Orthographic projection in autocad pdf

On the Home tab of the ribbon, click the Ortho View tab. Note: You can also click the Ortographic DWG tab in the Project Manager, open an orthographic drawing, and click New View on the ribbon. In select orthographic drawing, do one of the following, and click OK: Under Orthographic drawings in your project, click an existing drawing. Click Create New. In the New DWG dialog box, type a name in the File name box. Enter an author name. Click OK. Click OrthoCube in the drawing. You can use the handles to move and resize the cube, or you can use AutoCAD to move, rotate, and adjust accurately. The highlighted (red) panel indicates the current view, such as Note: Click a corner of ViewCube, if necessary, to access the grips you want on OrthoCube. On the ribbon, the Ortho Editor tab, do the following: Click a view from the drop-down list in the drop-down list in the drop-down list (for example, front). Note: By default, the temporary drawing that appears is set to zoom in on all items currently in the drawing. ViewCube is set to top view using the world coordinate system (WCS). Click on the corners of ViewCube to find an angle that lets you work with OrthoCube grips easily. In the Output Size list (for example, 1:16), click a scale in the Scale drop-down list. You can also adjust the scale when you place the orthodrat. (Optional) In the Output Size panel, click Paper Checker to make sure the selected scale is appropriate. (Optional) In the Library panel, click Save OrthoCube. In the Save View dialog box, type a display name and click OK. In the Create panel, click OK. Specify the location of the lower-left corner of the display port in the orthographic drawing area. Note: At the command prompt, you can also specify the exact coordinates of the drawing. You can also specify the scale of the view before you place the view in the drawing by clicking ? to open the Select Ortho View Scale dialog box. You can now annotate the orthographic drawing. On the Home tab of the ribbon, click the Ortho View tab. Note: You can also click the Ortographic DWG tab in the Project Manager, open an orthographic drawing, and click New View on the ribbon. In select orthographic drawing, do one of the following, and click OK: Under Orthographic drawings in your project, click an existing drawing. Click Create New. In the New DWG dialog box, type a name in the File name box. Enter an author name. Click OK. Click OrthoCube in the drawing. You can use the handles to move and resize the cube, or you can use AutoCAD to move, rotate, and adjust accurately. The highlighted (red) panel indicates the current view, such as Note: Click a corner of ViewCube, if necessary, to access the grips you want on OrthoCube. 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Note: At the command prompt, you can also specify the exact coordinates of the drawing. You can also specify the scale of the view before you place the view in the drawing by clicking ? to open the Select Ortho View Scale dialog box. You can now annotate the orthographic drawing. So far in this level you have learned the basic 2D AutoCAD commands. What you have learned at this level will be a very large part of what you use in your daily compilation. This tutorial is not going to teach commands, but will instead show a common technique that is used a lot in 'Mechanical' drafting. It will also ask you to think about what you draw and how it should be represented. Mechanical drafting is an area of the drafting world. In simple terms, it is used to describe the methods of drafting and designing machines, assemblies and in a nut shell, the parts used in everything from a fork to a Formula 1 race car. It does not include anything that involves buildings and structures (Architectural & Structural Preparation) or landscapes and roads (Civil preparation and engineering). Now just because you dream of becoming an architect, it does not mean that you should skip this tutorial. In fact, you will probably need to use these terms or read charts related to this topic (such as with Elevation Drawings). So read on! What is orthographic projection? If you look at the picture below, you will see a drawing for a part. It displays the object with a Top view, a cover view, and a page view. These three points of view are projected and aligned. As you've seen with the Orthographic Mode (F8) option, the lines are either horizontal or vertical - and in Orthographic projection, the views are mostly horizontal or vertical relative to each other. It is this alignment (projection) that makes the drawing easy to read. You'll also see an isometric view that is sometimes used to give a more visual look. This tutorial will not cover Isometric drafting, as shown in Tutorial 3-2. Save the drawings you do in this exercise for more practice in that lesson. The reason why this method is used is that you can take a designed Draw it, dimension it and then give all the necessary information to the manufacturer. In some only 2 views are needed, but for something more than a simple part, 3 or more views are needed. Very complex parts need 6 or more. There are 2 ways to decide which views are used and where they are placed in the drawing. I'll borrow some info from Wikipedia to explain this: First angle projection (European standards) In first angle projection, the object is conceptually placed in quadrant I, ie it loats above and before viewing aircraft, the planes are opaque, and each view is pushed through the object on the plane furthest from it. (Mnemonic: an actor on a stage.) Extending to the 6-sided box, each view of the object is projected in the direction (sense) of the sight of the object, onto the (opaque) walls of the box; that is, each view of the object is drawn on the opposite side of the box. A two-dimensional representation of the object is then created by unfolding the box, to see all the interior walls. This yields two plans and four increases. A simpler way to visualize this is to place the object on top of an upside down bowl. If you push the object down to the right edge of the bowl, the right page view appears. Image of object in box, overlooking the object projected in the direction of sight on walls using first angle projection. Similar image showing the box unfolding from around the object. Image showing orthographic views positioned relative to each other according to first-angle projection. Third-angle projection (USA Standards) In the third angle projection, the object is conceptually placed in quadrant III, i.e. it lurks under and behind the view aircraft, the planes are transparent, and each view is drawn to the plane closest to it. (Mnemonic: a shark in a tank, esp. that has sunk into the floor.) Using the 6-sided view box, each view of the object is projected opposite the direction (meaning) of sight, onto the (transparent) exterior walls of the box; that is, each view of the object is drawn on the same side of the box. The box is then unfolded to see all its exterior walls. A simpler way to visualize this is to place the object at the bottom of a bowl. If you push the object up to the right edge of the bowl, the right page view appears. Here is the construction of third angle projections of the same object as above. Note that the individual views are the same, just arranged differently. Image of object in box, overlooking the object projected in the direction of sight on walls using first angle projection. Similar image showing the box unfolding from around the object. Image showing orthographic views positioned relative to each other according to third-angle projection. Ok - that was some funny theory - thanks for reading it. One last bit of info before we get back to cad stuff. Since there are 2 kinds of standards, how do you know which is like when you have a drawing in front of you? There is a standard symbol that in the title block to specify which method was used. This symbol shows a simple simple and displays the projection. Think about which symbol represents which method, and then move your mouse over the images to see if you were correct. Because I am in North America and was trained to use the Third Angle Projection method, that is what I will show in this tutorial. The techniques of drawing are the same, it's just a matter of which direction you 'project' or draw the lines. Here is a short video for you that explains these concept visually. For these exercises, we start by looking at an Isometric drawing of an object and then drawing the front, page, and top views using the dimensions we get. At work, you may find that you get a part to measure and then draw, or you can design the part yourself. Here is the part that we will draw in this tutorial: This is a very simple example to get you used to the concepts. You will have more exercises at the bottom. Ok, the first question that you want to ask yourself is where should I start?. I recommend that you start where you have the most information. This will sometimes be the front or top - it depends on each drawing. In this case, I will start with the front page and draw it. You don't have to worry about sizing it at this point – wait until you have all your views drawn. Ok, it should have been easy enough, so now you can start drawing the top view. To draw the top, you need to 'project' the lines up. Draw lines up from the main points of your front view. Make sure you're on your Osnaps (include Quadrant). Now that you have the vertical lines, draw the horizontal lines. Be sure to allow enough space to draw the width (via OFFSET) and spaces between views. Now you have almost 2 views drawn. Trim the lines so you're left with only the lines you need. Stop and check to make sure you haven't forgotten any lines. It is very easy to miss some. Now it's time to jump forward and take a side trip. Read Tutorial 4-3 to learn about Linetypes because you will need them here. After reading the tutorial, return to the drawing and load the Hidden and Centered line types. These are necessary to add more information to the drawing. If all went well, you should be able to load linetypes and scale them (LTS with a value of 10 or 12) to fit with your drawing. What you see above are the finished Top and Front views. Do you understand why there are hidden lines? They are there to indicate that the hole (circle) is drilled right through the block. Where would you draw the lines if the hole was only drilled halfway through? The center lines are used to show the hole, and the arc has the same center point. These are both common and standard CAD methods, and you need to understand them. Another view to draw. This will be the side/right view. Can you imagine that yet? To get started on page view, you need to determine where it will be placed in In this example and using 3. 3. View. For precise placement, draw multiple projection lines. What you see in the picture above is that I established the upper right corner of my front view by projecting 2 lines. So I pulled up a 45-degree line from the corner. Now I can start projecting lines from my top view to create the page view. The line specified by the 'Project Down' manager will be the left side of the right page view. I would project another one from the other side of the top view and it would establish the width of the right side. Also note that by using this technique, the top and page views are the same distance from the banishment. From there I just have to project to the right from my front view. Almost there. Now just trim some lines and change the line type for the hidden lines. End goal is to draw this: Once you have all your lines, your center lines and your hidden lines, you are ready to dimension and add any notes that are needed. Think about what commands you used in this tutorial. You used LINE, CIRCLE, OFFSET & TRIM. You also use Layers and then learned about Linetypes. What this shows is that you don't have to use a lot of commands, but it's your knowledge of how these commands work that makes your reputation as a cad user. Additional exercise: Draw the missing view on the right side and the other views (except Isometric) for this drawing. Extra practice: Here's a scan from an old (1919) compilation book that I found on Google Books. This image has 4 separate exercise to keep you busy. Draw what you get, and then draw the missing view. Additional exercise: Draw the 3 views needed for each of these blocks. Extra 1 - Extra 2 Just one more thing I should mention. This method of compiling can be used in other ways as well.3D channel in AutoCAD On the right is a simple piece of channel work pulled in AutoCAD. It is a straight piece that has been cut at an angle so that it can be connected to another piece to form a bend. Drawing in 3D is not very difficult when you know how. But if I sent it to the manufacturer, he wouldn't really know where to start. So I would have to send him a drawing of the pipe rolled out. This would give him a template to cut the shape and manufacture the part. Think about what this shape would look like if it were flat. How would you draw it? How would get the basket correct? Do you think that sounds like a job for Orthographic Projection? Yes it does. The drawing below shows how this could be drawn to show all aspects of this part. I have the diameter of the pipe and the length to the top and length to the bottom of the cut. Since I have the diameter, I also have the circumference. The circumference tells me the length of material that will be needed to make the piece. The part is drawn in Magenta, and the projection lines are in Blue. What I needed to do was use the DIVIDE command to split both the circle and line representing the perimeter. Then it was a matter of projecting down and to the right from the circle, the circle, up from the perimeter line. This gave me a grid to use for SPLINE to create the basket. The additional view was drawn with a pair of projection lines and an ELLIPSE. Try this if you like and follow the command line of the commands you don't know. It can be hard, but take your time. Video: Orthographic projection - Part 2 2