



# ANSYS ED Workbench Tutorial

## ANSYS Workbench Basics



## Using this Tutorial



The following guidelines are provided when taking the tutorial

Green boxes are guides describing various Workbench features but requiring no action on your part

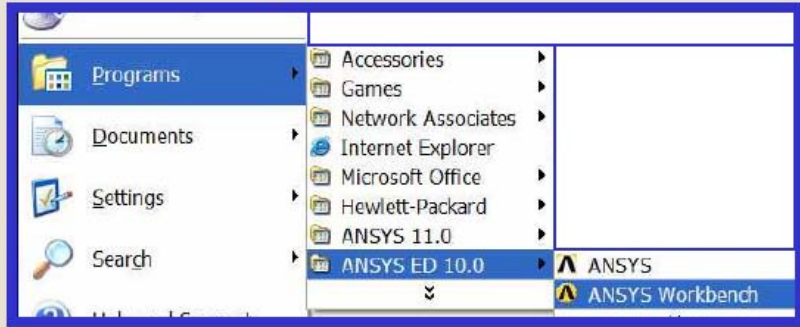
Blue boxes represent actions to be taken. When numbered they guide you through the sequence of the actions

Orange boxes present warnings or notes of interest or importance

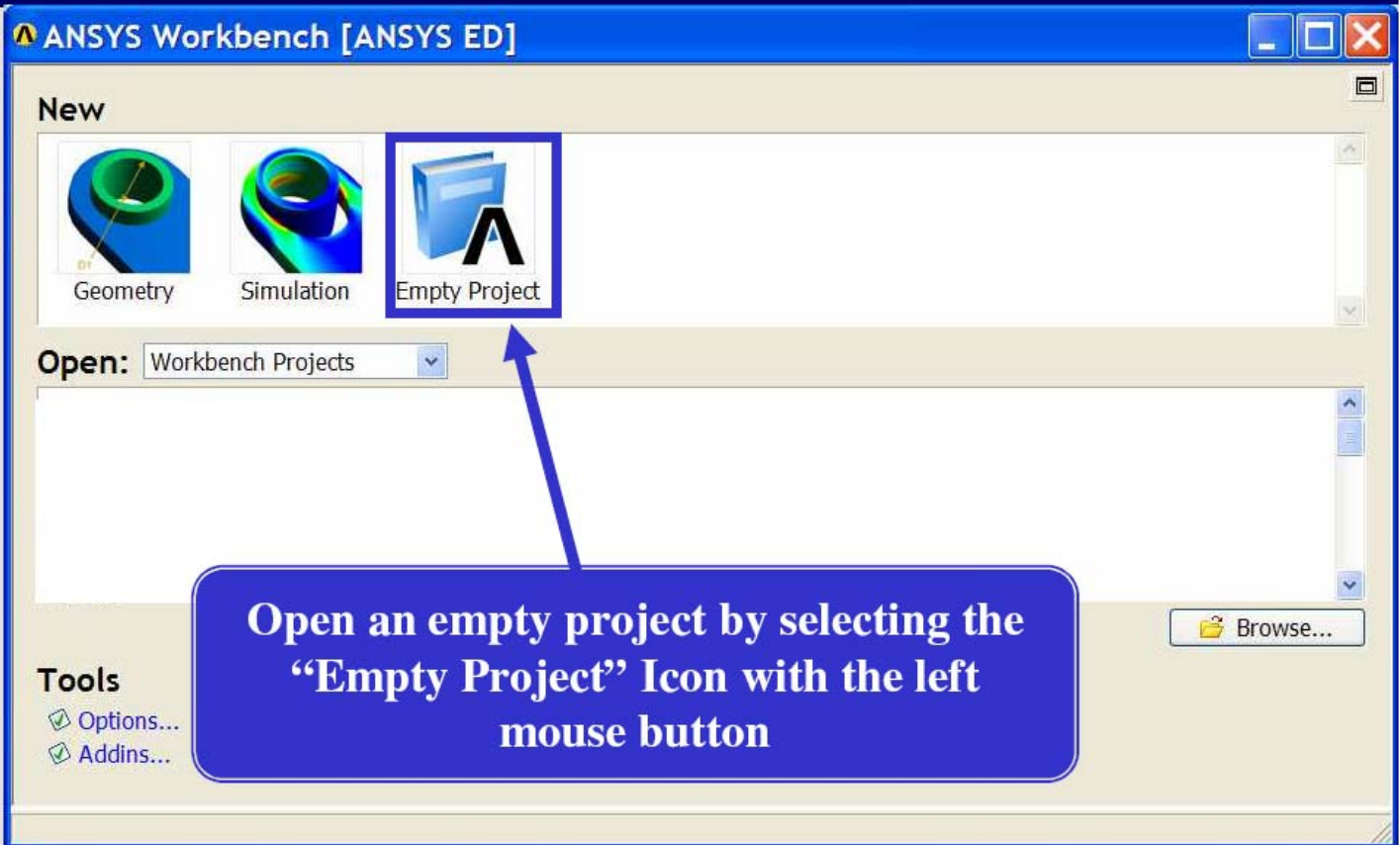
# Getting Started



Launch the ANSYS Workbench



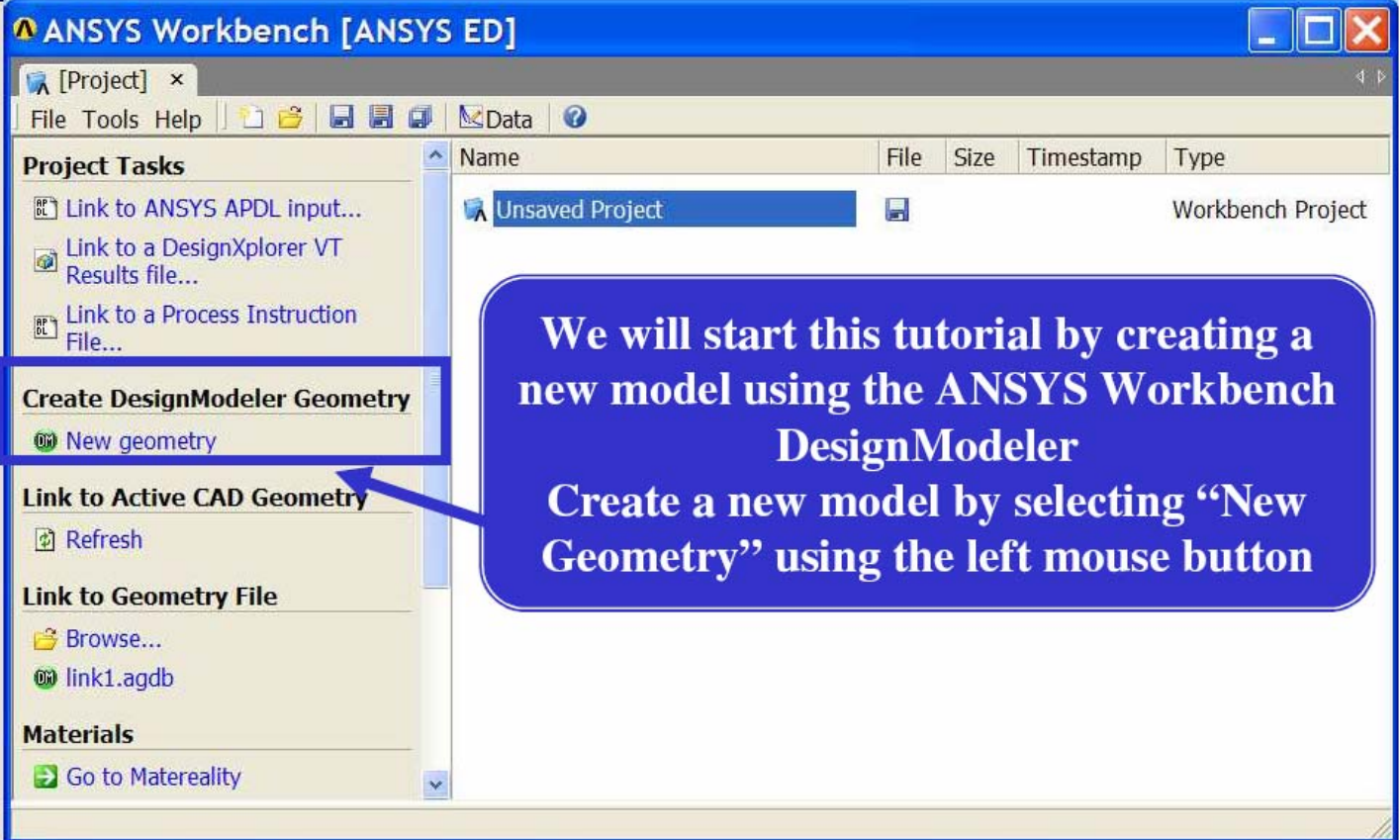
# Open an Empty Project



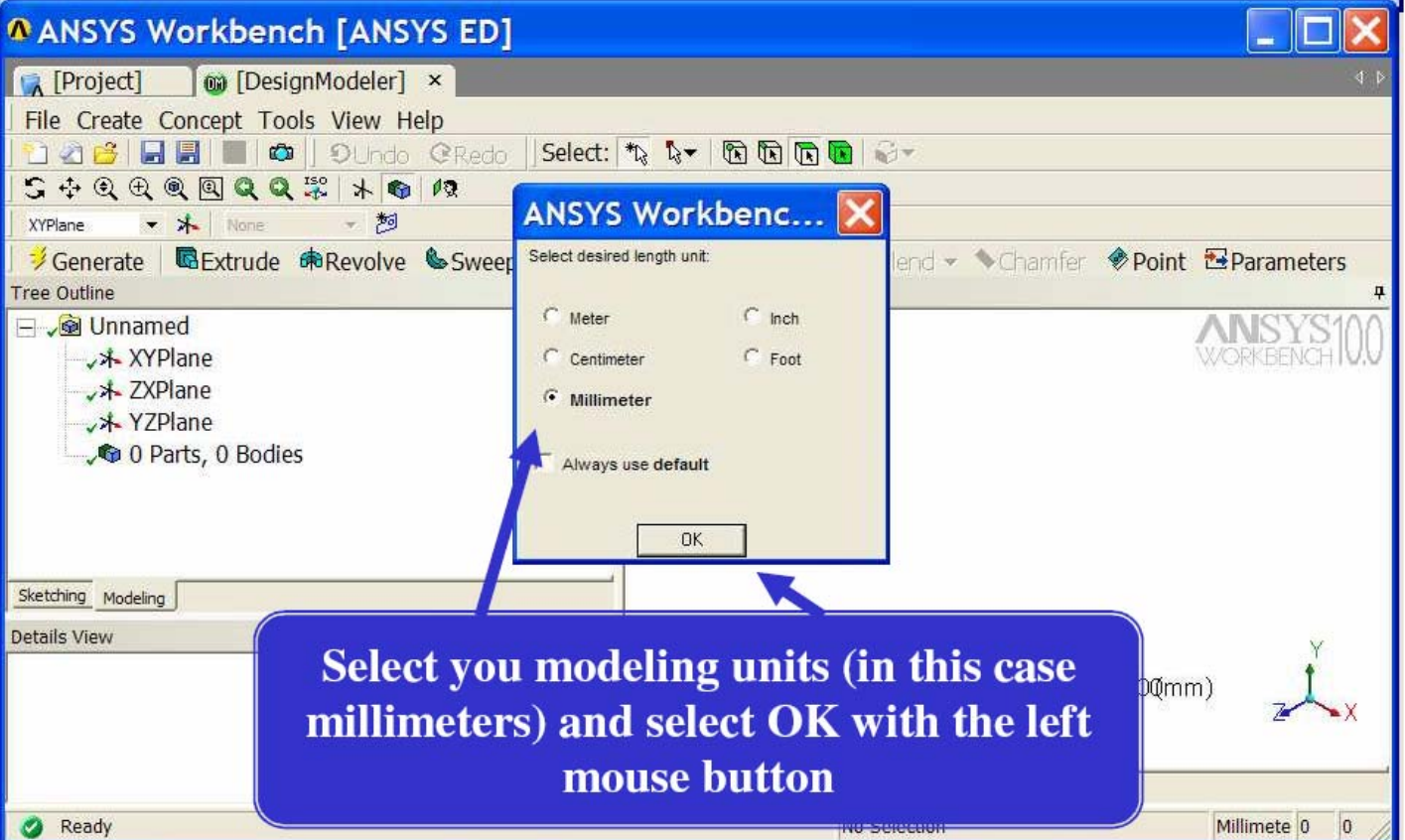
Open an empty project by selecting the “Empty Project” Icon with the left mouse button



# Creating initial geometry



# Building our initial model



- Rectangular Plate with a Uniform Pressure Load
  1. Create geometry
  2. Apply constraints
  3. Apply pressure load
  4. Define required results
  5. Solve
  6. Examine Results
  7. Generate Report

## Step 1A – Create Rectangle

**2. Select Rectangle from the Draw Menu**

**3. Select a starting point (left mouse click – or hold and drag)**

**1. Enter Sketching Mode**

**4. Create Rectangle (left mouse click or release drag)**



# Step 1B – Extrude Rectangle



ANSYS Workbench [ANSYS ED]

**1. Select the Extrude Operation**

© 2006 ANSYS, Inc. All rights reserved. 9 ANSYS, Inc. Proprietary

# Step 1B – Extrude Rectangle



ANSYS Workbench [ANSYS ED]

**2. Select Generate to complete the extrusion**

**1. In the details of extrusion set the depth of the extrusion**

Property	Value
Direction Vector	None (Normal)
Direction	Normal
Type	Fixed
FD1, Depth (>0)	3 mm
As Thin/Surface?	No
Merge Tonnalov?	Yes

Extrude Creation -- Click the Generate button to complete the extrude

© 2006 ANSYS, Inc. All rights reserved. 10 ANSYS, Inc. Proprietary

# Step 1C – Create a Simulation



**1. Select File Save and save your file in a directory of your choice as Exercise1.agdb**

**2. Select the [Project] folder tab**

Save in: Tutorial Files

File name: Exercise1.agdb

Save as type: DesignModeler Geometry (\*.agdb)

© 2006 ANSYS, Inc. All rights reserved. 11 ANSYS, Inc. Proprietary

# Step 2A – Open a new Simulation



**Select (left mouse click) "New Simulation"**

**Note that your new model and file have been added to your project**

DesignModeler Tasks

- Open
- Open copy
- New simulation**

Default Geometry Options

- Solid bodies
- Surface bodies
- Line bodies
- Parameters
- Attributes
- Named selections
- Material properties

Advanced Geometry Defaults

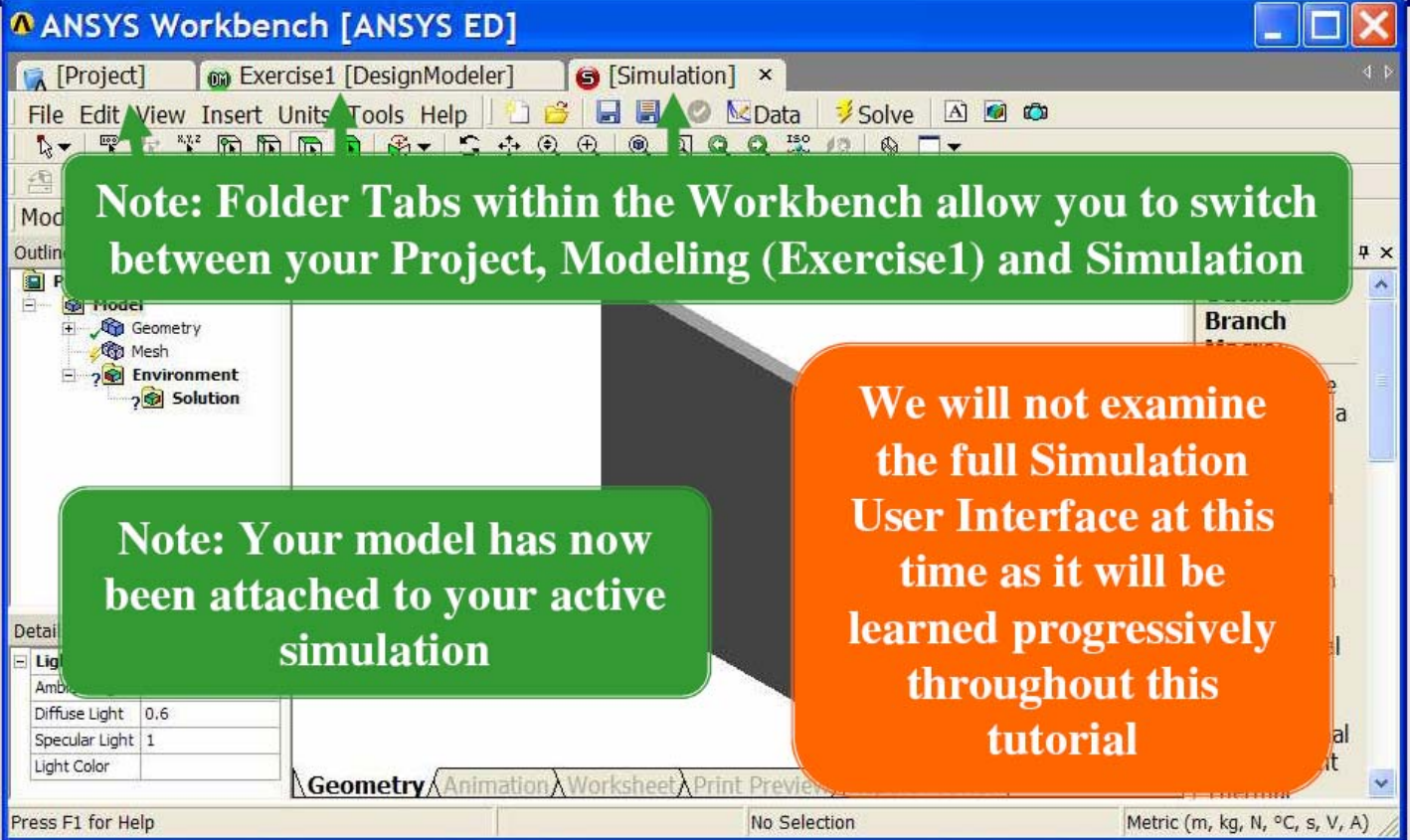
Edit Item

- Rename

© 2006 ANSYS, Inc. All rights reserved. 12 ANSYS, Inc. Proprietary



# Attaching Models & Folder Tabs

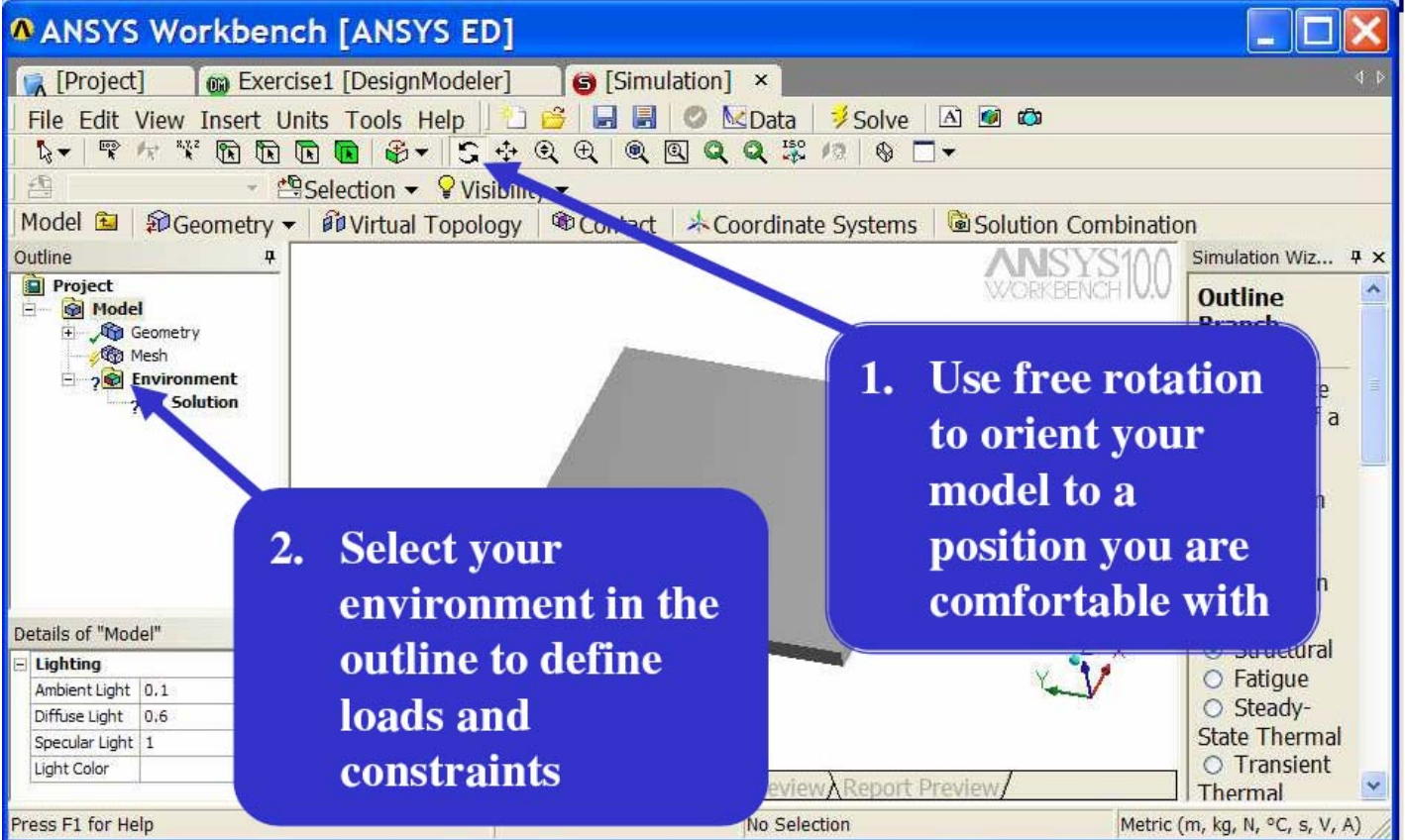


**Note: Folder Tabs within the Workbench allow you to switch between your Project, Modeling (Exercise1) and Simulation**

**Note: Your model has now been attached to your active simulation**

**We will not examine the full Simulation User Interface at this time as it will be learned progressively throughout this tutorial**

# Step 2B – Defining your Environment



**1. Use free rotation to orient your model to a position you are comfortable with**

**2. Select your environment in the outline to define loads and constraints**



## Step 2B – Defining your Environment



The screenshot shows the ANSYS Workbench [ANSYS ED] interface. The top menu bar includes File, Edit, View, Insert, Units, Tools, and Help. Below the menu bar is a toolbar with various icons. The main workspace displays a 3D model of a rectangular block. A green callout box with white text is overlaid on the interface, pointing to the Environment toolbar. The callout text reads: "Note that a new Environment toolbar has been added to the user interface. Feel free to browse available capabilities and options." The Environment toolbar is located at the top of the workspace and includes a folder icon, a dropdown menu set to "Static", "Steps: 1", and three radio buttons for "Structural", "Thermal", and "Electromagnetic". The "Structural" radio button is selected. The left sidebar shows the Project Outline with a tree view containing Model, Geometry, Mesh, Environment, and Solution. The bottom status bar displays "Press F1 for Help", "No Selection", and "Metric (m, kg, N, °C, s, V, A)".

**Note that a new Environment toolbar has been added to the user interface  
Feel free to browse available capabilities and options**

## Step 2B – Defining your Environment



The screenshot shows the ANSYS Workbench [ANSYS ED] interface, similar to the previous one. The Environment toolbar is highlighted, and the "Structural" radio button is selected. A green callout box with white text is overlaid on the interface, pointing to the "Structural" radio button. The callout text reads: "For our purposes we are going to focus on a single step 'Static' structural simulation." The rest of the interface, including the menu bar, toolbar, 3D model, Project Outline, and status bar, is identical to the previous screenshot.

**For our purposes we are going to focus on a single step "Static" structural simulation**



# Step 2B – Defining Supports



**At this point you have two options**

1. Selected “Fixed Support” from the “Structural” pull-down menu

© 2006 ANSYS, Inc. All rights reserved. 17 ANSYS, Inc. Proprietary

# Step 2B – Defining Supports



**Option 2**

1. Using the right mouse button select: “Insert>Fixed Support” from the pop-up menu

© 2006 ANSYS, Inc. All rights reserved. 18 ANSYS, Inc. Proprietary



## Step 2B – Defining Supports

2. Select the Apply function under the Fixed Support Details

1. Using the selection tools previously introduced select the four bounding edges of the your plate

Details of "Fixed Support"

Scope	
Scoping Method	Geometry ...
Geometry	Apply   Cancel
Definition	
Type	Fixed Sup...
Suppressed	No

4 Faces: Surface Area(approx.) = 7.0171e- Metric (m, kg, N, °C, s, V, A)

© 2006 ANSYS, Inc. All rights reserved. 19 ANSYS, Inc. Proprietary

## Step 3A – Defining Loads

2. Select the surface to which the load is to be applied and select apply in the details menu

1. Select a "Pressure" load from the Structural menu or right mouse button pop-up

Details of "Pressure"

Scope	
Scoping Method	Geometry ...
Geometry	Apply   Cancel
Definition	
Type	Pressure
Define As	Constant

1 Face: Surface Area(approx.) = 3.4172e-0 Metric (m, kg, N, °C, s, V, A)

© 2006 ANSYS, Inc. All rights reserved. 20 ANSYS, Inc. Proprietary



# Step 3B – Defining Loads



**3. Define the pressure to be applied to the selected surface(s)**

Details of "Pressure"

Type	Pressure
Define As	Constant
Magnitude	2. Pa
Suppressed	No

Simulation Wizard

Outline Branch Macros

To automate the setup of a simulation branch, choose from the options below then click the Run button.

- Structural
- Fatigue
- Steady-State Thermal
- Transient Thermal

1 Face: Surface Area(approx.) = 3.4172e-0 Metric (m, kg, N, °C, s, V, A)

© 2006 ANSYS, Inc. All rights reserved. 21 ANSYS, Inc. Proprietary

# Step 4A - Defining Results



**Once you have defined your loads and constraint they can be verified by selecting the object from the tree**

**Next you need to set up your solution by selecting "Solution" from the tree**

Details of "Pressure"

Scoping Method	Geometry ...
Geometry	1 Face

Simulation Wizard

Outline Branch Macros

To automate the setup of a simulation branch, choose from the options below then click the Run button.

- Structural
- Fatigue
- Steady-State Thermal
- Transient Thermal

No Selection Metric (m, kg, N, °C, s, V, A)

© 2006 ANSYS, Inc. All rights reserved. 22 ANSYS, Inc. Proprietary



# Step 4A - Defining Results



**ANSYS Workbench [ANSYS ED]**

[Project] Exercise1 [DesignModeler] [Simulation] x

File Edit View Insert Units Tools Help

Solution Static Steps: 1 Tools Stress Strain Deformation Thermal Shape

Project Model Geometry Mesh Environment Fixed Support Pressure Solution

Details of "Solution"

Adaptive Convergence Max Refine... 1 Refinement... 2

Options Save ANSYS... No Solver Type Program Co...

Geometry Animation Worksheet Print Preview Report Preview

Press F1 for Help No Selection Metric (m, kg, N, °C, s, V, A)

© 2006 ANSYS, Inc. All rights reserved. 23 ANSYS, Inc. Proprietary

Note that a the solution toolbar now appears in the User Interface

# Step 4A - Defining Results



**ANSYS Workbench [ANSYS ED]**

[Project] Exercise1 [DesignModeler] [Simulation] x

File Edit View Insert Units Tools Help

Solution Static Steps: 1 Tools Stress Strain Deformation Thermal Shape

Project Model Geometry Mesh Environment Fixed Support Pressure Solution

Details of "So"

Adaptive C Max Refine... Refinement...

Options Save ANSYS... Solver Type

Worksheet Print Preview Report Preview/ Thermal

Press F1 for Help No Selection Metric (m, kg, N, °C, s, V, A)

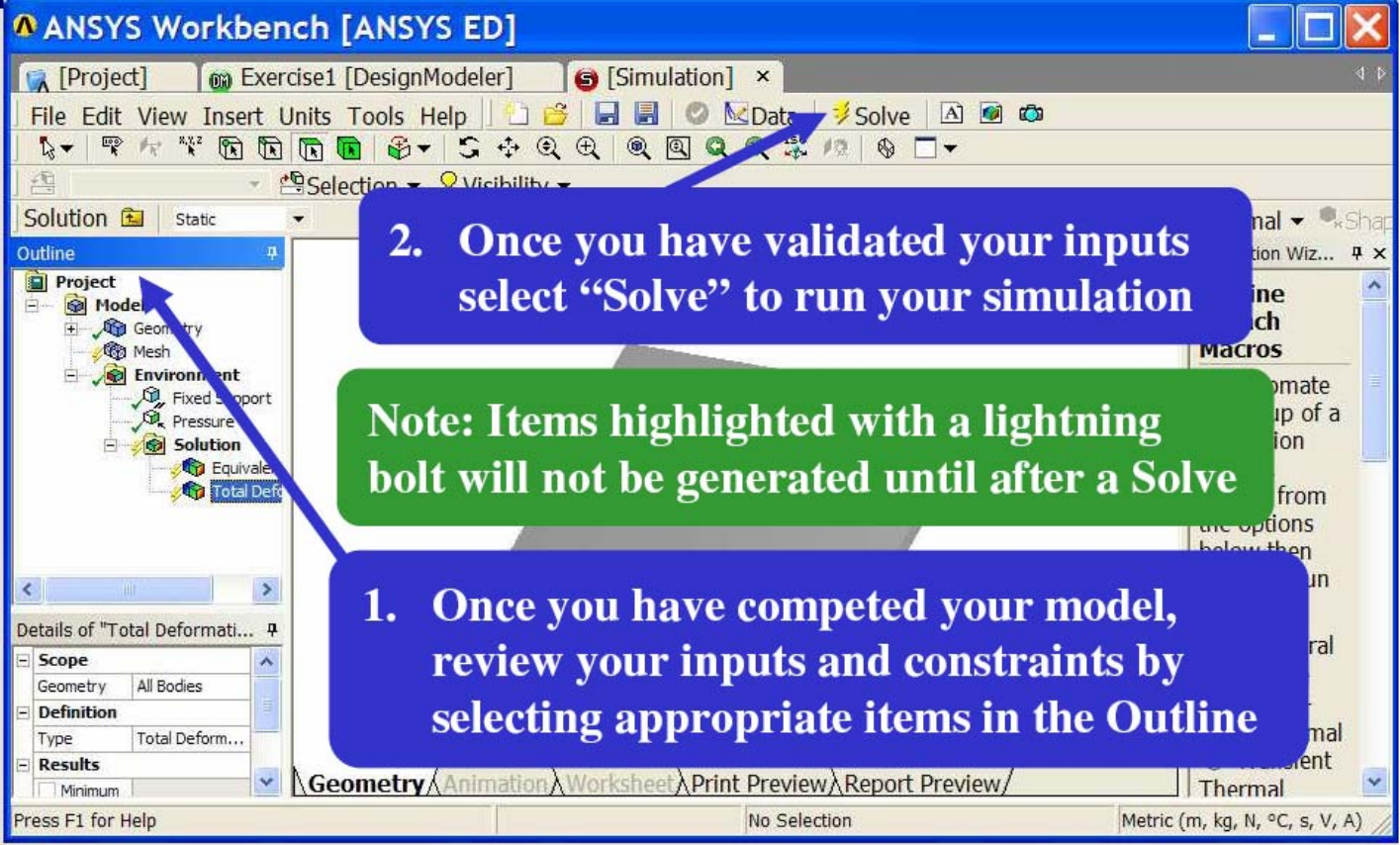
© 2006 ANSYS, Inc. All rights reserved. 24 ANSYS, Inc. Proprietary

1. From the Stress pull-down or right mouse menu define required stress results as "Equivalent (von-Mises)"

2. From the Deformation pull-down or right mouse menu define required deformation results as "Total"



# Step 5A – Validating Inputs

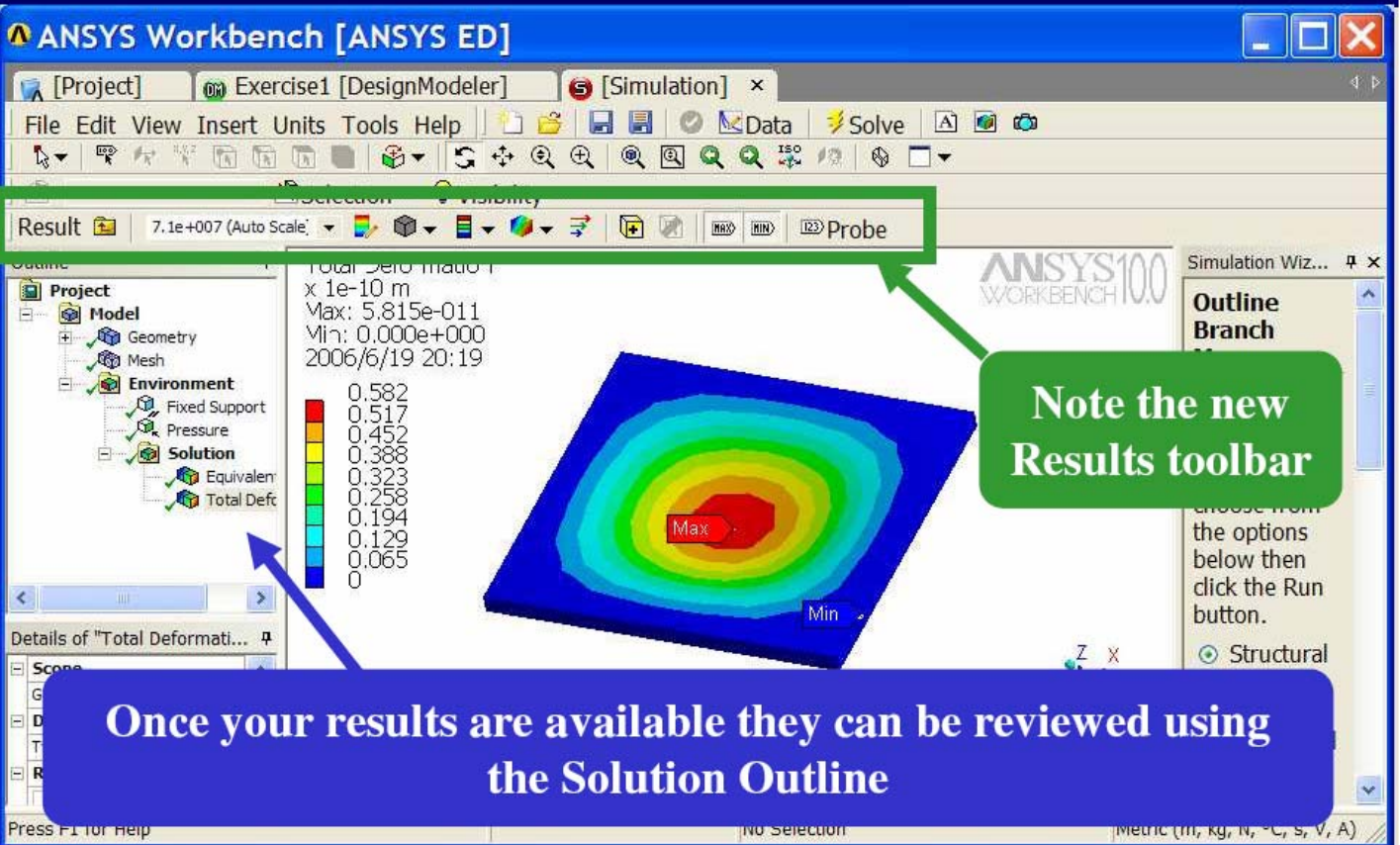


2. Once you have validated your inputs select “Solve” to run your simulation

Note: Items highlighted with a lightning bolt will not be generated until after a Solve

1. Once you have completed your model, review your inputs and constraints by selecting appropriate items in the Outline

# Step 6A – Reviewing Results



Note the new Results toolbar

Once your results are available they can be reviewed using the Solution Outline



# Step 6B – Reviewing Animation



**To animate results select the Animation tab at the bottom of the Results view**

© 2006 ANSYS, Inc. All rights reserved. 27 ANSYS, Inc. Proprietary

# Step 6C - Manipulation



**1. Start your animation by selecting Play in the toolbar**

**2. You can use view manipulation while running animations**

**Note the new Animation toolbar**

© 2006 ANSYS, Inc. All rights reserved. 28 ANSYS, Inc. Proprietary



# Step 7A – Define Figures



**3. Select figure creation from the Simulation toolbar**

**2. To create a report figure select an item in the outline and use view manipulation to define the figure**

**1. To add graphical figure to your report you must first define and save Figures**

**Reports can be generated at any time during a Simulation**

© 2006 ANSYS, Inc. All rights reserved. 29 ANSYS, Inc. Proprietary

# Step 7B – Generate a Report

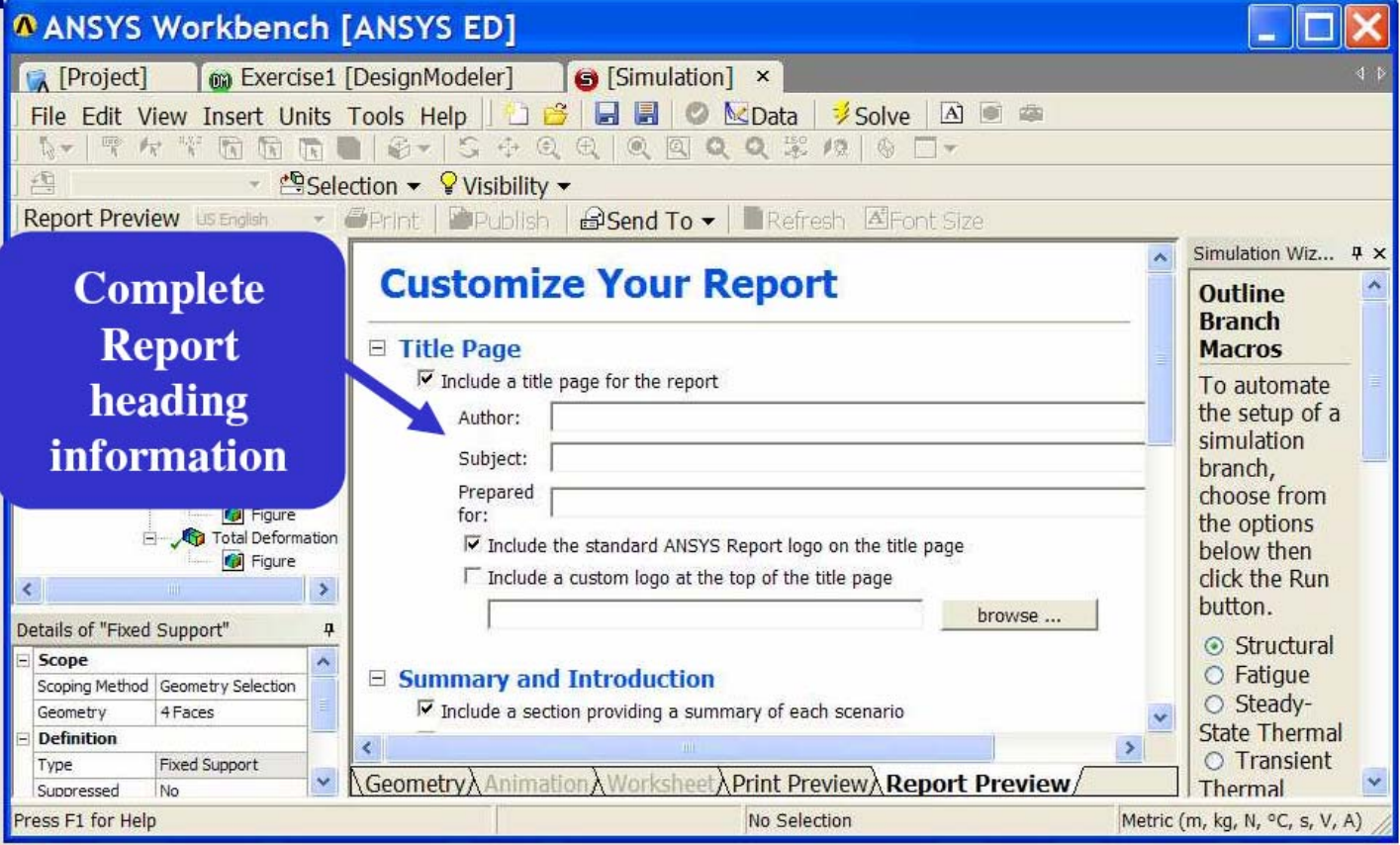


**1. To generate a report for your simulation select the “Report Preview” folder tab below your simulation window**

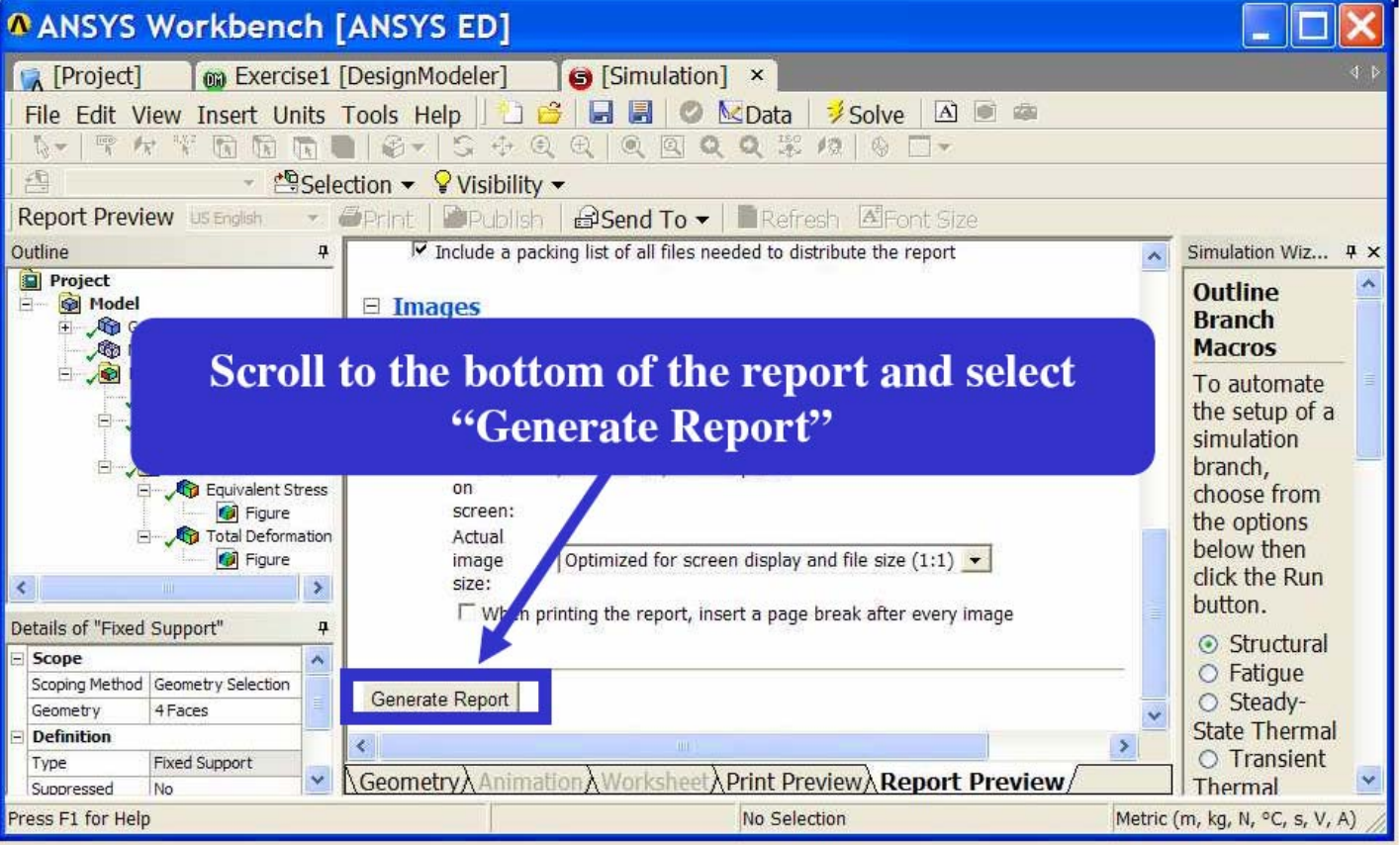
© 2006 ANSYS, Inc. All rights reserved. 30 ANSYS, Inc. Proprietary



# Step 7C – Complete Heading



# Step 7D – Generate the Report





# Step 7E – Publish Report



1. If you have modified your report select “Refresh”

2. Review your Report content

3. Publish or send your report to others for review in a Browser (No ANSYS License Required)

ANSYS Workbench [ANSYS FD1]

[Project] Exercise 1

File Edit View Insert Utilities

Selection Visibility

Report Preview US English Print Publish Send To Refresh Font Size

Outline

ANSYS Report

**Project**

**Project Created**  
Monday, June 19, 2006 at 4:36:08 PM

**Project Last Modified**  
Monday, June 19, 2006 at 4:36:08 PM

**Report Created**  
Monday, June 19, 2006 at 10:33:57 PM

Details of "Fixed Support"

Scope	
Scoping Method	Geometry Selection
Geometry	4 Faces

Definition	
Type	Fixed Support
Suppressed	No

Press F1 for Help

© 2006 ANSYS, Inc. All rights reserved. 33 ANSYS, Inc. Proprietary

# Congratulations



- You have complete Exercise 1 of the ANSYS ED tutorial
- Before you leave the ANSYS Workbench save the results of this exercise
- Return to the Project Page by selecting the [Project] folder tab

# Save your work



**Select "Save All" from the "File Menu" or toolbar ICON**

geometry from **Exercise1**.

Name	File	Size	Timestamp	Type
Exercise1	Exercise1.wbdb	2 KB	6/19/2006 10:56:32 PM	Workbench Pr
Exercise1	Exercise1.agdb	10 KB	6/19/2006 10:55:31 PM	DesignModeler
Model	Exercise1.dsdB	39 KB	6/19/2006 10:55:30 PM	Simulation

# Review your Project Contents



**Note: Your Project and associated files have now been saved in a common location**

**If you wish to exit this session and continue later Exit the Workbench**

Name	File	Size	Timestamp	Type
Exercise1	Exercise1.wbdb	2 KB	6/19/2006 10:56:32 PM	Workbench Pr
Exercise1	Exercise1.agdb	10 KB	6/19/2006 10:55:31 PM	DesignModeler
Model	Exercise1.dsdB	39 KB	6/19/2006 10:55:30 PM	Simulation