

**Examples for NC file generation (3 operations, 12 tool paths)  
Turning and milling**

**System:** Pro Engineer Wildfire 5

In order to start this study, basic knowledge of modeling in Pro Engineer Wildfire 5 is needed.

**Attachments:** the design part model, the workpiece model as inputs of this study,  
3 AVI simulations as outputs of this study

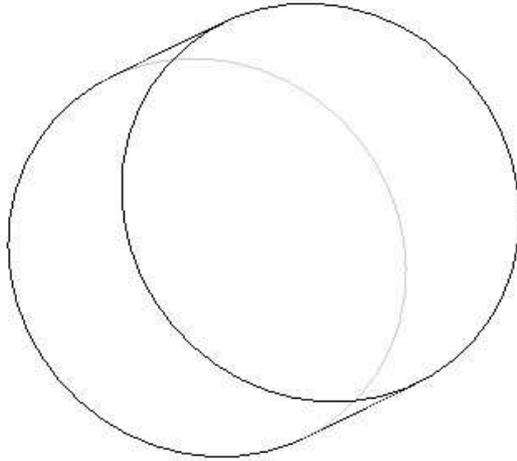
**Author: Dr. Ibolya Zsoldos**

2011

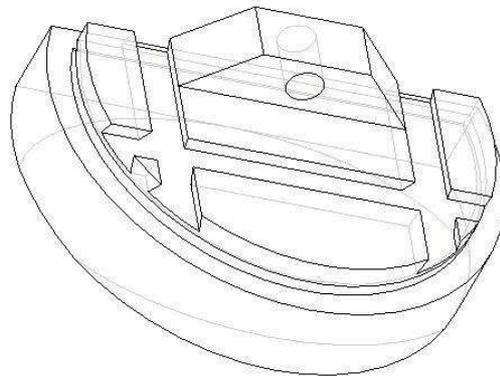
## GENERATING OF NC PROGRAMS FOR KNEE PROSTHESIS

In this exercise a detailed description is given about all steps of creating NC programs for machining of the knee prosthesis part. In order to start the study, the 3D model files of the design part and the workpiece are needed which are attached to this description. This detailed description can be used as a self-tutoring materials for the manufacturing modul of the Pro/ENGINEER Wildfire 5.0.

Create an own working directory on the Winchester. Start the Pro/Engineer. In Pro/Engineer choose **Set Working Directory** from the **File** menu and select the new directory created a moment ago. (Do the same after every start of the system.) All files we will create in Pro/Engineer will be placed automatically in this directory. The design part and the workpiece models need to be copied to this directory, Figure 1 and 2.



**Figure 1.** The workpiece model



**Figure 2.** The design or reference model

## 1. THE MANUFACTURING MODEL

This first part of the exercise will have we create the manufacturing model. We will need to bring in the design model and the workpiece. We do not show the fixture on the manufacturing model but we will take it into consideration during the manufacturing.

1. To create the manufacturing model, choose **File, New**.

At this point the *New* window should appear. The system needs to know what type of model is being created. Choose **Manufacturing** as type and **NC Assembly** as Sub-type.

Enter the name of this manufacturing model (e.g.exercise) and press **OK**.

If the New File Option window appears choose empty for the Template (we don't use master pattern or stencil for the modeling).

If the system defines a template automatically (without query), delete the three datum planes and the NC coordinate system from the model tree, which can be seen on the left side.

The exercise.mfg file will be created in your working directory, if you choose saving.

There is no automatic saving in Pro/Engineer.

2. The Manufacturing icon menu should appear on the right side of the screen. First we build our manufacturing model from our design part and workpiece. Choose from the **Insert** menu the **Reference model** and **Assembly**. The *Open* window should appear. Choose the design model file in the working directory and pick to **Open**. The design model should appear in the main work window. Close the *Create Reference Model* window (with the **Same Model** option) with **OK**.
3. Choose from the **Insert** menu the **Workpiece** and **Assembly**. Choose the workpiece model file in the working directory and pick to **Open**. The workpiece model should appear in the main work window, as well.
4. The system is now asking us to assemble the two components together. Below the main menu the assembly building tool appears. Open the **Placement** panel. For the first constraint choose **Align** on the Constraint Type list. Choose (pick) the A-1 axis on the design model, after that choose the A-1 axis on the workpiece. These two axes should stand together on the screen (the axis are the same). In order to

appear the hidden surfaces, it is useful to select the Hidden line view from this point.

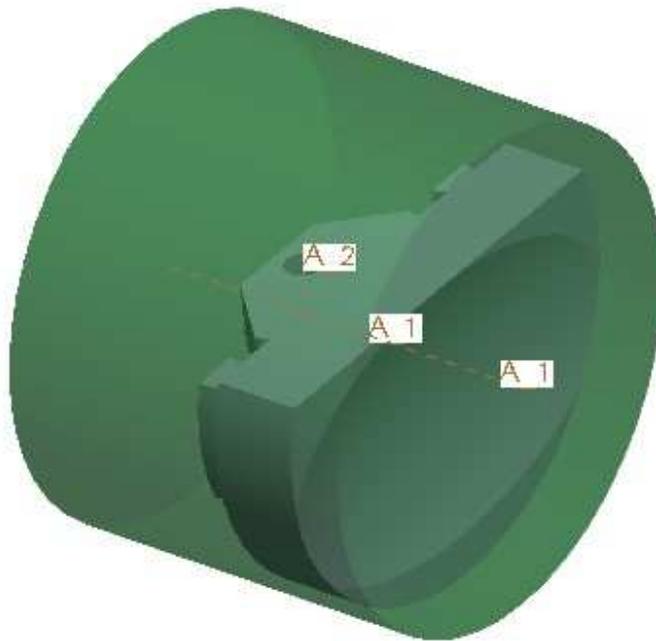
For the second constraint: choose **New Constraint** on the Placement panel and choose **Align** on the Constraint Type list again. Choose the boundary plane on the design model. Choose the boundary plane on the workpiece. Choose the **offset** option from the Offset list on the Placement panel, write **5** for it and press **enter** on the clipboard or pick to green type list again.

The *STATUS* of the assembly should be “*fully constrained*”.

Pick to green **OK** icon (the Nike pipe icon) on the right side of the component placement tool, below the main menu.

If the *Create Stock-workpiece* window appears, leave the Same Model option active and close it with **OK**.

Select *Shading* view from the upper main icon menu, the manufacturing model should be as we can see it in Figure 3.



**Figure 3.** The manufacturing model

You should notice that both sides of the model need to be machined. This will require three operations.

## **2. FIRST OPERATION**

The first operation will machine the bottom side of the reference model. As a first step, you will rotate the reference model to obtain a good 3D view from the bottom side of the model. Select a similar view as we see in the Figure 3.

5. To define the first operation choose the **Operation** from the **Steps** main menu. The *OPERATION SETUP* window should appear. The default operation name is "OP010". We can either accept the name or reenter it e.g."side1".
6. To define the NC machine pick to the icon button on the right side of the window, in the line of the 'NC machine'. The *MACHINE TOOL SETUP* window should appear. We can accept or modify the Machine Name: the default name is "Mach01". For the Machine Type choose **LATHE** on the popup list. To close the *MACHINE TOOL SETUP* window pick the **OK** button.
7. In the Fixture Setup option we could show the model of the fixture. An assembly containing the fixture can be here created. This step is not essential to generate NC programs, so we do not show it here in this example.
8. Machine Zero(machine coordinate system):

The coordinate system is a base feature we can create it any time on one of the design models (reference model or workpiece). The system requires the machine coordinate system and we can create it here, as well. The coordinate system represents the Home and Tram position for the manufacturing.

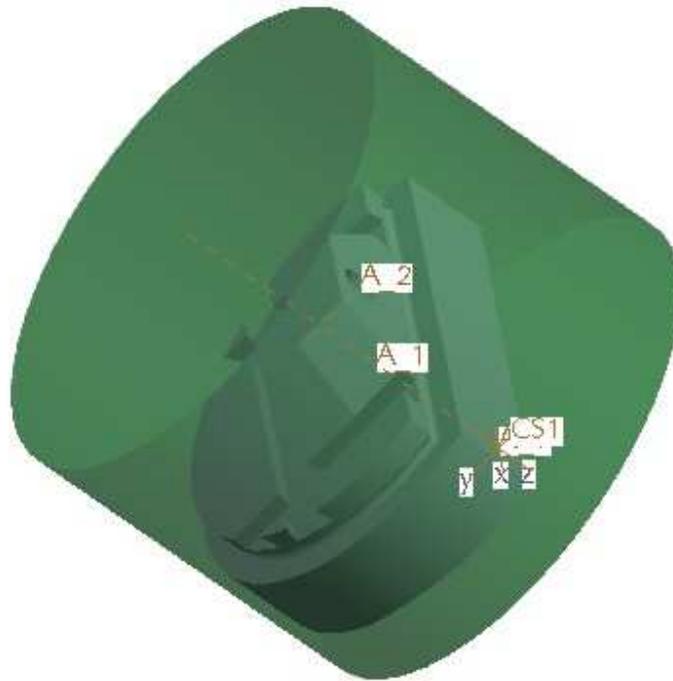
To create the coordinate system, pick the coordinate system icon in the main window, on the right side vertical icon menu. The *COORDINATE SYSTEM* window should appear. Pick inside the field of the 'References' and show references for our coordinate system as follows:

For example: select three perpendicular planes e.g. **DTM1, DTM2 and DTM3** datum planes on the workpiece (pick on them on the model, press the **Ctrl** button on the keyboard and keep it press during the selections). In this case the intersect point of the planes will be the origin and the intersect lines of the planes will be the axes of the coordinate system as you can see on the screen.

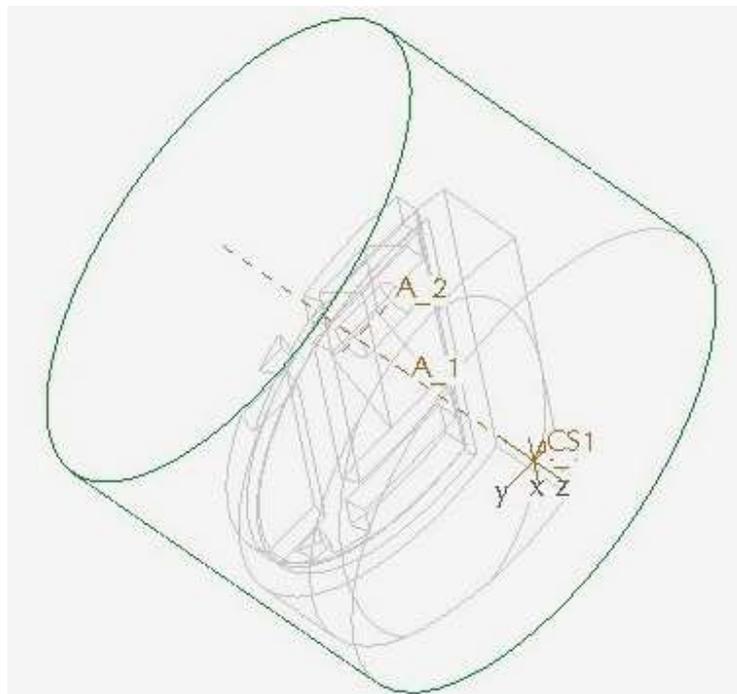
Second we have to define the orientation of the axes: in the *COORDINATE SYSTEM* window pick the **Orientation** panel. Define the Z-axis so, that it has to be parallel with the revolution axis of the part, Figure 4-5.

Close the *COORDINATE SYSTEM* window by picking the **OK** button in it.

To define this coordinate system as a machining coordinate system, pick the arrow icon near the 'Machine Zero' in the *OPERATION SETUP* window. When the *Select* popup menu appears, pick to the coordinate system created a moment ago, on the model. Close the *OPERATION SETUP* window by picking the **OK** button in it.



**Figure 4 .**The orientation of the NC machine coordinate system.



**Figure 5.** The orientation of the NC machine coordinate system.

## 2.1. NC sequence: 1. Area turning for machining the face plane

9. The icons of the possible turning NC sequences have appeared on the upper horizontal main icon menu.
10. The first NC sequence will be an Area turning sequence so pick the **Area Turning** icon. The NC SEQUENCE popup menu appears on the right side.
11. Form the SEQ SETUP menu accept the checked options of **Tool, Parameters, Tool Motions** and select **Start** and **End** options as well. These options has to be defined in the followings. Close the menu with **Done**.
12. The Tool Setup window should appear. Set the tooling parameters as follows:

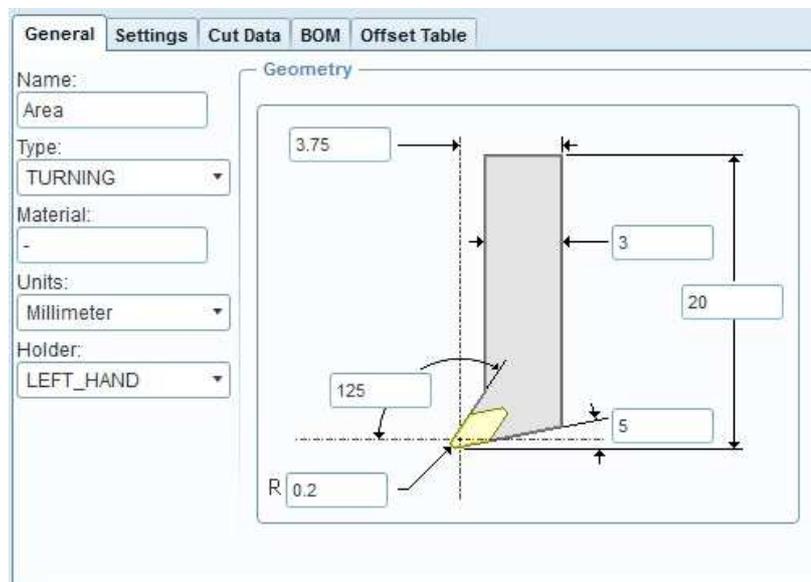
TOOL Name:	Area
TOOL TYPE	Turning

ON THE GEOMETRY PANEL:

TOOL_WITH	3
NOSE_RADIUS	0.2
SIDE_ANGLE	125
END_ANGLE	5
SIDE_WIDTH	3.75

ON THE SETTING PANEL:

TOOL_ NUMBER	1
--------------	---



**Figure3. 5.** The first tool

Before you close the TOOL SETUP window choose **APPLY** and **OK**.

13. The list of the NC Sequence parameters appears. To bring up all parameters required for this sequence type, pick to **ALL** button above the list. There are default values for a lot of parameters. It is obligatory to give the values only for the parameters where the field is filled by yellow. The most important parameters you can find in the basic list. Enter the parameters as shown below. (Verify the parameter of CUT\_ANGLE which has to be 90 at this NC sequence of the face turning.)

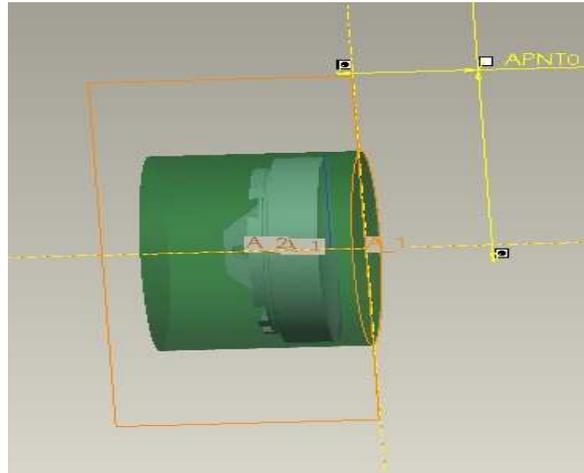
TOOL_ID	AREA
NCL_FILE	-
PRE_MACHINING_FILE	-
POST_MACHINING_FILE	-
ROUGH_OPTION	ROUGH_ONLY
SCAN_TYPE	TYPE_1_CONNECT
STEPOVER_ADJUST	YES
TRIM_TO_WORKPIECE	NO
CUT_DIRECTION	STANDARD
CORNER_FINISH_TYPE	FILLET
OUTPUT_POINT	CENTER
GOUGE_AVOID_TYPE	TIP_ONLY
STEP_DEPTH	1
MIN_STEP_DEPTH	-
TOOL_CLEARANCE	-
END_STEP_DEPTH	-
TOLERANCE	0.01
NUMBER_PASSES	0
NUM_PROF_PASSES	1
PROF_INCREMENT	0
PROF_STOCK_ALLOW	0
ROUGH_STOCK_ALLOW	0
CUT_ANGLE	90
Z_STOCK_ALLOW	-
START_OVERTRAVEL	0
END_OVERTRAVEL	0
CORNER_LENGTH	-
CHAMFER_DIM	-
CORNER_ANGLE	-
CONCAVE_RADIUS	-
CONVEX_RADIUS	-
BACK_CLEAR_ANGLE	5
CUT_FEED	75
ARC_FEED	-
FREE_FEED	-
RETRACT_FEED	-
CUT_UNITS	MMPM
RETRACT_UNITS	MMPM
SPINDLE_SPEED	800
SPINDLE_SENSE	CW
MAX_SPINDLE_RPM	-
SPEED_CONTROL	CONST_RPM
SPINDLE_RANGE	NO_RANGE
RANGE_NUMBER	0
CIRC_INTERPOLATION	ARC_ONLY
NUMBER_OF_ARC_PTS	1

COOLANT_OPTION	OFF
COOLANT_PRESSURE	NONE
CUTCOM	OFF
NUMBER_CUTCOM_PTS	1
CUTCOM_LOC_APPR	-
CUTCOM_LOC_EXIT	-
CUTCOM_REGISTER	-
FIXT_OFFSET_REG	-
COORDINATE_OUTPUT	MACHINE_CSYS
TOOL_ORIENTATION	90
END_STOP_CONDITION	NONE
OSETNO_VAL	-
Z_GAUGE_OFFSET	-
X_GAUGE_OFFSET	-
PLUNGE_FEED	-
PLUNGE_UNITS	MMPM
LEAD_RADIUS	0
TANGENT_LEAD_STEP	0
NORMAL_LEAD_STEP	0
RETRACT_RATIO	1.1
PLUNGE_ANGLE	0
PULLOUT_ANGLE	0
APPROACH_DISTANCE	-
EXIT_DISTANCE	-
RETRACT_CONTROL	NORMAL_TO_CUT
ENTRY_ANGLE	90
EXIT_ANGLE	90
START_MOTION	DIRECT
END_MOTION	DIRECT

After the parameters have been entered, close the list with **OK**.

**14.** Now we define the start tool position. The *DEF START* menu is active. Since there are no datum points on the models, we have to create one. Pick to the datum point icon on the right side icon menu and the DATUM POINT window appears. Pick in the 'References' field and pick inside the datum plane which is equal to the xz plane. The new point appears on the model as it can be seen in Figure 6. Click in the 'Offset reference' field and select a horizontal and a vertical reference on the model: e.g. the A1 axis and the vertical plane (the circle boundary plane) on the workpiece (keep press the Ctrl button on the keyboard for the selection of more than one reference). Enter the distances of the point from the references: **30** and **15**. It is important that the distance of 30mm is in the direction of the positive x axis, since the tool start to move in the positive xz plane. In the end pick **OK**.

**15.** Second we define the end tool positions. From the DEF END the Select option is active. Select (pick to) the same point we created a moment ago on the model.



**Figure 6.** Start and End tool position

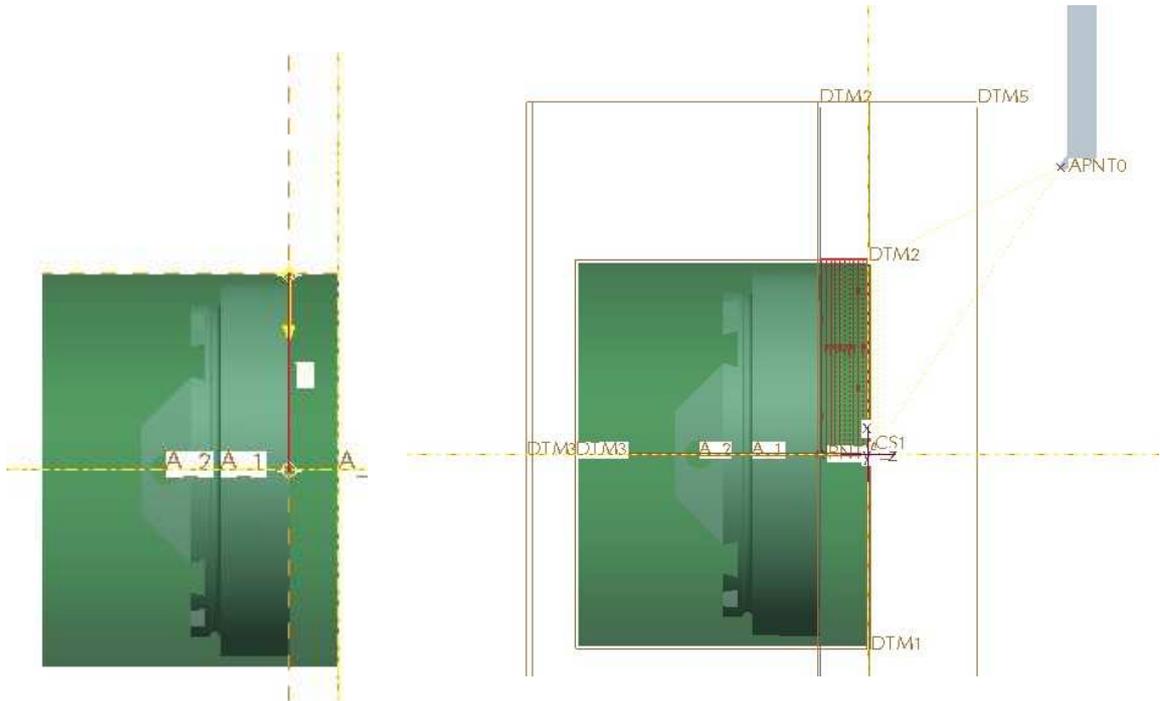
- 16.** In the *TOOL Motion* window we define the boundary surface of the material what we will remove from the workpiece in this NC sequence. It will be enough to show it by only a line (by a Turn profile) in the xz plane this boundary. Keep the default options of <end of tool path> and Area Turning Cut. Select **insert** and the AREA Turning Cut window appears. The Turn profile has to be defined. Pick the **Turn Profile** icon on the right side, vertical icon menu (perhaps it is hidden by the AREA Turning Cut window, so this should be moved).

The icon menu of the Turn profile command appears below the upper horizontal main icon menu. As the default option (Create an envelope on-the-fly), the system sign a boundary network with yellow colour. Select the **Use sketch to define turn profile** icon, and then **Define an internal sketch** icon. The SKETCH window appears, keep the default options and pick **Sketch**. The system revolves the model into the xz plane. Select appropriate references for sketching and sketch the only line shown in Figure 7, left. (Draw a line from up to the centre line along the boundary edge of the reference model as it can be seen in Fig 7, left). Close the sketcher (with ✓ icon). The system shows the removable material with the horizontal yellow arrow. If this arrow shows inside the model, pick to it in order to reverse it. Close the Turn profile icon menu with the green ✓ icon (above, on the right).

Activate the Area Turning Cut window by picking the left bottom arrow icon in it, and close it with the green ✓ icon.

Close the Tool motion window **OK**.

- 17.** In the last step of this NC sequence we see the simulation of the tool motion .There are two possibilities. First select **Play path** on the NC SEQUENCE menu after that select **Screen play**. See the simulation and the tool path (as it can be seen in Fig. 7, right), click to **Close**. Second select **Play path** again, after that select **NC Check**. In a new VERICUT application we can see the simulation. Close the NC SEQUENCE menu with **DONE Seq**.



**Figure7 . Left:** defining the boundary of the material which will be removed.  
**Right:** Tool path of the first NC Sequence

18. We remove the machined material from the model: from the main **Insert** menu select **Material removal cut**, and the list of the defined NC sequences appears. Select the **1:Area turning** from the list (now there is only this option), select **Automatic** and **Done**. The *Intersected comps* window appears. Select **Auto Add** and the file name of the workpiece model appears in the upper field, which means that material will be removed from this model. Select **OK**. The model appears without the removed material.

## 2.2. NC sequence: 2. Area Sequence for turning the profile

19. The second NC sequence will be an Area turning sequence again, so pick the **Area Turning** icon on the upper horizontal main manufacturing icon menu. The NC SEQUENCE popup menu appears on the right side.
20. Form the SEQ SETUP menu accept the checked options of **Tool, Parameters, Tool Motions** and select **Start** and **End** options, as well. These options have to be defined in the followings. Close the menu with **Done**.
21. The *Tool Setup* window should appear. Accept the first tool with **OK** (we use the same tool as in the last NC sequence).
22. The list of the NC Sequence parameters appears. To bring up all parameters required for this sequence type, pick to **ALL** button above the list. Enter the parameters as

shown below. (Verify the parameter of CUT\_ANGLE which has to be 0 at this NC sequence of the profile turning.)

TOOL_ID	AREA
NCL_FILE	-
PRE_MACHINING_FILE	-
POST_MACHINING_FILE	-
ROUGH_OPTION	ROUGH_ONLY
SCAN_TYPE	TYPE_1_CONNECT
STEPOVER_ADJUST	YES
TRIM_TO_WORKPIECE	NO
CUT_DIRECTION	STANDARD
CORNER_FINISH_TYPE	FILLET
OUTPUT_POINT	CENTER
GOUGE_AVOID_TYPE	TIP_ONLY
STEP_DEPTH	1
MIN_STEP_DEPTH	-
TOOL_CLEARANCE	-
END_STEP_DEPTH	-
TOLERANCE	0.01
NUMBER_PASSES	0
NUM_PROF_PASSES	1
PROF_INCREMENT	0
PROF_STOCK_ALLOW	0
ROUGH_STOCK_ALLOW	0
CUT_ANGLE	0
Z_STOCK_ALLOW	-
START_OVERTRAVEL	0
END_OVERTRAVEL	0
CORNER_LENGTH	-
CHAMFER_DIM	-
CORNER_ANGLE	-
CONCAVE_RADIUS	-
CONVEX_RADIUS	-
BACK_CLEAR_ANGLE	5
CUT_FEED	75
ARC_FEED	-
FREE_FEED	-
RETRACT_FEED	-
CUT_UNITS	MMPM
RETRACT_UNITS	MMPM
SPINDLE_SPEED	800
SPINDLE_SENSE	CW
MAX_SPINDLE_RPM	-
SPEED_CONTROL	CONST_RPM
SPINDLE_RANGE	NO_RANGE
RANGE_NUMBER	0
CIRC_INTERPOLATION	ARC_ONLY
NUMBER_OF_ARC_PTS	1
COOLANT_OPTION	OFF
COOLANT_PRESSURE	NONE
CUTCOM	OFF
NUMBER_CUTCOM_PTS	1
CUTCOM_LOC_APPR	-
CUTCOM_LOC_EXIT	-

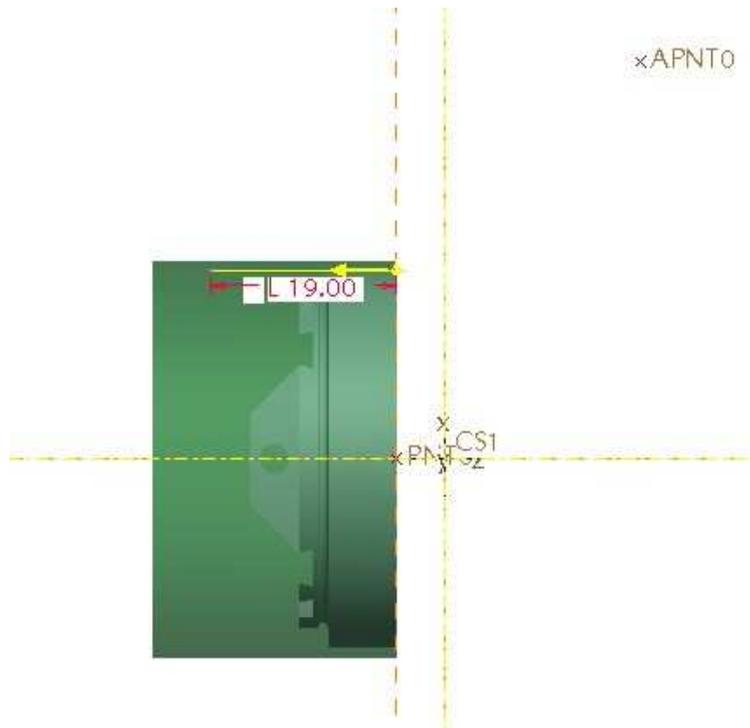
CUTCOM_REGISTER	-
FIXT_OFFSET_REG	-
COORDINATE_OUTPUT	MACHINE_CSYS
TOOL_ORIENTATION	90
END_STOP_CONDITION	NONE
OSETNO_VAL	-
Z_GAUGE_OFFSET	-
X_GAUGE_OFFSET	-
PLUNGE_FEED	-
PLUNGE_UNITS	MMPM
LEAD_RADIUS	0
TANGENT_LEAD_STEP	0
NORMAL_LEAD_STEP	0
RETRACT_RATIO	1.1
PLUNGE_ANGLE	0
PULLOUT_ANGLE	0
APPROACH_DISTANCE	-
EXIT_DISTANCE	-
RETRACT_CONTROL	NORMAL_TO_CUT
ENTRY_ANGLE	90
EXIT_ANGLE	90
START_MOTION	DIRECT
END_MOTION	DIRECT

After the parameters have been entered, close the list with **OK**.

23. We are defining the start and end tool positions. For requiring of the DEF START, second the DEF END menu select the same point we created in the first NC Sequence on the model .
24. In the *TOOL Motion* window we define the boundary surface of the material what we will remove from the workpiece in this NC sequence. It will be enough to show it by only a line (by a Turn profile) in the xz plane. Keep the default options of <end of tool path> and Area Turning Cut. Select **insert** and the AREA Turning Cut window appears. The Turn profile has to be defined. Pick the **Turn Profile** icon on the right side, vertical icon menu (perhaps it is hidden by the AREA Turning Cut window, so this should be moved).  
 The icon menu of the Turn profile command appears below the upper horizontal main icon menu. As the default option (Create an envelope on-the-fly), the system sign a boundary network with yellow colour. Select the **Use sketch to define turn profile** icon, and then the **Define an internal sketch** icon. The SKETCH window appears, keep the default options and pick **Sketch**. The system revolves the model into the xz plane. Select appropriate references for sketching and sketch the only line shown in Figure 8. (Draw a line from right left along the boundary edge of the reference model as it can be seen in Fig 8). Close the sketcher (with ✓ icon).  
 The system shows the removable material with the vertical yellow arrow. If this arrow shows inside the model, pick to it in order to reverse it. Close the Turn profile icon menu with the green ✓ icon (above, on the right).

Activate the Area Turning Cut window by picking the left bottom arrow icon in it. If the field of Turn profile is empty, pick to the Turn profile created a moment ago. Select Positive Z for the Start extension and Positive X for the End Extension. Close the Area Turning Cut window with the green ✓ icon.

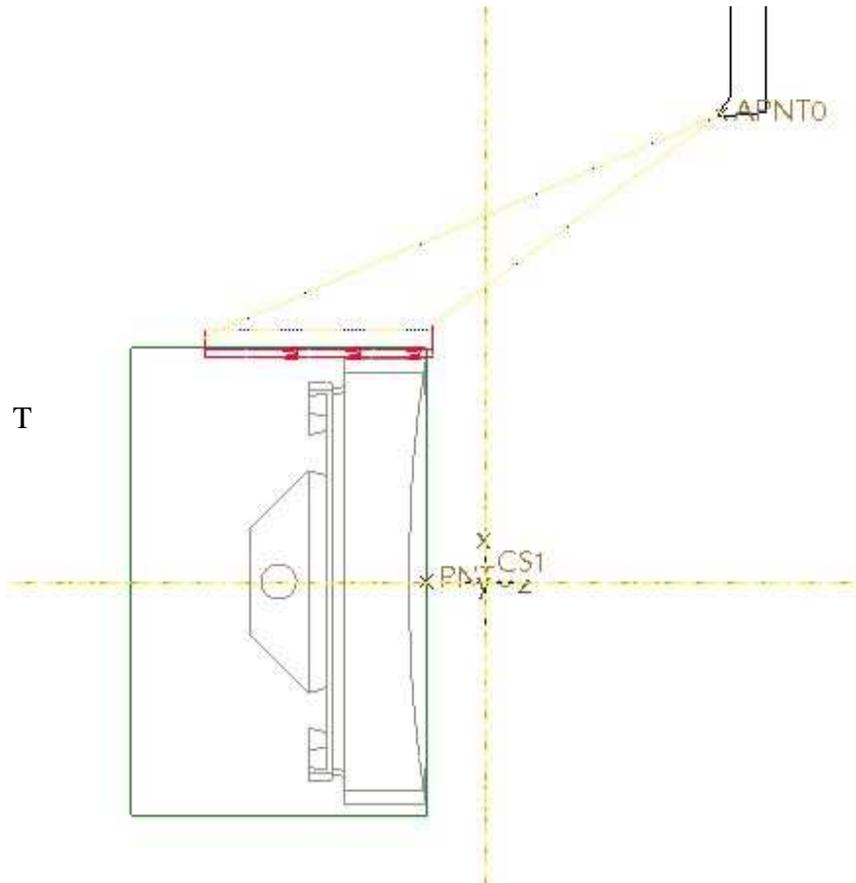
Close the Tool motion window **OK**.



**Figure 8.** Boundary edge for the material removing in the second NC Sequence

25. In the last step of this NC Sequence we verify the simulation of the tool motion. First select **Play path** on the NC SEQUENCE menu, after that select **Screen play**. See the simulation and click to **Close**. Second select **Play path** again after that select **NC Check**. In the VERICUT application we can play the simulation. Close the NC SEQUENCE menu with **Done Seq**.

26. We remove the machined material from the model: from the main **Insert** menu select **Material removal cut**, and the list of the defined NC sequences appears. Select the **2:Area turning** from the list (now there are two options on the NC SEQ LIST), select **Automatic** and **Done**. The *Intersected comps* window appears. Select **Auto Add** and the file name of the workpiece model appears in the upper field, which means that material will be removed from this model. Select **OK**. The model appears without the removed material.



**Figure 9.** The tool path of the second NC Sequence

### 2.3. 3. NC sequence: 1.Groove Sequence

27. In the Model Tree (left side of the screen) we can see how the new manufacturing entities are building after each other. The upper horizontal main manufacturing icon menu is active.
28. To start the 3. sequence choose **Groove Turning** from the upper horizontal main manufacturing icon menu. The NC SEQUENCE popup menu appears on the right side.
29. From the **SEQ SETUP** menu select the options of **Tool, Parameters, Tool Motions, Start and End**. These options have to be defined in the followings. Close the menu with **Done**.
30. The *Tool Setup* window should appear. Set the tooling parameters as follows:

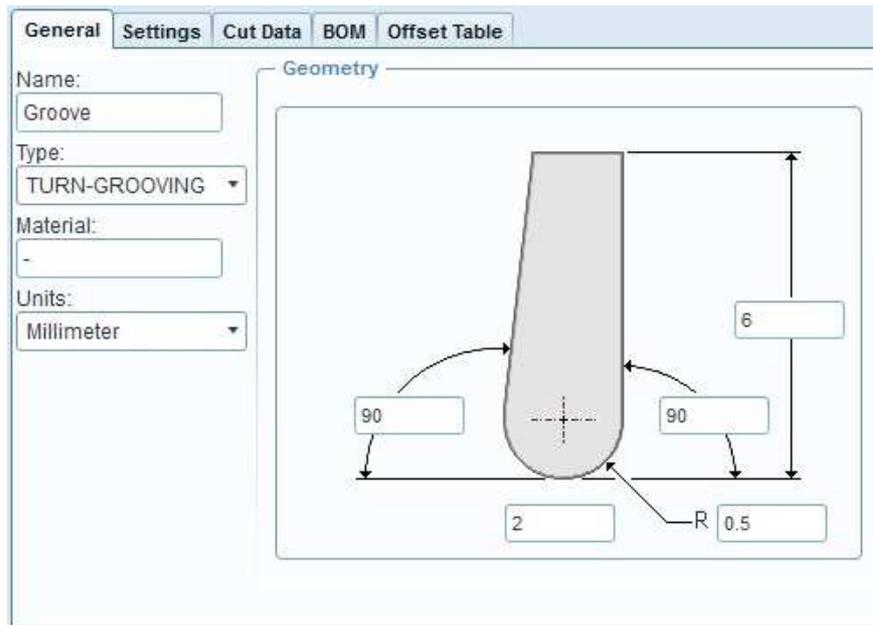
TOOL_ID	Groove
TOOL_TYPE	Turn-Grooving

ON THE GEOMETRY PANEL:

TOOL_WITH	2
LENGTH	6
NOSE_RADIUS	0.5
SIDE_ANGLE	90
END_ANGLE	90

ON THE SETTING PANEL:

TOOL_NUMBER	2
-------------	---



**Figure 10.** The Turn-Grooving tool

To close the *TOOL SETUP* window choose **APPLY** and **OK**.

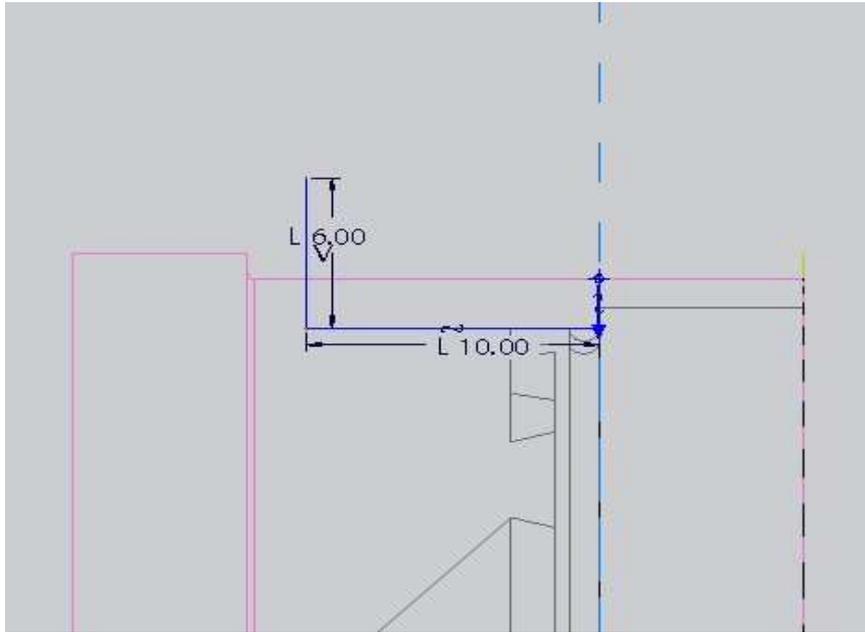
**31.** The list of the NC Sequence parameters appears. To bring up all parameters required for this sequence type, pick to **ALL** button above the list. Enter the parameters as shown below.

TOOL_ID	GROOVE
NCL_FILE	-
PRE_MACHINING_FILE	-
POST_MACHINING_FILE	-
ROUGH_OPTION	ROUGH_&_PROF
SCAN_TYPE	TYPE_1
STEPOVER_ADJUST	YES
TRIM_TO_WORKPIECE	NO
CUT_DIRECTION	STANDARD
CORNER_FINISH_TYPE	FILLET
OUTPUT_POINT	CENTER

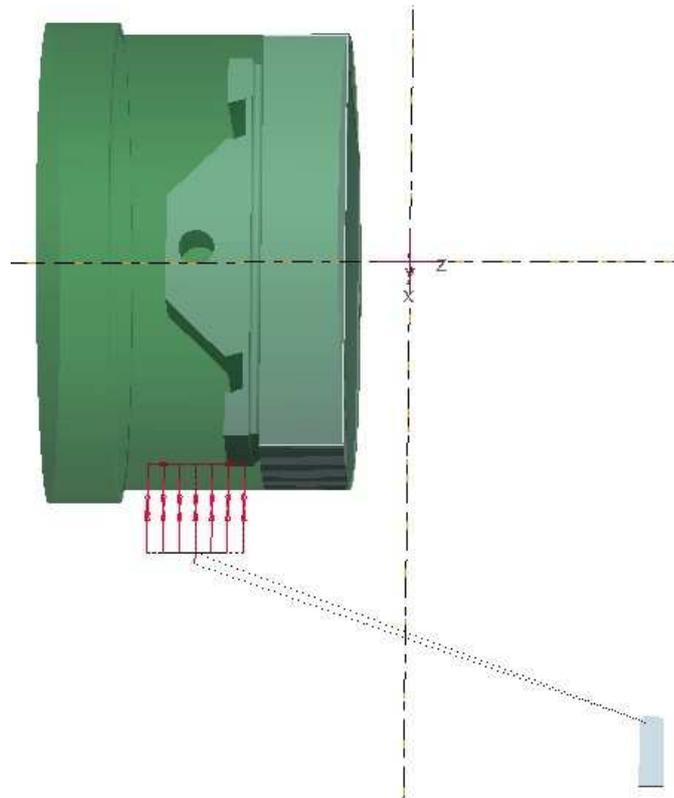
GROOVE_FINISH_TYPE	NO_BACKCUT
TOLERANCE	0.01
STEP_OVER	1.5
NUMBER_PASSES	0
NUM_PROF_PASSES	1
PROF_INCREMENT	0
PROF_STOCK_ALLOW	0
ROUGH_STOCK_ALLOW	0
Z_STOCK_ALLOW	-
SIDEWALL_OFFSET	-
PECK_DEPTH	0
DEEP_GROOVE_OPTION	BY_DEPTH
CUT_FEED	75
ARC_FEED	-
FREE_FEED	-
RETRACT_FEED	-
CUT_UNITS	MMPM
RETRACT_UNITS	MMPM
SPINDLE_SPEED	800
SPINDLE_SENSE	CW
MAX_SPINDLE_RPM	-
SPEED_CONTROL	CONST_RPM
SPINDLE_RANGE	NO_RANGE
RANGE_NUMBER	0
CIRC_INTERPOLATION	ARC_ONLY
NUMBER_OF_ARC_PTS	1
COOLANT_OPTION	OFF
COOLANT_PRESSURE	NONE
CUTCOM	OFF
NUMBER_CUTCOM_PTS	1
CUTCOM_LOC_APPR	-
CUTCOM_LOC_EXIT	-
CUTCOM_REGISTER	-
CUTCOM_REG_RIGHT	-
FIXT_OFFSET_REG	-
COORDINATE_OUTPUT	MACHINE_CSYS
DELAY	-
TOOL_ORIENTATION	90
END_STOP_CONDITION	NONE
OSETNO_VAL	-
Z_GAUGE_OFFSET	-
X_GAUGE_OFFSET	-
ALTERNATE_SIDE_OUTPUT	NO
ALT_OSETNO_VAL	-
PLUNGE_FEED	-
PLUNGE_UNITS	MMPM
CLEAR_DIST	1
PULLOUT_DIST	-
FULL_RETRACT_DEPTH	0
LEAD_RADIUS	0
TANGENT_LEAD_STEP	0
NORMAL_LEAD_STEP	0
APPROACH_DISTANCE	-
EXIT_DISTANCE	-
ENTRY_ANGLE	90
EXIT_ANGLE	90
START_MOTION	DIRECT

After the parameters have been entered, exit with **OK**.

- 32.** We are defining the start and end tool positions. For requiring of the DEF START, second the DEF END menu select the same point we created in the first NC Sequence on the model.
- 33.** In the *TOOL Motion* window we define the boundary surface of the material what we will remove from the workpiece in this NC sequence. It will be enough to show it by only a line (by a Turn profile) in the xz plane. Keep the default options of <end of tool path> and Area Turning Cut. Select **insert** and the AREA Turning Cut window appears. The Turn profile has to be defined. Pick the **Turn Profile** icon on the right side, vertical icon menu (perhaps it is hidden by the AREA Turning Cut window, so this should be moved).  
The icon menu of the Turn profile command appears below the upper horizontal main icon menu. As the default option (Create an envelope on-the-fly), the system sign a boundary network with yellow colour. Select the **Use sketch to define turn profile** icon, and then the **Define an internal sketch** icon. The SKETCH window appears, keep the default options and pick **Sketch**. The system revolves the model into the xz plane. Select appropriate references for sketching and sketch the line network shown in Figure 11. (Two vertical and one horizontal lines as it can be seen in Fig 11). Close the sketcher (with ✓ icon).  
The system shows the removable material with the vertical yellow arrow. If this arrow shows inside the model, pick to it in order to reverse it. Close the Turn profile icon menu with the green ✓ icon (above, on the right).  
Activate the Area Turning Cut window by picking the left bottom arrow icon in it. If the field of Turn profile is empty, pick to the Turn profile created a moment ago. Select Positive X for the Start and for the End Extension.  
Close the Area Turning Cut window with the green ✓ icon.  
Close the Tool motion window **OK**.
- 34.** In the last step we see the simulations of the tool motion. First select **play path** on the NC SEQUENCE menu after that select **screen play**. See the simulation and click to **close**. Second select **play path** again after that select **NC Check**. In new application, in the **VERICUT** we can see the simulation, Figure12.



**Figure 11.**The edge net (with dark blue colour) for the removing material



**Figure 12.** The tool path of the 3. NC Sequence

35. We remove the machined material from the model: from the main **Insert** menu select **Material removal cut**, and the list of the defined NC sequences appears. Select the **3:Groove turning** from the list (now there are three options on the NC SEQ LIST), select **Automatic** and **Done**. The *Intersected comps* window appears. Select **Auto Add** and the file name of the workpiece model appears in the upper field, which means that material will be removed from this model. Select **OK**. The model appears without the removed material.

### 2.3. NC sequence: 2. Groove Sequence

36. To start the 4. sequence choose **Groove Turning** from the upper horizontal main manufacturing icon menu. The NC SEQUENCE popup menu appears on the right side.
37. From the **SEQ SETUP** menu select the options of **Tool, Parameters, Tool Motions, Start** and **End**. These options have to be defined in the followings. Close the menu with **Done**.
38. The *Tool Setup* window should appear. Set the tooling parameters as follows:

TOOL_ID	Groove2
TOOL_TYPE	Turn-Grooving
ON THE GEOMETRY PANEL:	
TOOL_WITH	1
LENGTH	4
NOSE_RADIUS	0.5
SIDE_ANGLE	90
END_ANGLE	90
ON THE SETTING PANEL:	
TOOL_NUMBER	3

To close the *TOOL SETUP* window choose **APPLY** and **OK**.

39. The list of the NC Sequence parameters appears. To bring up all parameters required for this sequence type, pick to **ALL** button above the list. Enter the parameters as shown below.:

TOOL_ID	GROOVE2
NCL_FILE	-
PRE_MACHINING_FILE	-
POST_MACHINING_FILE	-
ROUGH_OPTION	ROUGH_ONLY
SCAN_TYPE	TYPE_1
STEPOVER_ADJUST	YES
TRIM_TO_WORKPIECE	NO

CUT_DIRECTION	STANDARD
CORNER_FINISH_TYPE	FILLET
OUTPUT_POINT	CENTER
GROOVE_FINISH_TYPE	NO_BACKCUT
TOLERANCE	0.01
STEP_OVER	1
NUMBER_PASSES	0
NUM_PROF_PASSES	1
PROF_INCREMENT	0
PROF_STOCK_ALLOW	0
ROUGH_STOCK_ALLOW	0
Z_STOCK_ALLOW	-
SIDEWALL_OFFSET	-
PECK_DEPTH	0
DEEP_GROOVE_OPTION	BY_DEPTH
CUT_FEED	75
ARC_FEED	-
FREE_FEED	-
RETRACT_FEED	-
CUT_UNITS	MMPM
RETRACT_UNITS	MMPM
SPINDLE_SPEED	800
SPINDLE_SENSE	CW
MAX_SPINDLE_RPM	-
SPEED_CONTROL	CONST_RPM
SPINDLE_RANGE	NO_RANGE
RANGE_NUMBER	0
CIRC_INTERPOLATION	ARC_ONLY
NUMBER_OF_ARC_PTS	1
COOLANT_OPTION	OFF
COOLANT_PRESSURE	NONE
CUTCOM	OFF
NUMBER_CUTCOM_PTS	1
CUTCOM_LOC_APPR	-
CUTCOM_LOC_EXIT	-
CUTCOM_REGISTER	-
CUTCOM_REG_RIGHT	-
FIXT_OFFSET_REG	-
COORDINATE_OUTPUT	MACHINE_CSYS
DELAY	-
TOOL_ORIENTATION	90
END_STOP_CONDITION	NONE
OSETNO_VAL	-
Z_GAUGE_OFFSET	-
X_GAUGE_OFFSET	-
ALTERNATE_SIDE_OUTPUT	NO
ALT_OSETNO_VAL	-
PLUNGE_FEED	-
PLUNGE_UNITS	MMPM
CLEAR_DIST	1
PULLOUT_DIST	-
FULL_RETRACT_DEPTH	0
LEAD_RADIUS	0
TANGENT_LEAD_STEP	0
NORMAL_LEAD_STEP	0
APPROACH_DISTANCE	-
EXIT_DISTANCE	-

ENTRY_ANGLE	90
EXIT_ANGLE	90
START_MOTION	DIRECT
END_MOTION	DIRECT

After the parameters have been entered, exit with **OK**.

40. We define the start and end tool positions. Select twice the same point we created in the first NC Sequence in the model .

41. In the *TOOL Motion* select **insert** and the AREA Turning Cut window appears. The Turn profile has to be defined. Pick the **Turn Profile** icon on the right side, vertical icon menu (perhaps it is hidden by the AREA Turning Cut window, so this should be moved).

The icon menu of the Turn profile command appears below the upper horizontal main icon menu. As the default option (Create an envelope on-the-fly), the system sign a boundary network with yellow colour. Select the **Use sketch to define turn profile** icon, and then the **Define an internal sketch** icon. The SKETCH window appears, keep the default options and pick **Sketch**. The system revolves the model into the xz plane. Select appropriate references for sketching and sketch the line network shown in Figure 13. (A vertical line and a half circle as it can be seen in Fig 13). Close the sketcher (with ✓ icon).

The system shows the removable material with the vertical yellow arrow. If this arrow shows right, pick to it in order to reverse it. Close the Turn profile icon menu with the green ✓ icon (above, on the right).

Activate the Area Turning Cut window by picking the left bottom arrow icon in it. If the field of Turn profile is empty, pick to the Turn profile created a moment ago.

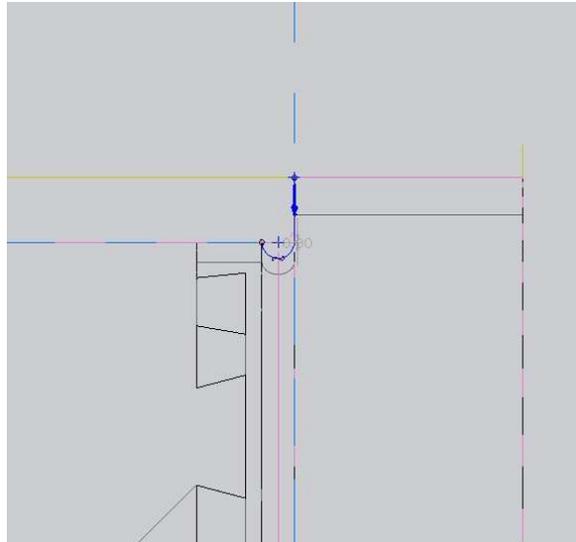
Select Positive X for the Start and for the End Extension.

Close the Area Turning Cut window with the green ✓ icon.

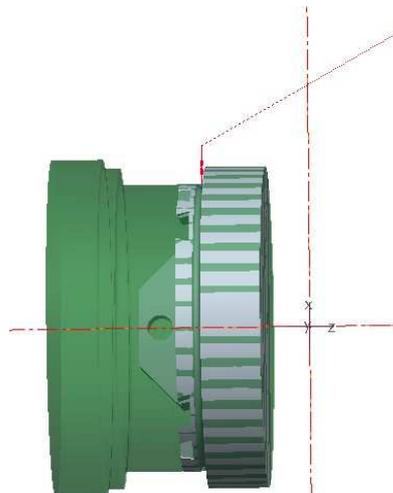
Close the Tool motion window **OK**.

42. In the last step we see the simulations of the tool motion. First select **play path** on the NC SEQUENCE menu after that select **screen play**. See the simulation and click to **close**. Second select **play path** again after that select **NC Check**. In new application, in the **VERICUT** we can see the simulation, Figure14.

43. It is not needed to remove the machined material from the model at this NC Sequence, since it does not influence the next tool path. If you intend to do it: from the main **Insert** menu select **Material removal cut**, and the list of the defined NC sequences appears. Select the **4:Groove turning** from the list (now there are four options on the NC SEQ LIST), select **Automatic** and **Done**. The *Intersected comps* window appears. Select **Auto Add** and the file name of the workpiece model appears in the upper field, which means that material will be removed from this model. Select **OK**. The model appears without the removed material.



**Figure 13.** The boundary edge for the removing material



**Figure 14.** Tool path of the 4.NC Sequence

## **2.5. MACHINING SIMULATION**

Every NC sequence has been created for the first turning operation. In the following steps we will create the machining simulation of the first operation.

44. From the main **Edit** menu choose **CL Data**, and **Output**. The SELECT FEAT menu appears on the right side. Select **Operation, OP\_010, File, Done**. Enter the name of the CL Data file (e.g. **Turning**). Close the popup menus.
45. From the main **Tools** menu choose **CL Data, Material Removal Simulation**, select the name of the CL Data file created a moment ago. Choose **Done**. In the VERICUT application we can see the full turning simulation.
46. Close the popup windows and menus and save the work.

### 3. SECOND OPERATION

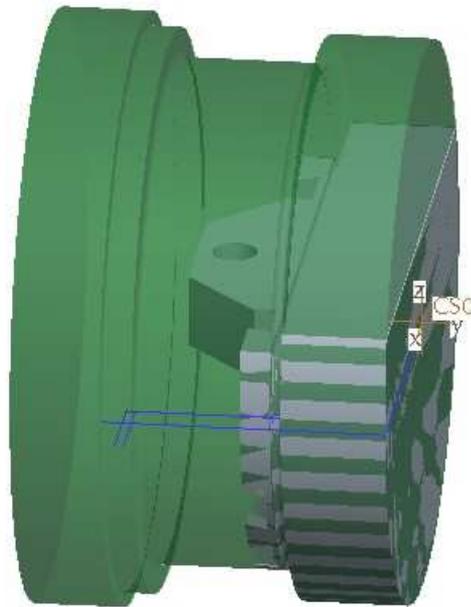
47. We are now going to start the second operation for this manufacturing model. An operation corresponds to a setup or tape file. We will need to turn over the model on the screen, Figure15. We will need to define new NC machine and Machining coordinate system.
48. In order to start the second operation, choose the **Operation** from the **Steps** main menu. The *OPERATION SETUP* window should appear. Choose **File** and **New** in this window and we get a new, empty surface. The default operation name is "OP020". We can either accept the name or reenter it e.g. "side2".
49. NC machine:  
To define the NC machine pick to its icon button on the right side of the window, in the line of the 'NC machine'. The *MACHINE TOOL SETUP* window should appear. Choose **File** and **New** in this window and we get a new, empty surface. We can accept or modify the Machine Name: the default name is "Mach02". For the Machine Type choose **MILL** on the popup list. Change the Number Of Axes from 3 to **5**. To close the *MACHINE TOOL SETUP* window pick the **OK** button
50. In the **Fixture Setup** option we can define the model of the fixture. An assembly containing the fixture can be created. This step is not essential to generate NC programs, so we not show it here in this example.
51. **Machine Zero** (machine coordinate system):  
The system requires the machine coordinate system and you can create it here, as well. To define the coordinate system, pick the arrow icon near the Machine Zero in the *OPERATION SETUP* window.  
Pick the coordinate system icon in the main window, on the right side vertical icon menu. The *COORDINATE SYSTEM* window should appear. Pick inside the field of the 'References' and show references for our coordinate system as follows:  
For example: select three perpendicular planes e.g. **DTM1, DTM2 and DTM3** datum planes on the workpiece (pick on them on the model, press the **Ctrl** button on the keyboard and keep it press during the selections). In this case the intersect point of the planes will be the origin and the intersect lines of the planes will be the axes of the coordinate system as you can see on the screen.

Second we have to define the orientation of the axes: in the *COORDINATE SYSTEM* window pick the **Orientation** panel. Define the Z-axis so, that it has to be parallel with the axis of the hole of the design part, see Figure 15. (From the positive Z-axis the tool starts to cut during NC milling.)

Close the *COORDINATE SYSTEM* window by picking the **OK** button in it.

#### 52. Retract plane

Pick the arrow icon near the „Surface” in the *OPERATION SETUP* window. The retract selection window should appear. Enter **30** to the Value which means the Z axis depth for the retraction of the tool. Close the window with **OK** and close the operation setup window with **OK**.



**Figure 15** Machine coordinate system of the second operation

### ***3.1. NC Sequence: Volume Rough***

53. New icons for the milling operation have appeared below the main horizontal menu. The first NC sequence will be a Volume Rough milling sequence so pick the **Volume Rough** icon. The NC SEQUENCE popup menu appears on the right side.

54. In the SEQ SETUP menu, make sure **Tool**, **Parameters** and **Window** are checked, then choose with **Done**.

55. The Tool Setup window should appear. Set the tooling parameters as follows and according to Figure 16:

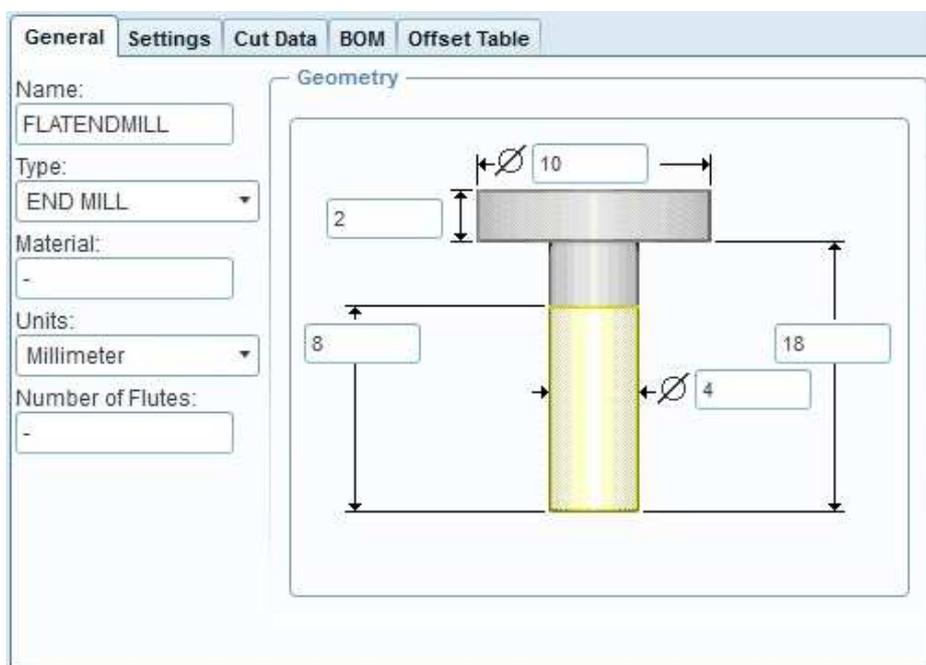
TOOL_ID	Flatendmil
TOOL_TYPE	END MILL

ON THE GEOMETRY PANEL:

CUTTER_DIAMETER	4
LENGTH	18

ON THE SETTING PANEL:

TOOL_NUMBER	4
-------------	---



**Figure 16:** The tool for the Volume Rough NC Sequence

Close the tool Setup window with **Apply** and **OK**.

56. The list of the NC Sequence parameters appears. To bring up all parameters required for this sequence type, pick to **ALL** button above the list. Enter the parameters as they are shown below:

TOOL_ID	FLATENDMLL
NCL_FILE	-
PRE_MACHINING_FILE	-
POST_MACHINING_FILE	-
ROUGH_OPTION	ROUGH_ONLY
SCAN_TYPE	TYPE_3
CUT_TYPE	CLIMB
STEPOVER_ADJUST	YES
RETRACT_OPTION	OPTIMIZE

RETRACT_TRANSITION	CORNER_TRANSITION
TRIM_TO_WORKPIECE	FULL_TRIM
CUT_DIRECTION	STANDARD
CORNER_FINISH_TYPE	FILLET
CUSTOMIZE_AUTO_RETRACT	YES
POCKET_EXTEND	TOOL_ON
PLUNGE_PREVIOUS	NO
STEP_DEPTH	1
MIN_STEP_DEPTH	-
TOLERANCE	0.01
STEP_OVER	2
CORNER_ROUND_RADIUS	-
TOOL_OVERLAP	-
NUMBER_PASSES	0
NUM_PROF_PASSES	1
PROF_INCREMENT	0
ROUGH_STOCK_ALLOW	0
PROF_STOCK_ALLOW	0
BOTTOM_STOCK_ALLOW	-
WALL_SCALLOP_HGT	0
BOTTOM_SCALLOP_HGT	0
AXIS_SHIFT	0
CUT_ANGLE	0
CUT_FEED	75
FREE_FEED	-
RETRACT_FEED	-
PLUNGE_FEED	-
RAMP_FEED	-
ARC_FEED	-
ARC_FEED_CONTROL	TOOL_CENTER
TRAVERSE_FEED	-
CUT_UNITS	MMPM
RETRACT_UNITS	MMPM
PLUNGE_UNITS	MMPM
SPINDLE_SPEED	800
WALL_PROFILE_SPINDLE_SPEED	-
SPINDLE_SENSE	CW
MAX_SPINDLE_RPM	-
SPEED_CONTROL	CONST_RPM
SPINDLE_RANGE	NO_RANGE
RANGE_NUMBER	0
LINTOL	-
CIRC_INTERPOLATION	ARC_ONLY
NUMBER_OF_ARC_PTS	1
COOLANT_OPTION	OFF
COOLANT_PRESSURE	NONE
CUTCOM	OFF
NUMBER_CUTCOM_PTS	1
CUTCOM_LOC_APPR	-
CUTCOM_LOC_EXIT	-
CUTCOM_REGISTER	-
FIXT_OFFSET_REG	-
COORDINATE_OUTPUT	MACHINE_CSYS
END_STOP_CONDITION	NONE
OSETNO_VAL	-
Z_GAUGE_OFFSET	-
WALL_PROFILE_CUT_FEED	-

RAMP_ANGLE	90
CLEAR_DIST	1
PULLOUT_DIST	-
HELICAL_DIAMETER	-
LEAD_IN	NO
LEAD_OUT	NO
LEAD_RADIUS	0
TANGENT_LEAD_STEP	0
NORMAL_LEAD_STEP	0
APPR_EXIT_PATH	TRIM_BOTH
APPR_EXIT_HEIGHT	DEPTH_OF_CUT
APPROACH_DISTANCE	-
EXIT_DISTANCE	-
OVERTRAVEL_DISTANCE	0
APPR_EXIT_EXT	0
ENTRY_ANGLE	90
EXIT_ANGLE	90
START_MOTION	DIRECT
END_MOTION	DIRECT
RETRACT_RADIUS	-

It is important to verify, that the parameter TRIM\_TO\_WORKPIECE has to be FULL\_TRIM at this NC Sequence.

After the parameters have been modified, exit the parameter tree window by choosing **OK**.

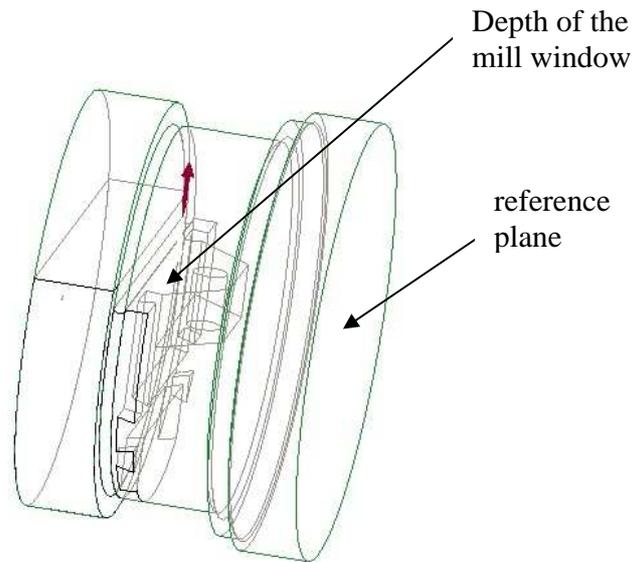
57. For this sequence we will create a Mill Window. When the DEFINE WIND menu appears choose the **Mill Window** icon from the right side, vertical icon menu. The Define window plane icon menu appears below the upper horizontal main menu.

The Mill Window will be created by sketching a rectangle, using the outside edges of the workpiece. The starting plane (sketching plane) will be a plane above the reference model and the depth of the cut will be an internal plane of the model shown in Figure 17.

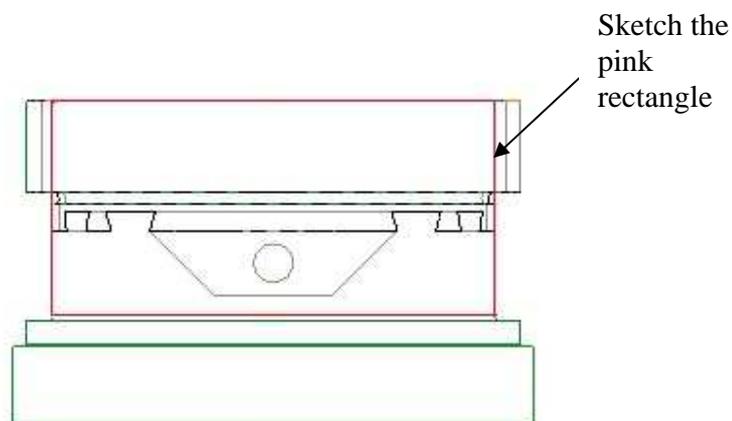
58. From the Define window plane icon menu select **Sketch window type** (the middle icon), since we will sketch the region which will be machined.
59. Open the **Placement** panel (pick to it). A window plane has to be selected. The window plane has to be above the model. Since there is no such plane on the model, we have to create it now. Pick to the **Datum plane tool** icon from the right side, vertical icon menu. The DATUM PLANE window appears. As a reference select the **DTM1** datum plane on the model. Enter the Offset translation of **20** mm. Verify that the new datum plane is above the model. Close the DATUM PLANE window with **OK**. If the Define window plane icon menu is not active, activate it by picking to the arrow icon on the right side. It is probable, that you need to open again the Placement panel and pick to the new datum plane which will be the starting window plane.
60. We have to define the area (a rectangle) on the new datum plane, since we have to define the region which will be machined. Pick to the **Define an internal sketch** icon on the Define window plane icon menu. The SKETCH window appears. As a reference select a plane on the model which is perpendicular to the starting window plane (the

new datum plane), Fig. 17. Select **Sketch** in the SKETCH window. Select the appropriate references for the sketching and sketch the rectangle shown in Figure 18. Close the sketcher.

61. In the end the depth of the Mill window has to be defined. Open the **Depth** panel on the Define window plane icon menu and sign the **Specify the depth**. From the Depth options select the **To selected** option and pick to the internal plane of the manufacturing model as it is shown in Figure 17. Close the Define window plane icon menu with the green ✓ icon.

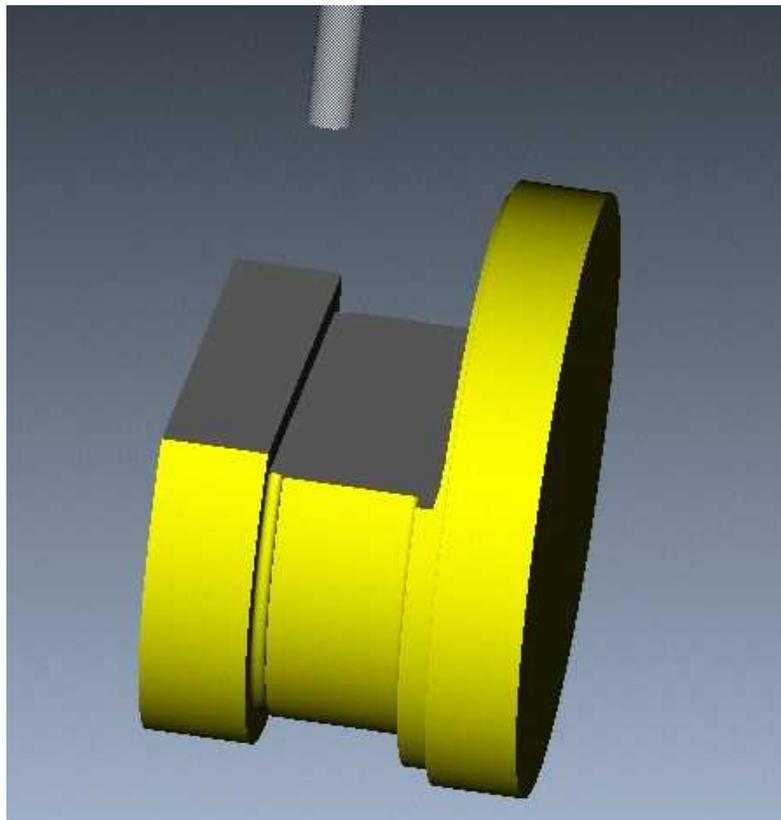


**Figure 17.** The depth of the mill window and the reference plane for the sketching



**Figure 18.** The Internal sketch of the Mill window

62. In the last step we see the simulations of the tool motion. First select **play path** on the NC SEQUENCE menu after that select **screen play**. Play the simulation and click to **close**. Second select **play path** again after that select **NC Check**. In new application, in the **VERICUT** we can see the simulation. The model is shown in Figure 19 after the machining.



**Figure19.** The model after the volume milling

63. In order to remove the material from the model, select **Material removal cut** from the main **Insert** menu, and the list of the defined NC sequences appears. Select the **1:Volume milling** from the list, select **Automatic** and **Done**. The *Intersected comps* window appears. Select **Auto Add** and the file name of the workpiece model appears in the upper field, which means that material will be removed from this model. Select **OK**. The model appears without the removed material.

### 3.2. NC Sequence: Trajectory milling

In this NC Sequence the linear groove at the neck will be cut. The Trajectory milling type of NC sequence will be used. It is characteristic that there is no tool path computation at this NC sequence, the toll path is sketched.

64. The second NC sequence will be a Trajectory milling sequence so pick the **Trajectory** icon. For the number of the machining axes accept the default **3 Axis** option and close the MACH AXES menu with **Done**. The NC SEQUENCE popup menu appears on the right side.

65. In the SEQ SETUP menu, make sure **Tool, Parameters, Tool Motion** are checked, then choose **Done**.

66. The Tool Setup window should appear. Set the tooling parameters as follows:

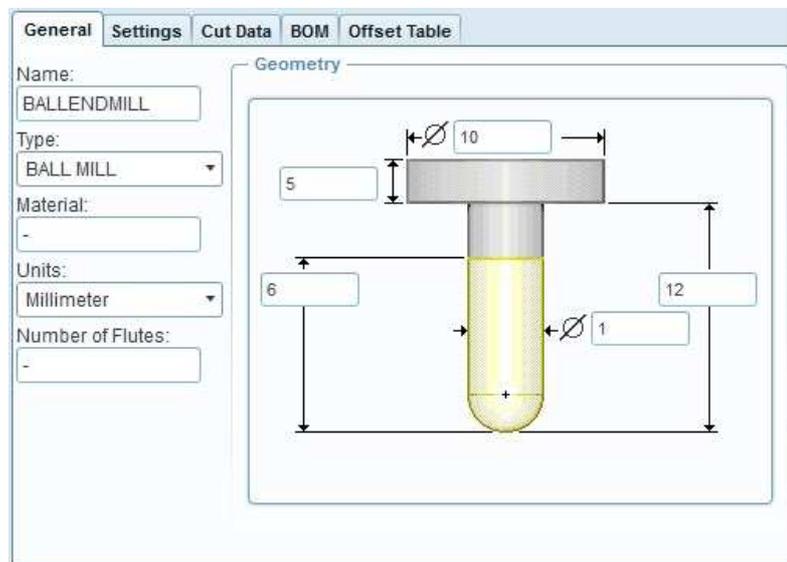
TOOL_ID	ballendmil
TOOL_TYPE	BALL MILL

ON THE GEOMETRY PANEL:

CUTTER_DIAMETER	1
LENGTH	12
CORNER RADIUS	0.5

ON THE SETTING PANEL:

TOOL_NUMBER	6
-------------	---



**Figure 20.** The tool for the Trajectory milling

We close the tool Setup window with **Apply** and **OK**.

67. The list of the NC Sequence parameters appears. To bring up all parameters required for this sequence type, pick to **ALL** button above the list. Enter the parameters as they are shown below:

CUT_FEED	75
SPINDLE_SPEED	800
CLEAR_DIST	1

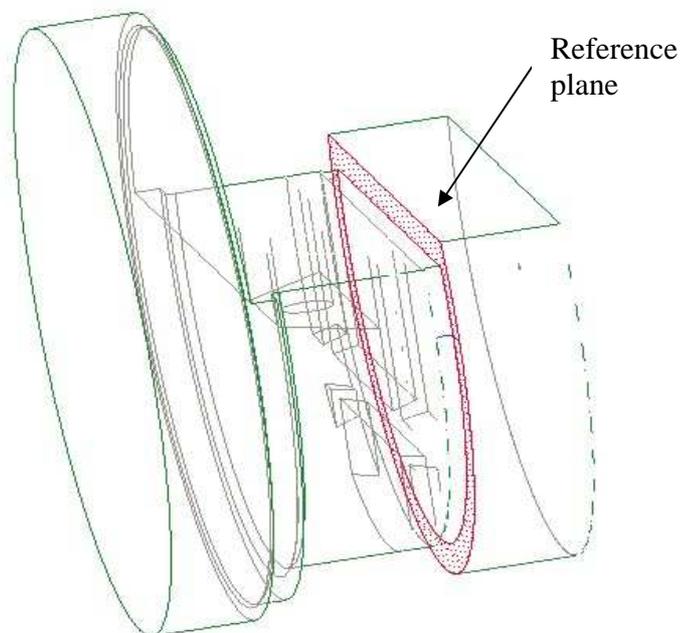
After the parameters have been modified, exit the parameter tree window by choosing **OK**.

68. The Tool Motions window appears. Choose **Insert** and the Curve Trajectory Setup window appears. Now we have to show the curve of the tool path. Since we do not have such curve, we will create it here. Select the Sketch tool from the right side, vertical main icon menu. The sketch window appears. For the Sketching plane select the plane signed with red colour in Figure 21 and for the reference plane select the perpendicular plane.

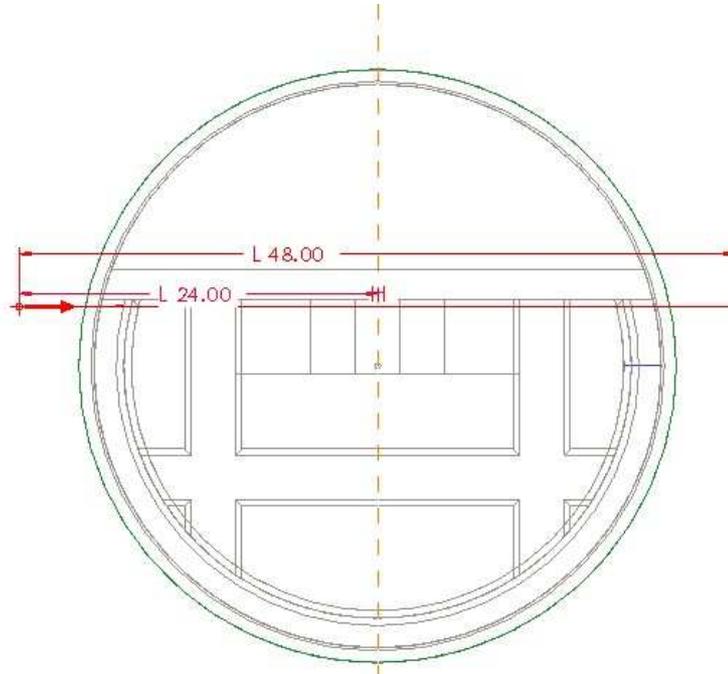
Select appropriate references for the sketching and sketch the line according to Figure 22. It has to be mentioned that this line is at the bottom of the groove.

Close the sketcher and activate the Curve Trajectory Setup window if it is needed. (For the activation pick to the arrow icon bottom in the window.) Select **Left** for Tool Offset and verify that the offset arrow shows inside the groove. Close the Curve Trajectory Setup window with **OK**.

Close the Tool Motions window with **OK**.

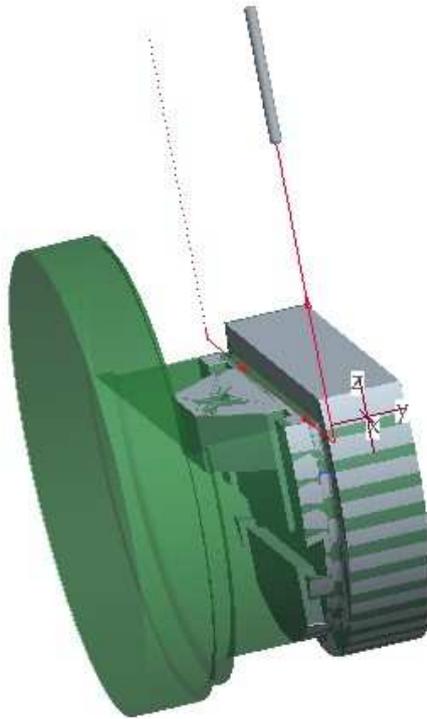


**Figure 21.** Definition of the new datum plane



**Figure 22** Definition of the trajectory

69. Select **play path** on the NC SEQUENCE menu after that select **screen play**. Play the simulation and click to **close**. Second select **play path** again after that select **NC Check**. In new application, in the **VERICUT** we can see the simulation. The model is shown in Figure 23 after the machining.



**Figure 23.** Tool path of the trajectory NC Sequence

### ***3.3. NC Sequence: Holmaking***

70. The third NC sequence will be a Holmaking sequence so pick the **Insert a drilling cycle** icon and select **Standard**. For the number of the machining axes accept the default **3 Axis** option and close the MACH AXES menu with **Done**. The NC SEQUENCE popup menu appears on the right side.

71. In the SEQ SETUP menu, make sure **Tool, Parameters, Holes** are checked, then choose **Done**.

72. The Tool Setup window appears. Set the tooling parameters as follows:

TOOL_ID	DRILLING
TOOL_TYPE	DRILLING

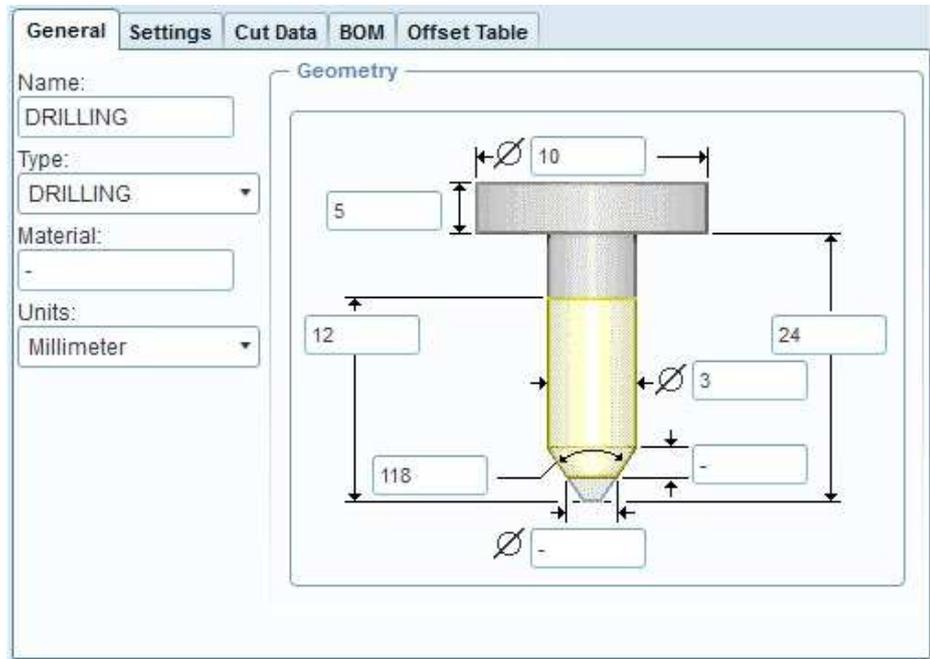
ON THE GEOMETRY PANEL:

CUTTER_DIAMETER	3
-----------------	---

LENGTH	12
POINT ANGLE	118

ON THE SETTING PANEL:

TOOL_ NUMBER	7
--------------	---



**Figure 24:** Tool for the drilling

We close the Tool Setup window with **Apply** and **OK**.

73. The list of the NC Sequence parameters appears. To bring up all parameters required for this sequence type, pick to **ALL** button above the list. Enter the parameters as they are shown below:

CUT_FEED	75
SPINDLE_SPEED	800
CLEAR_DIST	1

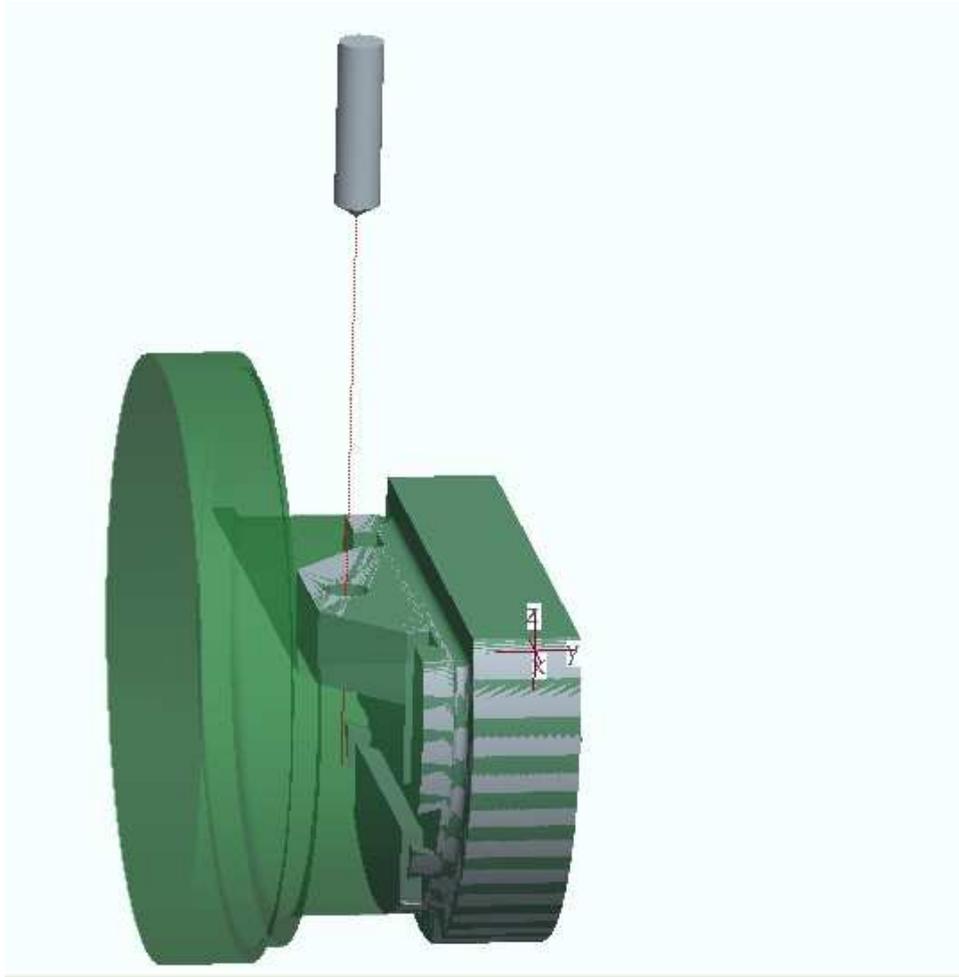
After the parameters have been modified, exit the parameter tree window by choosing **OK**.

74. The hole set window appears. To define the Depth accept the option of Auto for the Start option, and enter **7** (mm) for the End option.

For the Hole axes Set select the axis (**A-2**) of the hole of the design model.

Close the Hole Set window with the green ✓ icon.

The tool path definition is ready as it is shown in the figure 25. We can play the simulation as we have done it in the previous NC Sequences. We close the new NC Sequence with **Done Seq.**



**Figure 25:** Tool path of the holemaking

### ***3.4. NC Sequence: Surface milling***

75. The fourth NC sequence will be a Surface Milling sequence so pick the **Surface Milling** icon. For the number of the machining axes choose **5 Axis** option and close the MACH AXES menu with **Done**. The NC SEQUENCE popup menu appears on the right side.

76. In the SEQ SETUP menu, make sure **Tool, Parameters, Surfaces, Define cut** are checked, then choose **Done**.

77. The Tool Setup window should appear. Set the tooling parameters as follows:

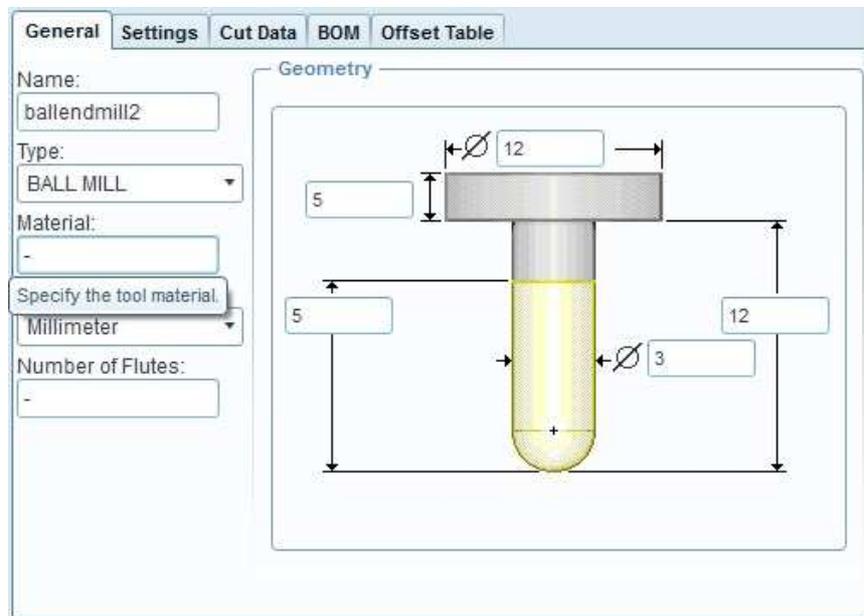
TOOL_ID	BALLENDMILL2
TOOL_TYPE	BALL MILL

ON THE GEOMETRY PANEL:

CUTTER_DIAMETER	3
LENGTH	12
CORNER_RADIUS	1.5

ON THE SETTING PANEL:

TOOL_NUMBER	8
-------------	---



**Figure 26:** Tool for the Surface Milling

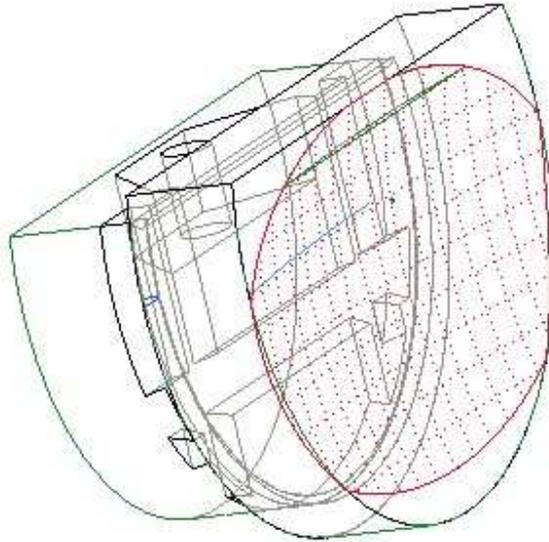
We close the tool Setup window with **Apply** and **OK**.

78. The list of the NC Sequence parameters appears. To bring up all parameters required for this sequence type, pick to **ALL** button above the list. Enter the parameters as they are shown below:

CUT_FEED	75
STEP_OVER	1
SPINDLE_SPEED	800
CLEAR_DIST	1

After the parameters have been modified, exit the parameter tree window by choosing **OK**.

79. The SURF PICK menu appears. Select the **Model** option and **Done**. Click to the surface which we will machine as it is shown in Figure 27.

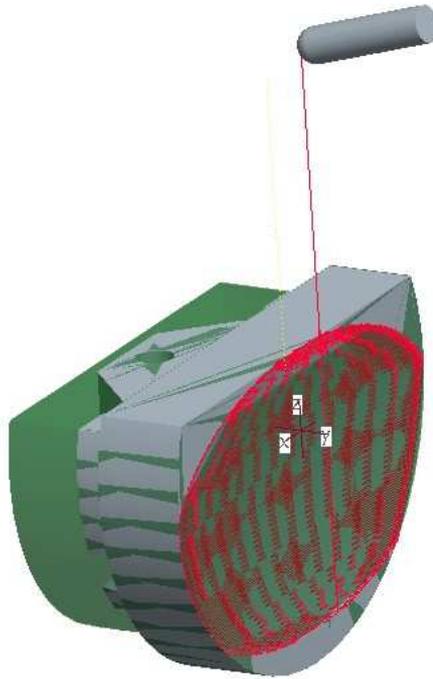


**Figure 27.** The surface to be machined

Select **Done Return** for the SELECT SRFS menu and again **Done Return** for the NCSEQ SURFS menu.

The Cut Definition window appears. Select **From surface isolines** option from the upper list. Pick to the selected surface on the surface list (surface Id = 464). The cut direction has to be horizontally. Verify it with the arrow icon (left, bottom in the Cut Definition window): pick to this icon while the yellow arrow is not horizontal (on the model). Choosing **Preview** we can see the tool slices. Close the Cut Definition window with **OK**.

The tool path definition is ready as it is shown in the Figure 28. We can play the simulation as we have done it in the previous NC Sequences. We close the new NC Sequence with **Done Seq**.



**Figure 28:** The tool path of the surface milling

### 3.5. MACHINING SIMULATION

Every NC sequence has been created for the second, milling operation. In the following steps we will create the machining simulation of this operation.

80. From the main **Edit** menu choose **CL Data**, and **Output**. The **SELECT FEAT** menu appears on the right side. Select **Operation, OP\_020, File, Done**. Enter the name of the CL Data file (e.g. **Milling1**). Close the popup menus.
81. From the main **Tools** menu choose **CL Data, Material Removal Simulation**, select the name of the CL Data file created a moment ago. Choose **Done**. In the VERICUT application we can see the full turning simulation.
82. Close the popup windows and menus and save the work.

## 4. THIRD OPERATION

During the third operation the rest material will be removed from the workpiece. It is assumed that the material is cut to the top plane of the model shown in Figure 29.

83. In order to start the second operation, choose the **Operation** from the **Steps** main menu. The *OPERATION SETUP* window should appear. Choose **File** and **New** in this window and we get a new, empty surface. The default operation name is "OP030". We can either accept the name or reenter it e.g. "side3".

84. NC machine:

To define the NC machine pick to its icon button on the right side of the window, in the line of the 'NC machine'. The *MACHINE TOOL SETUP* window should appear. Choose **File** and **New** in this window and we get a new, empty surface. We can accept or modify the Machine Name: the default name is "Mach03". For the Machine Type choose **MILL** on the popup list. To close the *MACHINE TOOL SETUP* window pick the **OK** button.

85. In the **Fixture Setup** option we can define the model of the fixture. An assembly containing the fixture can be created. This step is not essential to generate NC programs, so we not show it here in this example.

86. Machine Zero (machine coordinate system):

To define the coordinate system, pick the arrow icon near the Machine Zero in the *OPERATION SETUP* window.

Pick the coordinate system icon in the main window, on the right side vertical icon menu. The *COORDINATE SYSTEM* window should appear. Pick inside the field of the 'References' and show references for our coordinate system as follows:

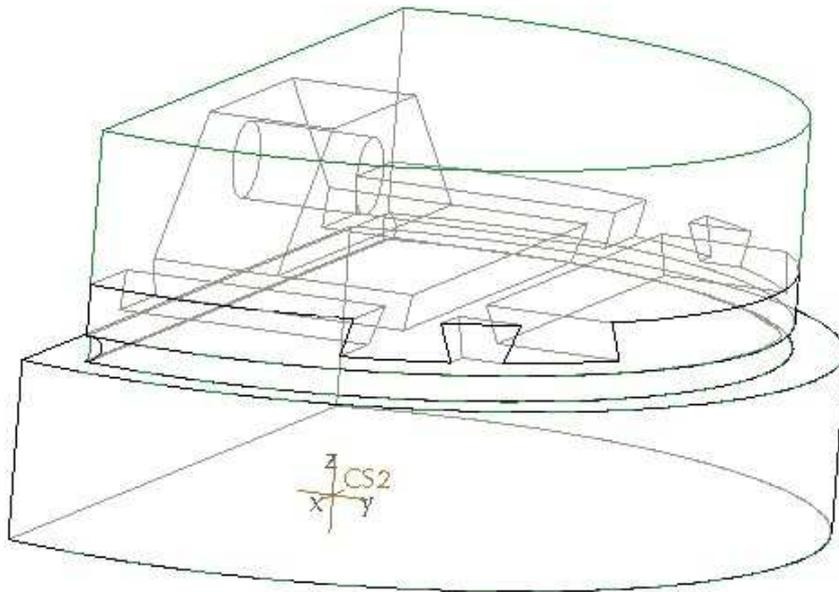
For example: select three perpendicular planes e.g. **DTM1**, **DTM2** and **DTM3** datum planes on the workpiece (pick on them on the model, press the **Ctrl** button on the keyboard and keep it press during the selections).

Second we have to define the orientation of the axes: in the *COORDINATE SYSTEM* window pick the **Orientation** panel. Define the Z-axis so, as it can be seen in Figure 29. (From the positive Z-axis the tool starts to cut during NC milling.)

Close the *COORDINATE SYSTEM* window by picking the **OK** button in it.

87. Retract plane

Pick the arrow icon near the „Surface” in the *OPERATION SETUP* window. The retract selection window should appear. Enter **40** to the Value which means the Z axis depth for the retraction of the tool. Close the window with **OK** and close the operation setup window with **OK**.



**Figure 29.**The machine coordinate system of the third operation

#### ***4.1. NC Sequence: Volume milling***

88. To start the sequence, choose the **Volume Rough** from the manufacturing icon menu (below the main upper horizontal menu).
89. In the SEQ SETUP menu, verify that **Tool, Parameters, Window** are checked, then choose **Done**.
90. The Tool Setup window appears. Select **File, Use previous** .From the tool list select the first tool of the Volume Milling sequence of the second operation and select **OK**. (We use the same tool which we defined in our previous Volume milling NC sequence.)

Close the tool Setup window with **OK**.

91. On the MFG PARAMS list enter the parameters as shown below:

TOOL_ID	FLATENDMLL
ROUGH_OPTION	ROUGH_ONLY
SCAN_TYPE	TYPE_3
TRIM_TO_WORKPIECE	FULL_TRIM
STEP_DEPTH	1
STEP_OVER	2
CUT_FEED	75
SPINDLE_SPEED	800
CLEAR_DIST	1

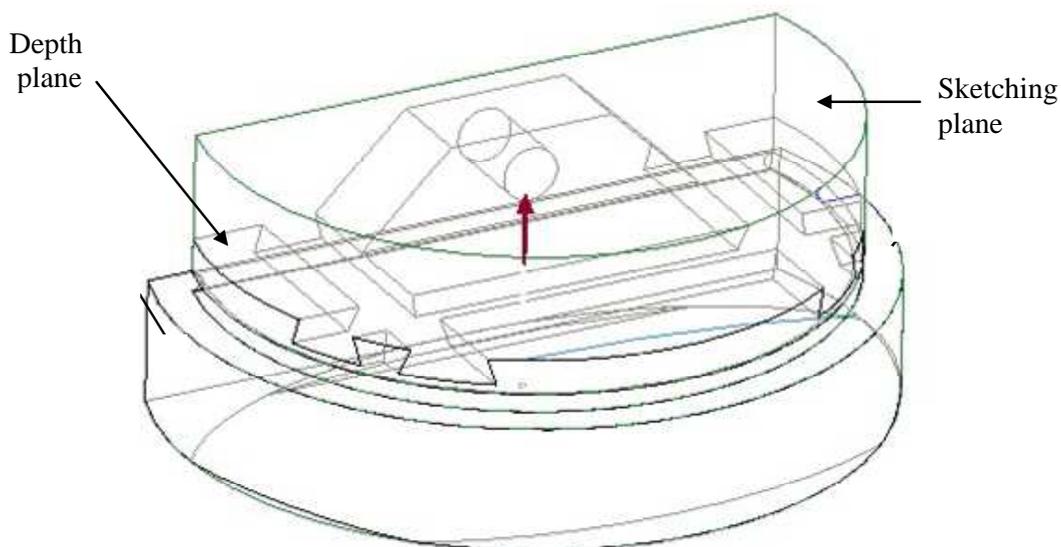
It is important to verify, that the parameter TRIM\_TO\_WORKPIECE has to be FULL\_TRIM at this NC Sequence.

Close the tool parameter list with **OK**.

92. For this NC sequence we will create a Mill Window. When the DEFINE WIND menu appears choose the **Mill Window** icon from the right side, vertical icon menu. The Define window plane icon menu appears below the upper horizontal main menu.

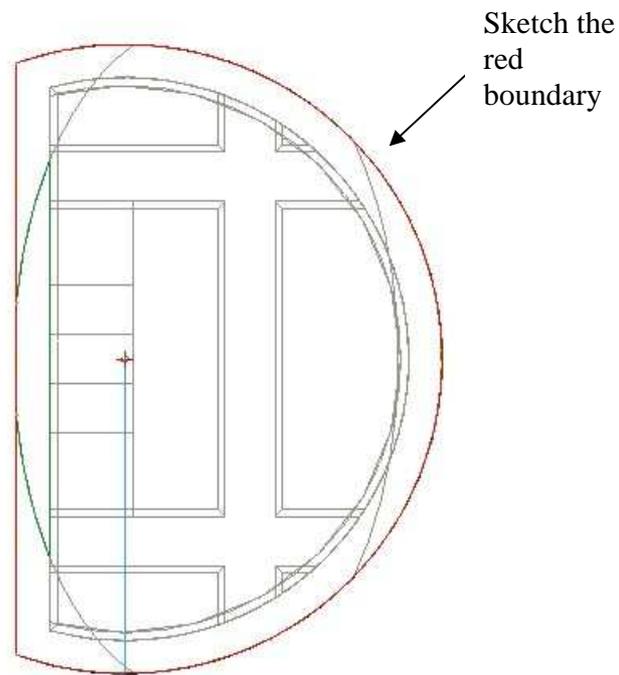
The Mill Window will be created by sketching an area, using the outside edges of the workpiece. The starting plane (sketching plane) will be the top plane of the workpiece model and the depth of the cut will be an internal plane of the model shown in Figure 30.

93. From the Define window plane icon menu select **Sketch window type** (the middle icon), since we will sketch the region which will be machined.
94. Open the **Placement** panel (pick to it). A window plane has to be selected. The window plane (sketching plane) has to be the top plane of the workpiece model. Pick to it, Fig. 30.



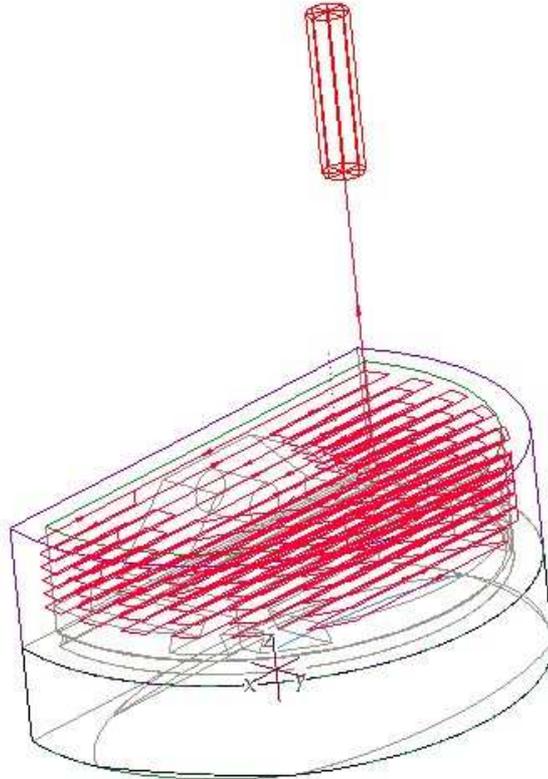
**Figure 30.** Sketching planes and the depth of the Mill Window

95. We have to define the area on the window plane, since we have to define the region which will be machined. Pick to the **Define an internal sketch** icon on the Define window plane icon menu. The SKETCH window appears. Pick **Sketch** in the SKETCH window. Select the appropriate references for the sketching and use the boundary edges of the workpiece model for the sketching as it is shown in Figure 31. Close the sketcher.



**Figure 31.** For the extrude selection

96. In the end the depth of the Mill window has to be defined. Open the **Depth** panel on the Define window plane icon menu and sign the **Specify the depth**. From the Depth options select the **To selected** option and pick to the internal plane of the manufacturing model as it is shown in Figure 30. Close the Define window plane icon menu with the green ✓ icon.
  
97. The NC sequence has now been set up. Choose **Play Path, Done** to view a tool path similar to what is shown in figure 32. Choose **Done Seq** from the NC SEQUENCES menu and save the work.



**Figure 32.** Tool path of the VOLUME MILLING

#### ***4.2. NC Sequence: 1. Trajectory milling***

98. To start the NC sequence choose **Trajectory** and **Done**. For the number of the machining axes accept the default **3 Axis** option and close the MACH AXES menu with **Done**. The NC SEQUENCE popup menu appears on the right side.
99. In the SEQ SETUP menu, make sure **Tool, Parameters** and **Tool Motions** are checked, then choose **Done**.
100. The Tool Setup window should appear. We use the same tool as was used in the last NC Sequence: from the tool list select the first tool named **FLATENDMILL** and select **OK**.
101. On the MFG parameter list enter the parameters as shown below:

CUT_FEED	75
SPINDLE_SPEED	800
CLEAR_DIST	1

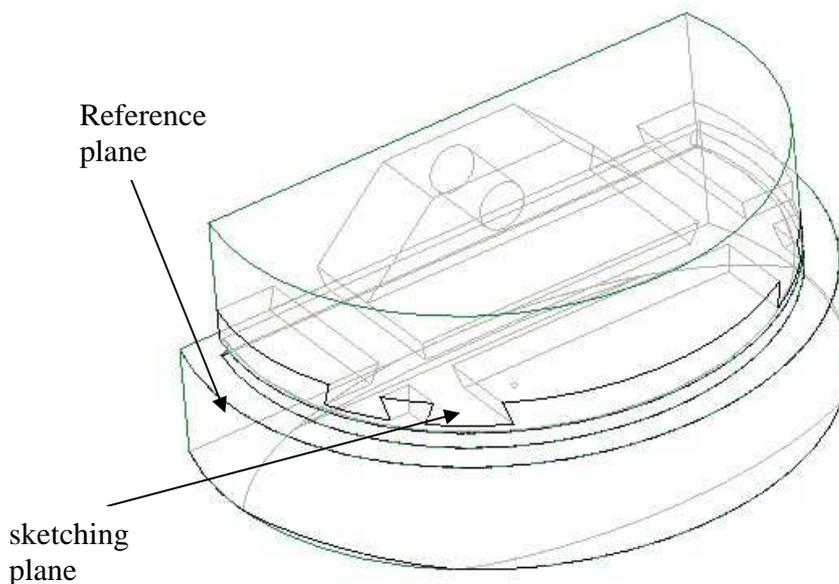
Close the MFG PARAMS list with **OK**.

102. The Tool Motions window appears. Choose **Insert** and the Curve Trajectory Setup window appears. Now we have to show the curve of the tool path, we will sketch it here. Select the Sketch tool from the right side, vertical main icon menu. The sketch window appears. For the Sketching plane select the bottom plane of the grooves of the reference model and for the reference plane select a perpendicular plane, Figure 33.

Select appropriate references for the sketching and sketch the line net according to Figure 34.

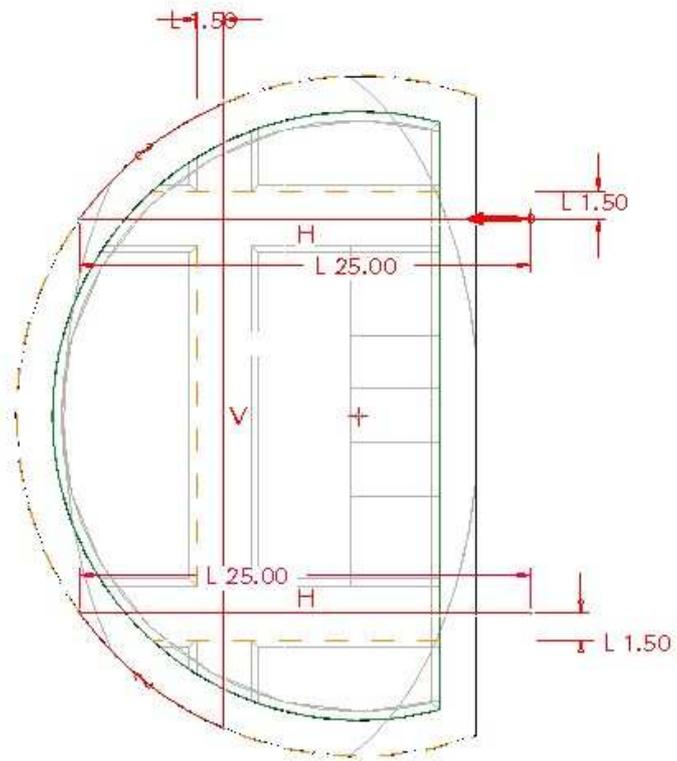
Close the sketcher and activate the Curve Trajectory Setup window if it is needed. (For the activation pick to the arrow icon bottom in the window.) Select **None** for Tool Offset, the trajectory is not needed to move here. Close the Curve Trajectory Setup window with **OK**.

Close the Tool Motions window with **OK**.

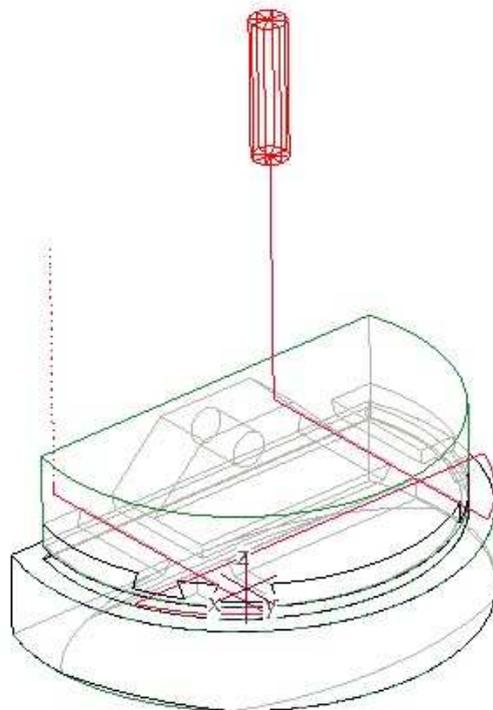


**Figure 33.** Sketching and reference plane for sketching

103. The NC sequence has now been set up. Choose **Play Path, Done** to view a path similar what is shown in figure 35. Choose **Done Seq** from the NC SEQUENCES menu and save the work.



**Figure 34.**The red line net is the trajectory



**Figure 35.** The trajectory tool path

### 4.3. NC Sequence: Surface milling

104. To start the NC sequence choose **Surface Milling** and **Done**. The NC SEQUENCE popup menu appears on the right side.
105. In the SEQ SETUP menu, make sure that **Tool, Parameters, Surfaces, Define cut** are checked, then choose **Done**.
106. The Tool Setup window should appear. Define a new tool with the following parameters:

TOOL_ID	BALLMILL2
TOOL_TYPE	BALL MILL
ON THE GEOMETRY PANEL:	
CUTTER_DIAMETER	2
LENGTH	12
ON THE SETTINGSs PANEL:	
TOOL_NUMBER	9

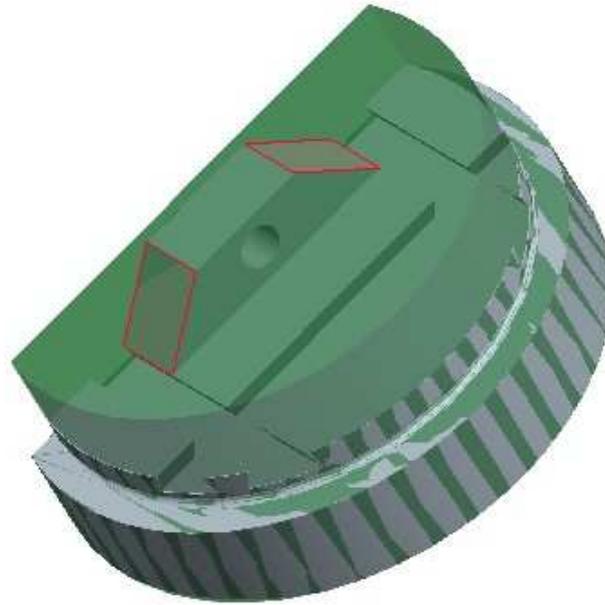
Close the Tool Setup window with **Apply** and **OK**.

107. From the MFG PARAMS menu, choose **Set**. Enter the parameters as shown below:

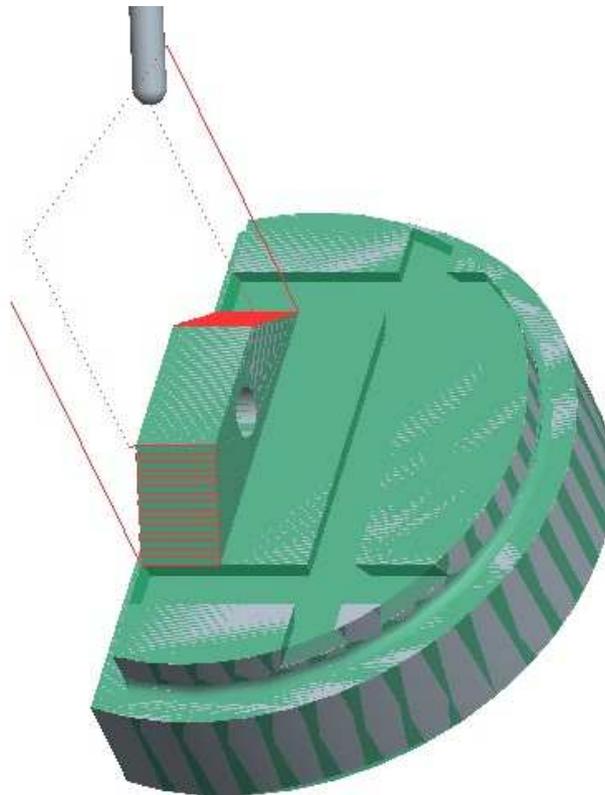
CUT_FEED	75
STEP_OVER	0.5
SPINDLE_SPEED	800
CLEAR_DIST	1

After the parameters have been modified, exit the Param tree window.

108. The **Surf pick** menu appears. Select the **Model** option and **Done**. Pick to the surface which we will machine as it is shown in figure 36. Close the NCSEQ SURF menu with **Done Return**.
109. The Cut Definition window appears. Select **From surface isolines** option from the upper list. Pick to the selected surfaces on the surface list. The cut direction has to be horizontally. Verify it with the arrow icon (left, bottom in the Cut Definition window): pick to this icon while the yellow arrow is not horizontal (on the model). Choosing **Preview** we can see the tool slices. Close the Cut Definition window with **OK**. Close the Cut Definition window with **OK**.
110. The tool path definition is ready as it is shown in the F igure 37. We can play the simulation as we have done it in the previous NC Sequences. We close the new NC Sequence with **Done Seq**.



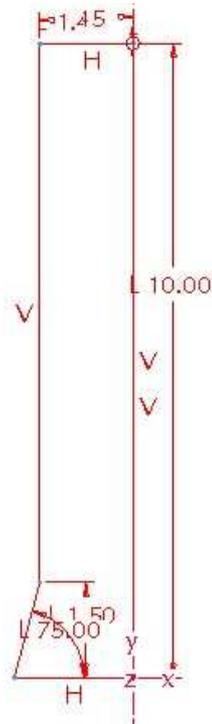
**Figure 36.** The surfaces to be machined are signed with red colour



**Figure 37.** The tool path of the surface milling

#### 4.4. NC Sequence: 2.Trajectory milling

111. To start the sequence choose the **Trajectory** icon on the manufacturing icon menu below the main horizontal icon menu, and **Done**. For the number of the machining axes accept the default **3 Axis** option and close the MACH AXES menu with **Done**. The NC SEQUENCE popup menu appears on the right side.
112. In the SEQ SETUP menu, make sure **Tool, Parameters** and **Tool Motions** are checked then choose **Done**.
113. The Tool Setup window should appear. We define a new tool but this new tool type can not be found in the tool library of the Pro/Engineer so we need to create the tool model. Select **File, New** in the Tool Setup window. Choose the tool type: **MILLING** in the General panel. Select **Sketch** from Edit menu in the Tool Setup window and select **Sketcher**. The system get us in the Sketcher, we can start to create our tool sketch as we can see in Figure 38. It is important that a symmetry axis is drawn vertically on the right side of the sketch, and a coordinate system is defined at the right bottom corner of the sketch. (The tool for defining of the coordinate system can be found in the Sketch main menu.)



**Figure 38.** The sketch of the new tool

Close the sketcher with the ✓ icon .

Set the tooling parameters as follows:

TOOL\_ID

TRAJECTORY

ON THE SETTING PANEL:  
TOOL\_NUMBER 9

Close the Tool Setup window with **Apply** and **OK**.

114. From the MFG PARAMS menu, choose **Set**. Enter the parameters as shown below:

CUT_FEED	75
SPINDLE_SPEED	800
CLEAR_DIST	1

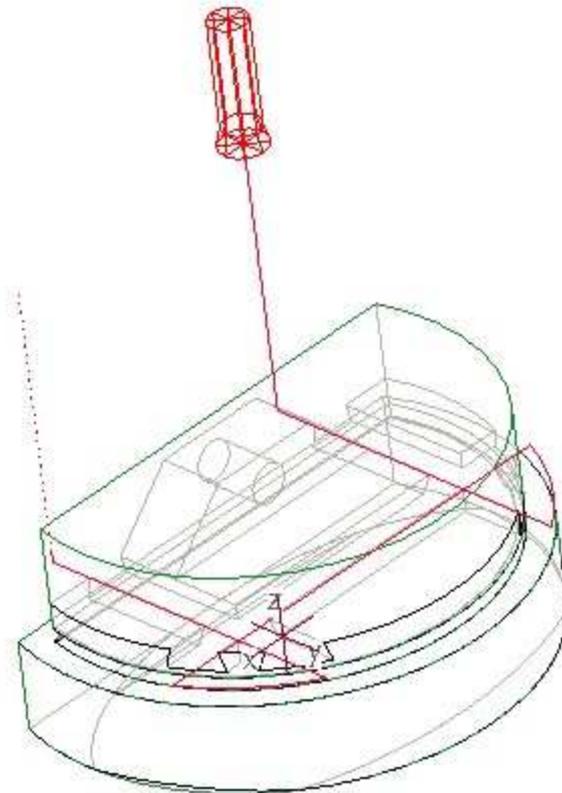
After the parameters have been modified, exit the Param tree window by choosing **File, Exit** and close the MFG PARAMS menu by choosing **Done**.

115. The Tool Motions window appears. Choose **Insert** and the Curve Trajectory Setup window appears. Now we have to show the curve of the tool path. On the manufacturing model pick to the trajectory defined in step of 105, Figure 33-34.

Close the Curve Trajectory Setup window with the green ✓ icon.

Close the Tool Motions window with **OK**.

116. The NC sequence has now been set up. Choose **Play Path, Done** to view a path similar what is shown in Figure 39. Choose **Done Seq** from the NC SEQUENCES menu and save the work.



**Figure 39.** Tool path of the trajectory

## 4.5. MACHINING SIMULATION

Every NC sequence has been created for the second, milling operation. In the following steps we will create the machining simulation of this operation.

117. From the main **Edit** menu choose **CL Data**, and **Output**. The SELECT FEAT menu appears on the right side. Select **Operation, OP\_030, File, Done**. Enter the name of the CL Data file (e.g. **Milling2**). Close the popup menus.
118. From the main **Tools** menu choose **CL Data, Material Removal Simulation**, select the name of the CL Data file created a moment ago. Choose **Done**. In the VERICUT application we can see the full turning simulation.
119. Close the popup windows and menus and save the work.