



3D Design from Concept to Completion in AutoCAD

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AutoCAD has evolved considerably over recent years to become a powerful 3D design application. In this class, you'll explore new and enhanced AutoCAD functionality that enables you to apply your existing 2D AutoCAD experience to the visual world of 3D modeling. Learn how to take your design ideas from concept to completion by creating and editing solid primitive objects. Create smooth, free-form shapes with new mesh tools. Control selection and editing with sub-object selection filters and 3D gizmos. In addition to exploring these basic 3D tools, you'll learning valuable tips and tricks for navigating and visualizing your AutoCAD model. If you thought 3D in AutoCAD was just too hard, think again.

About the Speaker:

David has more than 25 years of hands-on experience with AutoCAD® as a user, developer, author, and consultant. He is an independent CAD consultant, 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 teaches 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 3-D 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?

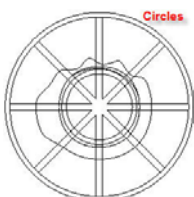
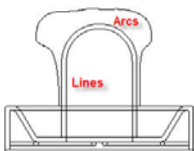
Most designers take the 3D conceptual designs from their imaginations and flatten them into 2D CAD drawings. The 2D CAD drawings are then interpreted by builders or manufacturers to produce 3D objects. This 3D to 2D to 3D design process was accepted without question because, originally, 2D CAD was simply 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 2-part 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 to world of 3D conceptual design, consider your thought process as you create a 2 dimensional drawing in AutoCAD. You probably start with an idea... an image in your head. You then flatten the image in your head by picturing it from different angles and using your drafting knowledge 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 3 dimensional image in your head.

The process for designing in 3D is similar... but even easier because you don't have to flatten the image! Instead, skip that step and jump right into identifying basic shapes. 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?



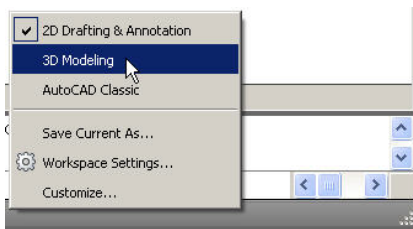


The air hockey paddle has several cylinders, an array of boxes, a sphere at the top and even an inverted cone of removed material. The house has many boxes and wedges, and a cylinder.



3D Modeling Workspace

AutoCAD 2010 includes three default workspaces: 2D Drafting and Annotation, 3D Modeling, and AutoCAD Classic. They're accessible from the status bar.



The 3D Modeling workspace provides easy access to 3D modeling and visualization tools via the AutoCAD ribbon.



Creating and Editing Solid Cylinders

Solid primitives are essential to creating typical 3D designs with ease and efficiency. They're available in the Primitives panel on the Home tab of the 3D Modeling workspace. The Box tool is displayed by default but the flyout will update to display whatever tool you used last.

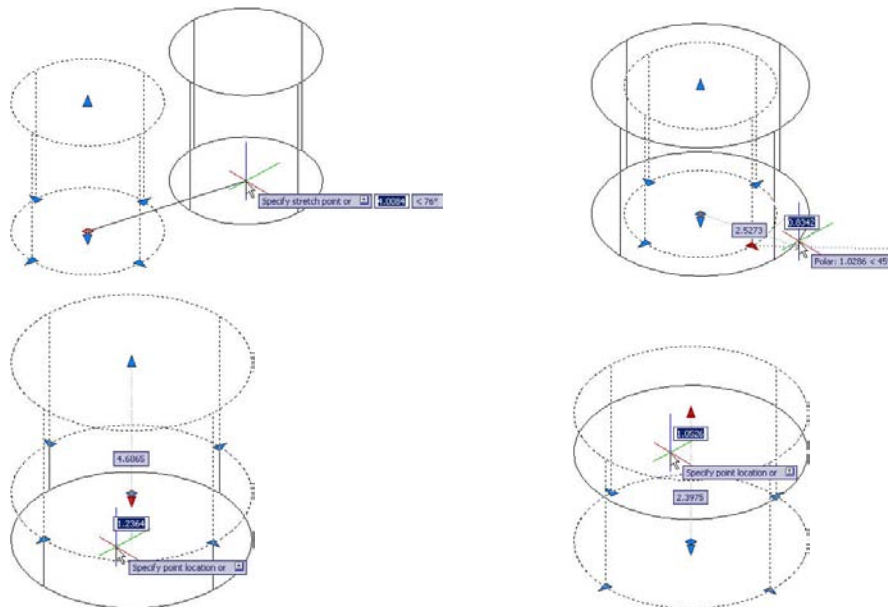
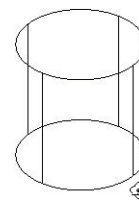
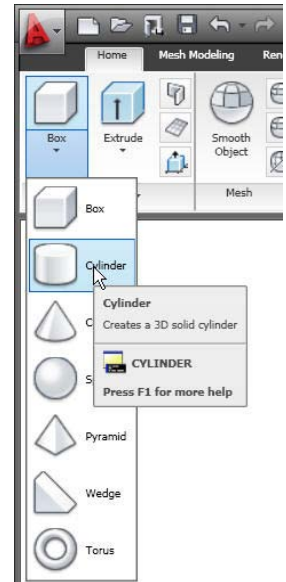
Drawing a cylinder feels very much like drawing a circle. You can create a cylinder using the same options (3P, 2P, Ttr). In addition, the cylinder tool includes an Ellipse option that enables you to create an elliptical cylinder (similar to drawing an ellipse). If you know exactly what you want the 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.

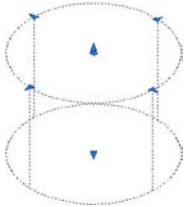
Since a cylinder is 3 dimensional, unlike a circle, it requires a height. From the top view, the height isn't obvious. It just looks like a circle. When you work in 3D, you'll want to view your model from different angles, not just from the top as we're used to in 2D design. 3D viewing and navigation is an entire topic on its own but you don't have to learn everything there is to know about it in order to get started with 3D modeling. One of my favorite viewing tools is Orbit. You can orbit the model by pressing the Shift key and middle mouse button and moving the mouse. The orbit symbol appears at the cursor and you can easily and intuitively adjust your view of the model.

AutoCAD has had the ability to create solid primitive objects, like cylinders, all the way back to AutoCAD Release 12. However, until AutoCAD 2007, those objects were not easily editable. You practically had to know exactly what you wanted to draw before you started. That might work for creating as-built models but it made conceptual design challenging and time-consuming. Fortunately, AutoCAD 2007 took a huge leap towards true conceptual design. You can easily edit primitive objects using grips or the Properties window.

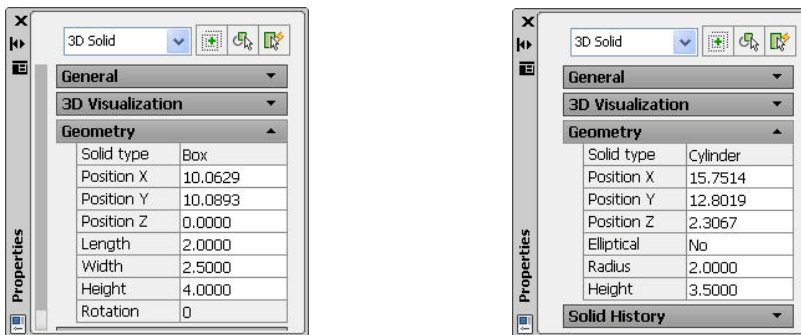
Selecting a cylinder displays grips a move grip (square) at the center of its base. Picking that grip makes it easy for you to move the cylinder to another location (just as you would expect from your 2D experience). Selecting one of the four quadrant grips enables you to edit the radius of the cylinder (again, just like you expect when editing a 2D circle). Selecting stretch grip at the center of the base enables you to change the height and, at the same time, change the location of the base. Selecting the grip at the other end of the cylinder enables you to edit the height of the cylinder without changing its base.



Did you notice that I described that grip as being "at the other end" rather than being at the "top"? That's because the base could just as easily be on top (or on the side or anywhere else). You could, for example, grip edit the "top" of this cylinder, and drag it below the base.



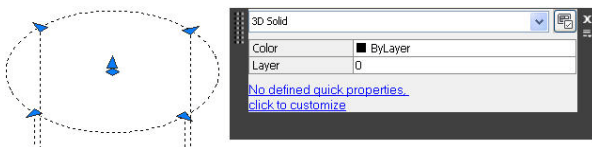
Grip editing is one way to change the shape of a solid primitive object. Another way is to enter specific values for key properties. The properties vary depending on the type of solid primitive. For example, a solid box has properties for length, width, and height; whereas a cylinder has properties for radius and height. You can view and modify these properties in the Geometry panel of the Properties window. As you change the values, the geometry in the drawing editor updates.



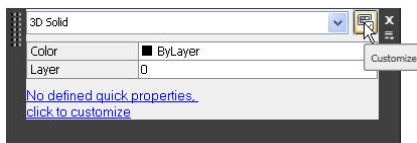
Quick Properties is another option (in AutoCAD 2009 or 2010) for modifying object properties. You can turn it on from the status bar.



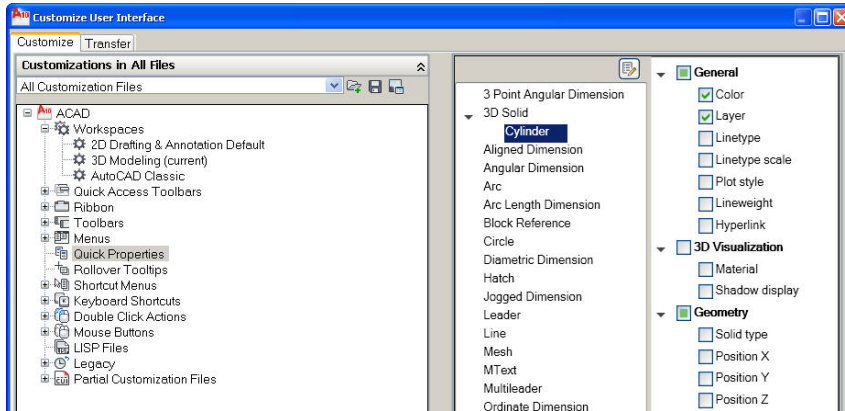
The purpose of Quick Properties is the same as Properties (to view and edit object properties) but Quick Properties is customizable so that can display only the useful properties rather than a long list of every property. You can also control where the Quick Properties palette is displayed. By default it's near the cursor giving you quick and easy access to key properties right when you select an object. For solid primitive objects, only the Color and Layer properties are displayed by default. However, you can add geometry values, such radius and height for a cylinder, to the Quick Properties list. Since these are properties that you'll want to edit often, it's worth taking a few minutes to make them easily accessible.



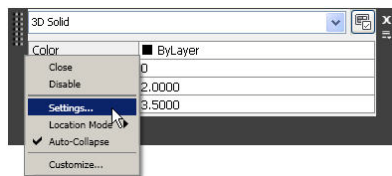
The “quickest” way to add properties for a specific type of object is to select that object with QP turned on (type QP or use the toggle on the status bar). In the Quick Properties palette, select the Customize button.



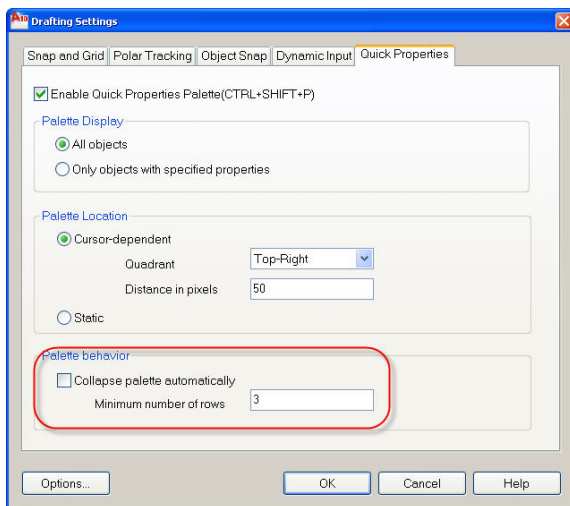
The Customize User Interface (CUI) dialog box is displayed and the select object, a cylinder in my case, is automatically added to the Quick Properties object type list (if it wasn't already there). Now you can simply turn on the properties you want QP to display for that particular type of object.



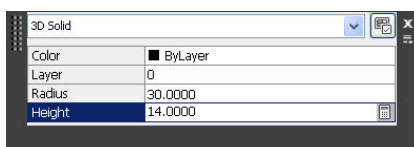
The Quick Properties palette only displays three rows by default. When you pass the cursor over it, the QP windows extends as necessary to display the remaining quick properties. If you'd like, you can modify the QP settings to display more properties.



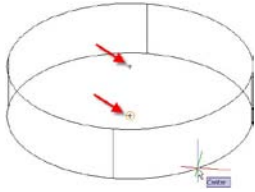
You can specify the number of rows to display or you can display all of the quick properties by disabling the option to Collapse palette automatically.



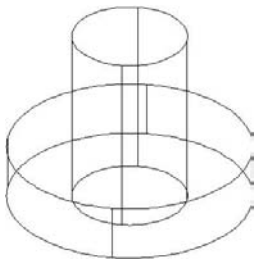
Now, when you select a cylinder, you can easily modify the radius and height in the Quick Properties window right near your cursor! For the air hockey paddle, change the radius and height of the cylinder to 30 and 14.



Moving along the design process for the air hockey paddle, you can create a second cylinder within the original cylinder. Since you want the two cylinders to be concentric, use the center object snap. As you might imagine, a cylinder has two center object snap points- one at the base and one at the top.

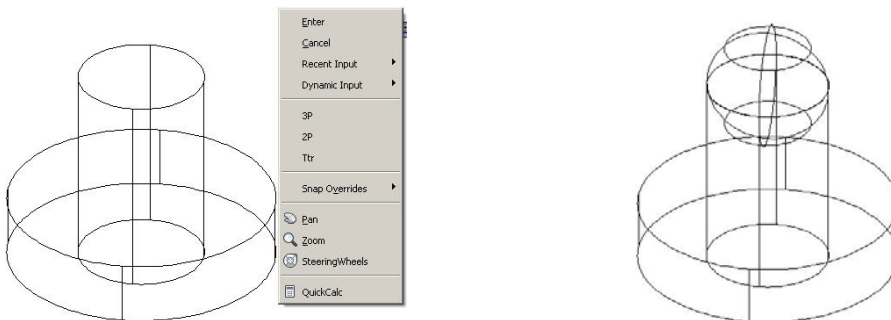


It doesn't matter which one you choose because it's easy enough to change later as your design evolves. But, for now, go ahead and pick the center osnap at the bottom of the existing cylinder and pick rough points to define the radius and height for the new cylinder.



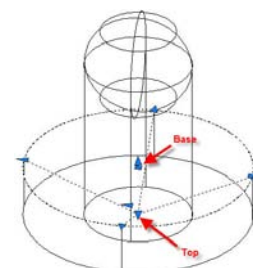
Creating and Editing Solid Spheres and Cones

The Sphere tool (SPHERE command) is available in the solid primitives flyout of the Modeling panel (with the 3D Modeling workspace current). Drawing a sphere is much like drawing a circle. You can use the default center/radius option or choose from 3Point, 2Point, and TanTanRadius. Use the default center/radius option and place it at the center of the top of the cylinder then use the quadrant osnap to specify the same radius as the cylinder.



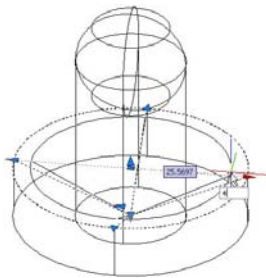
The Cone tool (CONE command) is available in the solid primitives flyout of the Modeling panel (with the 3D Modeling workspace current). The options for creating a cone are similar to creating a cylinder (and a circle). First, you specify the location and size of the base of the cone. You can use the default center/radius option or choose from 3Point, 2Point, TanTanRadius, and Elliptical. Just like with drawing a circle in 2D, the option you choose depends on what you're trying to create, what existing objects you can snap to, and what you know. Since we want the cone to be located at the center of the existing cylinder, we can 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

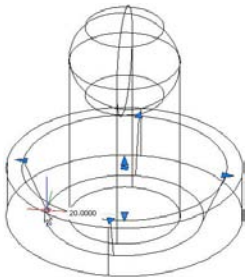


additional options as well. A right-click menu (and the Command line) enable you to pick 2 points to determine the height, pick a point to specify the height and change the orientation at the same time, or enter a value for the top radius. If you don't enter a value for the top radius, it assumes a value of zero and creates the cone with a sharp point. Also, keep in mind that, like a cylinder, the base and top of a cone don't necessarily mean that the base has to be on the bottom and the top has to be on the top. For the air hockey paddle, I created the cone using a default top radius of zero and snapping to the center at the bottom of the cylinder to specify the height.

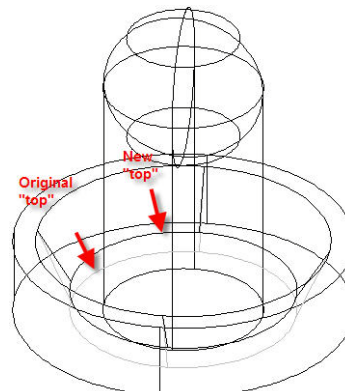
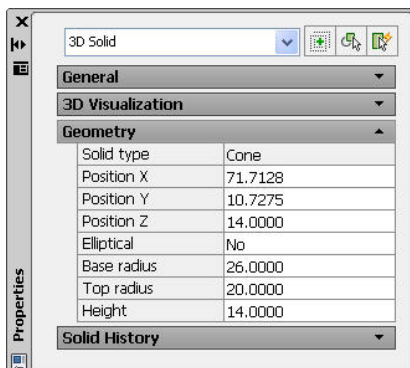
Since the purpose of the cone is to subtract material from the large cylinder, we need to modify the base radius so there will be a rim around the top of the cylinder after the cone is subtracted. Modifying a solid cone is just as easy as modifying a solid cylinder. To ensure a rim of 4 units, drag the cone's quadrant grips in 4 units.



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.

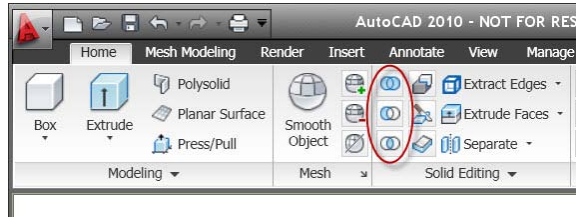


Even though AutoCAD doesn't care if the Top Radius is really on top, there is a difference. If you change the height of the cone, the base maintains its position while the location of the "top" adjusts accordingly. For example, to ensure you have 4 units of material left at the bottom of the cylinder after you subtract the cone, change the height of the cylinder from 14 to 10 units.

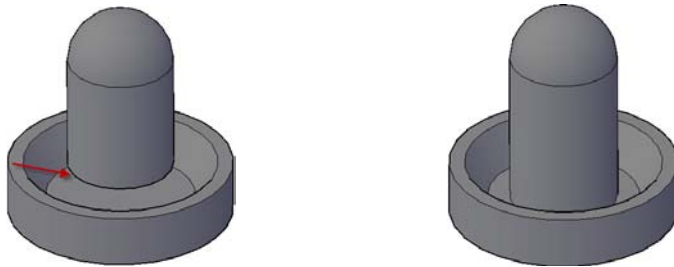


Subtracting and Unioning Solids

The cone primitive represents material we want to remove from the paddle. To add and remove solid objects, you can use Boolean tools. You'll find these tools on the Home tab of the ribbon when the 3D Modeling workspace is active.



The Union tool joins the selected solid objects into a single composite solid object. The Subtract tool removes selected objects from other selected objects; also forming a composite solid. The order in which you perform a combination of Union and Subtract operations can create different results. For example, if you union the cylinders and sphere for the air hockey paddle and then subtract the cone, you get a big gap between the upper part of the model (the handle) and the lower part of the model (the striker). This is still a single composite solid but it looks like two separate objects, which isn't the result we're looking for. The solution is easy. It's just a matter of changing the order of the Boolean operations and the objects that you select.

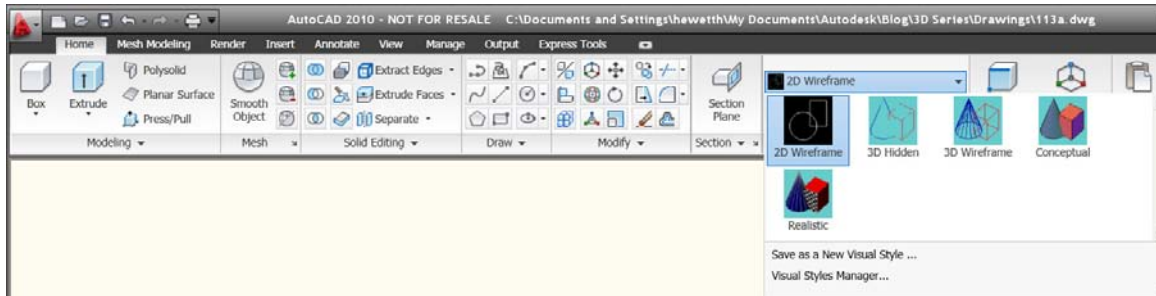


When using the Subtract tool, it's important to read the command prompts to ensure you are selecting objects in the appropriate order. You must first select the objects you want to subtract from and then select the objects you want to subtract from them. Since you can select multiple objects, you must remember to press enter to finish the "select from" option before you select the object(s) you want to subtract from them. This may seem obvious but, speaking from experience, it's easy to make a mistake with the selections if you're new to the Subtract tool or if you haven't used it for a while. This is one of the few AutoCAD commands that prompts for two selection sets.

To create the air hockey paddle, first subtract the cone from the lower cylinder and then union the newly created composite solid with the remaining cylinder and sphere.

Setting a Visual Style

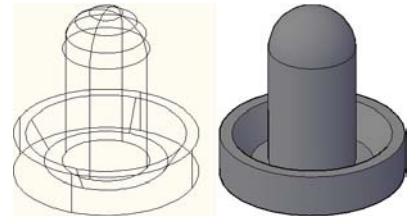
As you create 3D models in AutoCAD, you'll discover that it can be difficult to visualize the geometry using the default wireframe visual style. Visual Styles were introduced in AutoCAD 2007 and are a powerful replacement for the old "shade modes". AutoCAD includes five visual styles: 2D Wireframe, 3D Hidden, 3D Wireframe, Conceptual, and Realistic. You can access Visual Styles in the View panel of the Home tab when the 3D modeling workspace is active.



Here are some brief descriptions of the five predefined visual styles (directly from Help system):

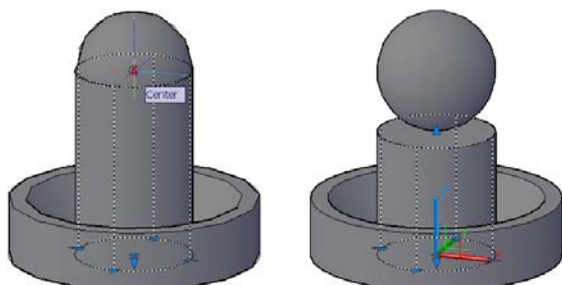
- **2D Wireframe.** Displays the objects using lines and curves to represent the boundaries. Raster and OLE objects, linetypes, and lineweights are visible.
- **3D Wireframe.** Displays the objects using lines and curves to represent the boundaries.
- **3D Hidden.** Displays the objects using 3D wireframe representation and hides lines representing back faces.
- **Conceptual.** Shades the objects and smooths the edges between polygon faces. Shading uses the Gooch face style, a transition between cool and warm colors rather than dark to light.
- **Realistic.** Shades the objects and smooths the edges between polygon faces. Materials that you have attached to the objects are displayed.

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 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 Conceptual visual style, it becomes clear that the cone has been subtracted.

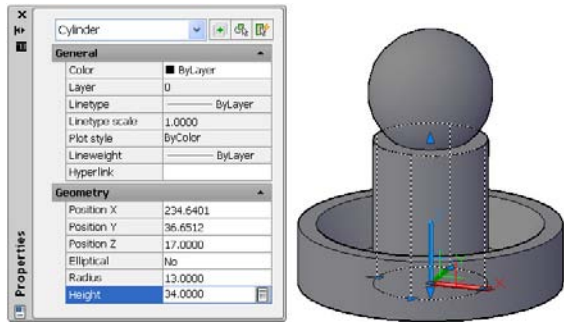


Editing Composite Solids

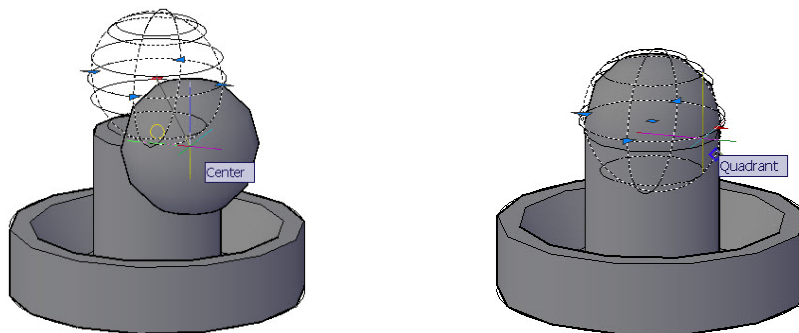
When you union or subtract solid primitive objects, AutoCAD creates a composite solid. If you click to select any of the primitive objects, the entire composite is selected. Fortunately you can still edit the primitive objects. The key is control... or I should say CTRL is the key! If you hold down the 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 way 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).



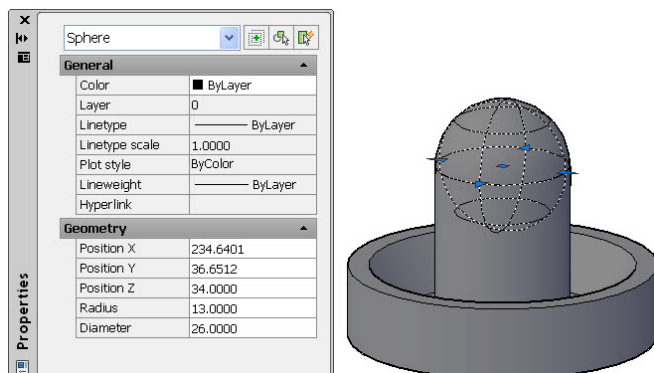
As I mentioned, you can also edit sub-objects using the Properties palette. Notice that even though the cylinder is a sub-object in the composite solid, all the typical cylinder properties are available for editing. Change the radius of the cylinder to 13 and the height to 34.



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 osnap 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 osnap of the cylinder to automatically match the cylinder's radius... just like editing a circle in 2D!

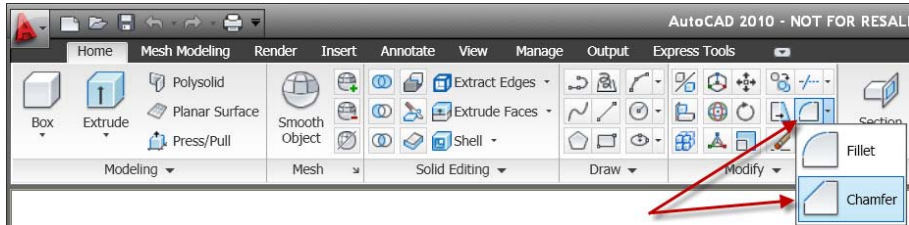


To verify that you snapped to the right point, thus creating a sphere with the correct radius, you can view the sphere properties in the Properties palette. The radius should equal 13 just like the cylinder radius we edited previously.

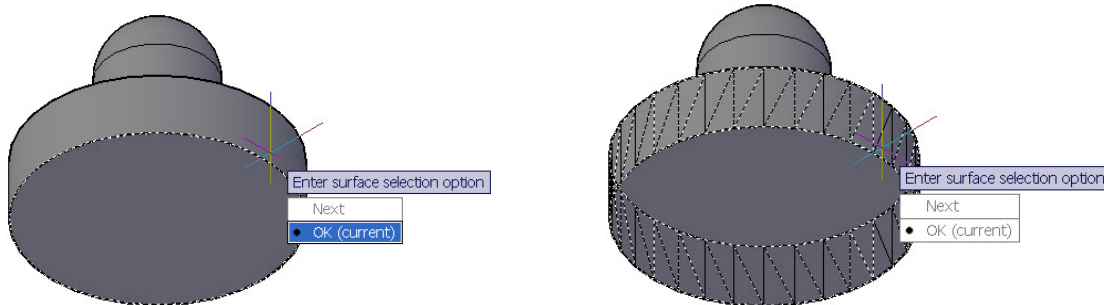


Creating and Editing Chamfers and Fillets

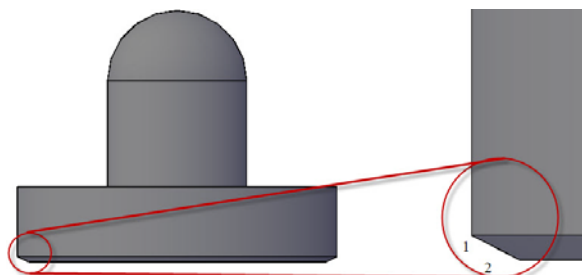
All the objects we've used to create the air hockey paddle are basic primitive shapes; cylinders, sphere, and cone. It conveys the general shape that we want, but the edges are hard. We'll soften the bottom edge by adding a chamfer. You can use the same Chamfer command that you've probably been using for 2D design. When the 3D modeling workspace is active, you'll find the Chamfer tool in the Modify panel of the Home tab. It's in the same button flyout as the Fillet tool. Whichever one you use 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 edge 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.



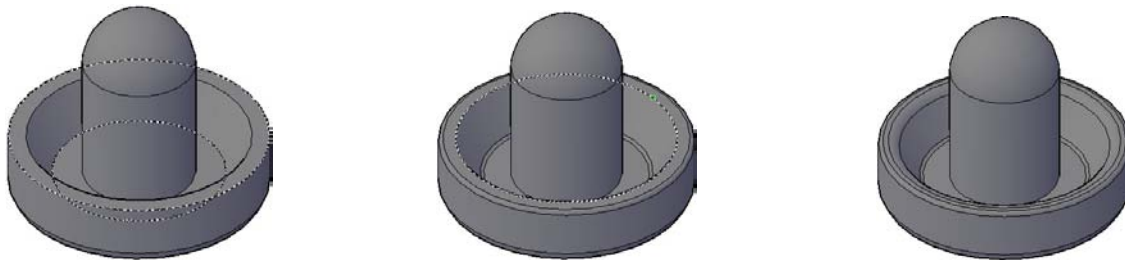
In this particular case it doesn't matter which one you choose as the base surface as long as you enter the appropriate values for the following prompts. Go ahead and accept the bottom circular face as the base surface. AutoCAD will then prompt for the base surface chamfer distance (i.e. the distance 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.



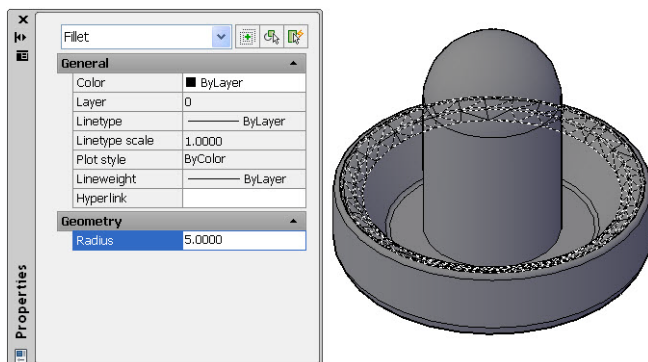
If you had entered the values in a different order (1 for the base distance and 2 for the other distance) or if you had specified the cylindrical face as the base surface, the chamfer would have been 1 unit along the bottom and 2 units along the side.

If you want to create rounded edges on your 3D model, you can use the Fillet tool... the same one you use for 2D design. You can access the Fillet tool from the Modify panel of the Home tab. It's in the same button flyout at the Chamfer tool so if 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.



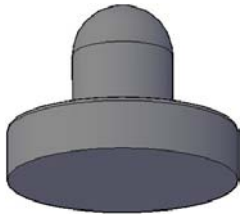
When you add chamfers and fillets to a composite solid, the chamfers and fillets become subobjects for that composite solid. You can select chamfer and fillet subobjects using the CTRL key. As you pass the cursor over a composite solid, the chamfers and fillets highlight. You can then click to select them just as you can with solid primitive subobjects. Unlike solid primitive subobjects, chamfers and fillets don't have object grips. However, they do have object properties. 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. For example, I applied the 1 unit fillet to two edges at the same time. When I pass the cursor over either fillet (with the CTRL key pressed), both are highlighted. If I change the Radius value in the Properties window, both fillets are updated.

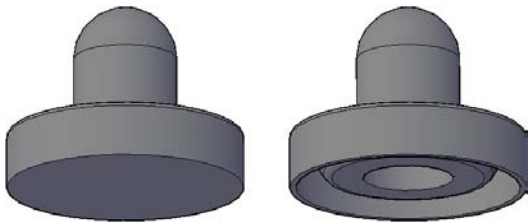
You can edit 3D chamfers in a similar way. When you CTRL-select a chamfer, the Properties palette displays the First Distance and Second Distance. In addition to changing the fillet and

chamfer values you can also remove fillets and chamfers. For example, if you decide you don't want a chamfer on the bottom of the air hockey paddle, you can use CTRL-pick to select the chamfer and then use the Erase tool to remove it.

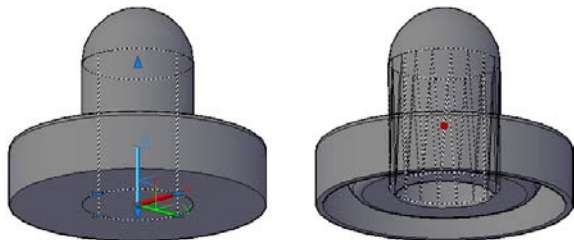


Shelling a Solid

All the tools we've used to design the air hockey paddle so far have enabled AutoCAD to maintain the integrity of the sub-objects. Even after we combined the cylinders, sphere, and cone into a composite solid using Boolean operations (union and subtract) we could still access and edit the primitive properties by pressing the CTRL key. We could continue to modify those properties (in addition to the chamfer and fillet properties) even after we added chamfers and fillets to the model. There are some tools, however, that cause the model to lose that editing capability. You can save a lot of editing time if you avoid those tools 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.



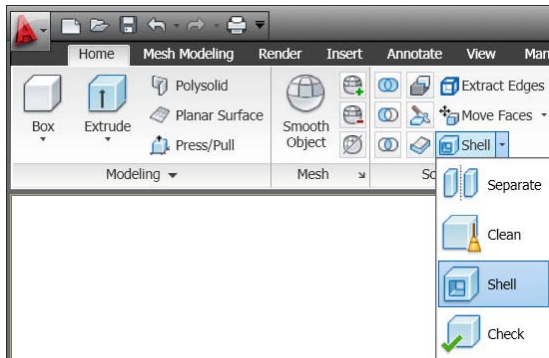
Unfortunately, the Shell tool strips the individual components (cylinder, sphere, cone, fillet, etc) of their intelligence. For example, when you press the CTRL key and click on the tall cylinder in a model before using the Shell tool, AutoCAD knows it's a cylinder and displays the appropriate grips. When you click on the same object after using the Shell tool, AutoCAD treats it as a face with limited editing capability.



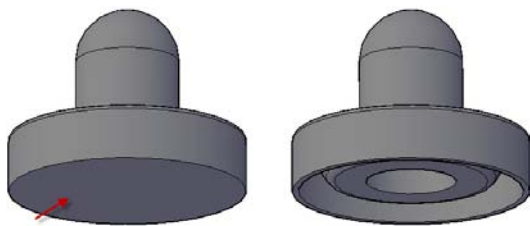
It's not a big deal if those sub-objects already have the perfect values. But, in conceptual design, that's rarely the case. You want the ability to edit those values for as long as possible. Nevertheless, the Shell tool is handy for creating thin-walled objects such as the air hockey paddle. So, being 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 tab. You'll probably have to open the flyout toolbar to find it. The Shell tool is actually an option within the SOLIDEDIT command but it's kind of buried (you'll first need to choose the Body option). If you like to use the keyboard you might be tempted to type SHELL. That's not what you want. The SHELL

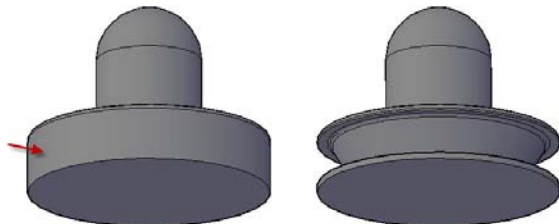
command (unlike the Shell option of the SOLIDEDIT command) has been in AutoCAD for as long as I can remember and it launches an OS window. I make that mistake sometimes (too often) but the brief flashback to DOS always makes me laugh.



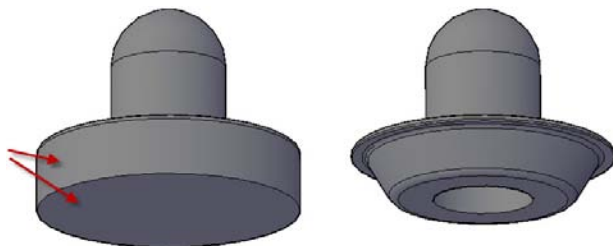
After you successfully launch the right Shell tool, you're prompted to select a 3D solid object and then you're prompted to remove faces. "Remove faces" doesn't mean it's going to remove faces from the object. It's giving the option to exclude faces from the shell operation. For example, when I performed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



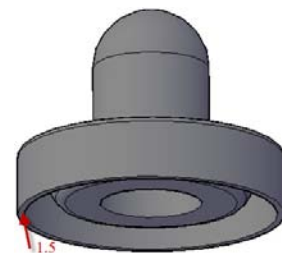
If I had "removed" the cylindrical face instead, it would look like this:



If I had removed the circular and the cylindrical faces, it would look like this:

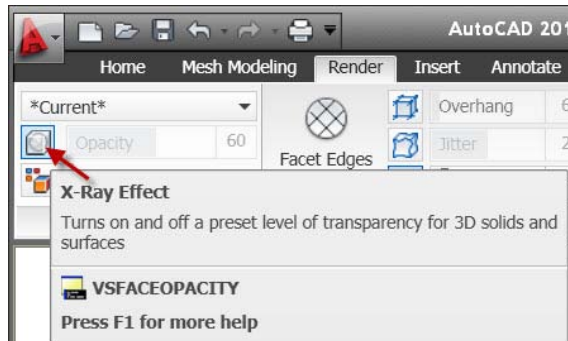


If you don't select any faces (just press Enter to end the selection set), the outside of the model won't look any different but it will be hollowed out. After you specify which, if any, faces to remove from the shell operation, you must enter the shell offset distance. I entered a distance of 1.5 for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

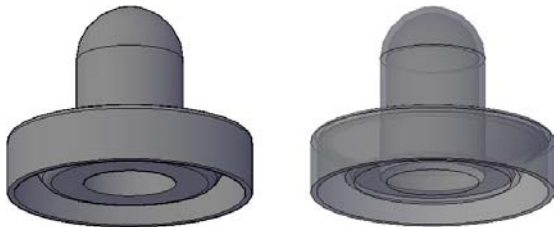


Apply the X-Ray Effect

There are times that it can be helpful to view your model with the edges only slightly obscured... a compromise between the wireframe and conceptual visual styles. For those times you can use the conceptual visual style with an X-ray effect. The X-ray effect is accessible in the Visual Styles panel of the Render tab.



You can toggle the X-Ray effect on or off and you can use the slider bar to control the opacity of the X-Ray effect. In the images below, you can see the difference with the X-Ray effect turned off and on (with opacity of 60).



I turned the X-Ray mode on while I created a thin cylinder for the bottom of the air hockey paddle (using the Cylinder tool in the Modeling panel of the Home tab). It enabled me to see into the model, ensuring I select the proper points, yet it's easier to visualize than with the wireframe mode. For the center point of the base, I snapped to the center of the bottom of the paddle and for the base radius I used the quadrant osnap, ensuring that I got the quadrant of the inner ring. To specify the height, I dragged the cylinder up and entered a value of 1.5. Since X-Ray mode is on, I could see the top edge of the new cylinder; reassuring me that I drew it in the right direction and at an appropriate size.



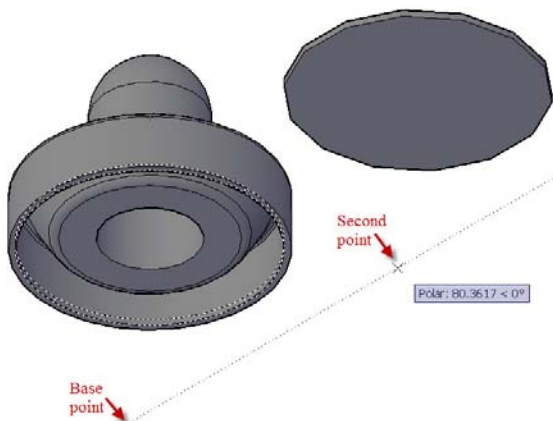
Moving with Polar Tracking

You can create and modify 3D geometry regardless of which visual style is active. Visual style options such as the X-Ray effect enable you to see through the model and you can snap to points on inside/obscured objects. For example, if you want to create an array of box-shaped ribs on the inside of the air hockey paddle, you can draw them directly in the model viewing and snapping to key points.



Just because you *can* do it, however, doesn't mean you should. It might save a few steps by drawing geometry on the inside of a model but the mental energy it requires may not be worth it. Unless I'm performing a few quick picks and I'm very confident about my selection/pick points, I prefer to move geometry out where I can visualize it more easily. To draw the boxes on the inside of the air hockey paddle, for example, you might want to move the bottom plate of the air hockey paddle away from the rest of the model. You could be sloppy and move it anywhere in 3D space, but in order to take advantage of existing geometry, a more deliberate move might save you some clicks in the long run. First, I turned off the X-Ray effect and orbited the model to view it from below.

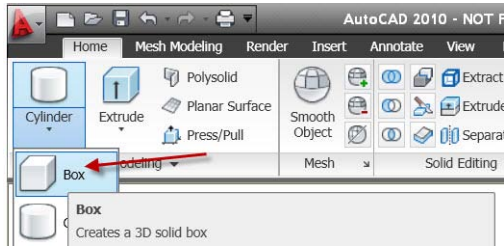
You could choose from several different tools for moving the plate in 3D but I often find that the plain old MOVE command works just fine. For the base point, select anywhere in space (being careful not to snap to any objects). Then drag the cursor along the X or Y axis (with polar tracking turned on) and pick to specify the second point (again being careful not to snap to any objects). Isn't this exactly what you do with the MOVE command in 2D design? We just happen to be looking at the model from a 3D viewpoint.



As long as you were careful not to snap to existing geometry, you can be assured that the cylinder has not moved along the Z-axis. Why is that important? Because we'll use the Z-values from both of our parts (the plate and the main part of the paddle) when we draw the box ribs.

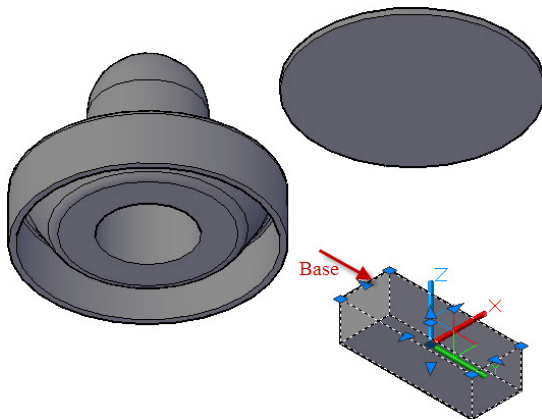
Creating and Editing a Solid Box

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 tab. It will prompt (by default) for two corners (just like the RECTANGLE command) and a height.

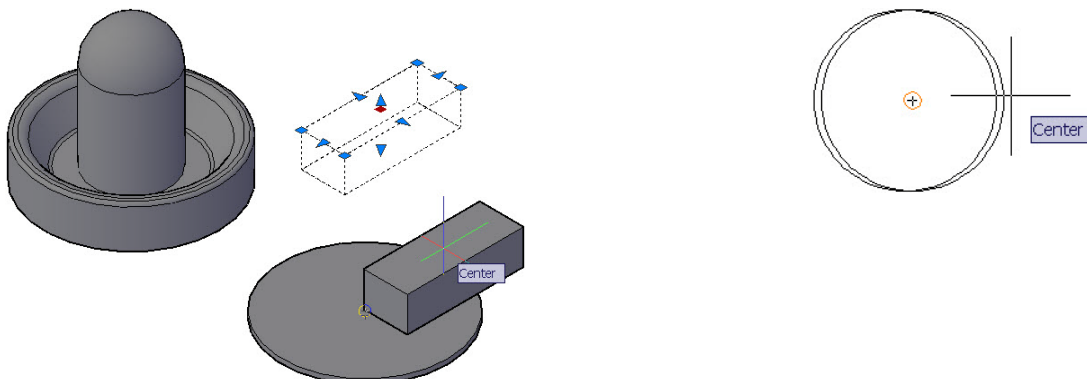


You could save some clicks if you draw the box exactly where you want it to go but maybe you don't know that yet... After all, this is conceptual design and you may not know where you're going until you get there. For now, just draw the box anywhere in 3D space.

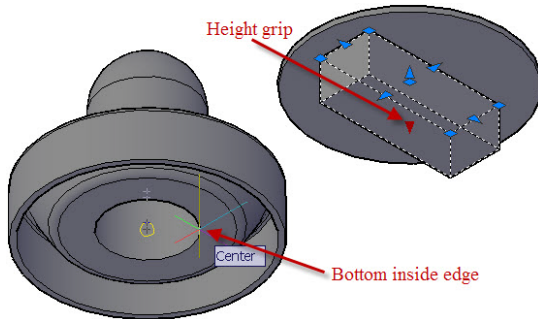
I was **really** sloppy. I even drew the height down the Z-axis so the base of the box is actually on top. Thank the AutoCAD developers for grips!



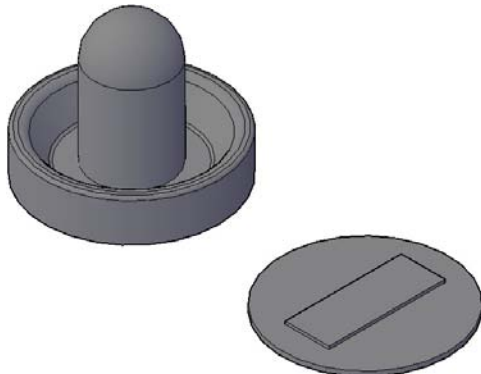
As I thought about the design, I realized I need the box rib to be flush with the top of the cylindrical plate. First, I selected the base grip (the square grip in the middle) and moved the box to the center of the top of the cylinder (using the CEN osnap of course). Since the cylinder is thin, the top and bottom center points are close together. You have to be careful to select the right one. You can apply the same trick that you use in 2D (and probably take for granted). For example, if you have a 2D drawing with two circles that are very close to each other and you want to pick the center of the circle on the right, you would position your cursor to the right rather than in the tiny gap between the two circles. This ensures that you get the "right" circle. The same logic applies in 3D!



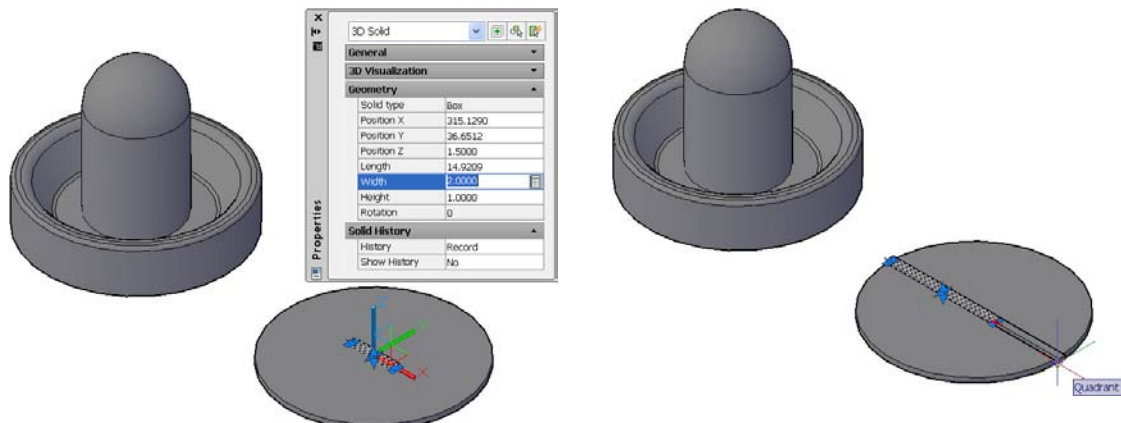
Now the base of my box is in the right position but the height is going the wrong direction. We want the top of the box to be flush with the inside of the air hockey paddle. If we know the value for that height, we could enter it in the Properties palette but it would still be going the wrong direction. We'd first have to grip-edit the height so that it's above the base and then enter the appropriate height value. I don't know the value and I don't want to think that hard. Since we know the top of the box needs to be flush with the inside of the paddle, we can grip-edit the box snapping to existing geometry. Select the box, pick the height grip, and select the bottom inside edge on the air hockey paddle. Since the height grip is constrained to moving in one direction (along the Z-axis in this case), it doesn't matter that we're snapping to an object with random XY values.



Viewing the model from above, you can see that the box is now protruding above the cylindrical plate and you can rest assured that when the plate is inserted in the paddle, the top of the rib will be flush with lower inside face of the paddle.

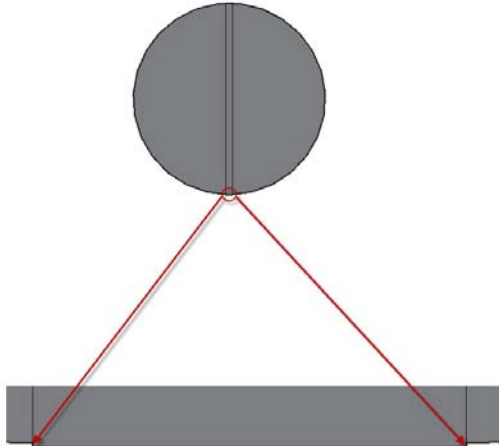


Next, we need to modify the width and length of the box. We can use the Properties palette to set the width to 2 units. We want the length to be equal to the diameter of the cylinder. If we know the diameter of the cylinder we can enter it in the Properties palette. But, if we don't know it or don't want to think that hard, we can grip-edit by snapping to the appropriate quadrants on the cylinder. The length grip is automatically constrained to move along the length axis so it doesn't matter if we pick the upper or lower quadrant grips.

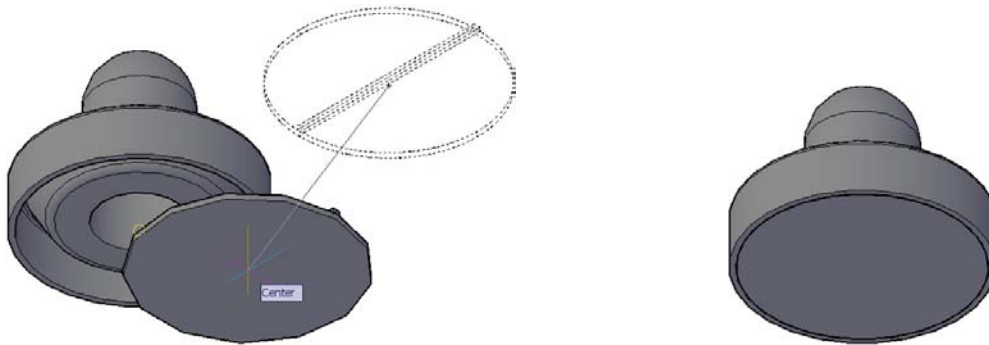


Checking for Interferences

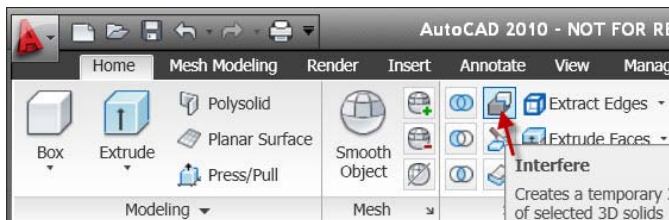
A close (very close) look at the ends of the box shows the corners extend past the edge of the cylinder. It's tough to see because it's so small but it's a potential problem when we try to insert the bottom plate into the air hockey paddle. We can use interference checking to identify and eliminate interferences.



First we need to move the cylinder and box objects into the proper position at the bottom of the air hockey paddle. You can use the ordinary MOVE command snapping from the bottom center of the cylindrical plate to the bottom center of the rim of the air hockey paddle. Since the inner and outer edges of the rim are concentric, it doesn't matter which one you pick. If, when specifying the base point, you pick the top center of the cylinder instead of the bottom, the cylinder won't be flush with the bottom rim of the air hockey paddle.

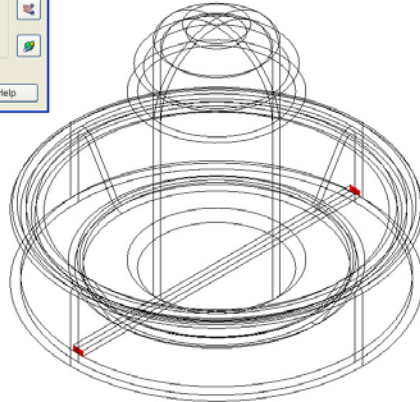
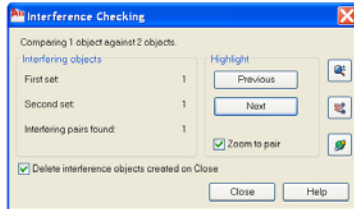


You can access the Interfere tool from the Solid Editing Panel of the Home tab.

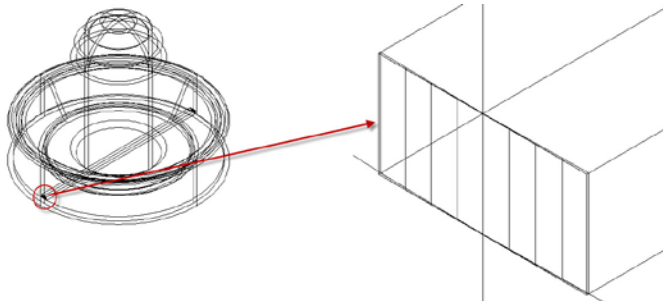


The Interfere tool is one of the rare AutoCAD commands (like Subtract) that require two selection sets. To check for interferences between the air hockey paddle and the cylinder and box objects, we select the air hockey paddle as the first selection set (pressing enter to end the selection set). Then select the box and cylinder as the second selection set. After completing the second selection set, AutoCAD displays the Interference Checking dialog box and creates

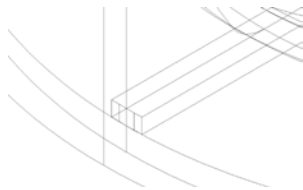
temporary solid objects (in red) where any interferences occur. The dialog box offers various tools that enable you to examine the interferences while they're temporarily displayed.



When you close the dialog box, the red interference objects are automatically removed unless you deselect the option to “Delete interference objects created on Close”. With that option deselected, 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 one solid. The interferences are just slivers in this case.

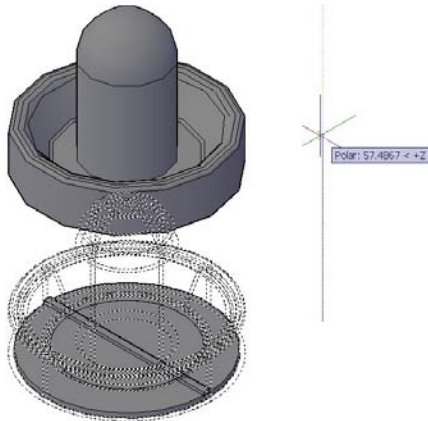


Using the subtract tool, we can first select the box and then (after ending that selection set), select the interference solid. The change is barely noticeable but could be the difference between a successful design and a costly mistake!

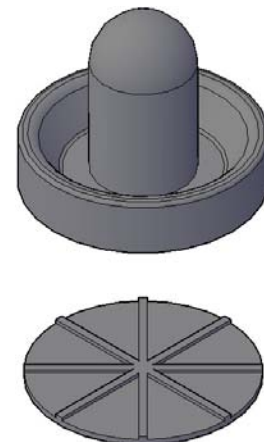
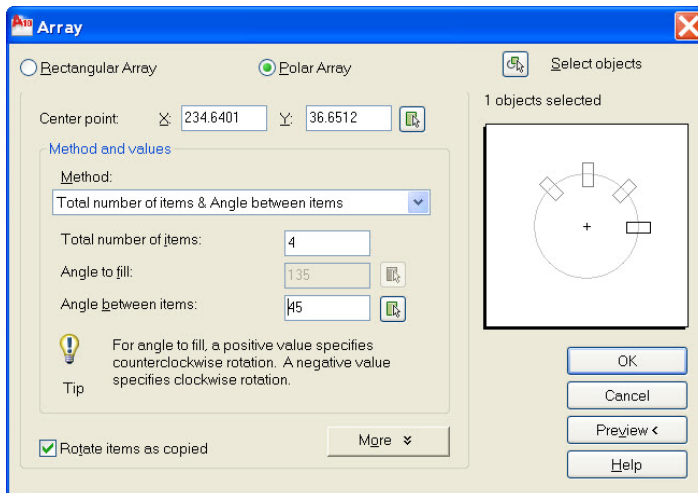


Arraying a Box

Our final task, before moving to the rubber grip, is to array the box-shaped rib and then union the ribs to the cylindrical plate. For a better view of the box and plate, you can move the paddle straight up the Z-axis using polar tracking in exactly the same way you move objects along the X and Y axis for 2D design.

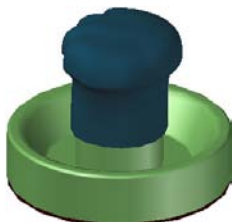


AutoCAD has a 3D array tool (on the Modify panel of the Home tab), which you can use to array an object in three dimensions. I've found, however, that the plain old 2D array is sufficient for most of my 3D modeling needs. The rib for the air hockey paddle, for example, is on the XY plane and will be arrayed around a Z-axis (just like in 2D). In the Array dialog box, choose the Polar option, select the rib as the object to be arrayed, and specify the center of the cylinder to be the center point for array. For the method, it works well to specify the total number of items (4) and the angle between the items (45). After arraying the box, you can union the boxes with the cylindrical plate.

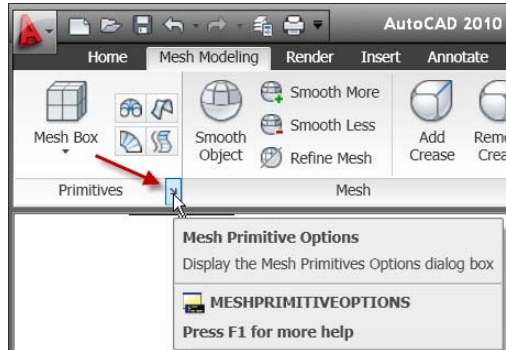


Creating a Mesh Box

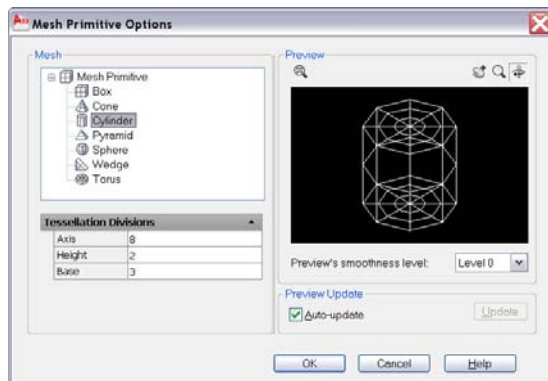
Now we're ready for the new stuff! Using the new mesh modeling tools, introduced in AutoCAD 2010, we'll design a freeform shape for the rubber grip.



The 3D Modeling workspace in AutoCAD 2010 includes a new Mesh Modeling ribbon tab. The Mesh Modeling tab has six panels including the Primitives panel. Notice the small arrow icon in the lower right corner of the Primitives panel. You'll see the same icon in many other panels throughout the AutoCAD ribbon. These icons launch relevant dialog boxes for each specific panel. Clicking the icon in the Primitives panel opens the Mesh Primitive Options dialog box.

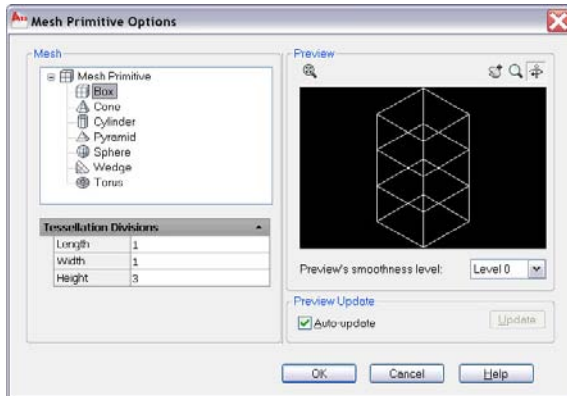


The Mesh Primitive Options dialog box enables you to specify the number of tessellation divisions for the mesh primitives (box, cone, etc) you create. The tessellation options vary for the different types of primitives. For example, the tessellation divisions for a box are length, width, and height whereas the divisions for a cylinder are axis, height, and base. The number of tessellation divisions defines the editable faces on each surface. A cylinder with 8 axis, 2 height, and 3 base tessellation divisions, for example, has 24 faces on each of the top and bottom surfaces and 16 faces around the side.

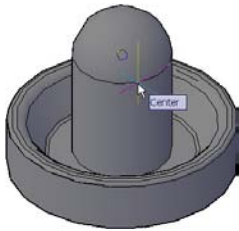


You can preview the primitive shapes based on the tessellation divisions that you assign. If Auto-update is enabled, the preview image automatically updates. If it's not enabled, you can choose the Update option to view the results. In addition to the tessellation divisions, you can assign a default smoothness level between 0 and 4. A level of 0 creates the basic primitive shape with only 1 facet per face. As you increase the smoothness level, the number of facets increases creating a smoother, more rounded shape. You can change the smoothness level even after you've created a mesh primitive object.

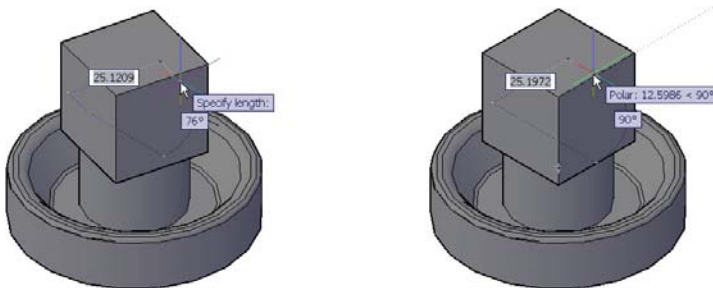
To create the rubber grip for the air hockey paddle, we'll start with a mesh box with relatively few tessellation divisions. Set the length and width to 1 and the height to 3. Leave the smoothness at 0 for now.



Now we're ready to create the mesh box. We could create the mesh box by specifying two corners of the base, similar to creating a rectangle. But, we already have the main paddle model and we want the grip to be centered on this model, so we can use the Center option and select the center osnap on the sphere.

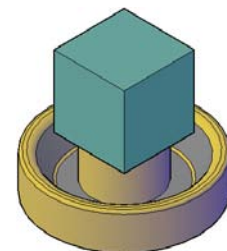


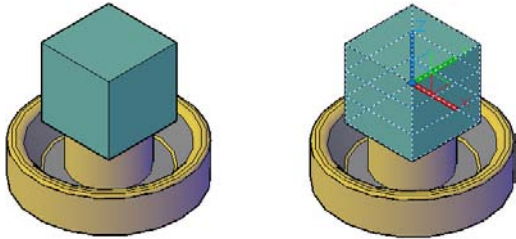
Next, we can pick the corner of the base, enter values for the length of each side, or select the Cube option. The Cube option is handy in this case because it enables you to quickly create the box by specifying only one value for length, width, and height. You can pick a point or type a value, just as you're used to with other AutoCAD drawing tools. Even if you type a value, the direction in which you drag the cursor affects how the cube is positioned. For example if you drag in a random direction, the faces of the box are not parallel with the coordinate system. If you drag along a polar axis when you enter the value, the planes of the cube are parallel with the coordinate system.



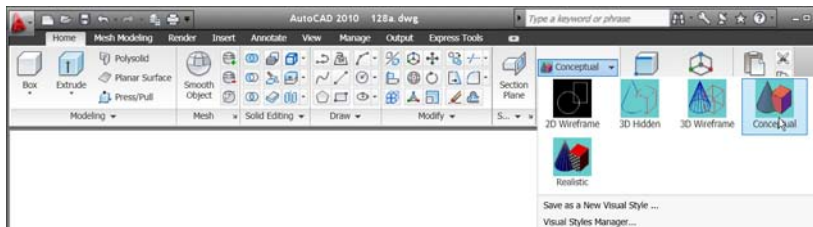
To create this rubber grip, I dragged the cursor along a polar axis (since it's a cube, it doesn't matter which axis) and entered a length of 30, which is slightly larger than the diameter of the existing plastic handle. To help differentiate between the plastic paddle and the rubber grip, it's a good idea to put them on different layers (with different colors).

At first glance, the mesh box appears to have only a single tessellation division for height as well as the length and width. In reality, the height does have 3 tessellation divisions (as we specified in the Mesh Primitive Options) but you can't see them with the current Conceptual visual style. If you select the mesh box, you'll see the tessellation divisions.

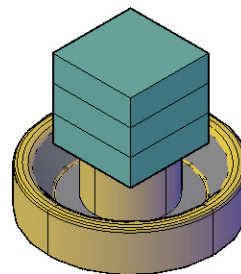
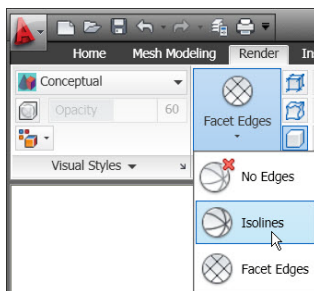




By default, the Conceptual visual style doesn't display the tessellation divisions. The 3D wireframe visual style does, but it's tough to always work in a wireframe style. To get the best of both styles when working with mesh objects, you can use the Conceptual visual style with isolines turned on. If the Conceptual visual style is not already set, you can select it from the View panel of the Home tab.

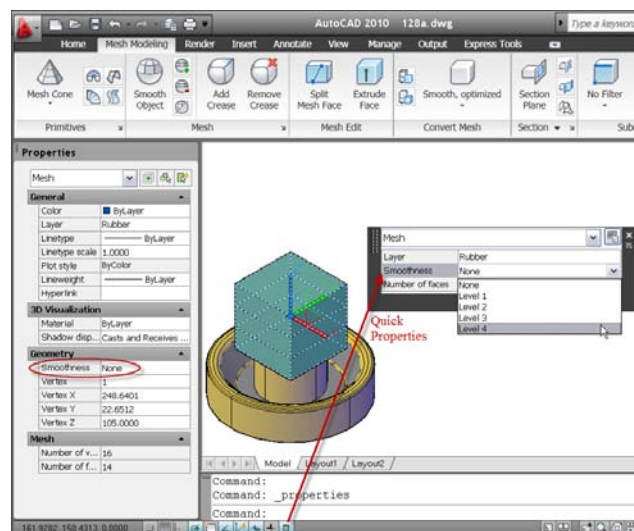


Then, on the Render tab, select Isolines from the Edge Effects settings panel. Now you can see the tessellation divisions in the length, width, and height!

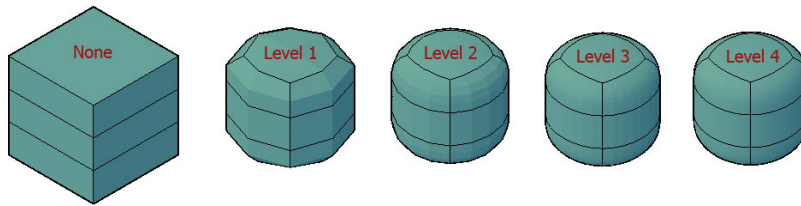


Editing a Mesh Box

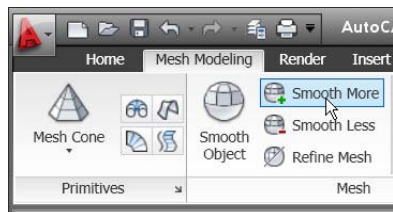
When you create a mesh primitive with a smoothness level of 0, each face (the surface between tessellation lines) is represented by a single flat facet. You can increase the smoothness level to create mesh objects that are more rounded. There are several ways to modify a mesh's smoothness level. If you select the mesh object, you can modify the smoothness level using the Properties palette or with Quick Properties.



You can choose from 4 levels of smoothness in addition to zero smoothness (None). As you increase the smoothness level, the number of facets per face also increases.

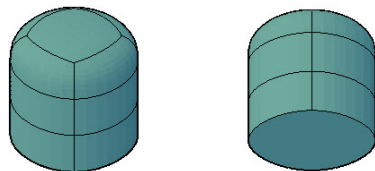


In addition to using the Properties and Quick Properties palettes to change the smoothness level, you can increase or decrease the smoothness one level at a time with the Smooth More and Smooth Less tools on the Mesh Modeling ribbon tab. For the air hockey paddle, set the smoothness at Level 3.



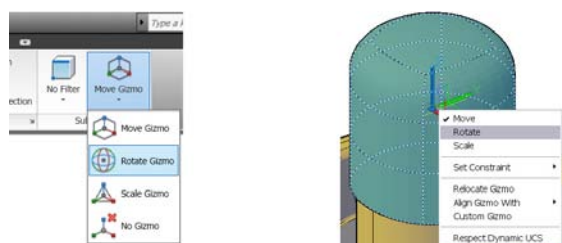
Sometimes you may want smoothness on one area of a mesh object but not on another. For example, on the rubber grip we want the top to be rounded and smooth. But on the bottom, where it fits over the plastic air hockey paddle, we want a sharp edge. You can produce this effect by adding creases to selected subobjects within the mesh. Subobjects can be faces, edges, or vertices. To make it easier to view the bottom of the rubber grip, I turned off the layer with the paddle geometry. Viewing the rubber grip from below, you can then hold down the CTRL key and click on the bottom face to select it.

The default option, Always, means that it will be creased at all levels of smoothness even if you increase the smoothness after applying the crease. I used the default Always option for the rubber grip.

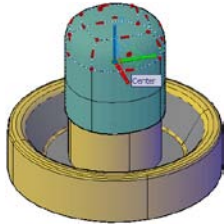


The top of the rubber grip is nice and rounded but we'll make it a little larger to fit better with an adult hand. To scale the top of the rubber grip, you can use the 3D gizmo. The gizmo is a handy 3D editing tool that was first introduced in AutoCAD 2007 and was dramatically enhanced in AutoCAD 2010. Before editing the rubber grip, turn on the Plastic layer so you can use the Center object snap of the sphere as the base point for scaling the grip.

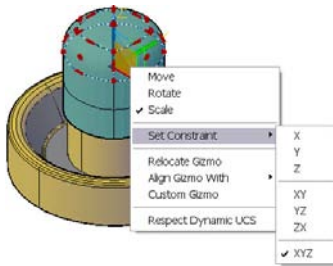
When you select 3D objects and sub-objects, AutoCAD automatically displays one of three 3D gizmos: Move, Rotate, and Scale (Scale is new in AutoCAD 2010). You can switch between these three gizmos using the Gizmo list in the subobject selection panel of the ribbon or by right-clicking on the gizmo icon. The gizmo you set remains persistent until you change it.



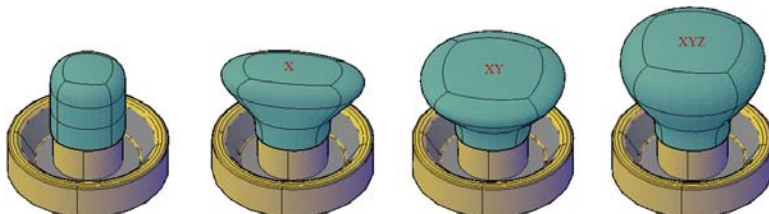
To modify the upper part of the rubber grip, press the CTRL key and create a window selection around the upper 1/3 of the mesh. Right-click and select the Scale option. The gizmo location changes, jumping from grip to grip, as you move the cursor over the selected objects. To move the gizmo to a specific location you can choose the Relocate Gizmo option from the right click menu and then snap to the location. Since we want the gizmo to scale from the center of the air hockey paddle we can snap to the center of the sphere.



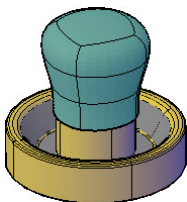
One of the primary benefits of the 3D gizmo is the ability to constrain the editing behavior to a specified axis or plane. You can specify the axis or plane by clicking on the gizmo when the proper axis/plane is highlighted or you can choose it from the Set Constraint option in the right-click menu.



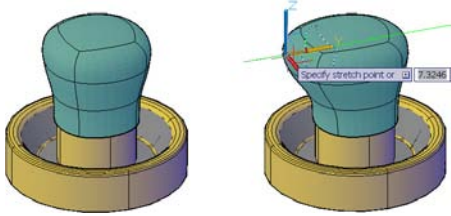
The following image illustrates the differences when constraining the scale operation for the rubber grip.



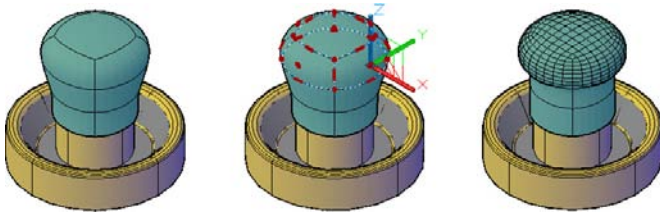
For the rubber grip, select XYZ to scale evenly in all directions and then enter a factor of 1.3.



The next step in designing the rubber grip for the air hockey paddle is to make the finger impressions. You can push and pull subobjects of a mesh (faces, edges, and vertices) to create that type of ergonomic/organic shape. However, the amount of detail that you can create is limited by the number of editable subobjects. Currently, the top of the rubber grip doesn't have enough editable subobjects to enable the detail required for the individual finger impressions.



The key for adding detail to a mesh is to have more editable subobjects in the area that needs detail. To do this, you can use the Refine tool. When you refine a mesh, AutoCAD converts the facets on the selected faces into editable faces with corresponding editable edges and vertices. Select the subobjects at the top of the rubber grip, which includes five faces, and then choose the Refine tool.



The rubber grip is taking shape. With the additional faces created by refining the top part of the mesh, we're ready to make the finger impressions. This is a creative process where you can experiment by selecting different subobjects (faces, edges, and vertices) and using different gizmos (move, rotate, scale) to form the finger impressions. As you select subobjects on the mesh, you can filter on which type of subobject to select.

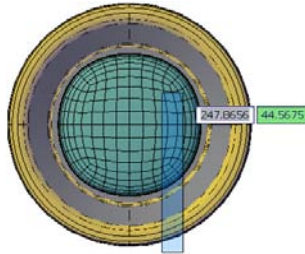


If you leave the subobject selection filter as the default, No Filter, your selection is less predictable. For example, if you pick on the corner of a face, AutoCAD could potentially select the face, edge, or vertex. When you use a window selection, it selects all the subobjects that fall within the window, which may or may not be what you intended. Set the subobject selection to Face to select the faces for making the narrow finger impressions.

The mesh object has many small faces. Even with subobject selection set, it can be tedious to select each face individually. Fortunately you can use implied windowing just as you do for 2D design. Since this is a 3D model, however, you might find it helpful to view the model from different angles as you select the subobjects. You can use the ViewCube to easily change to a standard viewpoint. As you pass the cursor over different areas of the ViewCube, they highlight indicating which viewpoint will be applied.



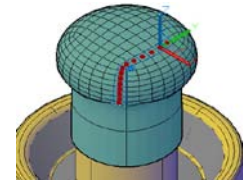
Select Top to view the air hockey paddle from a top view. Then, using implied windowing (left to right), I select the faces you want to edit.



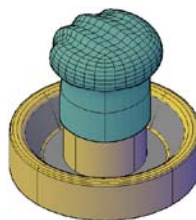
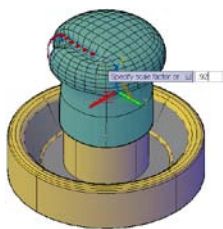
You can continue changing your viewpoint as necessary to view and select subobjects. The ViewCube includes hotspots at the very corners of the cube to display isometric viewpoints.



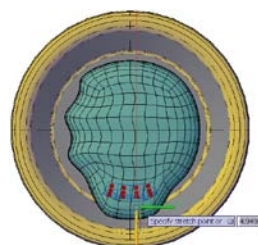
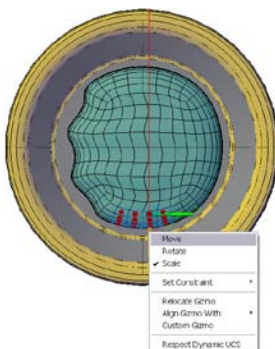
If you accidentally select unwanted subobjects, you can press the Shift key and click on the red grip of the unwanted subobject. The grip turns blue indicating that it's cold and won't be used in the editing operation. If your window selection missed some subobjects, you can press the CTRL key to continue adding subobjects to the selection set.



To create the finger impressions, I experimented with Move and Scale operations, relocating the gizmo to various locations. Ultimately, I decided to scale the selected faces towards the lower back of the rubber grip, making them more pronounced at the upper front. I also decided to constrain the scale operation to the YZ axis. Using a similar process, I created the remaining finger impressions.



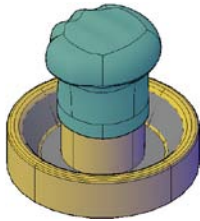
We're almost finished creating the free-form shape for the rubber grip of our air hockey paddle. As a final step in the mesh editing process, we'll use the Move gizmo to stretch one side of the rubber grip where the thumb wraps underneath it. You can use the ViewCube to set a viewpoint that makes selecting the appropriate faces easy. Ensure the 3D Move gizmo is active and then select the appropriate axis (in this case, the X-axis) to constrain movement.



Integrating Meshes with Solids

At some point you may want to integrate free-form mesh shapes with the precision solid shapes. For example, while we want the rubber grip to fit the shape of a hand, it must also fit snugly over the plastic air hockey paddle.

Conversion tools on the Mesh Modeling tab enable you to convert mesh objects to solid objects. You can select from different combinations of smooth or faceted; optimized or not optimized. First we'll look at smooth versus faceted. Notice the smoothness of the original mesh object is maintained when you choose one of the Smooth conversion options. If you choose one of the faceted options, the original smooth mesh faces are converted to flat faces, which can cause the model to become angular. When you choose the optimized option, coplanar faces are merged together. If you choose not-optimized, each face in the original mesh object is retained in the converted solid. After choosing the conversion option, you can select the Convert to Solid tool. There's also a Convert to Surface tool. I converted the rubber grip mesh to a solid using the smooth optimized option.

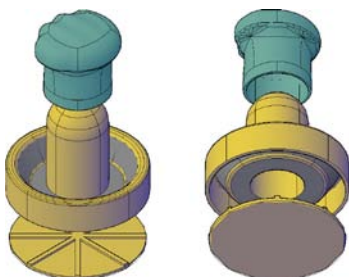


The rubber grip is currently a solid mass that passes right through the handle of the air hockey paddle, which you can see if you view the paddle from below. We need to remove the excess material from the rubber grip so that it can slide over the plastic paddle. First, we'll make a copy of the paddle and move it off to the side. Then, select the Subtract tool from the Solid Editing panel of the Home tab and select the rubber grip as the object you want to subtract from. Don't forget to press enter to end the first selection set (the objects you want to subtract from). Next, select the plastic paddle that you want to subtract from the rubber grip. The result is an object with two parts, the inside mass and the outside mass.



You can use the Separate tool on the Home ribbon tab to separate the rubber grip into two solids. Finally, erase the inside solid. Ta-dah... A composite shape with both style and precision!

To better visualize how the rubber grip and plastic air hockey paddle work together, turn on the Plastic layer and move the paddle down the Z-axis.



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 tool palette and it includes more than 300 materials 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. Access the Materials editor from the dialog box launcher in the lower right corner of the Materials panel on the Render tab.

To view materials on your model, you'll need to set the Realistic visual style (available from the upper left corner of the Render ribbon tab). The Realistic visual style displays the model with Isolines turned on. If you'd like to view it without Isolines, you can select No Edges in the Edge Effects panel of the Render tab.

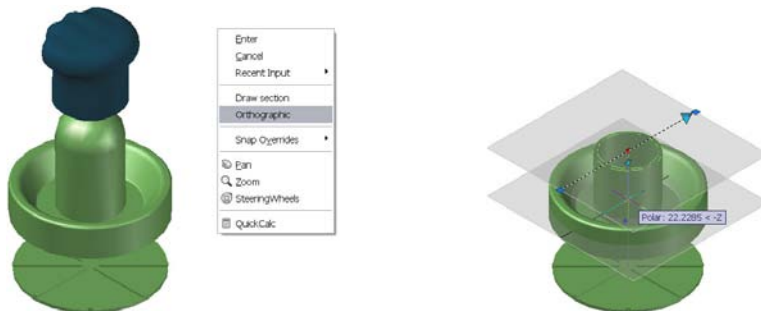


Creating a 2D Drawing from a 3D Model

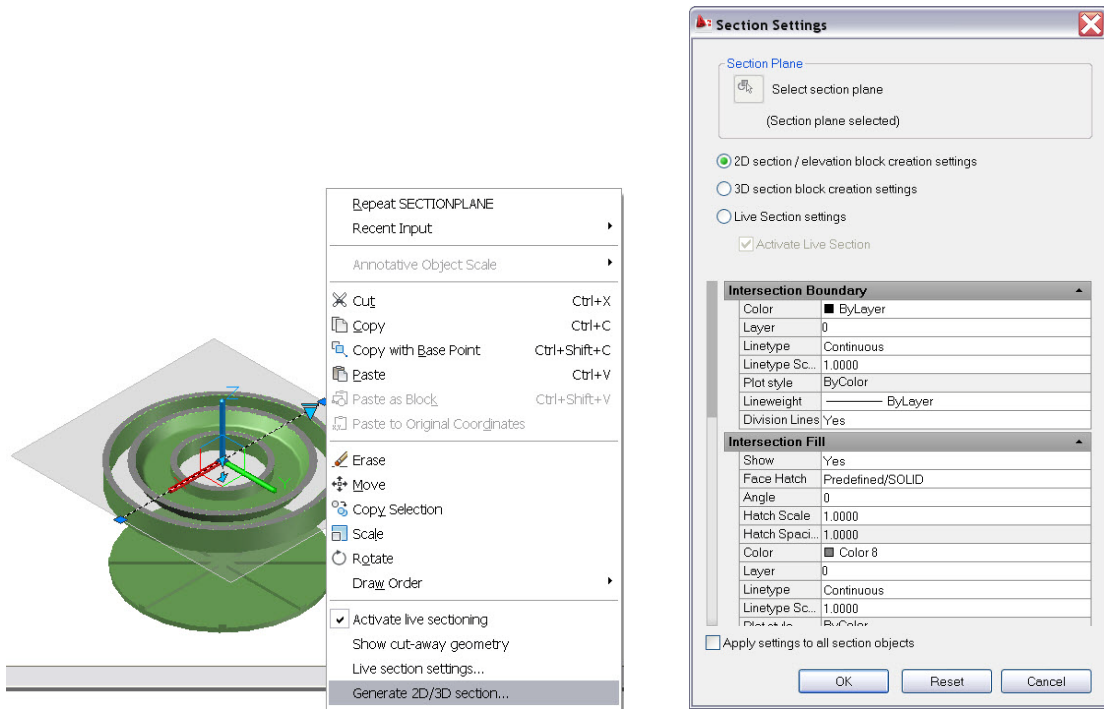
When you're ready to document your design, you can take advantage of your existing data by creating 2D sections from the 3D model. It's easy to do using the Section Plane tool, which is available on the Home Tab, in the Section panel.



You can define a section plane using various methods. You might want to pick points or select from standard orthographic options. If you choose the Orthographic option, you can then specify front, back, top, bottom left or right. AutoCAD automatically creates a section plane aligned with that view. You can then select the section plane and use grips to flip its direction, or change its location.



A right-click menu offers additional options including the ability to generate 2D or 3D sections from the model. You can control how sections are displayed using the Section Settings dialog box, which is accessible from the right-click menu.



Once you've created the 2D sections, you can use your familiar 2D drawing editing tools to add dimensions and create the necessary 2D documentation.

