Click Here

Click Draft Angle .The Draft Definition dialog box appears. An arrow appears on a plane, indicating the default pulling direction. Select Selection by neutral face to determine the selection mode. Select the upper face as the neutral element. This selection allows the application to detect the face to be drafted. The neutral element is now displayed in blue, the neutral curve is in pink. The faces to be drafted are in dark red. The default angle value is 5. Enter 7 degrees as the new angle value. The new angle value is displayed in the geometry. Click Preview to see the draft to be created. Click More to access additional options. To know how to use the options Parting Element and Draft Form, see Drafts with Parting Elements in Part Design User guide. Click the Limiting Elements box. While drafting a face, you can limit it by selecting one or more faces or planes that intersect it completely. Select Plane.1 as the limiting element. The arrow points to the portion of material that will be kept after performing the operation. Select Plane.2 as the second limiting element. Note that the number of limiting elements you select is indicated in the dialog box, just in front of the Limiting Elements box. Click the arrow to reverse its direction, and therefore retain the opposite side of the feature. Click OK to confirm the operation. The faces are drafted but the part areas included between both limiting planes have not been modified, as specified through the limiting element option. CATIAs innovative CAD software allows users to create a wide range of precision parts quickly and effectively. But while CATIAs modeling tools may allow you to check the functionality of your finished components, you must also ensure that your parts can be successfully removed from their molds. With CATIA Draft Analysis, draft angles can be evaluated and optimized for your design for easy ejection. Here is a brief guide on how to make use of this intuitive feature.The Keys to CATIA Draft AnalysisTo begin, you need to have a finished 3D model in CATIAs Workbench. From here, you enter Material Mode and access the CATIA Draft Analysis box. Amongst other details, the box will show the color scale and the upper and lower limit values for draft analysis. With this tool, you can determine whether your draft angle is positive or negative and thus where the parting line and/or shutoff for the part should be. We use these insights to specify surfaces, pull direction, and the draft angle required during the physical prototyping stage.The analysis tool works by assigning different color ranges to the 3D model. By following these signals you can confirm that draft directions at each point fall within the specified parameters for your part. These parameters can always be adjusted through the unit settings. It is important to note that color accuracy can vary according to the video card used, so if youre seeing a color representation that doesnt match up to the stated values then this may just be a display error.You can perform a draft analysis on any surface of your model. In order to select a surface, check the On the Fly box and hover over the intended surface. Once you have highlighted a surface, corresponding details will show up in the draft analysis box. The parameters option should show the direction in which the component will be ejected from its tool. The pull will always be perpendicular to the plane of your surface. You can click the Inverse button to reverse the draft direction.Once the direction has been set you can set minimum limit against which the draft angle will be measured on various surfaces. For example, if you set this parameter to 3 degrees then all surfaces with an angle of 3 degrees or higher in the direction of the pull will show up as positive, while all surfaces with an angle of 3 degrees or higher away from the pull will show up as negative. You can see the colors that correspond to each value in the quick mode option box.Remember, there will always be a negative and positive face to your part, your sole purpose here is to identify any surfaces that require further adjustment to fall in line with their corresponding plane.As you can see, theres a lot to unpack when it comes to CATIA Draft Analysis. For more information on how we provide outsourced CATIA design services or an estimate on our CAD services feel free to contact us. Were always ready to help. Contextual commands creating the pulling directions you need are available from the Selection field: Create Line: For more information, see Creating Lines. Create Plane: see Creating Planes. X Axis: the X axis of the current coordinate system origin (0,0,0) becomes the direction. Y Axis: the Y axis of the current coordinate system origin (0,0,0) becomes the direction. Z Axis: the Z axis of the current coordinate system origin (0,0,0) becomes the direction. If you create any of these elements, the application then displays the corresponding icon next to the Selection field. Clicking this icon enables you to edit the element.Click Draft Analysis in the Shape Analysis toolbar. The Draft Analysis dialog box is displayed. Set up your analysis: Select a surface for analysis. The quick analysis is displayed on the surface.The draft direction is the w/z axis of the compass. In the Mode area of the dialog box, select Full Analysis Mode . The color scale and range of values in the Draft Analysis.x dialog box are expanded.The corresponding colors are displayed on the surface. Optional: Modify the colors and/or values in the color scale (right-click a color or value). In the Display area of the dialog box, select On the Fly and move the pointer over the surface. The following information about the current pointer location is displayed:Green arrow, N: The normal to the surface Red arrow, D: The draft direction Blue arrow, T: The tangent to the surface Circles: The plane tangent to the surface Value (reverse highlighted): The angle between the draft direction and the tangent to the surface (the value is expressed in the units set in Tools > Options > General > Parameters > Units tab) The cone Value (unhighlighted): The cone angle (the angle between the axis of the cone and the draft direction D) As you move the pointer over the surface, this information is updated dynamically. Note that you can activate the On the fly option even when not visualizing the materials. It gives you the tangent plane and the deviation value. Click Inverse to reverse the draft direction. This command affects all selected elements. Right-click the cone angle to display the Angle Tuner dialog box and then modify the angle using the up/down arrows. The value is automatically updated in the color scale dialog box, on the geometry and the cone diameter is adjusted.The value in the color scale dialog box (16.961deg in the example shown below) is highlighted in a different color . Right-click the red arrow, D (draft direction) to display a contextual menu.From the contextual menu you can: Hide/show the cone. Hide/show the angle. Hide/show the tangent. Lock/unlock the analysis position. Keep the point at this location (a Point.xxx appears in the specification tree). Define a new draft direction and then lock the direction: When you have finished working with this command, do one of the following:To create the analysis, click OK. The analysis is added to the specification tree where it is identified as Command_Name.Analysis.x.The analysis will be updated automatically whenever you modify any of the elements in the analysis (for example when you deform an element using control points). To abandon the analysis, click Cancel. Note: Note that settings are saved when exiting the command, and redisplayed when you select Draft Analysis again. Be careful, when selecting the direction, not to deselect the analyzed element. A draft analysis can be performed just as well on a set of surfaces. If an element belongs to an analysis, it cannot be selected simultaneously for another analysis; you need to remove the current analysis by deselecting the element to be able to use it again. In some cases, even though the rendering style is properly set, it may happen that the analysis results are not visible. Check that the geometry is up-to-date, or perform an update on the involved geometric elements. The analysis results depend of the current object. May you want to change the scope of analysis, use the Define in Work object contextual command. Click Draft Analysis in the Shape Analysis toolbar. The Draft Analysis dialog box is displayed. Set up your analysis: Select a surface for analysis. The quick analysis is displayed on the surface.The draft direction is the w/z axis of the compass. In the Mode area of the dialog box, select Full Analysis Mode . The color scale and range of values in the Draft Analysis.x dialog box are expanded.The corresponding colors are displayed on the surface. Optional: Modify the colors and/or values in the color scale (right-click a color or value). In the Display area of the dialog box, select On the Fly and move the pointer over the surface. 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Here the part should get easily removed from the panel dies (in case of Sheet metal part) or Mold (in case of plastic).No parts can be ejected from the tool if it is having a straight 90-degree profile from its base surface. A small draft angle is provided to each and every area of the component in order to get easy removal of the component from the tool.Here we are providing a direction in which components should eject in order to get easy removal of components from the tool. In other words, it also lets us know whether the model is having a draft angle and how much is the draft angle from the tool direction.The Types of Components for Draft Analysis Draft analysis is performed on the components where the ejection of materials happens after the material is made inside the tools. In the case of Automotive Sheet Metal manufacturing, the blank sheet after turning into automotive panels like Hood, Fender, Roof, Side Doors, Back Doors etc. need to be removed from Panel Dies. If a considerable amount of draft angle is not provided, the removal of components from the tool will be difficult. Its the same in the case of mold(plastic component) after getting filled inside the core and cavity, it needs to get ejected easily with the help of ejectors. Procedure To Perform A Draft Analysis in Catia Step 1Change the view mode to Customize View Parameters and select the Material option from the Mesh. Step 2 Next, you need to create a rough line, without considering a particular direction or parameters. The direction of the line may vary from component to component. To make the Draft Analysis easier, it will be better if the line is created from the CG point. For that, we need to select the measure inertia option from the measuring toolbar. After that select the body from the tree for the Measure Inertia dialogue box. The CG point is represented inside a box on the intersection point of red, blue, and green. If we want to get to that point permanently, we need to select the created geometry and select both the center of gravity and the Axis system. Step 3 Create a random line from the CG point using the point and direction option from the line command. Step 4 Draft Analysis options are available inside the following workbenches:Part Design Generative Shape Design Wireframe and Surface for BuildingFreeStyle Shaper Insert > Analysis > Draft AnalysisThe draft Analysis dialog box with colour scale (with quick analysis mode) appears on the screen. So the first mode option is the quick analysis mode and the second is the full analysis mode. Here the quick analysis mode is used. Change the value of green according to the draft angle provided to the part. Here for automotive sheet metal components, the ideal draft angle is 7 degrees. Step 5 Drag the compass and place it over to the line. Select the compass icon from the direction of the Draft Analysis dialog box. The compass may get misplaced from its original position after clicking on the compass icon. Select the compass again and place it over to the line properly one more time. Step 6 After placing it, click anywhere on the part. The appearance of the component will change afterwards. To get a clear picture, either we need to adjust the compass by holding the marked region of the compass and dragging it towards the opposite direction or by selecting the inverses draft direction option inside the direction of Draft Analysis Step 7 After getting the green colour on the component, try to adjust the compass using the knob, and adjust the direction of the compass to get green on the entire top panel region. Try zooming in and checking if each and every region of the panel is showing green, if in some regions, red or blue is showing instead of green, those negative regions are below 7 degrees and have no draft applied on those regions. Those blue and red negative regions need to be changed. Step 8If we are ok with the result, right-click on the compass and lock its current position using lock current orientation. Step 9 Click ok to the draft analysis dialog box to close it. Double click on the line and right click on the direction option and change the line direction to compass direction. The line will get oriented along with the previously locked compass draft direction. After the use, right-click again on the compass and unlock it. Also, reset the compass using the shortcut Alt + v + r. The compass will get reset to its original position. Later we can provide thickness to the line and change the appearance colour to green by right-clicking and selecting properties (which is an ideal colour used to represent draft direction). Exploring More Display OptionsShow or Hide the Colour ScaleThe first option inside the display is to show or hide the colour scale. This option is used to change the draft angle value according to the type of component used. Analysis under the Running PointThis option is used to highlight the exact draft angle of that face. For that, we need to move the point along the faces. No Highlight Representation / Light EffectNo Highlight Representation and Light Effect options are used to change the view type to get a clear highlight of the Draft Analysis result. Draft Analysis of Plastic ComponentsThe ideal draft angle for the plastic components is 3 degrees. This may vary according to the functional requirement. Apart from the draft angle consideration, the remaining procedures for the draft analysis are similar to the above-mentioned steps. The draft angle provided for the plastic features is 0.5 degrees. This may also get varied accordingly. The above figure shows the tooling direction, slider direction, dog house slider direction, and snap slider direction of the plastic component. This is determined with the help of draft analysis with a draft angle of 0.5 degrees. Author Note that settings are saved when exiting the command, and redisplayed when you select the Draft Analysis icon again. Be careful, when selecting the direction, not to deselect the analyzed element. A draft analysis can be performed just as well on a set of surfaces. If an element belongs to an analysis, it cannot be selected simultaneously for another analysis, you need to remove the current analysis by deselecting the element to be able to use it again. In some cases, even though the rendering style is properly set, it may happen that the analysis results are not visible. Check that the geometry is up-to-date, or perform an update on the involved geometric elements. The analysis results depend of the current object. May you want to change the scope of analysis, use the Define in Work object contextual command.

What is draft analysis in catia. Draft angle catia v5. How to use draft angle in catia v5. How to check draft analysis in catia v5. Catia draft angle. How to measure draft angle in catia v5. How to check draft angle in solidworks.