Magnificent Modeling: 3D Design from Concept to Completion

David Cohn – 4D Technologies

AC3362-P   Modeling in 3D can be easy. Learn how to use new and enhanced AutoCAD® software functionality to apply your 2D AutoCAD experience to the world of 3D modeling. In this class, I will show you how to take your ideas from concept to completion by creating and editing solid objects; creating smooth free-form shapes using meshing tools; and tapping into the power of surface modeling. Take advantage of the new context-sensitive PressPull and surface curve extraction capabilities. I will also explain the differences between history- and non-history-based solids and show how you can use add-ins like Autodesk® Inventor® Fusion software to directly manipulate 3D AutoCAD models. If you thought 3D in AutoCAD was just too hard to do, think again.

Learning Objectives

At the end of this class, you will be able to:

- Create 3D models by combining basic shapes to create more complex objects
- Use meshes and surfaces to create organic shapes
- Modify 3D models using sub-object selection and direct manipulation
- Convert between solids, meshes, and surfaces

About the Speaker

David has more than 25 years of hands-on experience with AutoCAD® and 12 years with Revit® as a user, developer, author, and consultant. He is the technical publishing manager with 4D Technologies/CADLearning, a contributing editor to *Desktop Engineering* magazine, the former publisher and editor-in-chief of *CADCAMNet* and *Engineering Automation Report*, the former senior editor of *CADalyst* magazine, and 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 is always a popular presenter at both Autodesk University and AUGI® CAD Camps.

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Introduction

There's no denying it—it's a 3D world! Everywhere we look. Everything we touch. We're surrounded by 3D objects. All of these real-world 3D objects, even some you might never consider, require conceptualization, design, and production to take them from an idea in someone's head…

…to reality!

So, if our world is 3 dimensional, why do so many of us design in 2 dimensions?

Most designers take the 3D conceptual designs from their imaginations and flatten them into 2D CAD drawings. The 2D CAD drawings are then interpreted by builders or manufacturers to produce 3D objects. This 3D to 2D to 3D design process was accepted without question because, originally, 2D CAD was simply a replacement for traditional 2D drafting.

Fortunately, advances in technology have enabled AutoCAD to evolve from a 2D drafting tool to a 3D conceptual design tool. You can design the way you think—in three dimensions! If you think 3D design in AutoCAD is just too hard, it's time to give it another try. In this course we'll explore new and enhanced AutoCAD functionality that enables you to apply your existing 2D AutoCAD experience to the world of 3D conceptual design. It's time to begin the transition from tradition to innovation!

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 head. You then 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 the mental image of the 3D object.

The process for designing in 3D is similar, but it's actually easier, because you don't have to flatten the image! Instead, skip that step and jump right into identifying basic shapes. But instead of identifying 2D shapes, focus on 3D shapes. Can you identify some of the basic shapes in the air hockey paddle and the house?

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

3D Modeling Workspace

AutoCAD includes four default workspaces: 2D Drafting and Annotation, 3D Basics, 3D Modeling, and AutoCAD Classic. They're accessible from both the status bar and the Quick Access toolbar.
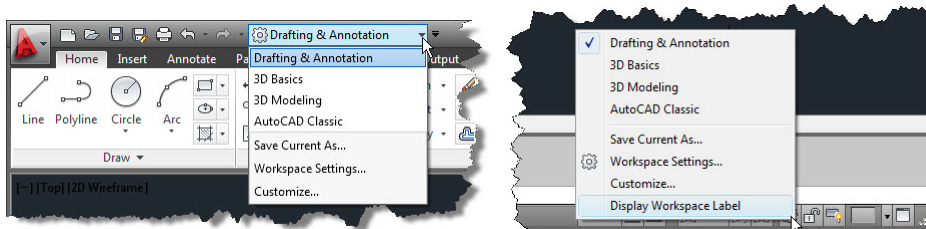
The 3D Modeling workspace provides easy access to all of the 3D modeling and visualization tools via the AutoCAD ribbon, whereas 3D Basics includes just a subset of those tools.
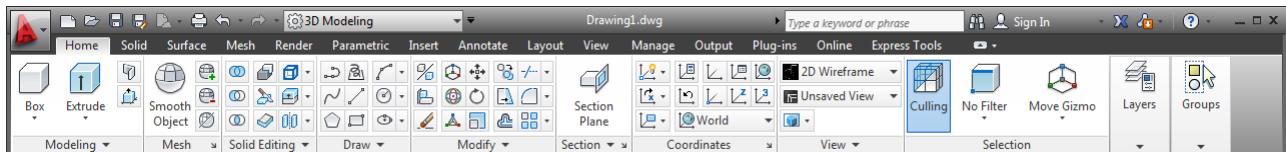
Creating and Editing Solid Cylinders

Solid primitives are essential to creating typical 3D designs with ease and efficiency, and they're the easiest way to start building up a 3D object. The solid primitives are available in the Modeling panel on the Home ribbon of the 3D Modeling workspace. The Box tool is displayed by default but the flyout updates to display whatever tool you used last.

These 3D tools work very much like their 2D counterparts. For example, drawing a cylinder feels 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's 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 exact points or enter specific values (just like drawing a circle). But, if you're in the conceptual design phase and starting with a blank drawing, that may be more information than you know. Fortunately, it's easy to modify the cylinder even after you create it.

When you work in 3D, you'll want to be able to view your model from different angles, not just from the top down in plan view, like we do when
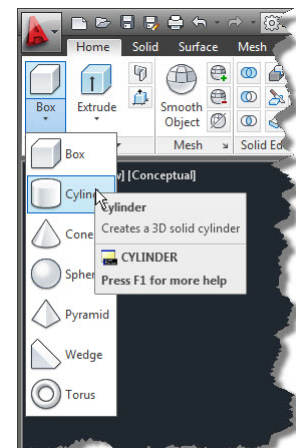
working in 2D. Fortunately, AutoCAD provides some quick and easy tools for viewing and navigating in 3D. If you press the SHIFT key and middle mouse button (the roller wheel), AutoCAD activates its 3D Orbit mode. You can then move around in 3D even while you're in the middle of another command. In addition, the ViewCube updates to help you understand the current 3D orientation. You can also use the ViewCube itself to move around in 3D and switch to isometric, plan, and elevation views.

AutoCAD has had the ability to create solid primitive objects, such as boxes and cylinders, since Release 12. But until AutoCAD 2007, those objects were not easily editable. If you didn't create the object exactly correct the first time, you'd have to erase the object and start over. That's no longer the case. You can now easily edit solid primitives using grips or the Properties palette.

For example, if you select a cylinder, you'll see a number of grips. The square grip at the center of the base enables you to move the cylinder. Selecting one of the four triangular quadrant grips lets you change the radius of the cylinder. And selecting the triangular grip at either end of the cylinder lets you change the height of the cylinder.
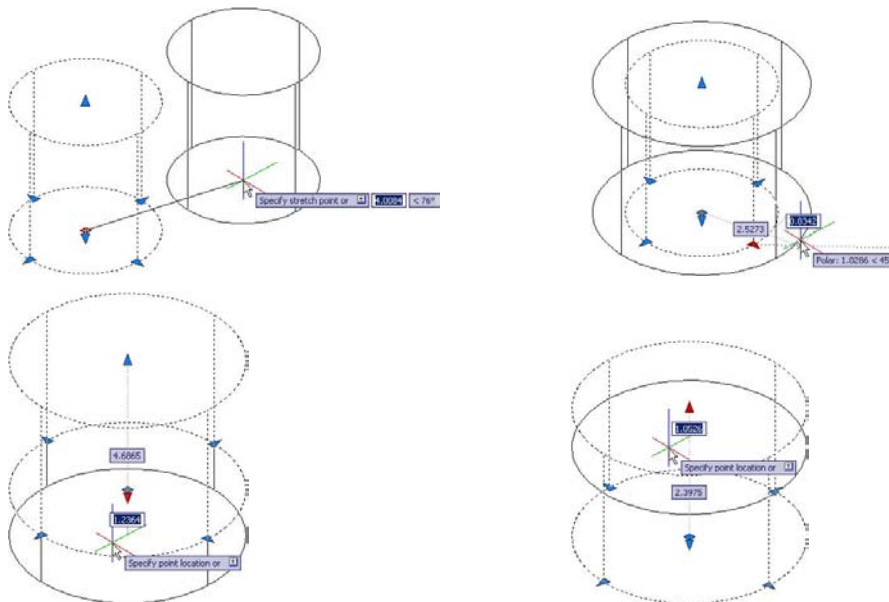

Did you notice that I described that grip as being "at either end" rather than being at the "top or bottom?" That's because you could easily drag the grip at the "top" and drag it so that it was below the other grip, thus becoming the bottom.

If you don't 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
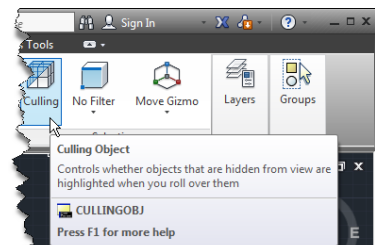
In addition to grip editing, you can select a solid and then change key values in the Properties palette. The properties available will vary depending on the type of solid primitive you select. For example, a box has properties for length, width, and height; whereas a cylinder has properties for radius 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.
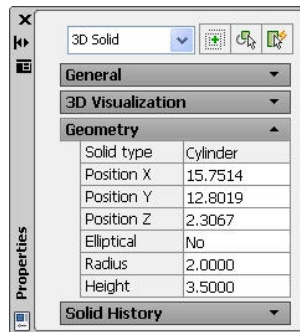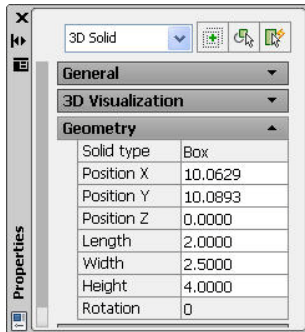
You can also use the Quick Properties palette to modify a solid. To display Quick Properties, first turn it on using the control on the Status bar.
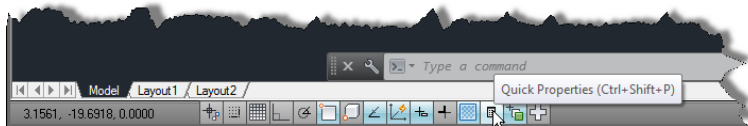

The Quick Properties palette serves the same purpose as the Properties palette, but the Quick Properties palette is customizable. You can configure it so that it shows just the properties you need to view and change, rather than a list of all of the properties associated with a selected object. You can also control where the Quick Properties palette is displayed. By default, when active it displays above and to the right of the cursor, giving you quick and easy access to key properties right when 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, I'll change the radius and height of the cylinder to 30 and 14, respectively.
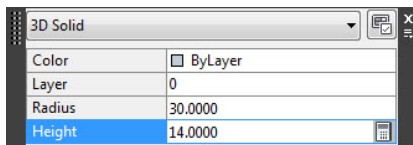

Moving along in the design process for the air hockey paddle, I'll create a second cylinder. Since I want the two cylinders to be concentric, I'll use the Center object snap to snap the center of the new cylinder to the center of the existing cylinder. Remember that a cylinder has two center object snap points: one at the base and one at the top.
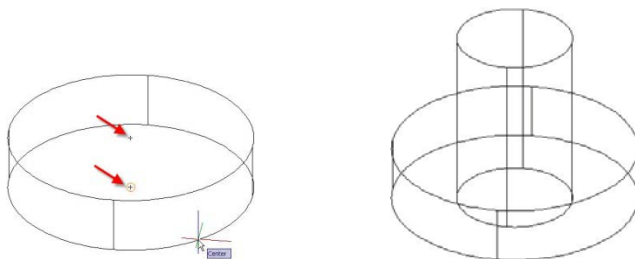
It doesn't matter which one you choose because it's easy enough to change later as your design evolves. But, for now, go ahead and pick the center object snap at the bottom of the existing cylinder and pick rough points to define the radius and height for the new cylinder.

# Creating and Editing Solid Spheres and Cones

A sphere is another type of solid primitive, and you'll find the Sphere tool in the solid primitives drop-down in the Modeling panel of the Home ribbon. Drawing a sphere is much like drawing a circle. You can use the default center/radius option or choose from 3Point, 2Point, and TanTanRadius. I'll use the default center/radius option and place the sphere at 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.
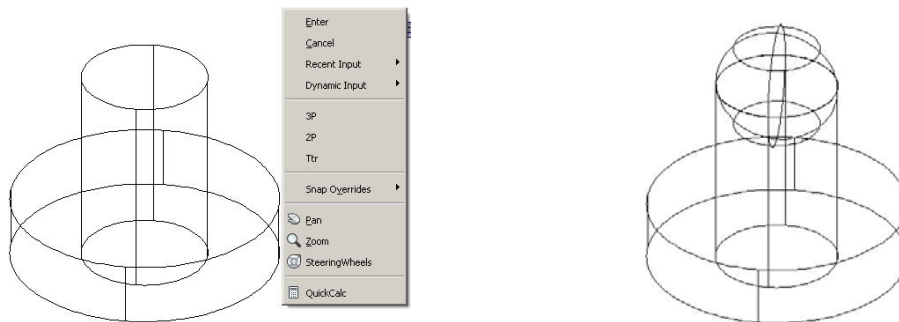
A cone is another type of solid primitive available in the drop-down in the Modeling panel. Options for creating a cone are very similar to those available when creating a cylinder (or a circle). You first specify the location and size of the base of the cone, and you can use the default center/radius option or choose from 3Point, 2Point, TanTanRadius, and Elliptical. Since I want the cone to be located at the center of the cylinder, I'll use the center/radius option.

After specifying the base of the cone, the default option is to enter a value or pick a point to specify the height. However, there are additional options as well, which can be easily accessed by right-clicking and choosing from the shortcut menu. For example, you can pick two points to determine the height and change the orientation of the cone at the same time, or specify a value for the top radius. If you don't specify a top radius, the cone will taper to a point. Again, remember that, like a cylinder, the base doesn't necessarily have to be below the top. For the air hockey paddle, I'm going to create the cone so that it tapers toward the bottom of the paddle.

Since we're going to use the cone to subtract material from the large cylinder, we need to modify the base radius so there will be a rim around the top of the cylinder after the cone is subtracted. Modifying a solid cone is just as easy as modifying a solid cylinder. To ensure a rim of 4 units, just drag the cone's quadrant grips in 4 units toward the center.
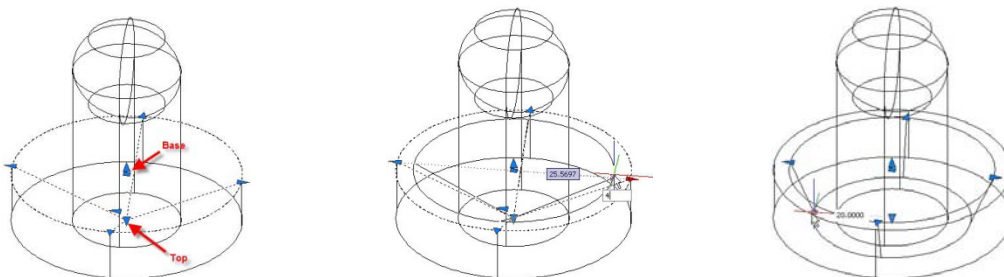
Notice the top of the cone is below the base because of the way we originally created it. AutoCAD doesn't care where the top is in relation to the base. If you look at the properties for the cone you'll see a Base Radius, a Top Radius, and a Height. Widen the top (which in this case is the bottom) of the cone by dragging the radius out 20 units.

Even though AutoCAD doesn't 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.
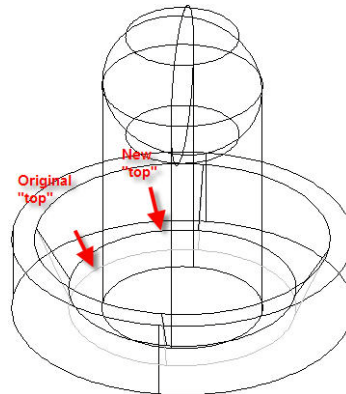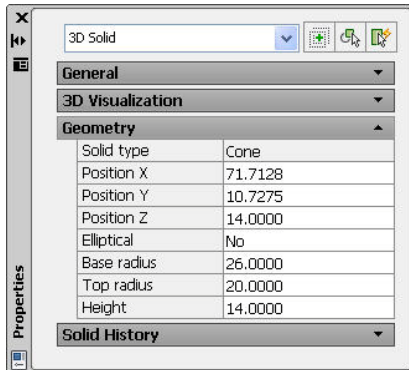

Subtracting and Unioning Solids

The cone represents material we want to remove from the paddle. To add and remove solid objects, we use Boolean tools. You'll find these tools in the Solid Editing panel of the Home ribbon in the 3D Modeling workspace.
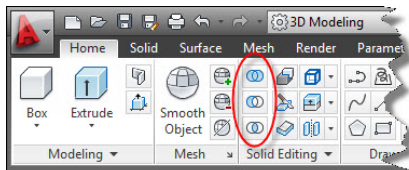
The Union tool joins all of the selected solid objects to form a single composite solid object. The Subtract tool also creates a single composite solid: you first select one or more objects from which other objects will be subtracted and then select the objects you want to subtract from the first set.

When you perform Boolean operations, the order in which you perform those operations is important, because differences in the order can create different results. For example, if you were to union the cylinders and sphere for the air hockey paddle and then subtract the cone, you would end up with a big 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. That's certainly not the result we're looking for. But if we first subtract the cone from the lower cylinder and then union the remaining objects together, we'll get the desired results.
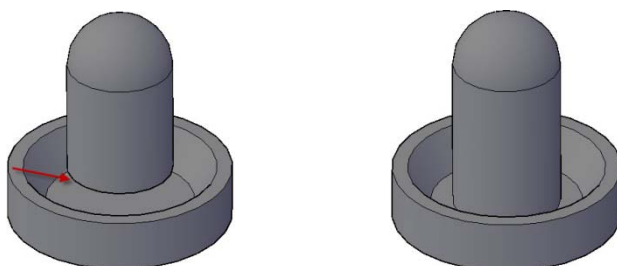

Remember that when using the Subtract tool, you first select the objects you want to subtract from, then press ENTER to finish that selection, and then select the objects you want to remove

from the first set. It's easy to forget if you're new to AutoCAD or haven't used this tool in a while since it's one of the few AutoCAD commands that prompts for two selection sets.

Solid History

In previous versions of AutoCAD, the program tracked each change you made 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. In previous versions of AutoCAD, this variable had its value set to 1 (on). Starting in AutoCAD 2012, the default SOLIDHIST value is now set to 0 (off).

When the Solid History is on, you can press the CTRL 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

composite solid with
Show History set to yes
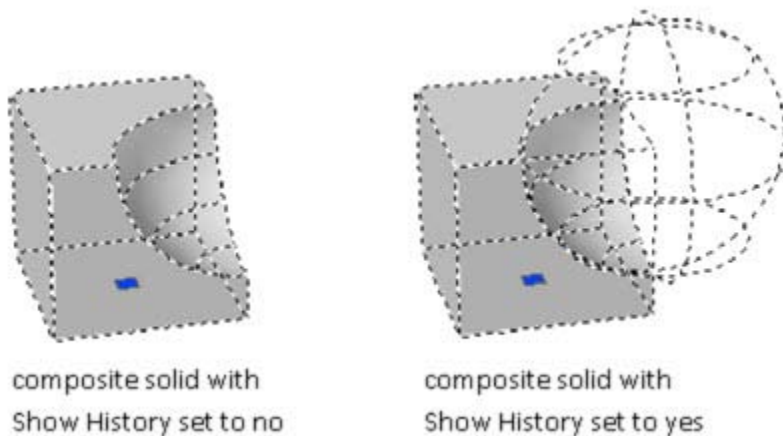
In order for AutoCAD to track the Solid History, you must enable it **before** creating the solid primitives you plan to combine. To change the SOLIDHIST value, you must type the system variable.

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. As we'll also see, some operations remove any existing History regardless of the SOLIDHIST setting.

Autodesk Inventor Fusion

Inventor Fusion provides direct modeling capabilities so that you can quickly make changes to a composite solid regardless of whether the solid includes history or not.

When you select a 3D model in AutoCAD, you can then click on the Edit in Fusion button to immediately open the model in Inventor Fusion. You can then use tools in Fusion to make changes to the model.
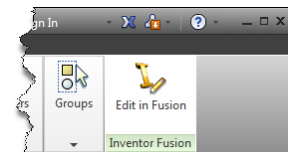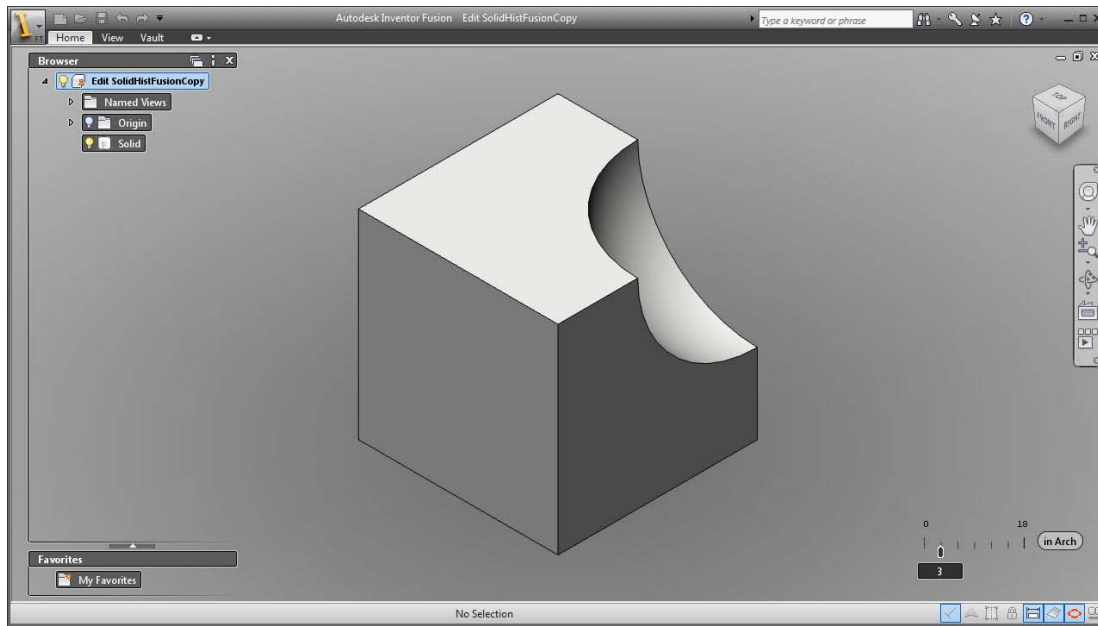
After making changes, you don't need to save those changes. You can simply click the Return to AutoCAD button to close Fusion. The modified model is returned to AutoCAD and you immediately see your changes.

Setting a Visual Style

As you create 3D models in AutoCAD, you'll discover that it can be difficult to visualize the geometry using the default wireframe visual style. Visual Styles were introduced in AutoCAD 2007 and are a powerful replacement for the old "shade modes." AutoCAD now includes ten visual styles: 2D Wireframe, Conceptual, Hidden, Realistic, Shaded, Shaded with Edges, Shades of Gray, Sketchy, Wireframe, and X-Ray. You can access Visual Styles in the View panel of the Home ribbon when the 3D modeling workspace is active.
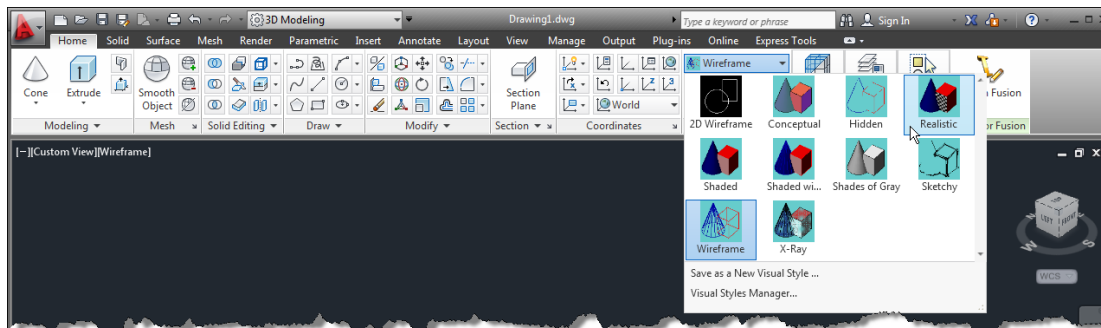
Viewing the model as a wireframe can be helpful if you want to see geometry that would otherwise be obscured by shaded faces. Viewing the model with shading, like with the Conceptual or Realistic visual style, can help you better understand the model. For example, if you view the air hockey paddle in wireframe mode, you can't tell if the cone is an extra object that overlaps the cylinders or if it has been removed from the cylinder. Changing to the Realistic visual style, it becomes clear that the cone has been subtracted.
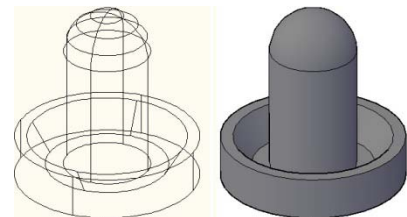
Editing Composite Solids

When you union or subtract solid primitive objects, AutoCAD creates a composite solid. If you click to select any of the primitive objects, the entire composite is selected. If you model without history, you can only modify component solids using Inventor Fusion. If you model with history, however, you can still edit the primitive objects by using the CTRL key. If you hold down the CTRL key as you pass the cursor over the composite solid, AutoCAD highlights each primitive sub-object. You can then click to select the highlighted sub-object. Once it's 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).
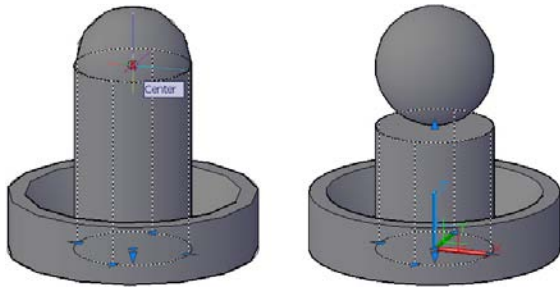

As I mentioned, you can also edit sub-objects using the Properties palette. Notice that even though the cylinder is a sub-object in the composite solid, all the typical cylinder properties are available for editing. Change the radius of the cylinder to 13 and the height to 34.
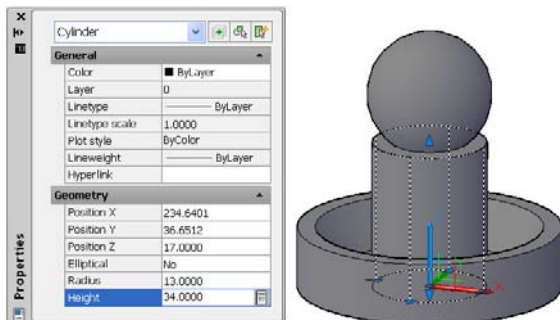
Press the CTRL key and select the sphere. A red, green, and blue gizmo displays at the center of the sphere. Even if the gizmo functionality is turned off, the sphere's center grip displays and behaves as you expect from your 2D experience with grips. Selecting the grip makes it hot and automatically enables the Move editing mode. You can then select an object snap point on an existing object, such as the Center of the top of the cylinder. The sphere automatically moves to that location. Selecting one of the sphere's quadrant grips enables you to modify the sphere's radius. You can snap to the quadrant object snap of the cylinder to automatically match the cylinder's radius, just like when editing a circle in 2D.
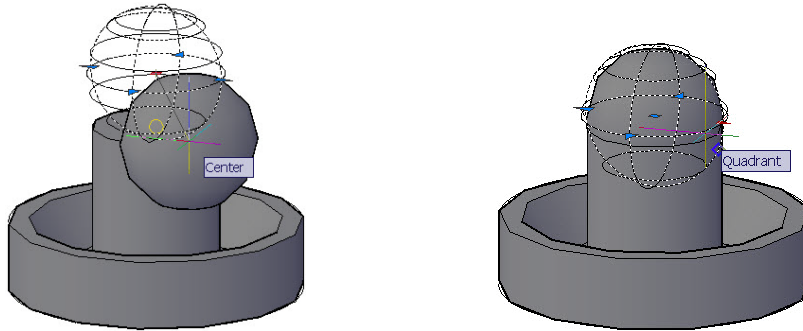
To verify that you snapped to the right point, thus creating a sphere with the correct radius, you can view the sphere properties in the Properties palette. The radius should equal 13 just like the cylinder radius we edited previously.
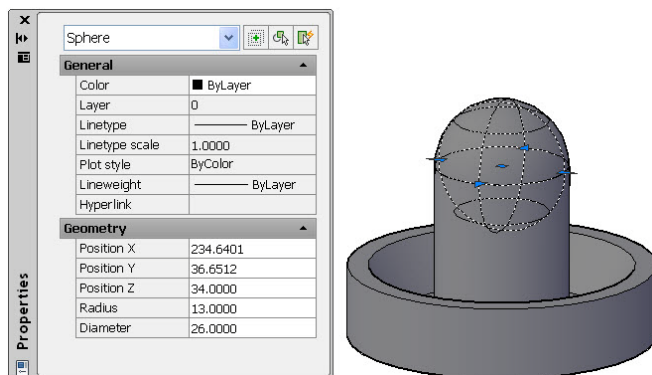


Creating and Editing Chamfers and Fillets

All the objects we've used to create the air hockey paddle are basic primitive shapes: cylinders, a sphere, and a cone. The result conveys the general shape that we want, but the edges are hard. We'll soften the bottom edge by adding a chamfer. You can use the same Chamfer command that you've probably been using for 2D design. When the 3D modeling workspace is active, you'll find the Chamfer tool in the Modify panel of the Home ribbon. It's in the same button flyout as the Fillet tool. Whichever one you used last will be displayed on top.
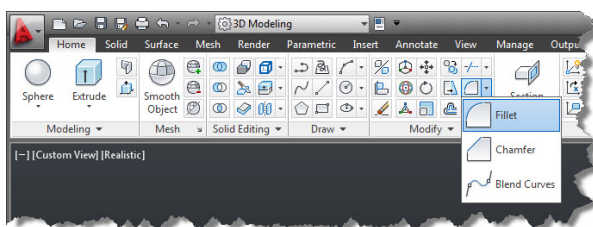
When you launch the Chamfer command, select the edge you want to chamfer. AutoCAD will highlight one of the surfaces adjacent to the selected edge. If it has highlighted the face you want to use as the base surface, you can select OK. If you want to use a different surface as the base, you can choose Next. AutoCAD will cycle through the adjacent surfaces, highlighting each one until you choose. For example, when I select the bottom edge of the air hockey paddle, AutoCAD highlights the circular face. If I choose Next, AutoCAD highlights the cylindrical face. Choosing next again cycles back to the circular face.
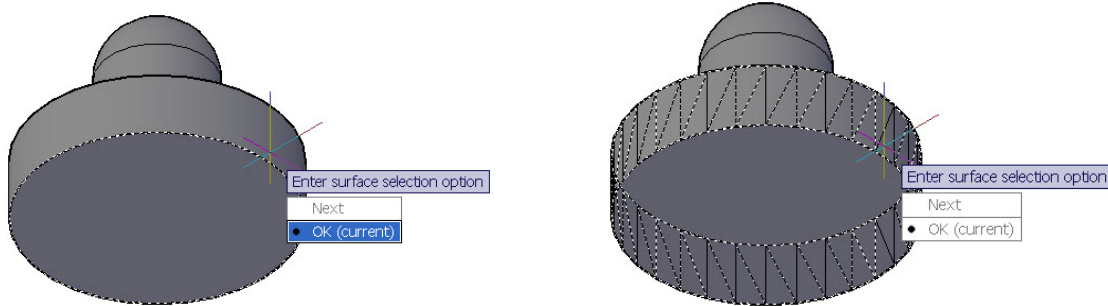
In this particular case it doesn't matter which one you choose as the base surface as long as you enter the appropriate values for the following prompts. Go ahead and accept the bottom circular face as the base surface. AutoCAD will then prompt for the base surface chamfer distance (i.e. the distance that will be trimmed off the edge of circular face). Enter a value of 2. It then asks for the other surface chamfer distance (i.e. the distance that will be trimmed off the edge of the cylindrical face). Even though you selected the chamfer edge at the beginning of the chamfer operation, AutoCAD prompts you to select it again at the end. It seems a little strange but at this point you also have the opportunity to choose the Loop option, which selects all the edges on the base surface. Since the base surface is a circular shape, there is only one edge. Below is a side view of the resulting chamfer.
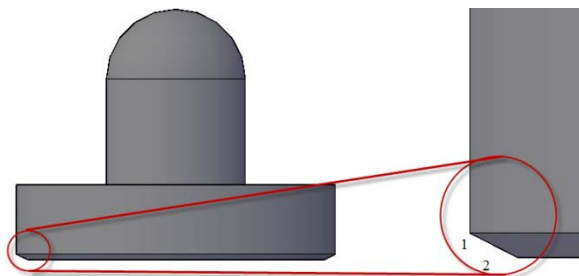

If you had entered the values in a different order (1 for the base distance and 2 for the other distance) or if you had specified the cylindrical face as the base surface, the chamfer would have been 1 unit along the bottom and 2 units along the side.

If you want to create rounded edges on your 3D model, you can use the Fillet tool. Again, this is the same tool you use when working in 2D, and it's located in the Modify panel of the Home ribbon, in the same button flyout as the Chamfer tool. Since you used the Chamfer tool last, you'll have to open the flyout to access the Fillet tool.

After launching the Fillet tool, you select the 3D object you want to fillet. It doesn't matter where you select the object because you'll be prompted to select the edges you want to fillet later. First, however, it prompts for a fillet radius. I entered a value of 1 to create fillets on the air hockey paddle and I selected the two edges highlighted in the first image. If you want to create additional fillets at a different radius, you'll need to repeat the Fillet command (just as you do in 2D design). For the air hockey paddle, I specified a radius value of 3 for the edge as shown in the second image. The resulting model has smooth edges.
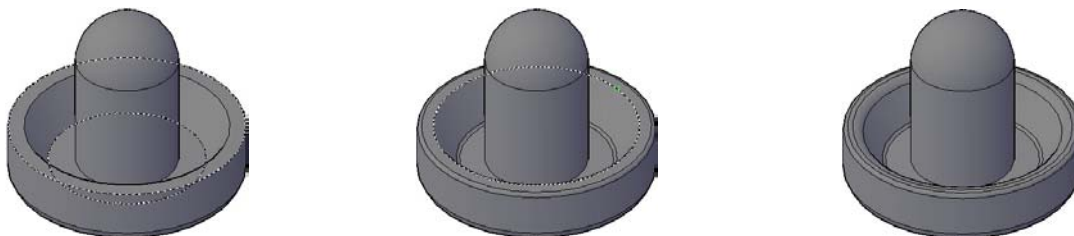
When you add chamfers and fillets to a composite solid, the chamfers and fillets become sub-objects within the composite solid. That means that if History is on, you can select chamfer and fillet subobjects using the CTRL key. As you pass the cursor over a composite solid, the chamfers and fillets highlight. You can then click to select them just as you can with solid primitive subobjects. And like other solid primitives, beginning in AutoCAD 2012, 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.
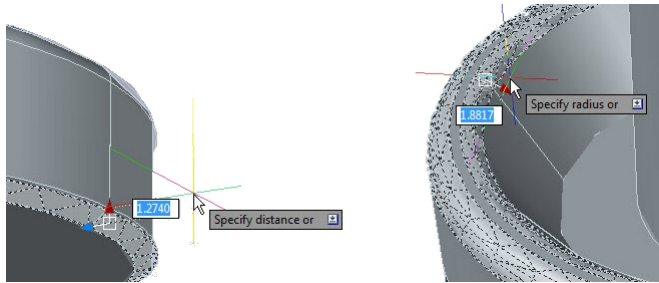
For example, if you press the CTRL key and select the large fillet on the air hockey paddle, the Radius property is displayed in the Properties palette. When you enter a different value, the model dynamically updates to reflect the change.
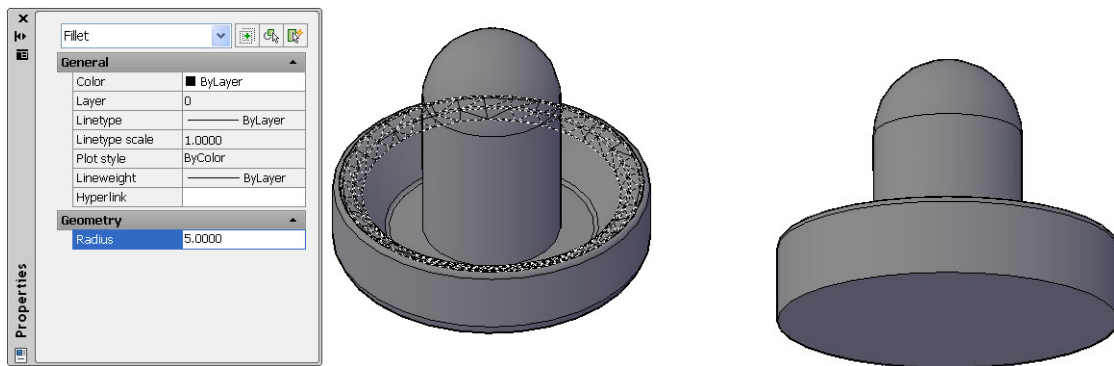

If, when you used the Fillet tool, you selected multiple edges to fillet, all of the fillets created by that single operation are treated as one object. In that case, if you change the radius, both fillets will change.

If you decide that you don't 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 fillets and chamfers using Inventor Fusion.

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 Inventor Fusion, but you can save a lot of editing time if you avoid those operations until later in the design process.
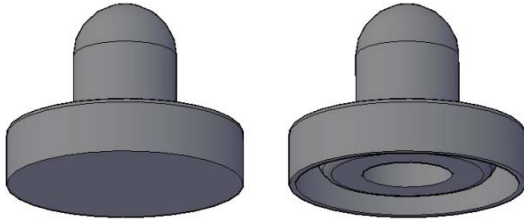
For example, we ultimately want the air hockey paddle to be a thin shell rather than a solid mass and AutoCAD has a Shell tool that is perfect for this type of operation. Unfortunately, the Shell tool is one of those tools that removes the History. For example, in the version of the model with History enabled, when you press the CTRL key and click on the tall cylinder before using the Shell tool, AutoCAD knows it's 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.
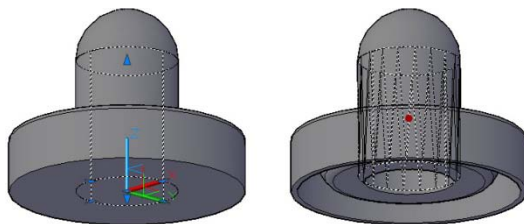
That's fine if those sub-objects are already the right size, but when doing conceptual design, that's rarely the case. You want to be able to modify those objects as long as possible. Once you shell the objects that will no longer be possible. Nevertheless, the Shell tool is handy for creating thin-walled objects such as the air hockey paddle. So, being satisfied with the design thus far, we'll perform the shell operation.

The Shell tool is available on the Solid Editing panel of the Home ribbon. You'll probably have to open the button flyout to find it. The Shell tool is actually an option within the SOLIDEDIT command. (You can't just type SHELL; doing that opens an OS command window.)
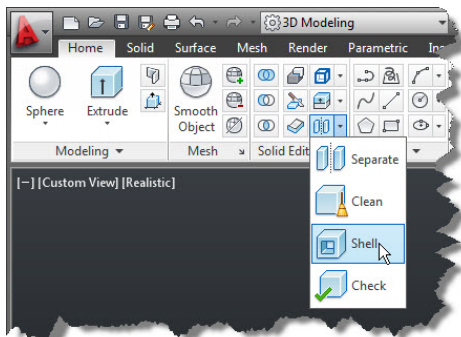

After you successfully launch the right Shell tool, you're prompted to select a 3D solid object and then you're prompted to remove faces. "Remove faces" doesn't mean it's going to remove faces from the object. AutoCAD is giving you the option to exclude faces from the shell operation. For example, when I performed a shell on the air hockey paddle, I selected the circular face at the bottom of the cylinder, resulting in an open bottom.
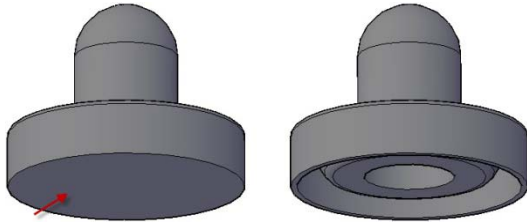
If I had "removed" the cylindrical face instead, it would look like this:
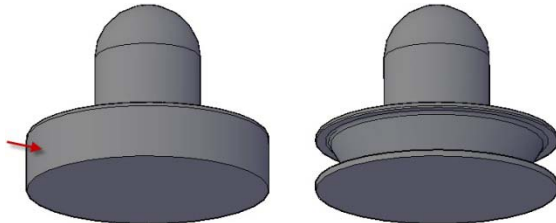
If I had removed the circular and the cylindrical faces, it would look like this:
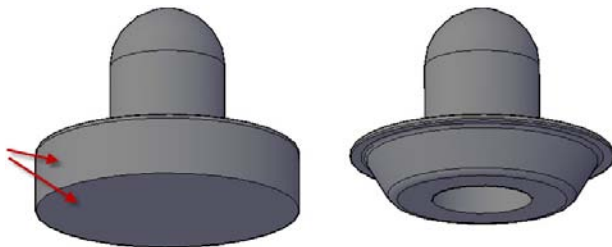

If you don't select any faces (just press ENTER to end the selection set), the outside of the model won't look any different but it will be hollowed out. After you specify which faces, if any, to remove from the shell operation, you must enter the shell offset distance. I entered a distance of 1.5 for the air hockey paddle.
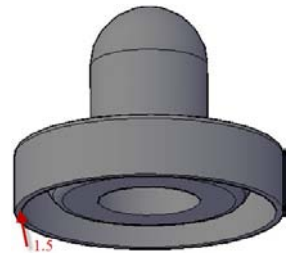

Adding a Second Part

Now I'm ready to add a second part, a thin base that will snap into the bottom of the paddle.

I'll 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, I dragged the cylinder up and entered a value of 1.5. With the X-Ray visual style active, I can easily see that the new cylinder lines up properly.
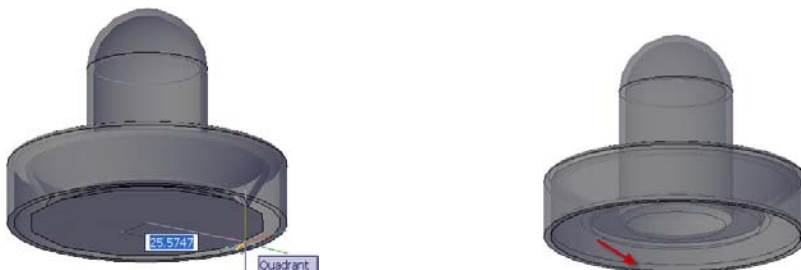
Hiding and Isolating Objects

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. In previous versions of AutoCAD, you could solve this problem by temporarily moving objects out of the way and then moving them back into place to "reassemble" your model once you were done editing. But you no longer need to do this. You can just hide objects temporarily.
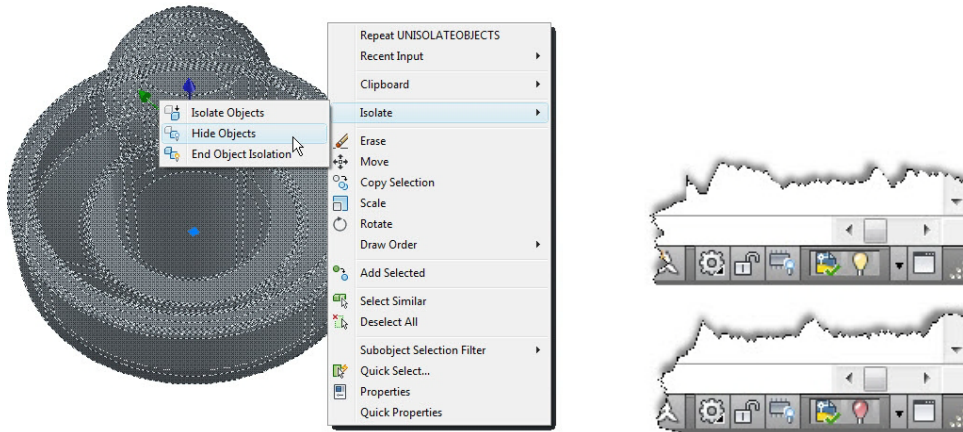
Temporarily hiding objects is very easy. You can simply click to select the object and then right-click and choose Isolate. Notice that the Isolate tool has three options:

- 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 light bulb in the button is yellow, no objects have been hidden. When the light bulb is red, this indicates that one or more objects have been hidden.

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

When you want to see the other objects, you can simply end object isolation, and any objects that were hidden 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

The BOX command is one of the many AutoCAD tools that enable you to create 3D solids. You can access the Box tool from the Modeling panel of the Home ribbon. By default, the command prompts for two opposite corners (just like the RECTANGLE command) and then for a height.

You could save some clicks if you draw the box exactly where you want it to go, but maybe you don't know that yet. After all, this is conceptual design and you may not know where you're going until you get there. For now, just draw the box anywhere in 3D space.
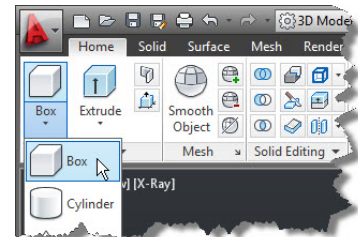
Once you create the box, you can use grips to move the box, and grips or the Properties palette to resize the box.
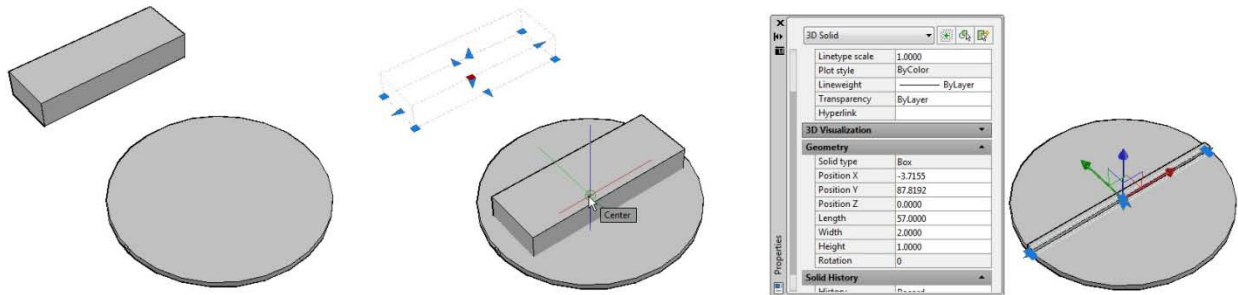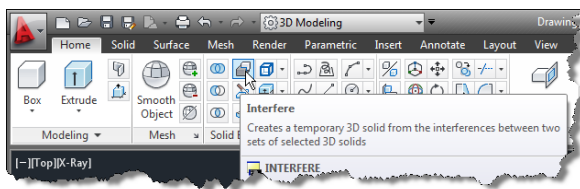
Checking for Interferences

If you look very closely at the ends of the box, you'll see that the corners of the box extend past the edge of the cylinder. You'll have to zoom in quite a bit to see it. It's tough to see, because the problem is very small. But this would pose a serious problem when we try to insert the bottom plate into the air hockey paddle. We can use interference checking to identify and eliminate interferences.
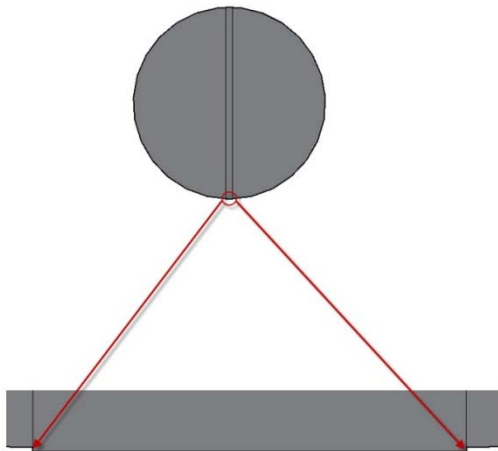
First, we'll "unhide" the paddle. Then, we'll use the Interfere tool, which is located in the Solid Editing panel of the Home ribbon.

The Interfere tool is one of the rare AutoCAD commands (like Subtract) that require two selection sets. To check for interferences between the air hockey paddle and the cylinder and box objects, we select the air hockey paddle as the first selection set (pressing ENTER to end the selection). Then select the box and cylinder as the second selection set. Thanks to Selection Cycling, it's easy to see that we've selected the correct objects. After completing the second selection set, AutoCAD displays the Interference Checking dialog box 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're temporarily displayed.
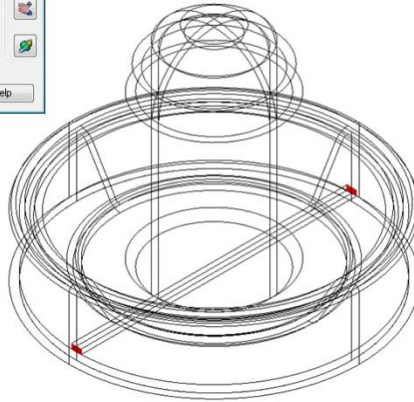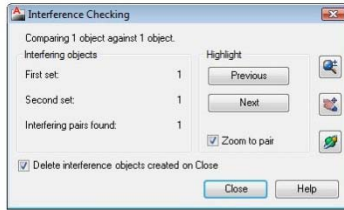
When you close the dialog box, the red interference objects are automatically removed unless you clear the "Delete interference objects created on Close" check box. With that option turned off, AutoCAD automatically creates a new solid object representing the interferences. Even though these particular interferences are at opposite ends of the box, they're created as a single solid. The interferences are just slivers in this case.
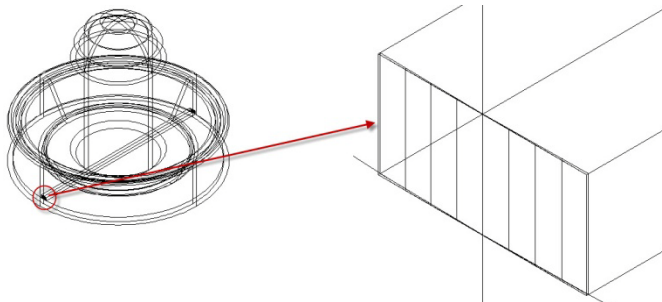
Using the subtract tool, we can first select the box and then (after ending that selection set), select the sliver of the interfering solid. The change is barely noticeable but could be the difference between a successful design and a costly mistake!
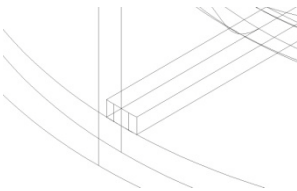
Arraying a Box

Our 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.

We can use the new associative Polar Array tool to array the rib. The advantage of creating the ribs as an associative array is that if I need more ribs, I can immediately update the array. I'll create a total of 4 ribs, with 45 degrees between each rib (which fills 135 degrees).
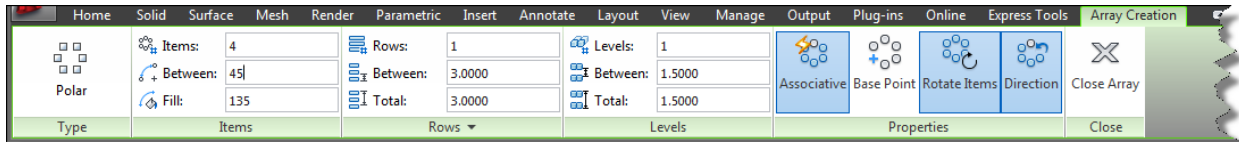
Once I've created the ribs, I want to perform a Boolean Union to join them together with the cylindrical base. But AutoCAD cannot union the ribs to the base because the ribs are now part of an associative array. So before I can perform the Boolean operation, I must Explode the array to reduce it back into individual solids. Then I can perform a Boolean Union to complete the base.
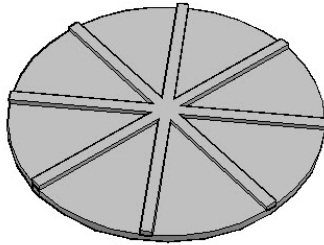
Creating a Surface

Next, we'll use some of the surface modeling tools introduced in AutoCAD 2011 to create a more free form shape for the rubber grip on the air hockey paddle.
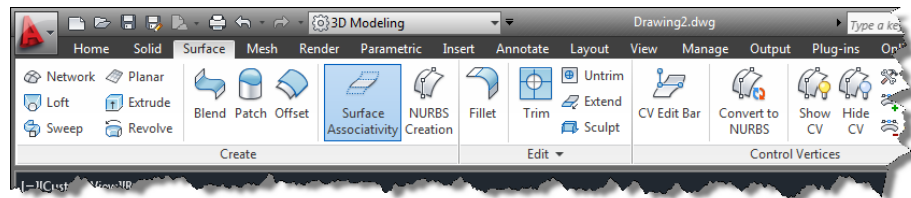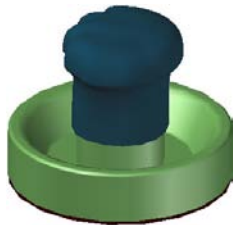
The 3D Modeling workspace includes a Surface ribbon tab. The Surface ribbon has six panels including the Create panel. Tools you can use to create surfaces include:

- **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 notice 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.

I've moved the objects comprising the base of the paddle to their own layer. Then, on a new layer, I drew 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 we're trying to achieve when modeling surfaces.
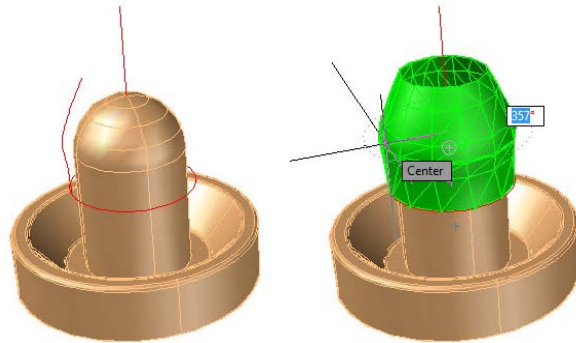
I'll use the Revolve tool to create a new 3D surface by revolving the spline 360-degrees around an axis formed by the line.

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, I can modify the spline by grip editing its control vertices, until I get the shape I want. If necessary, I 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. I also find it very helpful to use 3D object snaps so that I only snap to control vertices. You can select the Vertex 3D object snap using the 3D Object Snap tool on the Status bar. It's also a good idea to turn off the standard 2D Object Snap mode when you enable 3D Object Snap so that AutoCAD doesn't get confused.
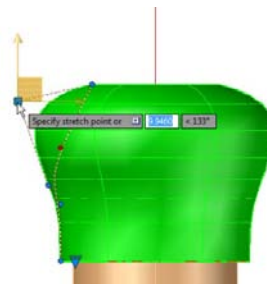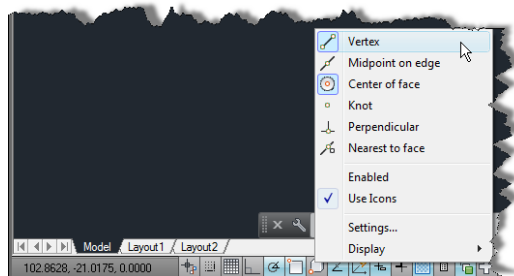
In addition, to make it easier to work in three-dimensional space, I also find it extremely helpful to use the Move gizmo to ensure that I only move vertices as I intend. I relocate the gizmo to the vertex that I want to move, and then restrict my movements to the XY-plane of the spline.

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, we're going to want to convert this object into a solid. Before I can do that, I need to create a watertight object—a surface that is completely enclosed.

Right now, my surface is still open at the top and bottom. But we can easily close those openings using 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 use additional guide curves, if necessary, to further control the shape of the surface.
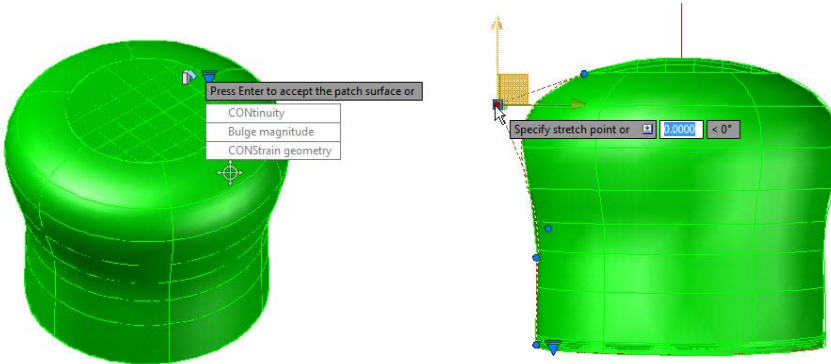
Continuity measures how smoothly surfaces flow into each other. AutoCAD supports three types of continuity:

- **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, I'll specify G2 continuity and a bulge magnitude of 0.5. For the surface patch on the bottom, I'll specify G0 continuity and no bulge. Then I'll add a surface fillet to round off the intersection between the two surfaces. When I'm finished, I'll end up with a watertight surface.

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

Integrating Surfaces with Solids

Now I'm ready to integrate the more free-form surface with the precision solid. I sculpted the rubber grip so that it would be contoured to better fit the shape of a hand. But it must also fit snuggly over the plastic air hockey paddle.

Since I made the surfaces watertight, I can use the Surface Sculpt tool to union all of the surfaces together and convert them into a solid.

Once I've done that, I still have 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. We need to remove the excess material from the rubber grip so that it can slide over the plastic paddle.

First, we'll make a copy of the paddle and move it off to the side (because AutoCAD deletes the object being subtracted). Then, I'll use the Subtract tool and select the rubber grip as the object I want to subtract from, press ENTER, and then select the plastic paddle as the object that I want to subtract from the rubber grip. The result is an object with two parts, the inside mass and the outside mass.


This is referred to as a disjointed solid. I can then use the Separate tool, located in the Solid Editing panel on the Home ribbon to separate the rubber grip into two separate solids. Then, it's a simple matter to 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.

For those who do wish to work with meshes, however, I'll briefly outline the steps one might use 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.
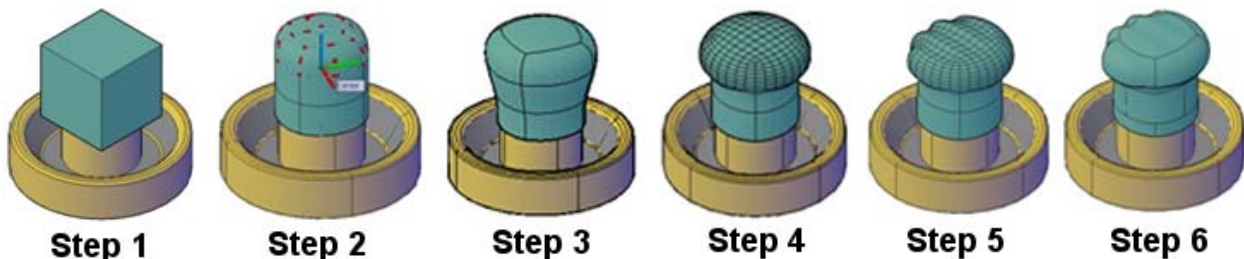
After converting the mesh into a solid, the remaining steps are the same (in other words, subtract the plastic base from the handle, separate the solid, and so on). As you can see, however, there are quite a few more steps involved when working with meshes and less quantitative control over the shape 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 don't find exactly the material you're looking for in the materials library, you can create your own using the Materials editor.

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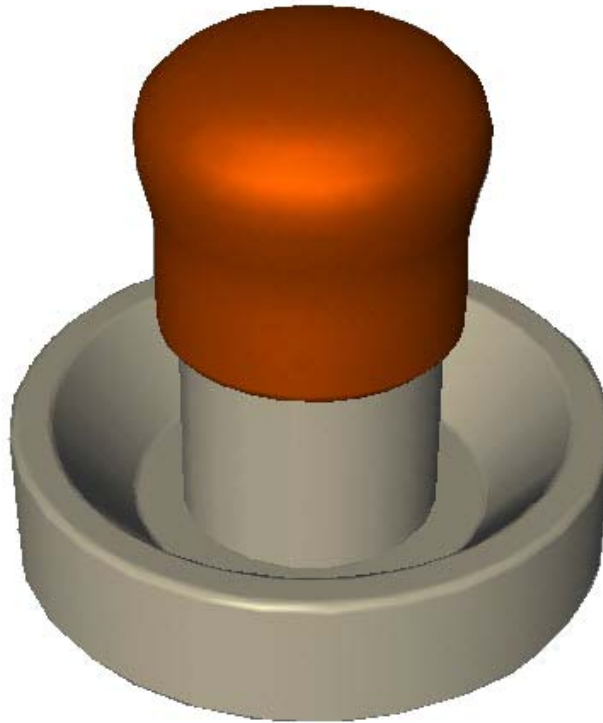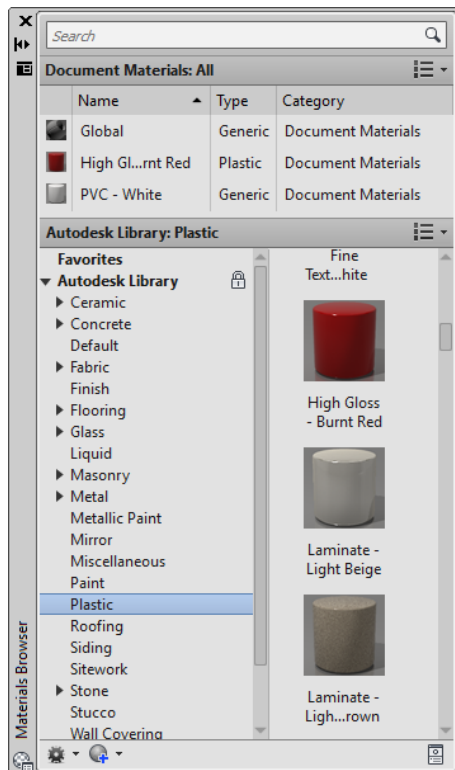
Creating a 2D Drawing from a 3D Model

When you're ready to document your design, you can take advantage of your existing data by creating 2D drawings directly from the 3D model, by using AutoCAD's model documentation tools.
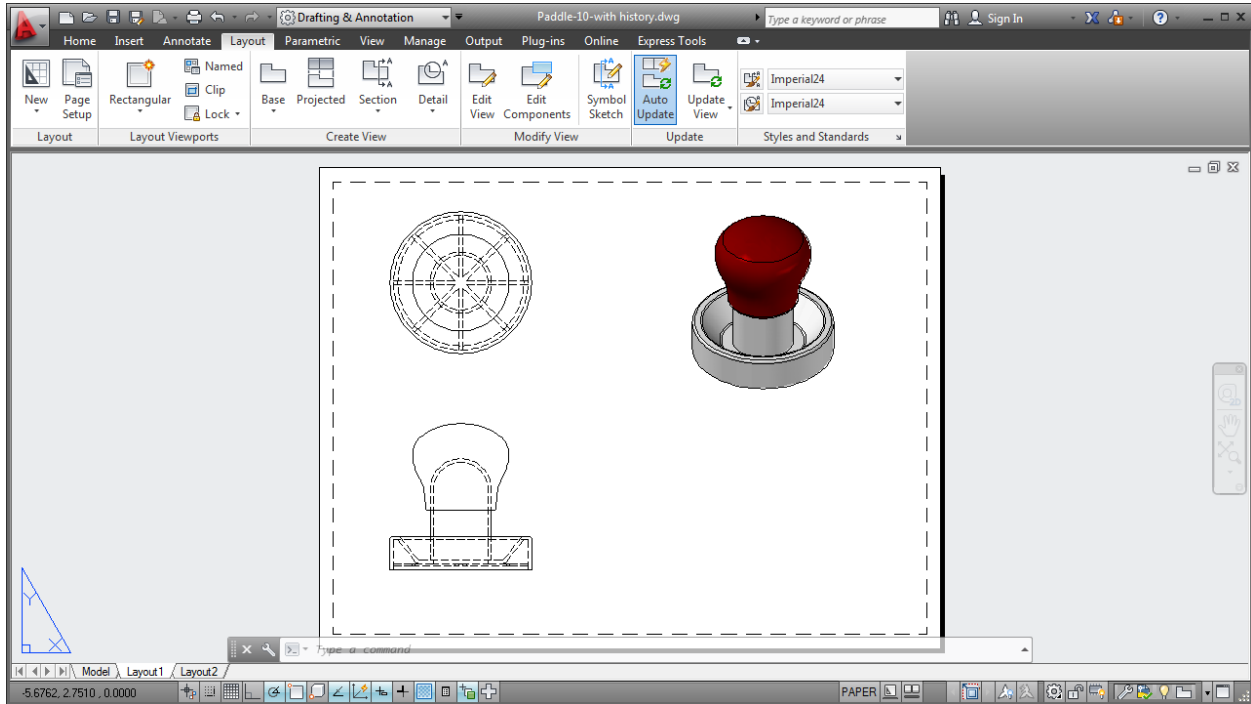
Once you've created a base view and any necessary projected, section, and detail views, you can use familiar annotation tools to add the dimensions and notes necessary to complete your 2D documentation. Yet any changes you make to the 3D model will continue to be reflected in those 2D drawings.
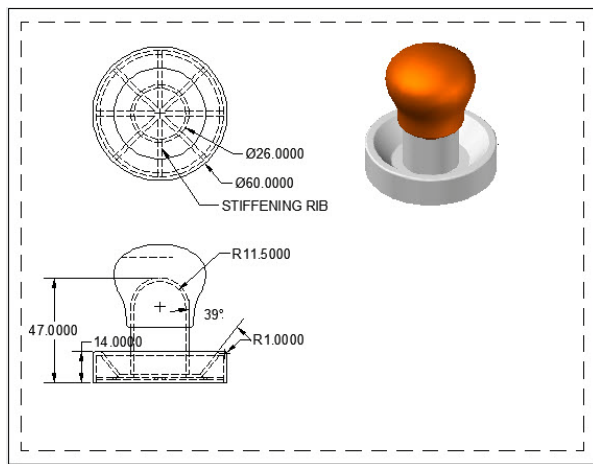
To learn more about creating 2D documentation from your 3D models, refer to my course AC3374: Smarter Drawings: Intelligent Model Documentation Made Easy.

Other New and Enhanced 3D Modeling Tools

There are several other new and enhanced 3D modeling tools that simply did not fit into the workflow used on the air hockey paddle. Nevertheless, they should be included in your AutoCAD 3D modeling arsenal.

Press/Pull

Press/Pull has been around for a while, but it's been enhanced.

The Press/Pull tool is located in both the Modeling panel of the Home ribbon and the Solid panel of the Solid ribbon.
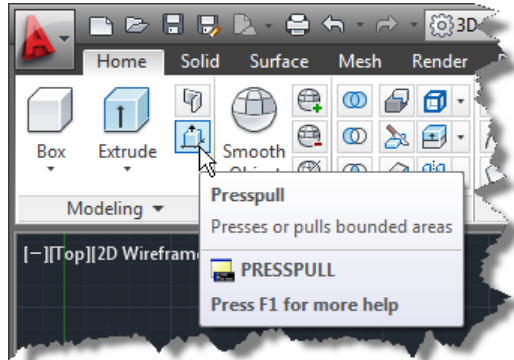

The Press/Pull tool lets you press or pull bounded areas and responds differently based on the type of object you select. For example, if you start the Press/Pull tool and click inside a bounded area, you can pull to add volume to a solid.
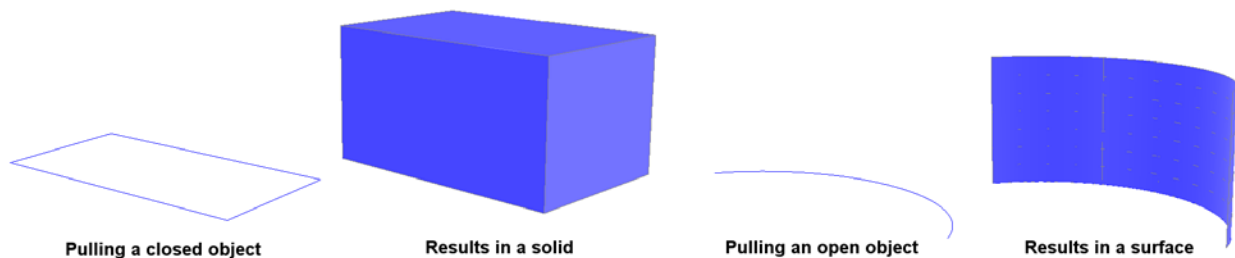

Before   **After**
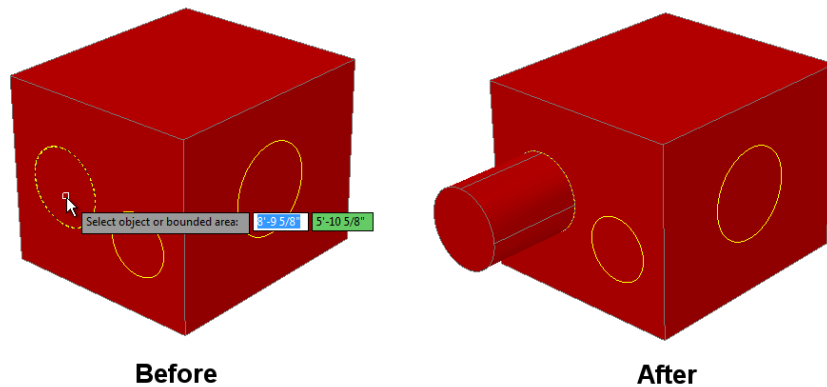
If you click on the object itself (instead of inside the bounded area), you can pull to extrude that object to create a separate solid.
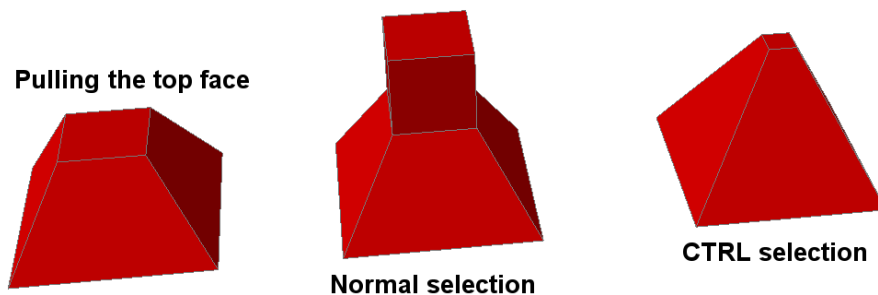

Before   **After**
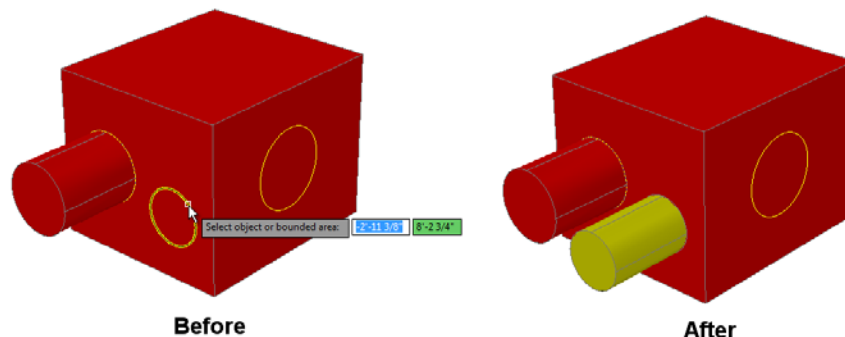
If you click inside a bounded area and press through the solid, you can remove that volume from the solid.

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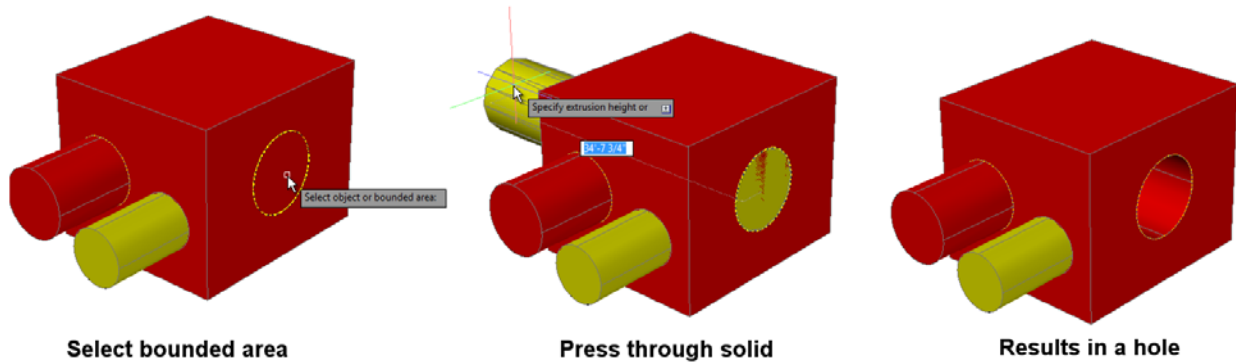
If you select a closed 2D object, you can pull to create a solid. If you select an open 2D object, you can pull to create a surface.

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 click to 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 notice that after you select an object or bounded area, but before you specify the extrusion height, you can choose the Multiple option and then click to select additional bounded areas. Once you're done, you can right-click and then extrude all of those areas at once.
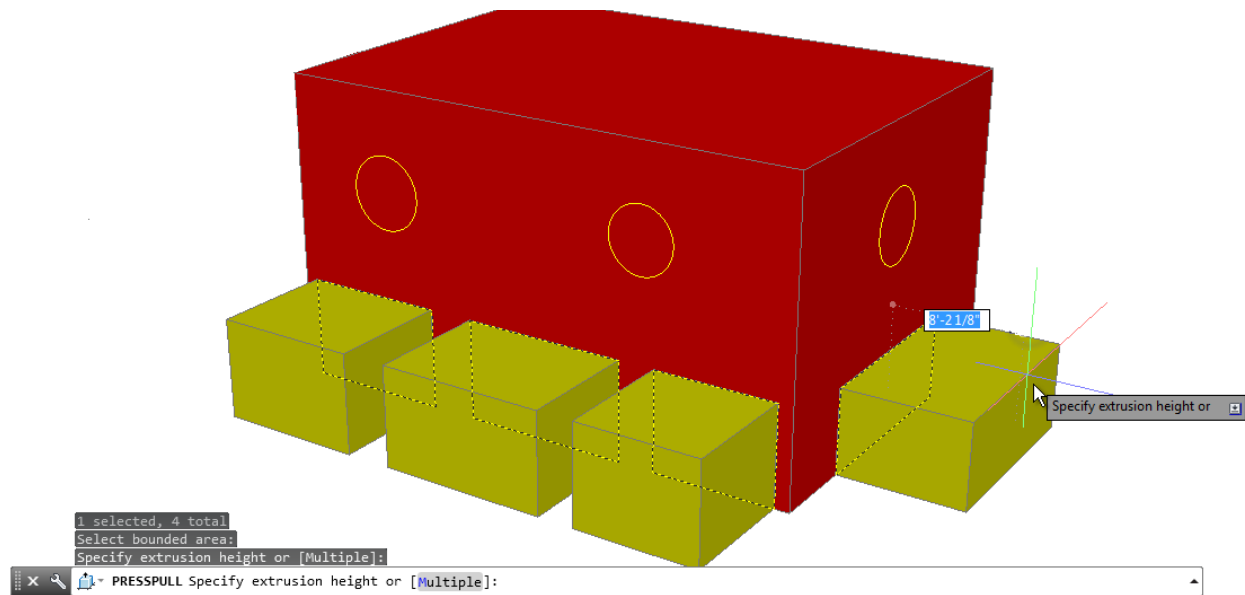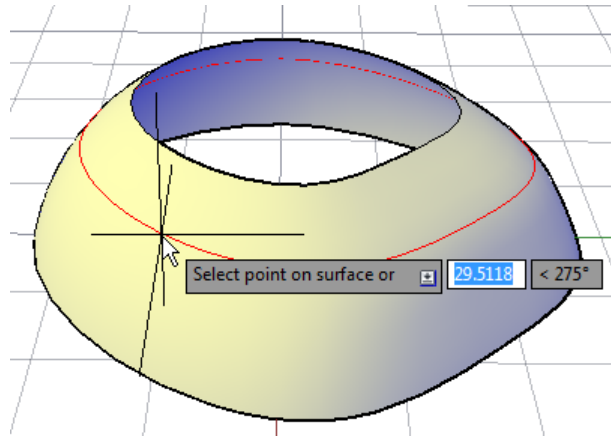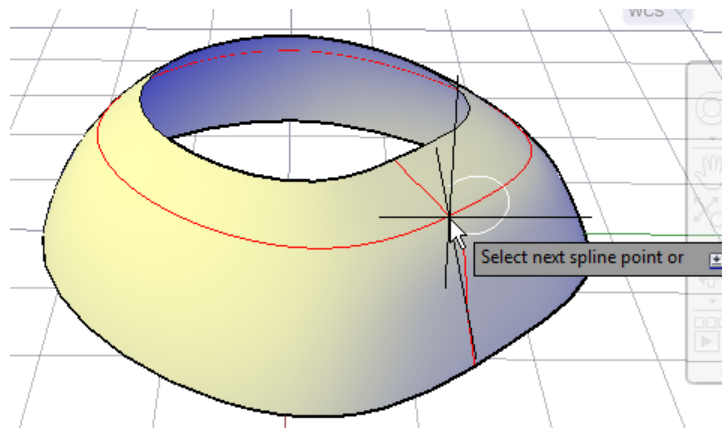

Extract Isolines

Extract Isolines is a new surface modeling tool introduced in AutoCAD 2013. This 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. You'll find the Extract Isolines tool in the Curves panel of the Surface ribbon.
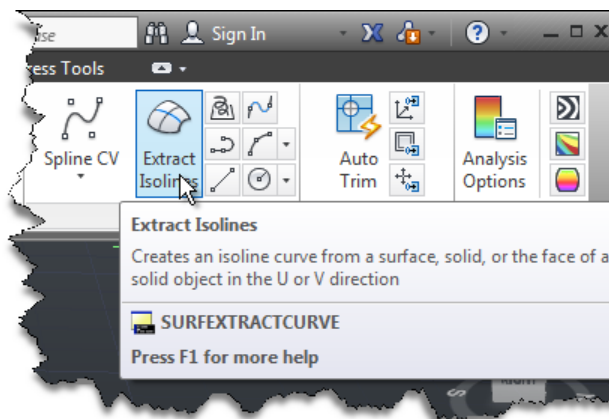
When you select this tool, AutoCAD prompts you to select a surface, solid, or face. If you click to select a surface, AutoCAD is immediately ready to place an isoline on that surface, and you can simply click to place it. Note that you can also use the direction option to change the direction of the isoline.


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


Once you've 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.
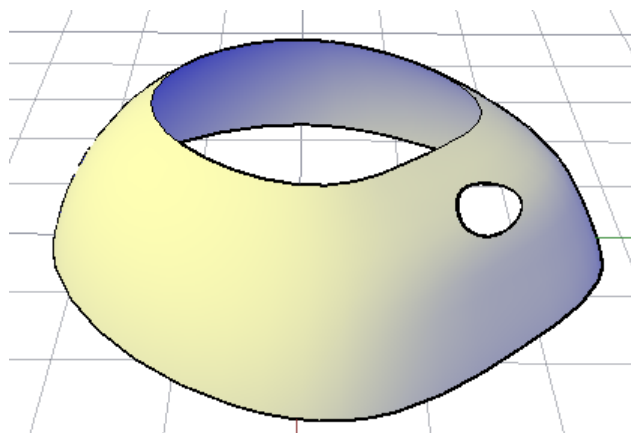
You could then use the Extrude tool (located in the Create panel of the Surface ribbon) to extrude the spline, creating a new surface.
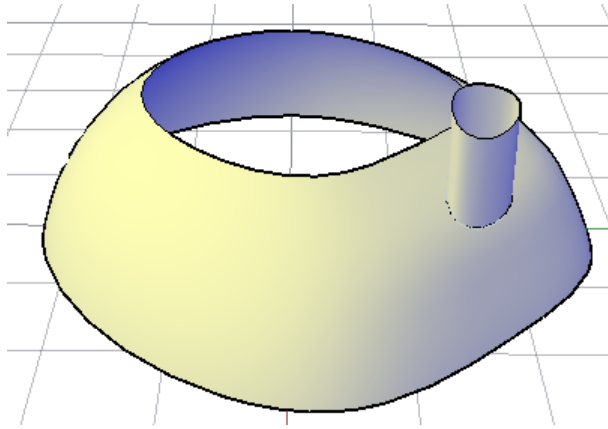
And since the two surfaces form a perfect intersection, you could do additional surface modeling, such as adding a surface fillet.
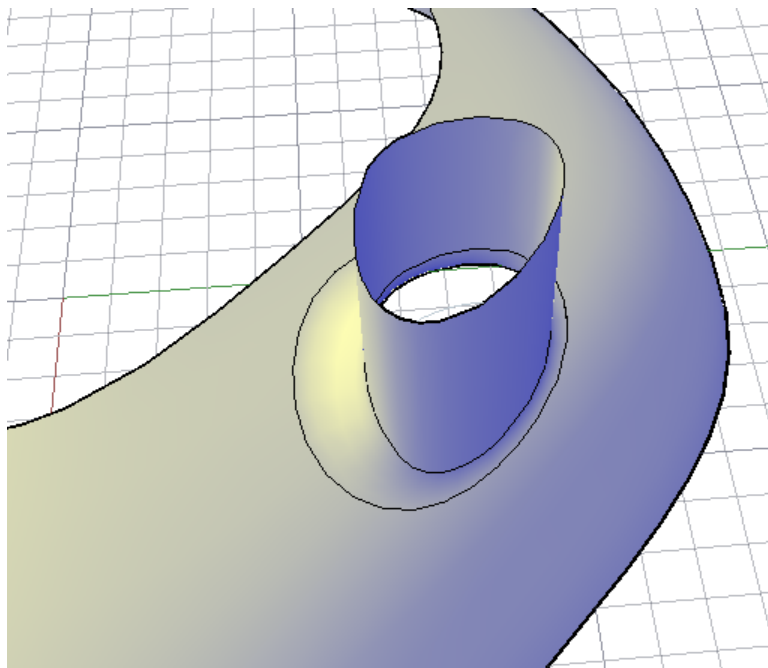
So as you can see, Extract Isolines can be a very powerful tool.

AutoCAD provides a wealth of powerful tools that truly enable you to take your ideas from concept to completion entirely in 3D using methods very similar to those that you've used for years working in 2D.

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