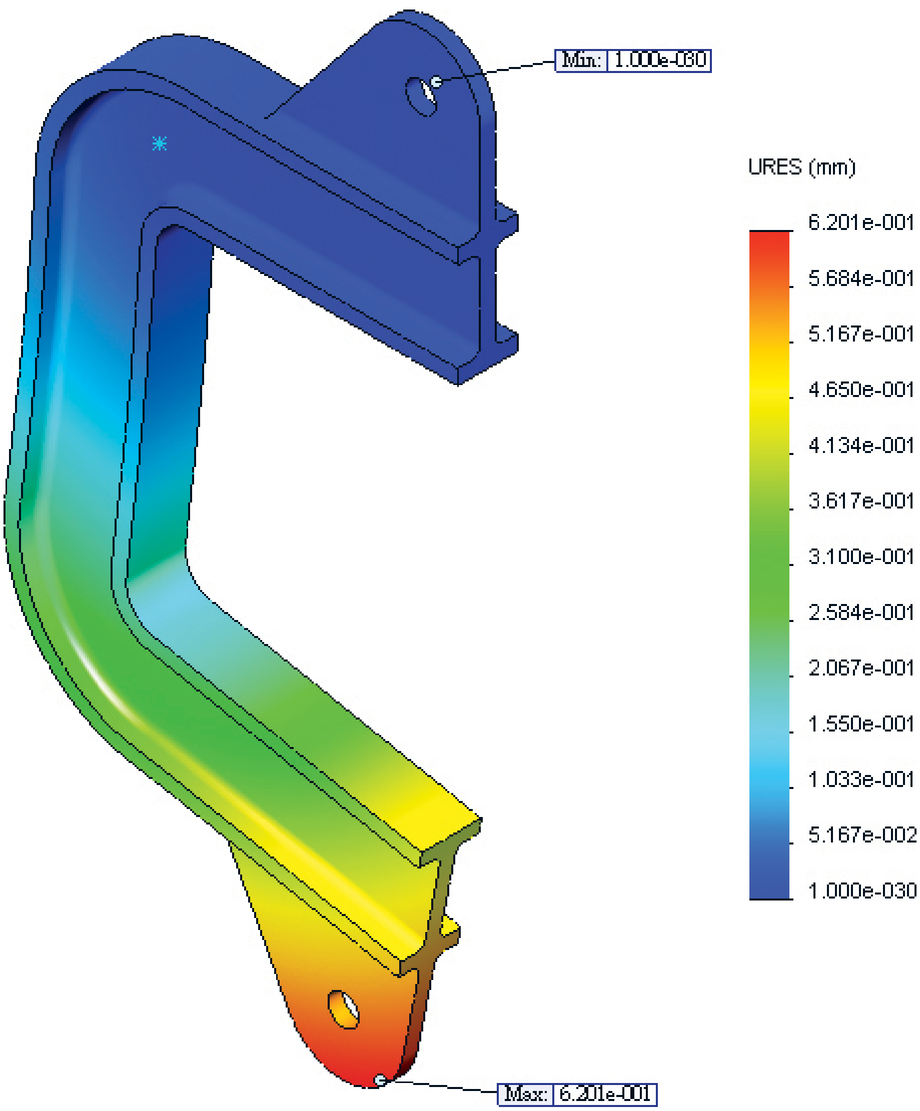
Mechanics of Materials Labs

with SolidWorks® Simulation 2013



Huei-Huang Lee

SDC PUBLICATIONS

Better Textbooks. Lower Prices.

www.SDCpublications.com

Chapter 1

Stresses

Stresses are quantities to describe the intensity of force in a body (either solid or fluid). Its unit is force per unit area (i.e., N/m^2 in SI). It is position dependent.

Imagine that your arms are pulled by your friends with two forces of the same magnitude but opposite directions. What are the stresses in your arms? Assuming the magnitude of the forces is 100 N and the cross-sectional area of your arms is 100 cm^2, then you may answer, "the stresses are 1 N/cm^2 . everywhere in my arms." This case is simple and the answer is good enough. For an one-dimensional case like this, the stress σ may be easily defined as

$$\sigma = \frac{P}{A}$$

where P is the applied force and A is the cross sectional area.

In general 3D cases, things are much more complicated. Now, imagine that you are buried in the soil by your friends, and your head is 100 m deep below the ground surface. How do you describe the force intensity (i.e., stress) on your head?

If the soil is replaced by still water, than the answer would be much simpler. The magnitude of the pressure (stress) on the top of your head would be the same as the pressure on your cheeks, and the direction of the pressure would always be perpendicular to the surface where the pressure applies. You've learned these concepts in your high school. And you've learned that the magnitude of the pressure is $\sigma = \rho g h$, where ρ is the mass density of the water, g is the gravitational acceleration, and h is the depth (100 meters in this case). In general, to describe the force intensity at a certain position in water, we place an infinitesimally small body at that position, and measure the force per unit surface area on that body.

In the soil (which is a solid material rather than water), the situation is much more complicated. First, the magnitude of the pressure on the top of your head may not be the same as that on your cheeks. Second, the direction of pressure is not necessarily perpendicular to the surface where the pressure applies. However, the above definition of stresses for water still holds. Let me restate as follows:

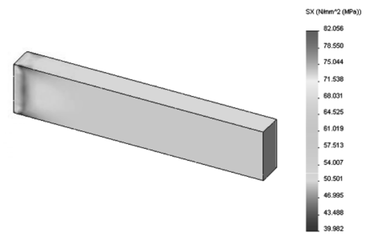
> *The stress at a certain position in a solid material is defined as the force per unit surface area on an infinitesimally small body placed at that position.*

Note that, the infinitesimally small body could be any shapes, and if we know the stresses on a certain shape of small body, we can infer the stresses on other shapes. We usually take a small cube to describe the stresses.

This chapter will guide you to learn the concepts of stresses.

Section 1.1

Stress Components



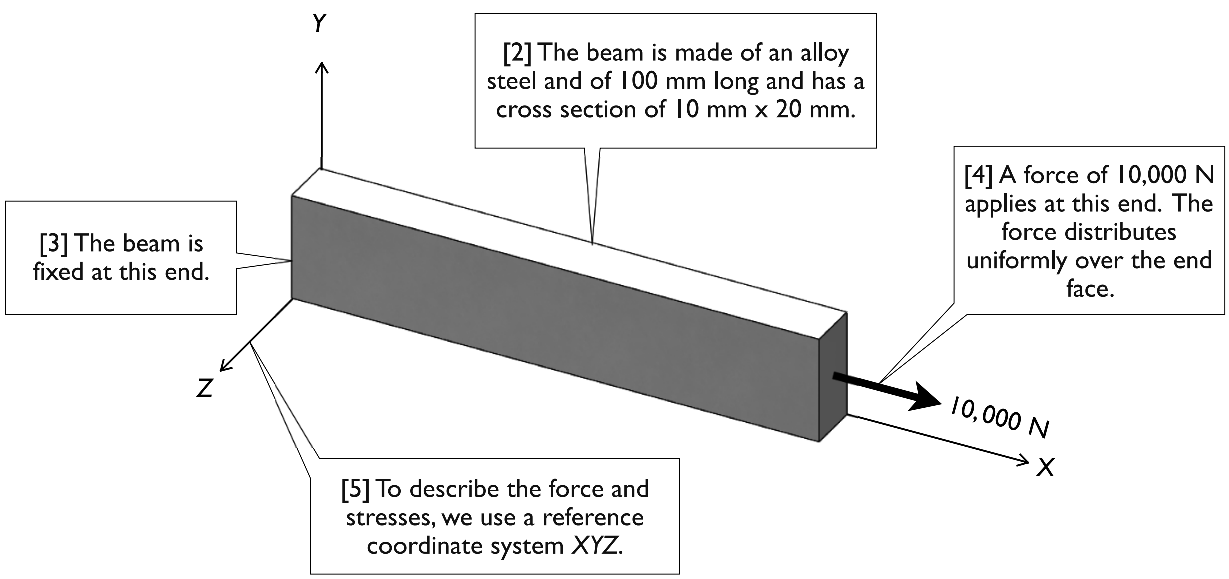
1.1-1 Introduction

[1] Consider a cantilever beam made of an alloy steel and of dimension 10 mm x 20 mm x 100 mm [2], which is fixed at one end [3] and subjected to a force applied on the other end [4]. The force is in positive *X*-direction and has a magnitude of 10,000 N. Note that, we've used a reference coordinate system as shown in [5].

In theory, the stress is uniform over the body; i.e., every point in the beam has the same stress. How do we describe this stress? Can we simply say, the stress is 50 MPa, which is calculated by

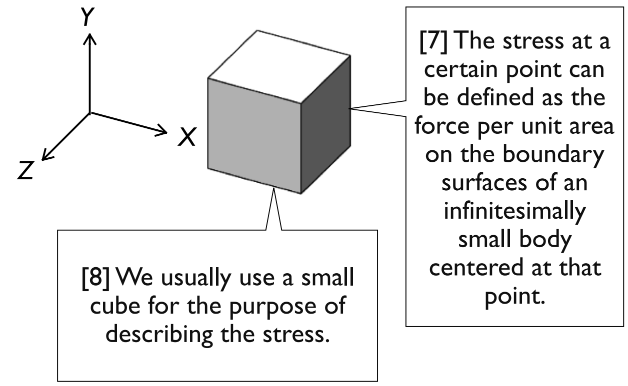
$$\frac{10{,}000 \text{ N}}{10 \text{ mm} \times 20 \text{ mm}} = 50 \text{ MPa?}$$

For a simple case like this, that may be adequate. In order to apply to more general cases, we need to say something more, specifically, what is the direction of the stress? What is the surface on which the stress acts?



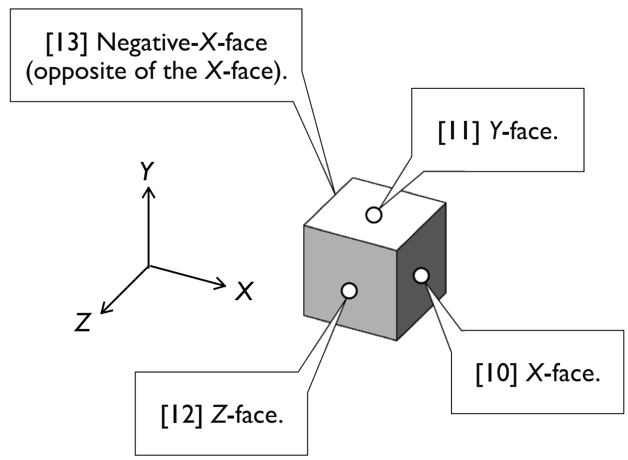
[6] Definition of Stress

The stress at a certain point can be defined as *the force per unit area acting on the boundary surfaces of an infinitesimally small body centered at that point* [7]. The stress values may be different at different locations of the boundary surfaces. The small body can be any shape. However, for the purpose of describing the stress, we usually use a small cube [8] of which each edge is parallel to a coordinate axis.



[9] *X*-Face, *Y*-Face, and *Z*-Face

Each of the six faces of the cube can be assigned an identifier as *X*-face, *Y*-face, *Z*-face, negative-X-face, negative-Y-face, and negative-Z-face, respectively [10-13].

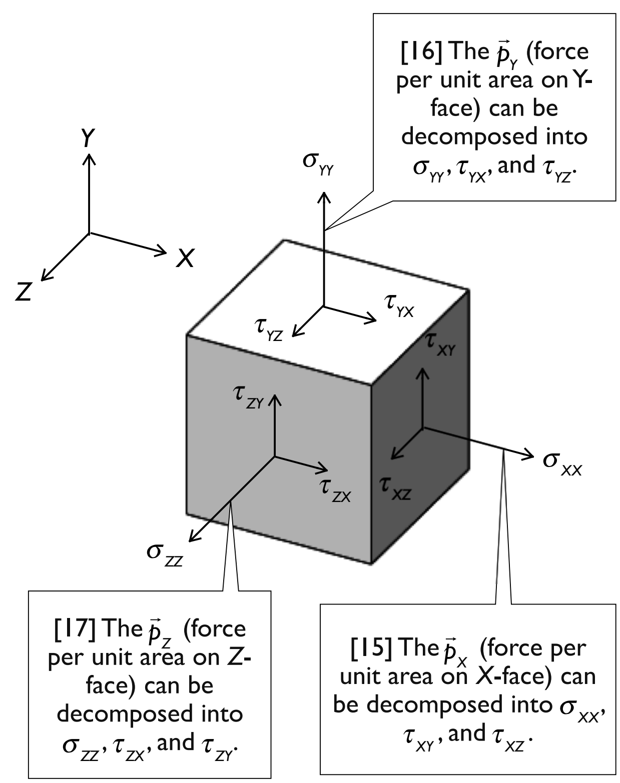


[14] Stress Components

Let $\vec{p}_X$ be the force per unit area acting on the *X*-face. In general, $\vec{p}_X$ may not be normal or parallel to the *X*-face. We may decompose $\vec{p}_X$ into *X*-, *Y*-, and *Z*-component, and denote σ_{XX}, τ_{XY}, and τ_{XZ} respectively [15]. Note that, the first subscript (*X*) is used to indicate the **face** on which the stress components act, while the second subscript (*X, Y, or Z*) is used to indicate the **direction** of the stress components. Also note that, σ_{XX} is normal to the face, while τ_{XY}, and τ_{XZ} are parallel to the face. Therefore, σ_{XX} is called a **normal stress**, while τ_{XY}, and τ_{XZ} are called **shear stresses**. In Mechanics of Materials, we usually use the symbol σ for a normal stress and τ for a shear stress.

Similarly, let $\vec{p}_Y$ be the force per unit area acting on the *Y*-face and we may decompose $\vec{p}_Y$ into a normal component (σ_{YY}), and two shear components (τ_{YX} and τ_{YZ}) [16]. Also, let $\vec{p}_Z$ be the force per unit area acting on the *Z*-face and we may decompose $\vec{p}_Z$ into a normal component (σ_{ZZ}), and two shear components (τ_{ZX} and τ_{ZY}) [17]. Organized in a matrix form, these stress components may be written as

$$\{\sigma\} = \begin{pmatrix} \sigma_{XX} & \tau_{XY} & \tau_{XZ} \\ \tau_{YX} & \sigma_{YY} & \tau_{YZ} \\ \tau_{ZX} & \tau_{ZY} & \sigma_{ZZ} \end{pmatrix} \qquad (1)$$



[18] Stress Components on Other Faces

It can be proven that the stress components on the negative-*X*-face, negative-*Y*-face, and negative-*Z*-face can be derived from the 9 stress components in Eq. (1). For example, on the negative-*X*-face, the stress components have exactly the same stress values as those on the *X*-face but with opposite directions [19]. Similarly, the stress components on the negative-*Y*-face have the same stress values as those on the *Y*-face but with opposite directions [20], and the stress components on the negative-*Z*-face have the same stress values as those on the *Y*-face but with opposite directions [21].

The proof can be done by taking the cube as free body and applying the force equilibria in *X*, *Y*, and *Z* directions respectively.

On an arbitrary face (which may not be parallel or perpendicular to an axis), the stress components can be calculated from those on *X*-face, *Y*-face, and *Z*-face. We'll show that this can be done using a Mohr's circles (Section 10.1).

[22] Symmetry of Shear Stresses

It also can be proven that the shear stresses are symmetric, i.e.,

$$\tau_{XY} = \tau_{YX}, \quad \tau_{YZ} = \tau_{ZY}, \quad \tau_{ZX} = \tau_{XZ} \qquad (2)$$

The proof can be done by taking the cube as free body and applying the moment equilibria in *X*, *Y*, and *Z* directions respectively.

[23] Stress Components

We now conclude that 3 normal stress components and 3 shear stress components are needed to describe the **stress state** at a certain point. We usually write them as follows

$$\{\sigma\} = \{\ \sigma_X \quad \sigma_Y \quad \sigma_Z \quad \tau_{XY} \quad \tau_{YZ} \quad \tau_{ZX}\ \} \qquad (3)$$

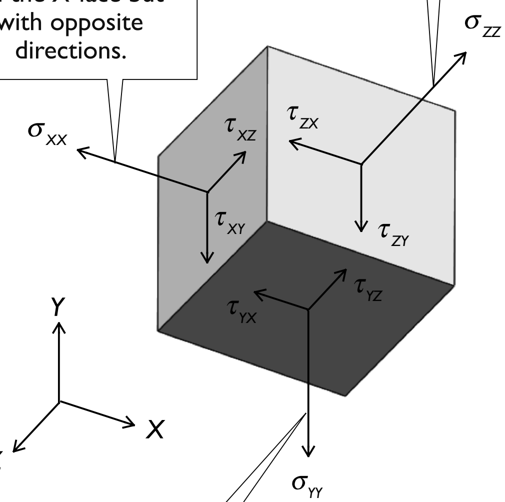
Note that, for more concise, we use σ_X in place of σ_{XX}, σ_Y in place of σ_{YY}, and σ_Z in place of σ_{ZZ}.

The purpose of this section is to familiarize the 6 stress components in Eq. (3). The stress field in this section is uniform over the entire body. In the next section, we'll explore a nonuniform stress field.

Another purpose of this section is to familiarize the **SolidWorks Simulation** user interface.

[19] The stress components acting on the negative-*X*-face have exactly the same stress values as those on the *X*-face but with opposite directions.

[21] The stress components acting on the negative-*Z*-face have exactly the same stress values as those on the *Z*-face but with opposite directions.



[20] The stress components acting on the negative-*Y*-face have exactly the same stress values as those on the *Y*-face but with opposite directions.

1.1-2 Launch **SolidWorks** and Create a New Part

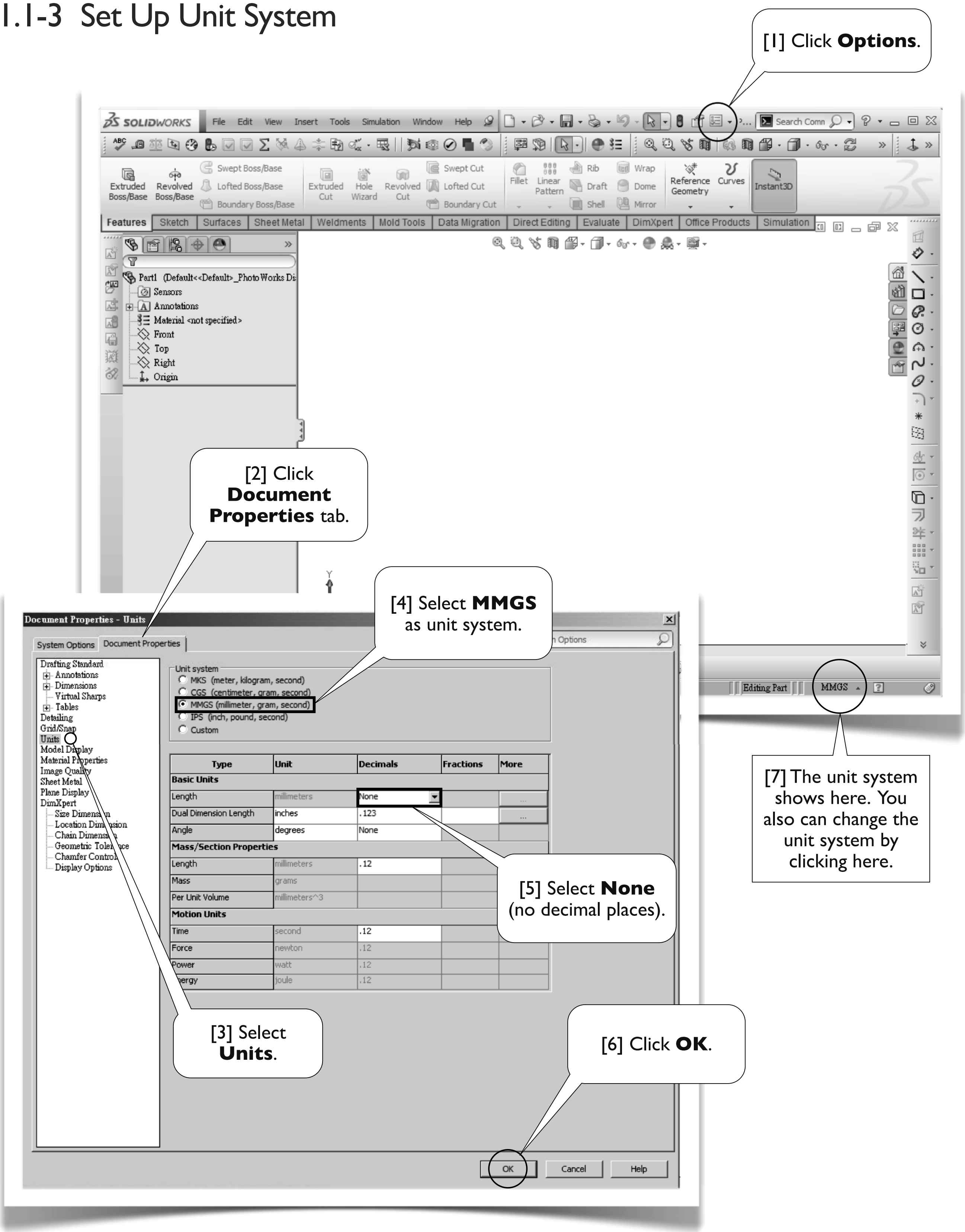
About the Text Boxes

1. Within each subsection (e.g., 1.1-2), text boxes are ordered with numbers, each of which is enclosed by a pair of square brackets (e.g., [1]). When you read the contents in a subsection, please follow the order of the text boxes.
2. The text box numbers also serve as reference numbers when referred from other text. In the same subsection, we simply refer to a text box by its number (e.g., [1]). From other subsections, we refer to a text box by its subsection identifier and the text box number (e.g., 1.1-2[1]).
3. A text box is either round-cornered (e.g., [1, 3, 5]) or sharp-cornered (e.g., [2, 4]). A round-cornered box indicates that **mouse or keyboard actions** are needed in that step. A sharp-cornered box is used for commentary only: no mouse or keyboard actions are needed in that step.

SolidWorks Terms

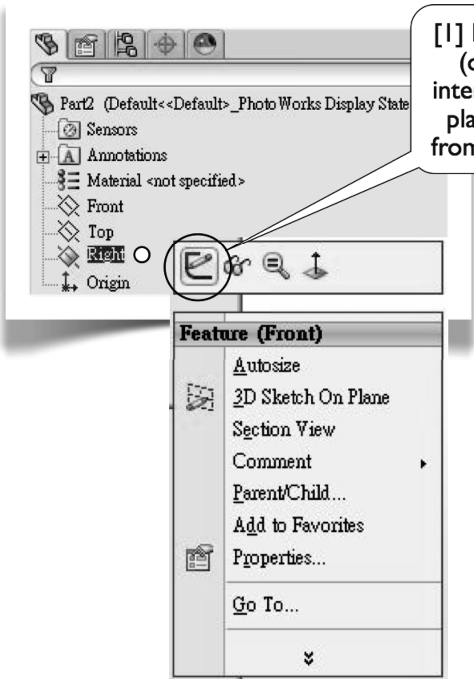
In this book, terms used in the **SolidWorks** are boldfaced (e.g., **Part** in [4, 5]).

1.1-3 Set Up Unit System

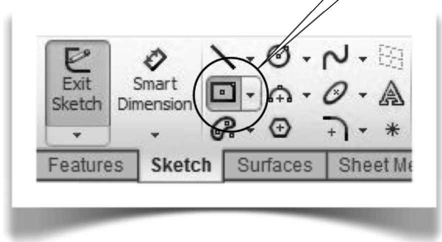


1.1-4 Create a Geometric Model

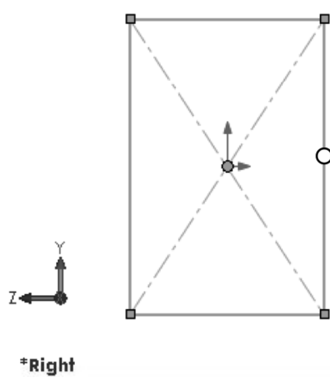
[1] In the **Features Tree** (on the left of the user interface), right-click **Right** plane and select **Sketch** from the **Context Menu**.



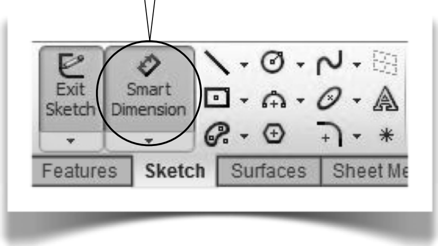
[2] In the **Sketch Toolbar**, select **Center Rectangle**.



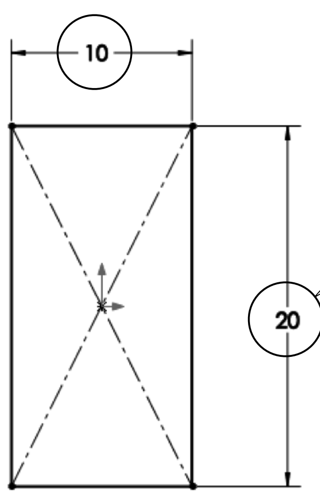
[3] Draw a rectangle centered at the origin (the sizes are arbitrary for now).



[4] In the **Sketch Toolbar**, click **Smart Dimension**.

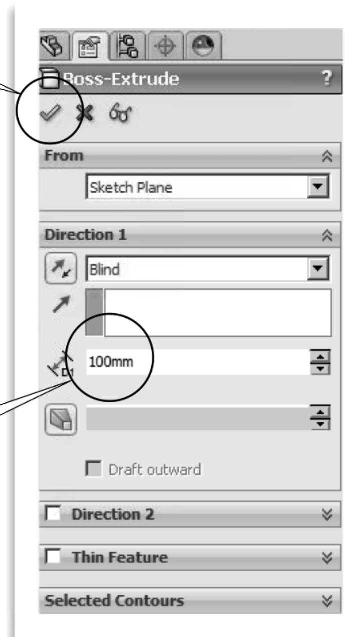


[5] Specify dimensions (10 mm and 20 mm) like this.

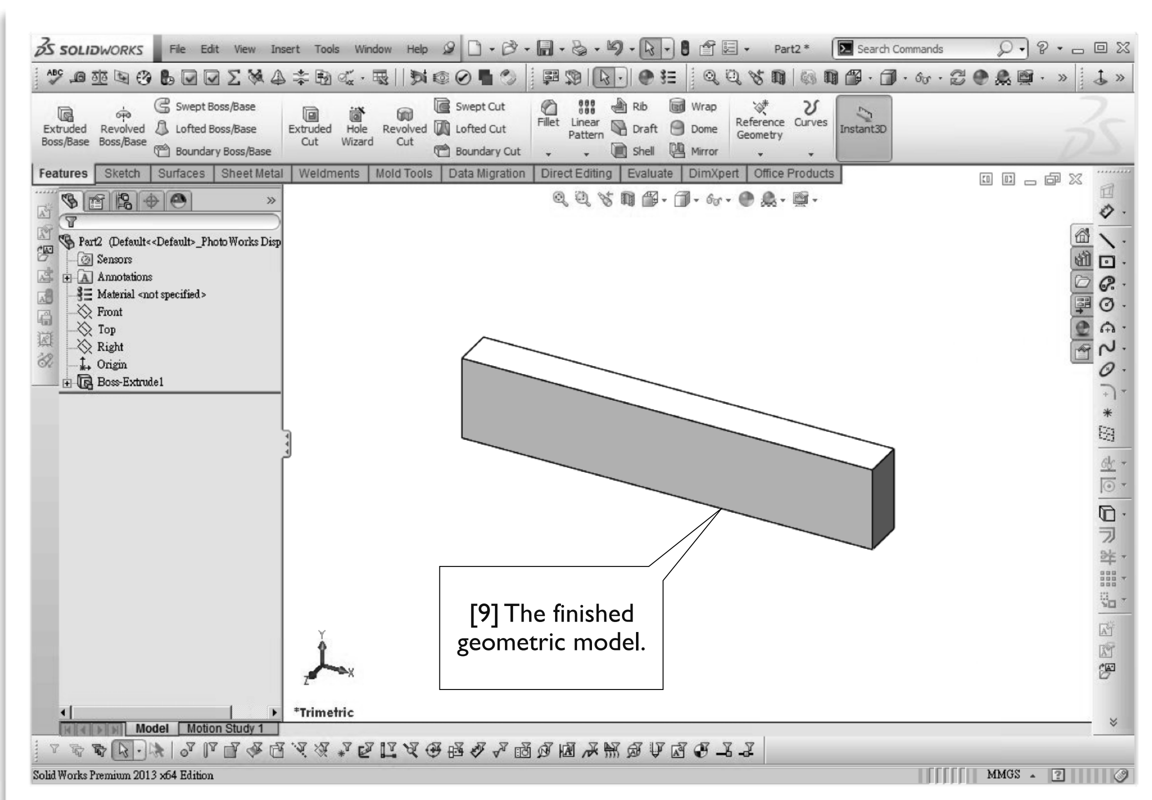


[6] In the **Features Toolbar**, click **Extruded Boss/Base**.

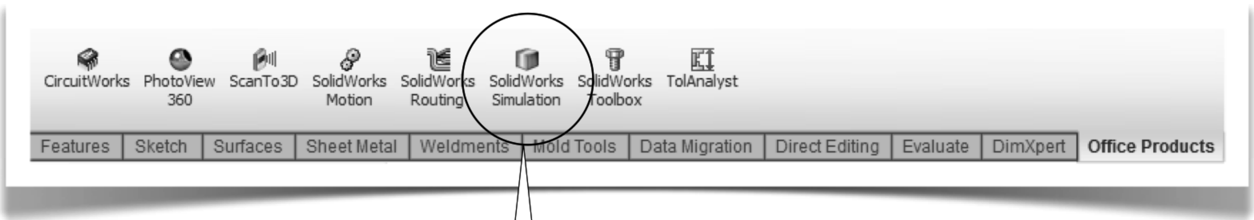
[8] Click **OK**.



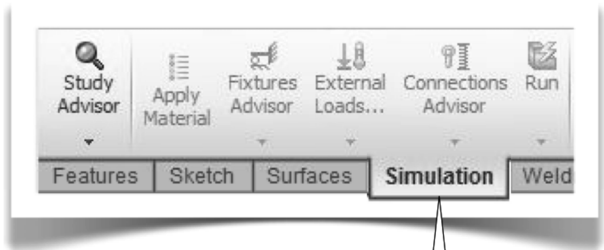
[7] In the **Property Box**, type 100 (mm) for **Depth**.



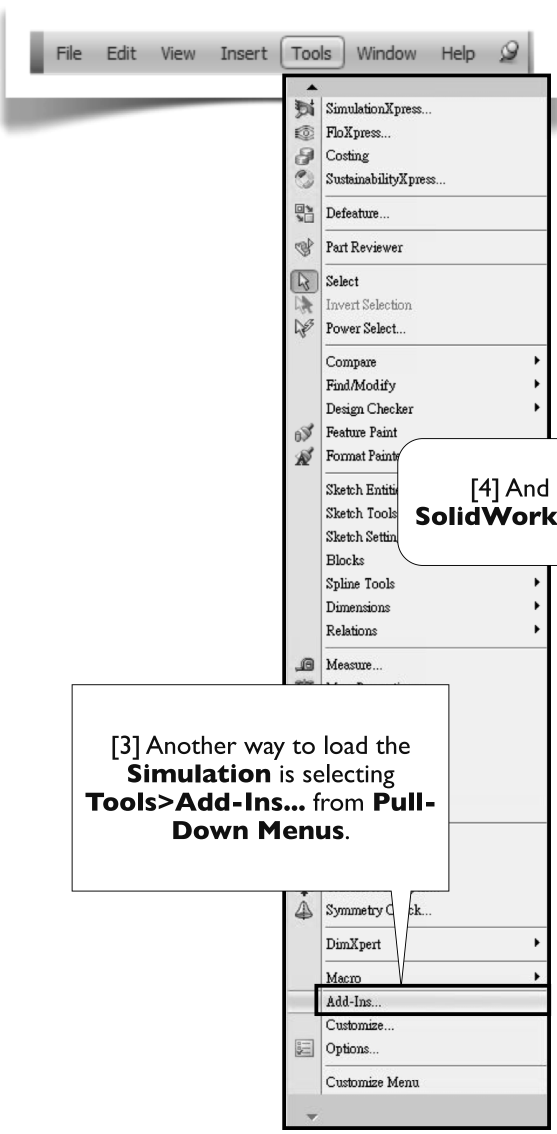
1.1-5 Load **SolidWorks Simulation**



[1] If **SolidWorks Simulation** is already loaded (i.e., if **Simulation Toolbar** is available), skip this step. Otherwise, in the **Office Products Toolbar**, click to load the **Simulation**.

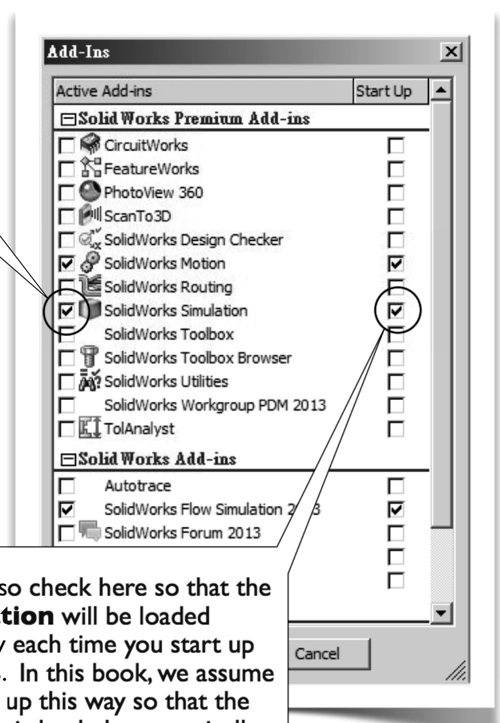


[2] The **Simulation Toolbar** becomes available.



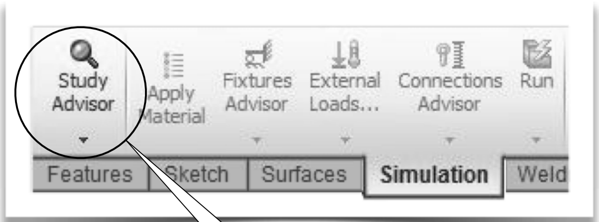
[3] Another way to load the **Simulation** is selecting **Tools>Add-Ins...** from **Pull-Down Menus**.

[4] And then select **SolidWorks Simulation**.

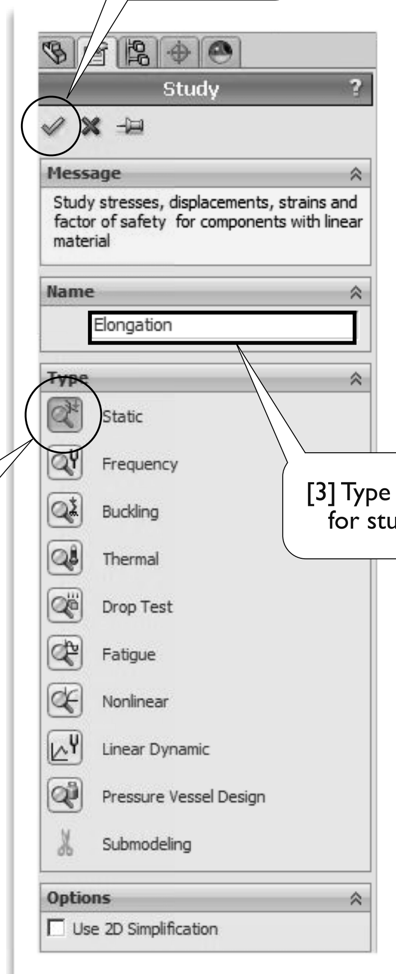


[5] You may also check here so that the **Simulation** will be loaded automatically each time you start up **SolidWorks**. In this book, we assume that you set up this way so that the **Simulation** is loaded automatically each time you start up **SolidWorks**.

1.1-6 Create a Static Structural Study



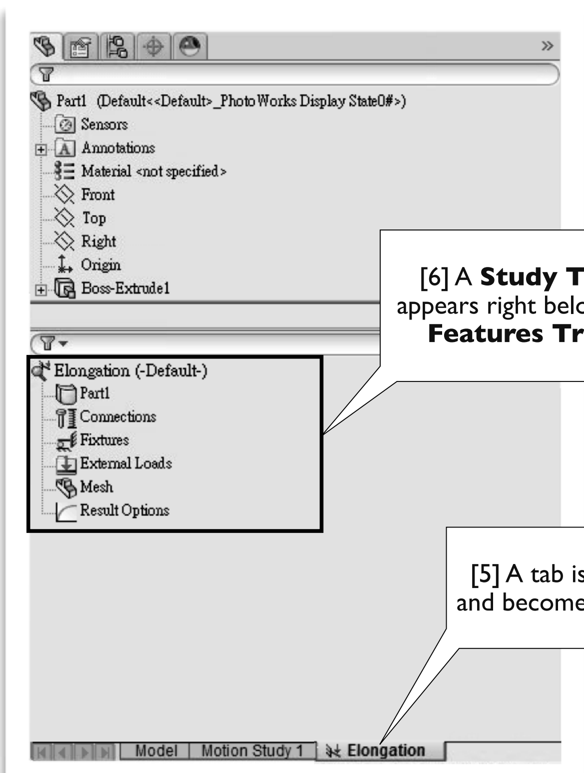
[1] In the **Simulation Toolbar**, select **Study Advisor>New Study**.



[2] By default, **Static** (static structural study) is selected as study **Type**.

[3] Type **Elongation** for study **Name**.

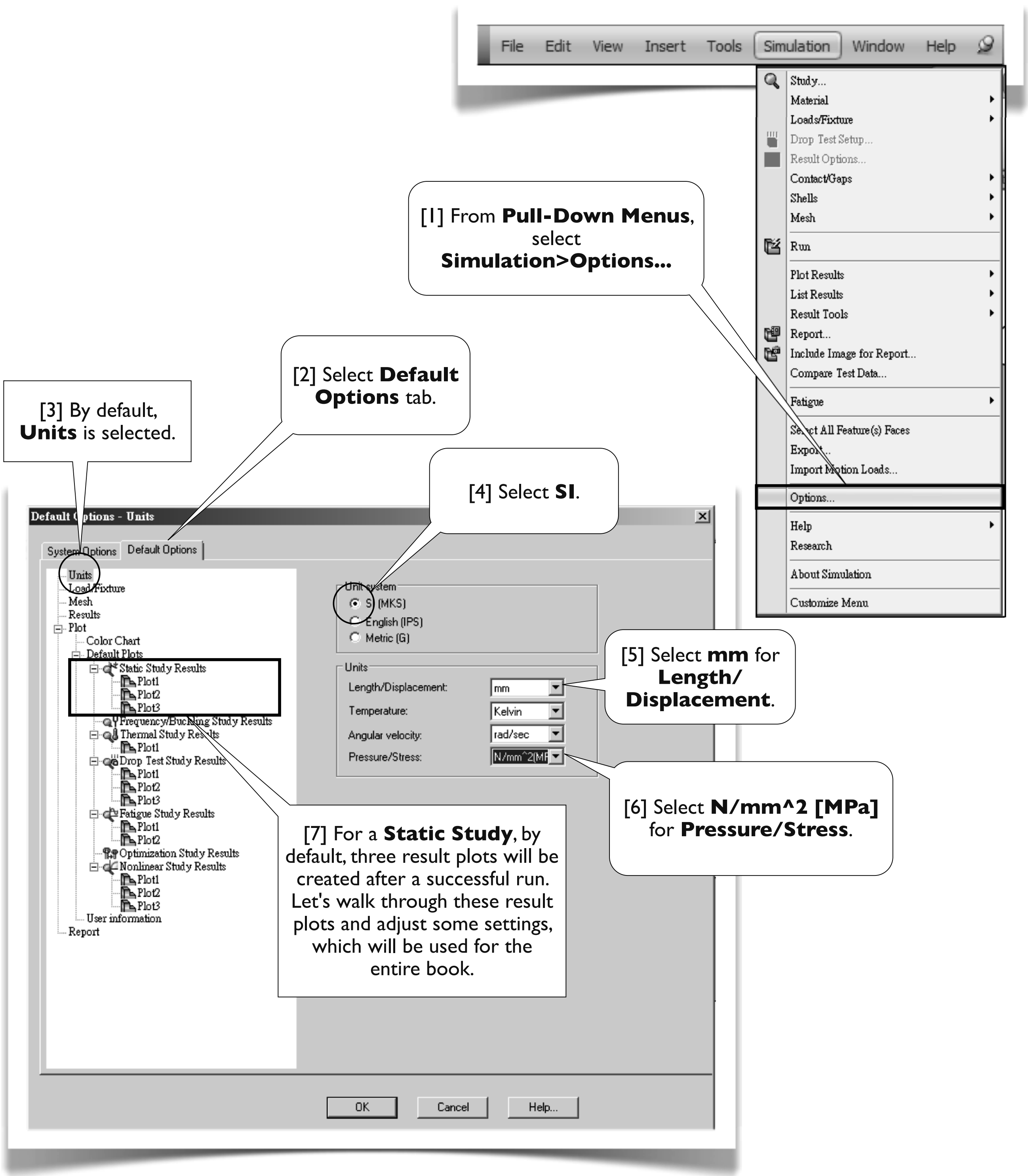
[4] Click **OK**.

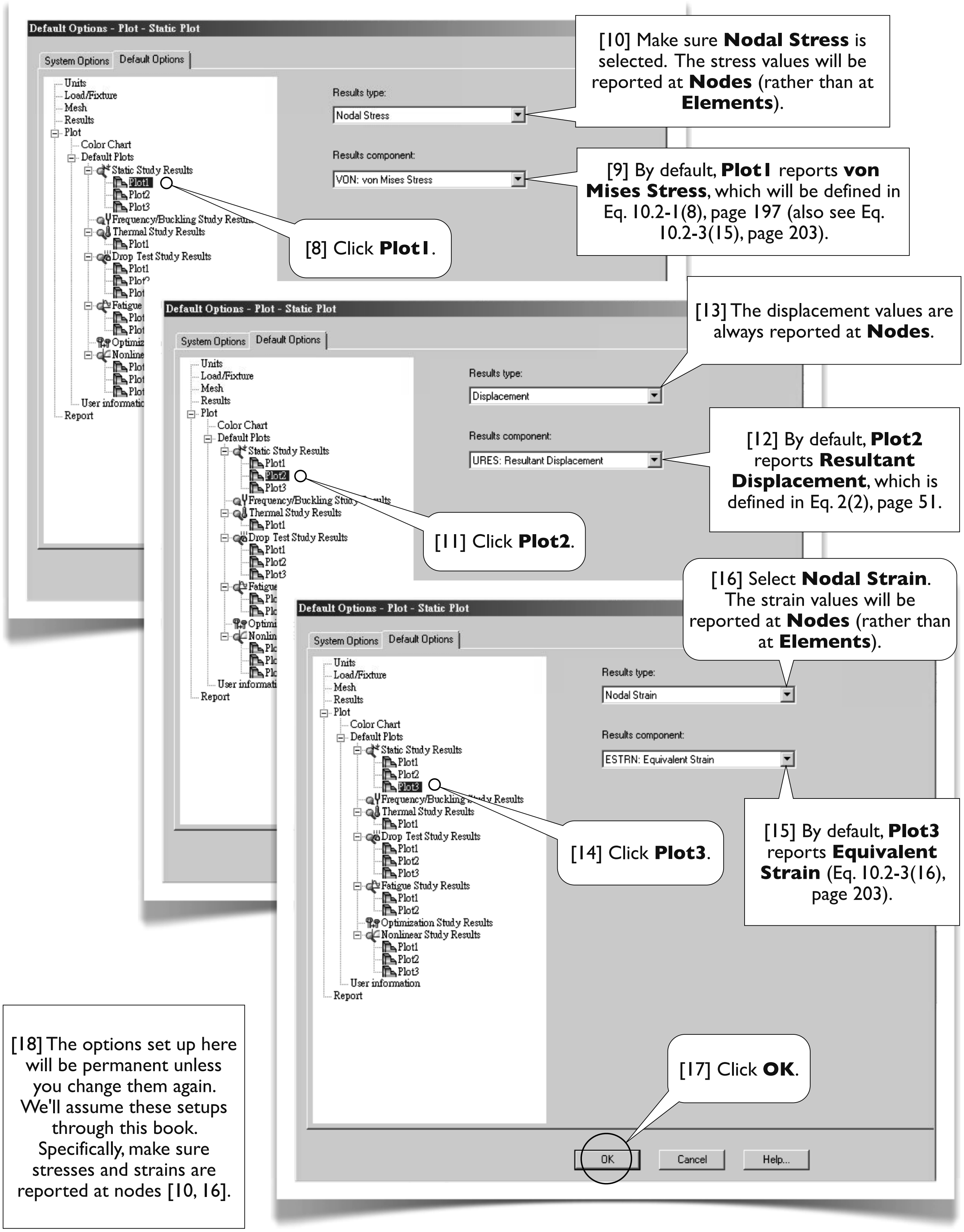


[5] A tab is added and becomes active.

[6] A **Study Tree** appears right below the **Features Tree**.

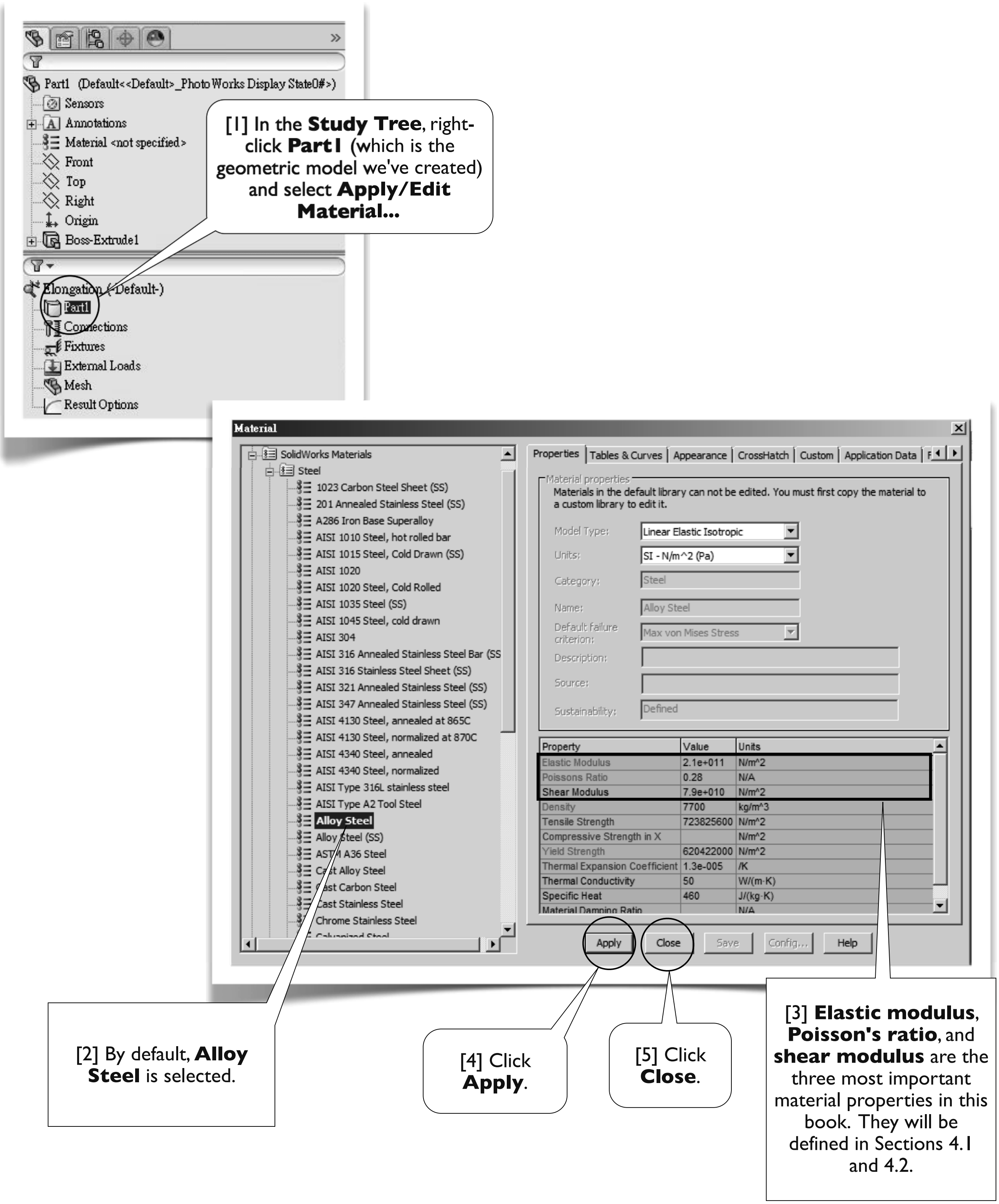
1.1-7 Set Up **Options** for **SolidWorks Simulation**



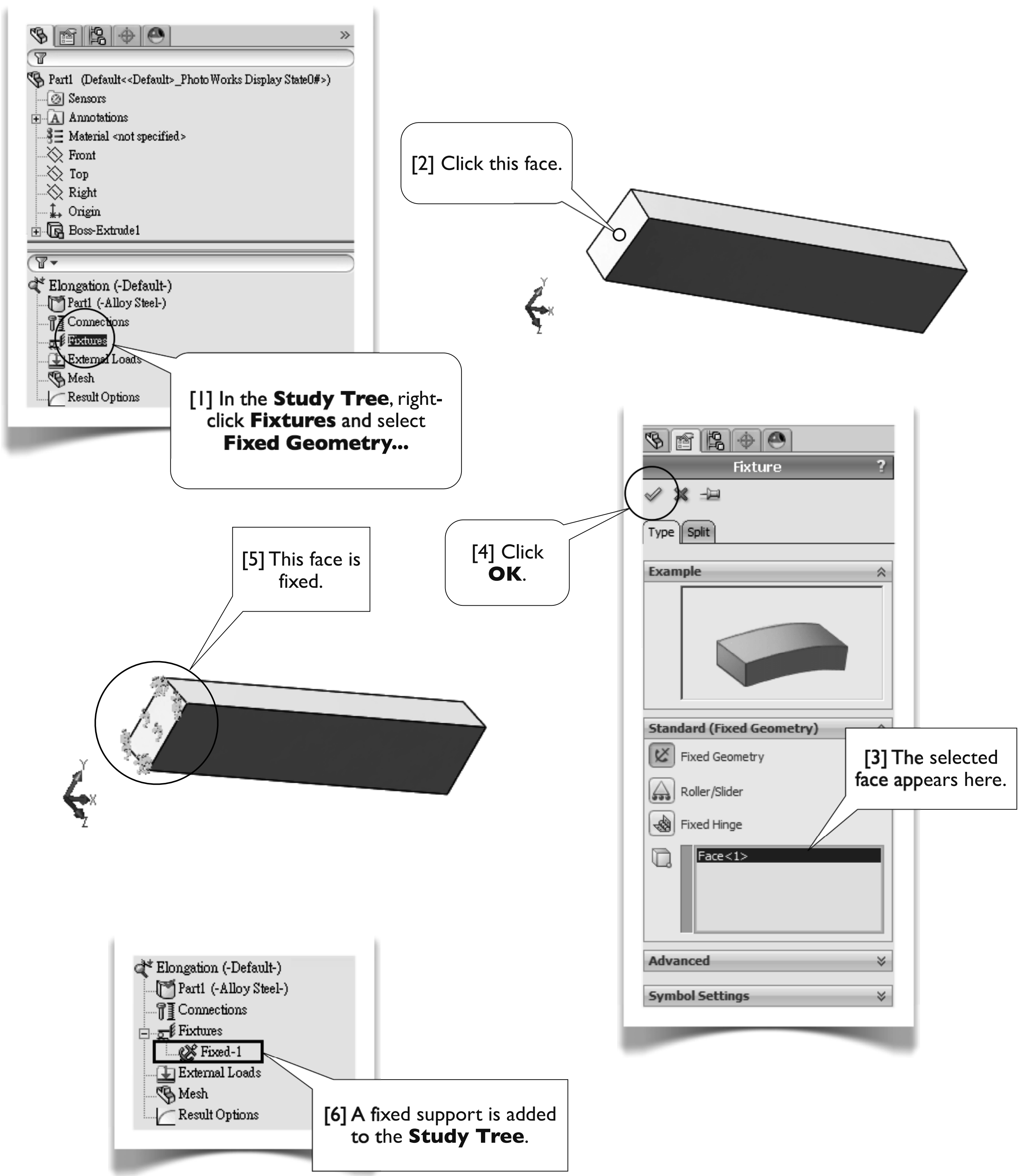


[18] The options set up here will be permanent unless you change them again. We'll assume these setups through this book. Specifically, make sure stresses and strains are reported at nodes [10, 16].

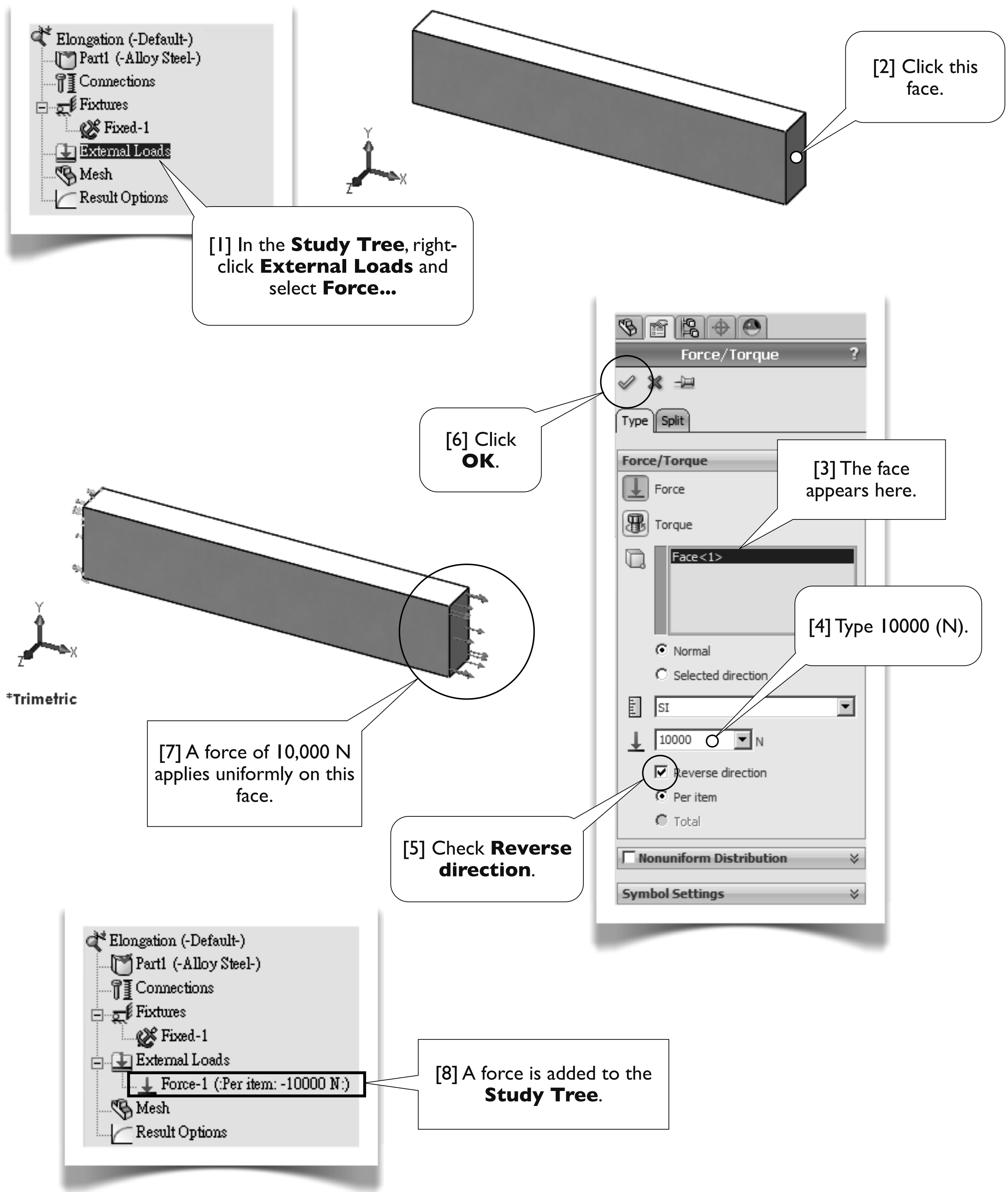
1.1-8 Apply Material



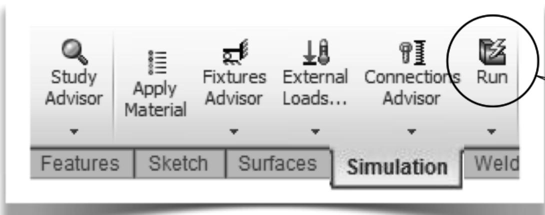
1.1-9 Apply Support



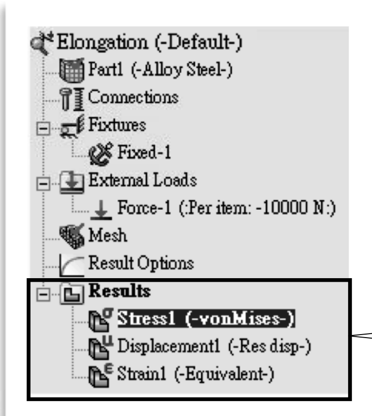
1.1-10 Apply Load



1.1-11 Solve the Model

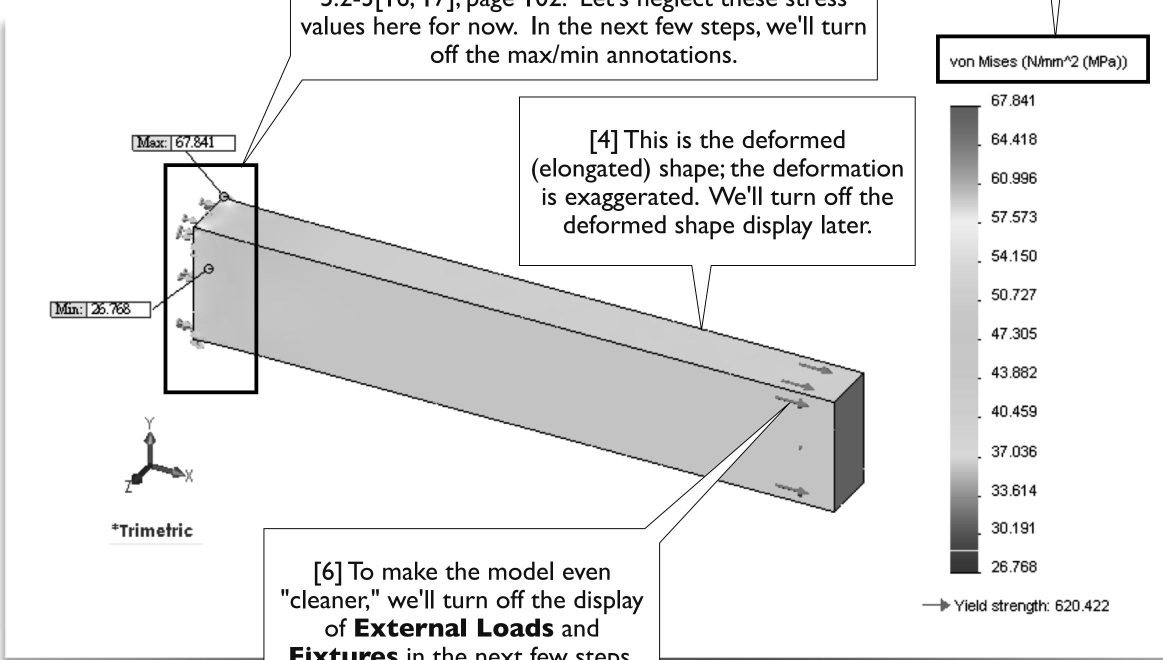


[1] In the **Simulation Toolbar**, click **Run**. It takes only a few seconds to solve the model.



[2] As mentioned earlier (1.1-7), by default, results for **Von Mises Stress, Resultant Displacement**, and **Equivalent Strain** are created. Note that **Stress1** is highlighted, meaning it is active and displayed in the **Graphics Area**.

[3] By default, **Von Mises Stress** is displayed. We'll change to display σ_x later.

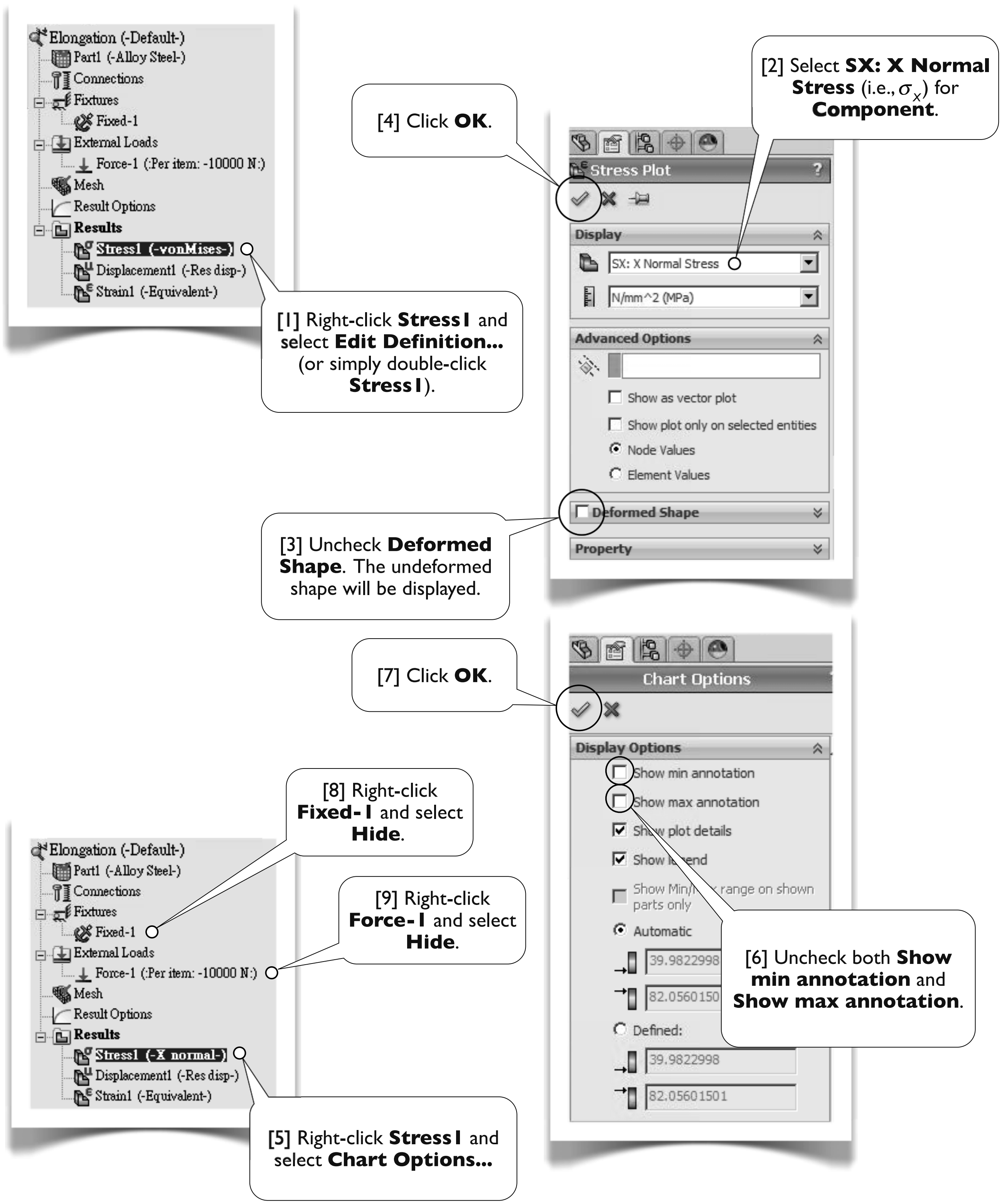


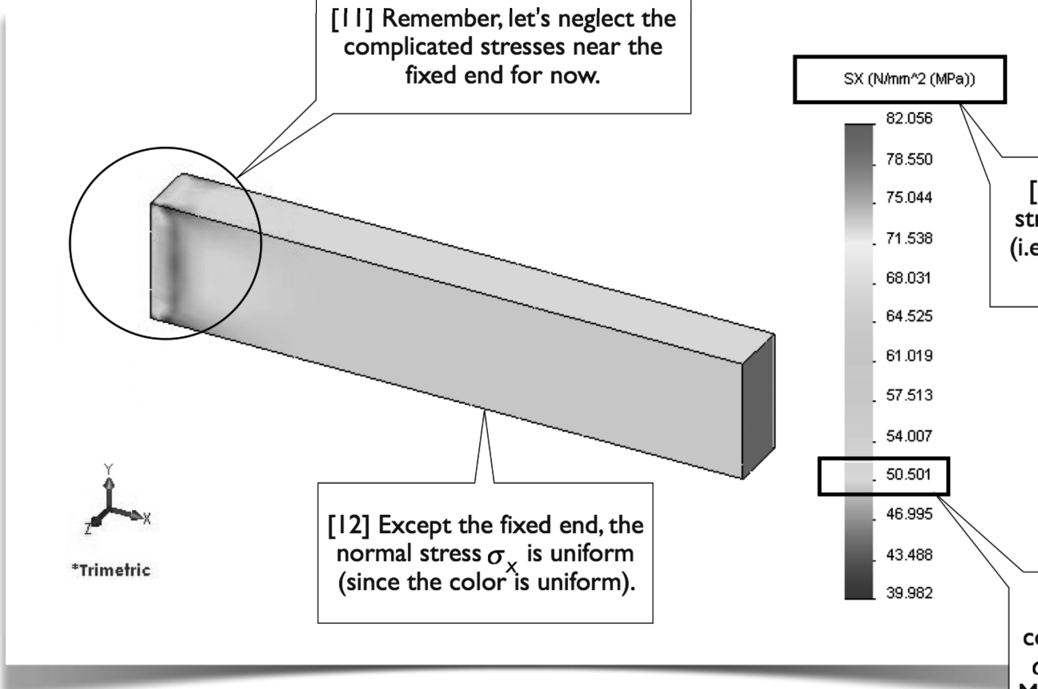
[5] The stress is uniform over the entire body except the area near the fixed end, where the stresses are complicated and we'll explain this phenomenon in 5.2-3[16, 17], page 102. Let's neglect these stress values here for now. In the next few steps, we'll turn off the max/min annotations.

[4] This is the deformed (elongated) shape; the deformation is exaggerated. We'll turn off the deformed shape display later.

[6] To make the model even "cleaner," we'll turn off the display of **External Loads** and **Fixtures** in the next few steps.

1.1-12 View the Normal Stress σ_x



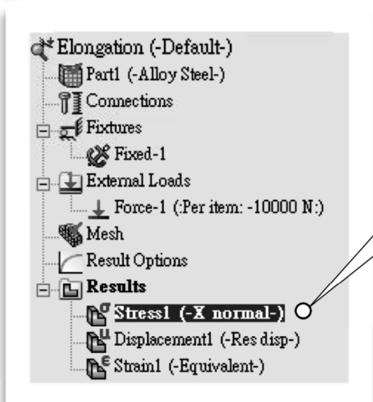


[11] Remember, let's neglect the complicated stresses near the fixed end for now.

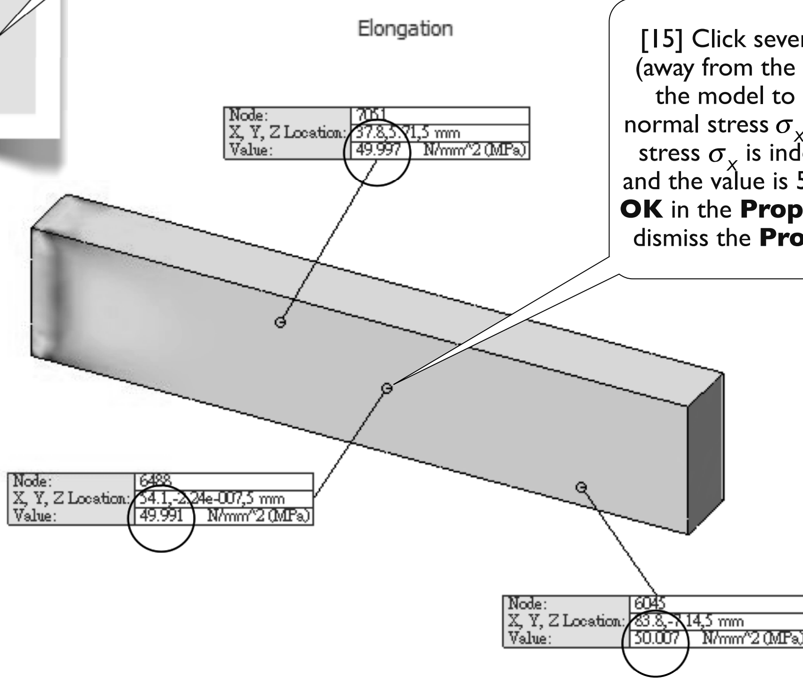
[10] Now, the normal stress in the X-direction (i.e., σ_x) is displayed. The unit is MPa.

[12] Except the fixed end, the normal stress σ_x is uniform (since the color is uniform).

[13] From the legend colors, the normal stress σ_x is approximately 50 MPa. Let's use **Probe** to obtain a more accurate value.

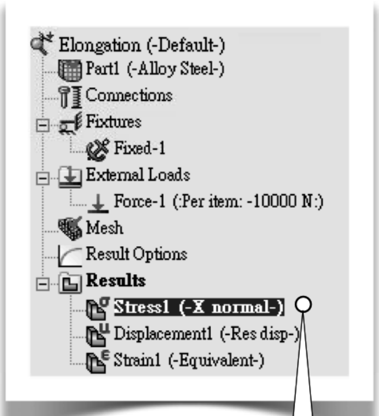


[14] Right-click **Stress1** and select **Probe**.



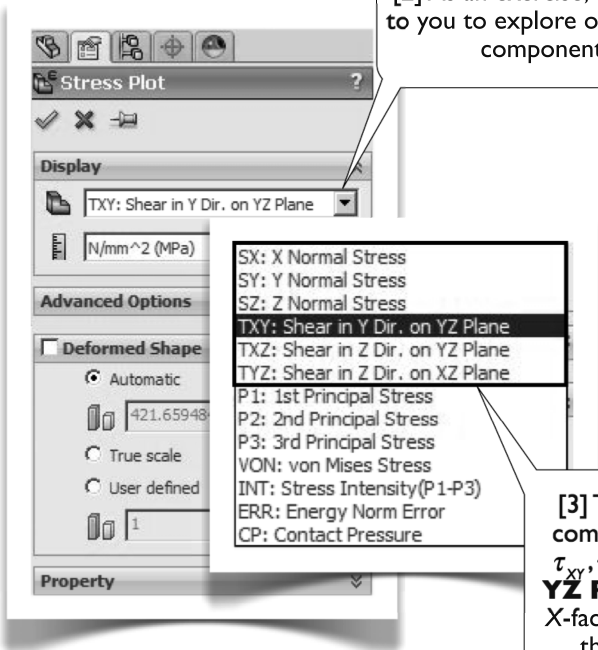
[15] Click several locations (away from the fixed end) in the model to display the normal stress σ_x. The normal stress σ_x is indeed uniform and the value is 50 MPa. Click **OK** in the **Property Box** to dismiss the **Probe Result**.

1.1-13 View Other Stress Components



[1] Right-click **Stress1** and select **Edit Definition...** (or double-click **Stress1**).

[2] As an exercise, we leave it to you to explore other stress components.

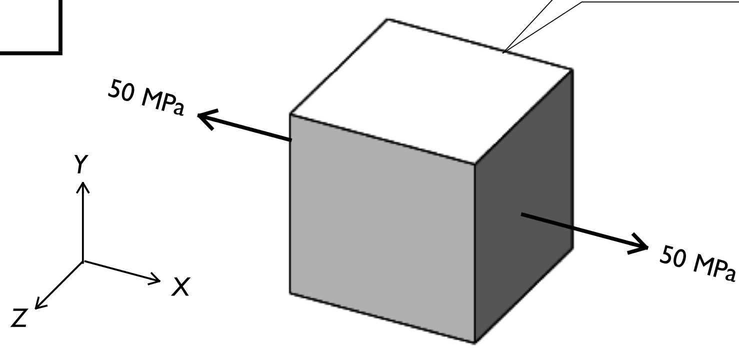


[3] These are six stress components ($\sigma_X, \sigma_Y, \sigma_Z, \tau_{XY}, \tau_{XZ}, \tau_{YZ}$). Note that, **YZ Plane** is the same as *X*-face, and **XZ Plane** is the same as *Y*-face.

Stress Component	Stress Value
σ_X	50 Mpa
σ_Y	0
σ_Z	0
τ_{XY}	0
τ_{XZ}	0
τ_{YZ}	0

[4] Write down each stress component value. The results should be like this. In this example, all stress components are essentially zeros, except σ_X (which is 50 MPa)

[5] The stress state of any point in the cantilever beam can be represented like this. In the next exercise, we'll explore a case in which a shear stress component is non-zero and the stress states are non-uniform.



1.1-14 Save the Document and Exit **SolidWorks**

[1] Click **Save** to save the document with the name **Cantilever**. Two files are created in your working folder: **Cantilever.SLDPRT** and **Cantilever-Elongation.CWR**; the former is the main project file, while the later stores the result data generated by a finite element solver. Other files, if any, are not relevant; they can be deleted.

[2] From the **Pull-Down Menus**, select **File>Exit** to exit **SolidWorks**.