# 2018 Composites Webinar

Sean Harvey

Technical Services Manager

# Agenda

- Composites Overview
- Composites Simulation Workflow and Capabilities
- Delamination and Progressive Failure
- Multiscale Modeling (ACT)
- Composite Cure Simulation
- What’s New in R19?
- Q&A

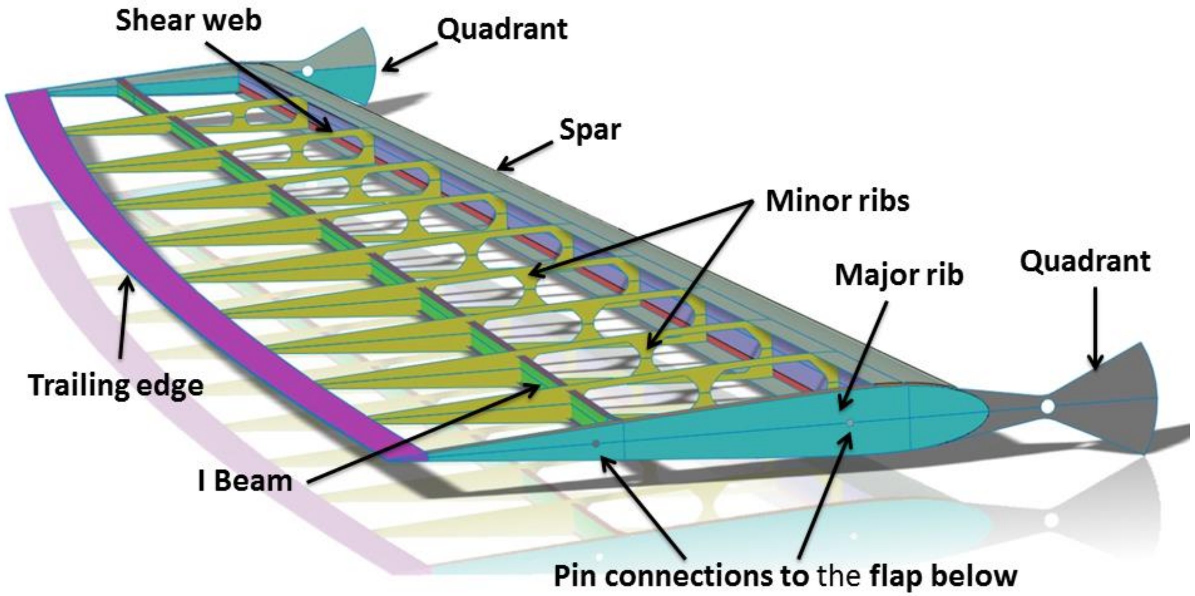
ANSYS®

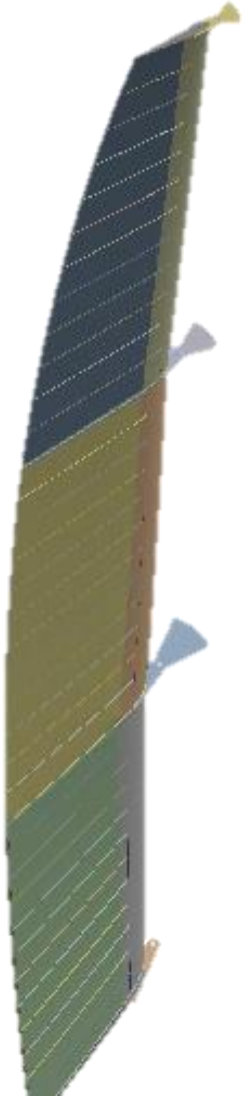
# Pushing the Boundaries

# 2017 America's Cup with ANSYS

## Why Important/Engineering Challenge

- Simple. In competitive Sports, Winning is Everything.
- Make the strongest, lightest, most efficient Sail

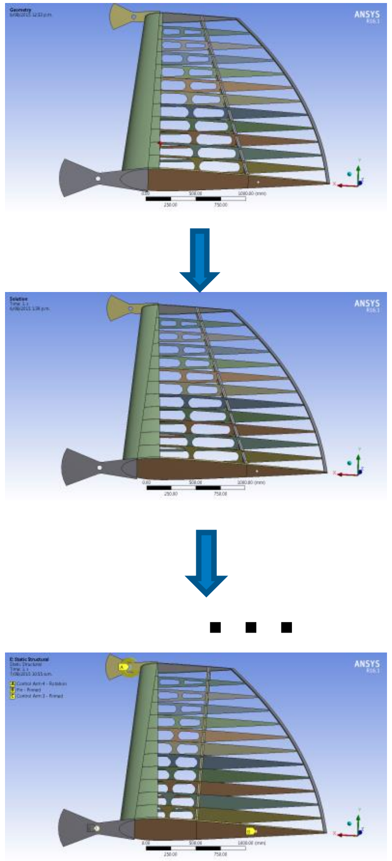


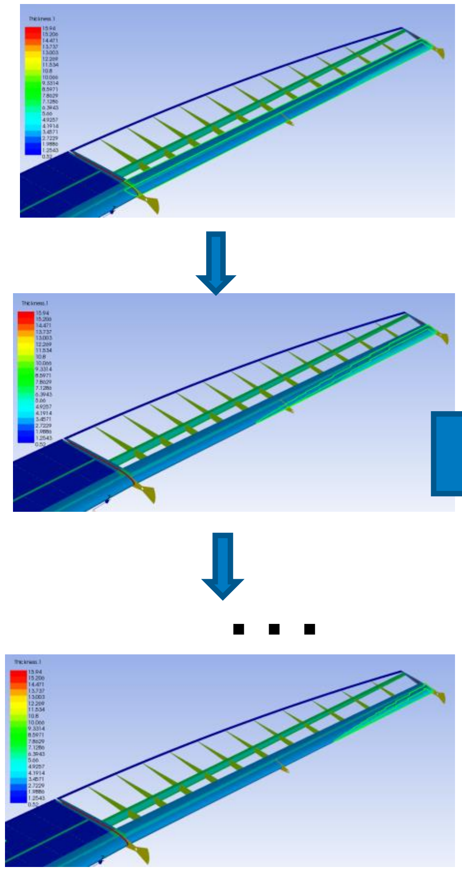
  July 5, 2018 

# How? Finding the Optimal Solution with ANSYS

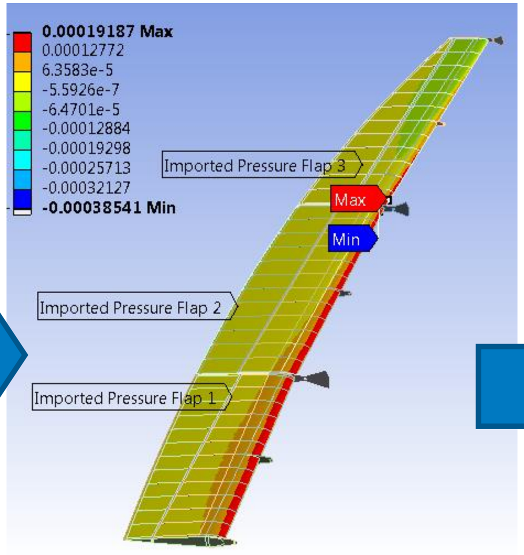
Geometric Variations

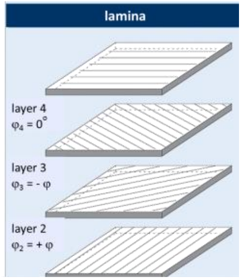


Composite Laminate Variations

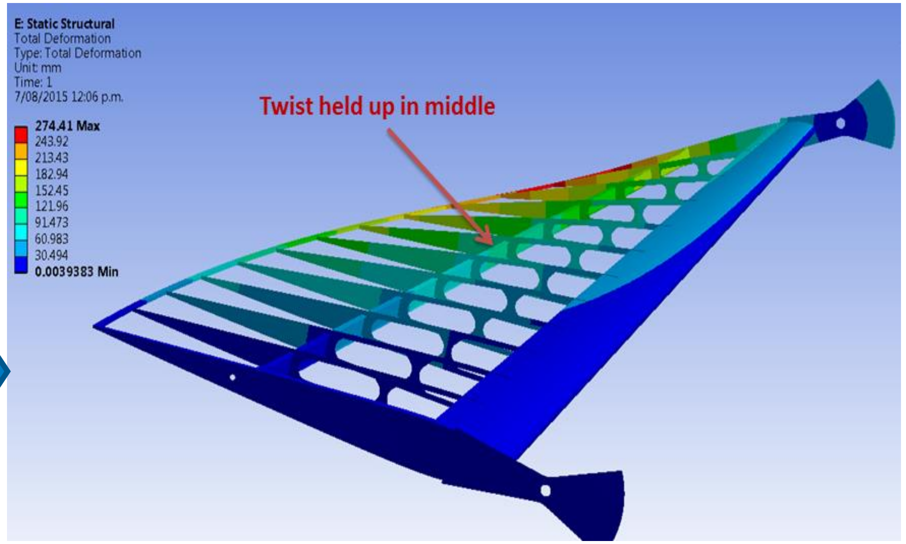


Pressure Loading




Maximize Strength
Minimize Weight and Deflection



## Outcome:

**Team New Zealand Won the America's Cup in June 2017**

"ANSYS provided an edge over our competitors by helping us change the game of traditional sailing," said Nick Hutchins, CFD engineer, Emirates Team New Zealand.

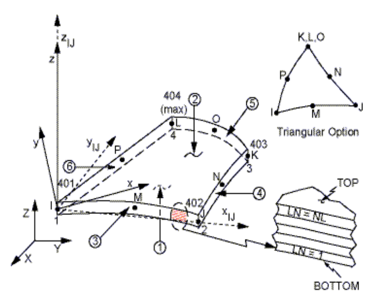
A cornerstone of our design process, ANSYS' simulation tools delivered valuable productivity gains that ultimately led to a faster, more competitive craft."

# Modeling Layered Composites Efficiently

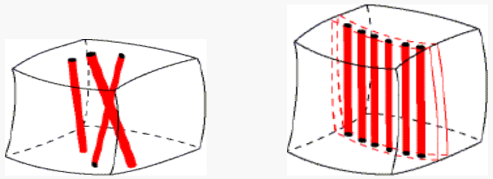
## ANSYS is not new to modeling composites:

- Layered shell and solid elements for >3 decades
- Multi-material beams, with support for multiple layers and multiple section integrations
- Temp-Dep ortho material props with structural temps at each layer
- Composite PrepPost adds ease of use and results evaluation
- VCCT, CZM, Progressive Damage to characterize delamination, debonding and damage mechanics.
- Integration into Workbench allows for rapid design studies with ACP
- Composite Cure (ACCS) for manufacturing and post cure deformation.
- ACT Extensions for multiscale modeling

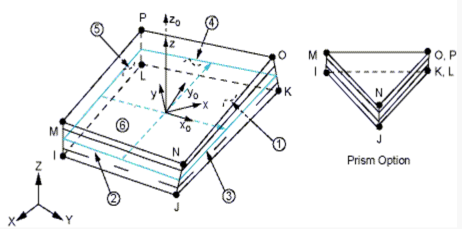
*Shell*



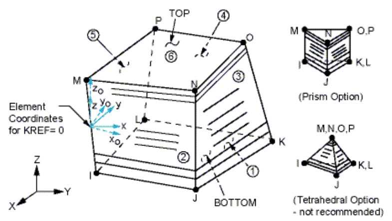

*Discrete/Smeared Reinforcement*



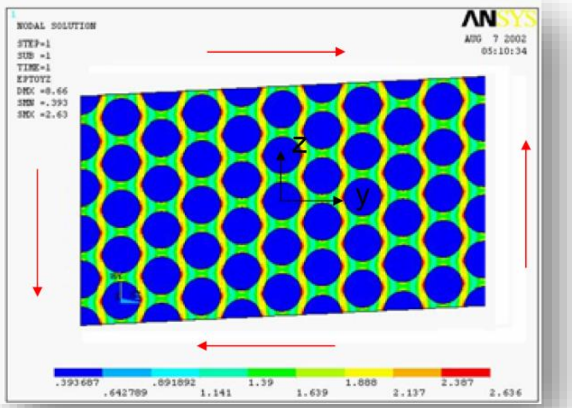
*Solid-Shell*




*Layered Solid*

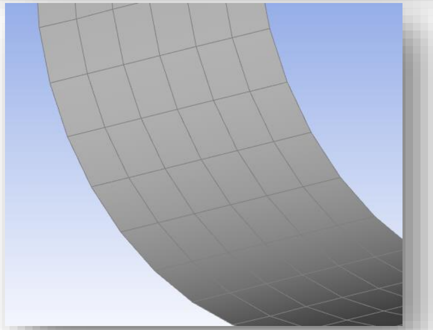


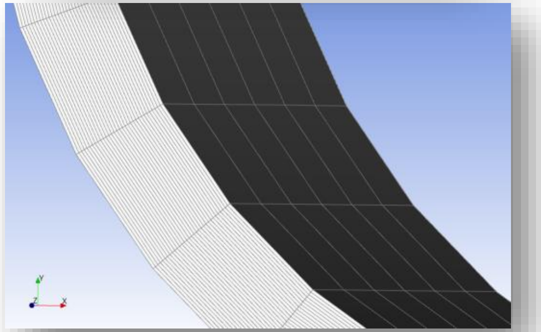
# Modeling Layered Composites Efficiently




***What are some of the numerical approaches?***

- Micro-Scale Approach – Detailed fiber matrix interactions via RVEs
- Meso-Scale (Laminate Level)
  Approach – Layered Shells and Solids, extract displacements, modes, overall stiffness behavior, and detailed stresses and strains
- Macro-Scale Approach – Smeared or homogenous shell/solid, with no detailed ply stresses nor strains.
- Pre-Integrated Shell (A,B,D,E) matrices (Sectype – GENS)





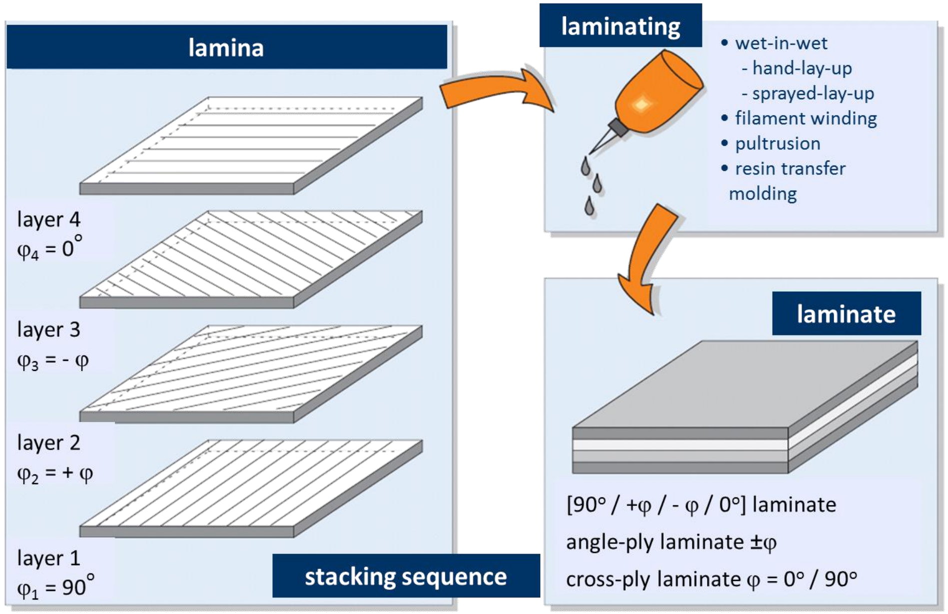
# Modeling Layered Composites Efficiently

- **Highlight several focus area:**
- Allow for geometric design changes to model
- Make changes to layup, ply locations, and orientations
- Solid Models generated automatically by extruding layup
- Incorporate fluid/thermal loading directly from CFD
- Incorporate loading from external data or other simulations
- Solve structural simulations all off the same unified model
  - Static (linear & non-linear)
  - Buckling (linear & non-linear)
  - Transient Dynamic
  - Linear Dynamics
  - Explicit (Bird Strike, Drop Test, Crash and Impact, etc.)
- Ease of post-processing of composite results

# Modeling Layered Composites Efficiently

## *What makes composites analysis challenging?*

- **Layered composites simulation requires:**
  - **Orthotropic material stiffness terms (E's, G's, nu's)**
  - **Lamina (ply) thickness**
  - **Fiber Orientation – draping**
  - **Understanding of stacking sequence implications (ABD matrix coupling, bending and twisting during cure)**
  - **Orthotropic strengths**
  - **Understanding of numerous potential modes of failure and the various failure theories**



# Modeling Layered Composites Efficiently

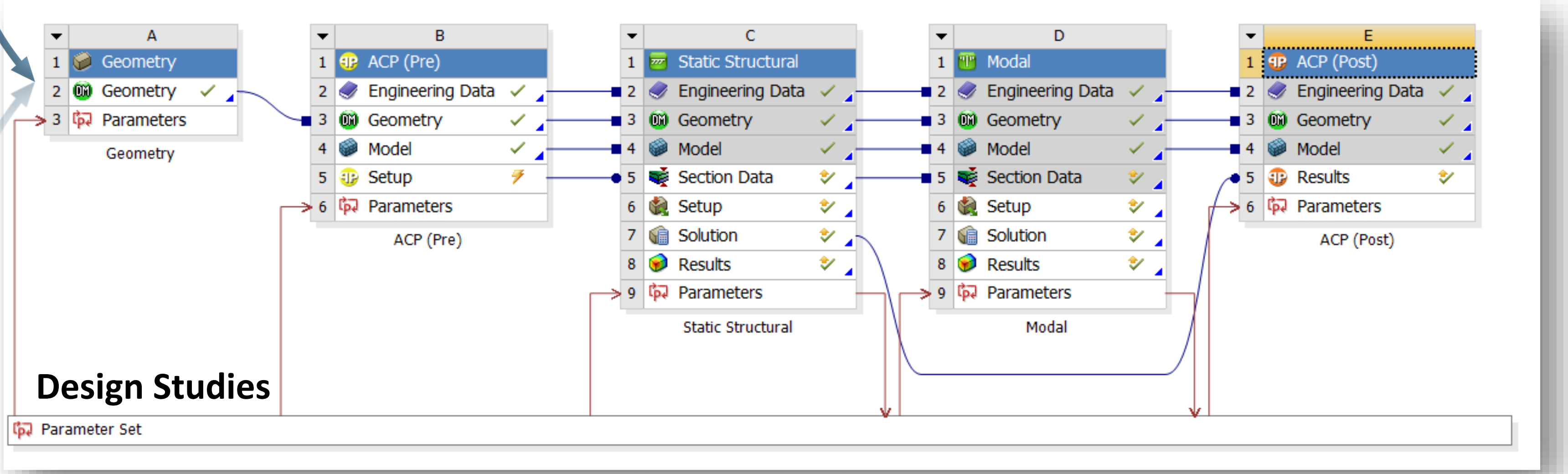
***How can ANSYS facilitate composite simulations ?***

- **Let's start with the workflow in ANSYS Workbench for incorporating composites in a typical design:**

Pro|ENGINEER
CoCreate Modeling
Autodesk Inventor
SolidWorks
CATIA
NX SIEMENS
Solid Edge

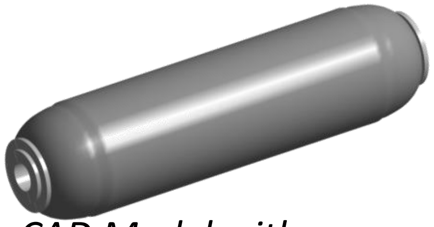
- Composite PrepPost integrated into Workbench workflow
- Upstream changes in design propagate through to the results
- In this example we might look at sensitivity of geometry size, shape or layups to deformation, natural frequencies, and failure margin

**Model/Cleanup** | **Materials, Layups & Mesh** | **B/Cs and Simulations** | **Composite Post-Processing**

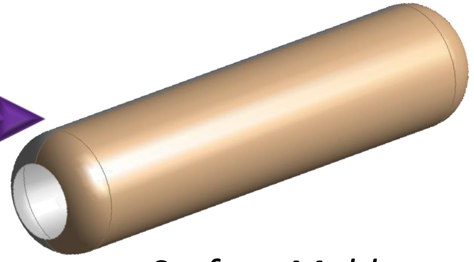



  July 5, 2018 

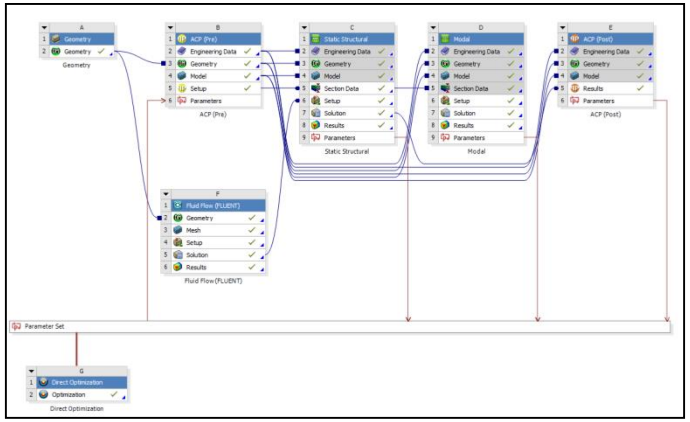
# From the CAD model to the composites design



CAD Model with
Arbitrary Thicknesses



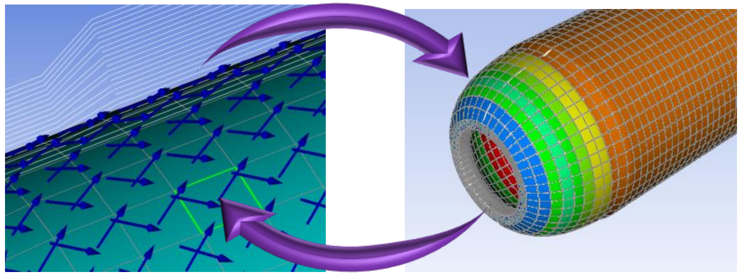
Surface Mold



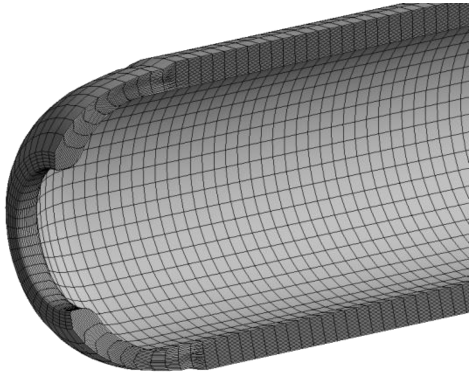
Ply definitions

Failure Analysis

Orientations
Thickness
Material
Location
Number of plies



**Iterate until good design is found**

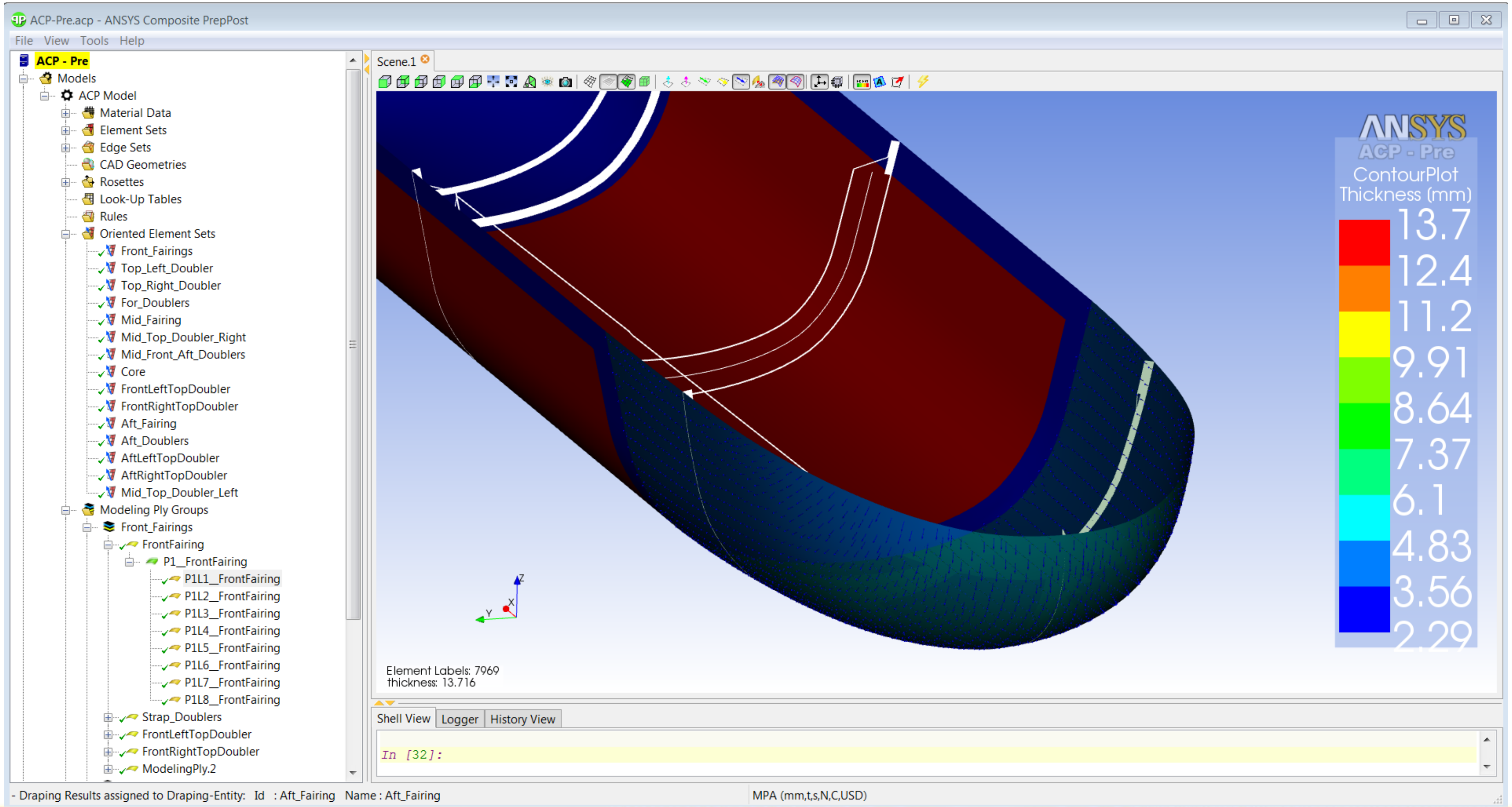


Final Model with Exact
Thicknesses

ANSYS

# Modeling Layered Composites Efficiently

***Composite PrepPost (ACP) is used to define the layup. Here we see the part thickness, cross-sections, and fiber orientations***



# Modeling Layered Composites Efficiently

***Make changes to layup, ply locations, orientations***

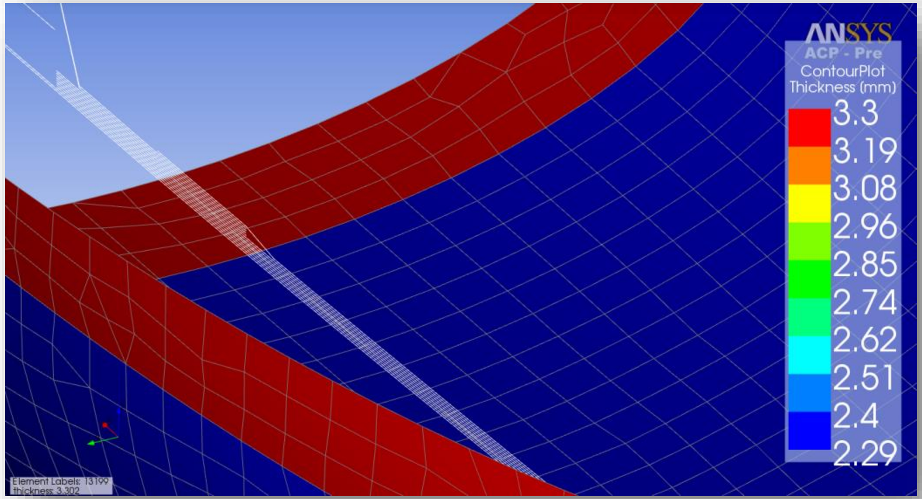
**Here we change the width of the doubler shown in red**

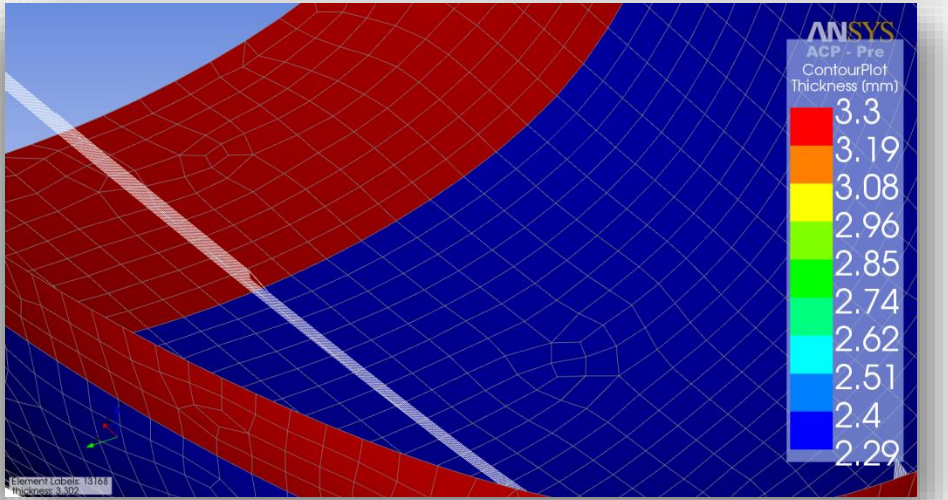
Outline of All Parameters

| | A | B | C | D |
|---|---|---|---|---|
| 1 | ID | Parameter Name | Value | Unit |
| 2 | ⊟ Input Parameters | | | |
| 3 | ⊟ Radome Geometry with Spot Welds (A1) | | | |
| 4 | P1 | front_doubler_width | 50 | |
| * | New input parameter | New name | New expression | |
| 6 | ⊟ Output Parameters | | | |
| 7 | ⊟ Static Structural (C1) | | | |
| 8 | P2 | Total Deformation Maximum | ⚡ | in |
| * | New output parameter | | New expression | |
| 10 | Charts | | | |

Outline of All Parameters

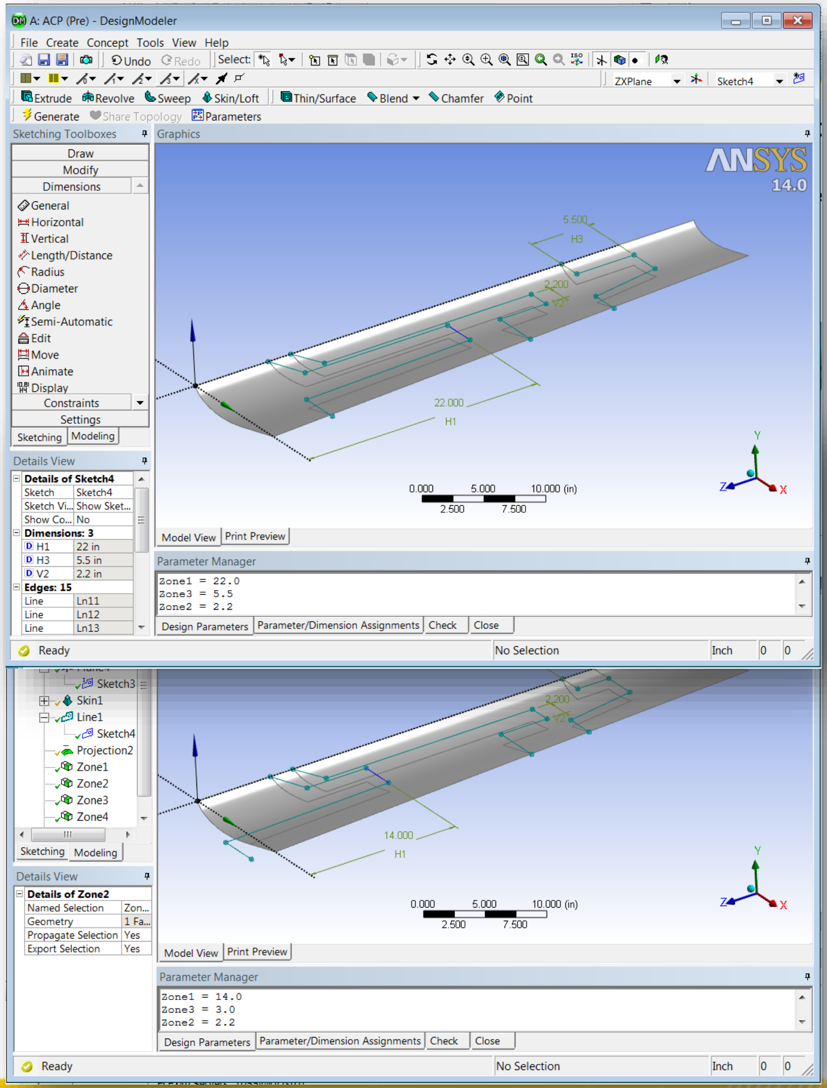
| | A | B | C | D |
|---|---|---|---|---|
| 1 | ID | Parameter Name | Value | Unit |
| 2 | ⊟ Input Parameters | | | |
| 3 | ⊟ Radome Geometry with Spot Welds (A1) | | | |
| 4 | P1 | front_doubler_width | 200 | |
| * | New input parameter | New name | New expression | |
| 6 | ⊟ Output Parameters | | | |
| 7 | ⊟ Static Structural (C1) | | | |
| 8 | P2 | Total Deformation Maximum | ⚡ | in |
| * | New output parameter | | New expression | |
| 10 | Charts | | | |

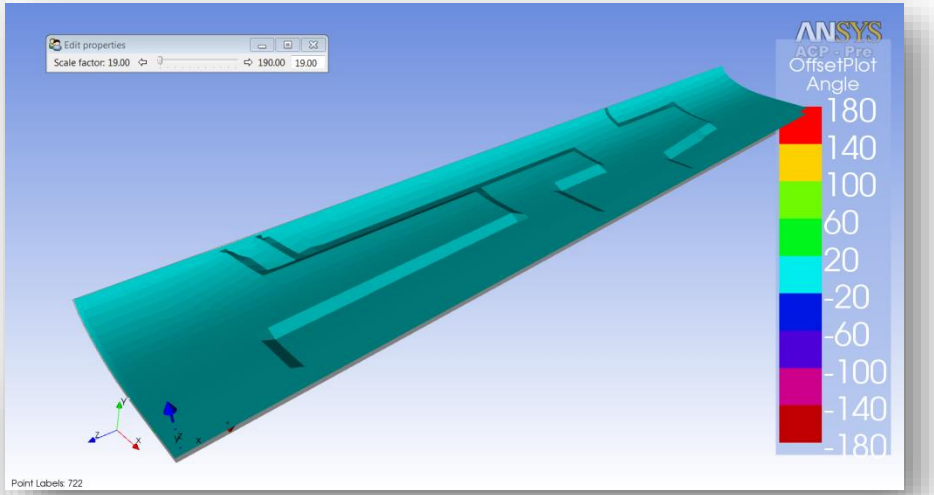


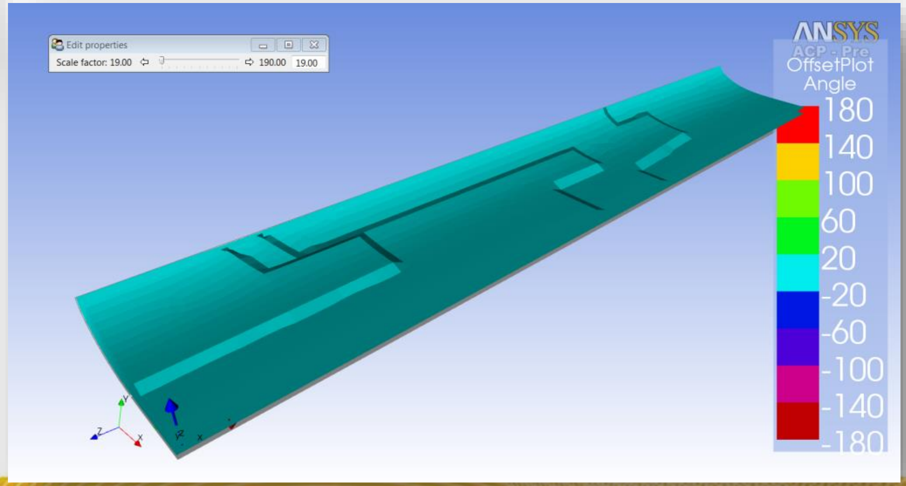



# Modeling Layered Composites Efficiently

***Make parametric changes to ply dimensions***



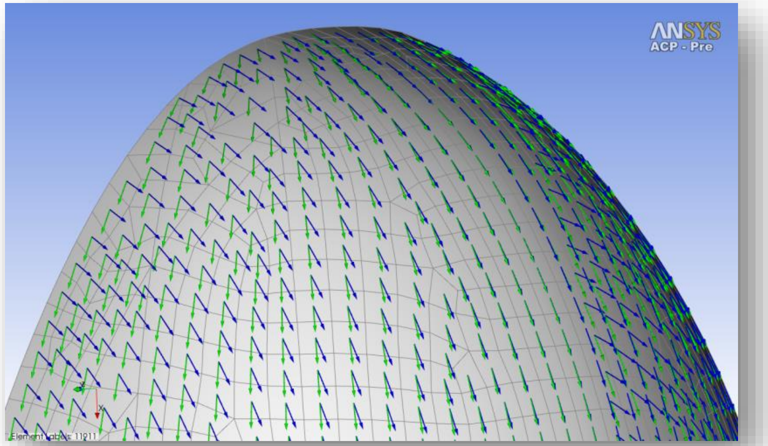



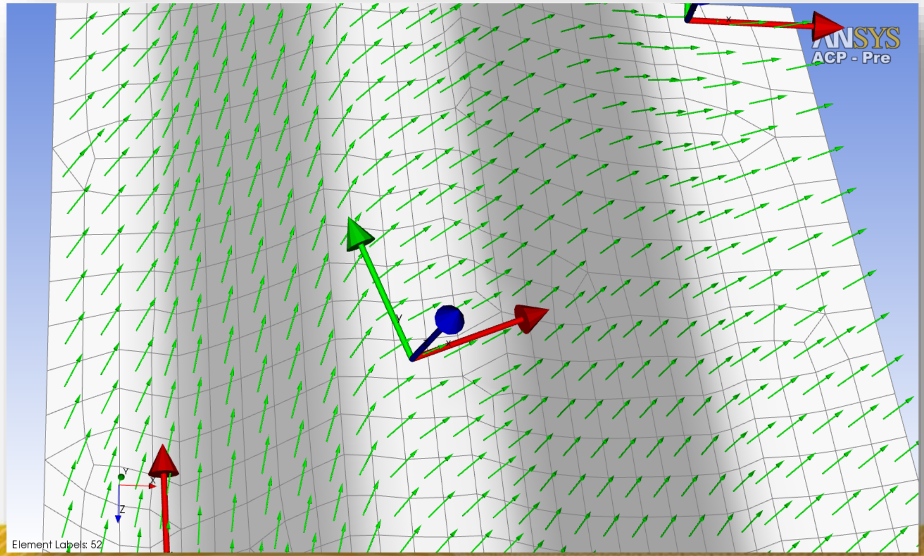
# Modeling Layered Composites Efficiently

## *Fiber orientation prediction and modification*

- Use internal draping calculations
- Interface with Siemen's FiberSim
- Modify the fiber orientation to something that is actually observed on the manufacturing floor.
- For this we use multiple rosettes (local coordinate systems) specifying the angle to match the measured at known locations, and let PrepPost interpolate in between.



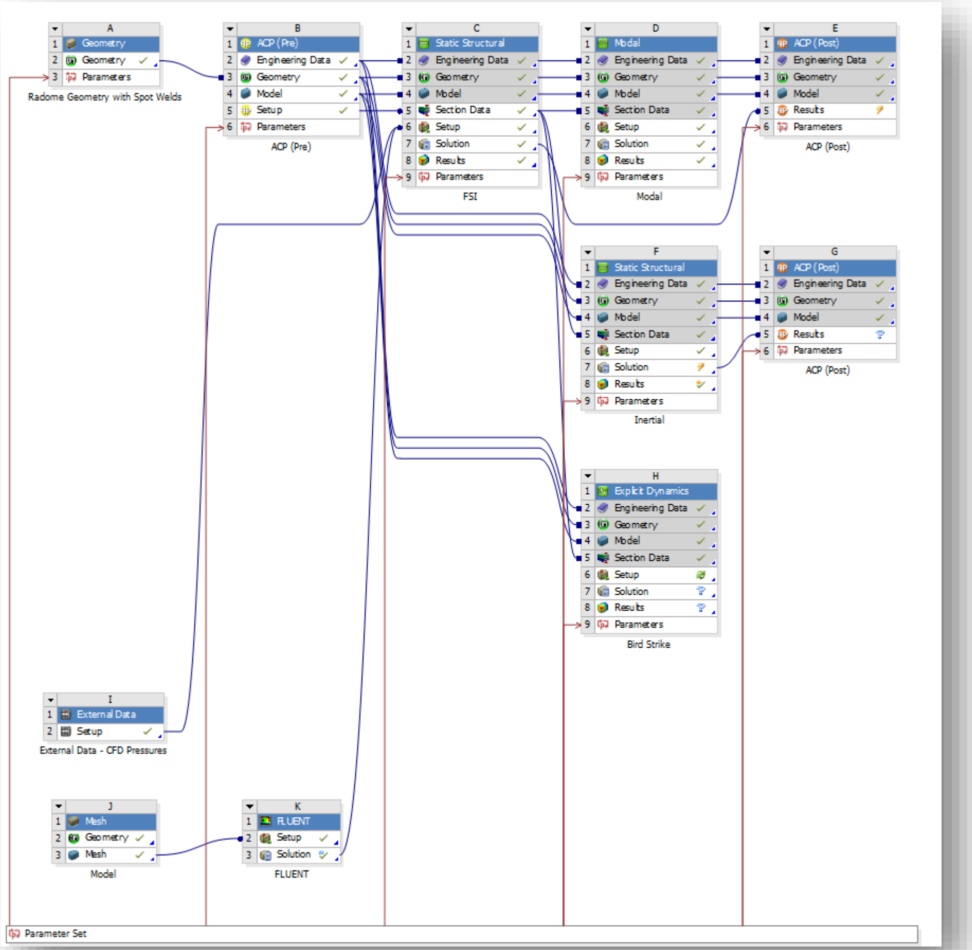

**Green – parallel to rosette,** Blue**– draped prediction**

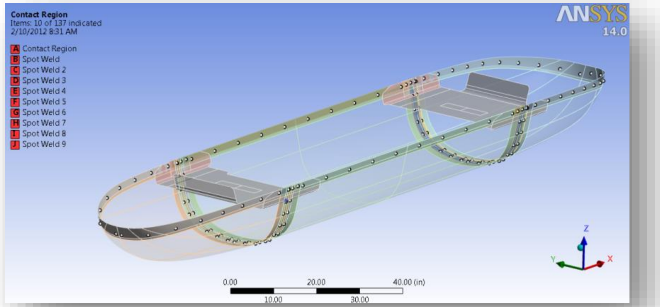


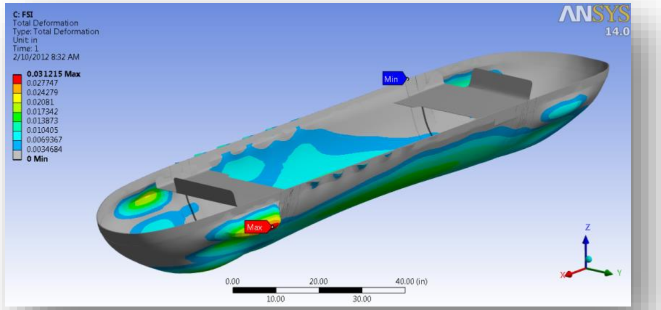
# Modeling Layered Composites Efficiently

## *Radome Case Study*

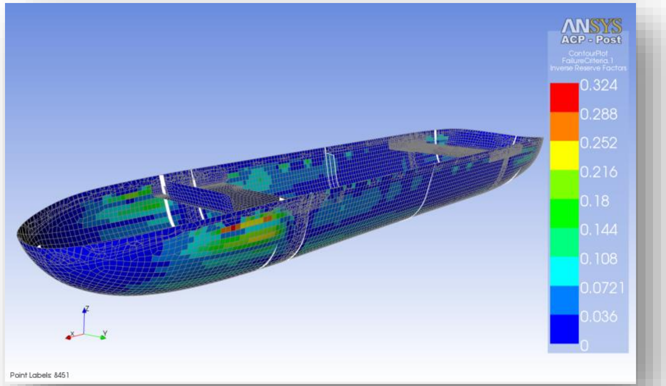




Connections



Deformations from FSI

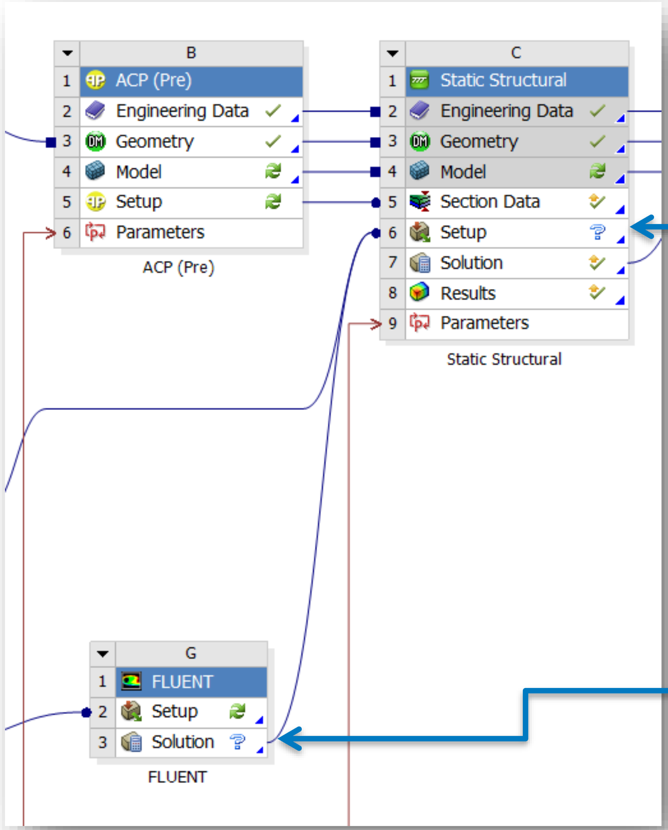


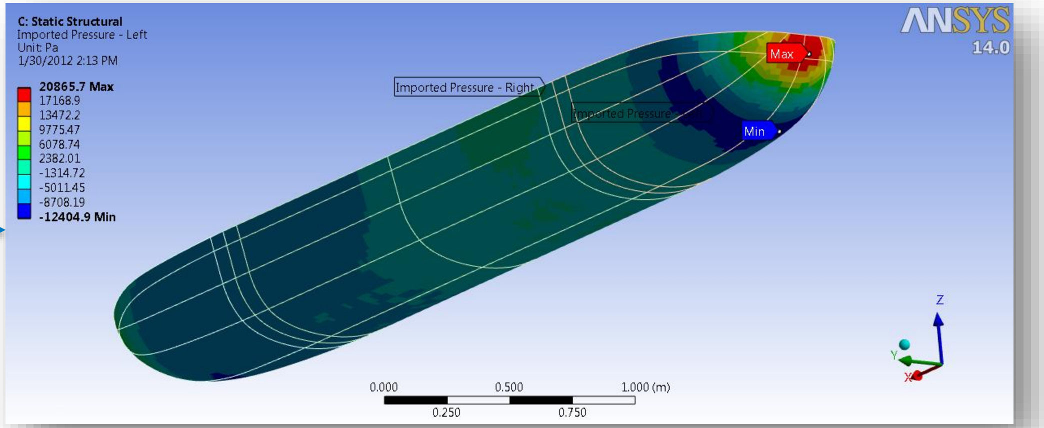
Failure Criteria

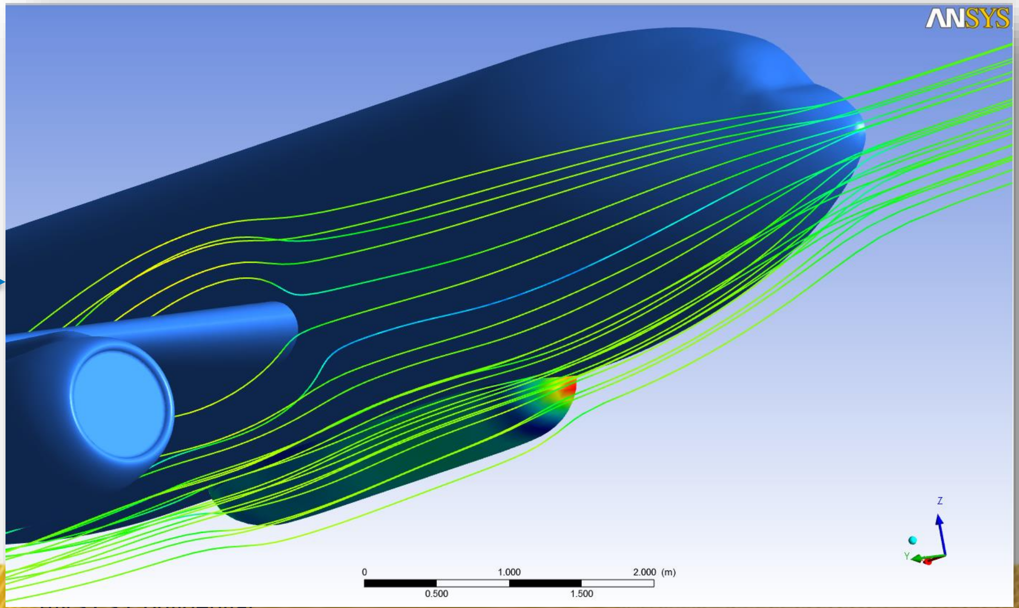
   

# Modeling Layered Composites Efficiently

***Incorporate fluid/thermal loading directly from CFD (1-way and 2-way FSI)***

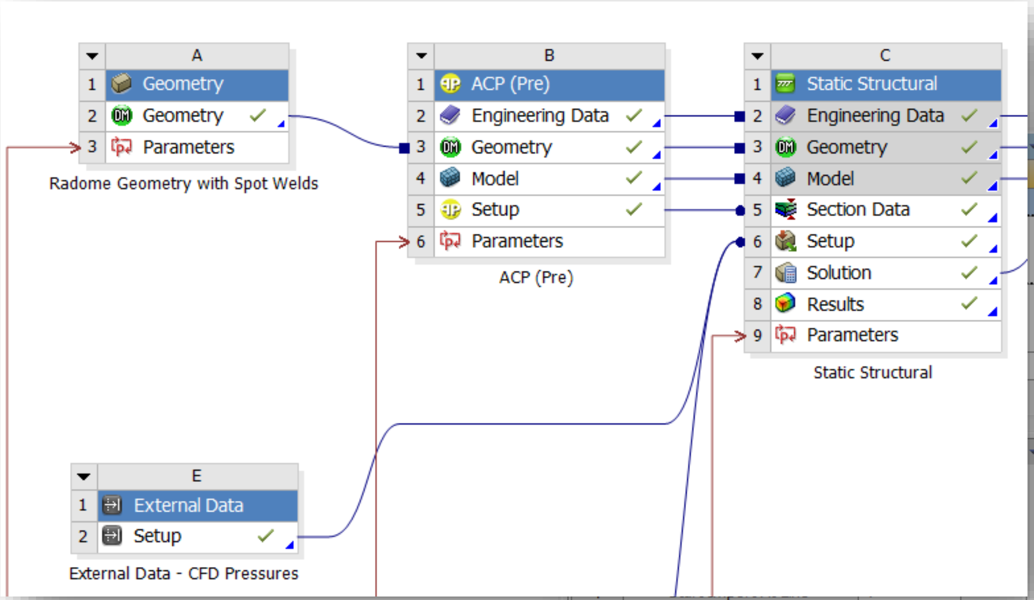


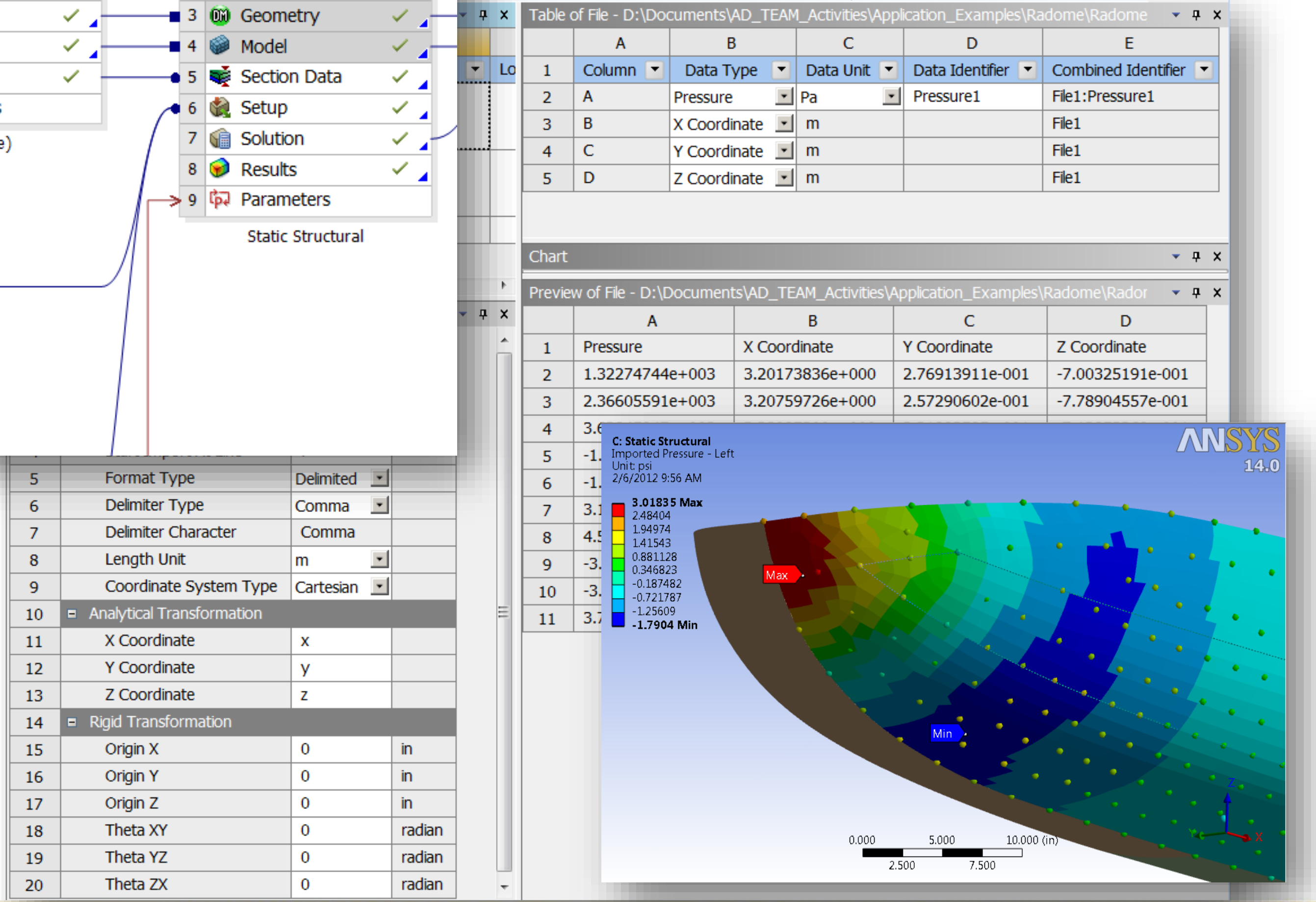



# Modeling Layered Composites Efficiently

## *Incorporate, transform, and validate loading from other simulations*




**Data Mapping**

- Pressure
- Temperature
- Heat Transfer
- Thickness
- Displacements
- Many more..

# Modeling Layered Composites Efficiently

## *Perform what-if studies*

Here we look at how the tip deflection and 2$^{nd}$ natural frequency change with the 1$^{st}$ rib spanwise location.

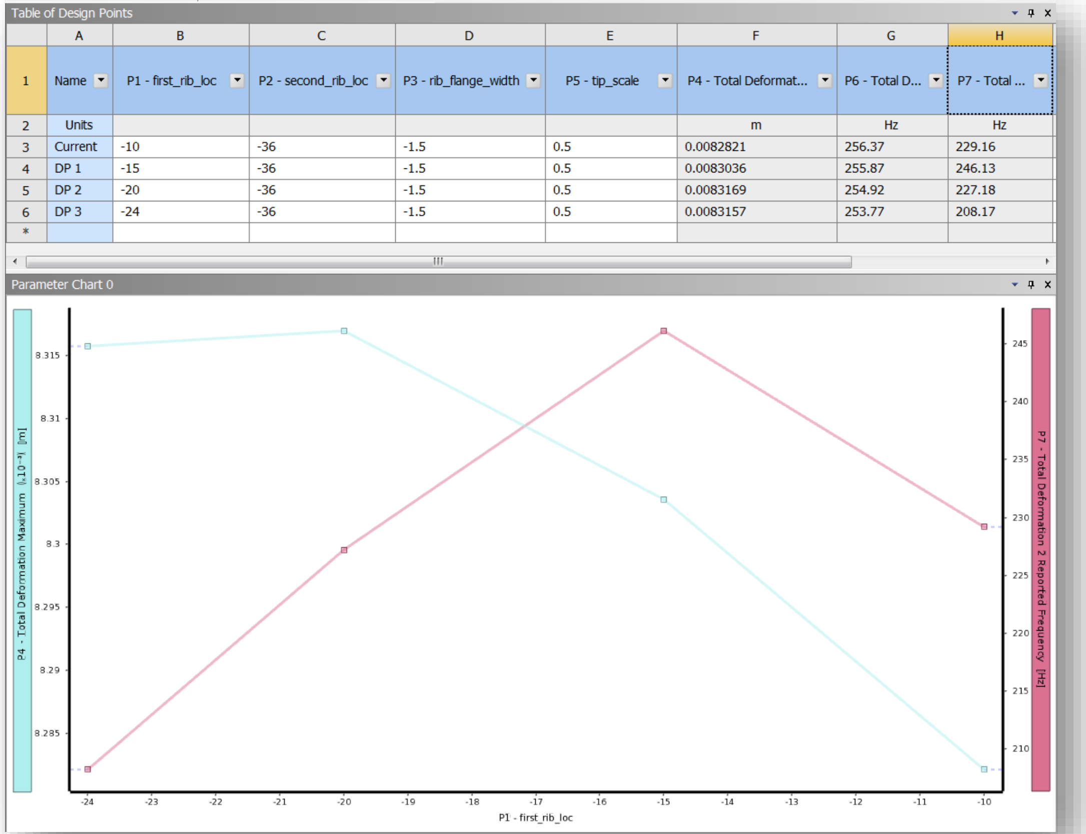
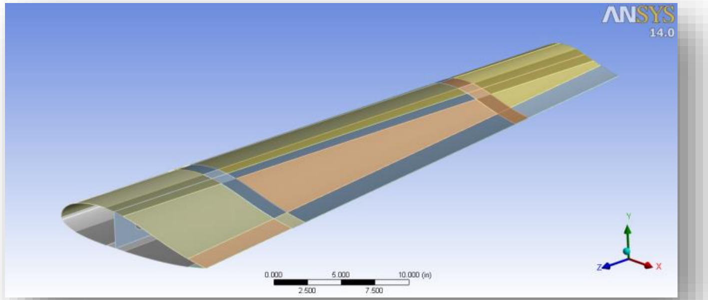


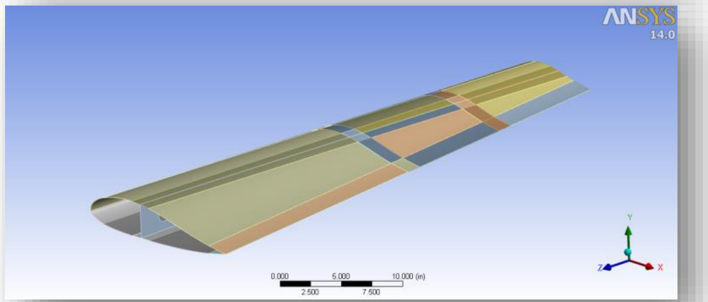
Table of Design Points

| | A | B | C | D | E | F | G | H |
|---|---|---|---|---|---|---|---|---|
| 1 | Name | P1 - first_rib_loc | P2 - second_rib_loc | P3 - rib_flange_width | P5 - tip_scale | P4 - Total Deformat... | P6 - Total D... | P7 - Total ... |
| 2 | Units | | | | | m | Hz | Hz |
| 3 | Current | -10 | -36 | -1.5 | 0.5 | 0.0082821 | 256.37 | 229.16 |
| 4 | DP 1 | -15 | -36 | -1.5 | 0.5 | 0.0083036 | 255.87 | 246.13 |
| 5 | DP 2 | -20 | -36 | -1.5 | 0.5 | 0.0083169 | 254.92 | 227.18 |
| 6 | DP 3 | -24 | -36 | -1.5 | 0.5 | 0.0083157 | 253.77 | 208.17 |
| * | | | | | | | | |

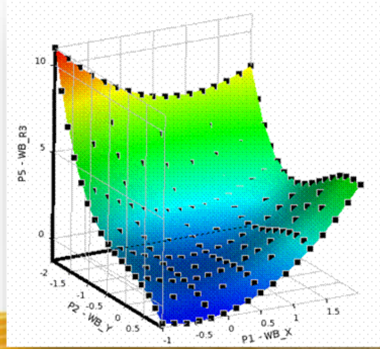










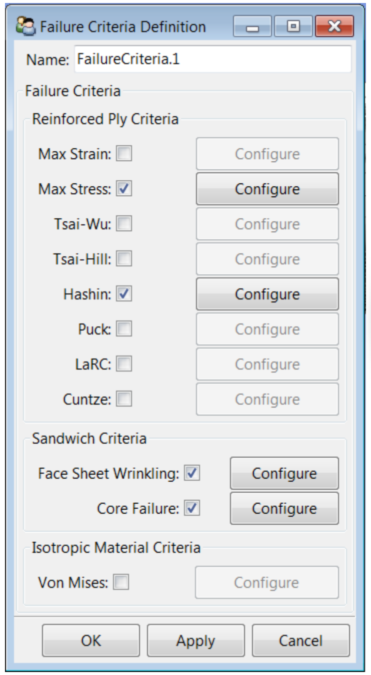


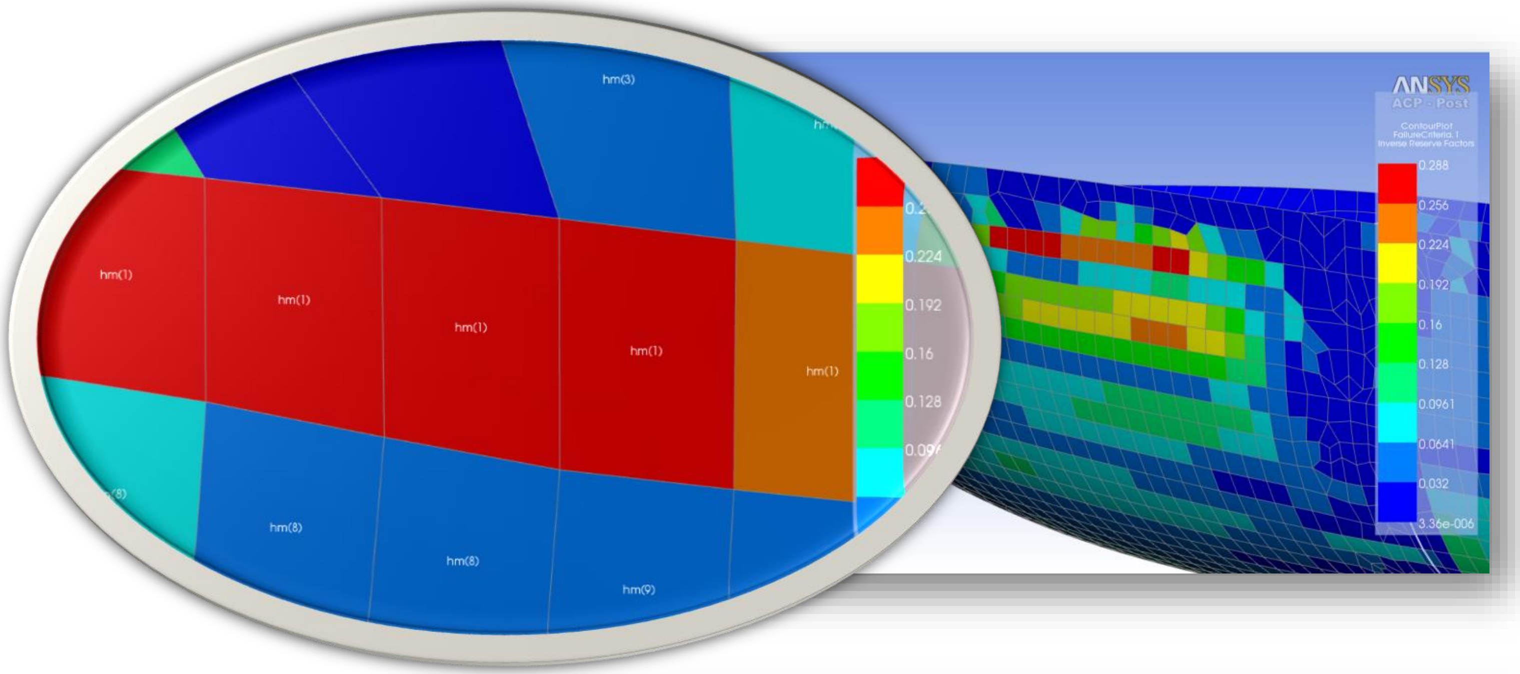


*Create response surfaces for optimization*

# Modeling Layered Composites Efficiently

## *Ease of post-processing of composite results*



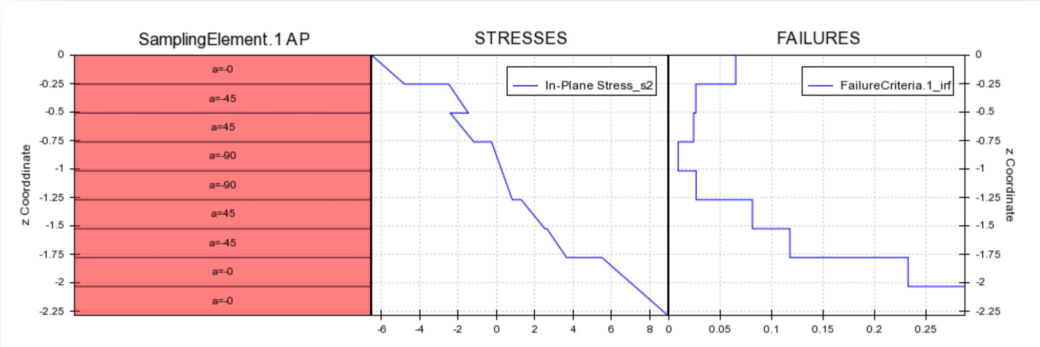




**Results Interrogation**

•Worst case failure criteria over all layers shows

•Hm (1) – **H**ashin **m**atrix – Layer **1**

•Pick element and see stress/strain and failure through the thickness



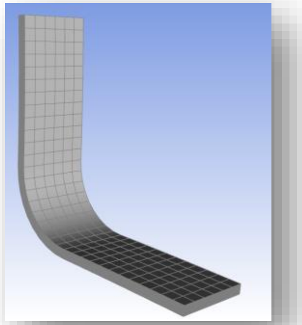

ANSYS

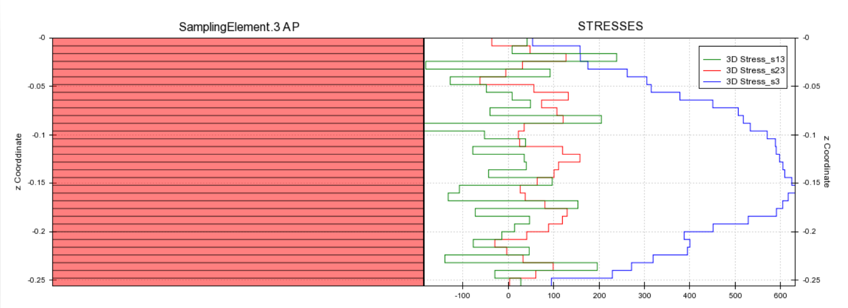
# Modeling Layered Composites Efficiently

## *Ease of post-processing of composite results*

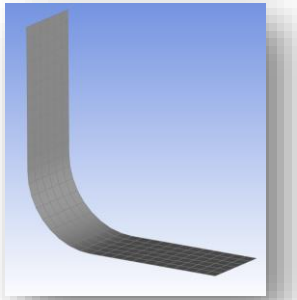
**Make Thru-Thickness Stress Plots and account for Interlaminar normal stresses with shells**

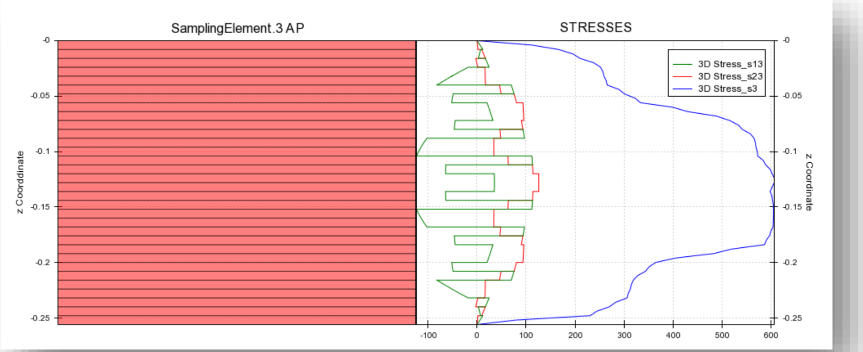
- Below we look at the stress through the thickness in the bend
- Traditional shell approaches can not account for interlaminar normal stress (shown in blue curve)
- ANSYS Composite PrepPost can predict these base on the work of Roos, Kress & Ermanni
- Make more accurate assessments without the need for large 3D models






**Thru-Thickness Stress Plots – Solids**






**Thru-Thickness Stress Plots – Shells**

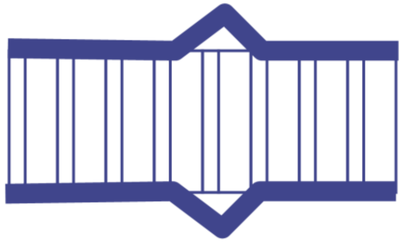
   

# Modeling Layered Composites Efficiently

## *Ease of post-processing of composite results*

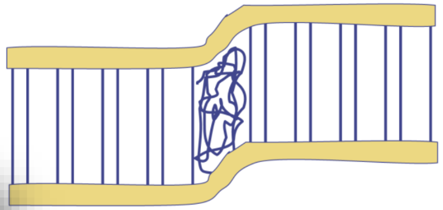
### Wrinkling

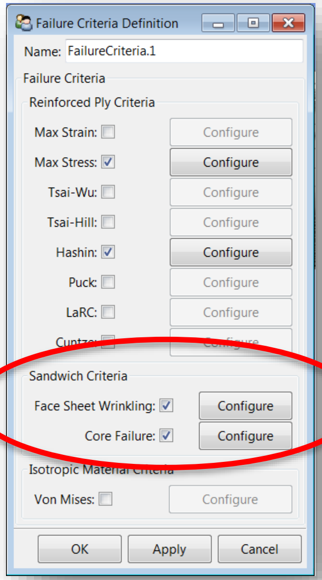
- Local buckling of a face sheet under compression
- Failure indicator available using shell  modeling of sandwich

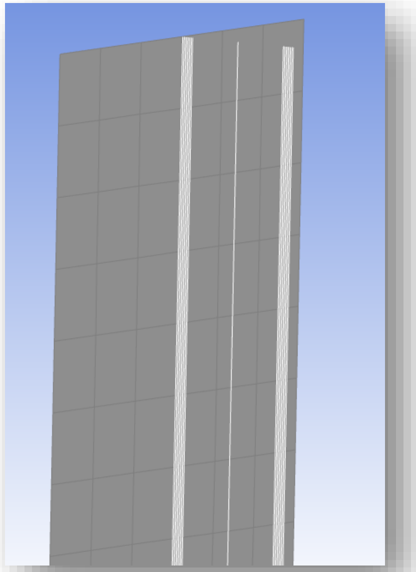


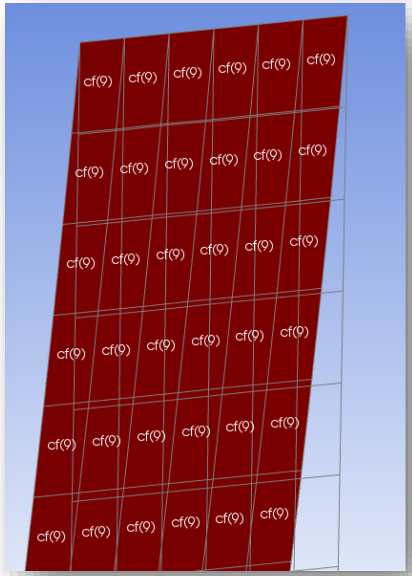
### Core Failure

- Local failure of core in shear or tensile loading
- Failure indicator available using shell  modeling of sandwich

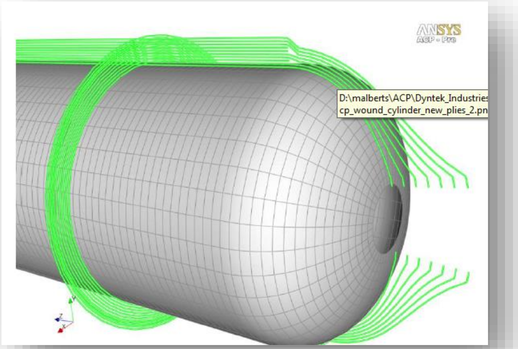


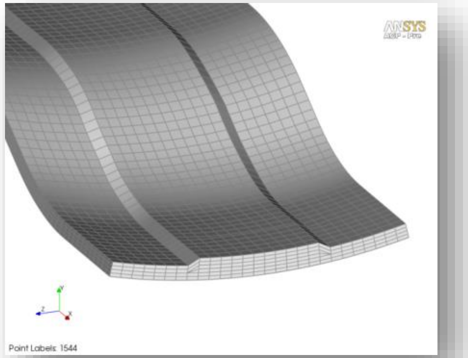




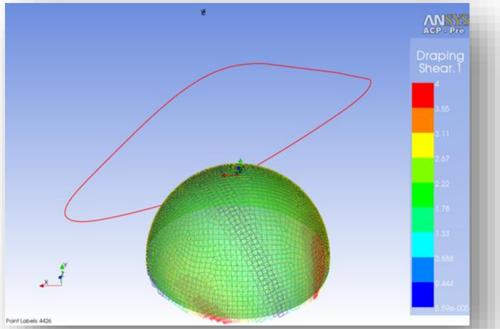
# Modeling Layered Composites Efficiently

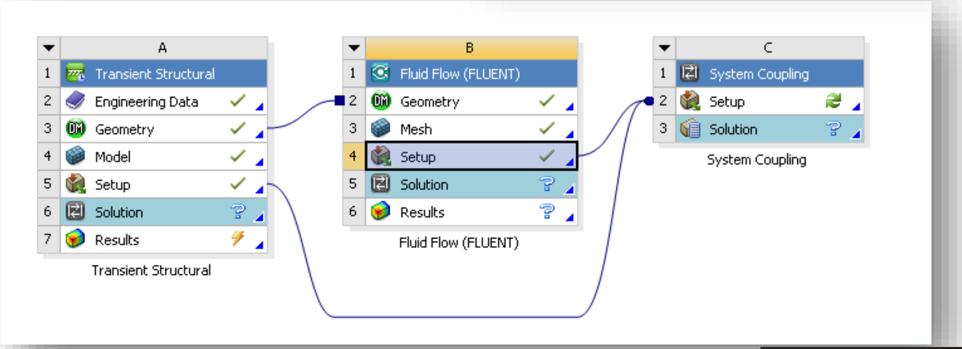





## *Even more..*

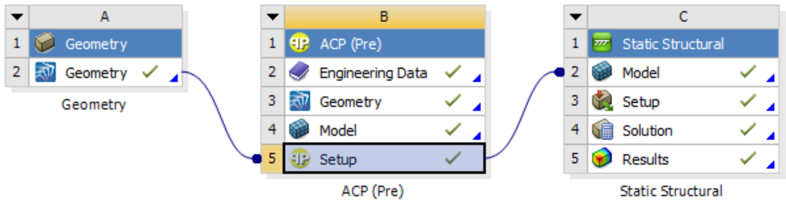
- •3D Curves used to guide fiber orientation
- •Fiber orientation using interpolation between known locations (Superposed method)
- •Flat pattern prediction
- •Right click suppress parts or plies
- •Map composite thickness from 3D CAD
- •Complex Model Assembly with contacts
- •Customize layups using scripts and tables (example: Filament Winding)
- •Python Scriptable
- •1 and 2 Way FSI with composites via System Coupling



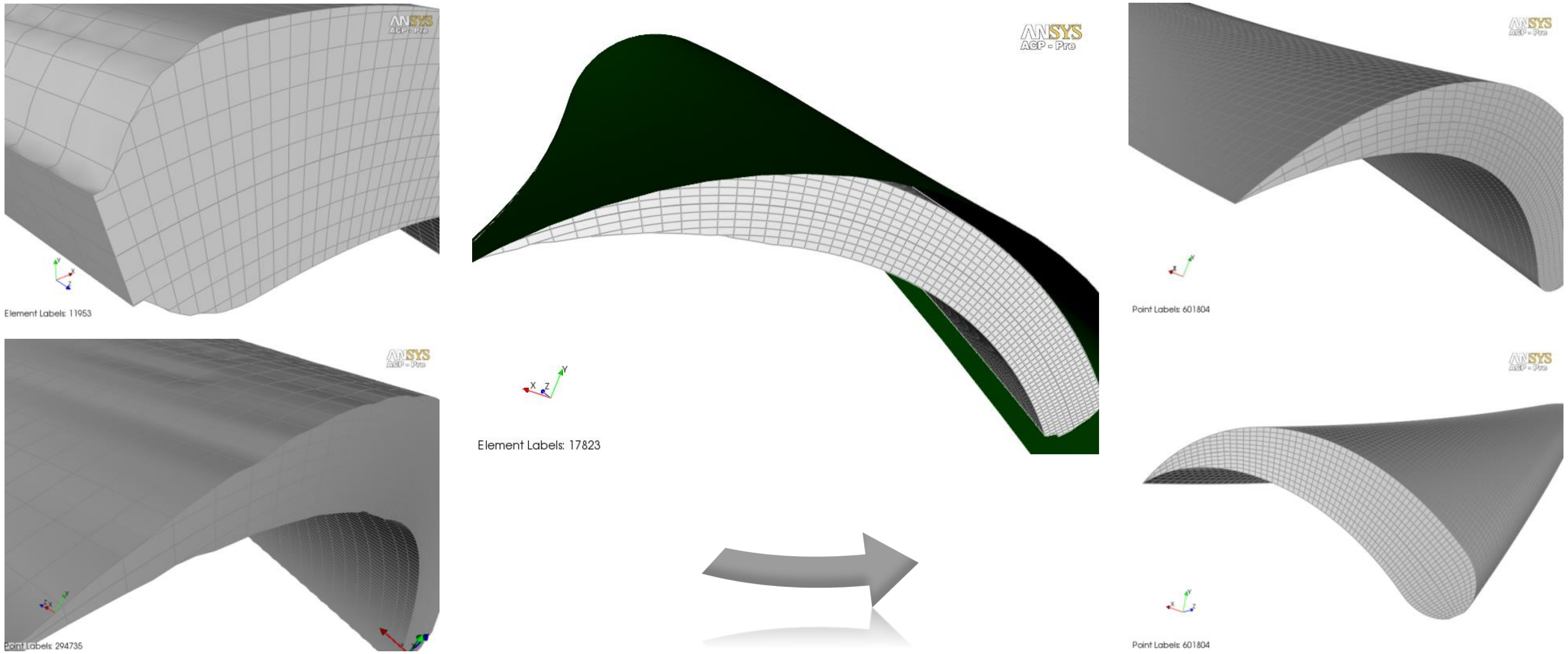

  

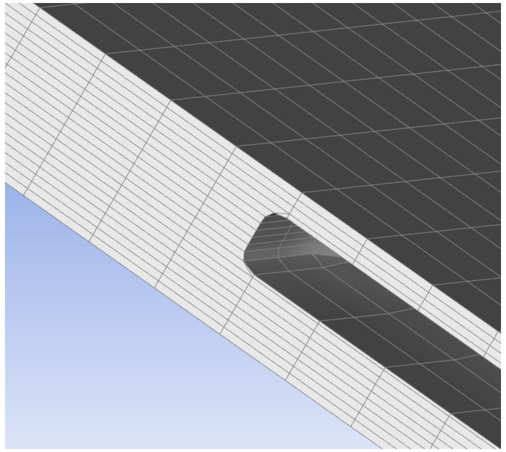
# Modeling Layered Composites Efficiently




## Solid Modeling of Complex Geometries

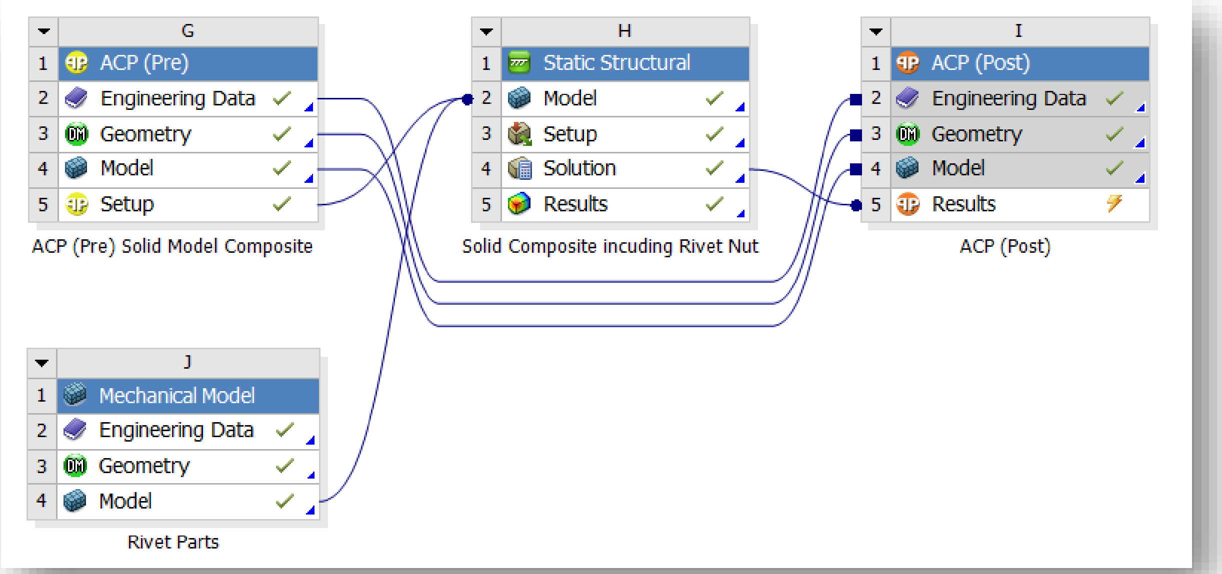
- Use complex geometry to cut, guide and smooth layup

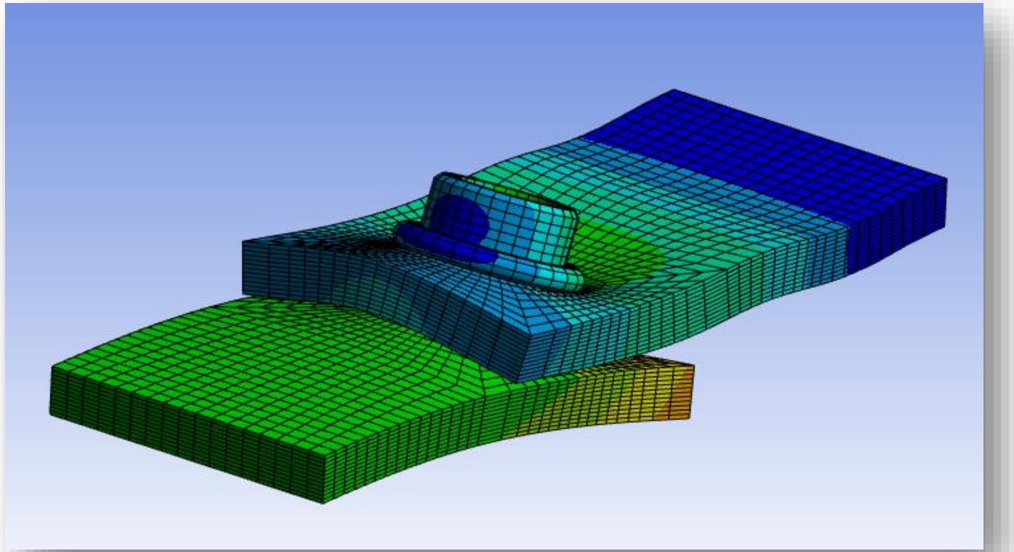


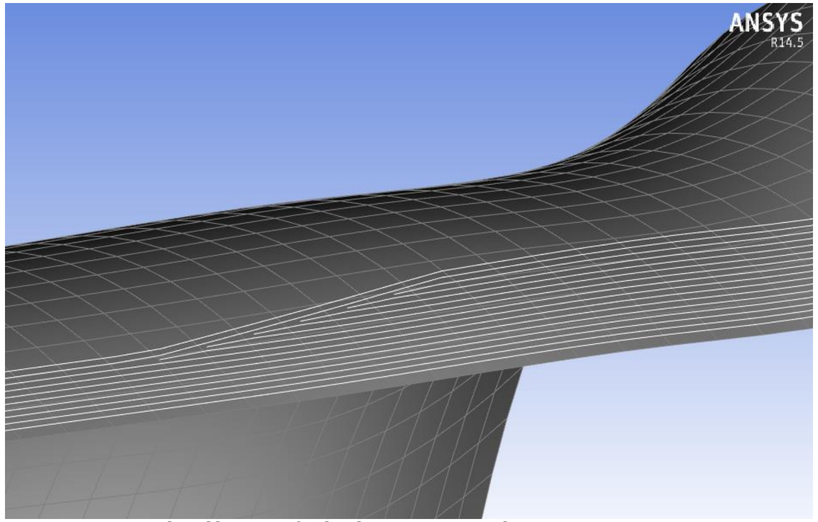
  July 5, 2018 

# Solid Composite Workflow – Model Assembly

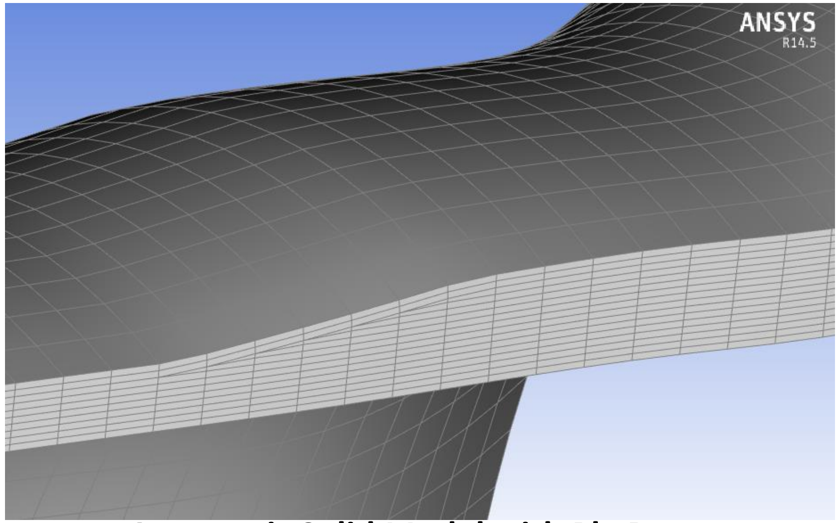


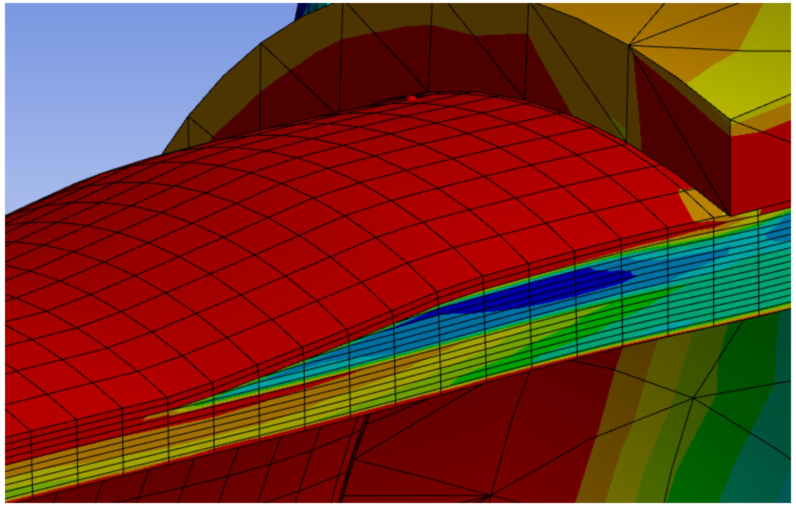
# Solid Composite Workflow (Thermal & Structural)



Shell Model shown with Section Cut



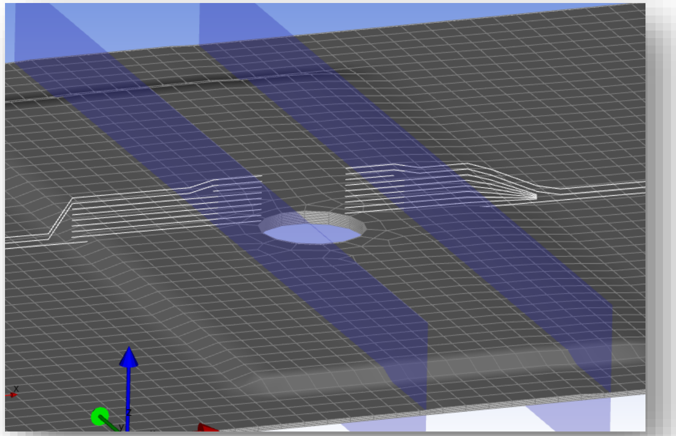
Automatic Solid Model with Ply Drops

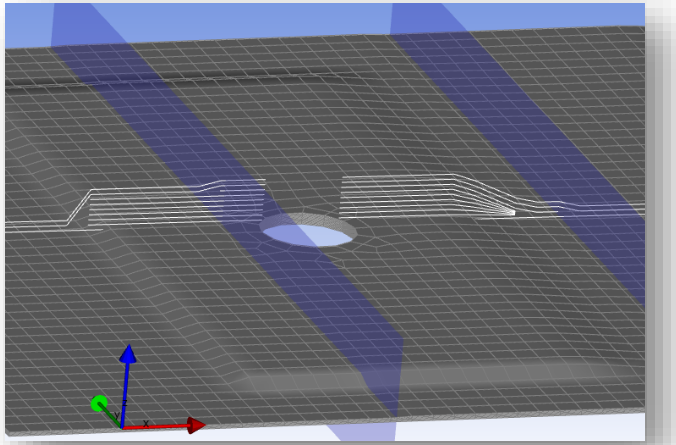


Solid Model Passes to ANSYS Mechanical

# Parametric Studies of Layup

- **Ply Drop off location and angles are varied via table**
- **Solid Model gets reconstructed**
- **Results Updated - *Automatically***





Outline: No data

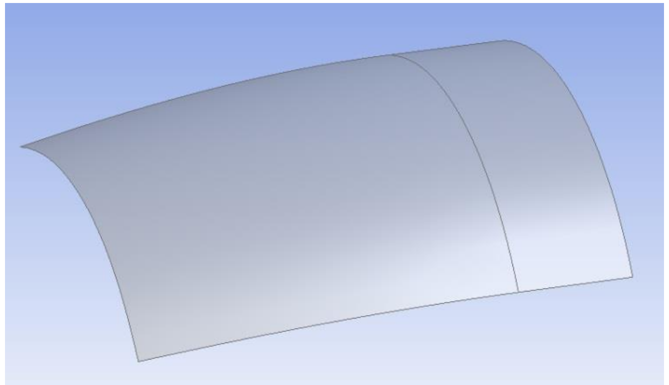
| | A | B | C |
|---|---|---|---|
| 1 | ID | Parameter Name | Value |
| 2 | Input Parameters | | |
| 3 | ACP (Pre) (A1) | | |
| 4 | P1 | ParallelRule.1.pos_distance | 2 |
| 5 | P2 | ModelingPly.1.ply_angle | 30 |
| 6 | P3 | ModelingPly.2.ply_angle | 45 |
| * | New input parameter | New name | New expressi |
| 8 | Output Parameters | | |
| 9 | Static Structural (B1) | | |
| 10 | P4 | Total Deformation Maximum | 0.13313 |
| * | New output parameter | | New expressi |
| 12 | Charts | | |

Table of Design Points

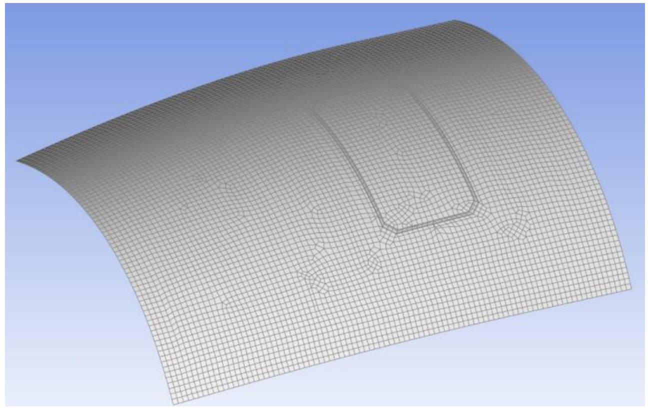
| | A | B | C | D | E | F | G |
|---|---|---|---|---|---|---|---|
| 1 | Name | P1 - Par... .1 .po... | P2 - Mode... .1 .ply_a... | P3 - Modelin... .2 .ply_an... | P4 - Total Deformation Maximum | Exported | Note |
| 2 | Units | in | None | None | in | | |
| 3 | Current | 2 | 30 | 45 | 0.13313 | | |
| 4 | DP 1 | 2 | 45 | 30 | 0.18103 | | |
| 5 | DP 2 | 3 | 30 | 90 | 0.11064 | | |
| * | | | | | | | |

ANSYS

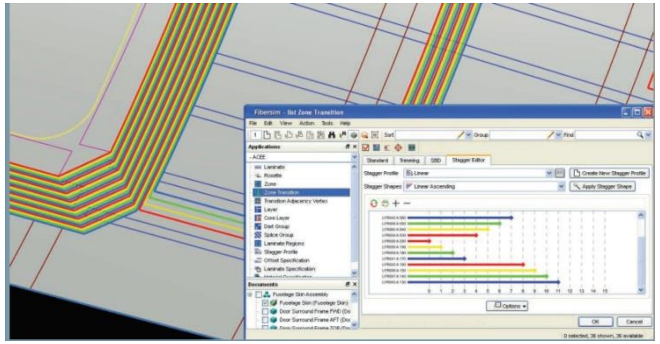
# FiberSim and ACP Workflow



CAD Geometry

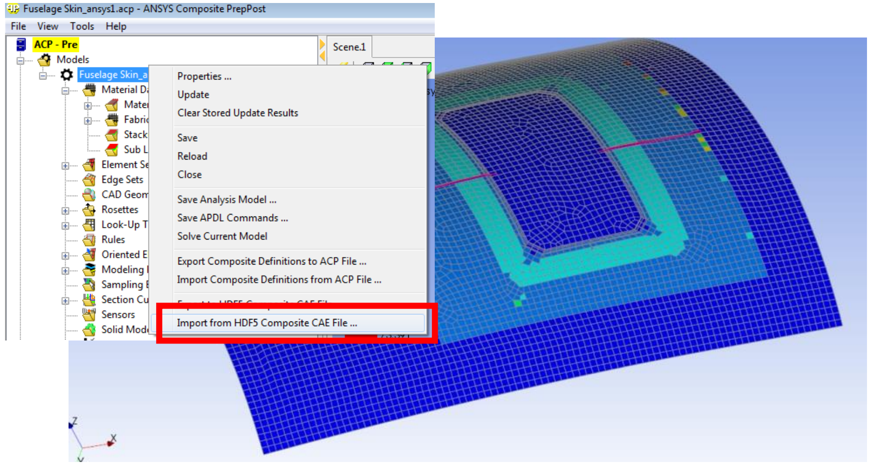


ACP Mesh



Siemens FiberSim

HDF5 Export

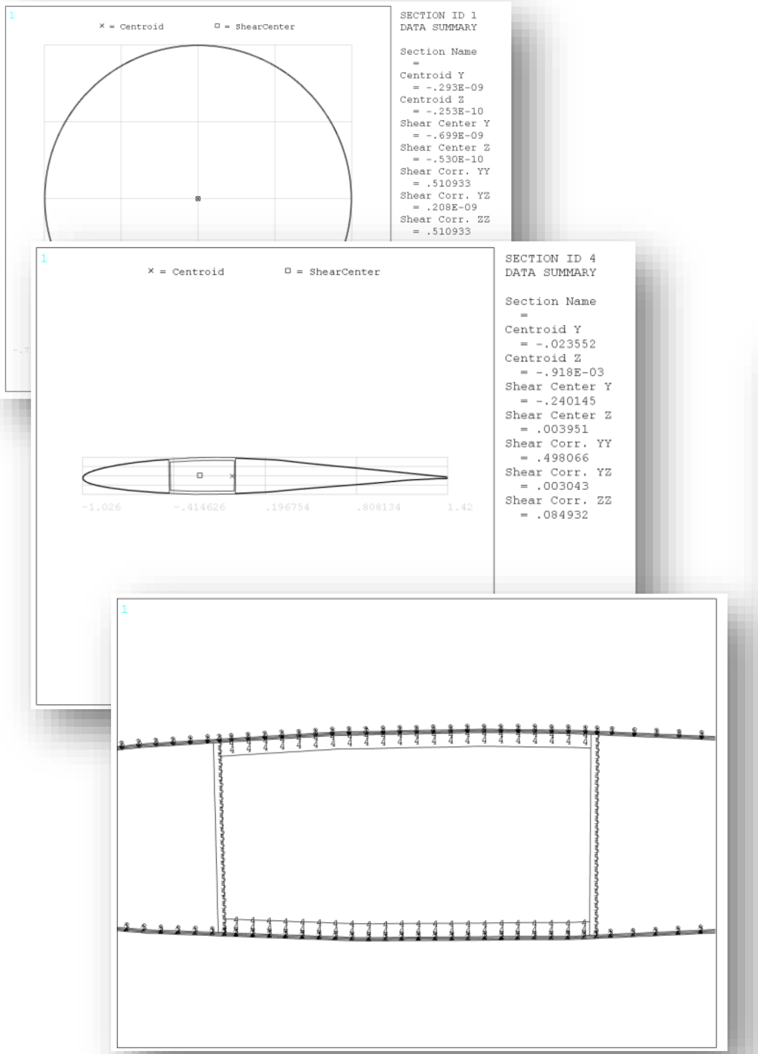


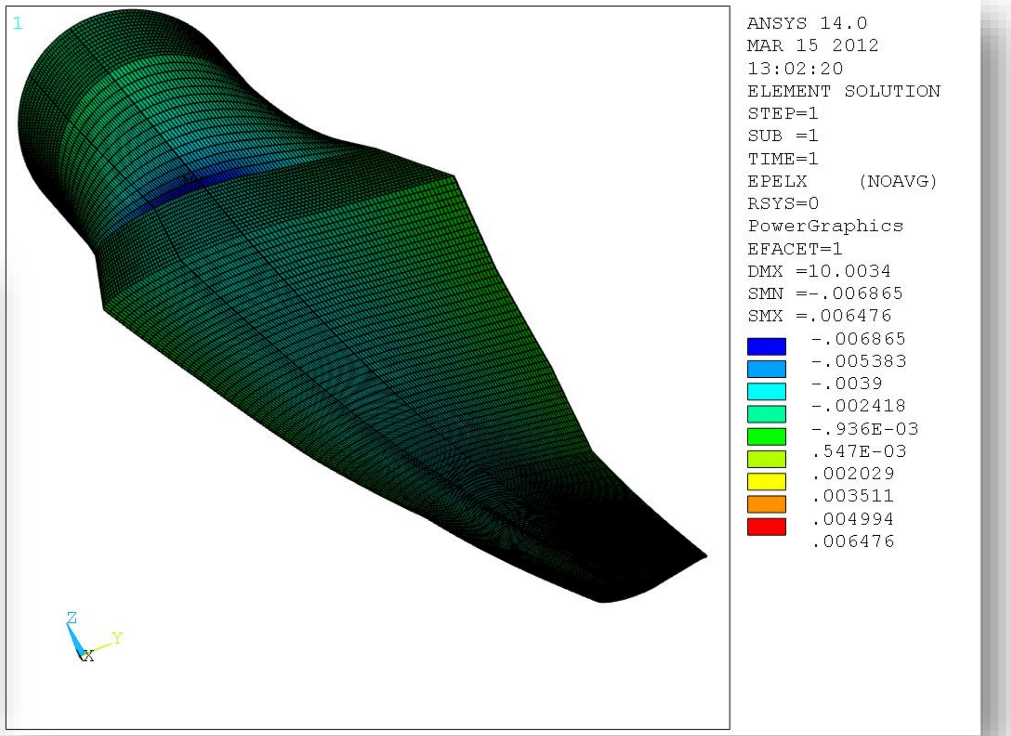

ACP Setup

# Beam Modeling – Advanced Capabilities

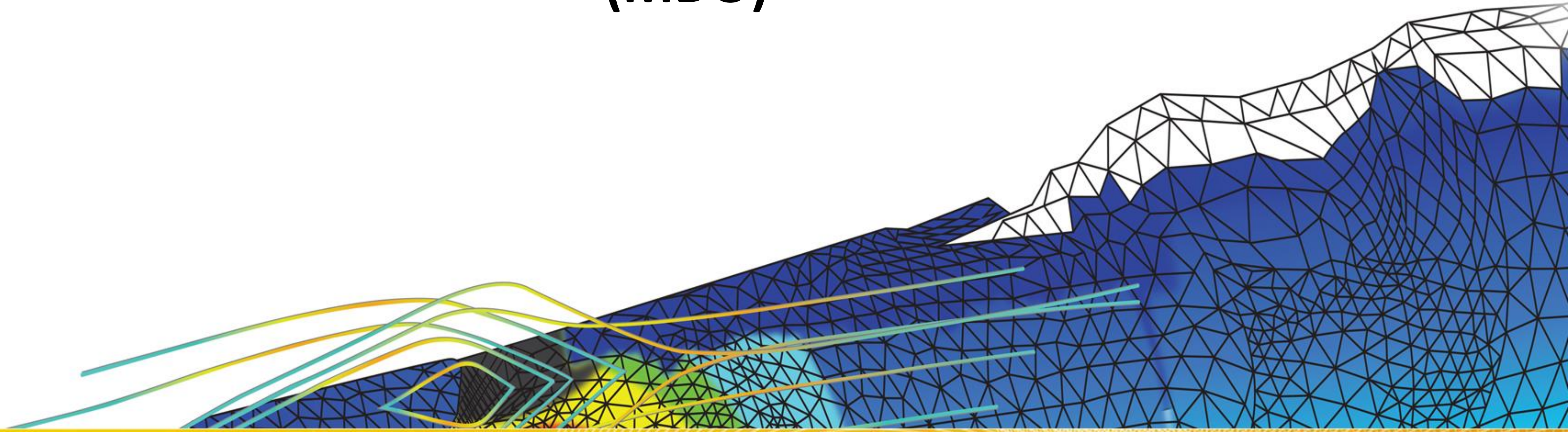
- User Defined Sections
- Composite Sections
- Tapering
- Results (Disp, Strain, Stress) shown on Beam



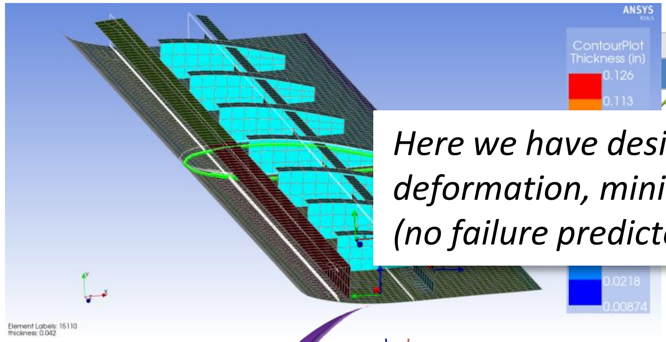



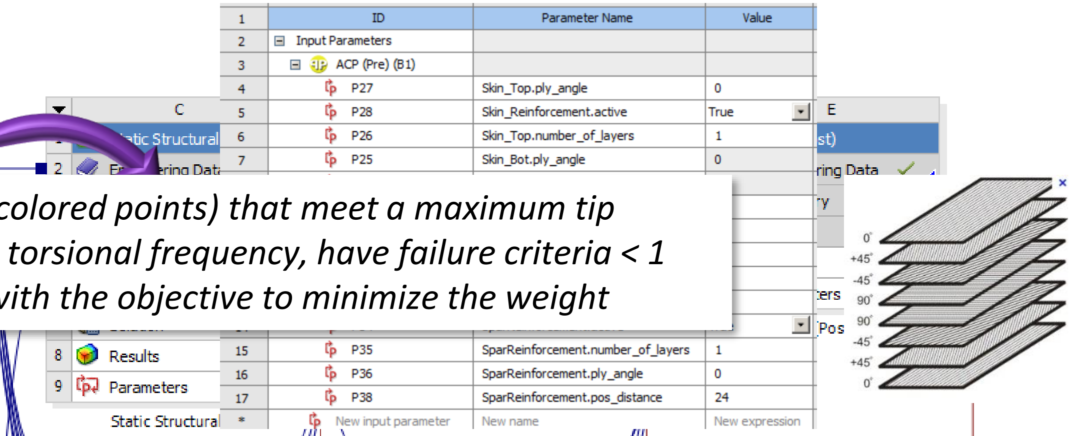
# Multidisciplinary Design Optimization (MDO)



# Composite Optimization with FSI



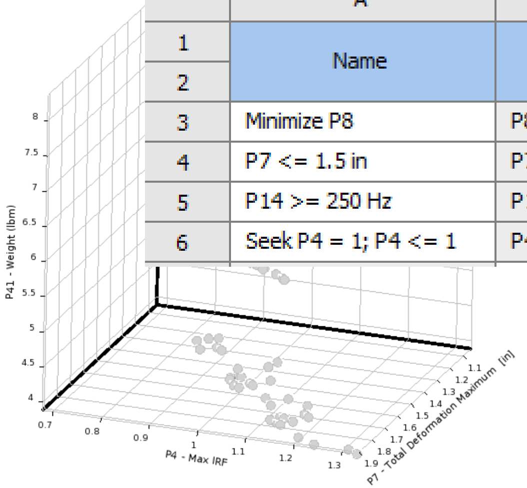

Specify Initial Design in Composite



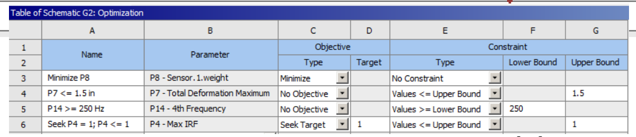

*Here we have designs (colored points) that meet a maximum tip deformation, minimum torsional frequency, have failure criteria < 1 (no failure predicted), with the objective to minimize the weight*

Table of Schematic G2: Optimization

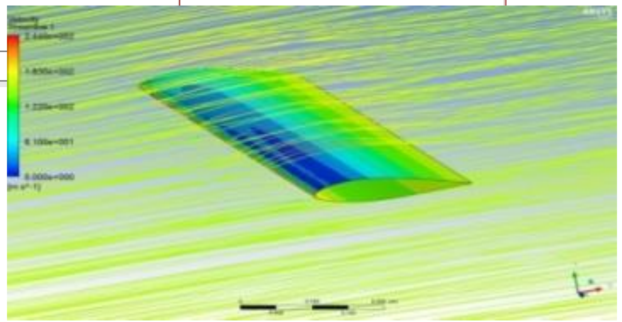
| | A | B | C | D | E | F | G |
|---|---|---|---|---|---|---|---|
| 1 | Name | Parameter | Objective | | Constraint | | |
| 2 | | | Type | Target | Type | Lower Bound | Upper Bound |
| 3 | Minimize P8 | P8 - Sensor.1.weight | Minimize | | No Constraint | | |
| 4 | P7 <= 1.5 in | P7 - Total Deformation Maximum | No Objective | | Values <= Upper Bound | | 1.5 |
| 5 | P14 >= 250 Hz | P14 - 4th Frequency | No Objective | | Values >= Lower Bound | 250 | |
| 6 | Seek P4 = 1; P4 <= 1 | P4 - Max IRF | Seek Target | 1 | Values <= Upper Bound | | 1 |




Have Design Explorer find Candidate Objective Solutions

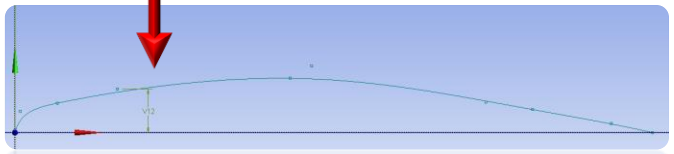


Put Constraints on Acceptable Frequencies, Displacements and Failure Criteria

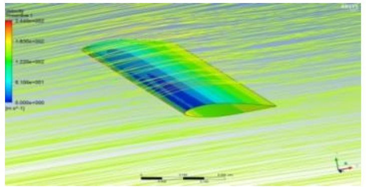


Add Fluid Flow from Fluent/CFX

# MDO of Aerostructural systems



*Original Airfoil*



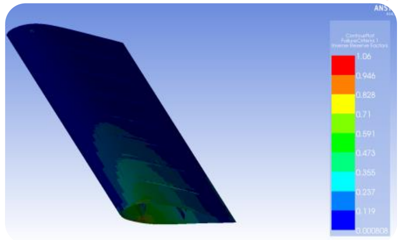
*CFD-FSI*

*Modified Airfoil*

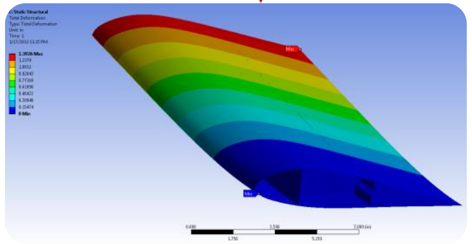
*User types in the value of the parameter, and ANSYS updates the models automatically and solves for the requested results*

Table of Design Points

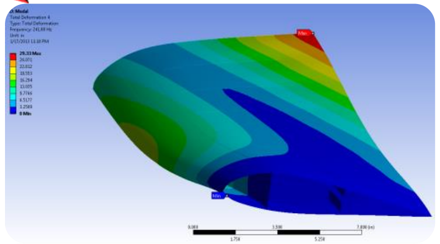
| | P44 - FoilTopH | P46 - AOA | P47 - Mach | P4 - Max IRF | P7 - Total Deformation Maximum | P14 - 4th Frequency | P41 - Weight (lbm) | P42 - Drag | P43 - Lift | P45 - Lift to Drag Ratio Fluent |
|---|---|---|---|---|---|---|---|---|---|---|
| 2 | | | | | in | Hz | | lbf | lbf | |
| 3 | 0.6 | 5 | 0.5 | 1.1179 | 1.6986 | 209.43 | 4.9601 | 37.838 | 637.37 | 16.845 |
| 4 | 1.2 | 5 | 0.5 | 1.064 | 1.3926 | 241.88 | 5.0918 | 39 | 762.7 | 19.556 |
| * | | | | | | | | | | |



*Composite Failure*



*Max Deflection*



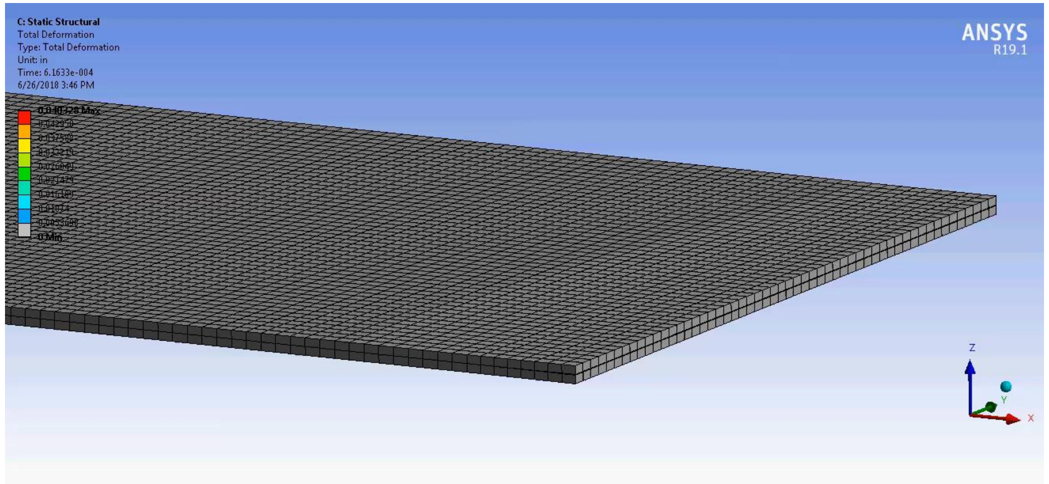
*Torsional Frequency*

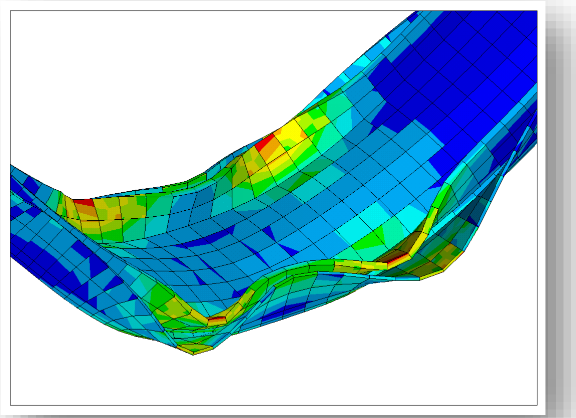
*Modified Airfoil Shows Aerodynamic and Structural Performance Increase*

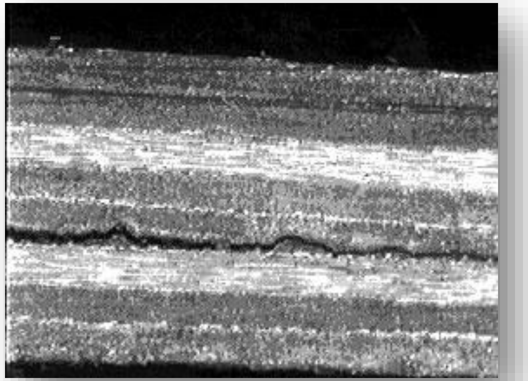
# Delamination and Progressive Damage

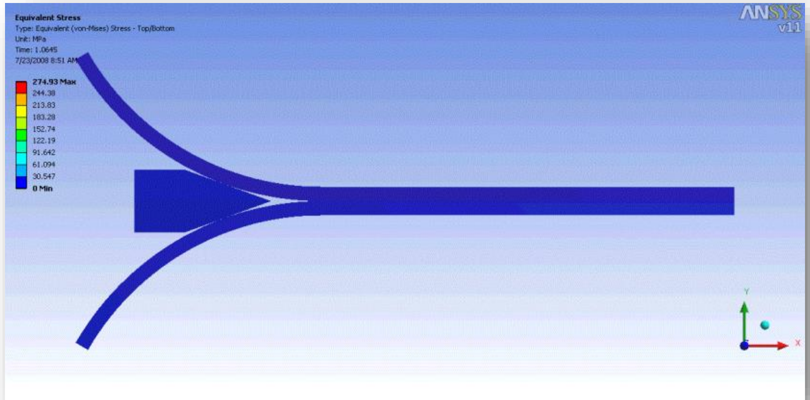
# Delamination and Damage (Cont.)

## *Examples*



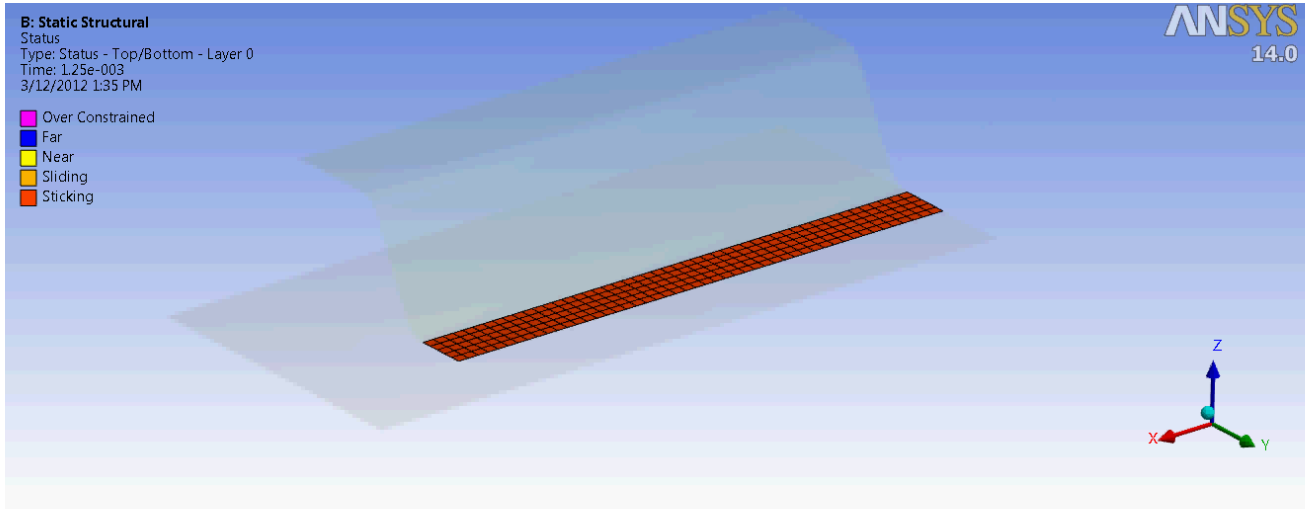


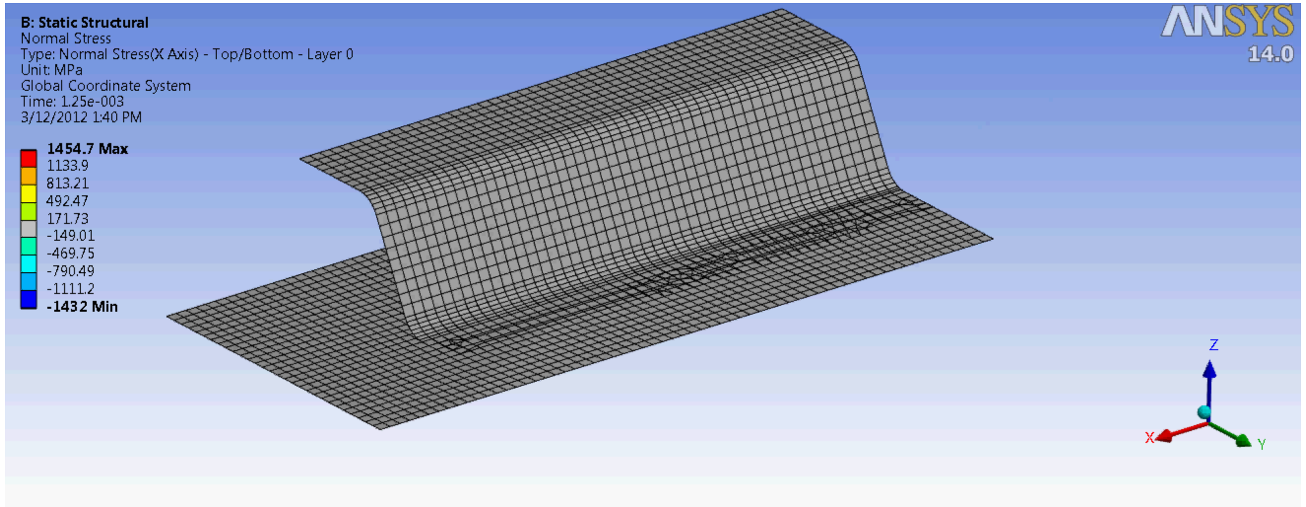


# Delamination and Damage (Cont.)

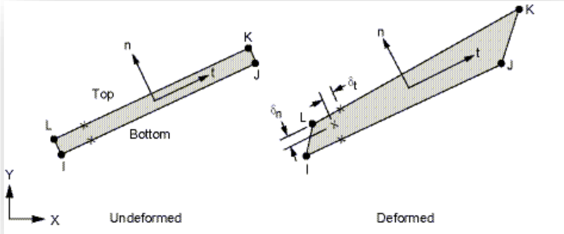
## Bilinear Cohesive Zone Modeling Example – Skin -Stringer



# Delamination and Damage (Cont.)

- **Two Approaches for Delamination (CZM and VCCT)**
- **Cohesive Zone Model - CZM** is typically used to simulate:
  - Interface delamination of composite structures
  - Separation of adhesive joints
  - Fracture process in a homogenous medium
  - This approach introduces failure mechanisms by using the hardening-softening relationships between the separations and incorporating the corresponding forces across the interface
  - An interface delamination and failure simulation is performed by first separating the model into two components or groups of elements. A cohesive zone is then defined between these two groups. ANSYS offers two options to model the interface:
    – Interface elements with CZM model
    – Bonded Contact elements & CZM model
  - Contact Debonding is exposed in Mechanical in Fracture Object

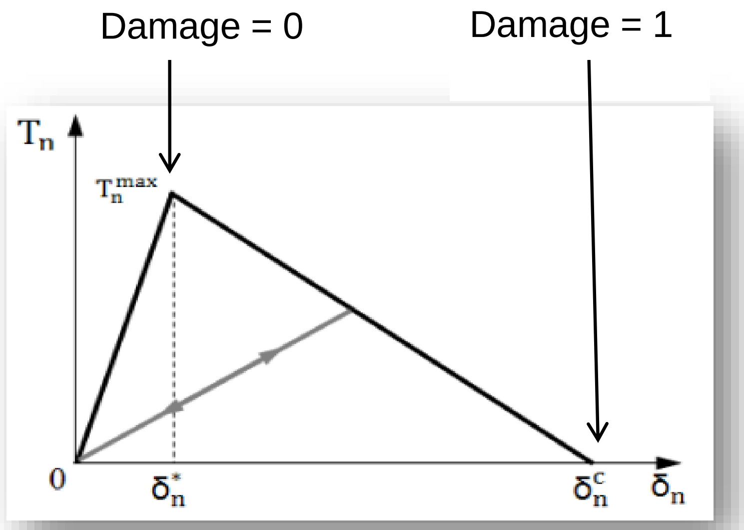



**Interface Elements**

ANSYS

# Delamination and Damage (Cont.)

- Cohesive Zone Models - CZM
  - Exponential & Bilinear
- Consideration of different contribution of Shear component from Normal component
- Consideration of irreversibility of CZM. Unloading behavior properly simulated (damage included).
- Support mode I, mode II/III, and mixed mode fracture

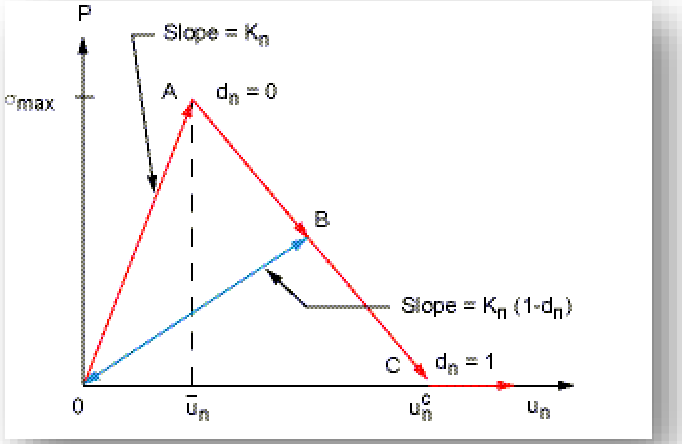


# Delamination and Damage (Cont.)

- **Bilinear Cohesive Zone Modeling using Contact**
  - The bilinear cohesive zone material is based on the model proposed by Alfano and Crisfield
  - Area Under the curve is the Normal Critical Fracture Energy Gcn (units Energy/Area)
  - Mode II/III Failure follows the same
  - Debonding is complete when this equation is satisfied

$$\left(\frac{G_n}{G_{cn}}\right) + \left(\frac{G_t}{G_{ct}}\right) = 1$$

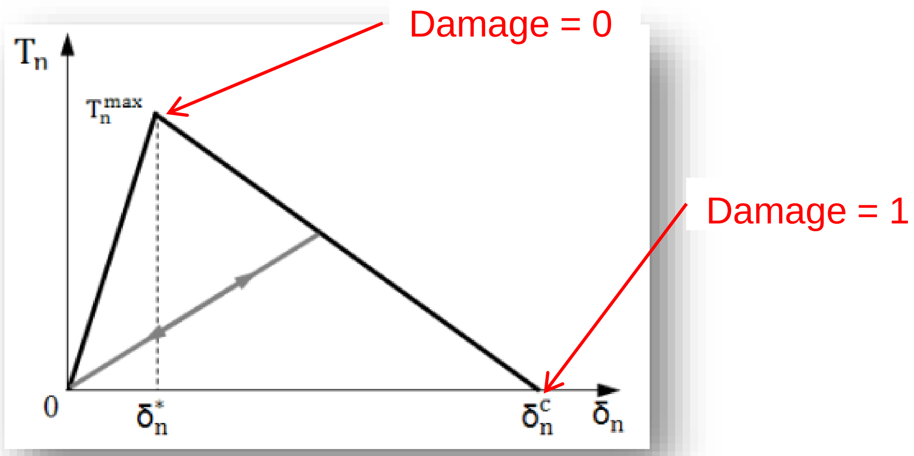
- **Options to specify**
  - CBDD -- Bilinear material behavior with linear softening characterized by maximum traction and maximum separation (valid for contact elements only).
  - CBDE -- Bilinear material behavior with linear softening characterized by maximum traction and critical energy release rate (valid for contact elements only).
  - User can specify normal, shear, or both

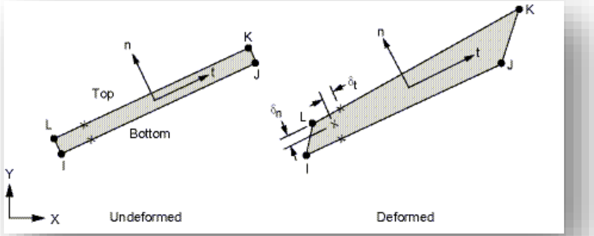


# Delamination and Damage (Cont.)

**Bilinear CZM Law using Interface Elements**

- A bilinear traction-separation law provides the material behavior with TB,CZM,,,,BILI
- Based on the work of Alfano and Crisfield
- Five or six material constants required
- Damage is included
- User inserts interface elements using czmesh command (MAPDL) which splits the mesh at the nodes and inserts the interface elements.
- Mechanical can create using a match mesh (to force matching) or node match (where node are already matching). Upcoming Slide.

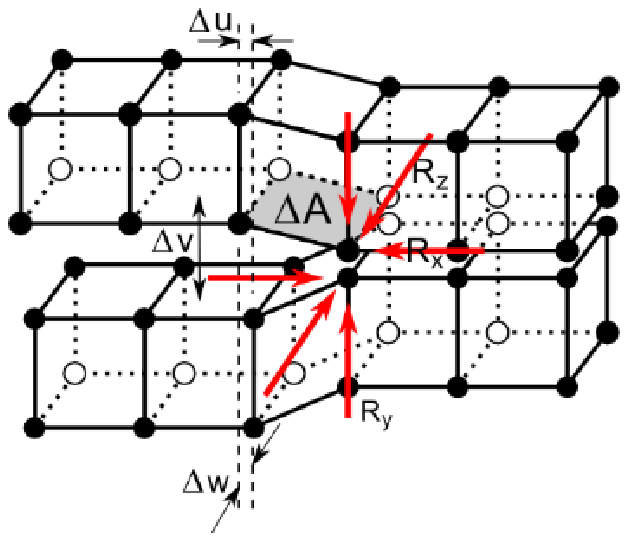



Interface Elements - INTERxxx

# Delamination and Damage (Cont.)

- **Virtual Crack Closure Technique – VCCT:**
  - Most commonly employed fracture mechanics approach to compute the debonding of composites structures
  - Based on the assumption that the energy needed to separate a surface is the same as the energy needed to close the same surface
  - VCCT was initially developed to calculate the energy-release rates of a cracked body and has been extended to crack propagation using interface elements
  - Crack growth occurs along a predefined crack path
  - The path is defined via interface elements.
  - The analysis is quasi-static and does not account for transient effects.
  - The material is linear elastic and can be isotropic, orthotropic or anisotropic.



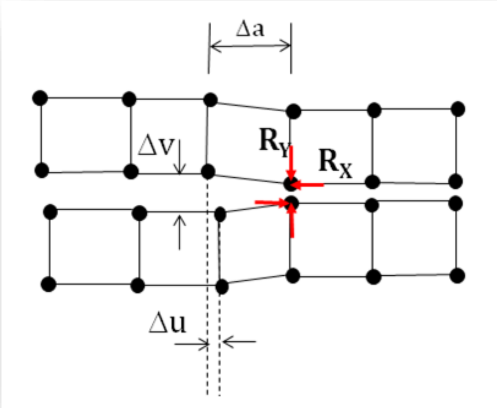
# Delamination and Damage (Cont.)

***How to model delamination and debonding?***

- Virtual Crack Closure Technique – VCCT:
  - The crack can be located in a material or along the interface of the two materials.
  - The fracture criteria are based on energy-release rates calculated using VCCT.
  - Several fracture criteria are available are:
    - Linear Fracture Criterion
    - Bilinear Fracture Criterion
    - B-K Fracture Criterion
    - Modified B-K Fracture Criterion
    - Power Law Fracture Criterion
    - User-Defined Fracture Criterion

$$f = \frac{G_T}{G_T^c}$$

$$f = \frac{G_I}{G_I^c} + \frac{G_{II}}{G_{II}^c} + \frac{G_{IIII}}{G_{III}^c}$$




- Multiple cracks can be defined in an analysis
- VCCT doesn't need a collapsed mesh as SIFS or JINT calculation
- Crack growth simulation is a nonlinear structural analysis
- Multiple cracks can grow simultaneously and independently
- Cracks can merge to a single crack when on the same interface

ANSYS

# Delamination and Damage (Cont.)

## *VCCT Based Crack Growth*

**Virtual Crack Closure Technique – VCCT**:

- Fracture occurs when the fracture criterion index is met, expressed as

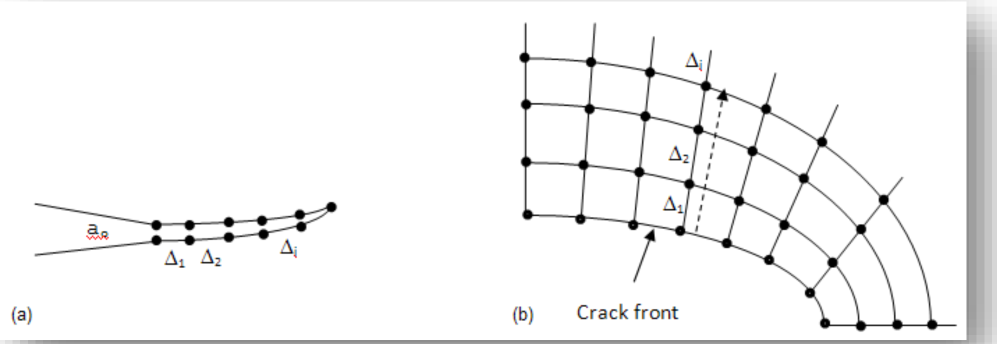
$$f \geq f_c$$

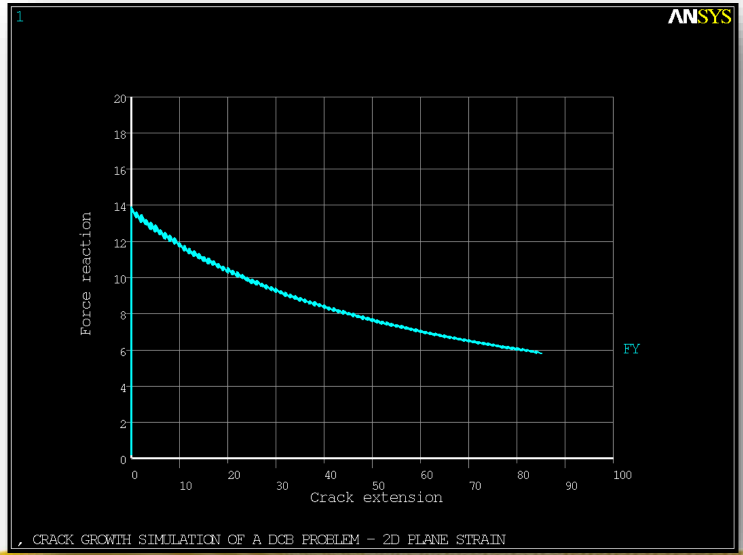
- In a crack growth simulation, a quantity of interest is the amount of crack extension. VCCT measures the crack extension based on the length of the interface elements that have opened, as expressed by the following equation and in the subsequent figure:

$$\Delta a = \Sigma \Delta_i$$

- Time step size is generally small during growth.

- Mesh size considerations are important

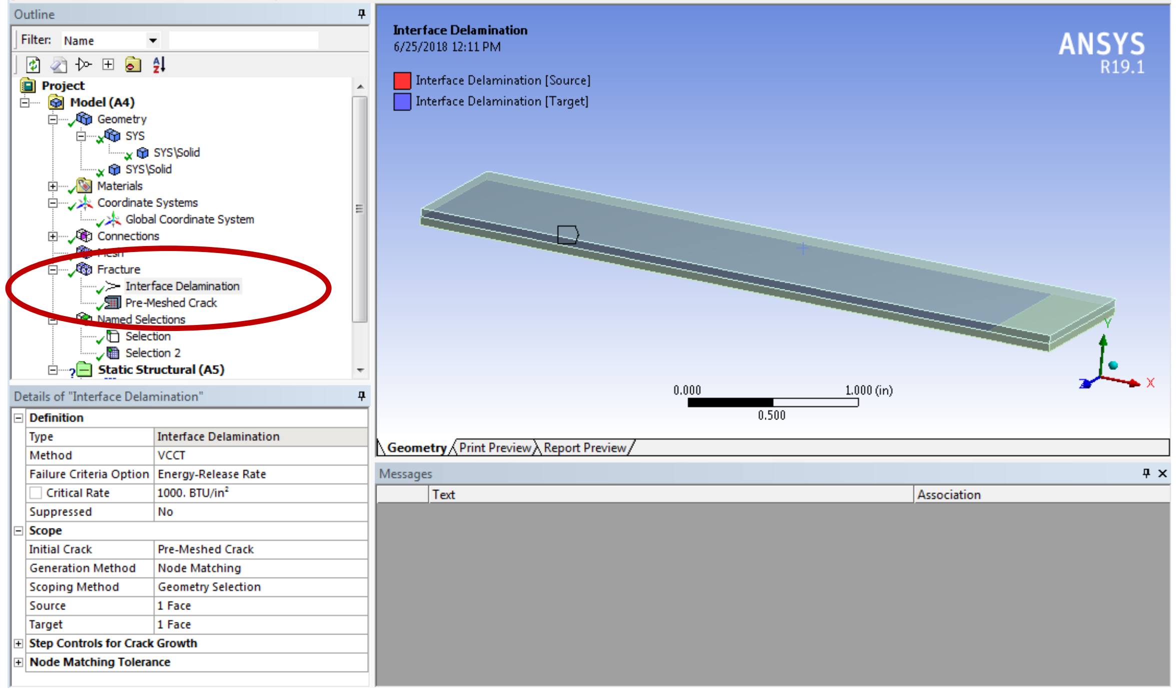


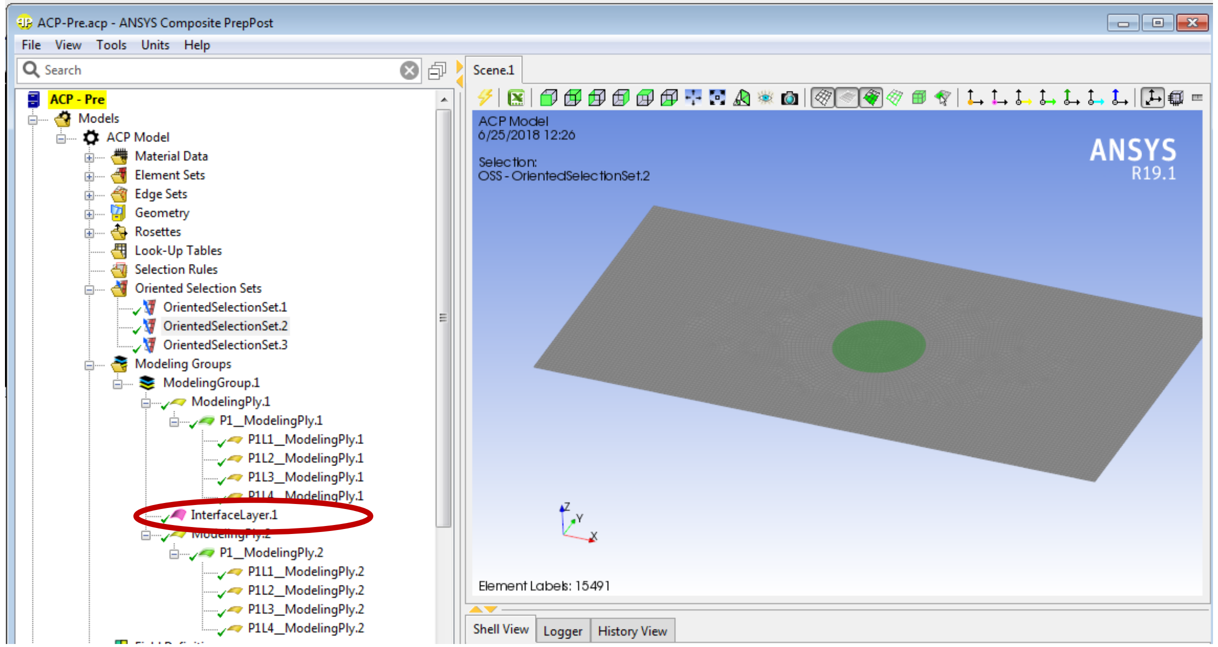
# Delamination and Damage (Cont.)

- **VCCT and CZM- How to define Initial Delamination?**
  - **Mechanical**
  - **ACP**

**Define Delamination in Mechanical**




**Define Delamination in ACP**

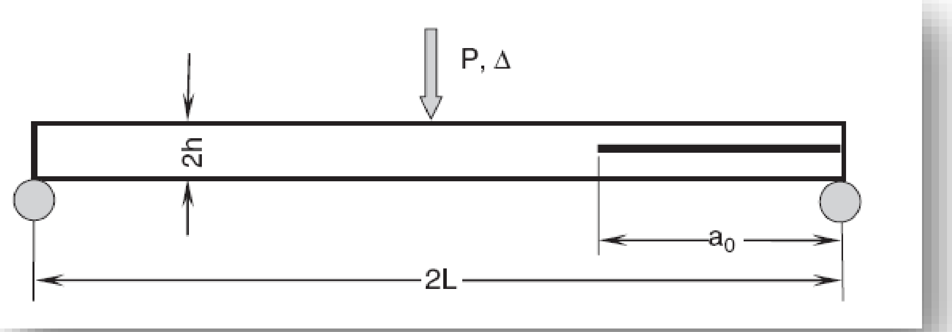


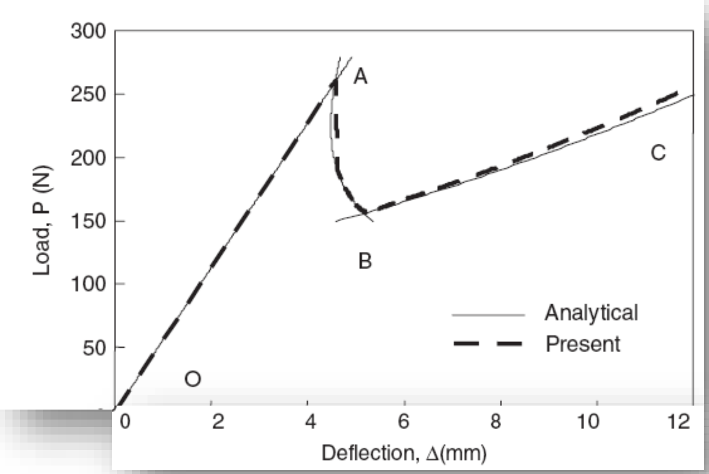

  

# VCCT-Based Crack Growth Simulation

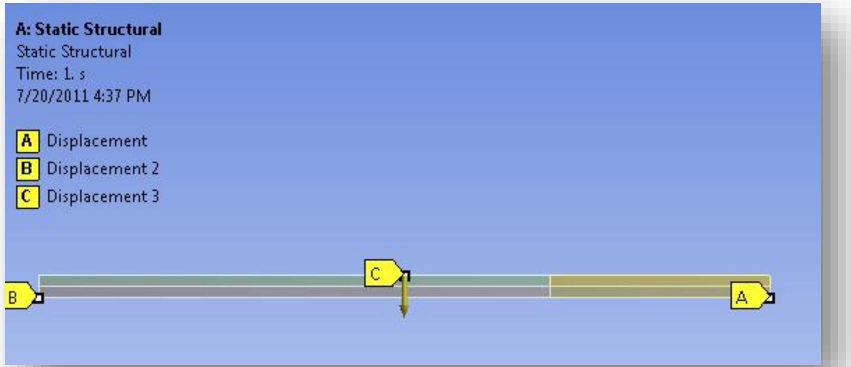
## *VCCT Examples*

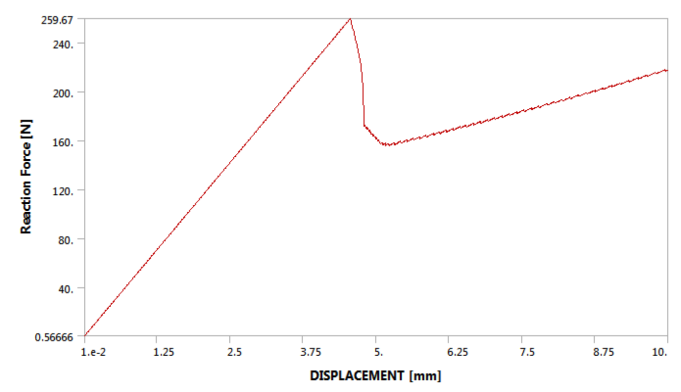
End Notched Flexure (ENF) Specimen




Results from published paper




ANSYS

Reference: Progressive crack growth analysis using interface element based on the virtual crack closure technique by De Xiea *and Sherrill B. Biggers Jr.* Finite Elements in Analysis and Design 42 (2006) 977 – 984

# VCCT-Based Crack Growth Simulation

## *VCCT Examples*

Mixed Mode Bending (MMB) Specimen

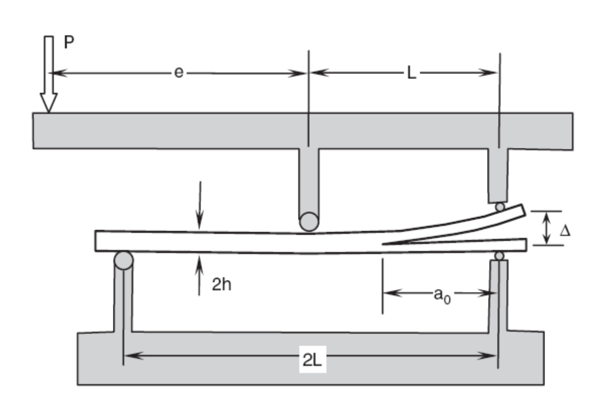
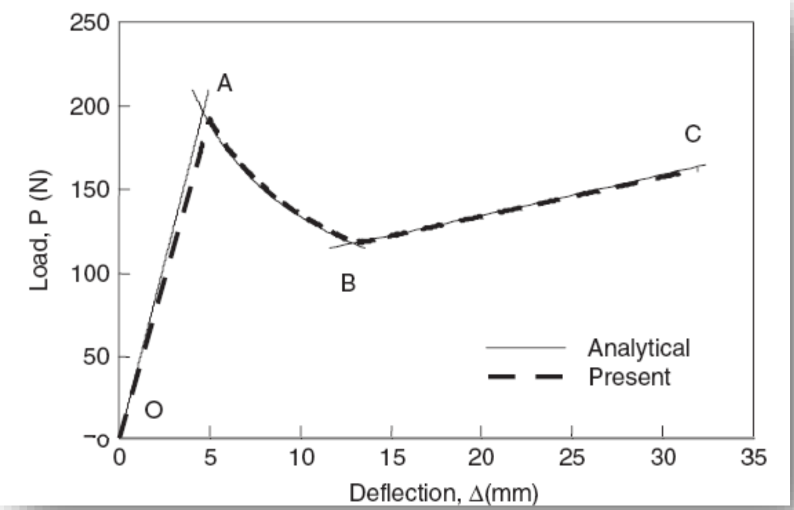
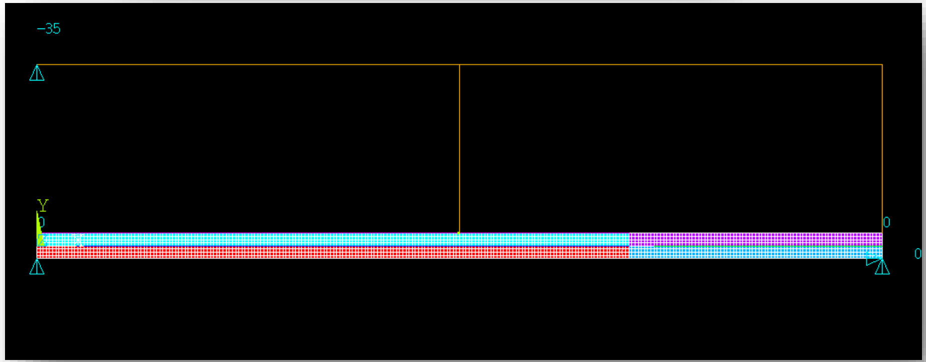
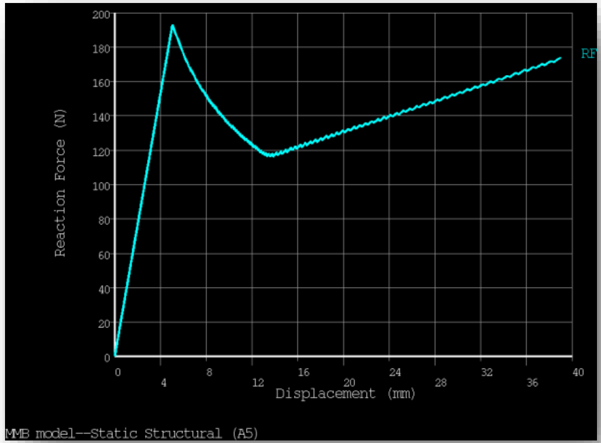



Fig. 12. Mixed-mode bending (MMB) test apparatus.




Results from published paper



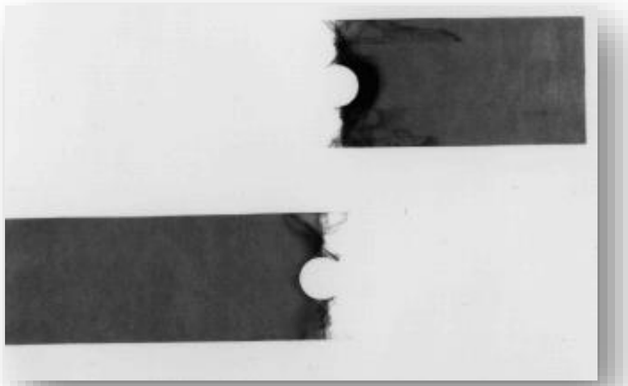

ANSYS

Reference: Progressive crack growth analysis using interface element based on the virtual crack closure technique by De Xiea *and Sherrill B. Biggers Jr.* Finite Elements in Analysis and Design 42 (2006) 977 – 984

# Composite Progressive Damage

***What happens beyond first ply failure?***

Progressive Damage



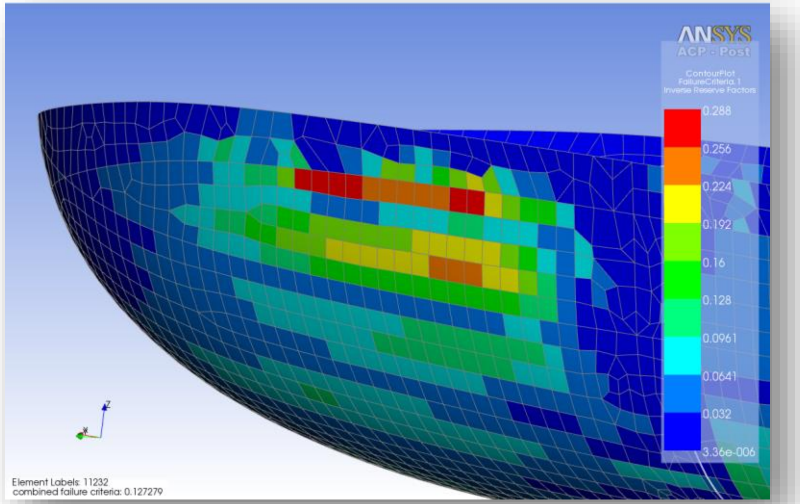
# Composite Progressive Damage

***What happens beyond first ply failure?***

Plotting failure criteria allow us to estimate the strength of the structure based on the first ply failure using one or multiple failure criteria (Puck, Hashin, Max Stress, etc.)

**Progressive damage**

- Progressive damage generally refers to degradation of stiffness
- ANSYS has the MPDG option and others such as the Microplane model (not covered here)



# Composite Progressive Damage

## *What happens beyond first ply failure?*

**Progressive Damage MPDG Model**

- The damage initiation and propagation in fiber-reinforced composites can be simulated with a nonlinear solution process.
- This capability allows you to estimate ultimate composite strength under complex stress states.
- The difference between first ply failure and ultimate is very dependent on the layup and loading.  For some layups, the difference can be just a few percent, while with others, it can be much greater
- Damage Initiation Criteria  - This defines the criteria type for determining the *onset* of the material damage under loading. The available failure criteria are as follows:
  - Maximum strain, Maximum stress, Puck, Hashin, LaRc03, LaRc04, User-defined
- Damage Evolution Law -  This defines the way material degrades following the initiation of damage

# Composite Progressive Damage

**Compare ANSYS Damage capability TB,DMGI & TB,DMGE with published work of Camanho and Matthews to showcase the capabilities**

**Paper Title:**

**A Progressive Damage Model for Mechanically Fastened Joints in Composite Laminates**

**P. P. Camanho and F. L. Matthews** *

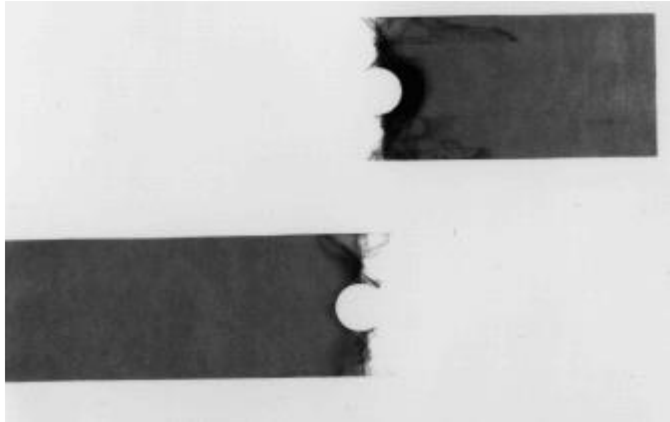
*Centre for Composite Materials*
*Imperial College of Science, Technology and Medicine*
*Prince Consort Road*
*London SW7 2BY, UK*

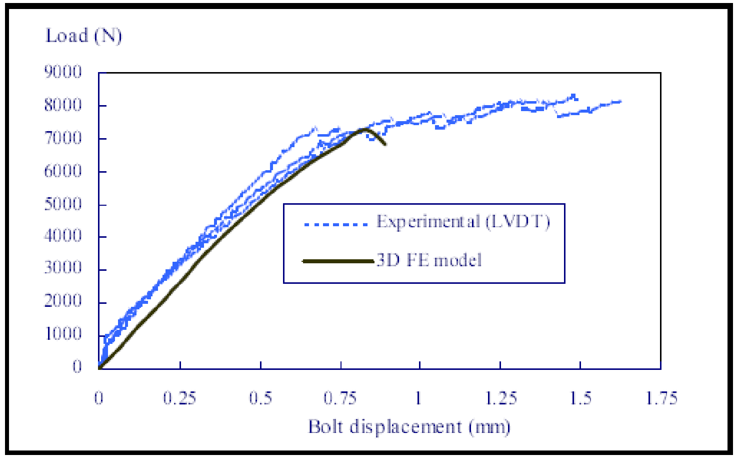
**(Paper accepted for publication in the *Journal of Composite Materials*, 2000)**

**Journal of Composite Materials December 1999 33: 2248-2280,**

# Composite Progressive Damage

- In this work, the Authors have experimental Load/Disp curves of composite coupons of Pin Loaded Net Tension, Shear-Out, and Bearing failure modes.
- Authors are using Abaqus Explicit with reduced integration and hourglass control.
- Hashin Failure on [0/90/-45/45]2s laminate made of T300/914.
- Simulation model is solid with one element per ply.
- Net Tension Failure experimental and Abaqus results shown below.





# Composite Progressive Damage

- **ANSYS model built in Workbench Mechanical.**
- **Composite Setup in ACP.**
- **Failure and damage commands added to engineering data**
- **Layered composite SHELL181 with pin as rigid contact.**
- **Damage Evolution stiffness reduction per the cited paper**

```
!  Specify Hashin strength terms
TB,FCLI,my_mat_num,1,20,1
TBTEMP,71.6
TBDATA,1,1439.,-1318.,98.,-125.,98.,-125.
TBDATA,7,79.,79.,79.

! Specify damage initiation Hashin
TB,DMGI,my_mat_num,1,4,FCRT
TBTEMP,71.6
TBDATA,1,4,4,4,4
```

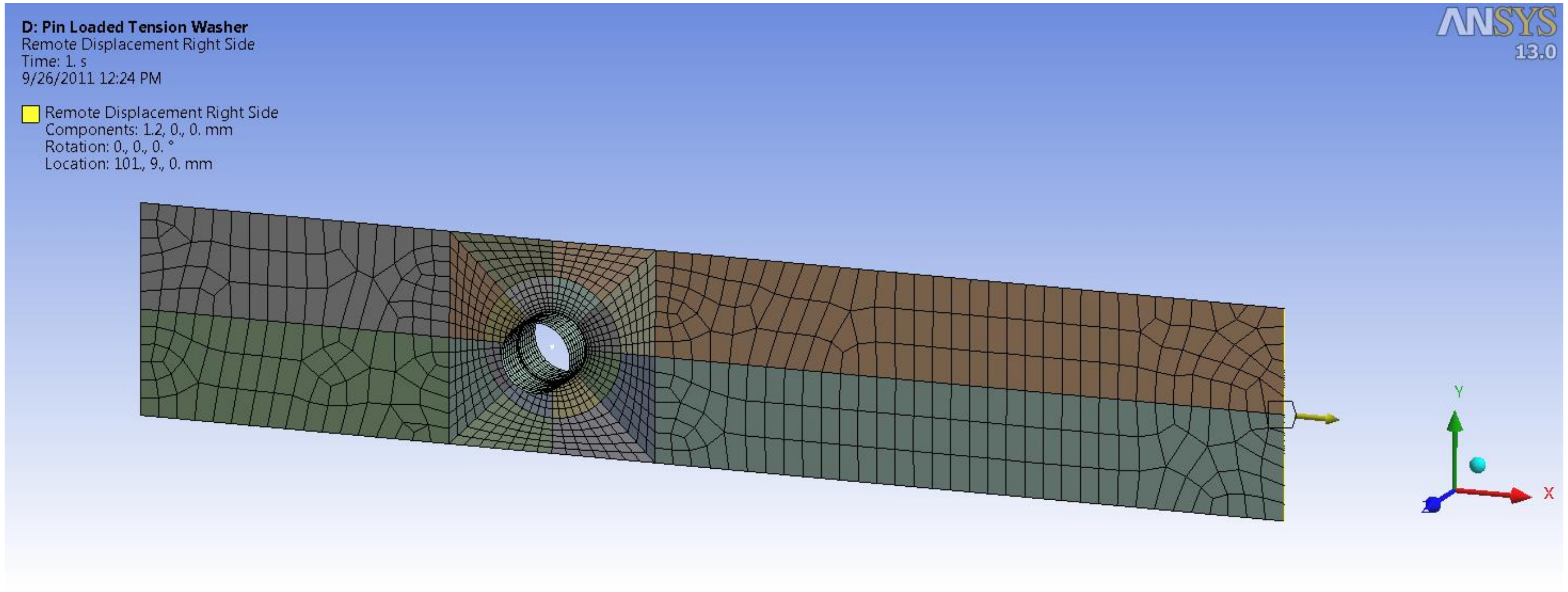
```
! Specify damage evolution Hashin

TB,DMGE,my_mat_num,1,4,MPDG
TBTEMP,71.6
TBDATA,1,.93,.86,.8,.6
```

Properties of Outline Row 4: T300/914

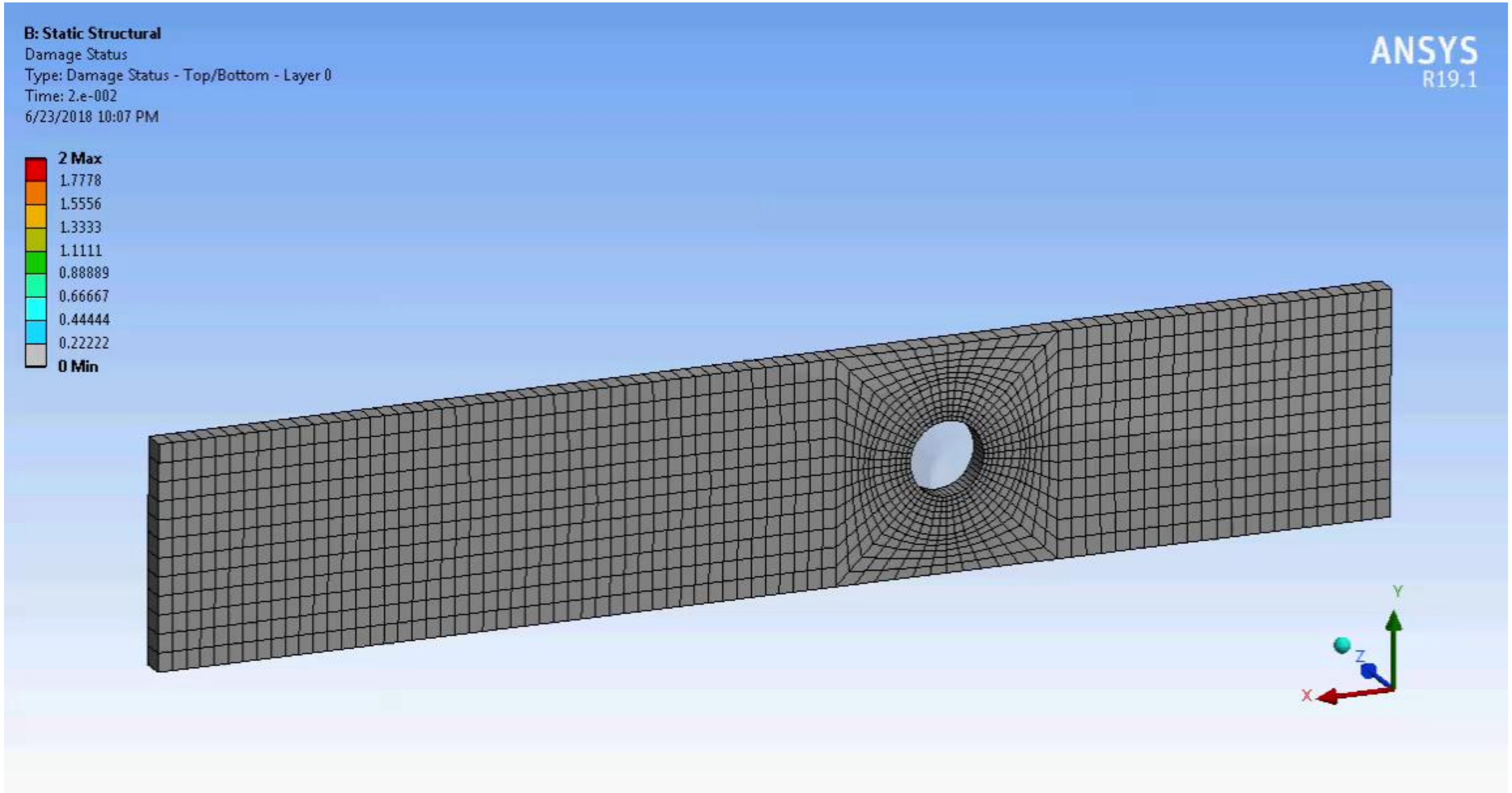
| | A | B | C | D | E |
|---|---|---|---|---|---|
| 1 | Property | Value | Unit | | |
| 2 | Material Field Variables | Table | | | |
| 3 | Orthotropic Elasticity | | | | |
| 4 | Young's Modulus X direction | 1.29E+05 | MPa | | |
| 5 | Young's Modulus Y direction | 9500 | MPa | | |
| 6 | Young's Modulus Z direction | 9500 | MPa | | |
| 7 | Poisson's Ratio XY | 0.34 | | | |
| 8 | Poisson's Ratio YZ | 0.52 | | | |
| 9 | Poisson's Ratio XZ | 0.34 | | | |
| 10 | Shear Modulus XY | 4700 | MPa | | |
| 11 | Shear Modulus YZ | 3200 | MPa | | |
| 12 | Shear Modulus XZ | 4700 | MPa | | |
| 13 | Orthotropic Stress Limits | | | | |
| 14 | Tensile X direction | 1439 | MPa | | |
| 15 | Tensile Y direction | 98 | MPa | | |
| 16 | Tensile Z direction | 98 | MPa | | |
| 17 | Compressive X direction | -1318 | MPa | | |
| 18 | Compressive Y direction | -125 | MPa | | |
| 19 | Compressive Z direction | -125 | MPa | | |
| 20 | Shear XY | 79 | MPa | | |
| 21 | Shear YZ | 79 | MPa | | |
| 22 | Shear XZ | 79 | MPa | | |
| 23 | Damage Initiation Criteria | | | | |
| 24 | Tensile Fiber Failure Mode | Hashin | | | |
| 25 | Compressive Fiber Failure Mode | Hashin | | | |
| 26 | Tensile Matrix Failure Mode | Hashin | | | |
| 27 | Compressive Matrix Failure Mode | Hashin | | | |
| 28 | Damage Evolution Law | | | | |
| 29 | Active Table | Material ... | | | |
| 30 | Tensile Fiber Stiffness Reduction | 0.93 | | | |
| 31 | Compressive Fiber Stiffness Reduction | 0.86 | | | |
| 32 | Tensile Matrix Stiffness Reduction | 0.8 | | | |
| 33 | Compressive Matrix Stiffness Reduction | 0.6 | | | |

# Composite Progressive Damage

- Specimen held ALL DOF at left edge, and at pin.
- Specimen displaced 1.2mm on right edge.
- Coupon dimension specified in this case to generate net tension failure, with some local bearing failure.
- Pin Contact is Normal Lagrange



# Composite Progressive Damage

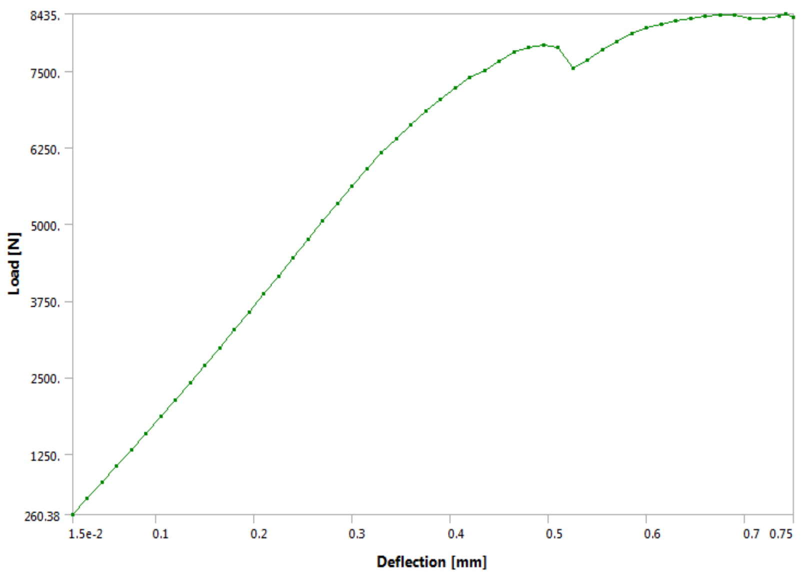



Damage status
0 = undamaged
1 = damaged
2 = completely damaged

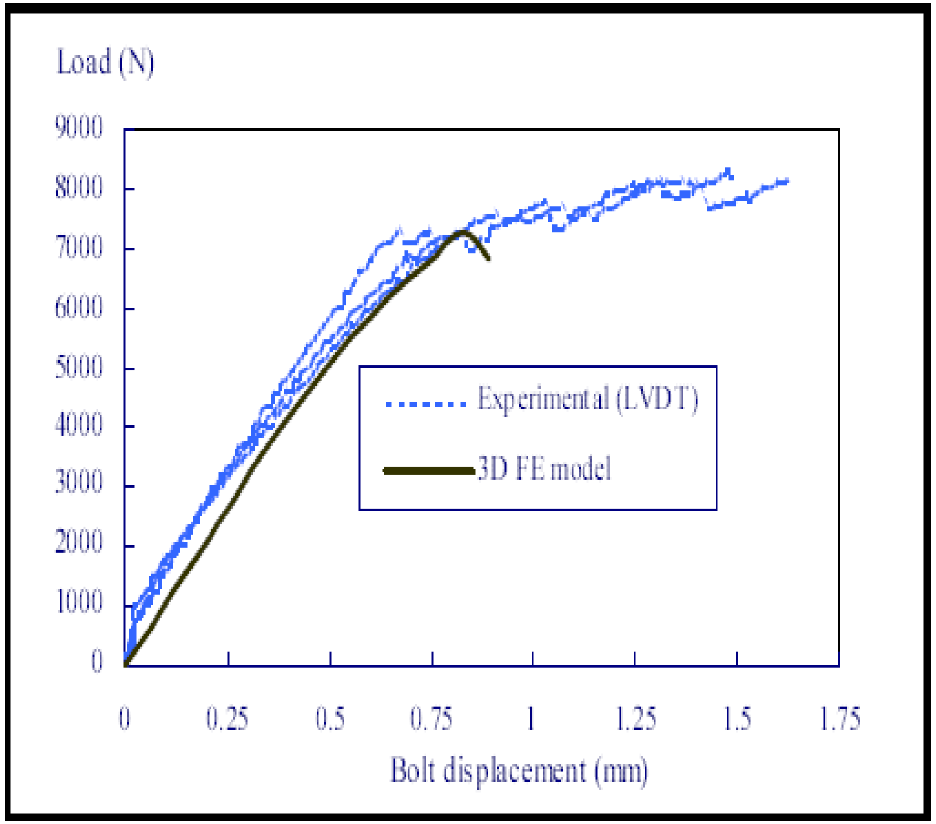
# Composite Progressive Damage

## Compare Load vs Displacement Curves

**ANSYS R19.1 Beta**



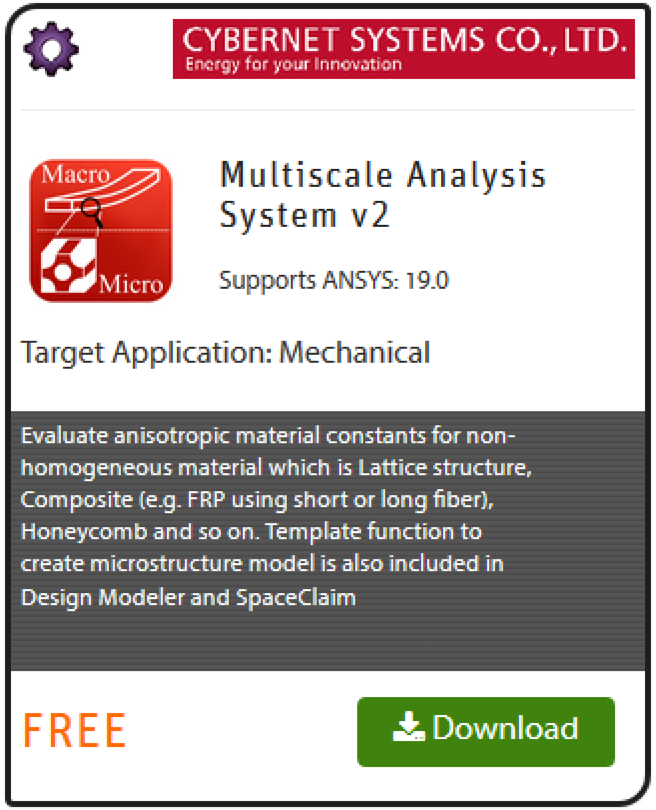

**Experiment & Abaqus/Explicit**

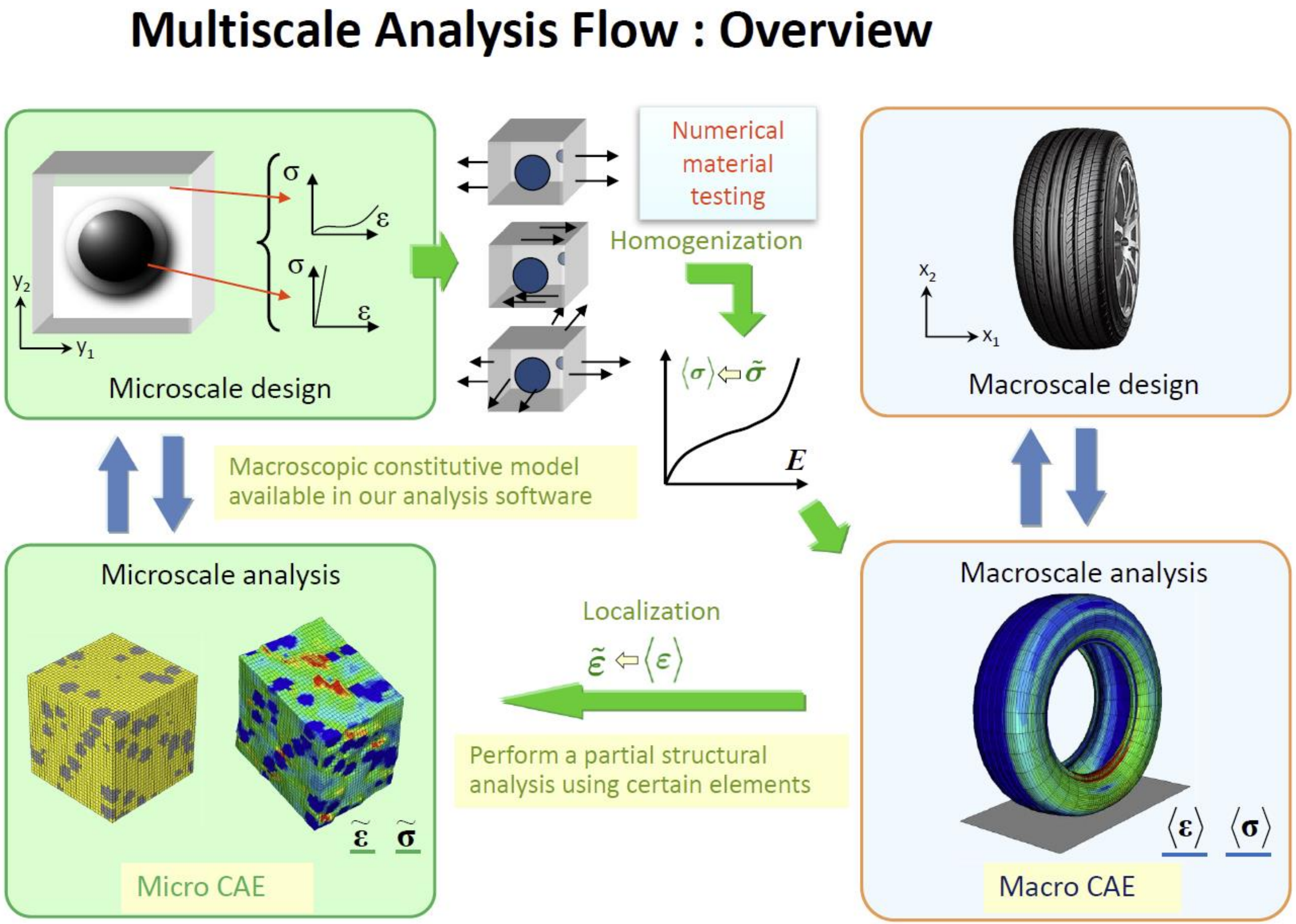


# Multiscale Modeling

# Workbench Apps for Composites – Multiscale CMAS

- **ANSYS ACT**
- **https://appstore.ansys.com/search?q=multiscale**



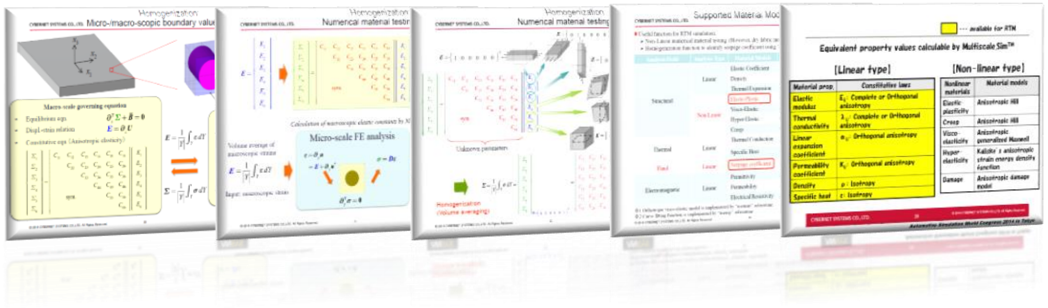



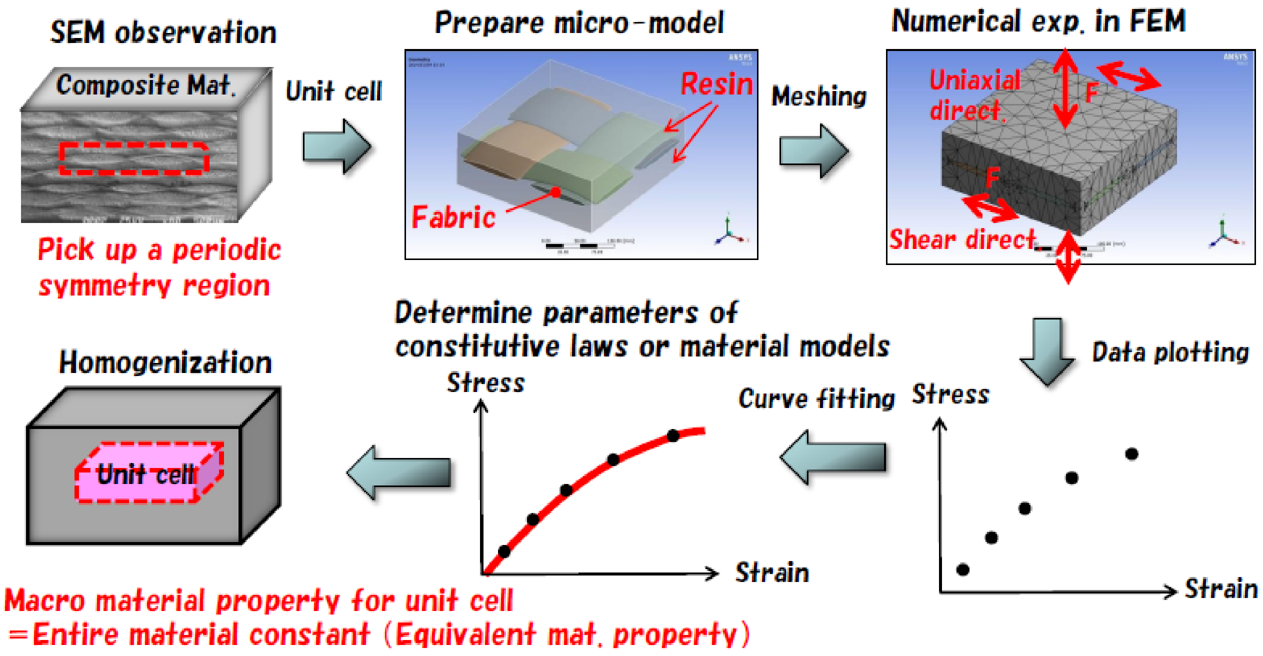
# Multiscale CMAS

ANSYS ACT

CYBERNET



- **Calculate the material properties** with accuracy.
- **Homogenization** analysis for each microstructure model.




ANSYS

# Multiscale CMAS

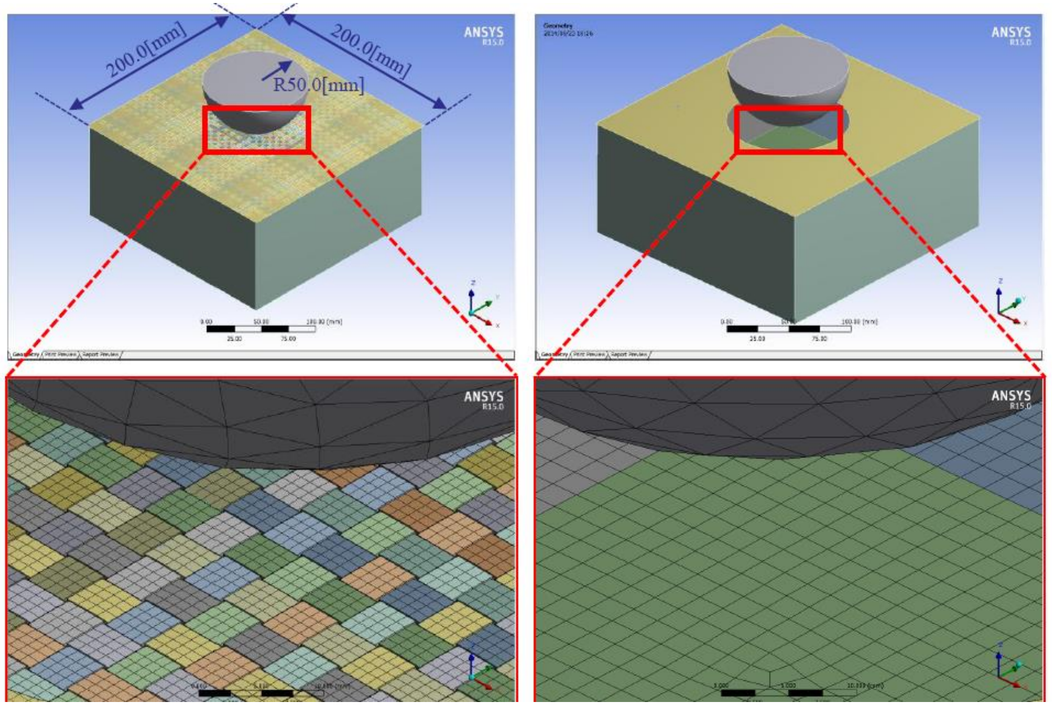
ANSYS ACT

CYBERNET

- Homogenized model validated vs direct model

**Press Velocity :** 5m/s
**Material Model**
Punch : *MAT_RIGID
Die : *MAT_RIGID
Tow : *MAT2
Weave : *MAT235 (Homog)
**Model Statistics**

| | Direct | Homogenized |
|---|---|---|
| Nodes | 115 k | 16 k |
| Elements | 130 k | 42 k |

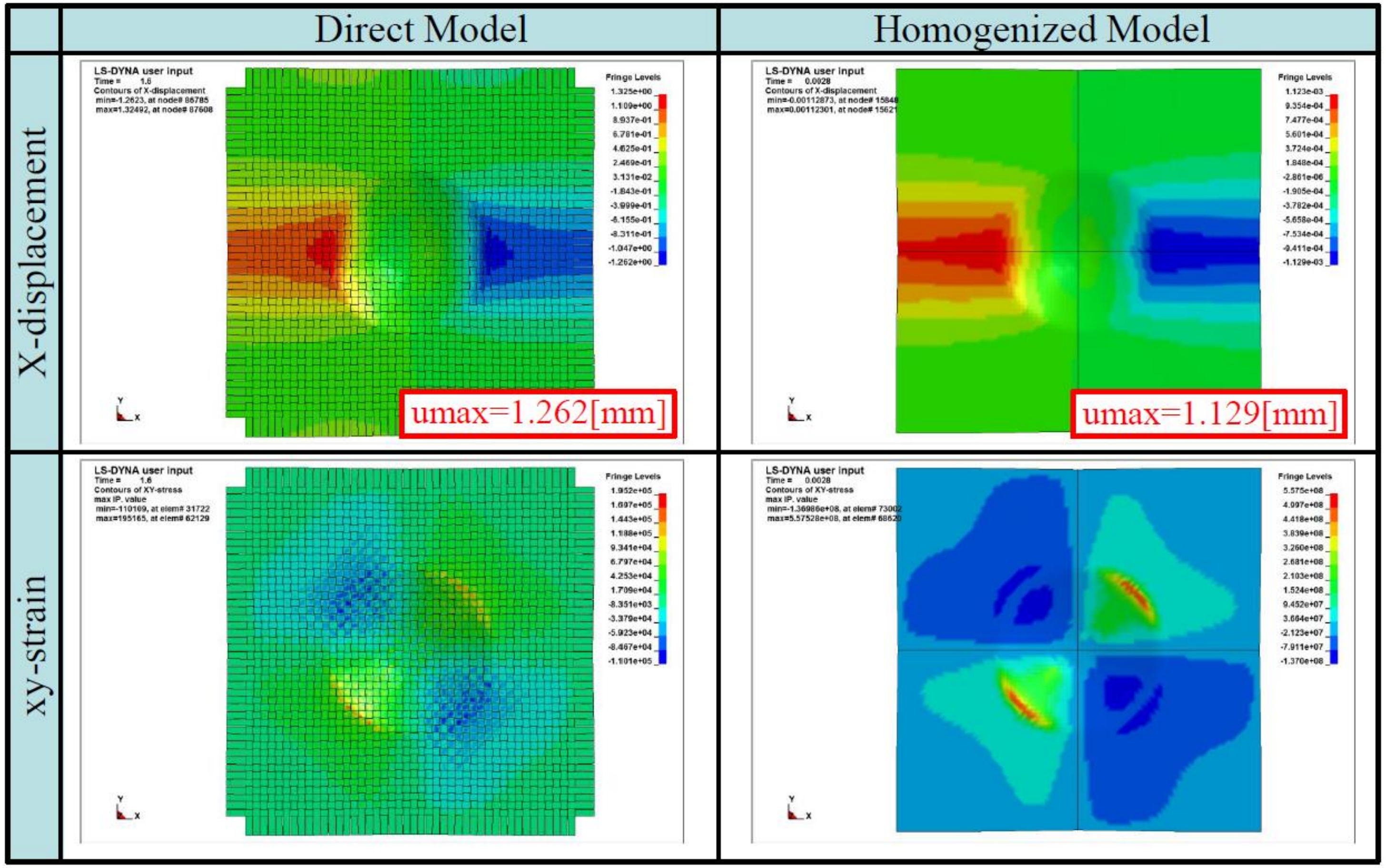


ANSYS

# Multiscale CMAS

- Homogenized model validated vs direct model



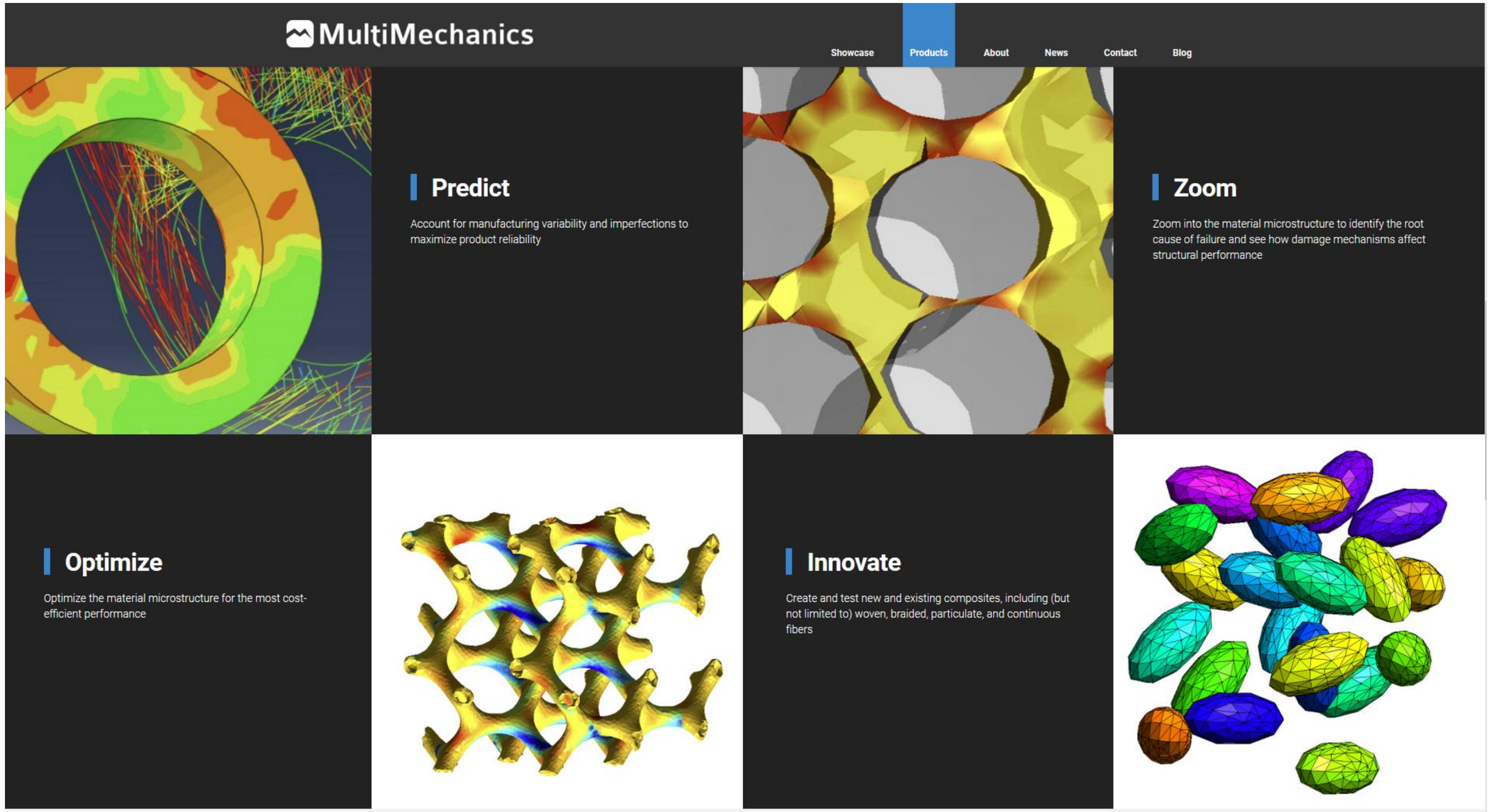

Direct model: 5h31min
Homogenized model : 0h22min

# Workbench Apps for Composites – MultiMechanics

- **MultiMech**
  - https://www.ansys.com/resource-library/webinar/composite-performance-multimech-2018

MultiMech for ANSYS

Intuitive TRUE Multiscale FE analyses within ANSYS/Workbench

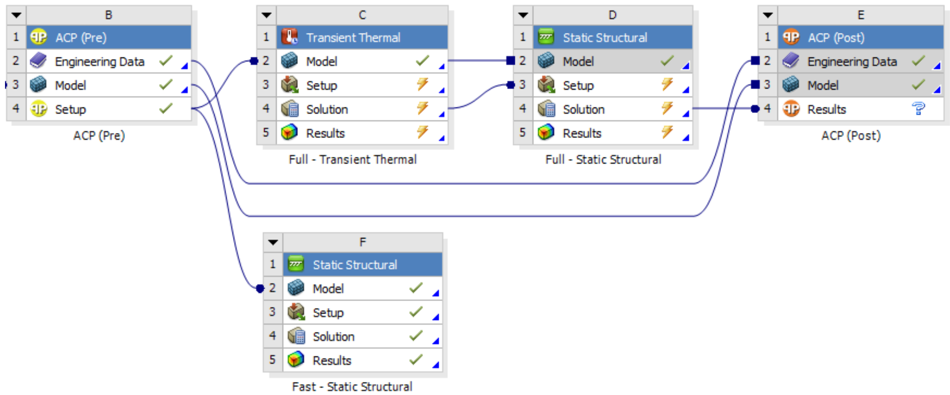


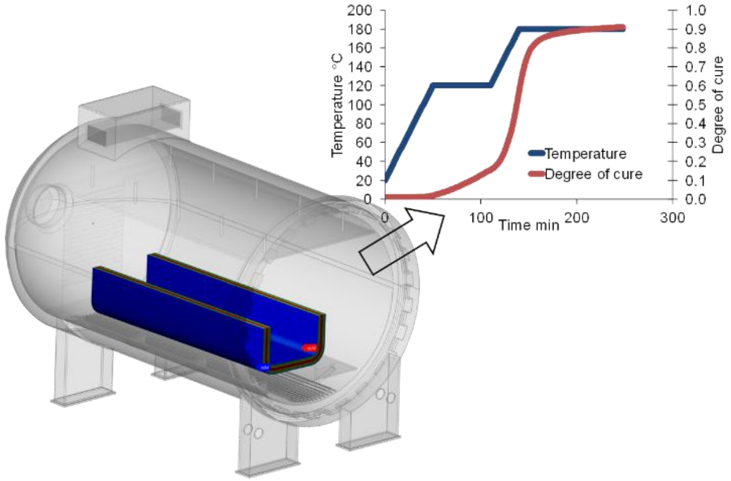

  

# ANSYS COMPOSITE CURE SIMULATION

# Curing / ANSYS Composite Cure Simulation (ACCS)

- Residual Stresses during Cure lead to **distortion/cracking**
- Spring-in, Cure **Shrinkage,** Consolidation
- Enthalpy, **Tool Part interaction**
- Design inverted tooling surface
- Temperature and Power During Cure

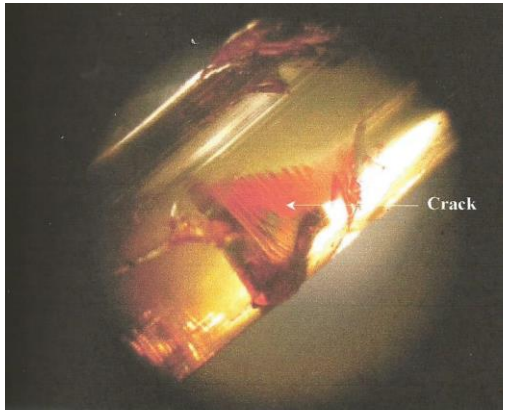


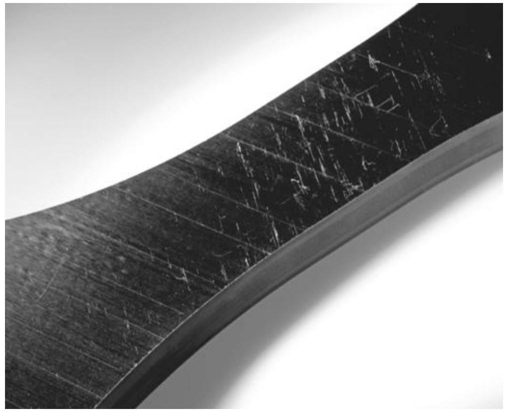




*The ACCS chemical solver is embedded within transient thermal module and simulates development of polimerisation, glass transition temperature as well as internal heat generation related to exothermic cross-linking reactions*

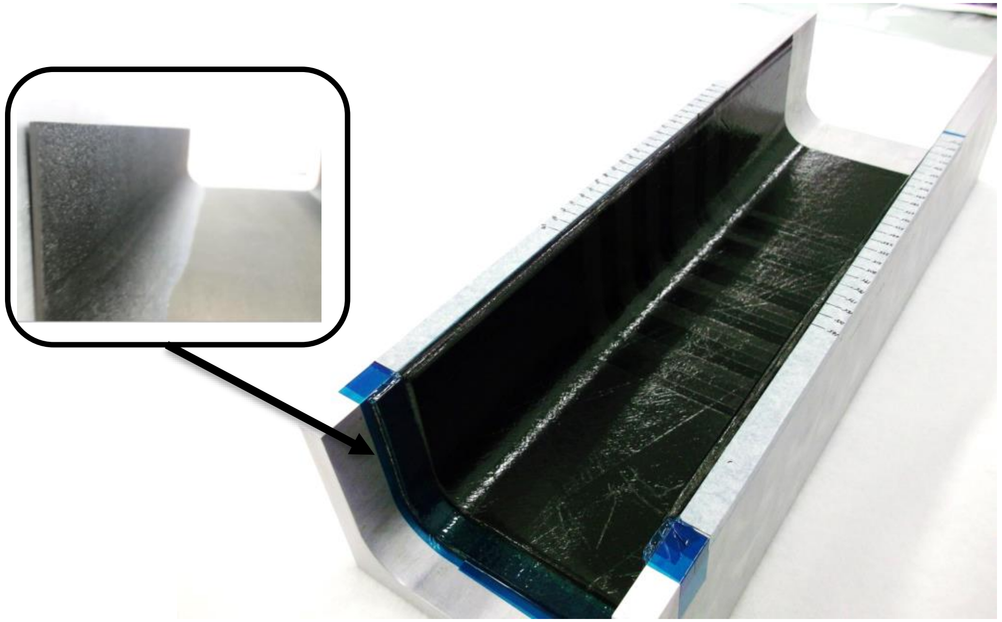
# Distortions and Cracking in Composites

**Residual stress induced cracking**



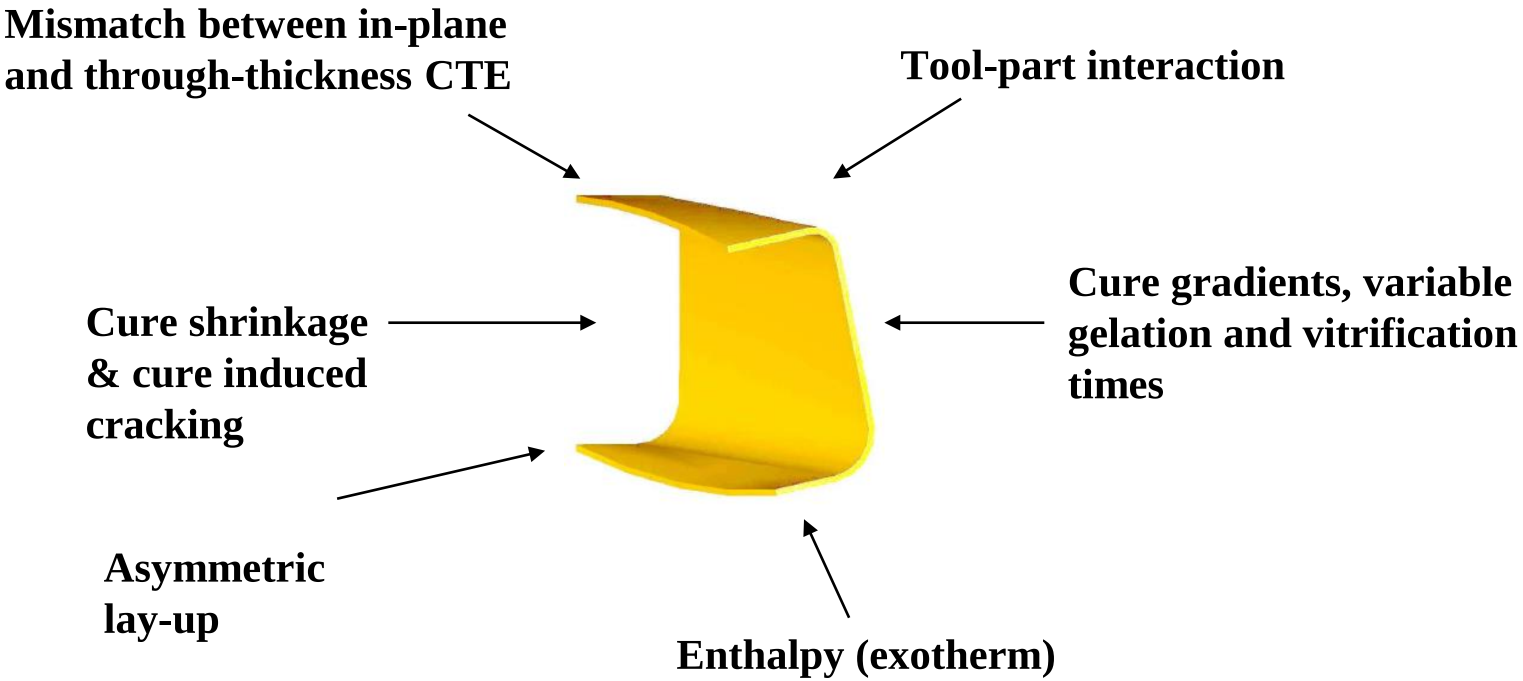



**Spring-in in a composite spar section**



# What is causing the problem?

Residual stresses that develop during cure lead to distortion or cracking significantly impacting on the final product performance



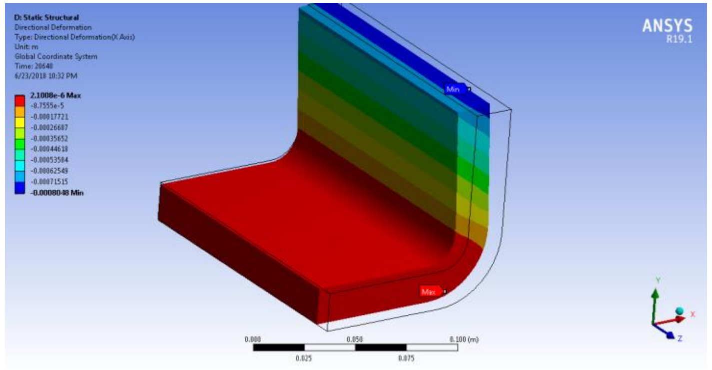

from T Garstka. Separation of process induced distortions in curved laminates, PhD thesis, Bristol,2005

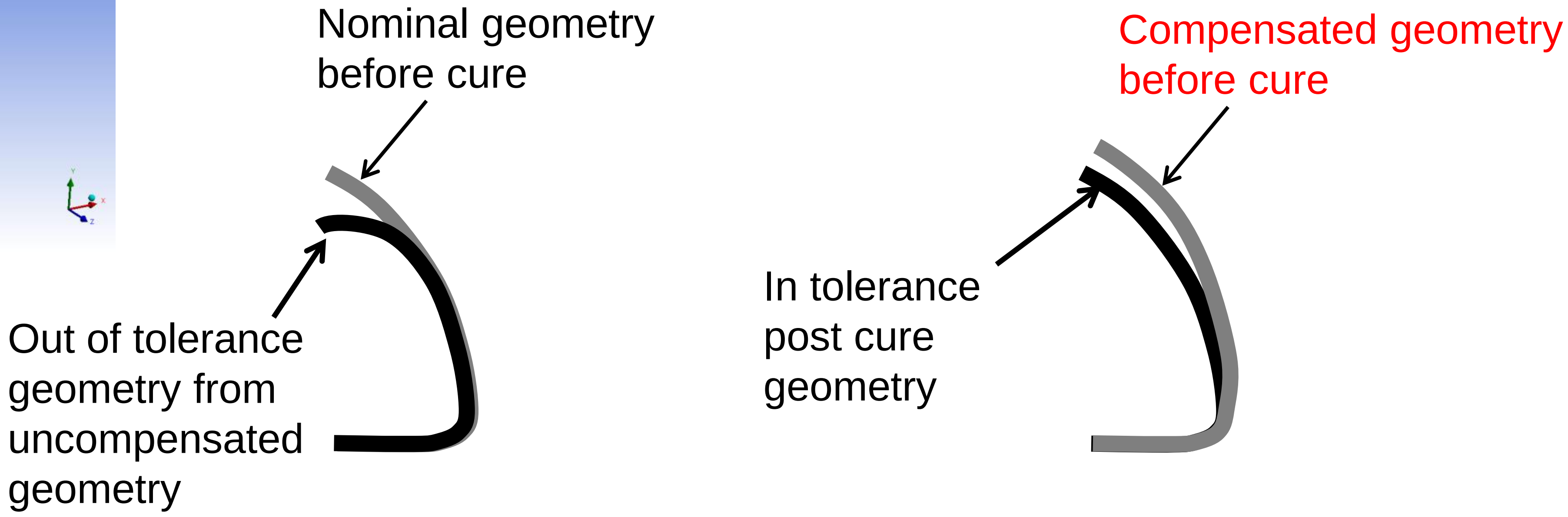
   

# Solution, numerical analysis

1) Predict distorted postcure geometry of the composite part

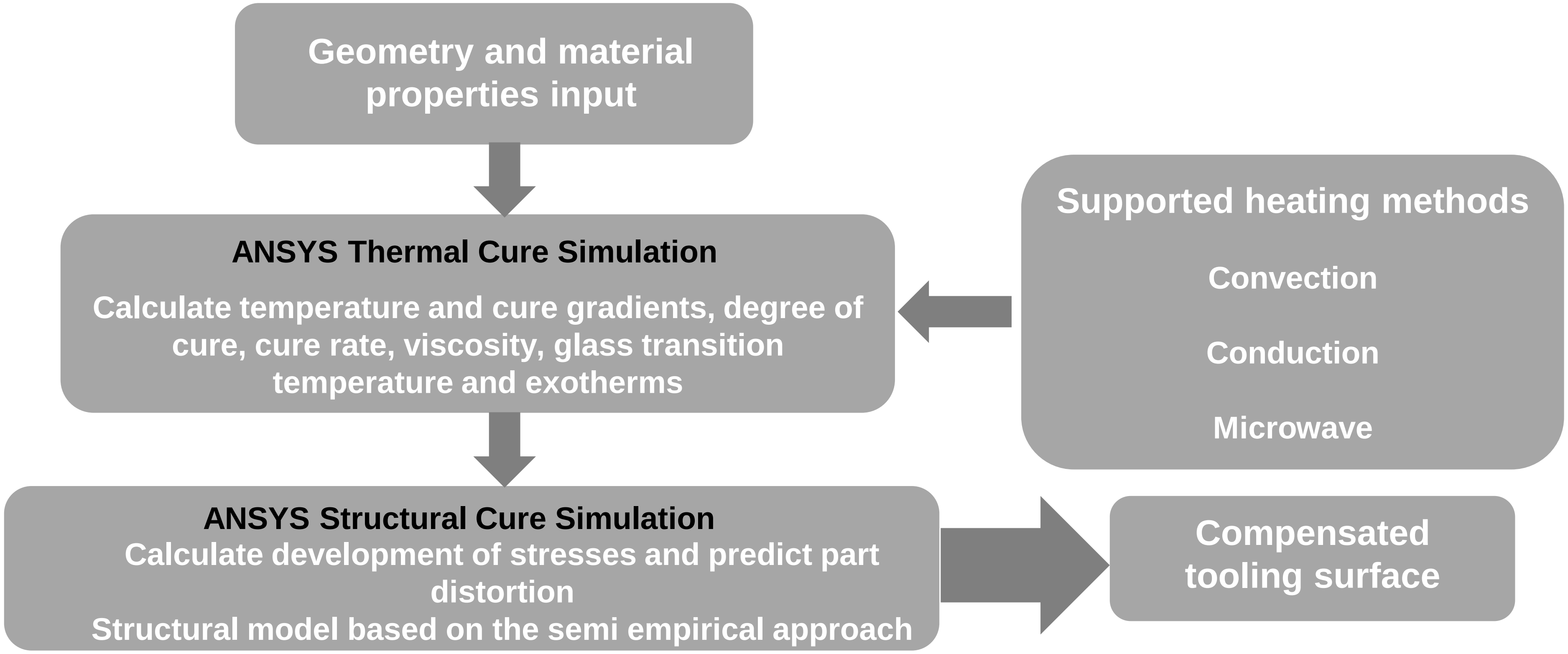
2) Invert part distortion and use it to design tooling



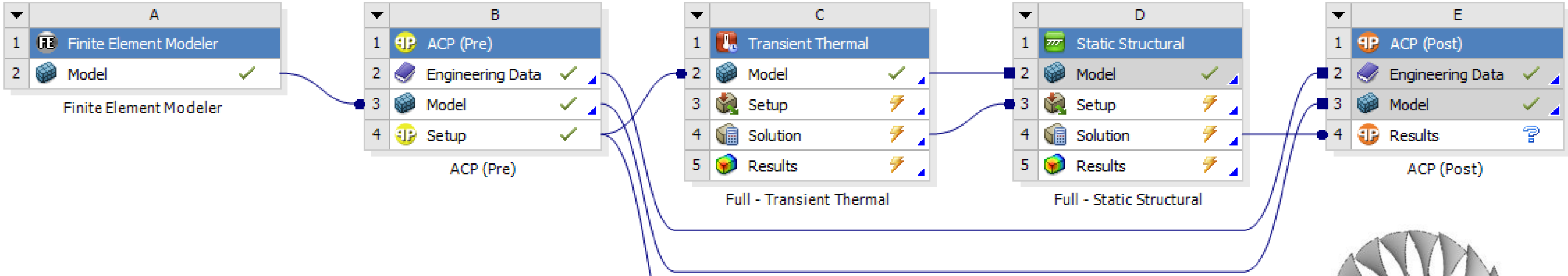

  

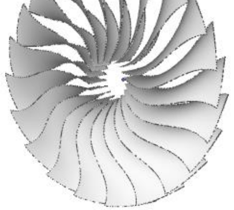
# ANSYS CURE SIMULATION SOLUTION



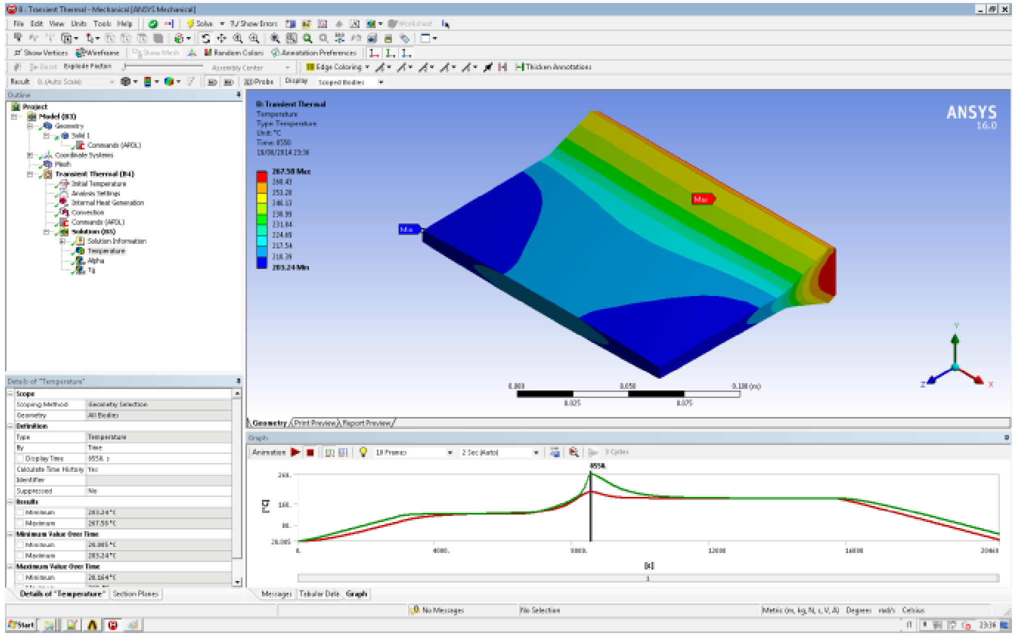

The analysis is semi coupled. First we evaluate temperature and degree of cure using thermal solver and then degree of cure and temperature is passed to structural simulation to analyse development of stresses and distortions

# Simulation workflow and postprocessing



- **Temperature and exotherms**
- **Degree of cure**
- **Glass transition temperature development**
- **Cure optimisation**




ANSYS

# Example where ACCS technology was used

Tooling corrections for landing flap nose caps

Tooling design and compensation for the leading edges. Optimisation of the bonding process


Tooling corrections for flaps including ribs, skins and spars

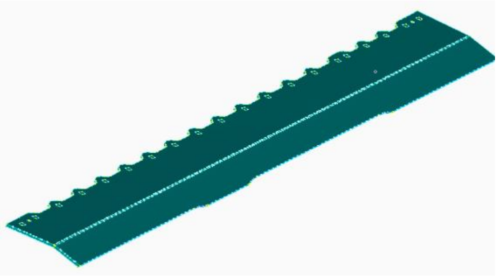
Tooling modifications to fan track liners

Tooling design corrections for rear spars

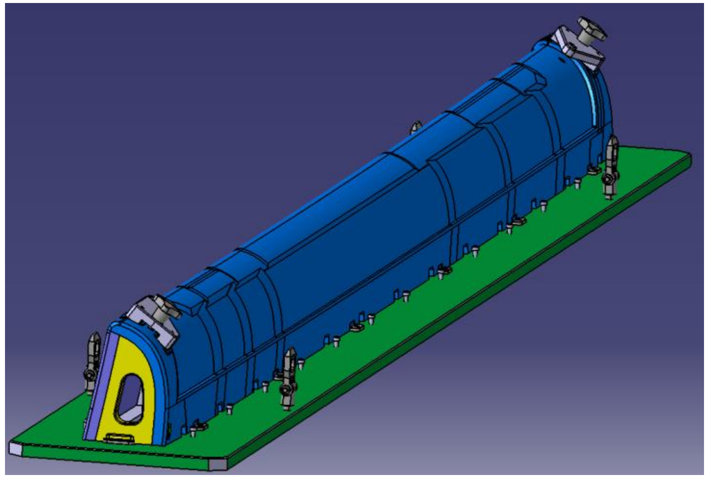
ANSYS

# Example problems

**Sewing angles linking fuselage and wing**
**Steel tooling**

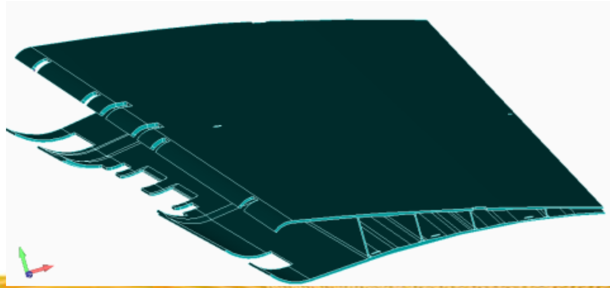


**Composite nose caps on invar tooling**



**Tooling for the Vestas v164 wind turbine**
**Carbon/Glass tooling**

**Landing flaps**
**Invar tooling**

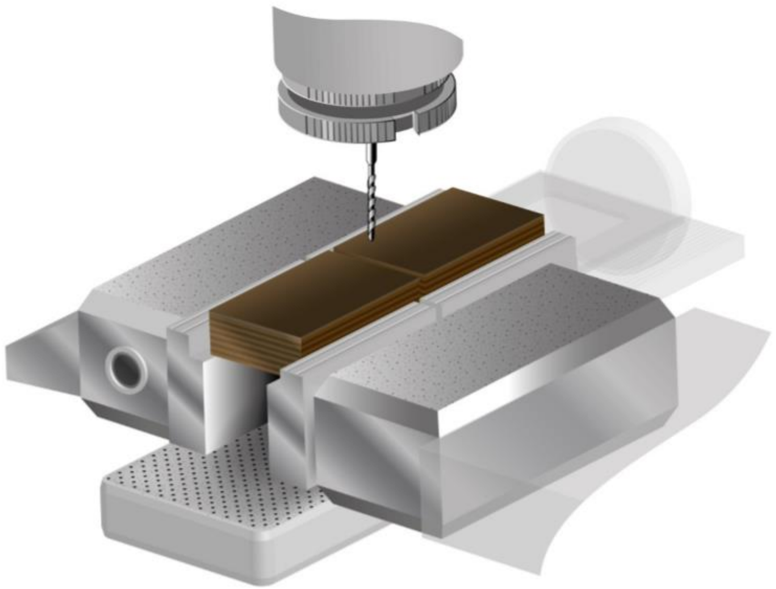


# Residual stress measurement vs prediction

**Validation of residual stresses prediction using incremental slitting method.**

**Courtesy National Physics Laboratory in the UK.**



The material used in this study was SE84 LV (Toray T700 UD HS) carbon fibre-reinforced epoxy unidirectional tape, supplied by Gurit (UK) Ltd.

The nominal cured ply thickness was 0.3 mm.

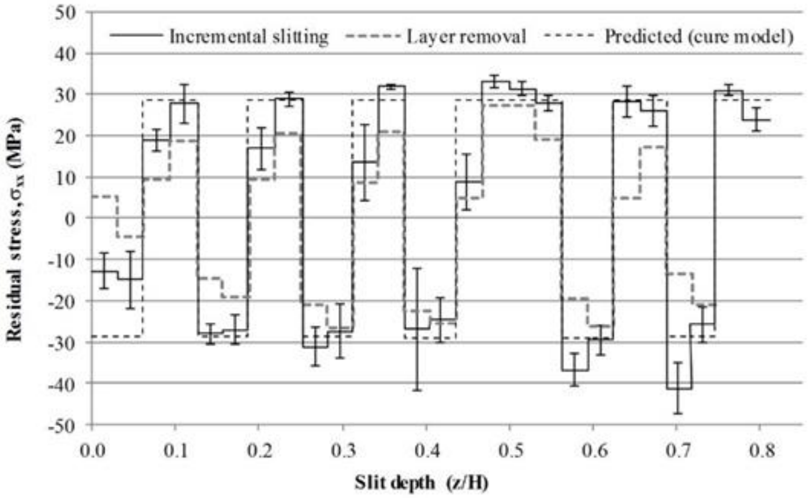
The stacking sequence was $[0°_2/90°_2]_{4s}$

Determination of residual stresses in a laminated thermoset composite using the incremental slitting method, Gower M, Shaw R, Wright L, Hughes J, Garstka T, Submitted to Composites Part A

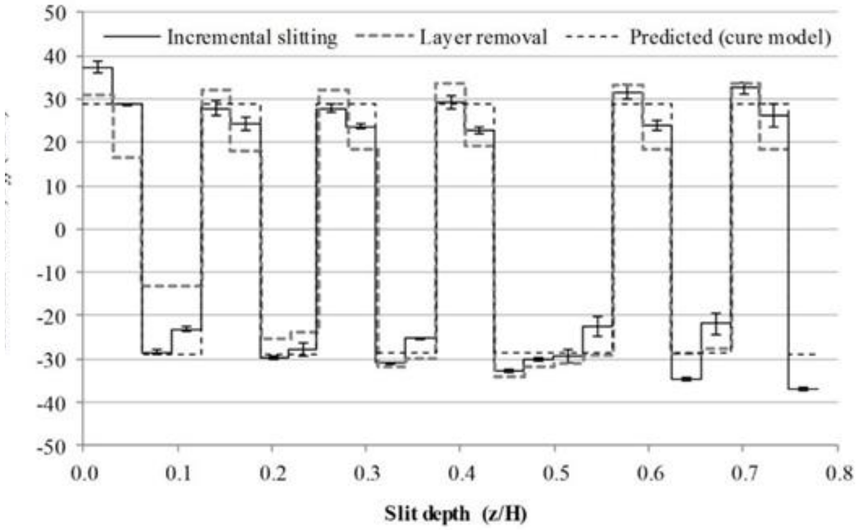
# Residual stress measurement vs prediction

**Courtesy National Physics Laboratory in the UK**

**Fibre direction stresses**




**Transverse stresses**



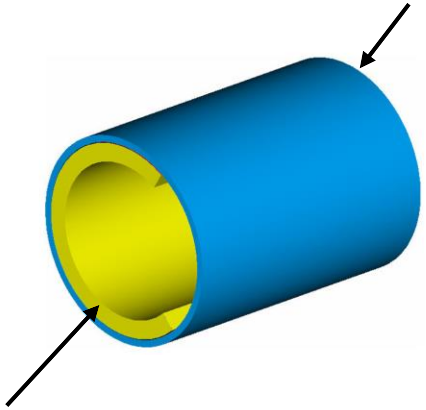

Determination of residual stresses in a laminated thermoset composite using the incremental slitting method, Gower M, Shaw R, Wright L, Hughes J, Garstka T, Submitted to Composites Part A

# Distortions in composite C-shaped laminates

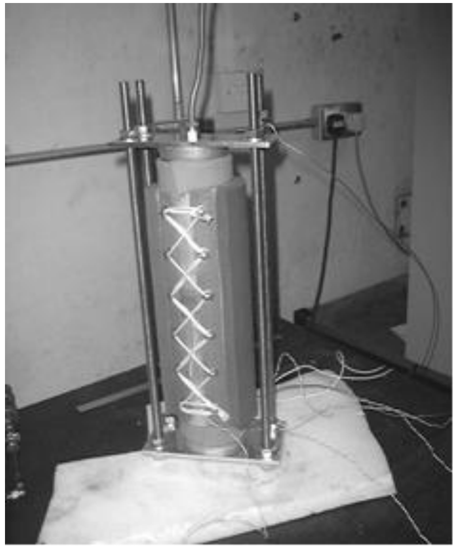
**Composite C-rings were cured on the inside of the composite and aluminium tool following manufacturers recommended cure cycle**

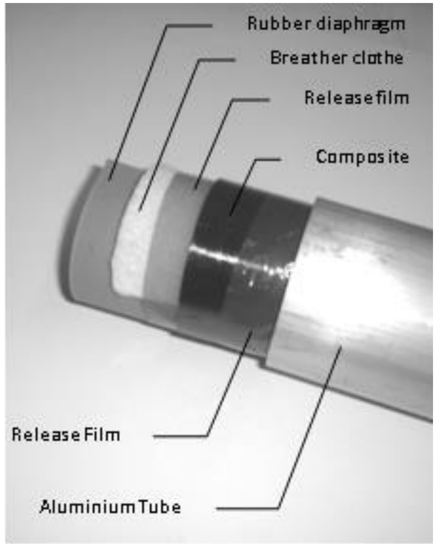
**Specimens were manufactured from AS4-8552 laminate with stacking sequence $[0,90]_{ns}$**

**Carbon and Al Tooling**

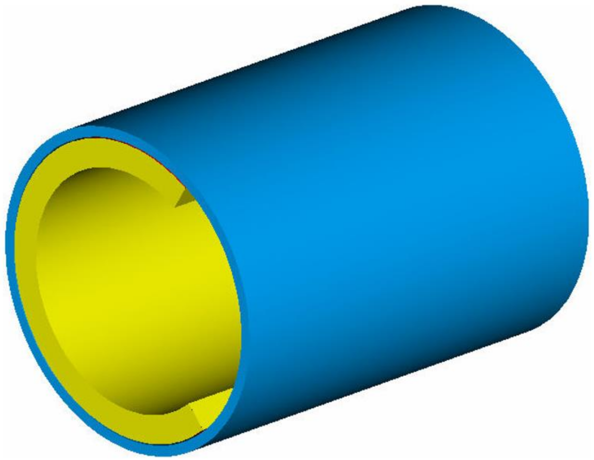


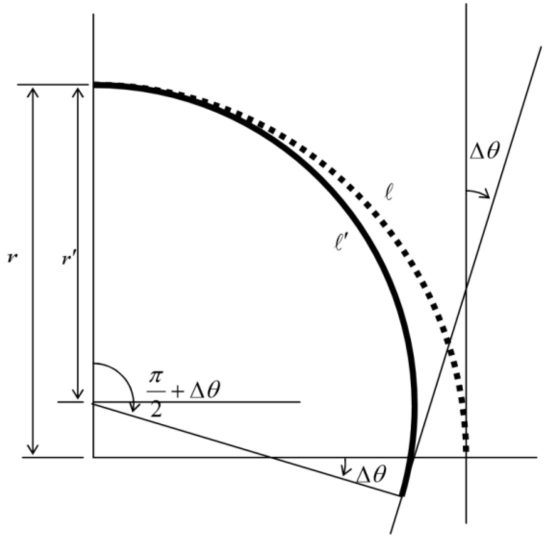
**Composite C-shape**
**Thickness 0.25mm-4mm**

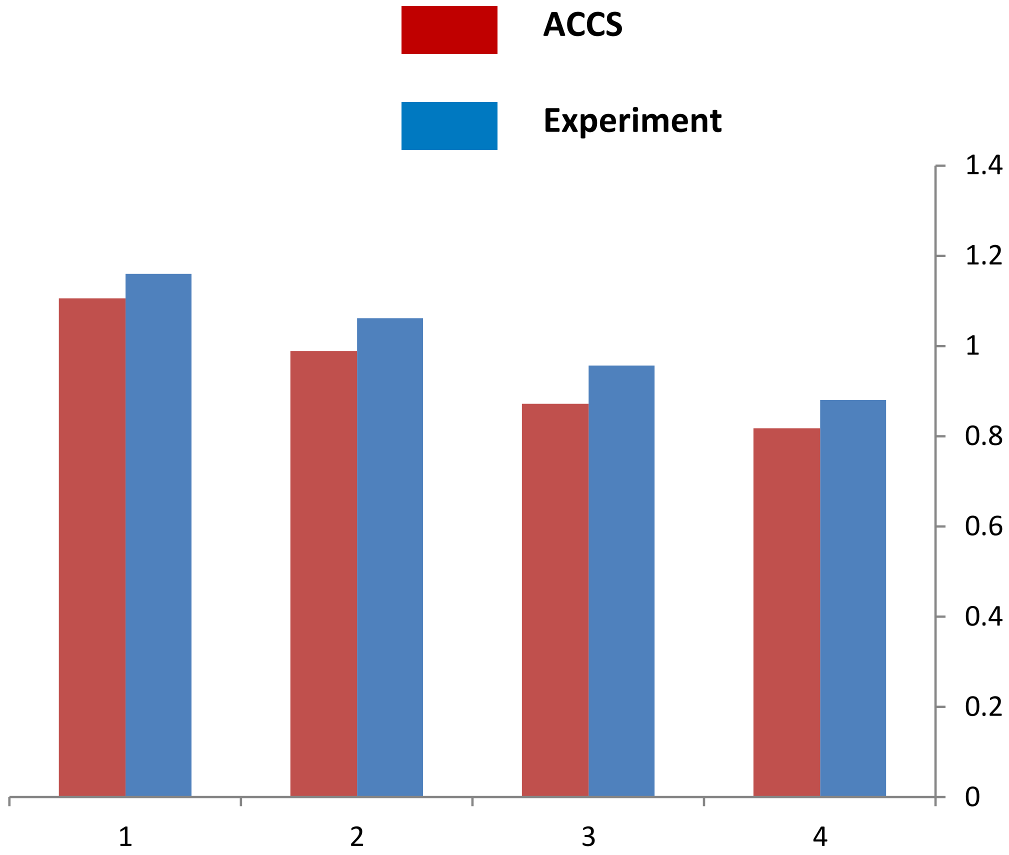




# Distortions in composite C-shaped laminates



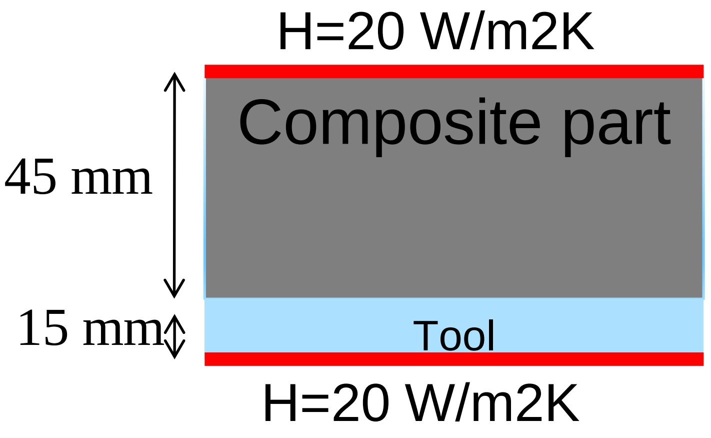


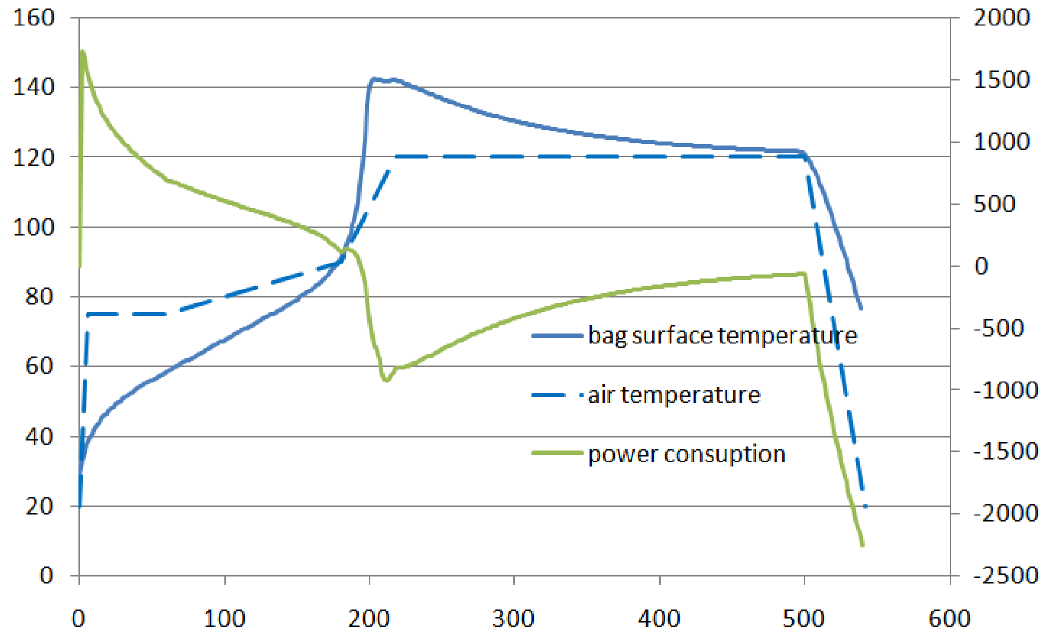



# Energy consumption and exotherms during cure

ACCS allows to calculate power at every step of the cure cycle

Total energy consumed by the laminate as well as energy released by the exothermic reaction can be calculated allowing for better process control and oven design



# ACCS Summary and conclusions

Curing of epoxy composites is complex and may lead to significant built in stresses and distortions

The exothermic reaction in epoxies may also cause considerable problems during manufacturing especially in the case of thick laminates

ACCS allows to predict development of cure and subsequently build up of residual stresses

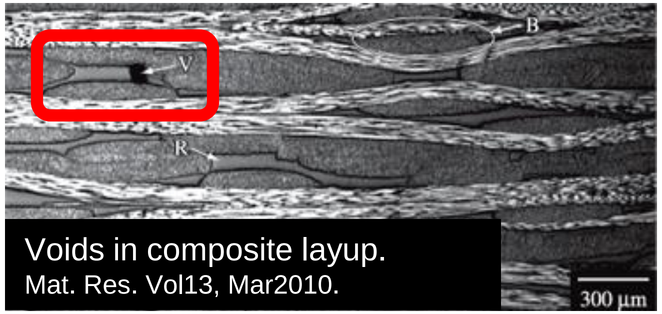
ACCS allows removes the need for costly trial and error approach in tooling design. It significantly shortens product development time and the overall process cost.

# R19.0 Composites Update

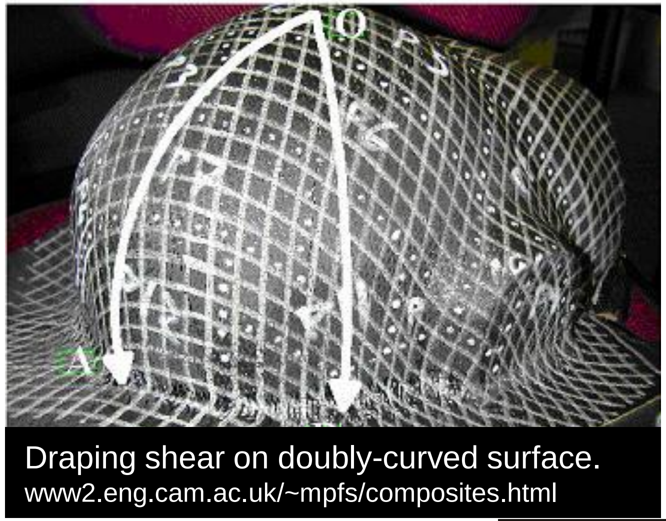
# Field-dependent Material Properties

**Control properties as a function of scalar fields such as Draping Shear, Temperature and any user-defined field.**

- Variable properties
  - Stiffness
  - Strength
  - Density
- NEW: Scope field definitions to
  - Element Sets (Mechanical Named Selections)
  - Oriented Selection Sets
  - Modeling Plies
- Up to 9 user-defined scalar fields




Voids in composite layup.
Mat. Res. Vol13, Mar2010.



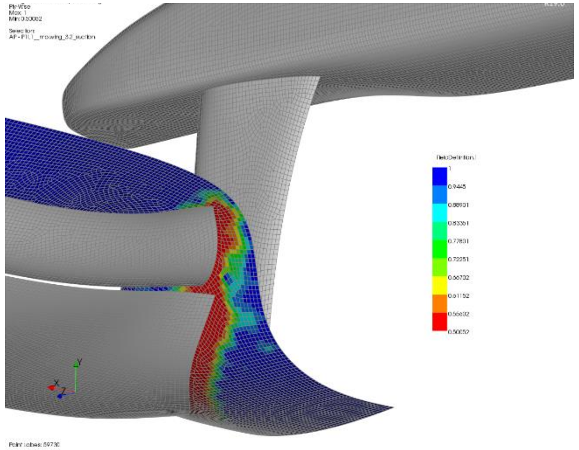
Draping shear on doubly-curved surface.
www2.eng.cam.ac.uk/~mpfs/composites.html

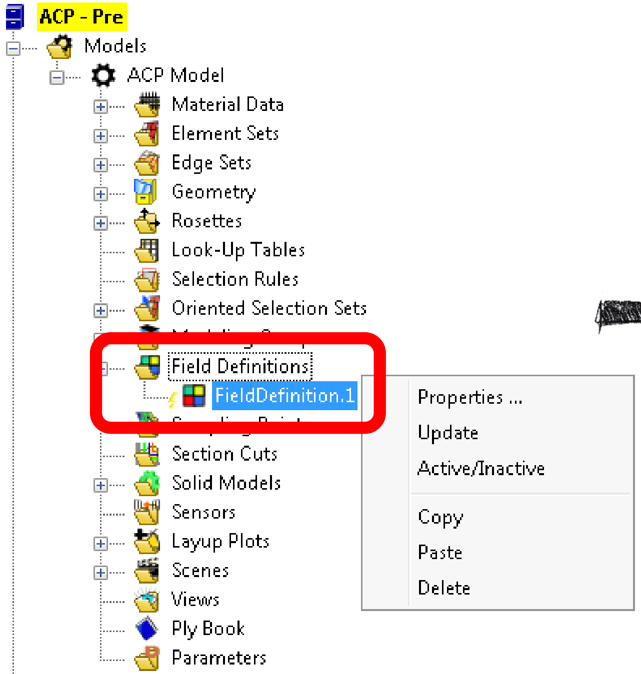
   

# Field-dependent Material Properties

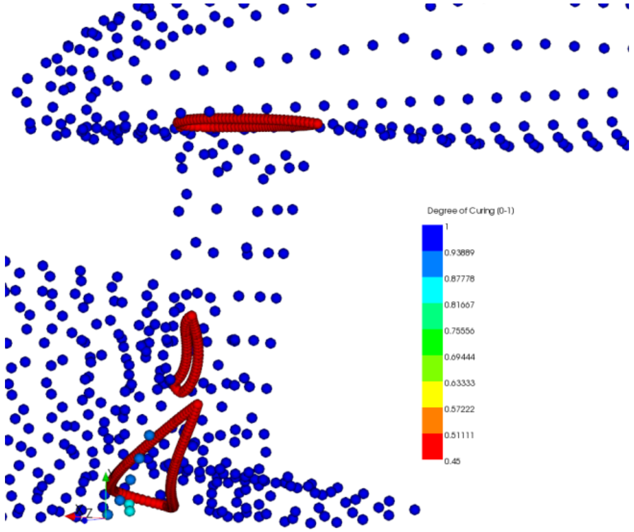
- **Scalar field definition using ACP 3D-Tables**
- **NEW: Scoping on model down to ply-level**
- **Undefined regions assume default field state**

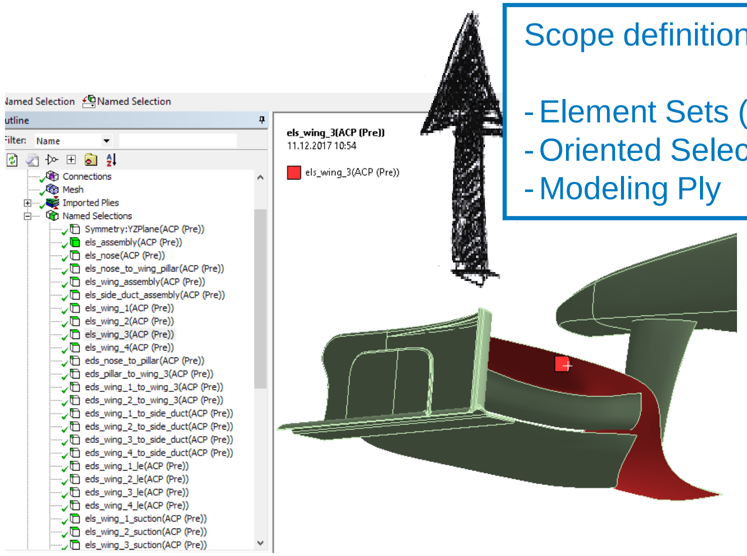
Final result of Incomplete Curing for ply P1L1_mp.wing_3.1_suction






3D Look-up Table point cloud, which will interpolate on the selected model scope.



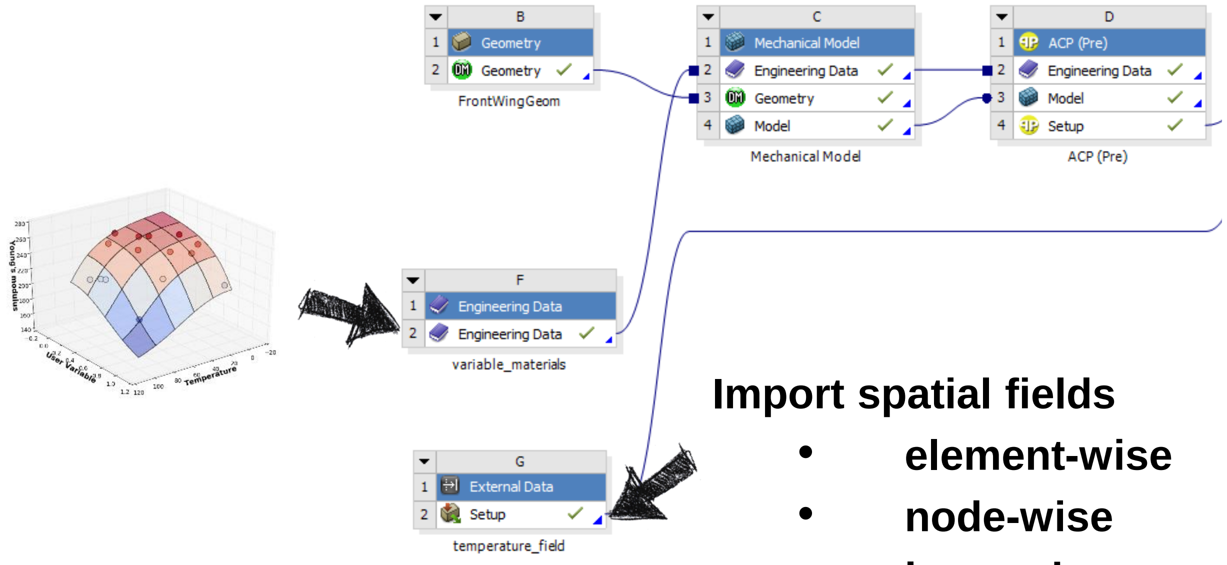




Scope definition to

- Element Sets (Named Selections)
- Oriented Selection Sets
- Modeling Ply

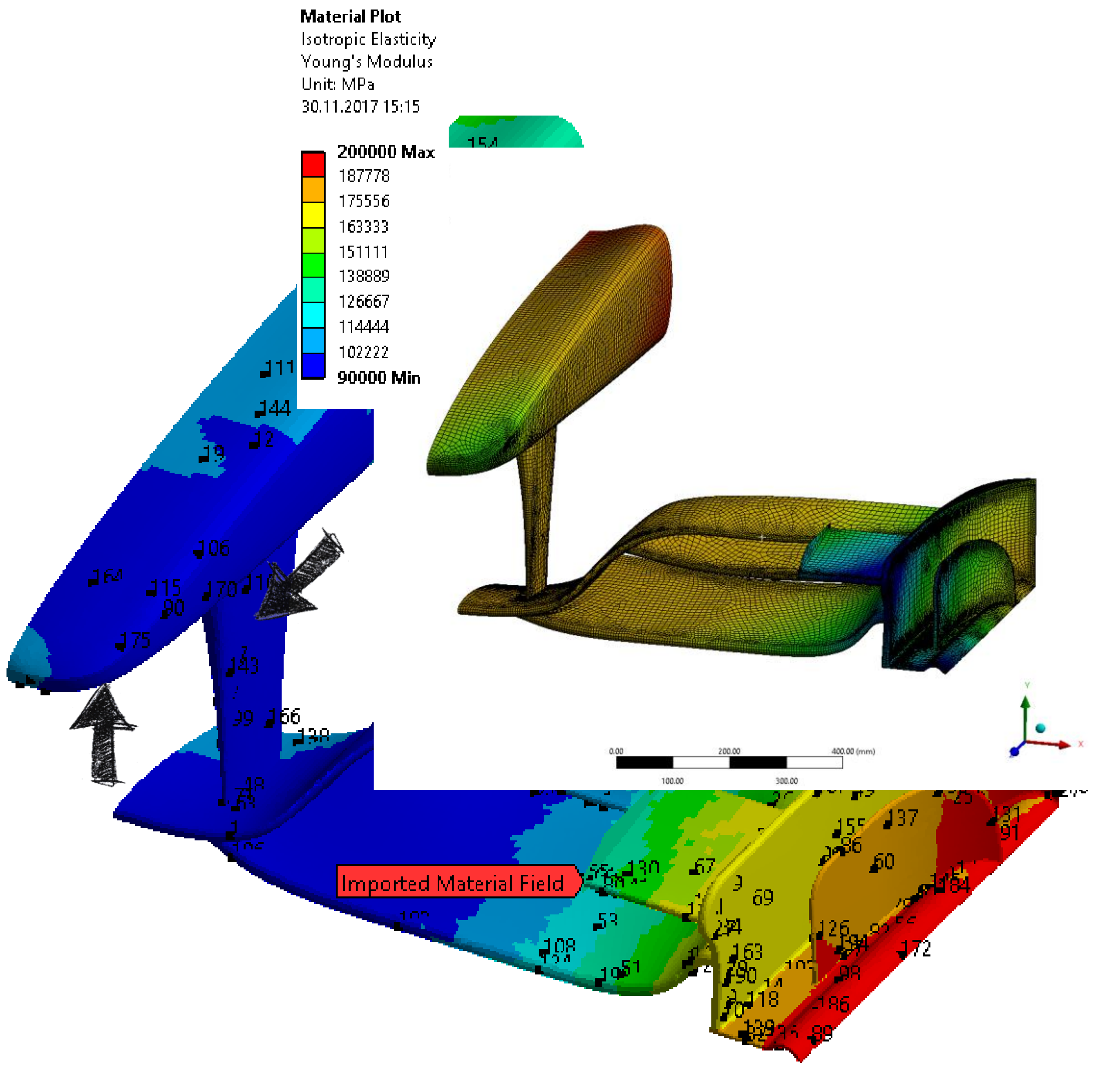
# Field-dependent Material Properties: Other Enhancements

- **Variable density**
- **Material plots in Mechanical**
- **Improved interpolation algorithms**




**Import spatial fields**

- element-wise
- node-wise
- layer-wise

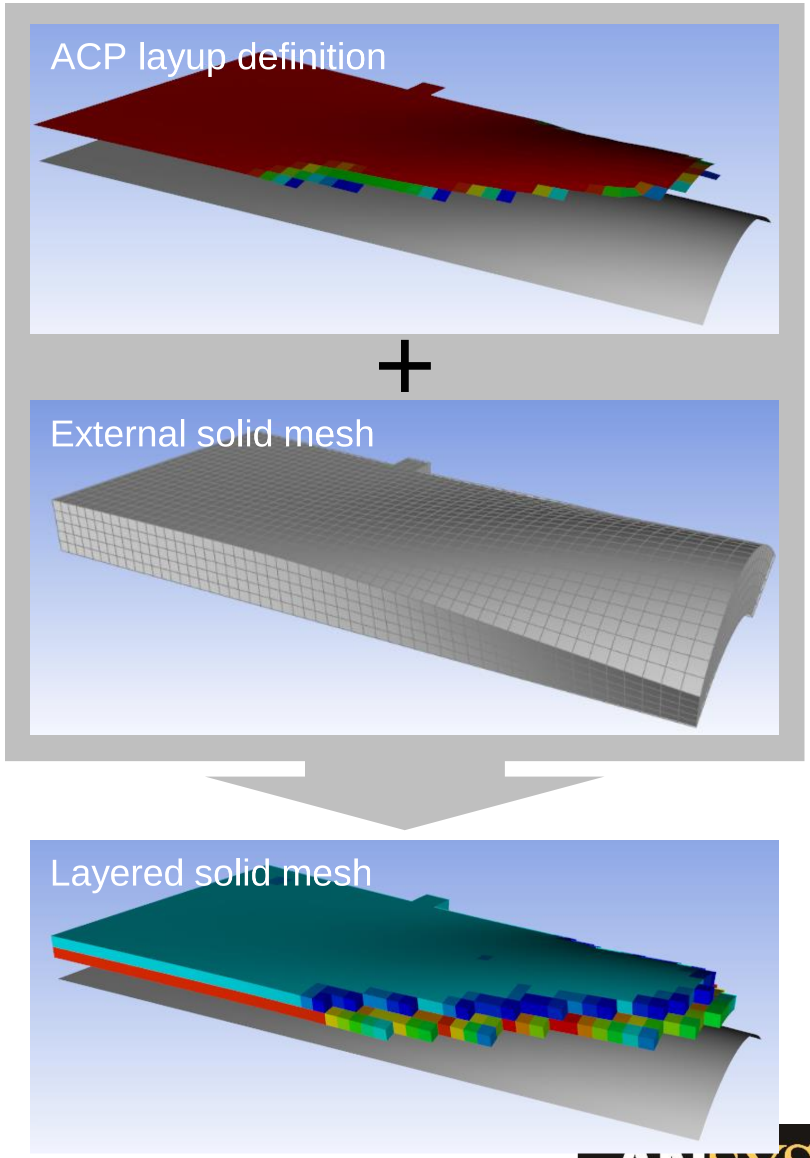


# Mapping of a Composite Lay-up onto a Solid Mesh

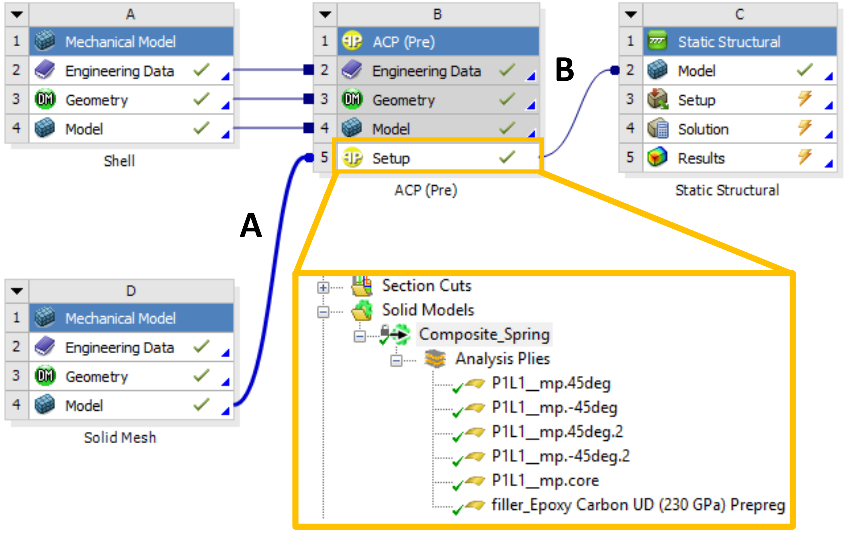
**Map shell-based composite definitions of ACP onto an external solid mesh to build a layered composite solid model.**

- **ACP automatically builds a 3D representation of the lay-up**
- **Solid mesh is generated independently of the lay-up and outside ACP**
- **Use cases:**
  - **Full cross-section composites, turbine blade**
  - **Where the standard solid model (extrusion) of ACP is not feasible**

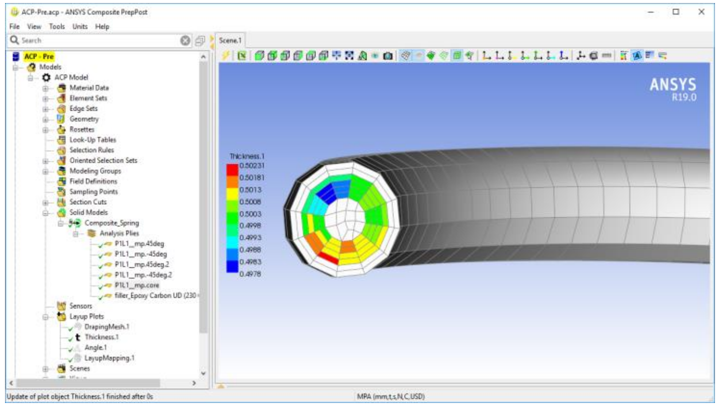


# Layup Mapping: Workflow

- Build the composite lay-up in ACP Pre
- Solid mesh is generated independently of the lay-up and outside ACP
- Pass the solid mesh to ACP (A)
- Configure the scope of the layup mapping
- Use layered solid mesh in downstream analysis (B) as used to



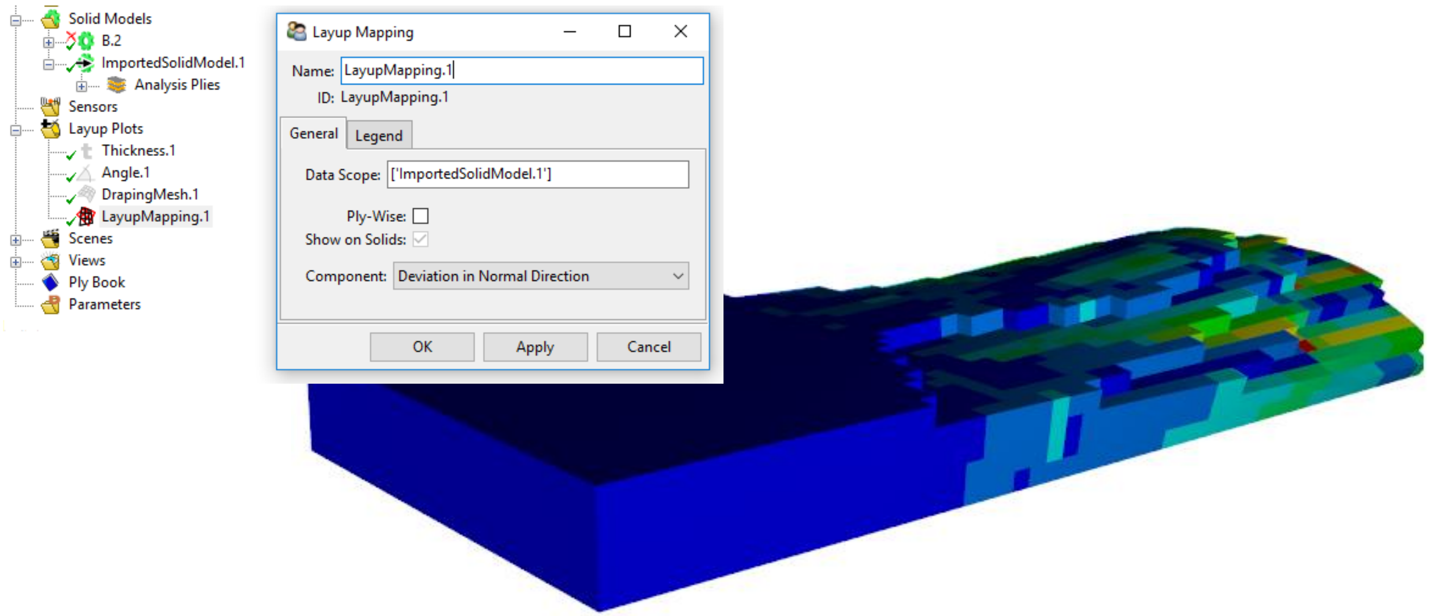

**Video**



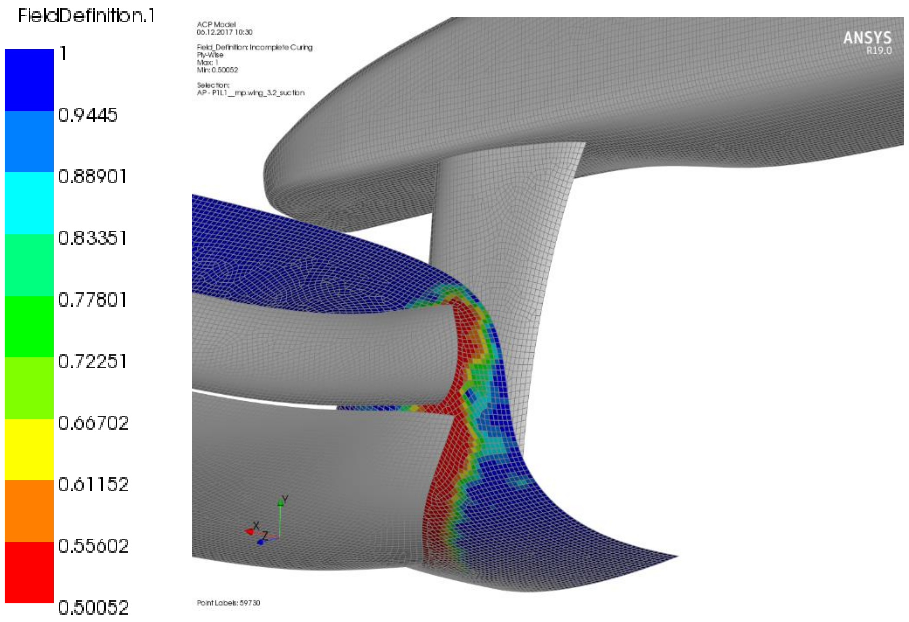

*Lay-up mapping for a full cross-section composite spring*

# New Plots

- **Field Definition Data**
  - **Plot field distributions**
- **Layup Mapping Data**
  - **Shows results of the layup mapping**



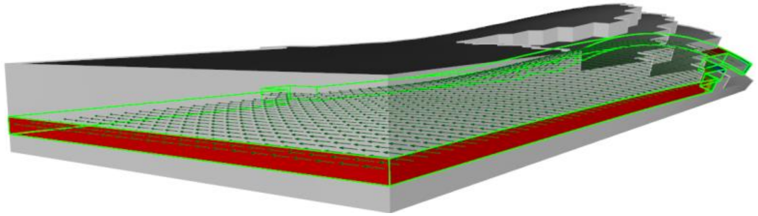

**Ply-wise field definition plot**



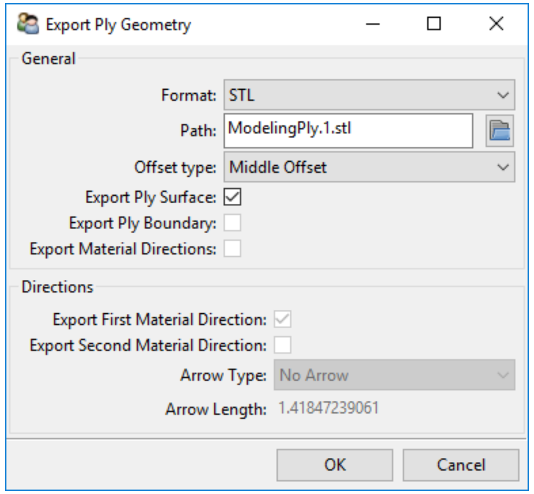
# Miscellaneous

- Direction arrows on solid elements (A)
- New export format for ply geometries: STL (B)
- Export of skin/envelop of the solid meshes as IGES, STEP, STL and CDB (C)
- Thickness plot: new component *relative thickness correction*
- Support of worksheet (mesh) based Named Selections (D)
- Performance: update of the layup definitions is up to 10% faster, solid model generation up to 40%.

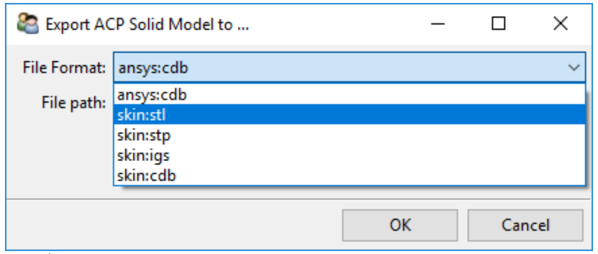
A



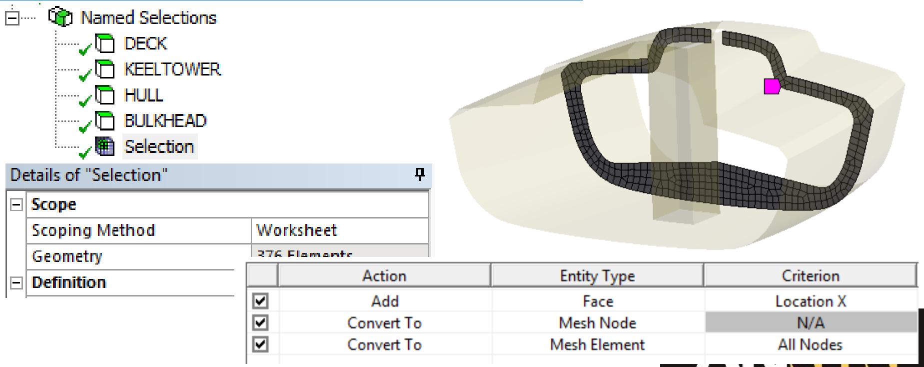
B




C



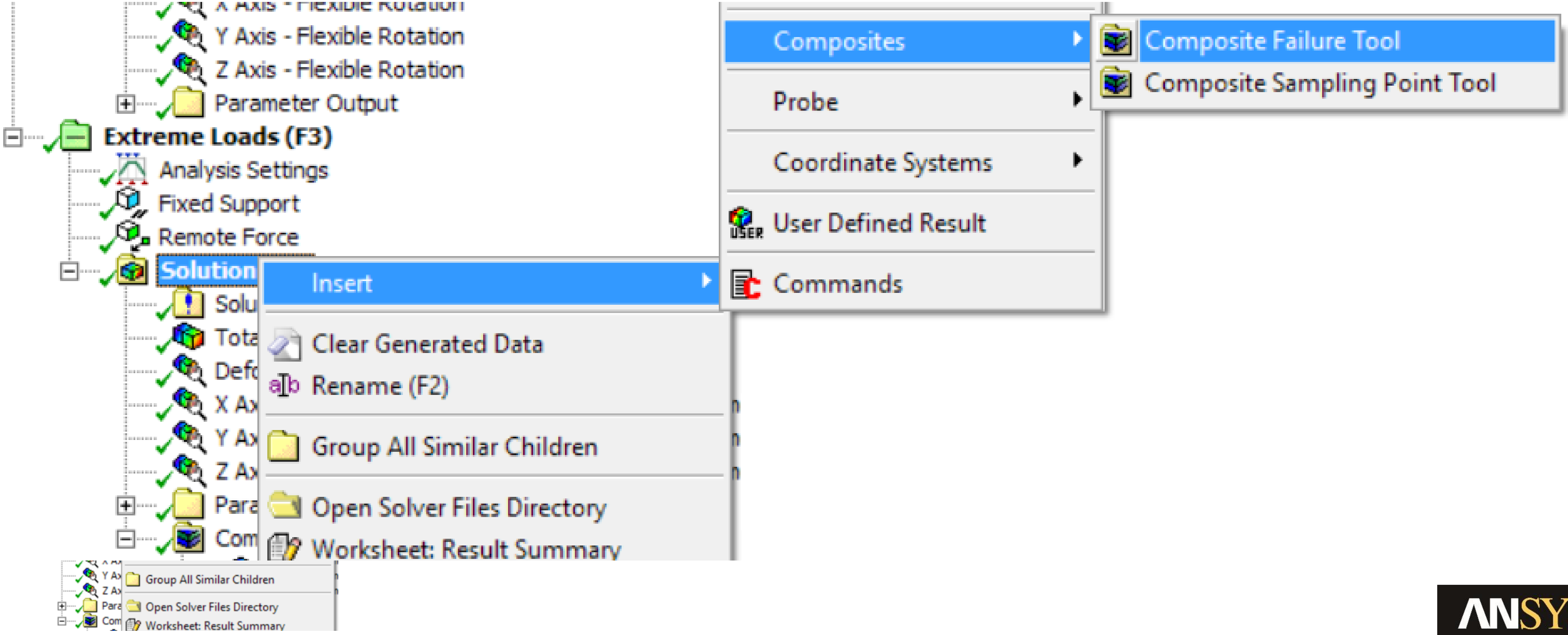

D




  July 5, 2018 

# Composite Failure Tool or ACP – what to use?

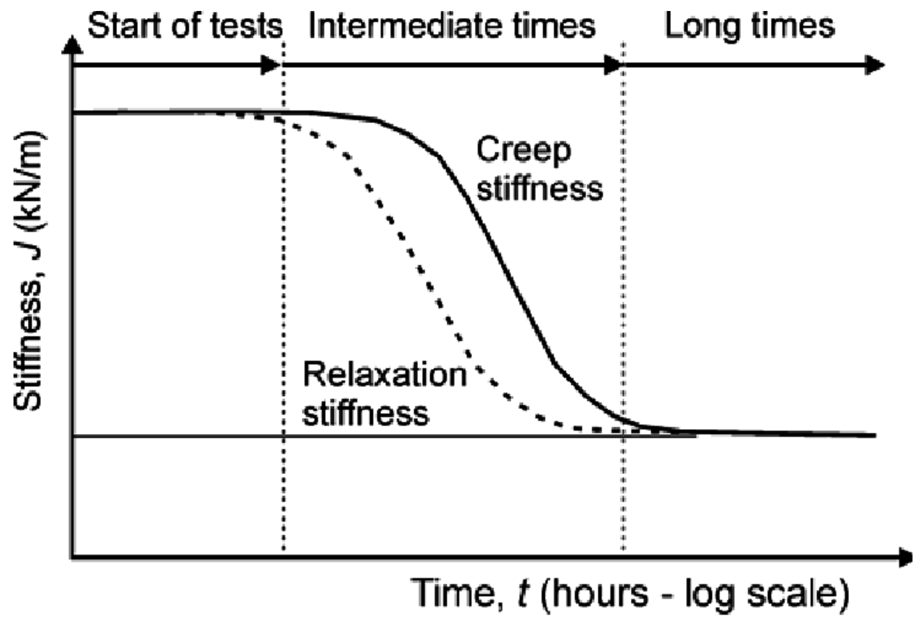
- You can use both Mechanical and ACP-Post to perform composite specific post-processing.



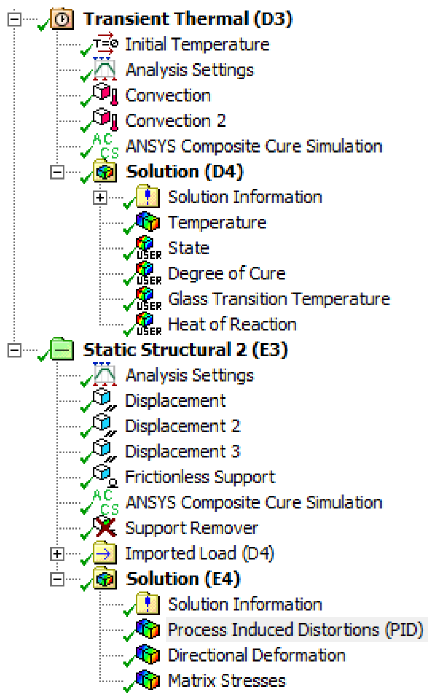
# Viscoelasticity in ANSYS Composite Cure Simulation (ACCS)

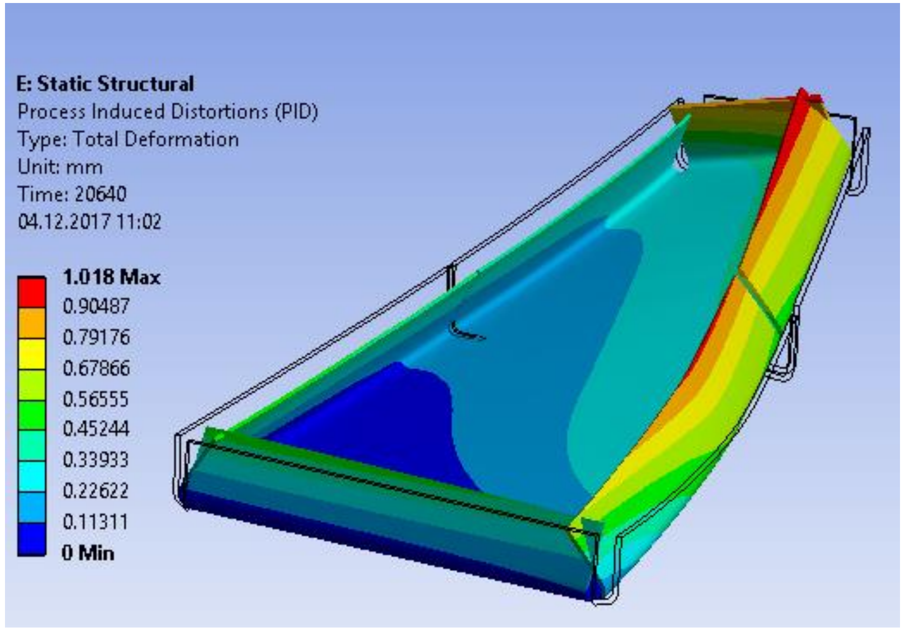
**Linear viscoelasticity was added to allow simulation of material behaviors with strong viscous relaxation effects. Stiffness relaxation are expressed in terms of Prony series. This capability allows to simulate residual stress relaxation effects when component is exposed to in-service loading.**



# New Cure Kinetic Equation in ANSYS Composite Cure Simulation (ACCS)

**AVRAMI – Erofeev cure kinetics equation was implemented to allow simulation of polymerization reaction with two competing reactions. The new equation allows to capture dual exothermic peak during polymerization.**



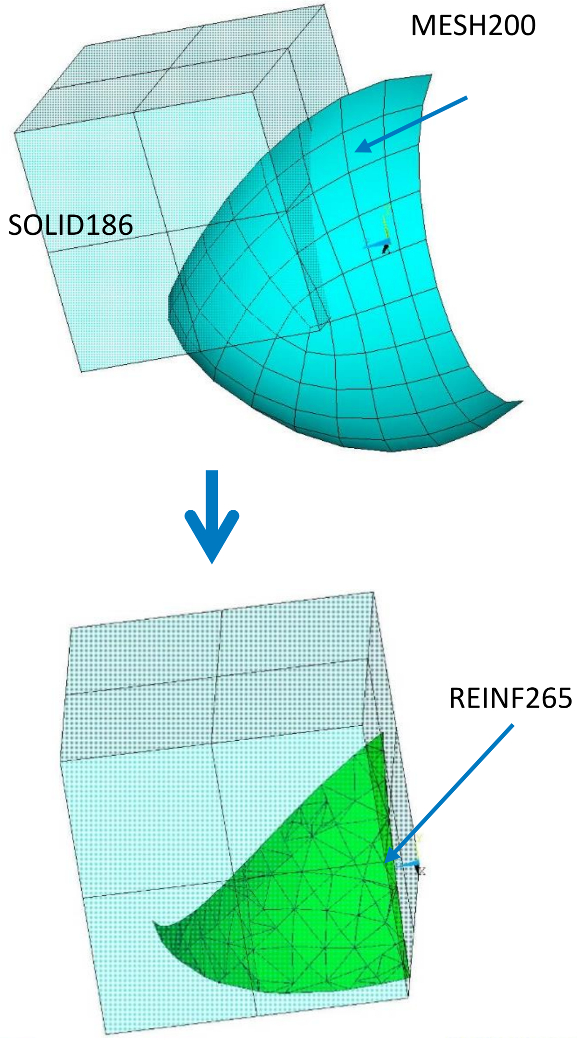
# New MAPDL Composite Related Features At 18.0 and 19.0

- **Mesh Independent Modeling with Higher Order Elements**
  - Supports higher order base elements, including SOLID186, SOLID187, and SHELL281 elements
  - Improved geometrical modeling and solution accuracy
  - Retain the high modeling efficiency
- **Full Membrane Reinforcing Stiffness**
  - Full plane stress state supported in both 2D and 3D smeared reinforcing
  - Suitable for modeling homogeneous reinforcing materials

# Mesh Independent Modeling with Higher Order Elements

- **Mesh Independent Method Introduced at REV180**
  - **Reinforcing defined independently via MESH200**
  - **Automatic intersecting procedure to create reinforcing elements**
  - **Supports linear and quadratic discrete and 2D smeared reinforcing**
  - **Supports linear 3D smeared reinforcing**

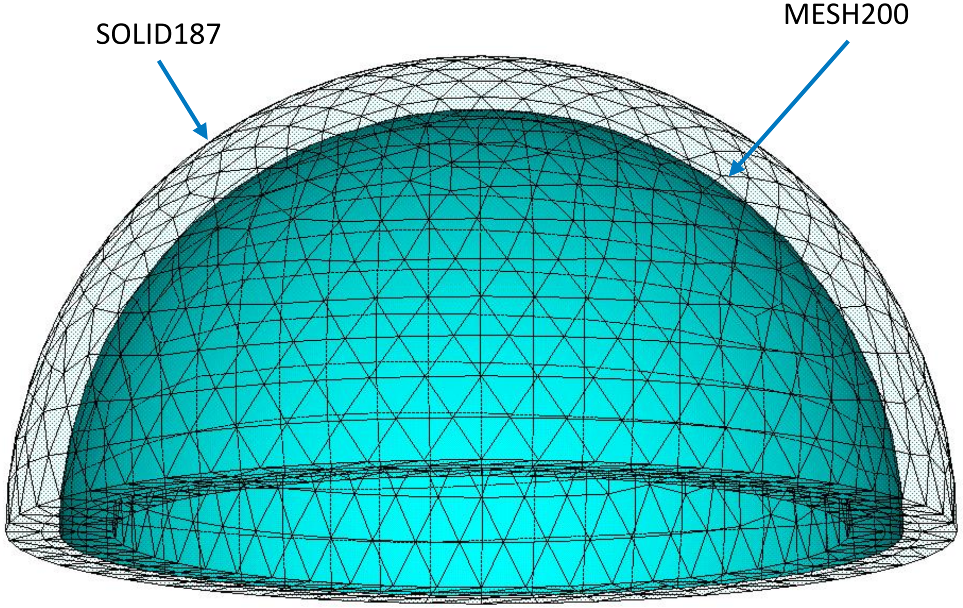
- **New at REV190**
  - **Quadratic 3D smeared reinforcing supported. Mesh independent method now applicable to all elements**
  - **A user friendly procedure introduced for defining reinforcing coordinate systems**

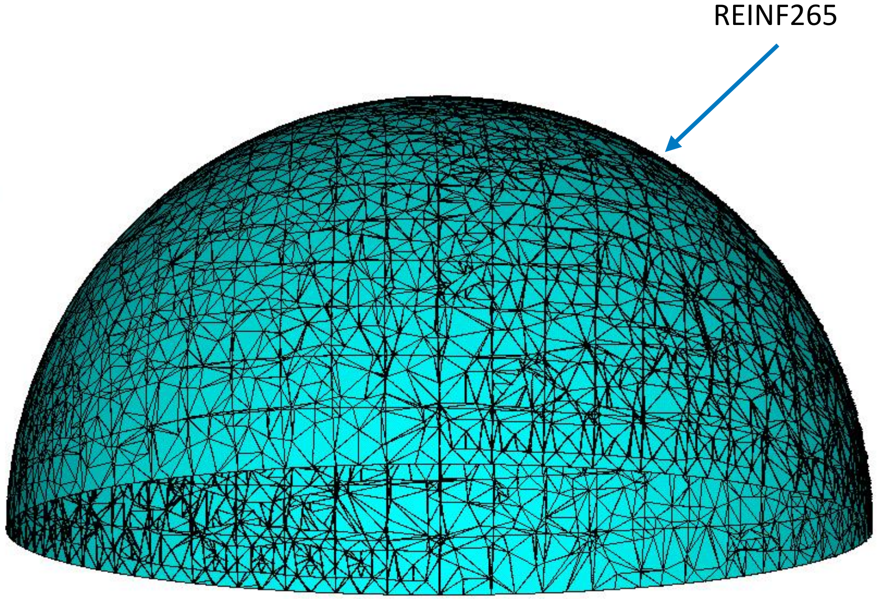



ANSYS

# Mesh Independent Modeling with Higher Order Elements

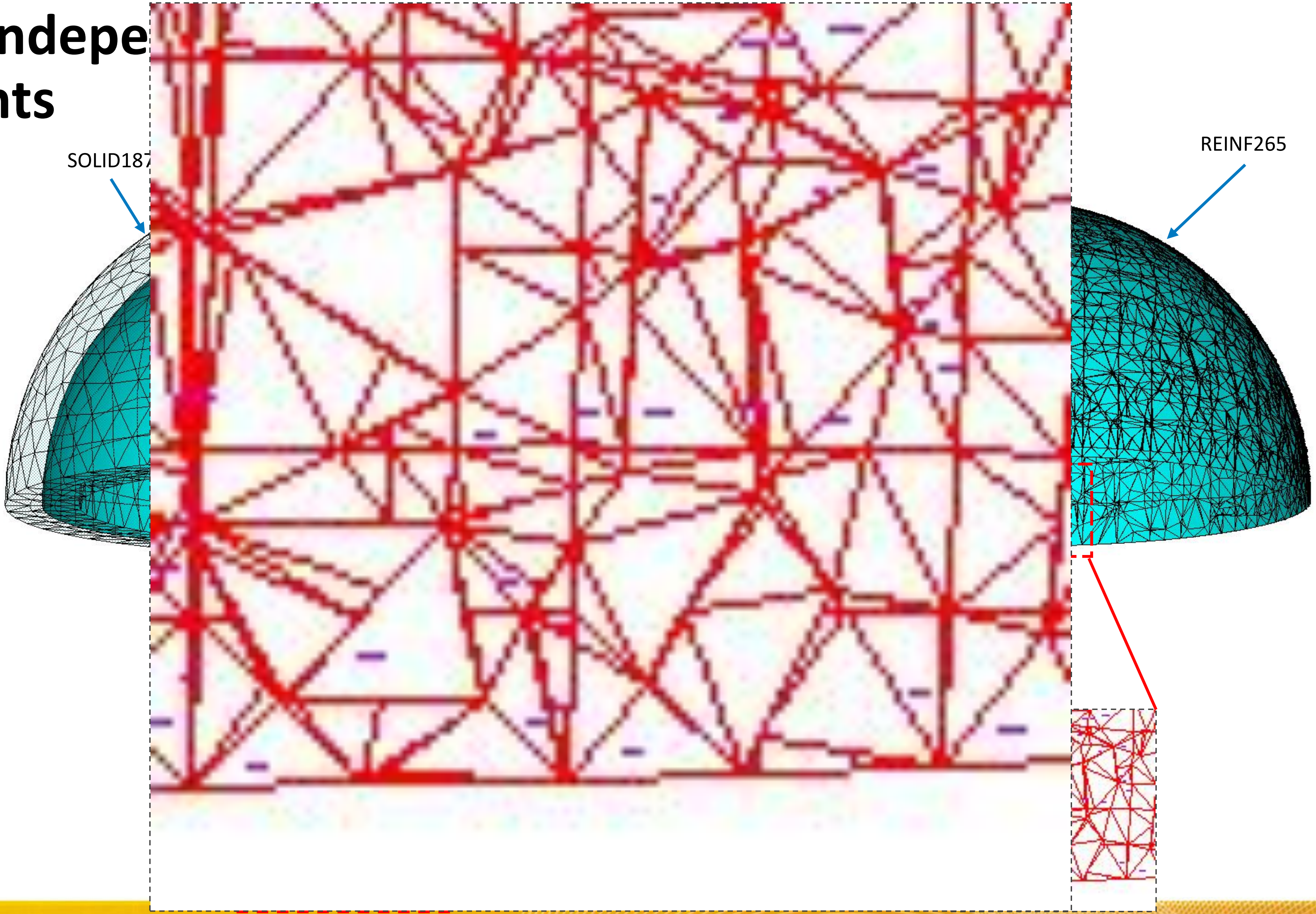
- **Example: Reinforced Cap on a Silo**



# Mesh Indepe
# Elements



ANSYS®

**Thank you!**