

FEMAP ANALYSIS – READ ME

Disclaimers

- This code was designed for use in an academic capacity. Purpose of this code is to assist in the analysis of composite materials in the Siemens FEMAP program. This code was tested on a variety of composite geometries including simplistic geometry for validation purposes (panels and tubes) and complex geometry with complex loading scenarios (boat hull and stringer).
- The MATLAB code is currently designed to identify the maximum values for stress (X/Y/Z Normal Stress and XY/YZ/XZ Shear Stress), strain (X/Y/Z Normal Strain and XY/YZ/XZ Shear Strain) and deflection (X, Y, and Z directions) for all nodes, elements, and layers and consolidate this information into a simple table.
 - Some simple vernacular changes could allow the code to analyze for the minimum or average values.
- This code only outputs useful information and data if the FEMAP analysis simulation is setup and run correctly.
- This code only runs correctly if the table names match.