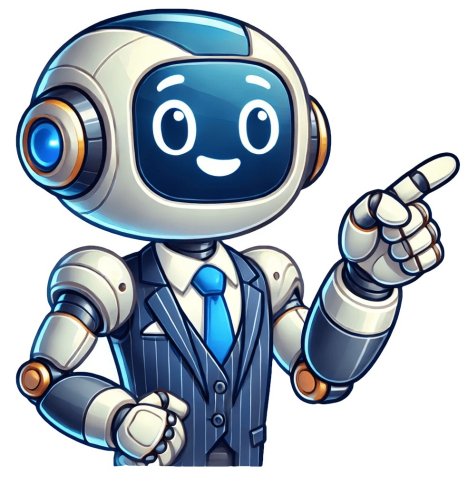Continue

You have to drill down into your drafting standard and change the XML file values. They are in the CATIA install location or possibly in other folders depending on your companies setup or use of ancillary automation tools. These are the files that populate your drafting standard under file >> Page Setup >> Standard. Hi! Thank your for your answer. What ha you used to edit the .XML file? You can use default editor available in Tools-Standards when running CATIA in admin mode (/admin switch). I have found it but I cannot edit it even in admin mode. 1. Create the following text, see Creating a Free Text 2. Select properties in the contextual menu (right-click). In font tab, select the bold italic style and in text tab increase the line spacing to 5 mm. Click ok. The text looks like this 3. Select the Set as default option from the contextual menu. The values you set are stored. Click in the drawing to finish the creation. The DS ISO 1 OpenType font is provided. This font is an OTF font that has been designed to display annotations with respect to the various ISO standards defining text representations. It is based on the ISO 3098 standards (ISO 3098-5:1997, ISO 3098-3:2000) and Unicode definition. DS ISO 1 is a variable-pitch font, designed as an OTF font with TrueType format outlines definition. It is named according to the following rules: Style Name Family Name Full Name Regular DS ISO 1 DS ISO 1 Italic DS ISO 1 DS ISO 1 Italic Bold DS ISO 1 DS ISO 1 Bold Bold Italic DS ISO 1 DS ISO 1 Bold Italic This font contains glyphs covering the following Unicode ranges: Range Unicode Defined Glyph in Range 0x0000 - 0x007F Controls and Basic Latin Present 0x0080 - 0x00FF Controls and Latin 1 Supplement 0x00A1 - 0x00FF 0x0100 - 0x017F Latin Extended A Present 0x0374 - 0x03F3 Greek 0x0388 - 0x3F9 0x0400 - 0x045F Cyrillic Present 0x2460 - 0x24EA Enclosed Alphanumeric Present There are also some extra Unicode ranges: Range (FontLab) Unicode Defined Glyph in Range 0x02B0 - 0x02FF Space Modifier Letters 0x02D9 (Dot above) 0x20A0 - 0x20CF Currency Symbols 0x20A7 and 0x20AC 0x2100 - 0x214F Letterlike Symbols 0x2104 and 0x2126 0x2190 - 0x21FF Arrows 0x2197, 0x21AC, 0x21A7, 0x21B7 0x2200 - 0x22FF Mathematical Operators 0x2205, 0x2211, 0x2220, 0x2222, 0x2225, 0x2229, 0x222B, 0x2259, 0x2260, 0x2261, 0x2264, 0x2265, 0x229A, 22A5, 22B2, 22B3 The bold style is not defined in the ISO 3098 standard. However, a thickness has been defined for the Bold and Bold Italic styles by adding 50% to the thickness of the regular style.Below is an example using DS ISO 1: Warning: DS ISO 1 font does not contain any Chinese, Japanese, or Korean character. DS ISO 1 is provided "as is". It is up to each user to decide whether this font fits requirements in terms of annotation representation and compliance to ISO standards (or any other standard). Four new Drafting standards are also provided. For more information, refer to the Version 5 Interactive Drafting User's Guide: Administration: Before You Begin. Select the dimension. Select Edit> Properties. You can also right-click the dimension and then select Properties from the displayed contextual menu. In the Properties dialog box that appears, click the Font tab. The associated dialog box is displayed. Hi ' Purpose: Macro will change all text greater then 2 into 10 in an active CATIA drawing ' Usage: 1 - A CATDrawing must be active with some dimensions on it ' 2 - Run macro ' Author: ferdo (Disclaimer: You use this code at your own risk) ' === Sub CATMain() Set drawingDocument1 = CATIA.ActiveDocument Set Sheets = drawingDocument1.Sheets Set activeSheet = Sheets.ActiveSheet Set views = activeSheet.Views For i=1 to views.Count if i 2 then Set view = views.Item(i) Set texts = view.Texts For j=1 to texts.Count Set text = texts.Item(j) text.SetFontSize 0,0,10.0 'change all text greater then 2 into 10 Next End if Next End Sub Regards Fernando //picasaweb.google.com/103462806772634246699/ Or even better ' Purpose: Macro will change text in an active CATIA drawing according to your inputs ' Usage: 1 - A CATDrawing must be active with some dimensions on it ' 2 - Run macro ' Author: ferdo (Disclaimer: You use this code at your own risk) ' === Sub CATMain() Set drawingDocument1 = CATIA.ActiveDocument Set Sheets = drawingDocument1.Sheets Set activeSheet = Sheets.ActiveSheet Set views = activeSheet.Views Dim myFontSize As Double myFontSize = "10" myFontSize = InputBox ("Please enter a font size.", "Enter Font Size", myFontSize) Dim myFontName As Double myFontName = "Courier" myFontName = InputBox ("Please enter a font name.", "Enter Font Name", myFontName) For i=1 to views.Count if i 2 then Set view = views.Item(i) Set texts = view.Texts For j=1 to texts.Count Set text = texts.Item(j) text.SetFontSize 0,0,myFontSize text.SetFontName 0,0, "myFontName" Next End if Next End Sub Regards Fernando //picasaweb.google.com/103462806772634246699/ Hi, I'll try to loop through as you said above. Still one doubt and thought to get the same clarified. Do the text in dimension should be considered as a text and accessed separately? I tried to do this but didn't succeeded. For dc = 1 To oDims.Count ' Dimension loop Set oMyDimension = oDims.Item(dc) Set oDimText = oMyDimension.GetValue oDimText.GetValue Value Next End Sub Regards Fernando "Courrier 10 BT" oDimText.SetFontSize 0, 0, 10 Next Regards, Maddy Tried to display all the text. But while doing so i'm not getting any dimension text displayed at all. You mean you want something like this? Sub CATMain() Set drawingDocument1 = CATIA.ActiveDocument Set Sheets = drawingDocument1.Sheets Set activeSheet = Sheets.ActiveSheet Set views = activeSheet.Views Dim selection1 As Selection Set selection1 = drawingDocument1.Selection selection1.Search "CATDrwSearch.DrwDimension,all" For i=1 to selection1.count Set Dimension = selection1.Item(i).Value MsgBox Dimension.GetValue.Value Next End Sub Regards Fernando //picasaweb.google.com/103462806772634246699/ Yes, Exactly, That's what i want to change the height and name. Hi Guys, Could you please help me on this. Regards, Maddy Double-click the text to switch it to edit mode. Select the whole text (you can also select only part of the text) and then select Edit> Properties. You can also right-click the selected text, then choose Properties. In the Properties dialog box that appears, select the Font tab. The associated dialog box is displayed. Font, Style, Size, Underline and Color: Choose the font, size, style and color of the text, and underline it. Attributes: Draw a line through (Strikethrough) or above (Overline) the selected text, and superscript or subscript it. You can either underline or overline a text, but you cannot do both. Character: Ratio: Modifies character width. Slant: Modifies character slant (for Italic text style, slant=15 deg). Spacing: Changes the spacing between characters. Pitch: Sets a fixed or a variable pitch. As an example, create the free text Tools and apply the font ROM1. Fixed Pitch Variable Pitch   The Slant and Pitch options are available only for stroke fonts. The pitch of some stroke fonts cannot be modified. In that case, the Pitch combo list is disabled. If you use characters in some fonts that have no or very little spacing (i.e. i or l), you should not set the spacing to 0 mm. In such case they would look as if they are superimposed and only one character would seem to be displayed in your annotation. Kerning: Reduce space between letters. This option is available with open type fonts, and for text only. OTF are customer's fonts, they are not provided with CATIA. without kerning with kerning More button: Display extra options, if any are available.   Structure of the standard is defined by the administrator. A standard file is structured as a tree, as it appears in the Standards Editor (available via Tools > Standards). It contains several main sections, each dealing with a specific aspect of the customization: e.g. for DXF: e.g. for Drafting: In this scenario, administrators will learn how to customize mapping parameters using an example. This scenario provides an example of Line Type mapping but the procedure is the same when customizing other parameters.The lists of attributes are not the same in V5 and AutoCAD. Mapping options are used to come as close as possible to the AutoCAD attributes, or to switch them to V5 attributes. A default mapping is available, but you can customize the mappings according to your needs, and create many new DXF mapping standard files. The V5 parameters proposed are those of the current Drafting standard file. Refer to Setting Standard Styles in the Interactive Drafting User's Guide for more details. You should be familiar with the DXF/DWG styles. Select AutoCAD colors by their number. Select AutoCAD line types by their name. Select V5 line types or thicknesses by their number. For text fonts you can: map any DXF font with a V5 font, associate a X scale factor to reframe a font with a different geometry (characters too wide or too narrow) to achieve the best possible alignment , define a default font to be used when there is no mapping for a DXF font, define a default KANJI font, other than SSS4, to be used when the BigFont DXF font is not mapped. You need to work in administrator mode. To do this, proceed as follows: Set up the CATReferenceSettingPath variable. Start a V5 session using the -admin option. For more information, refer to the Managing Environments chapter in the Infrastructure Installation Guide. Set the variables CATCollectionStandard and CATDefaultCollectionStandard. If none of the conditions are respected, a warning message will appear to let you know that you will neither be able to modify nor save the XML files. Select Tools > Standards to launch the standards editor. Choose the dxf category, and then open the DXF.xml file from the drop-down list. Expand the LineTypeMapping node in the editor. Select the line type 1 on the left. On the right, its DXF name and CATIA type number are displayed. Use the spinner to change it to another CATIA type number. Repeat this step with the other line types. Click OK to save your modifications in the current file and exit the standards editor or Create a new mapping standard file using the File - > Save as New command. Call it DXFDoc.xml in the directory. If you have created a new DXF mapping standard file, it is now available in the drop-down list in the DXF settings (available through Tools/Options/General/Compatibility) sp kumar Dimensions text sizes are not changing once i change the view scale.i want text sizes to be updated with reference to the view scale......how can i do this??? hello, is there anyway to change default catia drafting font? it comes always "SSS3". i want to set it to arial, verdana etc.. The System Admin has to customize the Drafting Standard file to setup all the defaults for drafting. Drafting (IntDraft) / Wrong representation of the Font size from Tools > Standards. . Scenario: 1. Open CATIA. 2. Go to Tools > Standards > Drafting > ISO > Styles > Default > Font & change these settings (Font size: 14, Bold: Yes, Underline: Yes 7 Colour: Green). 3. Go to part design. 4. Create a Pad. 5. Go to Drafting (with the changed standard). 6. Create Front view. 7. Create Dimension. 8. Go to Tools > Options > General > Parameters & Measures > Units & change the setting to Inch. 9. Go to Drafting (with the same changed standard). 10. Create front view & create dimension. . Problem: The font size is different. . Expected result: It should be as per the standard. . Drafting (IntDraft) / Wrong representation of the Font size from Tools > Standards. Drafting (IntDraft) / Wrong representation of the Font size from Tools > Standards. . Scenario: 1. Open CATIA. 2. Go to Tools > Standards > Drafting > ISO > Styles > Length / Distance Dimension > Default > Font > & change these settings (Font size: 14, Bold: Yes, Underline: Yes 7 Colour: Green). 3. Go to part design. 4. Create a Pad. 5. Go to Drafting (with the changed standard). 6. Create Front view. 7. Create Dimension. 8. Go to Tools > Options > General > Parameters & Measures > Units & change the setting to Inch. 9. Go to Drafting (with the same changed standard). 10. Create front view & create dimension. . Problem: The font size is different. . Expected result: It should be as per the standard. . . THIS PROBLEM WILL BE FIXED ON CATIA VERSION 5 RELEASE 19 GA LEVEL. NOTE THAT THIS PROBLEM WILL ALSO BE FIXED ON V5R18SP05 INCIDENT DIAGNOSIS . Drafting (IntDraft) / Wrong representation of the Font size from Tools > Standards. . CORRECTION EXPLANATION . The was a problem with the combo of the font size for the Test Properties, which has now been corrected. . HD72573 CATIA V5 NT>XP 569151000 518 CLOSED PER NoPE NoHIPER NoSpecatt 2008-03-19 2008-04-03 2008-06-02 [{"Business Unit":{"code":"BU053","label":"Cloud & Data Platform"},"Product":{"code":"SSVJ2K","label":"CATIA"},"Component":"","ARM Category":[],"Platform":[{"code":"PF025","label":"Platform Independent"}],"Version":"518","Edition":"","Line of Business":{"code":"LOB10","label":"Data and AI"}}] Select the dimension. Select Edit> Properties. You can also right-click the dimension and then select Properties from the displayed contextual menu. In the Properties dialog box that appears, click the Font tab. The associated dialog box is displayed.