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Drawing views not showing in solidworks

I'm not having a problem with my drawing views. What I design is similar to a previous design, so I opened the previous design and made a save as a copy and checked the box to include all the reference components. Then I tweaked the new design to the correct specifications and updated my painting. Then I closed the painting and opened it again at a later date. The old reference design was visible in terms, but the dimensions pointed to the NEW design. There are no errors in drawing or in models. WHAT I TRIED THAT DIDN'T WORK:- Rebuild Button- CTRL+B and CTRL+Q- Open assembly and rebuild/go back to drawing- Uncheck Automatic View Update/Rebuild/Re-Check Automatic View Update- Check for display driver (it was up to date)- Close/Re-Open Solidworks- Restart computerWHAT I TRIED THAT DOES WORK:- Change the Display StyleIf I change from Hidden Lines to other display state, new geometry appears. Then I can change to any display mode I want and the geometry will still show right. The problem is that when I change to the original style of the show and save the painting, closing and reopening the painting will put me in square one: Solution: Change display style, save drawing. Then change the display style to the preferred style. Save again, I couldn't find the solution online, so hopefully this helps someone have the same problem. Mark Burke Answered on 19 Aug, 2019 10:54 AM If the source file has been moved from its original location, Solidworks will not know where to find the file associated with the drawing. It will still be relevant but will not show up. Try opening the related section and then painting, it should show. Then just save everyone and the forum will be restarted. Discover how to adjust your painting views in SOLIDWORKS by hiding pieces without affecting or how many organs there are in it. Hitfaster (mechanical) (OP) 6 February 12 07:57 I just opened a painting, which was sent to a client a week ago and was good (showing four views of a section) to find that all my views now just dash boxes with crosses through! What has gone wrong?! The part is in the same folder as before (all the pieces and drawings are in one folder) and although I re-name before sending to the client, I thought I'd done it through SW, not just windows. Help needed asap! Thanks guys! Thank you for helping to keep engineering forums free of inappropriate tips. Engineer staff will check this guidance and take appropriate action. Page 2 of the book - Functional examples using metal 3D printed functional samples are a key step in product development “ they give engineers a chance to test new ideas and designs while also revealing how the product will stand up to real-world use. And when it comes to functional prototypes, 3D Rewriting the rules what“™ is. 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Sheet metal formation is the most cost-effective method of formation today to make parts in large quantities. Download now don't you just create a hole wizard feature in your SOLIDWORKS section and still see cosmetic threads? Or are you seeing the cosmetic theme in the assembly section/surface but not in painting? THIS ARTICLE COVERS ALL THE POSSIBILITIES WHY YOU MAY HAVE ENCOUNTERED MISSING COSMETIC TOPICS IN SOLIDWORKS, WHY THEY HIDE, AND HOW TO FIX THEM. The level part of the review is the first thing we need to check out is that the 'cosmetic theme' option was selected in the Hole Wizard feature under 'Options'. This is the middle icon, as shown in the image. If the cosmetic subject option is selected and still does not appear, double-check the document properties. To get there, go to Tools > Options > Document Properties. In the details section, there are two options that need to be chosen, 'cosmetic threads' and 'show annotations'. If you're looking to add a shady view of cosmetic threads in the hole, the 'Shade Cosmetics' option also needs to be chosen. These options can also be changed via the 'Annotations' folder in the FeatureManager tree. To get there, right-click on the 'Annotations' folder and select 'Details', this will provide pointed properties where the options 'cosmetic topics', 'shaded cosmetic threads' and 'show annotations' are also selectable. These settings are connected to the settings in the Details section of the document properties, so if they are turned off in one, it will reflect the change in the other. If your settings are all good to go and you still don't see your cosmetic yarns, annotations may be set to hide. To check it out, go to View > Hide/Show. Make sure it's all The icon is selected (depressed), and 'hide all sorts'. This will show all the annotations, including their cosmetic topics. Assembly level check holes created at the assembly level similar to the part of the surface we are going to examine the document properties (Tools > Options > Document Properties > Details). You want to ensure 'cosmetic topics' and 'show annotations' are selected. Again, these selections in the document properties are connected to the Annotation properties. Also, like part of the level, you want to check the 'Hide/View' options. Here you want to make sure that 'High Level Annotations' is chosen (depressed), and not 'hide all sorts'. Holes created on the surface of the part if cosmetic threads were added on the part surface but did not show up on the assembly surface, there are a few things to check out. In the assembly file, in the Properties Documents (Tools > Options > Document Properties > Details), if the 'View Forum Description' is selected and 'Use assembly settings for all components' is selected, the cosmetic topics do not show. To ensure the selection of both, or the selection 'use assembly setting for all components'. Next, in hide/show options, make sure the components are selected (depressed), and hide not all types. Drawing the check level by default, cosmetic threads appear in the model display for part files. However, they need to be handy to display the input assembly model. To do this, go to the Annotations tab and select model items. Then select a show and a 'Show Destination(s)' section will appear with the desired view, and under 'Annotations' you want to select the cosmetic theme icon (as shown in the below screenshot). Then you should be good to go! I hope you found this SOLIDWORKS tutorial useful. Check out more tips and tricks below. solidworks tips ɪmp;#x201c; tricks How to... Use the Import Detection Tool remove solidworks toolbox flag setting format movie settings... Editing mirror body management external resources in the library of photor pieces using Misaligned pairs there are a few things that can affect the display of a cosmetic theme in your SOLIDWORKS parts, assembly, or painting, and each of them varies depending on the display component, feature and even the environment you are working on. This SOLIDWORKS tutorial will run you through all possible causes and show you how to fix your cosmetic issue forever. Editing holes and ensuring the cosmetic theme is clear your first need to double check that the features you are using are enabled by the option for the topic. If this is a hole wizard, or similar features, edit it and check that the cosmetic theme is active. It's also a good idea to tick with the contact option enabling the notes in a drawing, as highlighted below. You've checked this out, making sure this feature doesn't repress in the design tree. You need to expand the feature to see if the case is grey and suppressed. If you want to suppress it, right-click the item. Check the details page in the next location options to check the Options menu. Browse to tools menu, then Options>Document Services>Check on this screen the theme option will show the outline of the theme in our model. Shady makeup threads tick boxes show the shady look of yarn inside the hole. Right-clicking on the Annotations folder of the same options above can be accessed by right-clicking on the annotations folder in the Design tree. There is a detailed option that appears in the right-clicking menu. You should also select display annotations option in the same right-click menu. It needs to be happy in order for any annotations to be shown. These options are stated in the section, assembly and drawing. This allows visibility to be controlled in all environments. Displaying all items in the Display menu for parts and should make sure that the option of displaying all anouches (sections) or displaying high-level indexing (assembly) is turned on. Model items for assembly drawings if you create a drawing of the assembly - and you're in the drawing environment - you'll notice that cosmetic topics are hidden by default. For their unhide you must use the model items command to import them from the assembly. This command can be found on the Annotations tab in a drawing. Once activated there is an option for cosmetic topics, which can be found under the Annotations section. From here you have 2 options: Set the source/destination to the selected features and select the alternative hole feature of the source/destination set to the whole model and click on one of the drawing views. After the perfect hit the green tick. SETTING YOUR OWN COSMETIC THREADS TO HIGH QUALITY SOLIDWORKS ALLOWS YOU TO DETERMINE THE QUALITY OF THREADS DISPLAYED IN ONE QUERIES. To find this option, select a view and scroll down in the property manager on the left. At the bottom of the panel you'll find the option to adjust your cosmetic theme screen to the top, or draft quality. High quality beauty theme displays detailed fonts and trimming of threads. Basically, it only shows a portion of the cosmetic theme. However, please be aware that this can sometimes slightly reduce the performance of your software when working on painting. Make sure that displaying the theme of cosmetics in painting displays sometimes a cosmetic theme can be shown in one painting show, but not the other. In this case, cosmetic yarn may be hidden in the design tree. To correct this problem, you need to find a view that doesn't show the theme of cosmetics properly. Scroll down to this view, expand items up Find the hole you want. You should see the hole subject, if you right-click this case you should be able to show the subject if it is hidden in the drawing. The theme of your cosmetic issue should now be fixed! IF YOUR COSMETIC ISSUE ISSUE IS MORE SPECIFIC THAN THE SOLUTIONS DESCRIBED HERE, WE RECOMMEND CONTACTING YOUR SOLIDWORKS VENDOR FOR HELP. IF YOU ARE A CUSTOMER INNOVATION SYSTEMS WITH A SOLIDWORKS ACTIVE SUBSCRIPTION, GET IN TOUCH WITH OUR AWARD-WINNING SUPPORT TEAM NOW AND WE WILL BE MORE THAN HAPPY TO HELP! Hope you found that useful! Have you seen our blog archive where we posted plenty of useful articles? WE ALSO HAVE AN INCREDIBLY VIDEO LIBRARY FULL OF EASY-LOOKING VIDEOS ON A NUMBER OF TOPICS INSPIRED BY OTHER SOLIDWORKS USERS - TAKE A LOOK. ALSO, DON'T FORGET TO FOLLOW INNOVATION SYSTEMS ON TWITTER FOR DAILY BITE SIZE SOLIDWORKS TIPS, TRICKS AND VIDEOS. Share this page on social media

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