IOWA STATE UNIVERSITY
Center for Nondestructive Evaluation

Onur Rauf Bingol
Bryan Schiefelbein
Robert J. Grandin
Stephen D. Holland
Adarsh Krishnamurthy

A Framework for Solid Modeling and Structural Analysis of Composites with Defects

JUNE 2017

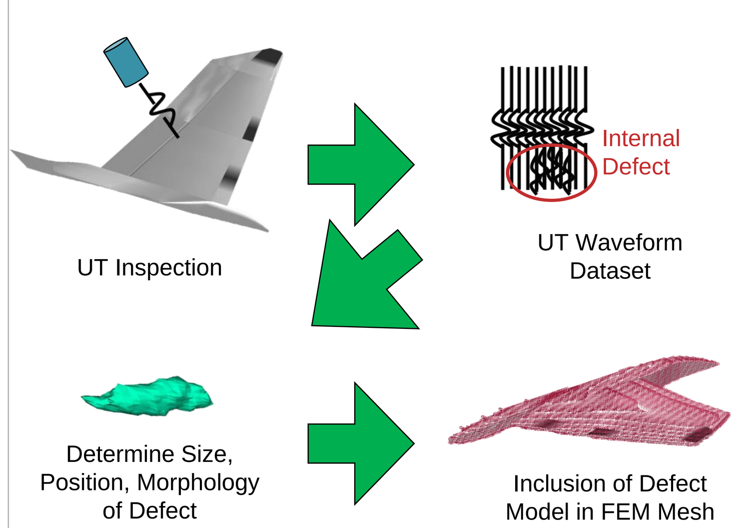
MOTIVATION

Fiber-reinforced composites are widely used in aerospace industry due to their combined properties of high strength and low weight. However, owing to their complex structure, it is difficult to assess the impact of manufacturing defects and service damage on their residual strength. While ultrasonic NDE is the preferred method to identify the presence of defects in composites, there are no accurate ways to model the damage and evaluate the structural integrity of composites.

We are developing **an automated framework** to incorporate flaws measured by ultrasonic testing (UT) into finite element (FE) models of composites with nontrivial geometry. These models will be useful for structural analysis to aid in assessing their residual life and make informed decisions regarding repairs.

METHODS

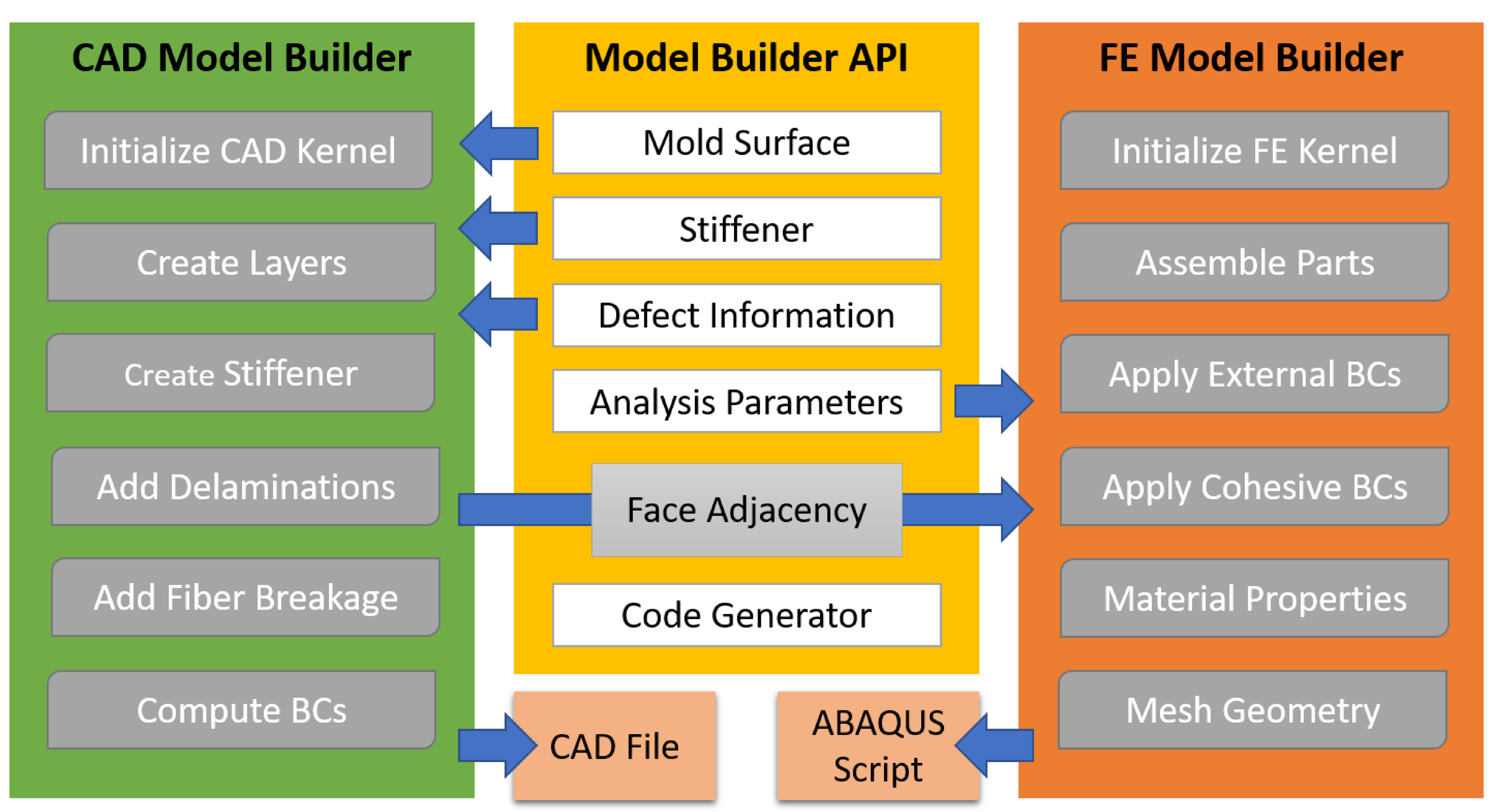
Incorporation of NDE Data



The figure illustrates the steps required to incorporate defects into finite element tools. There are three challenges in incorporating NDE flaw data into structural models of a part:

1. Mapping NDE indications onto their correct spatial locations within the part structure
2. Creating an accurate model of the defect based on the NDE response
3. Integrating the defect model(s) into the CAD / Finite Element mesh

The Automated Framework



The framework consists of 3 components:

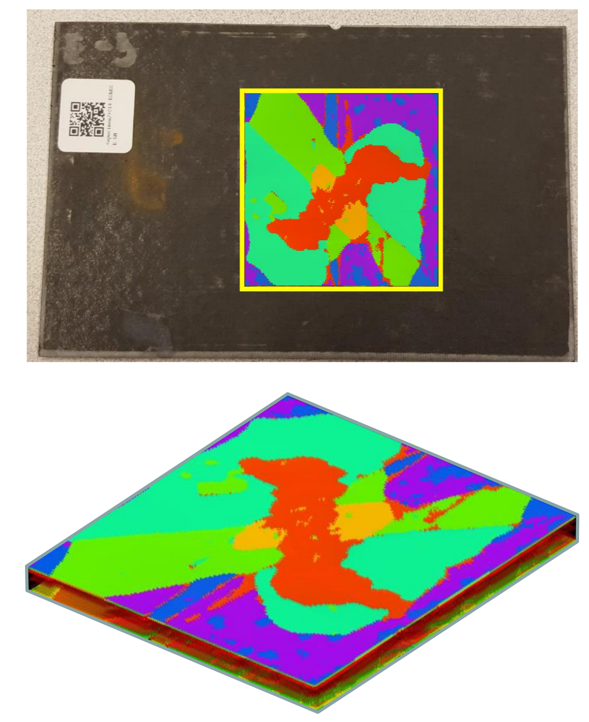
1. **CAD Model Builder** is responsible for generating the geometrical representations of the composite laminates. It also creates concrete representations of defects such as delaminations.
2. **FE Model Builder** is capable of using the geometrical representations to generate a finite element simulation model with appropriate FE mesh and boundary conditions.
3. **Model Builder API** is responsible for managing user inputs, converting data, and automatic code generation.

RESULTS

UT Sample and Delamination Example

The CFRP composite sample was provided by Dr. Stephen D. Holland and UT was performed by Dr. Ronald Roberts using bi-directional UT scan at CNDE.

The data is front-wall corrected by calculating the waveform envelope and aligning the first significant peak of this envelope. Hilbert Transform is applied to the corrected data to obtain a smooth waveform. The Hilbert-transformed data is scaled to unsigned char range (0 to 255) and the result is rendered using ray-casting on the GPU. The figure on the right shows the delaminated layer and the effect of indentation damage through the composite.

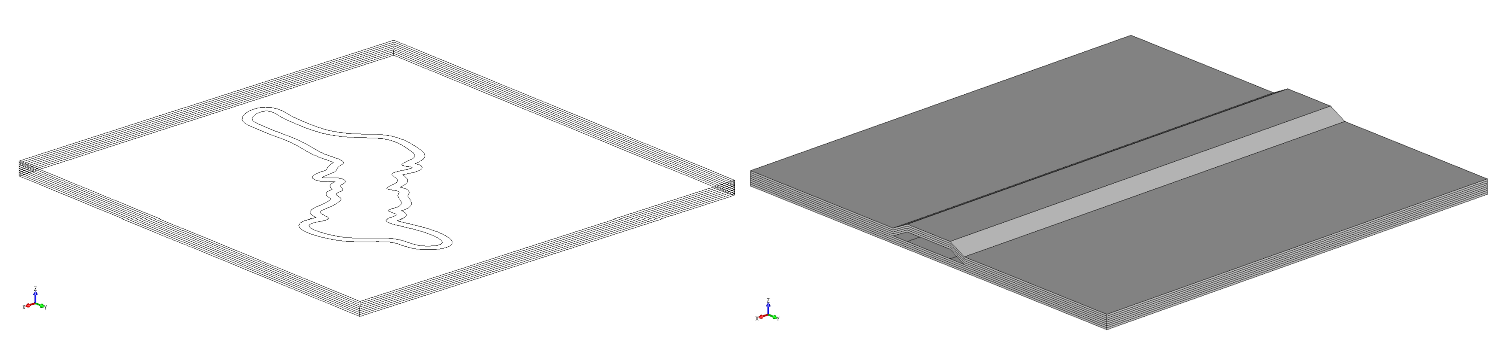


CAD Model Generation

The framework uses NURBS surfaces as molds to automatically generate the layered structure based on user-defined parameters. It can also generate composites with stiffeners.

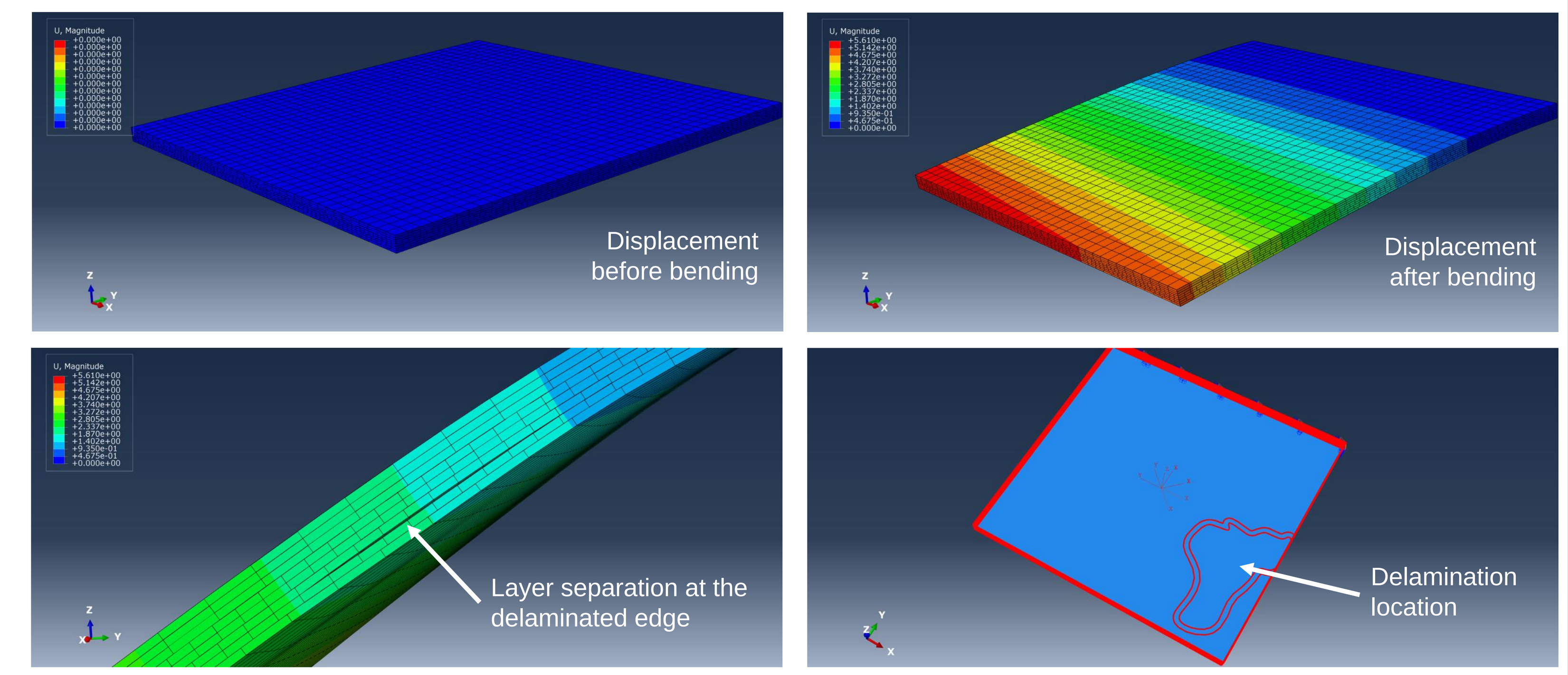
The framework is capable of incorporating delaminations between the layers of the composite model.

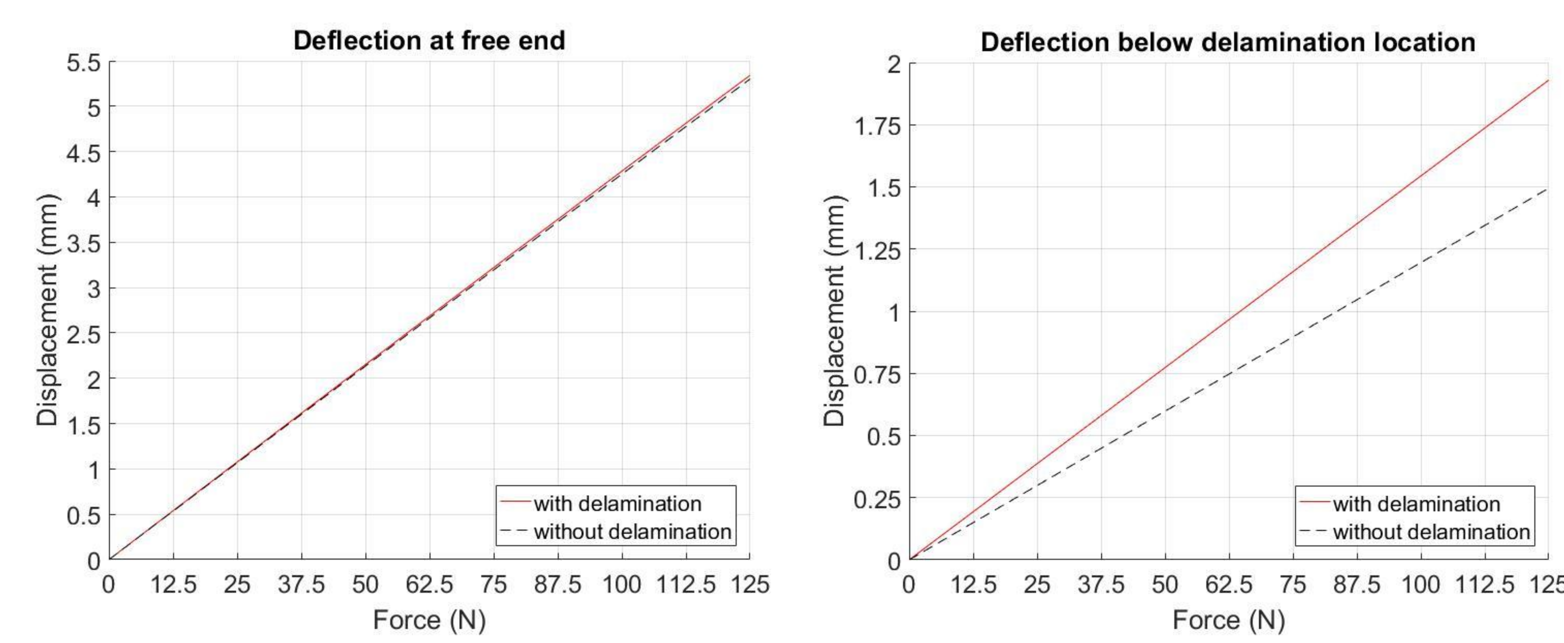
The figures below illustrate different examples of composited structures with delamination (left) and stiffener (right) generated by the framework



Finite Element Analysis of the Generated CAD Model

The FE model builder is responsible for reading the resulting CAD model and its calculated parameters and pass them to the FEA software, currently ABAQUS. FE model builder uses Cohesive Model for bonding composite laminae. Figures below illustrate the visualization of the delamination model in ABAQUS and the Force vs. Displacement graphs for both at the free end and at a location below the delamination. We can also observe the non-uniformity of the displacement field due to the delamination.



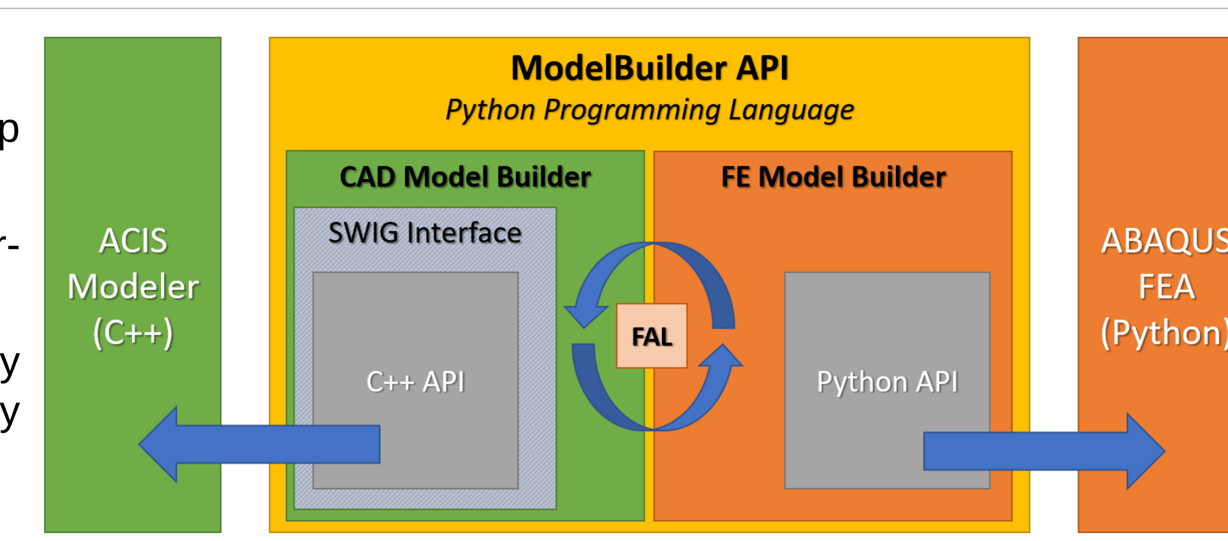


Framework Design

Users can interact with the framework using Python scripts or directly from the command line with the help of the ModelBuilder API.

The solid modeling kernel (ACIS) has C++ bindings, which is wrapped using a lightweight and user-friendly C++ API. This C++ API is then converted to Python using SWIG.

FE Model Builder provides ABAQUS FEA wrappers that can interface with the FEA using a user-friendly and robust interface. It interfaces with the data from the CAD Model Builder automatically without any additional user input.



FUTURE WORK

We have a working proof-of-concept for both CAD and FE model builders. These model builders will be improved to handle more complicated structures and various other types of delaminated surfaces.

ACKNOWLEDGEMENTS

We would like to thank William Winfree and Cheryl Rose from NASA for their valuable input and support for this project. We also would like to thank Ronald Roberts from CNDE at Iowa State University for the UT scans and the initial data. This project is a part of NASA Advanced Composites Project.

This work supported by National Aeronautics and Space Administration (NASA) under contract number NNL15AA12C through Center for Non-Destructive Evaluation (CNDE) at Iowa State University