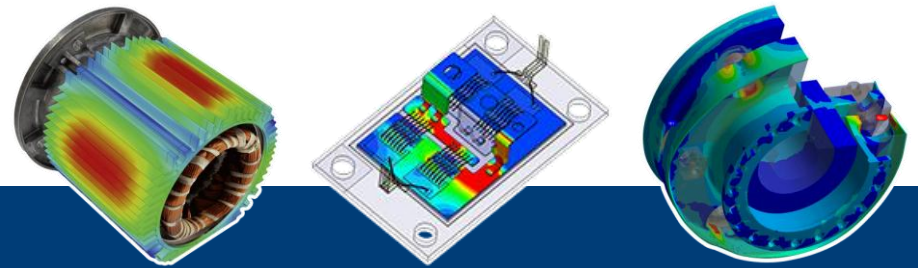


Simulation ist mehr als Software®



ANSYS Composite PrepPost (ACP)

„Composite modellieren, simulieren und bewerten“

5. Fachkongress Composite Simulation am 24. und 25. Februar 2016 in Hamburg

CADFEM

Shell-based Composite Analysis

Solid Composite Analysis

Additional Features

Define Variable Material Properties

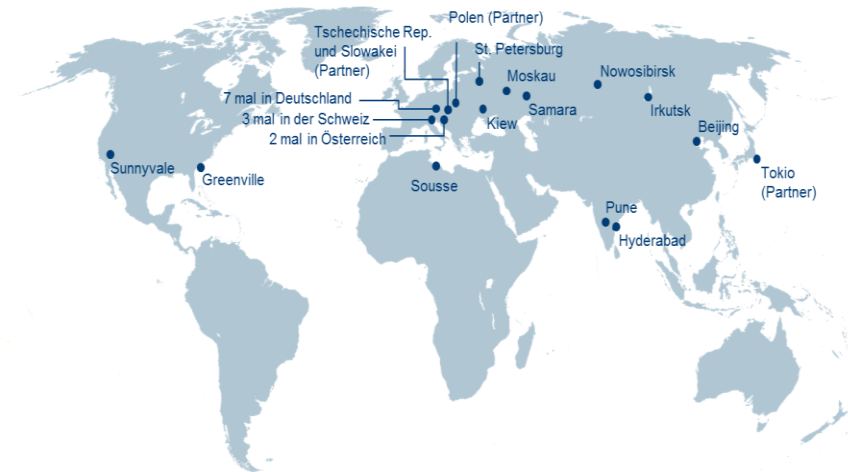
ANSYS Composite Cure Simulation

Simulation macht vieles möglich

- Wegweisende Innovationen entwickeln
- Maßstäbe in der Produktqualität setzen
- Kosten und Entwicklungszeiten reduzieren

CADFEM – CAE-Simulation seit 1985

- CADFEM in D, A, CH
 - 1985 gegründet
 - 12 Standorte
 - 185 Mitarbeiter
 - 60 Millionen Euro Umsatz
 - 2.300 Kunden
- CADFEM worldwide >250 Mitarbeiter
- ANSYS Competence Center FEM in Zentraleuropa
- Teamwork mit ANSYS Germany, ANSYS Competence Center CFD



Competence Center FEM

Weil Software allein noch keinen
Simulationserfolg garantiert, bietet CADFEM
alles, auf was es ankommt, aus einer Hand:

Produkte

Systemhaus:
Software und IT-
Lösungen

Service

Ingenieurdienstleister:
Beratung, Support,
Engineering

Wissen

Wissensanbieter:
Know-how-Transfer

CADFEM

Shell-based Composite Analysis

Solid Composite Analysis

Additional Features

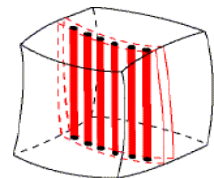
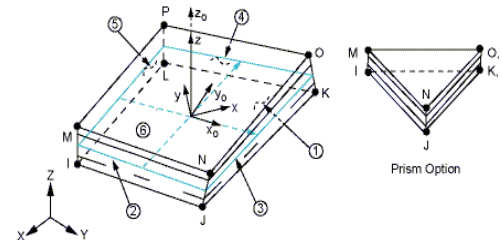
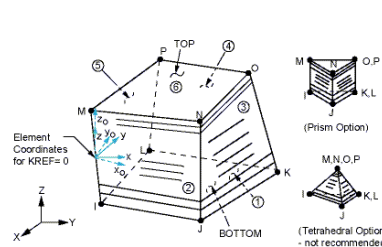
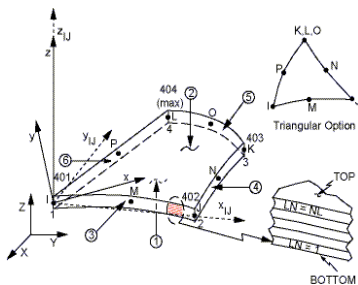
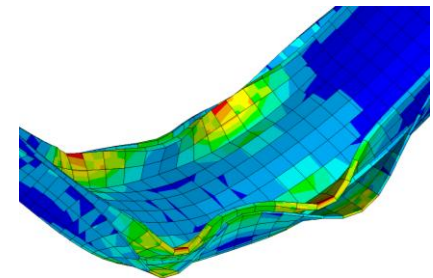
Define Variable Material Properties

ANSYS Composite Cure Simulation

Shell-based Composite Analysis

ANSYS and Composites

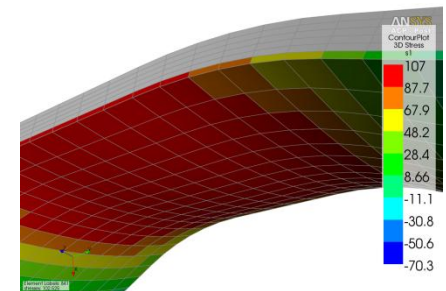
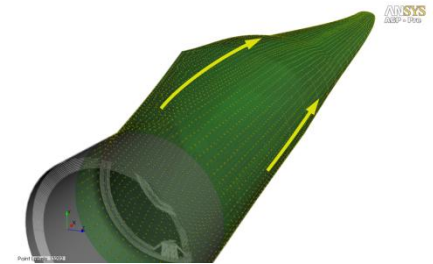
- ANSYS is not new to composite simulations.
 - Layered shell and solid elements for more than two decades.
 - Rich variety of elements for composite modeling from 1D to 3D.



Shell-based Composite Analysis

What can I do with ANSYS Composite PrepPost?

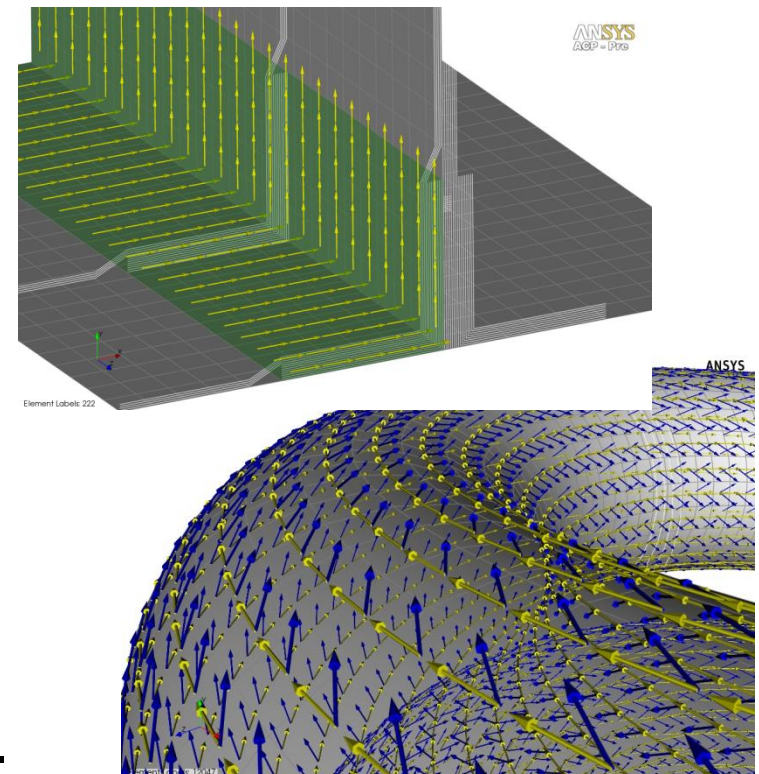
- Build up your Composite Design
 - Define the composite layup
 - Define fiber directions
- Evaluate your Composite Design
 - Evaluate stresses in the layers
 - Evaluate failure criteria



Shell-based Composite Analysis

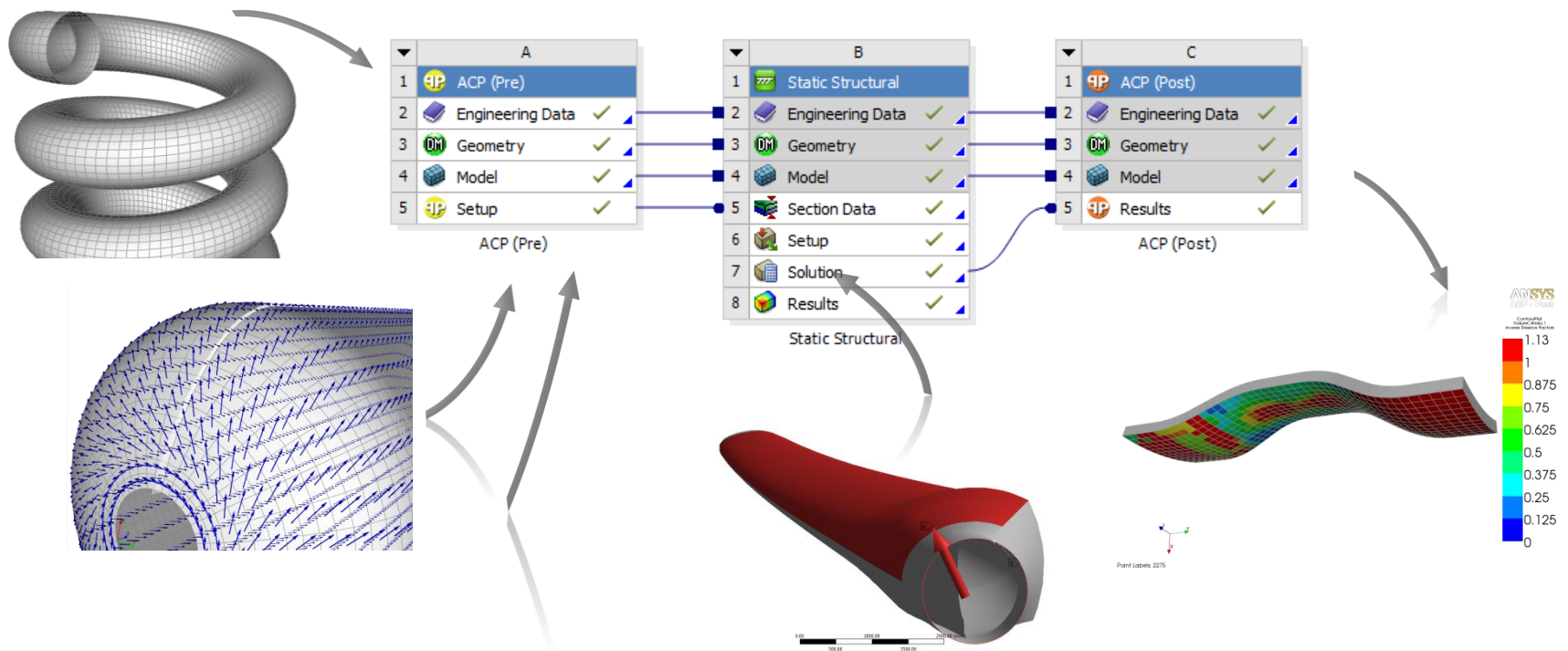
ANSYS Composite PrepPost is the Next Step

- Intuitive workflow for composite modeling.
- Different options to define fiber orientations.
- Quick setup of composite layup.
- Quick postprocessing.
- Allows design studies of different composite designs.



Shell-based Composite Analysis

The Workflow in ANSYS Workbench



Shell-based Composite Analysis

CAD model enhancement

Model preparation

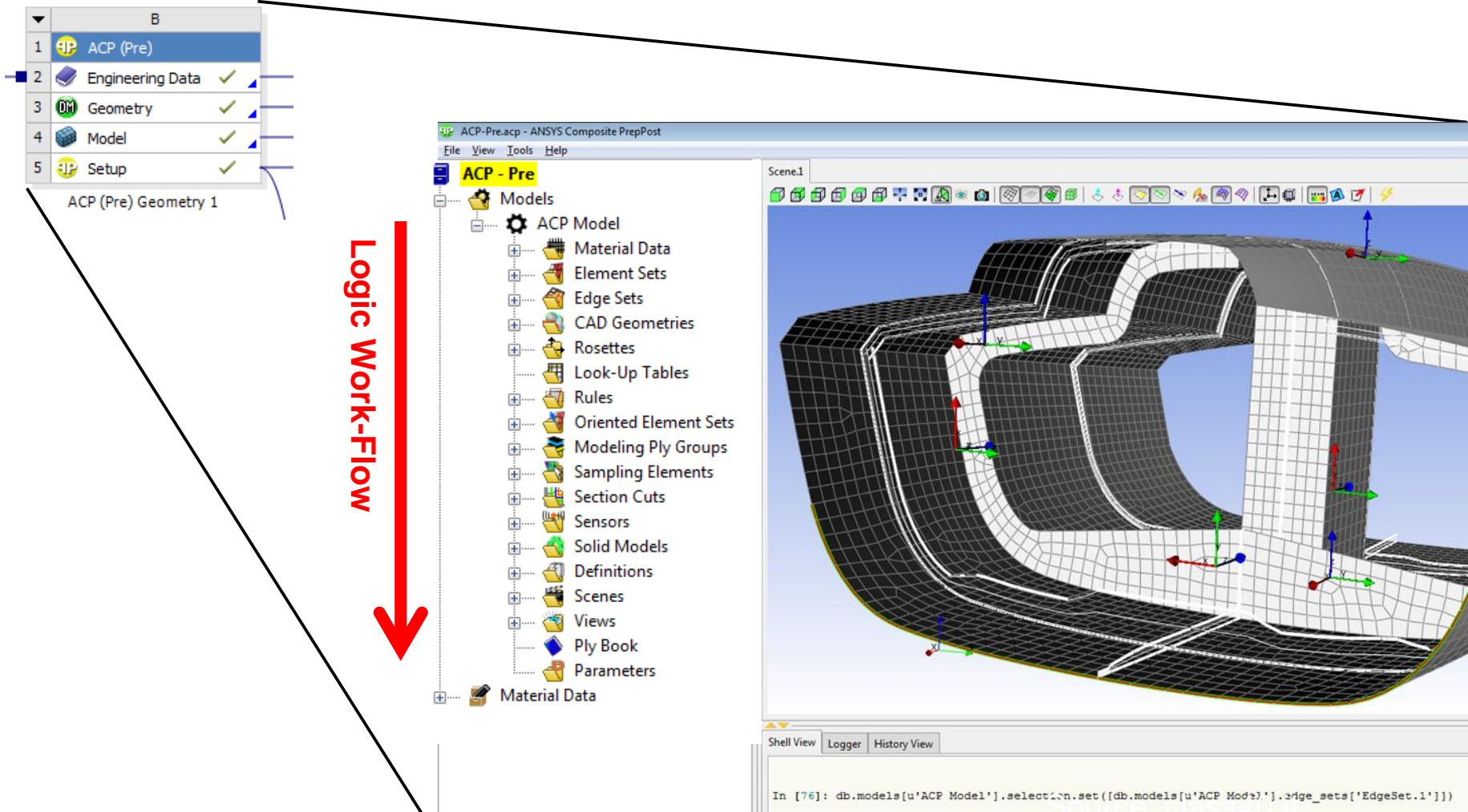
Design Loops & Tools

Design improvement

Composite postprocessing

Composite preprocessing

Shell-based Composite Analysis



Shell-based Composite Analysis

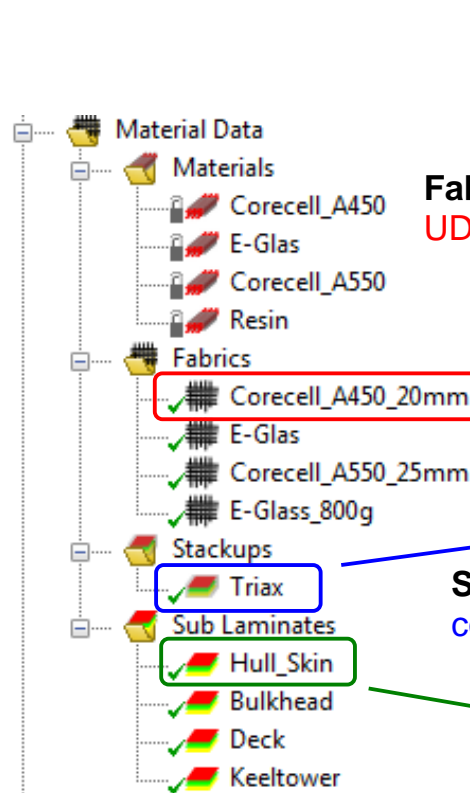
Typical **Fabric**:
Carbon UD



Typical **Stackup**:
Non-crimped
fabric



Material Level



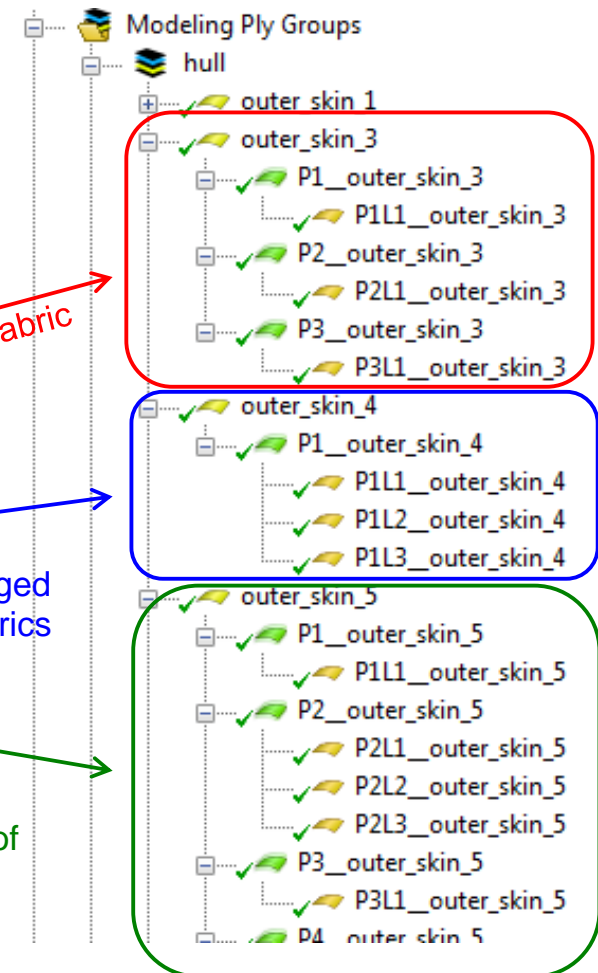
Fabric Single ply,
UD or Woven

3-times same fabric

Stackup Prearranged
combination of fabrics

Sub Laminate Collection of
fabrics or stackups

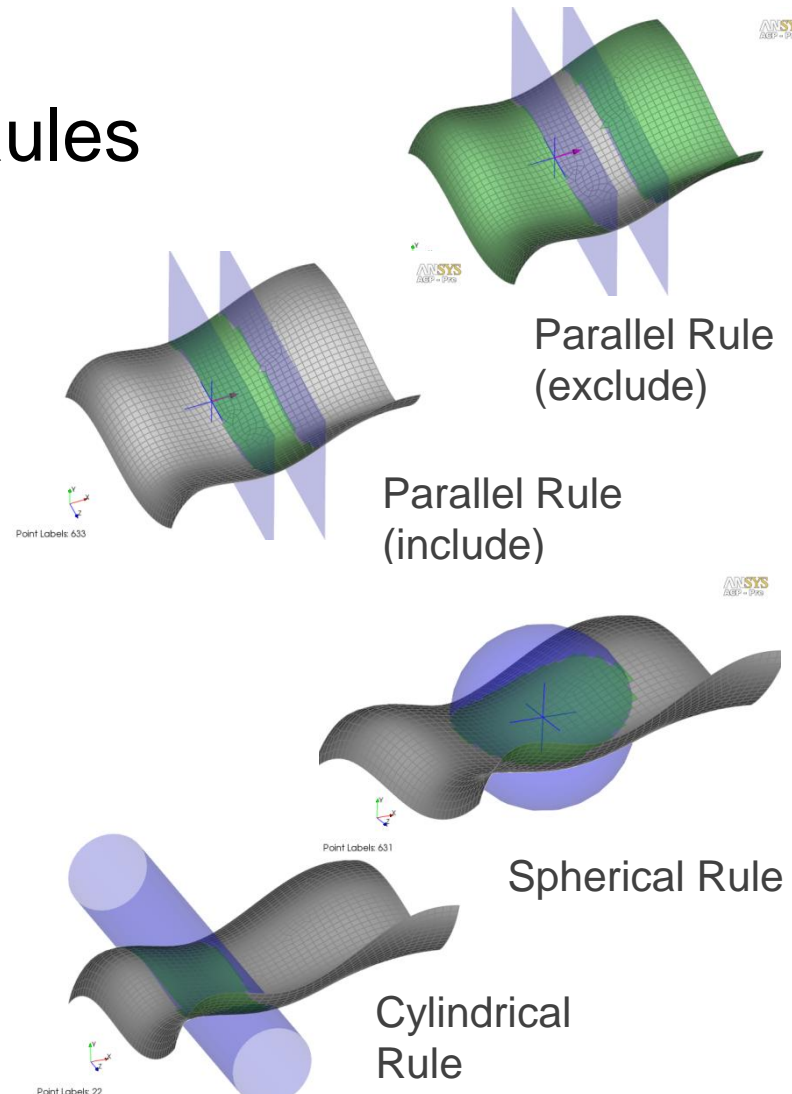
Ply Level



Shell-based Composite Analysis

Layup Surfaces defined by Rules

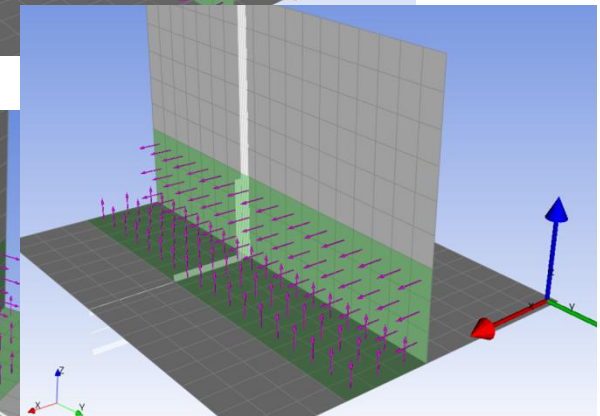
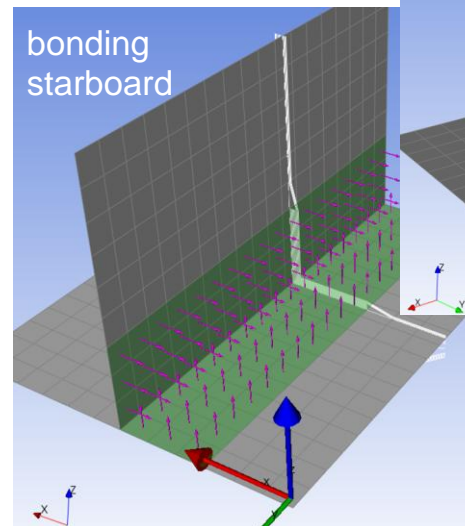
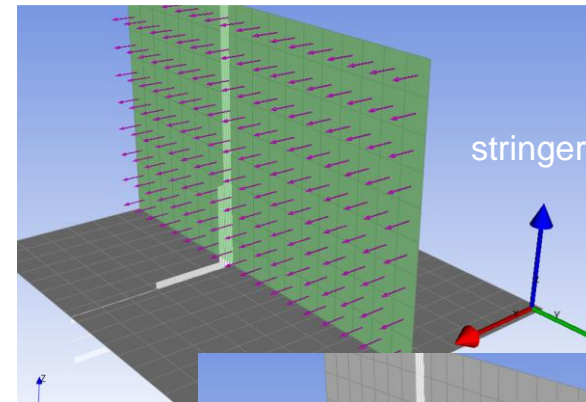
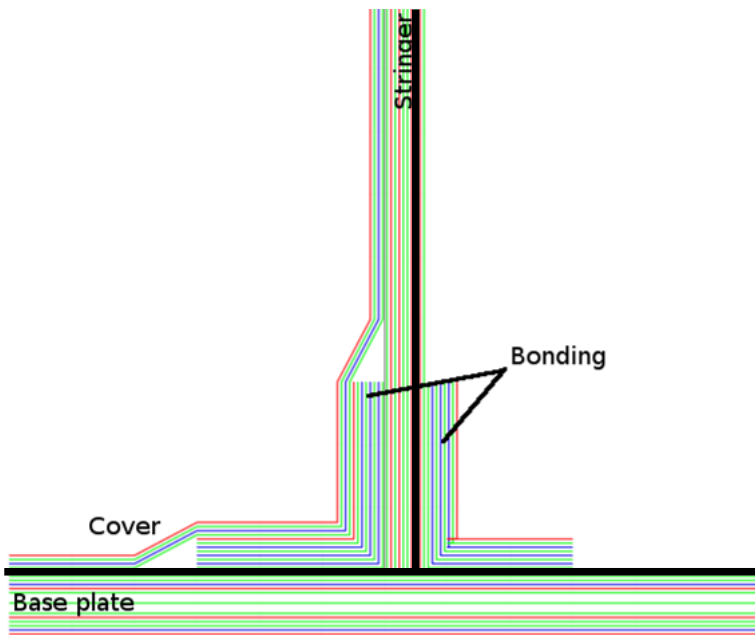
- Rules can be used as include or exclude rule.
- A parallel rule selects all elements between two locations.
- Cylindrical or spherical rules are used to reselect elements within a cylinder or sphere.



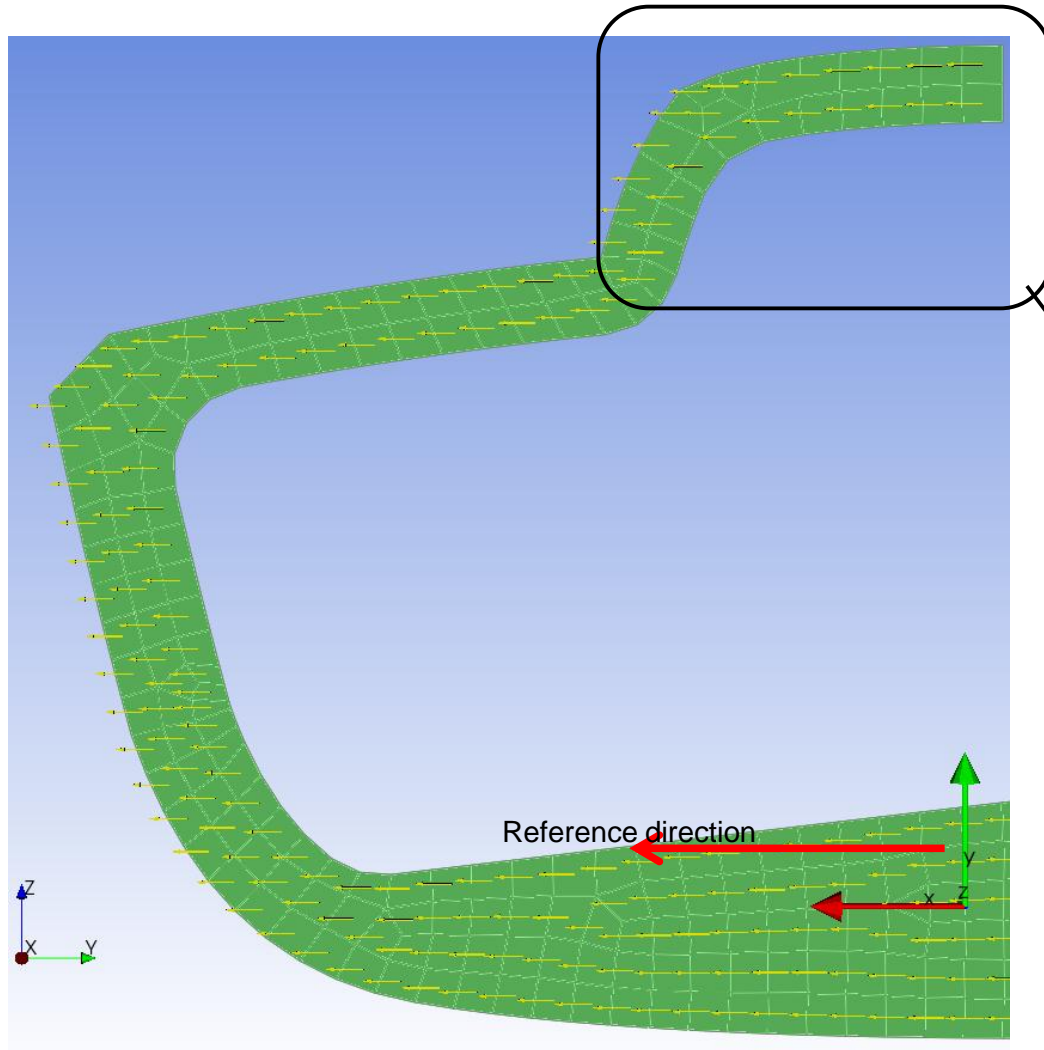
Shell-based Composite Analysis

How to manage arbitrary layup directions?

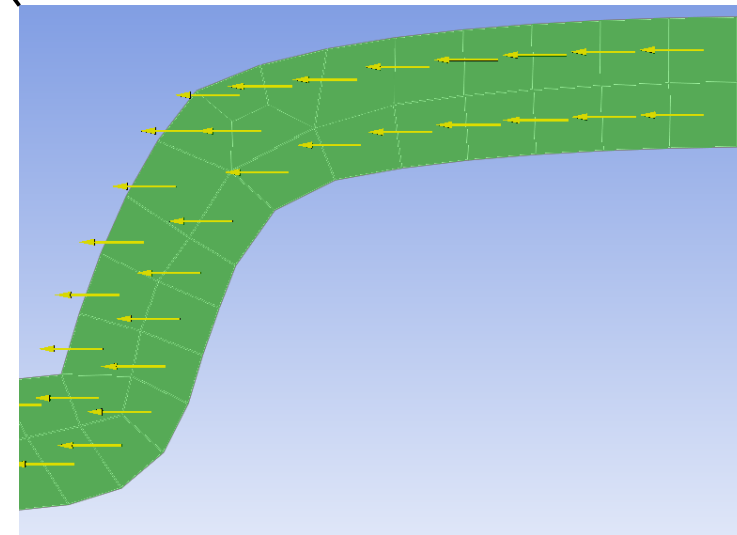
- Define Oriented Element Sets based on Mechanical's Named Selections
- Define multiple Layup Directions per Element



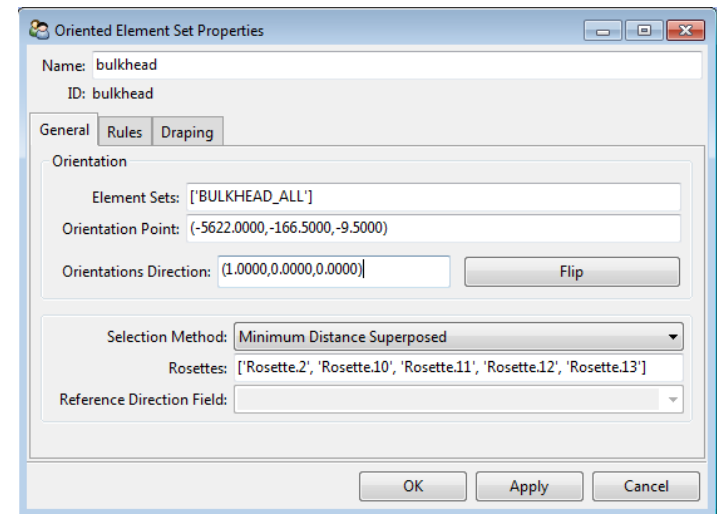
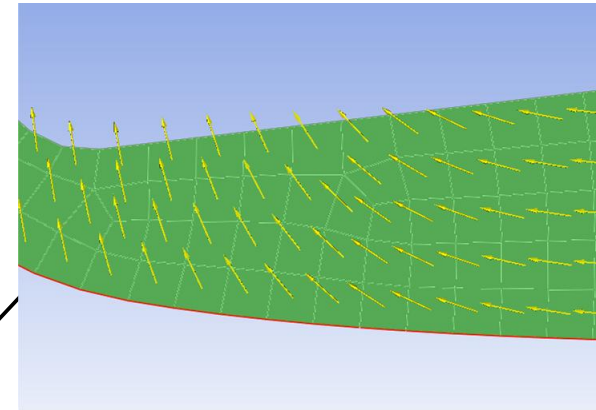
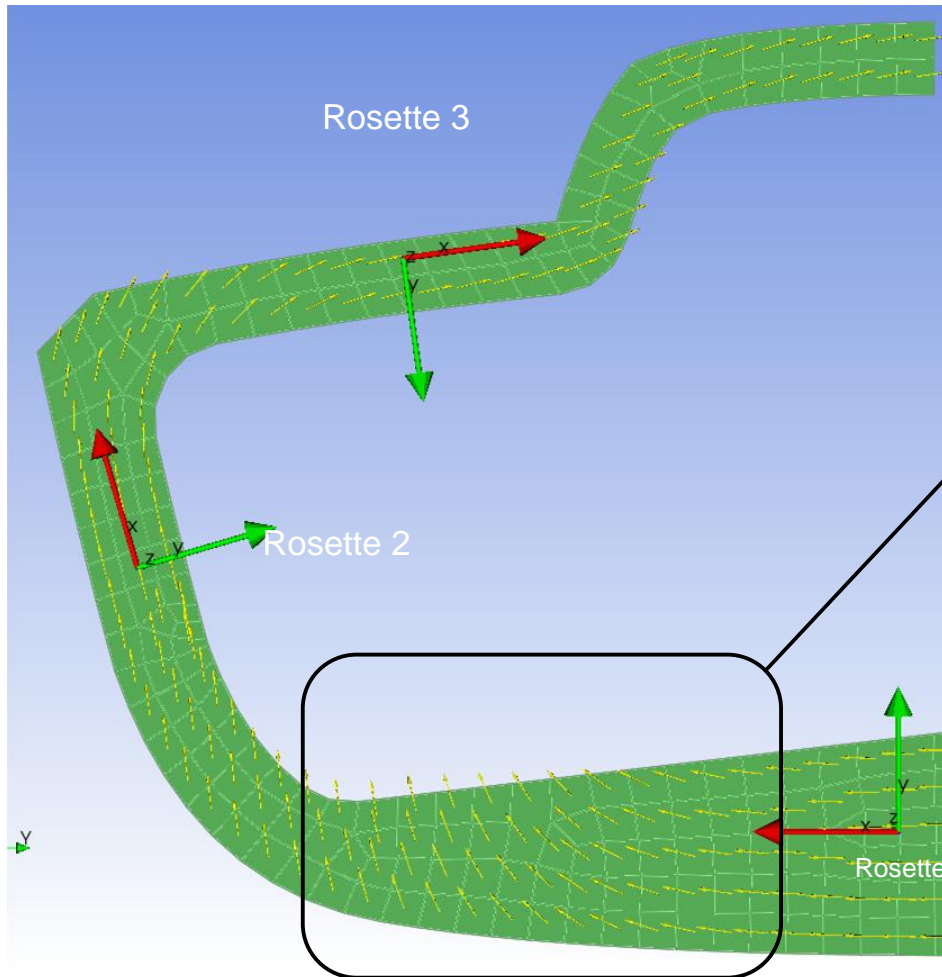
Shell-based Composite Analysis



- Cartesian, cylindrical, spherical and edge-wise rosettes
- Combination methods



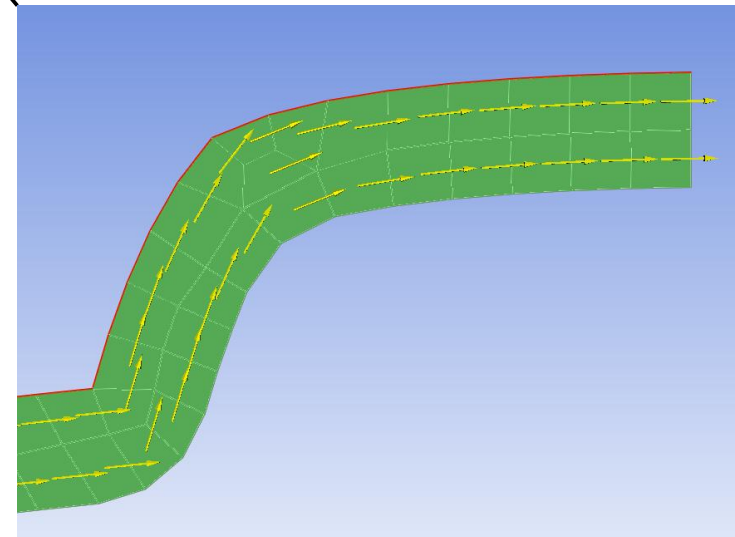
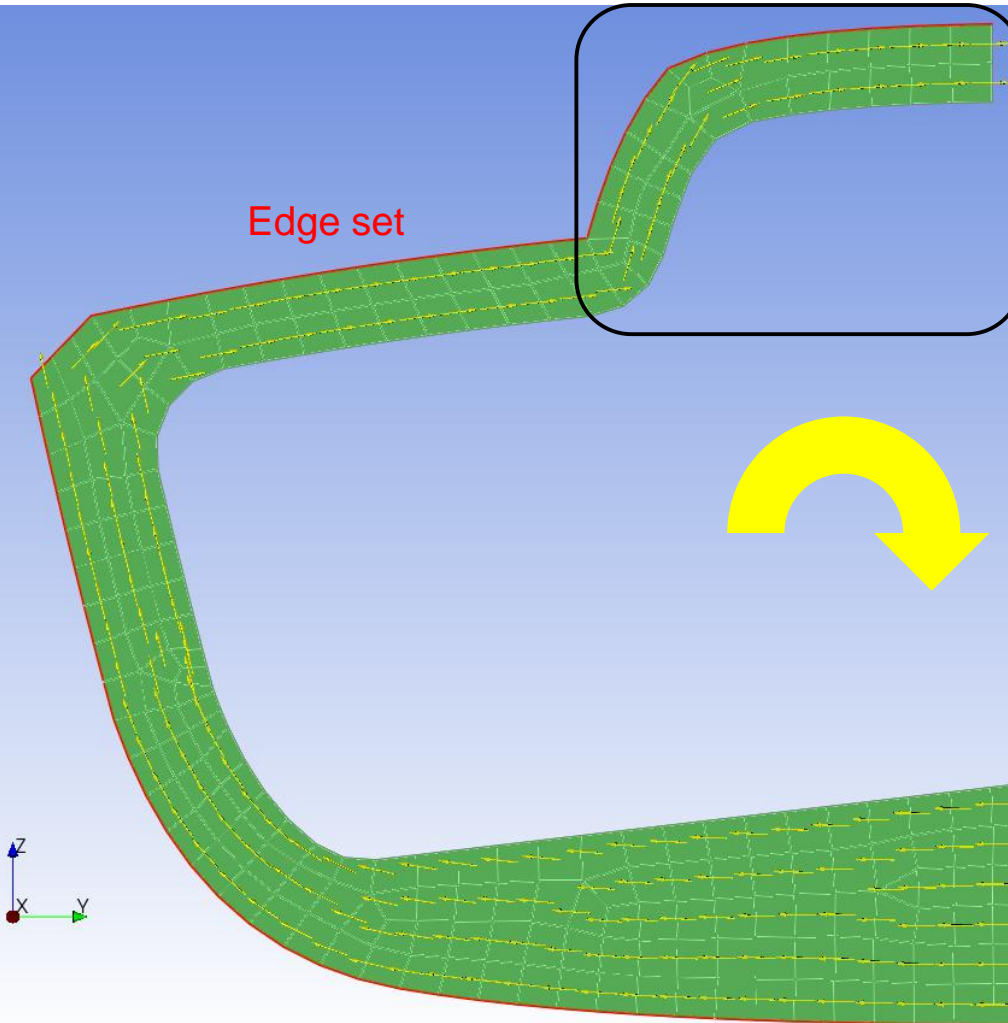
Shell-based Composite Analysis



Shell-based Composite Analysis

Edge set

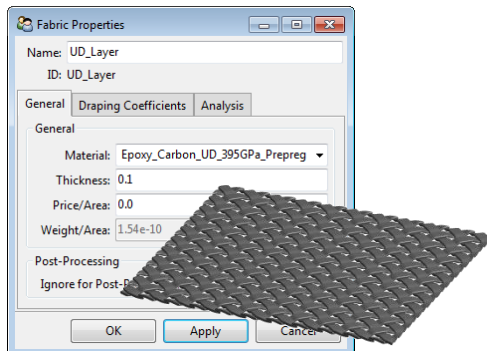
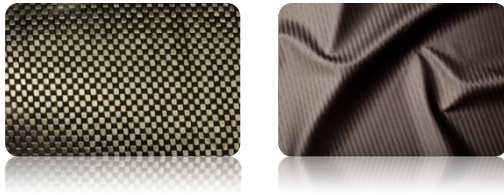
Edge-wise rosettes are very handy to define the reference direction along an edge guide



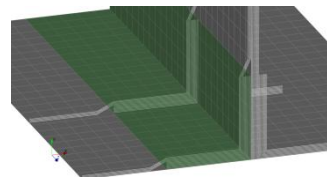
Shell-based Composite Analysis

Three steps to a composite layup

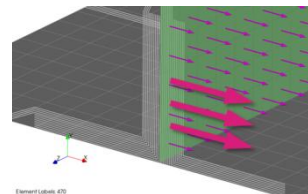
Material and Fabrics



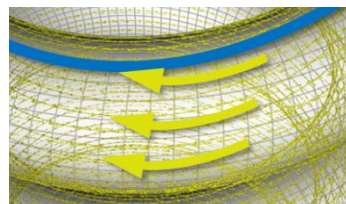
Oriented Element Sets



Layup Surface

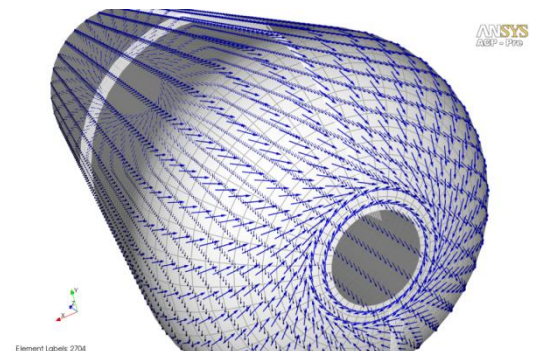
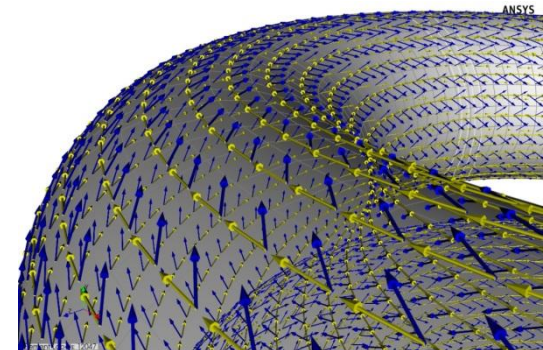


Layup Direction



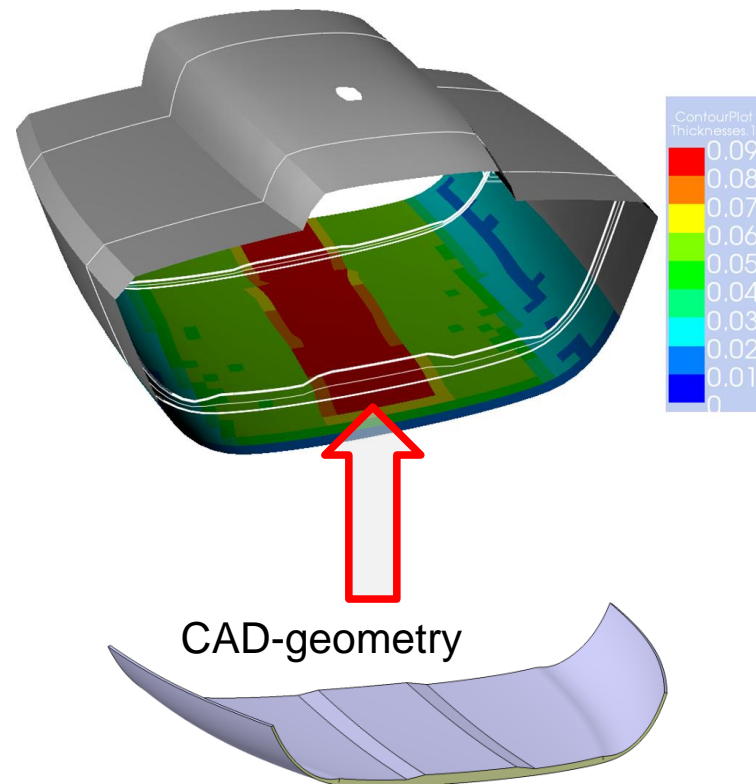
Fiber Orientation

Composite Layup

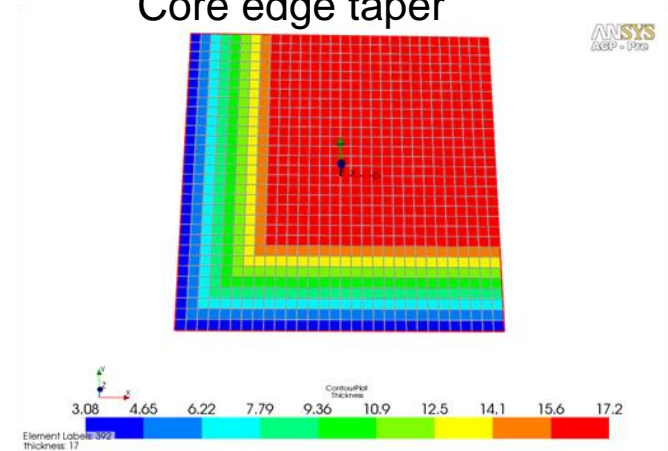


Shell-based Composite Analysis

Core thickness set by CAD geometry



Core edge taper



Ply thickness entered with a Look Up Table

Look Up Table Properties

Name: table

Values: Interpolation

	Location.x	Location.y	Location.z	-20	-40	20	40	90	Radius	Thickness										
0	-20	124110	-18	511083	-270	40	-27	72	-60	971187	27	726121	60	971187	90	000000	33	669359	0	272057
1	-20	007080	-17	147761	-271	95	-30	13	-70	649064	30	134522	70	649064	90	000000	31	202244	0	293549
2	-20	94367	-15	741248	-273	20	-33	13	90	000000	33	130761	90	000000	90	000000	28	654516	0	319470
3	-20	795262	-14	296281	-274	46	-36	99	90	000000	36	990215	90	000000	90	000000	26	034700	0	351818
4	-19	519142	-12	817651	-276	40	-42	13	90	000000	42	130117	90	000000	90	000000	23	351426	0	392267
5	-17	235383	-11	310261	-276	35	-49	45	90	000000	49	456410	90	000000	90	000000	20	613549	0	448248
6	-14	989218	-9	779142	-277	06	-61	46	90	000000	61	466476	90	000000	90	000000	17	838221	0	513735
7	-12	553883	-8	294760	-276	61	90	000000	90	000000	90	000000	90	000000	90	000000	15	010736	0	610230
8	-10	176401	-6	667318	-277	99	90	000000	90	000000	90	000000	90	000000	90	000000	12	164192	0	762906
9	-8	170560	-2	870949	-278	20	-27	72	-60	974917	27	727209	60	974917	90	000000	33	668243	0	872067
10	-22	913148	-21	177828	-271	95	-30	13	-70	655507	30	135672	70	655507	90	000000	33	201165	0	293579
11	-21	056861	-19	826297	-272	20	-33	14	90	000000	33	140134	90	000000	90	000000	29	653466	0	319482
12	-19	144893	-17	845511	-274	46	-36	99	90	000000	36	991991	90	000000	90	000000	26	032629	0	351853
13	-17	183554	-15	810390	-275	40	-42	13	90	000000	42	132452	90	000000	90	000000	23	350774	0	392265
14	-15	177933	-13	845173	-276	35	-49	45	90	000000	49	459709	90	000000	90	000000	20	612595	0	444288
15	-13	138765	-12	852382	-277	06	-61	47	90	000000	61	471402	90	000000	90	000000	17	827387	0	513759
16	-11	068705	-10	338251	-277	61	90	000000	90	000000	90	000000	90	000000	90	000000	15	010006	0	610260
17	-9	977204	-8	210284	-277	99	90	000000	90	000000	90	000000	90	000000	90	000000	12	164482	0	762960
18	-20	607203	-6	624608	-278	40	-27	72	-60	975714	27	727441	60	975714	90	000000	33	667883	0	872069
19	-19	128301	-24	685157	-271	95	-30	13	-70	659700	30	136277	70	659700	90	000000	31	200364	0	293586
20	-17	588165	-22	624854	-273	20	-33	14	90	000000	33	141845	90	000000	90	000000	28	652153	0	319497
21	-15	991911	-20	548681	-274	46	-36	99	90	000000	36	994827	90	000000	90	000000	26	032191	0	351876
22	-14	348778	-18	407842	-275	40	-42	13	90	000000	42	136977	90	000000	90	000000	23	348380	0	392318
23	-12	672104	-16	238861	-276	35	-49	46	90	000000	49	466724	90	000000	90	000000	20	610437	0	444435

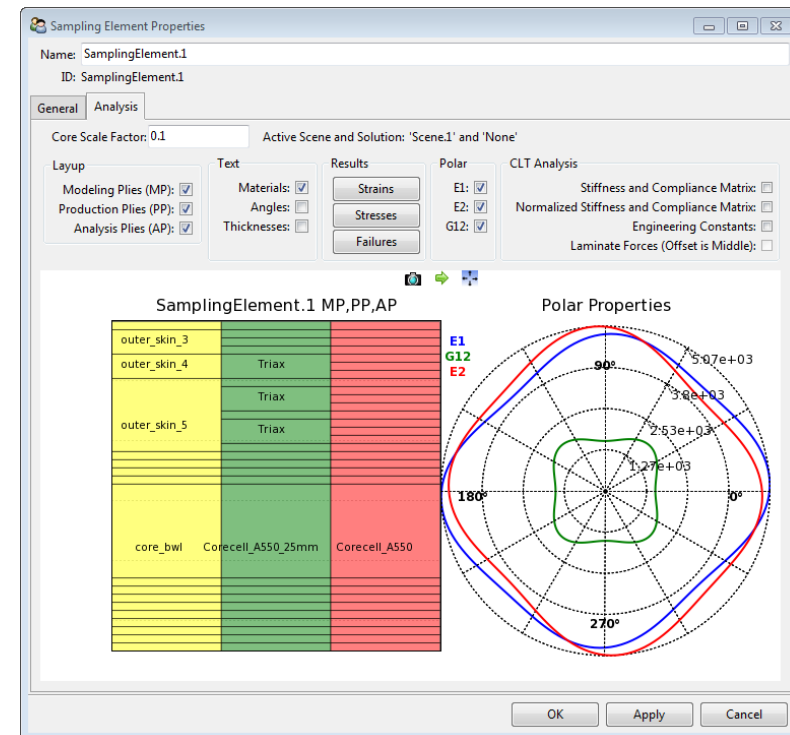
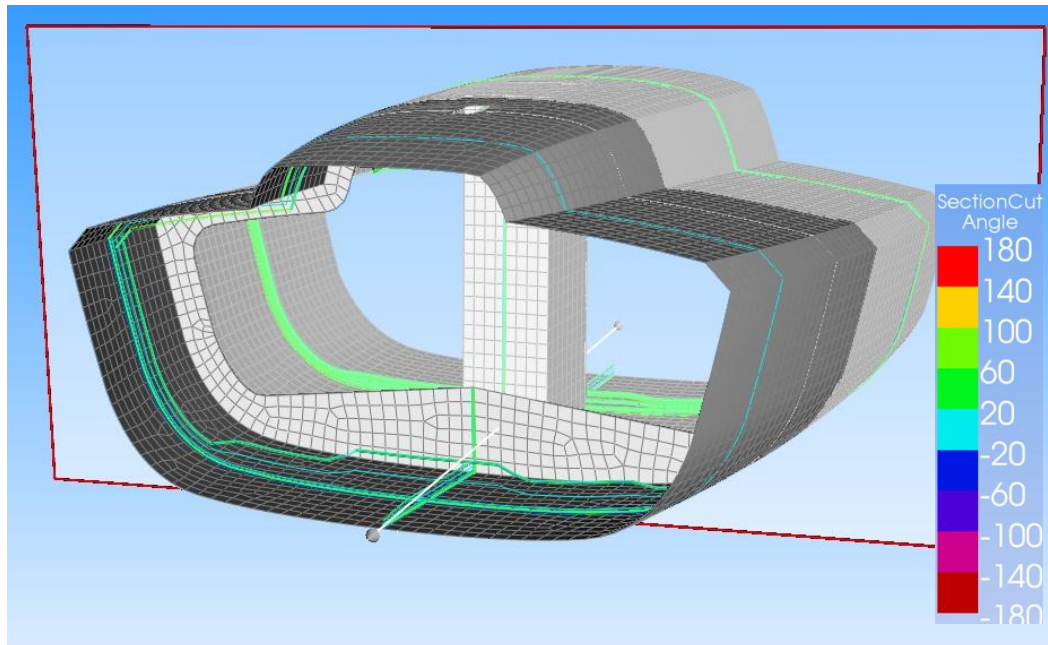
OK

Apply

Cancel

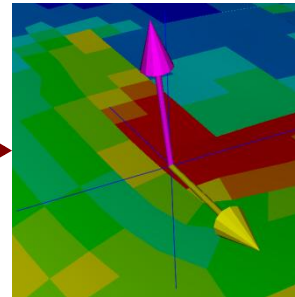
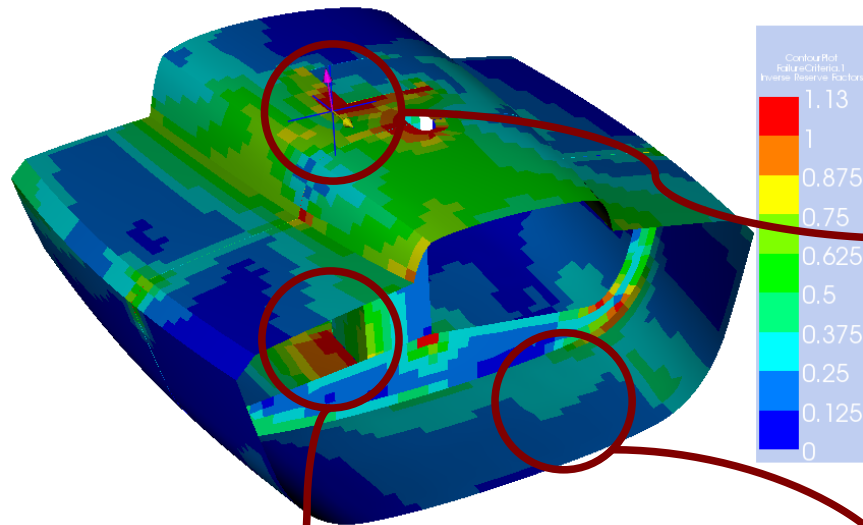
Shell-based Composite Analysis

- Stacking sequence and composite thickness can be displayed on arbitrarily Sections or the entire model
- Sampling elements are a powerful tool to analyze the layup

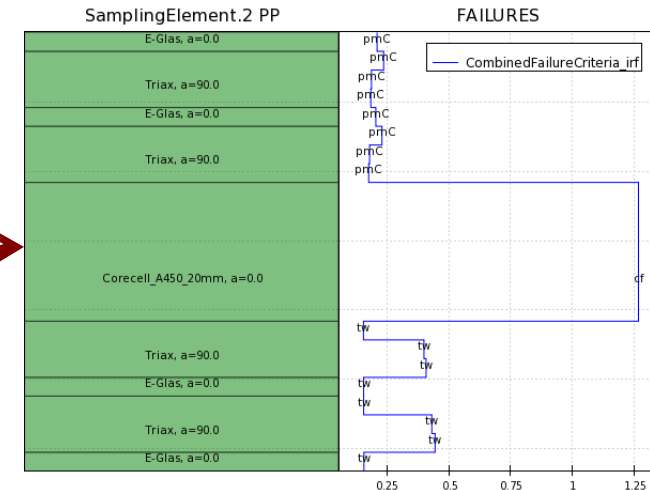


Shell-based Composite Analysis

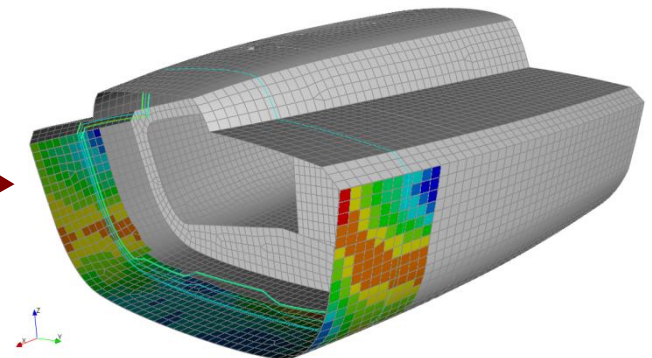
IRF plot of entire structure



Assess utilization of each ply with a Sampling Element



Analyze structure ply-by-ply



Tw(15)(2):
Tsai-Wu
Layer 15,
Loadcase 2

Gain an overview of
critical failure mode,
layer and load case

CADFEM

Shell-based Composite Analysis

Solid Composite Analysis

Additional Features

Define Variable Material Properties

ANSYS Composite Cure Simulation

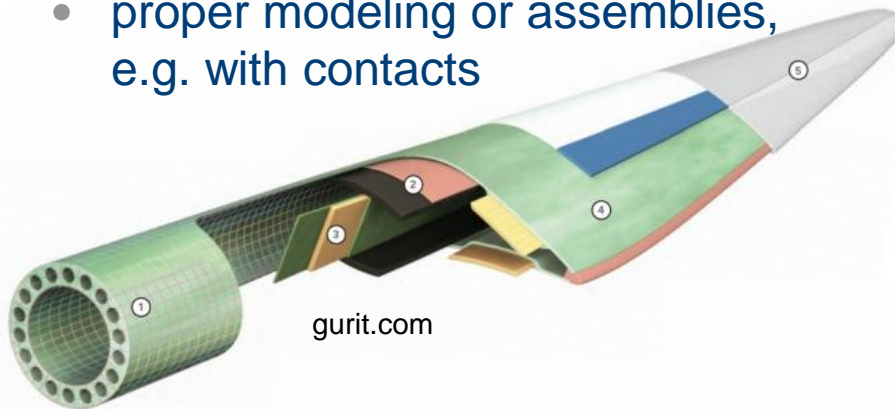
Why Solid Modeling?

Account for

- out-of-plane normal stresses
- out-of-plane shear stresses

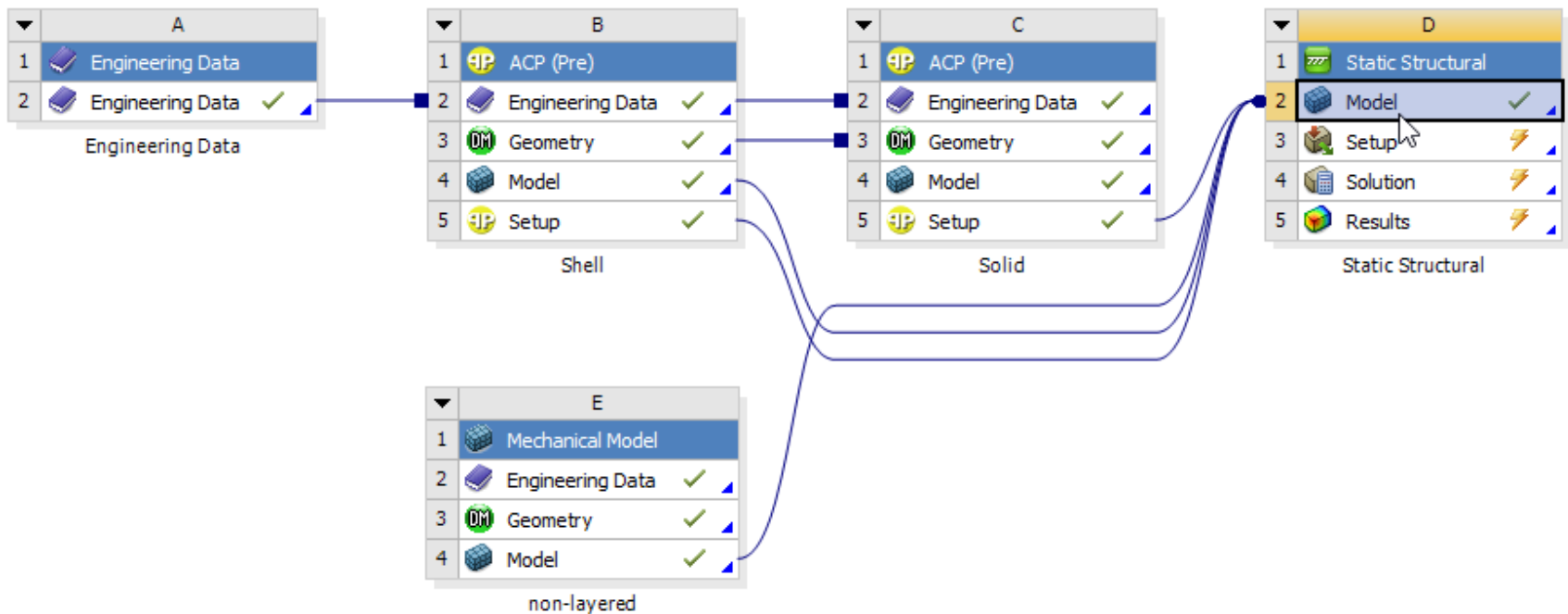
Relevant for

- „thick-walled“ composite structures
- e.g. buckling analyses of such
- proper modeling of assemblies, e.g. with contacts

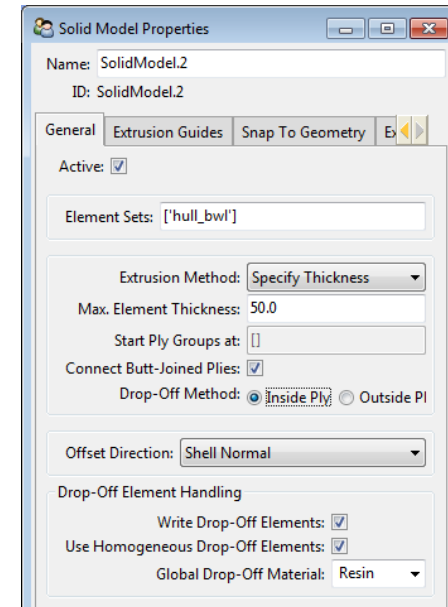
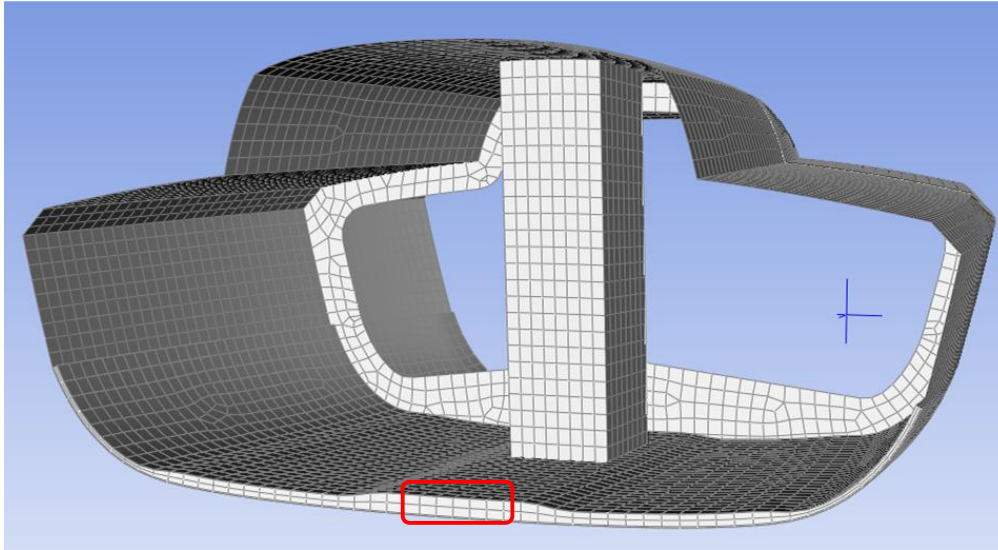


Unified approach to handle Shells and Solids in R17

- User can now assemble multiple Shell and Solid models from ACP in a single system.
- In addition, non-layered models (from Mechanical, External Model etc.) can be assembled along with the composite models.



Solid Composite Analysis

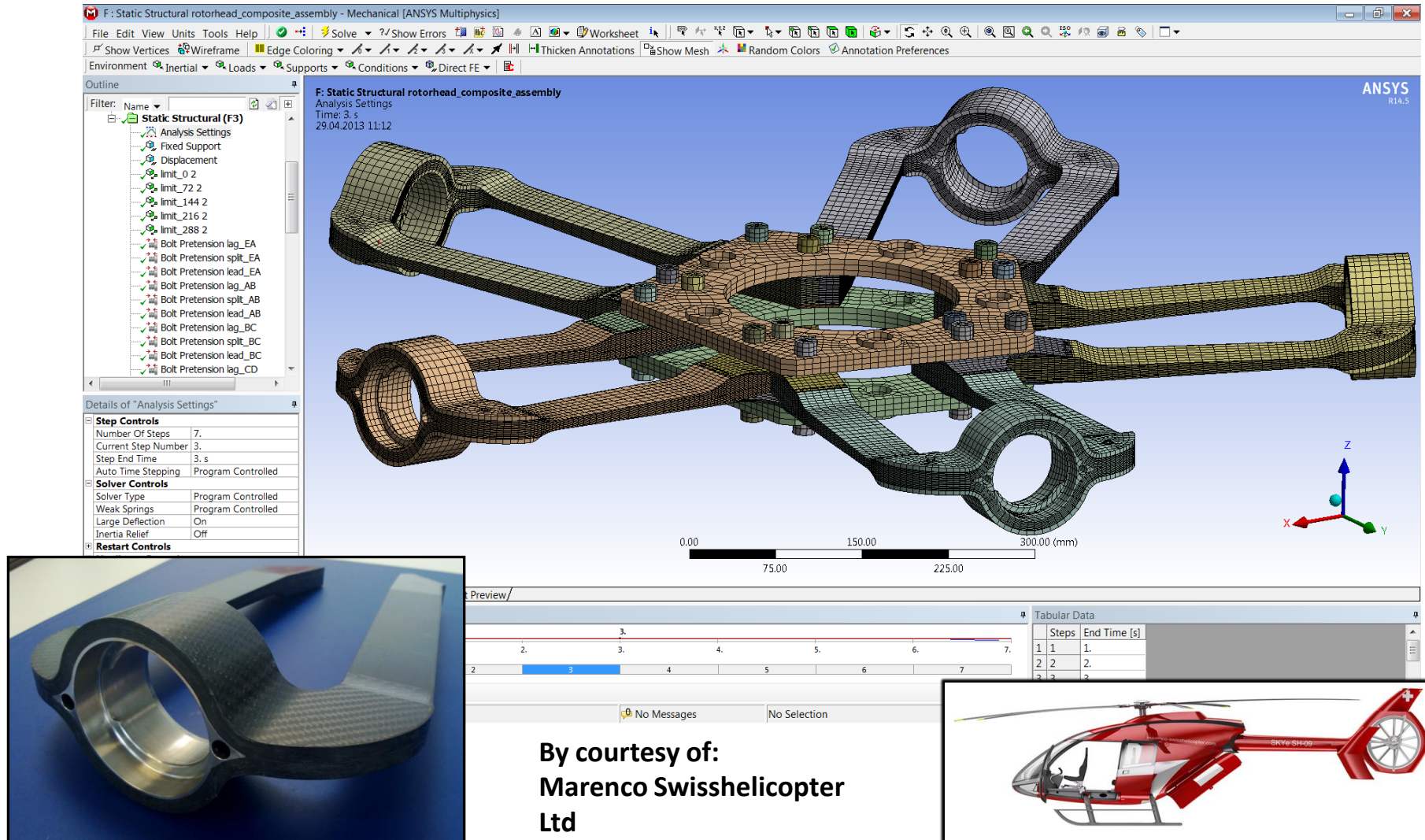


Monolithic

Material-wise

Analysis Ply-wise

Solid Composite Analysis



Solid Composite Analysis

- An Extruded Solid Model can be additionally shaped with external geometries to generate desired shapes

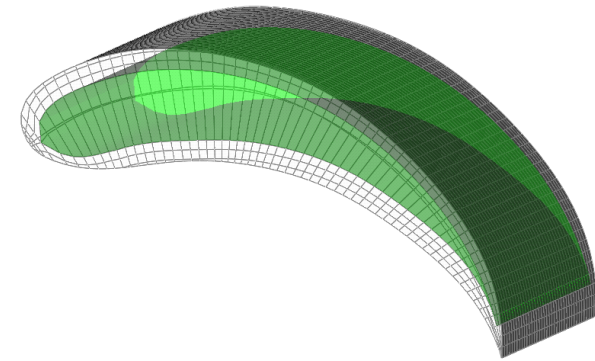
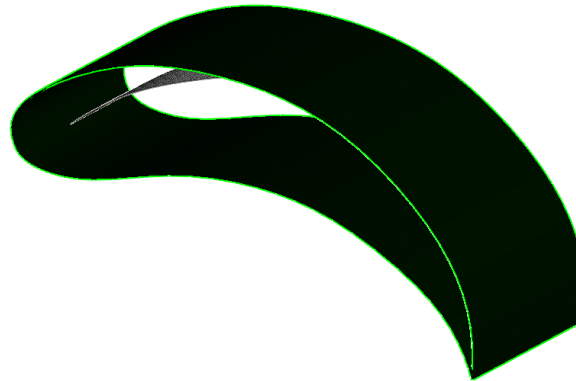
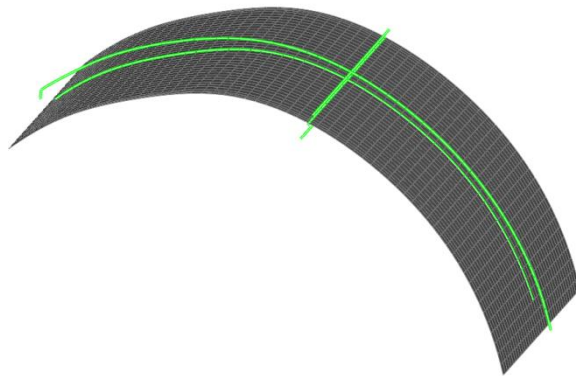
**Start with shell geometry
and composite layup**



**Define extrusion guides
and snap to geometry**

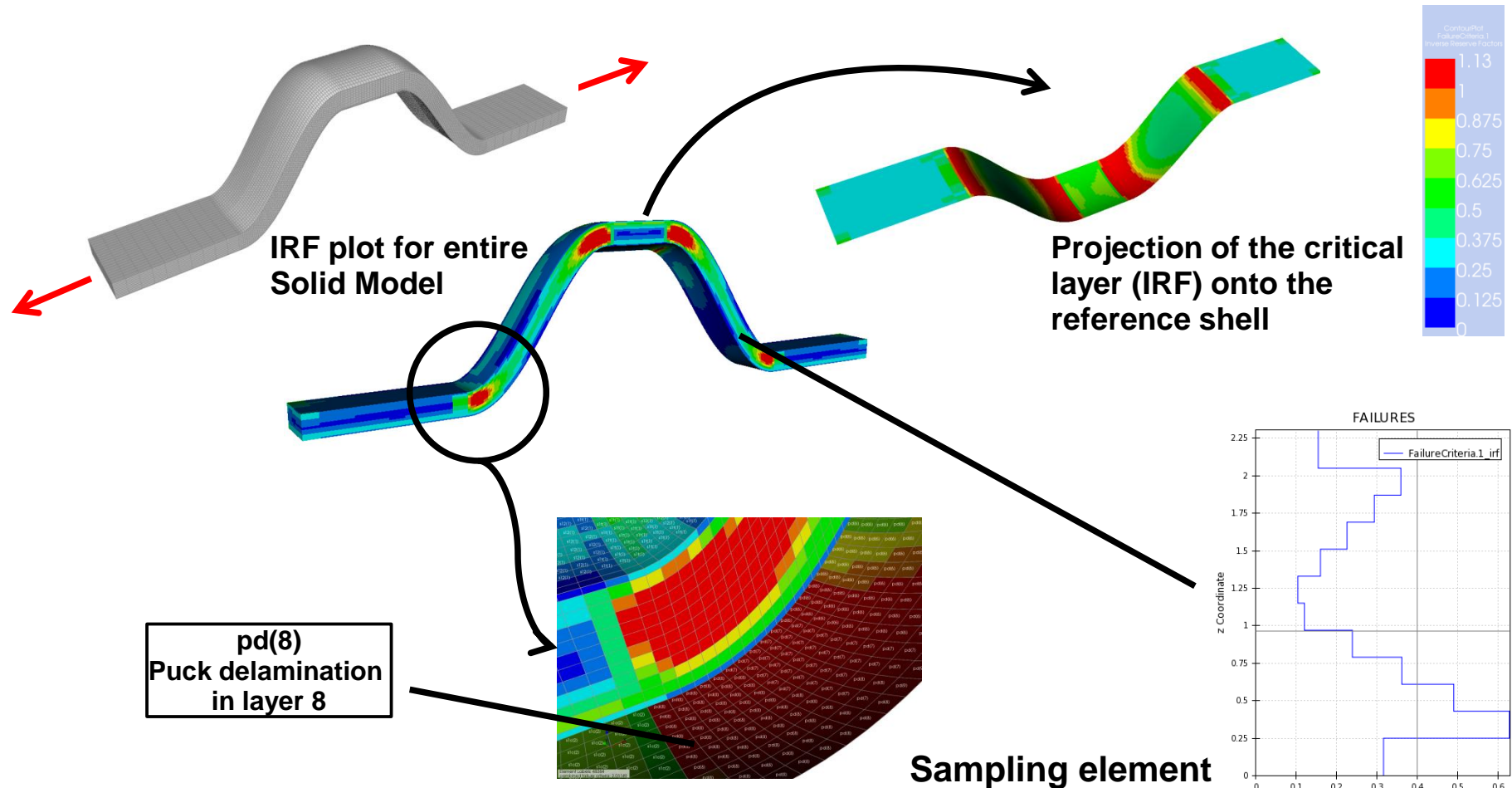


**Specify element thickness
and extrude Solid Model**



Solid Composite Analysis

All features of Shell-Model Postprocessing are retained



CADFEM

Shell-based Composite Analysis

Solid Composite Analysis

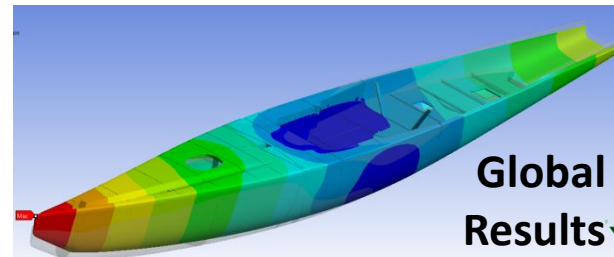
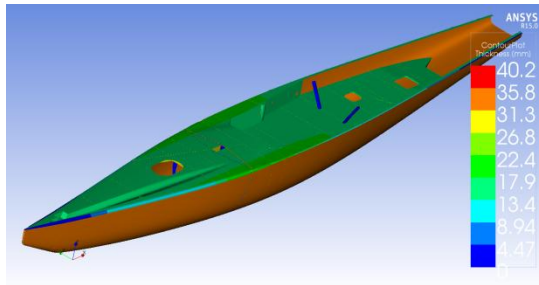
Additional Features

Define Variable Material Properties

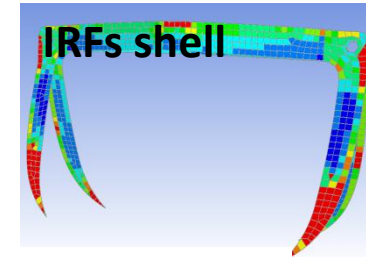
ANSYS Composite Cure Simulation

Additional Features

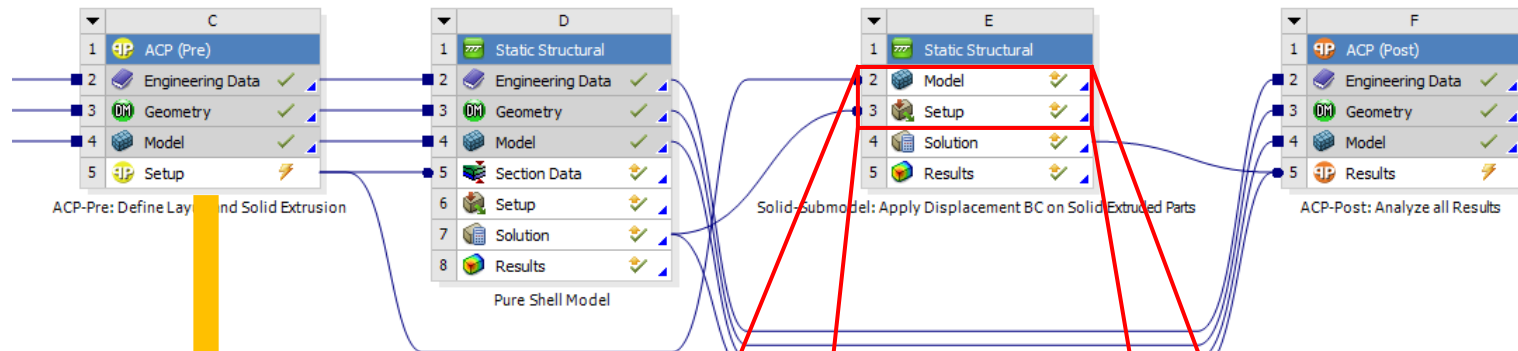
SHELL



Global Results



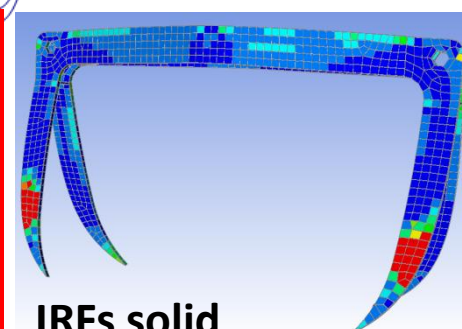
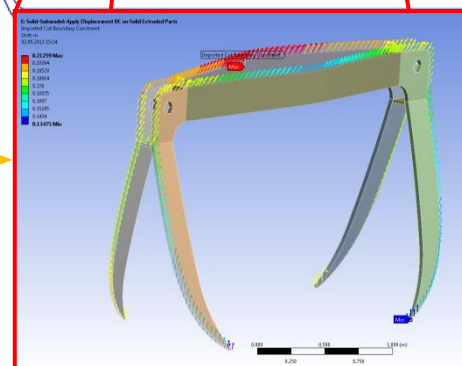
IRFs shell



SOLID

Create solid model of critical part (C5 – E2)

and map shell results to the cut faces (D7 - E3)

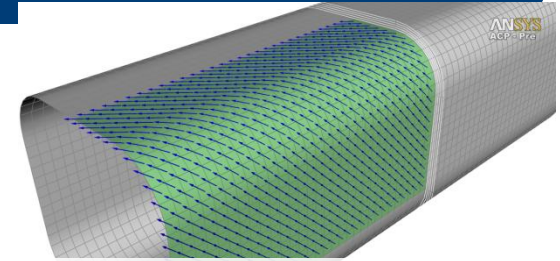


IRFs solid

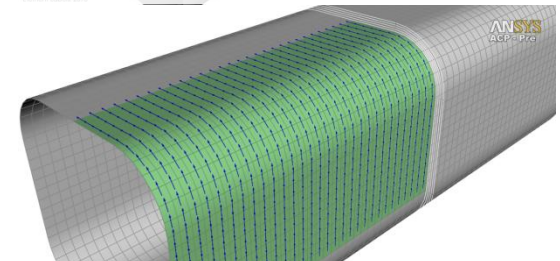
Additional Features

Composite Parameter Studies

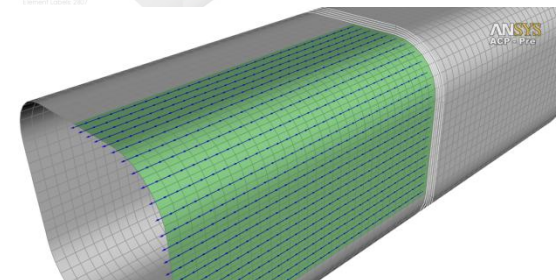
- ANSYS Composite PrepPost allows the user to define parameters for fiber angles.
- All parameters are accessible in the ANSYS Workbench parameter manager.



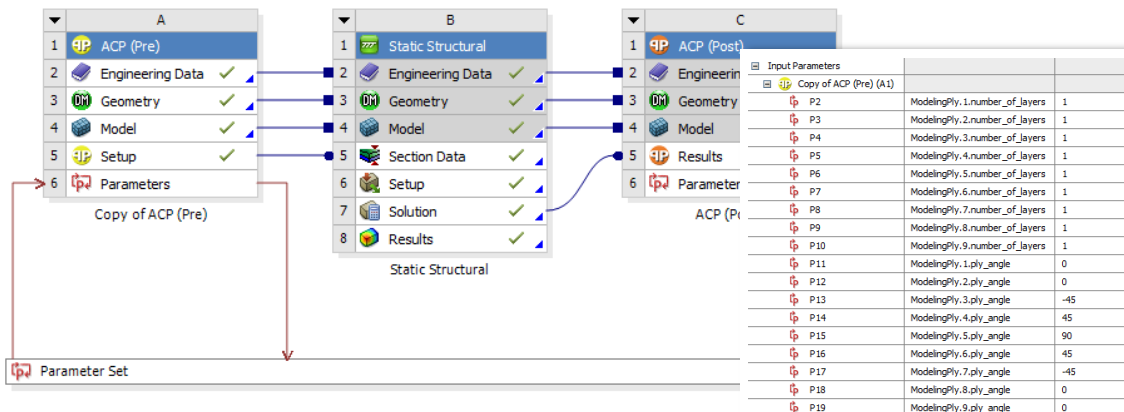
45° Layer



90° Layer



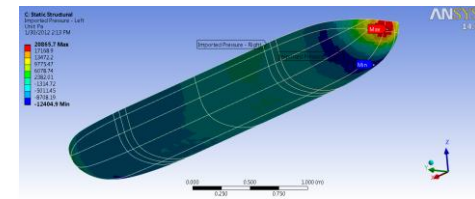
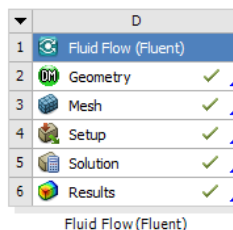
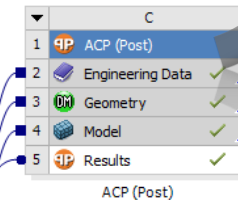
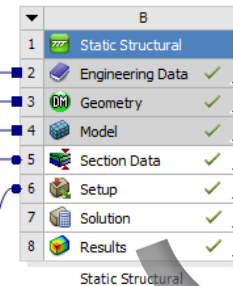
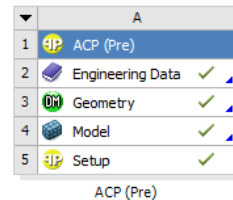
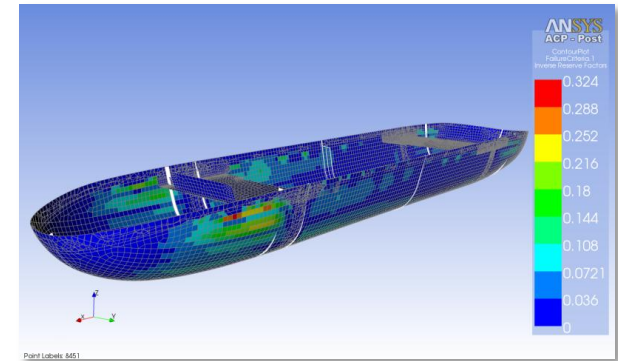
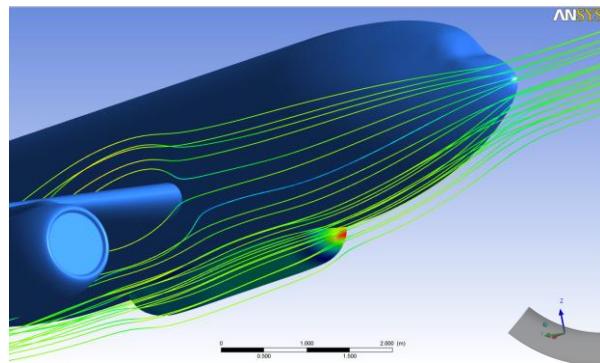
0° Layer



Additional Features

Fluid-Structure Interaction and Composites

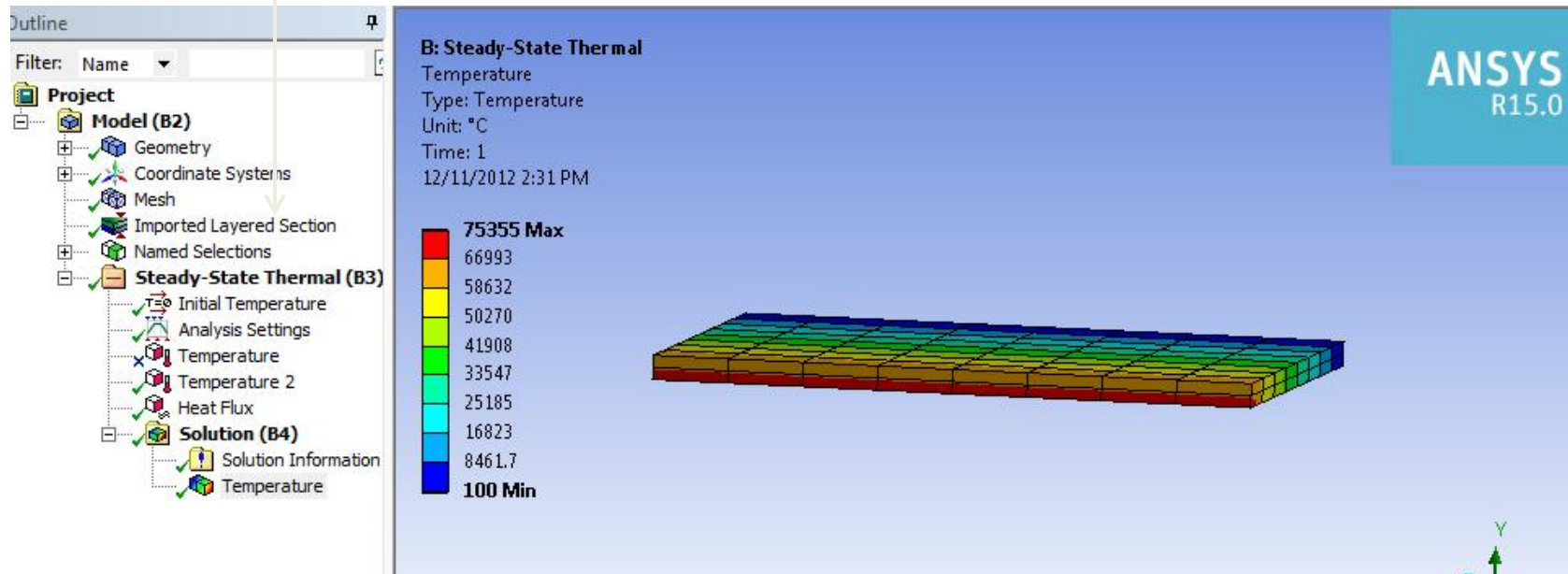
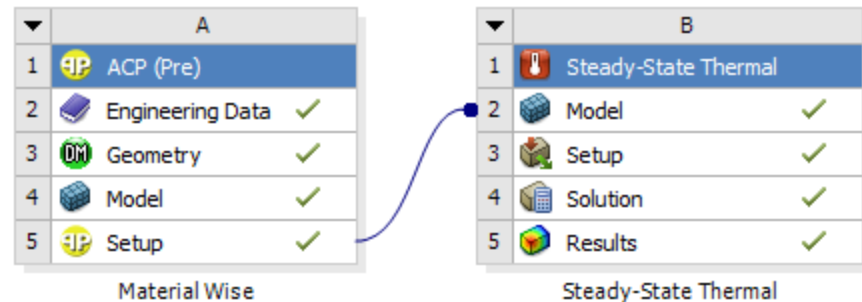
- Apply loads from CFD directly.
- One-way and two-way fluid structure interaction is possible.



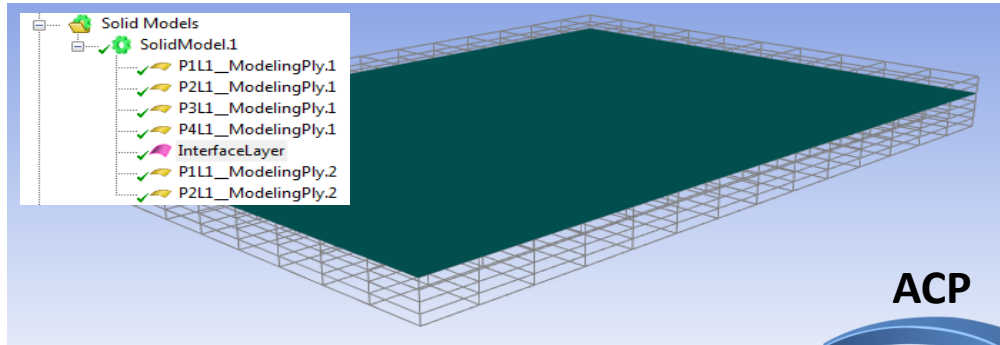
Additional Features

Thermal analysis now supports solid layer elements SOLID278/SOLID279

Thermal solid composites are now supported in models with Imported Layered Sections.

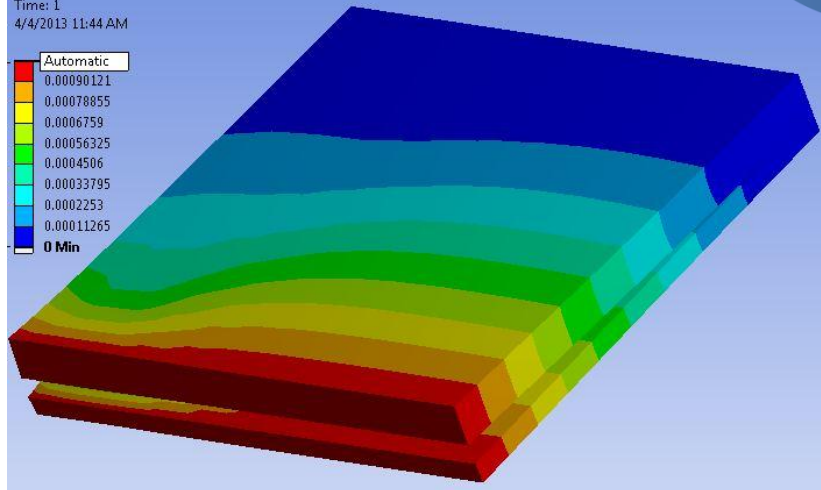
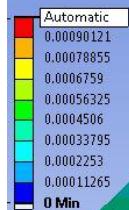


Additional Features



ACP

E: Static Structural
Total Deformation
Type: Total Deformation
Unit: m
Time: 1
4/4/2013 11:44 AM



Project

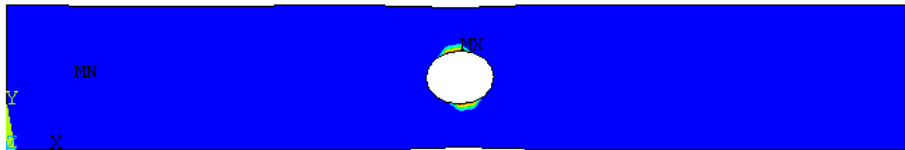
- Model (E2)
 - Geometry
 - Solid 1
 - Solid 2
 - Coordinate Systems
 - Connections
 - Mesh
 - Fracture
 - Pre-Meshed Crack
 - Interface Delamination
 - Imported Layered Section
 - Named Selections
 - Static Structural
 - Details of "Interface Delamination"

Definition	
Type	Interface Delamination
Method	VCCT
Failure Criteria Option	Energy-Release Rate
<input type="checkbox"/> Critical Rate	1. J/m ²
Suppressed	No
Scope	
Generation Method	Pre-Generated Interface
Interface	InterfaceLayer.1
Initial Crack	Pre-Meshed Crack
Step Controls for Crack Growth	

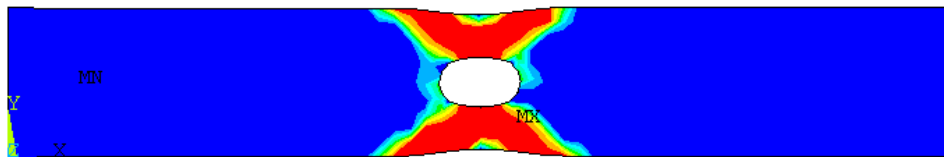
Center FEM

Additional Features

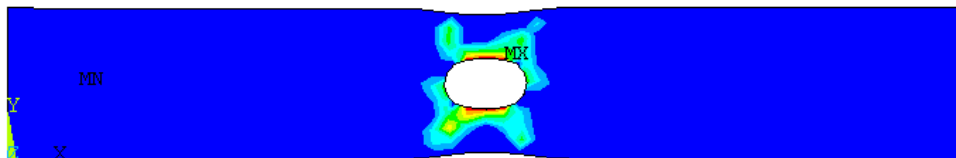
- New nonlinear solution process to simulate the damage propagation in composites beyond first ply failure
- Damage evolution with Continuum Damage Mechanics (CDM) method
- Example: progressive damage of a composite plate under stretch load



Start of damage (layer 1)



Progressed damage
(layer 1)



Progressed damage
(layer 3)

CADFEM

Shell-based Composite Analysis

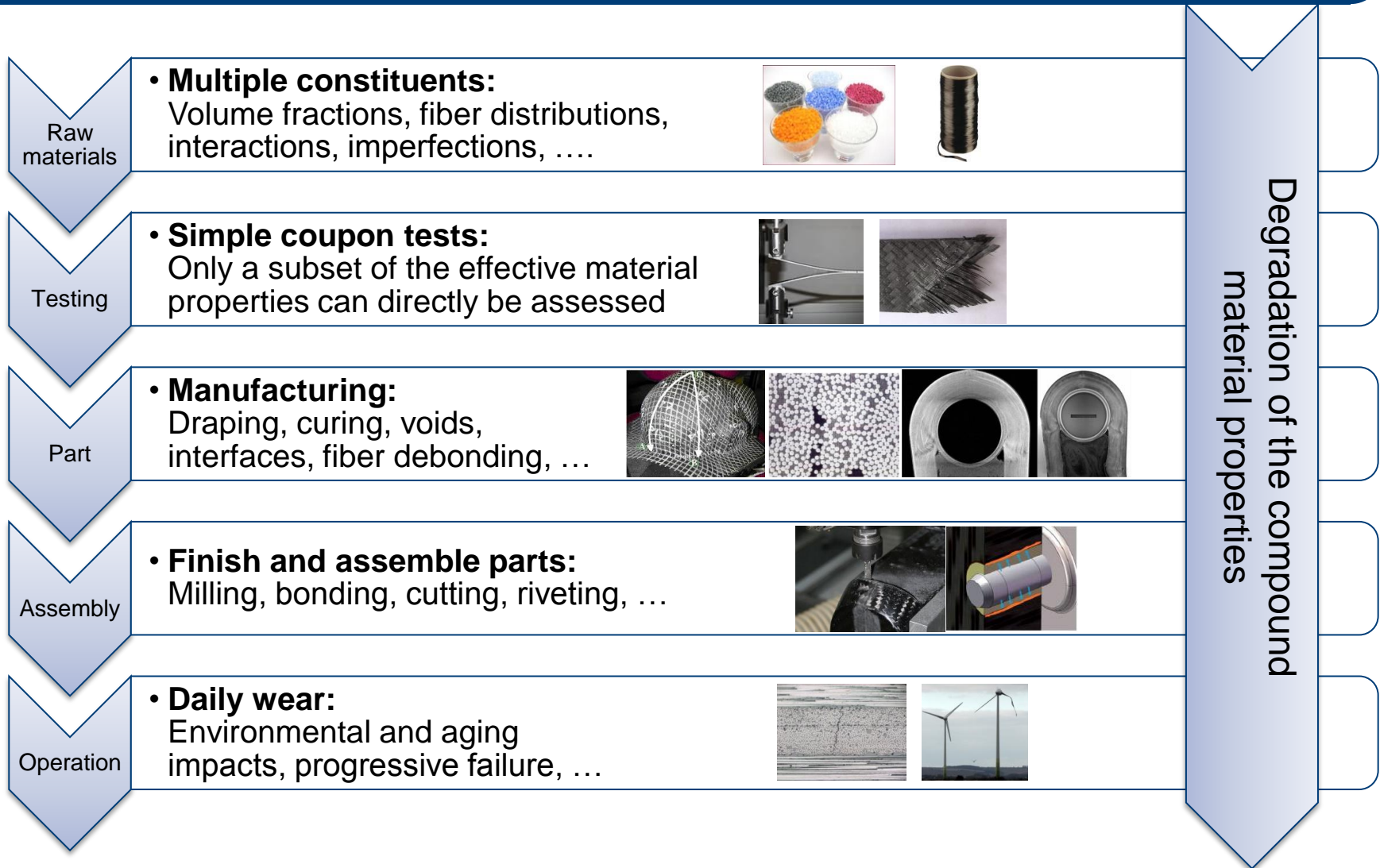
Solid Composite Analysis

Additional Features

Define Variable Material Properties

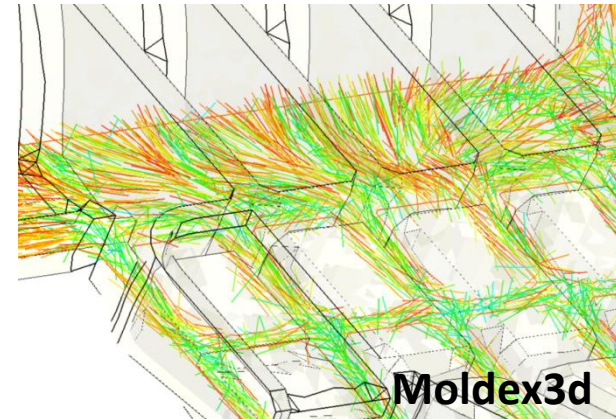
ANSYS Composite Cure Simulation

Variable Material Data: Background



Variable Material Data: Challenges

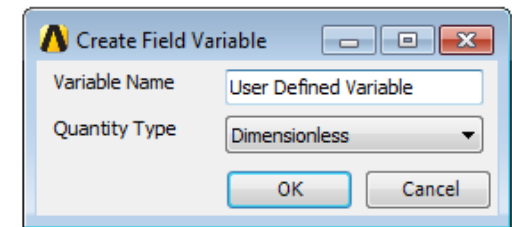
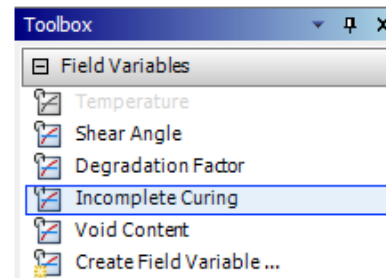
- Each composite structure has its individual challenges!
- Consider degraded material properties in region with low fiber volume content, voids and folding
- Analyse draping shear angle and fiber orientation
- Consider environmental conditions such as temperature and humidity















1. Define Variable Material Properties

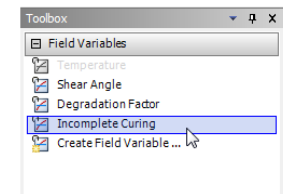
- In Engineering Data:
- Define up to 9 arbitrary field variables
 - Shear
 - Degradation Factor
 - Temperature
 - Void content
 - ...
- Supported properties in R17.0:
 - Elasticity
 - Stress limits
 - Strain limits
 - Puck for woven
- Benefit from new Excel interface to fill tabular data

	A
1	Contents of Engineering Data
2	Material
3	Epoxy_Carbon_UD_230GPa_Prepeg
4	Epoxy_Carbon_Woven_230GPa_Prepeg
5	PVC_Foam_60kgm3



Properties of Outline Row 4: Epoxy_Carbon_Woven_230GPa_Prepeg		
	A	B
1	Property	Value
2	 Material Field Variables	 Table
3	 Density	1420
4	 Orthotropic Secant Coefficient of Thermal Expansion	
10	 Orthotropic Elasticity	 Tabular
30	 Orthotropic Stress Limits	 Tabular
50	 Orthotropic Strain Limits	 Tabular
70	 Tsai-Wu Constants	
		

1

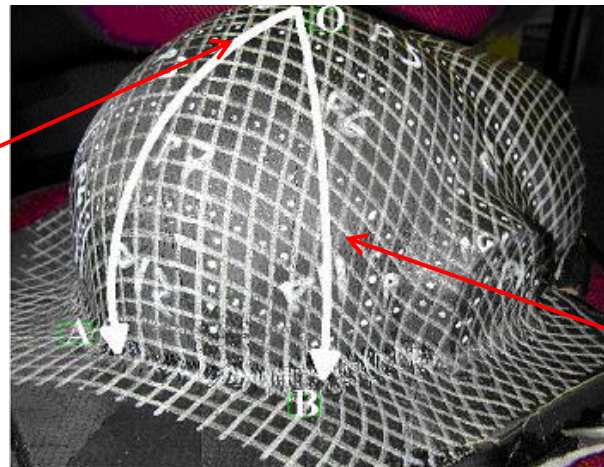
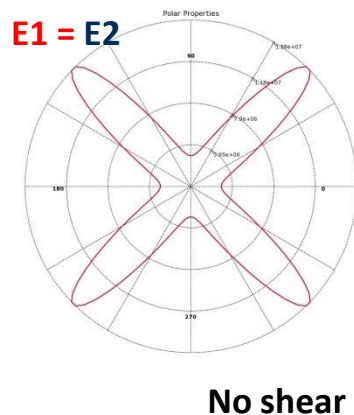


2

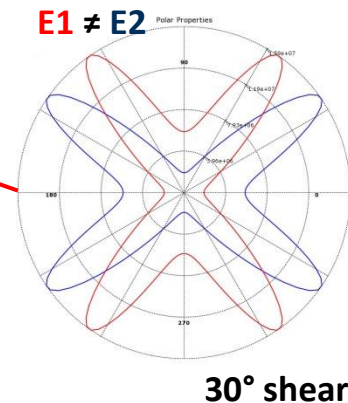
Table of Properties Row 10: Orthotropic Elasticity					
	A	B	C	D	E
1	Incomplete Curing	1	Temperature (C)	Young's Modulus X direction (Pa)	Young's Modulus Y direction (Pa)
2	0,5			1,2E+08	8,6E+06
3	1			9,6E+07	6,88E+06
*					

Why Shear Dependency?

- Draping simulation or direct measurements allow you to include the shear distortion effect of a fabric in your simulation.



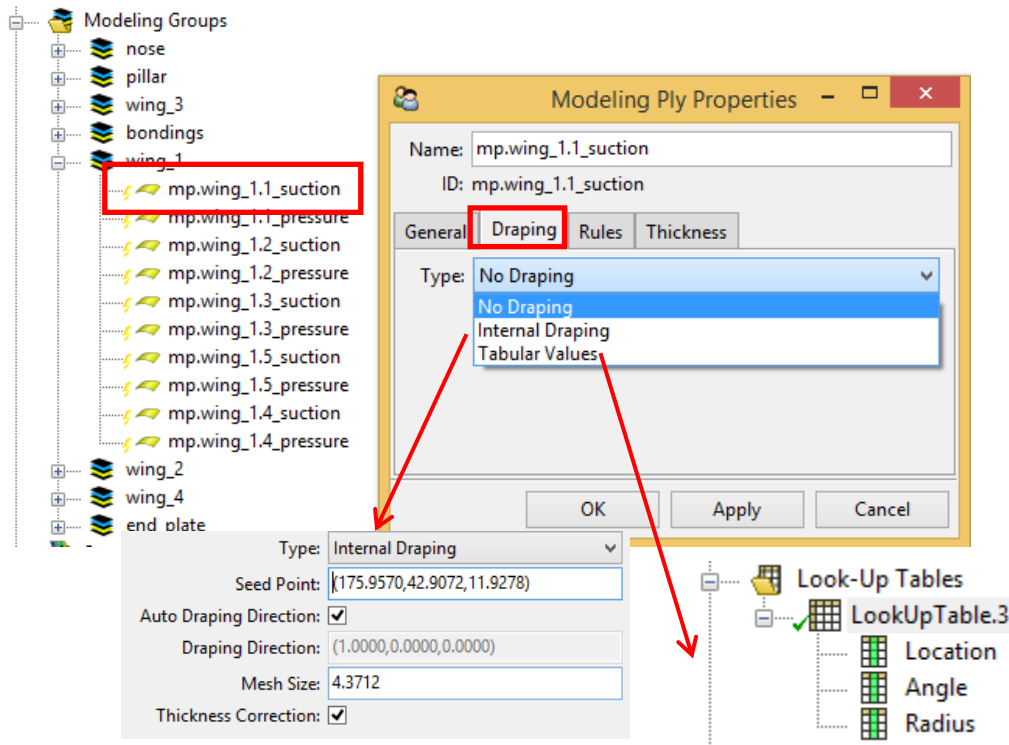
Composite draping: shear and wrinkling
(Dr M Sutcliffe, University of Cambridge, UK)



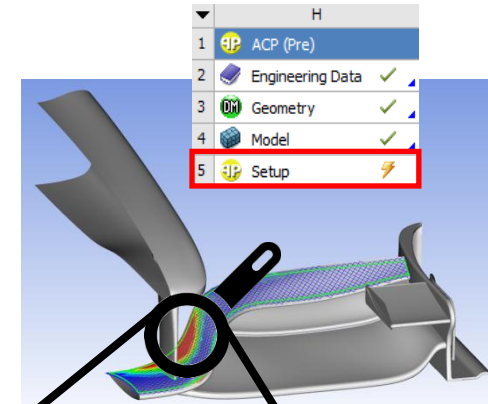
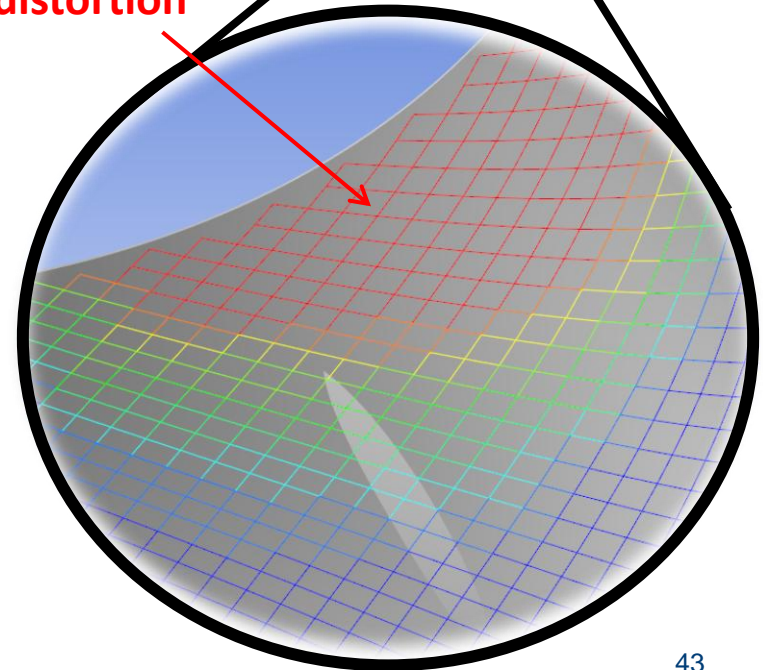
Shear has significant influence on the effective material characteristics.

2. Draping in ACP

- Draping allows you to evaluate the shear angle by
 - Simulation
 - External data: LookUp Table (CSV)



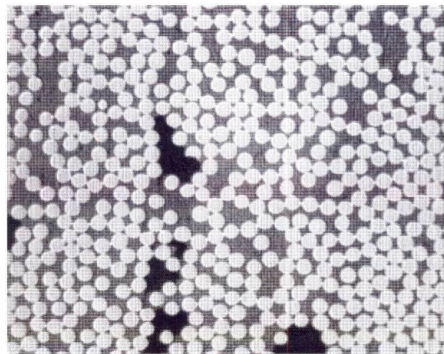
High shear distortion



Degradation of Composites

Voids in Carbon-epoxy cross-section

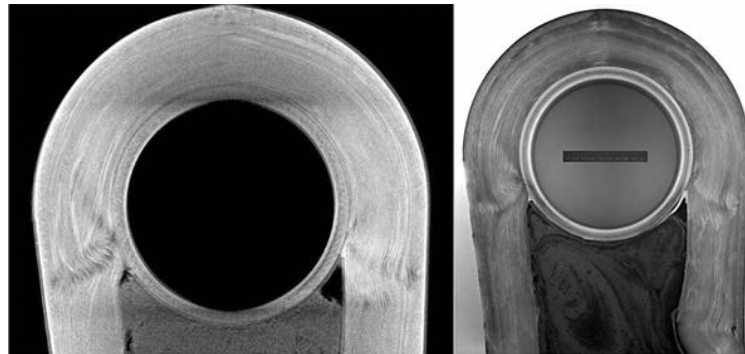
→ Reduction of the transverse and out-of-plane properties



(Prof IM Daniel, Northwestern University, USA)

Fiber waviness and uncured sections

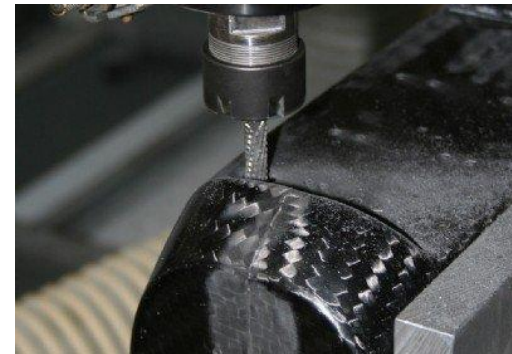
→ High variation in stiffness and strength



<http://www.dtbtest.com/>

Finishing (milling cutting etc.)

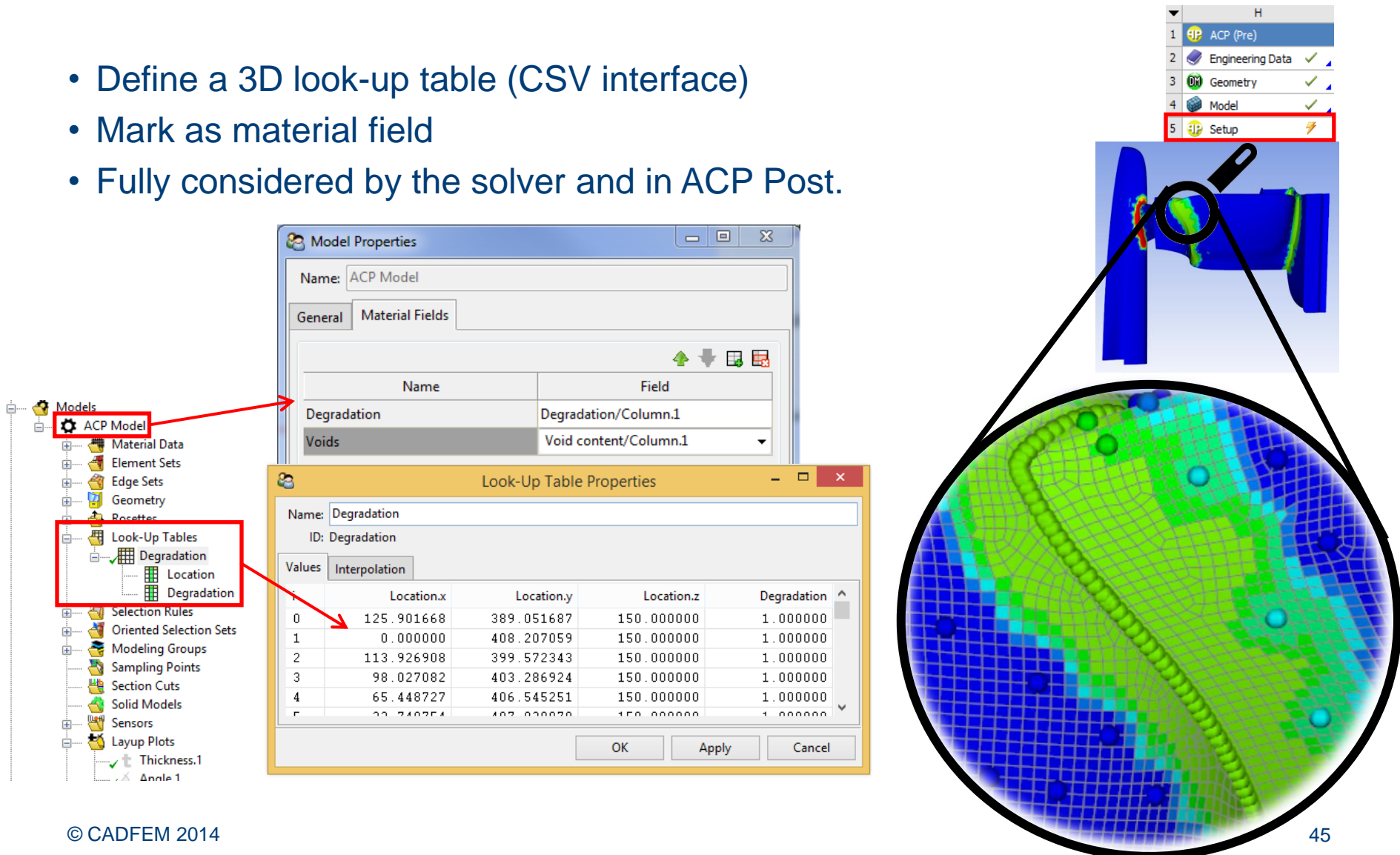
→ Introduction of failure origins



<http://www.mtdcnc.com/>

3. Specify Variable Material Fields

- Define a 3D look-up table (CSV interface)
- Mark as material field
- Fully considered by the solver and in ACP Post.



The screenshot illustrates the process of defining a 3D look-up table for material degradation in ANSYS. The 'Model Properties' dialog is open, showing the 'Material Fields' tab. The 'Look-Up Tables' section in the left tree is highlighted. The 'Look-Up Table Properties' dialog is open, showing the 'Degradation' table with a CSV interface. A circular inset shows a 3D model of a part with a color-coded degradation field.

Model Properties - Material Fields

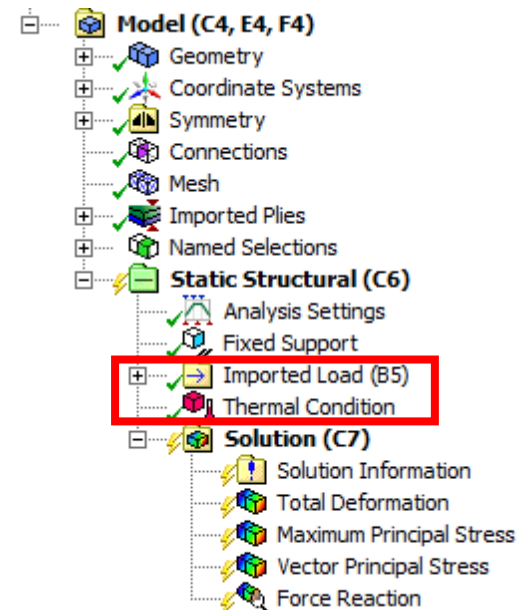
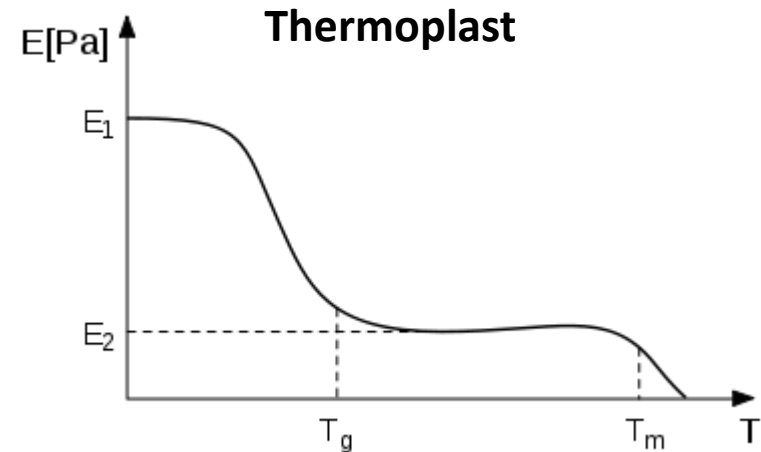
Name	Field
Degradation	Degradation/Column.1
Voids	Void content/Column.1

Look-Up Table Properties - Degradation

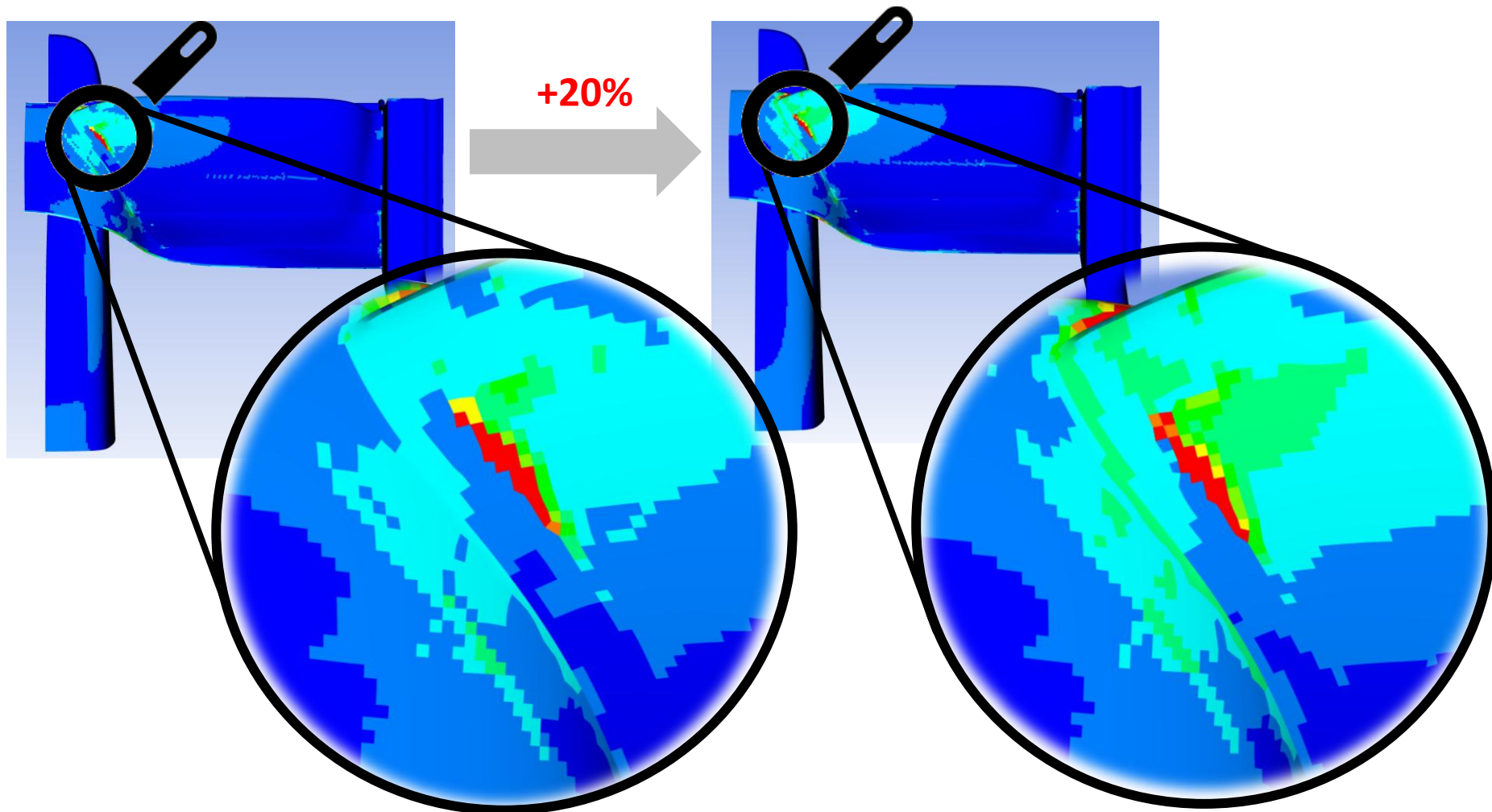
Values	Interpolation			
Location.x	Location.y	Location.z	Degradation	
0	125.901668	389.051687	150.000000	1.000000
1	0.000000	408.207059	150.000000	1.000000
2	113.926908	399.572343	150.000000	1.000000
3	98.027082	403.286924	150.000000	1.000000
4	65.448727	406.545251	150.000000	1.000000
5	32.710254	402.000000	150.000000	1.000000

4. Temperature

- Most resin systems and therefore also composites are sensitive to temperature.
- Thermal loading can be defined through different methods:
 - Thermal analysis
 - Mechanical
 - External data



Inverse Reserve Factors – Constant vs. Variable Material Data



CADFEM

Shell-based Composite Analysis

Solid Composite Analysis

Additional Features

Define Variable Material Properties

ANSYS Composite Cure Simulation

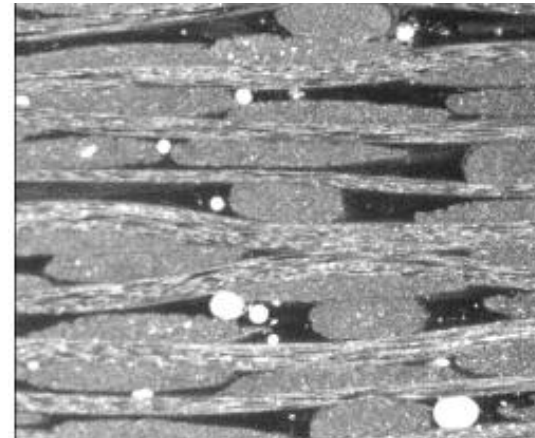
Why cure simulation is needed?

- Prediction of final shape
 - Springback
 - Compensation in tool geometry
 - Residual stresses
- Prediction of cure quality
 - Cure status
 - Resin burn
 - Thermal tool cooling, ...

→ Dimensional Accuracy

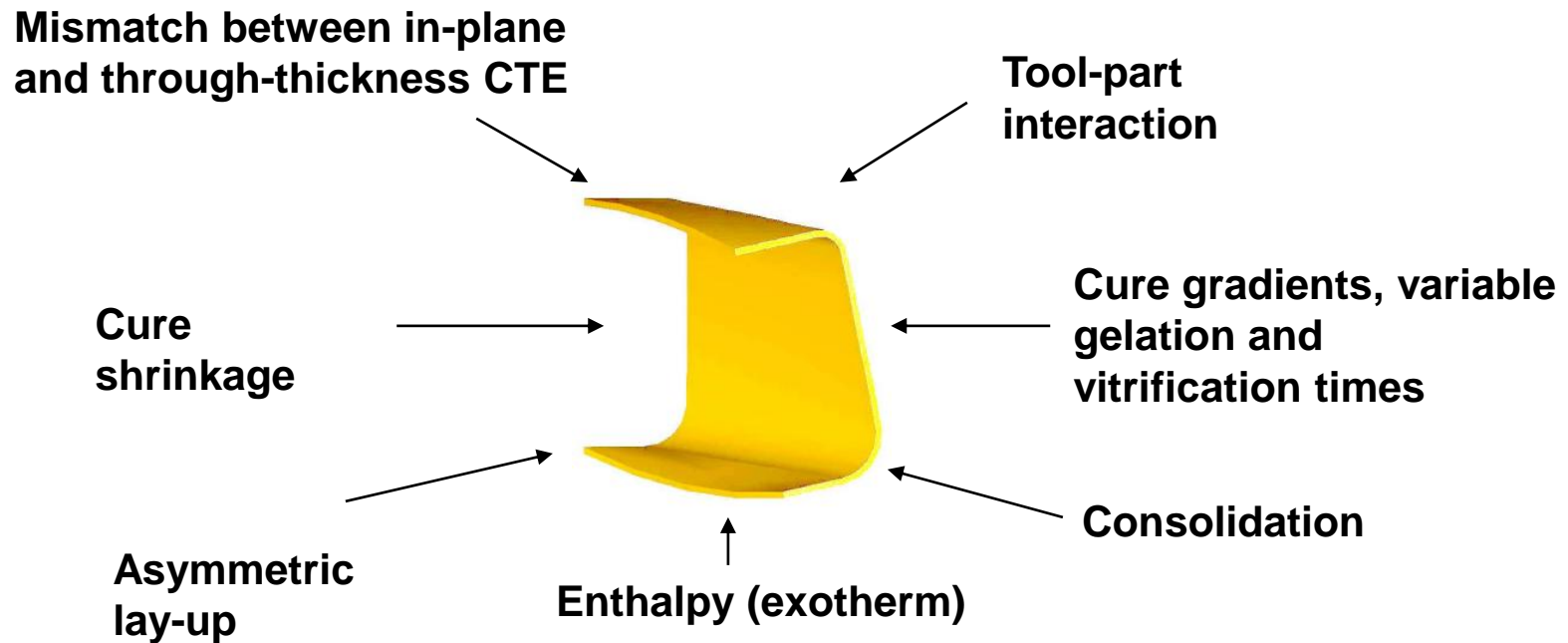
→ Structural Accuracy

→ impacting on final product performance and quality



<http://www.compositesworld.com/articles/composites-steady-radio-telescope-reflector> <https://www.fose1.plymouth.ac.uk/sme/MATS347/MATS347C6%20RTM.htm>

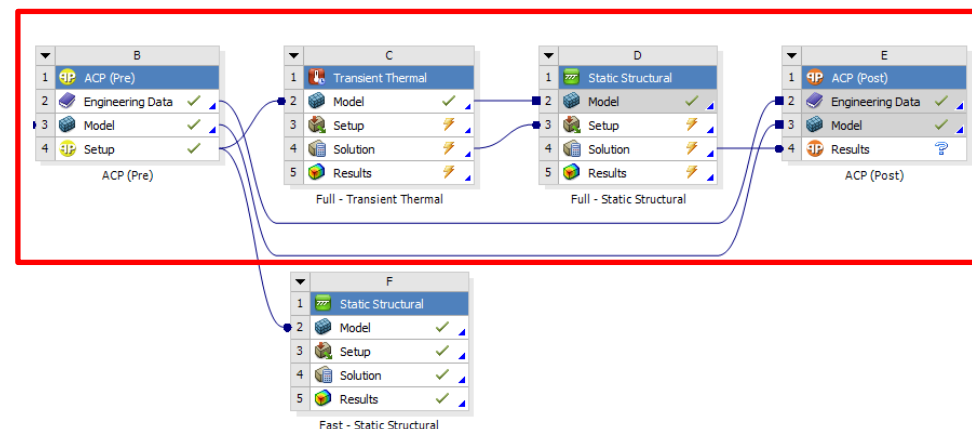
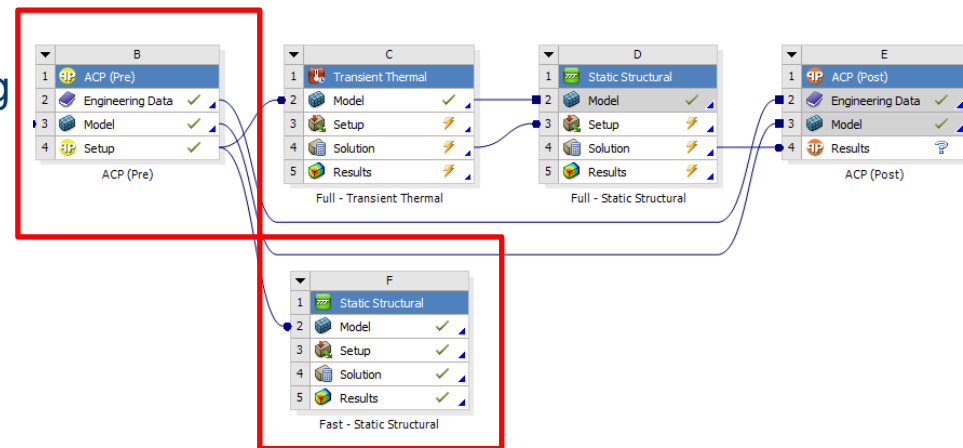
Sources of residual stresses and distortions



- Residual stresses that develop during cure lead to distortion and/or cracking significantly impacting on the final product performance
- Developed in partnership with LMAT Ltd.

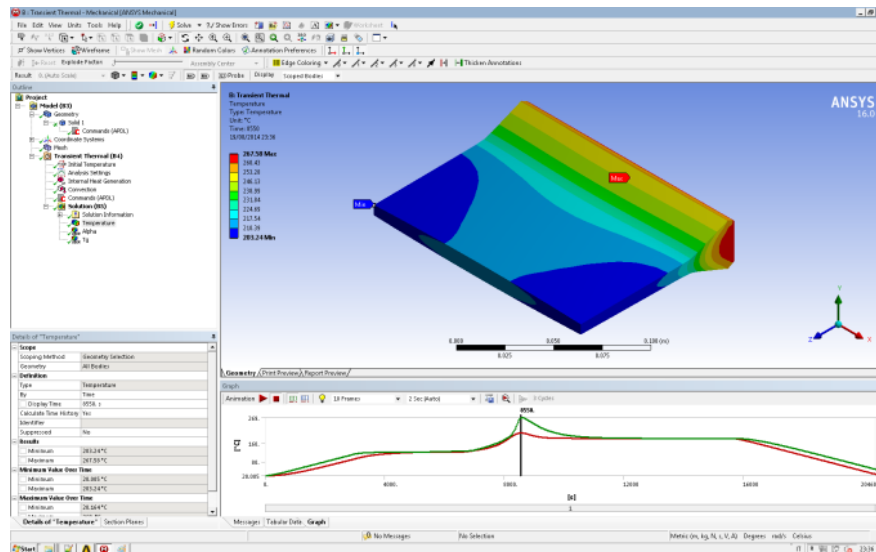
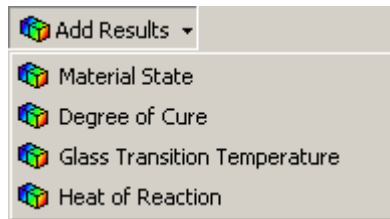
New ANSYS Cure Simulation Solution

- Fast cure simulation
 - For thin composite structures assuming known and uniform temperature distribution within the structure
 - Focus on prediction for distortion wrt. changes in laminate layup → designer level
- Full cure simulation
 - For thick composite structures where thermal heat generation from chemical reaction is dominant and non uniform
 - Transient thermal problem is solved and coupled to structural analysis
 - Focus on prediction for distortion and residual stresses as well as cure status, consolidation status → expert level with highest accuracy



New ANSYS Cure Simulation Solution – Full Cure Simulation

- Results in Thermal & Mechanical
- Results in Mechanical



Summary

- ✔ Intuitive workflow for composite modeling
- ✔ Quick postprocessing
- ✔ Solid modeling of composites
uses the same workflow
- ✔ Composite design studies - Optimize your design before you
build the first prototype
- ✔ Simple “drag-and-drop” multiphysics
- ✔ One single framework - ANSYS Workbench
- ✔ Input of Variable Material Properties
- ✔ ANSYS Composite Cure Simulation

Contact

CADFEM GmbH
Geschäftsstelle Frankfurt

Timo Frambach

Tel.: +49 (0) 6196-76708-21

Mobil: +49 (0)160 8970338

Fax: +49 (0) 6196-76708-44

E-Mail: tframbach@cadfem.de

Vielen Dank für Ihr Interesse!