

NxUg to Eureka



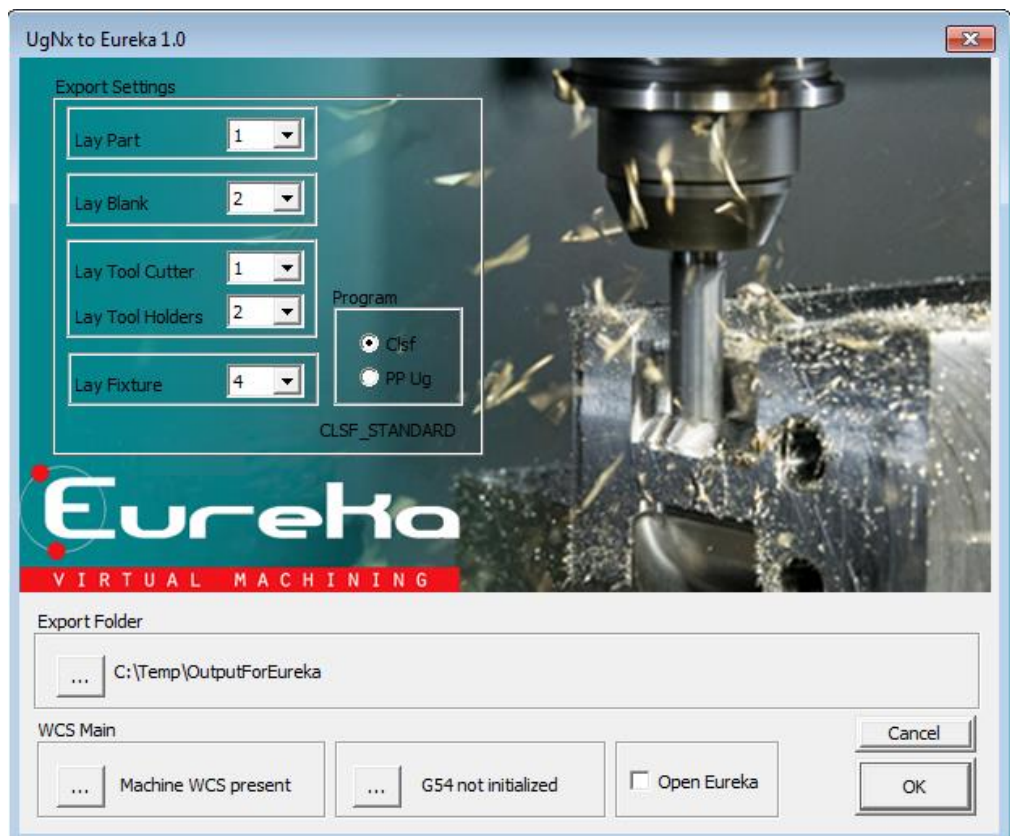
NxUg to Eureka

INTRODUCTION

Custom module in the UG NX environment to realize simulation setups in the Eureka Environment.

By a few steps, you can automatically create the Eureka setup and all the material you need to run.

The procedure, in addition to realizing the .epf file, exports the part-program and files of the finite and raw part, of the equipment, of the tools in the format.stl



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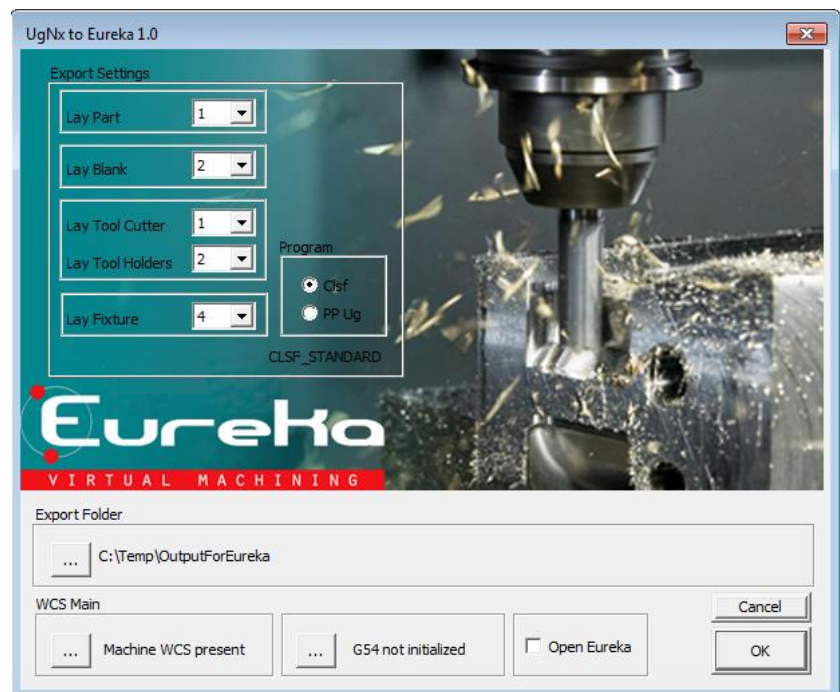
INSTALLATION

Run the file NxEurekaSetup.exe with administrator privileges for installer the program.



EXECUTION

In the UG NX software found the toolbar <<Nx_Eureka Interface>> with the button down with start the plug-in.

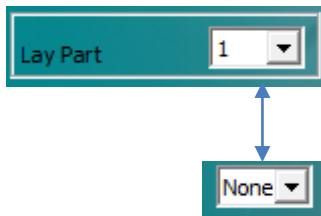


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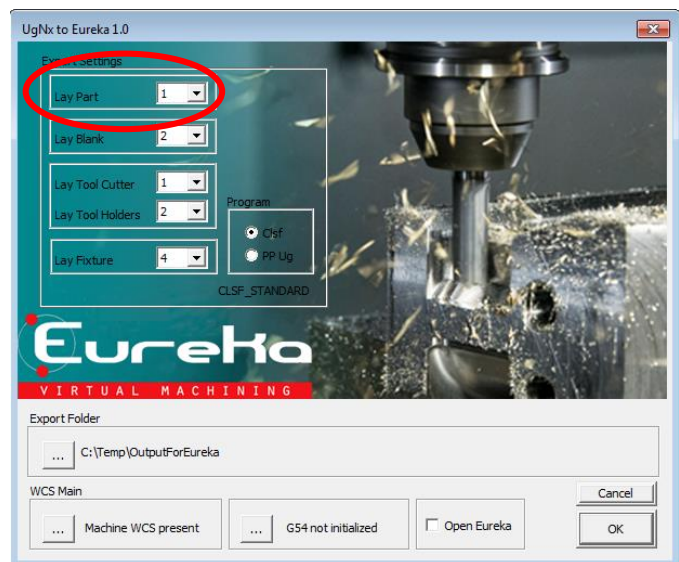
USE – Finite part

The procedure exports the template, or templates, of the finite part that is in the layer number defined in the "Lay part" menu (from 1 to 256). The procedure exports all solids that are on that layer as raw parts.

Select the layer in the interface where the finished part is located, or the parts, through the dropdown menu



If none is selected the procedure does not export this item

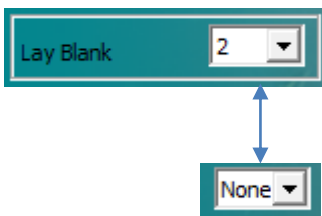


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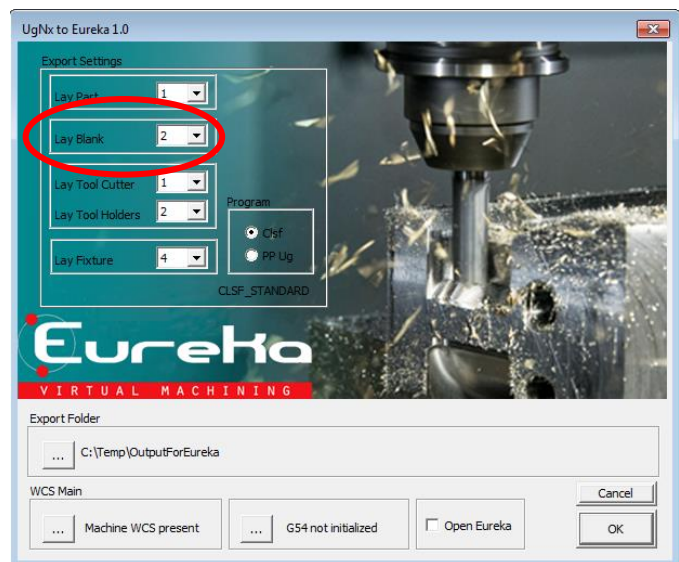
USAGE – Raw part

The procedure exports the model, or the templates, of the raw part that finds it in the layer number defined in the "Lay Blank" menu (from 1 to 256). The procedure exports all solids that are located on that layer as finite parts.

Select the layer in the interface where the raw part is located, or the parts, through a dropdown menu



If none is selected the procedure does not export this item

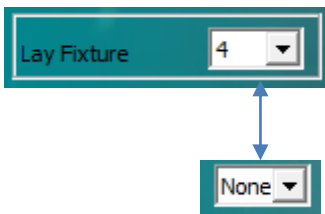


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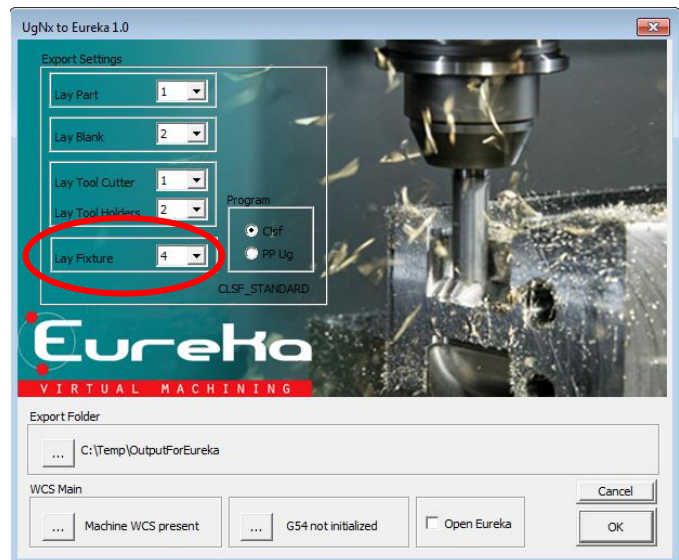
USAGE – Equipment

The procedure exports the model, or models, of the equipment that is in the layer number defined in the "Lay Fixture" menu (from 1 to 256). The procedure exports all solids that are located on that layer as equipment.

Select the layer in the interface where the equipment is located, or the equipment, through a drop-down menu



If none is selected the procedure does not export this item



NxUg to Eureka

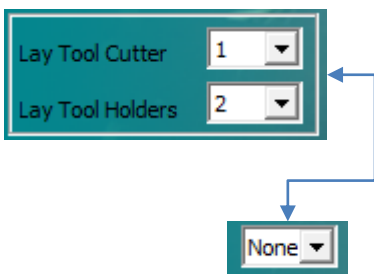
USAGE – Tools

Note To Extract tool solids, the procedure uses the .prt files that are linked to the definition of the UG Library tools ([Attachment 1](#)). If the kit is not in the library or there is no .PRT, the procedure does not export any solids to the kit itself.

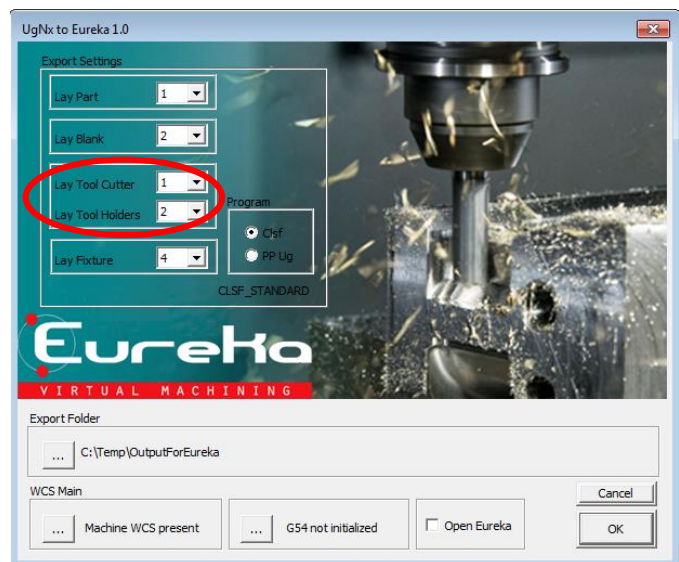
The procedure considers that the tools are composed of a holder and a cutter part. The holder part is the solid, or solids, of the kit without the sharp part. A kit can be composed of multiple holder.

The procedure exports the model, or models, of the sharp part that is in the layer number defined in the "Lay Tool Cutter" menu (1 to 256) and the holder part that is in the layer number defined in the "Lay Tool Holders" menu (1 to 256). The procedure exports all solids that are located on those layers as a sharp part and part holder.

Select the layer in the interface
Where the sharp part is
The holder, or parts, by means of
drop-down menus.



If none is selected the procedure does not export this item



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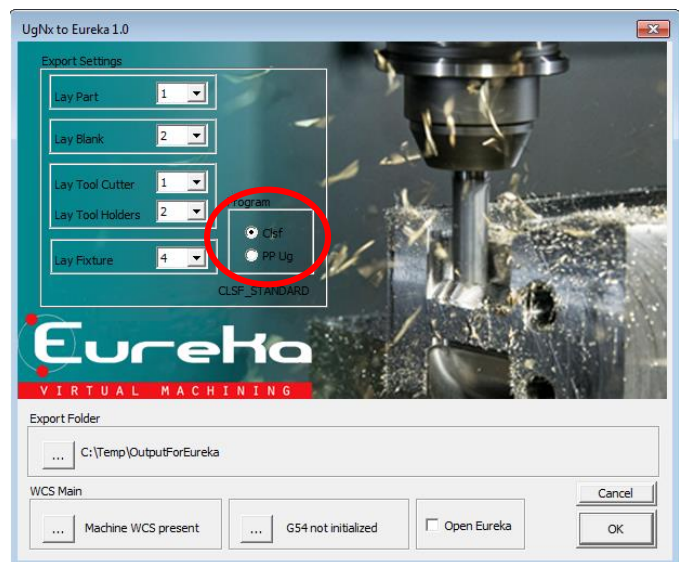
USAGE – tool path

The procedure exports the tool path in CLSF format or post processor. In the case of the tool path derived from post processor, the procedure considers the list of PostUG installed on UG NX.

Select the interface if exporting the tool path CLSF or PostUG.



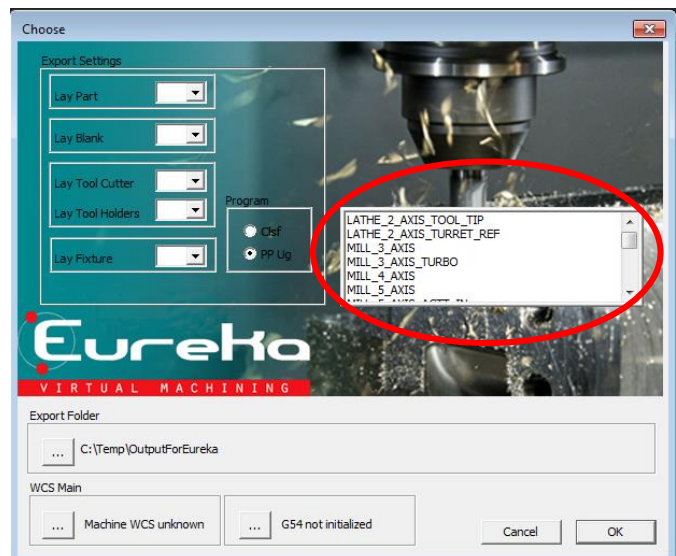
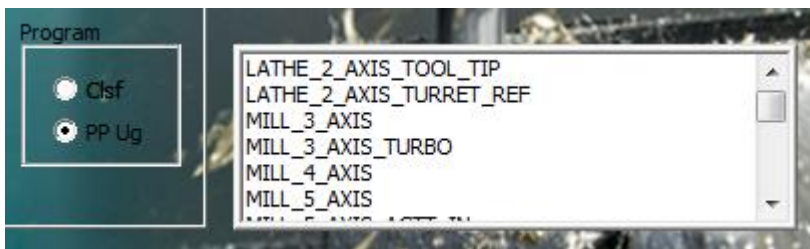
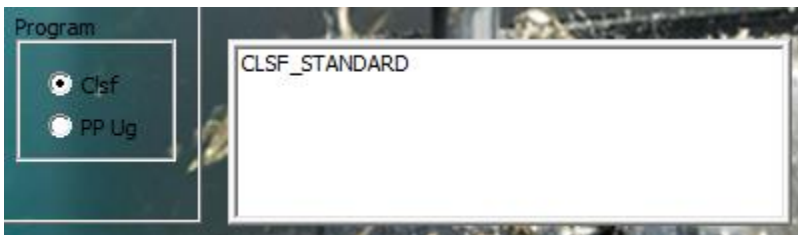
If something is selected is visible in the interface.



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USAGE – tool path

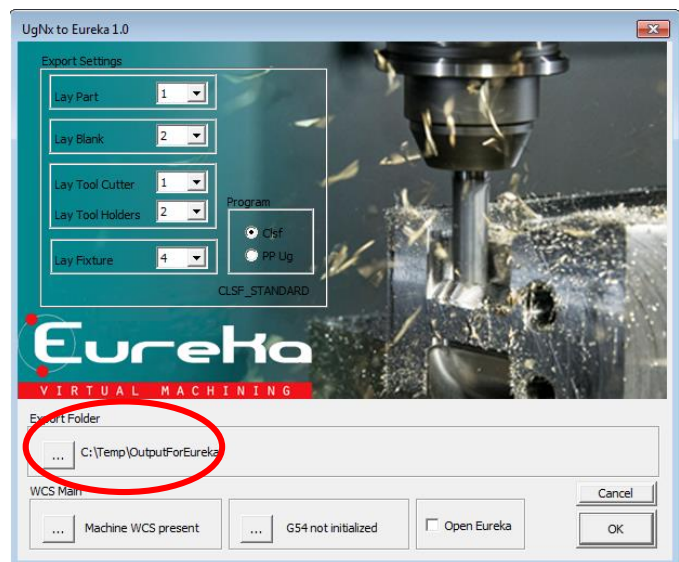
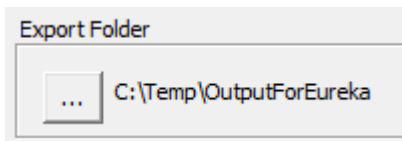
After you select the output type in the adjacent window, the available entries are displayed. Select the desired mode. The list of entries depends on the type of implementation of the UG NX.



NxUg to Eureka

USAGE – output file

You can select the folder where you want to export the files to be used for simulation on Eureka by clicking on the button in the "Export Folder" section.



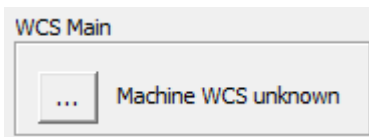
NxUg to Eureka

USAGE – zero machine

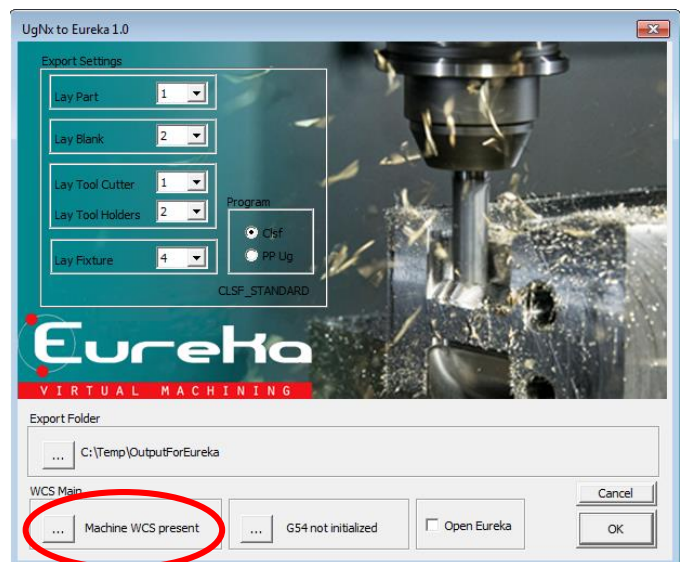
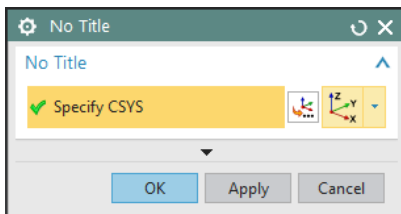
The zero machine means the point where each coordinate is equal to 0 in the machine's Eureka Setup.

To have a correct positioning of the 3d models on the Eureka Simulator, you must define the zero machine in the cam UG.

To define the zero machine, click on the button in the "WCS Main" section. Possibly the user can change the position of the 3d models on the simulator.



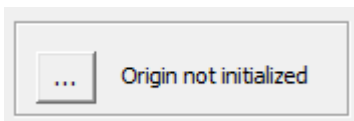
Using the Selection window, click the representative point of the zero machine



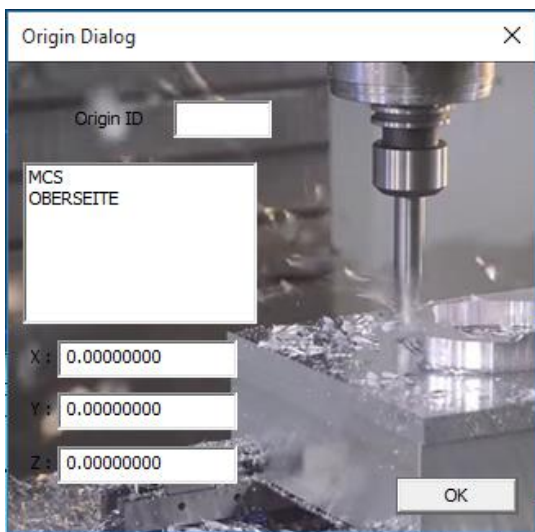
NxUg to Eureka

USAGE – Origin (Ex.:G54)

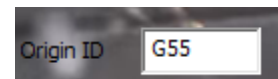
There is the possibility to define the origin of part-program through the button in the section <<Origin>>.



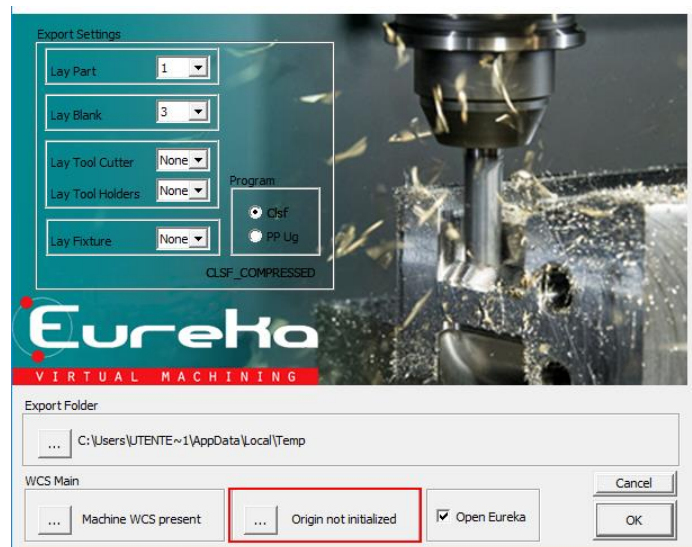
Clicking the button <<Origin>> are showing all MCS inside the part-program with the possibility to select one of these and you would modify the values (coordinates).



When you choose the cam MCS you have to insert the

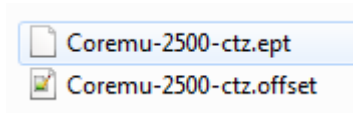


ID of the origin that you want to set too.

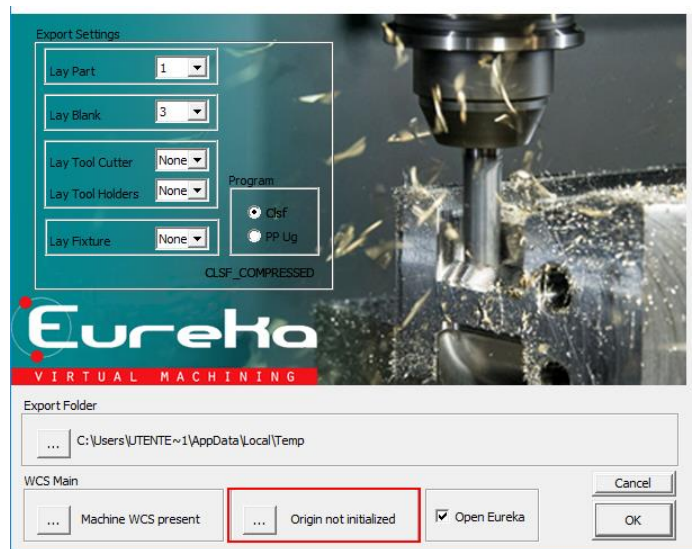


NxUg to Eureka

If the machine origin requires an offset respect to zero table, must be present a text file, with format below, with the values that will be added to the selected MCS coordinates. The file reside in the same folder as eureka machine template, file name will have the same name as template ept with extension <<.offset>>.



```
Coremu-2500-ctz.offset
1 # This file contains the offset values to be considered between
2 # the origin machine and point of table/plateau reference (point of WCS main selected).
3 # Enter values in the following format:
4 # Offset Origin:X,Y,Z
5 Offset Origin:100,100,200
```



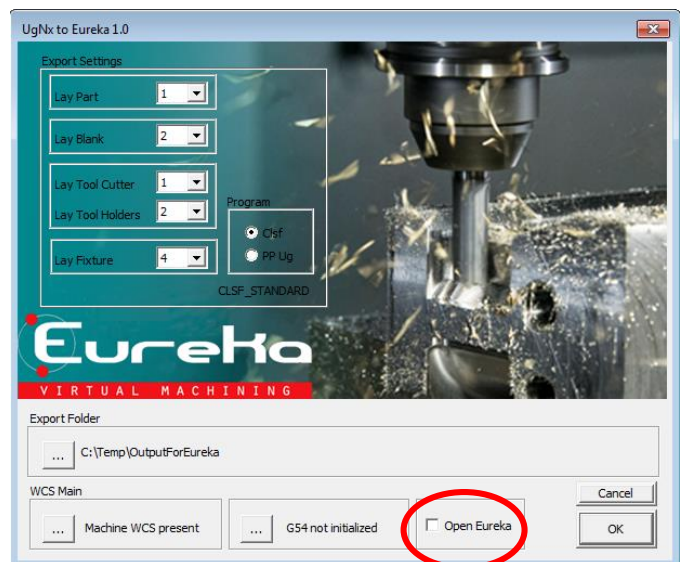
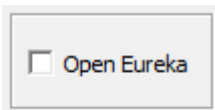
NxUg to Eureka

USAGE – opening Eureka

The user can choose whether to export the data and manually open Eureka, rather than automatically opening at the end of the procedure.

Select the "Open Eureka" flag if you want Eureka to open automatically at the end of the export.

Do not select the "open Eureka" flag if you want Eureka to do not open automatically at the end of the export.

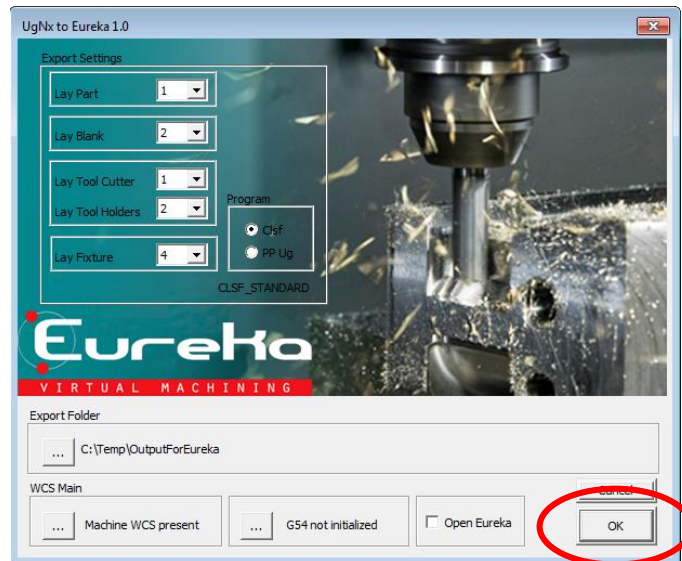
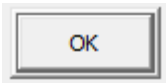


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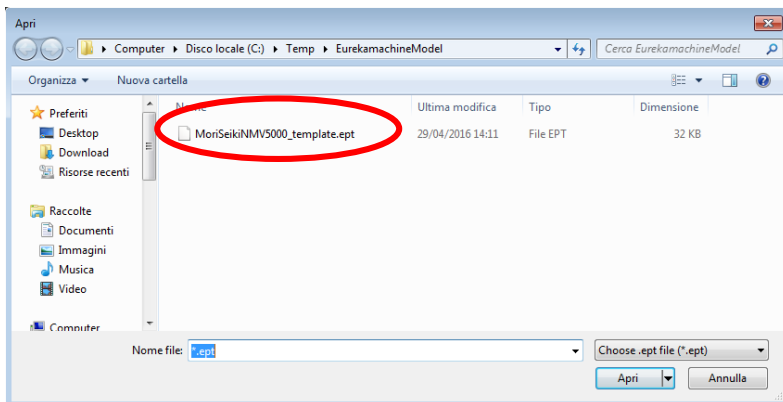
USAGE – Eureka Machine Setup Selection

Once the fields are filled in the form, click OK.

Caution-it is necessary that all fields are filled in order for the OK button to be completed and the procedure continues.



A procedure requires you to select the Eureka machine setup that you want to use for the simulation (.ept).

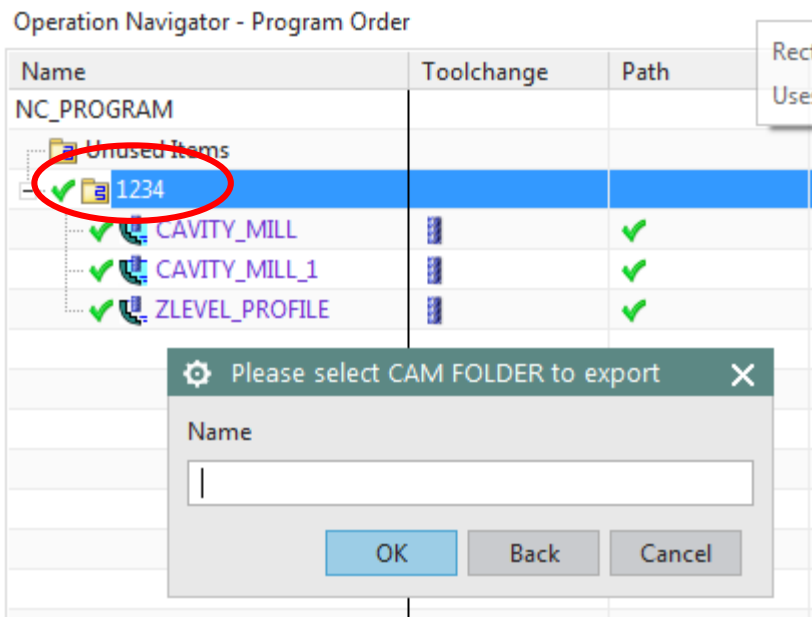


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USAGE – UG CAM Program Selection

The last action to run is to select the CAM program that you want to perform export/simulation.

In the "Operation Navigator" tree, select a single CAM operation or CAM folder. The procedure exports the previously mentioned files that belong to the single CAM operation or to all operations that reside in the selected folder.



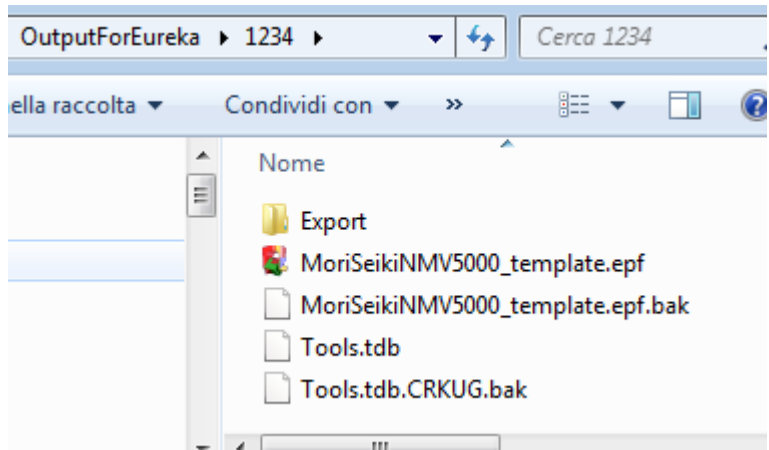
NxUg to Eureka

USAGE – Output File and Folders

At this point, the procedure exports the required files to the selected destination folder in the initial form.

The output files are:

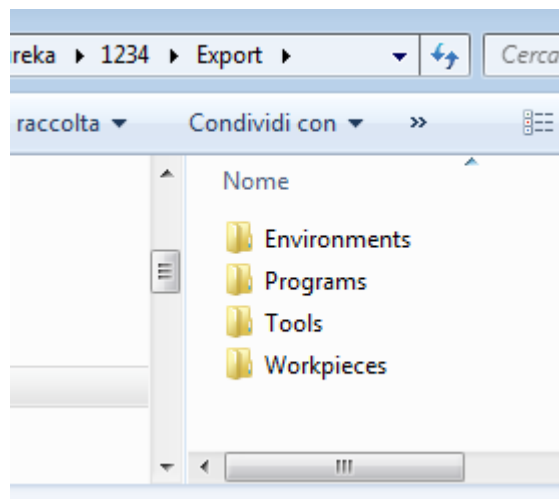
- **.epf** Setup Eureka Machine
With inside Solids finished,
Rough, equipment, tools
- **.tbd** Tool Library
- **"Export"** folder



In the "Export" folder, the folders are created:

Environments

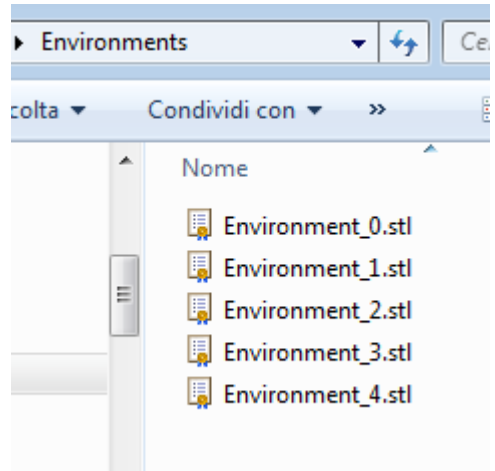
- Environments
- Programs
- Tools
- Workpieces



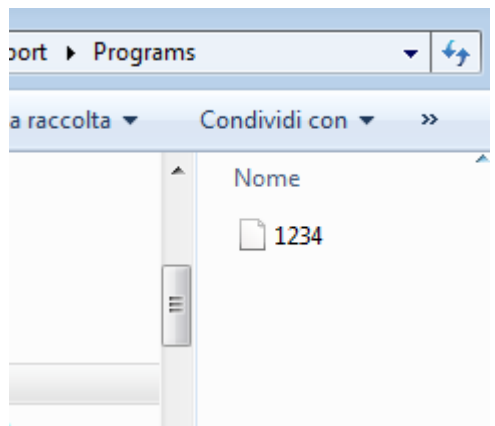
NxUg to Eureka

USAGE – Output File and Folders

The "Environments" folder creates files for the equipment models.
A **.stl** file for each 3d solid.



In the "Programs" folder, the file is created to the tool path (CLSF or postNX according to the selected form).



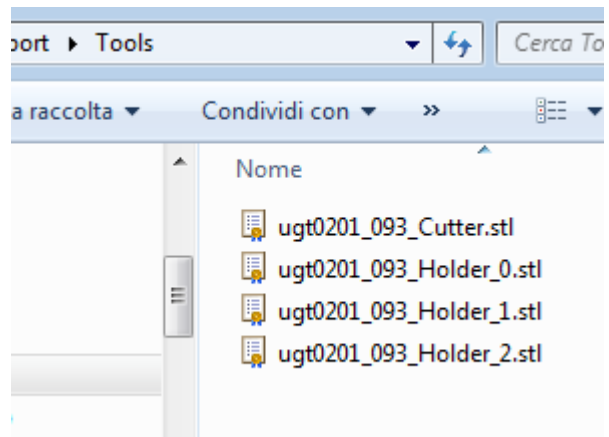
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USAGE – Output File and Folders

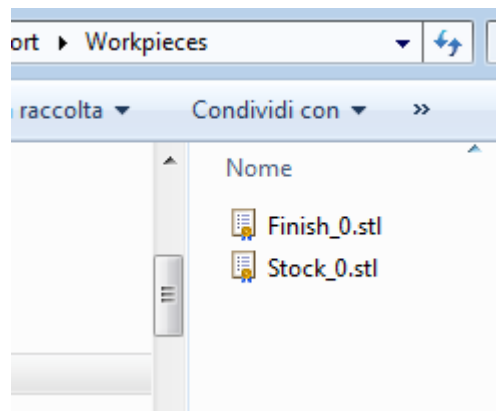
The "Tools" folder creates files for kit templates. The files are named as the kit name on UG.

A **.stl** file for each 3d solid that represents the cutter part.

A **.stl** file for each 3d solid that represents a kit holder part.



In the "Workpieces" folder, are created the files related to the 3d models of the part (or parts), of the finite and part, or parts of the stock.



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ATTACHMENTS 1 – NX Tools Library

Below an extract of the Official NX Manual for the Tool ASCII Library management.

tool library Advanced Search

Home > CAM > Manufacturing General > CAM libraries

Topics

- CAU
- CAM
 - Manufacturing General
 - Manufacturing introduction
 - CAM Express
 - Examining a manufacturing part
 - Creating a Manufacturing setup
 - Manufacturing operations
 - Cutting feeds and spindle speeds
 - Geometry, tool, method, and program parent
 - Generating and reviewing your tool path
 - Customizing the tool path display for an opera
 - Displaying cutter paths
 - Verify tool path
 - Operation Navigator
 - Shop Documentation
- CAM libraries
 - Library definition files
 - Library aliases
 - Library class hierarchies
 - Cutter, tool holder, and device libraries**
 - Retrieve a tool from a library
 - Retrieve a tool or device into a specific j
 - Example: Retrieve a turning tool with a
 - Retrieve a mill tool holder from the libr
 - Search Criteria dialog box
 - Turning tool tips and tracking points
 - Determining Search Criteria

Cutter, tool holder, and device libraries

You can store tools in and retrieve tools from the cutting tool, tool holder, and device libraries. All three libraries have the same structure, and you use and customize them in the same way.

When you use the libraries, you can define a tool once, and reuse the definition whenever you need it.

When you retrieve objects from a library, you:

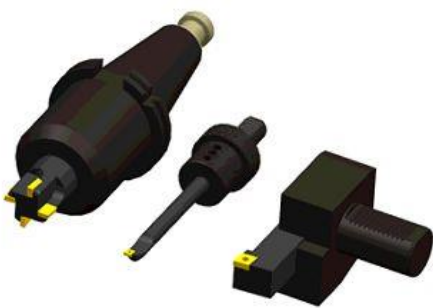
1. Specify a classification, such as mill, turning tool, or drill.
2. Specify parameters to filter the results, such as diameter, length, or tip angle.
3. Select a tool, holder, or device that matches your criteria.

Cutting tool libraries

A cutting tool library classifies cutting tools and stores tool parameters. NX uses these parameters to display standard tools and calculate material removal in simulations. A solid model of a standard tool need not exist. Use the solid tool library if the dimensions are non-standard.

If a cutting tool has a holder defined, NX loads the tool holder when it loads the tool.

Cutting tools can have an optional associated part file which NX can use to display a tool. This is usually an assembly of the components used to make up a tool, such as tool and holder, cutting inserts, and so on. The graphics folder of the tool library, `SUGII_CAM_LIBRARY_DIR\tool\graphics`, contains several examples of cutting tools.



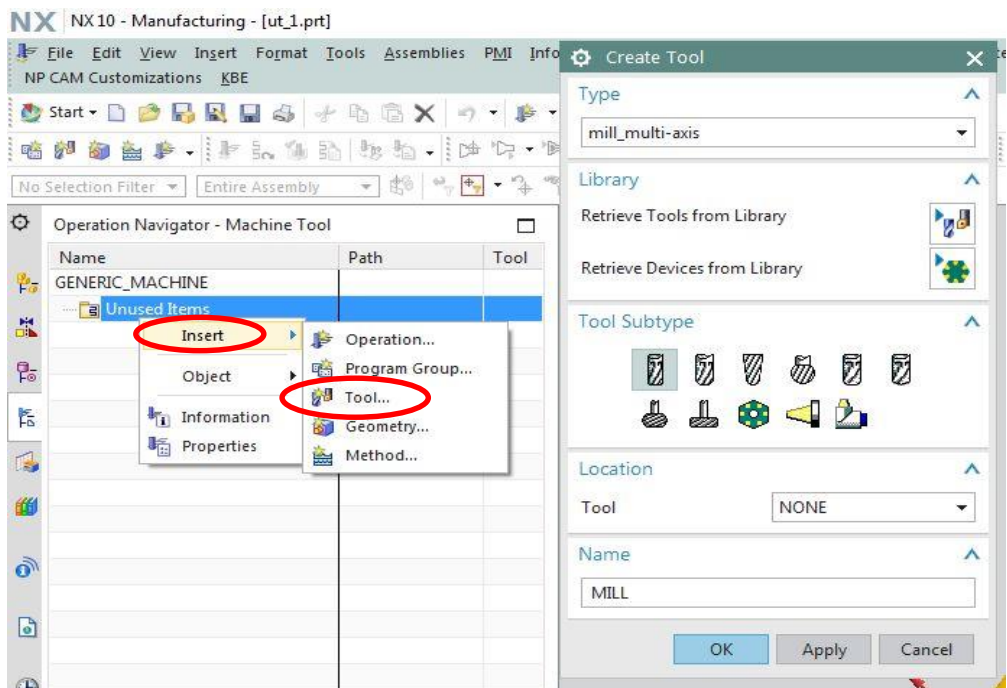
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ATTACHMENTS 1 – NX Tools Library

This part of manual illustrate the creation tools kit inside of a library and its recall in a part-program.

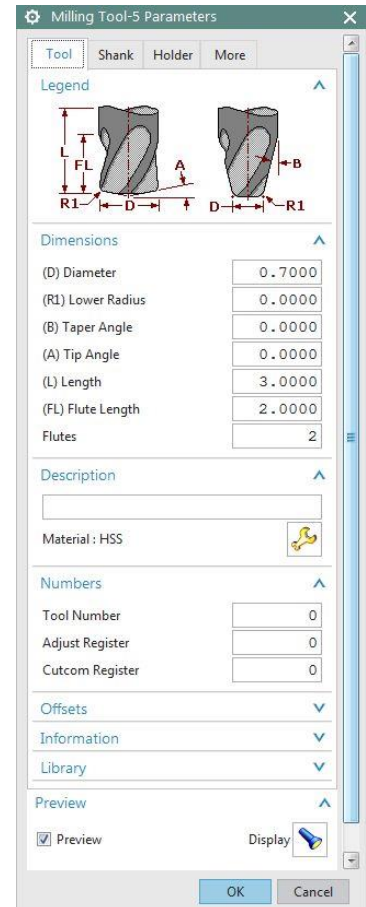
Creation kit for NX tool library

Presuppose you already have the assembly **.prt** with the components of the holder kit part, you set the type and the tool name of <<insert tool>> menu.



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In the <<Tool>> Tab you define geometry, dimensions and cut angles of the cutter part.

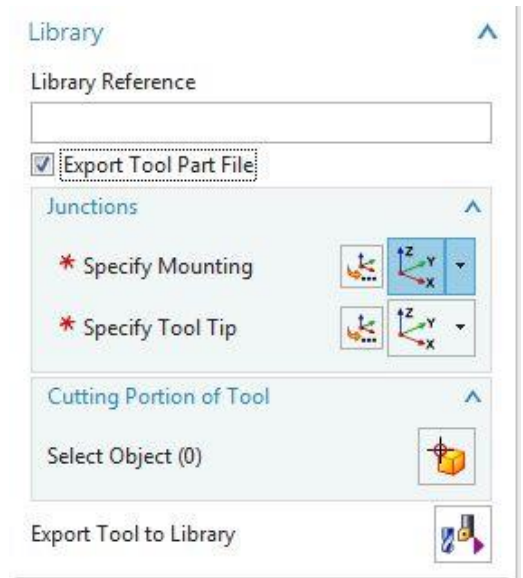


While the <<Library>> Section you flag <<Export Tool Part File>> and you set the export tool parameters :

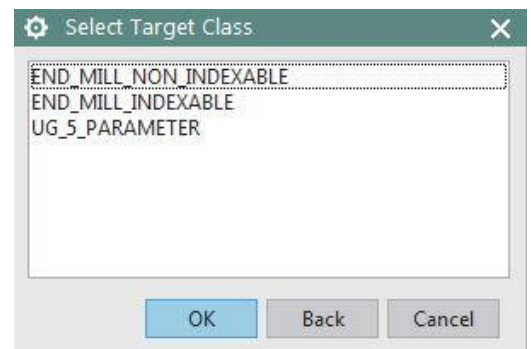
- Library Reference: Represents the reference name with export the kit inside the ASCII Library;
- Specify Mounting: Set the reference point with the tool attachment on the machine;
- Specify Too Tip: Set the tracking point, must coincident with the absolute WCS origin and CSYS orient same as MCS orient.

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At the end you select the single Tool component through button <<Select Object>> and you click <Export Tool To Library> to start the export on NX Library.



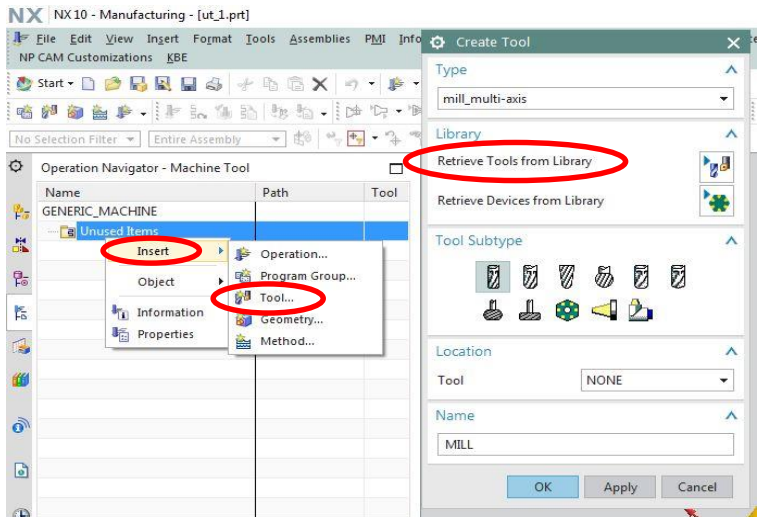
At this time, you can select the tool target class, how you can see in the picture below, so you have completed the export process.



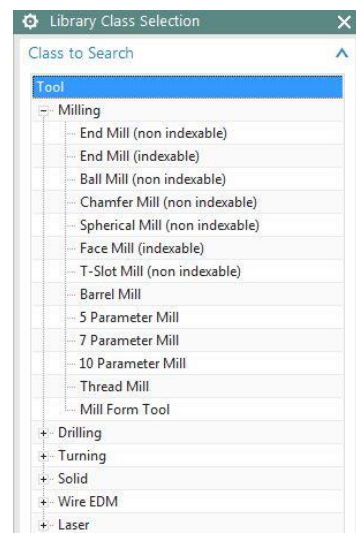
NxUg to Eureka

Choose NX LibraryTools

In the NX CAM Environment you can add tool through <<Insert Tool>> menu and after click << Retrieve Tools from Library>>.



After you can see another window where you can filter the tools selected their target class.



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After you can apply a second search filter to discover cutter geometry or tool name.

The 'Search Criteria' dialog box is shown with the following sections:

- Class Description:** 5 Parameter Mill
- Legend:** Two diagrams of a 5-flute mill. The left diagram labels dimensions L (total length), FL (flute length), R1 (corner radius), D (diameter), and A (taper angle). The right diagram labels dimensions D (diameter), R1 (corner radius), and B (taper angle).
- Search Parameters:**
 - Library Units: Inch (dropdown)
 - Libref: [text input]
 - (D) Diameter: [text input]
 - (FL) Flute Length: [text input]
 - (R1) Corner Radius: [text input]
 - (B) Taper Angle: [text input]
 - (SD) Shank Diameter: [text input]
 - Description: [text input]
- Additional Search Criteria:** Enter Query [text input]
- Actions:** List Results (icon), Clear Query (icon), Count Matches (icon)
- Buttons:** OK, Back, Cancel

At the end the window present a tools list to choose.

The 'Search Result' dialog box displays a table of matching items. The table has the following columns: Libref, Description, Material, Pocket Id, and Carrier Id.

Libref	Description	Material	Pocket Id	Carrier Id
ugti0363_001	Back-Countersinking Tool 0.362 inch (M8)	High Speed Steel		
ugti0363_002	Back-Countersinking Tool 0.445 inch (M10)	High Speed Steel		
ugti0363_003	Back-Countersinking Tool 0.528 inch (M12)	High Speed Steel		
ugti0363_004	Back-Countersinking Tool 0.610 inch (M14)	High Speed Steel		
ugti0363_005	Back-Countersinking Tool 0.693 inch (M16)	High Speed Steel		
ugti0363_006	Back-Countersinking Tool 0.776 inch (M18)	High Speed Steel		
ugti0363_007	Back-Countersinking Tool 0.858 inch (M20)	High Speed Steel		
ugti0363_008	Back-Countersinking Tool 0.346 inch	High Speed Steel		
ugti0363_009	Back-Countersinking Tool 0.386 inch	High Speed Steel		
ugti0363_010	Back-Countersinking Tool 0.472 inch	High Speed Steel		
ugti0363_011	Back-Countersinking Tool 0.890 inch	High Speed Steel		

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Where you can select the tool and with one click on <<OK>> button, you can add it in the part program.

