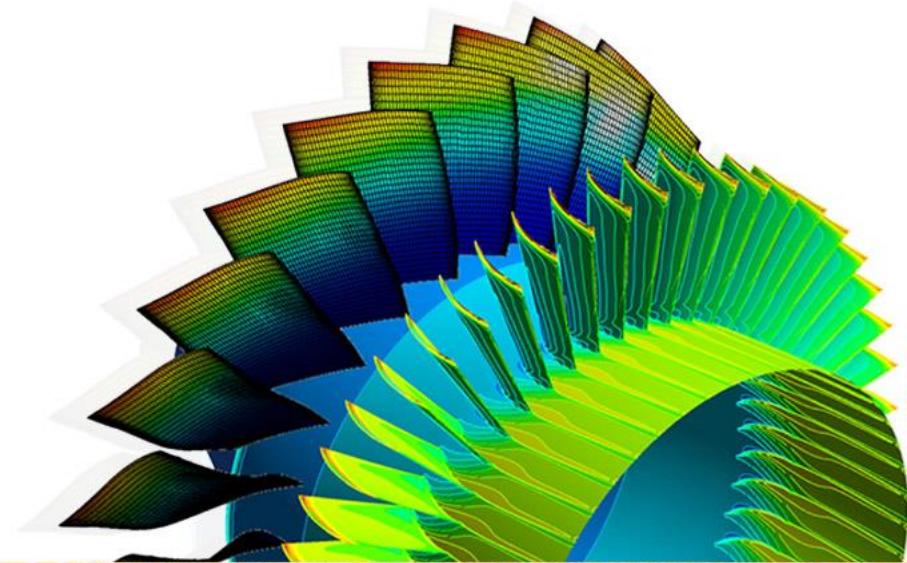# ANSYS Composite PrepPost 19.0
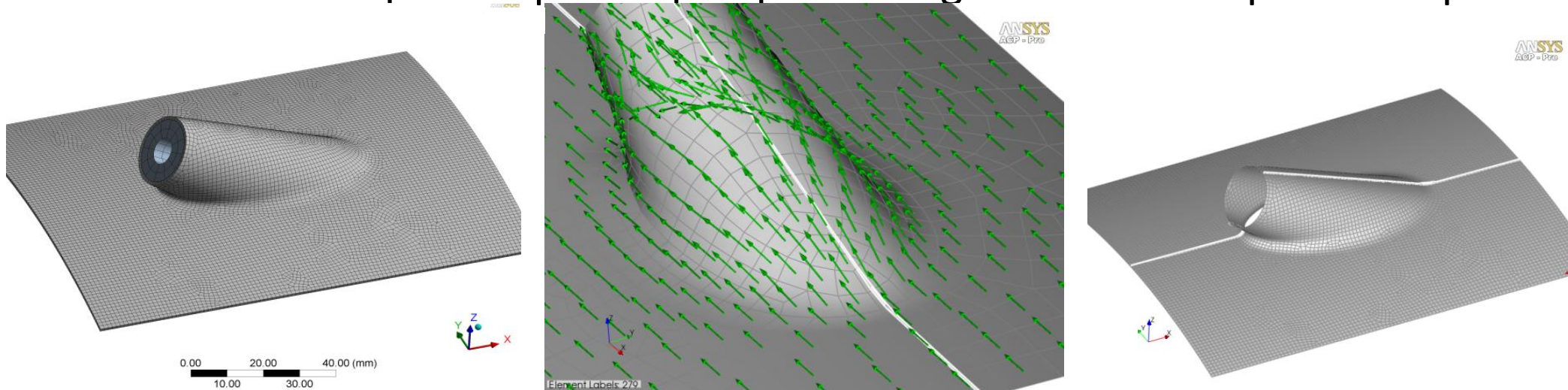
Module 2: ANSYS Composite PrepPost Demonstration

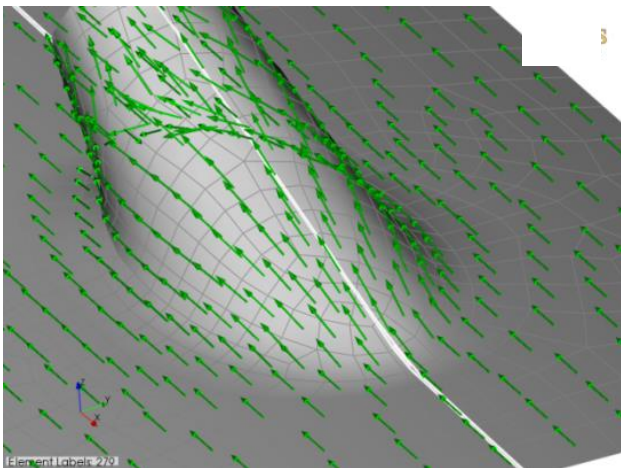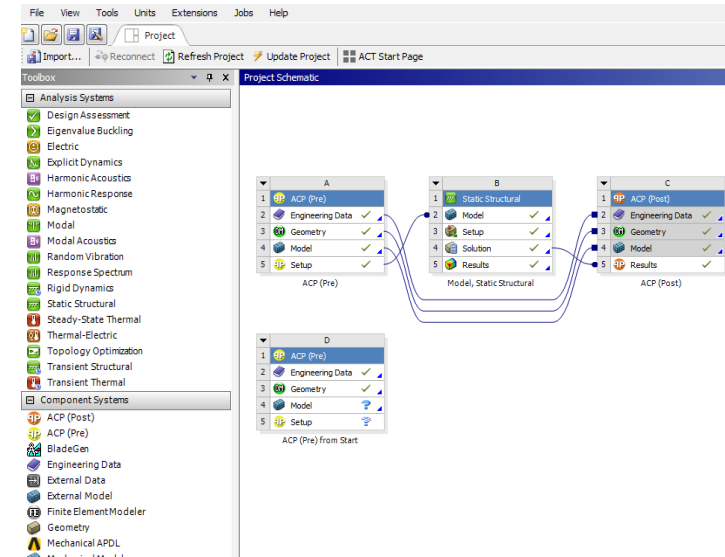# 2 ANSYS Composite PrepPost Introduction

Live Demonstration ANSYS Composite PrepPost

- Composite simulation workflow within ANSYS Workbench

- Defining a composite layup in ANSYS Composite PrepPost

- Loads and Boundary Conditions in ANSYS Mechanical

- Solution in ANSYS Mechanical

- Composite specific postprocessing in ANSYS Composite PrepPost

2 ANSYS Composite PrepPost Introduction

Live Demonstration ANSYS Composite PrepPost

- The simulation model used in the demonstration can be found in the folder:

  01_ACP_Demonstration

- Update the workflow from *ACP (Pre) SOLVED* to *Static Structural SOLVED* and *ACP (Post)* for the solved model including the composite layup (restore archive from .wbpz file, clean all generated data, update project)

- Start with *ACP (Pre) DEMO FROM START* to start modeling a new composite layup and for a new solution