ANSYS

ANSYS ED
Workbench Tutorial

ANSYS Workbench Basics

Using this Tutorial

ANSYS

The following guidelines are provided when taking the tutorial

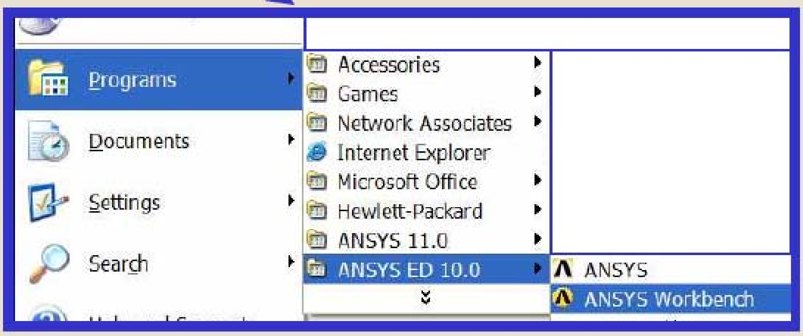
Green boxes are guides describing various Workbench features but requiring no action on your part

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

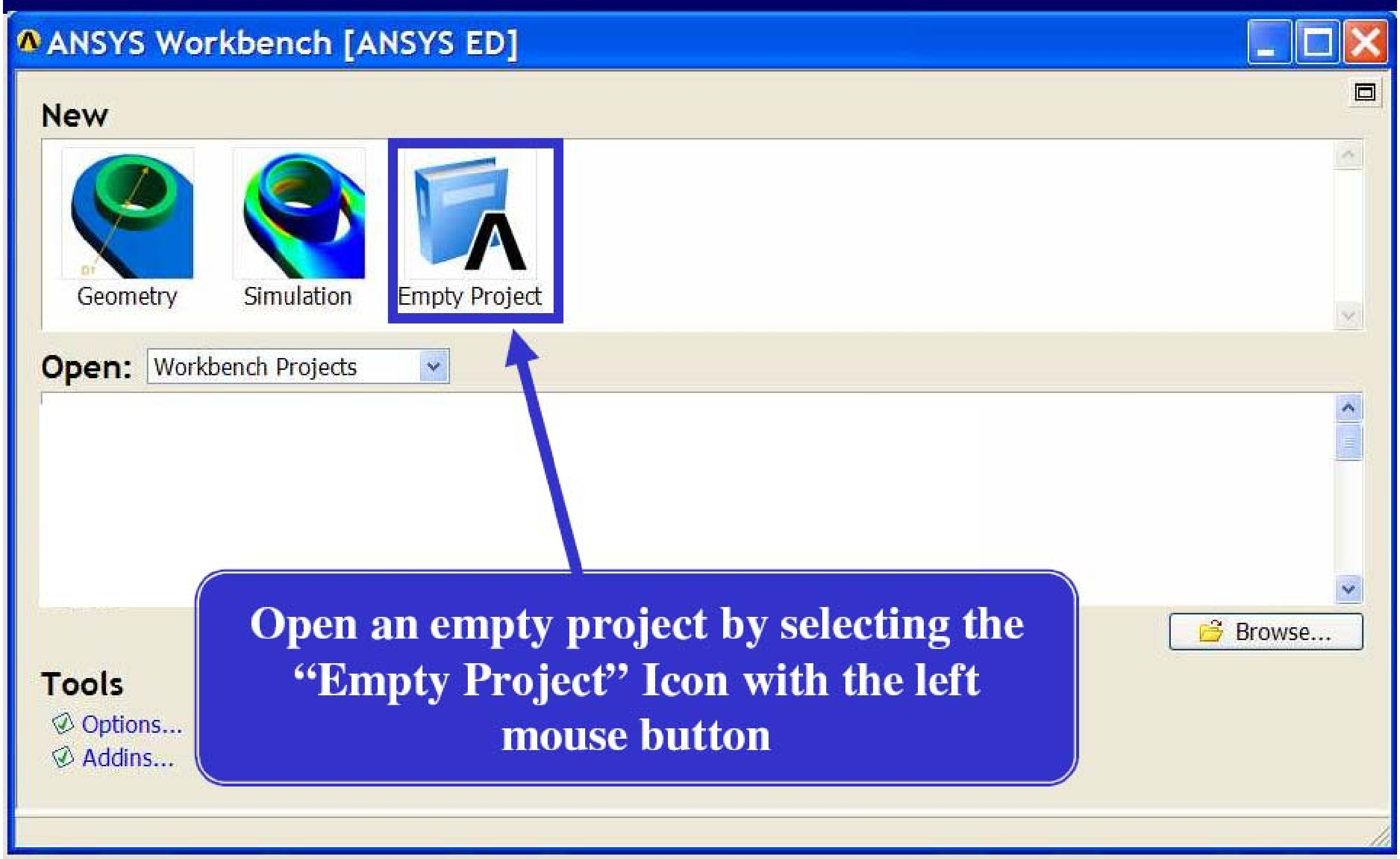
Orange boxes present warnings or notes of interest or importance

Getting Started

Launch the ANSYS Workbench

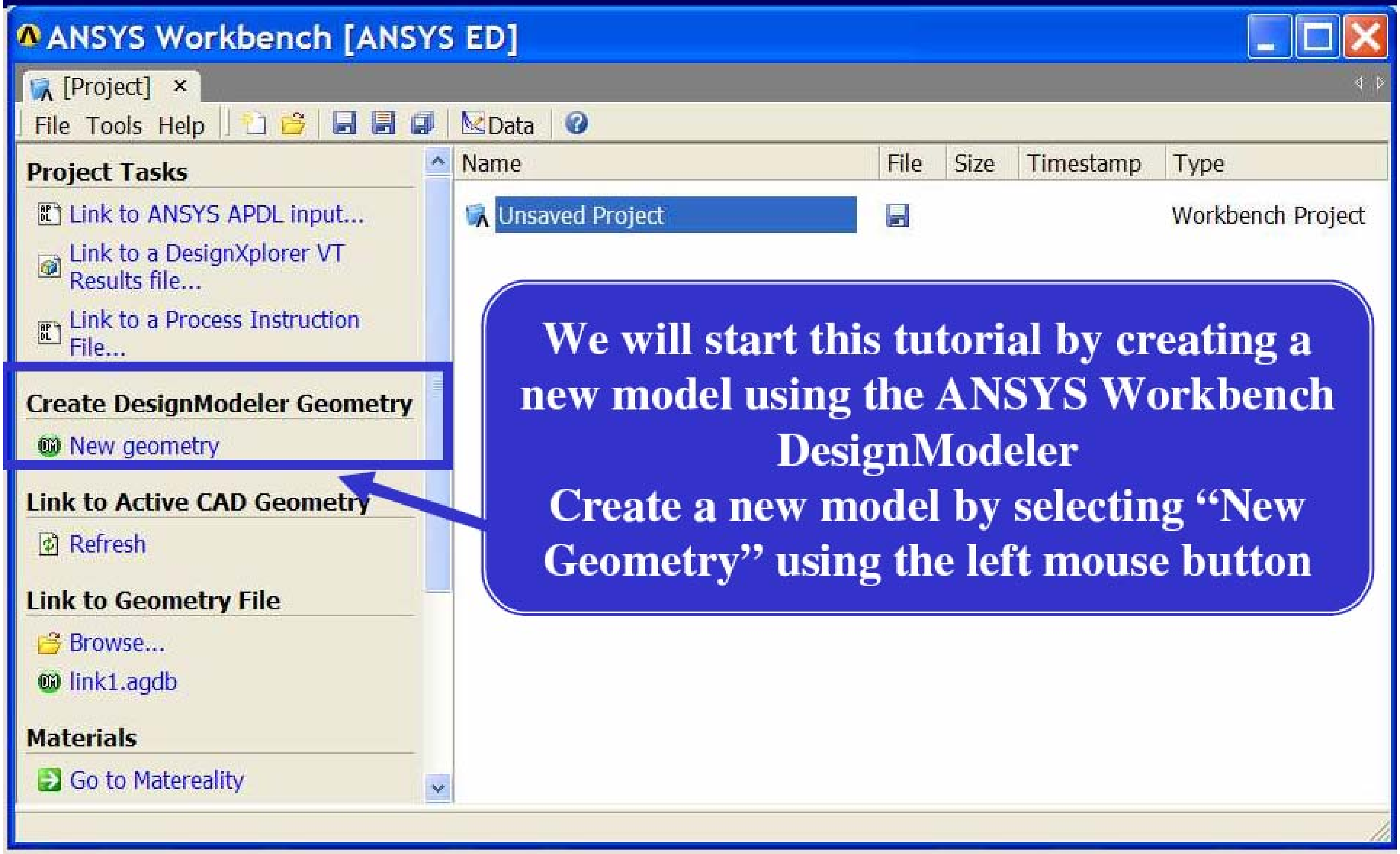


Open an Empty Project



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

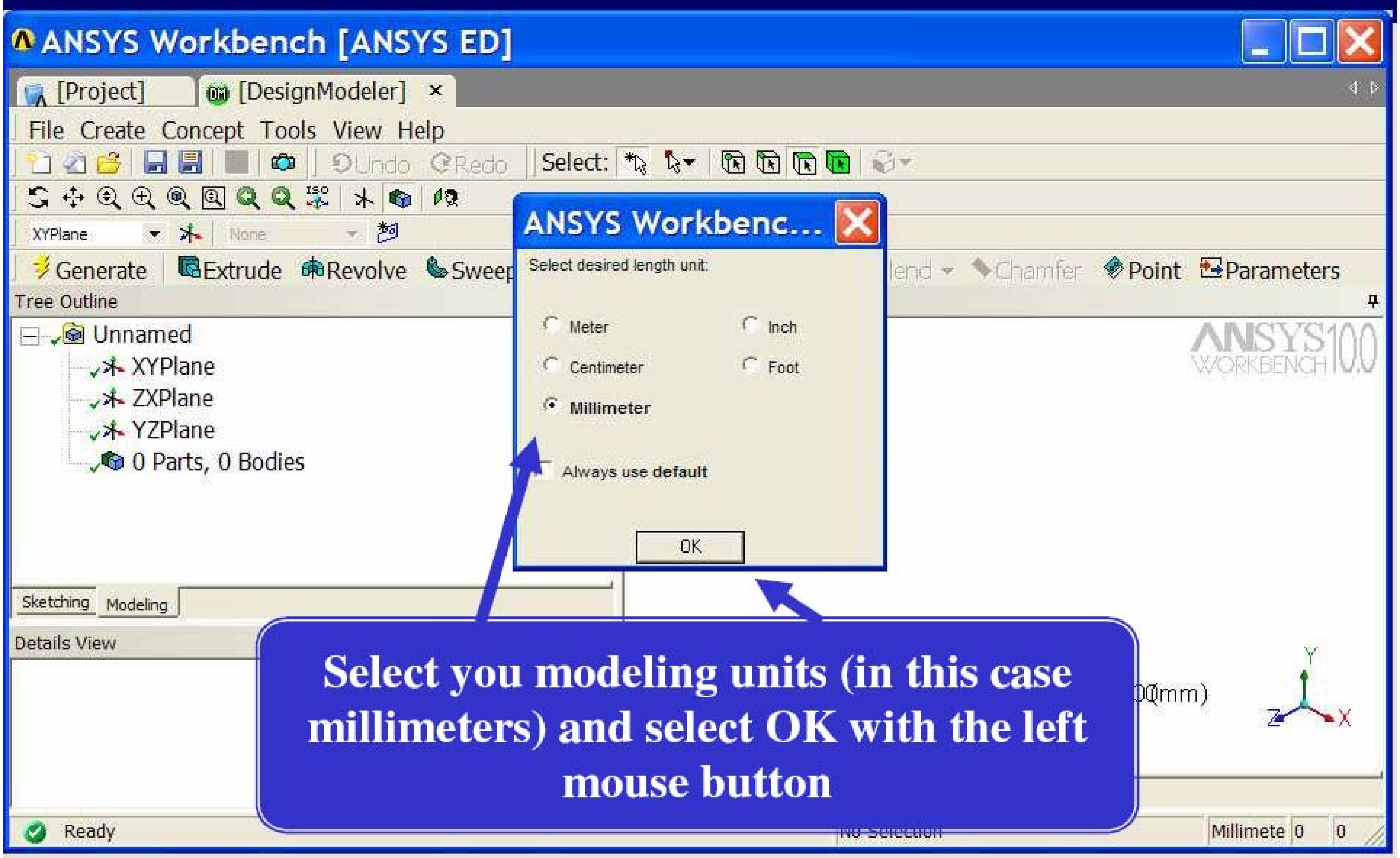
Creating initial geometry



We will start this tutorial by creating a new model using the ANSYS Workbench DesignModeler

Create a new model by selecting "New Geometry" using the left mouse button

Building our initial model

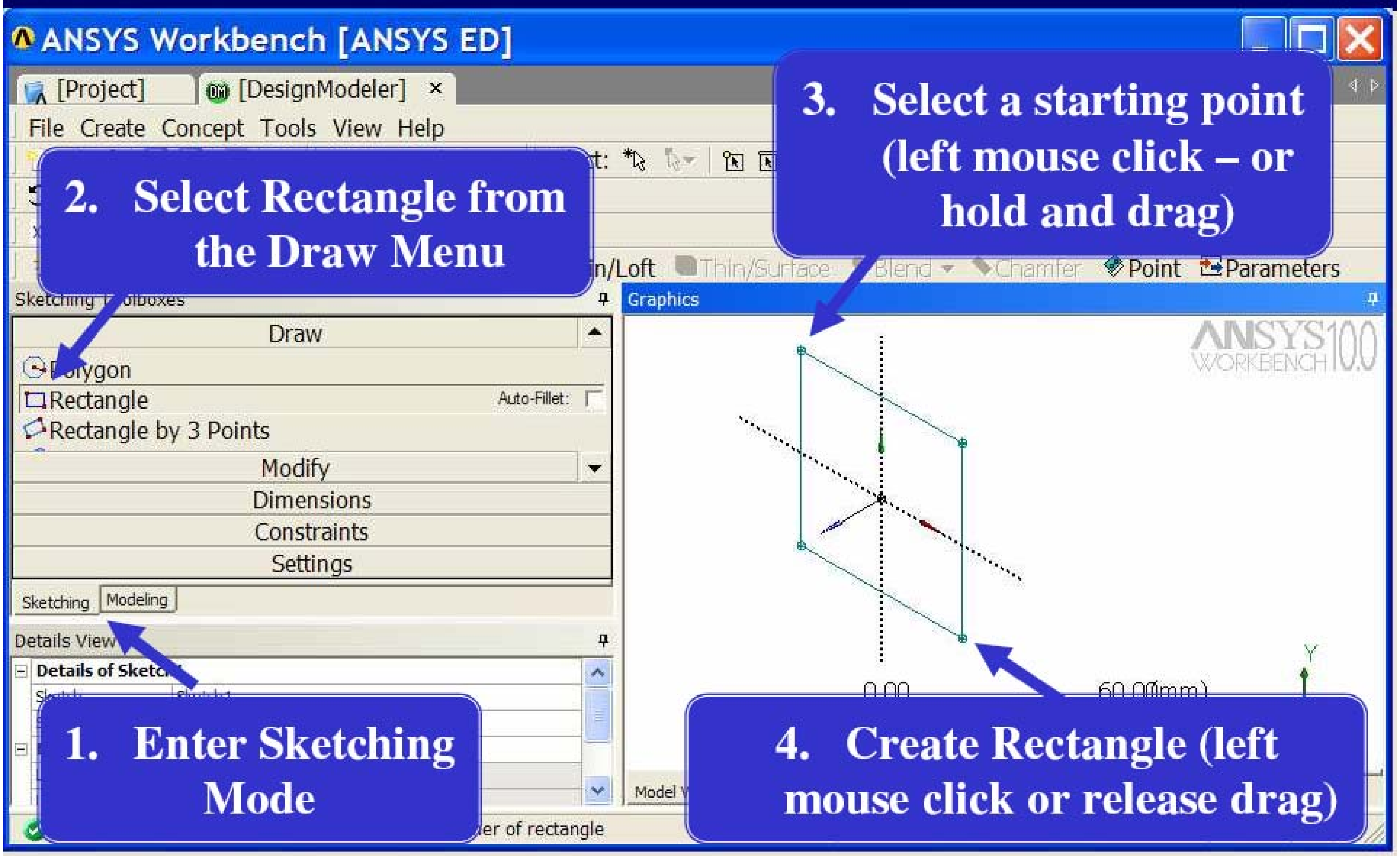


Select you modeling units (in this case millimeters) and select OK with the left mouse button

Exercise 1 - Outline

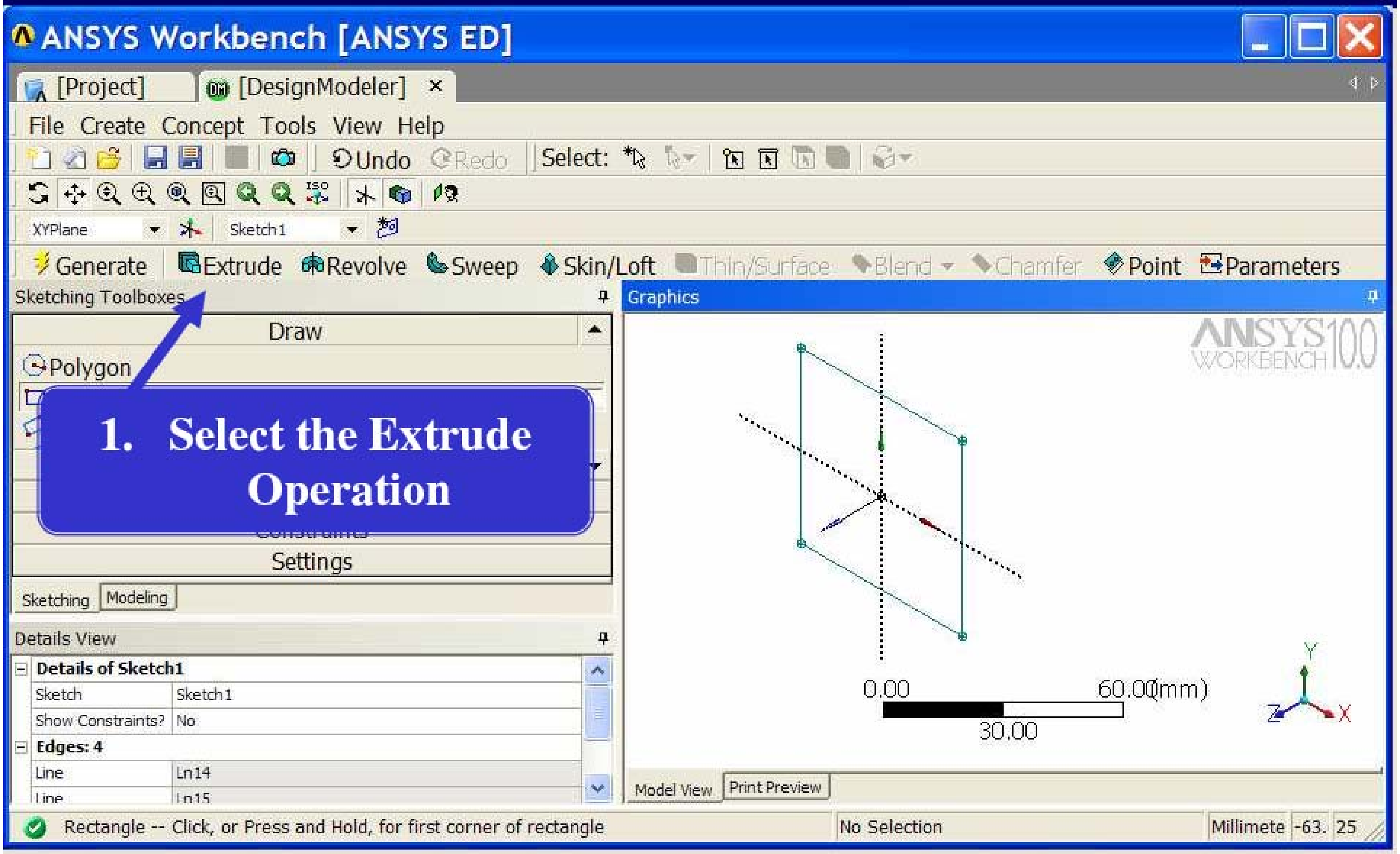
- Rectangular Plate with a Uniform Pressure Load
  1. Create geometry
  2. Apply constraints
  3. Apply pressure load
  4. Define required results
  5. Solve
  6. Examine Results
  7. Generate Report

Step 1A – Create Rectangle



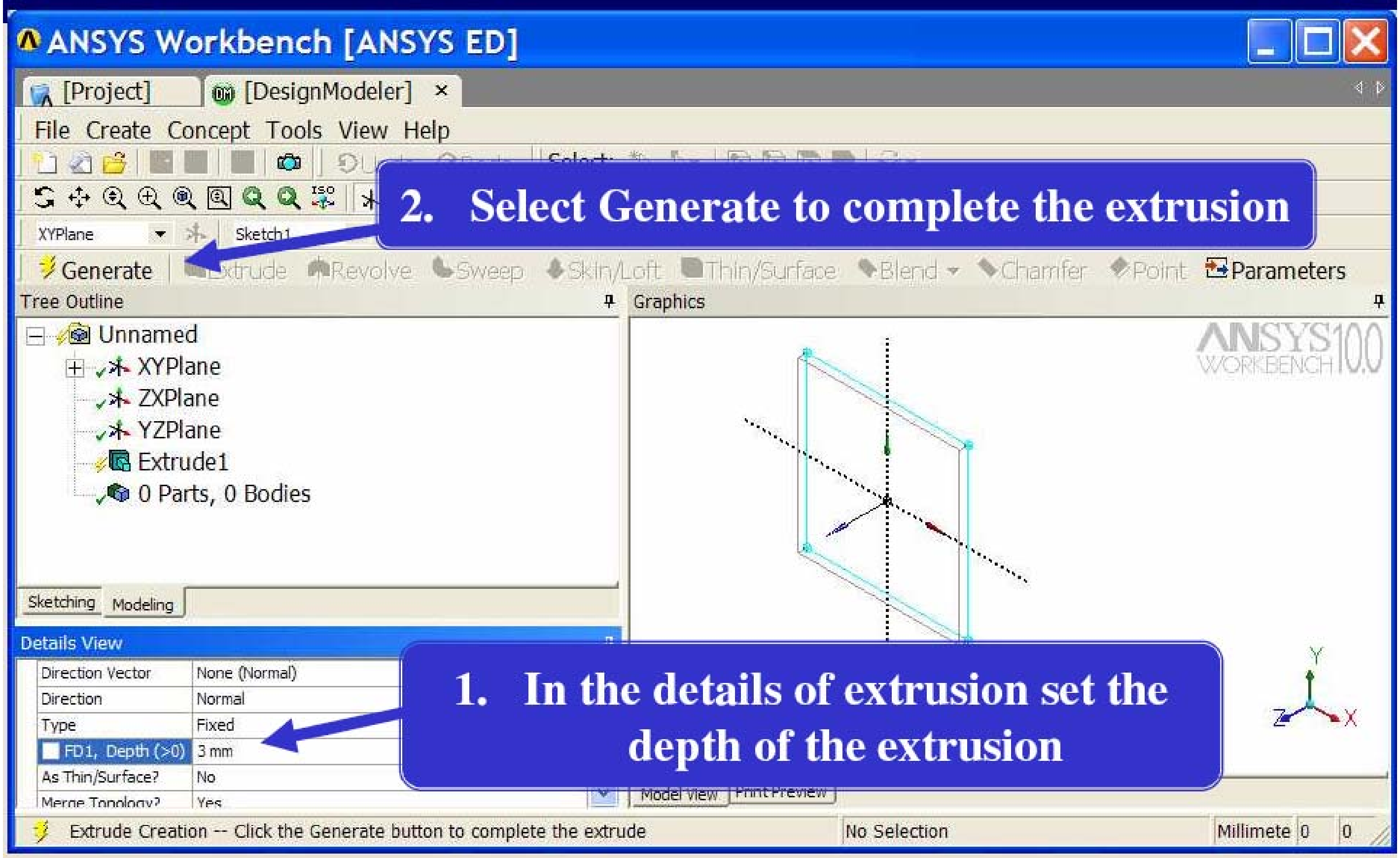
Step 1B – Extrude Rectangle

ANSYS

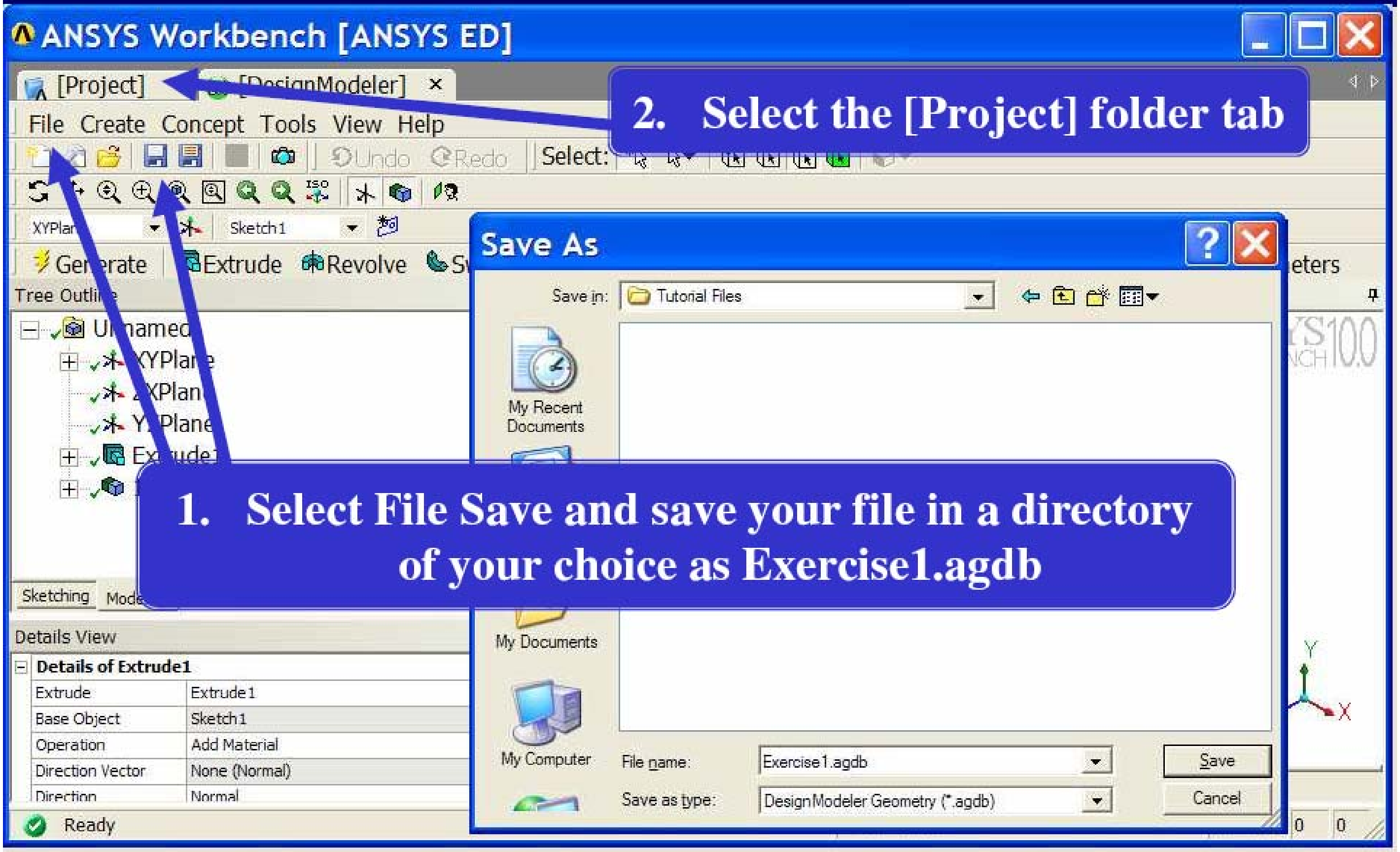


Step 1B – Extrude Rectangle

ANSYS



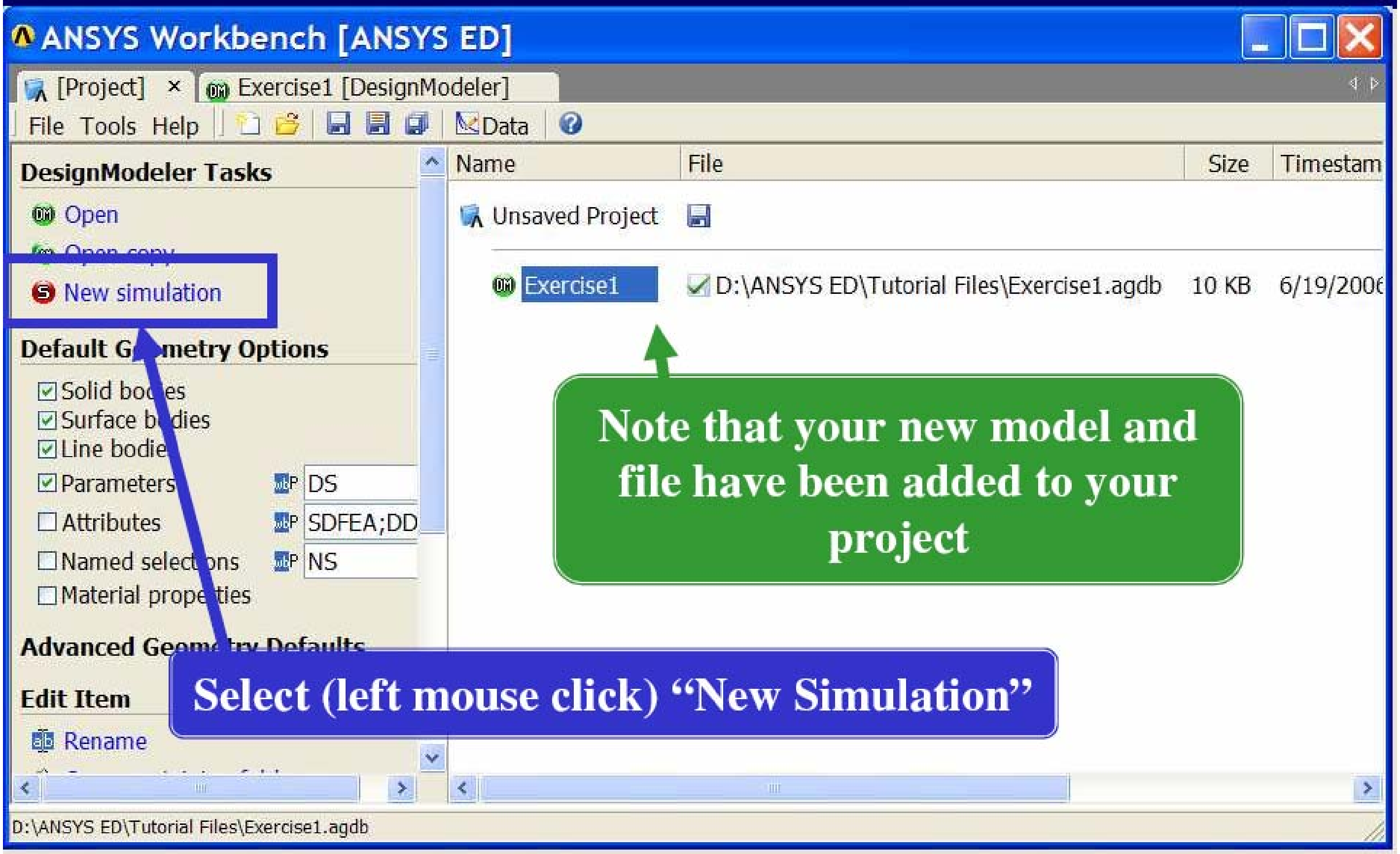
Step 1C – Create a Simulation



2. Select the [Project] folder tab

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

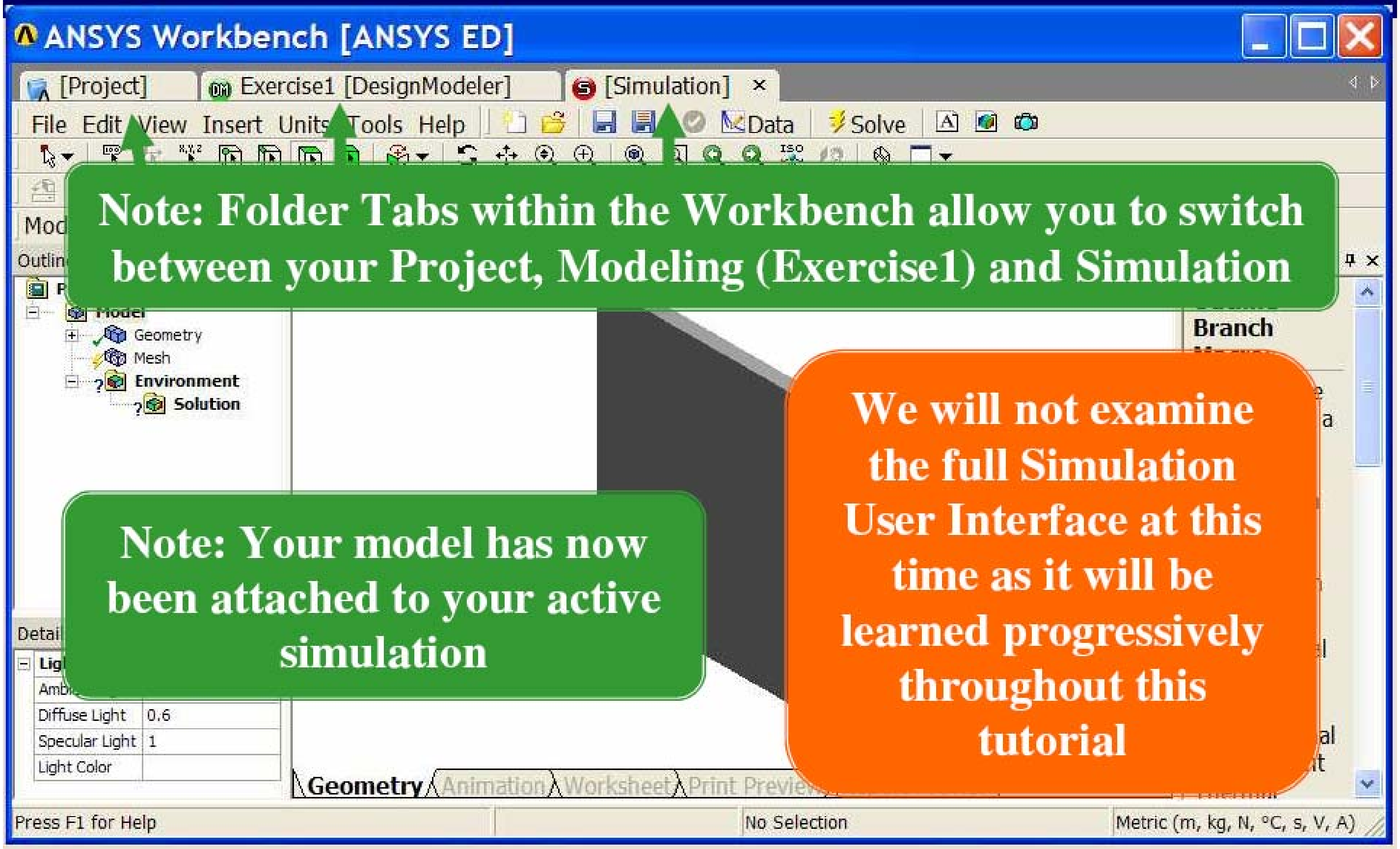
Step 2A – Open a new Simulation



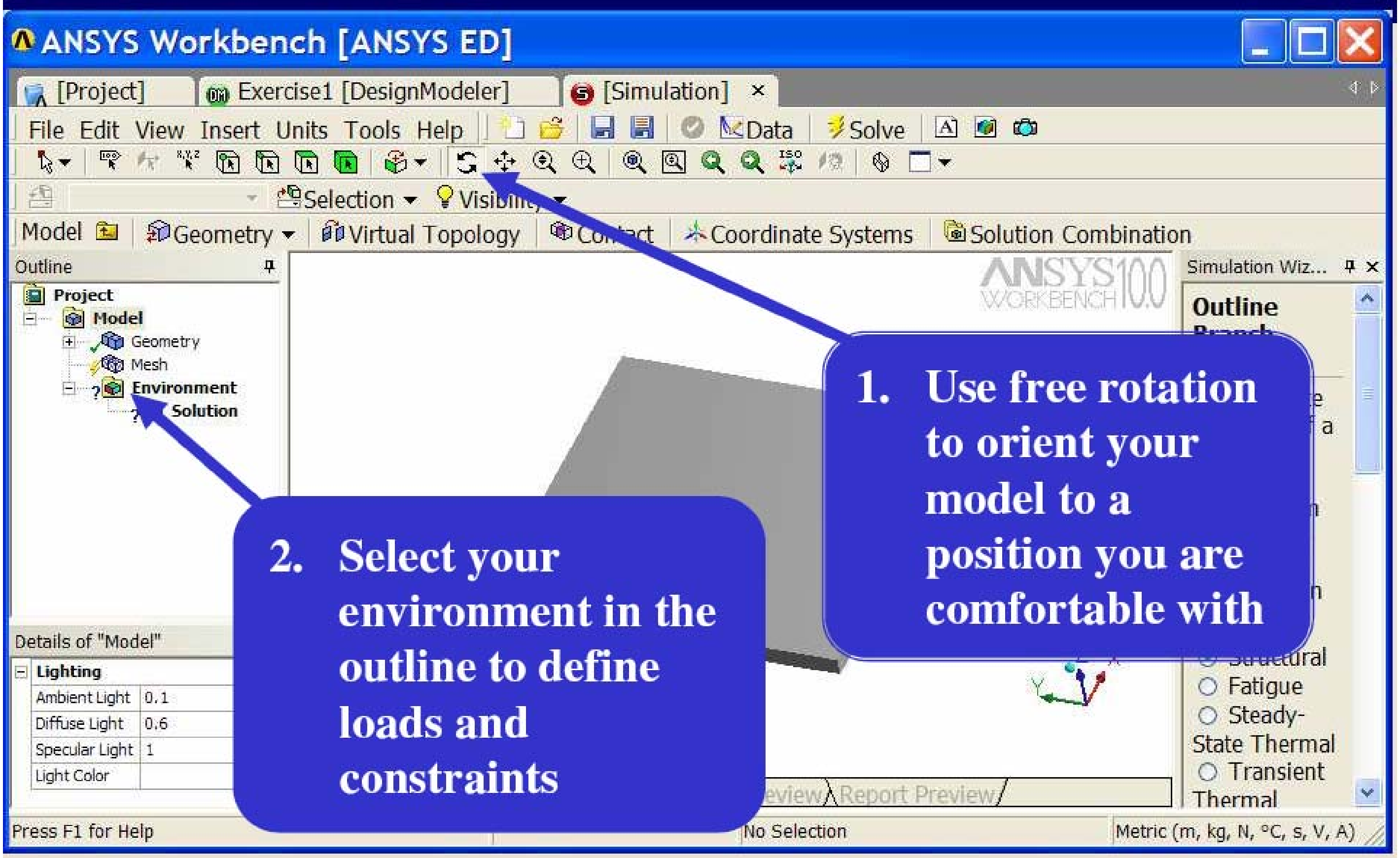
Note that your new model and file have been added to your project

Select (left mouse click) "New Simulation"

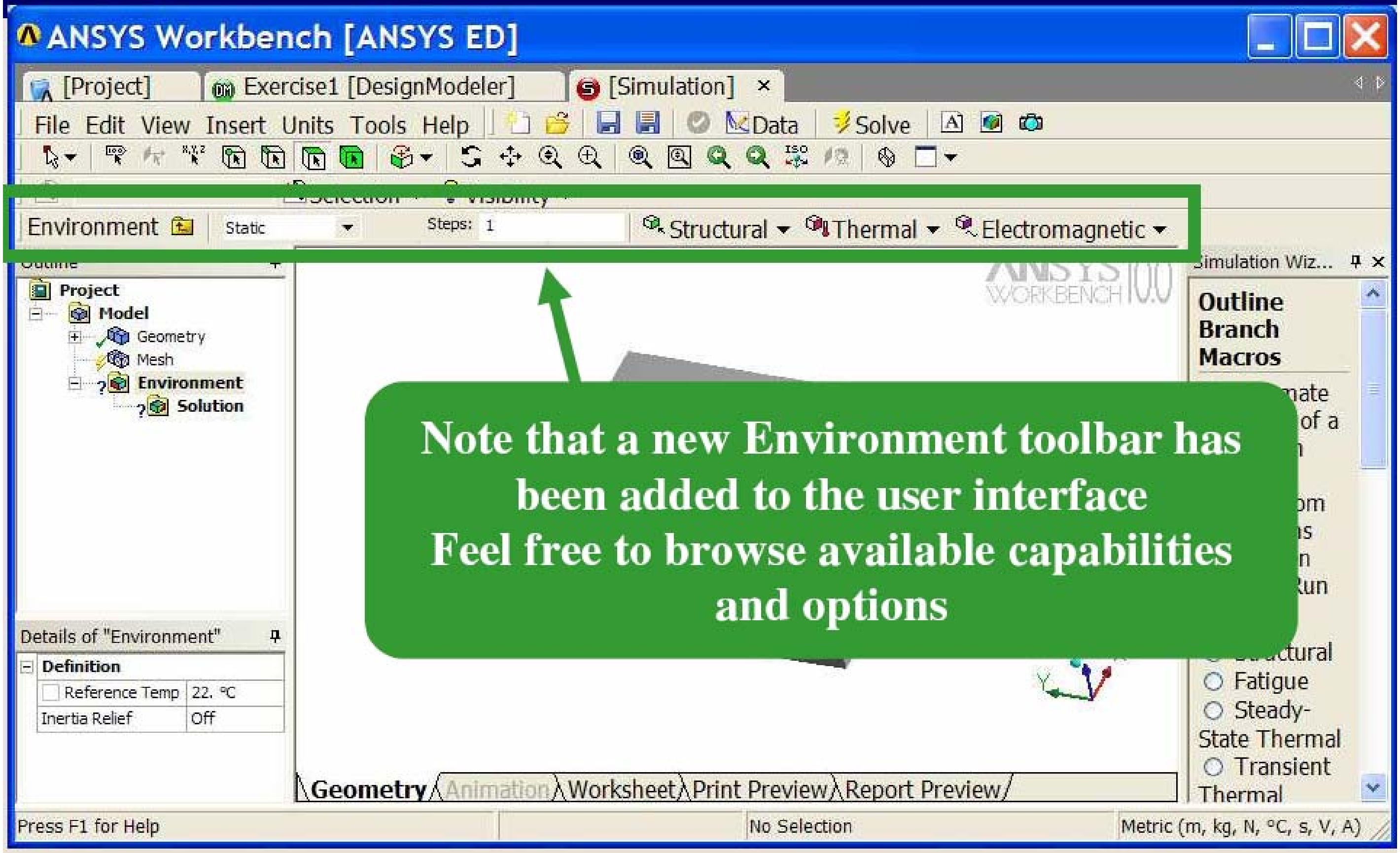
Attaching Models & Folder Tabs



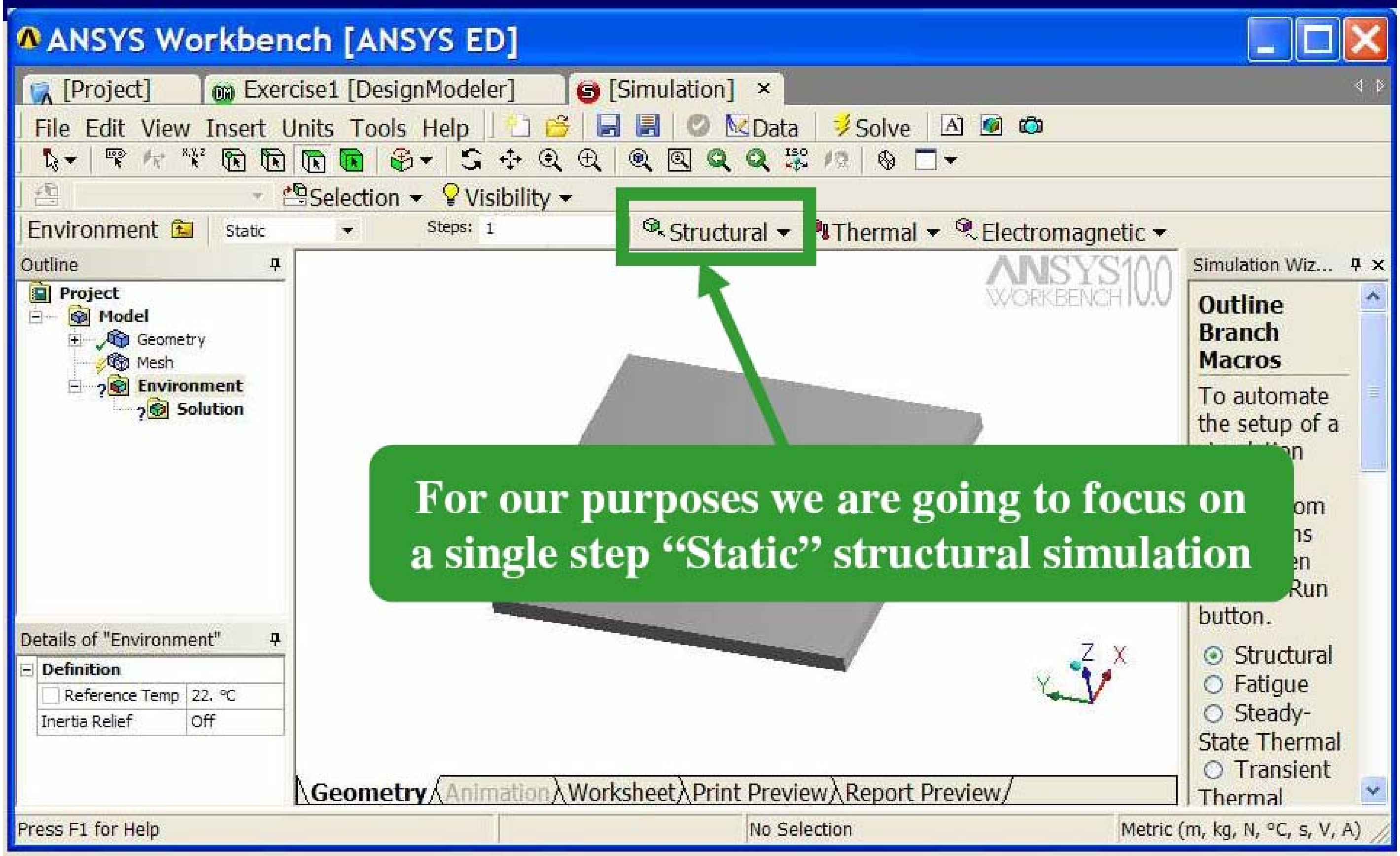
Step 2B – Defining your Environment



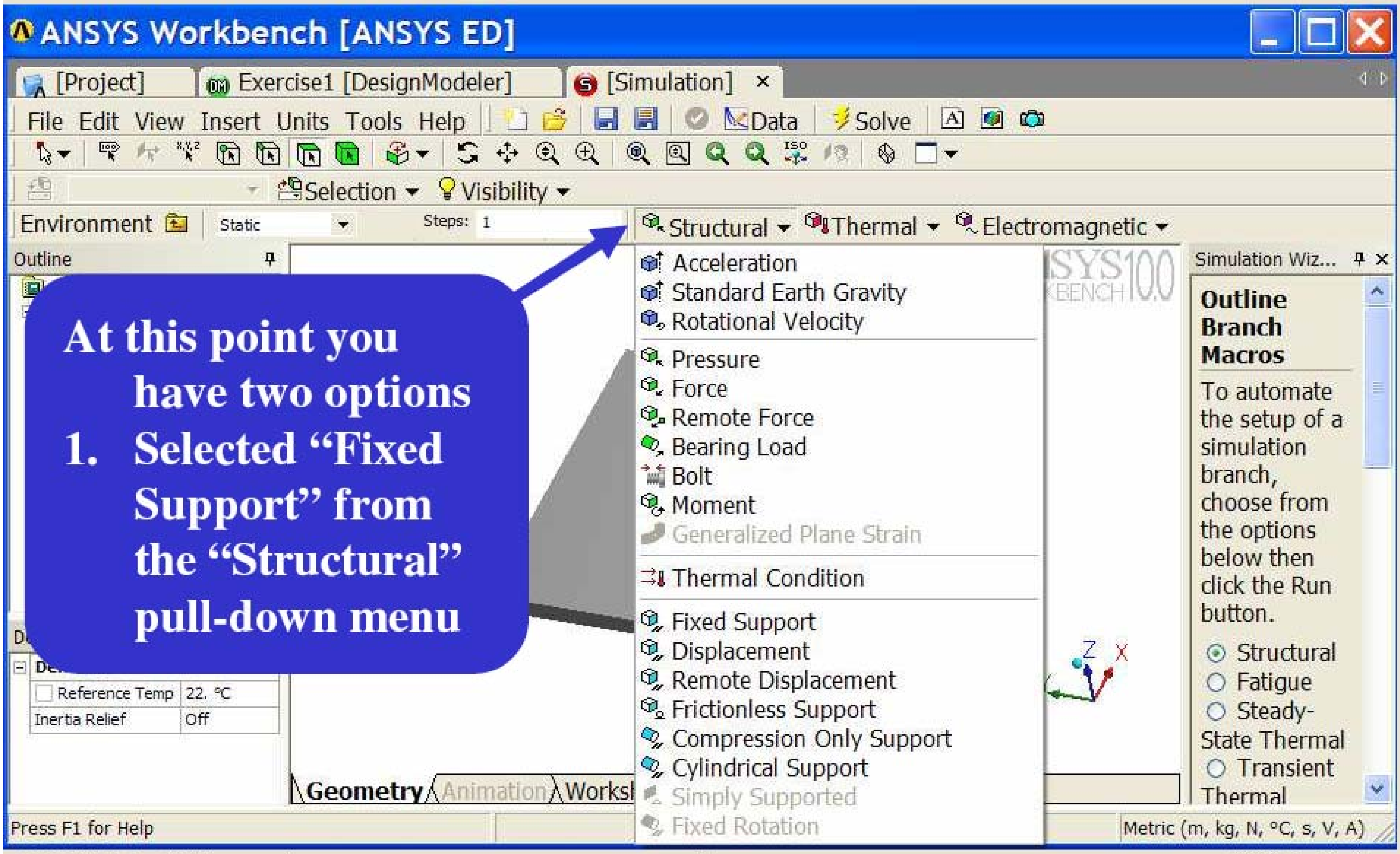
Step 2B – Defining your Environment



Step 2B – Defining your Environment



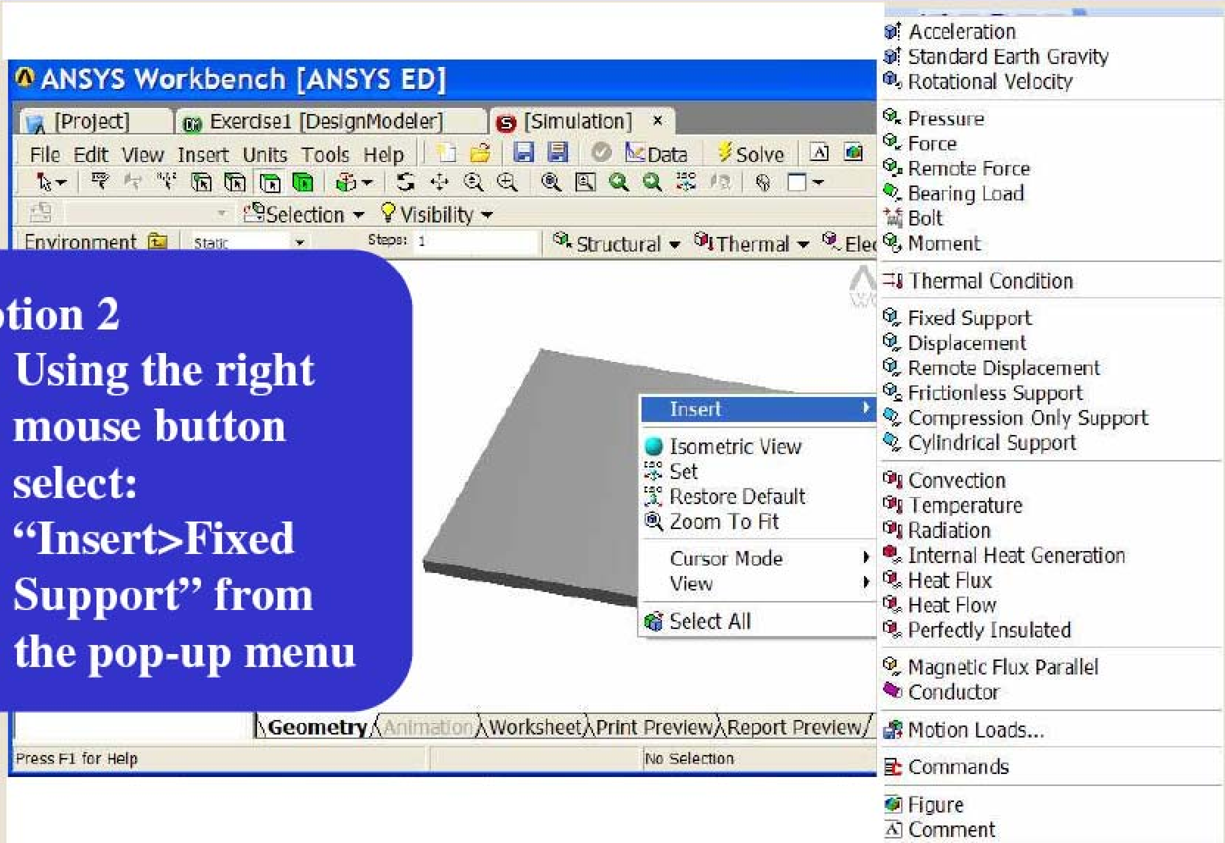
Step 2B – Defining Supports



At this point you have two options

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

Step 2B – Defining Supports

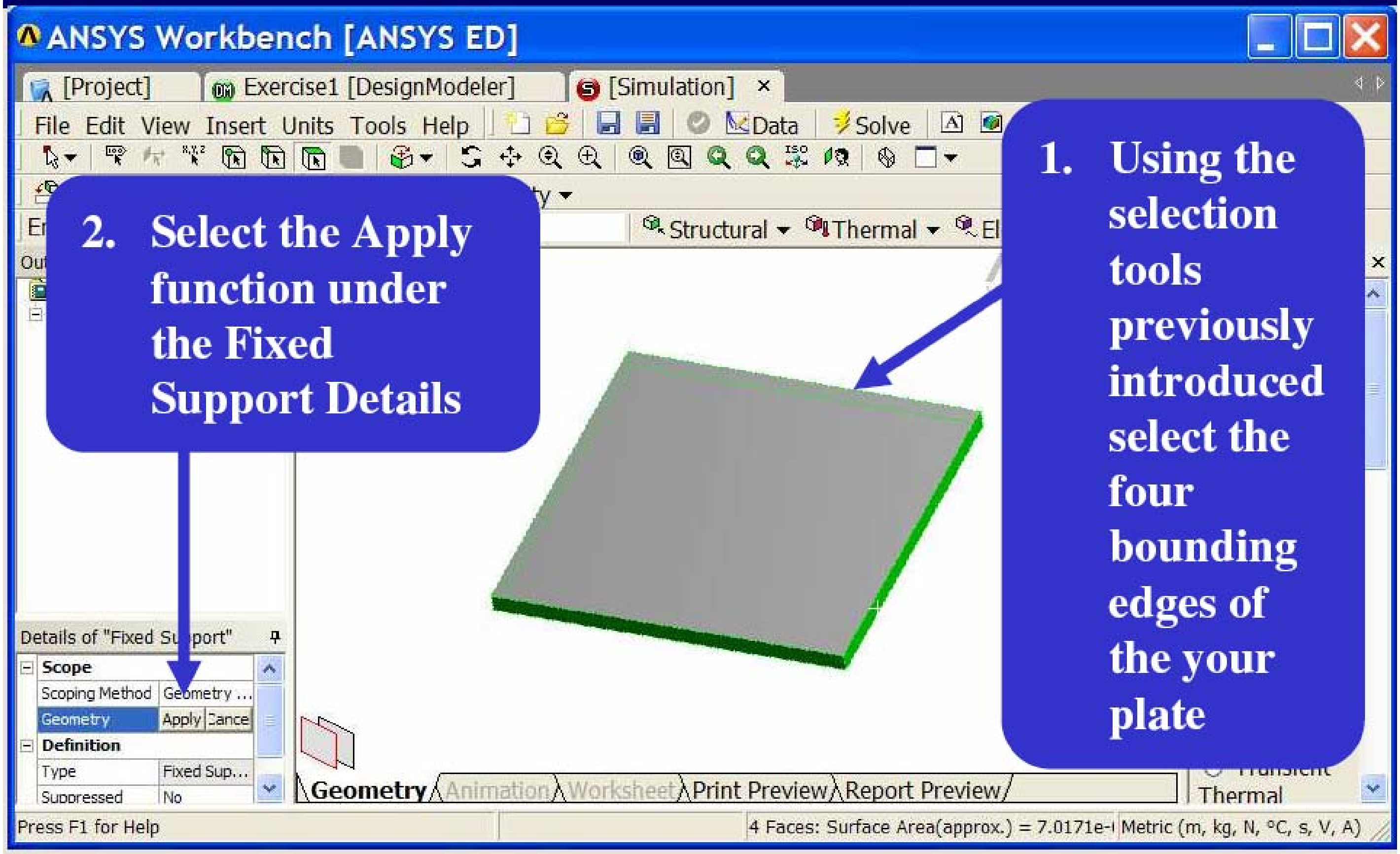


Option 2

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

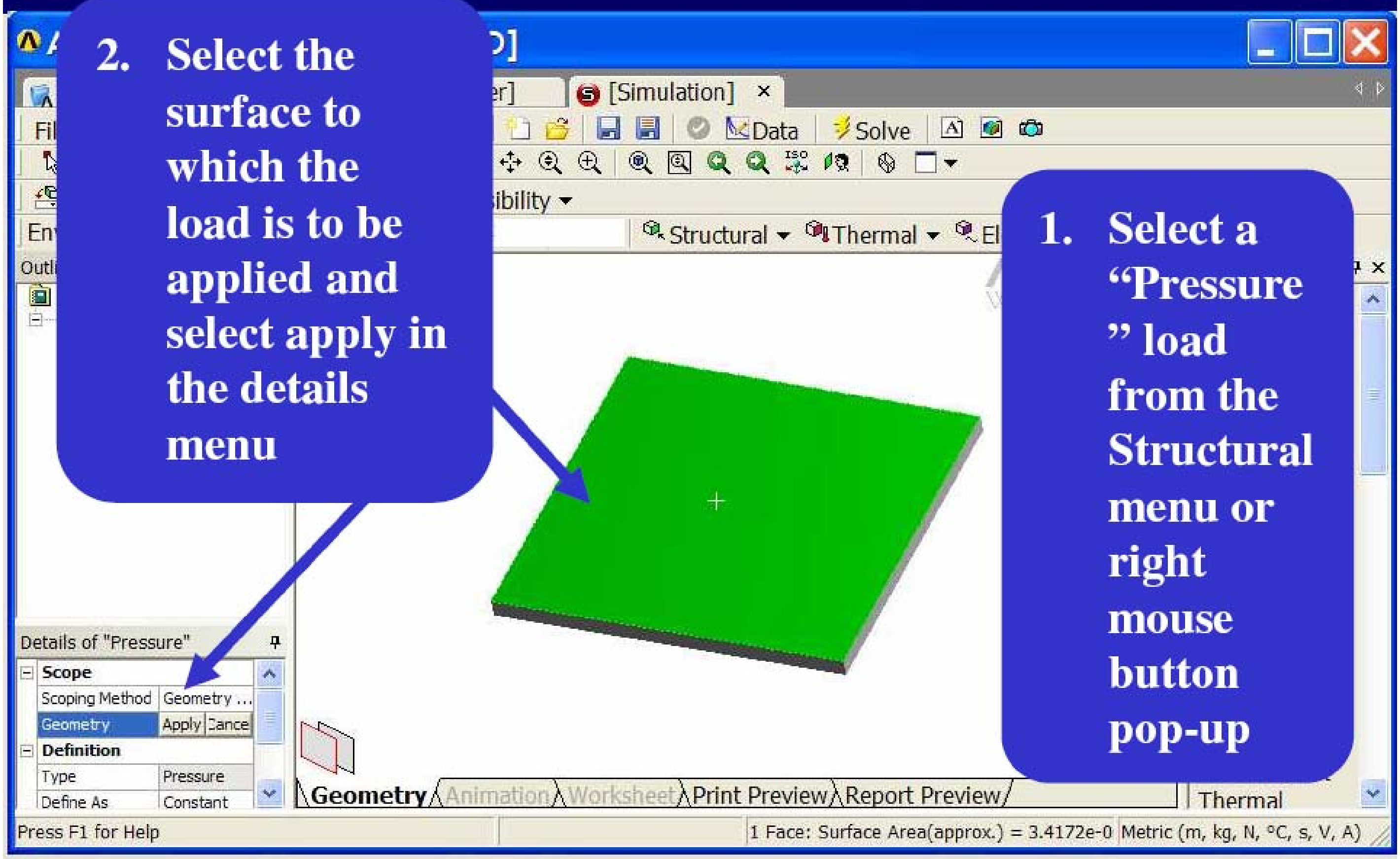
Step 2B – Defining Supports

ANSYS

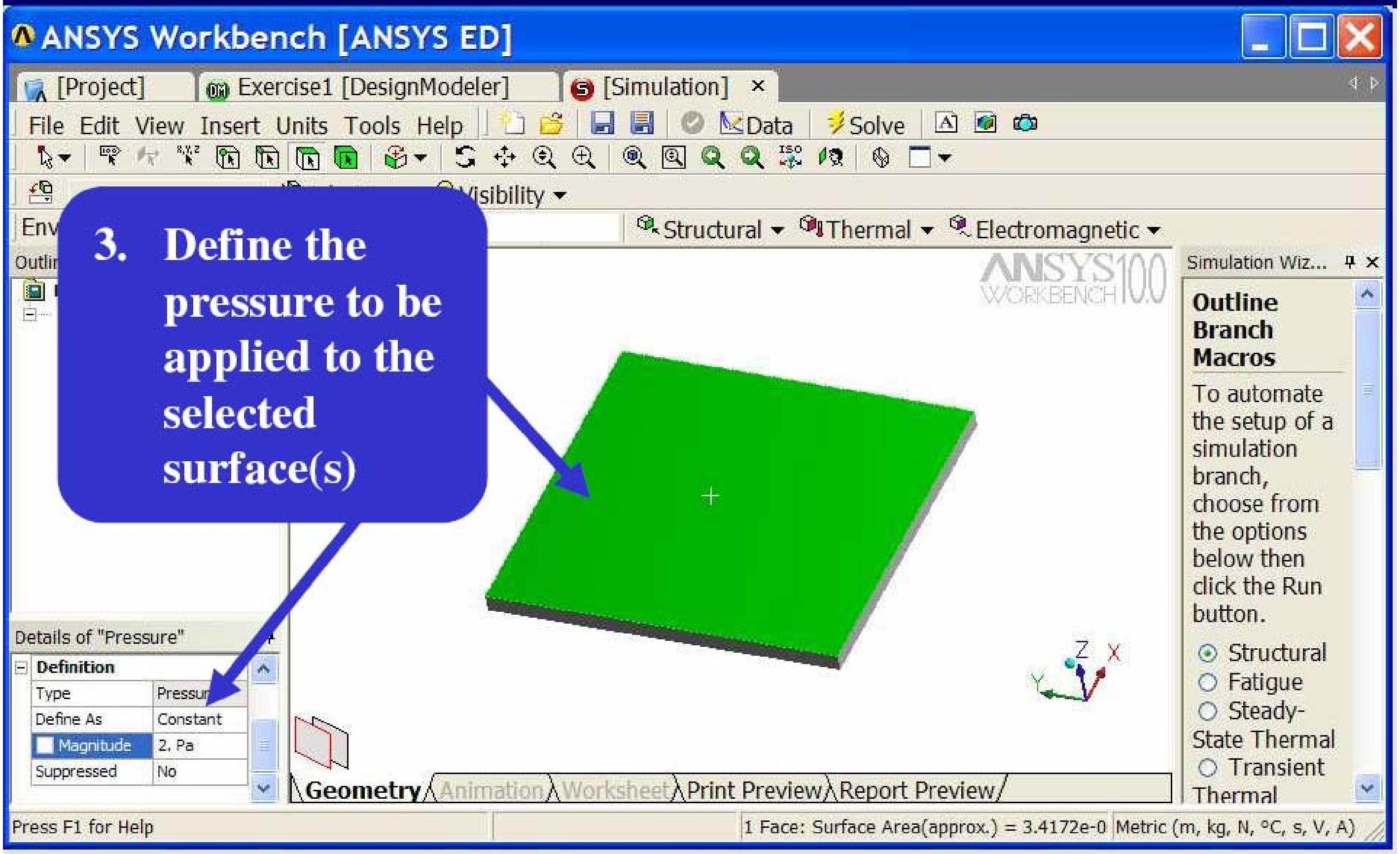


Step 3A – Defining Loads

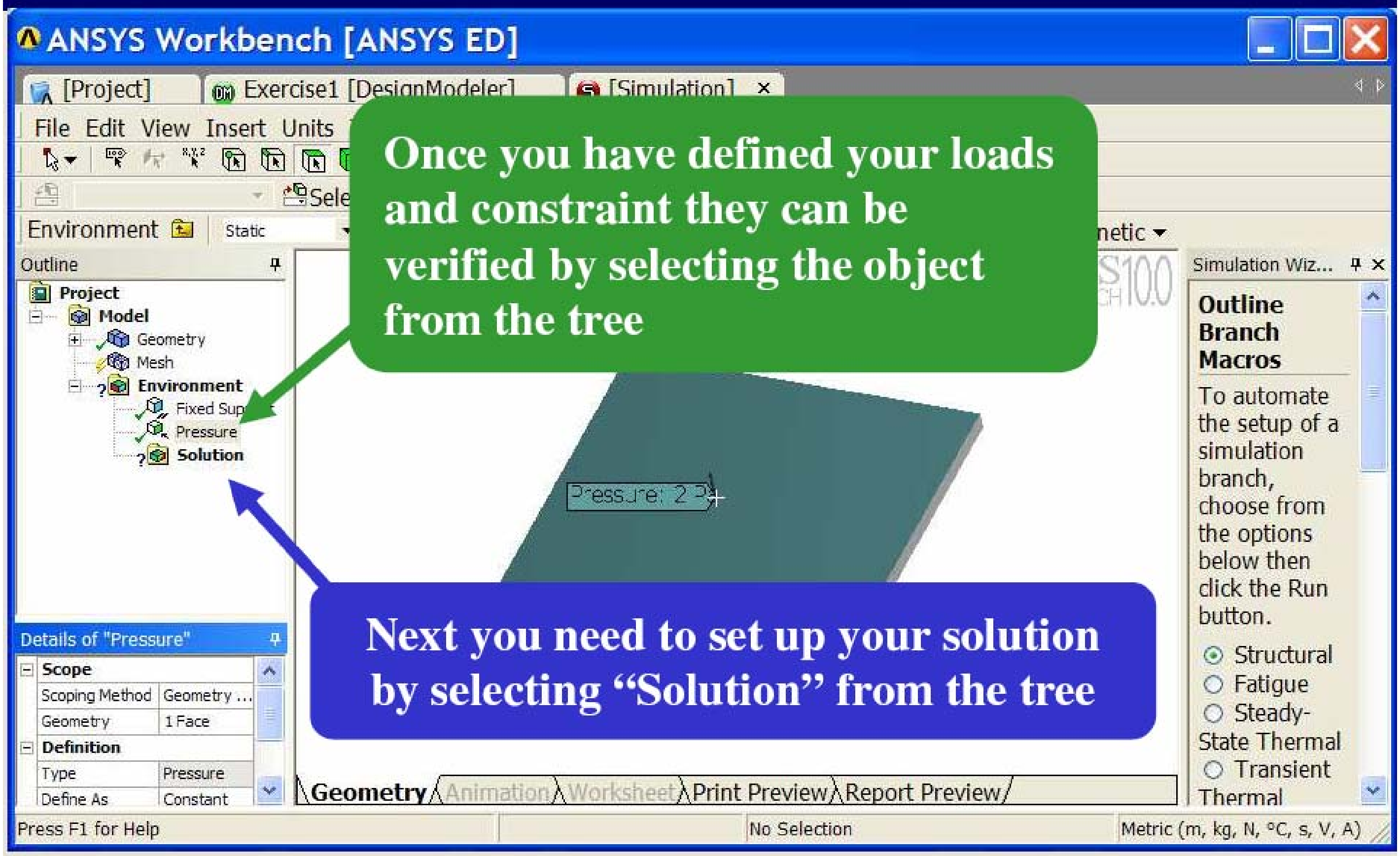
ANSYS



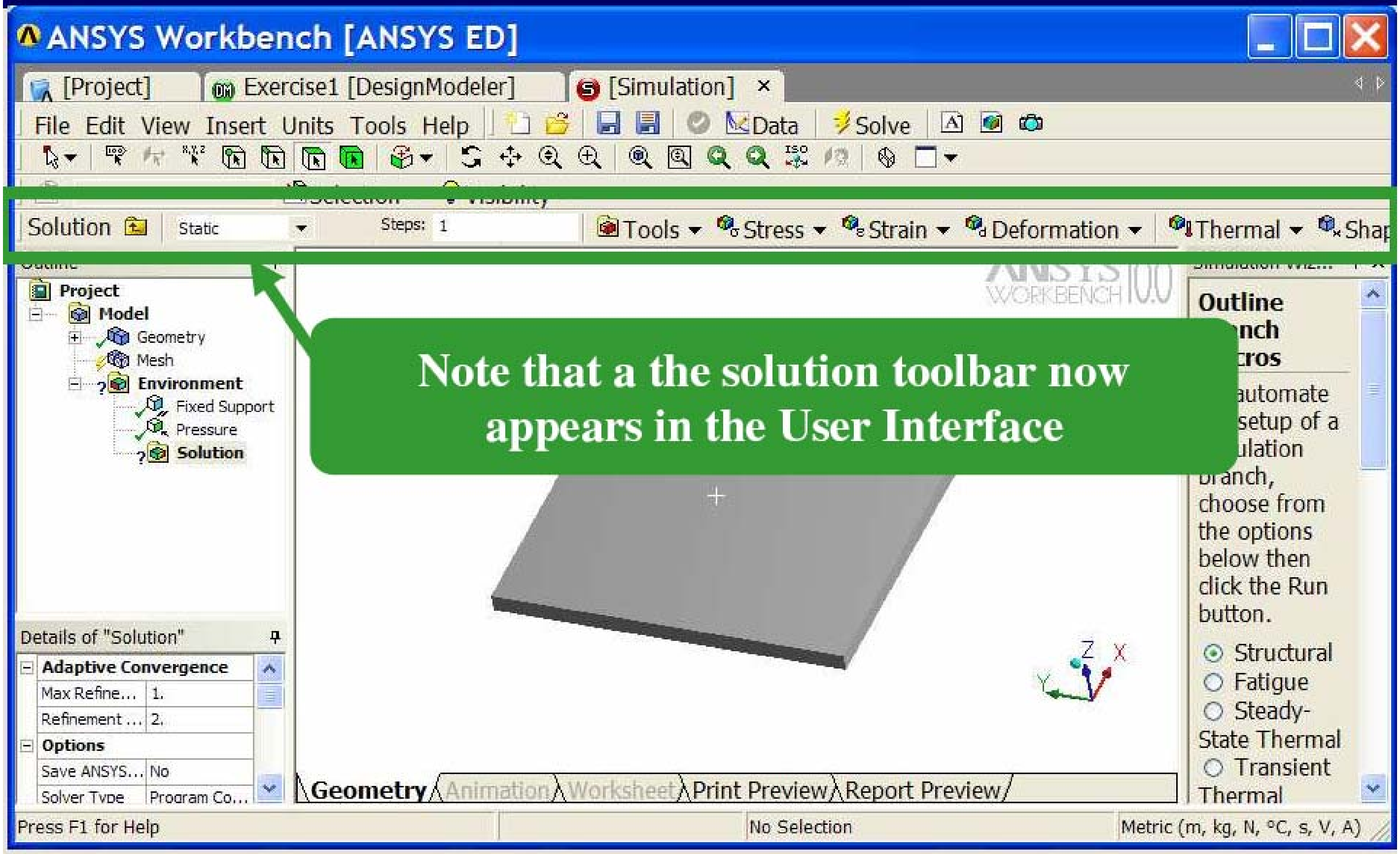
Step 3B – Defining Loads



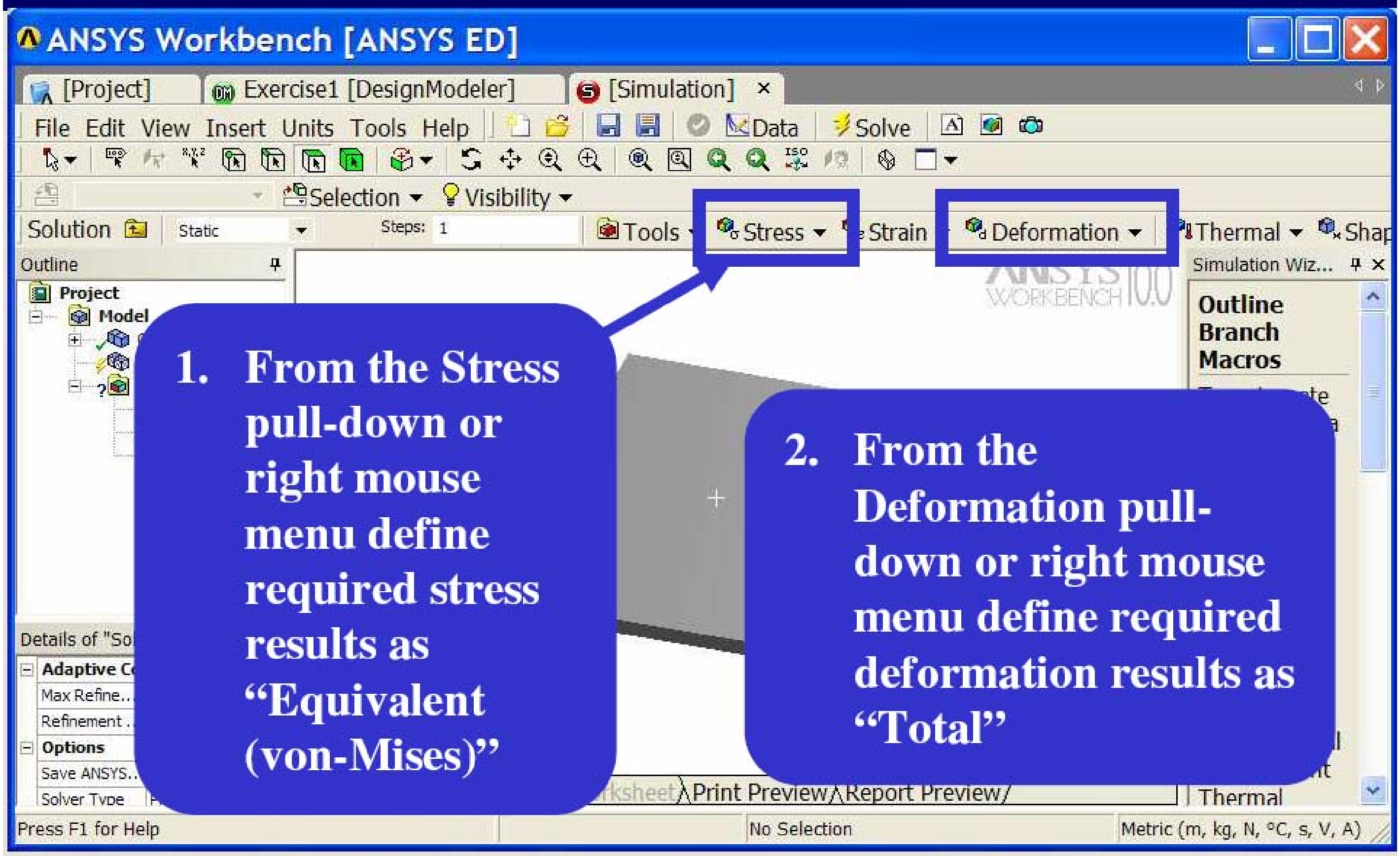
Step 4A - Defining Results



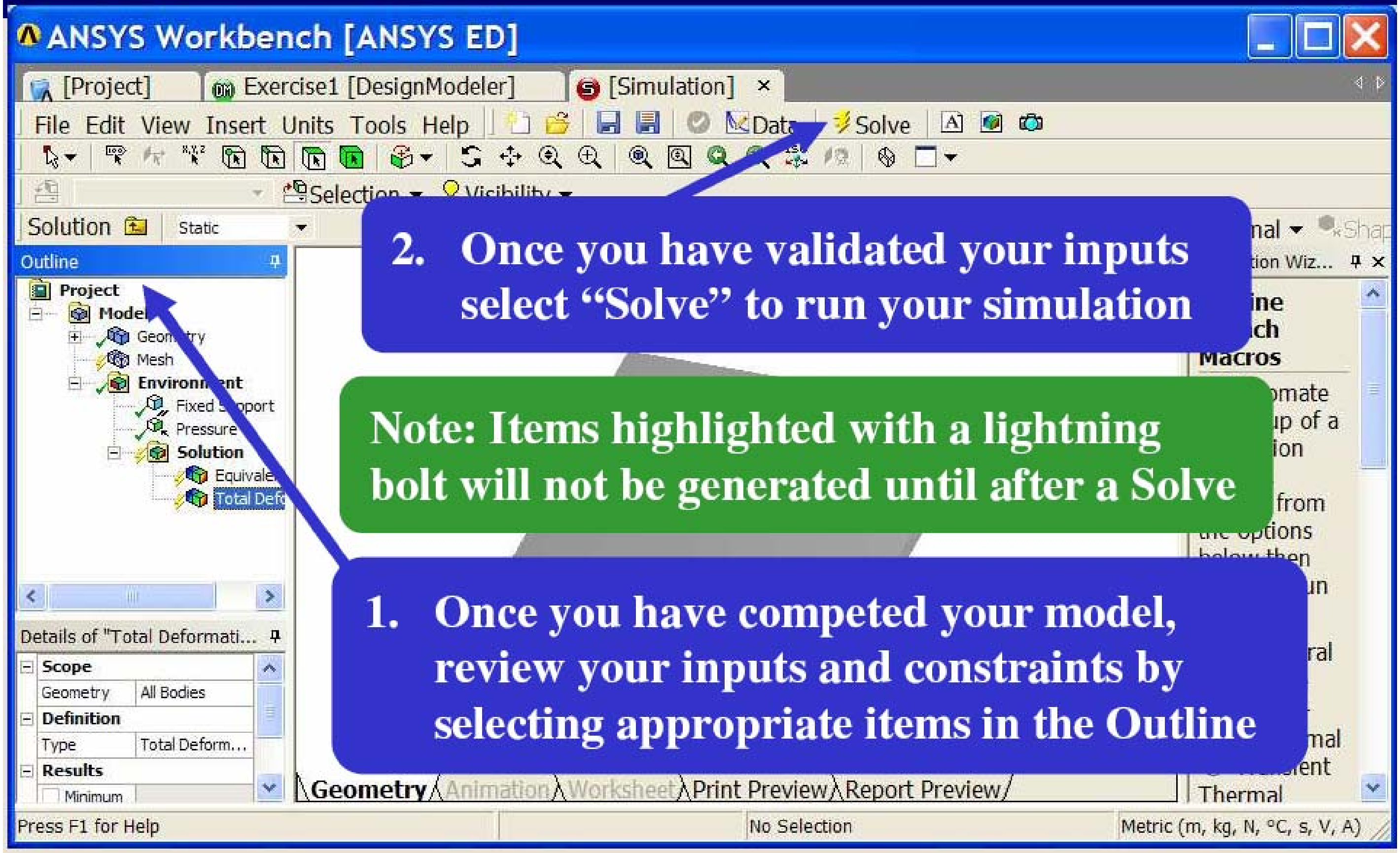
Step 4A - Defining Results



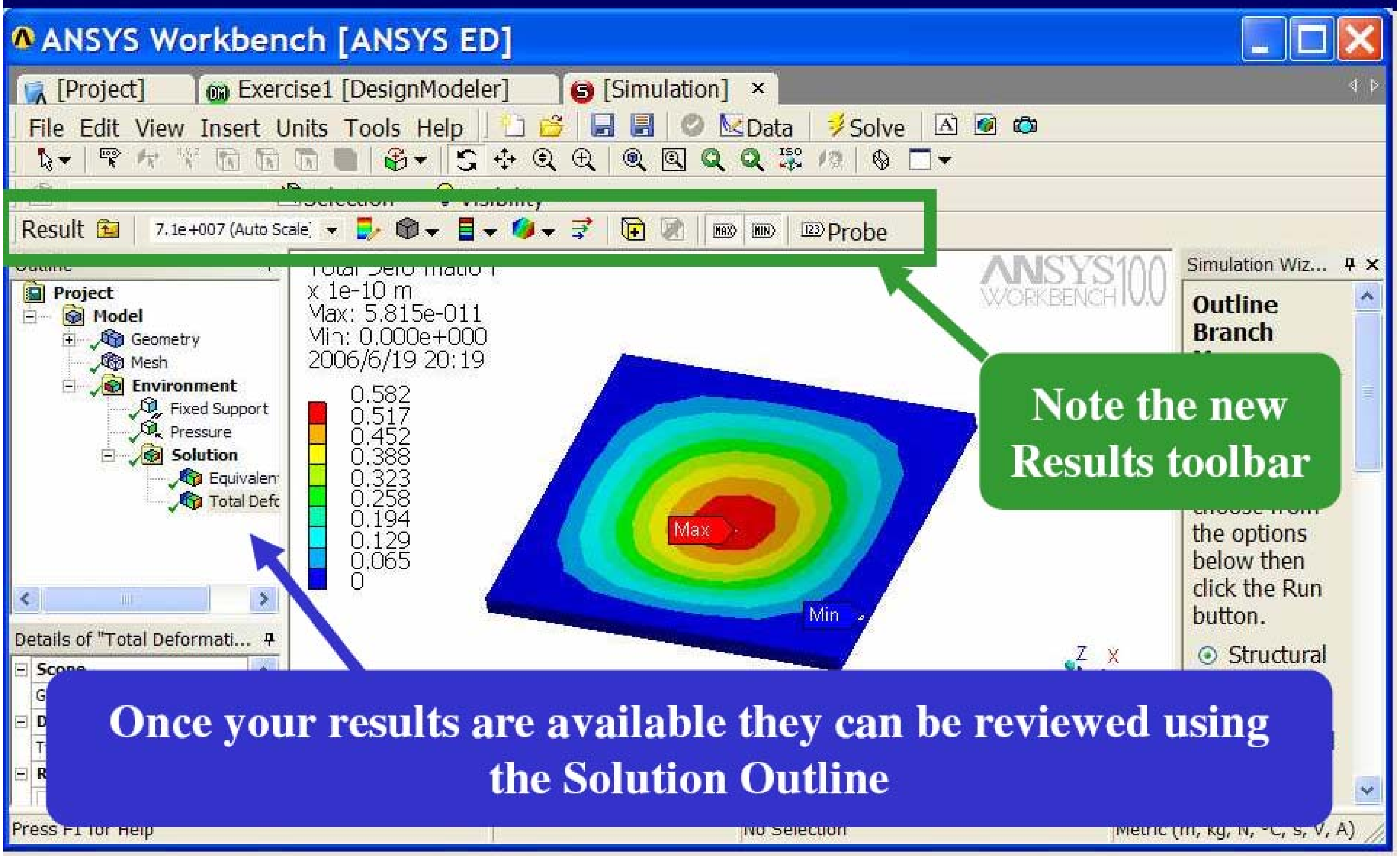
Step 4A - Defining Results



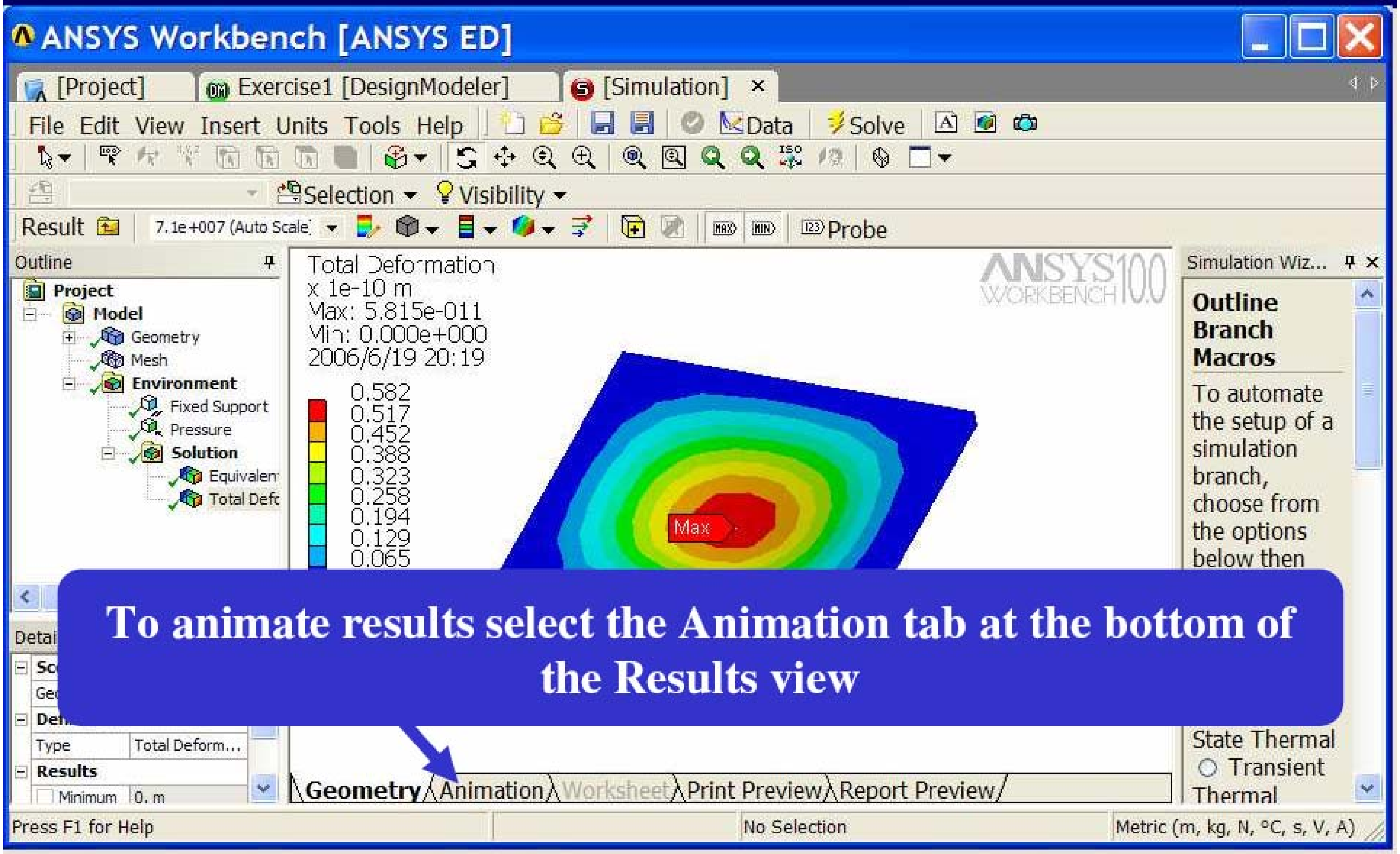
Step 5A – Validating Inputs



Step 6A – Reviewing Results

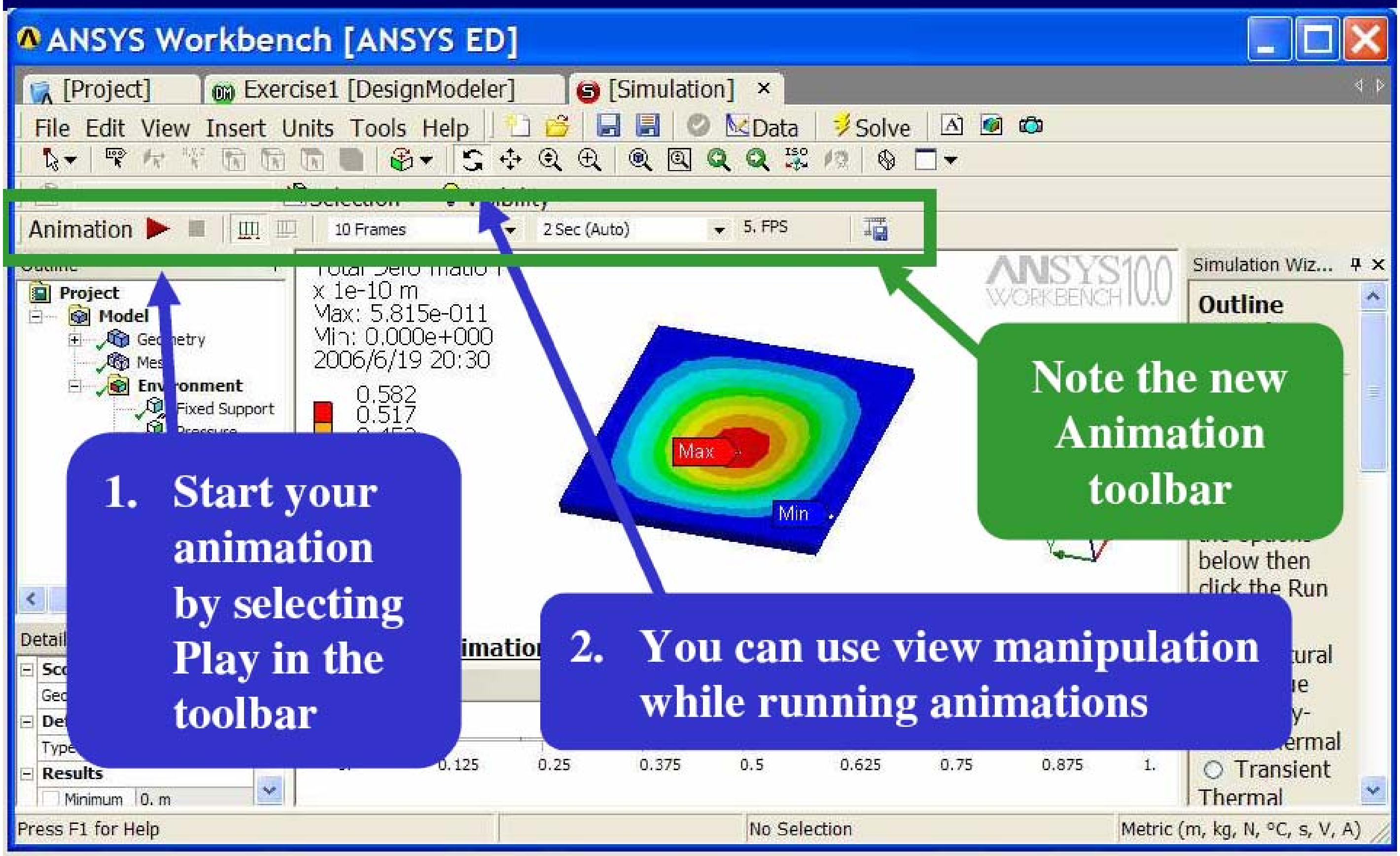


Step 6B – Reviewing Animation



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

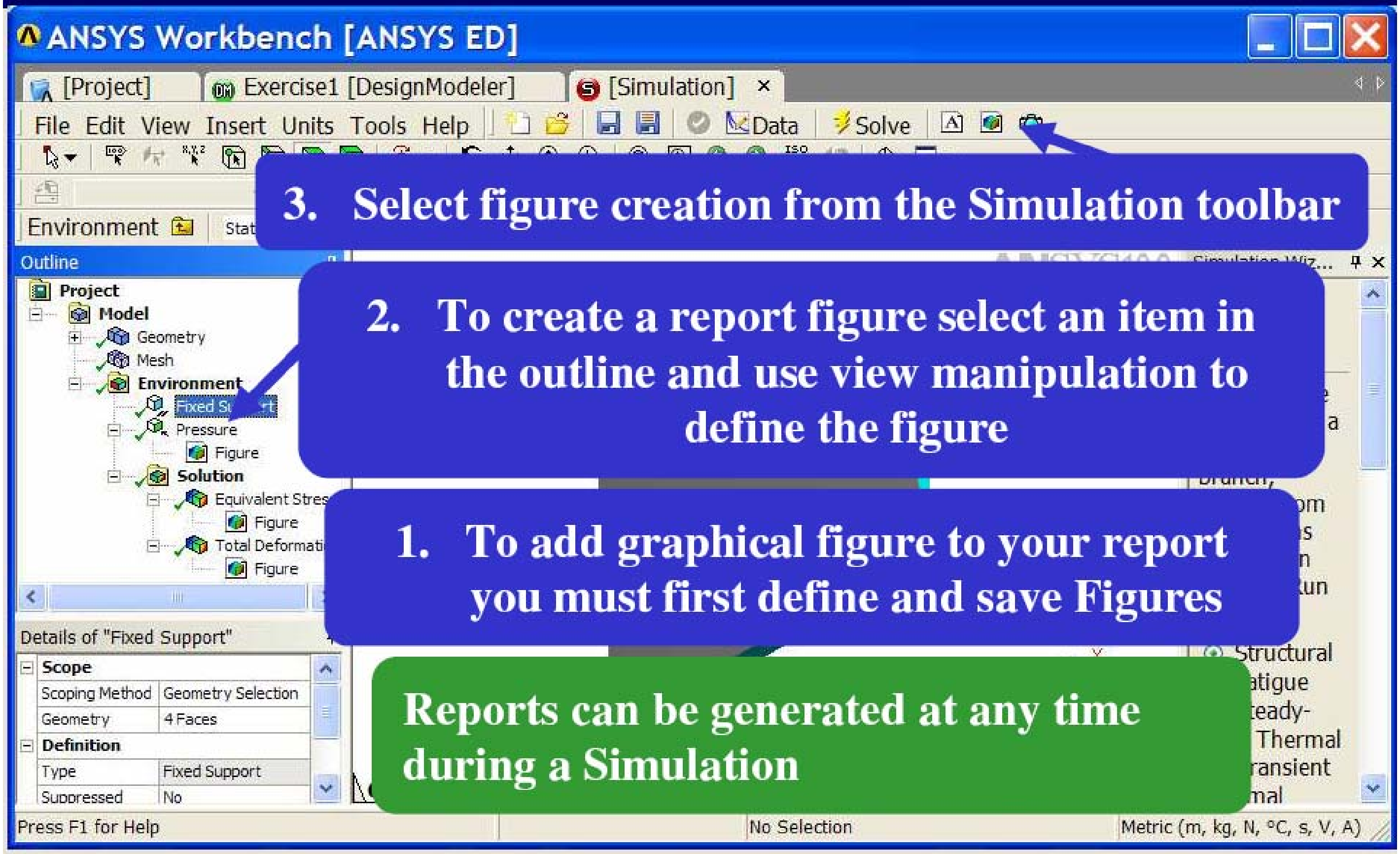
Step 6C - Manipulation



Note the new Animation toolbar

1. Start your animation by selecting Play in the toolbar
2. You can use view manipulation while running animations

Step 7A – Define Figures



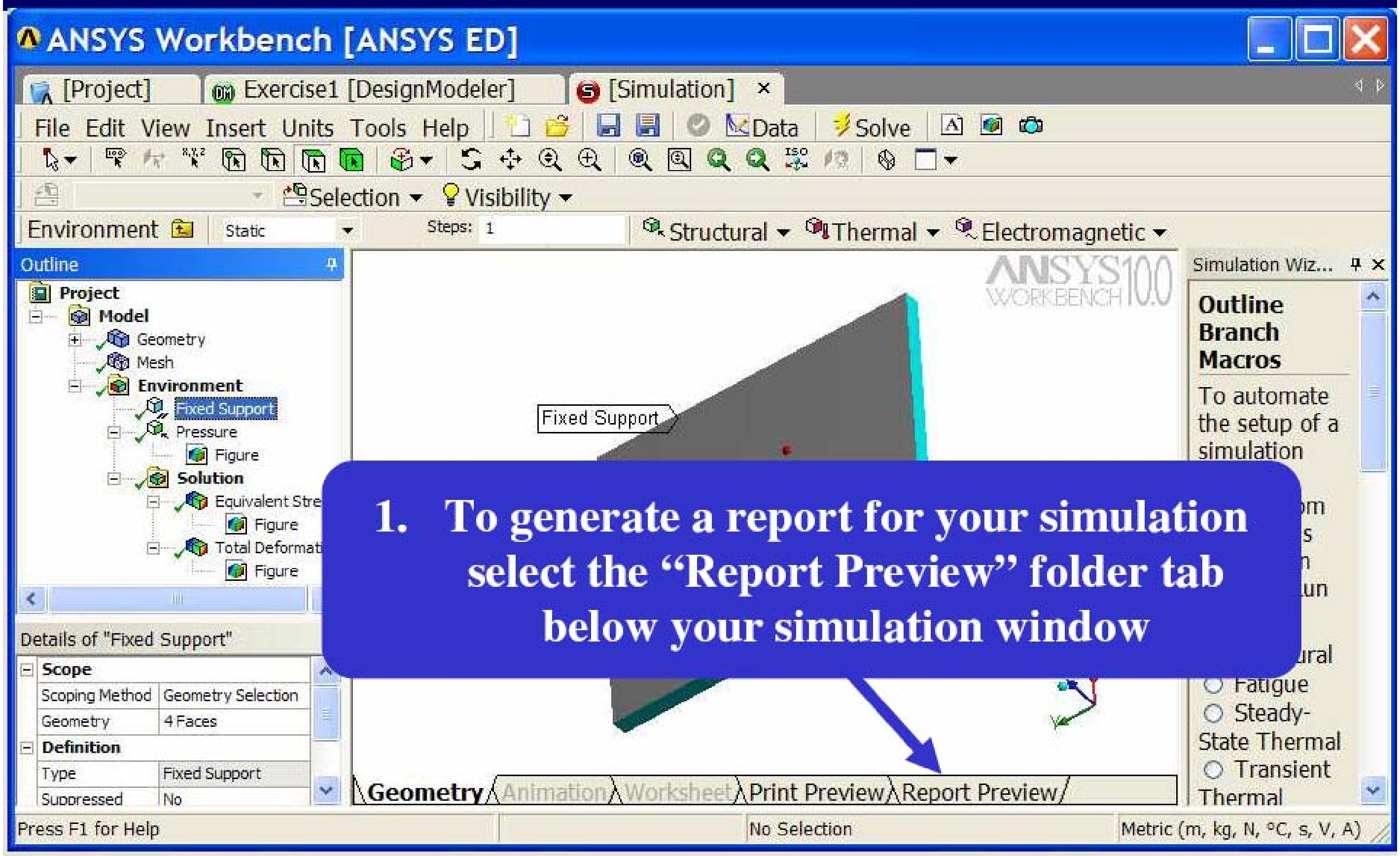
3. Select figure creation from the Simulation toolbar

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

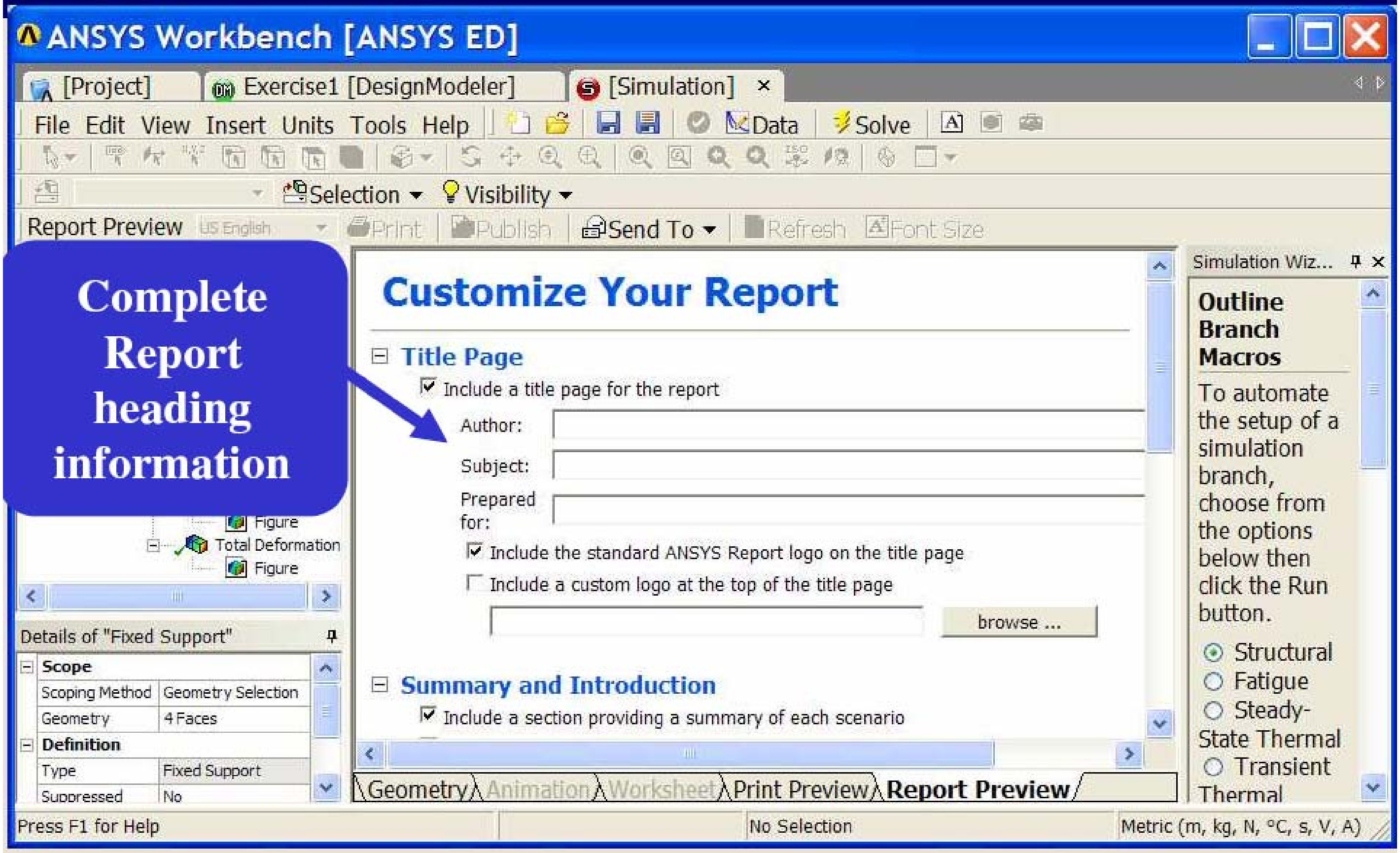
Reports can be generated at any time during a Simulation

Step 7B – Generate a Report



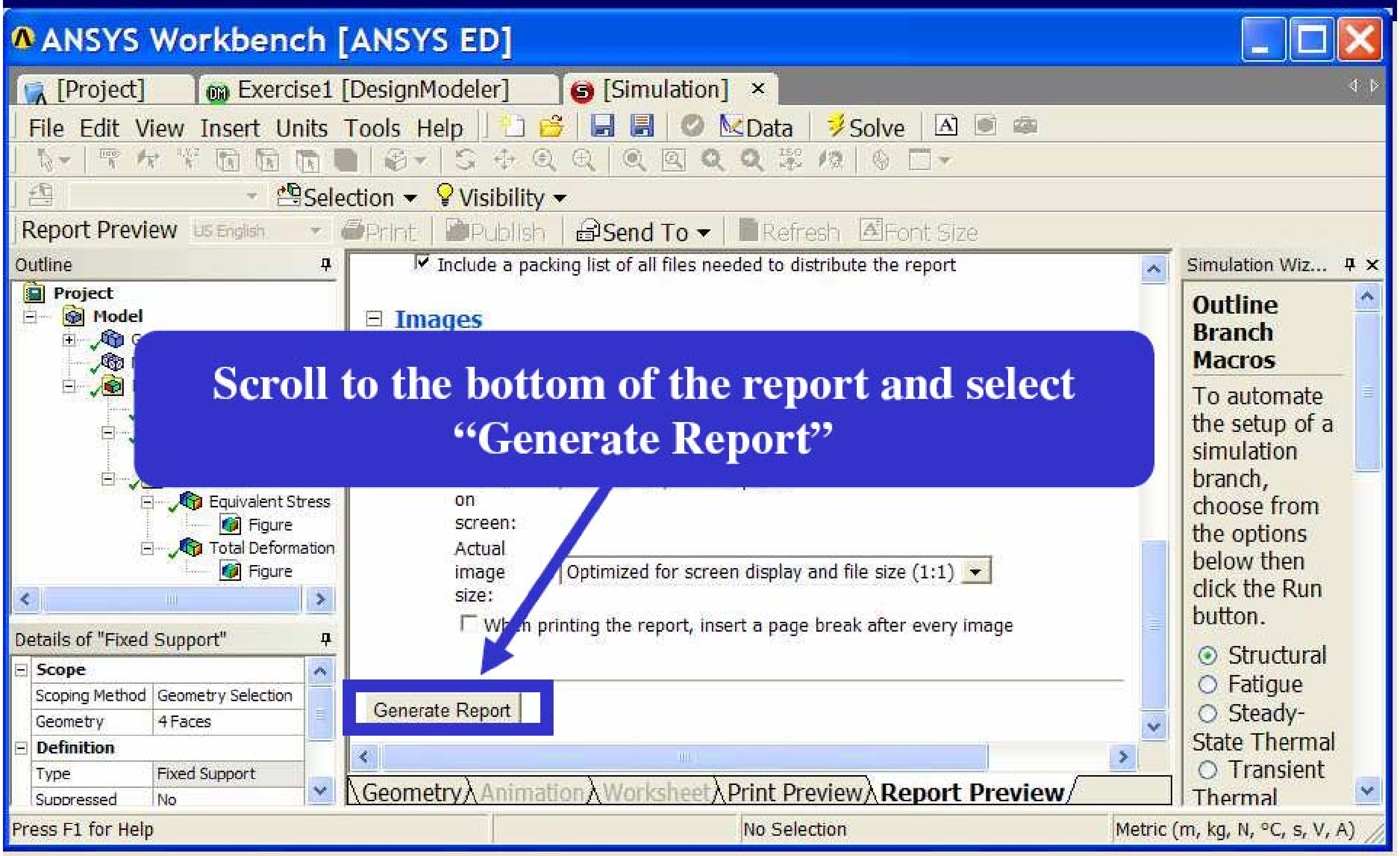
1. To generate a report for your simulation select the "Report Preview" folder tab below your simulation window

Step 7C – Complete Heading



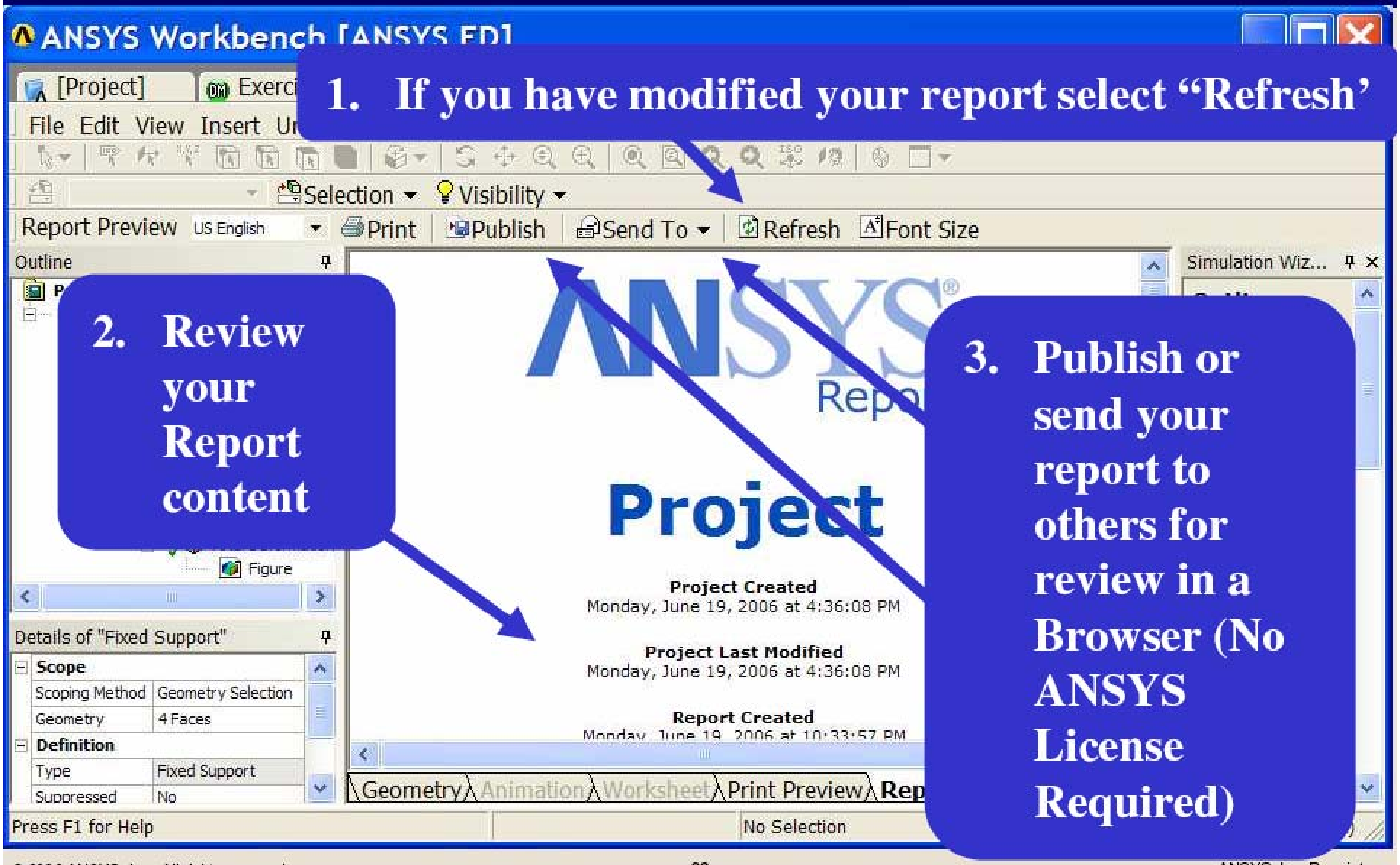
Complete Report heading information

Step 7D – Generate the Report



Scroll to the bottom of the report and select "Generate Report"

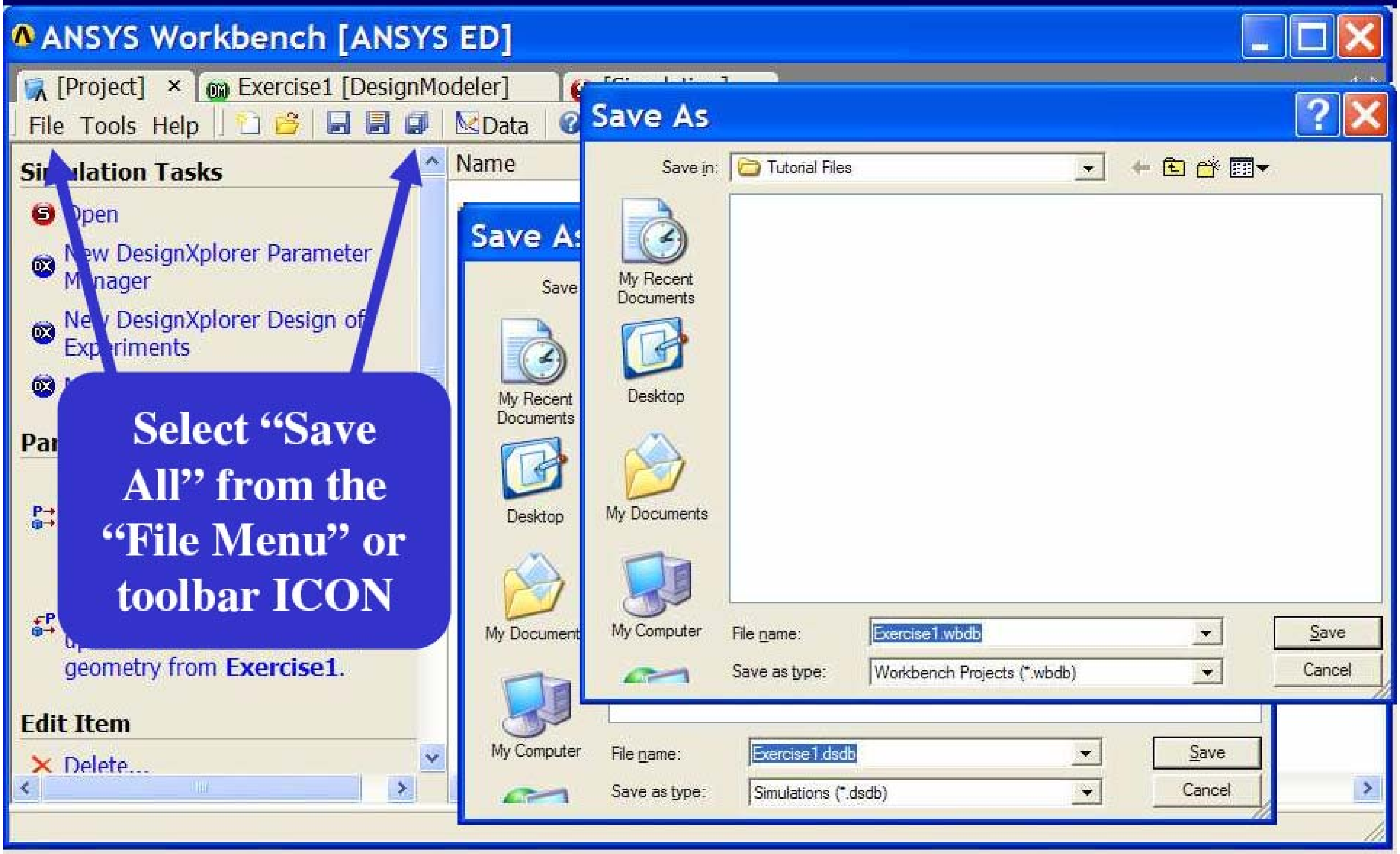
Step 7E – Publish Report



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



Review your Project Contents