

# GUIDELINE

## V3.261 Library Management Manual

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#### Adding a new component in the library

In order to allow the users to define a clamping using custom components, Guideline gives the possibility to add them to

the standard library. To do so, just press



in the toolbar and select the type of component you want to add (Base Gauge in our example).

N.B: only for Base Gauge, the software will ask you to Import or Create a new component. All the other categories can be imported only.





#### Adding a new component in the library: import the component

If you choose Import, the system will ask for a STEP file to be added to the library.

N.B: only STEP files are accepted.

A confirmation message is then displayed.







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### Adding a new component in the library: import the component

If you choose Import, the system will ask for a STEP file to be added to the library.

N.B: only STEP files are accepted.

A confirmation message is then displayed.

To see it in the library, open a new file (or re-open the piece you were working on) and it will be displayed.

To remove it, just press in the top-right corner of the icon. Import procedure works the same for base gauges, pallets, cubes, shoulders and machines.







#### Adding a new component in the library: Siemens NX

If you are using Siemens NX, you are required to add the correspondent .prt file in the file system in order for it to use the new elements in library.

Library is contained in \FCS\GuideLine3\FCSSystem folder and subfolders.

The picture on the right shows the library's strcture:

- Clamping components are in the named folders (CUBES, PALLETS ect). In those folders, the .prt files must be pasted for NX to search while creating the clamping assemblies;
- Stacks' components are contained in folders named by size (M6, M8 ect.). Here you can modify or add the stacks' elements and subractions tools (see page XX);





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#### Adding a new component in the library: create the Base Gauge

If you choose Create, the system will show a window where you can specify the characteristics and the name of the plate you want to add.







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Dropdown menu options are:

- TF: the plate will have a placement hole in the center;
- TR: the plate will have the center massive, surronded by four placement holes;







#### Adding a new component in the library: create the Base Gauge

If you choose Create, the system will show a window where you can specify the characteristics and the name of the plate you want to add.

Dropdown menu options are:

- TF: the plate will have a placement hole in the center;
- TR: the plate will have the center massive, surronded by four placement holes;
- Circular: creates a circular plate.







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#### Creating a new subtraction tool

Some of the functionalities of Guideline, like interference check and CAD-CAM export, rely on the seats geometry included in its library. If, in workshop, you use different seats' geometry, it may be the case that you want to implement those geometries in Guideline you can do it following the procedure listed in the next pages.

IMPORTANT NOTE: in this release it is possible to modify tools' geometry, but not to implement alternative ones. In other word, if you use different dimensions alternatively (i.e. 2 different lenghts) for the same size, that cannot be managed in Guideline at the moment. You should choose the one better fitting your specific needs (the largest, the one you use more often etc.) and manage the other casesoff line (modifying the geometry in CAD, writing a note in the documentations). BCK Solutions is at your disposal to help you working out your case.





#### Create a new subtraction tool – File structure

Bodies' and tools' 3D models are placed in \FCS\GuideLine3\FCSSystem folder, in the subfolders, divided by size. In the picture M16, H80 body (FCS catalogue code: 0003\_02101) is displaied highlighted as example.

For most of the bodies, the there are 7 files:

- A .bly file containing the 3D of the body used in Guideline (M16\_BODY\_H80\_0003\_02101.bly);
- 2. A bitmap (M16\_BODY\_H80\_0003\_02101.bmp);
- A .prt for Siemens NX integration with the tool for finishing (M16\_BODY\_H80\_0003\_02101.prt);
- A step file used for translating the geometry for finishing (M16\_BODY\_H80\_0003\_02101.stp);
- A .prt for Siemens NX integration with the tool for roughing (M16\_BODY\_H80\_0003\_02101\_Rough.prt);
- A step file used for translating the geometry for roughing (M16\_BODY\_H80\_0003\_02101\_Rough.stp);
- 7. A .bly file containing the 3D of the body and the tool used in Guideline (M16\_BODY\_H80\_0003\_02101\_Tool.bly).

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Home Share View			
→ 👻 🚹 > This PC > OS (C:)	>	FCS > GuideLine3 > FCSSystem > M16	~
OS (C:)	^	Name	Date
\$AVG		M16_BODY_H40_0003_02100_Rough.stp	20/1
bck_tools		M16_BODY_H40_0003_02100_Tool.bly	20/1
Brother		M16_BODY_H80_0003_02101.bly	20/1
Creative Suite		M16_BODY_H80_0003_02101.bmp	26/0
Creative Suite CS2		M16_BODY_H80_0003_02101.prt	20/1
		M16_BODY_H80_0003_02101.stp	20/1
discoi		M16_BODY_H80_0003_02101_Rough.prt	20/1
FCS		M16_BODY_H80_0003_02101_Rough.stp	20/1
📙 GuideLine3		M16_BODY_H80_0003_02101_Tool.bly	20/1
📜 Application		M16_BODY_H160_0003_02102.bly	21/0
📜 Converter		M16_BODY_H160_0003_02102.bmp	26/0
FCSSystem		M16_BODY_H160_0003_02102.prt	23/1
Help		M16_BODY_H160_0003_02102.stp	23/1
LicMan		M16_BODY_H160_0003_02102_Rough.prt	23/1
		M16_BODY_H160_0003_02102_Rough.stp	23/1
Misc		M16_BODY_H160_0003_02102_Tool.bly	21/0
📙 startup		M16_BODYES_H80_0003_00012.bly	21/0
FCS - Copy		M16_BODYES_H80_0003_00012.bmp	19/1
FMC Supervisor	$\checkmark$	<	
ns 7 items selected 348 KB			





- Create a new subtraction tool File structure
- M16\_BODY\_H80\_0003\_02101.bly: this file contains the body's geometry in a ligth format. This is the geometry used in Guideline for stack calculation;
- M16\_BODY\_H80\_0003\_02101.prt: this file contains body's, tool's and interfering body's geometry for Siemens NX. This is the component called by NX to build the assembly when Guideline transfers the clamping environment;
- M16\_BODY\_H80\_0003\_02101.stp: this file contains body's, tool's and interfering body's geometry used for translation in STEP/IGES using the option «Finishing»;
- M16\_BODY\_H80\_0003\_02101\_Rough.prt: this file contains body's, tool's and interfering body's geometry for building the clamping assembly for roughing in Siemens NX. This component is not managed by Guideline directly, but can be manually inserted by the user replacing the finishing version («Open Component As»). Seats creation will then use roughing geometry;





#### Create a new subtraction tool – File structure

• M16\_BODY\_H80\_0003\_02101\_Rough.stp: this file contains body's, tool's and interfering body's geometry used for translation in STEP/IGES using the option «Roughing»;

• M16\_BODY\_H80\_0003\_02101\_Tool.bly: this file contains the tool's geometry in a light format. This is the geometry used in Guideline for interference calculation;

Components of certain sizes might have not all the above files, or not all the solids in the files: this depends on the usage that they have in FCS Clamping System. Before modifications, please check the files/solids present for the element you want to work on and follow the procedure in the next pages.





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#### Create a new subtraction tool – Procedure

In order to modify the geometry of tools in the library, you should first create 3D models of the desired tools: you can do it using any CAD system. A good starting point can be to import the existing geometry and modify it. There are no particular rules to create the solid, just be sure to respect the nominal dimensions of FCS System.

Once you have the modified geometry ready, please do the following:

- 1. Within the same CAD file, create 3 solid bodies for:
  - A. The FCS element, standard or modified if the case;







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Once you have the modified geometry ready, please do the following:

- 1. Within the same CAD file, create 3 solid bodies for:
  - A. The FCS element, standard or modified if the case;
  - B. A body resulting from the union between the FCS element + the seat. This can be created with a boolean union or straight modeling it from scratch. Please, note that for this body, the diameter of the FCS element must be larger than the nominal dimension: for elements up to M10 it should be 3mm, for bigger components 5mm (referred to the diameter). Also, removing details (chamfers, blends, holes) from the FCS element is a good idea to minimize file size and maximise performance;







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Once you have the modified geometry ready, please do the following:

- 1. Within the same CAD file, create 3 solid bodies for:
  - A. The FCS element, standard or modified if the case;
  - B. A body resulting from the union between the FCS element + the seat. This can be created with a boolean union or straight modeling it from scratch. Please, note that for this body, the diameter of the FCS element must be larger than the nominal dimension: for elements up to M10 it should be 3mm, for bigger components 5mm (referred to the diameter). Also, removing details (chamfers, blends, holes) from the FCS element is a good idea to minimize file size and maximise performance;
  - C. A copy of the simplified FCS element, also with a larger diameter, but with no seat geometry. For sizes larger than M10, this body should be 2mm higher than the FCS element.







#### Creating a new subtraction tool – Procedure (cont'd)

- 2. Create a STEP file with all the solids above named as the element in library (M16\_BODY\_H80\_0003\_02101.stp in the example) and place it in the proper folder in libray (C:\FCS\GuideLine3\FCSSystem\M16 in the example)
- 3. Use Bly 3D Viewer to create Main and Tool file containing the FCS Element and the tool body respectively. Name them as convention (M16\_BODY\_H80\_0003\_02101.bly and M16\_BODY\_H80\_0003\_02101\_Tool.bly in the example) and place them in the proper folder in libray (C:\FCS\GuideLine3\FCSSystem\M16 in the example). If you don't have Bly 3D Viewer, you can send the STEP file to BCK Solutions and they will generate the .bly for you;
- Repeat point 1 and 2 and create a STEP file with the smaller seat for roughing, name it \_Rough (M16\_BODY\_H80\_0003\_02101\_Rough.stp in the example) and place it in the proper folder in libray (C:\FCS\GuideLine3\FCSSystem\M16 in the example)
- 5. OPTIONAL: if you use Siemens NX with Guideline integration, you should create the .prt files for finishing a roughing, name them as convention (M16\_BODY\_H80\_0003\_02101.prt and M16\_BODY\_H80\_0003\_02101\_Rough.prt) and place them in the directory.

