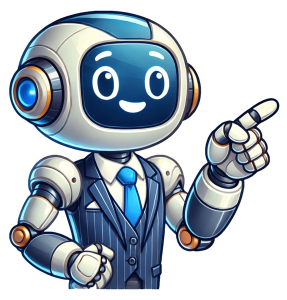

To Control the Display of Dimensions Perform one of the following operations: Set the sketcher_disp_dimensions and the sketcher_disp_weak_dimensions configuration options. Click or clear the Show dimensions and Show weak dimensions check boxes in the Sketcher area of the Creo Parametric Options dialog box. Toggle the Dimensions Display button in the Sketcher Display Filters on the Graphics toolbar.Setting the Color of DimensionsTo change the default colors for strong dimensions, weak dimensions, and locked dimensions, follow the steps below:1. Click > . The Creo Parametric Options dialog box opens.2. Click System Appearance.3. Click the arrow next to Sketcher.4. To redefine the color, click the color button to the left of a conductor type. A color palette appears.5. Select a color from Theme Colors. The color button shows the selected color.6. To revert to the default color, click Use Default .7. Click OK. Hello. We have met a issue more times, where Show Model Annotations in drawing doesnt work properly. The first picture shows dimensions of hole in Revolve feature in the model.The hole is fully constraint and all dimensions are strong.(This particullar hole is in pattern of 3 holes, but we have met this issue in many cases with another single features and models.) The First picture - a hole in the feature In Second picture you see dimensions which are shown by Show Model Annotations. As you can see, there are two of dimensios which are missing (d3.7 and d5.3). The Second picture - a hole in the drawing Have you ever met this issue?, possibly, what is the right procedure of showing model dimensions in drawing?Thanks for aswer. Hi Huntress, Did you check anything else out that may have over rid you drawing? Can you check the different version of frames to see if they modified version is still there?Tofflemire Tofflemire, No, I was not working on anything else at the time. I did try a complete shut down, reboot and then went to IntraLINK to see when things were modified. When I undid some of the frame, my assembly/part were more up to date than the prints. I opened the assembly first, the part then the drawing last. Much to my amazement, all the dimensions and geometric tolerances came back. WHY? Is this a glitch in Pro? I would ask here but I don't fell the heads in charge are much more help. I just want to know what I could have done to prevent this since it has made me lose time trying to get back what was lost. At least I will not have to re-do everything I checked everything into IntraLINK. I went back one frame, opened up the assembly then the part. I then checked the layers in the part since that is where all the dimensions are created; all the layers were turned off. That is why none of my dimensions were showing up. I will modify my config file to have all the layers on when opening up a drawing/part/assembly. I don't have this problem with other prints having their layers turn off when regenerating or opening the files up. Very Strange. Hi Huntress I was working on Wildfire 2 with a bespoke database in Germany, and had a similar problem with my geo tols disappearing - however, the dimensions were okay. I had never seen this type of issue in 2001, and in a way I'm glad that somebody else is having the same problems. I tried every trick I knew with layers, and drawing dispaly, but nothing improved the situations - GRRRRRRRRRRR! Have you contacted PTC or are they in the process of investigation? Best regardsDave Hi All, The drawing my of had the display status saved and set not to show the layers in question. They new dimensions would show up on new creations but would hide if the drawing was closed and reopened again. I would set the layer display status and save to show them. Hope that works.Tofflemire I opened my drawing up just fine this morning. I usually keep my part/assembly/drawing all up and switch back and forth between them to work on things. I wonder if doing so too many times created a glitch in my system and that made the layers hide. I have never told it to hide the layer with dimensions on it. Thanks for the advice. Now I know to check the layers first when my days work decides to hide from me. Enjoy! mapkey s1 @Mapkey_NAME Saves File Deletes Old .## Versions;\@MAPKEY_LABEL Save Single File on Disc;\~ Activate `main_dlg_cur` `File.psh_save`;\mapkey(cont) ~ Activate `file_saveas` `OK`;\mapkey(cont) ~ Activate `main_dlg_cur` `psh_purge`;; \ equals mirror of "/" slash ;; ends mapkey ;\ indicates continued mapkey on next line@MAPKEY NAME is actually the Description field@Mapkey Label is what shows in Mapkey listMichael Hi Michael, thank you for your suggestion. But I am afraid it's not the reason for my problem. Dimensions should be in colours as you can see on video.I made some modifications to a part and opened drawing to save changes. But dimensions wouldn.t come up. It was yesterday. Today after I restarted pro-e it's w/o problem..I can see them again now.. Really don't understand. Thank you anyways. When it happens again I surely will try your suggestion.Thank you for the mapkey. now I have to find out how to use it RegardsRadek. I found out how to use the mapkey. Thanks for that. Is there any way how to delete old file versions of all files in folder? What I want to do is be able to clean folder before backing up. Is it possible to backup all drawings along with assembly? When you backup assembly it automaticly saves all files. Can you set proe to automaticly save all drawings as well?Thanks Radek. Backup the drawing & it will also save all models & drawing formats.Open a system window to the working directory and type purge. It will run a batch file that will delete old versions of every Pro/E object in the directory. Hi dgallup. Thanks for the purge thing. It works well.Of course I know about backuip up drawings. What I ment was if I have opened main assembly and hit some buttom or whatever so it will backup parts AND drawing.. We had a big projects in last year and I'd have created more then 100 drawings for each.. You understand that to do backup for each drawing is not very convenient... But thank you anyway. Pro/E knows what dependent objects need to be stored with the active object but it has no way of knowing where else the object being stored is used. For that, you need a PDM system. A useful tool for those of us without PDM systems is a little utility called FindMyMother.exe. It will let you search for assemblies & drawings that use a model. I mainly use this before renaming a model so I don't break anything. So is it possible to backup part drawings along with main assembly? I store project files on pc. Time to time I need to load some projects onto a usb stick including drawings. Is there easy way to do it? Has anyone found a solution to this problem where the dimensions do not appear. I am having exactly the same problem. I guess I can wait till tommorrow but I do so have doubts the problem will go away by itself. ThanksMatthe To Set Decimal Places of Dimensions For Dimensions with Rounded Value Set the value of the default_dec_places configuration option for linear dimension and default_ang_dec_places for angular dimensions to required number of decimal places. Any dimension that you create will appear with the defined decimal places. For existing dimensions, select the dimension or select multiple dimensions using CTRL key. The Dimension ribbon tab opens. In the Precision group, select the Round Dimension option and specify number of decimal places.Alternatively, for a Drawing dimension, select the dimension, and on the Annotate tab, in the Format group, click Decimal Places. You are prompted to select the number of decimal places. Any dimensions selected or created after this are displayed with the designated number of decimal places.For Dimensions with Non-rounded ValueThe initial decimal places value of the dimension is the number equal to the number of significant digits after decimal point of actual dimension value.Select the dimension or multiple dimensions. The Dimension ribbon tab opens. In the Precision group, de-select the Round Dimension option and specify number of decimal places.If you select a driving dimension and modify the decimal places value, the actual dimension value is modified in case the defined number of decimal places is less than the number of digits after the decimal point of the current actual dimension value.However, if you select a driven or programmatically driven dimension, you cannot specify decimal places value that is less than the number of significant digits after decimal point of the actual dimension value. Showing Dimensions To display all dimensions on a drawing: 1. Click > > . Alternatively, in the Utilities group, click Show. The Show dialog box opens. 2. On the General tab, click the Dimension check box to display dimensions as a group. Any dimensions created after you clear the Dimension check box are not displayed on the screen till the Dimension check box is clicked. Create a dimension Dimensioning gives you the means to describe elements of your drawings in detail, providing valuable additional information about your model. To create a dimension,1. Click Annotation and then, in the Annotate group, click the arrow next to Linear or Circular. You can also click Angular in the Annotate group.2. Click the appropriate linear or circular dimension Dimension types under Linear or Circular respectively. The respective dialog box for the linear, circular, or angular dimension opens.3. Enter the following as necessary: For diameterTo dimension circular geometry where a tangential span exists and has unique center point, click Tangential Mode on the Diameter Dim dialog box. For radius and diameterTo pass the radius line through the element's center, click Centerline on the Radius Dimension or Diameter Dim dialog box. For tangentTo dimension a circle, arc, fillet, ellipse, or a B-spline (closed, both interpolation and control), click Tangential, right-click the viewport and select the required orientation for the dimension from the context menu.ExtremumThis option dimensions between the extremum points of the circular elements; it measures the longest or shortest possible distance, whichever is applicable. (This is the default option.)InclinedThis option allows you to orient the dimension at an angle to its extremum position. Creo Elements/Direct Annotation converts any value of the specified angle between 360 and -360 degrees.HorizontalThis option allows you to orient the dimension in horizontal plane.VerticalThis option allows you to orient the dimension in vertical plane.Parallel toThis option allows you to orient the dimension parallel to a reference line on your drawing.Perpendicular toThis option allows you to orient the dimension perpendicular to a reference line on your drawing. Extremum and Inclined are not available in Tangential Mode under Diameter. Tangential dimensions are not possible for open B-splines.Creo Elements/Direct Annotation always chooses the extreme tangent points of a closed B-spline which has more than two tangent points at the specified angle. If you select the points on a closed B-spline and a circle, arc, line, fillet or an ellipse, Creo Elements/Direct Annotation automatically chooses the tangent point which is closest to the selected point on that B-spline.4. If required, type a Prefix, Basic prefix, Postfix, Basic postfix, Subfix, or Superfix. Thread dimension properties are shown as keywords in the Prefix and Postfix fields.Basic prefix and Basic postfix are available only if you select a basic tolerance.5. If required, select a type of tolerance from the Tol Type box and type tolerance values.6. Click + or - under Decimal Places to increase or decrease the number of decimal places. Click the middle button to return to the default number of decimal places.7. Click the begin point for the dimensioning (or use box selection). For all types of dimensioning, you can specify a number of dimension points at the same time by drawing a selection box around them. Creo Elements/Direct Annotation determines the possible dimension points and creates the dimensioning automatically. Unless automatic placement mode is enabled, you then click the position for all dimension texts en masse. It is likely that some of the dimensions will need to be adjusted.For some dimensioning (such as single dimensions) you can specify a selection box from the start; but for others (such as datum dimensions) you must click the first point before drawing a box.When creating dimensions by box selection, the dimension reference elements cannot be re-selected during dimension creation. for angle dimensions, click one side of the angle. for tangential dimensions, click geometry where a tangential span exists. for a chamfer, click a chamfered edge.For single and angular dimensions, you can select a reference line as the begin point. For angular dimensions, the reference line must not have multiple segments.8. Enter the following as necessary: For a single dimensionClick the end point.You can also select a reference line as the end point. For a chained dimensionClick the first chain dimension point. Continue clicking chain dimension points and clicking the text positions as required. At any time while creating a chain dimension, you can undo the previously specified dimension point. Right-click and click Back to remove the last point, and then continue as normal. For a symmetry dimensiona. If you clicked Sym Single, click the end point. The dimension reference elements stay highlighted until you create the next dimension.b. If you clicked Sym Long, click the first symmetric datum point. The dimension reference elements stay highlighted until you create the next dimension. At any time while creating symmetry long dimensions, you can undo the previously specified dimension point. Right-click and click Back to remove the last point, and then continue as normal. For a datum dimensionContinue clicking datum dimension points and clicking the text positions as required. At any time while creating datum dimensioning, you can undo the previously specified dimension point. Right-click and click Back to remove the last point, and then continue as normal. For an angle dimensionClick the other side of the angle. If you have clicked the angle references in the wrong order, click Swap to take the angle that would arise if the two sides had been clicked in the opposite order.You can also select a reference line as the end point. The reference line must not have multiple segments. For a circular dimensiona. If you clicked Radius or Diameter, click a circle, arc, or fillet.b. If you clicked Arc, click the Length or the Angle check box to dimension the length or the angle of the arc. Click the two end points on an arc.Right-click the viewport, select Length or Angle dimension from the context menu.The order in which you select the endpoints determines whether the major or minor arc is dimensioned:Clicking clockwise along the arc measures the major arc.Clicking counterclockwise along the arc measures the minor arc.9. Under Appearance, at any time during creation of the dimension, you may specify various attributes. Most of these settings are intuitive, but it may be helpful to know:Style:Select a style to apply to the dimension.Color:Specify the color for the dimension.Font:Select the font for the dimension.Fill:Specify whether block fonts or arrows should be filled.Frame:Select the dimension frame.Arrow AppearanceFor dimensions that have two lines, you can specify the 1st arrow type for one end of the line that connects them and a 2nd arrow type for the other end.1st Type:Radius and Chamfer dimensions have only the 1st Type box.2nd Type:Sym Single and Coordinate dimensions have only the 2nd Type box.10. Click the position for the dimension text. A box representing the text follows the cursor until you specify the position.11. Continue adding dimensioning to the drawing, or click to complete the operation.When you create a new thread dimension, change the pitch in Creo Elements/Direct Modeling, and update the view in Creo Elements/Direct Annotation, the dimension also updates accordingly.If you have an old thread dimension, this update will not occur. After an update all user data in old thread dimensions are lost. You must use the Convert thread dimensions command to change the thread dimension. About Showing Model Annotations When you import a 3D model into a 2D drawing, the 3D dimensions and stored model information maintain parametric associativity with the 3D model. By default, they are invisible. You can then selectively choose 3D model information to show on a particular view, which is the concept of showing. Items that you make visible are referred to as shown. These shown dimensions are associative to the 3D model in both directions. That is, you can use these dimensions from both the drawing and the model environment to drive the model. After you place the model dimensions and detailing on the drawing, you can adjust their positions on the sheet and customize the format. Consider the following when you show model dimensions and detailing in your Creo Parametric drawings: Only one driving dimension for each model dimension may exist in a drawing. A drawing may have several views of the same object, but only one driving dimension for each feature of the model may be shown. You can unintentionally edit the model. If a driving dimension is edited, it turns white to indicate a discrepancy between the drawing and the model. When you regenerate the model, the drawing uses the new dimension. Using configuration options, you can break the link between model and drawing. This is not the usual Creo Parametric usage. You can create a Draft View with related imported annotations. When importing DXF, DWG, IGES, or CED files, select the Import associative dimensions option. In the imported file, select all the annotations that need to be related to the draft view, along with the draft geometry chosen for the creation of the draft view. Ordinate dimensions are not imported as associative when importing DXF, DWG, or IGES files. Hence, they cannot be associated with a draft view during its creation, but they can be related to an existing draft view. If you try to show legacy annotations associated with a model using the Show Model Annotations dialog box, which in turn, requires you to update those legacy annotations, even if you cancel the show operation and no annotations are displayed. This causes changes to the model. For example, the legacy annotations could be, a GTOL that does not have a corresponding note, or, a surface finish without corresponding symbol. When you do it that way, it adds a feature relation (as opposed to a part relation or an assembly relation). In the relation dialog box, change the "Look In" option to Feature, and pick your datum plane, and your relation should pop up. If you want the relations to all show up in the part level, make the plane (using just a number), then edit its value and enter your parameter. This will add a part-level relation for you.Hope it works GensetGuy, Relations in Pro/E are broken up into different types of relations. 1. Assembly relations d204:2=d204:5 where the :# specifies a part id which is different for each part.2. Part relations d4=2*d2 relations between top level dims3. Feature relations sd5=sd4 in a section4. Pattern relations If you hit use modify and click a feature the relation will be a part relation. It sounds like you entered the relation within the feature and it will be accessible only from feature relations where you'd select relations and choose feature relations and pick the feature whose relations you want to modify. If you like controlling all your relations from a single dialog, id suggest using only part level relations. Although it takes a little more work to exit a sketch and enter relations as part relations they'll be easier to modify later on. Hope this helps! Michael Justkeepgiviner gets another star. Thanks for the help. Good thoughts mjcole. I'll probably have to start doing it at the part level so I can see them all at once. Makes modifications that much easier.Thanks. There are also the FAQ relating to relations (sorry, could not resist):Mathematical Operators used in Pro/E Relations faq554-970How to activate the new user interfaces? faq554-211Using relation editor backdoor to parameters faq554-1132 Best regards, Matthew Ian Loew Please see FAQ731-376 for tips on how to make the best use of Eng-Tips Fora. Not sure if this is the solution because you mention they're not erased, but it's worth a try.While in the Annotate tab, try going to the drawing tree and expanding the annotations node for the view in question. Pick through the dimensions you see there to check if the missing dims highlight. If so, you can right-click and pick "unerase".One of the stupidest changes PTC implemented, IMO. In Reply to Anthony Troupe:We migrated to Creo Elements/Pro 5.0 M140 on Monday and I've been experiencing an issue with dimensions not showing on drawings. I've tried every iteration of Show Model Annotations conceivable and they do not show up. They don't show up as Annotations under the view and aren't erased. I've had to resort to drawing created dimensions for them, see the attached jpg.Has anyone else encountered this issue? This has happened on multiple drawings of different parts and assemblies. The only thing I can think of is they may be weak dims or driven ones. Try also suppress and unsuppress or delete/undo and see if they dissapear on next info. The other thing to try is using the trl+f finder or equations which can let you search for the offending d# symbol or value."It's not the size of the Forum that matters, It's the Quality of the Posts" Michael ColeBoston, MACSWP, CSWI, CSWTSFollow me on !w#$%@ TrajPar - @ mcSldWrx2008= ProE = SolidWorks I would make a drawing with several views and try showing the offending dimensions. ---------------------------------------- The Help for this program was created in Windows Help format, which depends on a feature that isn't included in this version of Windows. Hm, good points. I'll give them both a try, and report back. Many thanks! No go on Supress/Resume or Delete/Undo. The other thing to try is using the trl+f finder or equations which can let you search for the offending d# symbol or value. I assume you mean working with trail files? I deleted them long ago for this part, unfortunately. I would make a drawing with several views and try showing the offending dimensions. No go there as well...They just don't show up when I view the annotations.This may be one of those rare weird issues that I'll never run into again. Cntrl Alt Delete ---------------------------------------- The Help for this program was created in Windows Help format, which depends on a feature that isn't included in this version of Windows. Dimensions can be put on layers and blanked. I recall considering this for my list of 101 ways users can mess with each other. I used it when working with VSA software to hide the dimensions that had been added to the VSA model. If they are not hidden, then try driving them using a part relation to see if anything is affected. It is possible that it isn't a Creo flaw, but a failure of some earlier version of the software. It's one thing to correctly add and remove dimensions based on current user activities, but if this was done incorrectly previously, it's asking a lot for the software to correct a flawed database. mjcole was suggesting using crtl-f to start the Find function, though trail files could be a source if you still had them. I tried that and my screen went blank. ...Joking of course. 3DDave > I'll try your idea of forcing a relation on the dimensions. Good point, to check if anything changes as a result. 3dDave you are correct as to the find tool (binoculars) you can search for dims by name d## or od# for ordinate dims. The relations or equation editor also has an option to show dimension value for a given dim name. Hopefully it's not a declare value from a layout. It may just be an automatic weak dim. I usually use a wd mapkey to toggle the sketch options for the Weak Dims. Toggling show weak dims on may help."It's not the size of the Forum that matters, It's the Quality of the Posts" Michael ColeBoston, MACSWP, CSWI, CSWTSFollow me on !w#$%@ TrajPar - @ mcSldWrx2008= ProE = SolidWorks Heheh. This is funny. I wrote a relation as suggested and gave the two dimensions whacked out values, and everything regened just fine, with no visible changes to anything. So I think these two bits of information are like junk DNA left over from some previous Wildfire session.

**Creo dimensions not showing in sketch. How to add dimensions in creo drawing. Creo ptc drawing. Creo parametric drawing dimensions. Creo drawing dimensions. Creo drawing thread.**