


Introduction to Finite Element Modeler



Introduction to Finite Element Modeler

an ANSYS Tutorial



[Click Here to Start Tutorial](#)

© 2009 ANSYS, Inc. All rights reserved. ANSYS, Inc. Proprietary



Introduction to FE Modeler

Finite Element (FE) Modeler is used to import models and meshes from a variety of packages allowing you to analyze the physics/mesh and use the mesh to extract a parasolid geometry.

Topics Covered in this tutorial

- How to start FE Modeler
- Information obtained by FE Modeler
- Skin Components
- Geometry Synthesis
- Mesh Metrics
- Write Solver File

[Click Here to Continue](#)

© 2009 ANSYS, Inc. All rights reserved. ANSYS, Inc. Proprietary



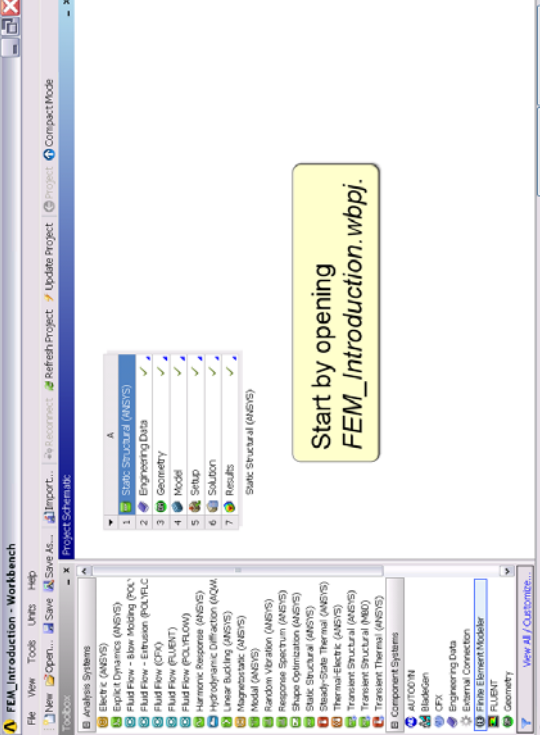
Introduction to FE Modeler

Prerequisites:

- Introduction to ANSYS Workbench
- Introduction to ANSYS DesignModeler
- Introduction to ANSYS Meshing

[Click Here to Continue](#)

© 2009 ANSYS, Inc. All rights reserved. ANSYS, Inc. Proprietary

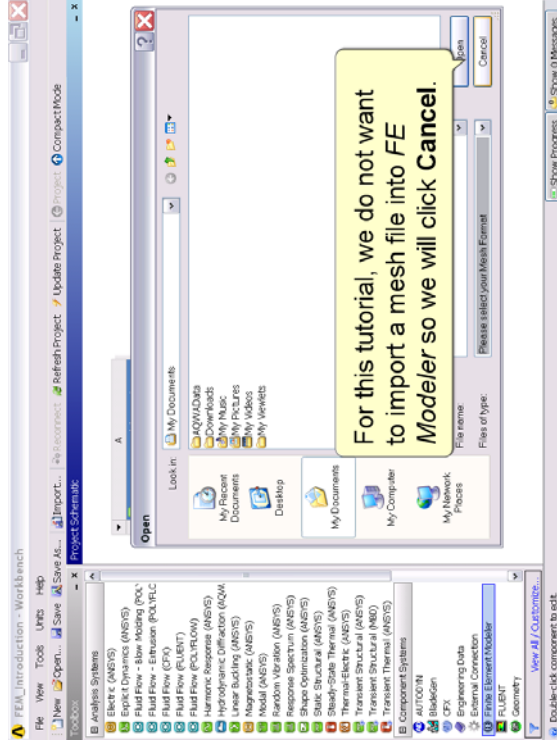
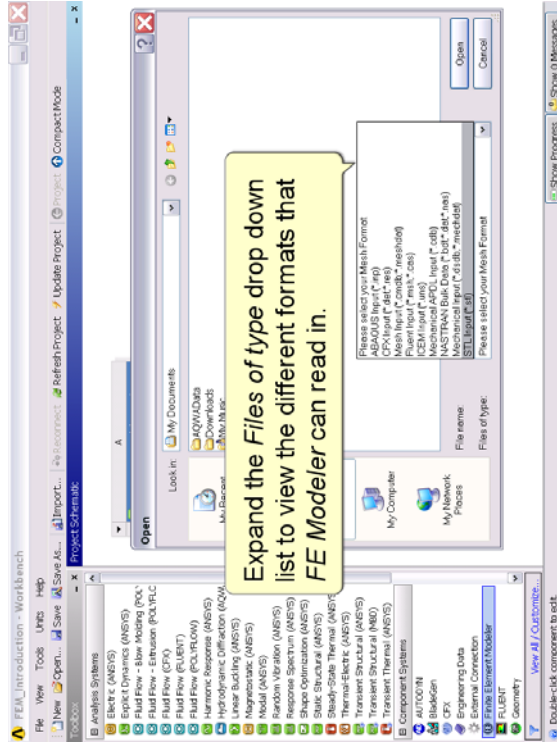
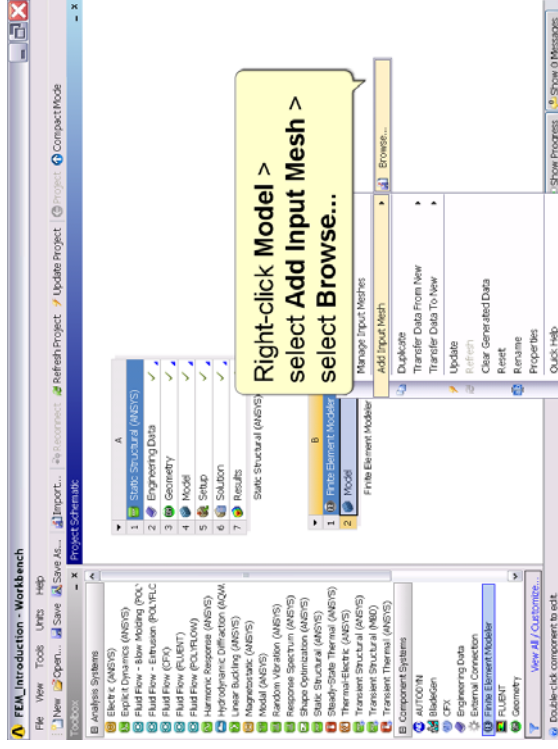
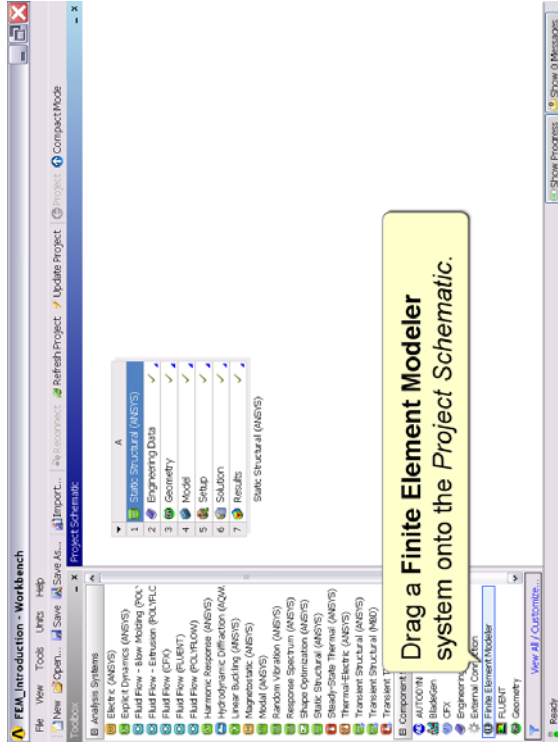


The screenshot shows the ANSYS Workbench interface. The 'Project Schematic' pane on the left lists the following steps: 1. Sub-Structural (ANSYS), 2. Engineering Data, 3. Geometry, 4. Model, 5. Setup, 6. Solution, 7. Results. The 'Analysis Systems' pane on the right lists various analysis types such as Explicit Dynamics (ANSYS), Fluid Flow - Slow Motion (FLUENT), Fluid Flow - Steady (FLUENT), Fluid Flow (FLUENT), Fluid Flow (POLYFLOW), Harmonic Response (ANSYS), Hydrodynamic Diffusion (AQUA), Linear Buckling (ANSYS), Modal (ANSYS), Random Vibration (ANSYS), Response Spectrum (ANSYS), Shape Optimization (ANSYS), Static Structural (ANSYS), Steady-State Thermal (ANSYS), Thermal-Electric (ANSYS), Transient Structural (ANSYS), and Transient Thermal (ANSYS). A yellow callout box with the text 'Start by opening FEM_Introduction.wbpj.' is overlaid on the interface.

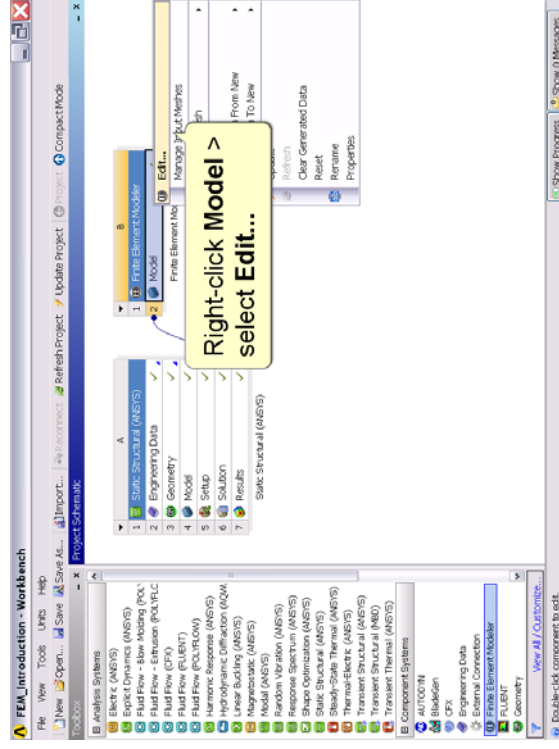
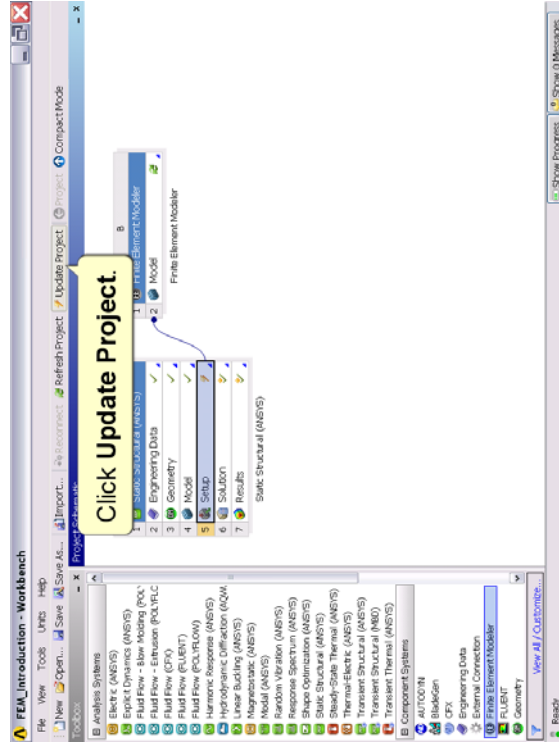
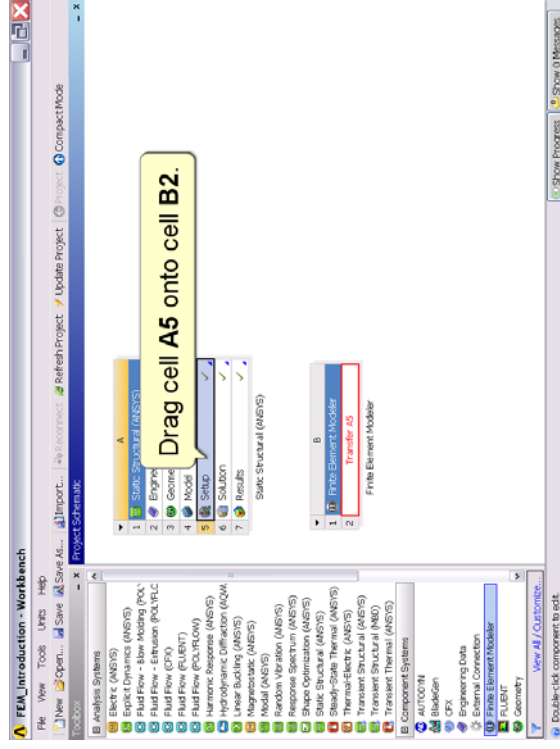
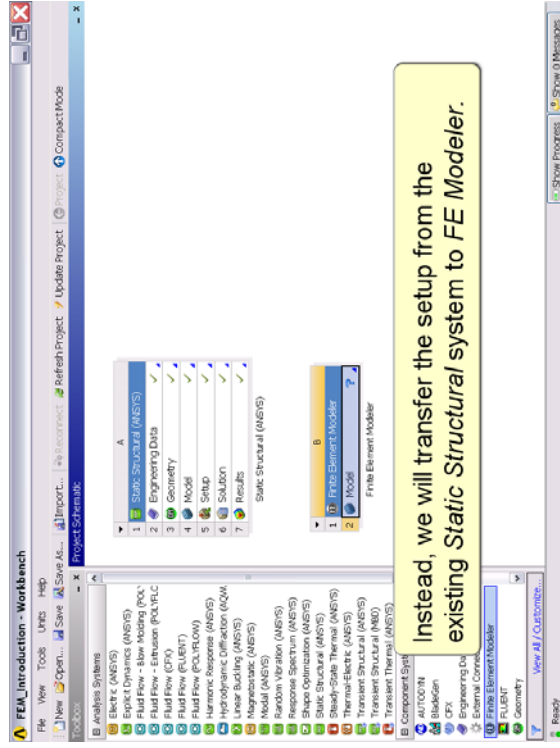
[Click Here to Continue](#)

© 2009 ANSYS, Inc. All rights reserved. ANSYS, Inc. Proprietary

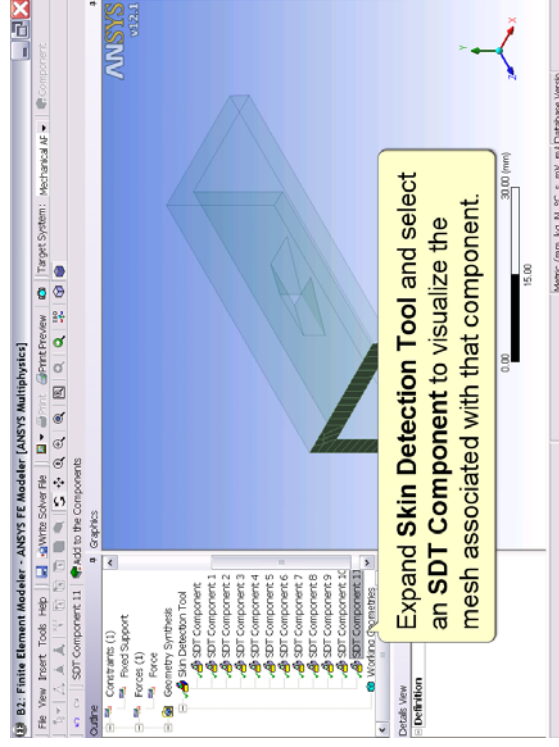
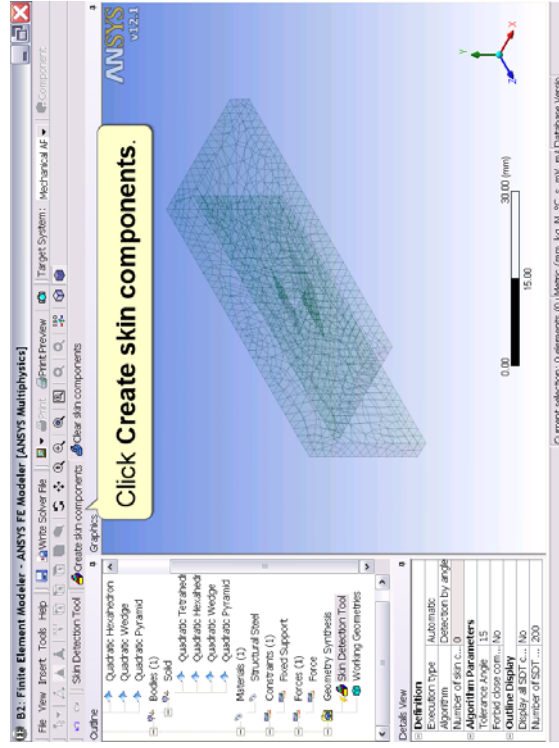
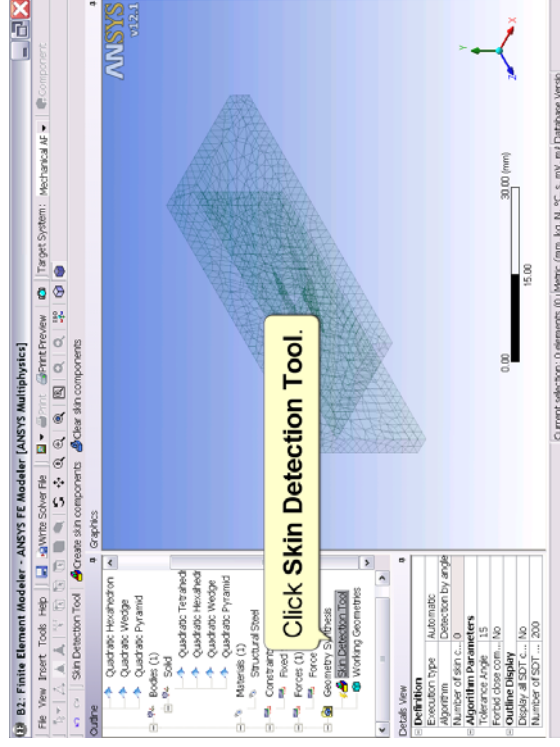
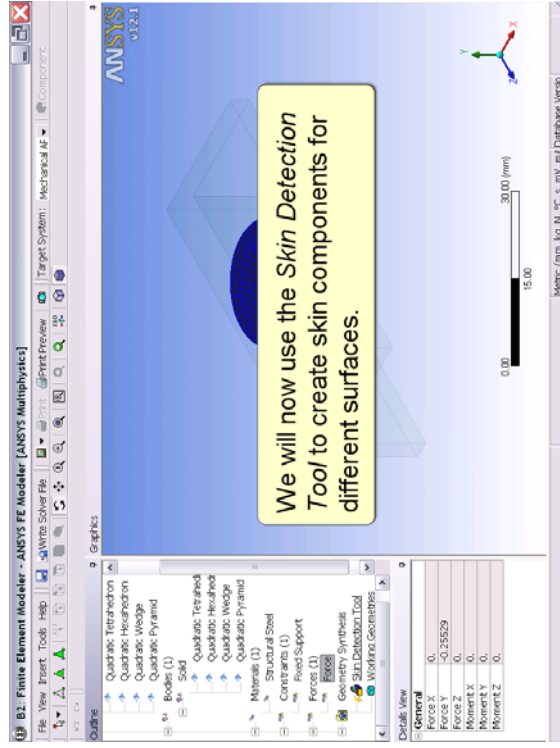
Introduction to Finite Element Modeler



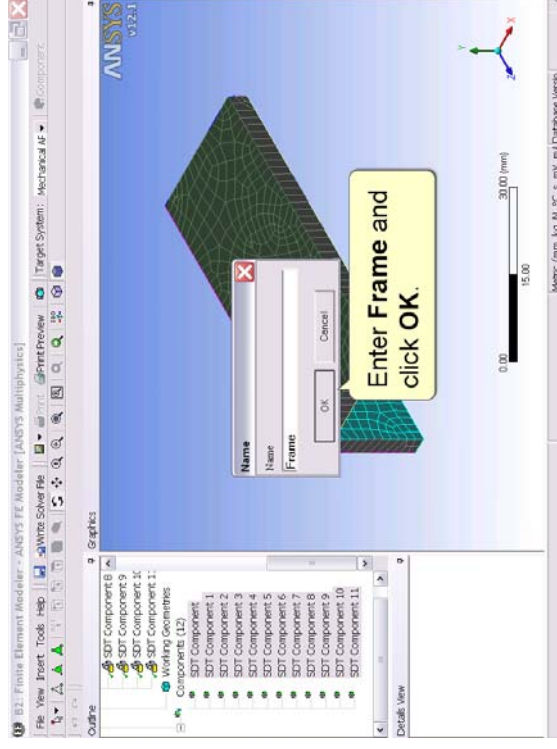
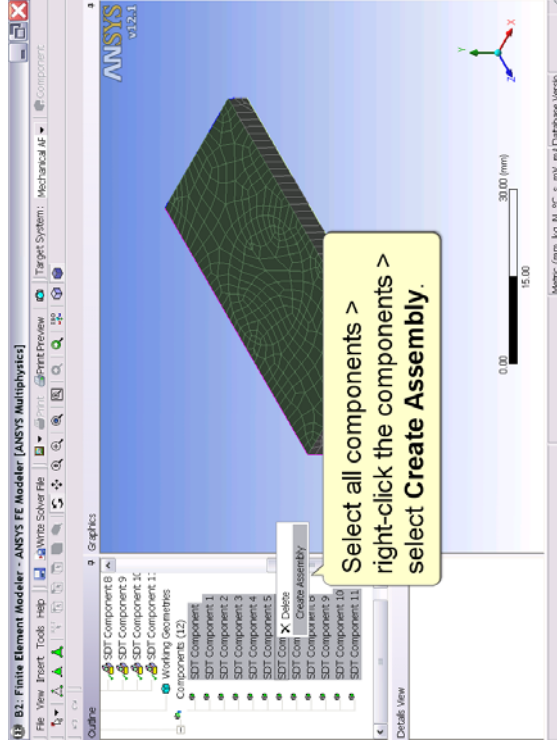
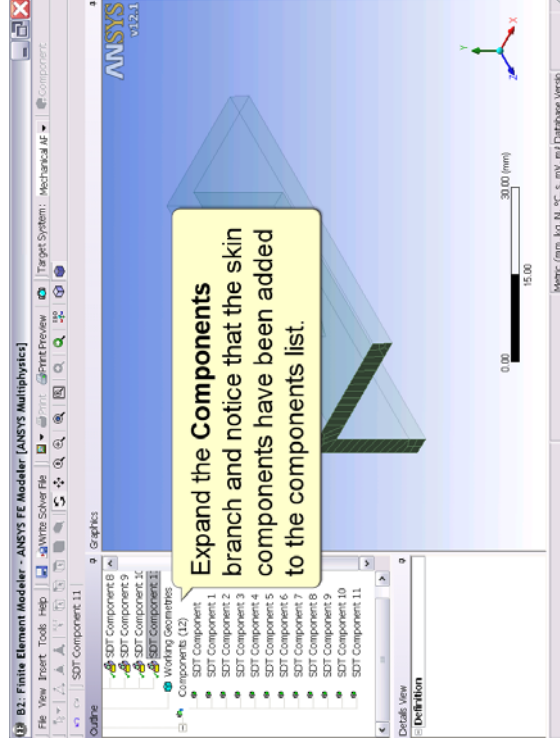
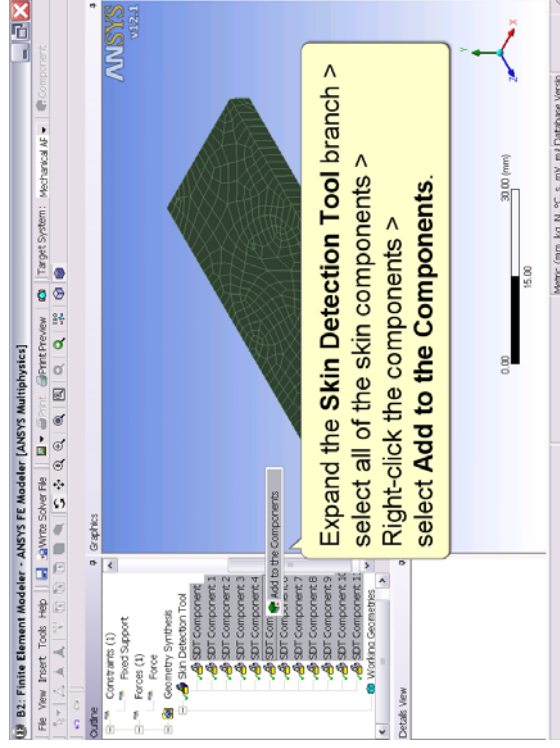
Introduction to Finite Element Modeler



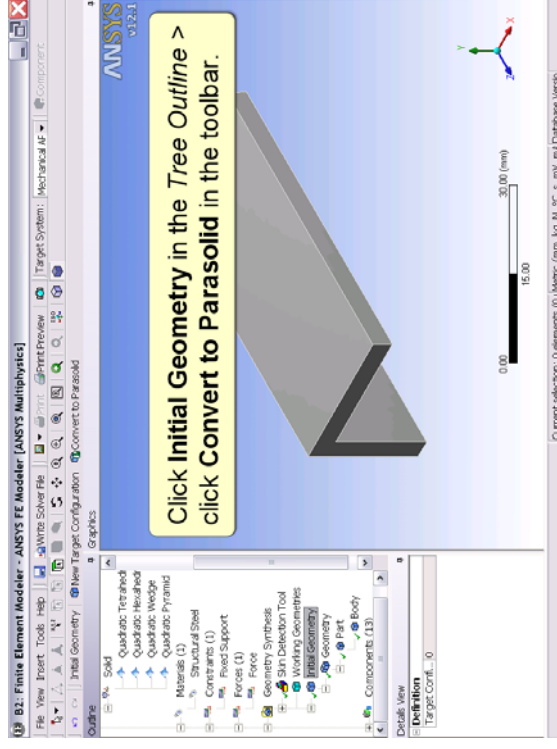
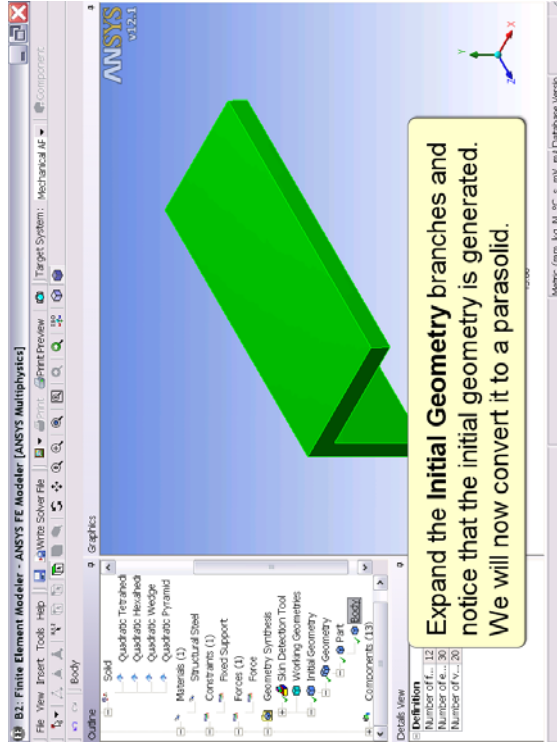
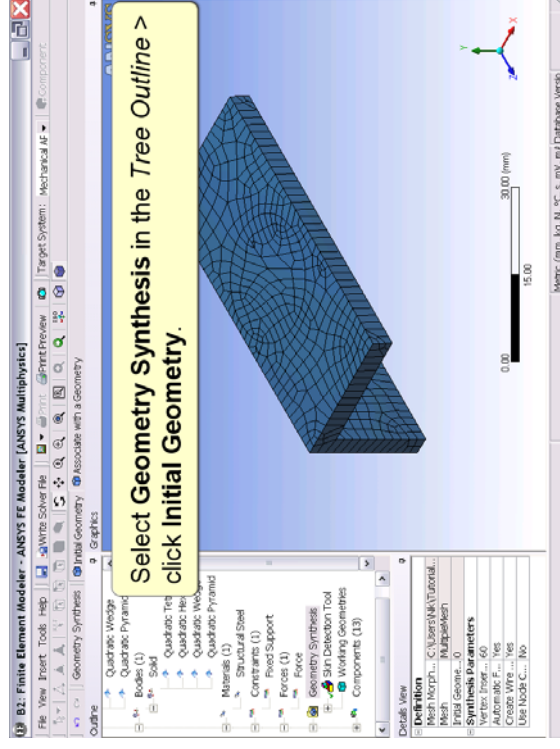
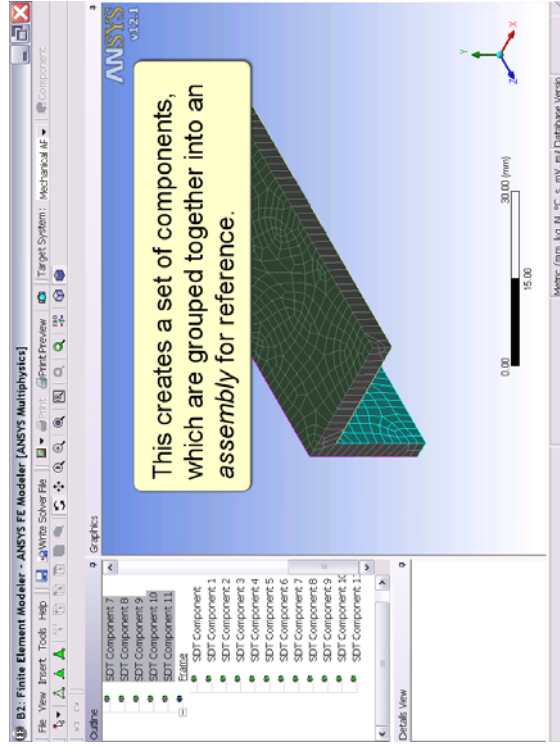
Introduction to Finite Element Modeler



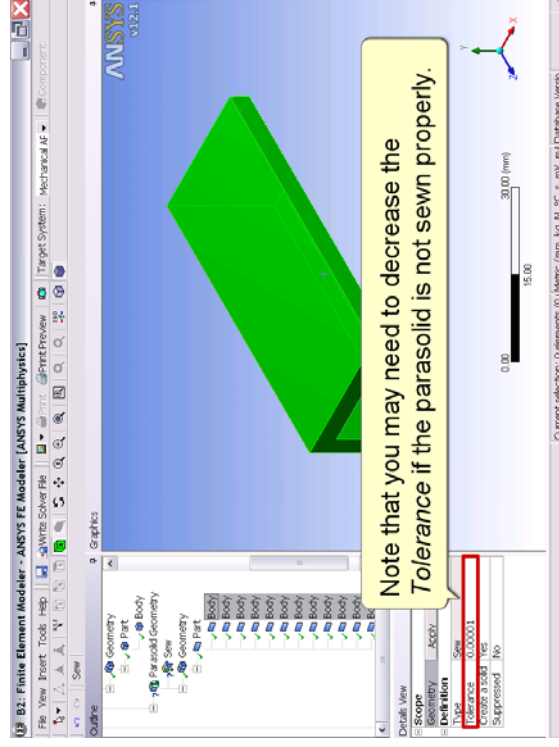
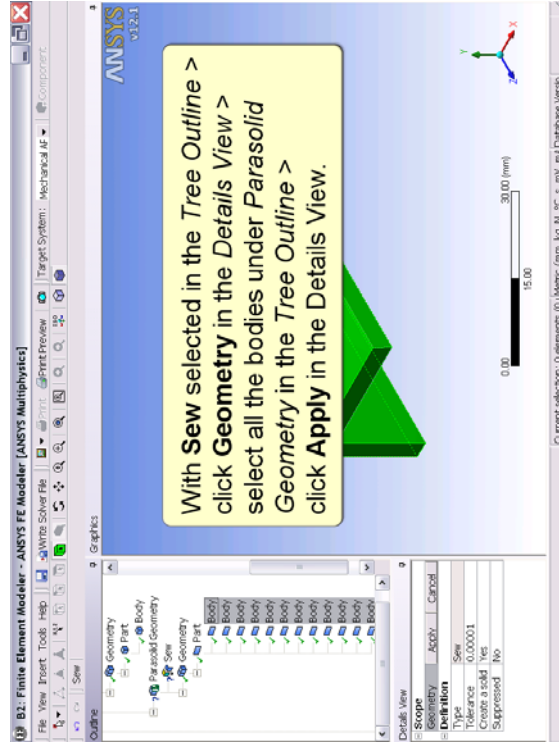
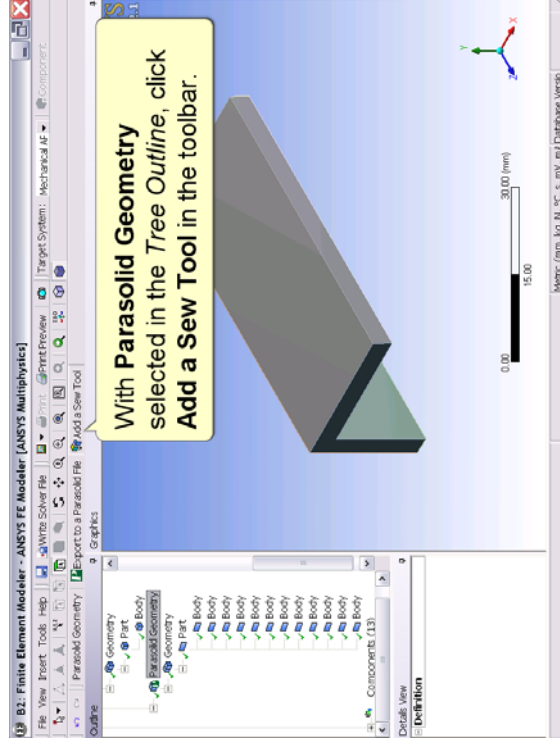
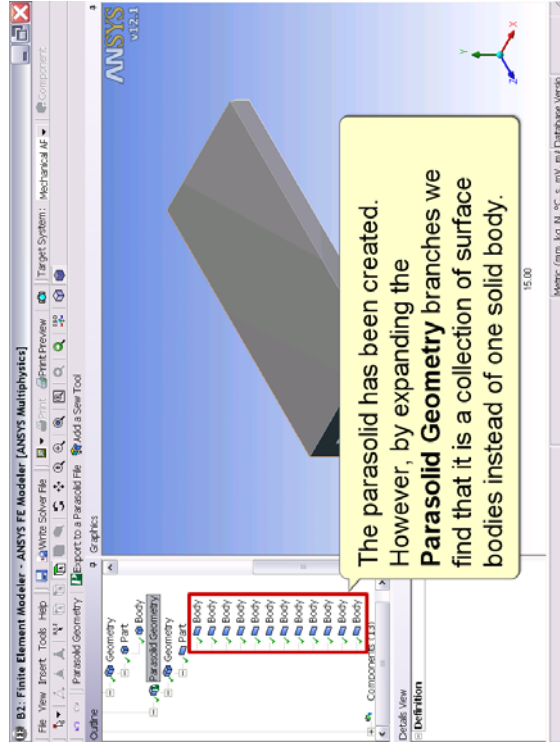
Introduction to Finite Element Modeler



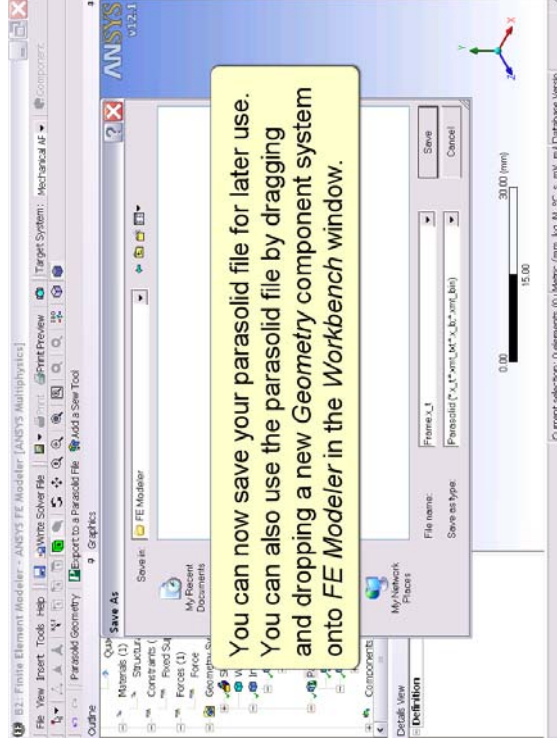
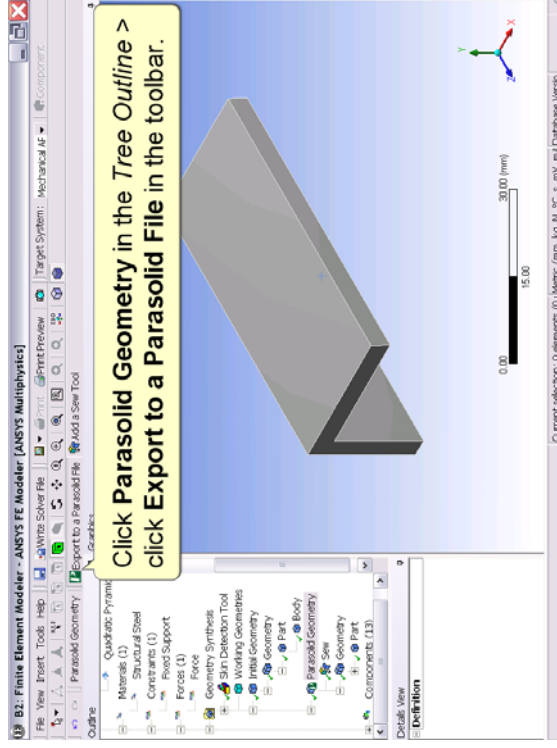
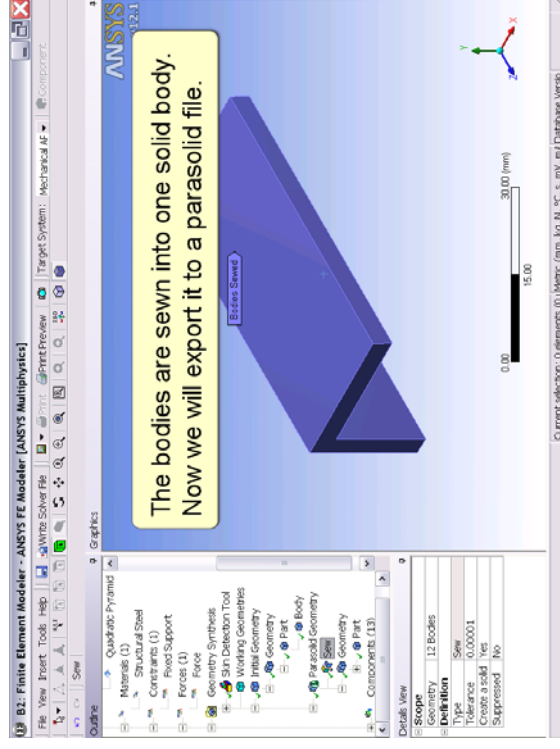
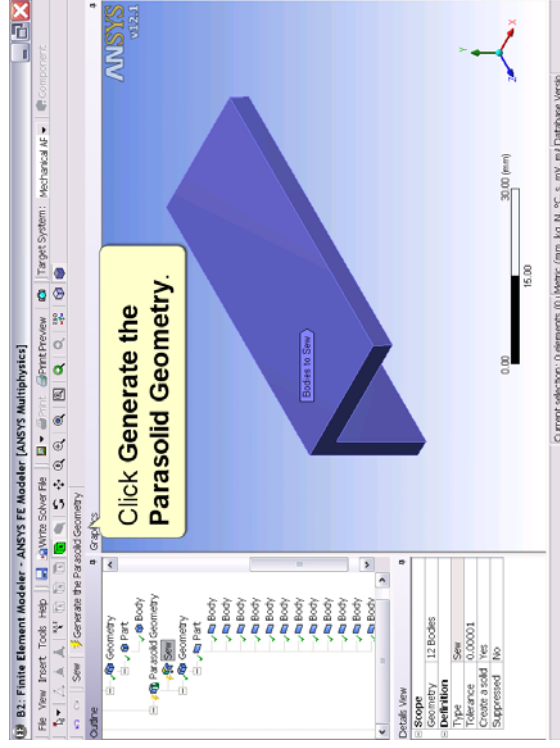
Introduction to Finite Element Modeler



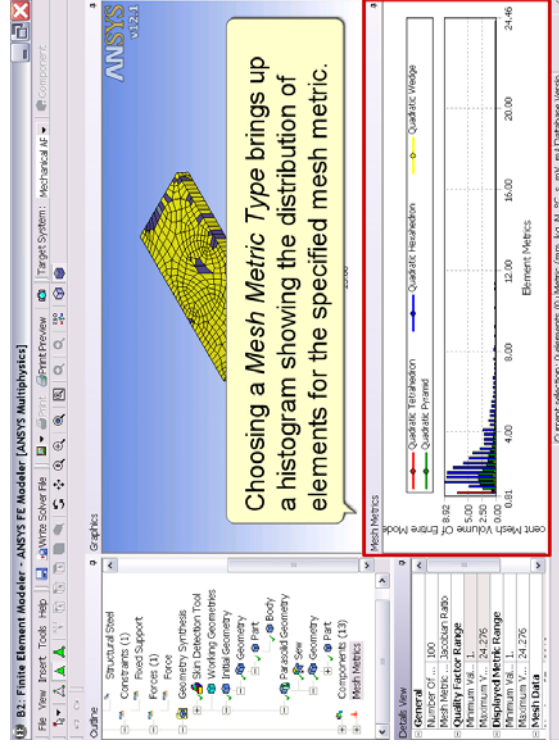
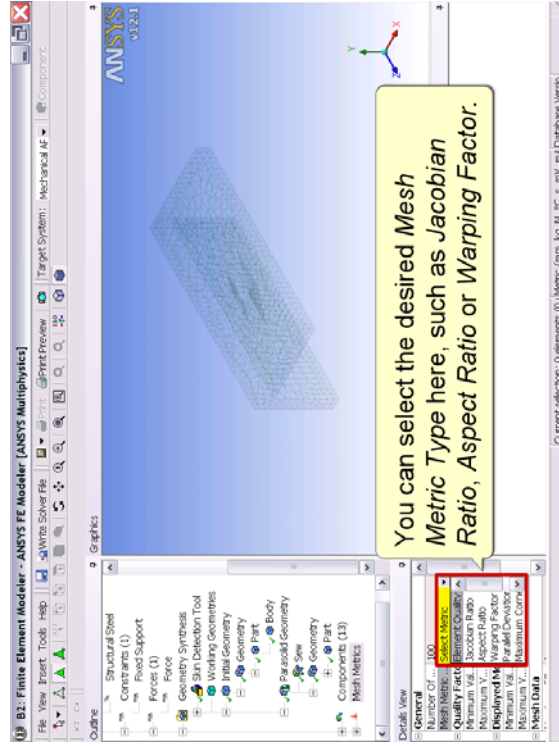
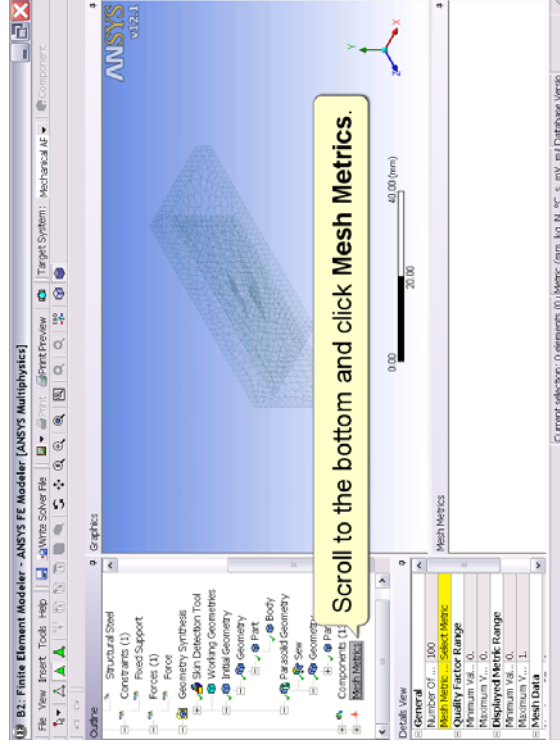
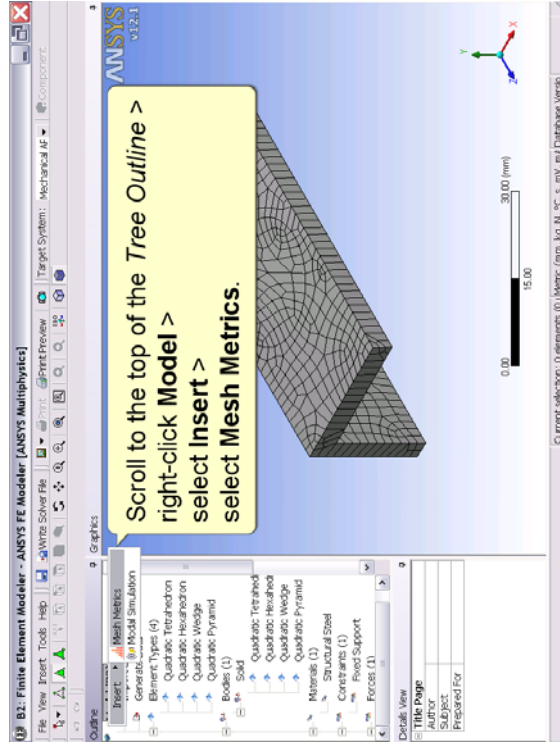
Introduction to Finite Element Modeler



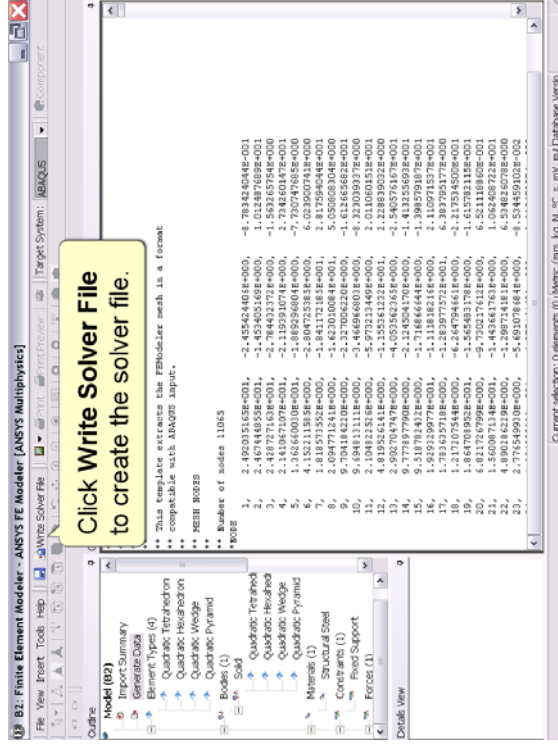
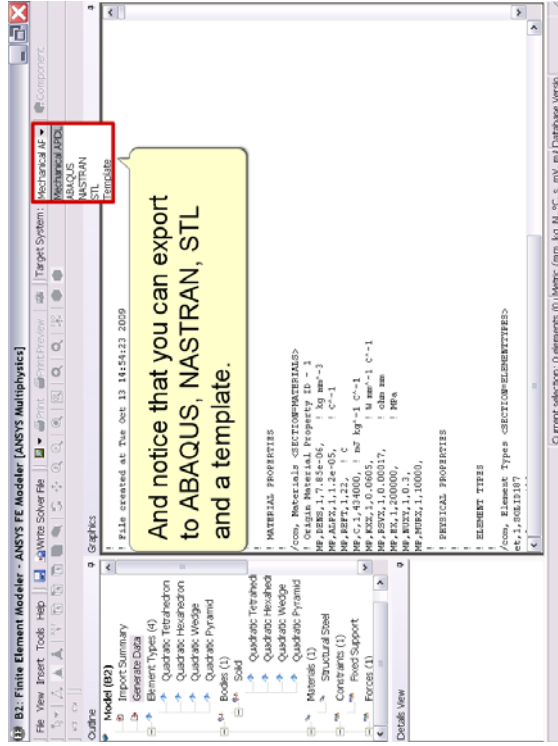
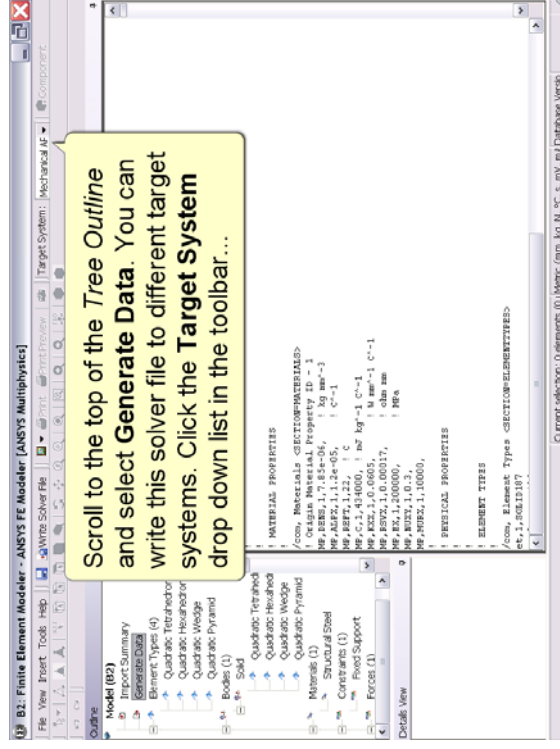
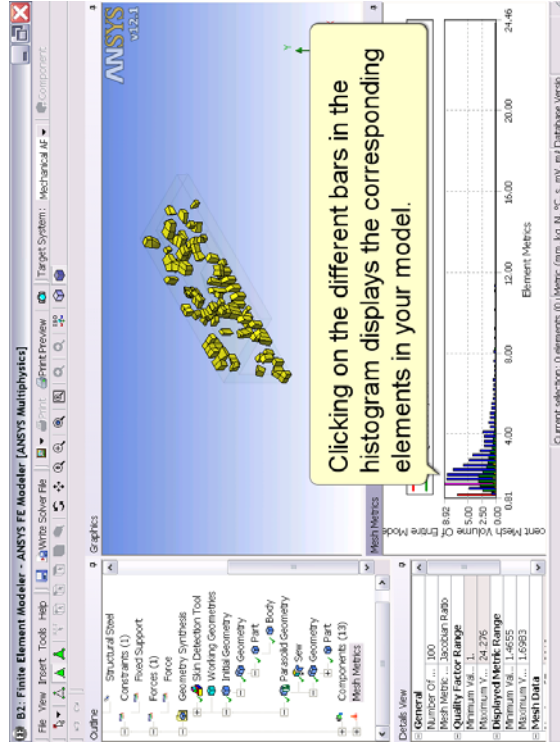
Introduction to Finite Element Modeler



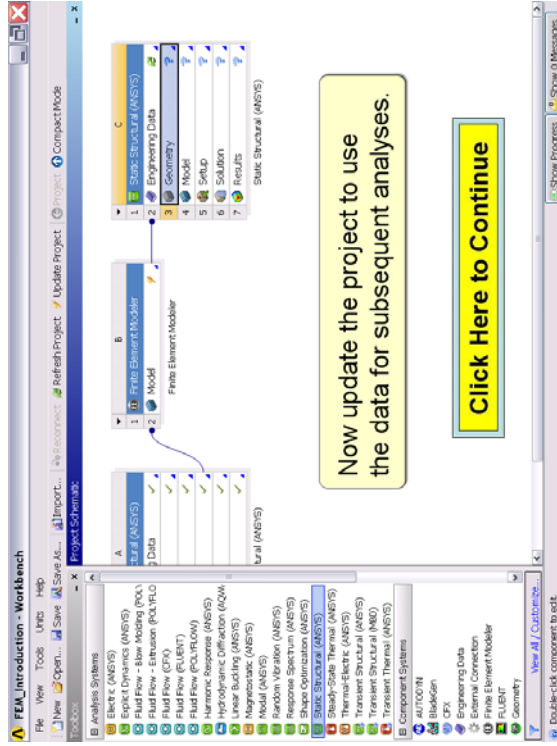
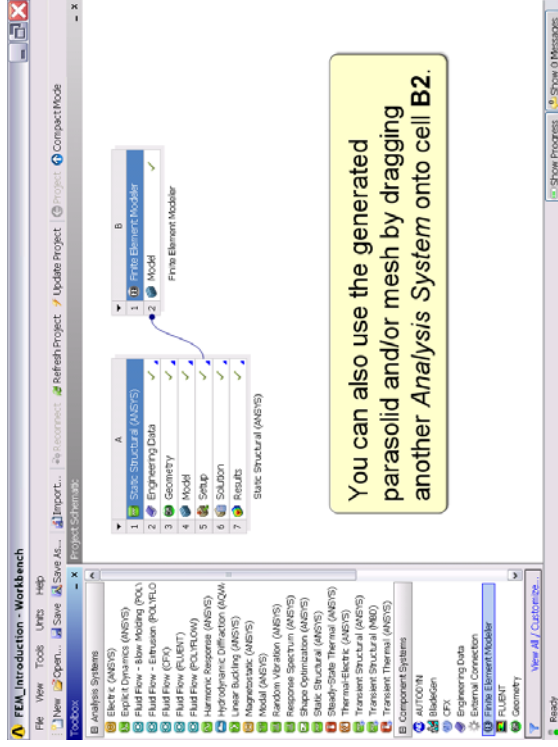
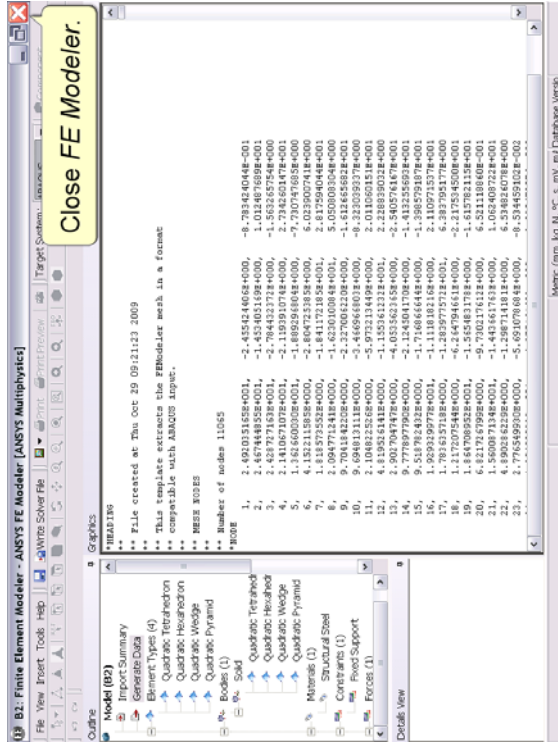
Introduction to Finite Element Modeler



Introduction to Finite Element Modeler



Introduction to Finite Element Modeler



- How to start FE Modeler
- How to view information obtained by FE Modeler
- How to create Skin Components and Assemblies
- How to create a parasolid from an input mesh
- How to use Mesh Metrics
- How to create a solver file