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How to measure in catia v5

How to measure a tube's bend radius on an isolated body. If you have ever worked with an imported geometry of a tube, you probably know, that CATIA doesn't provide any tool to measure a bend radius. If there is no centerline, measuring a bend radius or centerline radius requires an extra geometry. Our target is to create an UDF, which can be used for measuring bend radius, center line radius and bend angle of a dead geometry of a tube. 1. Let's start with a new CATPart, name it Bend Radius. Create 3 GeoSets, name them Tube, Inputs and Bend Radius Measuring. 2. In the Tube GeoSet define 3 points at any reasonable coordinates. Using these three points create a polyline and assign a radius - Fig.1. Fig.1 3. Now create a swept surface along the polyline. Profile type - Circle, Subtype - Center and radius, assign any reasonable radius, in my case it is 10mm. 4. Go to Input GeoSet and Extract two edges of the surface like it is shown on the Fig.2. Fig.2 5. Isolate the two extracted edges. Now the Inputs GeoSet should contain two isolated circles. The problem is that sometimes CATIA may not recognize these edges as circles on imported geometry, therefore it is better to use any curve as an input. To turn the two circle into curves use Curve Smooth command (Insert>Operations) - Fig.3. Isolate both curves and delete the circles, now the Inputs should contain two isolated curves. Fig.3 6. Now go to the Bend Radius Measuring and create a new GeoSet in it, name it Curve 1. Using one of the input, create a point on curve, see more details on Fig.4. Fig.4 7. Create two more points on the same curve. As a reference, create another point, type Circle / Sphere / Elypse center, to get the center point of the circle. 8. Create a plane using three equispaced points - Fig.5. Fig.5 9. Use this plane and the center point of the circle to create a new line - Fig.6. Fig.6 10. Curve 1 GeoSet is ready to duplicate it (Fig.7). To duplicate a GeoSet select it [1], use command Duplicate Geometrical Set [2], in the new window choose the destination [3], then select the second curve from the Inputs GeoSet, click OK. Name the new GeoSet - Curve 2. Fig.7 11. Using the two lines create a plane - Fig.8. Fig.8 12. Use this plane, center point and line from Curve 1 GeoSet to create another line - Fig.9. It has to be normal to the line from Curve 1 GeoSet. Create a line in the same way for the second curve. Fig.9 13. Intersection of these two lines is the center point of the bend radius (yellow). Name this point - Center Point - Fig.10. Fig.10 14. Create another intersection, like it is shown on Fig.11. Result of this intersection is a multiple result, it means it has more than one solution. In Multi-Result Management window select option - keep only one sub-element using a Near. As a reference element use the Center Point. Fig.11 15. It was the last geometry we need. Now create three parameters - two length type and one angle. Drag&drop them to the Bend Radius Measuring Geo Set - Fig.12. Fig.12 16. Edit their properties and change their local names - Fig.13. Fig.13 17. Next step is to create formulas for these parameters. Add a formula to the Bend Angle parameter (Fig.14). The formula measures the angle between two lines. Line.3 and Line.4 are the yellow lines created in step number 12. Fig.14 18. Formula of the Bend Radius parameter is shown on Fig.15. Fig.15 19. Similar formula has to be created for the last parameter, use the Center Point and one of the center points of the input curves - Fig.16. Fig.16 20. New formulas have been created, drag&drop them to the Bend Radius Measuring Geo Set - Fig.17. Fig.17 21. Hide all the geometry and show only Center Point, Inputs and surface of the tube - Fig.18. Fig.18 22. In this step an UDF will be created, go to Insert > Knowledge Templates > User Feature. Select the Bend Radius Measuring Geo Set, curve 1 and curve 2 should appear in Inputs of components - Fig.19. Fig.19 23. Go to the Parameters Tab and the three parameters created in step number 15. Publish these parameters and modify their names - Fig.20. Fig.20 24.Go to the tab Outputs and make sure that the main result is the Center Point, if it is not, replace the main result with it - Fig.21. You can also go to the Properties tab to change the icon of this UDF. Click OK. Fig.21 25. Save the file and close it. 26. To use the UDF create a new file and another tube in it. Go to Insert@Instantiate From Document and select the CATPart from previous point. In Insert Object window choose a destination and as an input select two curves - Fig.22. Fig.22 27. The bend has been measured and the parameters have been created - Fig.23. Fig.23 Click Measure Between . In DMU, you can also select Analyze > Measure Between from the menu bar. The Measure Between dialog box appears: By default: For reasons of legibility, angles between lines and/or curves equal to 0 degrees are not displayed in the geometry area Minimum distances and if applicable, angles are measured. Measures made on active products are done with respect to the product axis system. Measures made on active parts are done with respect to the part axis system. Note: This last distinction is not valid for measures made before Version 5 Release 8 Service Pack 1 where all measures are made with respect to the absolute axis system. Select the desired measure type. Notice that the image in the dialog box changes depending on the measure type selected. Set the desired calculation mode in the Calculation mode drop-down list box. Click to select a surface, edge or vertex, or an entire product (selection 1). Notes: The appearance of the cursor has changed to assist you. Dynamic highlighting of geometrical entities helps you locate items to click on. Click to select another surface, edge or vertex, or an entire product (selection 2). A line representing the minimum distance vector is drawn between the selected items in the geometry area. Appropriate distance values are displayed in the dialog box. Notes: By default, the overall minimum distance and angle, if any, between the selected items are given in the Measure Between dialog box. For more detailed information about the dialog box options, refer to More About the Measure Between Dialog Box Measure Item is accessible from the Measure Between dialog box. In DMU, the Measure Thickness command is also accessible from the Measure Between dialog box. For more information, see the DMU Space Analysis User's Guide. Select another selection and, if desired, selection mode. Set the Measure type to Fan to fix the first selection so that you can always measure from this item. Select the second item Select another item. Click Ok when done. The default measurement unit that comes with Catia v5 is in millimeters (mm) and the spacing for the grids are 1 mm for major grid lines (or also known as Primary Spacing) and 0.1 mm for minor grid lines (or Graduation). However, as you may have noticed, the Catia v5 Tutorials that is available here uses inches (in) as their measurement unit. In order to adjust the measurement unit to the one that you desire, all you have to do is to access the Options window located under theTools menu. In the Options window, select Parameters and Measure which is located on theOptions tree. In there, select the Units tab and you will be presented with all the parameters available in Catia v5. Select the parameter that you want to adjust and for this tutorial purpose, we will be changing the units for Length. Once you have selected a parameter, you can change its units by using the drop down menu. In the Sketcher menu, under the Mechanical Design tree, you can adjust the Primary Spacing and the Graduation values to suit your preference. As mentioned earlier, Primary Spacing is basically the length of each major grid lines whileGraduation is the spacing between minor grid lines. As you can see below, I have set my Primary Spacing to 1-inch and the Graduation to 10. This basically means that the length of my major grid lines will be 1-inch while the minor grid lines are 0.1-inch. In this article I will show you how to create bounding box in CATIA V5. This feature is very useful especially in the case if it is a part that has a complex shape. A bounding box is the physical shape (that you run into or that other physical shapes encounter) for a given object. It defines the physical shape of an object, though it may be much different in size and shape than the objects visual appearance. Open the given object. Select the part body, or whole part. Then click on the Measure Inertia button as it's shown on the picture below. Measure inertia window will open. You can see several values here. However, bounding box values are not shown. Therefore click on the button customize and select Principal axes. Close windows with the OK buttons. Then go to the part (or product) tree and expand item Measure and Inertia Volume on the bottom. There you can find three values: BBLX, BBLY and BBLZ. These are the dimensions of the desired bounding box. As you might have noticed, measure inertia window offer option to export data to .txt file. It's button beside to the customize button. Finally values included in .txt output: Anyway, it's up to you, which type of output you will select. In conclusion watch the video for better understanding: Insert the following sample model files: ATOMIZER.model, BODY1.model, BODY2.model, LOCK.model, NOZZLE1.model, NOZZLE2.model, REGULATION_COMMAND.model, REGULATOR.model, TRIGGER.model and VALVE.model. They are to be found in the online documentation file tree in the common functionalities samples folder cysm/samples. Learn from thousands of free Tutorials Join the Community Surya Mohan Tyagi There is a toolbar called "measure" which can measure all the types of dimensions, you only click on the toolbar and then select the surface to be measured. first of all click on the toolbar"measure between". then select the geometry which is to be measured. for radius select. measure item Community Library Challenges Groups Questions Tutorials Engineers Workbench Overview Features Compare Shop Overview Features Compare Pricing Resources Blog Resource Center Help Center © 2021 GrabCAD, a STRATASYS solution The Computer-Aided Design ("CAD") files and all associated content posted to this website are created, uploaded, managed and owned by third party users. Each CAD and any associated text, image or data is in no way sponsored by or affiliated with any company, organization or real-world item, product, or good it may purport to portray. who8myrice123 (Computer) (OP) 7 Apr 15 15:41 Hey guys, new here in the forums. Though I'm new to CATIA, I do have experience in NX9. So my question is: how would I measure just the vertical distance between two features/points? Whenever I try to take a measurement, it gives me the diagonal distance. I know in NX9, there is a tool called "Projected Distance" which allows you measure a fixed distance depending on the reference vector. Is there something similar here in CATIA? Thanks in advance! Thank you for helping keep Eng-Tips Forums free from inappropriate posts. The Eng-Tips staff will check this out and take appropriate action. Page 2 Low-Volume Rapid Injection Molding With 3D Printed Molds Learn methods and guidelines for using stereolithography (SLA) 3D printed molds in the injection molding process to lower costs and lead time. Discover how this hybrid manufacturing process enables on-demand mold fabrication to quickly produce small batches of thermoplastic parts. Download Now Design for Additive Manufacturing (DFAM) Examine how the principles of DFAM upend many of the long-standing rules around manufacturability - allowing engineers and designers to place a partâ€™s function at the center of their design considerations. Download Now Industry Perspective: Education and Metal 3D Printing Metal 3D printing has rapidly emerged as a key technology in modern design and manufacturing, so itâ€™s critical educational institutions include it in their curricula to avoid leaving students at a disadvantage as they enter the workforce. Download Now Taking Control of Engineering Documents This ebook covers tips for creating and managing workflows, security best practices and protection of intellectual property, Cloud vs. on-premise software solutions, CAD file management, compliance, and more. Download Now Something went wrong. Wait a moment and try again. When measuring a part using the "Measure Between" or "Measure Tool" to measure a part, is there a way to sum all of the measurements taken? I am trying to measure an assembly with complex curvature along it's edges but I am faced with adding all the edge measurements together in excel to get it's length. Is there way around this? Is there a Macro for this?Page 2Posted by4 years ago 6 comments WARNING: This will only work if any other screen sizing variable has been disabled! E.g. CATForceVideoSize In order to calibrate our display, you will need to set two windows environment variables; CATWidthMMOIScreen and CATHeightMMOIScreen. The variables that you will need to measure are the screen width in millimetres and the screen height in millimetres. e.g. My monitor screen has a width of 595mm and a height of 345mm. 1) Set the ENVIRONMENT VARIABLES in window's: a) Select the START button on the taskbar and right click on COMPUTER and select PROPERTIES: b) Select ADVANCED SYSTEM SETTINGS c) In the SYSTEM PROPERTIES dialog box, ADVANCED tab, select ENVIRONMENT VARIABLES: d) Select NEW... in the ENVIRONMENT VARIABLES dialog box: e) Enter two new variables; CATWidthMMOIScreen and CATHeightMMOIScreen with values corresponding to your monitor screen width and height: f) Press OK and close all dialog boxes. 2) Launch CATIA V5 a) Either open an existing component (Part/Product) or create a new component: b) Run command "C: SCALE PLANES": c) The SCALE PLANES PROPERTIES dialog box will inherit the WIDTH and HEIGHT from the environment variable set in 1). These can be left as default then press OK. d) Set a standard view e.g. Front/Top/Etc and using a ruler directly measure from your monitor screen for exact measurements. e) To turn off the screen scaling, simply run the same command from step 2b) again.

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