I'm not a bot

 Click the Replace Face icon . Select the horizontal face on the left. This face, which is the replacing face is now displayed in blue. Because it is tangent to the face produced by a fillet operation, that face is also selected and in turn includes the vertical face. Three faces are then selected as indicated by the blue color. Select the horizontal face on the right. Three tangent faces are selected as the faces to be replaced and are shown in purple: Click OK to confirm the operation. The result you obtain shows that replacing these faces affects the fillet radius value to the left. You can completely remove the particular face by selecting it from the 3D shape.Click Remove Face in the Dress-Up Features toolbar (Remove/Replace Face sub-toolbar) in the Part Design workbench. The Remove Face Definition dialog box appears. Important: For the Generative Shape Design workbench, Click Remove Face in the Operations toolbar (Trim-Split sub-toolbar). The Remove Face Definition dialog box is displayed as shown: In the Faces to remove field, select the inner face as shown. The face turns purple indicating that it will be removed. Important: In the Generative Shape Design workbench, the Support field is automatically filled in with the surface whose face belongs to. To modify the support, you need to remove all faces first. The contextual menu on the Faces to remove field displays the following contextual commands: Clear Selection: removes all selected faces from the selection. Tangency Propagation: includes all faces tangent to the selected faces from the selection. In the Faces to keep field, select both faces as shown.The faces turn blue, indicating that they will not be removed. Select the Show all faces to remove option to preview all the faces adjacent to the purple face that will be removed. Warning: In the Part Design workbench, faces to remove and faces to keep must belong to the same body while in the Generative Shape Design workbench, they must belong to the same support surface. If not, an error message is issued. Click OK to confirm.All of the faces have been removed. The new feature identified as RemoveFace.XXX is added to the specification tree. All Frameworks Object Hierarchy This Framework Indexes Represents the Replace Face operation. It replaces a face or a set of faces obtained by tangency continuity by a replacing element, such as a surface or a face or a skin. RemoveFace Returns the face to be removed. SplittingSide Returns or sets the splitting side . AddRemoveFace Sets the face to be removed. AddSplitPlane Sets the replacing element. DeleteRemoveFace Remove the face to be removed. o Property RemoveFace() As (Read Only) Returns the face to be removed. o Property SplittingSide() As Returns or sets the splitting side . The splitting side is the side of the body kept after the splitting. A positive side refers to the same orientation than the splitting element normal vector. o Sub AddRemoveFace(iRemoveFace) Sets the face to be removed. o Sub AddSplitPlane(iSplitPlane) Sets the replacing element. o Sub DeleteRemoveFace(iRemoveFace) Remove the face to be removed. Click the Replace Face icon . Select the horizontal face on the left. This face, which is the replacing face is now displayed in blue. Because it is tangent to the face produced by a fillet operation, that face is also selected and in turn includes the vertical face. Three faces are then selected as indicated by the blue color. Select the horizontal face on the right. Three tangent faces are selected as the faces to be replaced and are shown in purple: Click OK to confirm the operation. The result you obtain shows that replacing these faces affects the fillet radius value to the left. Bruce Clarke How do I use replace on a piece of surface data containing hundreds of faces which has had a change and now the faces of the surface as well as edges and vertices are no longer one for one from old to new. So the replace function has you pick the surface you want to replace and then select the surface to replace it with. In this instance I want to replace rev-38 with rev-46, see Replace_1.jpg. The Replace viewer opens after selection, see Replace_2.jpg. In the viewer you need to pick each element of that surface in the left window then select the corresponding element in the right window. It can be literally hundreds of mouse picks. With a complex model that has thousands of pointed elements after this geometry it can be a huge time saver. In this example the customer has changed the geometry so much that the new file is missing elements that the old file had. See Replace_3.jpg, you can see that Edge.52 does not exist in the new geometry. It has 3 pointing elements. Some pieces of geometry have many more pointed elements, see Replace_4.jpg. If I go through the selection process for all the geometry that corresponds I end up with elements I cant select to replace because they dont exist. If you click OK you get the error that you have not replaced all the elements, Replace_5.jpg. Once you say yes to this the specification tree is filled with errors, (Replace_6.jpg) as all the pointed elements are lost due to missing elements of geometry. Jack K Answered on 29 Jan, 2020 11:24 PM Generally speaking, you should be able to replace the old surface with the new one. But, it depends on several factors. Since you're asking the question, I guess you tried it and it didn't work. What did the error message say? Could you include a picture of the tree so we can see what the old surface is? plus a couple questions:- are you using Hybrid mode?- are there any children of the old surface, and do you want them to be children of the new surface?- is the new surface a child of the old surface? Have you ever found yourself working on a 3D model, only to realize that there were some unwanted faces that need to be removed? Fret not, CATIA V5, the powerful 3D modeling software, is here to the rescue with its handy Remove Face tool. In this blog, well walk you through the process of using this tool effectively, allowing you to streamline your design process and create more polished models. Step 1: Accessing the Remove Face tool To begin, ensure that you are in the Part Design mode withing CATIA V5. Look for the Remove Face tool icon, which can be found in the toolbar. Once youve located it, click on the icon to open the Remove Face tool dialog box. Step 2: Selecting the Faces to Remove Within the Remove Face tool dialog box, youll find a variety of options to manipulate your model. To remove unwanted faces, simply select them by clicking on each face individually. If you have a group of connected faces that need to be removed together, the Select Connected Faces option will be your best friend. With just a click, you can efficiently select multiple faces. Step 3: Previewing and Finalizing the Removal Before committing to the facial removal, its always a good idea to preview the changes. By clicking the Preview button in the Remove Face tool dialog box, you can see exactly how your model will look after the faces have been removed. This gives you the opportunity to ensure that the results align with your design vision. Once you are satisfied with the preview, its time to finalize the removal. Click OK in the Remove Face tool dialog box and watch as the selected faces disappear from your model. These may be faces that need to be removed because of a design change, or to clean up your model. Congratulations! Youve successfully learned how to utilize the Remove Face tool in CATIA V5. By integrating this tool into your workflow, you can save valuable time and effort by swiftly eliminating unwanted faces from your 3D models. Whether you are designing mechanical parts, architectural structures, or intricate prototypes, the Remove Face tool is an indispensable asset that enhances your design precision. Embrace the power of CATIA V5s Remove Face tool and unlock your creative potential. Streamline your design process, eliminate imperfections and craft exceptional 3D models with ease. Let your imagination run wild and watch as your designs flourish. I hope you found this tutorial helpful. If you have any questions or would like to explore more CATIA V5 features, feel free to reach out. Happy designing! Technical Training EngineerJoshua is a resident instructor for a large automotive OEM, working out of their Advanced Development Center. He implements customer specific training and brings eight years of industry experience to his classes. Joshuas areas of expertise include Advanced Part and Generative Shape Design, VPM Navigator and Generative Drafting. More Content by Joshua Sands Shell, Thickness, Thread, Tap, Remove and Replace Face COMMAND || CATIA V5 Tutorial || Part DesignPart Modeling -TOOLBAR - DRESS-UP FEATURESCOMMAND - Shell, Thickness, Thread, Tap, Remove and Replace face, Chamfer1. Shell, 2. Thickness, 3.Thread / Tap4. Remove and Replace face5. Chamfer Now discussed in detailed one by one,1. Shell - Shelling a feature means emptying it, while keeping a given thickness on its sides. Shelling may also consist of adding thickness to the outside. This task shows how to create a cavity.2. Thickness - Sometimes, some thicknesses have to be added or removed before machining the part.3. Thread / Tap -Thread must apply always on Outer Diameter And Tap must be applied inner Diameter.The Thread-Tap capability creates threads or taps, depending on the cylindrical entity of interest.4. Remove Face And Replace Face -Remove Face -When parts are far too complex for finite element analyses, there is a way of making them more simple.This tutorial shows you how to simplify apart by removing some of its faces.Replace Face -This tutorial shows how to modify the shape of a part by extruding one of its faces up to an external surface.Chamfer - Chamfering consists in removing or adding a flat section from a selected edge to create a beveled surface between the two original faces common to that edge. You obtain a chamfer by propagation along with one or several edges.This tutorial shows how to create two chamfers by selecting two edges. One case illustrates how the material is added, the other case shows how the material is retrieved.----------To download assignments drawings CATIA V5 Tutorial - CATIA V5 - Settings Dress-up feature toolbar----------To know more about cad software and Mechanical concepts, Please subscribe and press the bell icon to get the latest updates.Subscribe | Like | Share----------Thanks for Watching ...

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