Custom Bridge Parts from Inventor to InfraWorks to Revit

Learning Objectives

- Understand the basics of parametric modeling in Inventor
- Understand the parameters required by InfraWorks for parts to be dynamic
- Learn how to export bridges from InfraWorks to Revit
- Understand the workflow for creating custom bridge parts

Description

Learn hands-on the creation of basic bridge components in Inventor software. InfraWorks software can accept custom piers, girders, abutments, decks, and foundations created as IPT files in Inventor. In this class, you will model a parametric pier in Inventor and use Shape Modeler to prepare it for use in InfraWorks. Once we look at the options in Infraworks, we will export the project to Revit.

Speaker

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Lab Assistants

My sincere thanks to the following people:

Sean Hulbert, Designated Support Specialist, Autodesk
Michael Hurtado, Enterprise Priority Support Specialist, Autodesk
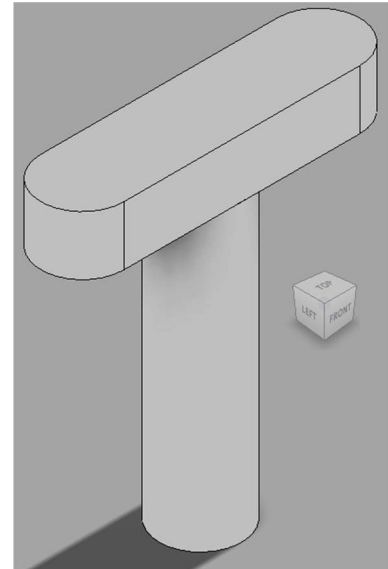Creg Dieziger, SR. Civil Engineer Designer, Morrison-Maierle

Introduction

Goal:

To give designers basic Inventor skills to create a simple bridge pier for use in Infraworks.

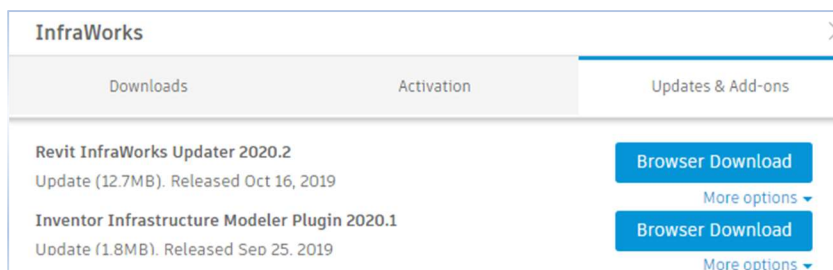Notes:

This is SUPER basic stuff. On purpose. Baby steps.

Software used in this presentation:

- Inventor Professional 2020.1
- Infraworks 2020.1
- Infrastructure Part Shape Utility (ShapeModeler.msi)
- Revit Infraworks Updater (IWExportToRevit.msi)

    The installation order will make a difference here.

    1. Inventor
    2. Infraworks
    3. ShapeModeler.msi
    4. IWExportToRevit.msi

★ Verify that the **Infrastructure Part Shape Utilities** appears in the Environments tab of Inventor.

★ ShapeModeler.msi is no longer a part of the default Infraworks installer and must be downloaded from manage.autodesk.com.
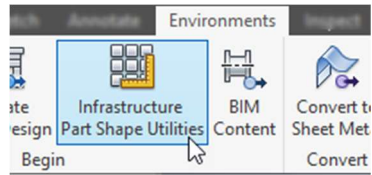
Assumptions

- You are already CAD user and familiar with 3D drafting concepts.
- You are familiar with Autodesk interface terminology, such as 'view cube', 'template', 'parametric' etc.

# The Inventor Environment for Civil People

Inventor is initially set up as if you were going to design a mechanical widget, so the default coordinate system and units are not ideal for a large-scale object like a bridge pier. In this section, we will set up inventor to be more civil-friendly.

Exercise 1: Make Inventor Feel More Like Home

First, change some settings to make working in Inventor feel more familiar.

When you first launch inventor, you are presented with a start screen. Like most Autodesk products, you can jump right in with a default template or pick an existing one.

1.1. Launch Inventor Professional 2020 if it is not already open. Click New Part.
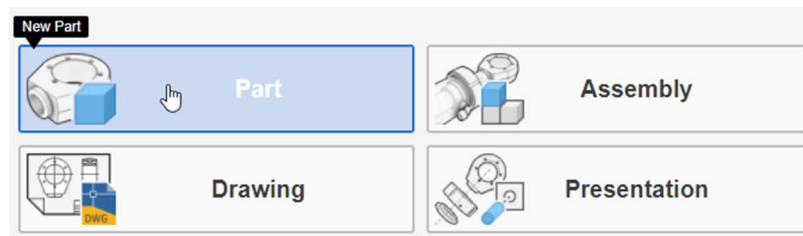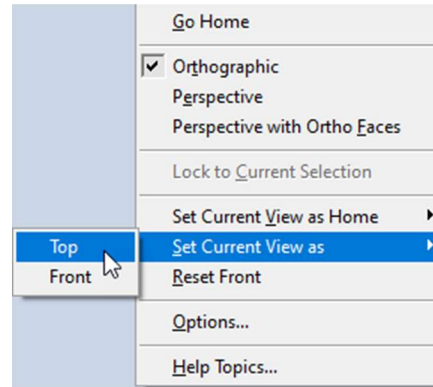

Initially, your drawing screen will show the view cube and the UCS icon. Notice that the XY plane is currently set to Front.

1.2. Click the down arrow near the view cube and select Set Current View to **Top**.

Setting the XY plane to Top will make the drawing environment feel more familiar.

Z is now "up."

Next, we can change the default zoom behavior of Inventor to make it more like AutoCAD. This is optional, but if you are used to AutoCAD, the zoom behavior may drive you crazy.

1.3. Go to the File tab and click **Options**.
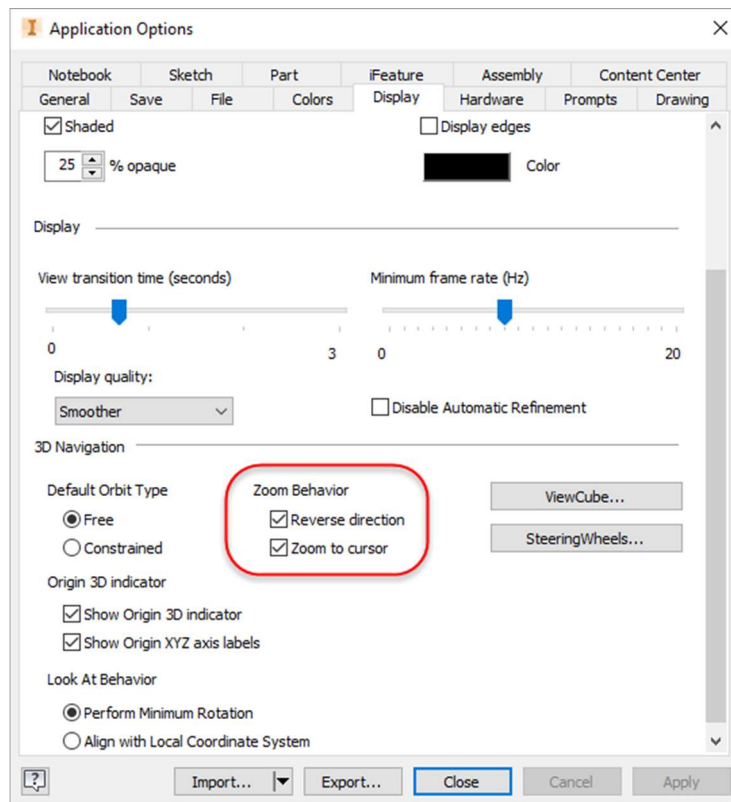
1.4. Switch to the Display tab.

1.5. Scroll down until you see Zoom Behavior.

1.6. Place a checkmark next to Reverse direction.

1.7. Click Close.

This changed setting will persist between Inventor sessions.

1.8. Save the file to your hard drive as `Basic Pier.ipt`.

Exercise 2: Units in Inventor

In this example, I started with the default US template which is in inches. Because we will be working with a large-scale bridge pier, let's change the units of the drawing to feet or meters.
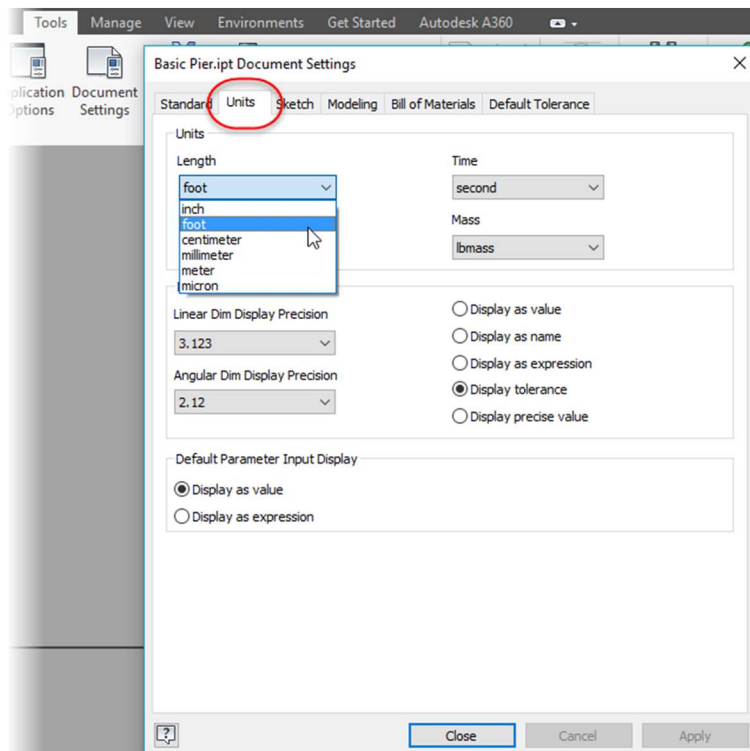
   2.1. Continue working in **Basic Pier.ipt** (created in the previous exercise).

   2.2. Go to the Tools tab > Options Panel and click **Document Settings**.

2.3. Switch to the **Units** tab.

2.4. From the Length dropdown, select **foot**.

2.5. Click Close, then Save the file.

Getting to Know Inventor

In the previous section, Make Inventor Feel More Like Home, you switched your mouse wheel zoom behavior to act more like AutoCAD.
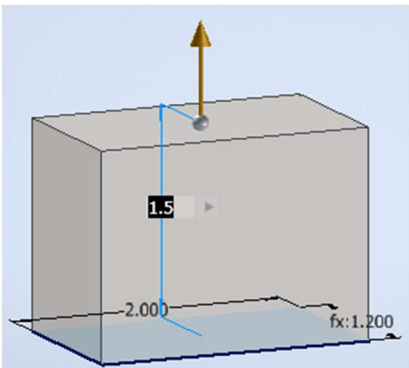
Like AutoCAD, holding shift and the middle mouse wheel will put you into 3D orbit mode.

Inventor Basics

Sketch First; then Extrude
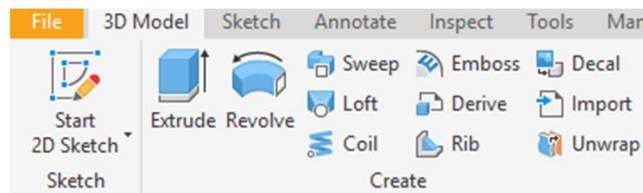
Inventor works by creating a 2D **Sketch** and then operating on the Sketch to become a 3D solid.
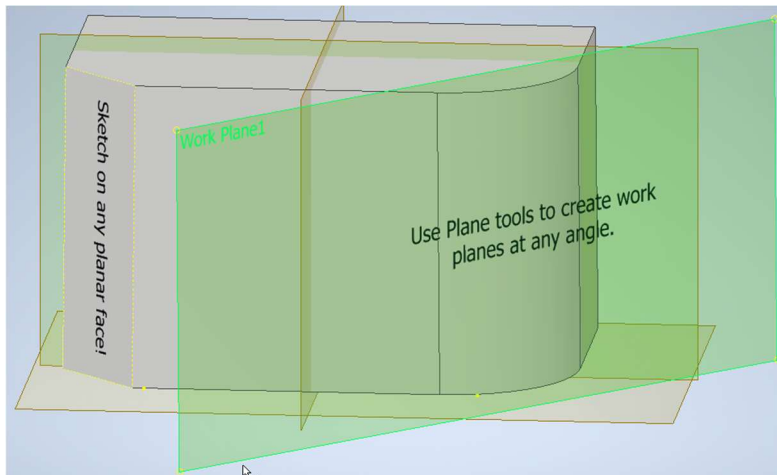

For example, to make a prism, one would first create a Sketch of a rectangle, then extrude the rectangle to the desired height.

Sketches can be created directly on a planar surface or on a work Plane.

Overview of Sketched features in Inventor

Sketches on a Plane

Sketches always need to start on a Work **Plane**. When you first open an IPT file, you will have XY, XZ and ZY planes already to sketch upon.

Once you have geometry in your file, you can create planes that relate to your part in any way.

Learn more about work planes and other work features in Inventor Help.

Snapping & Constraints In Inventor

Object snaps are always running in Inventor. Keep an eye out for the green dot while sketching that indicates you are about to click on significant geometry such as the endpoint or midpoint of geometry.
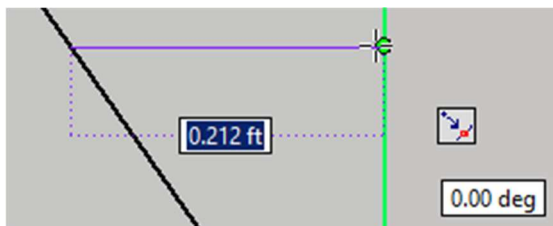
As you use snaps, Inventor will automatically constrain the part. This behavior is on by default and can be controlled in Constraint settings.
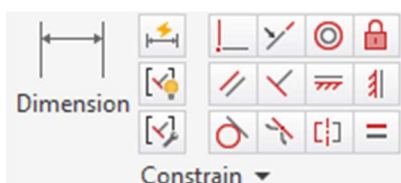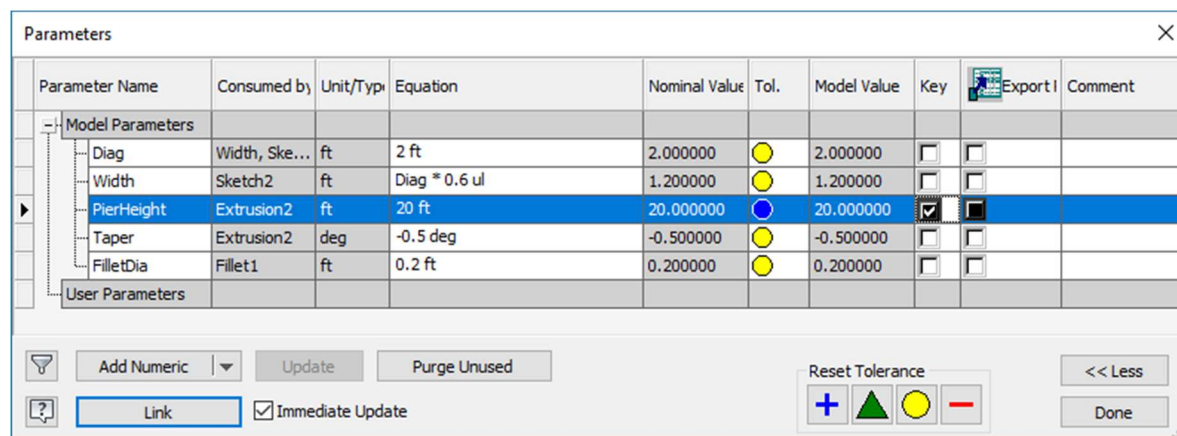

Geometric constraints allow geometry to be dynamic while abiding to design rules. Inventor adds constraints as we draw, but you can add more as needed.

More on geometric constraints in Inventor

Parameters! Parameters! Parameters!

Each bit of geometric data entered into Inventor is tracked as a **_Parameter_**. Model Parameters are created as you sketch and control the measurements of geometry. Unlike AutoCAD, where adding parametric behavior is an additional step, Inventor is always keeping track of how our geometry relates to itself.

You can name the parameters as you Sketch, or you can open the Parameters manager to add names later.

Parameter Name	Consumed by	Unit/Type	Equation	Nominal Value	Tol.	Model Value	Key	Export	Comment
− Model Parameters									
Diag	Width, Ske...	ft	2 ft	2.000000	○	2.000000	☐	☐	
Width	Sketch2	ft	Diag * 0.6 ul	1.200000	○	1.200000	☐	☐	
PierHeight	Extrusion2	ft	20 ft	20.000000	●	20.000000	☑	■	
Taper	Extrusion2	deg	-0.5 deg	-0.500000	○	-0.500000	☐	☐	
FilletDia	Fillet1	ft	0.2 ft	0.200000	○	0.200000	☐	☐	
User Parameters									

Add Numeric ▼   Update   Purge Unused   Reset Tolerance   << Less

Link   ☑ Immediate Update   + ▲ ● −   Done

Understanding parameters is critical when exporting to Infraworks. It is parameters that drive how (and if) Infraworks understands the design intent.

Infraworks uses the parameter name to "hook in" to the part. Any parameter that you would like to view in Infraworks will get a check placed in the Key column of the parameters manager.

Some significant changes in how Infraworks interprets equations and parameters have been recently incorporated into Infraworks 2020.1. Read more about those additions here.

Learn more about parameters in Inventor 2020 here.

Exercise 3: Your First Sketch

A sketch is a 2D drawing which can be used to create 3D geometry. We can turn 2D sketches into 3D objects in multiple ways such as using extrude, loft, revolve and sweep, just to name a few.

You can sketch directly on the face of an object or use a workplane.

Now that you have some basic facts about Inventor under your belt, you can now start sketching.

3.1. Continue working in the previous file. If you did not complete exercises 1 and 2, you can jump in by opening the file `Basic Pier (Blank file w units set to feet).ipt`.
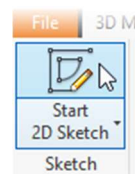
3.2. Switch to the 3D Model tab. In your Basic Pier.ipt file, click **Start 2D Sketch**

3.3. Select the XY plane by clicking it.
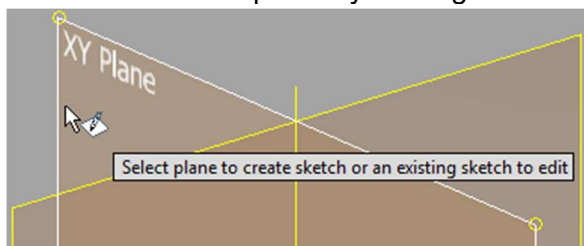
Even if you have the workplane visibility off, you will temporarily see them when beginning a sketch.

Notice that the tool tip is helping you figure out what input Inventor is looking for.

The Inventor interface will change to sketch mode and display tools that are related to creating 2D geometry.

3.3. Click the top portion of the circle tool.

There are several options for creating a circle, this will choose a simple center-radius type circle.
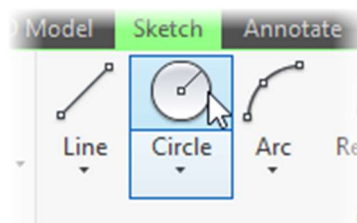
Object snaps are always on in Inventor. The green dot indicates that you will be snapping to the origin.
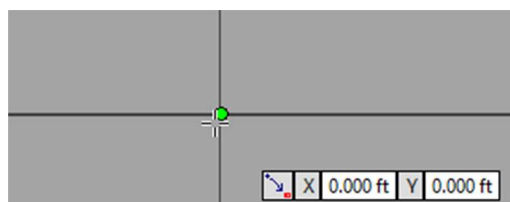
3.4. Move your cursor towards the origin point of the sketch.
Click when you see the green dot.

3.5. Type **`PierDia = 8`** and press enter on your keyboard. This creates an 8' diameter circle.

3.6. Next, click Finish Sketch.
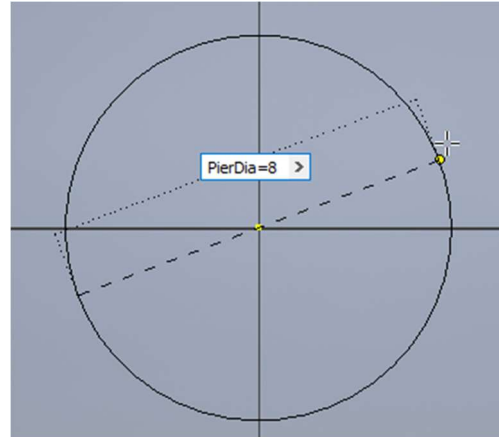
You may need to click Top on the view cube to see the entire circle from the top down.

Hint: Double-click the middle mouse wheel to zoom extents exactly like you would in AutoCAD.

Exercise 4: Extrude the Column

4.1. Continue working in the previous drawing.

4.2. If you were unable to complete the previous exercise but would like to continue, open the file `Basic Pier (Exercise 4).ipt.`

4.3. From the 3d Model tab > Create Panel, click Extrude.
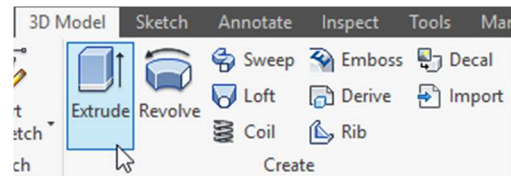
4.4. Set the extrusion height to **`PierHeight=20`**

4.5. Verify that your extrusion settings match what is shown here.

Click OK.
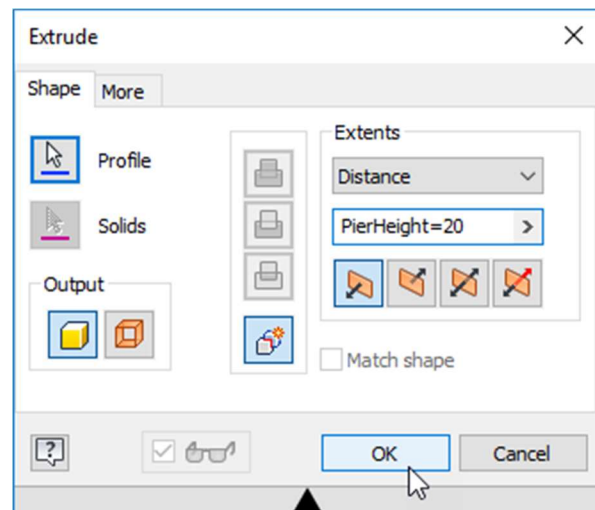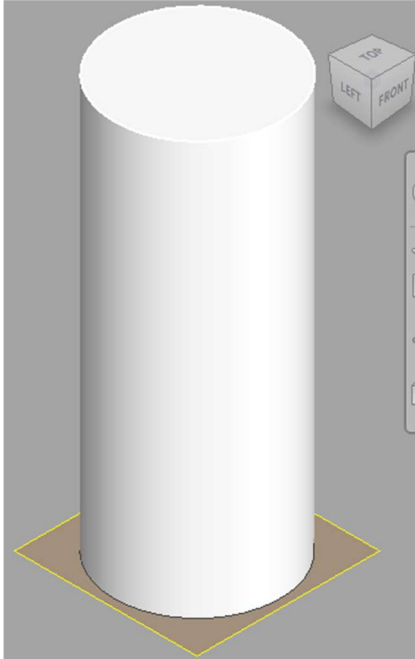
In the next several steps we will manipulate the view to make working with our geometry easier.

4.6. Using the view cube, manipulate your view so you are looking at the column from a SW isometric view.

Be sure the column is positioned upright on your screen.
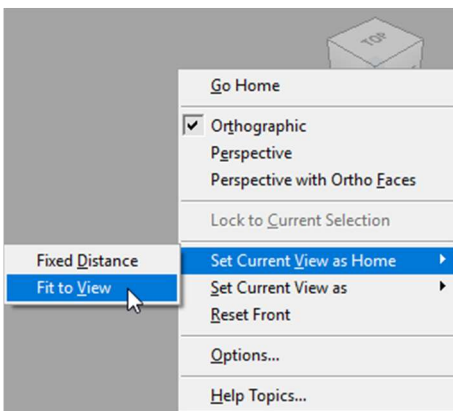
4.7. Right click the viewcube and select Set Current View as Home > Fit to View.

Now, any time you hit the home icon near the view cube you will be taken back to the SW Isometric view. 🏠

Exercise 5: Sketch the Top of Pier

Continue working in the previous drawing.

If you were unable to complete the previous exercise but would like to continue, open the file
`Basic Pier (Exercise 5).ipt`.
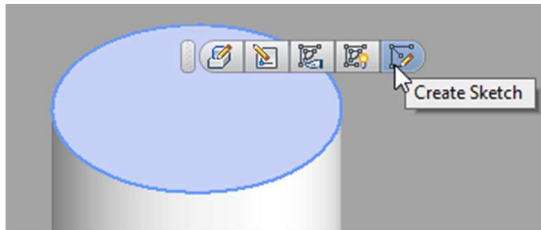
5.1. Select the top face of the cylinder.

5.2. Click Create Sketch from the onscreen tools (or from the ribbon)

Sketch mode will take your dreawing into a 2D drawing environment.

Here is an overview of what we will be drawing in the Inventor sketch.
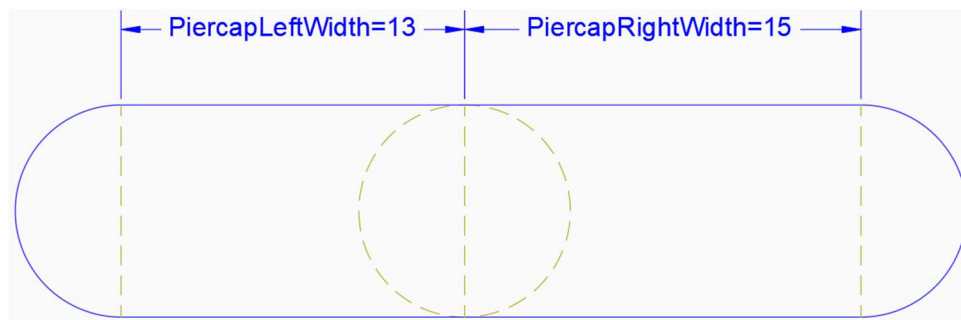
First we will draw the left and right sides of the pier cap as rectangles. We keep each side separate so they can be controlled by the **PiercapLeftWidth** and **PiercapRightWidth** parameters that are needed by Infraworks. We will then convert the vertical lines of the rectangles to construction lines, then finish off the pier with rounded ends.

Inventor behaves much like AutoCAD with polar tracking enabled.

As you draw, notice that Inventor is automatically adding contraints to your geometry.
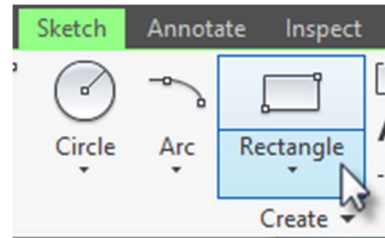
5.3. Click the Rectangle command

5.4. When drawing the rectangle start by snapping to the "12 o'clock" tangent position on the circle. Drag down and to the right until you see the width of the column is 8 feet wide.
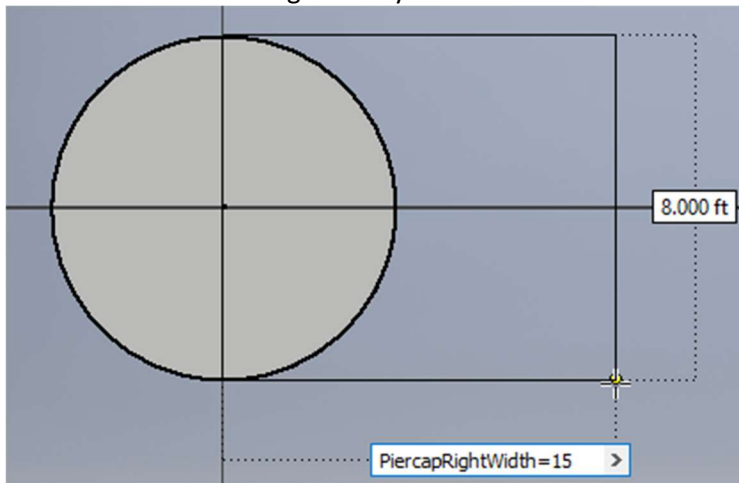
5.5. If your cursor is not active in the correct tooltip, press Tab.

5.6. Type in **PiercapRightWidth=15** and press enter.
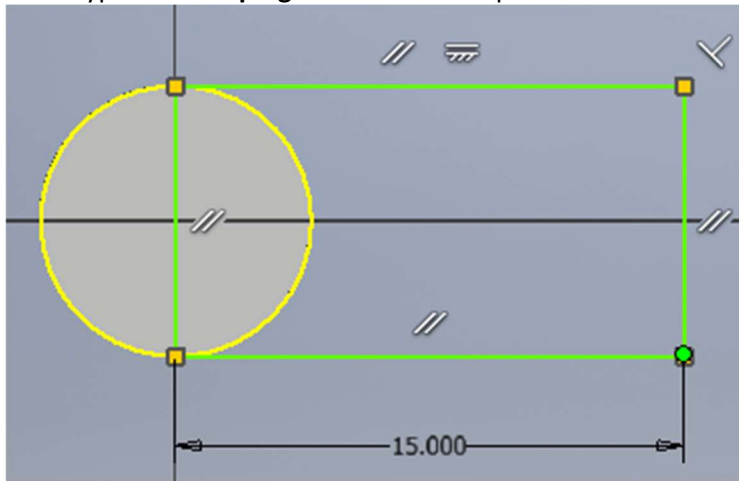
5.7. If you did the previous step correctly, Inventor will constrain the geometry as shown above.

5.8. Press escape to exit the rectangle command.

5.9. Delete the vertical line that crosses the diameter of the column.

★ This step is important, as Inventor is sensitive about overlapping sketch lines!

5.10. Similarly to the right side, draw a rectangle snapping from the top of the circle. Drag until you reach 8', the type **PiercapLeftWidth=13**. (4.5m). Use the tab key to verify you are typing in the correct text box.

5.11. Press ecape to finish the rectangle command.

5.12. Select the remaining vertical lines and the circle geometry by creating a right-to-left selection box. Just like AutoCAD, this will select objects that touch the selection.
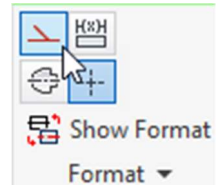
5.13. Now, click the construction line icon from the Format Panel. This hides this geometry when we are extruding the top of the pier in the next section.

The construction lines will appear as dashed.
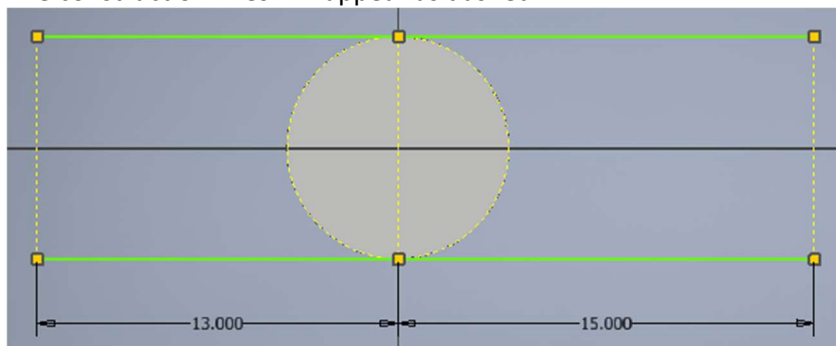
5.14. Go to the Sketch tab > Create panel. Select the Tangent type arc.

5.15. Sweep your cursor over one of the horizontal lines until you see the green dot at the end. This will indicate that the arc will be drawn tangent to the line highlighted in white.
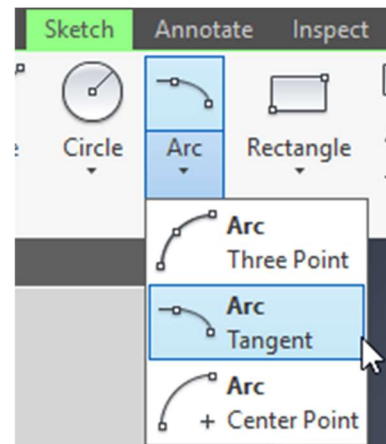
5.16. Click to create the arc along the right side of the pier.

5.17. Repeat to form the arc on the left side.

5.18. Press Esc on your keyboard to complete the arc command.

5.19. Click to select one of the horizontal lines.
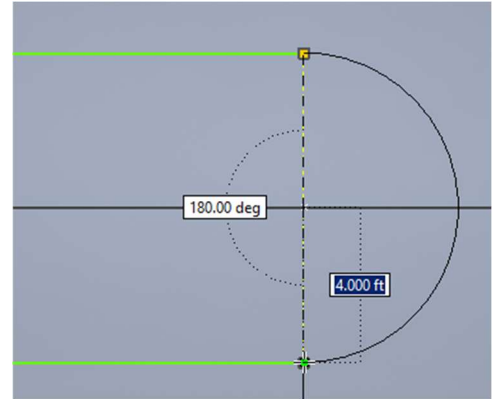
**5.20.** Right-click and select Close Loop.
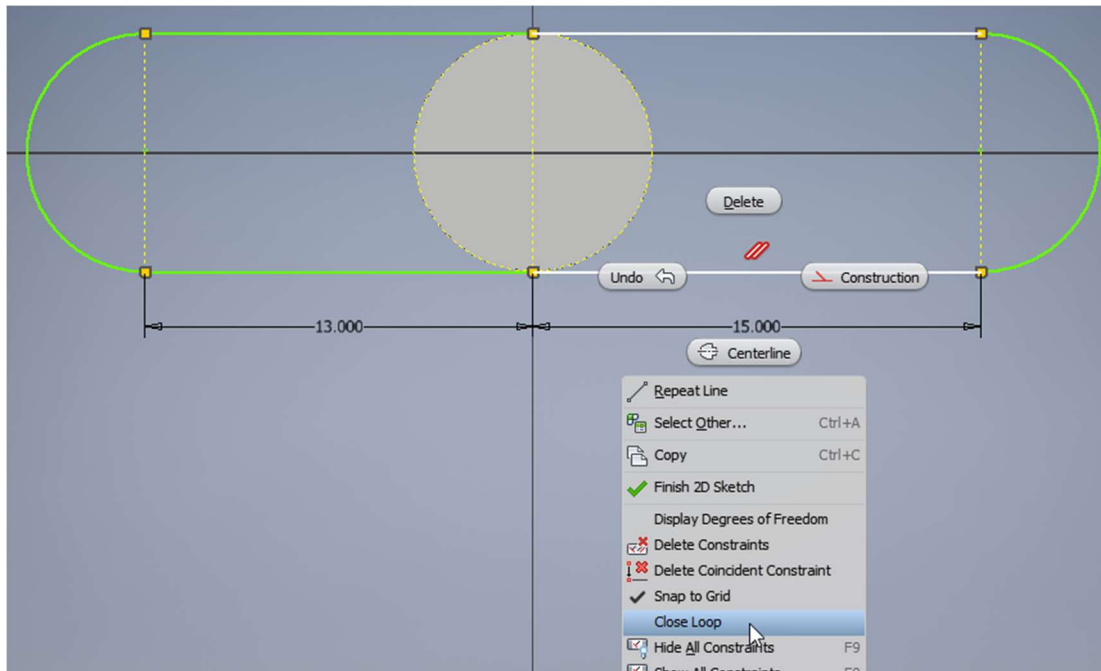
5.21. You may receive an instructional message, reminding you to select lines in order, until they form a closed shape. Click OK.
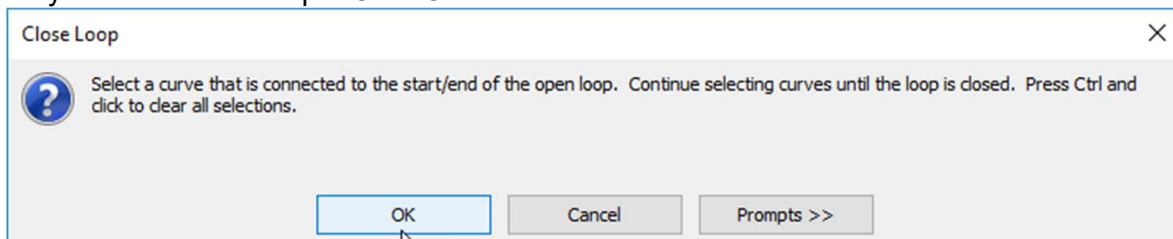
5.22. Click on each line of our bridge pier cap until the loop is closed.

   5.22a   As you click, you should receive a message that there are multiple sketch points where the highlighted curves connect. Would you like to combine the points into a single point?

   5.22b   Click Yes.

5.23. When you click the last line, you will receive a message indicating that the loop has been successfully closed. Click OK.
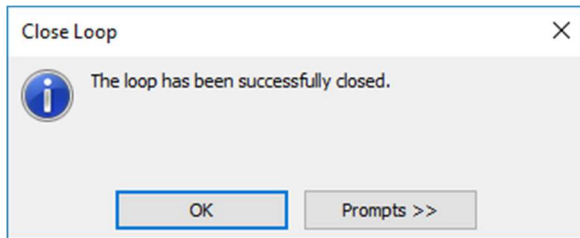
5.24. Click Finish Sketch.

5.25. Save the drawing.

Verifying your Sketch

Let's spend a few minutes checking out our new sketch.

★ Having a well-drawn and properly constrained sketch is critical for our bridge pier.

★ The top will fail to extrude if there are any open ends or the loop is not closed.

★ You may get unexpected geometry if the sketch is not constrained.

If you closed out of your sketch and need to return to edit it, there are several ways to get back in. the quickest way is to select the geometry and click Edit Sketch.
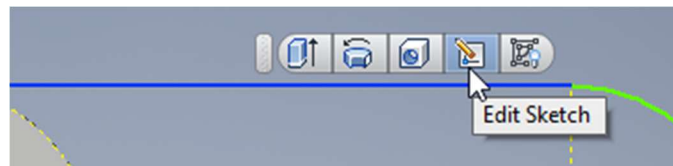
To explore your constraints, click the Show Constraints icon in the Constrain panel. Select various geometry until you see all the constraints.
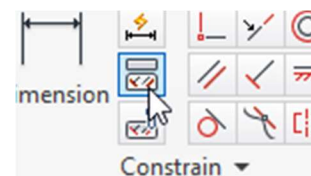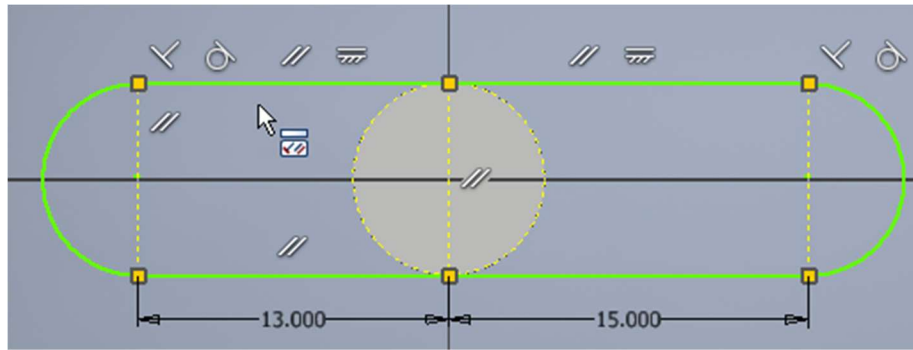
Right now, your lines are probably partially constrained. The geometry is green, meaning it could be constrained further.

When a line is fully constrained, it will turn blue.

Adding constraints in Inventor is like adding constraints in AutoCAD.

a. Click the icon for the constraint type you'd like to add.
b. Click the geometry to link parametrically.

Problem	Fix
Oops, I didn't snap to my column edge.	No sweat! Click the tangent constraint. Click the line, then the column edge. Repeat on the other edge of the column if needed.
I created my rectangle but didn't do the PiercapLeftWidth or PiercapRightWidth correctly.	Find the Parameters icon at the top of the Inventor Window. You can edit or add parameters in the resulting dialog box. We will get more into detail on the parameters dialog box later on in this lab.
My loop didn't close. ☹	You can force geometry to touch by using the coincident constraint, if there are gaps. Another common cause of loops not closing is overlapping geometry. If you forgot to delete the vertical line in step 5.9, the close loop command may pick the wrong path.

Exercise 6: Extruding the Top & Setting Up Parameters

If you are confident that you drew the piercap correctly, continue working in the previous drawing. If you are not so sure, open `Basic Pier (Exercise 6).ipt.`
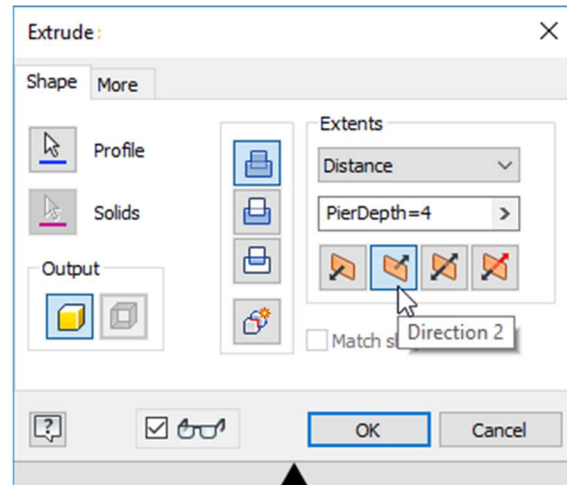
6.1. Click extrude from the create panel.

6.2. Type in **PierDepth=4** .

6.3. Set the direction to 2 (as shown).

★ That means we want the thickness of the pier to go *down* in relation to the PierHeight.

6.4. Click OK.

Are you ready to set up your parameters? Here we go!

6.5. Click the parameters button. f_x

6.6. Check your other model parameters.

★   Verify that you have spelled **PierHeight**, **PiercapRightWidth** and **PiercapLeftWidth** correctly and that the case matches what is shown above.

★   Place checkmarks in the **Key** column for:

     ✓   PierDia

     ✓   PiercapRightWidth

     ✓   PiercapLeftWidth

     ✓   PierDepth

     ✓   PierHeight

Parameter Name	Consumed by	Unit/Typ	Equation	Nominal	To	Model Value	Key		Export Param	Con
− Model Parameters										
PierDia	Sketch1	ft	8 ft	8.00...	○	8.000000	☑	☐		
PierHeight	Extrusion1	ft	20 ft	20.0...	○	20.000000	☑	☐		
d2	Extrusion1	deg	0.0 deg	0.00...	○	0.000000	☐	☐		
PiercapRightWidth	Sketch2	ft	15 ft	15.0...	○	15.000000	☑	☐		
PiercapLeftWidth	Sketch2	ft	13 ft	13.0...	○	13.000000	☑	☐		
PierDepth	Extrusion2	ft	4 ft	4.00...	○	4.000000	☑	☐		
d6	Extrusion2	deg	0.0 deg	0.00...	○	0.000000	☐	☐		
User Parameters										

6.7. At this point, your Parameter listing should match what is shown in the image above.

6.8. Test your parameters by changing the values in the Equation column.

★   This will ensure that your geometric constraints are set correctly.

For example, if you change the PierDia but the pier cap does not widen accordingly, your sketch may need the tangent constraint set.
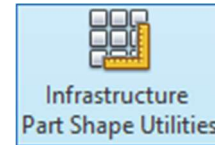
6.9. Save the file. 💾

Exercise 7: Exporting the IPT to Infraworks
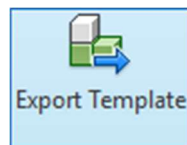
Continue working in the previous drawing.

If you were unable to complete the previous exercise but would like to continue, open the file `Basic Pier (Complete).ipt`.

7.1. Switch to the environments tab and click Infrastructure part shape utilities.

(If you do not see this button, it is because ShapeModeler.msi needs to be installed on your PC.)
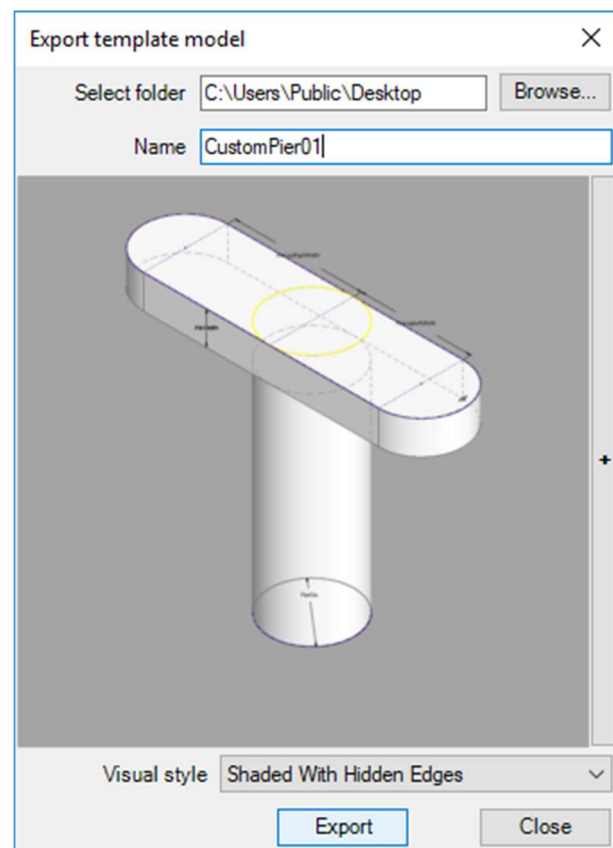
7.2. Click Export Template

7.3. In the Export Model Template dialog, click Browse.

7.4. Select the desktop of your PC and click OK.
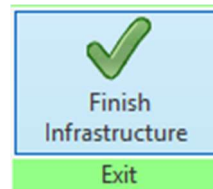
7.5. In the Name Field, type CustomPier01.
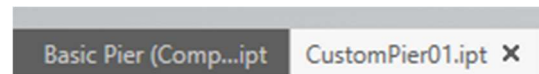
7.6. Click Export.

7.7. Click Close.

7.8. Click Finish Infrastructure.

Notice the tabs at the bottom left in the screen. The Infrastructure part shape utility has created a copy of your original Inventor file and saved a copy on the desktop.

7.9. Keep Inventor open, but close the `CustomPier01.ipt` file by clicking the X.

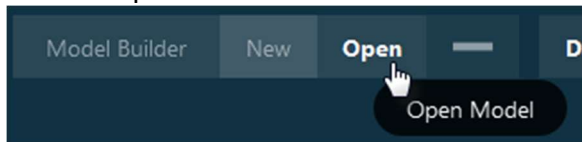**7.10.** Click Yes to save changes.

7.11. Minimize Inventor

Import Into InfraWorks
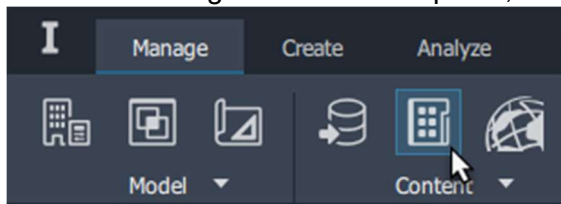
Exercise 8: At last, InfraWorks!

8.12. Launch Infraworks if it is not already started.
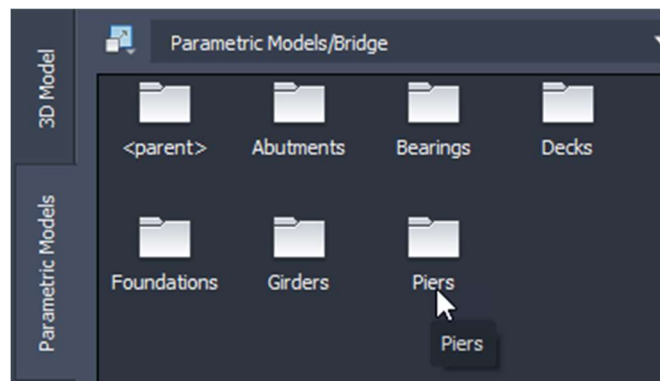
8.13. Click Open.

8.14. Browse to the folder for this class and open `Test Bridge.sqlite`

**8.15.** On the Manage tab > Content panel, click Style Palettes

8.16. In the Style Palette, switch to the Parametric Models tab.

8.17. Double-click to browse to the folder Bridge > Piers.

We will create a new folder to keep our custom pier separate from the out of the box ones.

8.18. Click the Green plus sign near the *top* of the Style Palette.

. CES319875-L- Custom Bridge Parts from Inventor to InfraWorks to Revit
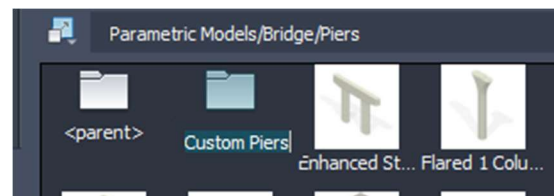
8.19. Click the Rename button and rename the folder to Custom Piers.

8.20. Double-click the folder to browse to it.

8.21. Click the plus icon at the *bottom* of the Style palette.

You should now see the Define New Parametric Models dialog box.

8.22. For the Model URI, click the ellipsis to browse to the Inventor file `CustomPier01.ipt` that you previously saved to the desktop.

**8.23.** If everything was defined properly in Inventor, your dialog box should look like the one below.

★ If you did not get this with the pier you created, browse back out and select the `CustomPier01 (Complete).ipt` file from your class files folder.

8.24. Switch to the Part Sizes Tab.

8.25. Rename the Part Size Name by double-clicking in the name field.
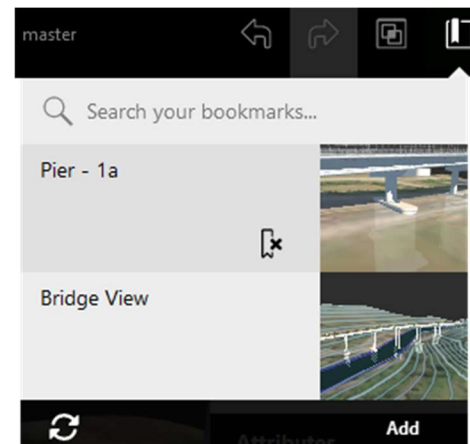
8.26. Set the name to AU2019.

At this point we could continue adding pre-configured part sizes by adding columns and changing the parameters as required. In the interest of time, we will not be adding more sizes.
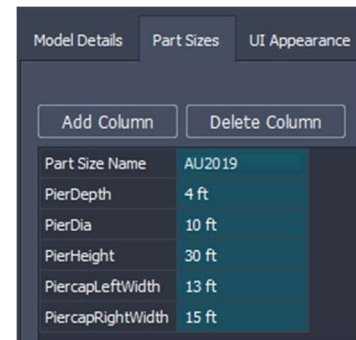
8.27. Click OK.

8.28. Click anywhere in the style palette to complete the style creation.

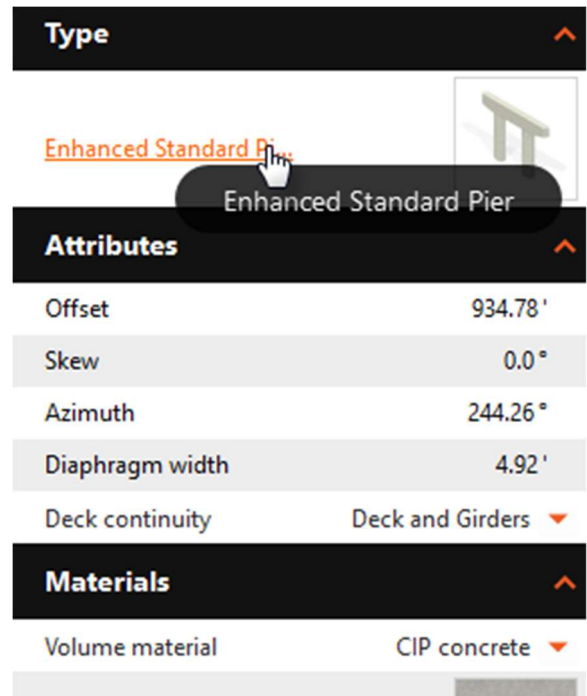**8.29.** Go to the Bookmarks, Select Pier – 1a.
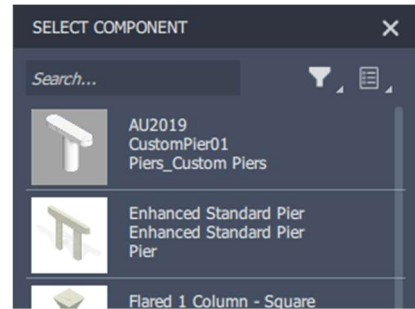
8.30. Select the bridge pier assembly.

8.31. Your first click will click the overall assembly, which includes the foundation, piers, girders and decks.

8.32. Click again to select the Pier.

8.33. The properties dialog box should reveal that it is an **Enhanced Standard Pier**. Click the hyperlink Enhanced Standard Pier.

Type	^
Enhanced Standard Pi...	
Enhanced Standard Pier	

Attributes	^
Offset	934.78 '
Skew	0.0 °
Azimuth	244.26 °
Diaphragm width	4.92 '
Deck continuity	Deck and Girders ▼

Materials	^
Volume material	CIP concrete ▼

8.34. In the Select Component asset card, double-click the **AU2019** pier from the top of the list.

Notice anything wrong with the pier? It is rotated incorrectly by 90°. This is easy to resolve, however.

8.35. Back in the style palette, double click to edit the CustomPier01.

	X	Y	Z
Translation	0.00000000	0.00000000	0.00000000
Rotation	0.000 °	0.000 °	90.000 °

8.36. Change the rotation in the Z axis to 90° as shown above.
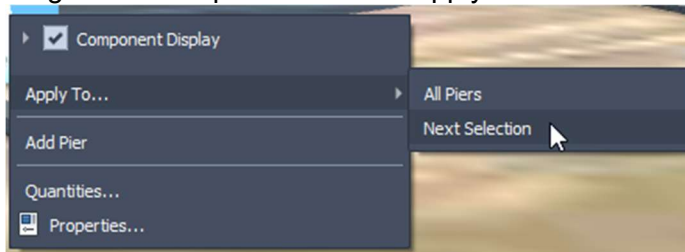
8.37. Click OK.

8.38. Press F5 on your keyboard to regenerate the model.

You should now see that the pier is placed correctly.

8.39. Go to the Bookmarks Select Pier – 1a.

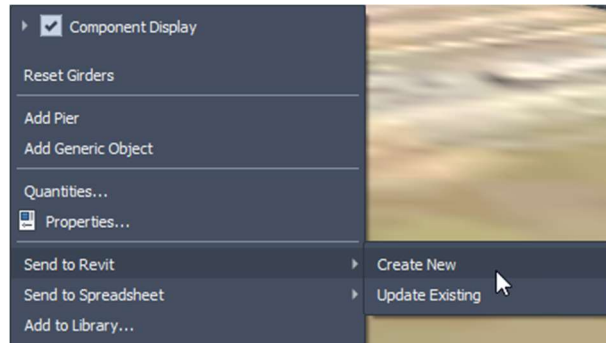**8.40.** Right-click the pier and select Apply To… > Next Selection.

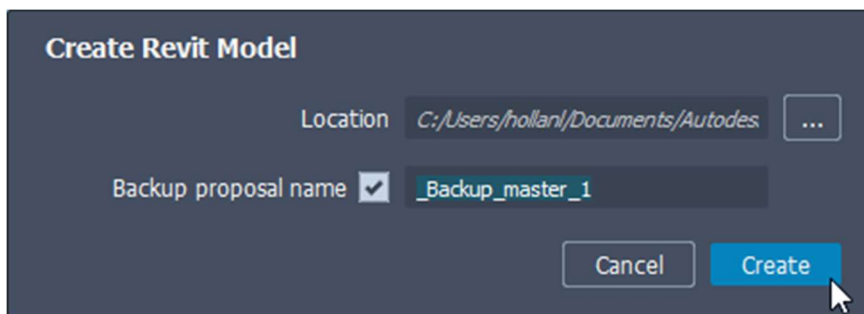**8.41.** Click any pier (or piers) and press enter.

8.42. Press escape to clear any selections.
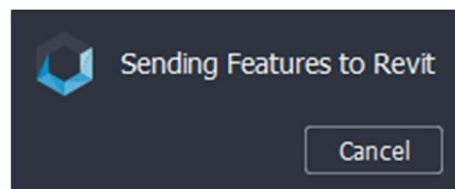
Exercise 9: A Bridge to Revit

9.1. Select the pier assembly. Right-click and select Send to Revit > Create New

9.2. Select the bridge. (to verify you have the bridge selected)

9.3. Leave the Location and proposal name as default. Click Create.
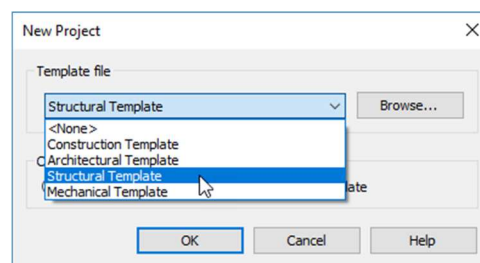

It will take several minutes as Infraworks packages up the bridge and creates an IMX file.
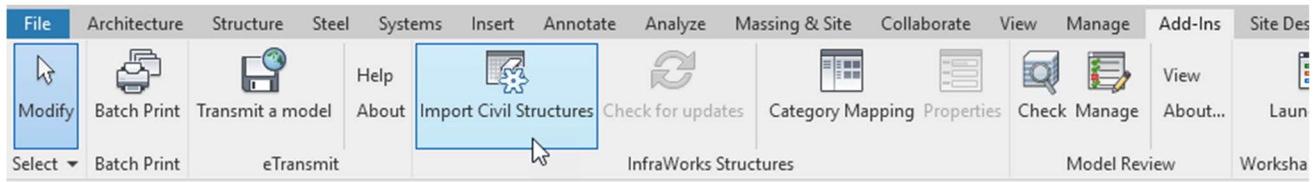

While this is exporting, see if any of your lab neighbors could use your help.

9.4. Launch Revit 2020 if it is not already open.
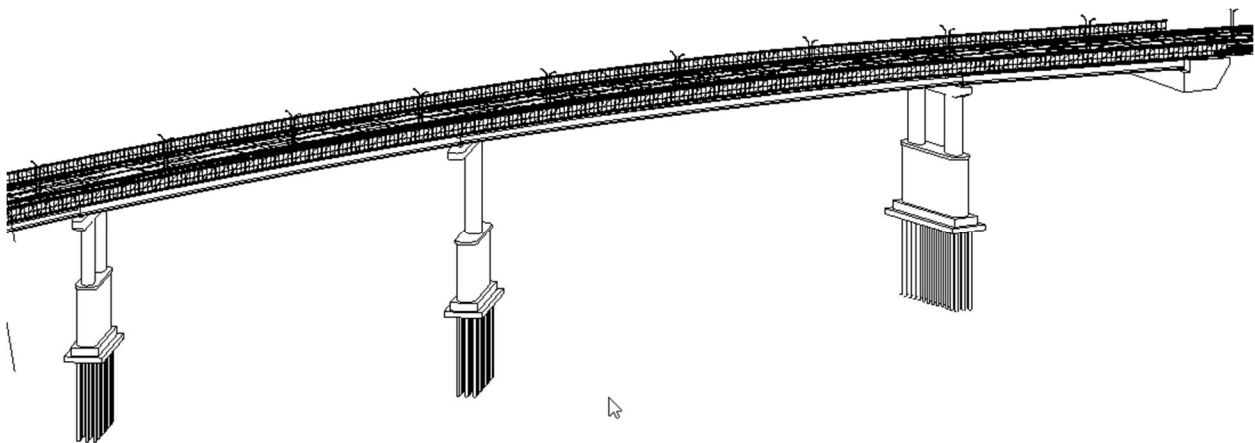
9.5. Click New…

9.6. Choose the Structural Template and click OK.

9.7. Go to the Add-ins tab > Infraworks Structures panel and click **Import Civil Structures**

It will take several minutes for all the data to import to Revit.

9.8. When the structures have finished importing, click the 3D view icon from the quick access toolbar.

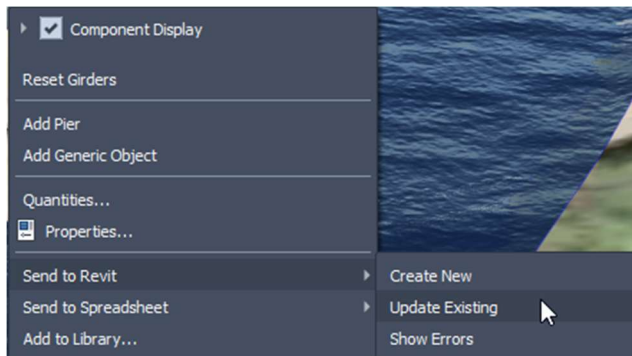

This is the end of the exercises.

Revit and Beyond

Revit for Rebar

Once your project is in Revit, you can start working on structural details such as placing rebar.

Check out this video on placing rebar in imported features.

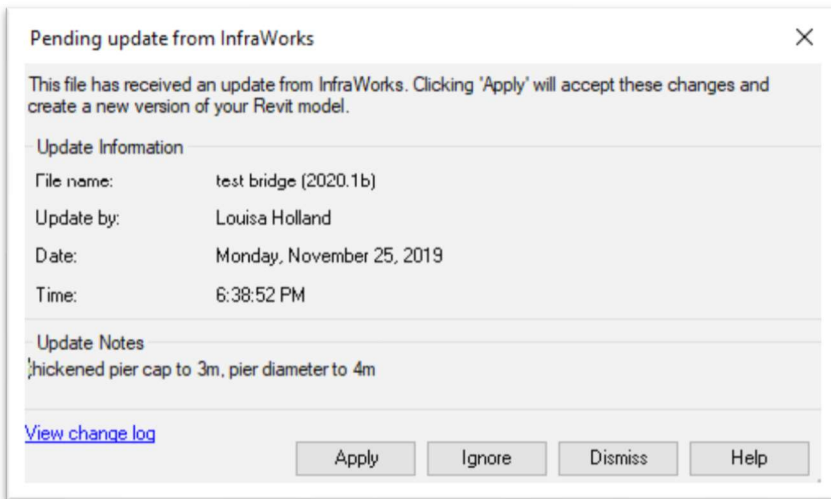## What Happens If a Design Change Occurs in Infraworks?

Design changes on your bridge can be updated by re-exporting from Infraworks. When working with design changes, be sure to use Update Existing as the option.
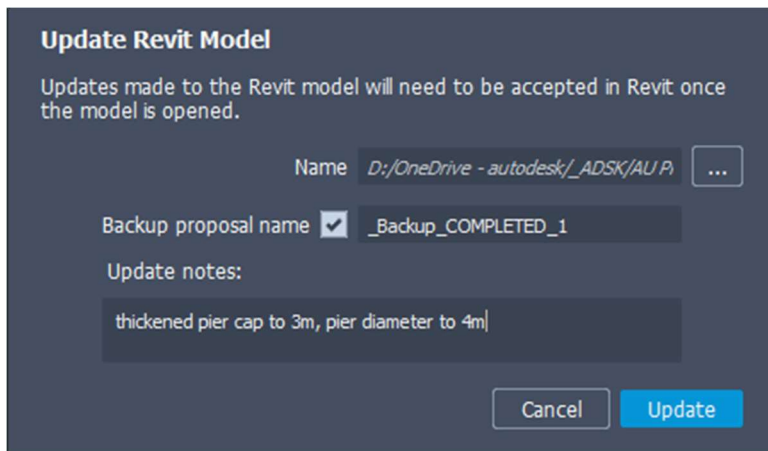
Update simply prompts you to overwrite the IMX created on the initial export.

On the Revit side, you will be prompted to accept the changes.

To accept the changes in Revit, click Apply.

Any rebar you placed will update as well.

Further Study and Other Helpful Links

All three parameters for the bridge pier *must be present*. PierHeight is the main driver of the geometry and is set by Infraworks. **Therefore, required parameters (such as PierHeight) cannot be a calculation in Inventor.**

★ If there are required parameters missing, Infraworks will not recognize the part.

Parameter Name	Required in:
PierHeight	Pier
PiercapLeftWidth	Pier
PiercapRightWidth	pier
CLBearingToBBW	abutment
LeftWidth	abutment
SeatDepth	abutment
SeatOffset	abutment
SeatWidth	abutment
LeftWidth	deck
RightWidth	deck

There are no required parameters for:

- Foundation
- Girders

Helpful Links

[About Parametric Models in Bridge Workflows](#)

[Sharing Parametric components](#)

[The official workflow from Inventor to Infraworks](#)

An entertaining introduction to workplanes: https://www.youtube.com/watch?v=KdJAYagKNVY

To learn more about using parametric constraints in Inventor, check out this [video by Faith Caplan](#).