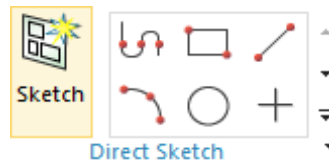# 1 Sketching

1.1 Introduction

Sketching is arguably one of the more difficult techniques to master in NX, but it is well-worth the effort. A single sketch can capture a tremendous amount of design intent, and inform the sequencing of subsequent solid modeling tools so that the design process becomes fluid and effortless.

NX has two sketch environments – which is a point of confusion for new users. The **Direct Sketch** environment is accessible via the **Sketch** icon on the far left side of the **Home** tab.


When you enter the **Direct Sketch** environment, the **Direct Sketch** group on the left side of the **Home** tab expands and appears as below.

The other sketch environment is accessible via **More / Open in Sketch Task Environment.**
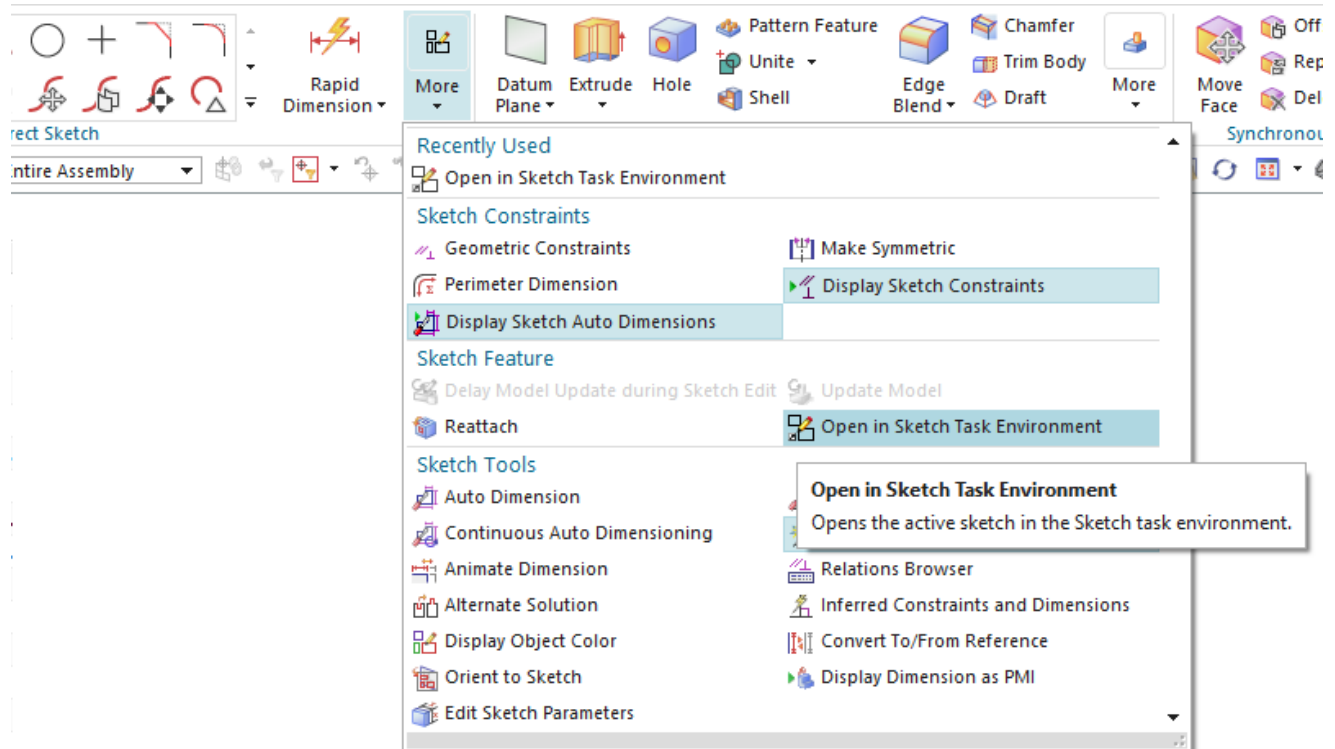
When sketching in the **Task Environment**, the ribbon changes completely – most tabs disappear, and **File** becomes **Task**. Additionally, the entire **Home** tab is populated with sketch-specific tools. Their names are displayed, too!
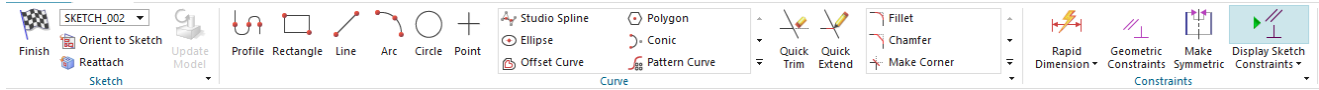
In either sketch mode, the sketch tools and **Finish Sketch** icon are always on the **Home** tab. For most of this chapter, we will be using **Direct Sketch**.

Our focus for **Direct Sketch** in NX11 at this stage is purely pedagogical – in our extensive training experience, **Direct Sketch** is by default the out of the box state of NX11, so we ask that you follow along and use **Direct Sketch** for the duration of this chapter. Here is a quick summary of the pros and cons of each:

Direct Sketch

- Is faster for simple sketches (less clicks to reach basic sketch tools)
- Does not require you to click **Finish Sketch** before selecting a non-sketch modeling tool.
- Allows the use of Animate Dimension.

Task Environment

- Is better for complex sketches (less clicks for many advanced sketch tools)
- Has a better layout on the ribbon
- Changes the Application Radials (**[Ctrl]**+**[Shift]**+click)
- Requires Customization on the Ribbon Bar.

1.1.1 Continuous Auto Dimensioning

When you first begin sketching, you will likely encounter **Continuous Auto Dimensioning**. This option automatically places dimensions for you as you sketch and ensures that your sketch is fully constrained. The figure below shows two versions of the same sketch – on the left, **Continuous Auto Dimensioning** is enabled, and on the right, it is disabled.
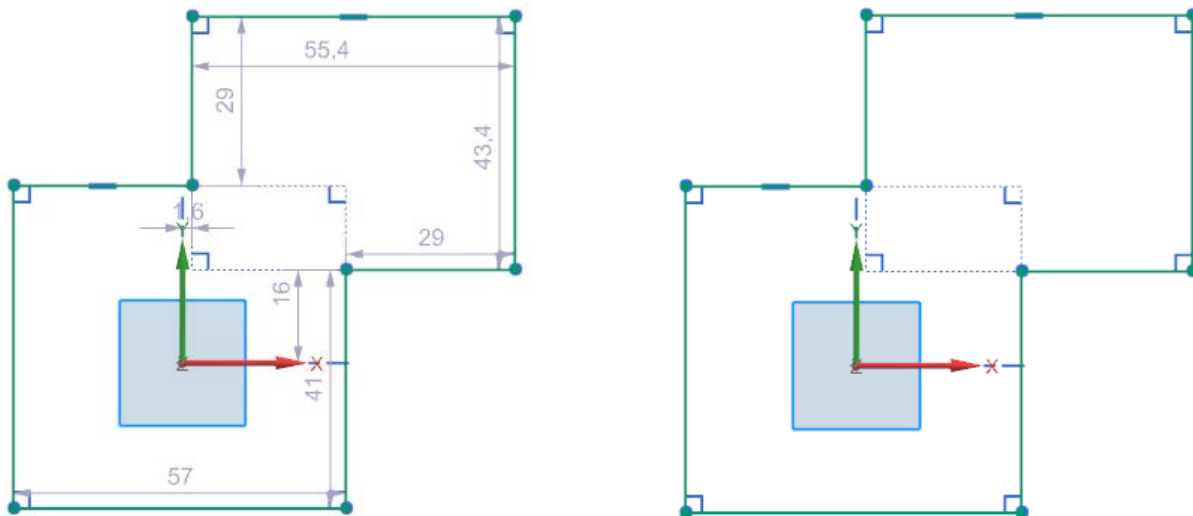

The dimensions placed when **Continuous Auto Dimensioning** is enabled are "weak" or *driven* dimensions, rather than *driving* dimensions – they can be overwritten by other constraints and dimensions and will not conflict with those other constraints. However, they are sometimes more difficult to edit, and tend to give sketches a cluttered appearance.

From within the **Sketch Task Environment**, you can disable **Continuous Auto Dimensioning** by toggling the switch at the bottom of the **Constraint Tools** drop-down menu on the far right of the **Constraints** group. Note that this will not affect dimensions already placed within the active sketch, and will not disable Continuous Auto Dimensioning for the next sketch you create. You can Toggle Continuous Auto Dimensions on or off from the More Menu.
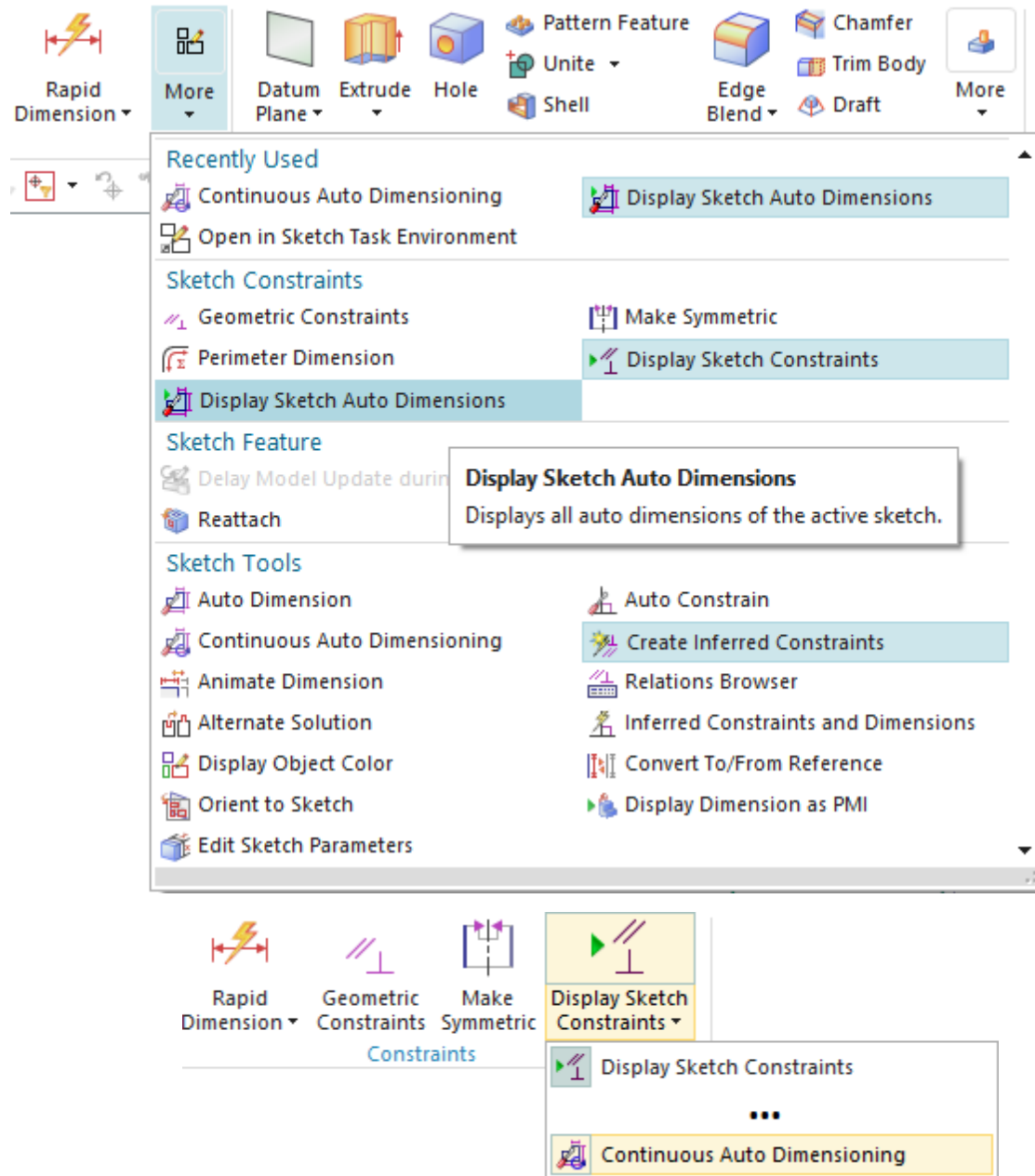
You enable or disable **Continuous Auto Dimensioning** for all future sketches on the **Sketch Preferences** menu, found at **File / Preferences / Sketch**. At this time, please UNCHECK the **Continuous Auto Dimensioning** checkbox.
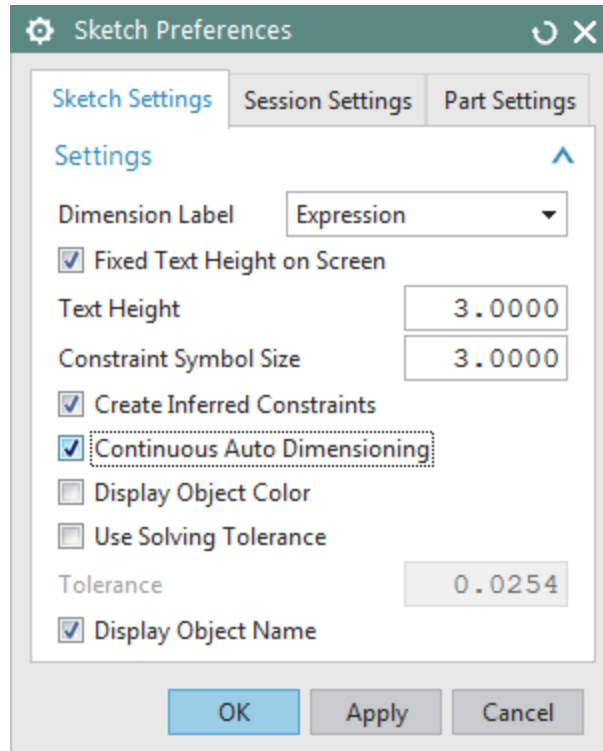
Once you are comfortable with the software, you can decide whether to use it or not, but while you are learning, we recommend that you DISABLE **Continuous Auto Dimensioning** in the **Sketch Preferences** menu.

1.1.2 Sketch Strategy

Before beginning a sketch, you will be better equipped to produce it efficiently if your sketching strategy is sound!

1. If you are working in a part file that contains no solid geometry, NX 11 will automatically create a datum CSYS that defines the sketch plane. By default, a sketch is created on the XC-YC plane of the WCS (Working Coordinate System). The datum axes are infinitely long lines that are coincident with the X and Y axes. The datum entities are displayed as shown below.
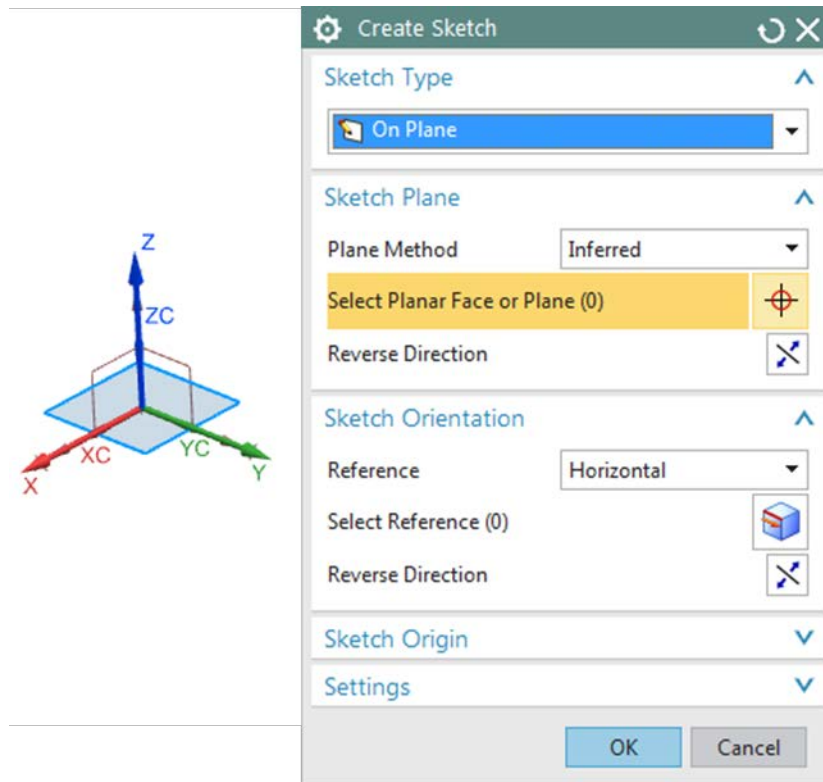
2. Orient your sketch the way you want it. When the **Create Sketch** dialog box is open and your cursor hovers over a face, the red-blue-green coordinate system that appears gives a preview of the sketch coordinate system's location and orientation. For example, clicking on the upper right corner of the face shown below will orient the sketch coordinate system as indicated.
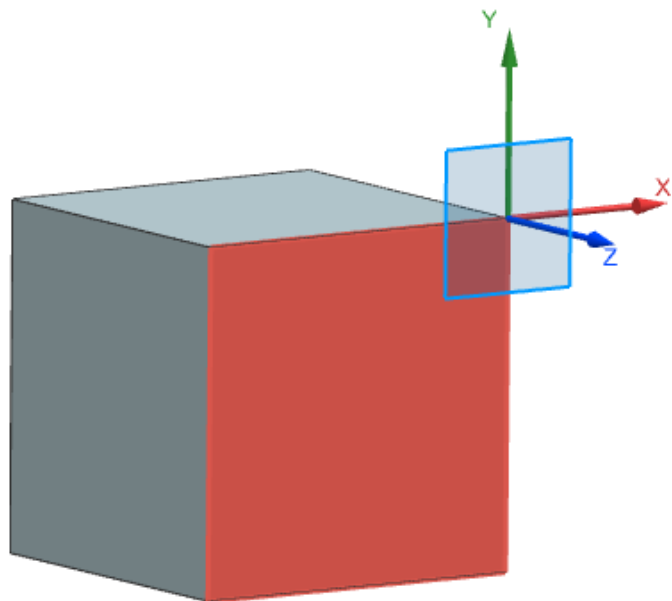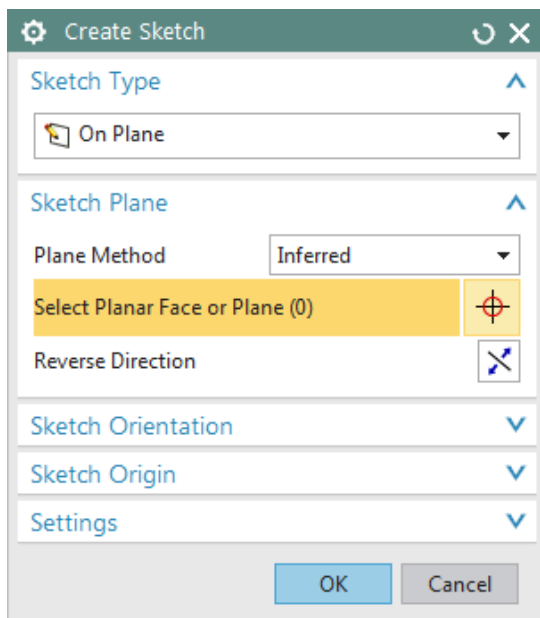

If you prefer your coordinate system in the bottom left corner, you would place your cursor near the bottom left corner before clicking, and the sketch coordinate system would then be placed as indicated below.
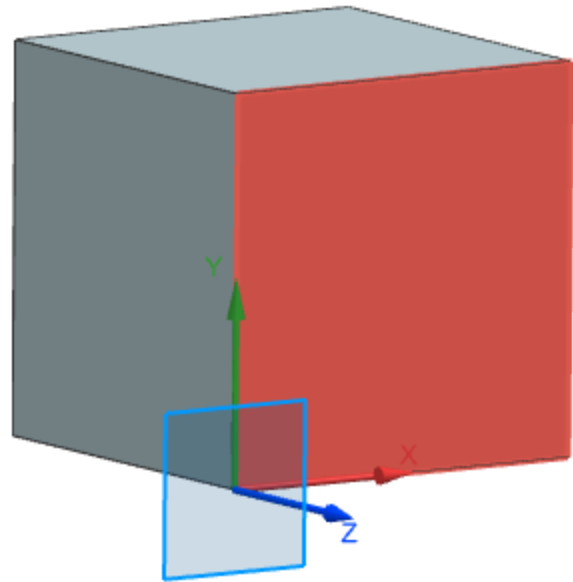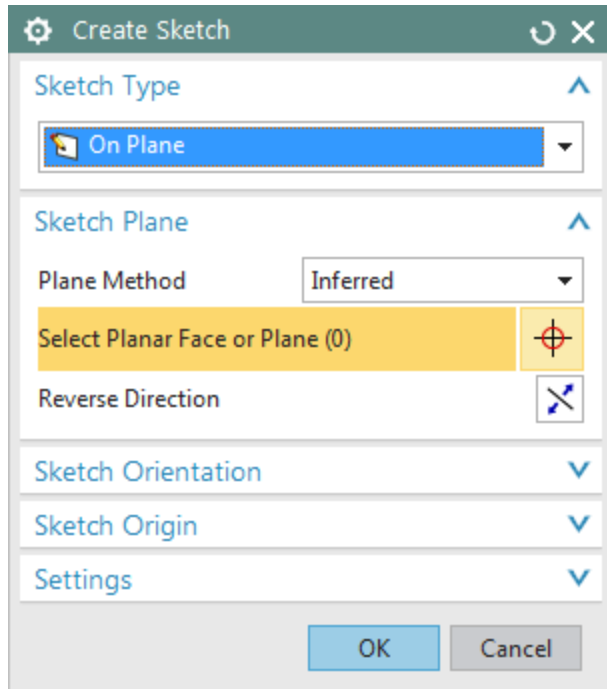
3. Imagine that you want to sketch a pitcher. The figure below illustrates the initial steps of the sketching process. The datum entities and the shape of the desired geometry have been created in three separate sketches. Make a sketch of the contour that resembles the final version. You may want to add a dimension or two at this step, but in general, dimensioning everything each step of the way will slow you down.
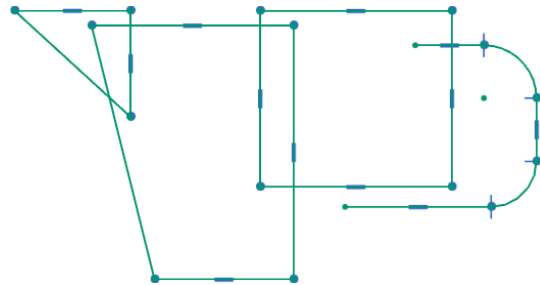
4. Once the basic shape is sketched, add the exact dimensions and constraints. In the next figure, you can see that the pitcher has been sized and all the geometry is ready to create the solid model. It is not necessary to fully constrain your sketch! The sketch below is six constraints away from being fully constrained.
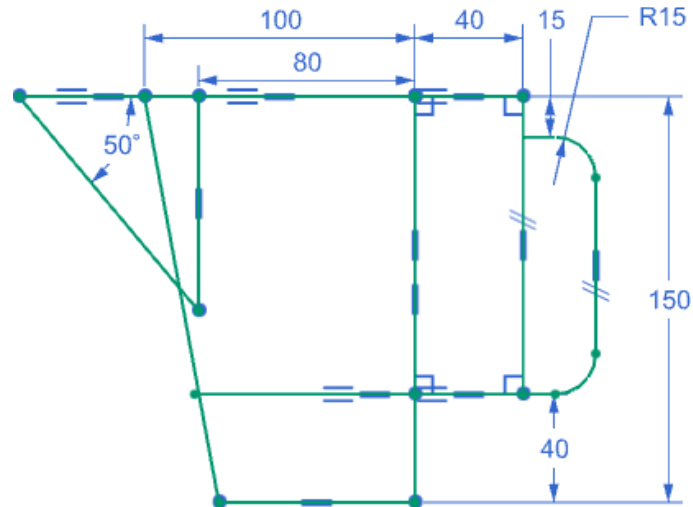
5. The final step in the sketching process is to choose the **Finish Sketch** icon located in the upper left corner of the Graphics Window when you are in sketch mode. This tells NX that you are finished sketching and are ready for bigger and better things!
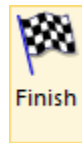

1.1.3   Dynamic Input Boxes

Dynamic input boxes are used to quickly view and modify input data related to a current action or feature. The input fields display labeled values you can change by clicking in the value field and typing a new value.
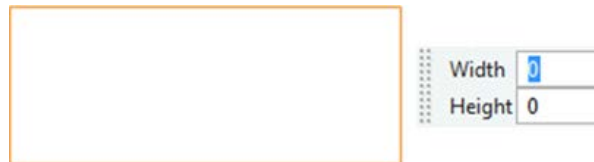
- To use the dynamic input box associated with a tool, select the tool, then click the first point, once the input box appears, you can enter values into it and press **[Enter]** to lock the size of the resulting sketch object.
- For certain tools (e.g., **Rectangle**), once you have entered values, an additional click may be required to determine the orientation. For others, entering the value into the dynamic input box will create the sketch object without an additional click.

1.1.4   Curve Selection

CONTROL POINTS: Every curve has a set of control points that are associated with it, as shown above. Lines have control points at their ends and at the midpoint. Arcs have control points at their ends, midpoint, and center. Circles have a control point at the center and at a 'start' point. In the **QuickPick** menu, you can see a variety of possible control points available for selection. Make a note of their icons!
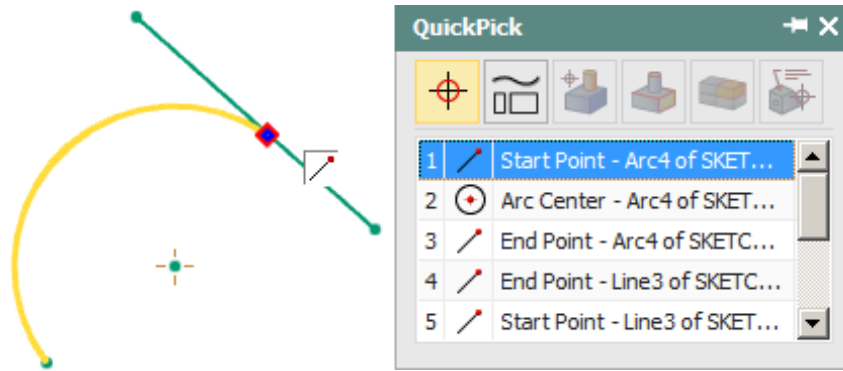
Control points are useful for a variety of tasks, such as building new geometry or making measurements of existing geometry. NX 11 is sensitive to how you pick curve entities and may provide different results based on whether you select the curve itself or its control points.

Important Note: When this book asks you to select a curve, pick anywhere on the curve, but **not** at the control points. When you are asked to select a center point, an end point, or a middle point, you should pick on the control point itself, not the curve. In NX 11, a control point is selected if it is inside the selection ball when you pick. You can also set your snap points to control what you pick as shown below.

Curves can be defined in many ways. They can be placed at screen picks, at *X, Y, Z* coordinates, or they can be connected to another curve at control points. They can also be placed by relations to other curves, such as parallel, perpendicular, tangent, or normal.

## 1.2   Elementary Sketch Curves
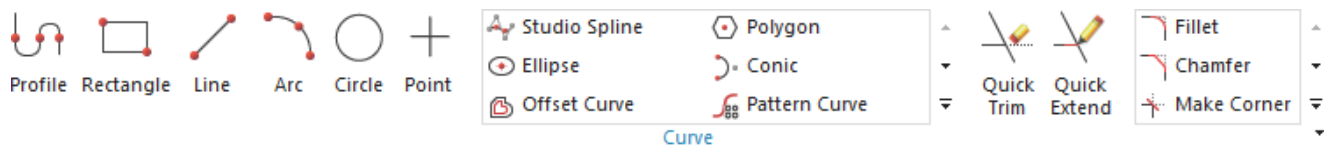
Let's get started by learning how to use some of the basic curve tools available in the sketch environment. Create a new part file and select the **Sketch in Task Environment** tool. Create a sketch on the X-Y plane.

1.2.1   Line

The **Line** tool produces a line segment with just two clicks.

- Create a new part file and select the **Sketch in Task Environment** tool. Select the **Line** tool. Click once to determine the start point of the line.
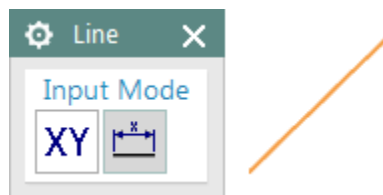

- Click again to finish the line. It's as easy as that!

1.2.2 Arc

The **Arc** tool has two methods for creating arcs – **Arc by 3 Points**, and **Arc by Center and Endpoints**.

- **Pan** to a different part of the sketch plane, away from the line you created in the last exercise.
- Select the **Arc** tool and set the **Arc Method** to **Arc by 3 Points**. Click once to specify the start point of the arc and move the cursor to the right.
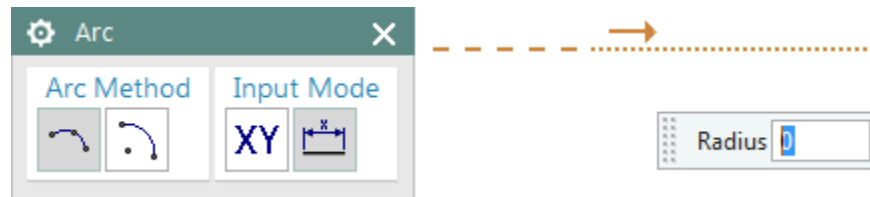

- Click again to specify the end point of the arc, and note that as you move the cursor, the **Radius** varies in the dynamic input box.
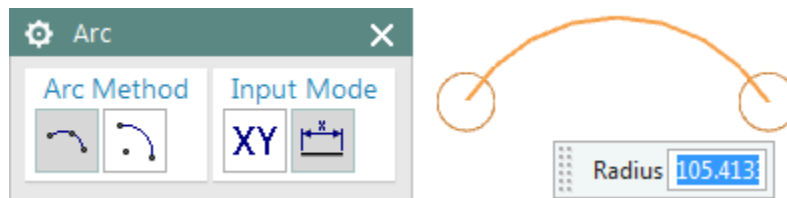

- Click a third time once you are pleased with the radius and the arc will be created.
- Practice making another arc using the **Arc by 3 Points** method, and this time specify the radius with the dynamic input box.

- **Pan** to a different part of the sketch plane, and change **the Arc Method** to **Arc by Center and Endpoints**. Click once to specify the arc center, and then again to specify the start point of the arc.
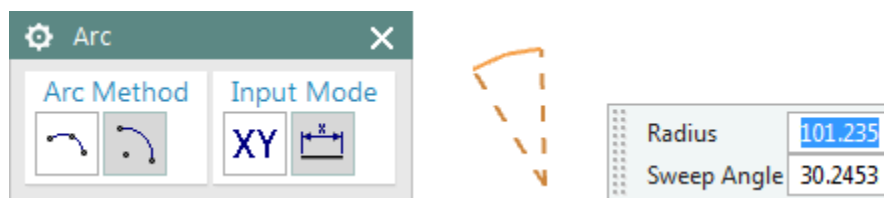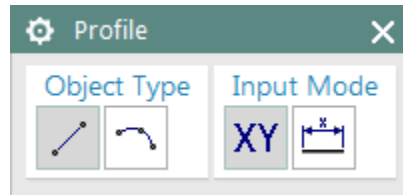

- As you move the cursor, the **Radius** and **Sweep Angle** will vary dynamically, and your third click will determine the arc end point.
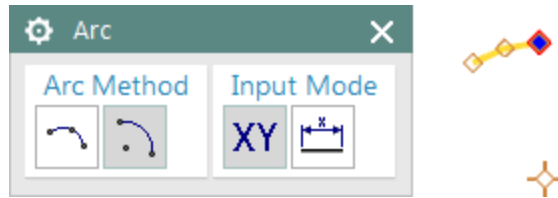
1.2.3   Profile

You will notice, that upon entering the sketch environment, the ribbon shows the **Profile** tool highlighted, and the **Profile** tool dialog box is open in the graphics window.


The **Profile** tool combines the essential functionality of the **Line** tool with that of the **Arc** tool. You can choose whether you would like a line segment or an arc by clicking on the corresponding icon in the **Object Type** section of the Profile dialog box, or you can switch back and forth between arcs and lines more fluidly as follows:

- A single click initiates a line segment. To start an arc, click and hold and then drag before releasing and clicking again. Practice this step by step as follows. Click once to start a line segment and then move the cursor vertically.
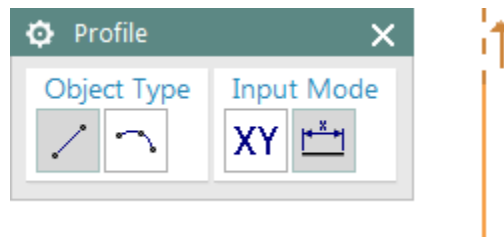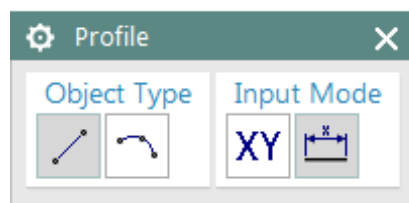

- Click again to end the line segment. With the cursor at the endpoint of the newly-created line segment, click and hold the left mouse button and drag the cursor – first vertically, then slowly to the right. You will notice that the arc comes out tangent to the line segment. Also note that the **Arc** icon is highlighted in the **Object Type** section of the **Profile** dialog box.
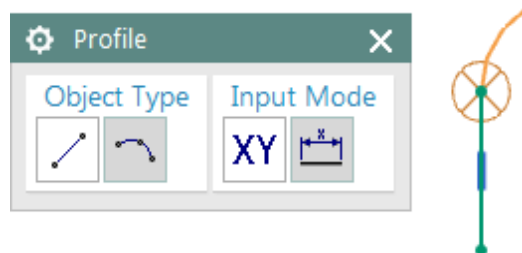

- Move the cursor further right and eventually down until the arc is a perfect half-circle. You will know when to stop with the help of the horizontal guide line.
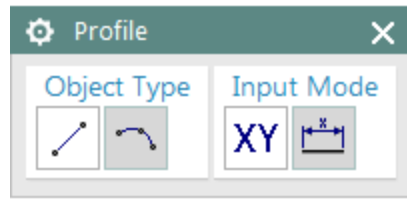
- Click again to end the arc. Note that the **Profile Object Type** immediately switches back to **Line**. Move your cursor down vertically until the dotted horizontal guide line appears to indicate that it has the same length as the first line segment.


- Click to end the line segment. Let your cursor remain at the end point, and again click and hold the left mouse button. This time, move the cursor to the right, and then move it slightly down. Notice that the arc created in this way is perpendicular to the last line segment!
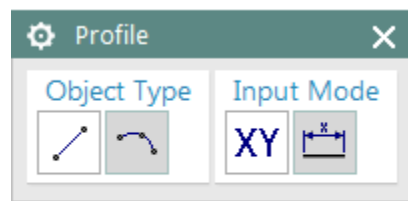

- This is an important point about creating arcs with the **Profile** tool – the arcs created by the **Profile** tool in this fashion are always either tangent or perpendicular to the previous object. The direction in which you move the cursor away from the last endpoint determines whether the arc will be tangent or perpendicular. If you make a mistake, you can always move the cursor back to the endpoint and sweep it out carefully in the desired direction. Move the cursor back to the start point of the arc, and this time, move the cursor straight down before moving it slightly to the right. Sweep it out until the arc is a perfect half circle, and click to end it. Move the cursor vertically and click to create another line segment of equal length to the last.
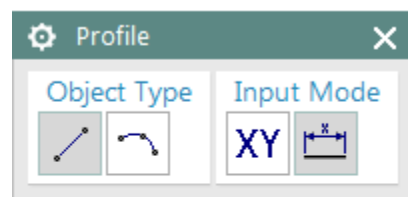
- Continue in this fashion until you've got the hang of alternating between line segments and arcs!
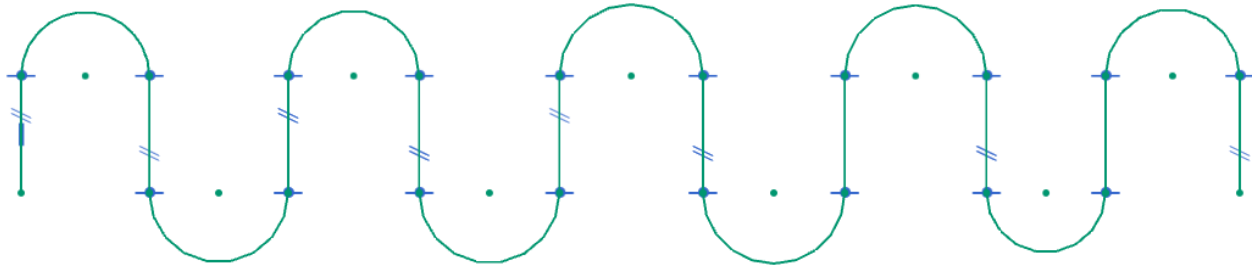
- Once you are satisfied with your progress, you can exit the **Profile** tool by clicking the middle mouse button twice, or by pressing **[Esc]**.
- **Pan** to a different (empty) part of the sketch plane, and select the **Profile** tool again. Now, let's sketch a contour beginning with an **Arc**. You can of course click the **Arc** icon under **Object Type** in the **Profile** dialog box, but you can also switch to **Arc** mode with a click-and-drag technique. Simply left-click and drag a short distance in the empty space in the graphics window, and you will note that the **Object Type** switches to **Arc** mode! To create your first arc will require three subsequent clicks. This works exactly in the same way as the **Arc by 3 Points** method of the **Arc** tool, as you practiced earlier.
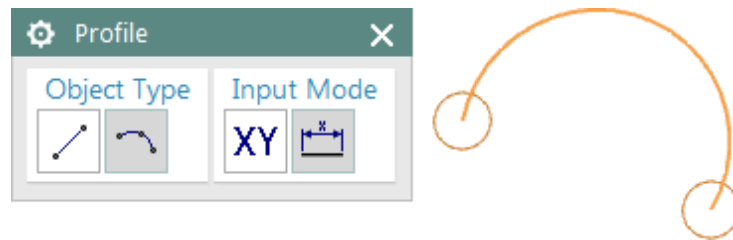

1.2.4 Circle

The **Circle** sketch tool has two methods for creating circles – **Circle by Center and Diameter**, and **Circle by 3 Points**.
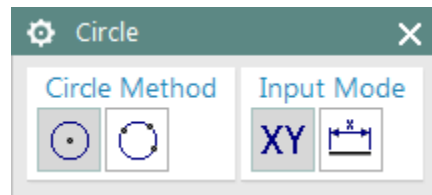
- **Circle by Center and Diameter** requires two clicks – the first click indicates the center, and the second click specifies a point on the circle.
- **Circle by 3 Points** requires three clicks – the behavior is similar to the **Arc by 3 Points** method in the **Arc** tool.
- **Practice both modes. Get comfortable using them with clicks alone, as well as with the help of the dynamic input boxes**!
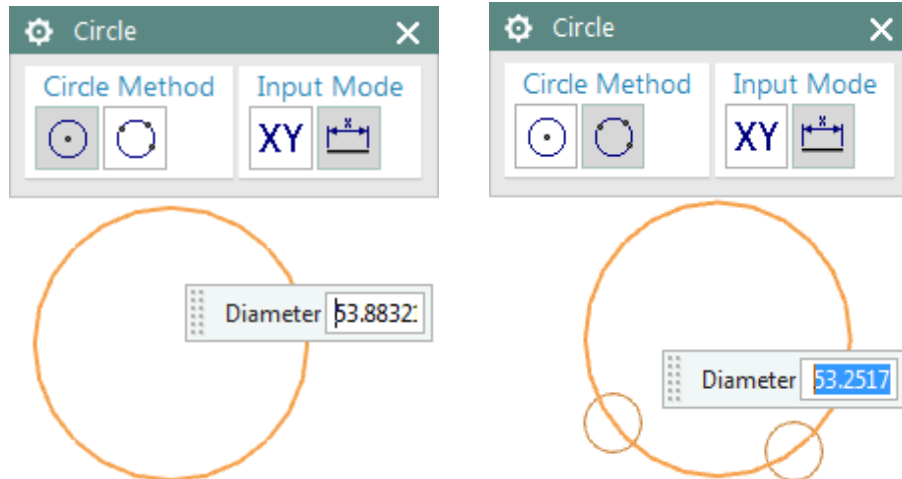
1.2.5 Point

The Point tool is completely self-explanatory. Select the tool and click anywhere in the graphics window to place a point on the sketch plane!

1.3 Geometric Constraints

Constraints are a set of rules that sketch geometry must follow, such as forcing the radii of all circles to be equal or making two lines parallel. Constraints are applied using the **Geometric Constraints** tool, accessed from the **Constraints** group on the **Home** tab in the **Sketch Task Environment**.
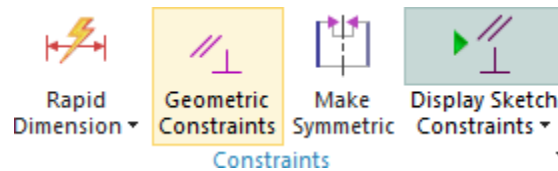

The **Geometric Constraints** dialog box behaves differently from most other dialog boxes in NX – there is no **OK** button and no **Apply** button. You simply select the constraint you wish to apply, then click on the object(s) you wish to constrain. Once you have specified at least one **Object to Constrain**, and one **Object to Constrain to**, the constraint is applied and the menu resets.
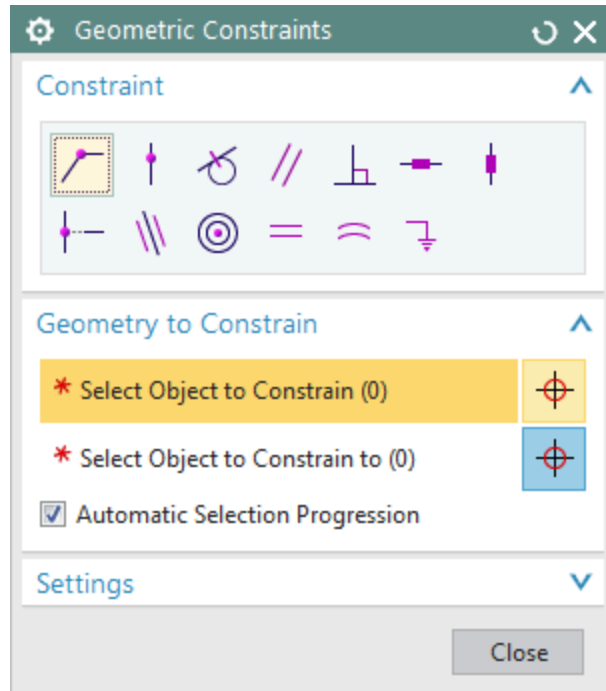
1.3.1 Degree Of Freedom Indicators

When applying dimensions or constraints, you will notice arrows are displayed on some of the vertices of sketched lines, or perhaps at the middle of sketched arcs. These are the *degree of freedom indicators* indicating which entities and vertices have not been fully constrained. As you place more and more dimensions and geometric constraints on a sketch, the degree of freedom indicators disappear one by one. To constrain a sketch fully, you must apply enough dimensions and geometric constraints to remove all of the degree of freedom indicators. You will also need to fix the sketch with respect to previously existing geometry. The figure below shows two lines meeting at a point, and the degree of freedom indicators at that point.
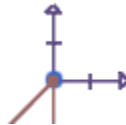
Note: It is not always necessary to fully constrain each sketch, but doing so makes a sketch behave more predictably.

1.3.2 Moving and Deforming Sketches

- Create a new file, select **Sketch in Task Environment**. Place your sketch on the X-Y plane. Using the **Profile** tool, sketch the contour shown below.
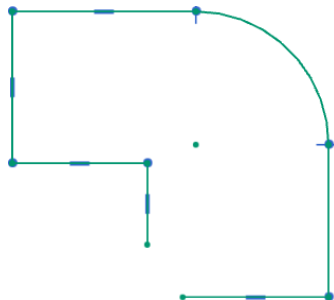
- Note the status display.

**Sketch needs 10 constraints**

- First, we will practice applying a rigid motion to an entire line segment. Left-click and hold on the vertical line segment shown below. Be sure to click on the body of the line itself – away from the endpoints, and away from the constraint icon at the center.
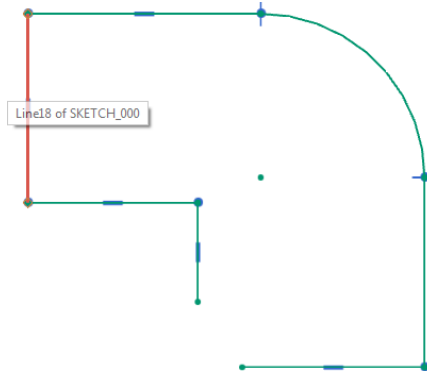

- Continue holding the left-mouse button and move your cursor out to the right. The sketch will deform as shown below. Note that the line retains its length and the vertical constraint no matter how you try to move it.
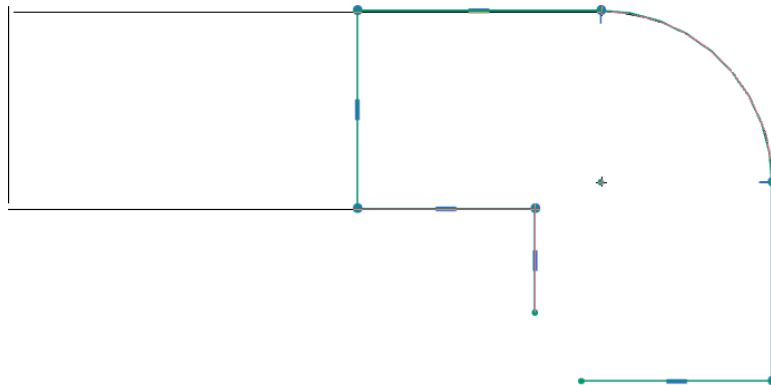
- Press **[Ctrl]+[Z]** to undo the change applied in the last step. Next, you will resize the arc subject to the existing constraints in the sketch. Left-click and hold on the bottom right endpoint of the arc, as shown below.
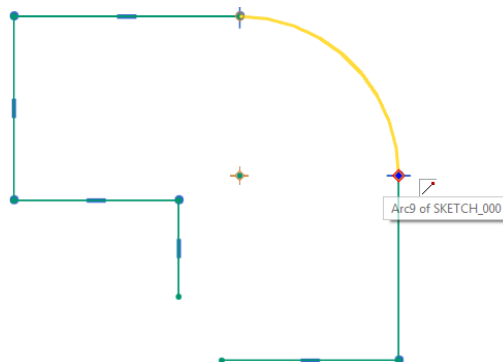
- Continue holding and drag your cursor to the right. Notice that since the bottom right vertical line segment does not have a vertical constraint, as you resize the arc, that line becomes diagonal in the preview.
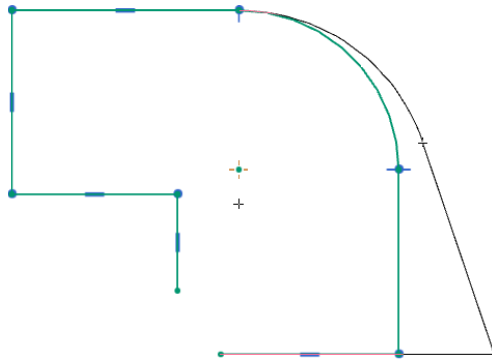
- **[Ctrl]+[Z]** to undo the change.

1.3.3 Fixed

The simplest of sketch constraints is the **Fixed** constraint. It is used to anchor one point or line on the sketch. To ensure that it has a "zero point," the **Fixed** constraint does not fix a point in space – it fixes a point with respect to all other geometry in the sketch. The **Fixed** constraint allows for a single sketch object as input.

- Select the **Geometric Constraints** tool, and select the **Fixed** constraint. Choose the arc center, as shown below.
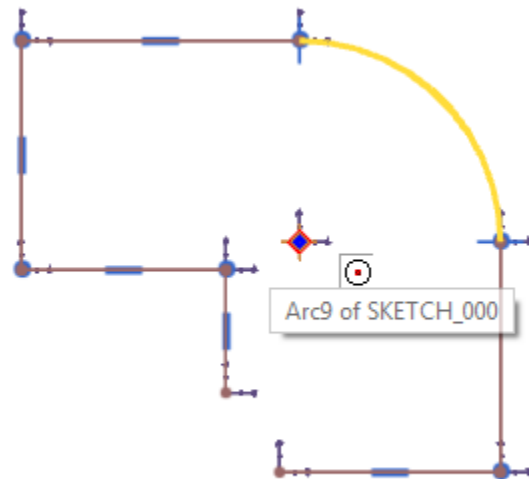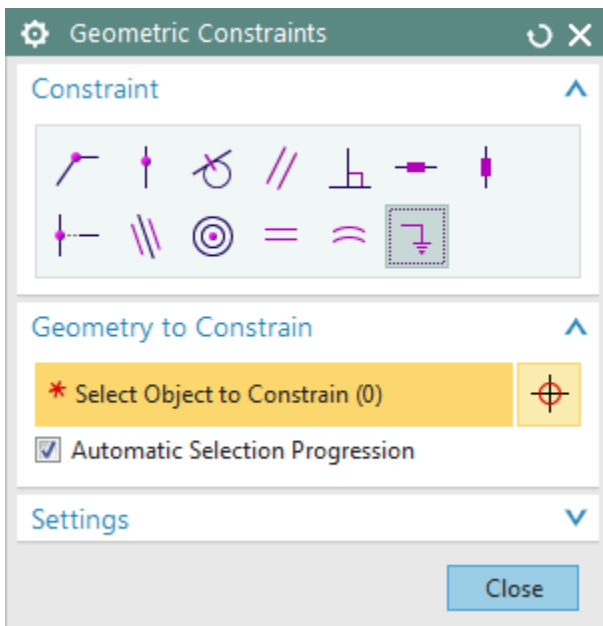
- Note that the degree of freedom indicators at that point have now been replaced by a **Fixed** constraint icon!
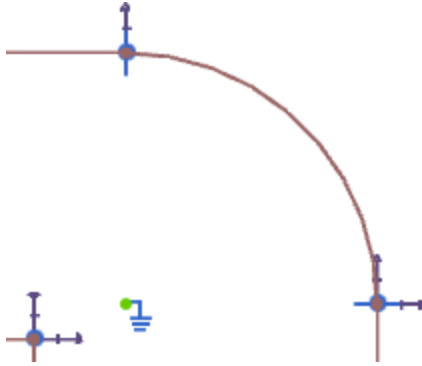
1.3.4 Coincident

The **Coincident** constraint forces two existing points in a sketch to coincide. A typical use case for this constraint is to make the endpoint of one curve coincide with the start point of another. The **Coincident** constraint requires two (or more) points as input.

You will use the **Coincident** constraint to close the gap created in the sketch from the previous exercise.

- Select the **Geometric Constraints** tool and choose the **Coincident** constraint. Select the endpoint of the vertical line segment as the **Object to Constrain**, and the start point of the horizontal line segment as the **Object to Constrain to**, as shown below.
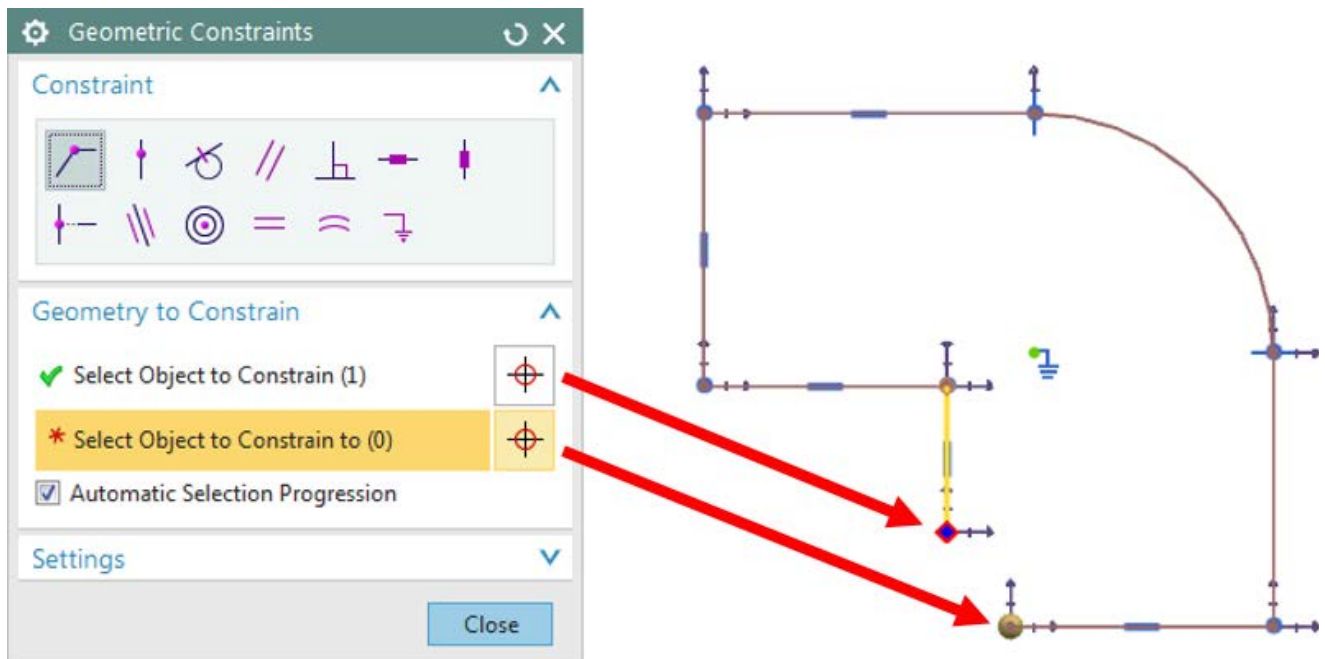
- Once the constraint is applied, the contour forms a closed loop!
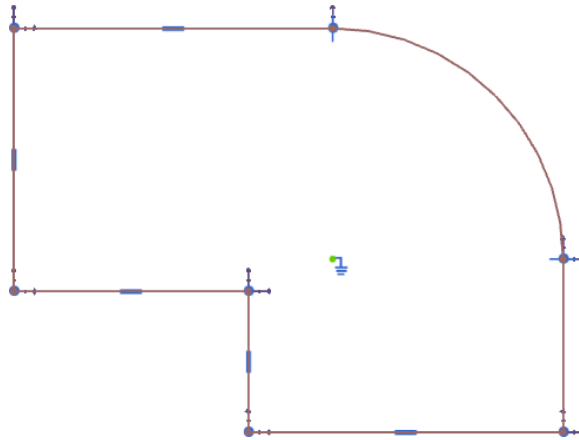
1.3.5 Point on Curve

Point on Curve is a very powerful constraint option. The name is a bit misleading– it should really be thought of as a point constrained to be on an infinite extension of a curve. It may be used to align points, endpoints of curves, or arc center points to lines or other curves. The **Point on Curve** constraint requires a single curve and a single point as input.

- Continuing with the previous sketch, select the **Geometric Constraints** tool and select the **Point on Curve** constraint. Select the horizontal line segment nearest to the arc center as the **Object to Constrain**, and the arc center as the **Object to Constrain to**, as shown below.
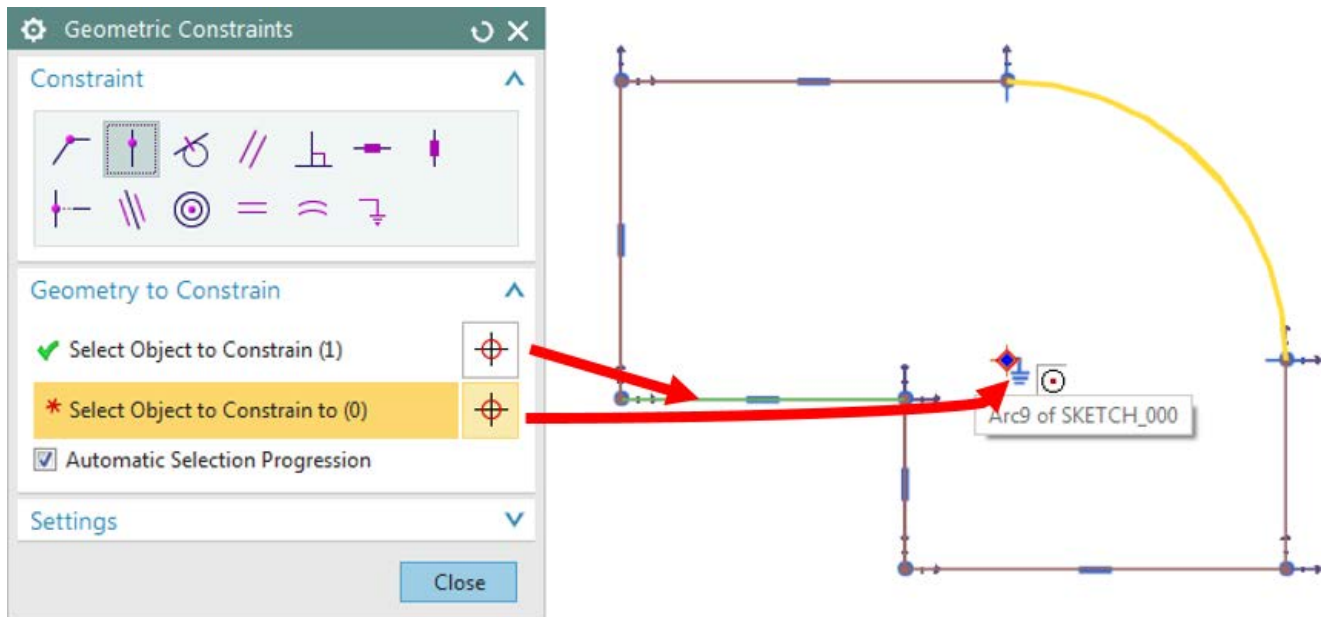
- Once the constraint is applied, a consequence is that the leftmost vertical line segment will have length equal to the radius of the arc on the top right. Try deforming the sketch to see for yourself!
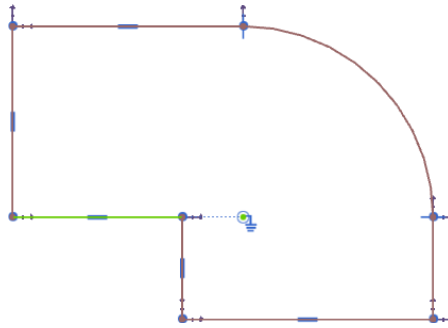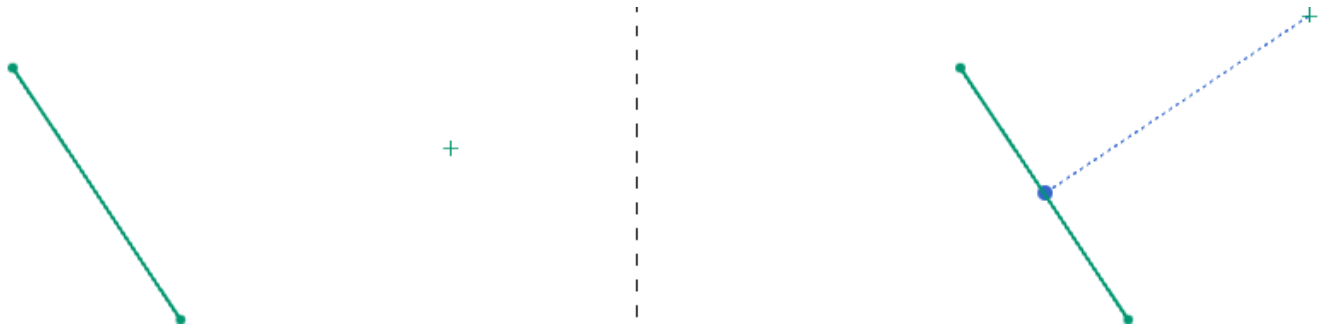
1.3.6   Midpoint

If you are accustomed to other CAD software, you may be surprised by the functionality of the **Midpoint** constraint. The **Midpoint** constraint is extremely useful for centering sketch objects relative to others. The **Midpoint** constraint requires a single line and a single point as input.


The graphic below illustrates the effect of the **Midpoint** constraint. Prior to the application of the constraint (right), the point (B) bears no obvious relation to the line (A). Upon application of the constraint (left), the point moves along a line parallel to (A) until the line segment joining it to the midpoint of (A) is perpendicular to (A).

- Continuing with the previous sketch, select the **Geometric Constraints** tool and choose the **Midpoint** constraint. Select the arc center as the **Object to Constrain**, and the lowest horizontal line as the **Object to Constrain to**, as shown below.
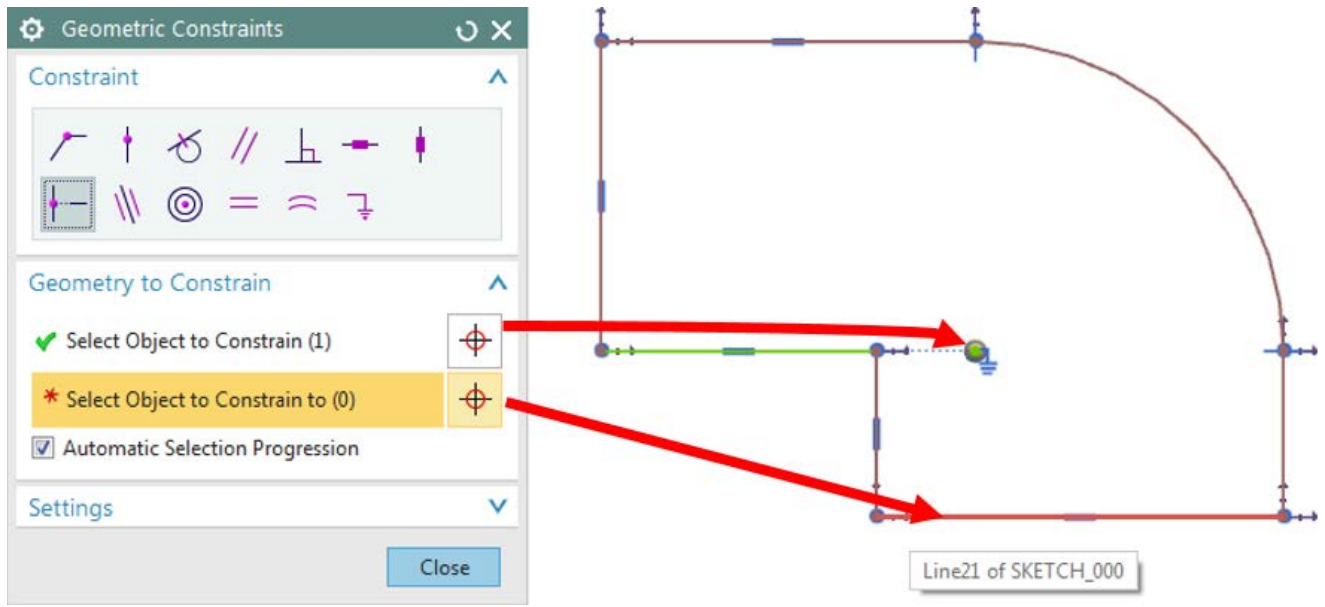
- After applying the constraint, the length of the lowest horizontal line will always necessarily be twice the radius of the arc. Try deforming the sketch to see for yourself!
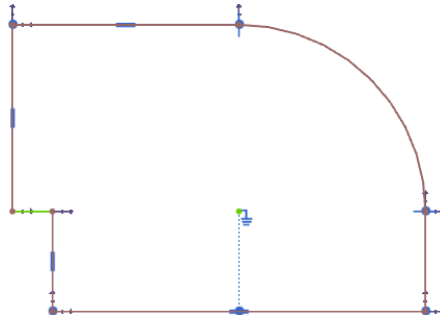
1.3.7 Equal Length

The **Equal Length** constraint is straightforward – it simply forces two lines to have the same length. Note that it only works on lines, and not arcs or any other kind of sketch curves. The **Equal Length** constraint requires (at least) two line segments as input.
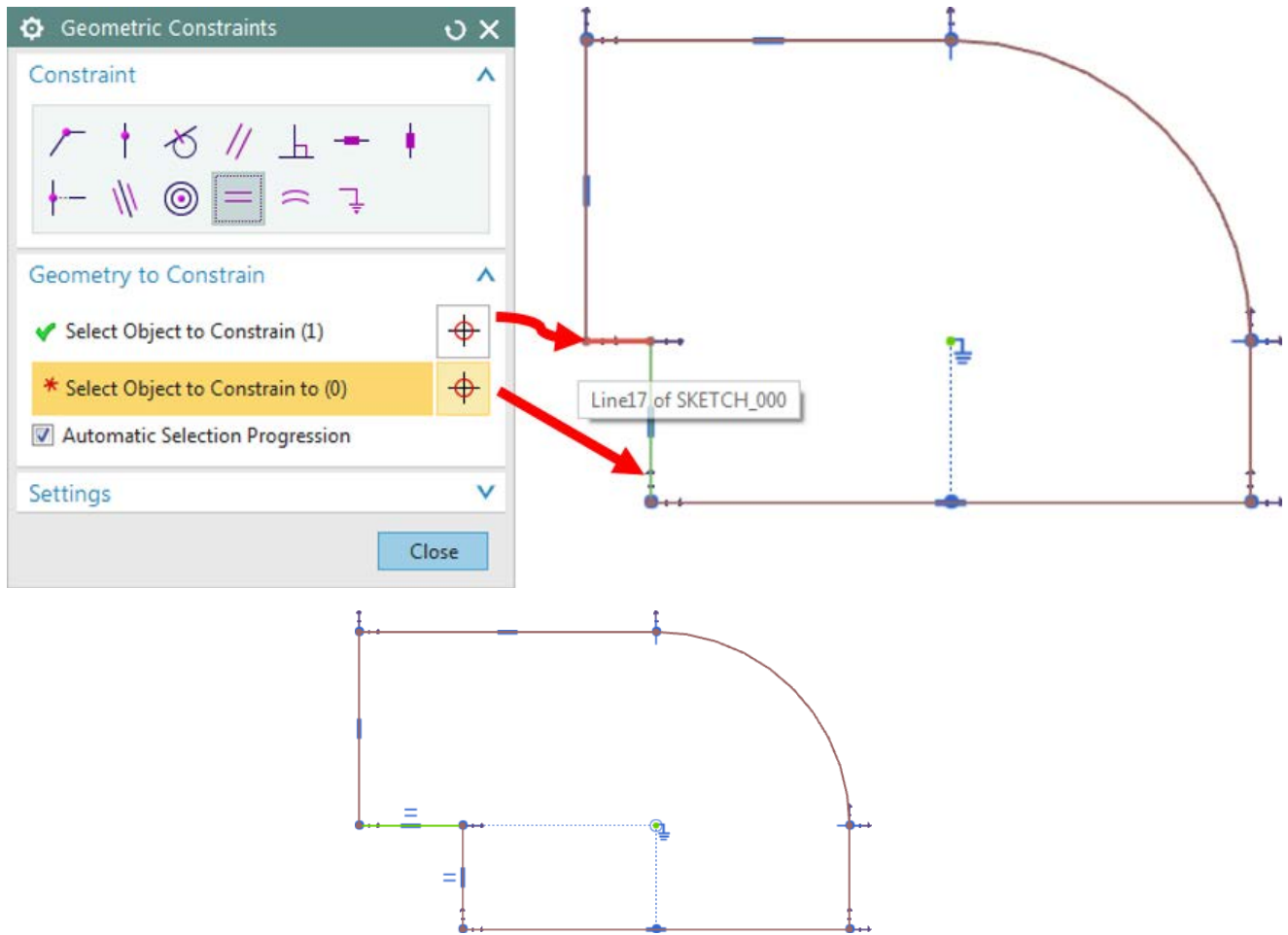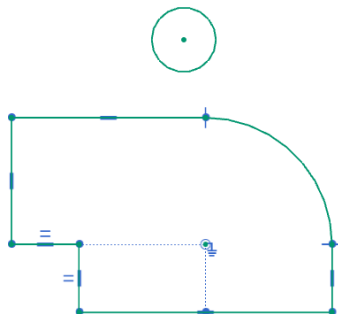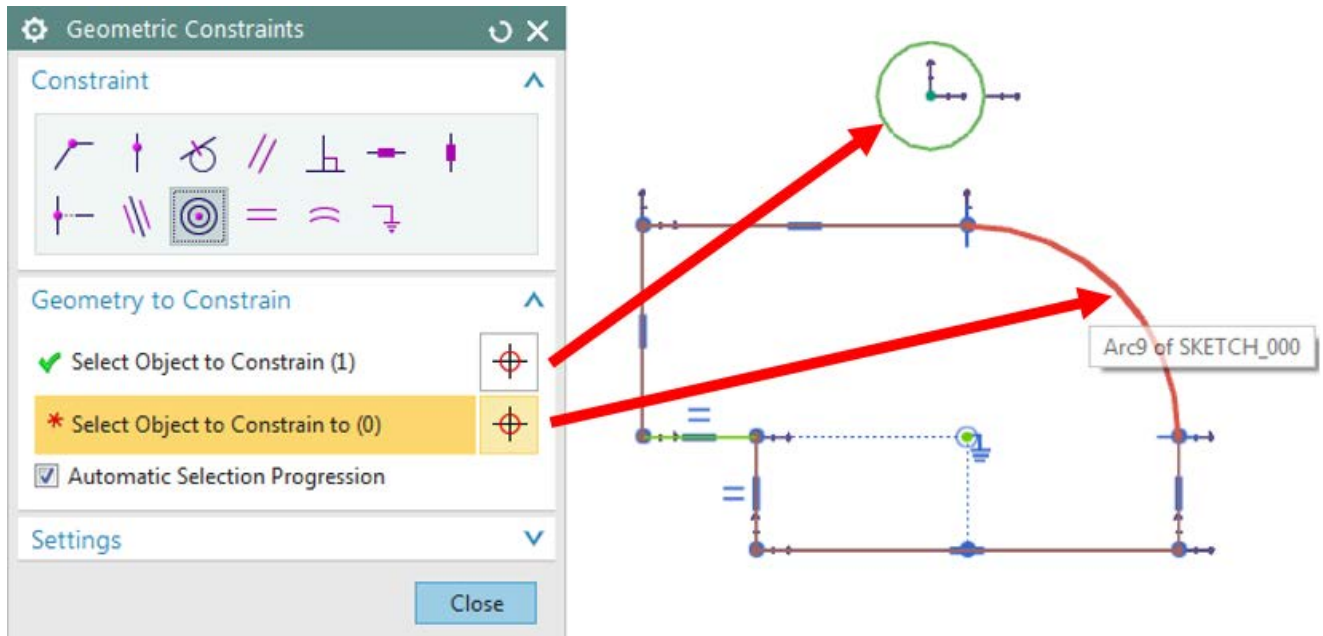
1.3.8 Concentric

The **Concentric** constraint is used to force the arc centers of circles or arcs to coincide – it offers no additional functionality over the **Coincident** constraint. The **Concentric** constraint requires two (or more) arcs or circles as input.
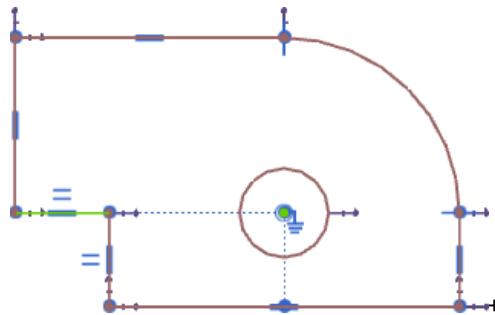

- Continuing with the previous sketch, use the **Circle** tool to add a circle above the contour sketched so far.

- Select the **Geometric Constraints** tool and select the **Concentric** constraint. Choose the newly-sketched circle as the **Object to Constrain**, and the pre-existing arc as the **Object to Constrain to**.

- After applying the constraint, the circle and arc are concentric!

1.3.9   Horizontal and Vertical

The **Horizontal** and **Vertical** constraints are used to force line segments to be parallel to the X and Y axes of the sketch coordinate system, respectively.

When you sketch a line that is nearly horizontal or vertical, NX 11 will often automatically apply a horizontal or vertical geometric constraint to them. You may also place these constraints manually if this does not occur.

For example, when using the **Profile** tool, if you see a horizontal dotted guide line, this is an indication that the software will infer and place a Horizontal constraint on the resulting line segment.


The thickened bar in the middle of the line segment is the **Horizontal** constraint.

1.3.10 Automatic Selection Progression

Automatic Selection Progression advances the menu automatically after you specify the **Object to Constrain**, so that your next click specifies the **Object to Constrain to**. In the next few exercises, we will illustrate some cases in which you might want to disable **Automatic Selection Progression**.

- **Pan** to a different part of the sketch plane, away from the sketch objects you have placed so far, and sketch the contour shown below using the **Profile** tool.
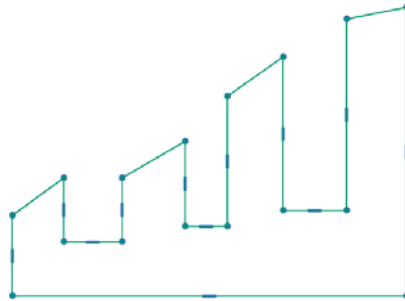
- Select the **Geometric Constraints** tool and uncheck the **Automatic Selection Progression** checkbox. Select the **Equal Length** constraint, and select the two lines indicated below as **Object to Constrain**. Since **Automatic Selection Progression** is disabled, you will have to manually click **Select Object to Constrain to** before selecting the third line.
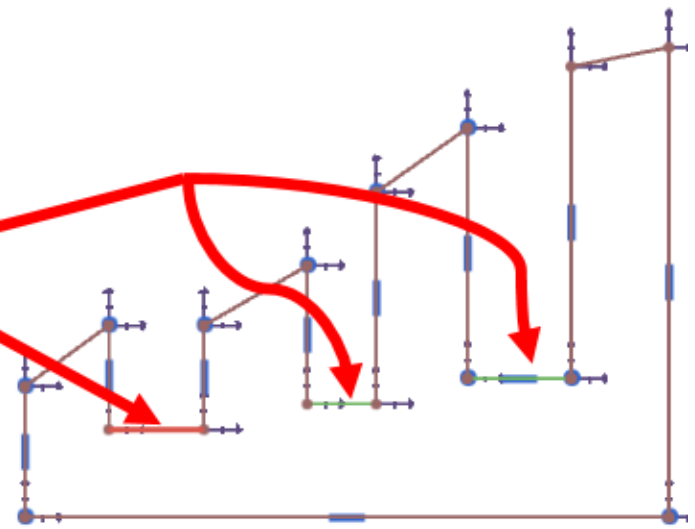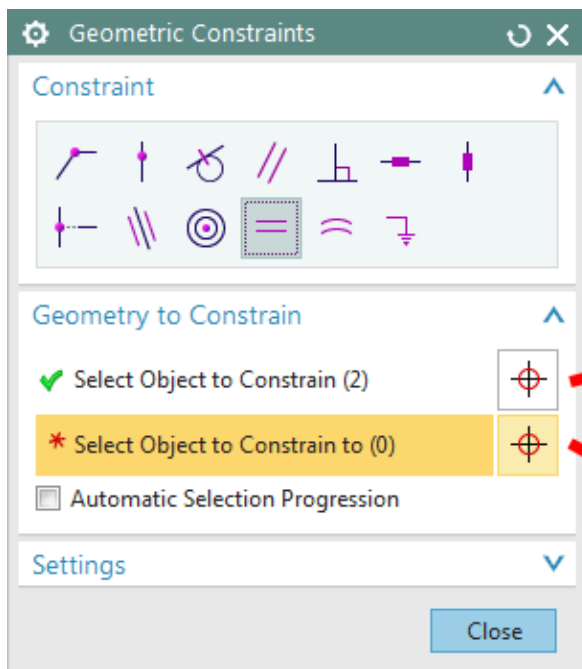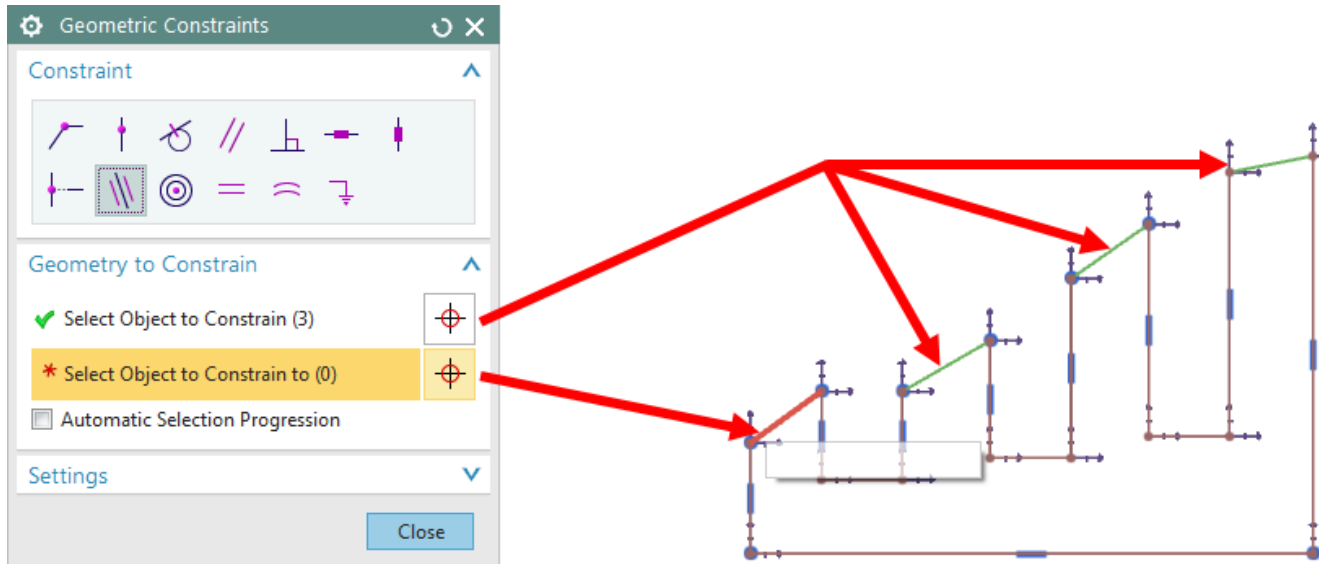
1.3.11 Collinear

The **Collinear** geometric constraint is similar to the **Point on Curve** constraint. The **Collinear** constraint aligns two or more lines with each other, as suggested by the icon. The **Collinear** constraint requires two (or more) lines as input.

- Select **Geometric Constraints** and choose the **Collinear** constraint. Select the three rightmost diagonal line segments as the **Object to Constrain**, and the leftmost as the **Object to Constrain to**.

- Use the **Collinear** constraint to align the three horizontal line segments of equal length. Feel free to either enable or disable **Automatic Selection Progression**. Your sketch should appear as below when you are done.
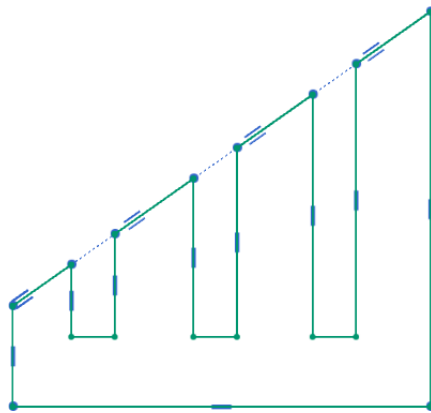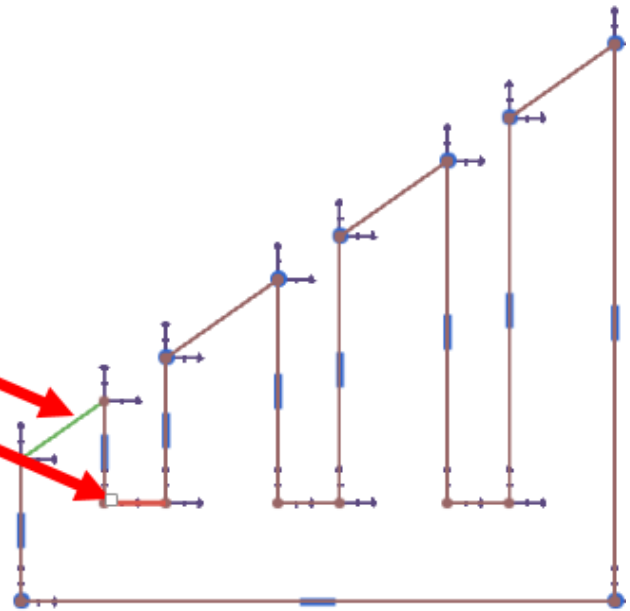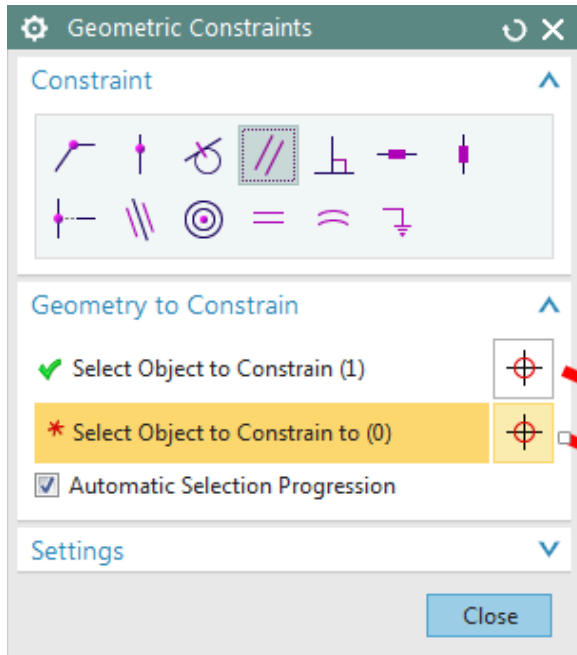

1.3.12 Parallel

The **Parallel** constraint forces two (or more) lines to be parallel.

- Continue with the previous sketch, and select the **Geometric Constraints** tool. Select the **Parallel** constraint. Choose the leftmost diagonal line as the **Object to Constrain**, and the leftmost horizontal line as the **Object to Constrain to**, as shown below.

- The **Horizontal** constraint on the short line segment forces the diagonal line to become horizontal, and the **Collinear** constraint force the other diagonal lines to become horizontal as well!
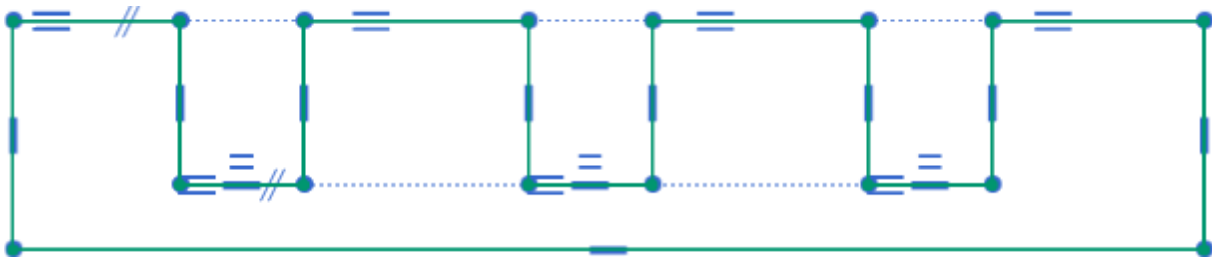

1.3.13 Tangent

The Tangent constraint forces tangency between two curves at a point. The appearance of the constraint in the graphics window is slightly unusual – it is a line passing through the coincident point at a 90° angle to the tangent line!

- Sketch a trapezoidal shape with rounded corners as shown below. Do a really bad job so that there will be plenty to clean up with the tangency constraint! Apply the tangency constraint to each coincident point joining an arc and a line.
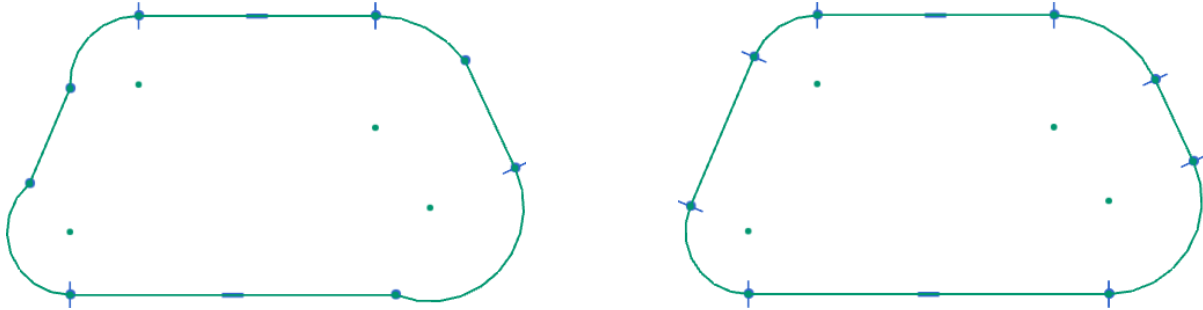
1.3.14 Equal Radius

The **Equal Radius** constraint is similar to the **Equal Length** constraint, but it can only be applied to arcs and circles. The Equal Radius constraint requires two (or more) arcs or circles as input.

Be sure not to confuse the icon when you use the tool – the **Equal Length** constraint will not allow you to select arcs! The appearance of the constraint in the graphics window is an equals sign, just like that of **Equal Length**.
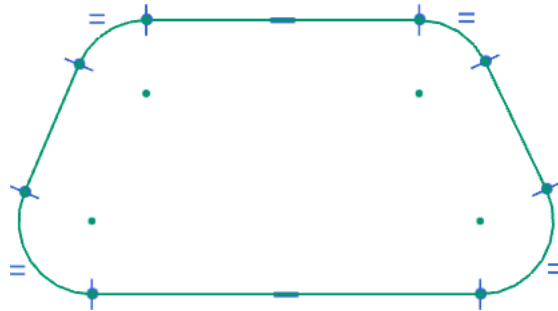
1.3.15 Perpendicular

The **Perpendicular** constraint forces two (or more) line segments to be perpendicular to each other.

- Explicit use of the **Geometric Constraints** tool is not the only way to apply sketch constraints. Consecutively left-clicking on curves and points in a sketch will result in relevant constraints appearing on the pop-up shortcut toolbar. Left-click on the bottom horizontal line segment of the previous sketch, and the right diagonal line. The pop-up shortcut toolbar will then appear with a summary of the constraints that are applicable to those two line segments. Find the **Perpendicular** icon and click to apply the constraint.

Perpendicular
Constrains two curves to be perpendicular.
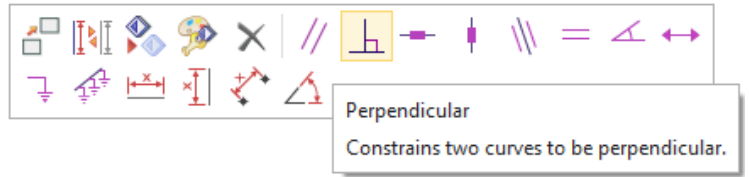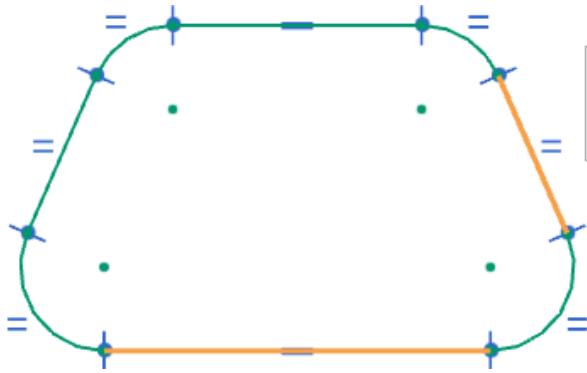
- This technique spares you the hassle of selecting the **Geometric Constraints** tool and finding the appropriate constraint each time you wish to constrain part of a sketch. There is also no issue with Automatic Selection Progression. We strongly recommend mastering this shortcut!