

Software

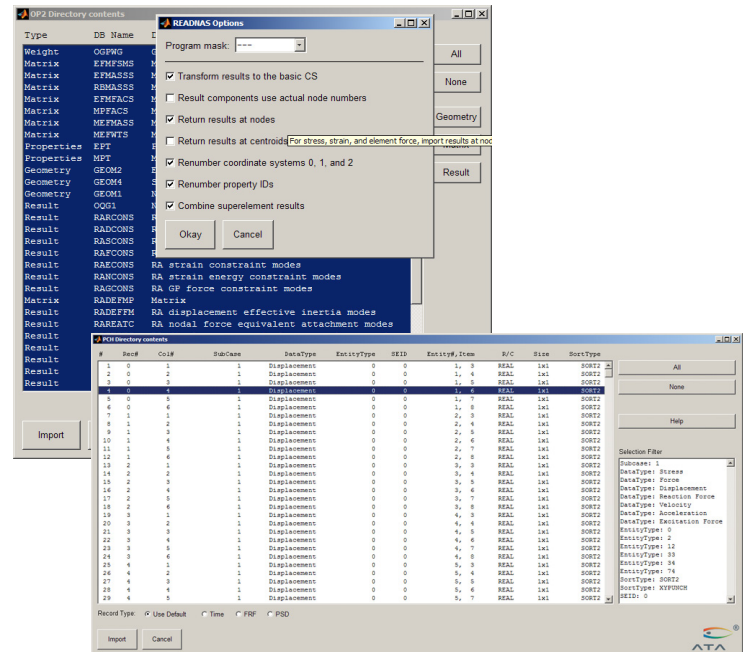
IMAT + FEA™ | IMAT Finite Element Analysis Interface

Overview

IMAT is a toolbox for use with MATLAB®, offered by ATA Engineering, Inc. (ATA). Designed for test and analysis engineers, it provides a framework for easily importing finite element (FE) simulation data and measured test data into the MATLAB environment to leverage the versatile programming and mathematical strengths of MATLAB. With the IMAT toolbox, test and simulation data such as mode shapes, time histories, spectra, and other functions can be viewed and manipulated in MATLAB while preserving all of the native data attributes.

IMAT+FEA extends the core capabilities of IMAT by providing significant additional functionality to manipulate simulation data. IMAT+FEA can directly access CAE data from many different analytical solvers, including Nastran, Abaqus, Femap, and I-deas. IMAT organizes this data into easy-to-use data types within MATLAB, allowing analysts to combine results, specify custom failure criteria, calculate margins, and otherwise evaluate CAE data using the power and flexibility of MATLAB.

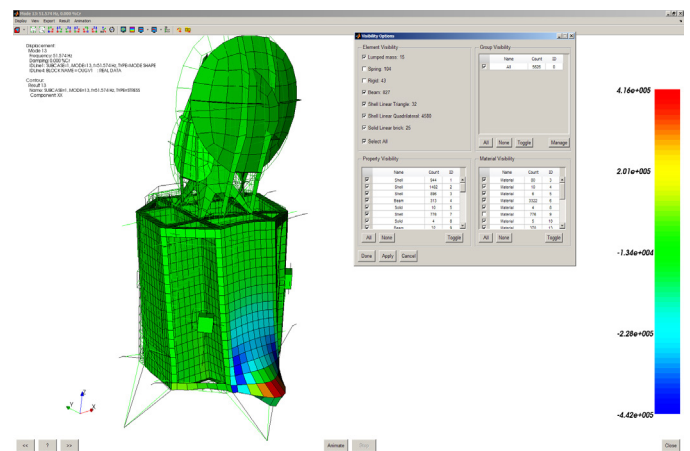
IMAT+FEA provides a powerful FEM, result, and coordinate trace visualization capability that utilizes the Visualization Toolkit developed by Kitware, Inc. This allows you to display very large datasets (greater than 1 million nodes) and visualize results on a deformed mesh with advanced visualization capabilities such as selective entity display (by element type, material, physical property, and/or group), transparency, and shaded or line display. Animate your shapes, interactively label elements and nodes, and “fly” through your FEM using VTK’s advanced dynamic translation and rotation in Perspective display mode.



- Selectively import data from OP2 or PCH files using the readnas utility in IMAT+FEA.

IMAT+FEA Key Features

- Nastran
 - Import bulk data (including DMI and DMIG), Output2, Output4, or Punch files
 - Select a subset of results to import
 - Import results into MATLAB in native (“raw”) OP2 format and use the provided datablock interpreter to translate the entire content of the results
 - Export to OP2 (then import into your CAE postprocessing tool)
 - Write bulk data entities such as DMIG, MPC, and some FEM entities such as coordinate systems, nodes, and several different element types and their associated physical and material properties
- Abaqus
 - Import xyData, step histories, and field data from ODB
 - Select a subset of results to import
- Femap
 - Supports Femap 11.3.x through 2022.2.x
 - Import select datasets from Femap Neutral file
 - Export to Femap Neutral file or directly to Femap via COM
 - Access the Femap COM API for bidirectional communication with Femap



- Display stress results on a deformed mesh, and selectively display FEM entities by type, group, or both with IMAT+FEA.

Specifications subject to change without notice. © 2023 ATA Engineering, Inc. IMAT is a trademark of ATA Engineering, Inc. I-deas and Femap are trademarks of Siemens Digital Industries Software. Abaqus is a trademark of Dassault Systèmes. MATLAB is a registered trademark of MathWorks.

© ATA Engineering, Inc. 2023

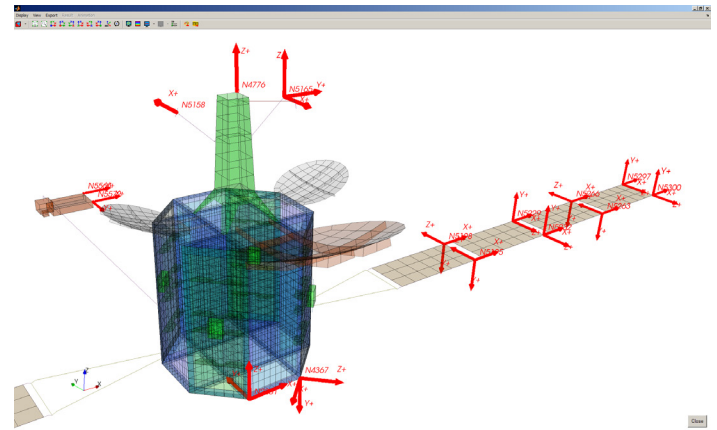
- Data Visualization
 - Display extremely large FEM datasets (millions of nodes)
 - Visualize deformed and contour results
 - Display coordinate traces

Hardware Platforms

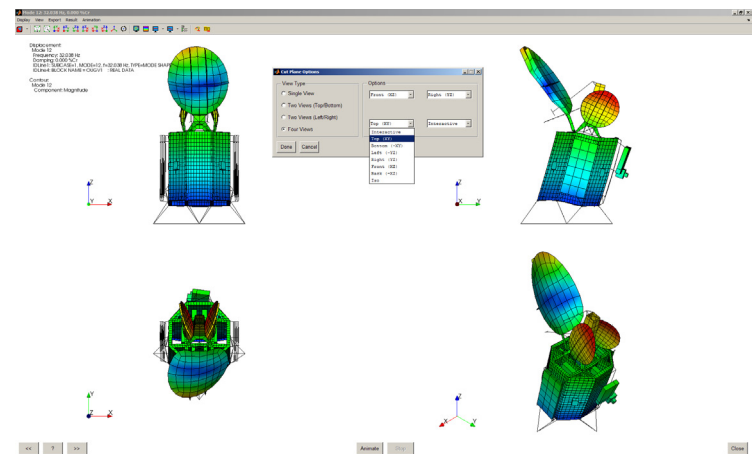
The IMAT toolbox was written primarily in the MATLAB language. Versions are available for 64-bit Windows, 64-bit Linux, and 64-bit Mac platforms. MATLAB R2020a (or higher) is a prerequisite for IMAT v8.0.

About ATA

ATA Engineering has more than 45 years of experience in structural dynamics analysis and testing, and is a world leader in test-analysis correlation and model updating. For more information please visit www.ata-e.com.



- Visualize coordinate trace locations and orientation on your FEM using VTK with IMAT+FEA.



- Animate your mode shapes with a multi-view capability using VTKPLOT.