Composites Analysis in ANSYS

© 2005 ANSYS, Inc.

ANSYS, Inc. Proprietary

Topics

- Composites and Advantages
- Traditional Composites Modeling
- Interface Delamination and Failure Simulation
- ANSYS-Fibersim Interface

© 2005 ANSYS, Inc.

ANSYS, Inc. Proprietary

Composites in the Industry

- By the broadest definition, a composite material is one in which two or more materials that are different are combined to form a single structure with an identifiable interface.
- The properties of that new structure are dependant upon the properties of the constituent materials as well as the properties of the interface.
- Additionally, where metal alloys have isotropic characteristics, composites can have very selective directional properties to meet specific application needs.

© 2005 ANSYS, Inc.

ANSYS, Inc. Proprietary

Composites in the Industry

- Composites used for typical engineering applications are advanced fiber or laminated composites, such as fiberglass, glass epoxy, graphite epoxy, and boron epoxy.
- ANSYS allows you to model composite materials with specialized elements called *layered elements*. You can perform any structural analysis (including nonlinearities such as large deflection and stress stiffening).

© 2005 ANSYS, Inc.

ANSYS, Inc. Proprietary

Benefits of Composites

- Stronger and stiffer than metals on a density basis
- Capable of high continuous operating temperatures
- Highly corrosion resistant
- Electrically insulating/conducting/selectively conducting properties
- Tailorable thermal expansion properties
- Exceptional formability
- Outstanding durability

© 2005 ANSYS, Inc.

ANSYS, Inc. Proprietary

Traditional Composites Analysis

Traditional Composites Modeling

- Element Technology
- Material Modeling and Layered Configuration
- Failure Criteria
- Additional Modeling and Post-Processing

© 2005 ANSYS, Inc.

Elements for Composites Analysis

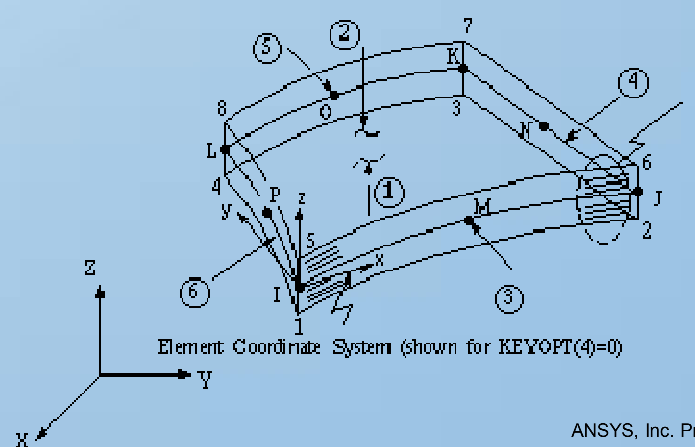
Element Type	Element Number
Shell	SHELL99; SHELL91; SHELL181
Solid	SOLID186; SOLID46; SOLID191; SOLID95
Solid-Shell	SOLSHL190
Beam	BEAM188; BEAM189

© 2005 ANSYS, Inc.

ANSYS, Inc. Proprietary

SHELL99
Linear Layered Structural Shell Element

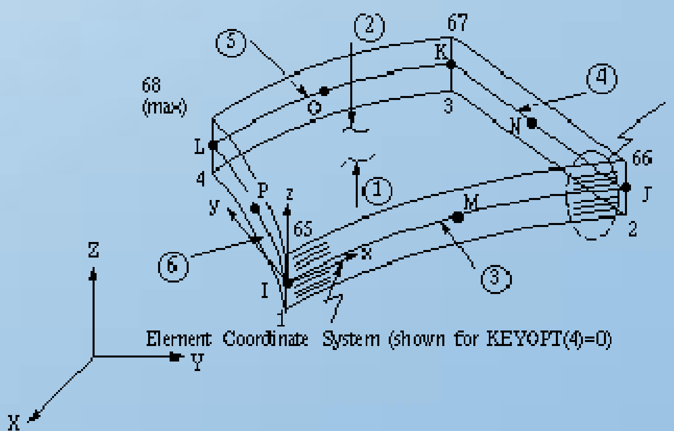
- Element Definition
  - 8-node, 3-D shell element with six degrees of freedom at each node
  - Thin to moderately thick plate and shell structures with a side-to-thickness ratio of roughly 10 or greater
- Layer Definition
  - Total of 250 uniform-thickness layers
  - 125 layers with bilinear thicknesses variation
  - Matrix form or Layer Form (Real Constants)
- Options
  - Option to offset the nodes to the top or bottom surface
  - NO Plasticity; large-strain behavior; sandwich option
  - Failure Modeling through TB, FAIL



Element Coordinate System (shown for KEYOPT(4)=0)

SHELL91
Nonlinear Layered Structural Shell Element

- Element Definition
  - 8-node, 3-D shell element with six degrees of freedom at each node
  - Used for modeling thick sandwhich structures
- Layer Definition
  - Total of 100 uniform-thickness layers
  - 125 layers with bilinear thicknesses variation
  - Layer Form (Real Constants)
- Options
  - No material property matrix input
  - Supports plasticity, large-strain behavior; laminated and sandwich structures
  - Failure Modeling through TB, FAIL

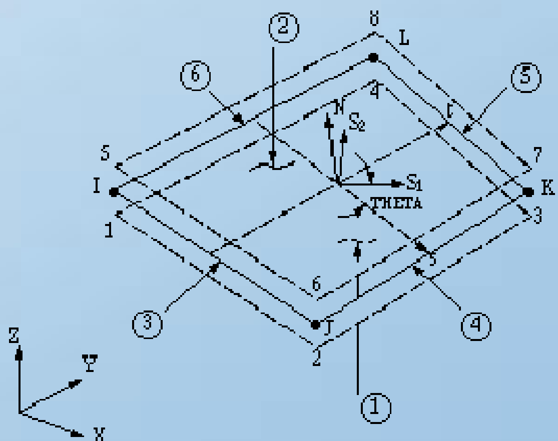


© 2005 ANSYS, Inc.

ANSYS, Inc. Proprietary

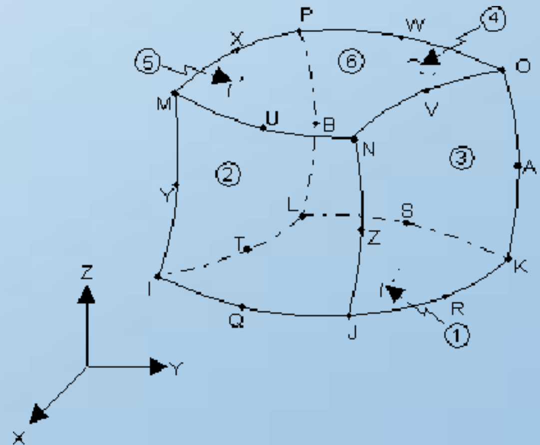
SHELL181
Finite Strain Shell

- Element Definition
  - 4-node 3-D shell element with 6 degrees of freedom at each node.
  - Used for layered applications for modeling laminated composite shells or sandwich construction. (First order shear deformation)
- Layer Definition
  - Total of 255 uniform/non-uniform section layers
  - Section commands rather than real constants.
  - Supports generalized section definition
- Options
  - Full nonlinear capabilities including large strain and material models
  - Failure criteria is available via **FC** and other **FC***xxx* commands.



SOLID186
3-D Layered Structural Solid Element

- Element Definition
  - Layered version of the 20-node 3-D solid element with three degrees of freedom per node (UX, UY, UZ).
  - The element can be stacked to model through-the-thickness discontinuities.
- Layer Definition
  - Total of 250 layers
  - Section commands rather than real constants.
- Options
  - Full nonlinear capabilities including large strain
  - Failure criteria is available via **FC** and other **FC***xxx* commands.

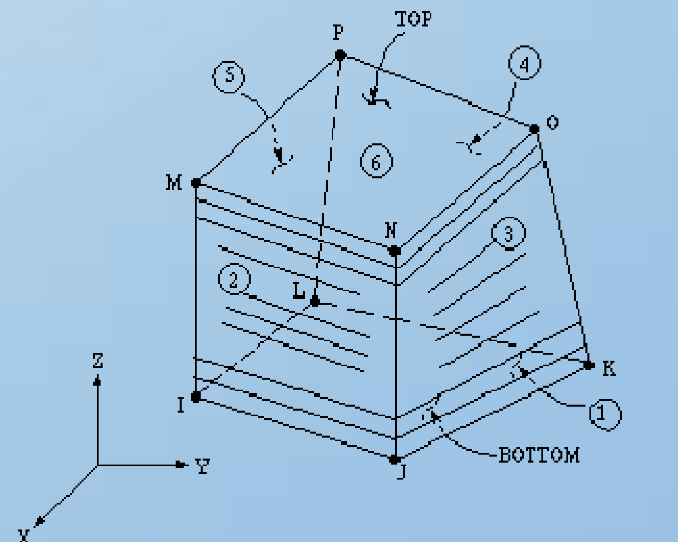


© 2005 ANSYS, Inc.

ANSYS, Inc. Proprietary

SOLID46
3-D Layered Structural Solid Element

- Element Definition
  - Layered version of the 8-node, 3-D solid element, SOLID45, with three degrees of freedom per node (UX, UY, UZ)
  - Designed to model thick layered shells or layered solids
  - Can stack several elements to model more than 250 layers to allow through-the-thickness deformation slope discontinuities.
- Layer Definition
  - Allows up to 250 uniform-thickness layers per element.
  - Allows 125 layers with thicknesses that may vary bilinearly
  - User-input constitutive matrix option
- Options
  - Nonlinear capabilities including large strain
  - Failure criteria through TB, FAIL option

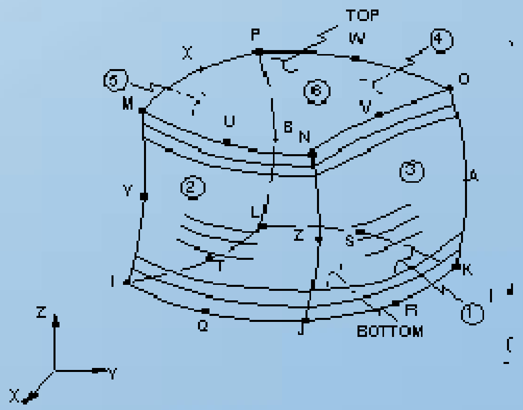


© 2005 ANSYS, Inc.

ANSYS, Inc. Proprietary

SOLID191
Layered Structural Solid Element

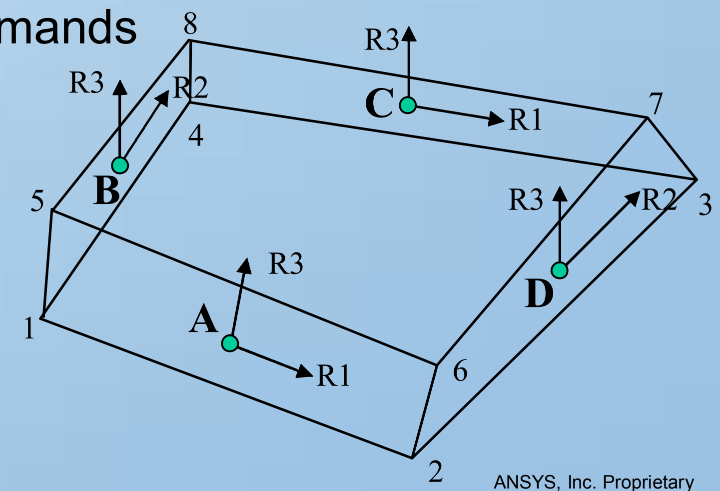
- Element Definition
  - Layered version of the 20-node 3-D solid element SOLID95, with three degrees of freedom per node (UX, UY, UZ).
  - Designed to model thick layered shells or layered solids
  - Can stack several elements to model more than 250 layers to allow through-the-thickness deformation slope discontinuities.
- Layer Definition
  - Allows up to 100 uniform-thickness layers per element.
  - Allows 125 layers with thicknesses that may vary bilinearly
  - User-input constitutive matrix option
- Options
  - No nonlinear capabilities
  - Failure criteria through TB, FAIL option



© 2005 ANSYS, Inc.

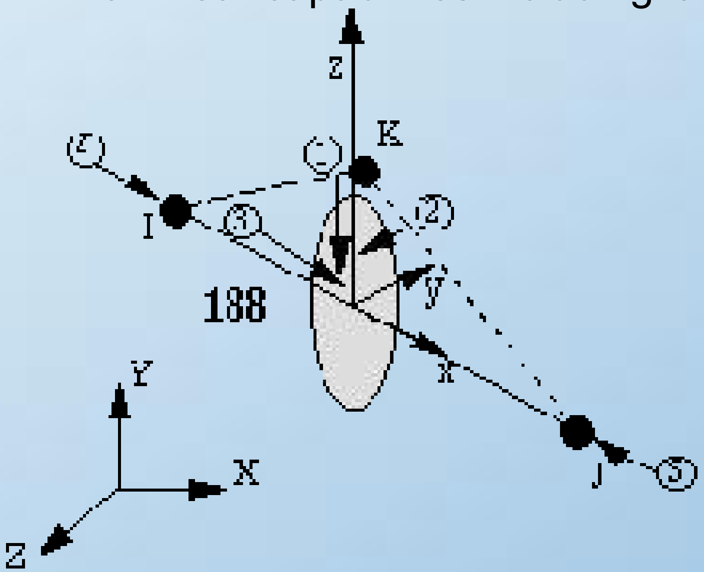
SOLSH190
3-D Layered Structural Solid Shell

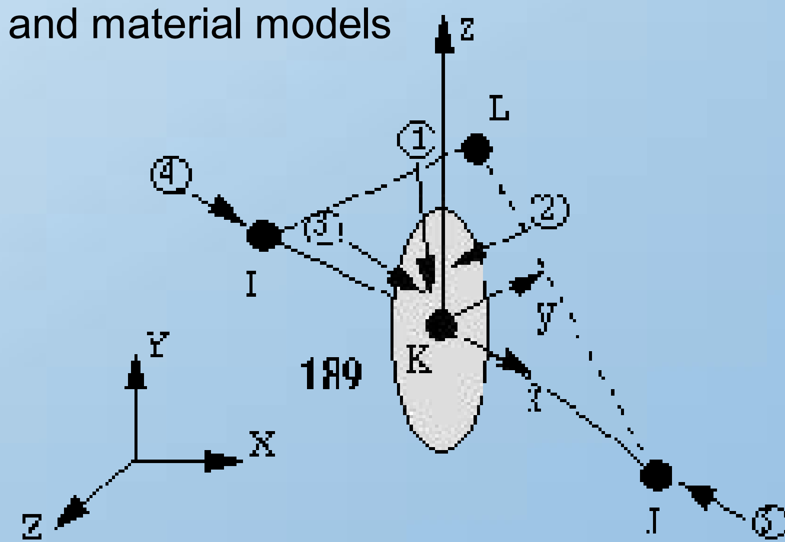
- Element Definition
  - 8-node 3-D solid-Shell element with three degrees of freedom per node (UX, UY, UZ).
  - The element can be stacked to model through-the-thickness discontinuities.
  - SOLSH190 can be used for simulating shell structures with a wide range of thickness (from thin to moderately thick).
- Layer Definition
  - The element has full nonlinear capabilities including large strain and allows 250 layers for modeling laminated shells.
- Options
  - Failure criteria is available using the FC commands
  - Full nonlinear capabilities including large strain and material models
  - Supported by EORIENT and VEORIENT commands



© 2005 ANSYS, Inc.

- Element Definition
  - BEAM188/BEAM189, linear and quadratic versions of the 3-D finite strain beam elements with six DOF
  - Suitable for analyzing slender to moderately stubby/thick beam structures. This element is based on Timoshenko beam theory. Shear deformation effects are included.
- Layer Definition
  - Through Section commands
  - Multi material cross section, custom cross section, inclusion and prediction of transverse shear stresses
- Options
  - Nonlinear capabilities including large strain and material models





© 2005 ANSYS, Inc.

Material Modeling & Layered Configuration

- The most important characteristic of a composite material is its layered configuration.
- Each layer may be made of a different orthotropic material and may have its principal directions oriented differently.
- For laminated composites, the fiber directions determine layer orientation.
- Two methods are available for defining the layered configuration:
  1. By specifying individual layer properties
  2. By defining constitutive matrices that relate generalized forces and moments to generalized strains and curvatures

© 2005 ANSYS, Inc.

ANSYS, Inc. Proprietary

Individual Layer Properties: First Method

- With this method, the layer configuration is defined layer-by-layer from bottom to top.
- The bottom layer is designated as layer 1, and additional layers are stacked from bottom to top in the positive Z (normal) direction of the element coordinate system.
- At times, a physical layer will extend over only part of the model. In order to model continuous layers, these dropped layers may be modeled with zero thickness.

© 2005 ANSYS, Inc.

ANSYS, Inc. Proprietary

Individual Layer Properties

- Material Property
  - **MP** command is used to define the linear material properties.
  - **TB** command is used to define the nonlinear material data tables
- The only difference is that the material attribute number for each layer of an element is specified in the element's real constant table or Section data
- The linear material properties for each layer may be either isotropic or orthotropic
- Material property directions are parallel to the layer coordinate system, defined by the element coordinate system and the layer orientation angle

© 2005 ANSYS, Inc.

ANSYS, Inc. Proprietary

Defining Layer Data

- Using Layered sections through the Section Tool
  - For each layer, the following are specified in the section definition through the section commands; or through the Section Tool (**SECTYPE**, **SECDATA**) (accessed with the **SECNUM** attributes).
  - Material properties (via a material reference number MAT)
  - Layer orientation angle commands (THETA)
  - Layer thickness (TK)
  - Number of integration points per layer (NUMPT)

© 2005 ANSYS, Inc.

ANSYS, Inc. Proprietary

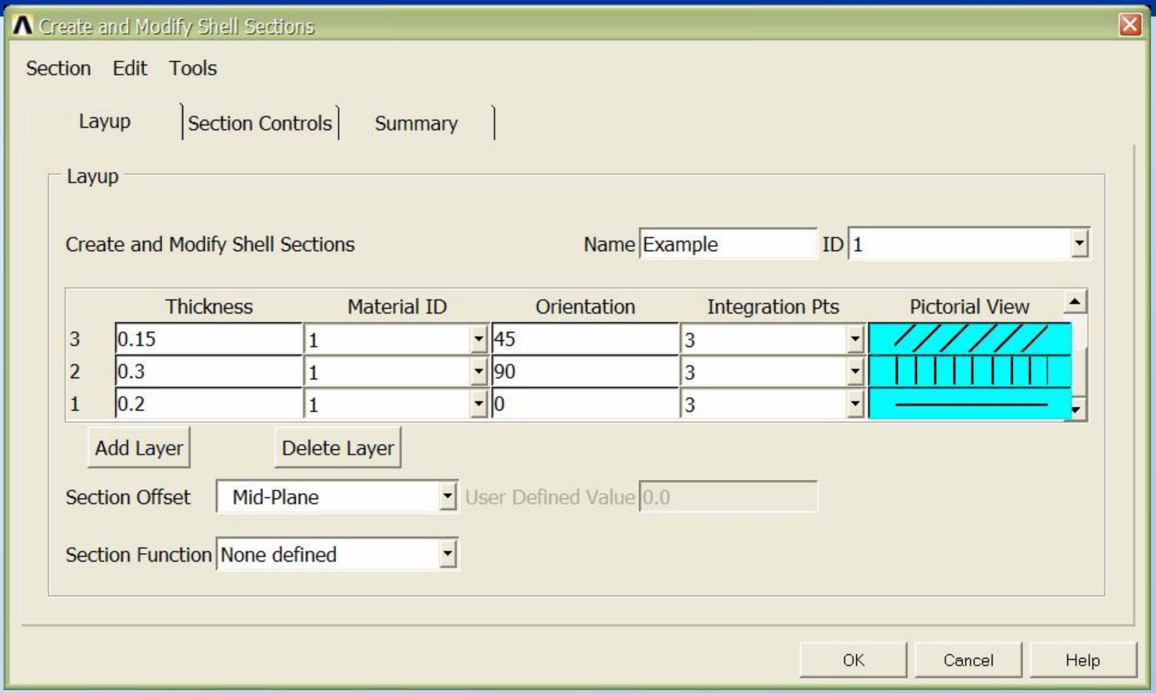
Layer Data Definition

- Layer Orientation Angle
  - This defines the orientation of the layer coordinate system with respect to the element coordinate system.
  - It is the angle (in degrees) between X-axes of the two systems. By default, the layer coordinate system is parallel to the element coordinate system.
- Layer Thickness
  - If the layer thickness is constant, you only need to specify TK(I), the thickness at node I. Otherwise, the thicknesses at the four corner nodes must be input.
  - Dropped layers may be represented with zero thickness.
- Number of integration points per layer
  - This allows you to determine in how much detail the program should compute the results.
  - For very thin layers, when used with many other layers, one point would be appropriate. But for laminates with few layers, more would be needed. The default is 3 points.

© 2005 ANSYS, Inc.

ANSYS, Inc. Proprietary

Shell Section Data



- SecType, SecId, SHELL, Name
- SecData, Thick, MatID, Ori, numSectPt

Thick	Layer Thickness
MatID	Material ID for layer
Ori	Layer Orientation
numSectPt	Number of integration points thru layer thickness

Repeat as many times as needed

Constitutive Matrices: Second Method

- This is an alternative to specifying the individual layer properties
- Available as an option KEYOPT(2) for SOLID46 and SHELL99.
- The matrices, which represent the force-moment and strain-curvature relationships for the element, must be calculated outside the ANSYS program
- They can be included as part of the solution printout with KEYOPT(10).
- The main advantages of the matrix approach are:
  - It allows you to incorporate an aggregate composite material behavior.
  - A thermal load vector may be supplied.
  - The matrices may represent an unlimited number of layers.
  - The terms of the matrices are defined as real constants.

© 2005 ANSYS, Inc.

ANSYS, Inc. Proprietary

Preintegrated General Shell Sections

- You can now define homogenous shell section behavior directly via preintegrated general shell sections, a method commonly used in analyses involving laminated composite structures.
- With preintegrated shell sections (**SECTYPE**,,GENS), you can directly specify the membrane, bending, and coupling properties.
- The preintegrated method also allows analysis of complex geometry (with repeated patterns such as corrugated sheets) using equivalent shell section properties.
- You can use preintegrated general shell sections when using the **SHELL181** element, provided that linear elastic material behavior is acceptable.
- Preintegration
  - Requires fewer system resources because numerical integration through the thickness of the shell is not required.
  - Allows import of homogenous section-stiffness constants evaluated in other analyses or by third-party, special-purpose software tools.

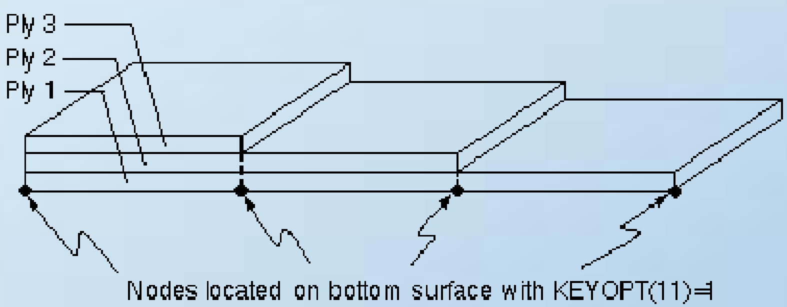
© 2005 ANSYS, Inc.

ANSYS, Inc. Proprietary

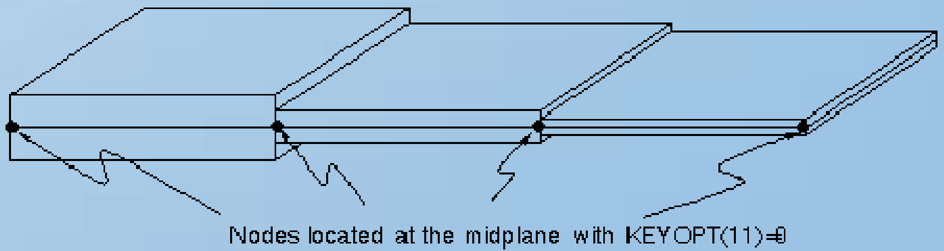
Node Offset

- For SHELL181 using sections defined through the section commands, nodes can be offset during the definition of the section, using the SECOFFSET command.
- For SHELL91, and SHELL99 the node offset option (KEYOPT(11)) locates the element nodes at the bottom, middle or top surface of the shell.

Layered Shell With Nodes at Bottom Surface for SHELL91 and SHELL99



Layered Shell With Nodes at Midplane for SHELL91 and SHELL99



© 2005 ANSYS, Inc.

Failure Criteria

- Failure criteria are used to learn if a layer has failed due to the applied loads. You can choose from three predefined failure criteria or specify up to six failure criteria of your own (user-written criteria). The three predefined criteria are:
  - *Maximum Strain Failure Criterion*, which allows nine failure strains
  - *Maximum Stress Failure Criterion*, which allows nine failure stresses
  - *Tsai-Wu Failure Criterion*, which allows nine failure stresses and three additional coupling coefficients. You have a choice of two methods of calculating this criterion.
- The failure strains, stresses, and coupling coefficients may be temperature-dependent.

© 2005 ANSYS, Inc.

ANSYS, Inc. Proprietary

Specifying Failure Criteria

- Failure criteria are commonly used for orthotropic materials. They can be input using either the **FC** commands or the **TB** commands
- **TB** Command: **TB, FAIL** Composite material failure data
  - Applies to SOLID46, SHELL91, SOLID95, SHELL99, SOLID191
- The failure criteria table is started by using the **TB** command (with *Lab* = FAIL). The data table is input in two parts:
  - the failure criterion keys
  - the failure stress/strain data.
- Data not input are assumed to be zero. The six failure criterion keys are defined with the **TBDATA** command following a special form of the **TBTEMP** command [**TBTEMP**,,CRIT] to indicate that the failure criterion keys are defined next

© 2005 ANSYS, Inc.

TB, FAIL and FC Commands

```
TB,FAIL,1,2 ! Data table for failure criterion, material 1, ! no. of temperatures = 2
TBTEMP,,CRIT ! Failure criterion key
TBDATA,2,1 ! Maximum Stress Failure Criterion (Const. 2 = 1)
TBTEMP,100 ! Temperature for subsequent failure properties
TBDATA,10,1500,,400,,10000 ! X, Y, and Z failure tensile stresses (Z value ! set to a large number)
TBDATA,16,200,10000,10000 ! XY, YZ, and XZ failure shear stresses
TBLIST TBTEMP,200 ! Second temperature
TBDATA,...
```

```
FC,1,TEMP,, 100, 200 ! Temperatures
FC,1,S,XTEN, 1500, 1200 ! Maximum stress components
FC,1,S,YTEN, 400, 500
FC,1,S,ZTEN,10000, 8000
FC,1,S,XY , 200, 200
FC,1,S,YZ ,10000, 8000
FC,1,S,XZ ,10000, 8000
FCLIST, ,100 ! List status of Failure Criteria at 100.0 degrees
FCLIST, ,150 ! List status of Failure Criteria at 150.0 degrees
FCLIST, ,200 ! List status of Failure Criteria at 200.0 degrees
PRNSOL,S,FAIL ! Use Failure Criteria
```

© 2005 ANSYS, Inc.

ANSYS, Inc. Proprietary

Failure Criteria …

- The **TB** commands (**TB**, **TBTEMP**, and **TBDATA**) can be used only for SHELL91, SHELL99, SOLID46, or SOLID191, but the **FC** and **FCLIST** commands can be used for any 2-D or 3-D structural solid element or any 3-D structural shell element.
- Some notes about specifying failure criteria:
  - The criteria are orthotropic, so you *must* input the failure stress or failure strain values for *all* directions. (The exception is that compressive values default to tensile values.)
  - If you don't want the failure stress or strain to be checked in a particular direction, specify a large number in that direction
  - User-written failure criteria may be specified via user subroutines USRFC1 through USRFC6. These subroutines should be linked with the ANSYS program beforehand

© 2005 ANSYS, Inc.

ANSYS, Inc. Proprietary

Pre & Post Processing Commands

- **LAYLIST** lists the layer stacking sequence from real constants and any two material properties
  - SHELL99, SHELL91, SOLID46, and SOLID191 elements
- **LAYPLOT** displays the layer stacking sequence from real constants in the form of a sheared deck of cards.
  - The layers are crosshatched and color coded for clarity.
  - The hatch lines indicate the layer angle (real constant THETA) and the color indicates layer material number (MAT).
- **SECPLOT** displays the section stacking sequence from sections in the form of a sheared deck of cards
  - The sections are crosshatched and color coded for clarity.
  - The hatch lines indicate the layer angle (THETA) and the color indicates layer material number (MAT) defined by the **SECDATA** command.
- Use the **LAYER** command in POST1 (or **LAYERP26** in POST26) to specify the layer number for which results are to be processed.
- The **SHELL** command specifies a TOP, MID, or BOT location *within* the layer

© 2005 ANSYS, Inc.

ANSYS, Inc. Proprietary

Interlaminar shear stresses

- Interlaminar shear stresses are usually important at the free edges of a model.
- For relatively accurate interlaminar shear stresses at these locations, the element size at the boundaries of the model should be approximately equal to the total laminate thickness.
- For shells, increasing the number of layers per actual material layer does not necessarily improve the accuracy of interlaminar shear stresses.

© 2005 ANSYS, Inc.

ANSYS, Inc. Proprietary

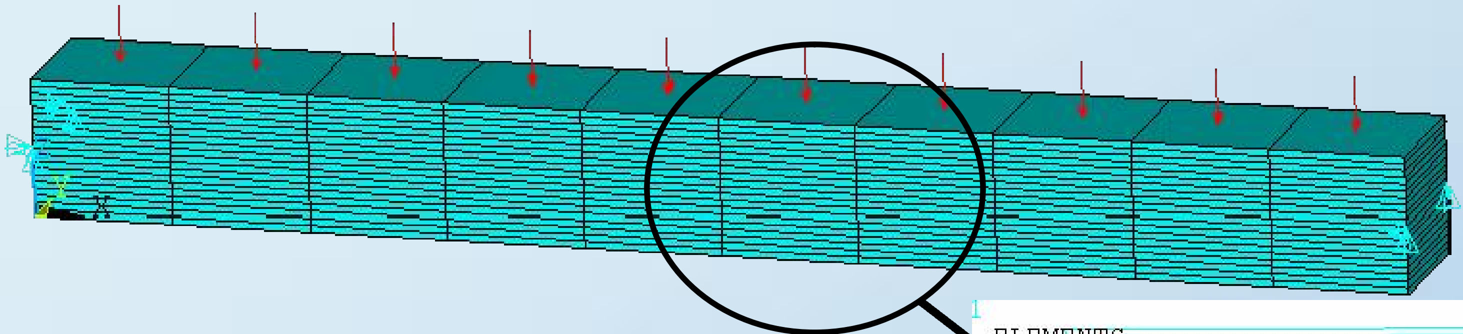
Interlaminar shear stresses

- With elements SOLID46, SOLID95, SOLSH190 and SOLID191, however, stacking elements in the thickness direction should result in more accurate interlaminar stresses through the thickness.
- Interlaminar transverse shear stresses in shell elements are based on the assumption that no shear is carried at the top and bottom surfaces of the element.
- These interlaminar shear stresses are only computed in the interior and are not valid along the shell element boundaries.
- Use of shell-to-solid submodeling is recommended to accurately compute all of the free edge interlaminar stresses.

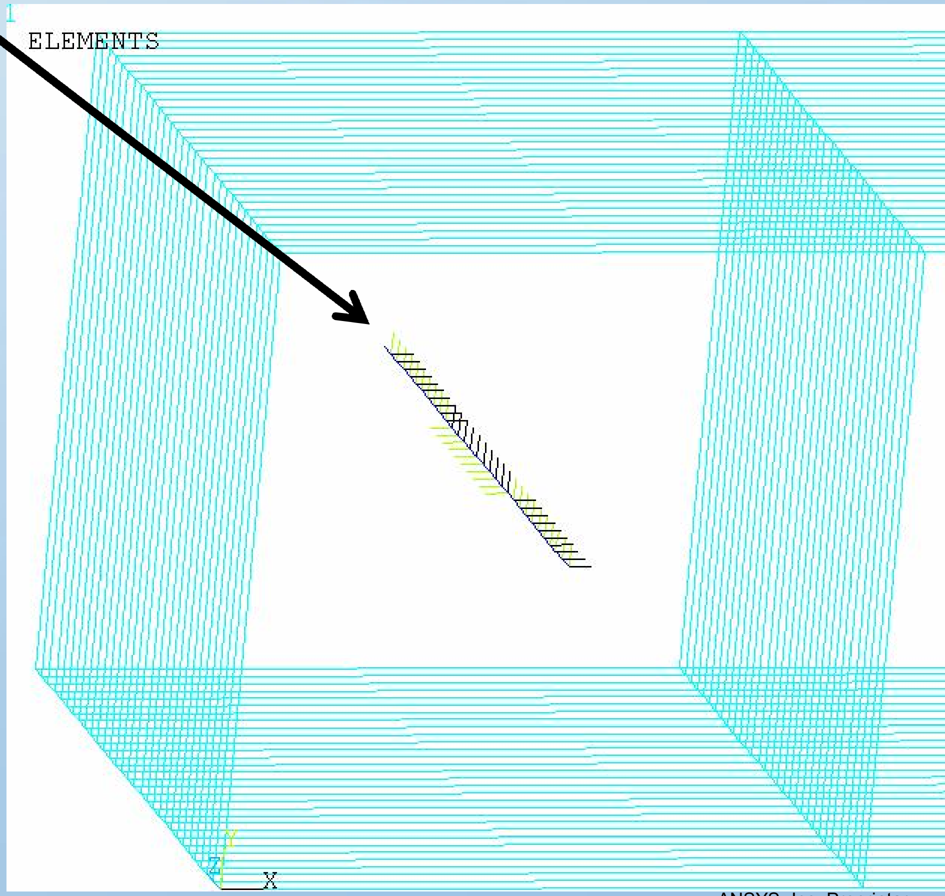
© 2005 ANSYS, Inc.

ANSYS, Inc. Proprietary

0/90 Symmetric Composite Example



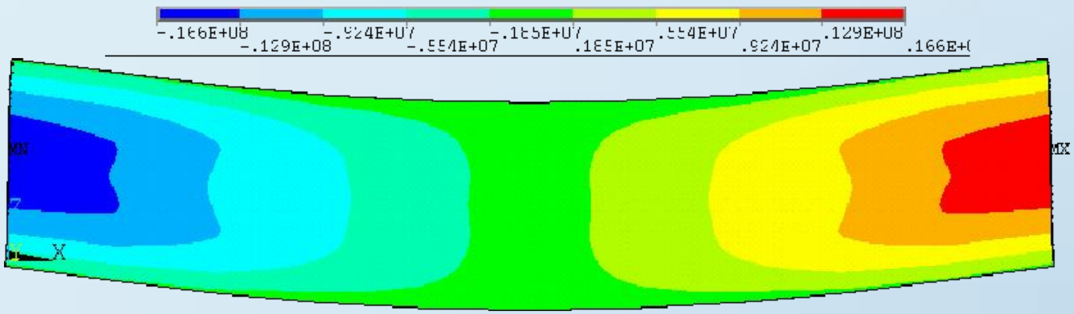
EX = 171GPa
EY = 3.42GPa
EZ = 3.42GPa
PRXY=0.25
PRYZ=0.25
PRXZ=0.25
GXY = 3.42GPa
GYZ = 1.37GPa
GXZ = 1.37GPa



© 2005 ANSYS, Inc.

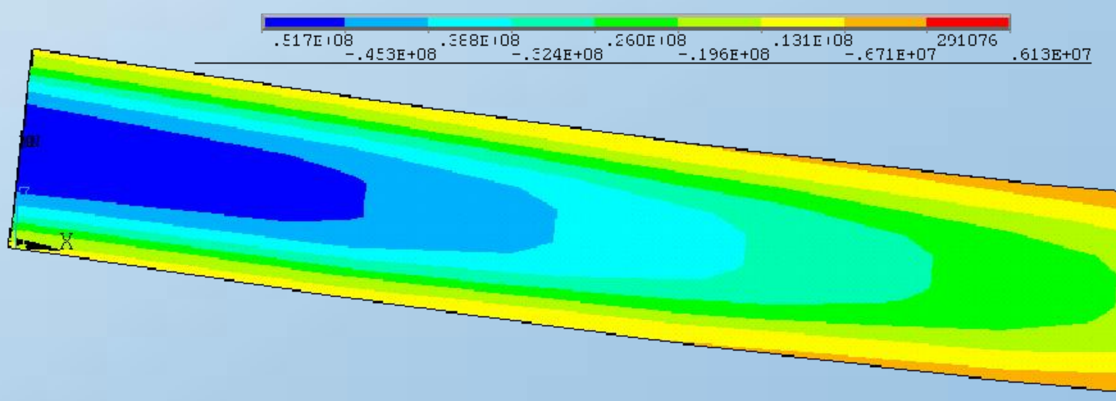
ANSYS, Inc. Proprietary

0/90 Symmetric Composite Example …



Solid Shell 190

h/L = 0.2; # layers: 30



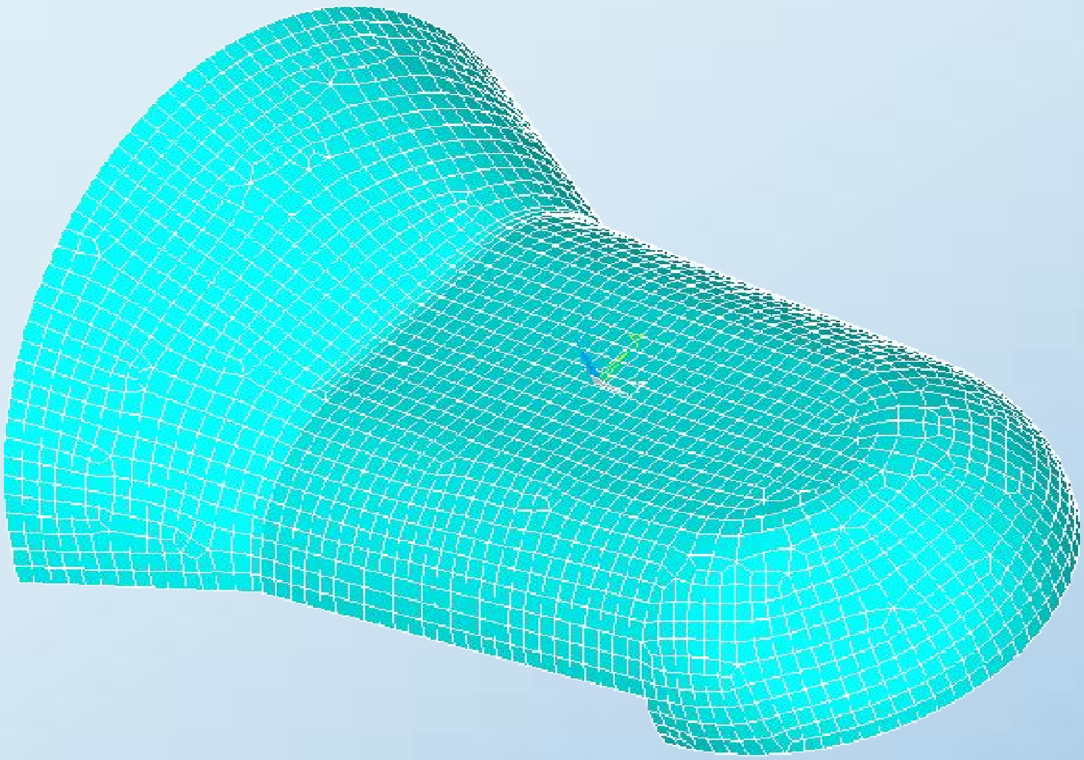
Solid Shell 190

h/L = 0.06; # layers: 12

© 2005 ANSYS, Inc.

ANSYS, Inc. Proprietary

Layered section Definition Example



Linear Orthotropic Properties for Material Number 1

Linear Orthotropic Material Properties for Material Number 1

Choose Poisson's Ratio

	T1
Temperatures	0
EX	171
EY	3.42
EZ	2.5
NUXY	0.3
NUYZ	0.2
NUXZ	0.15
GXY	3.42
GYZ	1.37
GXZ	1.37

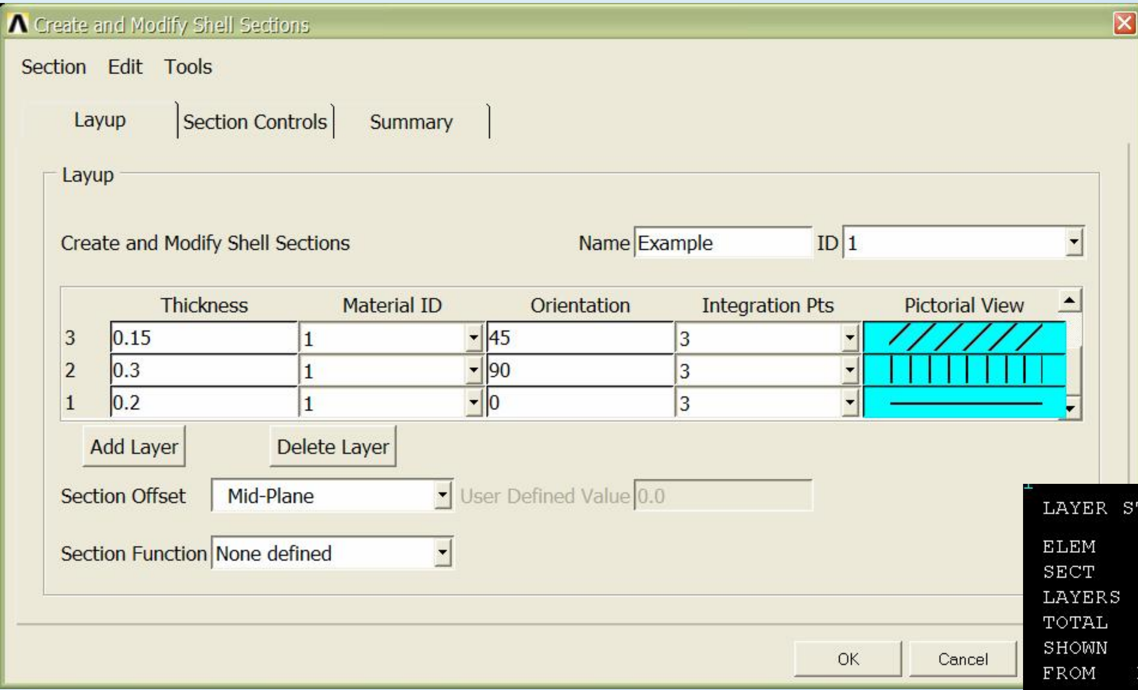
Add Temperature   Delete Temperature   Graph

OK   Cancel   Help

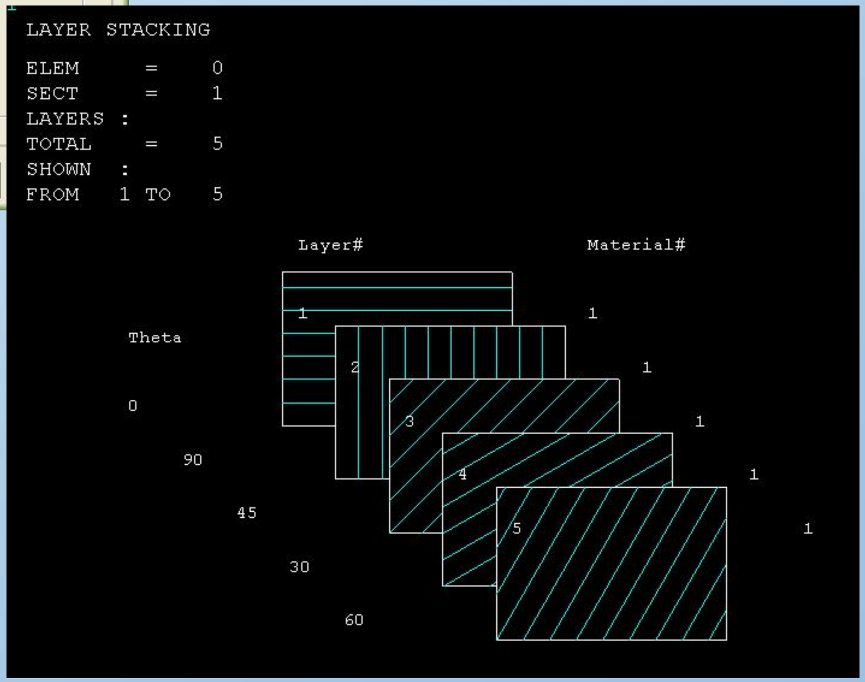
© 2005 ANSYS, Inc.

ANSYS, Inc. Proprietary

Layered section Definition Example …

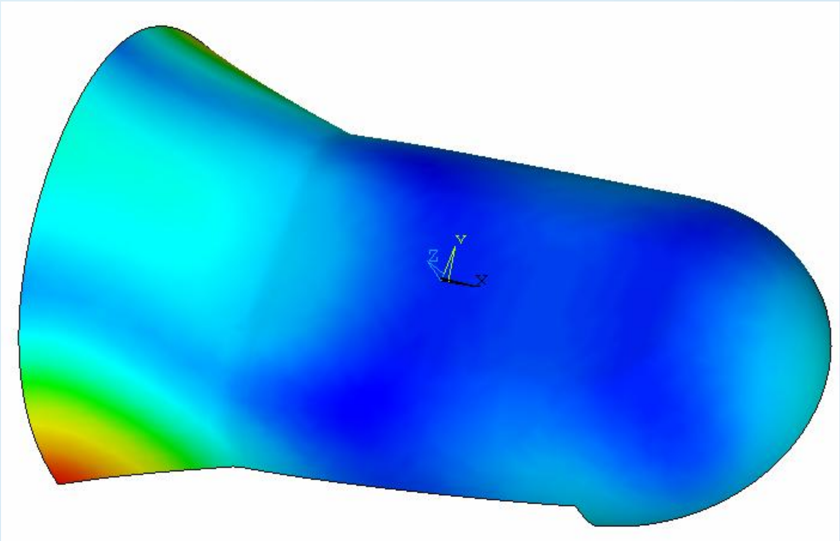


SECPLOT Command



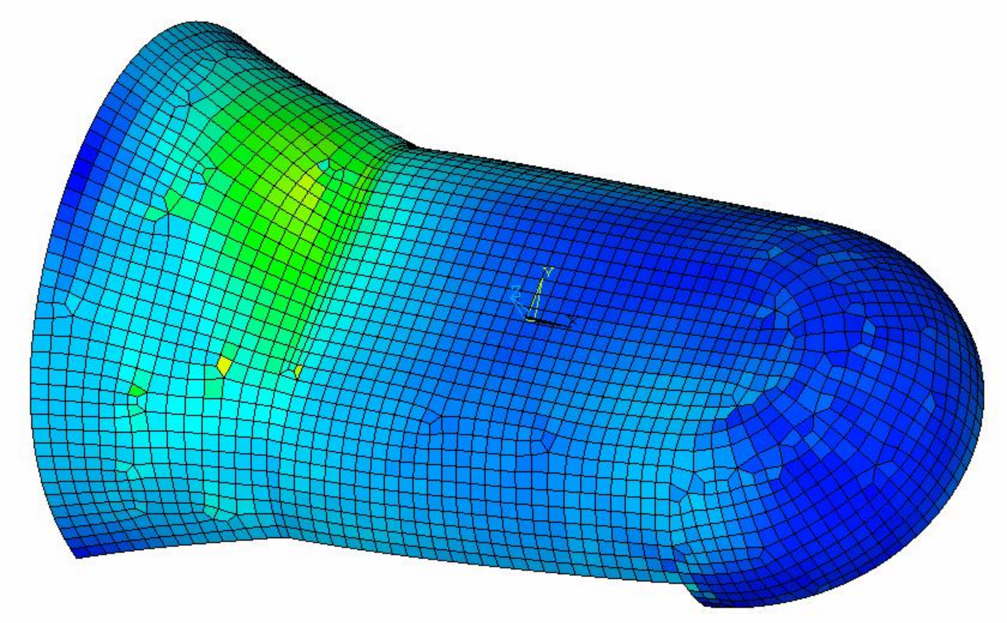
© 2005 ANSYS, Inc.

Layered section Definition Example …



Nodal displacement results

Element Stress result for layer 5



Interface Delamination and Failure

Interface Delamination and Failure

- An interface exists anywhere two materials are joined together.
- The interface between the layers of a composite structure is of special interest, because when this type of structure is subjected to certain types of external loading, the failure process (delamination) takes on a unique character.
- Interface delamination is traditionally simulated using fracture mechanics methods, such as nodal release technique.
- Alternatively, you can use the cohesive zone model to simulate interface delamination and other fracture pheneomenon.

© 2005 ANSYS, Inc.

ANSYS, Inc. Proprietary

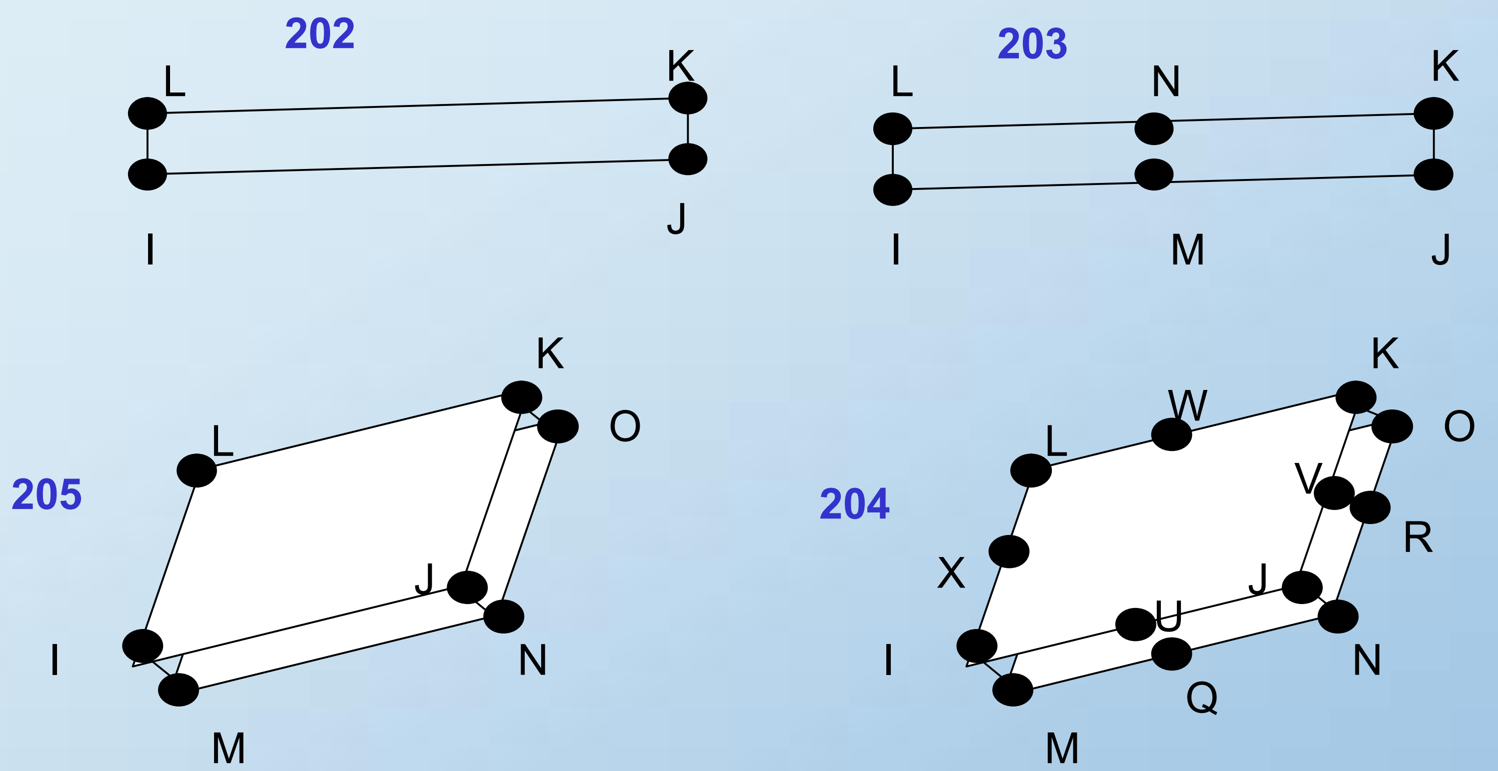
Cohesize Zone Model

- This approach introduces failure mechanisms by using the hardening-softening relationships between the separations and incorporating the corresponding tractions across the interface.
- This technique is also well suited for modeling the fracture process in a homogenous medium, since fracture can be viewed as a separation process between two surfaces
- 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 a set of interface elements designed specifically to represent the cohesive zone between the components and to account for the separation across the interface.

© 2005 ANSYS, Inc.

ANSYS, Inc. Proprietary

Cohesive Elements and Material



Interface elements with zero thickness

© 2005 ANSYS, Inc.

Interface Elements

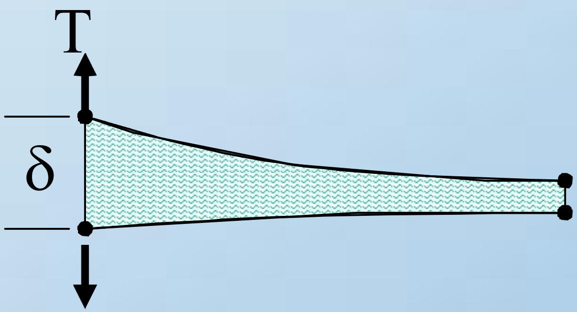
Elements Characteristics	Interface Element	Structural Elements
2-D, linear	INTER202	PLANE42, VISCO106, PLANE182
2-D, quadratic	INTER203	PLANE2, PLANE82, VISCO88, PLANE183
3-D, quadratic	INTER204	SOLID92, SOLID95, SOLID186, SOLID187
3-D, linear	INTER205	SOLID45, SOLID46, SOLID64, SOLID65, SOLID185, SOLSH190

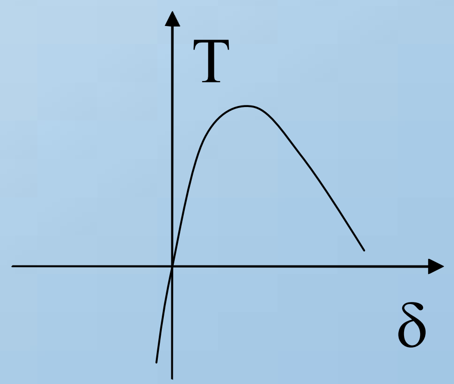
© 2005 ANSYS, Inc.

ANSYS, Inc. Proprietary

Cohesive Element and Material

- Element characteristics
  – Primary interest in tension/opening
  – Separation behavior is described by a traction separation law
  – Accounts for contact with a simple penalty approach
  – Works with contact elements for better contact modeling





Cohesive Element and Material

- Material definition
  - Material model parameters

    $\sigma_{max} \quad \bar{\delta}_n \quad \bar{\delta}_t$
  - Data input

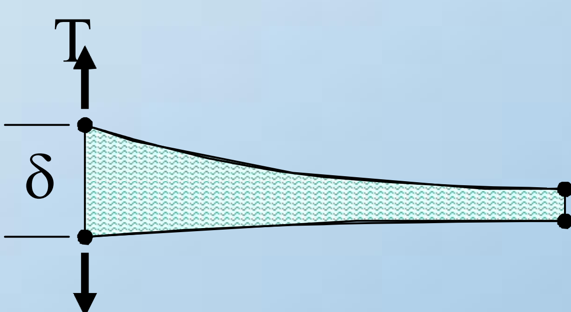
    TB,CZM,mat,ntemp,npts,EXPO

    TBDATA,1,c1,c2,c3

    C1 – σ_{max}

    C2 – $\bar{\delta}_n$

    C3 – $\bar{\delta}_t$



- Cohesive zone model
  - Exponential model

    Surface potential

Traction across the surface

$$\phi(\mathbf{\Delta}) = e\sigma_{max}\bar{\delta}_n\left[1-\left(1+\frac{\delta_n}{\bar{\delta}_n}\right)\exp\left(-\frac{\delta_t^2}{\bar{\delta}_t^2}\right)\exp\left(-\frac{\delta_n}{\bar{\delta}_n}\right)\right]$$

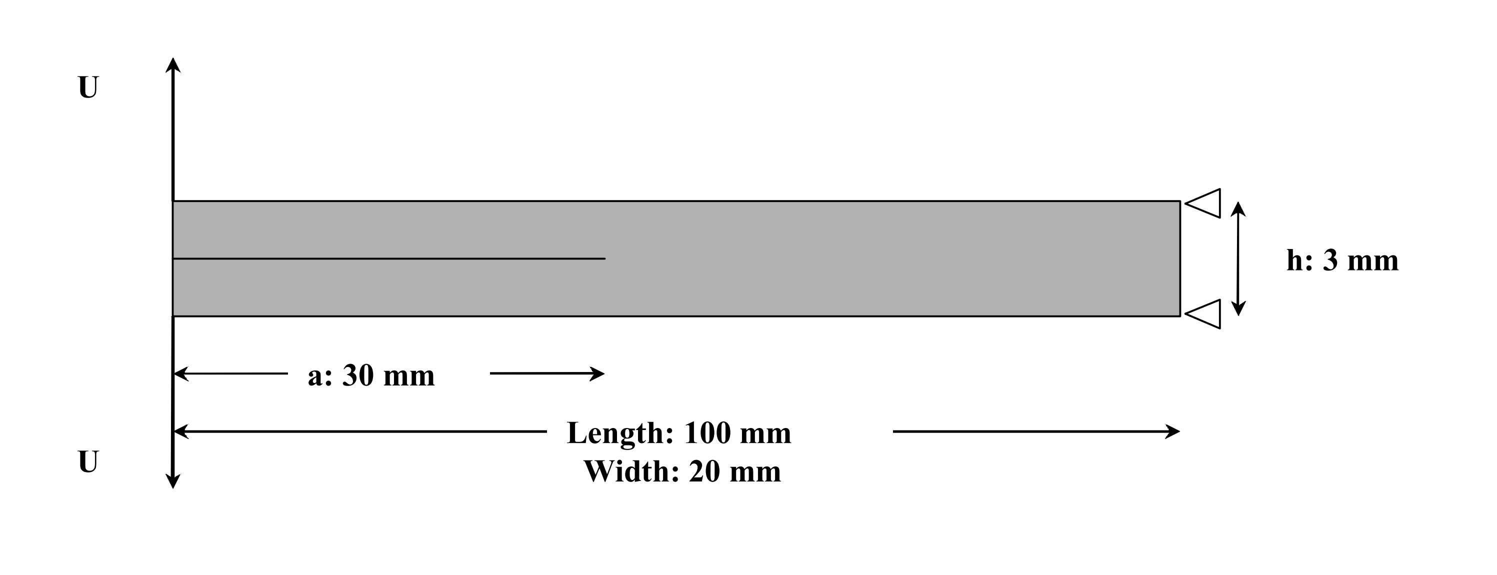
$$\mathbf{T} = \frac{\partial\phi(\mathbf{\Delta})}{\partial\mathbf{\Delta}}$$

© 2005 ANSYS, Inc.

ANSYS, Inc. Proprietary

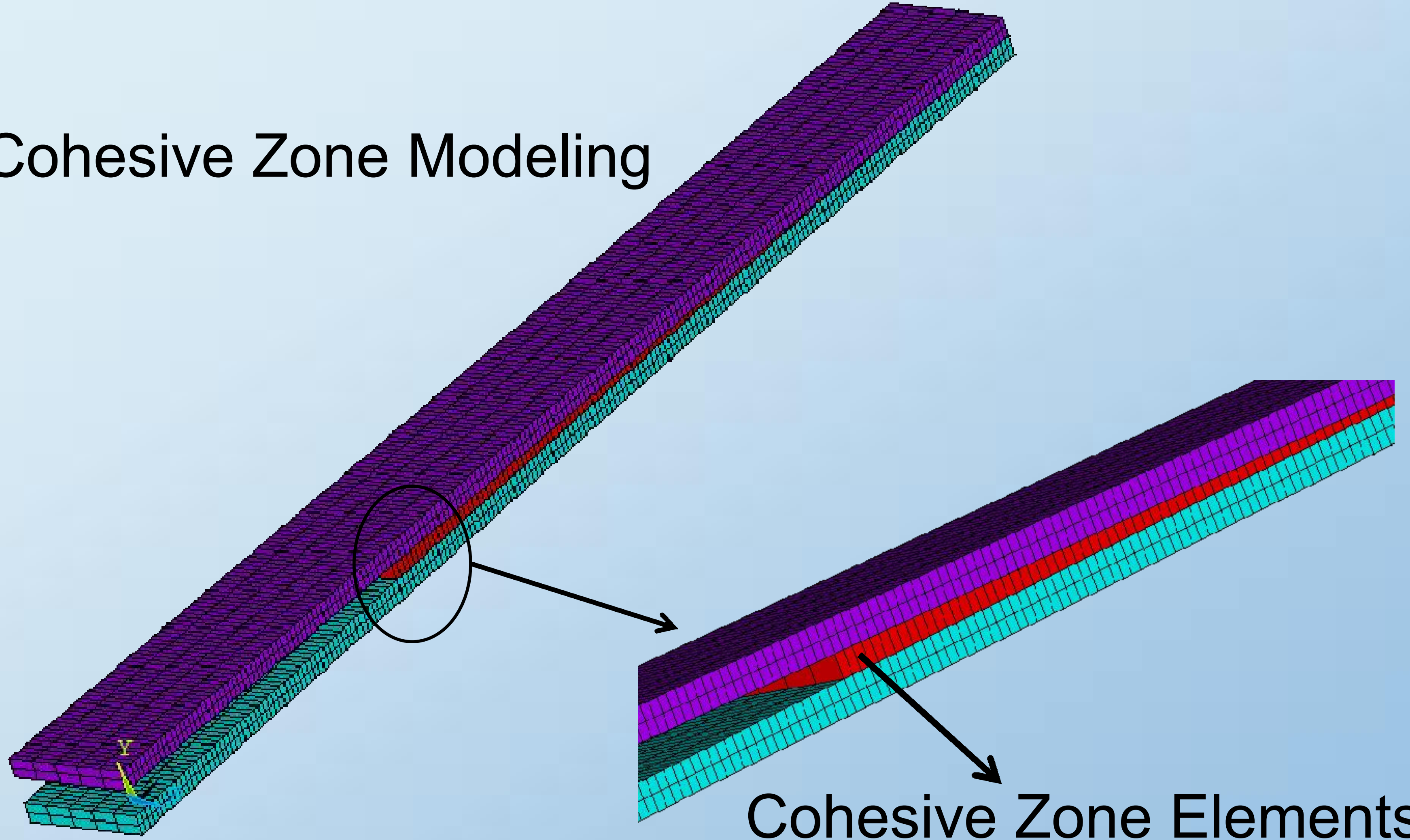
Interface Delamination Example

Problem description



Reference:
Alfano, G. and Crisfield, M. A., “Finite element interface models for the delamination analysis of laminated composites: mechanical and computational issues”, *International Journal for Numerical Methods in Engineering, 2001, 50:1701-1736.*

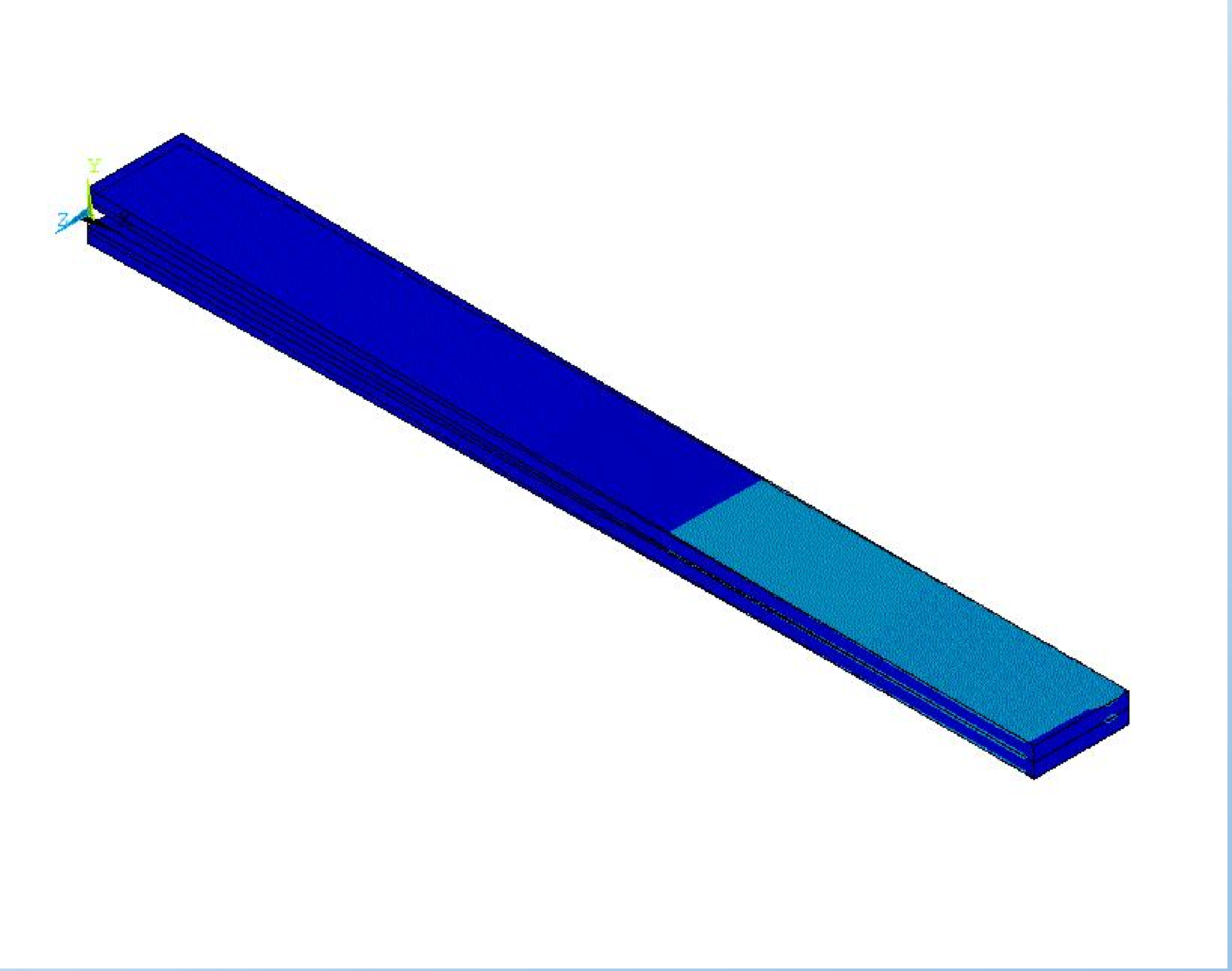
Interface Delamination Example …



© 2005 ANSYS, Inc.

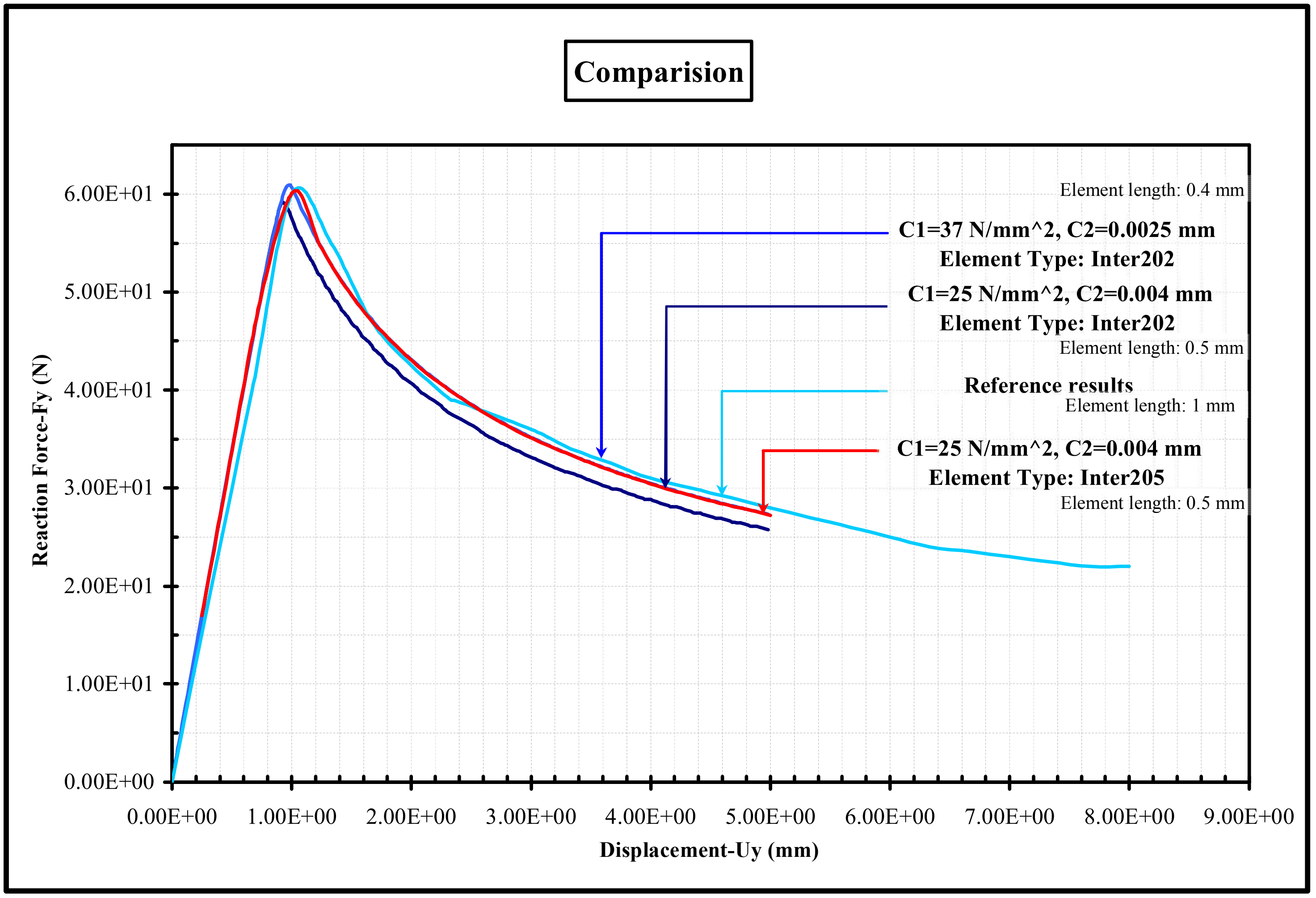
ANSYS, Inc. Proprietary

Interface Delamination Example …



© 2005 ANSYS, Inc.

Interface Delamination Example …



© 2005 ANSYS, Inc.

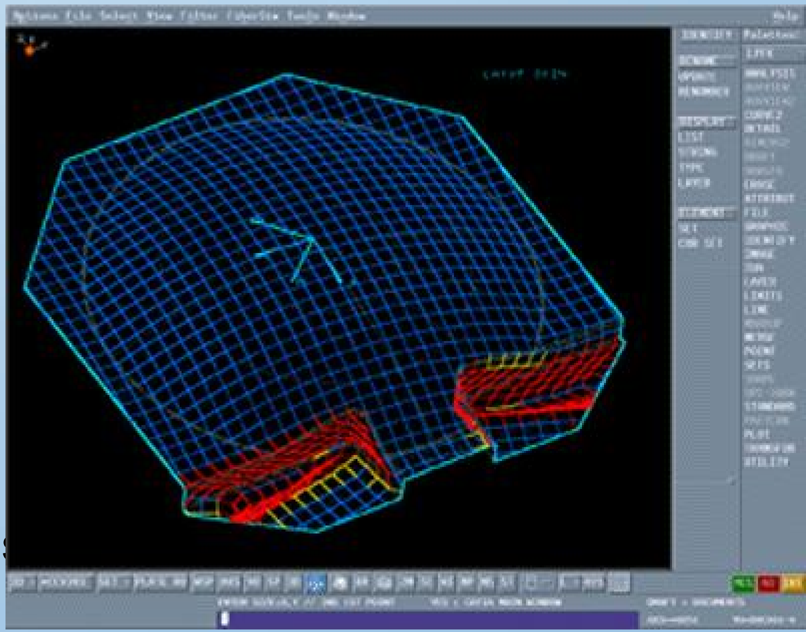
ANSYS – Fibersim Interface

© 2005 ANSYS, Inc.

ANSYS, Inc. Proprietary

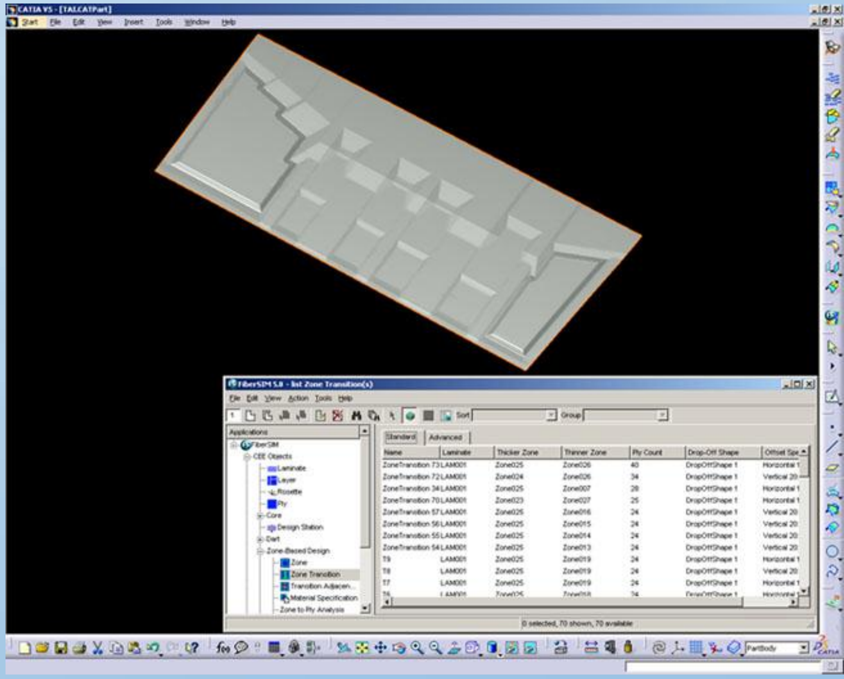
ANSYS Fibersim Interface

- VISTAGY, Inc. develops and markets the FiberSIM suite of software for composite design and manufacturing.
- www.vistagy.com/products/fibersim.htm
- By supporting the composite engineering process from the early stages of design all the way through to manufacturing, FiberSIM enables engineers to fully exploit these materials quickly and cost effectively.
- FiberSIM provides CAD-integrated software solutions for:
  - Managing plies
  - Assessing producibility
  - Creating flat patterns
  - Generating properties for analysis
  - Producing design documentation
  - Generating offset laminate surfaces
  - Exporting flat patterns to manufacturing
  - Programming laser projection systems
  - Interfacing with fiber placement machines



FiberSIM Interface

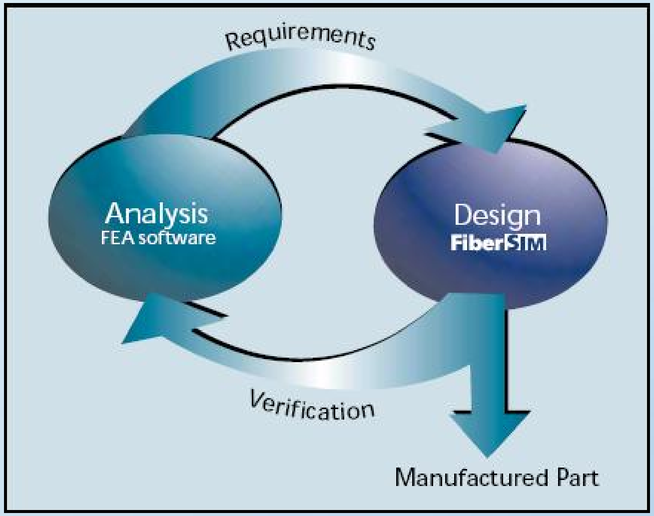
- The FiberSIM Analysis interface provides a complete and detailed description of the final part design, enabling accurate verification of the part's performance
- It has geometry available using .xml file that has the following information available:
  - Names of all layers
  - Which layers are used (not dropped-off)
  - The orientation angle of all layers



© 2005 ANSYS, Inc.

ANSYS, Inc. Proprietary

XML File Format




```xml
<?xml version="1.0"?>
<Laminate_Layup PART_NAME="TEST1">
<Orientation XYZ="0.0 0.0 0.0"/>
<Ply NAME="STEEL">
<Mesh_Point_Table COUNT="4"
RECORD="XYZ">
0.000000 0.000000 0.000000<R/>
30.000000 0.000000 0.000000<R/>
0.000000 30.000000 0.000000<R/>
3000000 30.000000 0.000000<R/>
</Mesh_Point_Table>
<Mesh_Polygon_Table COUNT="2"
RECORD="NODES,WARP,WEFT">
1 2 3,0.200000,1.745329<R/>
4 2 3,0.200000,1.745329<R/>
</Mesh_Polygon_Table>
</Ply>
... more plys
</Laminate_Layup>
```

© 2005 ANSYS, Inc. ANSYS, Inc. Proprietary

XML File

- The XML file has the following information available:
  - The names of all layers
  - Which layers are used (not dropped-off) at any one point
  - The orientation angle of all layers used at any one point

© 2005 ANSYS, Inc.

FibreSim Interface … contd.

- The XML file does not describe the laminate completely. Also needed are:
  - Layer thicknesses
  - Layer material properties
- These are provided using the SECDATA and MP commands in ANSYS.

```
sect,4,shell,fibersim, lamin1
secd,0.2,1,,,layer1
secd,1.0,2,,,layer2
```

- SHELL99 and SHELL181 are supported

© 2005 ANSYS, Inc.

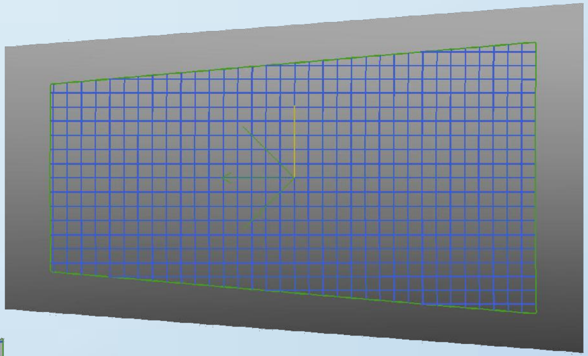
ANSYS, Inc. Proprietary

Other Input information

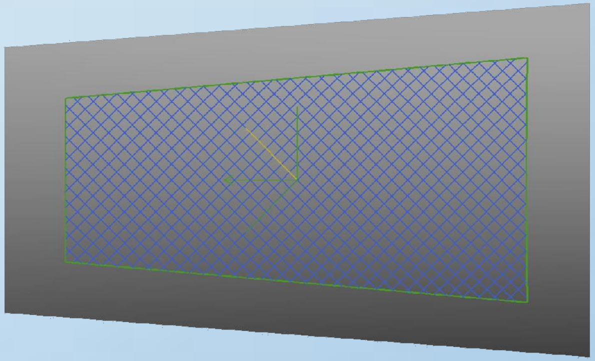
- Additionally, the following input are available:
  - Offset selection
  - Added mass per unit area
  - Distance and angle tolerances between the FS model and the ANSYS model
  - Debug output keys

© 2005 ANSYS, Inc.

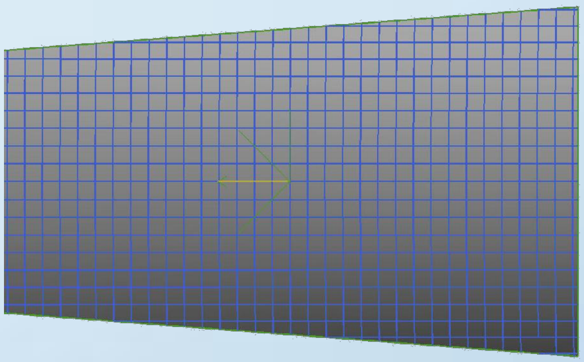
Fibersim Multilayer Input



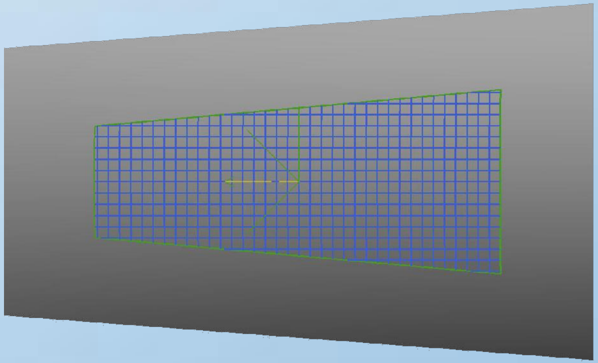
P002



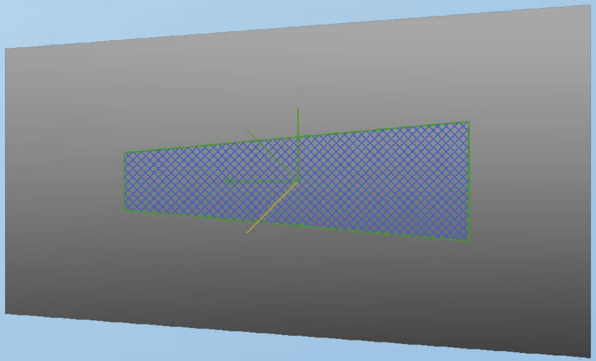
P003



P001, P006



P004

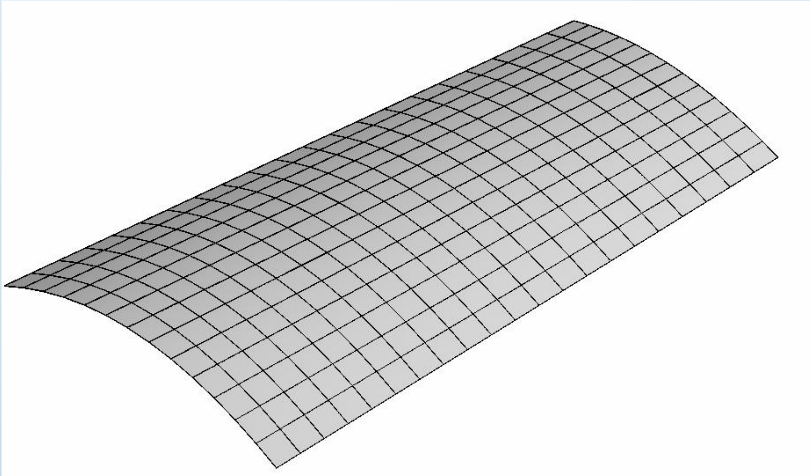


P005

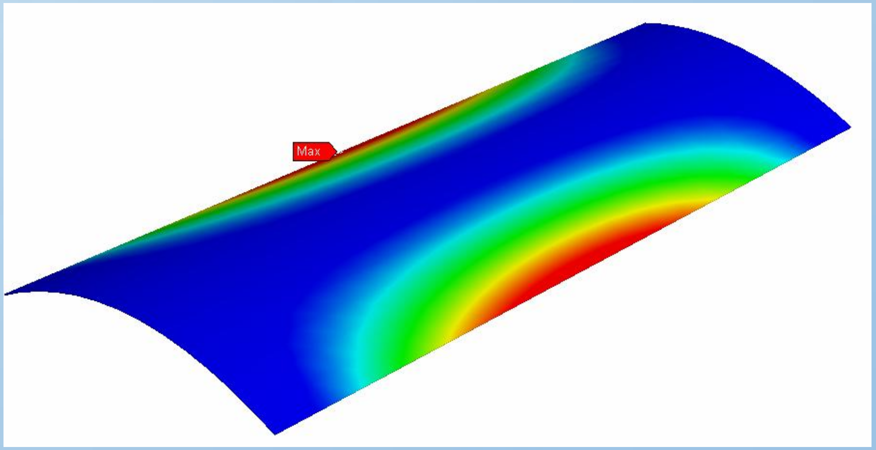
© 2005 ANSYS, Inc.
ANSYS, Inc. Proprietary

ANSYS-Fibersim Interface Example

```
t1 = 1.0e-3
!
! ************ Import FiberSIM data *******
!
sect,1,shell,fibersim,xml_filename
!
secd,t1,1,,,P001
secd,t1,1,,,P002
secd,t1,1,,,P003
secd,t1,1,,,P004
secd,t1,1,,,P005
secd,t1,1,,,P006
!
secr,xml_filename,xml
secn,1
!
thicktol = t1*50
edgetol = 100
angtol = 180
secc,,,,,,,,thicktol,edgetol,angtol,0
```



ANSYS Shell Mesh



Displacement Results

© 2005 ANSYS, Inc.

ANSYS, Inc. Proprietary

Conclusion

- ANSYS offers a range of capability for composites analysis
- Traditional composites capability includes shell, solid and beam elements with advanced section definition and failure criteria specification capability
- Advanced delamination modeling capability through cohesive zone elements extends the composites capability include failure modeling
- Interface to Fibersim brings composites analysis in ANSYS closer to CAD design and manufacturing by providing exact fiber orientations for use in generating laminate properties

© 2005 ANSYS, Inc.

ANSYS, Inc. Proprietary