\$\$						
<pre>\$\$ Template: optistr</pre>	uct					
\$\$						
\$\$						
\$\$	Case C	ontrol Ca	ards			
\$\$						
\$						
\$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					Bu	lk Data
\$ 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			

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 ${\tt INCLUDE}$ 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



PARAM, A	UTOSPC,YES						
PARAM, CI	HECKEL, YES						
\$ SYSTI	EM 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	178.53	8975490.8	734490.8	37340.0	981.746	9
\$							
MAT1	12100	00.0	0.3	7.901	E-09		
\$							
EIGRL	3			20			MAS
\$							
FORCE	2	11	01.0	0.0	0.0	-100.0	
\$							
SPC	1	31 123	4560.0				

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	Ν	v							

N	Name of Parameter
V	Value of Parameter

BEGIN B	ULK						
PARAM, A	UTOSPC,YES						
PARAM, C	HECKEL,YES						
\$ SYST	EM Data						
CORD2R	1	20.0	0 -20.0	0.0	20.0	-20.0	-100.0
+	-80.0 -20.	0.0					
\$							
GRID	1	50.0	-50.0	0.0			
GRID	2	50.0	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.	53975490.8	734490.	87340.0	981.746	59
\$							
MAT1	12100	00.0	0.3	7.901	E-09		
\$							
EIGRL	3			20			MASS
\$							
FORCE	2	11	01.0	0.0	0.0	-100.0	
Ş							
SPC	1	31 12	234560.0				
ENDDATA							
1							

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 CORD2P 1 20 0 -20 0 20 0 -20 0 -20 0 -100 0												
CORD2R	1	20.0	-20.0	0.0	20.0	-20.0	-100.0					
+	-80.0 -20.	0.0										
Ş												
GRID	1	50.0	-50.0	0.0								
GRID	2	50.0	-40.0	0.0								
Ş												
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	178.53	8975490.8	734490.8	87340.0	981.746	9					
\$												
MAT1	12100	00.0	0.3	7.901	E-09							
\$												
EIGRL	3			20			MASS					
\$												
FORCE	2	11	01.0	0.0	0.0	-100.0						
\$												
SPC	1	31 123	84560.0									
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)				
Ş								
GRID		1	50.	.0 -50.0	0.0			
GRID		2	50.	.0 -40.0	0.0			
Ş								
CBEAM	1	01	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.()	1		1	0.0
\$								
PBEAM		1	178.	.53975490.8	734490.	87340.0	981.74	69
\$								
MAT1		1210000	0.0	0.3	7.90	E-09		
\$								
EIGRL		3			20			MASS
\$								
FORCE		2	11	01.0	0.0	0.0	-100.0	
\$								
SPC		1	31 3	L234560.0				
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	СР	X1	X2	Х3	CD	PS		
ID		Unique grid point identification number. (Integer > 0)							
СР		Identification number of coordinate system in which the location of the grid point is defined. (Integer > 0 or blank)							
X1,X2,X	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)						on	
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, AUTOS PARAM, CHECH \$ SYSTEM I CORD2R + -80 \$	SPC,YES KEL,YES Data 1 D.0 -20.0	20.0 0.0	-20.0	0.0	20.0	-20.0	-100.0
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	178.53	3975490.8	734490.8	37340.0	981.746	9
MAT1 \$	1210000	0.0	0.3	7.901	5-09		
EIGRL \$	3			20			MASS
FORCE Ş	2	11	01.0	0.0	0.0	-100.0	
SPC ENDDATA	1	31 123	34560.0				

BULK DATA CARDS: ELEMENTS

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

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

Available elements include:

- **OD** CONM2, ...
- 1D CBEAM, CELAS2, CGAP, PLOTEL, RBE2, RBE3, ...
- 2D CQUAD8, CQUAD4, CTRIA6, CTRIA3, CSHEAR, ...
- **3D** CHEXA, CPENTA, CPYRA, CTETRA, ...

BEGIN B PARAM,A PARAM,C	ULK UTOSPC,YES HECKEL,YES						
\$ SYST	EM Data						
CORD2R	1	20.0	-20.0	0.0	20.0	-20.0	-100.0
+	-80.0 -20.	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	178.53	3975490.8	734490.	87340.0	981.746	59
\$							
MAT1	12100	0.00	0.3	7.90	E-09		
\$							
EIGRL	3			20			MASS
\$							
FORCE	2	11	01.0	0.0	0.0	-100.0	
\$							
SPC	1	31 123	34560.0				
ENDDATA							
1							

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 B	ULK						
PARAM, A	UTUSPC, IES						
C CVCT	ELCALL, ILS						
9 5151 CODD2D	IM Data	20.0	20.0	0 0	20.0	20.0	100 0
CORDZR	90 0 20 0	20.0	-20.0	0.0	20.0	-20.0	-100.0
т с	-80.0 -20.0	0.0					
CRID	1	50 0	-50 0	0 0			
GRID	2	50.0	-40.0	0.0			
S	2	50.0	-0.0	0.0			
CBEAM	101	1	94	550 0	1 0	0 0	
s	101	-	5.	000.0		0.0	
COUAD4	1	1	55	58	59	54	
COUAD4	2	1	54	59	50	51	
· ~ ·							
CQUAD4	3	1	58	43	44	59	
CQUAD4 \$	3	1	58	43	44	59	
CQUAD4 \$ PSHELL	3	1	58	43 1	44	59 1	0.0
CQUAD4 \$ PSHELL \$	3	1 11.0	58	43	44	59 1	0.0
CQUAD4 \$ PSHELL \$ PBEAM	3	1 11.0 178.5	58 3975490.8	43 1 734490.8	44	59 1 981.746	0.0
CQUAD4 \$ PSHELL \$ PBEAM \$	3 1 1	1 11.0 178.5	58 3975490.8	43 1 734490.8	44 37340.0	59 1 981.746	0.0
CQUAD4 \$ PSHELL \$ PBEAM \$ MAT1	3 1 1 1210000	1 11.0 178.5	58 3975490.8 0.3	43 1 734490.8 7.90E	44 37340.0 E-09	59 1 981.746	0.0
CQUAD4 \$ PSHELL \$ PBEAM \$ MAT1 \$	3 1 1 1210000	1 11.0 178.5 0.0	58 3975490.8 0.3	43 1 734490.8 7.90E	44 37340.0 2-09	59 1 981.746	0.0
CQUAD4 \$ PSHELL \$ PBEAM \$ MAT1 \$ EIGRL	3 1 1 1210000 3	1 11.0 178.5 0.0	58 3975490.8 0.3	43 1 734490.8 7.90E 20	44 37340.0 E-09	59 1 981.74	0.0 59 MASS
CQUAD4 \$ PSHELL \$ PBEAM \$ MAT1 \$ EIGRL \$	3 1 1 1210000 3	1 11.0 178.5	58 3975490.8 0.3	43 1 734490.8 7.90E 20	44 37340.0 2-09	59 1 981.74	0.0 59 MASS
CQUAD4 \$ PSHELL \$ PBEAM \$ MAT1 \$ EIGRL \$ FORCE	3 1 1 1210000 3 2	1 11.0 178.5 0.0	58 3975490.8 0.3 01.0	43 1 734490.8 7.90E 20 0.0	44 37340.0 2-09 0.0	59 1 981.746 -100.0	0.0 59 MASS
CQUAD4 \$ PSHELL \$ PBEAM \$ MAT1 \$ EIGRL \$ FORCE \$	3 1 1 1210000 3 2	1 11.0 178.5 0.0	58 3975490.8 0.3 01.0	43 1 734490.8 7.90F 20 0.0	44 37340.0 5-09 0.0	59 1 981.746 -100.0	0.0 59 Mass
CQUAD4 \$ PSHELL \$ PBEAM \$ MAT1 \$ EIGRL \$ FORCE \$ SPC	3 1 1 1210000 3 2 1	1 11.0 178.5 0.0 11 31 12	58 3975490.8 0.3 01.0 34560.0	43 1 734490.8 7.90F 20 0.0	44 37340.0 2-09 0.0	59 1 981.746 -100.0	0.0 59 Mass
CQUAD4 \$ PSHELL \$ PBEAM \$ MAT1 \$ EIGRL \$ FORCE \$ SPC ENDDATA	3 1 1 1210000 3 2 1	1 11.0 178.5 0.0 11 31 12	58 3975490.8 0.3 01.0 34560.0	43 1 734490.8 7.90F 20 0.0	44 37340.0 5-09 0.0	59 1 981.746 -100.0	0.0 59 Mass

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 B	ULK						
DADAM CI	UTOSPC, IES						
Ś SVST	EM Data						
CORD2R	I Data	20 0	-20 0	0 0	20 0	-20 0	-100 0
+	-80 0 -20 0	0.0	20.0	0.0	20.0	20.0	100.0
Ş	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
ş 	_						
PBEAM	1	178.5	3975490.8	734490.8	87340.0	981.746	9
Ş M2/01	101000	0 0	0.2	7 0.01	- 00		
MATI	121000	J.U	0.3	7.901	5-09		
₽ ETCDI	2			2.0			MACC
é ÉLGRE	3			20			MASS
FORCE	2	11	01 0	0 0	0 0	-100 0	
S	2	± ±	01.0	0.0	0.0	-100.0	
SPC	1	31 12	34560 0				
ENDDATA	Ŧ	JT 12.	51500.0				
BRODITA							



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 \rightarrow PSOLID
- CQUAD8, CQUAD4, CTRIA6, CTRIA3 → PSHELL
 OF PCOMP
- CSHEAR \rightarrow PSHEAR
- CBEAM \rightarrow PBEAM

BEGIN B	ULK						
PARAM, A	UTOSPC,YES						
PARAM, CI	HECKEL,YES						
\$ SYSTI	EM 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	
\$		\sim					
CQUAD4	1		55	58	59	54	
CQUAD4	2		54	59	50	51	
CQUAD4	3	(1)	58	43	44	59	
\$							
PSHELL		11.0		1		1	0.0
Ş							
PBEAM		178.53	975490.8	734490.8	37340.0	981.746	59
ş							
MAT1	1210000	.0	0.3	7.901	E-09		
Ş							
EIGRL	3			20			MASS
Ş							
FORCE	2	ΤT	01.0	0.0	0.0	-100.0	
⇒ a⊐a	1	21 102	45.00 0				
SPC	1	31 123	4360.0				
ENDDATA							

BULK DATA CARDS: PROPERTIES

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

PID	Unique shell element property identification number. (Integer > 0)
MID1	Material identification number for membrane. (Integer > 0)
т	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 BU	LK						
PARAM, AU	TOSPC,YES						
PARAM, CHI	ECKEL,YES						
\$ SYSTE	M Data						
CORD2R	1	20.0	-20.0	0.0	20.0	-20.0	-100.0
+ ·	-80.0 -20.0	0.0					
Ş							
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	178.53	3975490.8	734490.8	37340.0	981.740	59
\$							
MAT1	1210000	.0	0.3	7.901	E-09		
\$							
EIGRL	3			20			MASS
\$							
FORCE	2	11	01.0	0.0	0.0	-100.0	
\$							
SPC	1	31 123	34560.0				
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 $\ensuremath{\mathtt{MAT}}$ cards
- MATT1, MATT2, MATT8 **and** MATT9.

BEGIN B	ULK							
PARAM,A	UTOSPC,Y	YES						
PARAM,C	HECKEL,Y	YES						
\$ SYST	EM Data							
CORD2R		1	20	.0 -20.	0.0	20.0	-20.0	-100.0
+	-80.0	-20.0	0.0	0				
\$								
GRID		1	50	.0 -50.	0.0			
GRID		2	50	.0 -40.	0.0			
\$								
CBEAM	10	01	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	0	1		1	0.0
\$								
PBEAM		1	178	.53975490.	8734490.	87340.0	981.746	59
\$								
MAT1		1210000	0.0	0.3	7.901	E-09		
Ş								
EIGRL		3			20			MASS
Ş								
FORCE		2	11	01.0	0.0	0.0	-100.0	
Ş								
SPC		1	31 3	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		Е	G	NU	RHO	А	TREF	GE	
MID		Unique material identification number (Integer > 0)								
E		Yo	Young's Modulus							
G		Shear Modulus								
NU		Po	isson's l	Ratio						
RHO		Ma	iss dens	ity						
А		Th	ermal ex	pansi	on coef	ficient				
TREF		Reference temperature for thermal loading								
GE		Str	uctural I	Eleme	nt Dam	ping coef	ficient			

BEGIN B	ULK						
PARAM, A	UTOSPC,YES						
PARAM, CI	HECKEL, YES						
\$ SYST	EM Data			0.0			100.0
CORD2R	1	20.0	-20.0	0.0	20.0	-20.0	-100.0
+	-80.0 -20.0	0.0					
Ş	-	50.0	50.0	0.0			
GRID	1	50.0	-50.0	0.0			
GRID	2	50.0	-40.0	0.0			
Ş	1.0.1	1	0.4		1 0	0.0	
CBEAM	101	T	94	550.0	1.0	0.0	
2 COULD D 4	1	1		FO	FO	E 4	
CQUAD4	1	1	55	58	59	54	
CQUAD4	2	1	54	59	50	51	
CQUAD4	3	1	28	43	44	59	
Ş DOUDT I	1			1		1	0 0
C	± /	<u> </u>		T		Ţ	0.0
PDEAM	1	10 53	075400 0	724400 0	0 0 0 224	0.01 746	0
C	1	J	9/3490.0	/34490.0	5/340.0	901.740	9
	1310000		0.2	7 0.01	- 00		
MAII ¢			0.5	7.901	2-09		
PICDI	2			20			MACC
EIGRL C	5			20			MASS
FORCE	2	11	01 0	0 0	0 0	-100 0	
¢ rokce	2	± ±	01.0	0.0	0.0	-100.0	
SPC	1	31 123	4560 0				
FNDDAWA	1	JI 123					
BNUDAIA							

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 B	ULK						
PARAM, A	UTOSPC,YES						
PARAM, C	HECKEL,YES						
\$ SYST	EM 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			
Ş							
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	178.53	3975490.8	734490.8	37340.0	981.746	59
ş							
MAT1	121000	0.0	0.3	7.901	E-09		
Ś							
EIGRI	З			20			MASS
Ś							
FORCE	2	11	01.0	0.0	0.0	-100.0	
Ş							
SPC	1	31 123	34560.0				
ENDDATA							
2							

(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 B PARAM,A	ULK UTOSPC,YES						
PARAM, C	HECKEL,YES						
\$ SYST	EM 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			
Ş							
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	178.5	3975490.8	734490.8	87340.0	981.74	69
\$							
MAT1	1210000	0.0	0.3	7.901	E-09		
\$							
EIGRL	3			20			MASS
\$							
FORCE	2	11	01.0	0.0	0.0	-100.0	
\$							
SPC	1	31 12	34560.0				
ENDDATA							

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 B	ULK						
PARAM, A	UTOSPC,YES						
PARAM, C	HECKEL,YES						
\$ SYST	EM 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	
Ş							
PSHELL	1	11.0		1		1	0.0
\$							
PBEAM	1	178.53	975490.8	734490.8	37340.0	981.740	59
\$							
MAT1	121000	0.0	0.3	7.901	E-09		
\$							
EIGRL	3			20			MASS
Ş							
FORCE	2	11	01.0	0.0	0.0	-100.0	
ş	1	21 102					
SPC ENDDAWA	1	51 123	4360.0				
ENDDATA							

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

SID	Identification number of single-point constraint set. (Integer > 0)
G	Grid or scalar point identification number. (Integer > 0)
с	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 B	ULK UTOSPC,YES						
PARAM, C	HECKEL, YES						
Ś SYST	EM Data						
CORD2R	1	20	0 -20 0	0 0	20.0	-20 0	-100.0
+	-80 0 -3	20.0 0.0	20.0	0.0	20.0	2010	100.0
s							
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.	53975490.87	734490.8	37340.0	981.746	9
Ş							
MAT1	123	L0000.0	0.3	7.901	E-09		
\$							
EIGRL	3			20			MASS
\$							
FORCE	2	11	01.0	0.0	0.0	-100.0	
Ş							
SPC	1	31 1	234560.0				
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 / Time²
- As an illustration:

Mass	Length	Young' s Modulus	Density	Force	Stress	Time
kg	m	Ра	kg/m³	Ν	Ра	S
kg	mm	GPa	kg/mm ³	kN	GPa	ms
t	mm	MPa	t/mm ³	Ν	MPa	S

© 2019 Altair Engineering, Inc. Proprietary and Confidential. All rights reserved.

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
M∨W	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	НМ	OP2	РСН	OPT	PAT	Controlling I/O Option
Nodal Displacements	•	•	•	•	•	•	DISPLACEMENT
Element Strain Energy	•	•	•	•	0	0	ESE
Element Stresses	•	•	•	•	•	•	STRESS / ELSTRESS
Element Strains	•	•	•	•	•	0	STRAIN
Ply Stresses	•	•	•	•	•	0	<u>CSTRESS</u>
Composite Failure Indices	•	•	•	0	•	0	<u>CFAILURE</u>
Ply Strains	•	•	•	•	•	0	<u>CSTRAIN</u>
Element Forces	•	•	•	•	•	0	FORCE / ELFORCE
Grid Point Stresses	•	•	•	•	0	0	GPSTRESS / GSTRESS
SPC Forces	•	•	•	•	•	0	SPCFORCE
MPC Forces	0	0	•	•	•	0	MPCFORCE
Grid Point Forces	•	0	•	•	•	0	<u>GPFORCE</u>
Applied Loads	•	0	•	•	•	0	<u>OLOAD</u>
Contact Force and Pressure	•	0	٠	0	0	0	CONTE

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 <u>Results Output by OptiStruct</u> 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

© 2019 Altair Engineering, Inc. Proprietary and Confidential. All rights reserved.

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 \rightarrow 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



© 2019 Altair Engineering, Inc. Proprietary and Confidential. All rights reserved.





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

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 . ${\tt out}$ 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

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.

🖹 alu

📩 🦳 Titlas (1) .

Name

Name

Color

Defined

Include File

Card Image

ID.

Ε

G

NU

סנוח

- 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

mat1	Q
MAT1	
MAT10	



Hints (2/9)

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

Entities		ID 📀	Inclu	ıde		
🕀 즳 Ass	embly Hierarchy				ſ	🖹 💫 Prop
🕂 🍋 Con	nponents (3)					i 🕹
💋	🛃 torsion link	1 🔲		0		:
💋	🛃 RBE2	2 📘		0	1	lame
	🛃 RBE3	3 📒		0		Solverk
🕂 🙀 Mat	erials (1)					Name
2	alu	1 📘		0		ID
🗄 🎁 Title	es (1)					Color
	- ·	_		4 11		Include
	Lreate			Assembly Deven Continue Collector		Defined
	Expand All			Deam Section Collector		Card Im
	Collapse All			Black		Materia
	Configure Br	oweer	1	Component		
Name Varu	coningate bi	011301		Contact	100	
				Contact Surface		
				Cross Section		
				Curve		
				Feature	100	
			_	Field		
				Group	100	
			-	Include File	100	
				Laminate		. ₽ - 1
				Load Collector	100	
				Load Step	100	
				Material	100	
				Multibody		Nam
				Output Block		
				Parameter		
				Plot		
				Ply		
				Property	- 10	
		_		neyion		

📸 Properties (1)	
😓 bracket	1 🚺 0
ame	Value
Solver Keyword	PSOLID
Name	bracket
ID	1
Color	
Include	[Master Model]
Defined	
Card Image	PSOLID
Material	(1) alu



Hints (3/9)

- Click right mouse button in the model browser, select Create → Load Collector and create the two SPCs
- Click right mouse button in the model browser, select Create → Load Collector and create the force



Beam Section Collector
Beamsection Block Component Contact Contact Surface Cross Section Curve Feature Field Group Include File Laminate Load Collector Load Step



Hints (4/9)

- Click right mouse buttor in the model browser, select *Create* → *Load Step*
- 8. Click on button Solver Export Deck, choose file name and hit *Export*

Create 🕨 🕨	Assembly		0	
Expand All	Beam Section Collector	🕀 ॡ Materials (1)		Session Model Export X
Collanse All	Beamsection			
Collapse All	Block	Name	Value	
Configure Browser	Component	Solver Keyword	SUBCASE	File selection
	Contact	Name	Load	File type: OptiStruct
	Contact Surface	ID	1	
	Cross Section	Include	[Master Model]	Template: Standard format 💌
	Curve	User Comments	Hide In Menu/Export	The Institute Fill And the form
	Feature	Subcase Definition		File: torsion_link_training.rem
	Field	🗧 Analysis type	Linear Static	l v Export options
	Group	SPC	(1) Support	Europh Custom - Select Entition
	Include File	LOAD	(2) force	
	Laminate	SUPORT1	<unspecified></unspecified>	Solver options: Select Options
	Load Collector	PRETENSION	<unspecified></unspecified>	
	Load Step	MPC	<unspecified></unspecified>	Comments:
	Material	DEFORM	<unspecified></unspecified>	🗖 HyperMesh
	ka hit i	STATSUB (PRELOAD)	<unspecified></unspecified>	Connector
		STATSUB (PRETENS)	<unspecified></unspecified>	Part Assemblies /Parts
		SUBCASE OPTIONS		
		LABEL		Include files: Merge 👻
		SUBTITLE		
		ANALYSIS		I✓ ID Rules
		TYPE	STATICS	Prompt to save invalid elements
		ASSIGN		Prompt before overwrite

Hints (5/9)

9. Exported . fem file without HyperMesh comments

- Constraint reference
- Load reference
- Property reference
- Material reference



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.

torsion_link_training.fem torsion_link_training.h3d torsion_link_training.html torsion_link_training.mvw torsion_link_training.out torsion_link_training.res torsion_link_training.stat torsion_link_training_frames.html torsion_link_training_menu.html

HyperWorks S	olver Run Manag	er (@DEWLT257)		
File Edit View	/ Logs Solver	HyperWorks Help		
Input file(s):	torsion_link_train	ning.fem		Ž
Options:	-ncpu 4 -core in			
🥮 🗆 L	Jse SMP: -nt 2	Use MPI options	Use solver control	Sche
			Run C	lose

Sel		2.2		1						
oiver:	optistruct_2017	2.3_winb4.exe								
nput file:	torsion_link_tra	ining.fem		Job compl	eted					
tun comm	hand:/hwsolve	r.tcl/torsion_	ink_training.fe	m -screen -ncj	ou 4 -ce	ore in -dir -si	olver OS		_	_
/lessage lo	og:				Optin	nization sum	imary:		Grap	bh
lessage.	s for the]	: 00		<u> </u>	ple	Grid/El	em ID	Value		^
ANALYS	IS COMPLETE	Π.					742_Z	-1.04148		
(•			III	F	
lun summ	iary:									
*****	*******	******	********	******	*****	***** Fin	id:			4
**								**		
**		0	ptiStruct	2017.2.3				**		
**	Ad	vanced Eng	ineering J	analysis, 1	Desig	n and		**		
**	Optimiz	ation Soft	ware from	Altair En	ginee	ring, In	c.	**		
**								**		
**		Windows 7	SP1 (Buil	d 7601)	DEWLT	257		**		
**	8 CPU:	Intel(R) 24554	Core(TM) MB RAM,	17-4800MQ 65396 MB	CPU	@ 2.70GH	z	**		
**			,					**		Ξ
** Bui	1d tag: 090	0751_9433x	xx_Ce64RBW	8UH14M:13	8991- *****	-2 40000	200060	*******		
** CO	PYRIGHT (C)	1996-2017		Al	tair	Engineer	ing, I	inc. **		
** All	Rights Res Contai	erved. Co	pyright no ecrets of	tice does Altair En	not	imply pu ring. In	blicat c.	ion. **		
** Dec	ompilation	or disasse	mbly of th	is softwa	re st	rictly p	rohibi	.ted. **		
*****	*********	*******	********	******	*****	******	*****	******		
The am This r	ount of men un will use	ory alloca in-core p	ted for th rocessing	in the so	115 M lver.	1B.				
ANALYS	15 COMPLETE	.D.								
7.	nd of sol									
E	nu or solve	i screen o	ucput -200						h	*
					_					_

Hints (7/9)

11. Use HyperMesh's Quick Access Tool (Crtl+f)



🖨 🗑 Cards (2)	
CREEN	1 0
🖉 OUTPUT	2 0
🕀 🛜 Components (3)	
🛱 📭 Load Collectors (2)	
Name	Value
Include File	[Master Model]
Status	
number_of_outputs =	2
OUTPUT 1	
KEYWORD	H3D
FREQ	ALL
OPTION	
OUTPUT 2	
KEYWORD	HTML
OPTION	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

oher: optistruct_2017.2.3_win64.exe paut file tersion_link_done.fem Job completed un command: _/hwsolver.tcl _/torsion_link_done.fem -screen -ncpu 4 -core in -dir -solver OS tersage log: Optimization summary: Complete essages for the job: ANALYSIS COMPLETED. ANALYSIS COMPLETED. Un summary: Execution = Residual Strain Energy Ratio. ANALYSIS COMPLETED. ESOURCE USAGE INFORMATION EXECUTION STARTED COMPLETED.	n_ink_done.rem - nyperworks solver view	
un command: Job completed an command: ./hwsolver.td/torsion_link_done.fem -screen -ncpu 4 -core in -dir -solver OS lessage log: Optimization summary: seaagues for the job: Image: screen -ncpu 4 -core in -dir -solver OS ansLYSIS COMPLETED. Image: screen -ncpu 4 -core in -dir -solver OS ansummary: Optimization summary: Running in-core solution) Image: screen -ncpu 4 -core in -dir -solver OS olume -9.01016E-006 1 9.456442F.03 4.947119E-13 ote: Epsilon = Residual Strain Energy Ratio. scource USAGE INFORMATION HAXIMUM DEMORY USED 115 MB MAXIMUM MEMORY USED 115 MB MAXIMUM MEMORY USED 115 MB MAXIMUM MEMORY USED Tue Feb 06 08:29:30 2018 CMUCIE TIME INFORMATION O0:00:04 CHUTINE O0:00:07 CHUTINE O0:00:07 CHUTINE OO:00:07 CHUTINE OO:00:07 CHUTINE Chut CHUTINE Chut CHUTINE OF REPORT ***** For Useful OptiStruct Tips and Tricks, go to the URL: http://www.altarkhyperworks.com/tips.aspx <th>optistruct_2017.2.3_win64.exe</th> <th></th>	optistruct_2017.2.3_win64.exe	
un command: _/hwsolver.td/torsion_link_done.fem -screen -ncpu 4 -core in -dir -solver OS terage log: Optimization summary:	e: torsion_link_done.fem Job completed	
un command:/hwsolvertd/torsion_link_done.fem -screen -ncpu 4 -core in -dir -solver OS tetasge log essages for the job: ANALYSIS COMPLETED. ANALYSIS COMPLETED. ANALYSIS COMPLETED. un summany: Emming in-core solution) olume		
Hessge log: Optimization summary: Graph descages for the job: 1 1 Value 742_Z -1.09148 aNALYSIS COMPLETED. 742_Z -1.09148 742_Z -1.09148 742_Z -1.09148 un summary: = 9.01016E+006 Mass Find subcase Completed 1 9.456644E+03 4.347119E-13 oce: Epsilon 1 9.456644E+03 4.347119E-13 oce: Epsilon 115 MB 27 MB ESOURCE USAGE INFORMATION 115 MB 27 MB MAXIMUM MEMORY USED 115 MB 20106 MAXIMUM MEMORY USED 115 MB 0010004 CHUTHE INFORMATION The Feb 06 08:29:30 2018 00100104 CHUTHE INFORMATION 002:00:03 002:00:04 CHUTHE INFORMATION The Feb 06 08:29:30 2018 002:00:04 CHUTHE Feb 06 08:29:34 2028 002:00:04 CHUTHE CHUTHE 002:00:04 CHUTHE CHUTHE 002:00:04 CHUTHE CHUTHE 002:00:04 CHUTHE <t< td=""><td>nmand:/hwsolver.tcl/torsion_link_done.fem -screen -ncpu 4 -core in -dir -solver OS</td><td></td></t<>	nmand:/hwsolver.tcl/torsion_link_done.fem -screen -ncpu 4 -core in -dir -solver OS	
ANRLYSIS COMPLETED. ansames for the job: ansames for the job: a		anh
ANALYSIS COMPLETED. Unsummary: Running in-core solution) olume = 9.01016E+006 Mass Find Subcase Compliance Epsilon 1 9.456644E+03 4.97119E-13 ore : Epsilon = Residual Strain Energy Ratio. ANALYSIS COMPLETED. ESOURCE USAGE INFORMATION ESOURCE USAGE INFORMATION ESOURCE USAGE INFORMATION EXECUTION STARTED Tue Feb 06 08:29:30 2018 EXECUTION STARTED Tue Feb 06 08:29:30 2018 EXECUTION COMPLETED. EXECUTION COMPLETED Tue Feb 06 08:29:30 2018 EXECUTION STARTED Tue Feb 06 08:29:30 2018 EXECUTION STARTED Tue Feb 06 08:29:30 2018 EXECUTION COMPLETED Tue Feb 06 08:29:30 2018 EXECUTION STARTED TUE Feb 06 08:29:30 2018 EXECUTI	ges for the job:	apri
ANALYSIS COMPLETED. 742_Z -1.09148 742_Z -1.	Jie Gild/Elem ib Value	- 1
ANALYSIS CONTRILLS. Wun summany: Running in-core solution) 1 9.95644Er03 4.347119E-13 ore : Epsilon = Residual Strain Energy Ratio. ANALYSIS COMPLETED. ANALYSIS COMPLETED. ANALYSIS COMPLETED. MAXIMUM MEMORY USED 115 MB MAXIMUM MEMORY USED 27 MB 	YSTS COMPLETED 742_Z -1.04148	
un summany: Running in-core solution) olume = 9.01016E+006 Mass Find Subcase Compliance Epsilon 1 9.45644E+03 4.347119E-13 ore : Epsilon = Residual Strain Energy Ratio. ANALYSIS COMPLETED. ESOURCE USAGE INFORMATION ESOURCE USAGE INFORMATION ESOURCE USAGE INFORMATION MAXIMUM DISK SPACE USED 115 MB MAXIMUM DISK SPACE USED 27 MB COMPUTE INFORMATION EXECUTION COMPLETED. EXECUTION COMPLETED Tue Feb 06 08:29:30 2018 COMPUTE INFORMATION EXECUTION COMPLETED Tue Feb 06 08:29:30 2018 ELAPSED TIME 001:00:01 CONTINUE 000:00:07 CONTINUE 000:00:07 CONTINUE 000:00:07 CONTINUE 000:001 CONTINUE	ISIS COMPLETED.	
<pre>w summary: w summary: Nunning in-core solution) Subcase Compliance Epsilon 1 9.45644E+03 4.347119E-13 ote : Epsilon = Residual Strain Energy Ratio. ANALYSIS COMPLETED. ESOURCE USAGE INFORMATION MAXIMUM DESORY USED 115 MB MAXIMUM DESORY USED 27 MB COMPUTE TIME INFORMATION EXECUTION CONFLICTED Tue Feb 06 08:29:30 2018 ELAPSED TIME 00:00:01 CUU TIME 00:00:00:01 CUU TIME 00:00:00:01 CUU TIME 00:00:00:00 CUU TIME 00:00:00:00 CUU TIME 00:00:00:00 CUU TIME 00:00:00 CUU TIME 00:00:00 CUU TIME 00:00:00:00 CUU TIME 00:00:00 CUU TIME 00:00 CUU TIME 00:00:00 CUU TIME 00:00 CUU TIME 00:00 CUU TIME 00:00 CUU TIME 00:00 CUU TIME 00:00 CUU TIME 00:00 CUU TIME 00:00 CUU</pre>		
<pre>wsummary: Running in-core solution) olume = 9.01016E+006 Mass End Subcase Compliance Epsilon = 9.01016E+006 Mass End Subcase Compliance Epsilon = 1.9.00016E+006 Mass End Escurce UsAde INFORMATION ESCURCE USAde INFORMATION MAXIMUM MEMORY USED 115 MB MAXIMUM MEMORY USED 2.7 MB Compose Information Execution Completed USED Tue Feb 06 08:29:30 2018 ELAPSED TIME 00:00:00 CONTINUE OF REPORT ***** For Useful OptiStruct Tips and Tricks, go to the URL: http://www.altairhyperworks.com/tips.aspx ==== End of solver screen output ==== ==== Job completed =====</pre>		
<pre>mmuning in-core solution) Subcase Compliance Epsilon 1 9.486448+03 4.347119E-13 Over : Desilon - Residual Strain Energy Ratio. ANALYSIS COMPLETED. ESOURCE USAGE INFORMATION EXCURNE USAGE INFORMATION EXCURNE INFORMATION EXECUTION STARTED EXECUTION COMPLETED Twe Feb 06 08:29:30 2018 ELABSED TIME O0:00:01 Commented Strain Energy Ratio CFU TIME O0:00:01 Commented Strain Energy Ratio CFU TIME CFU Definition CFU Definition</pre>		
wsummany: Summing in-core solution) Subcase Compliance Epsilon 1 9.4566442+03 4.347119E-13 ote : Epsilon - Residual Strain Energy Ratio. ANALYSIS COMPLETED. ESOURCE USAGE INFORMATION MAXIMUM DENGRU USED 115 MB MAXIMUM DENGRU USED 27 MB COMPUTE TIME INFORMATION EXECUTION CONFLICED Tue Feb 06 08:29:30 2018 ELAPSED TIME 00:00:07 CUT TIME 00:00:07 For Useful OptiStruct Tips and Tricks, go to the URL: http://www.altain/perworks.com/tips.aspx ==== End of solver screen output ====		
usummary: Running in-core solution) 0lume = 9.01016E+006 Mass Subcase Compliance Epsilon = Residual Strain Energy Ratio. ANALYSIS COMPLETED. ESOURCE USAGE MAXIMUM MEMORY USED 115 MB MAXIMUM MEMORY USED 27 MB MAXIMUM DISK SPACE USED 27 MB MAXIMUM DISK SPACE USED 27 MB MAXIMUM MEMORY USED 115 MB MAXIMUM DISK SPACE USED 27 MB MAXIMUM DISK SPACE USED 27 MB MAXIMUM DISK SPACE USED 00:00:04 CONFUTE TIME WO:00:04 CUTIME 00:00:07 FLAPSED TIME 00:00:07 CUTIME 00:00:07 For Useful OptiStruct Tips and Tricks, go to the URL: http://www.altairhyperworks.com/tips.aspx ==== Job completed ====		
UNSUMMARY: Running in-core solution) = 9.01016E+006 Mass Find 1 9.456644E+03 4.347119E-13 occ : Epsilon = Residual Strain Energy Ratio. ANALYSIS COMPLETED. ESOURCE USAGE INFORMATION MAXIMUM MEMORY USED 115 MB NAXIMUM DISK SPACE USED 27 MB COMPUTE TIME INFORMATION EXECUTION STARTED Tue Feb 06 08:29:30 2018 EXECUTION STARTED Tue Feb 06 08:29:34 2018 EXECUTION STARTED Tue Feb 06 08:29:34 2018 EXECUTION STARTED Tue Feb 06 08:29:34 2018 COMPUTE TIME INFORMATION EXECUTION STARTED Tue Feb 06 08:29:34 2018 CONFUTE TIME INFORMATION EXECUTION COMPLETED Tue Feb 06 08:29:34 2018 CONFUTE TIME 00:00:01 From Started Time Conference Confer	· · · · · · · · · · · · · · · · · · ·	
AMINING INFORMATION Subcase Compliance Epsilon = 9.01016E+006 Mass Find 1 9.456442F.03 4.347119E-13 Ote : Epsilon = Residual Strain Energy Ratio. ANALYSIS COMPLETED. ANALYSIS COMPLETED. MAXIMUM MEMORY USED 115 MB MAXIMUM MEMORY USED 27 MB MAXIMUM MEMORY USED 27 MB MAXIMUM MEMORY USED 27 MB MAXIMUM MEMORY USED 27 MB MAXIMUM MEMORY USED 2018 EXECUTION CONFLETED Tue Feb 06 08:29:30 2018 ELAPSED TIME 00:00:07 CUT TIME 00:00:07 END OF REPORT For Useful OptiStruct Tips and Tricks, go to the URL: http://www.altainbygerworks.com/tips.aspx ==== Job completed ====	imary:	
Subcase Compliance Epsilon in 9, 936644E-03, 4.97(1)82-13 ore : Epsilon = Residual Strain Energy Ratio. ANALYSIS COMPLETED. ESOURCE USAGE INFORMATION MAXIMUM MEMORY USED 115 MB MAXIMUM DISK SPACE USED 27 MB MAXIMUM DISK SPACE USED 27 MB MAXIMUM DISK SPACE USED 27 MB MAXIMUM DISK SPACE USED 2010 EXECUTION COMPLETED Tue Feb 06 08:29:30 2018 ELAPSED TIME 00:00:07 END OF REPORT For Useful OptiStruct Tips and Tricks, go to the URL: http://www.altairhyperworks.com/tips.aspx ==== Job completed ====	= 9.01016E+006 Mass	Î
1 9.456644F-03 4.347119E-13 Ore : Epsilon = Residual Strain Energy Ratio. ANALYSIS COMPLETED. ESOURCE USAGE INFORMATION MAXIMUM DISK SPACE USED 115 MB MAXIMUM DISK SPACE USED 27 MB 	ase Compliance Epsilon	
ote : Epsilon = Residual Strain Energy Ratio. ANALYSIS COMPLETED. ESOURCE USAGE INFORMATION MAXIMUM MEMORY USED 115 MB MAXIMUM DISK SFACE USED 27 MB MAXIMUM DISK SFACE USED 2018 EXECUTION COMPLETED Tue Feb 06 08:29:30 2018 ELAPSED TIME 00:00:01 CUE TIME 00:00:01 CUE TIME 00:00:07 CUE TIME 00:07 CUE TIME 00:07	1 9.456644E+03 4.347119E-13	
ANALYSIS COMPLETED. ESOURCE USAGE INFORMATION MAXIMUM MEMORY USED 115 MB 27 MB MAXIMUM DISK SPACE USED 27 MB MAXIMUM DISK SPACE USED 27 MB MAXIMUM DISK SPACE USED 2018 The Feb 06 08:29:34 2018 DOI:00.010 The Feb 06 08:29:34 2018 The Feb 06 08:29:34 2018 The Feb 06 08:29:34 2018 The Feb	: Epsilon = Residual Strain Energy Ratio.	
ANALYSIS COMPLETED. ESOURCE USAGE INFORMATION MAXIMUM MEMORY USED MAXIMUM MEMORY USED MAXIMUM MEMORY USED MAXIMUM DISK SPACE USED MAXIMUM DISK SPACE USED MAXIMUM DISK SPACE USED Tue Feb 06 08:29:30 2018 ELAPSED TIME ELAPSED TIME CONVENTION From END OF REPORT ***** For Useful OptiStruct Tips and Tricks, go to the URL: http://www.altain/uperworks.com/tips.aspx ==== End of solver screen output ==== ==== Job completed ====	***************************************	
ESOURCE USAGE INFORMATION ESOURCE USAGE INFORMATION MAXIMUM MEMORY USED 115 MB MAXIMUM DISK SPACE USED 27 MB CMFUTE INFORMATION EXECUTION STARTED Tue Feb 06 08:29:30 2018 ELAPSED TIME 00:00:01 CU TIME 00:00:01 END OF REPORT For Useful OptiStruct Tips and Tricks, go to the URL: http://www.altairhyperworks.com/tips.aspx ==== Ich of solver screen output ==== ==== Job completed ====	YSIS COMPLETED.	
ESOURCE USAGE INFORMATION MAXIMUM MEMORY USED 115 MB MAXIMUM DISK SPACE USED 27 MB COMPUTE TIME INFORMATION EXECUTION STARTED Tue Feb 06 08:29:34 2018 EXECUTION COMPLETED Tue Feb 06 08:29:34 2018 CONTRACT ON CONTRACT		
SOURCE USAGE INFORMATION HAXIMUM MEMORY USED 115 MB MAXIMUM DISK SFACE USED 27 MB CMFUTE TIME INFORMATION EXECUTION STARTED Tue Feb 06 08:29:30 2018 ELAPSED TIME 00:00:01 CUU TIME 00:00:07 END OF REPORT For Useful OptiStruct Tips and Tricks, go to the URL: http://www.altairhyperworks.com/tips.aspx End of solver screen output ==== 	************************	
NALIMUM MEMORY USED 115 MB 27 MB NALIMUM DISK SPACE USED 27 MB COMPUTE TIME INFORMATION EXECUTION STARTED Tue Feb 06 08:29:30 2018 EXECUTION STARTED Tue Feb 06 08:29:34 2018 EXECUTION COMPLETED 00:00:00 CHU TIME 00:00:00 CHU TIME 00:00:00 END OF REPORT For Useful OptiStruct Tips and Tricks, go to the URL: http://www.altairhyperworks.com/tips.aspx ==== End of solver screen output ==== ==== Job completed ====	RCE USAGE INFORMATION	
ANNINGE DISK SPACE USED 27 MB 27 MB COMPUTE TIME INFORMATION EXECUTION STARTED Tue Feb 06 08:29:30 2018 EXECUTION STARTED Tue Feb 06 08:29:34 2018 00:00:04 00:00:04 00:00:07 END OF REPORT For Useful OptiStruct Tips and Tricks, go to the URL: http://www.altairhygerworks.com/tips.aspx End of solver screen output For Useful OptiStruct Tips and Tricks, go to the URL: http://www.altairhygerworks.com/tips.aspx 	VIII MEMORY USED 115 MB	
CMEUTE TIME INFORMATION EXECUTION STARTED EXECUTION CONFLICTED Tue Feb 06 08:29:30 2018 ELAPSED TIME COUTINE COUTINE COUTINE For Useful OptiStruct Tips and Tricks, go to the URL: http://www.altairhyperworks.com/tips.aspx ==== End of solver screen output ==== ==== Job completed ====	MUM DISK SPACE USED 27 MB	
OMFORE TIME INFORMATION EXECUTION STARTED Tue Feb 06 08:29:30 2018 EXECUTION COMPLETED Tue Feb 06 08:29:34 2018 ELAPSED TIME 00:00:01 CFU TIME 00:00:01 	***********	
OMPORT TIME INFORMATION EXECUTION STARTED Tue Feb 06 08:29:30 2018 ELAPSED TIME 00:00:01 CUU TIME 00:00:07 THE END OF REPORT ***** For Useful OptiStruct Tips and Tricks, go to the URL: http://www.altairhyperworks.com/tips.aspx ==== End of solver screen output ==== ==== Job completed ====	***************************************	
EXECUTION STARTED Tue Feb 06 08:29:30 2018 EXECUTION COMPLETED Tue Feb 06 08:29:30 2018 Tue Feb 06 08:29:34 2018 00:00:01 CFU TIME 00:00:01 For Useful OptiStruct Tips and Tricks, go to the URL: http://www.altairhyperworks.com/tip.aspx End of solver screen output	TE TIME INFORMATION	
EXECUTION COMPLETED THE FE 06 08:29:34 2018 00:00:04 00:00:05 THE FE 06 08:29:34 2018 00:00:04 00:00:07 THE FE 06 08:29:34 2018 00:00:07 THE FE 06 08:29:34 2018 00:00:07 THE FE 06 08:29:34 2018 00:00:04 THE FE 06 08:29:34 2018 00:00:04 THE FE 06 08:29:34 2018 THE FE 06 08:29:34 2	THE Feb 06 08:20:30 2018	
ELAPSED TIME 00:00:04 CFU TIME 00:00:07 END OF REPORT For Useful OptiStruct Tips and Tricks, go to the URL: http://www.altairhyperworks.com/tips.aspx ==== End of solver screen output ==== ==== Job completed ====	UTION COMPLETED Tue Feb 06 08:29:34 2018	
CFU TIME 00:00:07 END OF REPORT For Useful OptiStruct Tips and Tricks, go to the URL: http://www.altairhyperworks.com/tips.aspx End of solver screen output	SED TIME 00:00:04	
END OF REPORT For Useful OptiStruct Tips and Tricks, go to the URL: http://www.altairhyperworks.com/tips.aspx ==== End of solver screen output ==== ==== Job completed ====	FIME 00:00:07	
For Useful OptiStruct Tips and Tricks, go to the URL: http://www.altairhyperworks.com/tips.aspx For d solver screen output ==== Job completed ==== Pande View a Chee	***************************************	
For Useful OptiStruct Tips and Tricks, go to the URL: http://www.altairhyperworks.com/tips.aspx ==== End of solver screen output ==== ==== Job completed ====	***** END OF REPORT *****	
<pre>ror userul optistruct isps and iricks, go to the UKL: http://www.blaithypervorks.com/tips.aspx ==== End of solver screen output ==== ==== Job completed ====</pre>		
==== End of solver screen output ====	Jerui optistruct lips and fricks, go to the URL:	
==== End of solver screen output ====		
Job completed	//www.arcarrayperworks.com/crps.aspk	
Bandko View z Class	End of solver screen output ====	
Bank View - Clas	End of solver screen output ====	100
Pandro View - Chre	End of solver screen output ====	
Pavalter View - Class	Job completed ====	E
Parulta View - Clara	Job completed ====	
Parultz View - Clara	Job completed ====	
Results View + Close	Job completed ====	H +

Name AUTOSPC

> V1 AUTOSPRT BUSHRLMT

0

0

Π

Value

NO

2

3

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.

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 bel Recommended range violations: 	low.
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 bel Recommended range violations: 	low.
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 bel Recommended range violations: Element Property # of Recommended Range Max. Viol. Uiol. Lower Upper Value type TETRA Face Vertex Angle 19 15.00 165.00 11.95 L List of Auto-SPC d.o.f.s for loadcase 1 Total number of Auto-SPC d.o.f.s = 1344 Grid No. Component	low.
Total # of elements that exceeded recommended range (warning) = 19 Note: Only element with the highest violation of each check is listed bel Recommended range violations: Element Property # of Recommended Range Max. Viol. Viol. Lower Upper Value type TETRA Face Vertex Angle 19 15.00 165.00 11.95 L List of Auto-SPC d.o.f.s for loadcase 1 Total number of Auto-SPC d.o.f.s = 1344 Grid No. Component	low.
Total # of elements that exceeded recommended range (warning) = 19 Note: Only element with the highest violation of each check is listed bel Recommended range violations: Element Property # of Recommended Range Max. Viol. Viol. Lower Upper Value type 	low.
Recommended range violations: Element Property # of Recommended Range Max. Viol. Viol. Lower Upper Value type TETRA Face Vertex Angle 19 15.00 165.00 11.95 L List of Auto-SPC d.o.f.s for loadcase 1 Total number of Auto-SPC d.o.f.s = 1344 	
Element Property # of Recommended Range Max. Viol. Viol. Lower Upper Value type TETRA Face Vertex Angle 19 15.00 165.00 11.95 L List of Auto-SPC d.o.f.s for loadcase 1 Total number of Auto-SPC d.o.f.s = 1344 Grid No. Component	
Viol. Lower Upper Value type TETRA Face Vertex Angle 19 15.00 165.00 11.95 L List of Auto-SPC d.o.f.s for loadcase 1 Total number of Auto-SPC d.o.f.s = 1344 Grid No. Component	Elem.
TETRA Face Vertex Angle 19 15.00 165.00 11.95 L List of Auto-SPC d.o.f.s for loadcase 1 Total number of Auto-SPC d.o.f.s = 1344 	No.
List of Auto-SPC d.o.f.s for loadcase 1 Total number of Auto-SPC d.o.f.s = 1344 Grid No. Component	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	
Total number of Auto-SPC d.o.f.s = 1344 Grid No. Component	
Grid No. Component	
dila 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 456	
2000 4.5.6	

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



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/mm³)
- 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



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 $\tt H3D$ format and suppress the $\tt html$ 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

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





🗄 🔁 Components (1)

🗜 beam

1 0

ID

Color

Π

0

-> 14

Card Ima

PSHELL

EXERCISE 2B: STATIC ANALYSIS OF A SIMPLY SUPPORTED BEAM

Hints (1/6)

- The length system is reasonable to be millimeter, the consistent units are mm, MPa, N. 2. 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

mat1

Use HyperMesh's Quick Access 4. Tool (Crtl+f) to create according PSHELL property card and assign property to beam component

in the entity editor

🗄 🙀 Materials (1) MAT10 👔 steel 1 🗄 🌇 Properties (1) 🗄 🙀 Materials (1) 📩 shell 🗄 🚵 Properties (1) 1 🔲 🗄 间 Titles (1) 🖹 steel 0 🖢 shell 1 🔲 0 Name Value Value Value Name Name Solver Keyword PSHELL Name beam Solver Keyword MAT1 ID. 1 Name shell Name steel Color ID. ID. 1 Include File [Master Model] Color Color Property Include [Master Model] Include [Master Model] Material <Unspecified> Defined \checkmark Defined \checkmark Card Image PSHELL Select Property Card Image MAT1 Material (1) steel User Comments Hide In Menu/Export User Comments Hide In Menu/Export Enter Search String. 210000.0 G Т 10.0 Name NU 0.33 shell RHO 7.85e-009

pshell

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

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



hing

Create

Edit

\$

4

Hints (3/6)

input file:

•

export options:

include connectors

custom

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

M3c_Simple_Beam\beam_done

•

run options:

options

optimization

- n c p u



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)

2.
$$u_{\text{max}} = \frac{F l^3}{48 E l} = \frac{F l^3}{48 E \frac{t h^3}{12}} = \frac{F l^3}{4 E t h^3} = \frac{-1000 \cdot 1000^3}{4 \cdot 210000 \cdot 10 \cdot 20^3} = 14.881 \text{ mm}$$
 versus 14.895 mm

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

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 σ at ± h/2, but ± h/4:

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



 $\sigma_{
m max}$

0

0

Π

Value

H3D

YES

CORNER

stress

🗇 🏀 Cards (3)

C SCREEN

Components (1)
 Generation (1)
 Generation (1)

GLOBAL OUTPUT REQUEST 3

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



