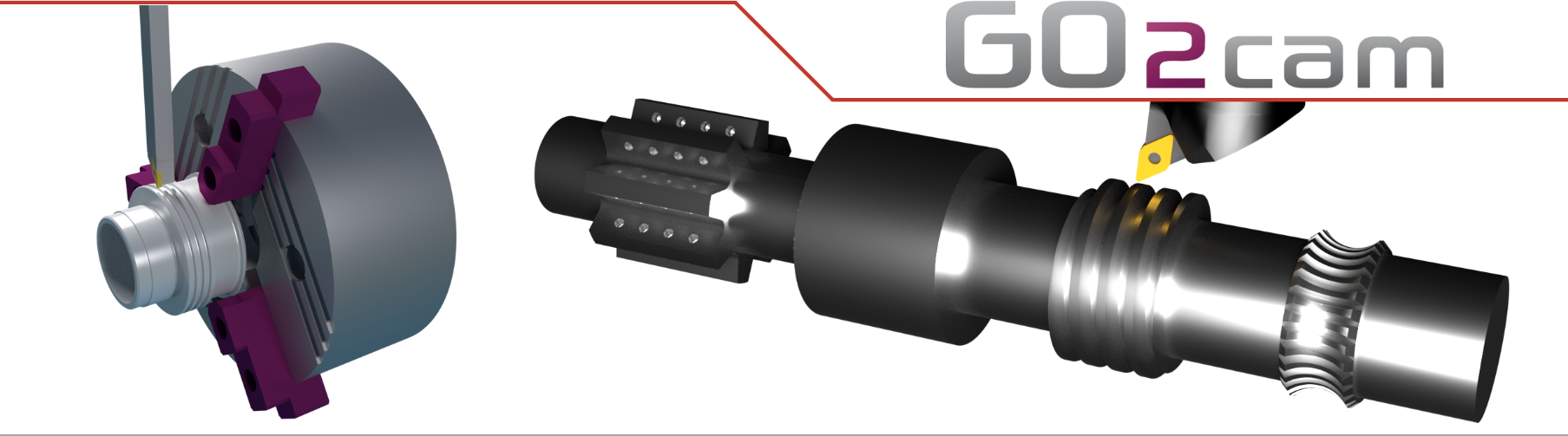
T05

GO2cam V6.10

Tutorial

T05 – TurnMill



T05 – TurnMill

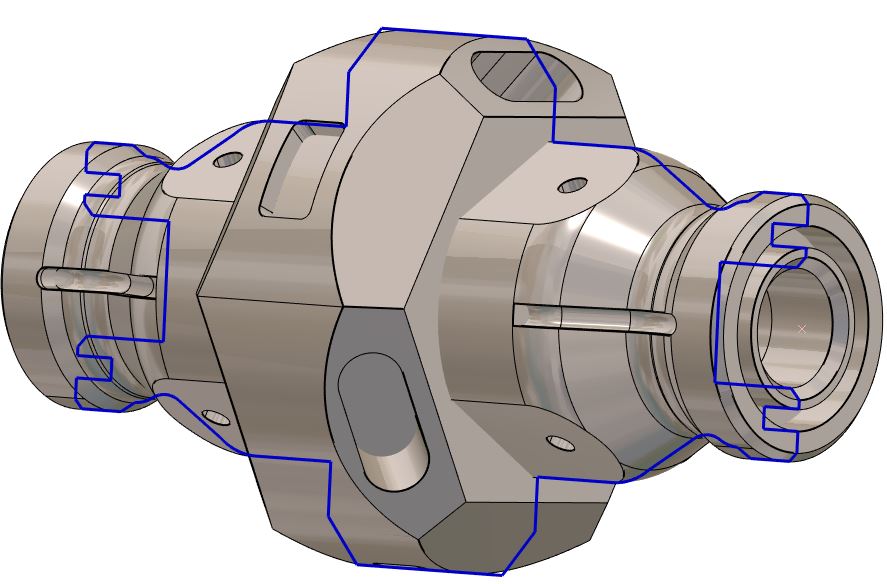
Introduction

|  |  |
| --- | --- |
| * Import of a revolved 3D solid * Use of Opelist for machining * Create axial, radial, and inclined planes | * Creation of developed plane * Projectionn of profiles in the developed plan * Application of turnmill operation |

**Additional Files**

You will find in the file :

* PDF file for this tutorial
* Part files for this tutorial: T05\_TurnMill.PCE
* 3D model in Parasolid file format: T05\_TurnMill.XT
* Opelist file: T05\_Opelist.OPL



### Import and positioning of parts :

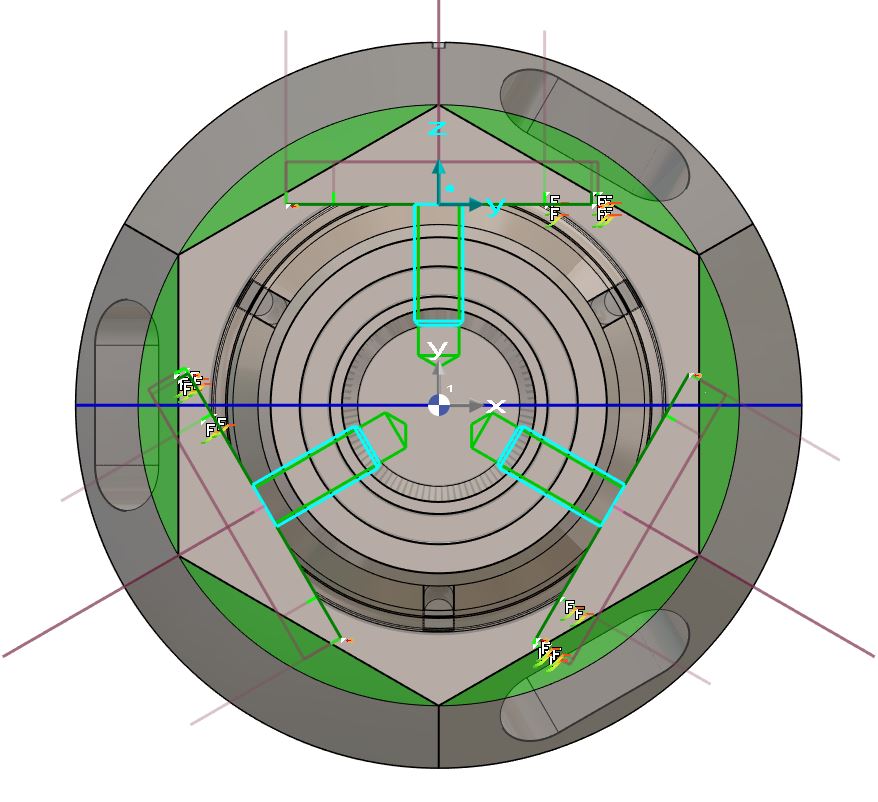
|  |  |
| --- | --- |
| Start :  * In **GO2cam,** left click on the TurnMill module.   The module opens with the Design tab active by default. |  |
| Import of solid part:Enter the **File** menuLeft click on **Graphical user interface, application  Description automatically generated**Select the file **T05\_TurnMill.XT** from your directory and left click on Import. The solid part is imported. |  |
| Positioning : **This command is still active. We must determine whether the part is properly oriented.**  **We will rotate the workpiece 180 ° around the Z axis to move the groove face towards us**   * Left click on **Turn around Z Axis** * In the field, type a value **180** and press Enter * Increase tolerance to 0.005 * Change the silhouette to Symmetrical silhouette. * Left click on  to validate |  |
| Defining the stock: **A cylindrical stock is automatically created for the workpiece with a constant overflow of 5 mm by default.**   * Left click on  and change the value to 120 to define the maximum diameter of the stock. * Left click on  to validate |  |
| Position of origin: By default, the origin is at the end of the part.   * Since the origin is in the correct position for this workpiece, Left click on to validate.   **Note:** Right click on the solid and choose Simplification.  **The part is now ready for machining.** |  |

### Turning operation:

|  |  |
| --- | --- |
| **Operation Menu:**  ****Note: This menu is dedicated to the opelist created by the user. If there is no opelist, this menu will not be displayed**.**  You must copy the file **T05\_Opelist.OPL**, in the opelist directory of the **GO2cam**  Standard path:  C:\GO2cam\_Intl\GO2camV610\opelist |  |
| Loading the opelist:Left click on the Opelists MenuGo to **My List****Note: If there is no other opelist, your opelist will automatically load, else you can click to expand it and find your opelist.**Select the opelist **T05\_Opelist.OPL** The following dialog box opens: If Opelist is customized, you can change the settings. And provide cutting conditions.   * Left click on **Validate** to apply the automatic opelist cycles. |  |
| Creation of the grooving cycle:Profile definition:In the **Turning** menu, Left click on GroovingSelect the geometry Define the tool:Left click on Left click twice on Groove Cutter, to access tool parameters settingsChange the Body Half Taper Angle, Cutter width and thickness values as per the image.In the machining type drop-down menu, select "Face" We will insert a solid holder and define it to forward direction. Left click on the icon  To add a holderLeft click on the menu Groove shank holder, Then replace the Groove shank holder with a Solid Insert HolderLeft click on the icon Select the tool holder provided with the document : **GHFGL25-80-8-AV.SYM**You must have copied this file in the sym directory in: C:\GO2cam\_Intl\GO2camV610\sym Change the orientation value to 0, So that the tool holder is in the forward positionChoice of cycle:Left click on Select the **Direct** cycleChange the Z step value to 2  Calculate | Graphical user interface, application  Description automatically generated |
| Simulation :In order to check our toolpath, we will simulate this:  Left click on SimulationCheck whether the dynamic mode is active, otherwise select this mode.In the display bar for solid on the right side of the screen, you can choose to view the solid with a **120** ° section during simulation :Left click on the blue arrow to expand **Solid Transparency**Select the check box with the section icon  Left click on  To start the simulation in step by step modeLeft click this icon again or press the space bar to advance step by step |  |

### Milling operations:

|  |  |
| --- | --- |
| Creation of a custom view: We will create two custom views, a **Revolution** view and **Reference** view, because the Reference view cannot be accessed in the Turning menu. This is the same for the rotating view which is not available in the Milling menu. Set the workpiece in the Revolution plan while in the Turning menuPress Ctrl+F2 on the keyboard at the same time The Revolution view is now saved as a custom view and assigned to the F2 key. Switch to the Milling menu and the solid will automatically be oriented in the Reference planePress Ctrl+F3 on the keyboard at the same time The Reference view is now saved as a custom view and assigned to the F3 key.  You can use Ctrl+F4 to create a third custom view**.** |  |
| Creation of a radial machining plane : We will create a machining plan using GO2cam's interactive axis system in order to machine the three radial flats faces and their holes. In the Milling menu, Press F2 on the keyboard to switch to the Revolution viewLeft click on the origin of the interactive axis system, hold the left click and move the mouse over the radial face. [You can release your hold on the left click at this point]The radial face is highlighted and the axis is snapped to the center of the face. Left click on the face to have the plane creation bar open.In the plane name field, type the Radial plane  Left click on  to move the plane origin to the center of the holePress and hold Ctrl+Shift key simultaneously and place the cursor on the edge of the hole, and the center point will be highlighted, left click to validate The axis is now in the center of the hole surface. To complete the axis must be oriented at 90°. In the orientation field, enter a value of 90  * Left click on  to validate   The Radial plane is now created. We will proceed to machine this face. |  |
| Display of axes: We have just created a machining plane.  In GO2cam, you can display the axis of the origin of the created plane. To do this, you must activate a filter. Go to the filter pageIn the list, Left click on ‘Elements and Colors’    * Select the "workplanes systems of axis" check box   Note:  To display the marks, you have two options:   * Show all marks * Show marks in the current plan |  |
| Machining on this plane:Machining of the face:Enter Left click on The selection mode is automatic by defaultLeft click on the **bottom face**, Start and end height is acquired from solid and stockLeft click on Choose **Flat End Mill**, then select the tool **"Flat End Mill D16"** in **Standard**  Left click on Choose Pocket+ContourLeft click on Drilling Operation - M8 :  * Enter in **Hole/Feature**, select **Manual Holes**    Left click on , And select the holeLeft click on , The information page is displayed in the lower right corner, showing the drilling depth and diameter. **Note: On the part, two entities of holes are visible, one corresponding to the drilled hole and the other to the tapped hole, with different depths. The drilled hole goes 5mm deeper than the tapped hole, indicating the depth.**   * Check if you are on the X+(radial) and C axis    Left click on Choose Drill, then select the tool **"Drill D6.75"** in **Standard**Left click on Choose **Drilling**, then select **Simple mode Auto**In the strategy, change the retract value to - 5 and Depth calculation to DiameterLeft click on Tapping operation of M8 :Left click on , then on the command **Wizard**   * In the machining tree, expand the operation Drilling, then Left click on , to copy the selection  Left click on Choose Tap, Download the standard library and select Tools **"TapM08x1.25"**Left click on Choose tapping, then select **Tapping mode Auto**Left click on | Text  Description automatically generated with low confidence      Graphical user interface, text, application, website  Description automatically generated |
| Copy operation by C rotation:  * In the machining tree, use Ctrl to select three cycles applied on the face * Right click in the machining tree, then Left click on Operation Copy * In the drop-down menu select C Rot, type 2 in the number field and 120 in the angle field      * To complete the operation, Left click on one of the three cycles in the machining axis   The operation copy is located below the original cycle.  You can start the simulation  to check your toolpath. | Text  Description automatically generated |



|  |  |
| --- | --- |
| Use the C-axis to machine on inclined planes : We will use the C-axis to machine all inclined planes and chamfer them at the same time. Change to the reference planeEnter manualLeft click on Select the edge of the hexagon, as shown in the figure below, and then select the edge path to be machinedCheck the Bottom Z box**,** then type - 92 in the field  * Set Axis to **C**  Left click on Select Chamfering Mill, download Otelo library. In the selection page, select tool **"16 120 943"**, then change the cut diameter to 35 and the mini diameter to 5.Left click on  , choose ChamferingLeft click twice on the Chamfering icon to enter the Strategy.In the strategy, the cycle must be on the left, the Z steps is 15, and the toolpath following slope must be "Yes"In "Movement", type 20 in the lead in and lead on arc radius fields; And the angle of lead in/out must be 0Left click on | Logo  Description automatically generated |
| Pocket on inclined plane :We will create an inclined plane in the same way as the radial plane with the interactive system of axes.Left click on the origin of the system of axes. While holding the left click, move the mouse over to inclined face, then click to select the face.The inclined plane is highlighted, the axis system is in the center of the face, and the plane creation bar is opened.In the plane name field, type “Inclined plane”In the orientation field, enter a value of 90    * Left click on  to validate |  |
| Pocket machining :  Select Left click on The selection mode is automatic by default.Left click on the **bottom face** of the pocket. The top and bottom height is acquired from the 3D part and stock.Left click on , the information page is displayed in the bottom right corner, which contains the value of the minimum radius to help you select the toolLeft click on Choose **Flat End Mill**, then select tool **"Flat End Mill D10"** from **Standard**Left click on Choose pocketSet the value of Z step to 2.5 and the value of allowance to 0Left click on The cycle is generated, if the Z plunge is not helix. You can change this parameter.Left click on  to expand the pocket operationRight click on **Strategy**, then left mouse click on **app/ret edition**Change Z plunge to Helix if required.Left click on  to recalculate the toolpath We will copy the operation in rotation C. Right click on the **pocket** cycle**,** then Left click on **operation copy**  * In the drop-down menu of C Rot, type 2 in the number field and 120 in the angle field      * Left click on the pocket cycle in the machining tree   The operation copy is applied below the original cycle.  You can start the simulation  to check the toolpath. | Graphical user interface, text, application  Description automatically generated      Logo  Description automatically generated      A picture containing graphical user interface  Description automatically generated |
| Pocket machining on developed plane: We will machine the pockets located on the largest diameter. It can be observed that the bottoms of these pockets follow the cylinder. It will be necessary to create a Developed plane in order to be able to machine these shapes. Creation of the developed plane :Left click on , then in the drop-down menu of the command bar, select "Measure on Solid" to measure the radius of the bottom of the pocket  Right click on the screen, select Topological InformationLeft click on bottom of pocket, value reads Radius 55mm This value will help us to create a plane developed directly on the curved bottom surface. Enter Design menu, click on WorkplanesLeft click on **Developed Plane**Left click on **icon**   The command bar appears. Before entering the diameter value, you must enter the parameter first.  The parameters for the developed plane are:   * Orientation **: Outside** * Ordinate Unit : **Mm,inch** * Ordinate Direction : **Top** * Name of plane : **Plane Dev1** * Enter a value of 110 in the diameter field and click on the green tick to confirm.   The development plane has now been created, and GO2cam will automatically put you in it. We will be able to realize projection in this plane. | Diagram  Description automatically generated with low confidence |
| Projection on developed plane :  * Still in the Workplanes menu, go to the reference plane to view the part * Left click on   **Note: When only one plane is available, the projected flat plane will be automatically selected**   * Left click on **Creation of an edge path on the solid** * Left click on an edge at the bottom of the pocket to get the continuity of the bottom edge, left click on Profile OK   A profile covers the edge path to be projected onto your developed plane. Now, the same step needs to be taken for the other two pockets before validating the function.   * After selecting three profiles, you can validate the command by clicking  Return in **Plane Dev1**, To check whether the geometry is correctly projected A picture containing diagram  Description automatically generated |  |
| Application of pocket on developed plane :In **Milling tab**, check whether the dev1 plane is still activeSelect Left click on Select the first profile on the developed plane.Enter a value of 5 in the Top Z box and 0 for Bottom Z, and then click the next two profiles. Ensure the values for Top and Bottom Z are similar.Left click on Choose Flat End Mill, then select tool **"Flat End Mill D04"**Left click on Choose Pocket, double click on **Pocket** to access the cycle parametersIn strategy, modify the below parameters:**Z step** to **2,5****Stepover value** to **0.50****XY and Z Stock Allowances** to **0**  * In page **movement**, modify:  **Rapid Plane** to **20****Approach and Return in Z to** Rapid PlaneLeft click on  You can start simulation  to check the toolpath. | Logo  Description automatically generated    Graphical user interface, application  Description automatically generated |
| Part return :Enter the **Turning** menuLeft click on the **Part** command and select part ManipulationLeft click on the function **Part return** **Note: In the command bar, the translation value will be automatically filled in the field.**Left click on   * At this point, if you have had a machine selected for the operation, you will be required to set the stock in the chuck and validate. [ In this case you will not encounter this] * Now, you can repeat and define all the cycles on this side, similar to those defined from T05\_Opelist to Pocket\_1 |  |