3D Modeling in AutoCAD

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Learning Objectives

- Discover how to create 3D models by combining basic shapes to create more complex objects
- Learn how to use meshes and surfaces to create organic shapes
- Learn how to modify 3D models using sub-object selection and underlying curves
- Learn how to convert between solids, meshes, and surfaces

Description

Capitalize on your knowledge of 2D to create 3D models entirely in AutoCAD software. In this class, you will learn how to take your ideas from concept to completion by creating and editing solid objects; creating smooth, free-form shapes using meshing tools; and capitalizing on the power of surface modeling. We'll look at the differences between history- and non-history-based solids and see how models can automatically update when you modify underlying curves or parameters. You will also learn how to control selection and editing with subobject selection filters, as well as glean valuable tips and tricks for navigating and visualizing your AutoCAD 3D models. If you thought 3D in AutoCAD was just too hard, think again.

Speaker(s)

David is the Senior Content Manager for CADLearning® products at 4D Technologies, where he develops content standards and creates affordable training solutions for Autodesk software, including AutoCAD, AutoCAD LT and ReCap. He has more than 30 years of hands-on experience with AutoCAD and 15 years with Revit as a user, developer, author and consultant, and is an Autodesk Certified Professional for both AutoCAD and Revit. A contributing editor to *Digital Engineering* magazine, he is also the former senior editor of *CADalyst* magazine, and is the author of more than a dozen books about AutoCAD. A licensed architect, David was also one of the earliest AutoCAD third-party software developers, creating numerous AutoCAD add-on programs. As an industry consultant, David has worked with many companies, including Autodesk. He has taught college-level AutoCAD courses and has consistently been a top-rated speaker at Autodesk University.

Introduction

We live in a 3D world. Every object that is built or manufactured—some that you may not have even considered—must be designed and modeled in order to take it from concept to reality.

Yet most people still work in 2D. That means that most designers take the 3D conceptual designs they have imagined and flatten them into 2D drawings. Those 2D drawings must then be interpreted by builders or manufacturers in order to produce 3D objects.

For years, this 3D to 2D to 3D process was accepted without question, because originally 2D CAD was simply a replacement for traditional 2D drafting.

Fortunately, AutoCAD has evolved over the years from a 2D drafting tool into a very capable 3D modeling tool that is quite suitable for conceptual design. Thanks to these changes, you can design the way you think—in three dimensions.

To get started on your journey into the world of 3D conceptual design, consider your thought process as you create a two-dimensional drawing in AutoCAD. You probably start with an idea— an image in your mind. Then you mentally flatten the image by picturing it from different angles, using your knowledge of drafting to identify basic 2D shapes such as lines, arcs, and circles.

From there, you apply your AutoCAD knowledge by selecting appropriate drawing tools such as LINE, PLINE, ARC, and CIRCLE, as well as editing tools such as OFFSET and TRIM. As you create those 2D drawings, you continually refer back to your mental image of the 3D object.

Designing in 3D is a similar process, but is actually easier because you do not have to mentally flatten your imagined image. Instead, you skip that step and begin by identifying basic 3D shapes. For example, can you identify some of the basic shapes used to create the air hockey paddle or the house in the images below?


The air hockey paddle is made up of several cylinders, with an array of boxes forming stiffening ribs in the base, a sphere forming the top, and an inverted cone used to remove material. Similarly, the house is comprised of multiple boxes, wedges, and a cylinder.

3D Modeling Workspace

AutoCAD includes three default workspaces: Drafting & Annotation, 3D Basics, and 3D Modeling. You can access workspaces from the Status bar or the Quick Access Toolbar.
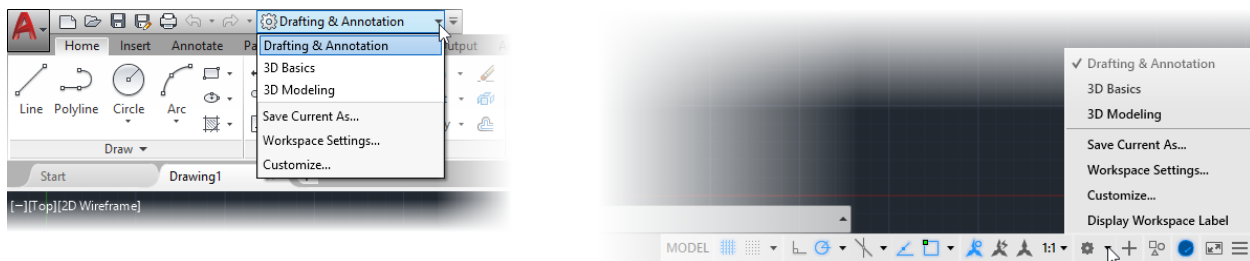
The 3D Modeling workspace provides easy access to all of the 3D modeling and visualization tools via the AutoCAD ribbon, whereas 3D Basics includes just a subset of those tools.
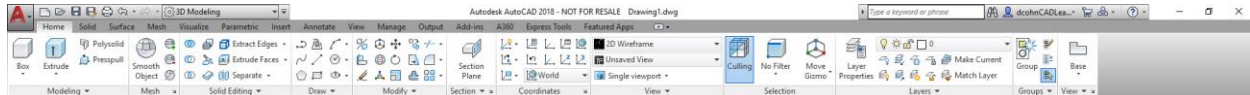

Creating and Editing Solid Cylinders

Solid primitives are essential to creating typical 3D designs and are the easiest way to start building up a 3D object. The tools for creating solid primitives are all available on the **Home** ribbon in the **Modeling** panel. For example, the **Box** tool initially displays in the split button, but when you choose a different tool, it floats to the top of the split button.

These 3D tools work very much like their 2D counterparts. For example, drawing a cylinder is very much like drawing a circle; you start by drawing a circle using the center/radius, 3P, 2P, or TTR options, and then add a height. There is also an option to create an elliptical cylinder that starts out similar to drawing an ellipse. If you know exactly what you want the cylinder to look like, you can pick precise points or enter specific values (just like drawing a circle). But if you are in the conceptual design phase and starting with a blank drawing, you can simply create a cylinder and then modify it later as your design evolves.


When you work in 3D, you will want to be able to view your model from different angles, rather than just from the top down (a plan view when working in 2D). AutoCAD provides easy-to-use tools to view and navigate models in 3D. You can press the **SHIFT** key and middle mouse button (the roller wheel) to activate 3D Orbit mode, so that you can move around the model even while in the middle of another command. You can also use the
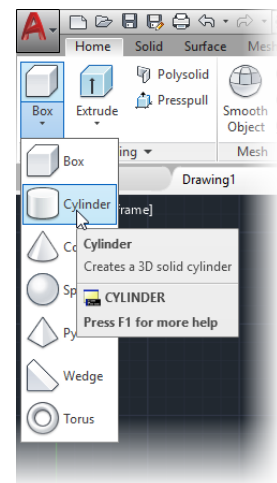
ViewCube to better understand the current 3D orientation, and even use the ViewCube itself to move around in 3D and switch to isometric, plan, and elevation views.

The ability to create solid primitive objects—such as boxes and cylinders—has existed in AutoCAD for many years. But in older versions, those objects were not easily editable. If you did not create the object correctly the first time, you had to erase it and start over. But that is no longer the case. AutoCAD lets you easily edit solid primitives using grips or the Properties palette.

For example, when you select a cylinder, you see several grips. You can use the square grip at the center of the base to move the cylinder. The triangular grips at the quadrants let you change the radius of the cylinder. And the triangular grips at either end of the cylinder lets you change the height of the cylinder.
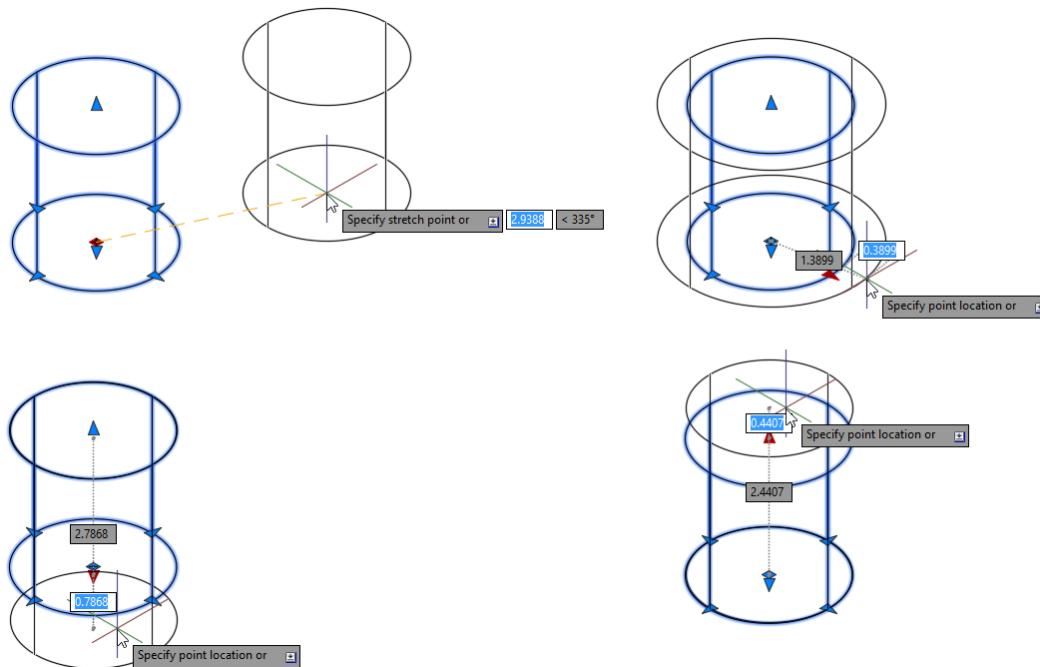

Culling

If you do not see all of the grips illustrated above, check to see if culling is enabled. Culling controls whether 3D objects that are hidden from view can be highlighted or selected:

- When enabled: you only see grips on visible faces (default)
- When disabled: you see all grips

Culling Object

Controls whether objects that are hidden from view are highlighted when you roll over them

CULLINGOBJ

Press F1 for more help
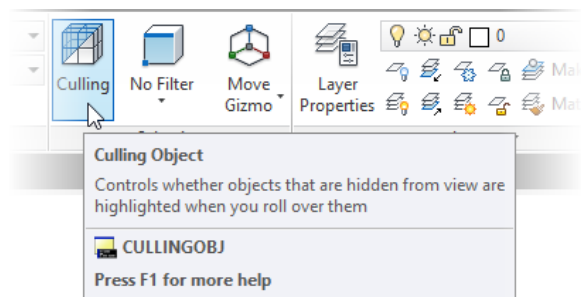
Changing Properties

In addition to grip editing, you can select a solid and then change key values in the Properties palette. The properties available will vary, depending on the type of solid primitive you select. For example, a box has properties for length, width, and height; whereas a cylinder has properties for radius and height. You can view and modify these properties in the Geometry panel of the Properties palette. As you change the values, the geometry in the drawing editor updates.
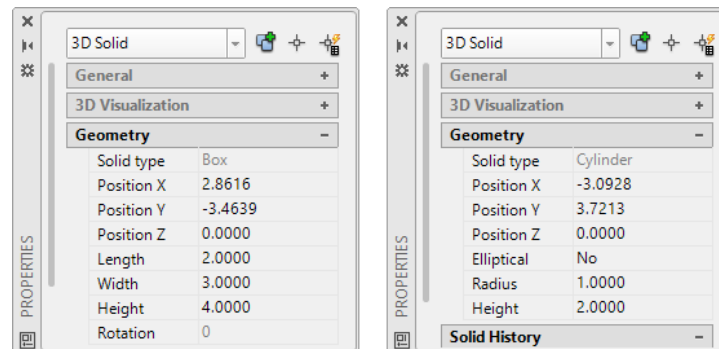
You can also use the Quick Properties palette to modify a solid. To display Quick Properties, toggle it on using the control on the **Status** bar, or simply double-click an object.


The Quick Properties palette serves the same purpose as the Properties palette, but the Quick Properties palette is customizable. You can configure it so that it shows just the properties you need to view and change, rather than a list of every property associated with a selected object. You can also control where the Quick Properties palette is displayed. By default, it displays above and to the right of the cursor, giving you quick and easy access to key properties whenever you select an object.

For example, when you select a cylinder, you can easily modify its radius and height in the Quick Properties palette, which will appear near your cursor. For the air hockey paddle, change the radius and height of the cylinder to **30** and **14**, respectively.
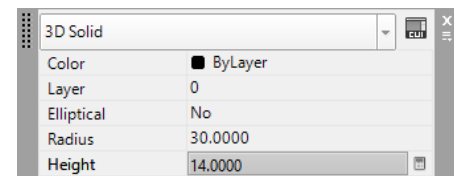

Adding a Second Cylinder

Continuing the design of the air hockey paddle, create a second cylinder. Since the two cylinders need to be concentric, use the **Center** object snap to snap the center of the new cylinder to the center of the existing cylinder. Remember that a cylinder has two center object snap points, one at the base and one at the top.
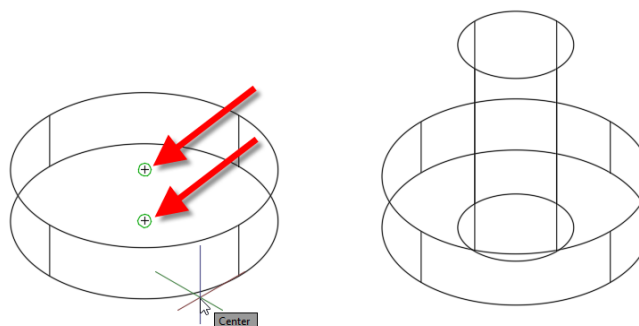
It does not matter which one you choose because it is easy enough to change later as your design evolves. But, for now, pick the center object snap at the bottom of the existing cylinder and pick other points to approximate the radius (**13**) and height (**34**) of the new cylinder.

## Creating and Editing Solid Spheres and Cones

You will create a solid sphere to produce the rounded top of the air hockey paddle. On the **Home** ribbon, in the **Modeling** panel, expand the split button and select **Sphere**. Creating a sphere is similar to drawing a circle. You can use the default center/radius option or choose from 3Point, 2Point, and TanTanRadius. Snap the center of the sphere to the center of the top of the cylinder, and then use the **Quadrant** object snap to make the sphere the same radius as the cylinder.
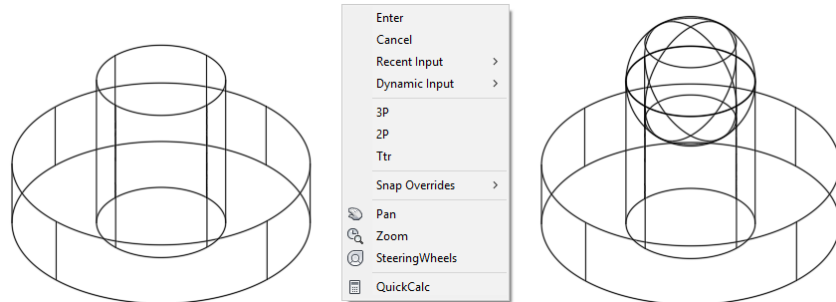
Next, you will create a solid cone, which will be used later to remove material from the lower cylinder. On the **Home** ribbon, in the **Modeling** panel, expand the split button and select **Cone**. The program prompts you to specify the center point of the base of the cone, and again, you can use the default center/radius option or choose from 3Point, 2Point, TanTanRadius, and Elliptical. Since you want the cone to be located at the center of the cylinder, use center/radius.

After specifying the base of the cone, the default option is to specify its height, but there are additional options at this step as well. For example, you can pick two points to determine the height and change the orientation of the cone, or you can specify a value for the top radius. If you do not specify a top radius, the cone will taper to a point. Also, remember that the base does not necessarily have to be below the top. For the air hockey panel, create the cone so that it tapers downwards toward the bottom of the paddle.

Since you will use the cone to subtract material from the large lower cylinder, modify the base radius so that a rim will remain around the top of the cylinder after the volume of the cone is removed. Modifying a solid cone is just as easy as modifying a solid cylinder. To ensure a rim of 4 units, drag the cone's quadrant grips **4** units toward the center.
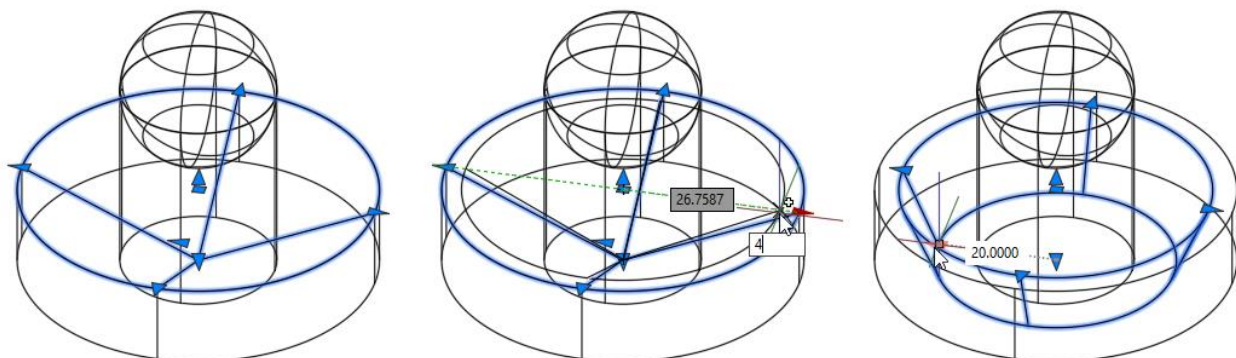
The top of the cone is below the base because of the way the cone was created. AutoCAD does not care where the top is in relation to the base. If you look at the properties for the cone, you

will see a Base Radius, a Top Radius, and a Height. Widen the top of the cone (which in this case is toward the bottom) by dragging the Top Radius out until it equals **20** units.

Although AutoCAD does not care if the Top Radius is really on top, there is a difference. If you change the height of the cone, the base maintains its position while the location of the "top" adjusts accordingly. For example, to ensure you have 4 units of material left at the bottom of the cylinder after you subtract the cone, change the height of the cylinder from 14 to **10** units.
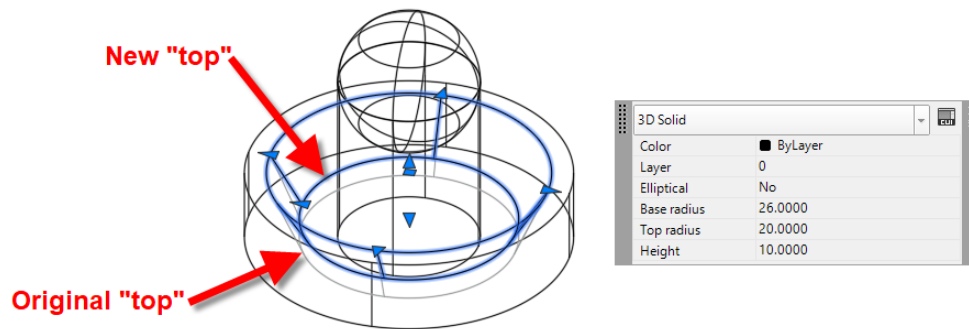
Subtracting and Unioning Solids

The cone represents material to be removed from the paddle. You can use Boolean tools to add and remove solid objects. These tools are located on the **Home** ribbon in the **Solid Editing** panel.
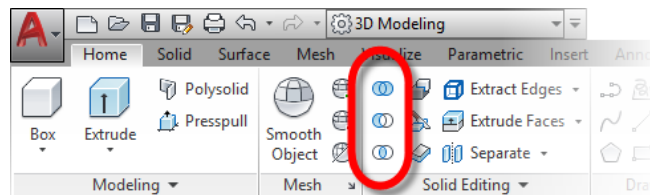
Union joins all selected objects to form a single composite solid object. Subtract also creates a composite solid. When using the Subtract tool, you first select one or more objects and then select the objects you want to remove (subtract) from the first set.

When performing Boolean operations, the order in which the operations are performed is important, because differences in the order can create different results. For example, if you were to first union the cylinders and sphere and then subtract the cone, you would end up with a gap between the upper part of the model (the handle) and the lower part (the striker). This would result in a *disjointed solid*, a single solid composed of what appears to be two separate objects (as shown on the left in the image below). To avoid this, first subtract the cone from the lower cylinder, and then union the remaining objects together.
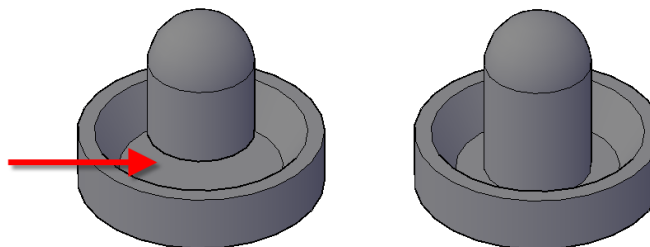

Remember that when using the Subtract tool, you first select the objects you want to subtract from, then press **ENTER** to finish that selection, and then select the objects you want to remove

from the first set. It is easy to forget this, since SUBTRACT is one of the few AutoCAD commands that prompts for two selection sets.

Solid History

AutoCAD can track the history of changes you make to a solid, similar to the way in which Autodesk Inventor tracks changes in its History Tree. This capability is controlled by the SOLIDHIST system variable. By default, however, this variable is set to 0 (off) rather than 1 (on).

When the Solid History is on, you can press the **CRTL** key to select primitive subobjects within composite solids. So even after performing a Boolean union or subtract, you could still select and edit the solids that were combined to create the composite solid object. But when the Solid History is off, you can no longer select the primitive subobjects after you perform Boolean operations.

Composite solid with Show History set to No (SOLIDHIST=0)

Composite solid with Show History set to Yes (SOLIDHIST=1)
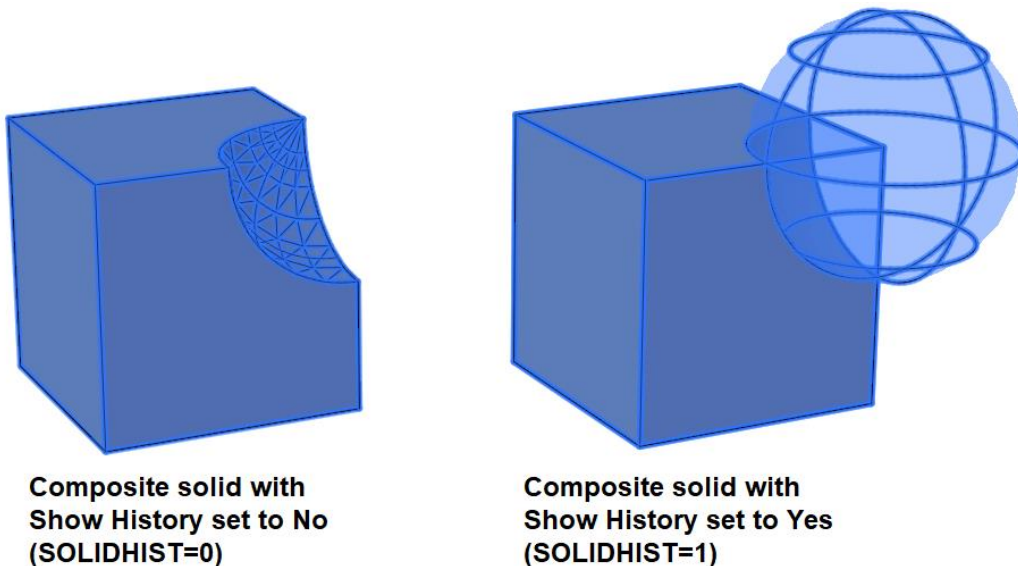
In order for AutoCAD to track the Solid History, you must enable it **before** creating the solid primitives you plan to combine. To change the SOLIDHIST value, you must type the system variable, or you can select the solids you want to combine and enable their Solid History in the Properties palette before combining the solid primitives.

You can also remove the history of a selected composite object by changing its History setting to None, or by using the BREP command. According to the Help file, "removing a composite history is useful when you work with complex composite solids," but really, the choice is yours. In addition, some operations remove any existing history regardless of the Solid History setting.

Setting a Visual Style

As you work with 3D models in AutoCAD, you can change the visual style to help you better visualize the geometry you are creating. AutoCAD includes ten visual styles—2D Wireframe, Conceptual, Hidden, Realistic, Shaded, Shaded with Edges, Shades of Gray, Sketchy, Wireframe, and X-Ray—and you can create custom visual styles. Tools for working with visual styles are located in several places in the ribbon, including the **View** panel on the **Home** ribbon.
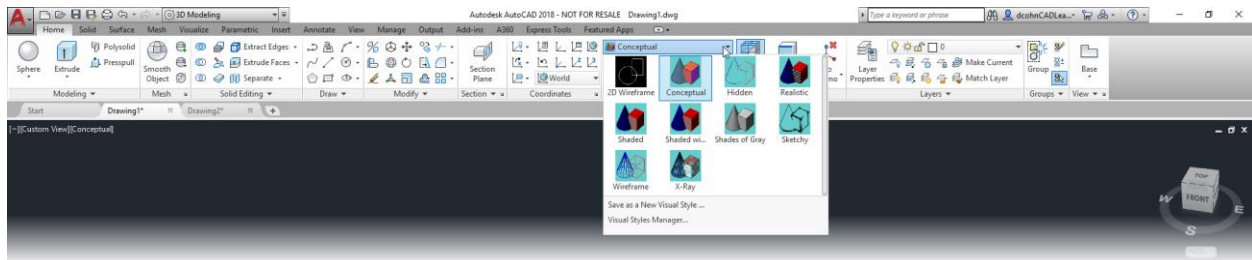
Viewing the model as a wireframe can be helpful if you want to see geometry that would otherwise be obscured by shaded faces. Viewing the model using the Conceptual or Realistic visual style can help you better understand the model. For example, when displayed using a wireframe style, you cannot tell if the cone is an extra object that overlaps the cylinders or if it has been removed from the cylinder. By changing to the Conceptual visual style, you can clearly see that the cone has been subtracted.
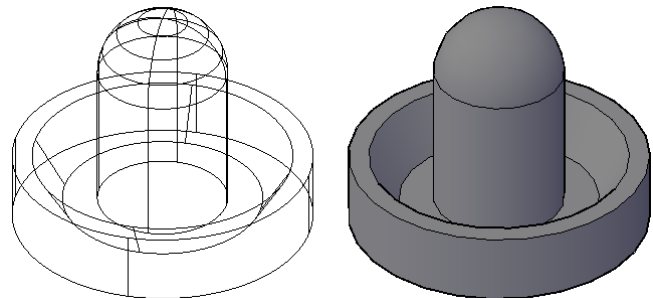
Editing Composite Solids

When you union or subtract solid primitive objects, AutoCAD creates a composite solid. If you select any of the primitive objects, the entire composite is selected. If you model without history, you can only modify component solids using direct manipulation of faces. If you model with history, however, you can still edit the primitive objects by using the **CTRL** key. If you press the **CTRL** key as you move the cursor over a composite solid, AutoCAD highlights each primitive subobject. You can then select the highlighted subobject. Once it is selected, you can use grips or the Properties palette to edit the primitive properties in the same ways that you could before performing the union or subtract operation. For example, if you want to shorten the handle on the air hockey paddle, you might select the stretch grip and move it down 15 units. Notice the edit only affects the cylinder. In AutoCAD, there are no constraints between 3D shapes (unlike parametric 3D design applications such as Inventor and Revit).
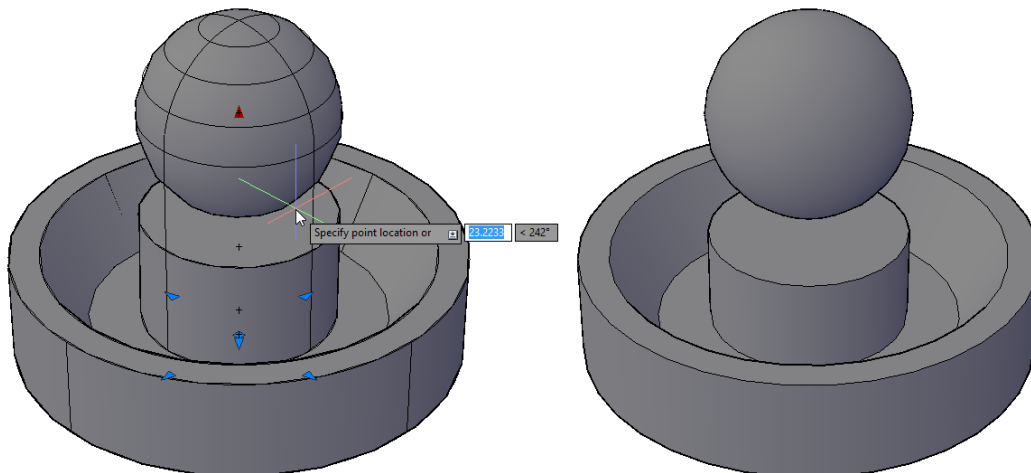
As previously mentioned, you can also edit subobjects using the Properties palette. Although the cylinder is a subobject in the composite solid, all the typical cylinder properties are available for editing. Change the radius of the cylinder to **13** and the height to **34**.
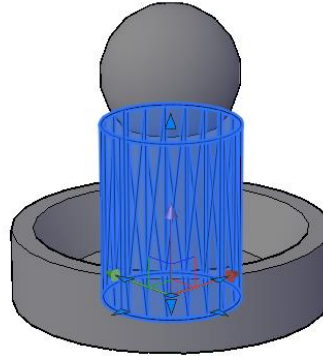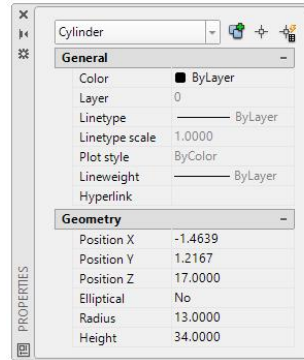
Press **CTRL** and select the sphere. A "gizmo" displays at the center of the sphere. Even if the gizmo functionality is turned off, the sphere's center grip displays and behaves similarly to when working in 2D. Selecting the grip makes it the hot grip and automatically enables the Move editing mode. You can then select an object snap point on an existing object. Snap the sphere to the center of the top of the cylinder. Then, select one of the sphere's quadrant grips and snap it to a quadrant point of the cylinder to change the radius of the sphere to match the radius of the cylinder.
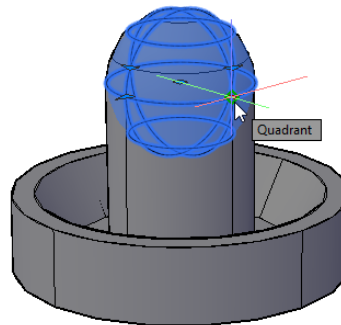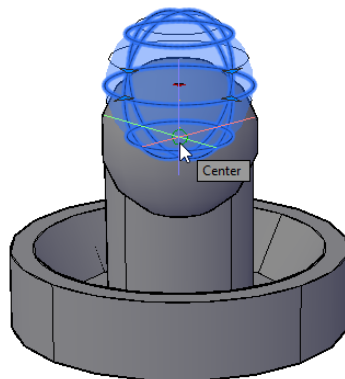
To verify that you snapped to the right point, thus creating a sphere with the correct radius, you can view the properties of the sphere in the Properties palette. The radius should equal 13, just like the radius of the cylinder you previously edited.
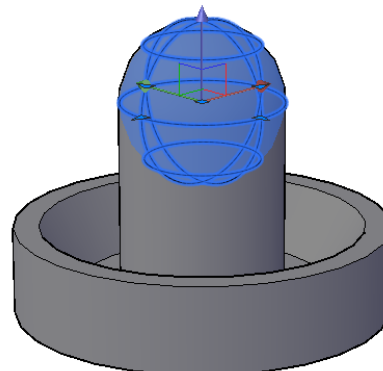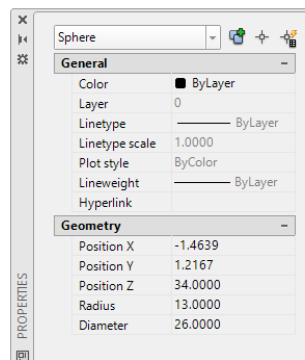
## Creating and Editing Chamfers and Fillets

All the objects used thus far to create the air hockey paddle are basic primitive shapes: cylinders, a sphere, and a cone. The result conveys the general shape, but the edges are hard. You can soften edges by adding chamfers and fillets. If you wish, you can use the same CHAMFER and FILLET commands used when working in 2D. When working in the 3D Modeling workspace, these tools are located on the **Home** ribbon in the **Modify** panel. The Fillet, Chamfer, and Blend Curves tools all share the same split button, and whichever tool you used last floats to the top.
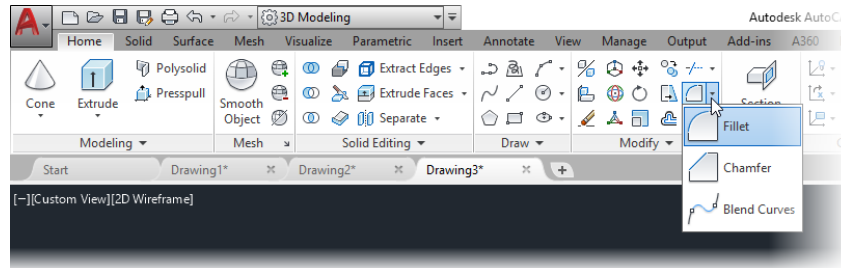
Click the **Chamfer** tool and select the edge you want to chamfer. AutoCAD highlights one of the surfaces adjacent to the selected edge, and you can toggle between these surfaces. The first chamfer distance will be measured along this surface.
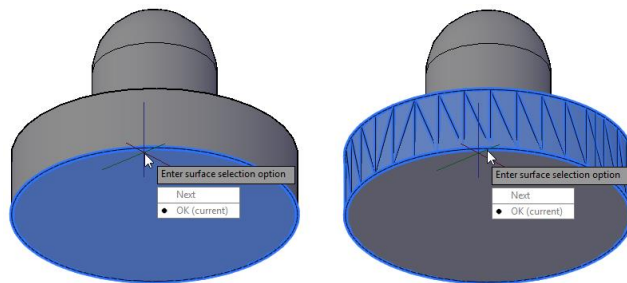

With the bottom surface selected, select **OK (current)**. When the program prompts you for the base surface chamfer distance, type **2** and press **ENTER**. The program then prompts you for the other surface chamfer distance. Type **1** and press **ENTER**. AutoCAD then prompts you to select an edge or loop. Click the edge again and then press **ENTER** to complete the chamfer. The image below shows a side view of the resulting chamfer.
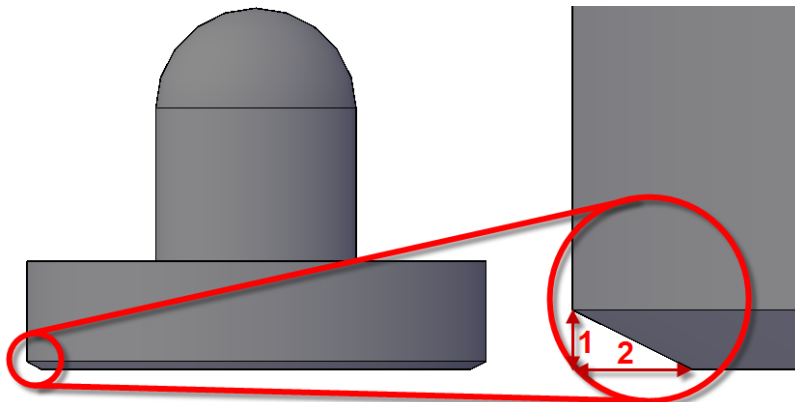

Undo the CHAMFER command and then see how you can accomplish the same thing by using similar tools specifically designed for working with 3D solids. These tools are located on the **Solid**

ribbon in the **Solid Editing** panel. The Chamfer Edge and Fillet Edge tools share the same split button, and whichever tool you used last will float to the top.
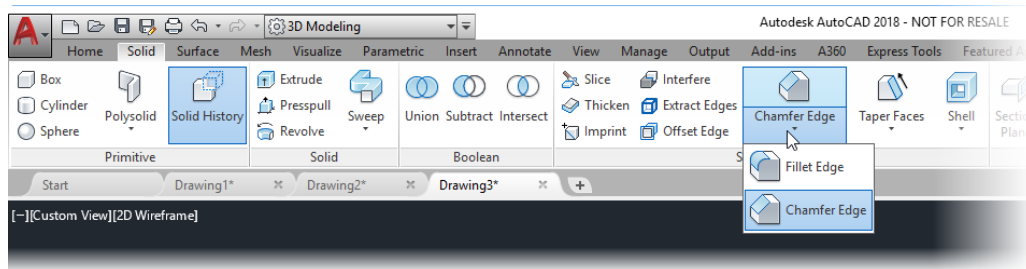

Click the **Chamfer Edge** tool and select the edge you want to chamfer, and then choose the **Distance** option.
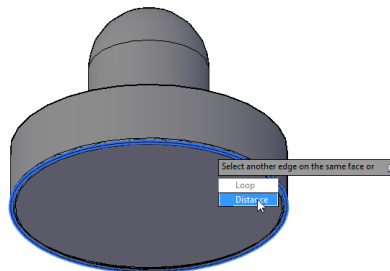
The program prompts you to specify the first distance. Type "**2**" and press **ENTER**. The program then prompts you to specify the second distance. Type "**1**" and press **ENTER**. Then, press **ENTER** twice to complete the command.

You can also use tools to create rounded edges on your 3D model. Again, you can either use the same FILLET command you use when working in 2D, or you can use the Fillet Edge tool.
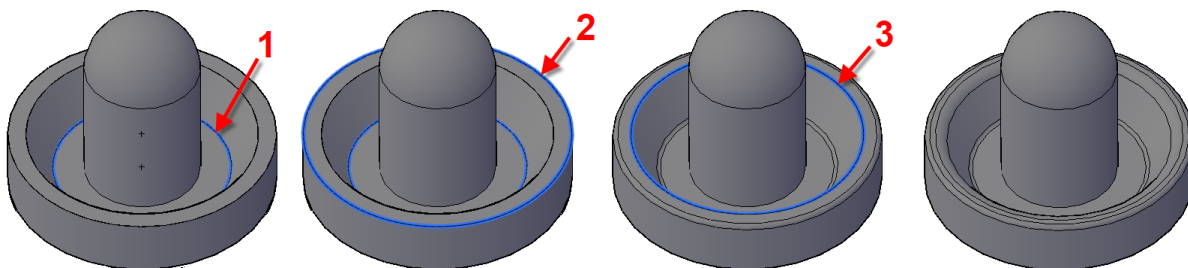
To use the FILLET command, on the **Home** ribbon, in the **Modify** panel, expand the split button and click the **Fillet** tool. AutoCAD prompts you to select the first object you want to fillet. Click one of the edges you want to fillet (edge 1 in the image above). That edge highlights, and the program prompts you to enter the fillet radius. Type "**1**" and press **ENTER**. The program then prompts you to select an edge. Click the other edge (edge 2 in the image above). Once both edges are highlighted, press **ENTER** to complete that fillet.

Use the Fillet Edge tool to create another fillet. On the **Solid** ribbon, in the **Solid Editing** panel, expand the split button and click the **Fillet Edge** tool. AutoCAD prompts you to select an edge. Click the edge you want to fillet (edge 3 in the image above). Then, select the **Radius** option, specify a radius of **3** units, and then press **ENTER** twice to complete the command. Now, the model has smooth edges (as shown in the image on the right, above).

When you add chamfers and fillets to a composite solid, the chamfers and fillets become subobjects within the composite solid. That means that if History is on, you can select chamfer and fillet subobjects using the **CTRL** key. As you move the cursor over a composite solid, the chamfers and fillets highlight. You can then select them, just as you can with solid primitive subobjects. And like other solid primitives, those chamfers and fillets have grips, so you can edit the chamfers and fillets by using grips, or you can change the chamfer distances or fillet radius in the Properties palette.
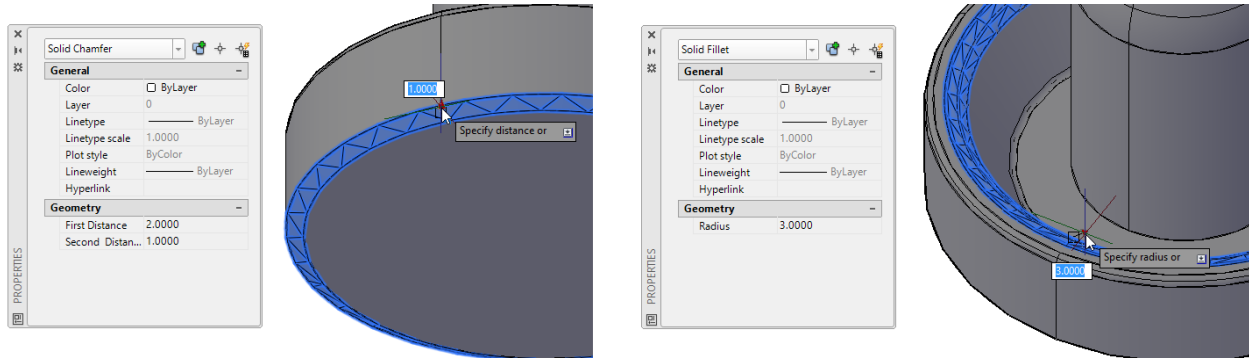

For example, if you press the **CRTL** key and select the large fillet on the air hockey paddle, the Radius property is displayed in the Properties palette. When you enter a different value, the model dynamically updates to reflect the change.

If you decide that you do not want a chamfer on the bottom of the air hockey paddle, you can use the **CTRL** key to select the chamfer, and then press the **DELETE** key (or use the Erase command) to remove it.

Of course, if History is off, you cannot use the **CTRL** key to select subobjects. In that case, you can still edit the model using direct manipulation.

Shelling a Solid

With all of the commands thus far, if History is on, you can use the **CTRL** key to select subobjects. But some operations remove the History, causing the model to lose the **CTRL** key editing capability. Of course, even after losing the History, you can still edit the model using direct manipulation, but you can save a lot of editing time if you avoid those operations until later in the design process.
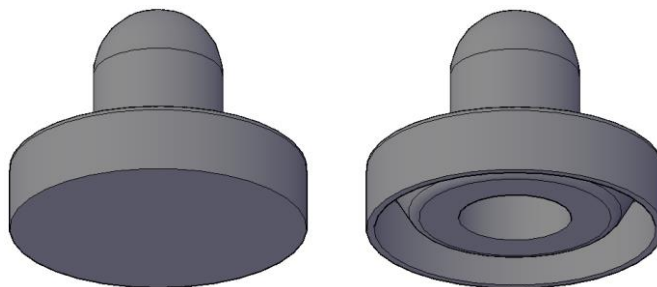

For example, the air hockey paddle eventually needs to be a thin shell (as would be produced by plastic injection molding) rather than a solid mass. The Shell tool in AutoCAD is perfect for performing this type of operation. Unfortunately, the Shell tool is one of those functions that removes the History. For example, when you press the **CTRL** key and click on the tall cylinder

before using the Shell tool, AutoCAD knows it is a cylinder and displays the appropriate grips. When you click on the same object after using the Shell tool, AutoCAD treats it as a face with limited editing capability.
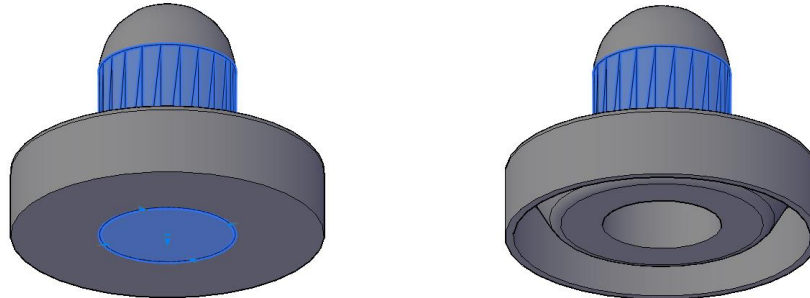

That is fine if those subobjects are already the right size, but when doing conceptual design, that is rarely the case. You want to be able to modify those objects as long as possible. Once you shell the objects, subobject editing will no longer be possible. Nevertheless, the Shell tool is handy for creating thin-walled objects such as the air hockey paddle. Since all of the components of the air hockey paddle are now the correct size, you are ready to perform the shell operation.
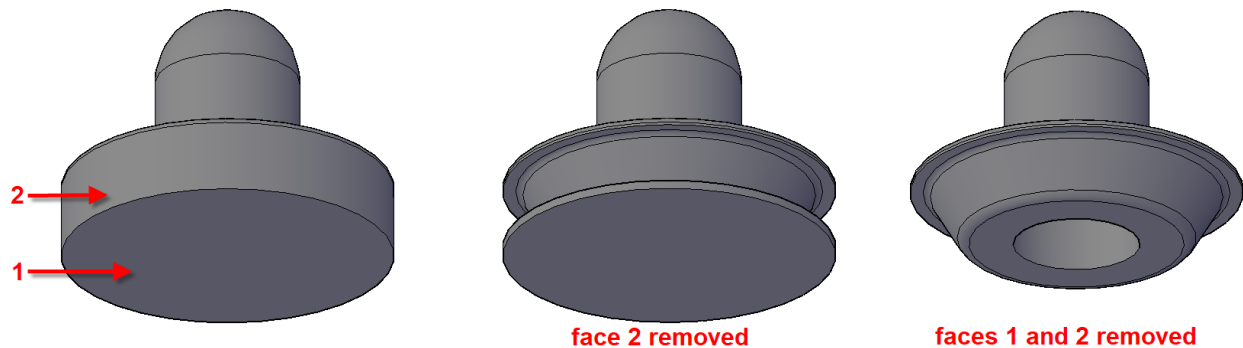
The Shell tool can be found on both the **Home** ribbon and on the **Solid** ribbon. (Note that this tool is actually one of the options of the SOLIDEDIT command, not a separate command. You cannot just type SHELL, since that opens an OS command window.)
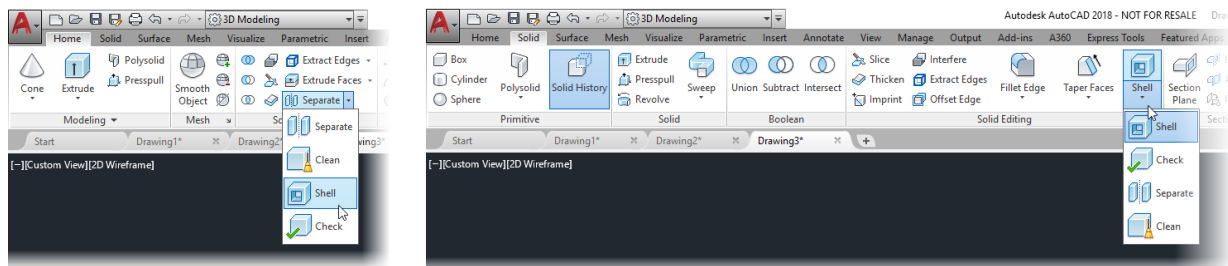
The program prompts you to select a 3D solid. Click anywhere on the model. Next, AutoCAD prompts you to remove faces. Faces are not actually removed from the object, but rather, this step gives you the option of excluding faces from the shell operation. Select the circular face on the bottom of the cylinder. This will result in an open bottom.
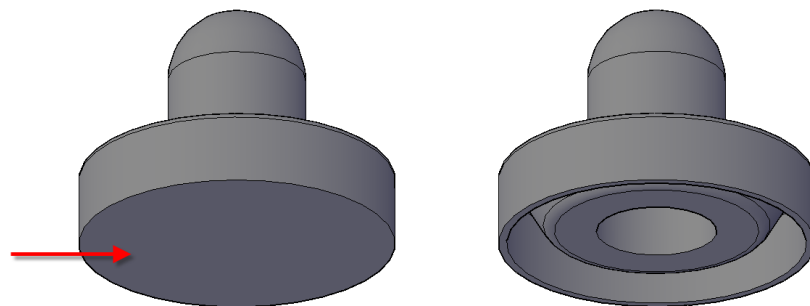
If you had "removed" the other faces, the resulting model would look much different.

2 →

1 →

<p style="text-align:center"><strong style="color:red">face 2 removed</p>

<p style="text-align:center"><strong style="color:red">faces 1 and 2 removed</p>

If you do not select any faces (just press **ENTER** to end object selection), the outside of the model will not look any different, but it will be hollowed out.

After selecting faces to remove from the shell operation, press **ENTER**. The program then prompts you to enter the shell offset distance. Type "**1.5**" and press **ENTER**. You must then press **ENTER** several more times to end the command. The resulting solid now has a thin wall exactly 1.5 units thick.

1.5
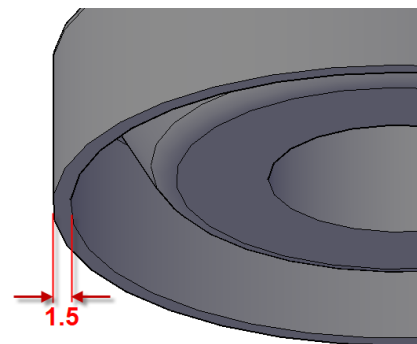
Adding a Second Part

Now you are ready to add a second part, a thin base that will snap into the bottom of the paddle.

Create another cylinder by snapping to the center point and then using the quadrant object snap to snap to the quadrant of the inner ring. To specify the height, drag the cylinder up and then enter a value of **1.5**. With the X-Ray visual style active, you can easily see that the new cylinder lines up properly.

22.3106

Specify base radius or
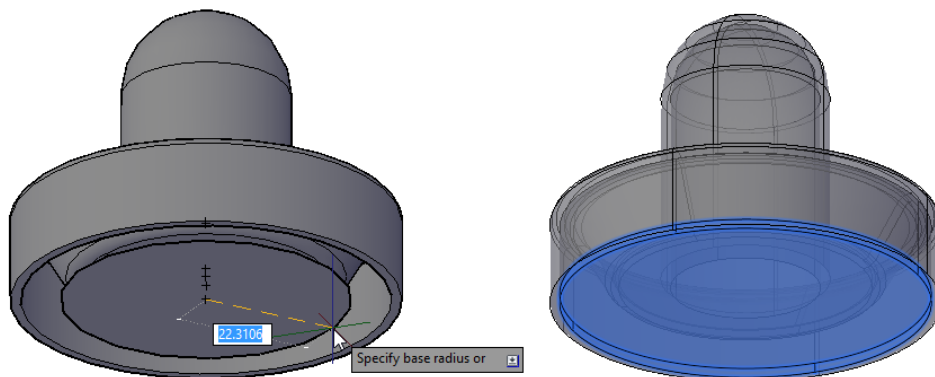
Hiding and Isolating Objects

You can create and modify 3D geometry, regardless of which visual style is active. Visual styles such as X-Ray enable you to see through the model, so in theory, you can snap to points inside other objects. But doing so is still difficult and prone to errors. While you could temporarily move objects out of the way and then later move them back into place to "reassemble" your model, this method is prone to error. Instead, hide objects temporarily.
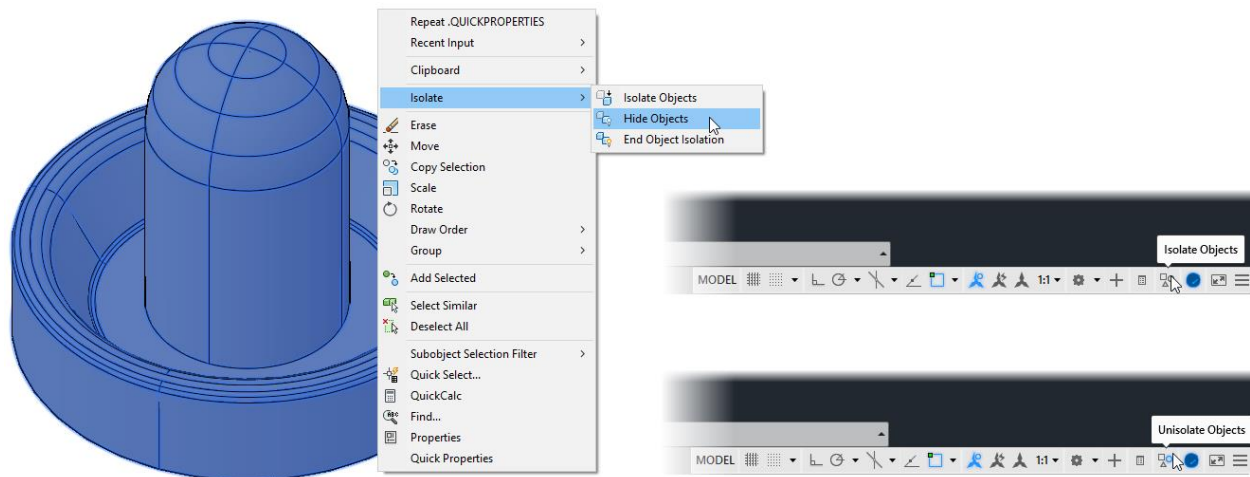
Temporarily hiding objects is very easy. You can simply select the object, and then right-click and choose **Isolate**. Notice that the Isolate tool has three options:

- Isolate Objects – hides everything except the objects you selected
- Hide Objects – hides the objects you selected
- End Object Isolation – makes all hidden objects visible again

You can also access this tool using the **Isolate/Unisolate Objects** button on the **Status** bar. This button provides the same options as those found in the shortcut menu, but it also serves an additional purpose. The button makes it very easy to see whether any objects have been hidden. When the entire button is gray (and its tooltip shows, "Isolate Objects"), no objects have been hidden. When the button includes a blue circle (and the tooltip says "Unisolate Objects"), this indicates that one or more objects have been hidden.

With the upper portion of the paddle hidden, it is then easy to draw the boxes on the top of the cylinder you just created, to add the stiffening ribs to the lower portion of the paddle.

To reveal the hidden objects, simply end object isolation. All objects that were hidden immediately become visible again. To do this, right-click on any object that is still visible, or click the button on the **Status** bar and then choose **End Object Isolation**.

Creating and Editing a Solid Box

You will use the BOX tool (**Home > Modeling > Box**) to create a 3D solid representing the first of several stiffening ribs. By default, the command prompts for the opposite corners (just like the RECTANGLE command) and then for a height.

You could save some clicks if you were to draw the box exactly where you want it to go, but maybe you do not know that yet. After all, this is
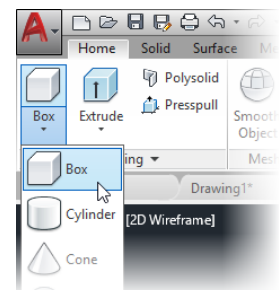
conceptual design, and you may not know where you are going until you get there. For now, just draw the box anywhere in 3D space. Initially, do not worry about its dimensions.

Once you create the box, you can use grips to move the mid-point of the bottom of the box to the center of the cylindrical plate, and then use grips or the Properties palette to resize the box. The resulting box should be 57 units long, 2 units wide, and 1 unit high.
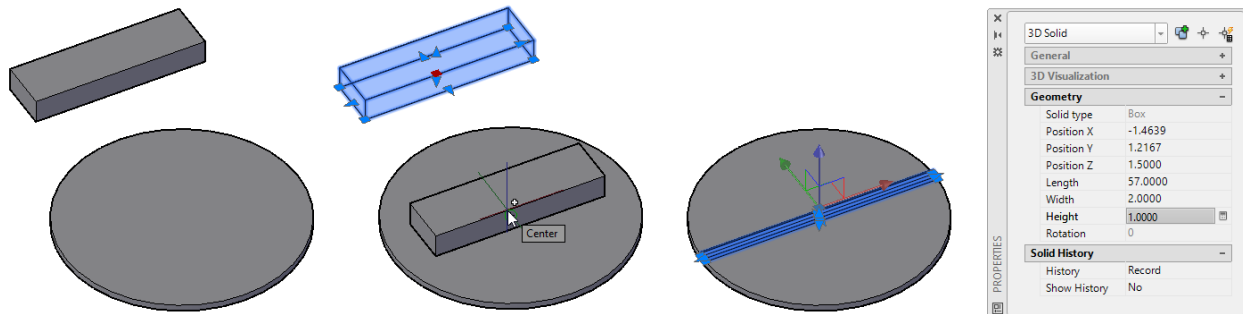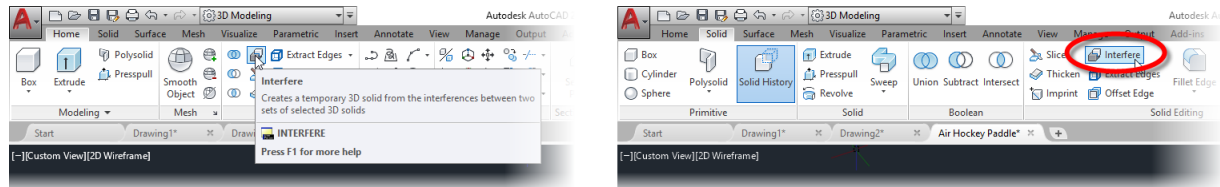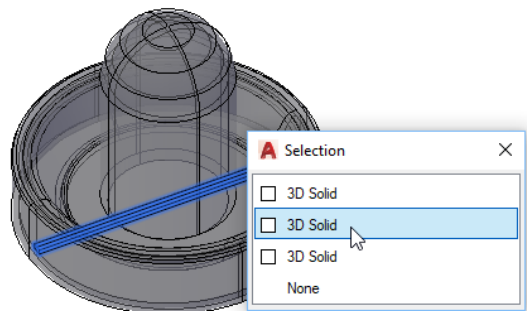

Checking for Interferences

If you look very closely at the ends of the box in a top-down view, you will see that the corners of the box extend past the edge of the cylinder. You will have to zoom in quite a bit to see it. Although it is difficult to see—because the problem is very small—this would prevent you from inserting the bottom piece into the air hockey paddle. You can use the program's interference checking tool to identify and eliminate interferences.

First, "unhide" the paddle. Then, switch to the X-Ray visual style.

You will use the Interfere tool, which is located in the **Solid Editing** panel of both the **Home** ribbon and the **Solid** ribbon.

The Interfere tool is another one of the few AutoCAD commands (like Subtract) that require two selection sets. To check for interferences between the cylindrical air hockey paddle and the box object, select the air hockey paddle as the first selection set and press **ENTER**. Then, select the box and base cylinder as the second selection set. Since selecting the proper objects can be problematic, this is a situation in which it helps to enable Selection Cycling. That way, when you select objects, you can clearly see that you have selected the correct objects.

After completing the second selection set, AutoCAD displays the **Interference Checking** dialog and creates temporary solid objects (shown in red) where any interferences occur. The dialog box offers various tools that enable you to examine the interferences while they are temporarily displayed.
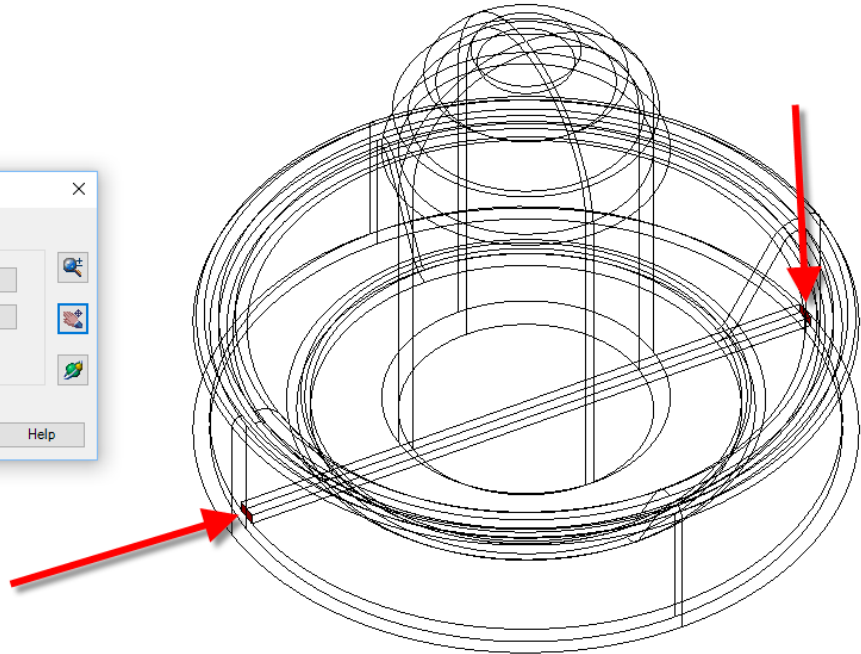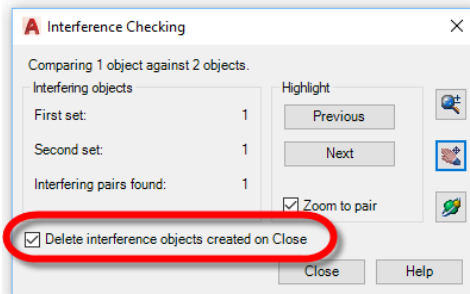
When you close the dialog box, the red interference objects are automatically removed, unless you first clear the **Delete interference objects created on Close** check box. With that option turned off, AutoCAD automatically creates a new solid object representing the interferences. Even though these particular interferences are at opposite ends of the box, they are created as a single solid object. The interferences are just slivers, in this case.

Next, use the **Subtract** tool. When prompted for the first selection set, select the solid box representing the rib and then press **ENTER**. Then, when prompted for the second selection set, select the sliver of the interfering solid. Then press **ENTER** to complete the command. The change is barely noticeable, but could be the difference between a successful design and a costly mistake!
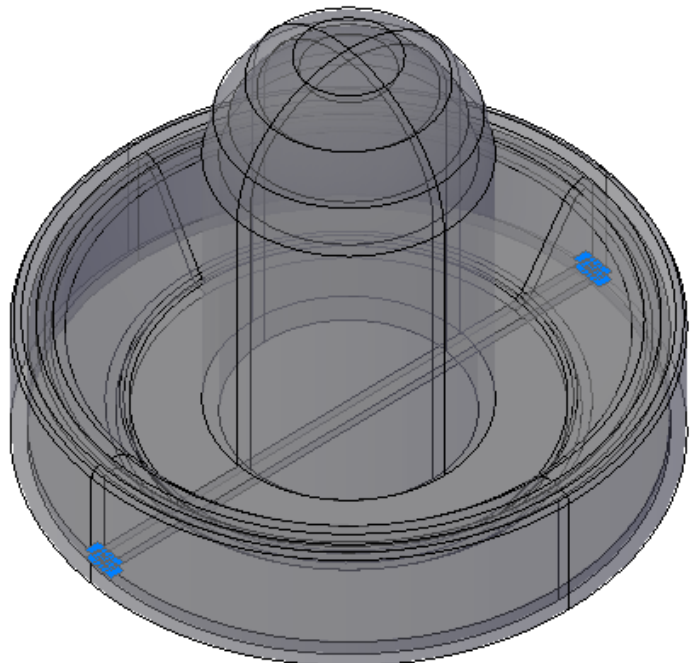
Array the Box

The final task before moving on to the rubber grip is to array the box-shaped rib and then union the ribs to the cylindrical plate. Again, before starting this step, simply hide the paddle again so you can see the plate and the rib.

Use the associative **Polar Array** tool to array the rib. The advantage of creating the ribs as an associative array is that if you need more ribs, you can immediately update the array. Create a total of 4 ribs, with 45 degrees between each rib (which fills 135 degrees).
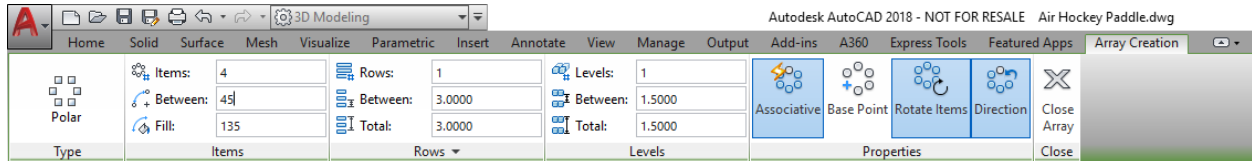

After creating the ribs, you need to perform a Boolean Union to join them together with the cylindrical base. But AutoCAD cannot union the ribs to the base because the ribs are now part of an associative array. So, before you can perform the Boolean operation, you must use the **Explode** tool on the array to reduce it back into individual solids. Then you can perform a Boolean Union to complete the base.
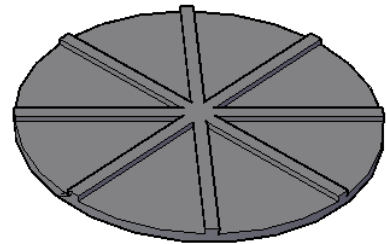
Creating a Surface

Next, you will use surface modeling tools to create a more free-form shape for the rubber grip on the air hockey paddle.
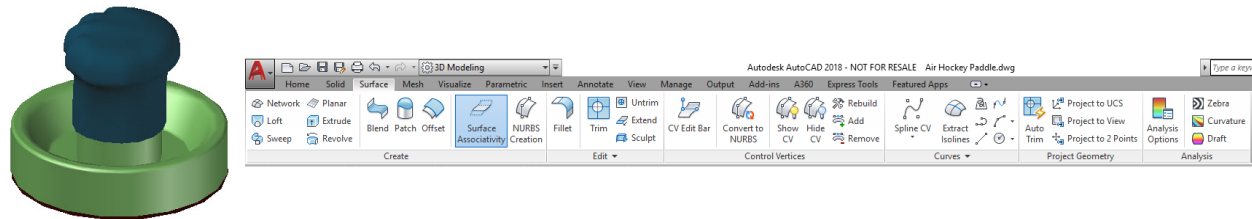

- **Network Surface** – creates a 3D surface in the space between several curves in the U and V directions
- **Loft** – creates a 3D surface in the space between several cross sections
- **Sweep** – creates a 3D surface by sweeping a 2D or 3D curve along a path
- **Planar Surface** – creates a planar surface
- **Extrude** – creates a 3D surface by extruding a 2D or 3D curve
- **Revolve** – creates a 3D surface by revolving a 2D or 3D curve around an axis
- **Blend** – creates a continuous blend surface between two existing surfaces
- **Surface Patch** – creates a new surface or cap to close an open edge of an existing surface
- **Surface Offset** – creates a surface parallel to an existing surface at a specified distance

Also note that when you create a surface, there are two additional controls that you can toggle on and off:

- **Surface Associativity** – maintains an associative relationship between the surface and curves used to create it. If you subsequently modify the underlying curves, the surface updates.
- **NURBS Creation** – controls whether the surface is created as a procedural surface or a NURBS surface. Note that NURBS surfaces, by definition, are not associative.

The objects comprising the base of the paddle were created on their own layer. On a new layer, draw three simple curves: a circle, a line, and a spline. Splines are particularly powerful curves to use when creating surfaces since the shape of the spline can be quickly modified by manipulating its control vertices. Yet the resulting spline remains a nice, smooth, curve, which is generally what you are trying to achieve when modeling surfaces.
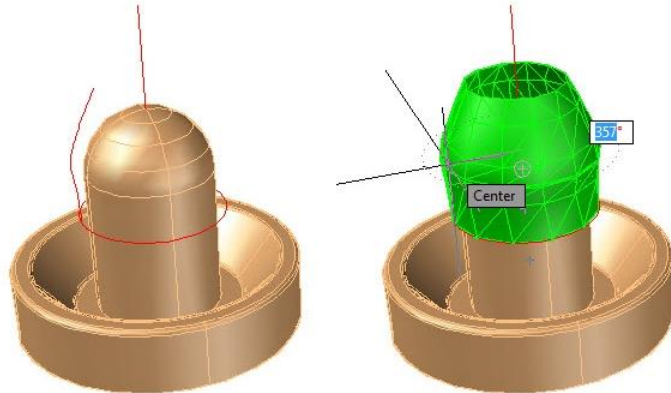
In the **Create** panel, use the **Revolve** tool to create a new 3D surface by revolving the spline 360-degrees around an axis formed by the line.

Editing the Associated Curves

When you create a procedural surface with Surface Associativity enabled, the resulting surface remains associated with the underlying curves. If you subsequently modify the curves, the surface automatically updates.

For example, you can modify the spline by grip editing its control vertices, until you get the shape you want. If necessary, you can add additional control vertices to obtain a finer level of detail.

You may find it helpful to switch to an elevation view in which you can better visualize the spline. You can also use 3D object snaps so that you only snap to control vertices. You can select the Vertex 3D object snap using the **3D Object Snap** tool on the **Status** bar. It is also a good idea to turn off the standard 2D Object Snap mode when you enable 3D Object Snap so that AutoCAD does not get confused.
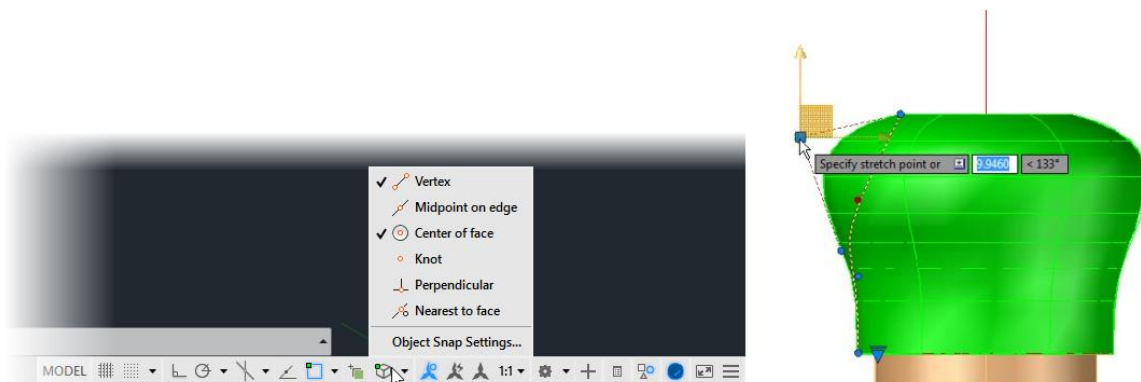
In addition, to make it easier to work in three-dimensional space, use the Move gizmo to ensure that you only move vertices as you intend. Relocate the gizmo to the vertex that you want to move, and then restrict movement to the XY-plane of the spline.

Making a Watertight Surface

Surfaces give you much greater control over the creation of three-dimensional shapes, enabling you to model much more organic-looking objects. But surfaces are abstract objects. Eventually, you will need to convert the surface back into a solid. Before you can do that, however, you must create a *watertight* object—a surface that is completely enclosed.

Right now, the surface is still open at the top and bottom. But you can easily close those openings by using the **Patch** tool to create surface patches.

When you add a surface patch, AutoCAD creates a new surface by fitting a cap over a surface edge that forms a closed loop. AutoCAD first prompts you to select the surface edges to patch, or you can select curves. Once you select the edge, you can also control the continuity and bulge magnitude, and you can use additional guide curves, if necessary, to further control the shape of the surface.
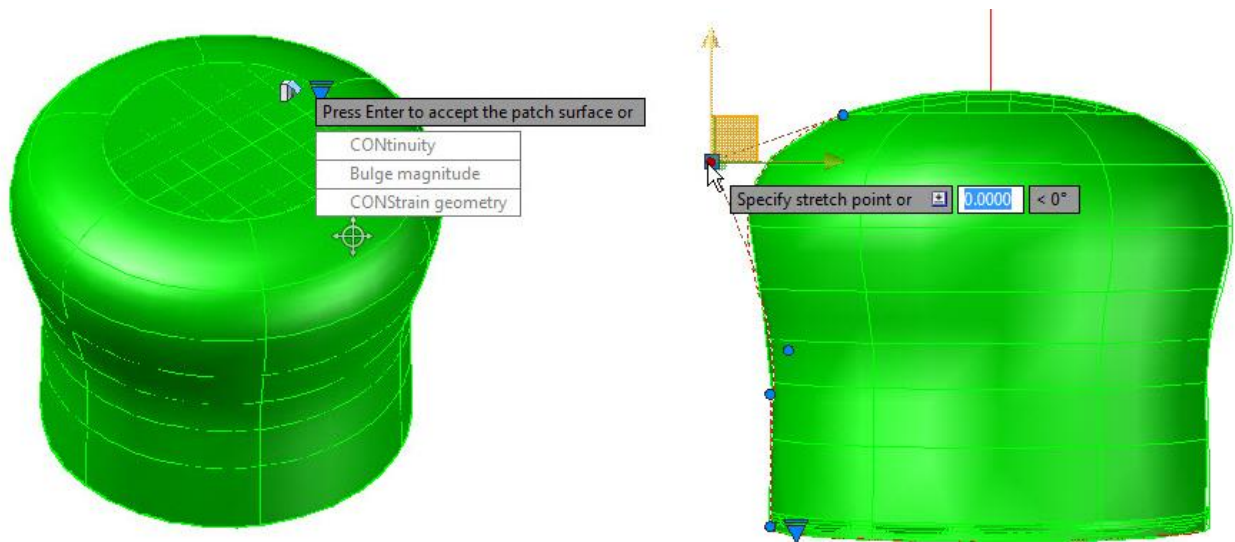
Continuity measures how smoothly surfaces flow into each other. AutoCAD supports three types of continuity:

- **G0 (Position)** – measures location only. If the edge of each surface is collinear, the surfaces are positionally continuous at the edges. Note that two surfaces can meet at any angle and still have positional continuity.
- **G1 (Tangency)** – includes both positional and tangential continuity. With tangentially continuous surfaces, the end tangents match at the common edges.
- **G2 (Curvature)** – includes positional, tangential, and curvature continuity. The two surfaces share the same curvature.

The bulge magnitude determines the roundness of the patch, in other words, how much it bulges. For best results, enter a value between 0 (no bulge, flat) and 1. The default is 0.5.

For the surface patch on the top, specify **G2** continuity and a bulge magnitude of **0.5**. For the surface patch on the bottom, specify **G0** continuity and no bulge. Then, add a surface fillet to round off the intersection between the two surfaces. The result is a watertight surface.

Even after adding those surface patches, you can still modify the spline curve. All of the surfaces will update to reflect the changes you make to the underlying curves.

Integrating Surfaces with Solids

Now you are ready to integrate the more free-form surface with the precision solid. You can sculpt the rubber grip so that it has a more contoured shape to better fit under the palm of your hand. But it must also fit snuggly over the plastic air hockey paddle.

On the **Surface** ribbon, in the **Edit** panel, click the **Surface Sculpt** tool to union the surfaces together and convert them into a solid.

Once you have done that, there is still a bit more work to do. The rubber grip is now a solid mass that shares the same volume as part of the plastic handle of the air hockey paddle. You need to remove the excess material from the rubber grip so that it can slide over the plastic paddle. You can use the Subtract tool to do that, but since AutoCAD deletes the objects being subtracted, first make a copy of the paddle and move it off to the side.

Then, start the **Subtract** tool. Select the rubber grip as the object you want to subtract from, press **ENTER**, and then select the plastic paddle as the object you want to subtract from the rubber grip. The result is a solid object with two parts, the inside mass and the outside mass.

This is referred to as a disjointed solid. On the **Home** ribbon, in the **Solid Editing** panel, expand the SOLIDEDIT split button and use the **Separate** tool to separate the rubber grip into two separate solids. Then, simply erase the inside solid.

Working with Meshes

The rubber grip could also be created using a mesh object, and prior to the addition of surface modeling in AutoCAD, this would have been the method of choice for creating this type of organic shape. But while you can modify meshes by pushing and pulling on mesh faces, edges, and vertices, surfaces are preferable since they can be controlled by adjusting the underlying associative curve.

You could use the following steps to create the rubber grip using a mesh object.

1. Create a mesh box approximating the size of the handle.
2. Edit the mesh box to increase its level of smoothness.
3. Select the upper portions of the mesh and scale them up to create a bulge.
4. Refine the upper portion of the mesh to increase the number of faces.
5. Selectively move individual mesh vertices to create finger impressions.
6. Convert the mesh into a solid.

Step 1   Step 2   Step 3   Step 4   Step 5   Step 6
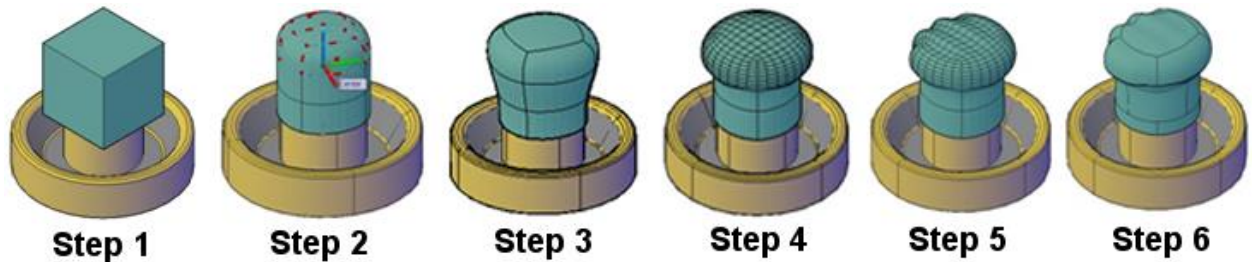
After converting the mesh into a solid, the remaining steps are the same (in other words, subtract the plastic base from the handle, separate the solid, and so on). As you can see, however, there are quite a few more steps involved when working with meshes and less quantitative control over the shape of the handle, although the incorporation of individual finger impressions is perhaps easier to accomplish using a mesh. Obviously, both tools are available, and the choice is yours.

Applying Materials

To convey your design intent and help others visualize your ideas, you can apply materials to your conceptual designs in AutoCAD. The AutoCAD materials library is easily accessible from the Materials Browser palette. The materials library includes more than 700 predefined materials and more than 1000 textures that you can simply drag and drop onto your model. If you do not find the exact material you are looking for, you can create your own using the Materials Editor.

To view materials on your model, switch to the Realistic visual style.
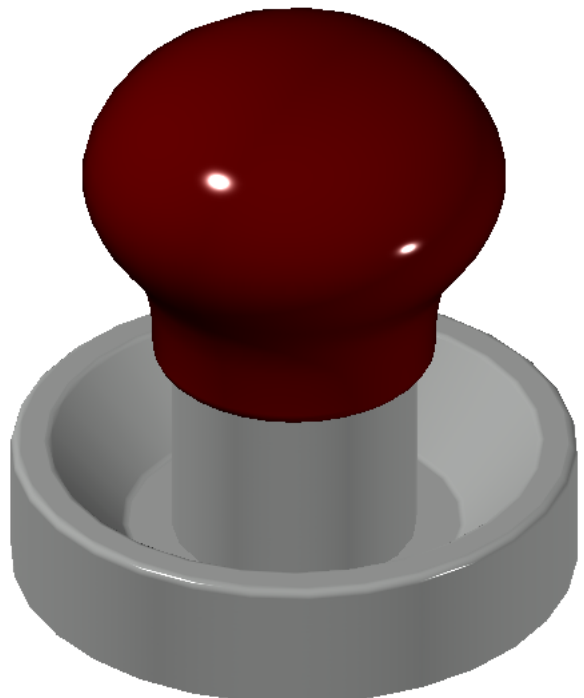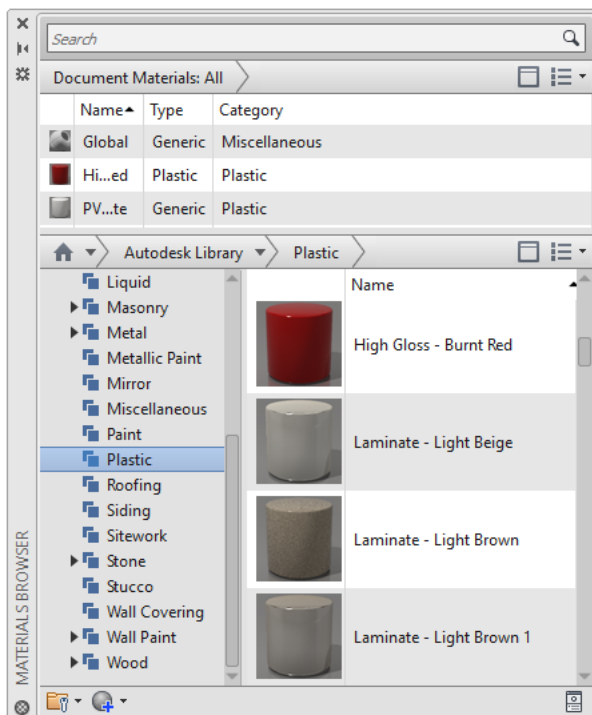
Creating a 2D Drawing from a 3D Model

When you are ready to document your design, you can take advantage of your existing data by using AutoCAD's model documentation tools to create 2D drawings directly from the 3D model.
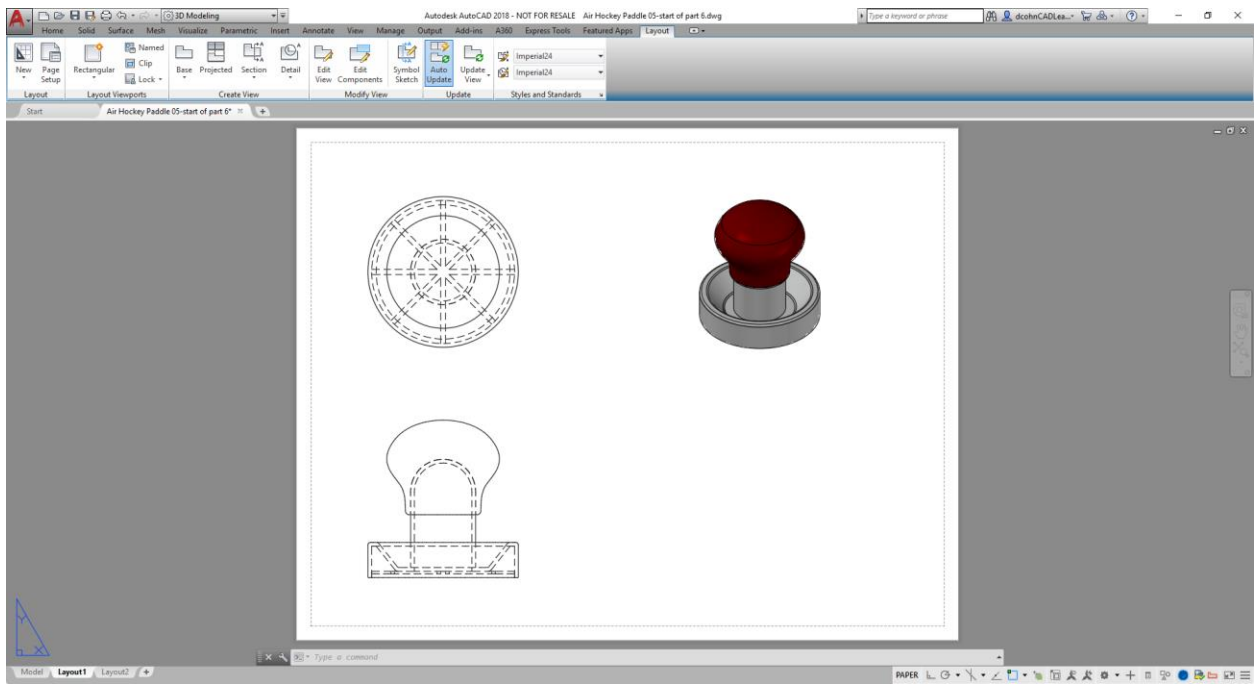
Once you have created a base view and any necessary projected, section, and detail views, you can use familiar annotation tools to add the dimensions and notes necessary to complete your 2D documentation. Yet, any changes you make to the 3D model will continue to be reflected in those 2D drawings.
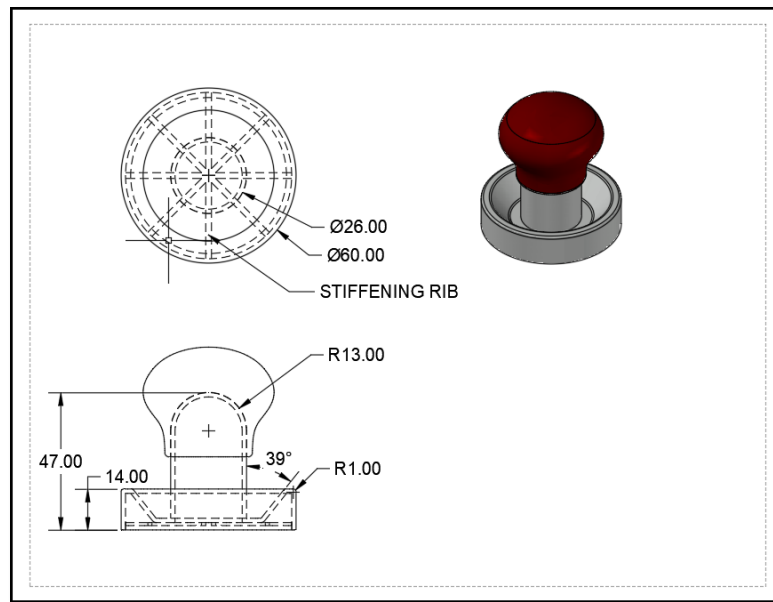
Other 3D Modeling Tools

There are certainly other 3D modeling tools available. Although they did not fit into the workflow used on the air hockey paddle, they should be included in your AutoCAD 3D modeling arsenal.

Press/Pull

The **Press/Pull** tool, located on both the **Home** and **Solid** ribbons, is one of the most versatile tools. It enables you to press or pull bounded areas, and responds differently based on the type of object you select. For example, if you start the Press/Pull tool and click inside a bounded area, you can pull to add volume to a solid.

Before   After
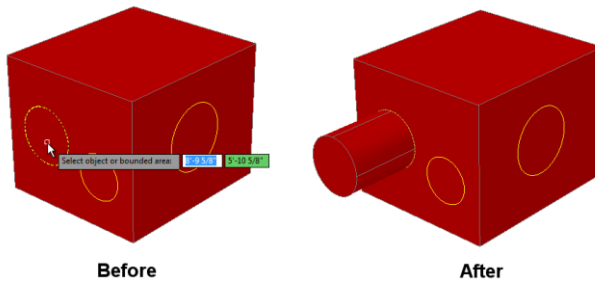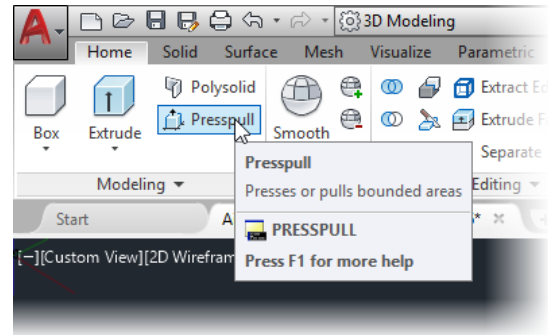
If you click on the object itself (instead of inside the bounded area), you can pull to extrude that object to create a separate solid.

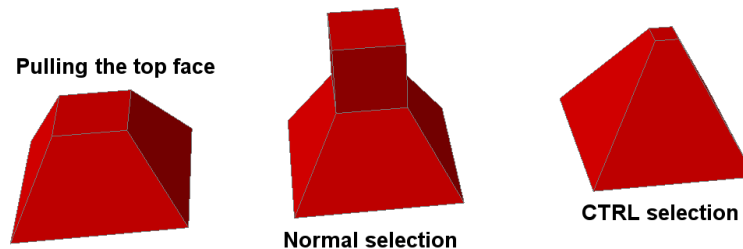Before   After
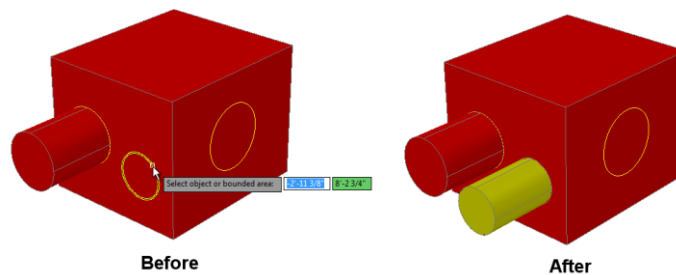
If you click inside a bounded area and press through the solid, you can remove that volume from the solid.

Select bounded area   Press through solid   Results in a hole
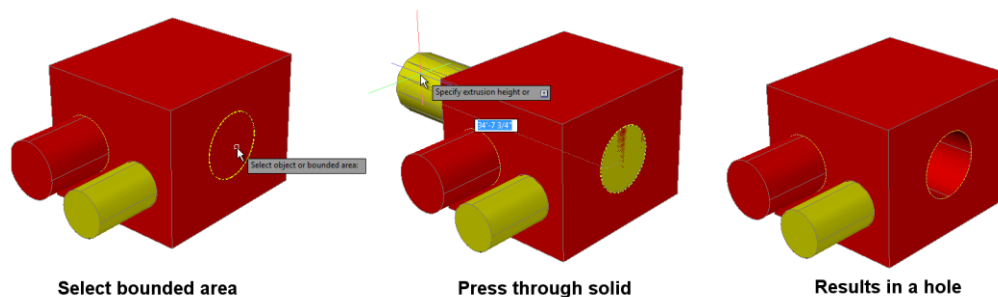
If you select a closed 2D object, you can pull to create a solid. If you select an open 2D object, you can pull to create a surface.

Pulling a closed object   **Results in a solid**   **Pulling an open object**   **Results in a surface**
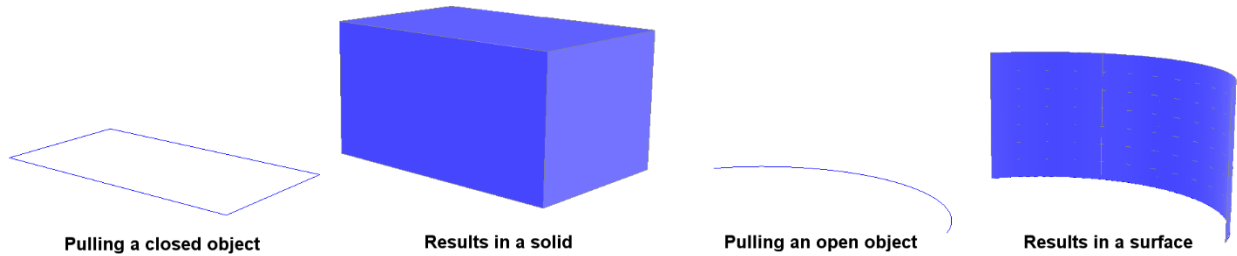
If you click on a face of an object, you can extrude that face, and the face extrudes without affecting the adjacent faces. But if you press the **CTRL** key and then select a face, the face is offset as it extrudes to follow the taper angle of the adjacent sides.

Pulling the top face

Normal selection

CTRL selection

Also note that after you select an object or bounded area, but before you specify the extrusion height, you can choose the **Multiple** option and then select additional bounded areas. Once you are done, you can right-click and then extrude all of those areas at once.
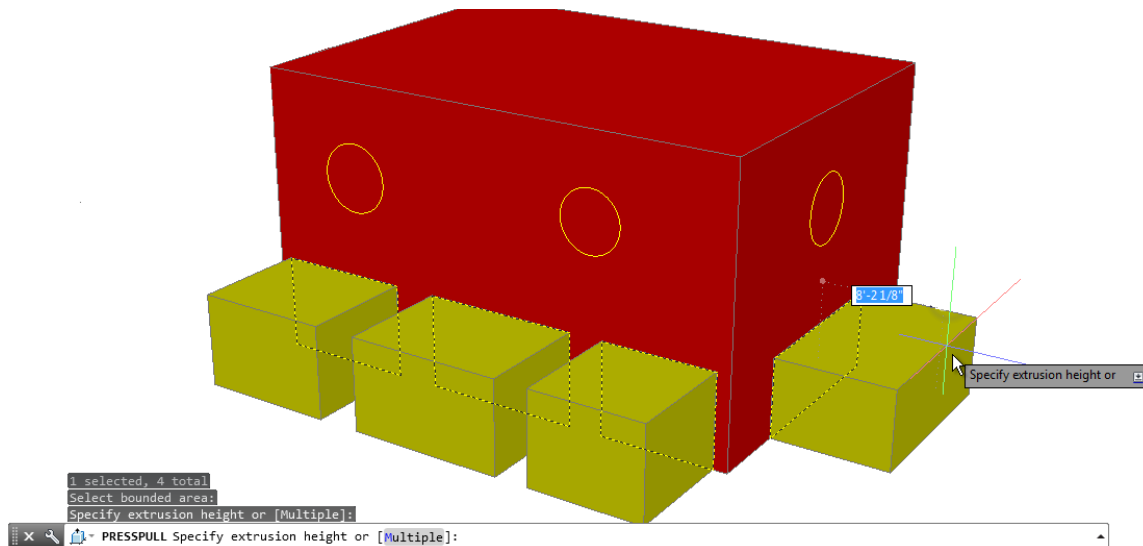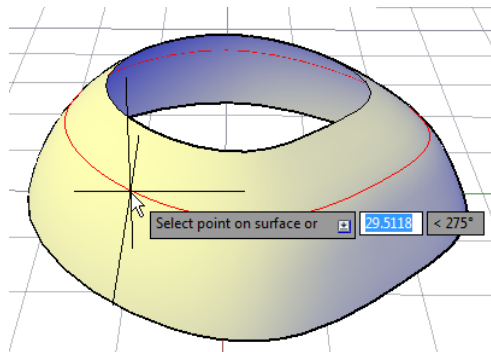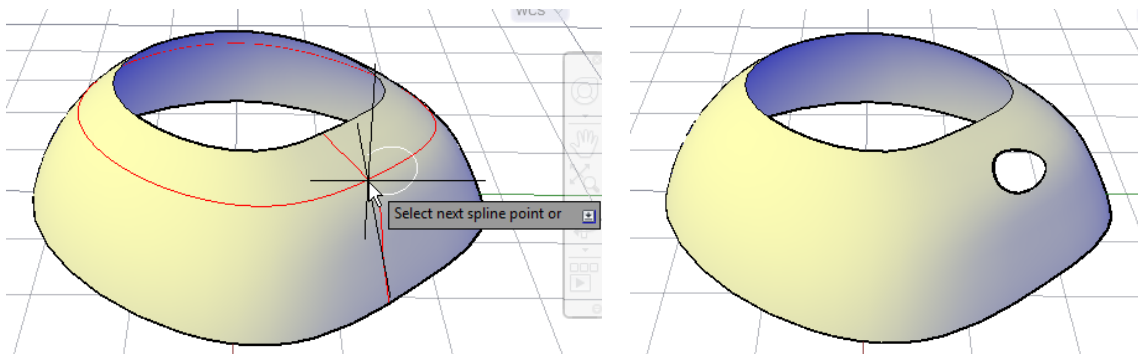
Extract Isolines

The Extract Isolines tool enables you to quickly create curves directly on surfaces and 3D solids so that you can then use those curves to create other objects. This tool is located on the **Surface** ribbon, in the **Curves** panel.
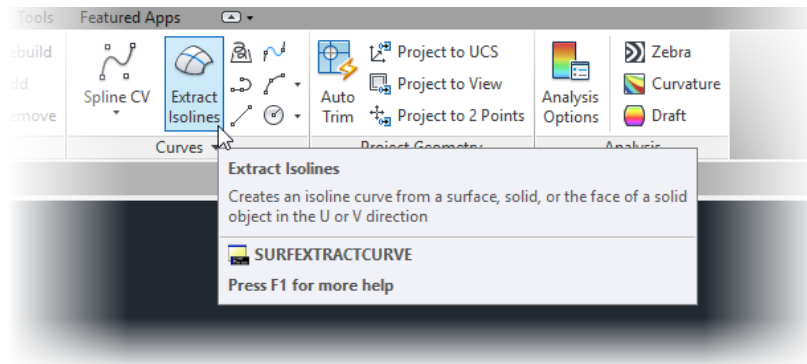
When you select this tool, AutoCAD prompts you to select a surface, solid, or face. If you select a surface, AutoCAD is immediately ready to place an isoline on that surface, and you can simply click to place it. Note that you can also use the direction option to change the direction of the isoline.


If you choose the **Spline** option, you can click to specify spline control points directly on the surface, and then, when you are finished, you can close the spline.

Once you have extracted isolines, you can use them to create other objects. For example, you could use the **Trim** tool (located in the **Edit** panel of the **Surface** ribbon) to trim away the portion of the surface inside the new spline.

You could then use the **Extrude** tool (located in the **Create** panel of the **Surface** ribbon) to extrude the spline, creating a new surface.
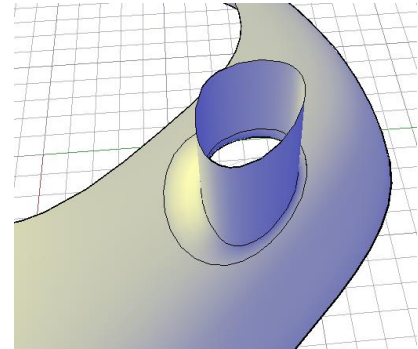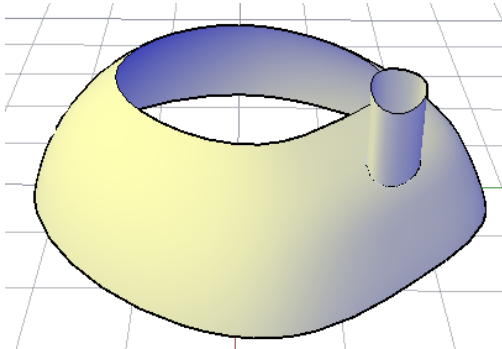

And since the two surfaces form a perfect intersection, you could model additional surface features, such as adding a surface fillet.
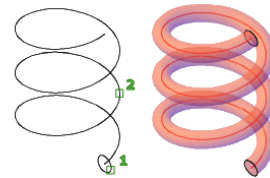
Other 3D Modeling Tools

There are lots of other 3D modeling tools you can use. For example, you can use **Extrude** to extrude a 2D or 3D curve, use **Loft** to create a 3D solid or surface in the space between several cross sections, and the **Sweep** tool to create a 3D solid or surface by sweeping a 2D or 3D curve along a path.
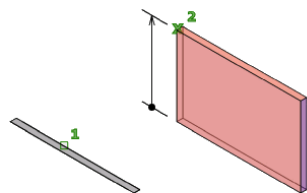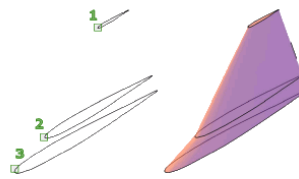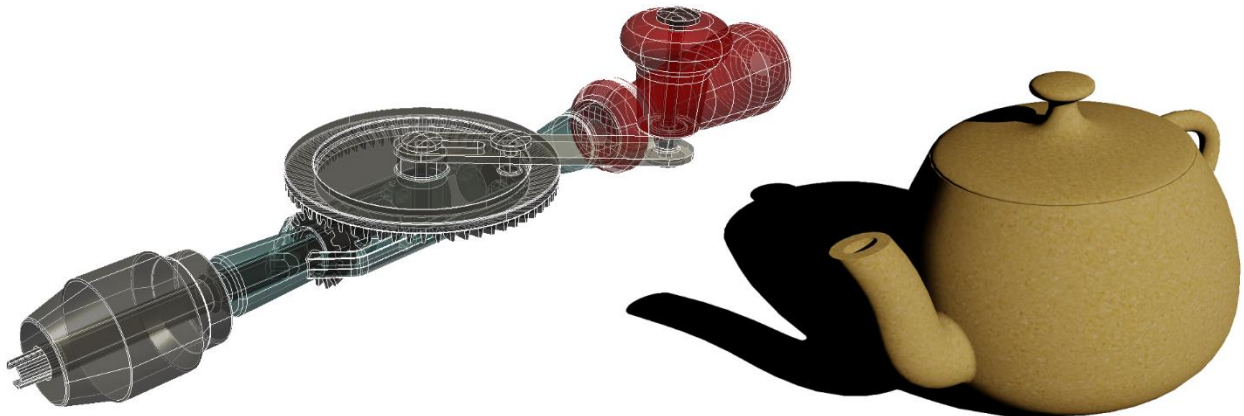

**Extrude**   **Loft**   **Sweep**

Conclusion

Used alone or in combination, you can use these tools, along with other AutoCAD commands to create just about any object you can imagine. These objects were created entirely in AutoCAD.

To learn more…

Do you want to learn more about modeling in 3D using AutoCAD? 4D Technologies offers a CADLearning course for AutoCAD that includes all of the tools covered in this class, plus lessons covering additional 3D modeling tools, as well as nearly every other aspect of AutoCAD—nearly 800 lessons totaling more than 60 hours of video-based tutorials in all.
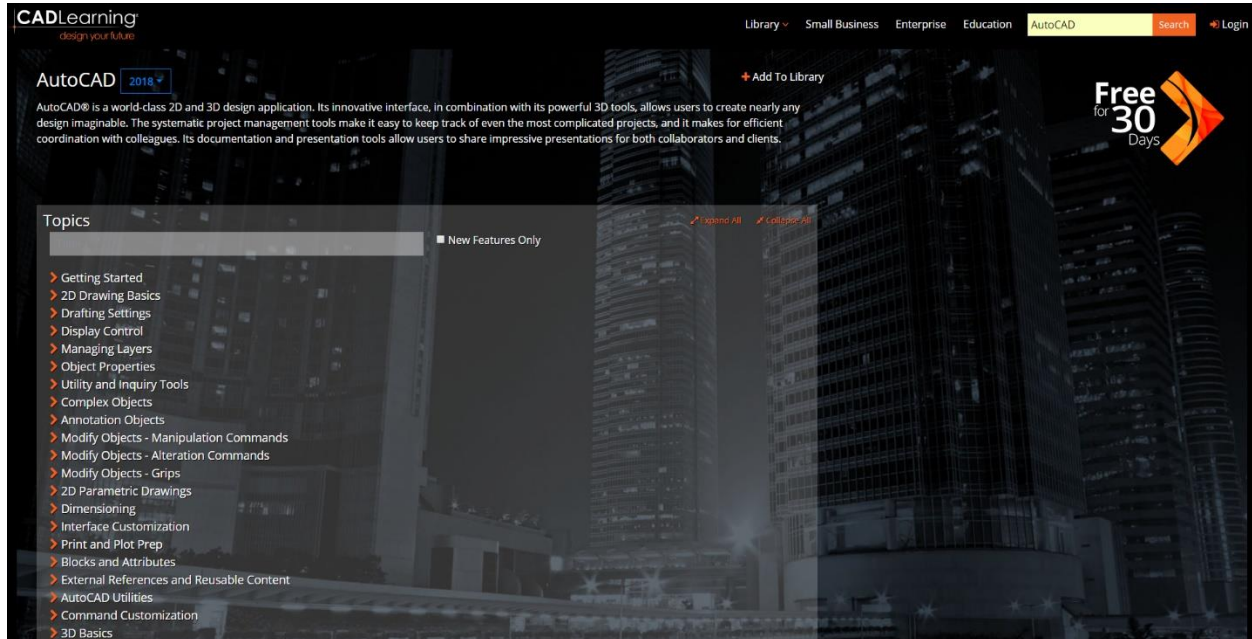
The AutoCAD course is just one of more than 100 CADLearning courses covering a wide range of Autodesk products for architecture, engineering, construction, manufacturing, and media & entertainment. For additional information, please visit www.cadlearning.com.