Composite Structural Analysis and Design Issues

ME 7502 – Lecture 13

Dr. B.J. Sullivan

Finite Element Analysis of Composite Structures

- Lamina stress analysis in FEA of composites
- Considerations in selection of element types
- Modeling individual layers with orthotropic elements
- FEA model construction
- Boundary conditions in FEA of composite plates and shells
- Thermal and Thermo-Structural Analysis Methods
- Assessment of Calculated Stresses and Strains
- Determination of Allowable Stresses and Strains
- Methodology for Fastener Location and Quantity in Bolted Structures
- Calculation of Margins and Safety

Lamina stress analysis in FEA of composites

It is not sufficient that a finite element code contain elements with anisotropic properties.

While this capability will allow the structural analysis to be performed, stresses will be available at discrete points (centroid, quadrature, points, nodes) only.

This will not be sufficient for investigating lamina failure.

To do lamina stress analysis, either the finite element code or a user-supplied post-processor must have the following capabilities:

Lamina stress analysis in FEA of composites

a) convert generalized nodal displacements (u_x, u_y, θ_x, θ_y, θ_z) into mid-plane strains $\{\varepsilon_x°\}$ and plate curvatures $\{K_x\}$ at, say, the centroids of each element

b) compute stresses at the appropriate through the thickness coordinate z_i corresponding to each ply I

$$\{\sigma_x^i\} = [\bar{Q}^i]\{\varepsilon_x^\circ + z^i K_x\}$$

c) Transform the laminate coordinate stresses to the material coordinate system stresses

$$\{\sigma_\ell^i\} = [\theta]^T\{\sigma_x^i\}$$

d) Use all of the stresses in an acceptable failure criterion to make judgements on the structural integrity of the laminate

Lamina stress analysis in FEA of composites

Some public domain FEA codes have at least the capabilities a), b) and c) above;

Capability d) can be user-dependent; i.e., the user may wish to be very specific about how lamina level stresses are combined within a failure criterion to make judgments regarding failure

Lamina stress analysis in FEA of composites

ANSYS, ABAQUS, NATRAN public domain FEA codes all have laminated composite plate and shell elements

Required input includes:

a. Lamina (ply) properties in local material directions
b. Orientation of ply relative to laminate (global) coordinate direction
c. Lamina (ply) thickness
d. Lamina failure algorithm (e.g., Hashin, Puck, etc.) and associated parameters

Output features:

a. Stresses in each ply in local material axes
b. Stress contour plots within plies across continuous elements
c. Margin of safety contour plots within plies across continuous elements, based on failure criterion and its parameters

Considerations in Selection of Element Types

- Three basic questions in element selection:
  - Should elements be represented by plate elements or solid elements?
    - Plate elements do a good job of capturing bending behavior and will require far fewer elements to simulate response
    - Solid elements provide a much better assessment of the interlaminar stresses
  - If plate elements are selected, should homogenous orthotropic properties be used, or should individual ply properties be used?
    - Homogenous orthotropic properties require single plate/shell elements through the thickness
    - Use of individual ply properties in FE analysis requires use of layered plate/shell elements through the thickness
  - Should elements with transverse shear deformation be employed?

Considerations in selection of element types

Laminate approximated by homogeneous orthotropic plate properties (average properties throughout)

- DUE TO STACKING SEQUENCE VARIATIONS IN LAMINATE WITH SET FIBER ORIENTATIONS, [D] MATRIX VARIES, BUT [A] MATRIX DOES NOT:

EXAMPLE:

Gr/Ep	$[0/\pm45]_S$	$[45/0/-45]_S$
D_{11}, MPa	5088	3133
D_{22} MPa	846	1570
D_{66} MPa	1325	2557

- THEREFORE, E^*_X, E^*_Y, ν^*_{XY} BACKED OUT USING [D] MATRIX WILL NOT AGREE WITH E's AND ν's BACKED OUT OF [A] MATRIX

$$D_{11} = \frac{h^3}{12}\frac{E_X^*}{\left(1-\nu_{XY}^*\nu_{YX}^*\right)},\ D_{22} = \frac{h^3}{12}\frac{E_Y^*}{\left(1-\nu_{XY}^*\nu_{YX}^*\right)},\cdots \Rightarrow \overline{E}_D \qquad \overline{E}_D \neq \overline{E}_A$$

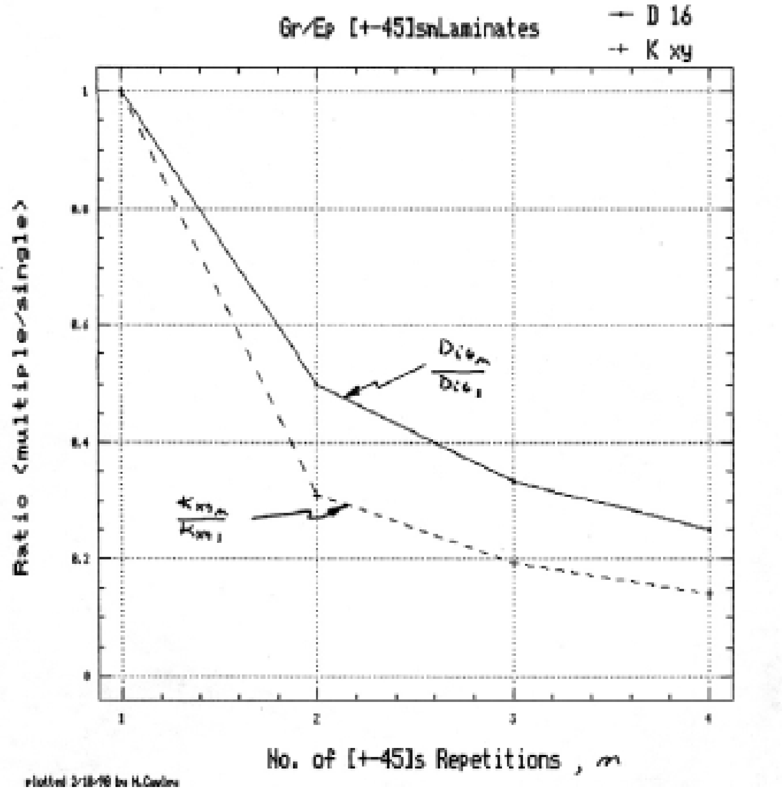
$$A_{11} = \frac{hE_X^*}{1-\nu_{XY}^*\nu_{YX}^*},\ A_{22} = \frac{hE_Y^*}{1-\nu_{XY}^*\nu_{YX}^*},\cdots \Rightarrow \overline{E}_A$$

USE OF ONE SET OF HOMOGENEOUS ORTHOTROPIC ELASTIC CONSTANTS WILL GIVE INCORRECT [A] OR [D] MATRICES.

Considerations in selection of element types

Laminate approximated by homogeneous orthotropic plate properties (average properties throughout)

- SYMMETRIC LAMINATES WITH ANGLE PLIES WILL HAVE BENDING/TWISTING COUPLING (D_{16}, D_{26}, …) WHICH WILL NOT BE NEGLIGIBLE UNLESS HAVE MANY REPEATED SEQUENCES OF SAME SUBLAMINATE, E.G., $[-45/0/45/]_{NS}$, N~10+



THEREFORE, USE HOMOGENEOUS ORTHOTROPIC PLATE <u>ONLY</u> IF MANY LAYERS, MANY STACKING SEQUENCE REPETITIONS, E.G., $[-45/0/45/]_{NS}$, N~10+

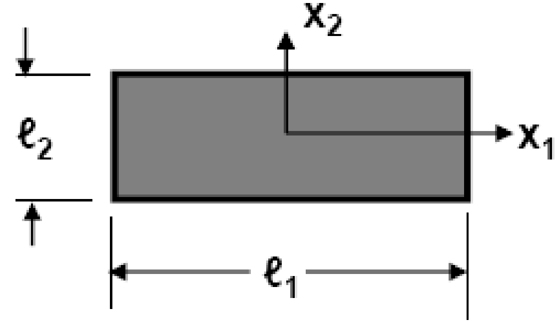
Modeling Individual Layers with Orthotropic Elements

1. STABILITY OF SOLUTION

- FINITE ELEMENT CODES SOLVE THE EQUATION $\{F\} = [K]\{x\}$

  {F} = APPLIED NODAL FORCE MATRIX
  [K] = GLOBAL STRUCTURAL STIFFNESS MATRIX
  {x} = NODAL DISPLACEMENTS

- *[K]* IS CONSTRUCTED FROM ELEMENT STIFFNESSES, *k*
- IF [K] IS NOT WELL-BEHAVED, SOLUTION MAY NOT CONVERGE
- WANT, FOR GOOD CONVERGENCE, $1/10 \le k_{11}/k_{22} \le 10$



$$k_{11} = \frac{\ell_2}{\ell_1} Q_{11}, \quad k_{22} = \frac{\ell_1}{\ell_2} Q_{22} \qquad \frac{k_{11}}{k_{22}} = \left(\frac{\ell_2}{\ell_1}\right)^2 \frac{Q_{11}}{Q_{22}}$$

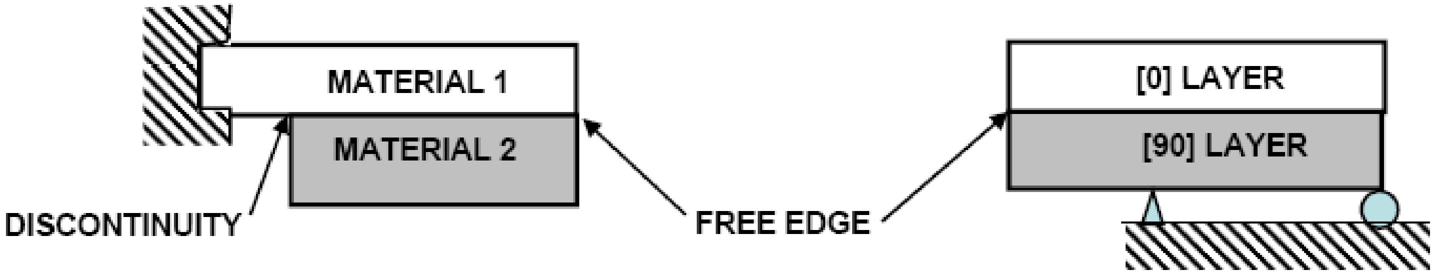
ISOTROPIC MATERIAL: $Q_{11} = Q_{22}$, $1/3 \le \ell_2/\ell_1 \le 3$
OPTIMAL $\ell_2/\ell_1 = 1$

Carbon/Epoxy: Q_{11} = 180 GPa, Q_{22} = 10 GPa , $1/10 \le \ell_2/\ell_1 \le 1$
OPTIMAL $\ell_2/\ell_1 = 1/4$

- ALWAYS DO A MESH SIZE CONVERGENCE STUDY - BEST CONVERGENCE OBTAINED USING 8-NODE QUADS (2-D) OR 20-NODE BRICKS (3-D)

Modeling Individual Layers with Orthotropic Elements

- STRESSES ARE THEORETICALLY INFINITE AT FREE EDGES OR DISCONTINUOUS GEOMETRIES WITH BIMATERIAL INTERFACES:



- STRESSES WILL *INCREASE* WITH *DECREASING* ELEMENT SIZE

- SAME IS TRUE AT DISPLACEMENT CONSTRAINT BOUNDARY CONDITIONS (EQUIVALENT TO INFINITE STIFFNESS MATERIAL!)

NEVER BELIEVE FE STRESS RESULTS AT EDGES OR DISCONTINUITIES BETWEEN MATERIALS OR AT BOUNDARY CONSTRAINTS!!

- NODAL STRESS SOLUTIONS IN FE CODES INTERPOLATE FROM GAUSS (INTEGRATION) POINTS OVER ALL ELEMENTS AT THAT NODE.

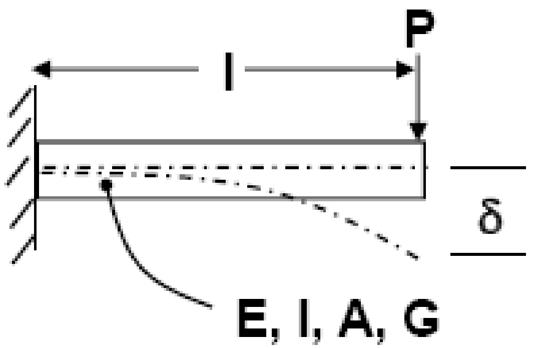
- INTERFACE STRESSES *WILL* BE DISCONTINUOUS

ALWAYS SELECT ELEMENTS WITH SAME MATERIAL PROPS (DESELECT OTHERS) WHEN EVALUATING STRESSES AT INTERFACE!!

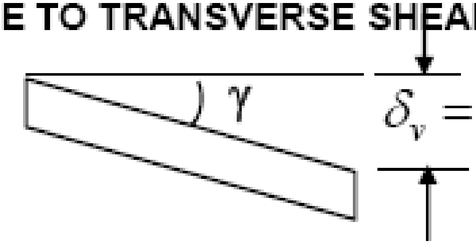
FEA Modeling of Laminated Plates

ISSUES WITH TRANSVERSE SHEAR DEFORMATIONS:

TIP DEFLECTION OF CANTILEVERED PLATE:



DUE TO TRANSVERSE SHEAR ONLY:



$$\delta_v = \gamma l\,, \quad \gamma \approx \frac{\tau}{G} \approx \frac{P}{AG}$$

BENDING

TRANS. SHEAR

$$\delta = \frac{Pl^3\left(1-\nu^2\right)}{3EI} + \frac{Pl}{GA}\,, \quad \therefore \delta = \frac{Pl^3\left(1-\nu^2\right)}{3EI}\left(1 + \underbrace{\frac{1}{4}\frac{E}{\left(1-\nu^2\right)G}\frac{h^2}{l^2}}_{}\right)$$

RATIO OF
TRANS. SHEAR
TO BENDING

FEA Modeling of Laminated Plates

RATIO OF TRANSVERSE SHEAR TO BENDING DEFLECTIONS

$$R_{TS/B} = \frac{1}{2(1-\nu)^2}\frac{h^2}{l^2} \text{ (ISOTROPIC)}, \quad \frac{1}{2(1-\nu_{XY}\nu_{YX})^2}\frac{h^2}{l^2} \text{ (ORTHOTROPIC)}$$

FOR ISOTROPIC MATERIAL:

$$E = 2G(1+\nu), \quad \nu \approx 0.3, \quad R_{TS/B} \approx \frac{h^2}{0.7l^2}$$

$$\text{FOR } R_{TS/B} \prec 1\%, \ \frac{h}{l} \prec \frac{1}{8} \quad ; \quad \text{FOR } R_{TS/B} \prec 5\%, \ \frac{h}{l} \prec \frac{1}{5}$$

ISO: CAN USE PLATE MODEL W/O TRANS SHEAR DEFS

FOR ORTHOTROPIC MATERIAL: EX: Gr/Ep $[0_2/x/x]_{ns}$

$$E_x \approx 120 \text{ GPa}, \ G_{xz} \approx 7 \text{ GPa}, \ \nu_{XZ} \approx 0.4, \quad \nu_{ZX} \approx 0.1 \quad R_{TS/B} \approx 5\frac{h^2}{l^2}$$

$$\text{FOR } R_{TS/B} \prec 1\%, \ \frac{h}{l} \prec \frac{1}{25} \quad ; \quad \text{FOR } R_{TS/B} \prec 5\%, \ \frac{h}{l} \prec \frac{1}{10}$$

ORTHO: TRANS SHEAR DEFS MUCH LARGER THAN ISO

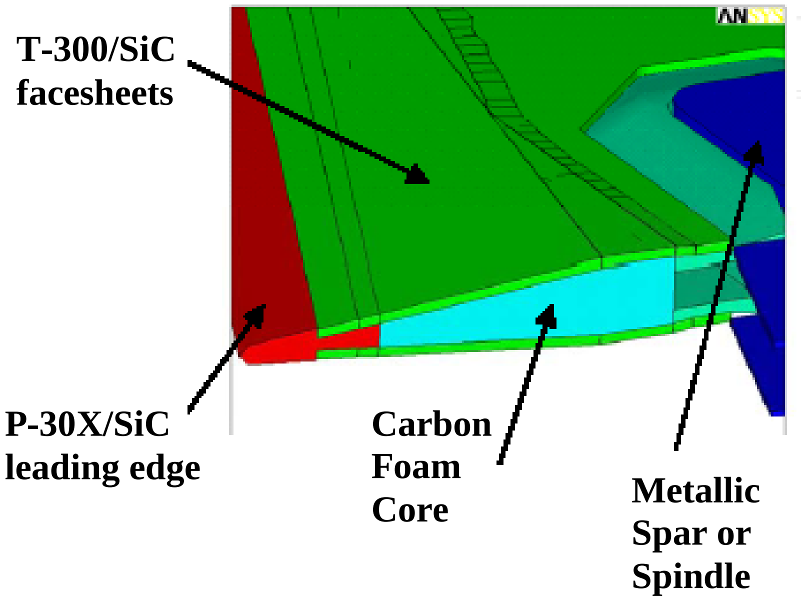
SINCE STRESS ERROR NORMALLY >> (3x – 5x) DEFLECTION ERROR, USE LAMINATED PLATE ELEMENT WITH TRANSVERSE SHEAR DEFORMATIONS!!

Use of Elements with Transverse Shear Deformation

- Most general purpose FEA codes provide plate and shell elements with transverse shear capability
- As we have seen, this can be an important aspect of composite structural analysis in two cases:
  - The ratio of in-plane Young’s modulus to through thickness shear modulus is relatively large (in some cases as low as 5; for metals, E/G < 3 is typically)
  - The span-to-thickness ratio is small (~25 or less)
- If you are not sure if shear deformation is important, try to perform identical analyses with and without this effect
  - Near identical results will indicate shear deformation is not important
  - Different results will indicate the importance of this feature in your analysis

FEA Model Construction

- Typical FEA model construction practice is not substantially different from metallic parts
  - Shell / flat plate elements for thin gauge members, e.g. facesheets
  - 3D solid elements for thicker parts, e.g. solid leading edge materials and core material of sandwich structure components
  - 3D solids or 3D beam elements for longerons and ribs

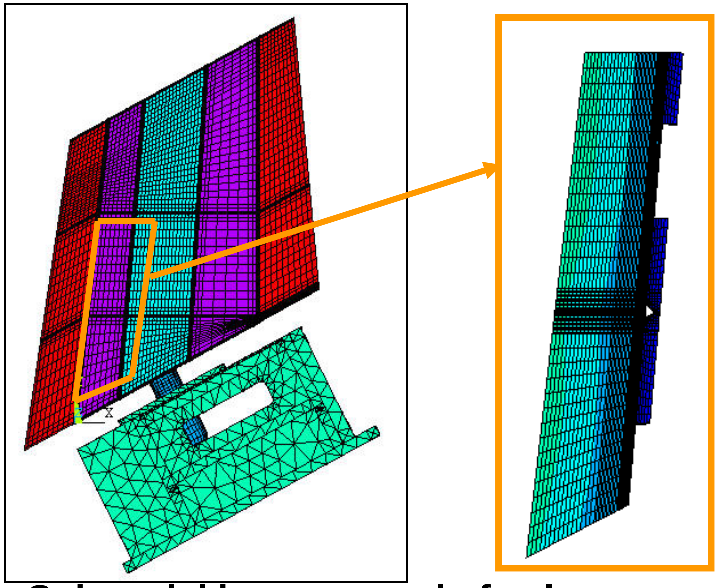


FEA Model Construction

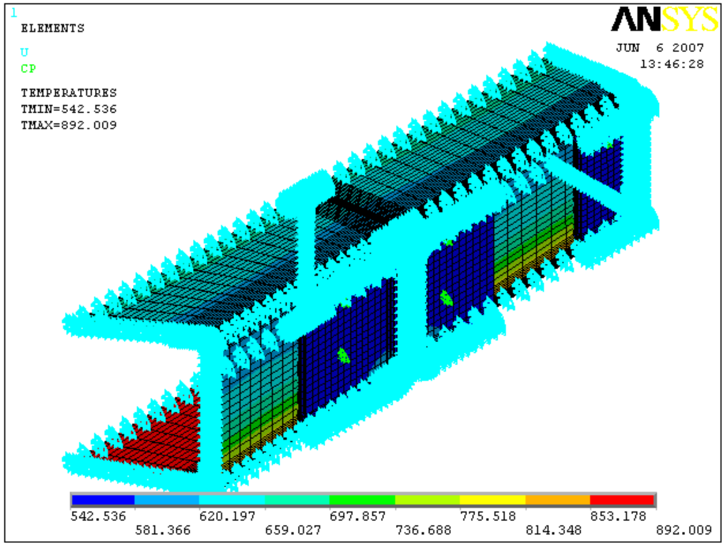
- One area of difference between FEA analysis of metallic parts and FEA analysis of composite parts is that more submodels are needed to accurately assess responses of composite parts, since interlaminar strengths of these materials are typically low
  - This is particularly true for refractory (e.g., Carbon-Carbon and Ceramic Matrix Composites)
- Submodeling effort is accomplished by creating a smaller model of the components where overstress in the full scale model is calculated
  - More elements through the thickness and better aspect ratio elements used

Example of FEA Submodel

- Cut Boundary Interpolation is performed by taking the resulting displacements from the full scale model that occur at the cut boundary of the sub model and applying them to the sub model (shown with bright blue arrows in picture on right)
- Submodel temperatures are applied using a Body Force interpolation, where temperatures are taken from the full scale model and applied to the submodel



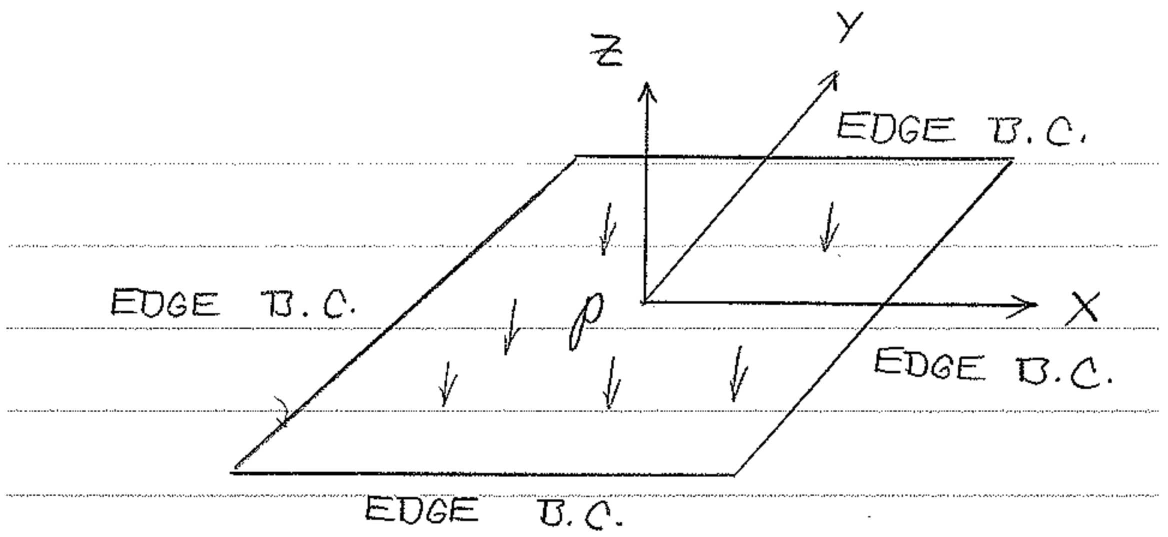
Submodel is composed of volumes outlined in orange.



Submodel shown with applied temperatures and displacements from full scale model

Boundary Conditions in FEA of Composites

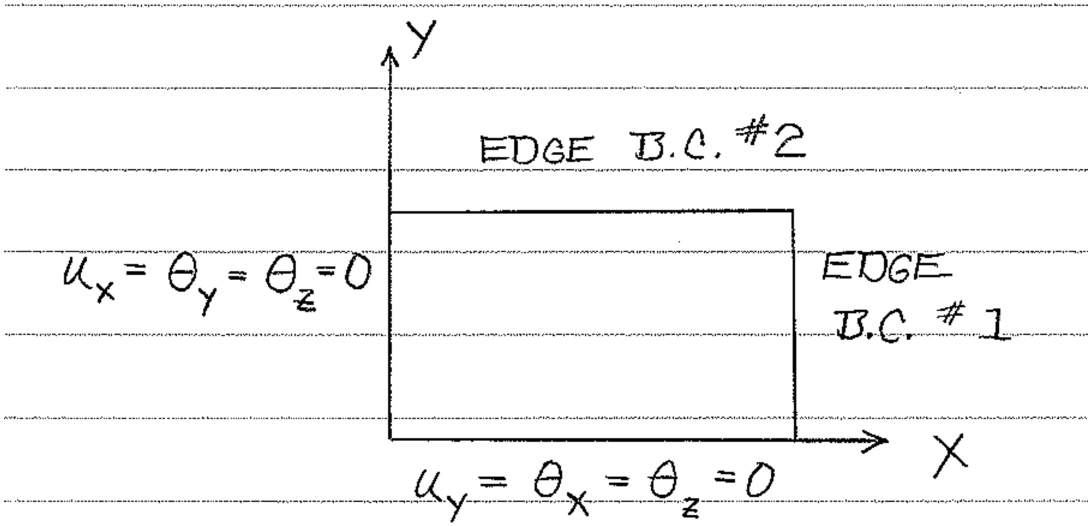
- For isotropic plates and shells, we frequently use symmetry B.C.'s to avoid having to analyze the entire body
  - For example, a plate under a load symmetric about one or two axes parallel to the edges can frequently be analyzed with a reduced model by employing symmetry B.C.'s:



Here, the pressure load p is symmetric about both X and Y axes

Boundary Conditions in FEA of Composites

- For loads and edge B.C.'s symmetric about both X and Y axes, an isotropic plate can be analyzed with a quarter segment and the following B.C.'s:

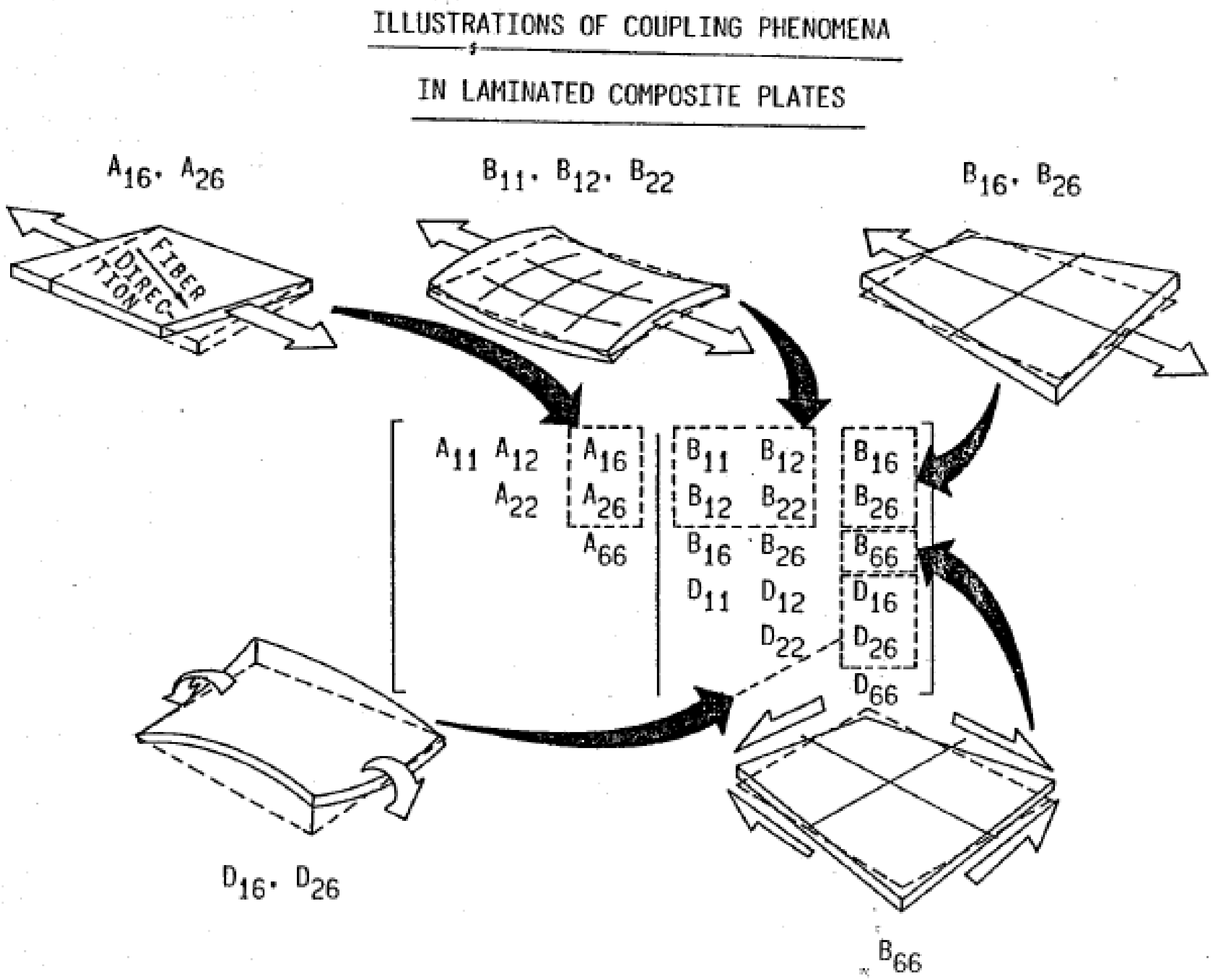


where Edge #1 could be simply-supported ($u_Z = \theta_X = \theta_Z = 0$) or clamped ($u_Z = \theta_X = \theta_Y = \theta_Z = 0$) and Edge #2 could be simply-supported ($u_Z = \theta_Y = \theta_Z = 0$) or clamped ($u_Z = \theta_X = \theta_Y = \theta_Z = 0$)

Boundary Conditions in FEA of Composites

- For composite plates, the elements of the A, B and D matrices must be examined before deciding if symmetry B.C.'s such as those used above can be employed to make the model smaller
  - For example, even if the load and edge B.C.'s are symmetric about the X and Y axes, if the laminate has shear-extensional coupling (i.e., if A_{16} and A_{26} are not zero), then symmetry B.C.'s are the type shown above cannot be used, since
    - u_Y is not zero across the X-axis cut, and
    - u_X is not zero across the Y-axis cut
  - If the laminate has no shear extensional coupling (i.e., if $A_{16} = A_{26} = 0$) but does have bending twisting coupling (i.e., if D_{16} and D_{26} are not zero), then θ_Z is not zero across either axis cut, so that here again symmetry B.C.'s could not be used
  - In either case, the entire plate would have to be analyzed.

Boundary Conditions in FEA of Composites



Boundary Conditions in FEA of Composites

- The same conclusion can be drawn for plates with bending-extensional coupling (i.e., non-zero B_{ij} coefficients) since any loads causing bending would mean
  - u_Y would not be zero across the X-axis cut, and
  - u_X would not be zero across the Y axis cut
- Suppose we have a balanced and symmetric laminate ($A_{16} = A_{26} = B_{ij} = 0$) making up a plate which has loads and edge boundary conditions symmetric about one or both axes
  - Typically, we will have bending-twisting coupling (i.e., D_{16} and D_{26} will be non-zero), so that, strictly speaking, the model cannot be made smaller by employing symmetry B.C.'s across the X-axis or Y-axis cuts
- Practically speaking, however, if there are many plies and the plies are well dispersed within the laminate, the magnitude of D_{16} and D_{26} relative to the other D_{ij} will be small
  - In these cases we treat the laminate as if it were specially orthotropic and employ symmetry B.C.'s

Boundary Conditions in FEA of Composites

Examples of laminate stack-ups and associated bending-twisting coupling coefficients:

Laminate	D_{11}	D_{22}	D_{12}	D_{16}	D_{26}	D_{66}
$[+60_4/-60_4]_s$	2.60×10^5	1.24×10^6	3.88×10^5	3.29×10^5	9.48×10^5	8.39×10^5
$[+60_2/-60_2]_{2s}$	2.60×10^5	1.24×10^6	3.88×10^5	1.65×10^5	4.74×10^5	8.39×10^5
$[\pm 60]_{4s}$	2.60×10^5	1.24×10^6	3.88×10^5	8.23×10^4	2.37×10^5	8.39×10^5

Ply Thickness 0.0625 in

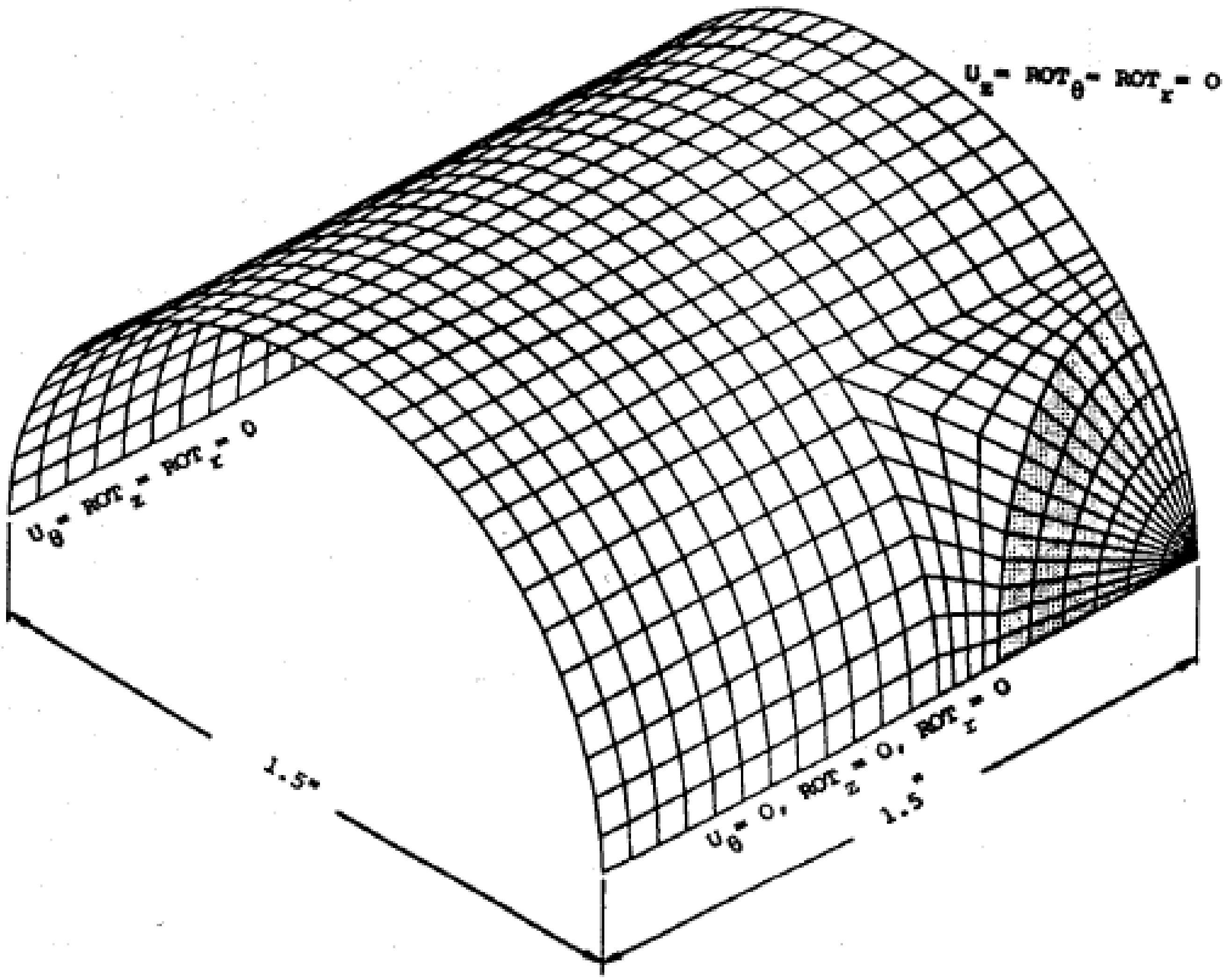
(Lamina properties from Table 1.)

The greater the number of plies and the more dispersed the plies within the laminate, the smaller the D_{16} and D_{26} coefficients relative to the others, the better the representation of the plate as Specially Orthotropic, and the more appropriate the use of symmetry B.C.'s if loads and edge B.C.'s are themselves symmetric

Boundary Conditions in FEA of Composites

- The same concepts apply to the analysis of composite shells
- For example, the half-symmetry model of the cylindrical shell shown on the next page would not be appropriate if
  - The laminate contained shear-extensional coupling, or
  - If loads causing bending were present and there was substantial bending-twisting coupling

Boundary Conditions in FEA of Composites

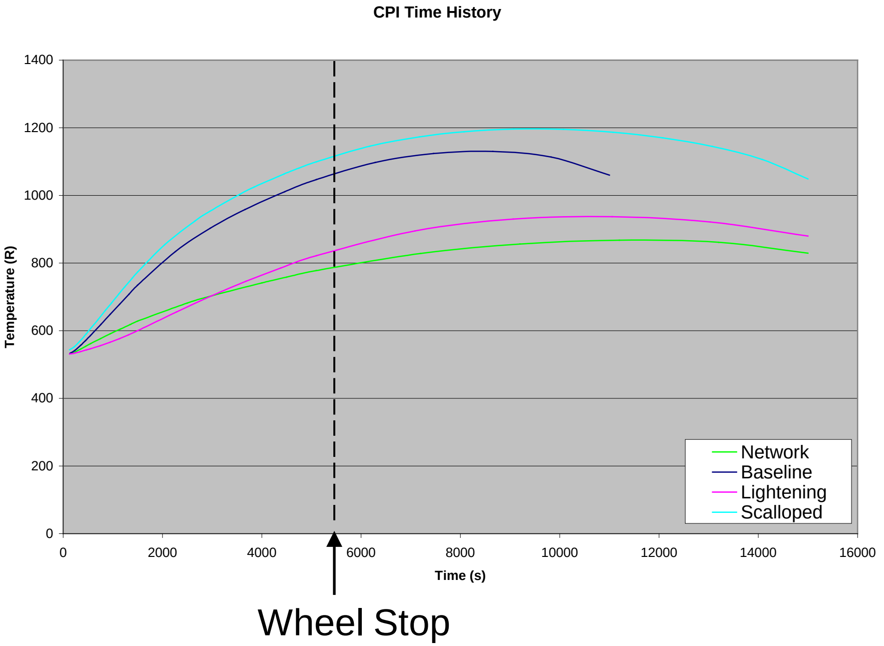


Thermal and Thermo-Structural Analysis Methodology

- In assessment of composite components on vehicles subjected to time-varying thermal loads, both transient heat transfer and thermal stress analyses are performed
- Stress analyses typically require greater discreteness in the FE grid than what is required for thermal analyses
  - Displacement, strain and stress spatial gradients are typically much greater than spatial temperature gradients
- Nevertheless, same FE model is used for both transient heat transfer and thermal stress analyses
  - Elements selected for the FE analyses must be capable of switching from thermal to stress types

Thermal and Thermo-Structural Analysis Methodology

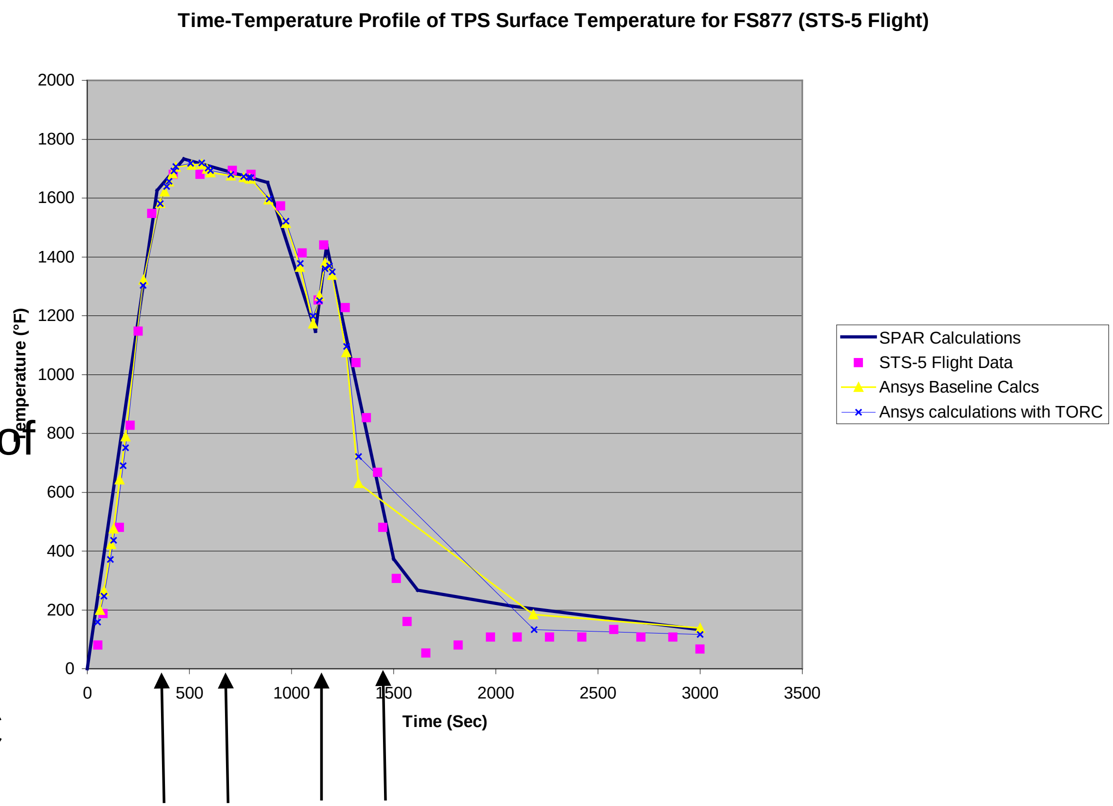
- Transient heat transfer analyses must usually be performed for some period of time beyond the cruise/re-entry period
- Thermal soak must be permitted to occur
  - Highest temperatures in thermal protection system (TPS) components often do not occur until after “wheel stop”



Period of thermal soak may actually be 2-3 times as long as period of aerothermal heating

Thermal and Thermo-Structural Analysis Methodology

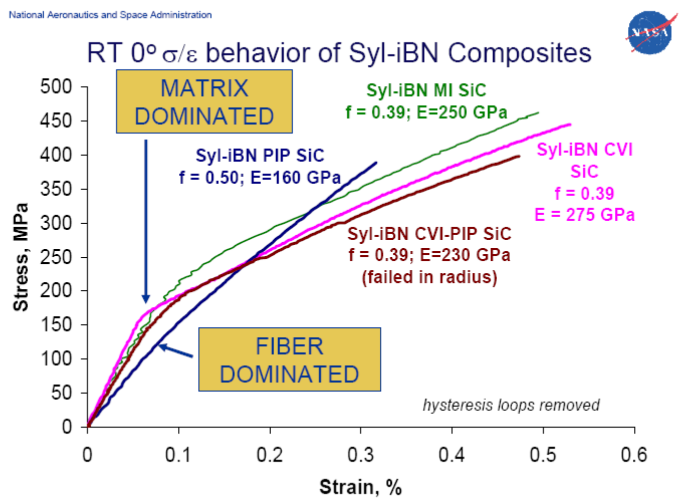
- “Snap shot” thermal stress analyses are performed for a few discrete times of flight
  - Times corresponding to peak thermal gradients
    - These times will lag time of peak gradient(s) in aero-thermal heating
  - Times corresponding to peak temperatures of different materials in CMC components
    - These times will lag time of peak aero-thermal heating



Candidate times for thermal stress analyses

Thermal and Thermo-Structural Analysis Methodology

- Measured stress-strain curves of composite material used in structural components will dictate type of analysis performed
  - Non-linear response requires nonlinear material analysis and strain allowable assessments
  - Linear response permits linear material analysis and allowable stress assessments



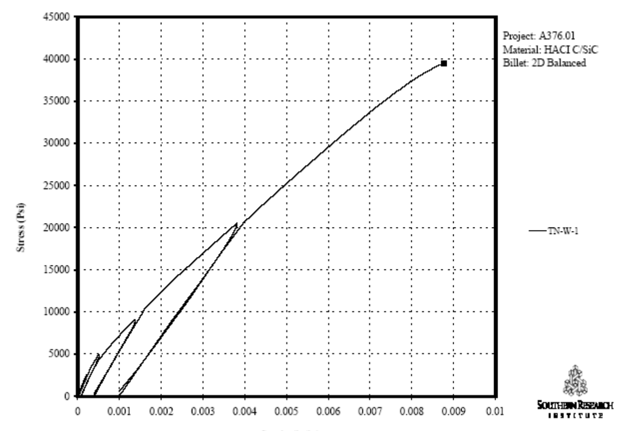
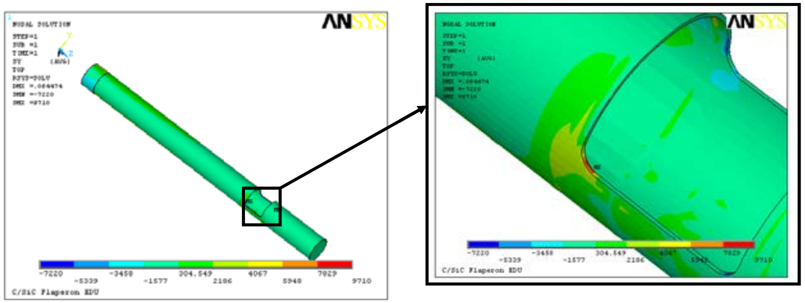


Figure 2.2.1-4 Warp Room Temperature Tensile Stress-Strain Response of HACI C/SiC 2D Balanced

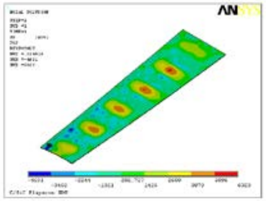
Assessment of Calculated Stresses and/or Strains

- Peak stresses or strains appearing on FEA contour plots are not appropriate for realistic assessment of component performance
  - When failure is initiated within a composite test specimen, the initial failure occurs over a region containing several (3-5 at a minimum) textile unit cells
  - A textile unit cell is defined as the "smallest volume of the fiber reinforcement containing all unique fiber orientations in the preform" (i.e., longitudinal, lateral, and through thickness)

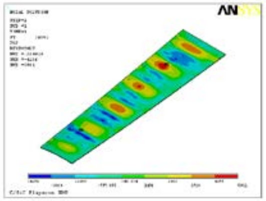


In-plane, hoop stress: 9.7ksi
MOS: 1.60

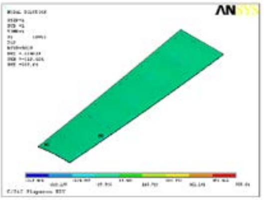
In-plane stress in torque tube of flaperon EDU resulting from mechanical loading



In Plane Stress
σ_x (6.3ksi)



In Plane Stress
σ_y (6.0ksi)



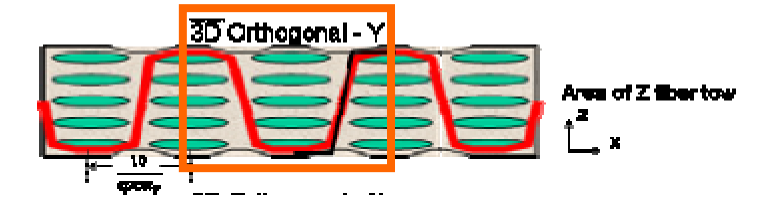
ILT and ILS are localized stresses
σ_z (6.0ksi)

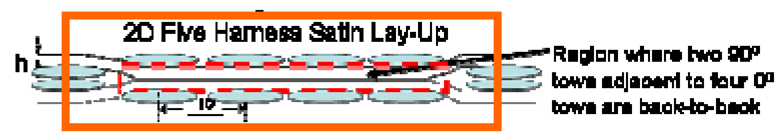
Stress Contours in access panel of flaperon EDU resulting from mechanical loading

Assessment of Calculated Stresses and/or Strains

- Accordingly, in the comparison of FEA calculated strains or stresses, an average of any given strain or stress component over a volume of at least three (3) unit cells should be compared to the measured failure strain or stress of the material

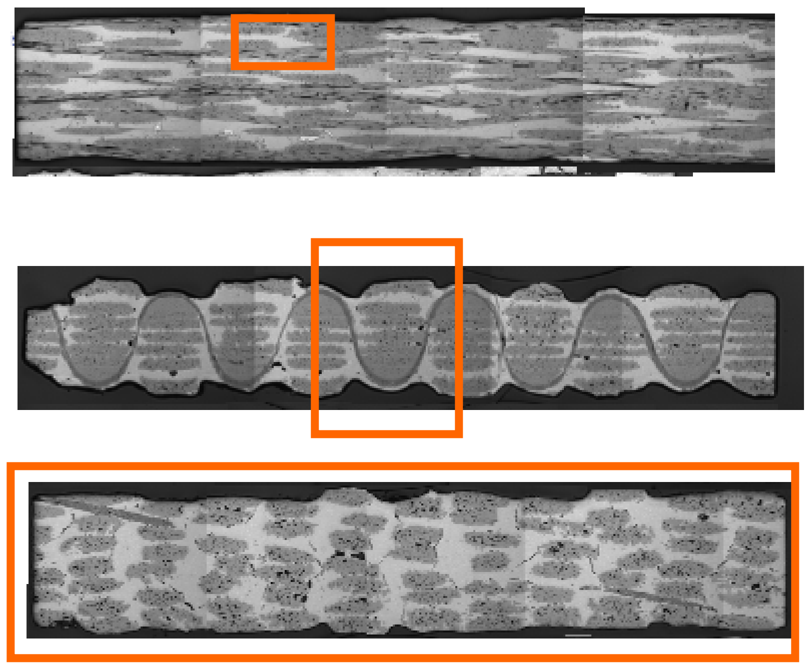
- This approach definitely makes the structural analysis more time consuming





One textile unit cell

Assessment of Calculated Stresses and/or Strains



Photomicrographs of Composites and Definition of Unit Textile Cells

One textile unit cell

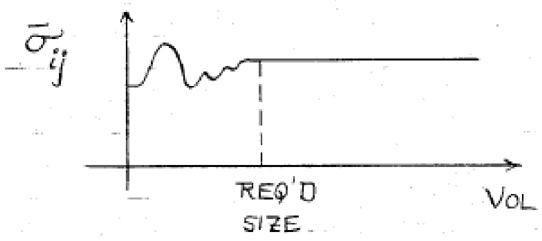
Assessment of Calculated Stresses and/or Strains

- What is the size of the unit cell for a specific balanced fabric reinforced material?
  - Suppose fabric reinforcement uses 3K T-300 yarns spaced 23 ends per inch (epi) for the warp yarns x 24 epi for the fill yarns
  - In addition, one ply is ~ 0.015 inch in thickness
  - One unit cell is therefore (1/23)” x (1/24)” x ~0.015” = 0.042” x 0.043” x 0.015”
  - The volume of three units cells is therefore ~ 0.13” x 0.13” x 0.05”
  - If a stress component, averaged over this volume, exceeds a measured strength, then failure is predicted
- What is the size of the unit cell for a specific unbalanced (e.g. 4:1) fabric reinforced material?
  - Supposed the fabric reinforcement uses 2K P-30X yarns spaced 20 epi warp x 5 epi fill; also, one ply is ~ 0.0125 inch in thickness
  - Volume of 3 unit cells is therefore 0.15” x 0.6” x 0.038”
  - If a stress component, averaged over this volume, exceeds a measured strength, then failure is predicted
- <u>Note</u>: if point stress component exceeds measured strength, but the volume averaged stress does not exceed strength, then no failure

Assessment of Calculated Stresses and/or Strains

- Why is this methodology acceptable? Isn't this a "non-conservative" approach to stress assessment?
- Need to remember two items:
  - First, the material properties being used in the analysis itself are not actually valid at a small point
    - The material properties, which relate average composite stresses to average composite strains, are valid only over a representative volume element. So too, therefore, are the calculated stresses.
  - Secondly, when failure is initiated within a composite test specimen, the initial failure occurs over a region containing several (3-5 at a minimum) unit cells
    - This, then, is the volume over which stresses should be averaged and compared to measured strengths

RVE must be large enough so that average stress in RVE is unchanged as size increases:



Generation of design properties through material property testing

How are the thermo-elastic properties used in the material models of the composite FE analyses determined?

How are the composite strength or strain-to-failure values measured?

How are the measured strengths used to define allowable values for the comparison with calculated stresses/strains?

Design Properties

- Most basic element of design properties database is material thermo-elastic properties themselves
  - Young's moduli (tension and compression)
  - Axial shear modulus
  - Poisson's ratios
  - Coefficients of thermal expansion
  - Thermal conductivities

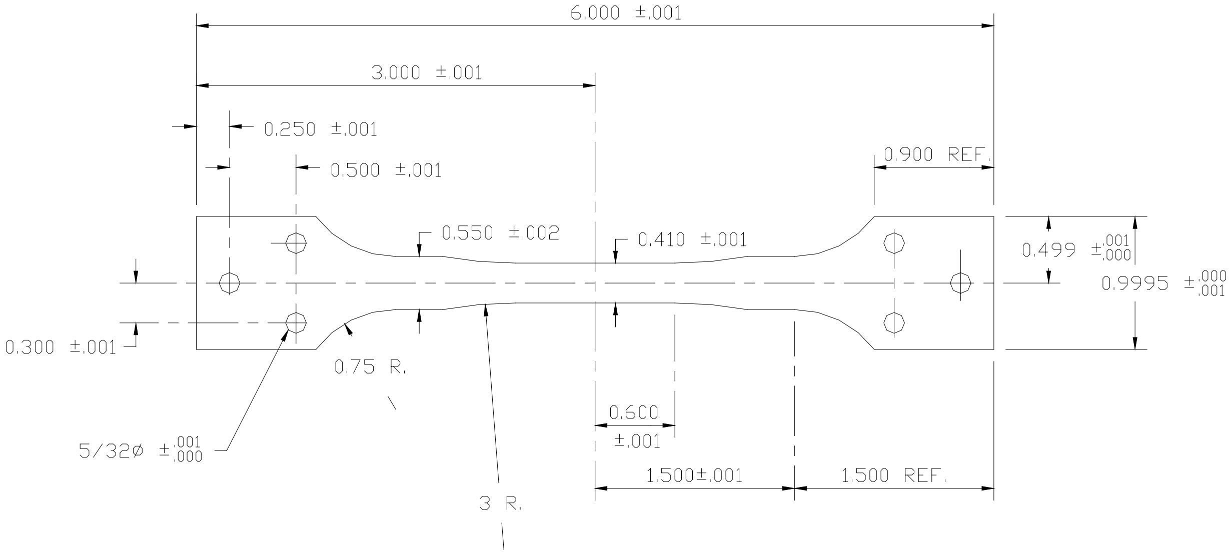
- Thermo-elastic moduli and thermal conductivities are typically temperature-dependent

Design Properties

- For full three-dimensional orthotropic materials, all properties (e.g., through thickness Young's and shear moduli) cannot be measured
- Micromechanics models are correlated using measurable data and then used to predict properties that cannot be measured
- Temperature-dependent strengths are frequently measured at same time as moduli
  - Axial tensile and compressive
  - In-plane shear strengths
- Through thickness strengths (tensile, compressive, shear) are measured alone

Tension Specimen

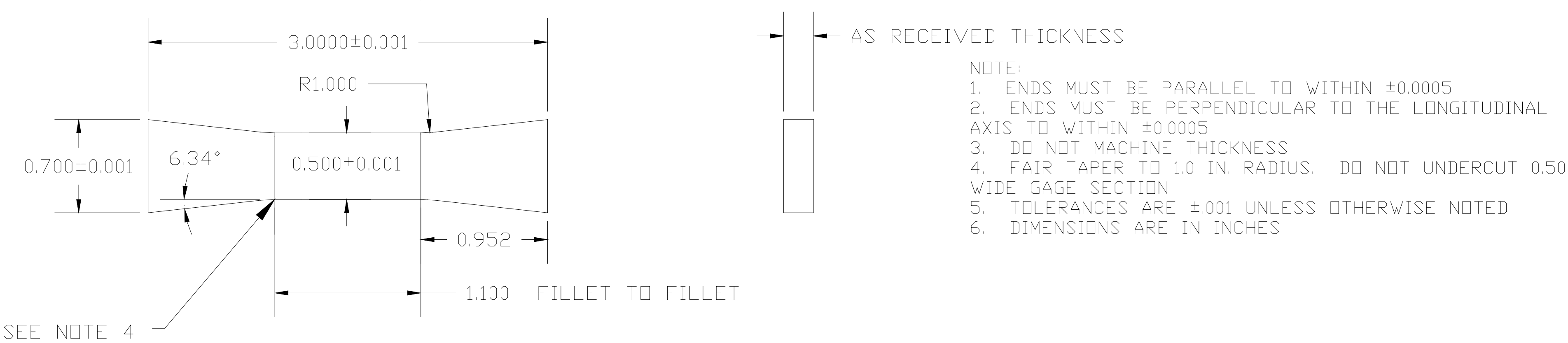
This specimen is used to measure the composite in-plane axial tensile modulus and strength



NOTES:
1. ALL DIMENSIONS ARE IN INCHES
2. TOLERANCES ARE ±.005 ON LENGTHS, ±.001 ON ALL OTHER DIMENSIONS UNLESS OTHERWISE NOTED
3. DO NOT UNDERCUT RADII AT TANGENT POINTS
4. DO NOT MACHINE THICKNESS IN THE GAGE SECTION
5. WIDTH OF GAGE SECTION CAN BE VARIED UP TO ~0.500 TO ACCOMODATE LAY-UPS SUCH AS [+/-45] & QUASI-ISOTROPIC LAY-UPS
6. SPECIMEN SHOULD NOT BE USED FOR 2DCC HYBRID LAY-UPS WITH HIGH AXIAL FIBER VOLUME FRACTIONS

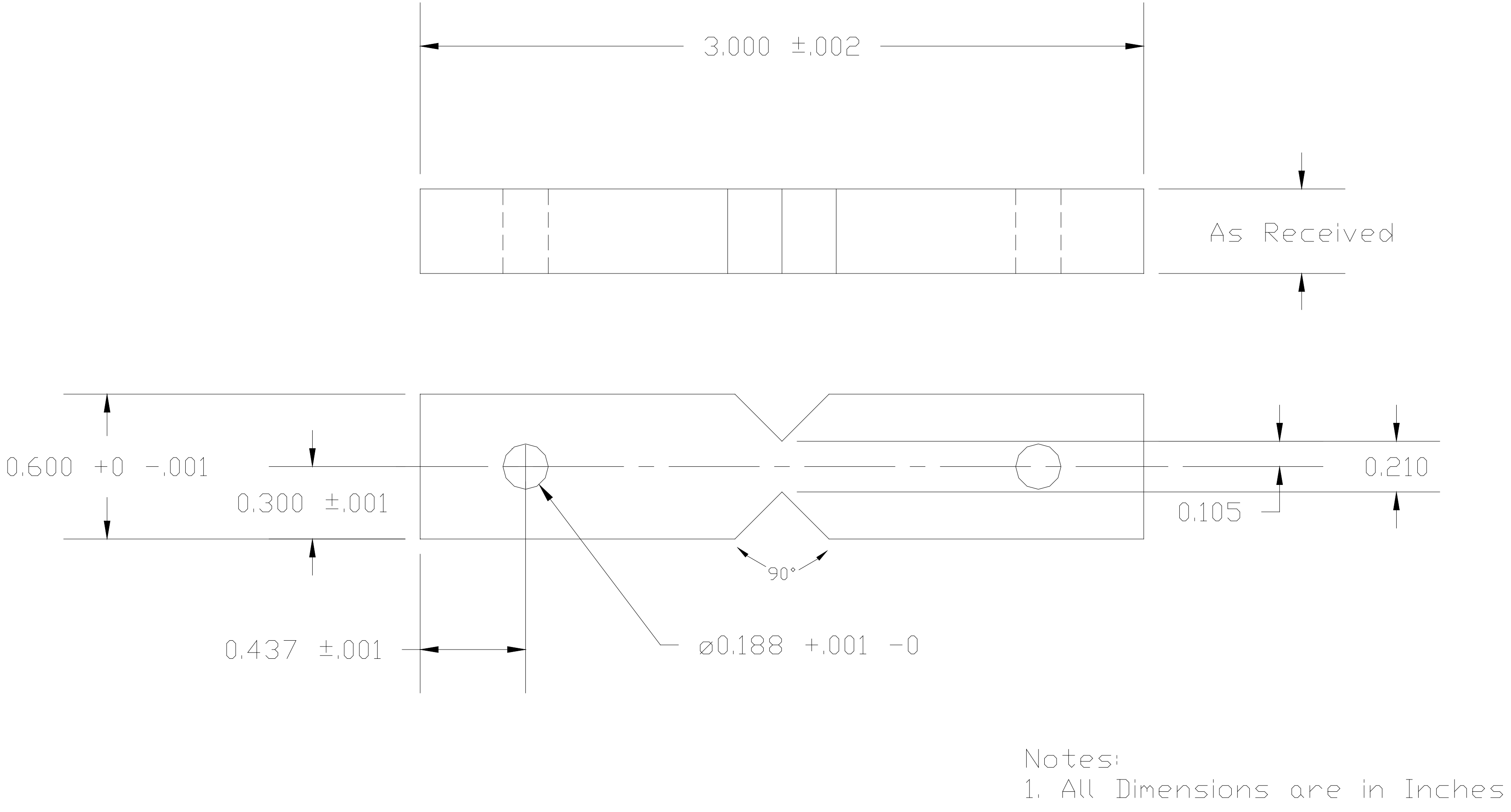
Compression Specimen

This specimen is used to measure the composite in-plane axial compressive modulus and strength

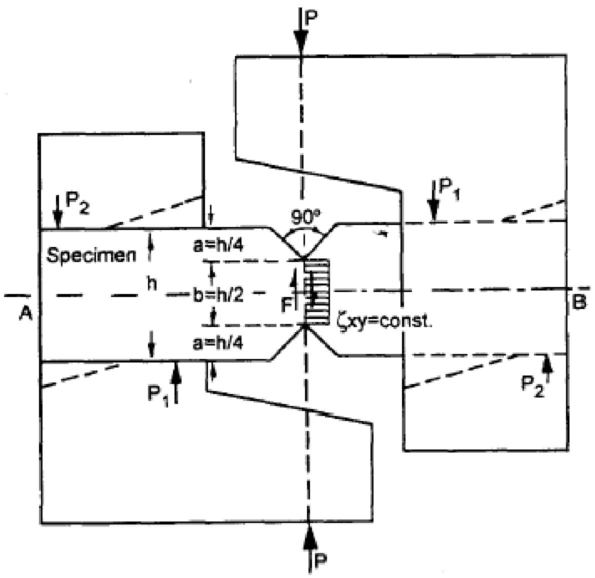


Rumanian Shear Specimen

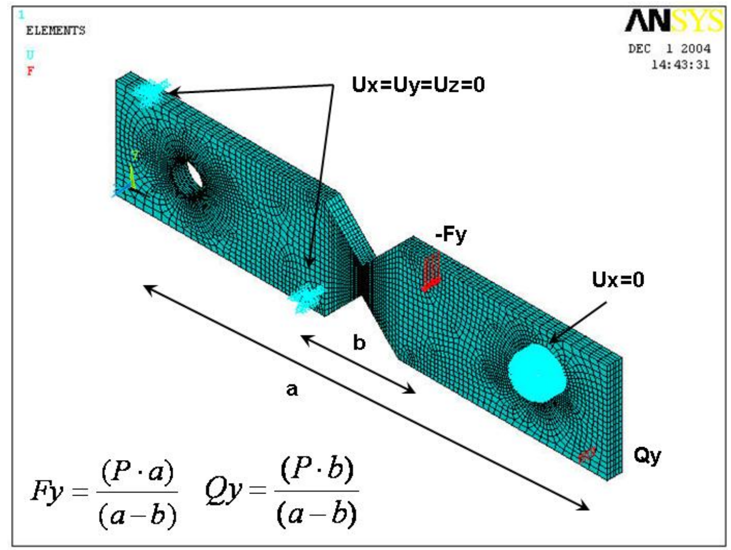
This specimen is used to measure the composite in-plane axial shear modulus and strength



Iosipescu Shear Specimen

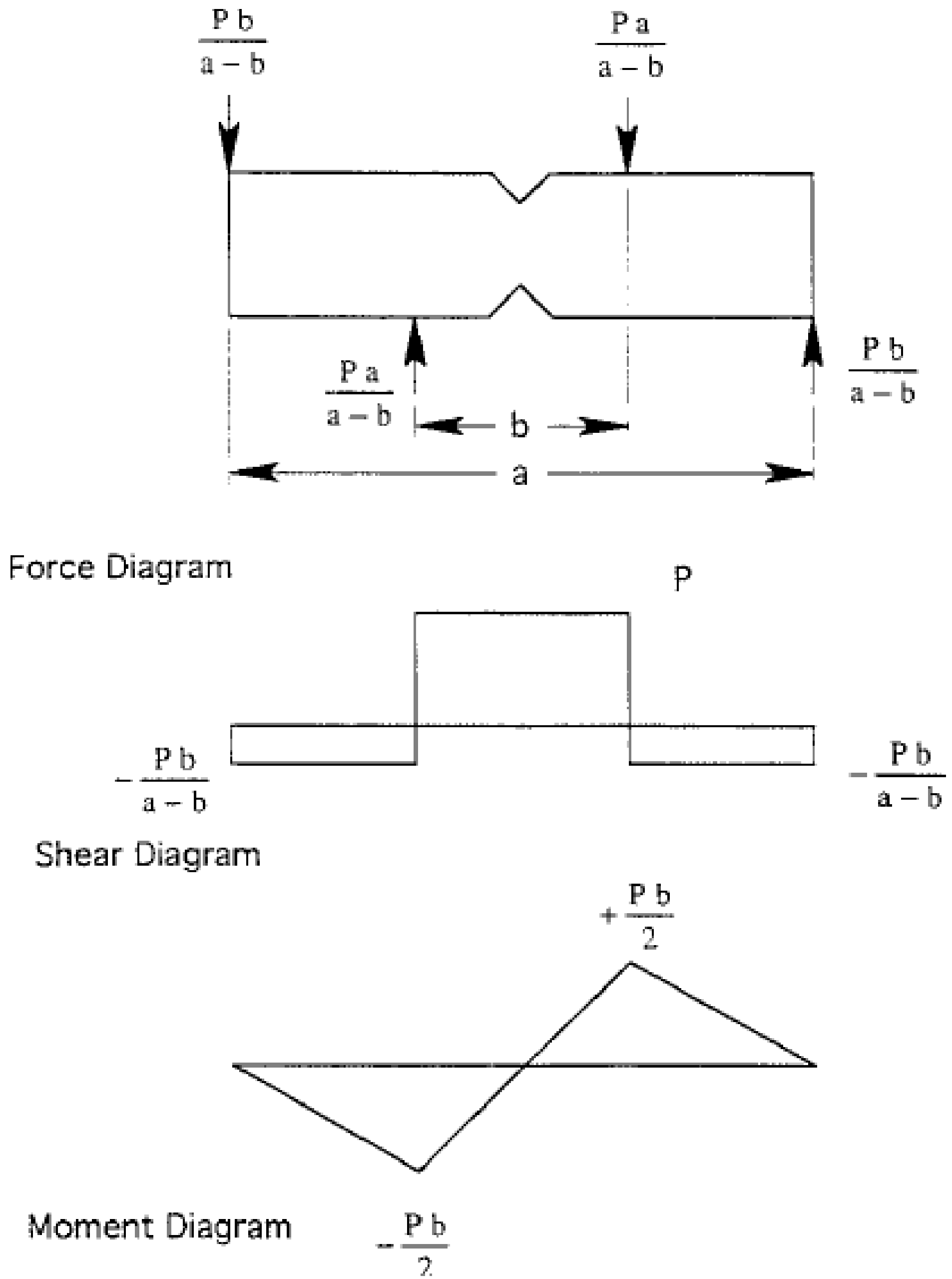


Free-body diagram

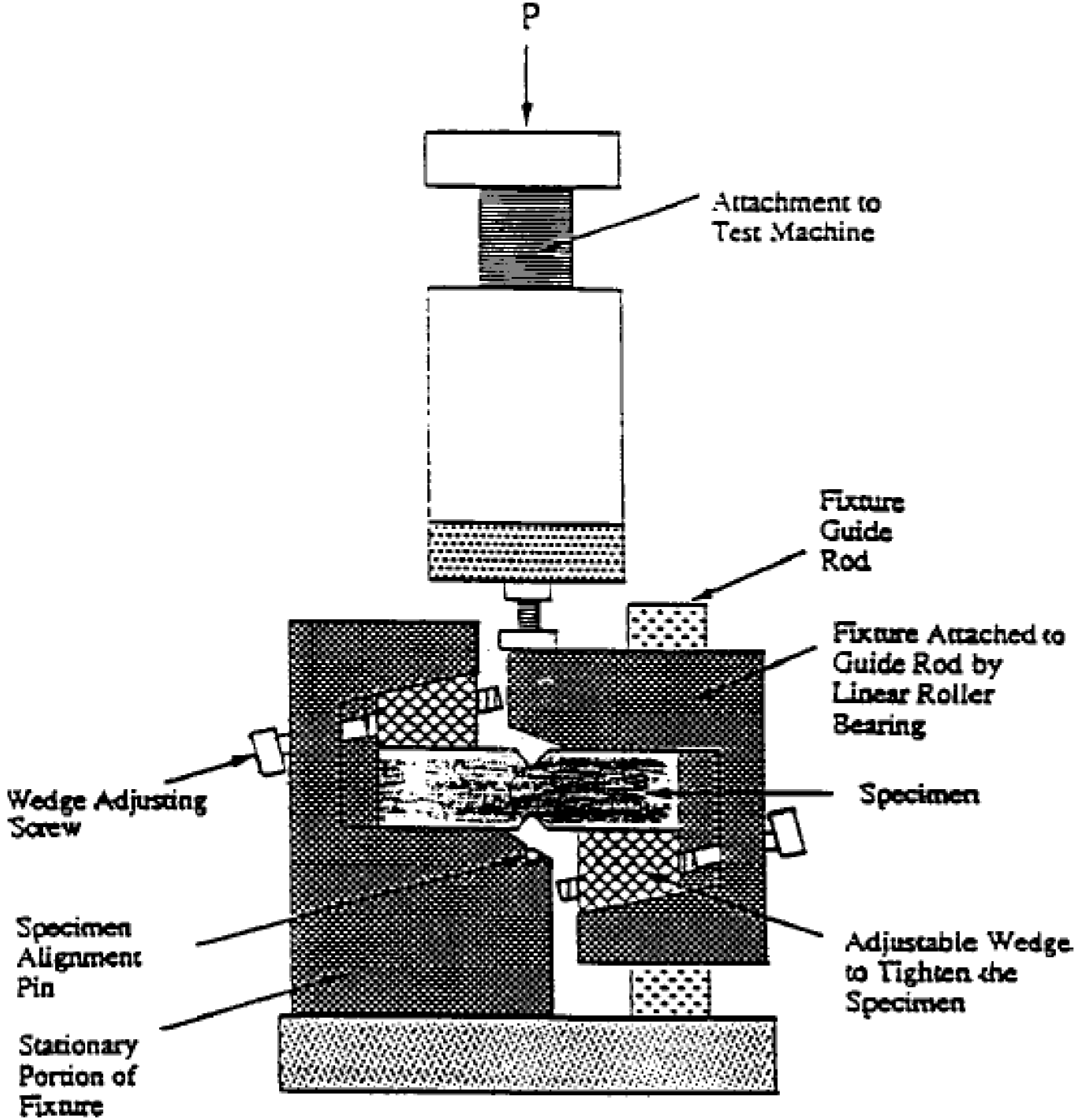


Displacement B.C.'s

Force, Shear and Moment Diagrams for Iosipescu Specimen

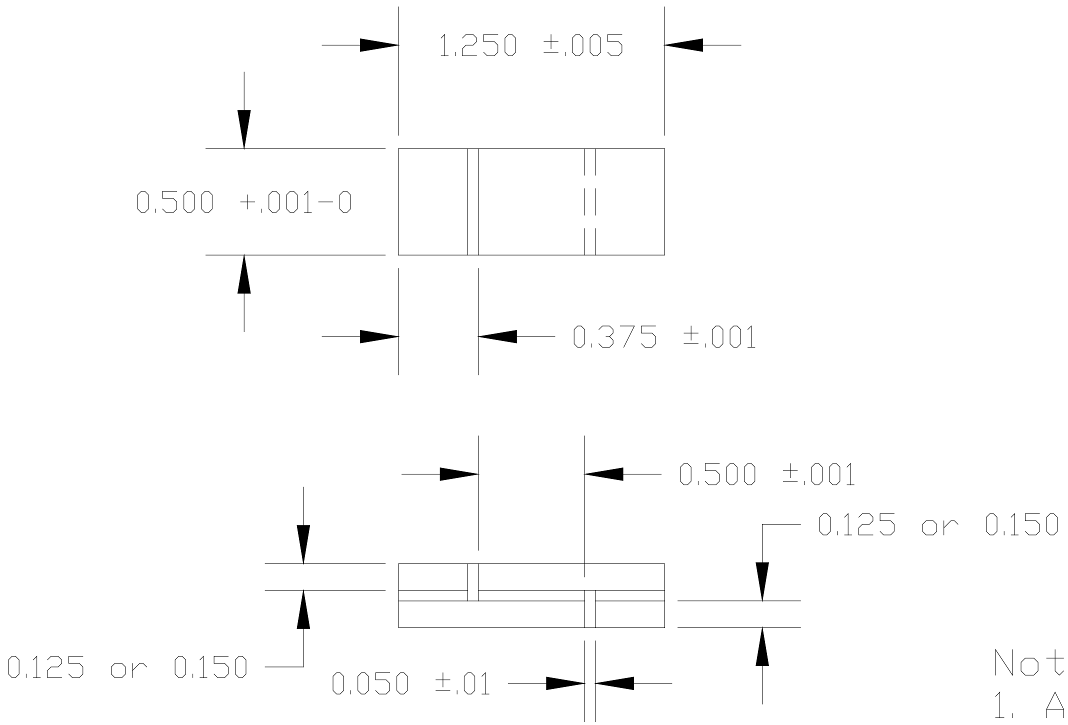


Schematic of Test Fixture for Iosipescu Test



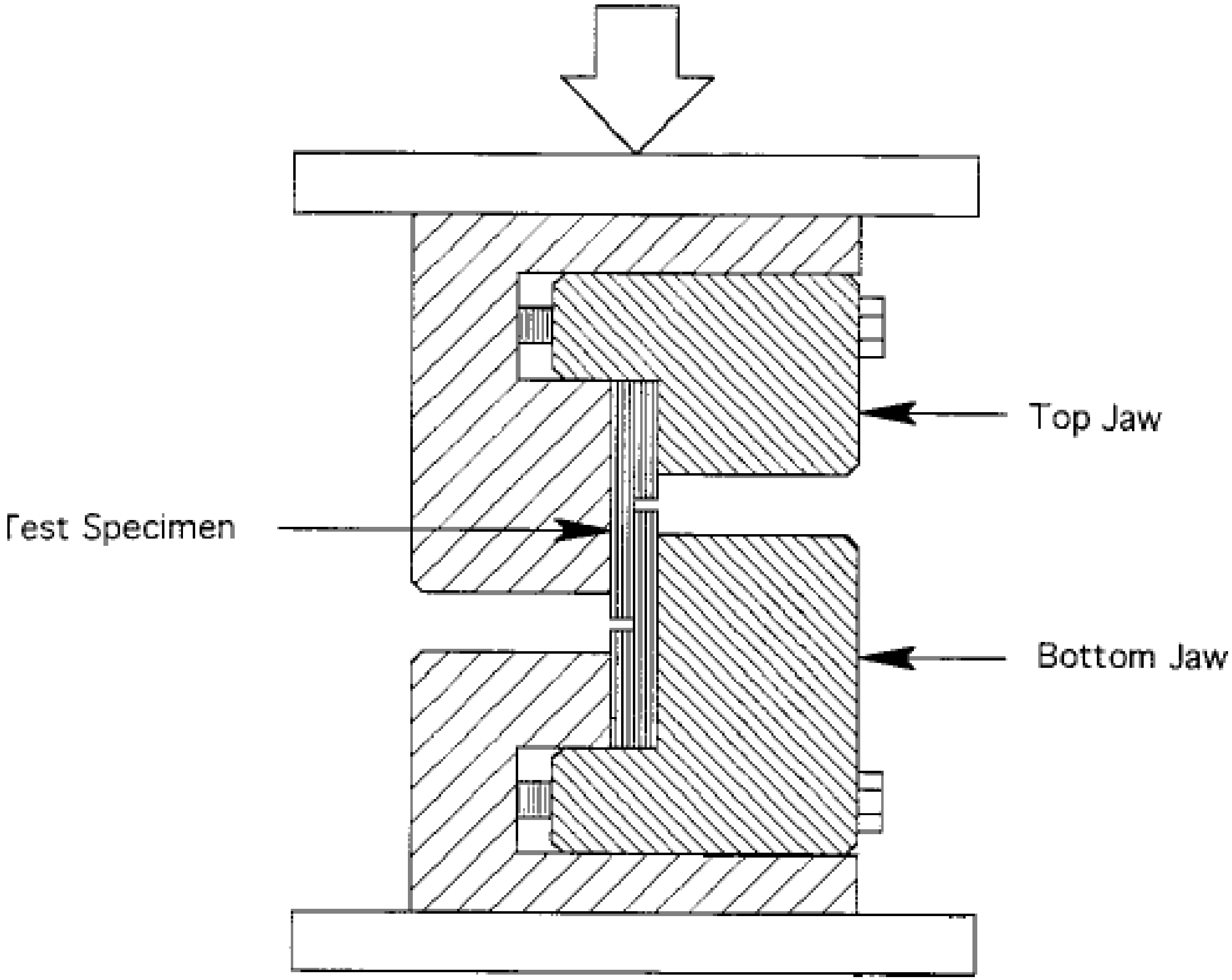
Double Notch Shear (DNS) Specimen

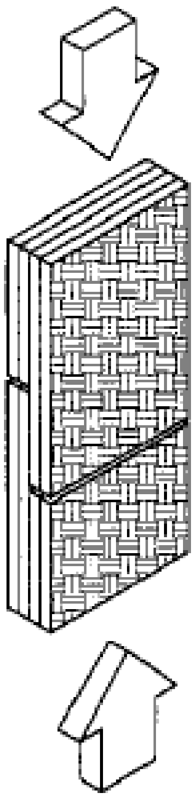
This specimen is used to measure the composite through thickness or interlaminar shear strength



Notes:
1. All dimensions are in inches.
2. C/SiC adherends may be 0.125" or 0.150" thick. Same thickness must be used for both adherends.
3. Thickness of bond is determined by candidate bond material.

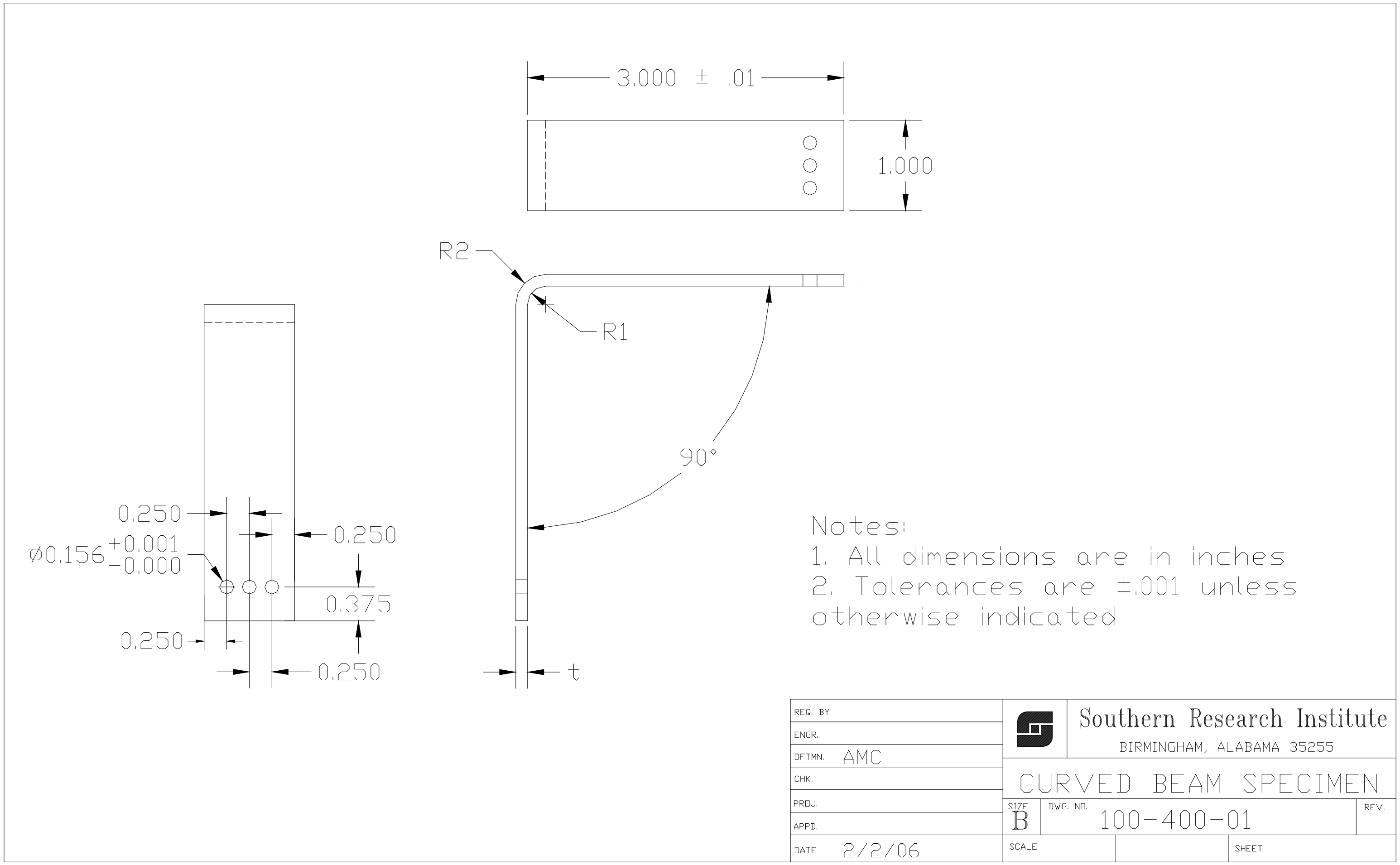
Schematic of Test Fixture and DNS Specimen



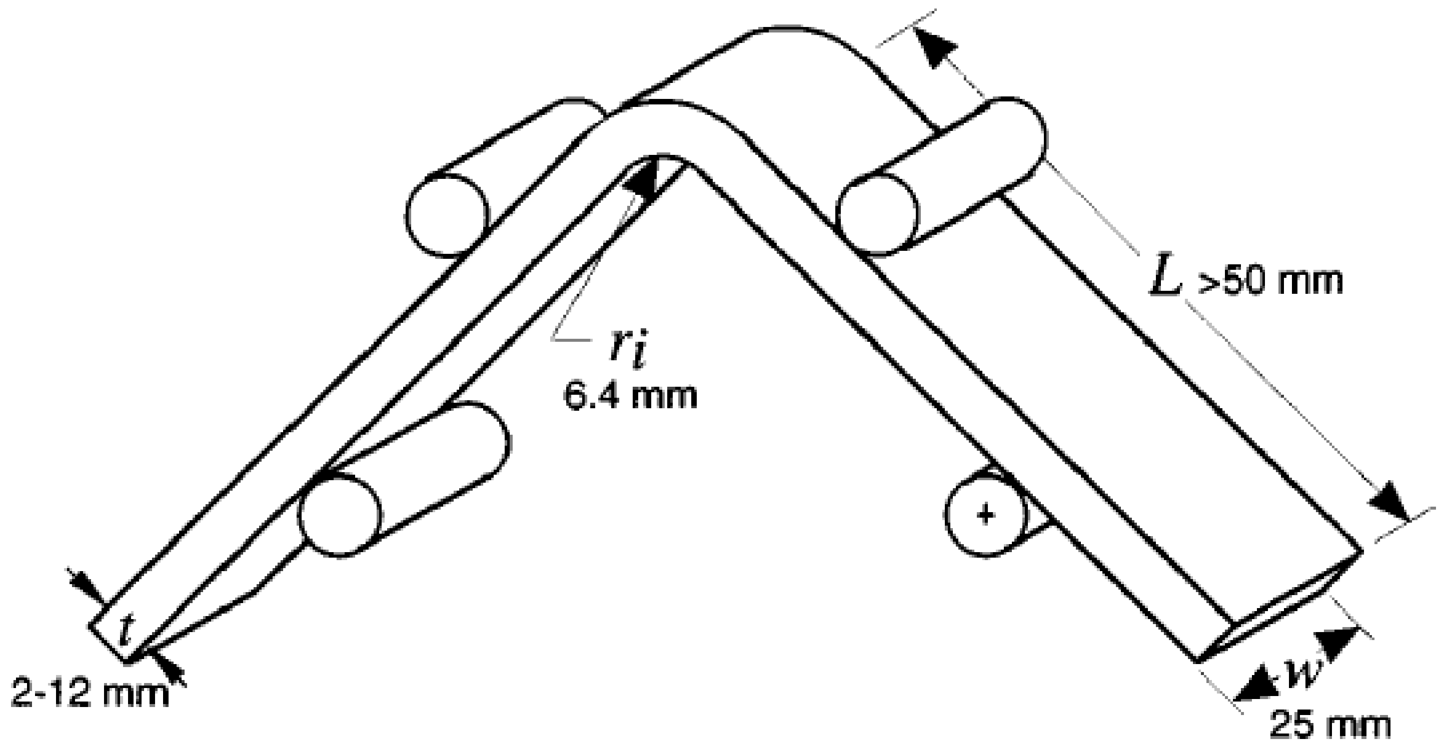


Curved Beam Specimen

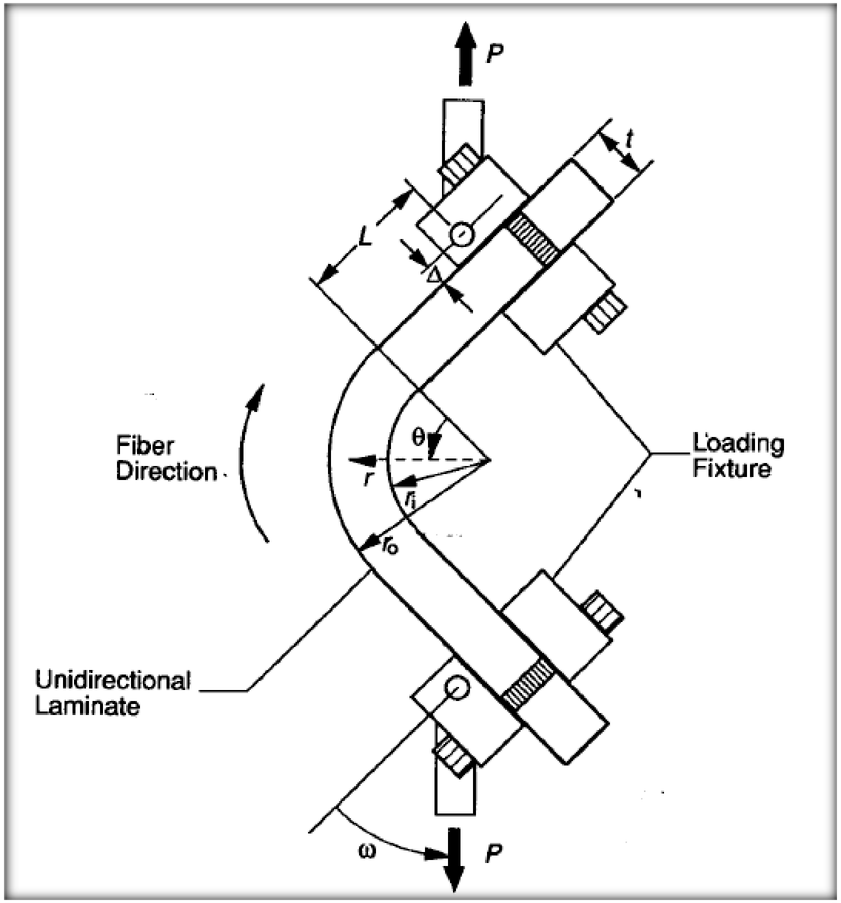
This specimen can be used to measure the composite through thickness tensile strength



Curved Beam Test Specimen and Geometry

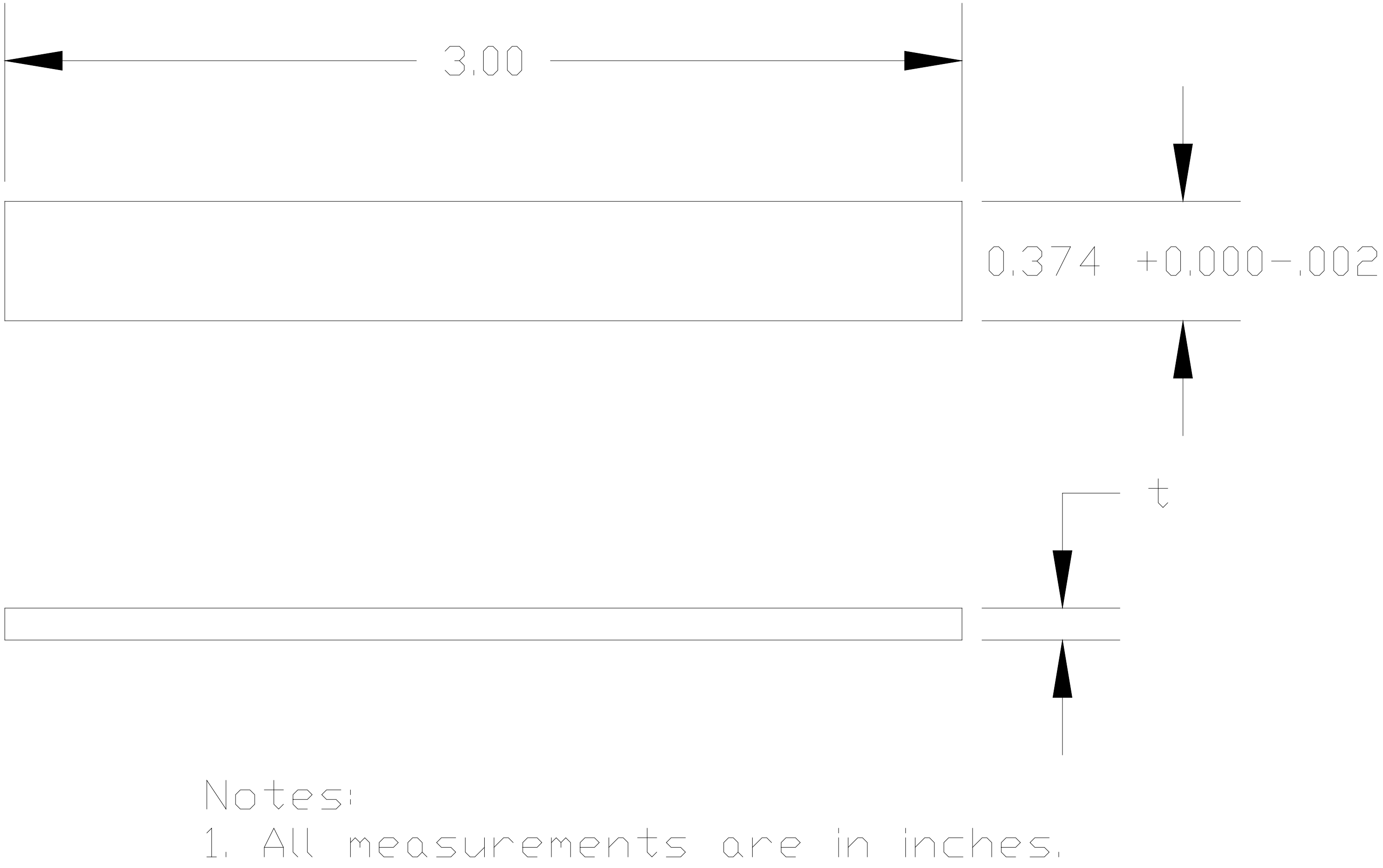


Curved Beam Test Specimen and Fixture



Thermal Expansion Specimen

This specimen can be used to measure the composite in-plane coefficients of thermal expansion (CTE)



Design Properties

- Definition of A-basis allowable property:
  - Design property for which there is a 95 percent confidence that 99 percent of the tested material samples will exceed this value
- B-basis allowable property defined as property for which there is a 95 percent confidence that 90 percent of the tested material samples will exceed this value
- Guidelines for composite material test programs to achieve A-basis or B-basis allowable properties are provided in MIL-HDBK-17

A-basis Allowable Test Matrix from MIL-HDBK-17

TABLE 2.3.6.1 *Cured laminate mechanical property test matrix designed for regression analysis.*

A-basis level matrix - 5 batches/90 data points per property

Mechanical Property	Test Methods[1]	Test Condition[2] and Number of Tests Per Batch[3]					Number of Tests
	See Handbook Section	Min	RT	ET1	ET2	ET3	
		Temp					
0° Tension (warp)	6.7.4.4[4]	3	4	3	4	4	90
90° Tension (fill)	6.7.4.4[4]	3	4	3	4	4	90
0° Compression (warp)	6.7.5.4	3	4	3	4	4	90
90° Compression (fill)	6.7.5.4	3	4	3	4	4	90
In-plane Shear	6.7.6.4	3	4	3	4	4	90
0° Short Beam Shear[5]	6.7.6.4	-	6	-	-	-	30
							480

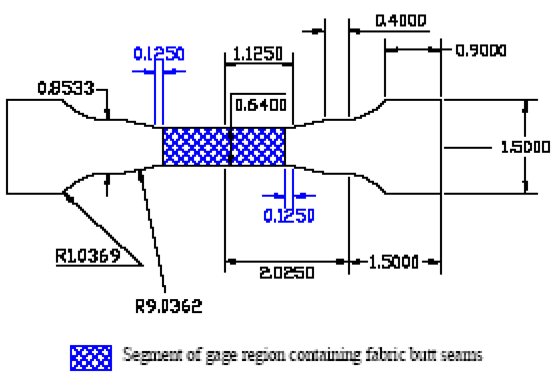
1. MIL-HDBK-17 is not currently in a position to make exclusive test method recommendations, but the referenced Handbook sections identify methods that are currently deemed acceptable for data submittals to MIL-HDBK-17.
2. Minimum and maximum temperature tests shall be performed within ±5°F (±2.8°C) of the nominal test temperature. Nominal test temperatures will be as agreed to by contractor and certifying agency. Dry specimens are "as-fabricated" specimens which have been maintained at ambient conditions in an environmentally-controlled test laboratory. Wet specimens are environmentally-conditioned by exposing them in a humidity chamber until they attain an equilibrium moisture content agreed to by the contractor and certifying agency, and then packaging them in a heat-sealed aluminized polyethylene bag until required for test. Tests shall be performed in a manner which maintains the moisture content in specimens at the levels agreed to by the contractor and certifying agency.
3. Tests shall be performed on each of the five batches.
4. For 0° and 90° tension, ASTM D 3039 and SACMA Recommended Method (SRM) 4-88 are acceptable test methods for MIL-HDBK-17 data submittals.
5. Short Beam Shear is for screening and quality control purposes only.

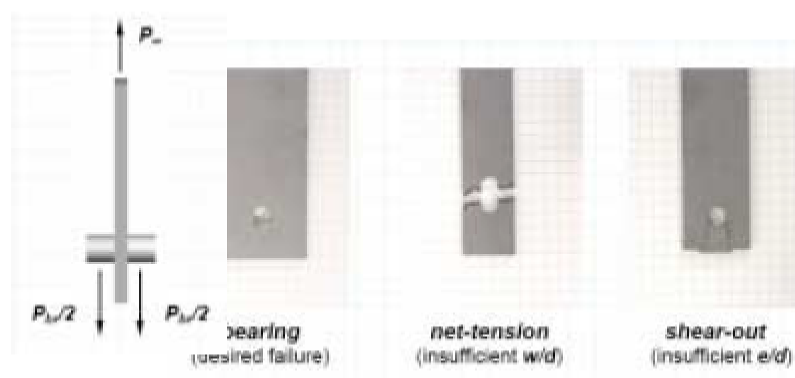
Allowable Material Properties

- A statistical software package known as STAT17, a by product of the MIL-HDBK-17 Working Group, is available for calculating A-basis and B-basis allowable properties from measured material property test data
  - A-basis and B-basis properties for polymer matrix composites used in military aircraft exist
  - Reinforced Carbon-Carbon used on Space Shuttle is the only refractory composite material for which A-basis properties exist
  - B-basis properties for specific material properties exist for ACC-6 and CVI C/SiC

Other Critical Design Properties

- Tensile strength of specimens with butt-joint plies
- Strength of notched specimens (i.e., specimens containing open holes)
- Strength of specimens containing loaded holes
  - Bearing strength
  - Net tension and/or compressive strength
  - Shear tear-out strength





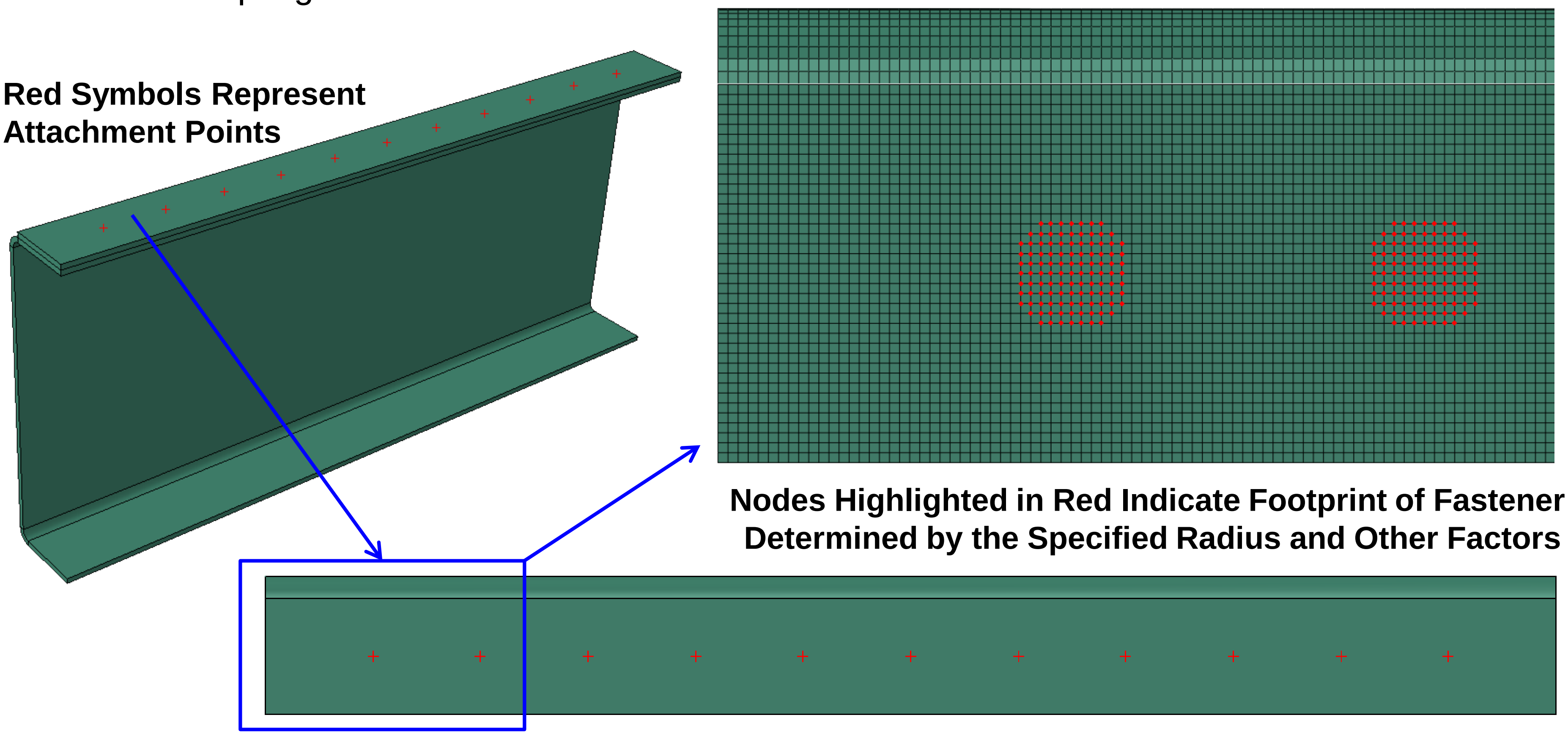
Methodology for Fastener Placement

- On large FEA models, analysts will often use coupling constraints between nodes on adjacent components to simply represent fasteners
  - However, coupling constraints require partitioning to ensure that nodes are located in regions where fasteners should be placed
  - If the mesh is sufficiently refined, there may be a node close-by the desired location, but it is unlikely that fasteners can be exactly placed in all locations using this approach, particularly if the analyst needs to iterate on the fastener patterns

- An alternative to this approach is Abaqus' mesh-independent fastener capability
  - The user positions fasteners by placing attachment points, which do not need to align with the existing mesh
  - The user can also specify the radius and mass of the fastener
  - Abaqus uses the attachment points and the fastener diameter to define a distributed coupling constraint between adjacent components; the footprint of the distributed coupling is driven by the fastener diameter
  - "Connector" output can be requested to query the bolt forces in each direction (for bolt calculations)

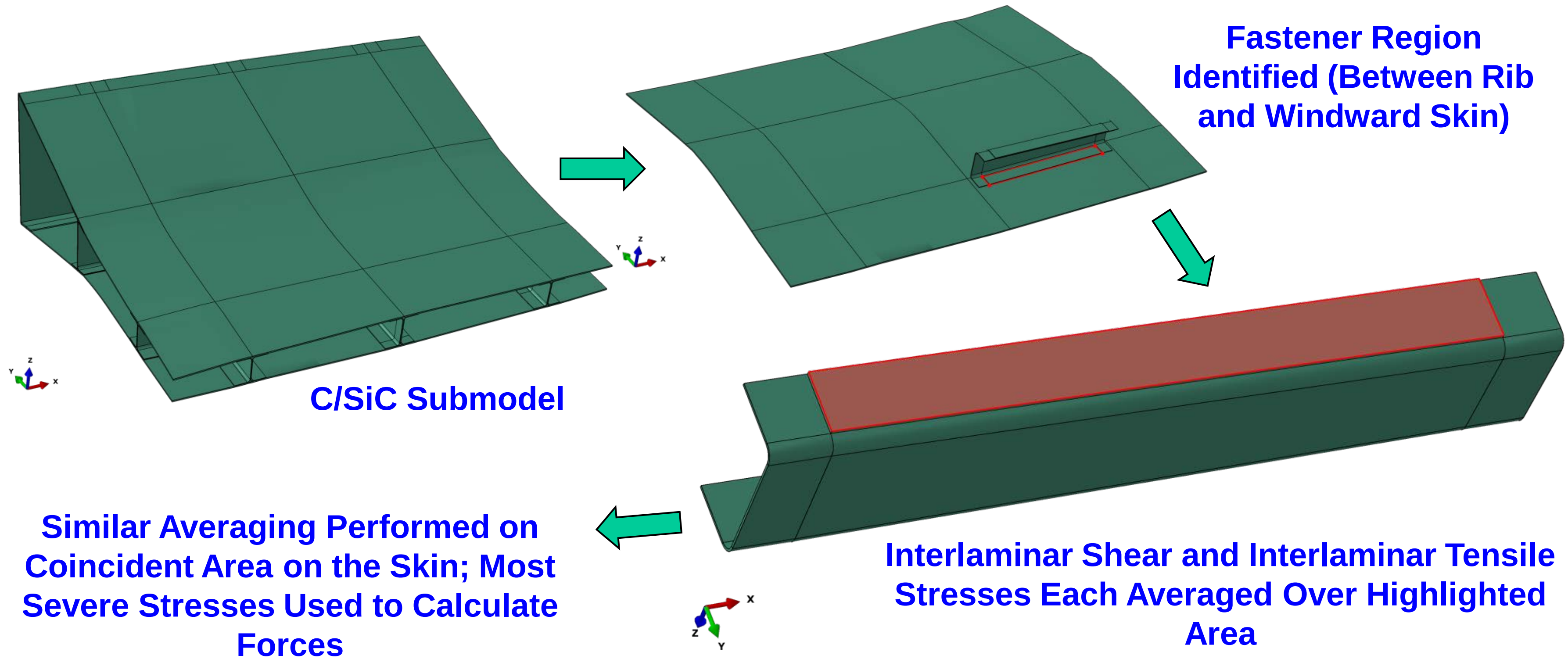
Methodology for Fastener Placement

- Mesh-independent fasteners are useful since the analyst can consider numerous fastener patterns without having to change the underlying mesh
  - Only the attachment points must be relocated, which is trivial
  - Abaqus determines which nodes on the underlying mesh should be involved in the coupling constraint



Methodology for Fastener Placement

- Results from the models solved with tie constraints are used to determine the number and placement of fasteners
- A region where fasteners are to be placed is identified, and stresses are averaged on each of the coincident areas



- Multiplying the average stress by the surface area of the selected face yields a force which must be resisted by the fasteners

Methodology for Fastener Placement

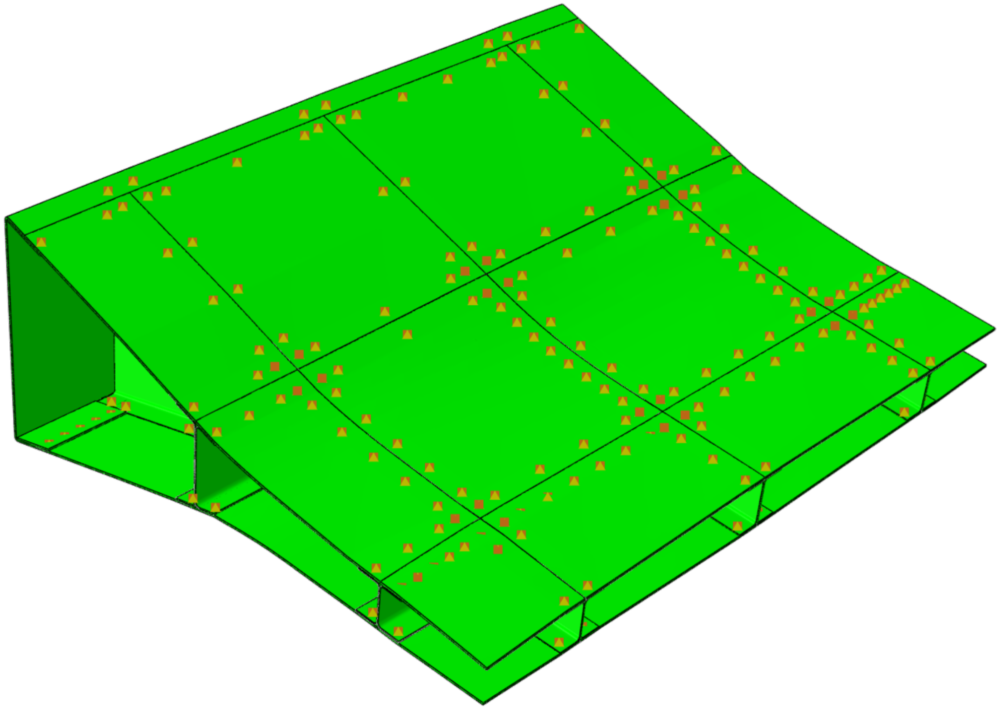
- Interlaminar shear and interlaminar tensile forces, corresponding to bolt shear and bolt tensile forces, are considered for both areas (in this example, the rib area and skin area), and whichever is more severe is used to calculate the force that must be resisted by the fasteners for a given location

- Knowing the allowables of the fastener and selecting a fastener diameter, the number of required fasteners can be determined

- This process is completed for each of the cases being considered for a given material

- The load case requiring more fasteners is used to place fasteners in a given location (the flange highlighted on the last slide, for example) for all load cases

Methodology for Fastener Placement



- Results from the submodels solved with tie constraints were used to determine the required number of fasteners for each load case
- All appropriate tie constraints in the submodels were removed and replaced with mesh-independent fasteners
  - Fasteners are shown as ■ symbols in the image to the right
- Submodels were re-analyzed with mesh-independent fasteners
- Bolt calculations will be performed to check for any failures; the number of fasteners will be adjusted as necessary depending on these results
  - Due to the high number of fasteners in these models, performing calculations for each fastener individually would be very time consuming
  - Post-processing script can be written to pull the relevant fastener output from the solved models and calculate bolt tensile and shear failures

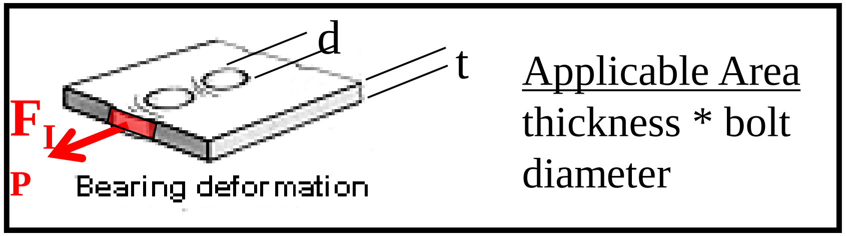
Post-Processing Methodology

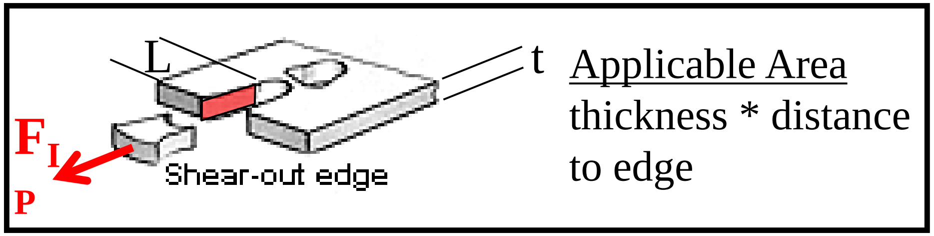
1. **Perform fastener calculations**
   - It is common to generate preliminary fastener layouts based on the results of tied submodels, but it is expected that some iteration will need to be performed to finalize these layouts
   - Fastener calculations will focus on bolt tension and bolt shear to determine whether a sufficient number of fasteners (or if too may fasteners) have been included in each submodel
   - The number and distribution of fasteners will affect local displacement and stress, so these details should be finalized before post-processing of the surrounding material takes place
   - Available measured fastener strengths and a factor of safety (FOS) of 2 should be used for all fastener calculations

2. **Perform bearing, net tensile, and shear-out calculations on the material surrounding the fasteners**
   - Loaded hole strengths for all three materials will be used in these calculations
   - A FOS of 2 is used for all calculations

3. **Consider material away from the fasteners; calculate margins of safety**
   - Estimated B-basis allowables and an acceptable FOS (e.g. 1.4) is used for all of these calculations
   - Based on available open-hole data on the composite material, a determination is made as to whether or not open-hole or notched strengths are required for post-processing the composite material design

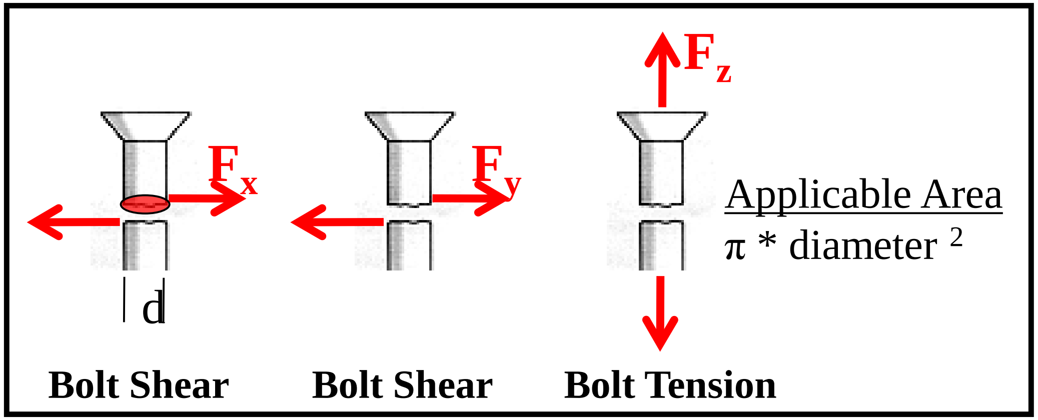
Example of Bolt Calculation Methodology





Potential Failure Mode	Applicable Force	Diameter	PIP FS Thickness	3D C-C Thickness	Applicable AREA	Calculated Stress	Allowable Stress Description	Temp at Peak Stress (°R)	Temp at Peak Stress (°F)	Predicted Strength	FOS	MOS
Facesheet Bearing	37.98	0.30	0.10	0.10	0.03	1.27	QI C-C σ_1^c	1609.50	1149.50	26.0	2.00	9.27
Facesheet Shear-Out	37.69	0.30	0.10	0.10	0.10	0.38	QI C-C τ^{12}	1609.50	1149.50	17.2	2.00	21.78
Spar Box Bearing	37.98	0.30	0.10	0.10	0.03	1.27	3D C-C σ_1^c	1609.50	1149.50	11.8	2.00	3.66
Spar Box Shear-Out	37.69	0.30	0.10	0.10	0.10	0.38	3D C-C τ_{12}	1609.50	1149.50	3.8	2.00	4.04
C/SiC Bolt Shear X	-4.71	0.30	0.10	0.10	0.07	-0.07	CVI C/SiC τ_{13}	1609.50	1149.50	2.9	2.00	20.61
C/SiC Bolt Shear Y	37.69	0.30	0.10	0.10	0.07	0.53	CVI C/SiC τ_{12}	1609.50	1149.50	32.3	2.00	29.28
C/SiC Bolt Tension	30.03	0.30	0.10	0.10	0.07	0.42	CVI C/SiC σ_1^{tens}	1609.50	1149.50	42.5	2.00	48.98

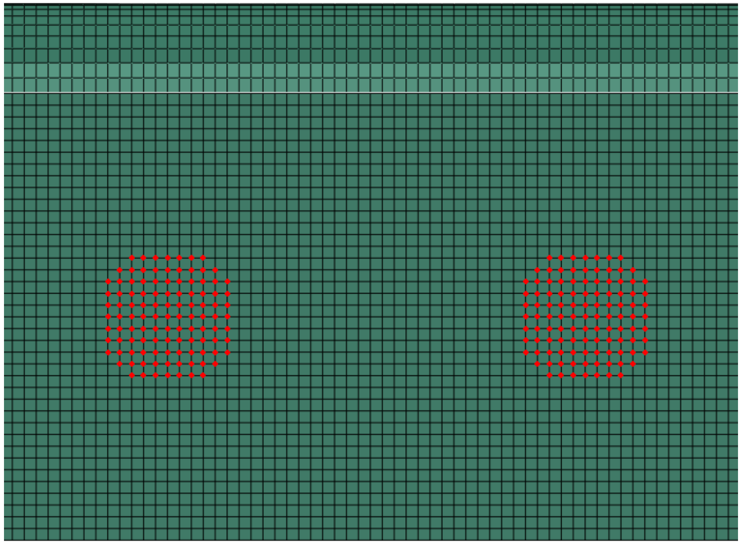
Fastener failure & facesheet / adherend failure modes must be checked for every fastener; note FOS set to 2 in all composite joints



Local fastener analysis used to determine material and fastener failure

Post-Processing the Surrounding Material

- The mesh-independent fastener definitions are essentially distributed coupling constraints whose footprints are equal to (or close to, depending on the mesh density) the specified diameter of the fastener
- The stresses that fall within the fastener footprint are artificially high due to the constraint – in reality there would be a hole in these locations, so the stresses that are present are not realistic
- These artificially high stresses are typically ignored when analyzing material away from the fasteners
- The reaction forces in the connector elements themselves are used to perform bearing, net tensile, and shear-out calculations for the material surrounding the fasteners
- As noted in the outline of post-processing tasks, the estimated B-basis allowables paired with an acceptable FOS (e.g. 1.4) are typically used when post-processing stresses away from the fasteners; loaded hole strengths and a FOS of 2 are used for bearing, net tensile, and shear-out calculations



Calculation of Margins of Safety

- Margins of safety (MOS) are used to quantify the state of stress or strain relative to design allowable values
- Margins of safety must be calculated using either calculated stresses or strains
- In expression below, substitute $\varepsilon_{Allowable}$ and ε_{Actual} for corresponding stress quantities, if strain allowable design approach is used
- Typically for composite stresses calculated via FEA, the MOS for one component at a time is calculated

$$MOS = \frac{\sigma_{Allowable} - \sigma_{Actual} \cdot FOS}{\sigma_{Actual} \cdot FOS} = \frac{\sigma_{Allowable}}{\sigma_{Actual} \cdot FOS} - 1$$

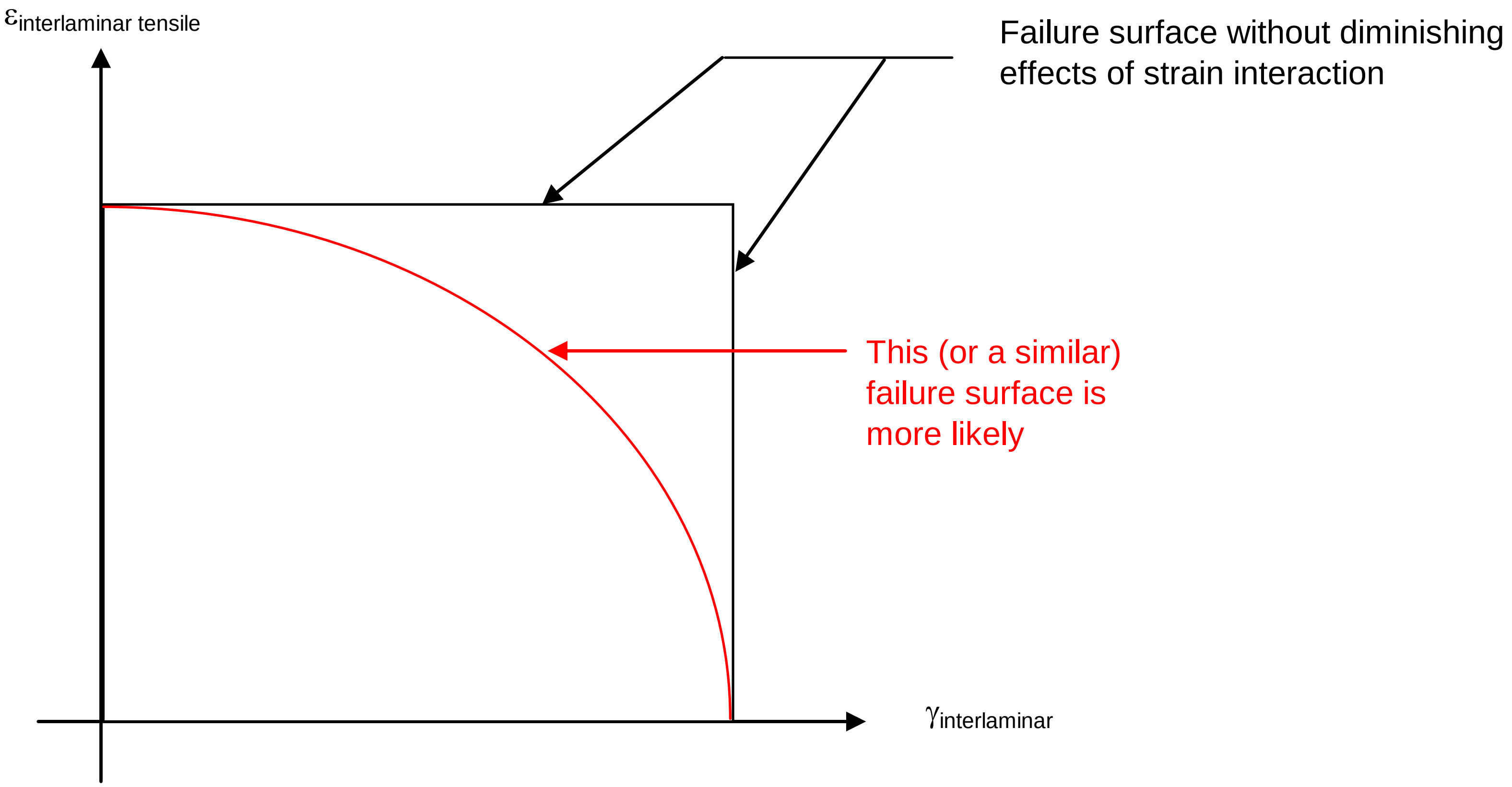
Calculation of Margins of Safety

- Example of a typical MOS table:

ANSYS Stress Component	Predicted Strength 70.00 (ksi)	Predicted Strength 2000.00 (ksi)	Temp Dep Predicted Strength (ksi)	Temperture at Peak Stress °R	Temperture at Peak Stress °F	ANSYS Calculated Stress (ksi)	Factor Of Safety	Margin Of Safety
σ_{XX}^{T}	37.9	46.5	39.73	945.00	486.33	6.19	1.0	5.42
σ_{XX}^{C}	51.7	53.7	52.61	1423.00	964.33	-25.31	1.0	1.08
σ_{YY}^{T}	37.9	46.5	39.78	956.00	497.33	6.03	1.0	5.60
σ_{YY}^{C}	51.7	53.7	52.63	1439.00	980.33	-12.62	1.0	3.17
σ_{ZZ}^{T}	1.5	2.0	1.58	904.00	445.33	0.50	1.0	2.13
σ_{ZZ}^{C}	14.382	13.94	14.17	1435.00	976.33	-3.52	1.0	3.03
t_{XY}	28.9	35.5	31.86	1400.00	941.33	7.63	1.0	3.18
t_{YZ}	1.6	3.0	1.89	961.00	502.33	0.90	1.0	1.09
t_{XZ}	1.6	3.0	1.75	779.00	320.33	0.78	1.0	1.24

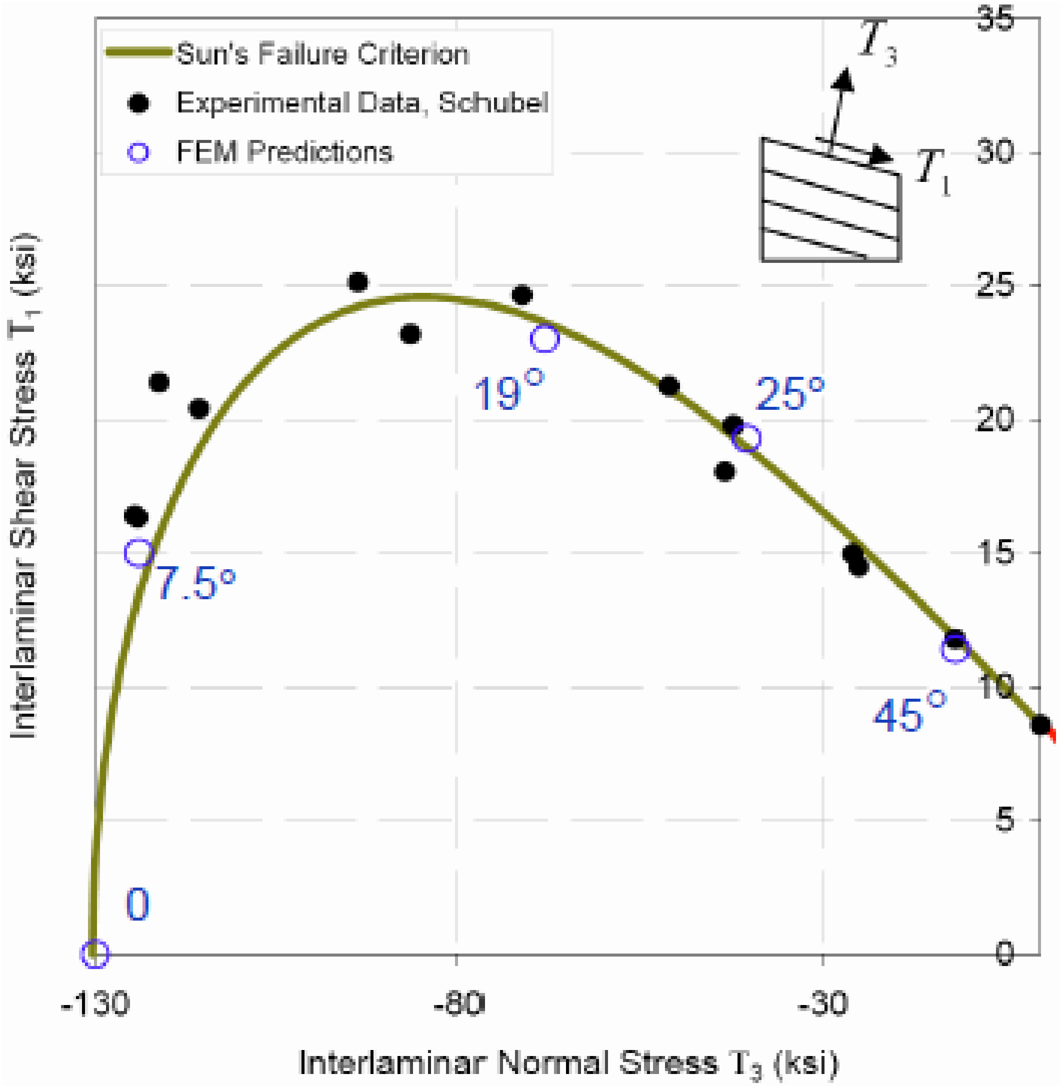
- This approach ignores potential adverse affects of stress or strain interaction

Interlaminar Strain Interaction



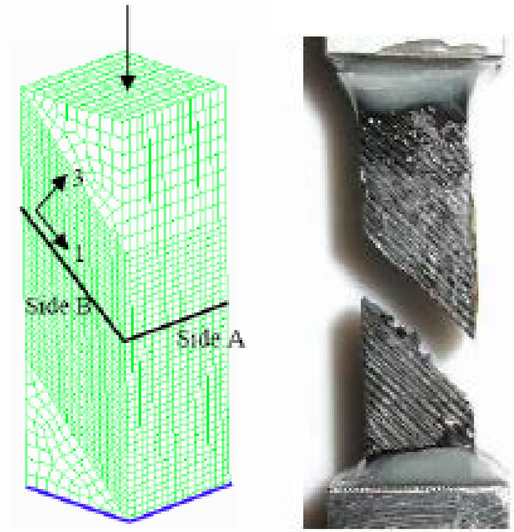
Material characterization testing to define interaction curves for all relevant components and at all temperatures of interest can be costly

Failure Criteria for Delamination Initiation



$$\left(\frac{T_3}{S_{3t}}\right)^2+\left(\frac{T_1}{S_1}\right)^2+\left(\frac{T_2}{S_2}\right)^2=1, \quad T_3\geq 0$$

$$\left(\frac{T_3}{S_{3c}}\right)^2+\left(\frac{T_1}{S_1-\eta T_3}\right)^2=1, \quad T_3< 0$$



FEM **Failed Specimen**

Margin of Safety Calculation

- Selected time for thermal stress analysis should correspond to the time of peak thermal gradient in the component
- Margins of safety (MOS) for each in-plane stress are individually calculated using a maximum stress failure criteria
- Interlaminar margins of safety are calculated using stress-interaction failure criteria (shown below)
- Factor of Safety = value agreed upon by all parties for all MOS calculations

Thru-Thickness Tension

$$MOS(T3>0)=\frac{1}{\sqrt{\left(\frac{FOS*\sigma_{xz}}{ILSS}\right)^2+\left(\frac{FOS*\sigma_{zz}}{ILTS}\right)^2+\left(\frac{FOS*\sigma_{yz}}{ILSS}\right)^2}}-1$$

Thru-Thickness Compression

$$MOS(T3<0)=\frac{1}{\sqrt{\left(\frac{FOS*\sigma_{zz}}{ILCS}\right)^2+\left(\frac{FOS*\sigma_{xz}}{ILSS-\eta\sigma_{zz}}\right)^2+\left(\frac{FOS*\sigma_{yz}}{ILSS-\eta\sigma_{zz}}\right)^2}}-1$$

η = friction coefficient; η = 0.28 provides best correlation for Gr/Ep composites; η > 0.28 is likely for C-C

Comments on Load Factors and Factors of Safety

- Load factors and factors of safety are used to amplify loads and calculated stresses, respectively, prior to determining MOS values
- Load factors are a reflection of the degree of uncertainty in the applied loads
- Factors of safety are intended to reflect the degree of uncertainty in the calculated stresses due to the complexity of the structure and/or the uncertainty in the math model representation of physical structure
- Typical factors of safety for composite structures (from NASA-STD-5001: Structural Design and Test Factors of Safety for Spaceflight Hardware)

TABLE III. Minimum Design and Test Factors for Composite/Bonded Structures

Verification Approach	Geometry of Structure	Ultimate Design Factor	Qualification Test Factor	Acceptance or Proof Test Factor
Prototype	Discontinuities	2.0*	1.4	1.05
	Uniform Material	1.4	1.4	1.05
Protoflight	Discontinuities	2.0*	NA	1.2
	Uniform Material	1.5	NA	1.2

NOTE:
* Factor applies to concentrated stresses. For non-safety critical applications, this factor may be reduced to 1.4 for prototype structures and 1.5 for protoflight structures.

Hot Structure / TPS Components

Load Factors and Factors of Safety

Vehicle	Limit Load Factors		Factor of Safety		Margin of Safety Calculation
	Mechanical	Thermal	Mechanical	Thermal	
AHW		~1.2	1.25	1.25	
X-33	1.25	1.0	1.25	1.25	Estimated B-Basis Allowables
X-37	1.5	1.5	1.5	1.0	Estimated B-Basis Allowables
X-43	1.0	1.2	1.5	1.0	Estimated B-Basis Allowables
X-51	2.0	1.2	1.5 / 2.0*	1.0	Estimated B-Basis Allowables
Space Shuttle	1.4	1.4	1.4	1.0	**True A-Basis Allowables**
NASA	Depends on uncertainty (NASA-STD-5002)		1.5 (NASA-STD-5001A)	1.5 (NASA-STD-5001A)	A-Basis allowables if man-rated; B-basis allowable otherwise

* 1.5 if tested; 2.0 if not tested

Summary

- FEA modeling and analysis of composite structural components must be sensitive to relatively poor interlaminar properties, especially for refractory composites
  - Submodeling is frequently necessary
- Nonlinear material elastic analysis is more appropriate for certain types of composite materials, e.g. particular types of refractory composites, high strain-to-failure composites
  - Strain based design criteria applies in this case
- Post-processing of calculated stresses and strains must account for textile reinforcement architecture effects
  - 3-5 unit textile cell volume averaging
- FEA methodology exists for determining optimum number of fasteners for bolted structure
- Interaction criteria are important in calculation of MOS values, primarily for interlaminar components