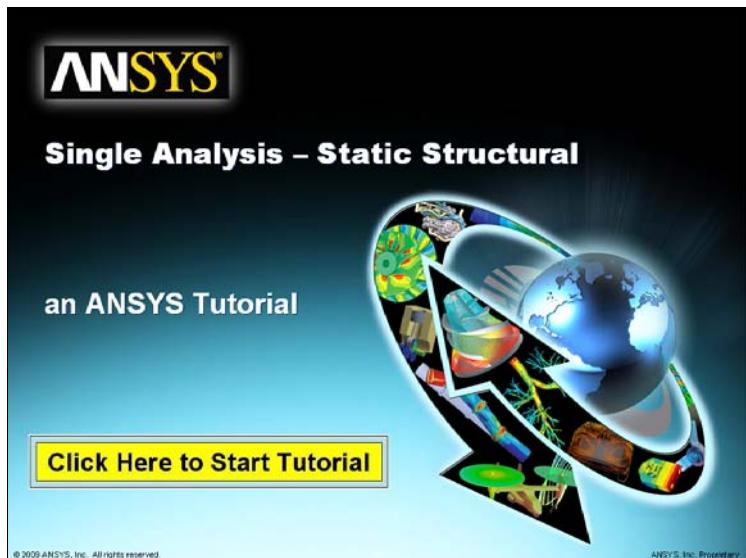


## Structural Mechanics - Static Structural Single Analysis



### Single Analysis - Static Structural **ANSYS**

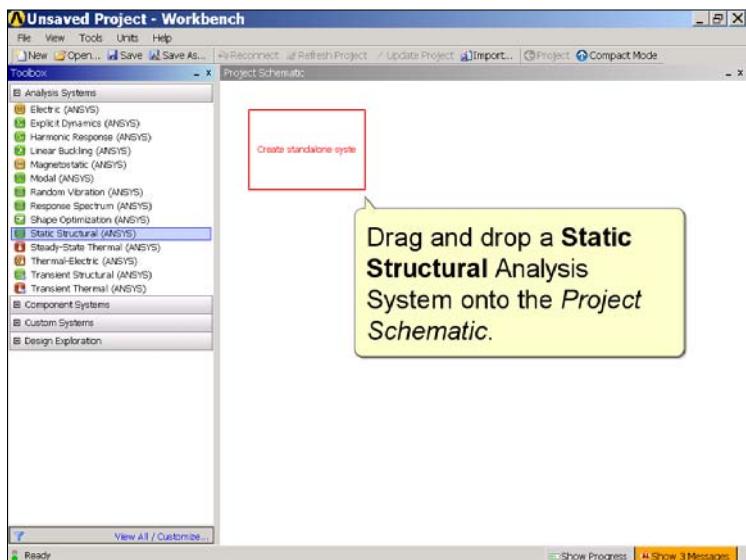
This tutorial will walk you through the simple Static Structural analysis. The geometry will be created in DesignModeler. In Mechanical Application we will define boundary conditions (supports and loads), solve the model, and evaluate results. Then, the geometry will be changed in DesignModeler and automatically updated in Mechanical application.

**Topics Covered in this tutorial**

- Geometry creation in DesignModeler
- Setting up and solving the Static Structural analysis in Mechanical application
- Updating geometry

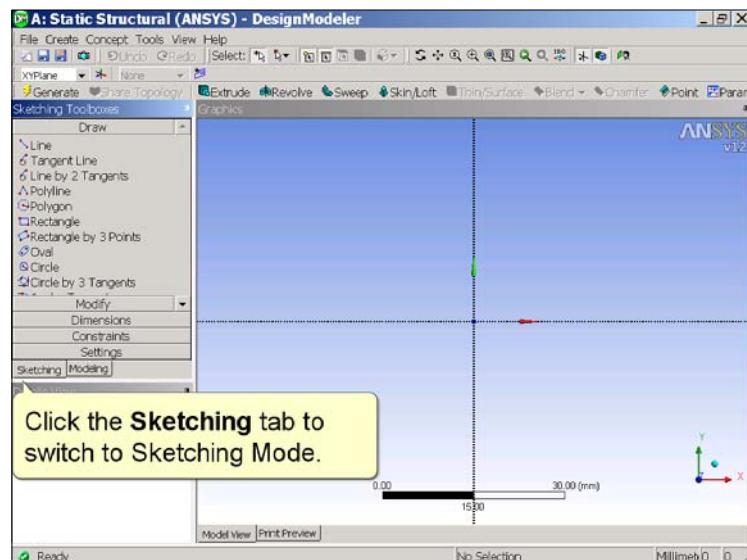
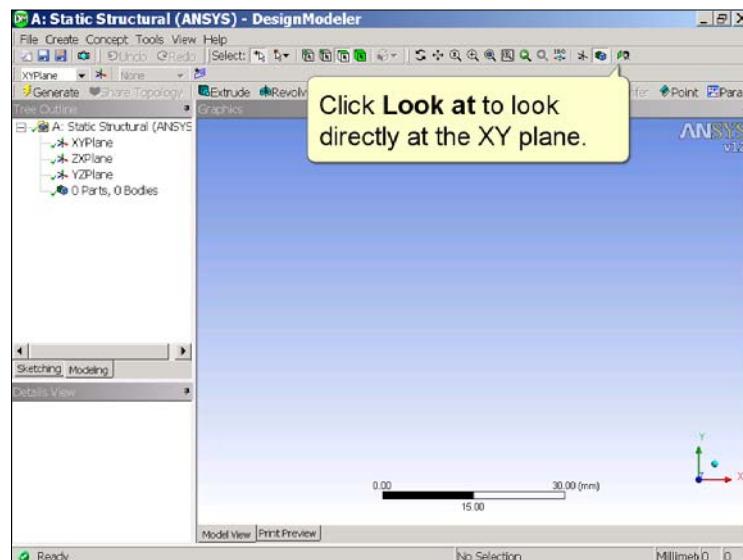
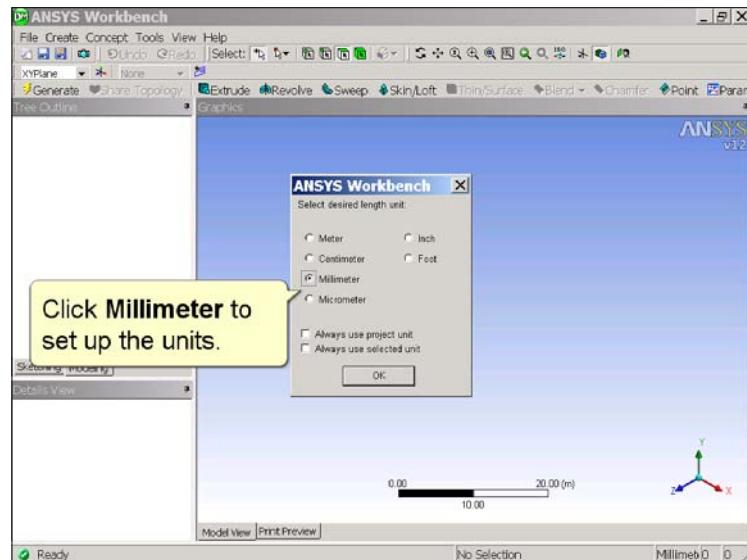
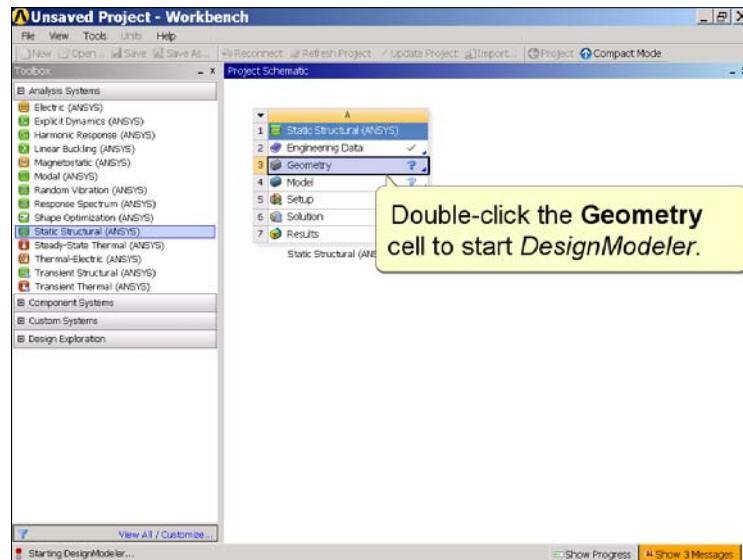
**Click Here to Continue**

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

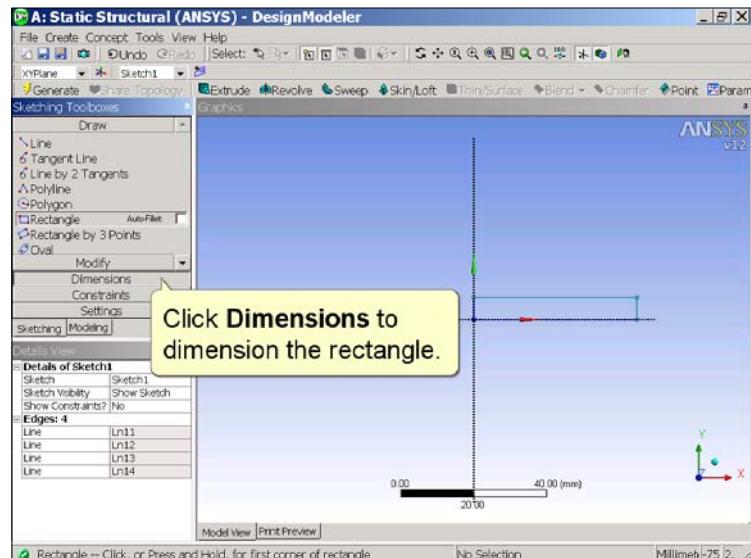
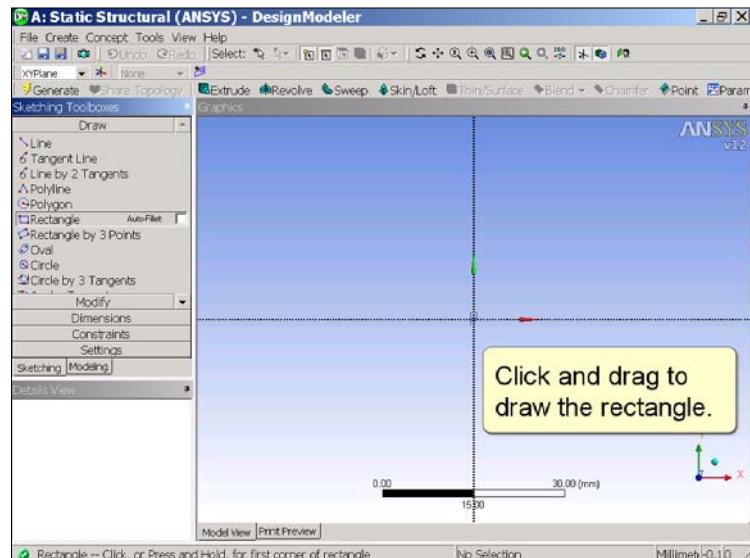
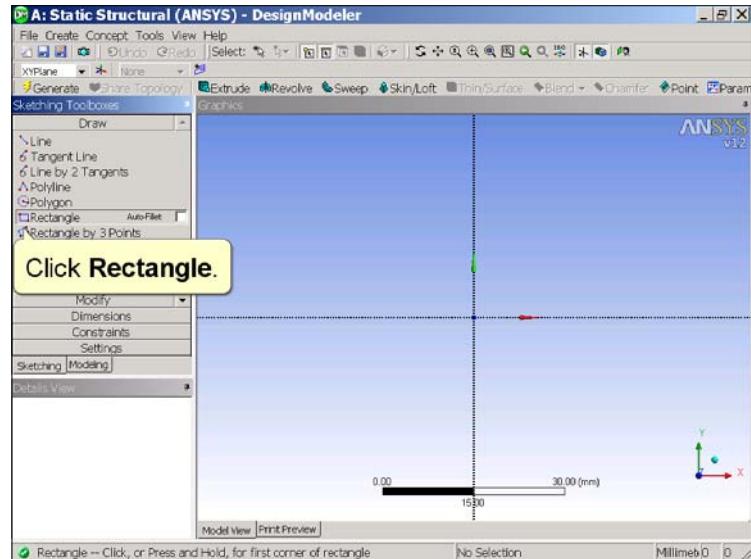
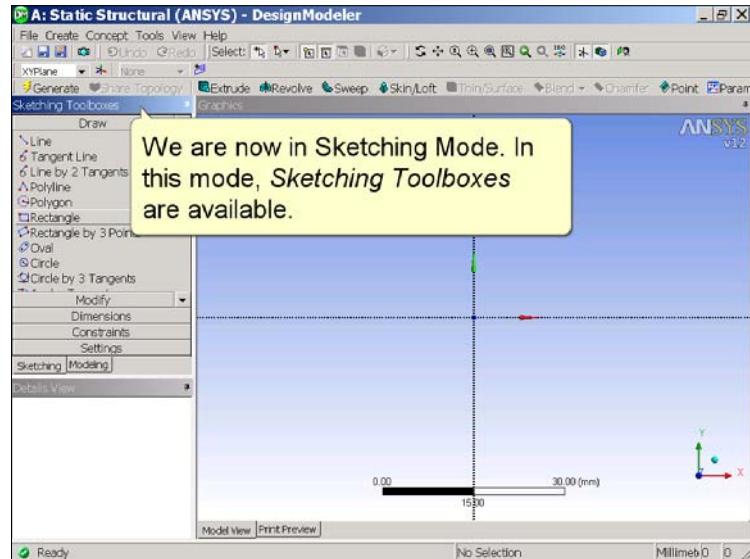


The screenshot shows the same "Unsaved Project - Workbench" window. The "Project Schematic" area now displays a numbered list of components: 1. Static Structural (ANSYS), 2. Engineering Data, 3. Geometry, 4. Model, 5. Setup, 6. Solution, and 7. Results. A callout box contains the text: "For this exercise, we will create a plate geometry in DesignModeler." The status bar at the bottom shows "View All / Customize...", "Show Progress", and "4 Show 3 Messages".

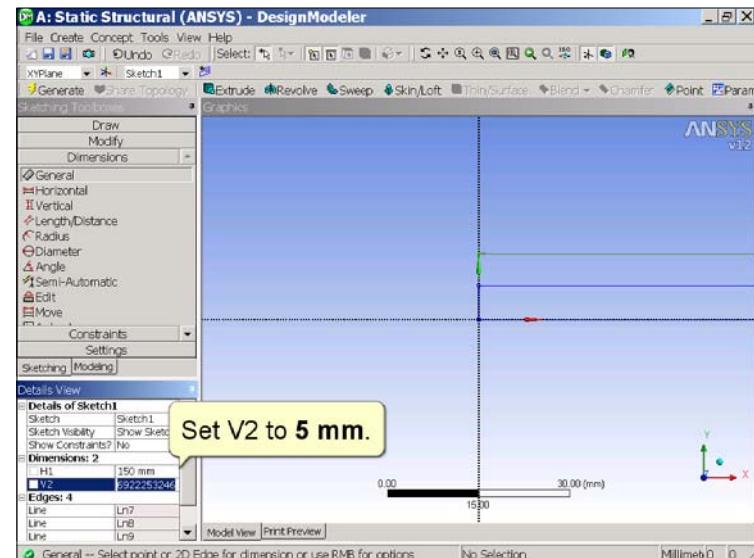
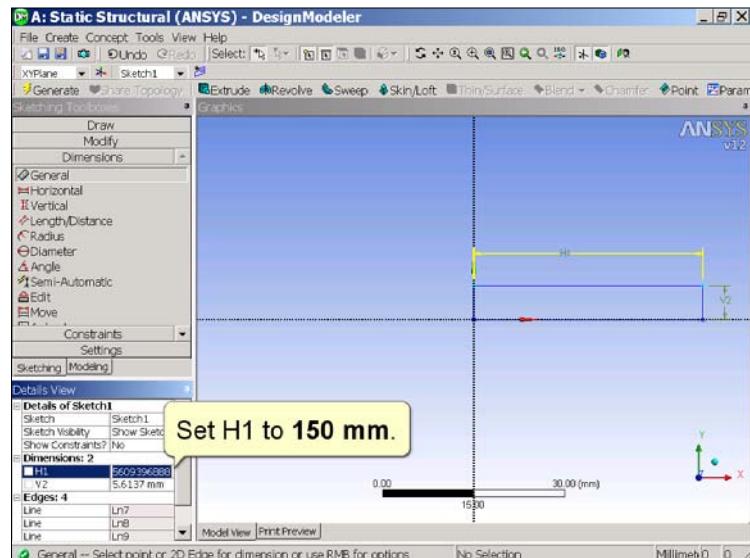
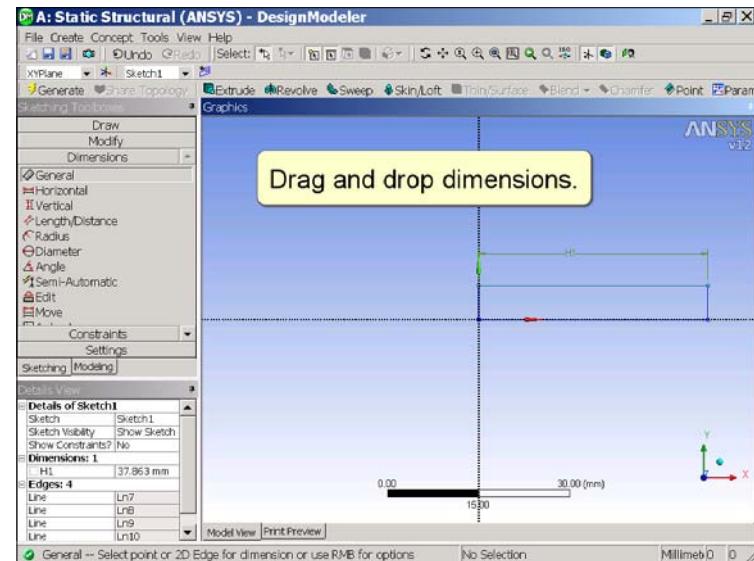
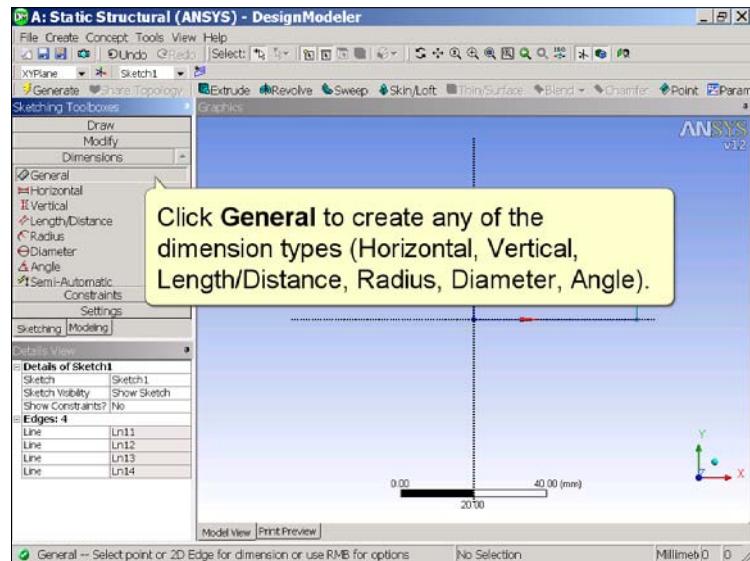
## Structural Mechanics - Static Structural Single Analysis



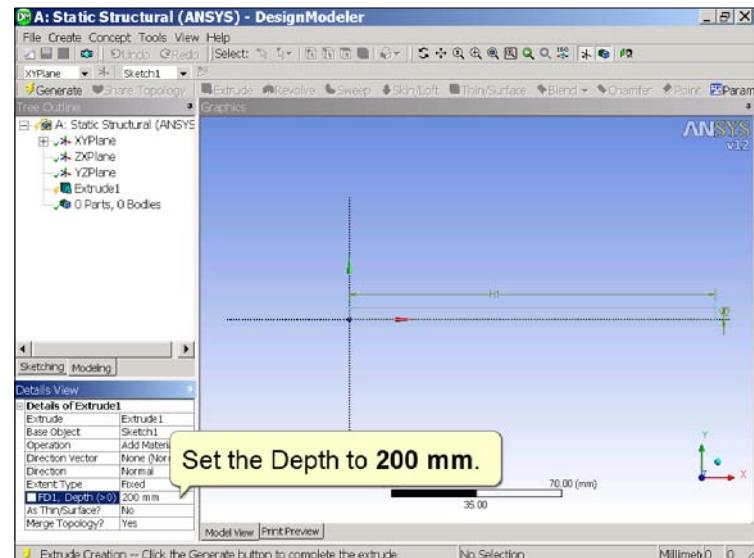
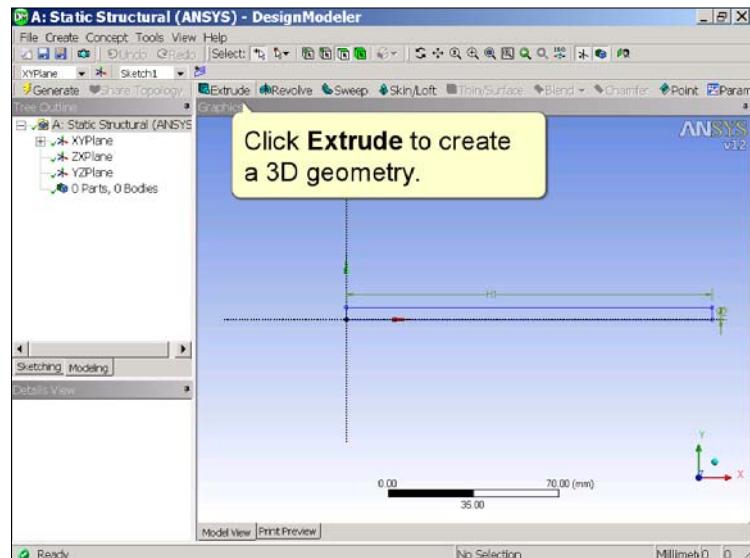
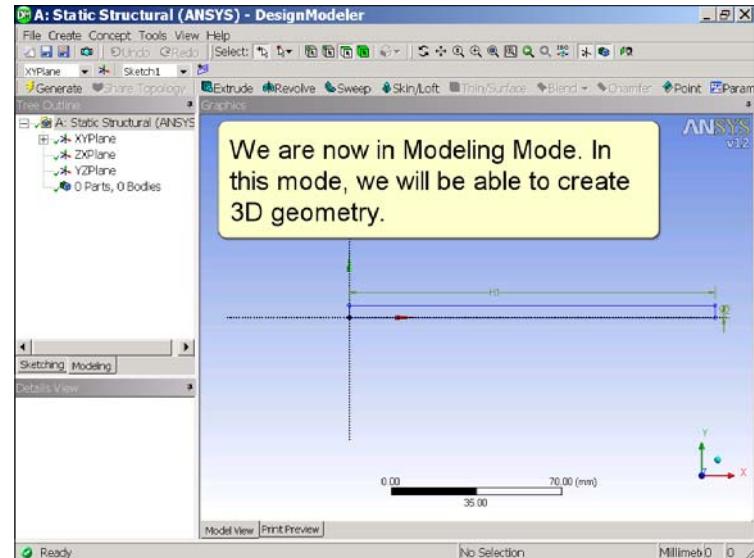
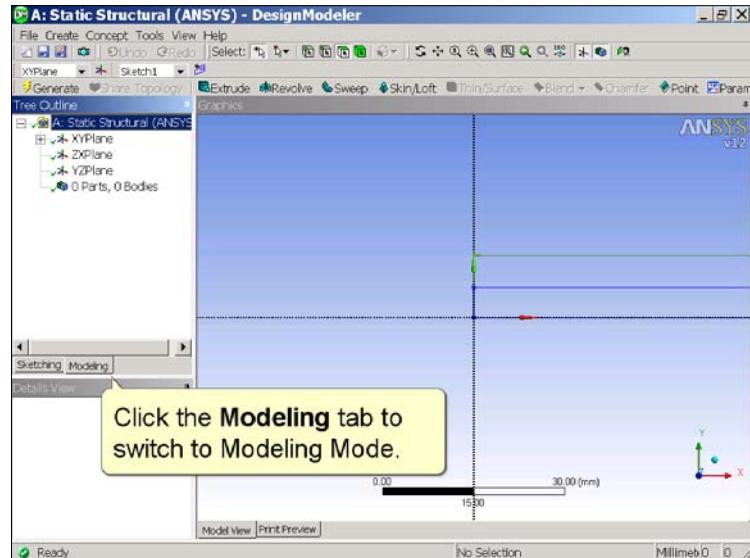
## Structural Mechanics - Static Structural Single Analysis



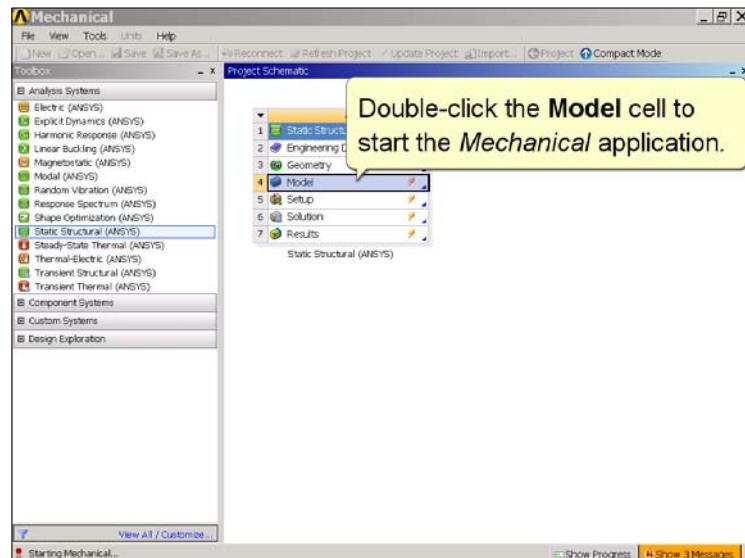
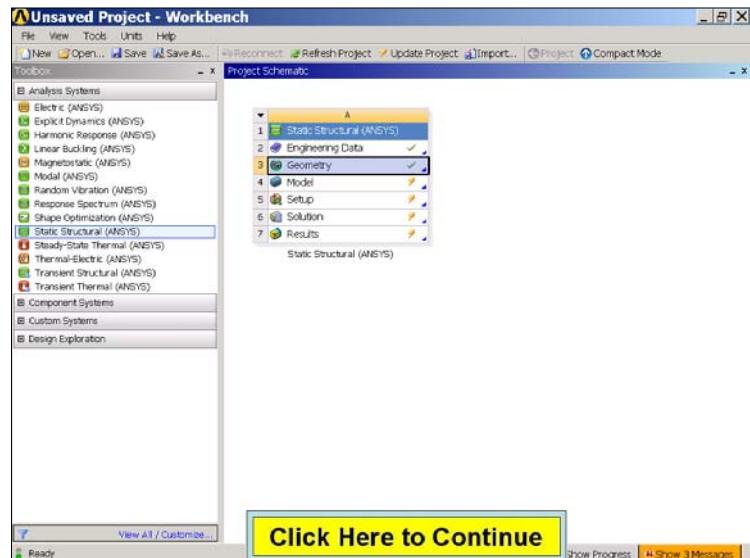
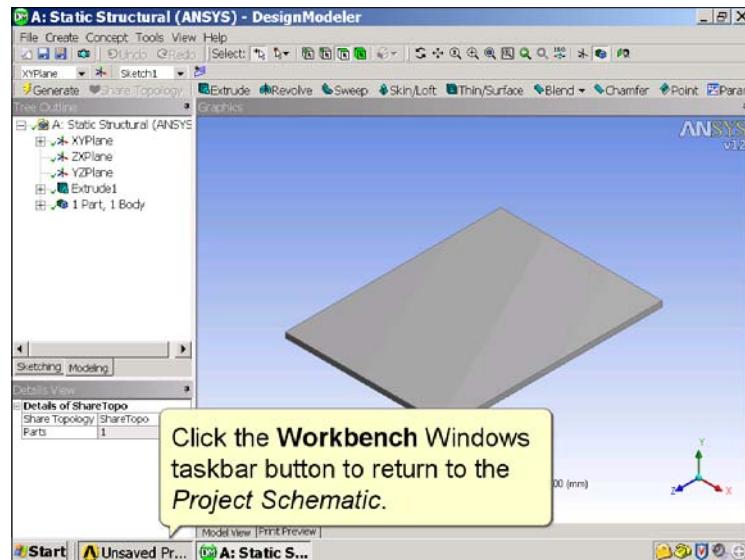
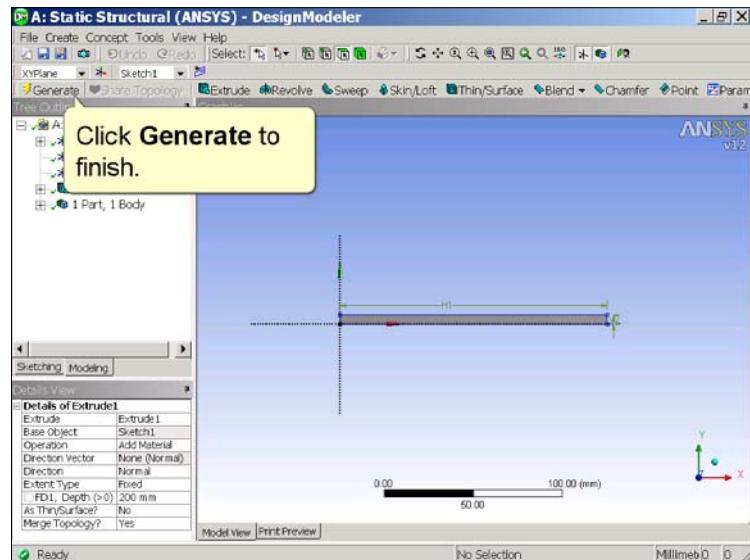
## Structural Mechanics - Static Structural Single Analysis



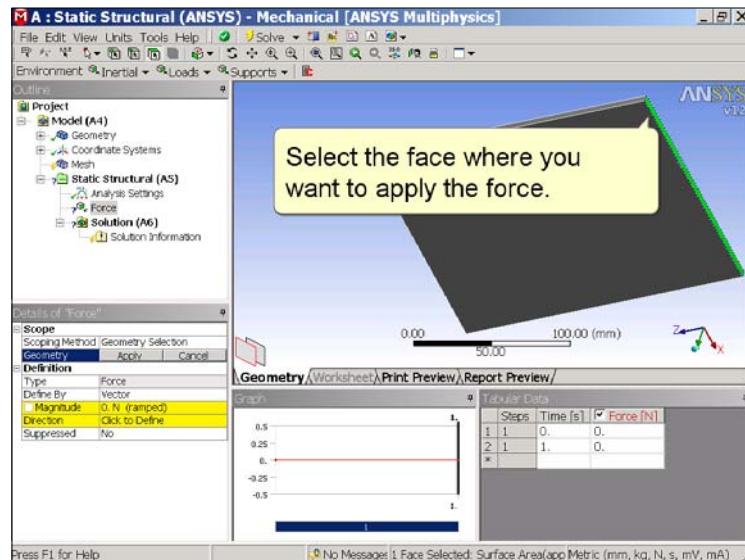
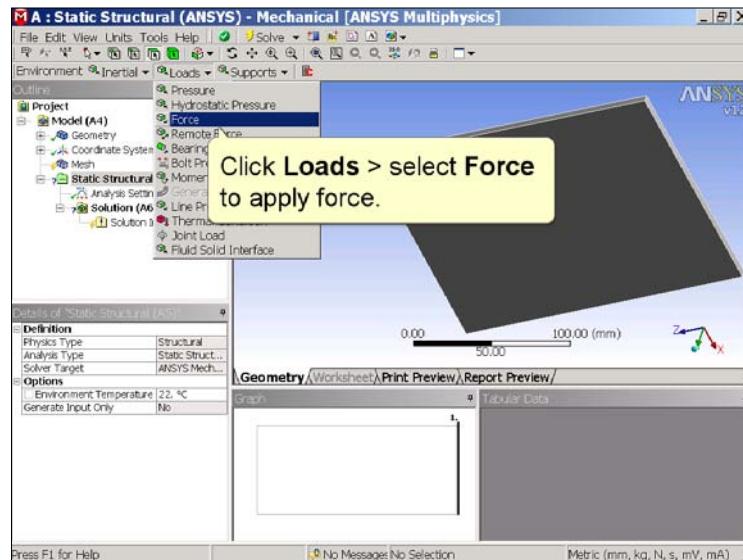
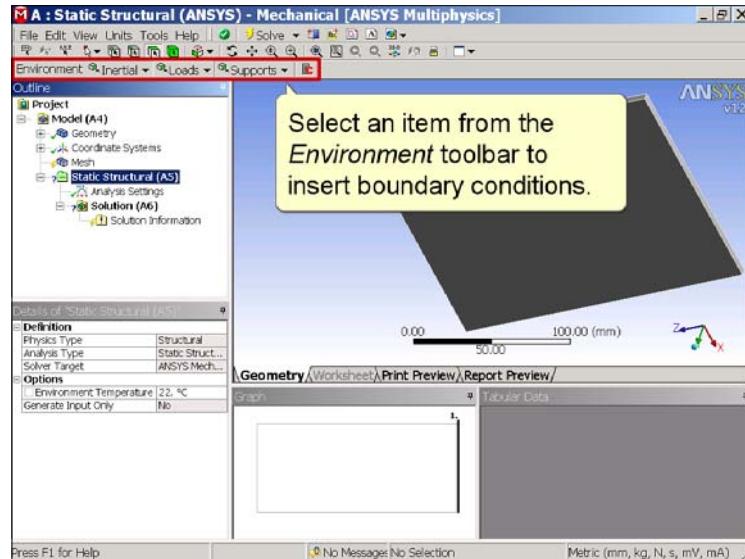
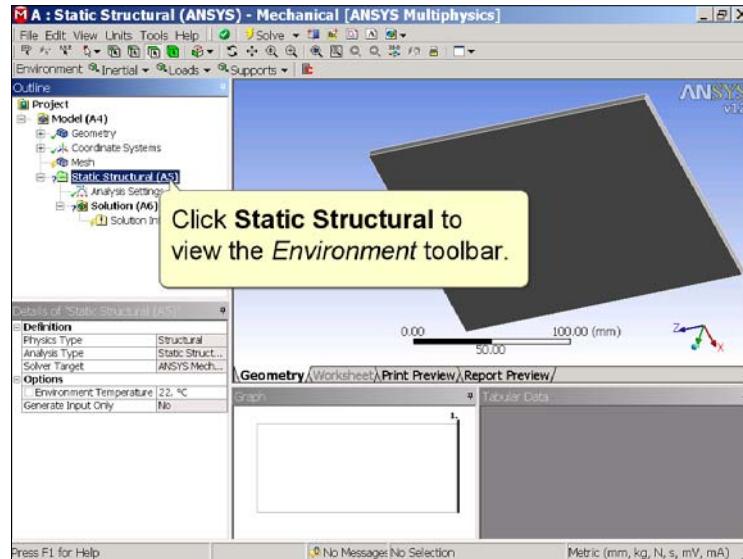
## Structural Mechanics - Static Structural Single Analysis



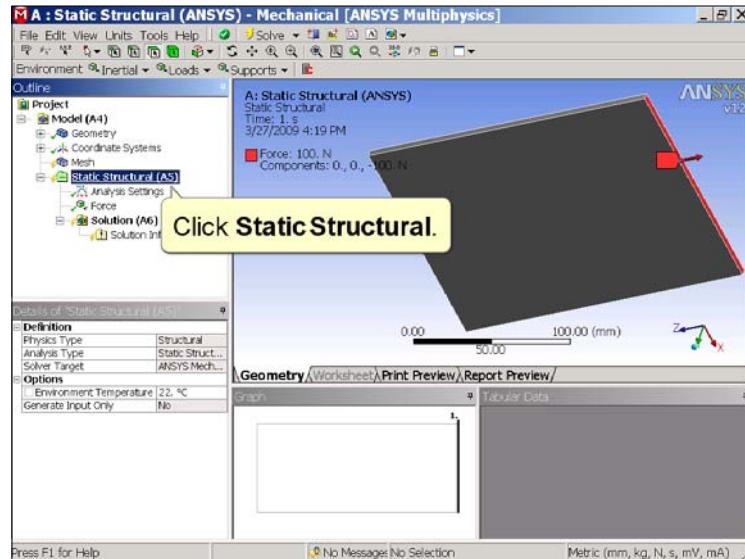
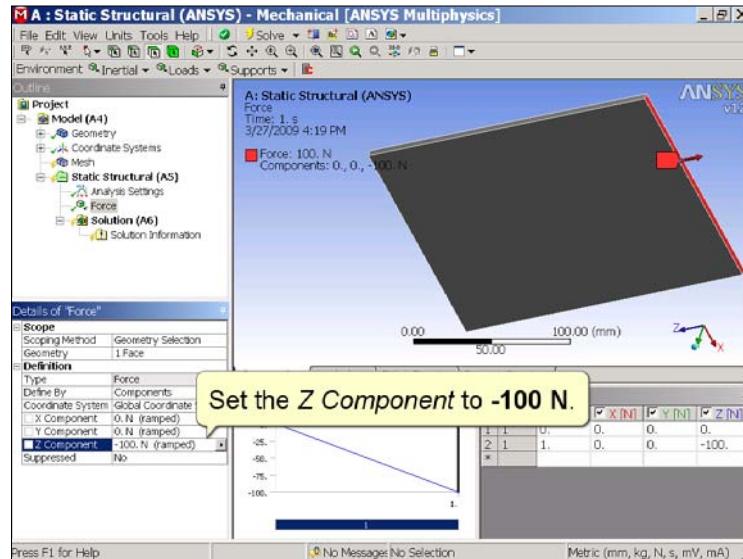
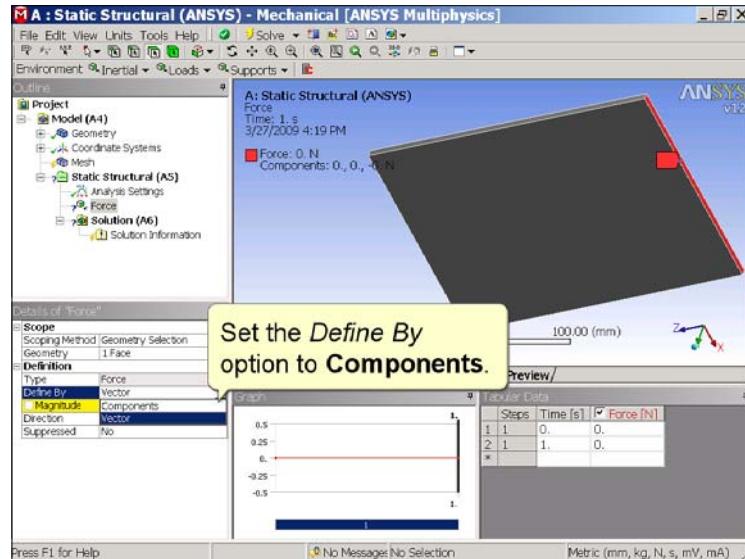
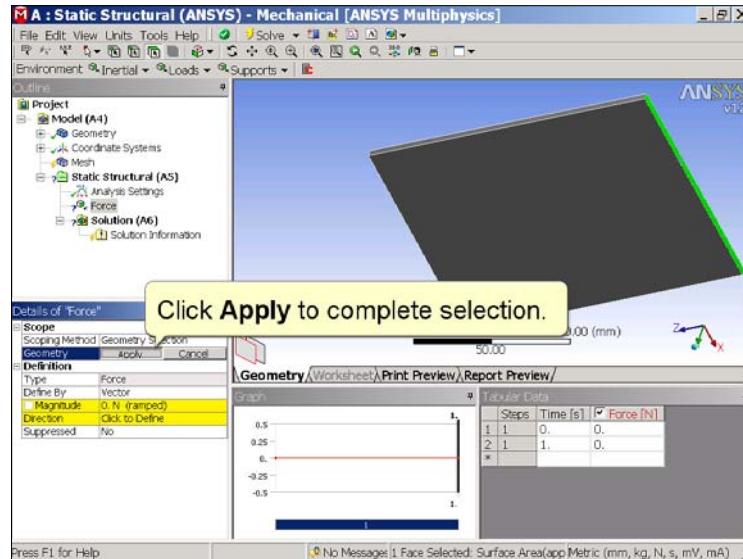
## Structural Mechanics - Static Structural Single Analysis



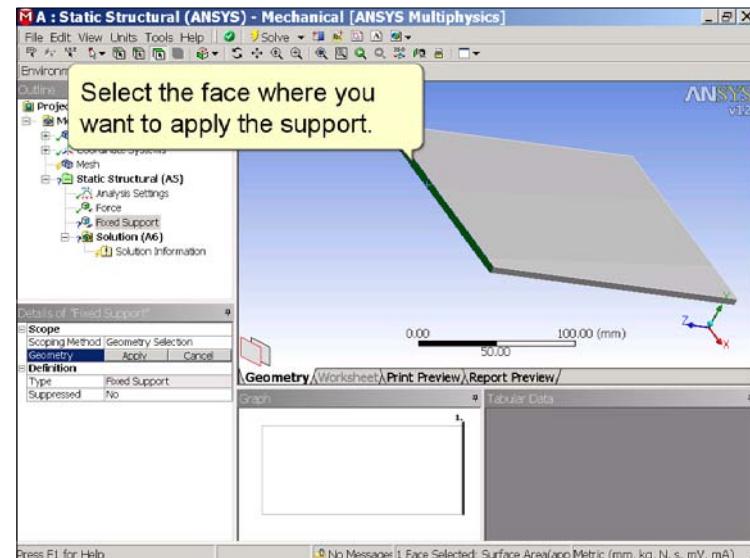
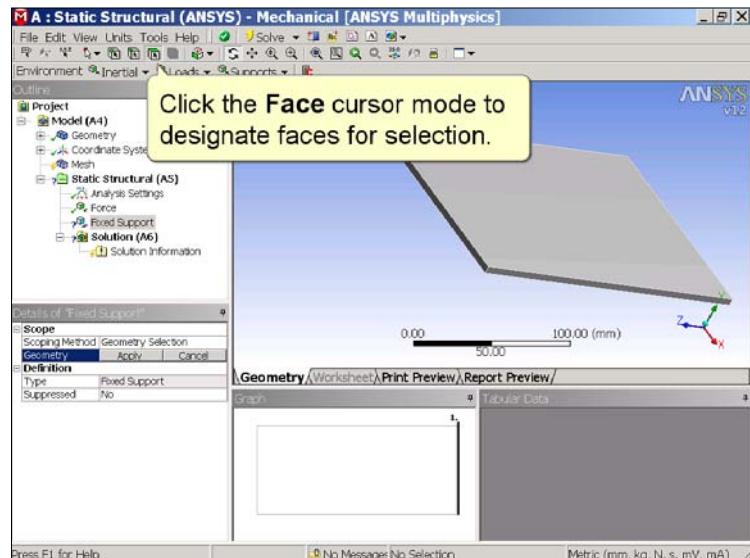
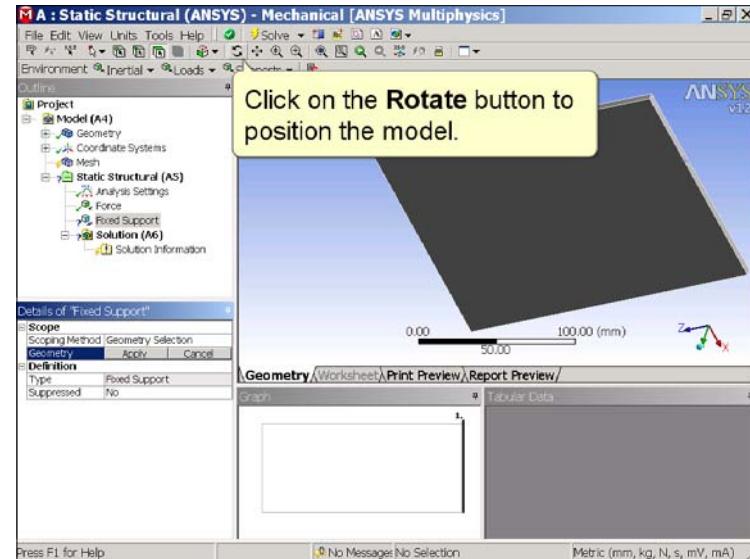
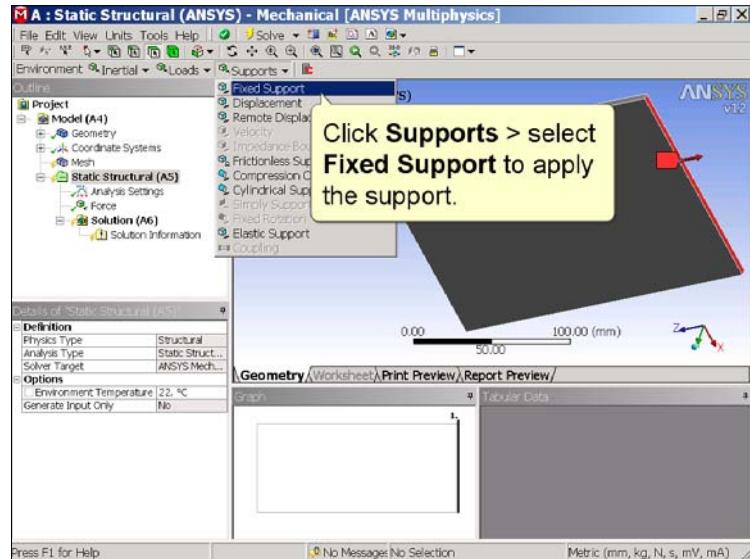
## Structural Mechanics - Static Structural Single Analysis



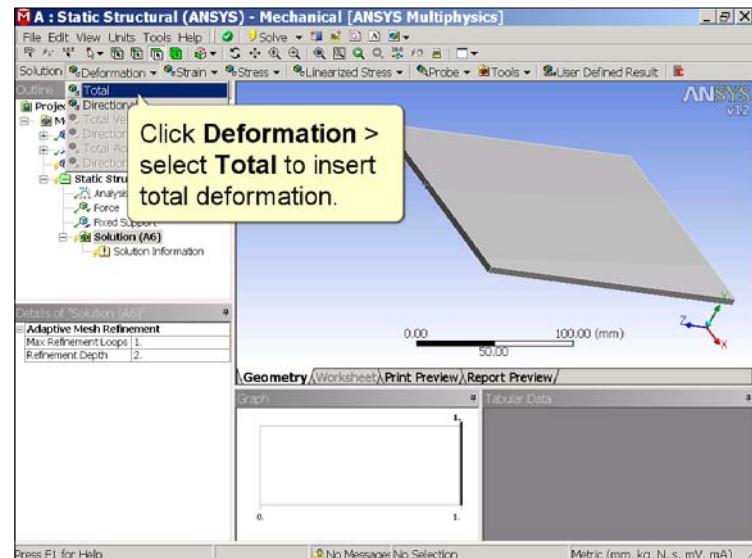
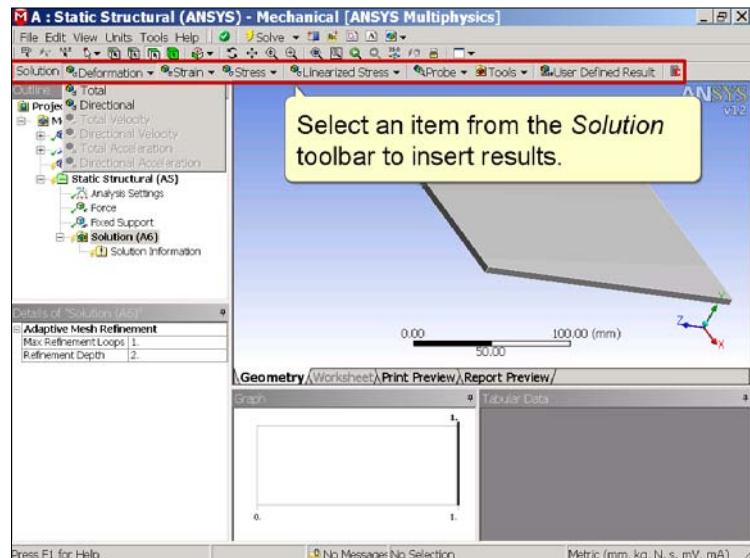
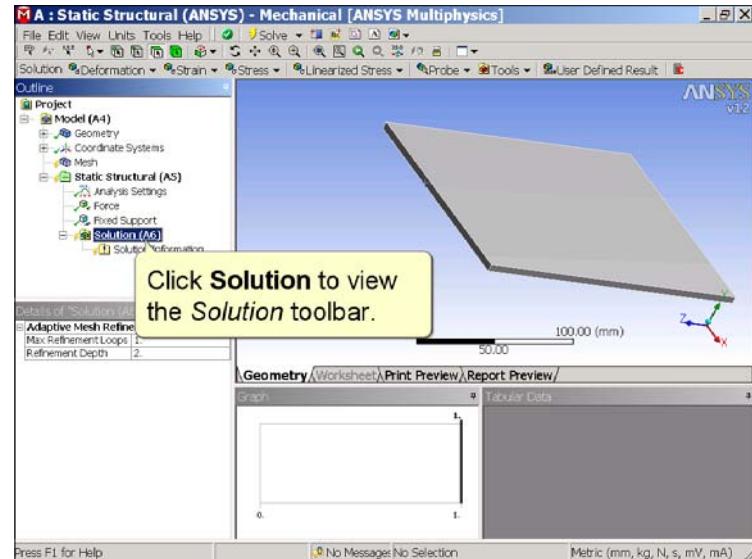
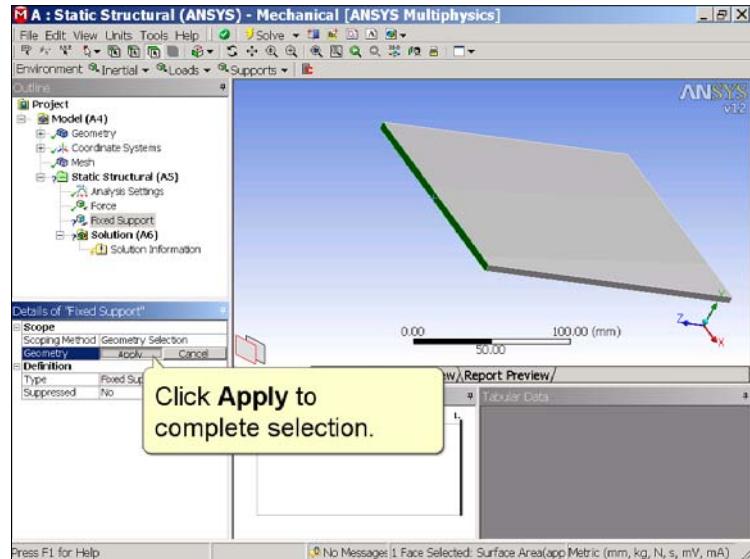
## Structural Mechanics - Static Structural Single Analysis



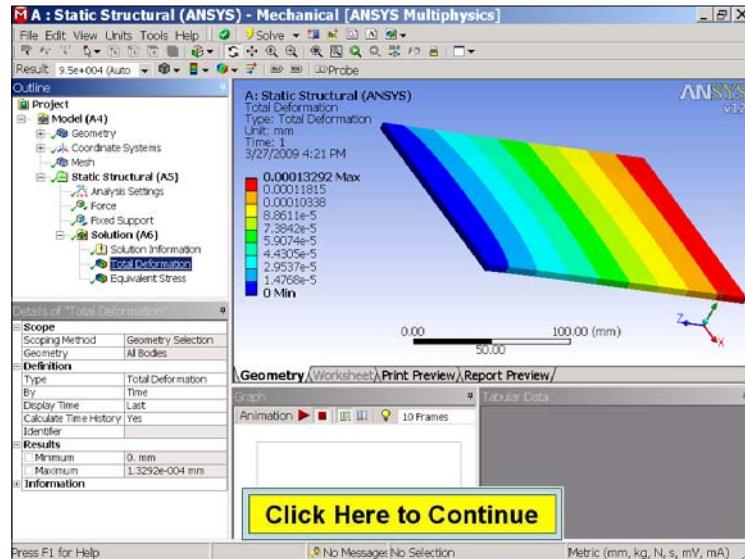
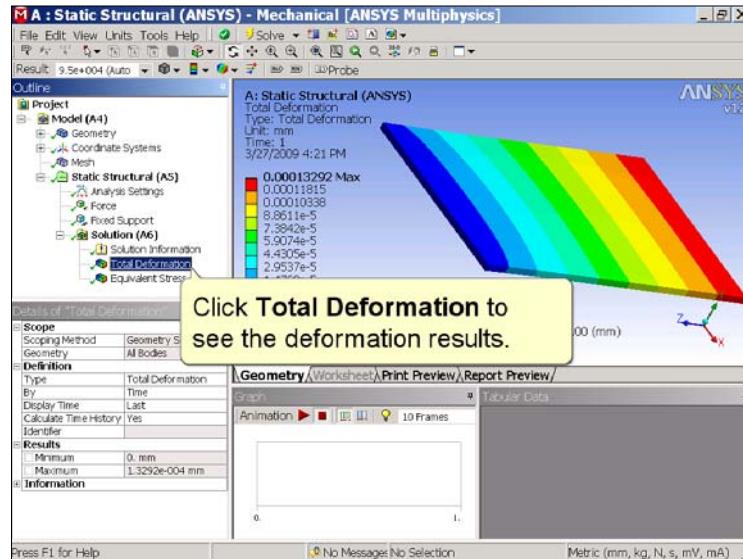
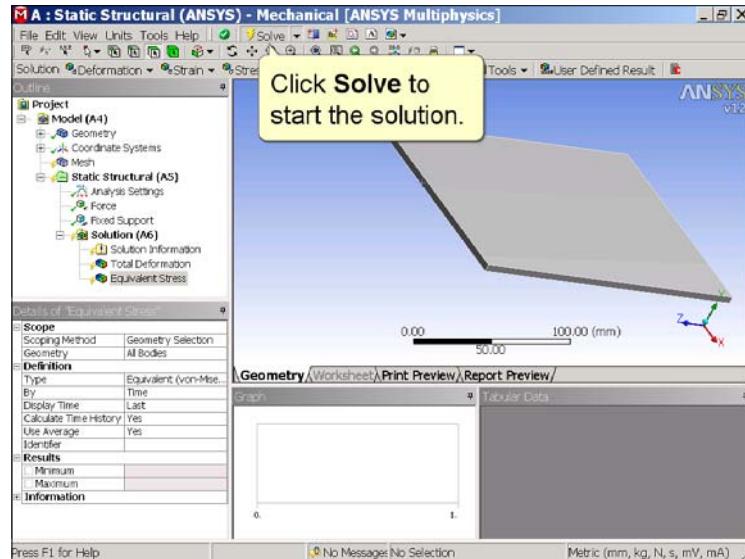
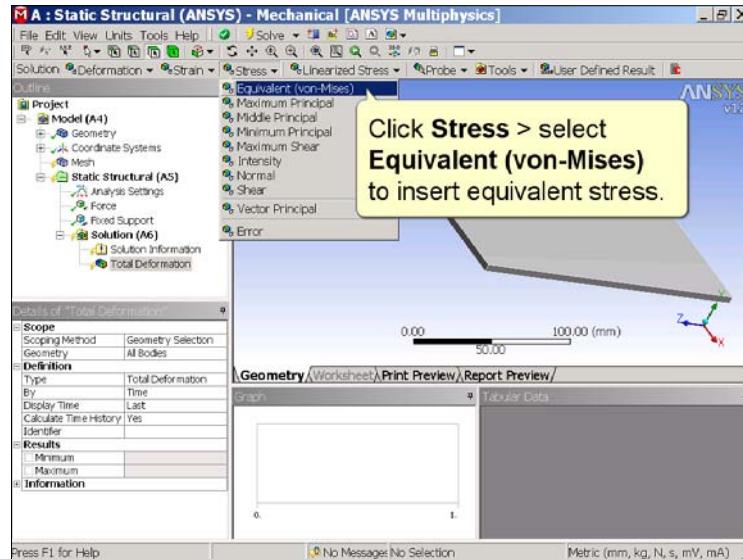
## Structural Mechanics - Static Structural Single Analysis



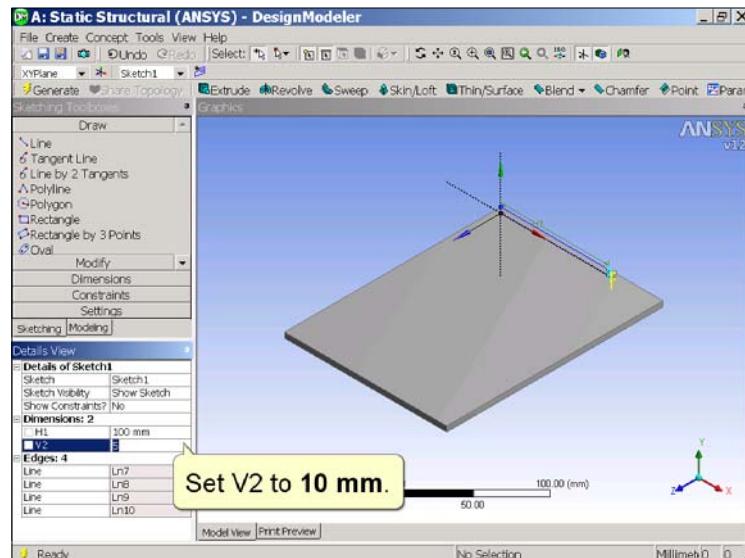
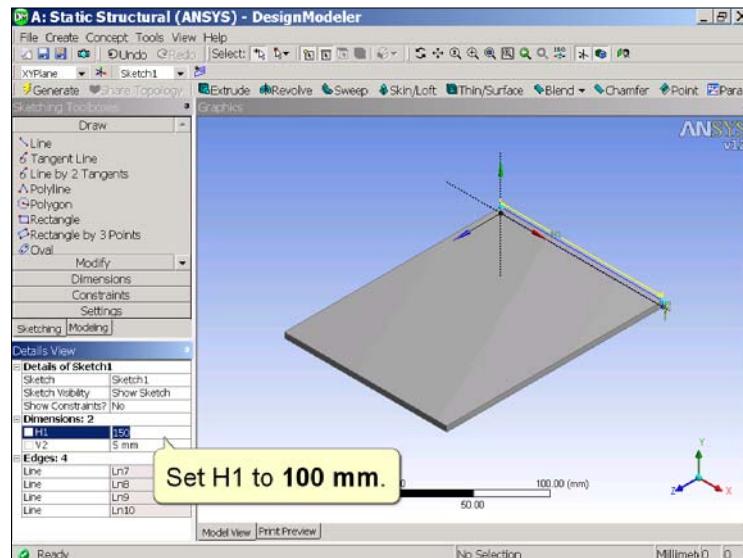
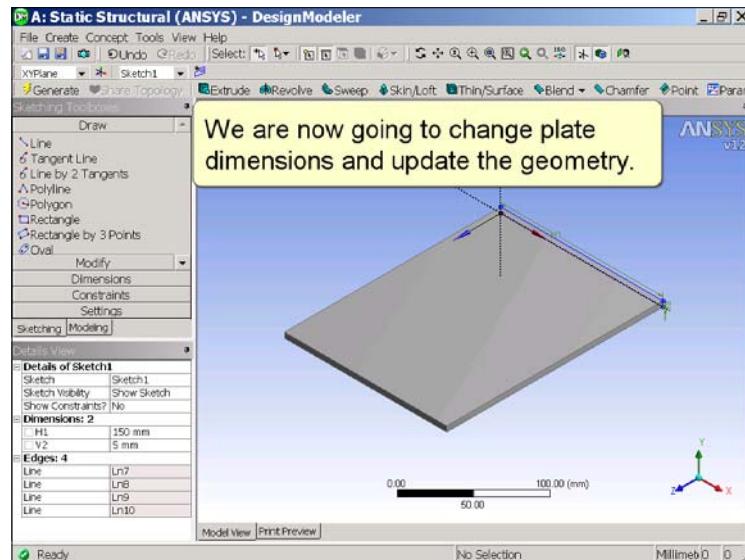
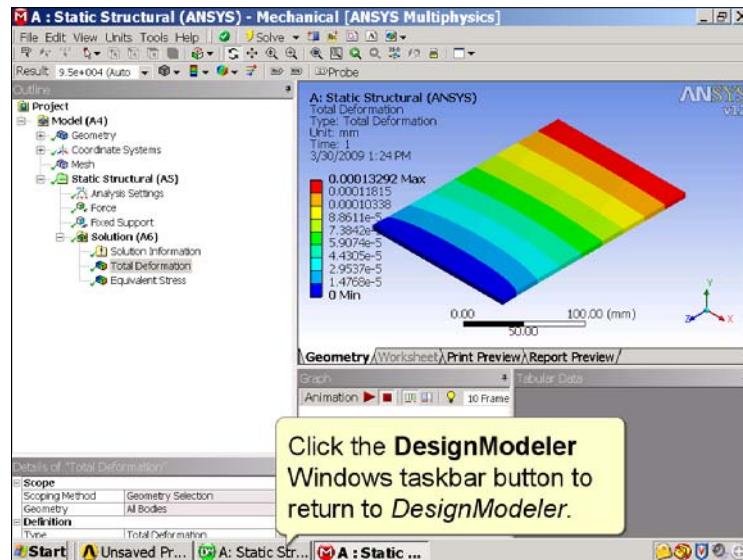
## Structural Mechanics - Static Structural Single Analysis



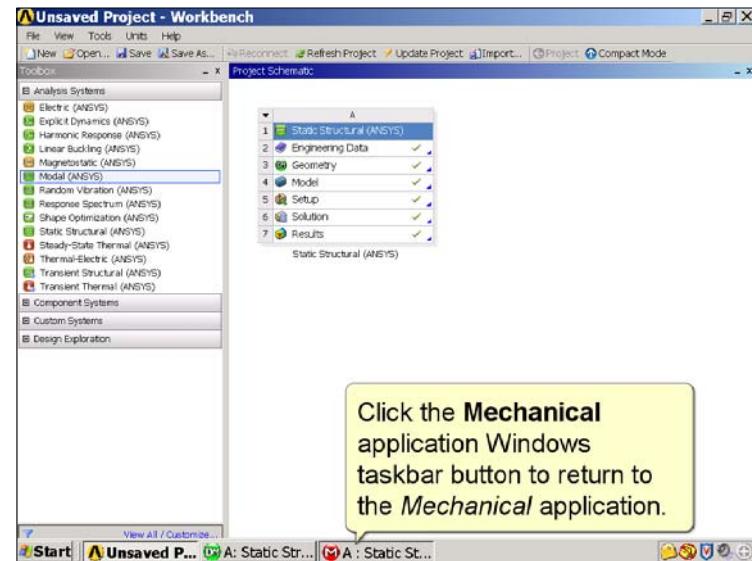
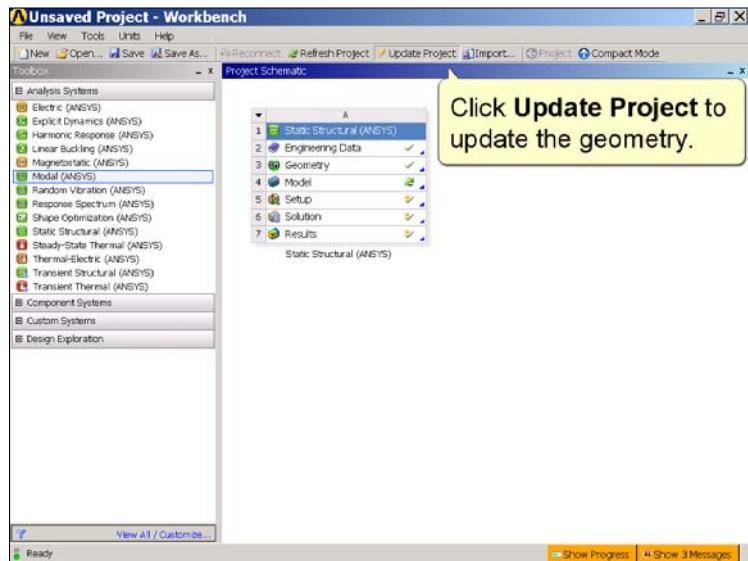
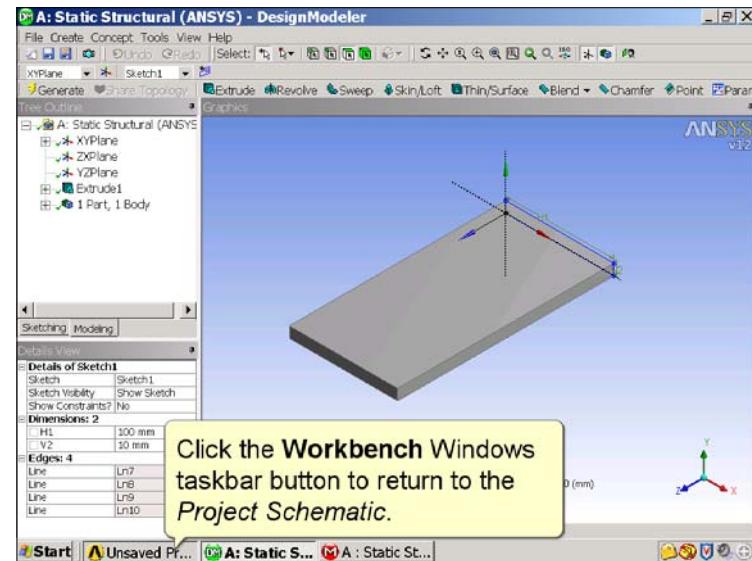
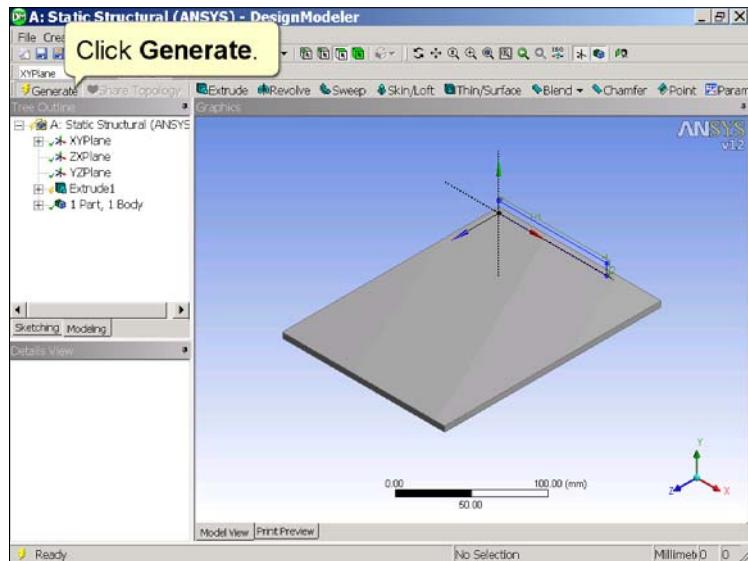
## Structural Mechanics - Static Structural Single Analysis



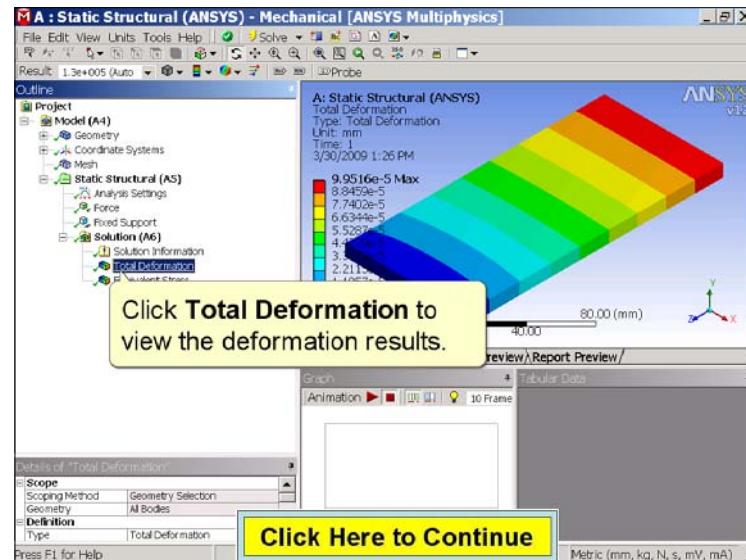
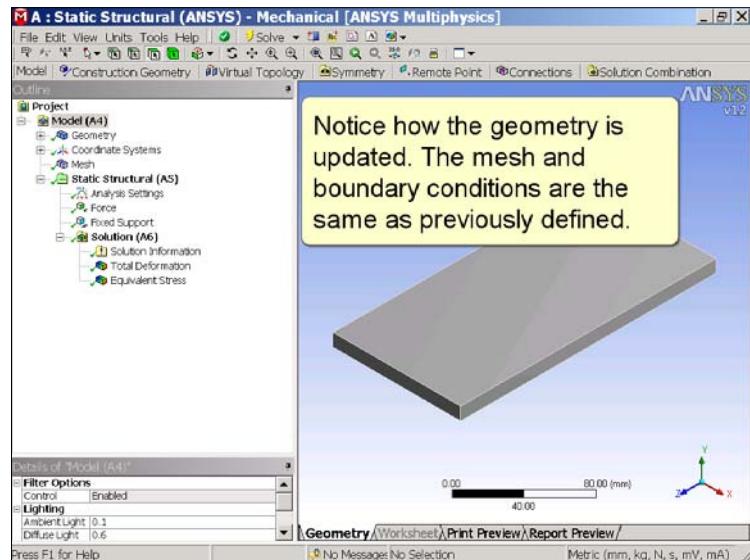
## Structural Mechanics - Static Structural Single Analysis



## Structural Mechanics - Static Structural Single Analysis



## Structural Mechanics - Static Structural Single Analysis



### What Have We Learned?

ANSYS®

- Creating simple geometry in DesignModeler
- Setting up single analysis
- Updating geometry

© 2009 ANSYS, Inc. All rights reserved.