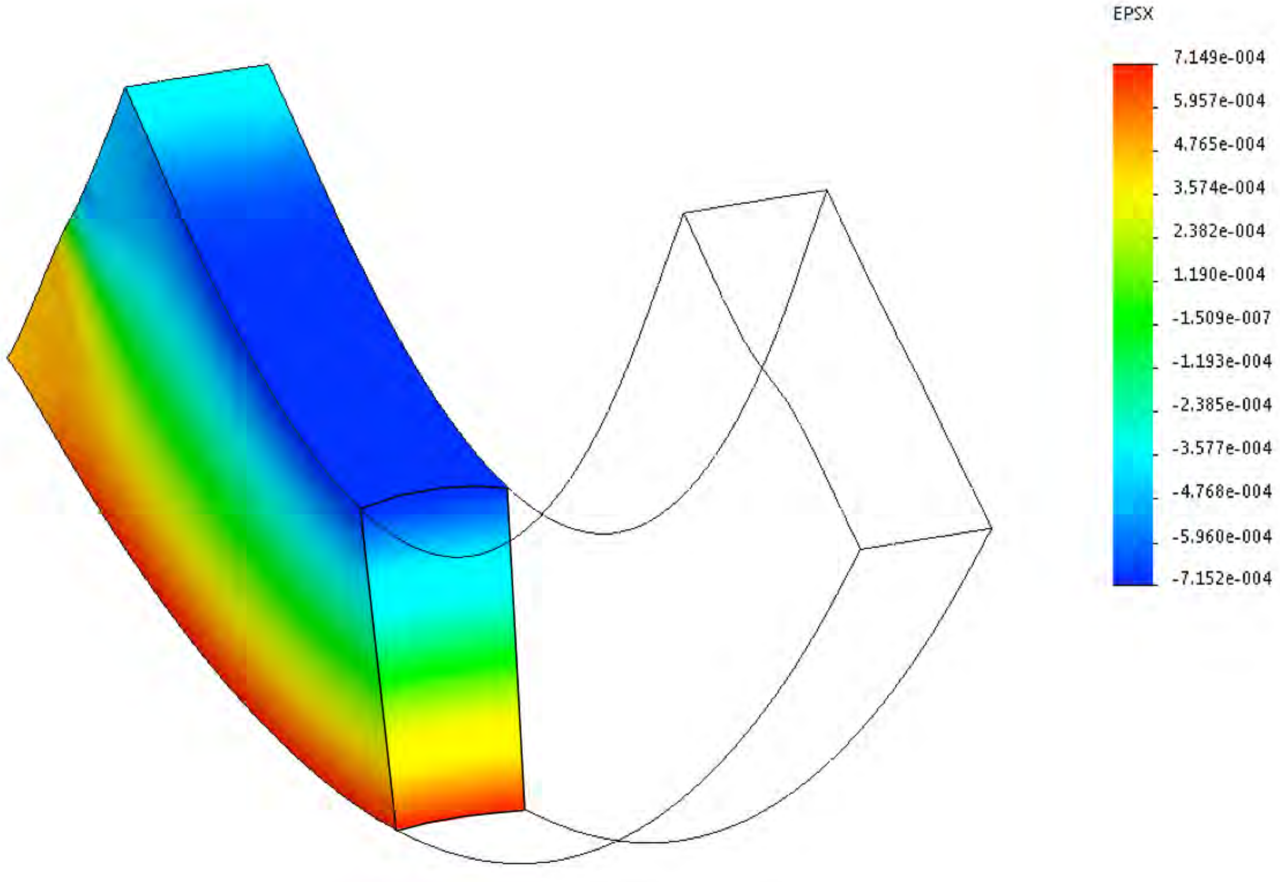
Mechanics of Materials Labs with SolidWorks® Simulation 2014



Huei-Huang Lee

Multimedia Disc

Includes Supplemental Files and Video Instruction

SDC PUBLICATIONS

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 a position-dependent quantity.

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 a 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 meters 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, then 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 $\sigma = \rho gh$, 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 behavior is quite different 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:

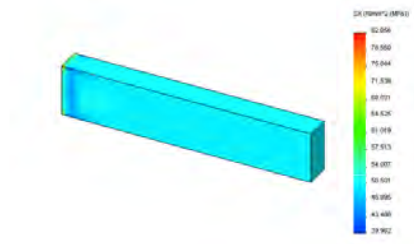
> *The stress at a certain position in a solid material is defined as the force per unit surface area on an infinitesimally small body placed at that position.*

Note that the infinitesimally small body could be any shapes. However, 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



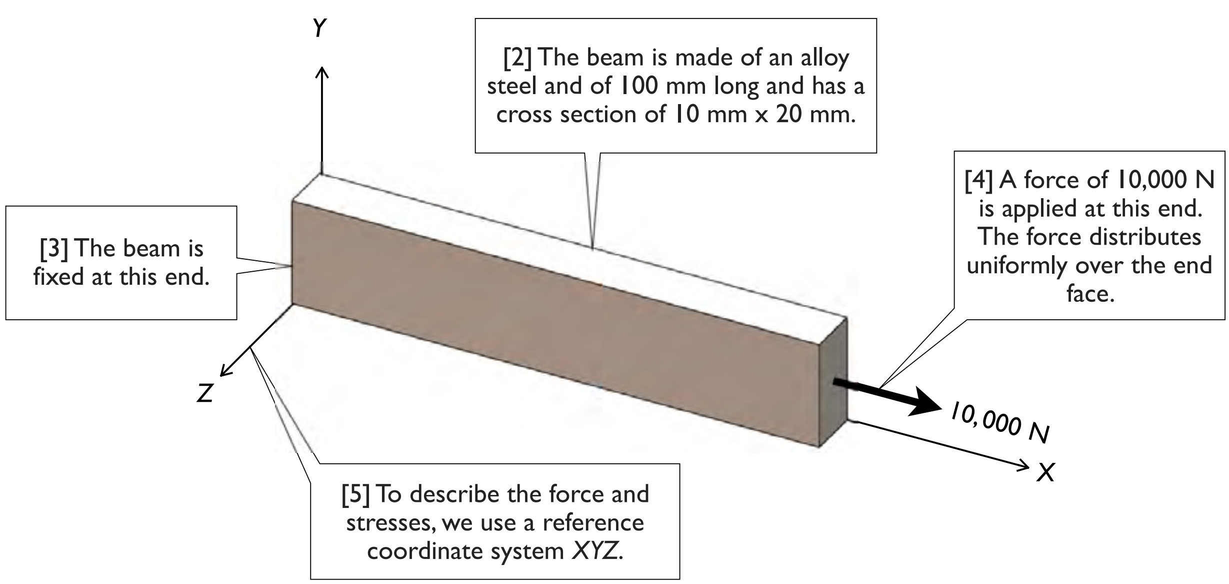
1.1-1 Introduction

[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 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

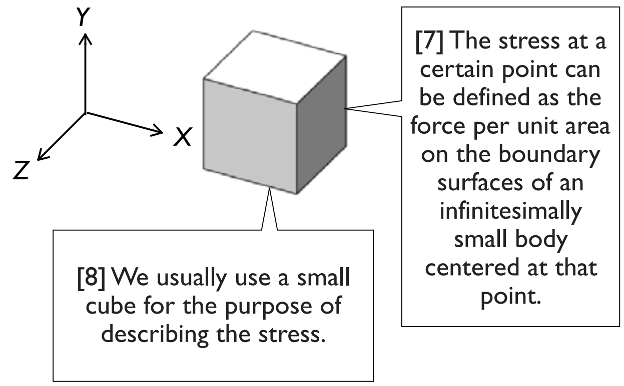
$$\frac{10{,}000\ \text{N}}{10\ \text{mm}\ \times\ 20\ \text{mm}} = 50\ \text{MPa?}$$

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?



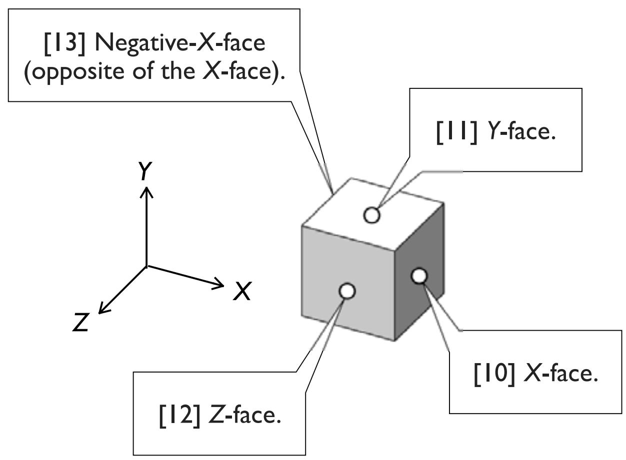
[6] Definition of Stress

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. If we can find the stresses on a small cube, we then can calculate the stresses on any other shapes of small body (see [18]).



[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].

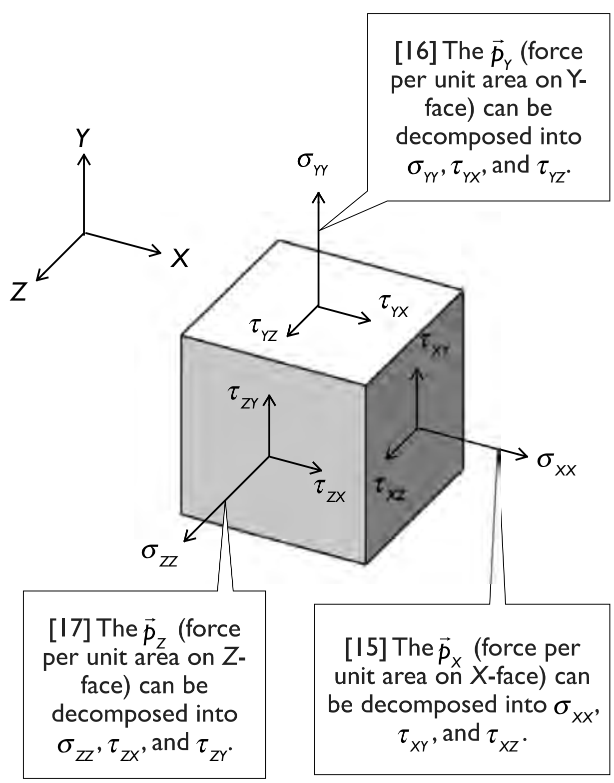


[14] Stress Components

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]. 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. 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

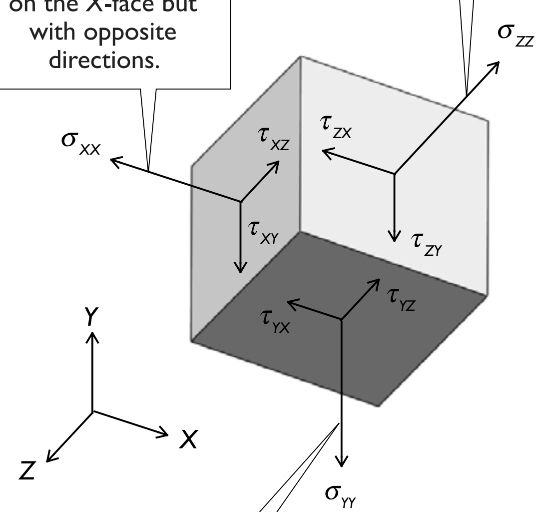
It can be proven that the stress components on the negative-*X*-face, negative-*Y*-face, and negative-*Z*-face can be derived from the 9 stress components in Eq. (1). For example, on the negative-*X*-face, the stress components have exactly the same stress values as those on the *X*-face but with opposite 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 also can be calculated from the 9 stress components in Eq. (1). We'll show that this can be done using Mohr's circles (Section 10.1).

[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.

[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.



[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.

[22] Symmetry of Shear Stresses

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

$$\tau_{XY} = \tau_{YX}, \quad \tau_{YZ} = \tau_{ZY}, \quad \tau_{ZX} = \tau_{XZ} \tag{2}$$

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 State**

We now conclude that 3 normal stress components and 3 shear stress components are needed to describe the **stress state** at a certain point, which may be written in a vector form

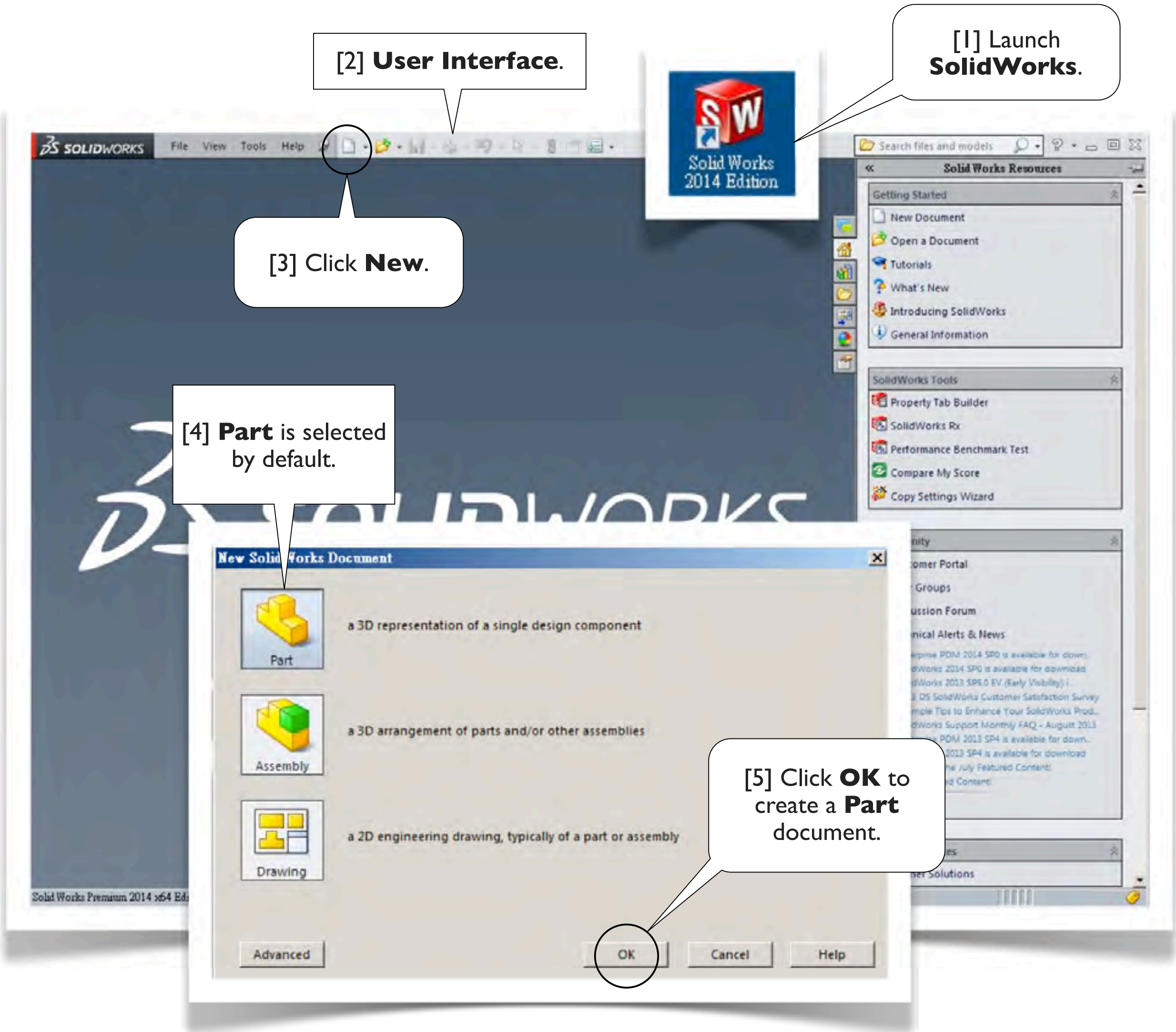
$$\{\sigma\} = \{ \sigma_X \quad \sigma_Y \quad \sigma_Z \quad \tau_{XY} \quad \tau_{YZ} \quad \tau_{ZX} \} \tag{3}$$

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 guide the students 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. #

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



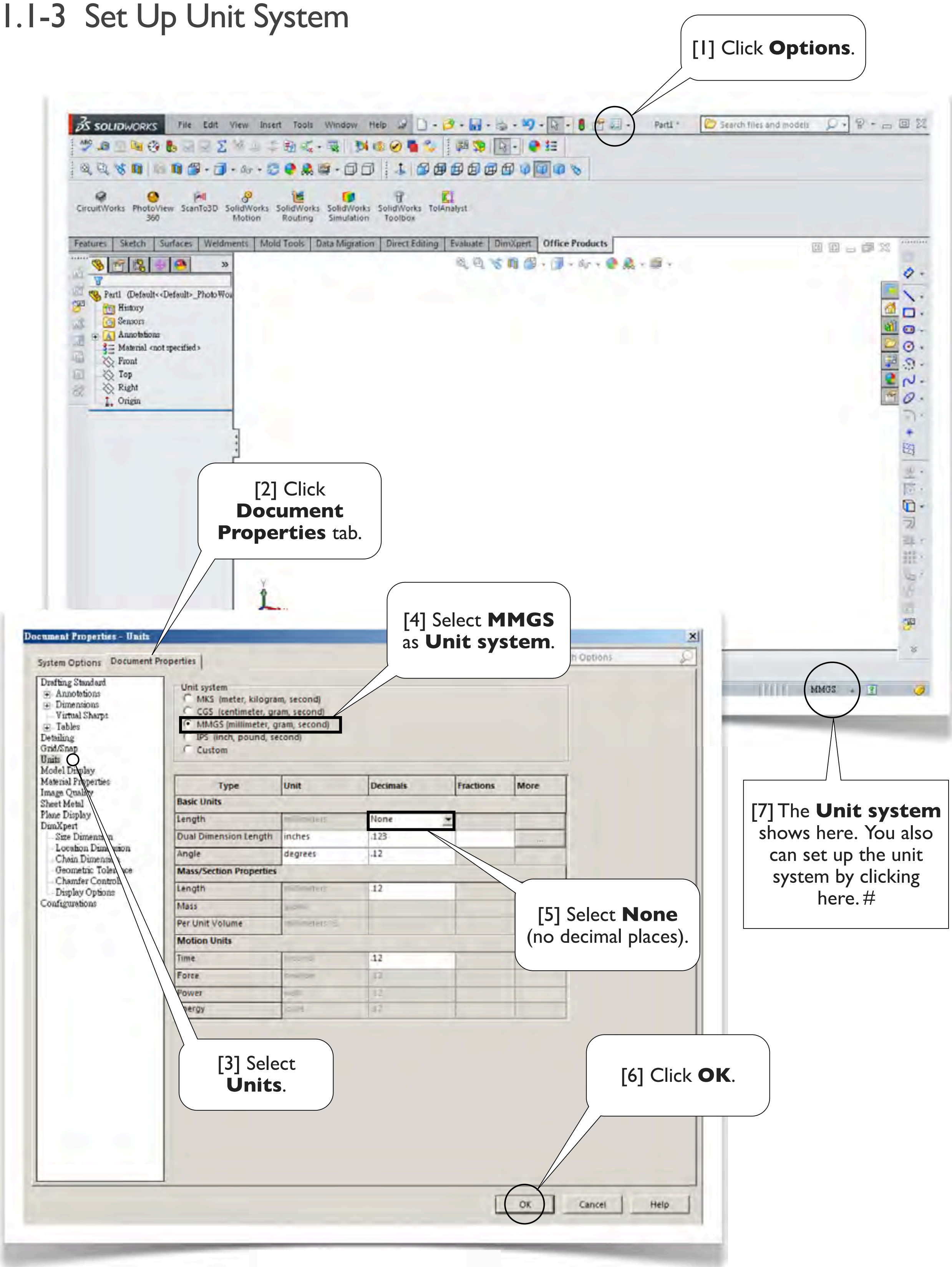
About the Text Boxes

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 of a subsection, please follow the order of the text boxes.
2. The text box numbers are also used as reference numbers. 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; i.e., mouse or keyboard actions are not needed in that step.
4. A symbol # is used to indicate the last text box of a subsection, so that you don't leave out any text boxes.

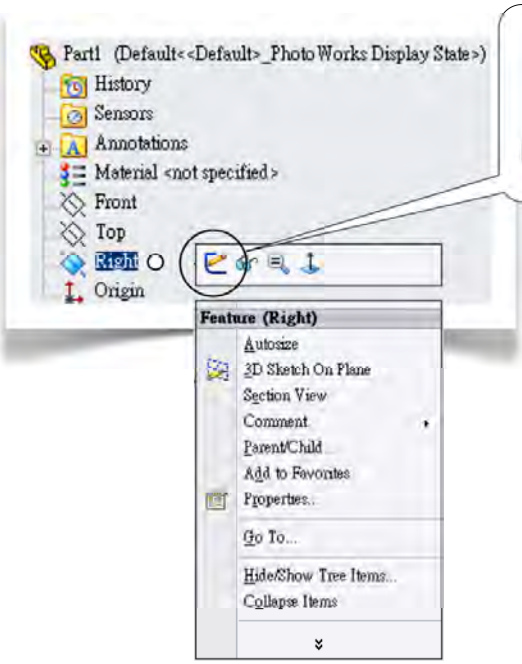
SolidWorks Terms

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

1.1-3 Set Up Unit System

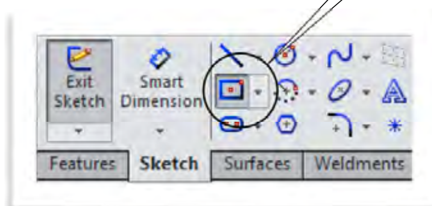


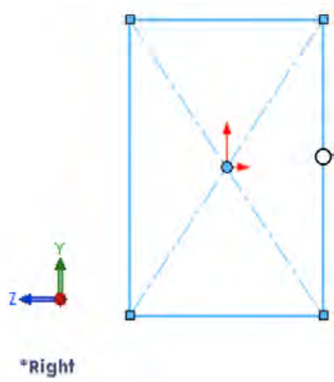
1.1-4 Create Geometric Model



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

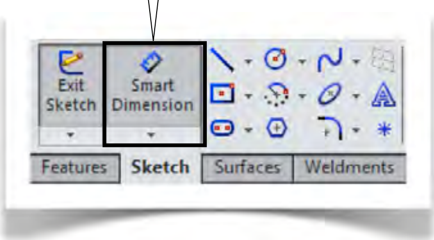
[2] In the **Sketch Toolbar**, select **Center Rectangle**.

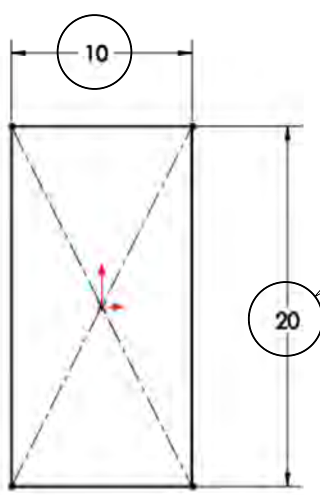




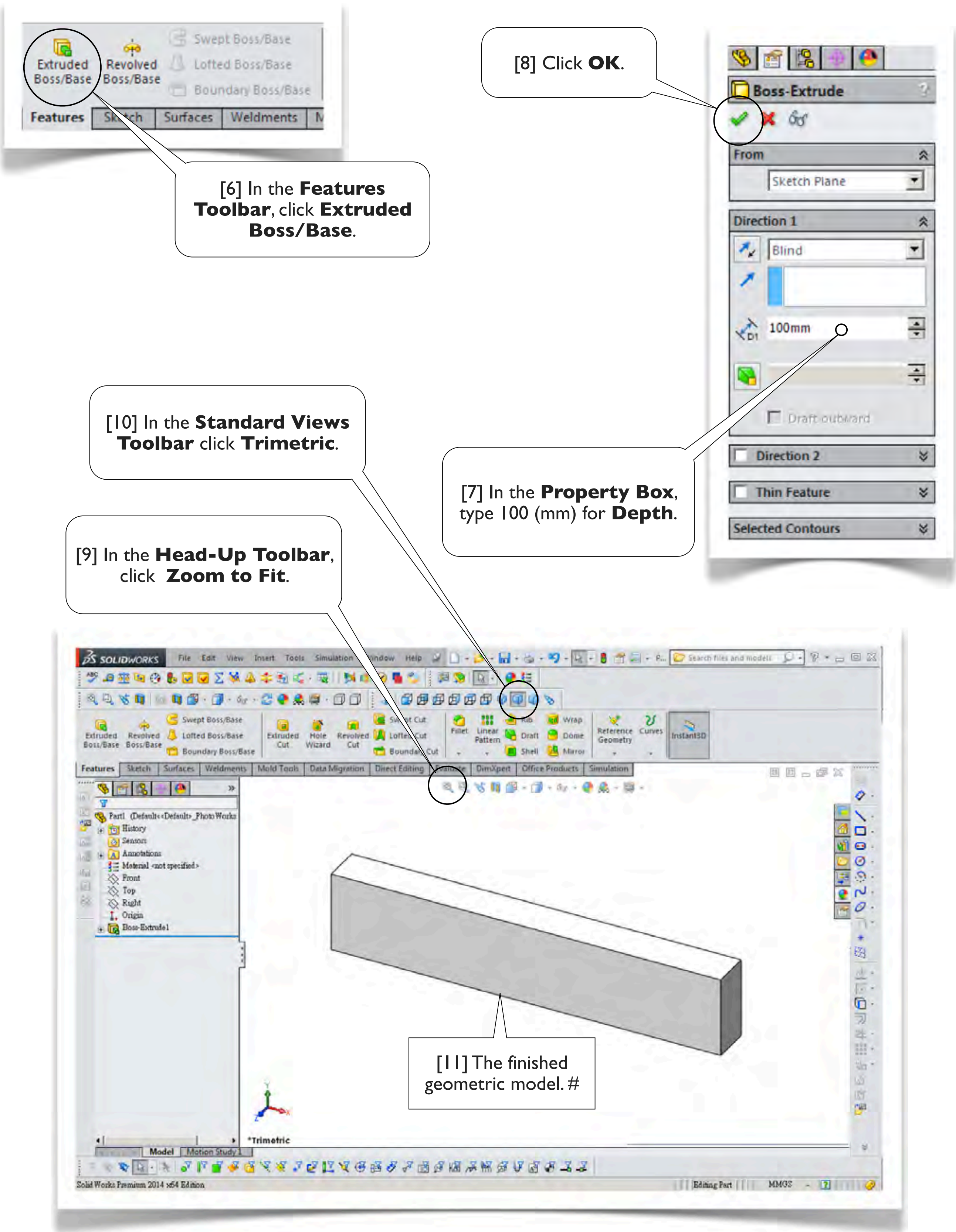
[3] Draw a rectangle centered at the origin (the sizes are arbitrary for now).

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

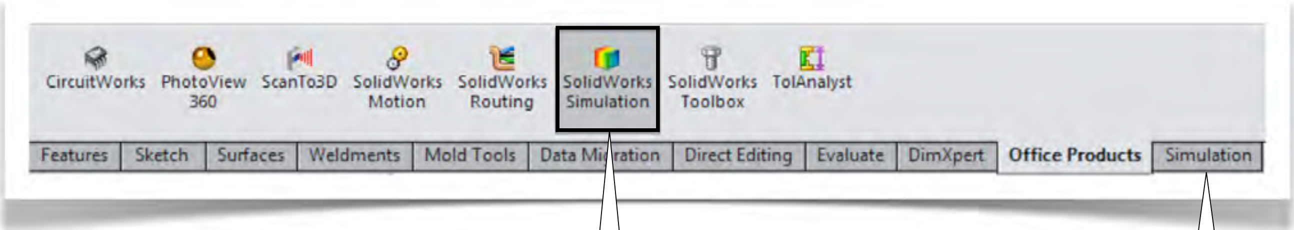




[5] Specify dimensions (10 mm and 20 mm) like this.

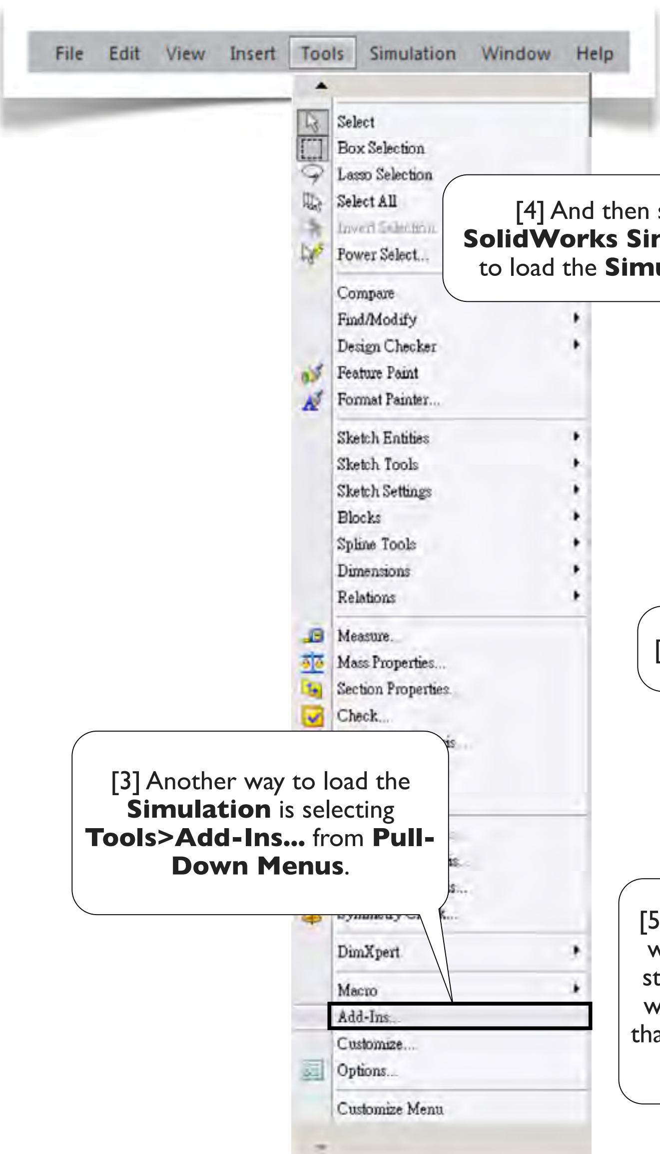

Extruded Boss/Base
Revolved Boss/Base
Swept Boss/Base
Lofted Boss/Base
Boundary Boss/Base
Features
Sketch
Surfaces
Weldments
[6] In the **Features Toolbar**, click **Extruded Boss/Base**.
[8] Click **OK**.
Boss-Extrude
From
Sketch Plane
Direction 1
Blind
100mm
Draft outward
Direction 2
Thin Feature
Selected Contours
[7] In the **Property Box**, type 100 (mm) for **Depth**.
[10] In the **Standard Views Toolbar** click **Trimetric**.
[9] In the **Head-Up Toolbar**, click **Zoom to Fit**.
*Trimetric
[11] The finished geometric model. #

1.1-5 Load **SolidWorks Simulation**



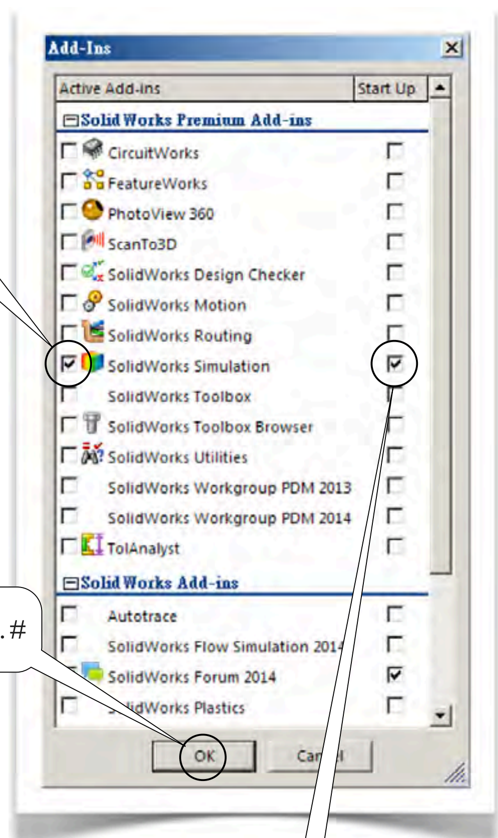
[1] If a **Simulation** tab is already present, which means the **SolidWorks Simulation** is already loaded, you may jump to 1.1-6 (next page) otherwise continue on step [2].

[2] In the **Office Products Toolbar**, click to load **SolidWorks Simulation**.



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

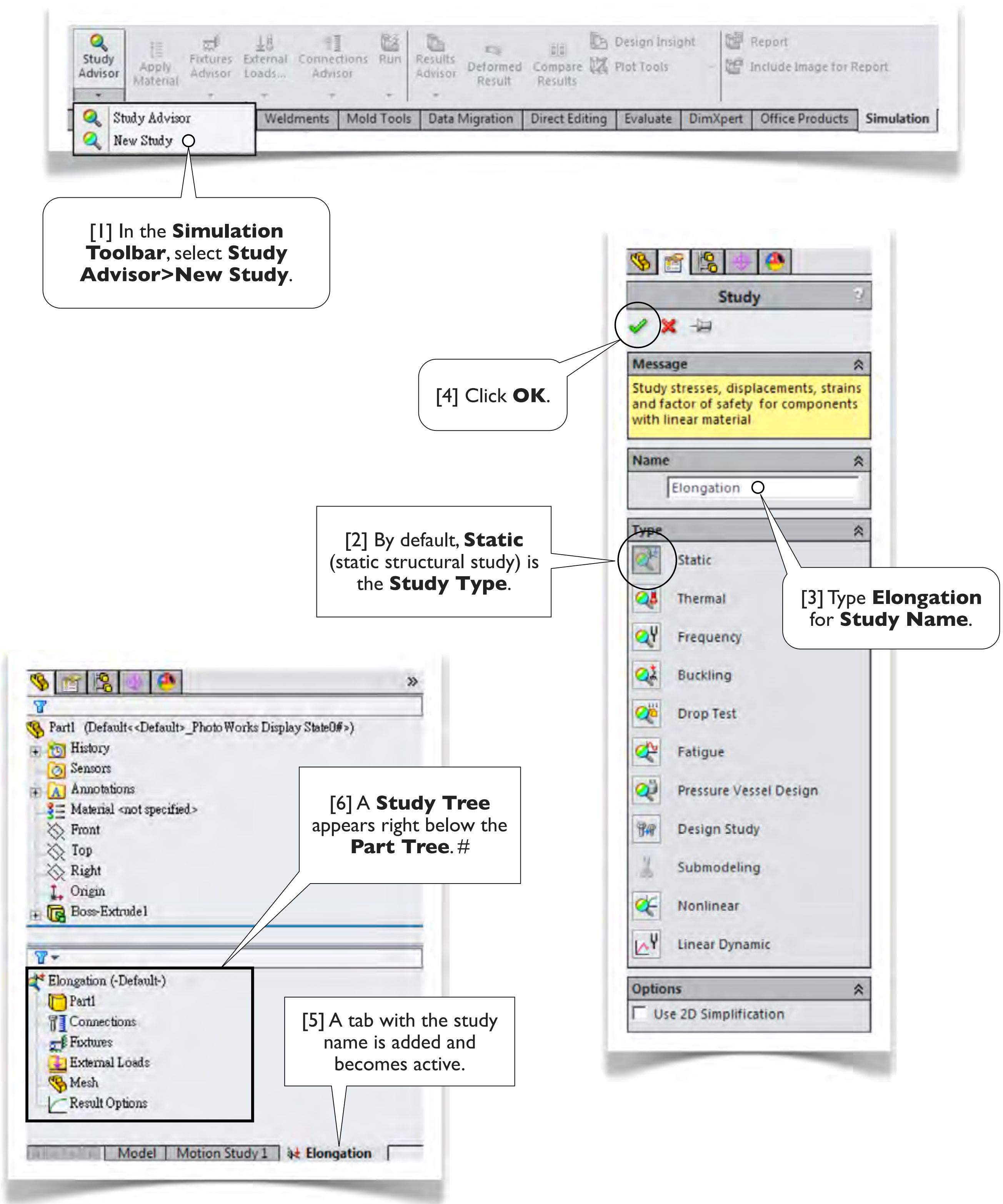
[4] And then select **SolidWorks Simulation** to load the **Simulation**.



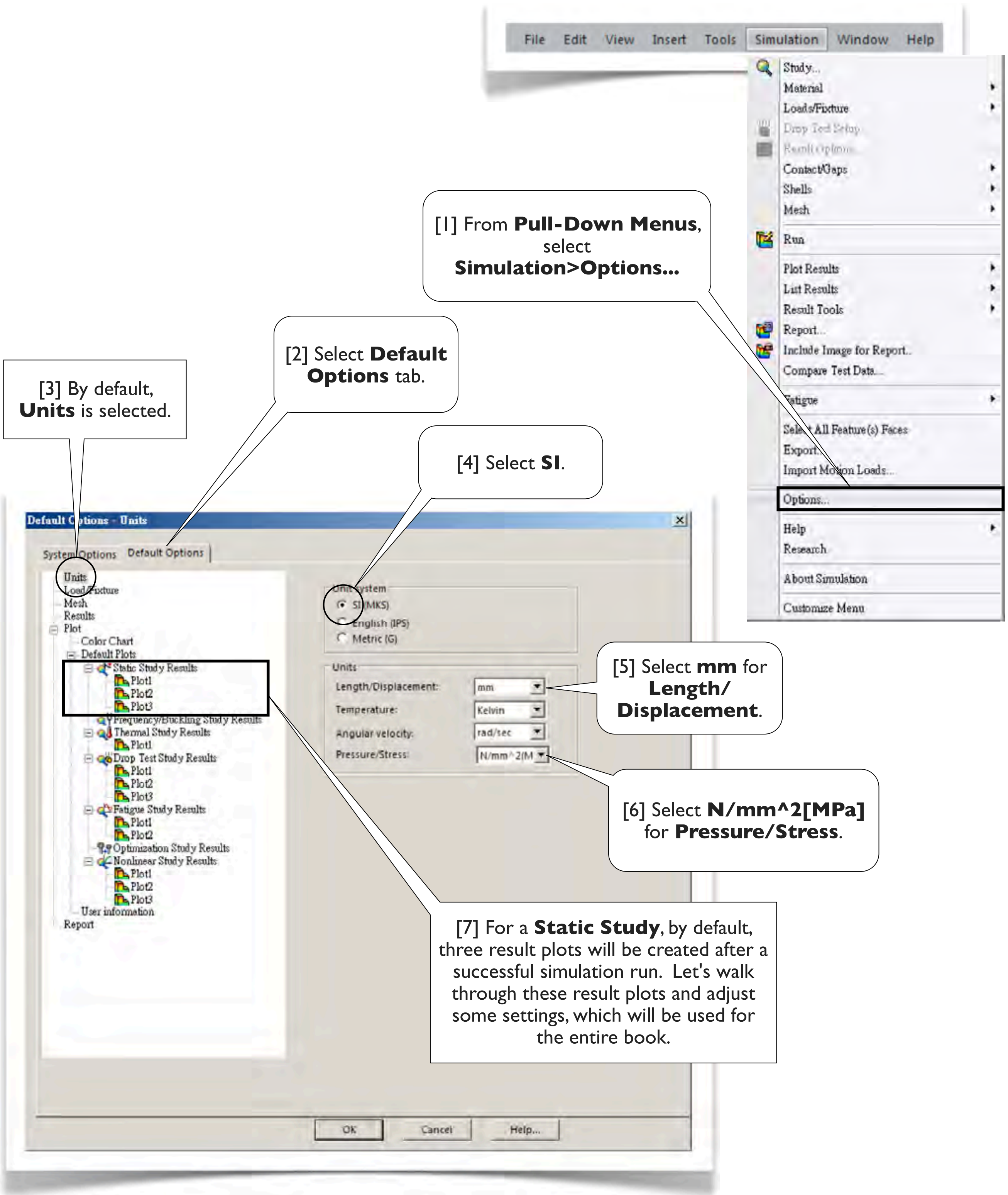
[5] Also click here so that the **Simulation** will be loaded automatically each time you start up **SolidWorks**. Through this book, we assume that you've checked this box so that the **Simulation** is loaded automatically each time you start up **SolidWorks**.

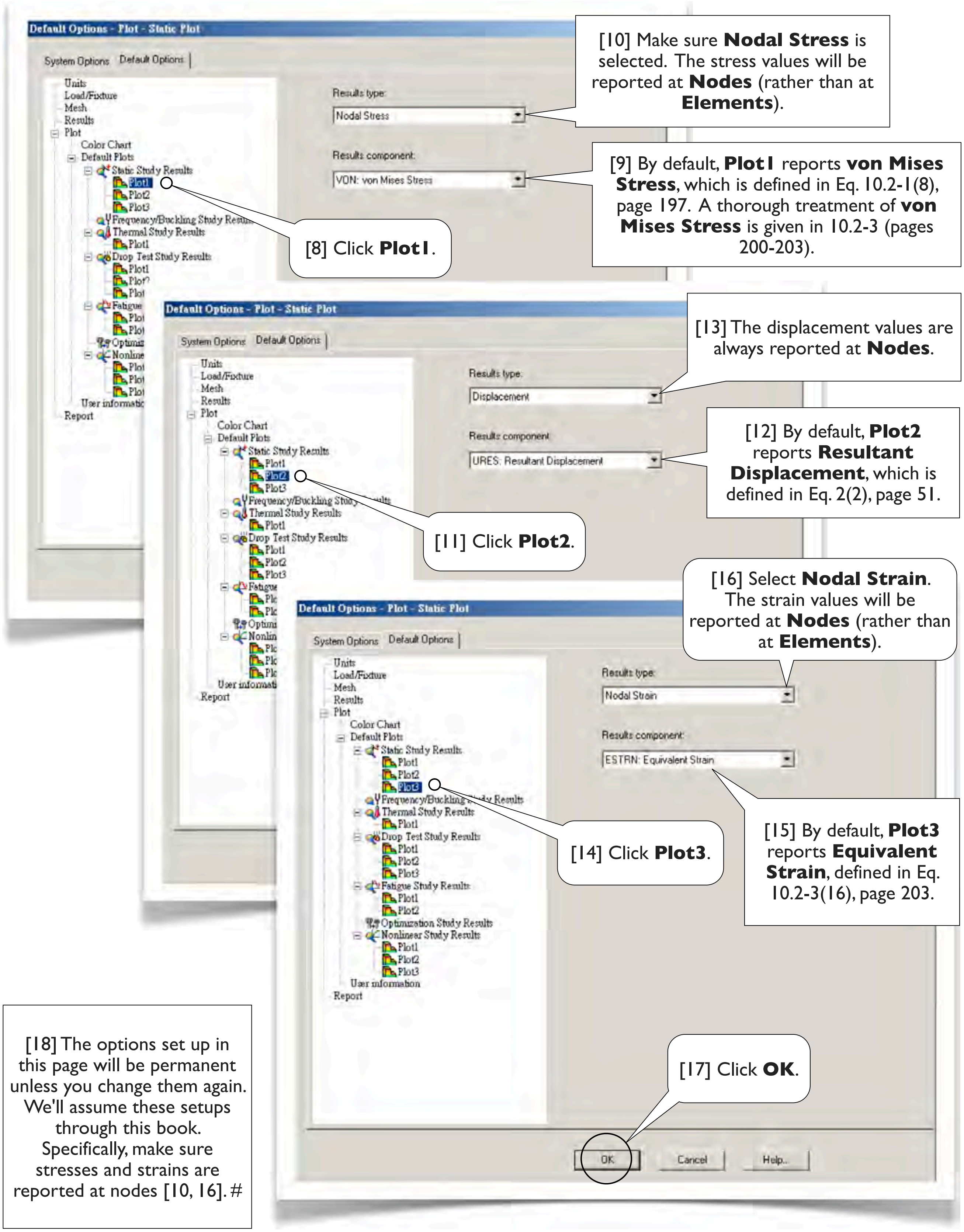
[6] Click **OK**. #

1.1-6 Create a Static Structural Study



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





[8] Click **Plot1**.

[9] By default, **Plot1** reports **von Mises Stress**, which is defined in Eq. 10.2-1(8), page 197. A thorough treatment of **von Mises Stress** is given in 10.2-3 (pages 200-203).

[10] Make sure **Nodal Stress** is selected. The stress values will be reported at **Nodes** (rather than at **Elements**).

[11] Click **Plot2**.

[12] By default, **Plot2** reports **Resultant Displacement**, which is defined in Eq. 2(2), page 51.

[13] The displacement values are always reported at **Nodes**.

[14] Click **Plot3**.

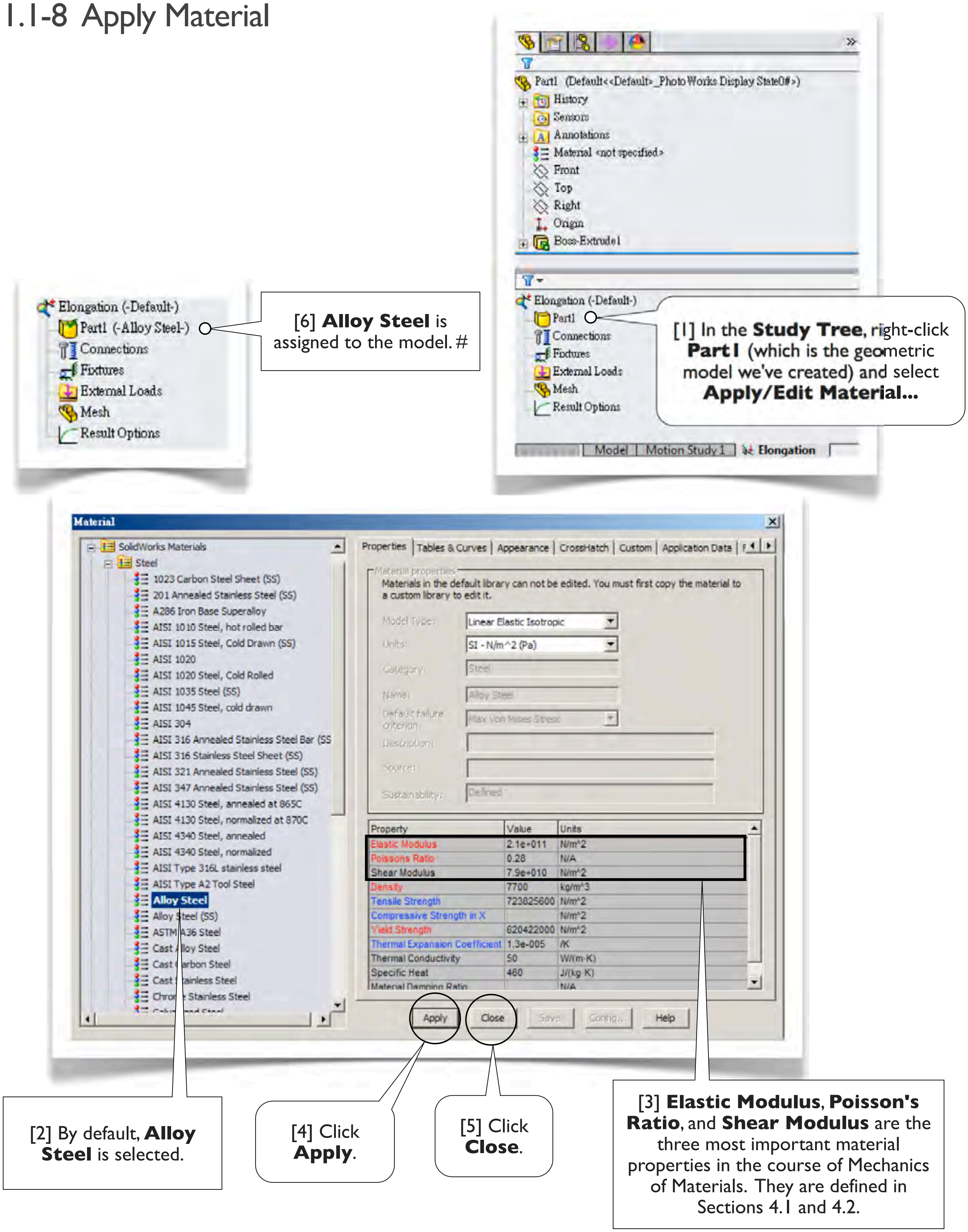
[15] By default, **Plot3** reports **Equivalent Strain**, defined in Eq. 10.2-3(16), page 203.

[16] Select **Nodal Strain**. The strain values will be reported at **Nodes** (rather than at **Elements**).

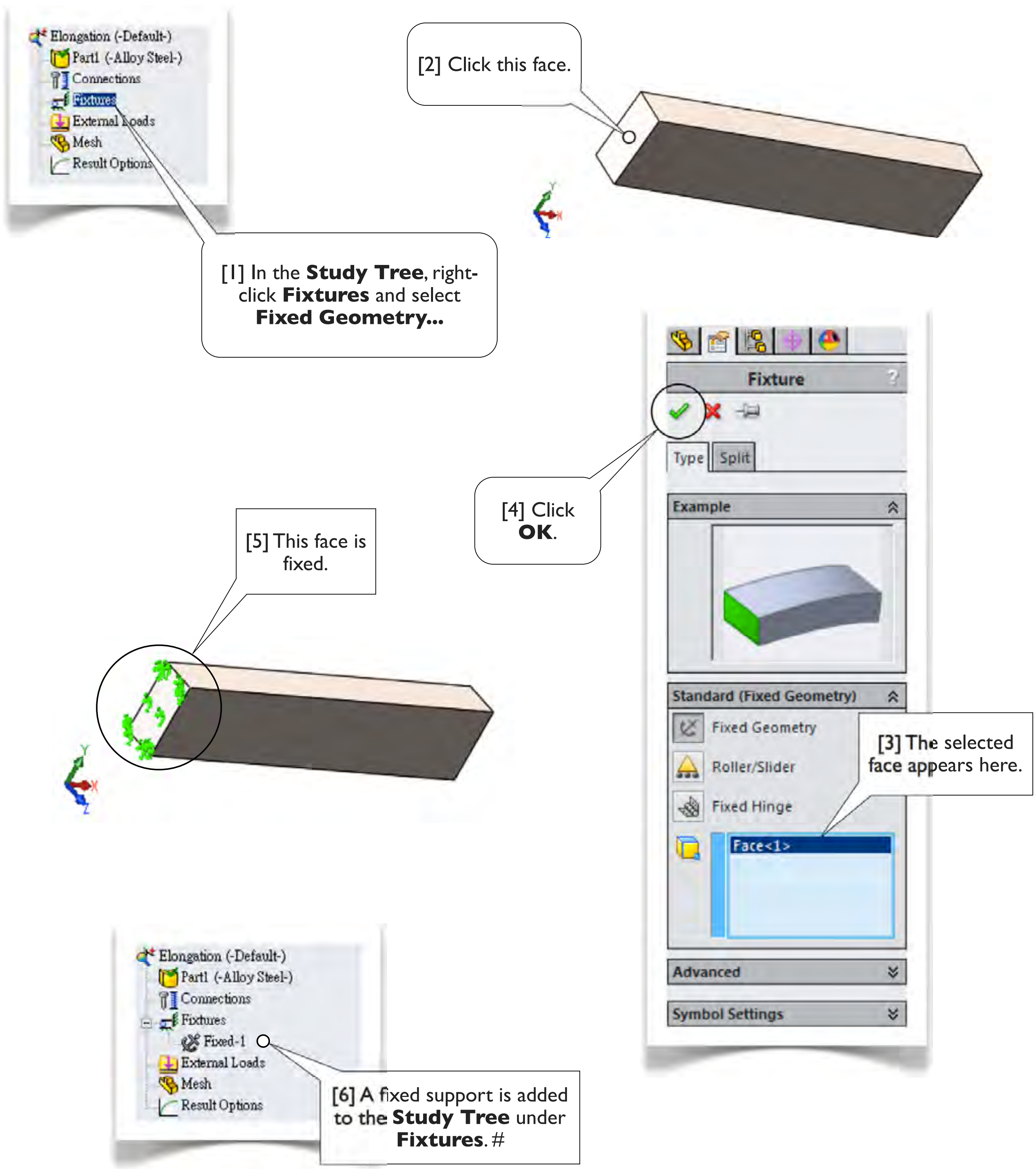
[17] Click **OK**.

[18] The options set up in this page 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]. #

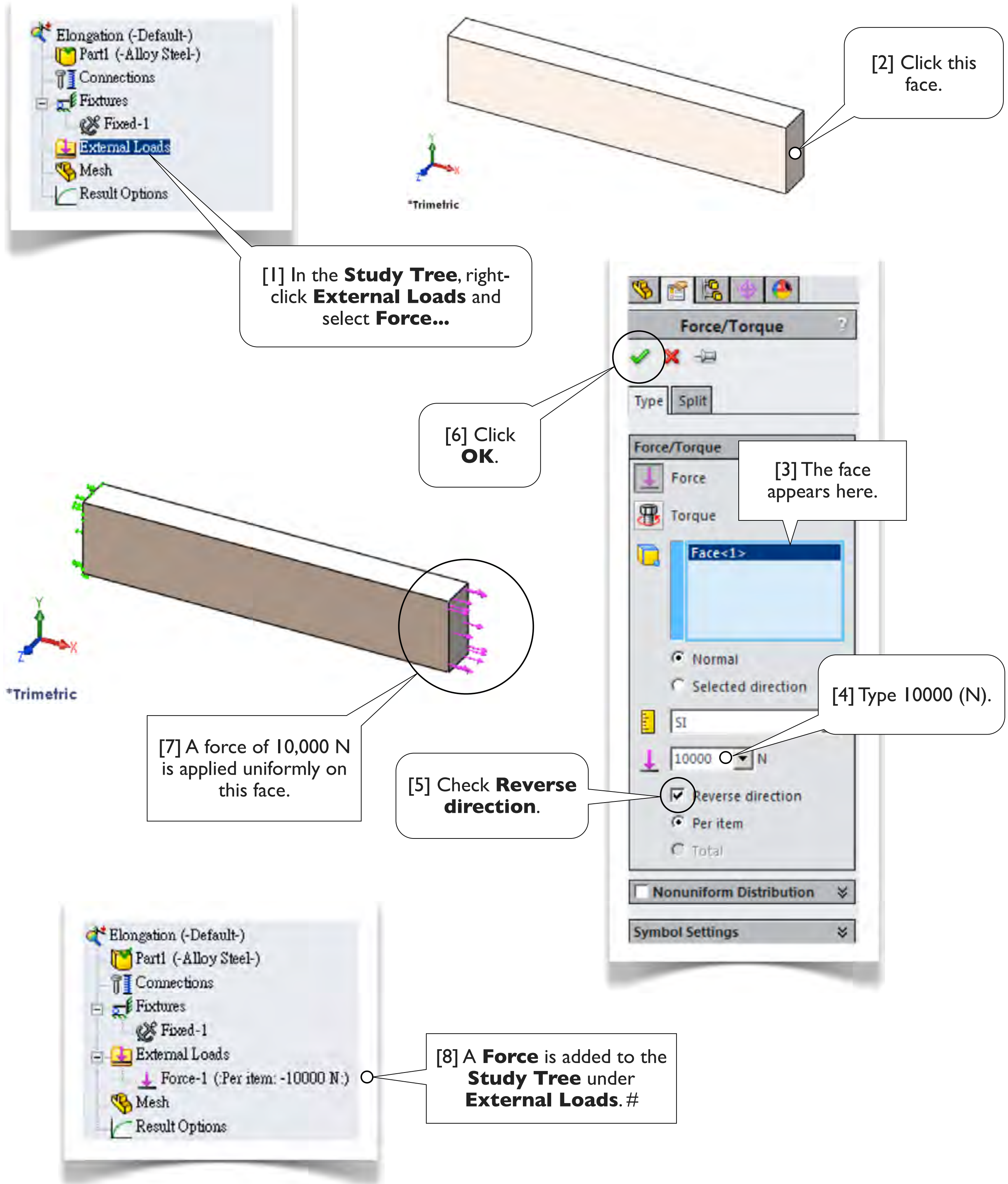
1.1-8 Apply Material



1.1-9 Apply Support

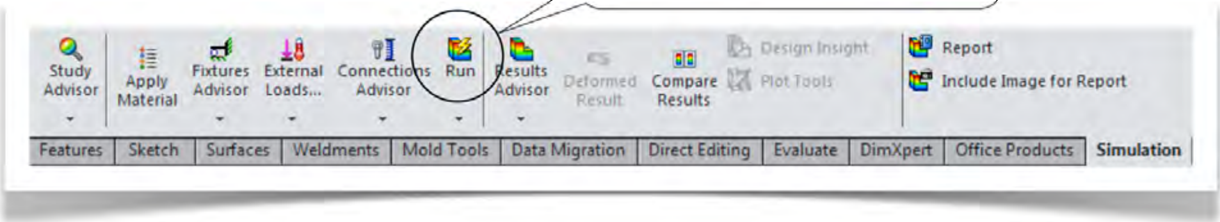


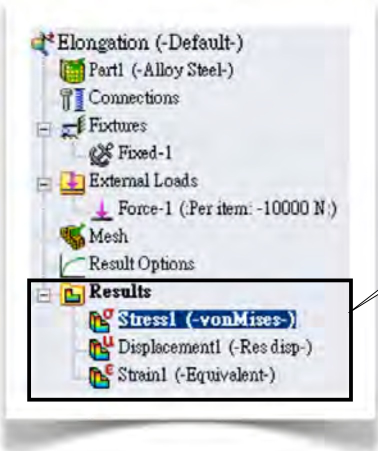
1.1-10 Apply Load



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[7], page 15), 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**.

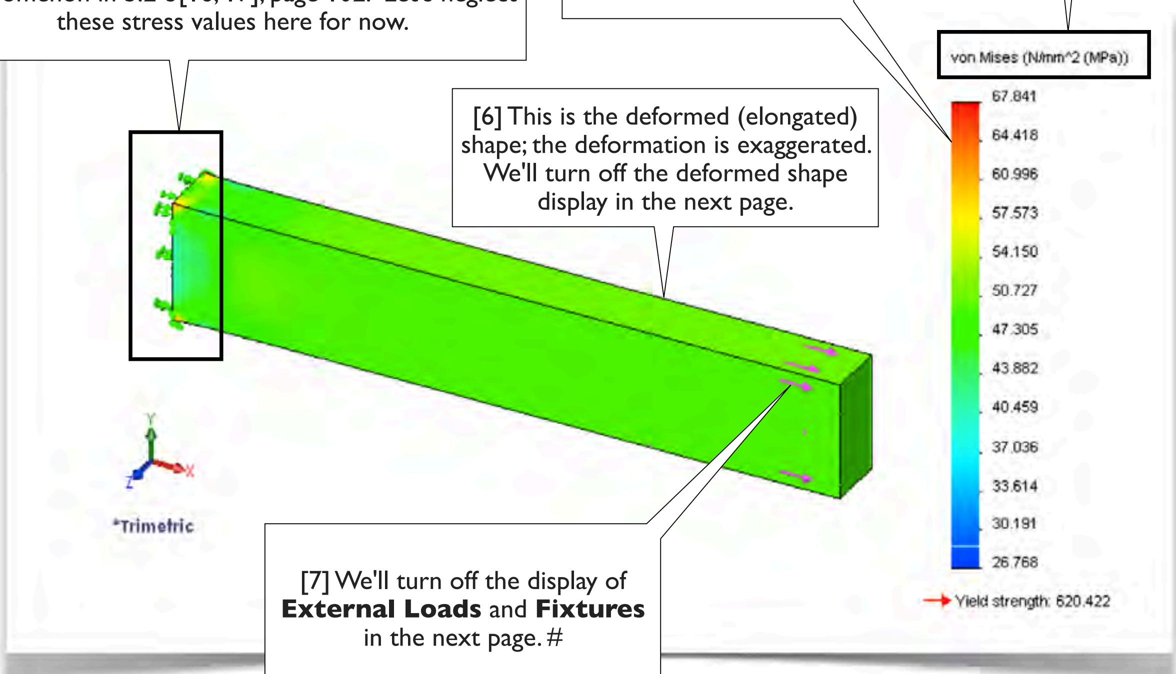
[4] Even you follow the exactly steps in this book, your results may deviate from the results in this book. It is normal, as long as the deviation is not too large.

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

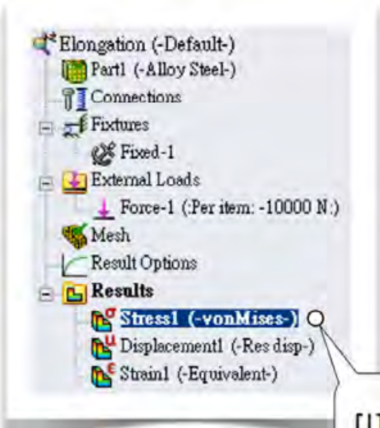
[5] The stress is uniform over the entire body except the area near the fixed end, where the stresses are complicated and we'll explain this phenomenon in 5.2-3[16, 17], page 102. Let's neglect these stress values here for now.

[6] This is the deformed (elongated) shape; the deformation is exaggerated. We'll turn off the deformed shape display in the next page.

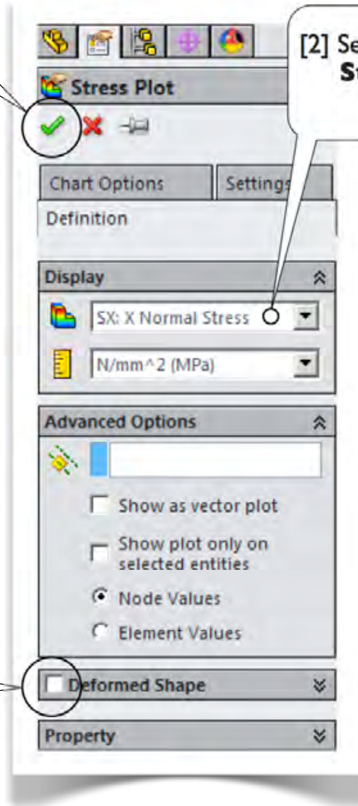
[7] We'll turn off the display of **External Loads** and **Fixtures** in the next page. #



1.1-12 View the Normal Stress σ_x



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

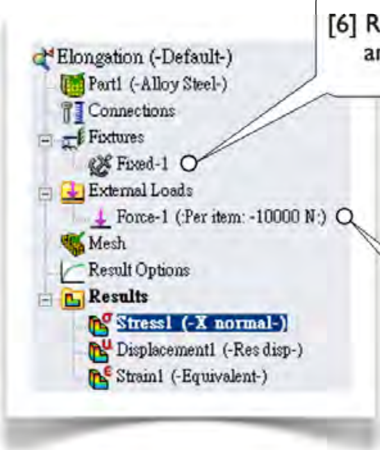


[2] Select **SX: X Normal Stress** (i.e., σ_x) for **Component**.

[3] Uncheck **Deformed Shape**. The undeformed shape will be displayed.

[4] Click **OK**.

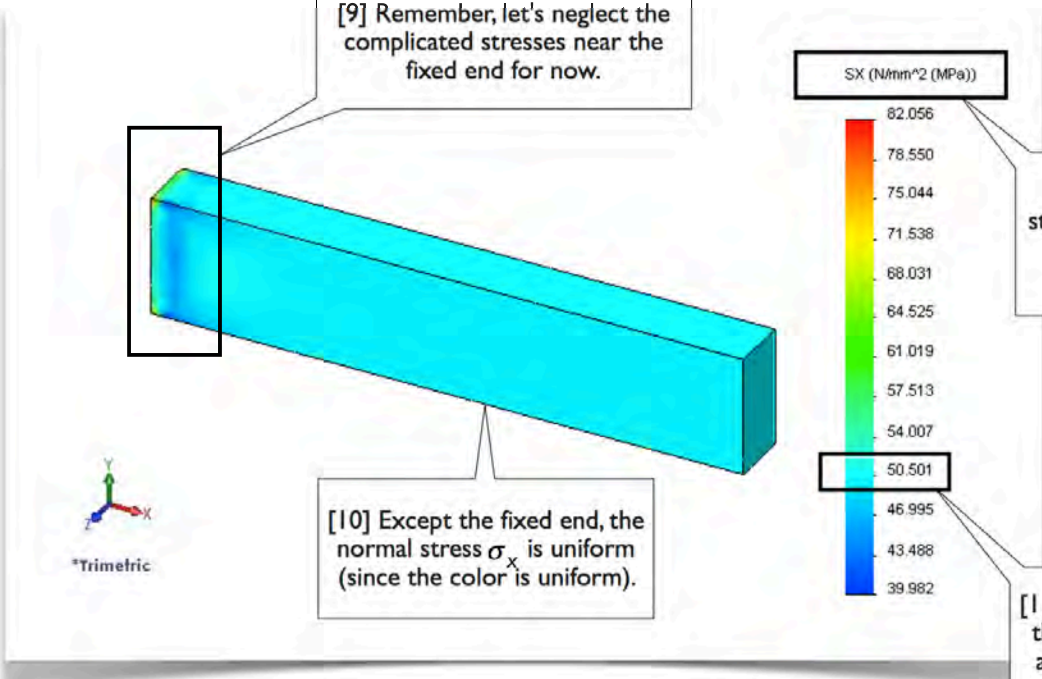
[5] **Deformed Shape** can be toggled on/off by clicking this button in the **Simulation Toolbar**.



[6] Right-click **Fixed-1** and select **Hide**.

[7] Right-click **Force-1** and select **Hide**.

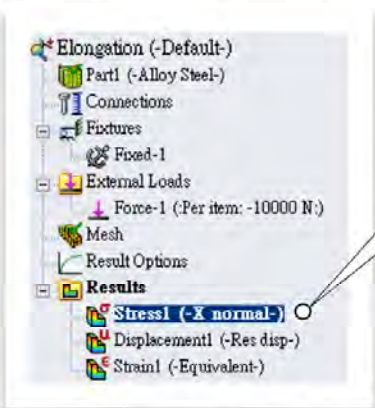
[9] Remember, let's neglect the complicated stresses near the fixed end for now.



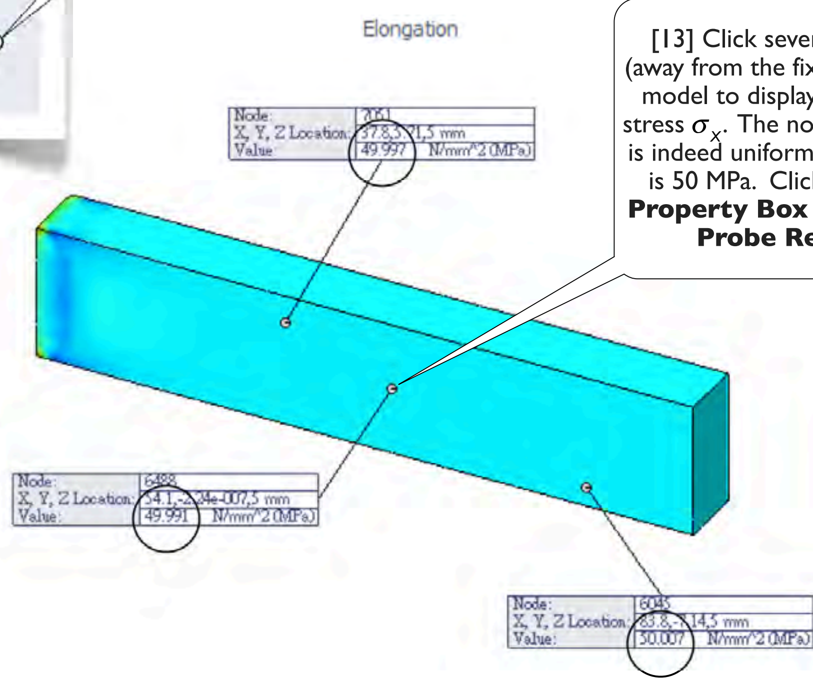
[8] Now, the normal stress in the *X*-direction (i.e., σ_x) is reported. The unit is MPa.

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

[11] By the legend colors, the normal stress σ_x is approximately 50 MPa. Let's use **Probe** to obtain more precise values.

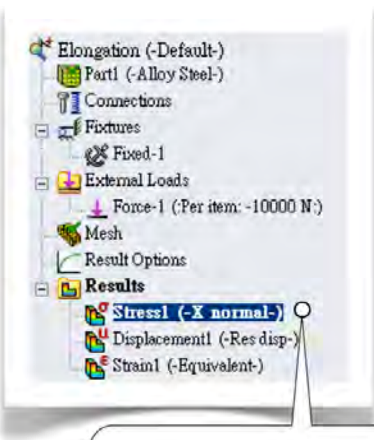


[12] Right-click **Stress1** and select **Probe**.

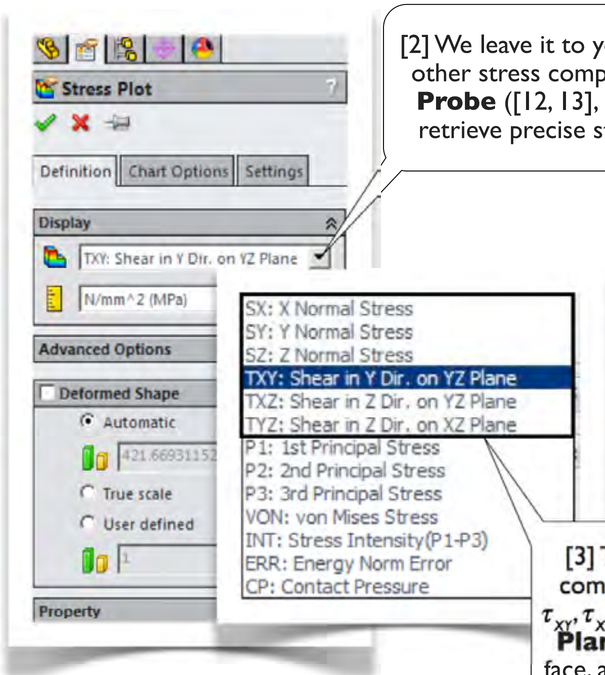


[13] Click several locations (away from the fixed end) in the model to display the normal stress σ_x. The normal stress σ_x is indeed uniform and the value is 50 MPa. Click **OK** in the **Property Box** to dismiss the **Probe Result**. #

1.1-13 View Other Stress Components



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



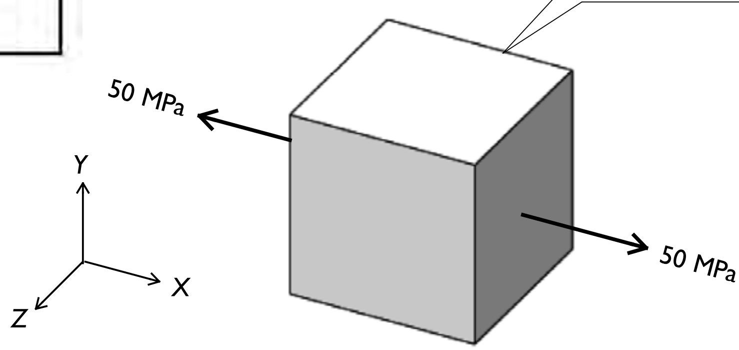
[2] We leave it to you to explore other stress components. Use **Probe** ([12, 13], last page) to retrieve precise stress values.

[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

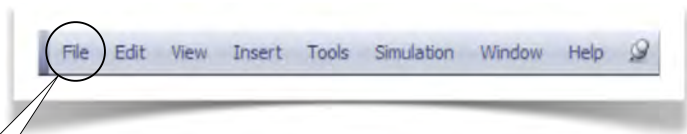
[4] Write down each stress component value. The results should be like this. In this example, all stress components are essentially zeros, except σ_X (which is 50 MPa).

[5] The stress state of any point in this 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**

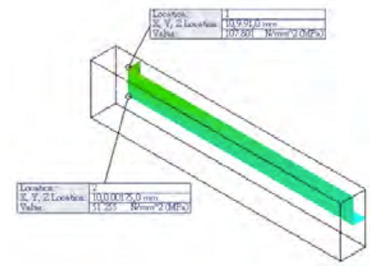
[1] Click **Save** and save the document with the name **Cantilever**. Two files are created in your working folder: **Cantilever.SLDPRT** and **Cantilever-Elongation.CWR**; the former is the main project file, while the later stores the result data generated by a finite element solver. Other files,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



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

Section 1.2

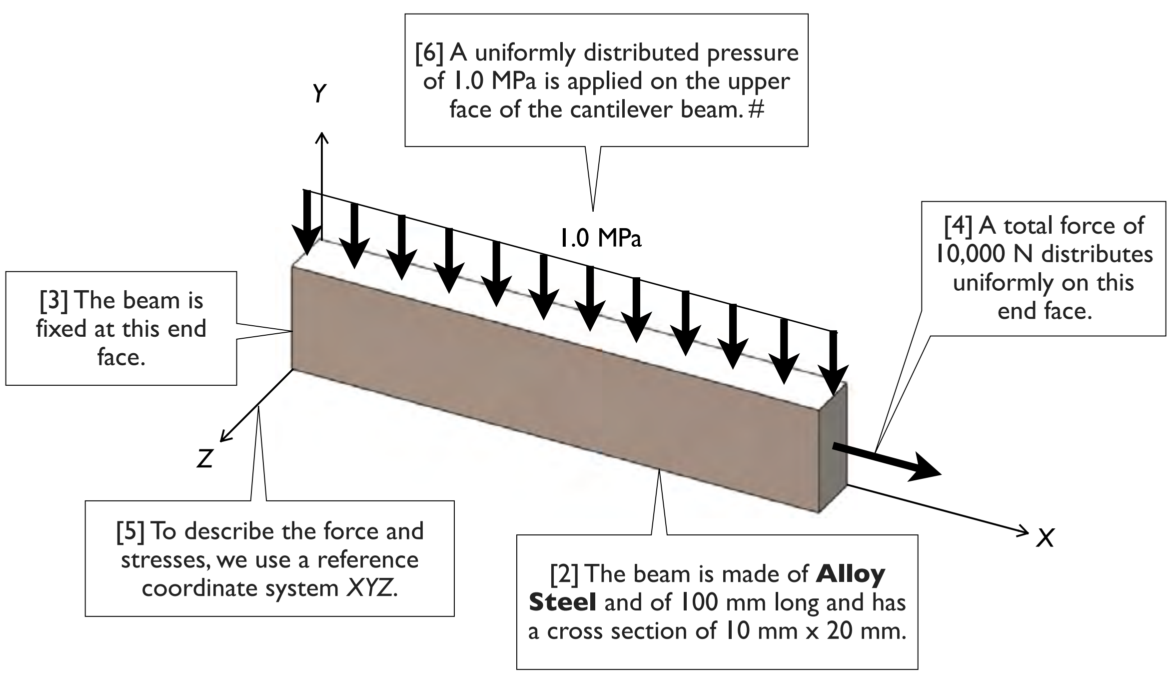
More on Stress Components



1.2-1 Introduction

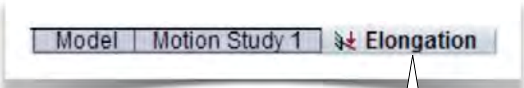
[1] In the last section, the stress field is uniform over the body and the only non-zero stress component is σ_x. In this section, we'll use the same model in the last section [2-5] but add a uniformly distributed transversal pressure of 1.0 MPa on the upper face of the beam [6]. In this case, the resulting stress will not be uniform, and non-zero shear stress components exist in the beam.

This section also demonstrates a way to retrieve results at specific locations in a body, namely the **Section Clipping** method.

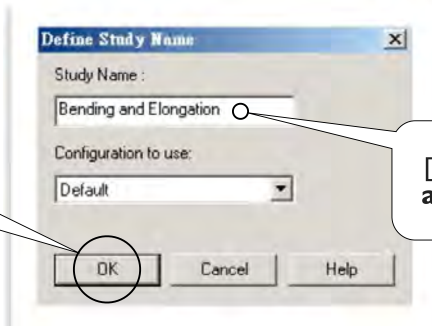


1.2-2 Start Up

[1] Launch **SolidWorks** and open the file **Cantilever** which was saved in Section 1.1. Make sure **SolidWorks Simulation** is loaded (1.1-5, page 13).

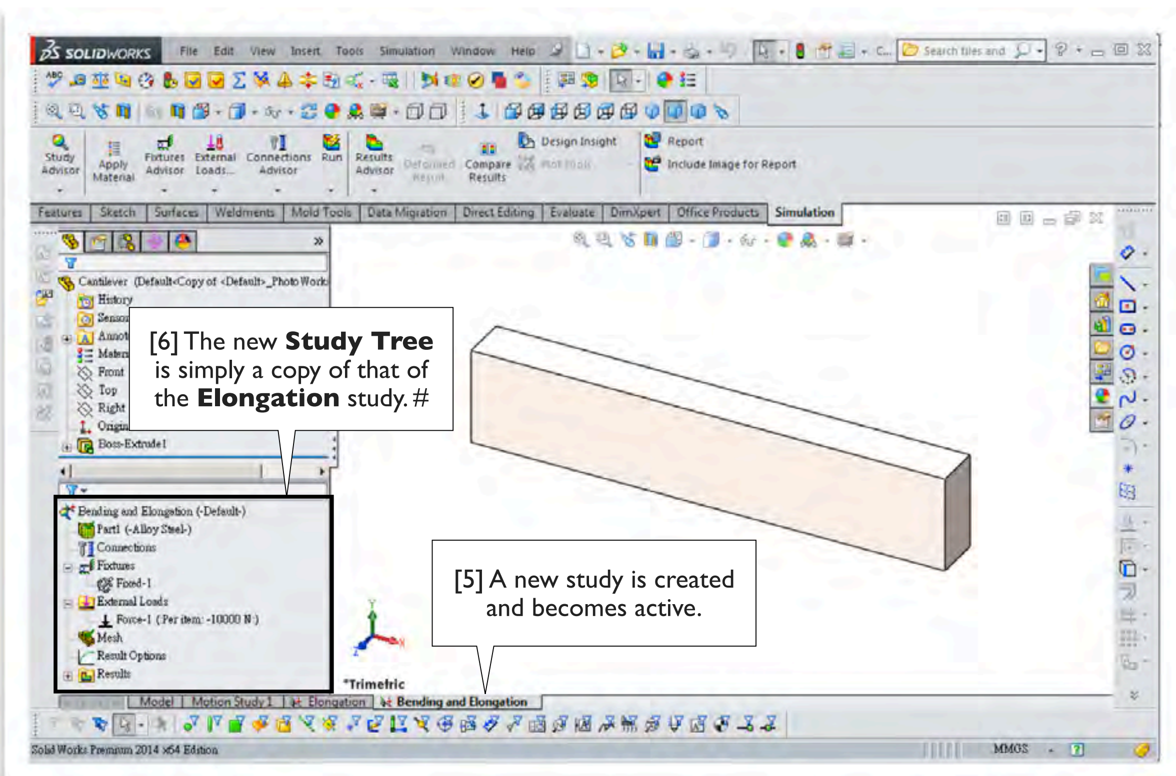


[2] Right-click **Elongation** tab and select **Duplicate** from the **Context Menu**.



[3] Type **Bending and Elongation**.

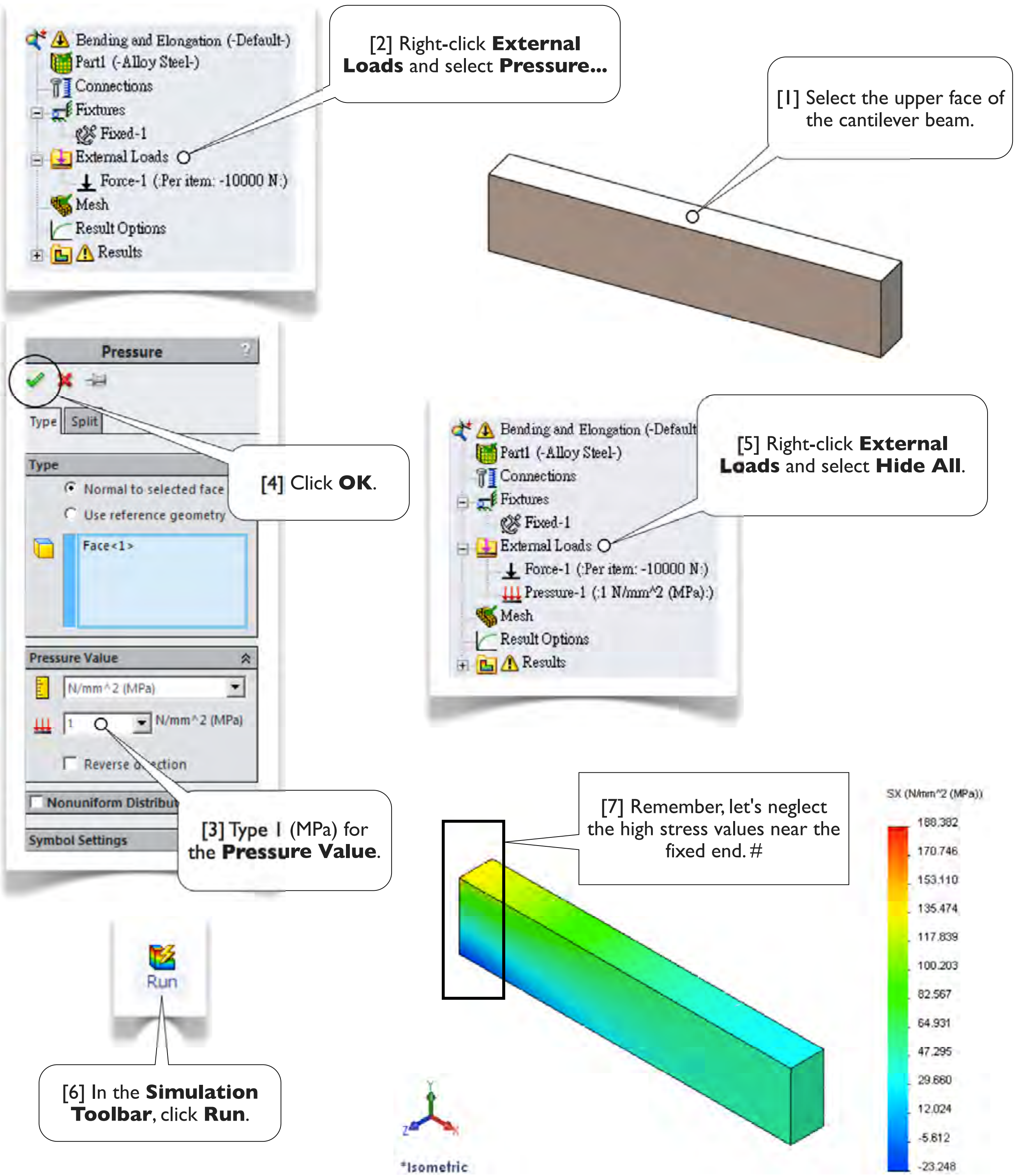
[4] Click **OK**.



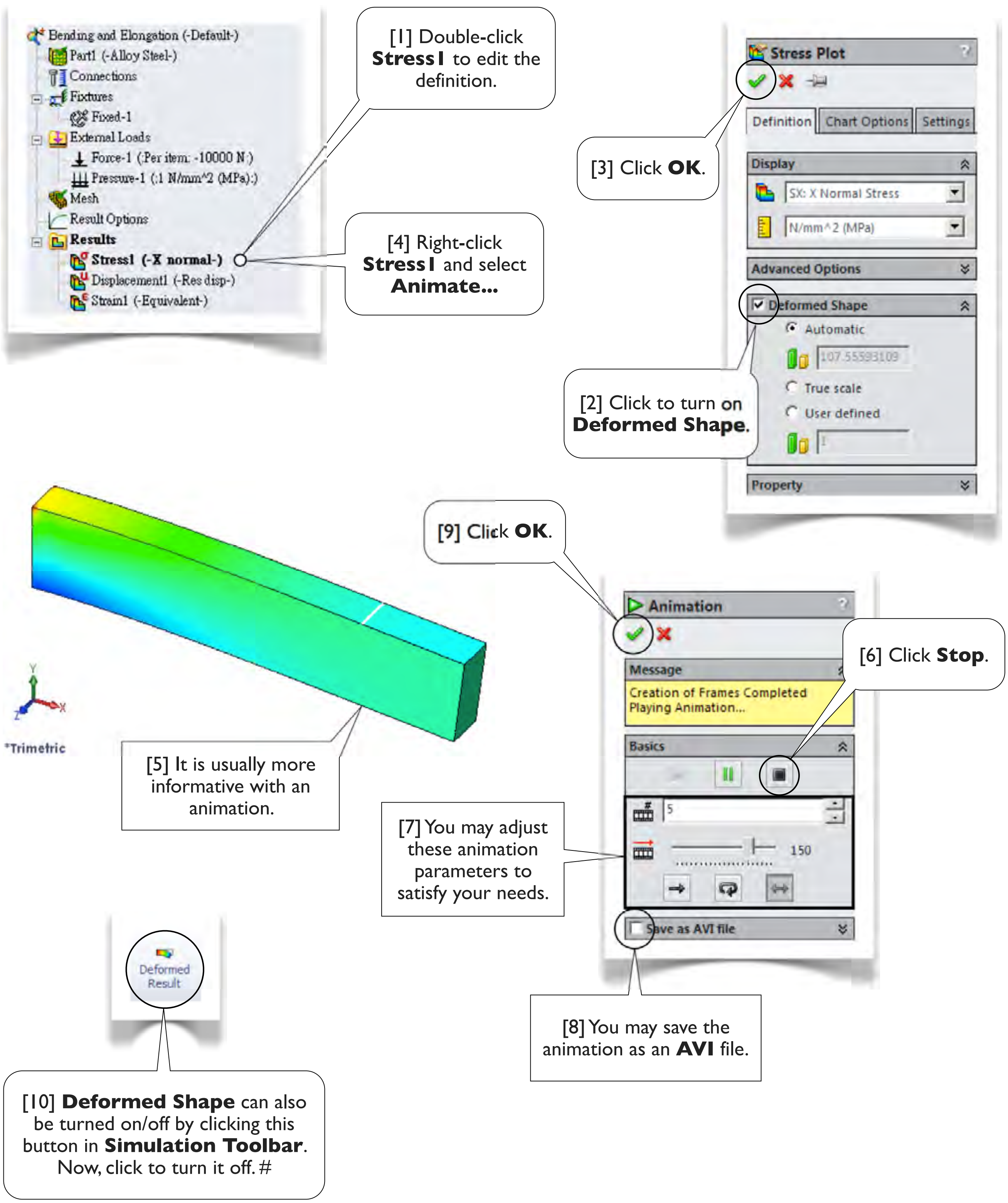
[5] A new study is created and becomes active.

[6] The new **Study Tree** is simply a copy of that of the **Elongation** study. #

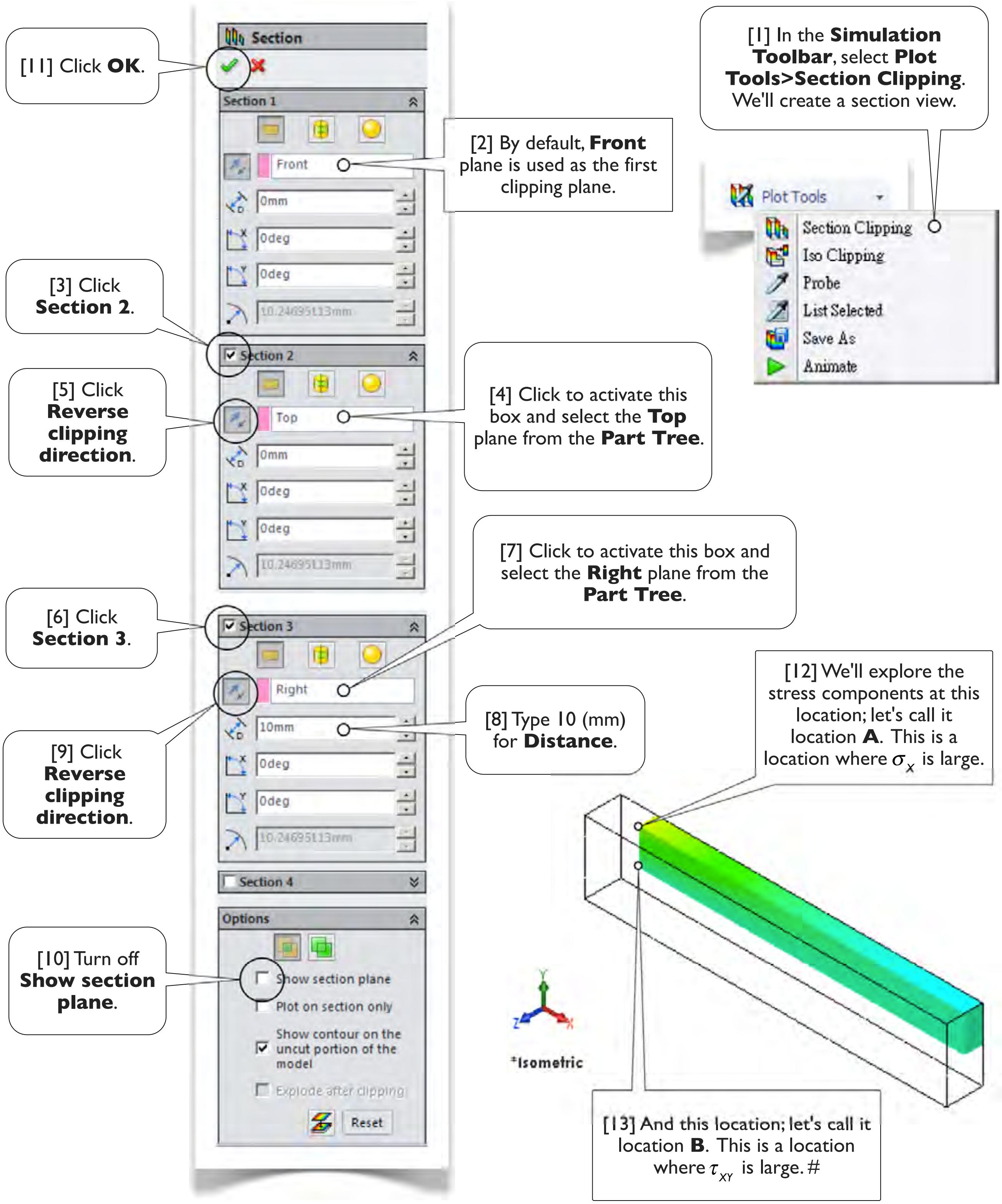
1.2-3 Add Transversal Load



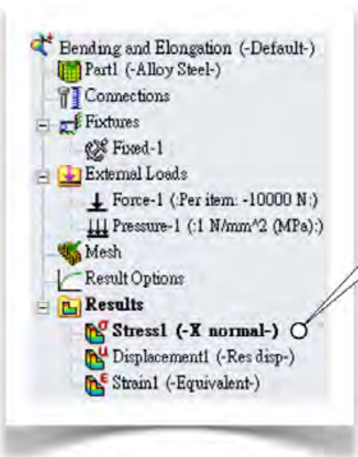
1.2-4 Animate the Deformation



1.2-5 Create Section View

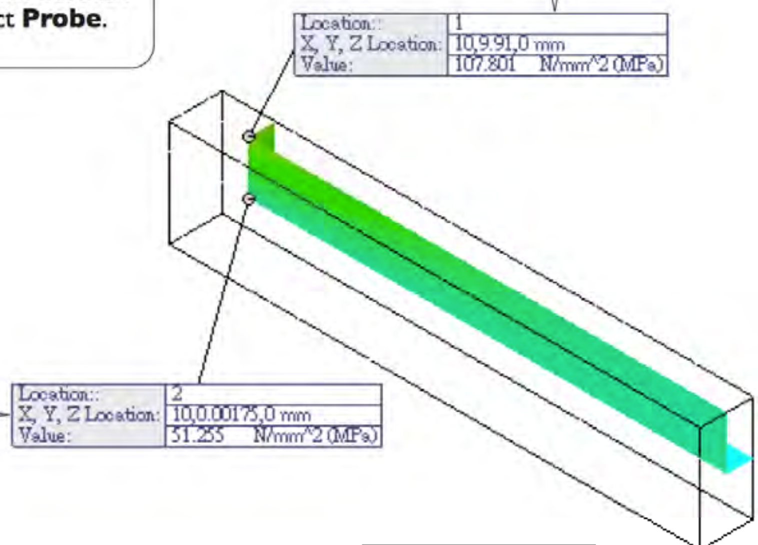


1.2-6 Stress Components at the Locations **A** and **B**



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

[2] Click location **A** to display the stress σ_X. You may enlarge the model to locate the corner more accurately.

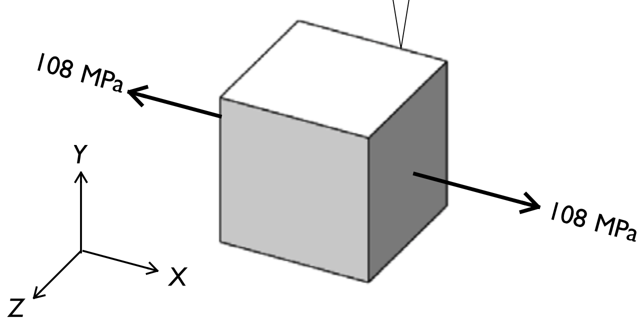


[3] Click location **B** to display the stress σ_X.

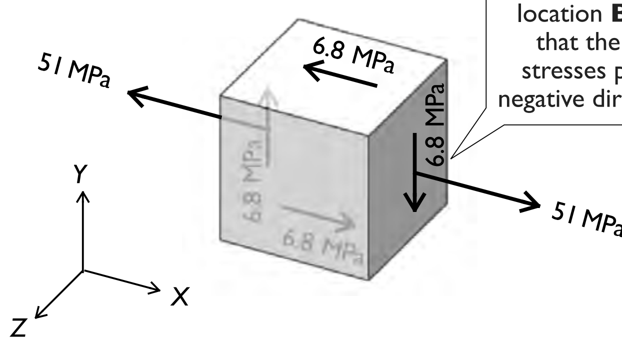
Stress Component	Location **A**	Location **B**
σ_X	107.801 Mpa	51.255 Mpa
σ_Y	0	0
σ_Z	0	0
τ_{XY}	0	-6.833 MPa
τ_{XZ}	0	0
τ_{YZ}	0	0

[4] Use **Probe** to explore other stress components and tabulate the data like this. Your stress values may not be exactly the same as here. Note that the shear stress τ_{XY} at location **B** is negative.

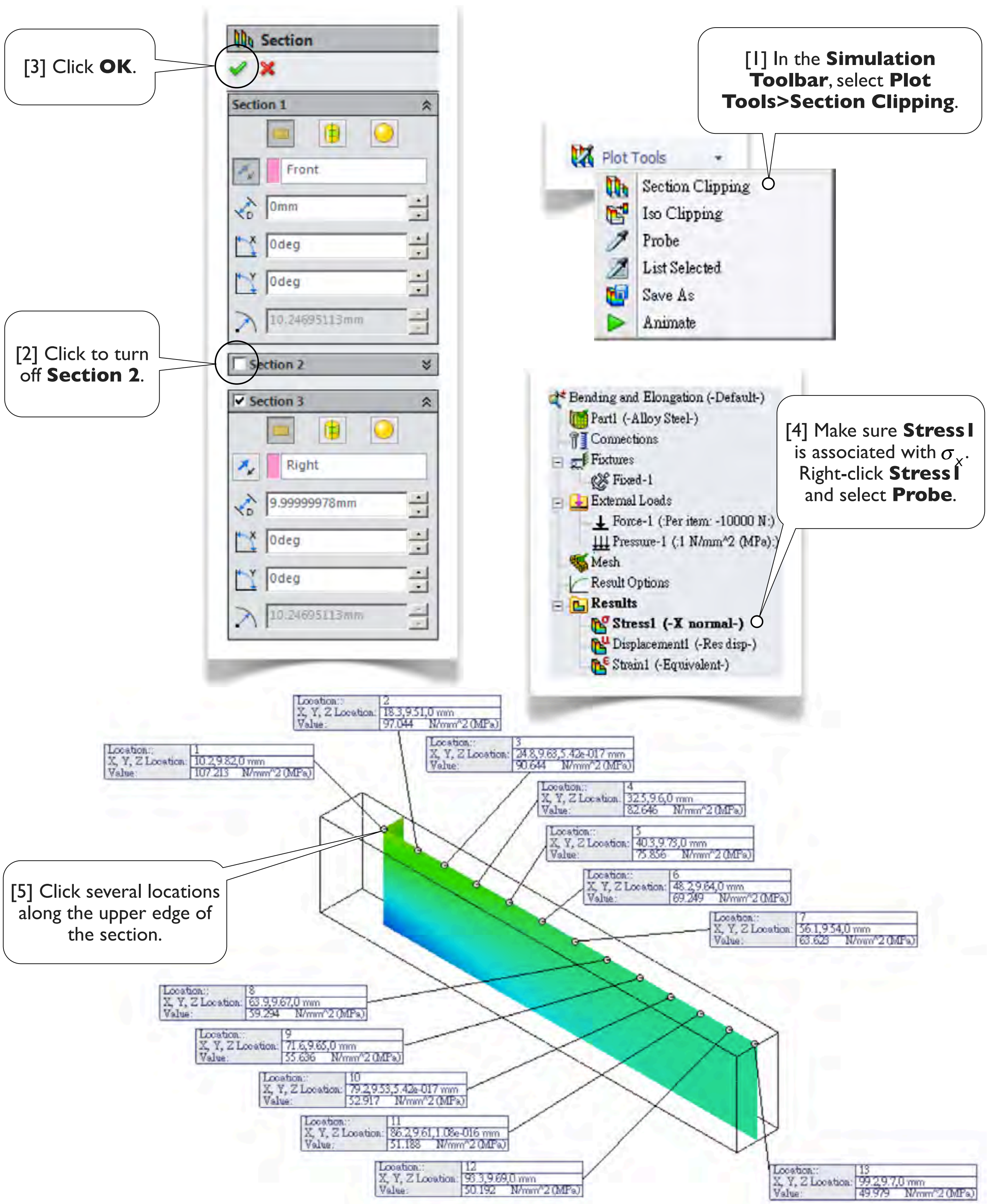
[5] The stress state at location **A**.

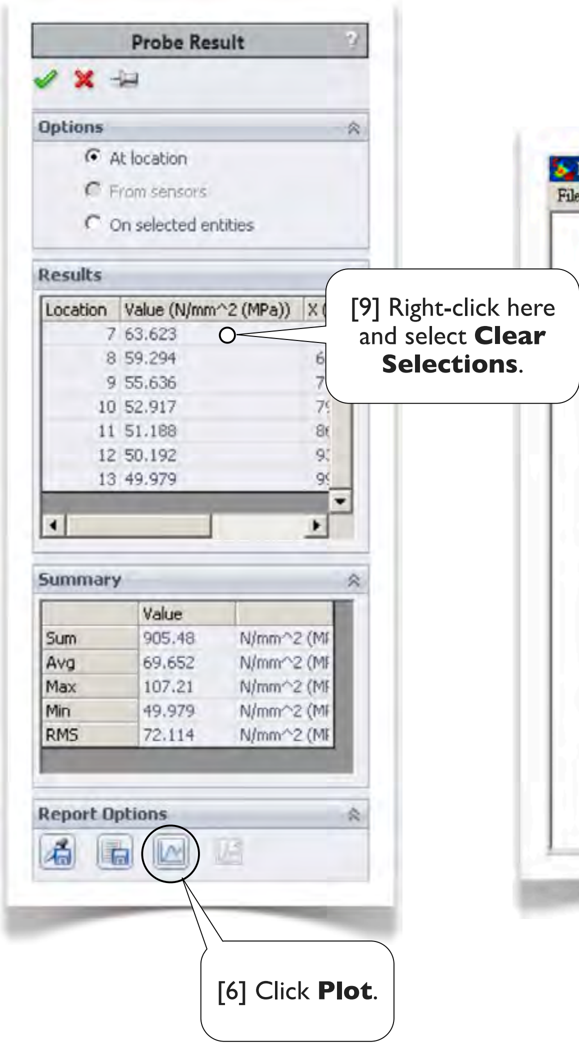


[6] The stress state at location **B**. Note that the shear stresses point to negative directions. #

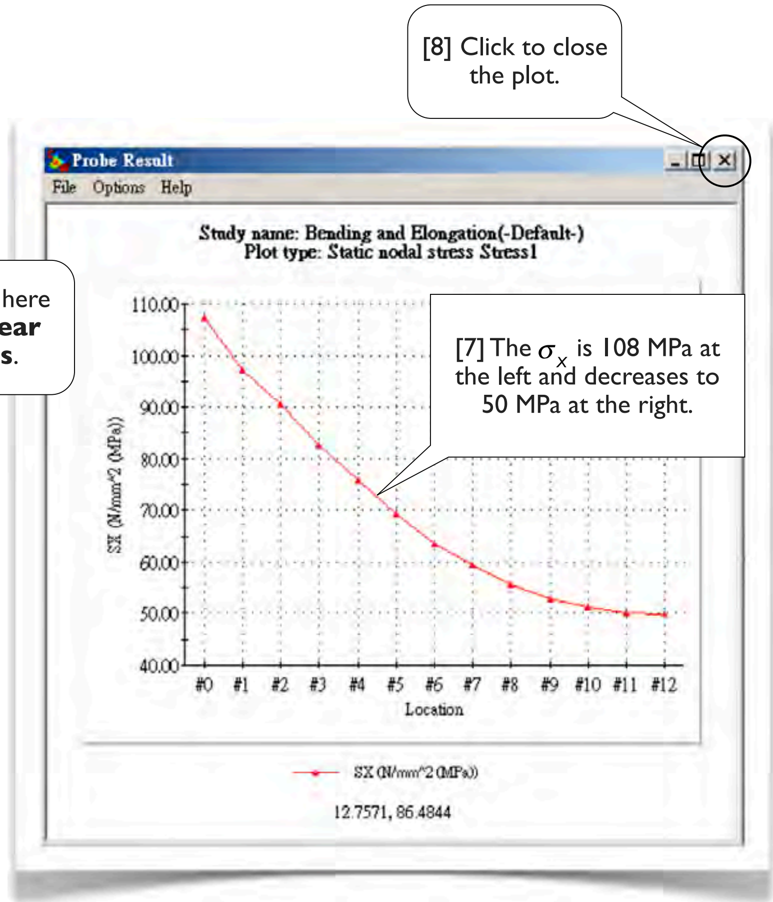


1.2-7 Distribution of σ_x Along Horizontal and Vertical Edges

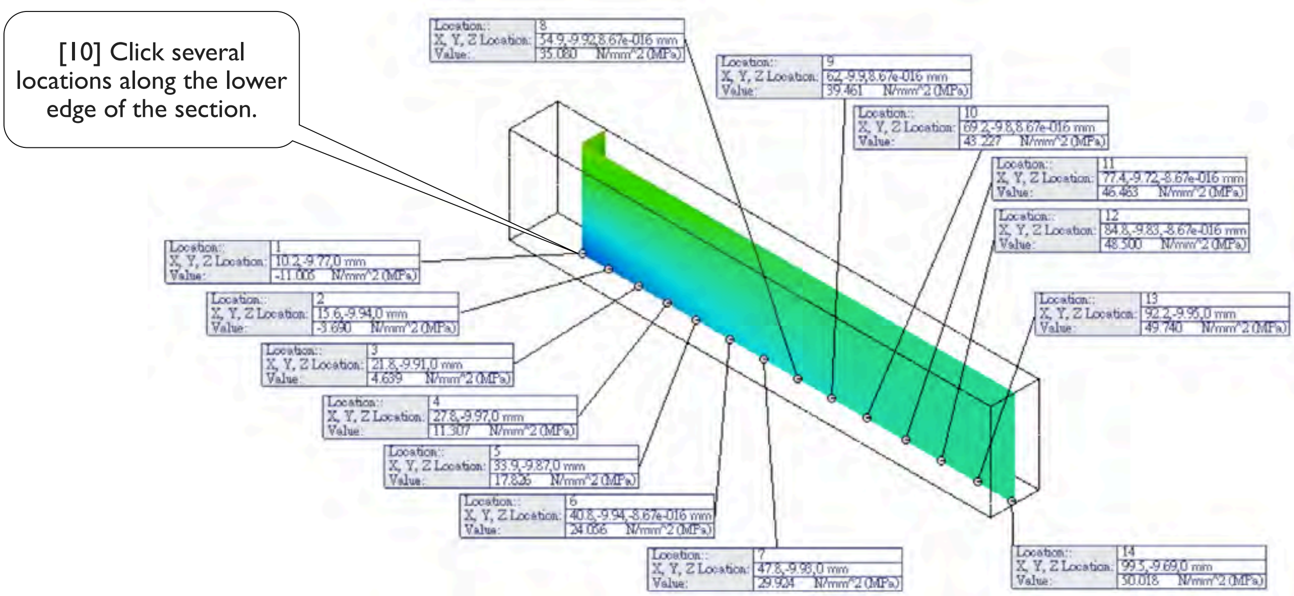




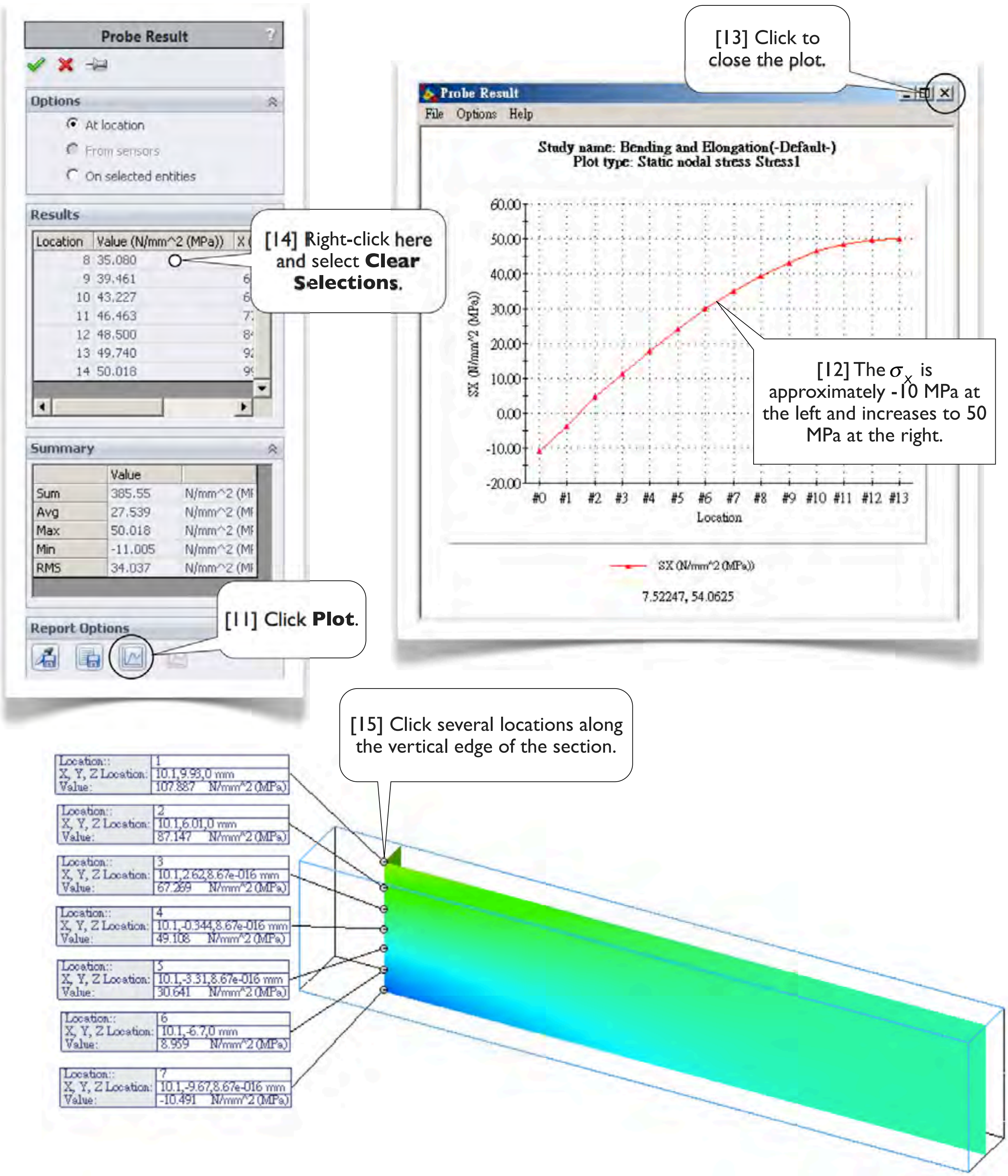
Probe Result
Options
At location
From sensors
On selected entities
Results
Location | Value (N/mm^2 (MPa))
7 63.623
8 59.294
9 55.636
10 52.917
11 51.188
12 50.192
13 49.979
Summary
Value
Sum 905.48 N/mm^2 (MPa)
Avg 69.652 N/mm^2 (MPa)
Max 107.21 N/mm^2 (MPa)
Min 49.979 N/mm^2 (MPa)
RMS 72.114 N/mm^2 (MPa)
Report Options
[9] Right-click here and select **Clear Selections**.
[6] Click **Plot**.

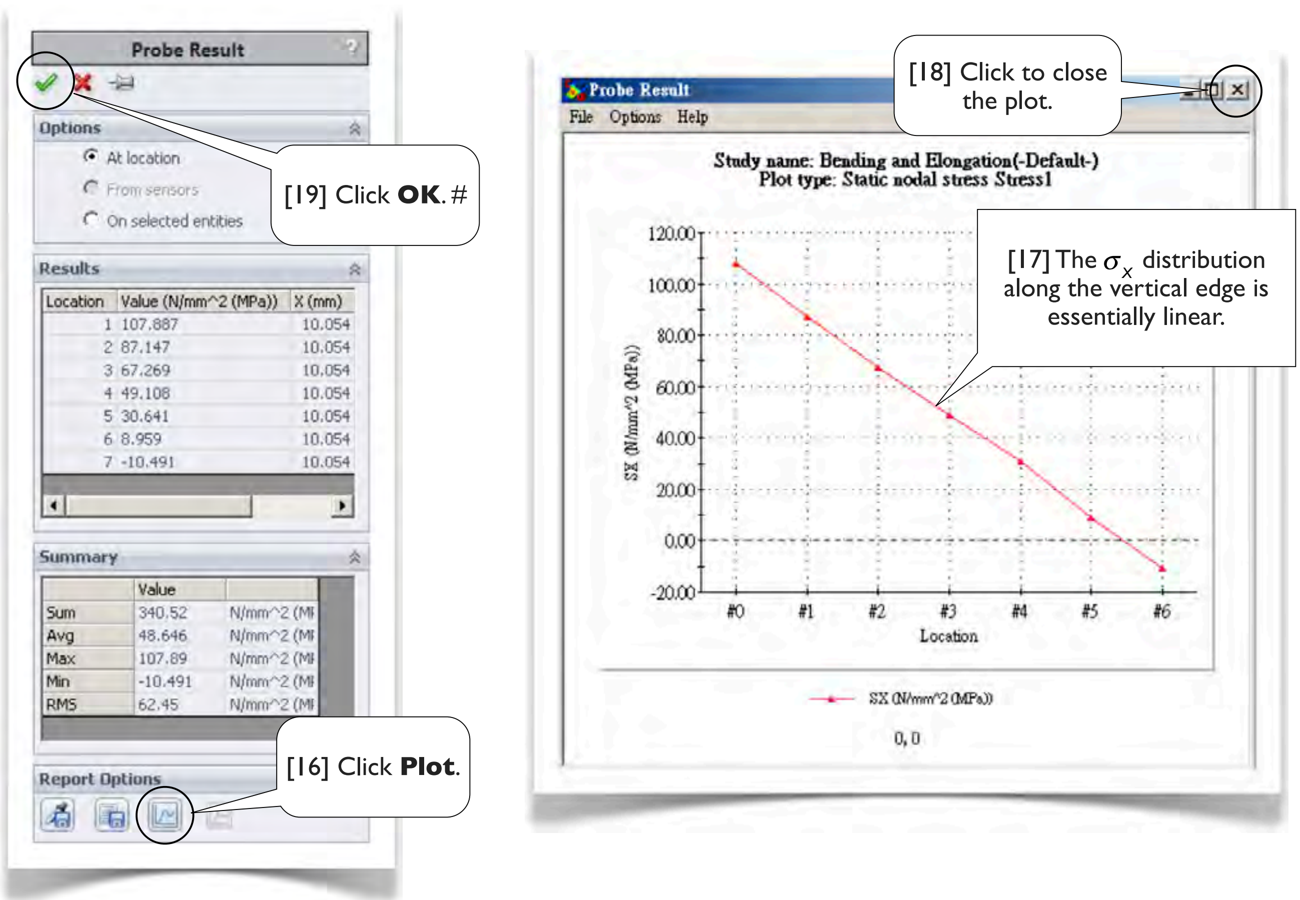


[8] Click to close the plot.
Probe Result
File Options Help
Study name: Bending and Elongation(-Default-)
Plot type: Static nodal stress Stress1
SX (N/mm^2 (MPa))
Location
#0 #1 #2 #3 #4 #5 #6 #7 #8 #9 #10 #11 #12
110.00 100.00 90.00 80.00 70.00 60.00 50.00 40.00
SX (N/mm^2 (MPa))
12.7571, 86.4844
[7] The σ_x is 108 MPa at the left and decreases to 50 MPa at the right.

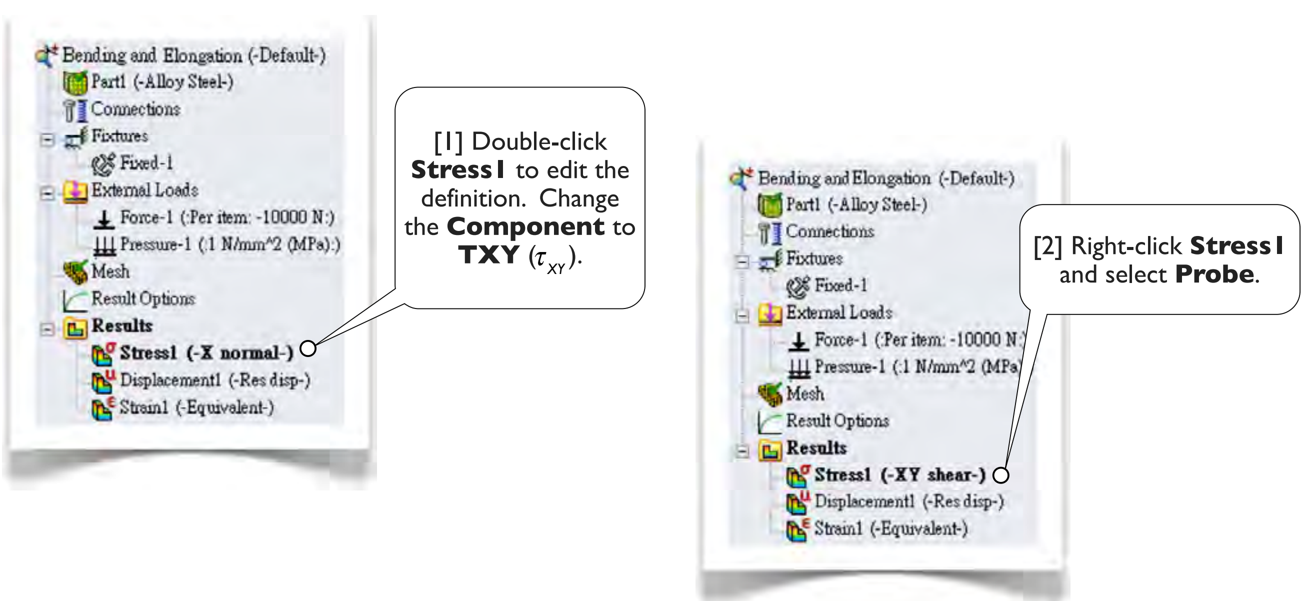


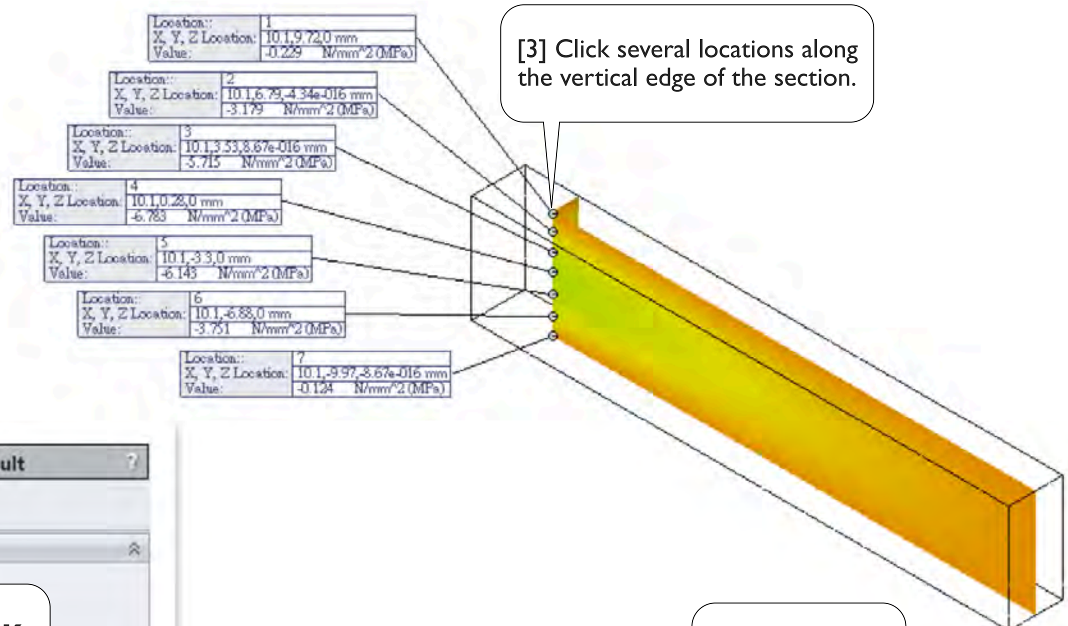
[10] Click several locations along the lower edge of the section.


[13] Click to close the plot.
[14] Right-click here and select **Clear Selections**.
[12] The σ_x is approximately -10 MPa at the left and increases to 50 MPa at the right.
[11] Click **Plot**.
[15] Click several locations along the vertical edge of the section.
Probe Result
Options
At location
From sensors
On selected entities
Results
Location
Value (N/mm^2 (MPa))
8 35.080
9 39.461
10 43.227
11 46.463
12 48.500
13 49.740
14 50.018
Summary
Value
Sum 385.55 N/mm^2
Avg 27.539 N/mm^2
Max 50.018 N/mm^2
Min -11.005 N/mm^2
RMS 34.037 N/mm^2
Report Options
File Options Help
Study name: Bending and Elongation(-Default-)
Plot type: Static nodal stress Stress1
SX (N/mm^2 (MPa))
Location
7.52247, 54.0625
Location: 1
X, Y, Z Location: 10.1,9.93,0 mm
Value: 107.887 N/mm^2 (MPa)
Location: 2
X, Y, Z Location: 10.1,6.01,0 mm
Value: 87.147 N/mm^2 (MPa)
Location: 3
X, Y, Z Location: 10.1,2.62,8.67e-016 mm
Value: 67.269 N/mm^2 (MPa)
Location: 4
X, Y, Z Location: 10.1,-0.344,8.67e-016 mm
Value: 49.108 N/mm^2 (MPa)
Location: 5
X, Y, Z Location: 10.1,-3.31,8.67e-016 mm
Value: 30.641 N/mm^2 (MPa)
Location: 6
X, Y, Z Location: 10.1,-6.7,0 mm
Value: 8.959 N/mm^2 (MPa)
Location: 7
X, Y, Z Location: 10.1,-9.67,8.67e-016 mm
Value: -10.491 N/mm^2 (MPa)

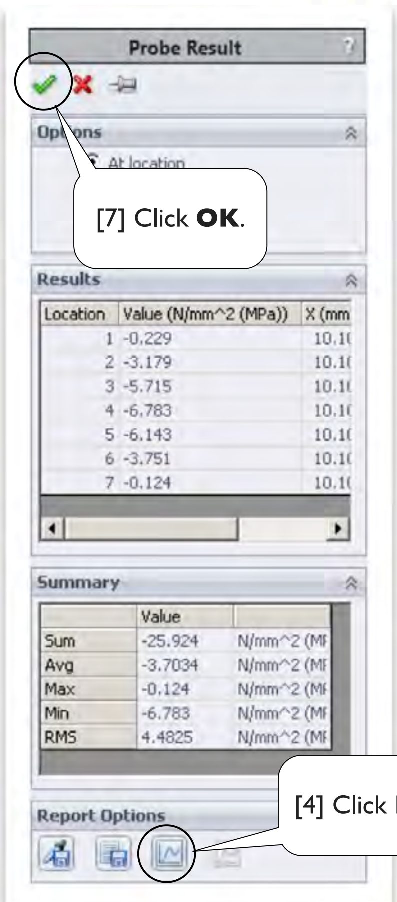


1.2-8 Distribution of τ_{XY} Along a Vertical Edge



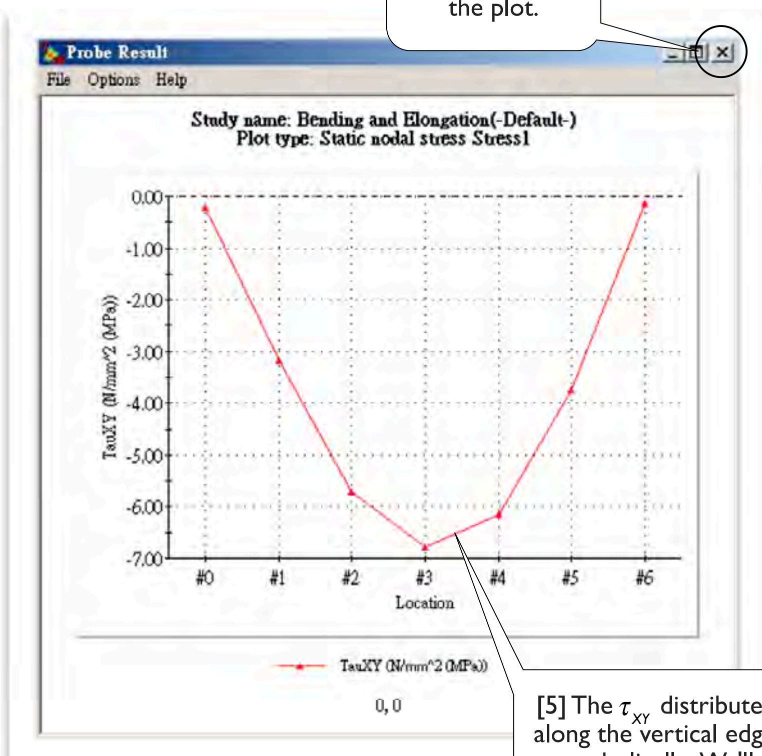


[3] Click several locations along the vertical edge of the section.



[7] Click **OK**.

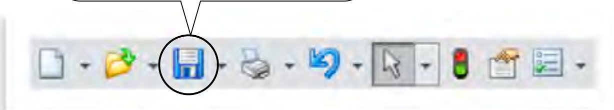
[4] Click **Plot**.



[6] Click to close the plot.

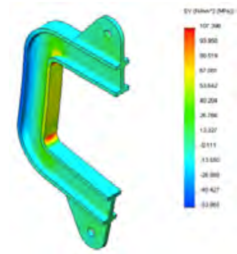
[5] The τ_{xy} distributes along the vertical edge parabolically. We'll discuss more on this curve in Section 8.1. A more accurate distribution is shown in 8.1-5[11], page 160.

[8] Save the document and exit **SolidWorks**. #



Section 1.3

Stresses in a C-Bar

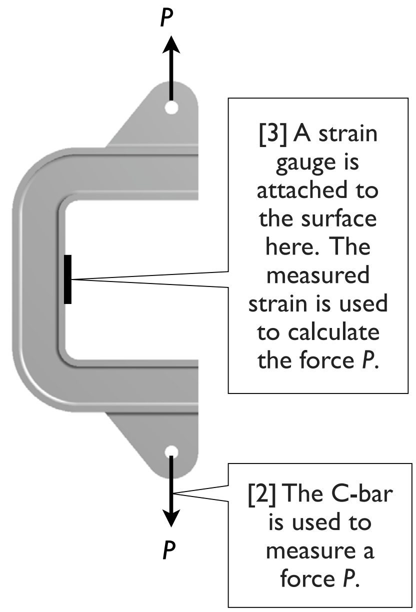


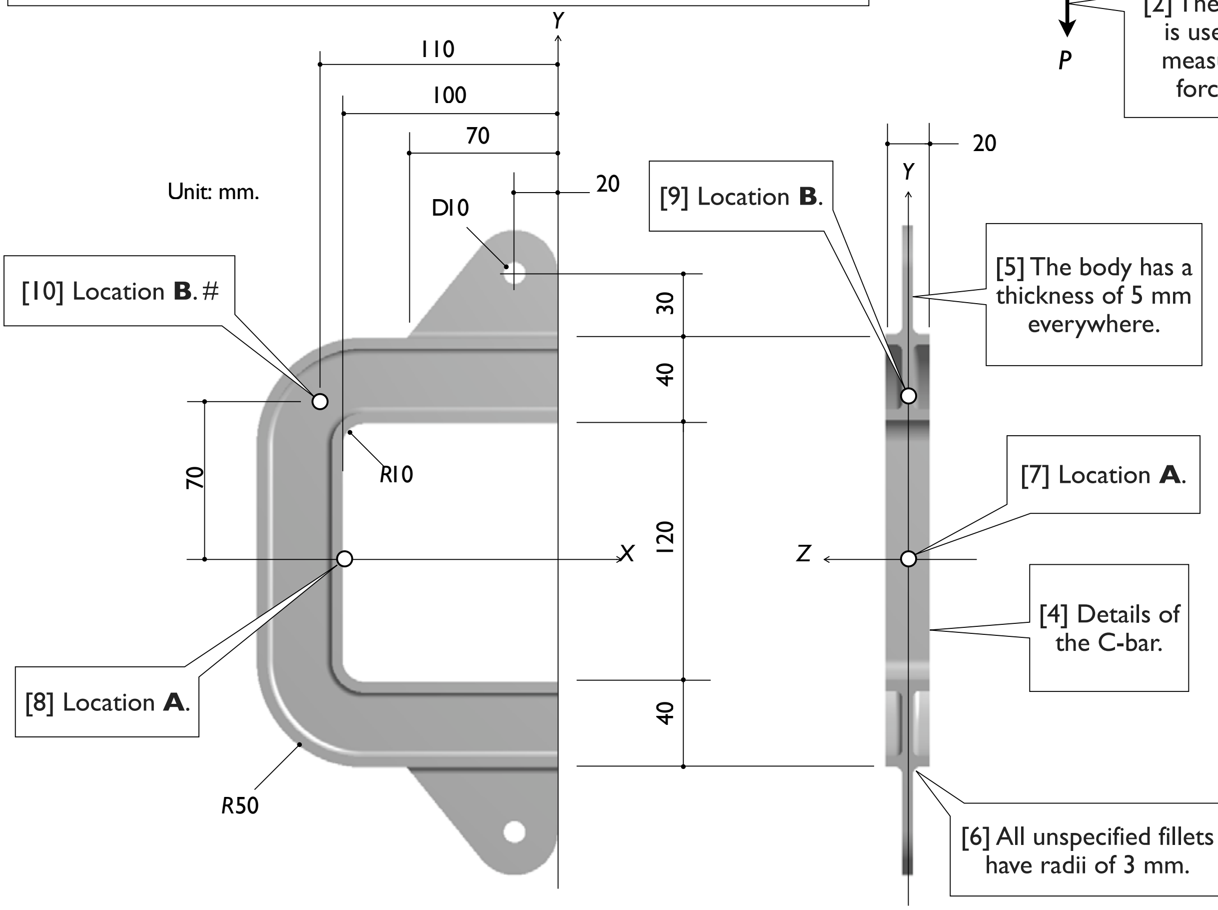
1.3-1 Introduction

[1] The C-shaped bar is made of an alloy steel and used as a dynamometer, a device to measure the magnitude of a force *P* [2]. A strain gauge is usually attached to the surface of a location as shown [3], and the measured strain is used to calculate the force *P*.

In this exercise, we will create a 3D solid model for the C-bar [4-6] and perform a static structural analysis under a force *P* = 2000 N. We'll examine the stress states at two locations, **A** [7, 8] and **B** [9, 10]. We examine location **A** since it is where the strain gauge situated and its normal stress σ_Y is high. Location **B** is arbitrarily chosen for its non-zero shear stress τ_{XY}.

This section also demonstrates a way to obtain stress results at specific location, namely using **Sensors**.

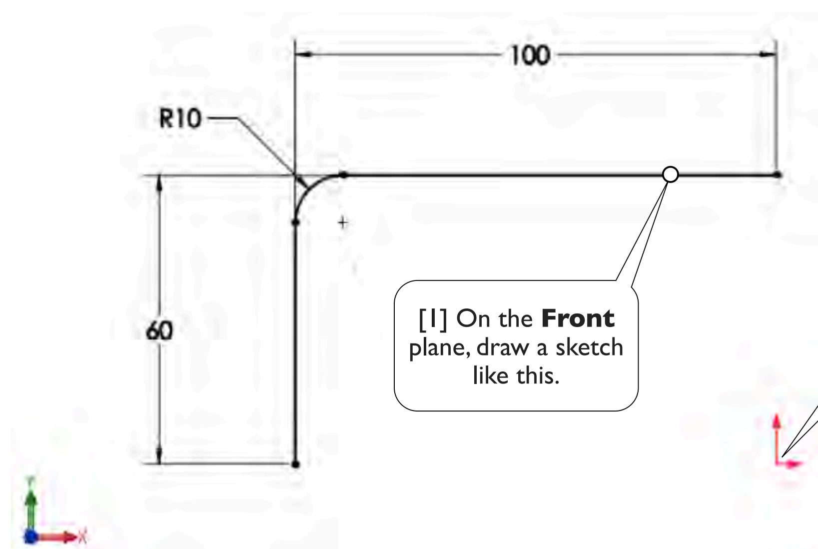




1.3-2 Start Up

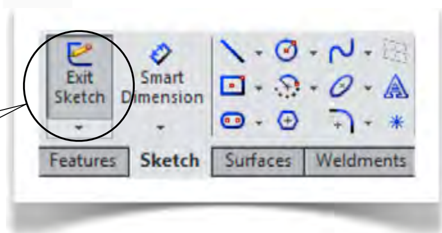
[1] Launch **SolidWorks** and create a new part. Set up **MMGS** unit system with zero decimal places for the length unit. #

1.3-3 Create a Sketch for the Sweeping Path



[1] On the **Front** plane, draw a sketch like this.

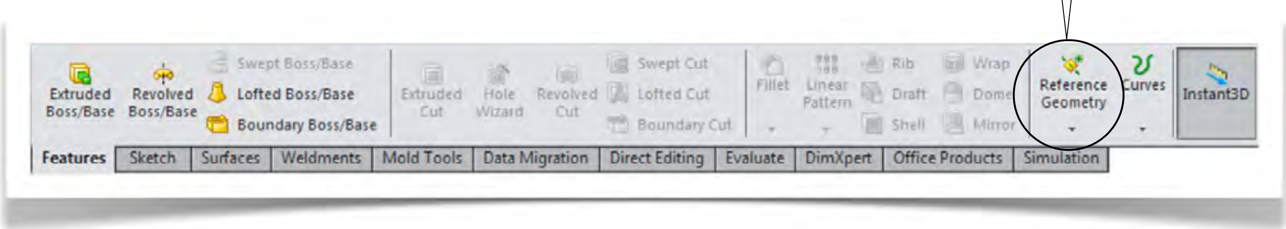
[2] The origin.

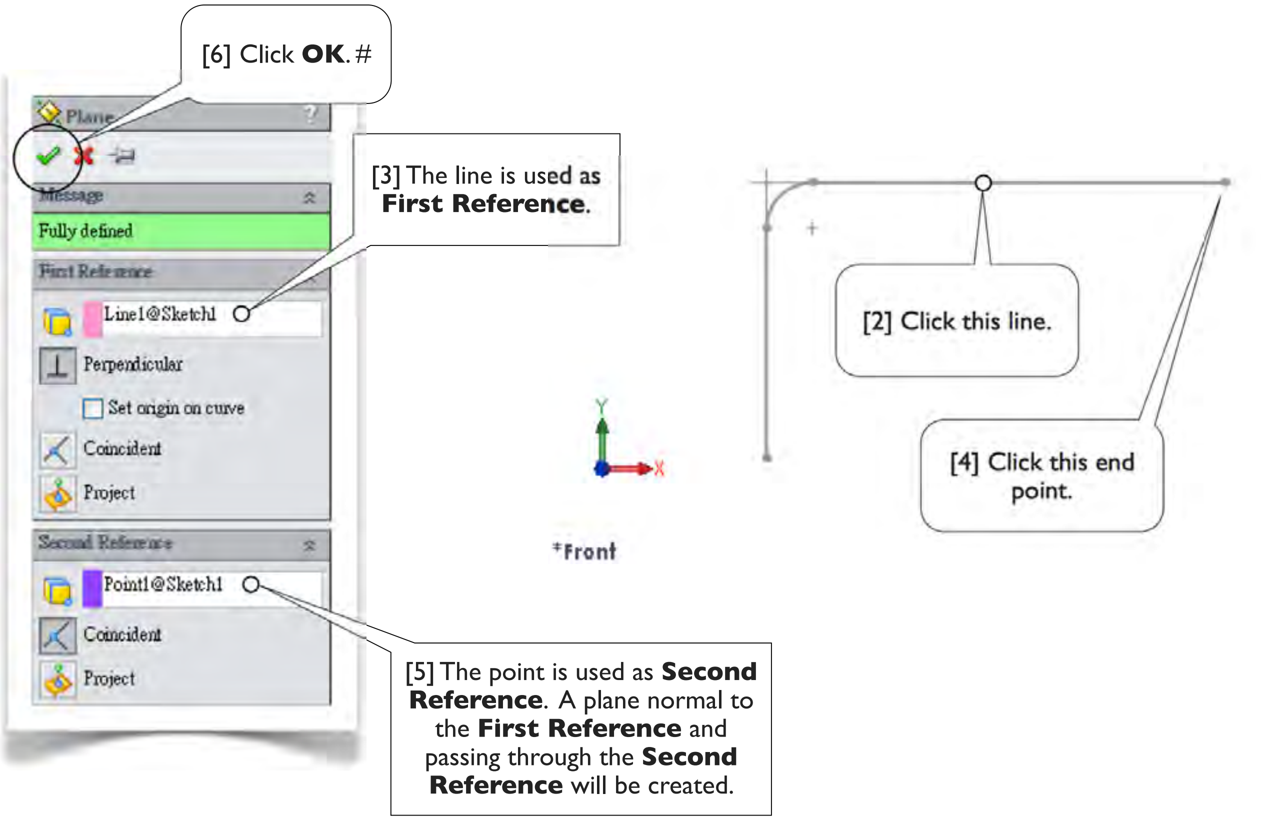


[3] Click **Exit Sketch**. #

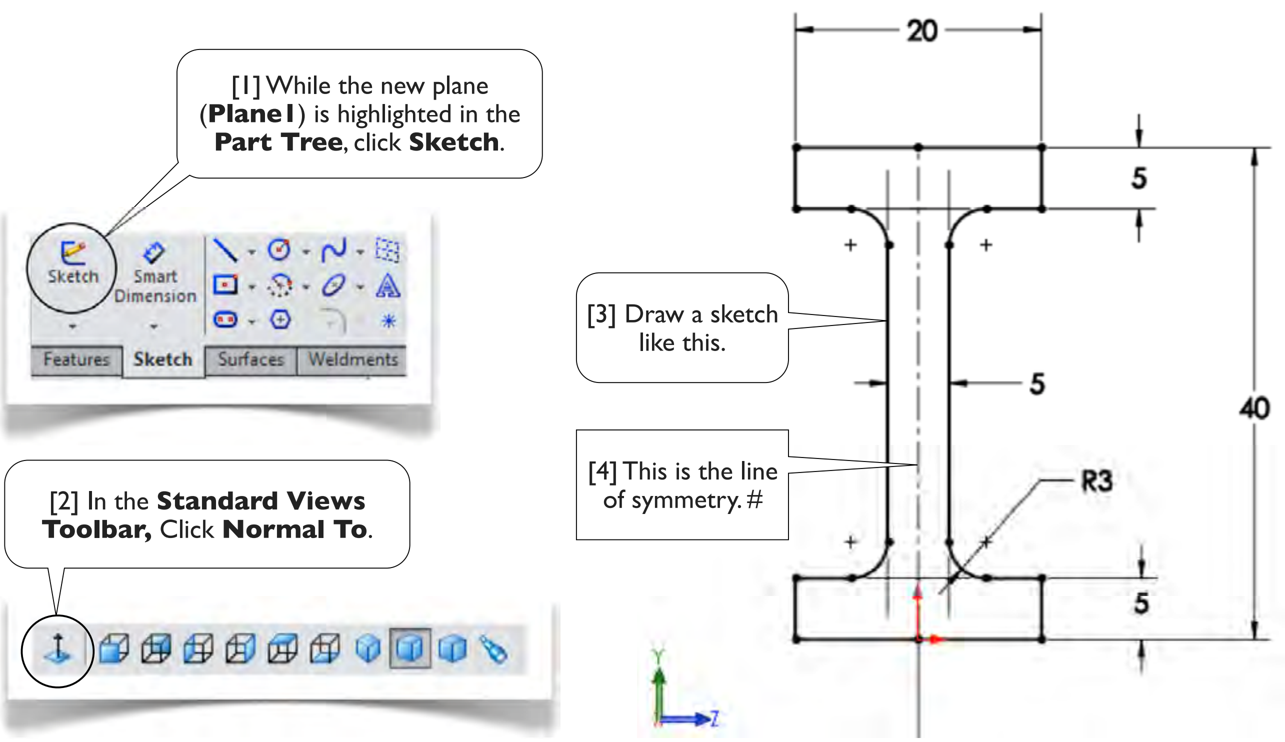
1.3-4 Create a New Plane

[1] In **Features Toolbar**, select **Reference Geometry>Plane**.

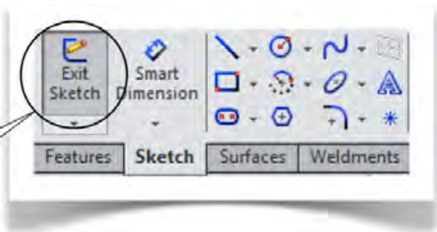




1.3-5 Create a Sketch for the Profile

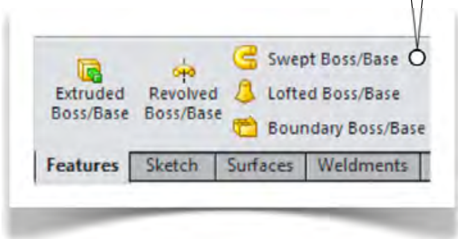


1.3-6 Create a Solid Body Using **Sweep**



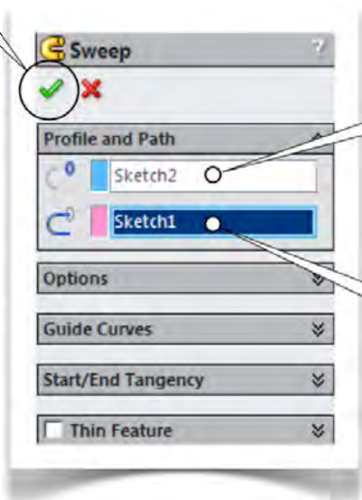
[1] Click **Exit Sketch**.

[2] In **Features Toolbar**, click **Swept Boss/Base**.



[5] Click **OK**.

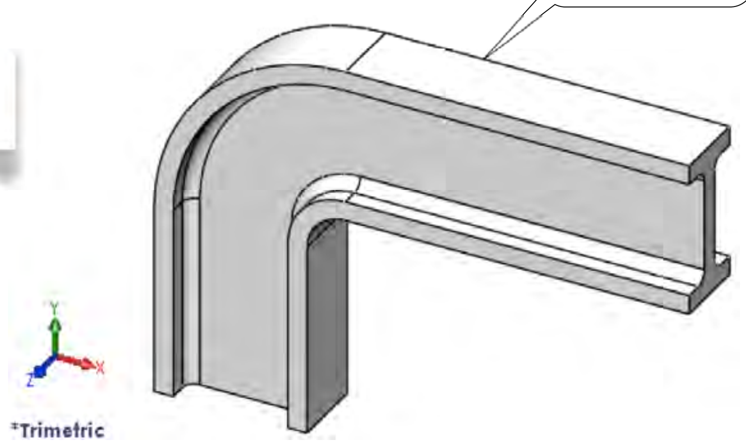
[3] The profile sketch (**Sketch2**) should be pre-selected. If not, select it from the **Part Tree**.



[4] Select the path sketch (**Sketch1**) from the **Part Tree**.

[7] A solid body is created. #

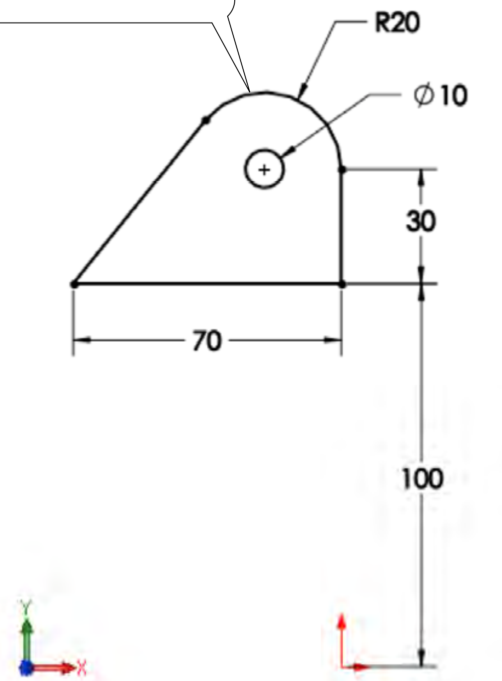
[6] In the **Standard Views Toolbar,** Click **Trimetric**. Also, in the **Head-Up Toolbar**, turn-off **View Planes**.



1.3-7 Create an Ear

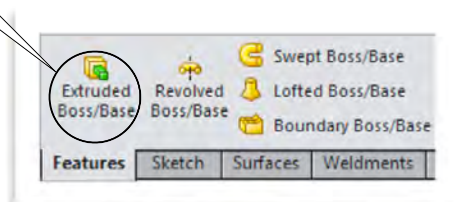
[1] In **Part Tree**, right-click **Sweep1** and select **Hide**.

[2] In the **Front** plane, draw a sketch like this. Click **Exit Sketch**.

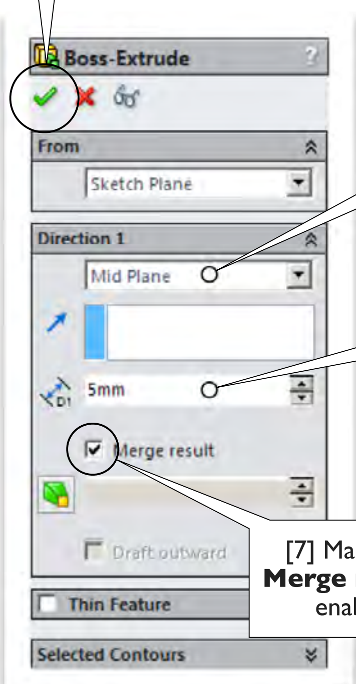


[3] Right-click **Sweep1** and select **Show**.

[4] In the **Part Tree**, click to highlight the newly created sketch (**Sketch3**) and, In **Features Toolbar**, click **Extruded Boss/Base**.



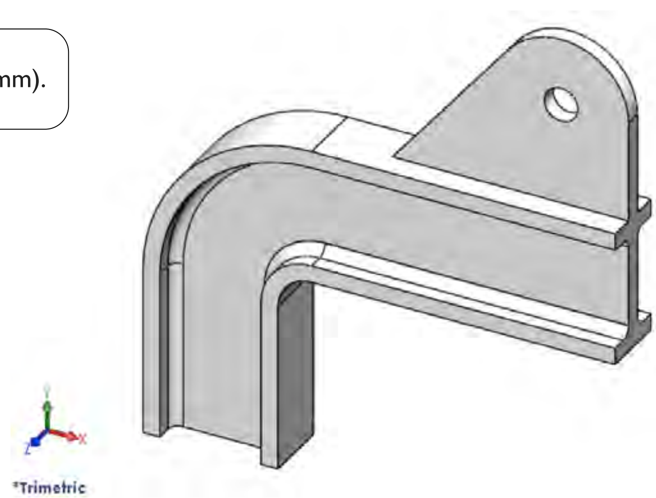
[8] Click **OK**. #



[5] Select **Mid Plane**.

[6] Type 5 (mm).

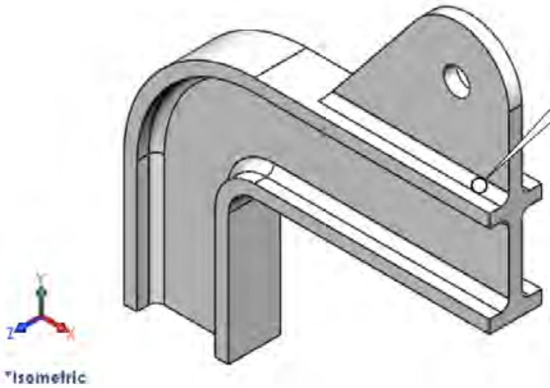
[7] Make sure **Merge result** is enabled.

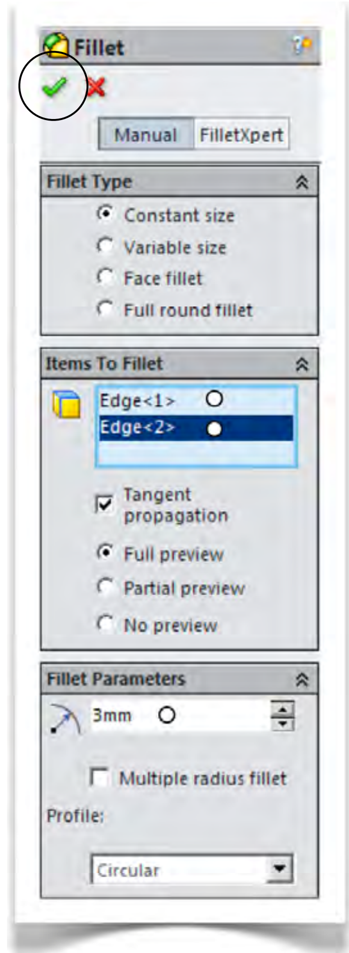


1.3-8 Create Fillets

[1] In **Features Toolbar**, click **Fillet**.

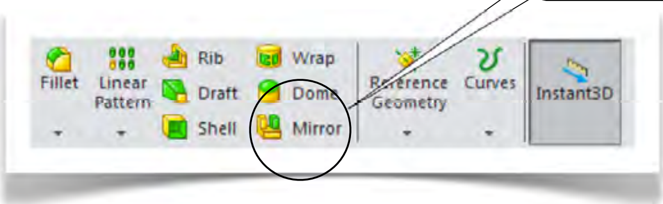
[2] Create fillets of 3 mm on both sides. #

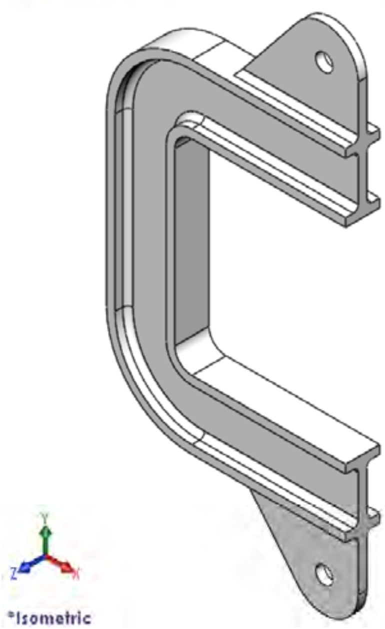


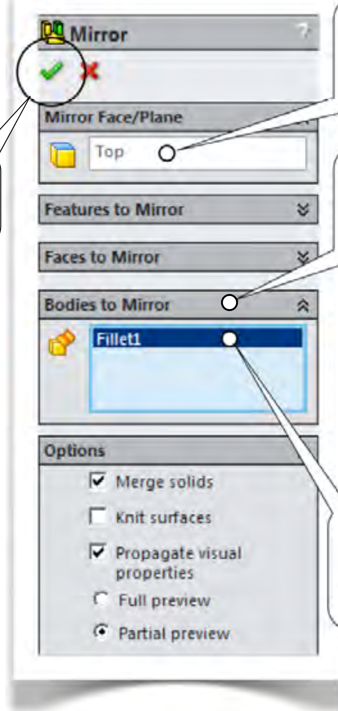


1.3-9 Mirror the Body

[1] In **Features Toolbar**, click **Mirror**.







[2] From **Part Tree**, select **Top** plane.

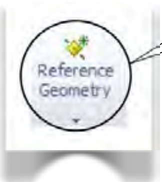
[3] Click **Bodies to Mirror**.

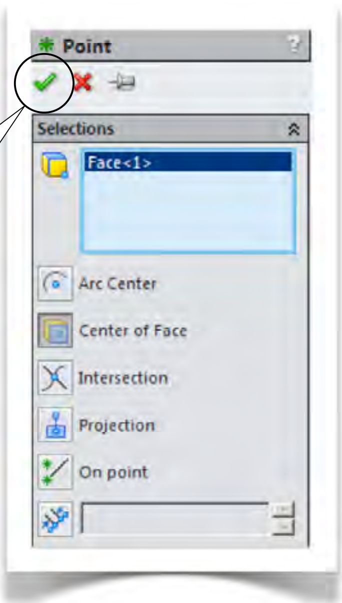
[4] In the **Graphics Area**, select the body.

[5] Click **OK**. #

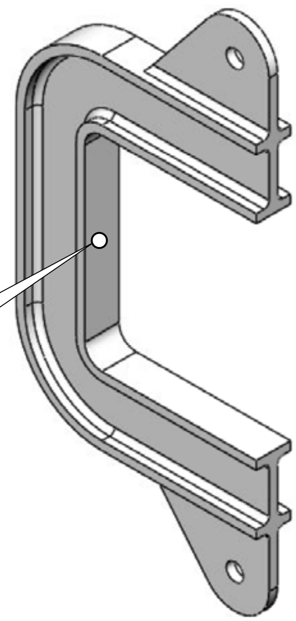
1.3-10 Create **Sensor** at Location **A**

[1] In **Features Toolbar**, select **Reference Geometry>Point**.



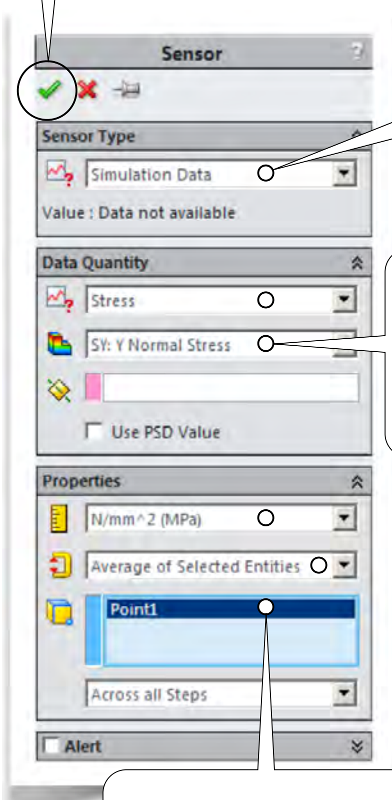


[3] Click **OK**.



[2] Click this face. A **Reference Point** is created at the center of the face. This is the location **A**, where we want to set up a **Sensor**.

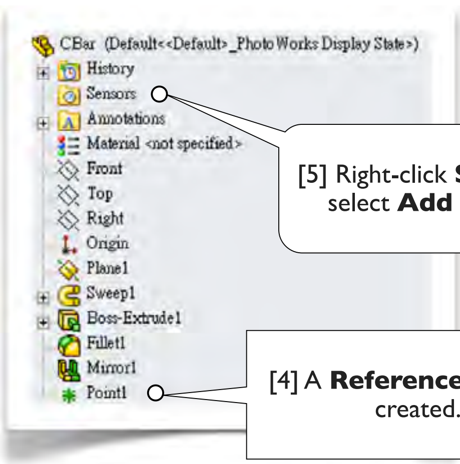
[9] Click **OK**. #



[6] Select **Simulation Data**.

[7] Select **SY**. The **Sensor** is initially set up to be associated with σ_Y. It can be changed later.

[8] Set up **Properties** like this and select **Point1** from **Part Tree**.

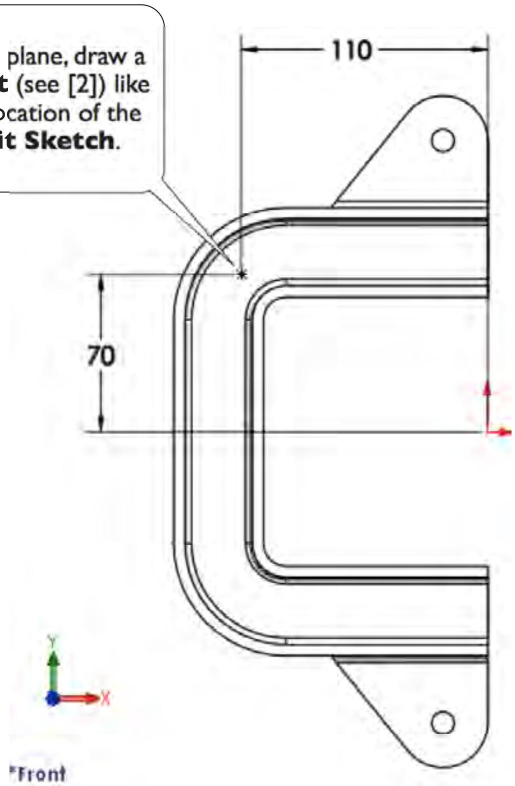


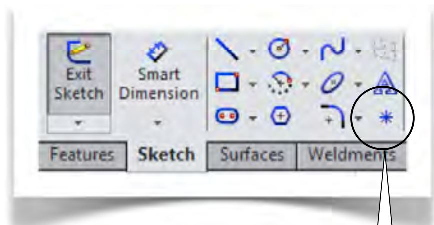
[5] Right-click **Sensors** and select **Add Sensor...**

[4] A **Reference Point** is created.

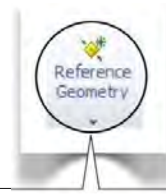
1.3-11 Create **Sensor** at Location **B**

[1] On the **Front** plane, draw a **Sketching Point** (see [2]) like this. Specify the location of the point. Click **Exit Sketch**.



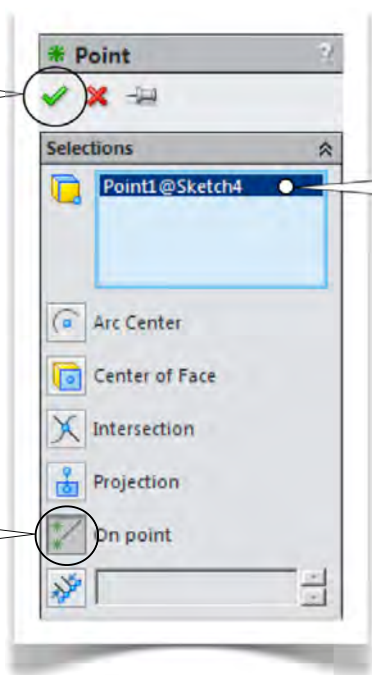


[2] **Sketching Point**.



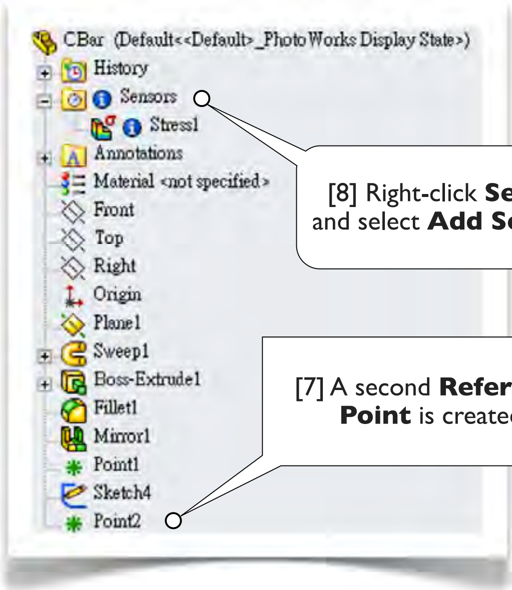
[3] In **Features Toolbar**, select **Reference Geometry>Point**.

[6] Click **OK**.



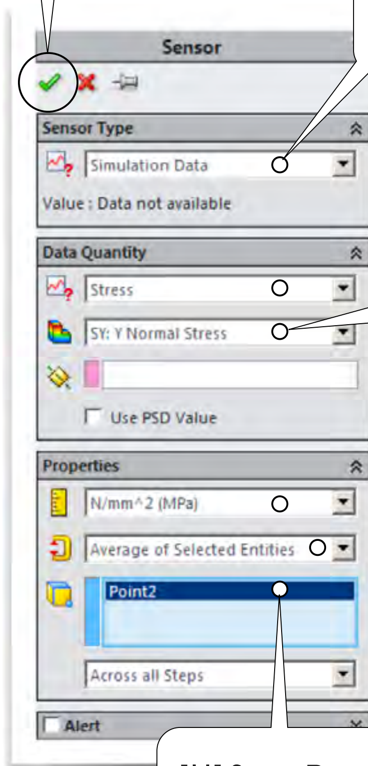
[5] In the **Graphics Area**, select the **Sketching Point**. A **Reference Point** is created at the location of the **Sketching Point**.

[4] Click **On Point**.



[7] A second **Reference Point** is created.

[8] Right-click **Sensors** and select **Add Sensor...**

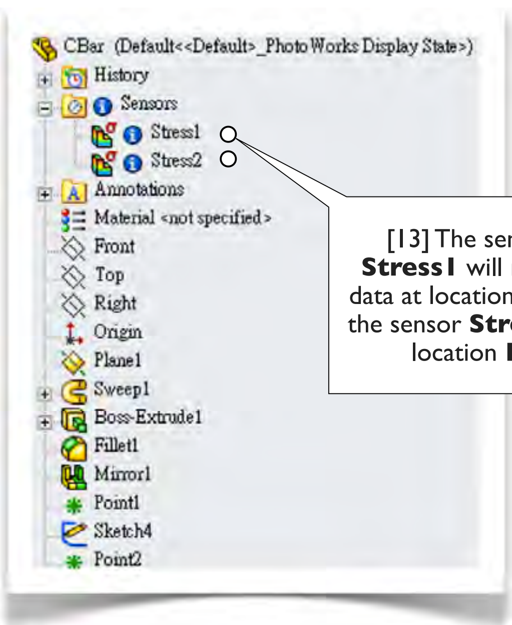


[9] Select **Simulation Data**.

[10] Select **SY**. The **Sensor** is initially set up to be associated with σ_Y. It can be changed later.

[11] Set up **Properties** like this and select **Point2** from **Part Tree**.

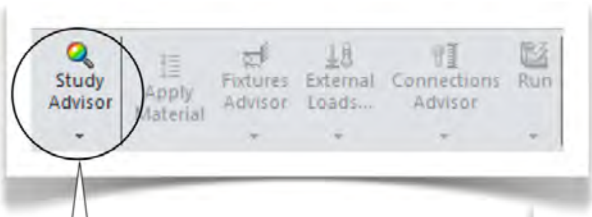
[12] Click **OK**.



[13] The sensor **Stress1** will report data at location **A**, and the sensor **Stress2** at location **B**.

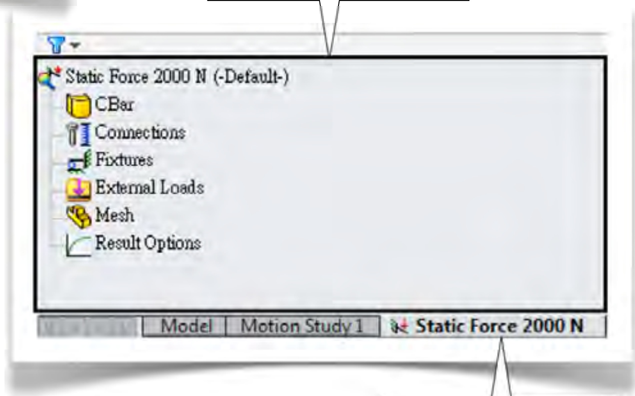
[14] Save the document with the name **CBar**. #

1.3-12 Create a Static Structural Study and Set Up Unit System

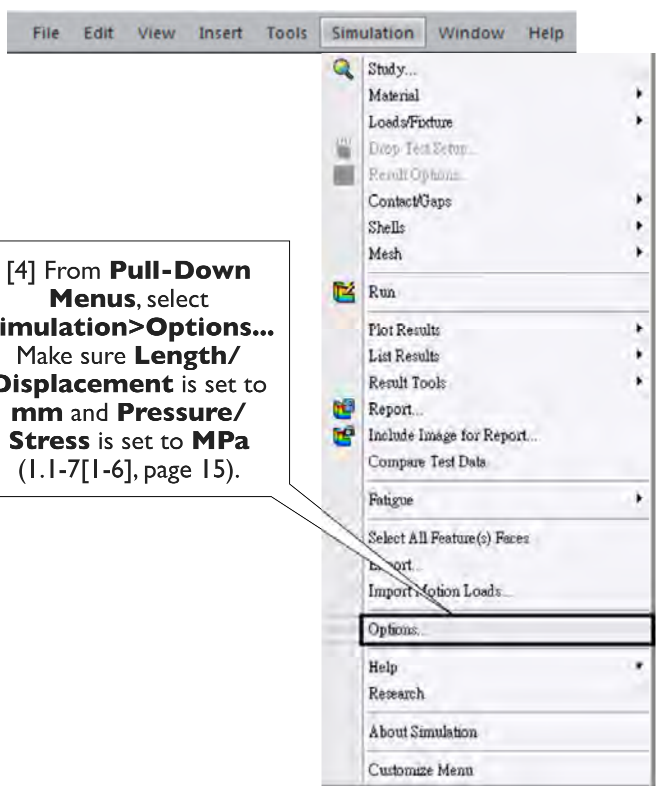


[1] Make sure **SolidWorks Simulation** is loaded (1.1-5, page 13). In the **Simulation Toolbar**, select **Study Advisor>New Study**. Type **Static Force 2000 N** for **Name**. Leave **Static** as default **Study Type**.

[3] The **Study Tree**.

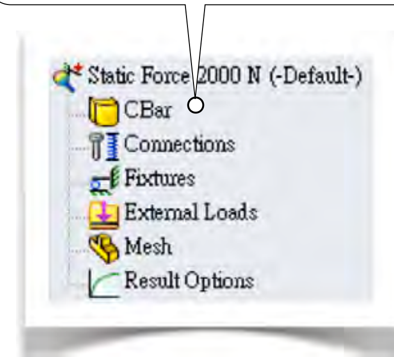


[2] A tab is added and becomes active.

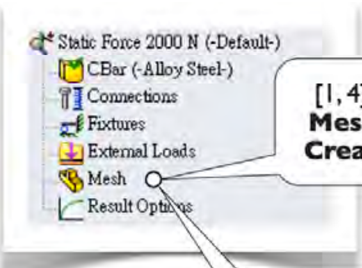


[4] From **Pull-Down Menus**, select **Simulation>Options...** Make sure **Length/ Displacement** is set to **mm** and **Pressure/ Stress** is set to **MPa** (1.1-7[1-6], page 15).

[5] In the **Study Tree**, right-click **CBar** and select **Apply/Edit Material...** and apply **Alloy Steel** for the model (1.1-8, page 17). #

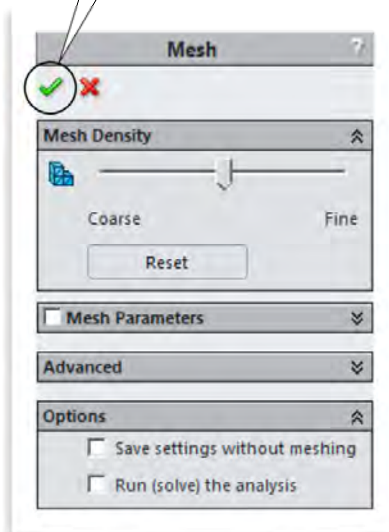


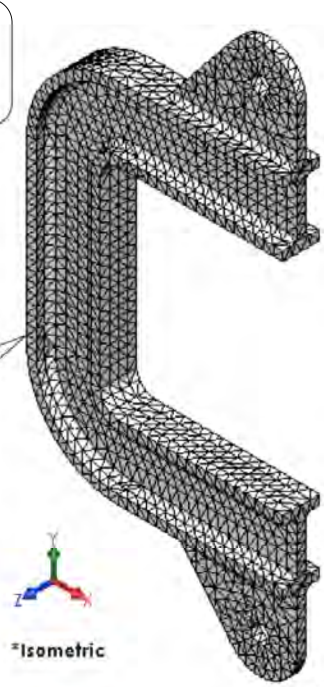
1.3-13 Create Mesh



[1, 4] Right-click **Mesh** and select **Create mesh...**

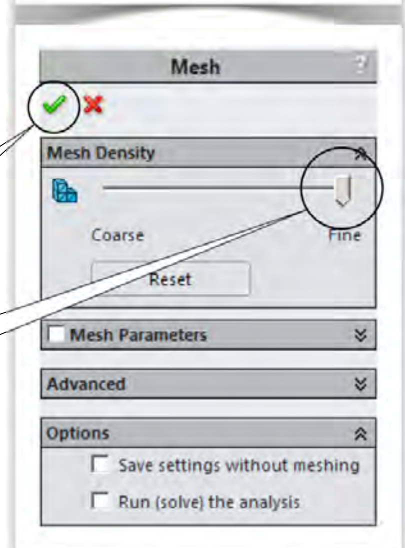
[2] Click **OK** to accept the default settings.





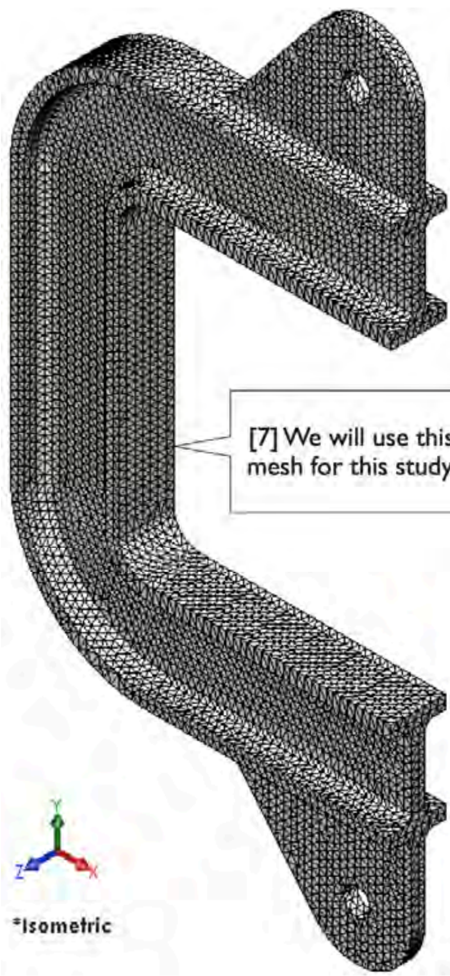
[3] This is the default mesh. In general, the finer the mesh, the more accurate the solutions. Let's make the mesh finer.

[8] Right-click **Mesh** and select **Details...**



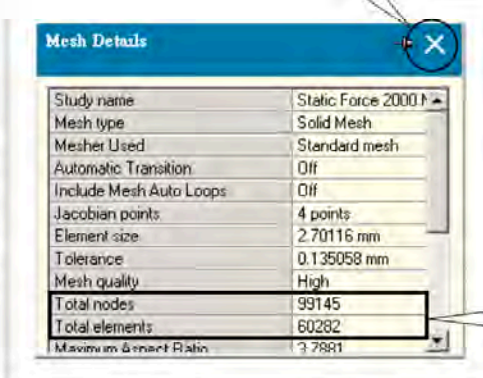
[5] Drag the slider all the way to the right. This is an easy way to refine the mesh.

[6] Click **OK**.



[7] We will use this mesh for this study.

[10] Click to close the **Mesh Details**. #

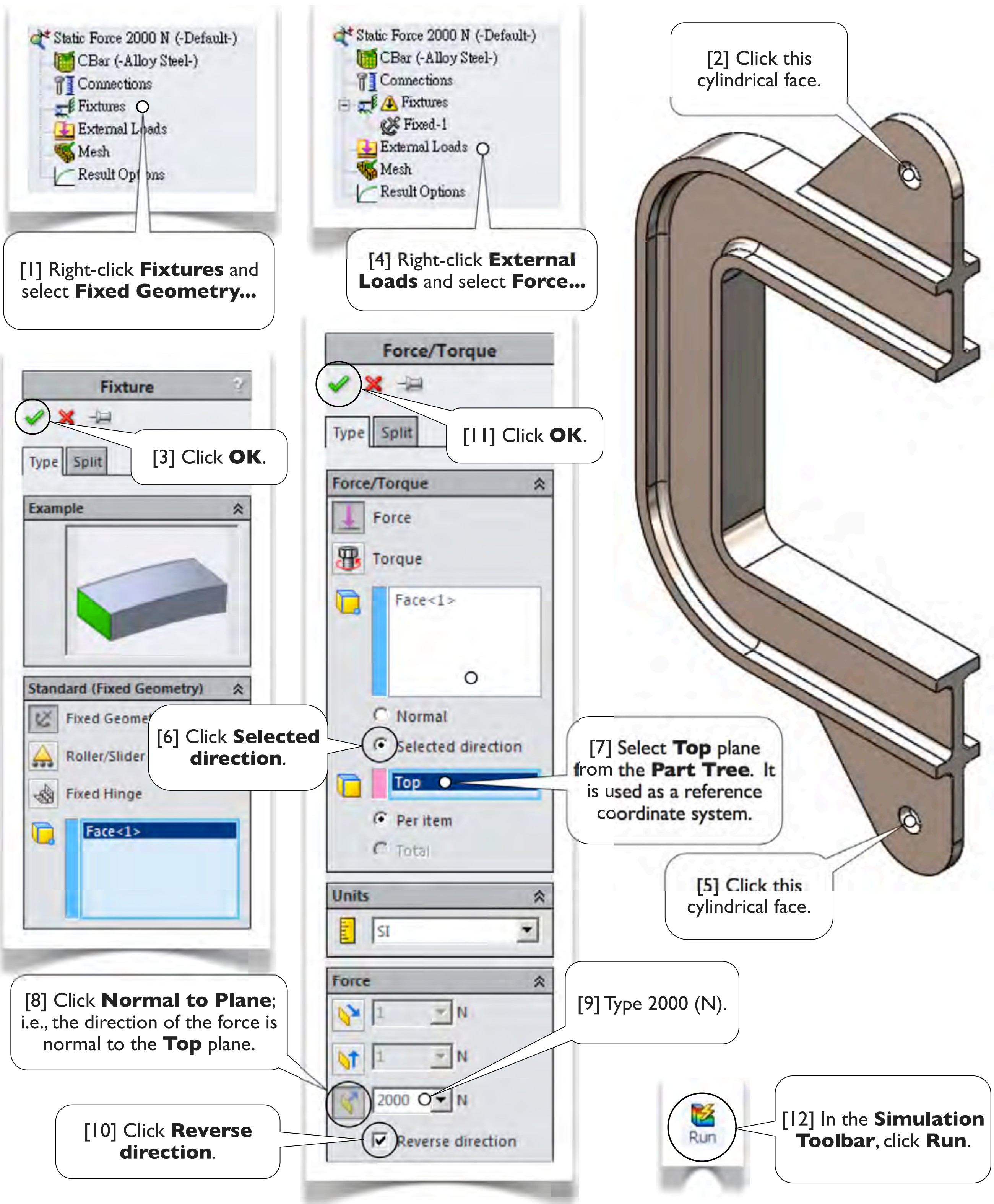


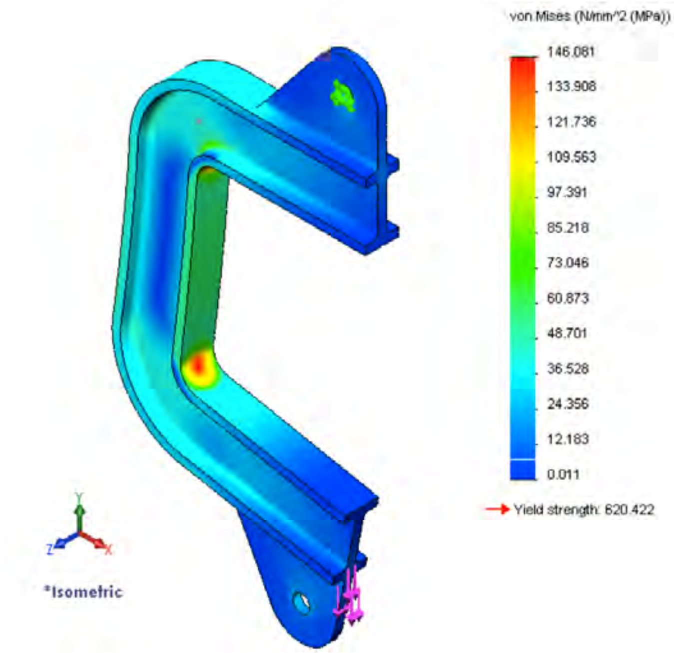
Mesh Details

Study name	Static Force 2000 N
Mesh type	Solid Mesh
Mesher Used	Standard mesh
Automatic Transition	Off
Include Mesh Auto Loops	Off
Jacobian points	4 points
Element size	2.70116 mm
Tolerance	0.135058 mm
Mesh quality	High
Total nodes	99145
Total elements	60282
Maximum Aspect Ratio	3.7881

[9] The **Mesh Details** shows that the mesh consists of 99,145 nodes and 60,282 elements. Your numbers may not be exactly the same as here. If you don't create a mesh, the program will automatically create a default mesh right before solving the model.

1.3-14 Set Up Boundary Conditions and Run the Model



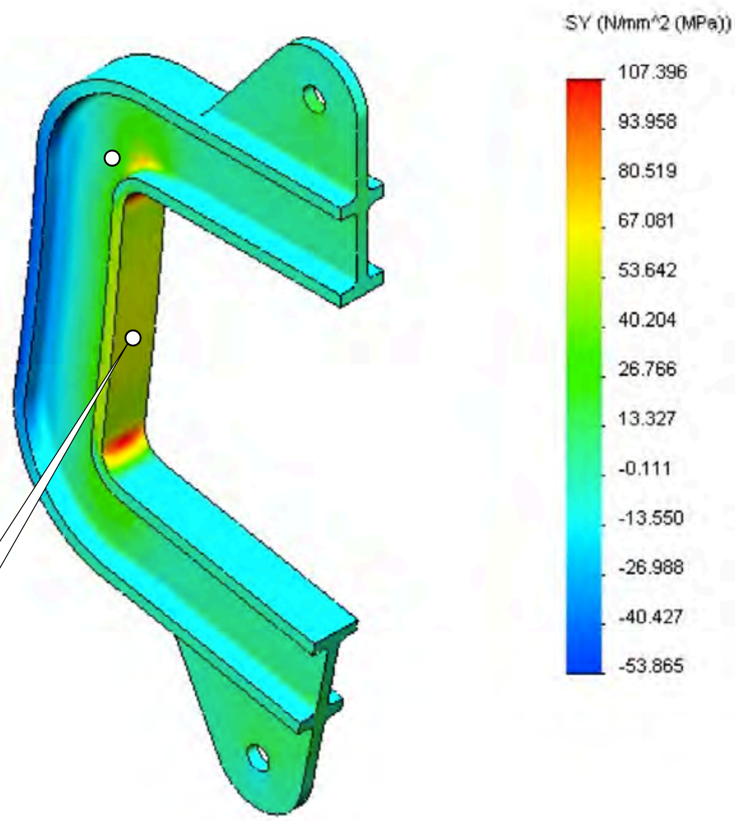


[14] Hide all **Fixtures** and **External Loads** (1.1-12[6, 7], page 21).

[15] Right-click **Stress1** and select **Animate...**

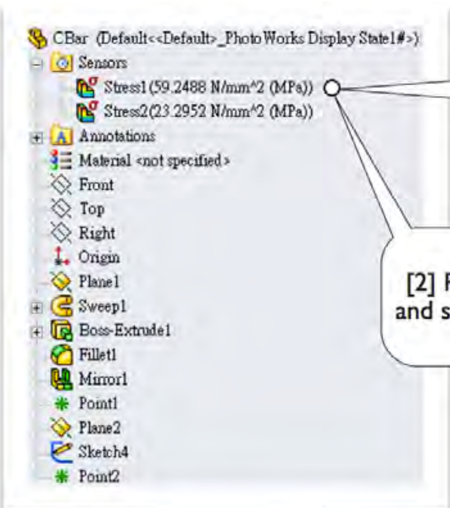
[13] Change to display **SY** (σ_Y) (1.1-12[1, 2, 4], page 21).

[16] Click **OK** after viewing the animation.



[17] Next, we'll explore the stresses at locations **A** and **B**. #

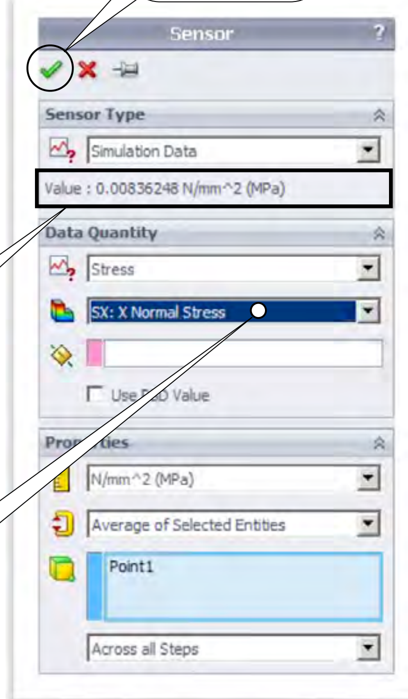
1.3-15 The Stresses at Location A



[1] Recall that the sensor **Stress1** is initially set up to be associated with σ_Y. Write down this stress value (59.2488 MPa) [5].

[2] Right-click **Stress1** and select **Edit Sensor**.

[6] Click **OK**.



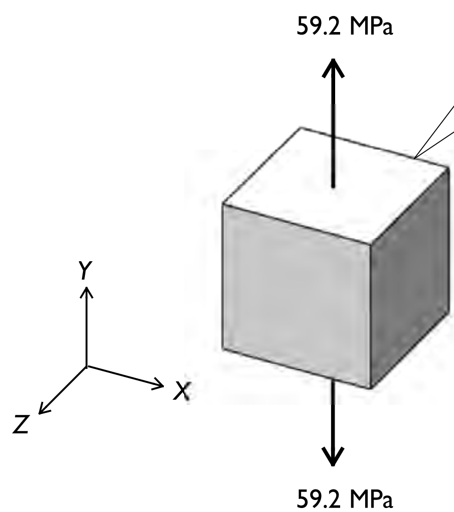
[4] The sensor **Stress1** reports σ_X value here. The σ_X is essentially zero.

[3] Change to **SX** (σ_X).

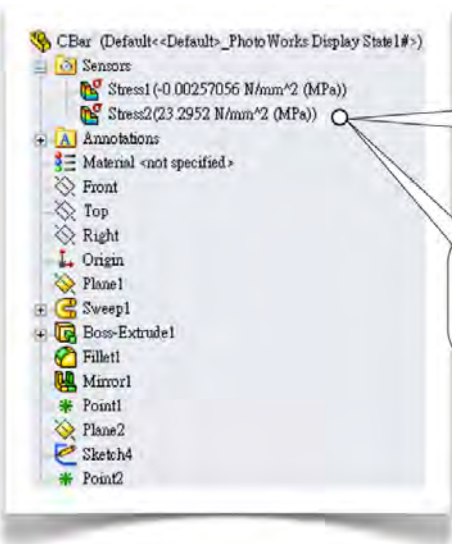
Stress Component	Stress Value
σ_X	0
σ_Y	59.2488 MPa
σ_Z	0
τ_{XY}	0
τ_{YZ}	0
τ_{ZX}	0

[5] Repeat steps [3, 4] to obtain other stress components. All other stress components than σ_Y are essentially zeros.

[7] The stress state at location **A**. #



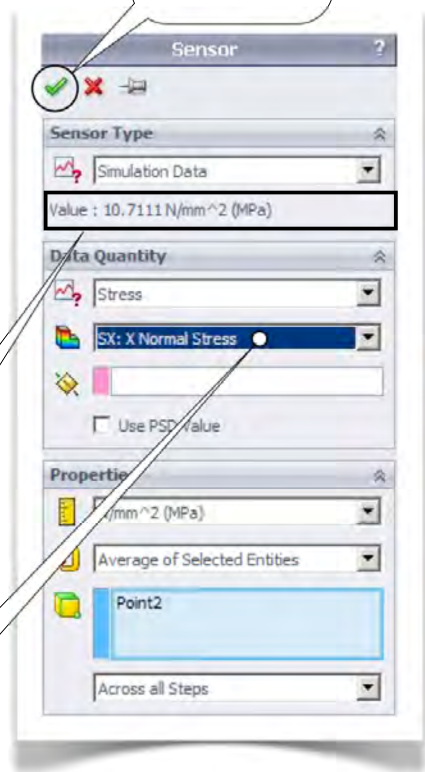
1.3-16 The Stresses at Location **B**



[1] The sensor **Stress2** is initially set up to be associated with σ_Y. Write down this stress value (23.2952 MPa) [5].

[2] Right-click **Stress2** and select **Edit Sensor**.

[6] Click **OK**.

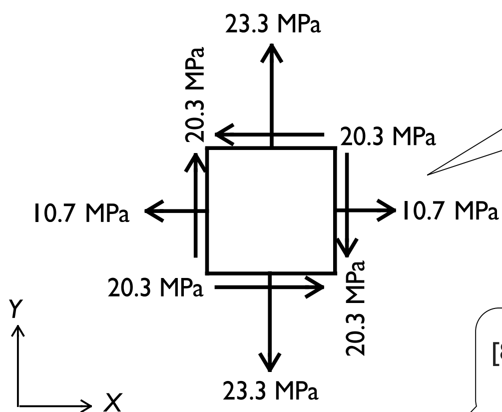


[4] The sensor **Stress2** reports σ_X value here (10.7111 MPa).

[3] Change to **SX** (σ_X).

Stress Component	Stress Value
σ_X	10.7111 Mpa
σ_Y	23.2952 MPa
σ_Z	0
τ_{XY}	-20.2619 MPa
τ_{YZ}	0
τ_{ZX}	0

[5] Repeat steps [3, 4] to obtain other stress components. Your results may not be exactly the same as here. Note that the Z-components (σ_Z, τ_{YZ}, τ_{ZX}) are all zeros; it is called a **plane stress state** (Section 12.1). The stress states discussed in this chapter are all plane stress states.



[7] The stress state at location **B**. Note that the shear stresses point to the negative directions.

[8] Save the document and exit **SolidWorks**. #