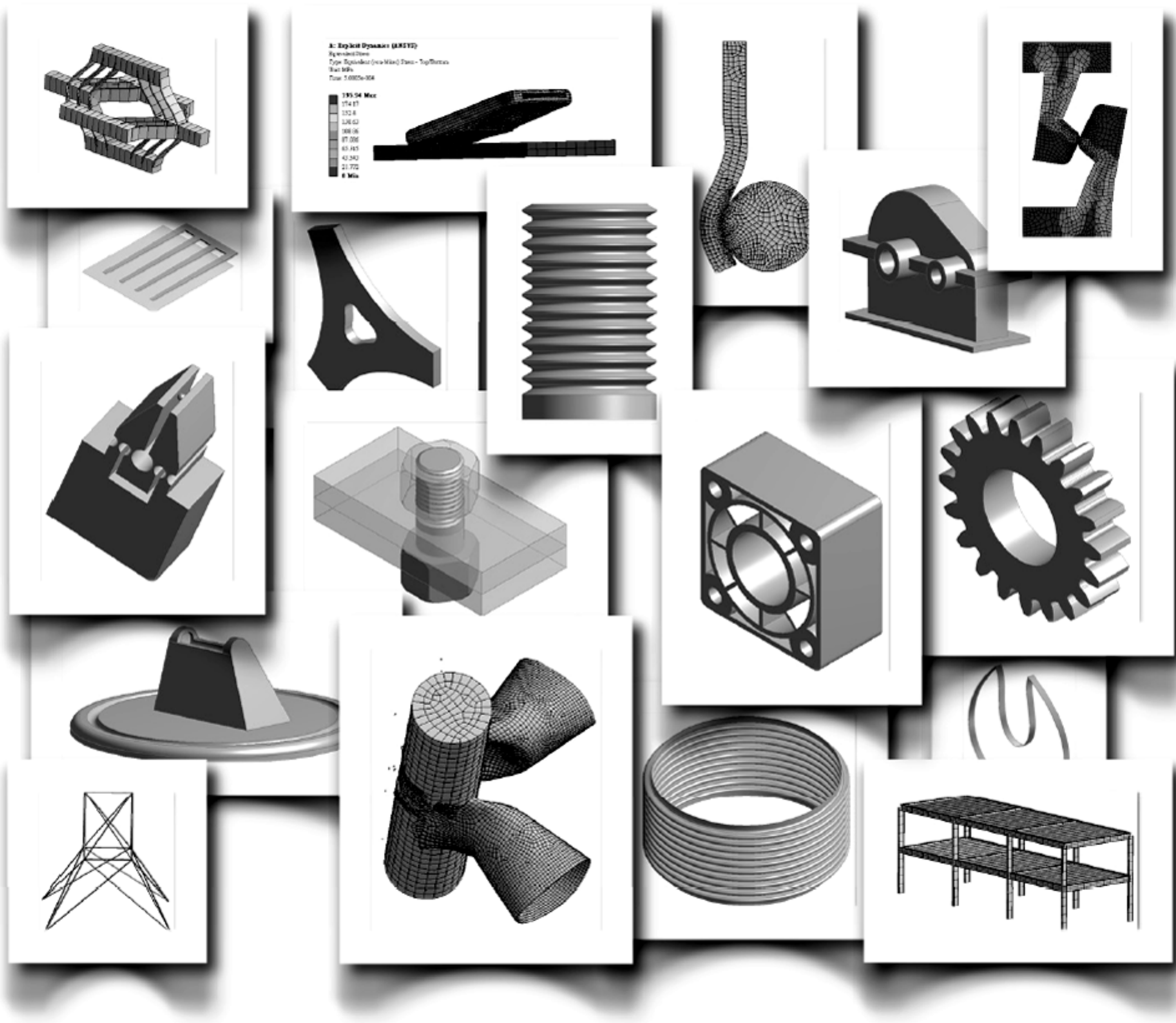
Finite Element Simulations with ANSYS Workbench 12

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 proficient at simulations, an engineer has to be proficient at geometric modeling first. In a simulation project, it is not uncommon to take the majority of human-hours to create a geometric model, that is particularly true in a 3D simulation.

A complex 3D geometry can be viewed as a collection of simpler 3D solid bodies. Each solid body is often created by first drawing a sketch on a plane, and then the sketch is used to generate the 3D solid body using tools 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. Five mechanical parts are sketched in this chapters. Although each sketch is used to generate a 3D models, the generation of 3D models is so trivial that we should be able to focus on the 2D sketches without being distracted. More exercises of sketching will be provided in later chapters.

About Each Section

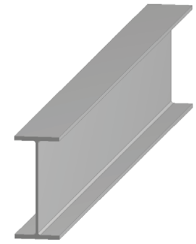
Each sketch of a mechanical part will be completed in a section. Sketches in the first two sections are guided in a step-by-step fashion. Section 1 sketches a cross section of W16x50; the cross section is then extruded to generate a solid model in 3D space. Section 2 sketches a triangular plate; the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

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. Sketches in these sections are in a not-so-step-by-step fashion; we purposely leave some room for the students to figure out the details.

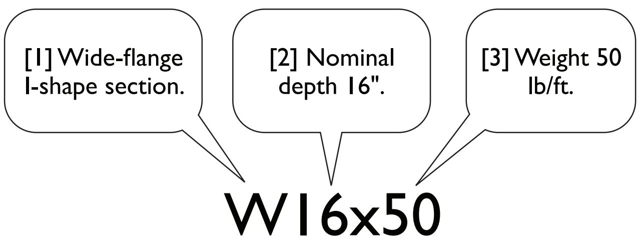
Section 2.1

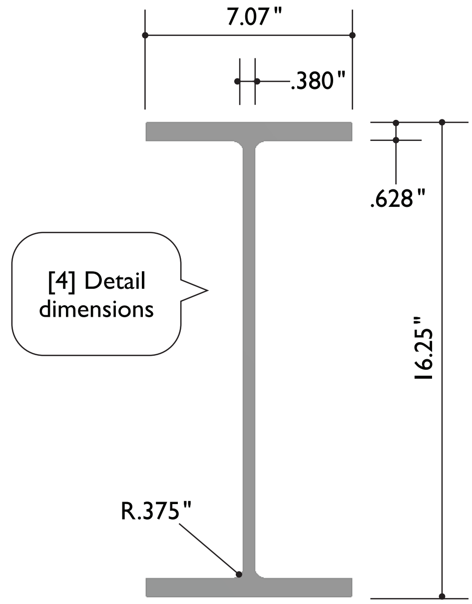
Step-by-Step: W16x50 Beam



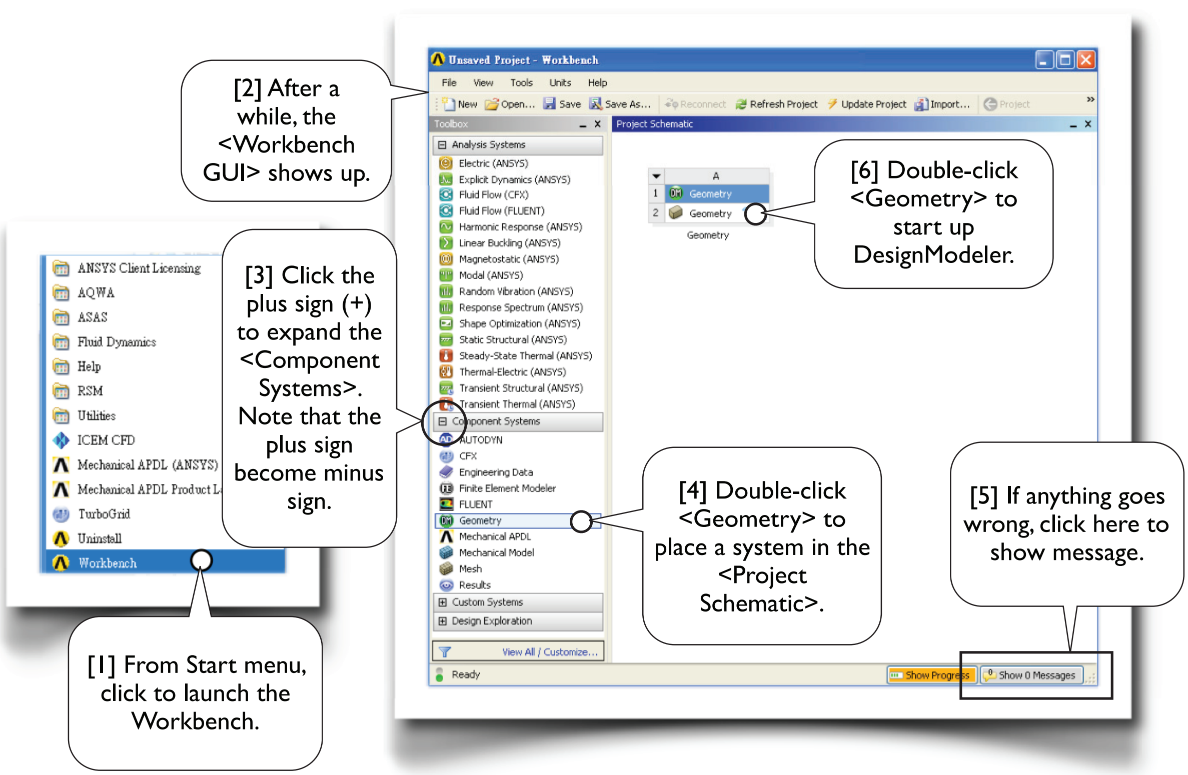
2.1-1 About the W16x50 Beam

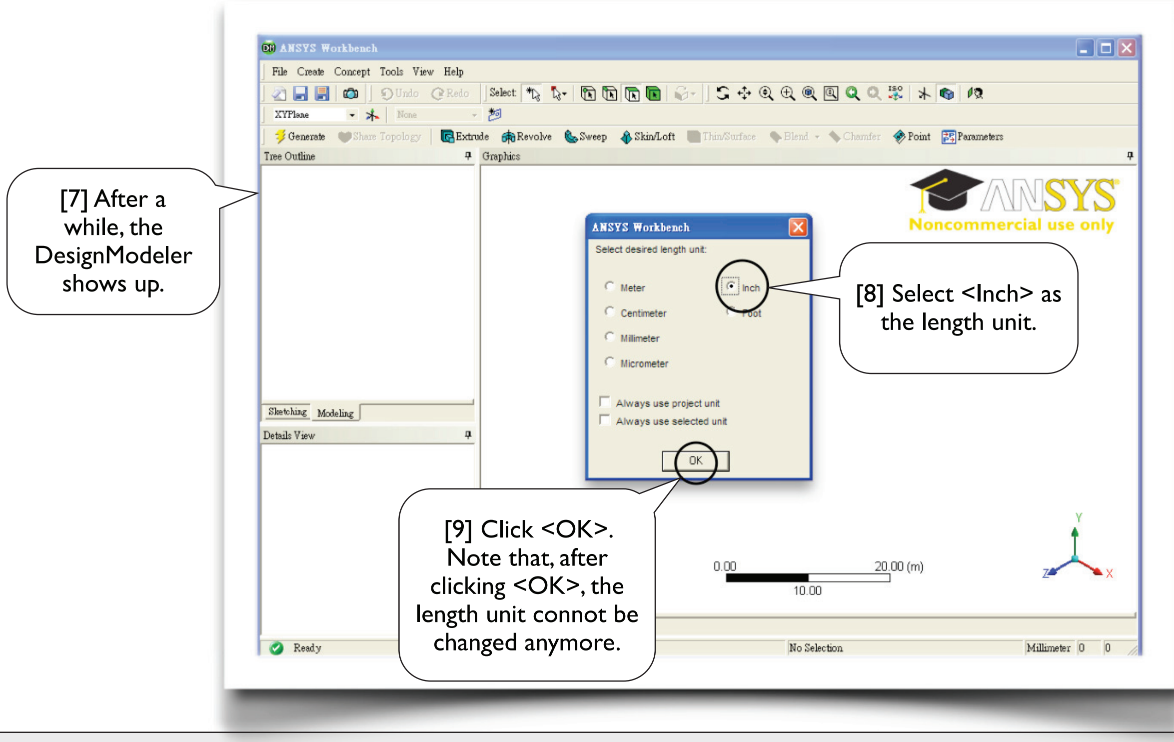
Consider a structural steel beam with a W16x50 cross-section [1-4] and a length of 10 ft. In this section, we will create a 3D solid body for the steel beam.





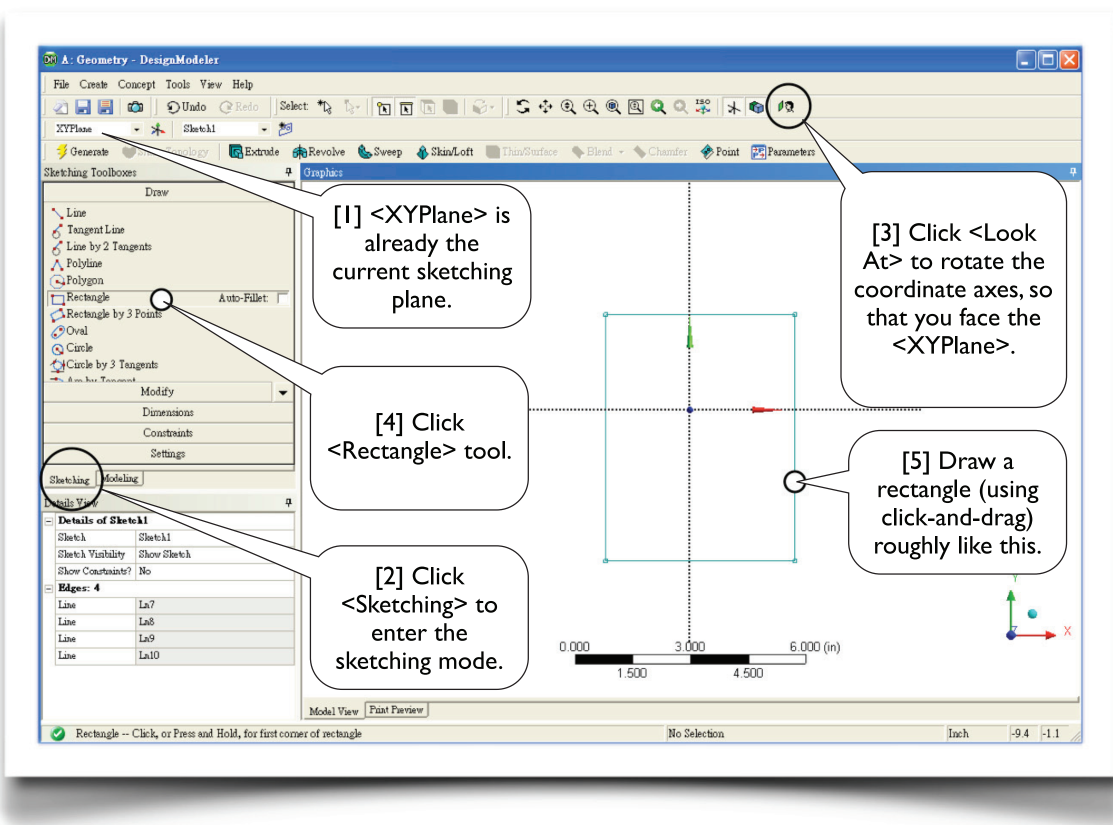
2.1-2 Start Up <DesignModeler>



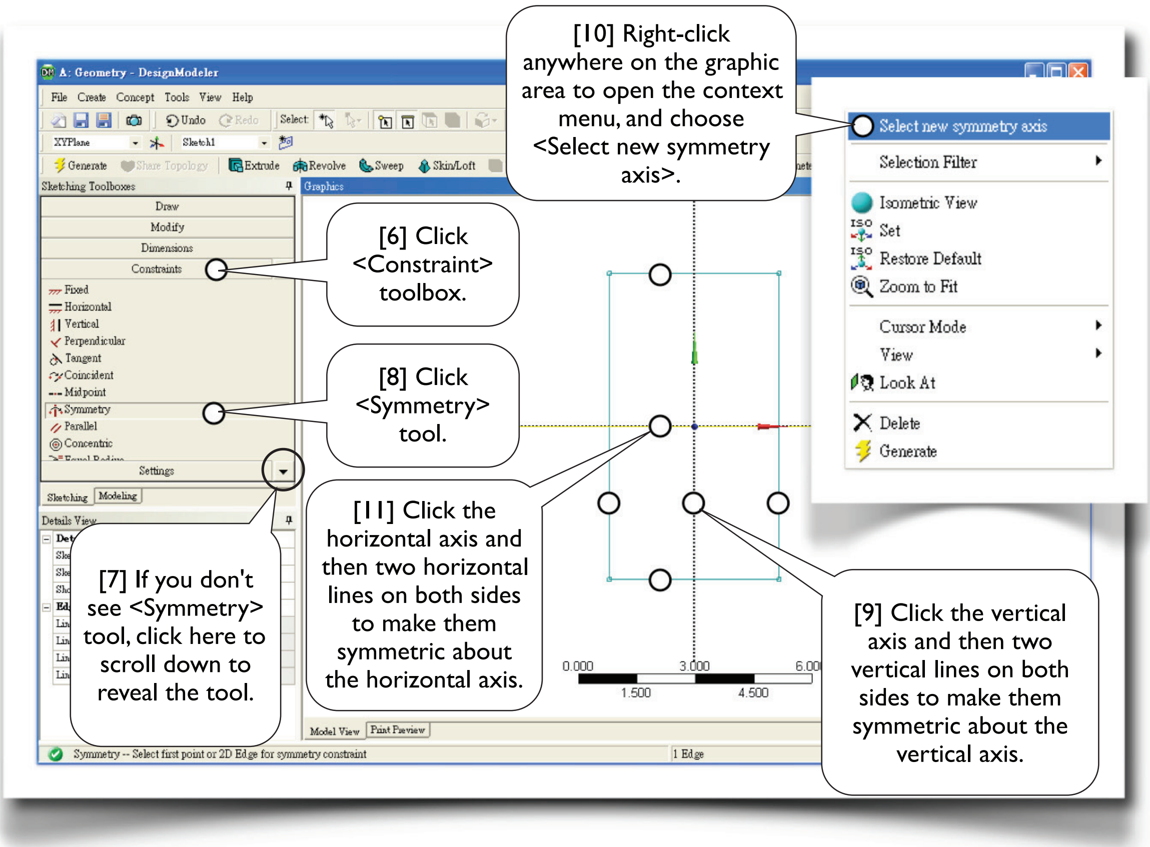


Notes: In a step-by-step exercise, whenever a circle is used with a speech bubble, it is to indicate that mouse or keynoard ACTIONS must be taken in that step (e.g., [1, 3, 4, 6, 8, 9]). The circle may be small or large, filled with white color or unfilled, depending on whichever gives more information. A speech bubble without a circle (e.g., [2, 7]) or with a rectangle (e.g., [5]) is used for commentary only, no mouse or keyboard actions are needed.

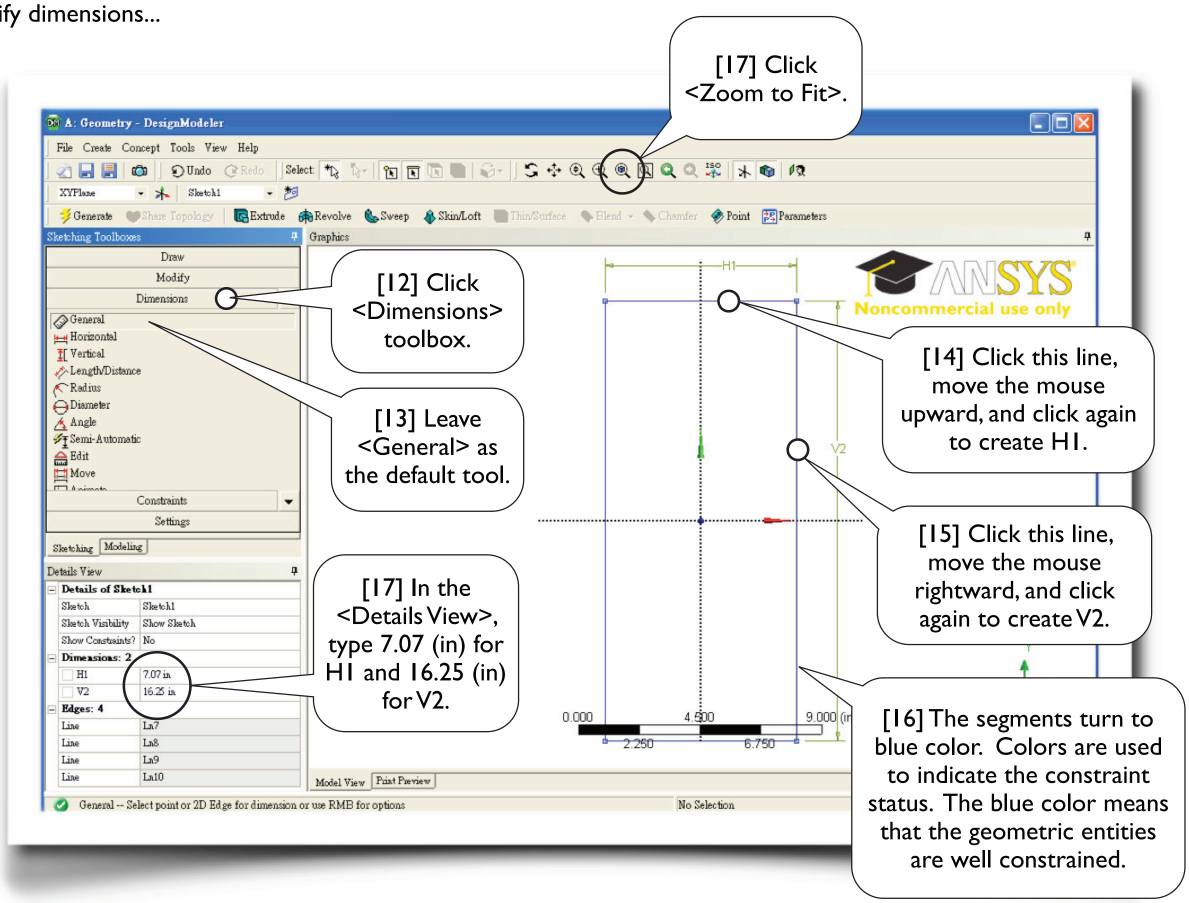
2.1-3 Draw a Rectangle on <XYPlane>



Impose symmetry constraints...

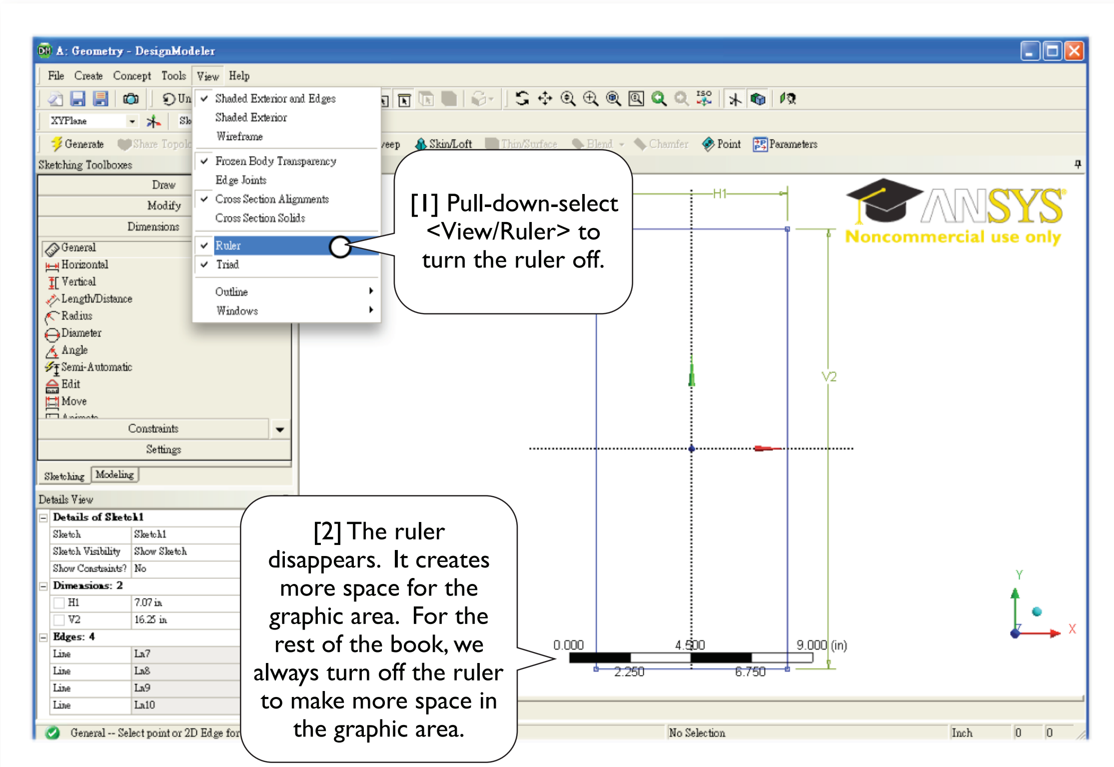


Specify dimensions...

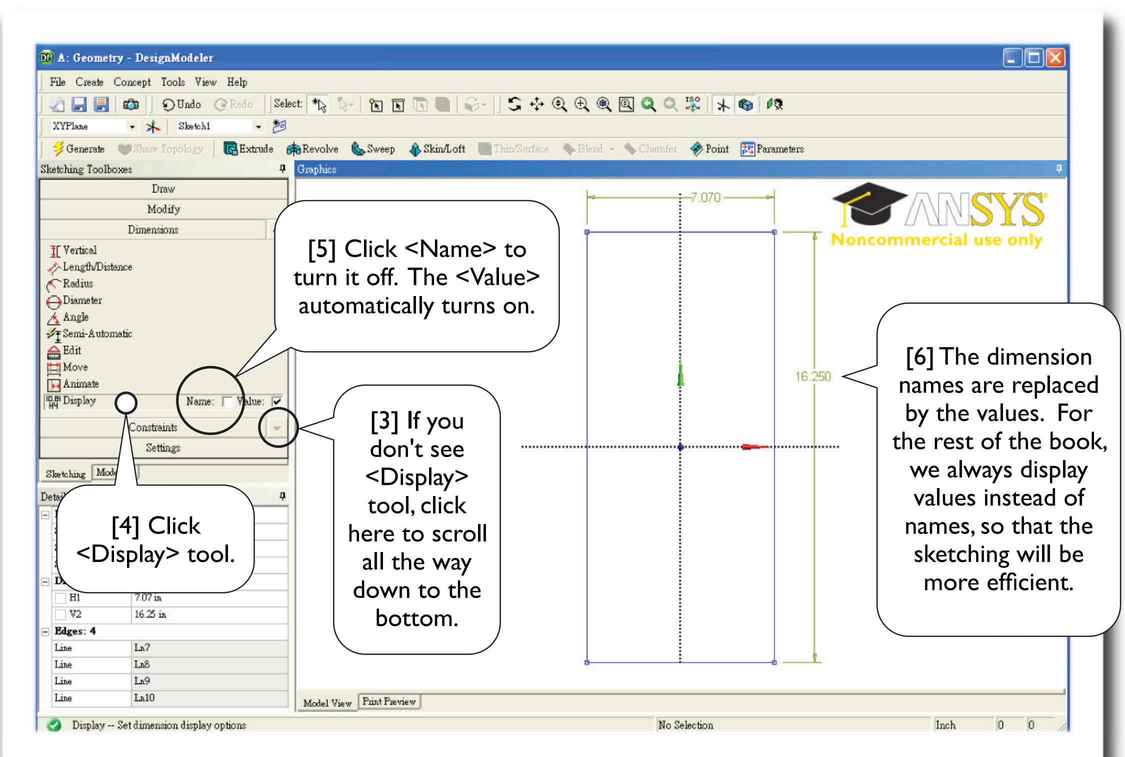


2.1-4 Clean up the Graphic Area

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

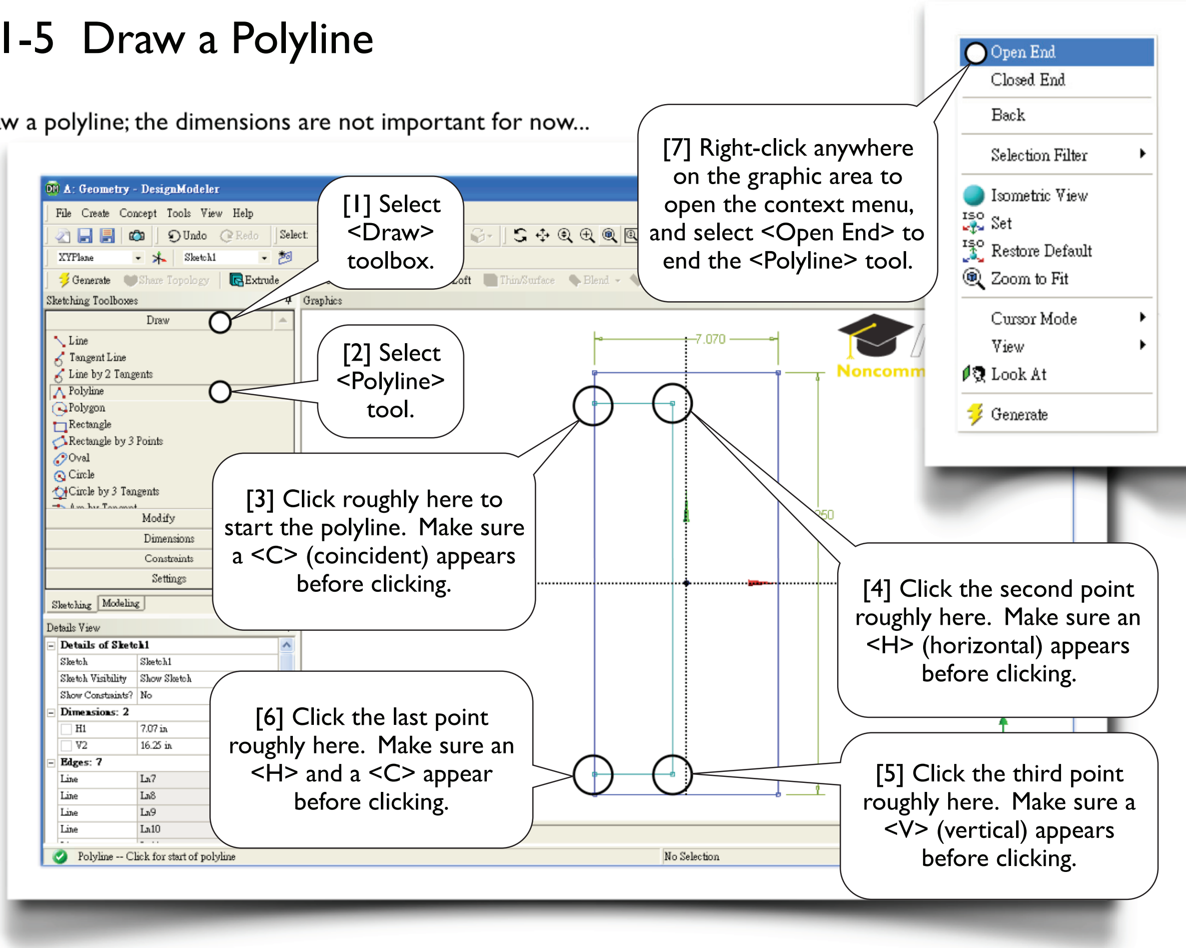


Let's display dimension values (in stead of names) on the graphic area...



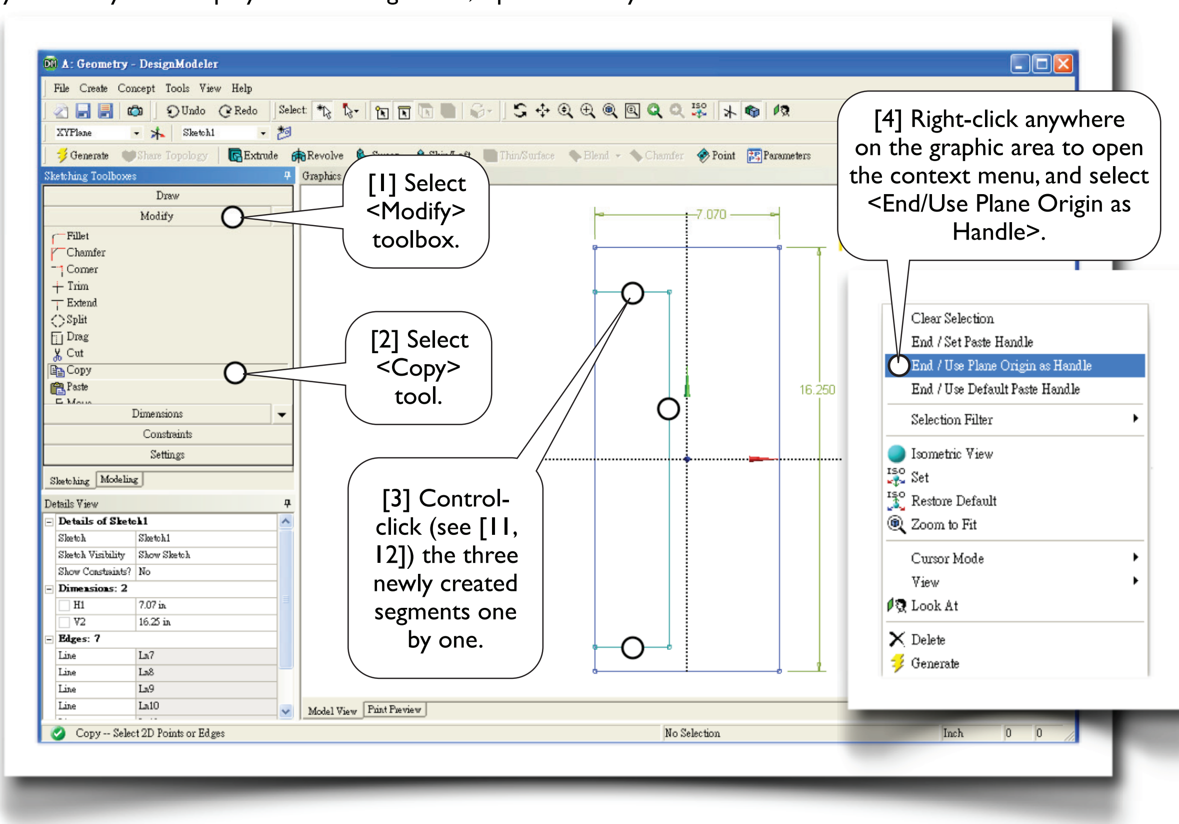
2.1-5 Draw a Polyline

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

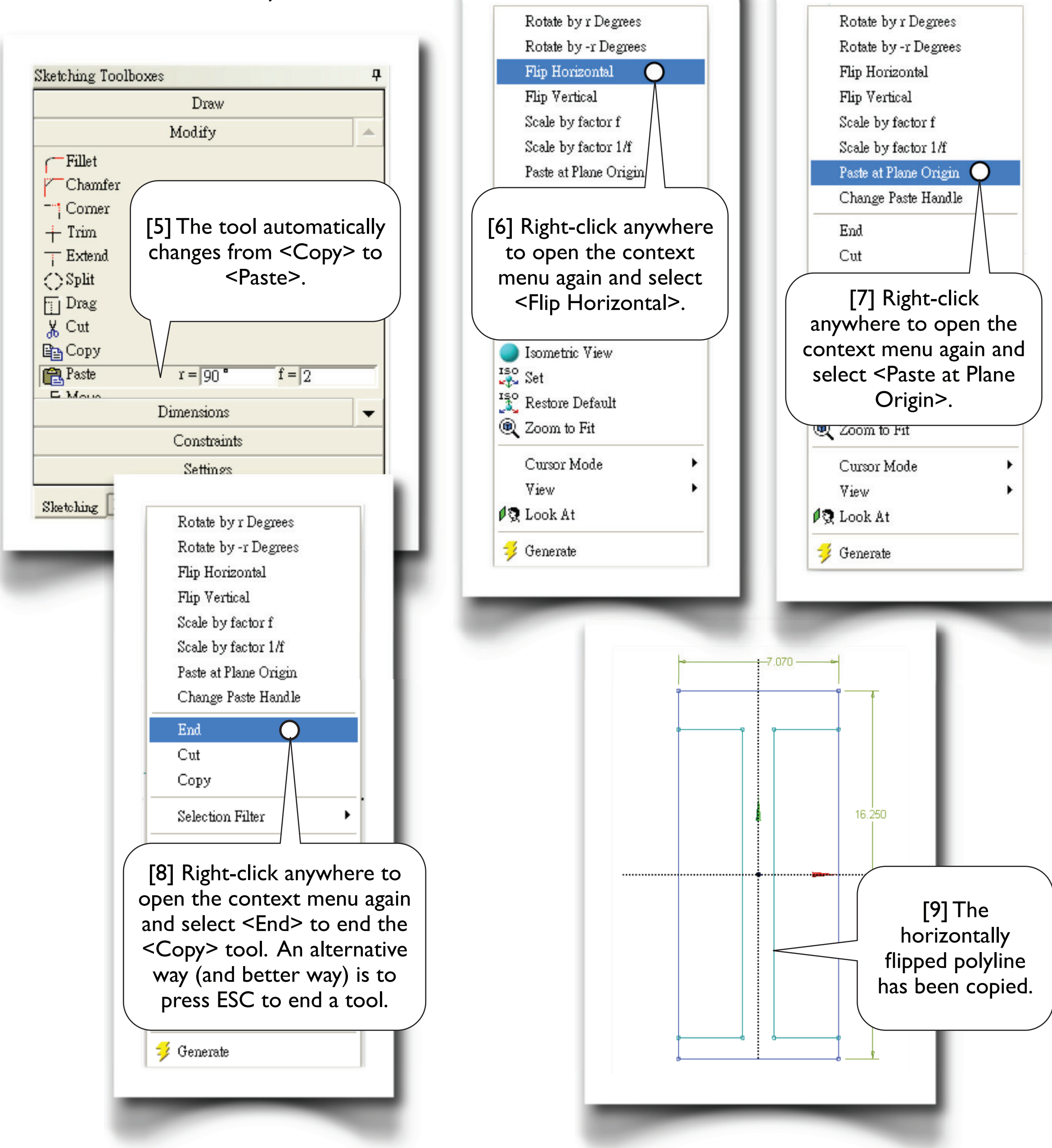


2.1-6 Copy the Polyline

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

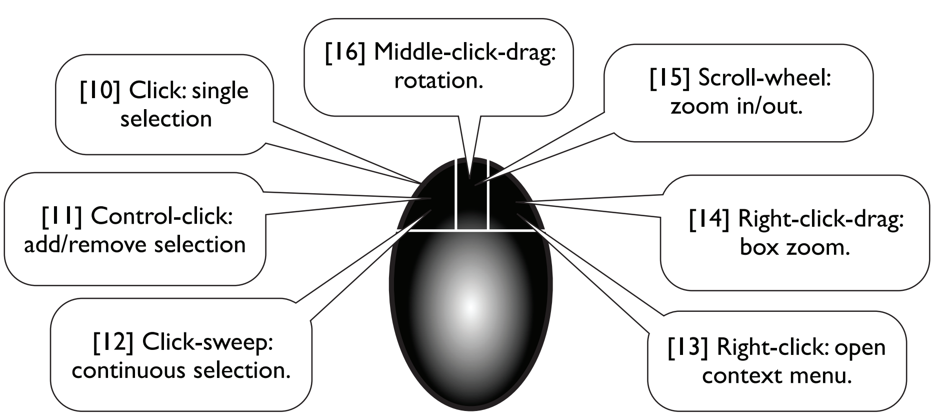


Context menu is used heavily...

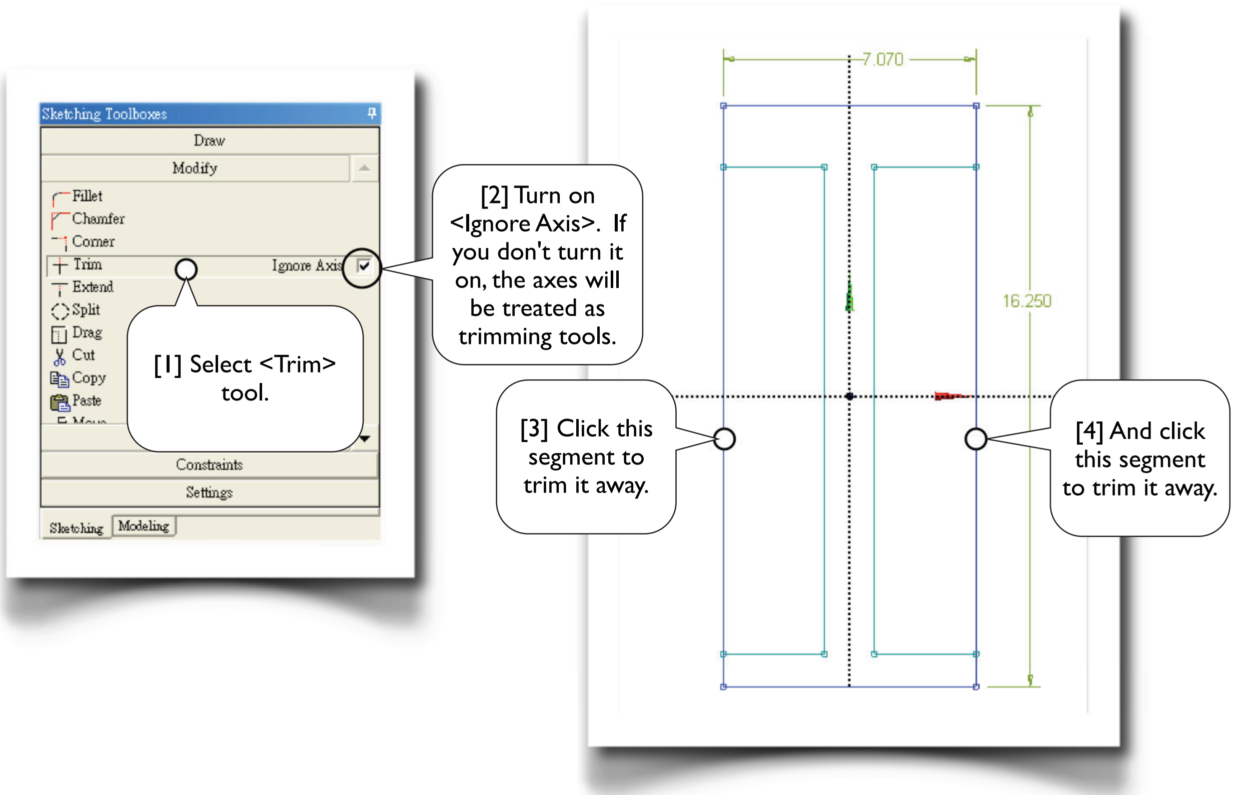


Basic Mouse Operations

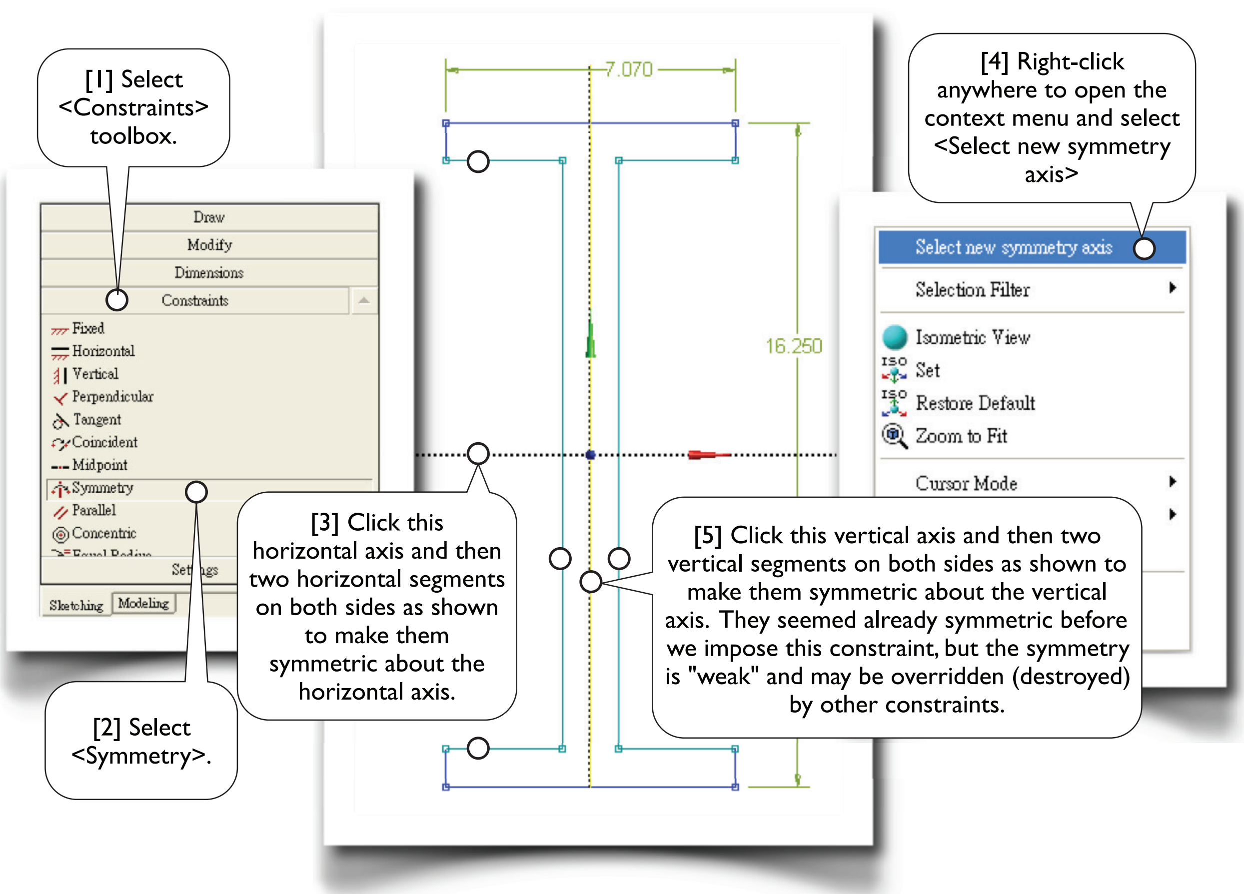
At this point, let's look into some basic mouse operations [10-16]. Skill of these operations is one of the keys to be proficient at geometric modeling.



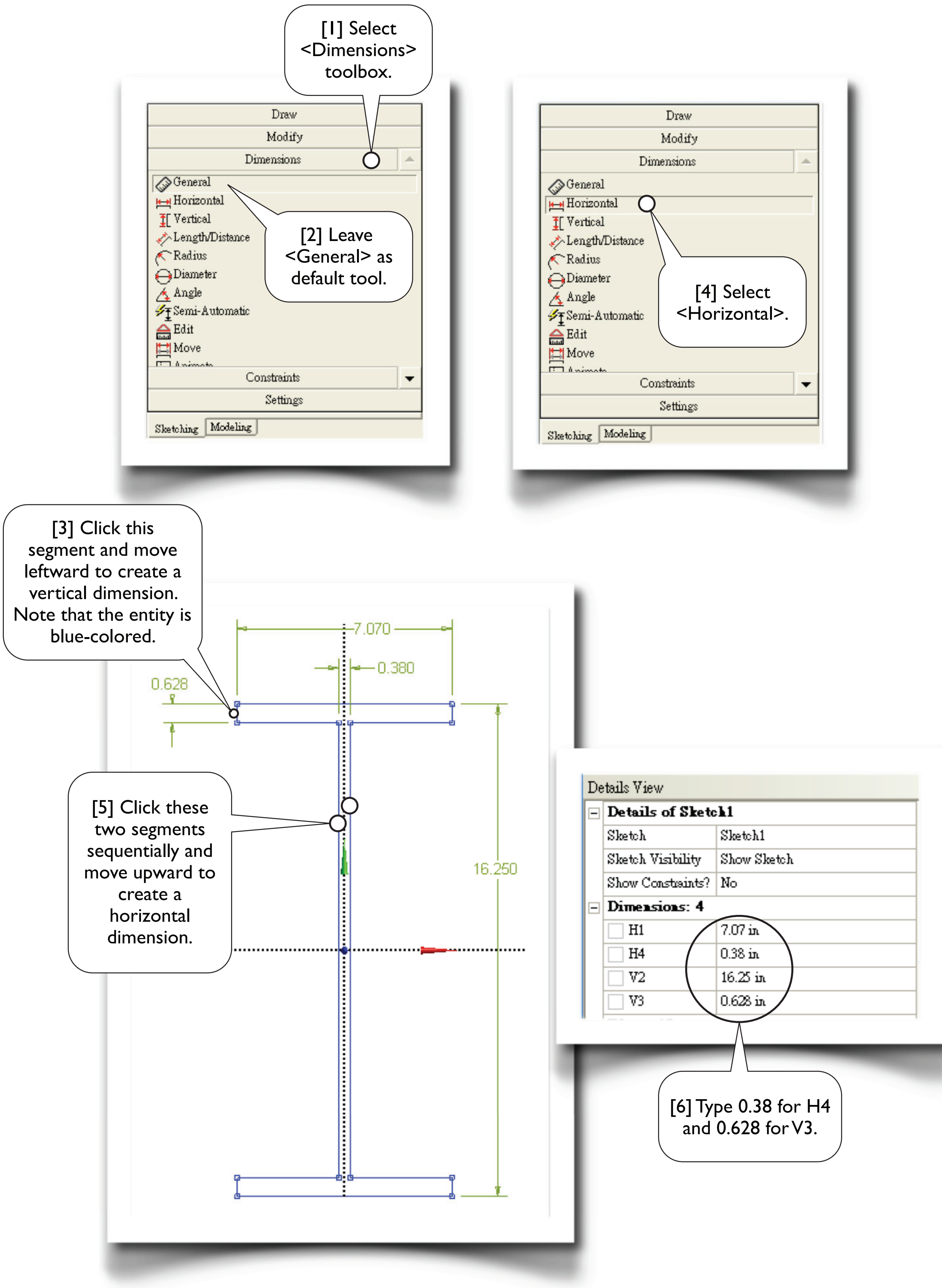
2.1-7 Trim Away Unwanted Segments



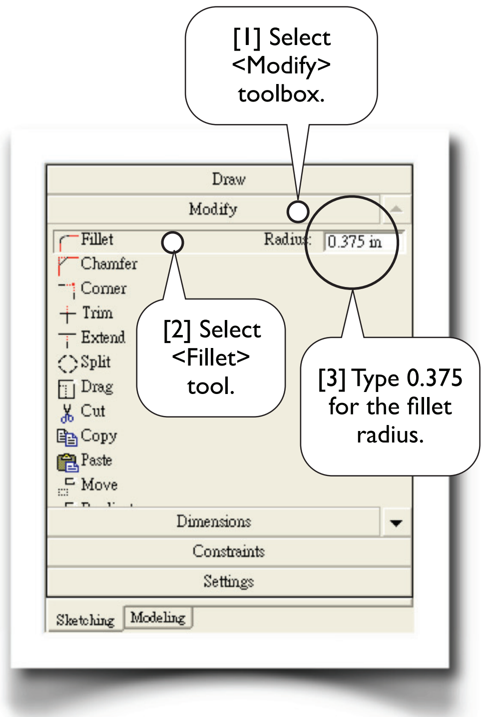
2.1-8 Impose Symmetry Constraints

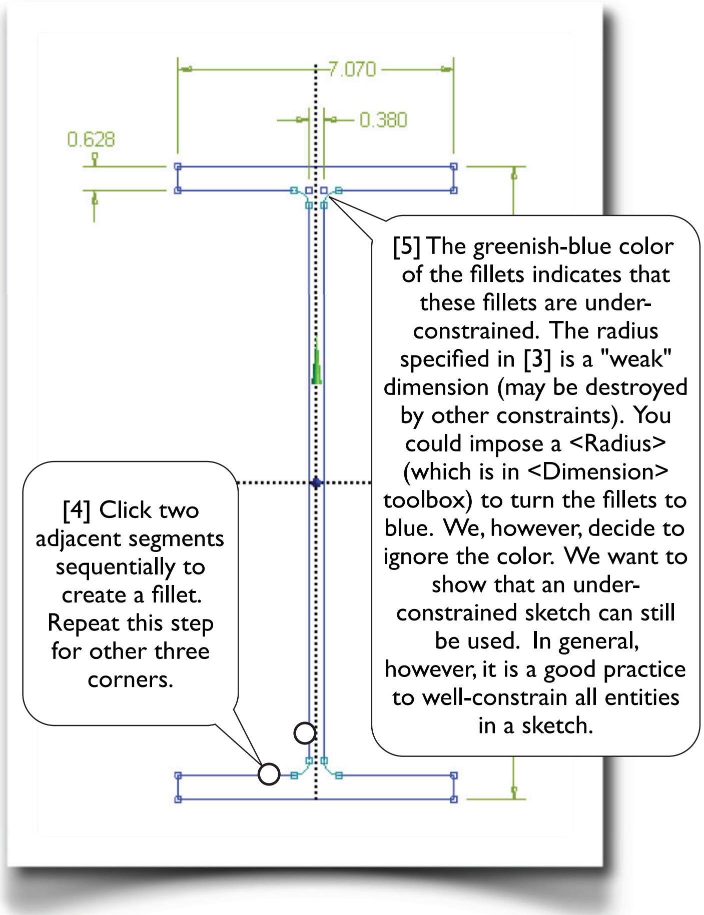


2.1-9 Specify Dimensions

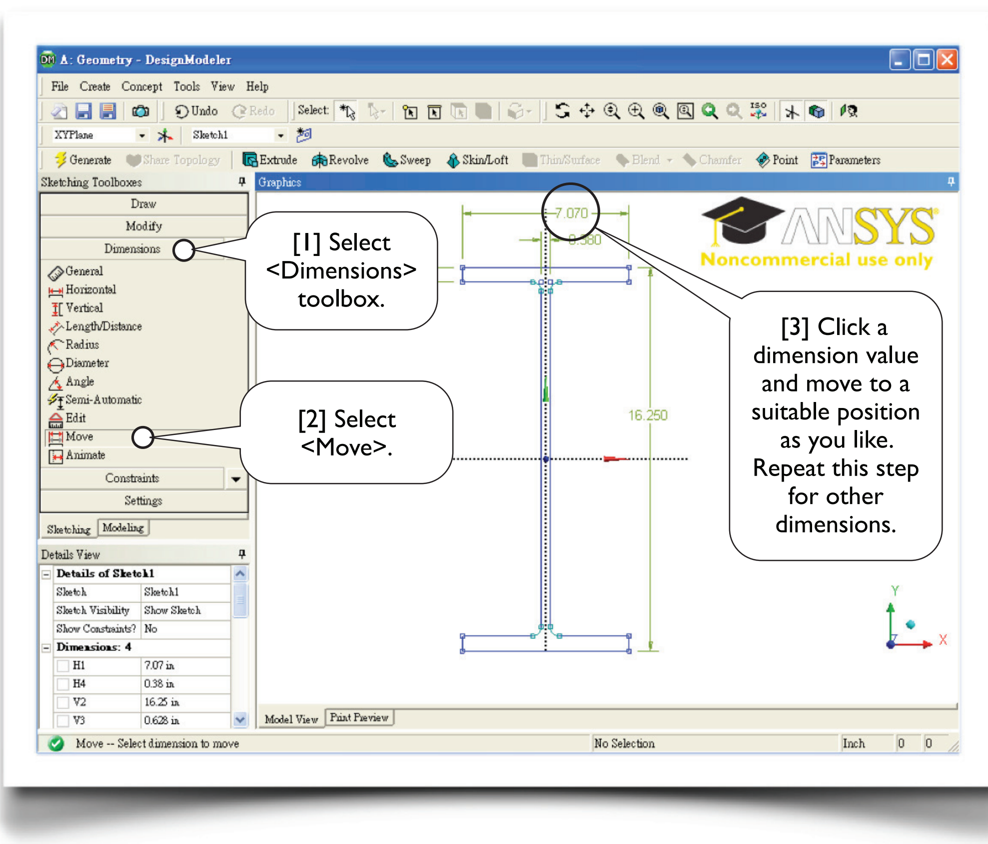


2.1-10 Add Fillets

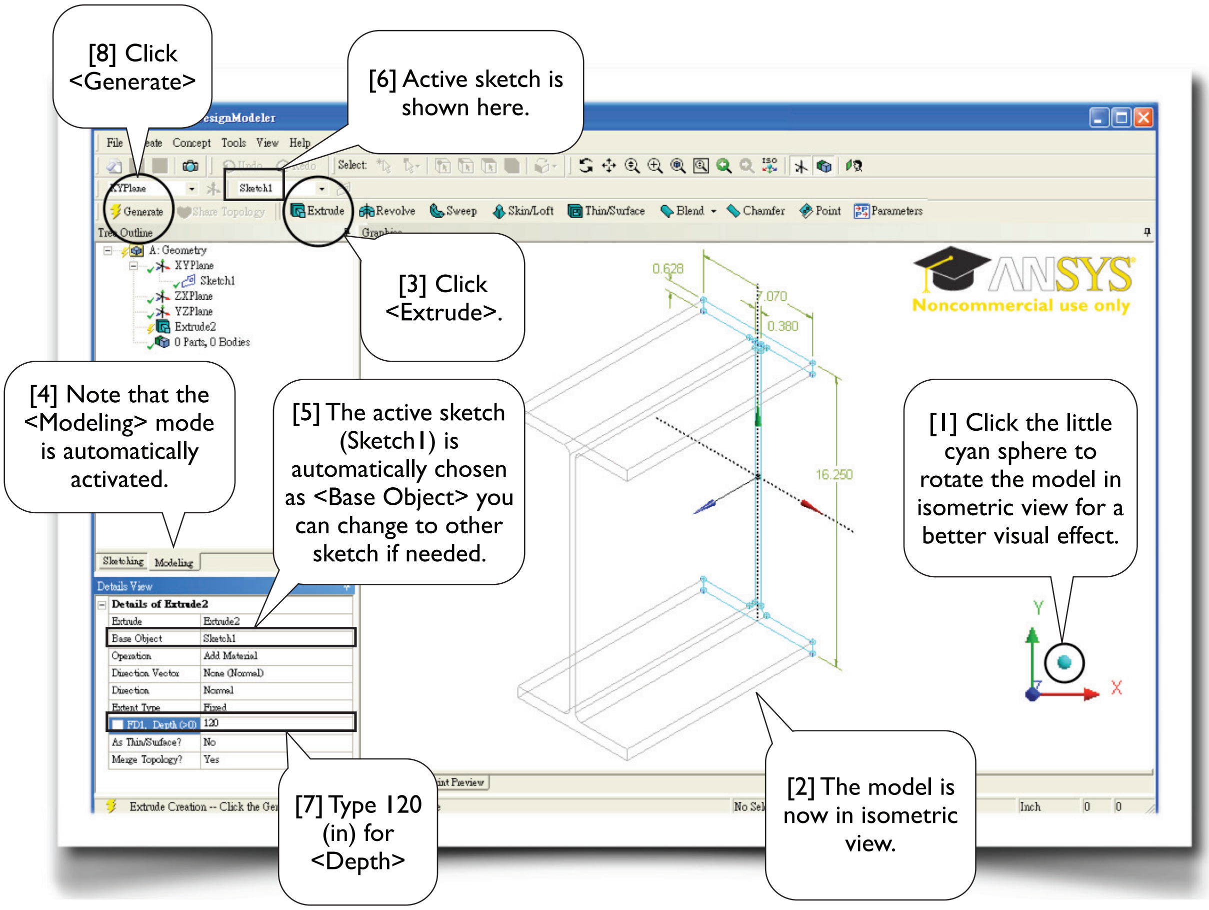


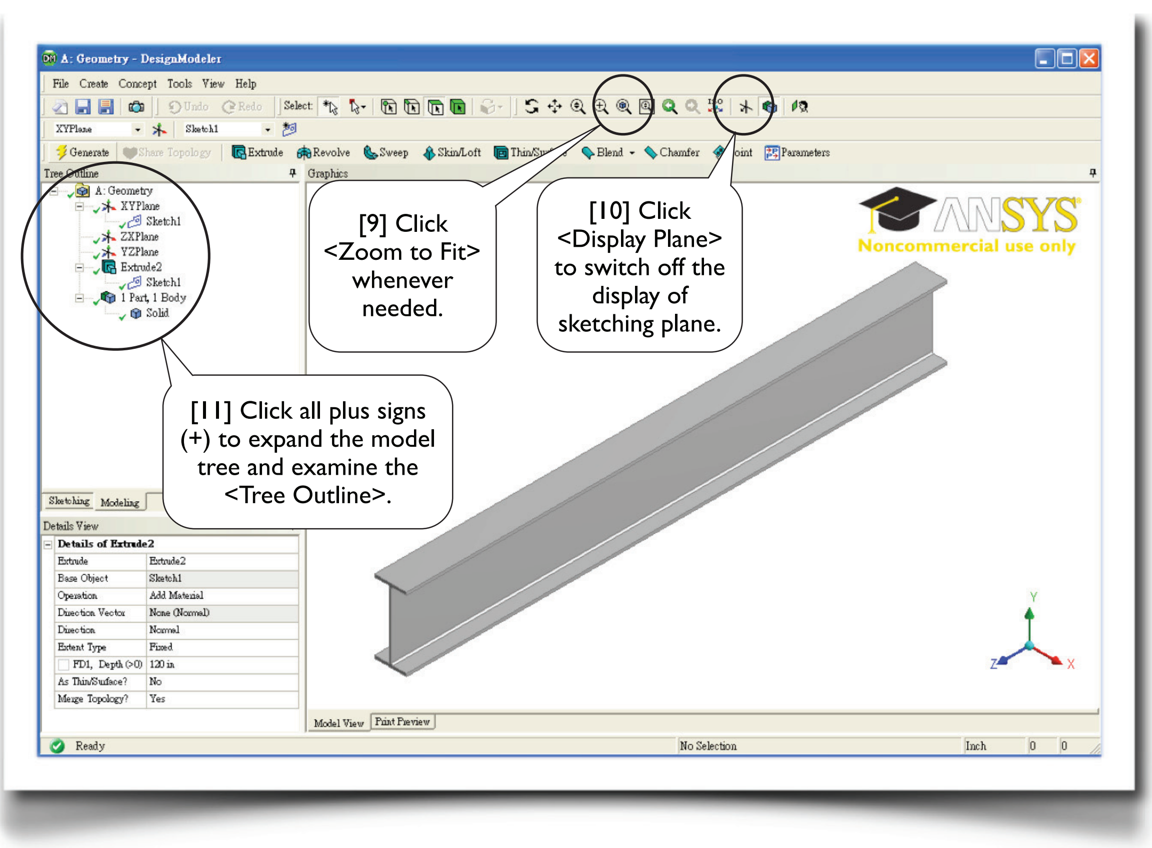


2.1-11 Move Dimensions

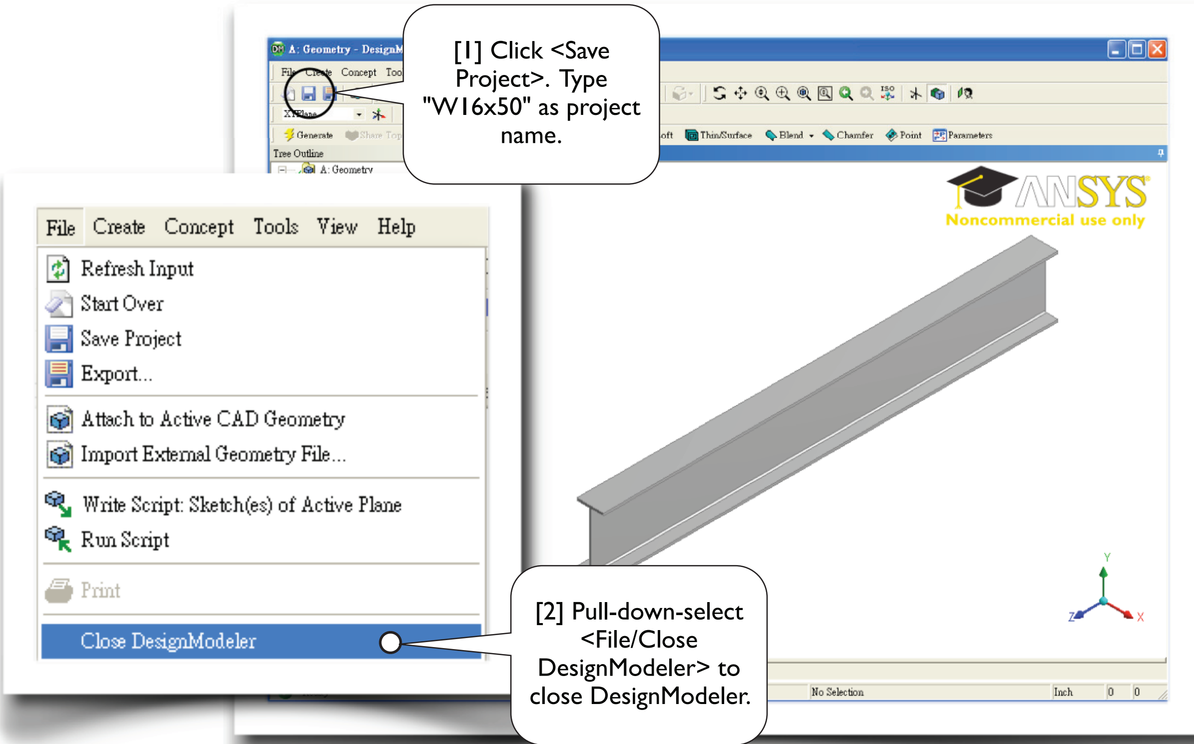


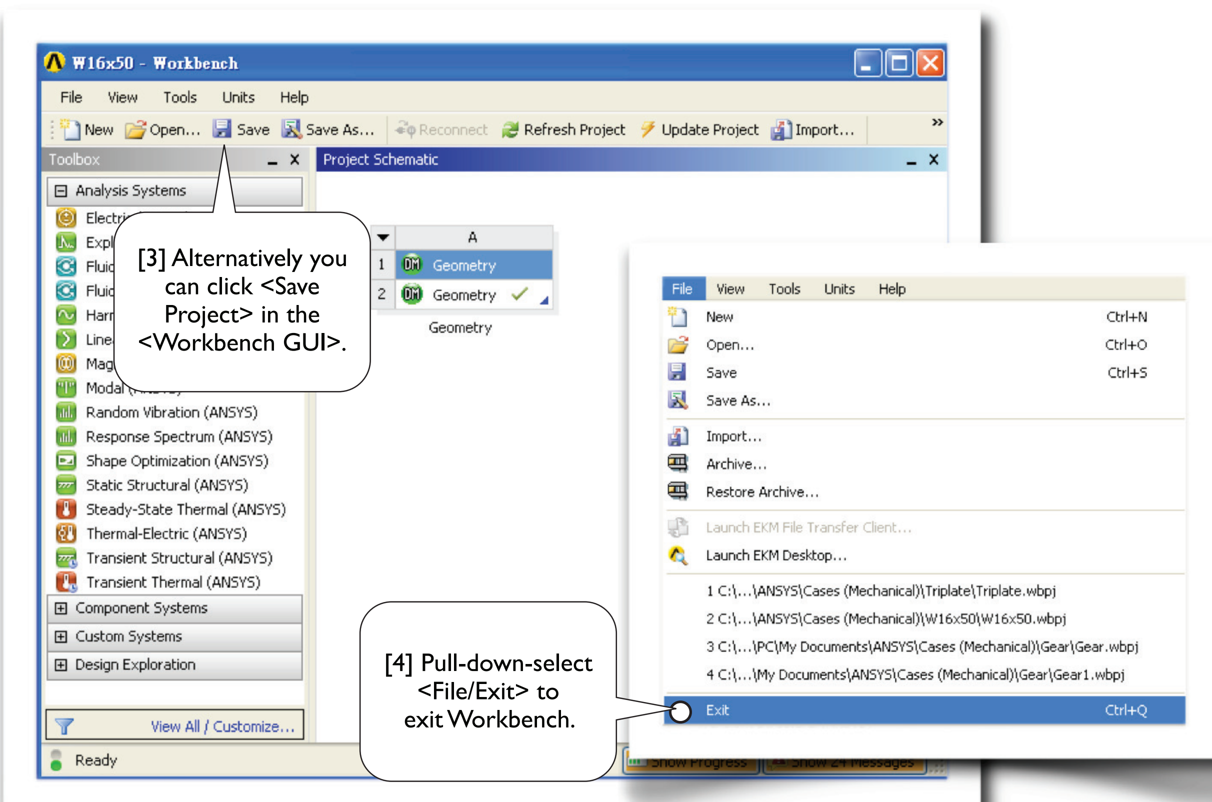
2.1-12 Extrude to Generate 3D Solid





2.1-13 Save the Project and Exit Workbench





Section 2.2

Step-by-Step: Triangular Plate

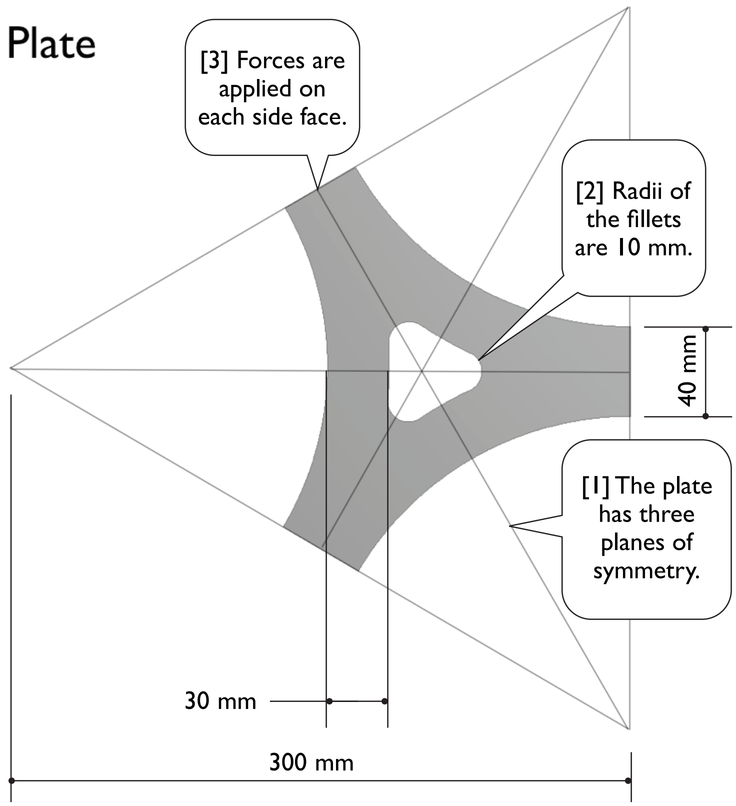
2.2-1 About the Triangular Plate

The triangular plate [1, 2] is made to withstand a tensile stress of 50 MPa on each side face [3]. The thickness of the plate is 10 mm. Other dimensions are shown in the figure.

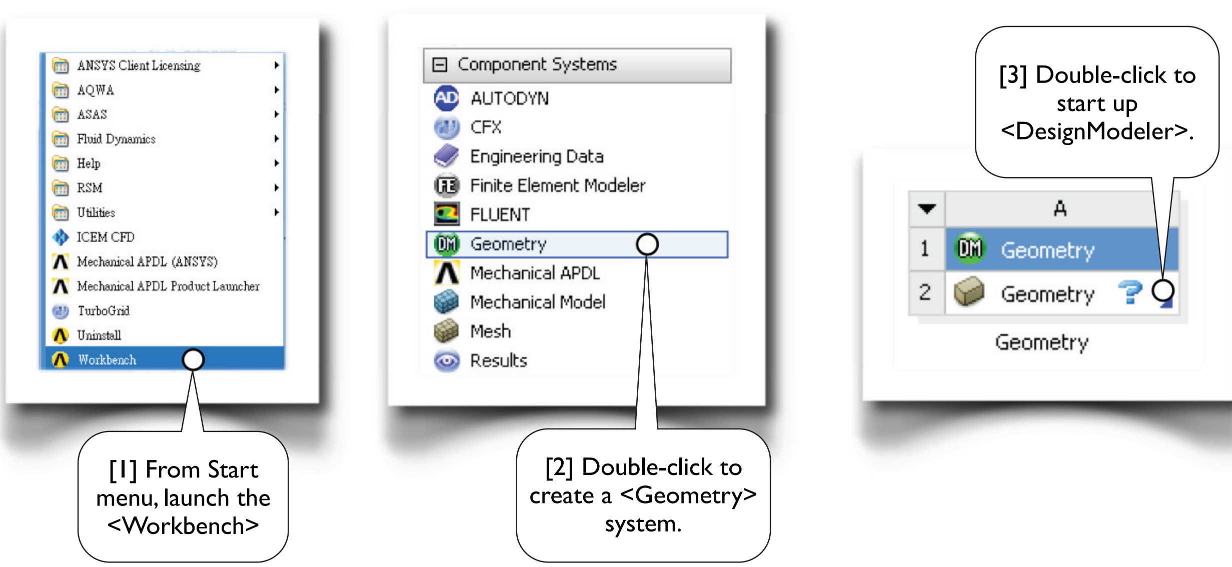
In this section, we want to sketch the plate on <XYPlane> and then extrude a thickness of 10 mm along Z-axis to generate a 3D solid body.

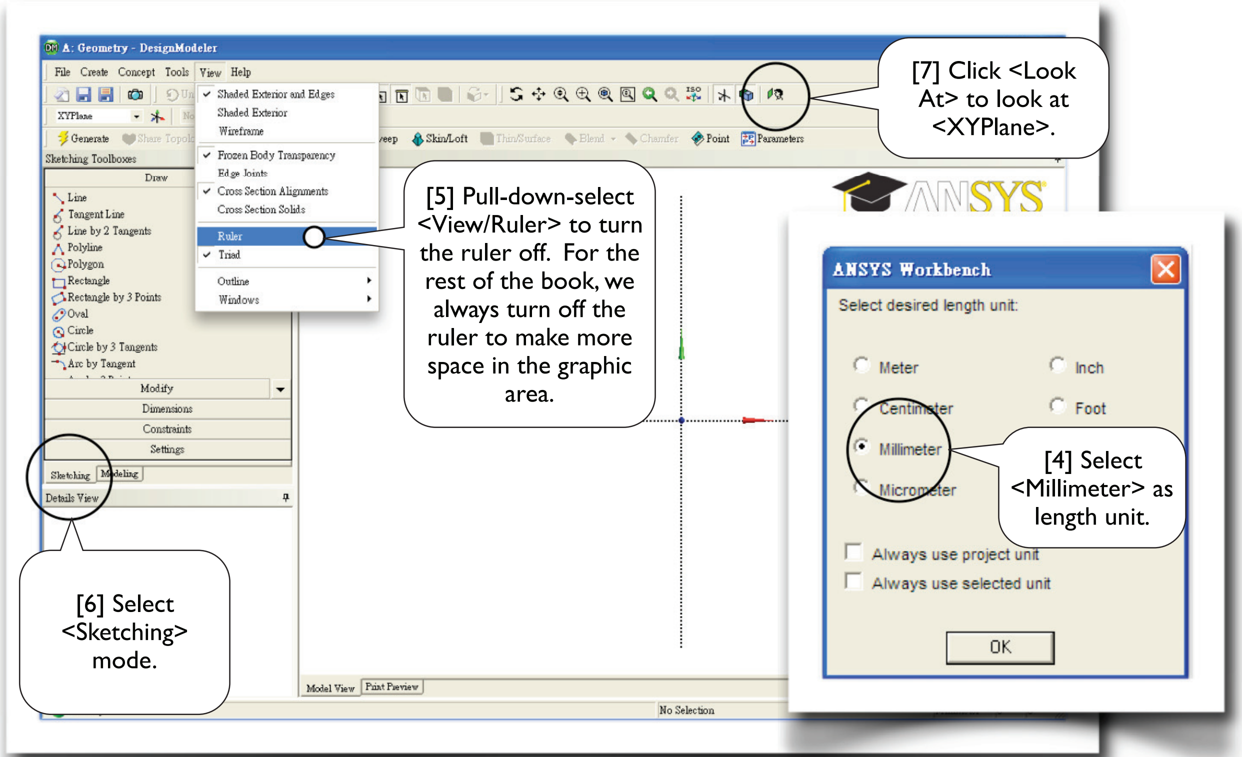
In Section 3.1, we will use this sketch again to generate a 2D solid model, and the 2D model is then used for a static structural simulation to assess the stress under the loads.

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

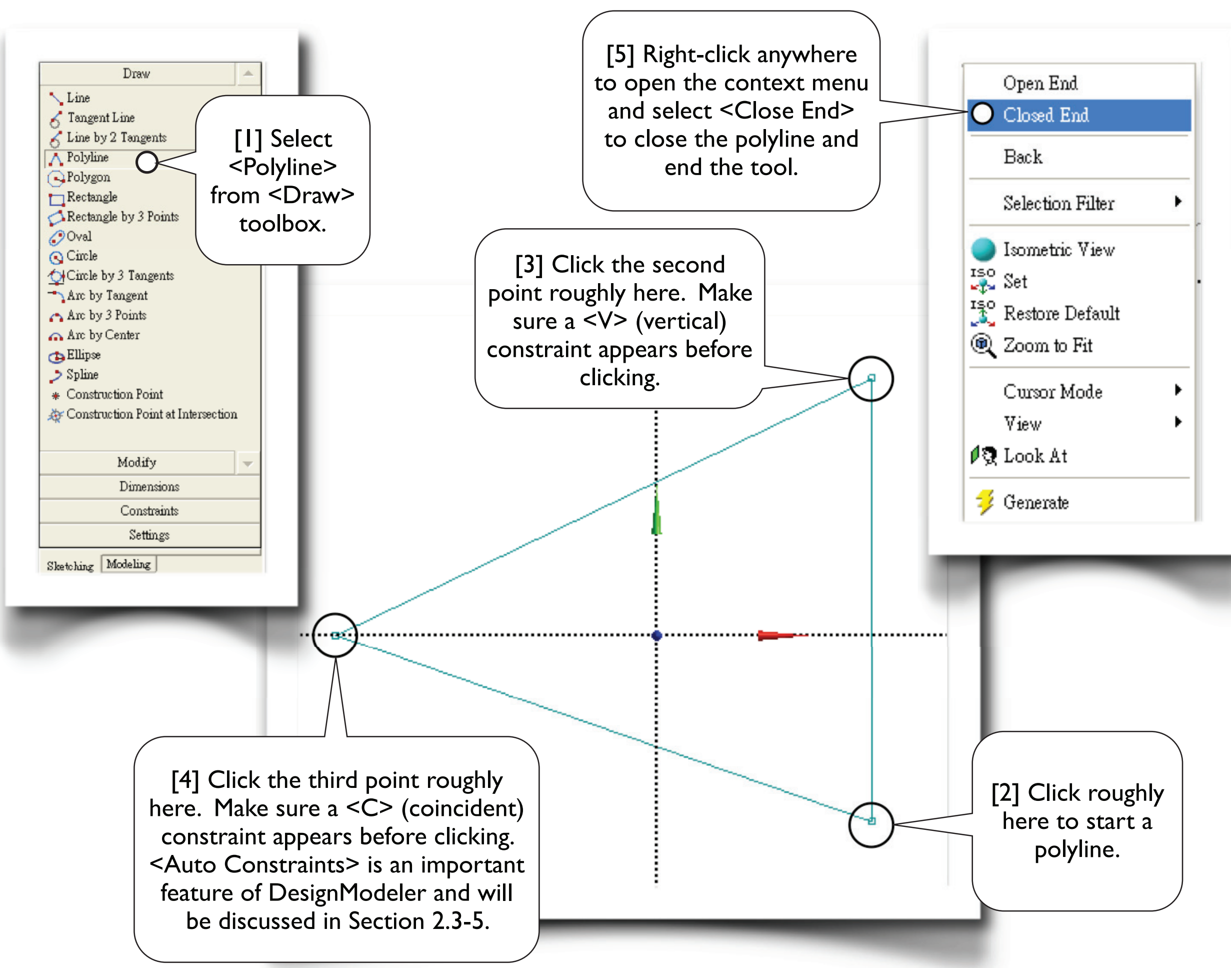


2.2-2 Start up <DesignModeler>

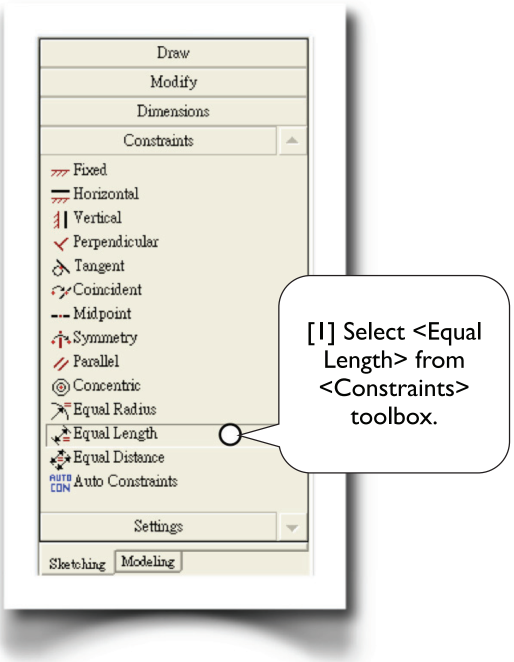


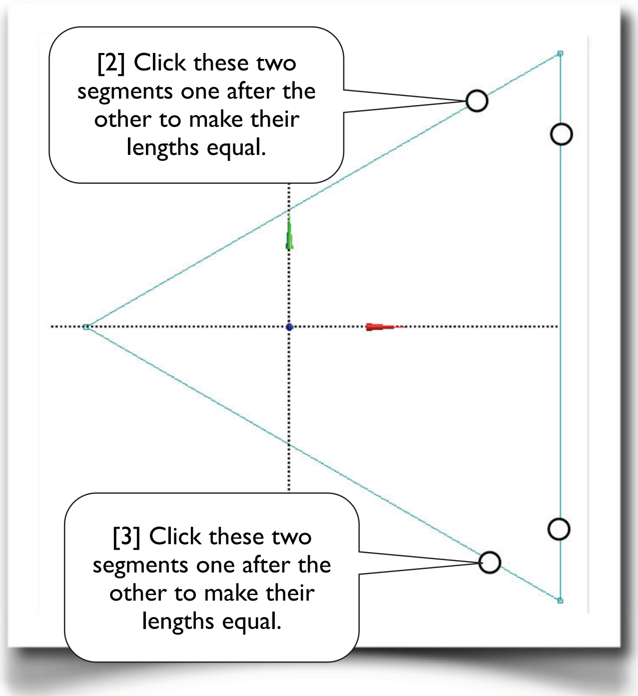


2.2-3 Draw a Triangle on <XYPlane>



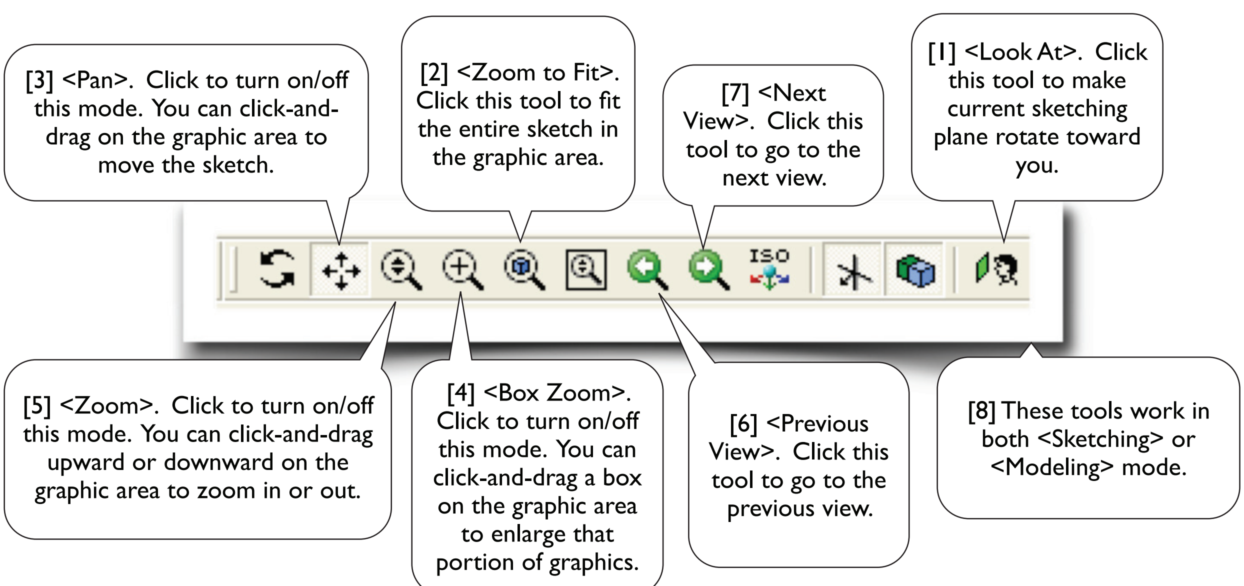
2.2-4 Make the Triangle Regular



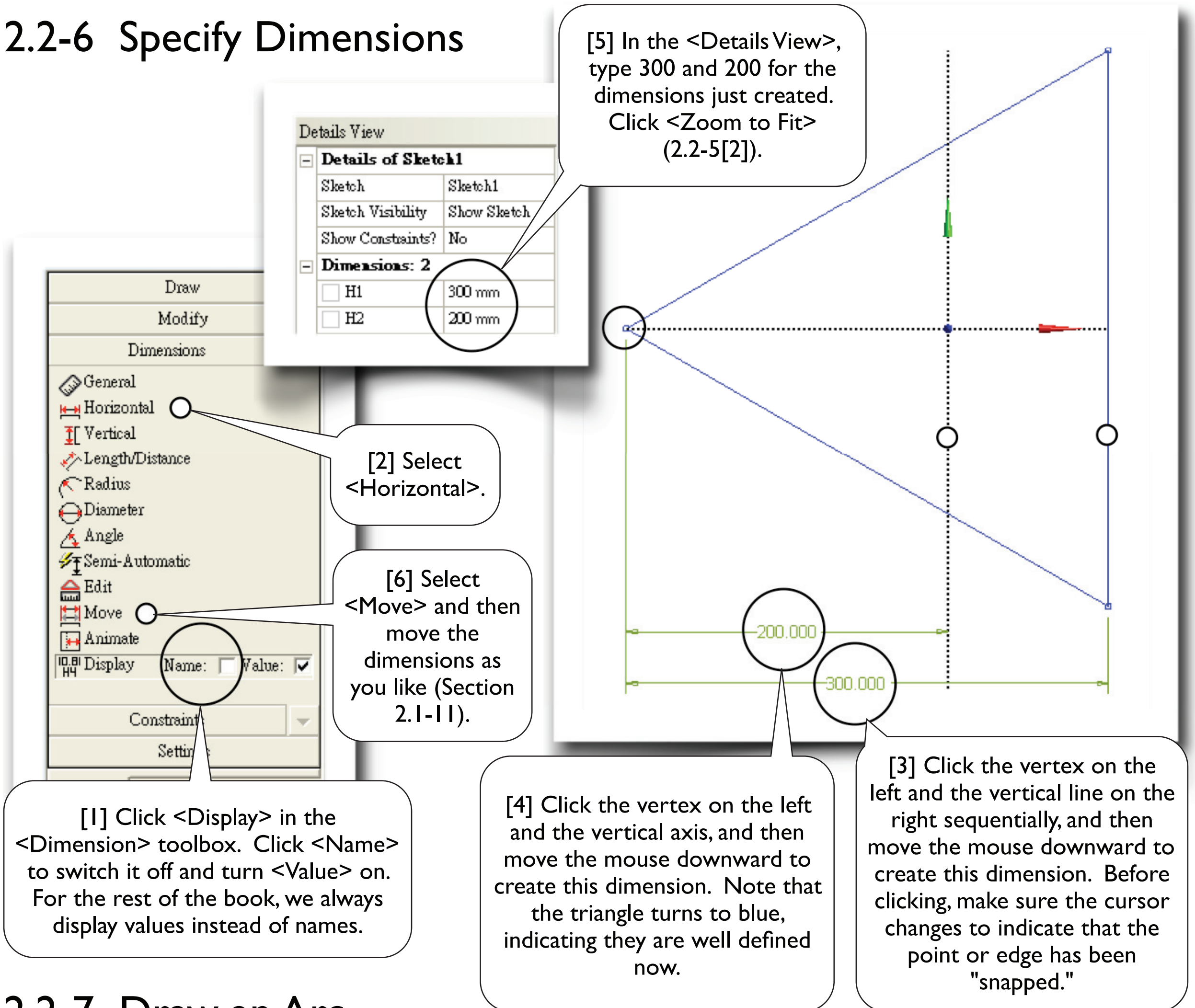


2.2-5 2D Graphics Controls

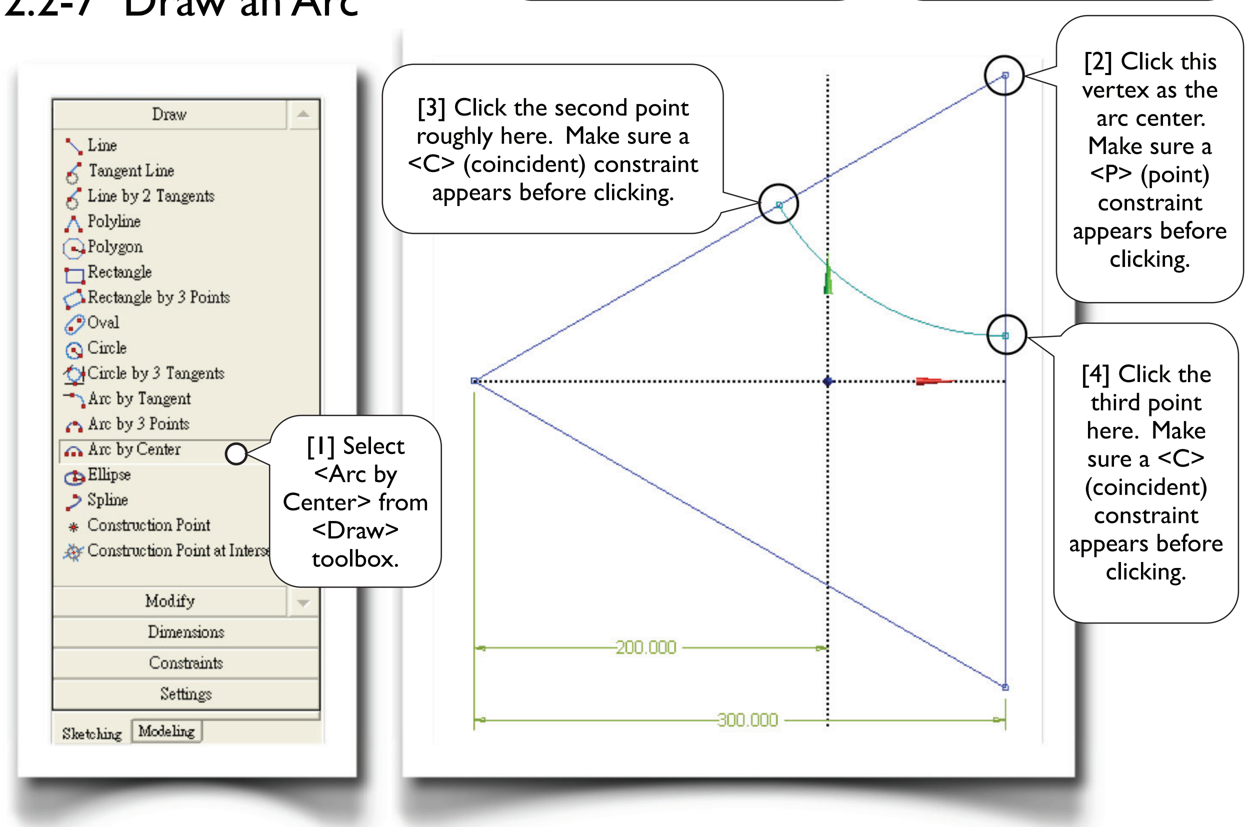
Before we proceed, let's spend a few minutes looking into some useful tools for 2D graphics controls [1-10]; feel free to use these tools whenever needed. The tools are numbered according to roughly their frequency of use. Note that more useful mouse short-cuts for <Pan>, <Zoom>, and <Box Zoom> are available; please see Section 2.3-4.



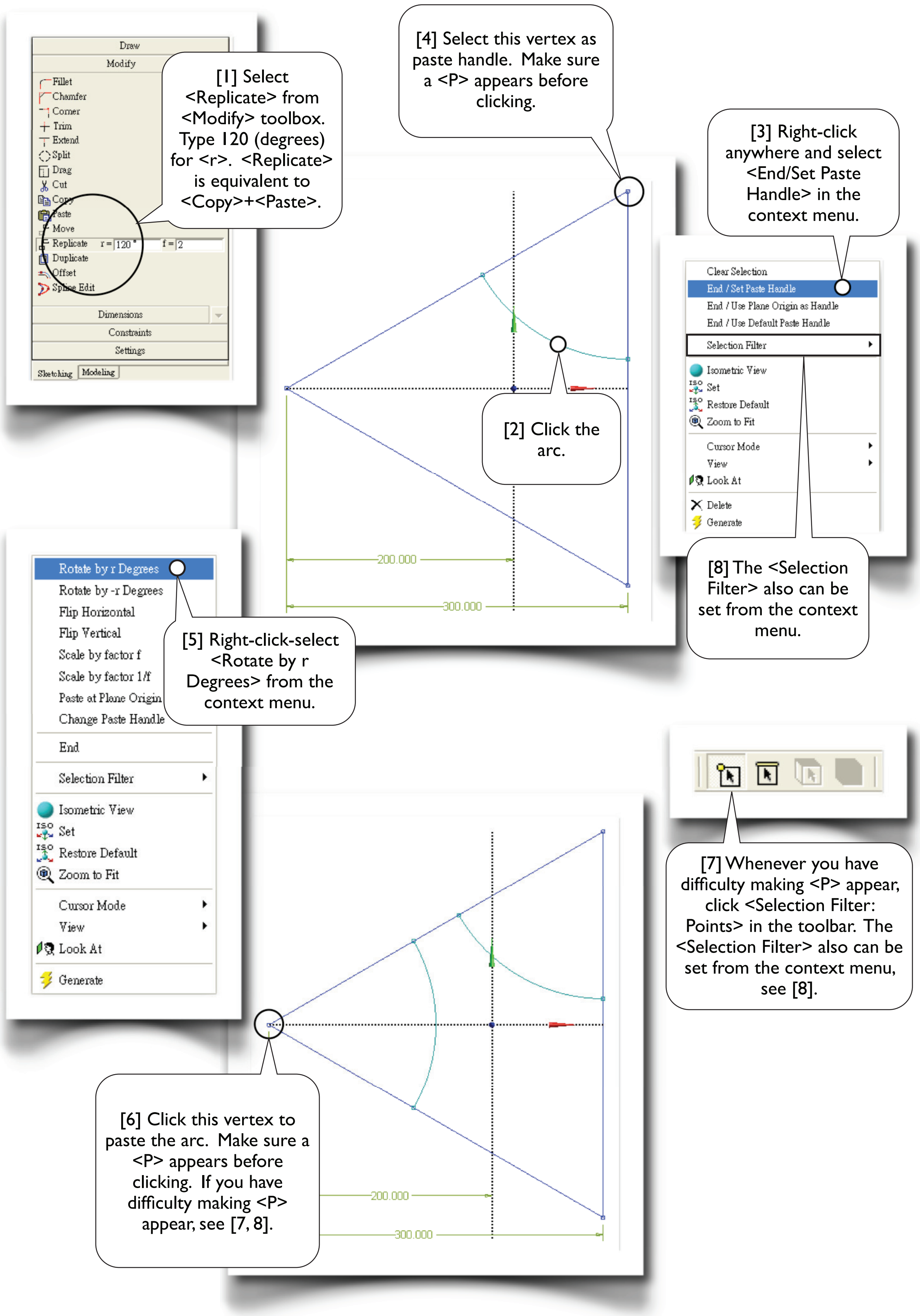
2.2-6 Specify Dimensions

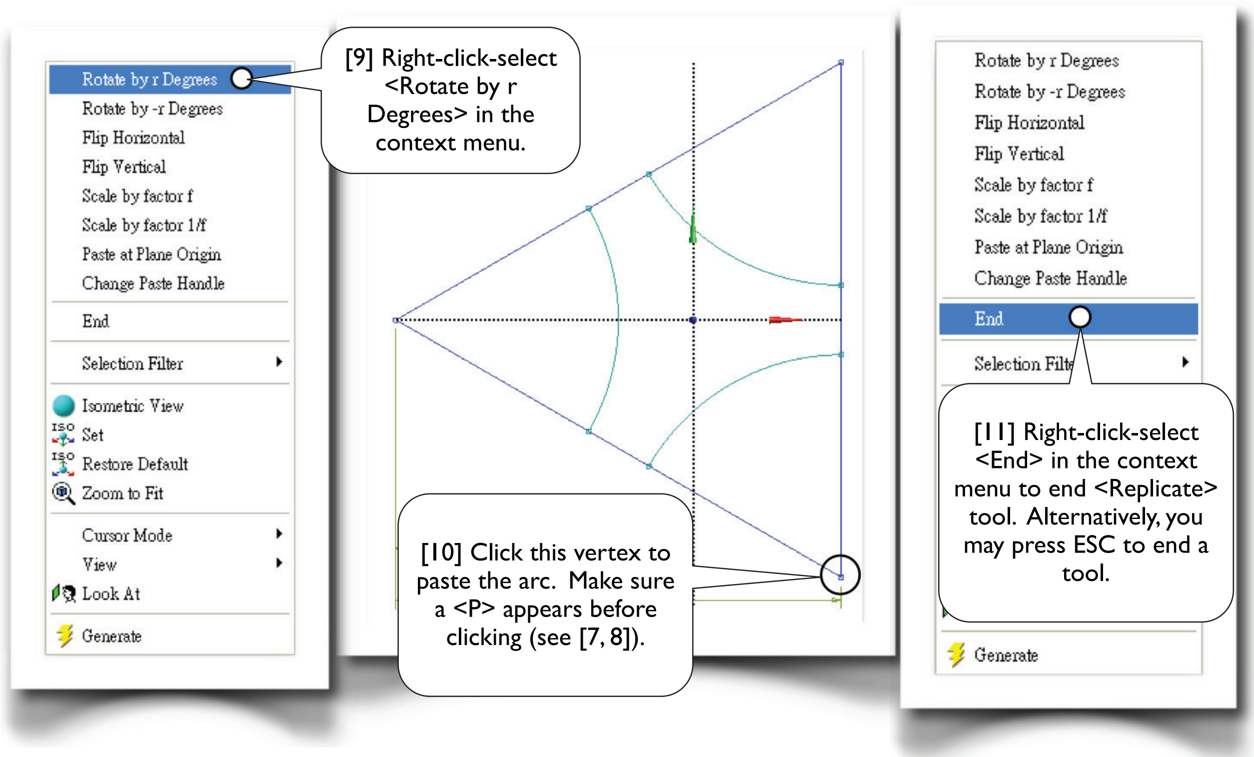


2.2-7 Draw an Arc



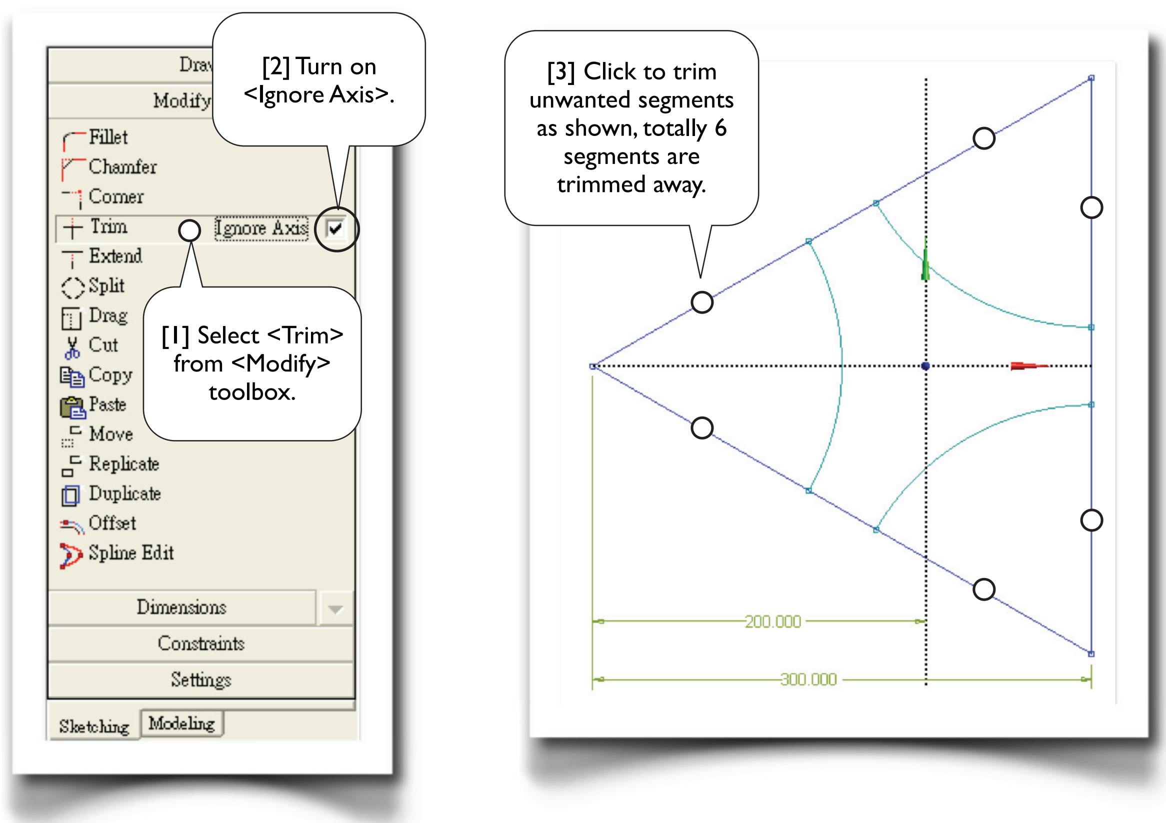
2.2-8 Replicate the Arc



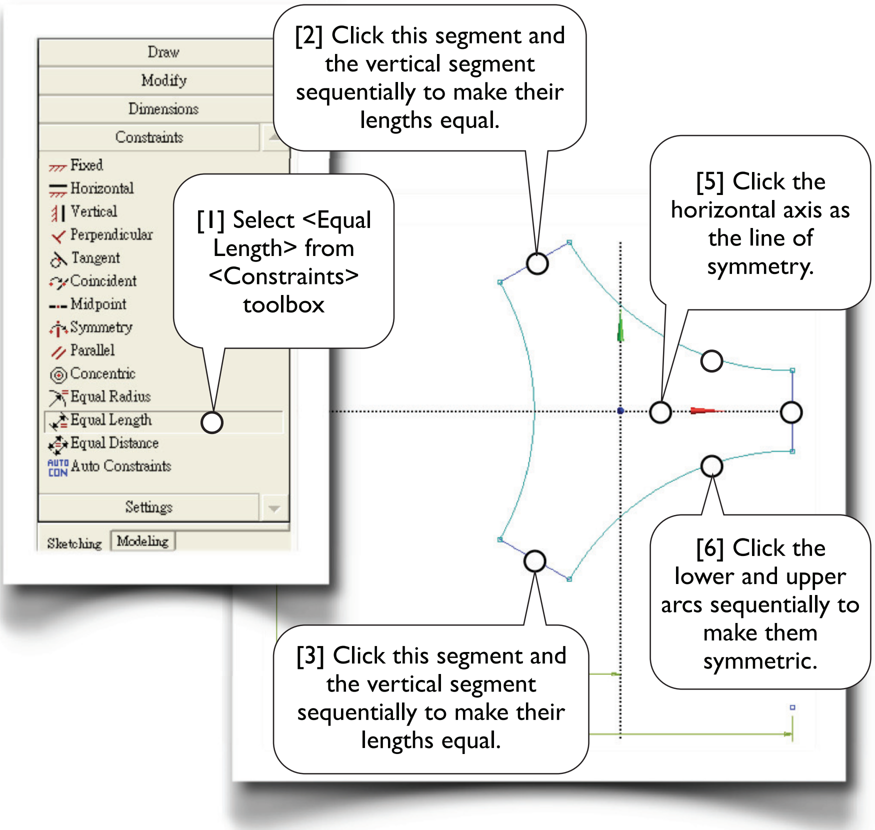


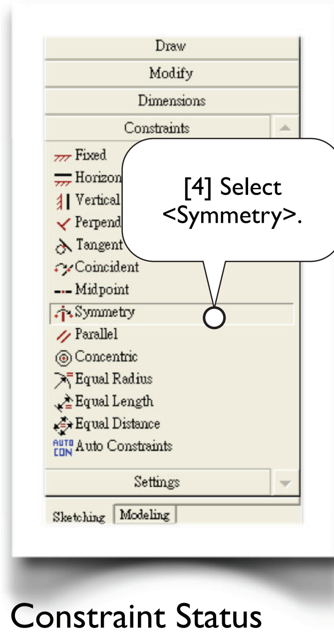
For instructional purpose, we chose to manually set the paste handle [3] on the vertex [4]. We could have used plane origin as handle. In fact, that would have been easier since we wouldn't have to struggle to make sure whether a <P> appears or not. Whenever you have difficulty to "snap" a particular point, you should take advantage of <Selection Filter> [7, 8].

2.2-9 Trim Away Unwanted Segments



2.2-10 Impose Constraints

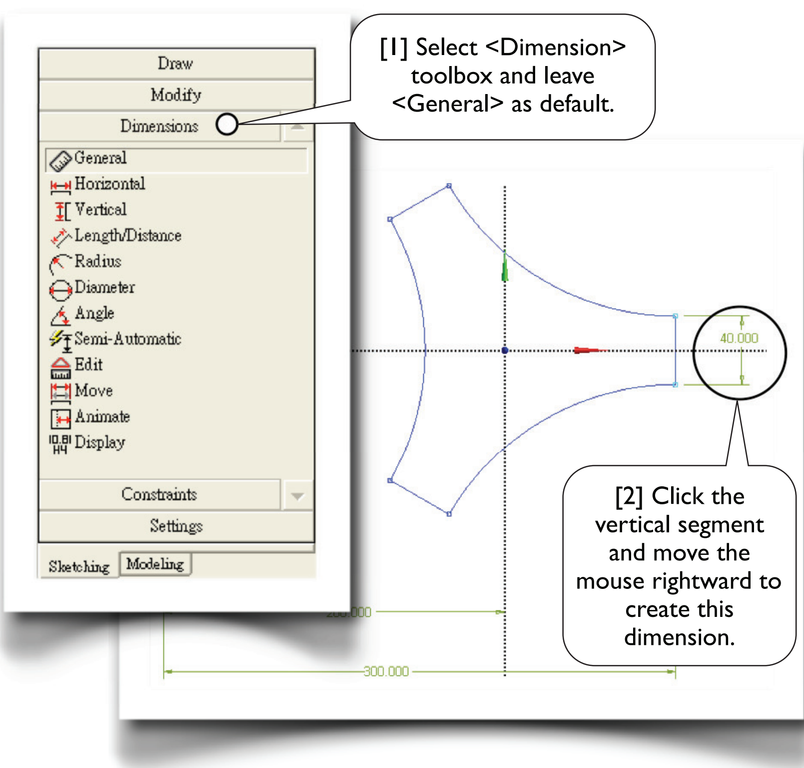


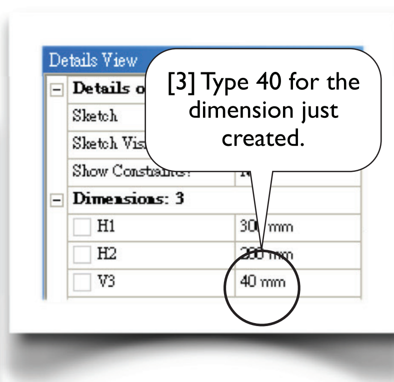


Constraint Status

Note 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.

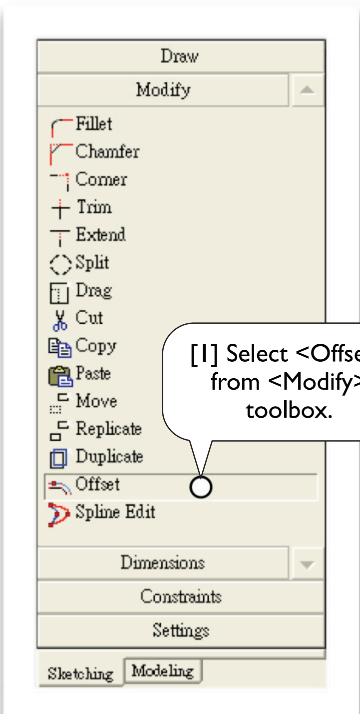
2.2-11 Specify Dimension of Side Faces



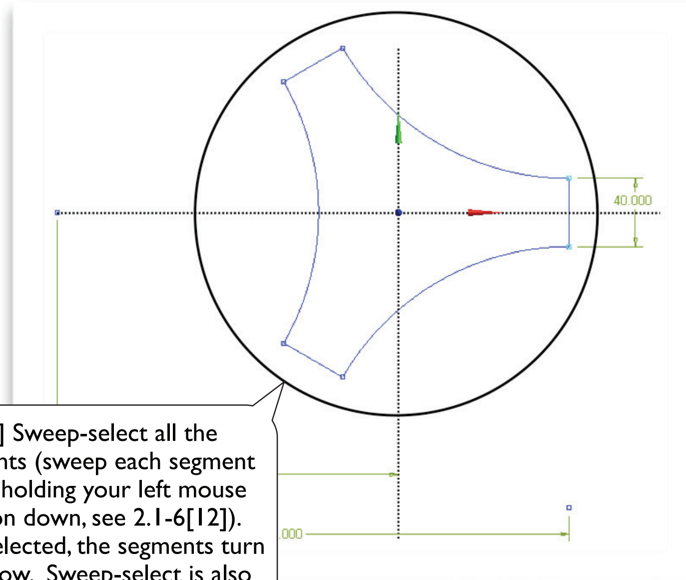


After impose dimension in [2], the arcs turns to blue, indicating they are well defined now. Note that we didn't specify the radii of the arcs; after well defined, the radii of the arcs can be calculated from other dimensions.

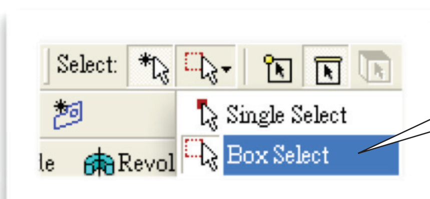
2.2-12 Create Offset



[1] Select <Offset> from <Modify> toolbox.

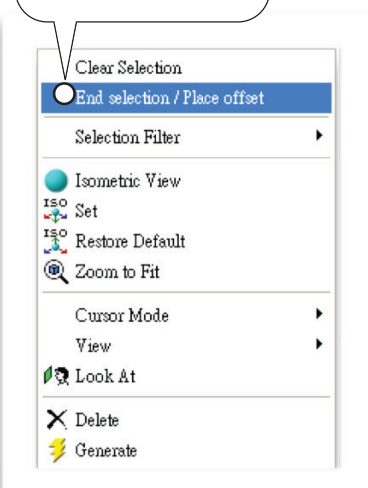


[2] Sweep-select all the segments (sweep each segment while holding your left mouse button down, see 2.1-6[12]). After selected, the segments turn to yellow. Sweep-select is also called paint-select.

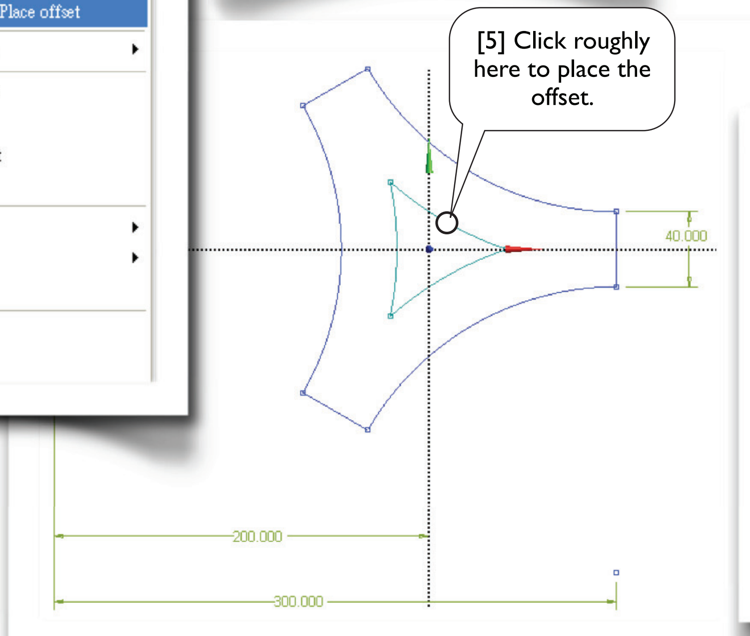


[3] Another way to select multiple entities is to switch the <Select Mode> to <Box Select>, and then draw a box to select all entities inside the box.

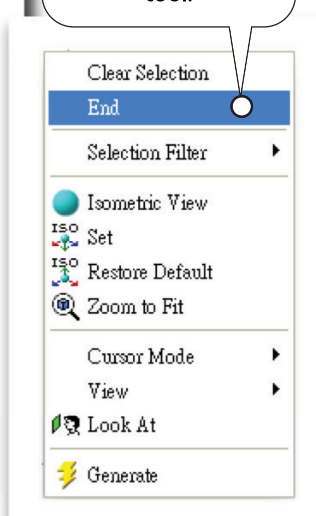
[4] Right-click-select <End selection/Place Offset> in the context menu.

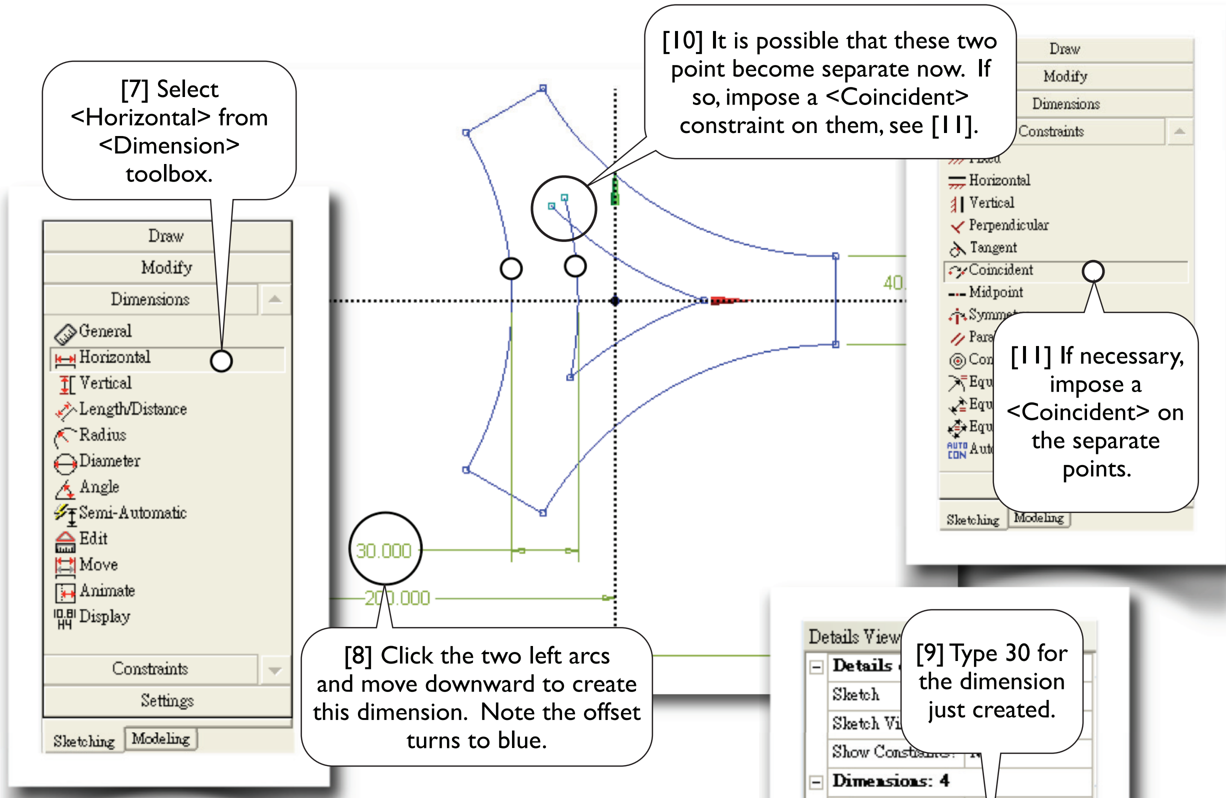


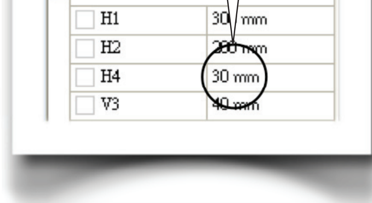
[5] Click roughly here to place the offset.



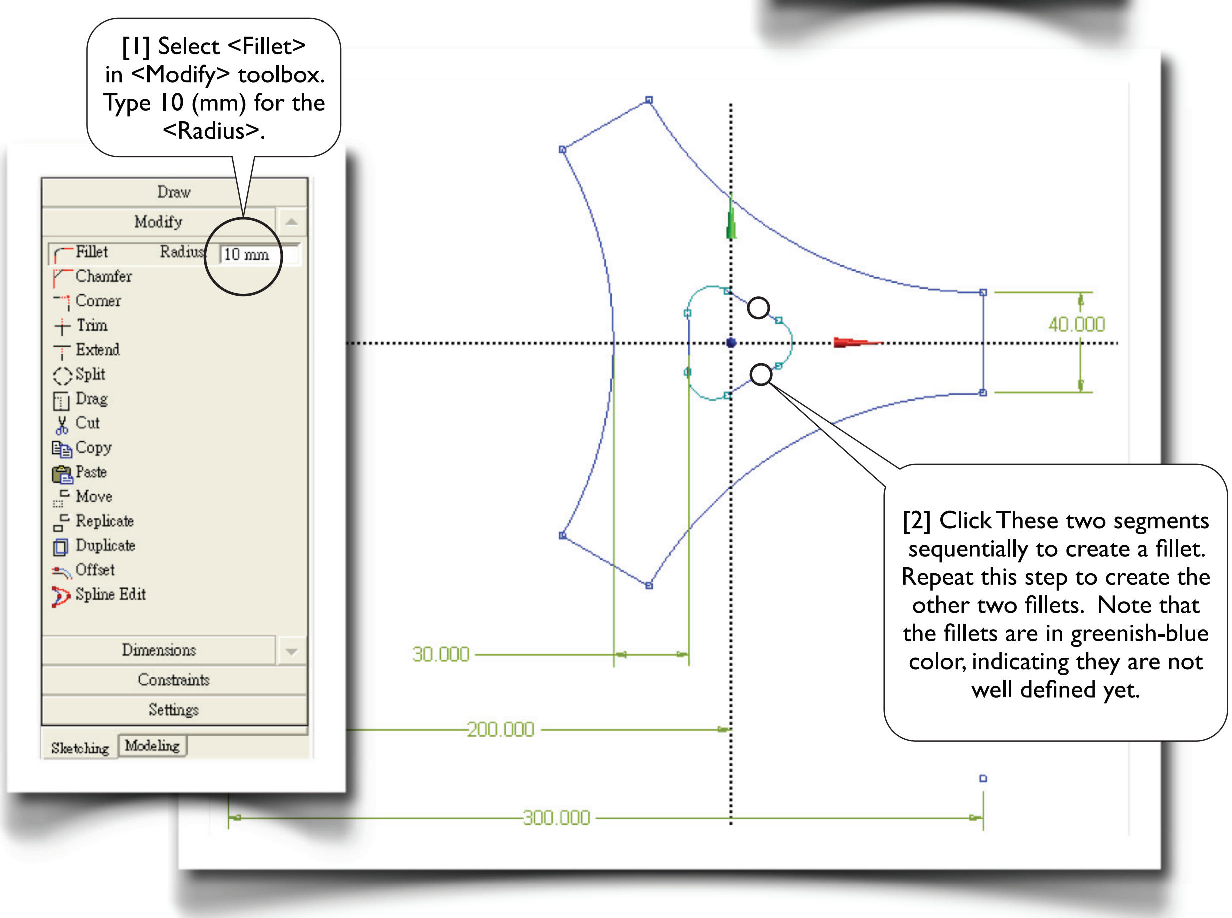
[6] Right-click-select <End> in the context menu, or press ESC, to close <Offset> tool.

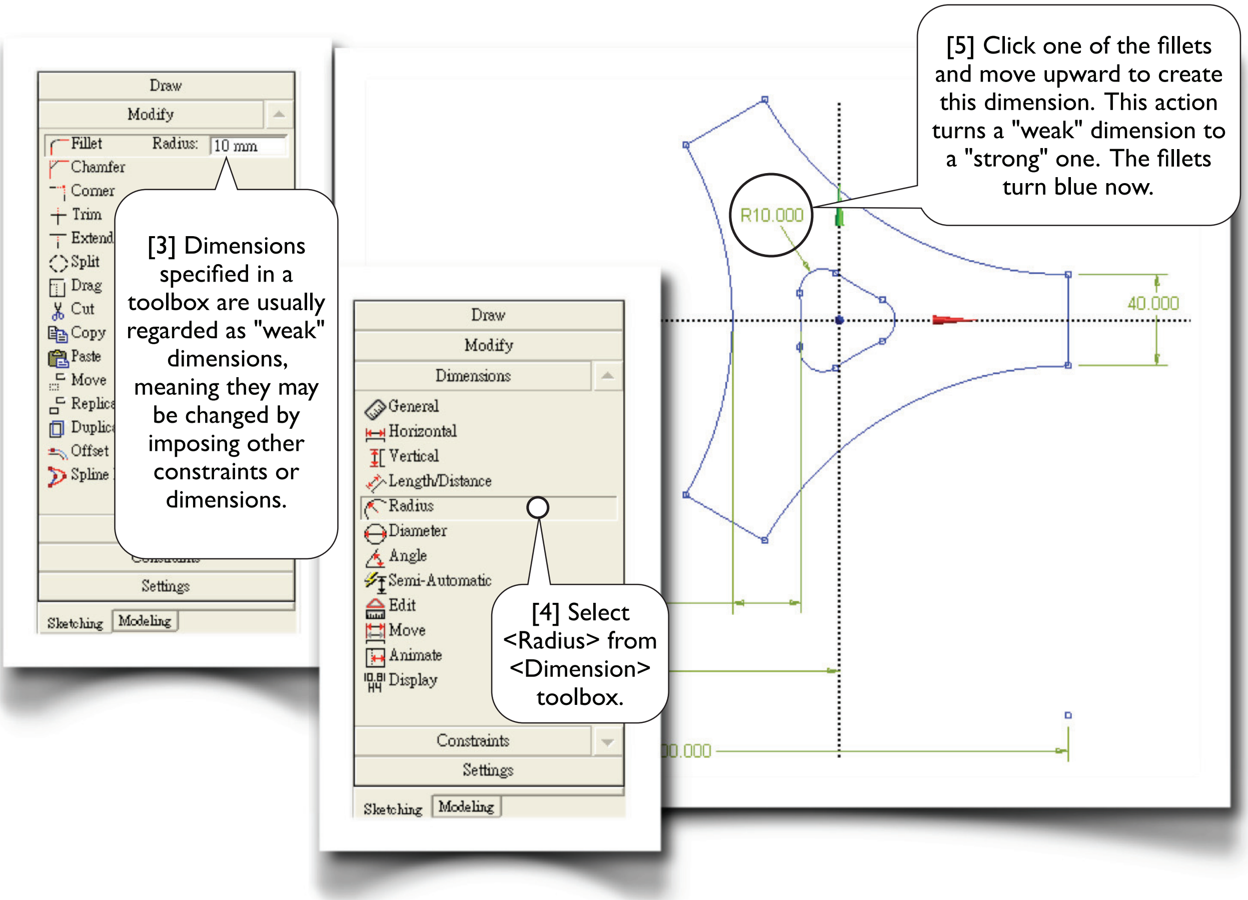




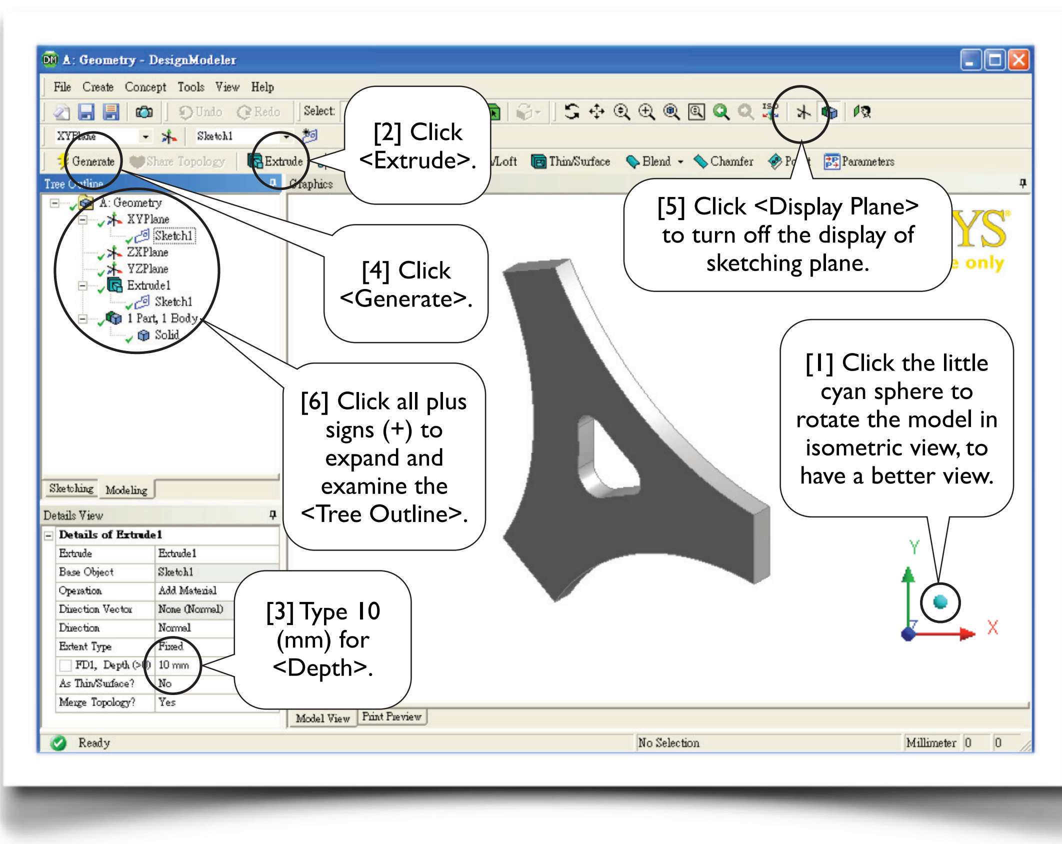


2.2-13 Create Fillets

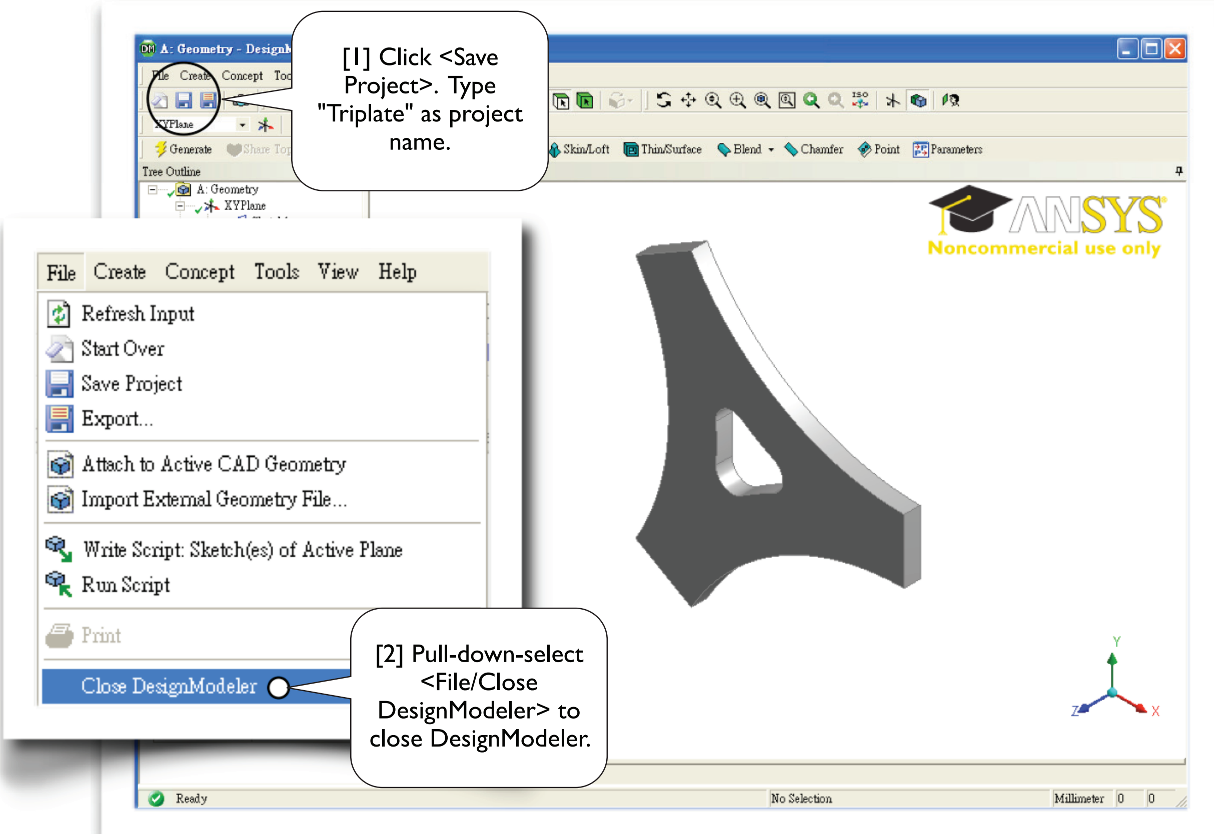


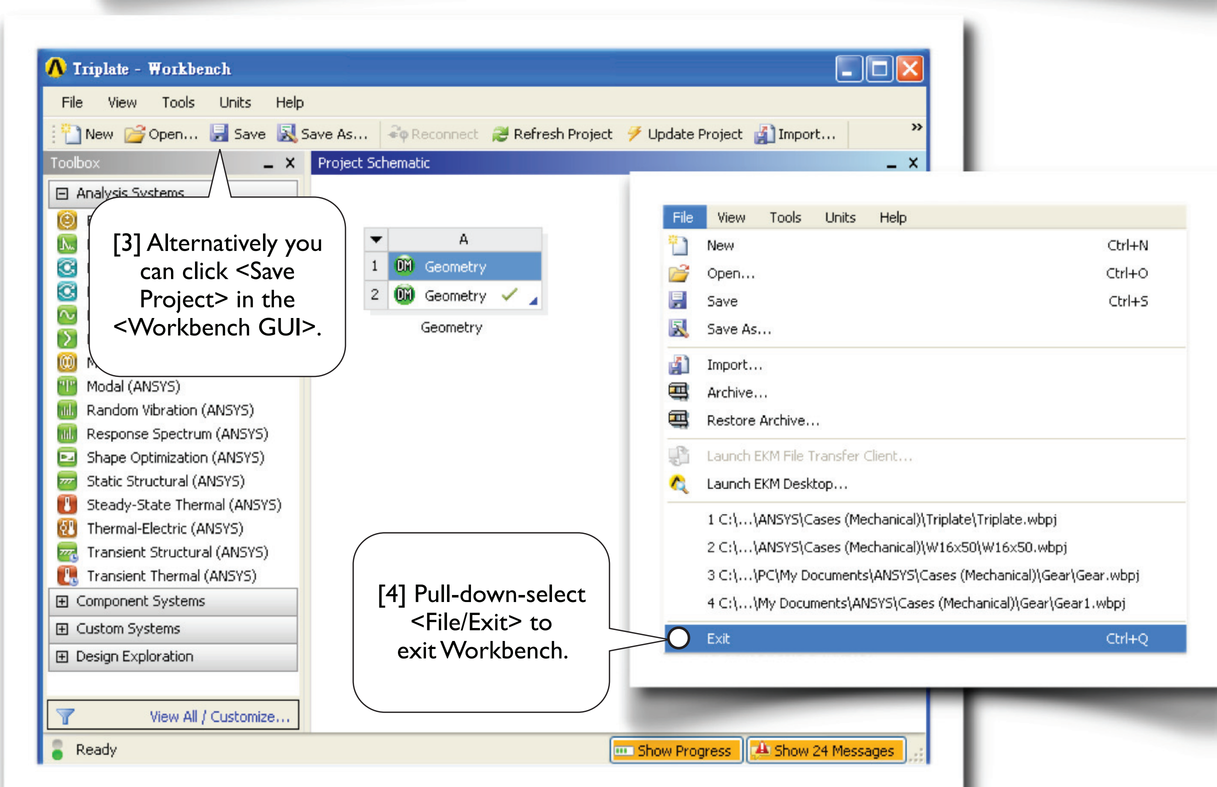


2.2-14 Extrude to Create 3D Solid



2.2-15 Save the Project and Exit Workbench





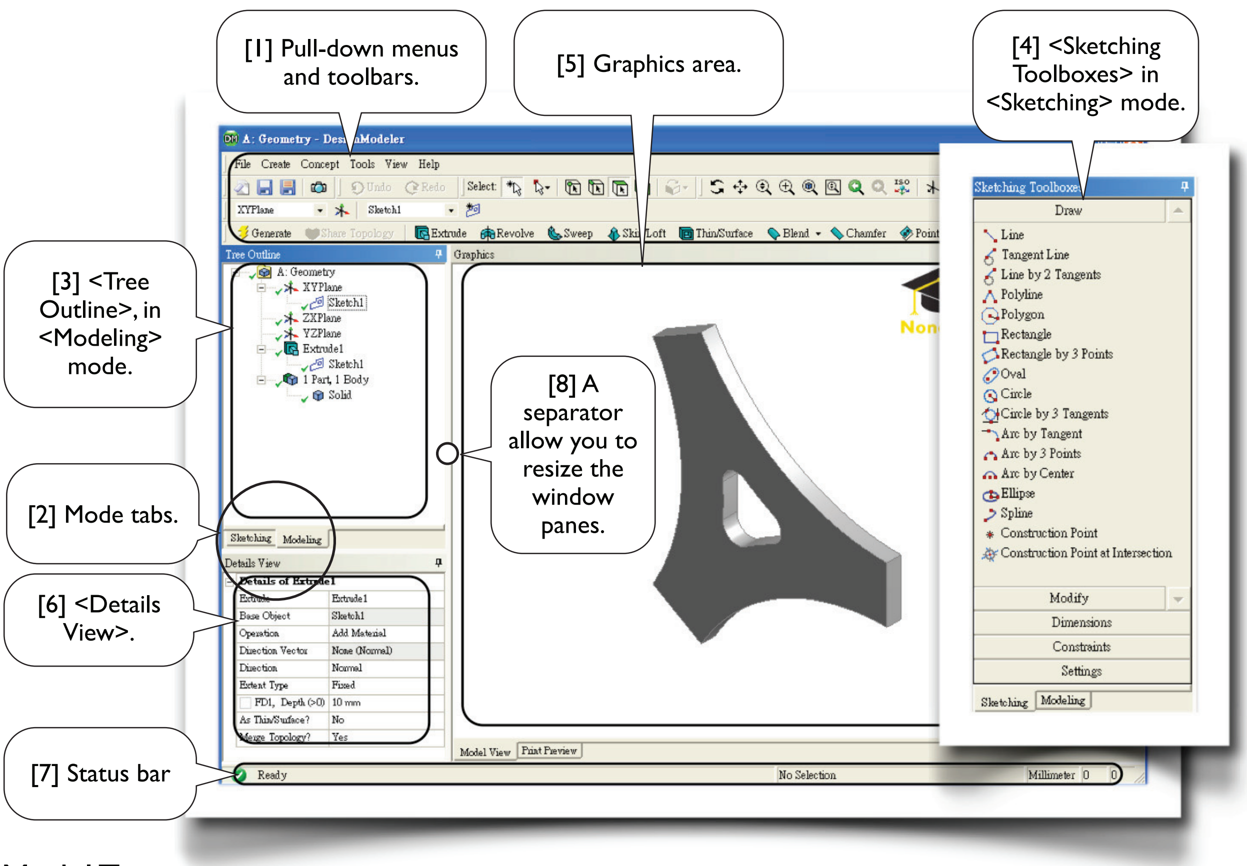
Section 2.3

More Details

2.3-1 DesignModeler GUI

The DesignModeler GUI is composed of 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 pane by dragging its title bar. Whenever you mess up the workspace, simply pull-down-select <View/Windows/Reset Layout> to reset the default layout.

The <Tree Outline> [3] shares the same area with the <Sketching Toolboxes> [4]; you switch between these two "modes" by clicking the "mode tab" [2]. The <Details View> [6] shows the detail information of the geometry you currently work with. The graphics area [5] displays the model when in <Model View> mode; you can click a tab to switch to <Print Preview>. We will cover more details of DesignModeler GUI in Chapter 4.



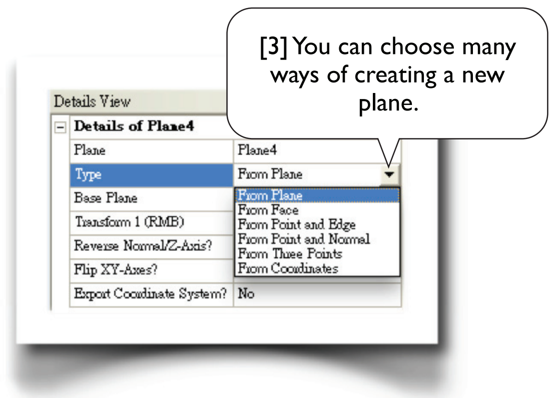
Model Tree

The <Tree Outline> contains an outline of the *model tree*, the tree representation of the geometric model. Each *leaf* and *branch* of the tree is called an *object*. A branch is an object containing one or more objects under itself. A model tree consists of *planes*, *features*, and a *part* branch. The parts are the only objects that are exported to <Mechanical>. Right-clicking an object and select a tool from the context menu, you can operate on the object, such as delete, rename, duplicate, etc.

The order of the objects is often relevant. DesignModeler renders the geometry according to the order. New objects are normally added one-by-one before the parts branch. 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 again.

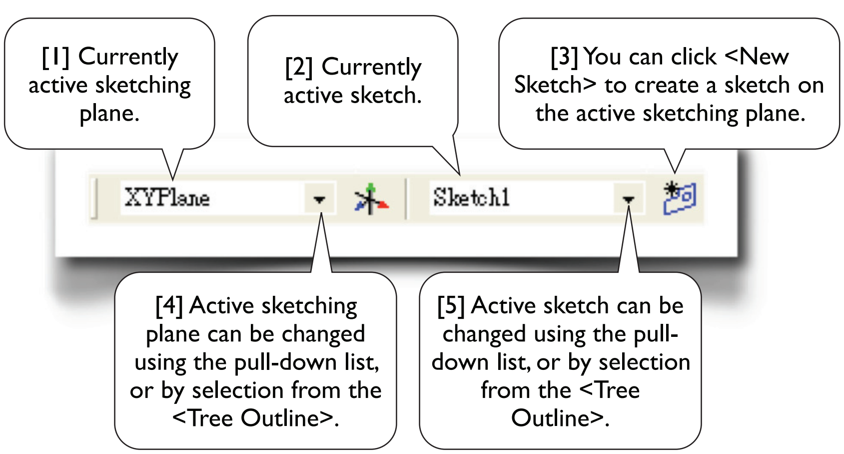
2.3-2 Sketching Planes

Sketches are created on *sketching planes*, or simply *planes*. Each sketch must be associated with a plane; each plane may have multiple sketches on it. In the beginning of a DesignModeler session, three planes are created automatically: <XYPlane>, <YZPlane>, and <ZXPlane>. Currently active plane is shown on the toolbar [1]. You can create new planes as needed [2]. There are many ways of creating a new plane [3]. In this chapter, since we assume sketches are created on the <XYPlane>, we will not discuss how to create sketching planes further, which will be discussed in Chapter 4. Usage of planes is not limited for storing sketches. Section 4.3-8 demonstrates another usage of planes.



2.3-3 Sketches

A sketch consists of *points* and *edges*; edges may be straight lines or curves. Along with these geometric entities, there are dimensions and constraints imposed on these 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 sketches, 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 need to click <New Sketch> [3]. Only one plane and one sketch is active at a time [1, 2]: newly created sketches are added to the active plane, and newly created geometric entities are added to the active sketch. In this chapter, we only work with a single sketch which is on the <XYPlane>. 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 a <Sketching Toolboxes> (2.3-1[4]). The <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 most efficient way to learn the tools is to try them out. During the tryout, whenever you want to clean up the graphics area, pull-down-select <File/Start Over>, or select all entities and then delete them. Some tools need further explanation, as described in the rest of this section.

Before we jump to discuss each of the toolboxes, some tips relevant to sketching are worth emphasizing first.

Pan, Zoom, and Box Zoom

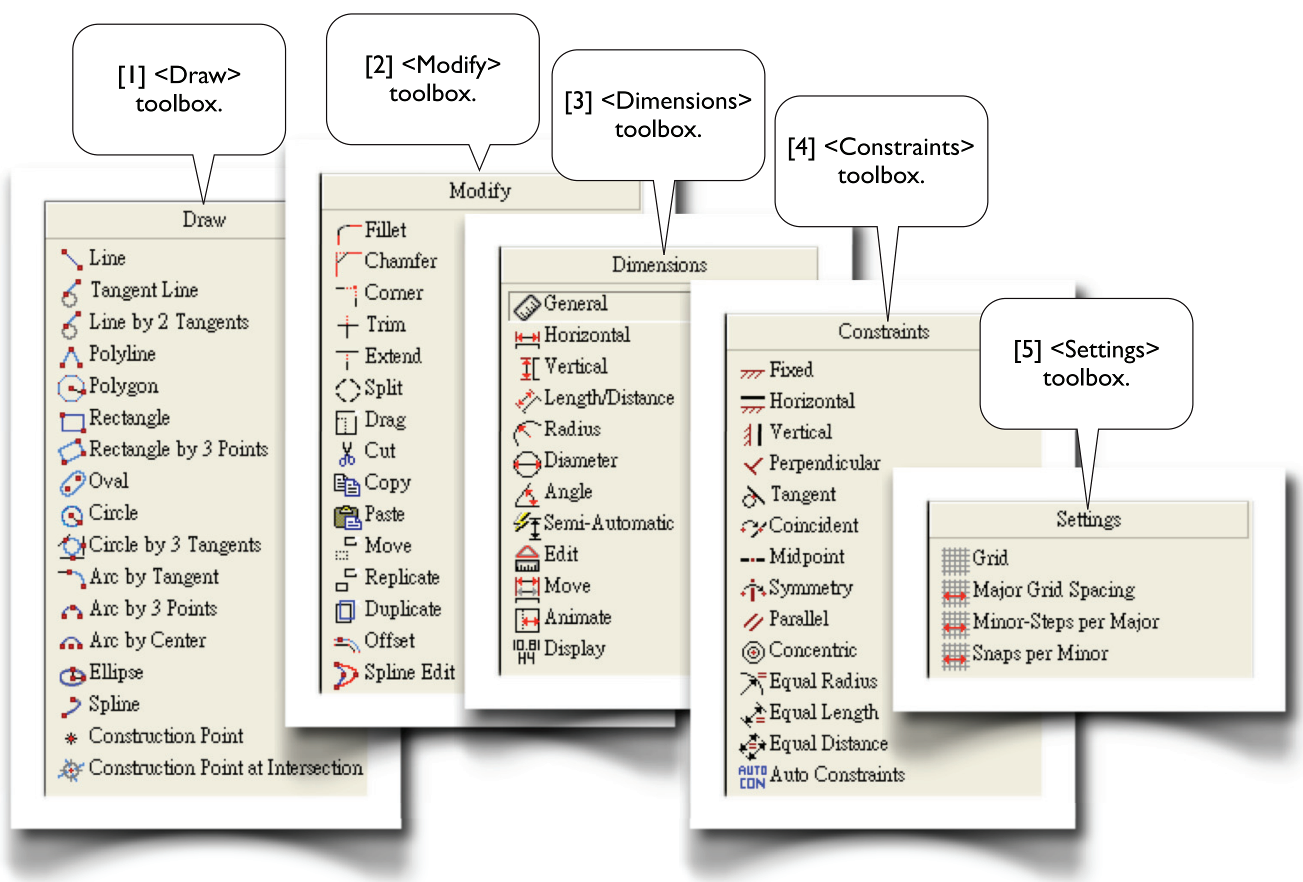
Besides the <Pan> tool (2.2-5[3]), the graphics can be panned by dragging your mouse while holding down both control key and the middle mouse button. Besides the <Zoom> tool (2.2-5[5]) the graphics can be zoomed in/out by simply rolling forward/backward your mouse wheel. The <Box Zoom> (2.2-5[4]) can be done by right-clicking and then dragging a rectangle in the graphics area. When you get use to these basic mouse actions, you probably don't need <Pan>, <Zoom>, and <Box Zoom> tools in the toolbar 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 wonder about what actions to do next. The coordinates of your mouse pointer are also shown in the status bar; they are sometimes useful.



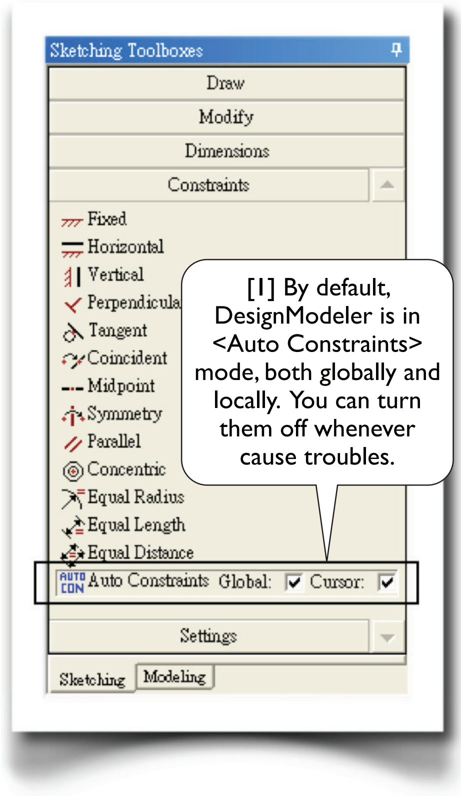
2.3-5 Auto Constraints[1, 2]

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 the points or edges. The following cursor symbols indicate the kind of constraints that will be applied:

C	- The point is coincident with a line.
P	- The point is coincident with another point.
H	- The line is horizontal.
V	- The line is vertical.
//	- The line is parallel to another line.
T	- The point is a tangent point.
⊥	- The point is a perpendicular foot.
R	- The circle's radius is equal to another circle's.

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].



2.3-6 <Draw> Tools[3]

Line by 2 Tangents

Select two curves, a line tangent to these two curves will be created. The curves can be circle, arc, ellipse, or spline.

Oval

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

Circle by 3 Tangents

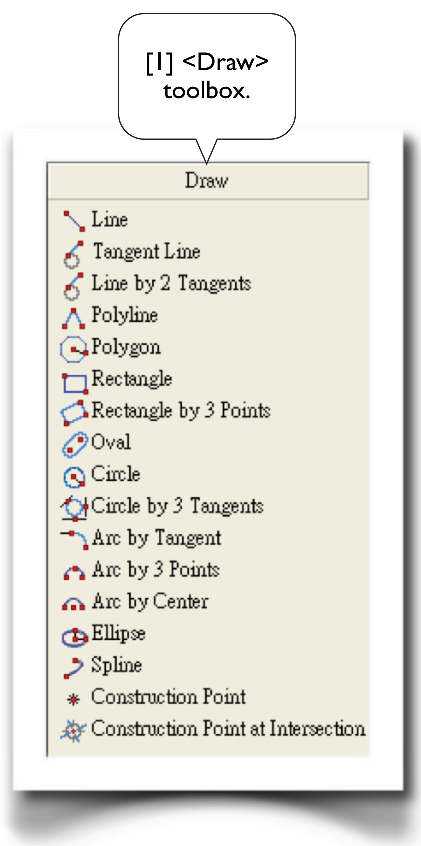
Select three edges, then a circle tangent to these three edges will be created. Remember that an edge can be a line or a curve.

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 point of the arc.

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 [2]: either open end or closed end; either with fit points or without fit points.

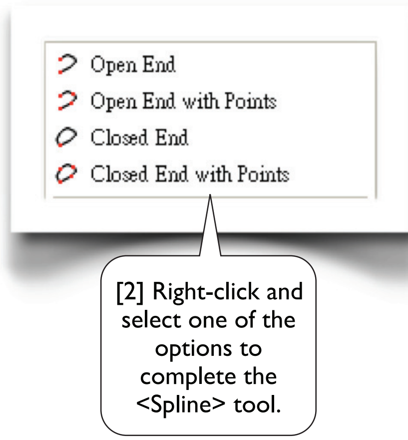


Construction Point at Intersection
Select two edges, a construction point will be created at the intersection.

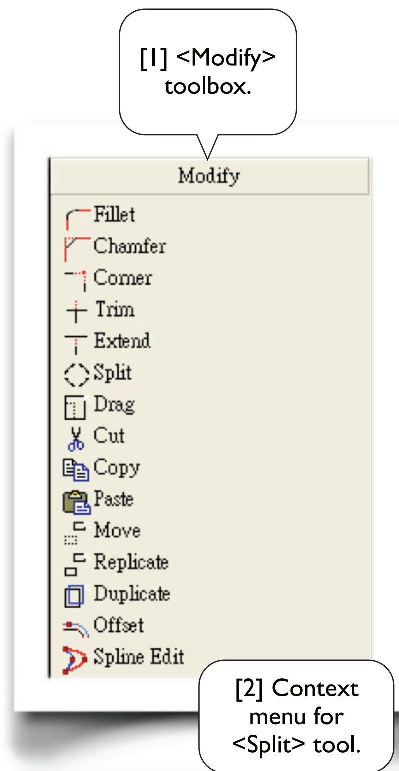
Delete Entities
There are no tools in the <Sketching Toolboxes> to delete entities. To delete entities, select them and right-click-select <Delete>. Multiple selection methods (e.g., control-selection and sweep-selection, see Section 2.1-6 and 2.2-12[2]), can be used to select entities.

Abort a Tool
To cancel a tool in any of toolbox, simply press <ESC>.

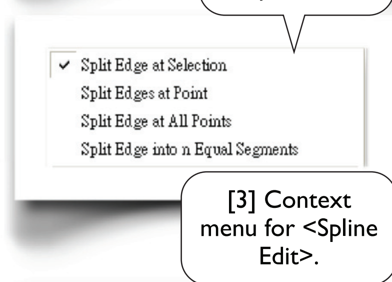
2.3-7 <Modify> Tools[4]



Corner
Click two entities, which can be lines or curves, the entities will be trimmed or extended up to the intersection point and form a sharp corner. The clicking points decide which sides to be trimmed.



Split
This tool split an edge into several segments depending on the options [2]. <Split Edge at Selection>: you click an edge, the edge will be split at the clicking point. <Split Edges at Point>: you click a point, all the edges passing through that point will be split at that point. <Split Edge at All Points>: you select an edge, the edge will be split at all points on the edge. <Split Edge into n Equal Segments>: You specify the value *n*, and select an edge, the edge will be split equally into *n* segments.



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

Cut
It is the same as <Copy>, except the originals are deleted.

Move
It is equivalent to a <Cut> followed by a <Paste>.

Replicate
It is equivalent to a <Copy> followed a <Paste>.

Duplicate
It is 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.

Spline Edit
It is used to modify flexible splines. You can insert, delete, drag the fit points, etc. For details, see the reference[4].

2.3-8 <Dimensions> Tools[5]

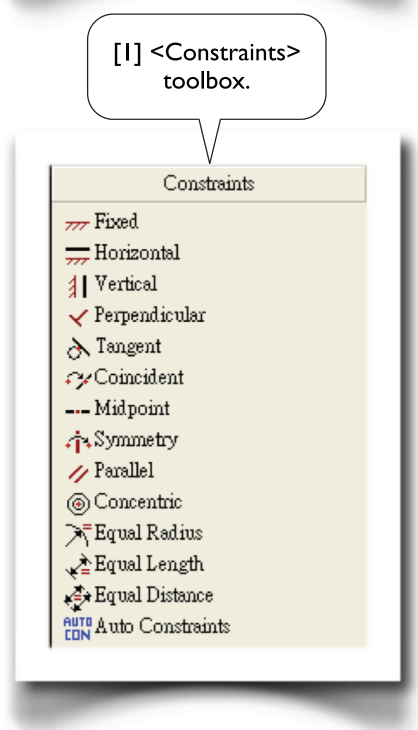
Semi-Automatic

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

Edit

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

2.3-9 <Constraints> Tools[6]



Fixed

It applies on any entity to make it fully constrained.

Horizontal

It applies on a line to make it horizontal.

Vertical

It applies on a line to make it vertical.

Perpendicular

It applies on two edges to make them perpendicular to each other.

Tangent

It 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. 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 then 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 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

It applies on two lines to make them parallel to each other.

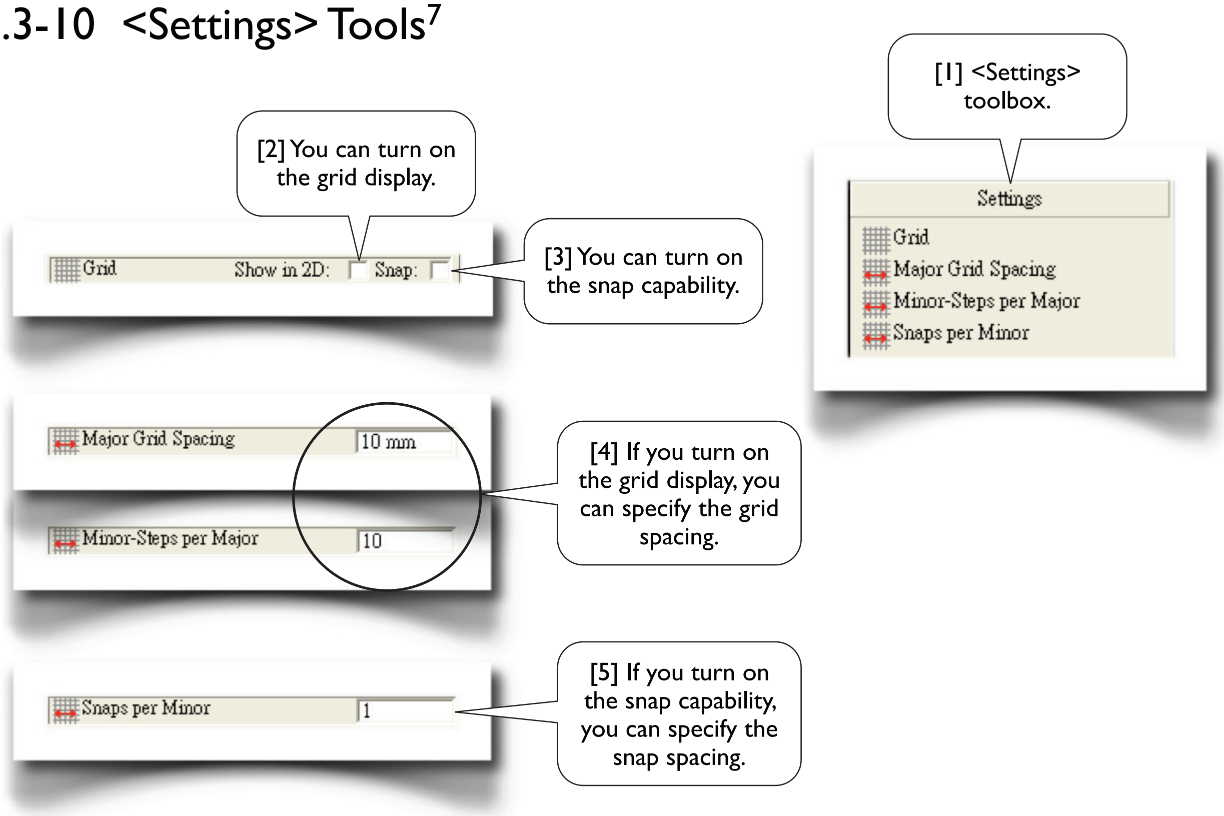
Concentric
It applies on two curves, which may be circle, arc, or ellipse, to make their centers coincident.

Equal Radius
It applies on two curves, which may be circle or arc, to make their radii equal.

Equal Length
It applies on two lines to make their lengths equal.

Equal Distance
It 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.

2.3-10 <Settings> Tools[7]

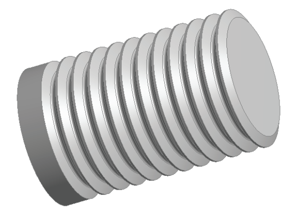


References

1. ANSYS Help System>DesignModeler>2D Sketching>Auto Constraints
2. ANSYS Help System>DesignModeler>2D Sketching>Constraints Toolbox>Auto Constraints
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

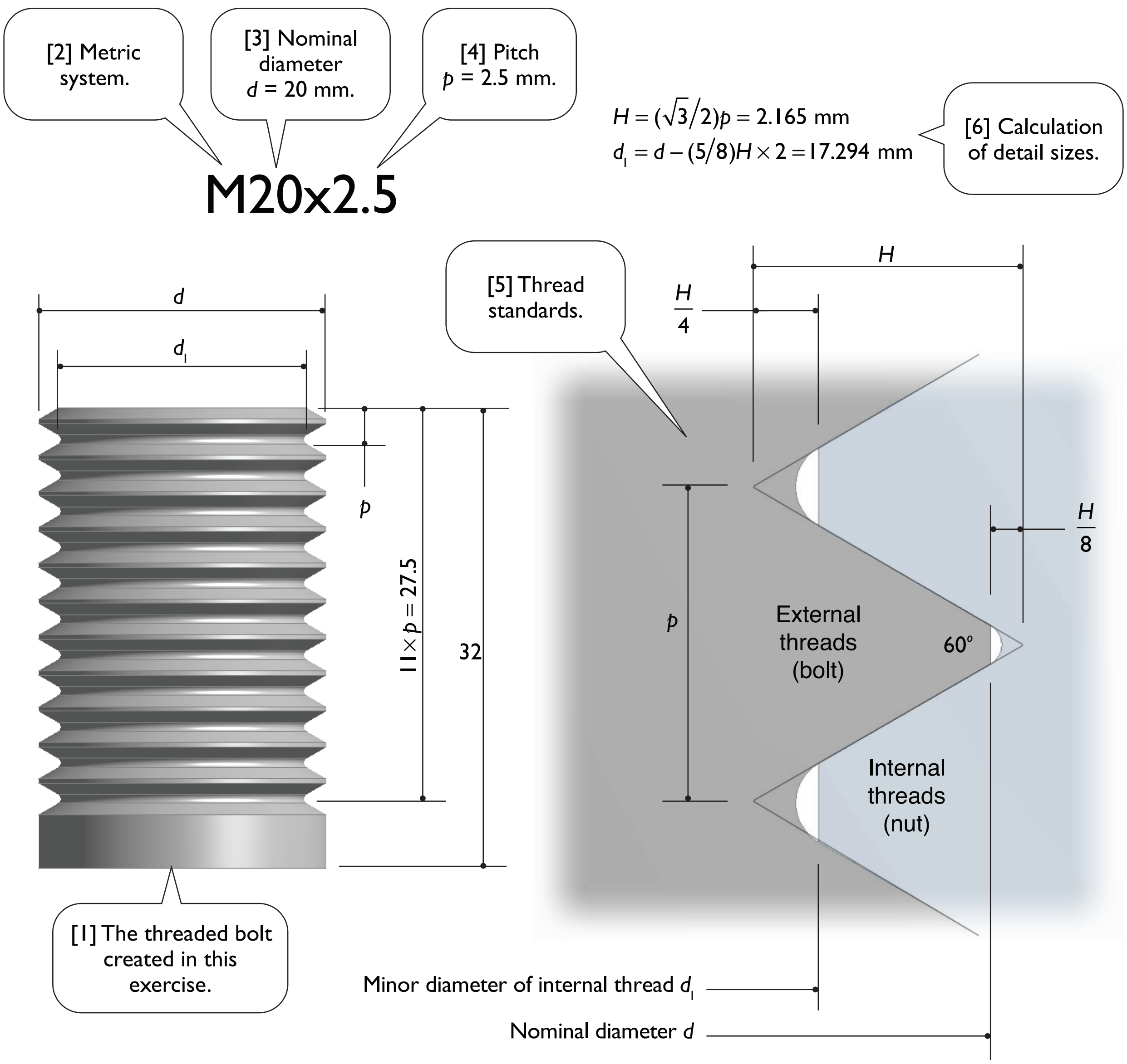
Section 2.4

Exercise: M20x2.5 Threaded Bolt



2.4-1 About the M20x2.5 Threaded Bolt

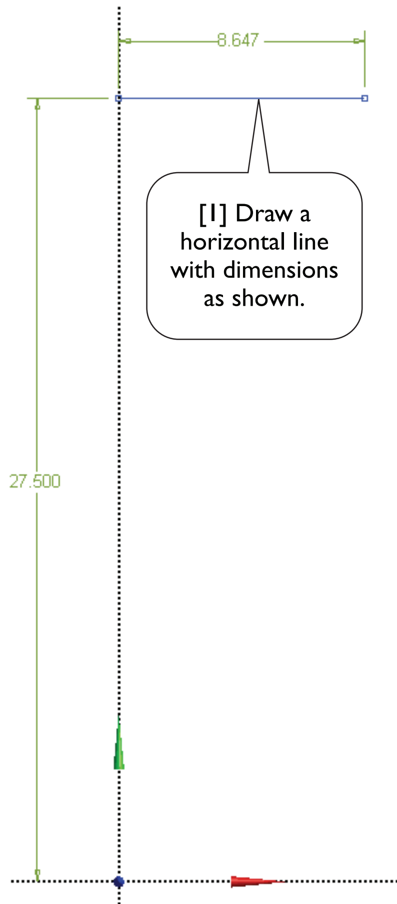
Consider a pair of threaded bolt and nut. The bolt has external threads while the nut has internal threads. This exercise is to created a sketch and revolve the sketch 360° to generate a solid body for a portion of the bolt [1] threaded with M20x2.5 [2-6]. In Section 3.2, we will use this sketch again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



2.4-2 Draw a Horizontal Line

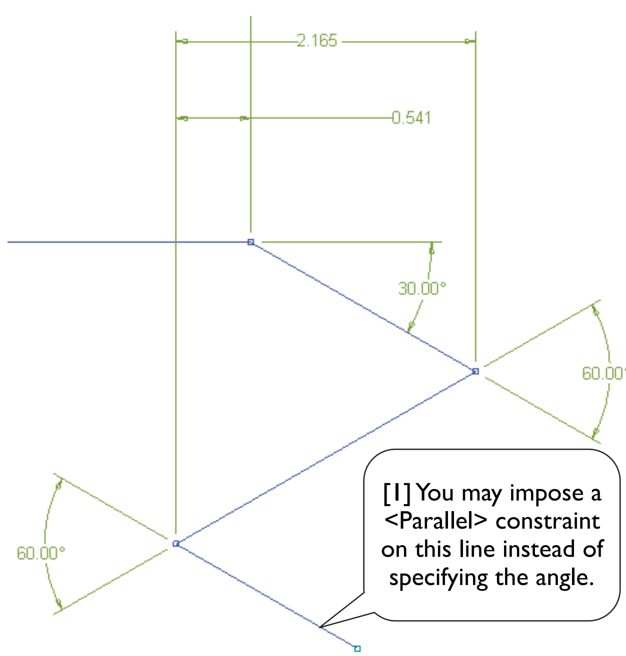
Launch <Workbench>. Create a <Geometry> System. Save the project as "Threads." Start up <DesignModeler>. Select <Millimeter> as length unit.

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



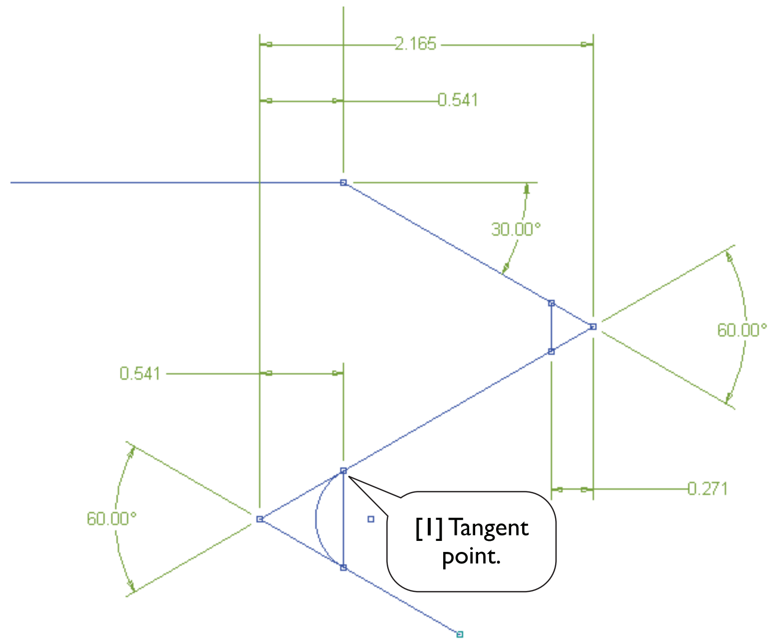
2.4-3 Draw a Polyline

Draw a polyline (totally 3 segments) and specify dimensions (30°, 60°, 60°, 0.541, and 2.165) as shown below. Note that, to avoid confusion, we explicitly specify all the dimensions. You may apply constraints instead. For example, using <Parallel> constraint in stead of specifying an angle dimension [1].

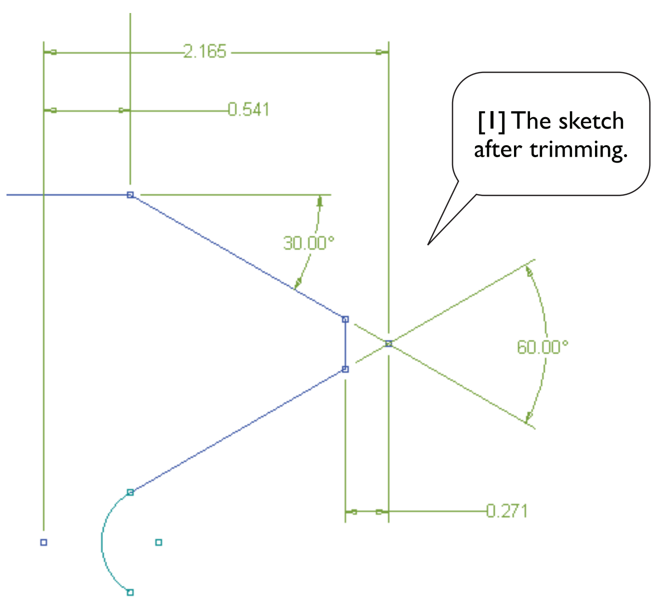


2.4-4 Draw Fillets

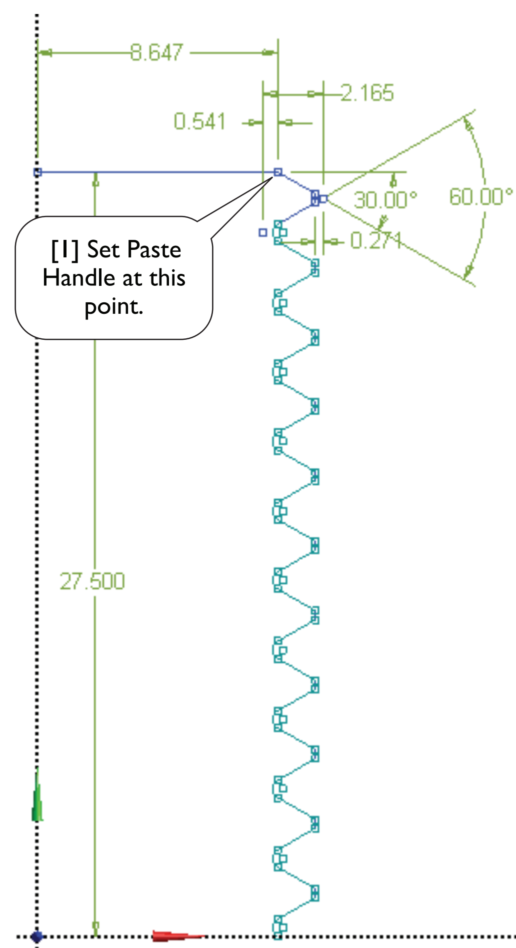
Draw two vertical lines and specify their positions (0.271 and 0.541). Draw an arc using <Arc by 3 Points>. If the arc is not in blue color, impose a <Tangent> constraint on the arc and one of its tangent line [1].



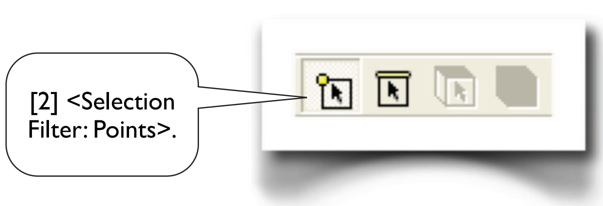
2.4-5 Trim Unwanted Segments



2.4-6 Replicate 10 Times

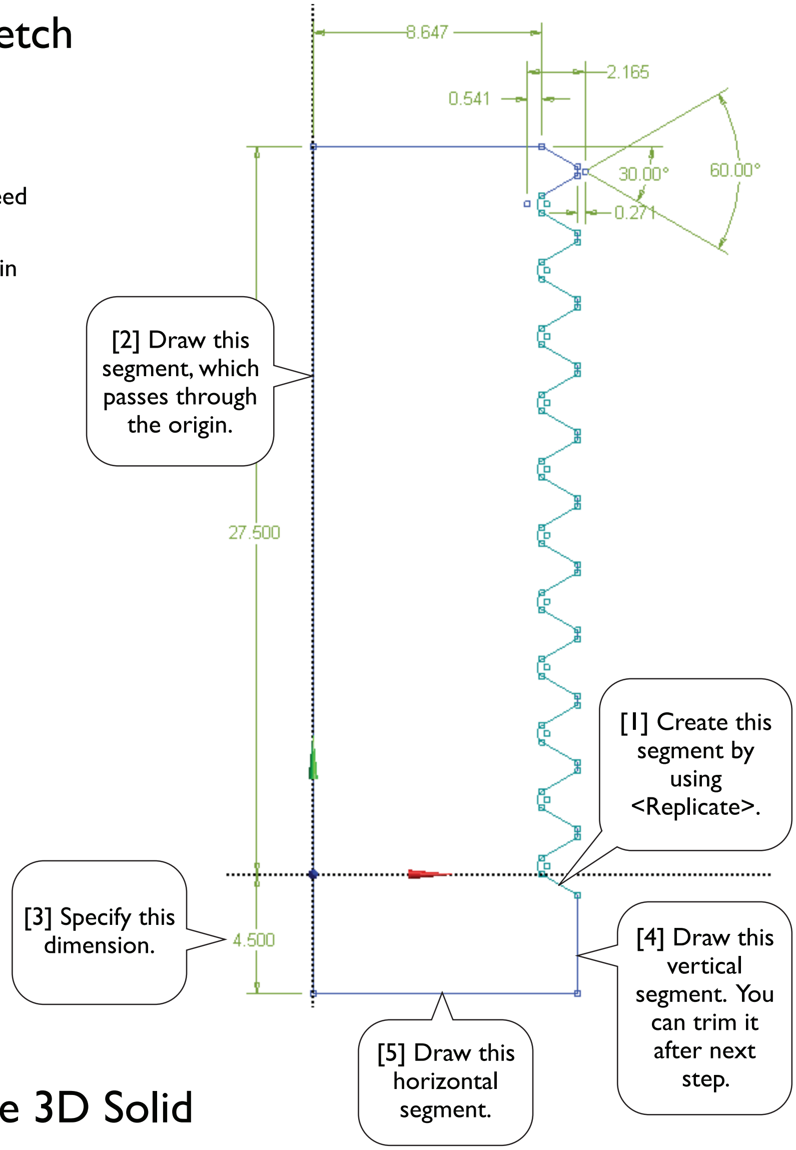


Select all segments except the horizontal one (totally 4 segments), and replicate 10 times. You may need to manually set the paste handle [1]. You may also need to use the tool <Selection Filter: Points> [2].



2.4-7 Complete the Sketch

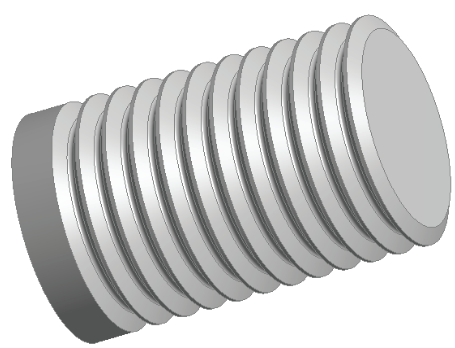
Follow the steps [1-5] to complete the sketch. Note that, in step [4], you don't need to worry about the length. After step [5], you can trim the vertical segment created in step [4].



2.4-8 Revolve to Create 3D Solid

Revolve the sketch to generate a solid of revolution. Select the Y-axis as the axis of revolution.

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

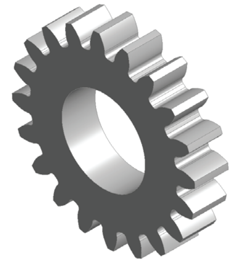


References

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

Exercise: Spur Gears

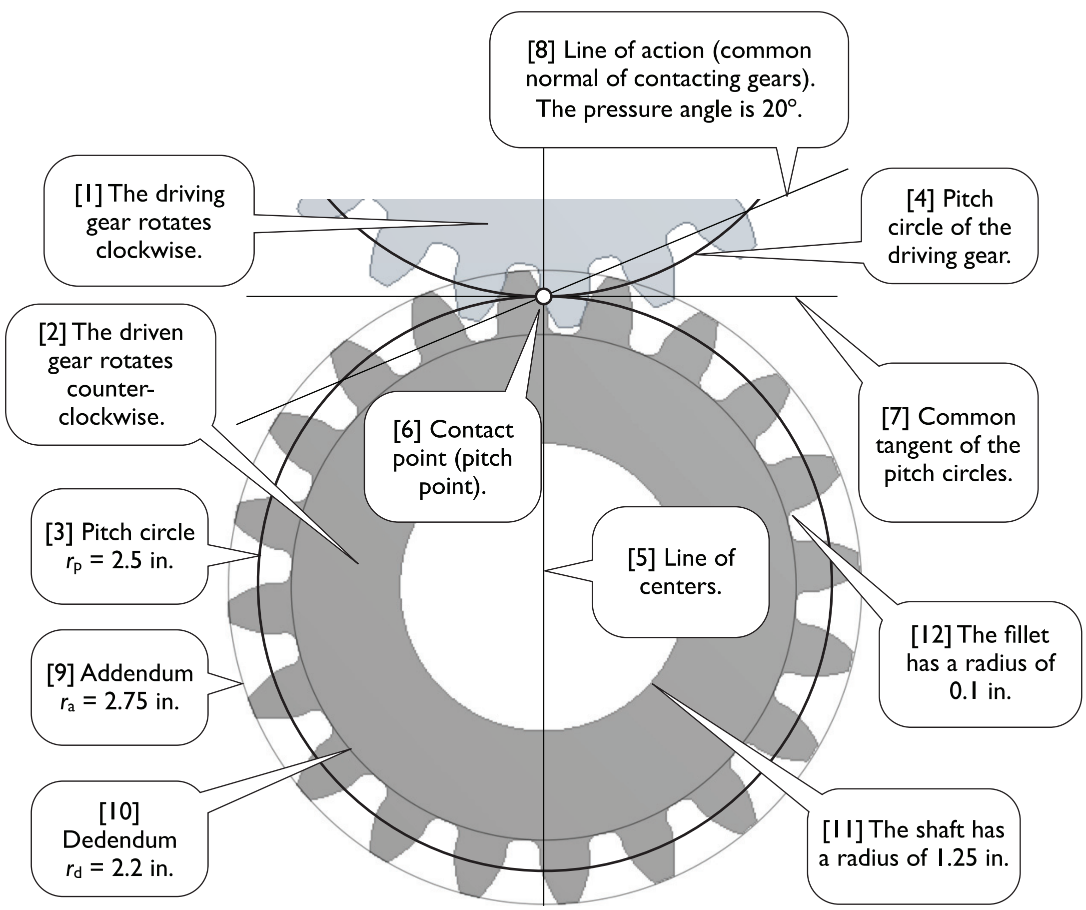


Geometric details of spur gears are important for a mechanical engineer. However, if you are not concerned about these geometric details for now, you may skip the first two subsections and jump directly to Subsection 2.5-3.

2.5-1 About the Spur Gears

The figure below shows a pair of identical spur gears in mesh [1-12]. Spur gears have their teeth cut parallel to the axis of the shaft on which the gears are mounted. Spur gears are used to transmit power between parallel shafts. In order that two meshing gears maintain a constant angular velocity ratio, they must satisfy the fundamental law of gearing: the shape of the teeth must be such that the common normal at the point of contact between two teeth must always pass through a fixed point on the line of centers[1] [5]. This fixed point is called the *pitch point* [6].

The angle between the line of action and the common tangent [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 (α = 20°) [8], and number of teeth (N = 20). In addition, 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 calculated 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 $\widehat{BCDEF}$ is the base circle, by the definition of involute curve,

$$\overline{BA} = \widehat{BC} + \overline{CP} = \widehat{BCDEF} \tag{1}$$

$$\overline{CP} = \widehat{CDEF} \tag{2}$$

From ΔOCP,

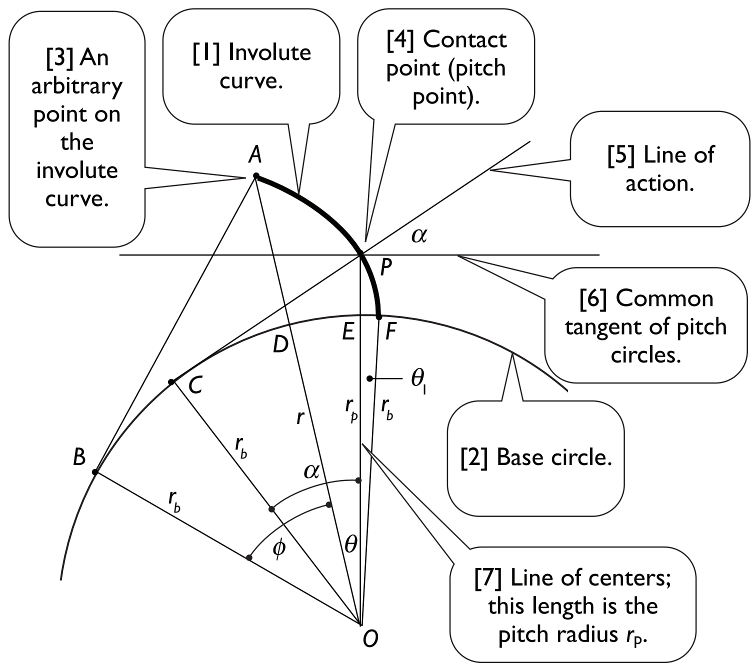
$$r_b = r_p \cos\alpha \tag{3}$$

From ΔOBA,

$$r = \frac{r_b}{\cos\phi} \tag{4}$$

Or equivalently,

$$\phi = \cos^{-1}\frac{r_b}{r} \tag{5}$$



To calculate θ, we notice that

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

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

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

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

$$\theta = (\tan\phi) - \phi - \theta_1 \tag{6}$$

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

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

$$\theta_1 = (\tan\alpha) - \alpha \tag{7}$$

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,

$$x = -r\sin\theta, \quad y = r\cos\theta \tag{8}$$

Numerical Calculations

In our case, the pitch radius $r_p = 2.5$ in, and pressure angle $\alpha = 20^\circ$; from Eqs. (2) and (7),

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

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

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.

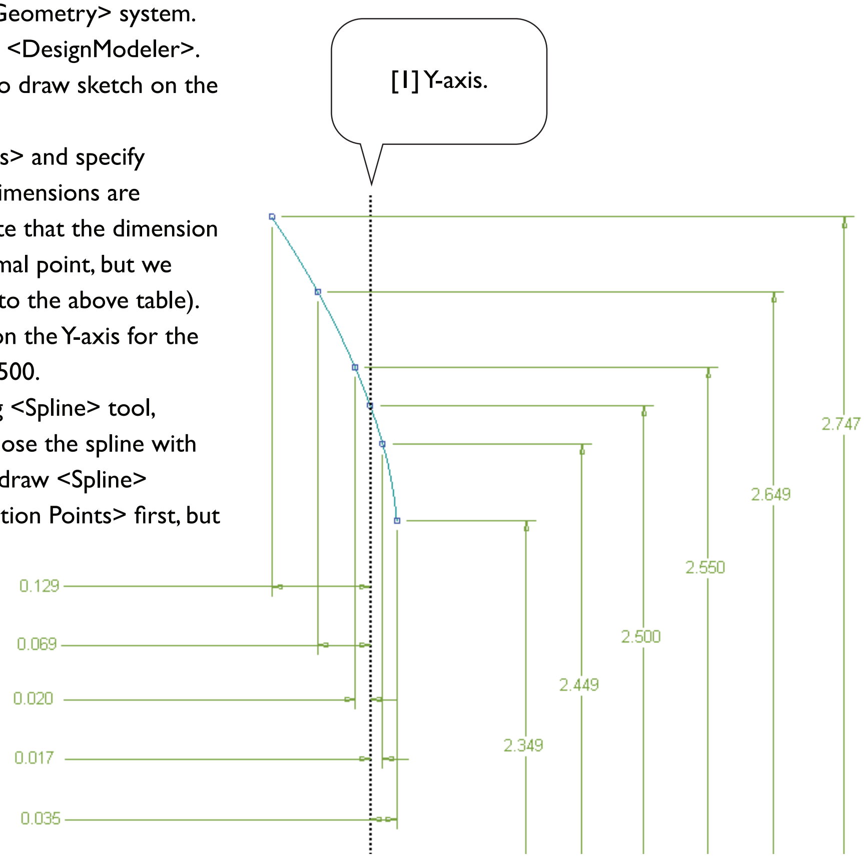
r in.	ϕ Eq. (4), degrees	θ Eq. (5), degrees	x	y
2.349232	0.000000	-0.853958	-0.03501	2.34897
2.449424	16.444249	-0.387049	-0.01655	2.44937
2.500000	20.000000	0.000000	0.00000	2.50000
2.549616	22.867481	0.442933	0.01971	2.54954
2.649808	27.555054	1.487291	0.06878	2.64892
2.750000	31.321258	2.690287	0.12908	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.

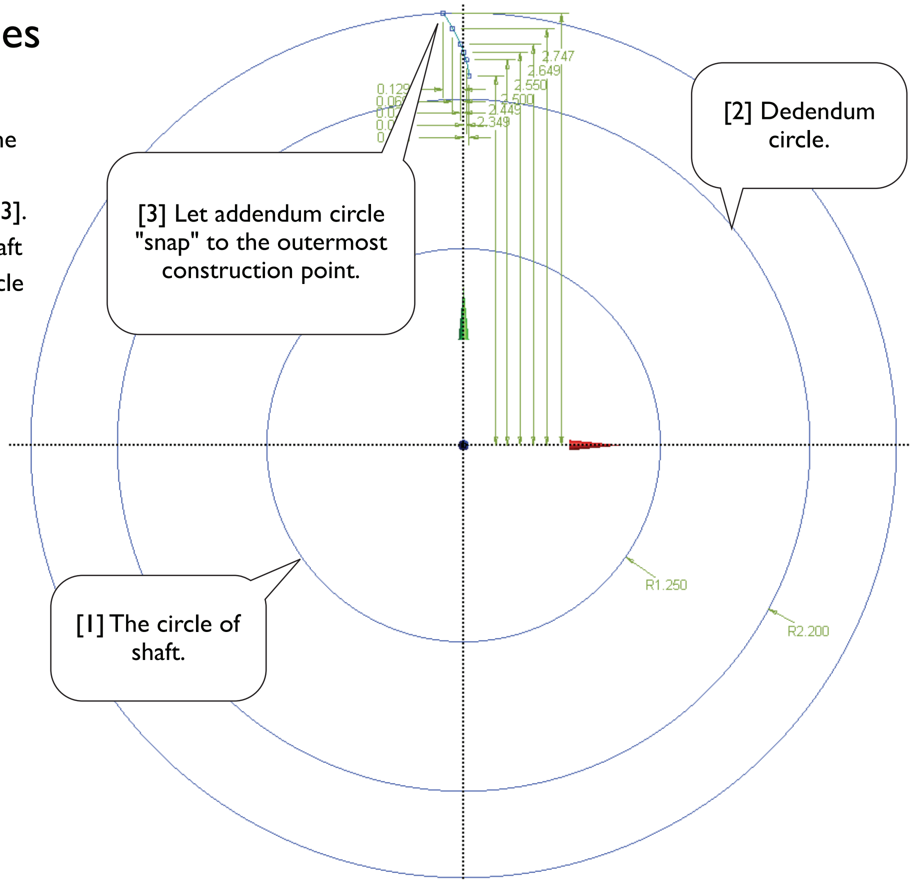
Draw six <Construction Points> and specify dimensions as shown (the vertical dimensions are measured down to the X-axis). Note that the dimension values display three digits after decimal point, but we actually typed with five digits (refer to the above table). Impose a <Coincident> constraint on the Y-axis for the point which has a Y-coordinate of 2.500.

Connect these six points using <Spline> tool, keeping <Flexible> option on, and close the spline with <Open End>. Note that you could draw <Spline> directly without creating <Construction Points> first, but that would be not so easy.



2.5-4 Draw Circles

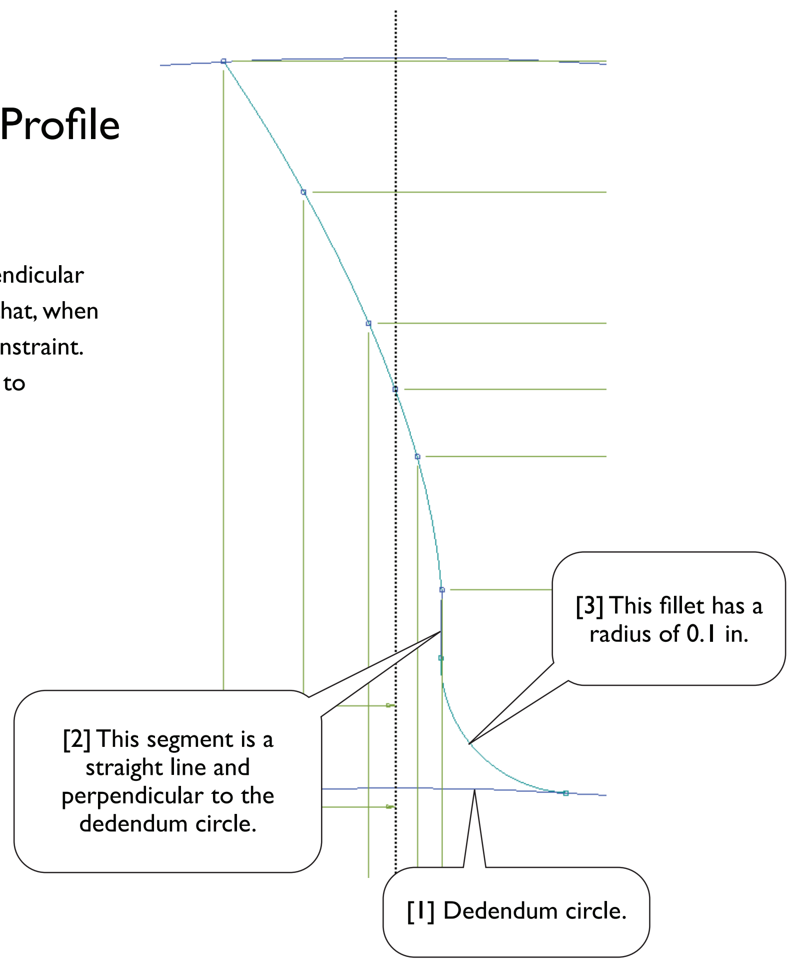
Draw three circles [1-3]. Let the addendum circle "snap" to the outermost construction point [3]. Specify radii for the circle of shaft (1.25 in) and the dedendum circle (2.2 in).



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.

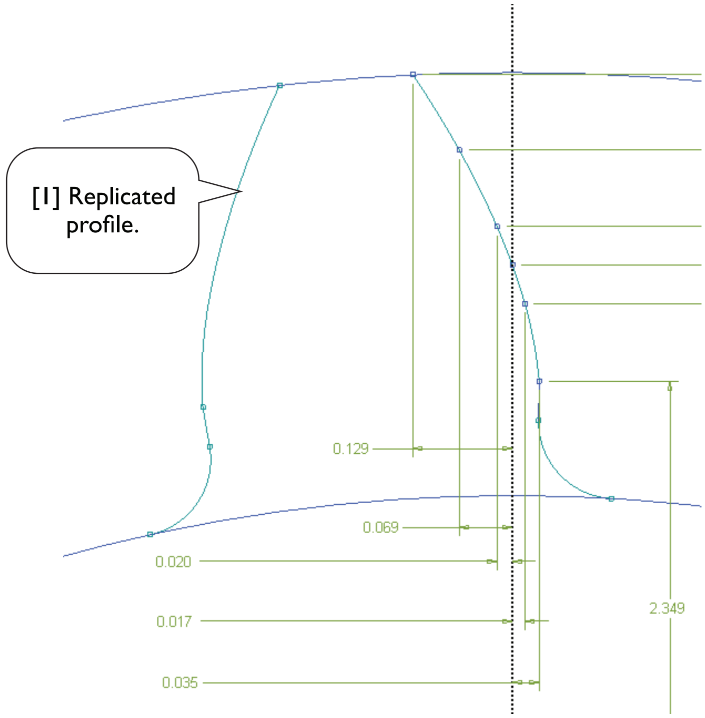
Draw a fillet [3] of radius 0.1 in to complete the profile of a tooth.



2.5-6 Replicate the Profile

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

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

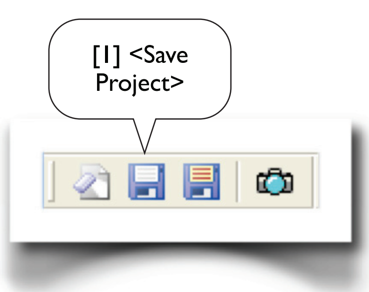


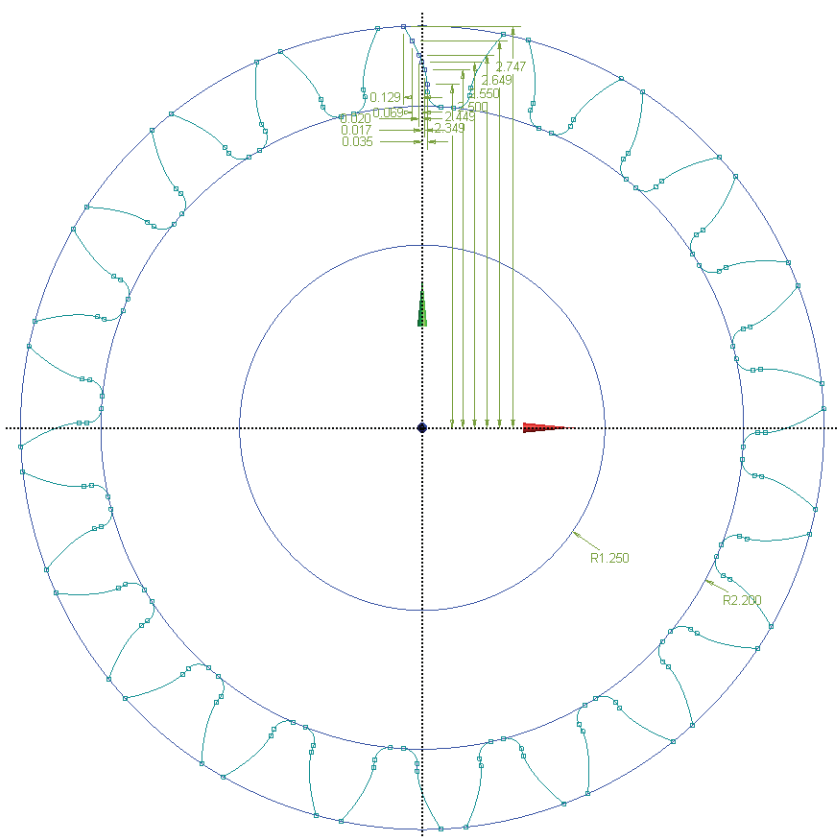
2.5-7 Replicate Profiles 19 Times

Activate <Replicate> tool again, type 18 (degrees) for <r>. Select both left and right profiles (totally 6 segments), <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).

As the geometric entities is getting more and complicated, the computer's processing time may be getting slower, depending on your hardware configuration.

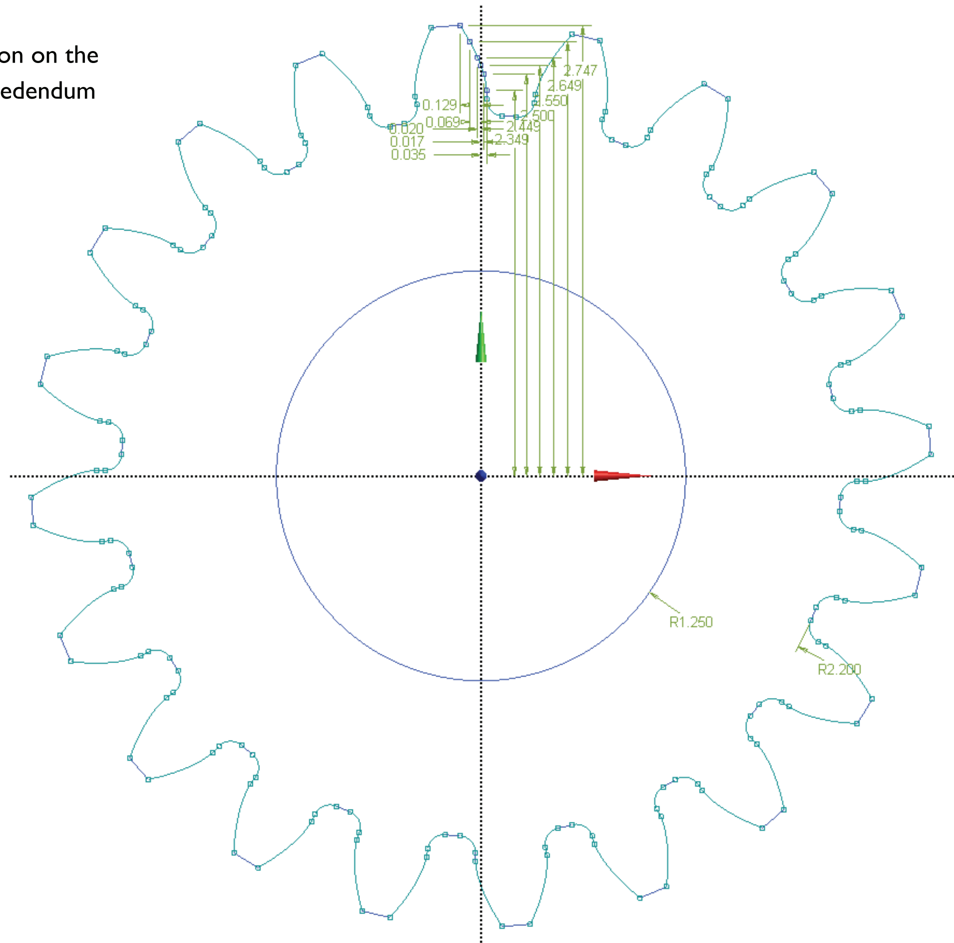
Save your project once a while by clicking the <Save Project> tool in the toolbar.





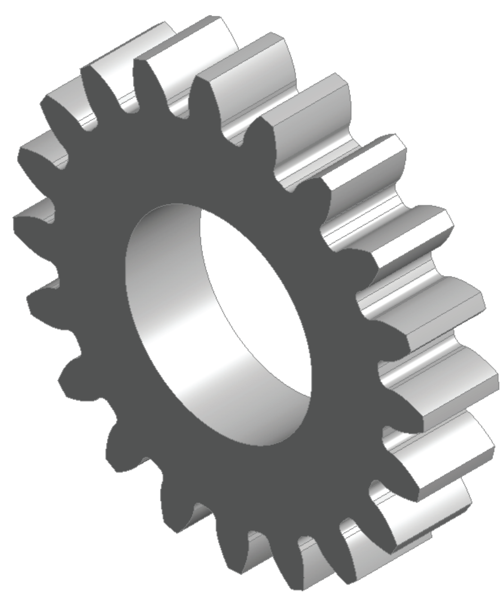
2.5-8 Trim Away Unwanted Segments

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



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.

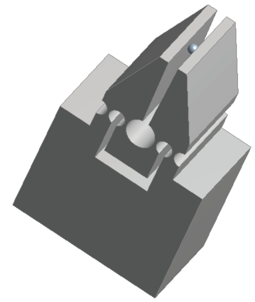


References

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

Exercise: Microgripper



2.6-1 About the Microgripper

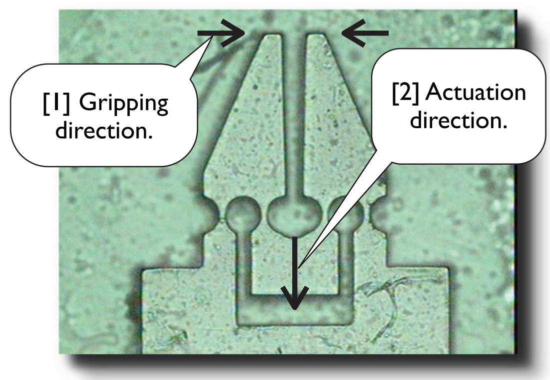
Many manipulators are designed as mechanisms, that is, they consist of bodies connected by joints, such as revolute joints, sliding joints, etc., and the motions are mostly governed by the laws of rigid body kinematics.

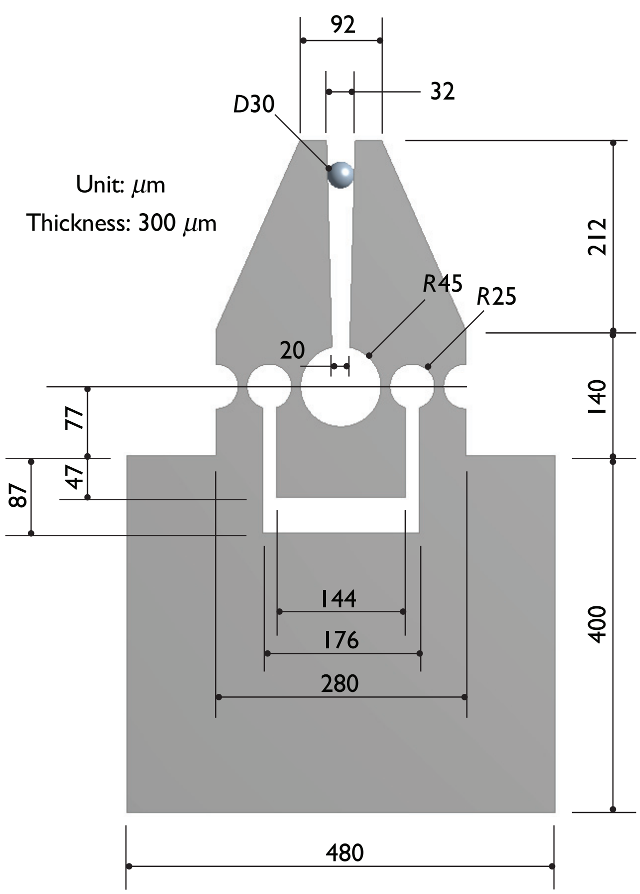
The microgripper discussed here [1-2] is a structure rather than a mechanism; the mobility are provided by the flexibility of the materials, rather than the joints.

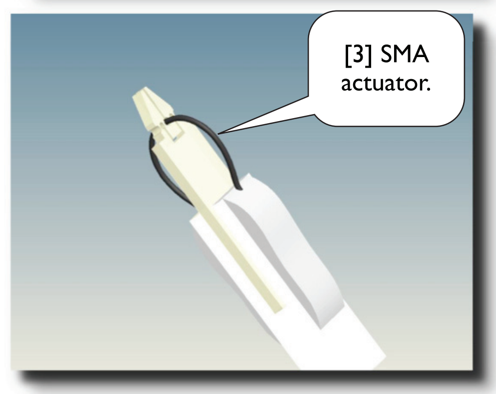
The microgripper is made of PDMS (polydimethylsiloxane, see Section 1.1-1). The device is actuated by a shape memory alloy (SMA) actuator [3], of which the motion is caused by temperature change, and the temperature is in turn controlled by electric current.

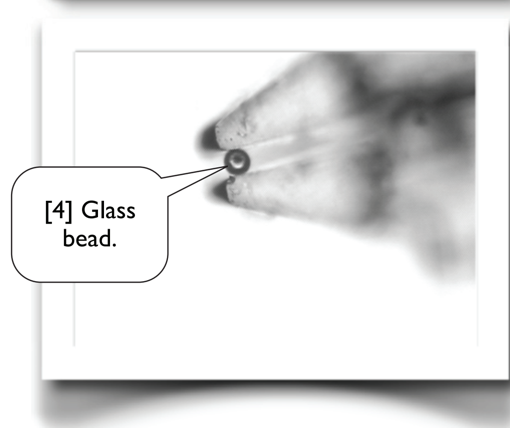
In the lab, the microgripper is tested by gripping a glass bead of a diameter of 30 micrometer [4].

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.





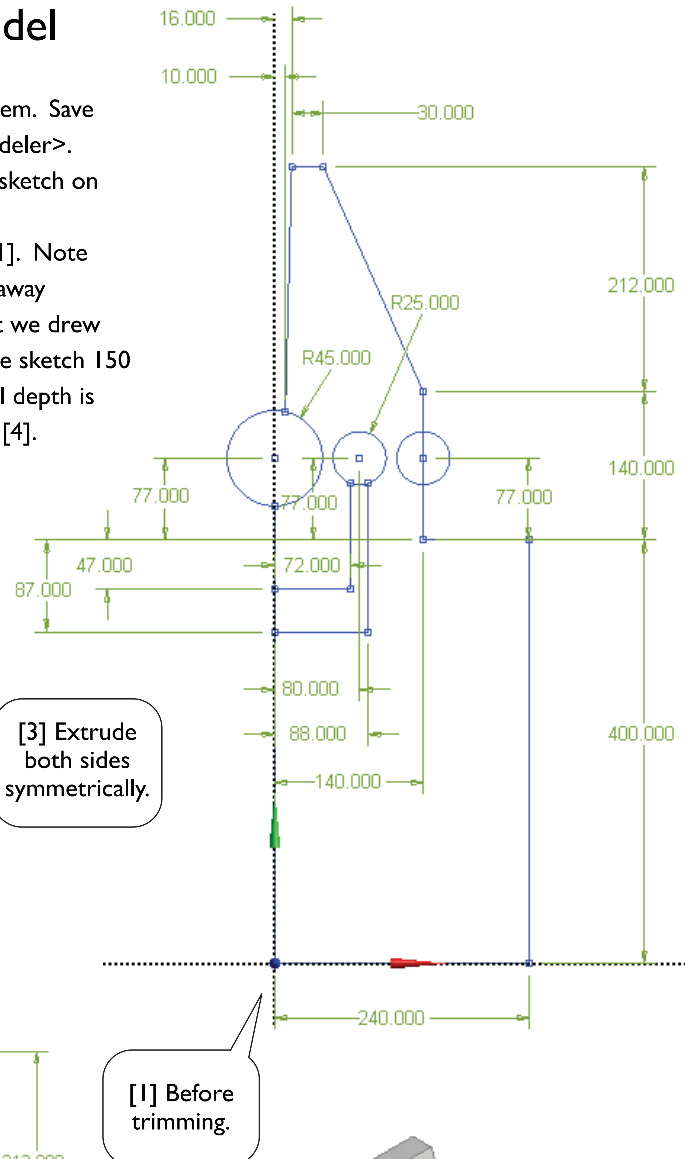




2.6-2 Create Half of the Model

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

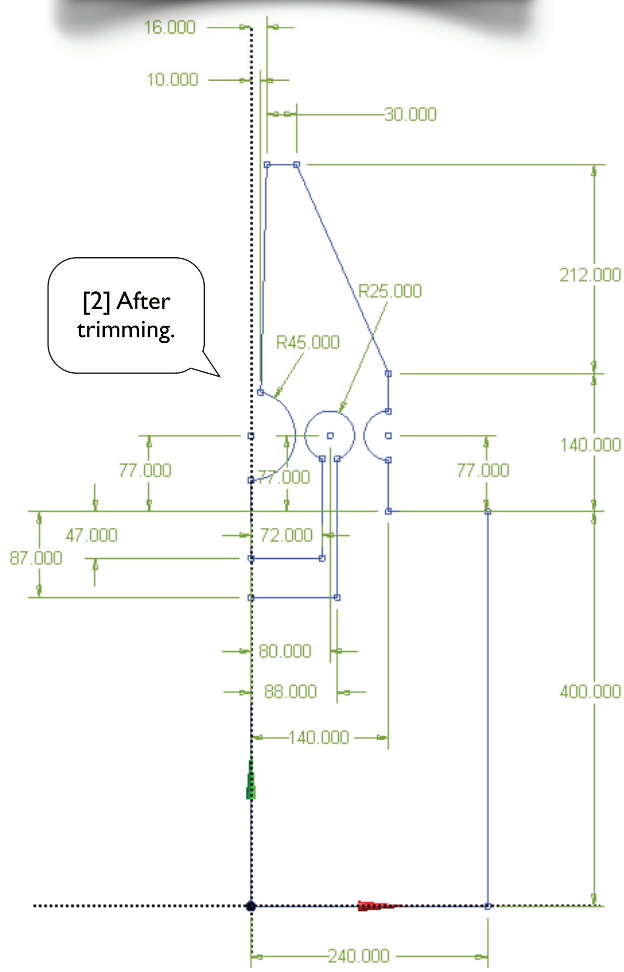
Draw the sketch as shown on the right side [1]. Note that two of the three circles have equal radii. Trim away unwanted segments as shown below [2]. Note that we drew half of the model, due to the symmetry. Extrude the sketch 150 microns both sides of the plane symmetrically (total depth is 300 microns)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

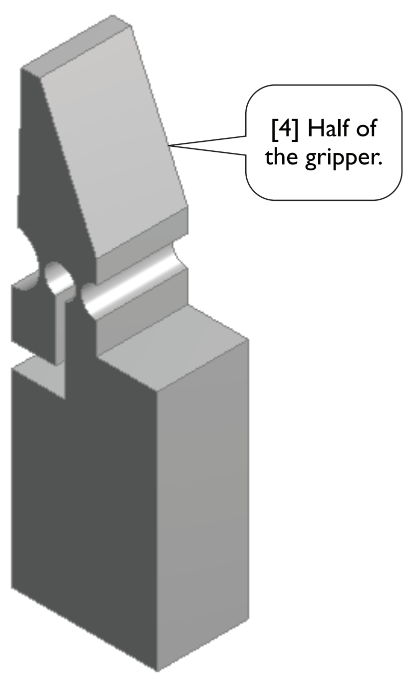


Details View

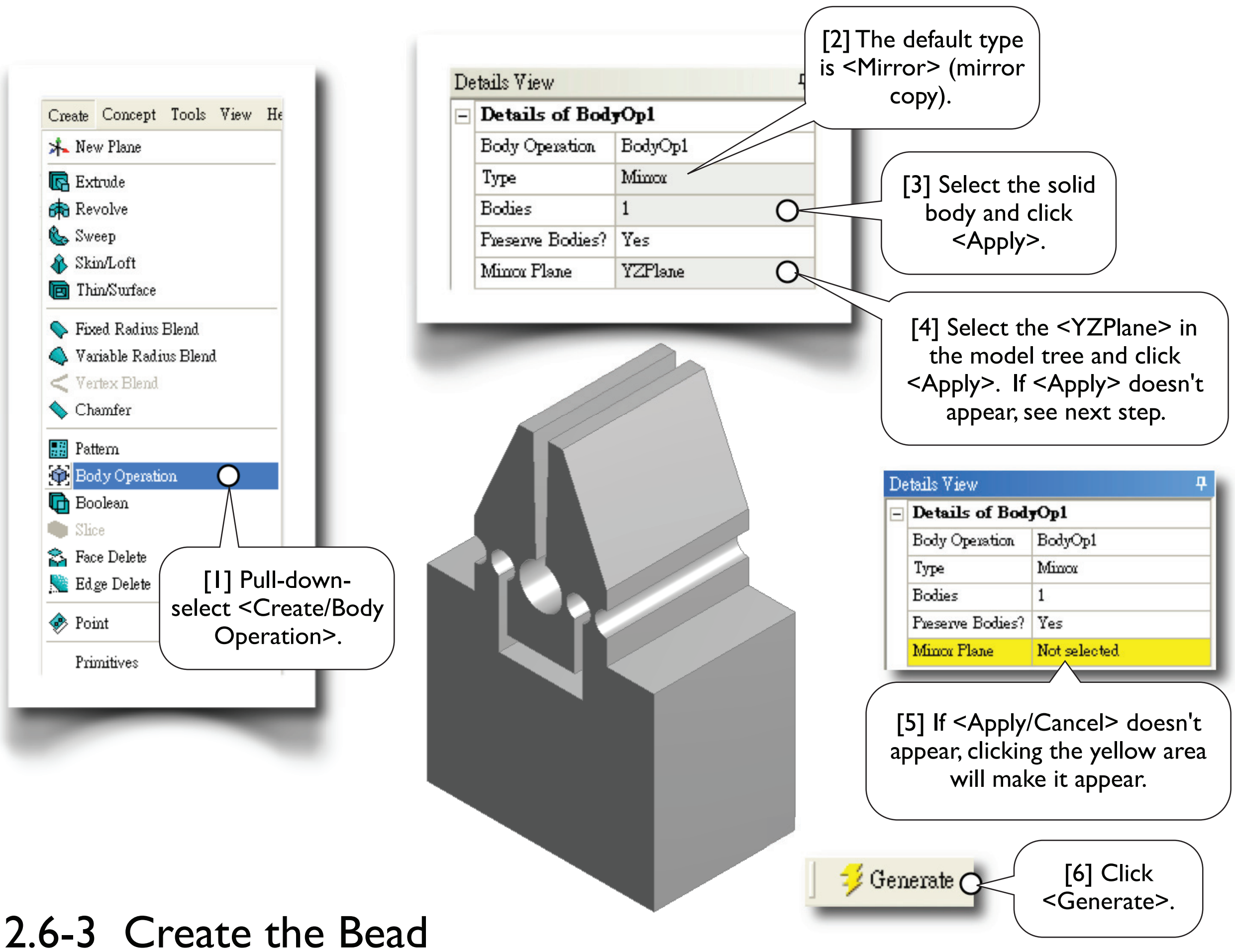
Details of Extrude1	
Extrude	Extrude1
Base Object	Sketch1
Operation	Add Material
Direction Vector	None (Normal)
Direction	Both - Symmetric
Extent Type	Fixed
FD1, Depth (>0)	150 µm
As Thin/Surface?	No
Merge Topology?	Yes

[3] Extrude both sides symmetrically.



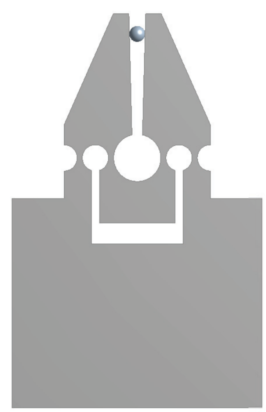


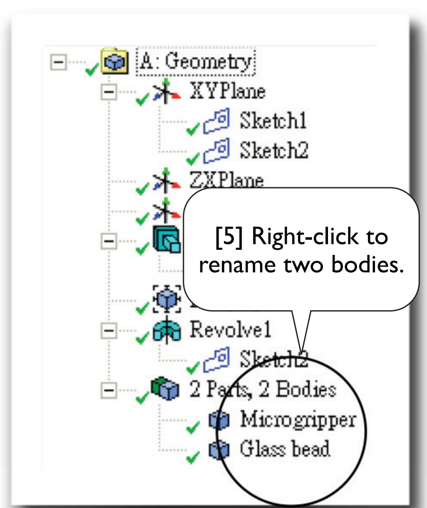
2.6-2 Mirror Copy the Solid Body

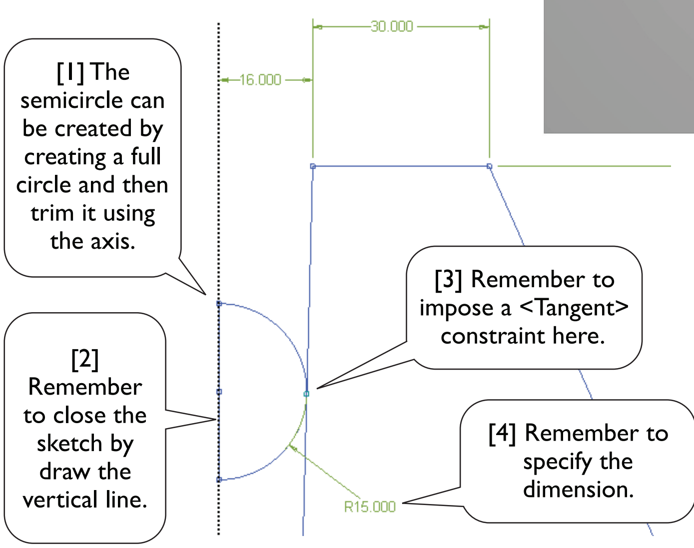


2.6-3 Create the Bead

Create a new sketch on XYPlane and draw a semicircle as shown [1-4]. Revolve the sketch 360 degrees to create the glass bead. Note that the two bodies are treated as two parts. Rename two bodies [5].







Wrap Up

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

Section 2.7

Problems

2.7-1 Key Concepts

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 edges 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 the <Tree Outline> in DesignModeler. A model tree consists of planes, 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 the <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 become selectable/unselectable. In <Sketching> Mode, there are two selection filters which corresponding to points and edges respectively. 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 constrain 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 Workbench Exercises

Create the Triangular Plate with Your Own Way

After so many exercises, you should be able to figure out an alternative way of creating the geometric model for the triangular plate (Section 2.2) on your own. Can you figure out a more efficient way?