PROJETO E MANUFATURA ASSISTIDOS POR COMPUTADOR
27260 A

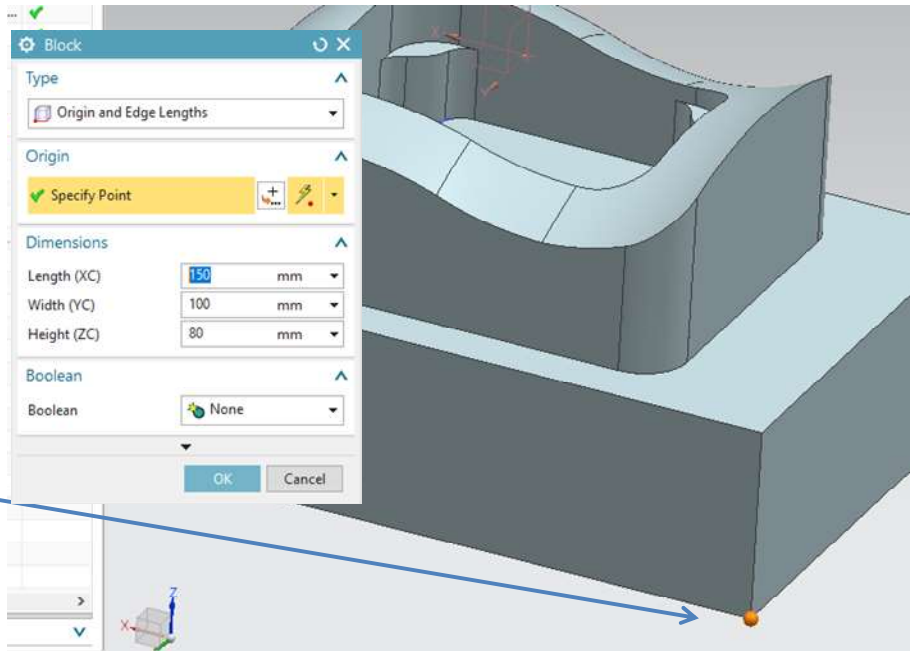
AULA 12 – MANUFACTURING CAD/CAM
LAB. 17

Departamento de Computação
Prof. Kelen Cristiane Teixeira Vivaldini

Departamento de Computação

Aula 12

- Open the file Die_cavity.prt of the exercise – Aula 12_Lab. 16
- **Insert** a **block** with the following dimensions and positioning
  - Length = 150 mm
  - Width = 100 mm
  - Height = 80 mm
- In the Point Constructor icon located on the toolbar choose the lower most edge of the base block, so that the new block created wraps up the whole previous model as shown.

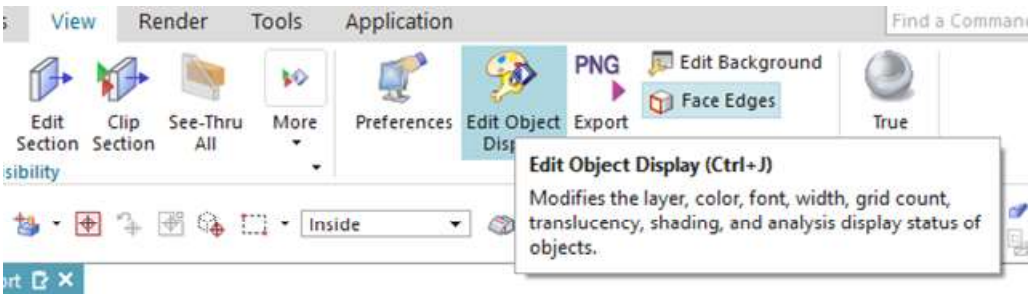


Departamento de Computação

Aula 12

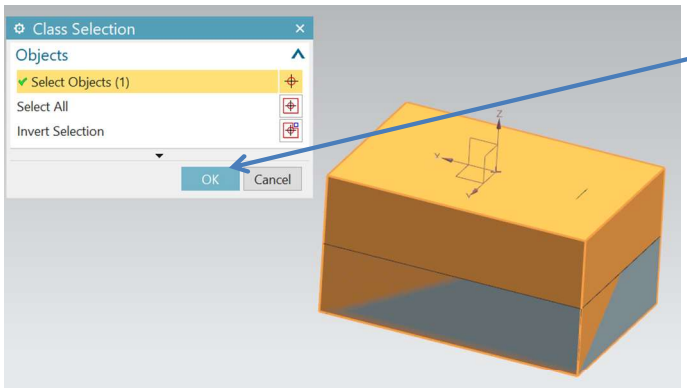
This block encloses the entire design part so we will change the display properties of the block

- **Click** on View tab -> Edit Object Display

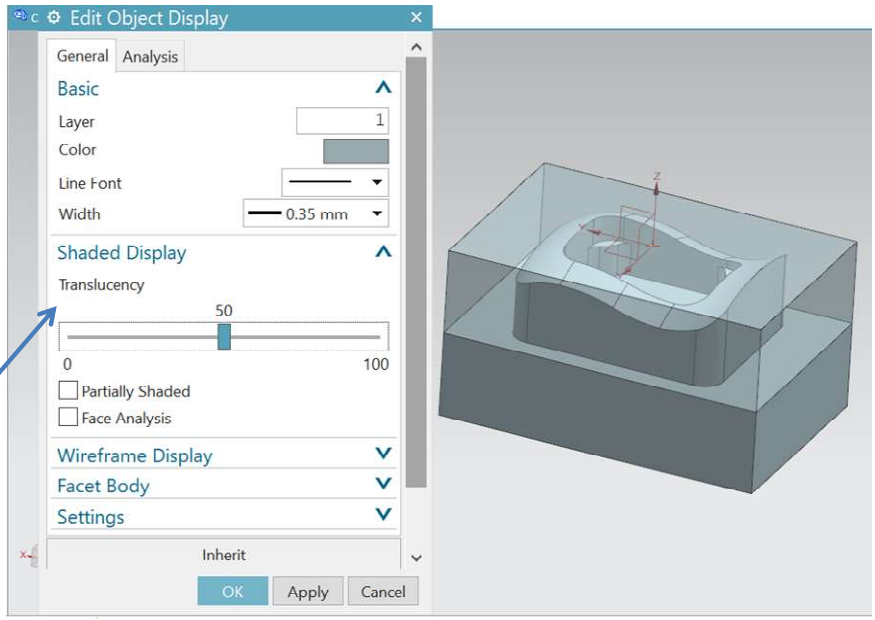


Aula 12

- Select the block you created and click **OK**



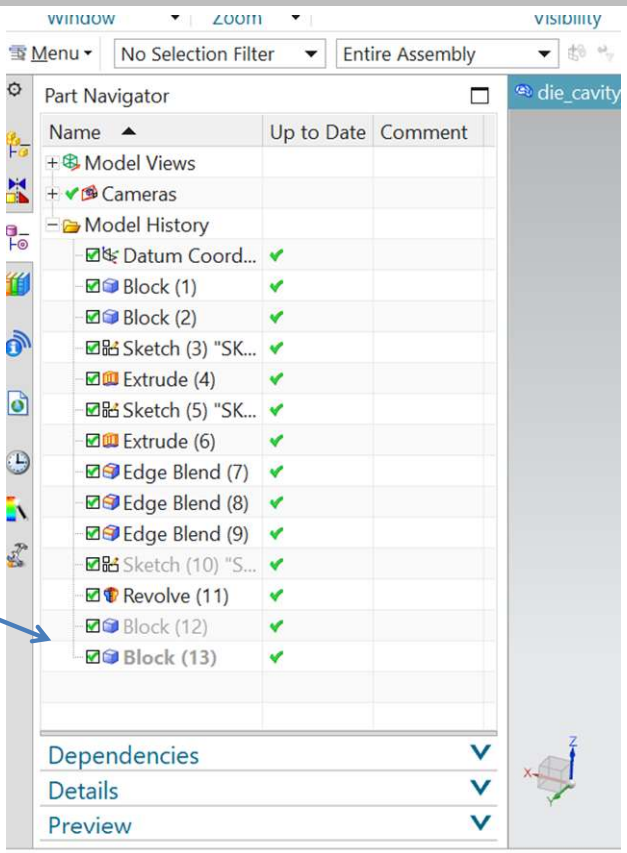
- When the window pops-up, change the display Color and change the Translucency to 50
- Click **OK**



Aula 12

- Hide the block you just created by right clicking on the block in the Part Navigator. This will make the raw block disappear from the environment. Whenever you want to view or work on this solid, reverse the blanks.

- This is done by pressing <Ctrl> + <Shift> + B.

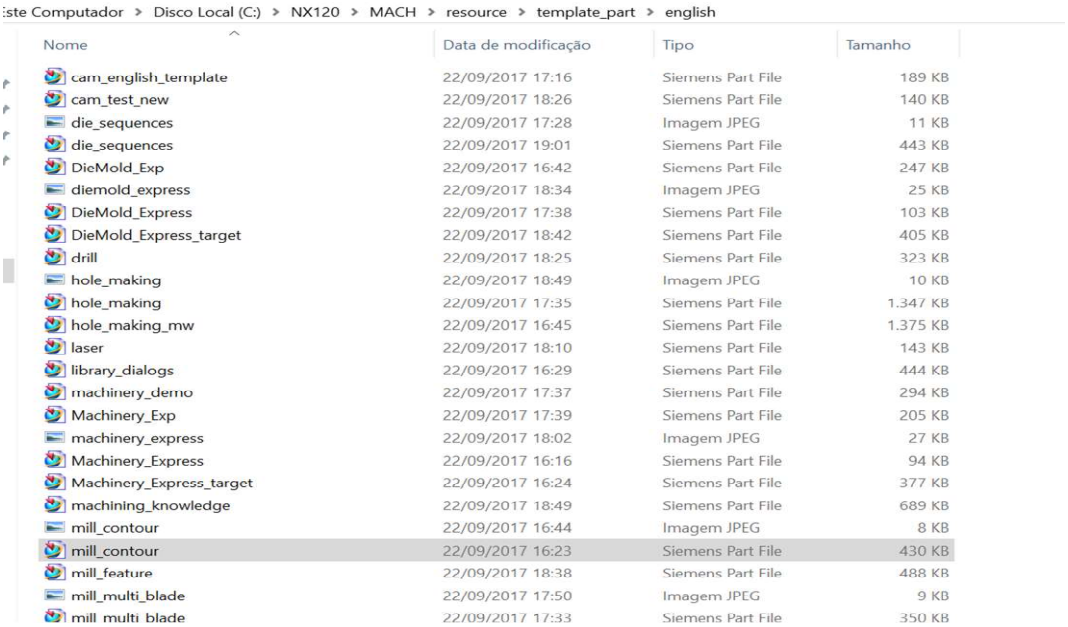


Departamento de Computação

Aula 12

Mysterious

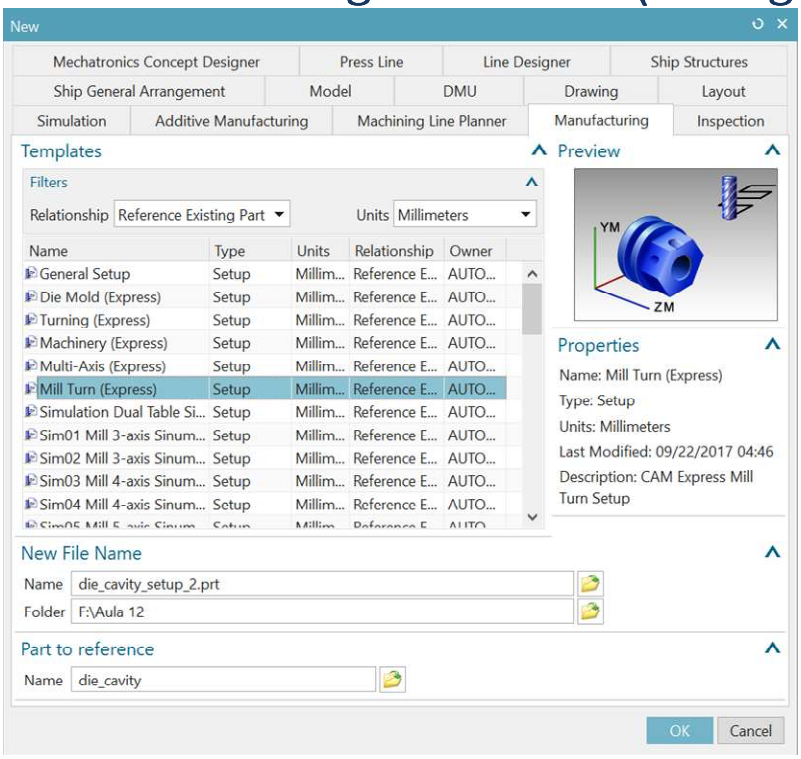
- Open the file mill-countour in the directory below.



Aula 12

Manufacturing module

- Select File →New →Manufacturing →Mill Turn (milling operation.

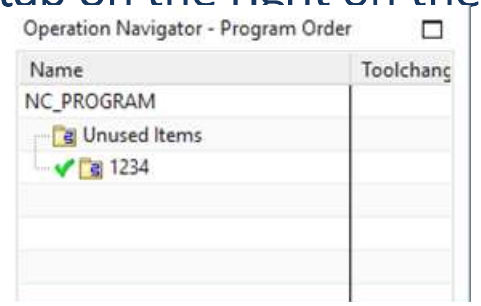


Departamento de Computação

Aula 12

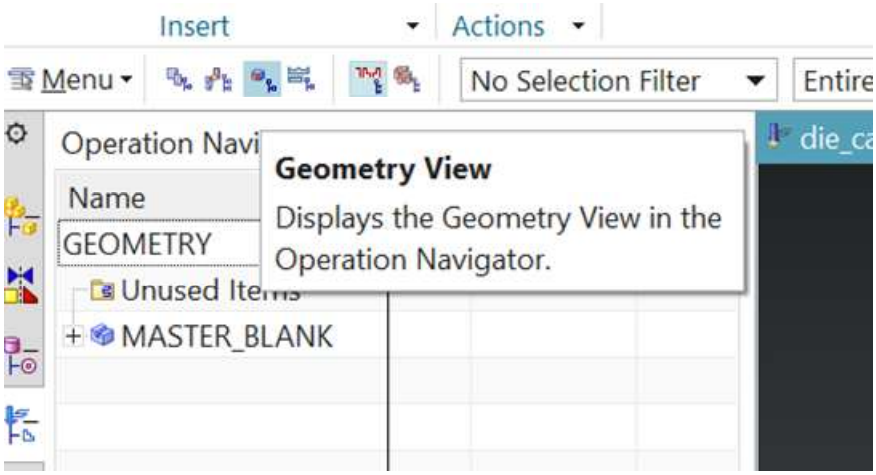
Operation Navigator

- Click on the **Operation Navigator** tab on the right on the **Resource Bar**



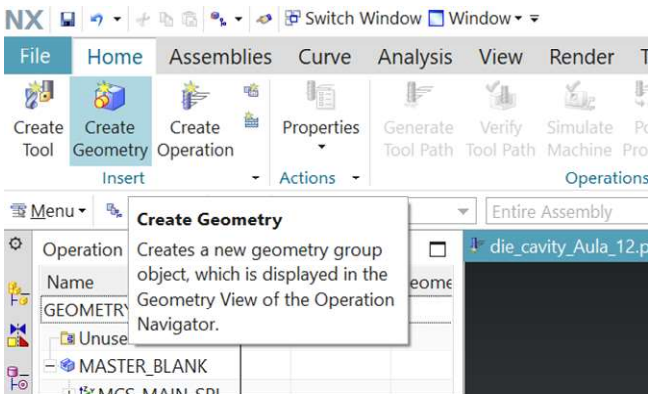
The Operation Navigator gives information about the programs created and corresponding information about the cutters, methods, and strategies. The list of programs can be viewed in different categorical lists. There are four ways of viewing the list of programs in the Operation Navigator. The four views are Program Order view, Machine Tool view, Geometry view and Machining Method view.

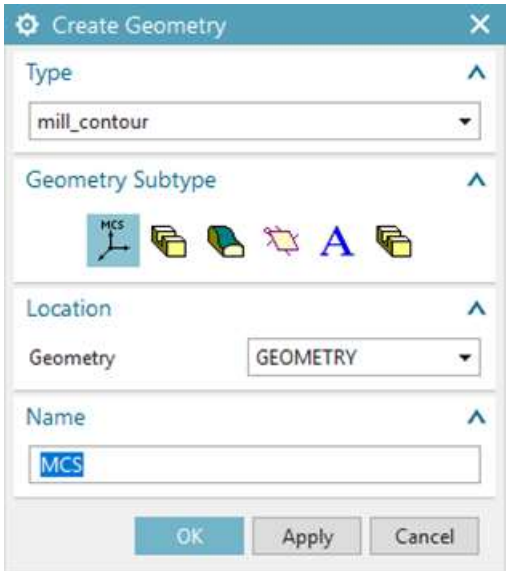
- Click on **Geometry View**



Machine Coordinate System (MCS)

- Click on **Home** tab -> **Create Geometry** icon in the **Insert** group to initiate setup for programming.

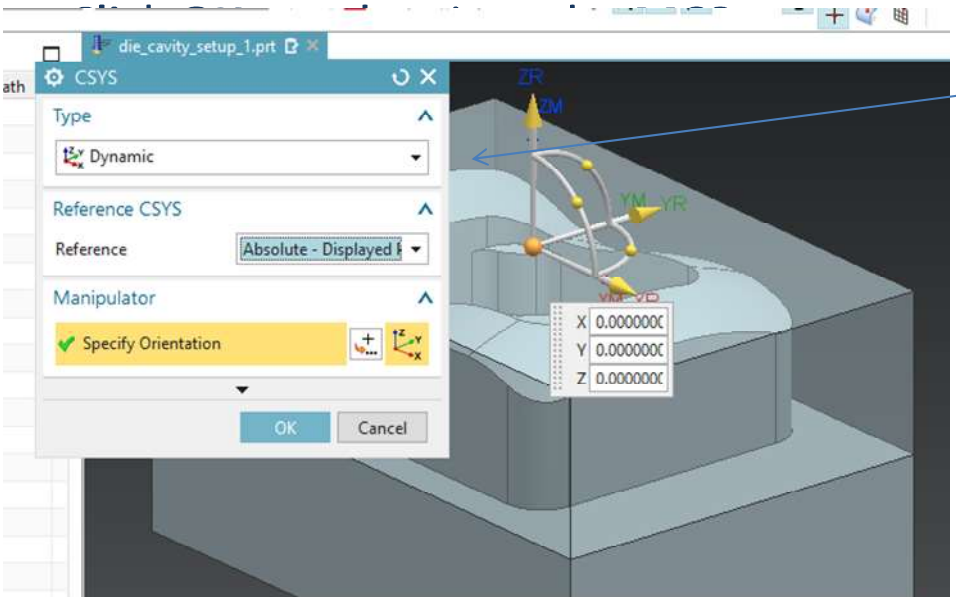


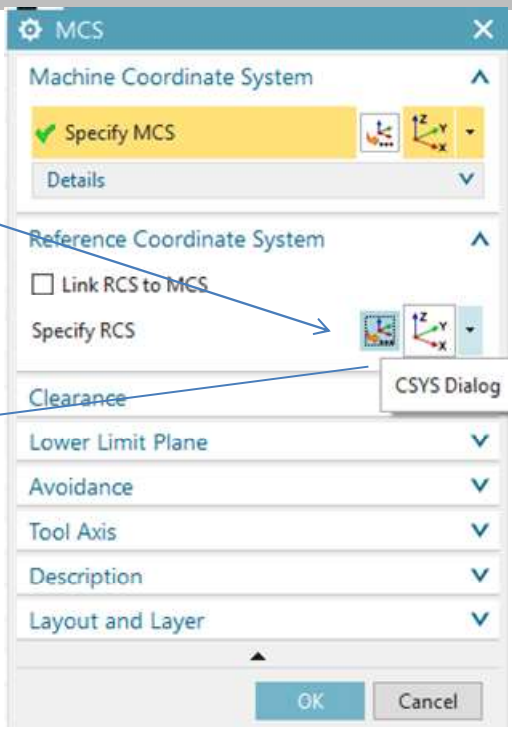


Departamento de Computação

Aula 12

- Click on the **CSYS Dialog** button in **Specify MCS**. This will highlight the default WCS of the part and assign it as the **MCS**



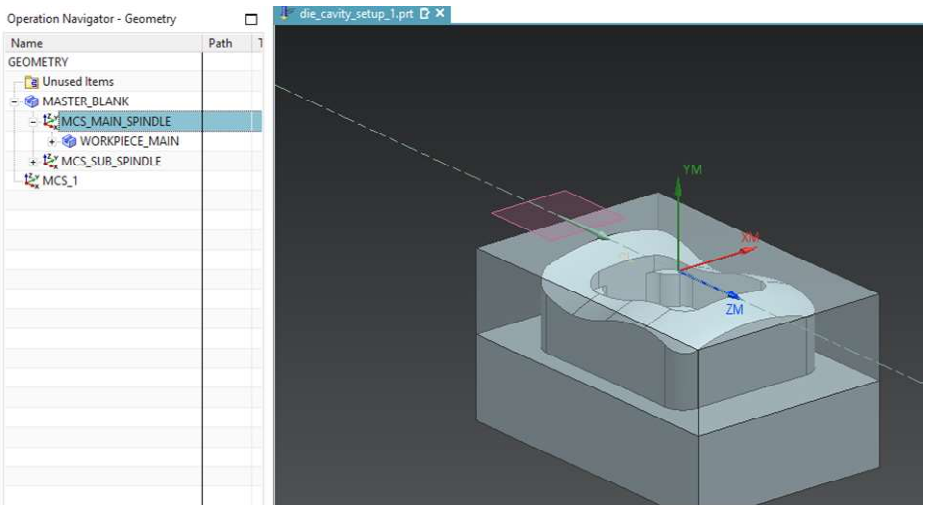


Departamento de Computação

Aula 12

Geometry Definition

- Click on **Geometry View**
- Expand **MCS_MAIN_SPINDLE** by clicking on the plus signs in the **Operation Navigator**

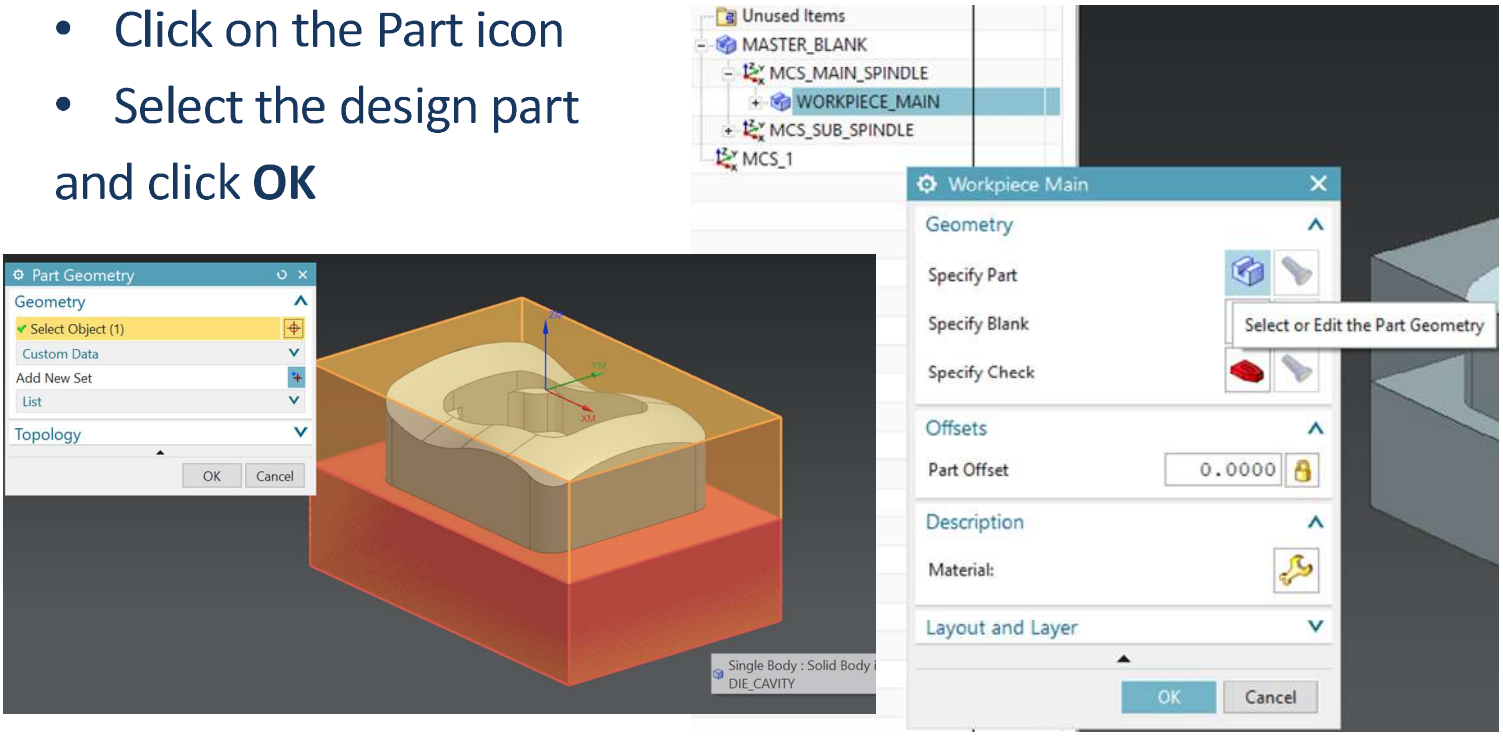


Departamento de Computação

Aula 12

- Double-click on WORKPIECE_MAIN
- Click on the Part icon
- Select the design part

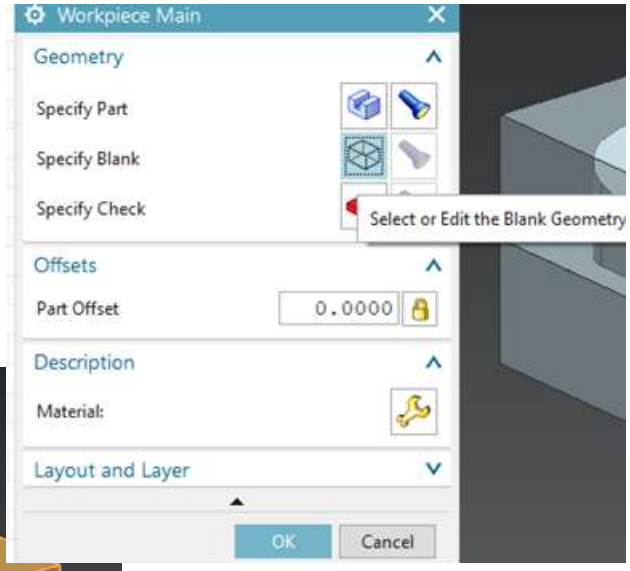
and click **OK**

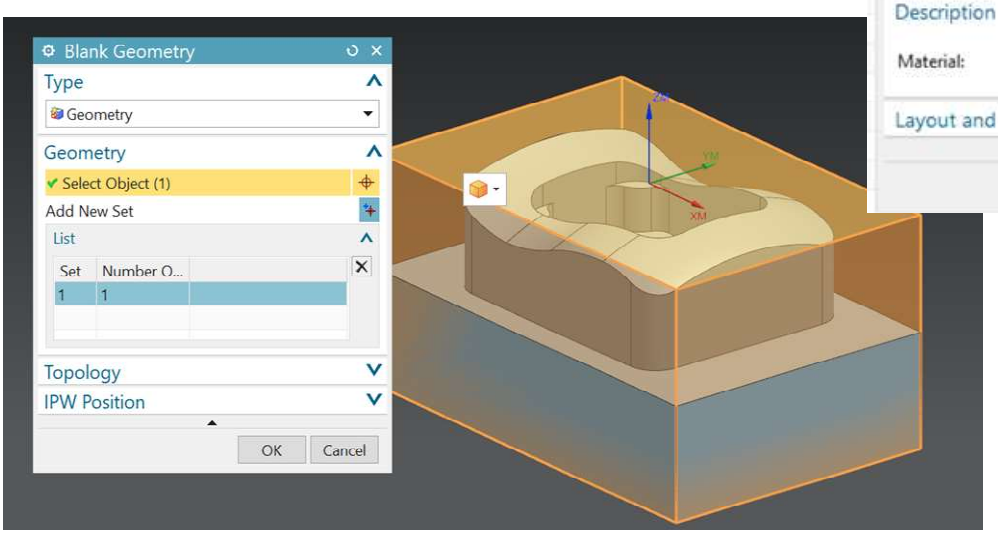


Departamento de Computação

Aula 12

- Click the **Blank** icon
- Click on the **Block** and press OK
- Click OK to exit the **Workpiece Main** window





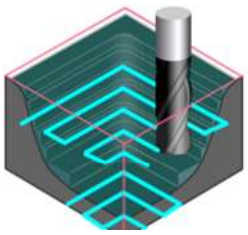
CREATING OPERATION - Programming Strategies

- Click on the Create Operation

Departamento de Computação

Aula 12

- In Operation Subtype, click on the Cavity_Mill as shown in the figure

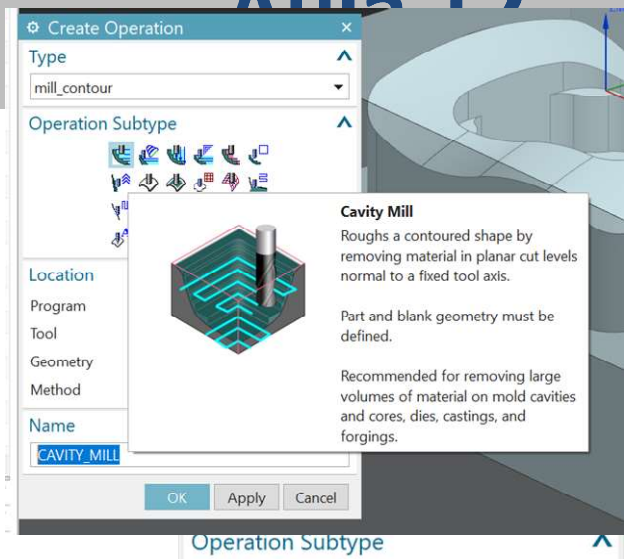


Cavity Mill

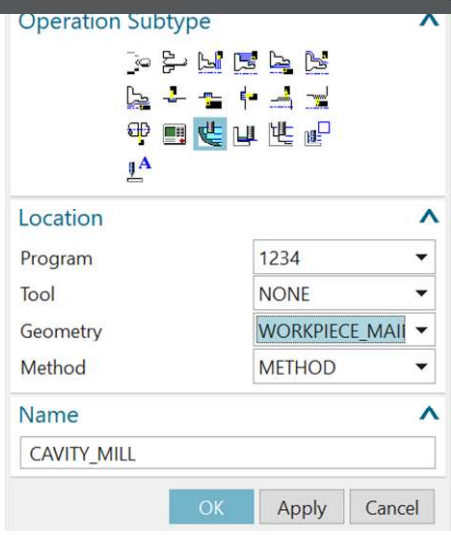
Roughs a contoured shape by removing material in planar cut levels normal to a fixed tool axis.

Part and blank geometry must be defined.

Recommended for removing large volumes of material on mold cavities and cores, dies, castings, and forgings.

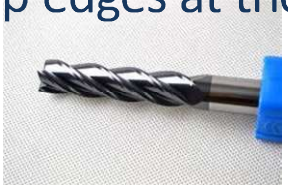


- **Change the Program from NC_PROGRAM to 1234**
- Change the Geometry to **WORKPIECE_MAIN**
- Click OK



Tool Creation and Selection

- **Flat End Mill Cutters**: These cutters have a sharp tip at the end of the cutter and are used for finishing parts that have flat vertical walls with sharp edges at the intersection of the floors and walls.



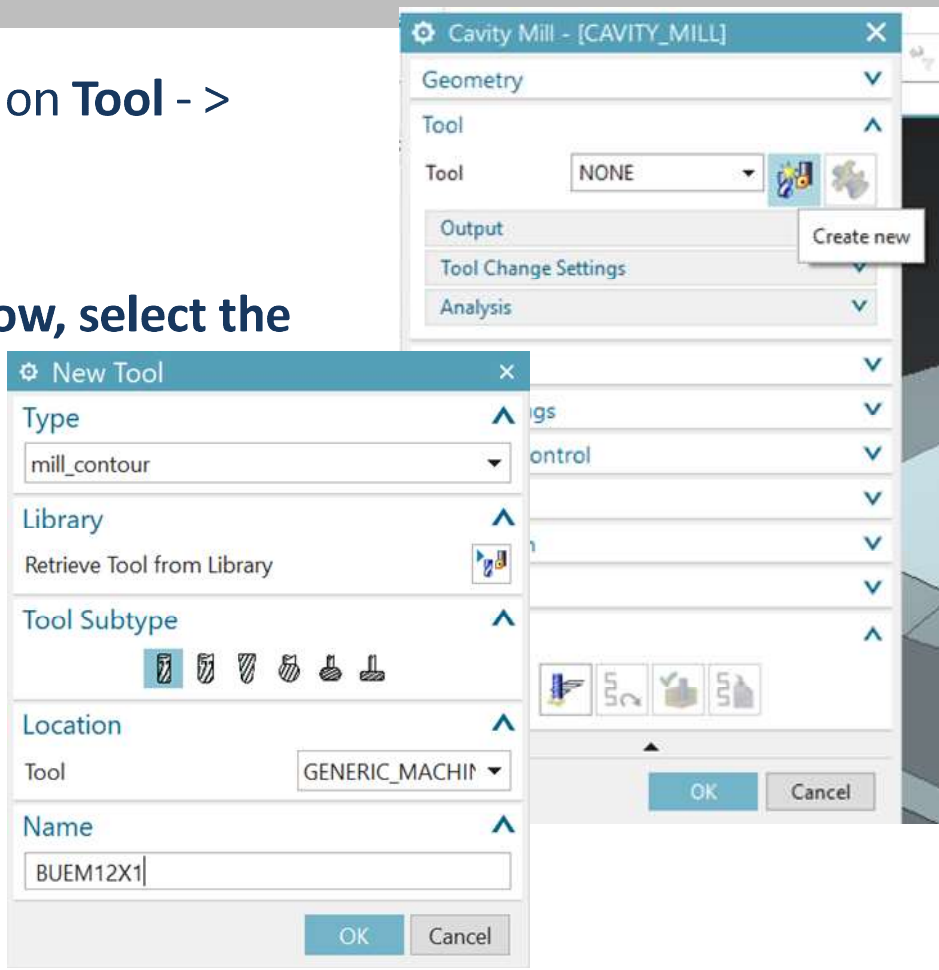
- **Ball End Mill**: These cutters have the corner radii exactly equal to half the diameter of the shank. This forms the ball shaped profile at the end. These cutters are used for roughing and finishing operations of parts or surfaces with freeform features.

- **Bull Nose Cutters:** These cutters have small corner radii and are widely used for roughing and/or semi-finishing the parts as well as for finishing of inclined and tapered walls.

The cutter that we are going to use to rough out this huge volume is BUEM12X1 (Bullnose End Mill with 12 diameter and 1 corner radius).

Aula 12

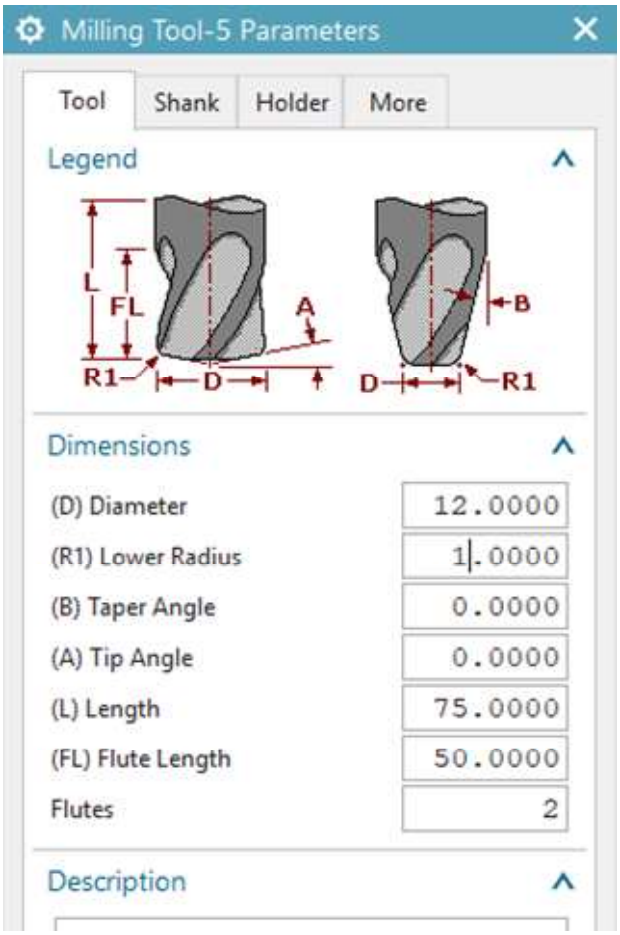
- In the **Cavity Mill**, click on **Tool** - > **Create New**
- Click **New**
- On the **New Tool window, select the Mill icon**
- Type in **BUEM12X1** as the **Name** and click **OK**



Aula 12

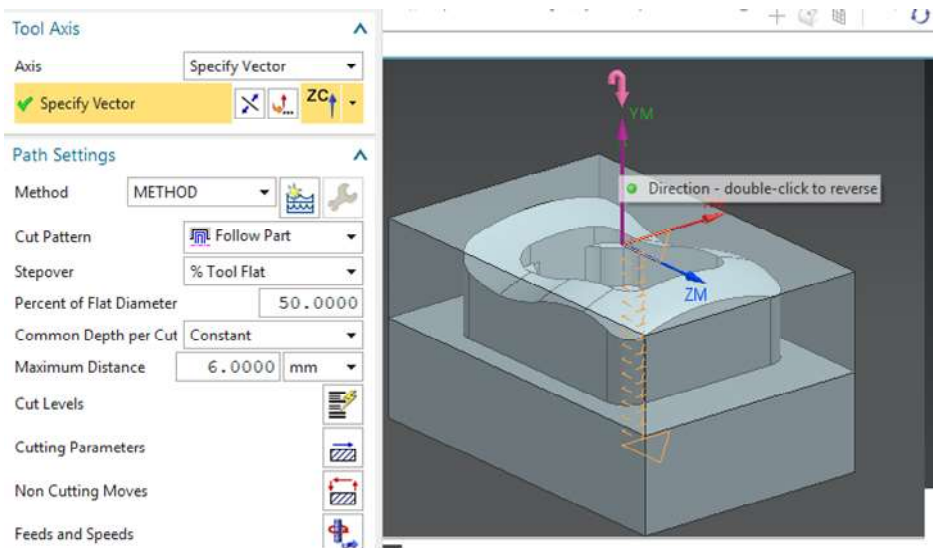
This will open another window to enter the cutter dimensions and parameters. You can also customize the list of tools that you would normally use and call the cutters from the lib

- Enter the values as indicated in **Dimensions**
- Click OK.



Tool Axis

- Make sure that the Tool Axis is perpendicular to the top surface on the block.
- Click on **Tool Axis** and choose **Specify Vector**
- Select the appropriate axis as shown



Path Settings

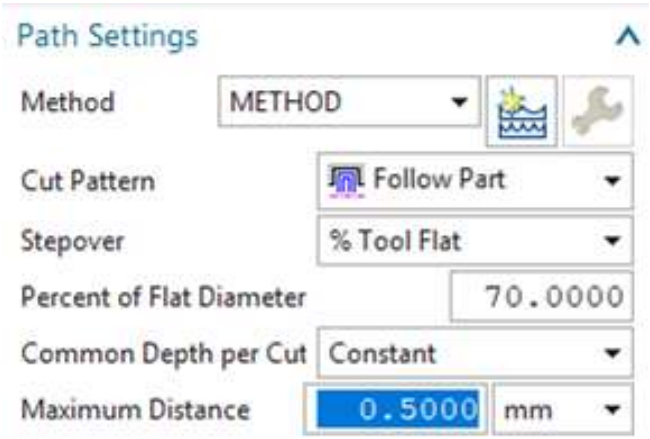
- In the **Cavity Mill** menu click on the **Path Settings**

Aula 12

- **Follow Part:** This is the most optimal strategy where the tool path is manipulated depending on the part geometry. If there are cores and cavities in the part, the computer intelligently considers them to remove the materials in an optimal way. This is widely used for roughing operations.
- **Follow Periphery:** This takes the path depending upon the periphery profile. For example, the outer periphery of our part is rectangular. So the tool path will be generated such that it gradually cuts the material from outside to inside with the Stepover value. This option is mostly used for projections and cores rather than cavities.
- **Profile**: This takes the cut only along the profile of the part geometry. It is used for semi-finishing or finishing operations.

- **Trochoidal**: This cutter is huge and is used for removing a large amount of material. The bulk of material is removed by gradual trochoidal movements. The depth of cut used will be very high for this strategy.
- **Zig**: This takes a linear path in only one direction of flow.
- **Zig Zag**: This tool takes a zigzag path at every level of depth. It saves time by reducing amount of air cutting time (idle running). The climb and conventional cuts alternate.
- **Zig with Contour**: This takes the path in one direction either climb or conventional. The unique thing is that it moves along the contour shape nonlinearly.

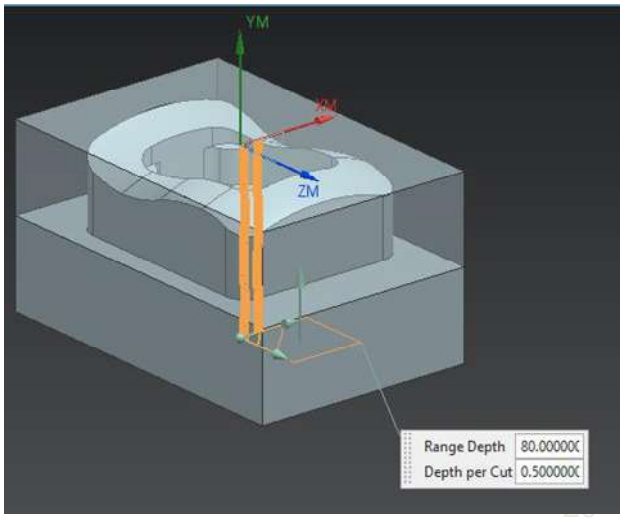
- Configure as the figure bellow:

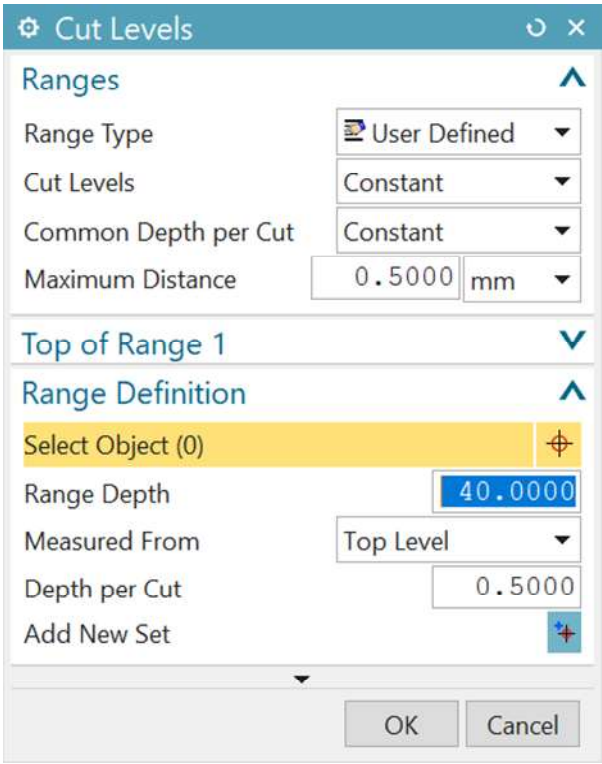


Departamento de Computação

Aula 12

- Click on **Cut Levels**
- Change the **Range Type** to **User Defined**
- Change the **Range Depth** to **80**
- Select **OK**



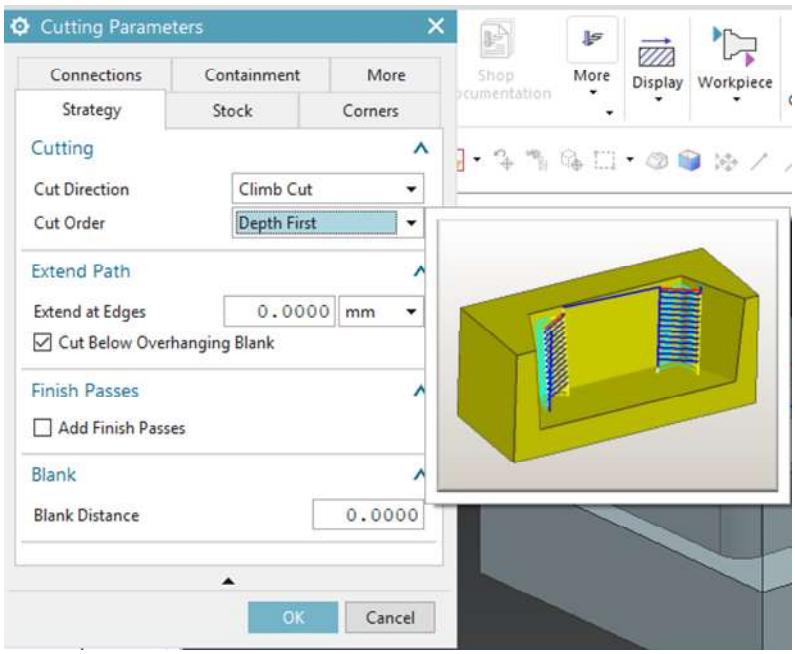


Departamento de Computação

Aula 12

Cutting Parameters

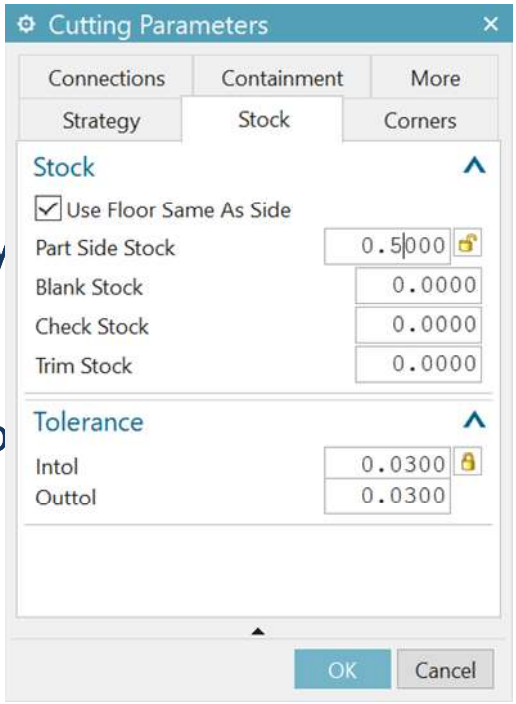
- On the **Path Settings** menu, click **Cutting Parameters**
- Under the **Strategy** tab button, change the **Cut Order** from **Level First** to **Depth First**



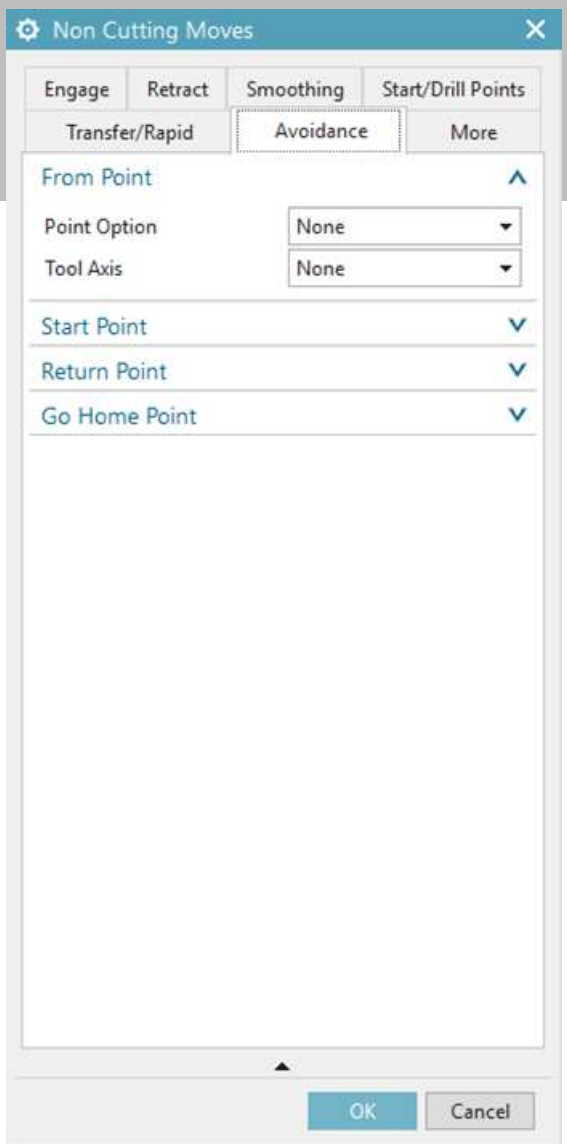
Departamento de Computação

Aula 12

- Click on the **Stock** tab
- Change the value of the **Part Side Stock** to **0.5**
- This value is the allowance given to every side of the part. If you want to give different values to the floors (or the flat horizontal faces) uncheck the box next to
- Use Floor Same As Side and enter a different value for Part Floor Stock.
- Click **OK**



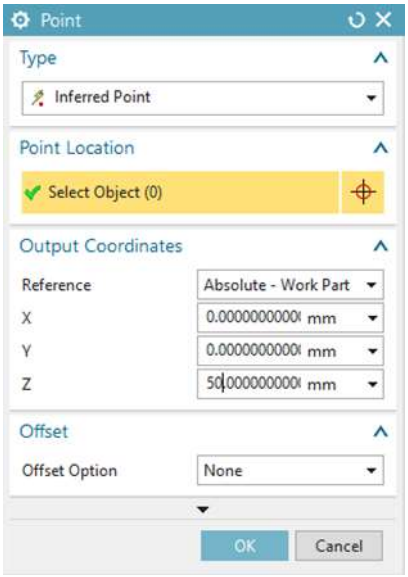
- Click the **Non Cutting Moves**
- Click the **Avoidance** tab



From Point

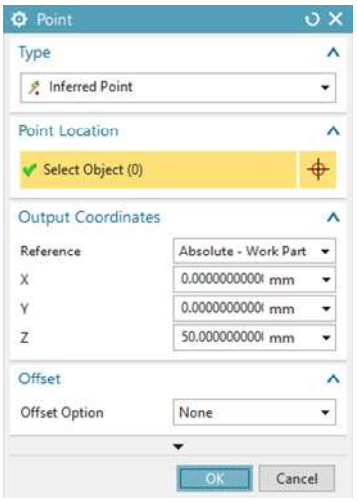
- This is the point at which the tool change command will be carried out. The value is normally 50 or 100 mm above the Z=0 level to enhance the safety of the job when the cutter is changed by the Automatic Tool Changer (ATC).
- Click **From Point**
- Choose **Specify** in the Point Option field
- In the **Point Dialog**, enter the coordinates

of XC, YC and ZC as **(0, 0, 50)**

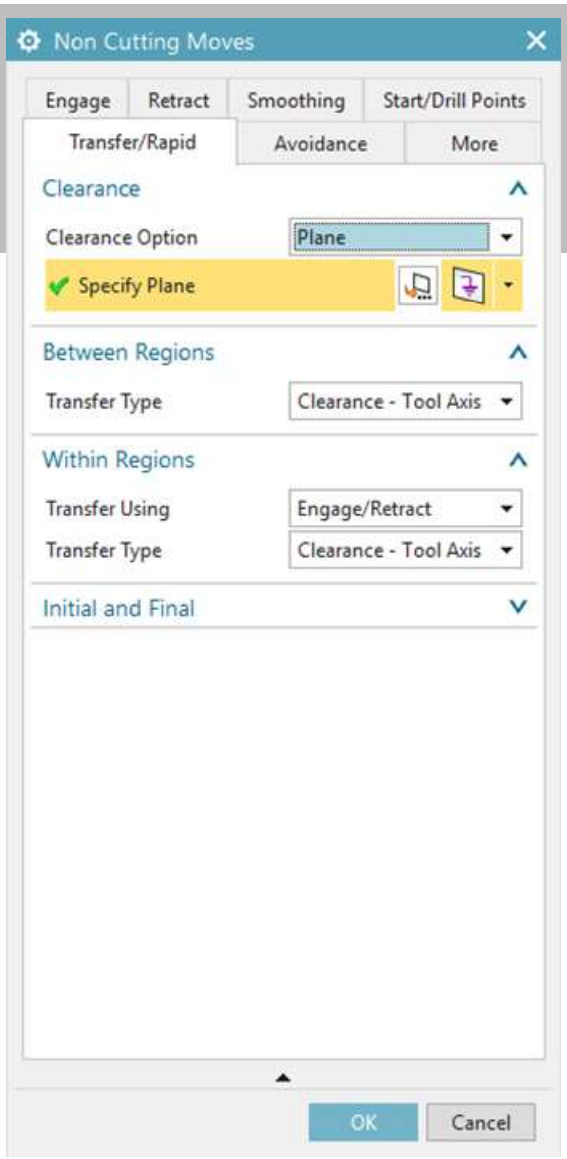


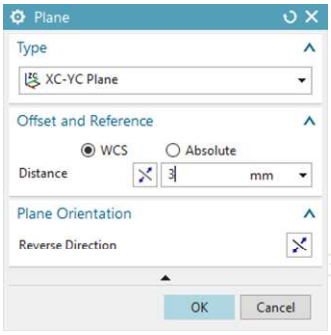
Start Point

- This is the point at which the program starts and ends. This value is also 50 or 100 mm above the Z=0 level to enhance safety. It is also the point at which the machine operator checks the height of the tool mounted on the spindle with respect to the Z=0 level from the job. This cross checks the tool offset entered in the machine.
- Click on **Start Point**
- Choose **Specify**
- Enter the coordinates **(0, 0, 50)**
- Click **OK**



- Click on the **Transfer/Rapid** tab
- **Clearance Option: Plane**
- **Specify Plane**: Choose the XC-YC Plane from the drop-down menu in Type tab
- Under the Offset and Reference tab enter the value as 3 as the Distance
- Click OK twice to go back to the Cavity Millparameters window



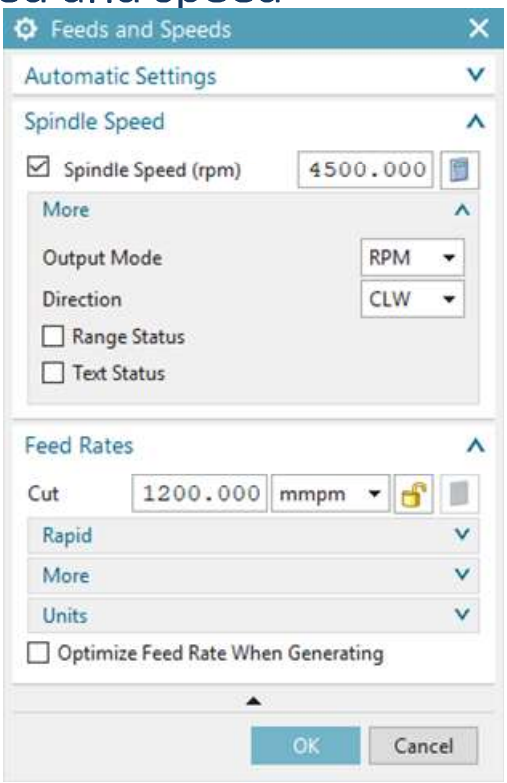


Departamento de Computação

Aula 12

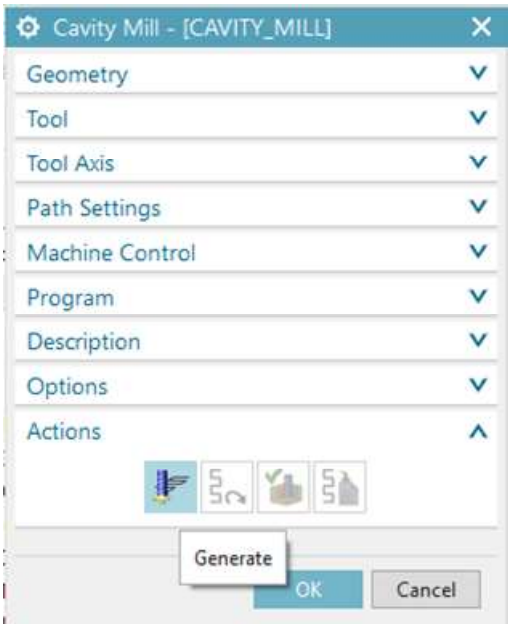
Speeds and Feeds

- Choose **Feeds** and **Speeds** to enter the feed and speed parameters
- Enter the **Spindle Speed** value as **4500 rpm**
- Enter the **Cut** value as **1200 mmpm**
- Click **OK**



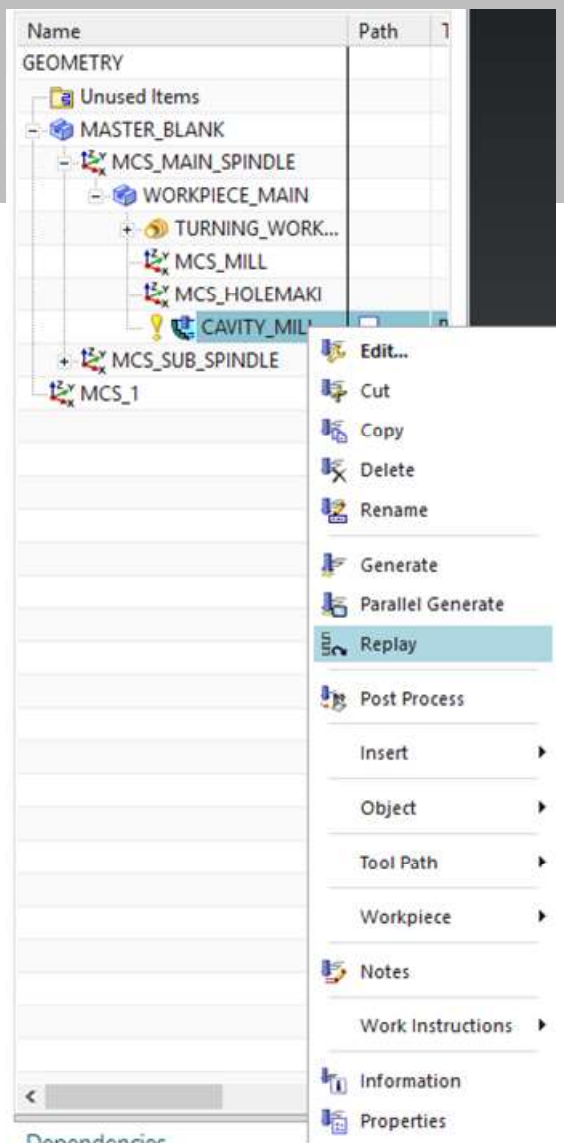
Generating Program

- Click on the Generate icon at the bottom of the window



Tool Path Display

- Whenever you want to view the entire tool-path of the program, right-click on the program in Operation Navigator and click Replay.

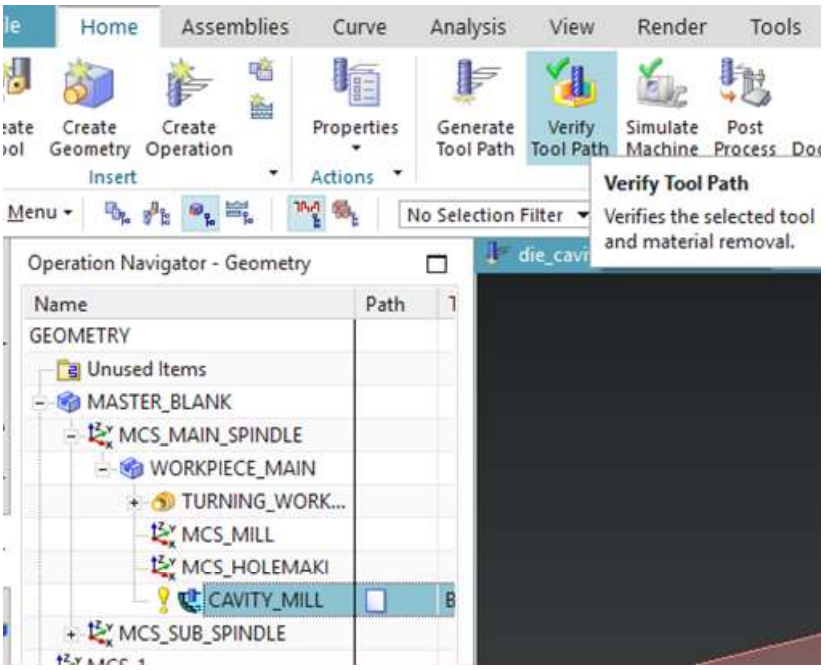


Departamento de Computação

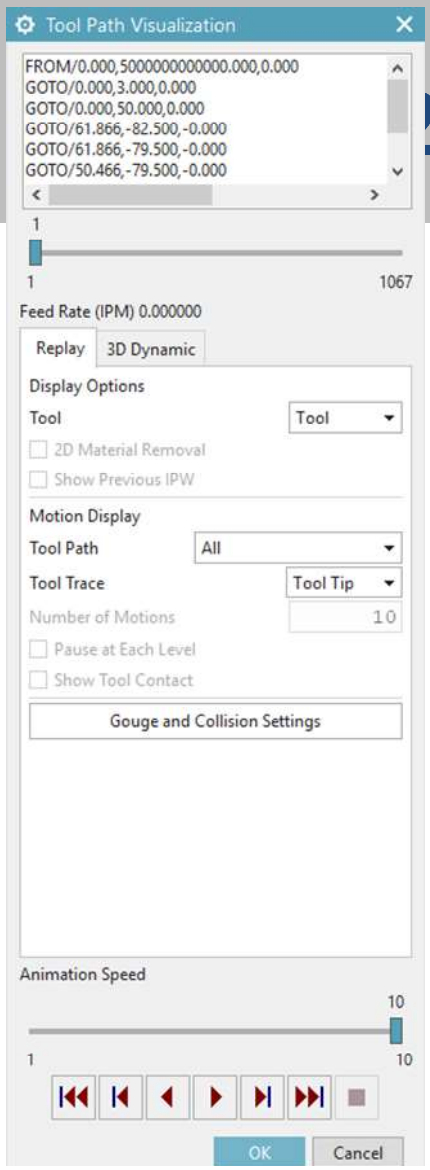
Aula 12

Tool Path Simulation

- Right-click on the program in the Operation Navigator and choose **Tool Path →Verify**



- On the Tool Path Visualization window, click on the Play icon to view the motion.

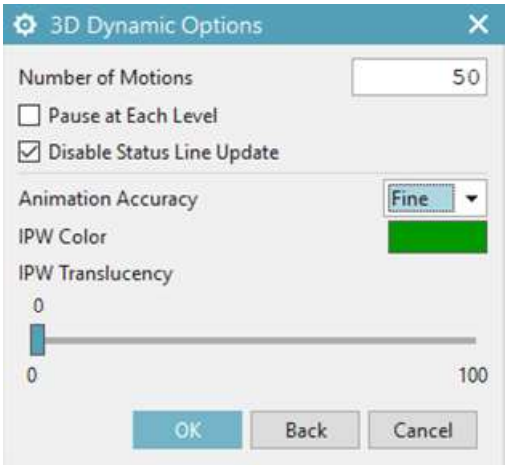


Departamento de Computação

Aula 12

- Click on **the 3D Dynamic** tab on the same window
- Click on the **Display Options** button on the same window
- Change **the Number of Motions** to **50**
- Change the **Animation Accuracy** to **Fine**
- Change the **IPW Color** to **Green**
- Click **OK**

- Click on the **Play** button again



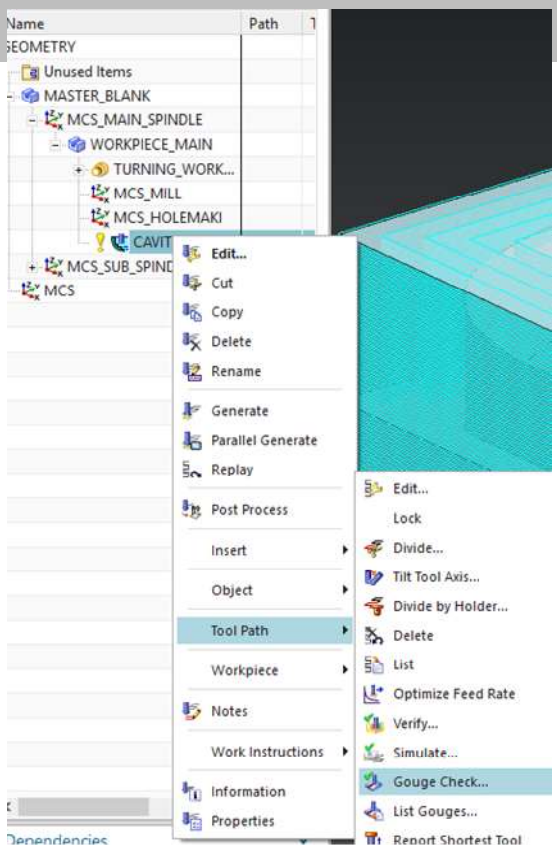
Aula 12

Gouge Check

- Right Click the program in the

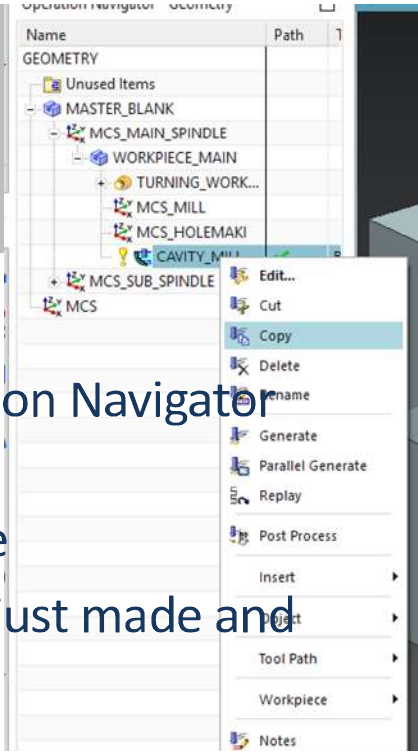
Operation Navigator

- Choose **Tool Path** →**Gouge Check**
- Click **OK**



Aula 12

OPERATION METHODS



Semi-Finishing

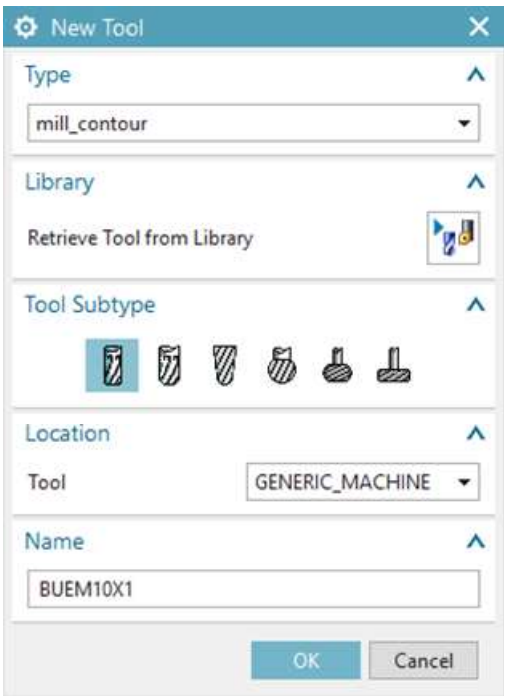
- Right-click CAVITY_MILL program in the Operation Navigator and click Copy
- Right-click CAVITY_MILL again and choose Paste
- Right-click the second CAVITY_MILL_COPY you just made and click Rename
- Rename the second program as CAVITY_MILL_1
- You can see that next to the newly created CAVITY_MILL_1 is a red mark, which indicates that the program is not generated.

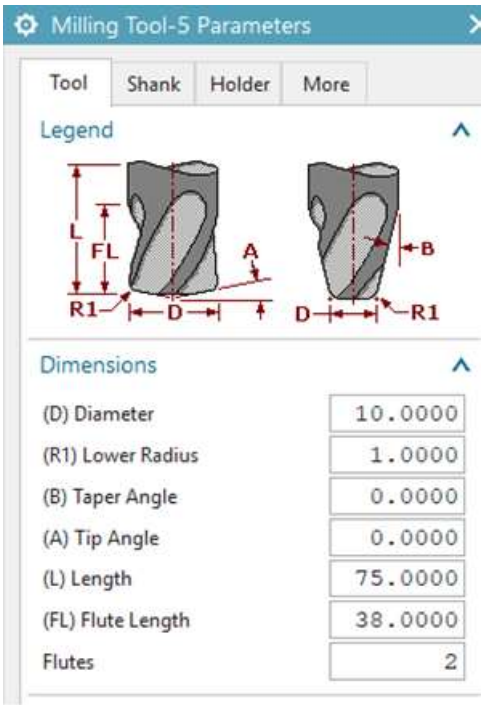
Aula 12

- Let us now set the parameters that need to be changed for the second program. Before we even start, we should analyze the part geometry to figure out the minimum corner radius for the cutter diameter. In our model, it is 5 mm and at the floor edges, it is 1 mm. Therefore, the cutter diameter can be anything less than 10 mm. For optimal output and rigidity, we will choose a Bull Nose Cutter with a diameter of 10 and a lower radius of 1.

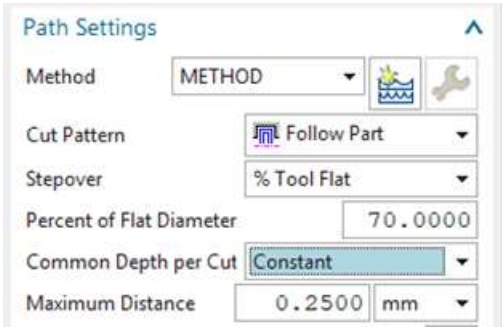
Aula 12

- Create a new **Mill** and name it **BUEM10X1**
- It should have a **Diameter** of **10**, a Lower **Radius** of **1** and a **Flute Length** of **38**
- Click **OK**





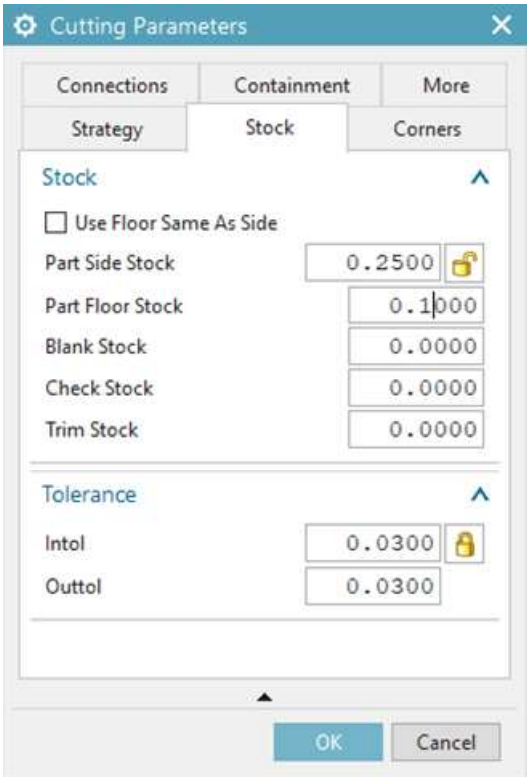
- Click the Common Depth per Cut as 0.25 in the Path Settings



Aula 12

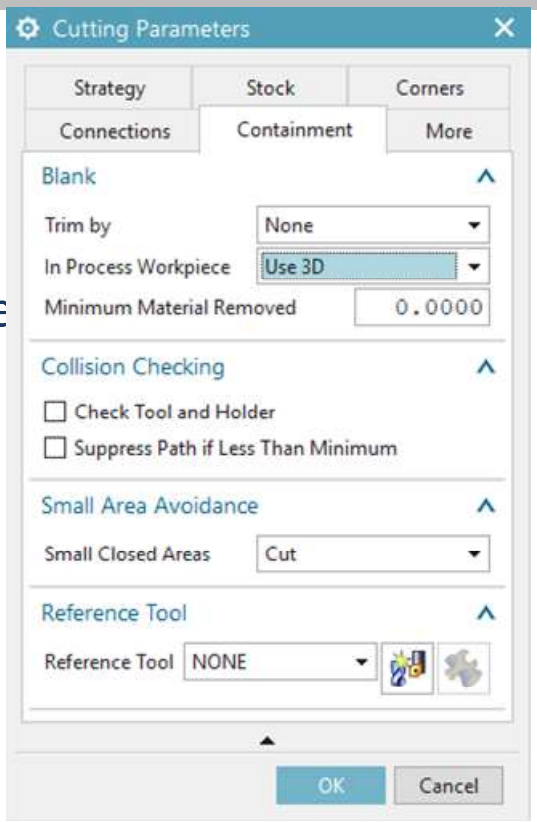
Cutting Parameters

- Click on the **Stock tab**
- Uncheck the box next to **Use Floor Same As Side**
- Enter **0.25** for **Part Side Stock**
- Enter **0.1** for **Part Floor Stock**



Aula 12

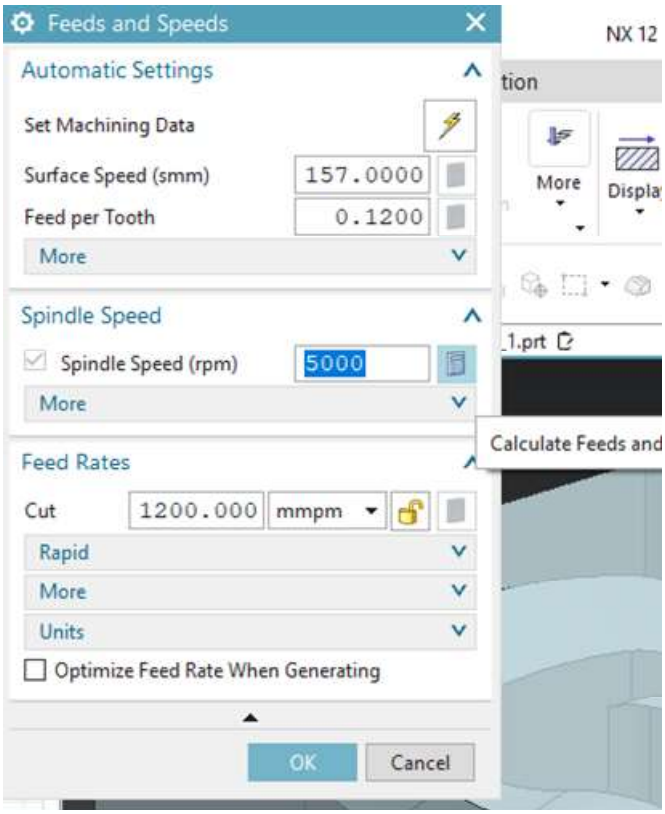
- Click on the **Containment:** In the drop-down menu next to In Process Workpiece, choose Use 3D.
- Choose OK to return to the Paramete



Departamento de Computação

Aula 12

- Click **Feeds** and **Speeds**
- Enter the **Spindle Speed** as **5000** and click on the **Calculator**
- Then click **OK**



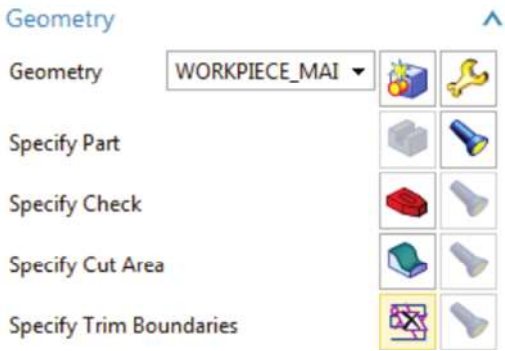
Aula 12

- Regenerate the program by clicking on the **Generate icon**
- After the software finishes generating click **OK**

- Then replay the Tool Path Visualization

Aula 12

- Repeat the same procedure as before to copy and paste **CAVITY_MILL_1** on **Operation Navigator**
- Rename the program **CAVITY_MILL_2**
- Double click **CAVITY_MILL_2** to make parameter changes
- In the pop-up parameters window, change the **Cut Pattern** to **Profile** and **the Stepover percentage** to **40**
- Click on the **Specify Trim Boundaries** tab



Aula 12

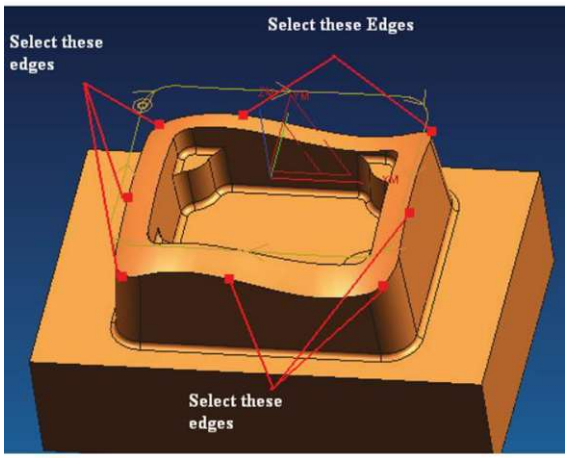
- Change the **Selection Method** to **Curves**
- Change the **Plane** from **Automatic** to **Specify** and click on the **Plane** Dialog

- Choose the **XC-YC Plane** from the drop-down menu under Type
- Enter a value of **3** next to **Distance**
- Click **OK**

Departamento de Computação

Aula 12

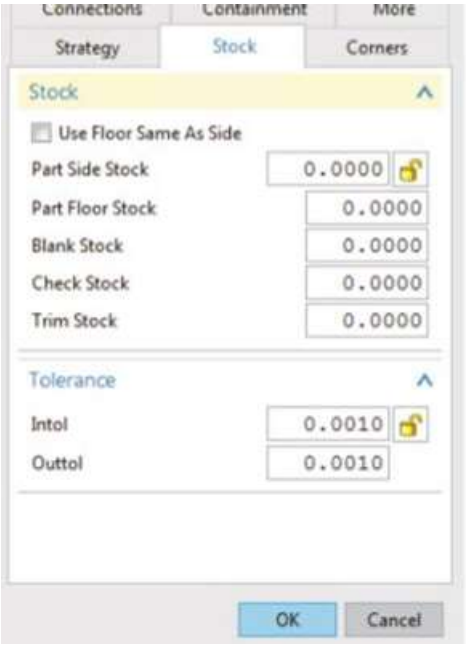
- Now we will start selecting edges from the part. These selected edges will be projected on the Z = 3 plane as curves and used as the boundary.
- Select all the top outer edges on the wall along the contour surface as shown in the figure.
- Make sure to select all **8 edges** and in a **continuous order**
- Choose **OK**



- Enter the Common Depth per Cut as 0.2
- Click **Cutting Parameters**
- In the pop up dialog box, click on Stock tab
- **Enter the Part Side Stock and Part Floor Stock values to be 0.00**

Aula 12

- Enter the Intol and Outtol values to be 0.001 as shown in the figure
- Click on Containment tab and change the In-process Workpiece to None
- Click OK
- Click on the Generate icon to generate the program in the Main Parameters window
- Click OK on the parameters window when the program generation is completed



Post Processing

- Click on a program in the Operation Navigator that you want to post process
- Click Menu →Tools →Operation Navigator →Output →Postprocess
- Select the MILL_3_AXIS machine andenter a location for the file
- Select OK

Obs. This will create the Post Processed file for the target machine. You can find the block numbers with G and M codes concerning the machine controller type. The extension of the file is .ptp