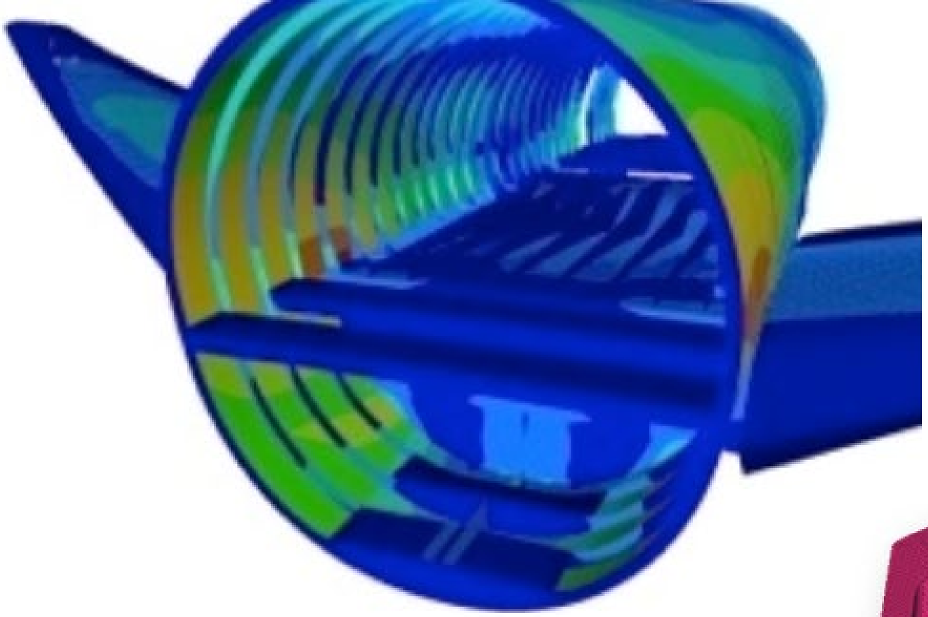
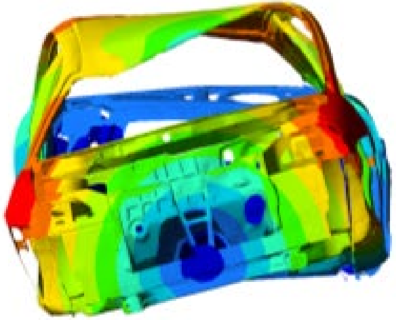
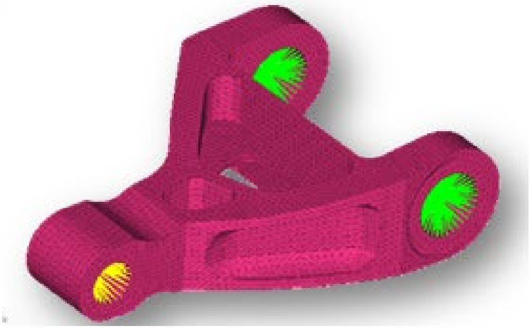






OPTISTRUCT FOR LINEAR ANALYSIS, V2019
CHAPTER 7: ADVANCED TOPICS

Altair

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

DEBUGGING GUIDE

GENERAL SANITY CHECKS

- Does a checkrun complete successfully (`-check`)?
- Are there any relevant warnings in the `.out` file?
- Does the model checker in HM show any issues (Tools → Model Checker)?
  - Check for any modeling errors (mesh discontinuities)
- Are the units consistent (mass, applied forces, etc. in the `.out` file make sense)?
- Does Groundcheck complete successfully (`GROUNDCHECK` Subcase Information Entry)? Check the element IDs for which the test fails.
- Plot animation (scale) will shown an mistakes in engineering judgement.
  - Missing boundary conditions or loading conditions
  - Material and property definitions
  - Element quality
  - Mass properties

GENERAL SANITY CHECKS

- Does a normal modes analysis show any issues? The number of rigid body modes should be as expected.
  - If free-free, are there six "rigid-body" (freq=0.0) modes?
  - Are there any mechanisms (freq=0.0)?
    - More than six "rigid-body" modes in free-free?
    - Any "rigid-body" modes in constrained modes?
- Is force balance satisfied (epsilon in out file)? Epsilon should be numerically zero.
  - Epsilon > 10E-9 may indicate trouble
- Check load paths – use grid point force balance to "trace" loads
  - Check stress contours for "consistency"
    - "Sharp" corners indicate bad modeling
    - Check stress discontinuities
- Try different memory options such as `-core in`, `-core out`, `-fixlen xy`, `-len xy`, `-maxlen xy`

DEBUGGING FOR NORMAL MODES ANALYSIS

- Try AMSES (`EIGRA`) and LANCZOS (`EIGRL`) – are results comparable?
- Try AMLS (`EIGRL` and `PARAM,AMLS,YES`) if available
- Try LANCZOS with `PARAM,AMLS,2` to enforce constraint reduction
- AMSES and AMLS will catch massless mechanisms automatically. It also outputs and constrains those DOFs
- Check if upper bound on `EIGRL`/`EIGRA` card is reasonable
- Try a non-blank for `ND`, `V1` and `V2` on `EIGRL`/`EIGRA` card
- Try small `V2`, e.g. 10 Hz (models with low `ND` but high `V2` might still fail due to too many modes)

DEBUGGING FOR INERTIA RELIEF ANALYSIS

- When comparing two models with `PARAM,INREL,-2` note that the models should
  - Have the same stresses and compliance
  - But not necessarily the same displacements

DEBUGGING FOR BUCKLING ANALYSIS

- Buckling is a very sensitive analysis. Even if the linear static run seems fine, small modelling issues can become apparent in buckling analyses.
- Run normal modes and make sure there are no rigid body modes.
- Try `PARAM,SHPBCKOR,2`
  - order of approximation used in plate bending geometric stiffness for linear shell elements
  - For 2, no transverse shear considered, only bending. Better for thin shells

DEBUGGING FOR HEAT TRANSFER ANALYSIS

- Nodal temperature input is defined thru `SPC` w/o dofs
- `CHBDYE` definition
- Temperature dependent conductivity only works with `NLHEAT`
- `TABLEM1` in `MATT4` defines multipliers, not actual conductivity
- Conductance/area is required in `PCONTHT` since v14.210

PARAMETERS

INTRODUCTION PARAMS

Parameters along with the parameters values are used generally in the Bulk Data entries to manage or control or requesting special features.

These Parameters are address with the command "`PARAM`" and these are classified here by the following:

- Sub-case
- Material
- Element
- Loads
- Output

PARAMETERS FOR DIFFERENT SUBCASE TYPES

Subcase Type	Output options	PARAM
LINEAR STATIC	DISPLACEMENT, ACCELERATION, STRESS, STRAIN, GPFORCE, GPSTRAIN, GPSTRESS, OLOAD, SPCF, PRESSURE, MPCFORCES	AUTOMSET, AUTOSPC, BUSHRLMT, BUSHSTIF, BUSHTLMT, COMP2SHL, CURVSHL2, ELASRLMT, ELASSTIF, ELASTTLMT, CHECKEL, CHECKMAT, CHKELSET, GRDPNT, GE_MOD
MODES	DISPLACEMENT, STRESS, STRAIN, GPFORCE, GPSTRAIN, GPSTRESS, OMODES	AMLS, AMLSMAXR, AMLSNCPU, AMLSUCON, AMSES, AUTOSPC, CHECKEL, CHECKMAT, CHKELSET, AUTOMSET, BUSHRLMT, BUSHSTIF, BUSHTLMT, COMP2SHL, CSTEVAL, ELASRLMT, ELASSTIF, ELASTTLMT, CHECKEL, CHECKMAT, CHKELSET, GRDPNT, GE_MOD
Inertia Relief	DISPLACEMENT, STRESS, STRAIN, GPFORCE, GPSTRAIN, GPSTRESS	AUTOSPRT, INREL, REFPNT, UCORD
BUCKLING	DISPLACEMENT, ACCELERATION, STRESS, STRAIN, GPFORCE, GPSTRAIN, GPSTRESS, OLOAD, SPCF, PRESSURE, MPCFORCES	AUTOMSET, AUTOSPC, BUSHRLMT, BUSHSTIF, BUSHTLMT, COMP2SHL, CURVSHL2, ELASRLMT, ELASSTIF, ELASTTLMT, CHECKEL, CHECKMAT, CHKELSET, GRDPNT, GE_MOD, CHECKEL, CHECKMAT, CHKELSET
THERMAL	THERMAL, OLOAD, SPCF	THCNTPEN

PARAMETERS FOR ELEMENTS, MATERIALS, OUTPUT

	PARAM
Element Specific	AMSE4CMS, CHECKEL, CHKELSET, CURVSHL2, EXCOUT, FLIPOK, KGRGD, NPRBAR, NPRBE2, RBE2FREE, RBE3FREE, RENUMOK, SHL2MEM, SHPBCKOR, TOLRSC, XPOST
Material Data Specific	PRESUBNL, OMID, MDK4OPT, CHECKMAT, ALPHA1, ALPHA1FL
Output Specific	AMLS, CMSALOAD, FLLWER, XPOST, EFFMAS, EXCEXB, EXTOUT, GPSLOC, ITAPE, OGEOM, POST, PRGPST, STRTHR, UCORD

TRANSITIONING ELEMENTS

INTRODUCTION TRANSITIONING ELEMENTS

When performing analyses of complex components or systems, the issue of connecting dissimilar mesh types often arises. Accuracy and efficiency are two conflicting aspirations and so analysts are often forced to use different types of elements in a single model. Due to limitations of recourses, analysts generally need to implement aggressive idealizations on their analysis models. When used correctly, transitioning can provides major cost savings while retaining quality of results. Transitioning schemes fall into two main categories.

1. Transitions from one element type to another that are of the same dimension (e.g. both are shell or solid elements).
2. Transition is where element of different dimension are joined (e.g. a shell to solid transition).

Dimensional reduction or model order reduction techniques are oftentimes used to transform a complex 3D or 2D problem into a lower order 1D or 2D system respectively. By doing so, computation times are significantly reduced, but in a way that does not compromise model accuracy. In dimensional reduction, the finite element model makes use of elements of reduced dimension, such as beams, plates and shells.

TRANSITIONING ELEMENTS TYPES

Transitioning Methods	Transitioning Elements	Additional Details
Interface	Tie Contact	Penalty-based and MPC-based (Lagrange Multipliers)
	Freeze Contact	Penalty-based
Gap or Friction Element	`CGAP`	
Shell-to-Solid Element Connector	`RSSCON`	
Kinematic coupling	Rigid Links	`RBE1`, `RBE2`, `RBAR`, `RROD`
	Load Distribution	`RBE3`

INTERFACE TYPES

Freeze & Tie Contact enforces zero relative motion on the contact surface, the contact gap opening remains fixed at the original value and the sliding distance is forced to be zero. Additionally, rotations at the slave node are matched to the rotations of the master patch.

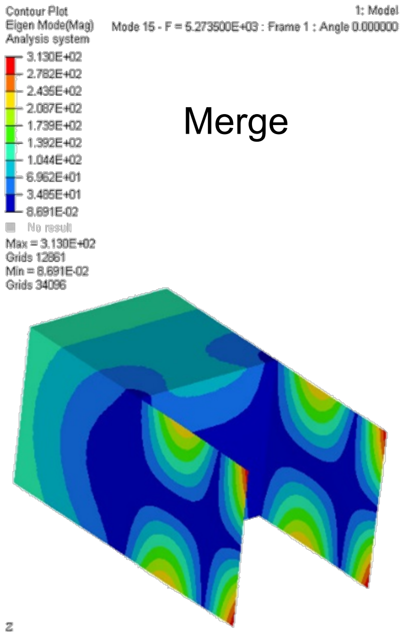
The `FREEZE` condition applies to all respective contact elements, regardless of whether they are open or closed.

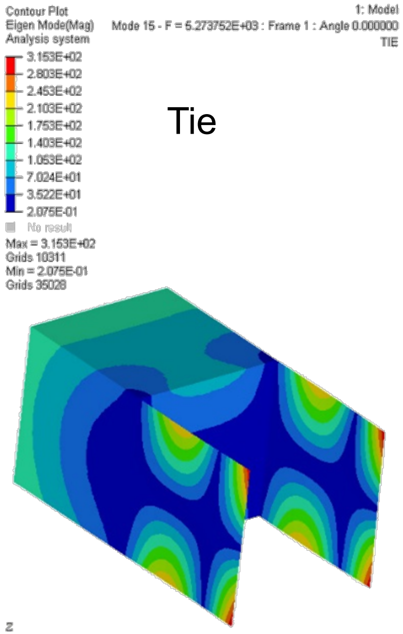
Two types of `TIE` contact are available, `PENALTY`-based and `MPC`-based. The two types can be switched using `CONTPRM`, `TIE`, `PENALTY`/`MPC`. The `MPC`-based `TIE` uses Multi-point Constraints to define a tied contact between the master and slave surfaces.

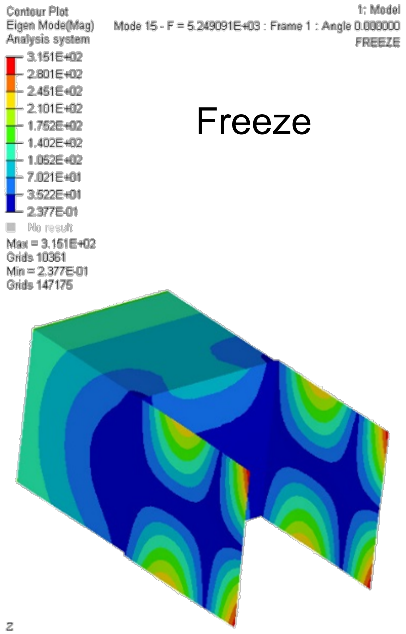
COMPARISON STUDY BETWEEN TIE AND FREEZE CONTACT

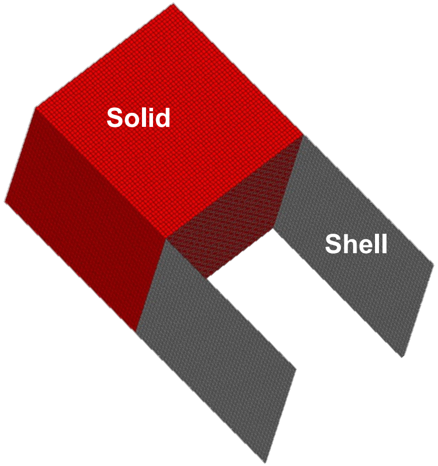
Free-Free Normal Mode analysis was conducted using a Solid cube & Shell plate FE-model

For evaluation these modes are connected using





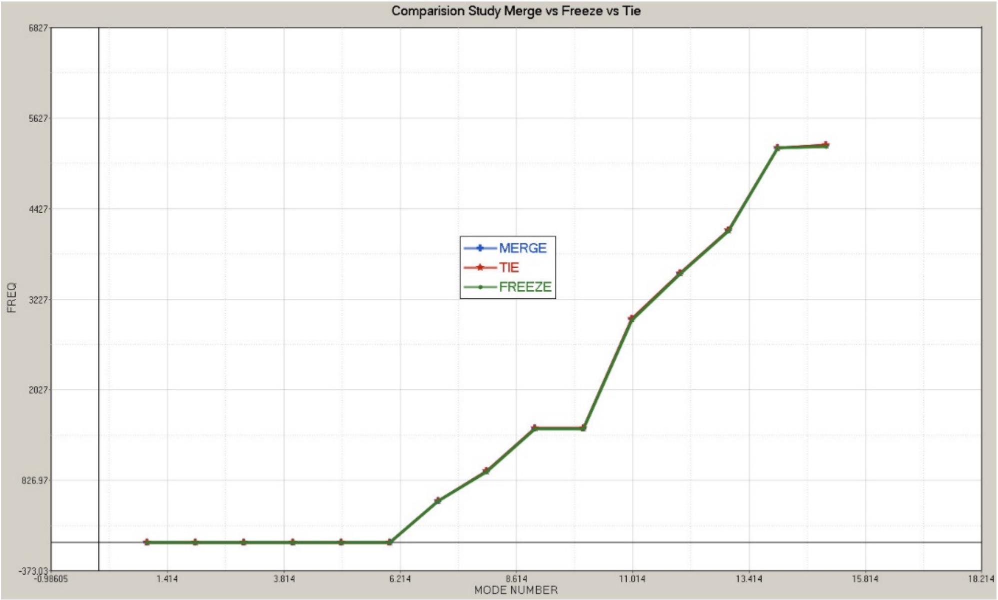




COMPARISON STUDY BETWEEN TIE AND FREEZE CONTACT

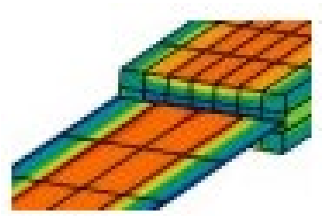
Observe that all the 15 modal frequency match very closely with Merge, compared to Tie & Freeze.

Mode	MERGE	TIE	FREEZE
1	9.96E-03	9.79E-03	8.86E-03
2	1.44E-02	1.47E-02	1.34E-02
3	1.53E-02	1.58E-02	1.48E-02
4	1.66E-02	1.69E-02	1.54E-02
5	1.71E-02	1.77E-02	1.63E-02
6	1.89E-02	1.95E-02	1.92E-02
7	5.56E+02	5.56E+02	5.54E+02
8	9.45E+02	9.45E+02	9.43E+02
9	1.52E+03	1.52E+03	1.51E+03
10	1.52E+03	1.52E+03	1.51E+03
11	2.97E+03	2.97E+03	2.95E+03
12	3.57E+03	3.57E+03	3.56E+03
13	4.14E+03	4.14E+03	4.13E+03
14	5.23E+03	5.23E+03	5.23E+03
15	5.27E+03	5.27E+03	5.25E+03



RSSCON – SHELL-TO-SOLID ELEMENT CONNECTOR

`RSSCON` generates a multipoint constraint that models a clamped connection between a shell and a solid element. The shell degrees-of- freedom are considered dependent. The translational degrees-of- freedom of the shell edge are connected to the translational degrees-of- freedom of the upper and lower solid edge.



Mixed dimensioned transition elements

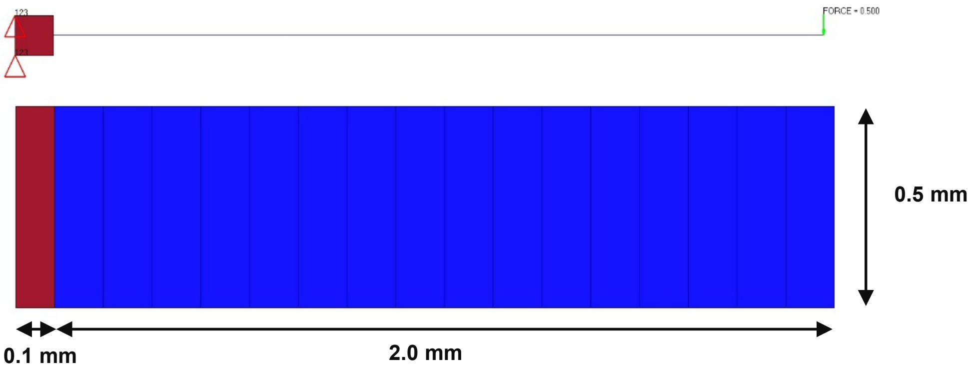
The rotational degrees-of-freedom of the shell are connected to the translational degrees-of-freedom of the lower and upper edges of the solid element face. Poisson’s ratio effects are considered in the translational degrees-of-freedom.

It is recommended that the height of the solid element’s face is approximately equal to the shell element’s thickness of the shell. The shell edge should then be placed in the middle of the solid face.

`RSSCON` is ignored in heat-transfer problems.

COMPARISON STUDY OF RSSCON WITH CANTILEVER MODEL

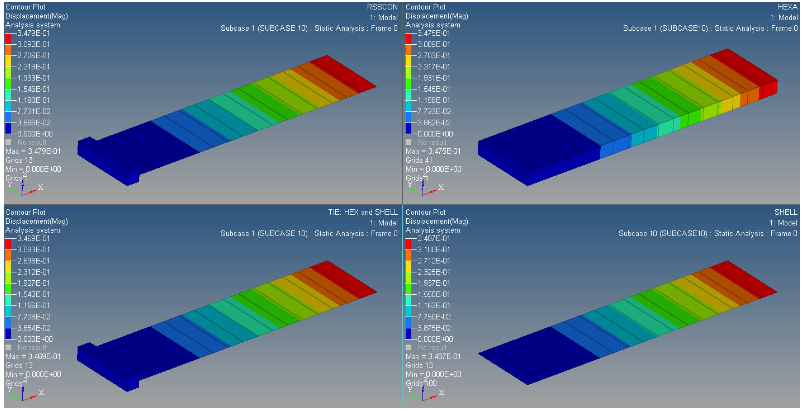
FE Model Details

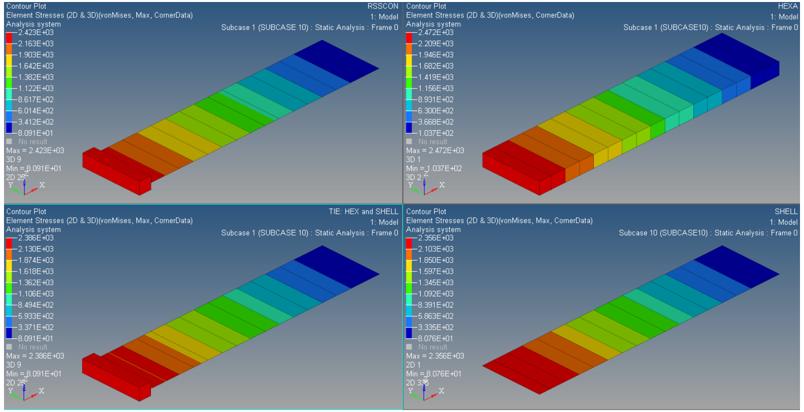


Types of FE Model used for Study	Material	Property	Force
CQUAD4 Elements	E = 2.1E5 Nu = 0.3	Thickness = 0.1 mm	F = 0.5 Newton
CHEXA Elements			
TIE (N2S) contacts with CHEXA & CQUAD			
RSSCON Model with CHEXA & CQUAD			

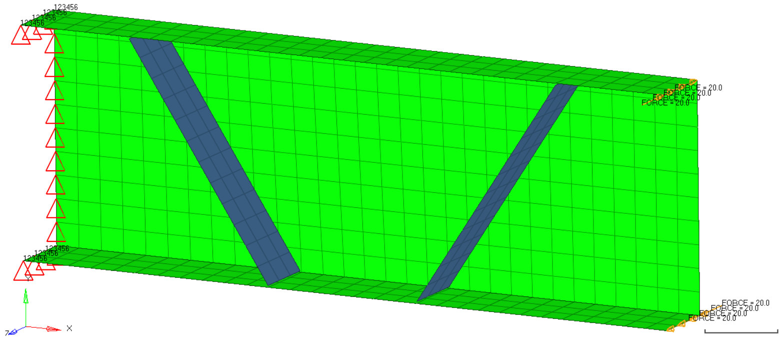
COMPARISON STUDY OF RSSCON WITH CANTILEVER MODEL

Results Summary	Displacements at tip (in mm)	Maximum Stresses at the Clamped End (in MPA)
CQUAD4	0.3487	2356
CHEXA	0.3475	2472
TIE-N2S (CHEXA-CQUAD)	0.3469	2386
RSSCON	0.3479	2423





EXERCISE 7A: STATIC ANALYSIS USING FREEZE CONTACT



File Name and Location

…\STUDENT-EXERCISE\7a_Beam_Rib\rib.hm

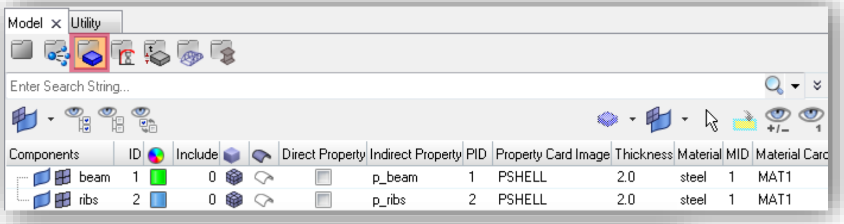
Objectives

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

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

1. Open the model in HyperMesh Desktop
2. Review the model (control cards, components, properties, materials, sets, load collectors & step)
3. Run the static analysis in OptiStruct, review the results in HyperView and check maximum static displacement
4. Create a group collector `freeze_contact` (card image `CONTACT`) with TYPE = `FREEZE`, SSID = slaves (grid) set (ID `1`) and MSID master (element) set (ID `2`)
5. Rerun the analysis with OptiStruct, review the results in HyperView and check if maximum static displacement is below 0.38

EXERCISE 7A: STATIC ANALYSIS USING FREEZE CONTACT



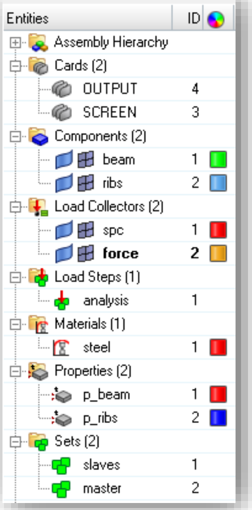
Hints (1/3)

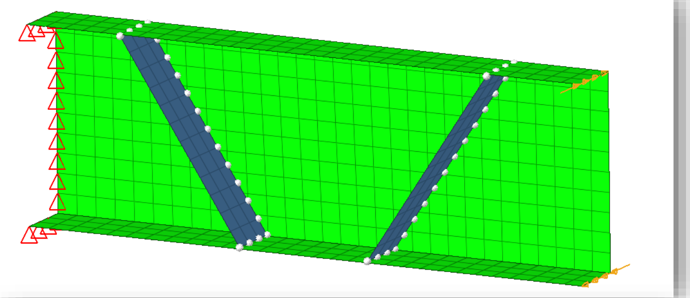
2. To review the model wrt components, properties and materials the component view in the model browser is well suited.

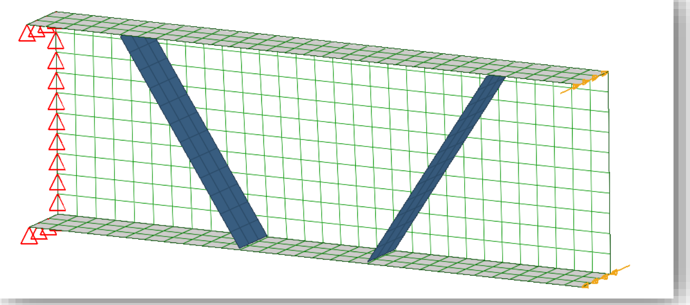
   Common control cards requests are set: `SCREEN` and `OUTPUT`

   There is one static load step defined: analysis

   There are two sets defined: slaves as grid set (ID 1) master as element set (ID 2)



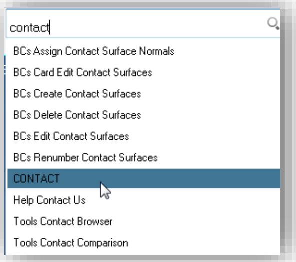


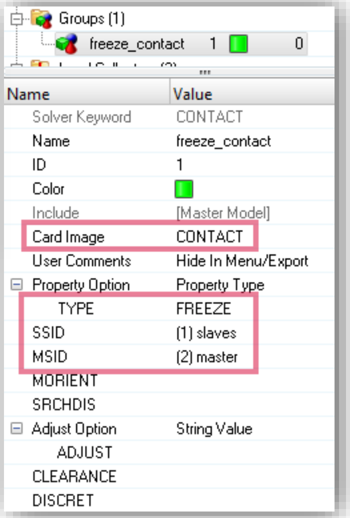


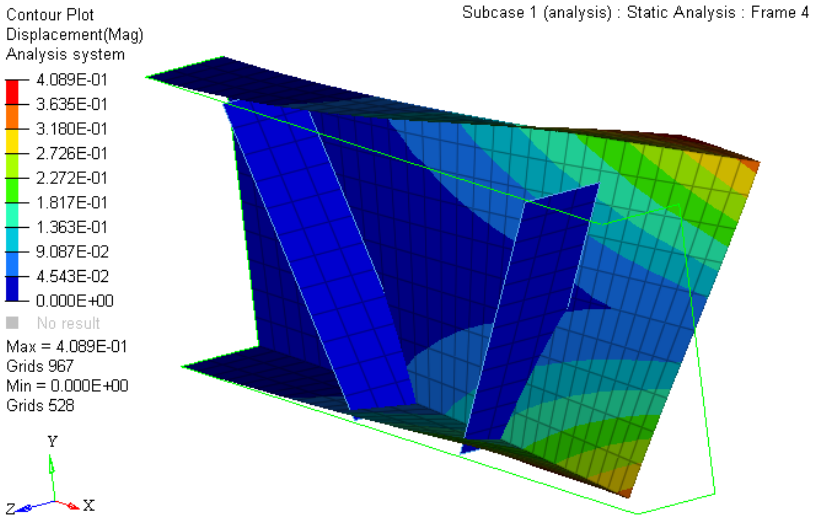
EXERCISE 7A: STATIC ANALYSIS USING FREEZE CONTACT

Hints (2/3)

3. Max. displacement = 0.4089
   (deformed shape inflated by 100)
   The two ribs are not connect to the beam.
4. Use HyperMesh's Quick Access Tool (`Crtl+f`) to create a `CONTACT` card.





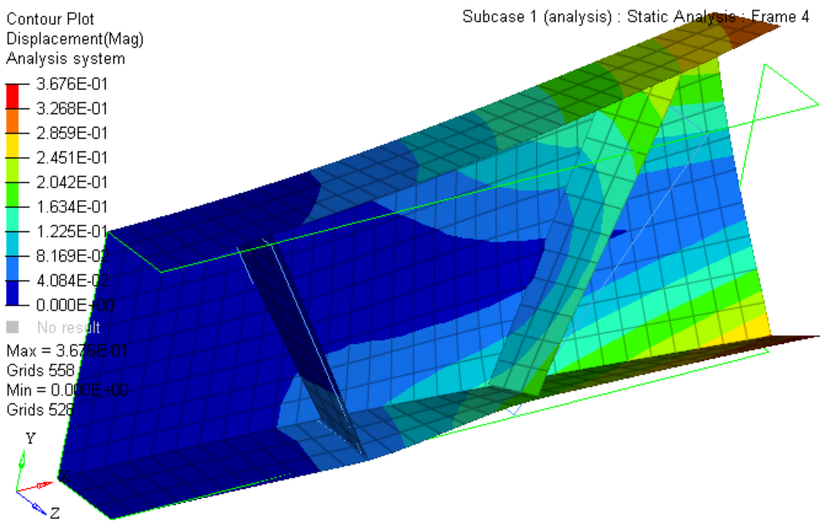


EXERCISE 7A: STATIC ANALYSIS USING FREEZE CONTACT

Hints (3/3)

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

   The two ribs are connect to the beam.



INTRODUCTION TO PARALLELIZATION

INTRODUCTION PARALLELIZATION

High Performance Computing

- Leverages computing resources – standalone or cluster
- Message Passing Interfaces
- Advanced Memory handling capabilities
- Large matrix factorizations, inversions, and manipulations across multiple degree-of-freedom systems

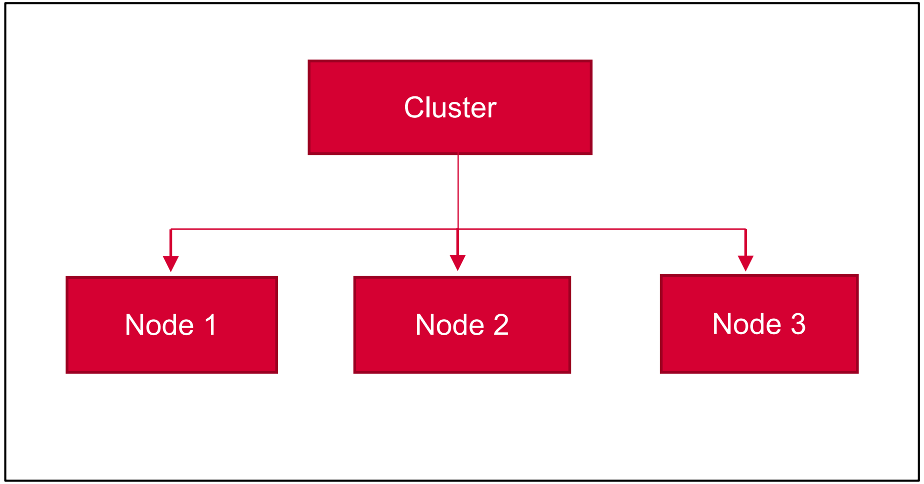
Computer Architecture

- Node – Computing machine with single or multiple sockets.
- Socket – Each socket contains one processor.
- Processor – Typically contains multiple cores/CPU's where computations are performed.
- Core/CPU – Computations are performed here.
- Thread – a single core, may be able to handle multiple parallel computations (via threads).

INTRODUCTION PARALLELIZATION

Cluster

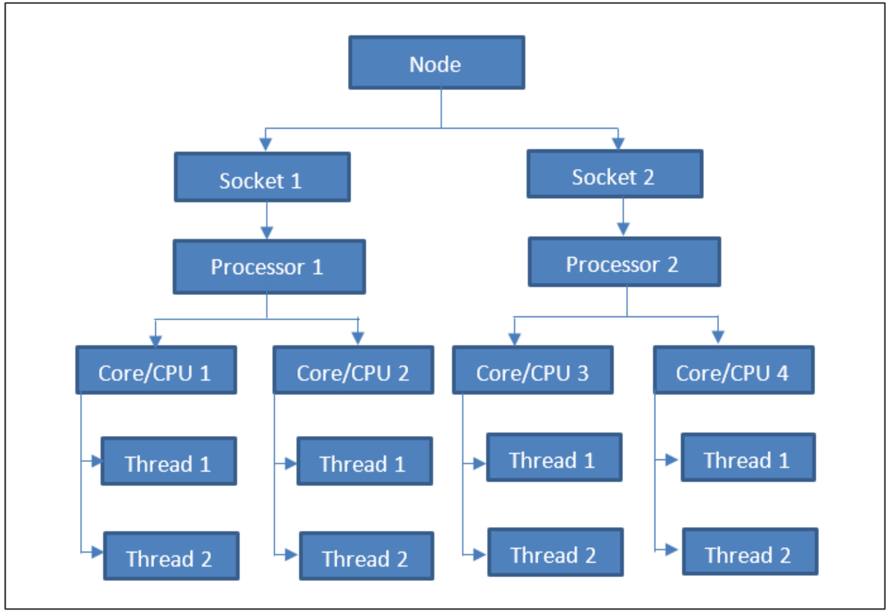
- A computational cluster is a collection of nodes that are connected together to perform as a single unit.
- The tasks assigned to a cluster can be internally distributed and reconfigured by various software to nodes within the cluster.



INTRODUCTION PARALLELIZATION

Node

- A node is a computing machine/workstation/laptop within a cluster.
- It consists of different electrical and electronic components, such as Central Processing Units/Cores, memory, and ports that communicate with each other through complex systems and electronic pathways.
- Typically, a node consists of one or more sockets, which further contain one Physical Processor each.



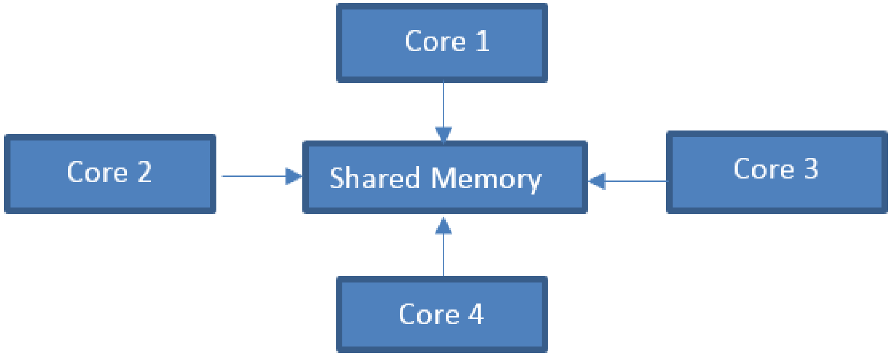
SERIAL VERSUS PARALLEL COMPUTING

Serial Computing	Parallel Computing
Solution is divided into discrete instructions.	Solution is divided into sections, which are in-turn divided into discrete instructions.
Sequential Execution of discrete instructions one logical processor.	Parallel Execution of discrete instructions of all sections simultaneously on multiple logical processors.
At each point in the time-domain, only a single discrete instruction is executed.	At each point in the time-domain, multiple discrete instructions relating to multiple parts are executed simultaneously.
Runtimes are typically high compared to Parallel Computing.	Runtimes are typically lower than Serial Computing.

OPTISTRUCT PARALLELIZATION – SMP

Shared Memory Parallelization (SMP)

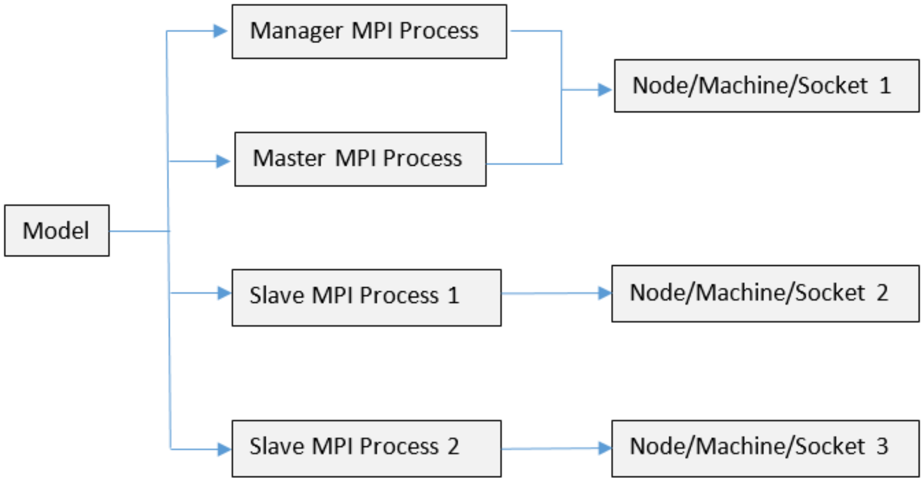
- Shared Memory Parallelization in OptiStruct does not require different executables or the installation of separate components for message passing.



OPTISTRUCT PARALLELIZATION – DMP

Distributed Memory Parallelization (DMP)

- Parallelization technique in computing that is employed to achieve faster results by splitting the program into multiple subsets and running them simultaneously on multiple processors/machines.
- Memory is distributed across multiple processors or nodes.



OPTISTRUCT PARALLELIZATION – DMP TYPES

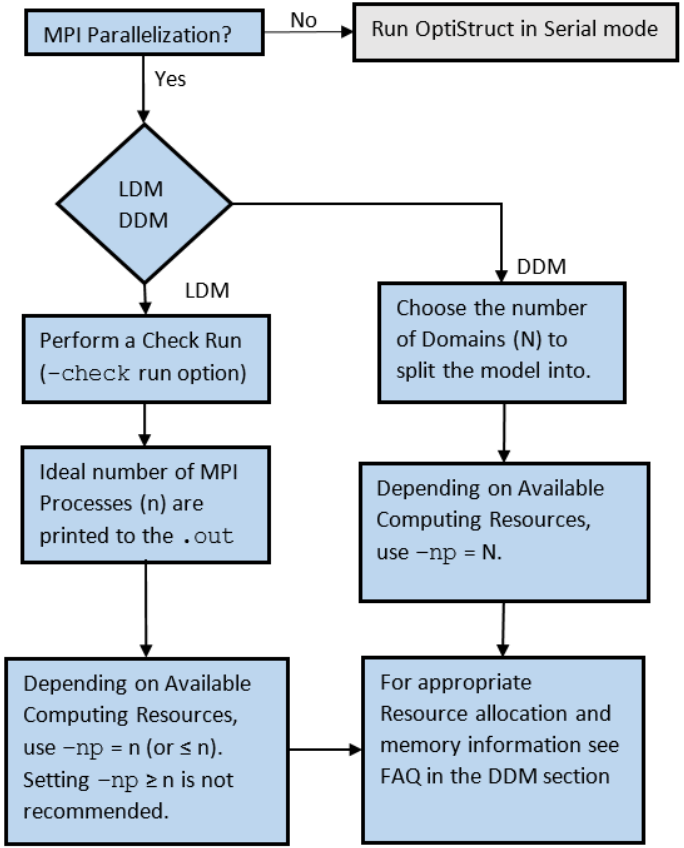
Distributed Memory Parallelization (DMP) Types

- Load Decomposition Method (LDM)
  - Large granularity
  - Efficient when there are a large number of loads/Boundary Conditions.
  - Ratio of loads/Boundary Conditions number to the number of nodes should be high.
- Not efficient for extremely large models
- Domain Decomposition Method (DDM)
  - Lower granularity
  - Efficient for extremely large models in a cluster.
- Multi-Model Optimization (MMO)
- Failsafe Topology Optimization (FSO)

OPTISTRUCT PARALLELIZATION – SPMD

Hybrid Parallelization – SPMD - Single Program Multiple Data (SMP + DMP)

- Combination of SMP and DMP
- Individual MPI process can be run in SMP configuration.
- Can generate enhanced runtime reduction when compared to either just SMP or DMP runs, individually.



OPTISTRUCT PARALLELIZATION – BEST PRACTICES DDM

OptiStuct run best in hybrid mode against pure MPI or pure SMP modes

- Always apply full nodes
- # of domain per node: 4
- # of threads per domain: (# of cores) / (# of domain per node)
  - E.g. on Ivy bridge E5-2697, total number of cores is 24, so # of threads per domain is 24/4=6

Proper number of nodes

- DDM needs to run in cluster to reduce memory requirement per host.
- Memory reduction for each domain is not linear
- The least number might be that fits for in-core mode.
- Do a check run first to get the memory estimation

OPTISTRUCT PARALLELIZATION – BEST PRACTICES DDM

Don't share nodes with other jobs.

- May cause serious load unbalancing problem.
- Each DDM job occupies the whole node.

Place all threads in one socket, e.g. on NUMA node, w/ dual socket, 12 cores, 24 cores total

- 6 MPI, 4 threads per MPI → okay when memory sufficient
- 4 MPI, 6 threads per MPI → recommended
- 3 MPI, 8 threads per MPI → load unbalancing
- 2 MPI, 12 threads per MPI → use when necessary

Local fast disk is crucial to OptiStruct performance. It should be at least used for scratch directory, through `-scr` or `-tmpdir` option.

For large model with big amount of output request, we suggest to put `.fem` file also on local disk and run OptiStruct there.

QUESTIONS & ANSWERS

1. What do SMP and DMP stand for?

RUN OPTIONS

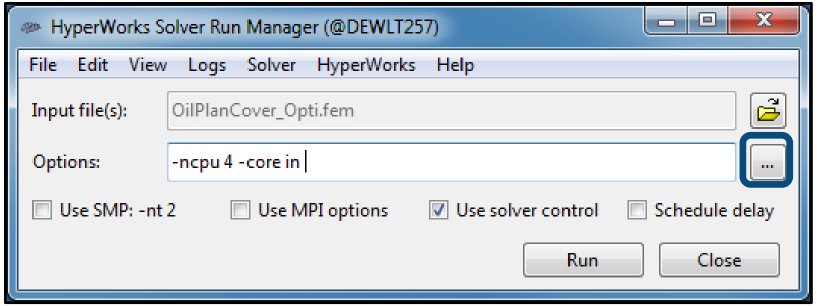
RUN OPTIONS FOR OPTISTRUCT

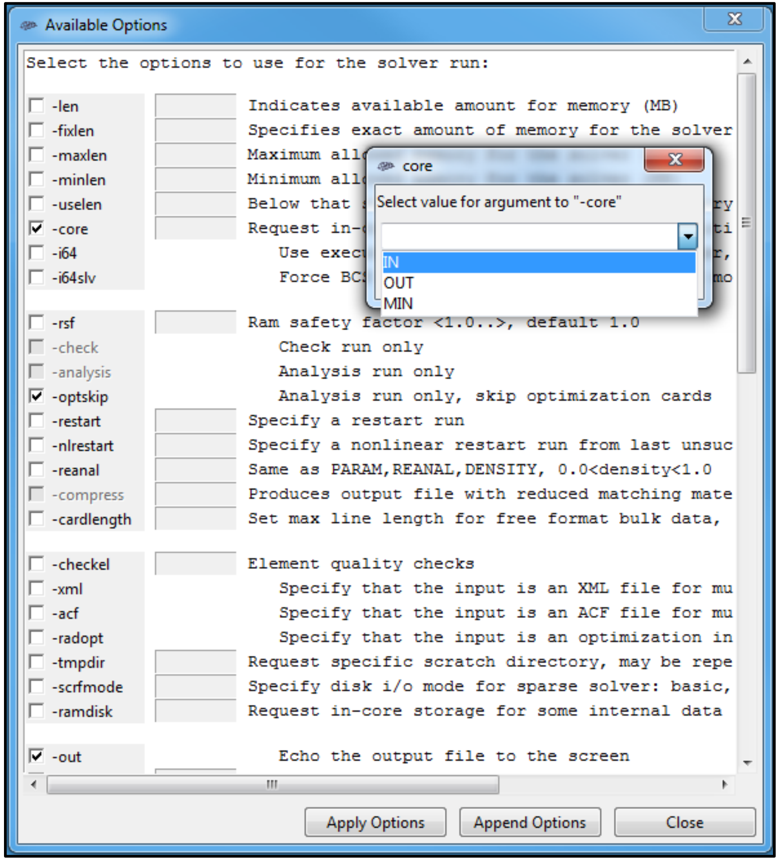
Selected Run Options for OptiStruct (see OptiStruct User's Guide for more)

`-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`	Echos the output file to the screen.

RUN OPTIONS FOR OPTISTRUCT – HW SOLVER RUN MANAGER

For a complete list of available options including plausibility checks use Options from the HyperWorks Solver Run Manager



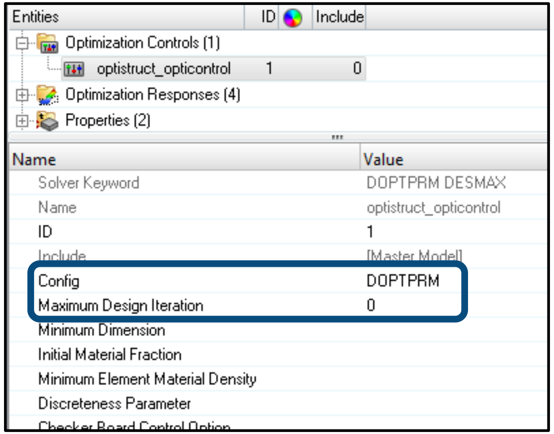


RUNNING ONE ITERATION ONLY

There are many cases where the user wants to run only one iteration

- to check the analysis results for the initial design
- to output sensitivity information for the initial design
- to output DREPORT information for the initial design
- etc.

Use `DOPTPRM, DESMAX, 0`



This will run a regular optimization but stops after the first iteration, i.e. the structural response depends on value of the design variables

If an analysis run is desired instead, use run options `-analysis` or `-optskip`

This will run an analysis with all optimization cards skipped, i.e. the structural response depends on FE data and not the design variables.

OUTPUT MANAGEMENT

DEFAULT OUTPUT FILES

Protocol Files

- `.out` file → provides a commentary on the solution process
- `.stat` file → provides details on CPU and elapsed time for each solver module

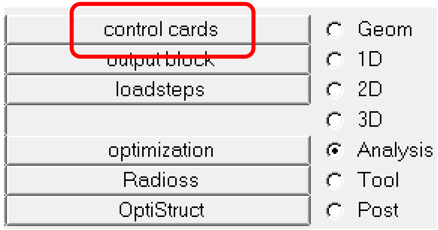
Result Files

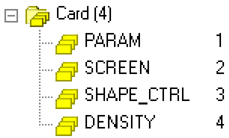
- `.h3d` file → compressed binary file, containing both model and result data
- `.res` file → is a HyperMesh binary results file
- `.mvw` file → HyperView/HyperGraph session file to open results in HyperWorks Desktop
- `.pch & .op2` file → Nastran Punch format and Output2 format

html Files

- `.html` file → contains a problem summary and results summary of the run.
- `frames.html` → opens the `.h3d` files using the HyperView Player browser plug-in
- `menu.html` → facilitates the selection of the appropriate `.h3d` file, for the HyperView Player browser plug-in

COMMON CONTROL CARDS

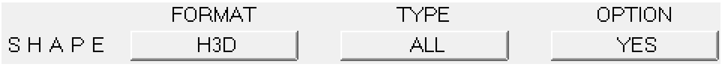





```
PARAM,CHECKEL,NO

DENSITY(H3D,ALL) = YES
SHAPE(H3D,ALL) = YES
SCREEN OUT
```

.out file is echoed to the screen



Shape result output in all simulations

Density result output in all simulations

For topography and shape optimization: element quality checks are not performed, but mathematical validity checks are

RECOMMENDATIONS

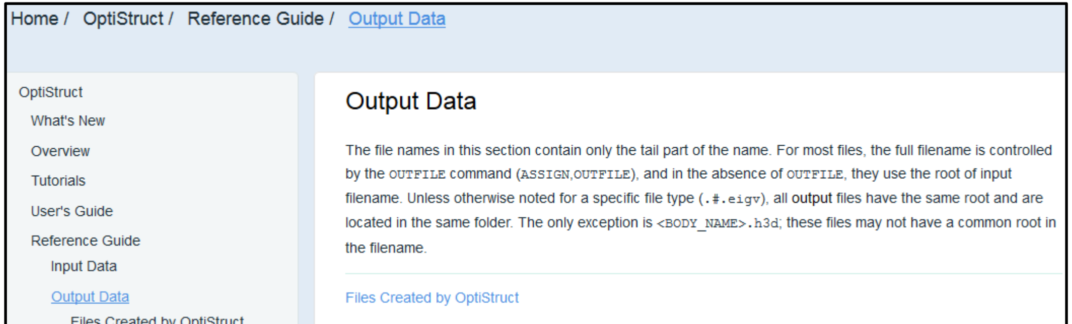
`OUTPUT` controls the format of default results output

Use result keywords like `DISPLACEMENT/STRESS` for detailed control

- Global or subcase dependency
- Additional output-Formats like `.pch` file
- Output only for a subset of nodes/elements/properties
- More detailed output specific control like stress type or stress location

Usage of other output control commands like `FORMAT` or `PARAM, POST` is possible, but not necessary/recommended as `OUTPUT` is more flexible.

Output data overview in OptiStruct help



EXAMPLE: ANALYSIS

Outputs

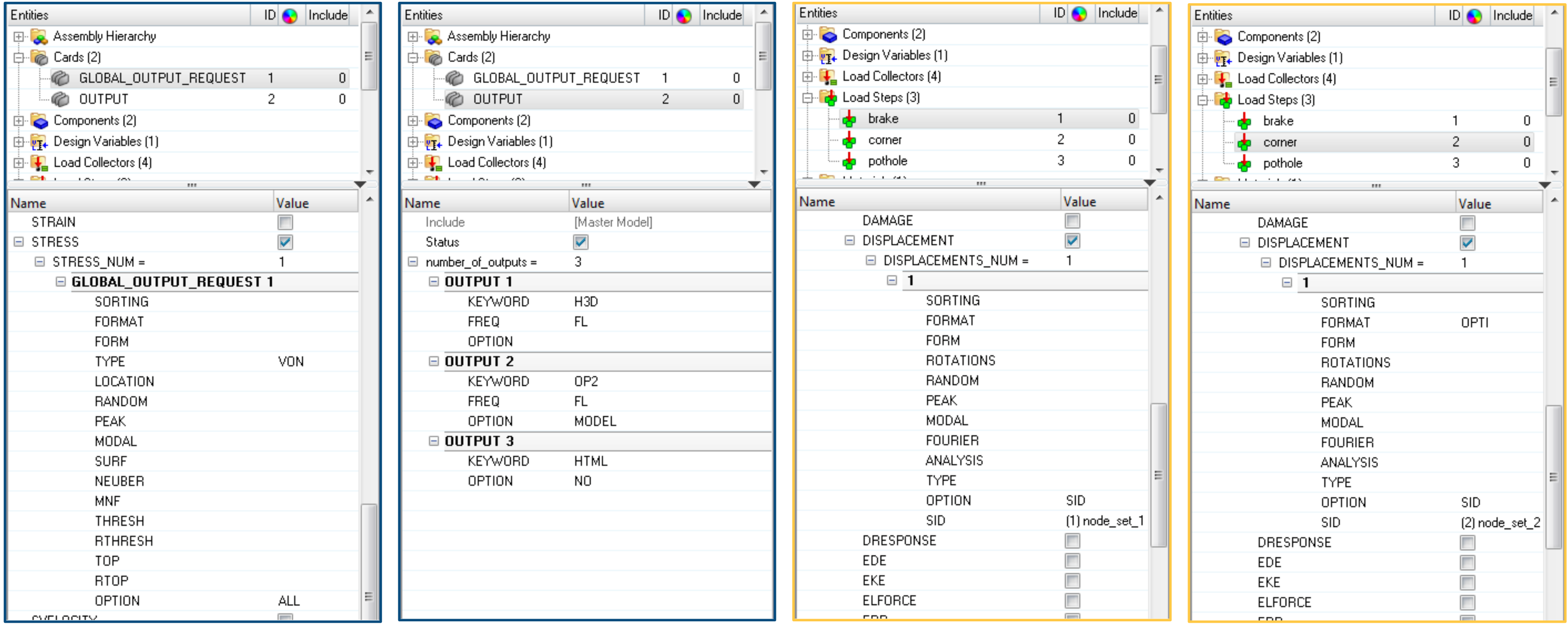
- Disable `.res` file (use `.h3d` instead) and `.html` files
- Enable `.op2` file including model data
- Reduce stress output to von Mises
- Displacements of node sets to ascii files
  - `.pch` file for load case brake (node set 1)
  - `.disp` (opti) file for load case pothole (node set 2)

In HyperMesh via

- Entity Editor (with HyperMesh's Quick Access Tool (`Crtl+f`))
- Load Step Browser
- Control Cards

```
$$ optistruct
$
OUTPUT,H3D,FL
OUTPUT,OP2,FL,MODEL
OUTPUT,HTML,,NO
STRESS(VON) = ALL
$$------------------------------
$$                         Case Co
$$------------------------------
$
SUBCASE       1
  LABEL brake
  SPC =         1
  LOAD =         2
  DISPLACEMENT(PCH) = 1
$
SUBCASE       2
  LABEL corner
  SPC =         1
  LOAD =         3
  DISPLACEMENT(OPTI) = 2
$
SUBCASE       3
  LABEL pothole
  SPC =         1
  LOAD =         4
```

EXAMPLE: ANALYSIS




```
OUTPUT,H3D,FL
OUTPUT,OP2,FL,MODEL
OUTPUT,HTML,,NO
STRESS(VON) = ALL
```

```
SUBCASE        1
  LABEL brake
  SPC =          1
  LOAD =          2
  DISPLACEMENT(PCH) = 1
```

```
SUBCASE        2
  LABEL corner
  SPC =          1
  LOAD =          3
  DISPLACEMENT(OPTI) = 2
```

OUTPUT

KPI (Key Performance Indicator)

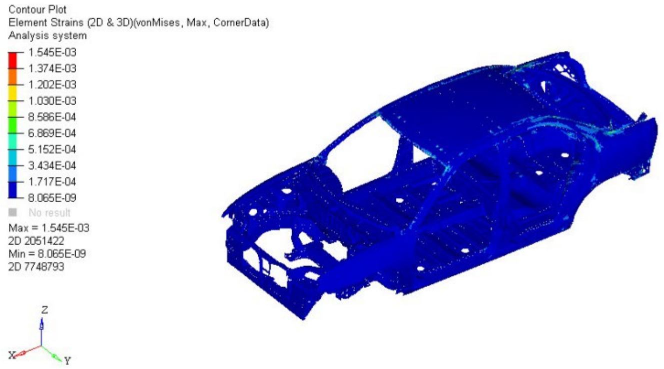
- OUTPUT,KPI or DISP(KPI) → .kpi file
- Currently supported for linear and nonlinear static analysis
- Max value for displacement/stress/strain/plastic strain based on groups by property
- Stresses and strains are supported for shells and solids

```
$ITERATION =      0
$DISPLACEMENTS
$SUBCASE ID =            2
       PID          GID          MAG          GID            X          GID            Y          GID            Z
         2         1114  5.31911E+00         1115  5.18782E+00         1114  1.80974E+00         1114 -2.92242E-02
$ITERATION =      0
$ELEMENT STRESSES
$SUBCASE ID =            2
       PID          EID          VON      XX/XX_1      YY/XX_2      ZZ/YY_1      XY/YY_2      YZ/XY_1      XZ/XY_2
         2         8621  1.11191E+02  7.32064E+00  1.03493E+02  2.21324E-01 -2.81675E+01  5.57553E-01 -2.31563E-01
$ITERATION =     41
$DISPLACEMENTS
$SUBCASE ID =            2
       PID          GID          MAG          GID            X          GID            Y          GID            Z
         2         1114  8.12851E+01         1115  8.00305E+01         1114  2.70630E+01         9705  2.43199E-01
$ITERATION =     41
$ELEMENT STRESSES
$SUBCASE ID =            2
       PID          EID          VON      XX/XX_1      YY/XX_2      ZZ/YY_1      XY/YY_2      YZ/XY_1      XZ/XY_2
         2         8621  4.46134E+02  8.43841E+00  3.99810E+02 -1.60386E+01 -1.08307E+02  4.88293E+00 -1.18679E+01
```

OUTPUT

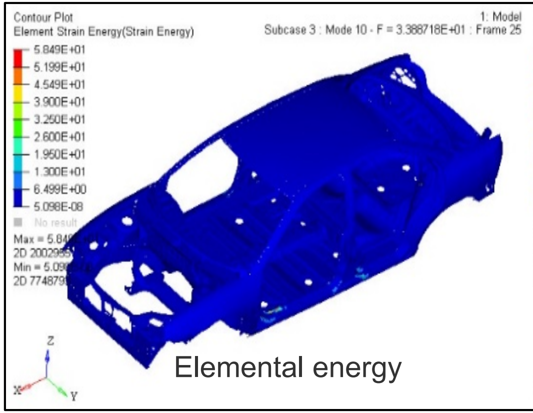
Strain output from Normal Mode Analysis

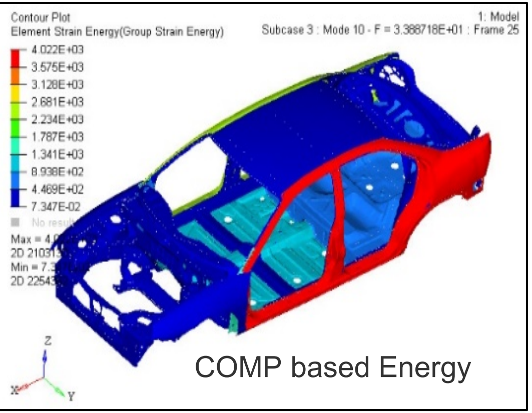
- H3D
- Centroid and Corner



Energy output per component (PROP, COMP, SET, OPROP, OCOMP, OSET)

- Component based energy output means the sum of elemental energy for each component





OUTPUT

Output with all different eigenvector normalization types (MAX, MAXT and MASS) is available with “ALL” in NORM field on EIGRA/EIGRL

- .out file has Generalized stiffness/mass output for MAX,MAT and MASS
- H3d file has eigenvector output for MAX,MAT and MASS

MASS

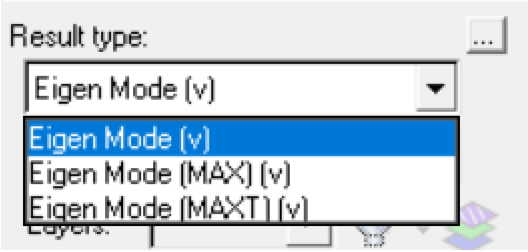
Subcase	Mode	Frequency	Eigenvalue	Generalized Stiffness	Generalized Mass
2	1	2.354957E+00	2.189404E+02	2.189404E+02	1.000000E+00
2	2	1.474904E+01	8.587906E+03	8.587906E+03	1.000000E+00
2	3	2.325721E+01	2.135379E+04	2.135379E+04	1.000000E+00
2	4	4.130079E+01	6.734052E+04	6.734052E+04	1.000000E+00
2	5	4.439779E+01	7.781842E+04	7.781842E+04	1.000000E+00
2	6	8.097436E+01	2.588539E+05	2.588539E+05	1.000000E+00

MAXT

Subcase	Mode	Frequency	Eigenvalue	Generalized Stiffness	Generalized Mass
2	1	2.354957E+00	2.189404E+02	5.458703E-01	2.493237E-03
2	2	1.474904E+01	8.587906E+03	2.142867E+01	2.495214E-03
2	3	2.325721E+01	2.135379E+04	5.385254E+01	2.521919E-03
2	4	4.130079E+01	6.734052E+04	1.679573E+02	2.494149E-03
2	5	4.439779E+01	7.781842E+04	1.288238E+02	1.655442E-03
2	6	8.097436E+01	2.588539E+05	6.442915E+02	2.489016E-03

MAX

Subcase	Mode	Frequency	Eigenvalue	Generalized Stiffness	Generalized Mass
2	1	2.354957E+00	2.189404E+02	5.458703E-01	2.493237E-03
2	2	1.474904E+01	8.587906E+03	2.142867E+01	2.495214E-03
2	3	2.325721E+01	2.135379E+04	5.385254E+01	2.521919E-03
2	4	4.130079E+01	6.734052E+04	1.679573E+02	2.494149E-03
2	5	4.439779E+01	7.781842E+04	1.288238E+02	1.655442E-03
2	6	8.097436E+01	2.588539E+05	6.442915E+02	2.489016E-03



OUTPUT AND DIAGNOSTICS

TIE and FREEZE diagnostics

- <filename>_nontied.fem for grids outside search distance

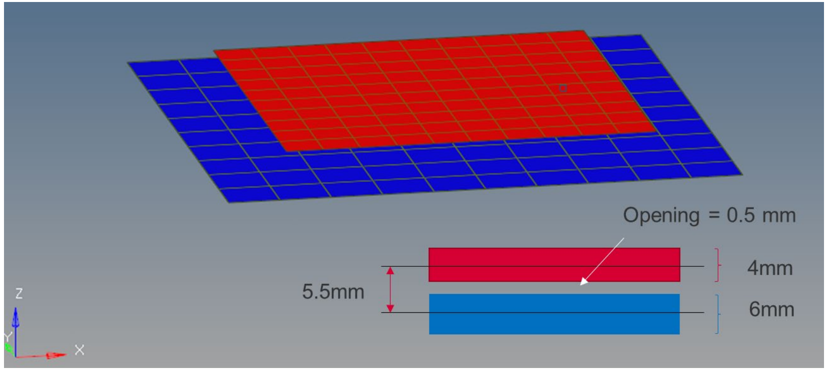
Initial Contact condition output (.cpr)

- CONTPRM, PREPRT,YES to request this output
- Breakdown for each CONTACT
- Check run only required

```
************************************************************************
Type  Name       Content
************************************************************************
(I)   CONTID     Contact interface ID
(I)   SLVGID     Core slave grid ID of the gap
(I)   NCMSTG     Number of connected master grids of the gap
(I)   NCSLVG     Number of connected slave grids beside the core slave
                 grid of the gap (only for S2S contact)
(C)   STATUS     Gap status (OPEN/CLOSED/FROZEN)
(R)   OPENING    Gap opening (negative value means penetration)
(R)   RAWGAP     Raw gap opening based on mesh coordinates
                 (adjusted mesh coordinates with N2S ADJUST)
(R)   PADDING    Gap padding due to GPAD, S2S ADJUST, CLEARANCE, etc.
(R)   SAREA      Nodal contact area on the core slave grid
                 (only for S2S contact)
************************************************************************
(I): Integer number
(R): Real number
(C): Characters
************************************************************************


Contact Interface (S2S):              1 "contact"
         Slave SURF/SET:              2 "slave"
        Master SURF/SET:              1 "master"
==========================================================================================================
  CONTID |  SLVGID |  NCMSTG |  NCSLVG |  STATUS |     OPENING     |     RAWGAP     |    PADDING     |     SAREA     |
==========================================================================================================
        1       302         6         5    OPEN     0.44950E+01     0.50000E+01     0.50500E+00     0.50000E+00
        1       303         6         5    OPEN     0.44950E+01     0.50000E+01     0.50500E+00     0.50000E+00
        1       304         6         5    OPEN     0.44950E+01     0.50000E+01     0.50500E+00     0.50000E+00
```

OUTPUT AND DIAGNOSTICS



Contact with CLEARANCE = 0.0

```
Contact Interface (S2S):        2 "shell_contact"
         Slave SURF/SET:        3 "slave"
        Master SURF/SET:        4 "master"
```

CONTID	SLVGID	NCMSTG	NCSLVG	STATUS	OPENING	RAWGAP	PADDING	SAREA
2	2081	16	8	OPEN	0.50000E+00	0.55000E+01	0.50000E+01	0.51840E+02
2	2082	12	8	OPEN	0.50000E+00	0.55000E+01	0.50000E+01	0.51840E+02
2	2083	9	8	OPEN	0.50000E+00	0.55000E+01	0.50000E+01	0.51840E+02
2	2084	12	8	OPEN	0.50000E+00	0.55000E+01	0.50000E+01	0.51840E+02
2	2112	9	8	OPEN	0.50000E+00	0.55000E+01	0.50000E+01	0.51840E+02
2	2113	12	8	OPEN	0.50000E+00	0.55000E+01	0.50000E+01	0.51840E+02
2	2133	12	8	OPEN	0.50000E+00	0.55000E+01	0.50000E+01	0.51840E+02
2	2134	9	8	OPEN	0.50000E+00	0.55000E+01	0.50000E+01	0.51840E+02
2	2151	9	8	OPEN	0.50000E+00	0.55000E+01	0.50000E+01	0.51840E+02
2	2166	12	8	OPEN	0.50000E+00	0.55000E+01	0.50000E+01	0.51840E+02
2	2167	9	8	OPEN	0.50000E+00	0.55000E+01	0.50000E+01	0.51840E+02
2	2184	9	8	OPEN	0.50000E+00	0.55000E+01	0.50000E+01	0.51840E+02
2	2204	12	5	OPEN	0.49999E+00	0.55000E+01	0.50000E+01	0.25920E+02
2	2205	9	5	OPEN	0.50000E+00	0.55000E+01	0.50000E+01	0.25920E+02
2	2206	9	8	OPEN	0.50000E+00	0.55000E+01	0.50000E+01	0.51840E+02

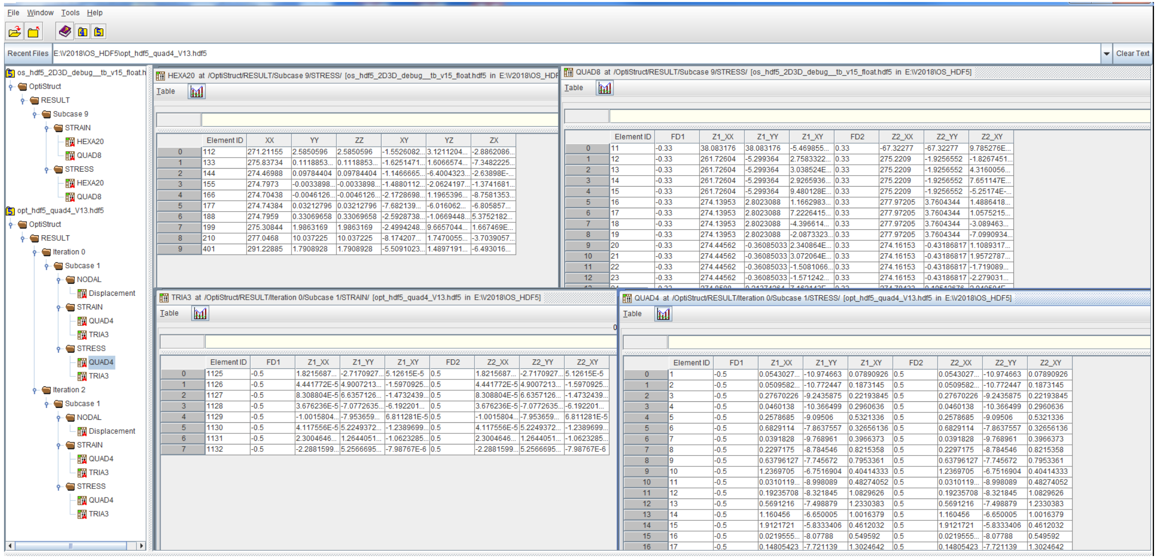
```
Contact Interface (S2S):        2 "shell_contact"
         Slave SURF/SET:        3 "slave"
        Master SURF/SET:        4 "master"
```

CONTID	SLVGID	NCMSTG	NCSLVG	STATUS	RAWGAP	PADDING	SAREA
2	2081	16	8	CLOSED	0.55000E+01	0.55000E+01	0.51840E+02
2	2082	12	8	CLOSED	0.55000E+01	0.55000E+01	0.51840E+02
2	2083	9	8	CLOSED	0.55000E+01	0.55000E+01	0.51840E+02
2	2084	12	8	CLOSED	0.55000E+01	0.55000E+01	0.51840E+02
2	2112	9	8	CLOSED	0.55000E+01	0.55000E+01	0.51840E+02
2	2113	12	8	CLOSED	0.55000E+01	0.55000E+01	0.51840E+02
2	2133	12	8	CLOSED	0.55000E+01	0.55000E+01	0.51840E+02
2	2134	9	8	CLOSED	0.55000E+01	0.55000E+01	0.51840E+02
2	2151	9	8	CLOSED	0.55000E+01	0.55000E+01	0.51840E+02
2	2166	12	8	CLOSED	0.55000E+01	0.55000E+01	0.51840E+02
2	2167	9	8	CLOSED	0.55000E+01	0.55000E+01	0.51840E+02
2	2184	9	8	CLOSED	0.55000E+01	0.55000E+01	0.51840E+02
2	2204	12	5	CLOSED	0.55000E+01	0.55000E+01	0.25920E+02
2	2205	9	5	CLOSED	0.55000E+01	0.55000E+01	0.25920E+02
2	2206	9	8	CLOSED	0.55000E+01	0.55000E+01	0.51840E+02

OUTPUT

HDF5

- Linear Static, Normal mode and buckling
  - DISP
  - STRESS/STRAIN (CORNER)
  - CSTRESS/CSTRAIN
  - FORCE
  - OLOAD
  - SPCF
  - GPF



OUTPUT AND DIAGNOSTICS

Gauss Strain for solids

Gauss and Corner option for Stress/Strain in composites

Corner option for Composite Stress

Corner option for Strain results in Transient

Gauss and Corner option for Neuber results in linear analysis

Neuber Stress/Strain for Frequency Response and Transient

Filter option for displacement output request
- Static and Buckling
- Punch and H3D file

PSD/RMS Principal Stress
- H3D file

Mass Matrix Output (.mgg)
- PARAM, PRTMGG,YES

Plastic Energy output with Neuber
- "PLAS" keyword is added in ESE
- H3D support

Neuber option for GPSTRESS/GPSTRAIN

RMS Stress/Strain for composite

ERP output in SE residual run

ESE output in SE residual run for Frequency response and transient

Energy output for small displacement analysis

Additional result types output on the fly (NLMON)
- STRESS, STRAIN, CSTRESS, CSTRAIN, CONTACT

REPCASE with H3D output

SPCF for intermediate steps in OPTI format for LGDISP analysis

QUESTIONS?