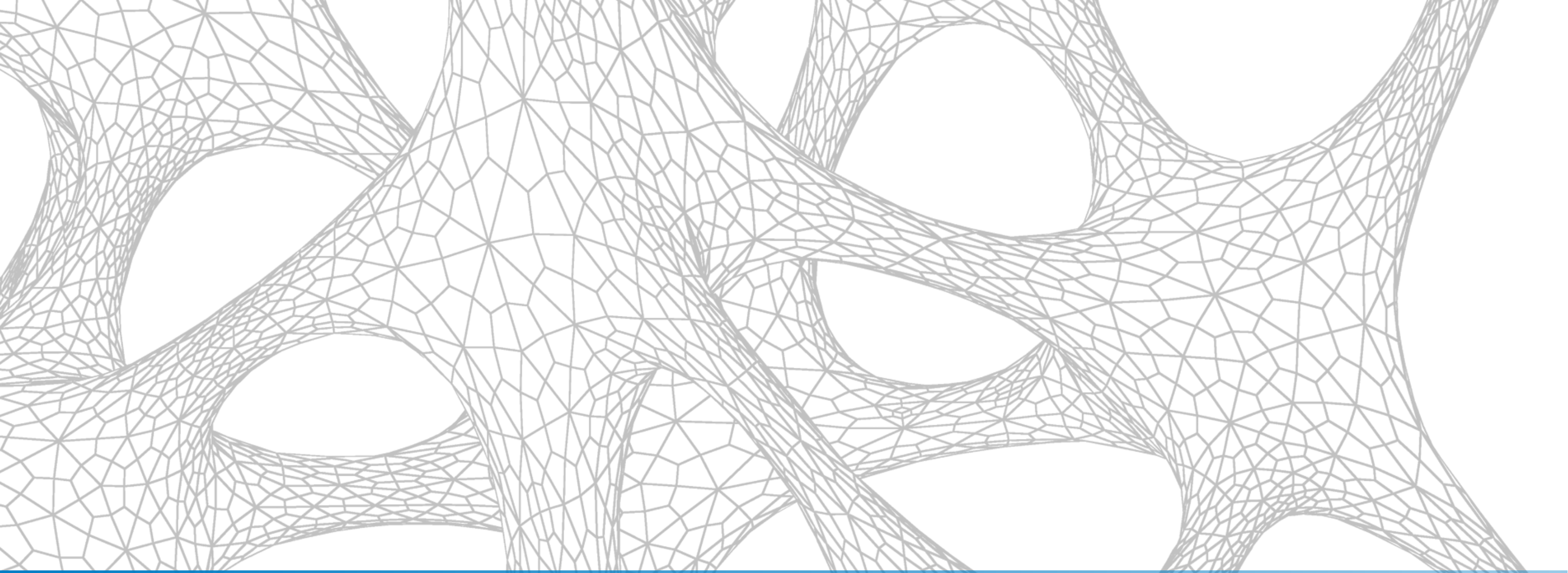
All for Composite Simulation as Manufactured: Autodesk NASTRAN, Helius PFA and Moldflow ecosystem

Jaesung Eom, David Veinbergs

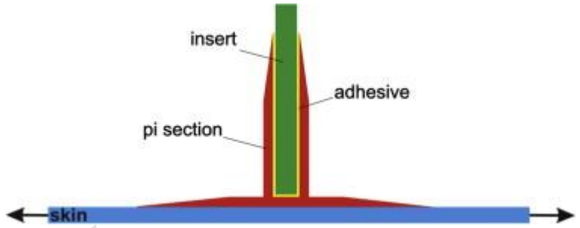
Research Engineers at Autodesk



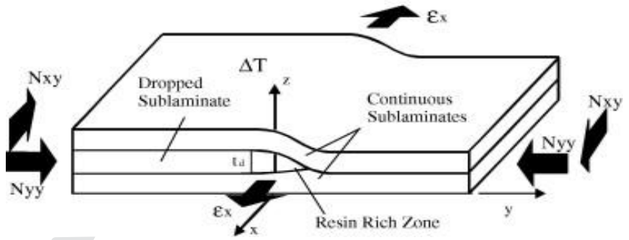
Why As-Manufactured simulation

Building a Composite Structure

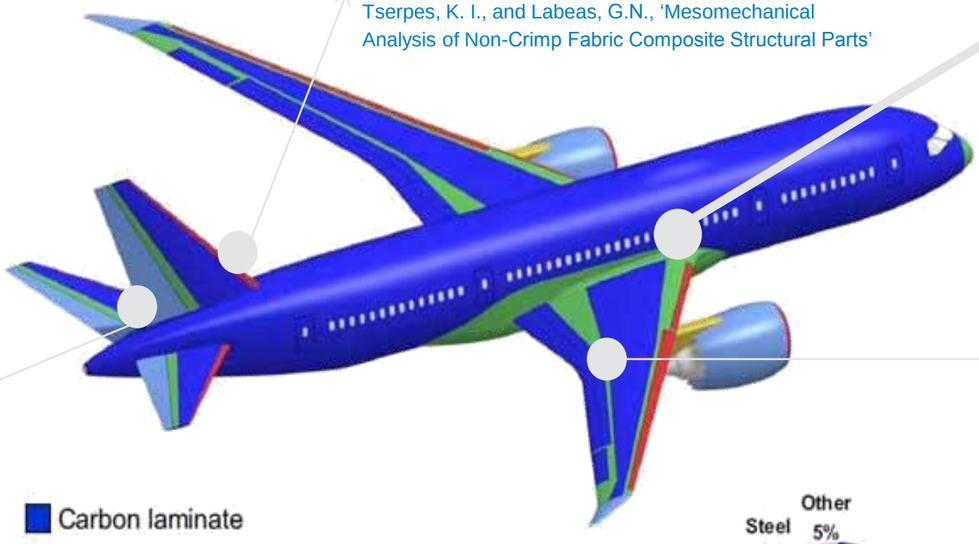
- Finally, laminates can be used to build a composite structure
  - Build ups / additional plies
  - Drop offs
  - Joints

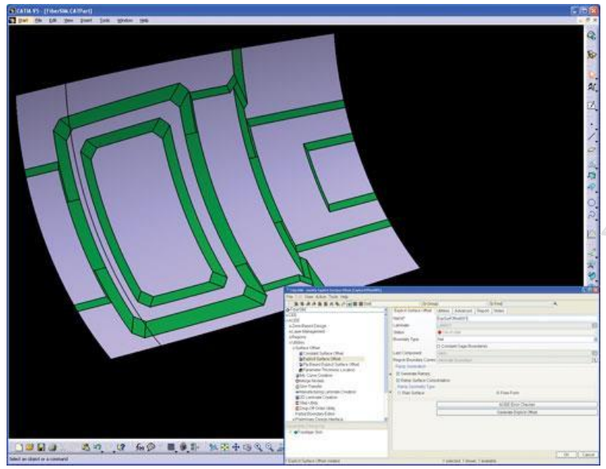


Tserpes, K. I., and Labeas, G.N., 'Mesomechanical Analysis of Non-Crimp Fabric Composite Structural Parts'

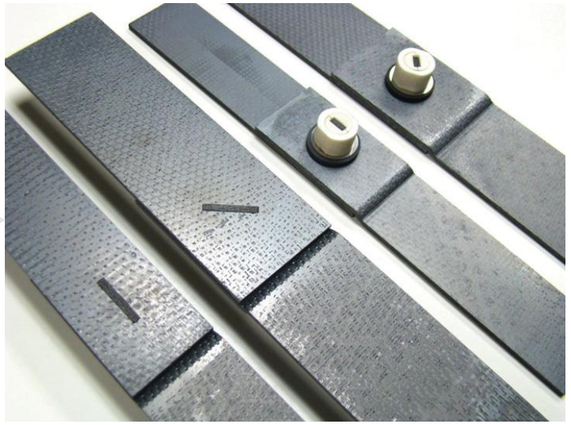


Kim, H.S, Rhee, S.Y., and Cho, M. 'Simple and Efficient Interlaminar Stress Analysis of Composite Laminates with Internal Ply-Drop'





LeGault, M. 'Aircraft Simulation Gets Composite Aware,' www.compositesworld.com

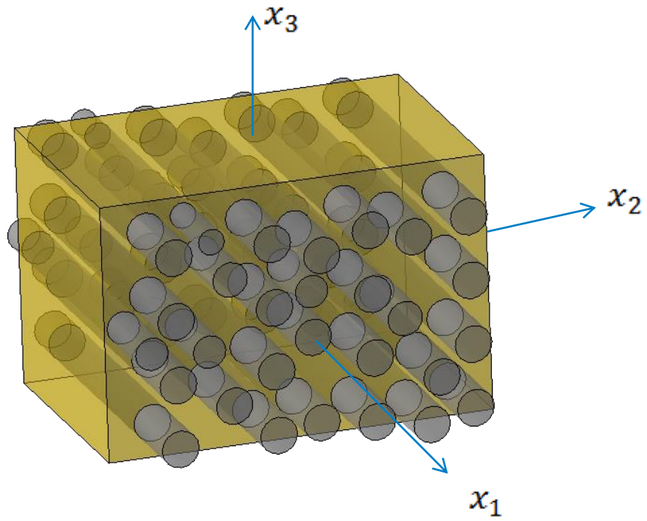


www.nasa.gov

www.1001crash.com

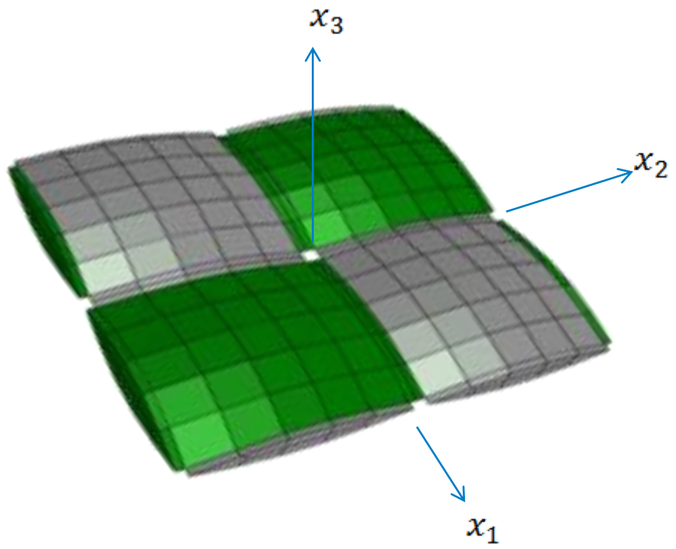
Building a Composite Structure –Lamina

- The foundational building block of a composite structure is an individual lamina (the homogenized ply)
  - Plies are typically stiffer and stronger in certain directions

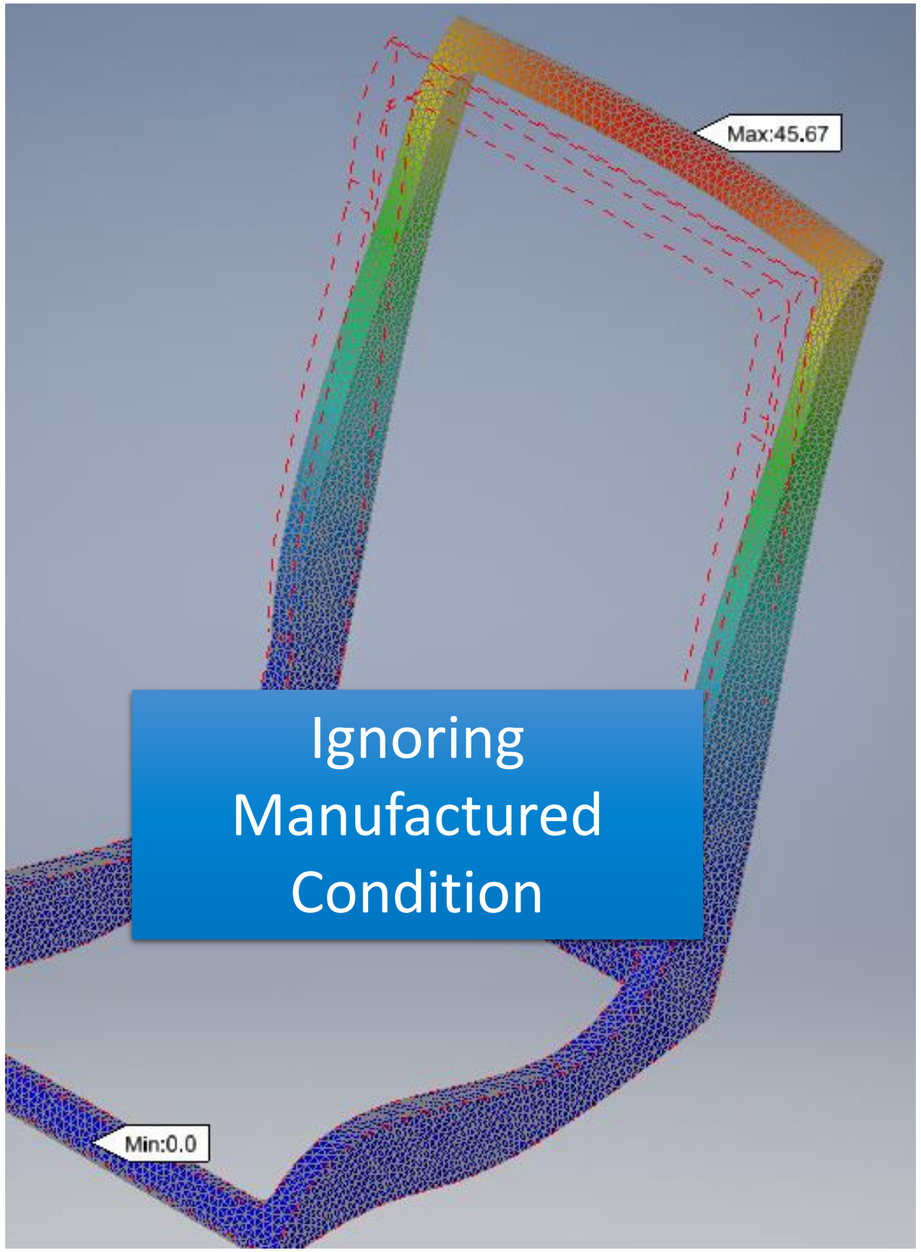


Material	AS4/ 3501-6	T300/ 5208	Kevlar/ epoxy	Boron/ Al	SCS-6/ Ti-15-3
Density, g/cm^3 (lb/in^3)	1.52 (0.055)	1.54 (0.056)	1.38 (0.05)	2.65 (0.096)	3.86 (0.14)
Axial modulus E_1, GPa (Msi)	148 (21.5)	132 (19.2)	76.8 (11.0)	227 (32.9)	221 (32)
Transverse modulus E_2, GPa (Msi)	10.50 (1.46)	10.8 (1.56)	5.5 (0.8)	139 (20.2)	145 (21)
Poisson's ratio ν_{12}	0.30	0.24	0.34	0.24	0.27
Poisson's ratio ν_{23}	0.59	0.59	0.37	0.36	0.40
Shear modulus G_{12}, GPa (Msi)	5.61 (0.81)	5.65 (0.82)	2.07 (0.3)	57.6 (8.35)	53.2 (7.78)
Shear modulus G_{23}, GPa (Msi)	3.17 (0.46)	3.38 (0.49)	1.4 (0.20)	49.1 (7.12)	51.7 (7.50)
Modulus ratio E_1/E_2	12.6	12.3	14.8	1.6	1.5
Axial tensile strength X_T, MPa (ksi)	2137 (310)	1513 (219.5)	1380 (200)	1290 (187)	1517 (220)
Transverse tensile strength Y_T, MPa (ksi)	53.4 (7.75)	43.4 (6.3)	27.6 (4.0)	117 (17)	317 (46)
Strength ratio X_T / Y_T	27	35	50	11	4.8

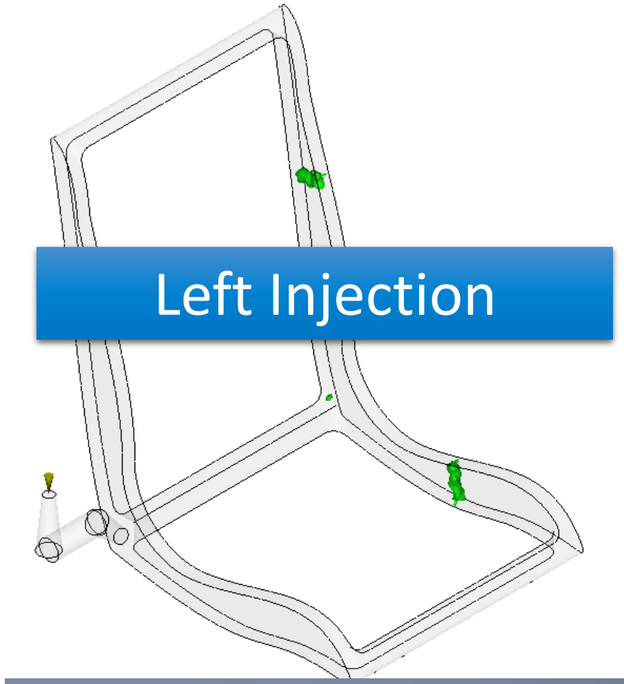
Herakovich, C. T., "Mechanics of Fibrous Composites," John Wiley & Sons, Inc., *1998*



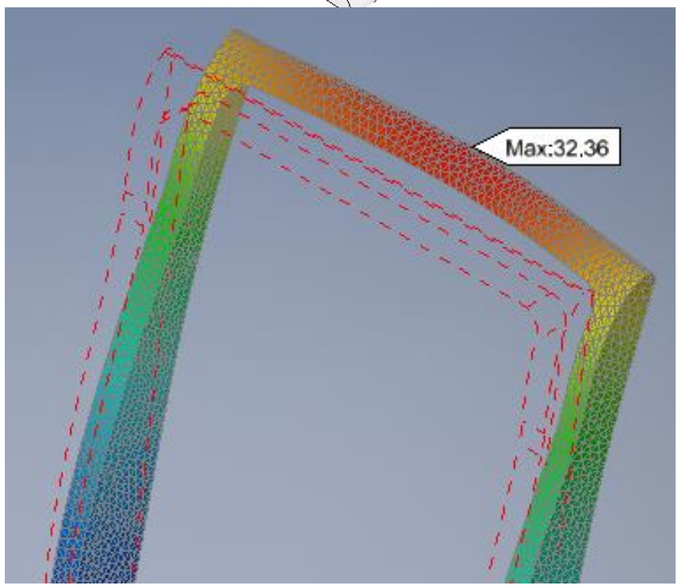
What if we simulated “As-Manufactured”

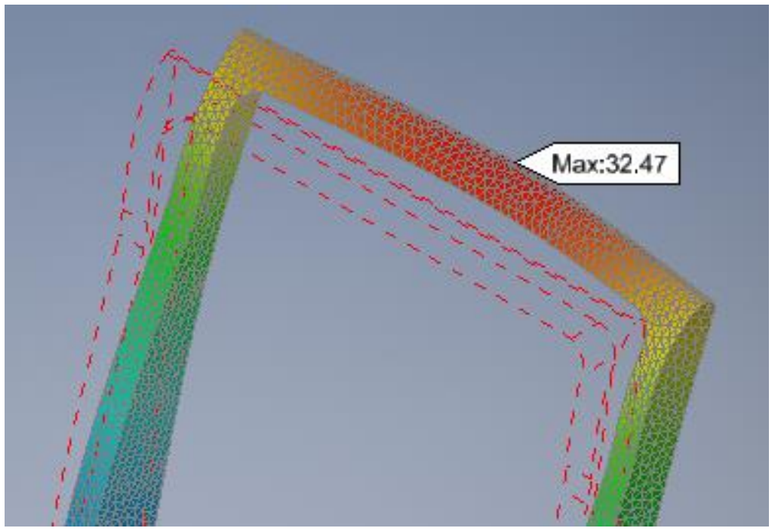


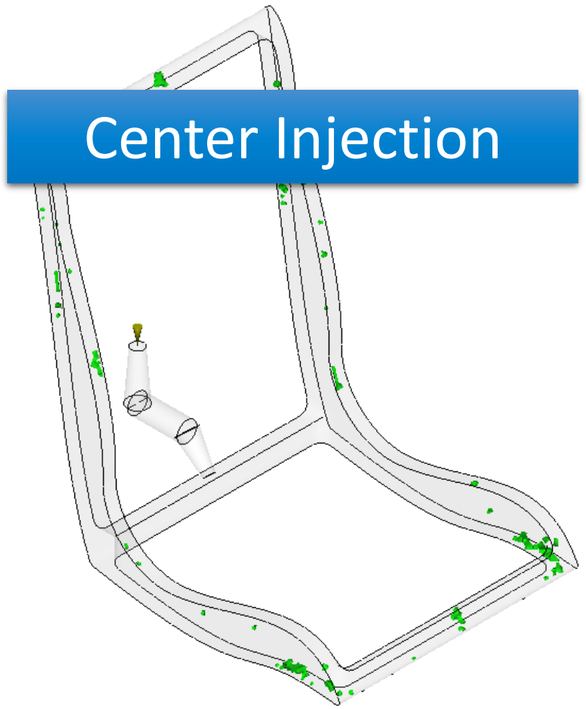
Ignoring Manufactured Condition



Left Injection



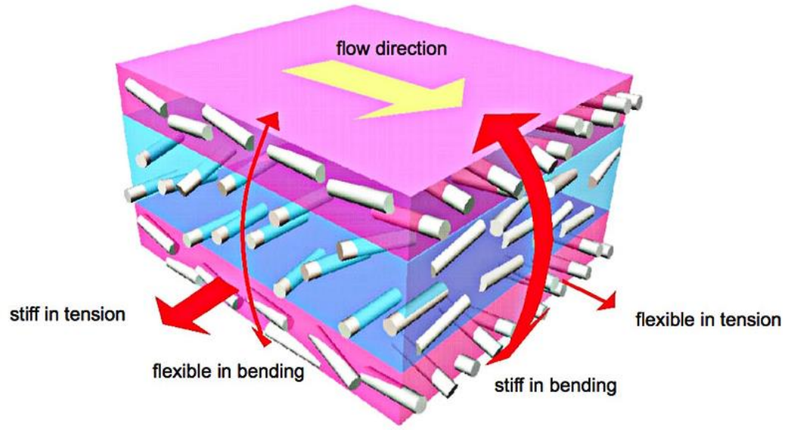


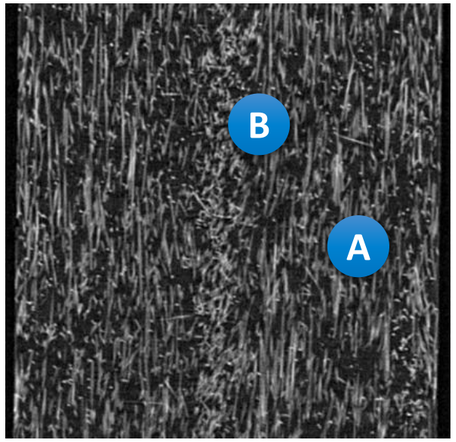


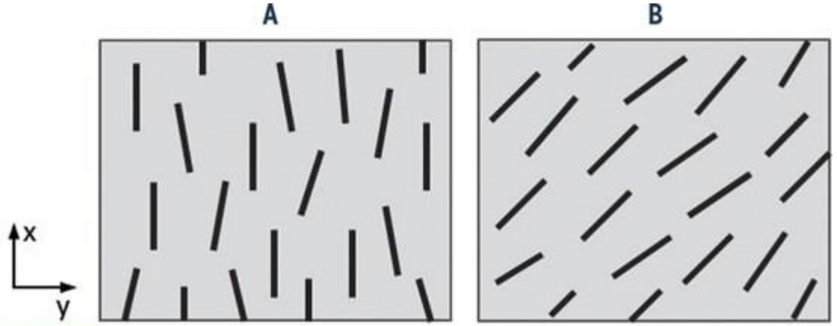
Center Injection

Fiber matters :
how it aligned tells, how it works

- Different fiber orientation = Different stiffness / strength
- Different fiber volume fraction = Different stiffness / strength

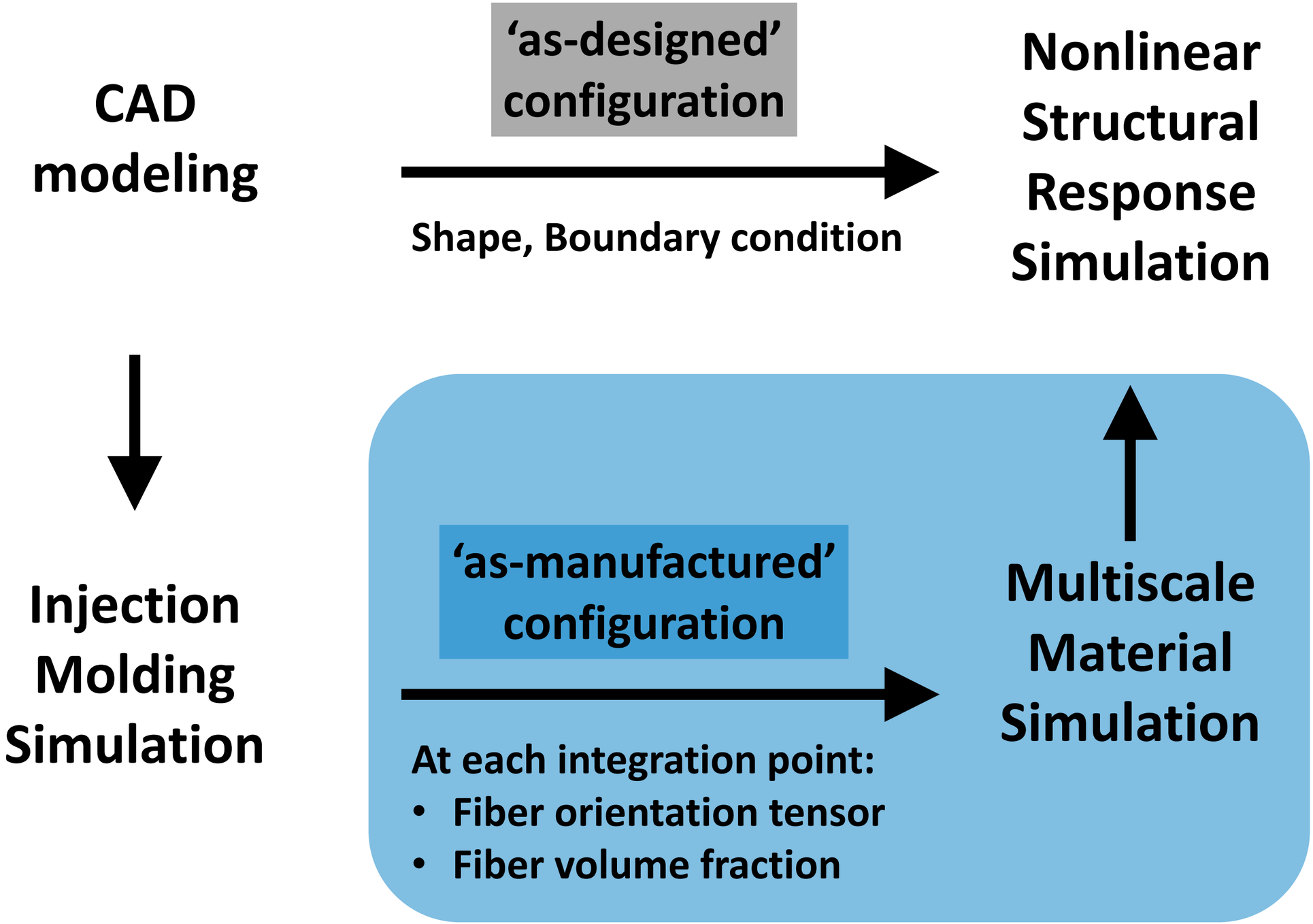


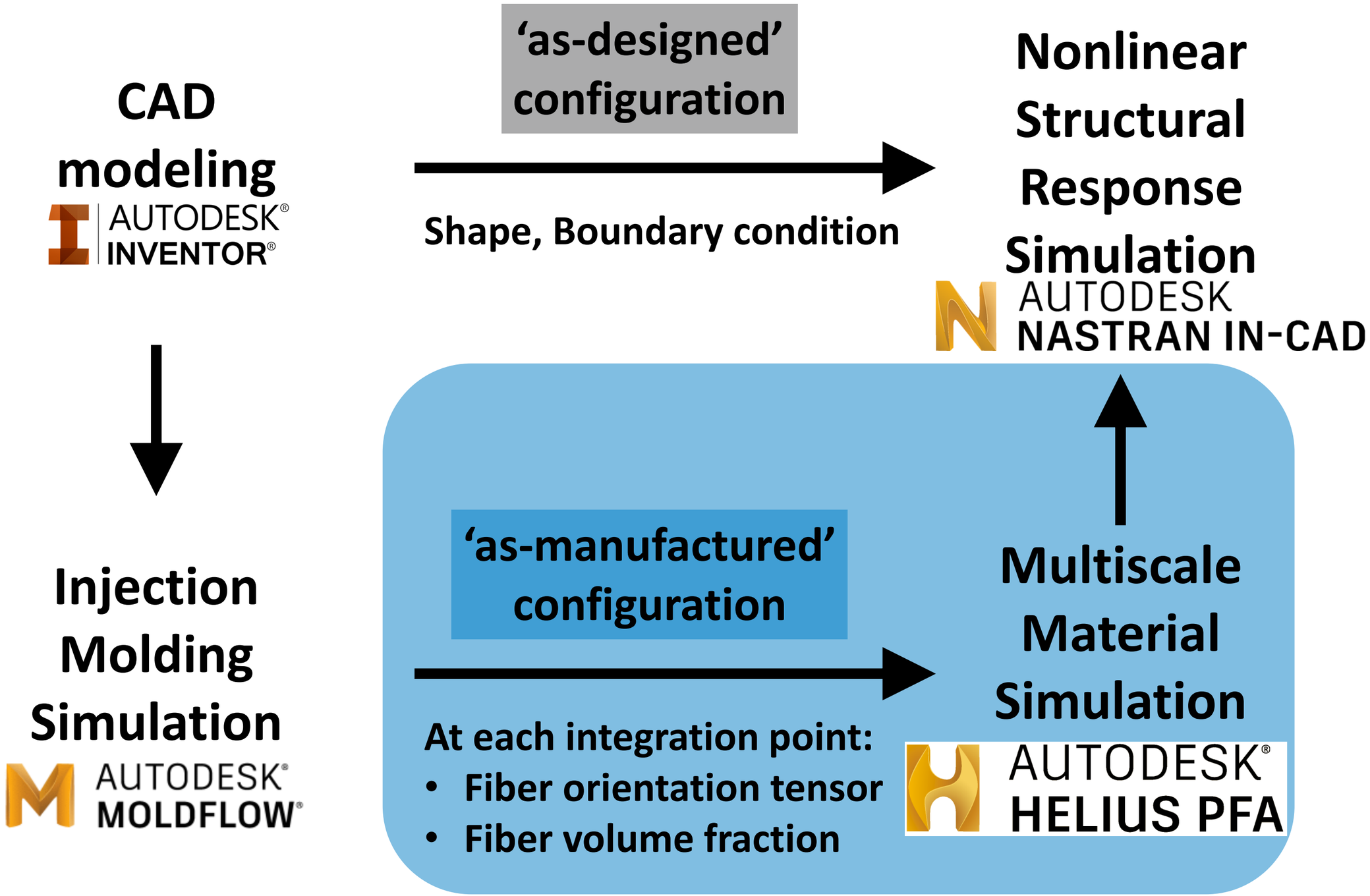




	Fibers in x direction	Fibers under 45°
E_x(MPa)	12,000	6,000
E_y(MPa)	4,000	6,000

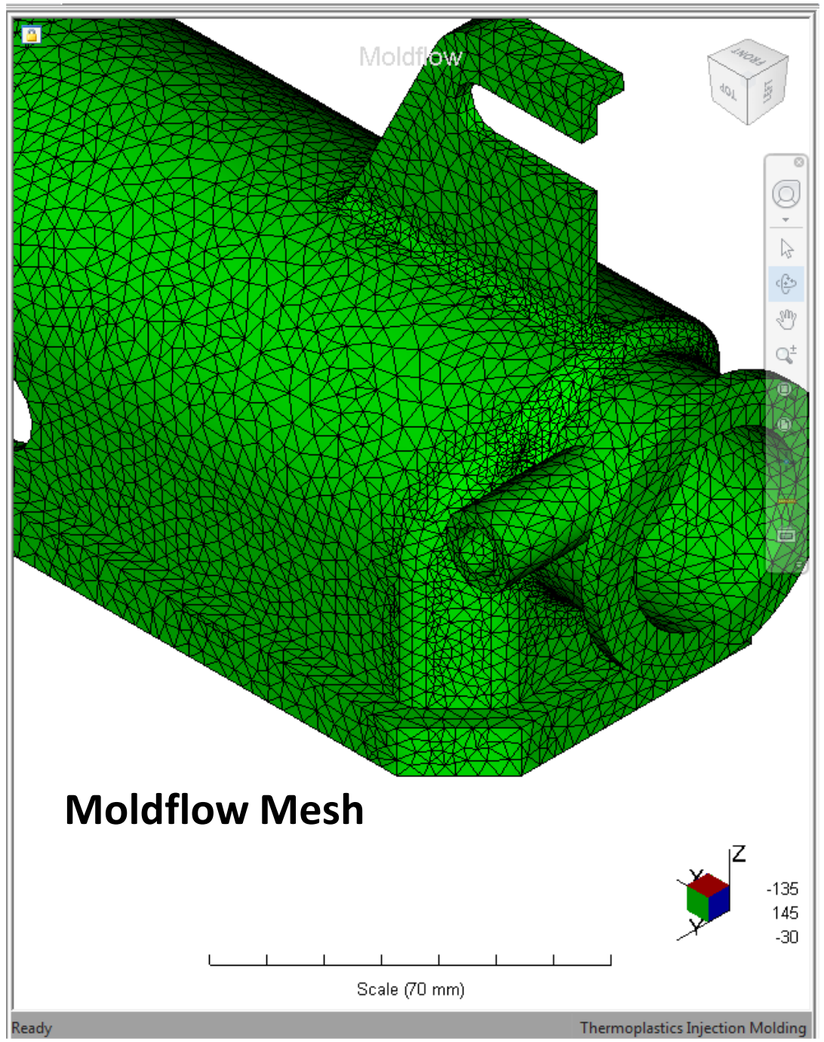
Fibers in material can add strength. Image courtesy of BASF Corporation.

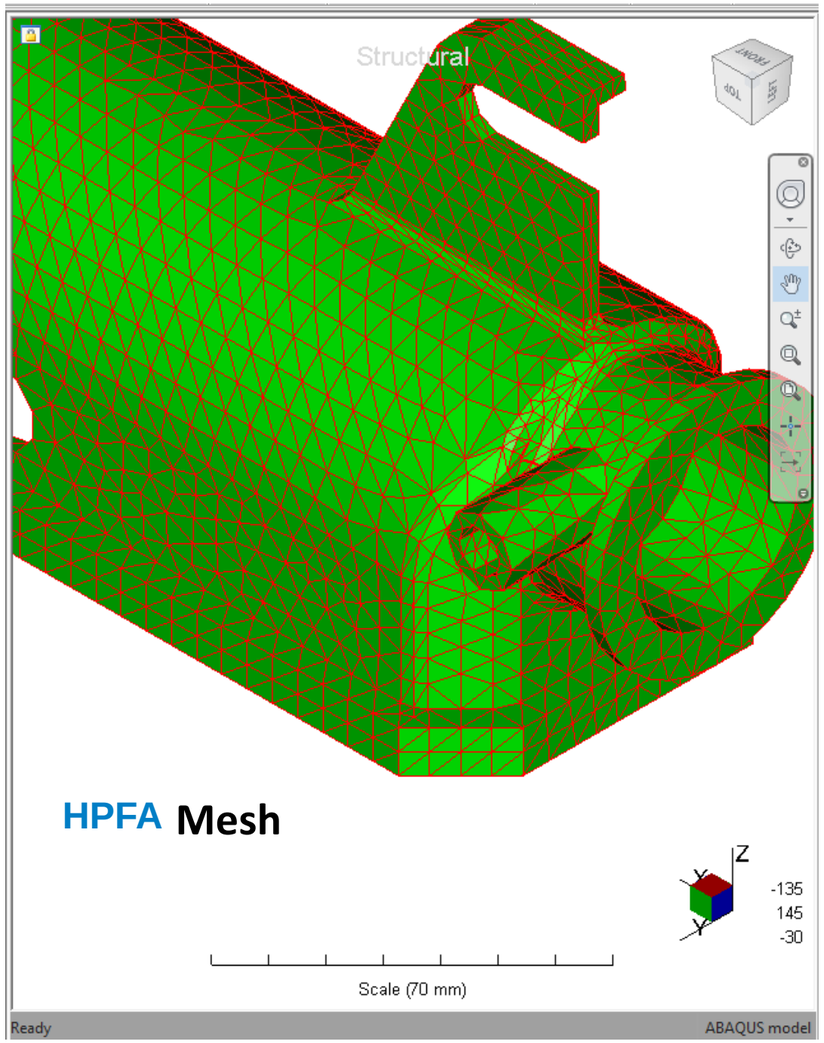

CAD modeling
'as-designed' configuration
Nonlinear Structural Response Simulation
Shape, Boundary condition
Injection Molding Simulation
'as-manufactured' configuration
Multiscale Material Simulation
At each integration point:
• Fiber orientation tensor
• Fiber volume fraction


CAD modeling
AUTODESK® INVENTOR®
'as-designed' configuration
Shape, Boundary condition
Nonlinear Structural Response Simulation
AUTODESK NASTRAN IN-CAD
Injection Molding Simulation
AUTODESK® MOLDFLOW®
'as-manufactured' configuration
At each integration point:
• Fiber orientation tensor
• Fiber volume fraction
Multiscale Material Simulation
AUTODESK® HELIUS PFA

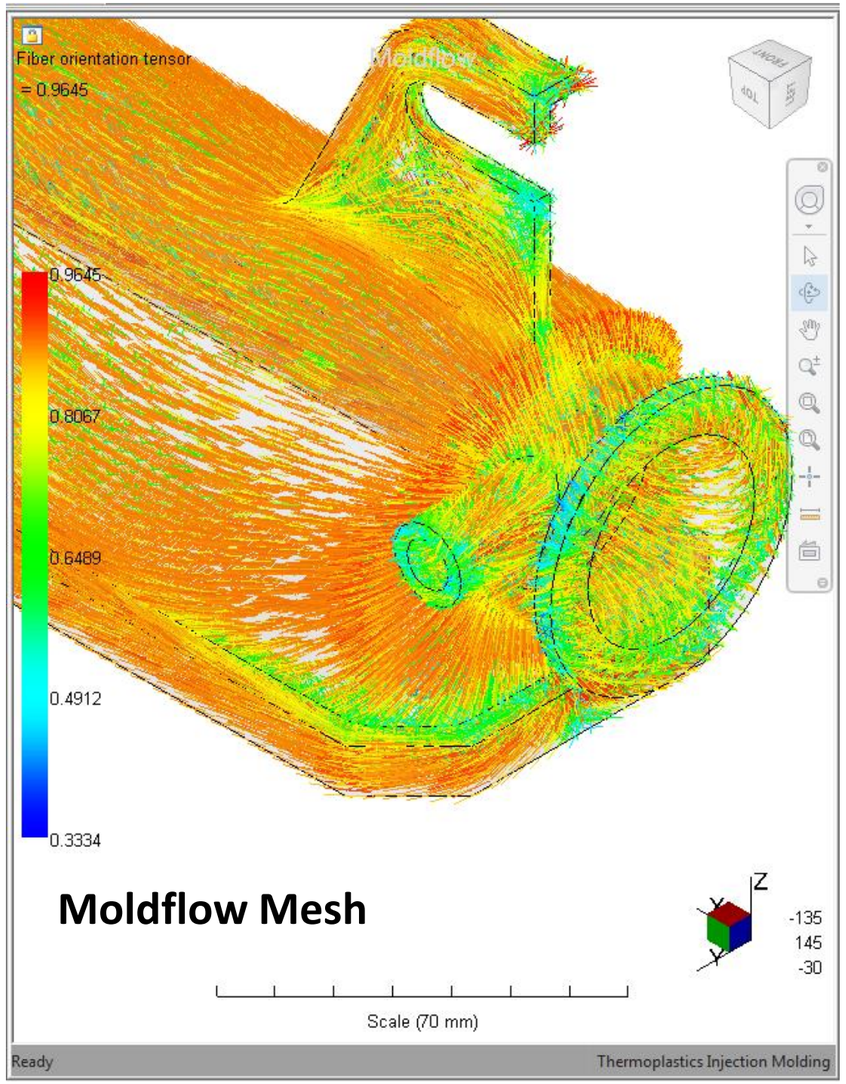
In general, the meshes used for injection molding simulation and mechanical response simulation are expected to be different.

→ Interpolate the Moldflow-predicted fiber orientation distribution and fiber volume fraction distribution onto the structural simulation mesh.

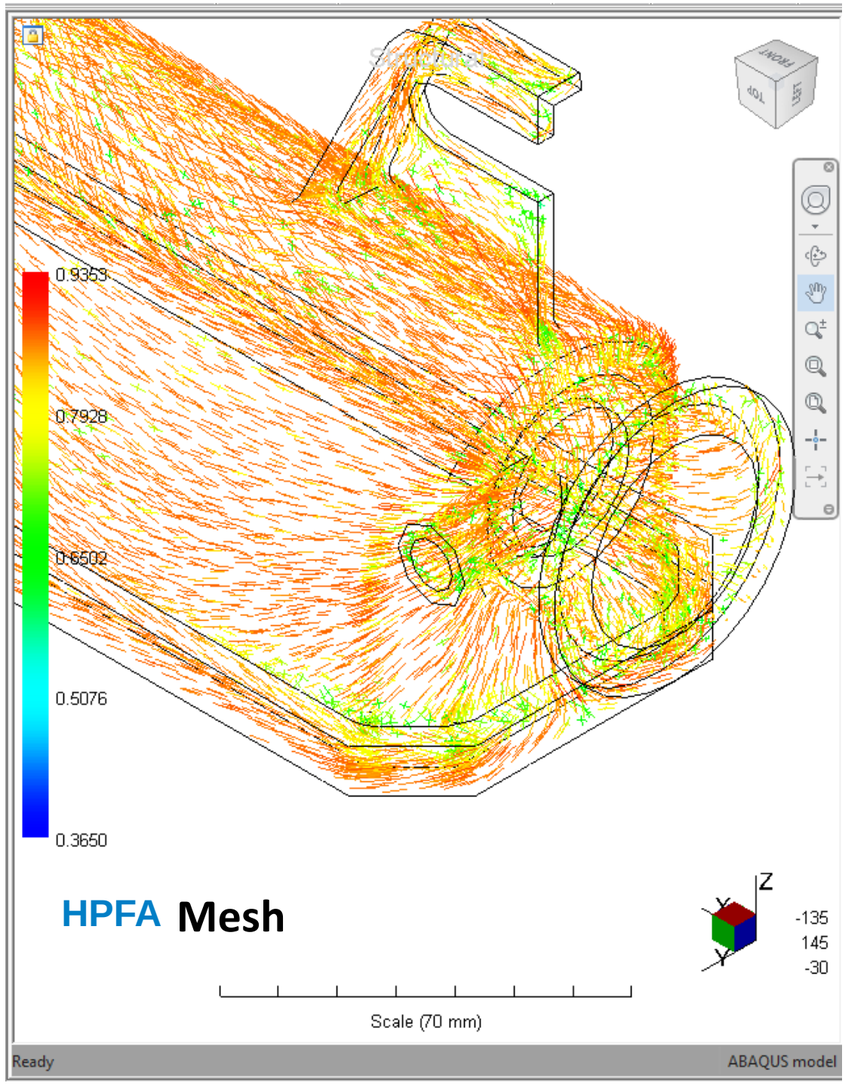




Moldflow-predicted fiber orientation

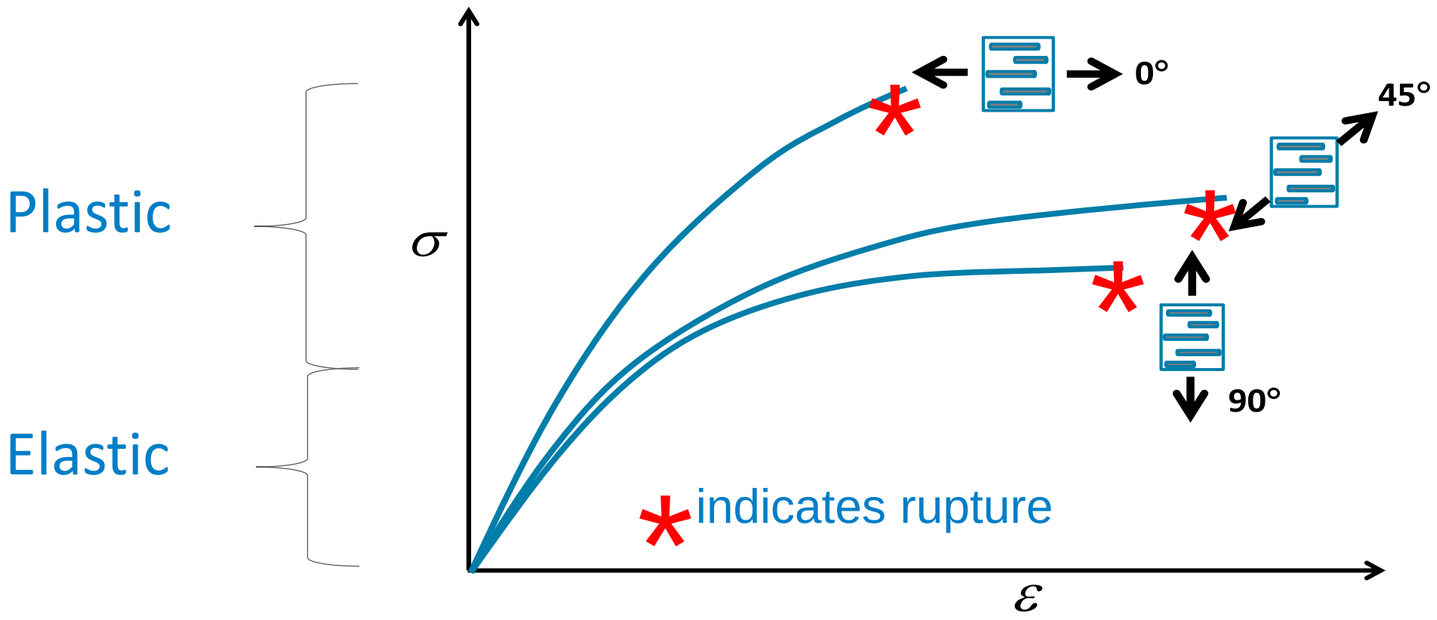


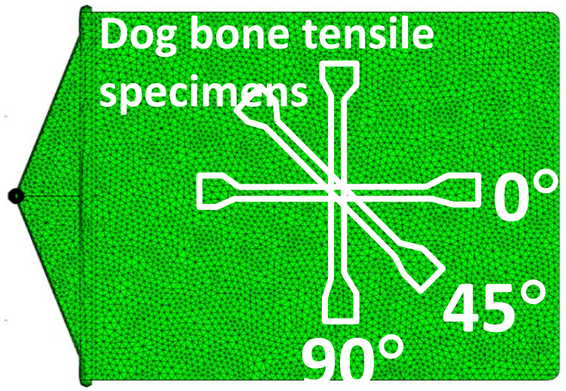
Interpolated fiber orientation



Bringing In-Situ Properties to Simulation

- Fiber Orientation Tensor & Fiber Volume Fraction
- Multiscale Progressive Failure Material Model
- Material Characterization



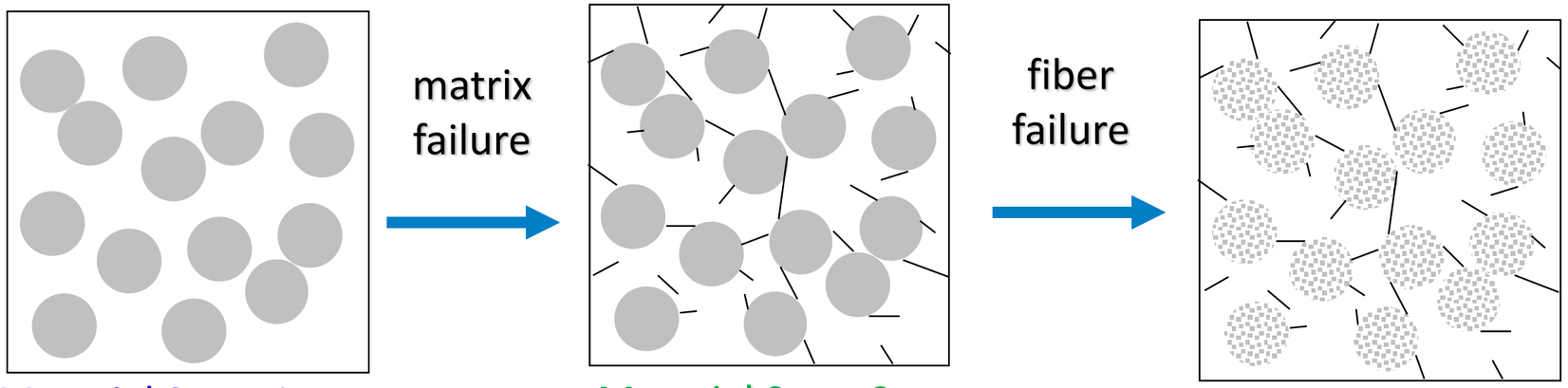


Nastran In-CAD w/ Helius PFA support

- Nastran In-CAD 2018.1 supports Advanced Material Exchange (AME) workflow in Helius PFA enabling as-manufactured workflow for plastic parts
- One stop shop solution for as-manufactured design and simulation with Autodesk platform
- Include as-manufactured earlier in the design process to make a better part and explore all opportunities for a great part
- Companion to existing FEA tools to allow better engineering decisions

Theory – how it works

Progressive Failure Analysis

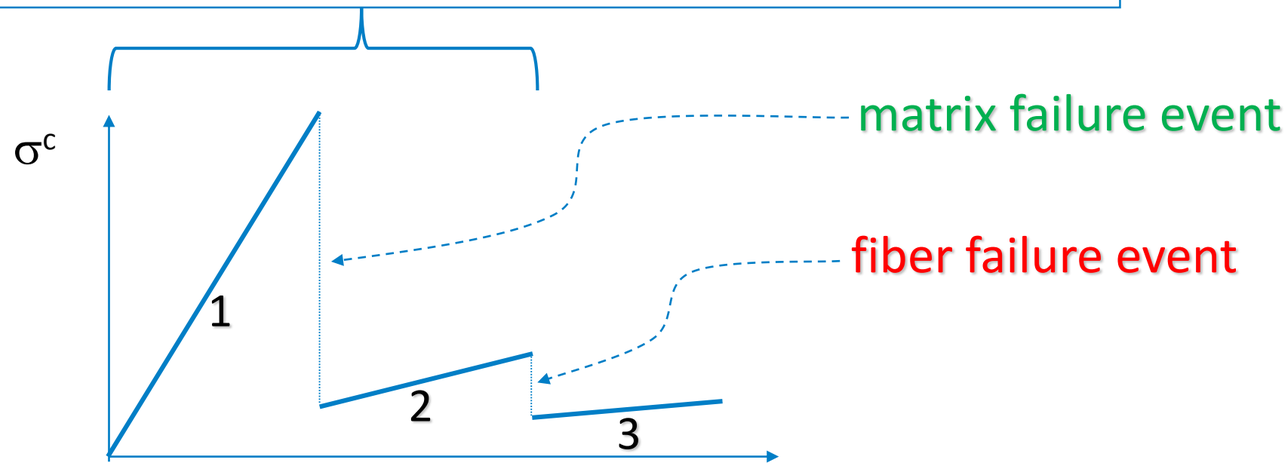


Material State 1
undamaged matrix,
undamaged fibers

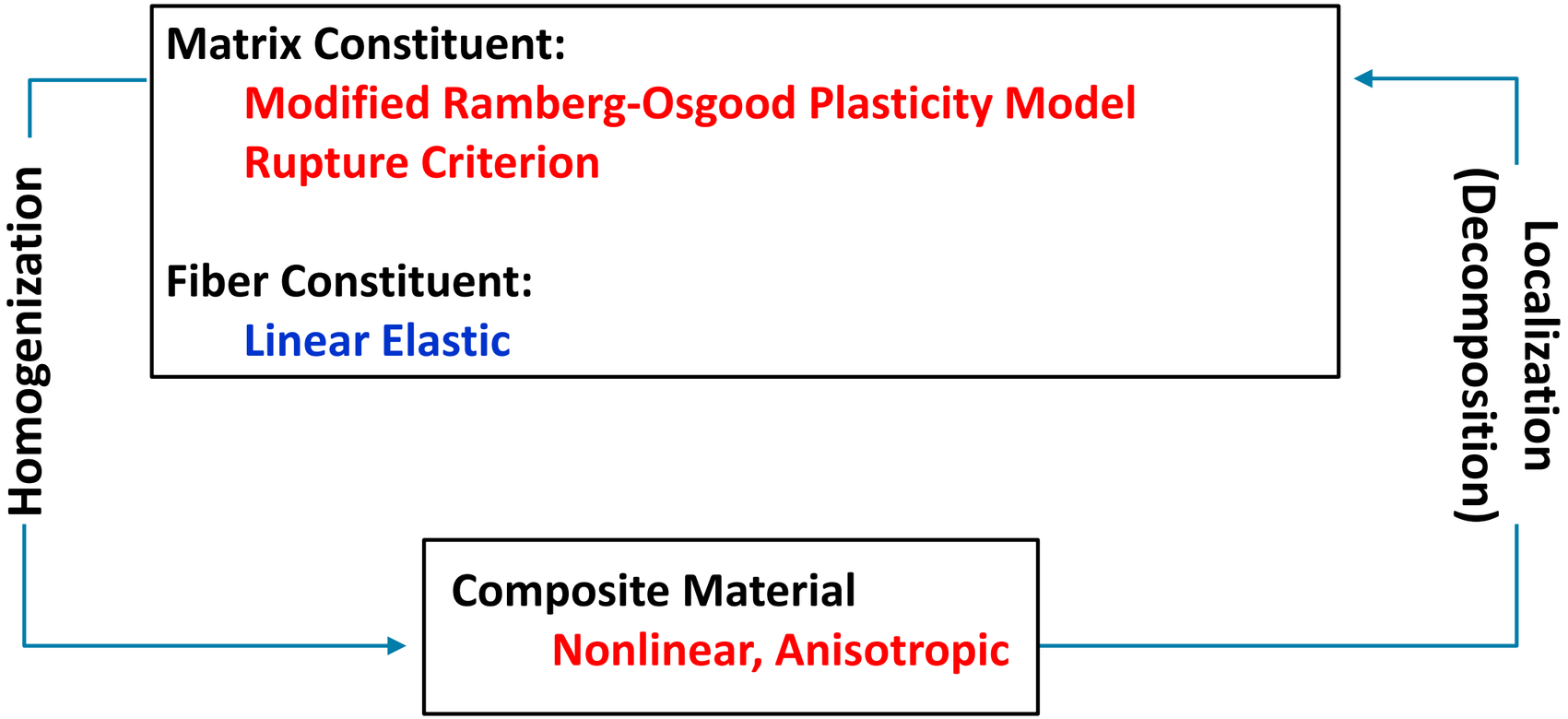
Material State 2
failed matrix,
undamaged fibers

Material State 3
failed matrix,
failed fibers

Injection molded Short fiber composite plastic



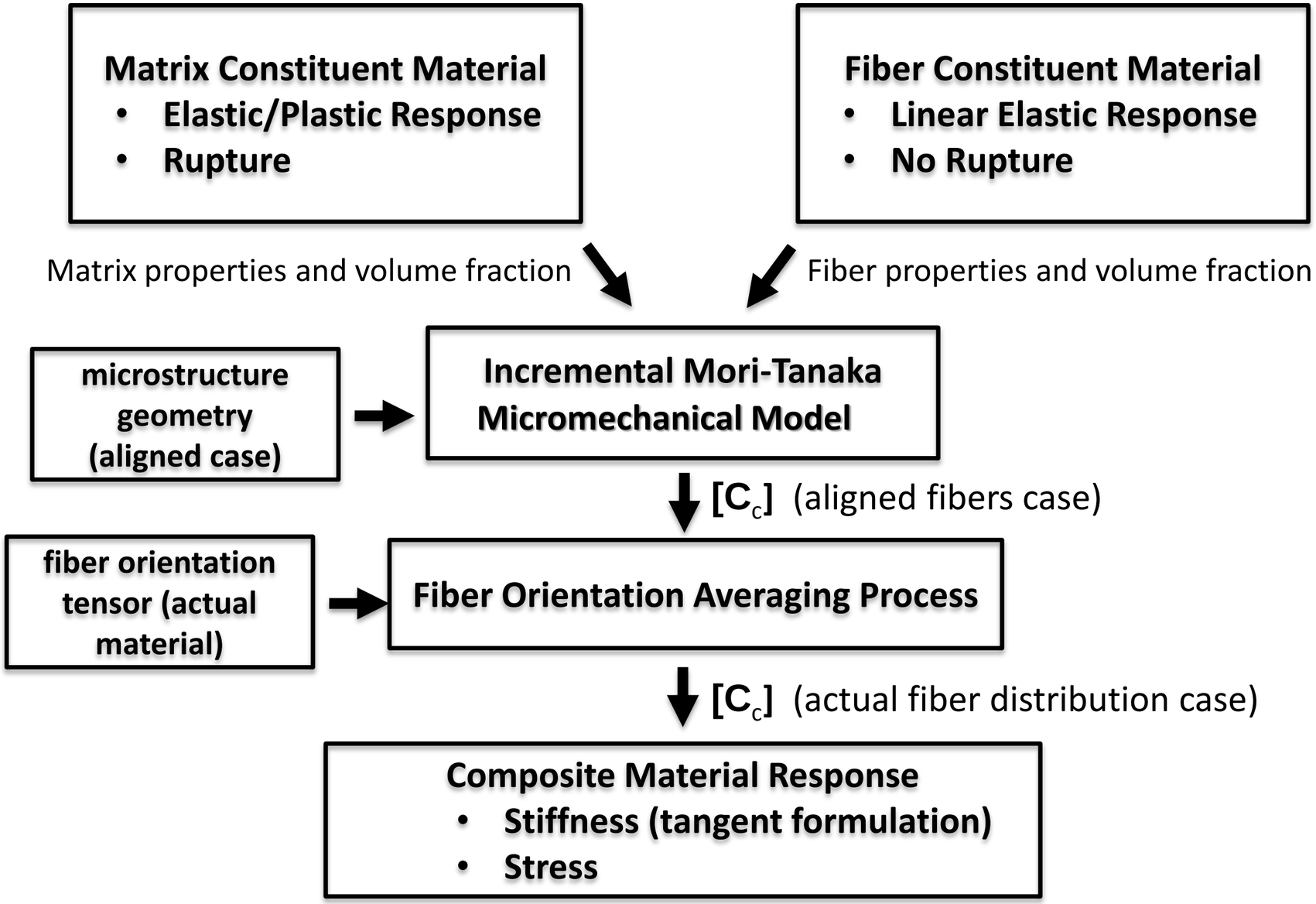
Multiscale Material Model (for progressive failure)



Constraints:

- Keep the material model as simple as possible while still retaining the essential material response features
- Hold the number of material model coefficients to a minimum so that the material model can be characterized using a minimum of simple test data

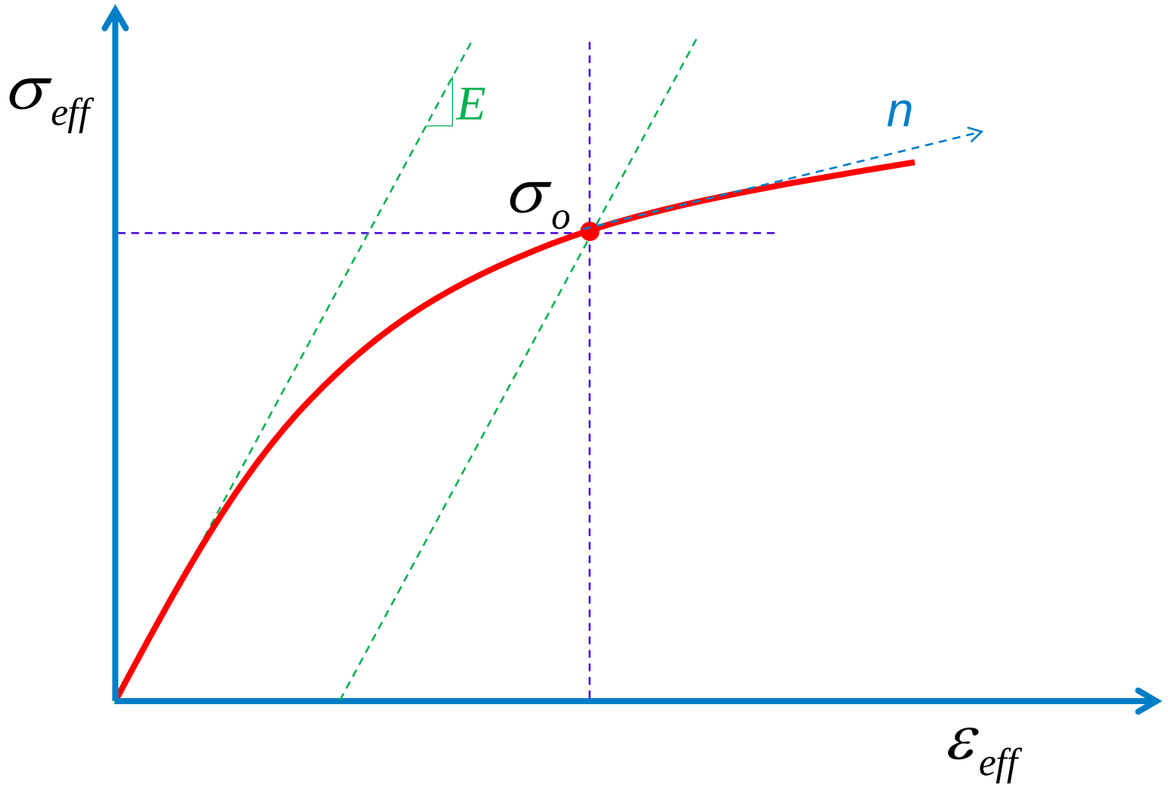
Material Model: Homogenization



Plasticity of the Matrix Constituent

Matrix Constituent Material

- **Elastic/Plastic Response**
- **Rupture**



σ_{eff} = von Mises stress

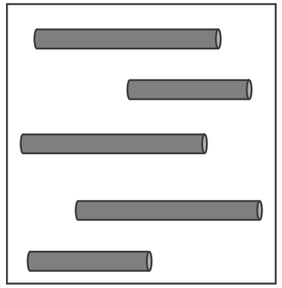
Ramberg-Osgood Plasticity Model

Two elasticity Parameters (E, μ)

Two Plasticity Parameters (σ_o, n)

- σ_o controls the effective stress level at which the plastic strain becomes as large as the elastic strain
- n controls the slope of the response at the effective stress level of σ_o
- matrix plasticity is driven by stresses in the matrix constituent material (requires the ability to decompose stress)

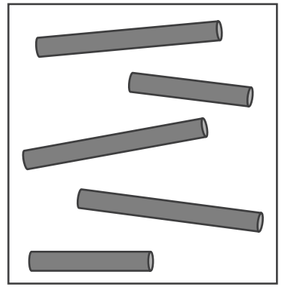
How does the model account for changes in the degree of fiber alignment?



Perfectly aligned

$$[FOT] = \begin{bmatrix} 1 & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & 0 \end{bmatrix}$$

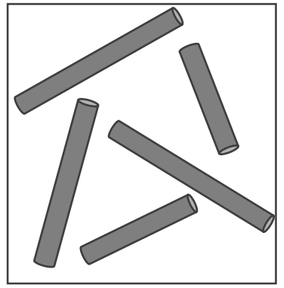
$\lambda_I = 1.0$



Mostly aligned

$$[FOT] = \begin{bmatrix} 0.836 & -0.056 & -0.018 \\ -0.056 & 0.098 & 0.129 \\ -0.018 & 0.129 & 0.066 \end{bmatrix}$$

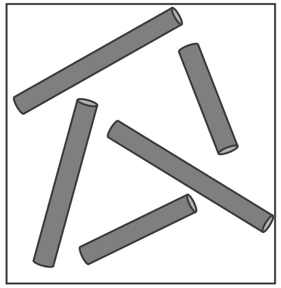
$\lambda_I = 0.841$



2-D random

$$[FOT] = \begin{bmatrix} 0.5 & 0 & 0 \\ 0 & 0.5 & 0 \\ 0 & 0 & 0 \end{bmatrix}$$

$\lambda_I = 1/2$



3-D random

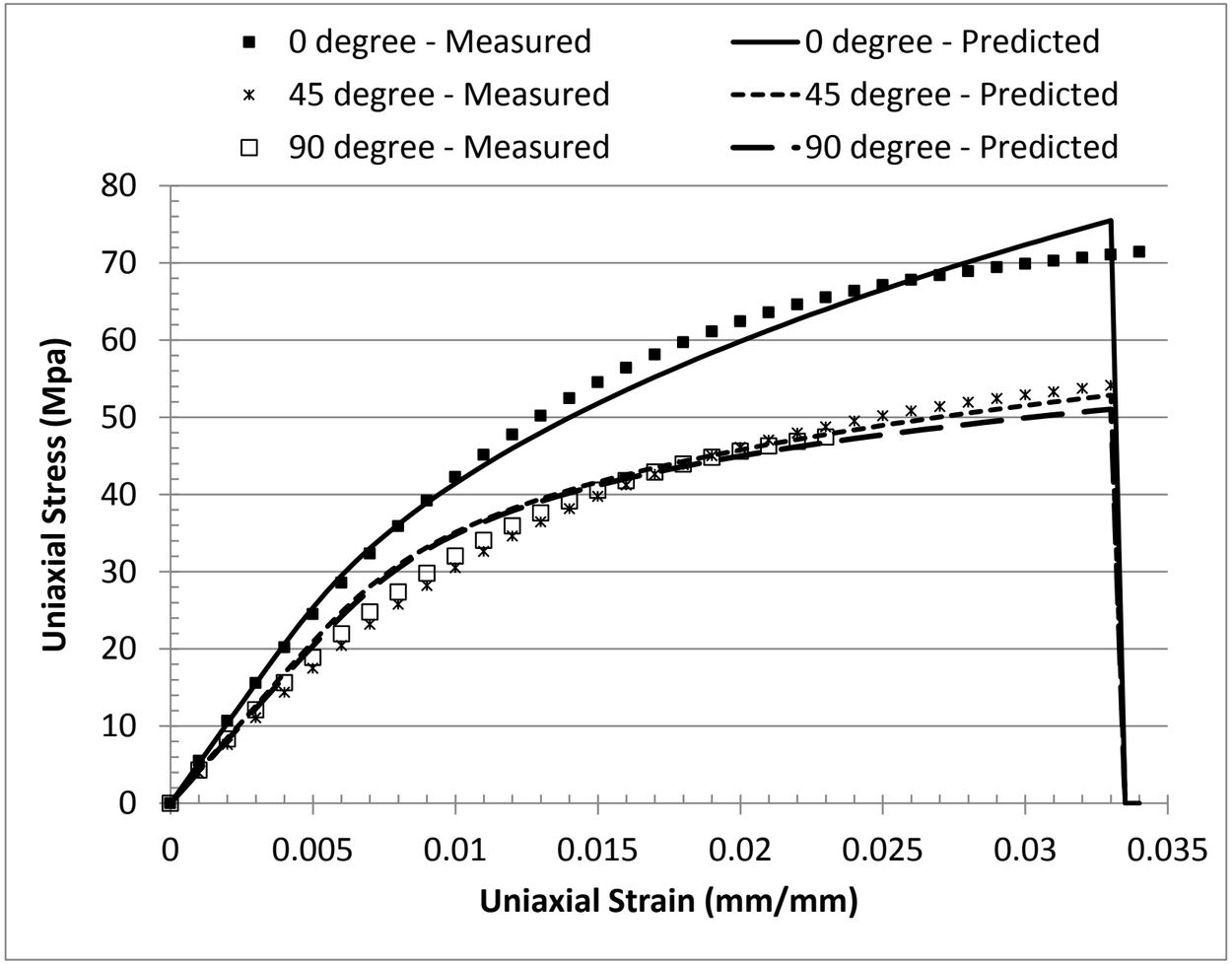
$$[FOT] = \begin{bmatrix} 1/3 & 0 & 0 \\ 0 & 1/3 & 0 \\ 0 & 0 & 1/3 \end{bmatrix}$$

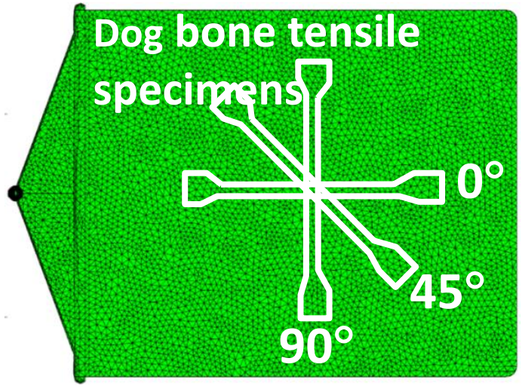
$\lambda_I = 1/3$

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 $\lambda_I = 1/2$.

Material Characterization:
Use measured uniaxial data to fit model coefficients

$$[\ \overbrace{(E_m,\ \mu_m,\ \sigma_o,\ n,\ \alpha,\ \beta)}^{\text{matrix}},\ \overbrace{(E_f,\ \mu_f)}^{\text{fiber}}\ ]$$

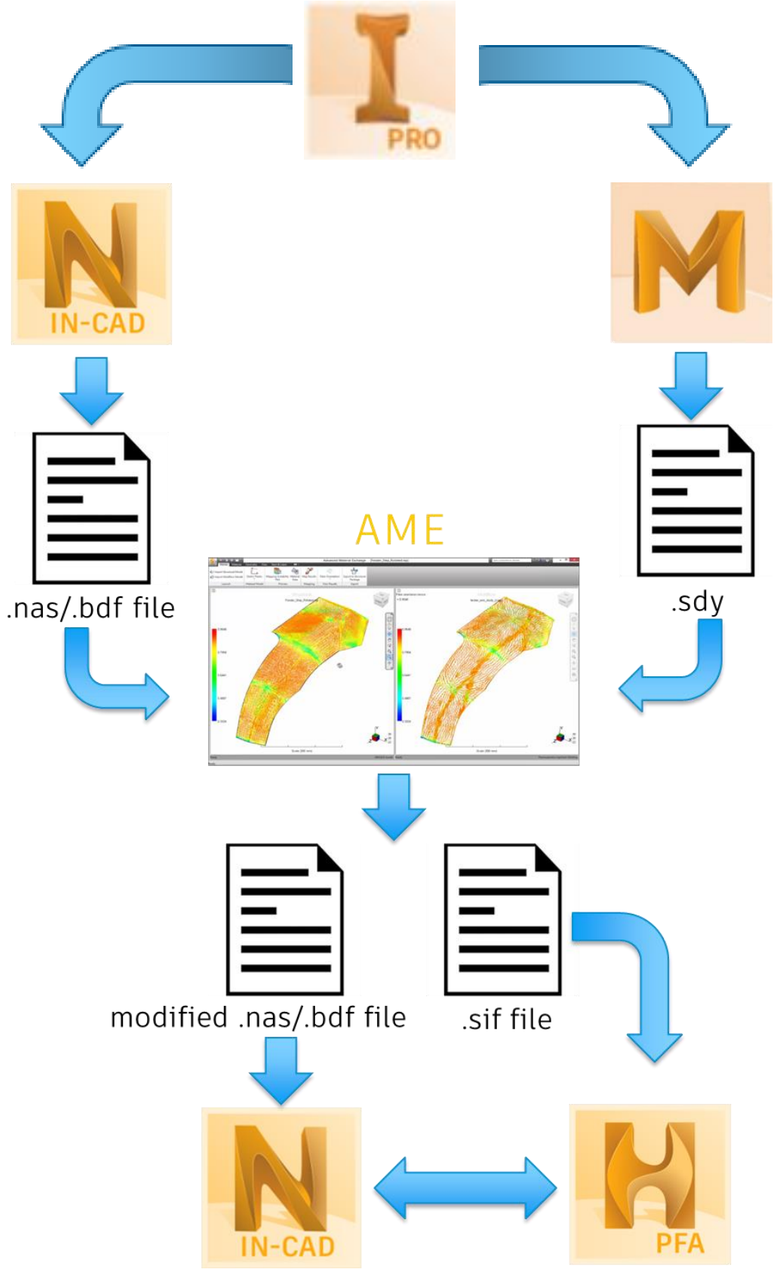




Workflow – Bringing everything together

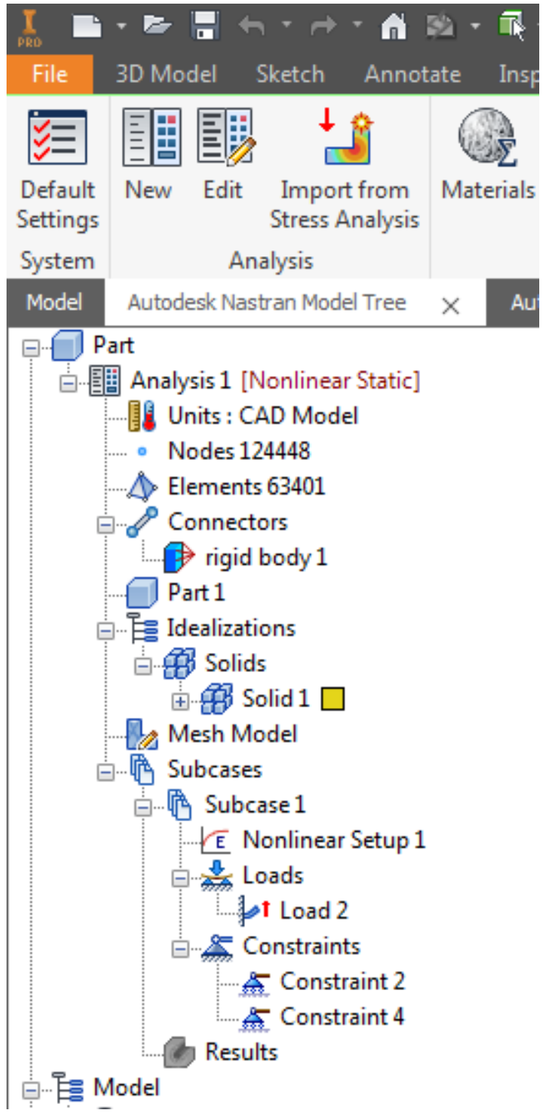
Basic Workflow

1. Create Geometry in Inventor
2. Moldflow Injection Molding Analysis
3. **Preprocess structural file in In-CAD**
4. **Map Moldflow results in AME**
5. **Solve and postprocess with In-CAD with Helius PFA**



Step 1: In-CAD Preprocessing

1. Meshing
   - Type: CTETRA
2. Materials
   - Define Materials and Solid Sections
3. Boundary Conditions
4. Applied Loads/Displacements
5. Nonlinear Static Analysis Settings
   - Turn large displacements off
   - Turn intermediate results on
   - Number of Increments: Min 30, Max 100

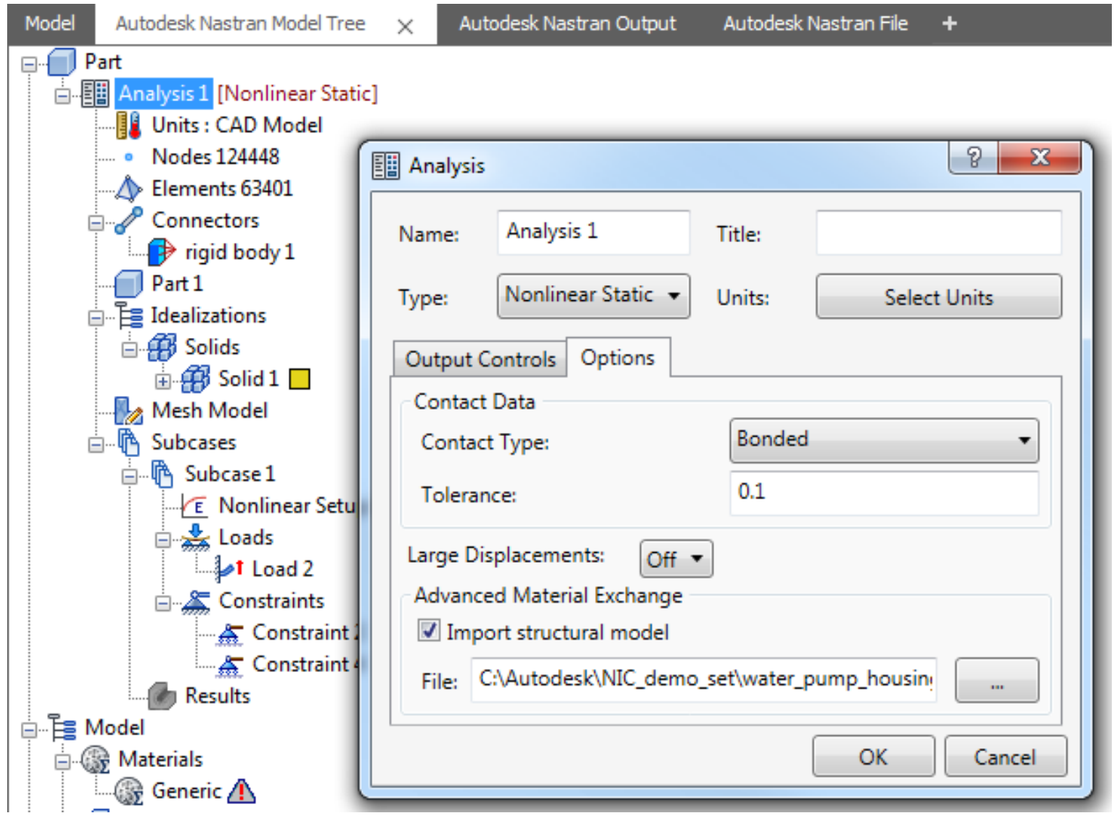


Step 2: AME

1. Pair models
   - Import Moldflow study
   - Import Nastran structural file
   - Align geometry
3. Nonlinear material
   - Select Environment or,
   - Add Nonlinear Material to supply nonlinear stress-strain data in .csv format
3. Map results
4. Export to Structural Package
   - Settings
     - Enable rupture
     - Enable weld lines
     - Enable warpage

Step 3: Solve and Post-process with In-CAD

1. Import AME mapped structural files
   - Imported/linked through Nonlinear Static dialog
2. Solve
3. Post-process
   - View Results
   - View Helius PFA results



Optional Steps

- After linking to location of mapped input file (Step 1 of previous slide), users are allowed to make some modifications to the model
- Adjust magnitude/direction of applied loads/displacements
- Adjust boundary conditions
- Adjust increments
- Add subcases
- **Changes to the mesh are not allowed after linking**

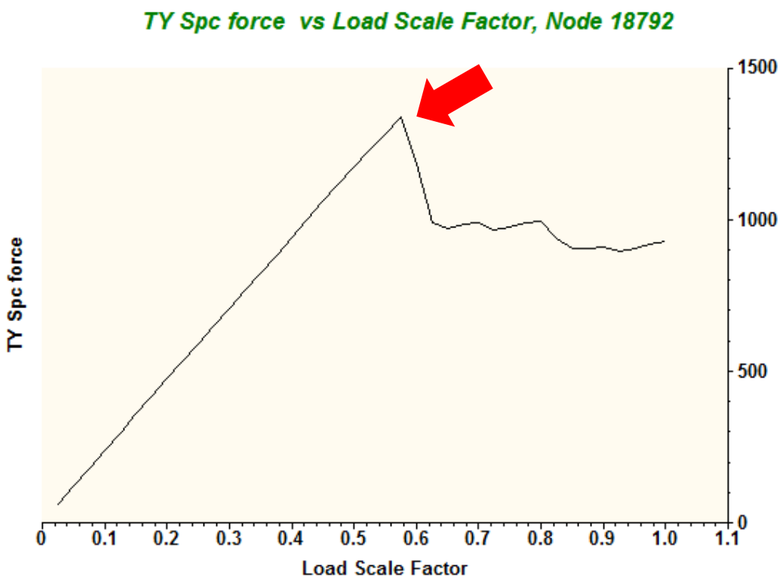
Nastran In-CAD and Helius PFA Advantages

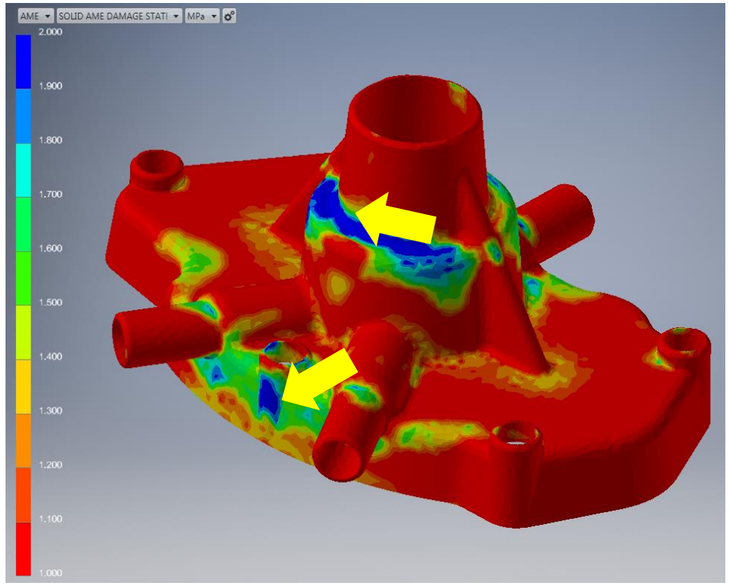
- Injection molded fiber filled and unfilled plastics
- Nonlinear Static Analysis
- Assemblies and Contact
- Progressive Failure
- Weld Line Failure Analysis
- Residual Stress Induced Deformation
- Linear Material Model
- Nonlinear Material Model

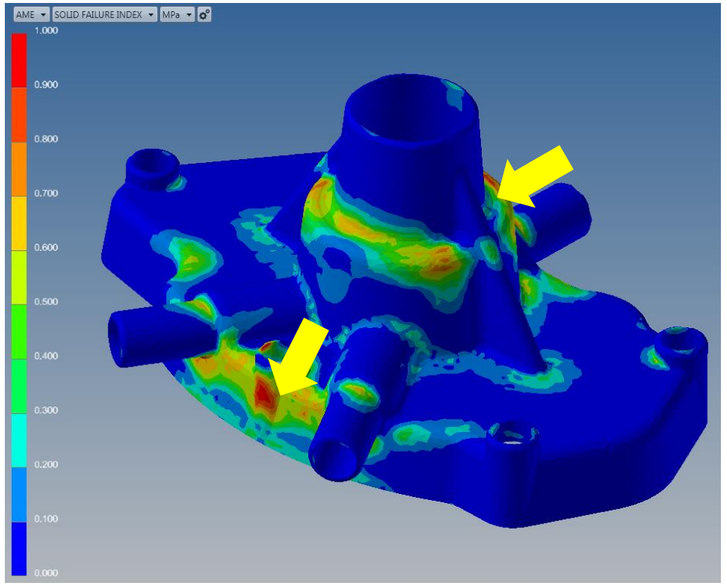
State Variable Outputs for Helius PFA

Result	Description
DAMAGE STATE	Gauss point degradation status (1 = Not Ruptured, 2 = Ruptured)
MATRIX TANGENT MODULUS	The tangent elastic modulus of the matrix constituent.
MATRIX EFFECTIVE PLASTIC STRAIN	The effective plastic strain of the matrix constituent.
MATRIX EFFECTIVE STRESS	The effective stress of the matrix constituent.
WELD SURFACE STRENGTH REDUCTION FACTOR	This result is only applicable to analyses that have weld surfaces activated. It is useful for visualizing the location of weld surfaces in your part and understanding the reduction in strength applied to each element on the weld surface.
FAILURE INDEX	This result is a continuous real variable that ranges from 0.0 to 1.0. It is used to indicate the fraction of the matrix failure criterion that has been satisfied.0.0 = the matrix stress state is zero. 1.0 = the matrix stress state has initiated failure. This result is computed as the left hand side of Eq. 9 in The Rupture Model.
FAILURE MODE	Failure mode. 0 represents no failure, 1 represents a tensile failure, and 2 represents a compressive failure. Note: If the material is forced to use a tensile response under compressive stress state, the failure mode is reported as tensile.

Useful Post-Processing Results



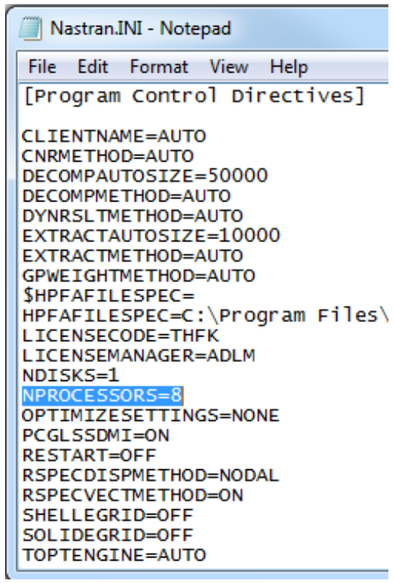




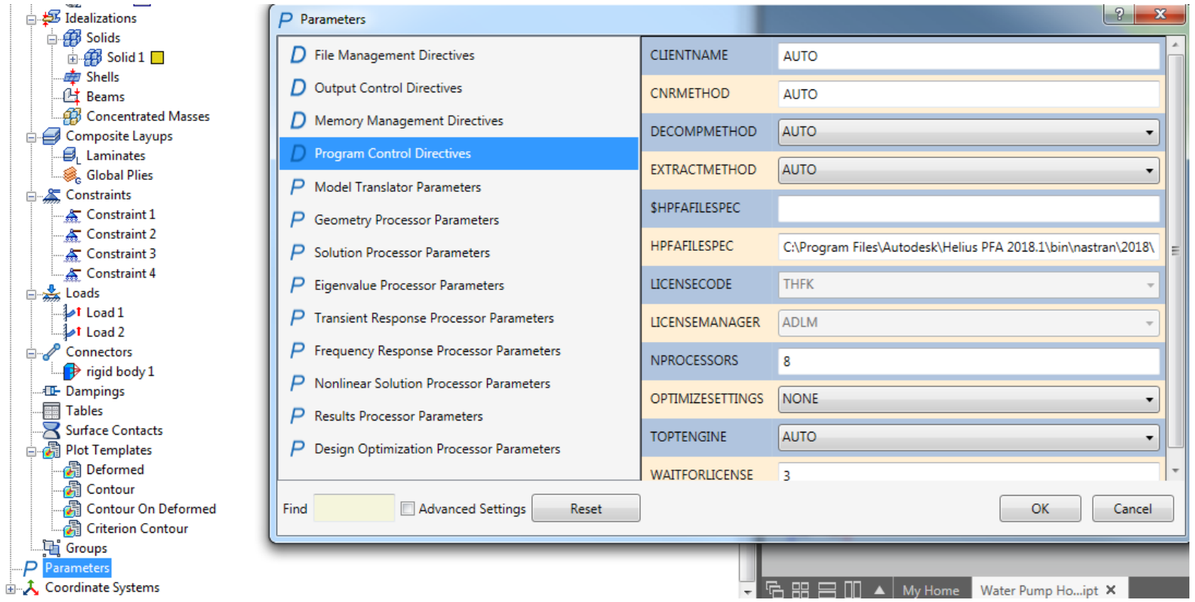
- Assess changes in global stiffness and determine the ultimate load
- Understand the location and extent of material damage
- Pinpoint regions that are close to rupture

Throttle the simulation: Setting the Number of Processors

- Via Nastran.ini file
  - Location *Install_Dir*\Autodesk\Nastran In-CAD 2018\System\Nastran



- Via Parameters in In-CAD



Demos

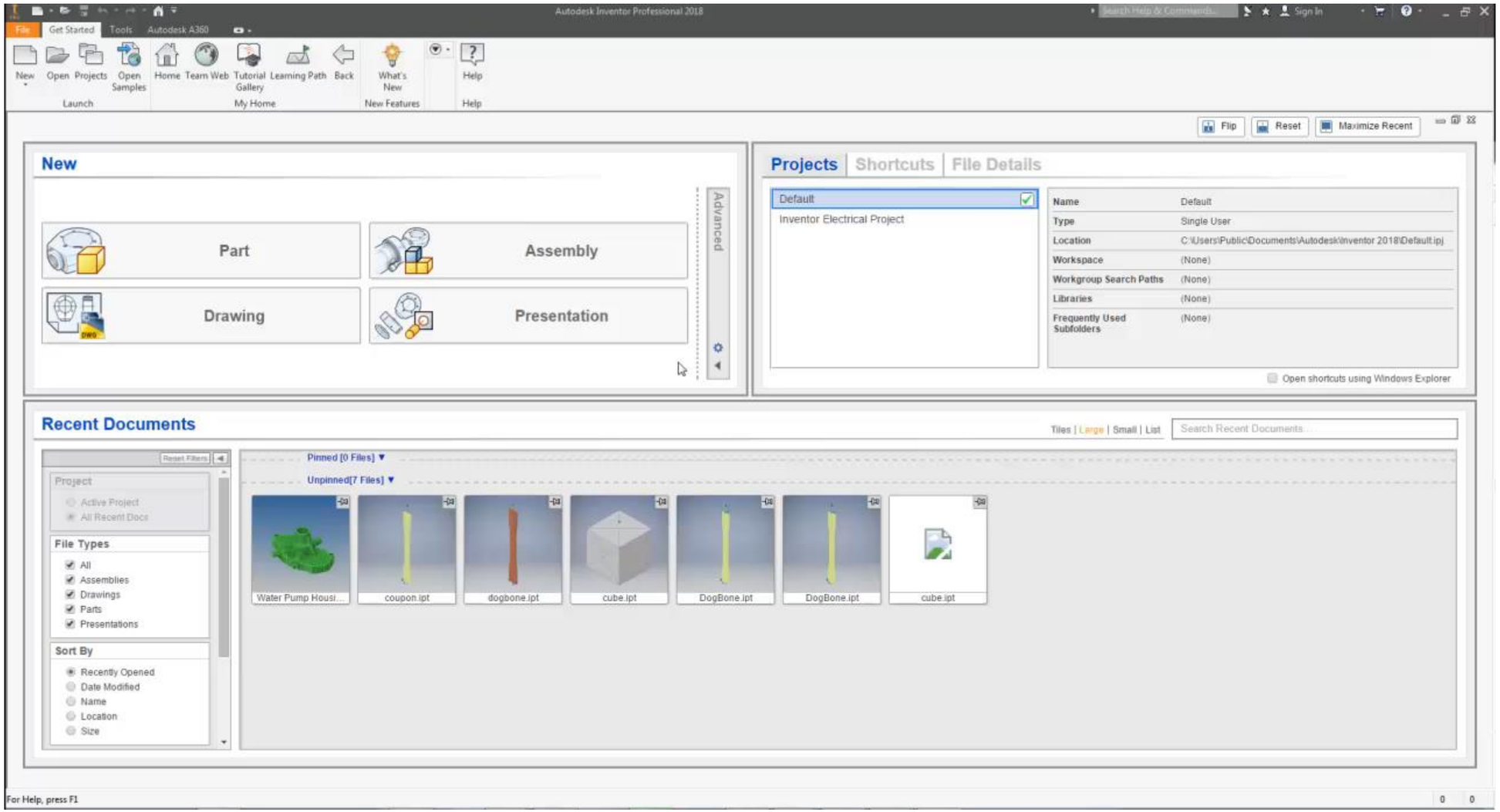
Workflow Demonstration: Video

- Nastran In-CAD: Part 1
  - Preprocess
  - Export to AME
- AME
  - Import Moldflow study and Nastran structural file
  - Alignment
  - Map Moldflow results
  - Export mapped Moldflow results

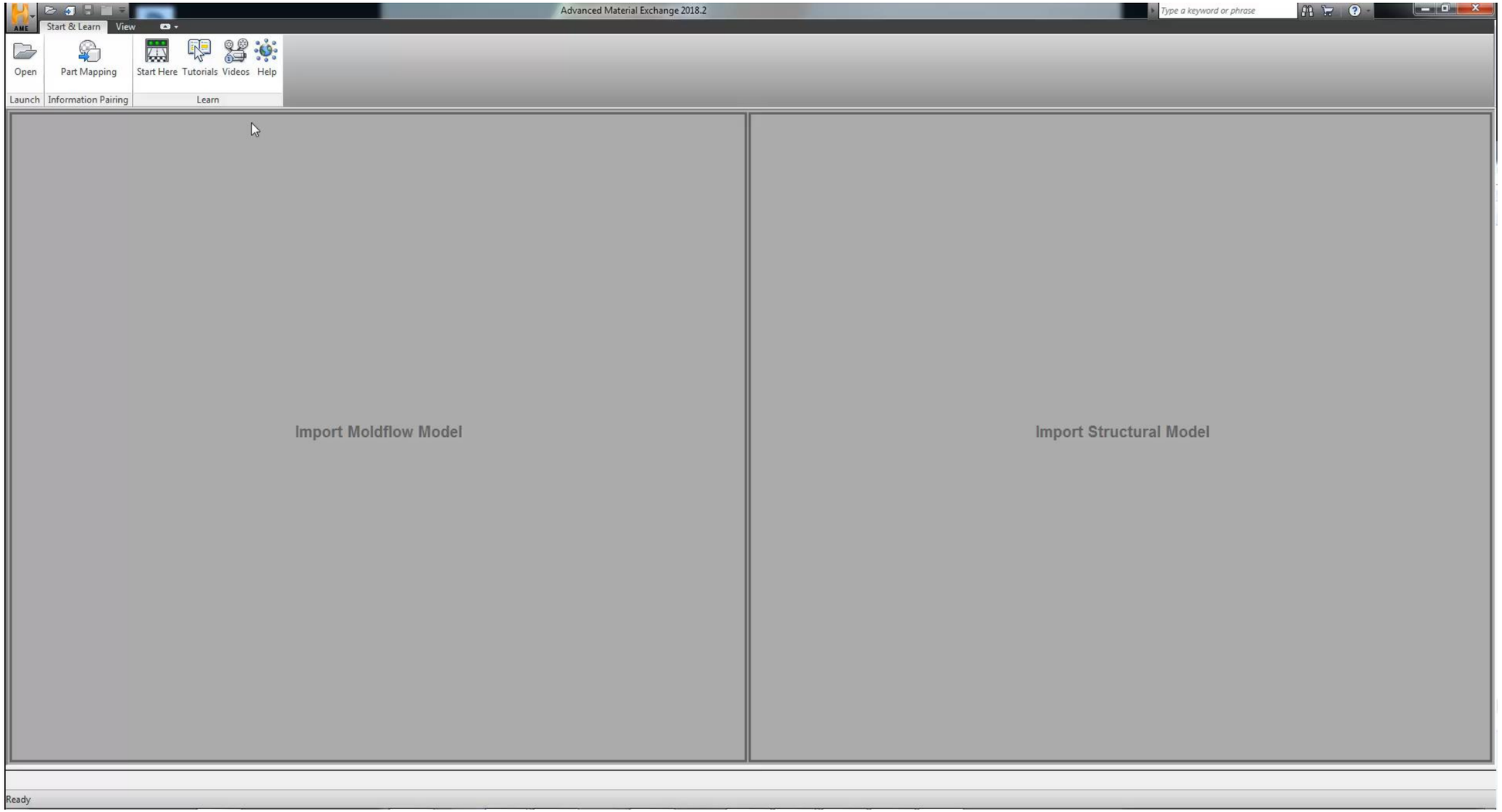
Workflow Demonstration: Video

- Nastran In-CAD: Part 2
  - Import Mapped Files
  - Run Structural Analysis
  - Interpret Helius PFA Results

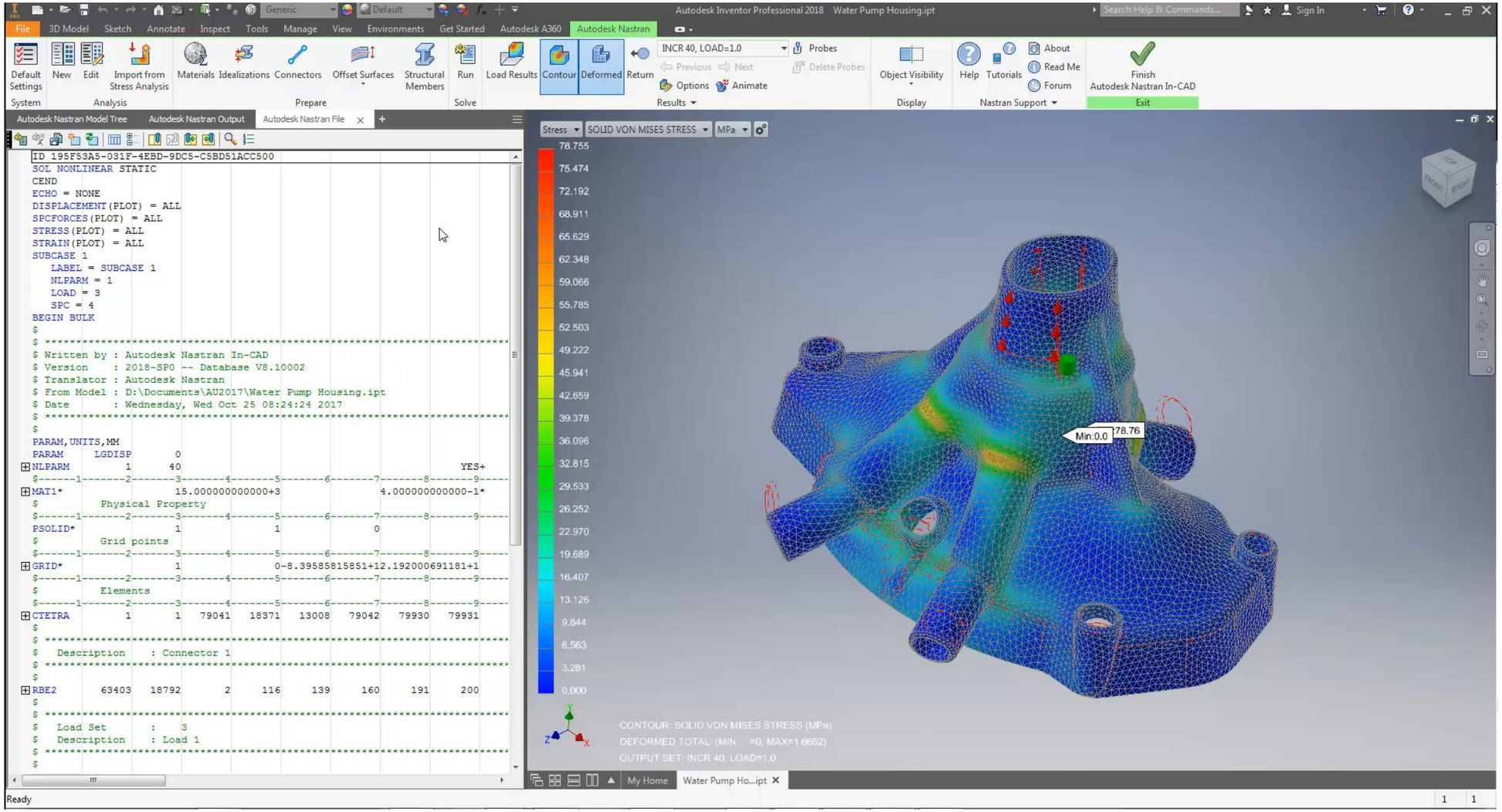
Nastran In-Cad: Preprocessing and Export



AME: Mapping Moldflow Results



Nastran In-CAD: Helius PFA Analysis



Live Demonstration

- Outline
  - View pre-processed model in In-CAD
  - View mapped models in AME and explore the UI
  - View completed run in In-CAD and explore the results

Summary

AUTODESK
NASTRAN IN-CAD

AUTODESK®
HELIUS PFA

AUTODESK®
MOLDFLOW®

What next – NASTRAN/Helius PFA/AME

- Broaden the support:
  - Additional analysis types and element types
- Deepen the support:
  - Advanced material model – more physics
- Optimization - Feedback loop to Moldflow

AUTODESK®

Make anything.