Part 1

A recent client had significant manufacturing issues with large composite parts that are bonded together. This series of articles will focus on the options available to develop a tool set for both new design and even more challenging repairs during manufacturing.

Typically, extensive testing is required to validate analytical tools. Finding schedule and budget to support the endeavor is a significant hurdle for engineering projects. Fortunately, there are intelligent short cuts that need to be part of any managers' plan.

This week, I will be presenting short discussions on how I've found adequate paths around that which contributed significantly to the overall success of the program. As always, reach out if I can help save schedule, cost and provide value.

Our discussion for this topic revolves around the work from NASA (TM-201922010). This memo provides specific challenges faced with bonded joints and the issues with completing analysis in designs that utilize them.

NASA/TM—2019-220210

Comparison of Design Tools for Stress Analysis of Adhesively Bonded Joints

Part 2

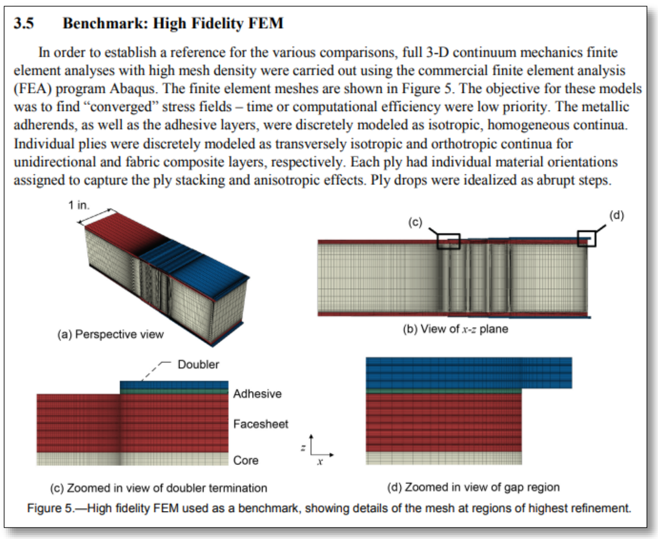
In NASA/TM-2019-2202 it provides insight into the analysis tools currently common and available. Specifically:

1. Hypersizer
2. Joint Element Designer (JED)
3. Parametric Continuum Solid Shell (CSS) FE
4. The Benchmark – ABAQUS High Fidelity FEM

As we dive into the details, please keep in mind I am goal oriented and these articles are written to achieve a specific task at hand. I won't be explaining the math behind the models but more importantly how can we use these data to get to the finish line of certification. In many cases, analysis is pushed due to previous unsuccessful efforts from other projects and a PM that found little value to the cost and schedule hit.

I have chosen to use the Altair® suite and can't say enough good things about that decision. This also means I am using Radioss for an explicit solver and for 2D efforts, ESA Comp. In the TM, several joint configurations are posed. I will be focusing on case 4 (sandwich panel) with a double lap or strap configuration. An image of the benchmark ABAQUS model in the TM is shown below.

With a high fidelity FEM, the stress concentrations can be visualized. This is still possible with the 2D analysis efforts and in most cases is preferred. Both approaches utilized together provide a powerful ability to understand the error associated with both methods.


3.5 Benchmark: High Fidelity FEM

In order to establish a reference for the various comparisons, full 3-D continuum mechanics finite element analyses with high mesh density were carried out using the commercial finite element analysis (FEA) program Abaqus. The finite element meshes are shown in Figure 5. The objective for these models was to find "converged" stress fields – time or computational efficiency were low priority. The metallic adherends, as well as the adhesive layers, were discretely modeled as isotropic, homogeneous continua. Individual plies were discretely modeled as transversely isotropic and orthotropic continua for unidirectional and fabric composite layers, respectively. Each ply had individual material orientations assigned to capture the ply stacking and anisotropic effects. Ply drops were idealized as abrupt steps.

Figure 5.—High fidelity FEM used as a benchmark, showing details of the mesh at regions of highest refinement.

Part 3

Finally, we are getting to the good stuff. We've identified the actions and how we will address the modeling.

Step 1: Replicate the 2D modeling from the TM previously described in both Part 1 and Part 2 of this series and compare to the results of the ESA Comp analysis to that of the TM. In this case we are looking for the failure stress within the bondline.

Step 2: With the process validated, we need to bound the error associated with the analysis. This is done with both the 2d modeling and the ABAQUS high fidelity FEM. In our case, this will be with Radioss.

This is also where we accept that all analysis models are estimates: some good and some not. We don't need the analysis to be exact: only that we can adjust for the error. This is also where we know our margin of safety policy. What minimum margin are we willing to accept in critical areas that have a high degree of variability. Keep in mind composites and bonded joints are anything but perfectly uniform. This isn't a machined part. Voids and other discontinuities are always present.

The images below show both the results from the TM (left hand side) and similar results from ESA Comp (right hand side). Both assume a bonded joint that has a double strap or symmetry about the thickness. The image of each joint represented in the models is provided at the top of each plot for bond line stresses.

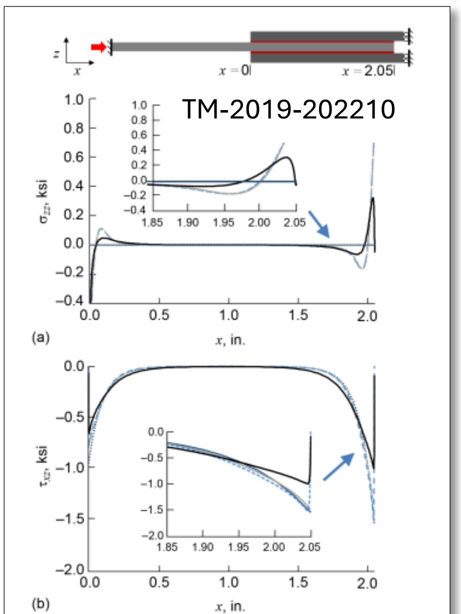
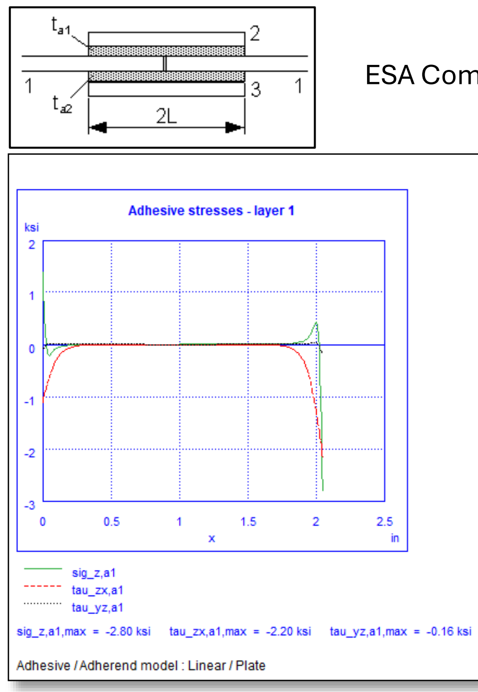


Figure 8.—Peel (a) and shear (b) stresses along the top adhesive centerline of Case 2.



Using the high fidelity FEM in the TM for a correction factor, we know that there is a 40% over reported stress within the bondline of the 2D model. This alone creates a significant value to future manufacturing defects or non-conformances. The time savings for small 2D models is critical.