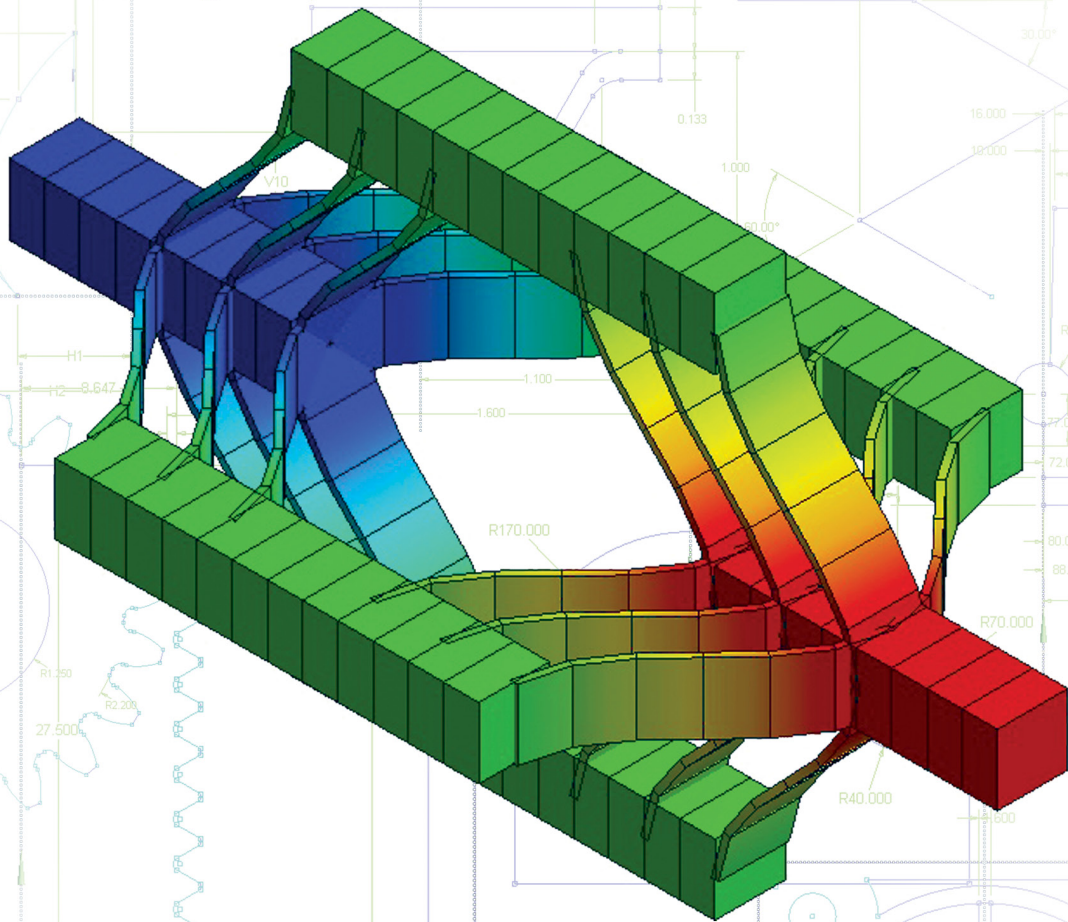


Finite Element Simulations with ANSYS® Workbench™ 14

Theory, Applications, Case Studies



Huei-Huang Lee



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Chapter 2

Sketching

A simulation project starts with the creation of a geometric model. To be successful on simulations, an engineer has to be proficient at geometric modeling first. In a simulation project, it is not uncommon to take majority of human-hours to create a geometric model; it is particularly true in a 3D simulation.

A complex 3D geometry can be viewed as a series of boolean operations (unions, subtractions, etc.) of simpler 3D solid bodies. Each solid body is often created by first drawing a sketch on a plane; the sketch is then used to generate the 3D solid body by using a tool such as <Extrude>, <Revolve>, <Sweep>, etc. In turn, to be proficient at 3D bodies creation, an engineer has to be proficient at sketching first.

Purpose of the Chapter

The purpose of this chapter is to provide exercises for the students so that they can be proficient at sketching using <DesignModeler>. Profiles of five mechanical parts are created in this chapter. Each sketch is used to generate a 3D model using a 3D tool of either <Extrude> or <Revolve>. The use of the 3D tools is trivial enough that we should be able to focus on the 2D sketches.

About Each Section

Each sketch of a mechanical part will be completed in a section. Sketches in the first two sections are guided in a step-by-step fashion. Section 1 sketches a cross section of W16x50; the cross section is then extruded to generate a 3D beam. Section 2 sketches a triangular plate; the sketch is then extruded to generate a 3D solid model.

Section 3 does not mean to provide a hands-on case. It overviews the sketching tools in a systematic way, attempting to complement what were missed in the first two sections.

Sections 4, 5, and 6 provide three cases for more exercises. We will present in a not-so-step-by-step fashion; we purposely leave some room for the students to figure out the details.

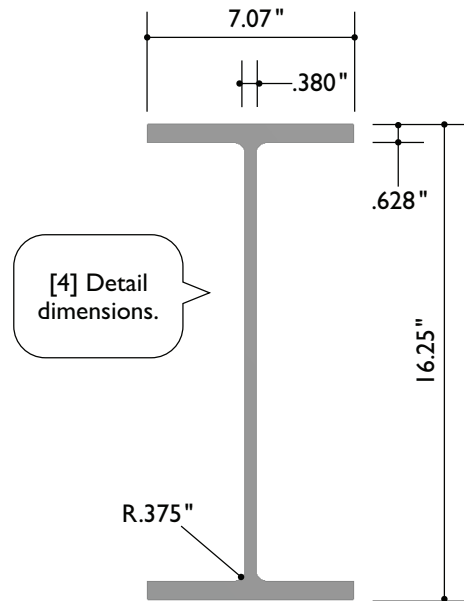
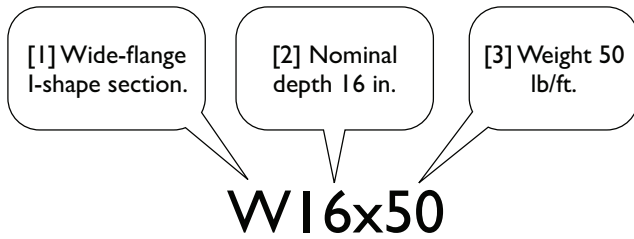
Section 2.1

Step-by-Step: W16x50 Beam



2.1-1 About the W16x50 Beam

In this section, we will create a 3D solid body for a steel beam. The steel beam has a W16x50 cross-section [1-4] and a length of 10 ft.



2.1-2 Start Up <DesignModeler>

[1] Launch Workbench.

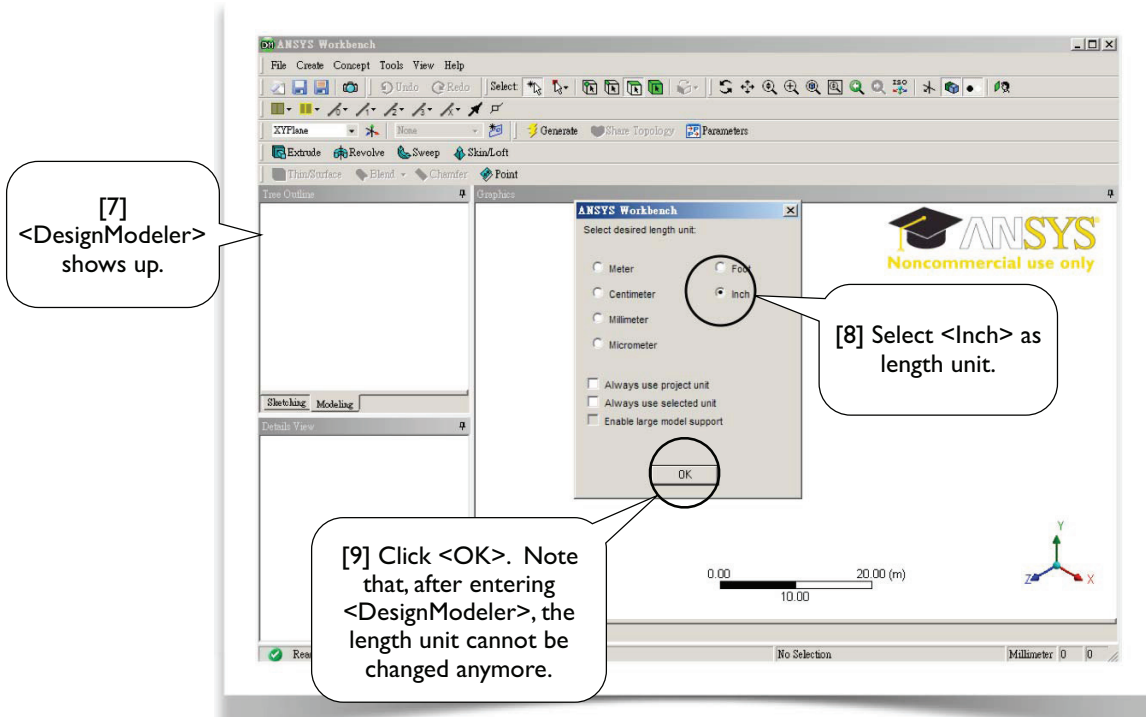
[2] <Workbench GUI> shows up.

[3] Click the plus sign (+) to expand <Component Systems>. The plus sign becomes minus sign.

[4] Double-click <Geometry> to create a system in <Project Schematic>.

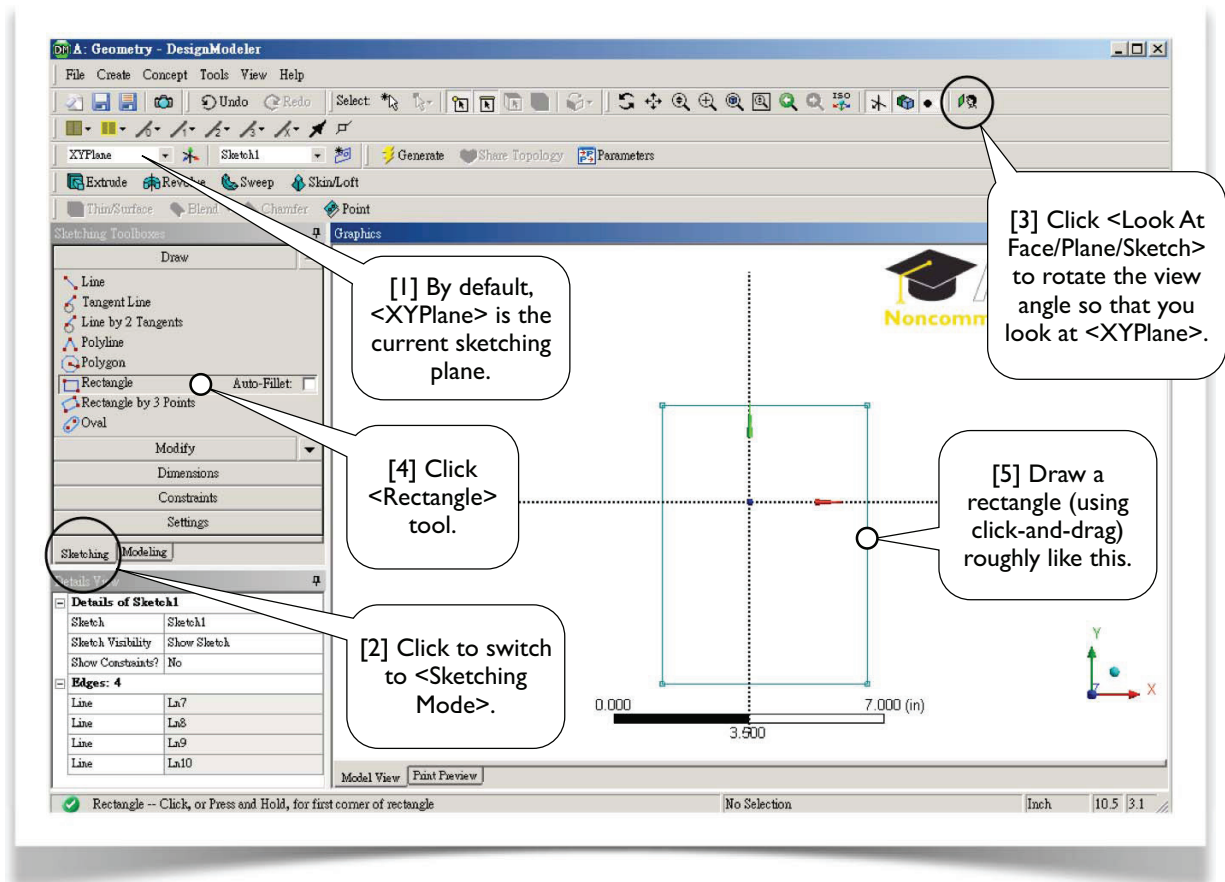
[5] You may click here to show the messages from ANSYS Inc. To hide the message, click again.

[6] Double-click <Geometry> to start up <DesignModeler>, the geometry editor.



Notes: In this book, when a circle is used with a speech bubble, it is to indicate that mouse or keyboard ACTIONS are needed in that step [1, 3, 4, 6, 8, 9]. A circle may be filled with white color [1, 4, 6] or unfilled [3, 8, 9]. A speech bubble without a circle [2, 7] or with a rectangle [5] is used for commentary only, i.e., no mouse or keyboard actions are needed.

2.1-3 Draw a Rectangle on <XYPlane>



Impose symmetry constraints...

[6] Click <Constraints> toolbox.

[8] Click <Symmetry> tool.

[7] If you don't see <Symmetry> tool, click here to scroll down until you see the tool.

[11] Click the horizontal axis and then two horizontal lines on both sides to make them symmetric about the horizontal axis.

[10] Right-click anywhere on the graphic area to open the context menu, and choose <Select new symmetry axis>.

[9] Click the vertical axis and then two vertical lines on both sides to make them symmetric about the vertical axis.

Specify dimensions...

[12] Click <Dimensions> toolbox.

[13] <General> is the default tool.

[17] In <Details View>, type 7.07 (in) for H1 and 16.25 (in) for V2.

[18] Click <Zoom to Fit>.

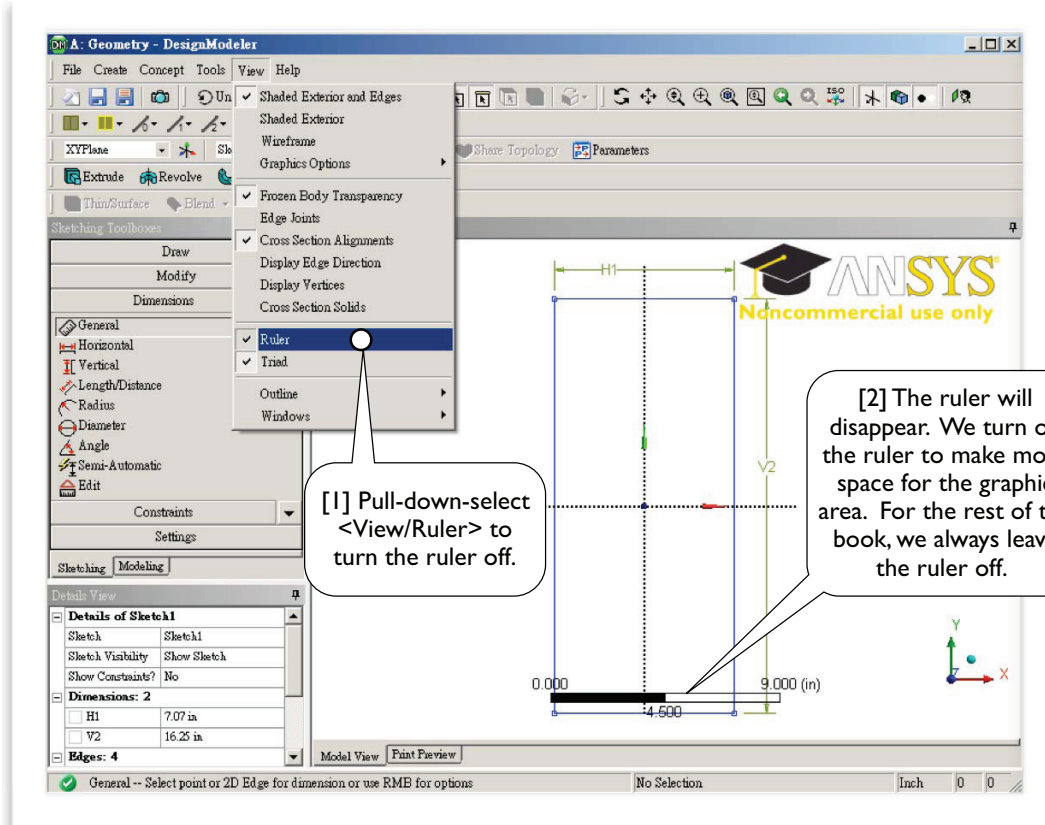
[14] Click this line, move the mouse upward, then click again to create H1.

[15] Click this line, move the mouse rightward, then click again to create V2.

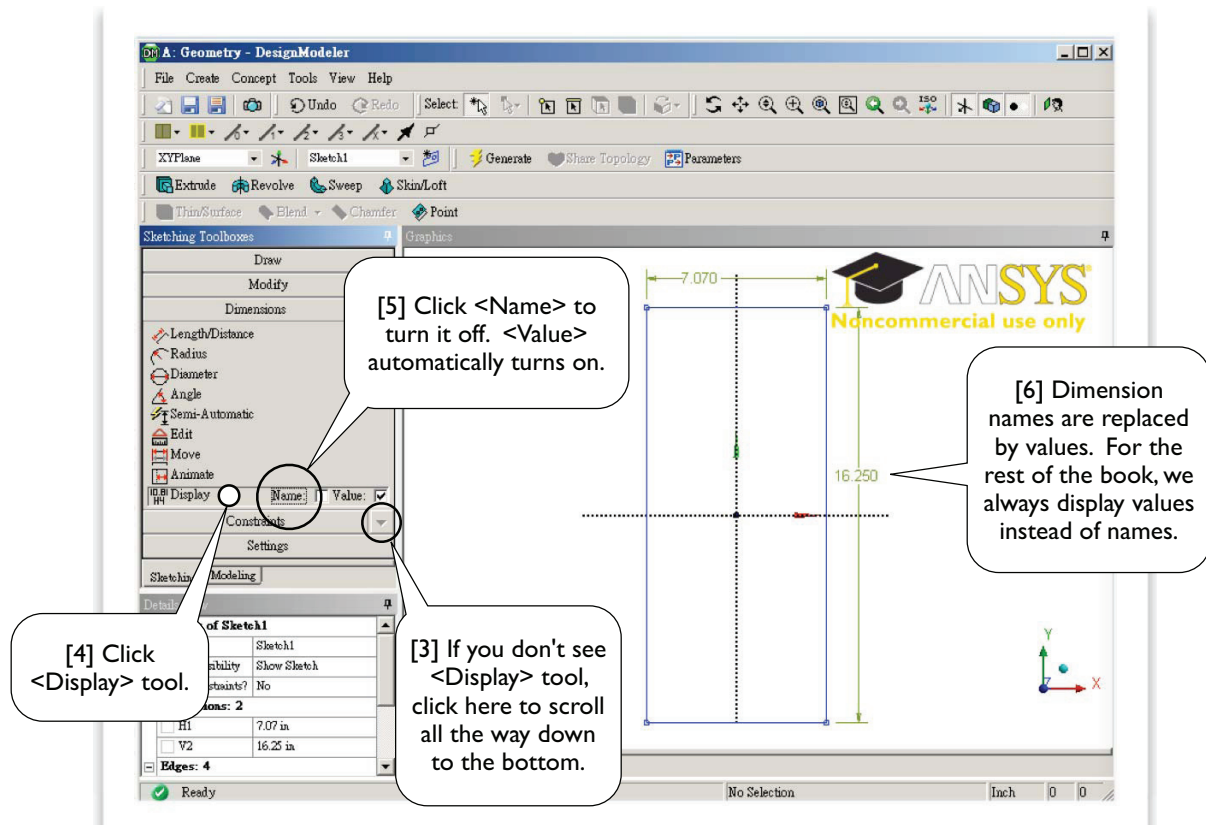
[16] All the lines turn to blue color. Colors are used to indicate the constraint status. The blue color means a geometric entity is well constrained.

2.1-4 Clean up the Graphic Area

The ruler occupies space and is sometimes annoying; let's turn it off...



Let's display dimension values (instead of names) on the graphic area...



2.1-5 Draw a Polyline

Draw a polyline; the dimensions are not important for now...

[1] Select <Draw> toolbox.

[2] Select <Polyline> tool.

[3] Click roughly here to start a polyline. Make sure a <C> (coincident) appears before clicking.

[4] Click the second point roughly here. Make sure an <H> (horizontal) appears before clicking.

[5] Click the third point roughly here. Make sure a <V> (vertical) appears before clicking.

[6] Click the last point roughly here. Make sure an <H> and a <C> appear before clicking.

[7] Right-click anywhere on the graphic area to open the context menu, and select <Open End> to end <Polyline> tool.

2.1-6 Copy the Polyline

Copy the newly created polyline to the right side, flip horizontally...

[1] Select <Modify> toolbox.

[2] Select <Copy> tool.

[3] Select the three newly created segments by control-clicking them (see [1]) one after another.

[4] Right-click anywhere on the graphic area to open the context menu, and select <End/Use Plane Origin as Handle>.

Context menu is used heavily...

[5] The tool automatically changes from <Copy> to <Paste>.

[6] Right-click to open the context menu again and select <Flip Horizontal>.

[7] Right-click to open the context menu again and select <Paste at Plane Origin>.

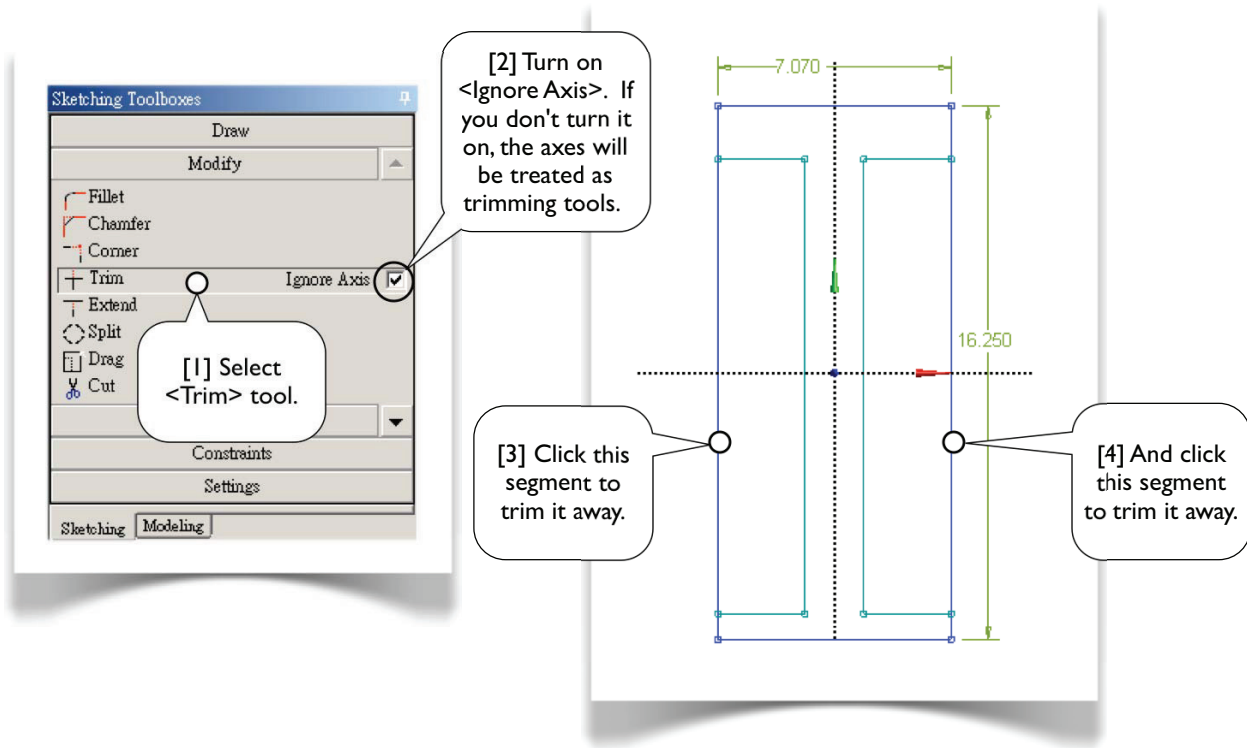
[8] Right-click to open the context menu again and select <End> to end <Copy> tool. An alternative way (and better way) is to press ESC to end a tool.

[9] The polyline has been copied.

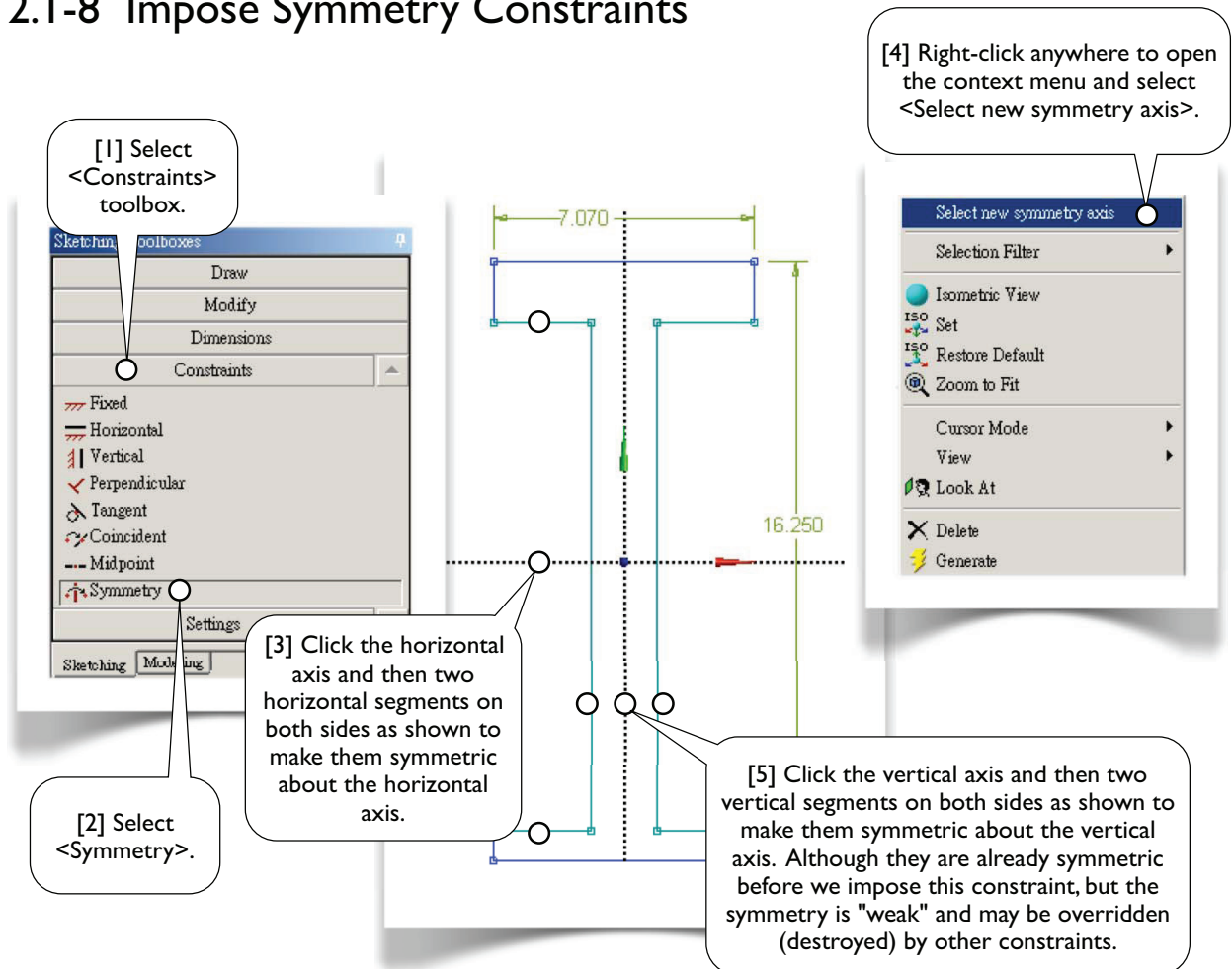
Basic Mouse Operations

- [10] Click: single selection.
- [11] Control-click: add/remove selection.
- [12] Click-sweep: continuous selection.
- [13] Right-click: open context menu.
- [14] Right-click-drag: box zoom.
- [15] Scroll-wheel: zoom in/out.
- [16] Middle-click-drag: rotation. Shift-middle-click-drag: zoom in/out. Control-middle-click-drag: pan.

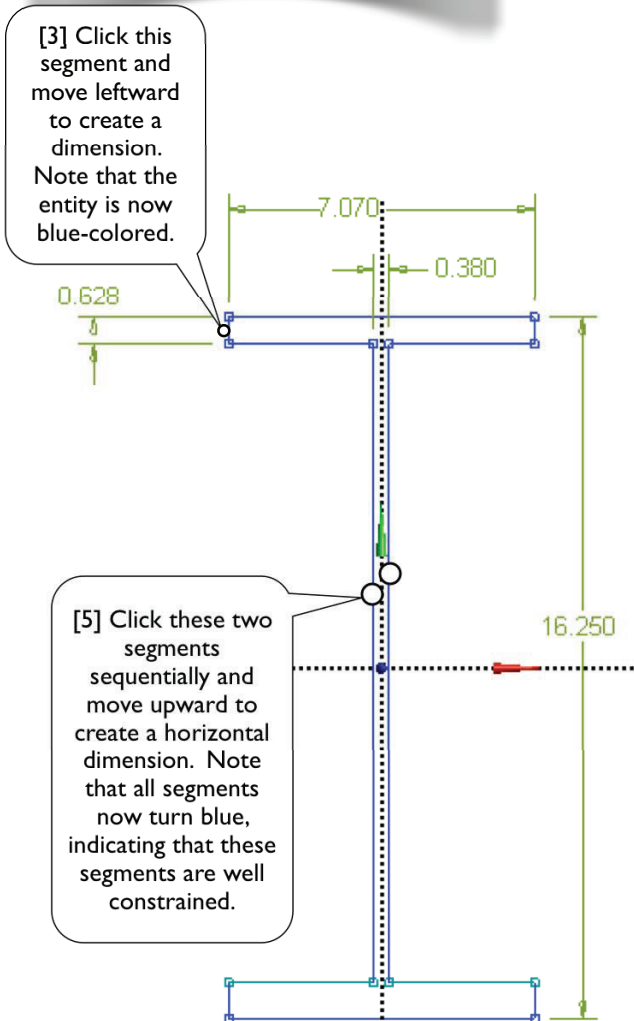
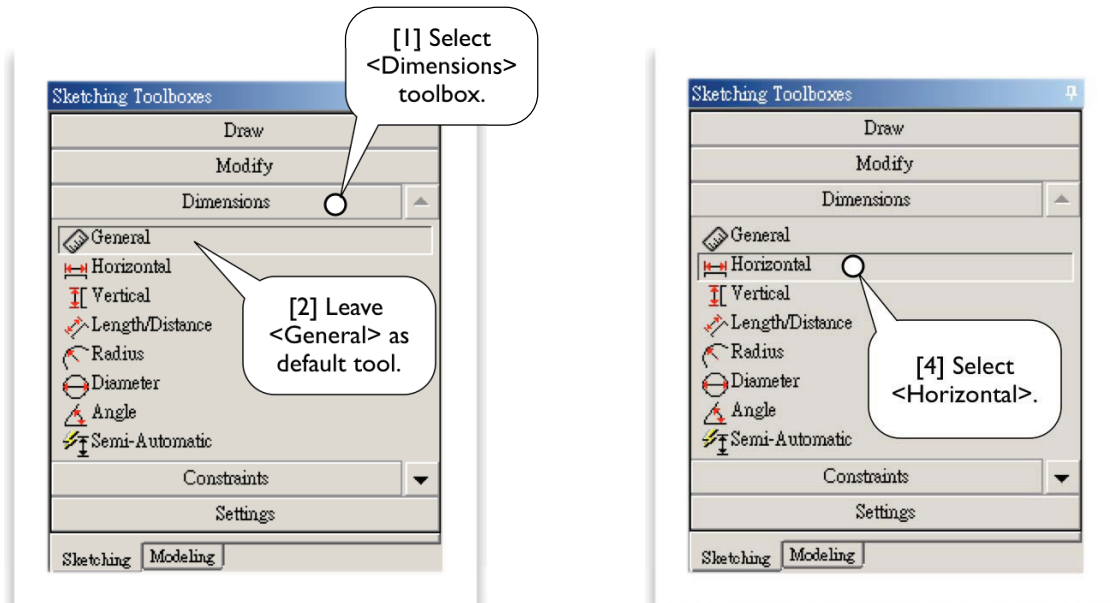
2.1-7 Trim Away Unwanted Segments



2.1-8 Impose Symmetry Constraints



2.1-9 Specify Dimensions

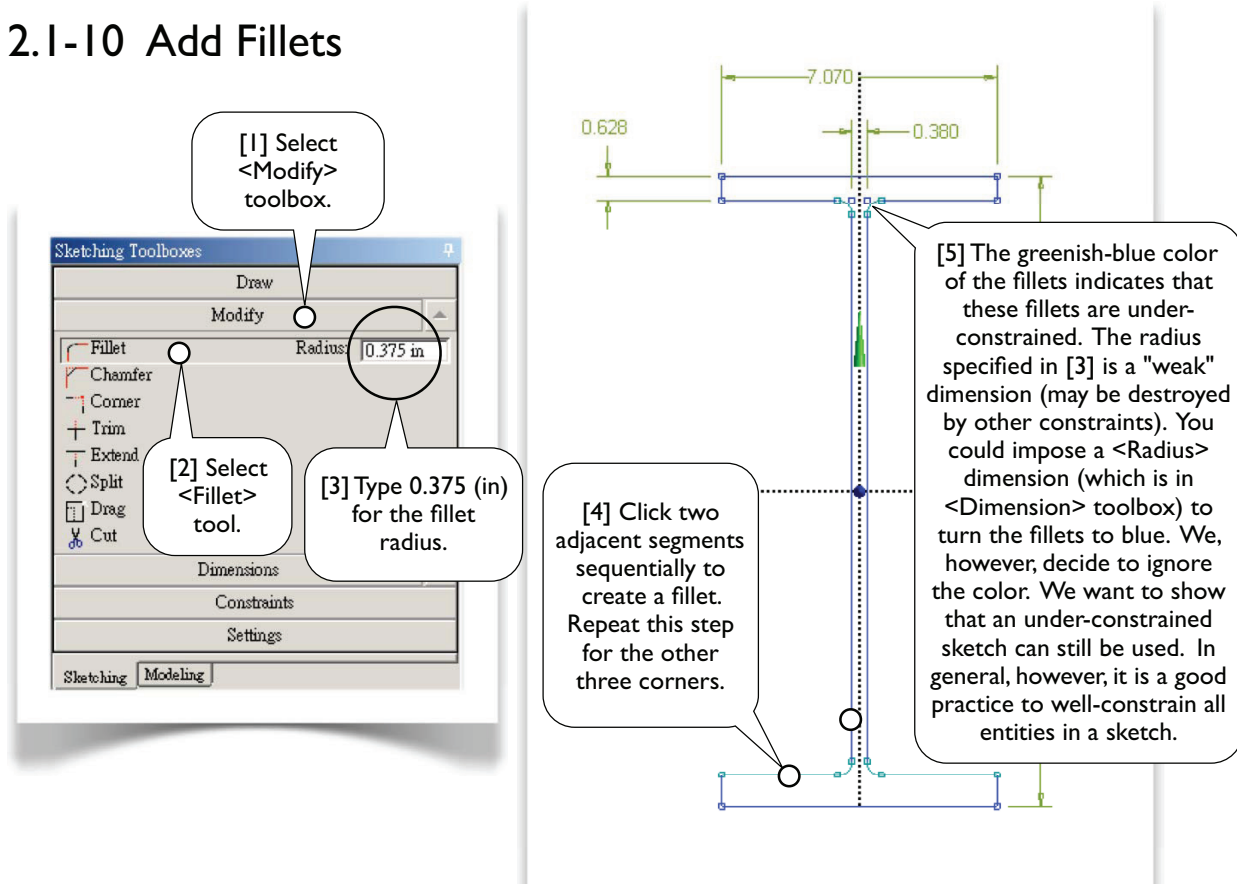


Details View

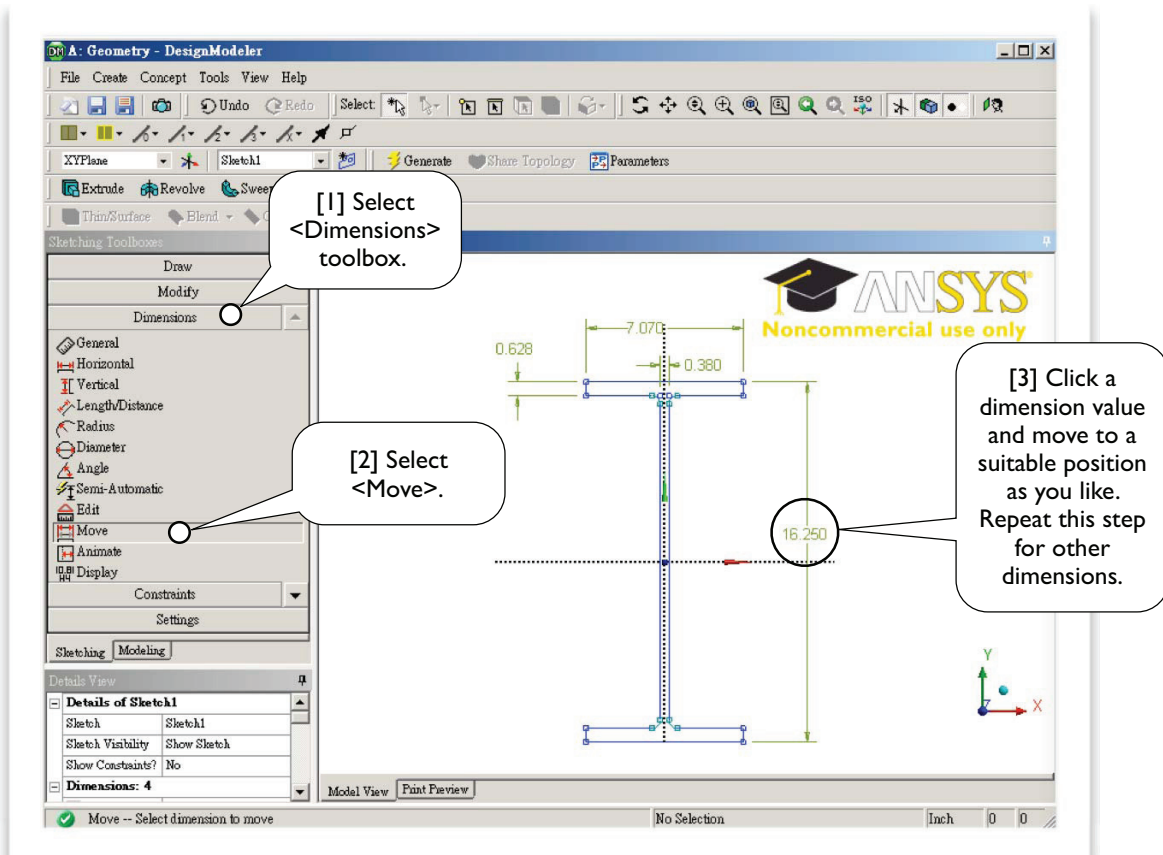
Details of Sketch1	
Sketch	Sketch1
Sketch Visibility	Show Sketch
Show Constraints?	No
Dimensions: 4	
<input type="checkbox"/> H1	7.07 in
<input type="checkbox"/> H4	0.38 in
<input type="checkbox"/> V2	16.25 in
<input type="checkbox"/> V3	0.628 in

[6] In <Details View>, type 0.38 (in) for H4 and 0.628 (in) for V3.

2.1-10 Add Fillets



2.1-11 Move Dimensions



2.1-12 Extrude to Generate 3D Solid

[3] Note that the active sketch (current sketch) is shown here.

[4] Click <Extrude>.

[5] Note that <Modeling> mode is automatically activated.

[6] An <Apply/Cancel> pair appears; click <Apply>. The active sketch (Sketch1) is selected as the default <Geometry>.

[7] In <Details View>, type 120 (in) for <Depth>.

[8] Click <Generate>.

[1] Click the little cyan sphere to rotate the world to an isometric view for a better visual effect.

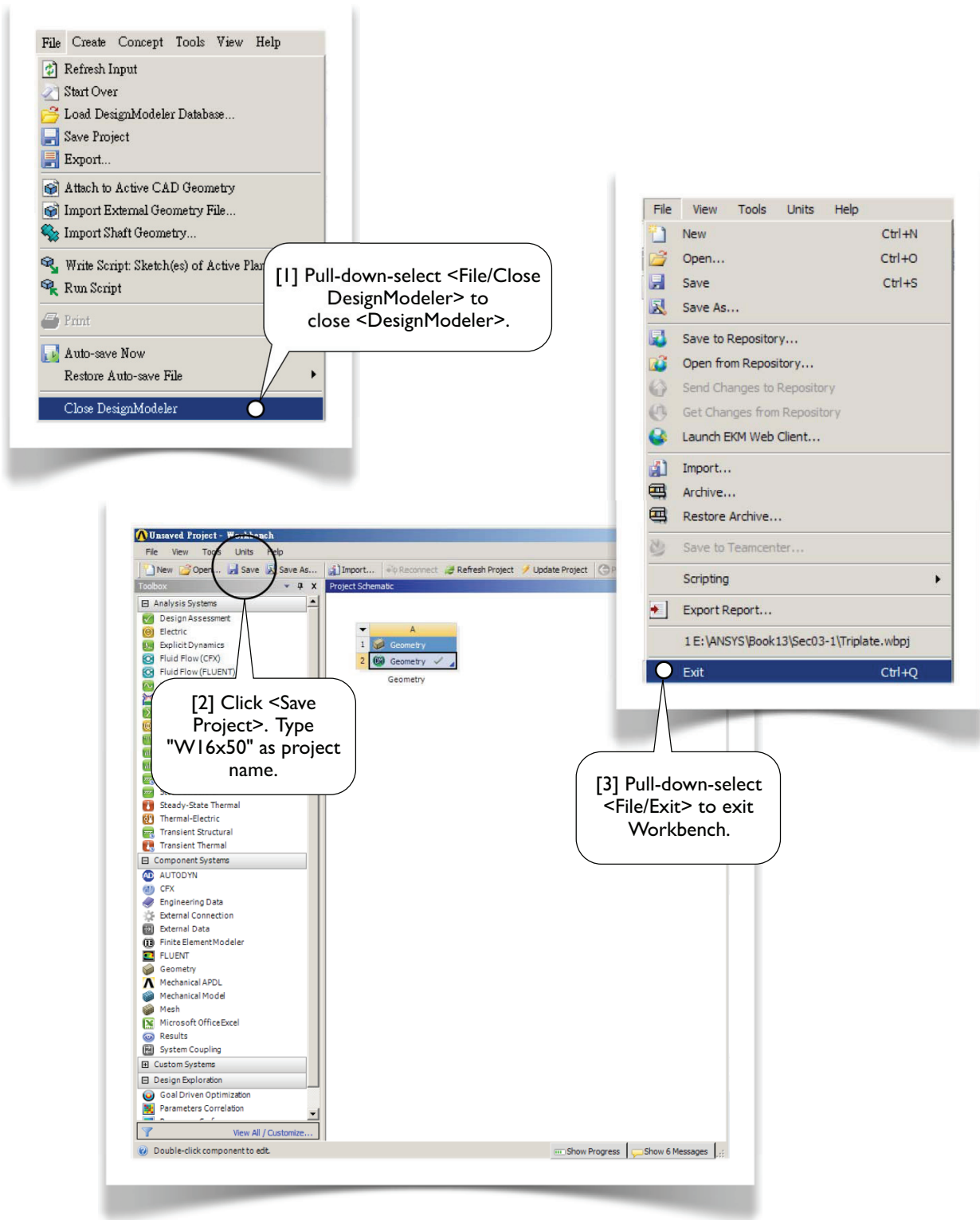
[2] The world rotates and is in isometric view now.

[9] Click <Zoom to Fit>. Use this tool whenever needed.

[10] Click <Display Plane> to turn off the display of sketching plane.

[11] Click all plus signs (+) to expand the model tree and examine the structure of <Tree Outline>.

2.1-13 Save Project and Exit Workbench



Section 2.2

Step-by-Step: Triangular Plate



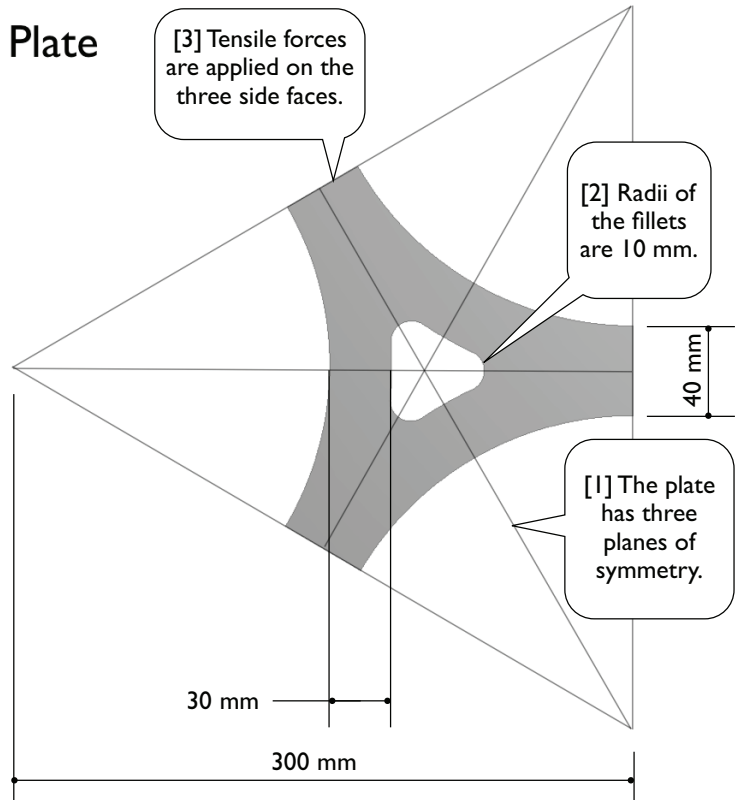
2.2-1 About the Triangular Plate

The triangular plate [1, 2] is made to withstand a tensile force on each side face [3]. The thickness of the plate is 10 mm. Other dimensions are shown in the figure.

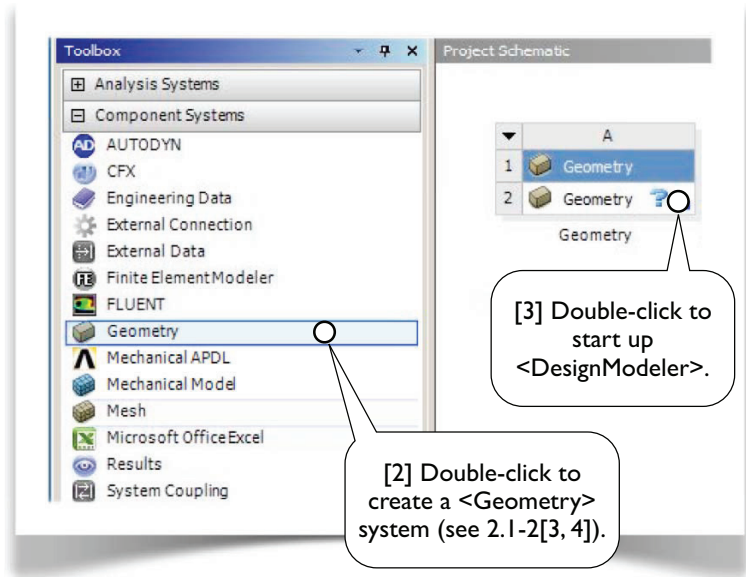
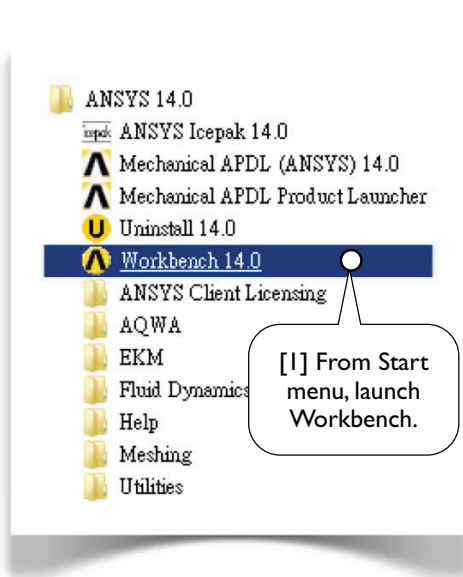
In this section, we want to sketch the plate on <XYPlane> and then extrude a thickness of 10 mm along Z-axis to generate a 3D solid body.

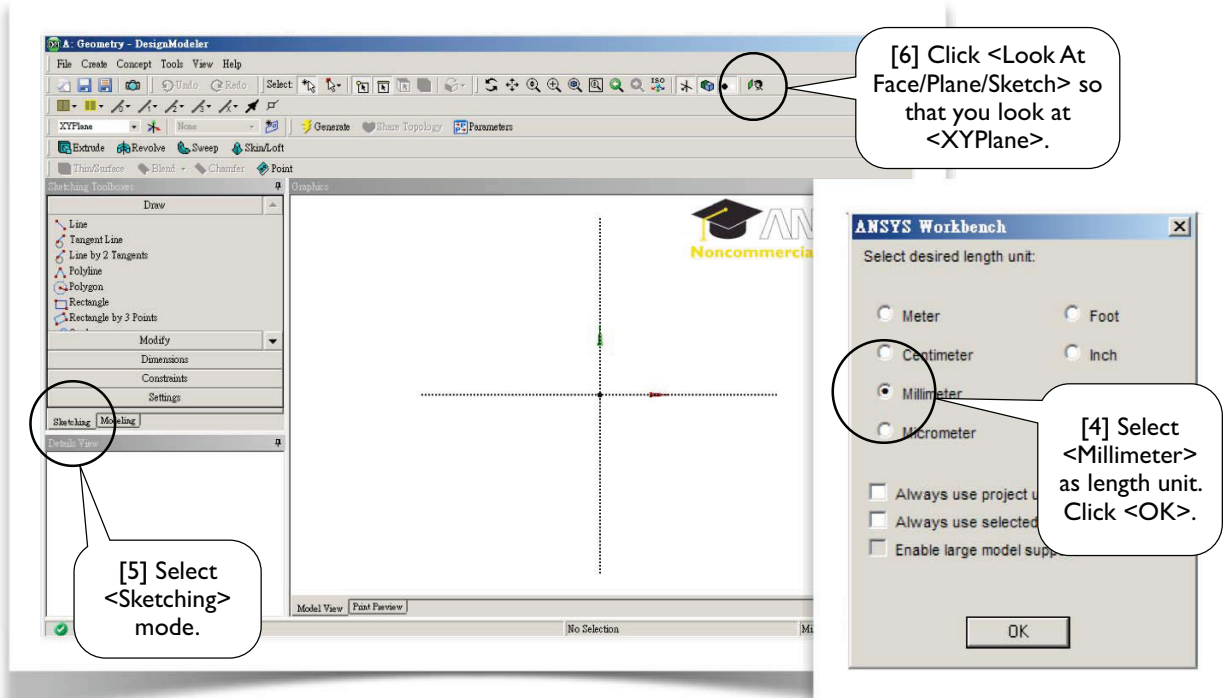
In Section 3.1, we will use this sketch again to generate a 2D solid model, and the 2D model is then used for a static structural simulation to assess the stress under the loads.

The 2D solid model will be used again in Section 8.2 to demonstrate a design optimization procedure.

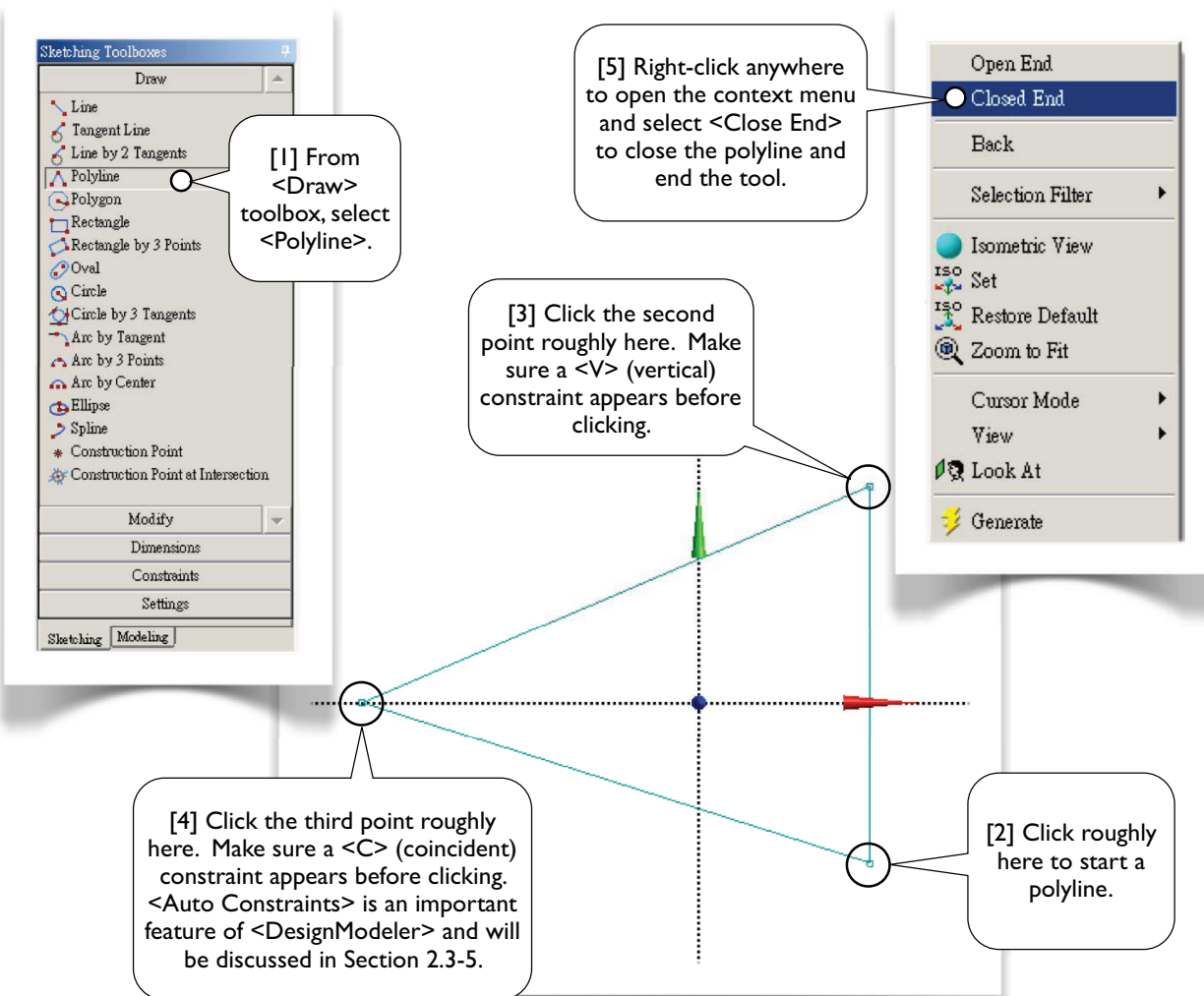


2.2-2 Start up <DesignModeler>





2.2-3 Draw a Triangle on <XYPlane>



2.2-4 Make the Triangle Regular

[1] From <Constraints> toolbox, select <Equal Length> tool.

[2] Click these two segments one after the other to make their lengths equal.

[3] Click these two segments one after the other to make their lengths equal.

2.2-5 2D Graphics Controls

Before we proceed further, let's look into some useful tools for 2D graphics controls [1-10]; feel free to use these tools whenever needed. Here, the tools are numbered according to roughly their frequency of use. Click to turn on a tool; click again to turn it off. Note that more useful mouse shortcuts for <Pan>, <Zoom>, and <Box Zoom> are available; please see Section 2.3-4.

[3] <Pan>. Click to turn on/off this mode. When on, you can click-and-drag on the graphic area to move the sketch.

[2] <Zoom to Fit>. Click this tool to fit the entire sketch in the graphic area.

[7] <Next View>. Click this tool to go to next view.

[1] <Look At Face/Plane/Sketch>. Click this tool to make current sketching plane rotate toward you.

[5] <Zoom>. Click to turn on/off this mode. When on, you can click-and-drag upward or downward on the graphic area to zoom in or out.

[4] <Box Zoom>. Click to turn on/off this mode. When on, you can click-and-drag a box on the graphic area to enlarge that portion of graphics.

[6] <Previous View>. Click this tool to go to previous view.

[8] These tools work for either <Sketching> or <Modeling> mode.

[9] <Undo>. Click this tool to undo what you've just done. Multiple undo's are allowed. This tool is available only in <Sketching> mode.

[10] <Redo>. Click this tool to redo what you've just undone. This tool is available only in <Sketching> mode.

2.2-6 Specify Dimensions

[1] In <Dimension> toolbox, click <Display>. Click <Name> to turn it off and automatically turn <Value> on. For the rest of the book, we always display values instead of names.

[2] Select <Horizontal>.

[3] Click the vertex on the left and the vertical line on the right (before clicking, make sure the cursor indicates that the point or edge has been "snapped,") and then move the mouse downward to create this dimension. (The value 300 will be typed in step [5].)

[4] Click the vertex on the left and the vertical axis, and then move the mouse downward to create this dimension. Note that all the segments turn to blue, indicating they are well defined now. (The value 200 will be typed in step [5].)

[5] In <Details View>, type 300 (mm) and 200 (mm) for the dimensions just created. Click <Zoom to Fit> (2.2-5[2]).

[6] Select <Move> and then move the dimensions as you like (2.1-11).

2.2-7 Draw an Arc

[1] From <Draw> toolbox, select <Arc by Center>.

[2] Click this vertex as the arc center. Make sure a <P> (point) constraint appears before clicking.

[3] Click the second point roughly here. Make sure a <C> (coincident) constraint appears before clicking.

[4] Click the third point here. Make sure a <C> (coincident) constraint appears before clicking.

2.2-8 Replicate the Arc

[1] From <Modify> toolbox, select <Replicate>. Type 120 (degrees) for <r>. <Replicate> is equivalent to <Copy>+<Paste>.

[2] Select the arc.

[3] Right-click anywhere to open the context menu and select <End/Set Paste Handle>.

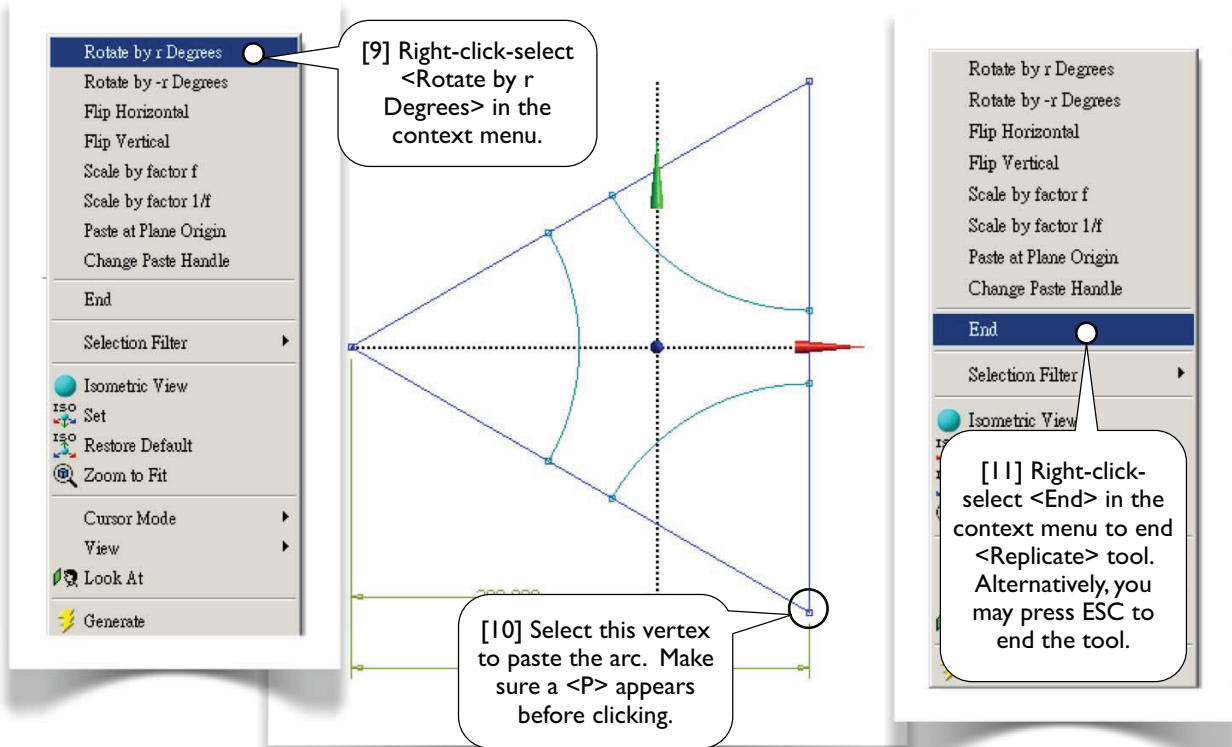
[4] Select this vertex as paste handle. Make sure a <P> appears before clicking. If you have difficulty making <P> appear, see [7, 8].

[5] Right-click-select <Rotate by r Degrees> from the context menu.

[6] Click this vertex to paste the arc. Make sure a <P> appears before clicking. If you have difficulty making <P> appear, see [7, 8].

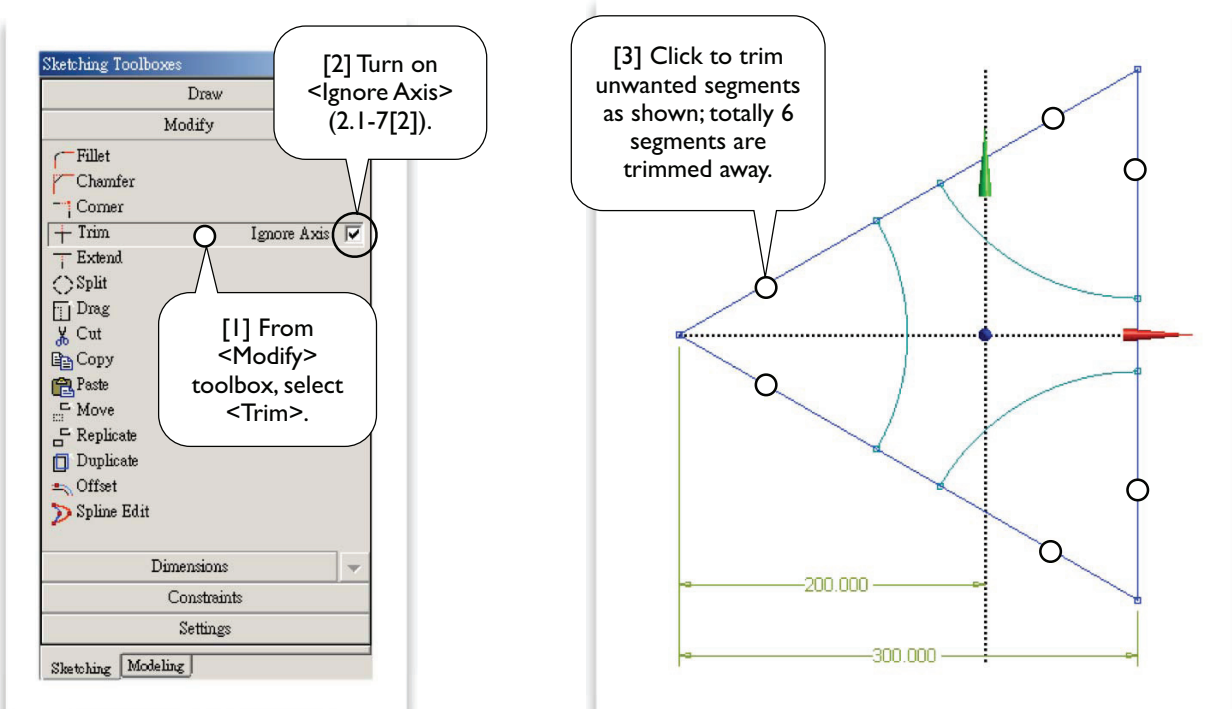
[7] Whenever you have difficulty making <P> appear, click <Selection Filter: Points> in the toolbar. <Selection Filter> also can be set from the context menu, see [8].

[8] <Selection Filter> also can be set from the context menu.



For instructional purpose, we chose to manually set the paste handle [3] on the vertex [4]. In this case, we actually could have used plane origin as handle.

2.2-9 Trim Away Unwanted Segments



2.2-10 Impose Constraints

[1] From <Constraints> toolbox, select <Equal Length>.

[2] Select this segment and the vertical segment sequentially to make their lengths equal.

[3] Select this segment and the vertical segment sequentially to make their lengths equal.

[4] Select <Symmetry>.

[5] Select the horizontal axis as the line of symmetry.

[6] Select the lower and upper arcs sequentially to make them symmetric.

Constraint Status

Note that the arcs have a greenish-blue color, indicating they are not well defined yet (i.e., under-constrained). Other color codes are: blue and black colors for well defined entities (i.e., fixed in the space); red color for over-constrained entities; gray to indicate an inconsistency.

2.2-11 Specify Dimension for Side Faces

[1] Select <Dimension> toolbox and leave <General> as default.

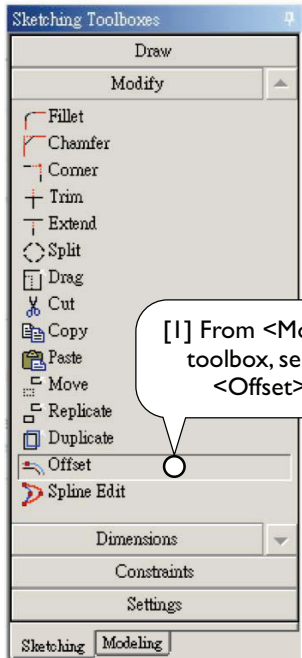
[2] Click the vertical segment and move the mouse rightward to create this dimension. (The value 40 will be typed in the next step.)

[3] Type 40 (mm) for the dimension just created.

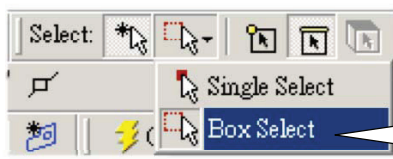
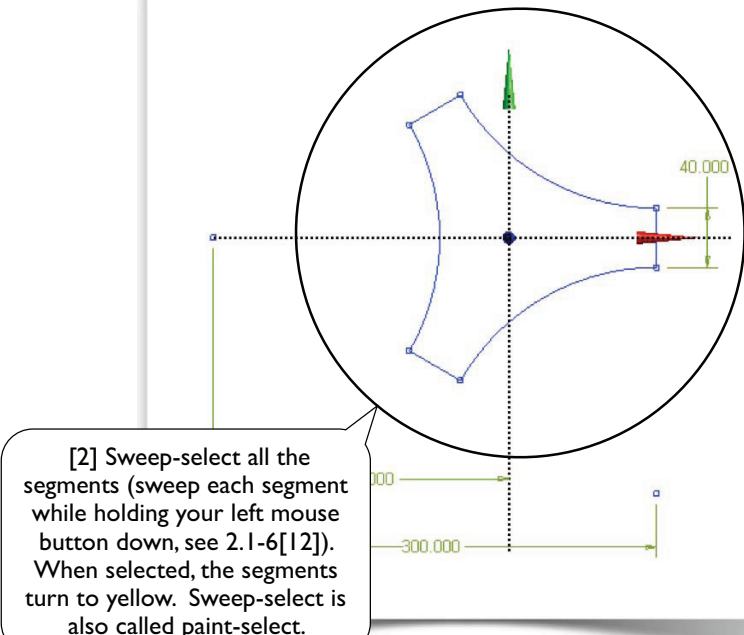
Details of Sketch1	
Sketch	Sketch1
Sketch Visibility	Show Sketch
Show Constraints?	No
Dimensions: 3	
<input type="checkbox"/> H1	300 mm
<input type="checkbox"/> H2	200 mm
<input type="checkbox"/> V3	40 mm

After impose dimension in [2], all segments turn to blue, indicating they are well defined now. Note that we didn't specify the radii of the arcs; the radii of the arcs are automatically calculated.

2.2-12 Create Offset

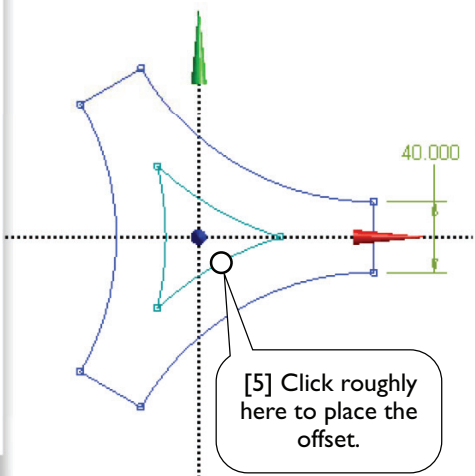
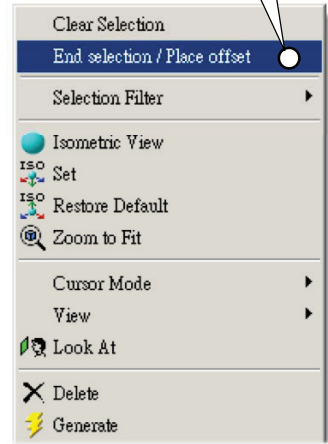


[1] From <Modify> toolbox, select <Offset>.

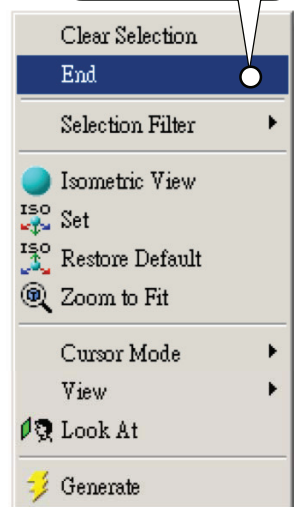


[3] Another way to select multiple entities is to switch <Select Mode> to <Box Select>, and then draw a box to select all entities inside the box.

[4] Right-click-select <End selection/Place Offset> in the context menu.



[6] Right-click-select <End> in the context menu, or press ESC, to close <Offset> tool.



[7] From <Dimension> toolbox, select <Horizontal>.

[8] Click the two left arcs and move downward to create this dimension. Note that all the segments turn to blue now.

[9] Type 30 (mm) for the dimension just created.

[10] It is possible that some points become separate after imposing the dimension. If so, impose a <Coincident> constraint on them, see [11].

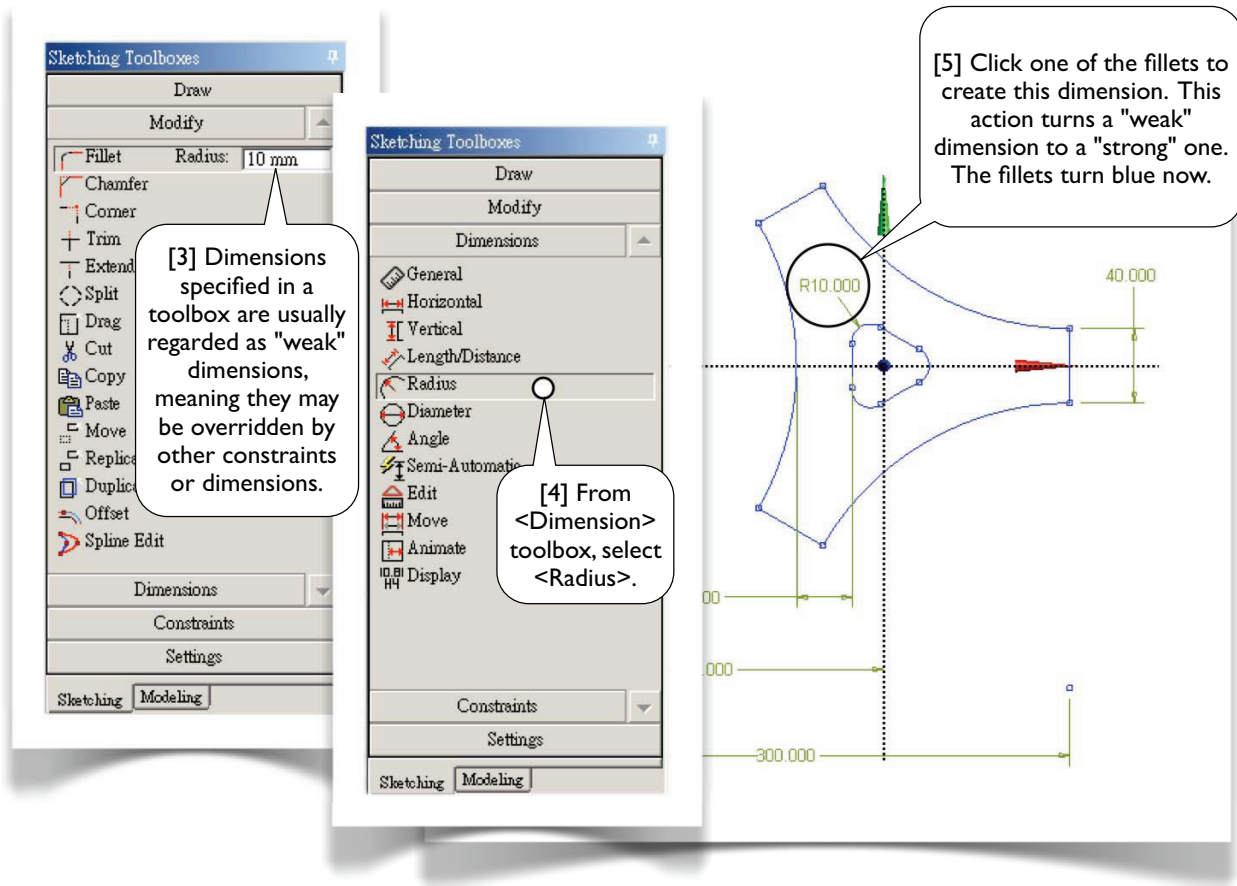
[11] If necessary, impose a <Coincident> on the separated points.

Details of Sketch1	
Sketch	Sketch1
Sketch Visibility	Show Sketch
Show Constraints?	No
Dimensions: 4	
<input type="checkbox"/> H1	300 mm
<input type="checkbox"/> H2	200 mm
<input type="checkbox"/> H4	30 mm
<input type="checkbox"/> V3	40 mm

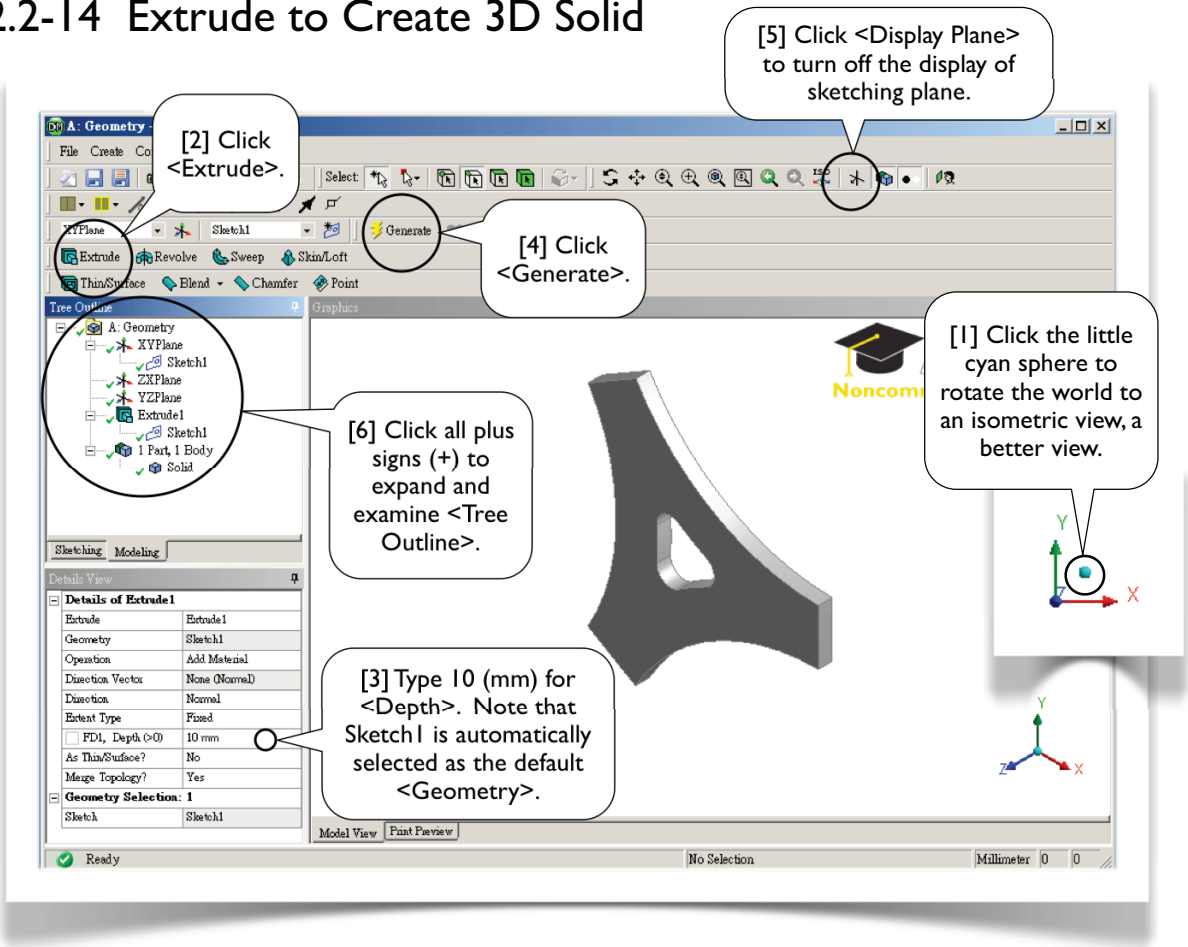
2.2-13 Create Fillets

[1] In <Modify> toolbox, select <Fillet>. Type 10 (mm) for <Radius>.

[2] Click two segments sequentially to create a fillet. Repeat this step to create the other two fillets. Note that the fillets are in greenish-blue color, indicating they are only weakly defined.



2.2-14 Extrude to Create 3D Solid



2.2-15 Save the Project and Exit Workbench

[1] Pull-down-select <File/Close DesignModeler> to close <DesignModeler>.

[2] Click <Save Project>. Type "Triplate" as project name.

[3] Pull-down-select <File/Exit> to exit Workbench.

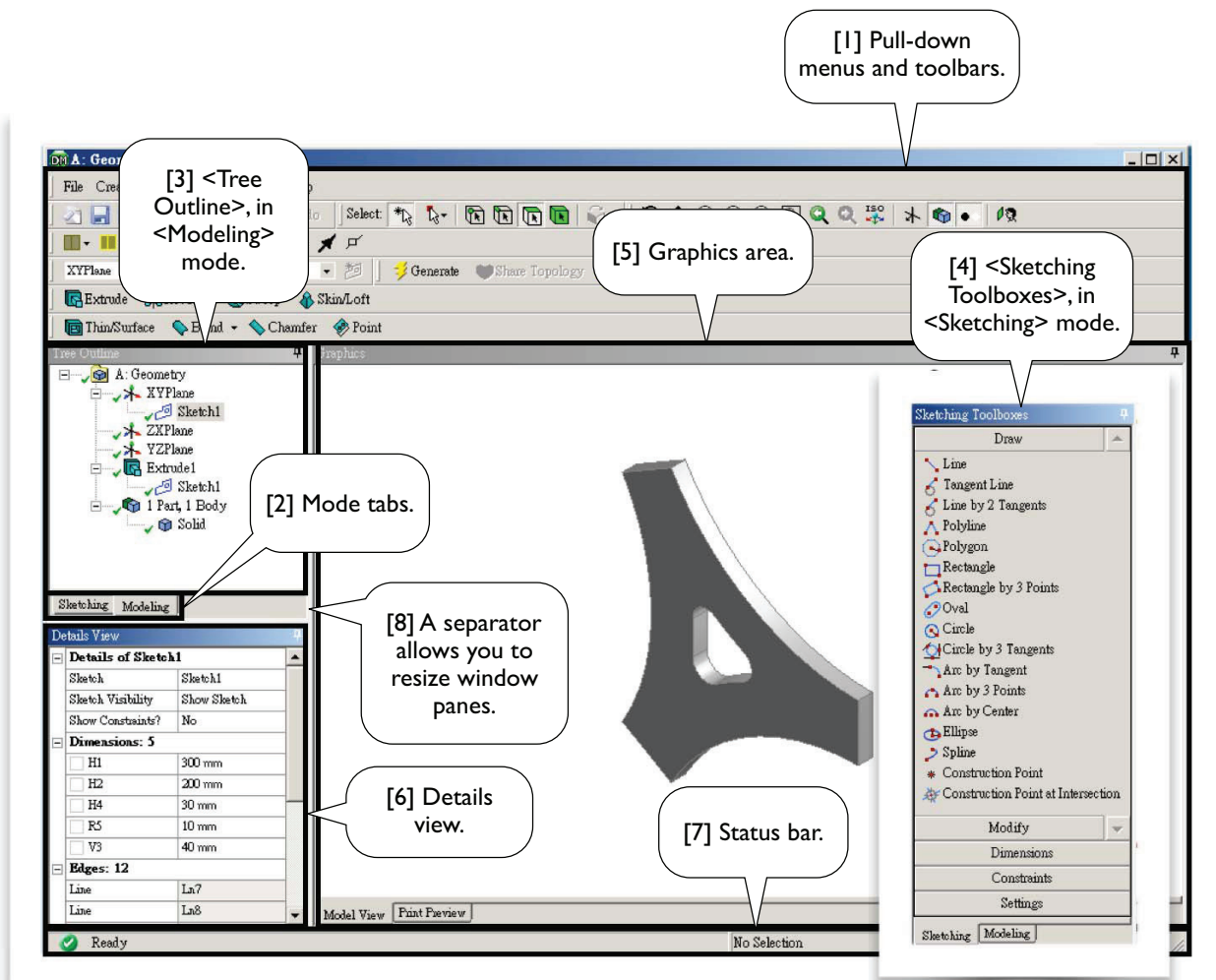
Section 2.3

More Details

2.3-1 DesignModeler GUI

<DesignModeler GUI> is divided into several areas [1-7]. On the top are pull-down menus and toolbars [1]; on the bottom is a status bar [7]. In-between are several "window panes." A separator [8] between two window panes can be dragged to resize the window panes. You even can move or dock a window pane by dragging its title bar. Whenever you mess up the workspace, pull-down-select <View/Windows/Reset Layout> to reset the default layout.

<Tree Outline> [3] shares the same area with <Sketching Toolboxes> [4]; you can switch between <Modeling> mode and <Sketching> mode by clicking a "mode tab" [2]. <Details View> [6] shows the detail information of the objects highlighted in <Tree Outline> [3] or graphics area [5]. The graphics area [5] displays the model when in <Model View> mode; you can click a tab (at the bottom of the graphics area) to switch to <Print Preview>. We will introduce more features of <DesignModeler GUI> in Chapter 4.



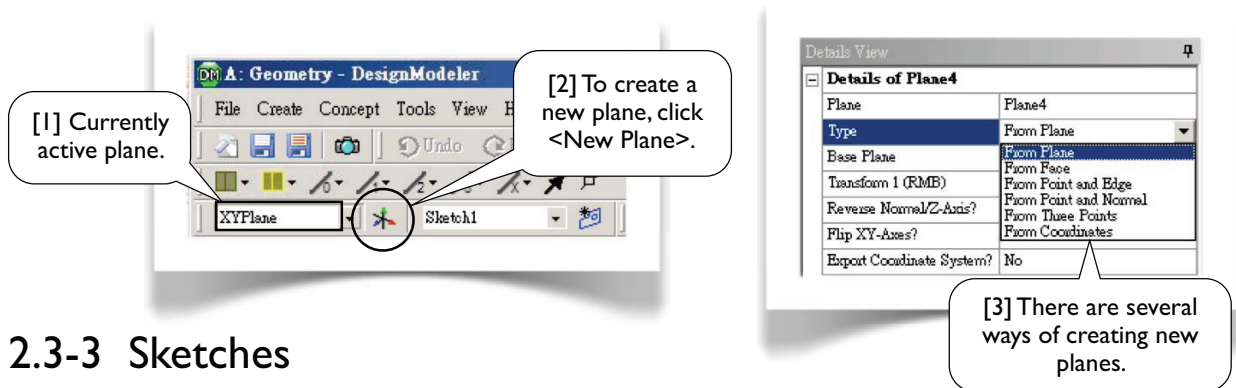
Model Tree

<Tree Outline> [3] contains an outline of the *model tree*, the data structure of the geometric model. Each *branch* of the tree is called an *object*, which may contain one or more objects. At the bottom of the model tree is a *part* branch, which is the only object that will be exported to <Mechanical>. By right-clicking an object and selecting a tool from the context menu, you can operate on the object, such as delete, rename, duplicate, etc.

The order of the objects is relevant. <DesignModeler> renders the geometry according to the order of objects in the model tree. New objects are normally added one after another. If you want to insert a new object BEFORE an existing object, right-click the existing object and select <Insert/...> from the context menu. After insertion, <DesignModeler> will re-render the geometry.

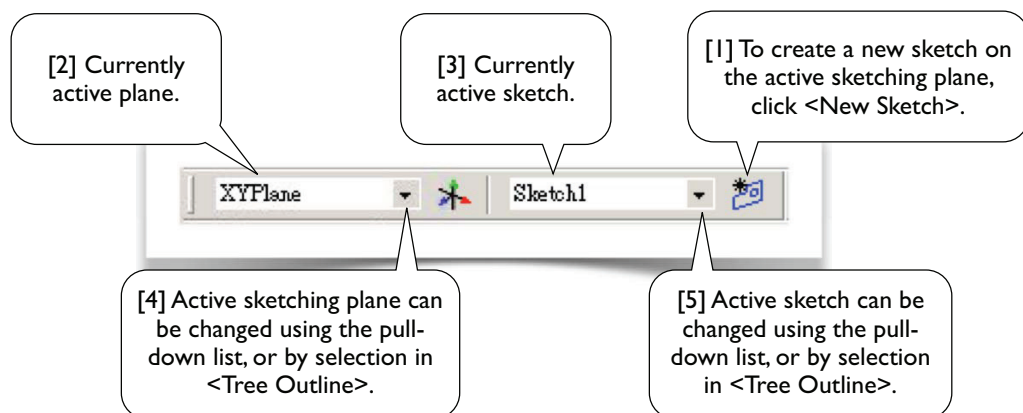
2.3-2 Sketching Planes

A sketch must be created on a *sketching plane*, or simply called *plane*; each plane, however, may contain multiple sketches. In the beginning of a <DesignModeler> session, three planes are automatically created: <XYPlane>, <YZPlane>, and <ZXPlane>. Currently active plane is shown on the toolbar [1]. You can create new planes as many as needed [2]. There are several ways of creating new planes [3]. In this chapter, since we always assume that sketches are created on <XYPlane>, we will not discuss how to create sketching planes further, which will be discussed in Chapter 4.



2.3-3 Sketches

A sketch consists of *points* and *edges*; edges may be straight lines or curves. Dimensions and constraints may be imposed on these geometric entities. As mentioned (Section 2.3-2), multiple sketches may be created on a plane. To create a new sketch on a plane on which there is yet no sketch, you simply switch to <Sketching> mode and draw any geometric entities on it. Later, if you want to add a new sketch on that plane, you have to click <New Sketch> [1]. Exactly one plane and one sketch is active at a time [2-5]; newly created sketches are added to the active plane, and newly created geometric entities are added to the active sketch. In this chapter, we almost exclusively work with a single sketch; the only exception is Section 2.6, in which a second sketch is used (2.6-4). More on creating sketches will be discussed in Chapter 4. When a new sketch is created, it becomes the active sketch.



2.3-4 Sketching Toolboxes

When you switch to <Sketching> mode by clicking the mode tab (2.3-1[2]), you will see <Sketching Toolboxes> (2.3-1[4]). <Sketching Toolboxes> consists of five toolboxes: <Draw>, <Modify>, <Dimensions>, <Constraints>, and <Settings> [1-5]. Most of the tools in the toolboxes are self-explained. The best way to learn these tools is to try them out individually. During the tryout, whenever you want to clean up the graphics area, pull-down-select <File/Start Over>. These sketching tools will be explained from 2.3-6 to 2.3-10.

Before we discuss these sketching tools, some tips relevant to sketching are emphasized below.

Pan, Zoom, and Box Zoom

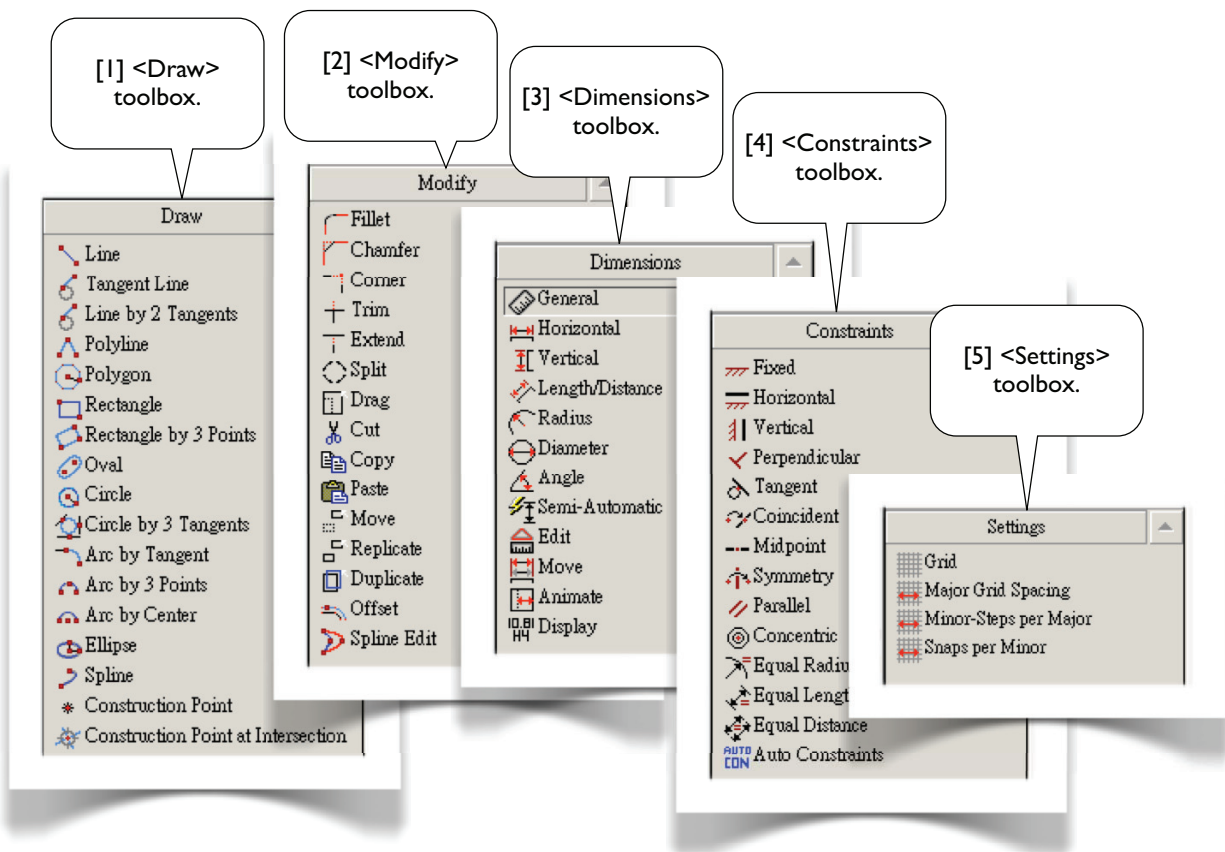
Besides <Pan> tool (2.2-5[3]), the graphics can be panned by dragging your mouse while holding down both control key and the middle mouse button. Besides <Zoom> tool (2.2-5[5]) the graphics can be zoomed in/out by simply rolling forward/backward your mouse wheel; the cursor position is the "zoom center." <Box Zoom> (2.2-5[4]) can be done by dragging a rectangle in the graphics area using the right mouse button. When you get used to these basic mouse actions, you usually don't need <Pan>, <Box Zoom>, and <Zoom> tools (2.2-5[3-5]) any more.

Context Menu

While most of operations can be done by issuing commands using pull-down menus or toolbars, many operations either require or are more efficient using the context menu. The context menu can be popped-up by right-clicking the graphics area or objects in the model tree. Try to explore whatever available in the context menu.

Status Bar

The status bar (2.3-1[7]) contains instructions on completing each operations. Look at the instruction whenever you don't know what is the next action to be done. Whenever a draw tool is in use, the coordinates of your mouse pointer are shown in the status bar.



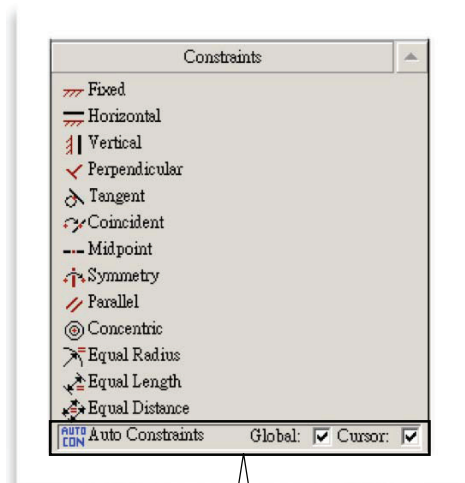
2.3-5 Auto Constraints^{1,2}

By default, <DesignModeler> is in <Auto Constraints> mode, both globally and locally. While drawing, <DesignModeler> attempts to detect the user's intentions and try to automatically impose constraints on points or edges. The following cursor symbols indicate the kind of constraints that will be applied:

- C - The cursor is coincident with a line.
- P - The cursor is coincident with another point.
- T - The cursor is a tangent point.
- ⊥ - The cursor is a perpendicular foot.
- H - The line is horizontal.
- V - The line is vertical.
- // - The line is parallel to another line.
- R - The radius is equal to another radius.

Both <Global> and <Cursor> modes are based on all entities of the active plane (not just the active sketch). The difference is that <Cursor> mode only examines the entities nearby the cursor, while <Global> mode examines all the entities in the active plane.

Note that while <Auto Constraints> can be useful, they sometimes can lead to problems and add noticeable time on complicated sketches. Turn off them if desired [1].



[1] By default, <DesignModeler> is in <Auto Constraints> mode, both globally and locally. You can turn them off whenever they cause troubles.

2.3-6 <Draw> Tools³ [1]

Line

Draws a straight line by two clicks.

Tangent Line

Click a point on an edge (an edge may be a curve or a straight line) to start a line. The line will be tangent to the edge at that point.

Line by 2 Tangents

If you click two curves (a curve may be a circle, arc, ellipse, or spline), a line tangent to these two curves will be created. If you click a curve and a point, a line tangent to the curve and ending to the point will be created.

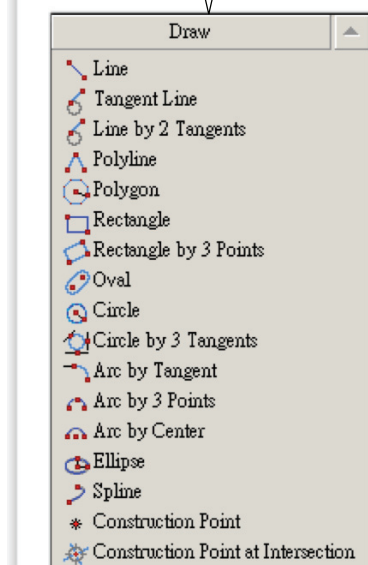
Polyline

A polyline consists of multiple straight line segments. A polyline must be completed by choosing either <Open End> or <Closed End> from the context menu [2].

Polygon

Draws a regular polygon. The first click defines the center and the second click defines the radius of the circumscribing circle.

[1] <Draw> toolbox.



Rectangle by 3 Points

The first two points define one side and the third point defines the other side.

Oval

The first two clicks define two centers, and the third click defines the radius.

Circle

The first click defines the center, and the second click defines the radius.

Circle by 3 Tangents

Select three edges (lines or curves), and a circle tangent to these three edges will be created.

Arc by Tangent

Click a point on an edge, an arc starting from that point and tangent to that edge will be created; click a second point to define the other end (and the radius) of the arc.

Arc by 3 Points

The first two clicks define the two ends of the arc, and the third click defines a point in-between the ends.

Arc by Center

The first click defines the center, and two additional clicks define the ends.

Ellipse

The first click defines the major axis and the major radius, and the second click defines the minor radius.

Spline

A spline is either rigid or flexible. The difference is that a flexible spline can be edited or changed by imposing constraints, while a rigid spline cannot. After defining the last point, you must right-click to open the context menu, and select an option [3]: either open end or closed end; either with fit points or without fit points.

Construction Point at Intersection

Select two edges, a construction point will be created at the intersection.

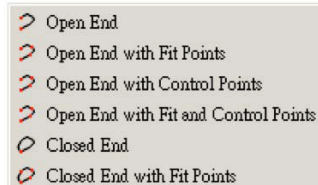
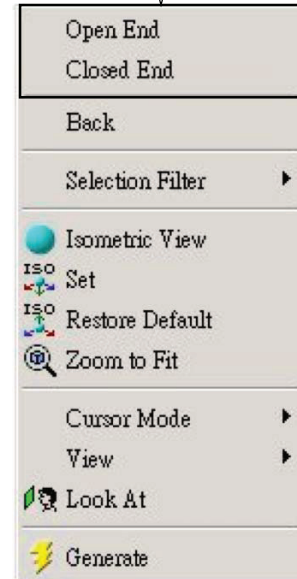
How to delete edges?

To delete edges, select them and choose <Delete> or <Cut> from the context menu. Multiple selection methods (e.g., control-selection or sweep-selection) can be used to select edges. To clean up the graphics area entirely, pull-down-select <File/Start Over>. A more general way of deleting any sketching entities (edges, dimensions, or constraints) is to right-click the entity in <Details View> and issue <Delete> (see 2.3-8[10] and 2.3-9[3, 4]).

How to abort a tool?

To abort a tool, simply press <ESC>.

[2] A polyline must be completed by choosing either <Open End> or <Closed End> from the context menu.



[3] A spline must be complete by selecting one of the options from the context menu.

2.3-7 <Modify> Tools⁴ [1]

Fillet

Select two edges or a vertex, and a fillet will be created. The radius of the fillet can be specified in the toolbox [2]. Note that this radius value is temporary and not a "formal" dimension or constraint, meaning that it can be changed by other dimensions or constraints.

Chamber

Select two edges or a vertex, and an equal-length chamber will be created. The lengths (distance between the vertex and the endpoints of the chamber line) can be specified in the toolbox, similar to [2].

Corner

Select two edges, and the edges will be trimmed or extended up to the intersection point and form a sharp corner. The clicking points decide which sides to be trimmed.

Trim

Select an edge, and the portion of the edge up to its intersection with other edge, axis, or point will be removed.

Extend

Select an edge, and the edge will be extended up to an edge or axis.

Split

This tool splits an edge into several segments depending on the options from the context menu [3]. <Split Edge at Selection>: select an edge, and the edge will be split at the clicking point. <Split Edges at Point>: select a point, and all the edges passing through that point will be split at that point. <Split Edge at All Points>: select an edge, the edge will be split at all points on the edge. <Split Edge into n Equal Segments>: Select an edge and specify a value n , and the edge will be split equally into n segments.

Drag

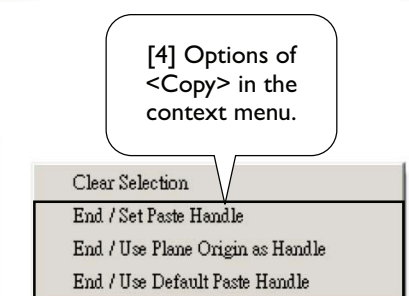
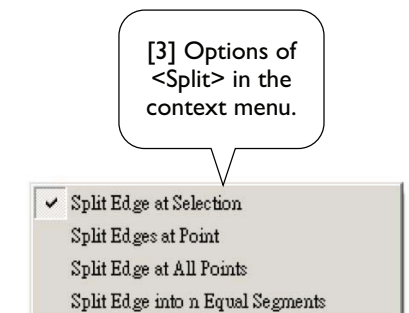
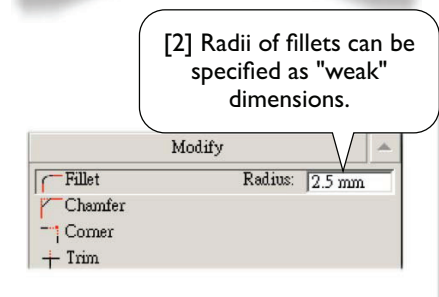
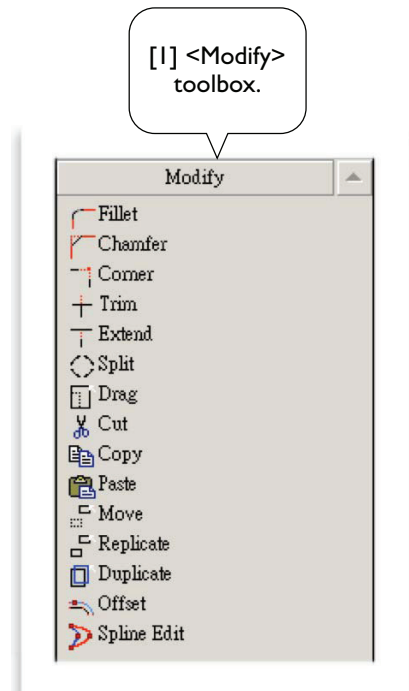
Drags a point or an edge to a new position. All the constraints and dimensions are preserved.

Copy

Copies the selected entities to a "clipboard." A <Paste Handle> must be specified using one of the methods in the context menu [4]. After completing this tool, <Paste> tool is automatically activated.

Cut

Similar to <Copy>, i.e., copy the selected entities to a "clipboard," except that the copied entities are removed.



Paste

Pastes the entities in the "clipboard" to the graphics area. The first click defines the position of the <Paste Handle> specified in the <Copy> or <Cut> tools. Many options can be chosen from the context menu [5], where the rotating angle r and the scaling factor f can be specified in the toolbox.

Move

Equivalent to a <Cut> followed by a <Paste>. (The original is removed.)

Replicate

Equivalent to a <Copy> followed by a <Paste>. (The original is preserved.)

Duplicate

Equivalent to <Replicate>, except the entities are pasted on the same place as the originals and become part of the current sketch. It is often used to duplicate plane boundaries.

Offset

Creates a set of edges that are offset by an equal distance from an existing set of edges.

Spline Edit

Used to modify flexible splines. You can insert, delete, drag the fit points, etc [6]. For details, see the reference⁴.

[5] Options of <Paste> in the context menu.

Rotate by r Degrees
 Rotate by $-r$ Degrees
 Flip Horizontal
 Flip Vertical
 Scale by factor f
 Scale by factor $1/f$
 Paste at Plane Origin
 Change Paste Handle

[6] Option of <Spline Edit> in the context menu.

Select New Spline
 Re-fit Spline
 Create Missing Fit Points
 Delete New Fit Points
 Create Missing Control Points
 Drag Fit Point
 Drag Control Point
 Insert Fit Point
 Delete Fit Point

2.3-8 <Dimensions> Tools⁵ [1]

General

Allows creation of any of the dimension types, depending on what edge and right mouse button options are selected. If the selected edge is a straight line, the default dimension is its length; you can choose other dimension type from the context menu [6]. If the selected edge is a circle or arc, the default dimension is the radius; you can choose other dimension type from the context menu [7].

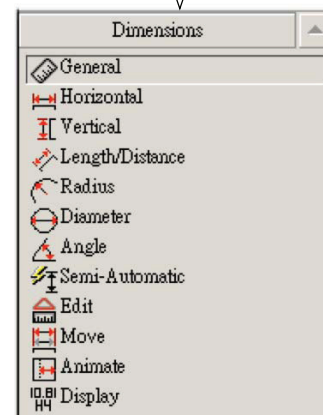
Horizontal

Select two points to specify a horizontal dimension. If you select an edge (instead of a point), the horizontal extremity of the edge will be assumed.

Vertical

Similar to <Horizontal>.

[1] <Dimension> toolbox.



Length/Distance

Select two points to specify a distance dimension. You also can select a point and a line to specify the distance between the point and the line.

Radius

Select a circle or arc to specify a radius dimension. If you select an ellipse, the major (or minor) radius will be specified.

Diameter

Select a circle or arc to specify a diameter dimension.

Angle

Select two lines to specify an angle. By varying the selection order and location of the lines, you can control which angle you are dimensioning. The end of the lines that you select will be the direction of the hands, and the angle is measured counterclockwise from the first selected hand to the second. If the angle is not what you want, repeatedly choose <Alternate Angle> from the context menu until the correct angle is selected [8].

Semi-Automatic

This tool displays a series of dimensions automatically to help you fully dimension the sketch.

Edit

Click a dimension, it allows you to change its name or value.

Move

Click a dimension and move it to an appropriate position.

Animate

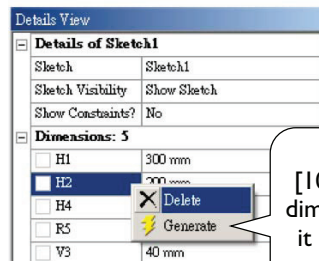
Click a dimension to show the animated effects.

Display

Allows you to decide whether to display dimension names, values, or both. In this book, we always choose to display dimension values [9] rather than dimension names.

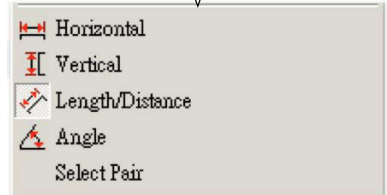
How to delete dimensions?

To delete a dimension, select the dimension in <Details View>, and choose <Delete> from the context menu [10]. You even can delete ALL dimensions by right-click <Dimensions> in <Details View>.



[10] You can delete a dimension by selecting it in <Details View>.

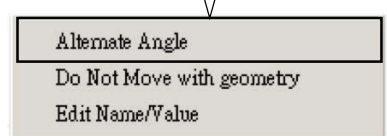
[6] Option of <General> in the context menu if you select a line.



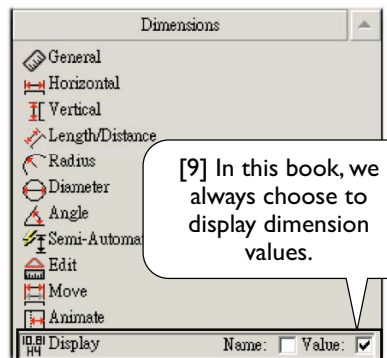
[7] Option of <General> in the context menu if you select a circle or arc.



[8] Repeatedly choose <Alternate Angle> from the context menu until the correct angle is selected.



[9] In this book, we always choose to display dimension values.



2.3-9 <Constraints> Tools⁶ [1]

Fixed

Applies on an edge to make it fully constrained if <Fix Endpoints> is selected [2]. If <Fix Endpoints> is not selected, then the edge's endpoints can be changed, but not the edge's position and slope.

Horizontal

Applies on a line to make it horizontal.

Vertical

Applies on a line to make it vertical.

Perpendicular

Applies on two edges to make them perpendicular to each other.

Tangent

Applies on two edges, one of which must be a curve, to make them tangent to each other.

Coincident

Select two points to make them coincident. Or, select a point and an edge, the edge or its extension will pass through the point. There are other possibilities, depending on how you select the entities.

Midpoint

Select a line and a point, the midpoint of the line will coincide with the point.

Symmetry

Select a line or an axis, as the line of symmetry, and then either select 2 points or 2 lines. If select 2 points, the points will be symmetric about the line of symmetry. If select 2 lines, the lines will form the same angle with the line of symmetry.

Parallel

Applies on two lines to make them parallel to each other.

Concentric

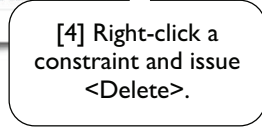
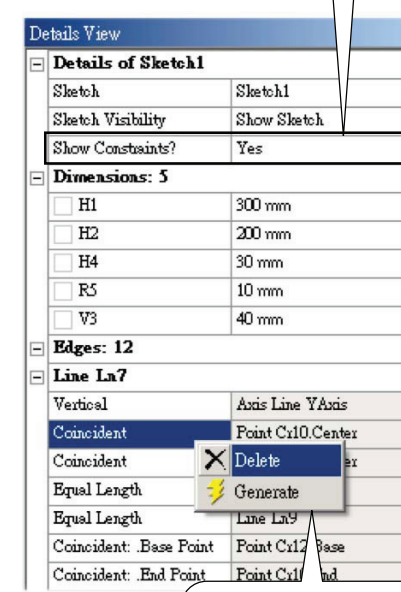
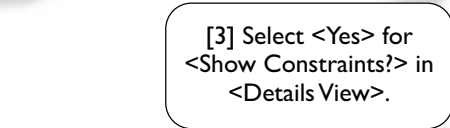
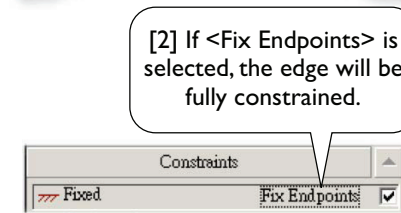
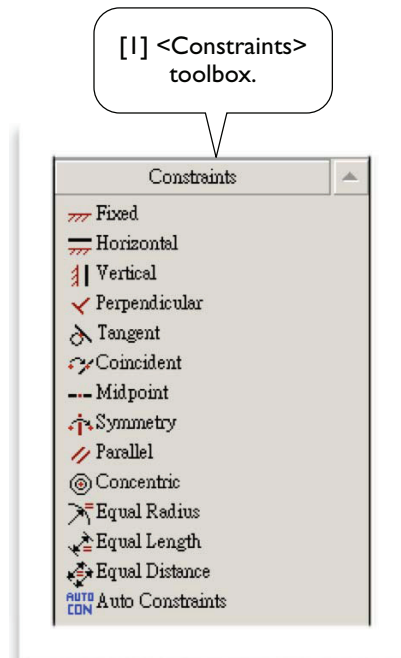
Applies on two curves, which may be circle, arc, or ellipse, to make their centers coincident.

Equal Radius

Applies on two curves, which must be circle or arc, to make their radii equal.

Equal Length

Applies on two lines to make their lengths equal.



Equal Distance

Applies on two distances to make them equal. A distance can be defined by selecting two points, two parallel lines, or one point and one line.

Auto Constraints

Allows you to turn on/off <Auto Constraints> (2.3-5[1]).

How to delete constraints?

By default, constraints are not displayed in <Details View>. To display constraints, select <Yes> for <Show Constraints?> in <Details View> [3] (previous page). You will see an edge has a group of constraints associated with it. To delete a constraint, right-click the constraint and issue <Delete> [4] (previous page).

2.3-10 <Settings> Tools⁷ [1]

Grid

Allows you to turn on/off grid visibility and snap capability. The grid is not required to enable snapping.

Major Grid Spacing

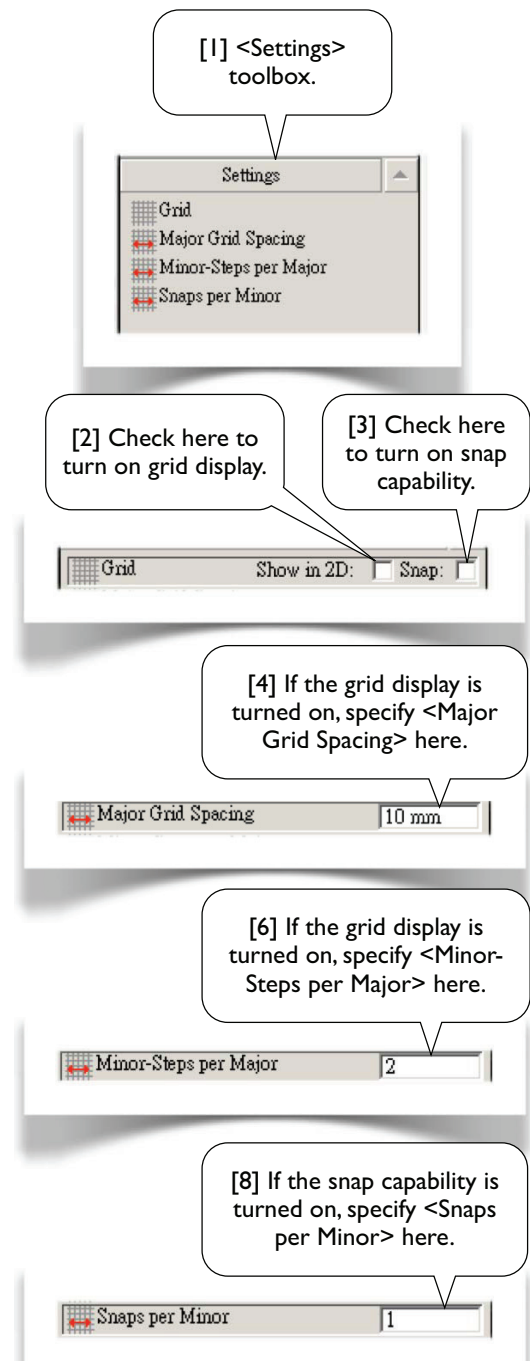
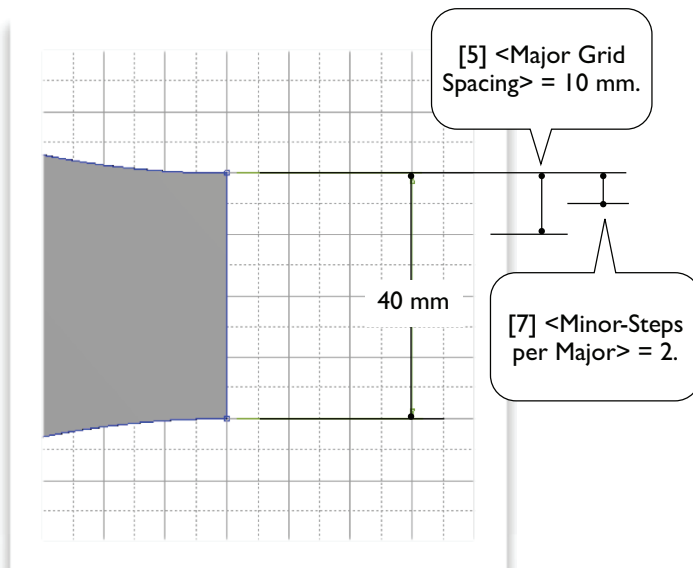
Allows you to specify <Major Grid Spacing> [4, 5] if the grid display is turned on.

Minor-Steps per Major

Allows you to specify <Minor-Steps per Major> [6, 7] if the grid display is turned on.

Snaps per Minor

Allows you to specify <Snaps per Minor> [8] if the snap capability is turned on.

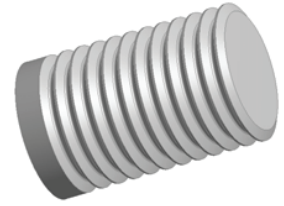


References

1. ANSYS Help System//DesignModeler//2D Sketching//Auto Constraints
2. ANSYS Help System//DesignModeler//2D Sketching//Constraints Toolbox//Auto Constraints :: 0
3. ANSYS Help System//DesignModeler//2D Sketching//Draw Toolbox
4. ANSYS Help System//DesignModeler//2D Sketching//Modify Toolbox
5. ANSYS Help System//DesignModeler//2D Sketching//Dimensions Toolbox
6. ANSYS Help System//DesignModeler//2D Sketching//Constraints Toolbox
7. ANSYS Help System//DesignModeler//2D Sketching//Settings Toolbox

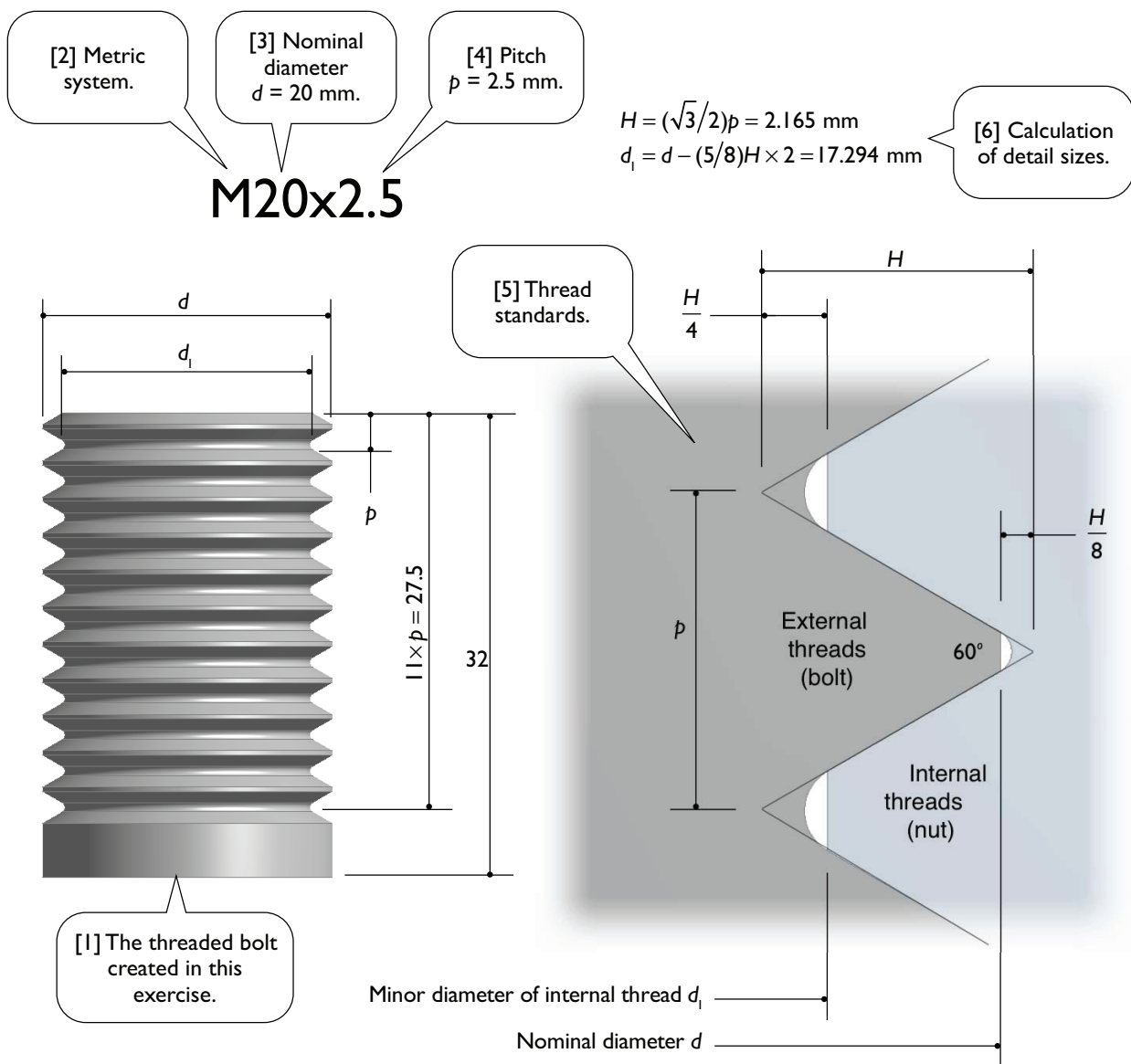
Section 2.4

More Exercise: M20x2.5 Threaded Bolt



2.4-1 About the M20x2.5 Threaded Bolt

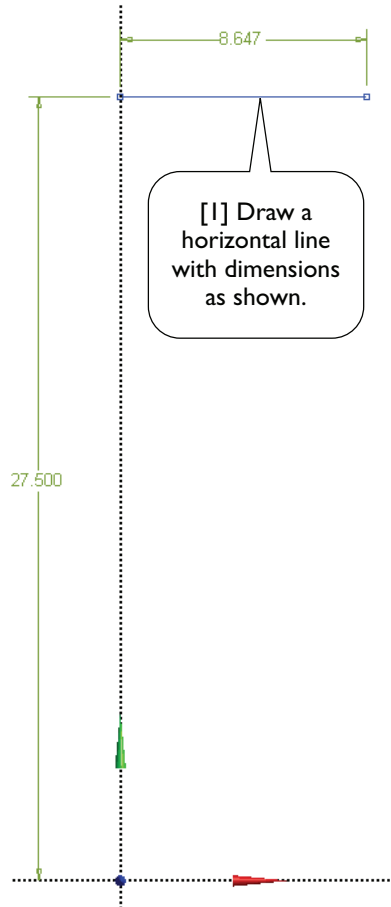
In a pair of threaded bolt-and-nut, the bolt has external threads while the nut has internal threads. This exercise is to create a sketch and revolve the sketch 360° to generate a 3D solid body representing a portion of the bolt threaded with M20x2.5 [1-6]. In Section 3.2, we will use this sketch again to generate a 2D solid body. The 2D body is then used for a static structural simulation.



2.4-2 Draw a Horizontal Line

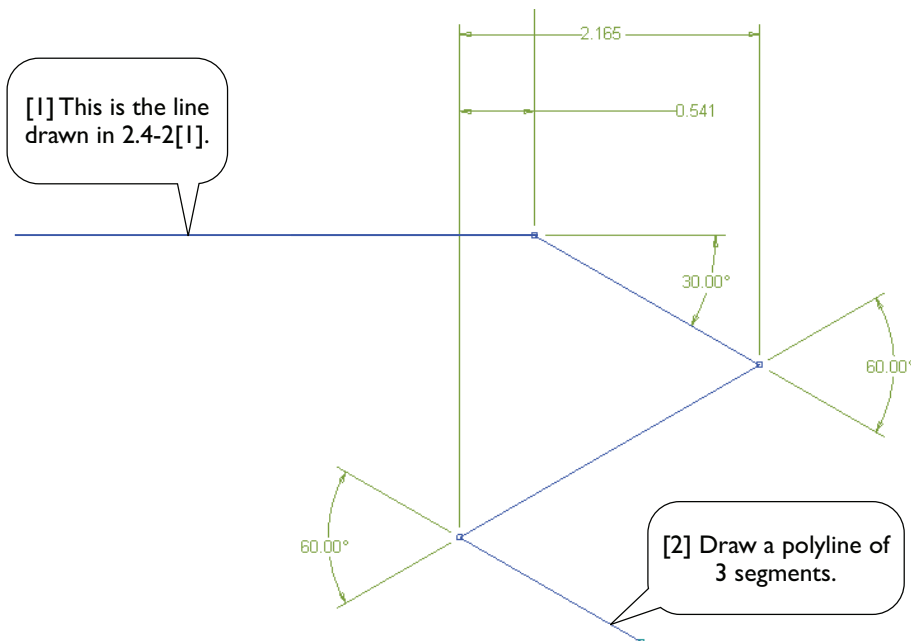
Launch Workbench and create a <Geometry> System. Save the project as "Threads." Start up <DesignModeler>. Select <Millimeter> as length unit.

Draw a horizontal line on <XYPlane>. Specify the dimensions as shown [1].



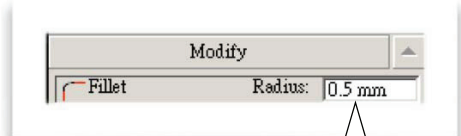
2.4-3 Draw a Polyline

Draw a polyline (totally 3 segments) and specify dimensions (30°, 60°, 60°, 0.541, and 2.165) as shown below [1-2]. To dimension angles, please refer to 2.3-8.



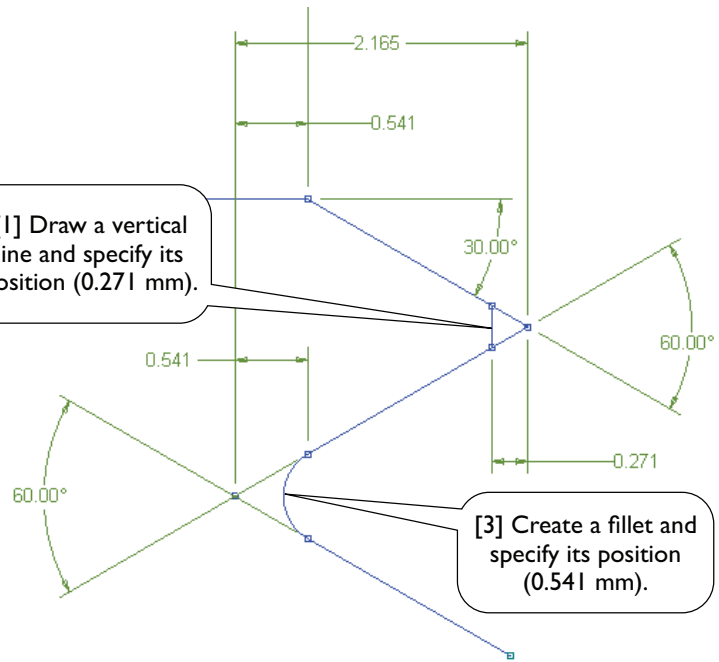
2.4-4 Draw Fillets

Draw a vertical line and specify its position (0.271 mm) [1]. Create a fillet and specify its position (0.541 mm) [2, 3].



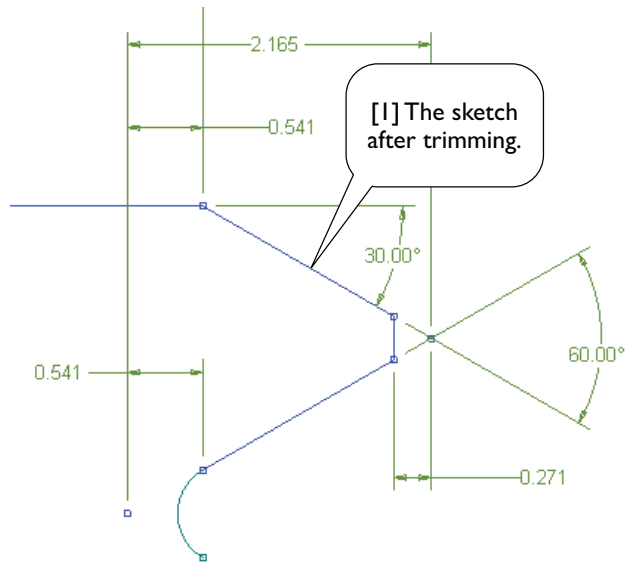
[2] Before creating fillets, specify an approximate radius value, say 0.5 mm.

[1] Draw a vertical line and specify its position (0.271 mm).

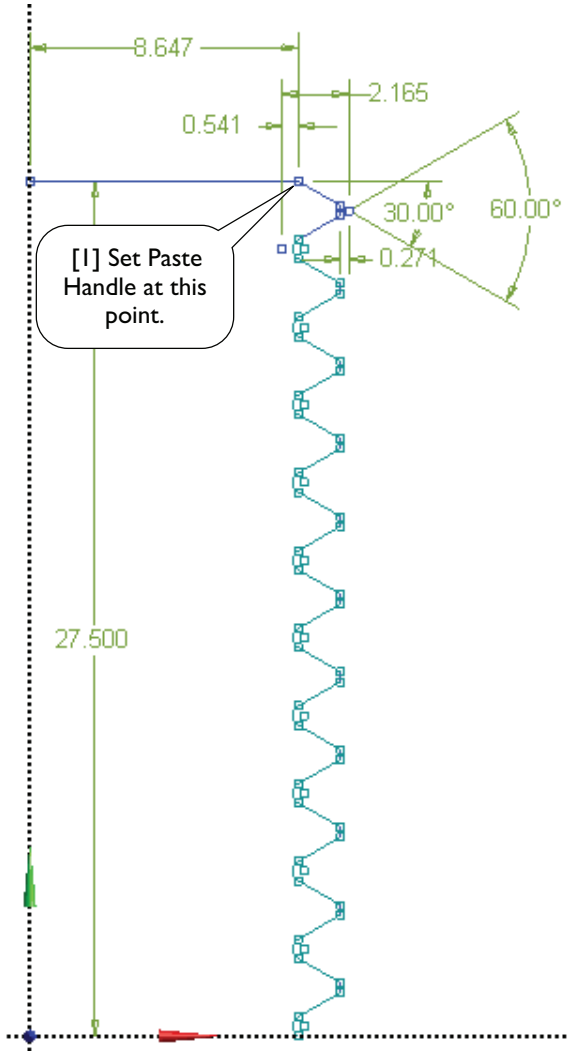


[3] Create a fillet and specify its position (0.541 mm).

2.4-5 Trim Unwanted Segments



[1] The sketch after trimming.



[1] Set Paste Handle at this point.

2.4-6 Replicate 10 Times

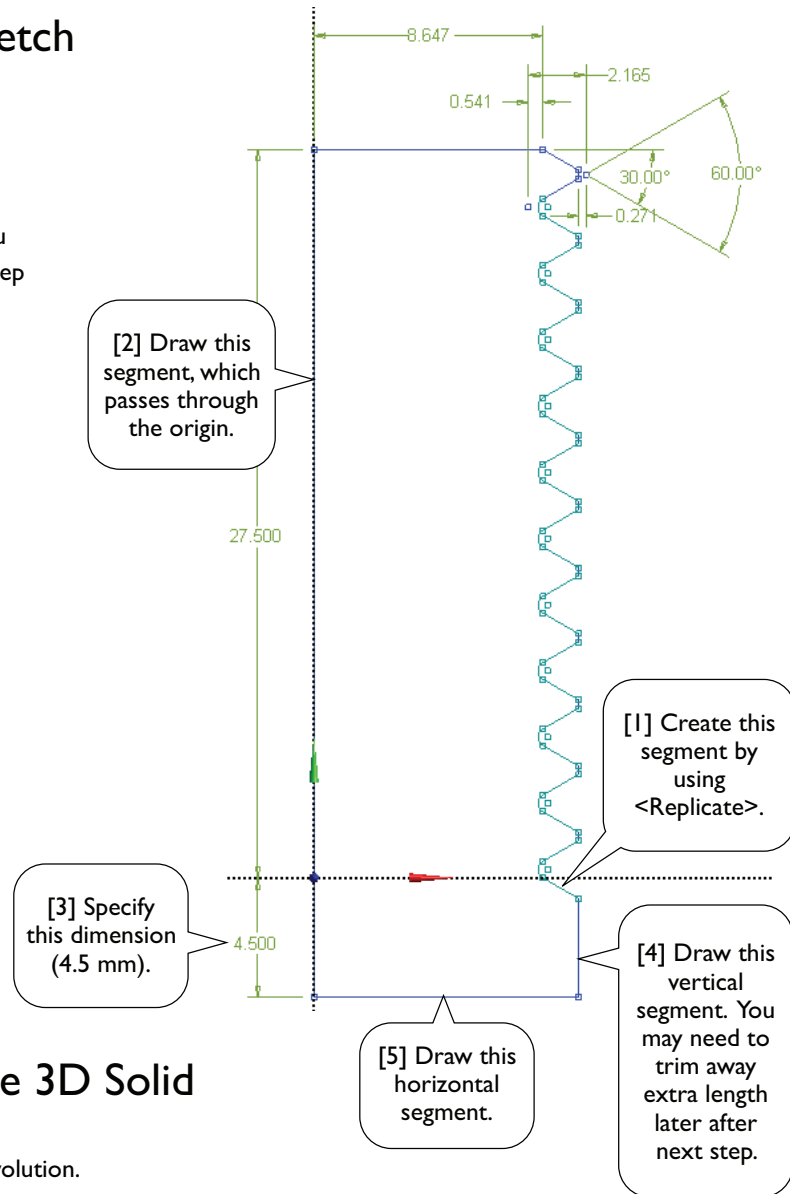
Select all segments except the horizontal line (totally 4 segments), and replicate 10 times. You may need to manually set the paste handle [1]. You may also need to use the tool <Selection Filter: Points> [2].

[2] <Selection Filter: Points>.



2.4-7 Complete the Sketch

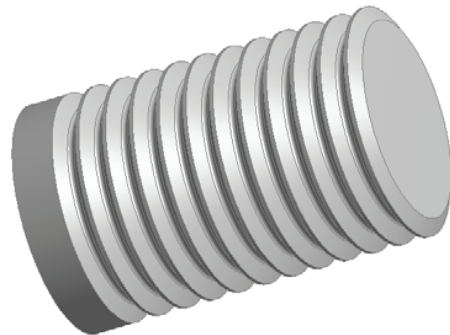
Follow steps [1-5] to complete the sketch. Note that, in step [4], you don't need to worry about the length. After step [5], you can trim the vertical segment created in step [4].



2.4-8 Revolve to Create 3D Solid

Click <Revolve> to generate a solid of revolution. Select the Y-axis as the axis of revolution. Don't forget to click <Generate>.

Save the project and exit from the Workbench. We will resume this project again in Section 3.2.



References

1. Zahavi, E., *The Finite Element Method in Machine Design*, Prentice-Hall, 1992; Chapter 7. Threaded Fasteners.
2. Deutschman, A. D., Michels, W. J., and Wilson, C. E., *Machine Design: Theory and Practice*, Macmillan Publishing Co., Inc., 1975; Section 16-6. Standard Screw Threads.

Section 2.5

More Exercise: Spur Gears

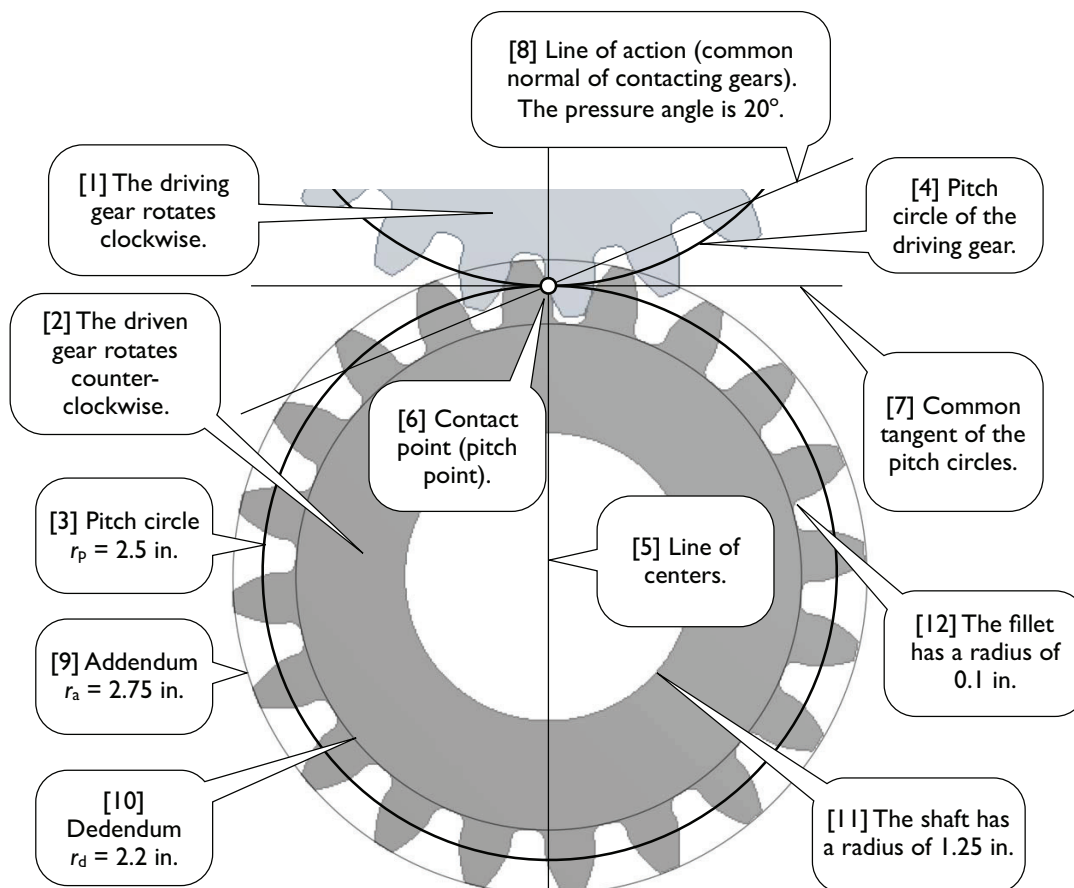


Geometric details of spur gears are essential for a mechanical engineer. However, if you are not interested in these geometric details for now, you may skip the first two subsections and jump directly to 2.5-3.

2.5-1 About the Spur Gears

The figure below shows a pair of identical spur gears in mesh [1-5]. Spur gears have their teeth cut parallel to the axis of the shaft on which the gears are mounted. In other words, spur gears are used to transmit power between parallel shafts. To maintain a constant angular velocity ratio, two meshing gears must satisfy a fundamental law of gearing: the shape of the teeth must be such that the common normal [8] at the point of contact between two teeth must always pass through a fixed point on the line of centers¹ [5]. This fixed point is called the *pitch point* [6].

The angle between the line of action [8] and the common tangent of the pitch circles [7] is known as the *pressure angle* [8]. The parameters defining a spur gear are its pitch radius ($r_p = 2.5$ in) [3], pressure angle ($\alpha = 20^\circ$) [8], and number of teeth ($N = 20$). The teeth are cut with a radius of addendum $r_a = 2.75$ in [9] and a radius of dedendum $r_d = 2.2$ in [10]. The shaft has a radius of 1.25 in [11]. The fillet has a radius of 0.1 in [12]. The thickness of the gear is 1.0 in.



2.5-2 About Involute Curves

To satisfy the fundamental law of gearing, most of gear profiles are cut to an *involute curve* [1]. The involute curve may be constructed by wrapping a string around a cylinder, called the *base circle* [2], and then tracing the path of a point on the string.

Given the gear's pitch radius r_p and pressure angle α , we can calculate the coordinates of each point on the involute curve. For example, consider an arbitrary point A [3] on the involute curve; we want to calculate its polar coordinates (r, θ) , as shown in the figure. Note that BA and CP are tangent lines of the base circle, and F is a foot of perpendicular.

Since APF is an involute curve and BCDEF is the base circle, by the definition of involute curve,

$$\widehat{BA} = \widehat{BC} + \widehat{CP} = \widehat{BCDEF} \quad (1)$$

$$\widehat{CP} = \widehat{CDEF} \quad (2)$$

From $\triangle OCP$,

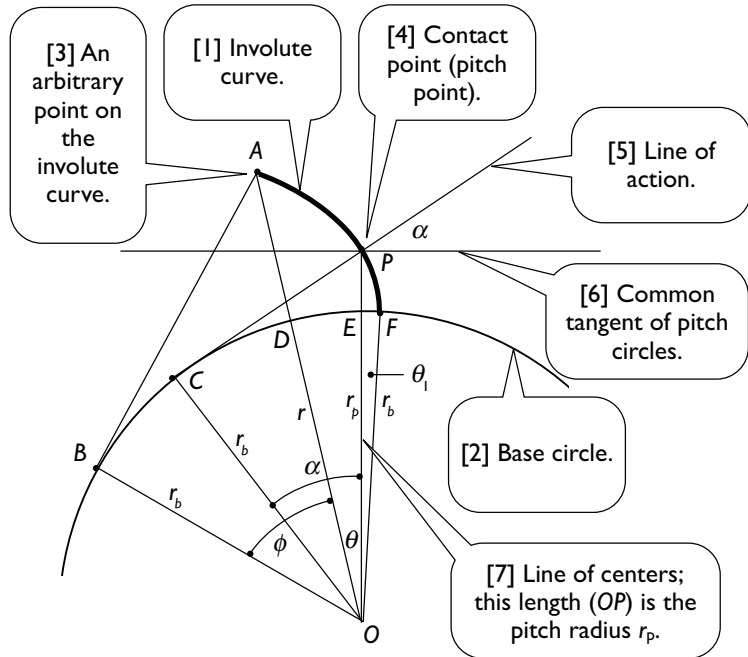
$$r_b = r_p \cos \alpha \quad (3)$$

From $\triangle OBA$,

$$r = \frac{r_b}{\cos \phi} \quad (4)$$

Or,

$$\phi = \cos^{-1} \frac{r_b}{r} \quad (5)$$



To calculate θ , we notice that

$$\widehat{DE} = \widehat{BCDEF} - \widehat{BCD} - \widehat{EF}$$

Dividing the equation with r_b and using Eq. (1),

$$\frac{\widehat{DE}}{r_b} = \frac{\widehat{BA}}{r_b} - \frac{\widehat{BCD}}{r_b} - \frac{\widehat{EF}}{r_b}$$

If radian is used, then the above equation can be written as

$$\theta = (\tan \phi) - \phi - \theta_1 \quad (6)$$

The last term θ_1 is the angle $\angle EOF$, which can be calculated by dividing Eq. (2) with r_b ,

$$\frac{\widehat{CP}}{r_b} = \frac{\widehat{CDEF}}{r_b}, \text{ or } \tan \alpha = \alpha + \theta_1, \text{ or}$$

$$\theta_1 = (\tan \alpha) - \alpha \quad (7)$$

Eqs. (3-7) are all we need to calculate polar coordinates (r, θ) . The polar coordinates can be easily transformed to rectangular coordinates, using O as origin and OP as y-axis,

$$x = -r \sin \theta, \quad y = r \cos \theta \quad (8)$$

Numerical Calculations

In our case, the pitch radius $r_p = 2.5$ in, and pressure angle $\alpha = 20^\circ$; from Eqs. (3) and (7) respectively,

$$r_b = 2.5 \cos 20^\circ = 2.349232 \text{ in}$$

$$\theta_1 = \tan 20^\circ - \frac{20^\circ}{180^\circ} \pi = 0.01490438$$

The calculated coordinates are listed in the table below. Notice that, in using Eqs. (6) and (7), radian is used as the unit of angles; in the table below, however, we translated the unit to degrees.

r in.	ϕ Eq. (5), degrees	θ Eq. (6), degrees	x	y
2.349232	0.000000	-0.853958	-0.03501	2.34897
2.449424	16.444249	-0.387049	-0.01655	2.44937
2.500000	20.000000	0.000000	0.00000	2.50000
2.549616	22.867481	0.442933	0.01971	2.54954
2.649808	27.555054	1.487291	0.06878	2.64892
2.750000	31.321258	2.690287	0.12908	2.74697

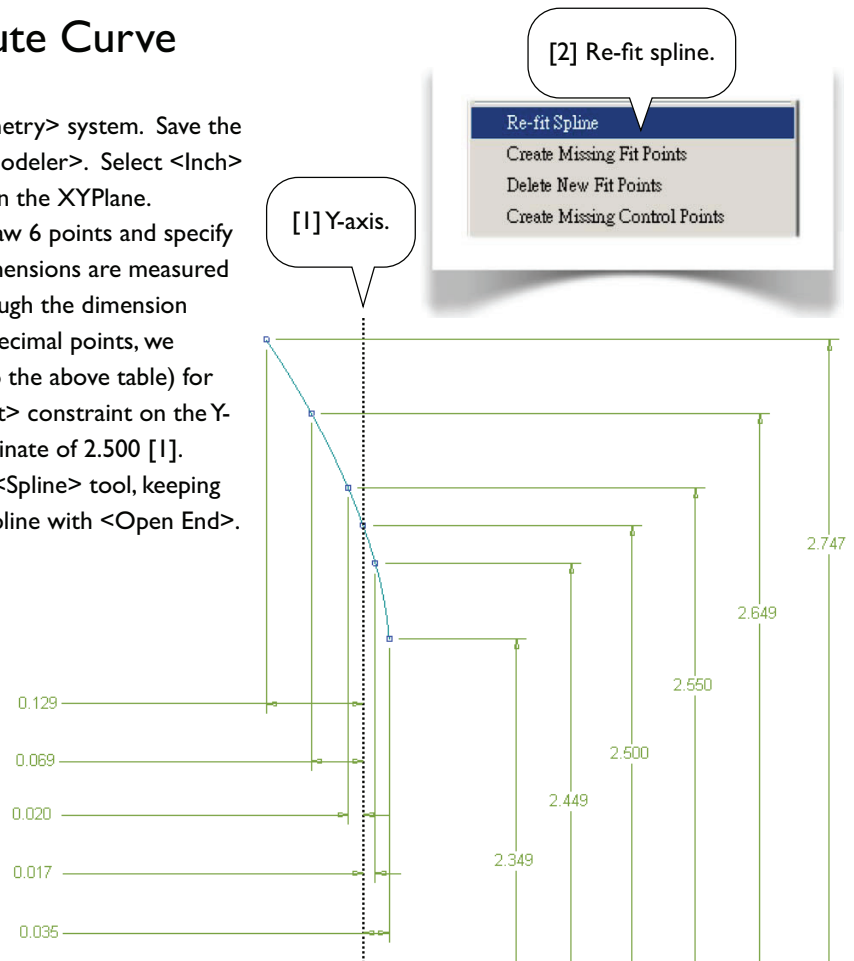
2.5-3 Draw an Involute Curve

Launch Workbench. Create a <Geometry> system. Save the project as "Gear." Start up <DesignModeler>. Select <Inch> as length unit. Start to draw sketch on the XYPlane.

Using <Construction Point>, draw 6 points and specify dimensions as shown (the vertical dimensions are measured down to the X-axis). Note that although the dimension values display with three digits after decimal points, we actually typed with five digits (refer to the above table) for more accuracy. Impose a <Coincident> constraint on the Y-axis for the point which has a Y-coordinate of 2.500 [1].

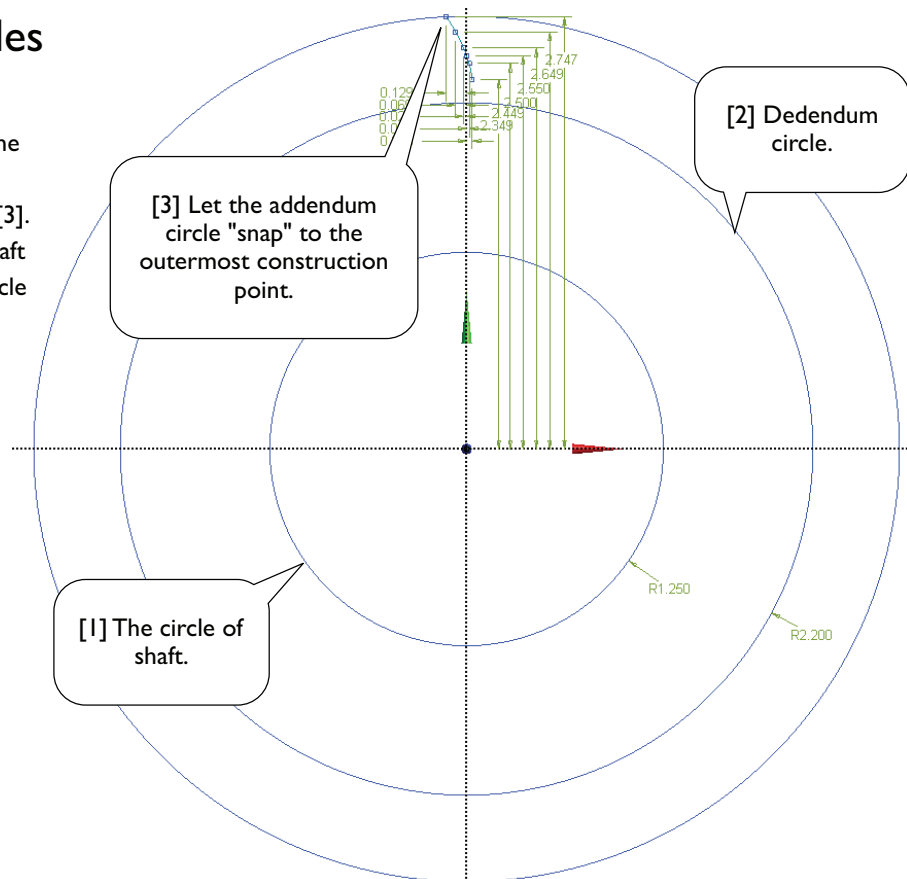
Connect these six points using <Spline> tool, keeping <Flexible> option on, and close the spline with <Open End>.

It is equally good that you draw the spline by using <Spline> tool directly without creating construction points first. To do that, issue <Open End with Fit Points> from the context menu at the end of <Spline> tool. After dimensioning each fitting points, use <Spline Edit> tool to edit the spline and issue <Re-fit Spline> [2].



2.5-4 Draw Circles

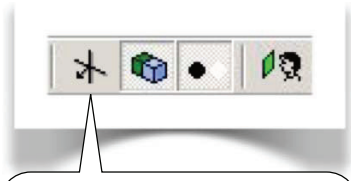
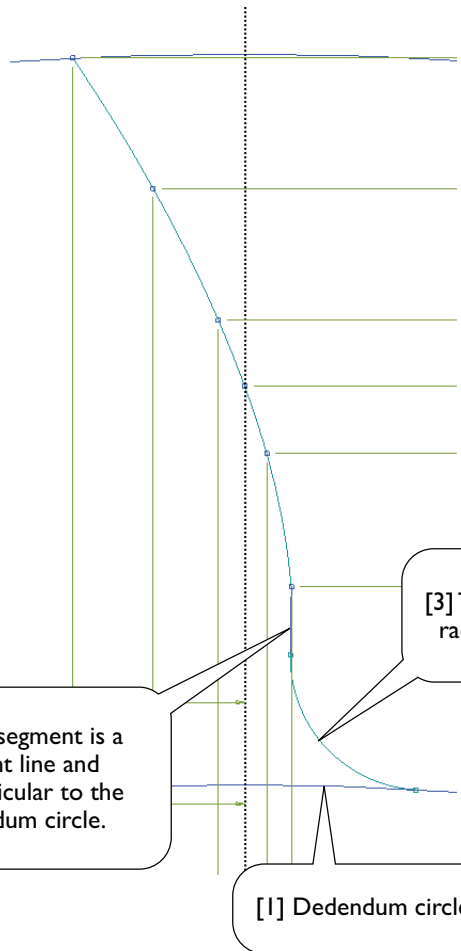
Draw three circles [1-3]. Let the addendum circle "snap" to the outermost construction point [3]. Specify radii for the circle of shaft (1.25 in) and the dedendum circle (2.2 in).



2.5-5 Complete the Profile

Draw a line starting from the lowest construction point, and make it perpendicular to the dedendum circle [1-2]. Note that, when drawing the line, avoid a <V> auto-constraint, (since this line is NOT vertical). Draw a fillet [3] of radius 0.1 in to complete the profile of a tooth.

Sometimes, turning off <Display Plane> may be helpful to clear up the graphics area. In this case, all the dimensions referring the plane axes disappear [4].

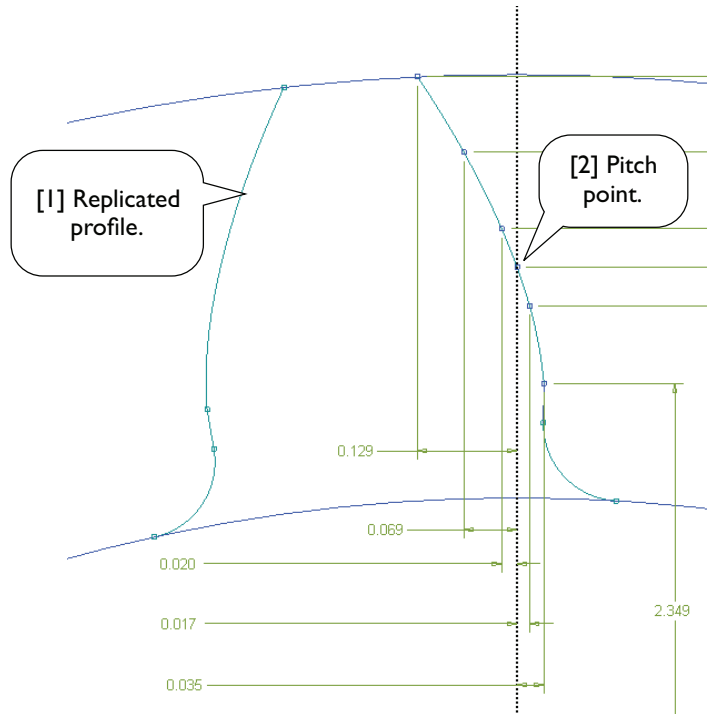


[4] Turn off <Display Plane> to clear up the graphics area.

2.5-6 Replicate the Profile

Activate <Replicate> tool, type 9 (degrees) for <r>. Select the profile (totally 3 segments), <End/Use Plane Origin as Handle>, <Flip Horizontal>, <Rotate by r degrees>, and <Paste at Plane Origin> [1]. End <Replicate> tool by pressing <ESC>.

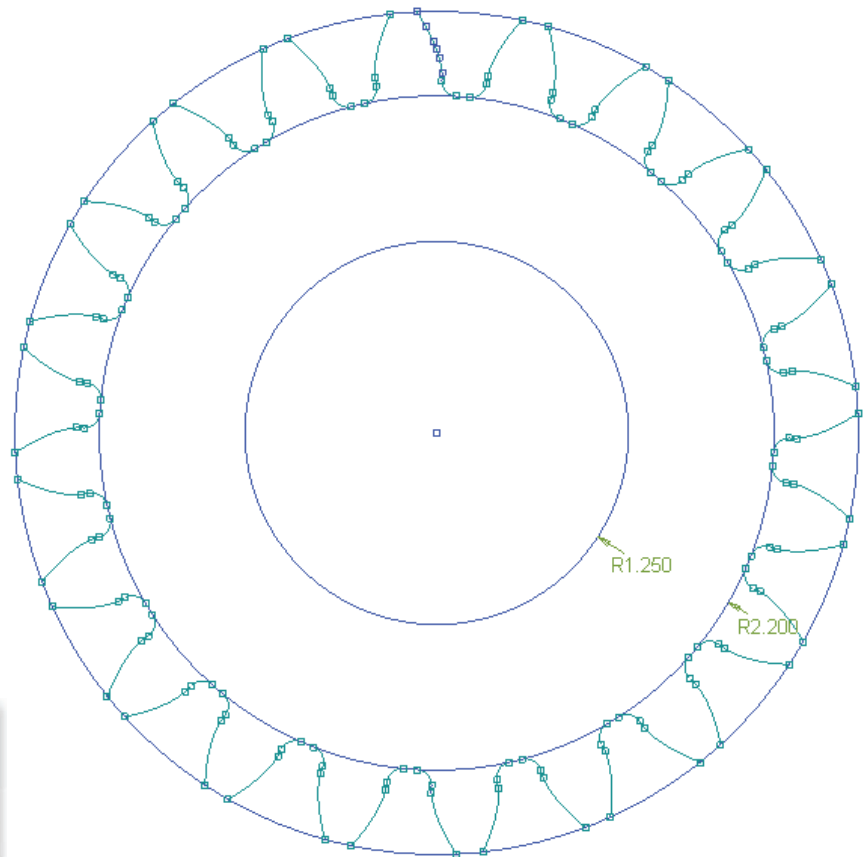
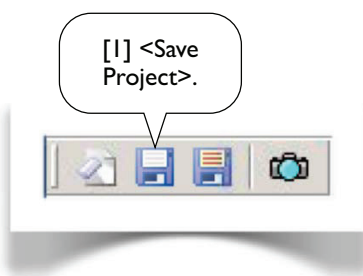
Note that the gear has 20 teeth, each spans by 18 degrees. The angle between the pitch points [2] on the left and the right profiles is 9 degrees.



2.5-7 Replicate Profiles 19 Times

Activate <Replicate> tool again, type 18 (degrees) for <r>. Select both left and right profiles (totally 6 segments), <End/Use Plane Origin as Handle>, <Rotate by r degrees>, and <Paste at Plane Origin>. Repeat the last two steps (rotating and pasting) until fill-in a full circle (totally 20 teeth).

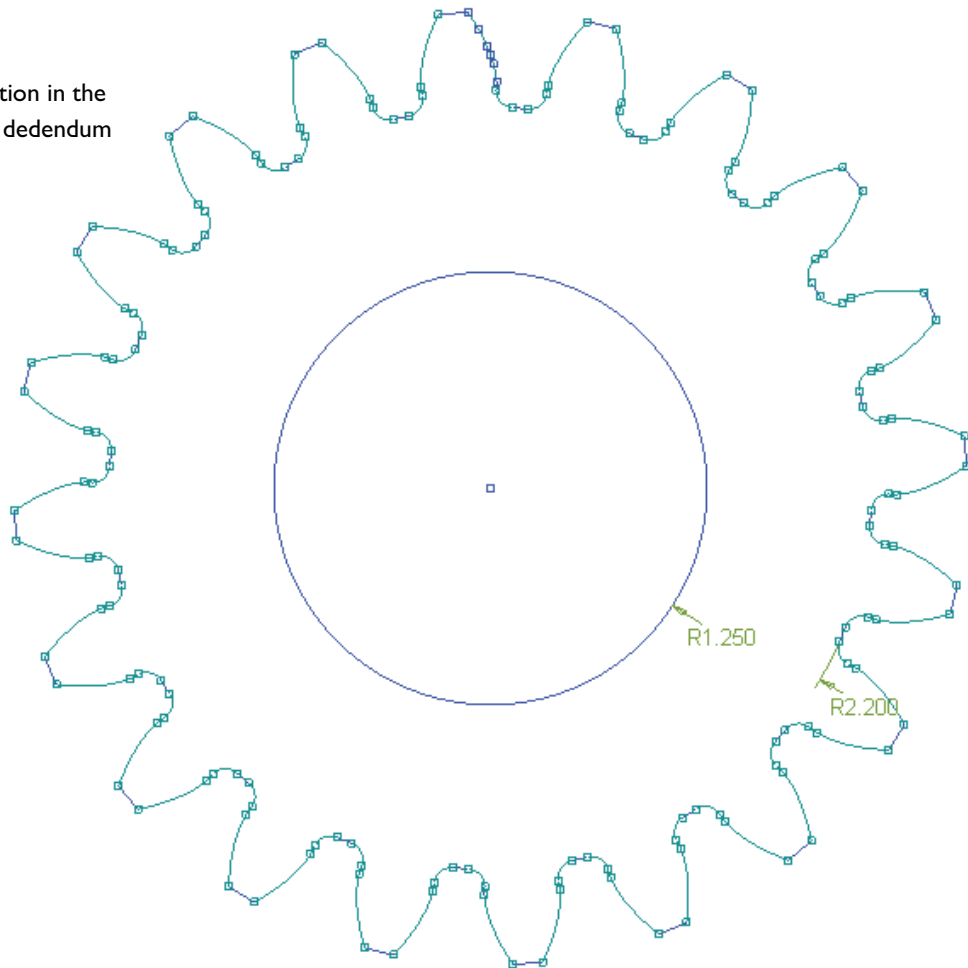
Save your project by clicking <Save Project> tool in the toolbar.



2.5-8 Trim Away Unwanted Segments

Trim away unwanted portion in the addendum circle and the dedendum circle.

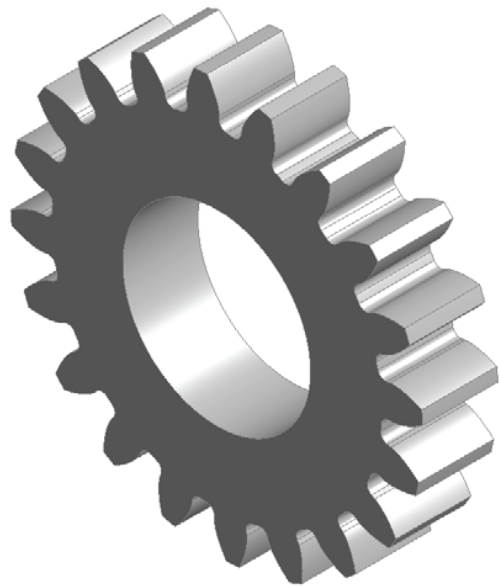
Remember, turning off <Display Plane> also turns off all the dimensions referring the plane axes (2.5-5[4]).



2.5-9 Extrude to Create 3D Solid

Extrude the sketch 1.0 inch to create a 3D solid as shown. Save the project and exit from Workbench. We will resume this project again in Section 3.4.

It is equally good that you create a single tooth (a 3D solid body) and then duplicate it by using <Create/Pattern> in <Modeling> mode. In this exercise, however, we use <Replicate> in <Sketching> mode because our purpose in this chapter is to practice sketching techniques.

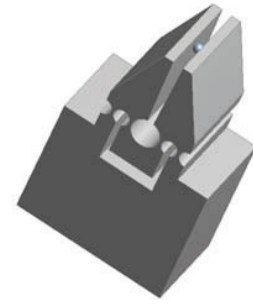


References

1. Deutschman, A. D., Michels, W. J., and Wilson, C. E., *Machine Design: Theory and Practice*, Macmillan Publishing Co., Inc., 1975; Chapter 10. Spur Gears.
2. Zahavi, E., *The Finite Element Method in Machine Design*, Prentice-Hall, 1992; Chapter 9. Spur Gears.

Section 2.6

More Exercise: Microgripper^{1,2}

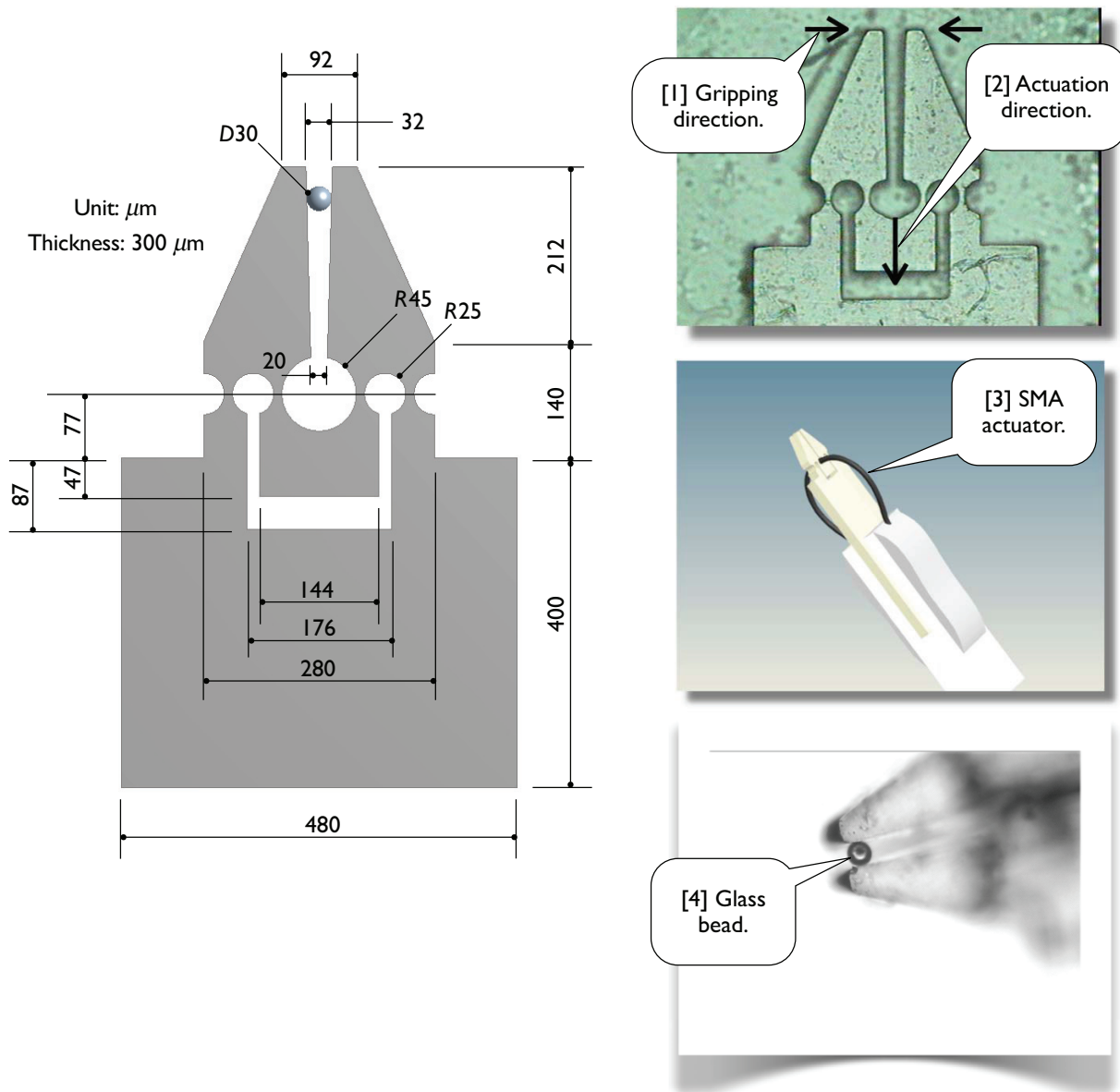


2.6-1 About the Microgripper

The microgripper is made of PDMS (polydimethylsiloxane, see 1.1-1). The device is actuated by a shape memory alloy (SMA) actuator [1-3], of which the motion is caused by temperature change, and the temperature is in turn controlled by electric current.

In the lab, the microgripper is tested by gripping a glass bead of a diameter of 30 micrometer [4].

In this section, we will create a solid model for the microgripper. The model will be used for simulation in Section 13.3 to assess the gripping forces on the glass bead under the actuation of SMA actuator.



2.6-2 Create Half of the Model

Launch Workbench. Create a <Geometry> system. Save the project as "Microgripper." Start up <DesignModeler>. Select <Micrometer> as length unit.

Draw a sketch on <XYPlane> as shown [1]. Note that two of the three circles have equal radii. Trim away unwanted segments as shown [2]. Also note that we drew half of the model, due to the symmetry. Extrude the sketch 150 microns both sides of the plane symmetrically (total depth is 300 microns) [3]. So far we have half of the gripper [4].

Details View

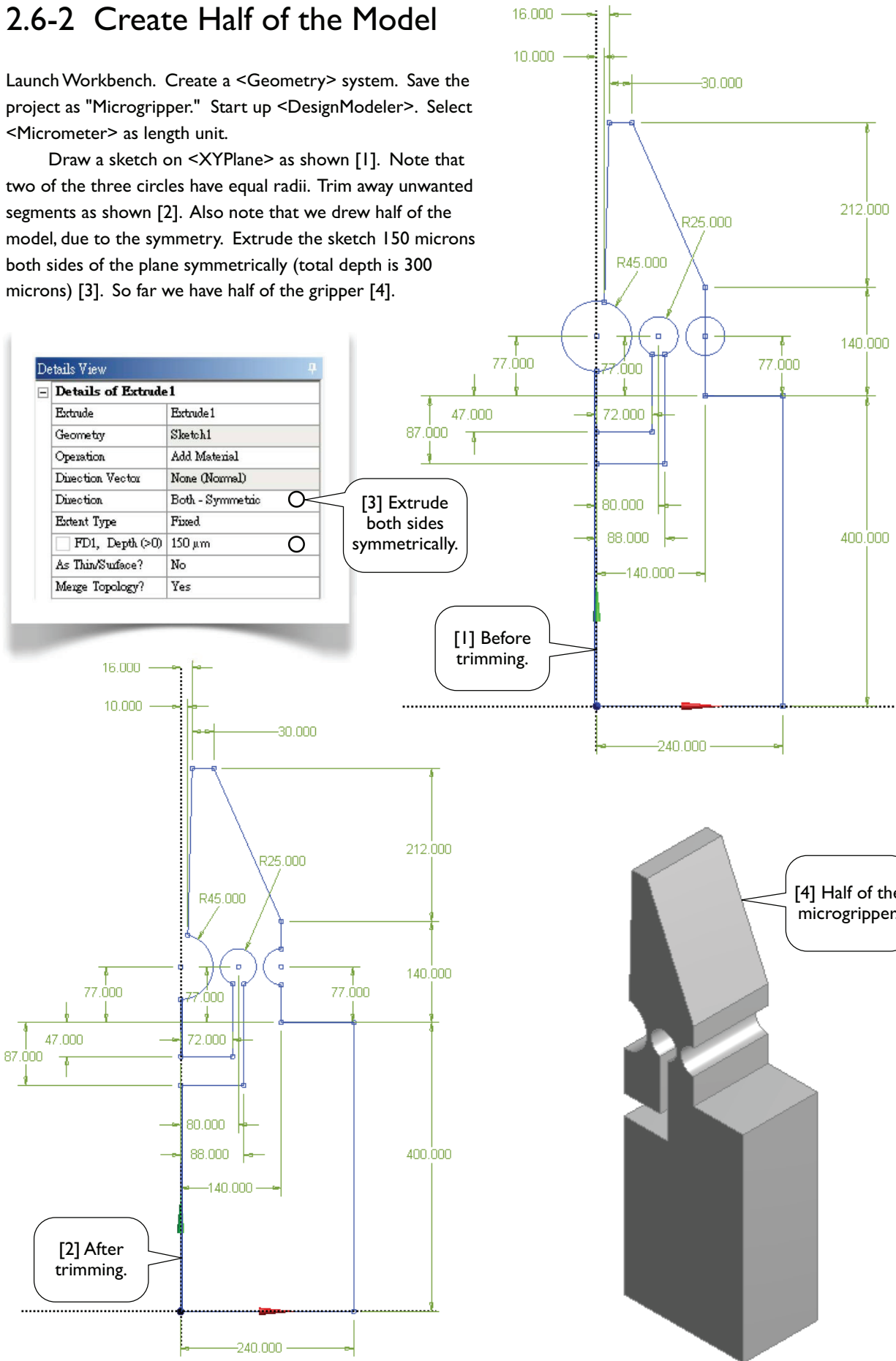
Details of Extrude1	
Extrude	Extrude1
Geometry	Sketch.1
Operation	Add Material
Direction Vector	None (Normal)
Direction	Both - Symmetric
Extent Type	Fixed
<input type="checkbox"/> FD1, Depth (>0)	150 μm
As Thin/Surface?	No
Merge Topology?	Yes

[3] Extrude both sides symmetrically.

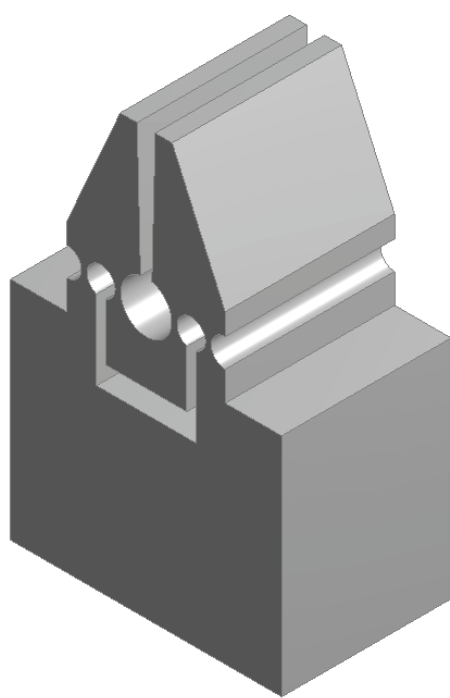
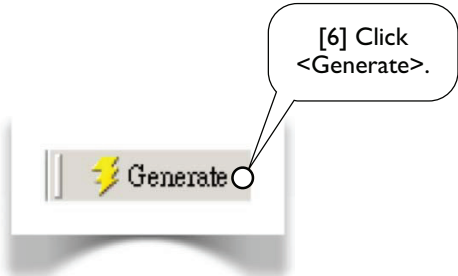
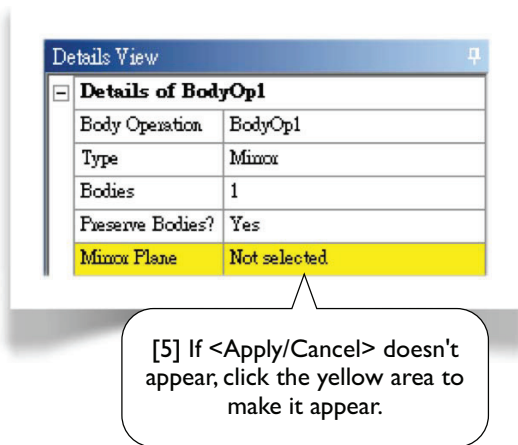
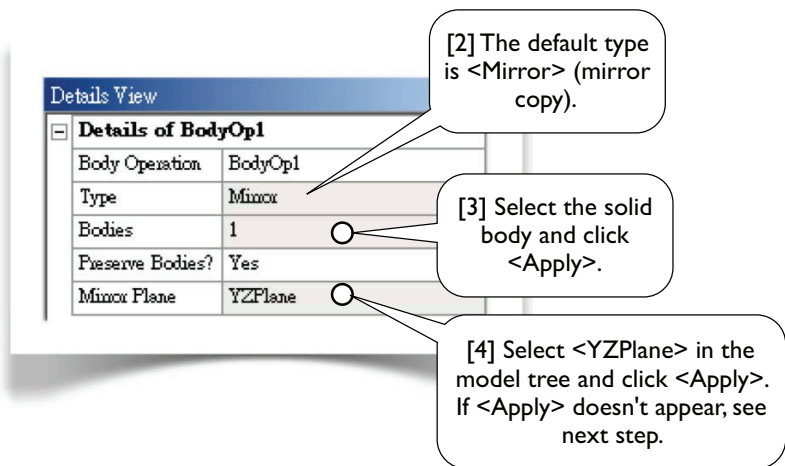
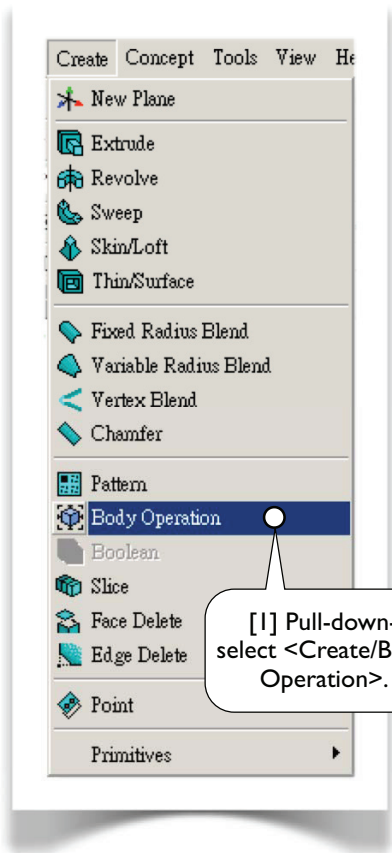
[1] Before trimming.

[2] After trimming.

[4] Half of the microgripper.

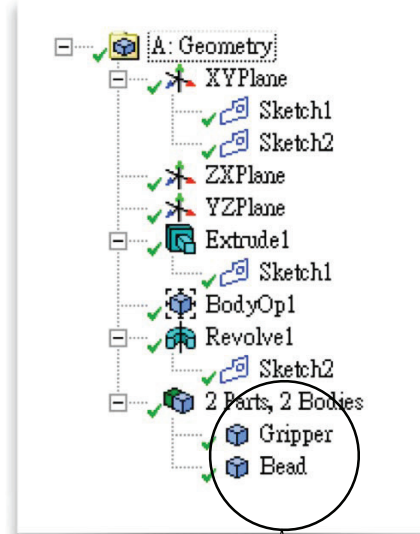
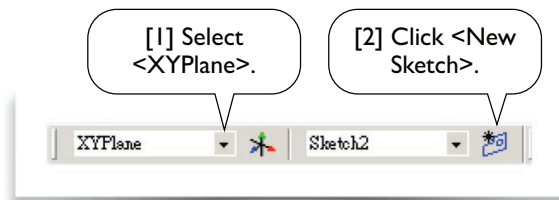
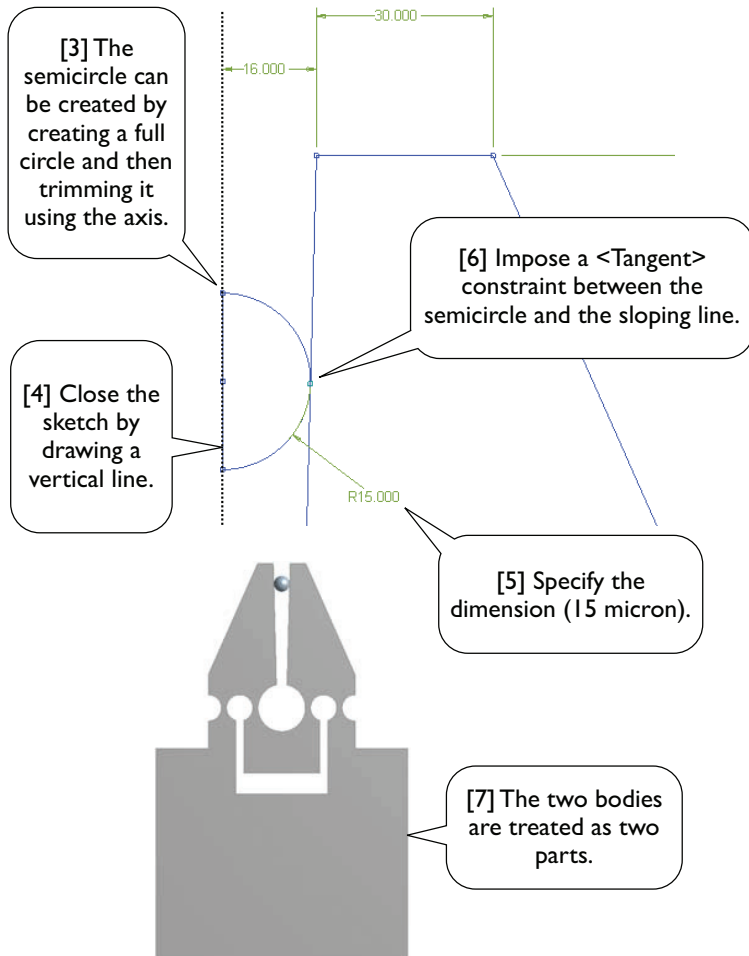


2.6-3 Mirror Copy the Solid Body



2.6-4 Create the Bead

Create a new sketch on XYPlane [1, 2] and draw a semicircle as shown [3-6]. Revolve the sketch 360 degrees to create the glass bead. Note that the two bodies are treated as two parts [7]. Rename two bodies [8].



Wrap Up

Close <DesignModeler>, save the project and exit Workbench. We will resume this project in Section 13.3.

References

1. Chang, R. J., Lin, Y. C., Shiu, C. C., and Hsieh, Y. T., "Development of SMA-Actuated Microgripper in Micro Assembly Applications," IECON, IEEE, Taiwan, 2007.
2. Shih, P. W., *Applications of SMA on Driving Micro-gripper*, MS Thesis, NCKU, ME, Taiwan, 2005.

Section 2.7

Review

2.7-1 Keywords

Sketching Mode

An environment under DesignModeler, configured for drawing sketches on planes.

Modeling Mode

An environment under DesignModeler, configured for creating 3D or 2D bodies.

Sketching Plane

The plane on which a sketch is created. Each sketch must be associated with a plane; each plane may have multiple sketches on it. Usage of planes is not limited for storing sketches.

Edge

In <Sketching Mode>, an edge may be a (straight) line or a curve. A curve may be a circle, ellipse, arc, or spline.

Sketch

A sketch consists of points and edges. Dimensions and constraints may be imposed on these entities.

Model Tree

A model tree is the structured representation of a geometry and displayed on <Tree Outline> in <DesignModeler>. A model tree consists of features and a part branch, in which their order is important. The parts are the only objects exported to <Mechanical>.

Branch

A branch is an object of a model tree and consists one or more objects under itself.

Object

A leaf or branch of a model tree is called an object.

Context Menu

The menu that pops up when you right-click your mouse. The contents of the menu depend on what you click.

Auto Constraints

While drawing in <Sketching Mode>, by default, <DesignModeler> attempts to detect the user's intentions and try to automatically impose constraints on points or edges. Detection is performed over entities on the active plane, not just active sketch. <Auto Constraints> can be switched on/off in <Constraints> toolbox.

Selection Filter

A selection filter filters one type of geometric entities. When a selection filter is turned on/off, the corresponding type of entities becomes selectable/unselectable. In <Sketching> mode, there are two selection filters, namely points and edges filters. Along with these two filters, face and body selection filters are available in <Modeling> mode.

Paste Handle

A reference point used in a copy/paste operation. The point is defined during copying and will be aligned at a specified location when pasting.

Constraint Status

In <Sketching> mode, entities are color coded to indicate their constraint status: greenish-blue for under-constrained; blue and black for well constrained (i.e., fixed in the space); red for over-constrained; gray for inconsistent.

2.7-2 Additional Workbench Exercises

Create Models with Your Own Way

After so many exercises, you should be able to figure out many alternative ways of creating the geometric models in this chapter. Try to re-create these models with your own way.