Introduction to NX CAE
Tips and Tricks

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ATA Engineering, Inc.

Innovative Solutions through Test and Analysis Driven Design

ATA Engineering offers services for:
- Design
- Analysis
- Test
- Process Improvement

ATA has 30+ years experience running Nastran on wide variety of problems

ATA is a value-added-reseller of Siemens PLM Software's CAD/CAE and PLM software packages.
For more information, visit our website
http://www.ata-plmsoftware.com/

Introduction to NX CAE & Tips and Tricks

Introductory Concepts in NX CAE

Demonstration - Static analysis of an electronics housing

- Analyze a component
- Update the design & analysis

Demonstration - NX CAE Tips and Tricks

1. Mesh morphing
2. Solver export options
3. Post-processing templates
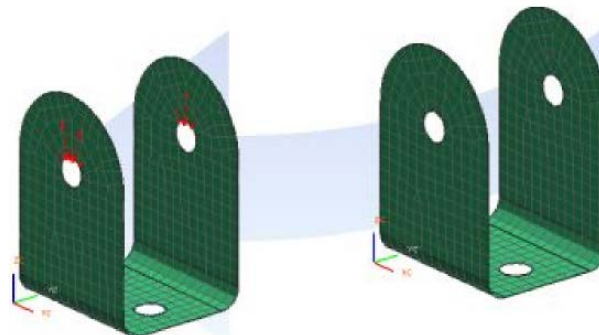
NX Advanced Simulation Has Distributed Files

Clockwise from top: A design part, a midsurfaced idealized part, a FEM file containing a mesh, a Simulation file containing the loads and boundary conditions.

bracket2.prt
(part, *.prt)
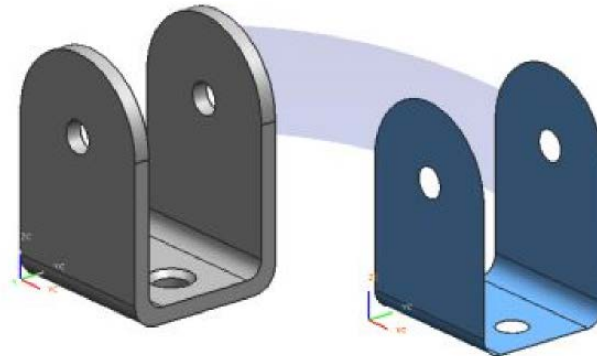
bracket2_i.prt
(idealized part, *i.prt)

bracket2_s.sim
(simulation, *.sim)

bracket2_f.fem
(fem, *.fem)

NX CAE Has Idealized Parts-
Use "Promote" to Idealize

With "Idealized Parts" you can:

- Simplify and remove design geometry details (idealize)
  - Examples: remove holes, fillets, chamfers, etc.
- Add new geometry useful for analysis not in the design (augment)
  - Examples: "split body" or "subdivide surfaces" for mesh control; create new geometry for analysis configurations.

Idealized Parts contain associative copies of design geometry. These copies are references to the design bodies that must be "enabled" by the **Promote** command before the geometry can be edited.

Associativity is one-sided:

- Changes to the Part allow the Idealized Part to update.
- Changes to the Idealized Part <u>do not</u> affect the original Part.

Distributed Files in NX CAE

To support concurrent engineering and data reuse, Advanced Simulation stores CAE model data in three primary and separate but related files:

- The *idealized part file* (_i.prt) is a special and optional NX file associated with a FEM file. The idealized part file usually contains simplifications or modifications of the design part geometry to make it suitable for analysis. The *design* or *master part* is referenced by the idealized part and permission to change the design part is not needed. All modifications are stored within the idealized part file, and the original master geometry is unaltered.

- The *FEM file* (.fem) provides the mesh for the Simulation file. Nodes, elements, and physical and material properties are stored with the FEM file. Typically, the FEM file also includes additional geometry derived from the idealized part file.

- The *Simulation file* (.sim) is an NX file that uses an NX FEM. Boundary conditions and solutions are stored with the Simulation file.

A fourth file type:

- An Assembly FEM file (.afm) is an optional and special type of CAE file for meshing assemblies. It is an NX FEM that behaves like an assembly of component FEMs. It is usually mapped to the design CAD assembly to provide the FEMs with orientation and position data. Simulation files can be built either from .afm or .fem files.

File -> Save As on NX CAE Files

When you perform a Save As on…	You are given the option to perform a Save As on…
The Simulation file (.sim)	The Simulation File
The FEM file (.fem)	The FEM file Any loaded Simulation files that reference it
The idealized part (_i.prt)	The idealized part file Any loaded FEM and SIM files that reference it
The design part (.prt)	The design part file Any loaded part files that reference it (parent assemblies) Any idealized part files that reference it Any loaded FEM and SIM files that reference it

Shown here:
Using **File→Save As**
To create a new "branch" of analysis
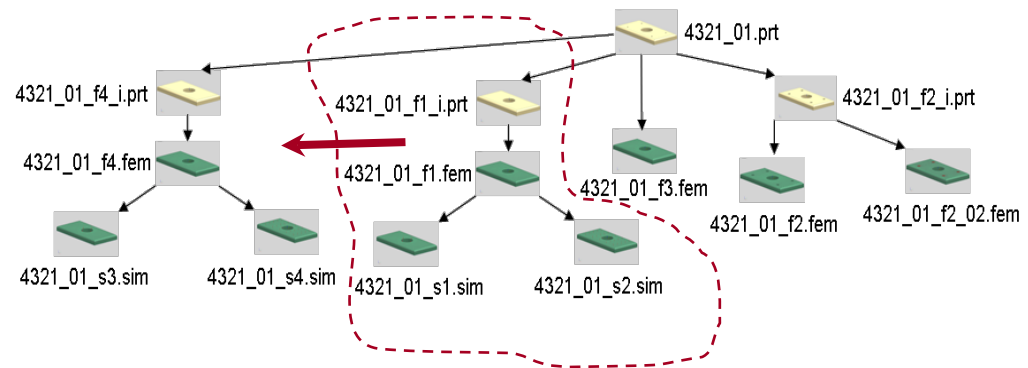
Demonstration – Introduction to NX CAE
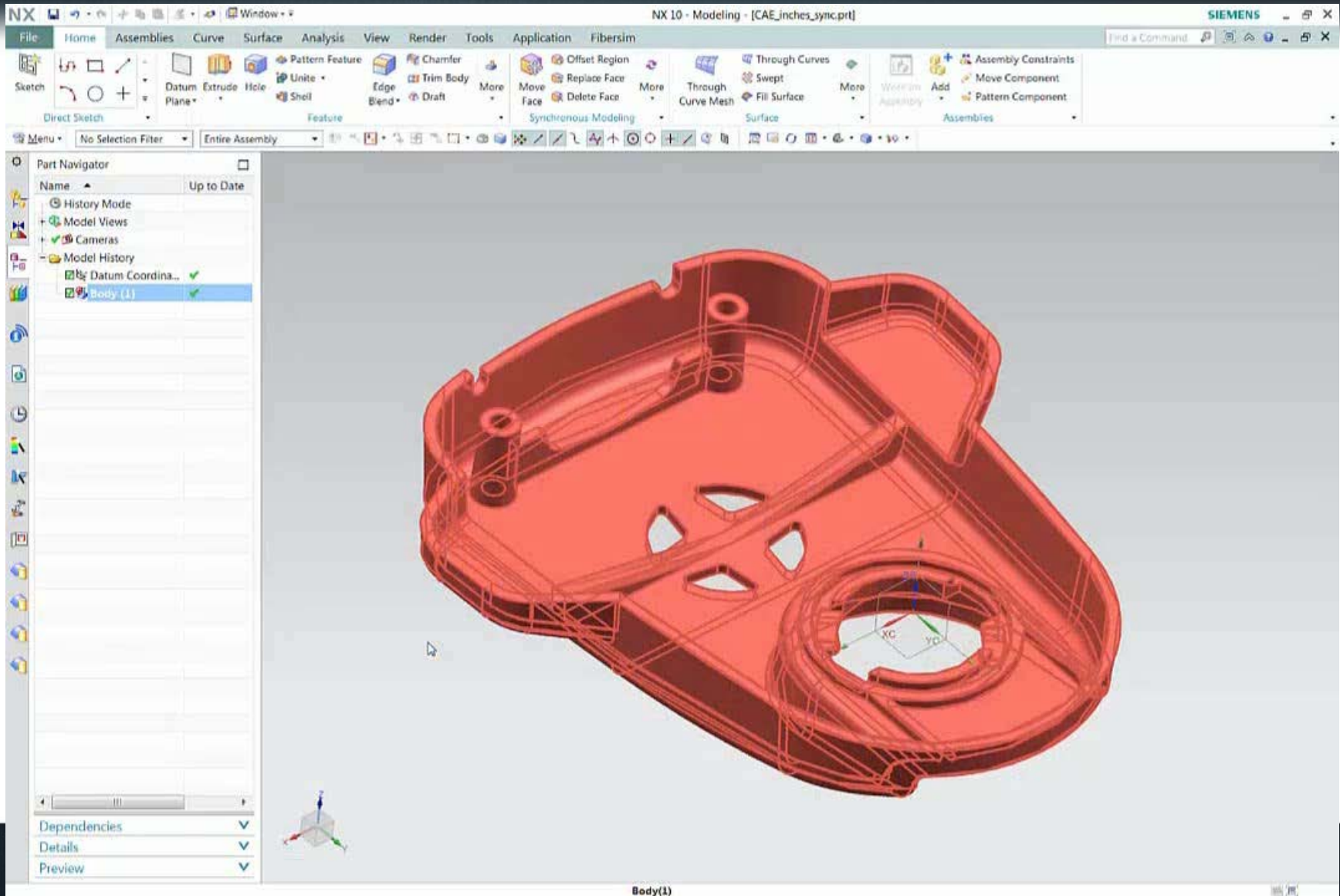
Demonstration & Discussion – User Tips and Tricks

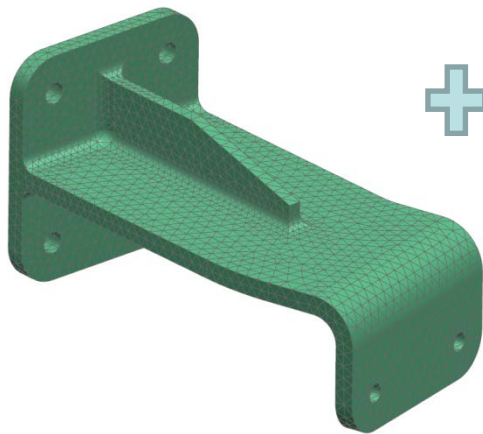Three favorites were selected for demonstration…

1. Pre-Processing / Node Association Command for Mesh Morphing

2. Advanced NXN Solver Options in NX Desktop

3. Post-Processing Templates

Many more details and tips in PLM World Presentation from 2015
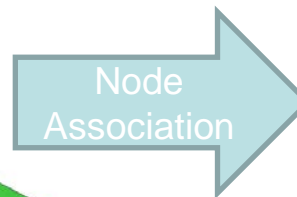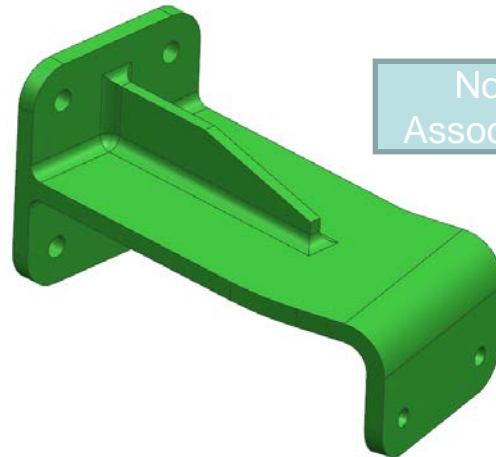
http://www.plmworld.org/p/do/sd/sid=6778

1 - Using the "Node Association" Command - Demonstration
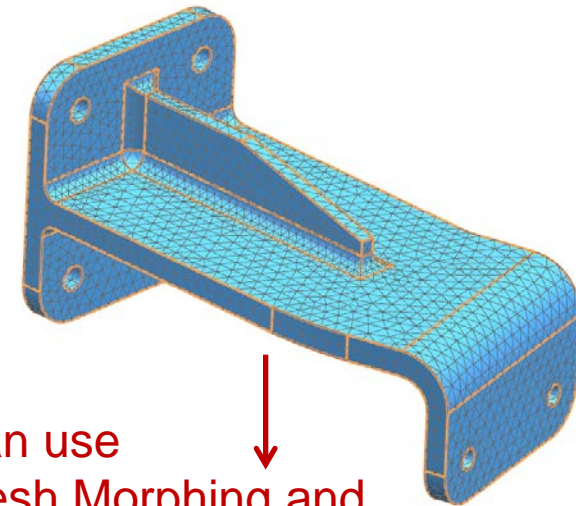
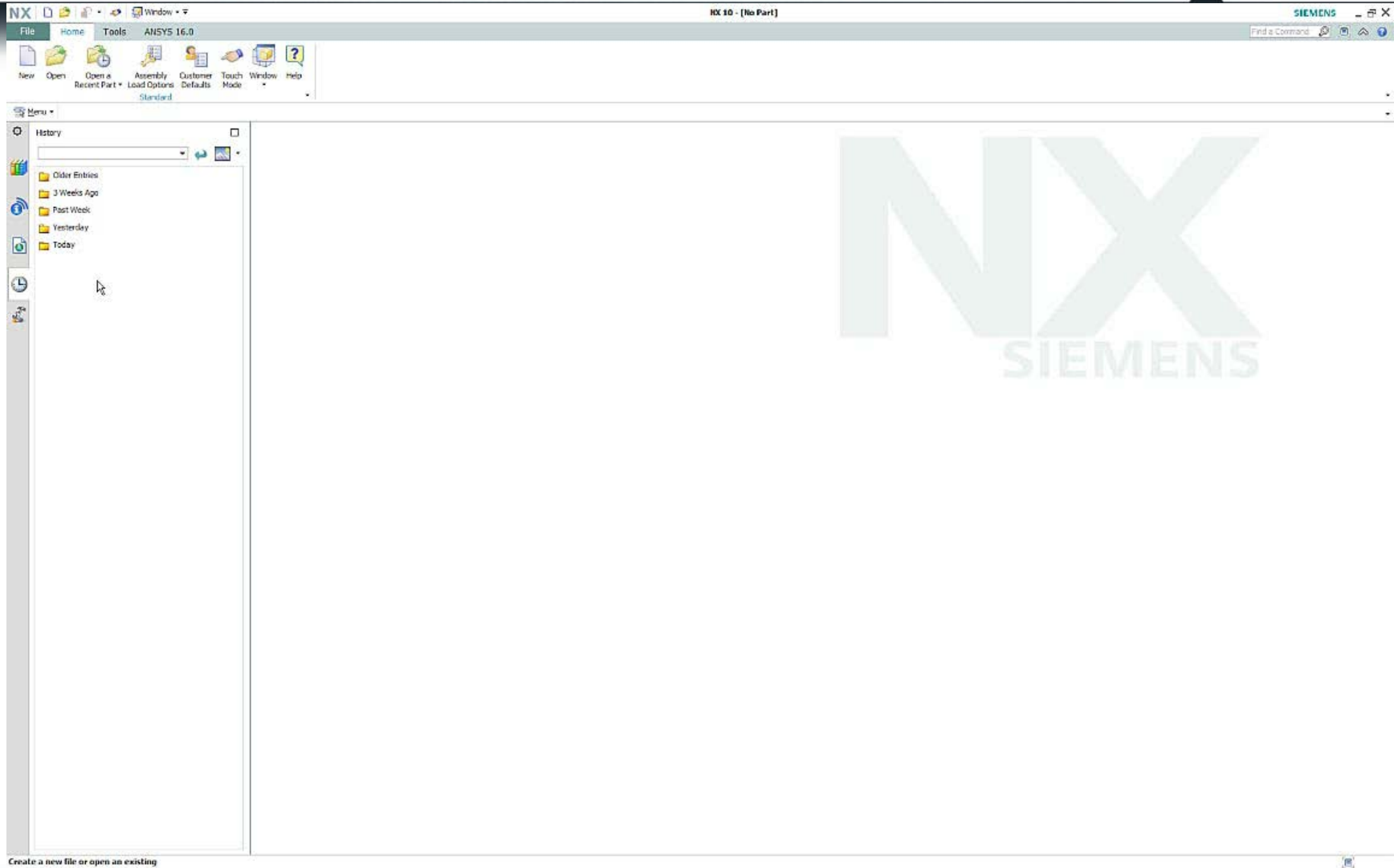Non-Associative
FEM

Representative
CAD Geometry

Associative FEM
Modify Mesh Geometry
Update Loads and BCs

Node Association

Can use
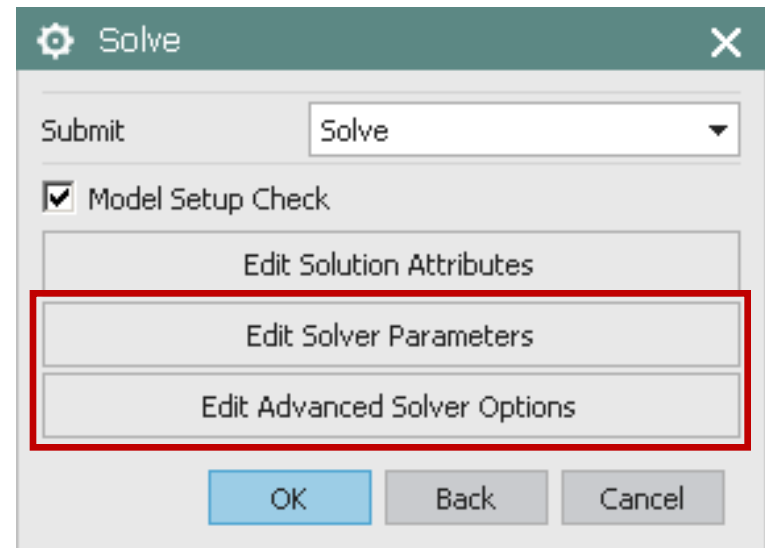Mesh Morphing and
Synchronous modeling
for powerful model
updates

1 - Using the "Node Association" Command - Demonstration

Using Solver Parameters and Advanced Solver Options

- Provides flexibility with what parts of the simulation get exported.

- Provides more control over how simulations are executed.

- NX gives you lots of solver control through these options

To learn more about options and memory for NXN performance, see presentation at:
http://www.ata-plmsoftware.com/resources/optimizing-nx-nastran-performance-on-an-hpc-cluster/

2.1 - Solver Parameters
Accessing Solver Parameters

Step 1: Initiate the solve by clicking the Solve icon on the Home ribbon or by right-clicking on the solution in the Simulation Navigator and selecting "Solve".
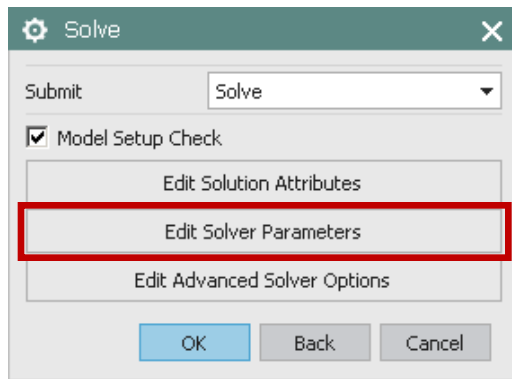

Step 2: In the Solve dialog box, select "Edit Solver Parameters."

Alternatively: Right click the solution in the simulation navigator and select "Edit Solver Parameters…".

Solver Parameters dialog box:

- Gives the user control over the optional parameters for submittal of the model to the NX Nastran or MSC Nastran solver. (equivalent to entering "Optional Keywords" in the solver command line)
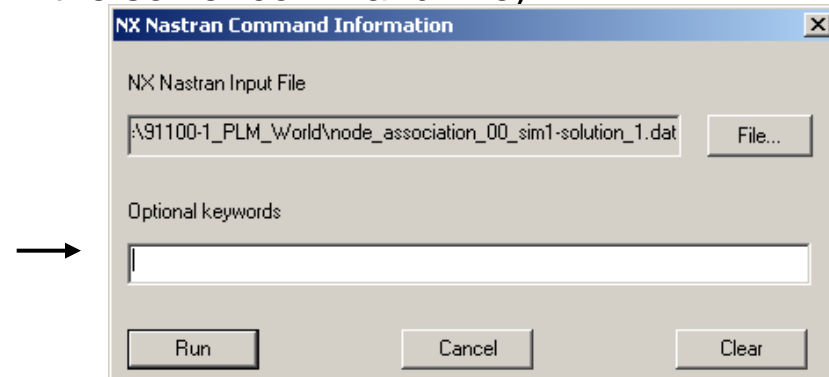
- The Solver Parameters dialog box also allows the user to specify the solver executable or version to use for the solution.
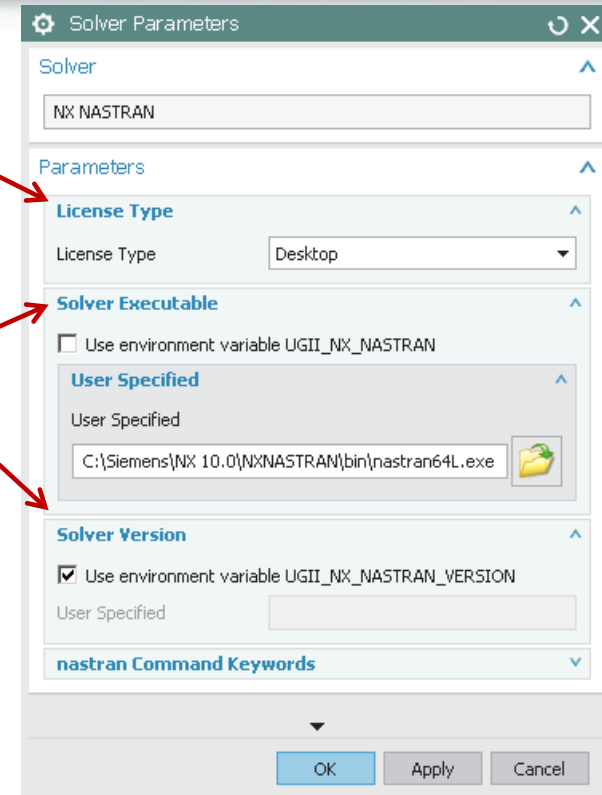
2.1 - Solver Parameters
Licenses and Environment Variables

License Type: May choose to use a Desktop or an Enterprise license for the solution

Solver Executable and Versions specified by environment variables: **UGII_NX_NASTRAN** and **UGII_NX_NASTRAN_VERSION**

- These variables can be reviewed in the "ugii_env_ug.dat" file and set in the "ugii_env.dat" file

- Default is to use the executable from the **UGII_NX_NASTRAN** environmental variable which is not the "L" version

- The "L" version of Nastran executable, **nastran64L.exe** allows for more than 8GB of RAM. In NX 10, this is the default, previously needed to set this.

- The solver version is the version number of NXN such as: UGII_NX_NASTRAN_VERSION=10

2.1 - Solver Parameters
Command Keywords

Allocate more resources

- Increasing the number of CPU's and RAM available to the solver can significantly decrease solution times for larger models.

- **memory** – specify the RAM used to complete the Nastran solution (must specify memory units, such as "gb")

- **old** – Enabling this checkbox causes solution results with the same name to be labeled incrementally (00, 01, 02, …) rather than overwriting previous results

- **scratch** – choose whether solution database files are stored or deleted after the solution completes. *Yes* will delete database files after the solution completes, *No* will store all database files, and *Mini* will store compact version of database files.

- **sdirectory** – choose the directory to which all database files and temporary solver files are written.
  - Hint: choose local drives to increase performance. Writing to a Solid State Drive (SSD) can further increase performance via decreased disk read/write times.

  parallel – specify the number of CPU's used to complete the solution (CPU's are used in parallel). Using more CPU's can significantly improve solve times.
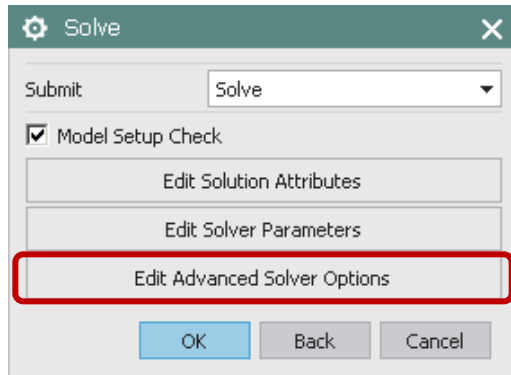
  Additional Keywords – see Nastran User's Guide

2.2 - Advanced Solver Export Options
## Accessing the Export Options

Step 1: Initiate the solve by clicking the Solve icon on the Home ribbon or by right-clicking on the solution in the Simulation Navigator and selecting "Solve".
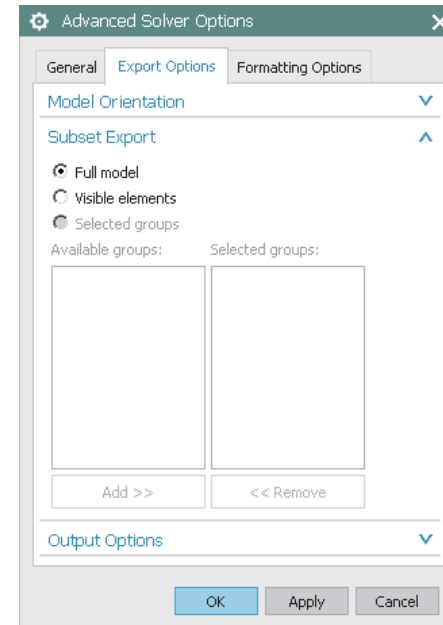

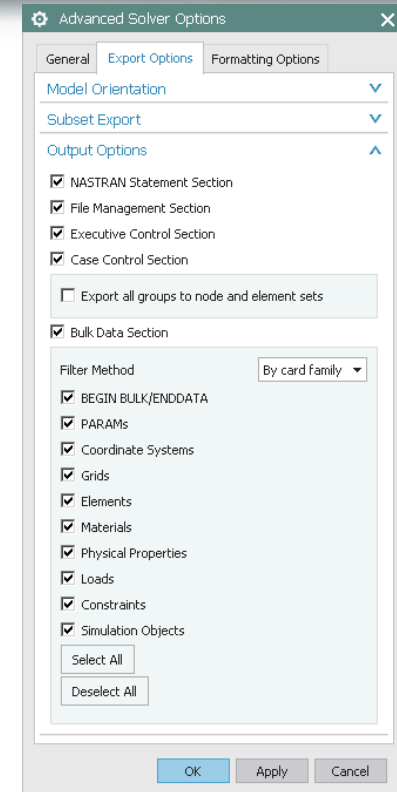**Step 2**: In the Solve dialog box, select "Edit Advanced Solver Options."
Alternatively: Right click the solution in the simulation navigator and select "Edit Advanced Solver Options…".
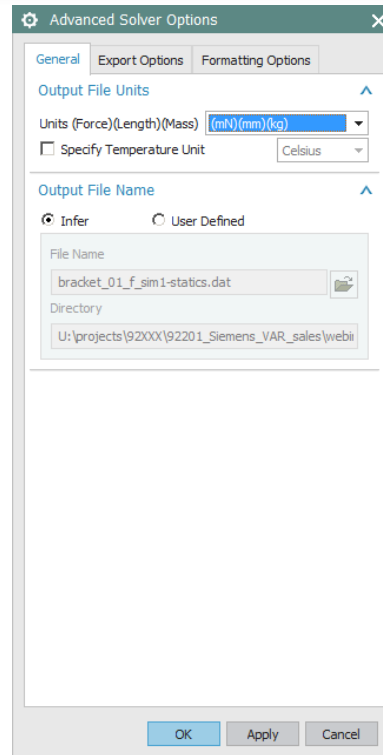
Step 3: Select units

Step 4: In the Advanced Solver Options dialog box, expand the Subset Export menu to select specific groups for export.

Step 5: Expand the Output Options menu to select specific bulk data objects for export. May filter by card type or by card name.

2.2 - Advanced Solver Export Options
Choices of what to export

The Export Options within the Advanced Solver Options dialog box give the user control over the bulk data written to the solver file.

The user can choose to withhold executive and case control information and certain bulk data information by simply deselecting data objects in the filter list. This allows for separate node/element, coordinate system, material, etc. files to be written.

In addition, the user has the option to export a subset of the model by "group" or export by "visible."

- **Export by "group"**: Only selected groups will be exported. Thus, two groups can have different elements and both can be solved in the same sim file without having to revise the FEM and either add/remove elements.

- **Export by "visible"**: Only the elements visible on the screen will be exported when the solve is called.
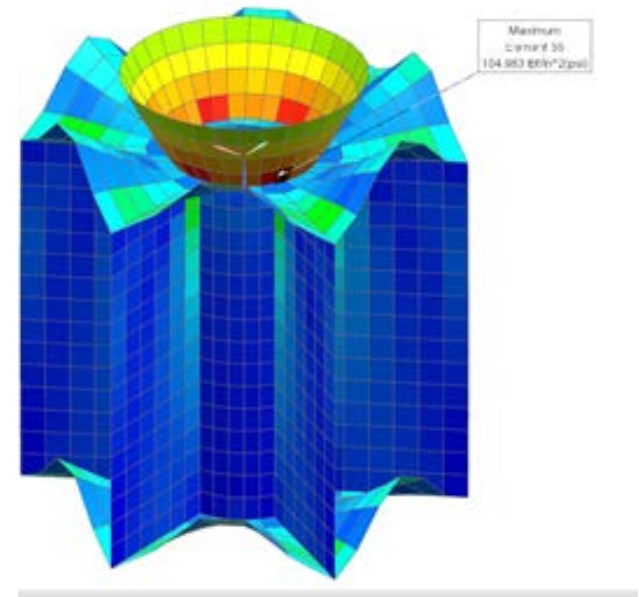
3 - Post-Processing Templates Demonstration

Post processing templates can be used to save information for viewing post processed results.

Templates can be created in NX10 in the post processing navigator by right clicking on a post view and picking the option for "new template".

The template definition is saved as an .xml file, which is a text file that can be edited to capture specifics that you want in the template.

Useful when results will be viewed in a specific format multiple times.

3 - Post-Processing Templates Demonstration
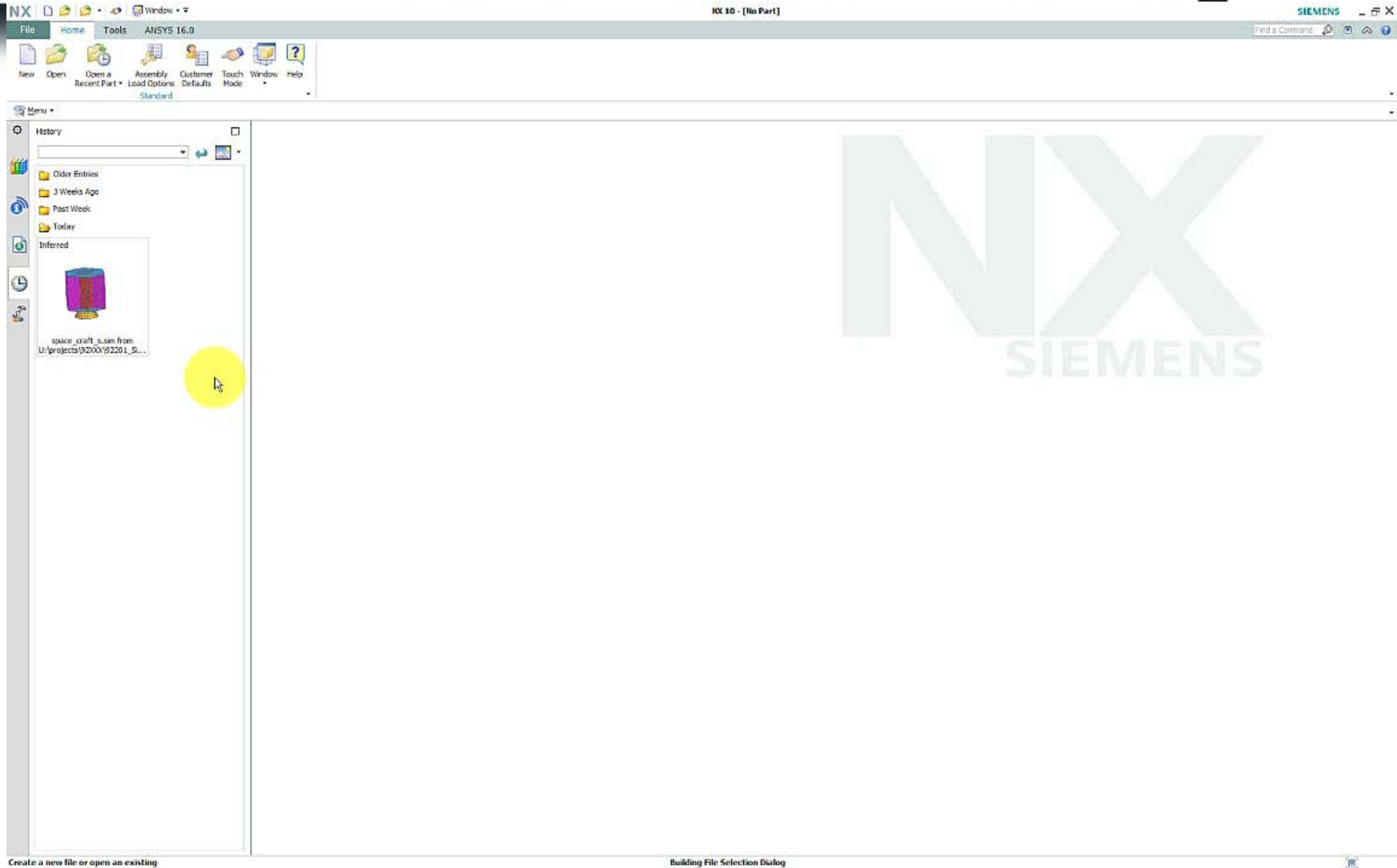
Tips and Tricks To Help You Get Even More Value Out of NX CAE

Demonstration - Static analysis of an electronics housing

- Analyze a component
- Update the design & analysis

Demonstration - NX CAE Tips and Tricks

1. Mesh morphing
2. Solver export options
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…And many more!

We've collected a wide array of other tips and tricks. Please contact us after the webinar if you are interested in learning more.

4. Matching Meshes at an Interface
5. Tetrahedral Meshing Control
6. Resolving Graphics Errors
7. Customizing Results Legend Text
8. Renumbering NX FEMs with NX Open
9. Miscellaneous Tips

Thank You!

Questions?

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https://www.surveymonkey.com/r/Y8YXSVT