

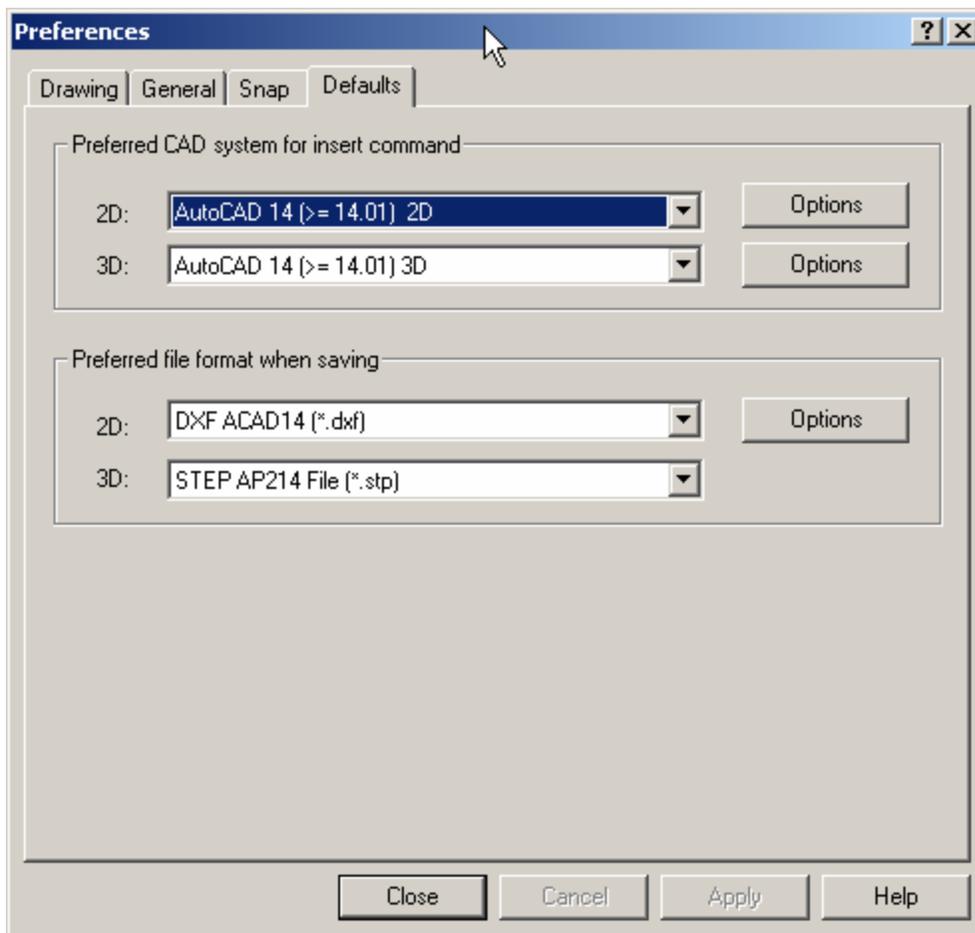
Using COMMON_PARTS Project and PartSpec with IDEAS

Tony Parker

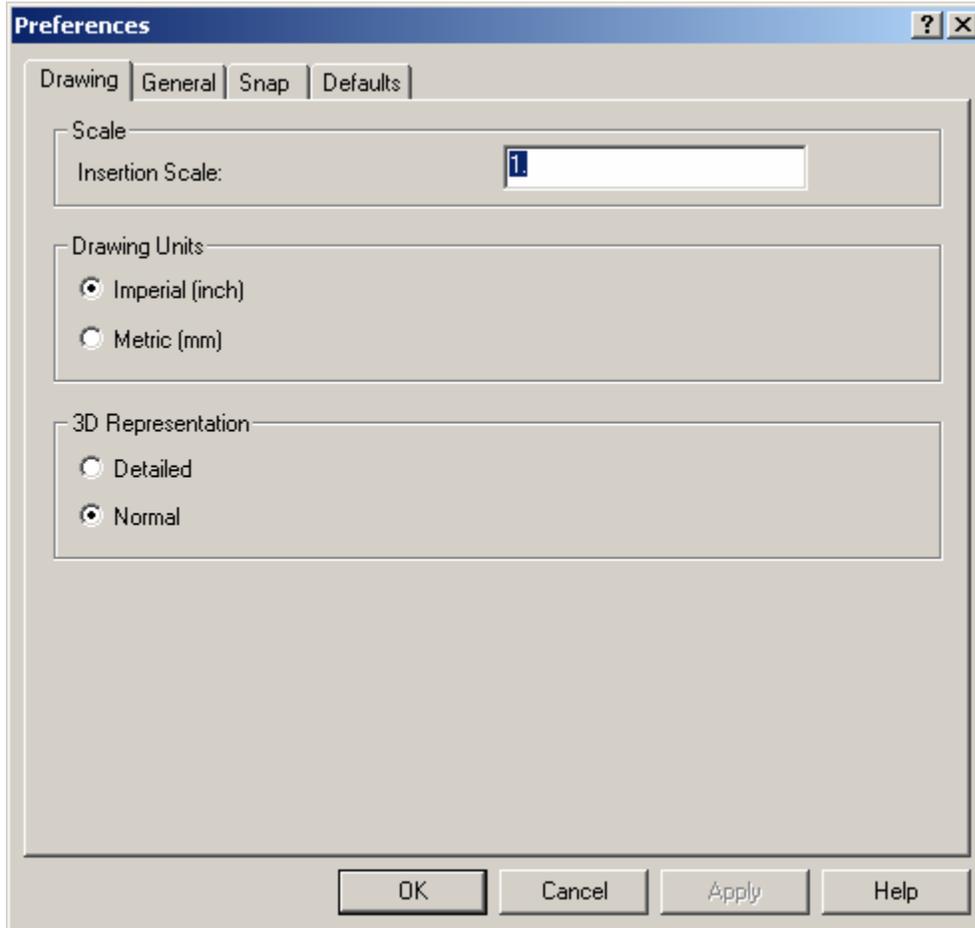
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While IDEAS is not one of the supported CAD systems that allow for direct insertion of symbols and 3D models with PartSpec, it is still a good resource for obtaining vendor parts and drawings for our use. This document explains how to setup and use PartSpec in ADMS with IDEAS. This is also the procedure you should use to contribute any standard parts that you create from scratch. While this procedure may appear convoluted it is the price we pay for multiple team areas, however the potential for time savings and increased efficiency far outweigh the problems.

- 1.) You will need Stewart or me to install the client portion on your local PC. All the data files have been loaded to T:\ PartSpec. You should also have a shortcut on your desktop to D:\Ideas\i9\bin\dxfreed.cmd so you can drop dxf files for translation to IDEAS asc drawing files. When the program is installed it should also be configured correctly for our use, if not you can fix Option settings to match the following two displays.
- 2.) Your Options -> Configuration -> Defaults should be:



Your Options -> Configuration -> Drawing should be:



There are more settings available under General and Snap, but they may not be of use with IDEAS. Feel free to explore them and let me know if you find anything we can utilize. **NOTE:** If you select Detailed under 3D Representation you will get actual thread form on the part, most often **NOT** what you want, and will just increase the model size.



3.) Using a 3D Catalog Part.

- a. Find the desired vendor part in the database. Open a text window in the editor of your choice and decide on an appropriate name for the item. Remember it has to be 32 characters or less, all upper case. See the online [naming rules](#) for guidance. Since these will be Vendor items they will also fall into the category of COMMON_PARTS so see the Common Parts [naming rules](#) some [sample names](#) are also available. You should also use the [Acronym list](#) to determine correct abbreviations. If in doubt ask.
- b. Double click the 3D item to activate it. Depending on the type of item, you may be prompted to select or supply size and material information. Enter this information in your text editor window so you can use it later to paste information about the item into IDEAS.
- c. When the item is generated and appears in the window, right mouse and choose Save As. Save the file to your SDRC_SCRATCH (D:\Ideasuser) directory, USE AN APPROPRIATE NAME.
- d. In Master Modeler select File -> Import -> STEP. Choose the folder icon and open the file you just created. Make sure Flavor is AP214. The translator should launch and the part should appear. Examine the part for dimensional accuracy and make sure it is a valid solid. Assign the correct material to the part. Orient the part with 0, 0, 0 at its most likely usable insertion point in an assembly. Set any attributes, i.e. color etc. as appropriate. You may also want to add reference points at key interface locations to help with assembly orientation, i.e. tube insertion depth for a fitting etc.
- e. Immediately name the part CORRECTLY. You can copy and paste from the editor window if you followed the above instructions. DO NOT PUT THE VENDOR NUMBER IN THE PART NUMBER FIELD. Leave the Part Number field blank for now. Enter the complete information for the part in the history. You can enter an 80 character description of the part in the description field. See 3D example at the end of this document.
- f. Once correctly named and identified, the part should be stored in the correct COMMON_PARTS library. The BTEV team has been designated as the Master Common Parts area so all items created must be transferred to this team with WRITE privileges. Once the part is stored in your local team COMMON_PARTS library you may use it in your assemblies. **ONCE THE ITEM IS STORED IN THE COMMON_PARTS LIBRARY IT IS NOT MODIFIABLE.** If the item is incorrect, a new item will be required; we do not want a simple item like an incorrect washer impacting multiple possibly large assemblies by marking them all out of date.
- g. If the item will have an associated Fermilab number then you will need to create a drawing for the item, generate or log the Fermilab number as the item Part Number, and then enter the drawing number assigned as the Part Number for the 3D item.
- h. You can now delete the STEP file and your text editor information. Notify your local support person to have the item(s) correctly exported to the

BTEV Master COMMON_PARTS team area and your local team copy set to reference only.

4.) Using a 2D Part View.

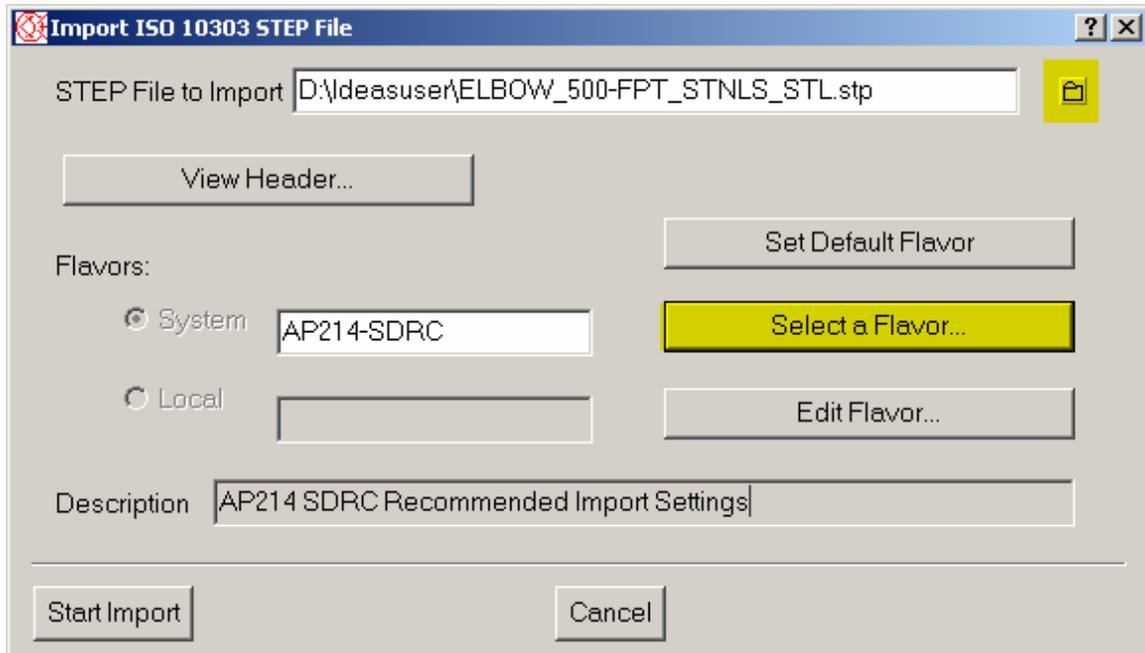
- a. Find the desired vendor part in the database. Open a text window in the editor of your choice and decide on an appropriate name for the item. Remember it has to be 32 characters or less, all upper case. See the online [naming rules](#) for guidance. Since these will be Vendor items they will also fall into the category of COMMON_PARTS so see the Common Parts [naming rules](#) some [sample names](#) are also available. You should also use the [Acronym list](#) to determine correct abbreviations. If in doubt ask.
- b. Double click the 2D item and view to activate it. Depending on the type of item you may be prompted to select or supply size and material information. Enter this information in your text editor window so you can use it later to paste information about the item into IDEAS.
- c. When the item is generated and appears in the window, right mouse and choose Save As. Save the file to your SDRC_SCRATCH (D:\Ideasuser) directory, USE AN APPROPRIATE NAME.
- d. Open a windows explorer window to your SDRC_SCRATCH (D:\Ideasuser) directory. Drop the dxf file you just created in step c onto your desktop dxftread shortcut. A cmd window will appear and the file will be translated to an IDEAS .asc drawing. You should see this file in the explorer window.
- e. In Master Drafting select File -> Import, set the filter to *.asc and the type to Drawing. Choose the asc file just created.
- f. The colors and line fonts may not be correct. Modify these as required. The entities will all be on layer 1 (AUTOCAD0) change this to an appropriate layer so the layer name will not be included with the symbol. If you have no preference use layer 200.
- g. Once the geometry is cleaned and dimensionally verified create a symbol using an APPROPRIATE name and export it. It's now ready to use in your drawings. If you contact me I'll include these symbols in the icon panels.
- h. Delete the open drawing, dxf, and asc files.

3D Part Example:

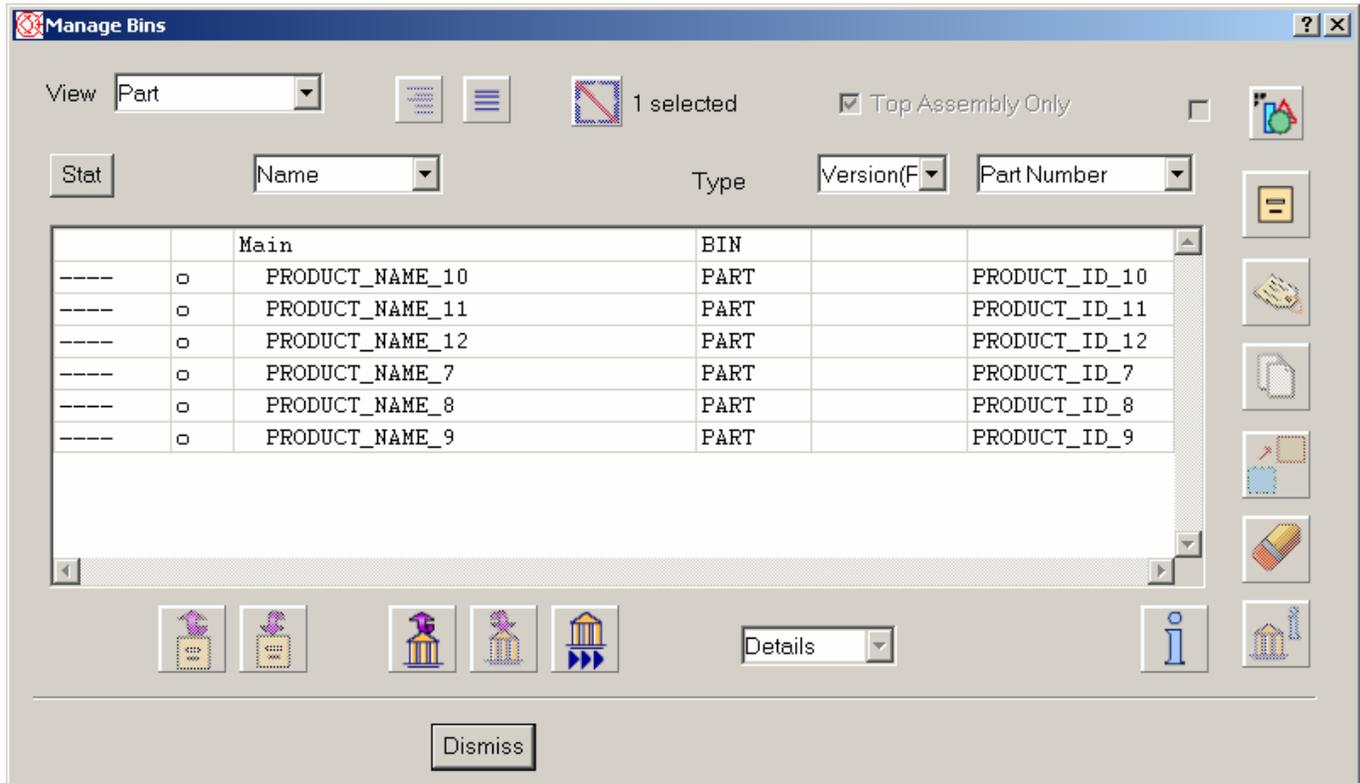
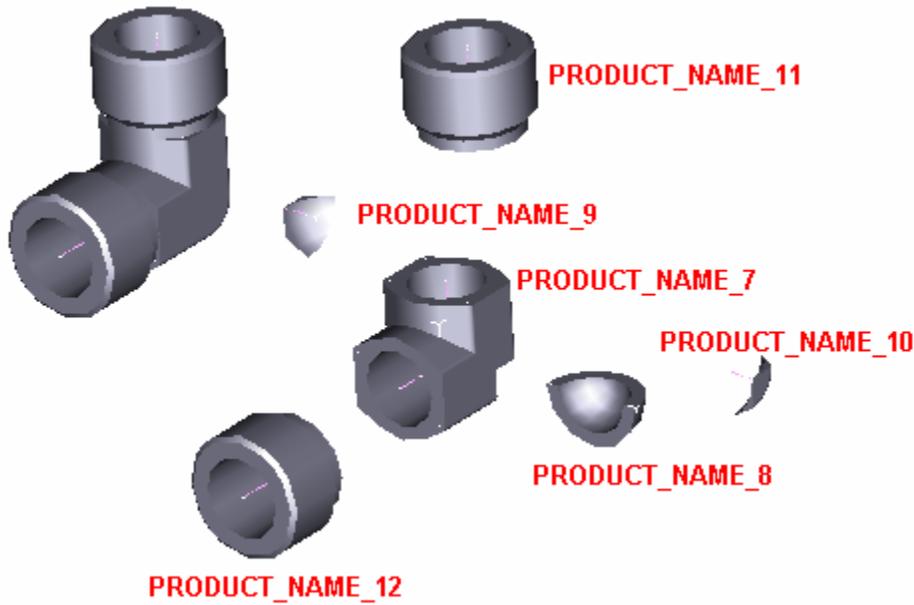
This example demonstrates importing and naming a 3D part for use in the IDEAS COMMON_PARTS libraries. It assumes that you have created a STEP file as described above. **I would recommend that you put away any parts on the workbench and save your model file before any import.**

In Master Modeler select File -> Import -> STEP

Choose the Folder icon to select the .stp file.

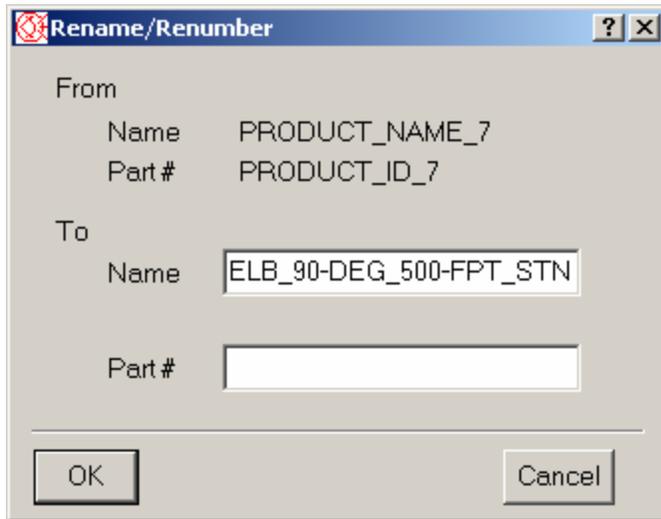


Make sure Flavor is set to AP214-SDRC, if not then choose Select a Flavor to set it. Choose Start Import when you're ready. The translator should connect and a progress window will appear. The part will appear. Some items will come in as multi part since they are parametrically generated and STEP is not flawless. For this example I imported a simple 1/2" female pipe thread 90 degree elbow which actually came in as six individual parts.



The items PRODUCT_NAME_8 and 9 are hard to see in the image but are actually the corners of the hex on the body. Determine the best order to join the parts and continue joining until you have a single part. In this case I started with the body (7), added (8), (9), (10), (11), and (12). At this point you should use Analyze Geometry -> Diagnose Part and examine at least Questionable Geometry, Free Edges, and Duplicate Trimmed Surfaces. Depending on the part you will need to use your judgment and modeling skills to determine if the part is acceptable. **Verify that the part is dimensionally correct.**

Use List Properties to assign the appropriate material to the part. Open Manage Bins and rename the item appropriately. Consult your local support person if necessary. Remove any existing part number entry.



The dialog box is titled "Rename/Renumber" and contains the following fields:

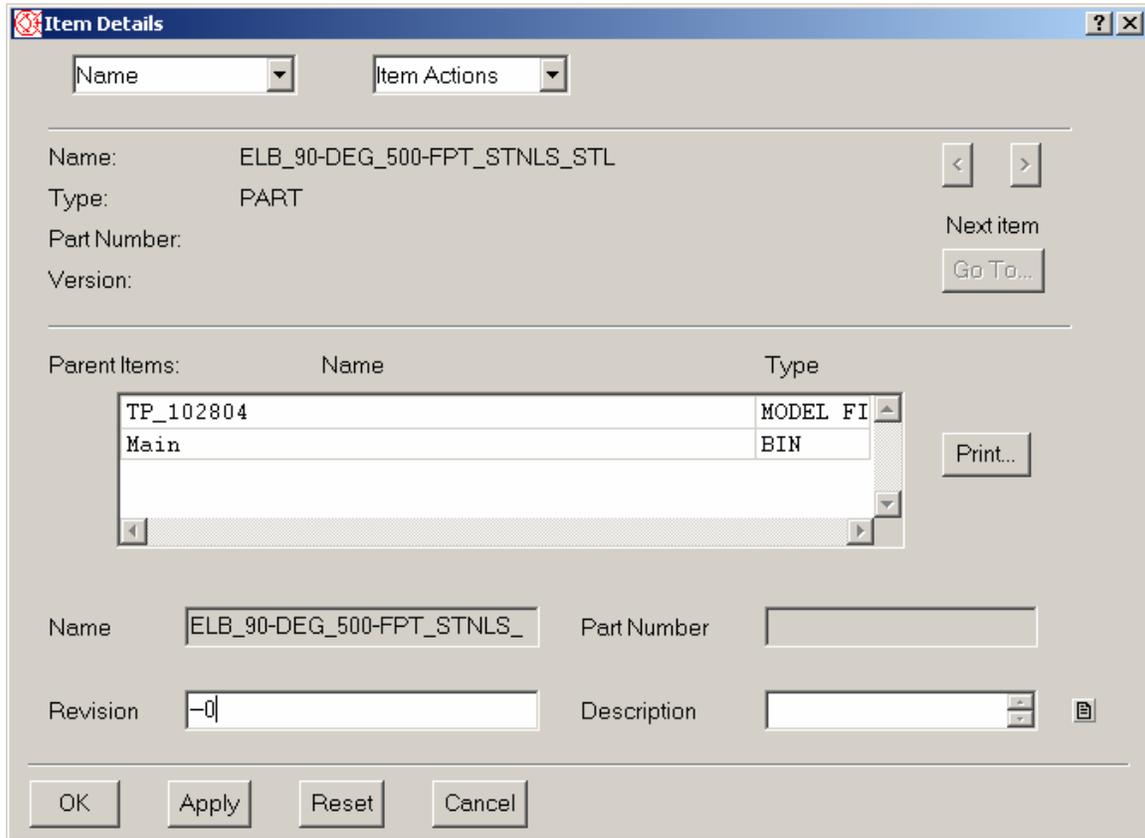
From	
Name	PRODUCT_NAME_7
Part #	PRODUCT_ID_7

To	
Name	ELB_90-DEG_500-FPT_STN
Part #	

Buttons: OK, Cancel

The full name below is **ELB_90-DEG_500-FPT_STNLS_STL** (28 characters) using abbreviations from the online acronym list.

While still in Manage Bins with the newly named part selected, choose Details -> Name and set the Revision to -0 (two dashes and a zero, not capital O).



The dialog box is titled "Item Details" and contains the following fields and controls:

Name: ELB_90-DEG_500-FPT_STNLS_STL
Type: PART
Part Number:
Version:

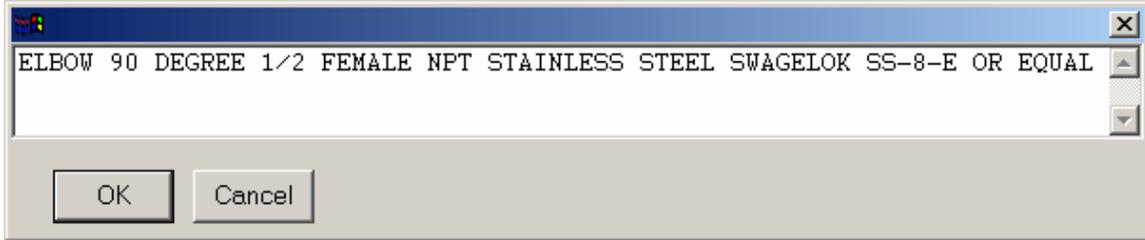
Parent Items:

Name	Type
TP_102804	MODEL FI
Main	BIN

Name: ELB_90-DEG_500-FPT_STNLS_
Part Number:
Revision: -0
Description:

Buttons: OK, Apply, Reset, Cancel

Select the Description editor icon and enter an 80 character maximum entry for the part, all Upper Case. Every effort to maintain this information through CAD and database migrations will be made. Users will be able to set the column display to Description during Get from Library to help identify parts.

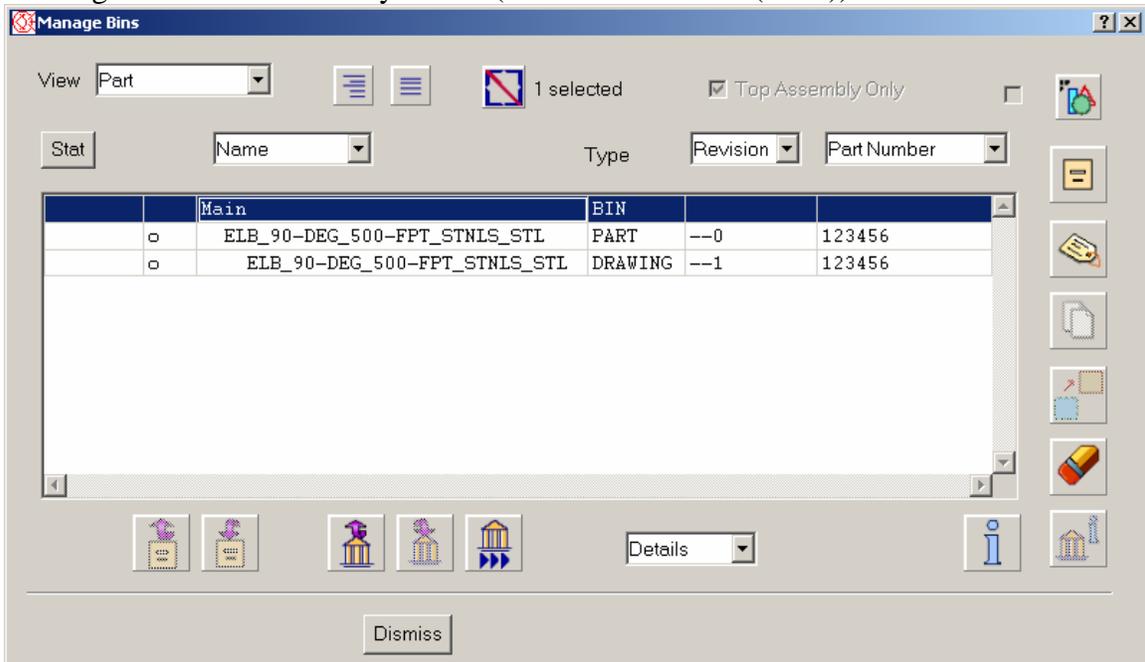


If you desire to add additional information to the part choose Details -> History and enter any information. Since this information is unlimited it may not be preserved during future migrations.

Now you're almost finished. Orient the part so that coordinate 0, 0, 0 is at a logical insertion point for instancing into an assembly. There will obviously be discrepancies about where the insertion point should be but it definitely should not be some arbitrary distance inside a volume of the part, and also not hundreds of inches or feet from the global origin. Since you are making the effort to contribute the part the ultimate decision and future complaints are yours.

Check the part into the appropriate COMMON_PARTS library, keep for reference only. You may now instance the item into your assemblies.

If the item will have an associated Fermilab number then you will need to create a drawing for the item, generate or log the Fermilab number as the item Part Number, and then enter the drawing number assigned as the Part Number for the 3D item. The revision for the 3D item should always be --0 (two dashes and a zero, not capital O) and the drawing revision should always be --1 (two dashes and a 1 (sheet)).



Contact your local administrator and make them aware of the addition to the library. It is their responsibility to package the part to the BTEV team (for check out) and make the other teams aware of the availability. NOTE: All COMMON_PARTS in the departmental team areas will be REFERENCE only, items exported from these teams to BTEV should be for check out. All items exported from BTEV to departmental teams should be REFERENCE only.