Integrated Workflow for Analyzing Composite Gas Turbine Components

James Zhang[1], Li Zheng[2], Peter de Diego[1], Bart Thomson[1]
Mahesh Turaga[3], Deepak Goyal[3], Etienne Ardouin[3]
[1]GE Power & Water
[2]GE Global Research Center
[3]Dassault Systèmes

Abstract: *Advanced composite materials are being used in an increasing number of gas turbine engine products where their unique combination of properties such as low density, high strength and fatigue resistance help to increase performance. High temperature composites are currently being investigated in many components of gas turbines where load and high temperature capability are critical. Development of efficient, accurate modeling methodologies is critical. Compared to conventional composite structures, turbine composite parts can be thick and of complex geometries, which requires specialized modeling techniques. In addition, lamina level (i.e., ply level) results are required. This presentation focuses on two methods to create a ply-level finite element model using the integrated Dassault Systèmes (DS) product suite. This includes CATIA Composites Part Design, Composites Link, and Composites Modeler for Abaqus and Abaqus. Two ply-level methods will be presented: first, a ply-by-ply method in which each ply is separately discretized; and second, a solid-fill method in which multiple plies are represented within one layered solid element. Steps including geometry creation, ply creation, draping simulation, ply inflation and ply level FEA are discussed using a simple, non-proprietary example. In addition, composite local inhomogeneity modeling using DS tool suites will be briefly presented. Challenges involved in the current process and the significant improvements obtained using this integrated process flow are discussed.*

Keywords: *Composites, Ceramic Matrix Composites, CMC, Abaqus, Abaqus Modeler for Abaqus, CMA, CATIA V5, CATIA CPD, CAD, FEA, Defects, Lamina, Ply, Gas turbine, Integrated, Workflow, Mesh trimming, Manufacturing, CEL, SFM, Solid Fill Method, Volume fraction, Inhomogeneity, Progressive Damage, CT Scan, Simpleware, Delamination*

1. Introduction

Hot Gas Path (HGP) turbine components have traditionally consisted of homogeneous metallic parts. Well-established methods for stress analysis and design limits are employed in design optimization in a straight-forward fashion. Composite HGP components, however, pose new challenges for analysis, design, and manufacturing. HGP components contain thick cross sections of complex geometries, within which both shear and bending deformation can be significant, and interlaminar stresses may be of fundamental importance.

Current analytical practice in this area includes a simplified representation of the composite by using laminate theory. The component is represented as a homogeneous, orthotropic material. Analyses can be performed, bulk laminate stresses can be calculated, and these results can be correlated to physical testing of actual parts to determine practical stress limits. While effective, this approach has several drawbacks: first, it is expensive, as it relies largely on testing of actual parts; second, it provides little insight into and optimization of the laminate architecture behavior; and third, it ignores the lamina-level and constituent-level character of the composite material.

To overcome these deficiencies, investigation on ply-level analysis methods using the DS tool suite was performed. Specifically, this includes CATIA for CAD-based operations and Abaqus for FEA-based operations. Taken as a whole, the process enables design engineers to build and manipulate ply-level FEA models, predict and optimize ply level stresses, and generate associated ply patterns for manufacturing. This paper will present two ply level analysis methods. First, the ply-by-ply method will be discussed. This approach explicitly discretizes each ply in the CAD and FEA models. Ply level details, including ply drops, are captured. This method takes relatively longer in modeling time, yet renders highly accurate ply-level stresses for detailed design study. Second, a solid (volume) fill method (SFM) will be presented. While this method also yields ply level stresses, plies are implicitly modeled as layers in a continuum element. As a consequence, some detail, such as ply drop effects, is lost. The method takes a relatively shorter modeling time, yet delivers results commiserate with needs in a preliminary, exploratory design environment. The theoretical background of the two methods, FEA implementation, and usage scope will all be addressed. In the final section, current efforts towards local inhomogeneity modeling and progressive damage will be summarized.

2. Ply-by-ply Method

The general workflow for the ply-by-ply method is illustrated using a composite dovetail example shown in Figure 1.

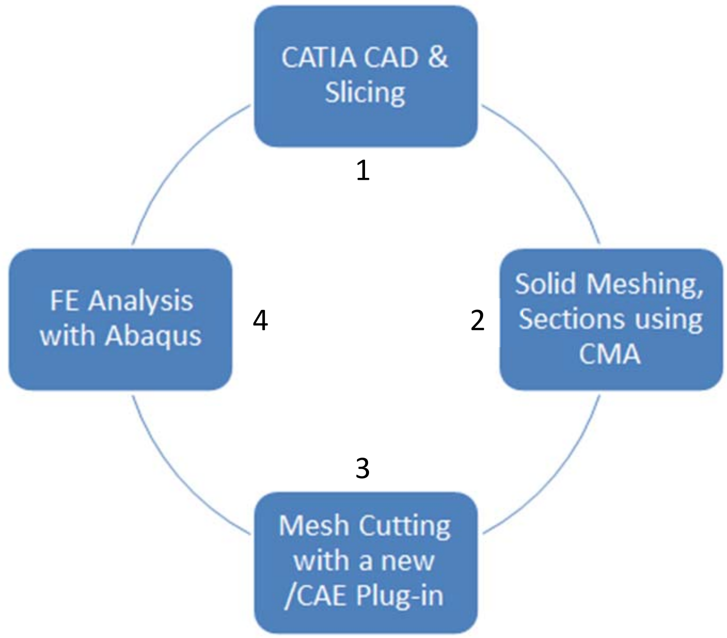

Figure 1. Ply-by-ply method workflow

Figure 1. General Workflow for the ply-by-ply method

Step 1: CATIA CAD & Slicing: Starting from CAD volumes of each ply pack, CATIA Composites Workbench determines the shape of each ply contour, while respecting the volume of each ply pack. This is done via advanced techniques such as Mid-Surface generation, with individual plies created from solid slicing, as shown in Figure 2. In this workflow, CATIA Meshing Workbench provides the tools needed to mesh each ply stack while respecting the ply boundaries. SIMULIA Composites Modeler for Abaqus (CMA) then transfers the Shell Mesh and associated layup information into Abaqus. This 3D model is also the reference used for manufacturing.

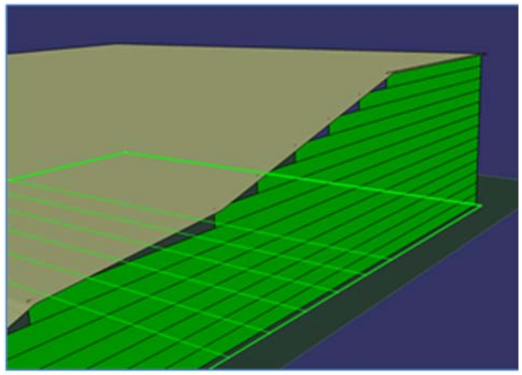

Figure 2. CATIA Solid slicing for ply boundary generation

Step 2: Solid Meshing, Sections using CMA: After the lay-up data and surface mesh have been created in CATIA workbench, the information is exported as a .layup file. The .layup file is then imported into Abaqus/CAE using CMA. Solid elements are generated by extrusion of the shell mesh for each layer and composite section. Element material orientations are automatically generated by CMA. Figure 3 shows a side-view of the fully meshed model, with color-coded sections. With this ply-by-ply method, each pack has several individually discretized laminas. Each of the packs has its own sets, sections, orientations and section assignments, which have been automatically generated by CMA. Some element overlap may occur in the CMA-generated solid element mesh. Mesh editing tools available in Abaqus/CAE give the user the ability to correct mesh imperfections.

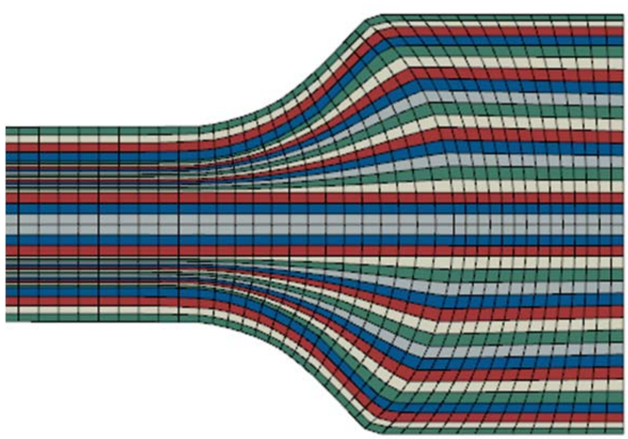

Figure 3: Cross-section for ply-by-ply mesh of dovetail pull-

Step 3: Mesh trimming plug-in: In Steps 1 and 2, mesh generation occurred on the composite component as it would be initially constructed. Subsequent operations may include machining. An Abaqus/CAE plug-in enables modification of the initial mesh relative to a user defined surface. This machining or cutting surface removes and/or modifies node locations such that the desired machined geometry is respected. Figure 4 shows the modification of the initial dovetail mesh relative to the machining surface.

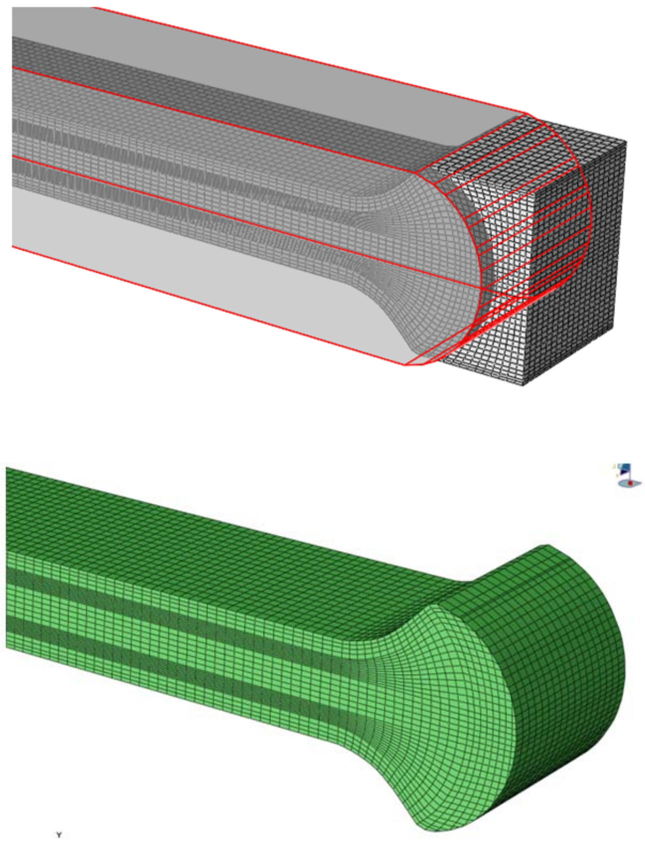

Figure 4. Meshing trimming to simulate machining process

Step 4: FE Model & Results: After machining, the component is ready for the next phase of the model construction. In analyses including assemblages of multiple machined parts, the part would be attached to other components. For this simple dovetail example, the component is ready for boundary condition and force application. FEA was conducted with Abaqus/Standard to simulate simple tension. Interlaminar tensile and shear stresses and other various ply level stress components were obtained. Figure 5 shows prediction of interlaminar shear stress (ILS) of the standard volume fiber plies (outer skin high volume fiber plies are not shown).

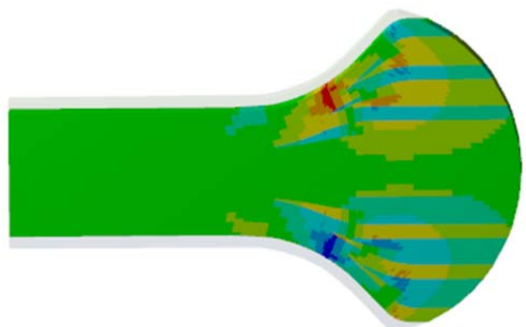

Figure 5. Ply level model and stress for a composite turbine

Creating a ply-by-ply model for more complicated geometry such as turbine blade follows the same procedure as described in step 1. Figure 6 shows a full ply –by-ply model and ply stress of a composite turbine blade.

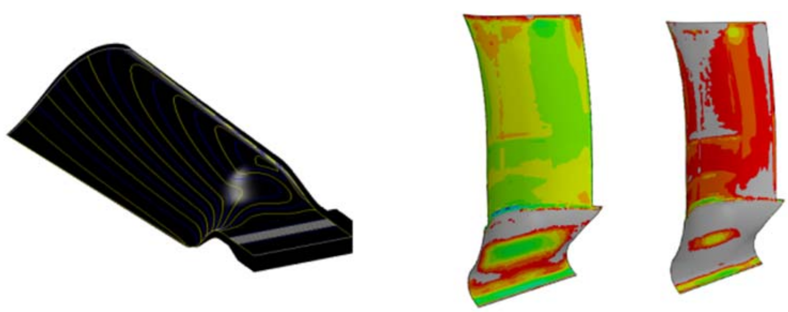

Figure 6. Ply level model and stress for a composite turbine

3. Ply level Solid-fill Method

The Solid-Fill Method (SFM) implicitly models each ply in the finite element model. The ply information is filled into appropriate elements based on geometry. The SFM is compared to laminate level and ply level analyses in Figure 7. For the laminate level, all elements through the cross section have the same property. For the ply-by-ply, each element corresponds to a specific ply. For the SFM, one element can include the properties of any number of plies. This can include fractions of plies as well. Element stiffness is then calculated relative to the volume fraction and location of each ply within the element.

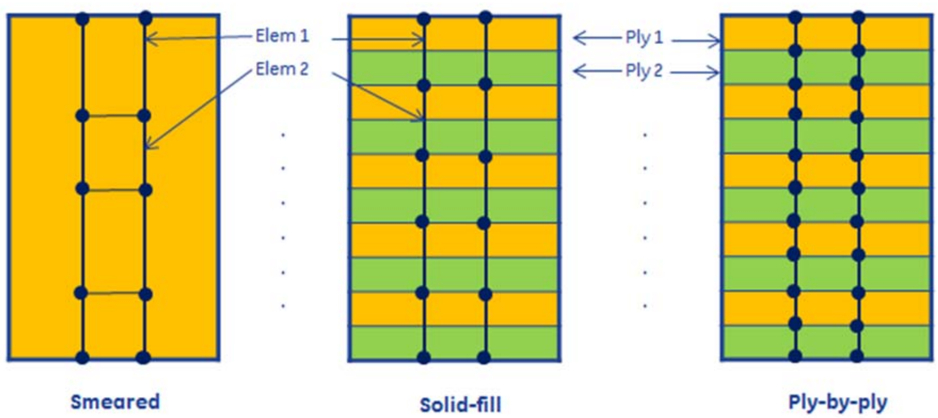

Figure 7 Smeared v.s Solid fill methods & ply-by-ply method

For in-plane properties, the element stiffness is calculated via Classical Lamination Theory (CLT). CLT evaluates the in-plane response of composite laminates given ply material properties and layup information. Out of plane shear and extension stiffnesses are modeled relative to the rule of mixtures. Further extension of this theory allows more accurate calculation of interlaminar stress [1, 2, 3, 4].

The SFM can be practiced analytically for simple geometry and simple boundary value problems. For complex geometry and loading conditions, such as blades, shrouds, and nozzles, finite element analysis is necessary. As with ply-by-ply, CMA provides the necessary link between CATIA and Abaqus. CMA fills the solid mesh with composite layup properties which either come from CATIA layup files or EXCEL layup tables. Material and section properties are therefore added in the correct orientation and correct position in the element. Thus, all needed information for a ply-level analysis is supplied. Figure 8 shows the general FEA process for SFM.

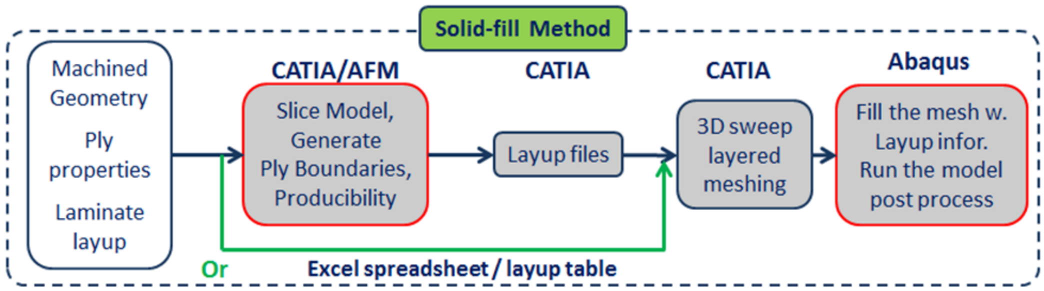

Figure 8 workflow of solid fill method

Briefly, the general steps for SFM are

Step 1. Prepare a clean and partitioned CAD model.
Step 2. Use CATIA composite Slicing method to generate plies and define the lamina, this layup file is associated with the solid in CATIA.
Step 3. Generate constant number layers of elements.
Step 4. Import CATIA/HYPERMESH mesh to Abaqus.
Step 5. Fill the solid mesh with layup properties and information.
Step 6. Assemble the meshes together in Abaqus.

Figure 9 shows the ply level in-plane stress in a composite blade using SFM

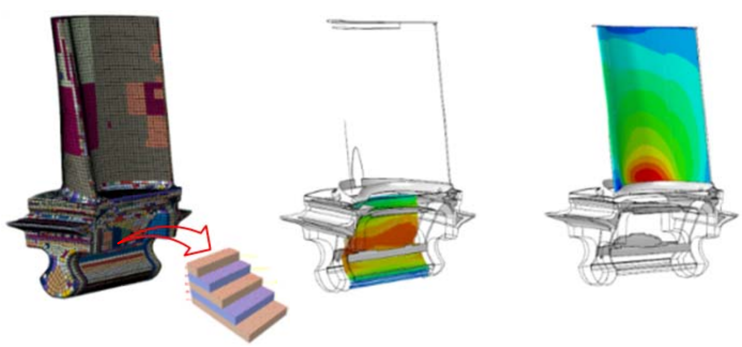

Figure 9. Composite turbine blade FEA model using Solid-fill method

4. Local inhomogeneity Modeling

Composite parts can contain certain inhomogeneity such as porosity, cracks and undesired particles from production processes. Those inhomogeneity may cause local stress concentrations and initiate pre-mature failure. It is important to identify those inhomogeneity and understand their effects. A methodology/workflow has been developed to create a finite element model that is mapped to inhomogeneity features. The goal is to understand the effect of the inhomogeneity on the composite part performance. This workflow has three steps.

Step 1: CT Scan data extracts inhomogeneity geometry using Simpleware software [5]: First, a digital photograph of a composite part cross section is taken at a desired resolution. Then, this image is imported into Simpleware using the *.tif file format. Outer surface geometry is identified via surface segmentation. Geometry segmentation identifies inhomogeneity geometries. The process is iteratively repeated until an acceptable contrast between the part surface geometry and inhomogeneity geometry is achieved. Finally, the resulting segmented geometry is exported as a ACIS .sat file.

Step 2: A novel Volume Fraction Tool approach in Abaqus/CAE captures inhomogeneity volume per element: In CAE, the original composite part without inhomogeneity is meshed with regular hexahedral elements. The volume fraction tool in CAE then operates on both the inhomogeneity geometry and the part mesh, performing a Boolean operation to determine which elements have inhomogeneity and the volume fraction associated with each inhomogeneity, on a per element basis. At the end of this process, each inhomogeneity is taken into account by degrading properties of the elements which geometrically coincide with inhomogeneity locations. The degradation magnitude can be scaled relative to the approximately measured inhomogeneity volume fraction.

Step 3: Material properties with degradation are mapped, and FE analysis is performed: A combination of *Elastic, dependencies and *Field, variable cards define degradation of material properties based on the volume fractions in the inhomogeneity region. Using this approach, when the inhomogeneity volume fraction is zero, non-degraded material properties of the composite are specified. As the inhomogeneity volume fraction increases, elastic moduli are decreased by an appropriate scale factor.

This process is shown in Figure 10.

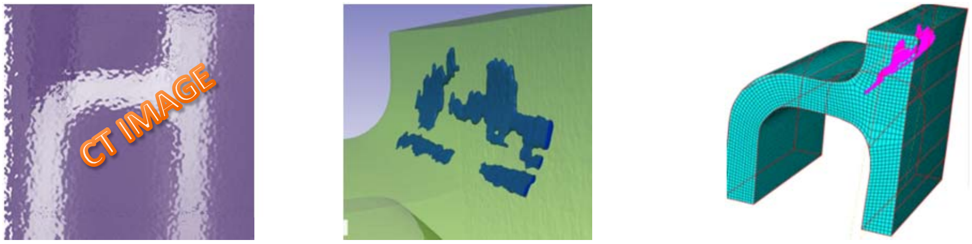

Figure 10. Inhomogeneity detection, identification and mapping to FEA model

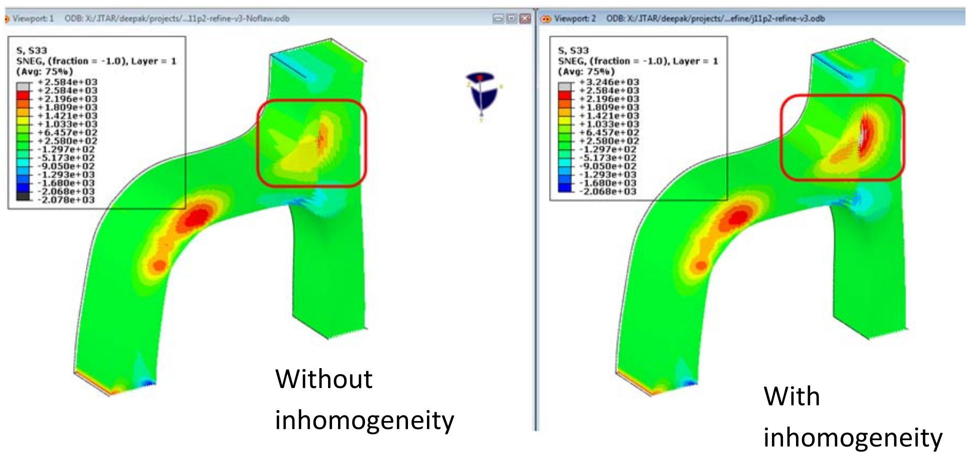

Figure 11. Effects of inhomogeneity (stress concentration in the inhomogeneity region)

FEA results of a composite part without and with inhomogeneity are shown in Figure 11. For this case, the presence of the inhomogeneity not only causes high stress concentrations, but also changes the load-path through the composite part.

Composite failure typically involves damage initiation and progression. Using DS suite one can model composite in-plane damage and progression, and progressive delamination in combination with ply level stress analysis. Efforts have been taken to model pinned composite panels pull test using progressive damage models and progressive delamination modeling of dovetail pull test specimens. This work is a possible candidate for future conference presentations.

5. Summary

The practice using DS tool to model composite parts in gas turbine engines has proven successful. DS composite tools and continued improved functions in CATIA and Abaqus enable design engineers to model the thick composite parts with complicated geometry in ply level for both design and analysis. The integrated composites workflow using CATIA CPD, Composites Link, Abaqus/CAE and CMA has significantly reduced the time required to design, optimize, and produce composite parts. As seen in sections 2 and 3, this workflow provides the ability to predict ply level stresses of turbine blade parts using Abaqus. The local inhomogeneity modeling methodology described in section 4 is critical to predicting the performance of composite parts that may contain small inhomogeneity during manufacturing. As a result, GEPW has the tools needed to define acceptable levels of defects for CMC parts.

6. Acknowledgement:

This material is based upon work supported by the US Department of Energy under Cooperative Agreement No. DEFC26-05NT42643.

7. Disclaimer:

8. References

1. Abaqus Analysis Users’ Manual, version 6.12
2. Fagiano, C., et al, “Domain Decomposition and Interlaminar Stress Recovery of Multilayer Composite Structures,” Composite Sci. and Tech.
3. Zhang, J., et al., "Failure Analysis of Adhesively Bonded Composite Joints, “46th AIAA/ASME/ASCE/AHS/ASC Structures, Structural Dynamics & Materials Conference, Austin, TX, April 2005.

4. Zhang, J., et al., "3D Stress Analysis of Adhesively Bonded Composite Joints," 46th AIAA/ASME/ASCE/AHS/ASC Structures, Structural Dynamics & Materials Conference, Austin, TX, April 2005.
5. Simpleware, http://www.simpleware.com/.