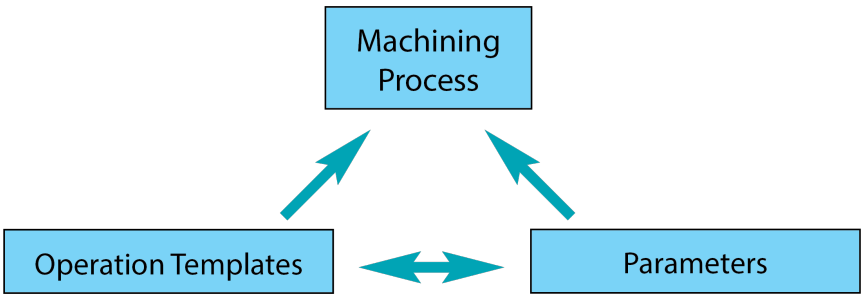
Machining Processes

SolidCAM enables you to automate repetitive tasks in CNC-Programming using the Machining Process feature.

Machining Process

Introduction & Basic concepts

SolidCAM enables you to automate repetitive tasks in CNC-Programming using the Machining Process feature. Machining Process (MP) is the structure of fully parameterized Operation Templates. In subsequent identical or similar situations, machining process templates can be either used directly or adapted to a new situation by giving values to their parameters.



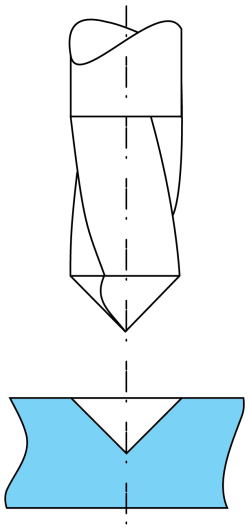
Operation templates

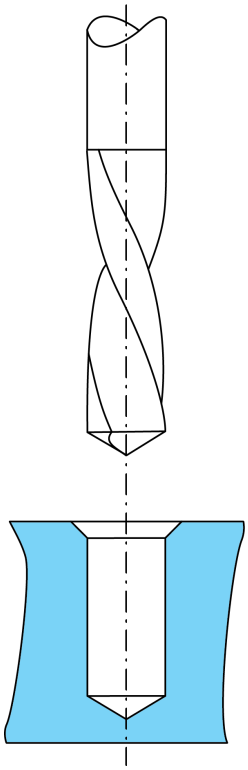
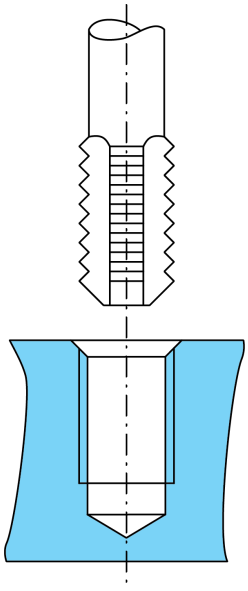
An Operation template is a parametric template of a single operation that is used within the Machining Process. There can be one or several operation templates in a Machining Process.

For example, to create a threaded countersink hole, we generally need three operations using the common geometry:

Centering

At this stage, preliminary centering is performed for the hole and the chamfer.



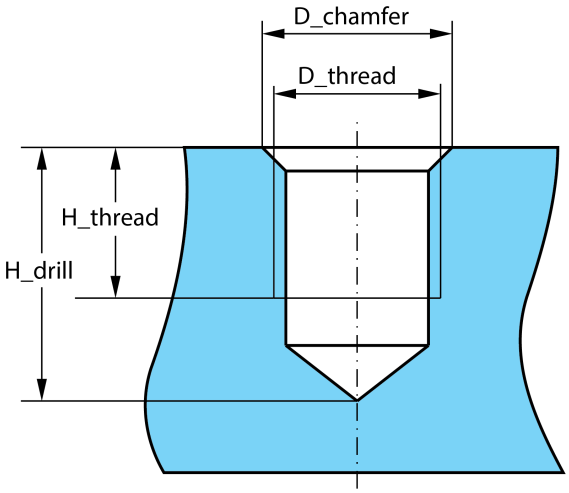
Drilling At this stage, the hole is drilled.	
Threading At this stage, the threading is performed.	

Parameters & Expressions

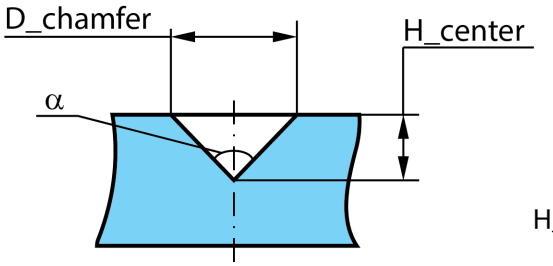
Parameters can be given a constant value or an expression.

In the above example, the following parameters are required:

- The tap size (D_thread);
- The tap depth (H_thread);
- The drill depth (H_drill);
- The chamfer diameter (D_chamfer).



In the above example, you can define the following expression for centering depth.

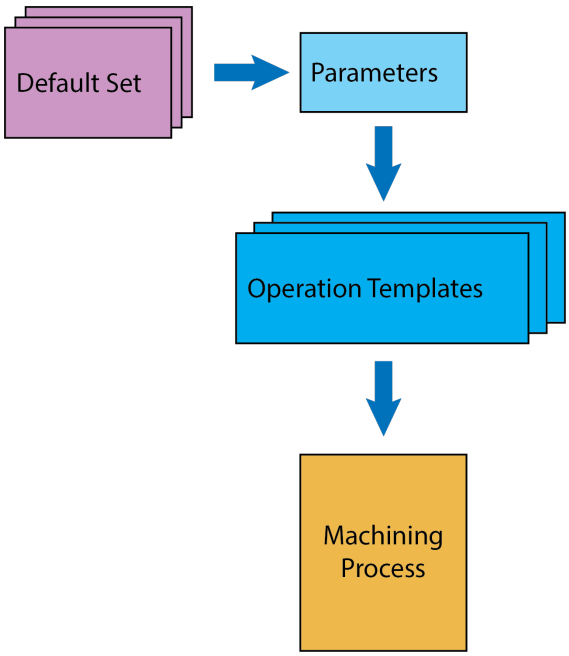


$$H\_center = \frac{D\_chamfer/2}{\tan(\alpha/2)}$$

With a group of such Operation templates, Parameters and Expressions, you can define the universal Machining Process.

Default sets

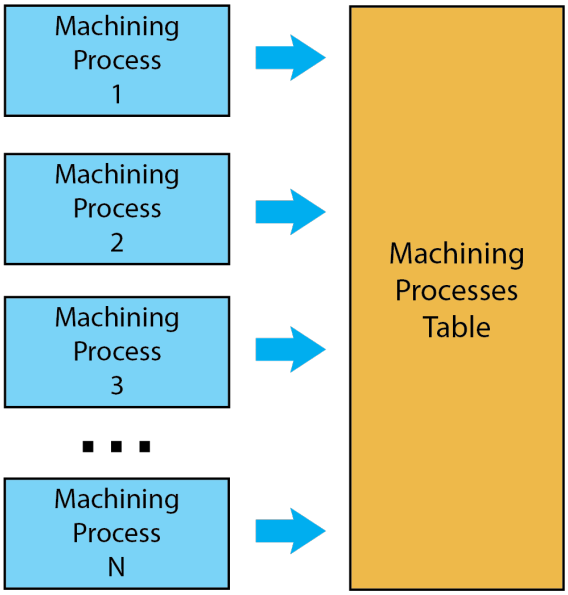
A **Default set** is a combination of parameters and expressions used for operation template customization. You can define a number of default sets by attaching different specific values or expressions to parameters. You can use default sets to adapt machining processes to specific tasks.



In the above example, you can create default sets for each tap size (e.g. M6, M8, etc.).

Machining Process Table

SolidCAM enables you to include your Machining Processes into a Machining Process Table.

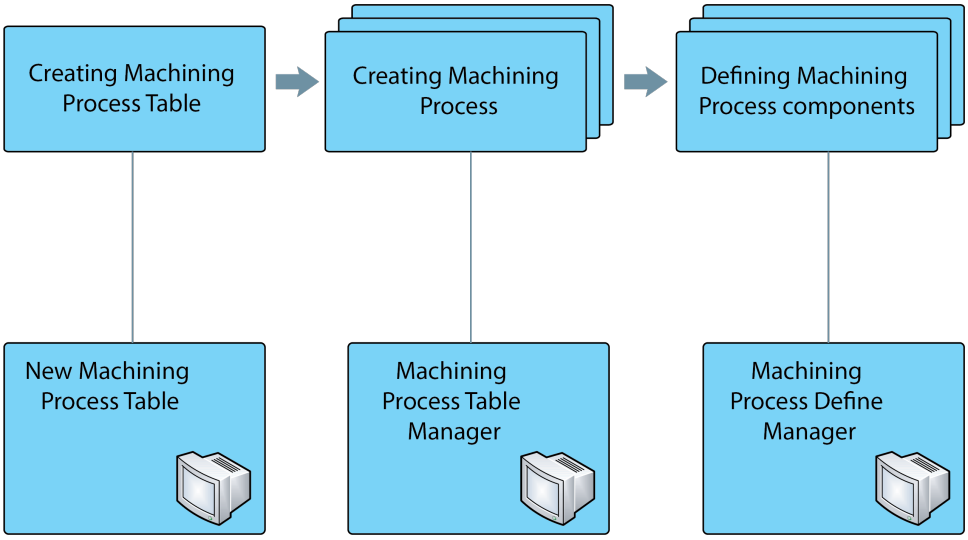


Related Topics

Creating Machining Processes - Introduction
Machining Processes usage - Introduction

Creating Machining Process

Three basic stages are required to create a Machining Process.



SolidCAM enables you to use three major interface tools for these tasks:

- **New Machining Process Table** - in this dialog box, you can define a new MP Table. SolidCAM also enables you to manage existing MP Tables.
- **Machining Process Table Manager** enables you to create and manage a single Machining Process in the Table.
- **Machining Process Define Manager** enables you to define components of a single Machining Process: Operation Templates, Default sets, Parameters and Expressions.

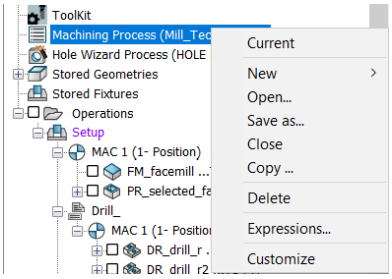
Related Topics

Machining Process - Introduction and Basic concepts
Machining Processes usage - Introduction

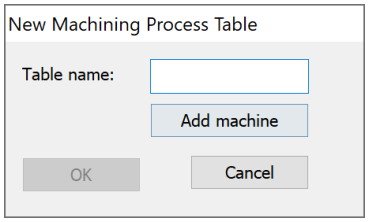
Defining Machining Process Table

The first stage of the Machining Process creation is the definition of the Machining Process Table.

1. In SolidCAM Manager, right-click the **Machining Process** header and choose the **Milling** type of the MP table from the **New** submenu.



The **New Machining Process Table** dialog box is displayed.



2. Enter a name for the Machining Process Table in the **Table name** field.
3. Click **Add machine** to define the machine controller for your Machining Process Table. Since not all machines support all of the options in SolidCAM, you must also specify the file that you will be using. This will assure that only the options supported by your machine are used.
4. Confirm the Machining Process Table creation with **OK.**

Related Topics

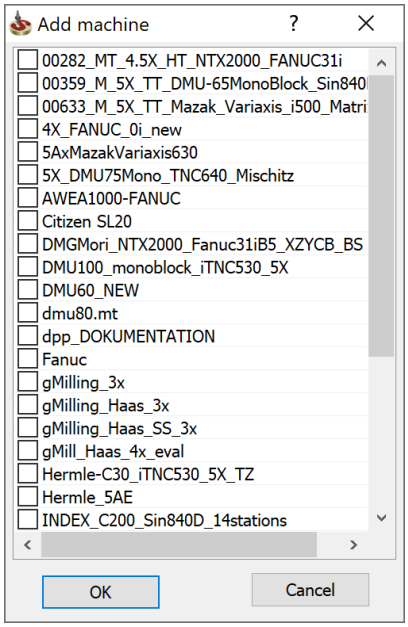
Managing Machining Process Tables
Machining Process Table Manager
Machining Process - Introduction and Basic concepts

Adding machine files

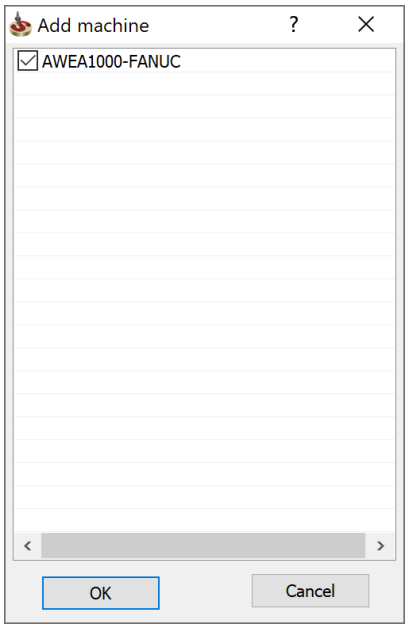
The Add machine dialog box enables you to choose a machine definition file (or a set of similar machine definition files) for your Machining Process. The machine definition file contains options oriented to the specific CNC-Machine. This file enables you to adapt your Machining Processes for your CNC-Machines.

Adding a machine

1. Click **Add** to choose the suitable machine definition file from the list.
2. In the Add machine dialog box, click on the check box adjacent to the machine definition file you wish to work with and click **OK**.

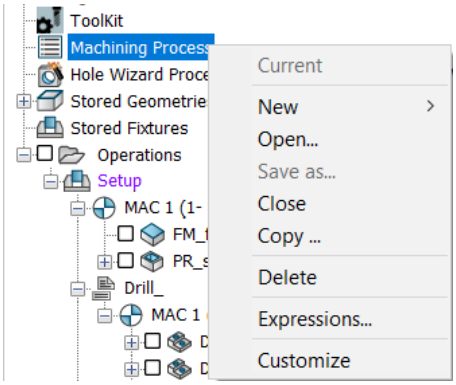


The Add machine dialog box is displayed again with the chosen machine definition file.



Managing Machining Process Tables

The Machining Process Tables menu contains the following options:



- **Current**

This option enables you to load the current MP Table in the Machining Process Table Manager.

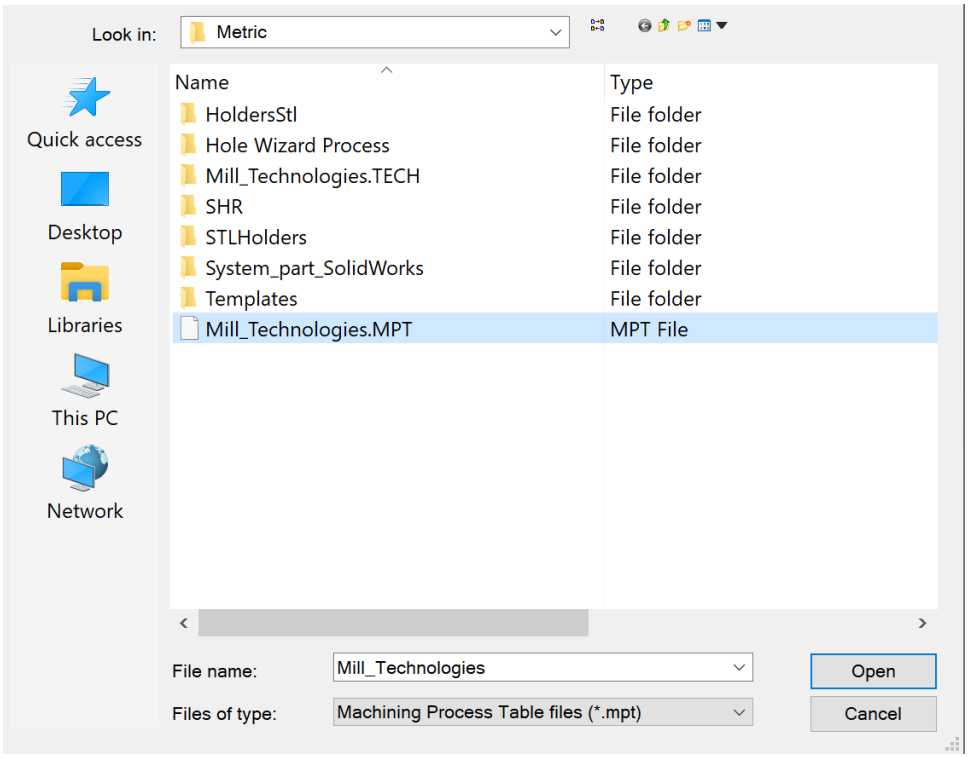
- **New**

With this option you can create a new MP Table. SolidCAM enables you to create four types of MP Tables: Milling, Turning, Mill-Turn, and Wire Cut.

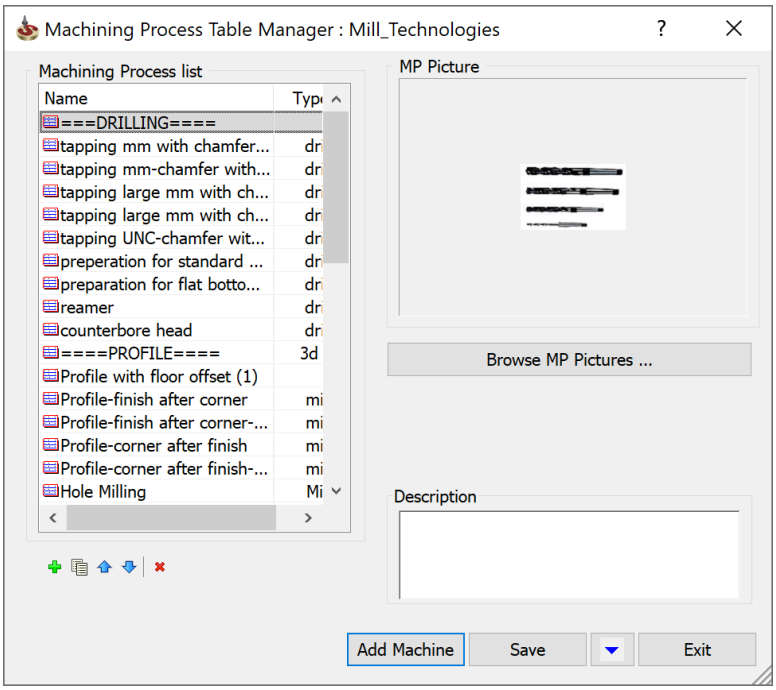
- **Open**

  This option enables you to load an MP Table via the standard Windows browser.

  1. Right-click the **Machining Process** header and choose **Open** from the submenu to load the Machining Process Table. The browser window is displayed.
  2. Choose a Machining Process Table file (***.mpt**).

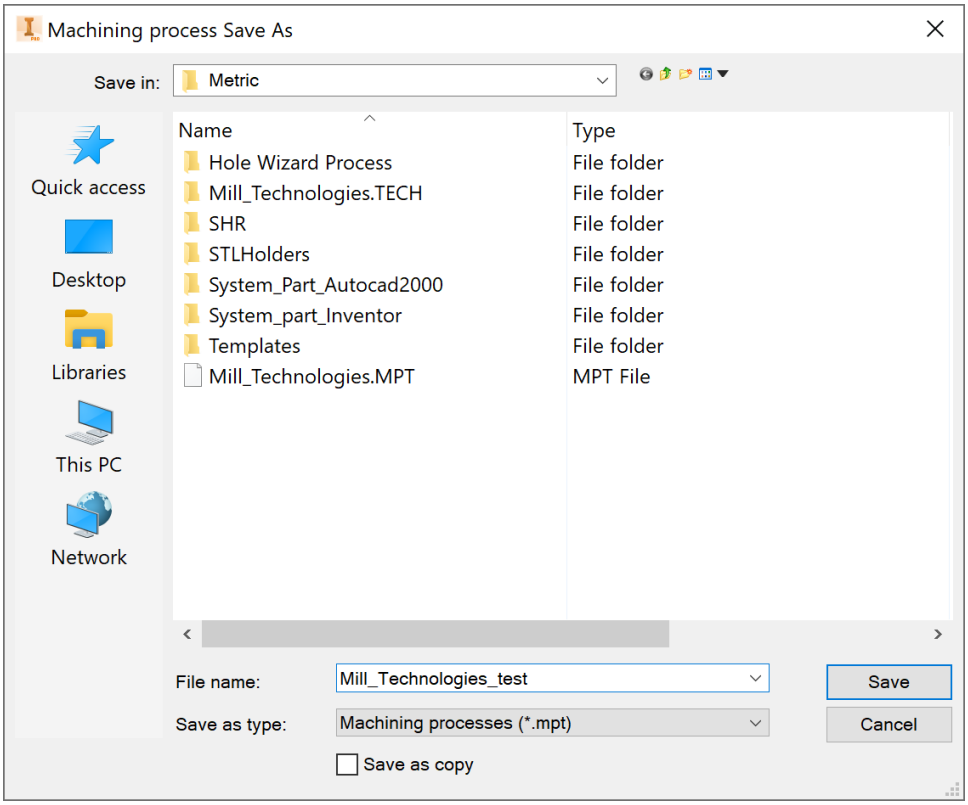


The Machining Process Table is loaded and the Machining Process Table Manager is displayed.



- **Save as**

This option enables you to save the current Machining Process Table to the disk with a new name or in a new location. When this option is chosen, the Machining process Save As dialog box is displayed.



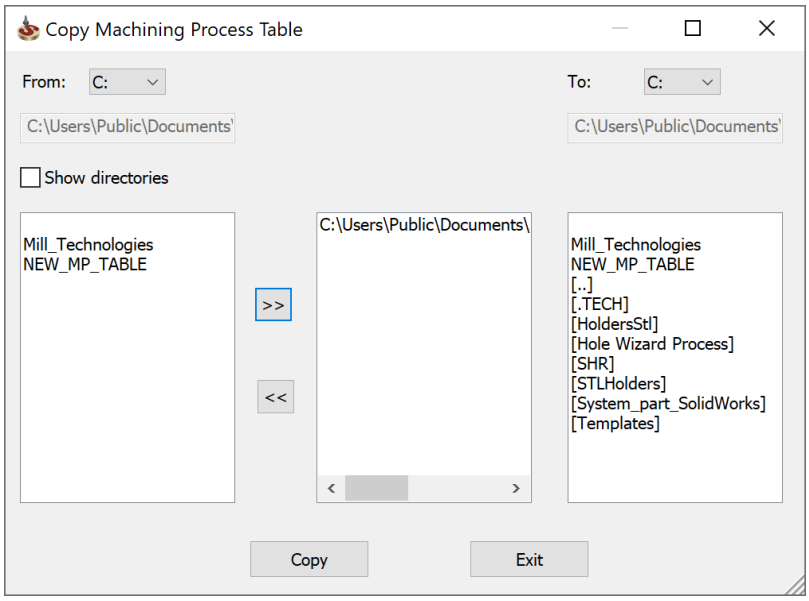
- **Close**

This option enables you to close the current Machining Process.

- **Copy**

  This option enables you to copy the MP Table.

  The Copy Machining Process Table dialog box is displayed.



1. In the left field of the dialog box, choose an MP Table.
2. Click >> to move the MP Table in the copy-container field. You can use the << button to remove MP Tables.
3. Choose the copy location in the right field.
4. Click **Copy** to confirm the operation.
5. Click **Exit** to close the dialog box.

- **Delete**

Choose this option to delete the MP Table via the standard Windows Browser.

- **Recent MP Tables**

This option displays a list of the last MP Tables you loaded in **SolidCAM**. You can open the file by clicking the file you want to open.

- **Group Types**

This option enables you to create your own MP Group Type.

- **Expressions**

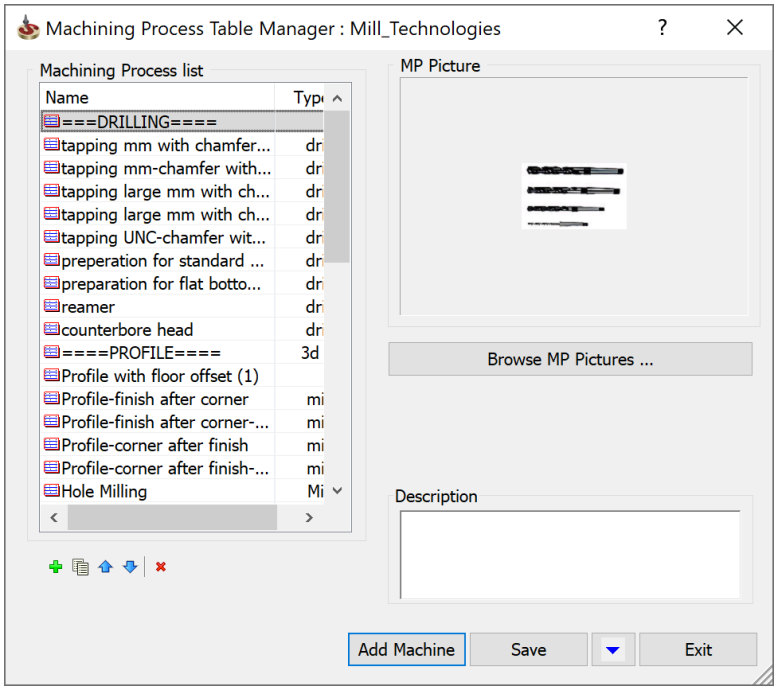
This option enables you to create and manage your own list of expressions that will be available in each MP you design.

Related Topics

Defining Machining Process Table
Machining Process Table Manager
Machining Process - Introduction and Basic concepts

Machining Process Table Manager

With the Machining Process Table Manager, SolidCAM enables you to control single Machining Processes in the Machining Process Table.



The Machining Process Table Manager dialog box contains the following fields:

- **Machining Process list**

This field contains the list of all Machining Processes from the current table.

Double-click on the Machining Process name in the list to edit MP in the Machining Process Define Manager.

Right-click on the Machining Process name in the list to open the menu of Managing Machining Processes.

During the Machining Process definition, SolidCAM enables you to divide the Machining Processes in the MP Table into groups. Each group name has to be started with the "**==**" symbols. All the Machining Processes following the group name (till the next group name) will be included into the group.

In the Add Machining Processes menu, each such group will be displayed as a separate submenu.

In the **Type** column, you can define and use your own **MP Group Types** for easier identification and classification of Machining Processes.

The following buttons enable you to manage the Machining Processes contained in the list:

- adds a new Machining Process to the Machining Process Table.
- creates a copy of the selected Machining Process and adds it to the bottom of the list.
- moves the selected Machining Process up the list.
- moves the selected Machining Process down the list.
- removes the selected Machining Process from the Machining Process Table.

- **MP Picture**

This option enables you to attach a picture to your MP.

Click the **Browse MP Pictures** button and choose a bitmap file (***.bmp**) in the browser window.

- **Description**

In this field, you can enter a text describing the Machining Processes.

During the Machining Process definition SolidCAM enables you to divide the Machining Processes in the MP Table into groups. Each group name has to be started with the "==" symbols. All the Machining Processes following the group name (till the next group name) are included into the group.

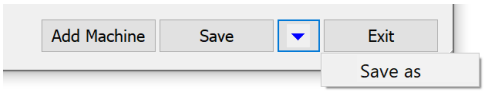
In the Add Machining Processes menu, each such group is displayed as a separate submenu.

- **Add Machine**

- **Save**

This button enables you to save the current state of the Machining Process Table.

- **Save as**



This button enables you to save the current Machining Process Table to the disk with a new name or in a new location.

- **Exit**

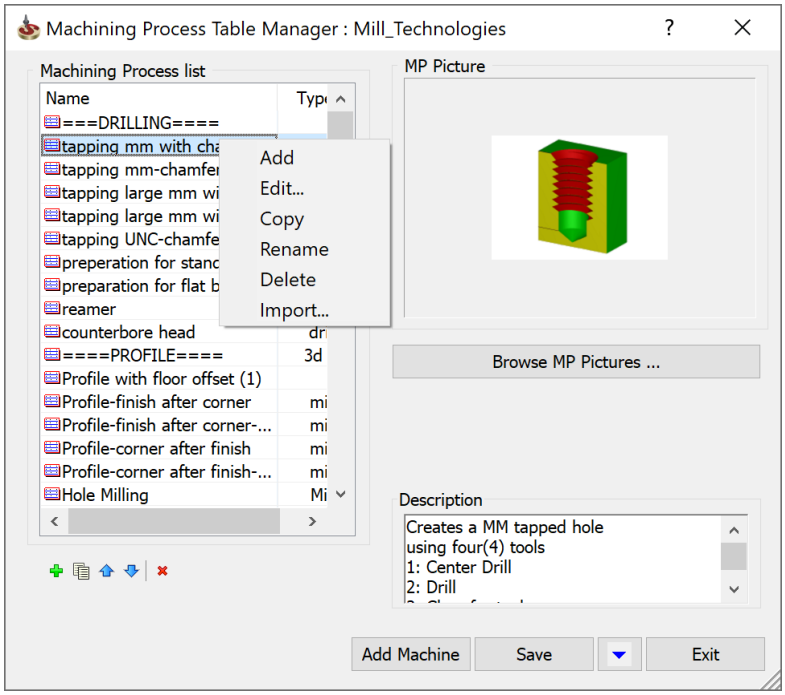
This button enables you to exit the Machining Process Table Manager dialog box.

Related Topics

Defining Machining Process Table
Managing Machining Processes
Machining Process Define Manager

Managing Machining Processes

Right-click the Machining Process name in the list to open the menu of Machining Process managing options.



- **Add**

This option enables you to add a new Machining Process to the Machining Process Table.

- **Edit**

This option enables you to edit the components of the Machining Process in the Machining Process Define Manager.

- **Copy**

This option enables you to create a copy of the selected Machining Process and adds it to the bottom of the list.

- **Rename**

This option enables you to rename the selected Machining Process.

- **Delete**

This option enables you to remove the selected Machining Process from the Machining Process Table.

- **Import**

This option enables you to insert a Machining Processes from another MP Table into the current MP Table.

	You can insert Machining Processes from MP Tables defined with similar MAC files.

1. Right-click on the Machining Process name and choose **Import** from the menu.
2. Choose the MP Table from the Browser. The MP Export Table Manager dialog box is displayed.
3. Choose the Machining Process from the Manager and confirm with **Export**.

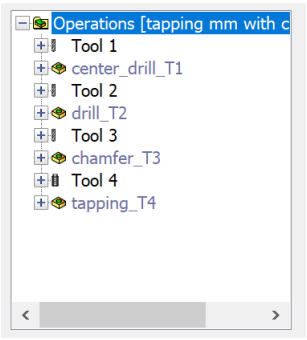
Related Topics

Machining Process Table Manager
Machining Process Define Manager

Machining Process Define Manager

Operation Templates page

On this page, you can work with the set of operation templates in the same way as you work with conventional SolidCAM operations via SolidCAM Manager.



There are four options to manage your operation templates:

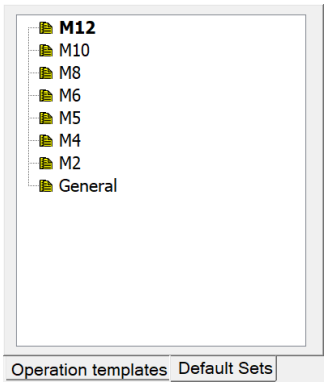
- **Add**
- **Edit**
- **Delete**
- **Change Tool**

See Also

Managing operation templates

Default Sets page

On this page you can work with the set of Default Sets created in the actual Machining Process.



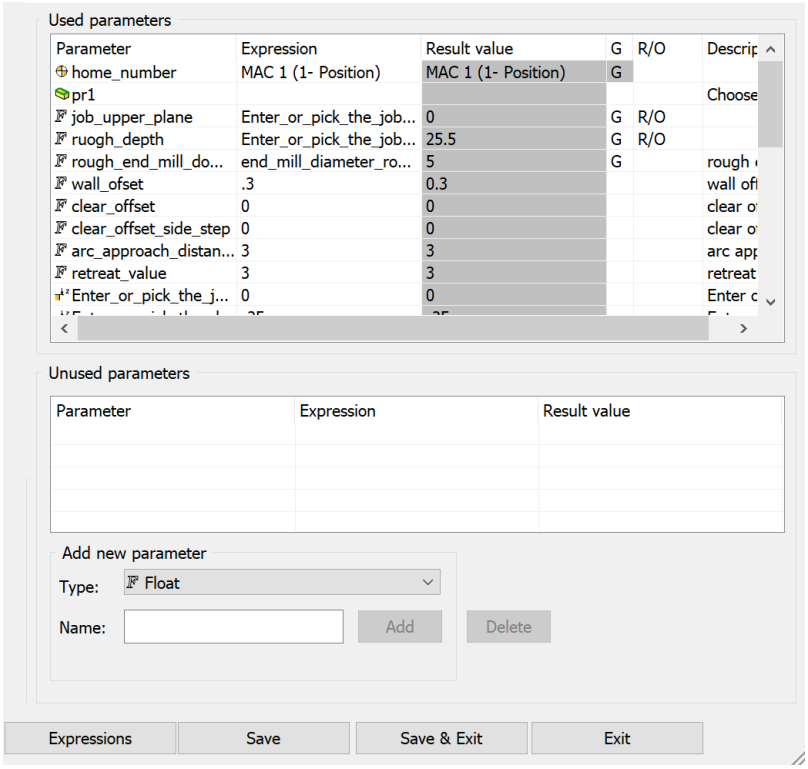
There are four options to manage Default Sets:

- **Set as Current**
- **Rename**
- **Delete**
- **Copy**
- **Add**

See Also

Managing Default Sets

Parameters & Expressions Tables



There are two areas in the Machining Process Define Manager that enable you to manage your parameters and expressions:

- **Used Parameters Table**

This area shows the table of parameters that were used in the Operation Templates with the actual Default Set.

- **Unused parameters Table**

This area shows the table of parameters that were not used in the Operation Templates with the actual Default Set.

See Also

Used/Unused Parameters Tables

Add new parameter

This field enables you to add a new parameter.

1. 1. Choose the new **parameter type** from the list.
   2. Type the **parameter name**.
   3. Click **Add** to enter the parameter to the table.

	With **Delete**, you can delete the selected parameter item from the Unused Parameters table.

Related Topics

Machining Process Table Manager
Defining Machining Process Table

Managing operation templates

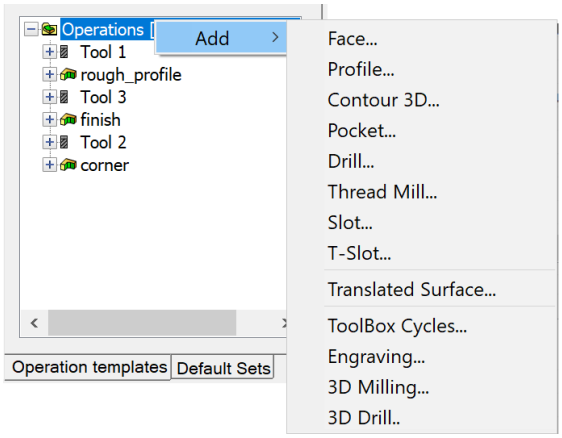
An Operation Template is a parametrically defined single SolidCAM operation. In addition to numerical values for the definition of technological parameters, you can use variables and expressions in the operation templates.

Double-click the **Machining Process** in the CAM tree to open the Machining Process dialog box. Double-click the required operation in the Machining Process list to open the Machining Process Define Manager where the following options enable you to manage your operation templates:

- **Add**

  The process of the operation template creation is the same as creating operations via SolidCAM Manager.

1. Right-click the **Operations** field in the **Operation Templates** tab.



2. Choose an operation template type from the menu.

See Also

Operation types

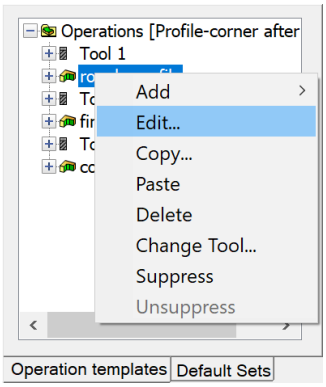
The corresponding operation template dialog box is displayed.

See Also

Defining operation templates

- **Edit**

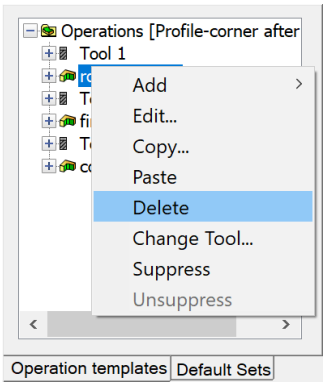
This option enables you to edit the operation template.



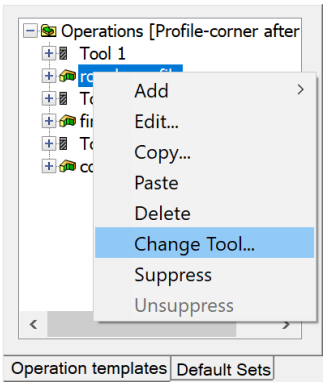
The corresponding operation template dialog box is displayed.

- **Delete**

This option enables you to delete the operation template.



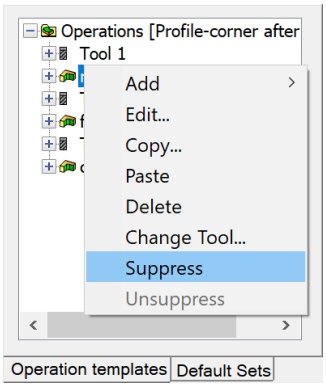
- **Change Tool**



This option enables you to change the tool used in the specific operation template. The Part Tool Table dialog box is displayed.

- **Suppress/Unsuppress**

SolidCAM enables you to suppress and unsuppress operation templates in the Machining Process.



The rules for suppressing and unsuppressing of the operation templates are the same as for operations. The suppressed/unsuppressed status of an operation template is the property of the operation template per default set, i.e. the same operation template can have a different status in different default sets. SolidCAM inserts suppressed operation templates from the Machining Process into the CAM-Part as suppressed operations.

Related Topics

Machining Process Define Manager
Managing Default Sets

Defining operation templates

The operation template dialog box contains the same fields you can find in the SolidCAM operation dialog box.

You can now fill the fields of the operation template in the same way as if you define a conventional operation. Note that some fields in the operation templates are marked with the button. These fields are parametric. In these fields, you can enter the following data:

- Variable
- Expression
- Value

	If a variable you entered does not exist, SolidCAM adds it to the Used Parameter Table.

To edit this parametric field, you can use the parametric field menu by clicking the button.

If you entered incorrect data in the parametric field (unacceptable type of variable, error in expression, etc.), SolidCAM marks this parametric field with the button.

	In the operation template, the definition of the Geometry and the Tool is mandatory.

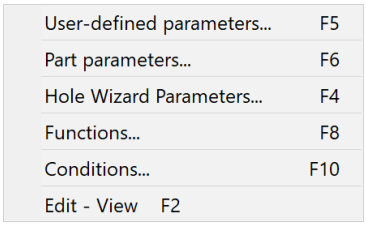
Related Topics

Machining Process Define Manager
Managing operation templates

Parametric field menu

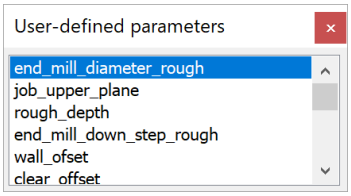
The parametric field menu enables you to choose parameters and expressions easily and effectively.

When you click the button, the following menu is displayed:



- **User-defined parameters**

This option displays a list of existing user-defined parameters.

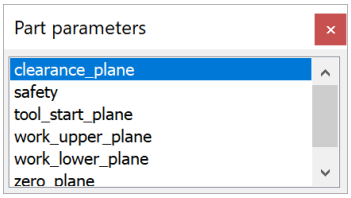


Choose the variable name from the list.

	You can open this list with the F5 hotkey when the cursor is placed in the parametric field.

- **Part parameters**

This option displays a list of existing part parameters.

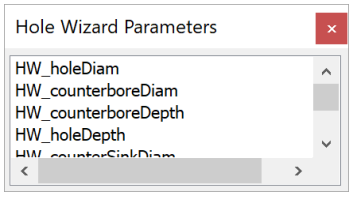


Choose the variable name from the list.

	You can open this list with the F4 hotkey when the cursor is placed in the parametric field.

- **Hole Wizard Parameters**

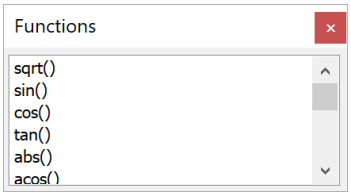
This option displays a list of existing hole wizard parameters.



	You can open this list with the F6 hotkey when the cursor is placed in the parametric field.

- **Functions**

This option displays the list of standard SolidCAM functions.



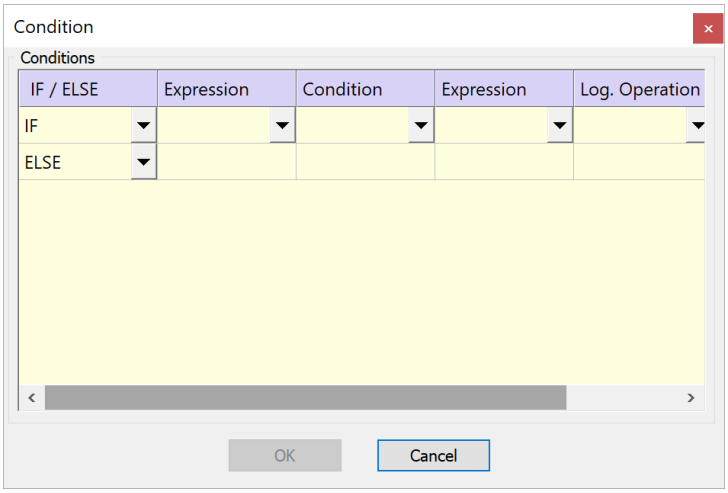
	You can open this list with the F8 hotkey when the cursor is placed in the parametric field.

See Also

Standard functions

- **Conditions**

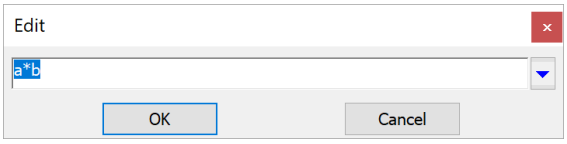
This option displays the table of standard SolidCAM conditions.



	You can open this list with the F10 hotkey when the cursor is placed in the parametric field.

- **Edit-View**

This option displays the Edit dialog box.



In this dialog box, you can see and edit the full parameter name or expression string.

	1. You can open this list with the F2 hotkey when the cursor is placed in the parametric field. 2. In this dialog box, you can also use the operational menu by clicking the button.

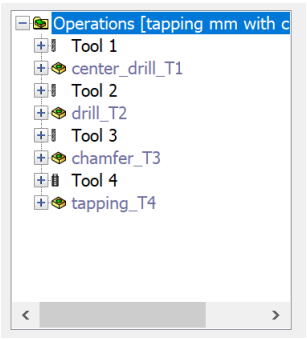
Related Topics

Variables & expressions
Used/Unused Parameters Tables
Machining Process Define Manager

Machining Process Define Manager

Operation Templates page

On this page, you can work with the set of operation templates in the same way as you work with conventional SolidCAM operations via SolidCAM Manager.



There are four options to manage your operation templates:

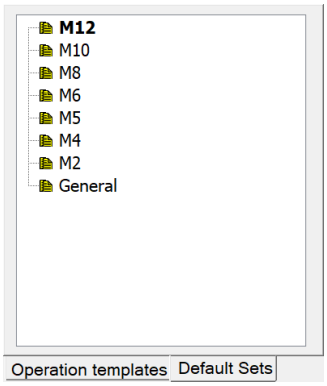
- **Add**
- **Edit**
- **Delete**
- **Change Tool**

See Also

Managing operation templates

Default Sets page

On this page you can work with the set of Default Sets created in the actual Machining Process.



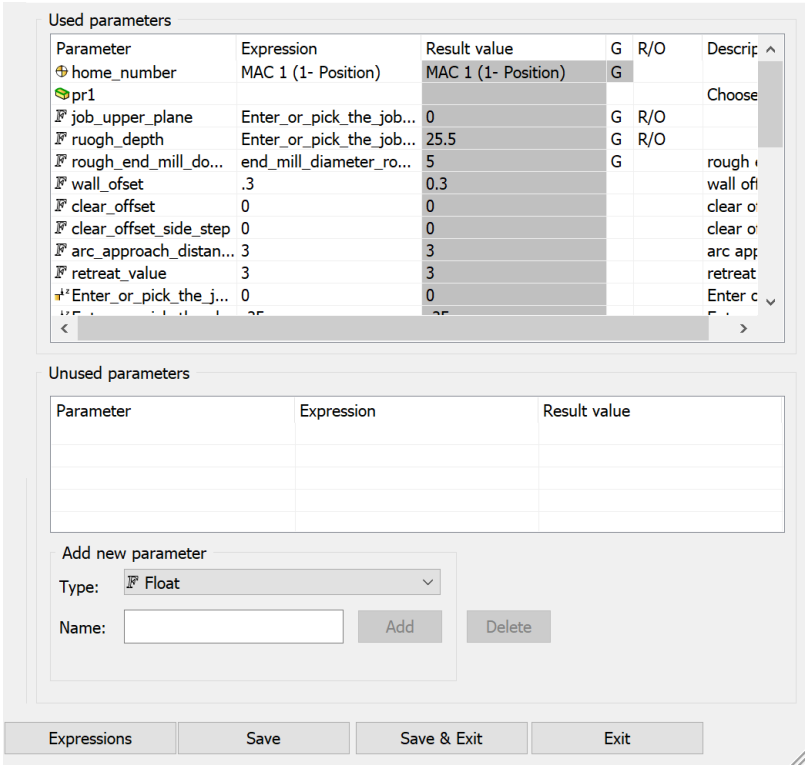
There are four options to manage Default Sets:

- **Set as Current**
- **Rename**
- **Delete**
- **Copy**
- **Add**

See Also

Managing Default Sets

Parameters & Expressions Tables



There are two areas in the Machining Process Define Manager that enable you to manage your parameters and expressions:

- **Used Parameters Table**

This area shows the table of parameters that were used in the Operation Templates with the actual Default Set.

- **Unused parameters Table**

This area shows the table of parameters that were not used in the Operation Templates with the actual Default Set.

See Also

Used/Unused Parameters Tables

Add new parameter

This field enables you to add a new parameter.

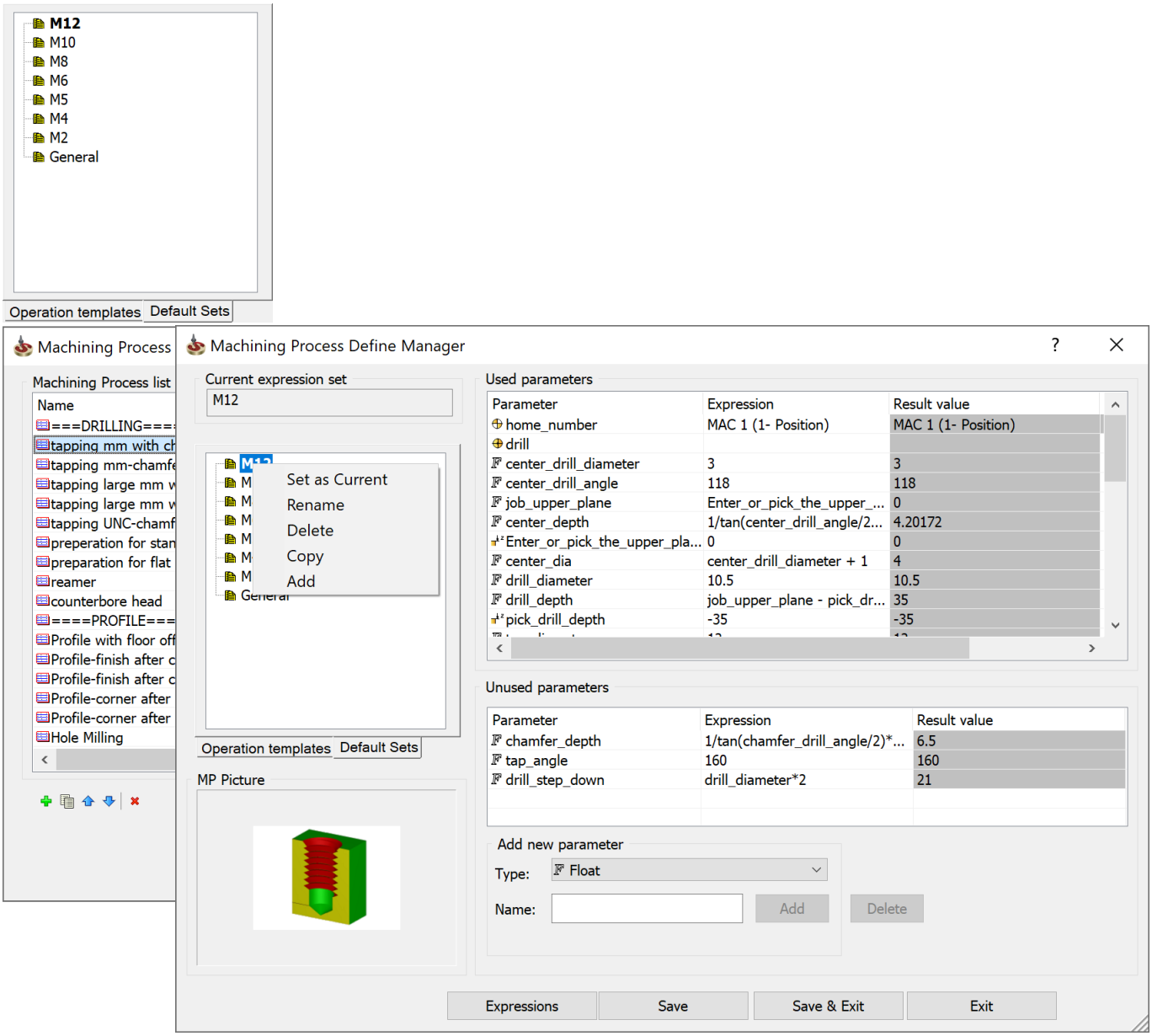
1. 1. Choose the new **parameter type** from the list.
   2. Type the **parameter name**.
   3. Click **Add** to enter the parameter to the table.

	With **Delete**, you can delete the selected parameter item from the Unused Parameters table.

Related Topics

Machining Process Table Manager
Defining Machining Process Table

Managing Default Sets



There are five options to manage Default Sets:

- **Set as Current**

This command enables you to activate the selected Default Set.

- **Rename**

This command enables you to rename the selected Default Set.

- **Delete**

This command enables you to remove the selected Default Set.

- **Copy**

This command enables you to copy the selected Default Set.

- **Add**

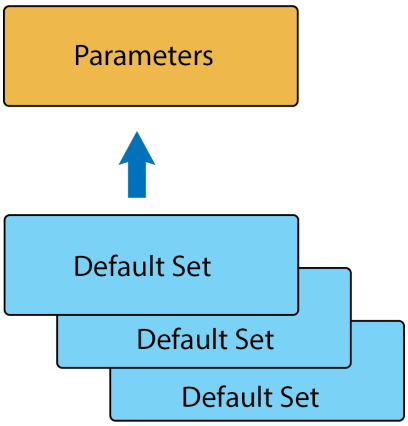
This command enables you to create a new Default Set.

Related Topics

Managing Default Sets
Machining Process Define Manager

Working with Default Sets

You can create a number of Default Sets and assign the specific combination of values and expressions to the parameters in each of them. The Default Sets enable you to adapt the Machining Process to specific tasks.



1. Create a new Default Set with the **Copy** option.
2. Activate the created Default Set with the **Current** option.
3. Change parameters definition via **Parameter Tables**.

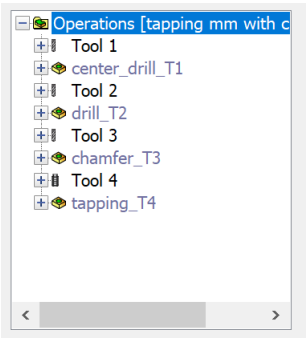
Related Topics

Managing Default Sets
Machining Process Define Manager

Machining Process Define Manager

Operation Templates page

On this page, you can work with the set of operation templates in the same way as you work with conventional SolidCAM operations via SolidCAM Manager.



There are four options to manage your operation templates:

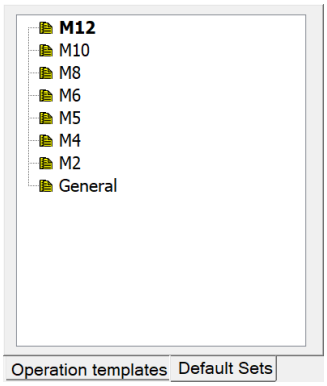
- **Add**
- **Edit**
- **Delete**
- **Change Tool**

See Also

Managing operation templates

Default Sets page

On this page you can work with the set of Default Sets created in the actual Machining Process.



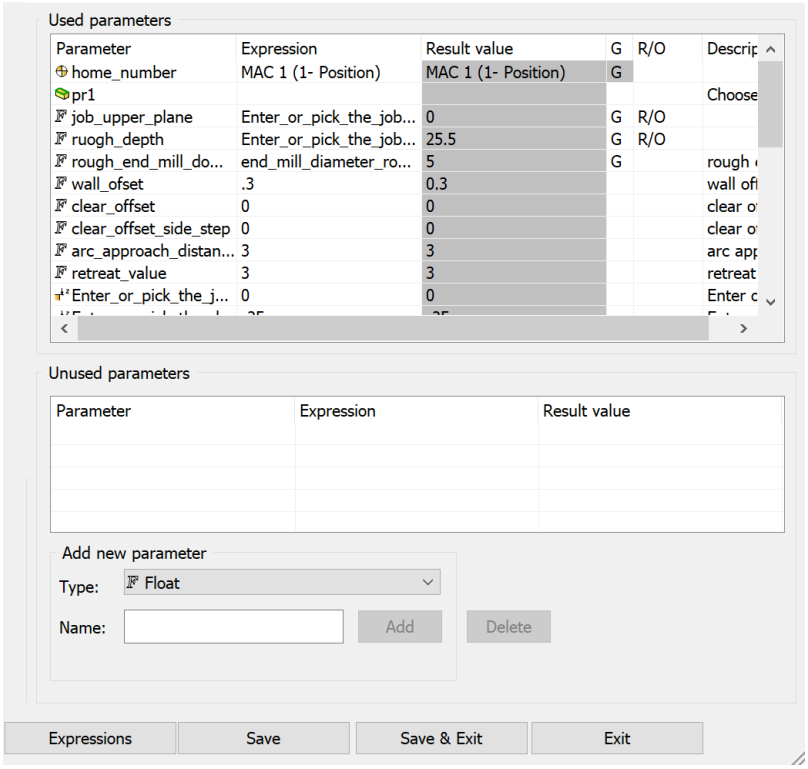
There are four options to manage Default Sets:

- **Set as Current**
- **Rename**
- **Delete**
- **Copy**
- **Add**

See Also

Managing Default Sets

Parameters & Expressions Tables



There are two areas in the Machining Process Define Manager that enable you to manage your parameters and expressions:

- **Used Parameters Table**

This area shows the table of parameters that were used in the Operation Templates with the actual Default Set.

- **Unused parameters Table**

This area shows the table of parameters that were not used in the Operation Templates with the actual Default Set.

See Also

Used/Unused Parameters Tables

Add new parameter

This field enables you to add a new parameter.

1. 1. Choose the new **parameter type** from the list.
   2. Type the **parameter name**.
   3. Click **Add** to enter the parameter to the table.

	With **Delete**, you can delete the selected parameter item from the Unused Parameters table.

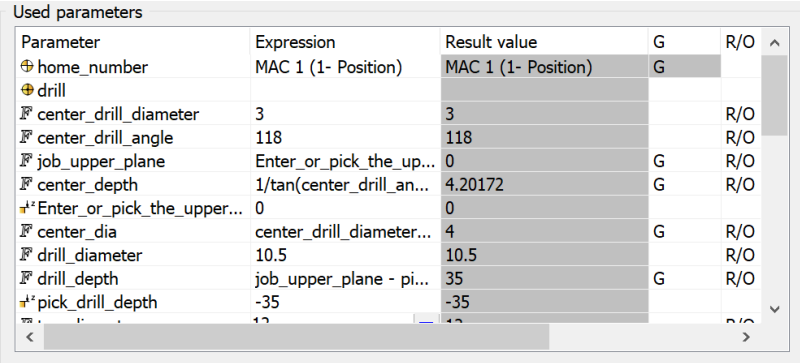
Related Topics

Machining Process Table Manager
Defining Machining Process Table

Used and Unused Parameter Tables

There are two specific fields in the Machining Process Define Manager that enable you to control your parameters and expressions:

Used Parameters Table



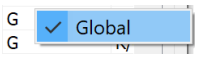
This area shows the table of parameters that were used in the operation templates with the actual Default Set.

This table contains the following columns:

- **Parameter** - this column shows the parameter name. You can change the name of the parameter. In this case the name will be updated in all operation templates that use this parameter.
- **Expression** - this column shows the parameter definition. There is either an expression or value here. To edit this parametric field you can use the parametric field menu available by clicking the button.

  See Also

  Parametric field menu
- **Result value** - this column shows the result value returned by the parameter.
- **G** - this column shows the status of the parameter. If the status is Global, the G letter is shown. In this case the expression of this value will be the same in all Default Sets.



By right-clicking this field, you can change the status of the parameter.

- **R/O** - this column shows the access status of the parameter. If the status is Read-only, the "R/O" letters are shown. In this case, the parameter definition cannot be changed during Machining process insertion.

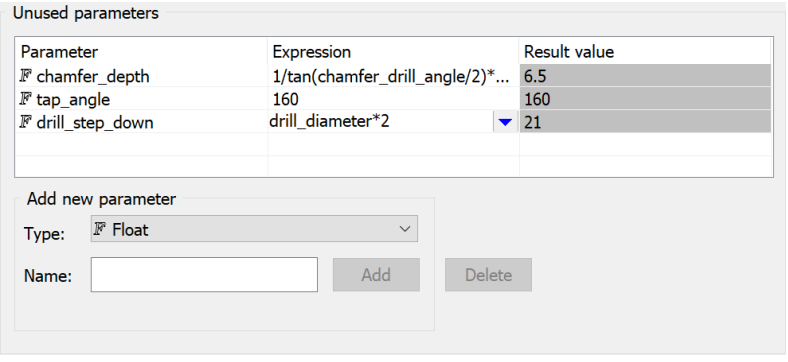
By right-clicking this field, you can change the read-only status of the parameter.

- **Description** - this column shows your own text description of the parameter. To edit this field, use the context menu available by clicking the button.

  See Also

  Parametric field menu#Edit

Unused parameters Table



This area shows the table of parameters that were not used in the operation templates with the actual Default Set.

This table contains the following columns:

- **Parameter** - this column shows the parameter name. In this field you can change the name of the parameter. In this case, the name will be updated in all operation templates that use this parameter.
- **Expression** - this column shows the parameter definition. There are either expressions or values here. To edit this parametric field, you can use the operational menu available by clicking the  button.

  See Also

  Parametric field menu

- **Result value** - this column shows the result value returned by the parameter.

Related Topics

Machining Process Define Manager
Variables & expressions

Variables and Expressions

SolidCAM enables you to use variables and expressions in your operation templates.

Variables

All variables can be divided into two groups depending on the way values are given to the variable.

- **Part parameters**

SolidCAM enables you to use a set of the standard built-in variables. All these variables get their values from the SolidCAM CAM-Part when you insert the Machining Process.

- **clearance_plane** - the Z-value of the Clearance Level.
- **safety -** the Z-value of the Safety Distance.
- **tool_start_plane -** the Z-value of the Tool Start Level.
- **work_upper_plane -** the Z-value of the CAM-Part Upper Level.
- **work_lower_plane -** the Z-value of the CAM-Part Lower Level.
- **zero_plane -** the Z-value of the Zero Level.

- **User-defined parameters**

You can define your own variables and use it in Operation Templates definition.

SolidCAM supports the following types of variables:

Profile Geometry variables are available for Profile Operation Templates.

Slot Geometry variables are available for Slot Operation Templates.

Limit Geometry variables are available for the Working Area definition in 3D Milling operation templates.

3D Model Geometry variables are available for 3D Milling operation templates.

Section Geometry variables are available for Section geometry in Slot operation templates.

Drill Geometry variables are available for Drilling operation templates.

Pocket Geometry variables are available for Pocket operation templates.

Z-Value variables enable you to store the Z-coordinates. The value of a variable of this type can be defined by picking on the solid model; the Z-coordinate of the picked position is used as the variable value.

Distance Value variables enable you to store the distance. The distance can be defined by picking two positions on the solid model.

String variables are used to store strings of symbols.

Float variables are used to store positive and negative floating point numbers.

Integer variables are used to store positive and negative whole numbers.

Tool Number variables are used to store the positive whole numbers used for the tool number definition.

Diameter variables enable you to store the diameters. The diameter value can be defined by picking a circular edge on the model.

	Enter the names for variables according to the following rules: • Use a letter as the first character; • Only capital and lowercase English letters, numbers and the underscore symbol (_) are acceptable; • Names cannot exceed 255 characters in length.

Expressions

SolidCAM enables you to use expressions for parameters definition.

You can use the following symbols in expressions:

- Addition (**+**)
- Subtraction (-)
- Multiplication (*****)
- Division (/)

Parentheses are acceptable.

When multiplication and division occur together in an expression, each operation is evaluated as it occurs from left to right. When addition and subtraction occur together in an expression, each operation is evaluated in the order of appearance from left to right. Parentheses can be used to override the order of precedence and force some parts of an expression to be evaluated before others. Operations within parentheses are always performed before those outside. Within parentheses, however, operator precedence is maintained.

SolidCAM enables you to use the set of standard mathematical functions such as sine, cosine, etc. in your formulas.

Functions list

Standard functions

Example:

For the **H_center** variable in the following formula $H\_center = \frac{D\_chamfer/2}{\tan(\alpha/2)}$ you can write the following expression:

H_center=(D_chamfer/2)/(tan(Alpha/2))

Related Topics

Machining Process Define Manager
Parametric field menu
Used/Unused Parameters Tables

SolidCAM standard functions

sqrt()

Returns a value specifying the square root of a number.

sin()

Returns a value specifying the sine of an angle.

cos()

Returns a value specifying the cosine of an angle.

tan()

Returns a value specifying the tangent of an angle.

abs()

Returns a value of the same type that is passed to it specifying the absolute value of a number.

	The absolute value of a number is its unsigned magnitude. For example, **abs(-1)** and **abs(1)** both return 1.

acos()

Returns a value specifying the arccosine of a number.

asin()

Returns a value specifying the arcsine of a number.

atan()

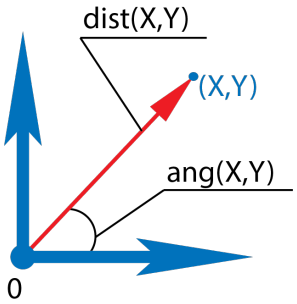
Returns a value specifying the arctangent of a number.

atan2(,)

Returns a value specifying the arctangent of quotient first and second arguments.

dist(,) and ang(,)

Returns polar coordinates: distance and angle (in radians) of the point defined in Cartesian coordinates.



pow(,)

Returns a value of the first argument raised to the power of the second argument.

For example: **pow(2,3)** returns **8**.

log10()

Returns a value specifying the base-10 logarithms of a number.

ln()

Returns a **value** specifying the natural logarithm of a number.

sum3(, ,)

Returns the sum value of three arguments.

int()

Returns the integer portion of a number.

rad()

Converts degrees to radians.

deg()

Converts radians to degrees.

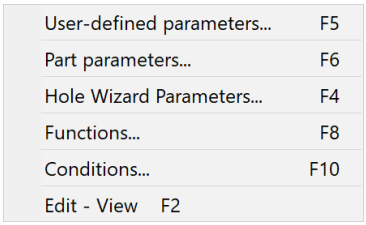
Related Topics

Calculator
Parametric field menu
Variables & expressions

Parametric field menu

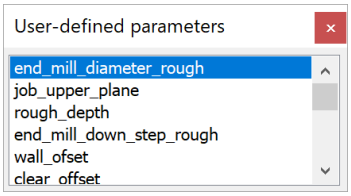
The parametric field menu enables you to choose parameters and expressions easily and effectively.

When you click the button, the following menu is displayed:



- **User-defined parameters**

This option displays a list of existing user-defined parameters.

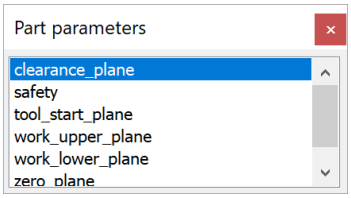


Choose the variable name from the list.

	You can open this list with the F5 hotkey when the cursor is placed in the parametric field.

- **Part parameters**

This option displays a list of existing part parameters.

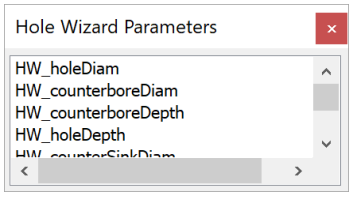


Choose the variable name from the list.

	You can open this list with the F4 hotkey when the cursor is placed in the parametric field.

- **Hole Wizard Parameters**

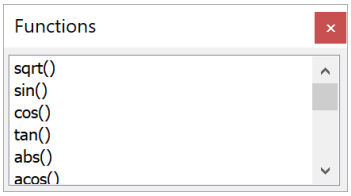
This option displays a list of existing hole wizard parameters.



	You can open this list with the F6 hotkey when the cursor is placed in the parametric field.

- **Functions**

This option displays the list of standard SolidCAM functions.



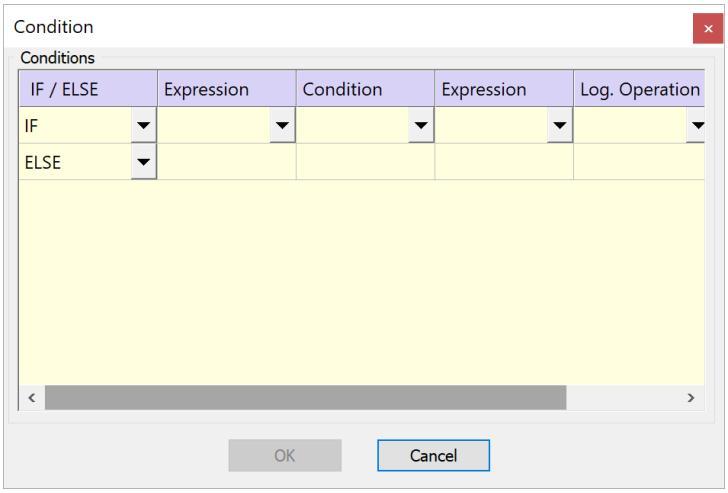
	You can open this list with the F8 hotkey when the cursor is placed in the parametric field.

See Also

Standard functions

- **Conditions**

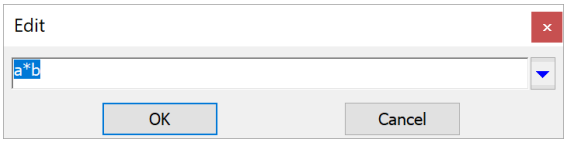
This option displays the table of standard SolidCAM conditions.



	You can open this list with the F10 hotkey when the cursor is placed in the parametric field.

- **Edit-View**

This option displays the Edit dialog box.



In this dialog box, you can see and edit the full parameter name or expression string.

	1. You can open this list with the F2 hotkey when the cursor is placed in the parametric field. 2. In this dialog box, you can also use the operational menu by clicking the button.

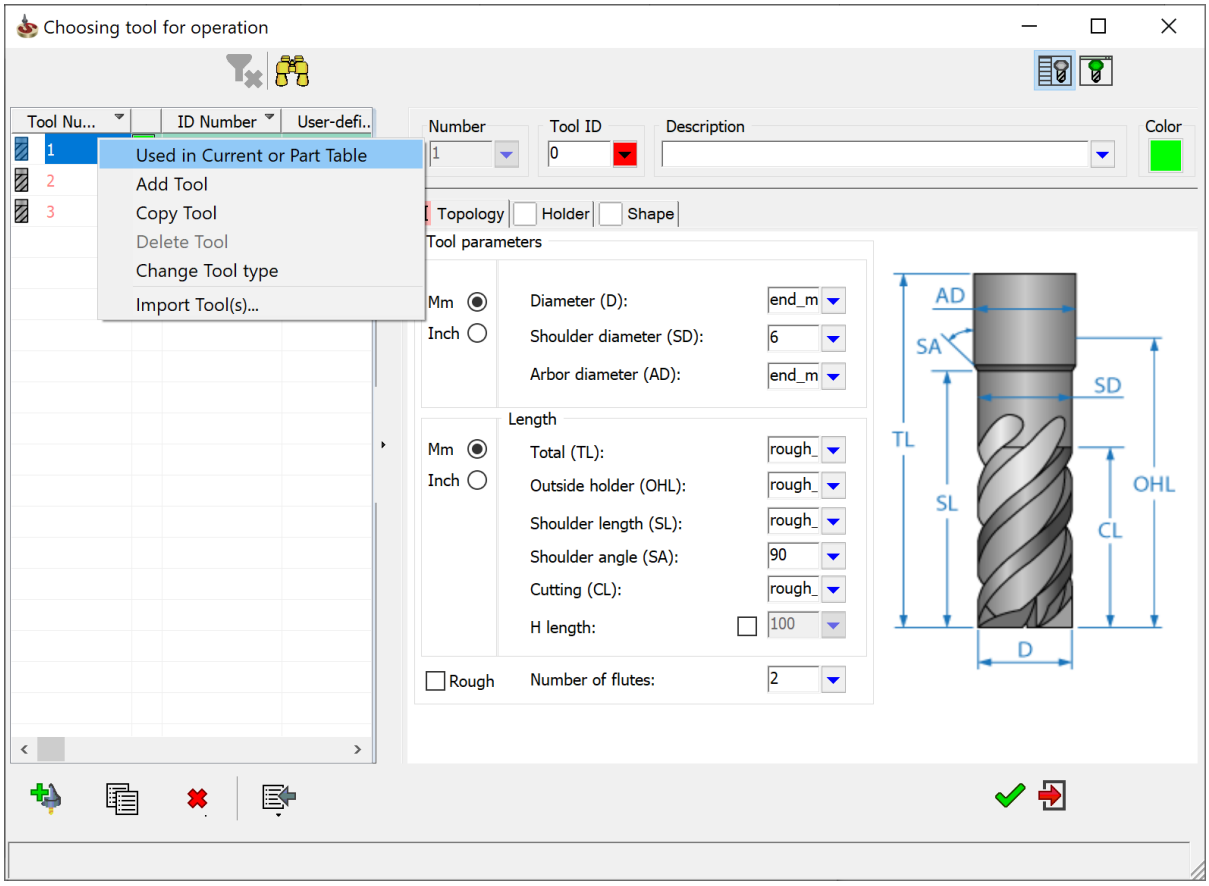
Related Topics

Variables & expressions
Used/Unused Parameters Tables
Machining Process Define Manager

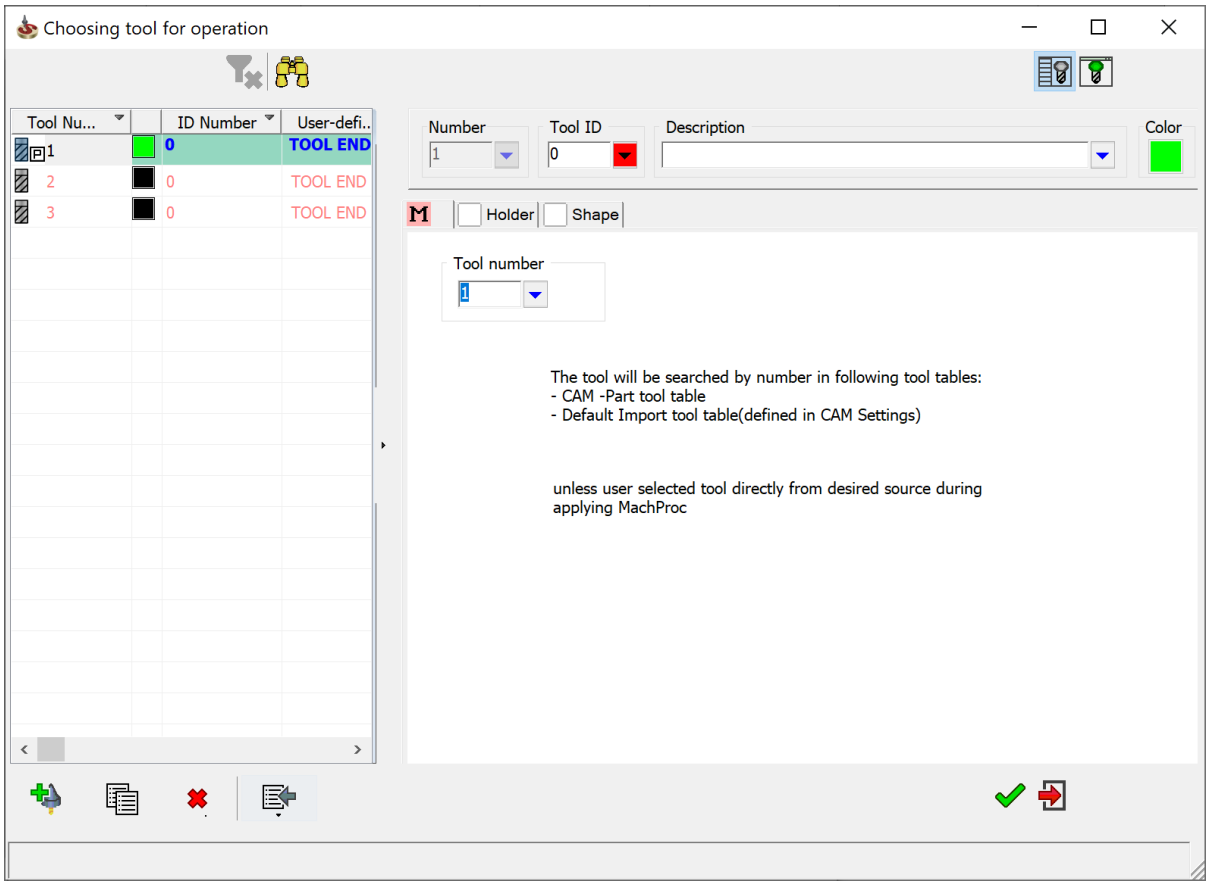
Parametric tool definition

SolidCAM enables you to define the tool number as a variable, similar to the other tool data that can be defined parametrically in the operation templates. The expression for the tool number variable can vary in the different Default sets. The tool number variable is assigned its value during the insertion of a MP into the CAM-Part.

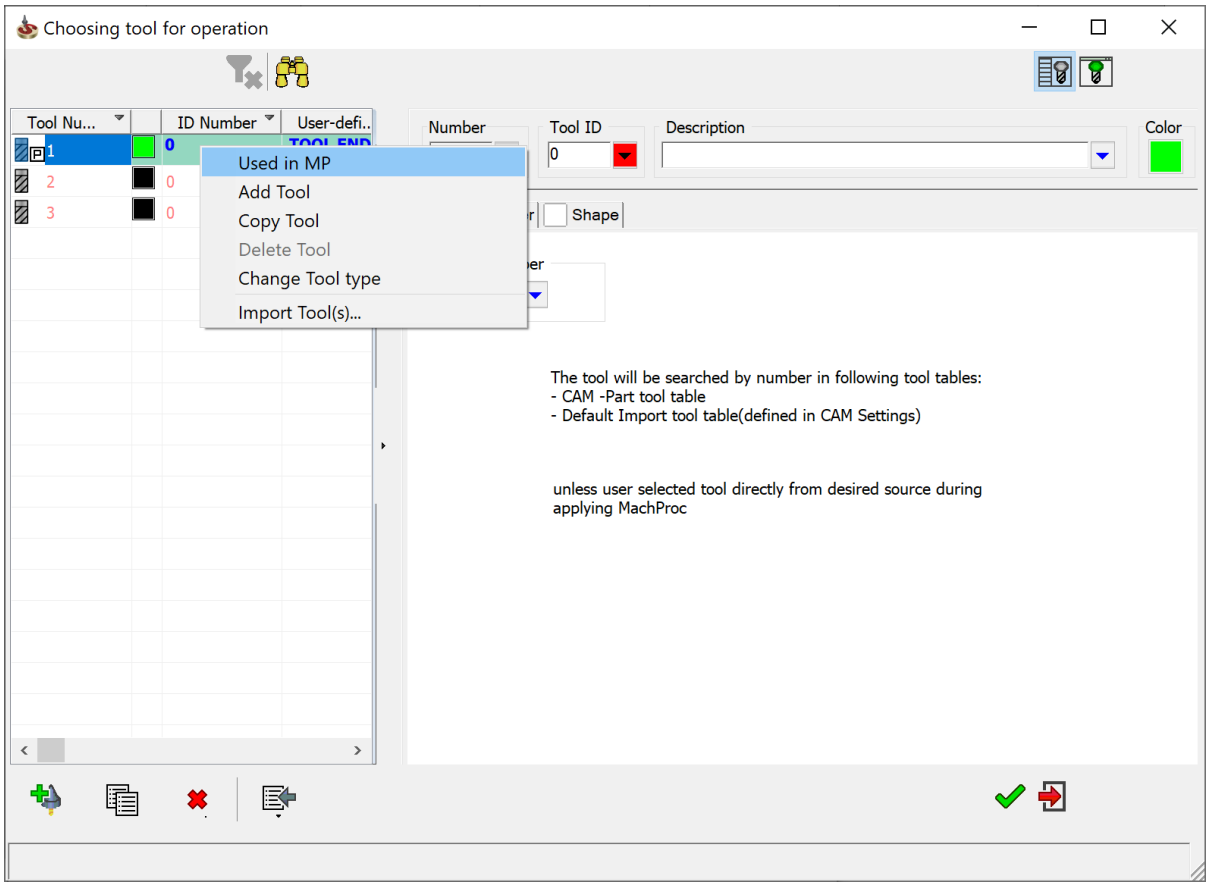
By default, this option is disabled when you define the tool for an operation template in a MP. To enable it and define the tool number parametrically, right-click the tool name in the **Tool list** area of the Part Tool table and choose **Used in Current or Part Table**.



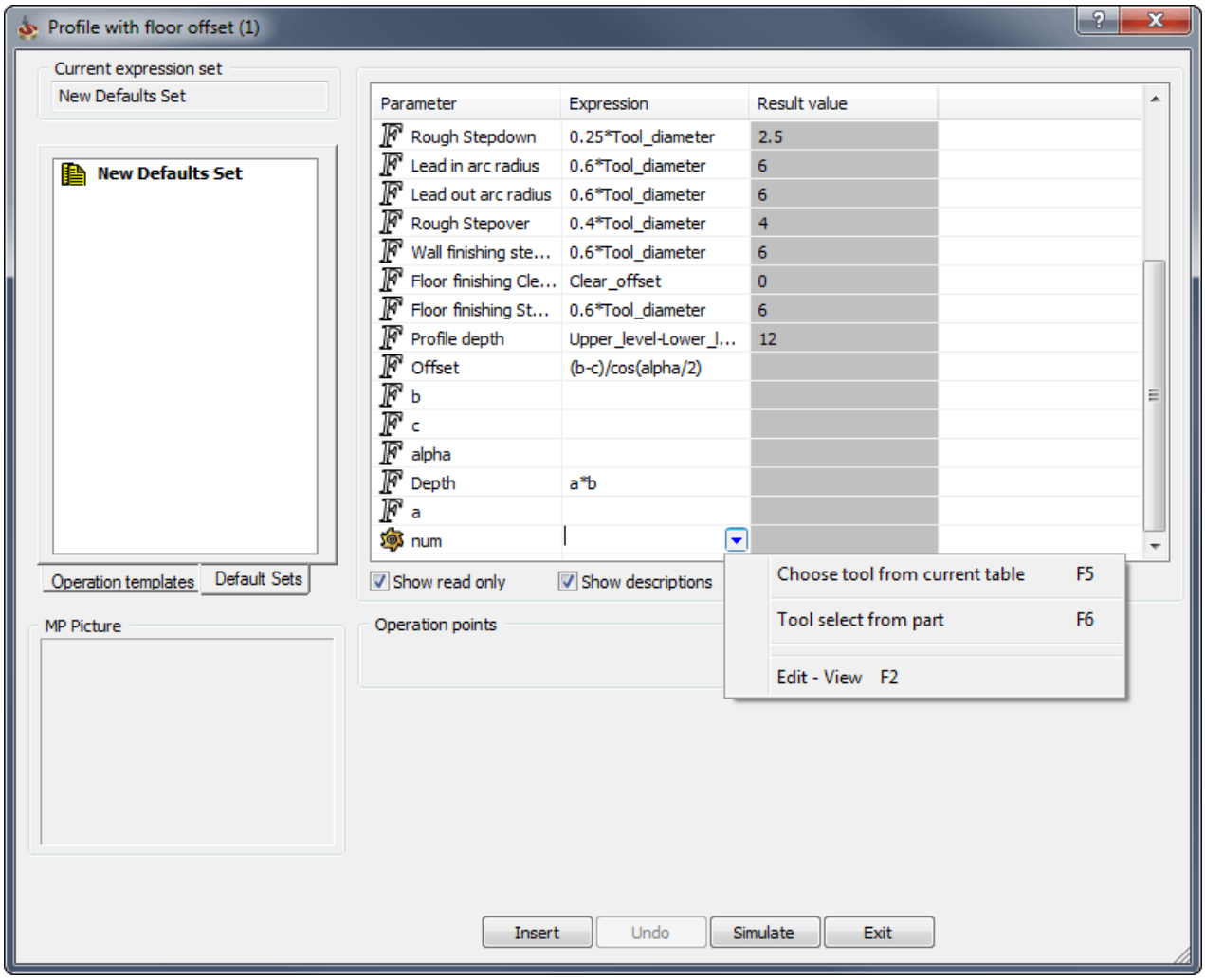
The **Tool number** field becomes available for parametric definition. All other tool parameters are disabled.



You can switch back to the method where the **Tool number** is assigned automatically by right-clicking on the tool name in the list and choosing **Used in MP**.



When you insert the MP into the CAM-Part, you have to assign values for the tool parameters that were defined as variables in the MP. If the tool was defined with the **User-defined tool number** option, SolidCAM automatically checks the existence of the tool with the specified number in the Part Tool Table and Current Tool Table; if a tool with this number is not found, the number is not acceptable.



The parametric menu available for the tool number variables contains three options:

- **Choose tool from current table**

This option displays the Current Tool Library from which you can choose a tool for the MP insertion. When you have selected a tool, its tool number from the Current Tool Table appears in the Expression field of the relevant variable. This option is available only when the Current Tool Library is defined.

- **Tool select from part**

This option displays the Part Tool Table from which you can choose a tool for the MP insertion. When you have selected a tool, its number from the Part Tool Table appears in the Expression field of the relevant variable.

- **Edit – View**

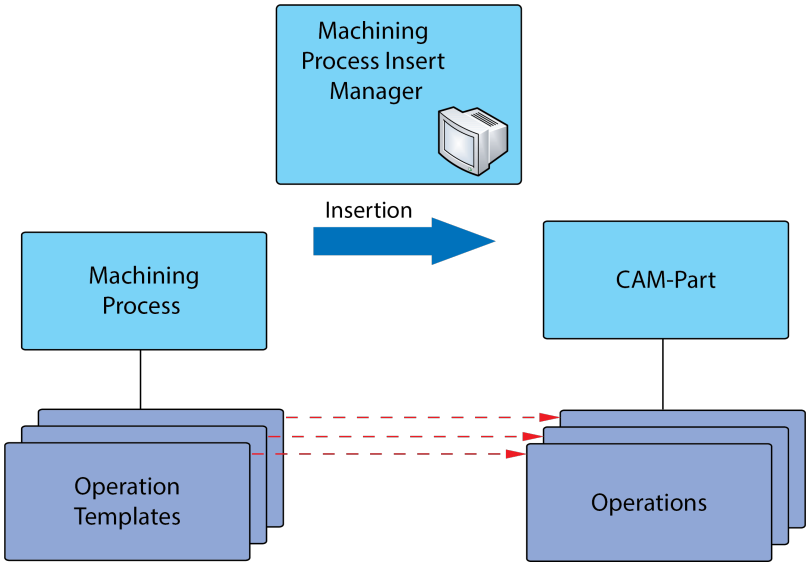
This option displays the Edit dialog box in which you can enter the expression manually.

Related Topics

Machining Process Define Manager

Using Machining Process

You use the defined Machining Process by inserting it into a CAM-Part. During the insertion, SolidCAM transforms each Operation template to a single operation.



The main interface tool that enables you to insert a Machining Process into a CAM-Part is the Machining Process Insert Manager.

Related Topics

Machining Process - Introduction and Basic concepts
Creating Machining Processes - Introduction

Inserting the Machining Process

Before inserting the Machining Process into your CAM-Part, do the following:

1. Load the CAM-Part in which you want to use the technology defined in the Machining Process.
2. Load the Machining Process Table.

All Machining Processes from the current Machining Process Table can be inserted into the CAM-Part directly using the **Add Machining Process** menu.

The Machining Processes in the menu are classified according to pre-defined groups.

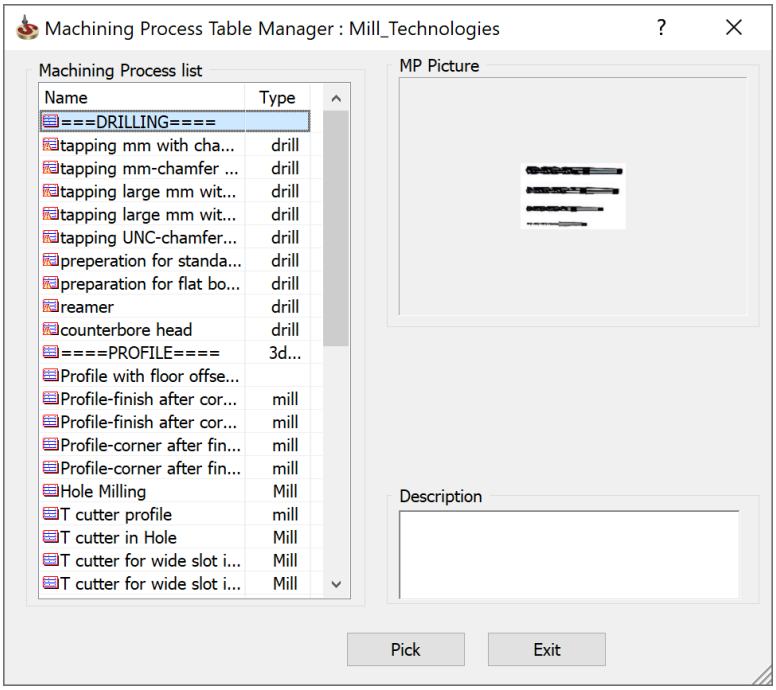
If the MP contains operation templates with operation options incompatible with the operation options of the current MAC file, the operation is marked with the icon. Such MP can be inserted into the current CAM-Part, but operation options will not be inserted.

When the MP contains a Drilling operation template with drill options incompatible with the drill options of current MAC file, this MP is marked with the ⚠ icon.

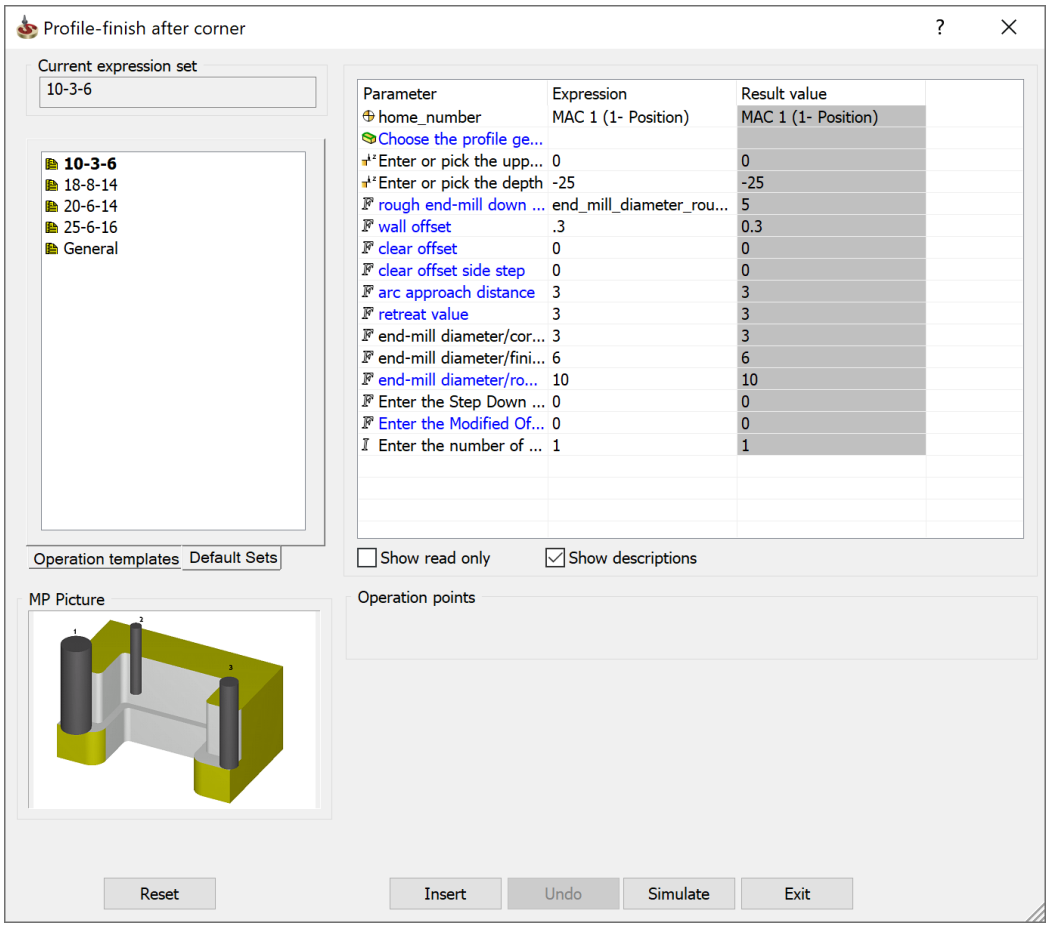
The **Add Machining Process** command is not available in the following cases:

- When there is no current Machining Process Table;
- When the current Machining Process Table is incompatible with the VMID file used in the current CAM-Part.

Right-click **Operations** in the CAM tree and select **Add Machining Process**. The **From MP Table** command enables you to choose the Machining Process to insert from the Machining Process Table Manager dialog box.



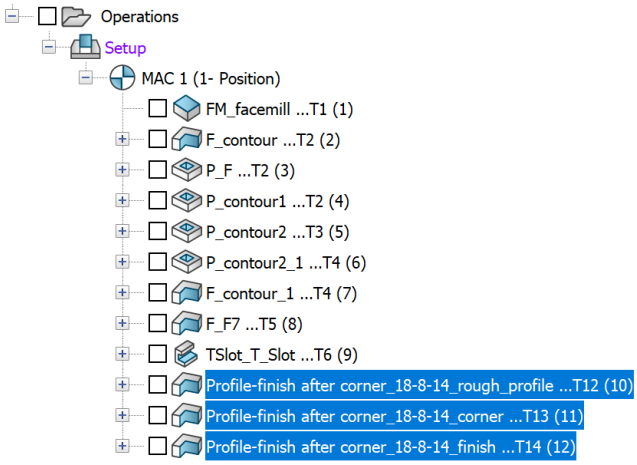
Choose a Machining Process from the list and click the **Pick** button. The Machining Process Insert Manager is displayed.



While the **Insert Manager** is loading, SolidCAM receives values of Part Parameters from the CAM-Part. Then SolidCAM receives expressions for other parameters from the current Default Set and calculates them.

To prepare the Machining Process for insertion, do the following:

1. On the **Default Sets page**, choose the required Default Set.
2. With the **Parameters Table**, fill all the blank expression fields. You can also change all expressions that do not have **Read-Only** status.
3. When the Machining Process is defined, click the **Insert** button to insert the Machining Process into the current CAM-Part.
4. SolidCAM chooses tools for the Machining Process.
5. The operations are inserted into the CAM-Part. The inserted operations are calculated according to the **Part Settings**.



- To undo the last insertion, use **Undo**.
- **Simulate** enables you to simulate operations of the current CAM-Part.

Choosing the tool

SolidCAM automatically performs the tool search during the machining process insertion. The tool search is performed according to the Used in Current or Part Table/Used in MP property of the tool used in the operation template.

When the **Used in Current or Part Table** option is used for the tool definition, the tool search is performed according to the tool number only. The tool search is first performed in the Part Tool Table. If the tool is found, it is used. If the tool with the defined

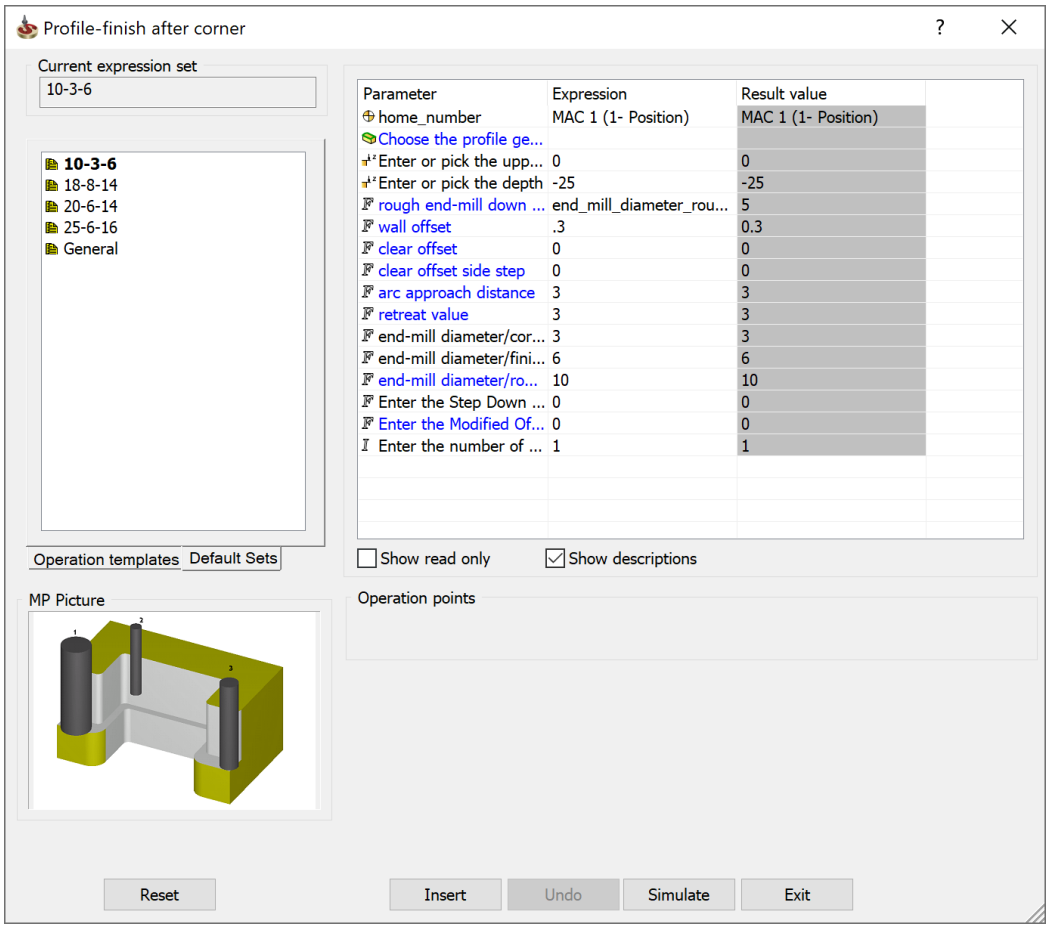
number is not found in the Part Tool Table, SolidCAM performs an additional search in the Current Tool Table. If the tool with the defined tool number is found in the Current Tool Table, it is copied into the Part Tool Table (with the same number) and used.

When the **Used in MP** option is used for the tool definition, the tool search is performed according to the tool parameters. The tool parameters used for the tool search are defined in the SolidSolidCAM Settings dialog box. The tool search is first performed in the Part Tool Table. If the tool is found, it is used. If the tool is not found in the Part Tool Table, SolidCAM performs an additional search in the Current Tool Table. If the tool is found in the Current Tool Table, it is copied into the Part Tool Table and used. If the tool is not found in the Current Tool Table, a new tool with the defined parameters is created in the Part Tool Table. SolidCAM automatically chooses the first available tool number for the newly created tool.

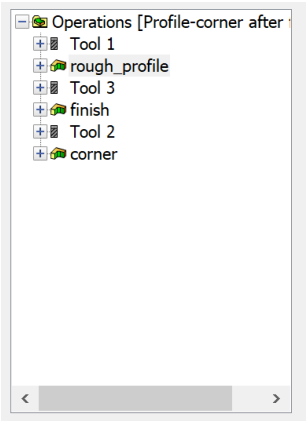
Related Topics

Machining Process Insert Manager
Parameters definition

Machining Process Insert Manager

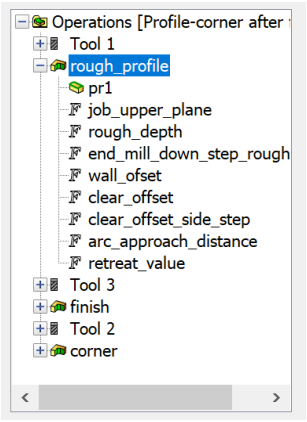


Operation Templates Page



This page displays the list of operation templates that you inserted into the CAM-Part.

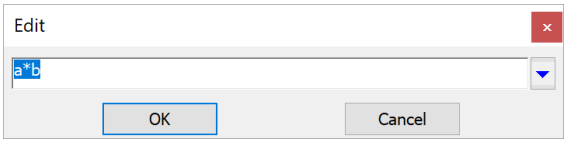
- By clicking on the (+) sign near each operation template, you can see the list of parameters used in the selected operation template.



- To see the operation template parameters and settings, right-click on the operation template name in the list and choose the **Show** command from the list. The operation template dialog box is displayed.

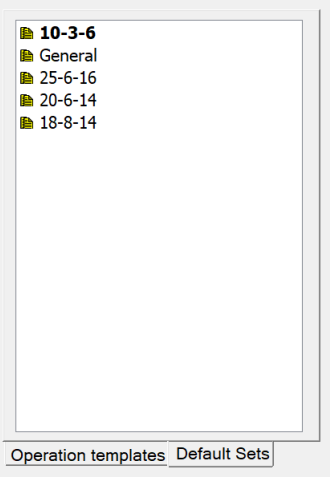
In this dialog box you cannot edit the Operation Template definition. To edit the operation template, go to the Machining Process Define Manager**.**

In the operation template dialog box, all parametric fields are marked with the button. Click this button to see the parametric field menu. The **Edit-View** option is available in this menu. This option opens the parameter edit dialog box.



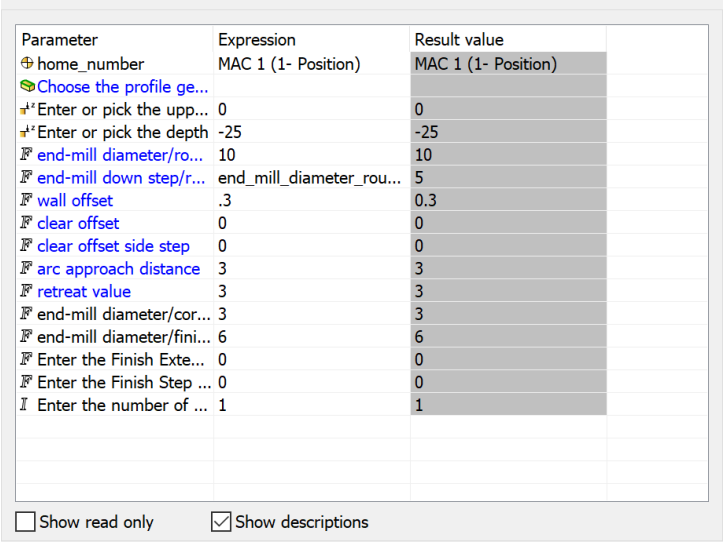
This dialog box enables you to see and edit the full parameter name or expression string.

Default Sets Page



Double-click on the Default Set name to make it current. The Parameters and values from the selected Default set will be shown in the Parameters Table.

Parameters Table



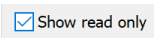
There are three columns in this table:

- **Parameter** - shows the parameter name.
- **Expression** - shows the parameter definition. In this field you have to define missed expressions for parameters. You can also edit all expressions that do not have **Read-Only** status.

See Also

Parameters definition

- **Result value** - shows the result value returned by the parameter.



With the **Show read only** check box, you can allow or forbid showing parameters marked Read-Only.

The **Show descriptions** check box enables you to show the parameter description instead of the parameter name in the **Parameter** column.

Operation Points

This field contains buttons to define specific points of the operation.

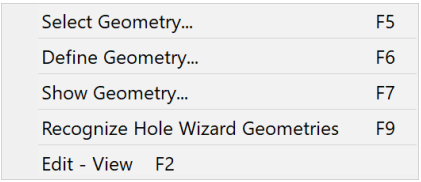
Related Topics

Inserting Machining Process
Parameters definition

Parameters definition

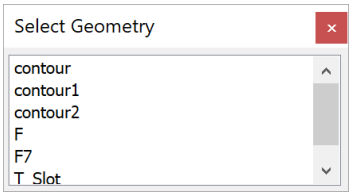
Click the button to display menus for each parametric field.

Parametric field menu for Geometries



- **Select Geometry**

This option displays a list of geometries of a **suitable type** that is defined in the CAM-Part.



Choose the desired variable name from the list.

	You can also open this list with the **F5** hotkey when you are in the parametric field.

- **Define Geometry**

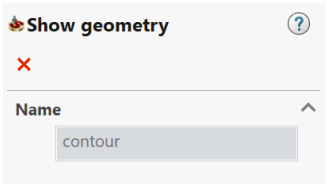
This option enables you to define a geometry from the current model.

	You can also open this list with the **F6** hotkey when you are in the parametric field.

- **Show Geometry**

The current 3D model, drill or chain geometry is displayed in the CAD graphics area.

To exit from this mode, use in the **Show Geometry** dialog box.



	You can also open this list using the **F7** hotkey when you are in the parametric field.

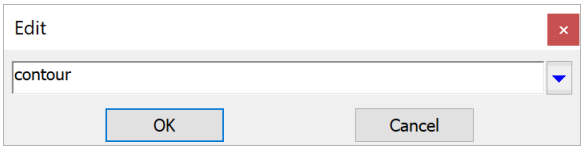
- **Recognize Hole Wizard Geometries**

This option displays the Drill Geometry Selection dialog box which enables you to select the Geometry for a Hole Wizard Process.

	You can also open this list using the **F9** hotkey when you are in the parametric field.

- **Edit-View**

This option displays the parameter **Edit** dialog box.



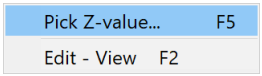
Here you can see and edit the full parameter name or expression string.

	1. You can also open this list with the **F2** hotkey when you are in the parametric field. 2. In this dialog box you can also use the operational menu by clicking the button.

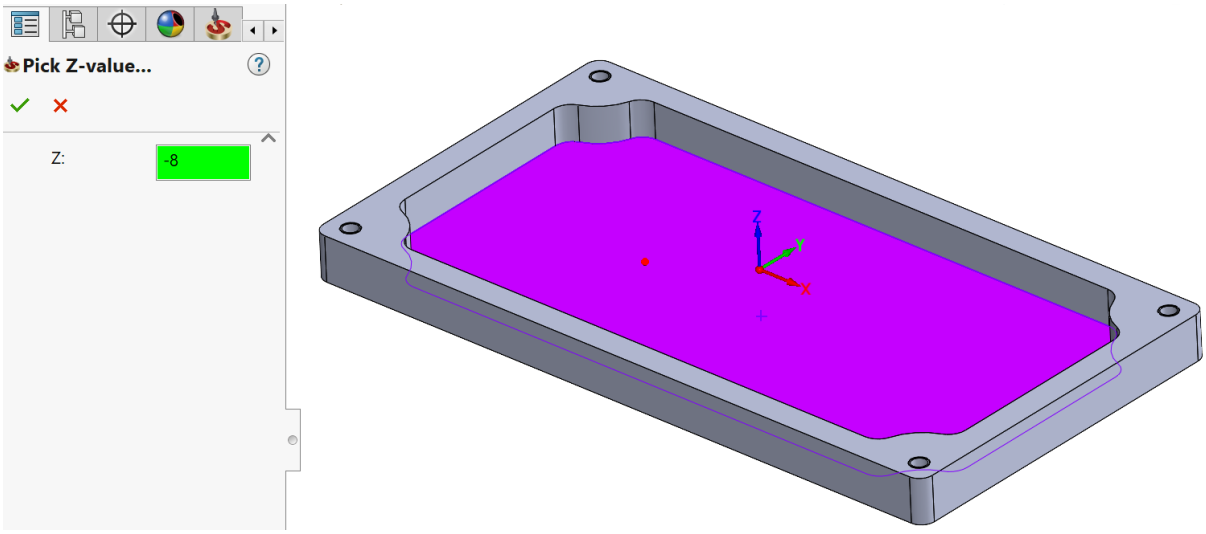
Parameters of Z-value and Distance types

For parameters of these types, SolidCAM enables you to define values by picking them from the active CAD model.

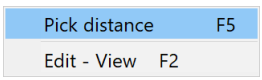
- To define the **Z-value**, choose the **Pick Z-value** (**F5**) item from the parametric menu.



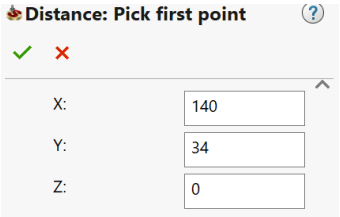
The **Pick Z-Value** dialog box will be displayed. Pick the desired point on the model and click the **OK** button to confirm the selection.



- To define the **Distance**, choose the **Pick Distance** (**F5**) item from the parametric menu.



1. In the CAD window, pick the first point.
2. Confirm the selection with the **OK** button in the **Distance: Pick first point** dialog box.



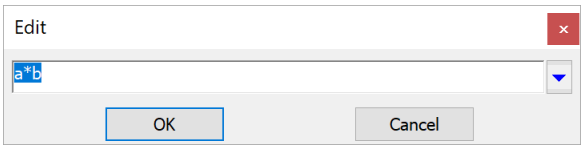
3. Pick the second point and confirm it in the same way.

The **Distance** value will be calculated.

Other Parameters

For parameters of other types, the parametric field menu contains only one item: **Edit-View** (**F2**).

Choose it to edit the parameter definition in the **Edit** window.



Related Topics

Inserting Machining Process
Machining Process Insert Manager