

MINISTRY OF EDUCATION AND SCIENCE OF UKRAINE
National Aerospace University
«Kharkiv Aviation Institute»

O. Shorinov, Yu. Neveshkin, M. Kalinichenko

**DEVELOPMENT OF THE CAM PROGRAM FOR CNC MACHINE
MANUFACTURING OF THE PART**

Training Manual

Kharkiv «KhAI» 2024

Наведено приклад проєктування операції механічного оброблення деталі фрезеруванням та отримання керуючої програми в програмному середовищі SolidCAM для її виготовлення на верстаті з числовим програмним керуванням. Подано основні налаштування в SolidCAM з описом інтерфейсу програми. Розглянуто послідовність дій при обробленні деталі фрезеруванням – від створення CAM-проєкту до отримання керуючої програми. Показано алгоритм дій при виборі поверхні для оброблення, налаштувань технології, інструменту та режимів фрезерування, візуалізації траєкторій переміщень інструменту тощо.

Для студентів механічних спеціальностей.

Reviewers: Yu. V. Dudukalov, PhD, Associate Professor,
V. F. Sorokin, DrS, Professor

Shorinov, O.

S55 Development of the CAM program for CNC machine manufacturing of the part [Electronic resource] : Training Manual / O. Shorinov, Yu. Neveshkin, M. Kalinichenko. – Kharkiv : National Aerospace University «Kharkiv Aviation Institute», 2024. – 57 p.

An example of the development of the operation of mechanical processing of a part by milling and generating a numerical control program in the SolidCAM software for its manufacture on a CNC machine is presented. Basic settings in SolidCAM are given with a description of the program interface. The sequence of actions during the milling of the part, from the creation of the CAM project to the generation of the numerical control program, is given. The algorithm of actions while choosing a geometry for milling, technology settings, tool and milling modes, visualization of trajectories of tool movements, etc., is outlined.

For students of mechanical specialties.

Figs 85. Bibliogr.: 2 references

UDC 004.896:621.9.06-529:621.914(076.5)

© Shorinov O., Neveshkin Yu.,
Kalinichenko M., 2024
© National Aerospace University
«Kharkiv Aviation Institute», 2024

INTRODUCTION

As the leading CAD/CAM solution, SolidCAM provides seamless single-window integration and full associativity with the SOLIDWORKS design model. All machining operations are defined, calculated, and verified without leaving the SOLIDWORKS window. With SolidCAM, you can convert SOLIDWORKS parts to GCode for any CNC machine.

From modeling to manufacturing, geometries are defined in operation (e.g., profile, pocket, drill, etc.) to determine where the model will be processed. The tools required for cutting can be assigned and selected from libraries. Familiar sources of error, such as missing data or entering incorrect coordinates, can be avoided by seamlessly transferring values from SOLIDWORKS. Any changes to the SOLIDWORKS design model are automatically detected, and toolpaths are corrected. Even the costly repair of surface defects can be avoided thanks to CAD data integration and transfer.

Other CAD systems can also be used for modeling. Once created, the geometry of your 3D models can be imported for programming into the fully functional CAD/CAM system offered by SolidCAM.

After programming, you can simulate the tool path and check for possible collisions with the workpiece in SolidCAM. The work holding is included in the collision check. Programming errors can be detected and corrected at an early stage.

During the simulation, the material is also detected and can be displayed. SolidCAM allows you to see the path of the wireframe tool on the model, the cutting tool as it moves through the solid material of the workpiece, and machining simulations using the kinematics of the CNC machine.

Once the toolpath has been successfully modeled, SolidCAM allows you to generate GCode specific to your CNC controller (e.g., Siemens or Heidenhain). The programmer can continue to work in SolidCAM throughout the entire process. For production processes, in general, this offers great flexibility.

By selecting a postprocessor in the CAM system, the generated GCode can produce the workpiece on different machines with different controls.

The final GCode can be sent to the machine via a USB drive or network. The machine operator prepares the machine and the workpiece. The tools are designed according to the Tool Table. When everything is set up, you can start manufacturing.

1. MANUFACTURING WITH SOLIDCAM

1.1. Programming in SolidCAM

SolidCAM 2.5D milling technologies are used in the following tutorial to complete CNC programs.

Programming in SolidCAM includes several steps [1]:

Step 1: Open the part in SOLIDWORKS, then start a New Milling CAM-Part in SolidCAM.

Step 2: Determine where the CAM data will be stored and create the CAM-Part.

Step 3: Select the CNC-Machine controller to start the CAM-Part definition.

Step 4: Define the Coordinate System (origin for all machining operations).

Step 5: Define the Stock model and the Target model.

Step 6: Add the operations required to machine the part.

Step 7: Generate the GCode.

Step 8: Send the GCode to the machine.

Depending on your CAM settings, you can skip steps 2, 3, 4, and 5. However, it is recommended that you familiarize yourself with the CAM-Part definition before SolidCAM automates the process for you.

In SolidCAM settings, make sure the following CAM-Part options are disabled [1]:

1. Auto select project location and name... (Fig. 1).

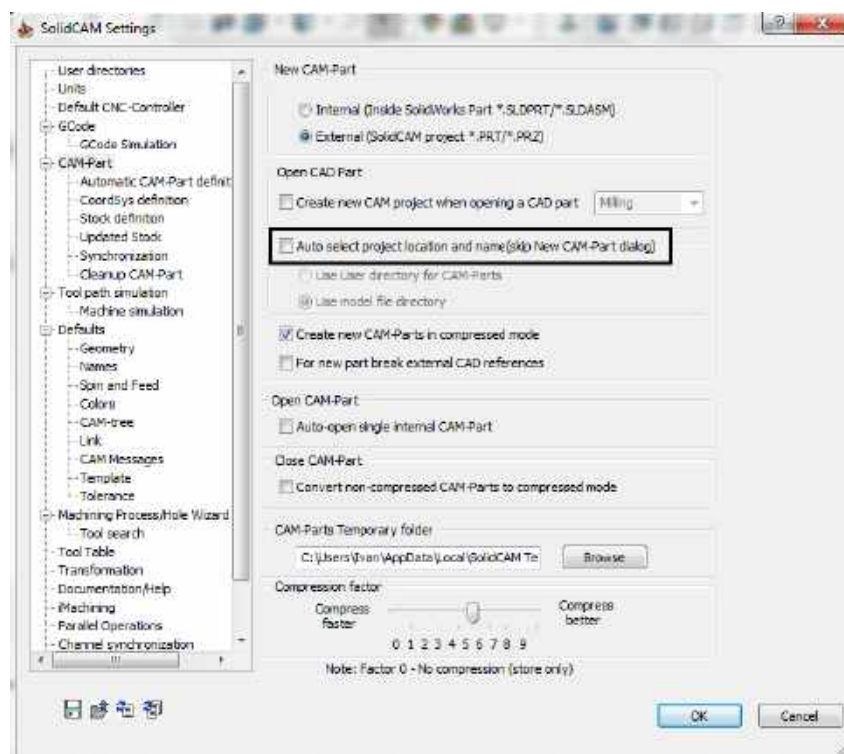


Fig. 1. SolidCAM Settings

2. Parameters listed in the settings for automatic CAM-Part definition (Fig. 2):
- *Definition of the CNC-controller;*
 - *CoordSys definition;*
 - *Create Machine Setup;*
 - *Definition of the Stock;*
 - *Definition of the Target.*

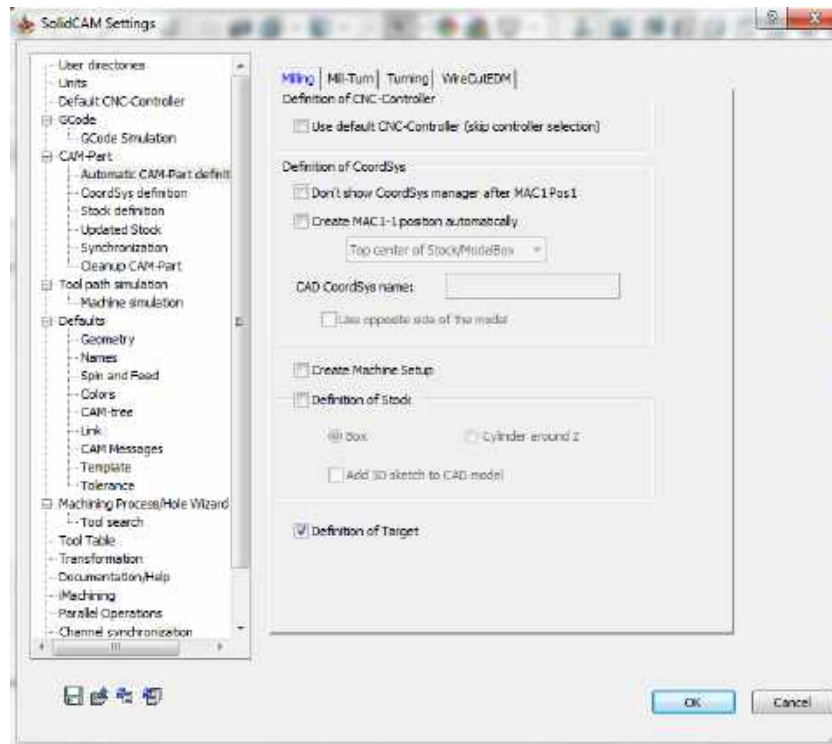


Fig. 2. Setting up the Automatic Definition of a CAM-Part

In the SolidCAM settings, turn off the *Create new CAM-Parts in compressed mode* option (Fig. 3). In the following tutorial, you will need to access the CAM-Part folders.

1.2. Creating and Defining a CAM-Part

Open the part in SOLIDWORKS and start a New Milling CAM-Part in SolidCAM (Fig. 4).

Open the already developed 3D model of the part (the part drawing is presented in the Appendix).

From the SOLIDWORKS menu bar, click *Tools > SolidCAM > New > Milling*.

You can also use the available controls of SolidCAM in SOLIDWORKS (SolidCAM CommandManager tabs or SolidCAM toolbar).

The following four steps will create and define the CAM-Part.

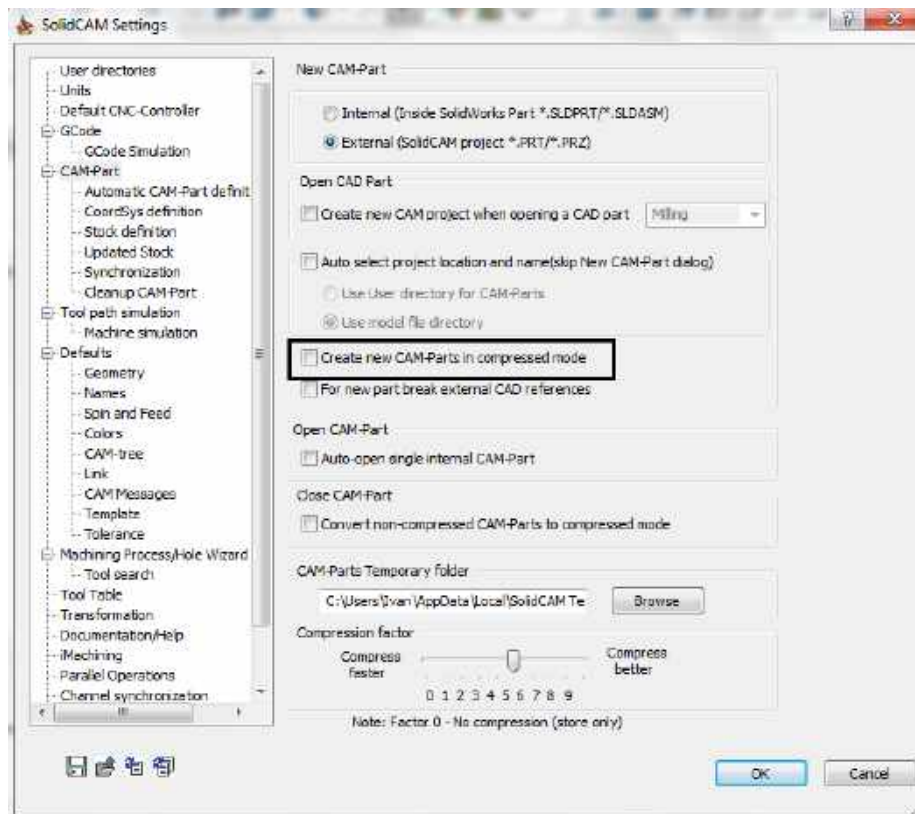


Fig. 3. Setting up a New CAM-Part

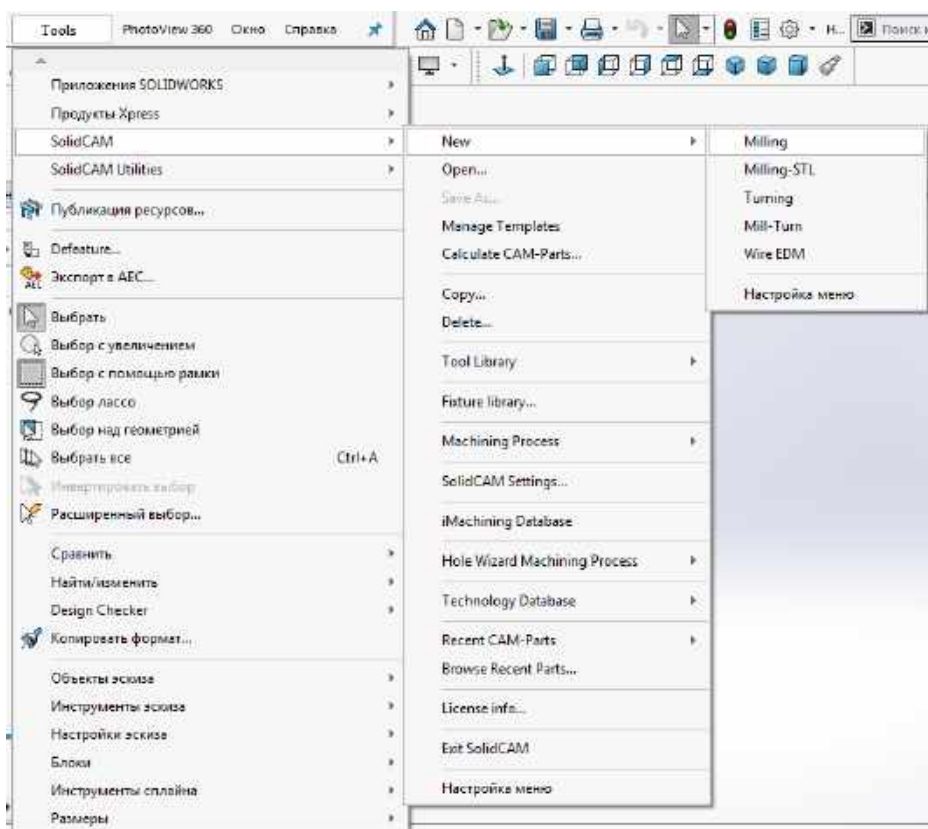


Fig. 4. Creating a New CAM Operation

1.2.1. Defining the Directory for Saving the CAM-Part

Determine where the CAM data will be stored and create the CAM-Part. Define the following (Fig. 5):

- External or internal mode to create a CAM-Part. All parts will be created using *External* mode. All CAM data is stored in the SolidCAM project;
- The CAM-Part name defines the new name of the SolidCAM file (by default, the SOLIDWORKS part name);
- The directory defines the location for storing the CAM part (*Use Model file directory* is the default location).
- The model name shows the path to the SOLIDWORKS part.
- *Metric* should be used for Unit of measurement.

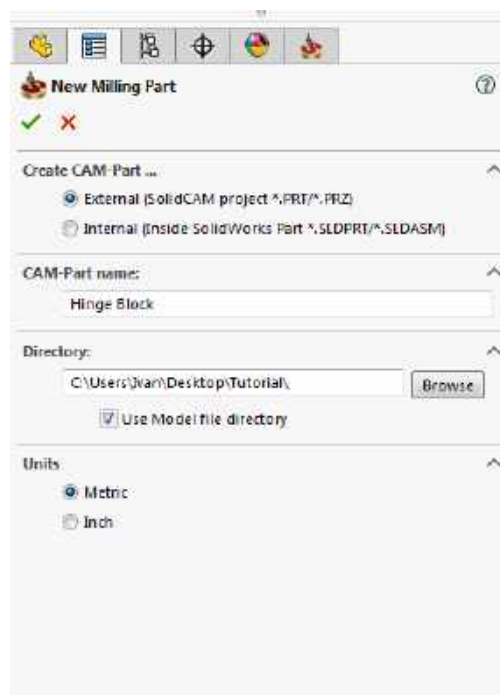


Fig. 5. Setting up a New Milling Part

Click *OK* to confirm your entries .

1.2.2. Selecting the CNC-Machine Controller

After creating a CAM-Part, the Milling Part Data dialog box appears. This dialog box allows you to complete the definition of the CAM-Part:

- Select the CNC-Machine Controller (Fig. 6);
- Click the drop-down menu to display a list of post-processors that are currently installed on the system;
- For this tutorial, the machining will be performed on a 3-Axis milling machine. Select the *gMilling_3X* post-processor from the list.

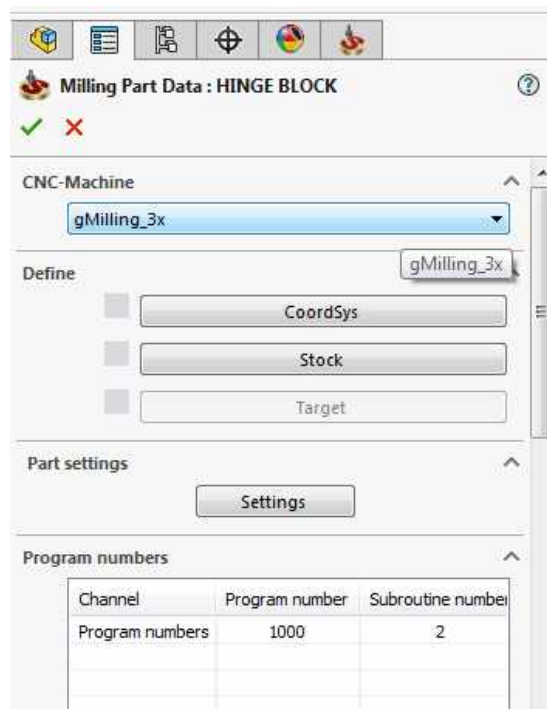


Fig. 6. Selection of a CNC-Machine Controller

1.2.3. Defining the Machine Coordinate System

SolidCAM changes the model orientation to an isometric CAM view. This orientation is suitable for machining on a CNC-Machine (with the Z-axis pointing vertically upwards (Fig. 7) [2]). Use the SolidCAM Views in the tabs to see the SolidCAM Coordinate System. These views are related to the orientation of the selected CoordSys position.

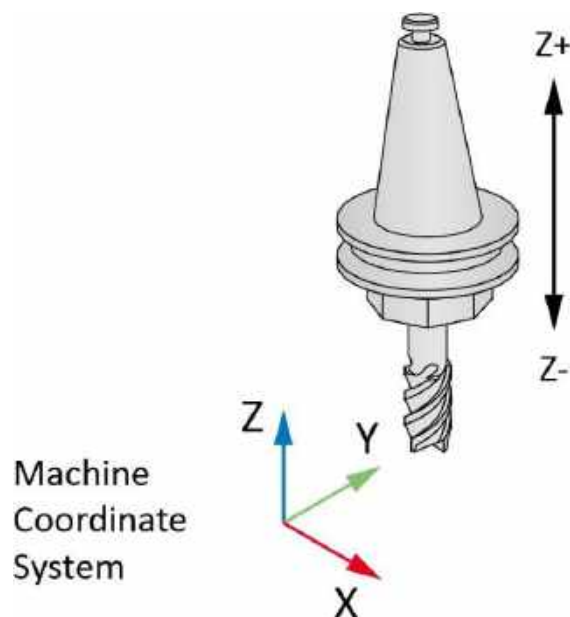


Fig. 7. Machine Coordinate System

To complete the CAM-Part Definition, you must define at least one Machine Coordinate System.

The Machine Coordinate System determines the origin of all CAM-Part operations. This corresponds to the built-in functions of the Controller (G54, for example).

The Z-direction of the Machine Coordinate System is parallel to the tool rotation axis. In SolidCAM, the tool is approached from the positive direction of the Z-axis (as on a vertical CNC machine).

For 3-axis milling machines, each Machine Coordinate System represents a separate clamping. If you need to machine a workpiece from different sides, use multiple machine coordinate systems with the Z-axis oriented normally (perpendicular) to the sides to be machined.

To start defining the coordinate system, click the *CoordSys* button to open the CoordSys dialog box (Fig. 8).

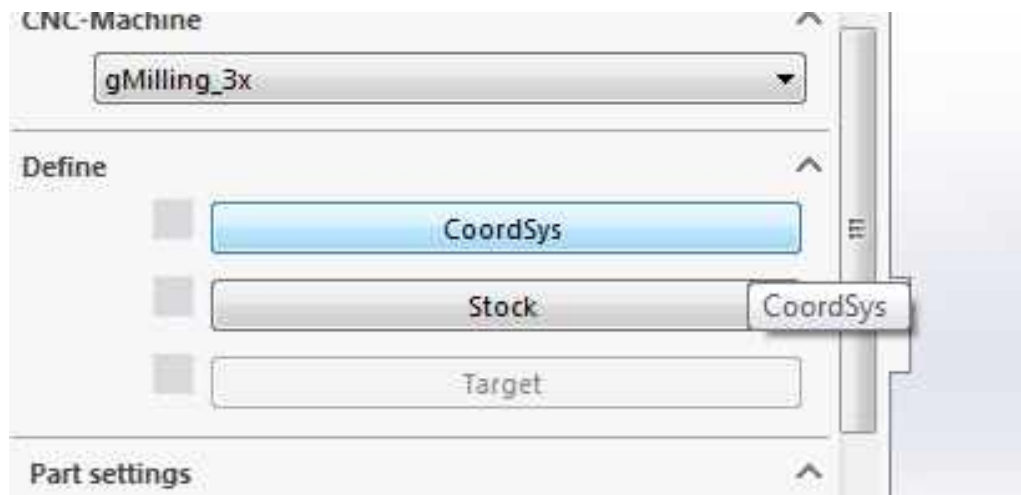


Fig. 8. Setting up the Coordinate System

The Coordinate System can be defined directly on the solid model. You can determine the starting position and orientation of the axes by selecting model faces, vertices, edges, or predefined SOLIDWORKS Coordinate Systems.

The machining geometries will also be selected directly on the solid model. Now, we start the defining of the Coordinate System.

Each newly created Coordinate System is automatically assigned a new sequence number.

In the Definition options box, click *Select face* and select from the drop-down menu *Center of revolution face* (Fig. 9).

Select a cylindrical hole surface, as shown in Fig. 10. The Z-axis of the coordinate system is parallel to the countersink's axis of rotation, and the origin is centered (X0, Y0) on top of the workpiece. The outer contour is machined in the first setup, and the countersink is drilled.

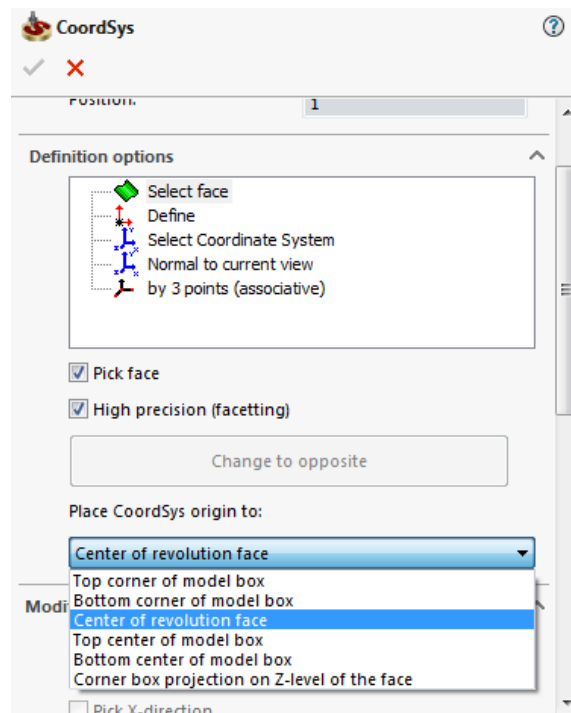


Fig. 9. Setting up CoordSys

To confirm the Coordinate System definition, click **OK** .

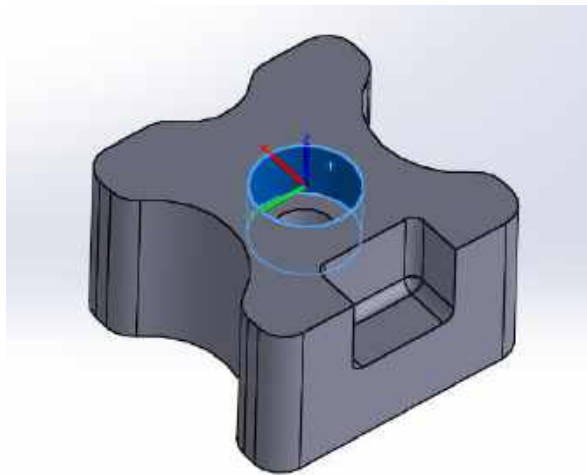


Fig. 10. Selection of Coordinate System on the Part

The CoordSys Data is displayed (Fig. 11). This allows you to define the default Machining levels, such as Tool start level, Clearance level, Part upper and lower levels, and Tool Z-level. The following parameters can be defined:

1. *MAC Number* – displays the number of the Coordinate System on the CNC-Machine. The default value is 1 (G54). If you use a different number, the GCode file contains a G-function that tells the machine to use the specified number stored in the controller;

1 — MAC Number: 1

2 — Position: 1

☐ Create planar surface at Part Lower level

Edit CoordSys

3 — Tool start level: 120

4 — Clearance level: 25

5 — Part Upper level: 0

6 — Part Lower level: -30

7 — Tool Z-level: 250

Levels: Radial

Tool start level: 129

Clearance level: 34

Part Upper level: 9

Part Lower level: 0

Levels: Rear

Translation Data

Plane

Fig. 11. Dialog Box Part Zero Manager

- 2. *Position* – displays the ordinal number of the Coordinate System;
- 3. *Tool start level* – determines the Z-level at which the tool starts working;
- 4. *Clearance level* – defines the Z-level to which the tool quickly moves from one operation to another (if the tool does not change);
- 5. *Part Upper level* – determines the height of the top surface of the workpiece that is milled;
- 6. *Part Lower level* – defines the bottom level of the surface of the workpiece that is milled;
- 7. *Tool Z-level* – determines the height the tool moves before rotating the 4/5 axes to avoid collisions between the tool and the workpiece. This level is linked to the CoordSys position, and you must ensure it does not exceed the machine limit. You should send the tool to a control point or a point associated with a control point.

Click OK  to accept the default machining levels and close the CoordSys Data dialog box.

The Milling Part Data dialog box (Fig. 12) is displayed in the SOLIDWORKS PropertyManager again. The defined Machine Coordinate System is shown.

Click **OK**  to confirm the Machine Coordinate System and close the Milling Part Data dialog box.

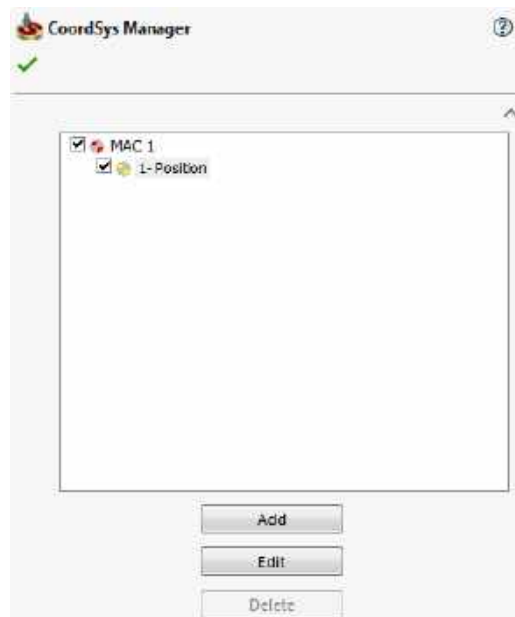


Fig. 12. CoordSys Manager Dialog Box

1.2.4. Defining the Stock and Target Models

Click the Stock button in the Milling Part Data dialog box (Fig. 13, a). The Stock dialog box (Fig. 13, b) is displayed, which allows you to select the method of defining the Stock.

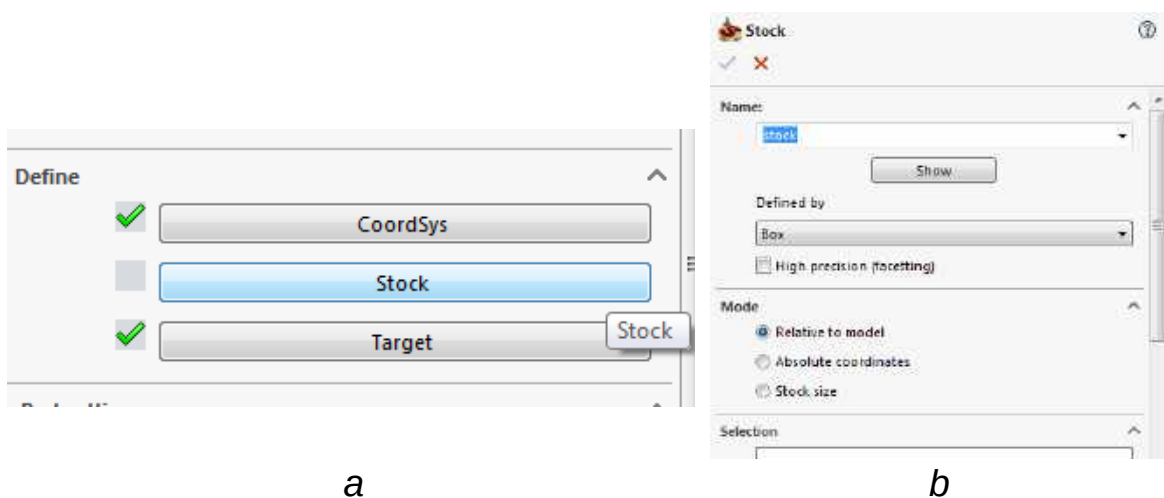


Fig. 13. The Milling Part Data Dialog Box:
a – Stock button; b – Stock dialog box

In the Defined by a drop-down menu (Fig. 14), you can select the following methods for defining the Stock model.

1. *Box* – this option allows you to define a Stock model as a box surrounding the selected target model;

2. *Extruded boundary* – this option allows you to define a standard model as a closed chain of wireframe geometry using one of the model sketches in the XY plane. The chain is stretched in the Z-axis to define the Stock model;

3. *3D Model* – this option allows you to define a Stock model by selecting a 3D solid model, which can be helpful when machining castings;

4. *Cylinder* – this option allows you to define the Stock model as a cylinder (or tube) surrounding the selected target model;

5. *STL* – this option allows you to define the Stock model based on an existing STL file.

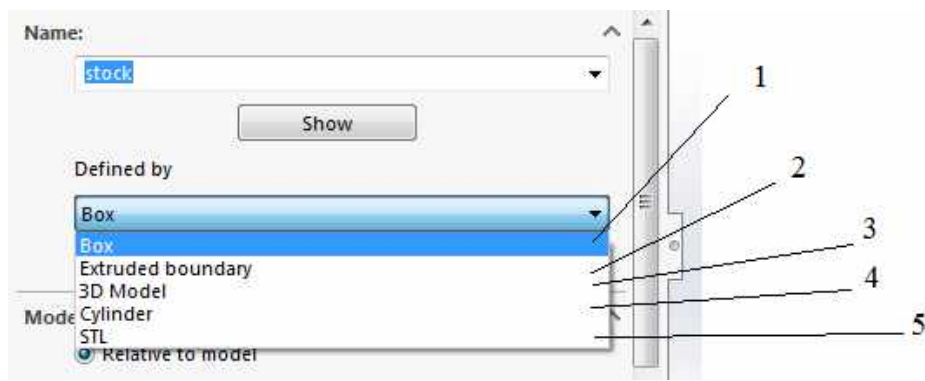


Fig. 14. Methods for Defining the Stock Model

Select *Box* from the list.

Optionally, you can define offsets that are relative to the model. In the Expand box area (Fig. 15), specify an offset of 1 mm for the X+, X-, Y+, Y-, and Z+ directions. Enter a Z-offset of 6 mm.

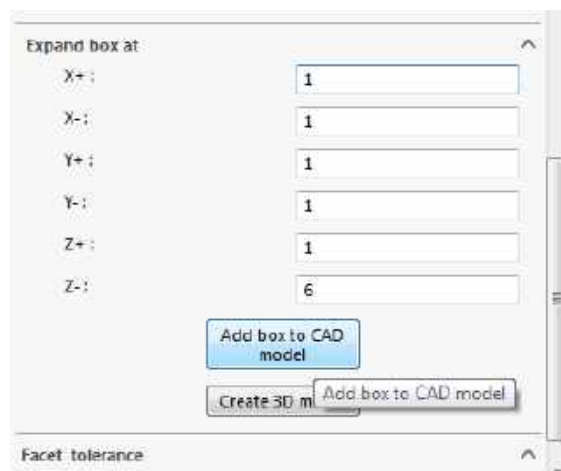


Fig. 15. Side Offsets

Now select *Solid* in the Graphics Area. SolidCAM automatically creates a box surrounding the Target model that defines the geometry of the Stock model.

When you click *Add box to CAD model*, the surrounding box is displayed as a 3D sketch. Click OK ☒ to confirm the Stock model definition.

The Milling Part Data dialog box is displayed again.

SolidCAM allows you to define a Target model, which is the final shape of the CAM-Part after machining. The Target model is used for group checking in the SolidVerify simulation.

Click the Target button in the Milling Part Data dialog box (Fig. 16).

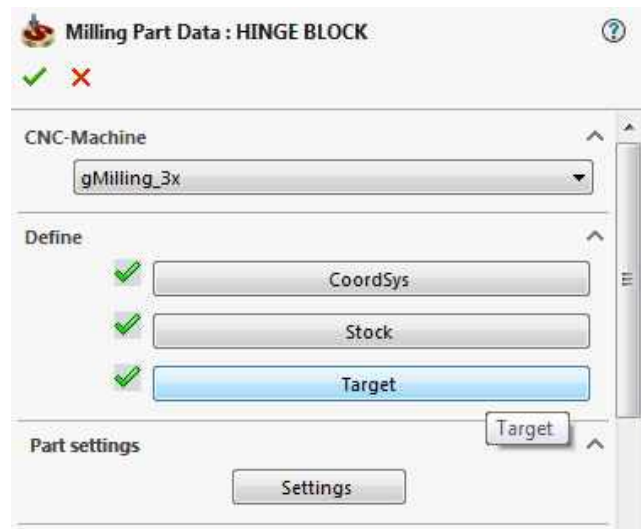


Fig. 16. Milling Part Data Dialog Box

The Target model dialog box is displayed, enabling you to define the Target model using the 3D Model selection method. Depending on the CAM Settings, the target model may already be defined by default. The recommended value for the Facet tolerance parameter is 0.01 mm (Fig. 17).



Fig. 17. Facet Tolerance

You can use a target model to define the geometry in some operations.

The Facet tolerance parameter defines the triangulation accuracy of the Target model. This parameter can also be defined for the workpiece model and clamping devices. Triangulated models are used later in SolidVerify simulations. The smaller the tolerance, the better the visualization quality of the Target model.

Click OK ☒ to confirm the Target model definition.

1.3. Saving the CAM Part

Confirm the CAM-Part Definition by clicking OK .

The Milling Part Data dialog box closes (Fig. 18), and SolidCAM Manager is displayed. The CAM-Part is saved and stored in the specified directory. At this point, the definition of the CAM-Part is finished.

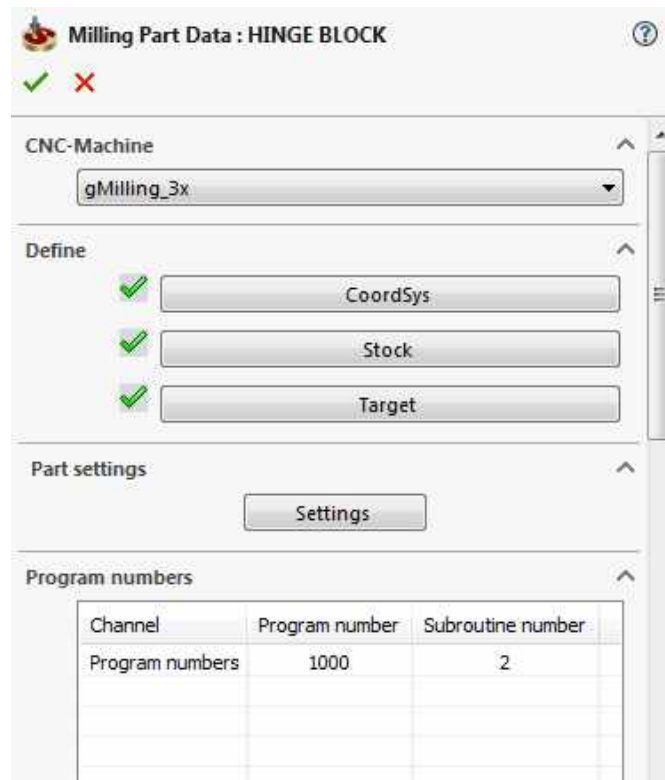


Fig. 18. The Milling Part Data Dialog Box

SolidCAM creates and works with a copy of the original SOLIDWORKS model. Any changes to the original model are automatically detected, and SolidCAM will prompt you to accept the changes in the CAM-Part. The changes will be applied to the copied model (DesignModel.SLDPRT), but you must also synchronize so that the associated CAM data is automatically adjusted.

SolidCAM uses the name of the SOLIDWORKS part to name the CAM-Part but with all capitalization. A SolidCAM part uses the file extension **.prt* (**.prz* if compressed mode).

SolidCAM also creates an assembly with the same name as the CAM-Part, with a file extension of **.SLDASM*. SolidCAM allows you to develop auxiliary geometries (e.g., sketches) without modifying the original model by using the assembly environment of the SOLIDWORKS. You can add components to the assembly file, such as a standard model, CNC machine table, fixtures, and other tooling elements.

1.4. SolidCAM User Interface

The SolidCAM Manager tree is the main function of the SolidCAM interface (Fig. 19). It is located on the left side of the SOLIDWORKS window (same as FeatureManager, PropertyManager, or ConfigurationManager). All information about the CAM-Part is displayed in SolidCAM Manager. SolidCAM always works with a copy of the original SOLIDWORKS model. This ensures that the original model cannot be easily modified.

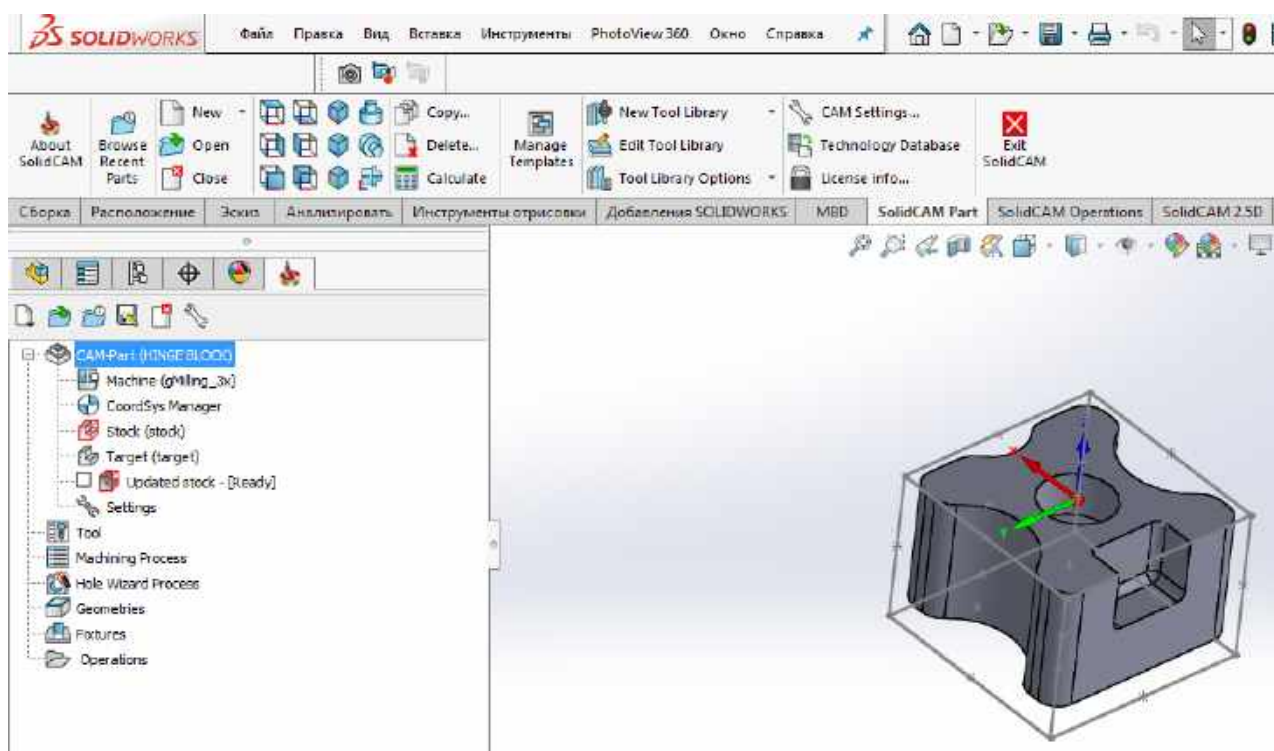


Fig. 19. SolidCAM Interface

SolidCAM performs a date/time comparison to recognize any later changes. If changes have been made to the original model, the message shown on the right will appear when you open the CAM-Part. If you click Yes, the changes are reflected in the copy, and you will need to synchronize and recalculate existing machining operations.

The SolidCAM Manager contains the following main elements (Fig. 20).

They represent the project tree. Each branch can be accessed after the project is created and is carried out by double-clicking on the element, or by right-clicking to access additional menus.

In the following sections, we will consider working with different branches using a specific example.

The main processing transitions are created and placed in the Operation header.

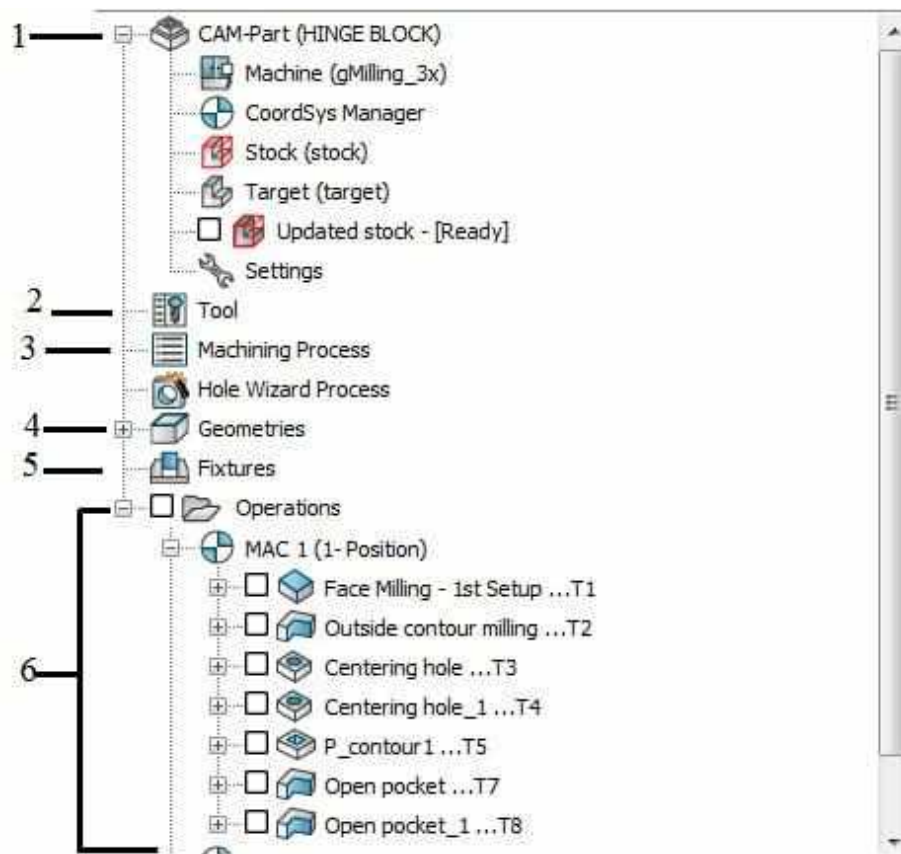


Fig. 20. SolidCAM Manager:
 1 – CAM-Part header; 2 – ToolKit header; 3 – Machining Process header;
 4 – Geometry header; 5 – Fixtures header; 6 – Operation header

2. DEVELOPMENT OF MILLING OPERATIONS

2.1. Adding a Face Milling Operation (1st Setup)

In the SolidCAM Manager, correct mouse-click the *Operations* header and select *Face...* from the *Add Milling Operation* submenu (Fig. 21).

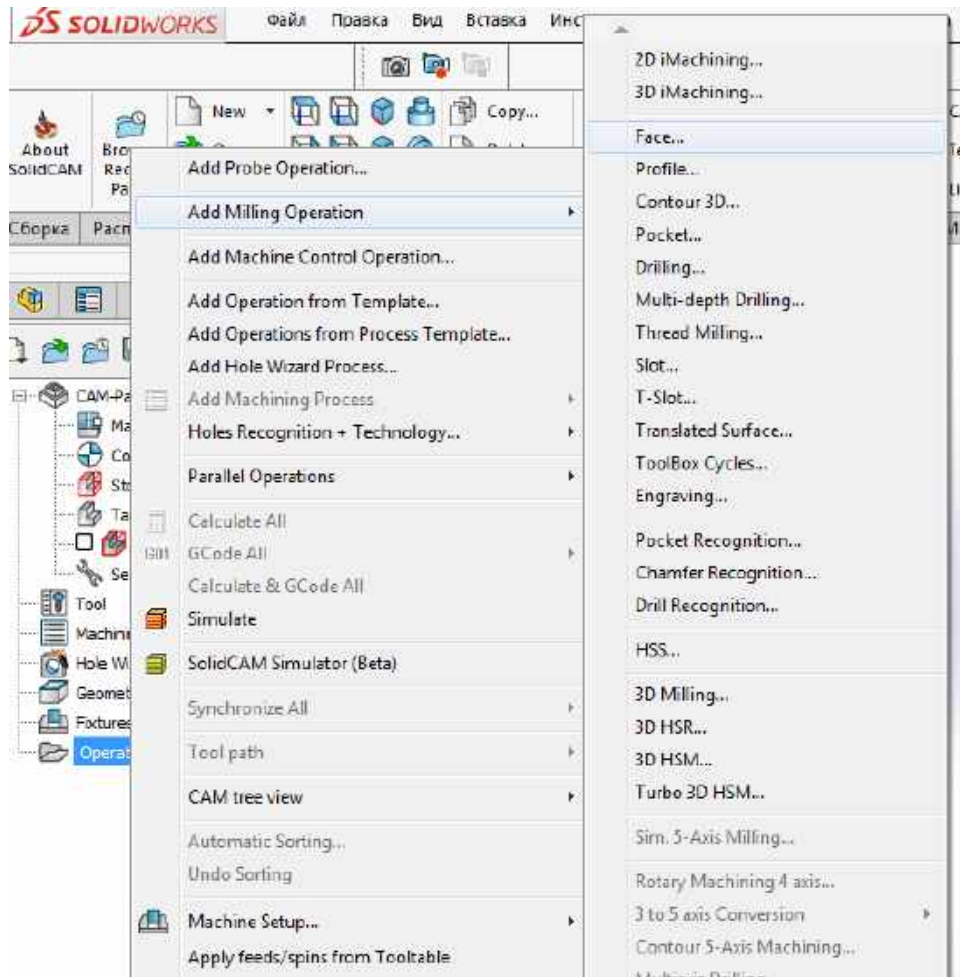


Fig. 21. Creating a Face Milling Operation

You can also add operations using the appropriate SolidCAM tabs in SOLIDWORKS CommandManager or the SolidCAM Operations toolbars.

The Face Milling Operation dialog box will appear (Fig. 22).

Description of the operating interface:

1. *Technology* – displays the selected technology according to the type of operation;
2. *Geometry* – defines the geometry for the operation;
3. *Tool* – defines the tool for the operation;
4. *Levels* – defines the machining levels for the operation;
5. *Technology* – defines technology parameters such as offsets, roughing and finishing data, depth type, etc;

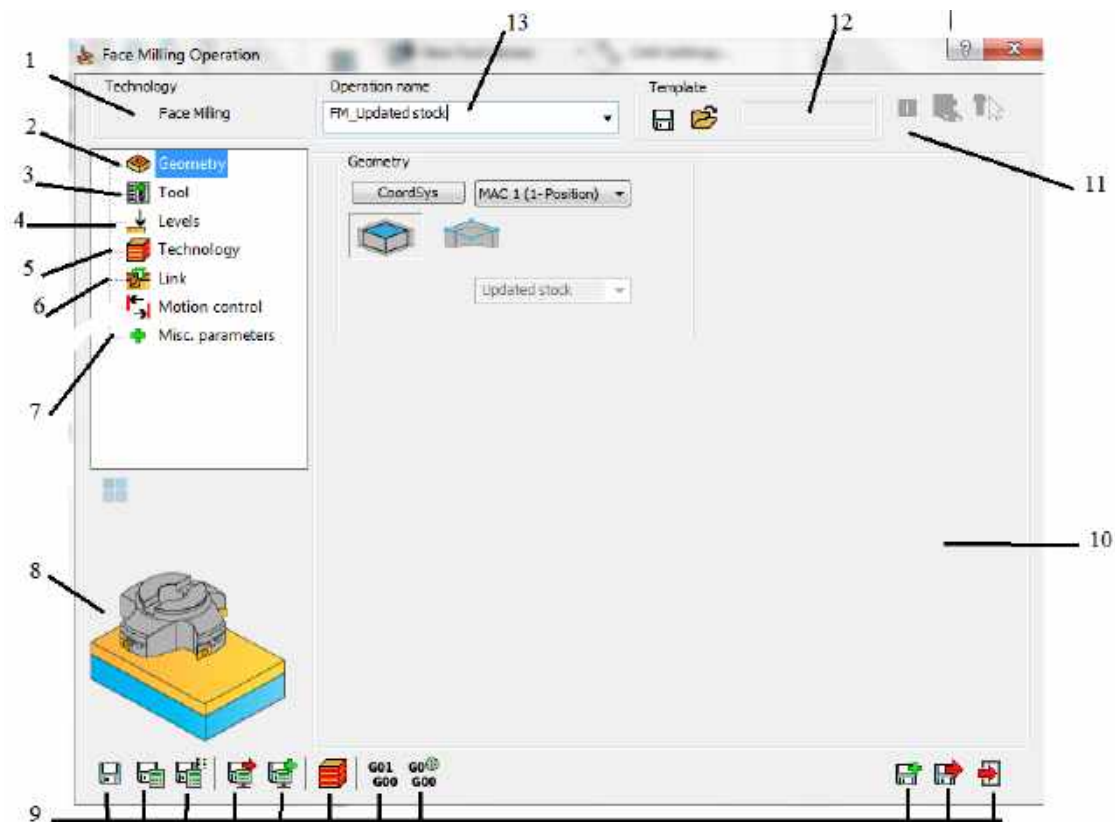


Fig. 22. Face Milling Operation Dialog Box:

1 – Technology; 2 – Geometry; 3 – Tool; 4 – Levels; 5 – Technology; 6 – Link; 7 – Miscellaneous parameters; 8 – Operation illustration; 9 – Operation buttons; 10 – Parameter page; 11 – Information; 12 – Template; 13 – Operation name

6. *Link* - defines the approach and indentation of the tool approach and retreat, the relationship between toolpaths, etc;

7. *Miscellaneous parameters* – defines various parameters of the operation, such as messages, additional parameters, etc;

8. *Operation illustration* – displays an image of the active item to make it easier to identify the corresponding parameter;

9. *Operation buttons* – allows you to perform functions such as *Save and Calculate*, *Simulate*, etc;

10. *Parameters page* – displays the parameters of the active item from the list on the left side of the dialog box;

11. *Information* – allows you to display a window that contains a summary of information about the operation;

12. *Template* – allows you to save the current operation as a template and load another existing template;

13. *Operation name* – displays the operation's name and allows you to select another existing transaction as a template.

In the Operation name field, enter *Face Milling – 1st Setup* (Fig. 23).



Fig. 23. Entering an Operation Name

By default, the *Stock-based boundary* is used to determine the Face milling geometry automatically. This option detects and identifies the initial face that needs to be machined using SolidCAM's Updated Stock model (USM).

After each operation, SolidCAM calculates the exact shape of the remaining allowance, which is displayed in the USM geometry.

Some operations can use USM data for object recognition and toolpath optimization.

2.1.1. Defining the Tool

For the next step, you have to determine the tool to be used for face milling. Go to the *Tool* page and click the *Select* button to define the Tool (Fig. 24).

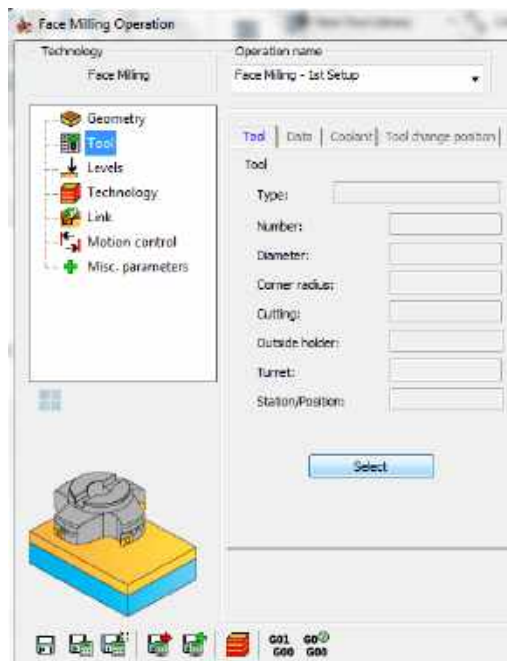


Fig. 24. Choosing a Tool for the Operation Button

The ToolKit interface is shown in Fig. 25.

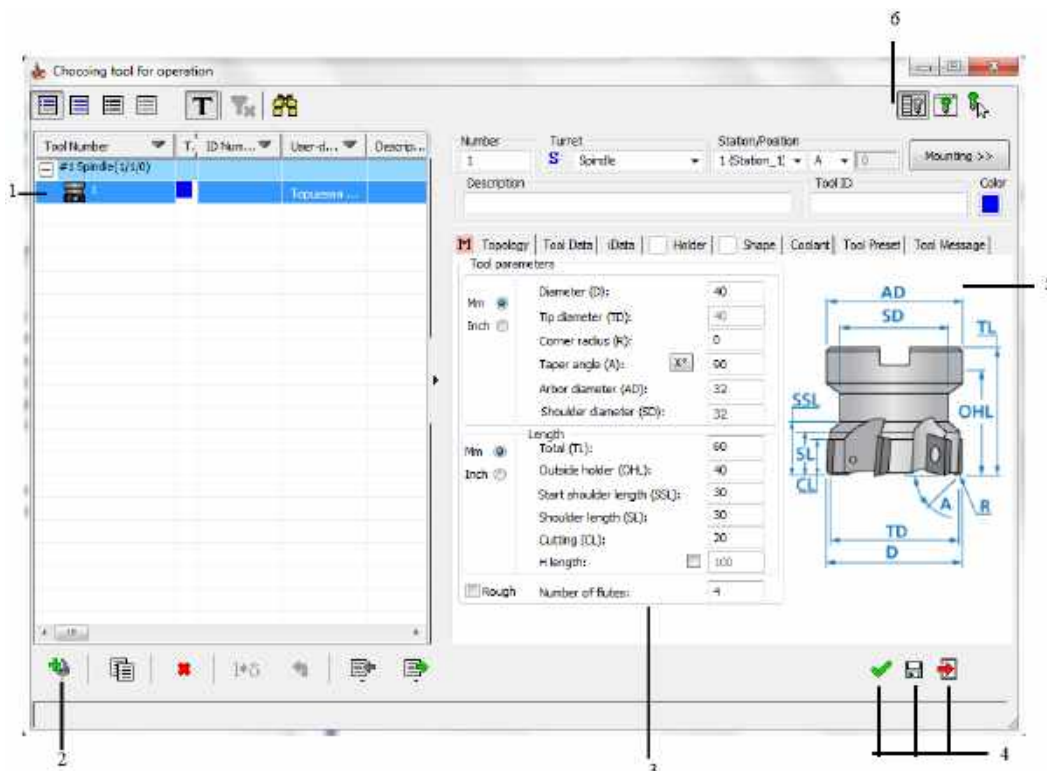


Fig. 25. Choosing Tool for Operation Interface

Here, the following options are available:

1. *Part Tool Table* – list of all available for use in the current CAM-Part;
2. *Add/Delete tool* – Option for adding/deleting tools;
3. *Tool parameters* – page with parameters that define tool geometry and other tool data;
4. *Toolkit buttons* – buttons for confirming selected tools, saving tool list, exiting from the tool window;
5. *Tool view* – illustration of the selected tool with parameters;
6. *Visualization Tools* – allow you to dynamically visualize tool definitions using various methods.

In the Toolkit dialog box with the Part Tool Table, you can manage the tools in the Part Tool Library.

The Part Tool Library is stored within the CAM-Part. Depending on the CAM settings, SolidCAM can load default tools into the Part Tool Table according to Operation default templates.

The default templates are deactivated for this exercise, leaving the current Part Tool Library empty. You need to define a new tool suitable for face milling.

The Selection Pane of the Toolkit dialog box displays the tools available for the current operation.

Double-click Face Mill in the Selection Pane or drag and drop it into the Tool Table window (Fig. 26).

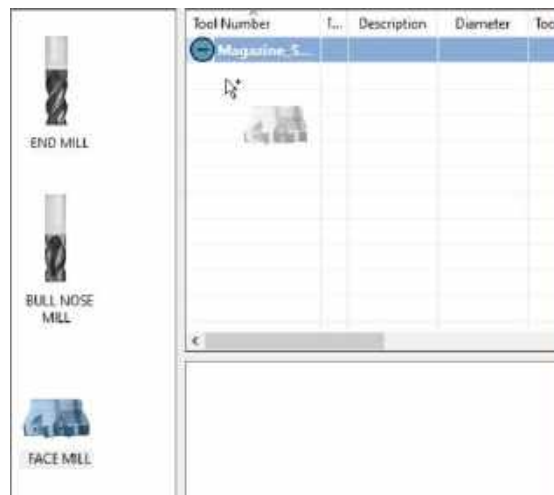


Fig. 26. Tool Table Window

The corresponding Tool item is created and displayed in the Tool Item Manager.

On the Data Page, define the Tool parameters shown in Fig. 27.

Click *Select* to confirm the Tool definition, select the tool to operate, and close the ToolKit dialog box.

The End Milling Operation dialog box appears.

Number: 1 Turret: S Spindle: 1 (Station_1) A D Mounting >>

Description: Tool ID: Color: Blue

M Topology Tool Data iData Holder Shape Coolant Tool Preset Tool Message

Tool parameters

Mm Inch

Diameter (D): 60

Tip diameter (TD): 60

Corner radius (R): 0

Taper angle (A): 90

Arbor diameter (AD): 32

Shoulder diameter (SD): 32

Length

Total (TL): 60

Outside holder (OHL): 40

Start shoulder length (SSL): 35

Shoulder length (SL): 30

Cutting (CL): 20

H length: 100

Rough Number of flutes: 4

3D Model Dimensions: AD, SD, TL, OHL, SSL, SL, CL, TD, D, A, R

Fig. 27. Tool Parameters

The *Tool page > Data* tab of the Operation dialog box lets you define the tool's Feed, Spin, and Offsets for the operation. SolidCAM populates the fields with the default cutting data specified for the Tool definition.

Switch to the *Data* tab to define the tool-cutting data for the operation (Fig. 28). This tab allows you to edit the data only for the current operation.

Enter a Spin rate value of 1000 rpm. Enter 500 mm/min for Feed XY and 250 mm/min for Feed Z.

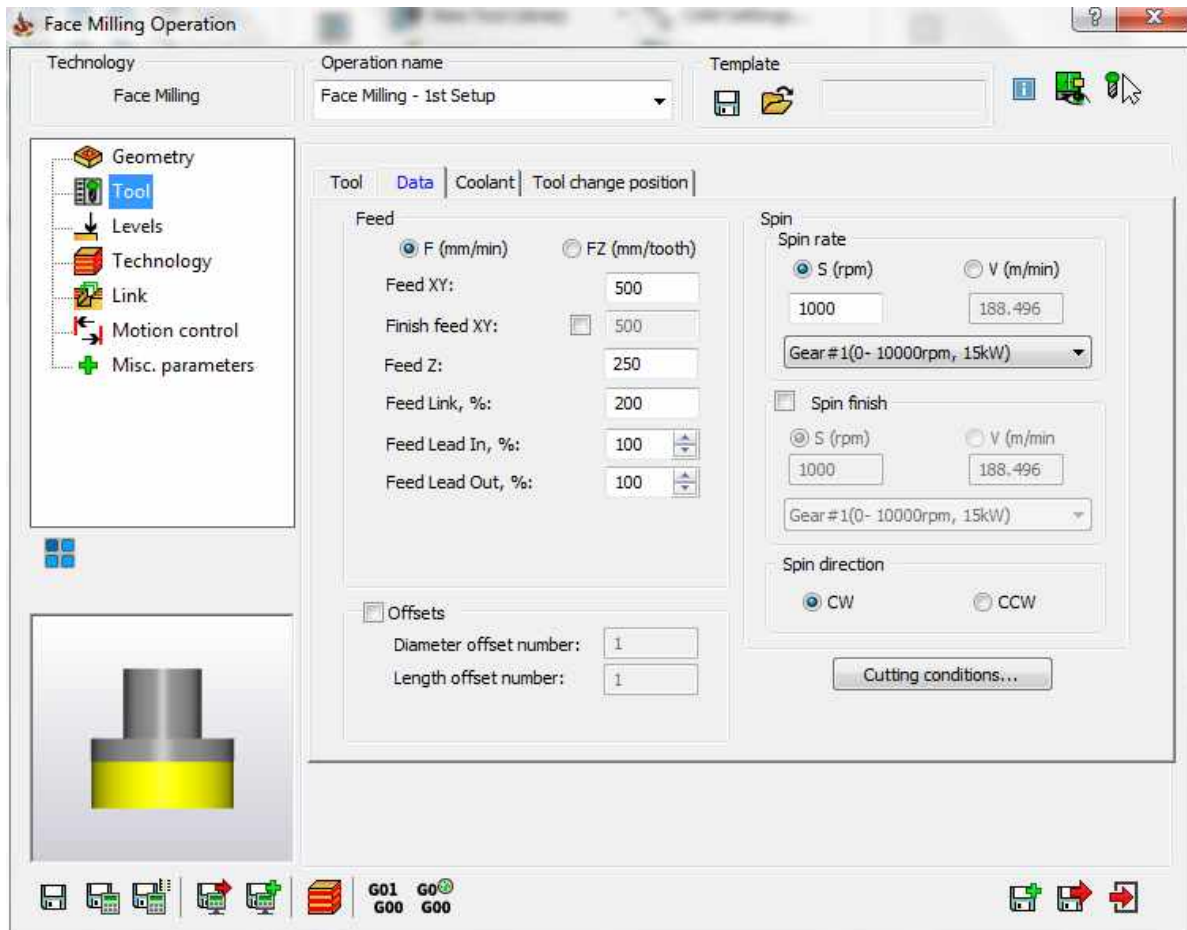


Fig. 28. Face milling Operation Settings

2.1.2. Defining the Levels

The next step is to define the Milling levels for the operation. SolidCAM allows you to specify the Z-levels at which the tool moves using the following parameters:

- *Upper level* – defines the Z-level from which machining starts;
- *Lower level (Depth)* – defines the Z-level below which the tool does not machine. This plane is not included in any milling strategy.

Go to the *Levels* page to define the Milling levels (Fig. 29).

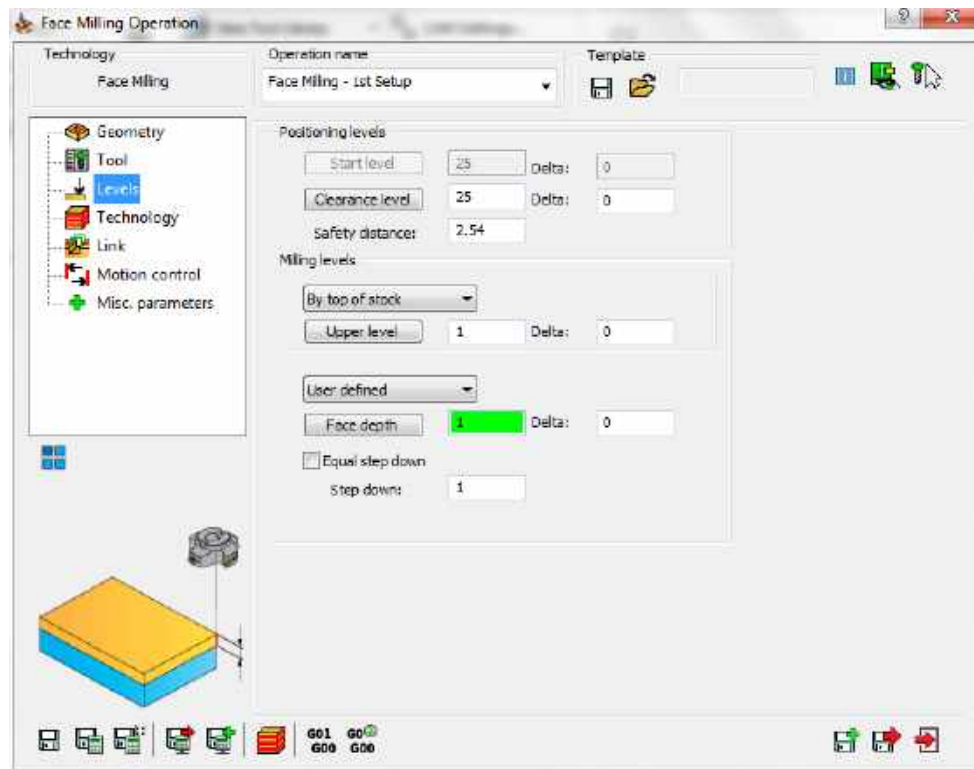


Fig. 29. The Levels Page

The default values of the Positioning levels are set according to the CAM-Part values specified in the CoordSys Data dialog box.

The Upper Level and Face Depth parameters define the Z-levels at which the tool moves in Face milling operations. SolidCAM automatically determines the levels for most operations based on the Geometry definition.

Since the face milling geometry was defined via *Stock based boundary*, the operation automatically uses the options *By the top of the stock* and *By the top of the target* to determine the Milling levels.

If preferred, you can define the levels using solid model data instead. This definition method, shown in the following steps, provides associativity of the Milling levels with the SOLIDWORKS model.

2.1.3. Defining the Technology

Switch to the *Technology* page to define the technological parameters of the Face Milling technology (Fig. 30).

Select a One Pass strategy for the technology since the tool is wider than the workpiece.

In the Extension area, use the default value of 10 for % of the tool diameter (Fig. 31).

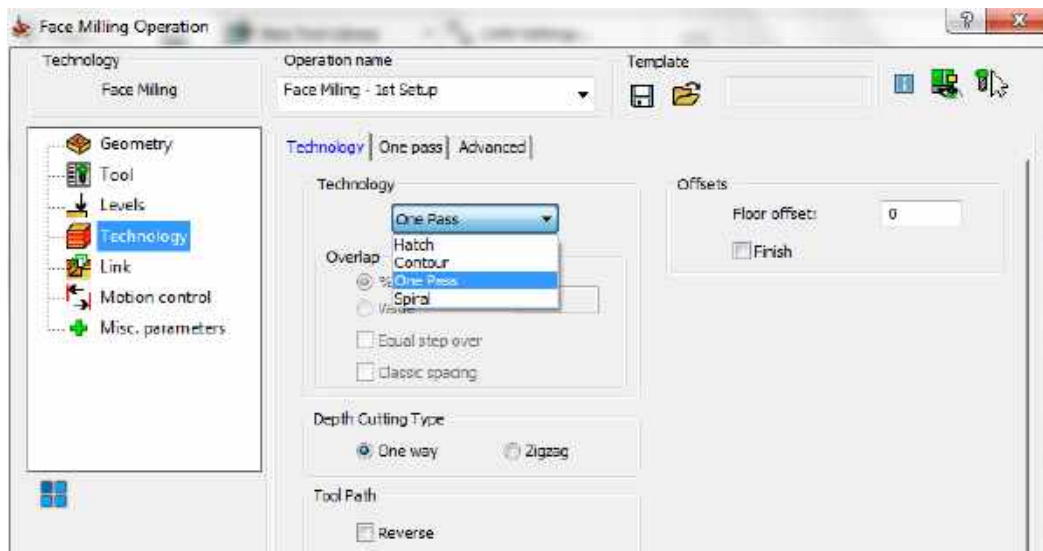


Fig. 30. The Technology Page

This means the toolpath will be extended over the face edges by 10 % of the tool diameter. In this case, there is no need to define any Lead in or Lead out parameters on the Link page.

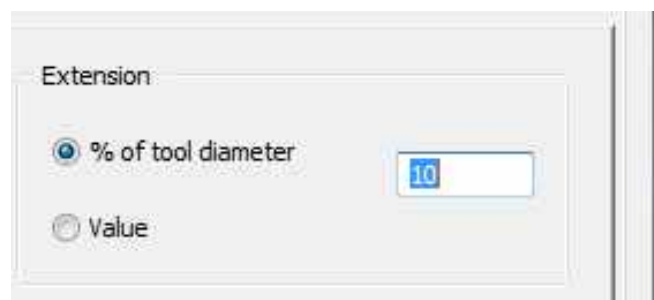


Fig. 31. Extension of the Tool Diameter

Click *Save and Calculate* to save the Face milling data and calculate the toolpath.

When the message appears, click *Yes* to confirm that the Upper level is above CoordSys Upper level of the operation (Fig. 32).



Fig. 32. Confirmation of the Upper level

2.1.4. Simulating the Operation

The SolidCAM Simulation option allows you to check and view the generated tool path after defining and calculating your machining operations. Simulation helps you avoid potential problems, such as mistakes in determining an operation or choosing an inappropriate milling strategy, that you would otherwise experience during an actual production run. Right-click the operation name in the SolidCAM Manager tree and then select *Simulate* from the context menu that appears (Fig. 33).

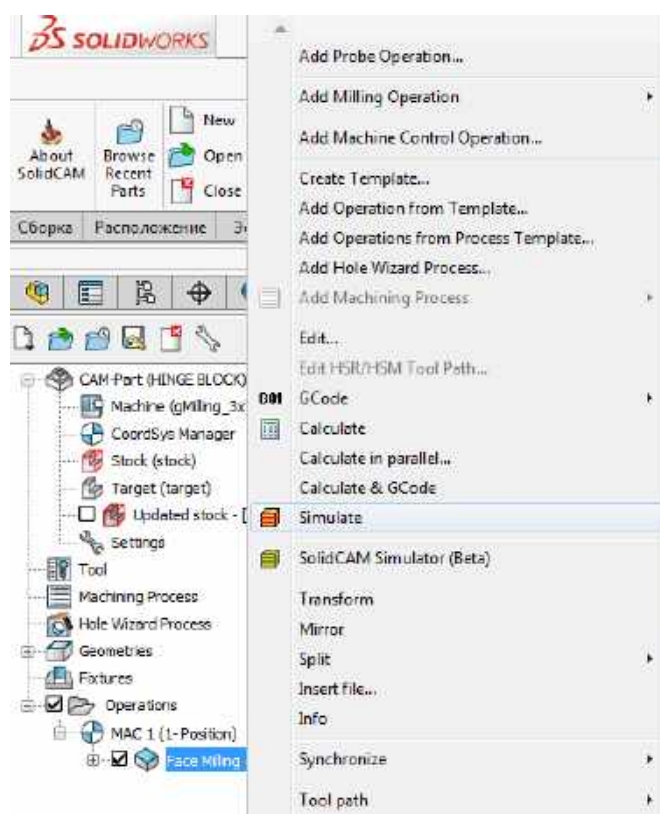


Fig. 33. Simulation Selection

The Simulation window appears (Fig. 34), which allows you to select and control the various simulation modes available in SolidCAM.

In the Simulation Data dialog box , click the Show Data button to display additional information about the toolpath, such as the current point coordinates, time, and feed.

The Simulation Control Panel includes:

1. *Simulation modes* – allows you to switch between the available SolidCAM simulation modes;
2. *Turbo* – performs the simulation in the computer's memory without displaying it on the screen. The image is displayed when the simulation is completed or when you click the *Pause* button;
3. *Play* – plays the simulation in a continuous mode;

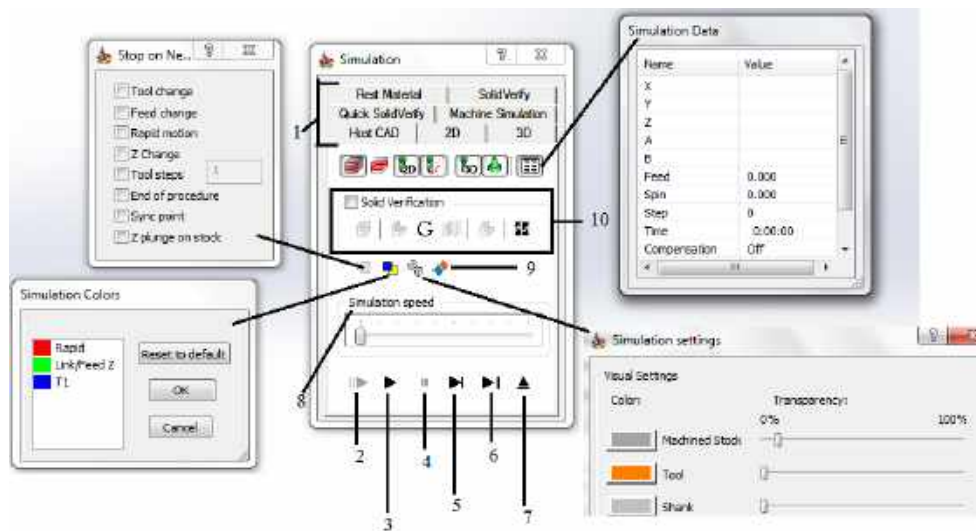


Fig. 34. The Simulation Dialog Box

4. *Pause* – pauses the simulation when playing in Continuous, Turbo, or Work Step modes;

5. *Single Step Mode* – performs the simulation step by step and then stops for each tool movement. To simulate the next step, press this button again or press the space bar on the keyboard;

6. *Continuous mode* – runs the simulation continuously until it reaches the end of the operation. For example, click this button to play only the first operation. Click it again to play only the second operation, and so on;

7. *Exit* – closes the simulation control panel;

8. *Speed slider* – allows you to control the speed of the simulation. Moving the slider to the left decreases the rate, and moving it to the right increases the speed;

9. *Clear* – clears the simulated toolpath;

10. *Solid Verification* – allows you to display the machining simulation directly on the solid model in the Graphics Area.

Click the *SolidVerify* tab and start the simulation by clicking the *Play* button. The Face milling operation is now simulated in SolidVerify (Fig. 35).

Close the Simulation Control Panel using the *Exit* button ▲.

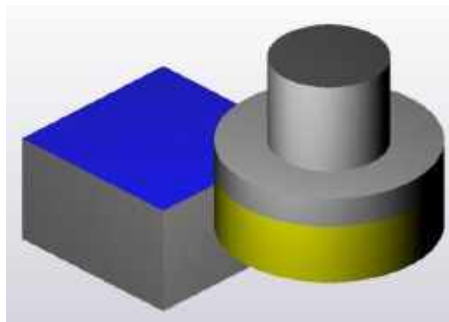


Fig. 35. SolidVerify Simulation

2.2. Adding a Profile Operation

Next, you need to machine the outside contour of the workpiece. In SolidCAM Manager, right-click any button in the Operations header or the last operation in the CAM tree.

Select *Profile* from the Add Milling Operation submenu (Fig. 36).

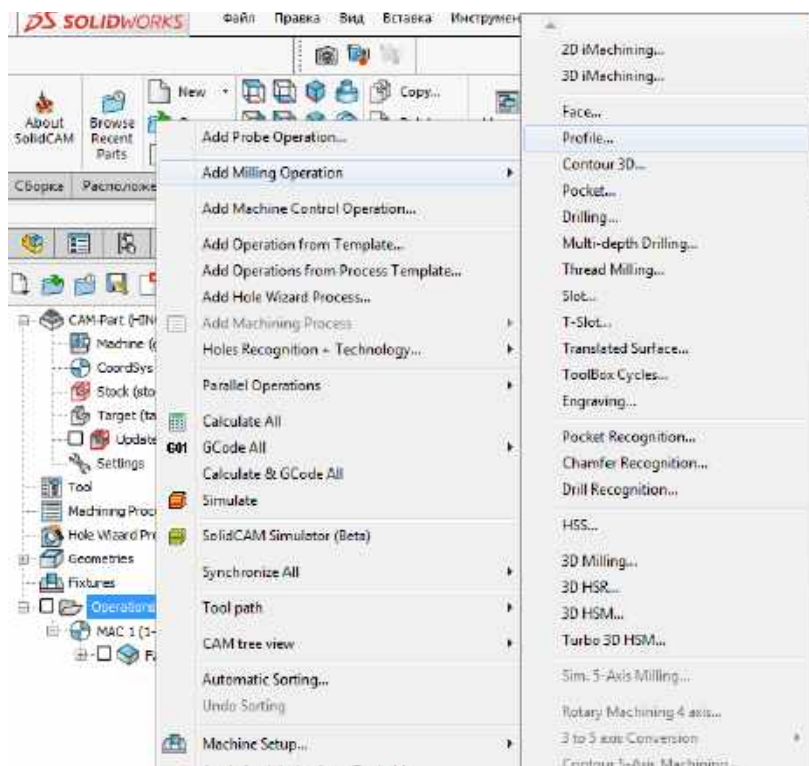


Fig. 36. Creating a Profile Operation

In the Operation name field, type *Outside Contour Milling* (Fig. 37).

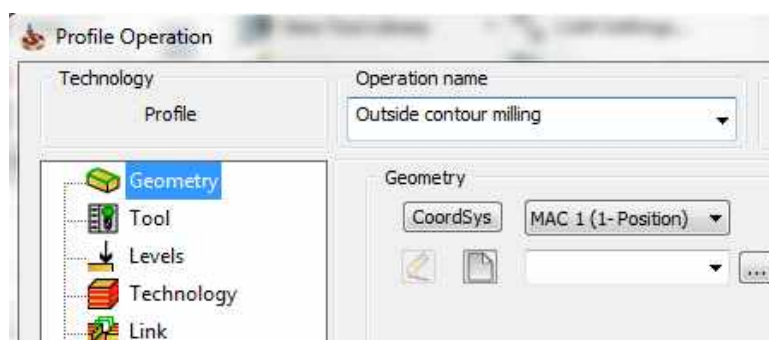


Fig. 37. Name of the Operation

On the Geometry page, click the *New* button to determine the machining geometry.

Select the top edge of the solid model, as shown in Fig. 38.

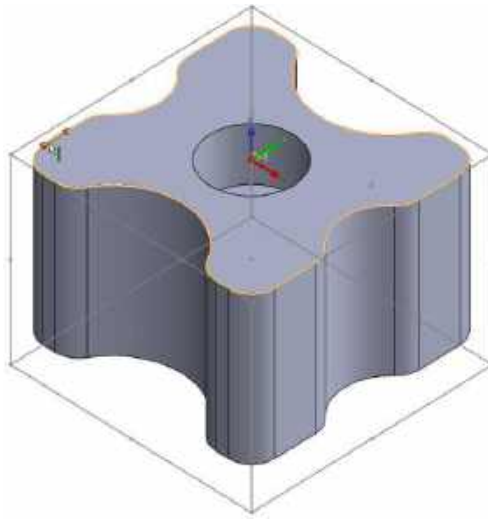


Fig. 38. Creating Geometry

The arrow at the chain start point indicates the direction of the geometry.

In SolidCAM operations, the direction of the chain geometry is used to calculate the tool path. When milling a profile, the tool moves in the geometry direction by default. In this tutorial, the geometry direction and clockwise tool rotation combination allow you to perform climb milling.

SolidCAM automatically connects all entities at the same Z-level to simplify the selection to close the chain.

Accept the choice in the corresponding message.

If you need to change the chain, for example, to change the direction, you can click *No* and use the chain buttons in the Chain List (Fig. 39).

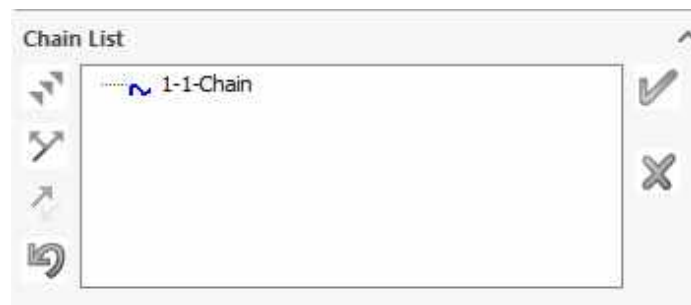


Fig. 39. Chain List

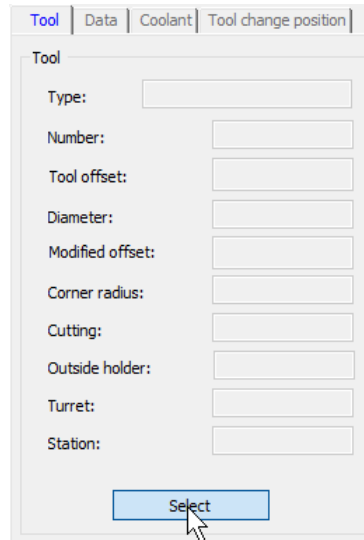
The Chain List allows you to manage existing chains of the current Geometry definition. To edit, right-click the chain and choose the appropriate command (*Replace*, *Insert*, *Edit*, *Reverse*, etc.) from the existing context menu.

Confirm the Geometry definition by clicking *OK* ☒. The Profile Operation dialog box appears again.

2.2.1. Defining the Tool

For the next step, you must define the tool for milling the contour.

Switch to the *Tool* page and click the *Select* button to define the tool (Fig. 40).



The screenshot shows a dialog box titled 'Tool' with four tabs: 'Tool', 'Data', 'Coolant', and 'Tool change position'. The 'Tool' tab is active. It contains several input fields for tool parameters: Type, Number, Tool offset, Diameter, Modified offset, Corner radius, Cutting, Outside holder, Turret, and Station. At the bottom of the dialog is a blue 'Select' button with a mouse cursor pointing at it.

Fig. 40. Tool Defining

The Part Tool Library is displayed. Only the tool used in the previous operation is listed and unsuitable for profile milling.

In the *Selection Pane*, double-click *End Mill* or simply drag and drop it into the Tool Table window.

The corresponding Tool Item is created and displayed in the Tool Item Manager (Fig. 41). On the Data Page, define the Tool parameters shown in Fig. 42.

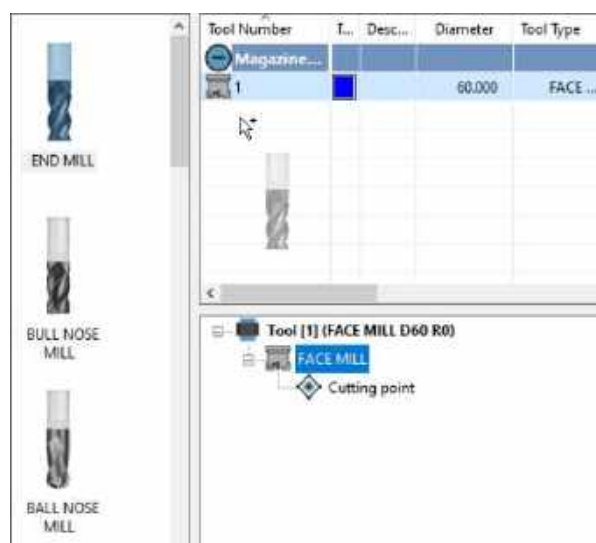


Fig. 41. Adding a Tool

Number: 2 Turret: S Spindle Station/Position: 1 (Station_1) A 0 Mounting >>

Description: Tool ID: Color:

M Topology Tool Data iData Holder Shape Coolant Tool Preset Tool Message

Tool parameters

☒ Mm Diameter (D): 20
☐ Inch Shoulder diameter (SD): 20
 Arbor diameter (AD): 20

☒ Mm Length
☐ Inch Total (TL): 80
 Outside holder (OHL): 60
 Shoulder length (SL): 40
 Shoulder angle (SA): 0
 Cutting (CL): 35
 H length: 100

☐ Rough Number of flutes: 4

3D diagram labels: AD, SA, TL, SL, SA, SD, OHL, CL, D

Icons: ✓ 📁 🔍

Fig. 42. Tool Parameters

Click the *Select* button to confirm the Tool definition, select the tool to operate, and close the ToolKit dialog box. The Tool page of the Profile operation dialog box appears.

Switch to the *Data* tab (Fig. 43) to define the tool-cutting data for the operation. Enter a Spin rate of 5000 rpm. For Feed XY, enter 500 mm/min and for Feed Z, enter 250 mm/min. If enabled, deselect the Finish feed XY and Spin finish override check boxes.

Tool **Data** Coolant Tool change position

Feed

☒ F (mm/min) ☐ FZ (mm/tooth)

Feed XY: 500
 Finish Feed XY: ☐ 500
 Feed Z: 250
 Feed Lead In, %: 100
 Feed Lead Out, %: 100

☐ Feed Z for penetration only
☐ Offsets
 Diameter offset number: 2
 Length offset number: 2

Spin

Spin rate
☒ S (rpm) ☐ V (m/min)
 5000 314.159
 Gear#1(0-10000rpm, 15kW)

☐ Spin finish
☒ S (rpm) ☐ V (m/min)
 5000 314.159
 Gear#1(0-10000rpm, 15kW)

Spin direction
☒ CW ☐ CCW

Cutting conditions...

Fig. 43. Determining the Cutting Modes

2.2.2. Determining the Profile Depth

For the next step, you need to define the Milling levels for the operation. Go to the *Levels* page (Fig. 44) to specify the Z-levels at which the tool moves.

In the Milling levels area, the Upper level is automatically detected because the profile geometry level was selected along the Upper level of the Solid model. In this tutorial, the Upper level is 0 because the Coordinate System was defined on the top surface of the part.

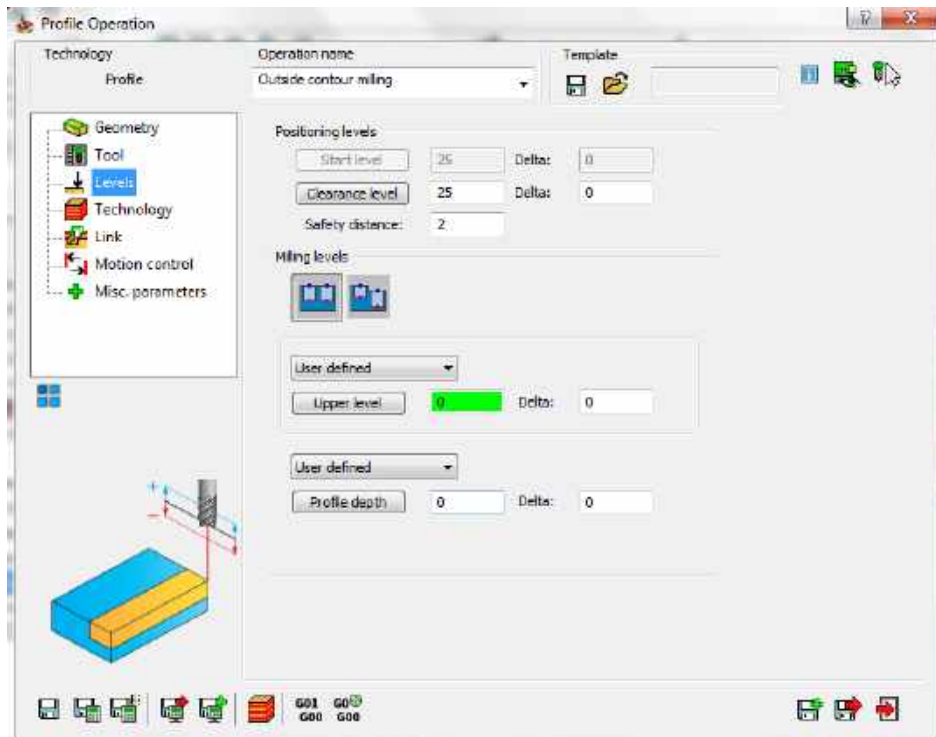


Fig. 44. The Levels Page

Click the *Profile depth* button, then select the Solid model's bottom edge, as shown in Fig. 45. The Profile depth value (–30) is displayed in the Pick Lower level dialog box.

Confirm the Profile depth by clicking **OK** .

The depth of the profile is automatically calculated from the difference between the Upper and Lower level values. This value (30 mm) is displayed in the Profile depth field of the Operation dialog box.

The Lower level parameter is associated with the SOLIDWORKS model. Associativity allows SolidCAM to synchronize with later model changes. When the model is changed, the CAM data is updated automatically.

The Profile depth parameter is indirectly associative. Associativity is set at the Lower level. When the Upper or Lower level is synchronized, the Profile depth is updated. The colored field indicates that the parameter is associated with the selected model entity (Fig. 46).

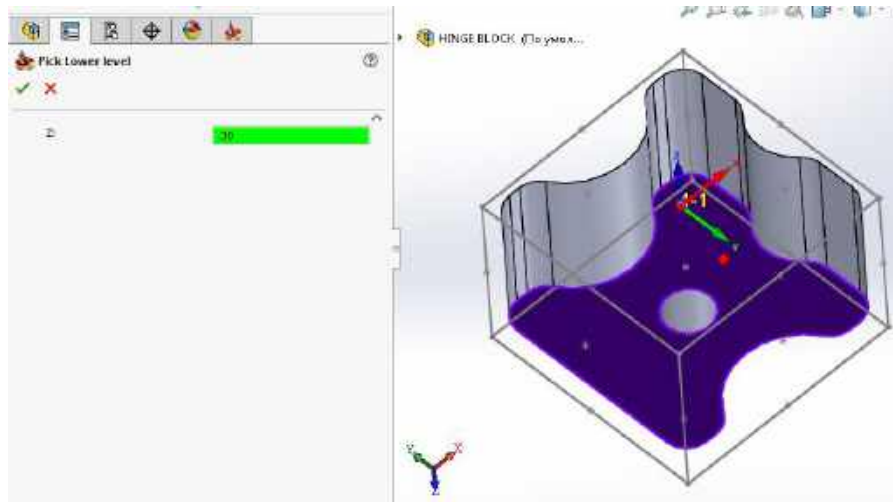


Fig. 45. Selecting the Profile Depth

Positioning levels

Start level	25	Delta:	0
Clearance level	25	Delta:	0
Safety distance:	2		

Milling levels

User defined

Upper level	0	Delta:	0
-------------	---	--------	---

User defined

Profile depth	30	Delta:	0
---------------	----	--------	---

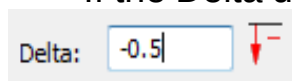
Fig. 46. Associativity of Milling Depth

In SolidCAM, the Delta parameter allows you to define an offset for the depth of cut that can be changed while maintaining associativity. This parameter refers to the Profile depth defined for the operation.

For example, you have a Profile depth of 12 mm, and you want to get a 0.5 mm allowance in the Z+ direction. If you enter 0.5 in the Delta field, the machining will be performed 0.5 mm above the defined Profile depth, even if the model is later changed.

For example, a negative value is often used when machining deeper than the bottom edge of a workpiece. Enter a depth delta value of 0.5 mm.

If the Delta depth value is negative, a red arrow is displayed next to the field



, indicating a negative offset value (in the negative direction of

the Z axis). If the Delta depth value is positive, a blue arrow is displayed next to the field Delta: 0.5, indicating a positive offset value (in the positive direction of the Z axis) (Fig. 47) [2].

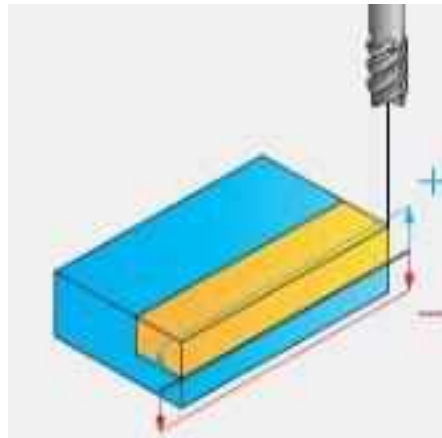


Fig. 47. Definition of Delta Depths

2.2.3. Defining the Technology

Switch to the *Technology* page to define the technological parameters of the Profile technology.

In the Modify area, define the position of the tool relative to the geometry:

1) On the left side – the tool cuts on the left side of the profile geometry (the tool is shown with a right-handed spiral and spindle direction M03) (Fig. 48, a);

2) On the right side – the tool cuts on the right side of the profile geometry (tool shown with right-hand spiral and spindle direction M03) (Fig. 48, b);

3) Center – the tool center moves precisely along the profile geometry (no compensation options can be used) (Fig. 48, c) [2].

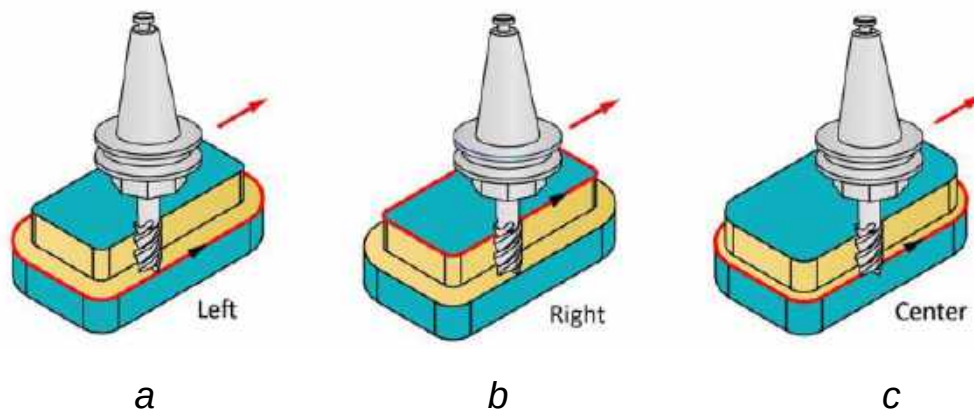


Fig. 48. Tool Position Options:
a – Left; b – Right; c – Center

Left's default setting for this tutorial meets the climbing milling requirements (Fig. 49).

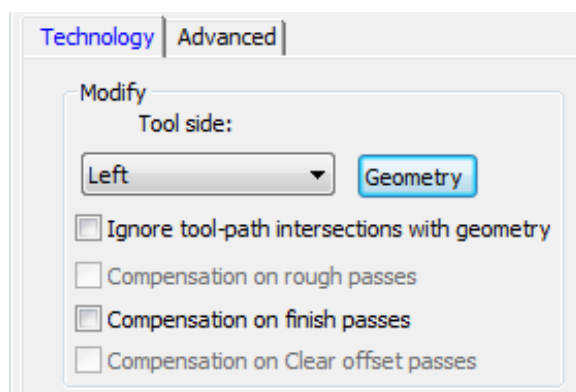


Fig. 49. Selecting the Tool Position

Click the *Geometry* button to check the tool position. Make sure that the position of the tool is correct.

Click *Cancel*  to return to the Profile Operation dialog box (Fig. 50). By default, the *Finish* operation is selected. In this case, roughing and finishing can be performed with the same tool on the same operation. Select both the *Equal step down* and *Rough* check boxes, and then enter a Maximum step down of 6 mm for roughing. In the Wall offset box, enter 0.3 mm.

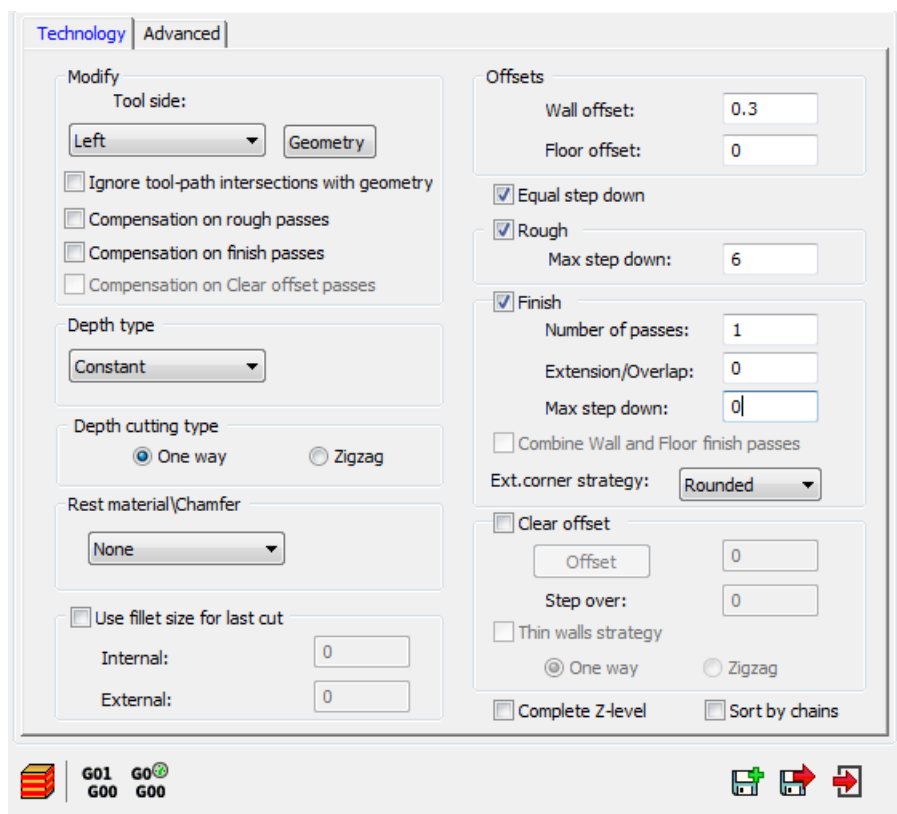


Fig. 50. The Technology PAGE

In the Finish area, if the Maximum step down is set to 0, the processing will be performed at the full Profile depth.

Enable *Compensation* for both rough and finish passes (Fig. 51). The CNC-Machine Controller's G4x tool radius compensation option will be used in GCode.

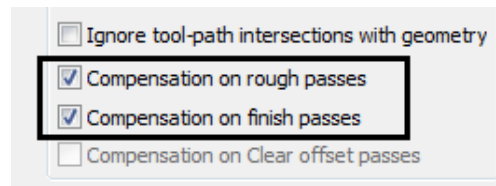


Fig. 51. Modifying the Geometry

2.2.4. Defining the Lead in and Lead out Tool Link Movements

On the *Link* page (Fig. 52), the the Lead in and Lead out areas enable you to control how the tool approaches and profile and then retreats away.

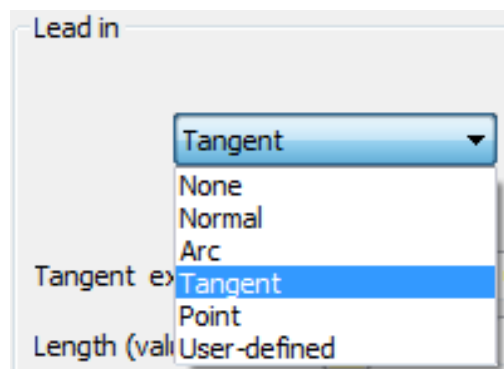


Fig. 52. The Tool Lead in Page

The lead-in movements are necessary to prevent the tool's vertical entering into the material. With the Lead in strategies, the tool descends to the machining level outside the material and then horizontally penetrates the material. The lead-out strategy allows for retraction movements outside the material.

The following options are available (Fig. 52):

1. *None* – the tool leads in and out of the cut from a milling level that is precisely adjacent to the profile start point;
2. *Normal* – the tool enters/leaves from a point normal to the profile. The distance between the point normal and the material is set in the tangent extension field. You can specify the normal length in the accompanying field;
3. *Arc* – the tool leads in/out via a tangent arc. The length of this extension is specified in the Tangent extension field. The arc radius is set in the Radius field. The Arc angle can be defined in the accompanying field. You can specify the arc angle in the accompanying field;

4. *Tangent* – the tool leads in/out via a line tangent to the profile. The distance to the material is specified in the Tangent extension field. The Length is specified in the accompanying field;

5. *Point* – the tool leads in/out from a user-defined position. From this position, the tool moves in a straight line to the start point of the profile. When chosen, the Pick button is activated to select a position directly on the solid model. The tangent extension field specifies the distance between the point and the material.

Select the *Tangent* option and specify a length (value) of  15 mm (Fig. 53).

If the *Same as Lead in* check box is enabled in the Lead out area, the same strategy and parameters defined for Lead in are used for Lead out. Often, this option can help to simplify the selection process.

Select the Arc option for Lead out and specify a Radius (value) of  2 mm with 90° an Arc angle.

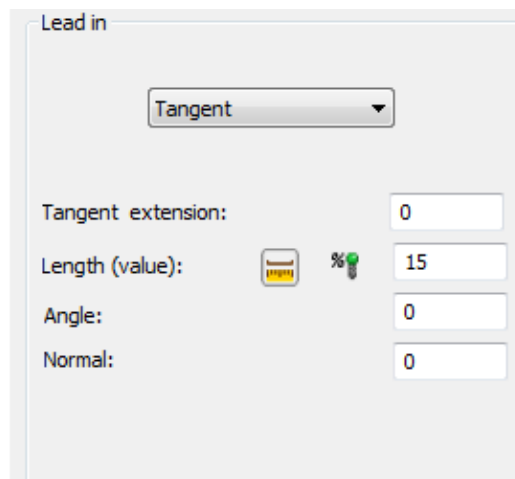


Fig. 53. The Lead in Set-up

For the Links between... passes, you should select the *Safety distance from the upper level* to prevent unnecessary movement of the tool to the Clearance level (Fig. 54).

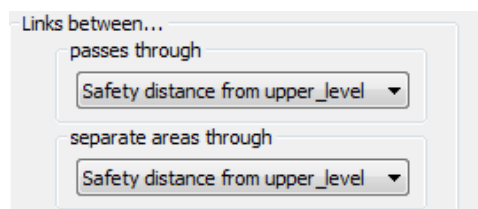


Fig. 54. Links between Setup

This completes the definition of the main technological parameters of profile milling.

2.2.5. Calculating and Simulating the Tool Path

Click the *Save and Calculate* button . The Profile's operation data is saved, and the Tool Path is calculated. Click the *Simulate* button , and then simulate the operation using the SolidVerify mode (Fig. 55).

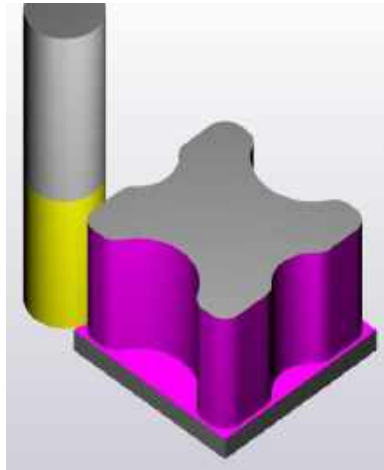


Fig. 55. SolidVerify Simulation

Now switch to the *Host CAD* mode and click *Play* button .

The Host CAD simulation mode (Fig. 56) allows you to display the tool path directly on the solid model in the Graphics Area. The SOLIDWORKS View Orientation options can be used to see the tool path from different viewpoints and zoom in on other areas of the model.

The Show Tool 2D  option turns on/off a simplified graphical representation of the tool.

The Show 3D  tool option enables/disables the three-dimensional graphical representation of the tool.

Close the Simulation control panel by clicking the *Exit* button .

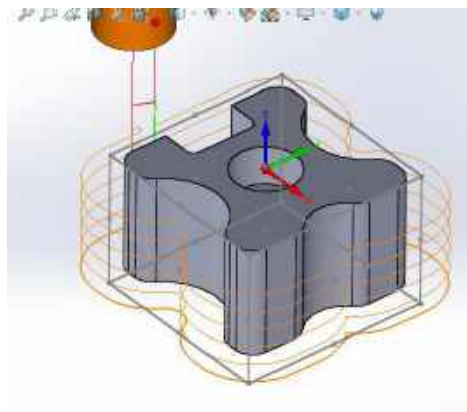


Fig. 56. Tool Path for Profile Operation

2.3. Centering the Through Hole

Next, define the Drilling operation for the preliminary center drilling of the through hole.

In the CommandManager, switch to the *SolidCAM 2.5D* tab and select *Drilling* (Fig. 57).

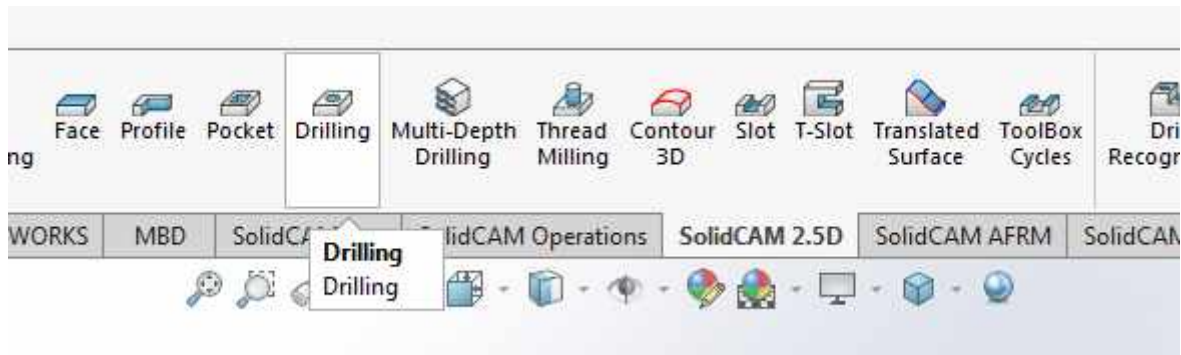


Fig. 57. Adding of the Drilling Operation

SolidCAM tabs provide you with an alternative method of adding operations. In the *Operation name* field, type *Through hole centering*.

On the *Geometry* page, click the *New* button, then select the solid model's top face (Fig. 58).

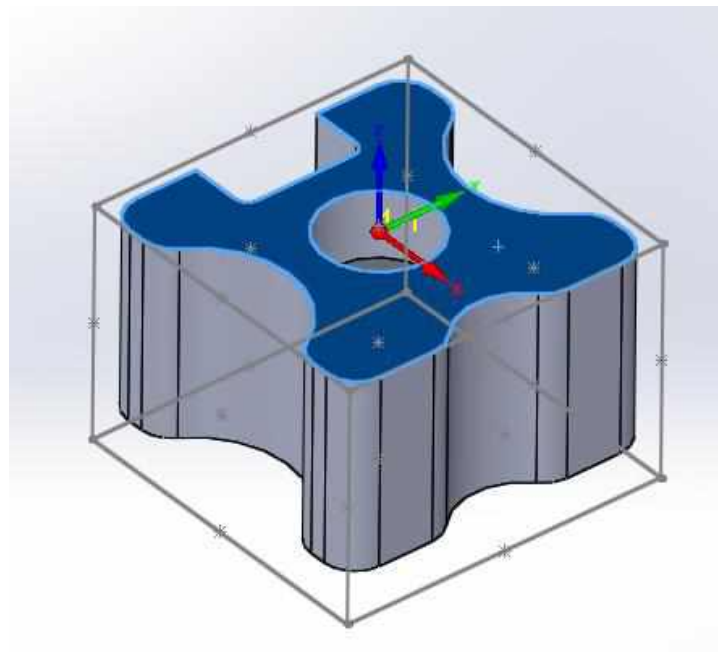


Fig. 58. Selecting a Geometry for the Drilling Operation

By default, *Multi-positions* are selected in the *Select centers by* field. *All circles* are set in the *Filter for the circle's area* (Fig. 59).

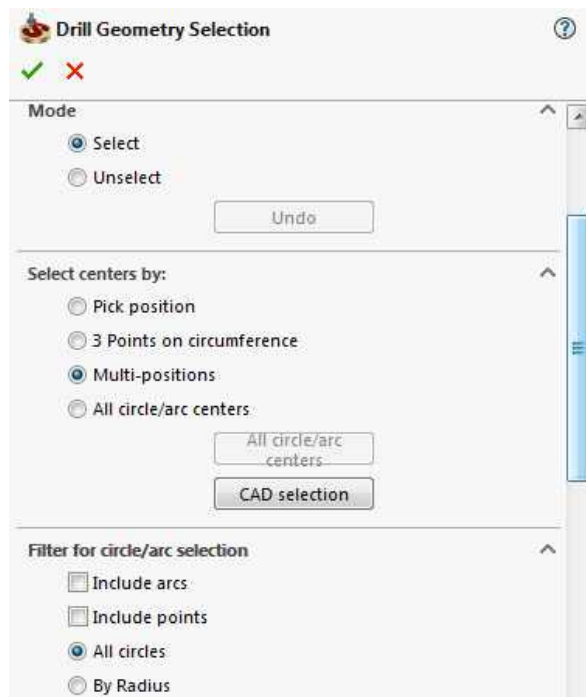


Fig. 59. Drill Geometry Selection

SolidCAM automatically recognizes all hole centers on the selected face and defines them as drilling positions.

Click OK  to confirm the *Select Drilling Geometry* dialog box.

2.3.1. Define the Tool and Tool Data

Go to the *Tool* page, then click the *Select* button to start defining the Tool.

In the ToolKit dialog box, click *Drilling Tools*  in the library Toolbar. The Selection pane displays the tools available for the current operation. Add the *Spot Drill* to the Part Tool Library and set the Diameter to 10 mm. Click the *Select*  button to confirm the tool for the operation.

On the *Tool* page> *Data* tab of the Drilling Operation dialog box, define the cutting data of the tool, as shown in Fig. 60.

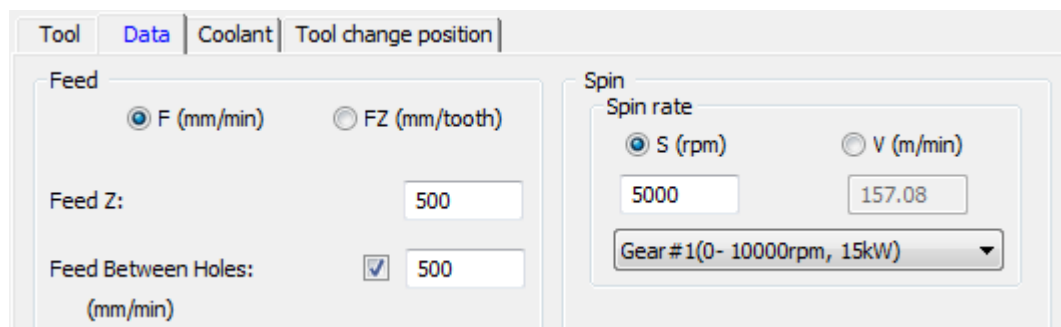


Fig. 60. Cutting Parameters for Drilling Operation

2.3.2. Defining the Drilling Depth

Go to the *Levels* page of the *Drilling Operation* dialog box (Fig. 61).

Positioning levels

Start level: 25 Delta: 0

Clearance level: 25 Delta: 0

Safety distance: 2

Milling levels

User defined

Upper level: 0 Delta: 0

Drill depth: 0 Delta: 0

Depth type

☐ Cutter tip

☐ Full diameter

☒ Diameter value: 4

Fig. 61. The Levels Setup

Click the *Upper Level* button, then select the solid model's top edge. Click the *Drill depth* button, then select the solid model's top edge again. The Drilling Depth value of 0 mm appears in the corresponding field.

The *Depth type* option can be used to deepen the drilled hole to obtain the specified diameter at the selected drilling depth.

Select the *Diameter value* for the *Depth type* and enter a value of 4 mm.

Click *Save and Calculate* . *Simulate*  the operation in SolidVerify mode (Fig. 62). Close the Simulation control panel by clicking the *Exit* button .

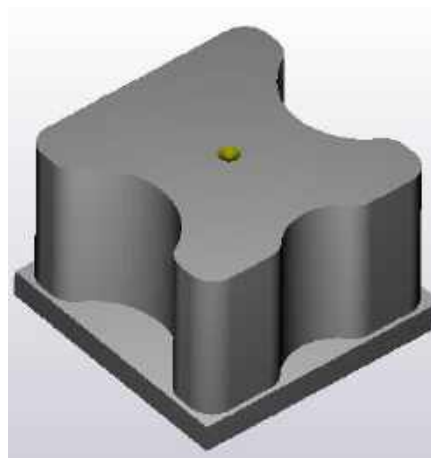


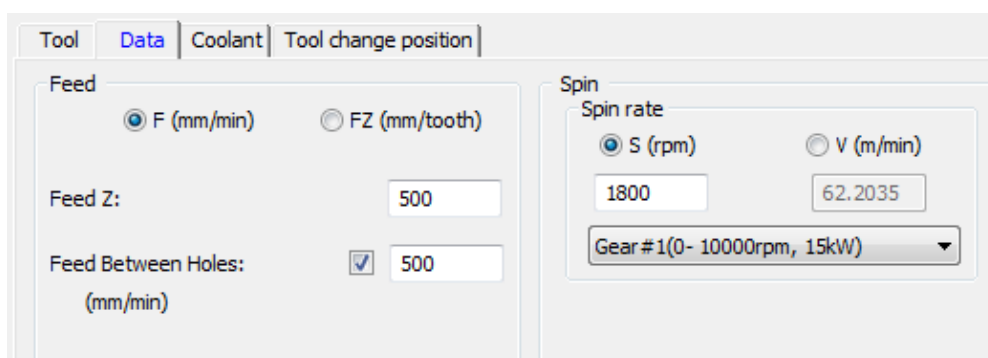
Fig. 62. SolidVerify Simulation

2.4. Drilling the Through Hole

Click the *Save and Copy*  button in the low right corner of the *Drill Operation* dialog box. This button saves the current operation data and automatically creates a new operation with the same parameters. The new operation is automatically opened for editing. Using the *Save and Copy* button, you can quickly create a new similar operation where most of the parameters are the same.

In the *Operation name* field, type *Through Hole drilling*.

For this operation, define a drill with a diameter of 11 mm. Define the tool-cutting data as shown in Fig. 63.

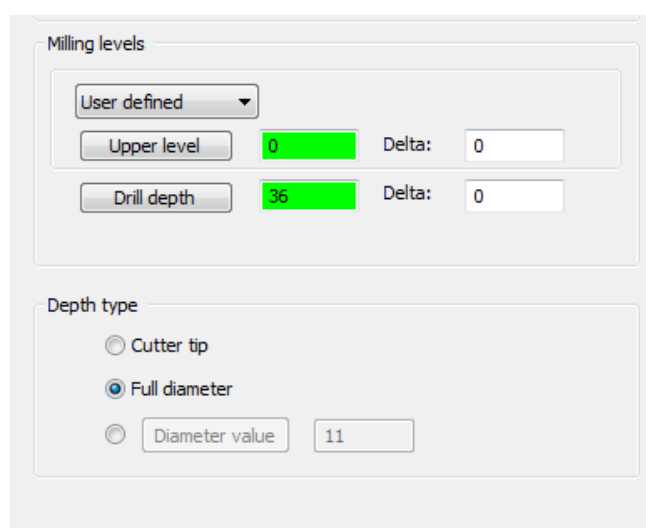


The screenshot shows the 'Data' tab of the 'Drill Operation' dialog box. It is divided into two main sections: 'Feed' and 'Spin'.
In the 'Feed' section, there are two radio buttons: 'F (mm/min)' (selected) and 'FZ (mm/tooth)'. Below them, 'Feed Z:' is set to 500. 'Feed Between Holes:' is checked and set to 500 (mm/min).
In the 'Spin' section, there are two radio buttons: 'S (rpm)' (selected) and 'V (m/min)'. Below them, 'Spin rate' is set to 1800. A dropdown menu shows 'Gear #1(0- 10000rpm, 15kW)'.

Fig. 63. Cutting Regimes for Drilling Operation

Go to the *Levels* page to define the new drilling depth and the method used to reach the depth.

Click the *Drill Depth* button, then select the model's bottom corner to determine the drilling depth. The drilling depth value is now 36 mm (Fig. 64). Since this depth is sufficient, no Depth delta is required.



The screenshot shows the 'Milling levels' dialog box. It has a dropdown menu set to 'User defined'. Below it, there are two rows of controls: 'Upper level' with a value of 0 and 'Delta: 0', and 'Drill depth' with a value of 36 and 'Delta: 0'. The values 0 and 36 are highlighted in green. Below this is the 'Depth type' section with three radio buttons: 'Cutter tip', 'Full diameter' (selected), and 'Diameter value' with a value of 11.

Fig. 64. Defining the Drilling Depth

2.4.1. Using Drill Cycles

Go to the *Technology* page. SolidCAM allows you to use several standard drilling cycles that your CNC-Machine Controller supports (e.g., Drilling or Boring).

Click the *Drill Cycle Type* button and select G73 from the list (Fig. 65). Click the *Parameters* button and enter the value. *Step down* 10 mm.

Click *Save & Calculate* , and then when prompted, click Yes.

Simulate the operation in SolidVerify mode .

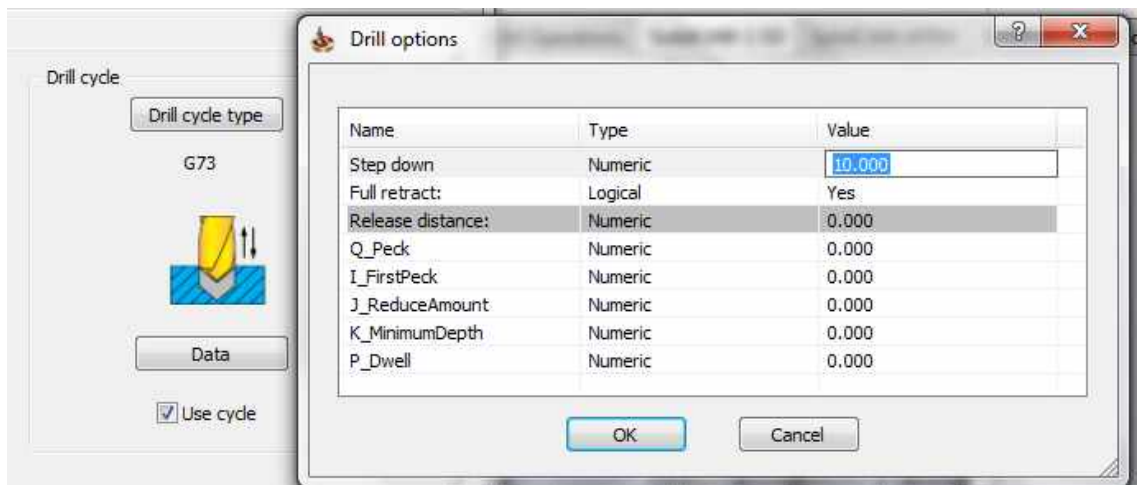


Fig. 65. Drill Cycle Type Set-up

2.5. Milling the Counterbore

To perform the machining of the counterbore, add a *Pocket Machining Operation*  and name it.

On the *Geometry* page, click *New* , and then select the counterbore contour. A confirmation message is automatically displayed since the shape is a closed circle. Click *Yes* and accept the selection. Confirm the definition of geometry by clicking *OK* .

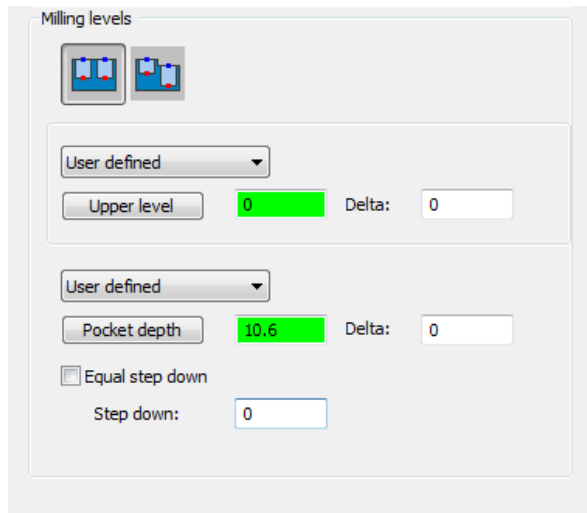
For this operation, define a Ø10 mm end mill. Define the tool cutting data as previously for the Ø20 end mill used for Profile milling.

Go to the *Levels* page, then define the Pocket depth directly on the solid model (Fig. 66).

In this case, Rough and Finish passes can be performed with the same tool. In the Finish area, select the *Walls* check box.

Enable *Compensation* for Rough and Finish passes to use the G4x option in GCode.

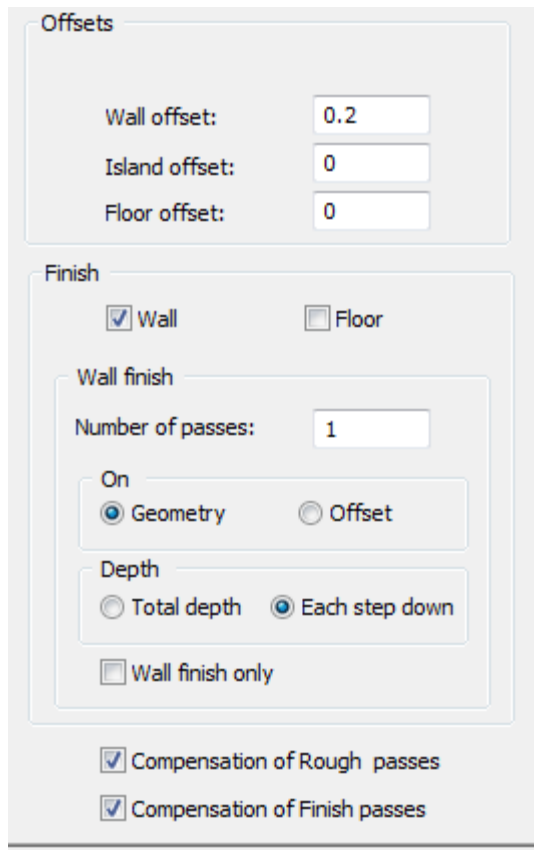
Since the workpiece is already drilled at this point, the vertical option of Ramping can be used for machining.



The 'Milling levels' dialog box contains two sections. The first section has a 'User defined' dropdown, an 'Upper level' button, a green field with '0', and a 'Delta: 0' field. The second section also has a 'User defined' dropdown, a 'Pocket depth' button, a green field with '10,6', and a 'Delta: 0' field. Below these is an 'Equal step down' checkbox and a 'Step down: 0' field.

Fig. 66. Definition of Pocket Depth

Go to the *Technology* page and enter 0.2 mm in the *Wall offset* field (Fig. 67).



The 'Offsets' section includes 'Wall offset: 0.2', 'Island offset: 0', and 'Floor offset: 0'. The 'Finish' section has 'Wall' and 'Floor' checkboxes, with 'Wall' selected. Under 'Wall finish', 'Number of passes' is '1'. There are two groups of radio buttons: 'On' with 'Geometry' selected and 'Offset' unselected; 'Depth' with 'Total depth' unselected and 'Each step down' selected. A 'Wall finish only' checkbox is also present. At the bottom, 'Compensation of Rough passes' and 'Compensation of Finish passes' are both checked.

Fig. 67. Definition of Allowances

Go to the *Link* page, and then define *Lead in* and *Lead out* parameters for the tool, as shown in Fig. 68.

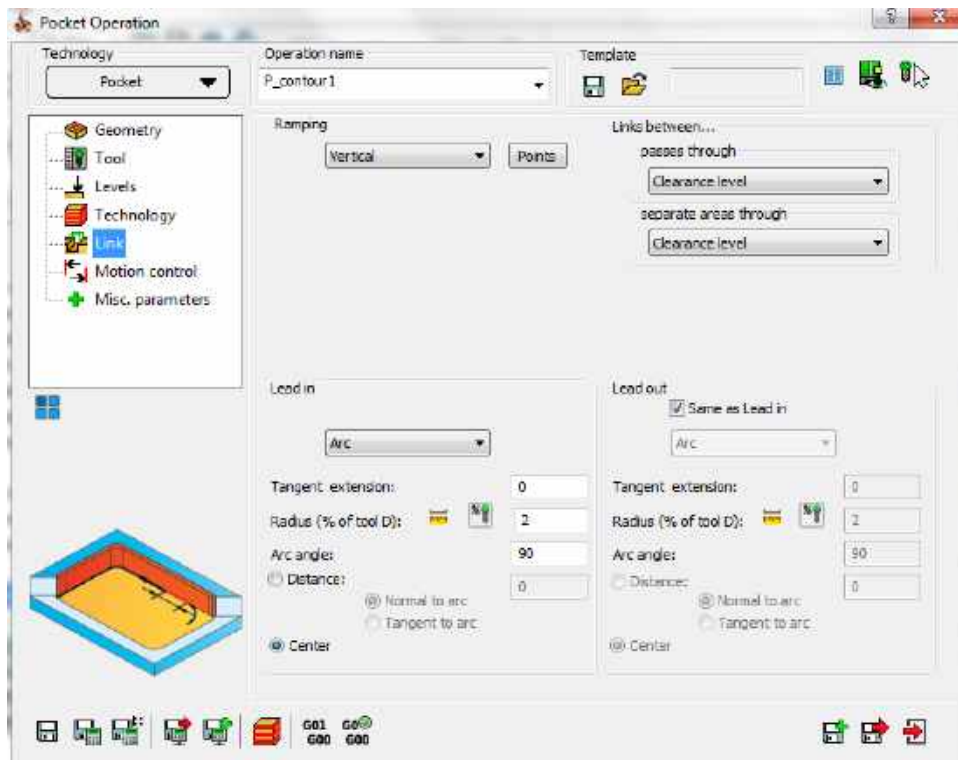


Fig. 68. Tool Lead in/Lead out Parameters

Fig. 69 shows Ramping points settings. Click **OK**  to confirm the dialog box.

Click **Save and Calculate** , and then simulate the operation in SolidVerify mode .

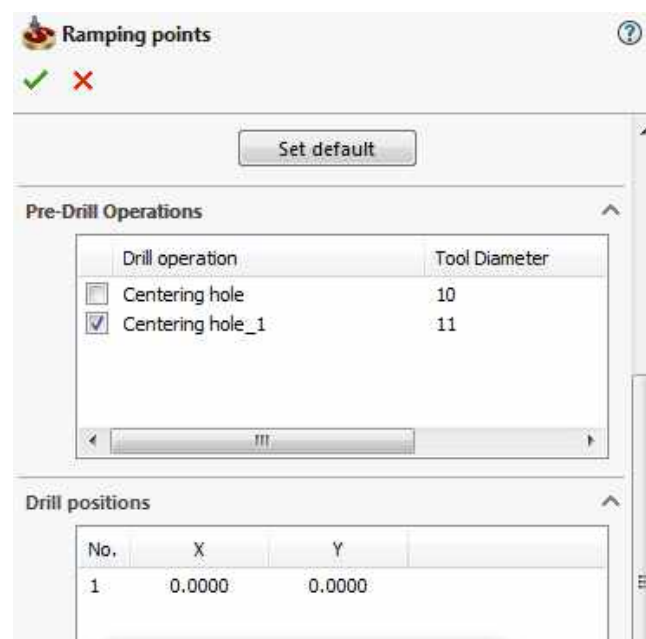


Fig. 69. Determining the Immersion Point

2.6. Defining the Open Pocket Milling

The Open pocket will be machined in two stages. First, Rough milling with the end mill, then Finish milling with a spherical end mill.

Geometry selection is shown in Fig. 70.

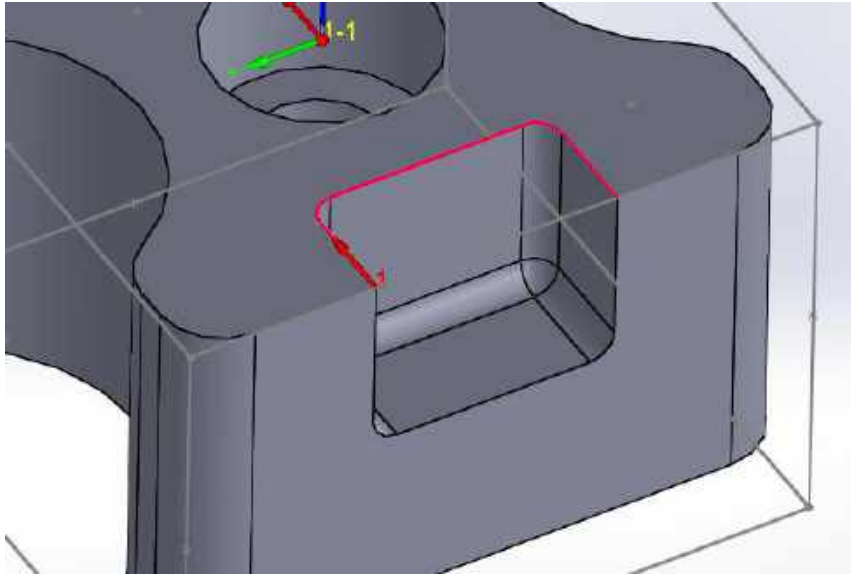


Fig. 70. Geometry Selection for Rough Milling of Open Pocket

Then go to the Tool and add an end mill (6 mm diameter) as described above (Fig. 71).

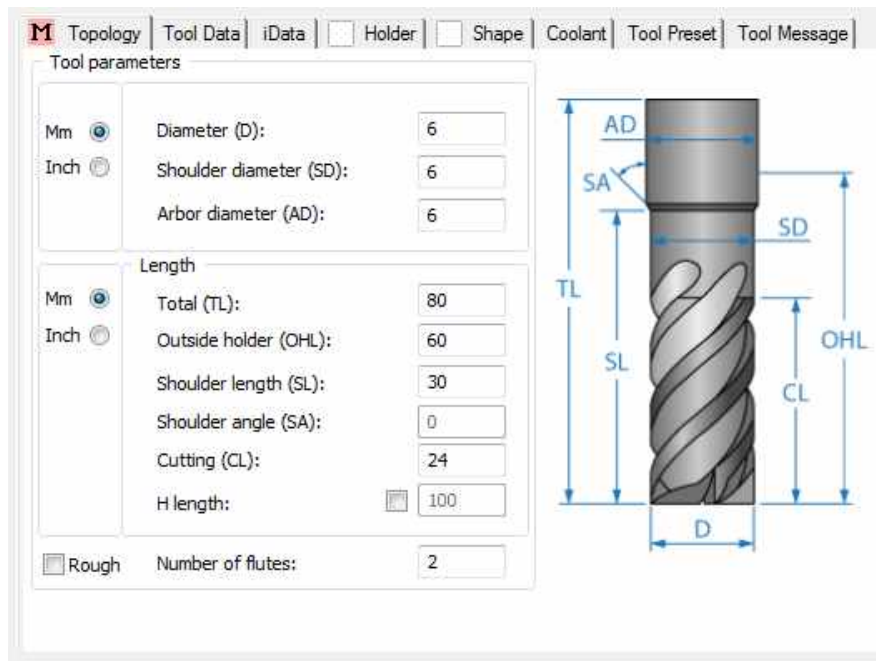


Fig. 71. Tool Defining for Rough Milling

On the Levels page, choose the Upper level and Profile depth (Fig. 72).

Positioning levels

Start level: 25 Delta: 0

Clearance level: 25 Delta: 0

Safety distance: 2

Milling levels

User defined

Upper level: 0 Delta: 0

User defined

Profile depth: 15 Delta: 0

Fig. 72. Levels Set-up

Then, set Technology parameters (only rough milling and 1.8 mm for wall offset for the following finish milling) as shown in Fig. 73.

Technology | Advanced

Modify

Tool side: Right Geometry

☐ Ignore tool-path intersections with geometry

☒ Compensation on rough passes

☐ Compensation on finish passes

☐ Compensation on Clear offset passes

Depth type: Constant

Depth cutting type: ☒ One way ☐ Zigzag

Rest material\Chamfer: None

☐ Use fillet size for last cut

Internal: 0

External: 0

Offsets

Wall offset: 1.8

Floor offset: 0

☒ Equal step down

☒ Rough

Max step down: 4

☐ Finish

Number of passes: 1

Extension/Overlap: 0

Max step down: 0

☐ Combine Wall and Floor finish passes

Ext.corner strategy: Rounded

☐ Clear offset

Offset: 3

Step over: 1

☐ Thin walls strategy

☒ One way ☐ Zigzag

☐ Complete Z-level ☐ Sort by chains

Fig. 73. Technology Page of Open Pocket Milling

Fig. 74 shows the Link parameters. In the Lead in, select the *Arc* option and use the same for Lead out.

Fig. 74. Lead in and Lead out Parameters

To define the Finish milling of the pocket, copy the Rough milling operation and add a new spherical end mill (4 mm diameter). Tool parameters are shown in Fig. 75.

Fig. 75. Spherical End Mill for Finish Milling

Finish passes need to be defined on the Technology page (Fig. 76).

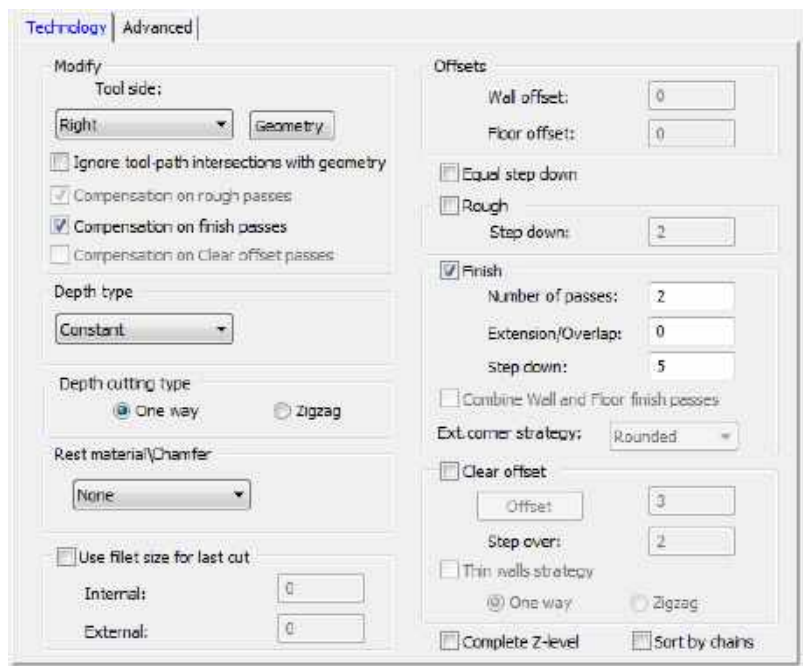


Fig. 76. Technology Set-up for Finish Milling

Click *Save and Calculate* , then simulate the operation in SolidVerify mode  (Fig. 77).

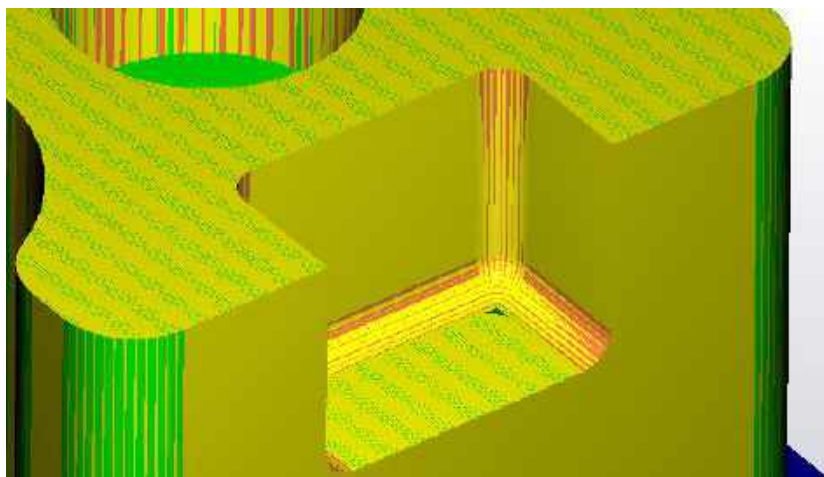


Fig. 77. Visualisation of the Obtained Pocket

2.7. Adding a New Coordinate System

Now, the bottom face of the workpiece must be machined. Since this requires a separate clamping, you must define another Machine Coordinate System for the next operation.

In the SolidCAM Manager, double-click the *CoordSys Manager* icon to display the corresponding dialog box, then select *Add* (Fig. 78).

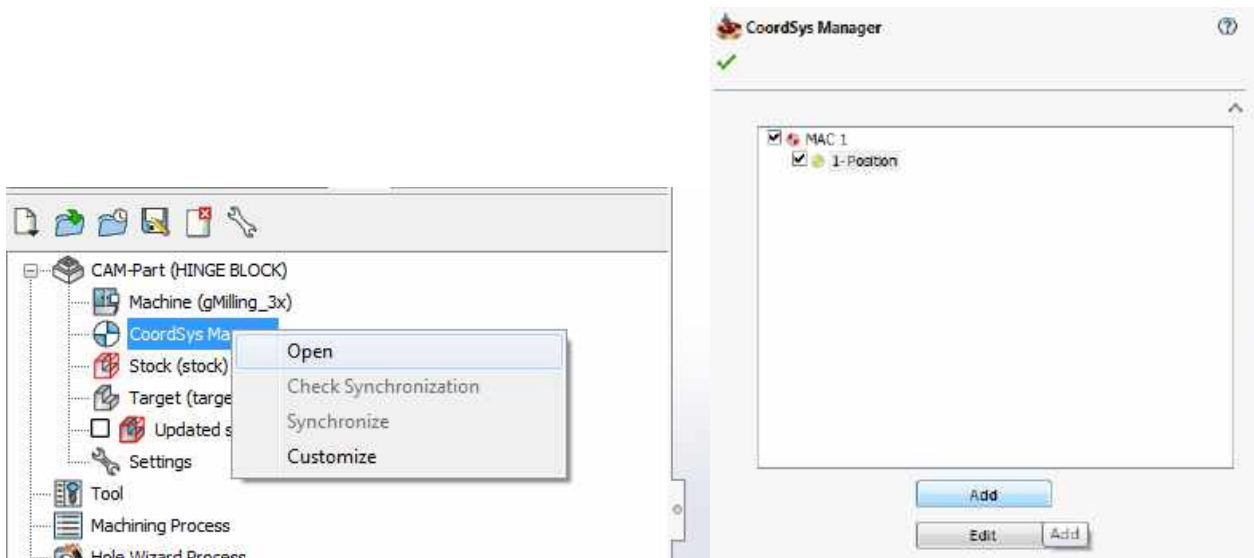


Fig. 78. Adding a New Coordinate System

In the CoordSys dialog box, change the Mac CoordSys number to 2 (e.g., MAC 2 - G55) (Fig. 79).

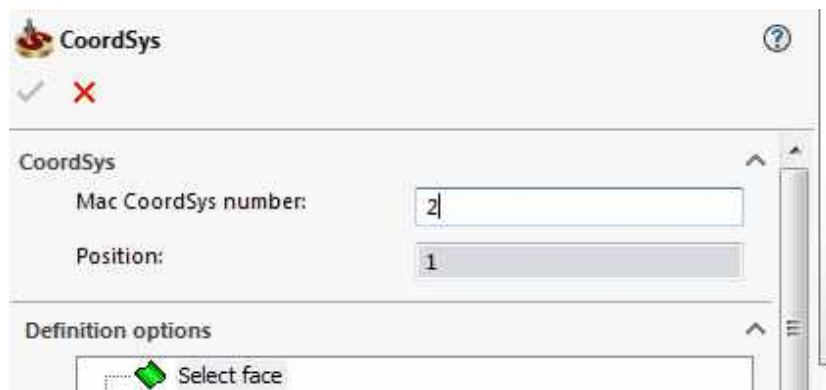


Fig. 79. Changing the Mac CoordSys Number

To center the origin of the Coordinate System, select the cylindrical surface of the through hole, as shown in Fig. 80.

The origin is located in the center of the solid model, but the Z-axis is facing the wrong direction.

Click the *Change to Opposite* button to quickly return the origin to the correct position (Fig. 81). To confirm the new Coordinate System definition, click *OK* .

Click *OK*  to accept the standard machining levels again and close the *CoordSys Manager* dialog box.

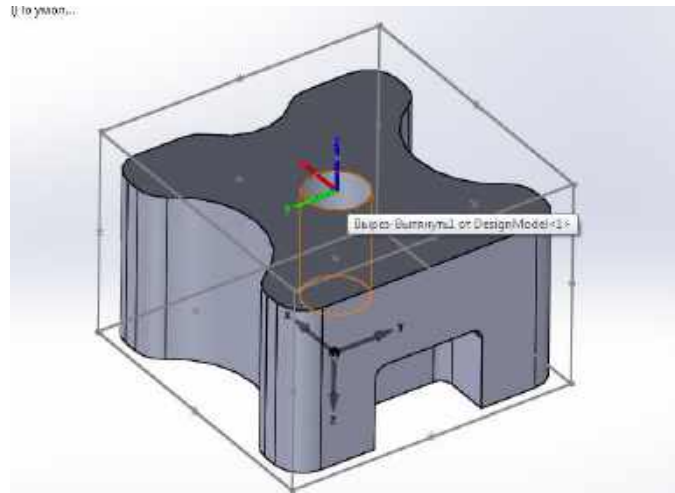


Fig. 80. Selecting a New Coordinate System on the Model

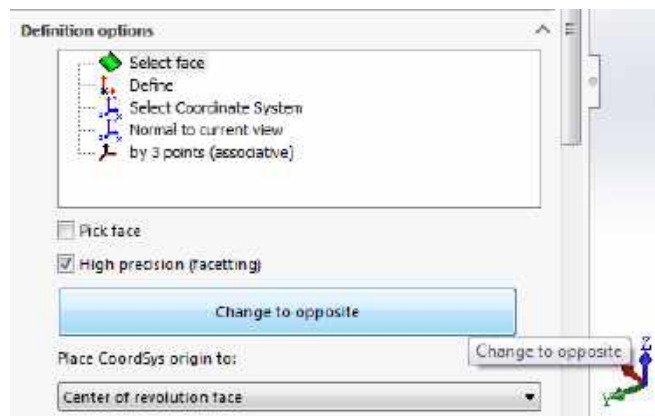


Fig. 81. Changing the Direction of the Z-axis

2.8. Adding a Face Milling Operation (2nd Setting)

Add another Face Milling operation to machine the workpiece finished height (30 mm).

Define the operation, following the same procedure as in section 1.5, with one difference: on the Geometry page, change CoordSys to *MAC 2* (2nd setting, Fig. 82).

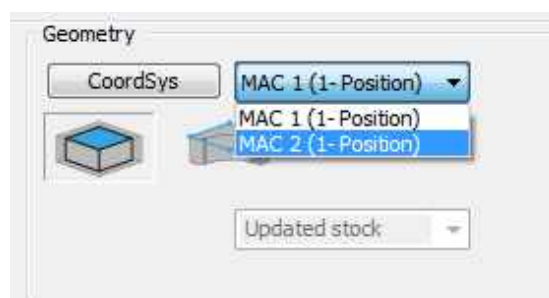


Fig. 82. Changing the Coordinate System

3. GENERATING G-CODE

Since this part is manufactured in two setups with separate clamping, each setup should generate a separate GCode file. SolidCAM allows you to insert Split in the program to do this easily.

Right-click the last operation under MAC 1 (Setup 1-1) in the CAM tree and select *After* from the *Split* submenu (Fig. 83).

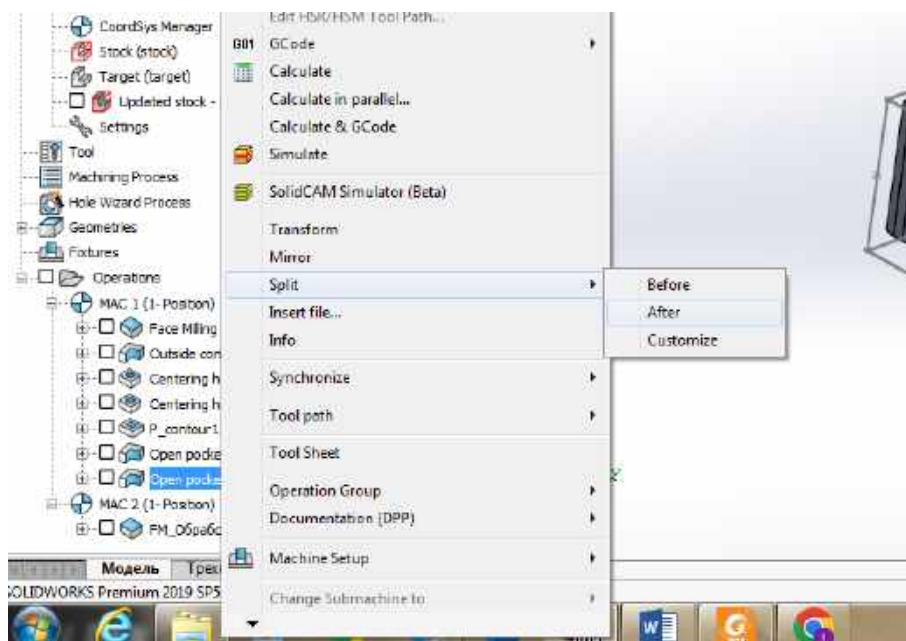


Fig. 83. Operation Split

Enter a name for the Split, such as the 2nd setup, and click *OK* (Fig. 84).



Fig. 84. Split Name

If several Coordinate Systems are defined, inserting a Split in the program before each one would be beneficial.

To generate GCode for all operations simultaneously, right-click the *Operations* header and select *Generate* command from the *GCode All* submenu (Fig. 85).

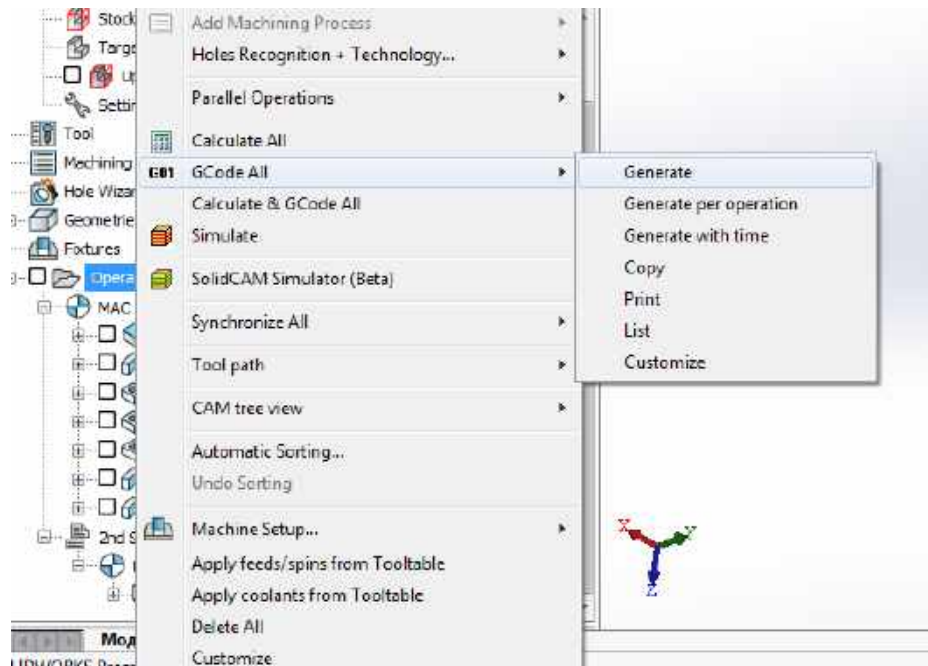
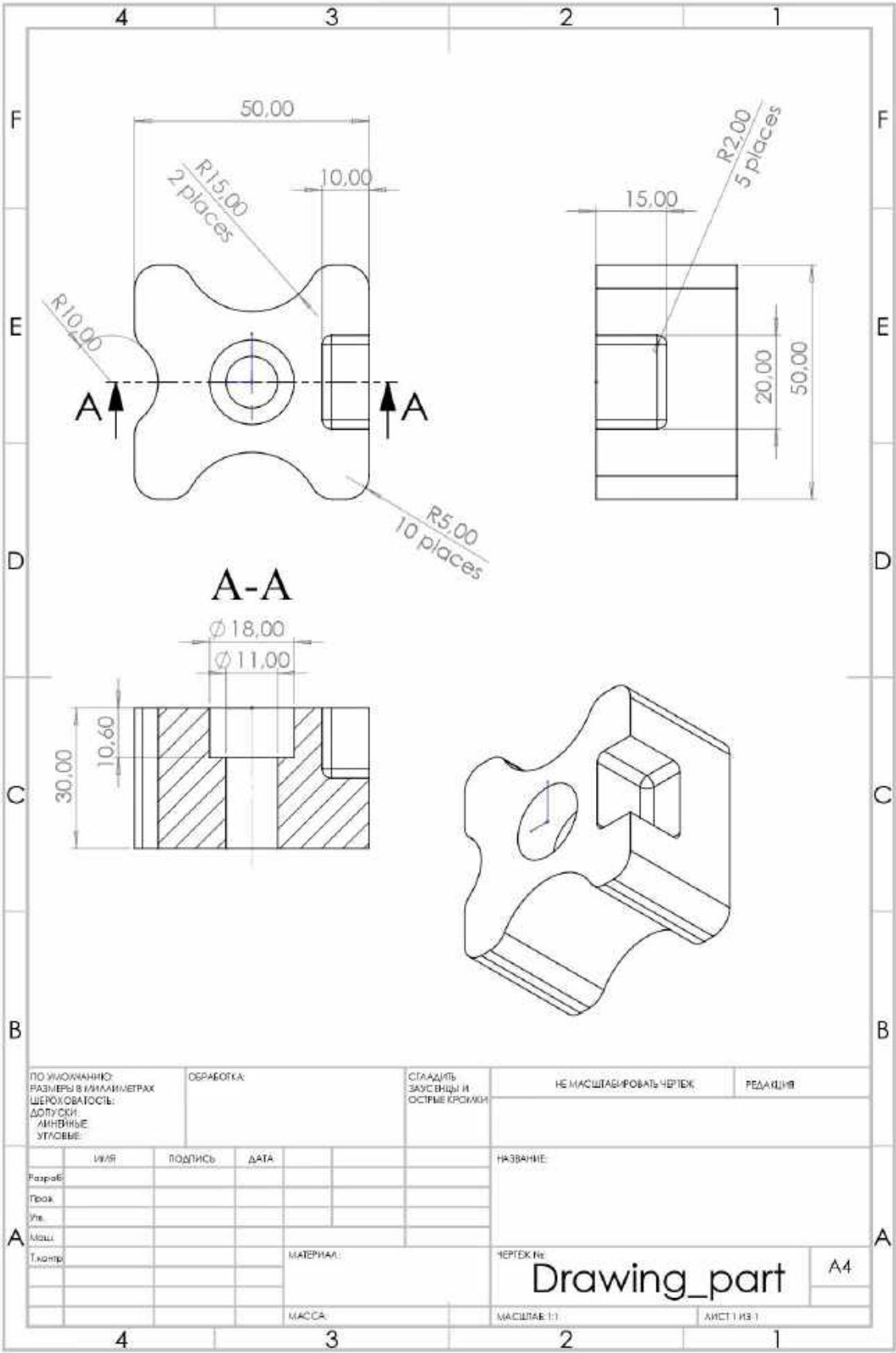


Fig. 85. Generating G-code

Depending on the post-processor, GCode is output as two separate files for both setups. The generated GCode files will be stored in the CAM-Part folder by default. You can print generated GCode using a text editor (such as Notepad) specified in the SolidCAM settings. You can also create GCode for a single operation or a specific group of operations.

The CAM program for the Hinge Block is completed.

The Part Drawing



REFERENCES

1. SolidWorks+SolidCAM EDU Training Course [Text]: Task-based CAD/CAM Training. – VDW-Nachwuchsstiftung GmbH : SolidCAM, 2021. - 233 p.
2. Milling Training Course: 2.5D Milling [Text]. – VDW-Nachwuchsstiftung GmbH : SolidCAM, 2016. - 311 p.

CONTENT

Introduction.....	3
1. MANUFACTURING WITH SOLIDCAM	4
1.1. Programming in SolidCAM	4
1.2. Creating and Defining a CAM-Part	5
1.2.1. Defining the Directory for Saving the CAM-Part	7
1.2.2. Selecting the CNC-Machine Controller.....	7
1.2.3. Defining the Machine Coordinate System	8
1.2.4. Defining the Stock and Target Models	12
1.3. Saving the CAM Part.....	15
1.4. SolidCAM User Interface	16
2. DEVELOPMENT OF MILLING OPERATIONS.....	18
2.1. Adding a Face Milling Operation (1st Setup).....	18
2.1.1. Defining the Tool	20
2.1.2. Defining the Levels.....	23
2.1.3. Defining the Technology.....	24
2.1.4. Simulating the Operation.....	26
2.2. Adding a Profile Operation	28
2.2.1. Defining the Tool	30
2.2.2. Determining the Profile Depth	32
2.2.3. Defining the Technology.....	34
2.2.4. Defining the Lead in and Lead out Tool Link Movements.....	36
2.2.5. Calculating and Simulating the Tool Path.....	38
2.3. Centering the Through Hole	39
2.3.1. Define the Tool and Tool Data	40
2.3.2. Defining the Drilling Depth.....	41
2.4. Drilling the Through Hole.....	42
2.4.1. Using Drill Cycles	43
2.5. Milling the Counterbore	43
2.6. Defining the Open Pocket Milling	46
2.7. Adding a New Coordinate System.....	49
2.8. Adding a Face Milling Operation (2nd Setting)	51
3. GENERATING G-CODE	52
Appendix. The Part Drawing.....	54
References	55

Навчальне видання

**Шорінов Олександр Володимирович
Невешкін Юрій Олександрович
Калініченко Микола Юрійович**

**РОЗРОБЛЕННЯ КЕРУЮЧОЇ ПРОГРАМИ ДЛЯ ВИГОТОВЛЕННЯ ДЕТАЛІ
НА ВЕРСТАТІ З ЧПК**

(Англійською мовою)

Редактор М. В. Кириленко
Технічний редактор А. М. Ємленінова

Зв. план, 2024

Підписано до видання 02.04.2024

Ум. друк. арк. 3,2. Обл.-вид. арк. 3,56. Електронний ресурс

Видавець і виготовлювач
Національний аерокосмічний університет ім. М. Є. Жуковського
«Харківський авіаційний інститут»
61070, Харків-70, вул. Чкалова, 17
[http:// www.khai.edu](http://www.khai.edu)
Видавничий центр «ХАІ»
61070, Харків-70, вул. Чкалова, 17
izdat@khai.edu

Свідectво про внесення суб'єкта видавничої справи
до Державного реєстру видавців, виготовлювачів і розповсюджувачів
видавничої продукції сер. ДК № 391 від 30.03.2001