CP125791

All for Composite Simulation as Manufactured: Autodesk Nastran, Helius PFA, and Moldflow Ecosystem

Jaesung Eom, David Veinbergs
Autodesk

Learning Objectives

- Learn how to use Autodesk Nastran and AME to run the short-fiber plastic composite model as manufactured material properties from Moldflow
- Learn how to use Autodesk Nastran In-Cad and AME to prepare Autodesk Nastran for progressive failure analysis of short-fiber composite materials
- Learn how to interpret progressive failure results of continuous-fiber and short-fiber composite materials

Description

This talk will present how the integration of Autodesk Nastran software, Helius PFA software with Advanced Material Exchange (AME) capabilities, and Moldflow software work together in the real world of complex composite simulations. Composite materials are an advanced family of materials that have been in development for decades and continue to gain usage in the aerospace, marine, automotive, and sporting goods industries. While composites enjoy benefits such as high strength-to-weight and stiffness-to-weight ratios, they are considerably more complicated than most metals and plastics. Helius PFA with its AME capabilities is part of the Autodesk Solution for digital prototyping, and it provides fast, accurate, and flexible tools for enhanced finite element analysis (FEA) of composite structures—manufactured from unidirectional or woven composites to injection-molded, short-fiber filled composites—including progressive failure analysis to help reduce testing and shorten design cycles by identifying potentially unforeseen design and material deficiencies.

Speaker(s)

Jaesung Eom is a Senior Research Engineer in Autodesk Digital Manufacturing Group simulation team. He is interested in nonlinear computational mechanics and the systematic identification of material properties by stochastics, ML and evolutionary strategy. He had his PhD on computation mechanics at KAIST and worked on biomechanical inverse problems on the soft tissue at RPI before joining Autodesk.

David Veinbergs is a Research Engineer in the Autodesk Digital Manufacturing Group simulation division. He has a B.S in Mechanical Engineering from the University of Wyoming. His primary interests are computational mechanics and progressive failure of composite materials.

Table of Contents

Introduction: What is As Manufactured Simulation?

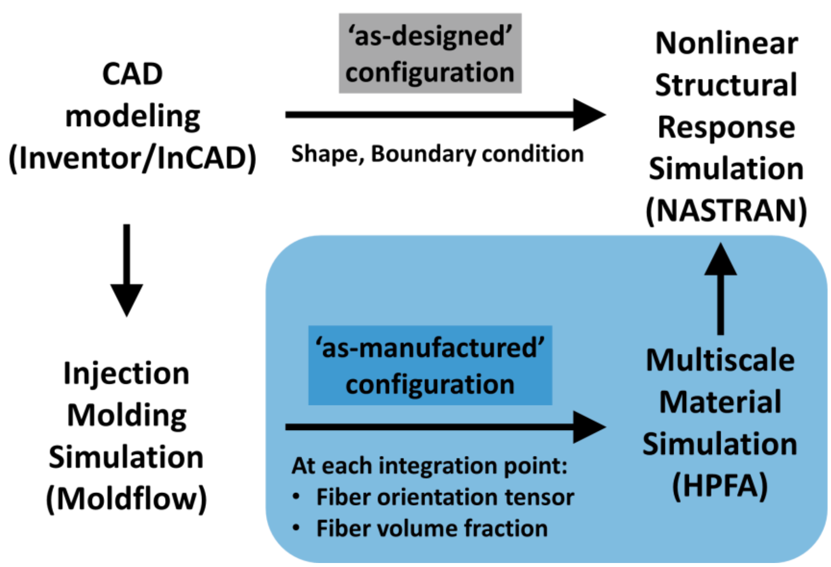
The use of composite has become common to achieve higher structural performance as higher stiffness-to-weight and higher strength-to-weight ratios from the high-tech aerospace industry to the consumer product industry. This talk will walkthrough how to build high-fidelity "As-Manufactured" composite simulation with Autodesk digital prototyping software ecosystem – Moldflow, NASTRAN, and HeliusPFA.

What is As-Manufactured Simulation of Composite?

The favorable structural characteristic of composite is due to the anisotropic nature of composite material (different depending on the direction of the applied force). For instance, the stiffness of a composite panel will often depend upon the orientation of the applied forces and/or moments.

As-"Injection molded" Short fiber Composite Simulation

Injection molding simulation software packages can be used to predict the distribution of fiber orientation throughout a part, in addition to the warped shape of the ejected, room-temperature part. To facilitate subsequent nonlinear (progressive failure) structural simulation of the short fiber filled part, Autodesk has developed new software to seamlessly link the results of injection molding simulation with nonlinear structural response simulation that features a multiscale progressive failure model for short fiber filled plastics.

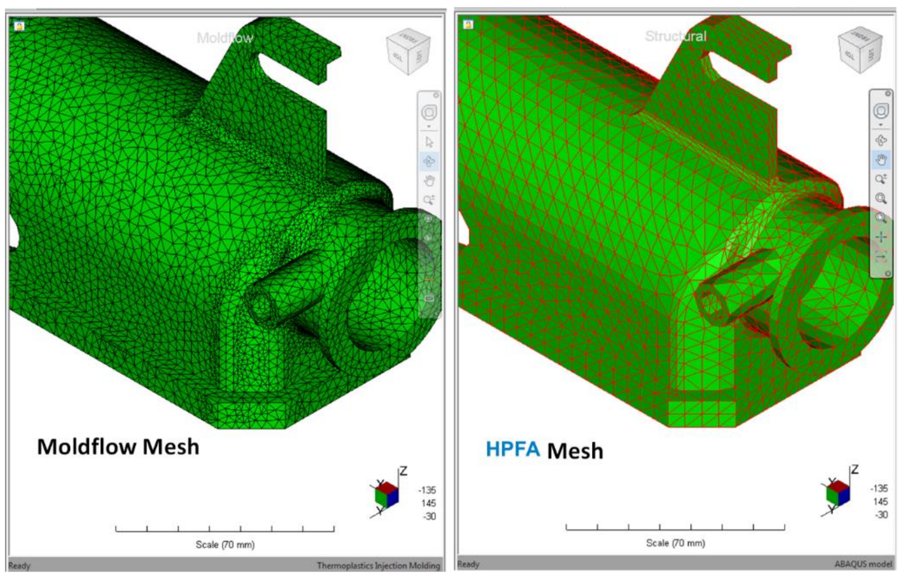


AS-"INJECTION MOLDED" SHORT FIBER COMPOSITE SIMULATION: INTEROPERATION OF MOLDFLOW, HELIUS PFA AND NASTRAN ENABLES HIGH FIDELITY SIMULATION WITH MANUFACTURING CONDITION INTO ACCOUNT.

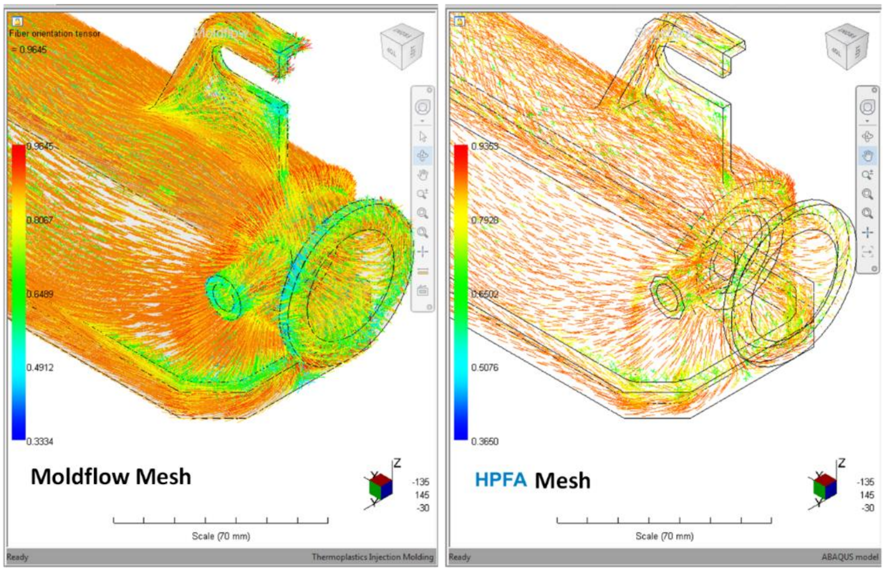
One-stop As-Manufactured solution from Autodesk

Autodesk offers one stop solution from design shape in CAD, injection molding simulation, structural simulation to evaluate shapes and manufacturing parameters during molding process for As-Manufactured simulation for injection molding composites.
Nastran In-CAD 2018.1 supports Advanced Material Exchange (AME) workflow in Helius PFA enabling as-manufactured workflow for plastic parts – this enables including as-manufactured performance earlier in the design process to make a better part and explore all opportunities for a great part.

- Moldflow predicts the spatial distribution of fiber orientation in short fiber filled, injection molded plastic parts. The 2nd order fiber orientation tensor at a point essentially provides a statistical description (in the continuum sense) of the orientation of fibers.
- AME workflow in Helius PFA: Automated mapping of the injection molding simulation predicted fiber orientation distribution and fiber volume fraction distribution onto the finite element mesh that will be used for the nonlinear structural response simulation,
- NASTRAN InCad solver: Enhancement of the structural response simulation with a multiscale, progressive failure, constitutive model for short fiber filled plastic materials that accounts for plasticity and rupture of the matrix constituent material, resulting in a composite material that exhibits an anisotropic, nonlinear response,



COMPARISON OF THE MESH USED FOR SIMULATION OF THE INJECTION MOLDING PROCESS (LEFT) AND THE MESH USED FOR STRUCTURAL LOADING SIMULATION (RIGHT)



COMPARISON OF THE DISTRIBUTION OF AVERAGE FIBER ORIENTATION PREDICTED ON THE INJECTION MOLDING MESH(LEFT) AND MAPPED ON THE STRUCTURAL MESH(RIGHT)

Theory – How "As-Manufactured" simulation works

Composite materials exhibit very complex multiscale constitutive behavior (including damage initiation, damage evolution, stiffness reduction and ultimate rupture). Most material models that are available in commercial FE codes do not adequately describe the response of composite materials to stress states outside the elastic range.

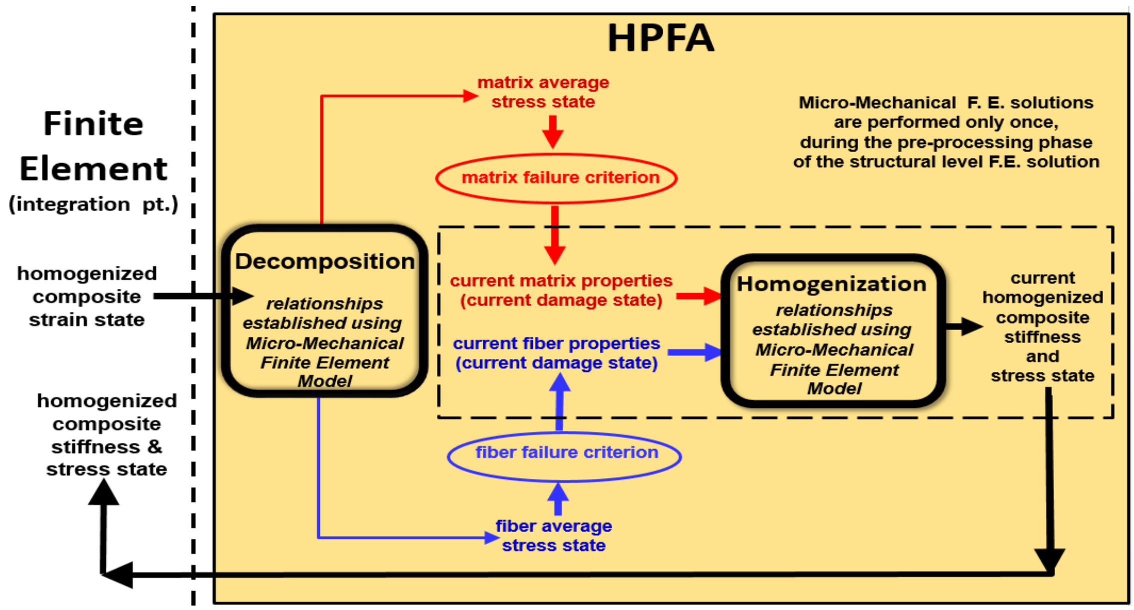
Even at relatively low structural load levels, inherent multiscale stress concentrations can cause:

- plasticity/damage/failure to the matrix constituent
- damage/failure to the fiber constituent
- damage/failure to the fiber/matrix bond
- damage/failure to the resin-rich interplay bond (delamination)

These material behaviors have specific stiffness reduction consequences on the composite structural response.

The individual composite load states in macro-scale finite element model influence the micro-mechanical finite element model thru discrete damage states and each constituent's average stress and strain state are accounted in micro-scale at the same time. The homogenization mapping and decomposition mapping are consistent within the multiscale finite element model.

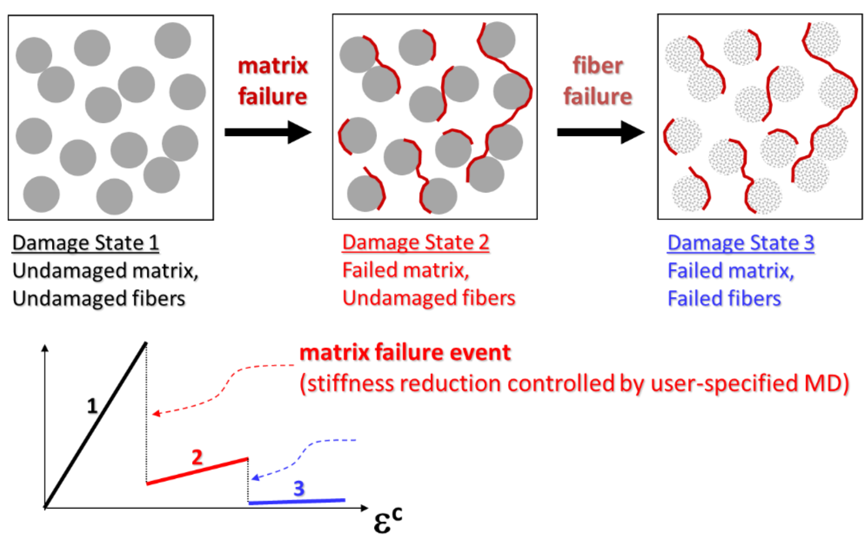
Multiscale Simulation Methodology



MULTISCALE SIMULATION SCHEME IN FINITE ELEMENT METHOD

Progressive failure analysis

The decomposition process gives us access to constitutive average stress and strain states at any integration point in a structural-level finite element model. The failure state of the composite material is completely determined by the failure states of its individual constituent materials. As a result, whole composite follows discrete damage state along the evolution of the local damage/failure with minimum computation cost.

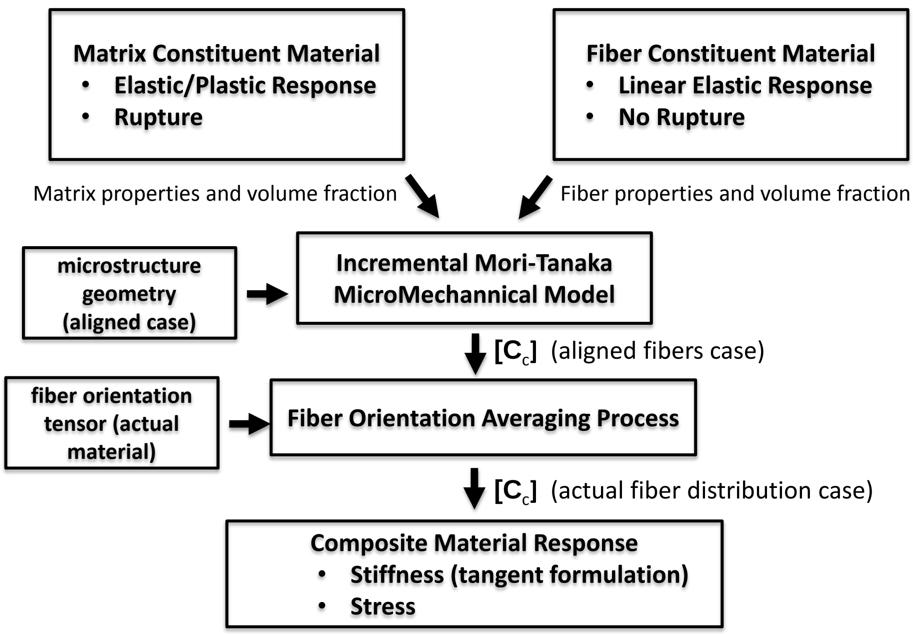


SIMPLE CASE: THREE DISCRETE DAMAGED STATES OF PROGRESSIVE FAILURE MODEL

Multiscale Short fiber composite model

During a structural-level finite element simulation of mechanical loading of the short fiber filled plastic part, the predicted deformation of the part is based on the stiffness of the homogenized composite material. However, to predict plasticity and rupture of the matrix material, Helius PFA must decompose the finite element code's homogenized composite strain into the average strain in the matrix constituent material. In addition, the multiscale material model can homogenize the response of the evolving heterogeneous microstructure into composite-level stress, strain and stiffness. For this, Helius PFA uses the individual constituent properties are input into an incremental Mori-Tanaka micromechanical model that can accommodate evolving matrix properties.

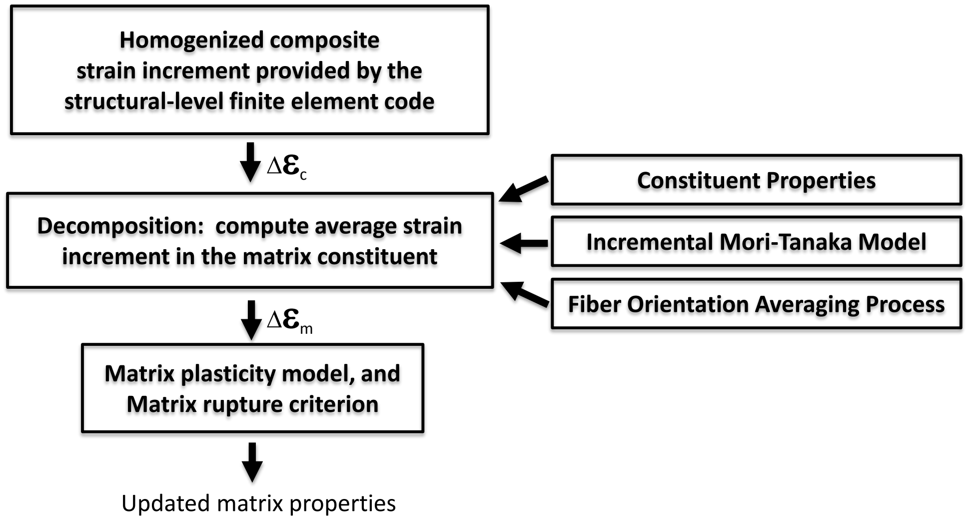
The incremental Mori-Tanaka micromechanical model produces homogenized composite properties for the idealized, perfectly aligned material. These properties, in turn, are operated upon by the fiber orientation tensor to produce the homogenized composite properties for the real material with the actual fiber orientation distribution.



SCHEMATIC DIAGRAM OF THE MULTISCALE MATERIAL MODEL'S HOMOGENIZATION PROCESS.

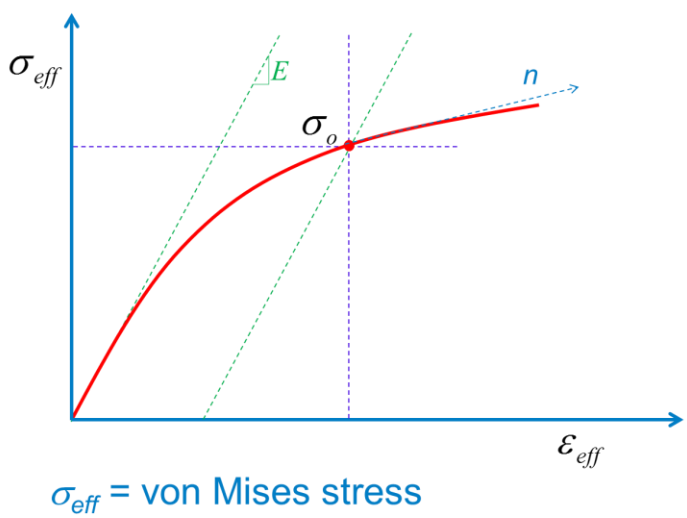
How individual matrix plasticity model evolved from macro change

The below diagram provides a schematic diagram of the decomposition process that maps homogenized composite strain increments into the average strain increments in the plastic matrix constituent material. The decomposition process is taken from Nguyen et al. [11,12] and makes use of the instantaneous (tangent) constituent properties, the incremental Mori-Tanaka micromechanical model and the fiber orientation tensor. The computed average strain increment in the matrix constituent is used to drive the matrix plasticity model and predict the evolution of the matrix tangent modulus.



SCHEMATIC DIAGRAM OF THE DECOMPOSITION PROCESS TO CONVERT THE COMPOSITE-LEVEL STRAIN STATE INTO THE MATRIX-AVERAGE STRAIN STATE

Matrix plasticity model



RAMBERG-OSGOOD PLASTICITY MODEL FOR MATRIX CONSTITUENT IN COMPOSITE

The response of the matrix constituent material is provided by a Ramberg-Osgood plasticity model that has been enhanced to allow the predicted plastic response to exhibit sensitivity to the direction of the loading relative to the fiber direction. The effective hardened yield strength of the matrix constituent material can be expressed as:

$$\sigma_Y^h(\varepsilon_{eff}^p) = E^{1/n}(\sigma_0)^{(n-1)/n}\varphi^{1/n} \tag{1}$$

where σ_0 and n are the typical material parameters that are used by the standard (isotropic) Ramberg-Osgood plasticity model, and φ is the effective plastic strain in the matrix constituent material. The yield function is satisfied when the effective stress in the matrix constituent matches the hardened yield strength.

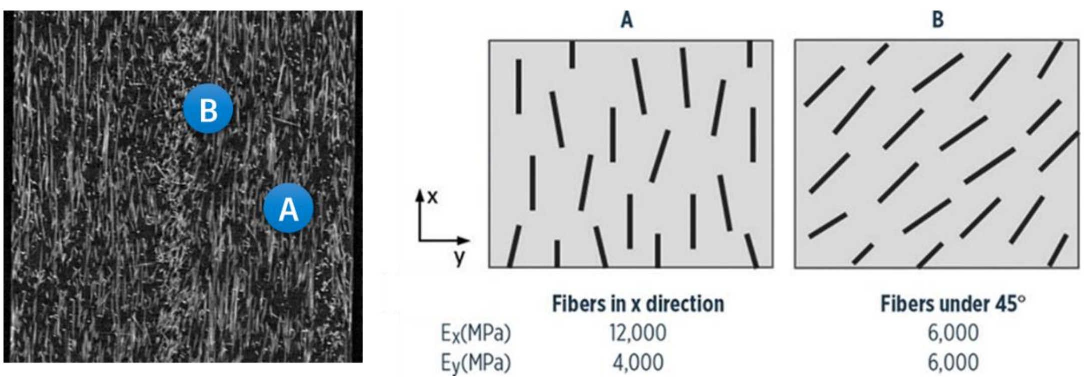
$$f(\varphi) \;=\; \sigma_{eff}(\varphi) \;-\; \sigma_Y^h(\varphi) \;=0 \tag{2}$$

For isotropic materials, the effective (scalar) stress is often represented by the von Mises stress

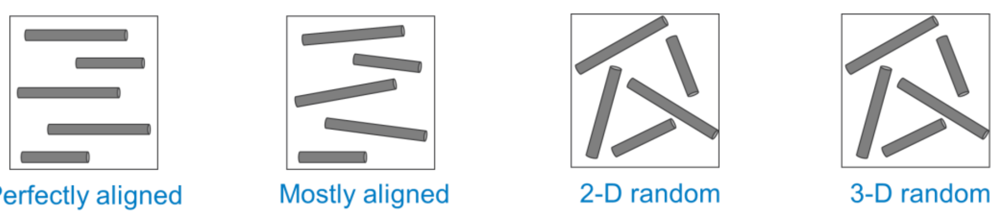
$$\sigma_{eff} \equiv \sqrt{\begin{array}{c}\left((\sigma_{11}-\sigma_{22})^2+(\sigma_{22}-\sigma_{33})^2+(\sigma_{33}-\sigma_{11})^2\right)/2 \\ +3[\,(\sigma_{12})^2+(\sigma_{23})^2+(\sigma_{31})^2]\end{array}} \tag{3}$$

where it is understood that the stress components represent the average stress in the matrix constituent material.

How fiber orientation affect material response



HOW FIBER ORIENTATION AFFECT MATERIAL RESPONSE (IMAGE COURTESY FROM COMPOSITE WORLD)



$$[FOT]=\begin{bmatrix}1&0&0\\0&0&0\\0&0&0\end{bmatrix} \qquad [FOT]=\begin{bmatrix}0.836&-0.056&-0.018\\-0.056&0.098&0.129\\-0.018&0.129&0.066\end{bmatrix} \qquad [FOT]=\begin{bmatrix}0.5&0&0\\0&0.5&0\\0&0&0\end{bmatrix} \qquad [FOT]=\begin{bmatrix}1/3&0&0\\0&1/3&0\\0&0&1/3\end{bmatrix}$$

$\lambda_I = 1.0$ $\qquad \lambda_I = 0.841$ $\qquad \lambda_I = 1/2$ $\qquad \lambda_I = 1/3$

HOW DOES THE MODEL ACCOUNT FOR CHANGES IN THE DEGREE OF FIBER ALIGNED-NESS?

Directional sensitivity of the plasticity model must disappear as the fiber alignment changes from mostly-aligned to mostly-random. This characteristic can be achieved by making α and β linear functions of λ_I where the condition $\alpha = \beta$ is achieved when the fiber alignment becomes random, i.e. at $\lambda_I = 1/3$ or possibly $\lambda_I = 1/2$.

Mapping Moldflow Results to Nastran In-CAD and Performing Progressive Failure Analysis with Helius PFA

Model Background and Overview

To demonstrate the full Autodesk workflow for as-manufactured simulation an automotive water pump housing cover will be considered. The part is a good example of complex geometry, varying fiber orientations, and injection molding weld lines. The full workflow involves the creation of the geometry in Inventor, the injection molding simulation in Moldflow, the creation of the FE model in In-CAD, the mapping of the Moldflow results to the FE model in AME, and the simulation of the structural model in In-CAD using the Nastran solver with Helius PFA composites functionality. In this example, the creation of the geometry in Inventor and the injection molding simulation in Moldflow will not be covered in depth. The focus will be on the creation of the FE model, the mapping of Moldflow results, and the post processing and interpretation of the progressive failure results provided by Helius PFA within the typical Nastran results. Completed files are available for downloaded here. Additional information may be found in the Moldflow, Inventor, Helius PFA, and Nastran In-Cad online help documents.

Moldflow Injection Molding Simulation

The part is constructed of PolyPacific Extron 3019 HS glass fiber filled polypropylene material. The filler represents 30% of the material, by weight, and has an aspect ratio of 25. The default recommended processing parameters provided by Moldflow are used for the simulation.

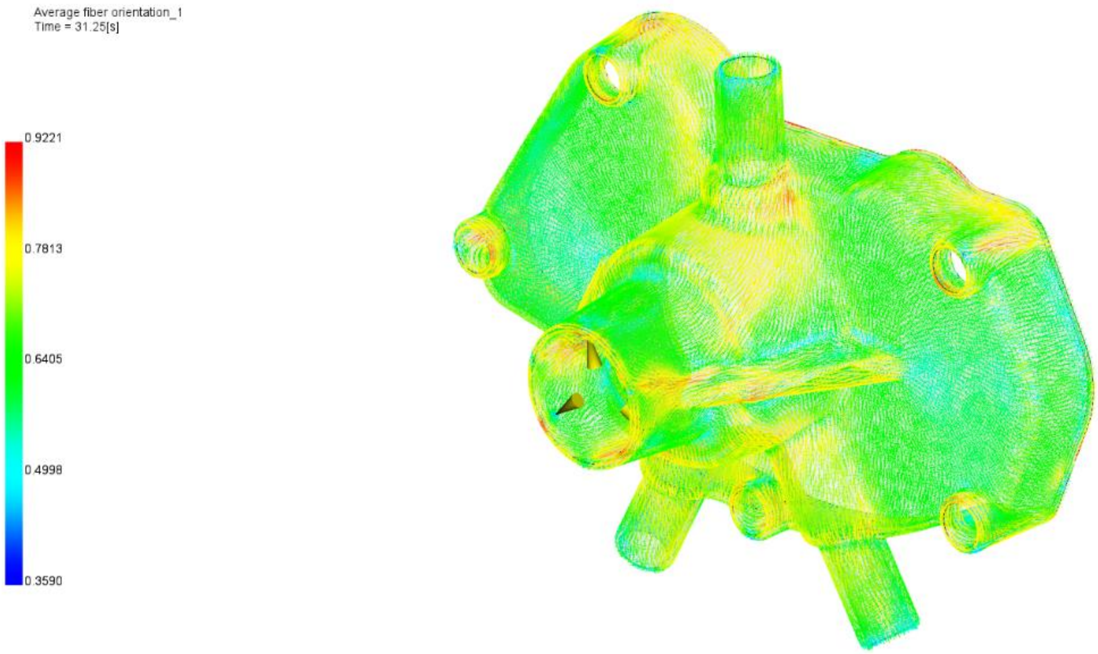
Moldflow Analysis Settings

To export the Moldflow results to Nastran In-CAD through AME a minimum Moldflow analysis sequence of "Fill + Pack" is required. To include the deformation resulting from residual stresses "Fill + Pack + Warp" is required. To include the temperature and pressure history at all weld surface points the following settings must be activated in Moldflow prior to the injection molding analysis, these settings activate the creation of the .ws3 file in the Moldflow results. This file is required for mapping of weld lines in AME.

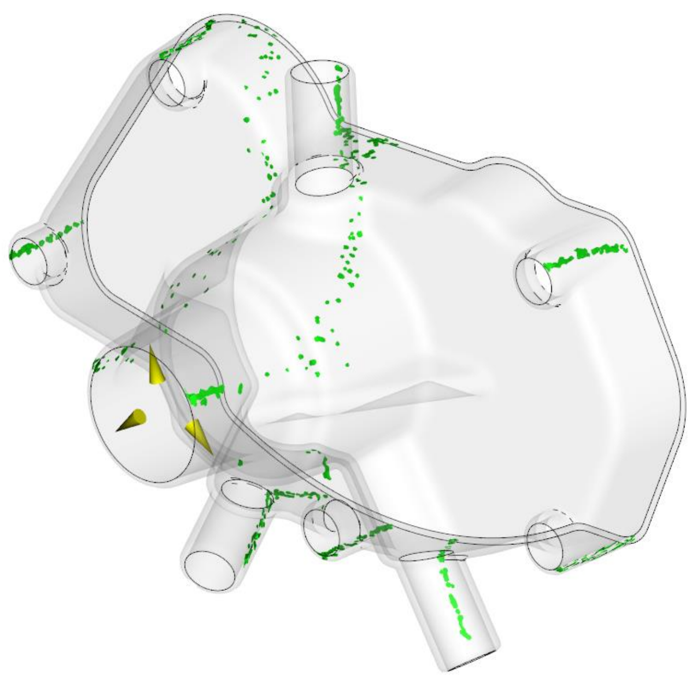
1. Click **Process Settings**
2. Click **Advanced Options**
3. Click **Edit** in the **Solver Parameters** frame
4. Select the **Interface** tab
5. Turn **ON** the **Weld surface strength analysis** option
6. Click **OK**

Moldflow Results

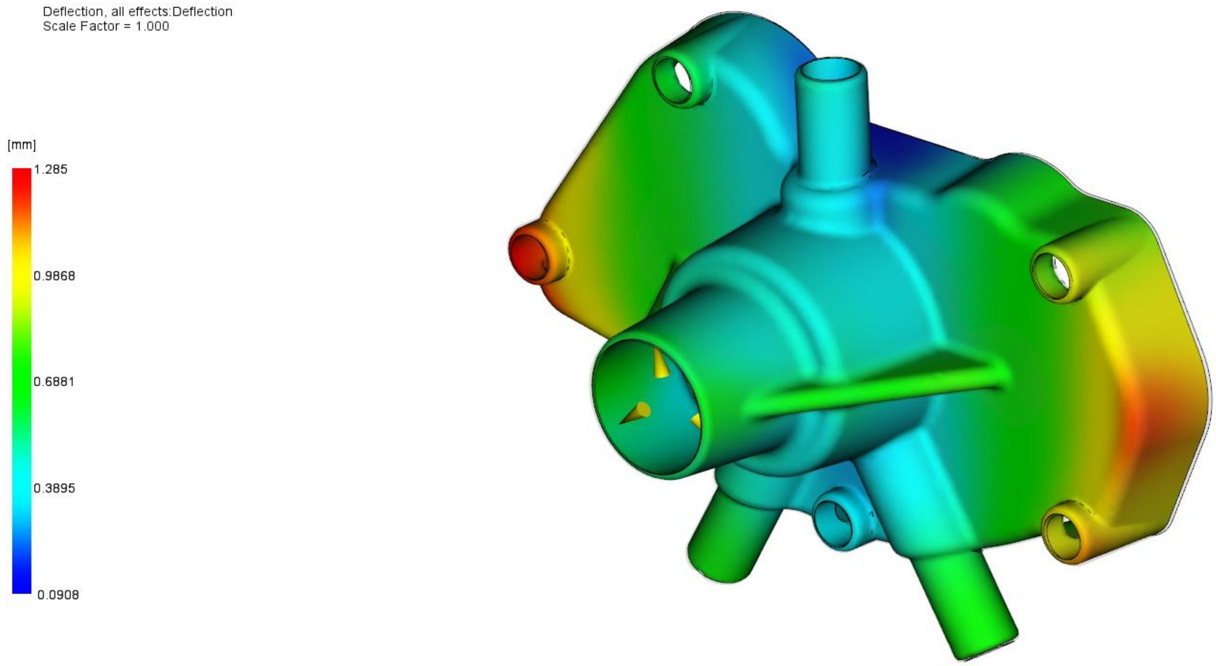
For this analysis, the deformation due to residual stresses and the weld line strength analysis are included for demonstration purposes only. The structural analysis will be run considering the as manufactured fiber orientations. Below are the results from the Moldflow simulation for fiber orientation, deformation due to residual stresses and the final location of the weld surfaces after packing.



MOLDFLOW PREDICTED FIBER ORIENTATION TENSOR.



MOLDFLOW WELD LINE MOVEMENT.



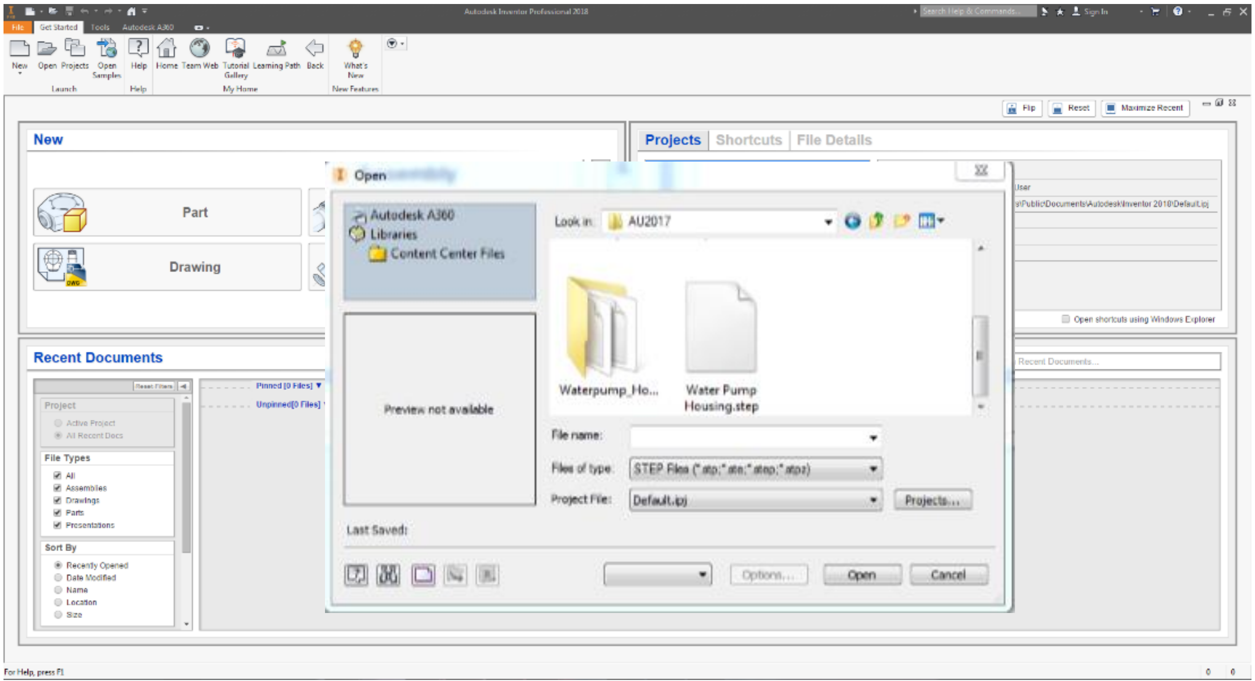
MOLDFLOW FREE DEFORMATION.

Inventor

For this case study we will use pre-generated geometry in the form of a STEP file. Generally, the geometry would be created within Inventor, saved, and used in Moldflow and In-CAD. Note that Moldflow will accept native Inventor files.

Importing Geometry

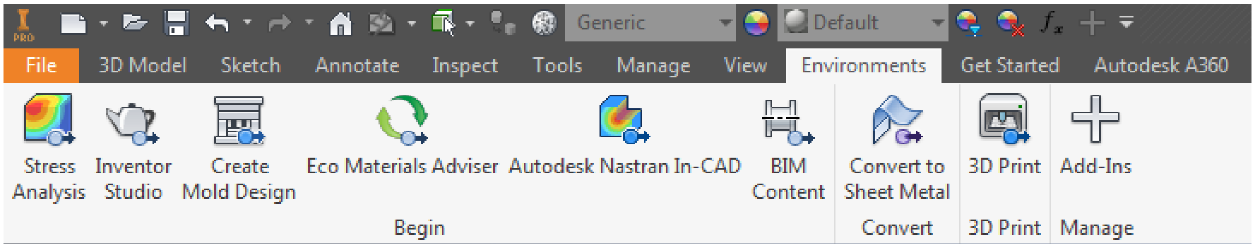
Open the step file by clicking **Open** and selecting the STEP file type from the pulldown.



IMPORTING STEP FILE INTO INVENTOR.

Nastran In-CAD

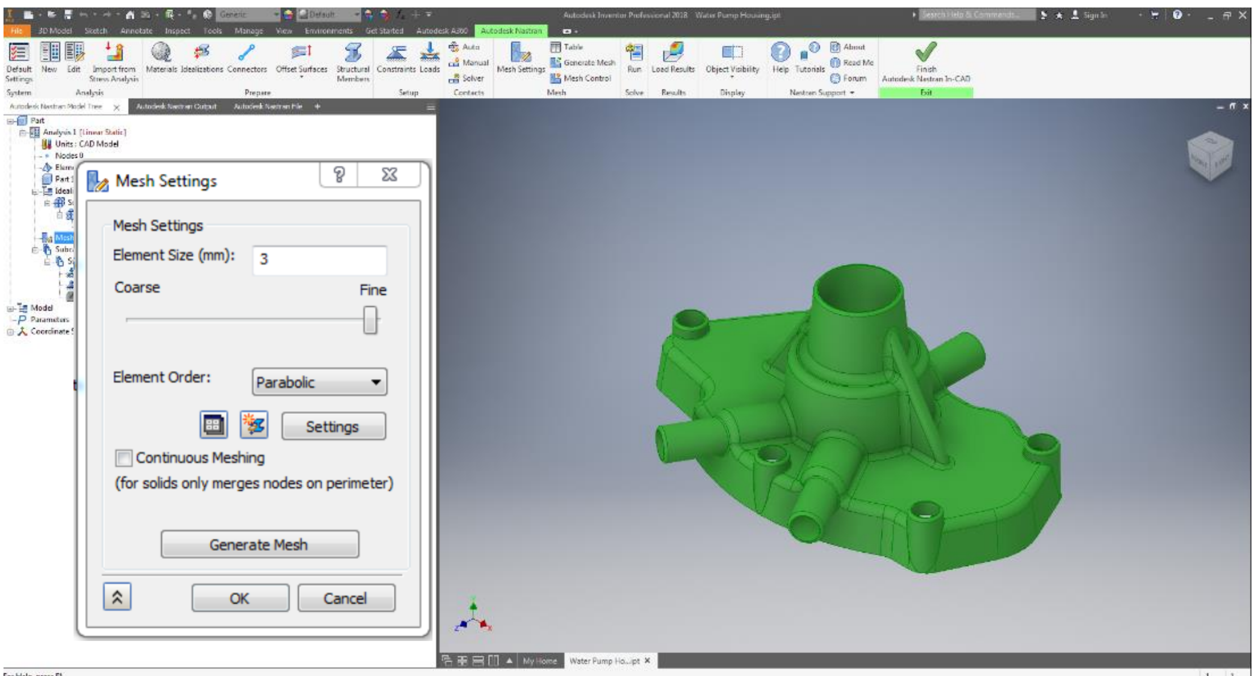
Nastran In-CAD exists in Inventor within the **Environments** tab shown below. To enter the Nastran In-CAD environment simply click on the **Autodesk Nastran In-CAD** icon.



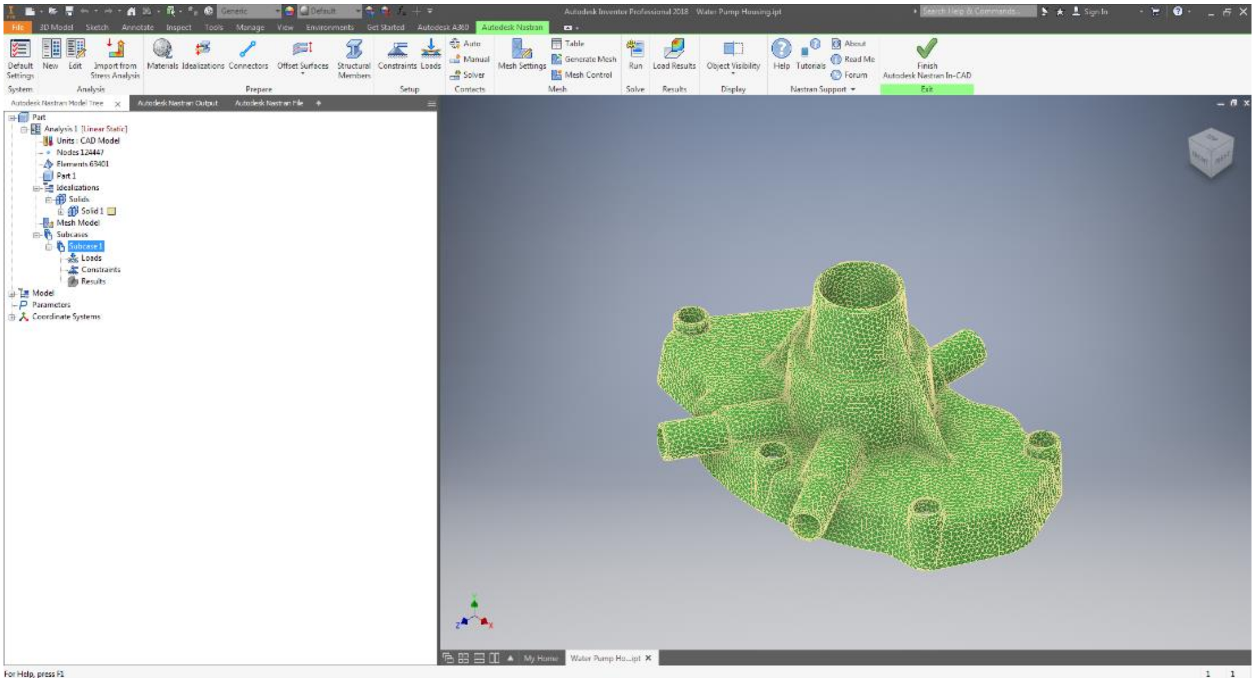
INVENTOR ENVIRONMENTS TAB.

Meshing Geometry

To mesh the geometry, click on the **Mesh Settings** icon and enter "3" in the **Element Size** field. Click **OK** to generate the mesh.



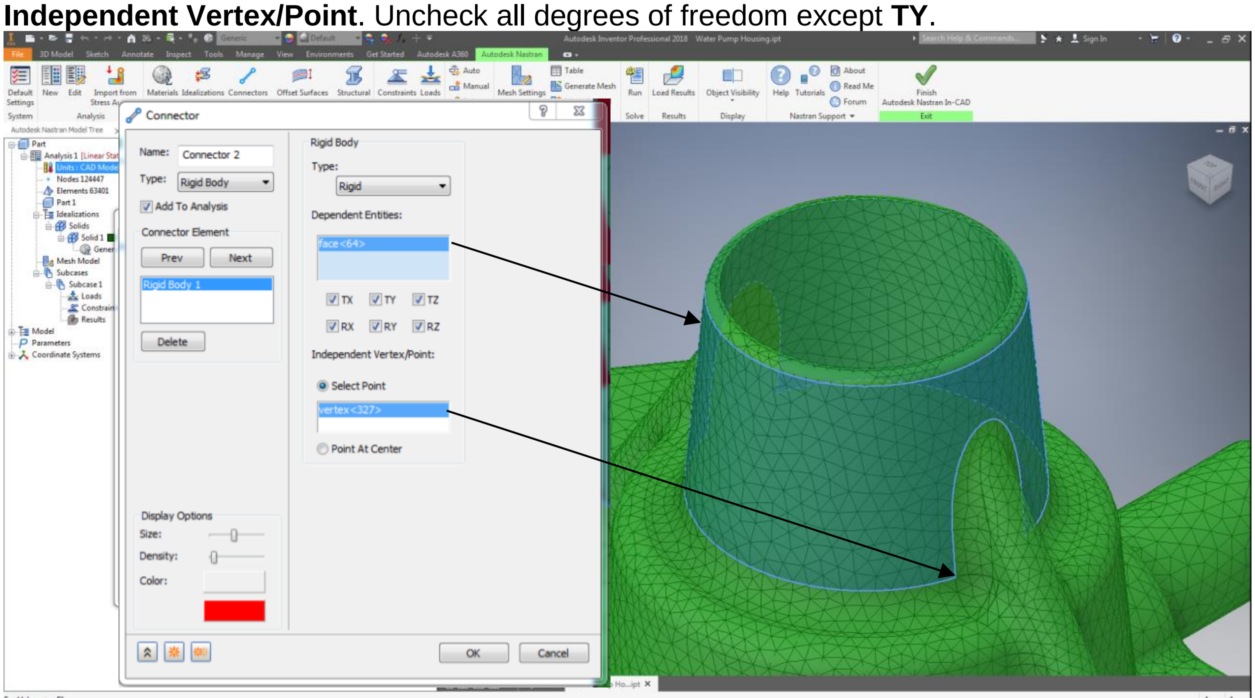
IN-CAD MESH SETTINGS.



MESHED GEOMETRY.

Rigid Body Tie Constraint

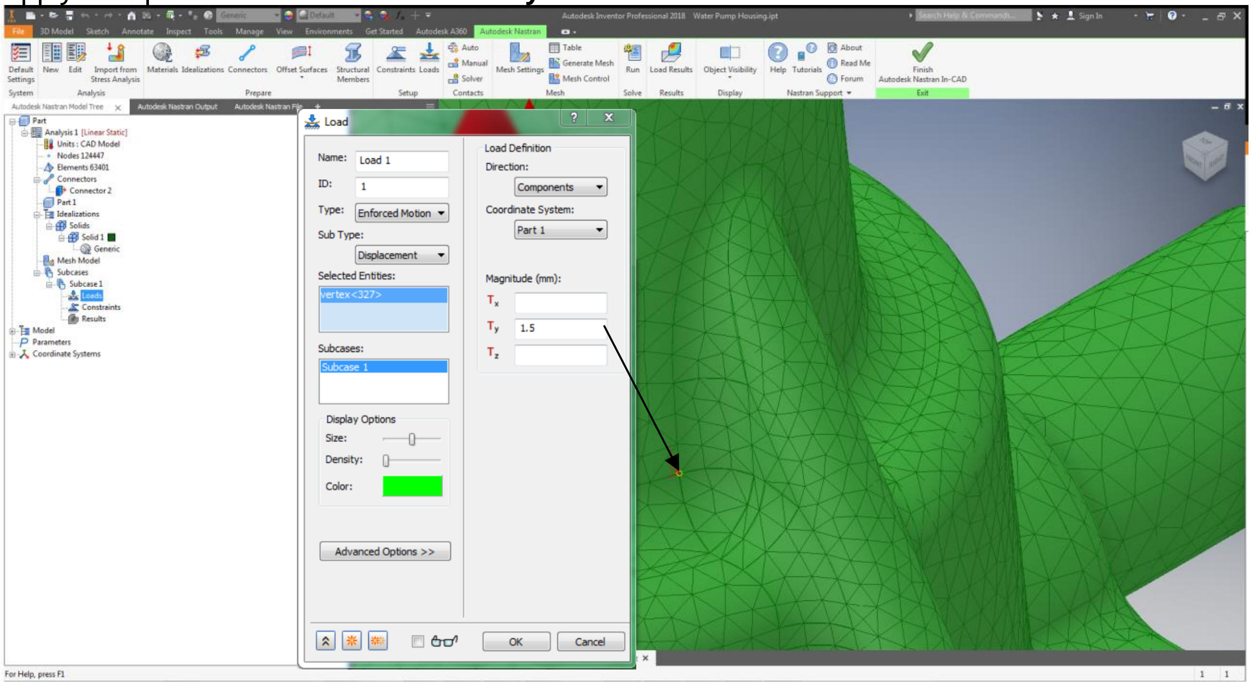
To simplify the application of an applied displacement load and post processing a rigid body tie constraint needs to be applied to the outer face of the surface shown below and a master vertex. This will allow the SPC forces for all the dependent nodes to be summed at the independent node. Select the **Connectors** icon, then select **Rigid Body** from the **Type** pull down. Select the face shown as the **Dependent Entities** and the corner node shown as the **Independent Vertex/Point**. Uncheck all degrees of freedom except **TY**.



TIED VERTICES/NODES.

Applied Displacement Load

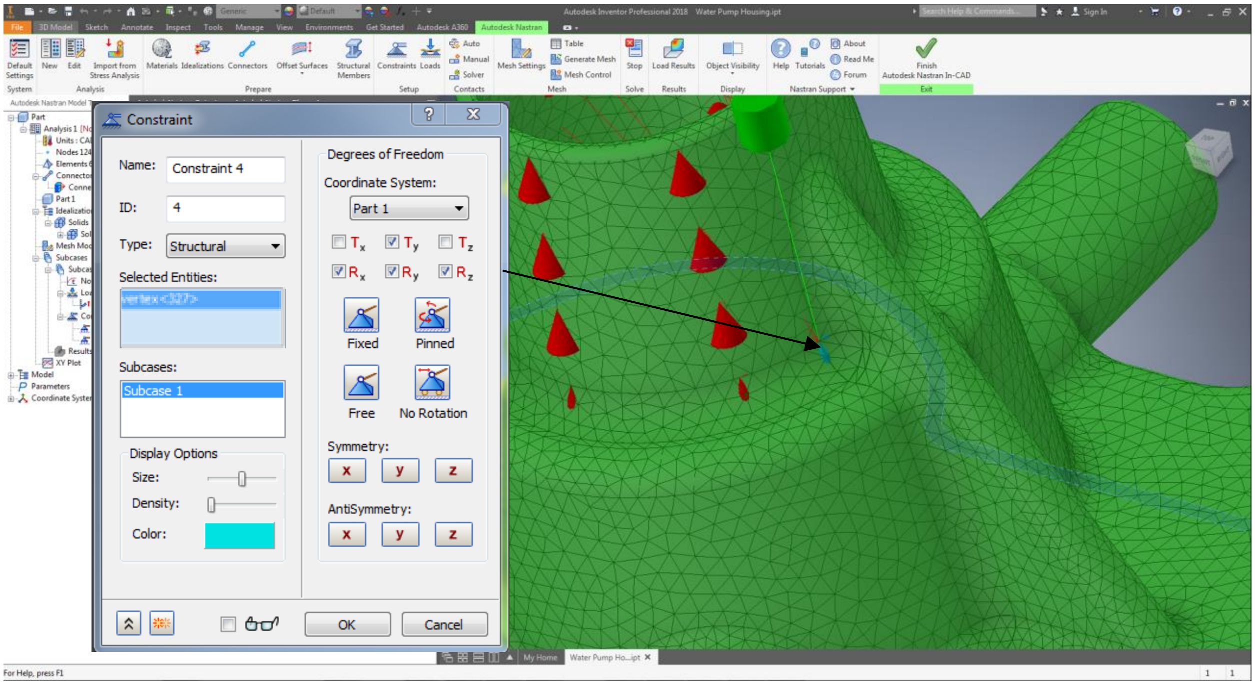
Click on the **Loads** icon and select **Enforced Motion** from the **Type** pulldown. Select the same node selected as the **Independent Vertex/Point** in the previous step as the **Selected Entities**. Apply a displacement of 1.5 mm in the **Ty** field.



DISPLACEMENT APPLIED TO MASTER VERTEX OF RIGID BODY CONSTRAINT.

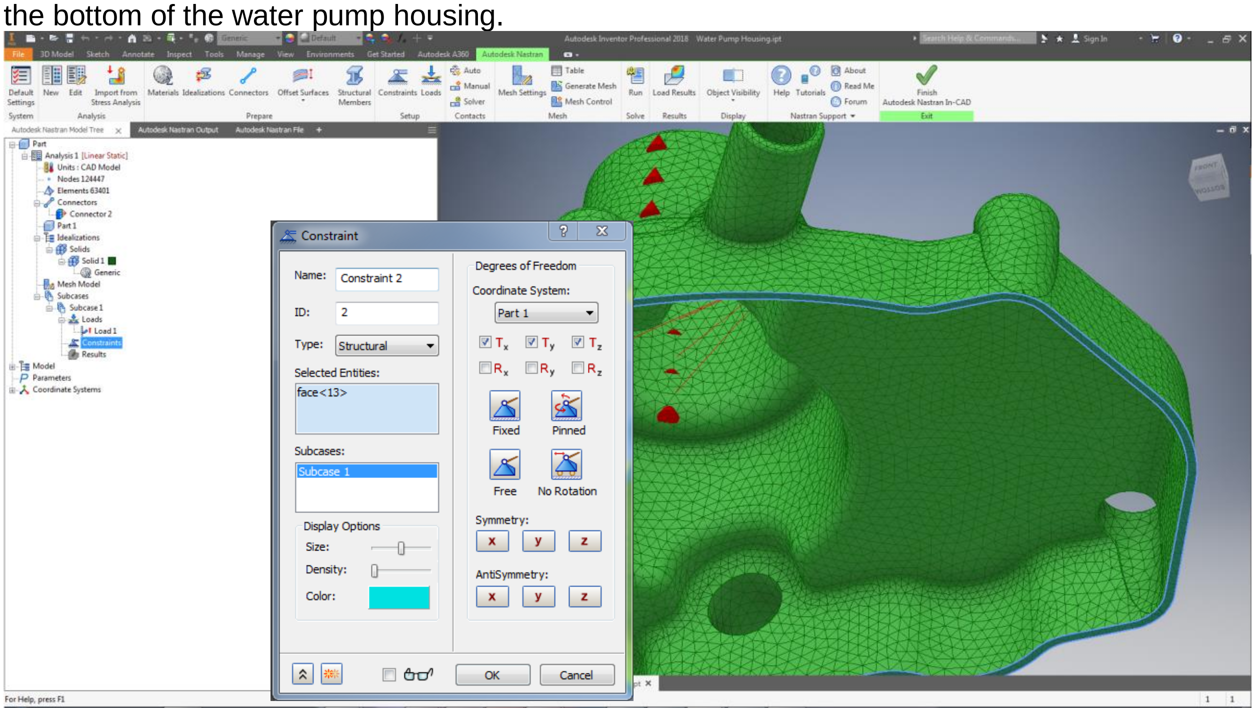
Constraints

When an enforced motion load is applied in Nastran, it is required that there also be an accompanying constraint applied on the same entity. In this case select the **Constraints** icon. Select the same node to which to the enforced motion was applied as the **Selected Entities**. Uncheck **Tx** and **Tz**.



CONSTRAINT APPLIED TO MASTER VERTEX OF RIGID BODY CONSTRAINT.

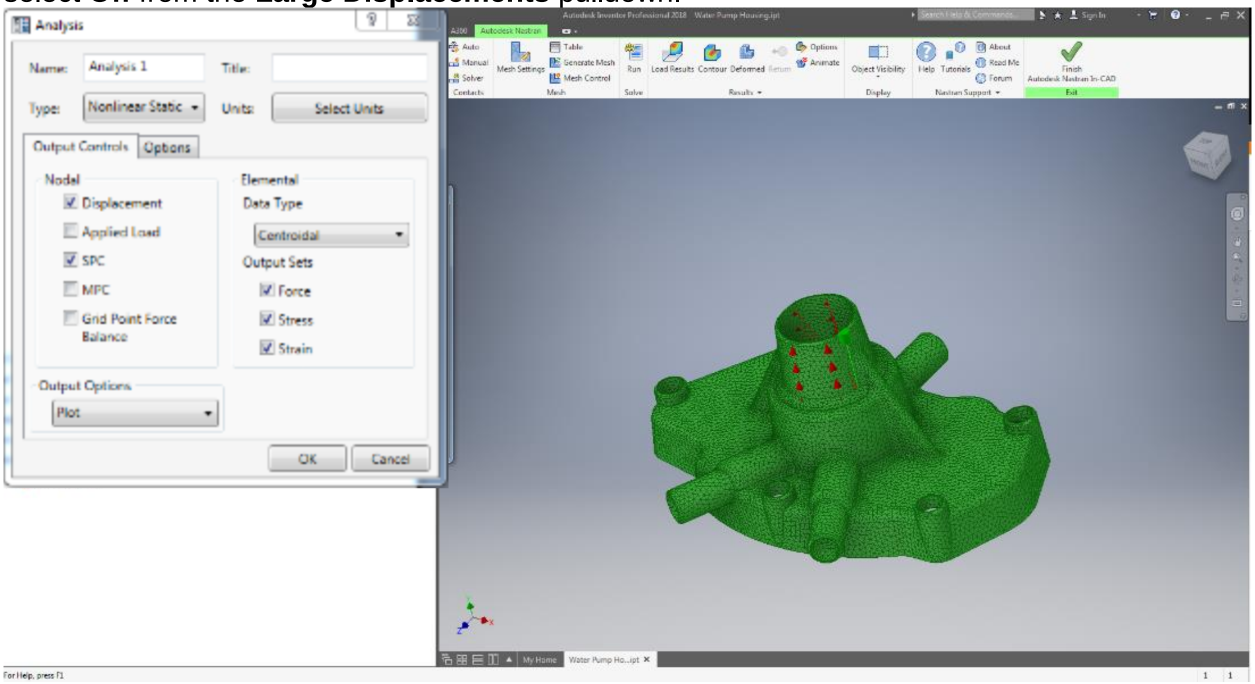
Select the **Constraints** icon again. Select the bottom face as shown below as the **Selected Entities**. Uncheck **Rx**, **Ry**, and **Rz**. This will constrain all translational degrees of freedom on the bottom of the water pump housing.



CONSTRAINTS APPLIED TO BOTTOM FACE.

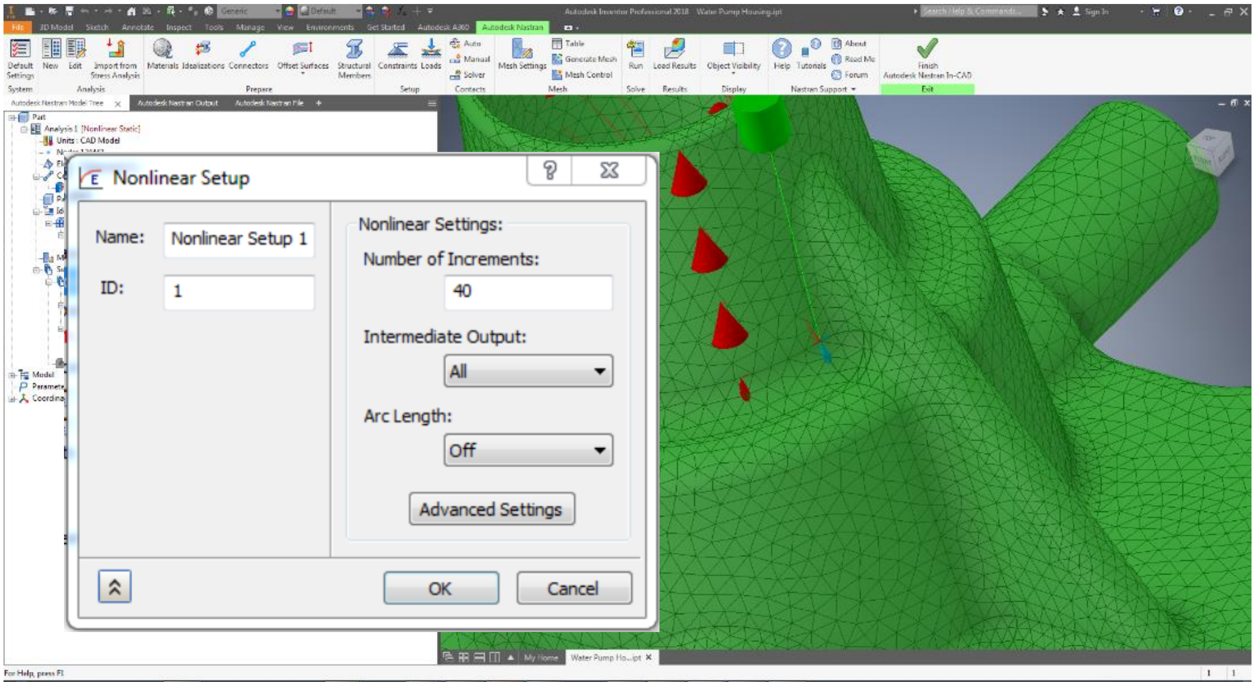
Analysis Settings

For the progressive failure analysis, we require that the solution type be Nonlinear Static. The default in In-CAD is Linear Static. To change this right click on Analysis 1 in the **Autodesk Nastran Model Tree** and select **Nonlinear Static** from the **Type** pulldown. Select **Centroidal** from the **Data Type** pulldown. Check the **Strain** radio button. Click on the **Options** tab and select **Off** from the **Large Displacements** pulldown.



NONLINEAR ANALYSIS SETTINGS.

After the analysis type has been changed to Nonlinear Static the **Nonlinear Setup 1** option will appear under **Subcase 1** in the **Autodesk Nastran Model Tree**. Right click **Nonlinear Setup 1** and select **Edit**. Enter **40** in the **Number of Increments** field. Select **All** from the **Intermediate Output** pulldown. Saving intermediate outputs to the results file allows for the identification of the failure event of interest during a progressive failure analysis.

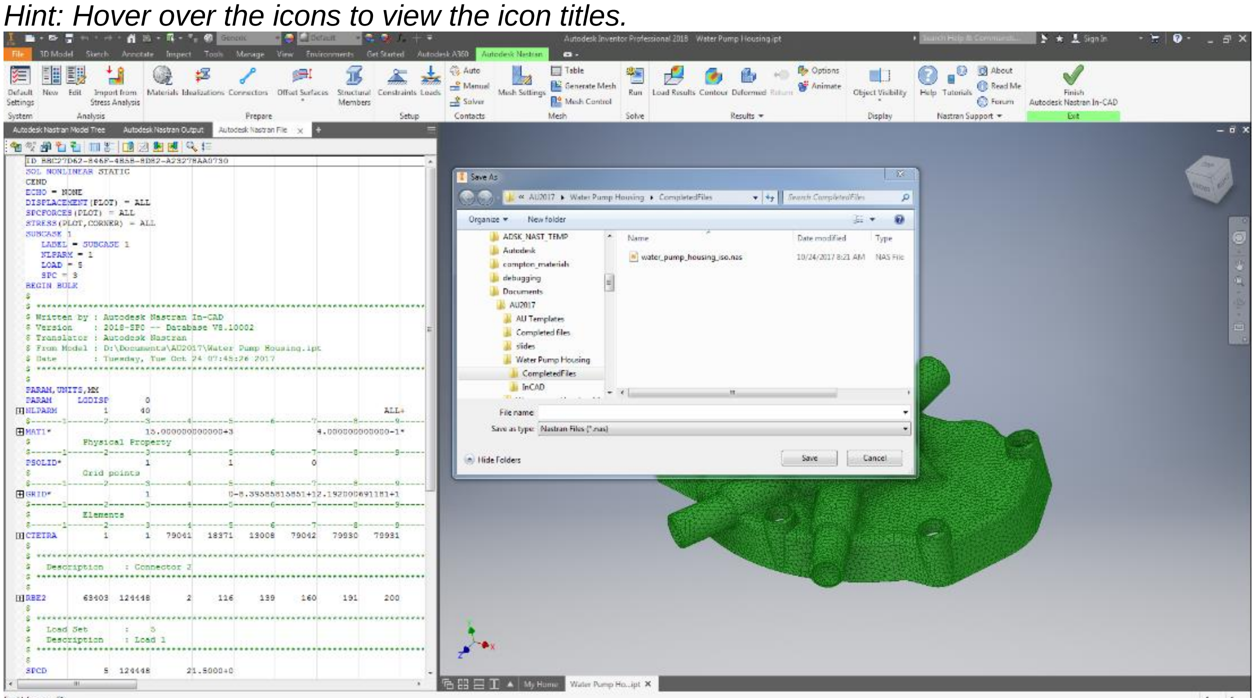


NONLINEAR SETUP.

At this point the In-CAD model is ready for export to AME for mapping Moldflow results.

Exporting structural input file

Click on the **Autodesk Nastran File** tab and select the **Update Nastran File** icon. This will write the Nastran structural input file. Now save the file by clicking on the **Save Nastran File** icon.
Hint: Hover over the icons to view the icon titles.



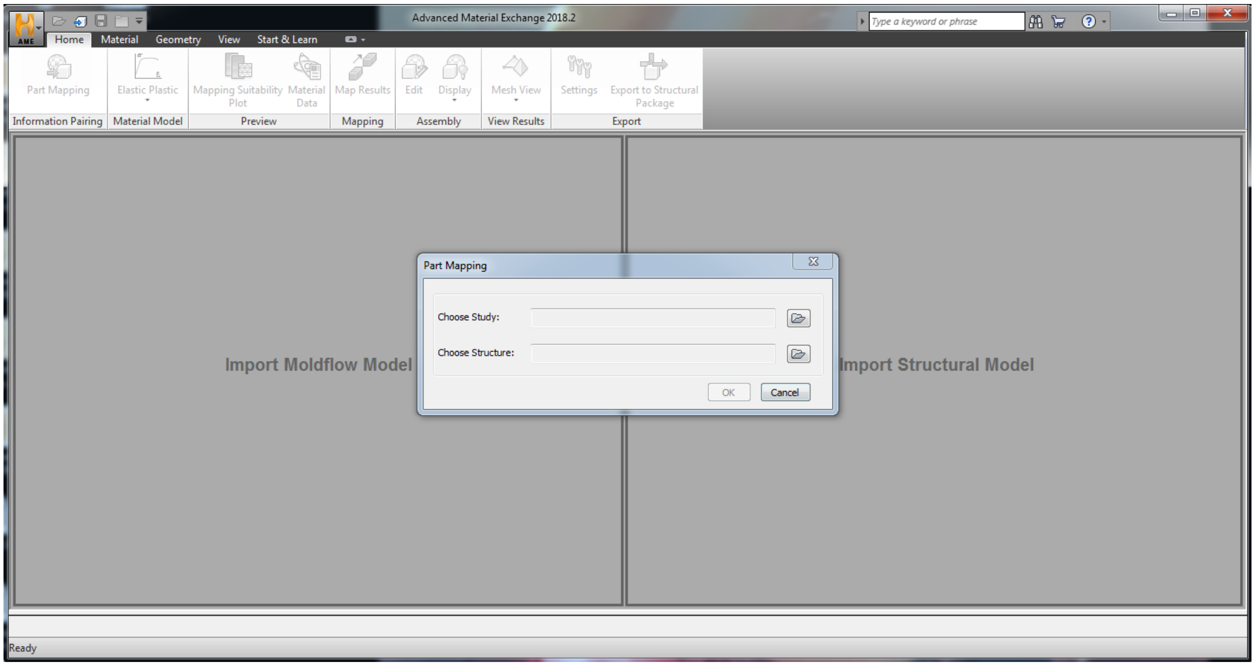
EXPORTING THE NASTRAN STRUCTURAL FILE.

AME

The AME utility is a component included with the Helius PFA installation. It is used to map injection molding results from a Moldflow mesh to a structural mesh. The results of this mapping are stored in the ".sif" file and exported with the augmented structural input file.

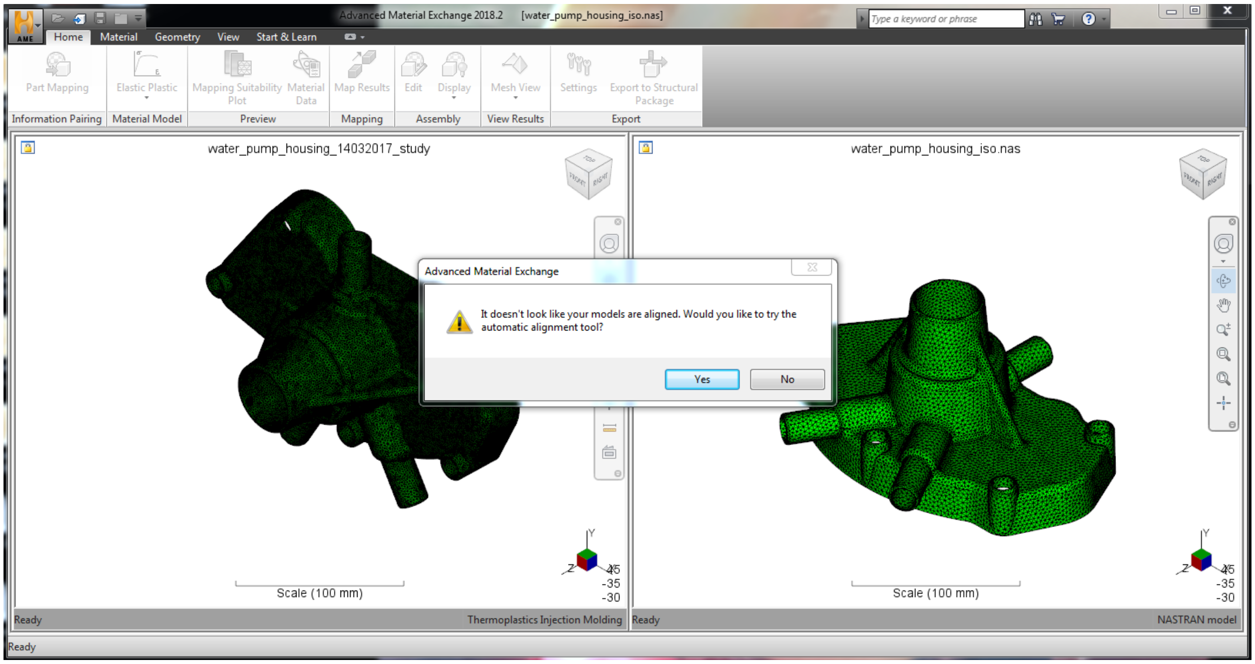
Importing Moldflow and Nastran files into AME

To begin the import process, select the **Part Mapping** icon in the AME UI. In the dialog that pops out, navigate to the Moldflow study previously run using the folder icon next to the **Choose Study** field. Navigate to the structural input file exported from In-CAD using the folder icon next to the **Choose Structure** field.



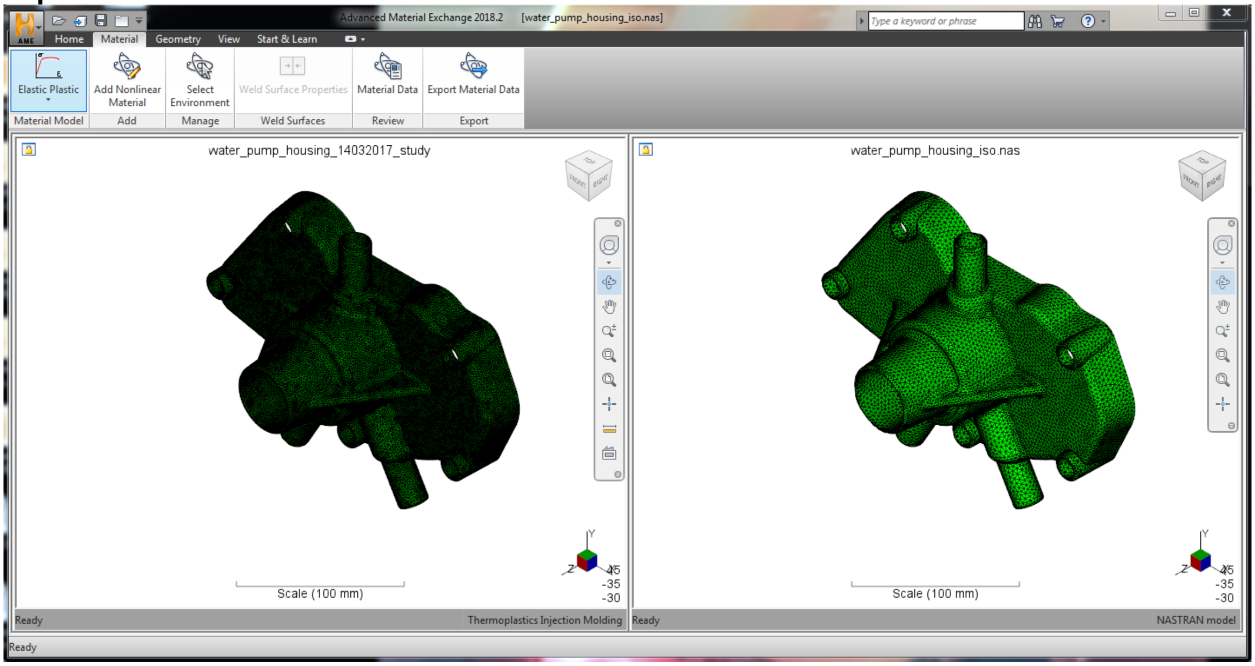
AME PART MAPPING DIALOG.

Once the Moldflow and In-CAD models are imported a dialog will pop up asking if you would like to use the automatic alignment tool. Accept this option by hitting **Yes**. *Hint: You can manually align the models by selecting the **Geometry** tab and clicking on the **Interactive Alignment** icon.*



AME PART MAPPING AUTOMATIC ALIGNMENT.

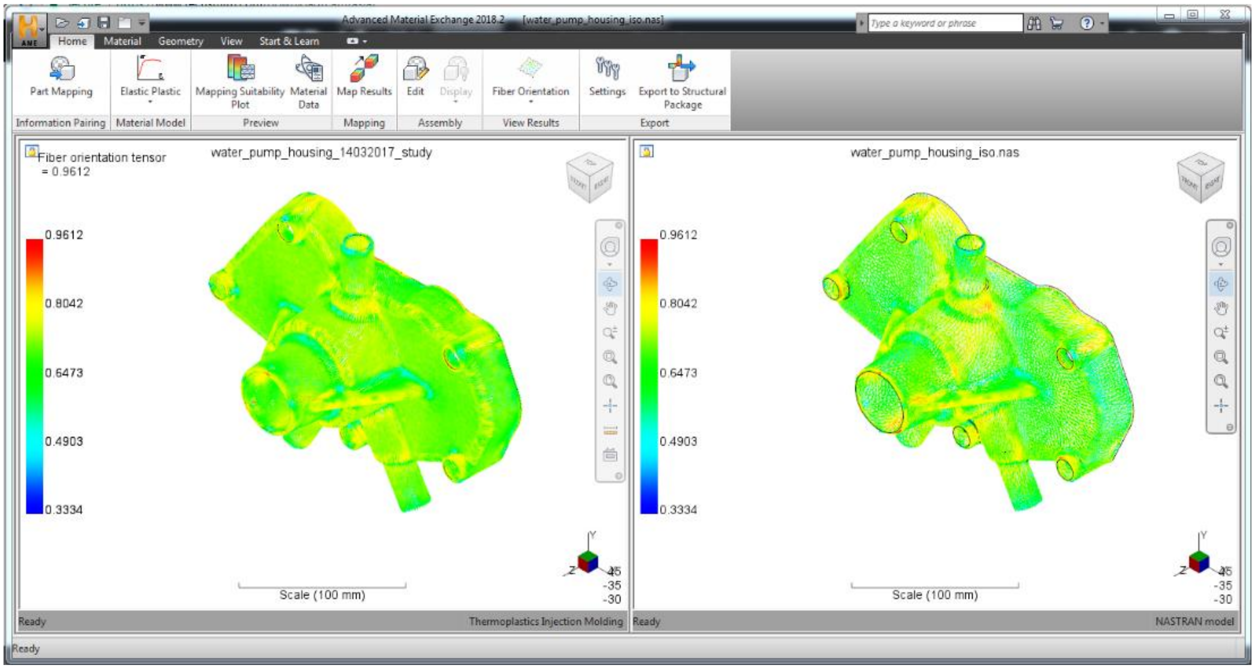
AME will align rotate and translate the structural mesh to align the parts as shown below. **Note that the actual structural mesh will not be affected! It will remain identical to what was exported from In-CAD.**



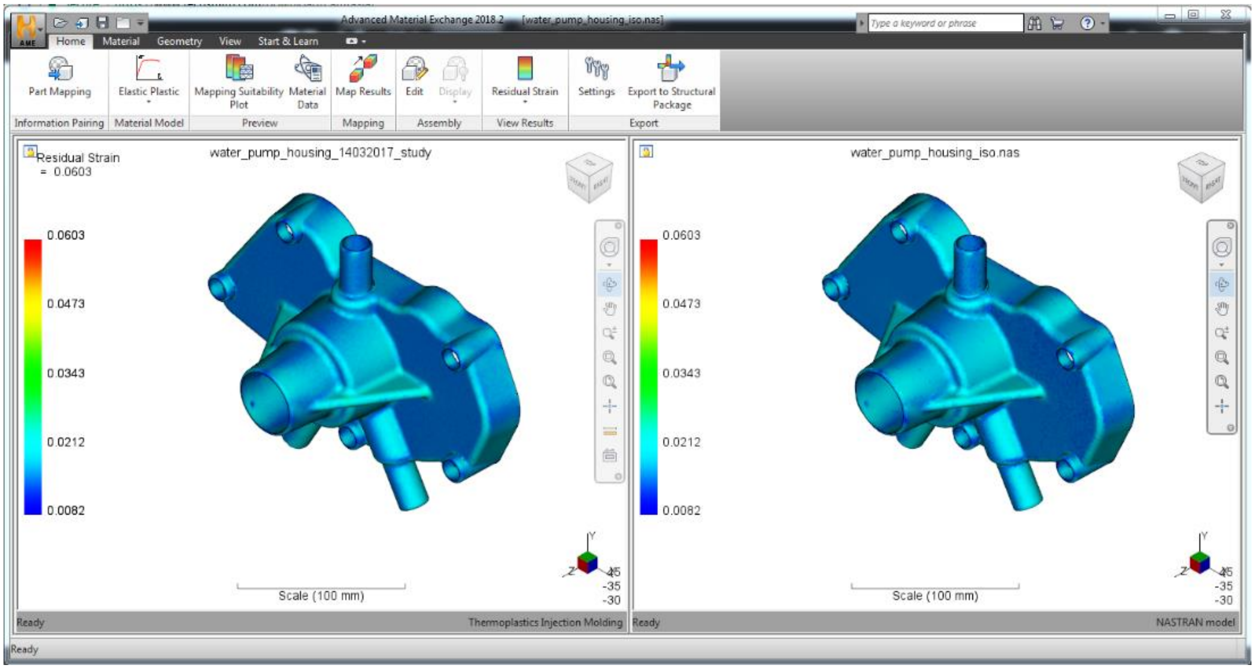
AME IMPORTED AND ALIGNED MOLDFLOW AND NASTRAN FILES.

Mapping Results

Click on the **Map Results** icon. After the mapping is complete **Fiber Orientation** will be the default plot type as shown below. To change the plot type, click on the small down arrow on the **Fiber Orientation** icon. The mapped fiber orientations and residual strains are shown below.



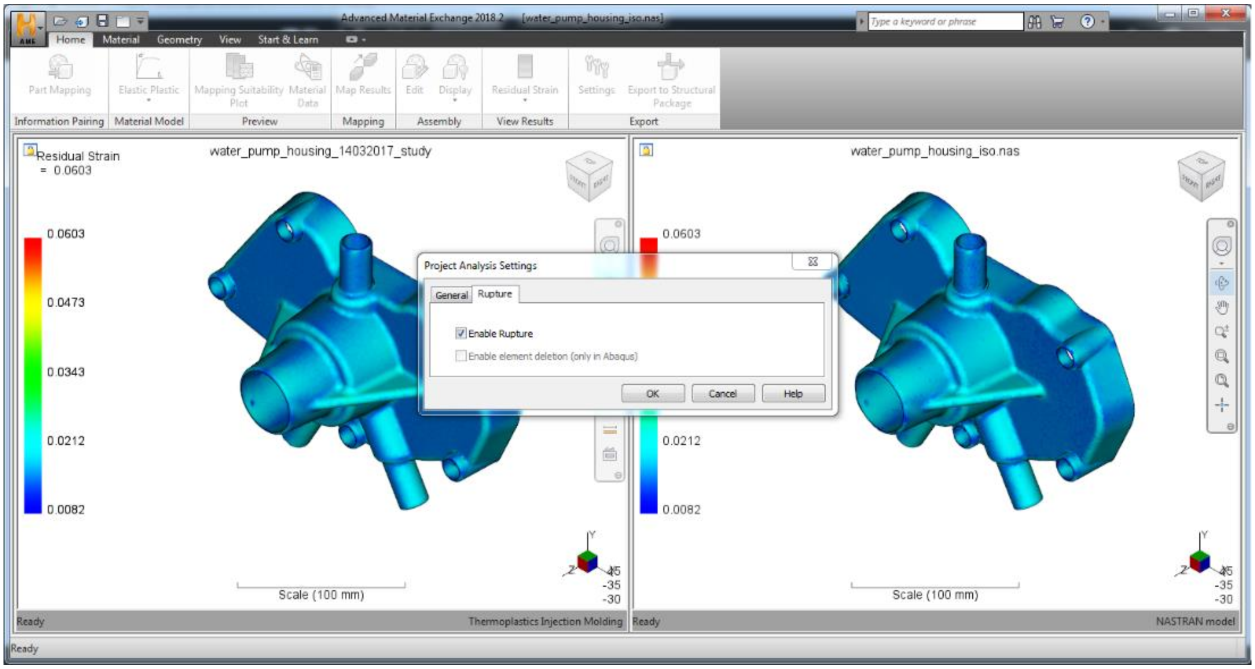
MAPPED FIBER ORIENTATIONS.



MAPPED RESIDUAL STRAINS.

Analysis Settings

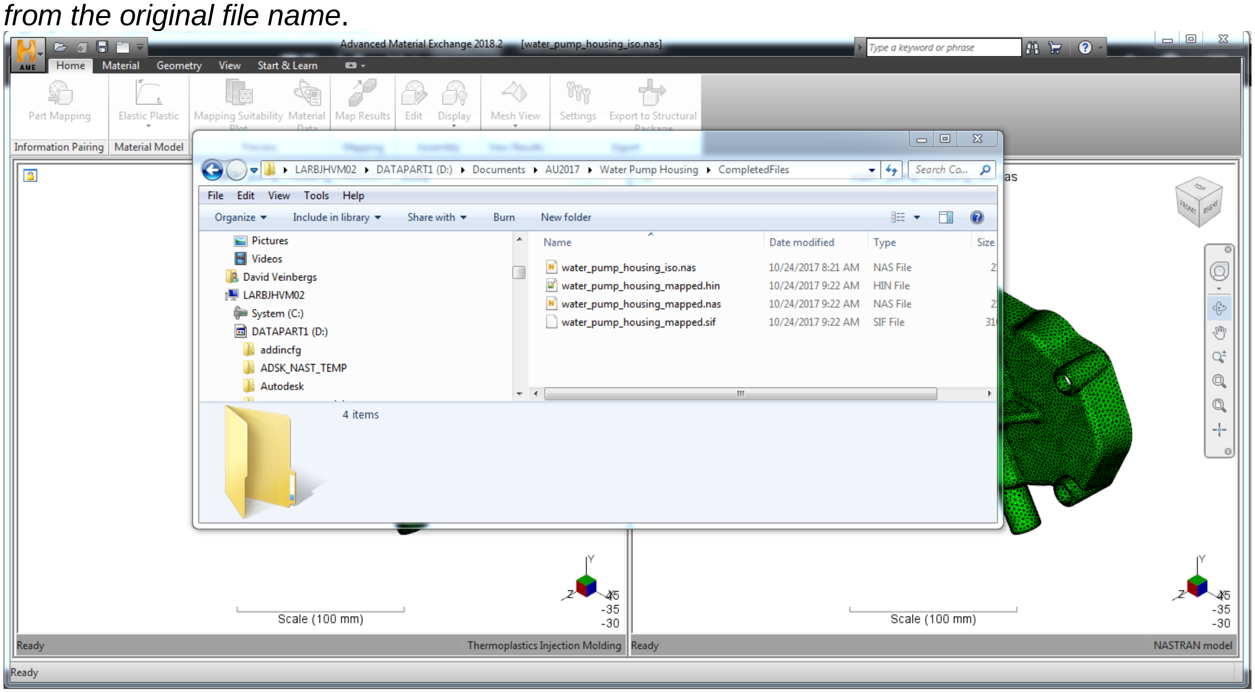
Prior to exporting the mapped structural input file several options are available through the **Settings** icon. Rupture is turned off by default in Helius PFA. To activate rupture for this analysis, select the **Settings** icon, click on the Rupture tab and click the **Enable Rupture** radio button, then click **OK**.



ENABLE RUPTURE.

Exporting Mapped Results

Once the mapping is complete and the settings required for the analysis are selected it is time to export the mapped files to be imported back into In-CAD. To export the files from AME select the **Export to Structural Package** icon. Save the file to a convenient location. Three files should be exported from AME. An augmented version of the original structural input file (.nas), the Structural Interface File (.sif), and the Helius INput file (.hin) will all be exported with the selected file name. *Hint: It is safest to rename the mapped files something to differentiates them from the original file name*.

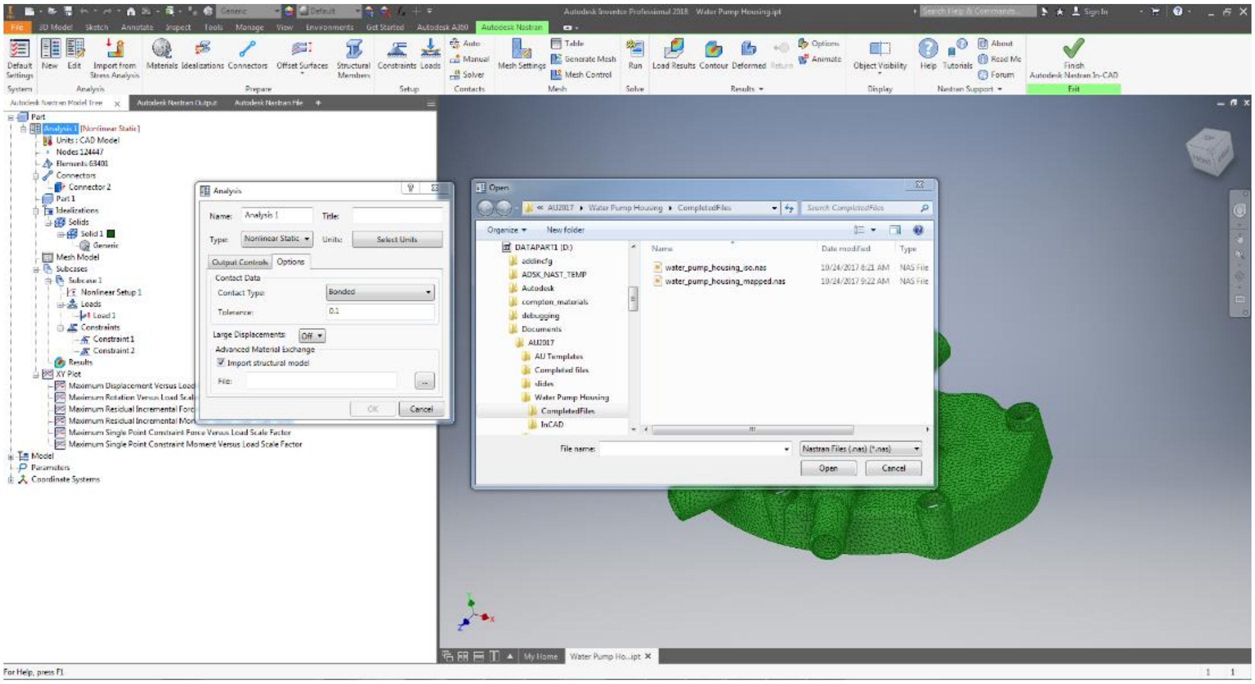


AME EXPORTED FILES.

Progressive Failure Analysis in Nastran In-CAD with Helius PFA

Importing Mapped Structural Analysis Files

Once the files have been exported from AME the mapped version can be imported into In-CAD. Right click on the **Analysis 1** tree item under the **Autodesk Nastran Model Tree**. Select **Edit** and click on the **Options** tab. Under the **Advanced Material Exchange** portion of the dialog check the radio button titled **Import structural model**. Select the .nas file that was exported from AME. Click **OK**.



In-CAD import dialog.

Running the Analysis

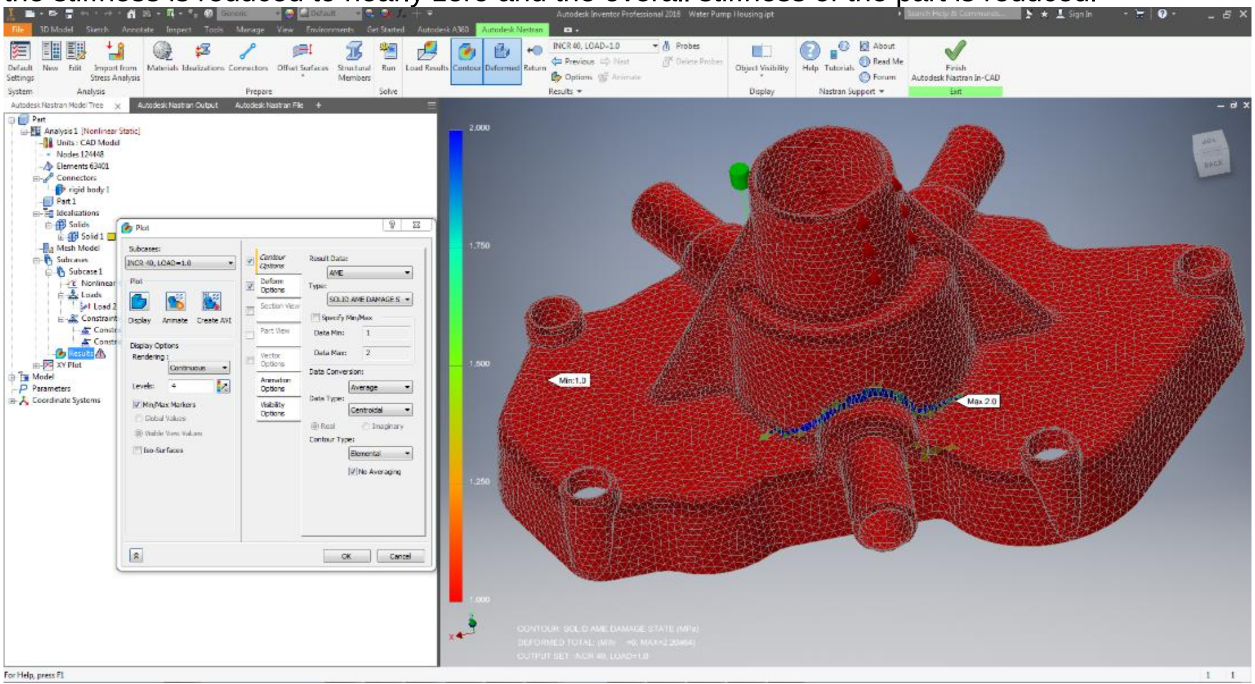
To run the structural analysis simply click on the **Run** icon in In-CAD. A dialog should appear indicating that the Nastran file is being generating and the analysis should launch as usual.

Helius PFA Results

Once the analysis is completed a number of additional AME/Helius PFA specific results will be available for viewing. To access the AME results click the **Load Results** icon. To view the results double click on **Results** in the **Autodesk Nastran Model Tree**. Select **AME** from the **Result Type** pulldown. Under the **Type** pulldown the following results types will be available:

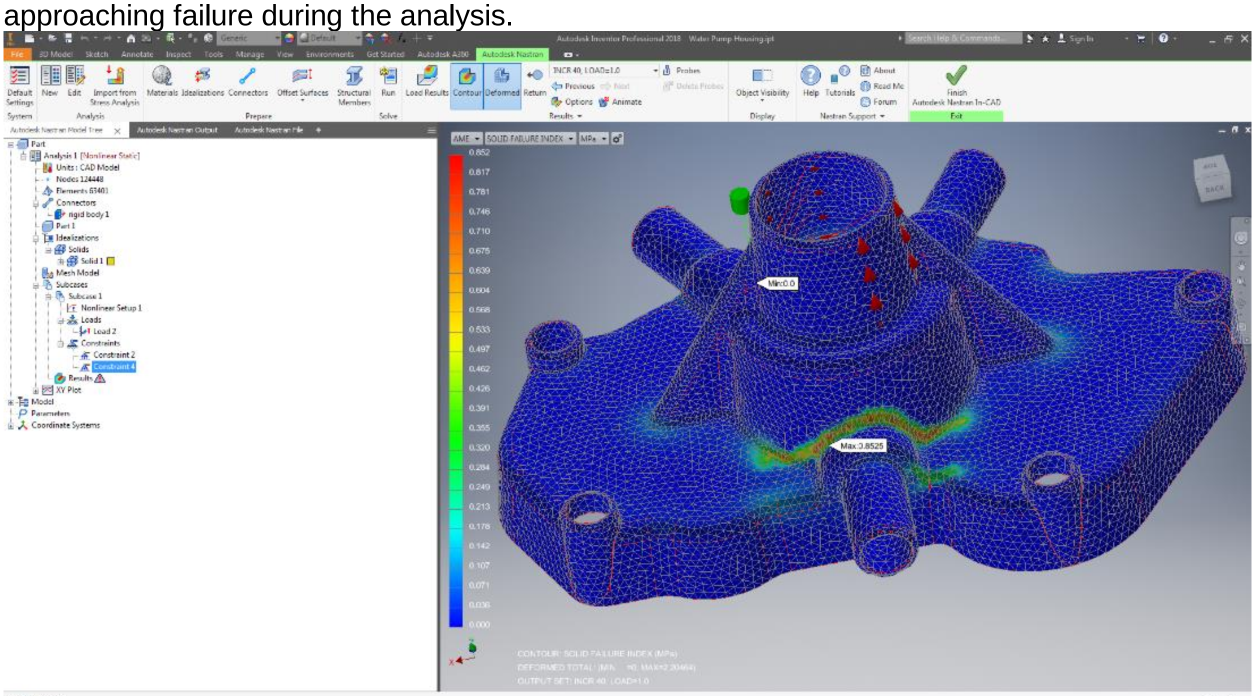
- **SOLID MATRIX TANGENT MODULUS**
- **SOLID MATRIX EFFECTIVE PLASTIC STRAIN**
- **SOLID MATRIX EFFICTIVE STRESS**
- **SOLID AME DAMAGE STATE**
- **SOLID WELD SURFACE STRENGTH FACTOR**
- **SOLID FAILURE INDEX**
- **SOLID FAILURE MODE**

In this progressive failure analysis, the SOLID AME DAMAGE STATE and SOLID FAILURE INDEX are of interest. The damage state is an element level result that indicates if the element has experienced failure or not. A value of 2 indicates complete failure and a value of 1 indicates that the material is pristine. In the below plot we see that all the elements in red are pristine and the small area in blue has experienced failure. Once the failure point is reached at an element the stiffness is reduced to nearly zero and the overall stiffness of the part is reduced.



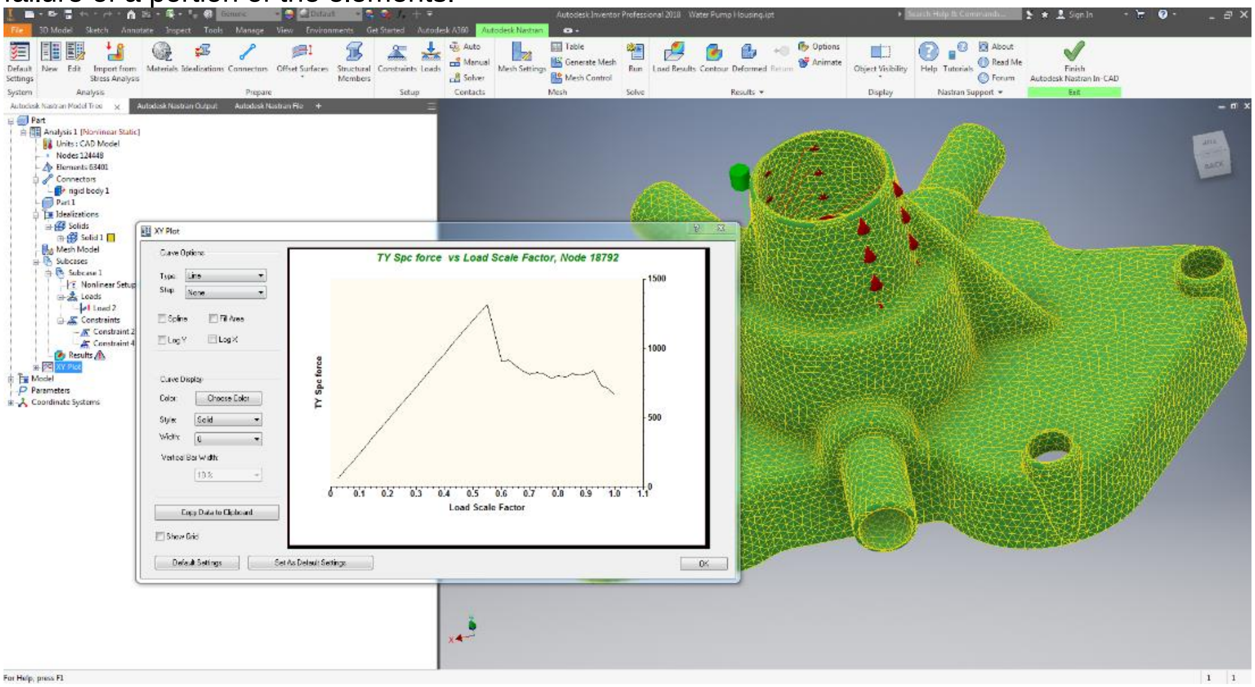
HELIUS PFA DAMAGE STATE.

In the plot below the failure index is shown. Unlike the damage state the failure index can take on any value between 0 and 1. This is helpful for determining which areas of the structure are approaching failure during the analysis.



HELIUS PFA FAILURE INDEX.

As an illustration of progressive failure experienced by the part and the consequences of the corresponding reduction in stiffness we can plot the SPC FY force at the independent node of the rigid body tie constraint we defined earlier. The SPC forces at all the nodes on the dependent faces will be summed up at this independent node. Plotting this SPC forces from increments 1 to 40 shows the reduction in the overall stiffness of the structure as elements begin to fail. As shown in the plot below, the structure can still carry load after the complete failure of a portion of the elements.



SUMMED TY SPC FORCE ON LOAD FACE.

Additional resources

Additional reading for workflow

- Required model files for demo are available here
- Helius In-CAD tutorials: http://help.autodesk.com/view/ACMPAN/2018/ENU/?guid=GUID-AB99D5FB-8A71-409C-BA39-6D07BCF2C9C5
- AME User's Guide: http://help.autodesk.com/view/ACMPAN/2018/ENU/?guid=GUID-CED5AF9F-EF61-4890-9253-F85DACA3B4A0
- Nastran In-CAD User's Guide: http://help.autodesk.com/view/NINCAD/2018/ENU/
- MATXM: http://help.autodesk.com/view/NSTRN/2018/ENU/?guid=GUID-2D9E5CDC-AF67-4AD7-BA85-007DFD476BC0
- NLPARM: http://help.autodesk.com/view/NSTRN/2018/ENU/?guid=GUID-B434F910-038E-43BD-B992-62A04EC95AFC

Theoretical references

1. Eshelby, J.D. (1957) "The Determination of the Elastic Field of an Ellipsoidal Inclusion and Related Problems," *Proceedings of the Royal Society London,* A241: 376–396.
2. Mori, T. and Tanaka, K. (1973) "Average Stress in Matrix and Average Elastic Energy of Materials with Misfitting Inclusions," *Acta Metallurgica*, 21: 571–574.
3. Benveniste, Y. (1987) "A New Approach to the Application of Mori-Tanaka's Theory in Composite Materials," *Mechanics of Materials*, 6: 147–157.
4. Nguyen, BAN, Bapanapalli, SK, Kunc, V, Phelps, JH and Charles L. Tucker, CL, (2009) "Prediction of the Elastic_Plastic Stress/Strain Response for Injection-Molded Long-Fiber Thermoplastics," *Journal of Composite Materials*, 43; 217.
5. Nguyen, BAN, and Kunc, V, (2009) "An Elastic-plastic Damage Model for Long-fiber Thermoplastics," *International Journal of Damage Mechanics*, (2009), published online doi:10.1177/1056789509338319.