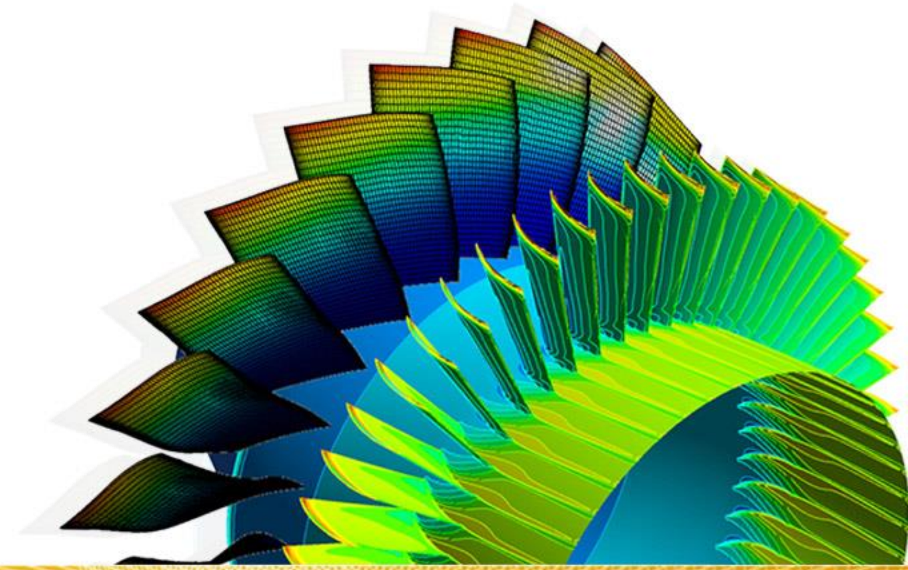
ANSYS Composite PrepPost 19.0

Workshop 10.3 – Modeling Progressive Damage – Advanced Example



12. Modeling Progressive Damage

Prerequisites:

The user is familiar with

- Standard workflow in ANSYS Workbench and ANSYS Composite PrePost
- Composite Solid Modeling
- Basics of Composite Engineering

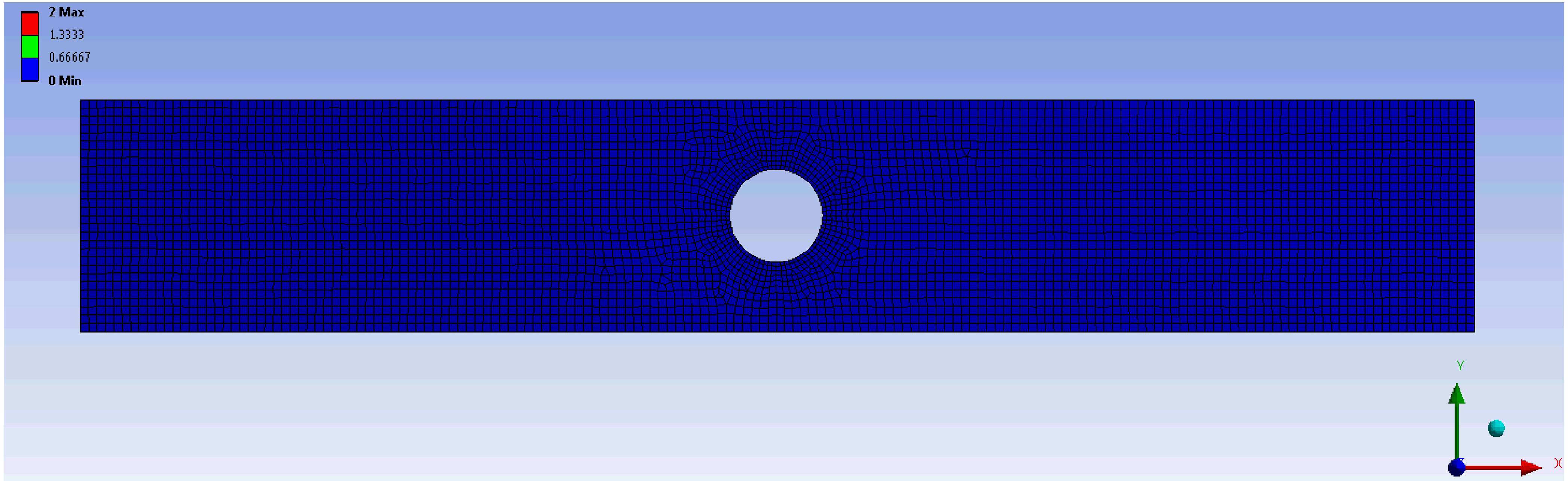
12. Modeling Progressive Damage

Agenda

- Workflow to model progressive damage modeling in ANSYS Workbench
    - Define the engineering properties
    - Define the damage initiation and damage progression variables
    - Setup the composite layup in ACP (Pre)
    - Apply loads and boundary conditions
    - Solve
    - Postprocessing

ANSYS

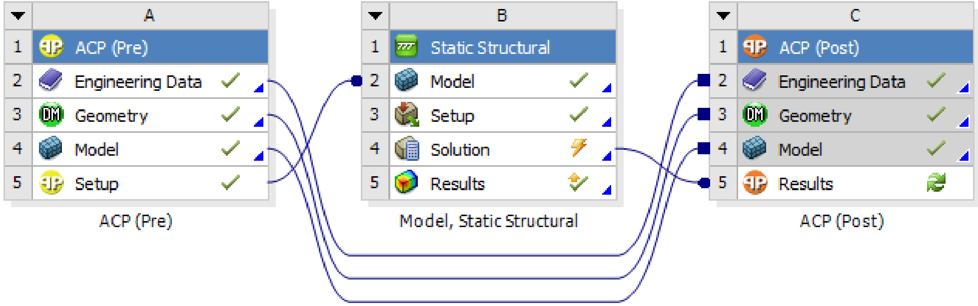
Progressive Damage



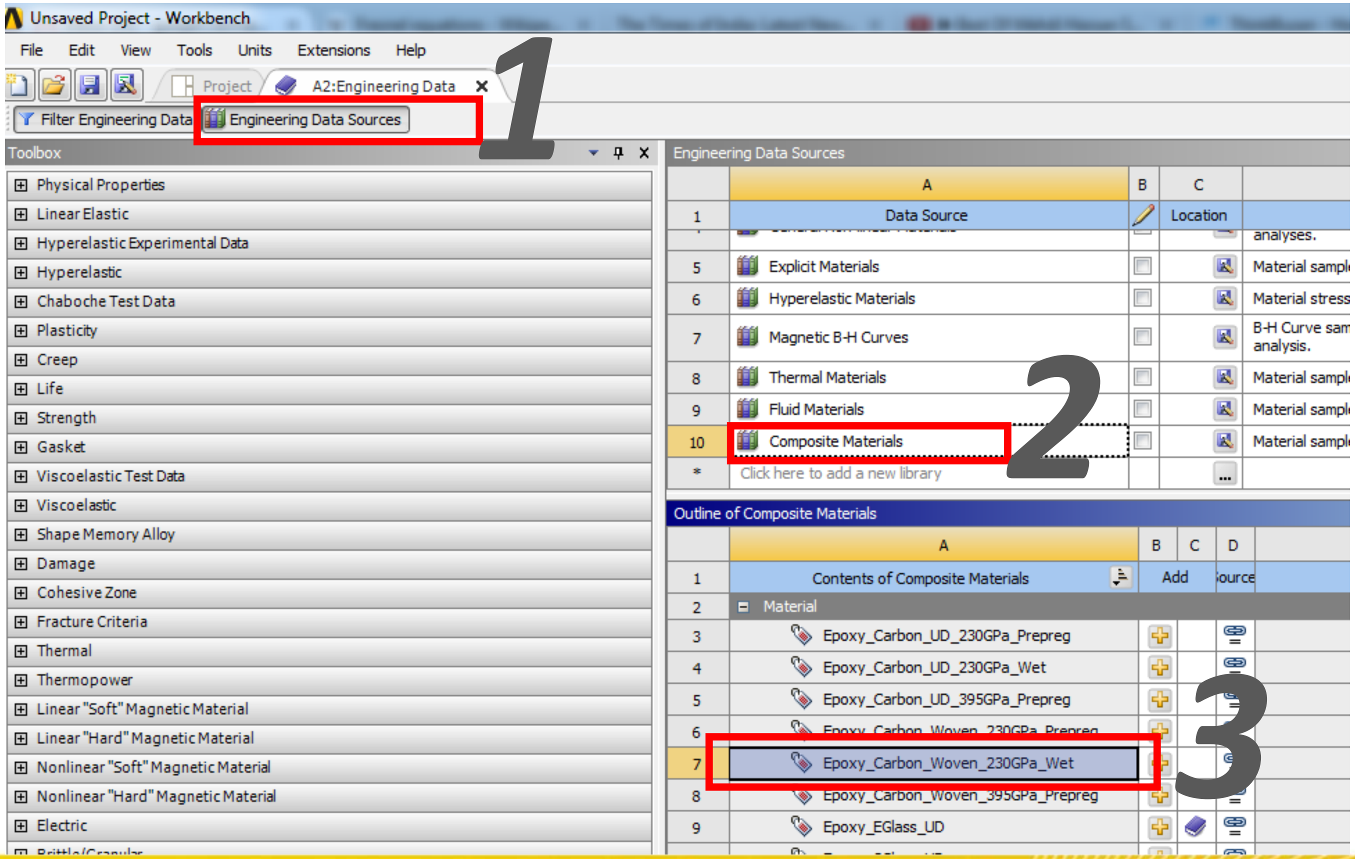
Animation of damage progression

12. Modeling Progressive Damage

- Open workbench archive *PD-WS_19.0.wbpz* and update the project



- Double click on Engineering Data



- Select Epoxy_Carbon_UD meterial from the composite materials library

ANSYS

12. Modeling Progressive Damage

Damage Initiation and Progression Variables

Properties of Outline Row 3: Epoxy_Carbon_UD

	A	B	C
1	Property	Value	Unit
2	Density	1.49E-09	mm^-3 t
3	⊞ Orthotropic Secant Coefficient of Thermal Expansion		
9	⊟ Orthotropic Elasticity		
10	Young's Modulus X direction	1.21E+05	MPa
11	Young's Modulus Y direction	8600	MPa
12	Young's Modulus Z direction	8600	MPa
13	Poisson's Ratio XY	0.27	
14	Poisson's Ratio YZ	0.4	
15	Poisson's Ratio XZ	0.27	
16	Shear Modulus XY	4700	MPa
17	Shear Modulus YZ	3100	MPa
18	Shear Modulus XZ	4700	MPa
19	⊞ Field Variables		
23	⊟ Orthotropic Stress Limits		
24	Tensile X direction	2231	MPa
25	Tensile Y direction	29	MPa
26	Tensile Z direction	29	MPa
27	Compressive X direction	-1082	MPa
28	Compressive Y direction	-100	MPa
29	Compressive Z direction	-100	MPa
30	Shear XY	60	MPa
31	Shear YZ	32	MPa
32	Shear XZ	60	MPa

Elastic Properties

Stress Limits

2

⊟ Damage Initiation Criteria	
Tensile Fiber Failure Mode	Maximum Stress
Compressive Fiber Failure Mode	Maximum Stress
Tensile Matrix Failure Mode	Maximum Stress
Compressive Matrix Failure Mode	Maximum Stress
⊟ Damage Evolution Law	
Tensile Fiber Stiffness Reduction	0.75
Compressive Fiber Stiffness Reduction	0.75
Tensile Matrix Stiffness Reduction	0.75
Compressive Matrix Stiffness Reduction	0.75

3

(1) Insert Damage Initiation and Damage Progression parameters

⊟ Damage
Damage Initiation Criteria
Damage Evolution Law

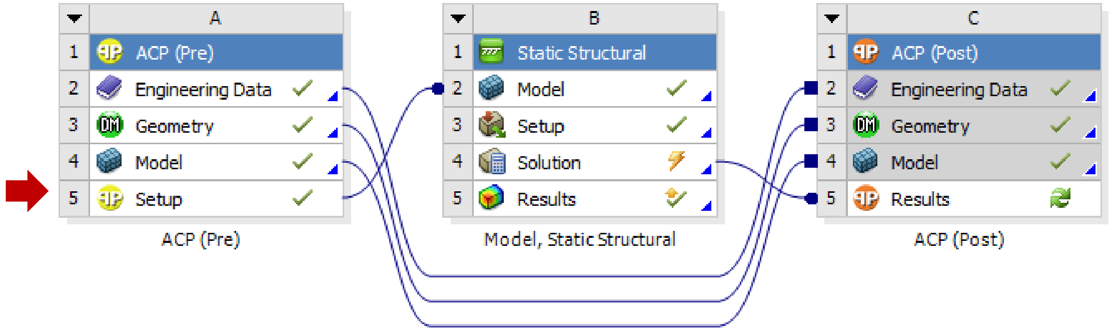
1

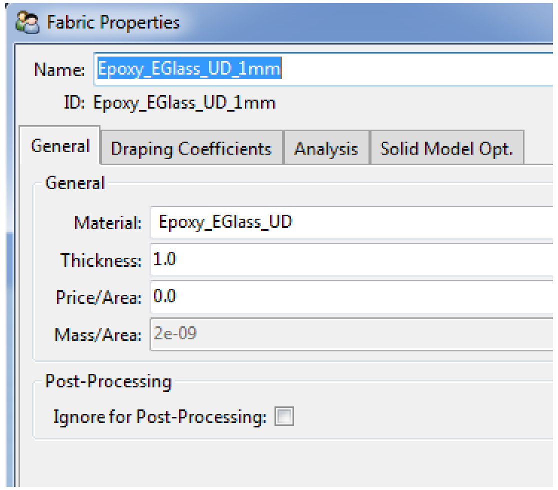
(2) Take the Maximum Stress criterion as the damage initiation criterion for all the modes. (3) Assign a damage reduction factor of 0.75 for all the inputs

ANSYS

12. Modeling Progressive Damage

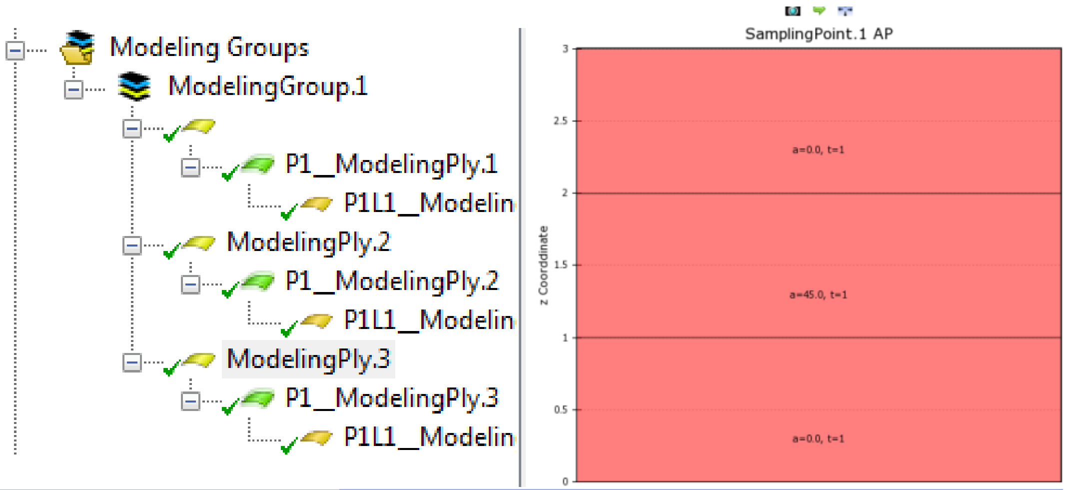
Open ACP (Pre)

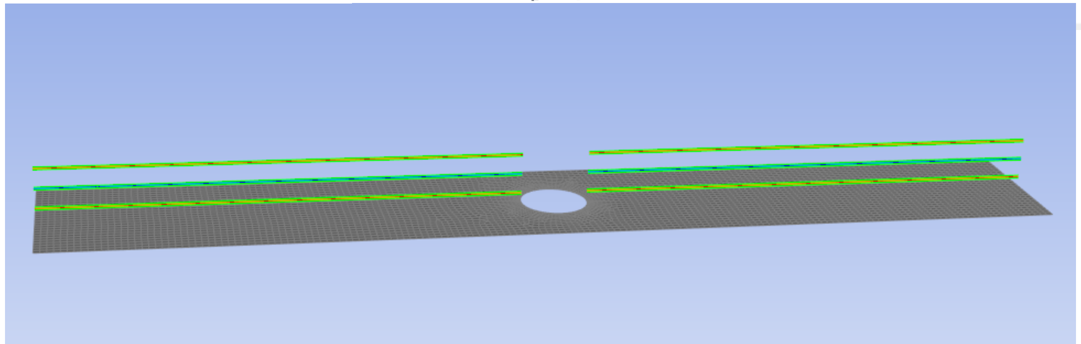




Fabric of thickness 1mm

3 plies of the same fabric are defined in Modeling Groups

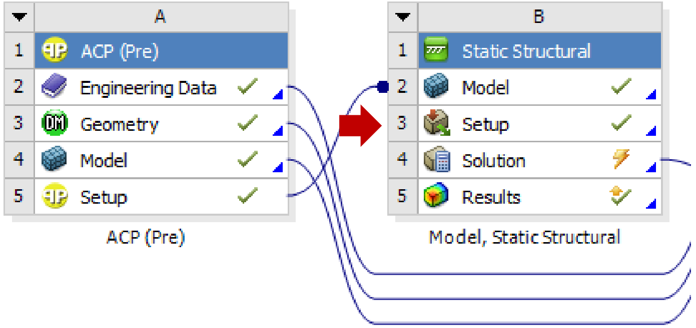




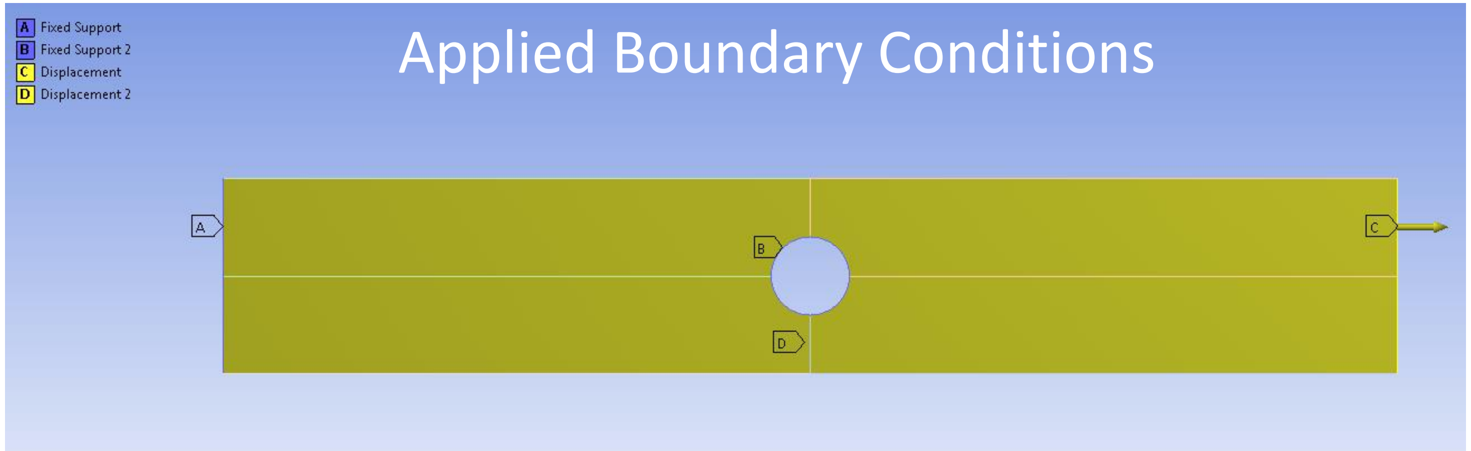
[0/45/0] layup throughout the geometry

12. Modeling Progressive Damage

Model Setup in Static-Structural



Open Mechanical

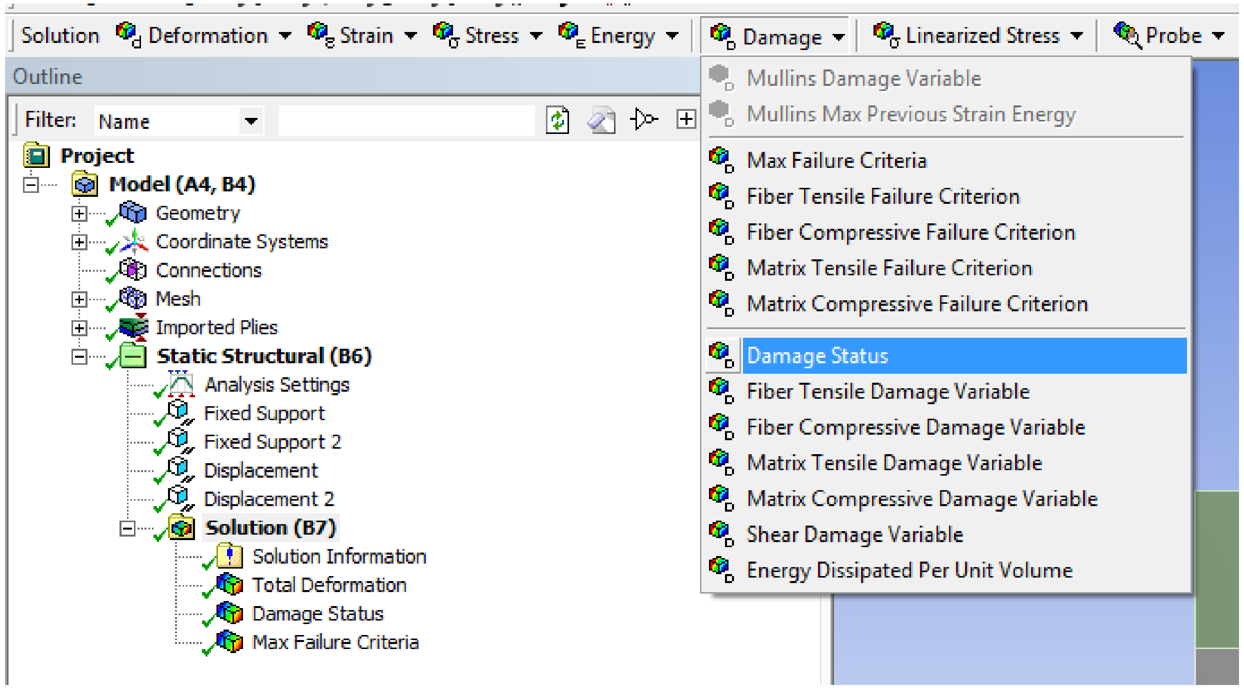


A : Edge on left side fixed in all directions
B : Edge of the hole fixed in all direction
C : A displacement of 2 mm applied to the edge on the right side
D: All nodes of the model constrained in the Z (normal to plane) direction

12. Modeling Progressive Damage

- In Mechanical users can postprocess maximum failure criterions for both fiber and matrix

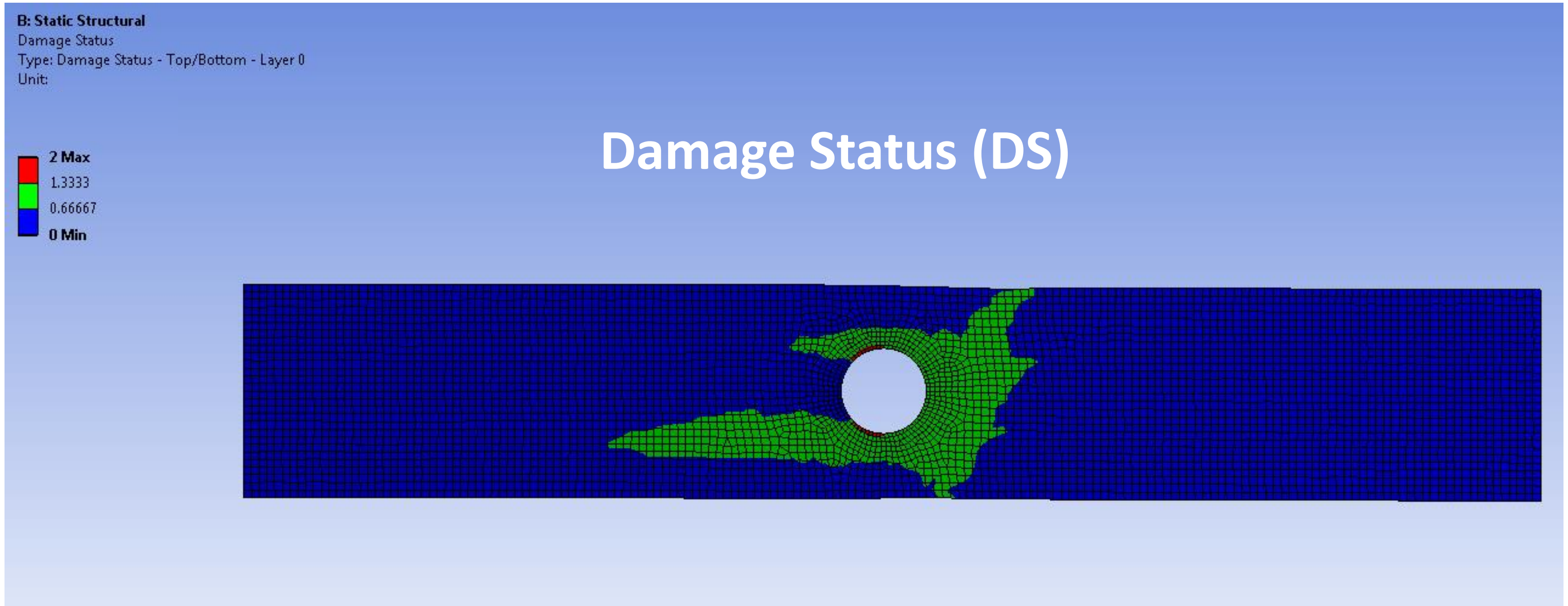
- Users can also access damage variables and observe damage progression through animation of the results.



click on Solution and select damage/failure criterion variables from the Solution Menu on the top

12. Modeling Progressive Damage

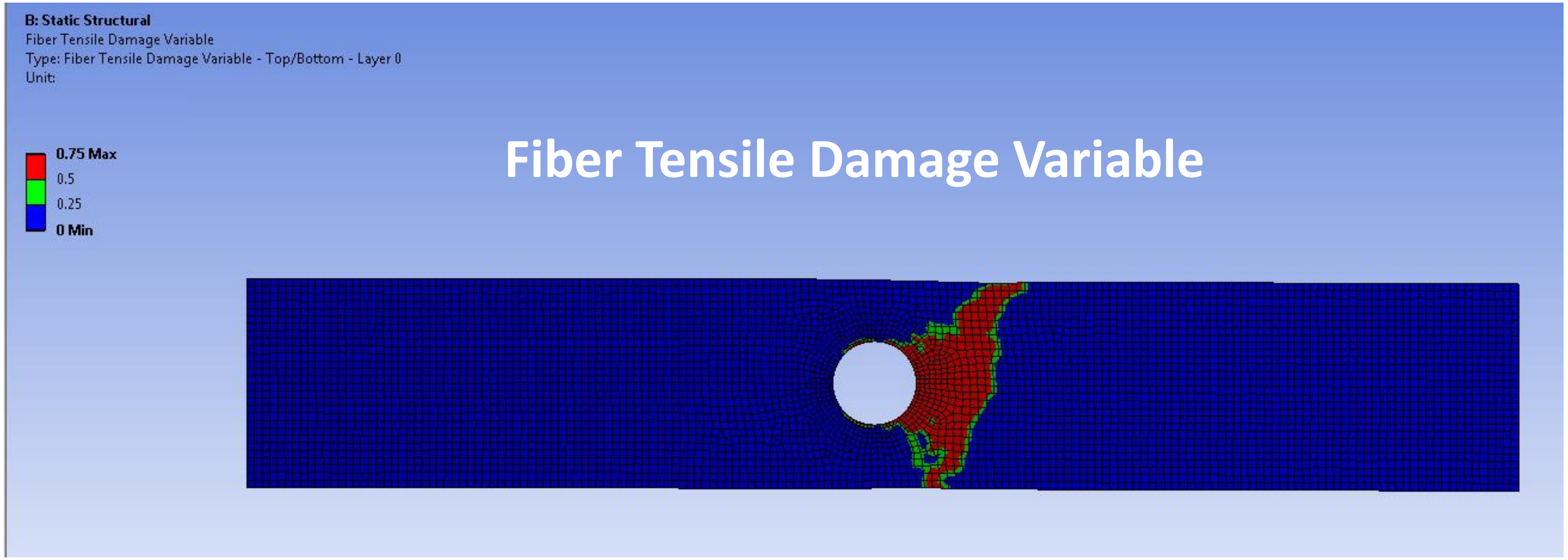
Results Analysis in Mechanical



- 3 color regions. Undamaged portion in blue (DS=0), partially damaged in green (DS=1), completely damaged in red (DS=2)
- The legend scale on the left has been reduced to show only 3 colors. By default Mechanical would shows a multiple colors band.

12. Modeling Progressive Damage

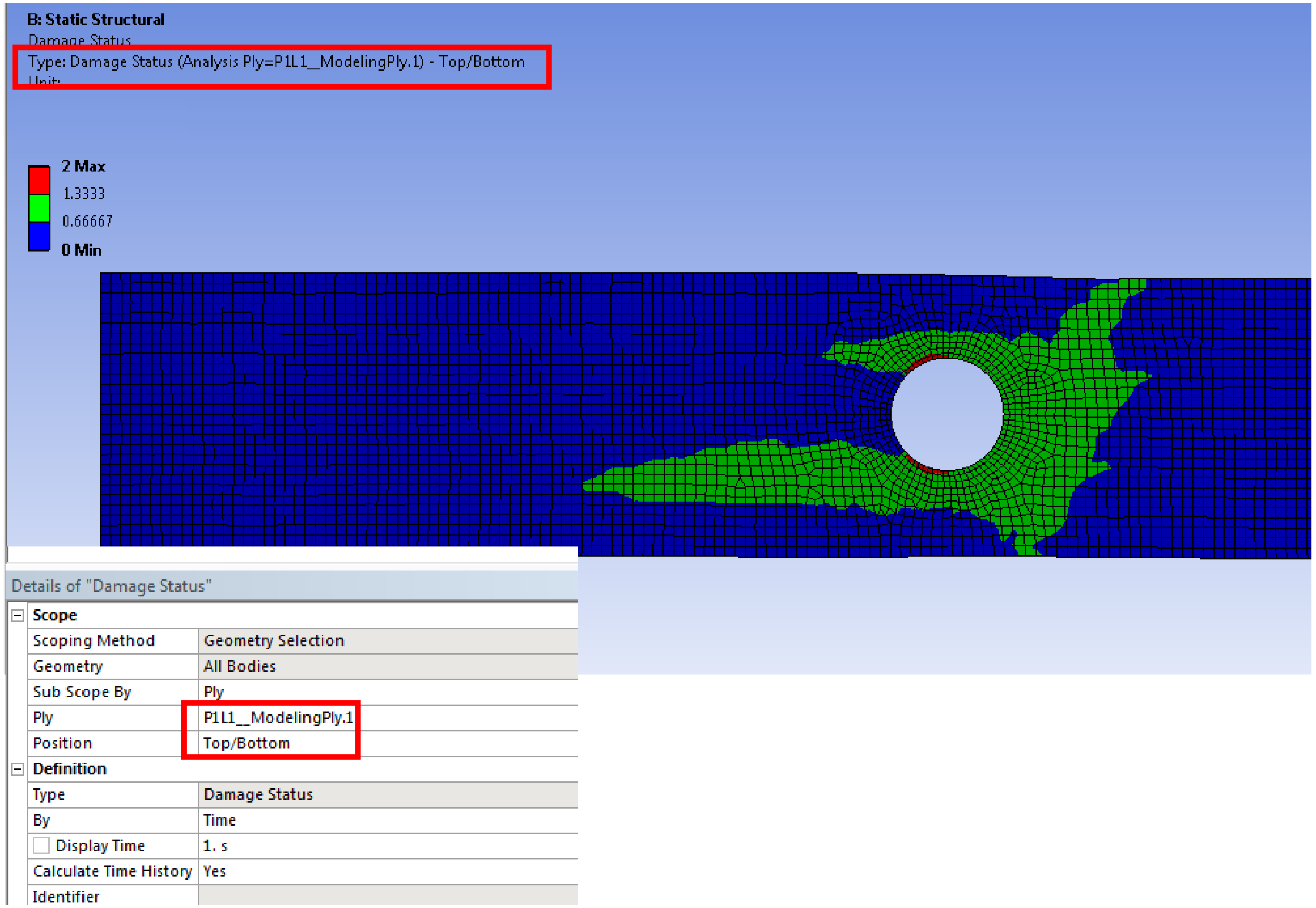
Results Analysis in Mechanical



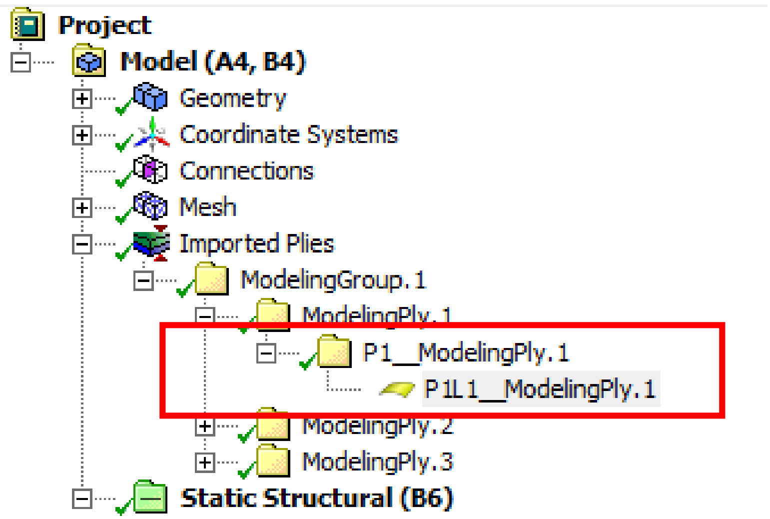
- In a small region the value of tensile damage is 0.75. In this region the fibers have failed and a damage factor of 0.75 is applied to the element stiffnesses. The other values in the plot are interpolation values computed by Mechanical to smoothen the results.

12. Modeling Progressive Damage

Results Analysis in Mechanical



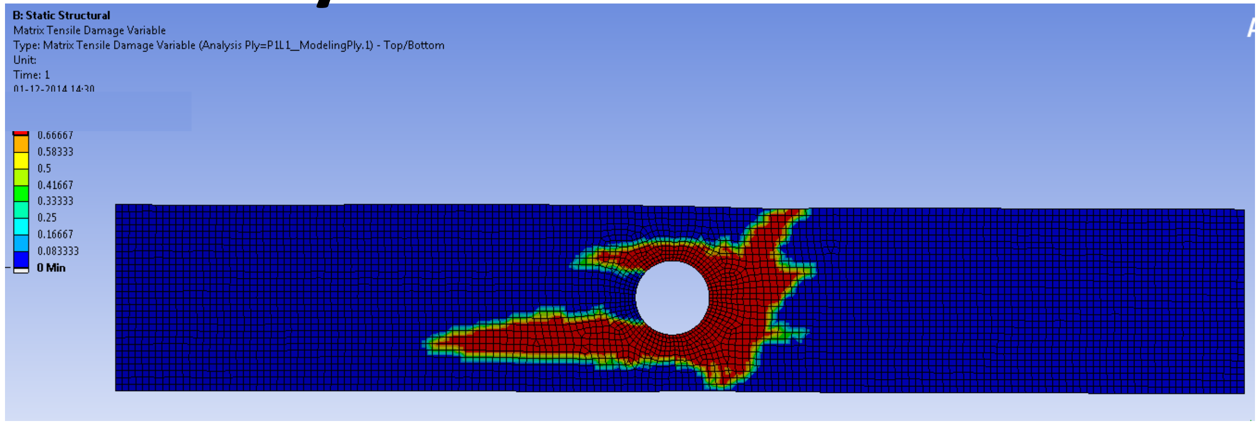
- Visualize the damage/failure criterion on individual layers or plies
- Select "Analysis Ply" from the Imported Plies object in the tree



12. Modeling Progressive Damage

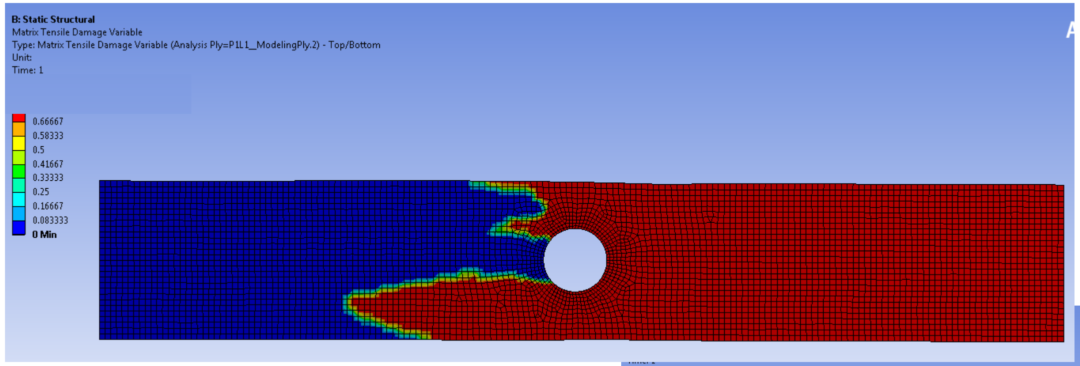
Results Analysis in Mechanical

Ply 1

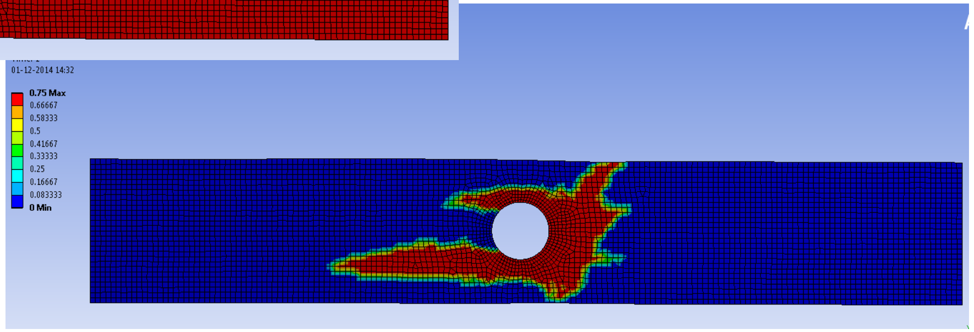


Viewing results on different plies

Ply 2

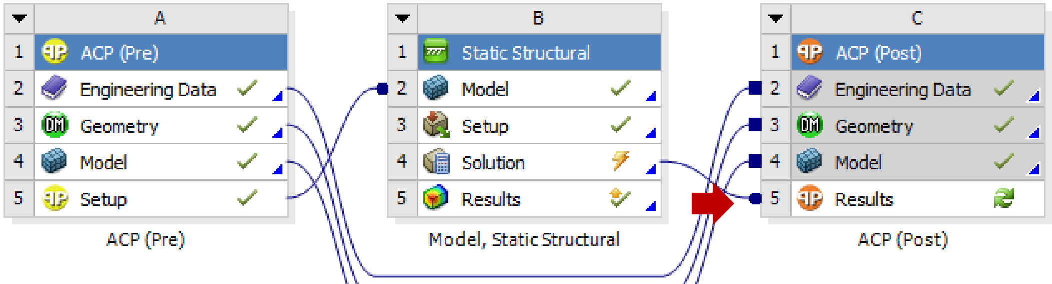


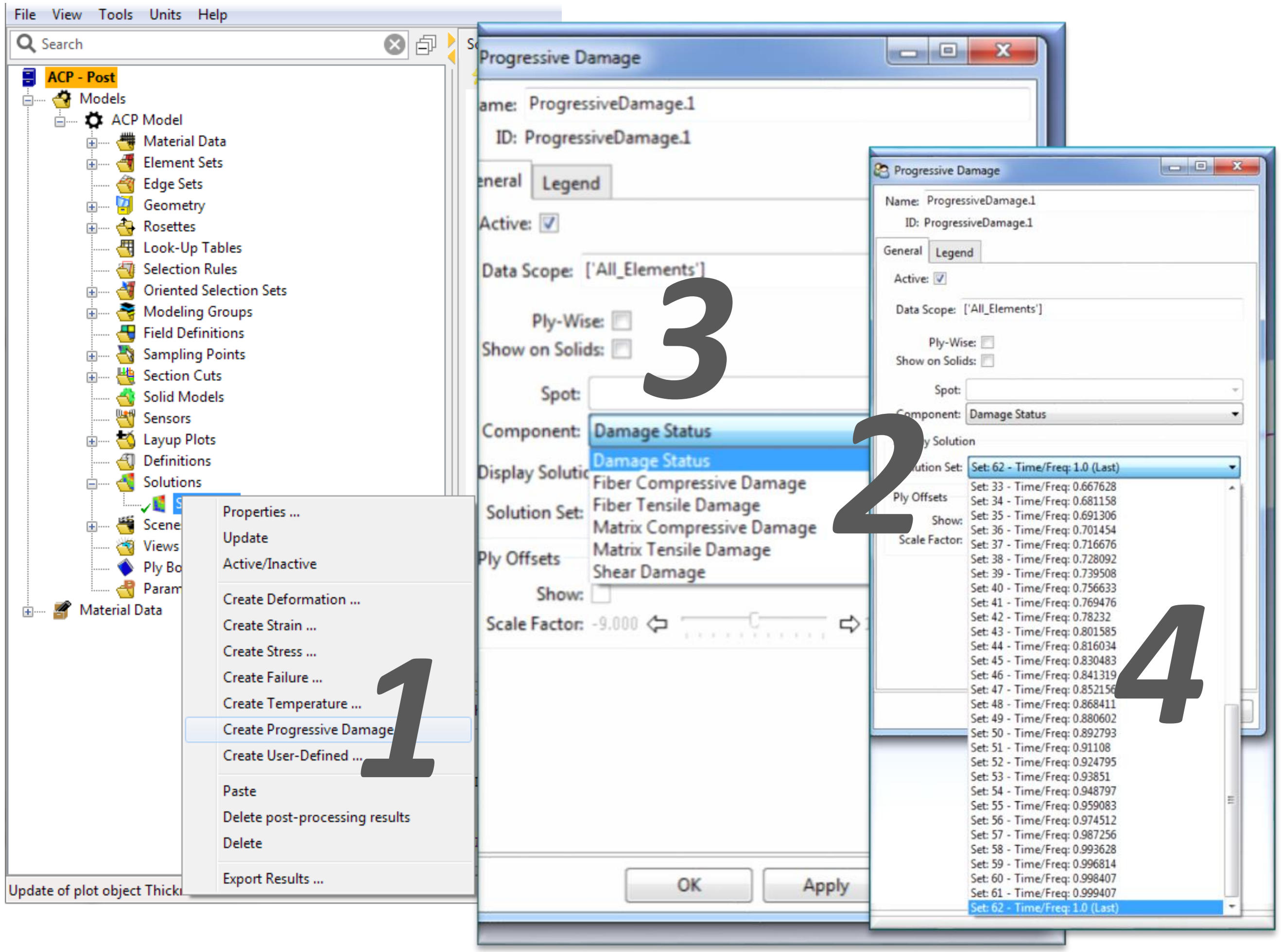
Ply 3



12. Modeling Progressive Damage

Results Analysis in ACP (Post)





1. RMB on “Solution 1” and select “Create Progressive Damage”

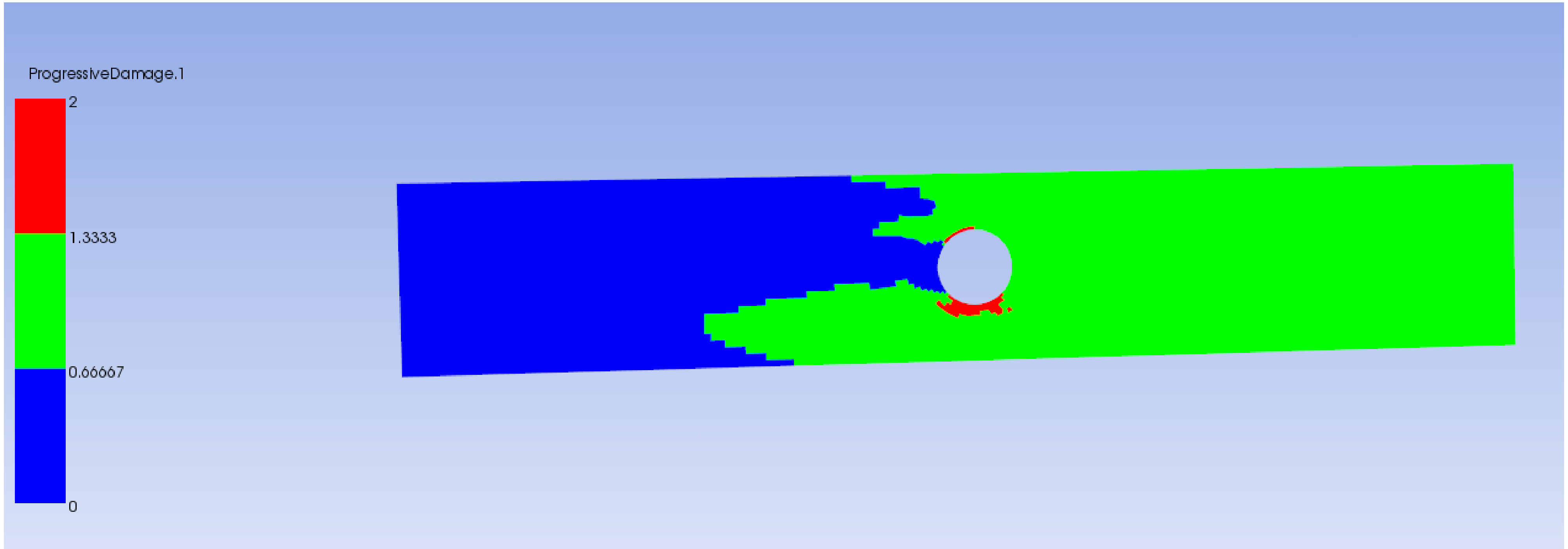
2. In the Progressive Damage panel select damage variables to post process

3. By default the results display only the maximum value of damage status for each element. Switch to Ply-Wise to see damage variables for individual ply

4. Select solution sets to evaluate damage variables at different time/sub steps

12. Modeling Progressive Damage

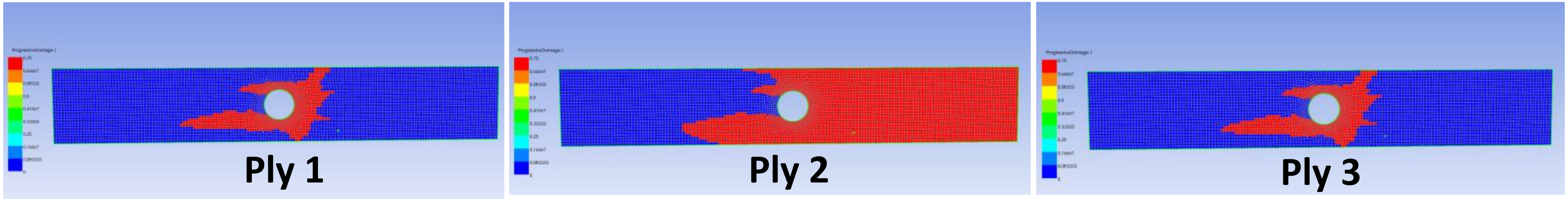
Results Analysis in ACP (Post)

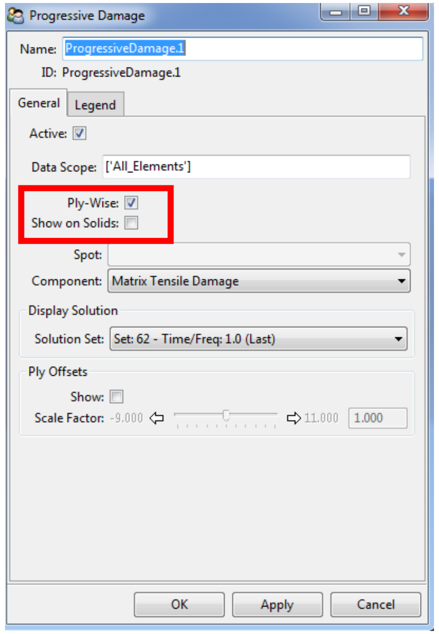


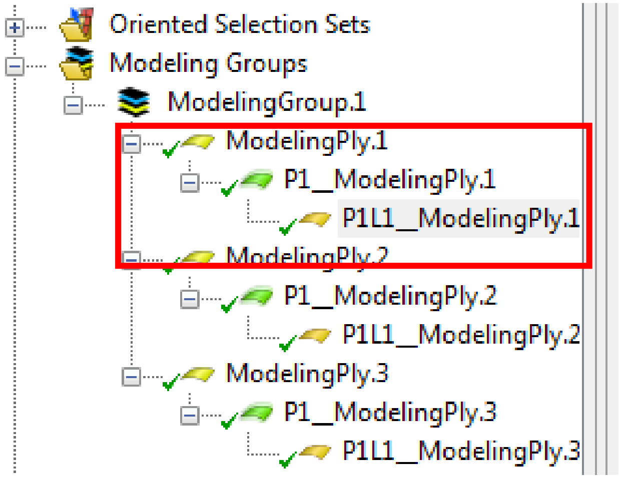
- This plot is different than the Damage Status plot in Mechanical (slide 10): ACP-Post displays the maximum damage status across all the integration point associated with an element while Mechanical displays the Damage status of the top/bottom layer

12. Modeling Progressive Damage

Results Analysis in ACP (Post)







- Damage status on each individual ply with “Ply-wise” option selected
- These results are similar to Mechanical results (slide 13)

12. Modeling Progressive Damage

Summary

- Setup engineering data with progressive damage input
- Post process for progressive damage in Mechanical and ACP-Post