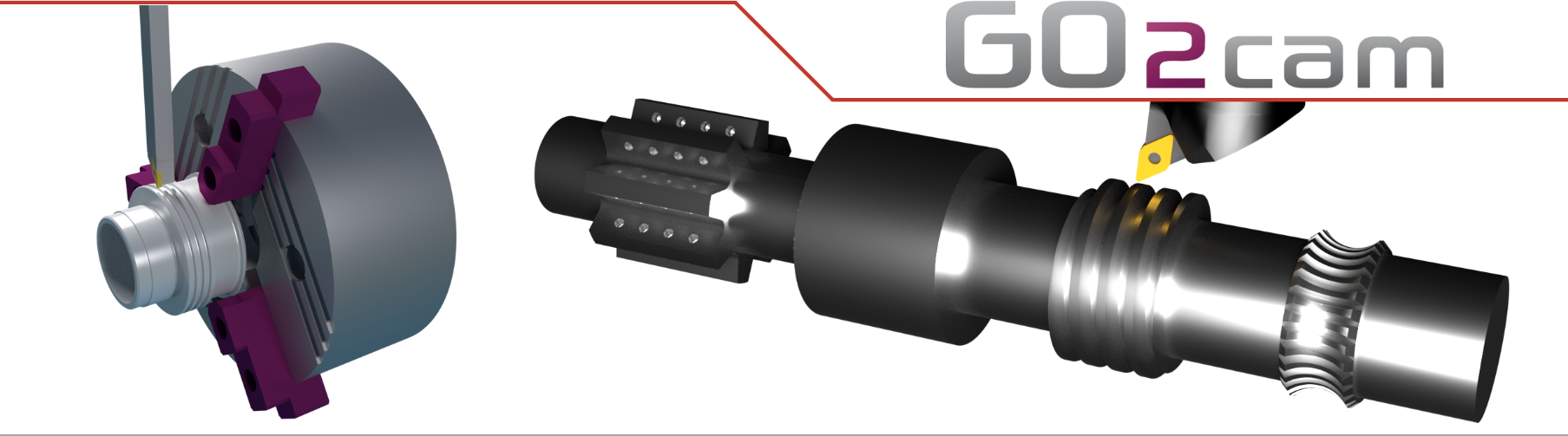
T02

T02

GO2cam V6.10

Tutorial

T02 – Cover



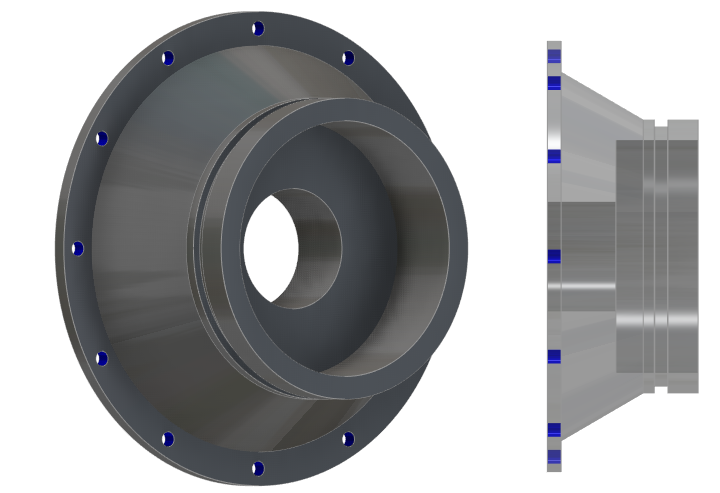
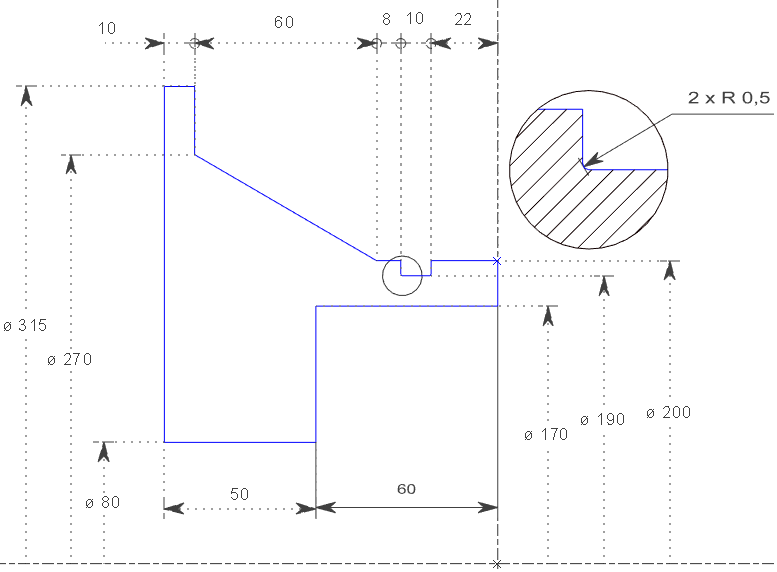
## T02 - Cover

**Introduction**

Welcome to this cover creation tutorial.

This tutorial has the objective to help the user:

* Learn how to use the turning environment in GO2cam
* Understand the uses and capabilities of GO2cam in turning.
* Get used to the user interface and the mouse actions by practicing on a simple example.

**** ****

**Extra files**

In the Basic Training Pack, you can find:

* the pdf file of this tutorial
* the layout design

The tutorials are additional resources to GO2cam's online help. You can access the online help by selecting the Help menu in GO2cam or by pressing F1.

Process for the Design:

|  |  |
| --- | --- |
| 1. Choice of the Turning environment in the homepage:  * Left-click on the Turning icon the most on the right   Note: the icons represent the type of product. If your licence do not include it, the product icon is greyed.  The software opens in design mode by default.   * Left-click on Shaft * Left-click on Shaft Creation |  |
| 1. Sketch of the part:  * Left-click on the Shaft Creation   Choice of the starting point: Z=0, D=200   * Type in 0 for Z and 200 for the Diameter * Left-click on  to confirm the starting point   ****Note:**** GO2cam proposes you alternate length and diameter. This choice is shown by an orange line linked to the motion of the cursor**.** |  |
| 1. Length L=22:  * Left-click on , then choose horizontal segment   You can enter positive or negative values of length depending on the direction of the + sign defined by  or .   * The button should be in that position , if it is not, click on the button to change the direction. * Type in 22 and press Enter. |  |
| Diameter Ø=190:  * Type in 190 and press Enter. | A picture containing box and whisker chart  Description automatically generated |
| Length L=10  * Type in 10 and press Enter. |  |
| Groove end: 2 solutions :   * Enter the value with the keyboard : type in 200 and press Enter   Or   * Left-click on the first horizontal segment which was first created and which has the same value. | Chart, diagram, box and whisker chart  Description automatically generated |
| Length L=8:  * Type in 8 and press Enter. |  |
| Creation of a cone L=60, Ø=270**:**  * Left-click on * Choose the mode of creation of the cone clicking onuntil it changes so that you can define the cone by its length and diameter * Type in 60 for the length, 270 for the diameter and press Enter * Left-click on |  |
| Diameter Ø=315:  * Left-click on * Type in 315 and press Enter.   You are no longer in the cone creation mode. GO2cam proposes you once again to define the length and diameter alternatively. |  |
| Length L=10:  * Type in 10 and press Enter. |  |
| Diameter Ø=80:  * Type in 80 and press Enter. |  |
| Length L=50:The direction of the + sign is still**,** there are 2 possible solutions**:**  * Type in - 50, Enter key   **Or**   * Change the direction with a click on and Left-click on sur , which becomes * Type in 50, Enter key |  |
| Diameter Ø=170: We are going to make a mistake on purpose**:**   * Type in 150 and press Enter. | Diagram  Description automatically generated |
| Finishing of the sketch: To finish the part, we can click on existing elements:   * Left-click on **t**he vertical axis * Left-click onthe first horizontal segment which was first created * Left-click on * Esc key to end the Shaft function |  |
| Modification : We are going to correct the mistake done on step 13; the actual diameter is 170, we typed a value of 150!**.**   * Left-click twice directly on 150   or Right-click on the valuethen choose ‘Edit element’     * Replace the value of 150 with 170 in the field, and enter * Left-click on **the Green Tick** to validate |  |
| Finishing, definition of fillets for the groove:  * Left-click on Zoom , then click on the screen twice to select the zone you want to zoom like shown on the picture * Click on **Esc** key to end the zoom function * Left-click on the **Fillets** function   To create a fillet you can either click directly on the corner where the fillet is or click on the 2 elements whose intersection is the fillet.   * Type in **0.5** and press Enter.     Note : the decimal symbol is the **point**, not the comma.   * Left-click on the **2 corners** of the groove to apply the fillet. * **Press ESC key** to end the Fillets function * Left-click on   The wireframe design is finished. |  |

Operating Procedure for the Machining:

Targets:

* Application of cycles for turning and grooving :
* Selection of geometry for machining
* Simple cycle parameters definition
* Selection of tool
* Use machining tree: modifying a cycle
* Tool path simulation

|  |  |
| --- | --- |
| Turning:  * Left-click on Turning * Left-click on   The stock is automatically created in relation to the profile.  It is a cylindrical stock defined with a constant 5 mm overflow around the profile.  We are going to modify the stock manually**.**   * In the Zmaxi field, type 2 to make the stock more than 2mm behind the origin     The stock is cored out:   * Change the value in D inner field to 70 and press ENTER      * Left-click on to validate |  |

Ope 10 Facing

|  |  |
| --- | --- |
| 1. Selection of geometry:  * Left-click on   Note: It is recommended to always zoom during selection to facilitate selection of profiles.   * Left-click onSelection * Left-click on the first element of the path as in the drawing and ensure that the arrow is pointing downwards. * Left-click on Profile OK * No need to modify the approach and return vectors which are tangent to the element shown. |  |
| * Tool selection: * Left-click on , then on Turning Facing Cutter   Graphical user interface, text, application, email  Description automatically generated   * Left-click on * Select **MT\_Sandvik\_2020\_Right** and   **MT\_Sandvik\_2020\_Left.**   * Left-click on **Confirm** to download the tools library. * After downloading and installing, select **MT\_Sandvik\_2020\_Left** in the library * In the list of tool, Left-click on   **" MS PCLNL 2020K 12"** | Logo  Description automatically generated |
| 1. Selection of the machining cycle:  * Left-click on  then on Roughing * In the list of techno, Left-click on **"Main Spindle - Face"** |  |
| 1. Toolpath calculation :  * Left-click on **Cycle Calculation** * Right-click on the cycle and rename it to ‘Facing’ |  |

Ope 20 Straight cutting (roughing)

|  |  |
| --- | --- |
| 1. Selection of geometry :  * Left-click on * Left-click on Selection * Left-click on the first path element, on the right (image A), click on OK * Left-click on the last path element, on the left (image B), click on OK * Left-click on Profile OK   **Note:** No need to modify the approach vector which are tangent to the element shown.   * **Select the Ret Vect check box, then manually enter the value 90 and press Enter** |  |
| 1. Selection of Tool :  * Left-click on , then on Turning Facing Cutter * Select **MT\_Sandvik\_2020\_Left** under user * In the list of tool, Left-click on **" MS PDJNL 2020K 11"** | Logo  Description automatically generated |
| 1. Selection of machining cycle :  * Left-click on  then on Roughing * In the list of techno, Left-click on **"Main Spindle - Outside"** |  |
| 1. Toolpath calculation :  * Left-click on **Cycle Calculation**   The toolpath is drawn, the evolutive stock is machined.   * Right-click on the cycle and rename it to ‘Outside Roughing’ |  |

Ope 30 Straight cutting (finish)

|  |  |
| --- | --- |
| 1. Selection of geometry :   **The menu**  **is still active.**   * Left-click on * In the machining tree left-click on  to expand the details for the outside roughing cycle * Left-click and drag and drop the profile on the screen.   **Note:** This method will copy the profile selection of the roughing cycle to the one currently being defined thus avoiding making the same selection multiple times. |  |
| 1. Selection of Tool :  * Left-click on , then on Turning Facing Cutter * Select **MT\_Sandvik\_2020\_Left** under user * In the list of tool, left click on **" MS PDJNL 2020K 11"** | Logo  Description automatically generated |
| 1. Selection of machining cycle :  * Left-click on Cycles then on Finish * In the list of techno, left-click on **"Main Spindle - Outside"** |  |
| * Toolpath calculation : * Left-click on Cycle Calculation * Right-click on the cycle and rename it to ‘Outside Finish’ |  |

Ope 40 Groove cutting

|  |  |
| --- | --- |
| 1. Start:  * Left-click on * Left-click on Zoom  then click twice on the screen to select the area you want to zoom like shown on the picture. |  |
| 1. Selection of geometry :  * Left-click on * Left-click on the first side of the groove (image A) * Left-click on the last side of the groove (image B)      * Left-click on **Profil OK** * No need to modify the approach and return vectors which are tangent to the elements shown. |  |
| * Selection of Tool : * Left-click on Groove cutter * Select **MT\_Sandvik\_2020\_Left** under user * In the list of tool, left mouse click on **" MS LF123J13-2020BM"** |  |
| 1. Selection of machining cycle :  * Left-click on , then on Direct |  |
| 1. Toolpath calculation :  * Left-click on **Cycle Calculation** |  |

Ope 50 Boring

|  |  |
| --- | --- |
| Selection of geometry :    * Left-click on * Left-click on **geometry selection** * Left-click on **the first element of the path, right (Figure A)** * Left-click on **the last element of the path, left (Figure B)**      * Left-click on **Profil OK** |  |
| Selection of Tool :  * Left-click on tools , then on Boring cutter   Graphical user interface, text, application, email  Description automatically generated   * Left-click on * Select **ST\_Sandvik\_D12\_Right** and   **ST\_Sandvik\_D12\_Left.**   * Left-click on **Confirm** to download the tools library. * After downloading and installing, select **ST\_Sandvik\_1212\_Left** * In the list of tool, select **MS A12M-SCLCL 06-R** |  |
| Selection of machining cycle :  * Left-click on Cycles , then on Roughing * In the list of techno, left mouse click on **"Main Spindle - Outside"** |  |
| * Toolpath calculation : * Left-click on **Cycle Calculation** * Right-click on the cycle and rename it to ‘Inside Roughing’ |  |

Simulation and NC blocks output

|  |  |
| --- | --- |
| 1. Simulation :  * Left-click on * Left-click on ****Simulation**** * Left-click onto activate the step by step mode**.** Then press the space key to go forward in the simulation step by step. * If you prefer continuous simulation, click on * Left-click onor *Esc* to stop the simulation   You can rotate the part by holding the left-click and moving the mouse. |  |
| 1. NC Code:  * Left-click on * Among the proposed list, left-click on the post-processor "**T67\_Fanuc**" * Left-click on **Open** * Left-click on **Confirm**   The NC Code is generated. | |

## **C axis operations**

|  |  |
| --- | --- |
| Switching to Reference Plane for Milling:  * Go back to **Design** menu * Left-click on current plane tab which is in the **Revolution** plan and change it to **Reference** plane |  |
| Design of a hole:  * Left-click on **Wireframe**, then select **Holes** * Left-click on the function **Standard Holes** * Select type of holes and choose **Through smoothed hole** * Left-click twice in the **Diameter** box and type 10      * Left-click twice in **Altitude** box and type -100      * Type the value for **X** as 292.5/2 and press **Enter** * Type the value for **Y** as 0 and press **Enter**      * Left-click on |  |
| Start the milling:  * Left-click on **Milling** * Left-click on **Hole/Feature**, then select **Manual Holes** |  |
| Selection of geometry :  * Left-click on * Left-click on the hole |  |
| Selection of Tool :  * Left-click on tools , then on drill   Graphical user interface, text, application, email  Description automatically generated   * Left-click on * Select **Standard** * Left-click on **Confirm** to download * After the download and installation, select **Standard** under **User** * Left-click on **"Drill D10"** | Graphical user interface, application  Description automatically generated |
| Selection of machining cycle :  * Left-click on Cycles  then on **Drilling** * In the list of techno, Left-click on **Simple mode Auto** |  |
| * Toolpath calculation : * Left-click on **Cycle Calculation** * Right-click on the cycle and rename it to ‘Drilling D10’   The path is displayed on the screen. | Diagram, schematic  Description automatically generated |
| Repeat the cycle in the machining tree:  * Left-click on **Drilling D10** cycle in machining tree * Select **Operation Copy** * Choose **Rotation** * Left-click twice in **Number** field, type in **11** and press **Enter** * Check that the coordinates of the center are **0, 0, 0** * Left-click twice in **Angle** field, type in **30**, **Enter** key      * Left-click on on **Drilling** in the machining tree to position the subprogram call after the drilling   The 12 holes are now machined. |  |
| 1. Simulation :  * Left-click on * Left-click on ****Simulation**** * Left-click on |  |
| 1. NC Code :  * Left-click on * Among the proposed list, click on the post-processor **T67\_Fanuc** * Left-click on **Open** * Left-click on **Confirm**   The NC Code is generated. | |