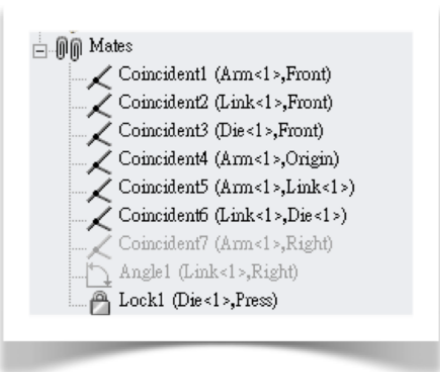
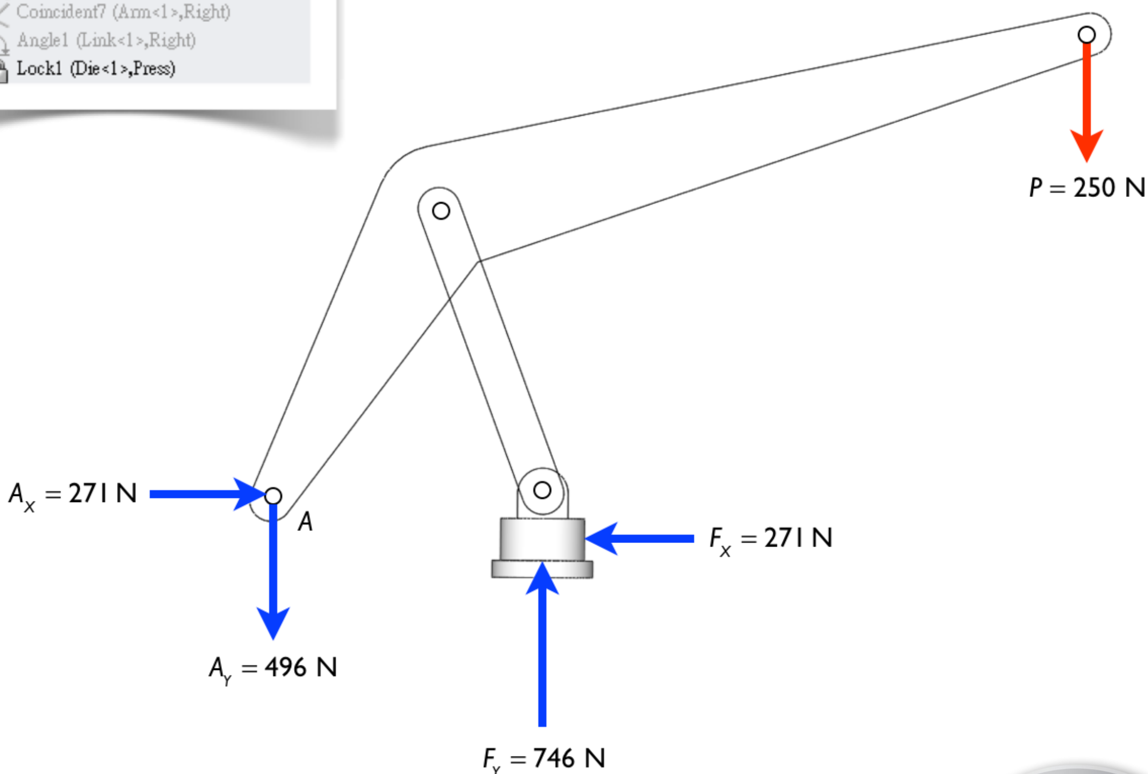
Engineering Statics Labs

with SOLIDWORKS® Motion 2015





Huei-Huang Lee

SDC PUBLICATIONS
Better Textbooks. Lower Prices.
www.SDCpublications.com

Chapter 1

Equilibrium of a Body

Engineering Statics

In Engineering Statics, we study the forces among rigid bodies at rest.

Since the bodies are not moving, the forces must satisfy Newton's equations of static equilibrium. The static equilibrium equations must be satisfied for a single body, a group of bodies in the system, or the entire bodies system. It is fair to say that Engineering Statics is the study of the static equilibrium equations and their application.

In "solving" a structural problem, we mean finding the forces acting on each structural member. In many cases, static equilibrium equations are enough to solve the problems. In these cases, we called these problems **statically determinate problems**. In other cases, we may need to seek other conditions, such as deformation conditions, to solve the problems. In these cases, we called these problems **statically indeterminate problems**. In Engineering Statics, we study statically determinate problems only.

Rigid Body Assumption

In the real world, all solid bodies are more or less deformable. There are no such things as **rigid bodies**. However, if the deformation of a body is not our concern and if the deformation is negligible, we can treat the body as a **rigid body**. In Engineering Statics, we assume all bodies studied in this book are **rigid bodies**. If deformation needs to be considered, we'll use a spring to represent a body. In Engineering Statics, springs are the only elements that are deformable.

Rigid body assumption might introduce errors. However, in many cases, the errors are negligible. In statically determinate problems, rigid body assumption usually introduce negligible errors.

Section 1.1

Supported Block: A 2D Case

1.1-1 Introduction

[1] In this section, we consider a block [2] supported by a hinge [3] and a roller [4] and find the reaction forces at the supports. Before we proceed to solve the problem using **SOLIDWORKS**, let's manually calculate the reaction forces.

From the free-body diagram [5], taking the moment equilibrium about *A*, we have

$$\sum M_A = 0$$

$$(150\text{ N})(1\text{ m}) + (300\text{ N})(0.5\text{ m}) - B_Y(1.5\text{ m}) = 0$$

$$B_Y = 200\text{ N}$$

Force equilibrium in *Y* direction,

$$\sum F_Y = 0$$

$$A_Y + B_Y - (300\text{ N}) = 0 \quad (\text{where } B_Y = 200\text{ N})$$

$$A_Y = 100\text{ N}$$

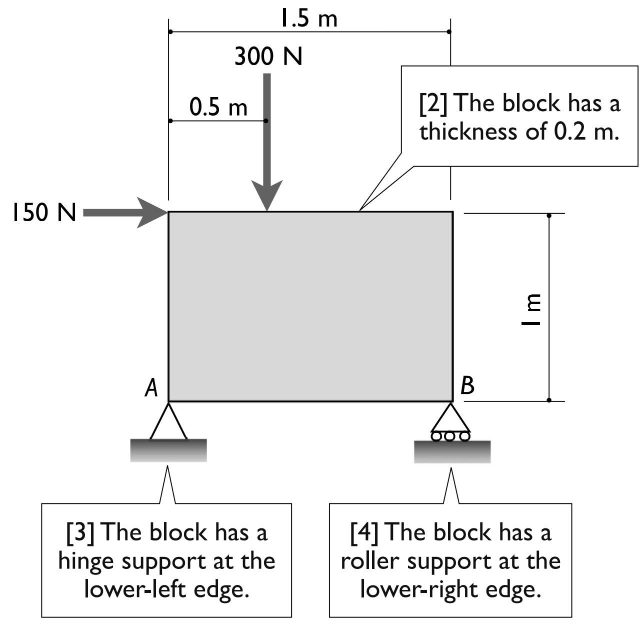
Finally, force equilibrium in *X* direction,

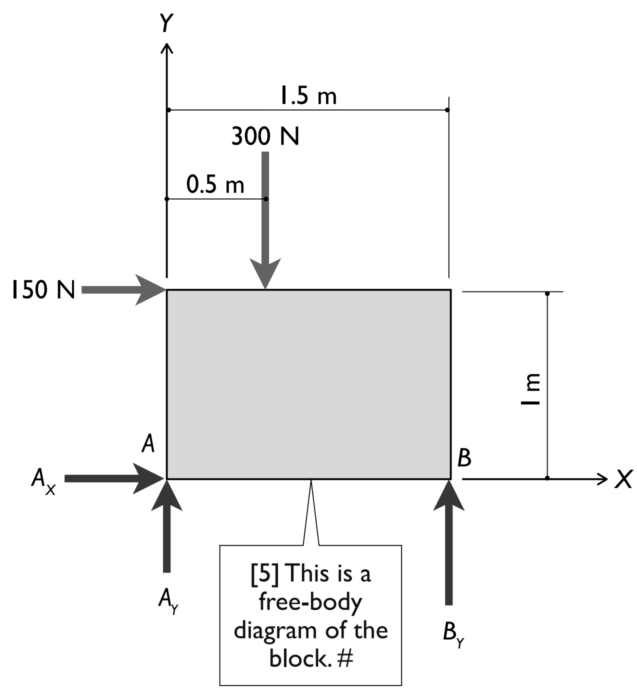
$$\sum F_X = 0$$

$$A_X + (150\text{ N}) = 0$$

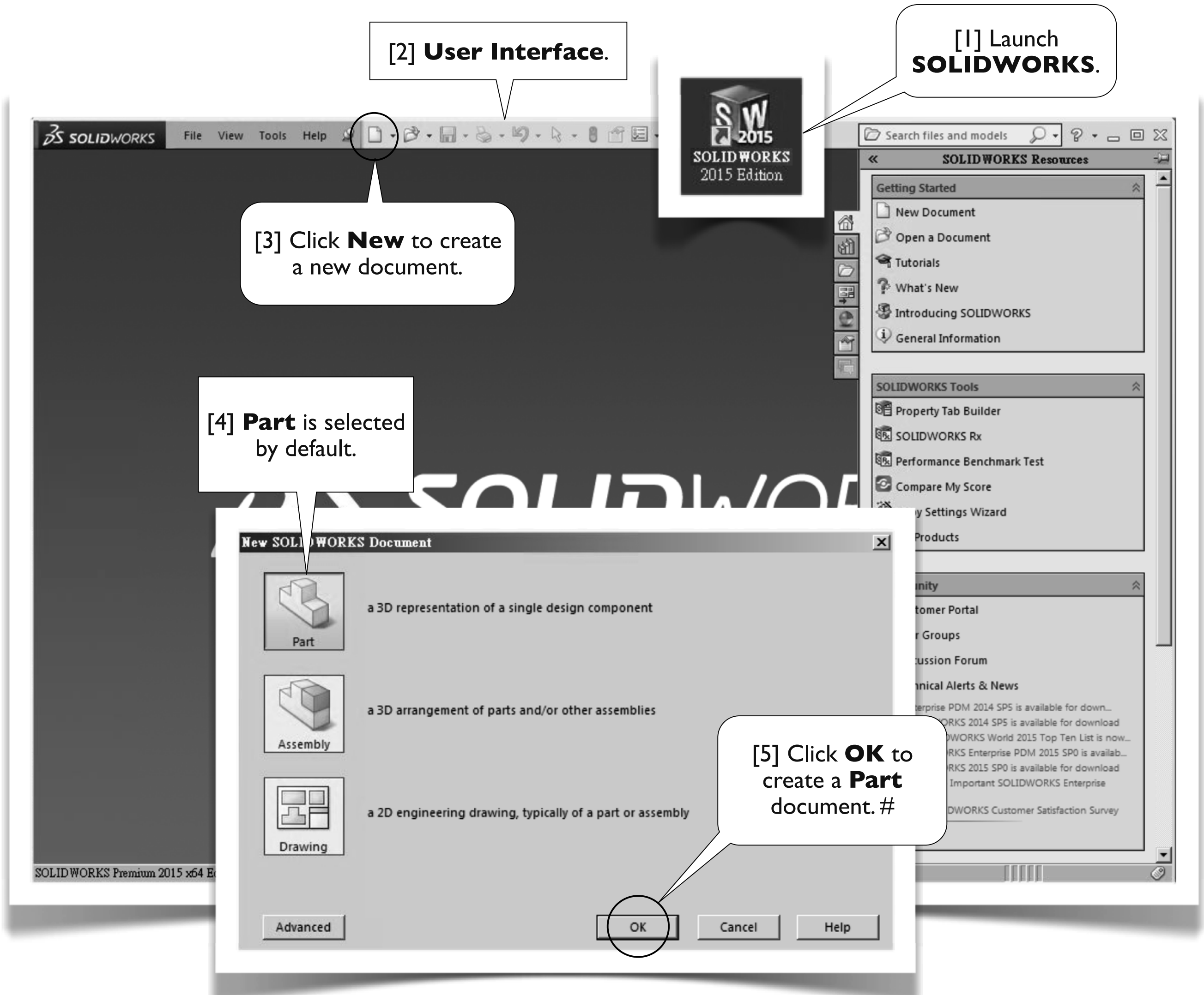
$$A_X = -150\text{ N}$$

Note that the negative sign of A_X indicates that it is opposite to the assumed direction shown in [5]. Now, let's solve this problem with **SOLIDWORKS**. If you know how to solve a simple problem like this, you may be able to solve a much more complicated problem.





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



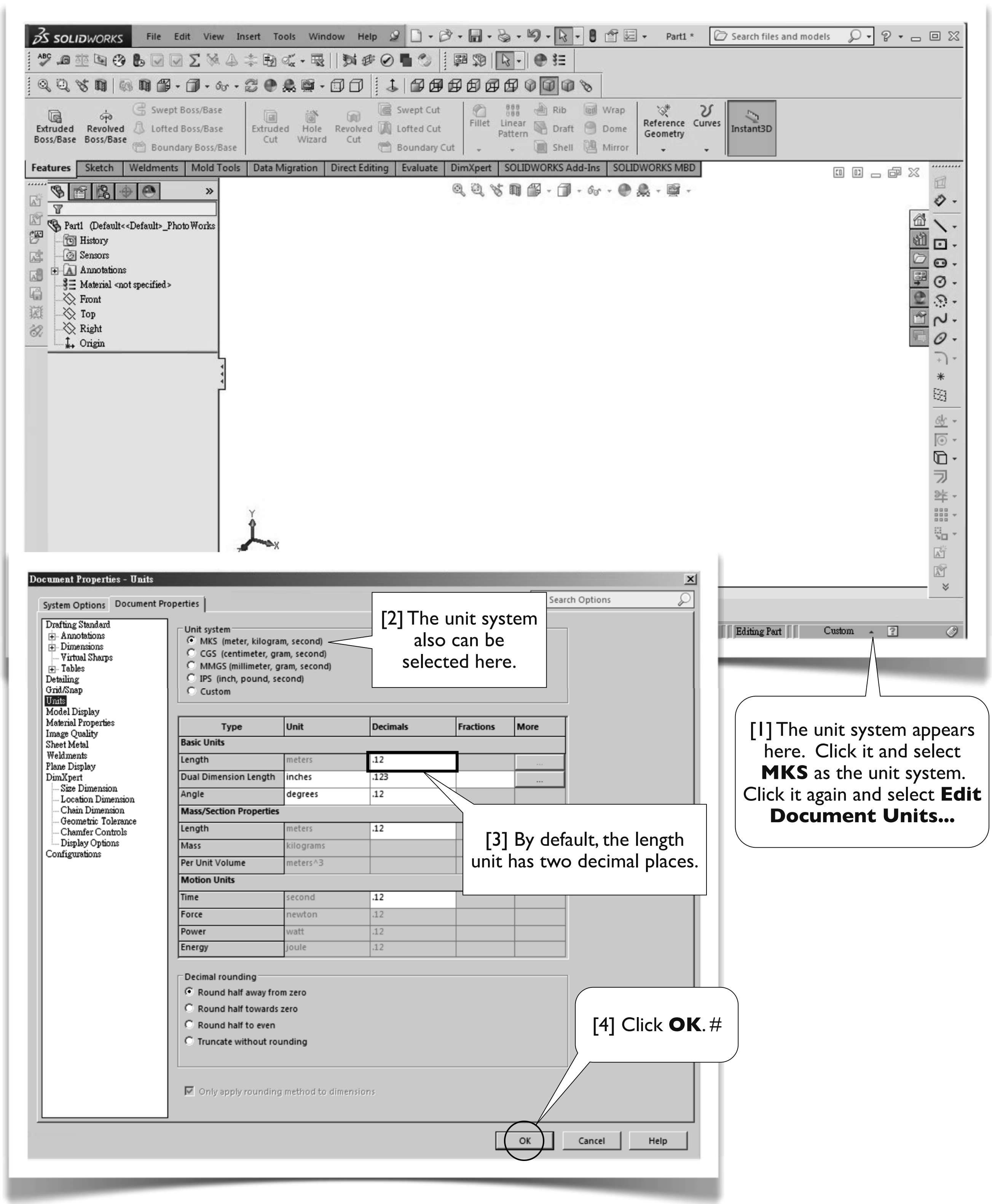
About the Textboxes

1. Within each subsection (e.g., 1.1-2), textboxes 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 textboxes.
2. The textbox numbers are also used as reference numbers. Inside a subsection, we simply refer to a textbox by its number (e.g., [1]). From other subsections, we refer to a textbox by its subsection identifier and the textbox number (e.g., 1.1-2[1]).
3. A textbox is either round-cornered (e.g., [1, 3, 5]) or sharp-cornered (e.g., [2, 4]). A round-cornered textbox indicates that **mouse or keyboard actions** are needed in that step. A sharp-cornered textbox 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 textbox of a subsection [5], so that you don't leave out any textboxes.

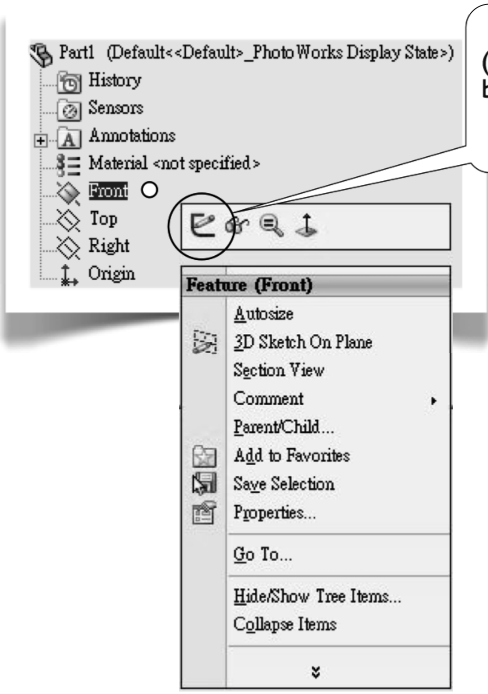
SOLIDWORKS Terms

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

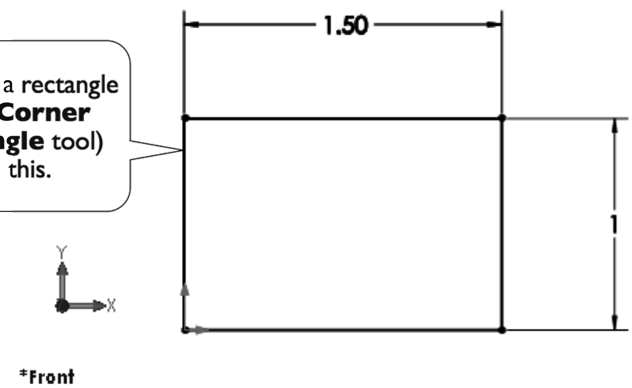


1.1-4 Create a Part: **Block**

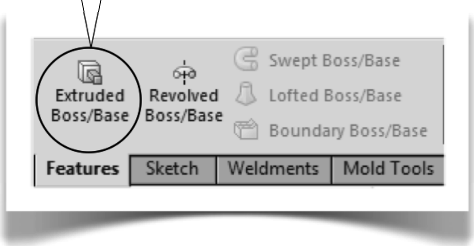


[1] In the **Features Tree** (also called **Part Tree** in this book), right-click **Front** plane and select **Sketch** from the **context menu**.

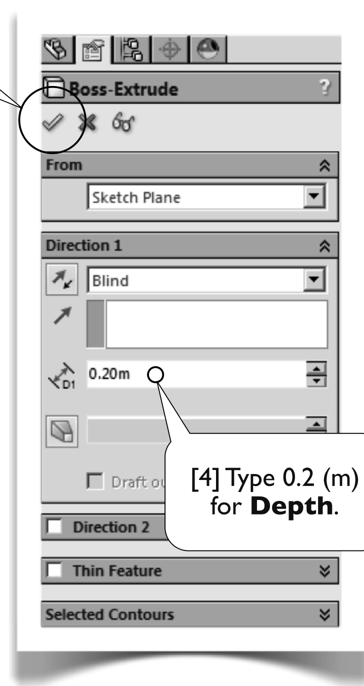
[2] Draw a rectangle (using **Corner Rectangle** tool) like this.



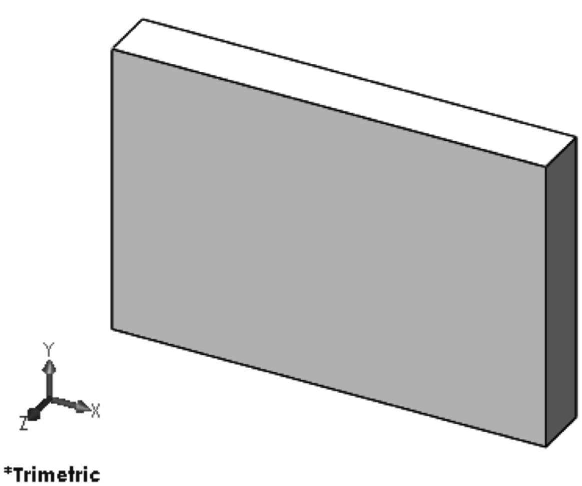
[3] In the **Features** toolbar, click **Extruded Boss/Base**.



[5] Click **OK**. #

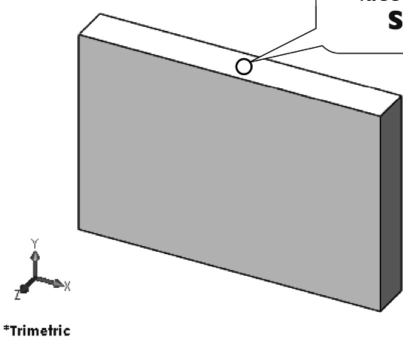


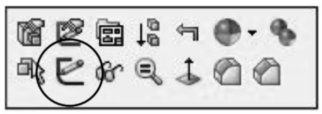
[4] Type 0.2 (m) for **Depth**.



1.1-5 Create Two **Points** on the **Block**

[1] Right-click this face and select **Sketch**.

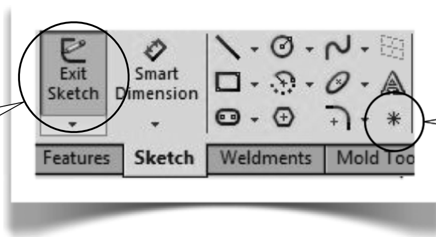




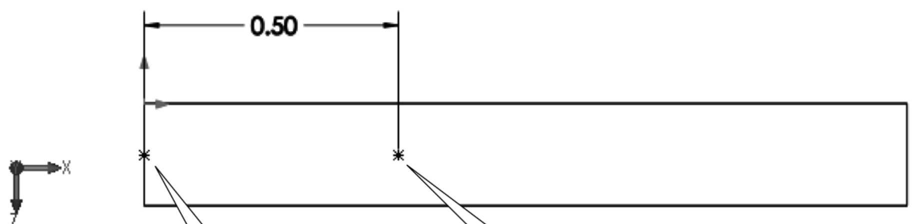
[7] Click **Trimetric**.

[2] In the **Standard Views** toolbar, click **Normal To**.

[6] Click **Exit Sketch**.



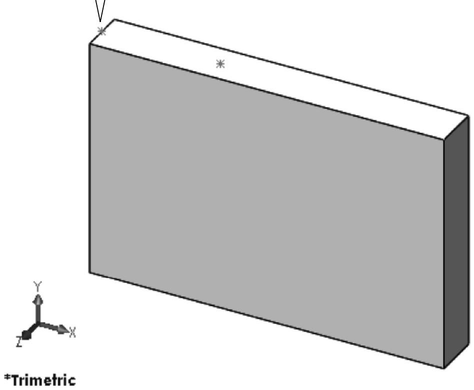
[3] In **Sketch** toolbar, select **Point**.



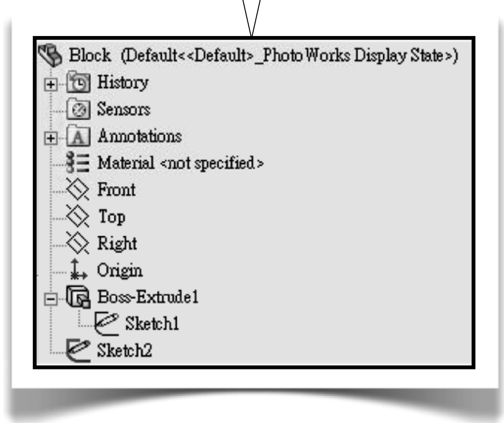
[8] The two **Points** will be used to locate the applied forces.

[4] Create a **Point** on the middle of this edge.

[5] Create a second **Point** like this. Impose a **Horizontal** relation between the two **Points**.



[10] This is the **Features Tree**. In this book, we also call it a **Part Tree**. #

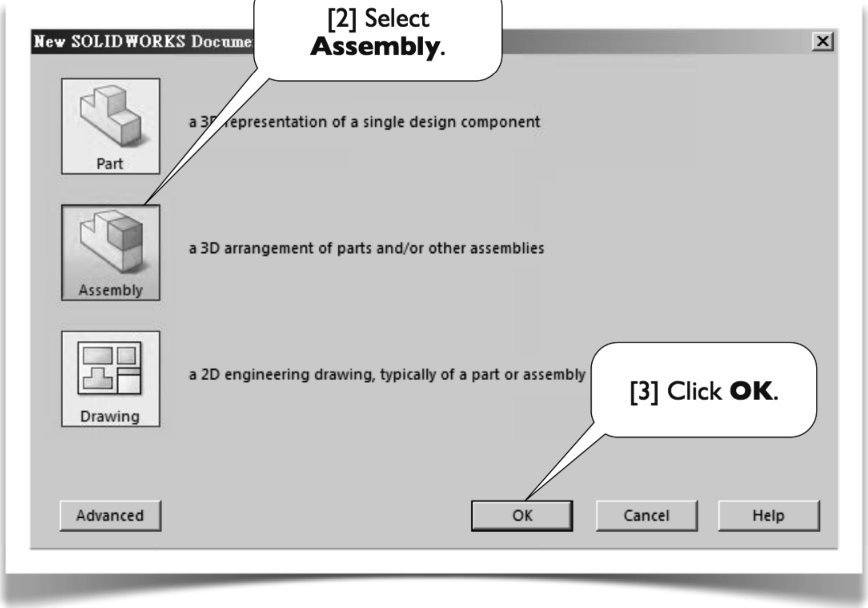


[9] Click **Save** and save the document with the name **Block**. A file **Block.SLDPRT** is created in your working folder.

1.1-6 Create an Assembly: **BlockSupported**

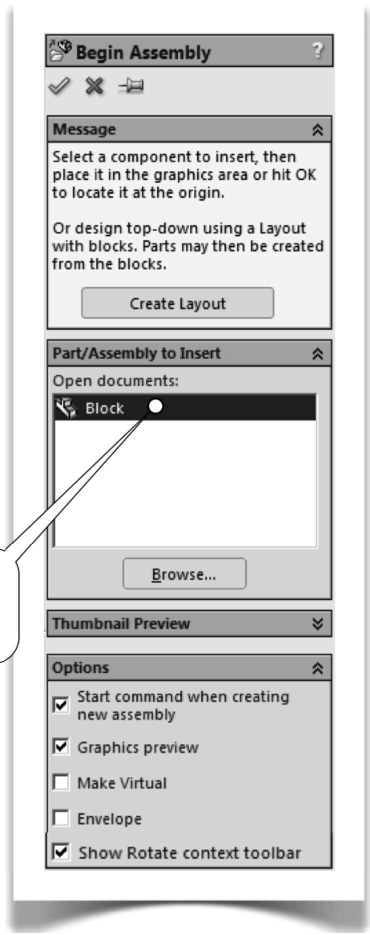
[1] Click **New**.

[2] Select **Assembly**.

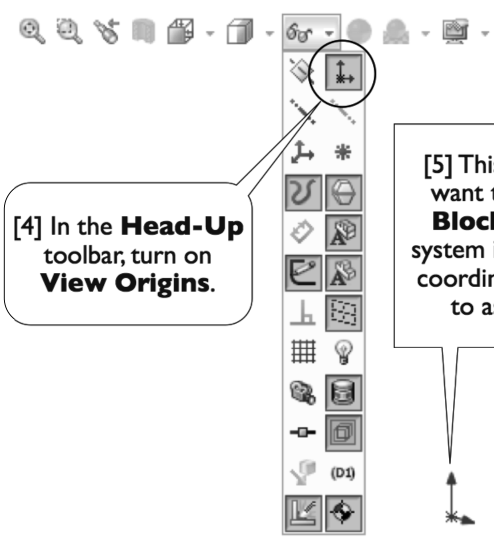


[3] Click **OK**.

[6] In the **Begin Assembly** box, select **Block**.

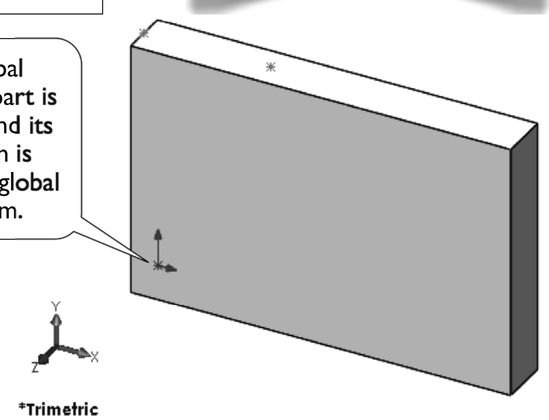


[4] In the **Head-Up** toolbar, turn on **View Origins**.



[5] This is the assembly's **Origin**. We want to show you how to insert the **Block** so that the part's coordinate system is coincident with the assembly's coordinate system, which we also refer to as the *global coordinate system*.

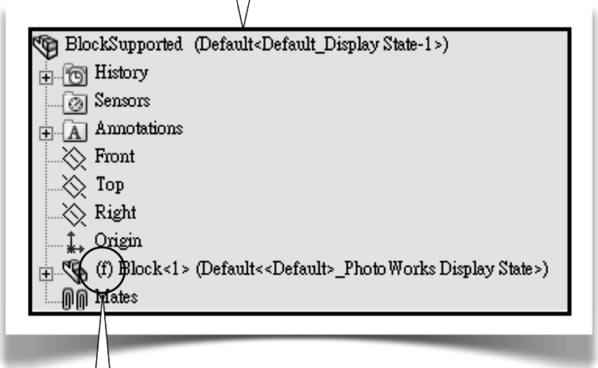
[7] Click the global **Origin**. Now the part is fixed in the space and its coordinate system is coincident with the global coordinate system.



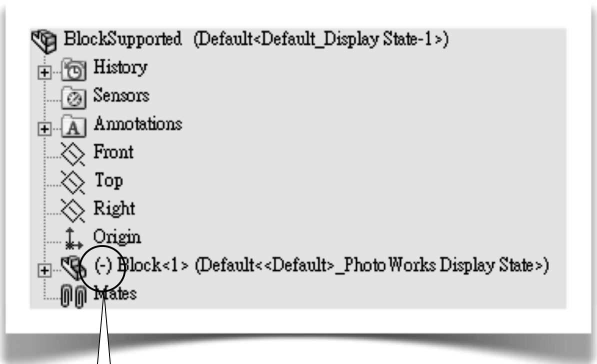
[8] Select **MKS** for the unit system with default decimal places. (1.1-3[1-4], page 7).

[9] Click **Save** and save the document with the name **BlockSupported**. A file **BlockSupported.SLDASM** is created in your working folder.

[10] This is the **Features Tree** of the **Assembly**. In this book, we simply call it an **Assembly Tree**.

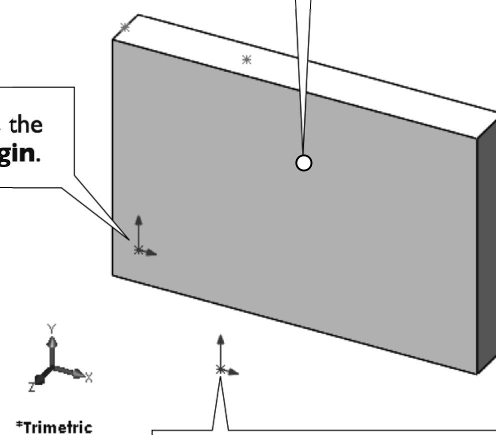


[11] An **(f)** sign indicates that the **Block** is fixed in the space. Now, right-click it and select **Float**.



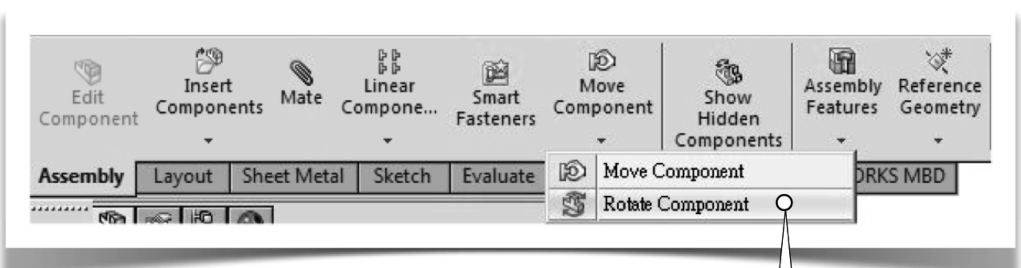
[12] The **(f)** sign changes to **(-)**, indicating that the **Block** is no more fixed. We'll show that it can be moved freely [13-16]. In 1.1-7 (next page), we'll set up constraints to fully support the **Block**.

[13] Left-click-drag the body to move it around the space.



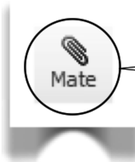
[14] This is the part's **Origin**.

[15] Now, the global **Origin** may not be coincident with the part's **Origin**.

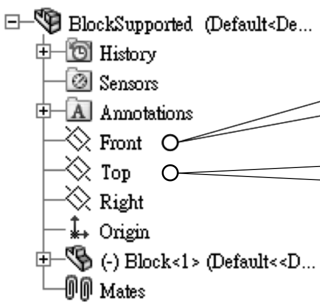


[16] Using **Move Component>Rotate Component** tool, you even can freely rotate the body. #

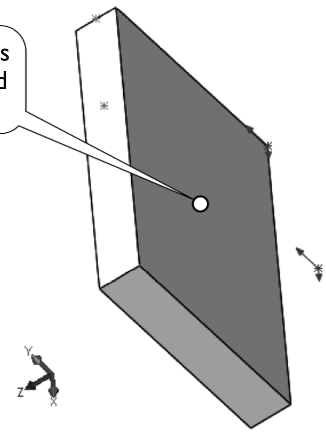
1.1-7 Set Up Supports



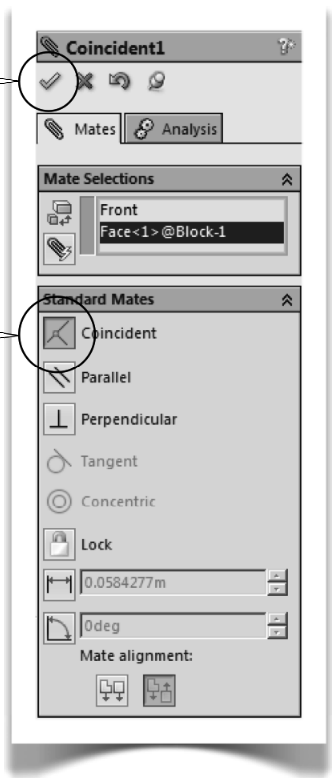
[1] In the **Assembly** toolbar, click **Mate**.



[2] In the **Graphics Area**, expand the **Assembly Tree**, and select the assembly's (global) **Front** plane...



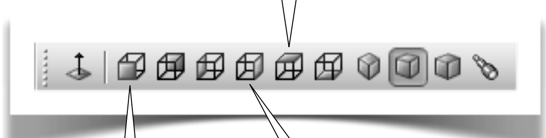
[3] And select the body's back face. You may need to rotate the view.



[4] **Coincident** mate is automatically selected.

[5] Click **OK**. Note that the **Mate** box is still open.

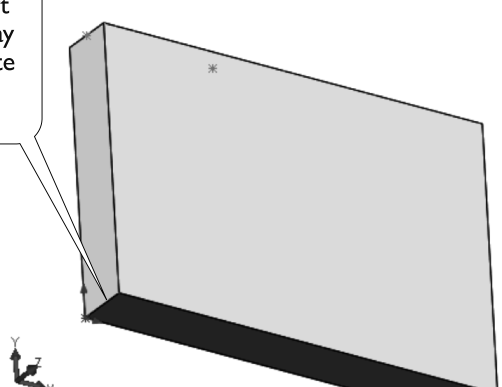
[6] Now, the body is constrained such that the back face is coincident with the global **Front** plane. To verify this, rotate to **Top** view, and left-click-drag the body.



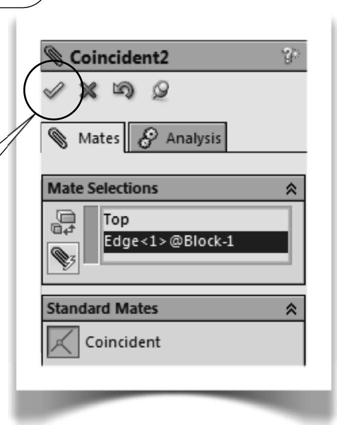
[7] Also, rotate to **Right** view, and left-click-drag the body.

[8] Finally, rotate to **Front** view, and left-click-drag the body. We conclude that the body is indeed constrained to move in XY-space.

[9] Select the global **Top** plane...

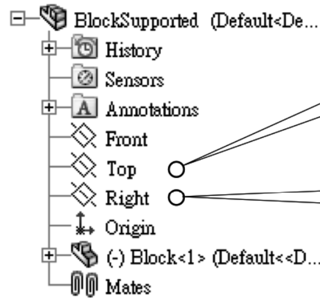


[10] And select the lower-left edge. You may need to rotate the view.



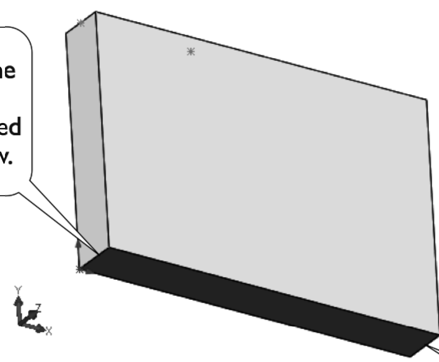
[11] Click **OK**.

[12] Now, the body is further constrained such that the lower-left edge is coincident with the global **Top** plane. To verify this, rotate to **Front** view, and left-click-drag the body.

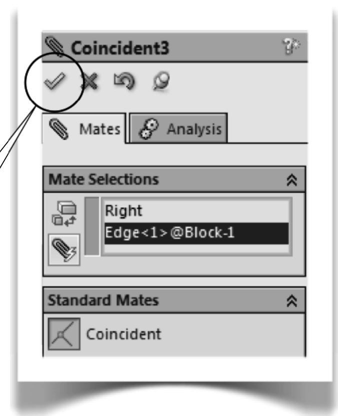


[13] Select the global **Right** plane...

[14] And select the lower-left edge again. You may need to rotate the view.



[15] Click **OK**.

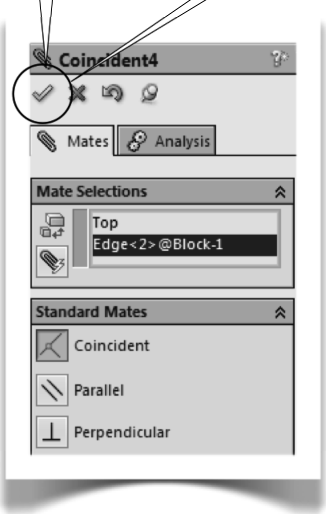


[16] Now, the body is further constrained such that the lower-left edge is coincident with the global **Right** plane. To verify this, rotate to **Front** view, and left-click-drag the body. Now, the body is restricted to rotate about the Z-axis.

[17] Select the global **Top** plane...

[18] And select the lower-right edge. You may need to rotate the view.

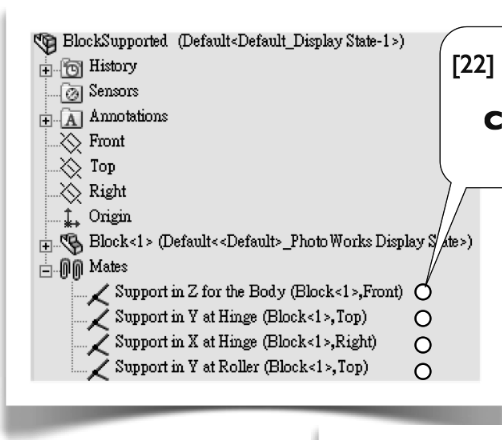
[19] Click **OK**.



[20] Now, the body is fully constrained (supported). To verify this, rotate to **Front** view again, and try to move the body. The body can't be moved now.

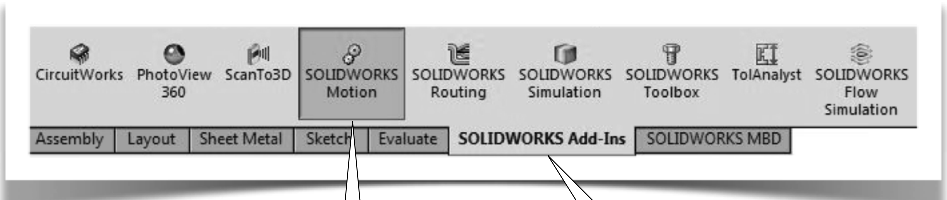
[21] Click **OK** again to dismiss the **Mate** box.

[22] In the **Assembly Tree**, expand **Mates** and rename the four **Coincident** mates like this for better readability.



[23] Click **Save**. #

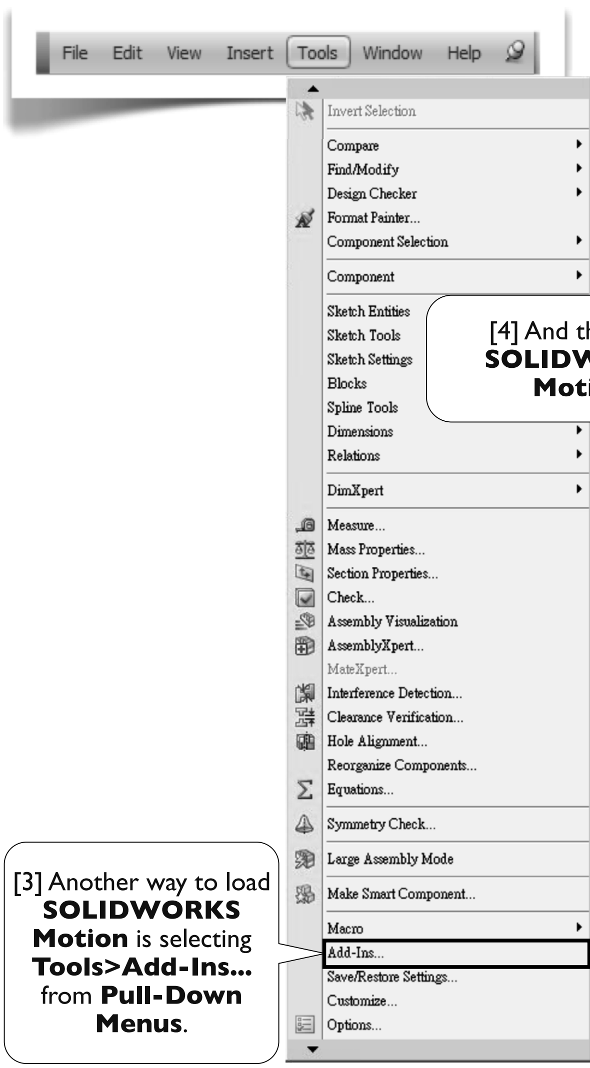
1.1-8 Load **SOLIDWORKS Motion**



[1] Click **SOLIDWORKS Add-Ins** tab.

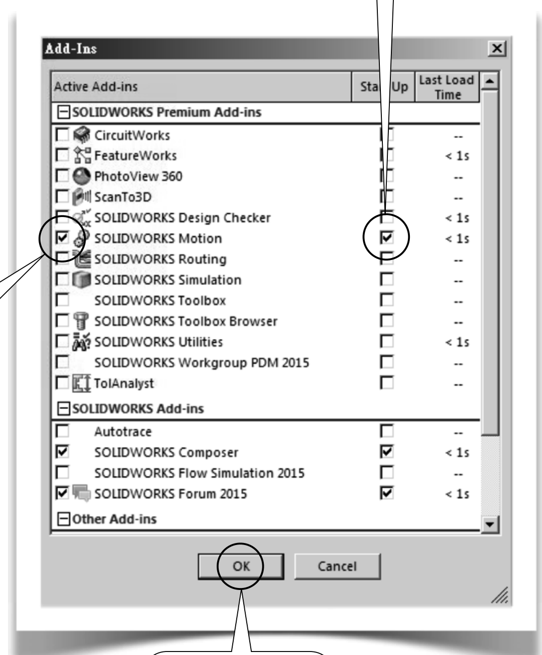
[2] If **SOLIDWORKS Motion** is highlighted, that means it has already been loaded; you may jump to next page. Otherwise, click it to load **SOLIDWORKS Motion**, or...

[5] Also click here so that the **Motion** will be loaded automatically each time you start up **SOLIDWORKS**. In this book, we assume that you set up this way so that the **Motion** is loaded automatically each time you start up **SOLIDWORKS**.



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

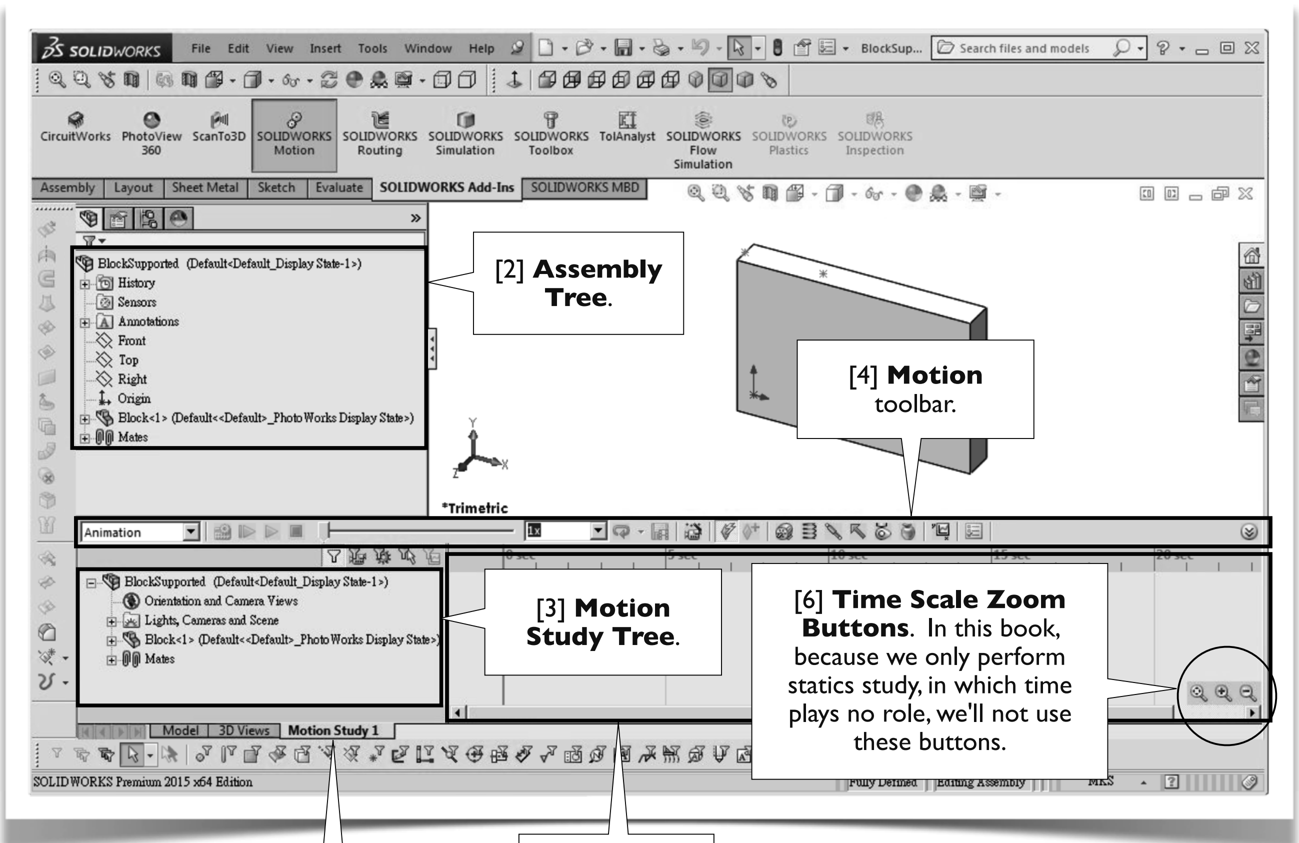
[4] And then click **SOLIDWORKS Motion**.



[6] Click **OK**.

[7] **SOLIDWORKS Motion** is designed to solve rigid-body mechanical problems, either static or dynamic. By rigid-body, it means the deformations of bodies are neglected. In **SOLIDWORKS Motion**, a static problem is treated as a special case (of dynamic problems) in which the response is independent of time. #

1.1-9 Create a **Motion Study**



[1] Click **Motion Study 1** tab.

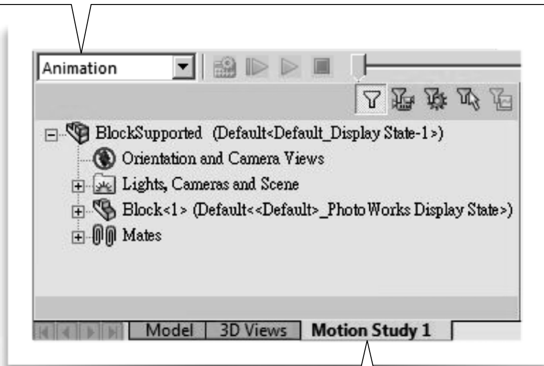
[2] **Assembly Tree**.

[3] **Motion Study Tree**.

[4] **Motion** toolbar.

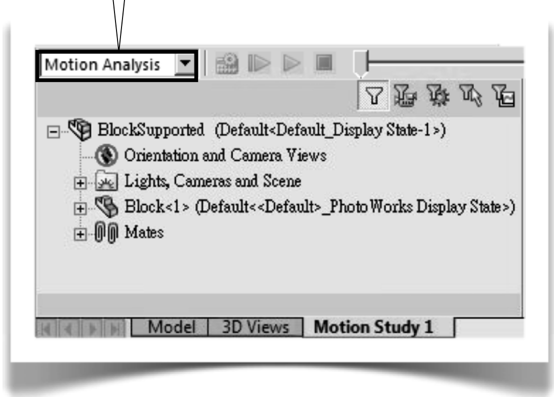
[5] **Timeline Area**.

[6] **Time Scale Zoom Buttons**. In this book, because we only perform statics study, in which time plays no role, we'll not use these buttons.



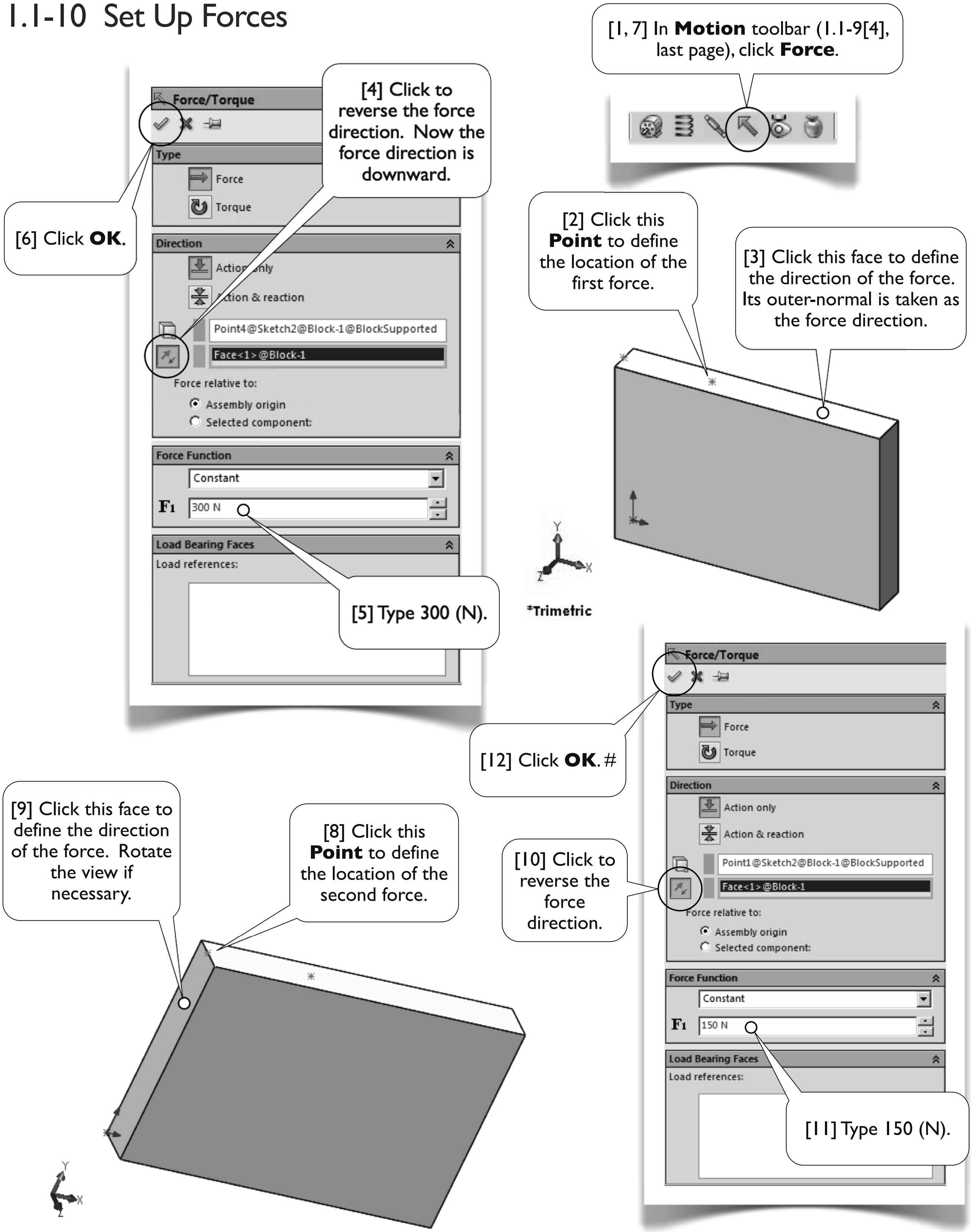
[7] You may double-click and change to a name that makes more sense to you. We, however, stick to this default name.

[8] By default, **Animation** is the **Type of Study**. It has limited capability for motion simulation.



[9] Select **Motion Analysis**, which provides full capability for rigid-body mechanical analysis. In this book, we always select **Motion Analysis** as the **Type of Study** and remember that we'll actually perform statics analyses. #

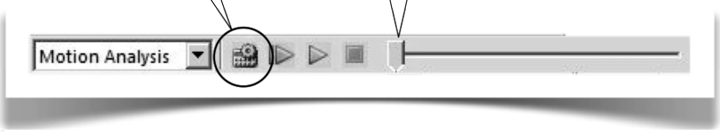
1.1-10 Set Up Forces

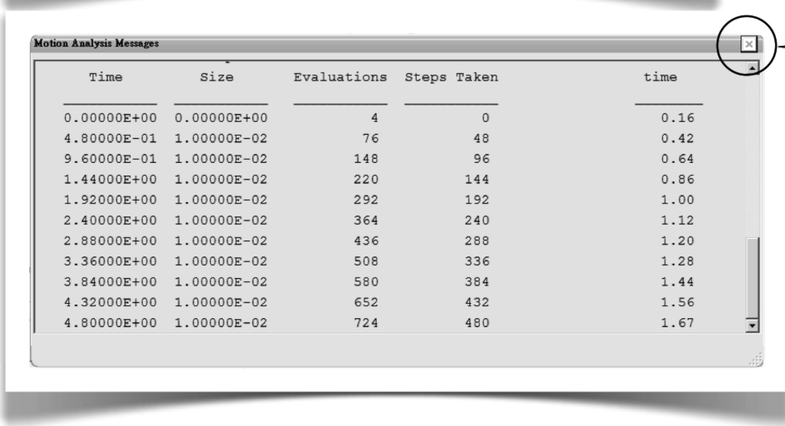


1.1-11 Calculate Results

[1] Click **Calculate**.

[3] The calculation completes in a few seconds, and the **Time Slider** proceeds to right-most position. By default, the simulation time is 5 seconds, which is an arbitrarily chosen time. Since the body is fully supported and the forces are constant, the results are time independent. We're in effect performing a static analysis. One thing you need to remember about this **Time Slider** is that whenever you want to re-define the forces, make sure the **Time Slider** is at the beginning (the left-most position). #





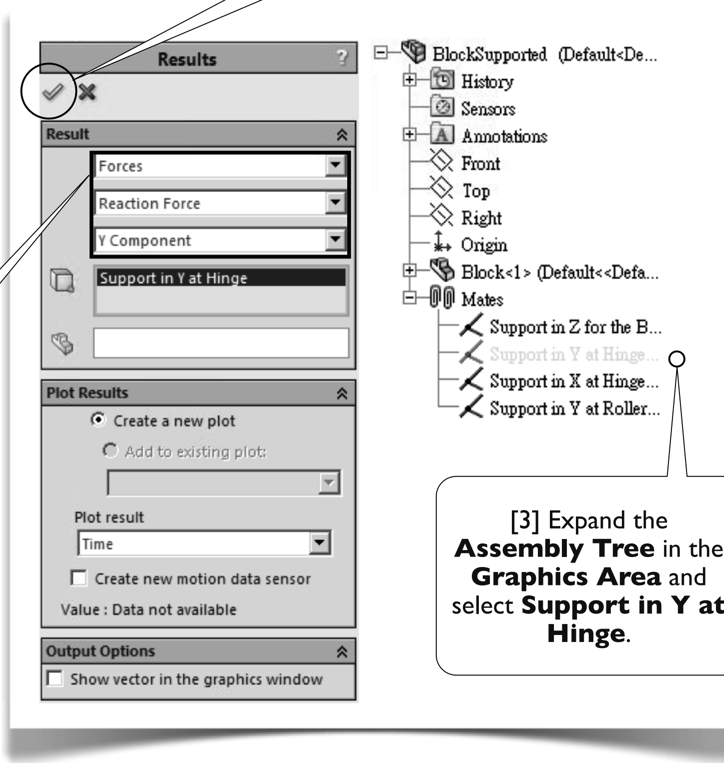
Time	Size	Evaluations	Steps Taken	time
0.00000E+00	0.00000E+00	4	0	0.16
4.80000E-01	1.00000E-02	76	48	0.42
9.60000E-01	1.00000E-02	148	96	0.64
1.44000E+00	1.00000E-02	220	144	0.86
1.92000E+00	1.00000E-02	292	192	1.00
2.40000E+00	1.00000E-02	364	240	1.12
2.88000E+00	1.00000E-02	436	288	1.20
3.36000E+00	1.00000E-02	508	336	1.28
3.84000E+00	1.00000E-02	580	384	1.44
4.32000E+00	1.00000E-02	652	432	1.56
4.80000E+00	1.00000E-02	724	480	1.67

[2] If a **Motion Analysis Messages** window appears, close it. In this book, always close this window right after calculating results.

1.1-12 View the Results

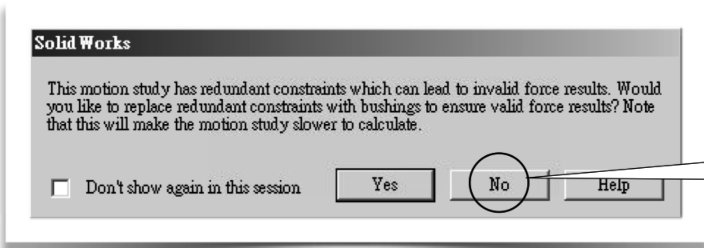
[4] Click **OK**.

[1] In **Motion** toolbar, click **Results and Plots**.

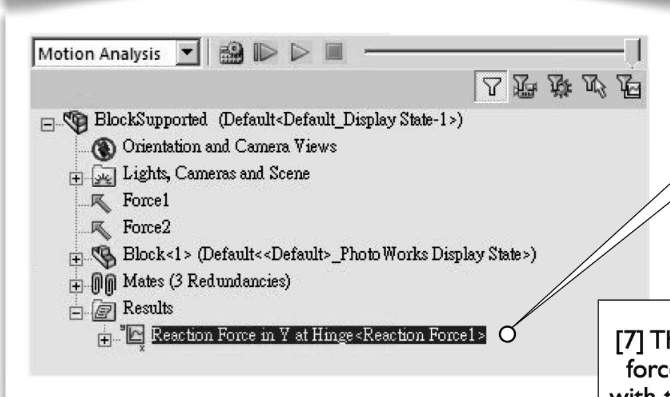


[2] Set up **Result** like this.

[3] Expand the **Assembly Tree** in the **Graphics Area** and select **Support in Y at Hinge**.

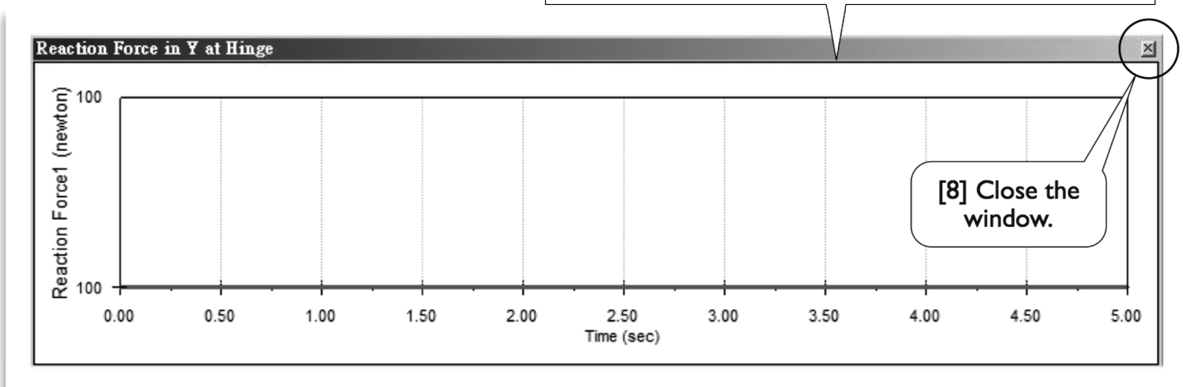


[5] Click **No**. In this book, always click **No** if this message appears.

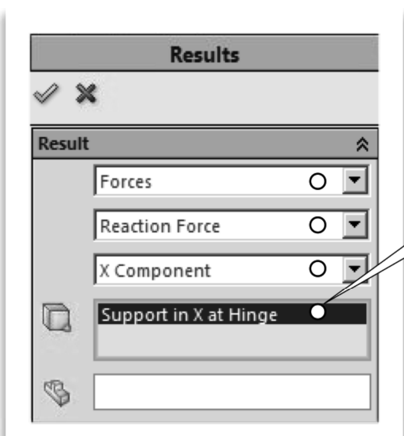


[6] In the **Motion Study Tree**, expand **Results**, click **Plot1** twice (not double-click) and change name to **Reaction Force in Y at Hinge**.

[7] The plot shows that the magnitude of the reaction force in Y-direction at the hinge is 100 N, consistent with the one calculated in 1.1-1[1] (page 5). Note that the reaction force is constant over time.

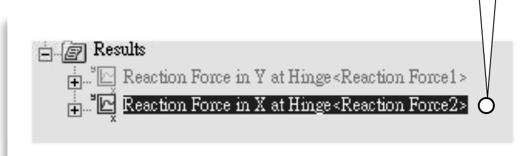


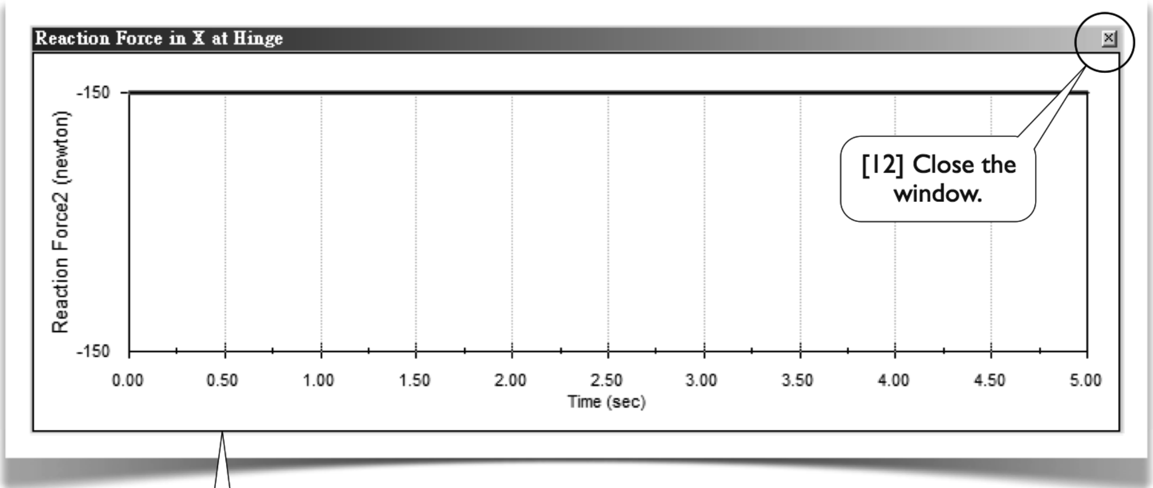
[8] Close the window.



[9] Repeat steps [1-5] except selecting **X Component** in step [2] and **Support in X at Hinge** in step [3],

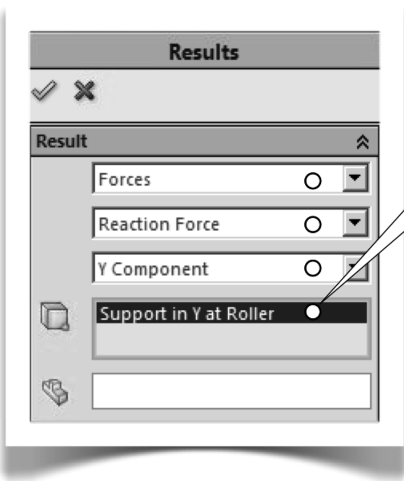
[10] In the **Motion Study Tree**, click **Plot2** twice and change name to **Reaction Force in X at Hinge**.



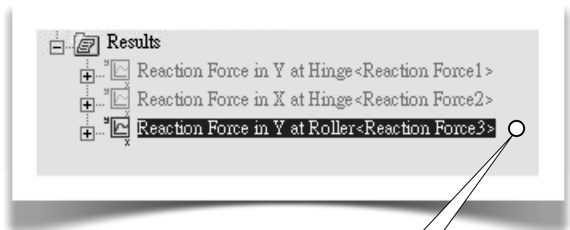


[12] Close the window.

[11] The plot shows that the magnitude of the reaction force in X-direction at the hinge is -150 N, consistent with the one calculated in 1.1-1[1] (page 5).

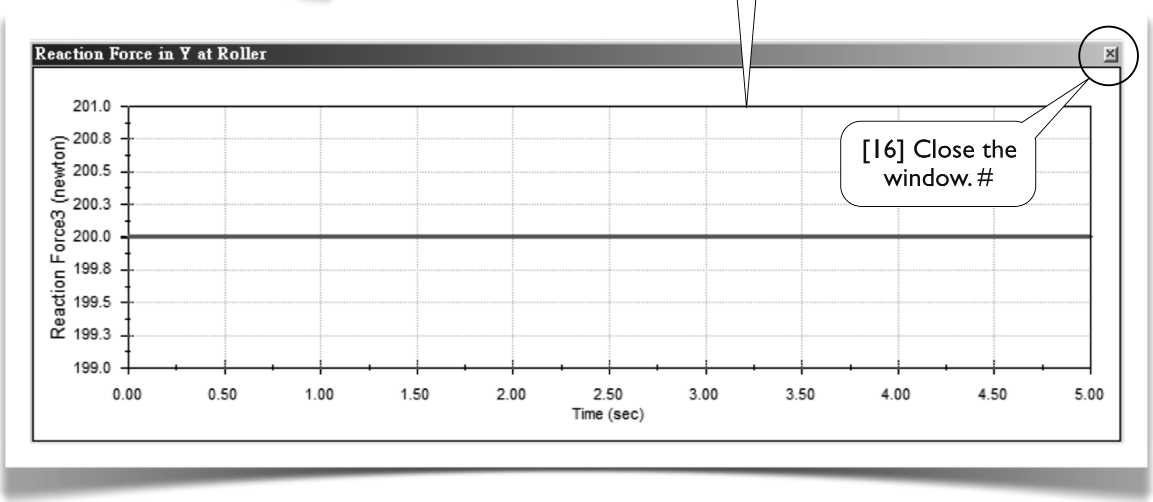


[13] Repeat steps [1-5] except selecting **Support in Y at Roller** in step [3],



[14] In the **Motion Study Tree**, click **Plot3** twice and change name to **Reaction Force in Y at Roller**.

[15] The plot shows that the magnitude of the reaction force in Y-direction at the roller is 200 N, consistent with the one calculated in 1.1-1[1] (page 5).



[16] Close the window. #

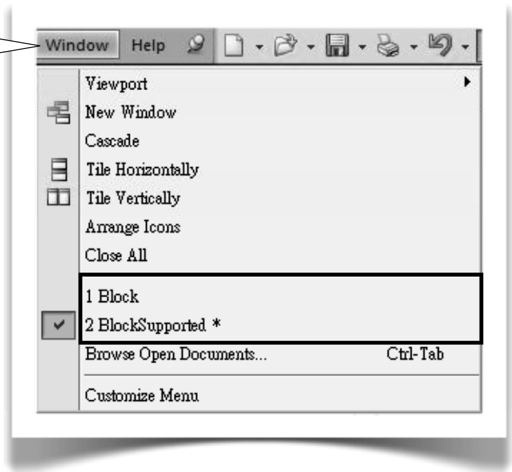
1.1-13 Do It Yourself: Validation of the Results

Do It Yourself

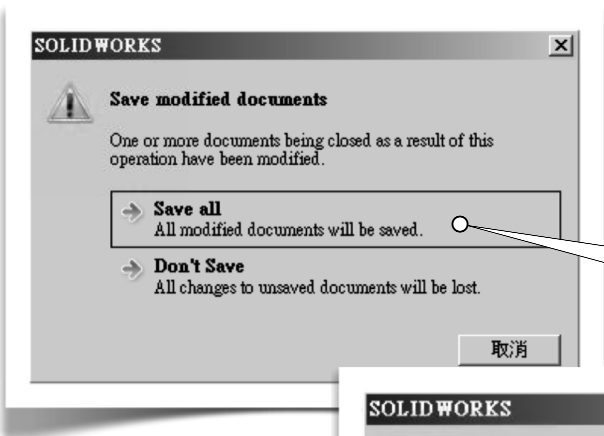
[1] To verify the validity of the results, you may check the force and moment equilibria for the body. In a 2D problem like this, you need to check 3 equilibrium equations to conclude the validity of the results. Of course, the 3 equilibrium equations must be independent. #

1.1-14 Wrap Up

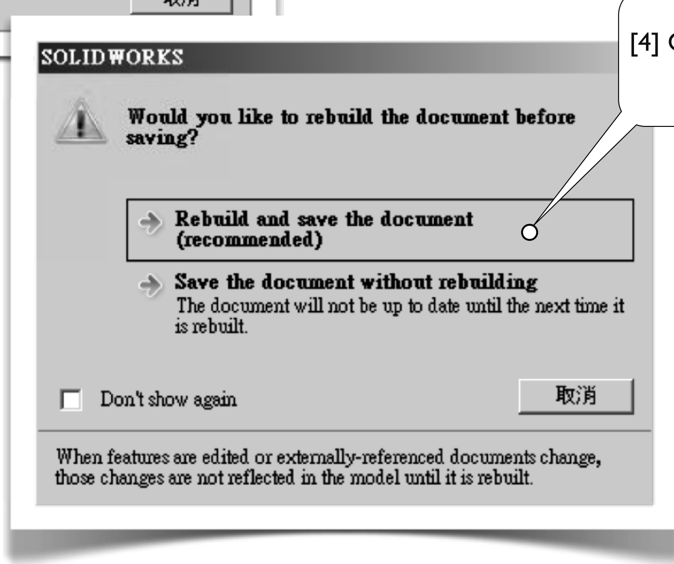
[1] From the **Pull-Down Menus**, click **Window** to see that there are two opened files: **Block** and **BlockSupported**.



[2] From the **Pull-Down Menus**, Select **File>Exit** to quit **SOLIDWORKS**.



[3] Click **Save all**.



[4] Click **Rebuild and save the document**. #

Section 1.2

Supported L-Plate: A 3D Case

1.2-1 Introduction

[1] Consider an L-shaped plate of thickness 5 mm [2] supported at three corners [3-5] and subject to a force *P* [6]. We want to find the reaction forces at the supports.

There are six reaction force components in this problem, namely D_X, D_Y, D_Z, B_X, B_Z, and C_Y. It is possible to establish six equations, according to force and moment equilibria, and solve these six reaction forces. However, we may calculate C_Y directly by considering the moment equilibrium about the axis passing through *B* and *D*,

$$\sum M_{BD} = 0$$

$$\vec{\lambda}_{BD} \cdot (\vec{r}_{A/B} \times \vec{P}) + \vec{\lambda}_{BD} \cdot (\vec{r}_{C/D} \times \vec{C}_Y) = 0$$

where the unit vector along *BD*,

$$\vec{\lambda}_{BD} = \frac{(-8\text{ cm})\vec{i} + (-9\text{ cm})\vec{j} + (12\text{ cm})\vec{k}}{\sqrt{(6^2 + 9^2 + 12^2)}\text{ cm}}$$

the position vectors,

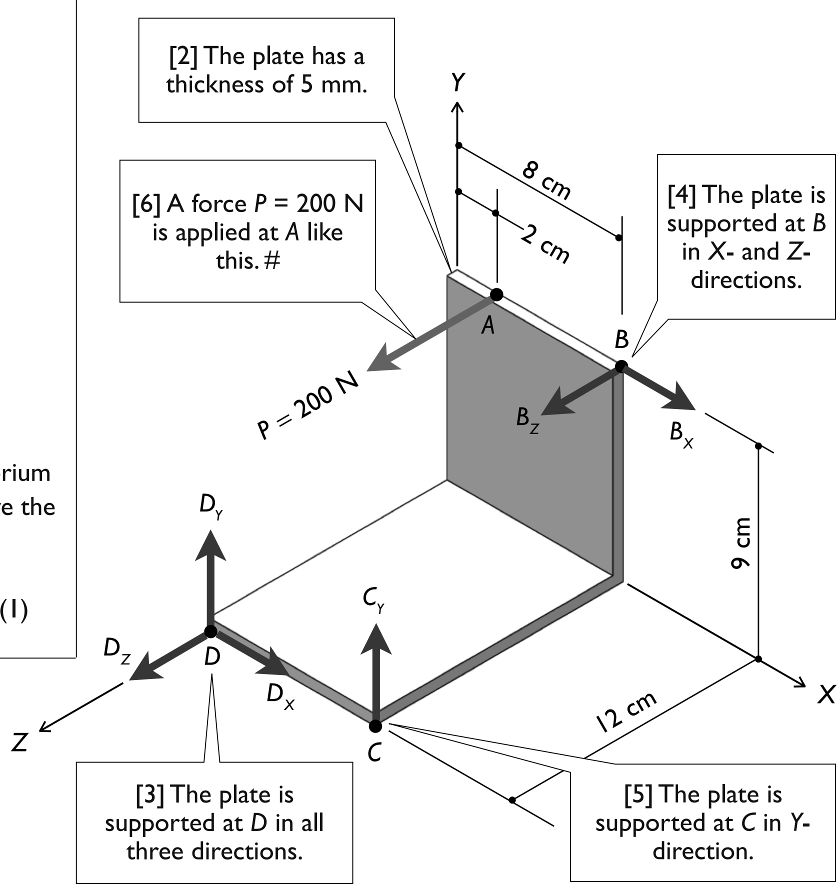
$$\vec{r}_{A/B} = (-6\text{ cm})\vec{i},\ \vec{r}_{C/D} = (8\text{ cm})\vec{i}$$

and the forces,

$$\vec{P} = (200\text{ N})\vec{k},\ \vec{C}_Y = (C_Y)\vec{j}$$

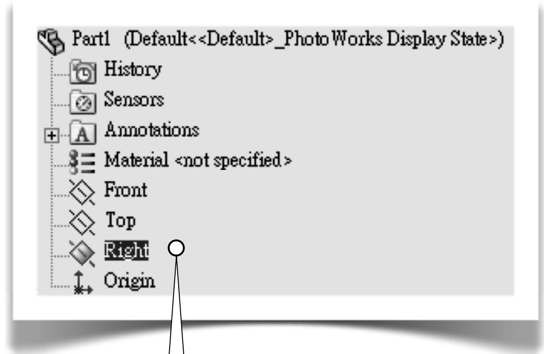
Substituting these into the moment equilibrium equation, and after mild calculation, we have the reaction force in *Y* at *C*,

$$C_Y = 112.5\text{ N} \qquad (1)$$

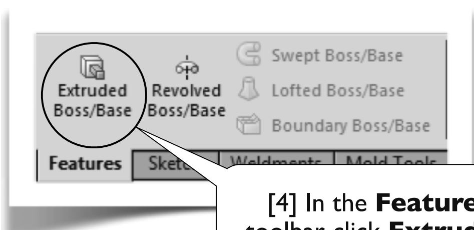


1.2-2 Start Up and Create a Part: **Plate**

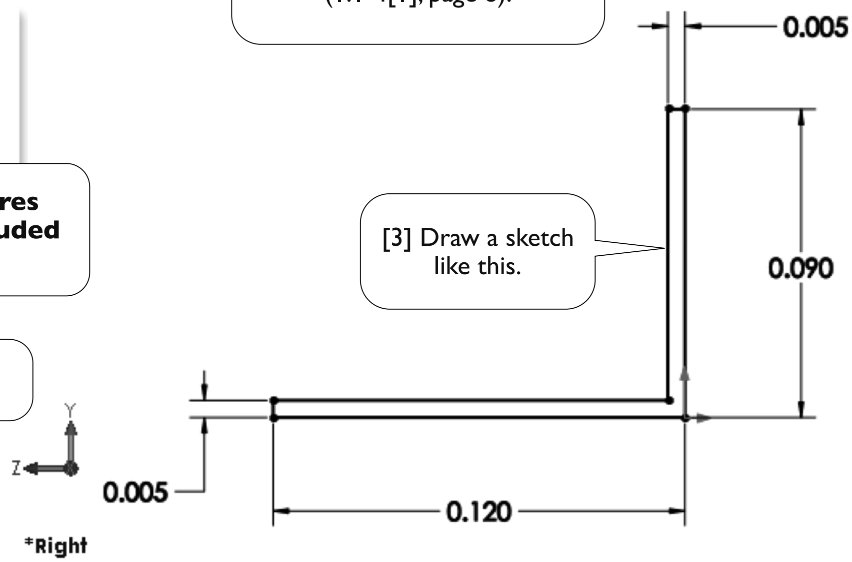
[1] Launch **SOLIDWORKS** and click **New** to create a new **Part** (1.1-2[1-5], page 6). Select **MKS** unit system with three decimal places for the length unit (1.1-3[1-4], page 7).



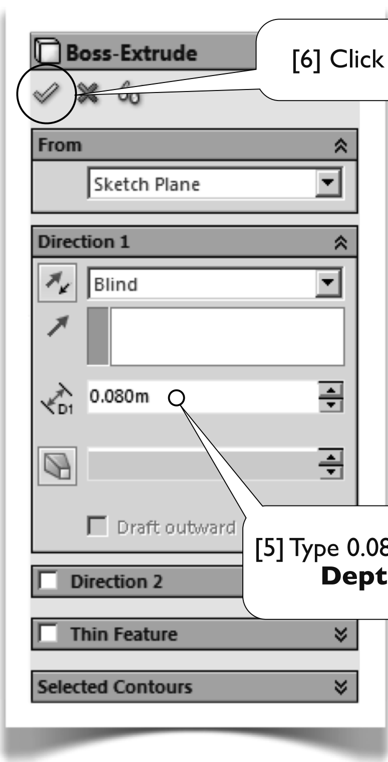
[2] In the **Part Tree**, right-click **Right** plane and select **Sketch** (1.1-4[1], page 8).



[4] In the **Features** toolbar, click **Extruded Boss/Base**.



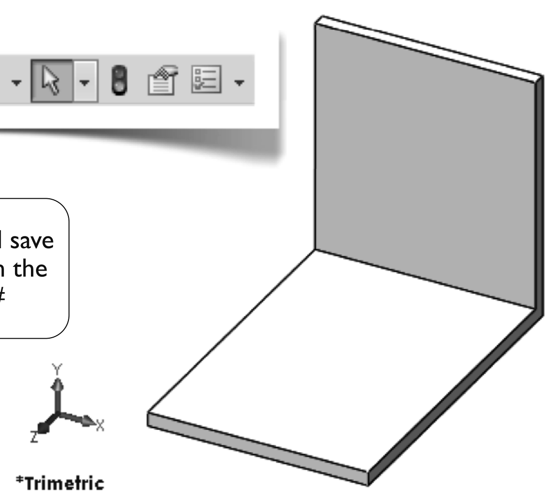
[3] Draw a sketch like this.



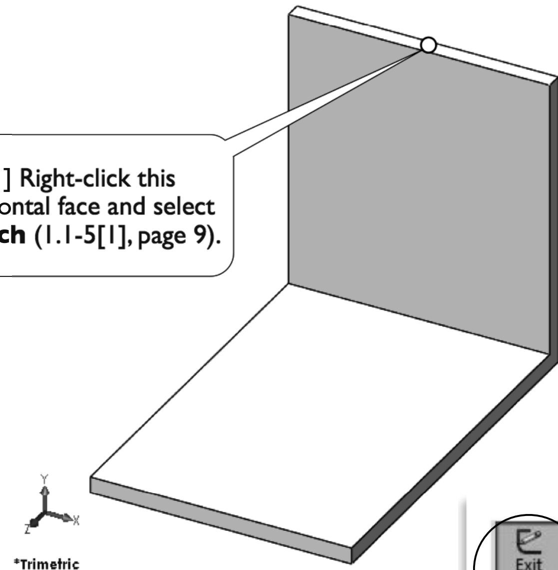
[6] Click **OK**.

[5] Type 0.08 (m) for **Depth**.

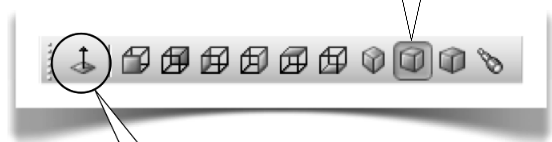
[7] Click **Save** and save the document with the name **Plate**. #



1.2-3 Create a **Point** on the **Plate**

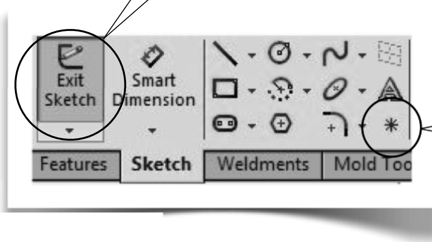


[1] Right-click this horizontal face and select **Sketch** (1.1-5[1], page 9).



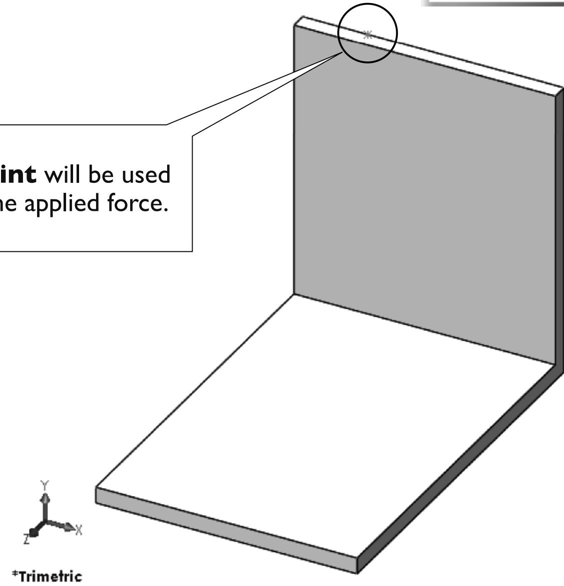
[2] In the **Standard Views** toolbar, click **Normal To**.

[6] Click **Trimetric**.

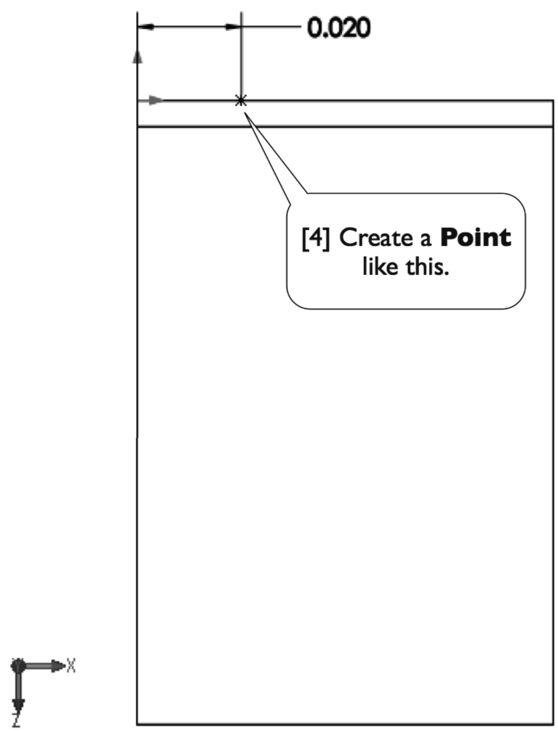


[5] Click **Exit Sketch**.

[3] In **Sketch** toolbar, select **Point**.



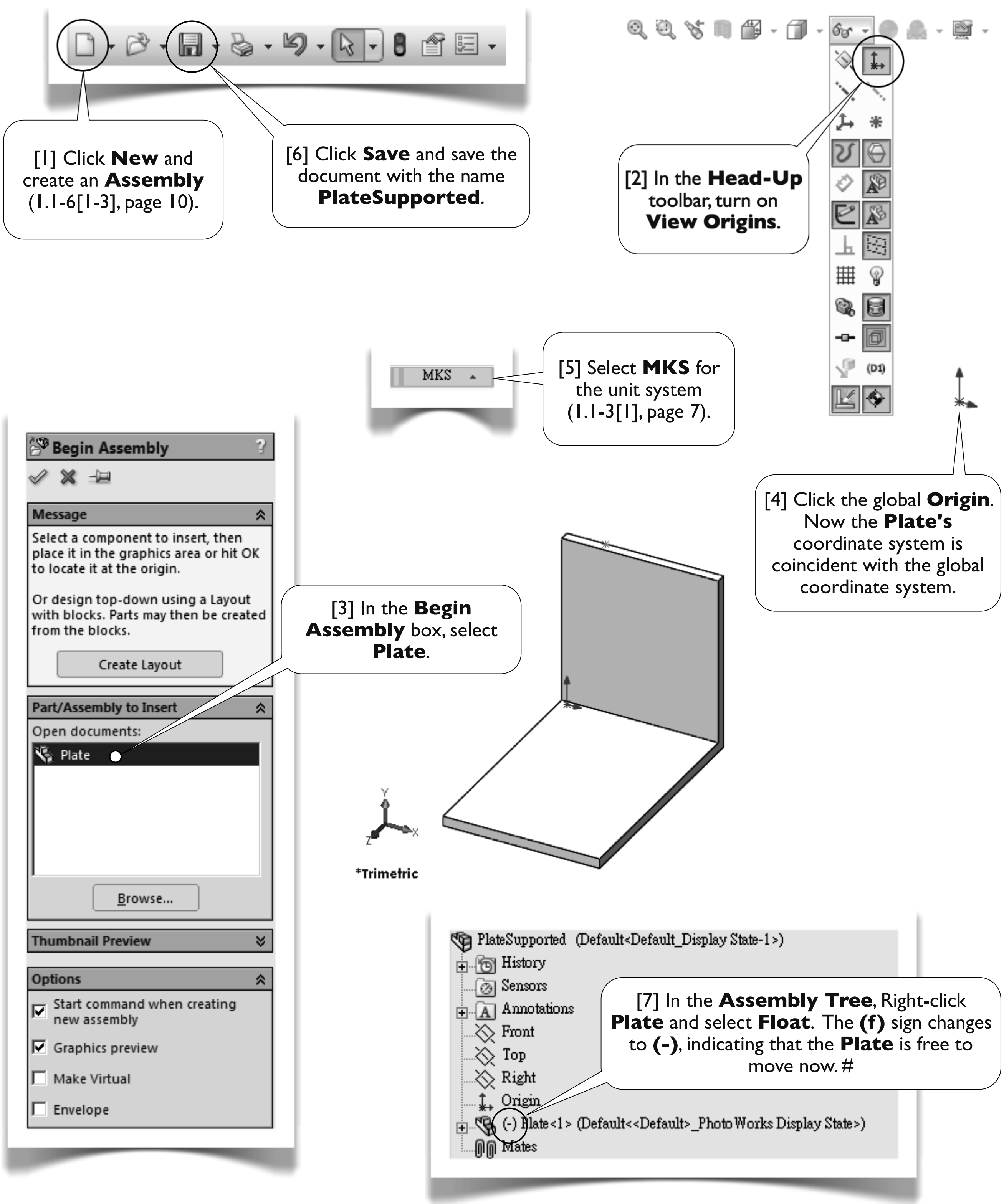
[7] This **Point** will be used to locate the applied force.



[4] Create a **Point** like this.

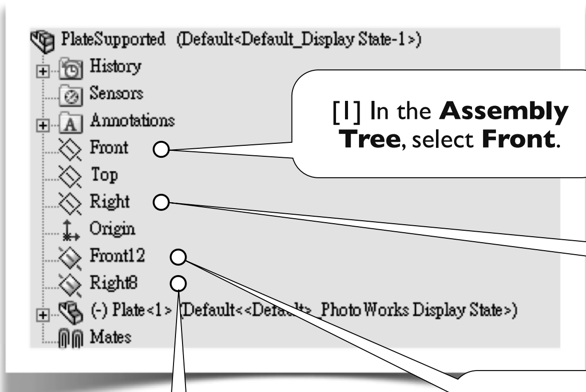
[8] Click **Save**. #

1.2-4 Create an Assembly: **PlateSupported**



1.2-5 Create Planes

[4] Click **OK**. A plane parallel to **Front** plane with distance +0.12 m is created.

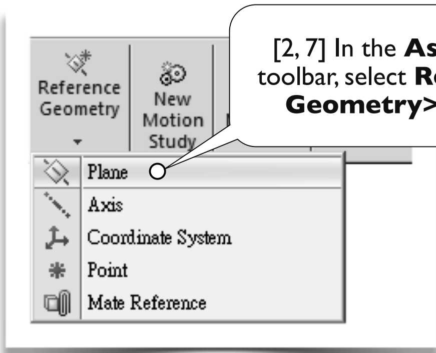


[1] In the **Assembly Tree**, select **Front**.

[6] Select **Right**.

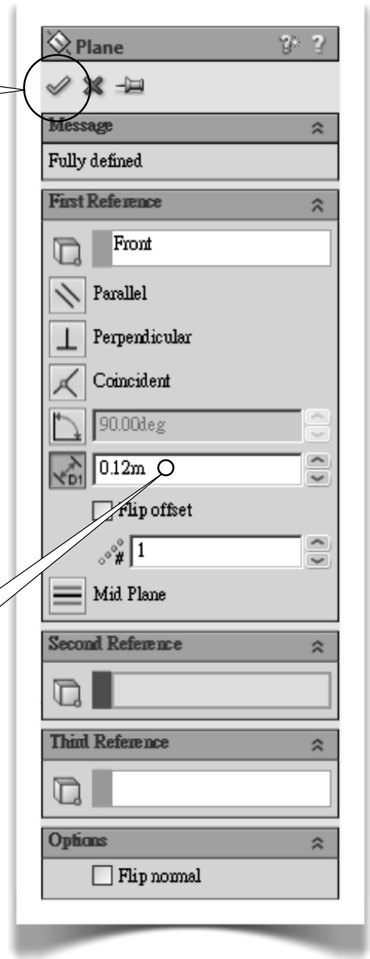
[5] Rename the newly created plane as **Front12**.

[10] Rename the newly created plane as **Right8**.



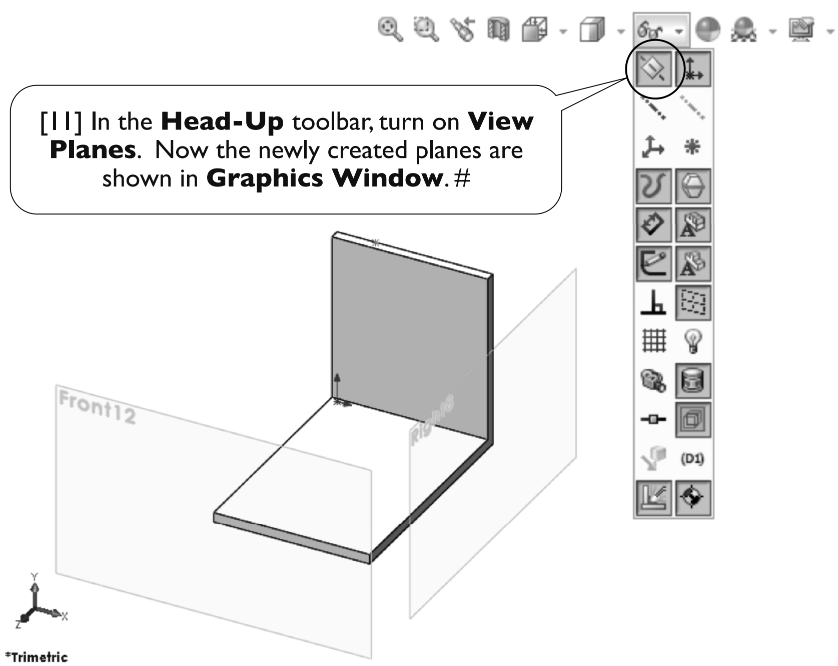
[2, 7] In the **Assembly** toolbar, select **Reference Geometry>Plane**.

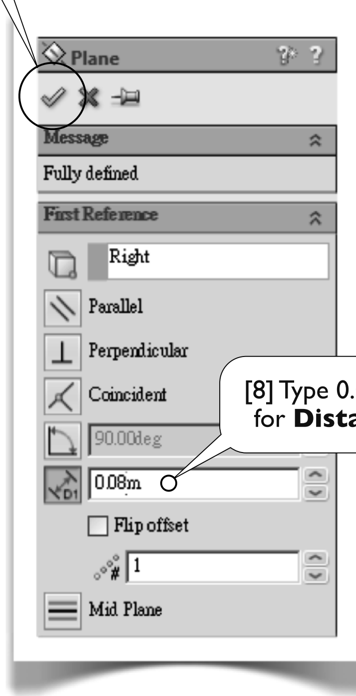
[3] Type 0.12 (m) for **Distance**.



[9] Click **OK**. A plane parallel to **Right** plane with distance +0.08 m is created.

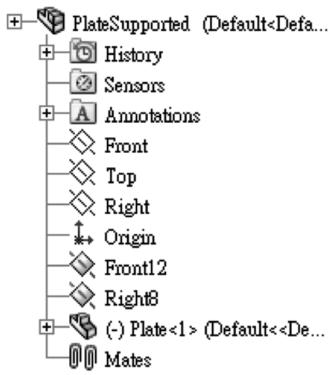
[11] In the **Head-Up** toolbar, turn on **View Planes**. Now the newly created planes are shown in **Graphics Window**. #

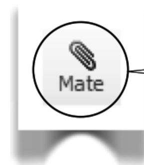




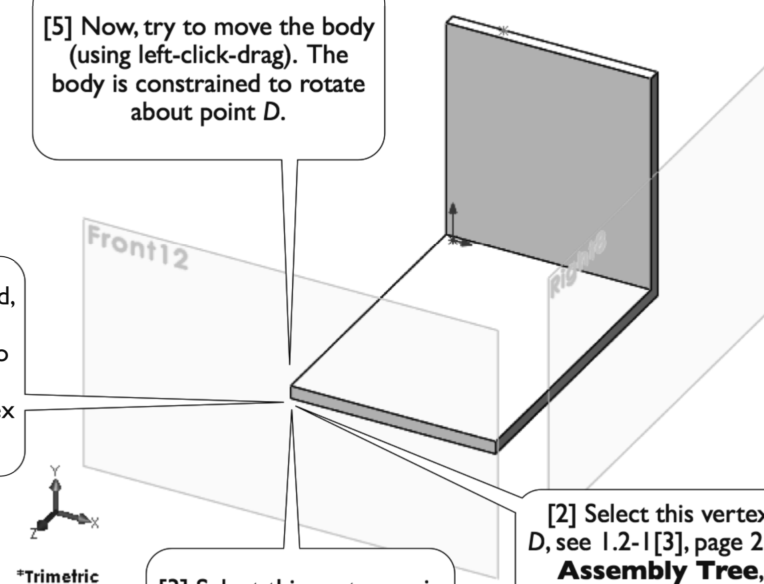
[8] Type 0.08 (m) for **Distance**.

1.2-6 Set Up Supports





[1] In **Assembly** toolbar, click **Mate**.

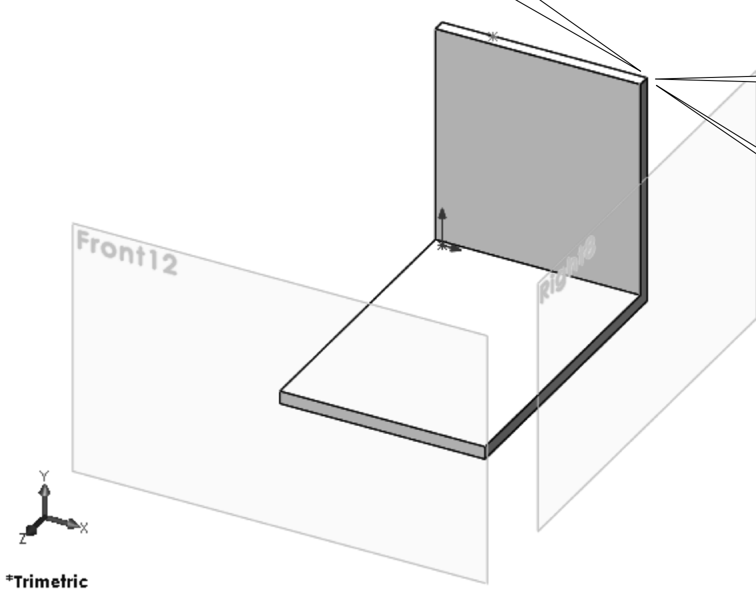


[2] Select this vertex (point *D*, see 1.2-1[3], page 21) and, in **Assembly Tree**, select global **Right** plane. Click **OK** to create **Coincident1**. This sets up a support at this vertex in *X*-direction.

[3] Select this vertex again and, in **Assembly Tree**, select global **Top** plane. Click **OK** to create **Coincident2**. This sets up a support at this vertex in *Y*-direction.

[4] Select this vertex again and, in **Assembly Tree**, select **Front12** plane. Click **OK** to create **Coincident3**. This sets up a support at this vertex in *Z*-direction.

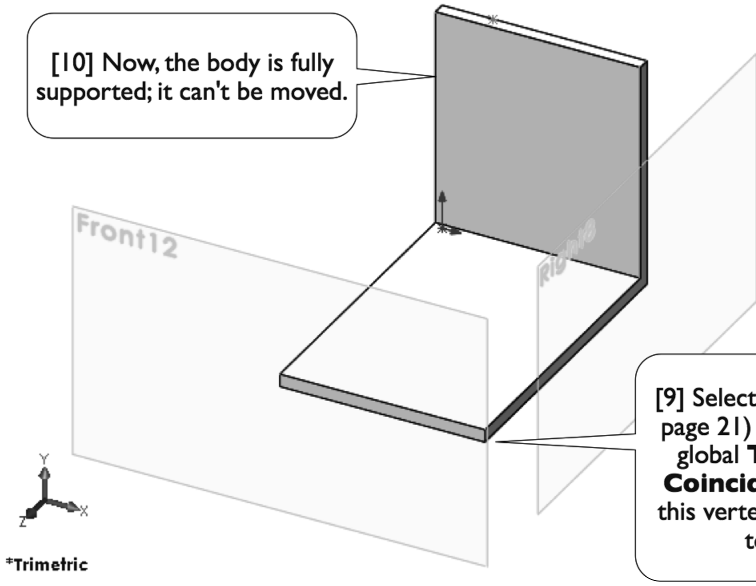
[5] Now, try to move the body (using left-click-drag). The body is constrained to rotate about point *D*.



[6] Select this vertex (point *B*, see 1.2-1[4], page 21) and, in **Assembly Tree**, select **Right8** plane. Click **OK** to create **Coincident4**. This sets up a support at this vertex in *X*-direction.

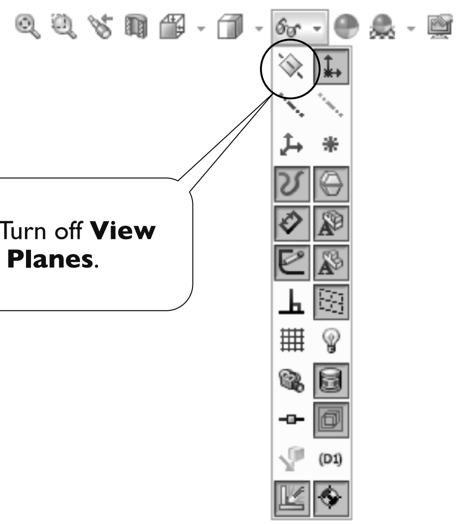
[7] Select this vertex again and, in **Assembly Tree**, select global **Front** plane. Click **OK** to create **Coincident5**. This sets up a support at this vertex in *Z*-direction.

[8] Now, try to move the body (using left-click-drag). The body is constrained to rotate about the axis passing through *BD*.

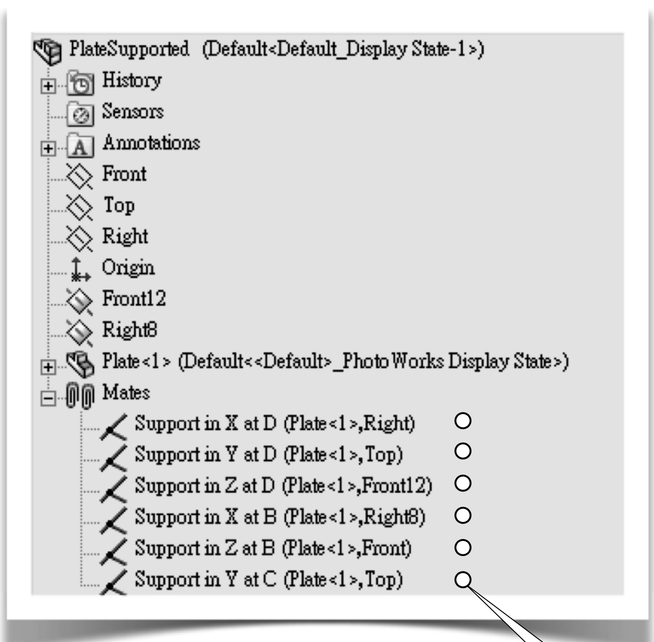


[10] Now, the body is fully supported; it can't be moved.

[9] Select this vertex (point *C*, see 1.2-1[5], page 21) and, in **Assembly Tree**, select global **Top** plane. Click **OK** to create **Coincident6**. This sets up a support at this vertex in *Z*-direction. Click **OK** again to dismiss the **Mate** box.



[11] Turn off **View Planes**.

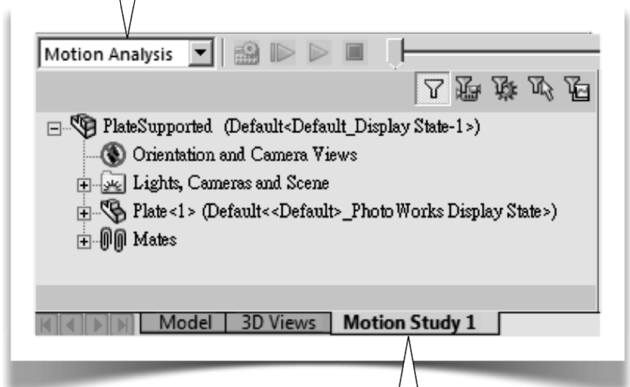


[12] In the **Assembly Tree**, expand **Mates** and rename the six **Coincident** mates like this (for better readability). #

1.2-7 Create a **Study**

[1] Load **SOLIDWORKS Motion** if it is not loaded yet (1.1-8, page 14).

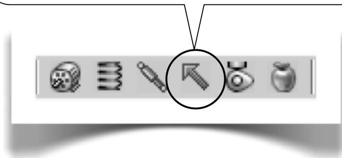
[3] Select **Motion Analysis** (1.1-9[8, 9], page 15). #



[2] Click **Motion Study 1** tab to create a new **Study**.

1.2-8 Set Up Forces

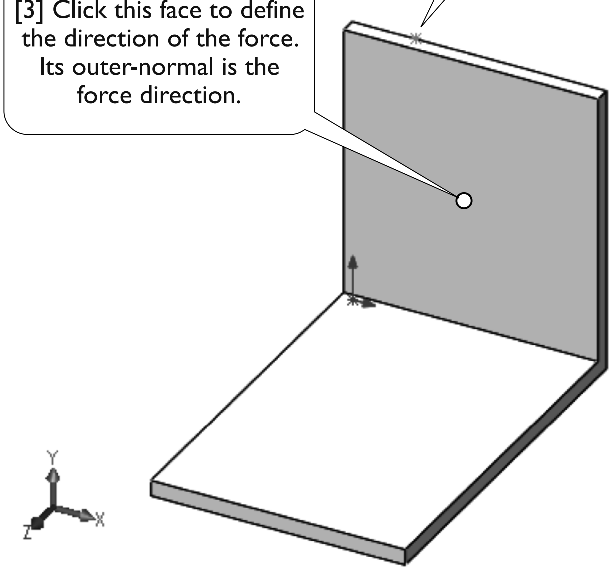
[1] In **Motion** toolbar, click **Force**.



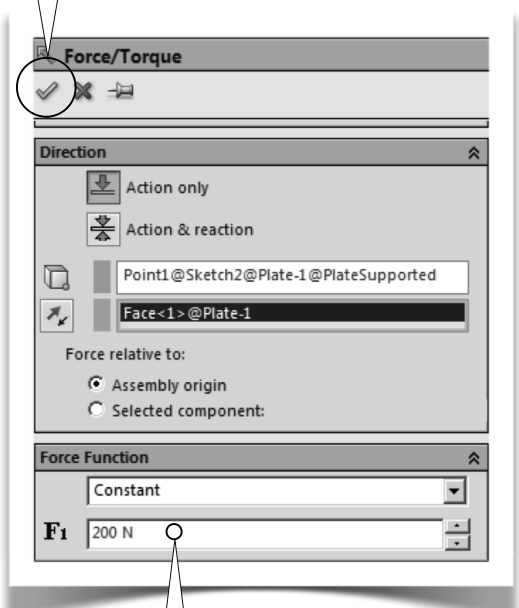
[5] Click **OK**. #

[2] Click this **Point** to define the location of the force.

[3] Click this face to define the direction of the force. Its outer-normal is the force direction.



*Trimetric



[4] Type 200 (N).

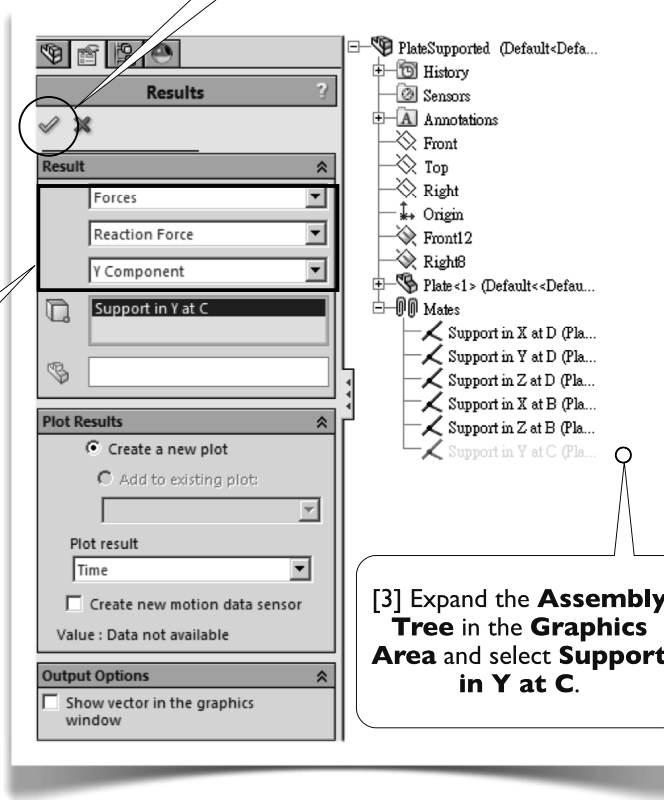
1.2-9 Calculate Results

[1] Click **Calculate**. Remember that, If a **Motion Analysis Messages** window appears, close it. #

1.2-10 Retrieve the Reaction Force at *C*

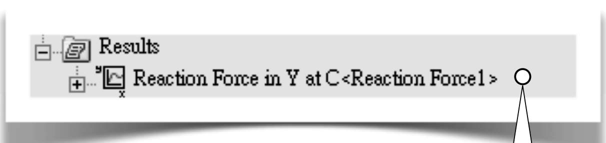
[1] In **Motion** toolbar, click **Results and Plots**.

[4] Click **OK**.

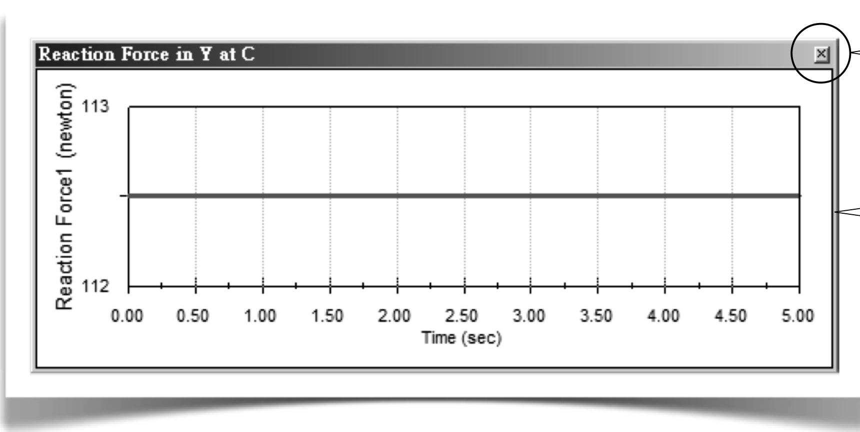


[2] Set up **Result** like this.

[3] Expand the **Assembly Tree** in the **Graphics Area** and select **Support in Y at C**.



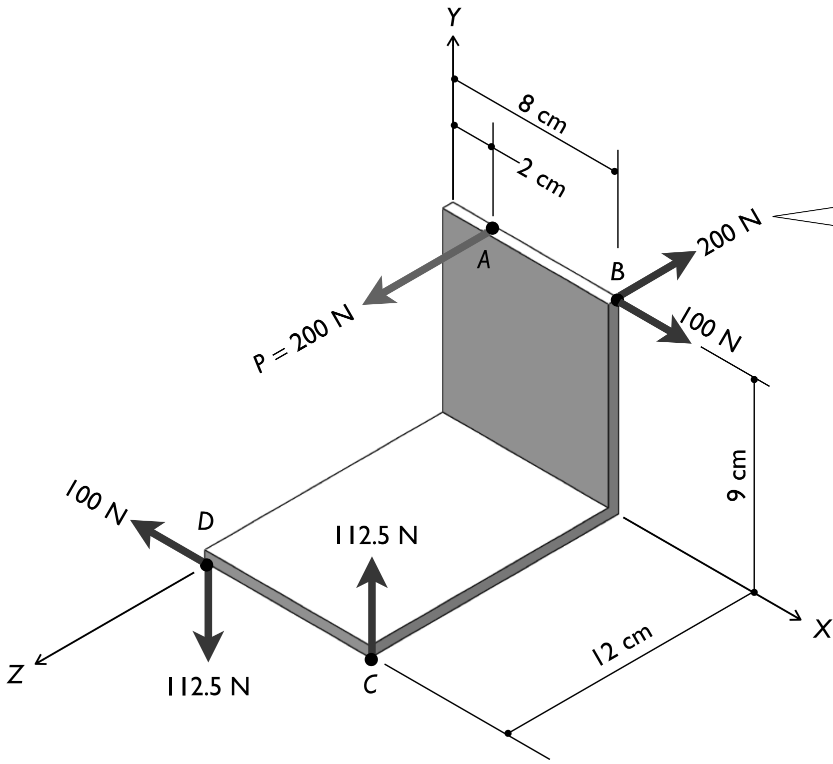
[5] In the **Motion Study Tree**, Rename **Plot1** as **Reaction Force in Y at C**.



[7] Close the window. #

[6] The plot shows that the reaction force in Y-direction at *C* is +112.5 N, consistent with the value in Eq. 1.2-1(1) (page 21).

1.2-11 Do It Yourself: Other Reaction Forces and Validation of the Results



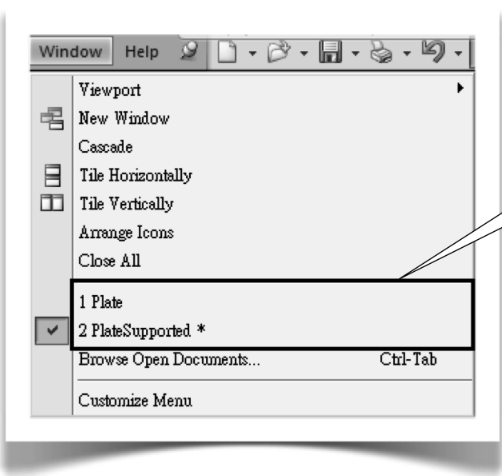
Do It Yourself

[1] It leaves you to retrieve other reaction forces. All reaction forces are summarized like this.

Do It Yourself

[2] To verify the validity of the results, we may check the force and moment equilibria for the body. A moment equilibrium can be with respect to a point or an axis. You need to check 6 equilibrium equations to conclude the validity of the results. The 6 equilibrium equations must be independent. #

1.2-12 Wrap Up



[1] From the **Pull-Down Menus**, click **Window** to see that there are two opened files.

[2] From the **Pull-Down Menus**, Select **File>Exit** to quit **SOLIDWORKS**. Click **Save all.** Click **Rebuild and save the document**. (See 1.1-14[2-4], page 20) #