GIBBSCAM **14**

Version 14 : September 2020

OPTICAM Tutorial

CAMBRIO

Proprietary Notice

Contents

Preface

GibbsCAM® Wire EDM, powered by OPTICAM is a new Wire EDM product offering at GibbsCAM 14, based on the partnership between CAMBRIO and Camtek and bringing more than thirty year of experience focused on the programming of wire EDM machines. OPTICAM is fully integrated into the user interface of GibbsCAM user interface.

Using automatic feature recognition and integrated technologies and strategies, OPTICAM offers a very high automation level and significantly reduces programming time. After a short training period, any beginner will be able to create a correct NC program with only a few clicks, while experienced users will still have a high flexibility and the possibility to operate on a detailed level.

The examples given in the following chapters are only supposed to give a general overview of the work with GibbsCAM Wire EDM, powered by OPTICAM. They do not come close to showing every possible machining function. The aim of the examples is to give a general overview of the different machining functions.

This tutorial assumes you are able to use GibbsCAM software. If this is not the case, we strongly advise you to participate in a GibbsCAM training.

The following sample files, available from GibbsCAM, are required for this tutorial:

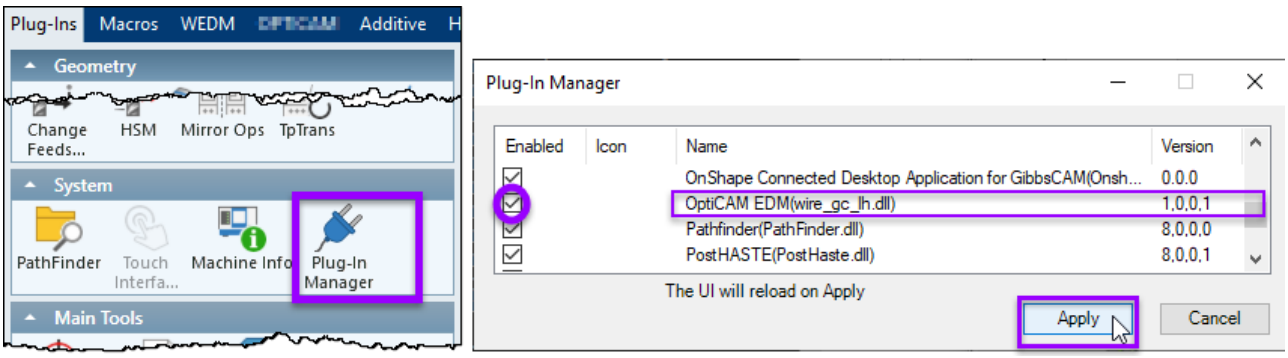
- GC14_Opticam_example_1.vnc
  GC14_Opticam_example_1.wire.opticam
- GC14_Opticam_example_2.vnc
  GC14_Opticam_example_2.wire.opticam
- GC14_Opticam_example_3.vnc
  GC14_Opticam_example_3.wire.opticam
- GC14_Opticam_example_4.vnc
  GC14_Opticam_example_4.wire.opticam
- GC14_Opticam_example_5.vnc
  GC14_Opticam_example_5.wire.opticam
- GC14_Opticam_example_6.vnc
  GC14_Opticam_example_6.wire.opticam
- GC14_Opticam_example_7.vnc
  GC14_Opticam_example_7.wire.opticam
- GC14_Opticam_example_8.vnc
  GC14_Opticam_example_8.wire.opticam
- GC14_Opticam_example_9.vnc
  GC14_Opticam_example_9.wire.opticam

Getting Started

With each tutorial, you will open a sample file, use the OPTICAM user interface that appears to the left of the GibbsCAM workspace, and exercise OPTICAM functions.

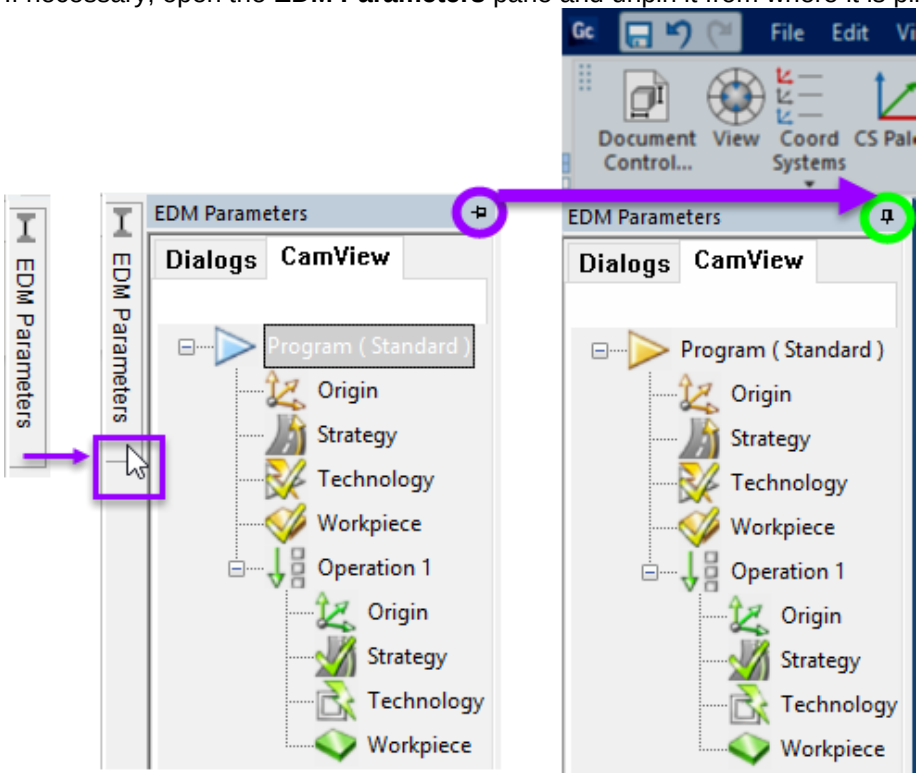
If the main menu bar does not have an OPTICAM menu

1. On the Plug-Ins menu, click Plug-In Manager.
2. In Plug-In Manager, tick the checkbox for OtiCAM EDM(wire_gc_lh.dll), and then click Apply.

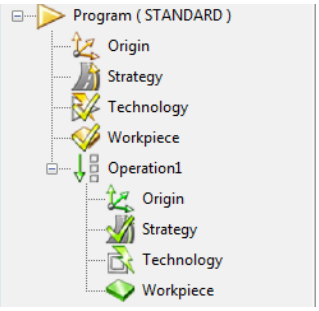


If the OPTICAM user interface does not appear

- Ensure that a <*filename*>**.wire.opticam** file exists in the same folder that contains the <*filename*>**.vnc** file
- If necessary, open the **EDM Parameters** pane and unpin it from where it is pinned or docked.



The OPTICAM Feature Manager



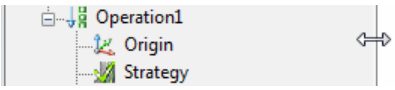
When the OPTICAM Feature Manager is active, the GibbsCAM lists (tools, processes, and operations) are not used and are suppressed.

The Feature Manager is the central operating element of the OPTICAM software. It lists all programmed machining operations.

***Setting the** OPTICAM Feature Manager*

If the Feature Manager is too narrow or too wide, you can set its width, as follows:

Position the cursor on the vertical right edge of the OPTICAM Feature Manager until the cursor turns into the following symbol:

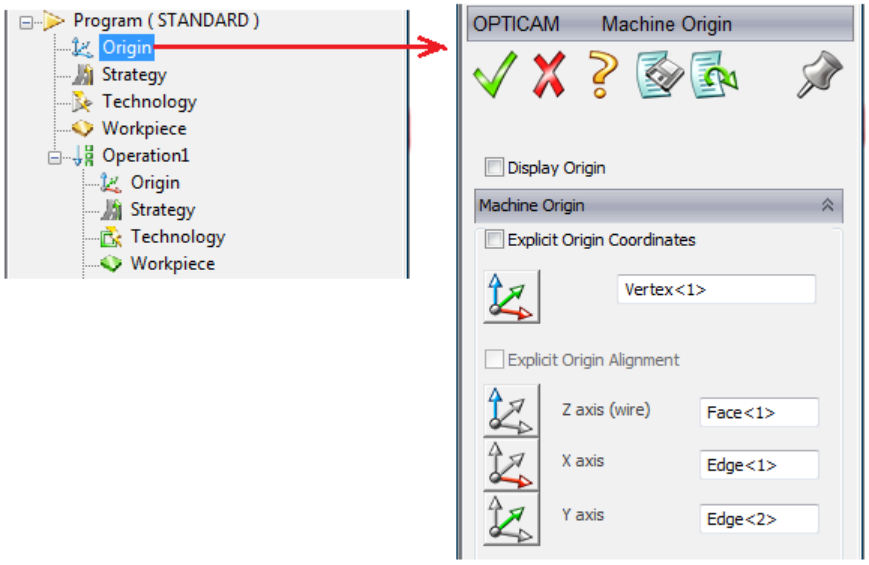


Click and hold the left mouse button and drag the cursor to the left or right. When the width is as you want it, release the left mouse button.

***Working with the** OPTICAM Feature Manager*

Double-clicking a branch

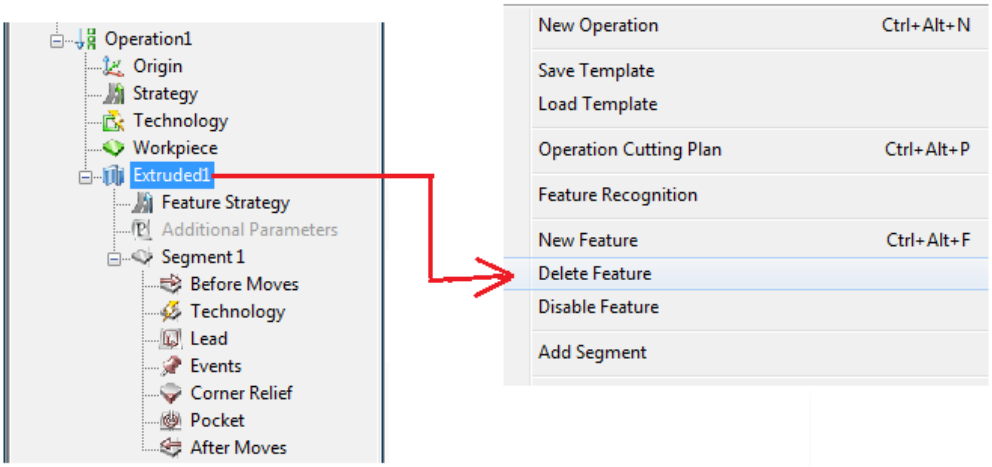
Double-clicking a branch of the Feature Manager usually opens a dialog box.



Use the dialog to set values. To accept changes, click  . To reject them, click  .

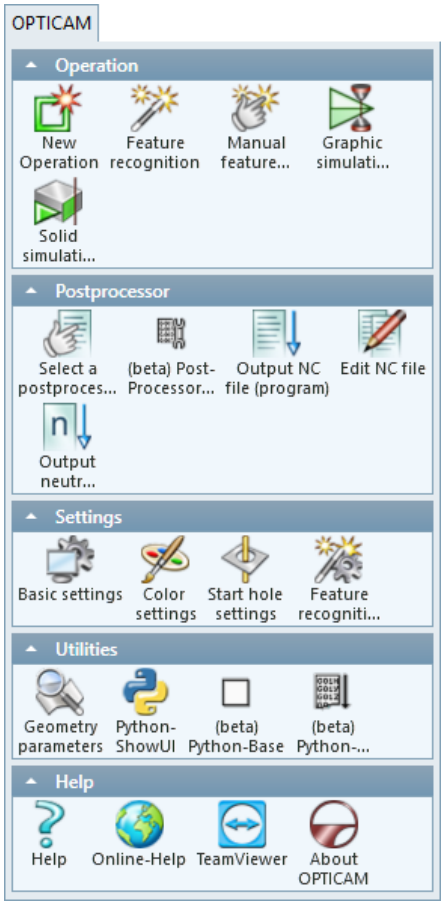
Right-clicking a branch

Right-clicking a branch of the Feature Manager displays a context menu.



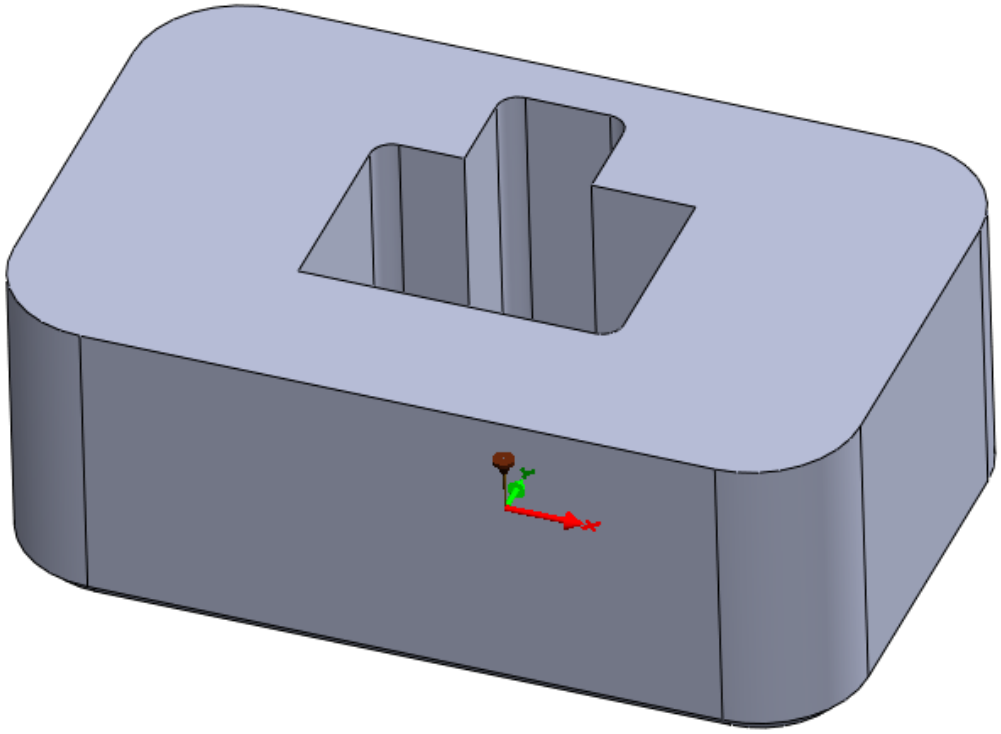
The displayed menus vary according to the selected branch. The example above shows the deletion of a feature.

The OPTICAM Menu



The OPTICAM menu on the top menu bar contains a range of setting functions that do not exist in the Feature Manager. It also offers the most important machining functions of the Feature Manager, although not all of them.

Example 1: Cylindrical Cut



Demonstrated in this Example

Opening a file: GC14_Opticam_example_1

The OPTICAM Feature Manager

Using the **Program** Dialog for Basic Settings

Machine Origin

Defining the Cutting Technology (Manual Input)

Creating the Workpiece (Billet) Geometry

Setting the Cutting Heights (Reference Plane Height and Secondary Plane Height)

Setting the Cutting Strategy

Manual Creation of the Cutting Machining

Machining Simulation (Graphic Simulation)

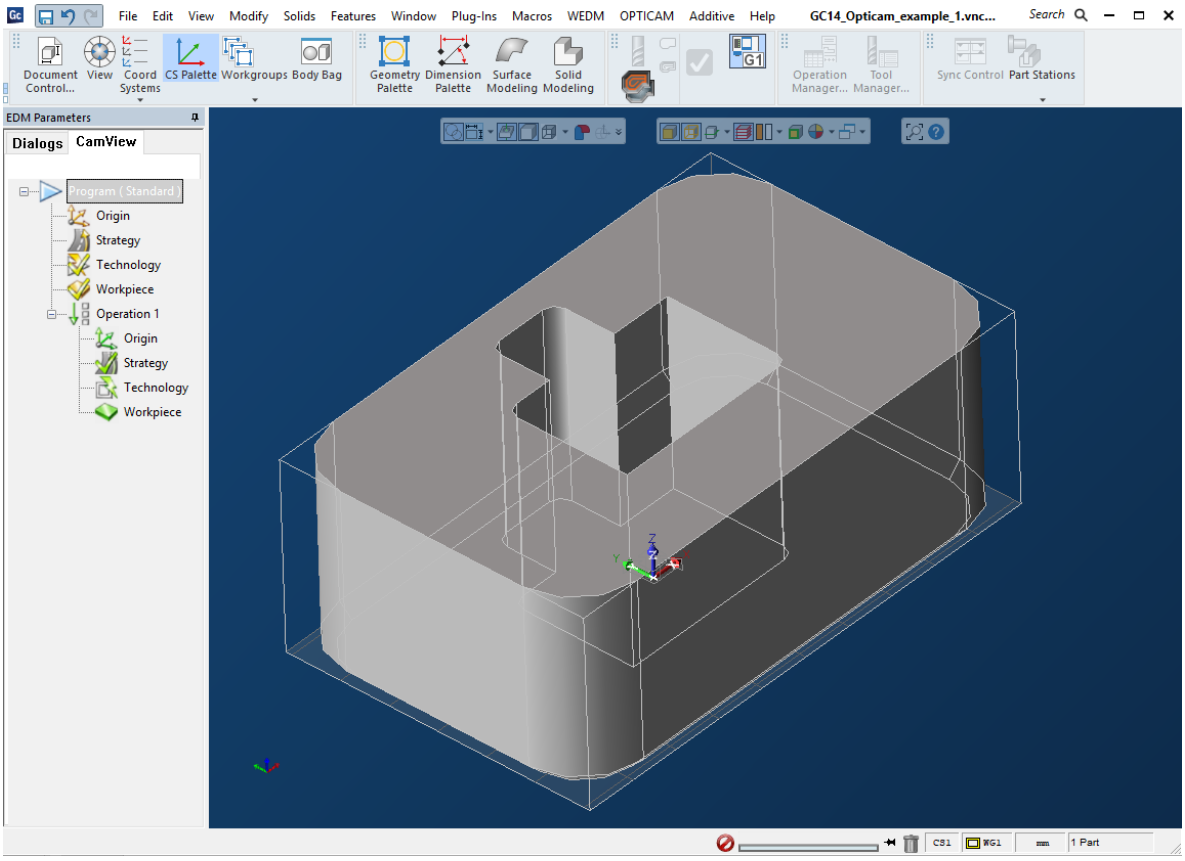
Further Functions of the OPTICAM “OPTICAM Feature Manager”

Creating the NC Program

Open GC14_Opticam_example_1

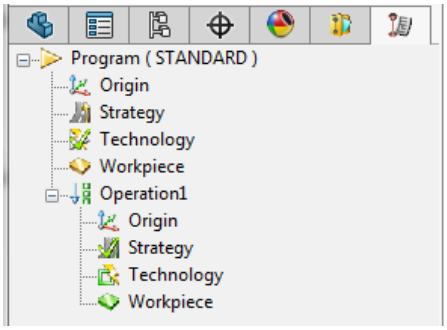
In GibbsCAM, open the example file **GC14_Opticam_example_1.vnc**.

The selected model is loaded into GibbsCAM.



The OPTICAM Feature Manager

To the left of the workspace, you should see the OPTICAM Feature Manager. If you do not, then refer to Getting Started on page 0.

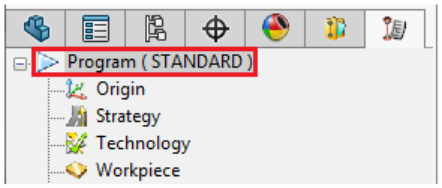


The OPTICAM Feature Manager is the central control element of OPTICAM. It allows you to program and modify machining operations and set machining parameters.

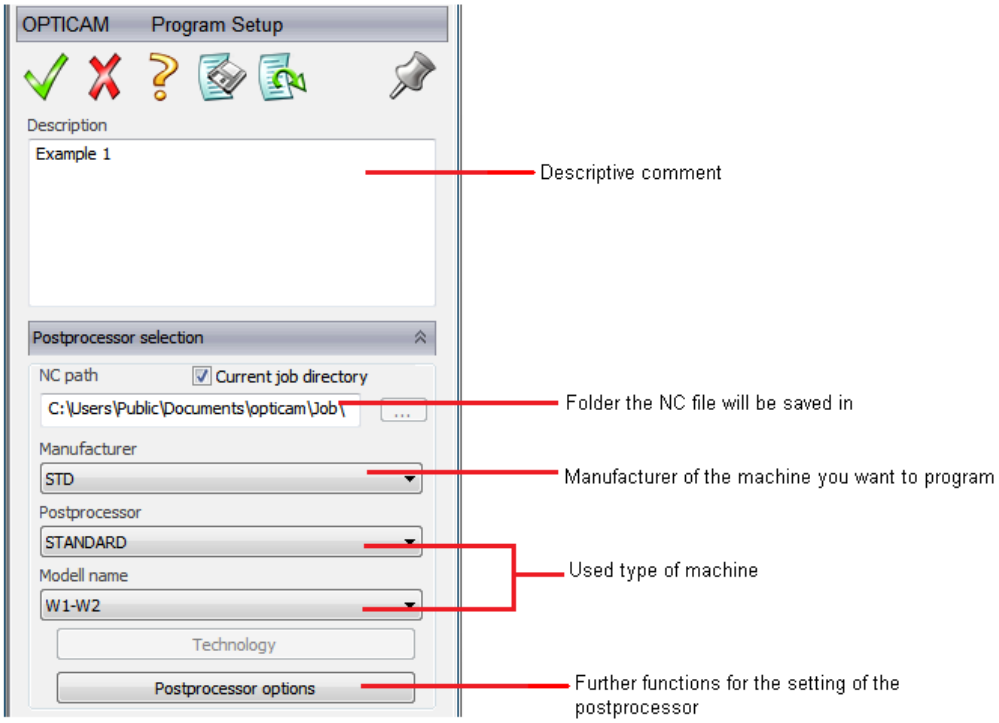
Using the Program Dialog to Set Defaults

As a first step, we define the basic settings. This means that we define the wire EDM machine for which the program will be created and where the NC output file will be placed; additionally, we define the start position of the program.

In the Feature Manager, double-click the branch **Program (STANDARD)**.



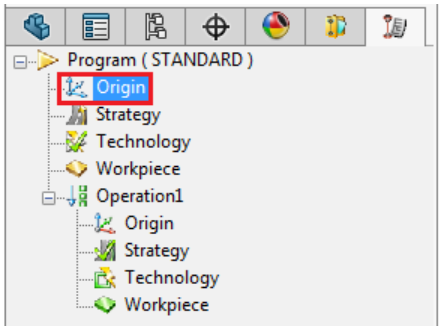
This opens the dialog for the definition of the program default settings. Please fill in the dialog as follows:



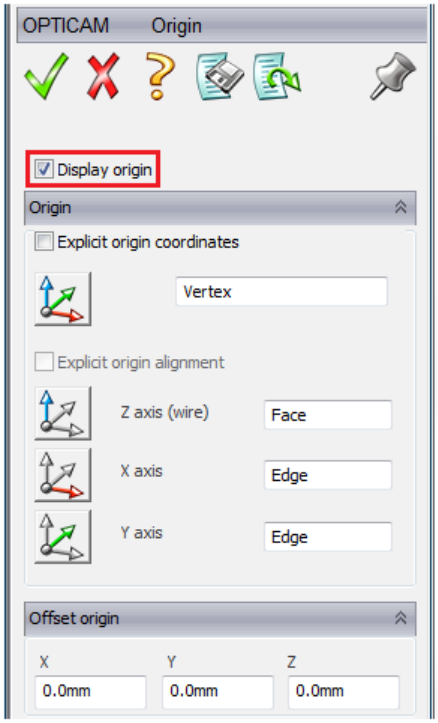
Accept changes by clicking  .

Defining the Machine Origin

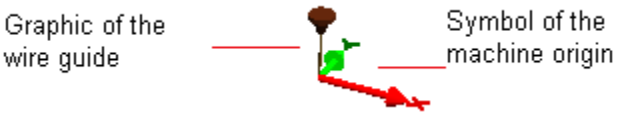
To define the machine origin, double-click the **Origin** branch in the Feature Manager.



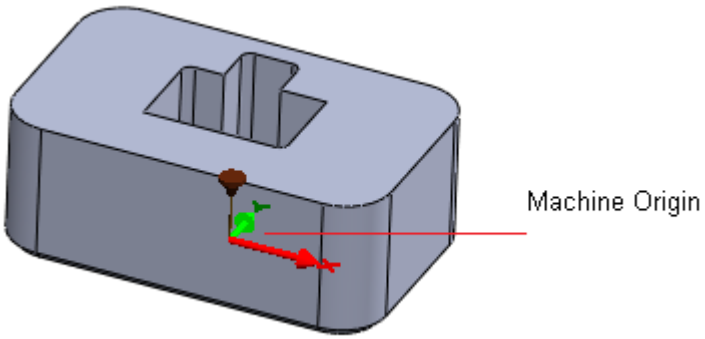
This opens the dialog for the definition of the machine origin.



The origin is graphically displayed by a symbol. The Z-axis of the origin is symbolized by a graphic of the wire guide.



Currently, the machine origin is in the center of the workpiece (see following figure).



Using the **Machine Origin** dialog, you can move the origin to any other position.

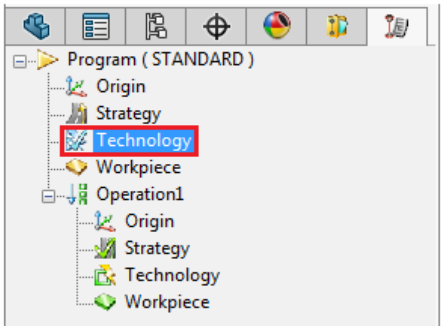
However, in Example 1, we want to accept the predefined origin. (Subsequent examples will explain the different possibilities of setting machine origins.

Close the **Machine Origin** dialog by clicking  .

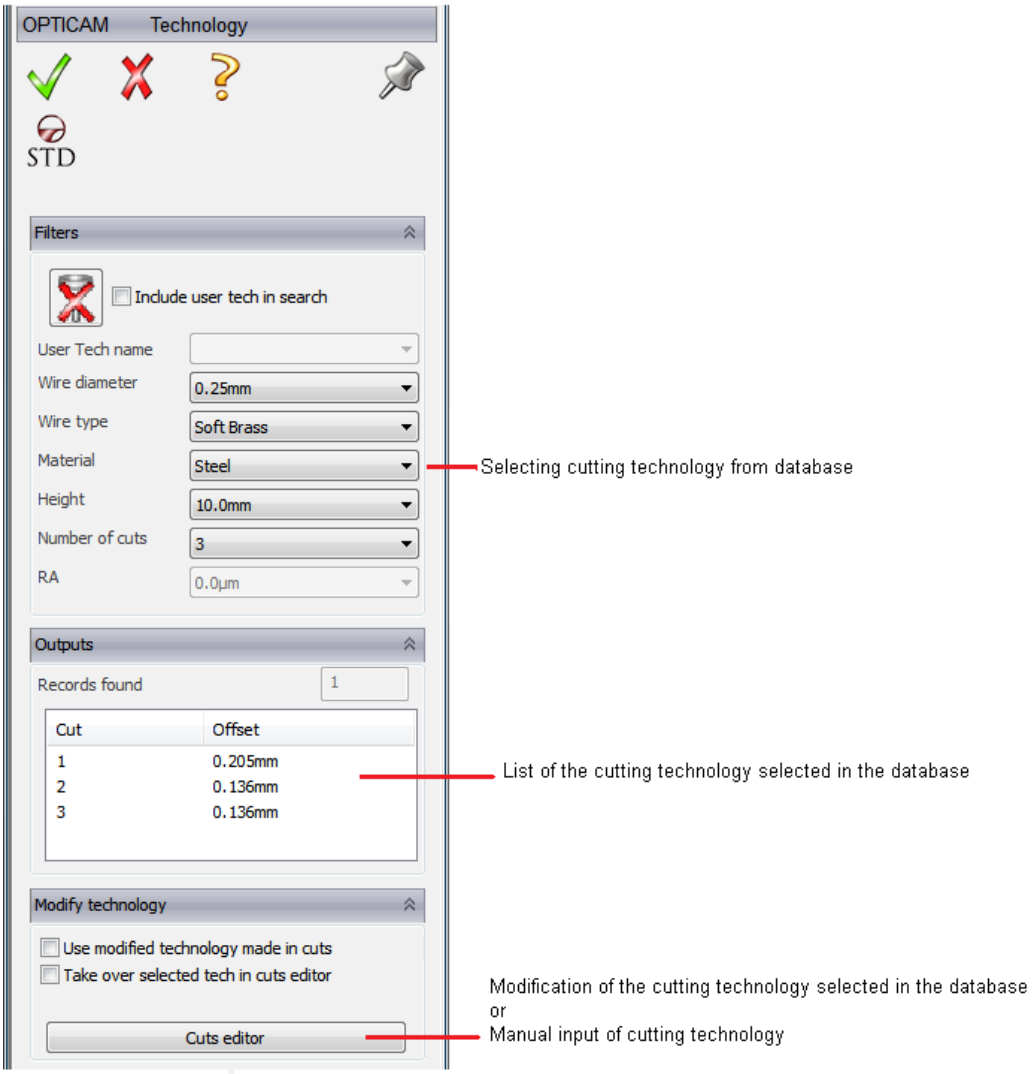
Defining the Cutting Technology (Manual Input)

In the next step, we will define with which cutting technology the workpiece is to be machined.

Double-click the branch **(Program) Technology** in the OPTICAM Feature Manager.

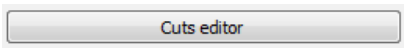


This opens the dialog for the definition of the cutting technology.

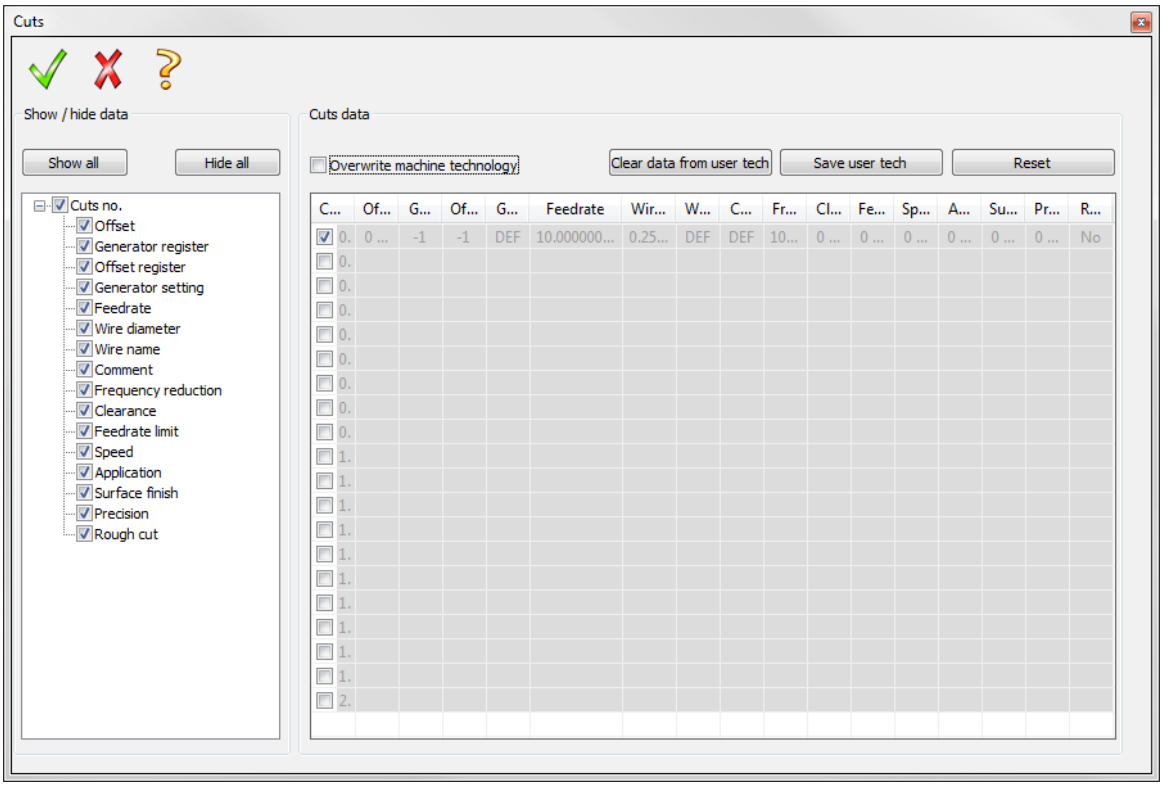


You can either select the parameters of the cutting technology from a database or enter them manually.

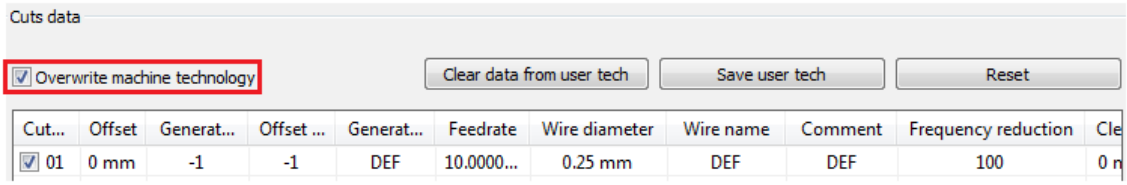
In the first few exercises, we will enter the cutting technology manually.

Click the 

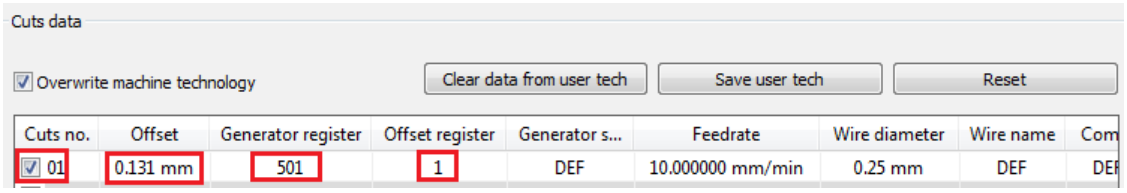
button to open a dialog where you can manually enter the cutting technology.



Activate the input fields by selecting the option  .



To machine the geometries with **1 cut**, please set the dialog as follows:



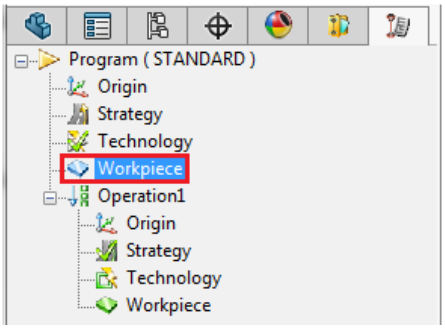
Accept changes by clicking  . The system will go back to the **Technology** dialog.

Close this dialog as well by clicking  .

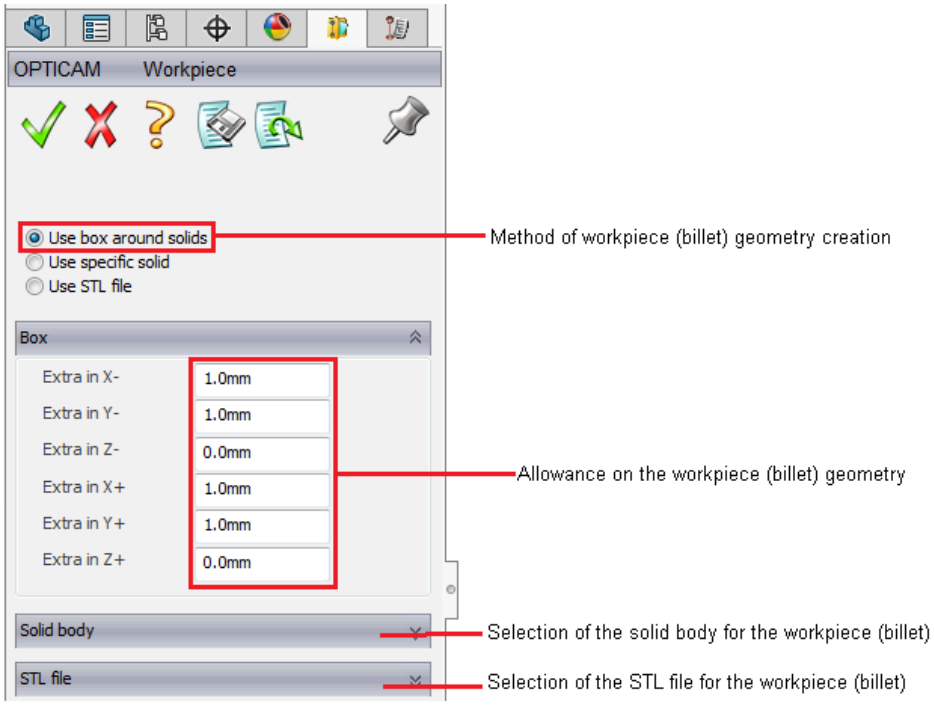
Creating the Workpiece (Billet) Geometry

In the next dialog, you can define the geometry of the workpiece (billet).

Double-click the branch **(Program) Workpiece** in the OPTICAM Feature Manager.



Fill in the dialog as follows:



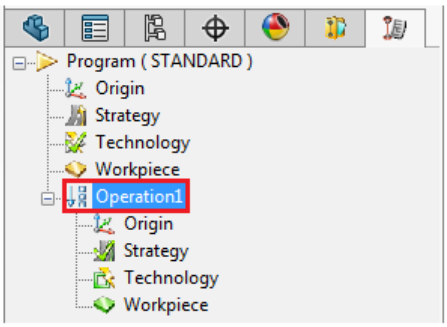
In the selected method, the workpiece (billet) is derived from the 3D model you want to machine. The system does this by creating a theoretical box which encloses the 3D model. Additionally, we define an allowance of 1 mm for both the x- and the y-axis.

Accept changes by clicking  .

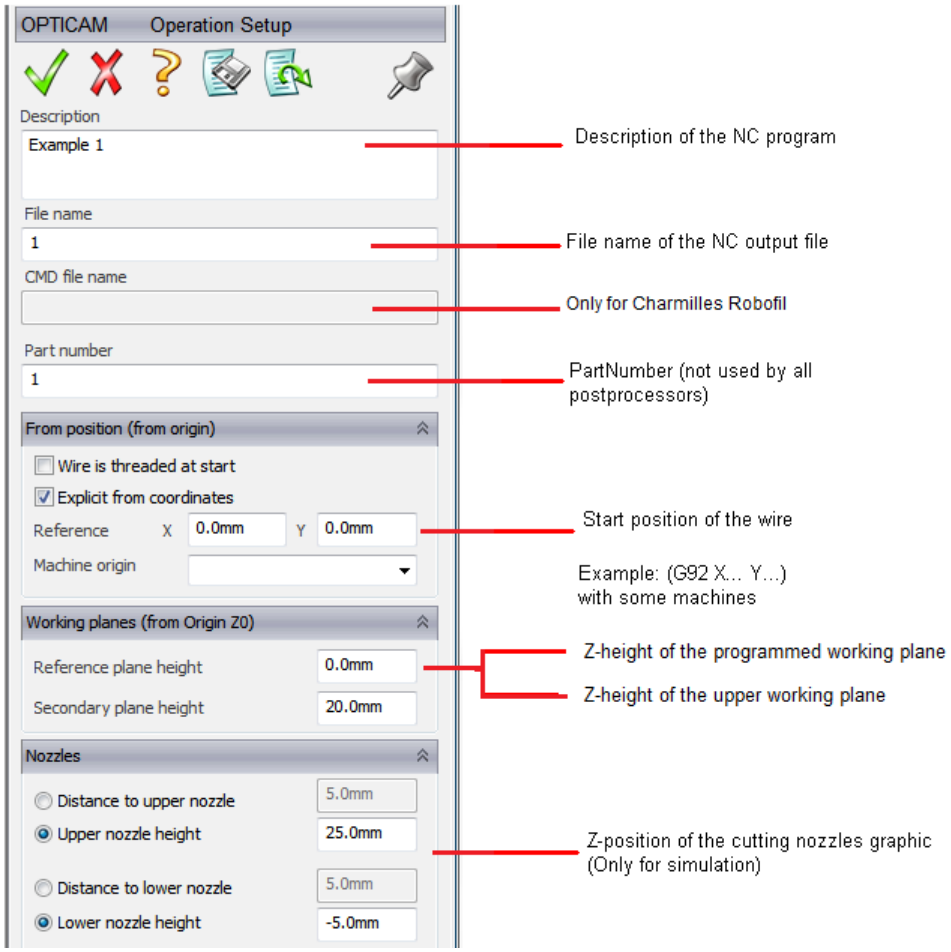
Setting the Cutting Heights (Reference Plane Height and Secondary Plane Height)

The following dialog sets among other things the name of the NC output file as well as the Z-height of the Reference and the Secondary Plane Height.

Double-click the branch **Operation1** in the OPTICAM Feature Manager.



In the field **Description**, you can enter a comment to the program. In the field **FileName**, the name of the NC output file is set. Additionally, you set the general **Start Position** and the height of the working planes (**Reference plane height** / **Secondary plane height**).

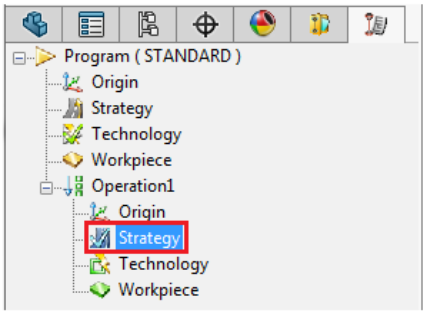


Accept changes by clicking  .

Setting the Cutting Strategy

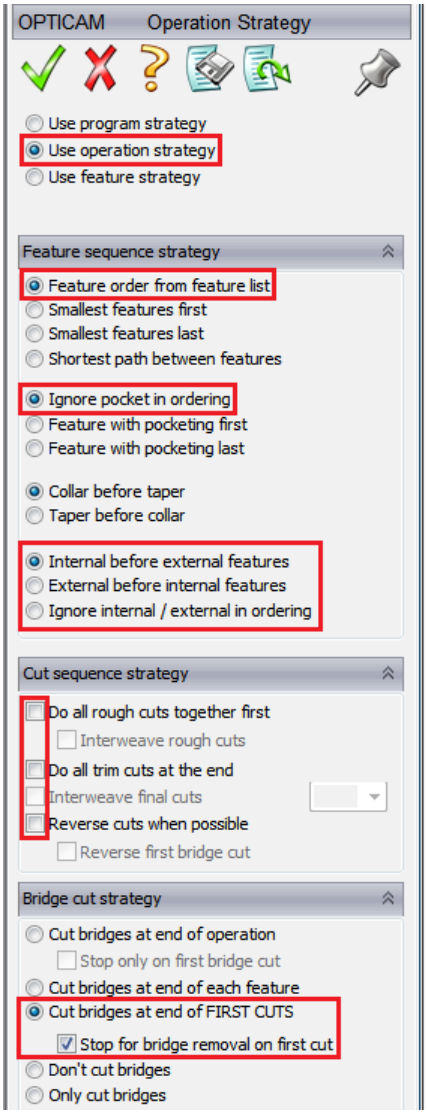
In the next dialog, you set the strategy with which the workpiece is to be machined.

Double-click the branch **(Operation) Strategy** in the OPTICAM Feature Manager.



The geometry is supposed to be machined in a way that at the end of the first cut a machine stop is carried out and the bridge is cut.

Set the dialog as follows:



Accept changes by clicking  .

Creating the Cutting Machining

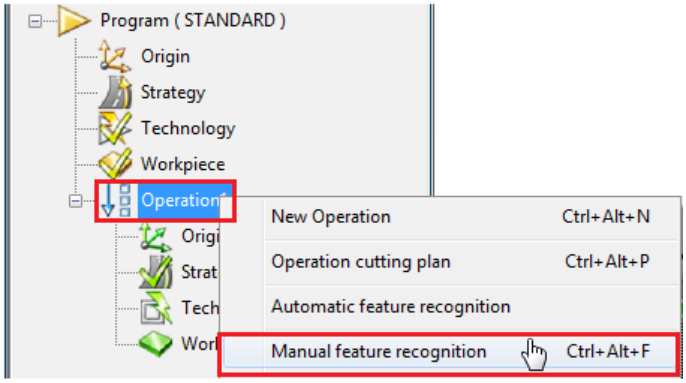
With OPTICAM it is possible to create the cutting machining automatically or by manually selecting faces or model edges.

Manual Creation of the Cutting Machining

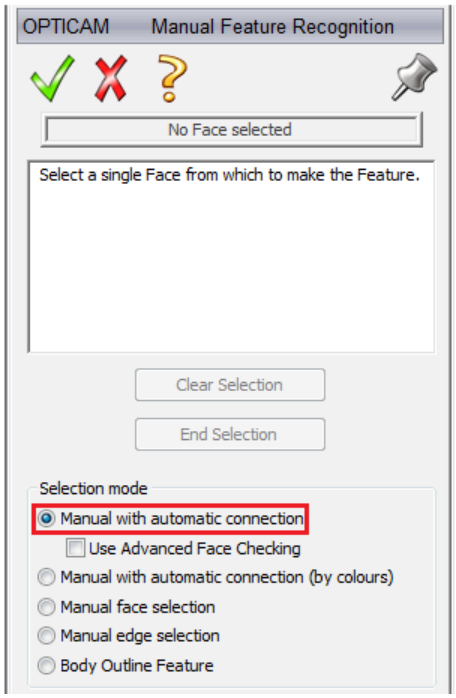
Initially, we want to show you how to machine a single geometry by manually selecting its faces with the mouse.

Right-click the branch **Operation1** in the OPTICAM Feature Manager.

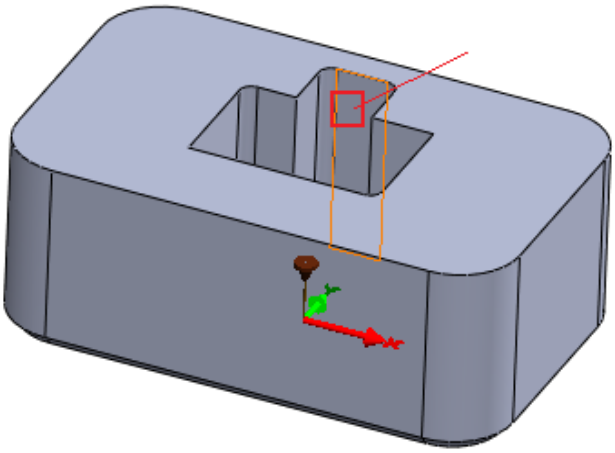
In the context menu, click **Manual feature recognition**.



In the **Manual Feature Recognition** dialog, activate the option **Manual with automatic connection**. After selecting this option, you click on a face and the system automatically finds all faces that can be wire cut and that are connected with the selected face.

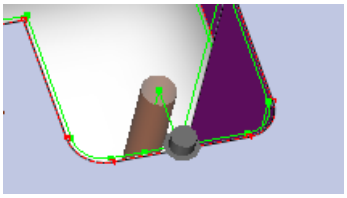


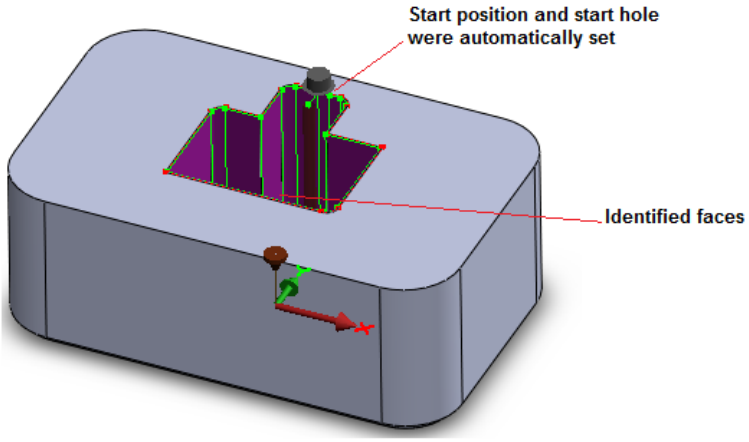
Click on a face of the internal aperture.



The system automatically identifies all faces belonging to the selected aperture and allocates the programmed cutting technology (1 cut). In addition, the start hole is positioned in front of the selected face and the start position in the center of the selected face.

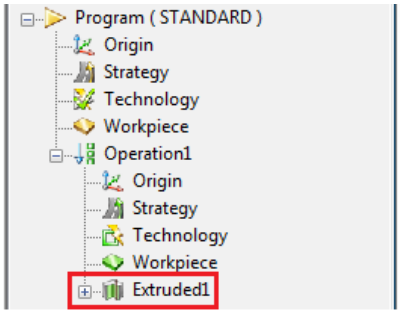
User advice: The **start position** is in the center of the selected face and the **start hole** is perpendicular to and in front of the selected face (see below).





Accept the selection by clicking  .

For every selected aperture, **OPTICAM** creates a **separate feature** in the OPTICAM Feature Manager. As only one aperture has been selected, there should also be only 1 feature:



If you click on a feature in the OPTICAM Feature Manager, OPTICAM highlights the geometry of the selected object and additionally displays the tool paths, the bridge, and the start hole.

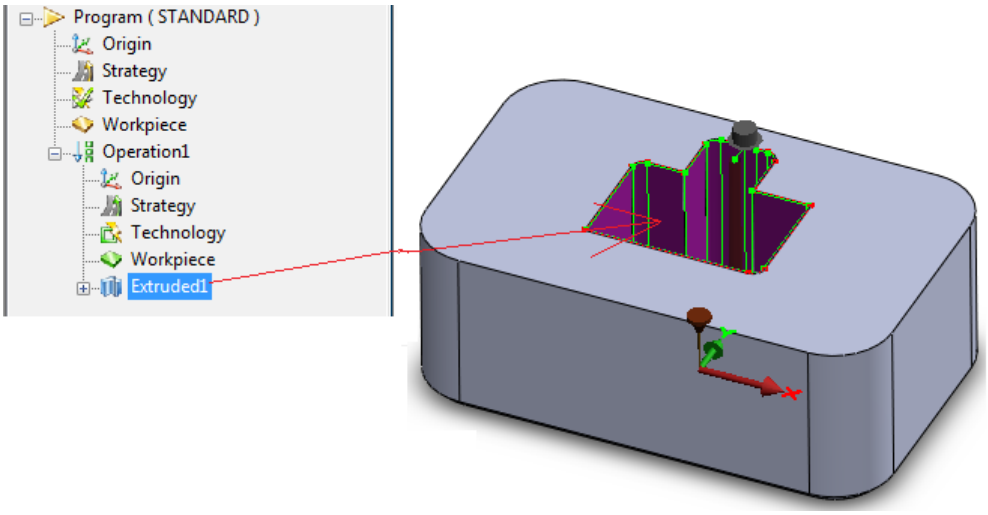
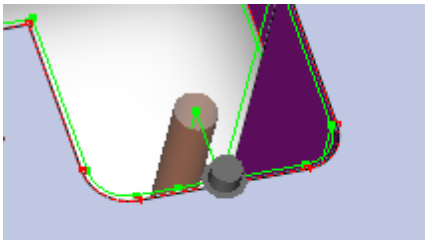


Diagram of the tool paths and the start hole.



Machining Simulation (Graphic Simulation)

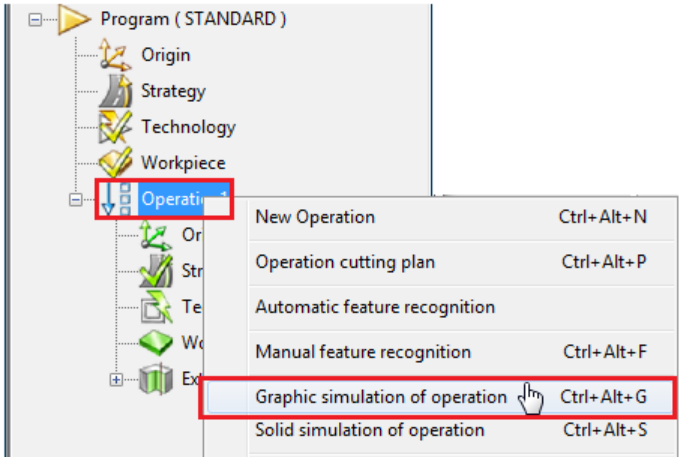
In the following, we will check the machining operations through a simulation. **OPTICAM** has two simulation forms:

Graphic Simulation simulates the tool paths WITHOUT depicting the material removal.

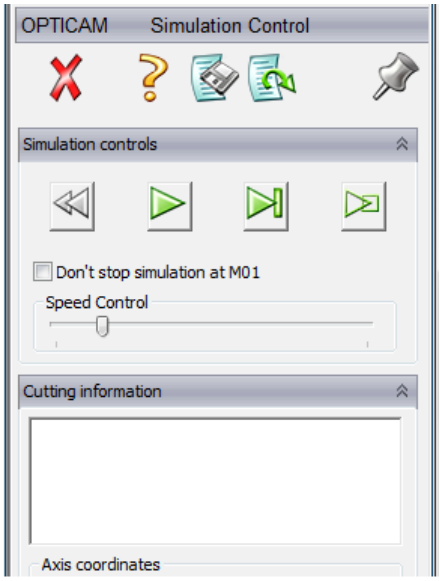
Solid Simulation simulates the tool paths and depicts the material removal.

Right-click the branch **Operation1** in the OPTICAM Feature Manager.

In the context menu, select **Graphic simulation of operation**.

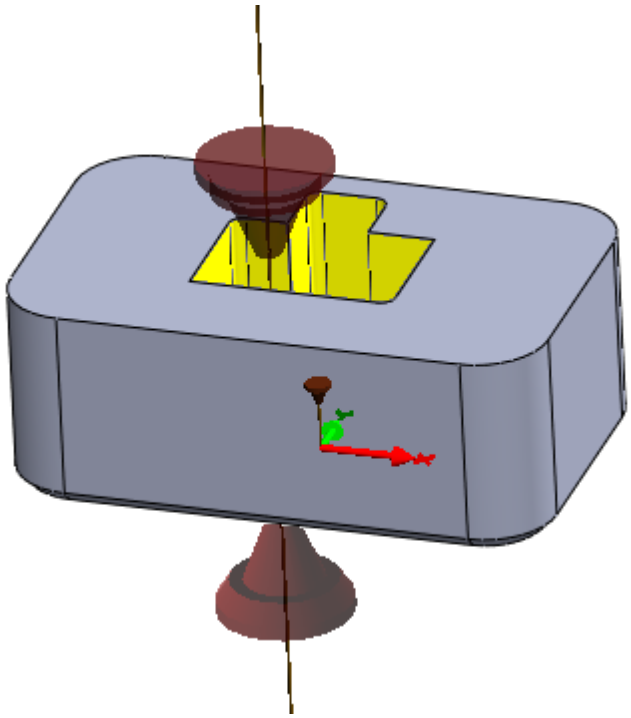


A dialog opens that allows you to control the simulation:



Click  to start the simulation. Use **Speed Control** to control the simulation speed.

At the end of the first cut, the simulation should stop as a machine stop (e.g. M01) is programmed before the cutting off of the bridge.



Use the  button to restart the simulation, and simulate the rest of the program.

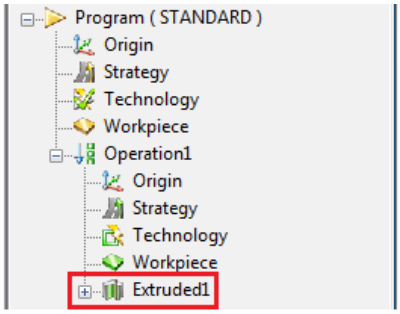
 **Tip:** With the  button you can reset the simulation and then restart it.

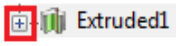
Close the simulation dialog by clicking  .

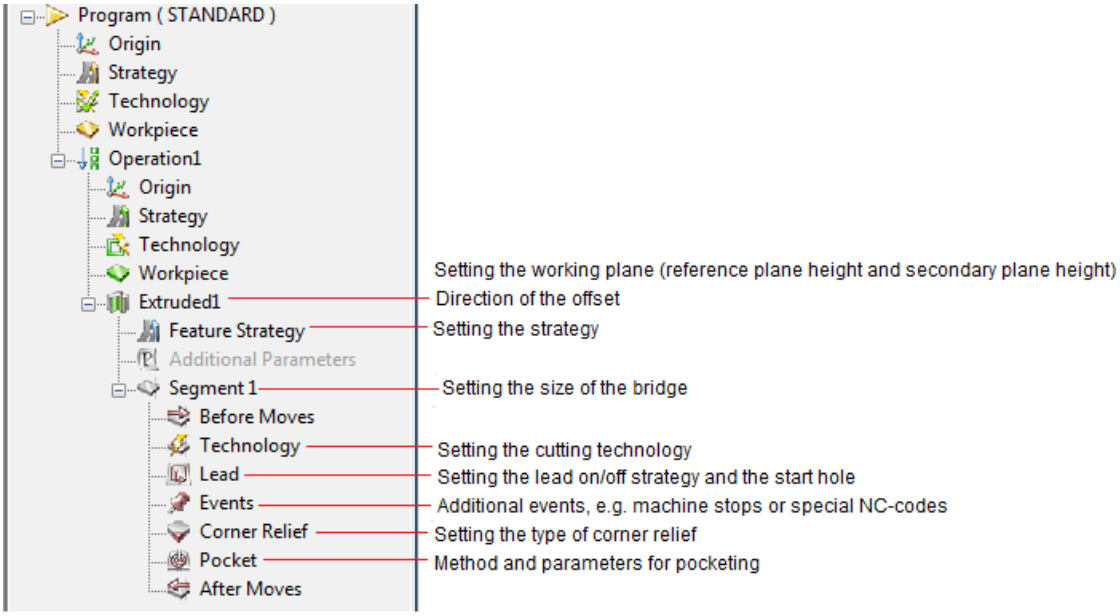
Further Functions of the OPTICAM Feature Manager

With the OPTICAM Feature Manager**,** you can influence and modify all cutting parameters (e.g. the start hole position, the lead on/off, the used cutting technology …).

Each programmed geometry has a separate feature in the OPTICAM Feature Manager.



Clicking the **+** symbol in front of a feature  opens the selected feature so that all cutting parameters are displayed in a tree structure.



If you want to modify for example the parameters of the **Lead on/off**, you have to Double-click the branch Lead.

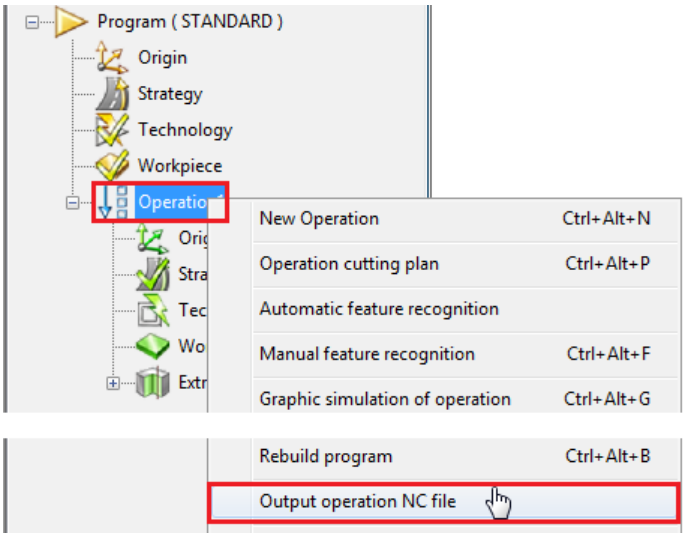
If you want to modify the **cutting technology**, Double-click the branch Technology.

In this first example, no modification of the cutting parameters shall be carried out; the following exercises will inform you in detail about the modification and adjustment of cutting parameters.

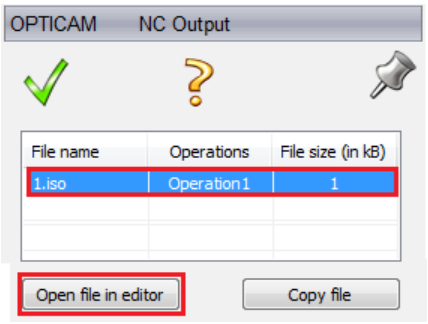
Creating the NC Program

At the end of this exercise, we create the NC program.

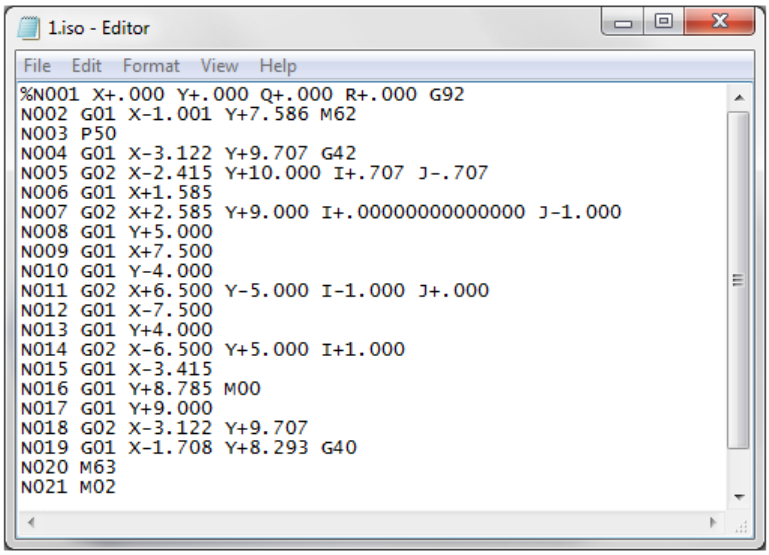
Right-click the branch **Operation1** in the OPTICAM Feature Manager and select **Output operation NC file**.



In the background, the NC program for the selected machine is created. Afterwards, a dialog for the editing of the NC file will be opened in the OPTICAM Feature Manager.



Click the NC file name and afterwards the Open file in editor button. A text editor displaying the just created NC program will be opened:



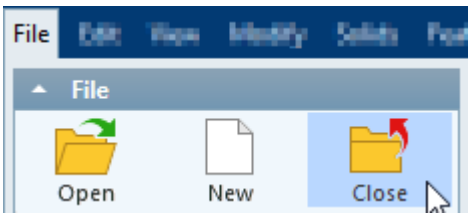
Close the editor  . Close the dialog **NC-OUTPUT** as well: 

Saving the OPTICAM Program

At the end, you save your work.

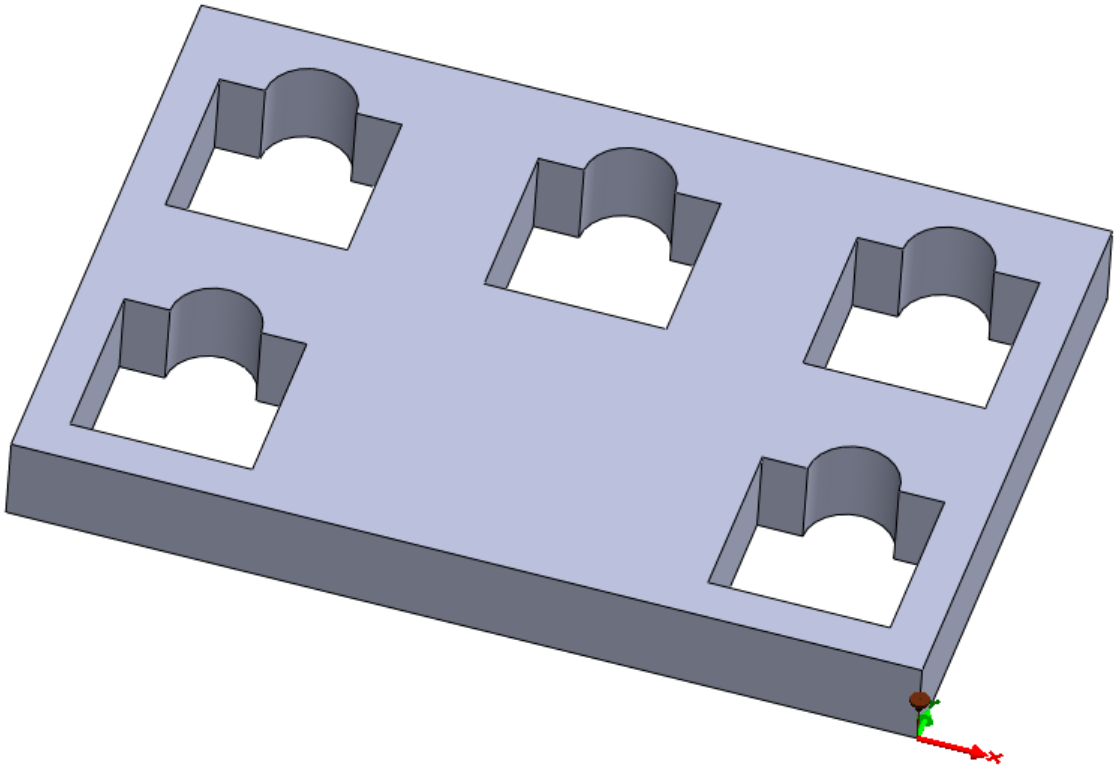
User advice: In addition to the GibbsCAM file **GC14_Opticam_example_1.vnc**, a second file containing the OPTICAM machining information for the current program will be saved. This OPTICAM file has the file extension .**wire.opticam**.

Close the file **GC14_Opticam_example_1.vnc**.



This exercise is now completed.

Example 2: Multiple Cylindrical Cuts

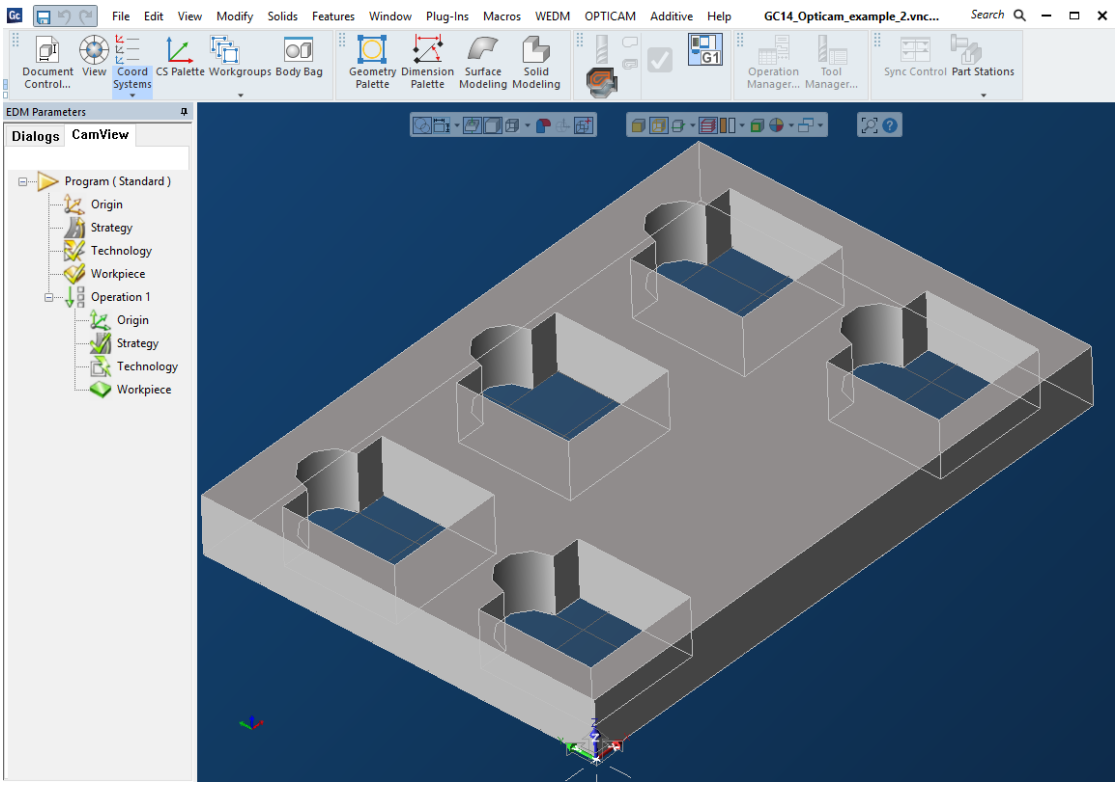


Demonstrated in This Example

Opening a file: GC14_Opticam_example_2
Using the **Program** Dialog for Basic Settings
Defining the Machine Origin
Defining the Cutting Technology (Manual Input)
Creating the Workpiece (Billet) Geometry
Setting the Cutting Heights (Reference Plane Height and Secondary Plane Height)
Setting the Operation Strategy
Manual Creation of the Cutting Machining
Machining Simulation (Graphic Simulation)
Machining Simulation (Solid Simulation)
Modifying the Operation Strategy
Modifying the Length of the Lead on/off
Modifying Various Machining Features at the Same Time
Creating the NC Program
Checking the NC Program at a Later Time

Open GC14_Opticam_example_2

In GibbsCAM, open the sample file **GC14_Opticam_example_2.vnc**.

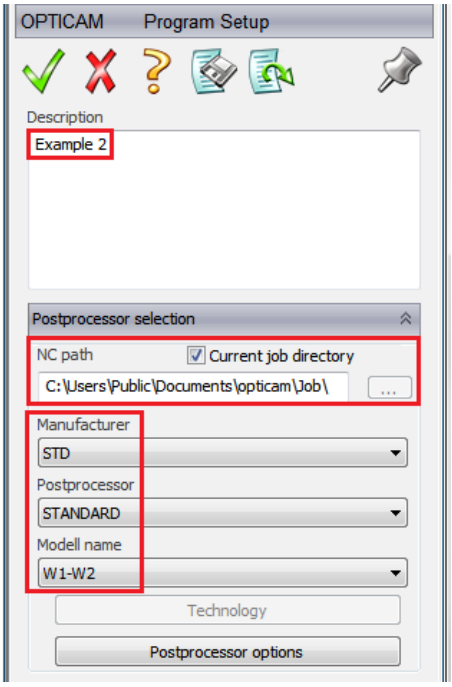


Using the Program Dialog for Basic Settings

As a first step, we define the basic settings. This means that we define the wire EDM machine for which the program will be created and where the NC output file will be placed; additionally, we define the start position of the program.

Double-click the branch **Program (STANDARD)** in the OPTICAM Feature Manager to open the dialog for the definition of the program default settings. Fill in the dialog as shown to the right:

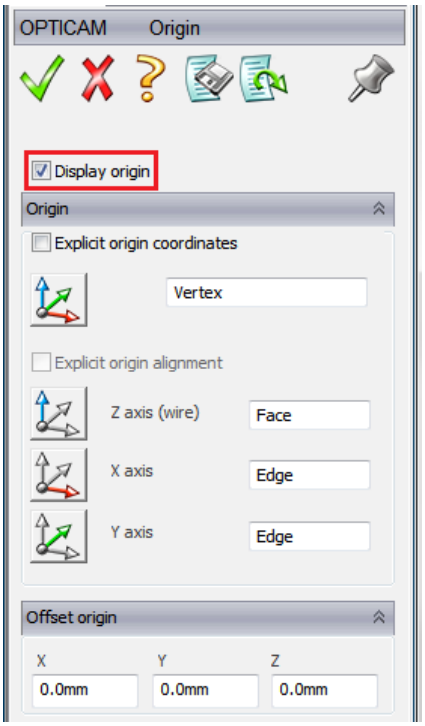
Accept changes by clicking  .



Defining the Machine Origin

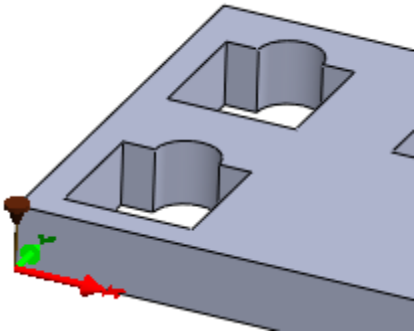
In the next step, the position of the machine origin will be placed at the lower right corner of the model.

Double-click the branch **(Program) Origin** in the OPTICAM Feature Manager to open the dialog for the definition of the machine origin.

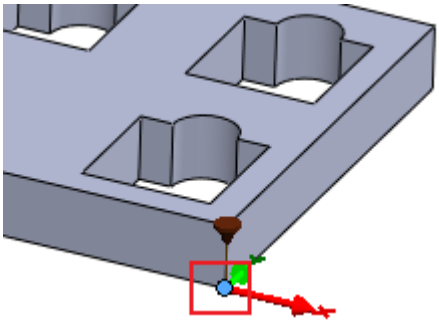


The origin is graphically displayed by a symbol. The Z-axis of the origin is symbolized by a graphic of the wire guide.

Currently, the machine origin is on the **left** side at the bottom of the workpiece (see figure at left).



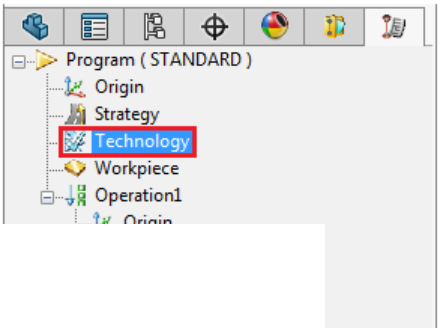
To define the origin at the lower **right** corner, put the cursor at the lower right corner and left-click. This moves the origin to the selected position.



Close the **Machine Origin** dialog by clicking  .

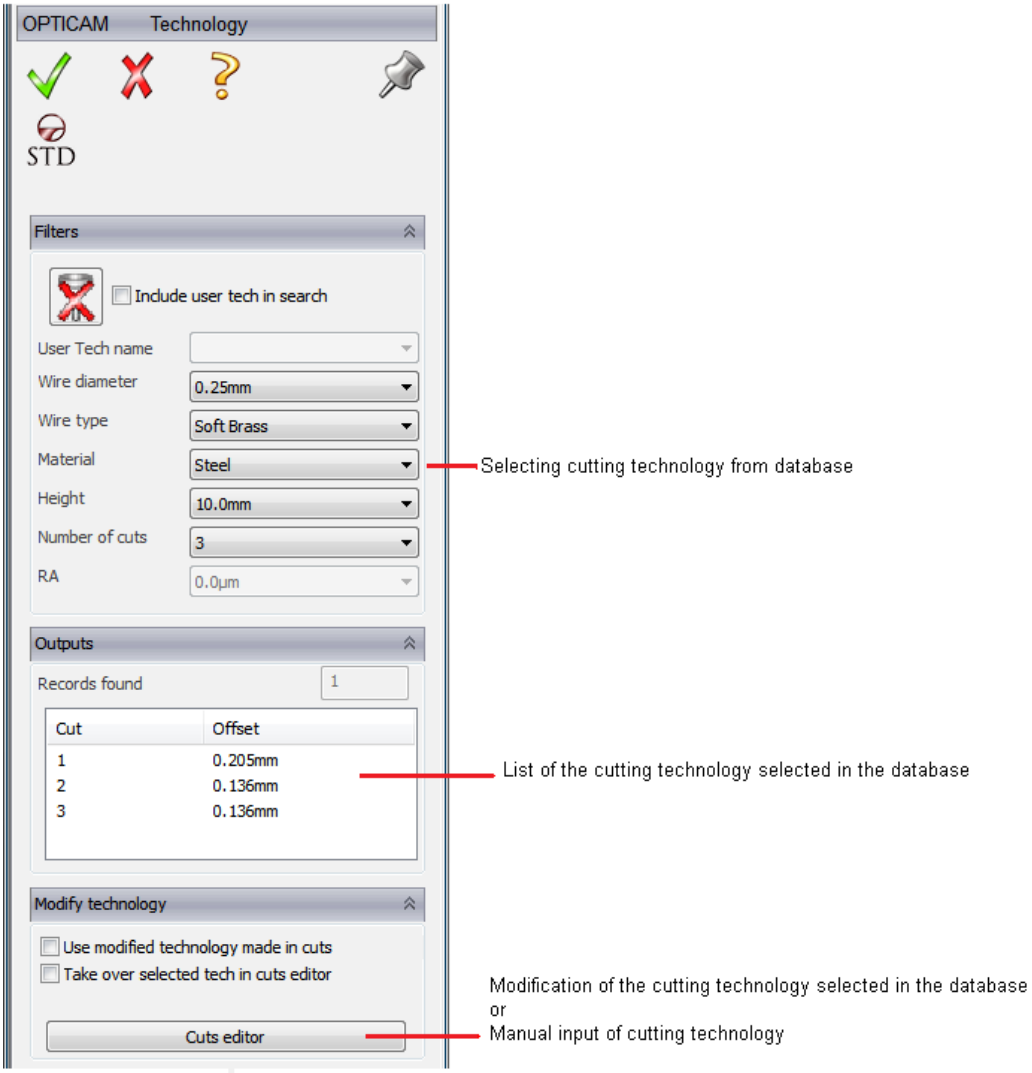
Defining the Cutting Technology (Manual Input)

In the next step, we will define with which cutting technology the workpiece is to be machined.



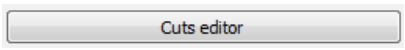
Double-click the branch **(Program) Technology** in the OPTICAM Feature Manager.

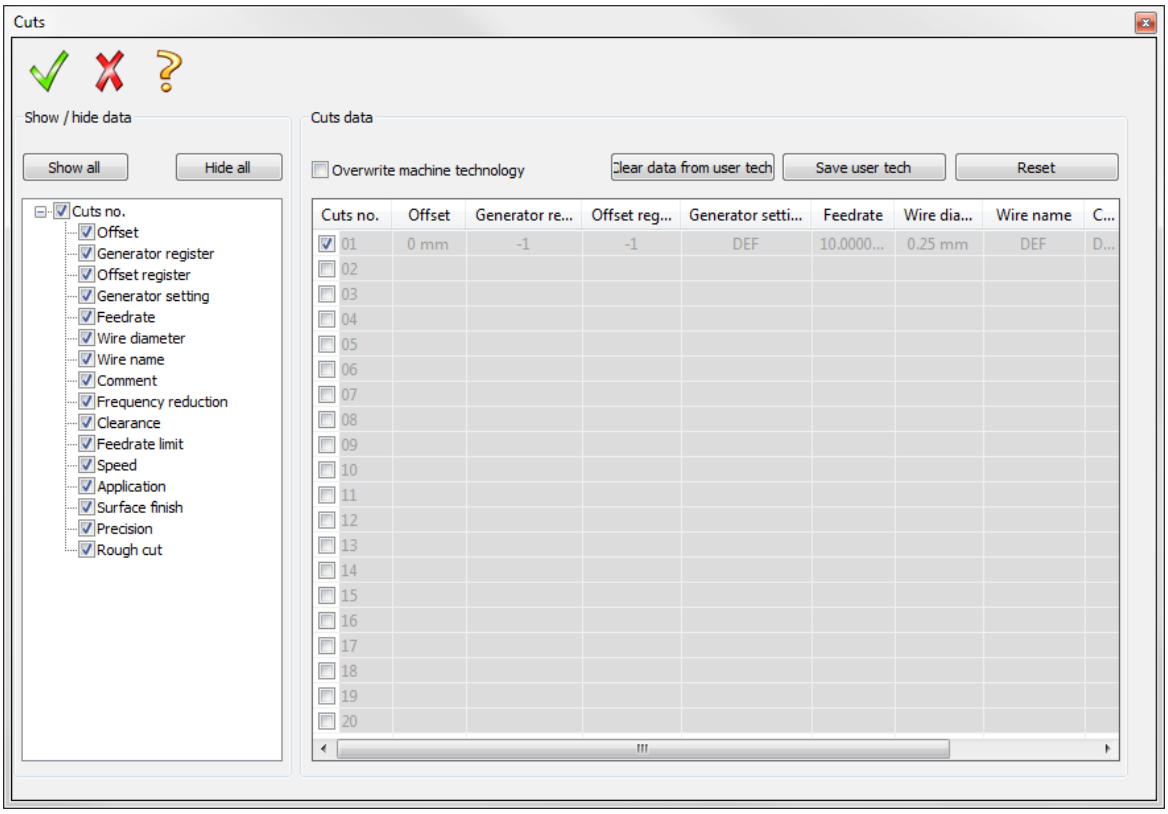
This opens the dialog for the definition of the cutting technology.



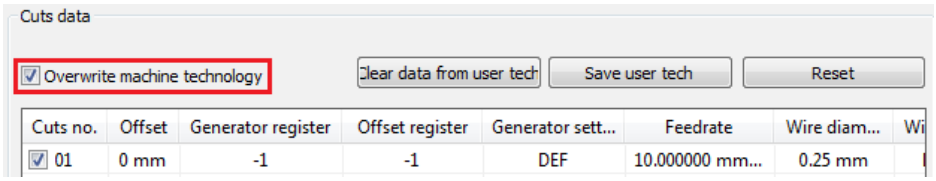
You can either select the parameters of the cutting technology from a database or enter them manually.

In the first few exercises, we will enter the cutting technology manually. You can do this as follows:

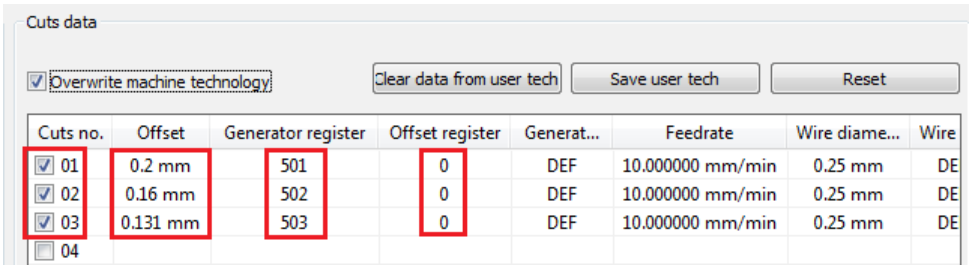
Click the  button. A dialog opens where you can manually enter the cutting technology.



Activate the input fields by selecting the option  .



To machine the geometries with **3 cuts**, please set the dialog as follows:



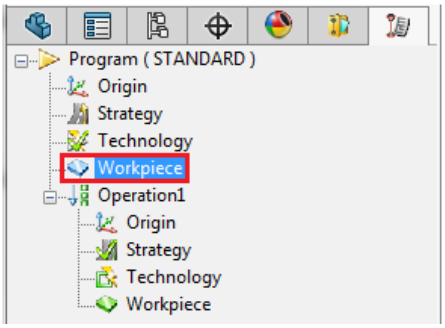
Accept changes by clicking  button. The system will go back to the **Technology** dialog.

Close this dialog as well by clicking on  .

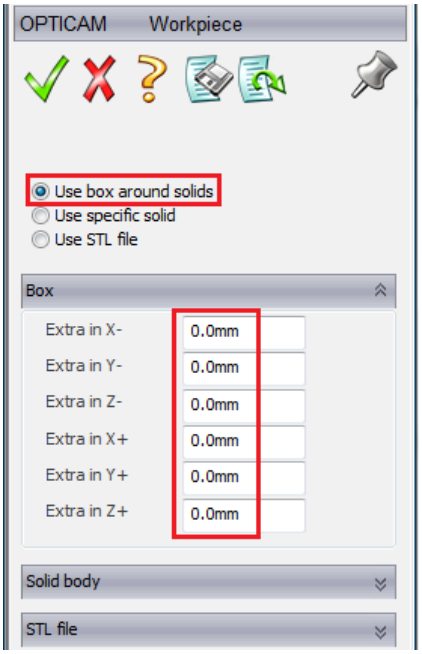
Creating the Workpiece (Billet) Geometry

In the next dialog, you can define the geometry of the workpiece (billet).

Double-click the branch **(Program) Workpiece** in the OPTICAM Feature Manager.



Fill in the dialog as follows:

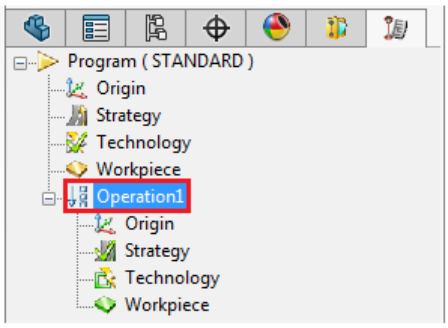


Accept changes by clicking  .

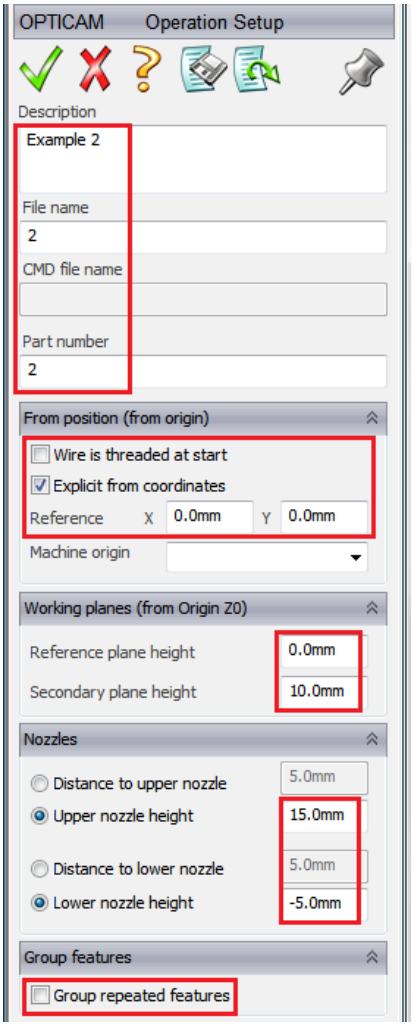
Setting the Cutting Heights (Reference Plane Height and Secondary Plane Height)

The following dialog sets among other things the name of the NC output file as well as the Z-height of the Reference and the Secondary Plane Height.

Double-click the branch **Operation1** in the OPTICAM Feature Manager.



In the field **Description**, you can enter a comment to the program. In the field **FileName**, the name of the NC output file is set. Additionally, you set the general **Start Position** and the height of the working planes (**Reference plane height** / **Secondary plane height**).



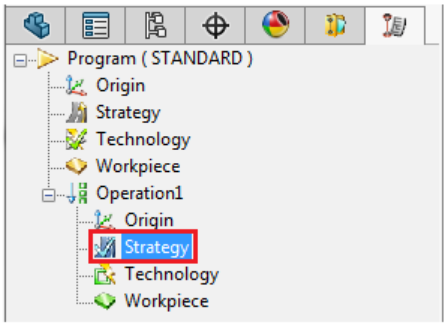
User advice: Do **NOT** activate the option **"Group repeated Features"**.

Accept changes by clicking  .

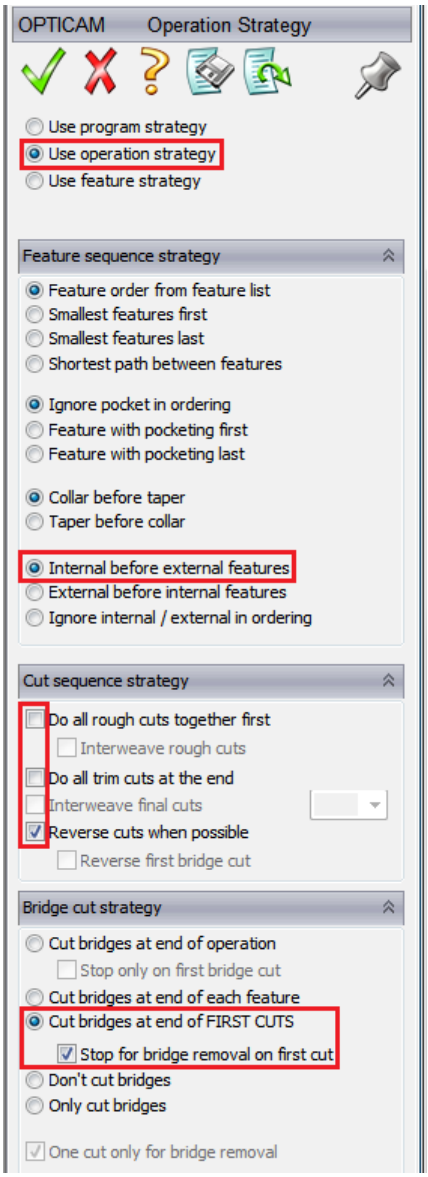
Setting the Cutting Strategy

In the next dialog, you set the strategy with which the workpiece is to be machined. The geometry will be machined in a way that the bridge will be cut at the end of the first cut and afterwards the two trim cuts will be done in reverse cuts (forwards/backwards).

Double-click the branch **(Operation) Strategy** in the OPTICAM Feature Manager.



Set the dialog as shown on the right, and then accept changes by clicking  .



Creating the Cutting Machining

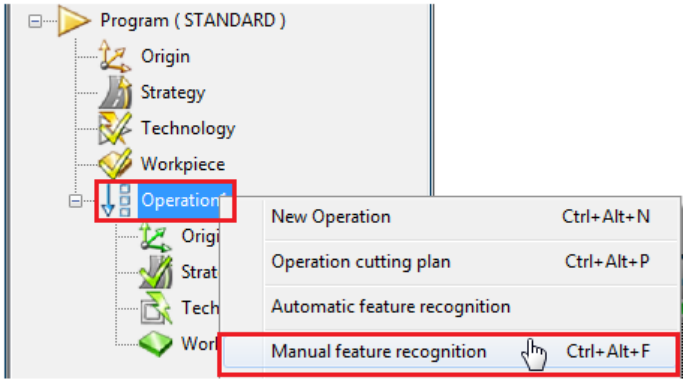
With OPTICAM it is possible to create the cutting machining automatically or by manually selecting faces or model edges.

Manual Creation of the Cutting Machining

Initially, we want to show you how to machine single geometries by manually selecting their faces with the mouse.

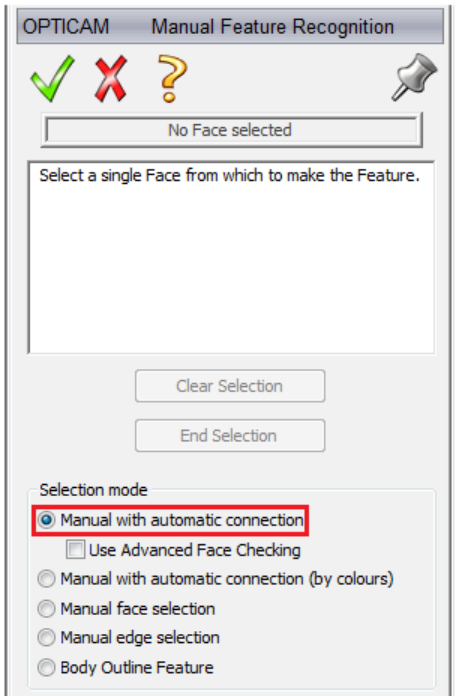
Right-click the branch **Operation1** in the OPTICAM Feature Manager.

In the context menu, click **Manual feature recognition**.



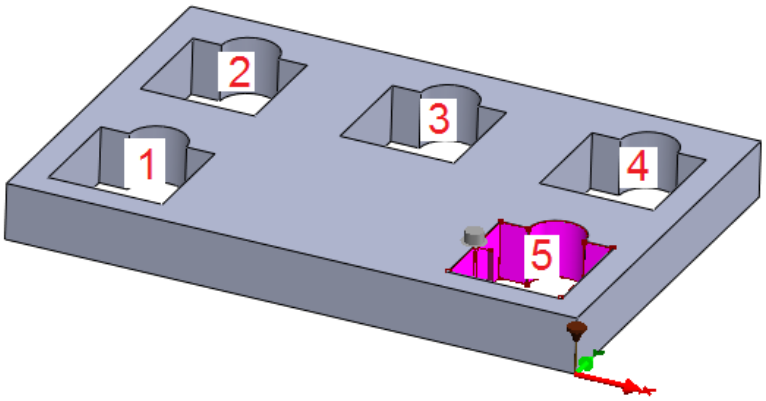
In the **Manual Feature Recognition** dialog, choose the option **Manual with automatic connection**.

After choosing this option, you can click on a face and the system will automatically find all faces that can be wire cut and are connected with the selected face.



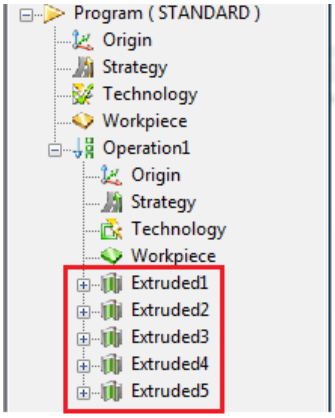
Click successively on one face of each of the five apertures.

Click each time on the **external surface on the left** so that the machining operation will be started in the center of the selected surface.



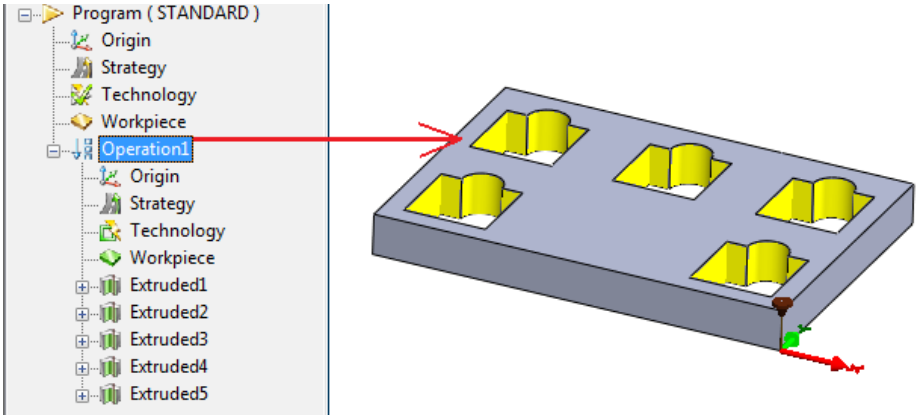
Accept the selection by clicking  .

For each selected aperture, OPTICAM creates a separate feature in the OPTICAM Feature Manager. Because five apertures have been selected, the tree shows five features:



User advice: If only one feature is shown, you have probably activated the option **"Group repeated Features"** in the dialog **Operation1**. This unites identical features into one feature.

If you click the branch **Operation1** in the OPTICAM Feature Manager**,** all geometry of all existing features will be highlighted.



If you click on one or various features in the OPTICAM Feature Manager, OPTICAM will highlight the geometries of the selected objects and additionally display the tool paths, the bridge and the start hole.

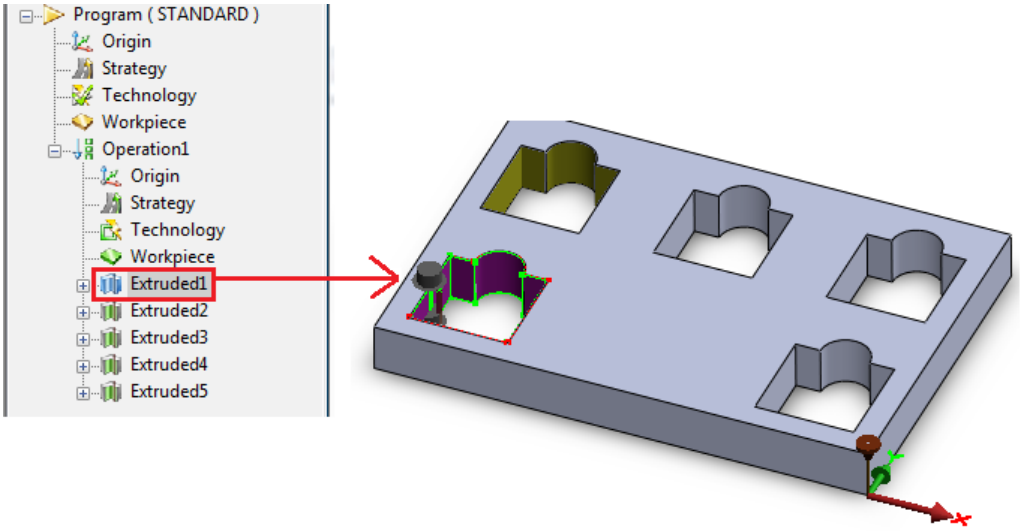
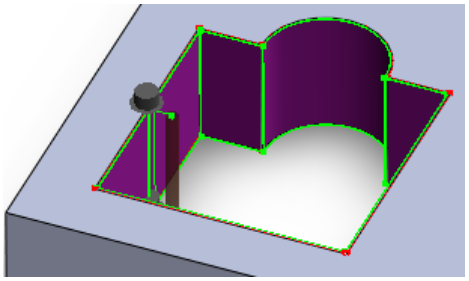


Diagram of the tool paths and the start hole:



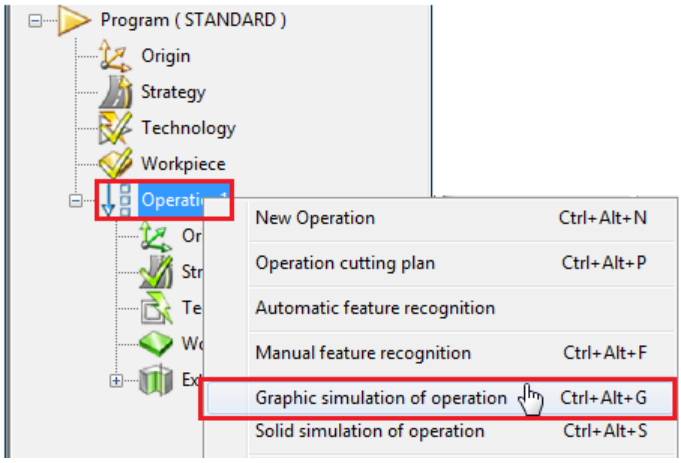
Machining Simulation (Graphic Simulation)

In the following, we will check the machining operations through a simulation. **OPTICAM** has two simulation forms:

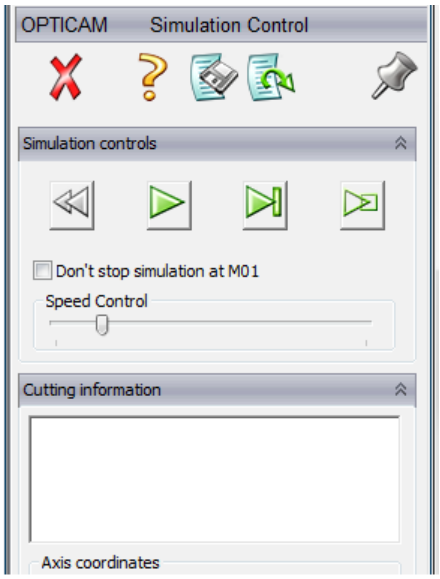
Graphic Simulation simulates the tool paths WITHOUT depicting the material removal.

Solid Simulation simulates the tool paths and depicts the material removal.

Right-click the branch **Operation1** in the OPTICAM Feature Manager. In the context menu, select **Graphic simulation of operation**.

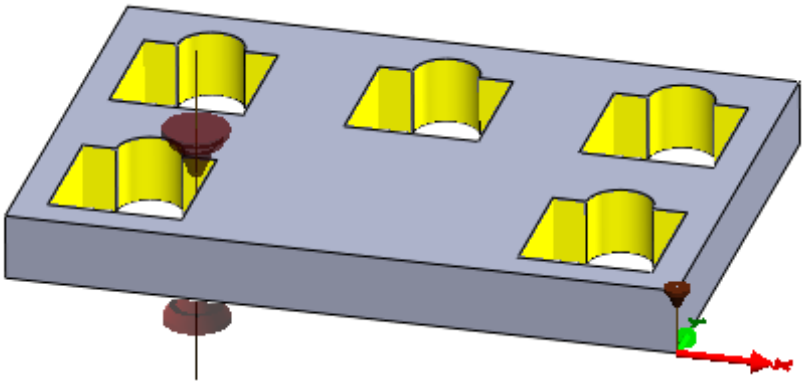


A dialog opens that allows you to control the simulation:

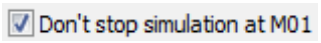


Click  to start the simulation. Use **Speed Control** to control the simulation speed.

At the end of the first cut, the simulation should stop as a machine stop (e.g. M01) is programmed before the cutting off of the bridge.



Use the  button to restart the simulation.

 **User advice:** Activate the option 

if the simulation is not supposed to stop at every machine stop (e.g. M01).

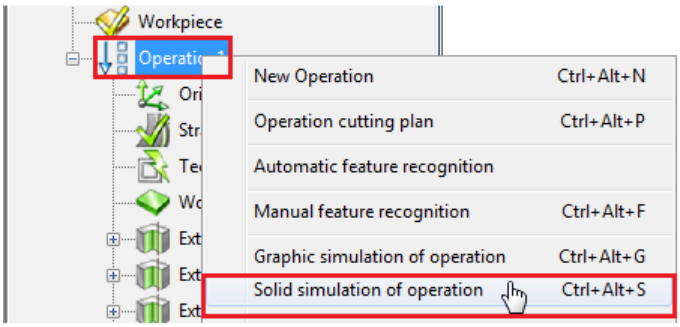
Simulate the complete program. Close the simulation dialog by clicking  button.

Machining Simulation (Solid Simulation)

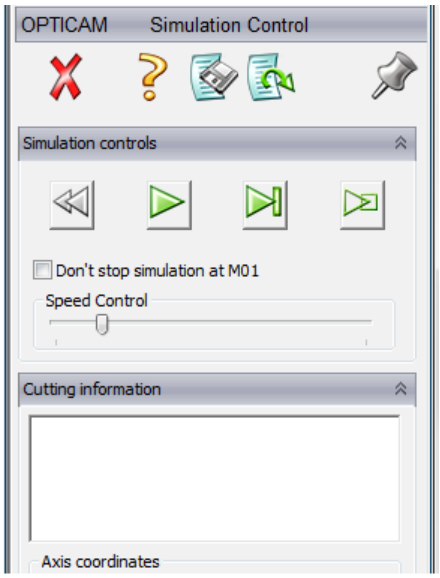
You may have noticed that the **Graphic Simulation** does not depict the material removal, and the bridge is not clearly depicted either.

Therefore, we will now use the **Solid Simulation** which will give you a more detailed overview of the tool paths.

Right-click the branch **Operation1** in the OPTICAM Feature Manager. In the context menu, click **Solid simulation of operation**.

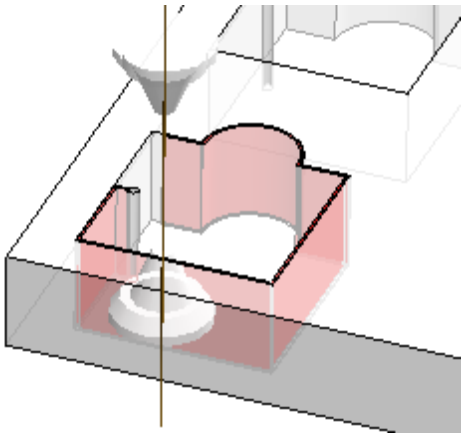


A dialog opens that allows you to control the solid simulation:



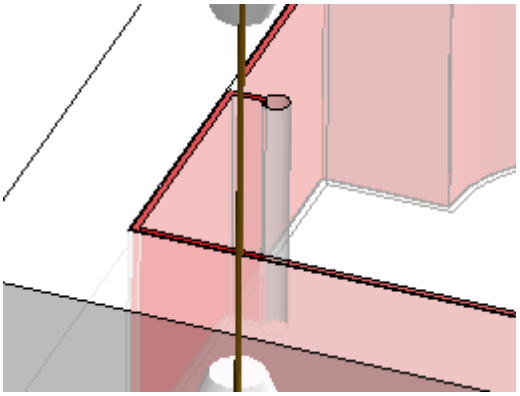
Click  to start the simulation. Use **Speed Control** to control the simulation speed.

The simulation type "**Solid Simulation**" clearly depicts the material removal and the position of the start holes.

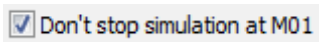


At the end of the first cut, the simulation should stop as a machine stop (e.g. M01) is programmed before the cutting off of the bridge.

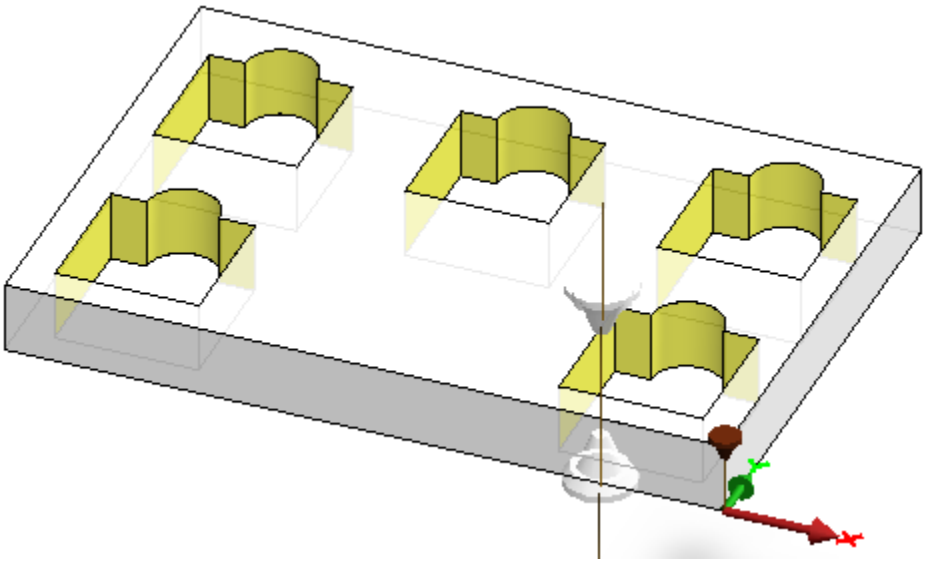
By extending the area around the start hole, you can clearly identify the bridge which has been created by the machine stop.



Use the  button to restart the simulation.

 **User advice:** Activate the option  if the simulation is not supposed to stop at every machine stop (e.g. M01).

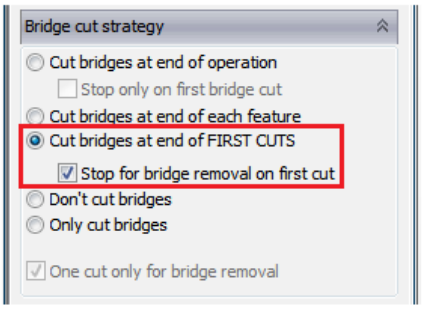
Simulate the complete program.



 **Tip:** With the  button you can reset the simulation and then restart it.

Close the simulation dialog by clicking  .

Modifying the Operation Strategy



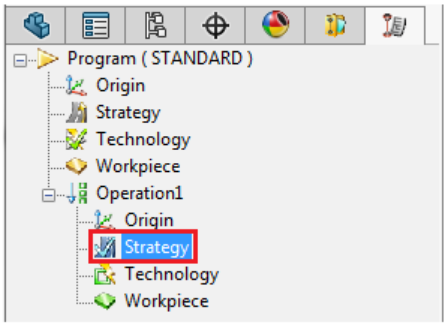
Perhaps you remember that at the beginning of the program we set "**Cut bridges at end of FIRST CUTS**" and "**Stop for bridge removal on first cut**" as our cutting strategy.

A disadvantage of this strategy is that the operator has to remove the slug after every main cut.

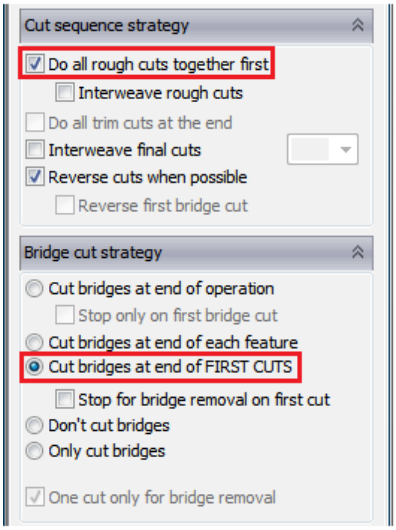
In many cases, it makes more sense to do **All Rough Cuts together first**, and then successively **cut all Bridges** and **do the Trim Cuts last**.

To set the above-mentioned strategy, proceed as follows:

Double-click the branch **(Operation) Strategy** in the OPTICAM Feature Manager.



Modify the strategy settings as follows:

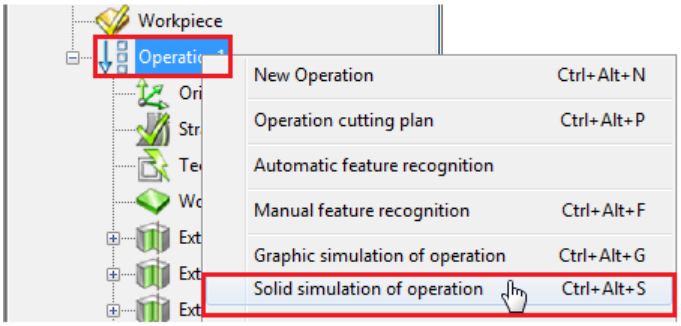


Accept changes by clicking  .

Repeated Machining Simulation (Solid Simulation)

To check the newly set strategy, we will restart the **Solid Simulation**.

Right-click the branch **Operation1** and select **Solid simulation of operation**.



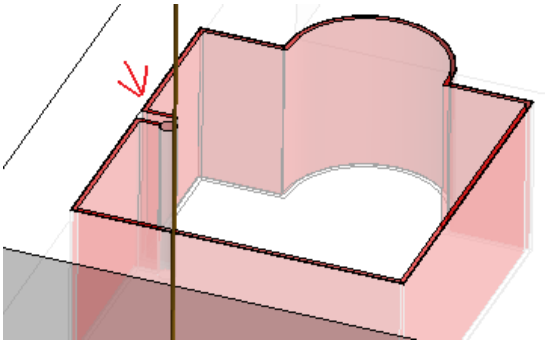
Use the  button to start the simulation.

Please continue working independently…

Close the simulation dialog by clicking  button.

Modifying Start Hole Position and Length of Lead off

In the **Solid Simulation,** you can see that the Start hole is too close to the contour and the Lead off is too long (see following figure).



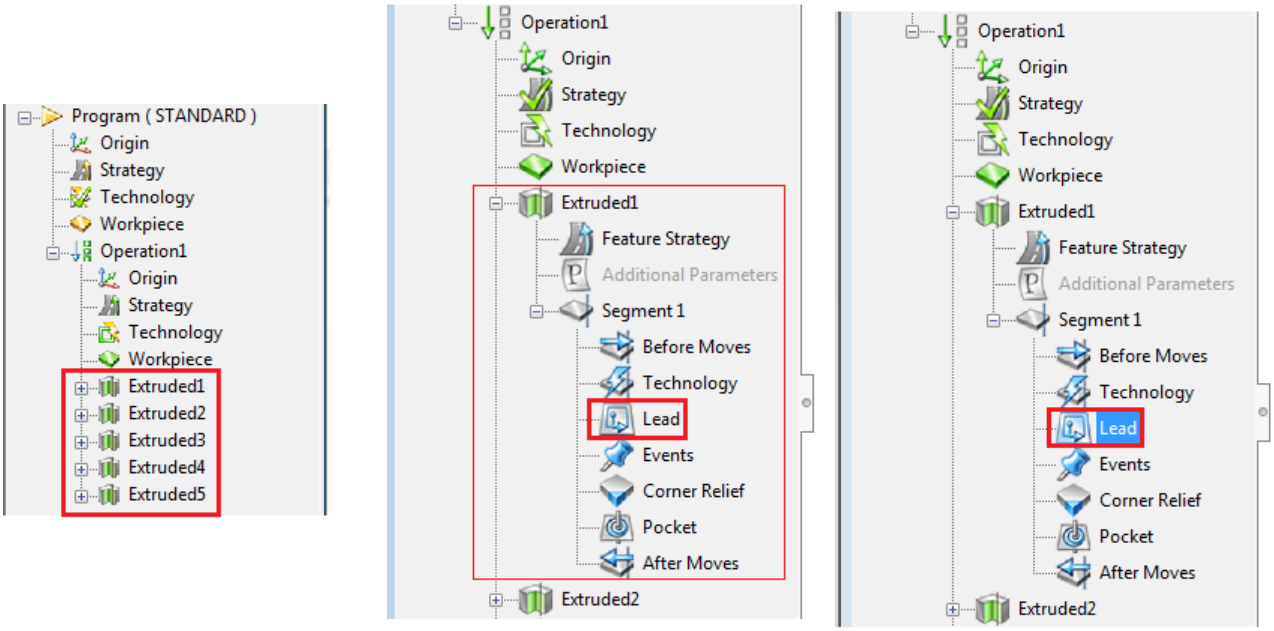
With the OPTICAM Feature Manager**,** you can influence and modify all cutting parameters. In this case, we want to modify the **Lead on/off parameters**, modify the start hole position and shorten the Lead off.

For each aperture, OPTICAM creates a separate feature in the OPTICAM Feature Manager.

Opening a feature leads to the display of all cutting parameters of the selected feature.

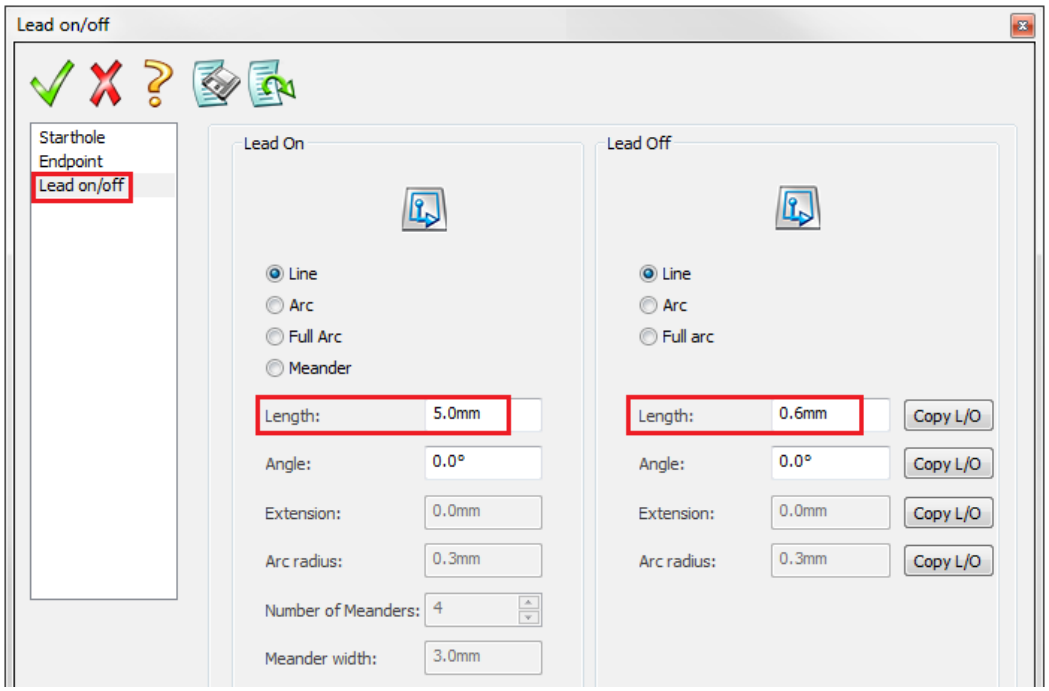
To modify the parameters of the **Lead on/off**, open the first feature of the OPTICAM Feature Manager and double-click the branch 

of that feature.



This opens the **Lead on / off** dialog.

Click the field **Lead on / off** and modify the length of the **Lead On** to **5** and of the **Lead Off** to **0.6** mm.



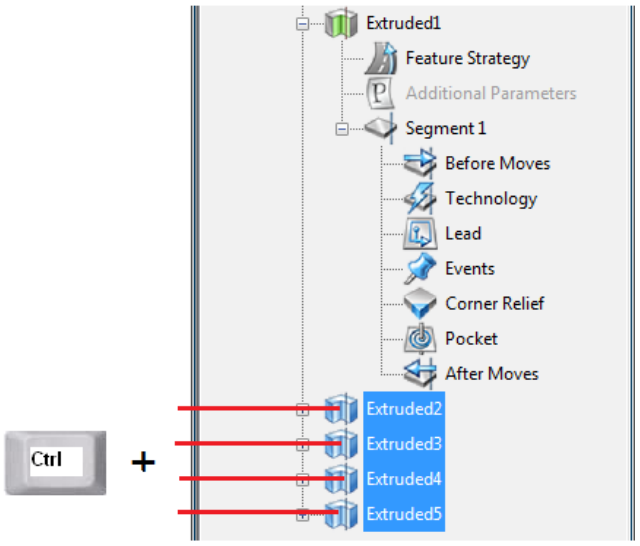
Accept changes by clicking  .

But the modification of the lead off only extends to the selected feature. To adjust the lead off of the other four features, you would have to repeat the process four times.

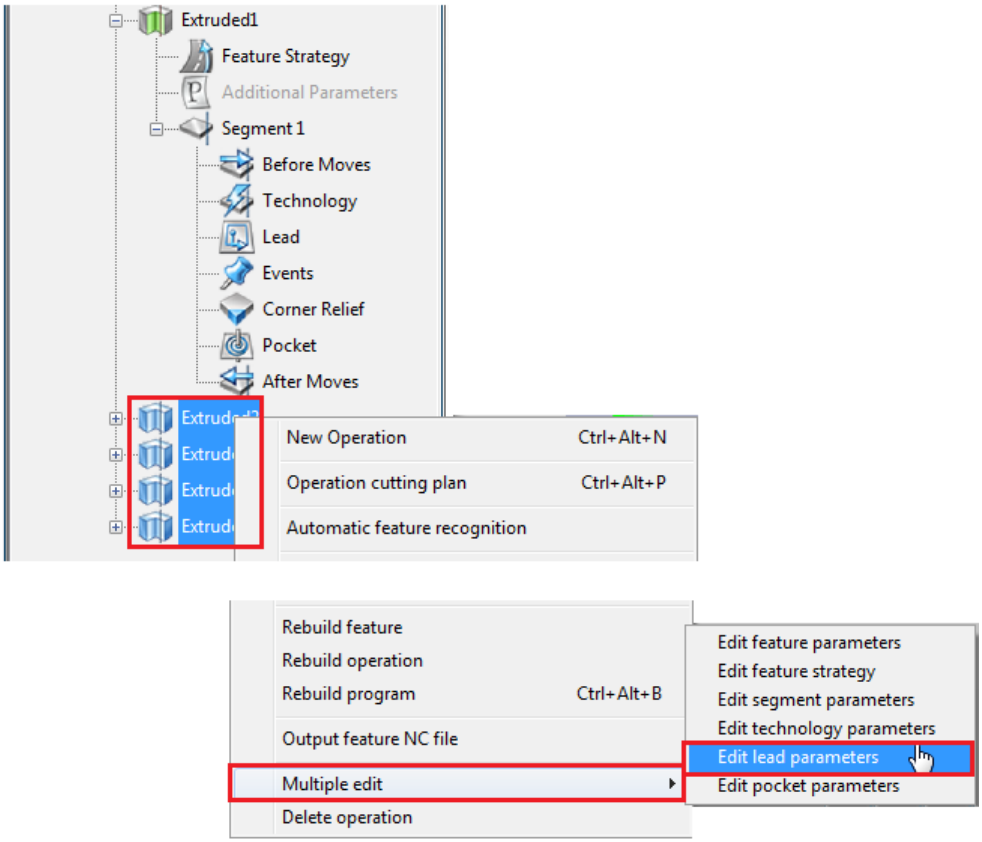
As this would be a bit laborious, OPTICAM offers the possibility to modify various features at the same time.

Modifying Various Machining Features at the Same Time

Holding down the **Ctrl** key, click successively on each of the four features in the OPTICAM Feature Manager that you have not yet modified.

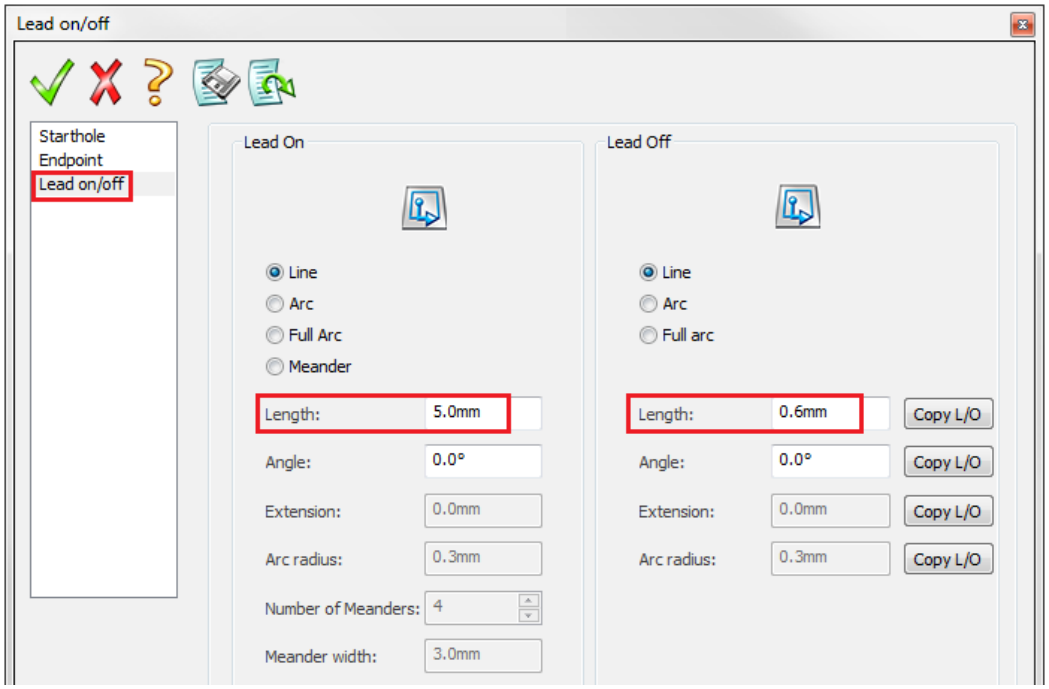


Now right-click one of the four features and, on the context menu, select **Multiple edit > Edit lead parameters**.



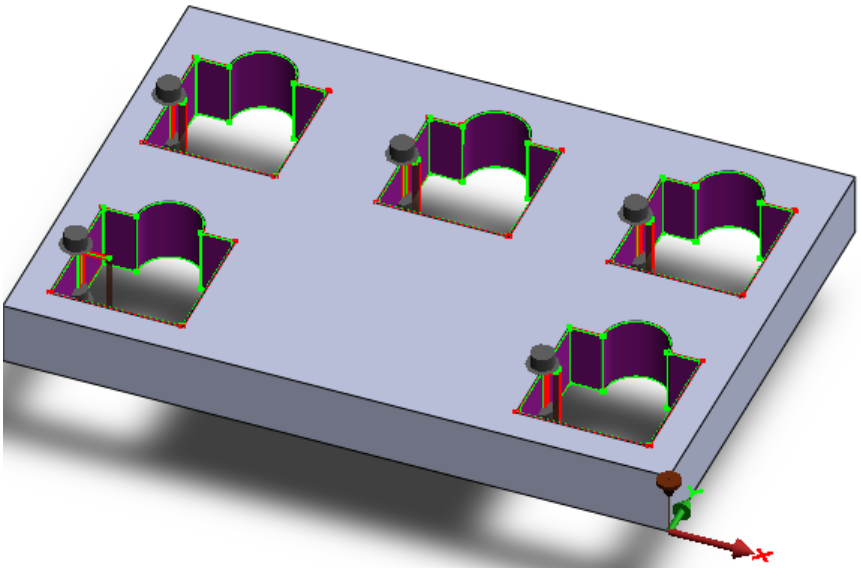
This opens the **Lead on/off** dialog.

Click the field dialog **Lead on/off** and modify the length of the **Lead On** to **5** and of the **Lead Off** to **0.6** mm.



Accept changes by clicking  .

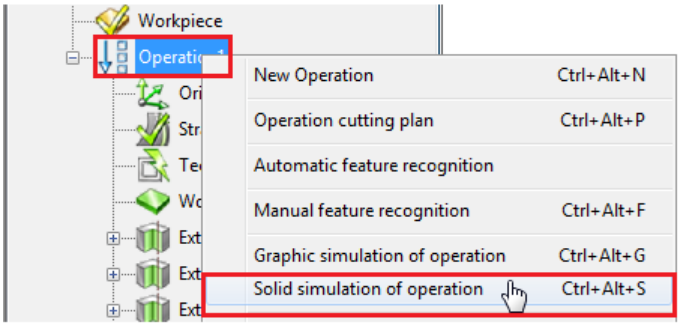
This time, the modification extends to **all** selected features.



Repeated Machining Simulation (Solid Simulation)

To check the modified Lead off, we will restart the **Solid Simulation**.

Right-click the branch **Operation1** in the OPTICAM Feature Manager and, on the context menu, click **Solid simulation of operation**.



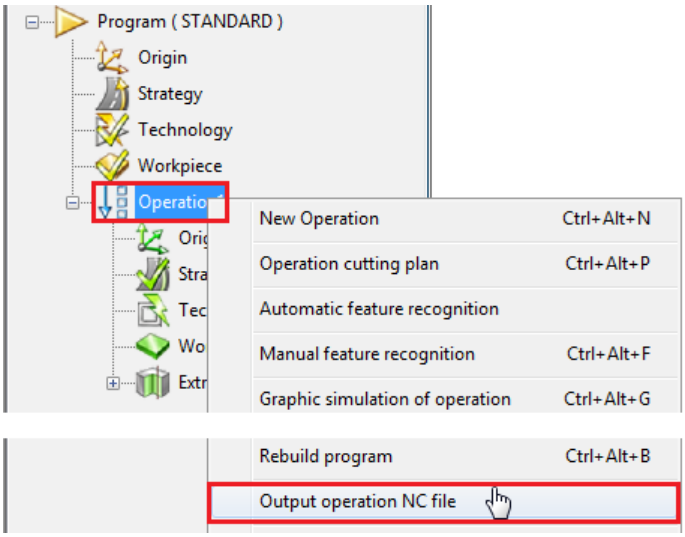
Please continue working independently…

Close the simulation dialog by clicking  button.

Creating the NC Program

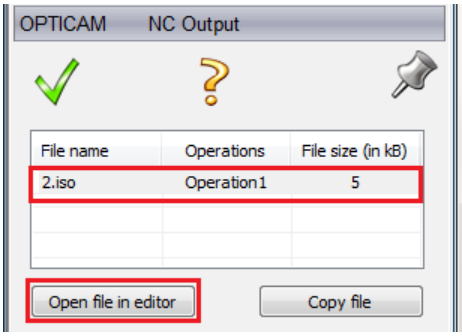
At the end of this exercise, we create the NC program.

Right-click the branch **Operation1** in the OPTICAM Feature Manager and select the command **Output operation NC file**.

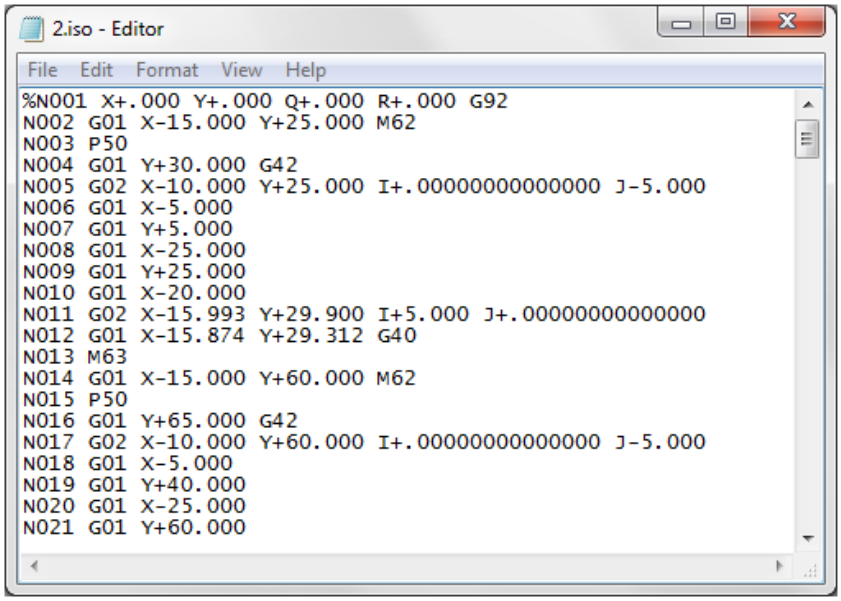


In the background, the NC program for the selected machine is created. Afterwards, a dialog for the editing of the NC file will be opened in the OPTICAM Feature Manager.

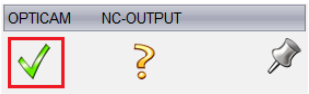
Click the file name in the dialog and then the Open file in editor button.



A text editor opens which displays the just created NC program:



Close the editor  .

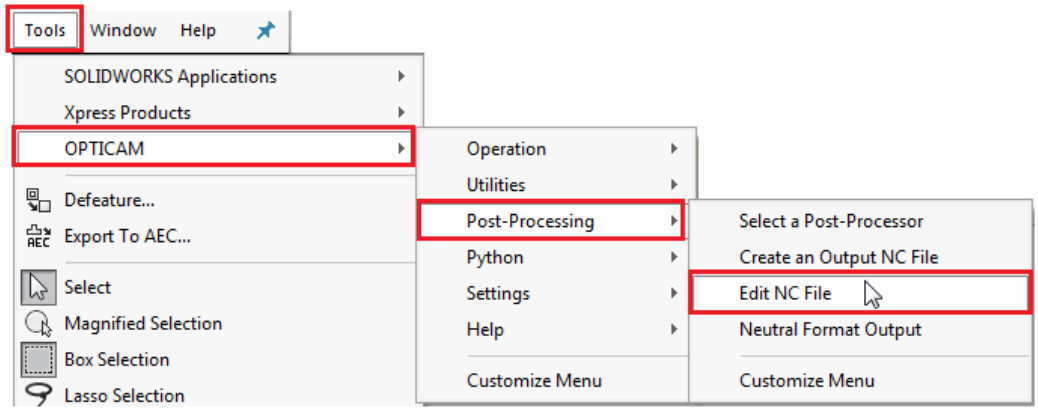
Close the dialog **NC-OUTPUT** as well 

.

Checking the NC Program at a Later Time

You can also examine the newly created NC program without having to create an NC file every time. You can do this as follows:

Select the menu order **Tools \ OPTICAM \ Post-Processing \ Edit NC File**.



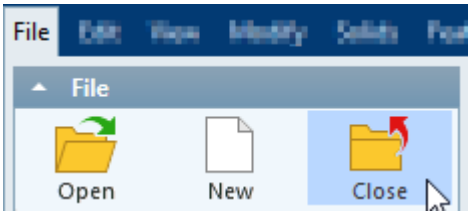
This opens the most recently created NC file in an editor.

Saving the OPTICAM Program

At the end, you should save your work.

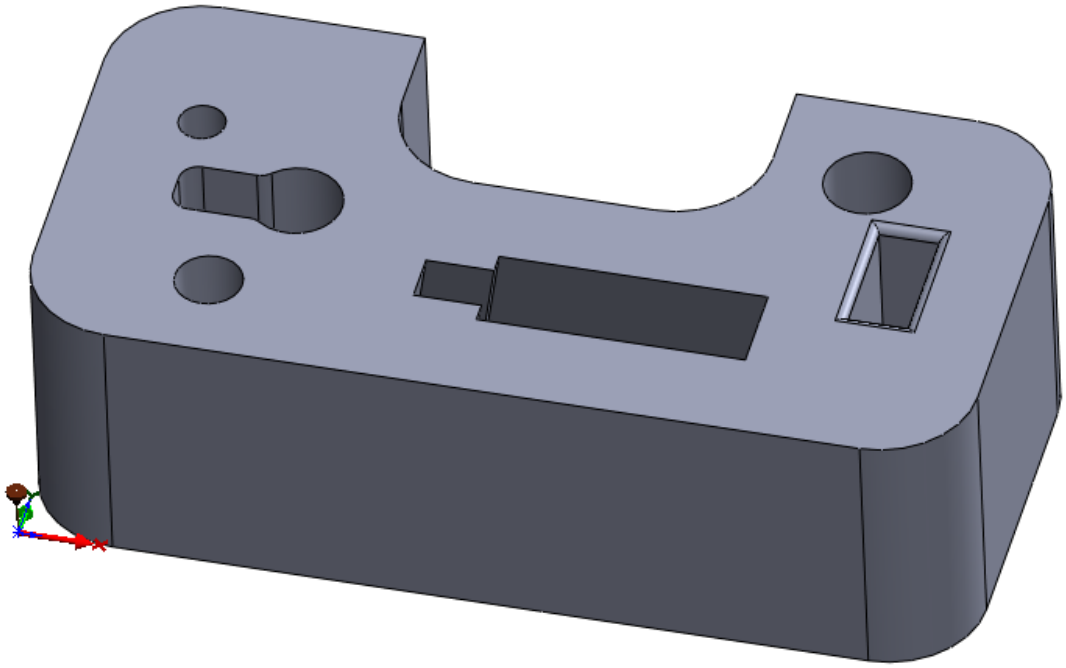
User advice: In addition to the GibbsCAM file **GC14_Opticam_example_2.vnc**, a second file containing the OPTICAM machining information for the current program will be saved. This OPTICAM file has the file extension .**wire.opticam**.

Close the file **GC14_Opticam_example_2.vnc**.



This exercise is now completed.

Example 3: Taper, Variable Taper

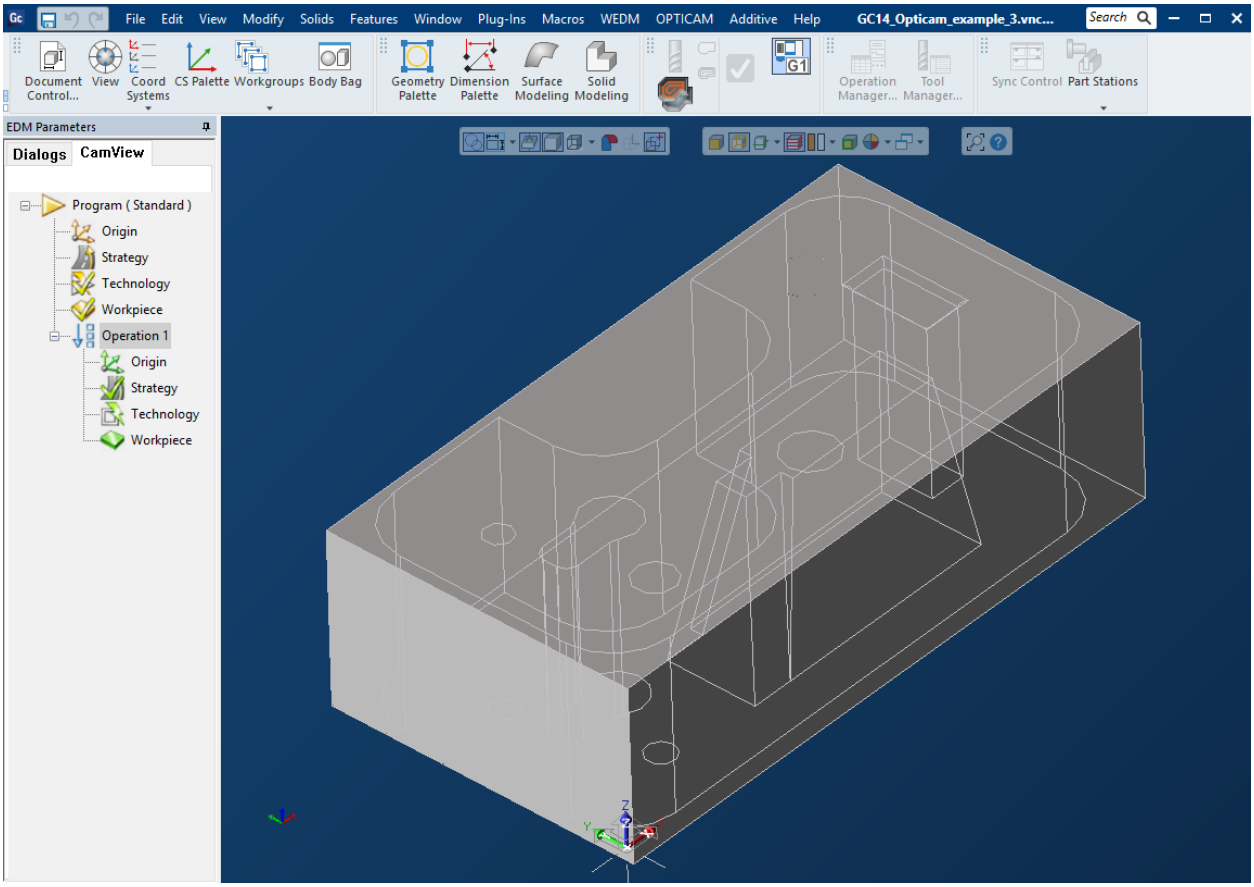


Demonstrated in This Example

Opening a file: GC14_Opticam_example_3
Positioning the Machine Origin in the Center of an Edge
Defining the Cutting Technology (Technology from Database)
Creating the Workpiece (Billet) Geometry
Setting the Cutting Heights (Reference Plane Height and Secondary Plane Height)
Setting the Cutting Strategy
Automatic Creation of the Cutting Machining through Feature Recognition
Setting the Position of the Start Hole and the Length of the Lead off
Adjusting the Size of the Bridge
Machining Simulation (Solid Simulation)
Depiction of Slugs during the Simulation
Creating the NC Program

Open GC14_Opticam_example_3

In GibbsCAM, open the sample file **GC14_Opticam_example_3.vnc**.

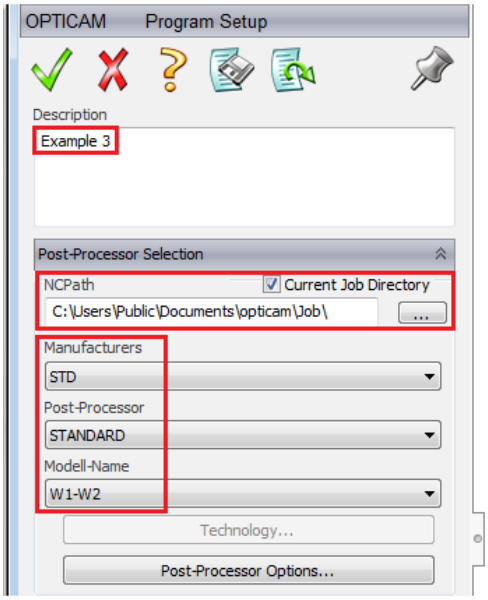


Using the Program Dialog for Basic Settings

As a first step, we define the basic settings again.

Double-click the branch **Program (STANDARD)** in the OPTICAM Feature Manager to open the dialog for the definition of the program default settings.

Fill in the dialog as shown to the right:

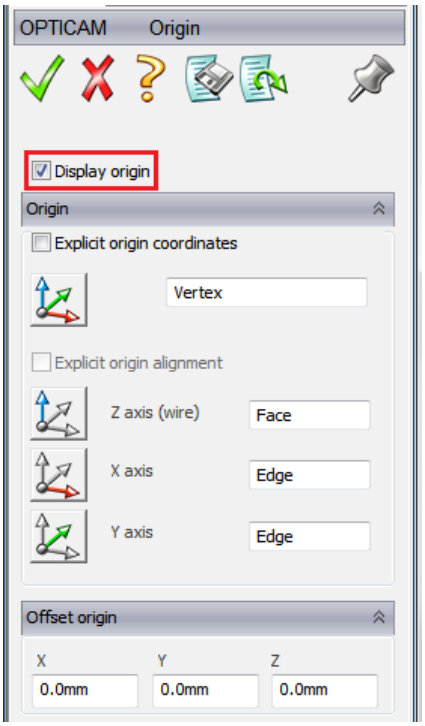


Accept changes by clicking  .

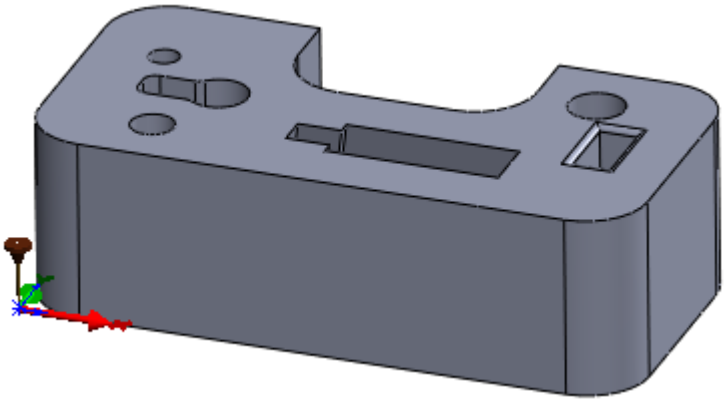
Positioning the Machine Origin in the Center of an Edge and Reversing the Direction of the Z-Axis

In the next step, we define the position of the machine origin which is to be placed in the center of the model's leading edge this time.

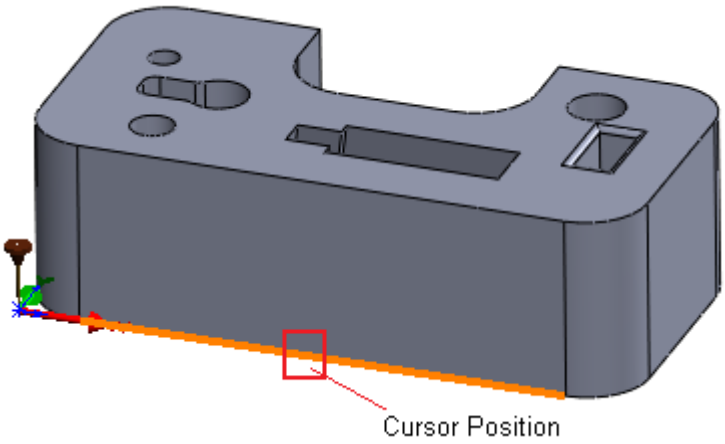
Double-click **(Program) Origin** to open the dialog to define the machine origin.



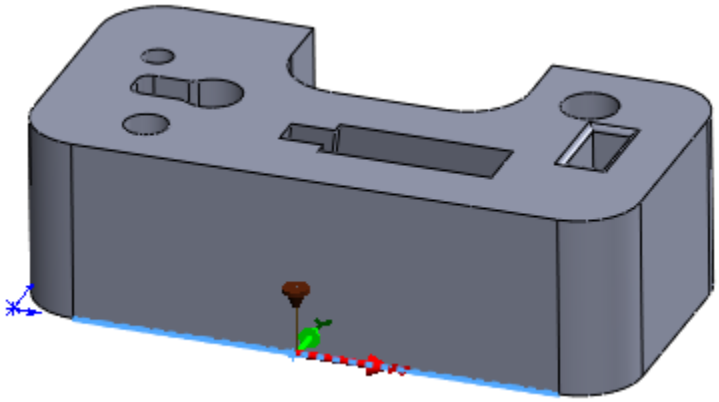
Currently, the machine origin is on the left side at the bottom of the workpiece (see following figure).



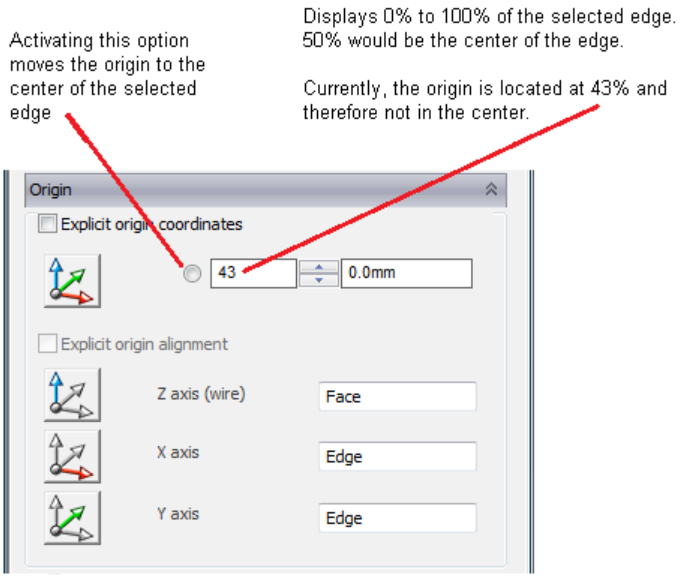
To position the origin in the center of the model's lower leading edge, position the cursor approximately in the center of the model's lower leading edge and click with the left mouse button.



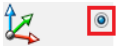
This moves the origin on the selected edge. However, it is unclear whether the new origin is exactly in the center of the edge.

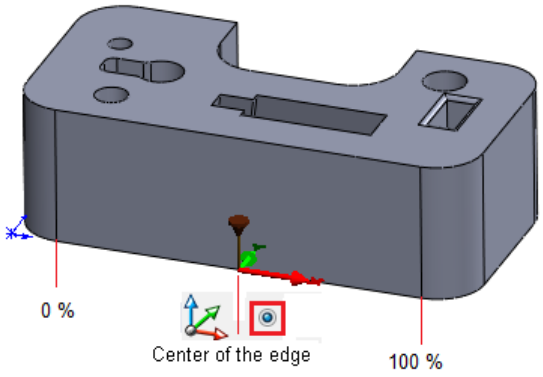


Looking at the dialog **Machine Origin** will help us (example).

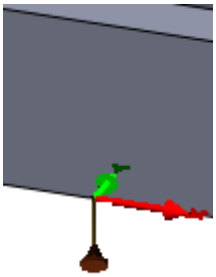


The origin is not exactly in the center of the selected edge, but 43% away from the origin of the edge. If you changed the value to 50%, the origin would be in the center of the edge. But it is easier to activate the option Origin  .

 This moves the origin exactly to the center of the edge.

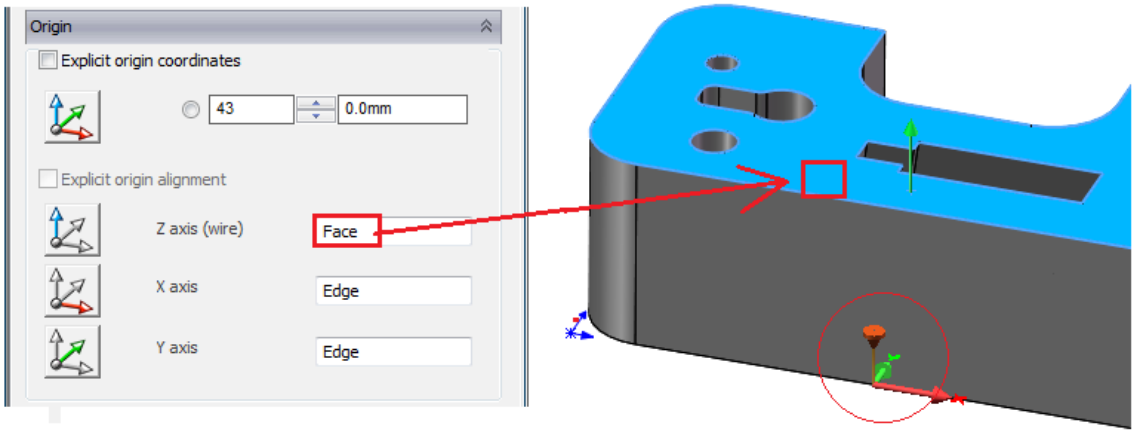


However, it may happen that the Z-axis of the origin points in the wrong direction.



But the nozzle icon along the Z-axis of the origin must point in the positive Z-direction.

To reverse the direction of the origin, position the cursor in the dialog box **Z axis (wire)**, and click on a plane face on which the Z-axis of the origin is to be aligned.



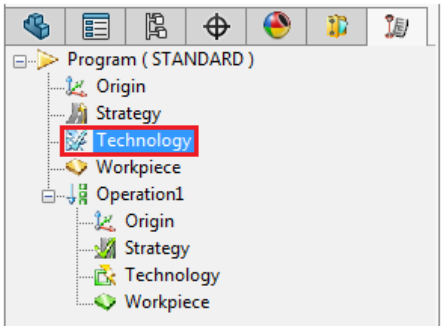
Thus, the direction of the Z-axis is reversed.

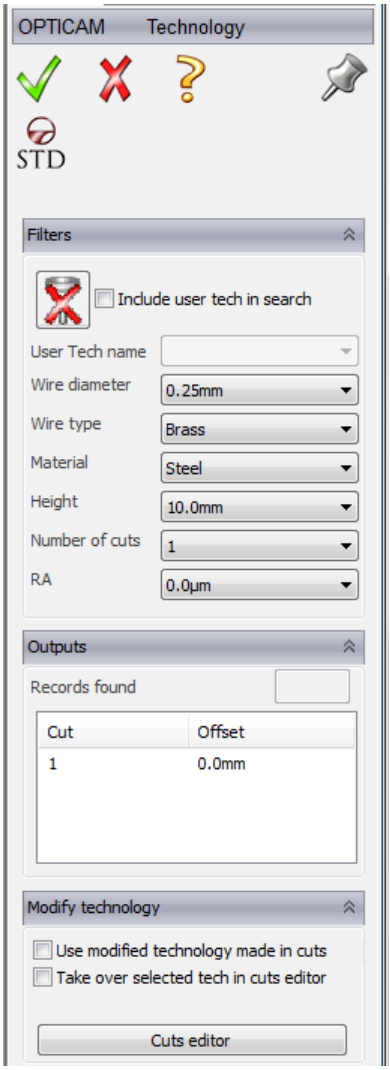
Close the dialog **Machine Origin** by clicking  .

Defining the Cutting Technology (from Database)

In the next step, we will define with which cutting technology the workpiece is be machined.

Double-click the branch **(Program) Technology** in the OPTICAM Feature Manager to open the dialog for the definition of the cutting technology.



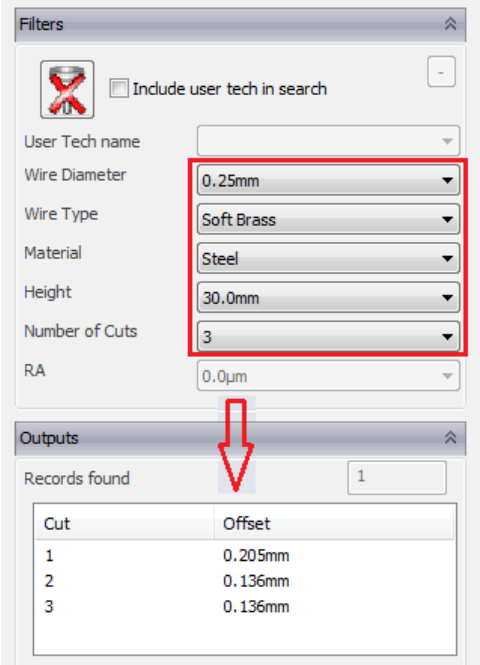


You can either select the parameters of the cutting technology from a database or enter them manually.

In the first few exercises, you have entered the cutting technology manually. However, in this example the technology is to be loaded from the Technology Database.

You can do this as follows:

Select suitable technology parameters from the listboxes **Wire Diameter**, **Wire Type**, **Material**, and **Height**, as follows:



In the dialog area **Records found** the selected cutting technology will be displayed.

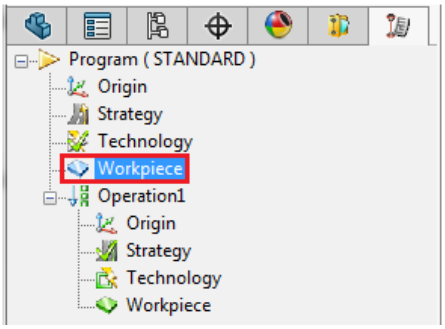
User advice: The input fields of the technology database vary according to the selected postprocessor and often only contain a few example technologies. For most wire EDM machines, complete technology databases are optionally available.

Accept the selection of the technology parameters by clicking .

Creating the Workpiece (Billet) Geometry

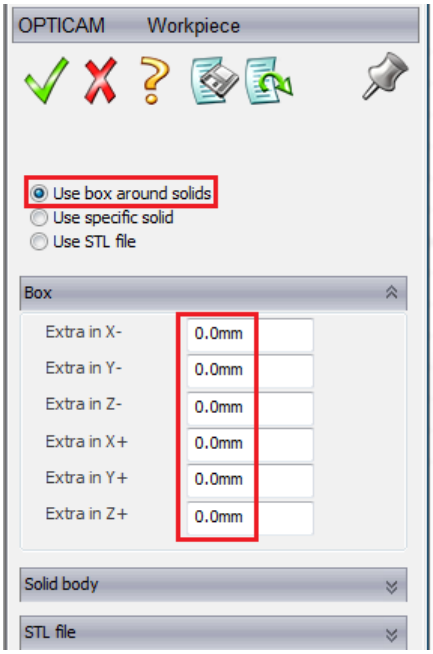
In the next dialog, you can define the geometry of the workpiece (billet).

Double-click the branch **(Program) Workpiece** in the OPTICAM Feature Manager.



The workpiece (billet) is to be derived from the solid body; additionally, no allowance values are to be defined.

Fill in the dialog as follows:

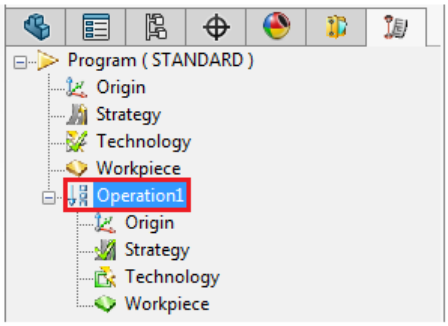


Accept changes by clicking  .

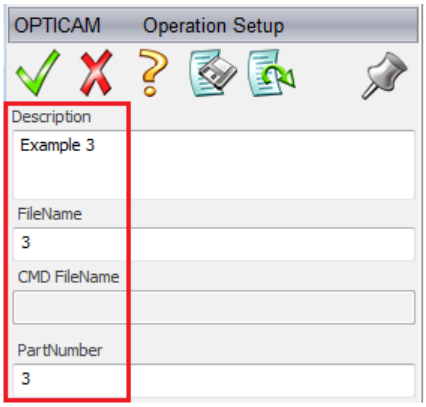
Setting the Cutting Heights (Reference Plane Height and Secondary Plane Height)

The following dialog sets among other things the name of the NC output file as well as the Z-height of the Reference and the Secondary Plane Height.

Double-click the branch **Operation1** in the OPTICAM Feature Manager.

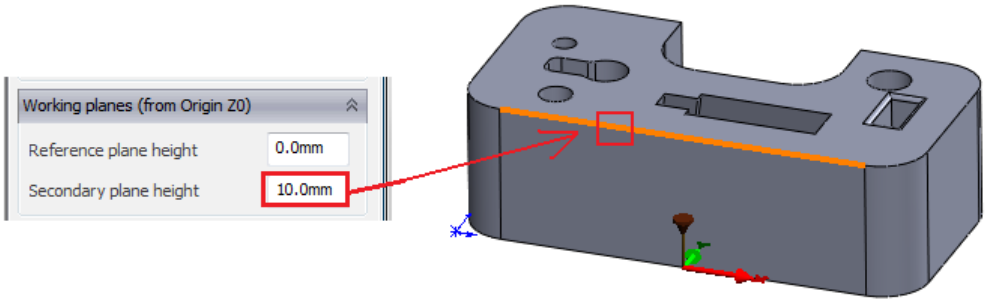


Fill in the fields **Description**, **FileName** and **PartNumber** as follows:

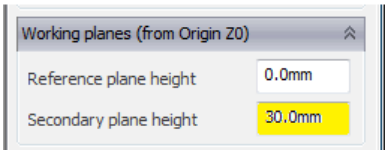


In this example, the **Secondary plane height** is to be identified in the model.

Position the cursor in the input field **Secondary plane height.**



Click on an **edge** (a **plain face** or a **vertex**) on the top level of the model. OPTICAM calculates the Z-height of the selected element and passes the value on to the dialog box **Secondary plane height**.

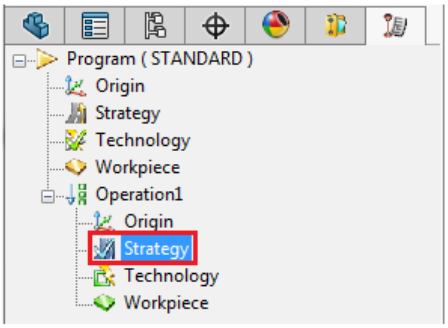


Accept changes by clicking  .

Setting the Cutting Strategy

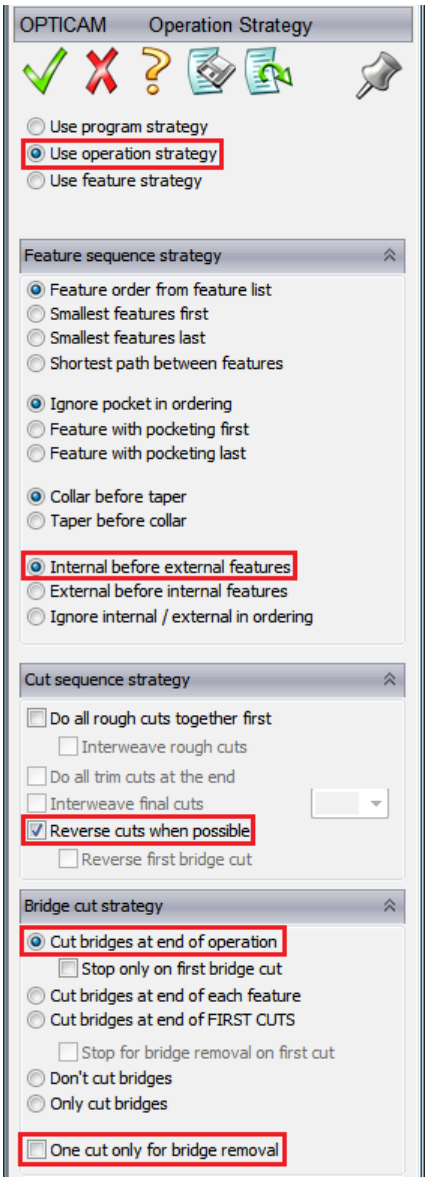
In the next dialog, you can define with which strategy the workpiece is to be machined.

Double-click the branch **(Operation) Strategy** in the OPTICAM Feature Manager.



The geometries are to be processed in a way that the **Bridges are Cut at the End of the Operation**. The main cut and the 2 trim cuts are to be done in **Reverse Cuts** (forwards/backwards).

Set the dialog as follows:



Accept changes by clicking  .

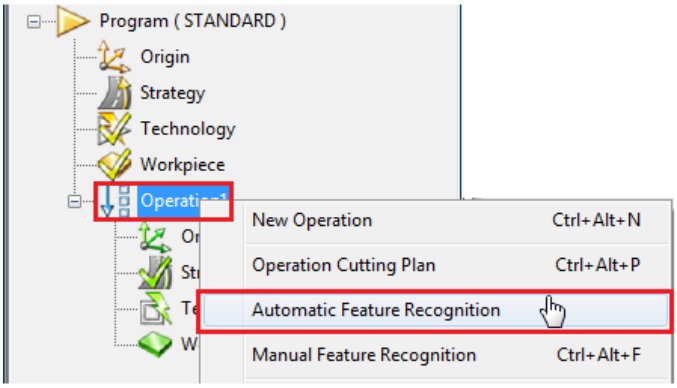
Creating the Cutting Machining

As you probably know from the first examples, with OPTICAM it is possible to create the cutting machining automatically or by manually selecting faces or model edges.

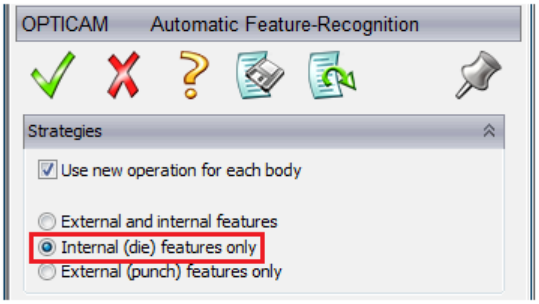
Because all internal apertures are to be machined in this example, the cutting machining will be automatically created by the system.

Automatic Creation of the Cutting Machining

Right-click the branch **Operation1** in the OPTICAM Feature Manager and, on the context menu, click **Automatic Feature Recognition**.

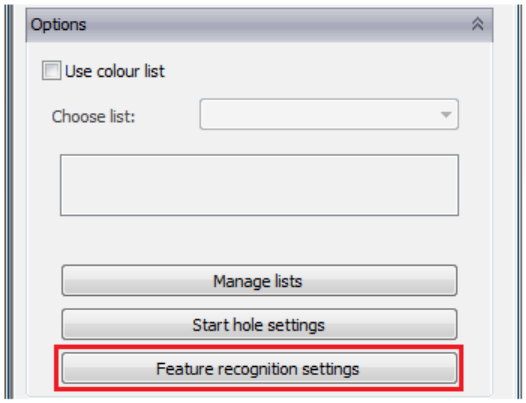


The dialog **Automatic Feature Recognition** opens. To machine only the internal apertures, set the dialog as follows:



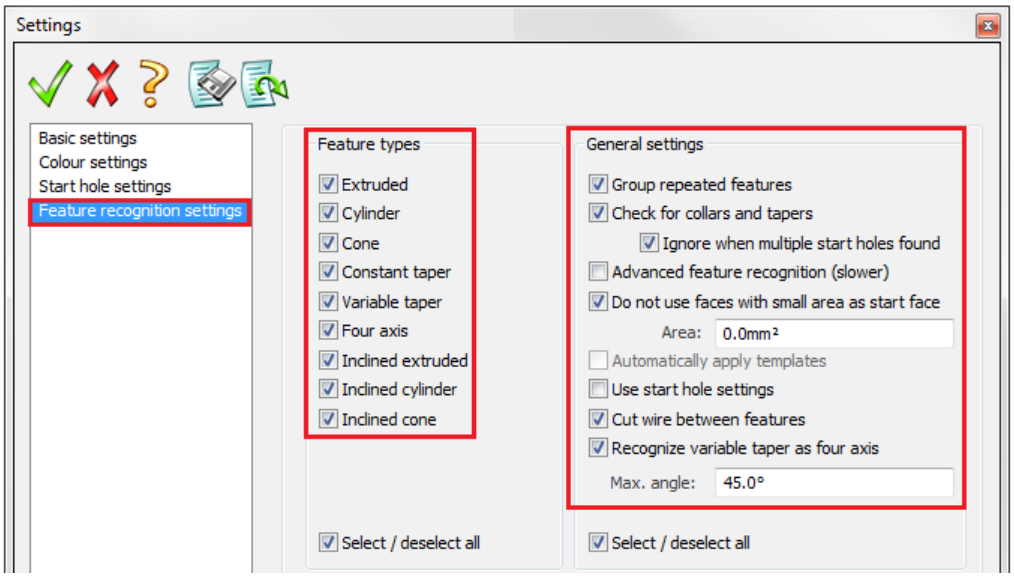
In order to set what exactly the **Feature Recognition** is supposed to recognize, click the 

button in the dialog.



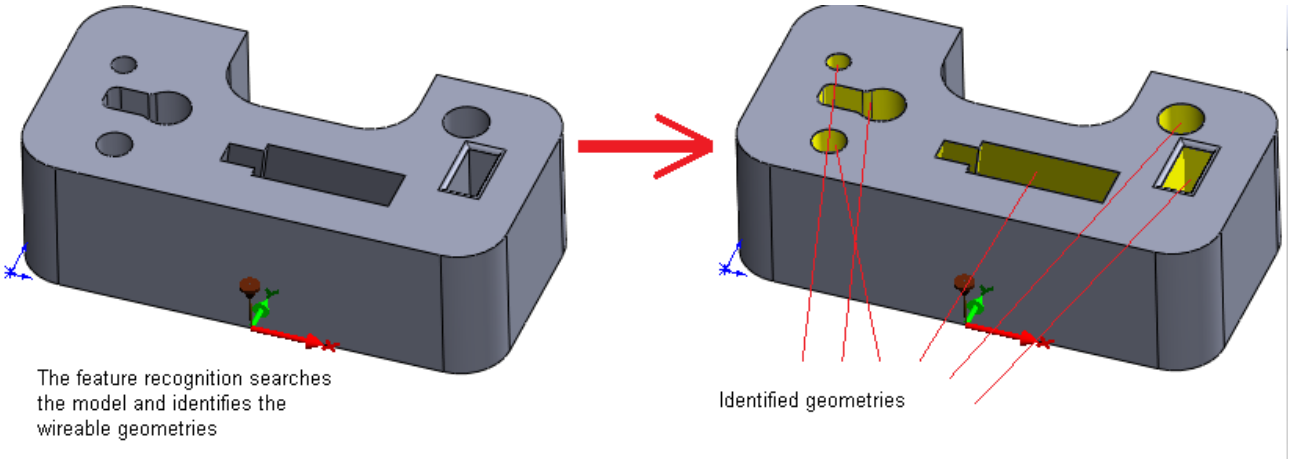
A dialog containing setting parameters for the **Feature Recognition** opens.

Make sure that the dialog looks as follows:

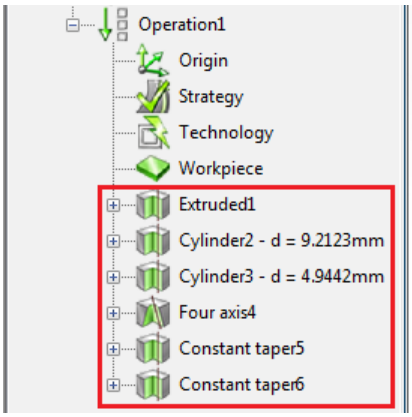


Accept changes by clicking  . Close the **Feature Recognition** dialog:  .

The system will now search the solid body for all geometries that can be wire cut (feature recognition) and create a machining suggestion with the cutting parameters previously set.



For each geometry that can be wire cut, OPTICAM creates a separate feature in the OPTICAM Feature Manager. As six geometries have been identified, there should also be six features:



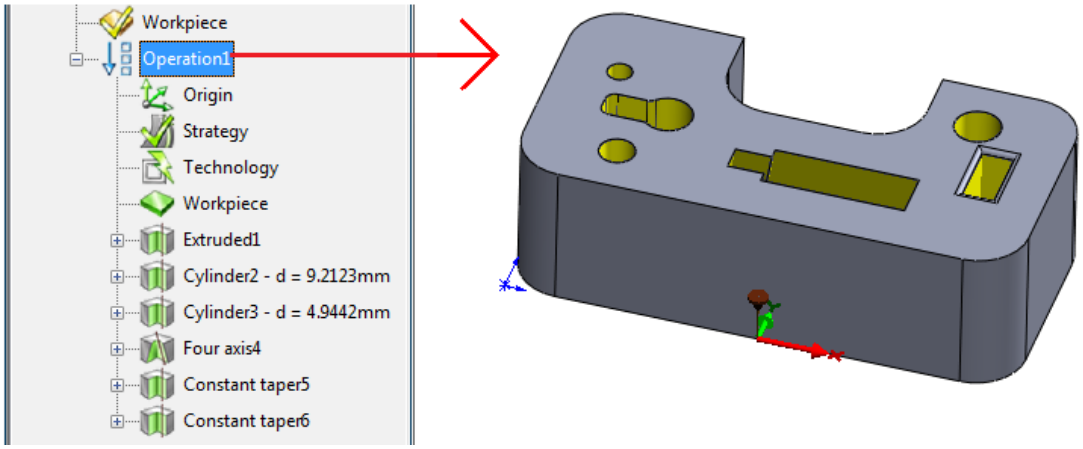
User advice: Please note that OPTICAM distinguishes between the identified geometries and divides them into categories:

- **Extruded** = Cylindrical cut on any feature
- **Cylinder** = Cylindrical round hole
- **Cone** = Tapered round hole
- **Constant Taper** = Constant taper (can be output to the NC data as a taper)
- **Variable Taper** = Taper changes within the feature (can be output as a variable taper)
- **Four Axis** = Upper and lower plain are different – (NC output as a four axis)
- **Inclined Extruded**, **Cylinder** and **Cone** = Features, type **Extruded**, **Cylinder** and **Cone**, introduced into the model diagonally and at an angle.

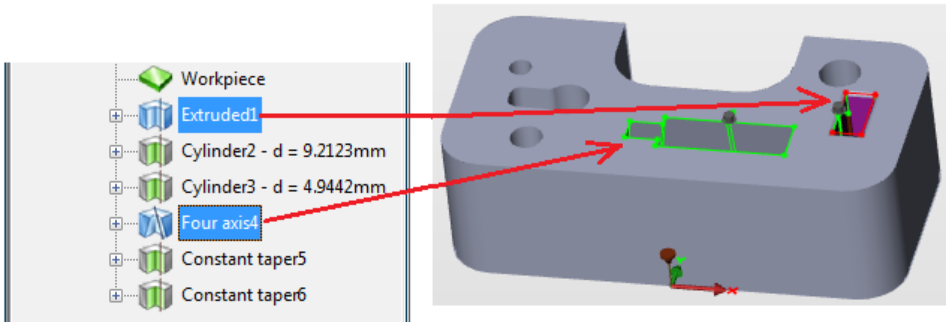
The geometry differentiation is very important for the NC output as the necessary NC output is automatically identified.

Remember:

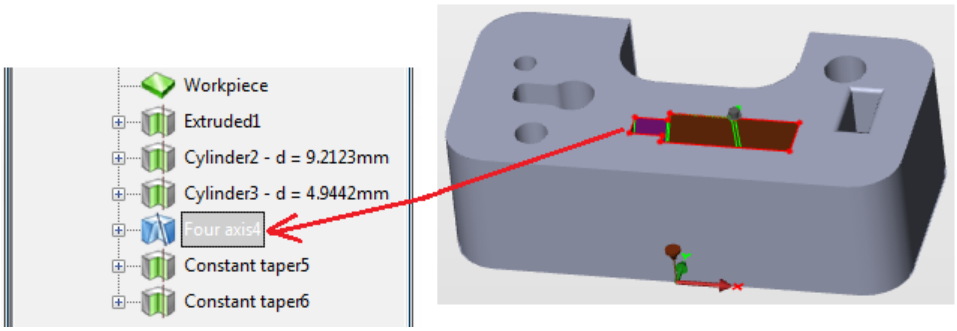
If you click the branch **Operation1** in the OPTICAM Feature Manager**,** the geometries of all existing features will be highlighted.



If you click on one or various features in the OPTICAM Feature Manager, OPTICAM will highlight the geometries of the selected feature(s) and additionally display the tool paths, the bridge, and the start hole.



However, it also works the other way round: If you click on a feature in the solid model, the corresponding entry will be highlighted in the OPTICAM Feature Manager.

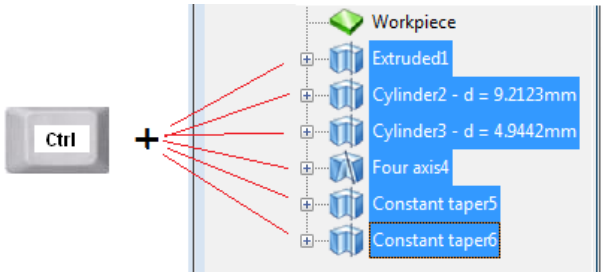


Setting the Start Hole Position and the Length of the Lead off

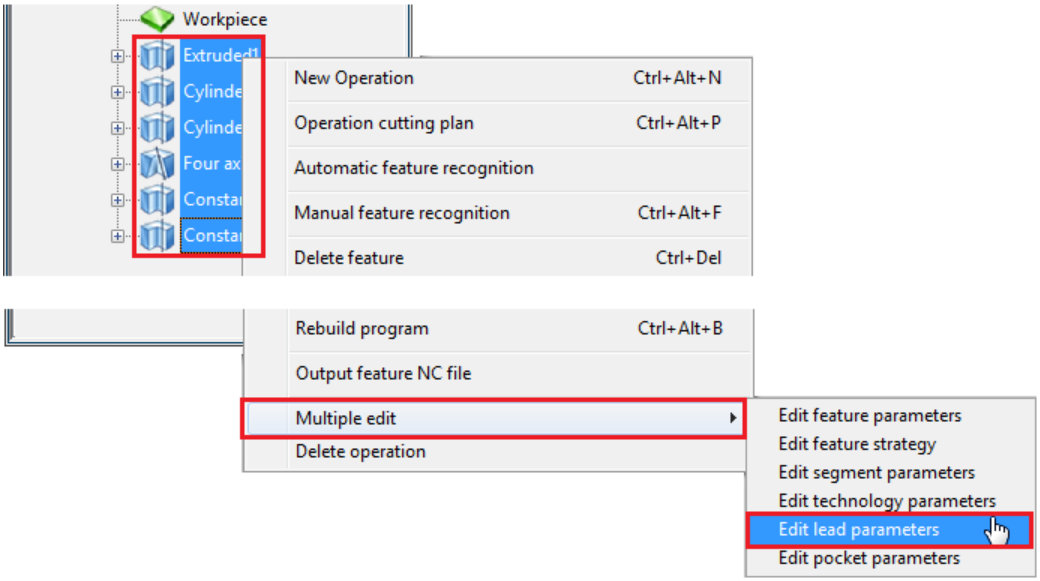
Using the OPTICAM Feature Manager, you can influence and modify all cutting parameters. In our case, we want to change the lead on / lead off parameters and the position of the start hole and shorten the lead off.

For each aperture, OPTICAM creates a separate feature in the OPTICAM Feature Manager.

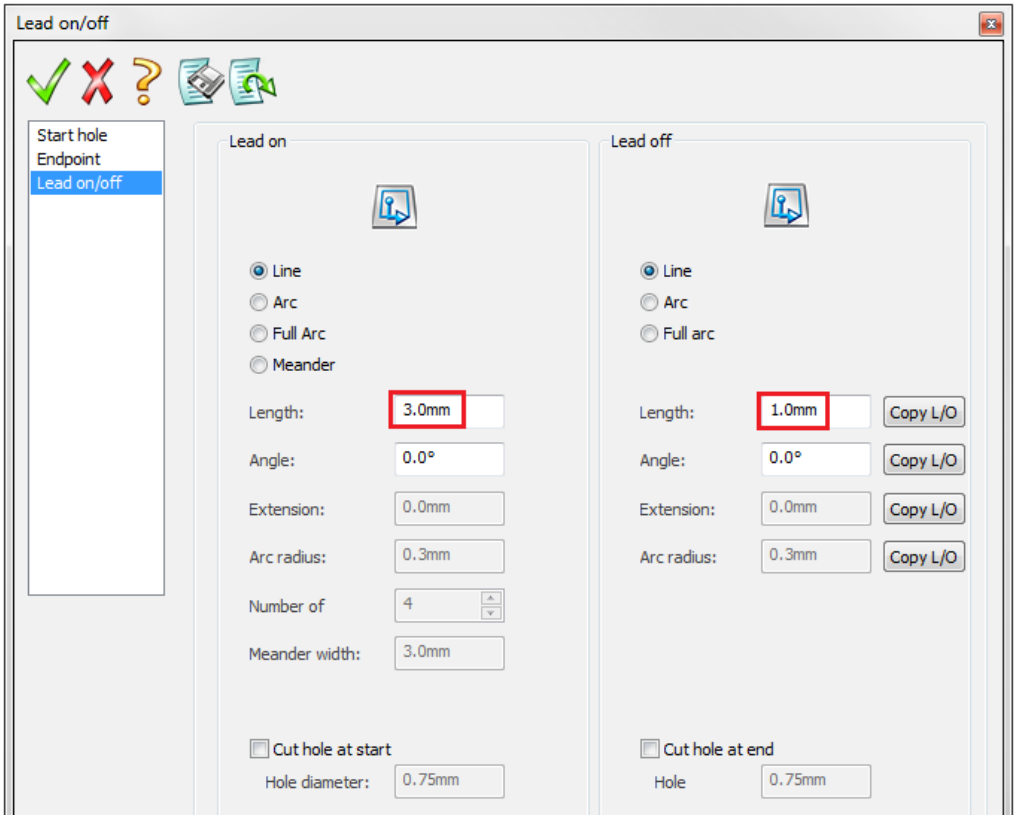
Holding down the **Ctrl** key, and click successively each of the six features:



Now right-click one of the six features and, on the context menu, click **Multiple edit > Edit lead parameters** to open the **Lead on / off** dialog.



On the left-hand side, click the feature **Lead on / off** and modify the length of the **Lead On** to **3** mm and of the **Lead Off** to **1** mm.

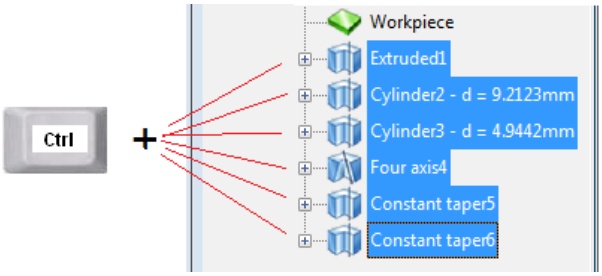


Accept changes by clicking  . The modification applies to all selected features.

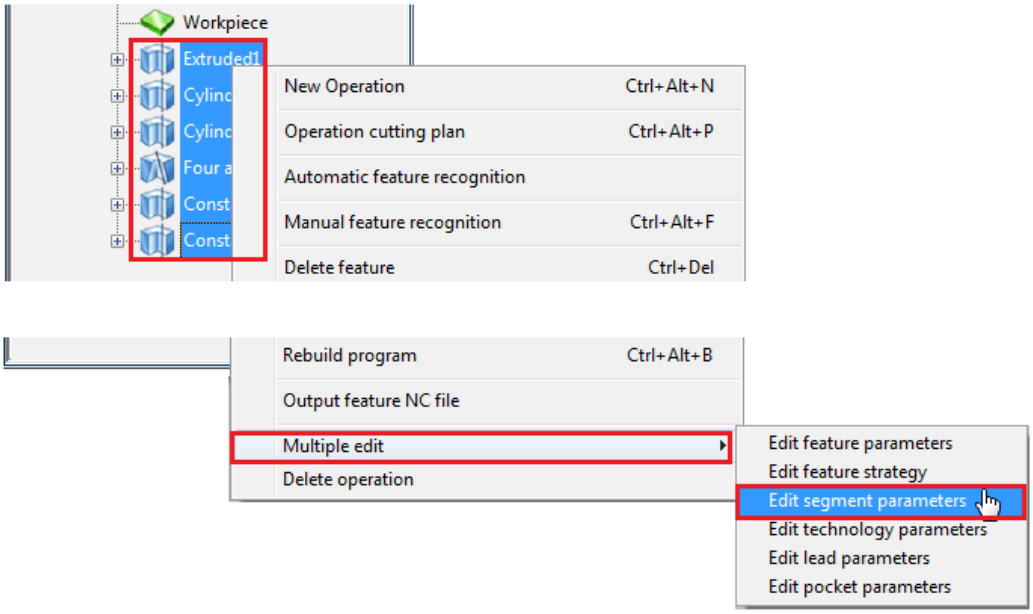
Adjusting the Size of the Bridge

Now we want to check the size of the bridge and adjust it, if necessary.

Holding down the **Ctrl** key, successively click each of the six features in the OPTICAM Feature Manager.

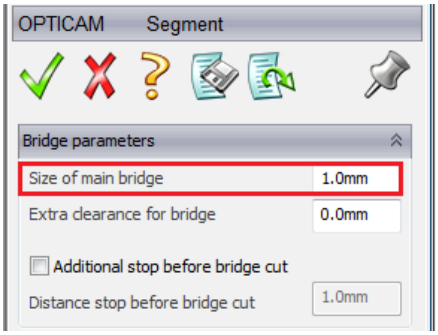


Now right-click one of the six features and, on the context menu, click **Multiple edit > Edit segment parameters**.



The dialog **Segment** opens.

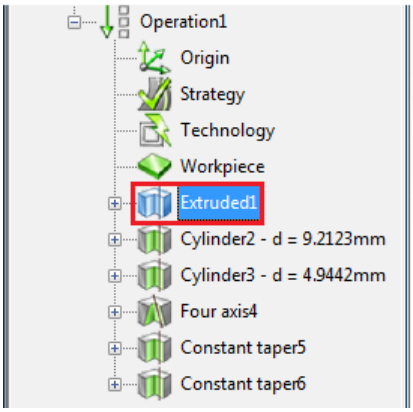
If necessary, change the **Size of main bridge** to **1.0** mm.



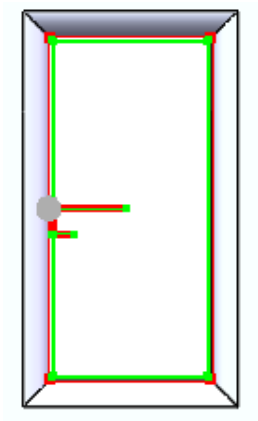
Checking the Changes

Since you have changed the **lead on / lead off** and the **size of the bridge**, you should check these changes graphically.

Click the feature **Extruded1** in the OPTICAM Feature Manager.

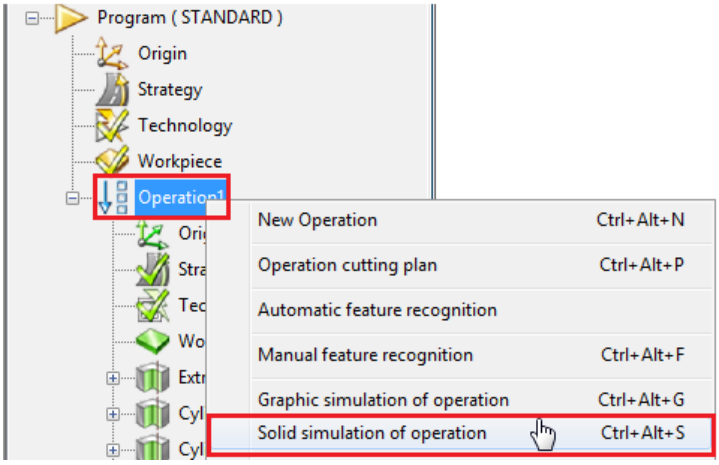


The figure below shows how the tool path graphic of the selected feature is highlighted in the solid body:

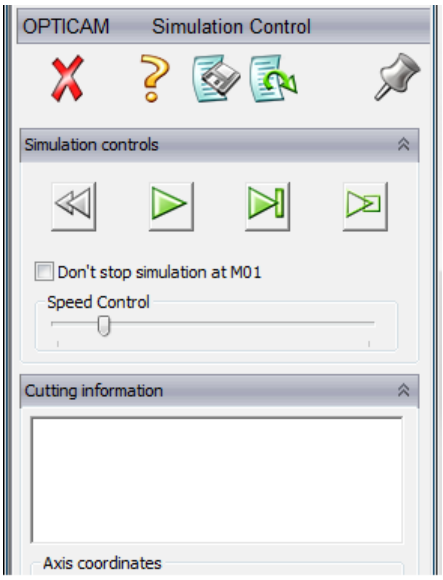


Machining Simulation (Solid Simulation)

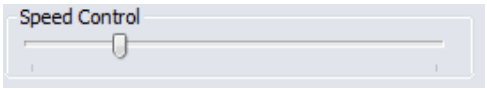
Right-click the branch **Operation1** and, on the context menu, click **Solid simulation of operation**.



A dialog opens that allows you to control the solid simulation:



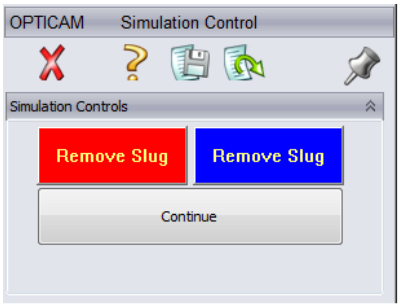
Click  to start the simulation. Use **Speed Control** to control the simulation speed.

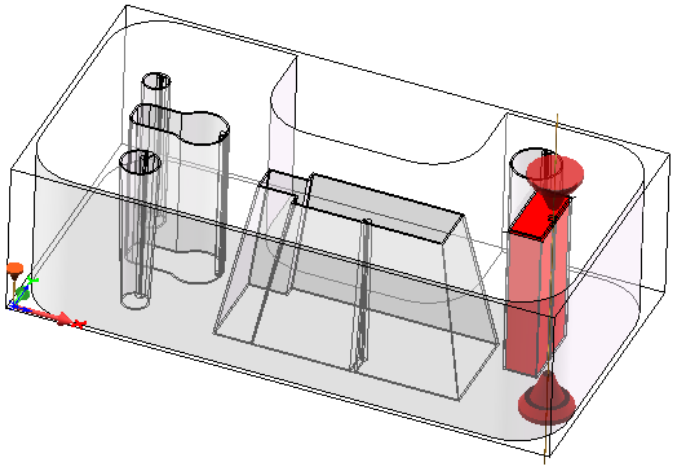


The simulation type **"Solid Simulation"** clearly depicts the material removal and the position of the start holes.

Depiction of Slugs during the Simulation

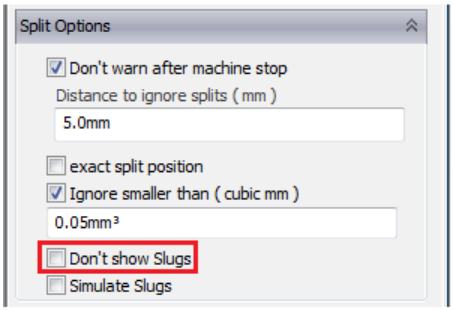
If the bridge is cut, normally a slug will be created. The solid simulation can visualize this slug and displays the button **Remove Slug**. The slug will be displayed in red.



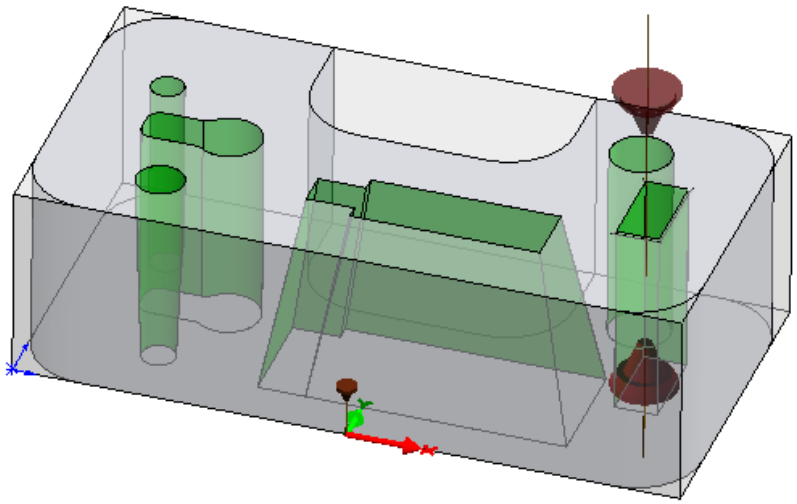


User advice: In the field **Split Options** in the simulation dialog you can set if a slug is to be displayed or not.

If you want to see the slug, you have to DEACTIVATE the option **Don't show Slugs**.



The following figure shows the complete program.



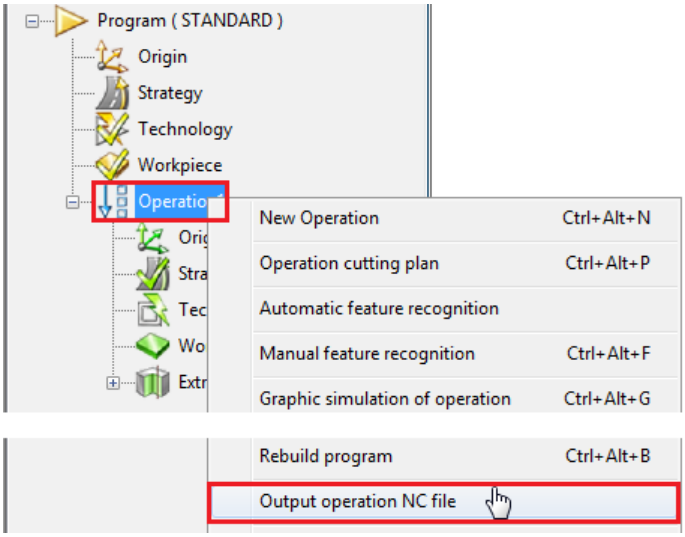
Tip: With the  button you can reset the simulation and then restart it.

Close the simulation dialog by clicking  button.

Creating the NC program

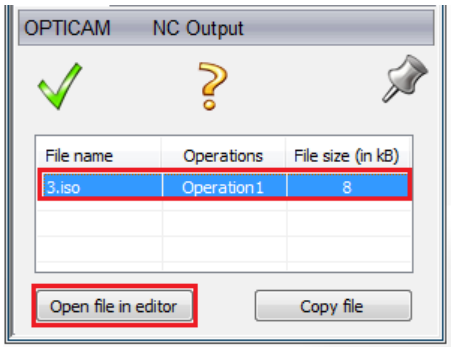
At the end of this exercise, we create the NC program.

Right-click the branch **Operation1** in the OPTICAM Feature Manager and select the command **Output operation NC file**.

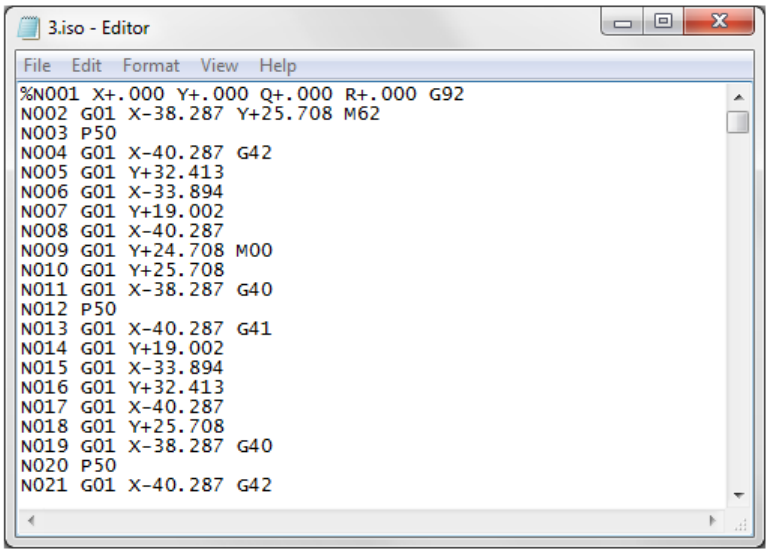


In the background, the NC program for the selected machine is created. Afterwards, a dialog for the editing of the NC file will be opened in the OPTICAM Feature Manager.

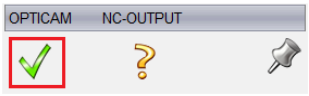
Click the NC file name and then the Open file in editor button.



A text editor opens which displays the just created NC program:



Close the editor  .

Close the dialog **NC-OUTPUT** as well 

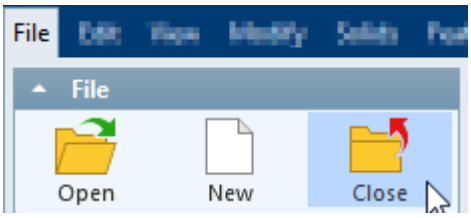
.

Saving the OPTICAM Program

At the end, you should save your work.

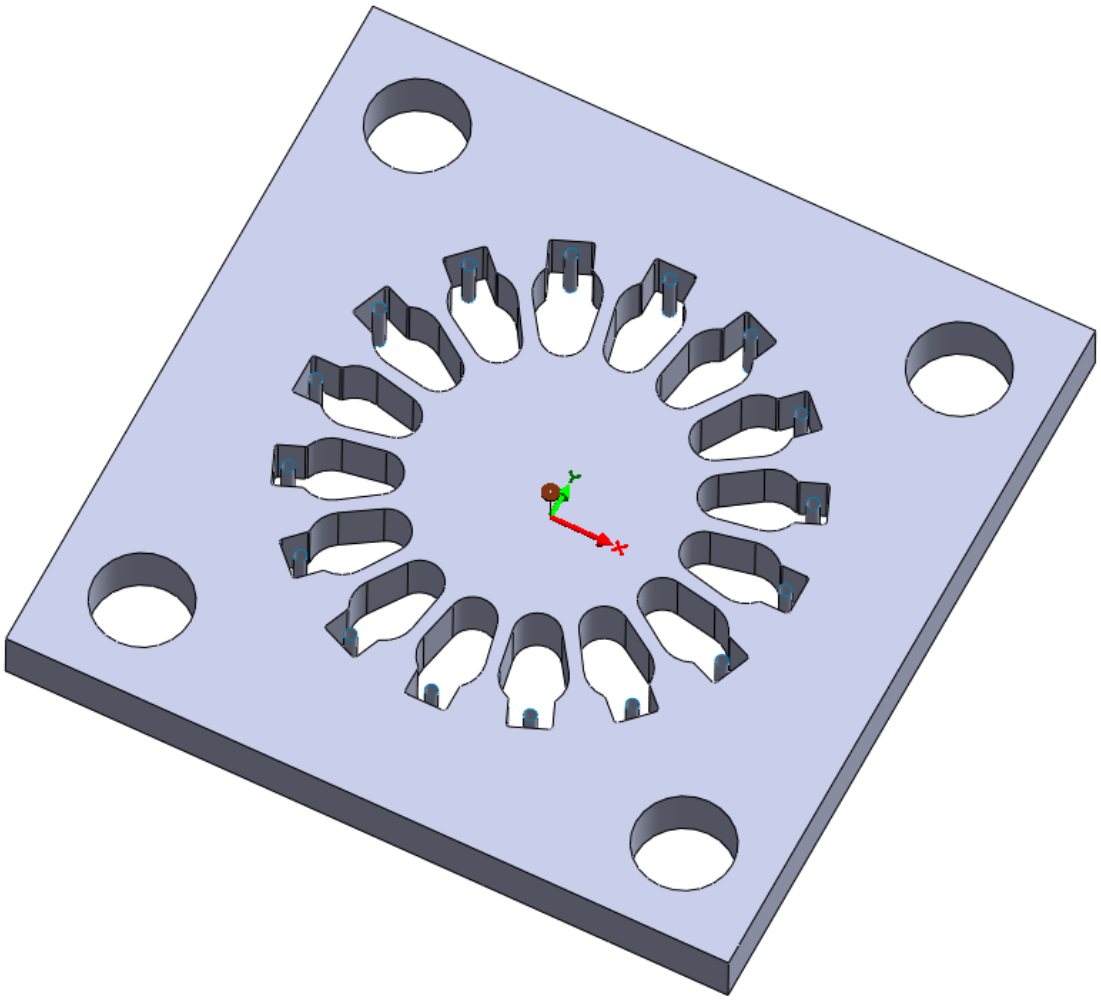
User advice: In addition to the GibbsCAM file **GC14_Opticam_example_3.vnc**, a second file containing the OPTICAM machining information for the current program will be saved. This OPTICAM file has the file extension *.**wire.opticam**.

Close the file **GC14_Opticam_example_3.vnc**.



This exercise is now completed.

Example 4: Automatic Start Hole Recognition



Demonstrated in This Example

Opening a file: GC14_Opticam_example_4

Defining the Machine Origin

Defining the Cutting Technology (Technology from Database)

Setting the Cutting Strategy

Automatic Creation of the Cutting Machining through Feature Recognition

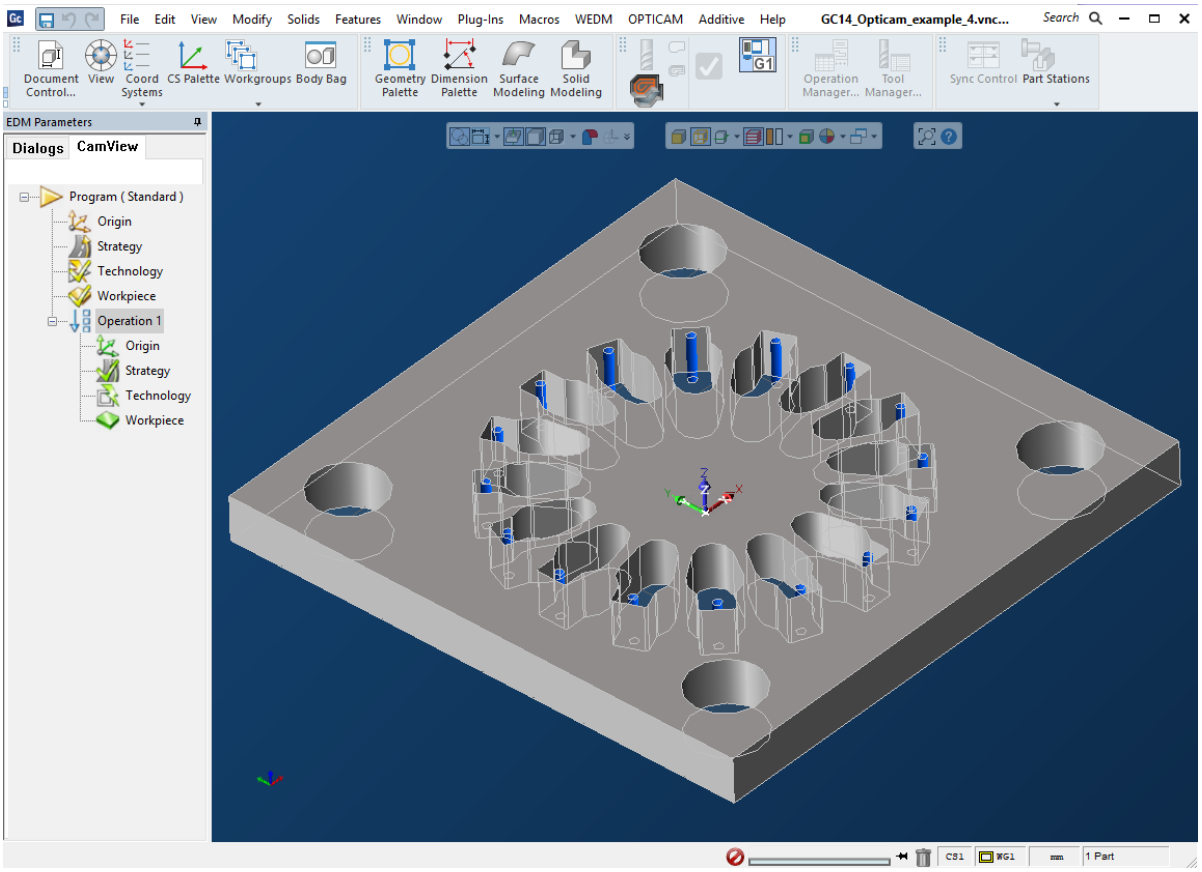
Automatic Start Hole Recognition

Grouping Repeated Features

Adjusting the Length of the Lead off

Open GC14_Opticam_example_4

In GibbsCAM, open the example file **GC14_Opticam_example_4.vnc**.

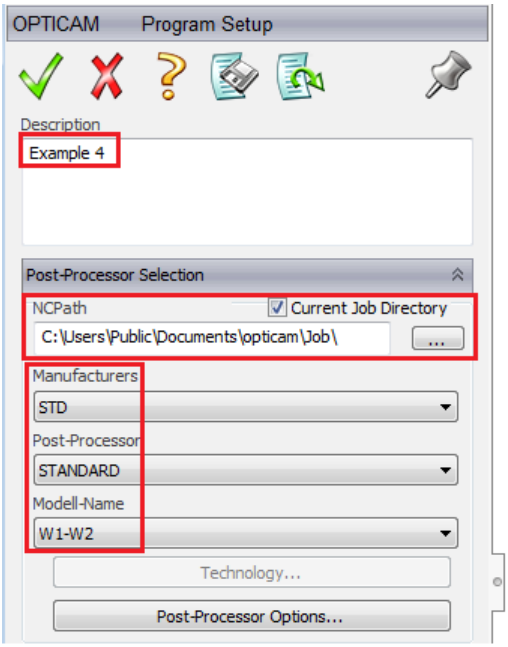


Using the Program Dialog for Basic Settings

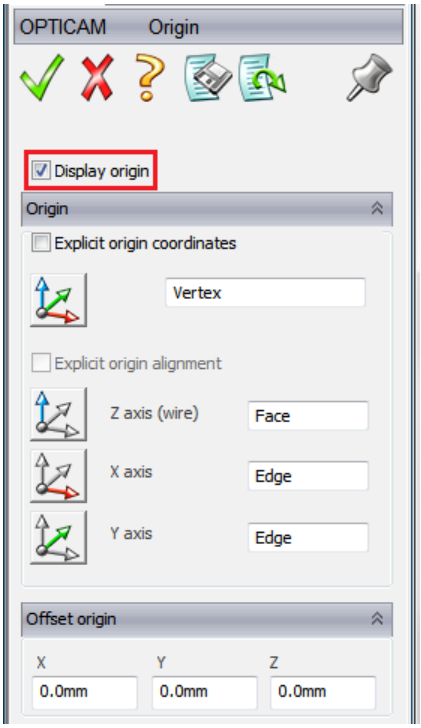
As before, double-click the branch **Program (STANDARD)** to open the dialog for the definition of the program default settings.

Fill in the dialog as shown to the right:

Accept changes by clicking  .



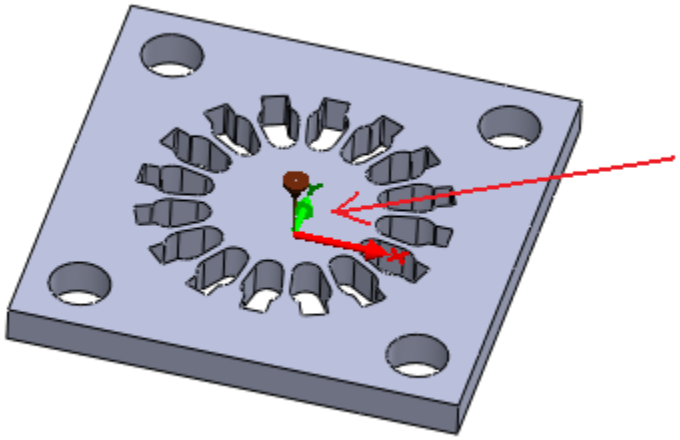
Defining the Machine Origin



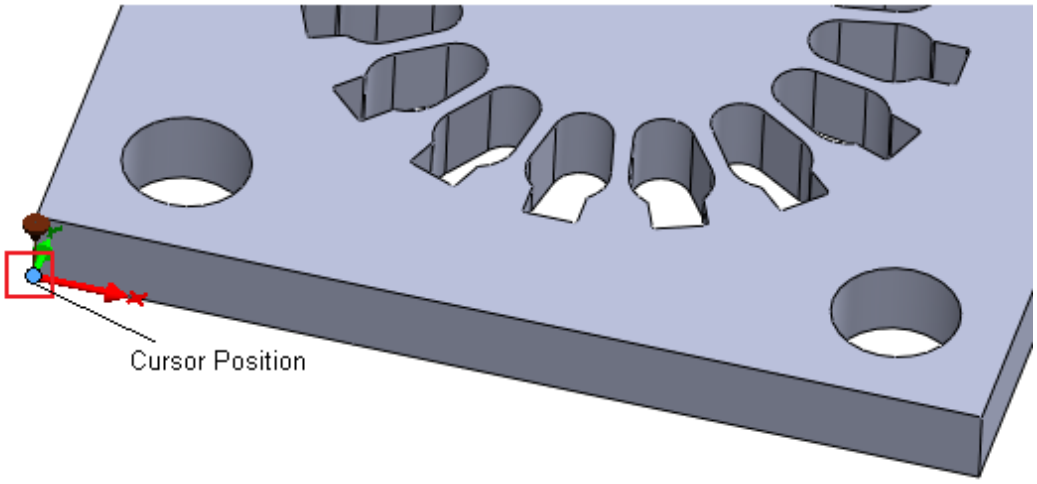
In the next step, the position of the machine origin will be defined. In this example, the origin will be placed at the lower left corner of the model.

Double-click the branch **(Program) Origin** to open the dialog for the definition of the machine origin.

Currently, the machine origin is in the center of the workpiece.



To move the origin to the lower left corner of the model, position the cursor at the lower left corner of the model and click with the left mouse button.



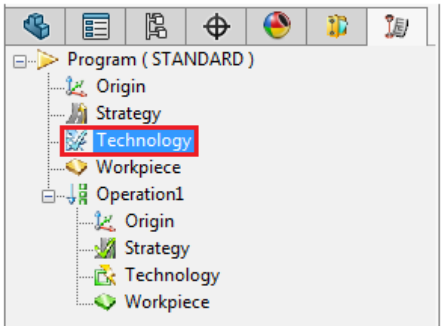
This moves the origin to the selected corner.

Close the **Machine Origin** dialog by clicking  .

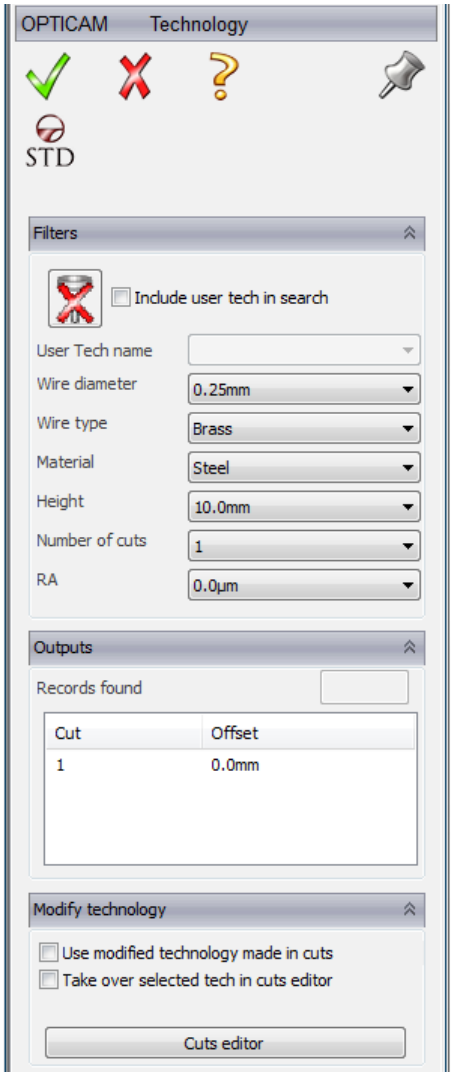
Defining the Cutting Technology (from Database)

In the next step, we will define with which cutting technology the workpiece is to be machined.

Double-click the branch **(Program) Technology** in the OPTICAM Feature Manager.

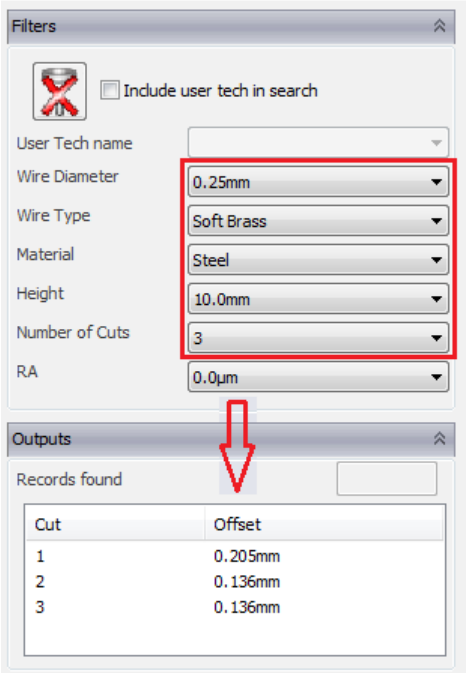


This opens the dialog for the definition of the cutting technology:



You can either select parameters of the cutting technology from a database or enter them manually. As in the last exercise, we want to load the technology from the Technology Database.

Select suitable technology parameters from the listboxes **Wire Diameter**, **Wire Type**, **Material** and **Height**.



In the dialog area **Records found** the selected technology will be displayed.

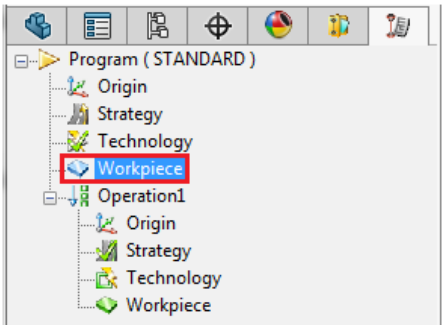
User advice: The input fields of the technology database vary according to the selected postprocessor and often only contain a few example technologies. For most wire EDM machines, complete technology databases are **optionally** available.

Accept the selection of the technology parameters by clicking .

Creating the Workpiece (Billet) Geometry

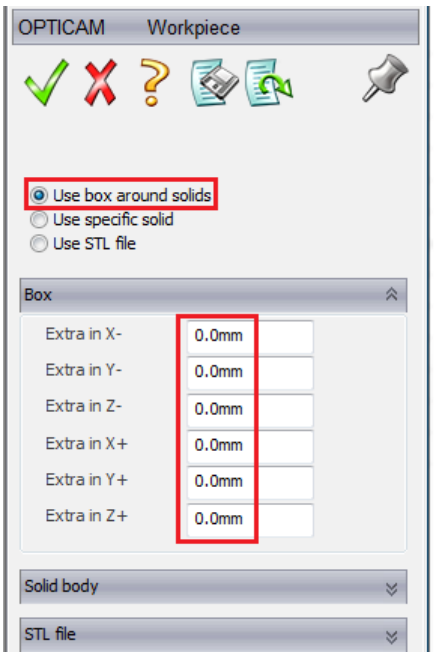
In the next dialog, you can define the geometry of the workpiece (billet).

Double-click the branch **(Program) Workpiece** in the OPTICAM Feature Manager.



The workpiece (billet) is to be derived from the solid body; additionally, no allowance values are to be defined.

Fill in the dialog as follows:

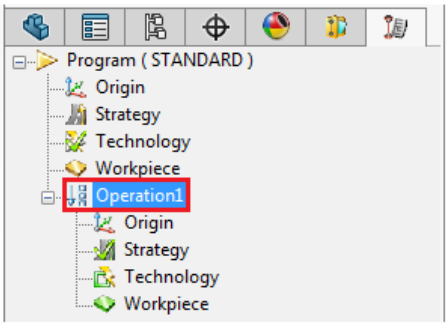


Accept changes by clicking  .

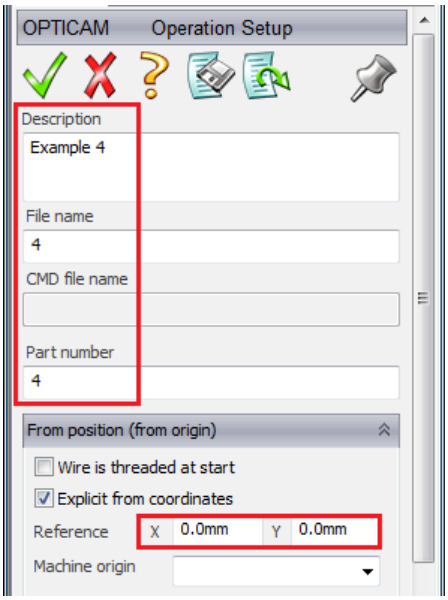
Setting the Cutting Heights (Reference Plane Height and Secondary Plane Height)

The following dialog sets (among other things) the name of the NC output file and the Z-height of the Reference and the Secondary Plane Height.

Double-click the branch **Operation1** in the OPTICAM Feature Manager.

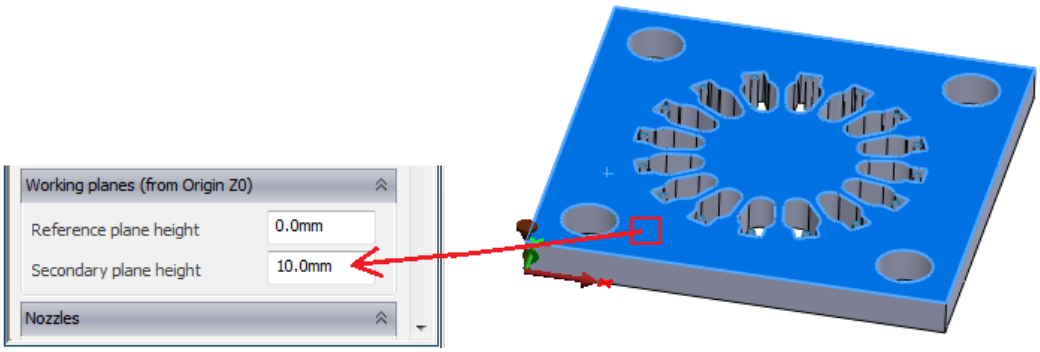


Fill in the fields **Description**, **File name**, **Part number** and **Position** as follows:

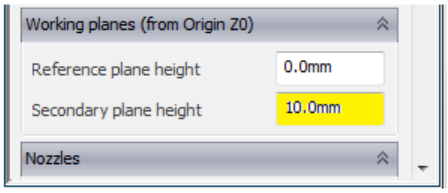


Identify the Secondary plane height in the model. You can do this by selecting an element (an edge, plain face or a vertex) which is located at the desired height.

Position the cursor in the input field **Secondary plane height**.



Click an edge or a vertex on the top level of the model. OPTICAM calculates the Z-height of the selected element and passes the value to the dialog box **Secondary plane height.**

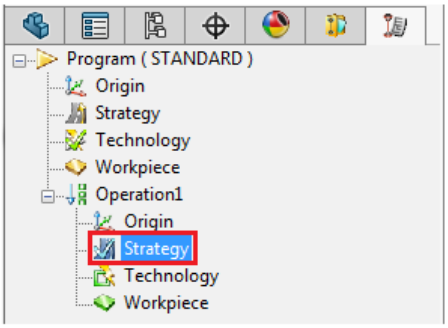


Accept changes by clicking  .

Setting the Cutting Strategy

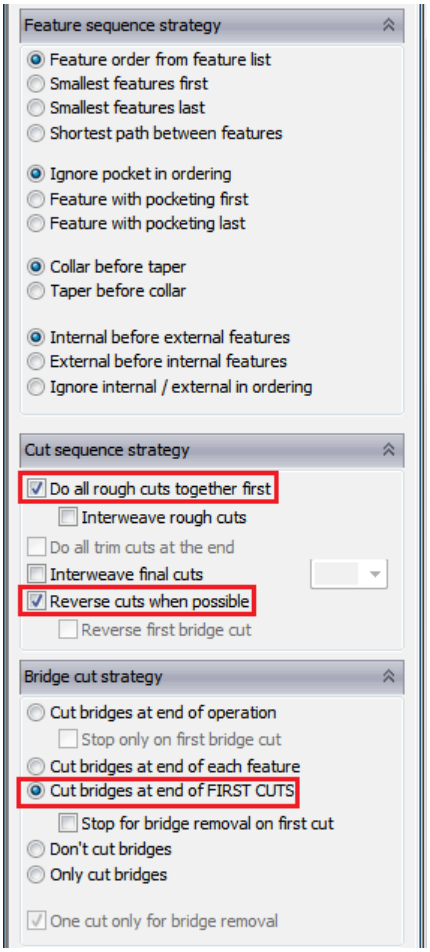
In the next dialog, you can define with which strategy the workpiece is to be machined.

Double-click the branch **(Operation) Strategy** in the OPTICAM Feature Manager.



The geometries are to be processed in a way that All Rough Cuts Are Done together first and then all Bridges Are Cut. In the end, the two Trim Cuts are to follow in Reverse Cuts (forwards/backwards).

Set the dialog as follows:



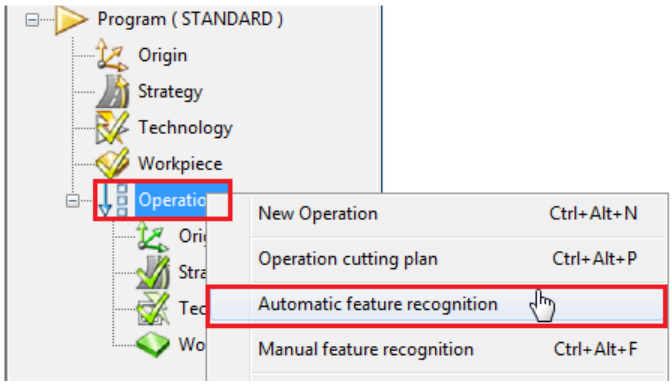
Accept changes by clicking  .

Creating the Cutting Machining

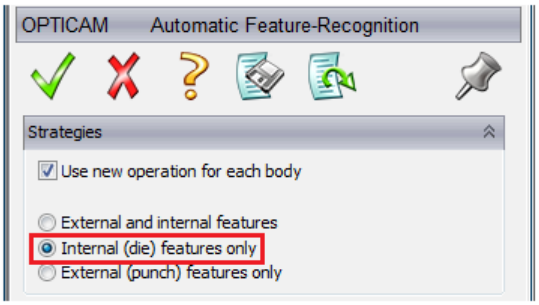
With OPTICAM it is possible to create the cutting machining automatically or by manually selecting faces or model edges. In this example, all internal apertures are to be machined. The cutting machining is to be automatically created by the system.

Automatic Creation of the Cutting Machining

Right-click the branch **Operation1** in the OPTICAM Feature Manager and, on the context menu, click **Automatic feature recognition**.

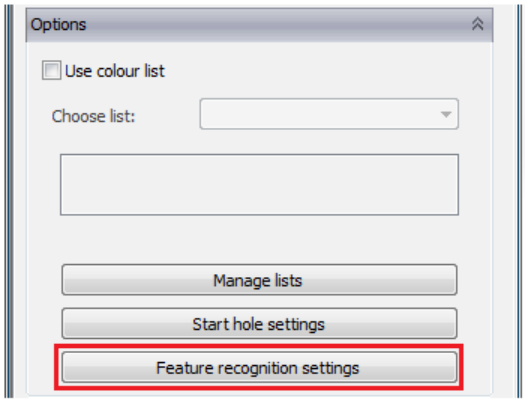


The dialog **Automatic Feature Recognition** will be displayed. To machine **exclusively the internal apertures**, set the dialog as follows:



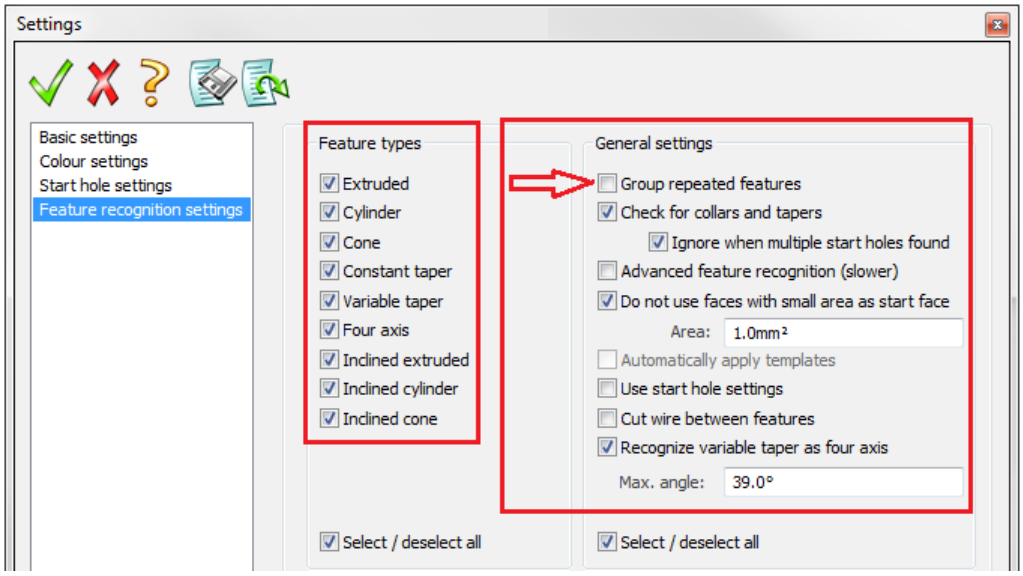
In order to set what exactly the **Feature Recognition** is supposed to recognize, click the 

button in the dialog.



A dialog containing setting parameters for the **Feature Recognition** will open.

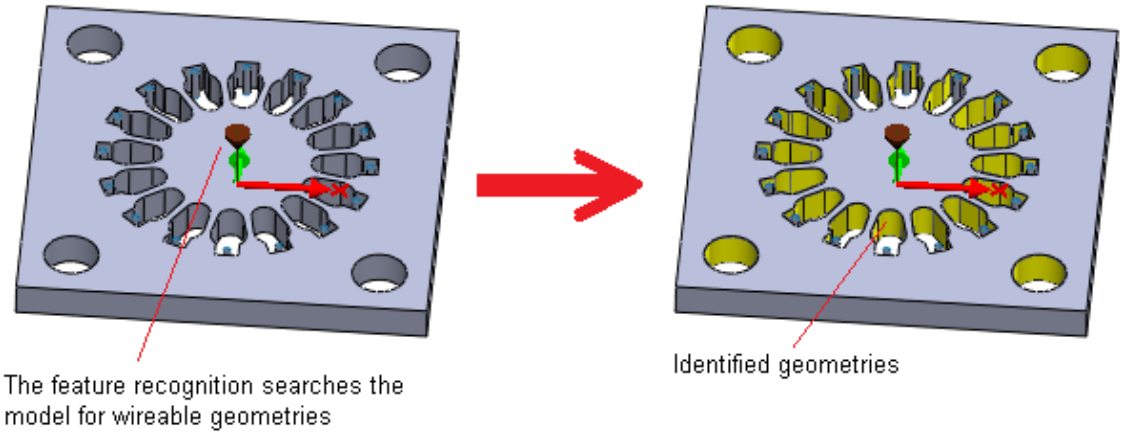
Make sure that the dialog looks as follows:



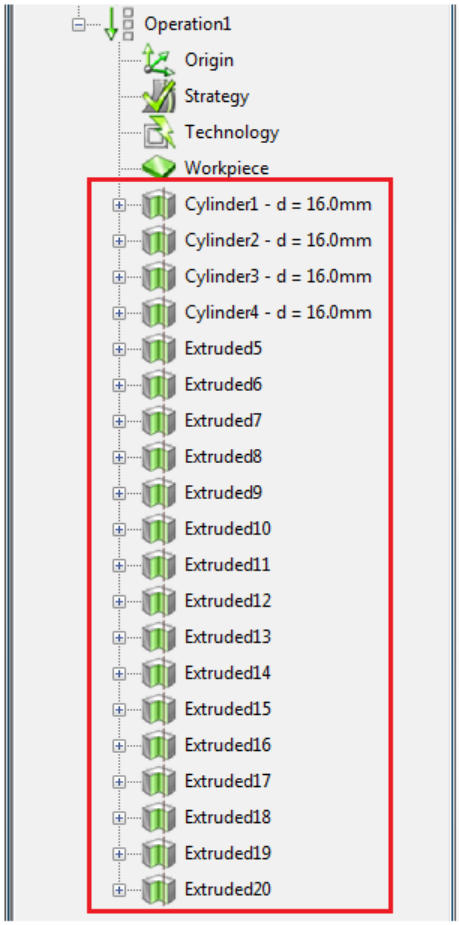
User advice: Do **NOT** activate the option **"Group repeated features"**.

Accept changes by clicking . Close the **Feature Recognition** dialog: .

The system will now search the solid body for all geometries that can be wire cut (feature recognition) and create a machining suggestion with the cutting parameters previously set.



For each aperture that can be wire cut, OPTICAM creates a separate feature in the OPTICAM Feature Manager.



User advice: Please note that OPTICAM distinguishes between the identified geometries and divides them into categories:

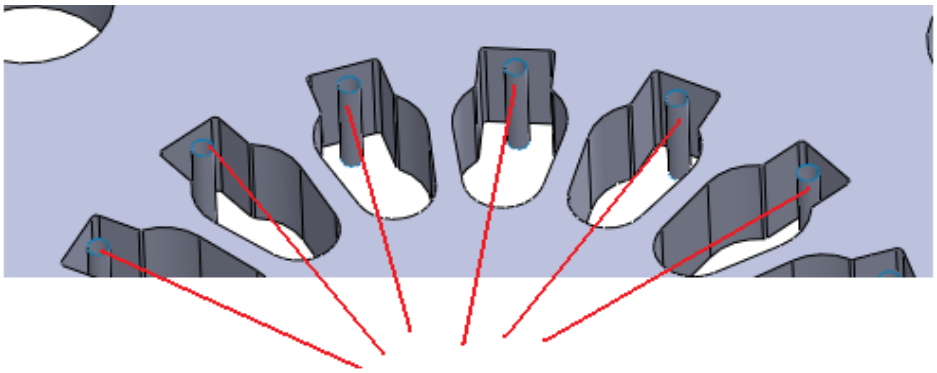
In this case, OPTICAM has identified and machined 16 cylindrical apertures (Extruded 5-20) and 4 cylindrical holes (Cylinder 1-4).

User advice: If only two features exist, you have probably activated the option **“Group repeated features”** in the **Feature Recognition** dialog, which unites identical features into one feature.

Automatic Start Hole Recognition

OPTICAM feature recognition is able to identify start holes in a 3D model and to use them as threading position for the cutting machining.

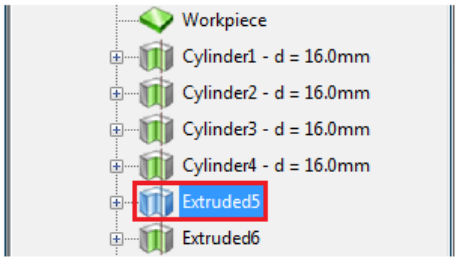
To take advantage of automatic start hole recognition, the start holes in the 3D model have to be drawn as “surface cylinders” (in GibbsCAM terminology, “Extruded Circles”). This is the case in this example.



Start holes, drawn as "cylinders"

Adjusting the Lead on/off

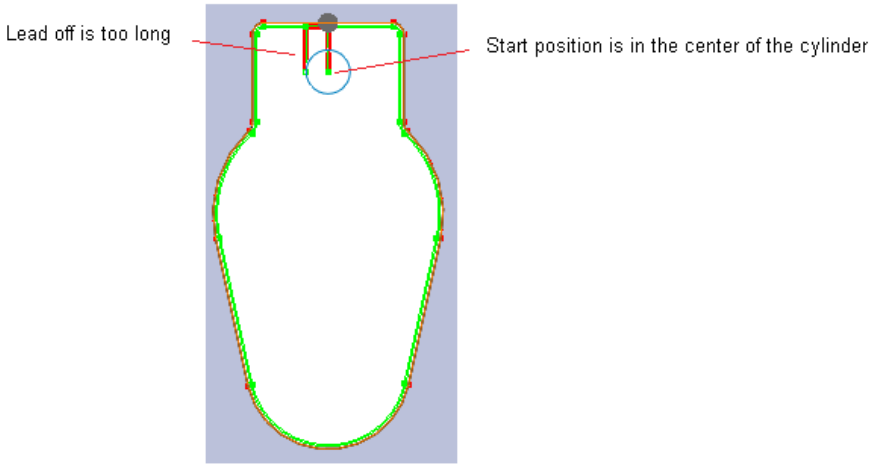
Click the feature **Extruded5** in the OPTICAM Feature Manager.



The tool paths of the selected feature are highlighted in the model.

If you extend the tool paths and look at them more closely, you will note several things:

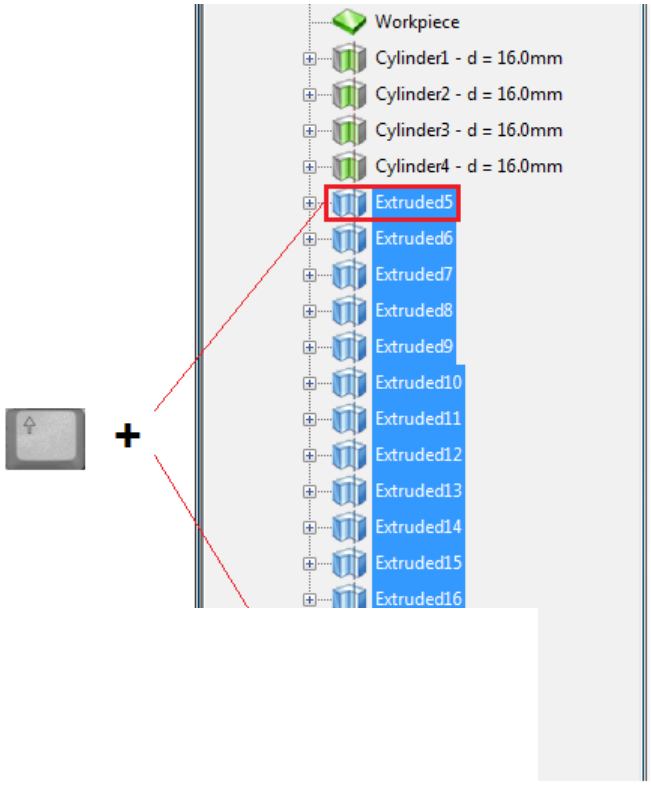
The start position is in the center of the cylinder. This is the result of the automatic start hole recognition of the OPTICAM Feature Recognition.



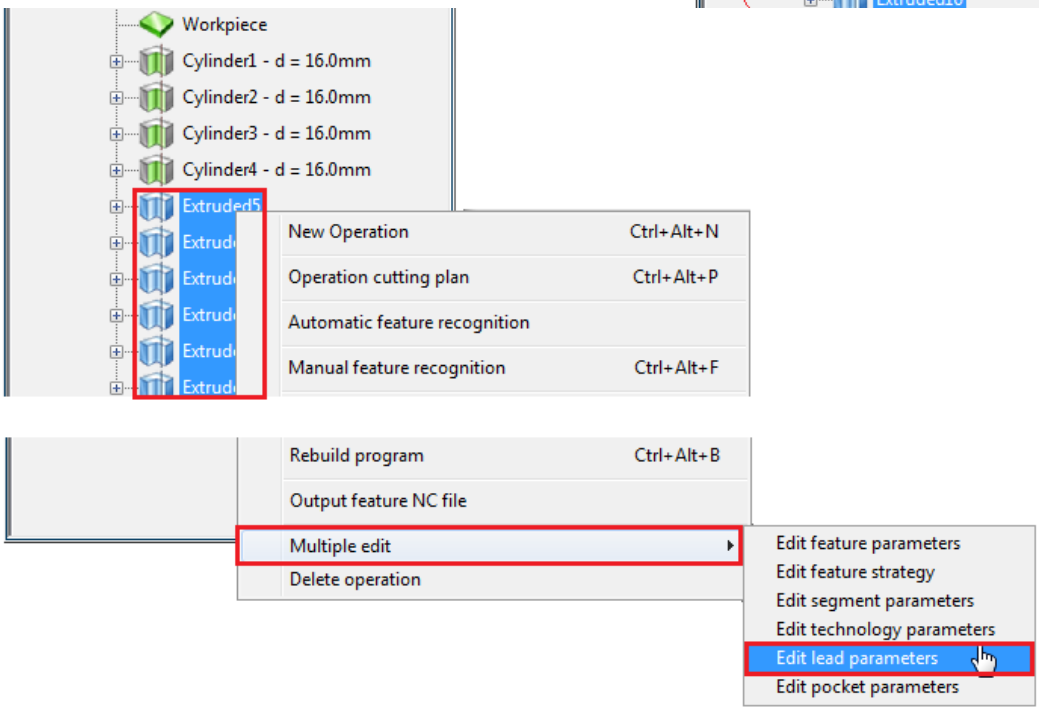
The lead off is too long and should be shortened. In the following, we will shorten the lead off.

Shortening the Length of the Lead off

In Feature Manager, click the first feature (**Extruded5**) and, while holding down the **Shift** key, click the last feature (**Extruded20**), easily selecting all 16 features.

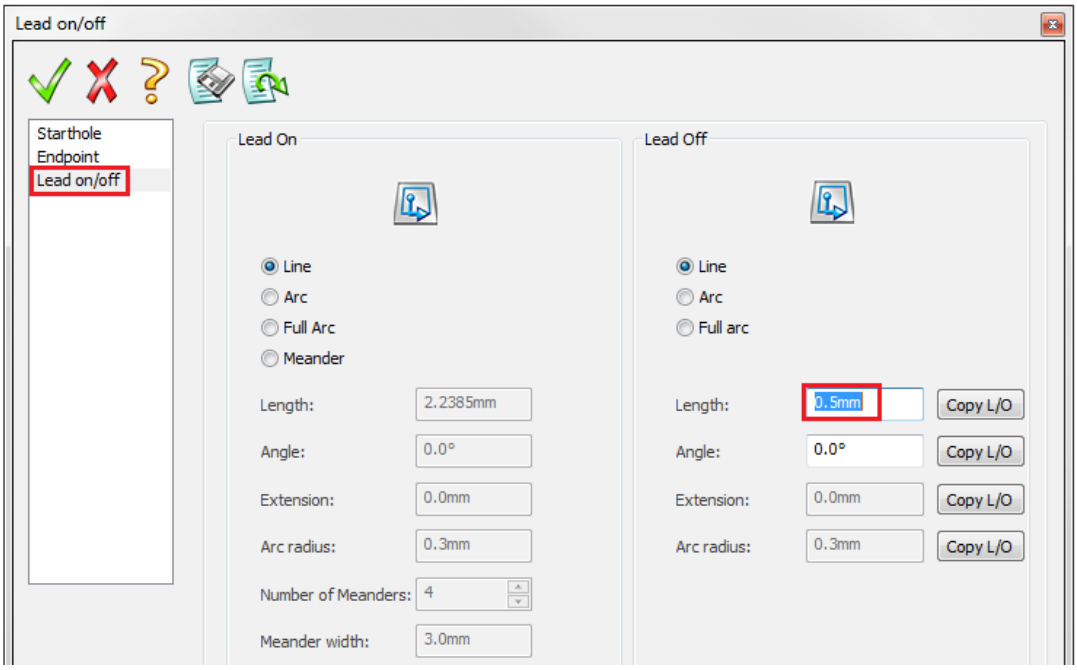


Now right-click one of the selected features and, on the context menu, select **Multiple edit > Edit lead parameters**.



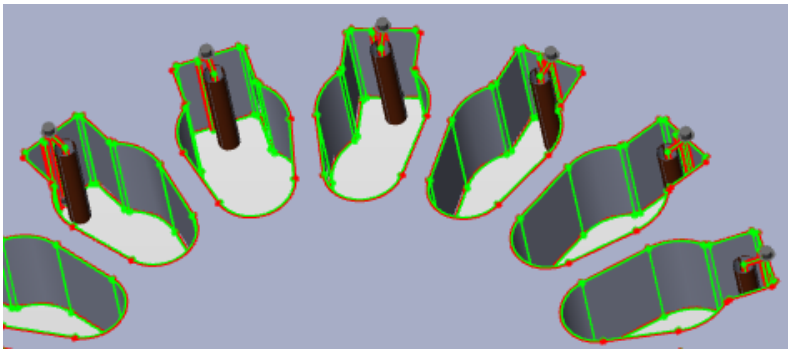
This opens the **Lead on/off** dialog.

On the left-hand side, click the feature **Lead on/off** and modify the length of the **Lead Off** to **0.5** mm.



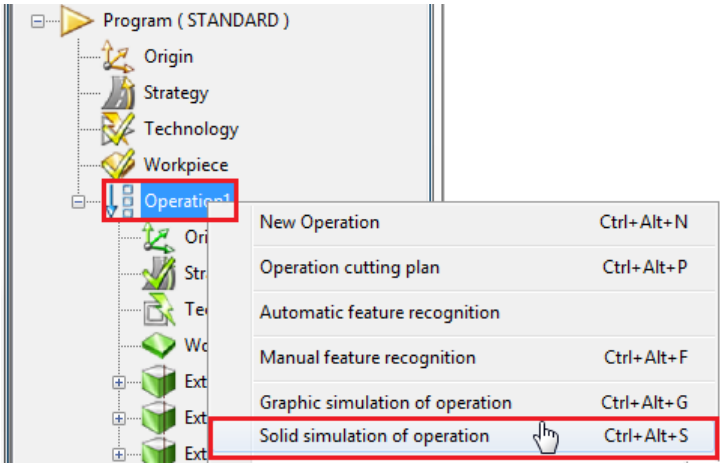
Accept changes by clicking  .

The adjustment of the lead off is applied to all selected features.

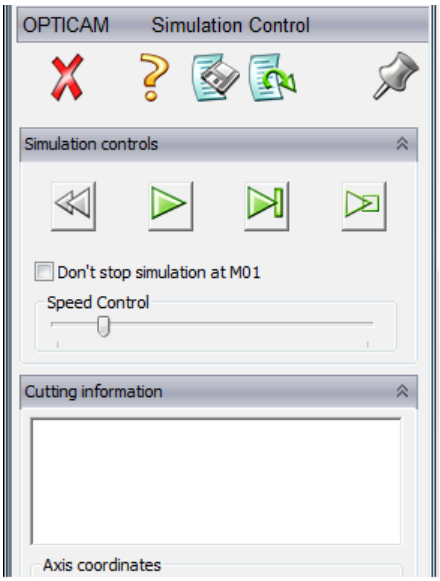


Machining Simulation (Solid Simulation)

Right-click the branch **Operation1** and, on the context menu, click **Solid simulation of operation**.

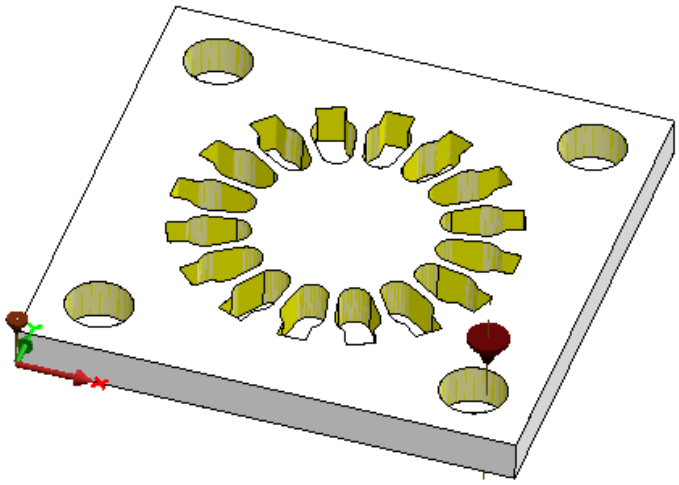


A dialog opens that allows you to control the solid simulation:



Click  to start the simulation. Use **Speed Control** to control the simulation speed.

Simulate the complete program.



Tip: With the button you can reset the simulation and then restart it.

Close the simulation dialog by clicking .

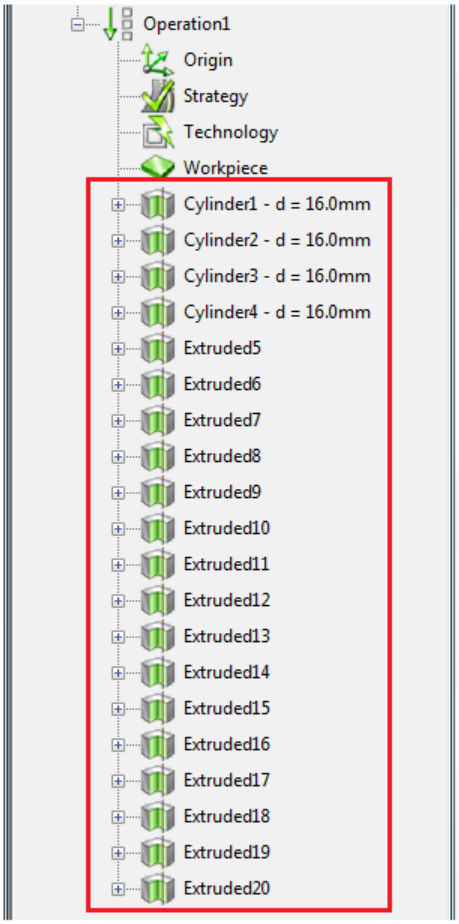
Grouping Repeated Features

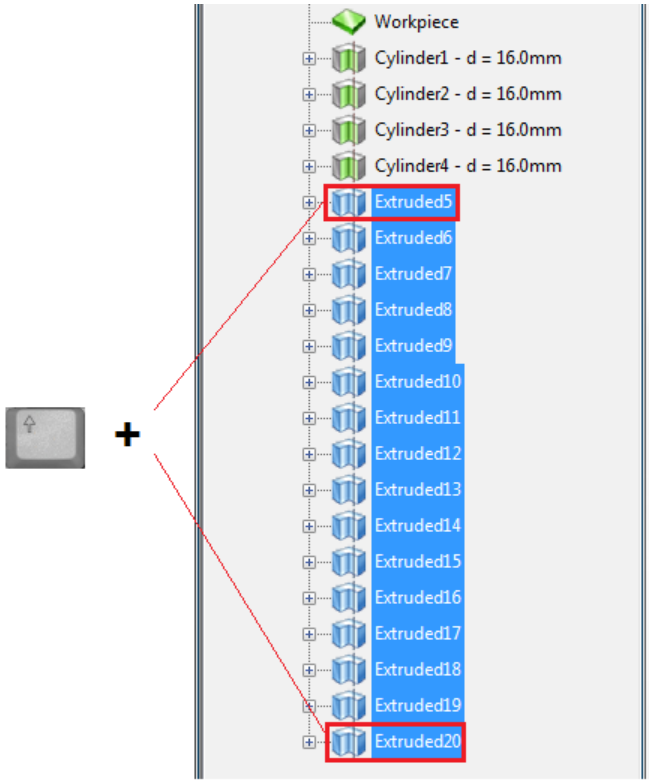
In the Feature Manager, a separate feature is created for each identified aperture. This can sometimes lead to a lot of features.

To prevent this effect, it is possible to unite several features into one object afterwards.

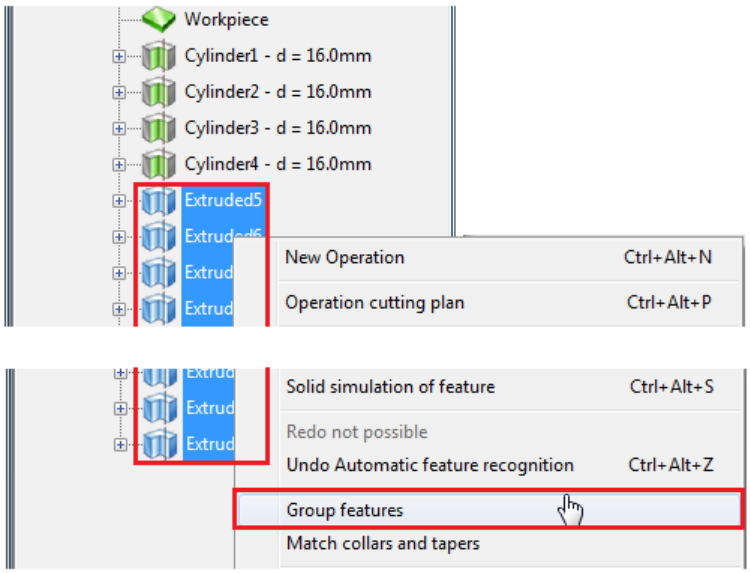
To demonstrate the procedure, we will unite the features Extruded 5-20 into one feature:

To select all 16 features at the same time, hold down the **Shift** key and click the first feature (Extruded5) and the last feature (Extruded20).

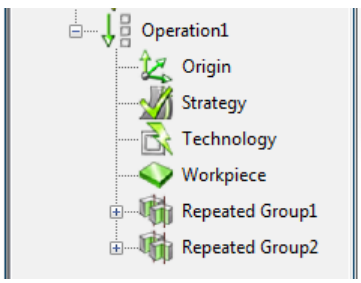




Right-click one of the selected features and, on the context menu, click **Group features**.



This unites the selected features into one feature.

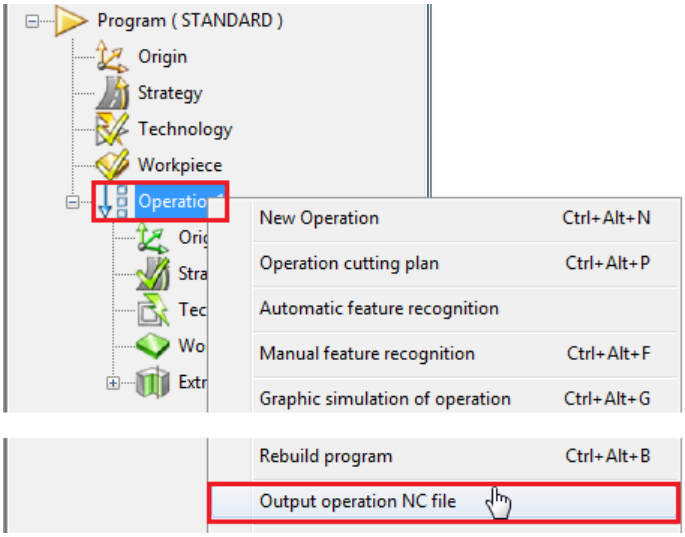


User advice: You do not have to manually group the features via the Feature Manager if you activate the option Group repeated features in the Settings dialog.

Creating the NC program

As we have already practiced in the preceding examples, at the end of this exercise we create the NC program.

Right-click the branch **Operation1** in the OPTICAM Feature Manager and, on the context menu, click **Output operation NC file**:

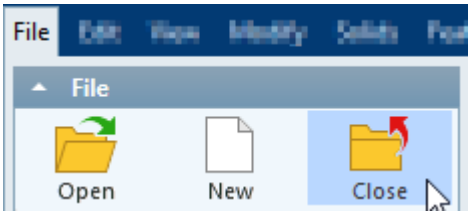


Saving the OPTICAM Program

At the end, you should save your work.

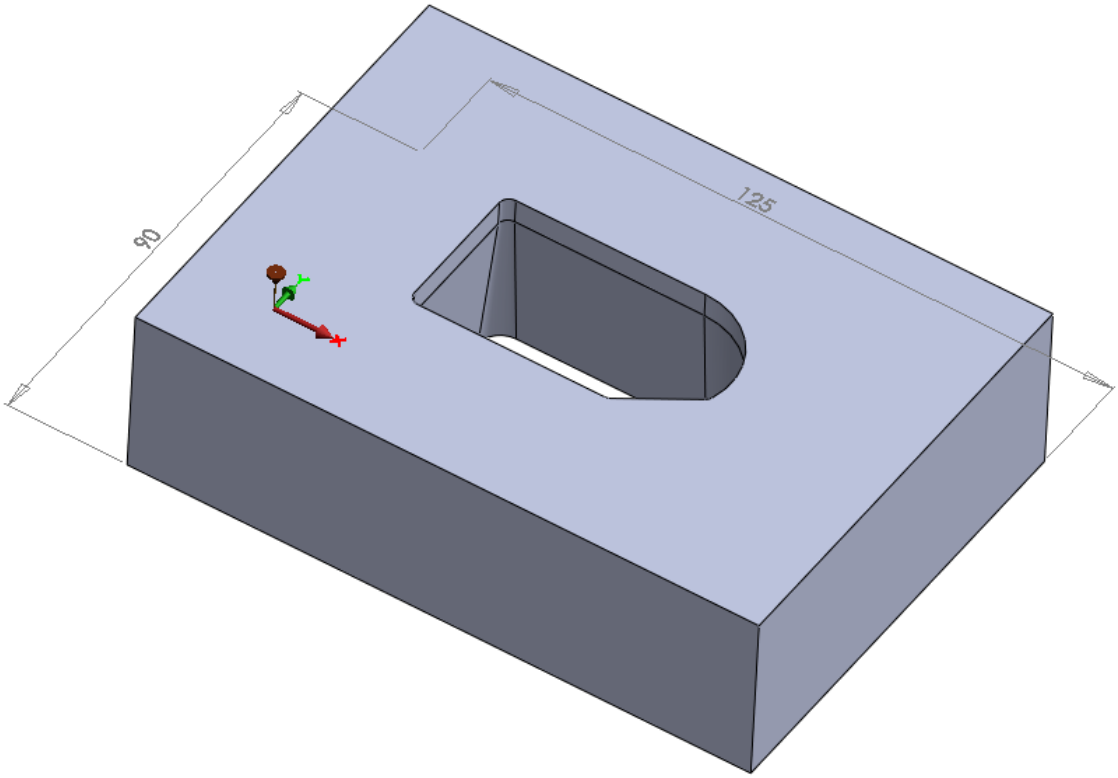
User advice: In addition to the GibbsCAM file **GC14_Opticam_example_4.vnc**, a second file containing the OPTICAM machining information for the current program will be saved. This OPTICAM file has the file extension .**wire.opticam**.

Close the file **GC14_Opticam_example_4.vnc**.



This exercise is now completed.

Example 5: Collar Machining



Demonstrated in This Example

Opening a file: GC14_Opticam_example_5

Using the **Program** Dialog for Basic Settings

Defining the Machine Origin

Defining the Cutting Technology (Technology from Database)

Creating the Workpiece (Billet) Geometry

Setting the Cutting Heights (Reference Plane Height and Secondary Plane Height)

Setting the Cutting Strategy

Automatic Creation of Collar Machining through Feature Recognition

Adjusting the Cut Sequence (Taper in Front of Cylindrical Section)

Modifying the Start and the Threading Position

Machining Simulation (Solid Simulation)

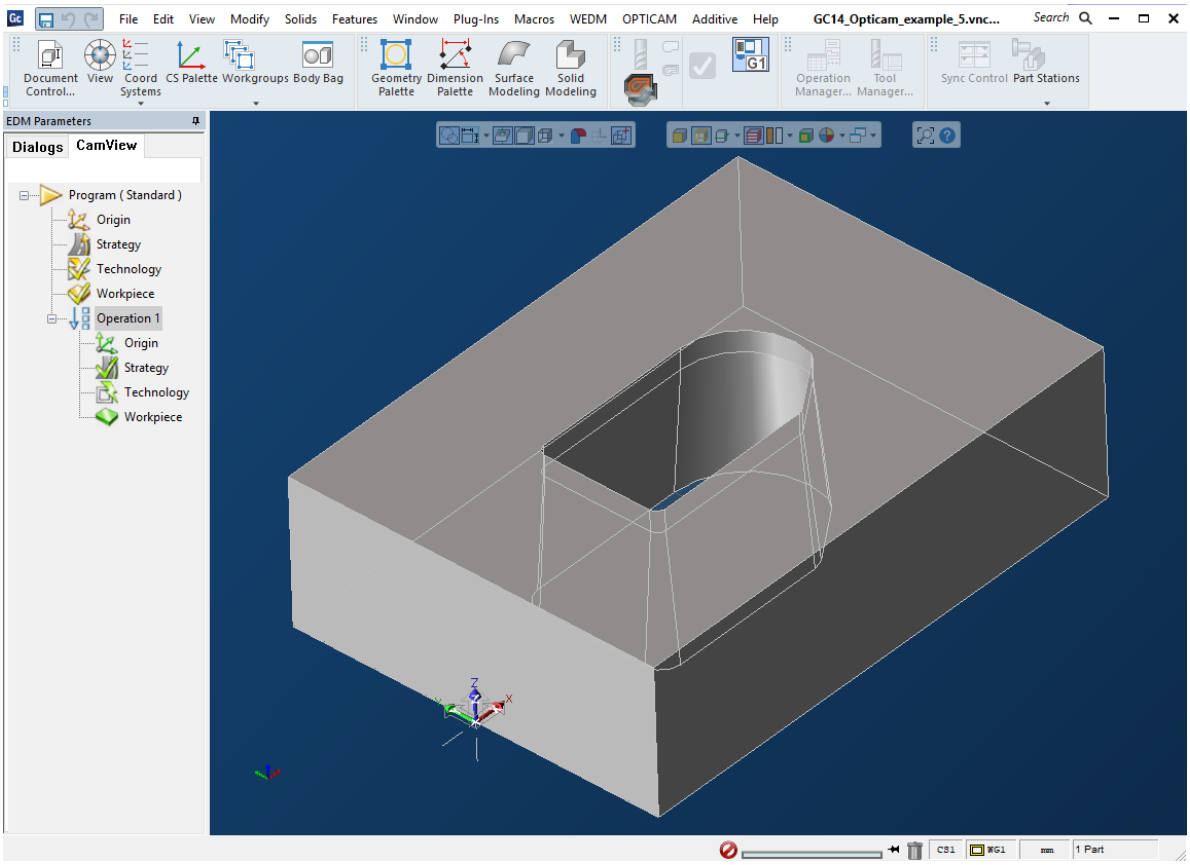
Adjusting the Number of Cuts and the Technology of the Tapered Section

Adjusting the Size of the Bridge

Creating the NC Program

Open GC14_Opticam_example_5

In GibbsCAM, open the example file **GC14_Opticam_example_5.vnc**.

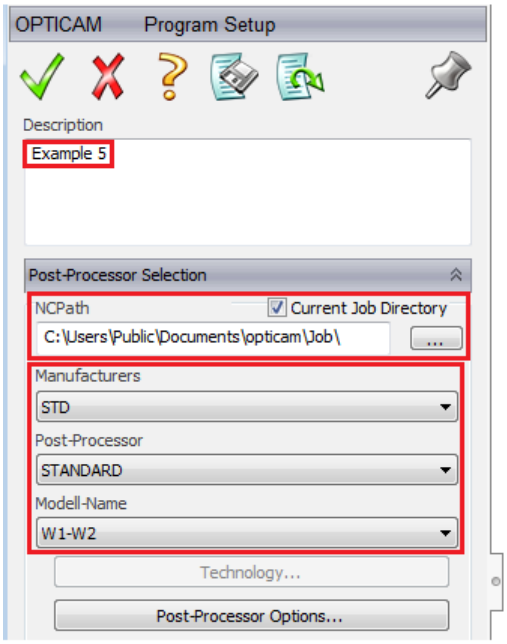


Using the Program Dialog for Basic Settings

Double-click the branch **Program (STANDARD)** in the OPTICAM Feature Manager.

Fill in the dialog as shown to the right:

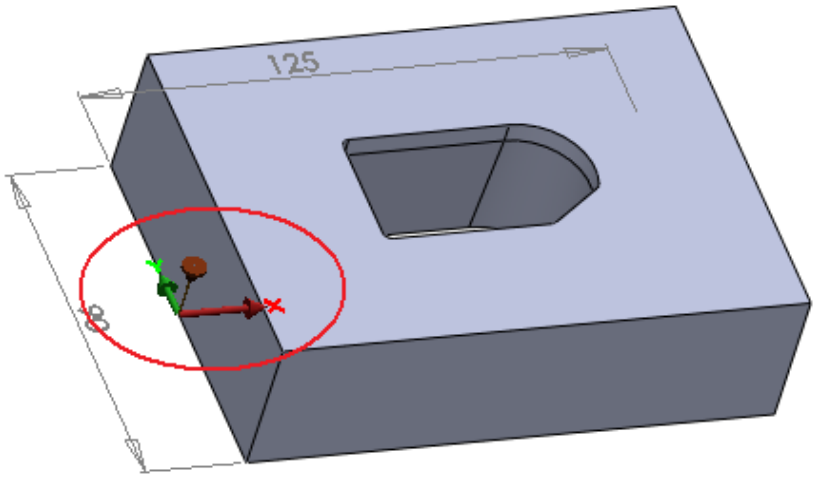
Accept changes by clicking  .



Defining the Machine Origin

In the next step, we will once more check the position of the machine origin. In this example, the origin is in the center of the lower left edge of the model.

As this is okay, you do not need to adjust the position.



Defining the Cutting Technology (from Database)

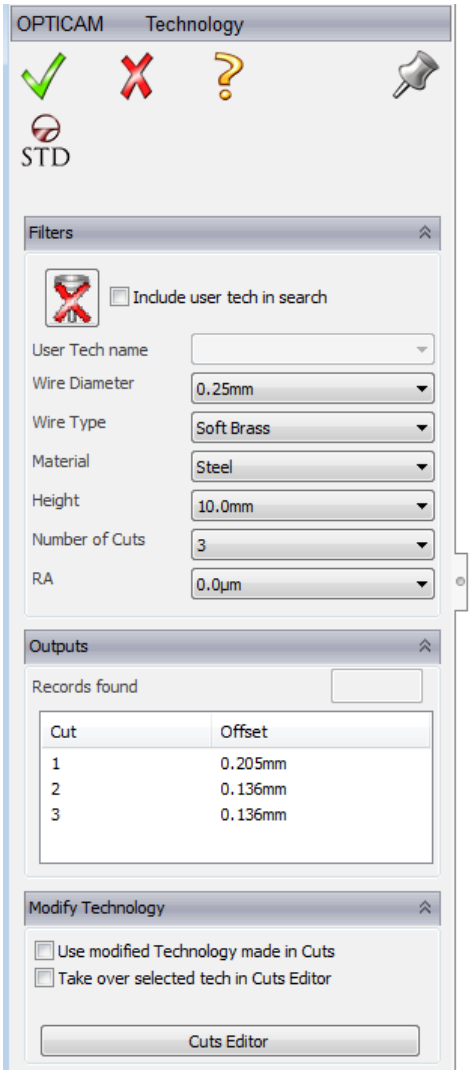
Now we will define the cutting technology the workpiece is to be machined with

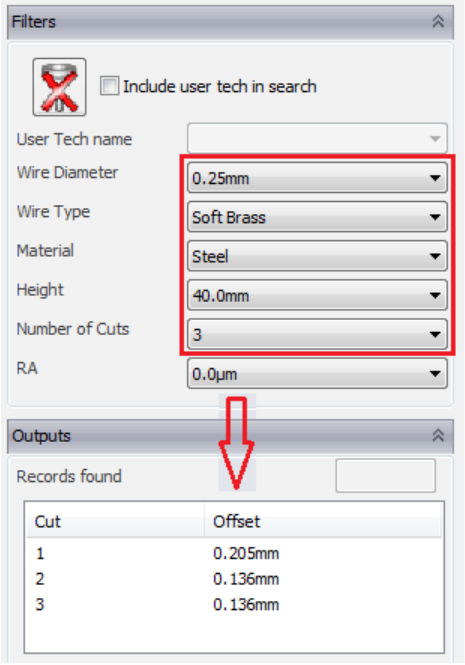
Double-click the branch **(Program) Technology** to open the dialog for the definition of the cutting technology.

You can either select the parameters of the cutting technology from a database or enter them manually.

We want to load the technology from the Technology Database. You can do this as follows:

Select suitable technology values from the listboxes **Wire Diameter**, **Wire Type**, **Material, Height** and **Number of Cuts.**





The selected technology is displayed in the dialog area **Records found.**

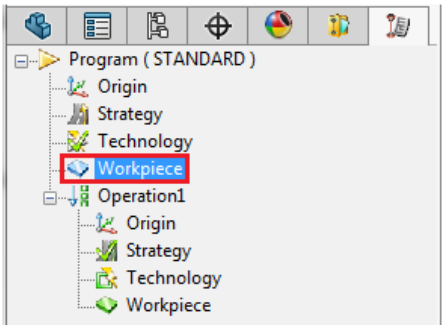
User advice: The input fields of the technology database vary according to the selected postprocessor and often only contain a few example technologies. For most wire EDM machines, complete technology databases are optionally available.

Accept the technology selection by clicking .

Creating the Workpiece (Billet) Geometry

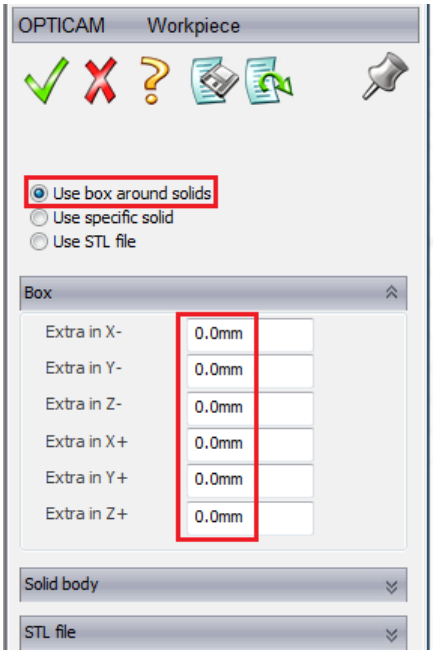
In the next dialog, you can define the geometry of the workpiece (billet).

Double-click the branch **(Program) Workpiece** in the OPTICAM Feature Manager.



Again, the workpiece (billet) is to be derived from the solid body; additionally, no allowance values are to be defined.

Fill in the dialog as follows:

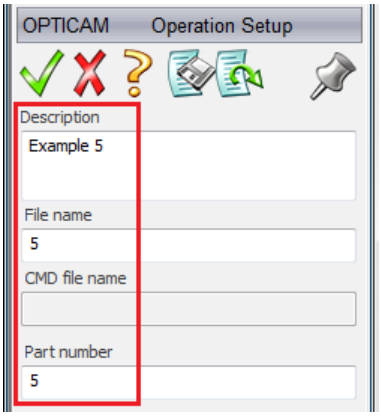


Accept changes by clicking  .

Setting the Cutting Heights (Reference Plane Height and Secondary Plane Height)

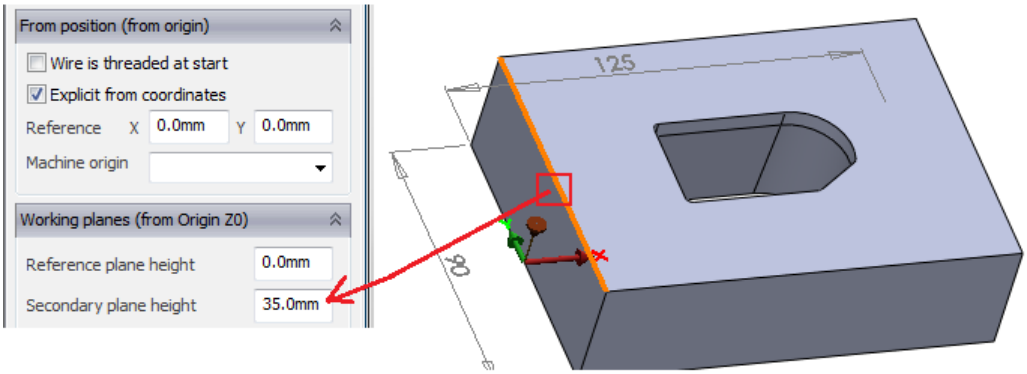
The following dialog sets among other things the name of the NC output file as well as the Z-height of the Reference and the Secondary Plane Height.

Double-click the branch **Operation1** in the OPTICAM Feature Manager, and fill in the fields **Description**, **File name** and **Part number** as follows:



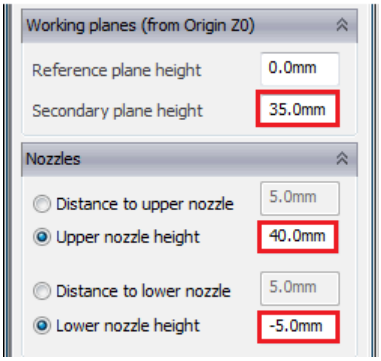
Identify the Secondary Plane Height in the model. You can do this by selecting an element (an edge, a plane face or a vertex) which is located at the desired height.

Position the cursor in the input field **Secondary plane height**.



Click an element on the top level of the model. OPTICAM calculates the Z-height of the selected element and passes the value to the **Secondary plane height** dialog box.

Please note that the **nozzle distances** in the lower dialog area (important for the simulation) have also been adjusted.

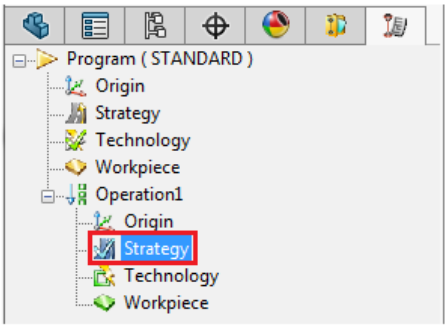


Accept changes by clicking  .

Setting the Cutting Strategy

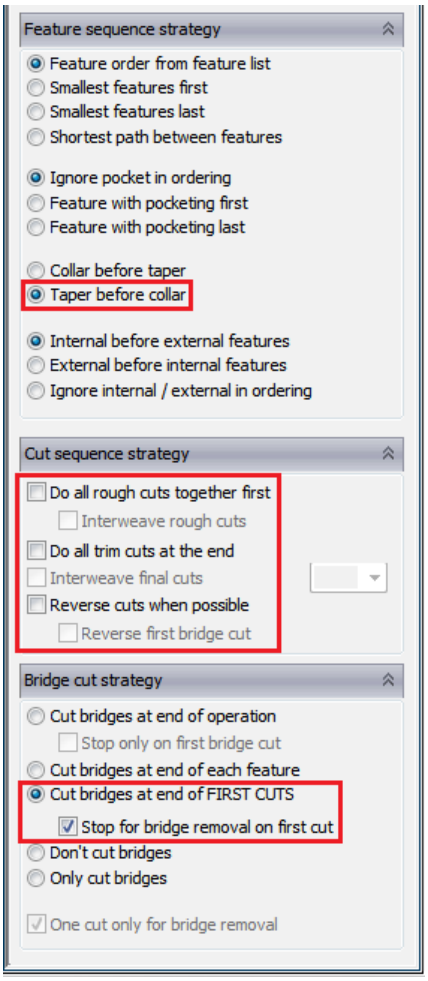
In the next dialog, you can define with which strategy the workpiece will be machined.

Double-click the branch **(Operation) Strategy** in the OPTICAM Feature Manager.



The internal aperture is to be machined in a way that **at the End of the Main Cut** there will be a **Machine Stop for the Removal of the Bridge**. Afterwards, the 2 Trim Cuts are to follow.

Set the dialog as follows:



Please note that due to the setting **Taper before Collar**, the tapered part of the collar machining will be cut before the cylindrical part.

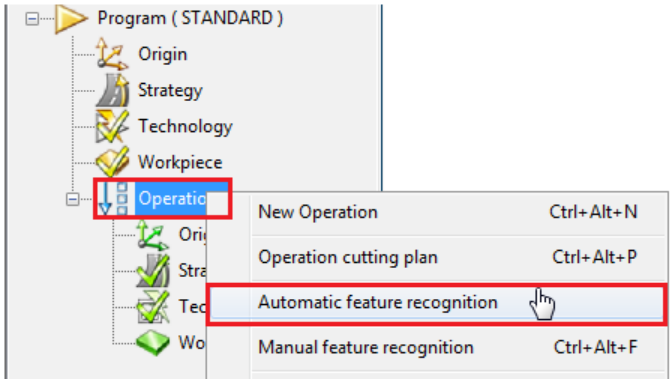
Accept changes by clicking  .

Creating the Cutting Machining

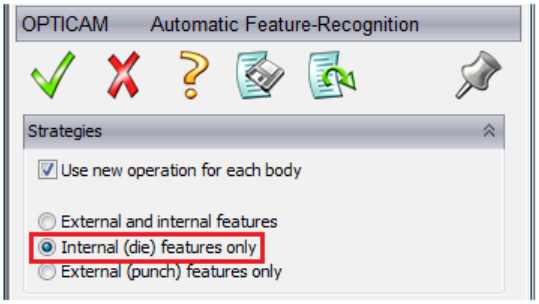
In this example, only the **internal aperture**, the collar machining, which consists of a tapered and a cylindrical section, will be machined.

Automatic Creation of the Cutting Machining

Right-click the branch **Operation1** in the OPTICAM Feature Manager and, on the context menu, click **Automatic feature recognition**.

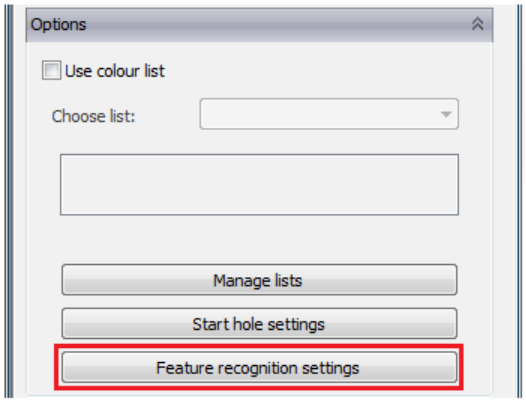


The dialog **Automatic Feature Recognition** is displayed. To machine **exclusively the internal aperture**, set the dialog as follows:



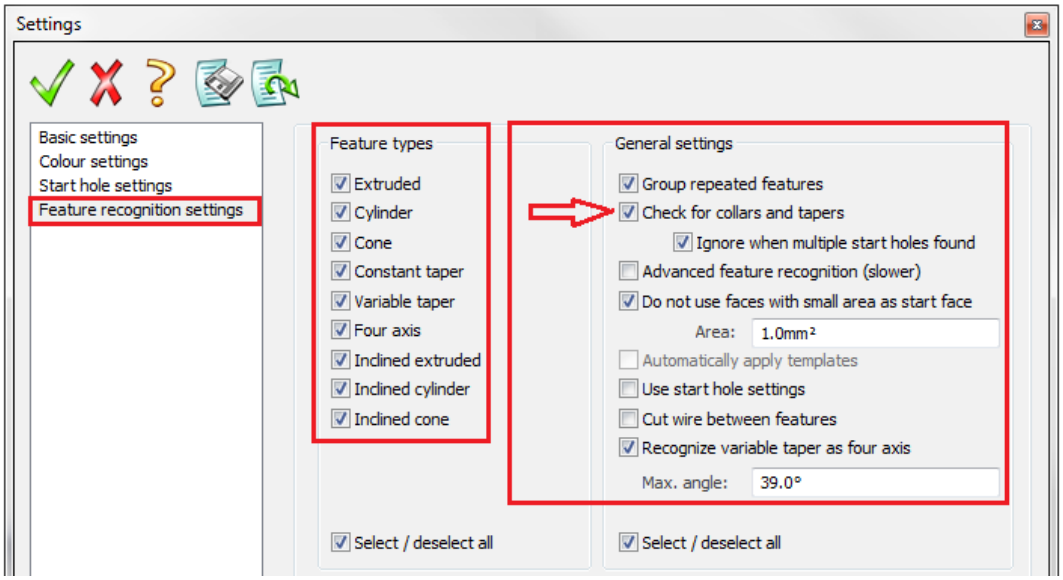
In order to set what exactly the **Feature Recognition** is supposed to recognize, click the 

button in the dialog.



A dialog containing setting parameters for the **Feature Recognition** opens.

Make sure that the dialog looks as follows:

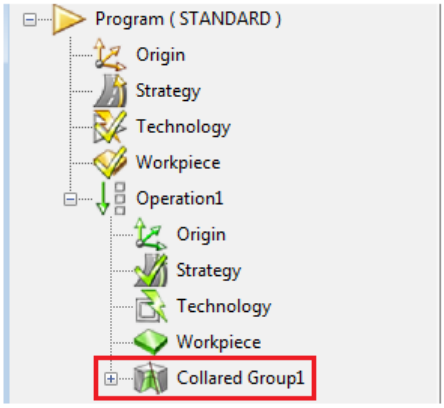


User advice: You have to activate the option **Check for collars and tapers**; if this option is not activated, the collar machining will be split into ***two*** **features** (cylindrical and tapered part).

Accept changes by clicking  .

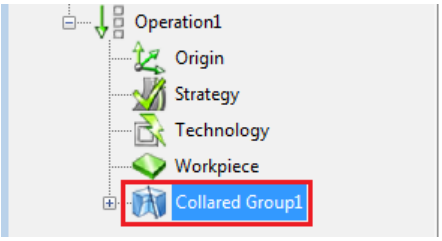
The system will now search the solid body for all geometries that can be wire cut (feature recognition) and create a machining suggestion with the cutting parameters previously set.

The collar machining geometry is automatically identified and listed in the OPTICAM Feature Manager.



Modifying the Start and the Threading Position

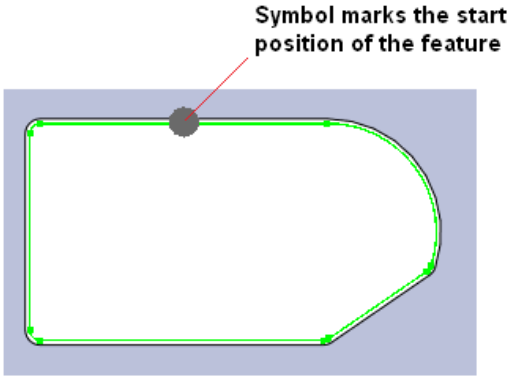
Click the Collared Group1 in the OPTICAM Feature Manager.



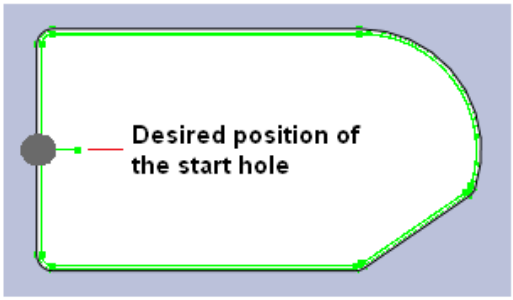
The cutting path and the start position of the selected object are highlighted in the model.

If you extend the feature and look at it more closely, you will note several things.

The start position (which is marked by a symbol) is in an unfavorable position. Additionally, the start hole should have a larger distance to the cutting contour.



The following figure shows the desired start position and the start hole in front of the cutting contour.

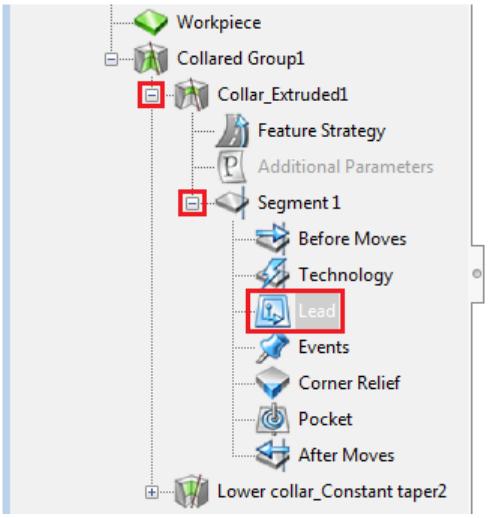


The lead off is too long and should be shortened. In the following, we will shorten the lead off.

Adjusting the Start and the Threading Position

Open **Collared Group1** and double-click the branch 

.

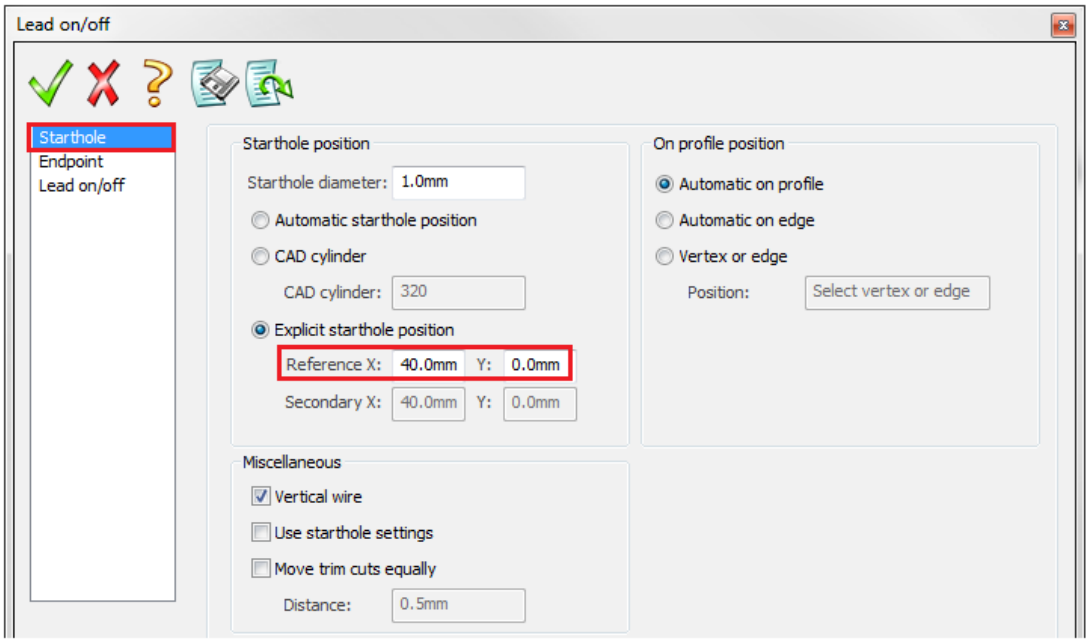


This opens the **Lead on/off** dialog.

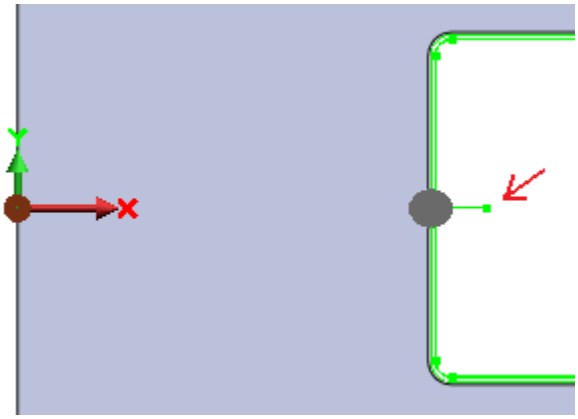
On the left-hand side, click the feature **Starthole**.

To enter the coordinates of the start hole directly or “catch” them in the model, you have to activate the option **Explicit starthole position**.

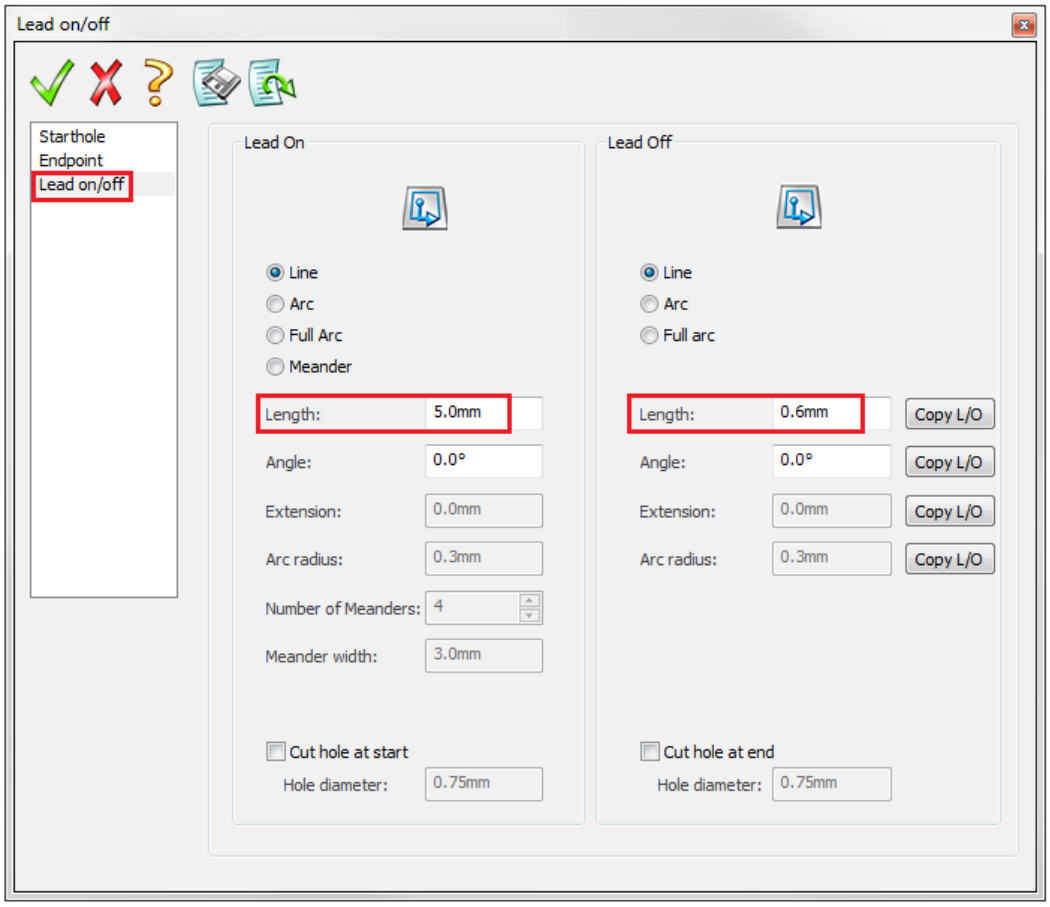
Enter **X40** and **Y0** as the position of the start hole.



Note how the input modifies the start position and the position of the start hole:



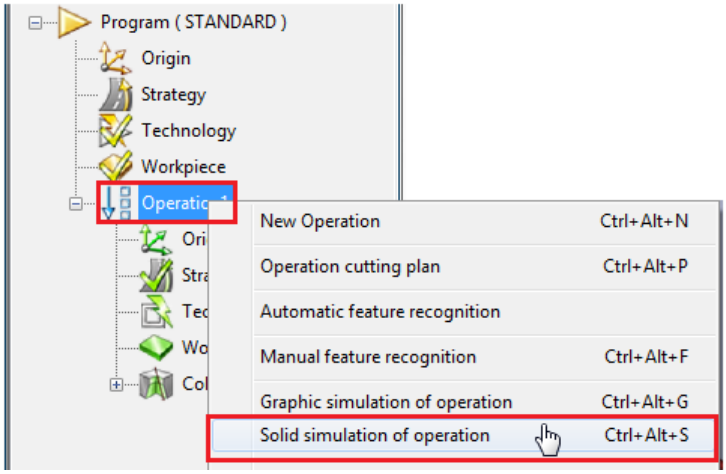
Click the feature **Lead on/off** and modify the length of the **Lead On** to **5** and of the **Lead Off** to **0.6** mm.



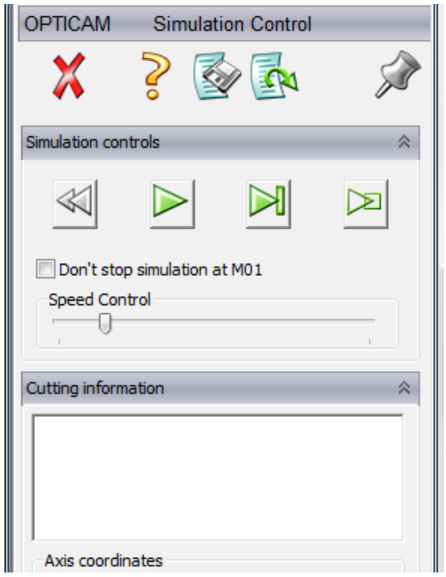
Accept changes by clicking  .

Machining Simulation (Solid Simulation)

Right-click **Operation1** and, on the context menu, click **Solid simulation of operation**.



A dialog opens that allows you to control the solid simulation:

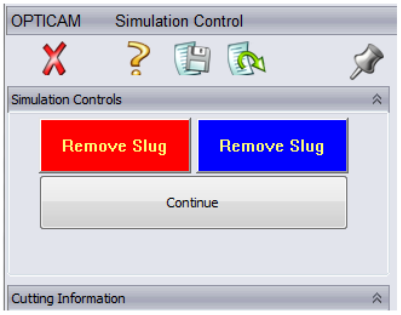


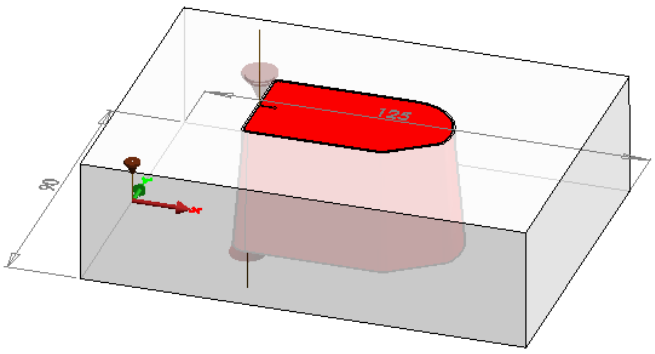
Click  to start the simulation. Use **Speed Control** to control the simulation speed.

Simulate the complete program.

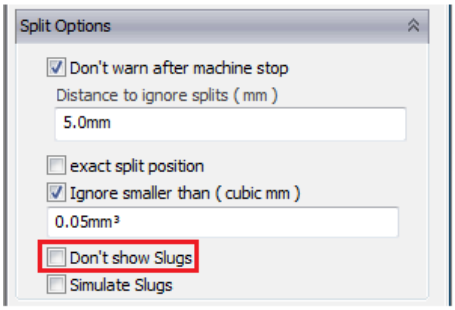
Depiction of Slugs During the Simulation

If the bridge is cut, normally a slug will be created. The solid simulation can visualize this slug and displays the button **Remove Slug**. The slug will be displayed in red.



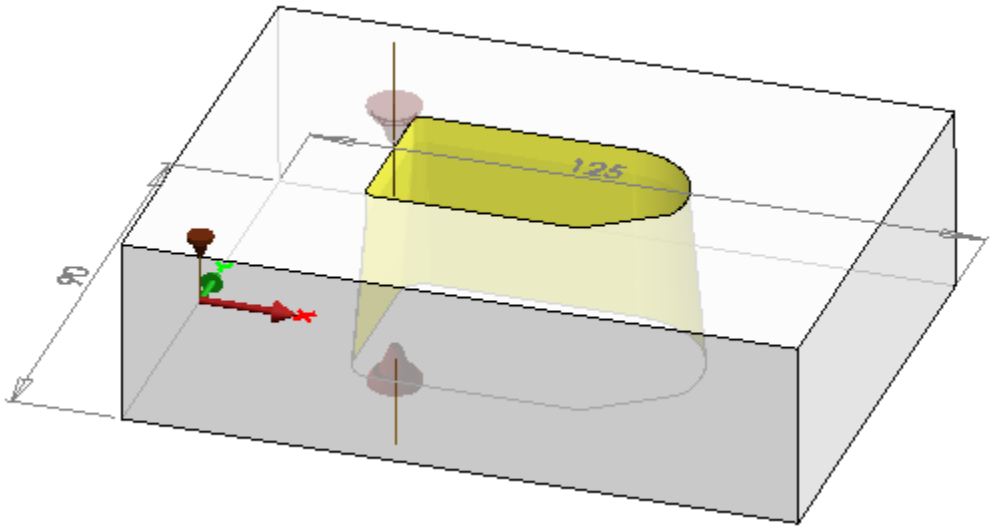


User advice: In the field **Split Options** in the simulation dialog you can set if a slug will be displayed or not.



If you want to see the slug, you have to *deactivate* the option **Don't show slugs**.

The following figure shows the final result of the simulation.



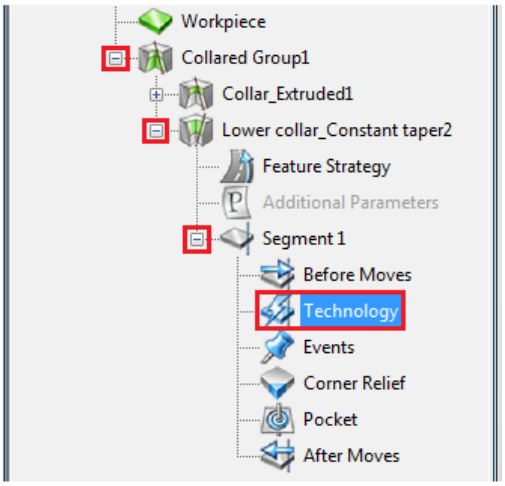
Close the simulation dialog by clicking .

Adjusting the Number of Cuts and the Technology of the Tapered Section

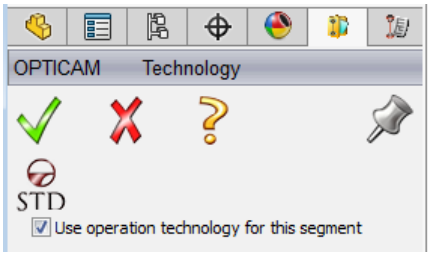
In the course of the simulation, you should have noticed that both the tapered and the cylindrical section were machined with three cuts. The fundamental reason for this is that we have previously selected a cutting technology with three cuts from the technology database. The selected technology is used for both objects.

In the following, we will adjust the technology of the tapered cut so that the taper is machined with only one cut.

Open the **Collared Group1** in the OPTICAM Feature Manager, then open the branch **Lower collar_Constant taper2**, and then double-click the branch **Technology**.

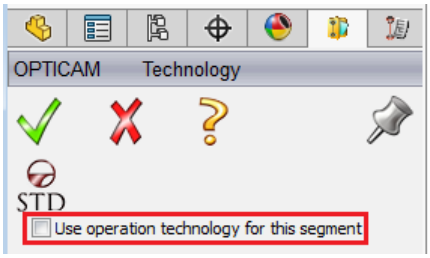


This opens the Technology dialog:



As the option **Use operation technology for this segment** is activated, the system uses the cutting technology previously set (3 cuts) for the machining of taper and collar.

If you *deactivate* the option **Use operation technology for this segment**, you can allocate your own technology to the selected taper.

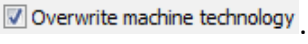


Deactivate the option **Use operation technology for this segment**.

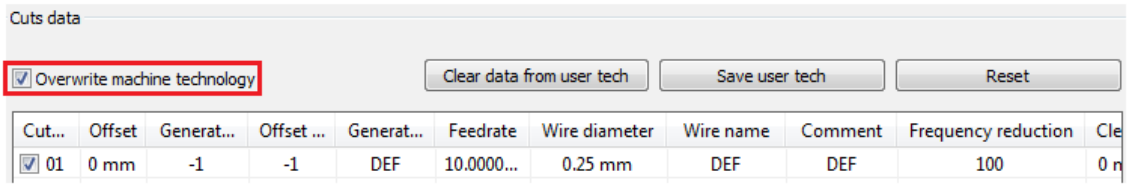
When the **Technology** dialog opens, click the 

button.

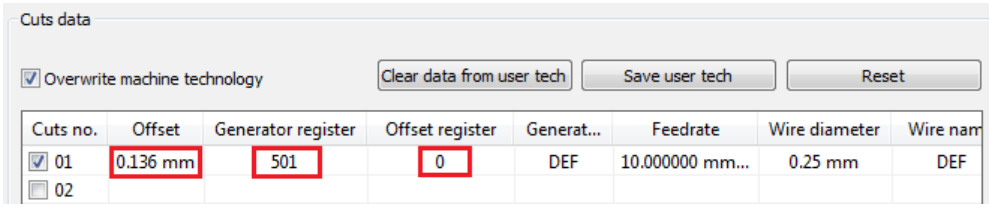
A dialog opens where you can manually enter the cutting technology.

Activate the input fields by selecting the option 

.

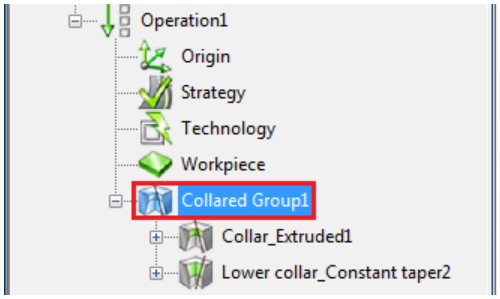


To machine the geometries with **1 Cut**, set the dialog as follows:



Accept changes by clicking  . The system returns to the **Technology** dialog. Close this dialog as well by clicking  .

Adjusting the Size of the Bridge for the Cylindrical Section



As we do not need a bridge for the Cylindrical Section of the collar machining, we will change this.

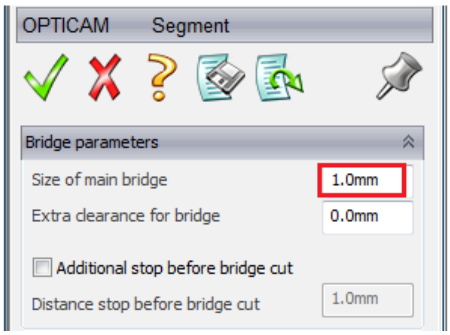
Double-click the **Collared Group1** in the OPTICAM Feature Manager.

Deactivate the option **Stop on lower taper** in the dialog area **Collar control**. The cylindrical part will now be executed without a bridge (machine stop).

Accept changes by clicking  .

Double-click the branch 

. This opens a dialog that lets you set the size of the bridge (distance of the stop point). Make sure that in the field **Size of main bridge** the value **1** is displayed, as shown to the right.

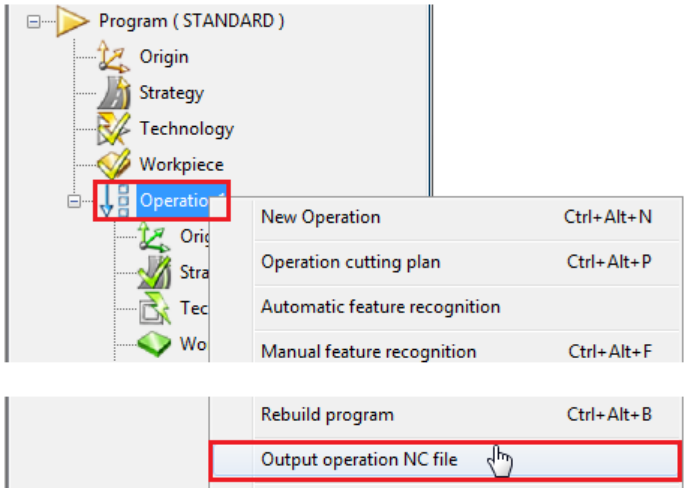


Accept changes by clicking  .

Creating the NC program

As we have already practiced in the preceding examples, at the end of this exercise we create the NC program.

Right-click **Operation1** and, on the context menu, click **Output operation NC file**.



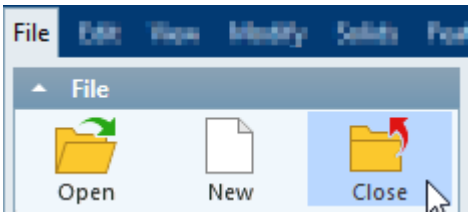
Please continue working independently…

Saving the OPTICAM Program

In the end, you should save your work.

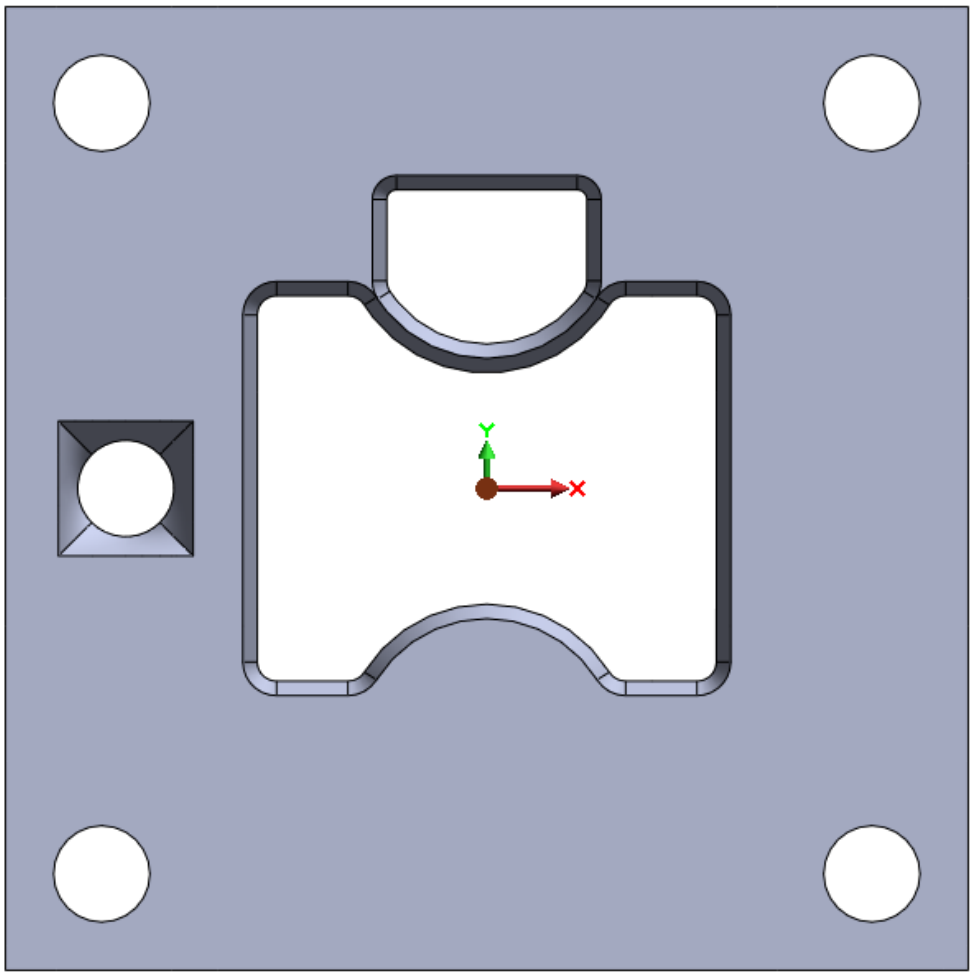
User advice: In addition to the GibbsCAM file **GC14_Opticam_example_5.vnc**, a second file containing the OPTICAM machining information for the current program will be saved. This OPTICAM file has the file extension .**wire.opticam**.

Close the file **GC14_Opticam_example_5.vnc**.



This exercise is now completed.

Example 6: Taper and Four Axes

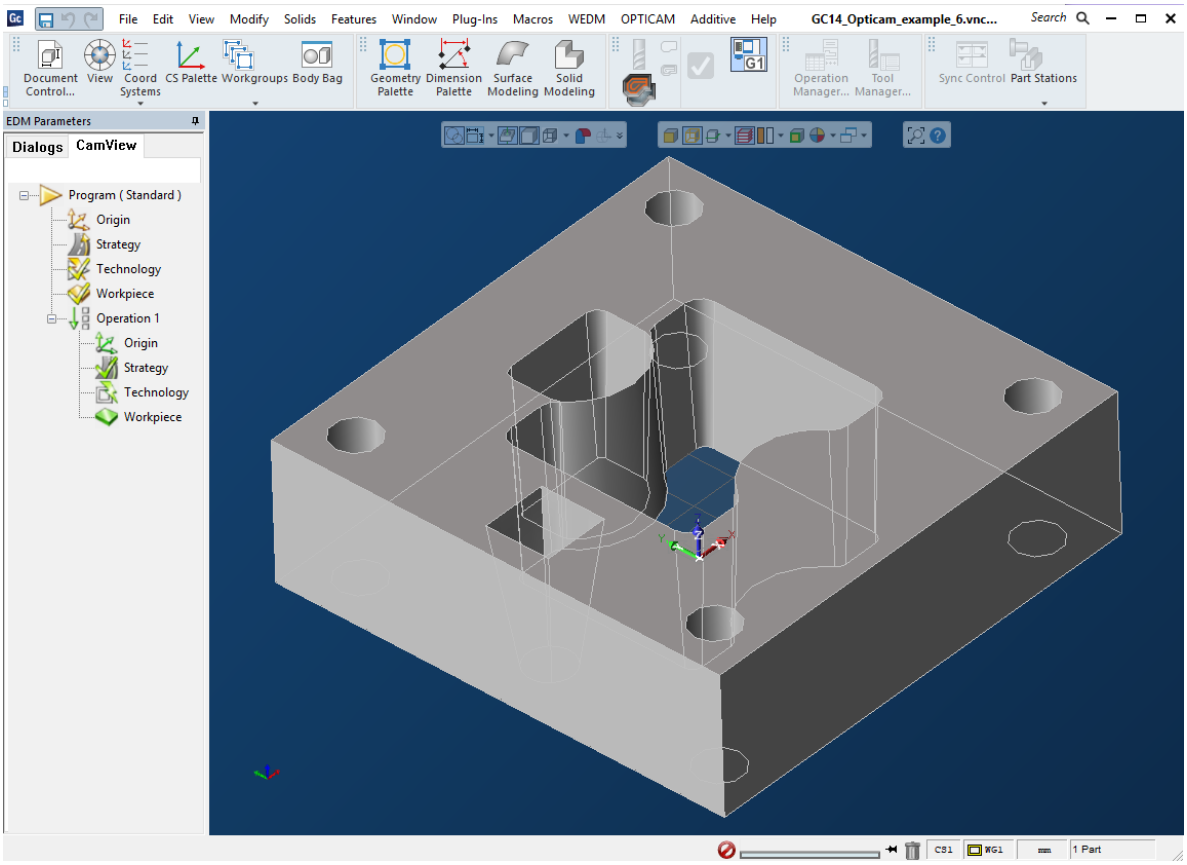


Demonstrated in This Example

Opening a file: GC14_Opticam_example_6
Positioning the Machine Origin in the Center of the Hole
Defining the Cutting Technology (Technology from Database)
Creating the Workpiece (Billet) Geometry with Allowance Values
Setting the Cutting Heights (Reference Plane Height and Secondary Plane Height)
Setting the Cutting Strategy
Automatic Creation of the Cutting Machining
Hiding Certain Geometry Types
Machining the External Geometry
Modifying the Length of the Lead on/off for all Features
Positioning the Start Hole in the Center of a Straight Line
Adjusting the Size of the Bridge for the External Feature

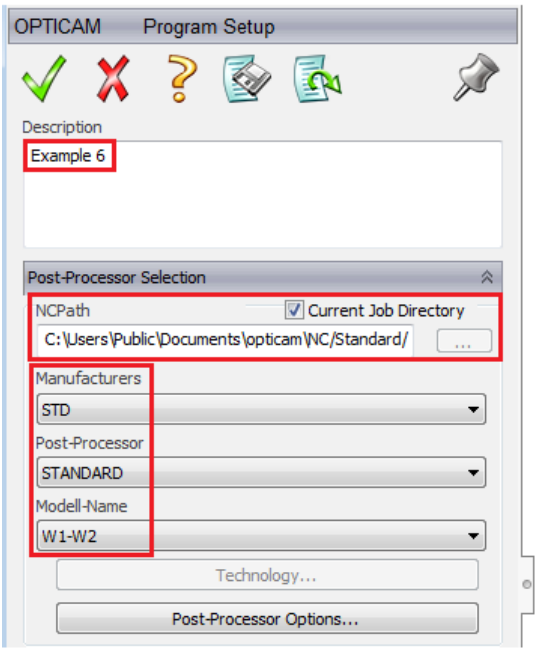
Open GC14_Opticam_example_6

In GibbsCAM, open the example file **GC14_Opticam_example_6.vnc**.



Using the Program Dialog for Basic Settings

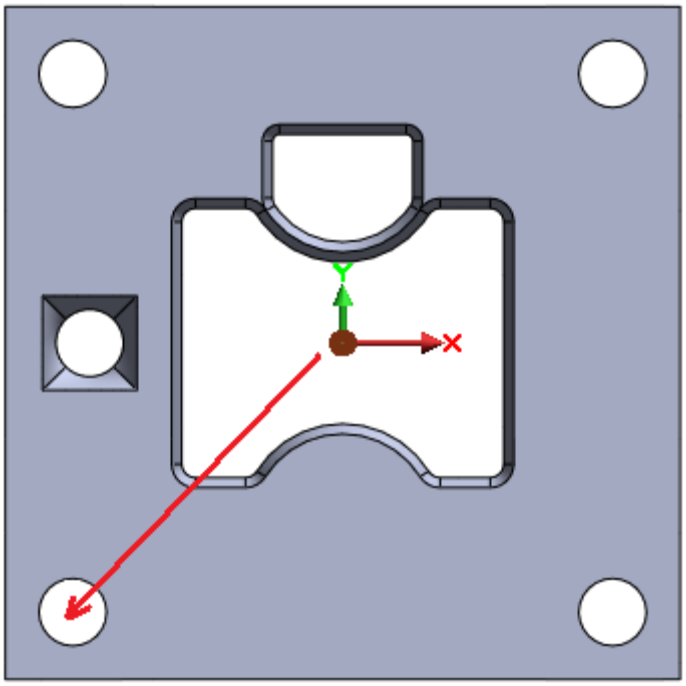
Double-click **Program (STANDARD)** and fill in the dialog as shown to the right:



Accept changes by clicking  button.

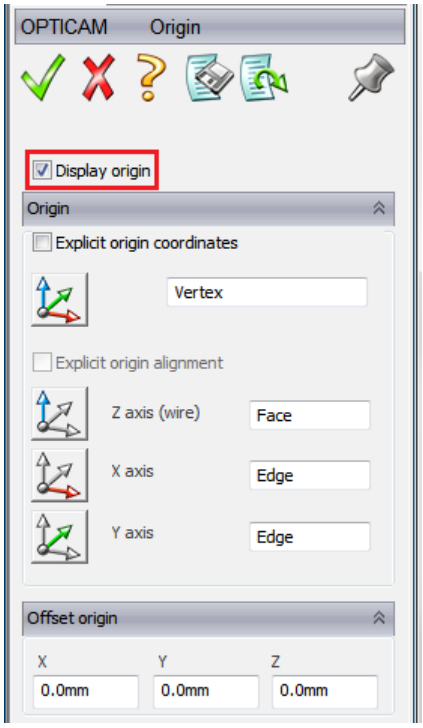
Positioning the Machine Origin in the Center of the Hole

In the next step, we will once more check the position of the machine origin. In this example, the origin is in the center of the workpiece.

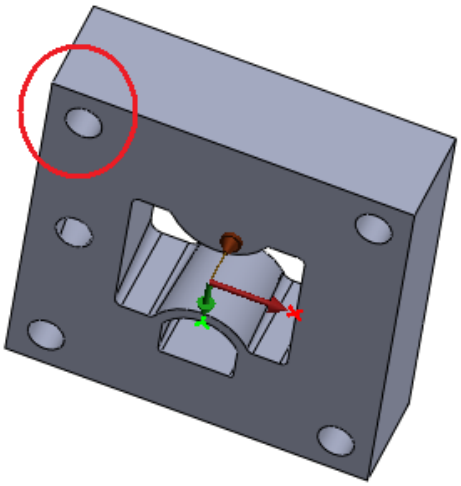


In this example, we want to move the origin to the center of the lower left hole.

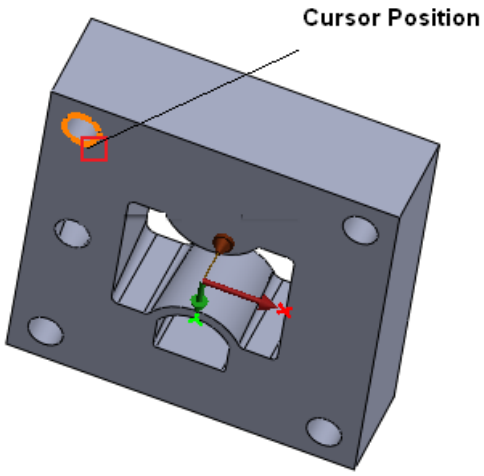
Double-click **(Program) Origin** to open the dialog for the definition of the machine origin.



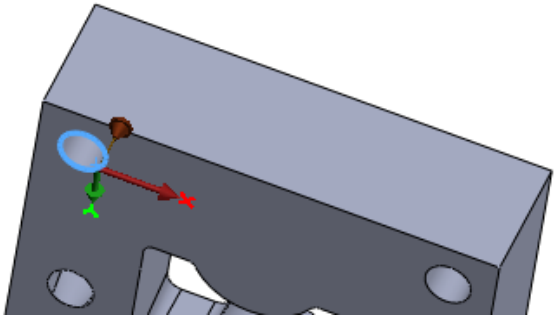
Rotate the model in such a way that you can see the lower left hole from *below* (see following figure):



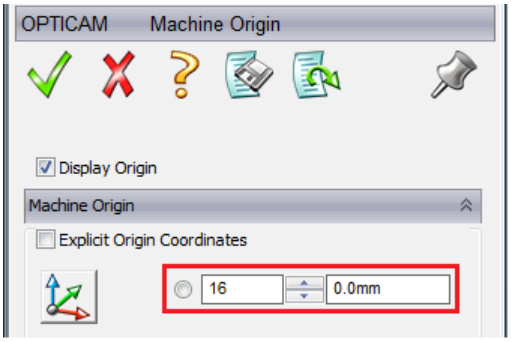
To move the origin to the center of the lower left hole, position the cursor on the model edge of the lower left hole and click with the left mouse button.



The origin moves to the selected edge, but is not in the center of the circle.

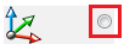
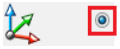


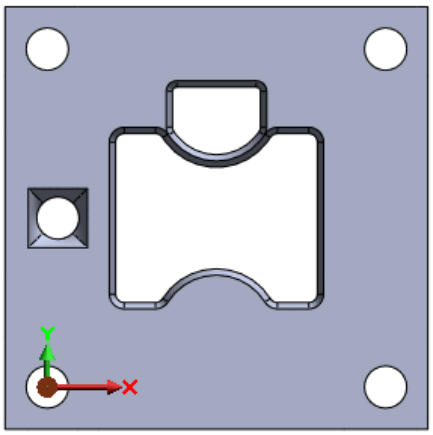
Looking at the Machine Origin dialog will help us (example).



The origin is not in the center of the hole, but *on* the selected model edge, namely 16% away from the origin of the edge. (Note that, depending on where you click the edge, a different percentage than 16% will result.)

If you were to change the value to 50%, the origin would be in the center of the edge—that is, 50% away from the origin of the edge, but still not in the center of the circle.

To move the origin to the center of the circle, choose the option  in the dialog, thus:  . This moves the origin exactly to the center of the circle.

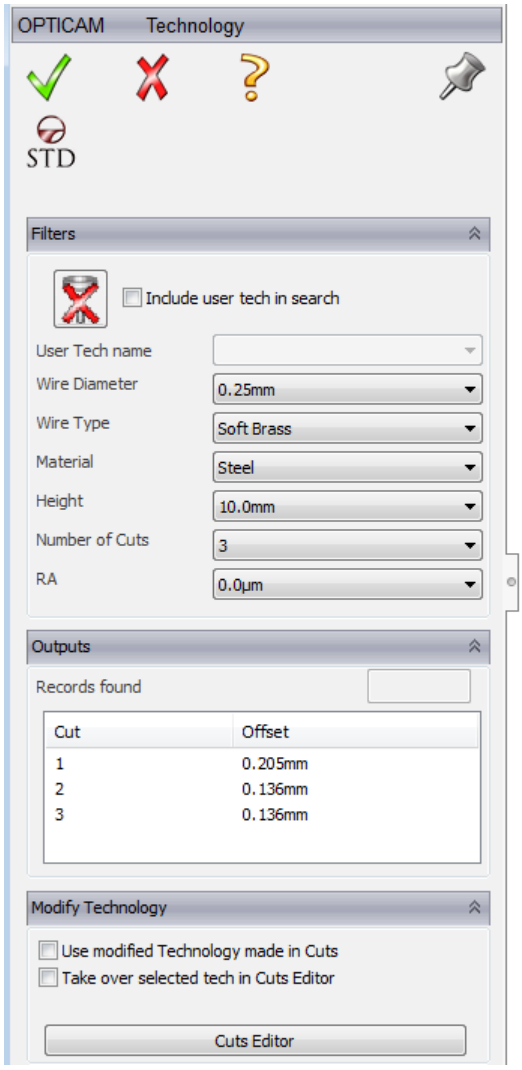


Close the dialog Machine Origin by clicking  .

Defining the Cutting Technology (from Database)

Now we will define with which cutting technology the workpiece is to be machined.

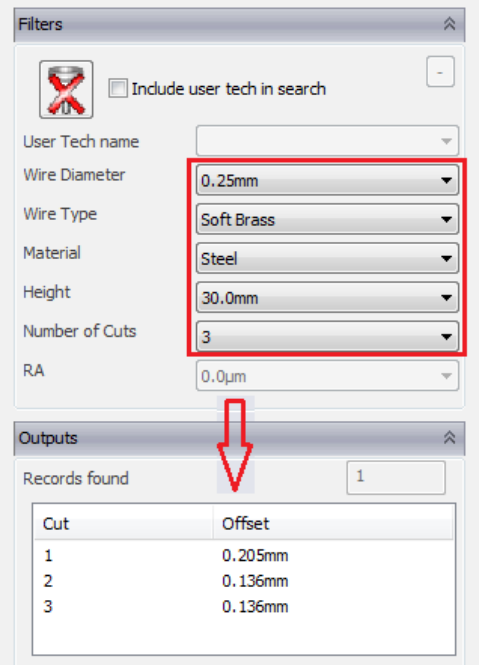
Double-click **(Program) Technology** to open the dialog to define cutting technology.



You can either select the parameters of the cutting technology from a database or enter them manually.

We want to load the technology from the Technology Database. You can do this as follows:

Select suitable technology values from the listboxes **Wire Diameter**, **Wire Type**, **Material, Height** and **Number of Cuts.**



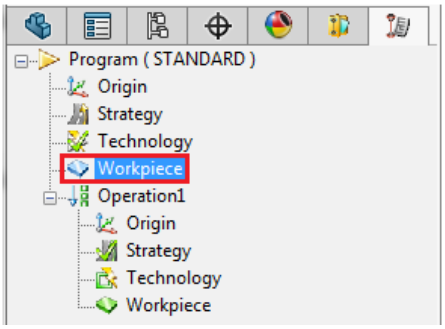
The selected technology will be displayed in the dialog area **Records found.**

Accept the technology selection by clicking  .

Creating the Workpiece Geometry with Allowance Values

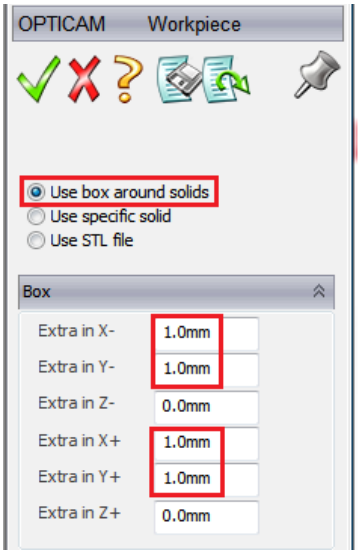
In the next dialog, you can define the geometry of the workpiece (billet).

Double-click the branch **(Program) Workpiece** in the OPTICAM Feature Manager.



Again, we want to derive the workpiece (billet) from the solid body; however, this time we will provide the workpiece (billet) with allowance values.

Fill in the dialog as follows:



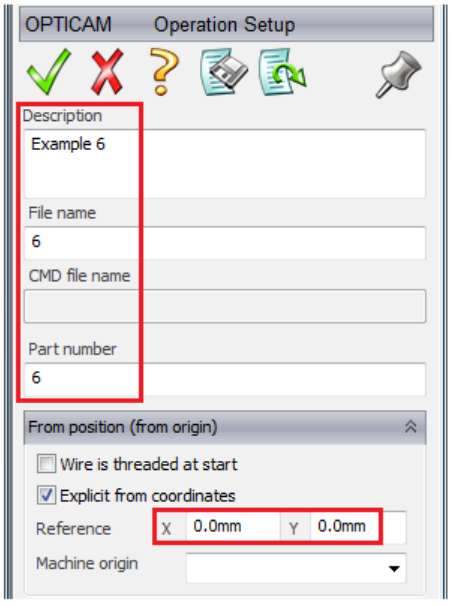
Due to the inputs, the workpiece (billet) is displayed in the **simulation** with a lateral allowance of 1 mm.

Accept changes by clicking  .

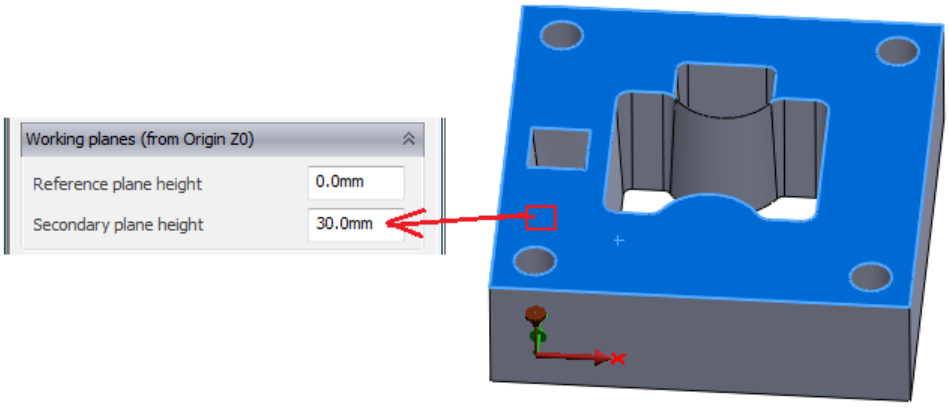
Setting the Cutting Heights (Reference Plane Height and Secondary Plane Height)

The following dialog sets (among other things) the name of the NC output file and the Z-height of the Reference and the Secondary Plane Height.

Double-click **Operation1** and fill in the fields **Description**, **File name**, **Part number** and **Position** as follows:



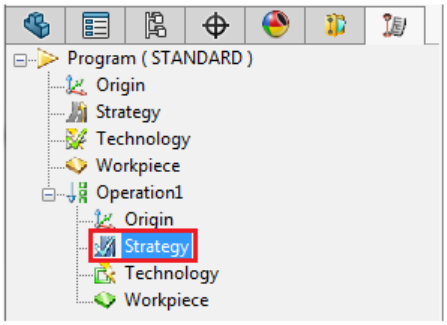
To identify the Secondary plane height in the model, position the cursor in the input field **Secondary plane height** and click the top face of the model. This passes the value on to the dialog box **Secondary plane height**.



Accept changes by clicking  .

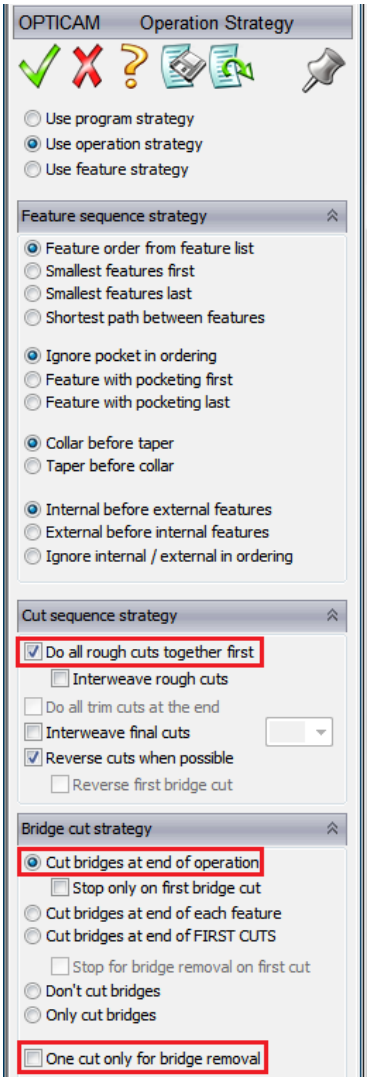
Setting the Cutting Strategy

To define with which strategy the workpiece will be machined with, double-click the branch **Strategy** in the OPTICAM Feature Manager.



The geometries are to be machined in such a way that **the Bridges are Cut at the End of the Operation**. The main cut and the 2 trim cuts are to be done in reverse cuts (forwards/backwards).

Set the dialog as follows:



Accept changes by clicking  .

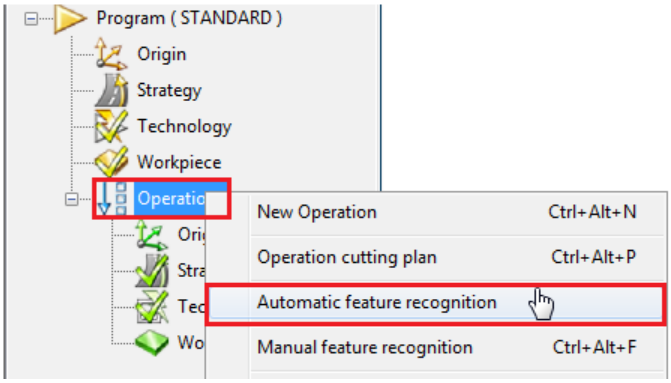
Creating the Cutting Machining

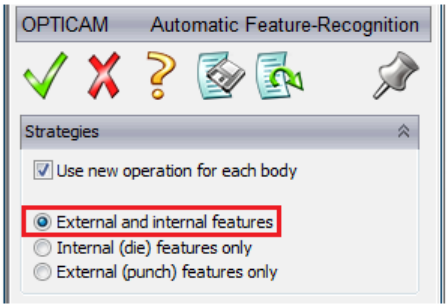
We want to machine the external feature and the tapered apertures.

User advice: The four cylindrical holes have to remain unmachined.

Automatic Creation of the Cutting Machining - Hiding Certain Geometry Types

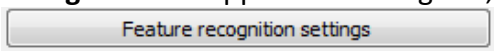
Right-click **Operation1** and, on the context menu, click **Automatic feature recognition**.



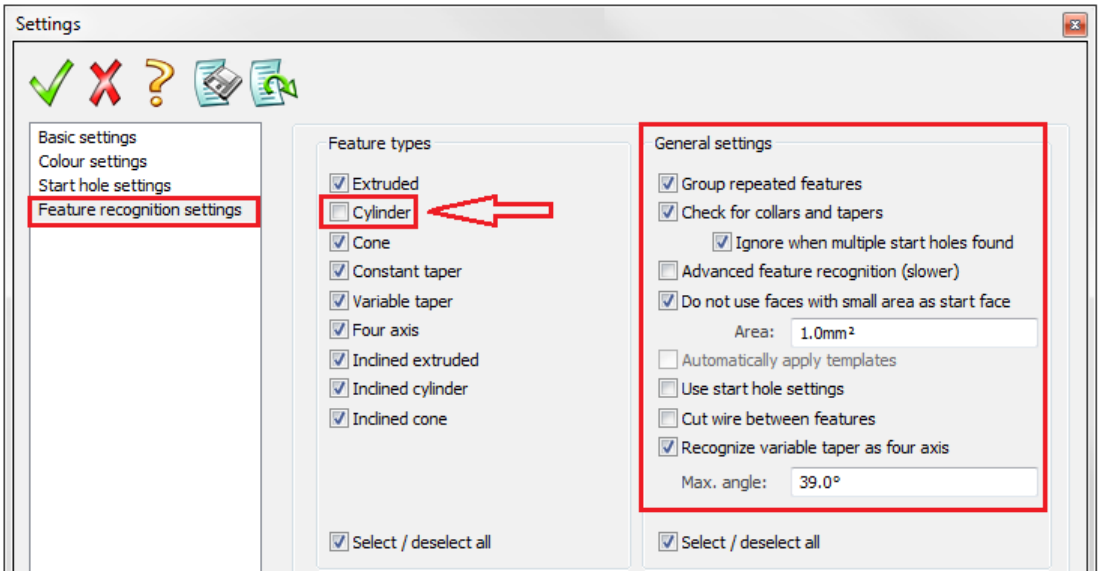


The **Automatic Feature Recognition** dialog opens. Because we want to machine the internal apertures *and* the external feature, set the dialog as follows:

In order to set what exactly the **Feature Recognition** is supposed to recognize, click the



button in the dialog.



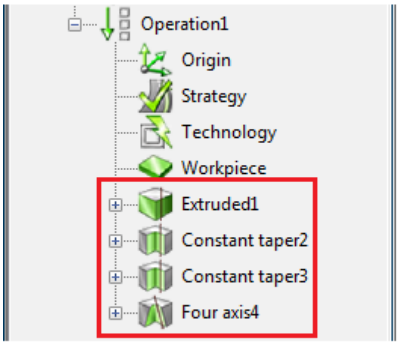
STOP **User advice:** The **Feature Recognition Settings** dialog has filters that let you exclude certain geometry types from the machining. To exclude the 4 cylindrical holes from the machining, you have to ***deactivate*** the option **Cylinder** (see figure above).

Accept changes by clicking  .

Close the dialog **Feature Recognition**.

The system will now search the solid body for all geometries that can be wire cut (feature recognition) and create a machining suggestion with the cutting parameters previously set.

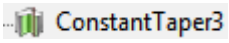
The tapered internal features and the external feature were automatically identified. Because we *deactivated* **Cylinder**, the four cylindrical holes remain unmachined, and the Feature Manager should display four features:



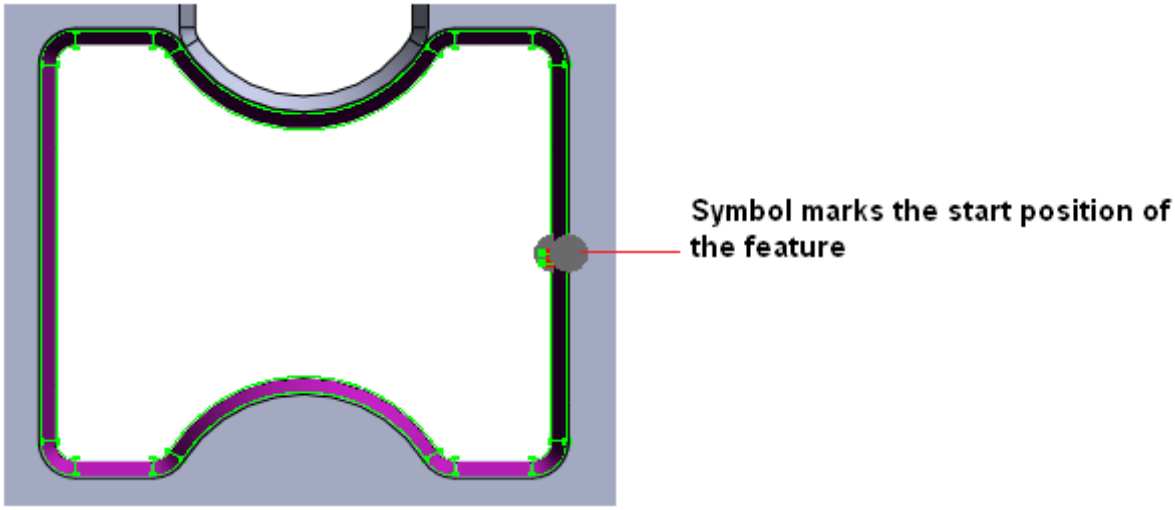
Checking the Created Features

The easiest way to check the created features is the visual check.

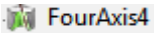
For this purpose, click on a feature in the OPTICAM Feature Manager. This highlights the cutting path belonging to the feature so that you can graphically check the start position of the feature, the number of cuts and the bridge.

Click the feature 

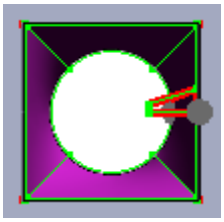
in the OPTICAM Feature Manager. The associated tool path is highlighted. The start position is marked by a symbol.



The visual check shows that position and size of the bridge are okay. However, you could shorten the length of the Lead off.

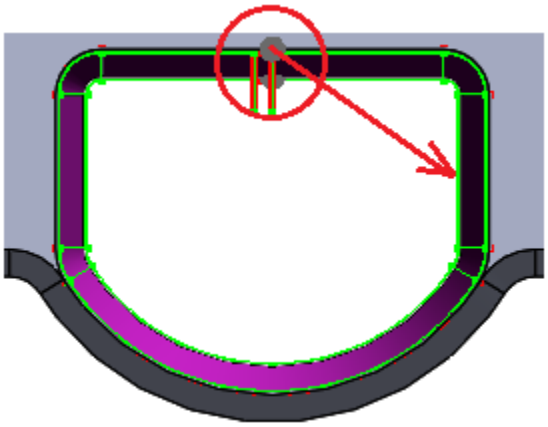
Click the feature 

in the OPTICAM Feature Manager.



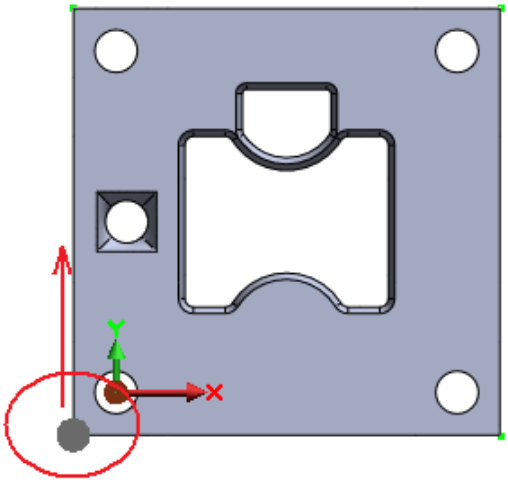
The visual check shows that position and size of the bridge are okay here as well. However, the position of the start hole is too close to the cutting feature and the length of the lead off should be shortened.

In Feature Manager, click the feature ConstantTaper2.



Although the start position is okay, for practicing purposes it should be moved from the long straight line to the center of the short straight line on the right-hand side. Additionally, here as well the position of the start hole should be moved further away from the cutting feature, and the length of the lead off should be shortened.

To check the external feature: In Feature Manager, click the feature Extruded1.

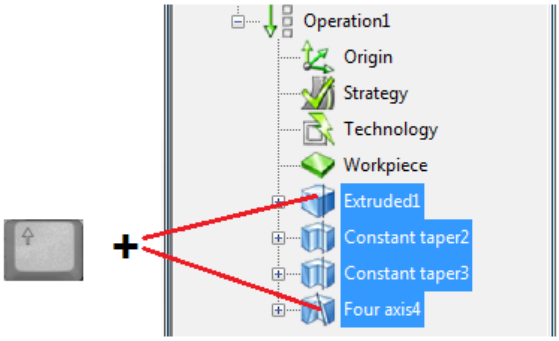


The start position is at the outside corner and should be moved to the center of the straight line. Additionally, the size of the bridge should be extended, the position of the start hole moved and the length of the lead off shortened.

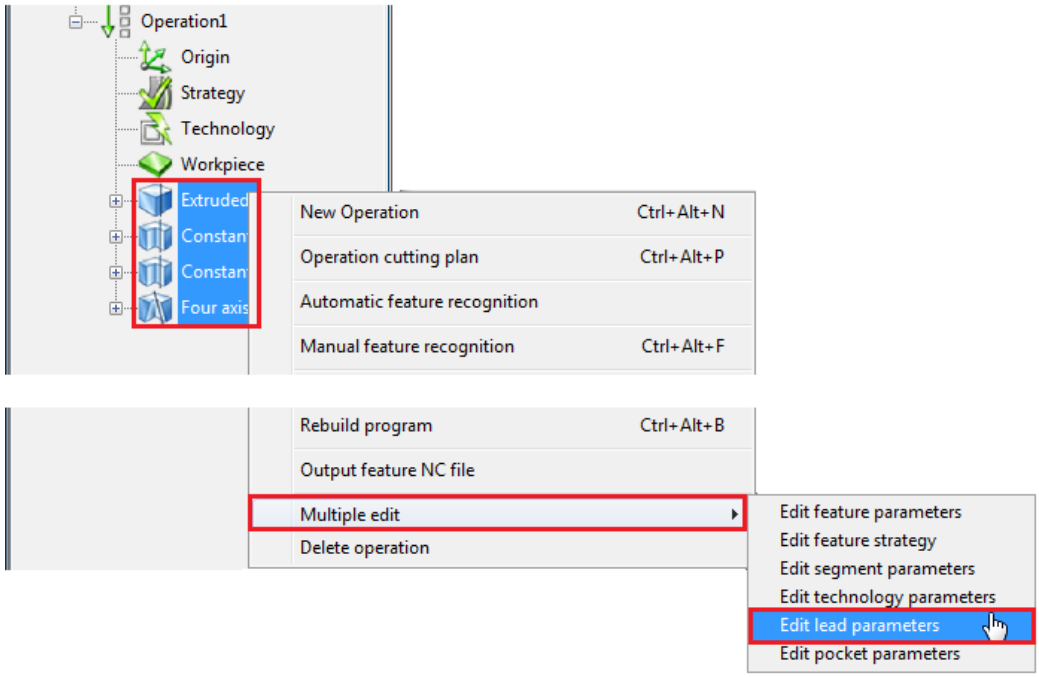
Modifying the Length of the Lead on/off for all Features

First, we modify the length of the Lead on to 5 mm and shorten the Lead off to 0.5 mm for all features.

Hold down the **Shift** button and click the **first** and the **last** feature in the OPTICAM Feature Manager. This way you can very easily select all 4 features.

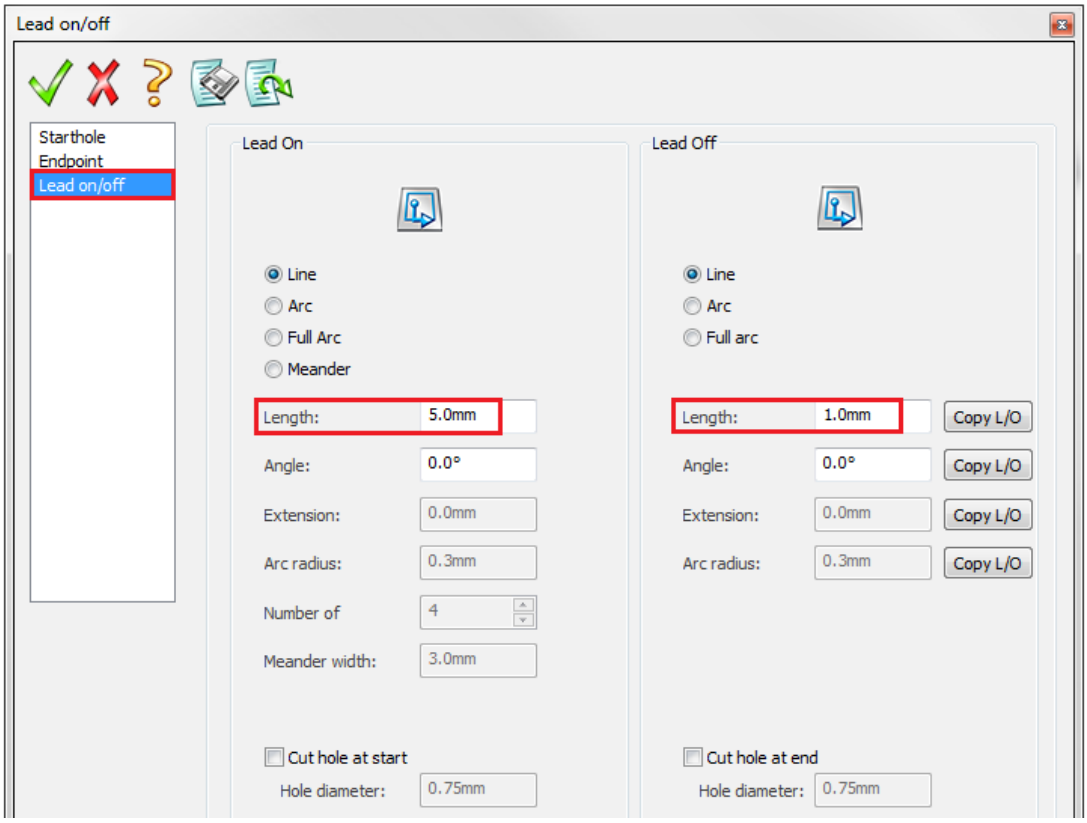


Now right-click one of the selected features and, on the context menu, click **Multiple edit > Edit lead parameters**.



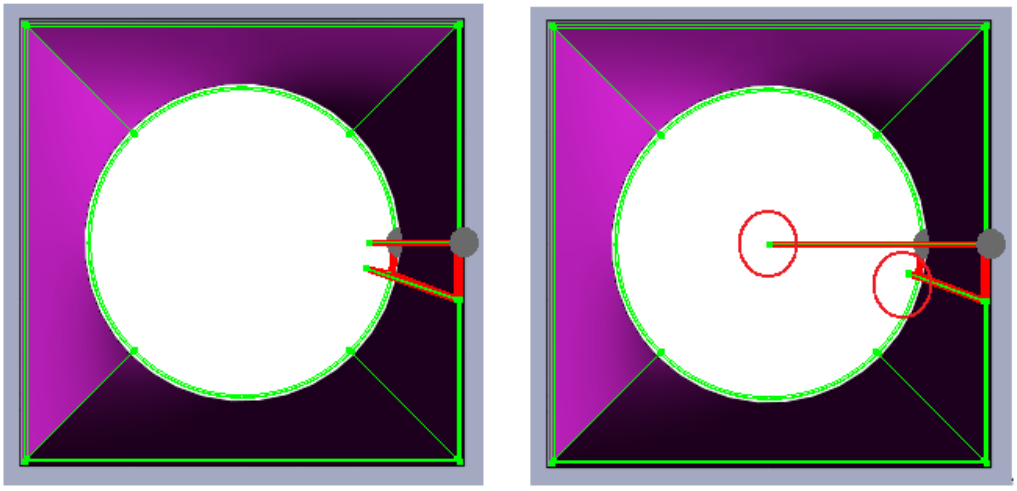
This opens the **Lead on/off** dialog.

On the left-hand side, click the feature **Lead on/off** and modify the length of the **Lead On** to **5** mm and of the **Lead Off** to **1** mm.



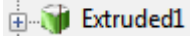
Accept changes by clicking  .

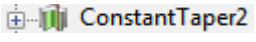
The adjustment of the lead on/off extends to all selected features.



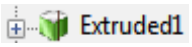
Before Afterwards

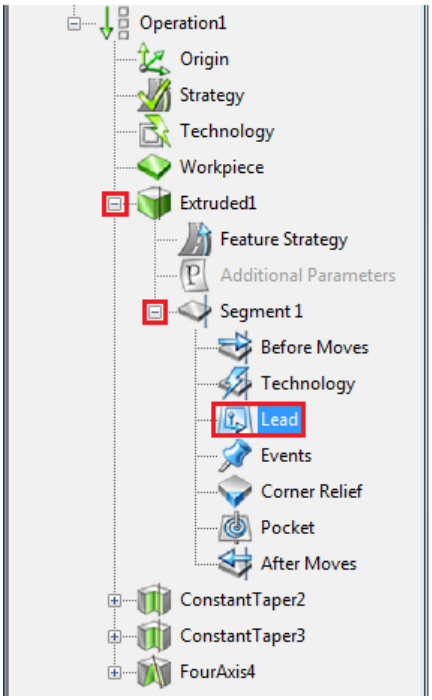
Positioning the Start Hole in the Center of a Straight Line

In the following, we will move the position of the start hole for the features 

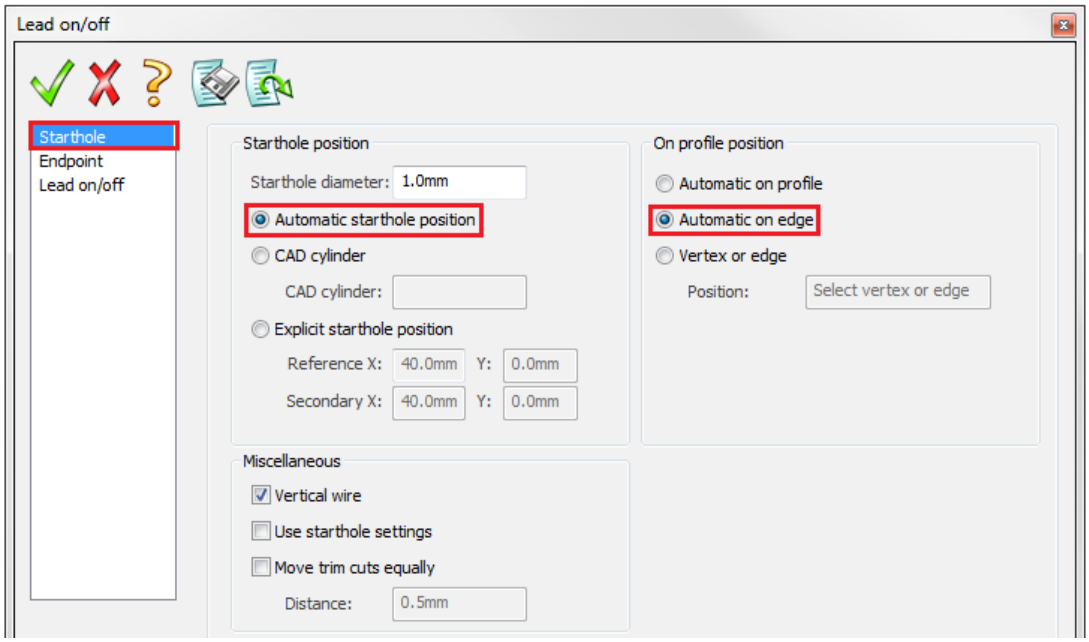
and 

to the center of a straight line.

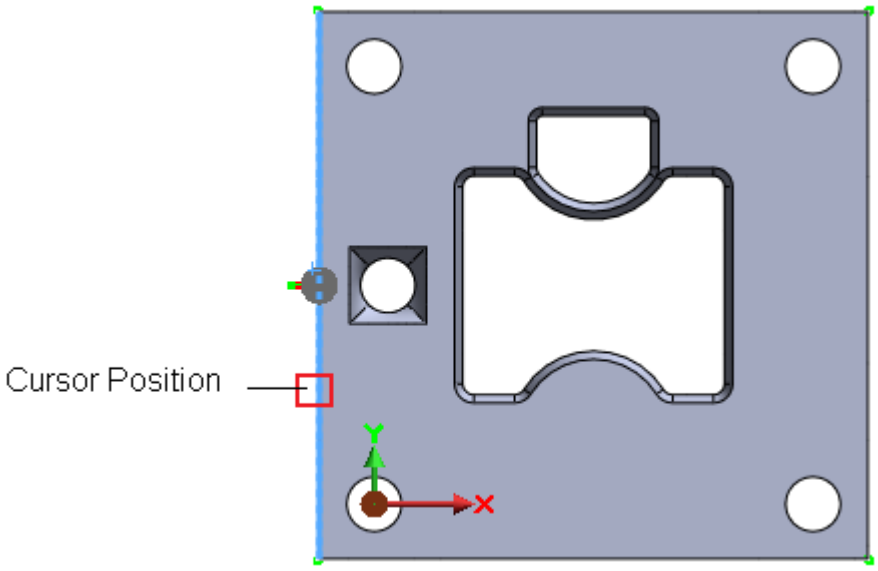
In Feature Manager**, o**pen the feature  and double-click the branch **Lead**.



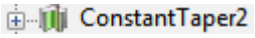
In the **Lead on/off** dialog, choose the option **Automatic on edge**. This option lets you automatically move the start position to the center of a straight edge. click the left outer edge of the workpiece to move the start position to the center of the selected edge.



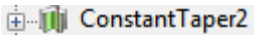
Note how the input modifies the start position and the position of the start hole:



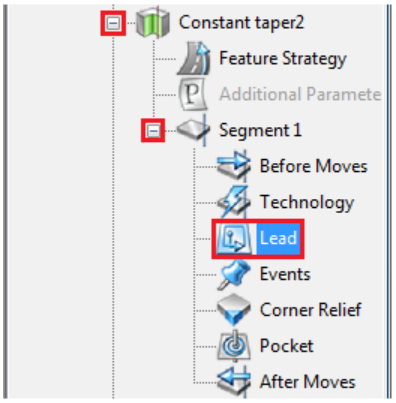
Accept changes by clicking on  .

Repeat the process for the adjustment of the start hole position for the feature 

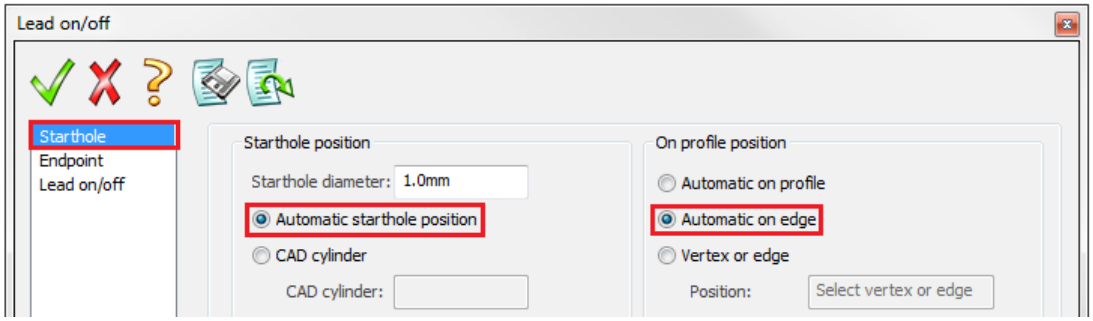
where the position of the start hole is to be moved to the center of the top straight line.

In the Feature Manager, open the feature 

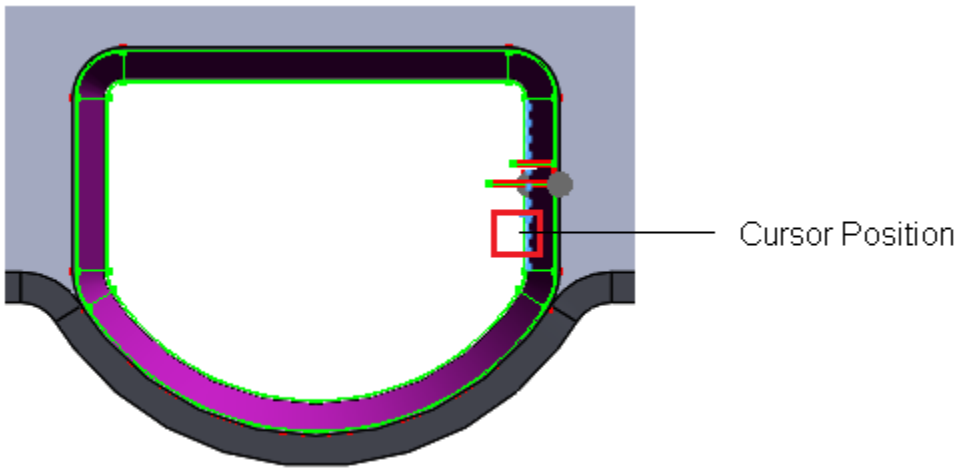
and double-click **Lead**.



Again, choose option **Automatic on edge** in the dialog **Lead on/off**. Click the straight left edge of the feature to move the start position to the center of the selected edge.



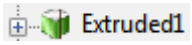
Note how the input modifies the start position and the position of the start hole:

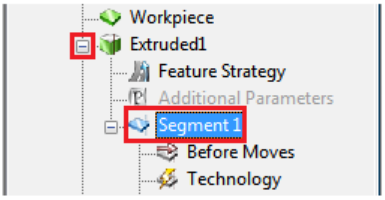


Accept changes by clicking  .

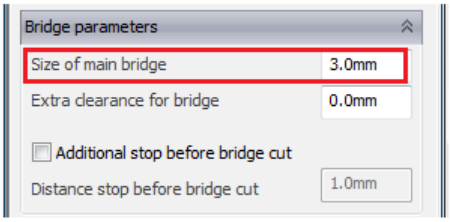
Adjusting the Size of the Bridge for the External Feature

As we need a wider bridge for the machining of the external feature, we will modify the size of the bridge to 3 mm.

In Feature Manager, open the feature  and double-click **Segment**.

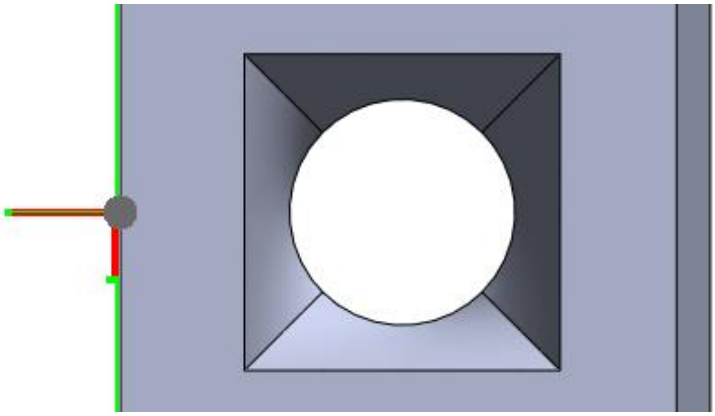


This opens a dialog where you can set the size of the bridge. Enter the value **3** in the field **Size of main bridge**.



Accept changes by clicking  .

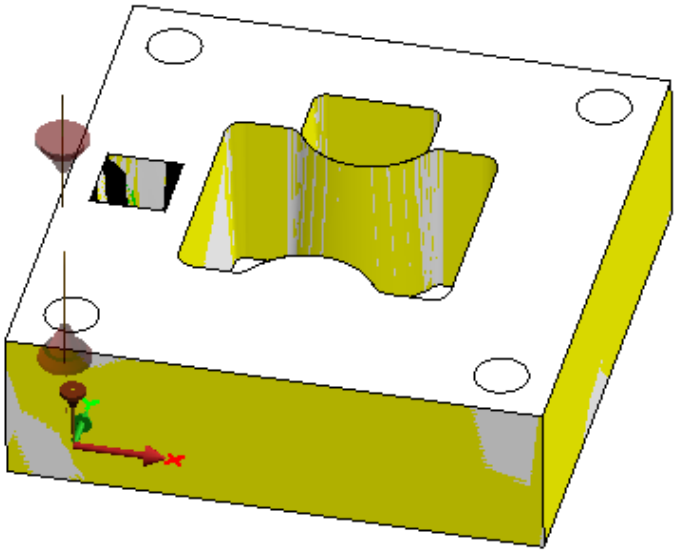
The following figure shows a magnification of the modified bridge.



Machining Simulation (Solid Simulation)

Right-click **Operation1** and, on the context menu, click **Solid simulation of operation**.

The following figure shows the final result of the simulation.

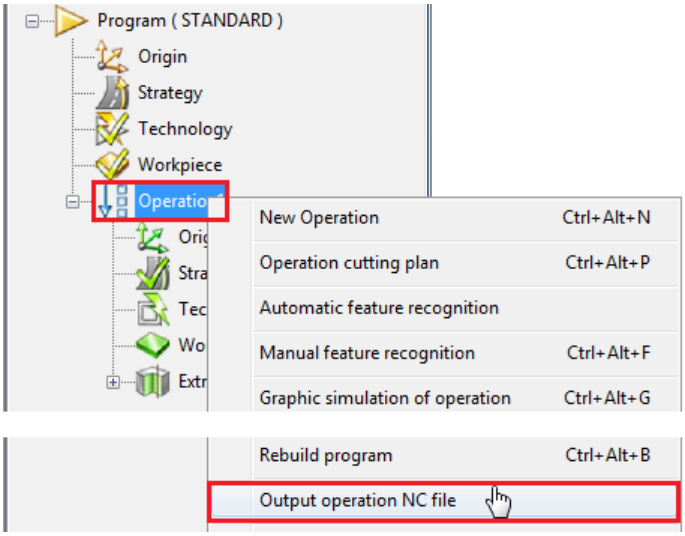


Close the simulation dialog by clicking  .

Creating the NC program

At the end of this exercise, we create the NC program again.

Right-click **Operation1** and, on the context menu, click **Output operation NC file**.



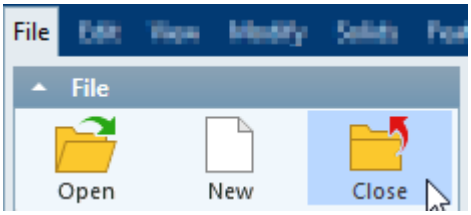
Please continue working independently…

Saving the OPTICAM Program

In the end, you should save your work.

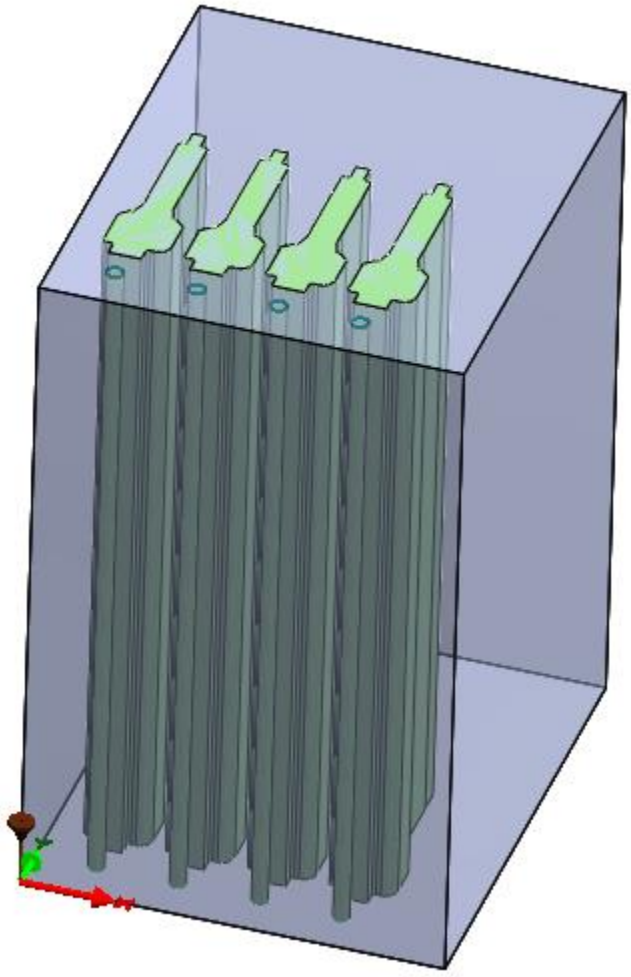
User advice: In addition to the GibbsCAM file **GC14_Opticam_example_6.vnc**, a second file containing the OPTICAM machining information for the current program will be saved. This OPTICAM file has the file extension .**wire.opticam**.

Close the file **GC14_Opticam_example_6.vnc**.



This exercise is now completed.

Example 7: Machining Punches and Multiple Punches



Demonstrated in This Example

Opening a file: GC14_Opticam_example_7

Deriving the Workpiece (Billet) Geometry from the Existing Model

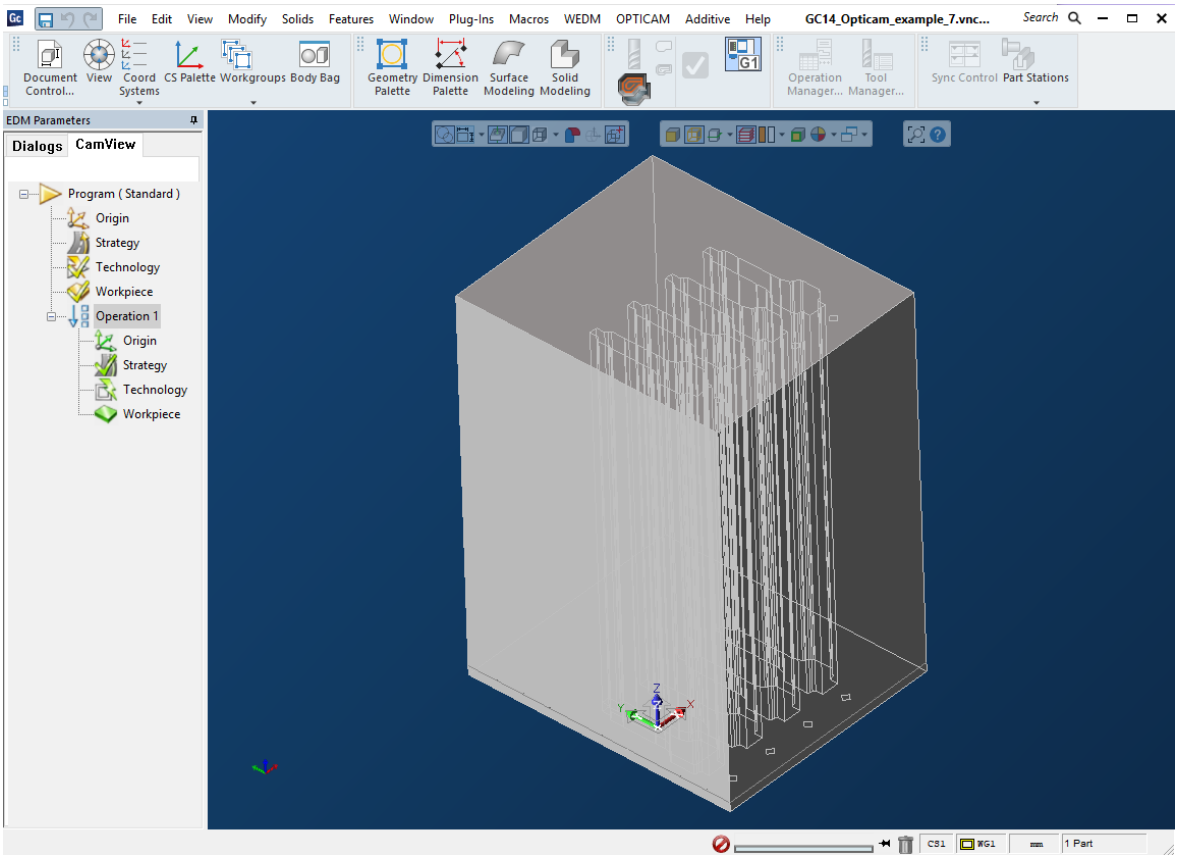
Hiding the Solid Body

Machining Punches / Multiple Punches

Modifying the Feature Order

Open GC14_Opticam_example_7

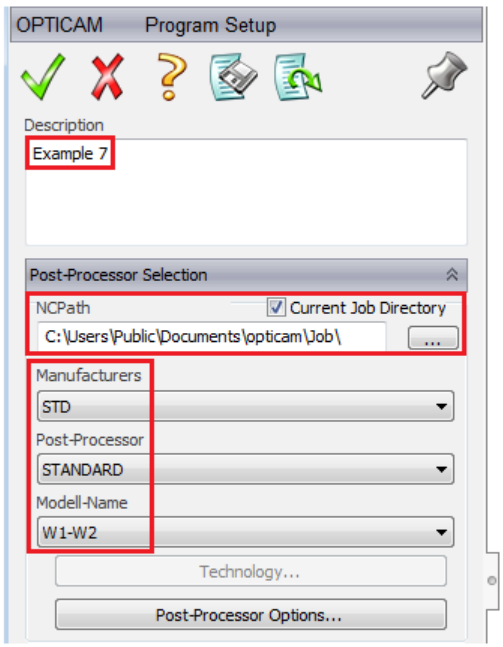
In GibbsCAM, open the example file **GC14_Opticam_example_7.vnc**.



Using the Program Dialog for Basic Settings

Double-click **Program (STANDARD)** and fill in the dialog as shown to the right:

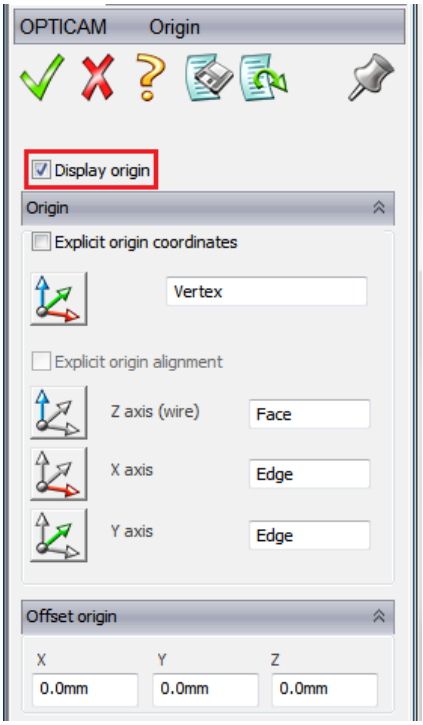
Accept changes by clicking  .



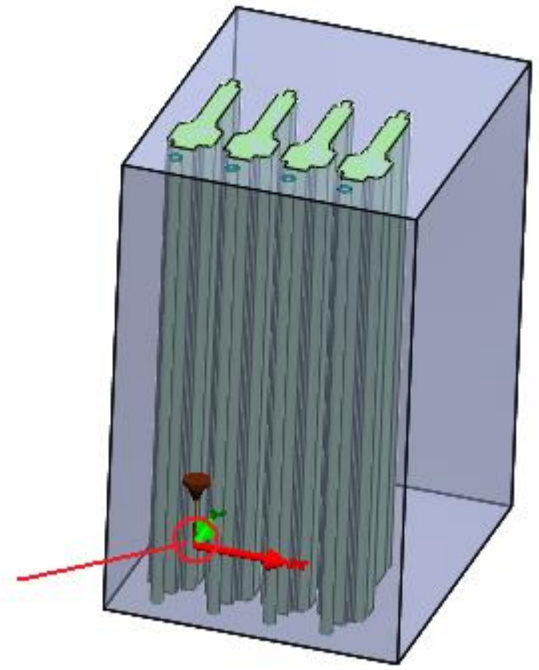
Defining the Machine Origin

In this example, the origin is to be placed at the lower left edge of the model.

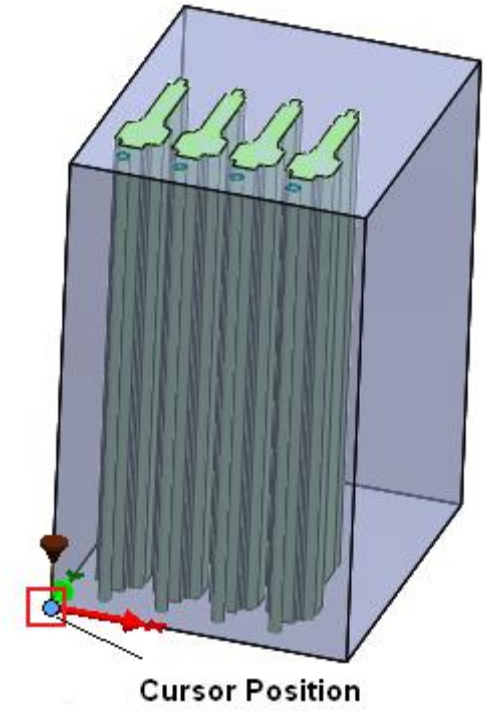
Double-click **(Program) Origin** to open the dialog to define the machine origin.



Currently, the machine origin is located at an undefined position.



To move the origin to the lower left corner of the center, position the cursor on the lower left corner of the model and click with the left mouse button.

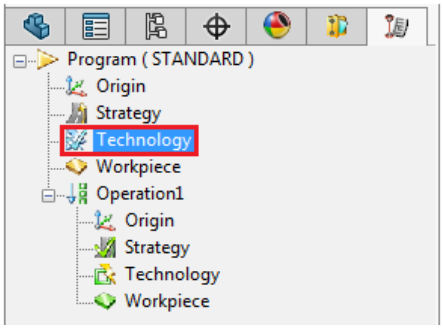


The origin is moved to the selected corner.

Close the dialog **Machine Origin** by clicking  .

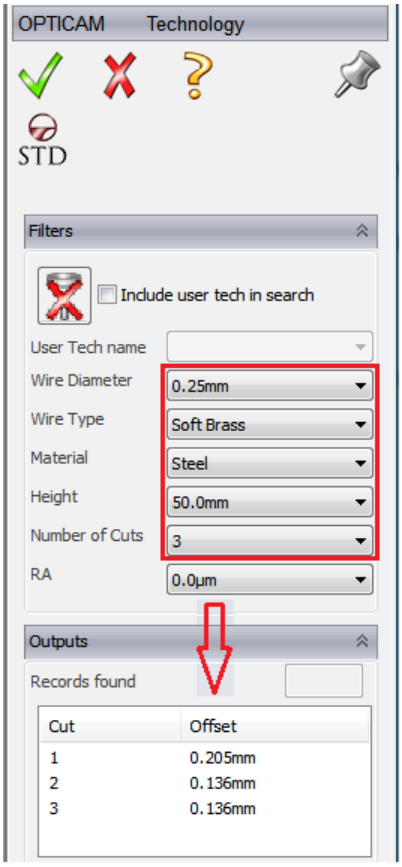
Defining the Cutting Technology (from Database)

Double-click the branch (Program) Technology in the OPTICAM Feature Manager.



This opens the **Technology** dialog to define the cutting technology. In this case, we will load cutting technology from the Technology Database, as follows.

Select suitable technology parameters from the listboxes **Wire Diameter**, **Wire Type**, **Material** and **Height**.

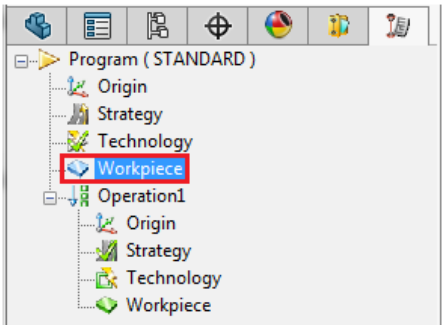


The selected cutting technology will be displayed in the dialog area **Records found**.

Accept the selection of the technology parameters by clicking  .

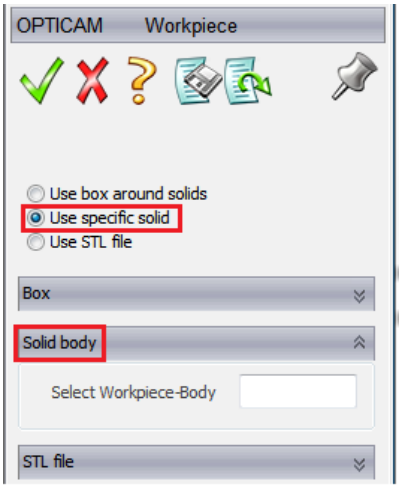
Deriving the Workpiece (Billet) Geometry from the Existing Model

In Feature Manager, double-click the branch **(Program) Workpiece**.

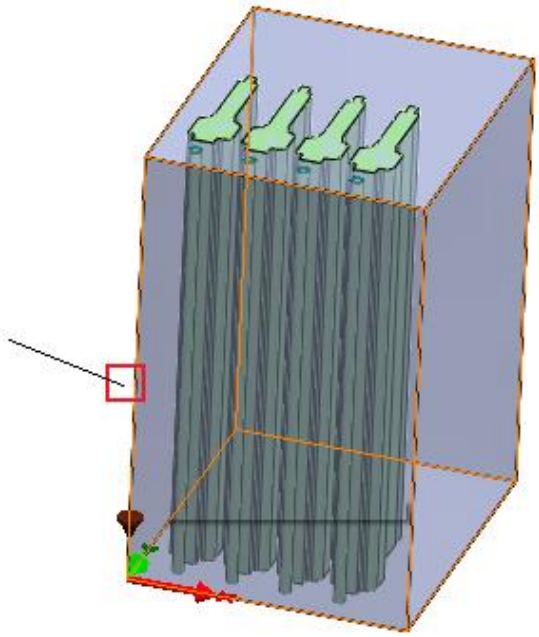


In the example file, the workpiece (billet) is predefined as a solid body.

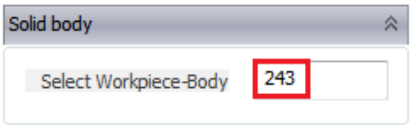
Choosing option **Use specific solid** activates the input field **Solid body**.



Click the geometry of the workpiece.



The name of the model is passed to the dialog.

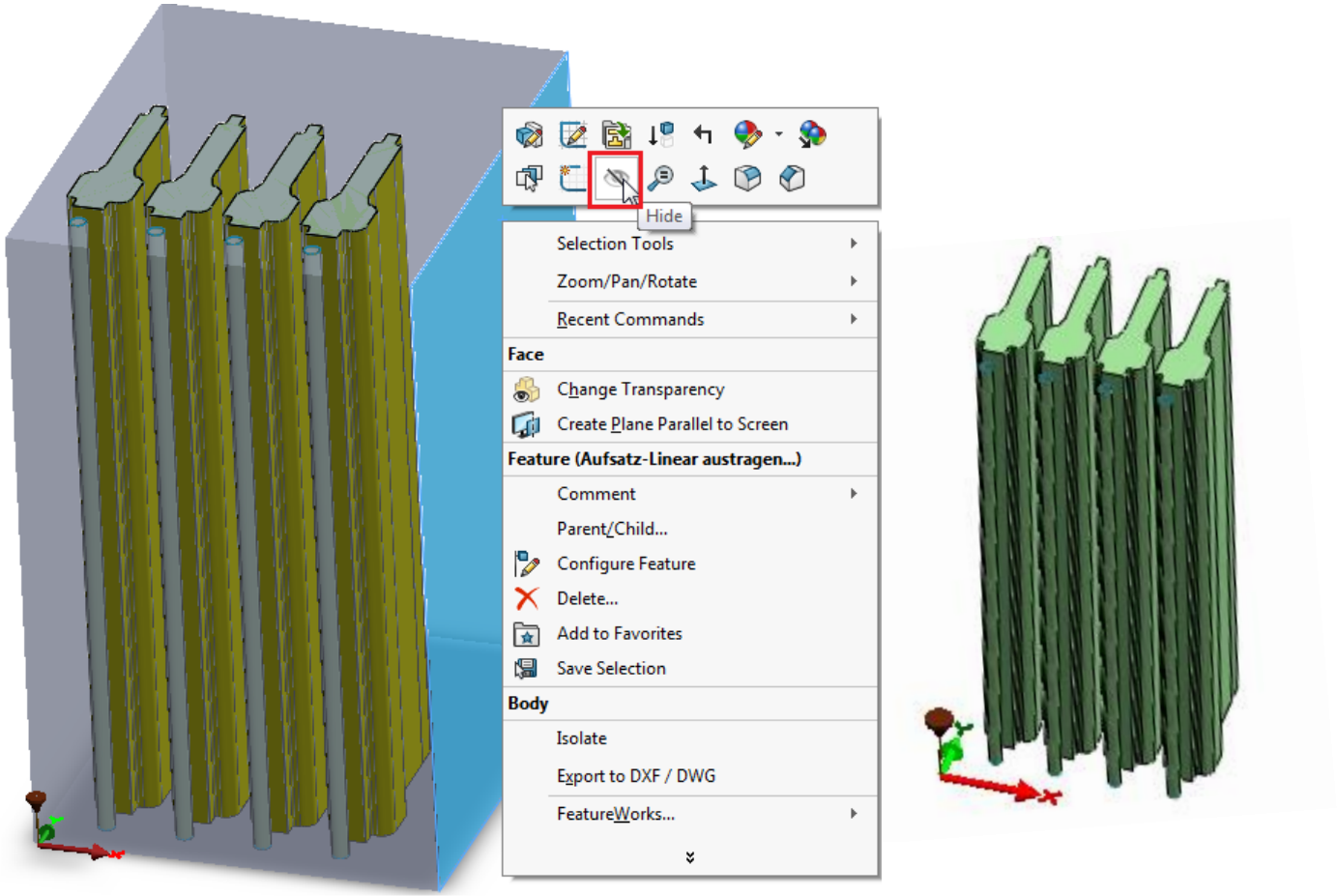


Accept changes by clicking  .

Hiding the Workpiece (Billet) Geometry

As the geometry of the workpiece (billet) would be identified and machined as a geometry by the automatic feature recognition, it is recommendable to hide the workpiece (billet) geometry.

Right-click the geometry of the workpiece (billet) and, on the context menu, click **Hide**.

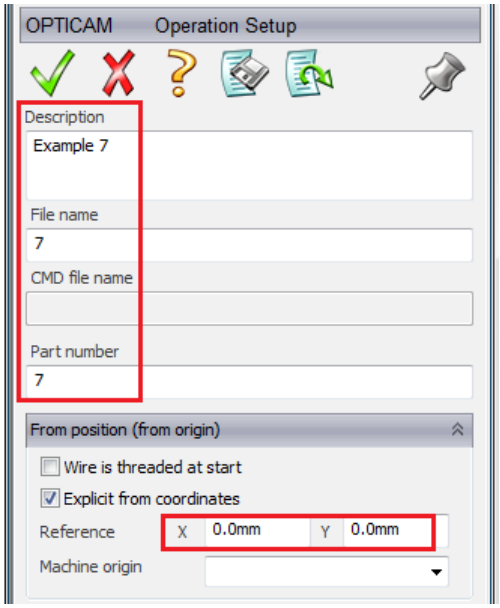


The model of the workpiece (billet) is hidden. The four punch geometries and the associated start holes remain.

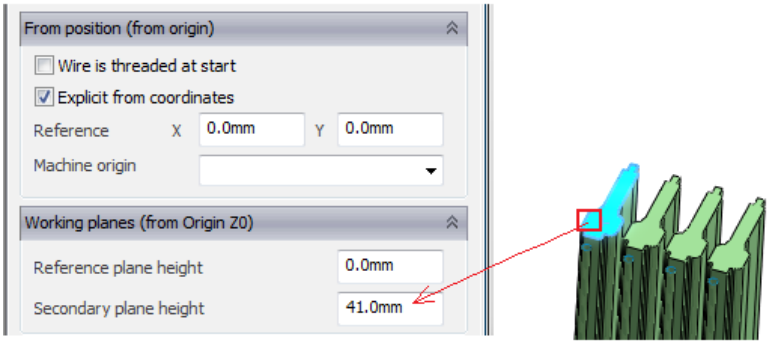
Setting the Cutting Heights

The following dialog sets (among other things) the name of the NC output file and the Z-height of the Reference and the Secondary Plane Height.

Double-click **Operation1** and fill in the fields **Description**, **File name**, **Part number** and **Position** as follows:



Identify the Secondary plane height in the model. To do this, position the cursor in the input field **Secondary plane height** and click an element (an edge, plane face or a vertex) on the top face of the model. OPTICAM calculates the Z-height of the selected element and passes the value on to the **Secondary plane height** dialog box.

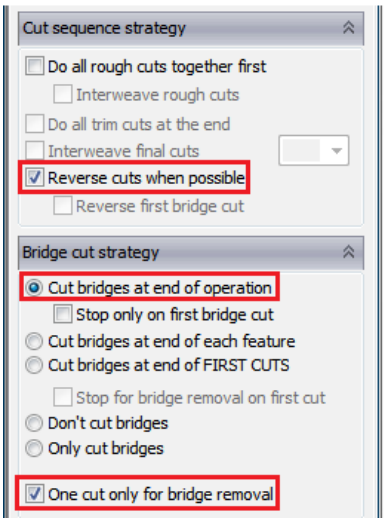


Accept changes by clicking  button.

Setting the Cutting Strategy

In the next dialog, you can define with which strategy the workpiece will be machined. The punches are to be machined individually, each one with 3 Cuts in Reverse Cuts (forwards/backwards). In the end, each Bridge is to be Removed with One Cut.

Double-click **(Operation) Strategy** and set the dialog as follows:



Accept changes by clicking on  .

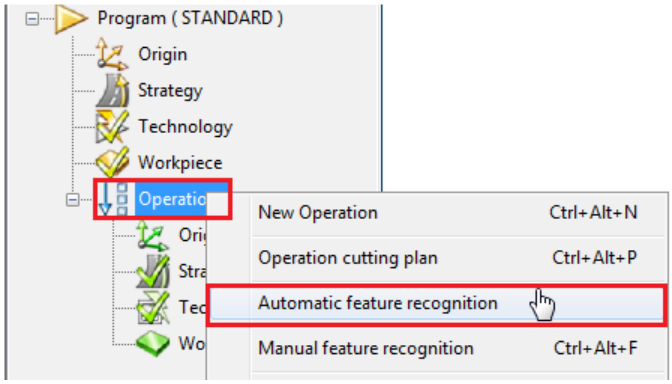
Creating the Cutting Machining

With OPTICAM it is possible to create the cutting machining automatically or by manually selecting faces or model edges.

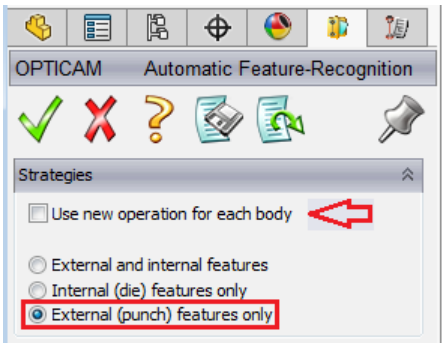
The 4 punches are to be machined automatically in this example. As the start holes exist as geometries, they will be automatically identified.

Automatic Creation of the Cutting Machining

Right-click **Operation1** and, on the context menu, click **Automatic feature recognition**.

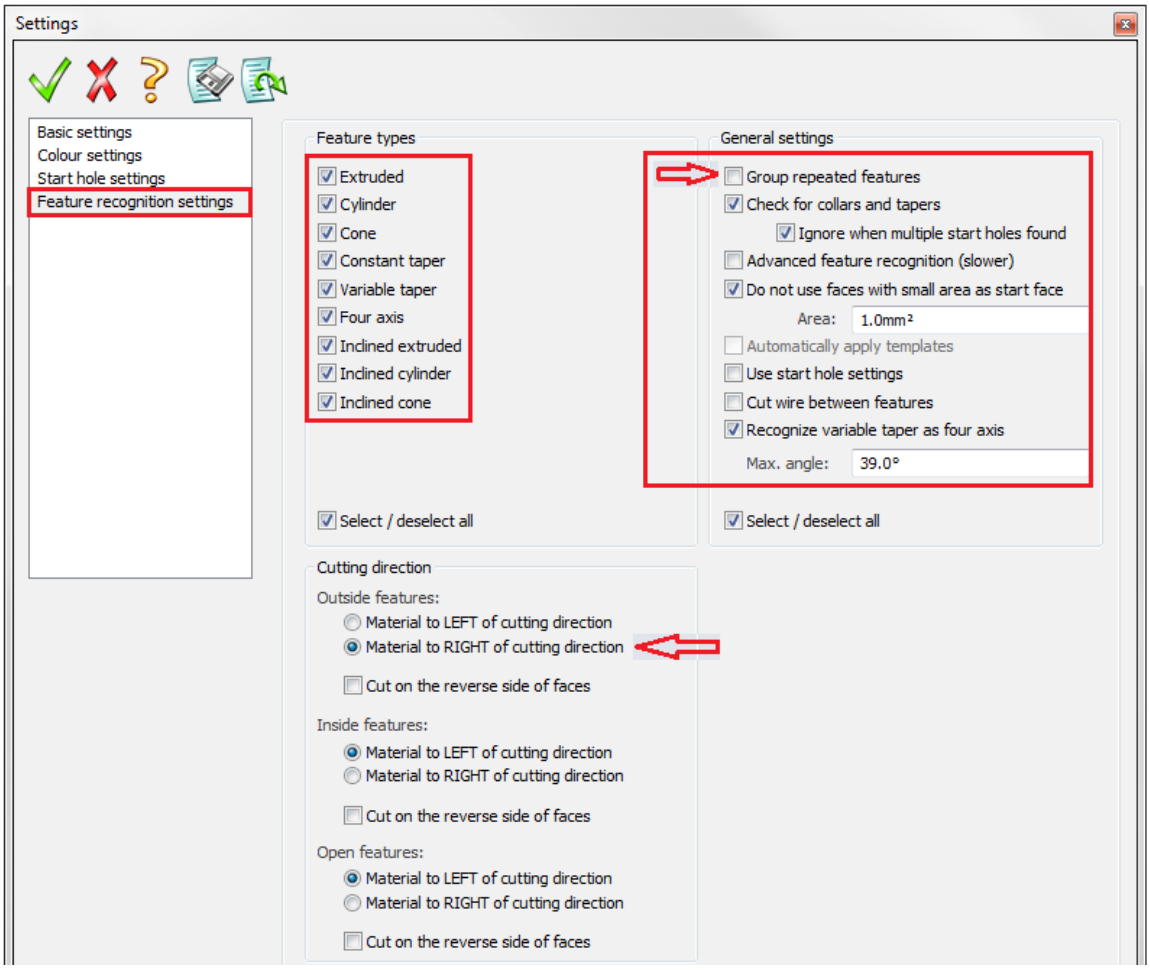


The dialog **Automatic Feature Recognition** opens. Because we want to exclusively machine the punches, choose **External (punch) features only** as the strategy.



Note: Do ***not*** select **Use new operation for each body**. This would create a separate **Operation** (separate NC file) in the OPTICAM Feature Manager for each geometry**.**

To set even more precisely what exactly the **Feature Recognition** is supposed to recognize, click the Feature recognition settings button.



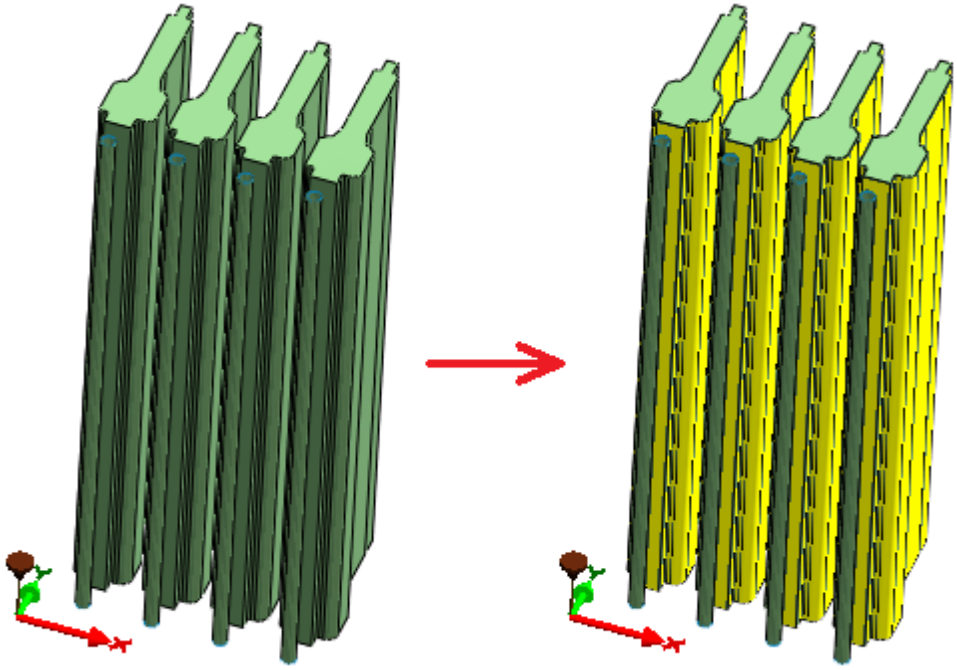
STOP **User advice:** Do ***not*** activate the option **Group repeated features**. Because the four punch geometries are identical, OPTICAM would unite the 4 punches into one feature in the OPTICAM Feature Manager. However, as we want to manipulate the feature order, we need 4 separate features in the OPTICAM Feature Manager.

User advice: Activate the option **Material to RIGHT of cutting direction**. This option defines the direction (clockwise or counterclockwise) in which our punches will be machined. **Material to RIGHT of cutting direction** means *clockwise*.

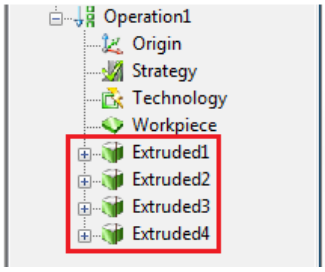
Accept changes by clicking  .

Close the dialog **Feature Recognition**:  .

The system will now search the solid bodies for all geometries that can be wire cut (feature recognition) and create a machining suggestion with the cutting parameters previously set.

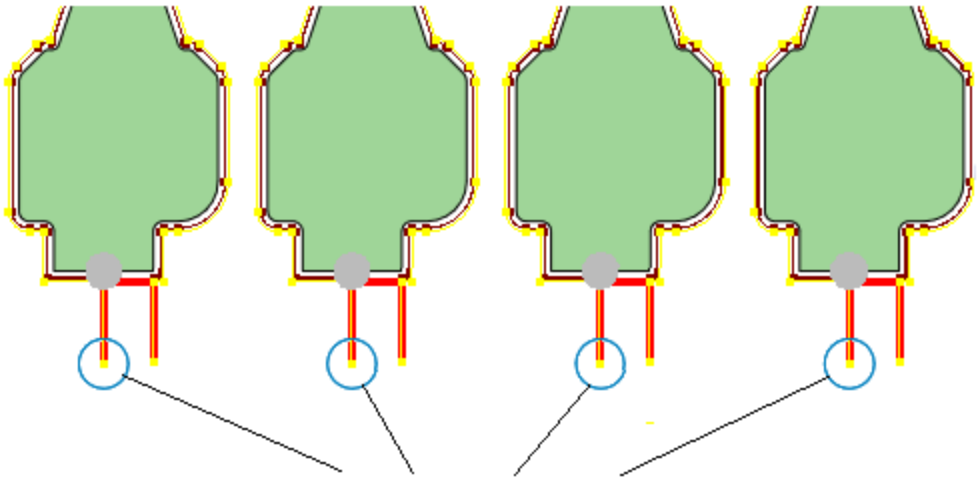


For each geometry that can be wire cut, OPTICAM creates a separate feature in the OPTICAM Feature Manager:



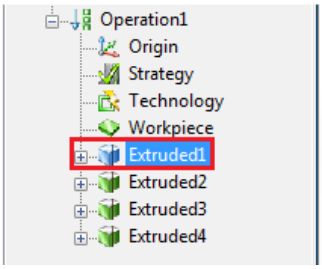
Automatic Start Hole Recognition

The OPTICAM feature recognition is able to identify start holes in a 3D model and to use them as threading position for the cutting machining. Start holes in the 3D model have to be drawn as GibbsCAM surfaces of the type Extruded Circle. As this is the case in this example, the start holes were automatically identified.

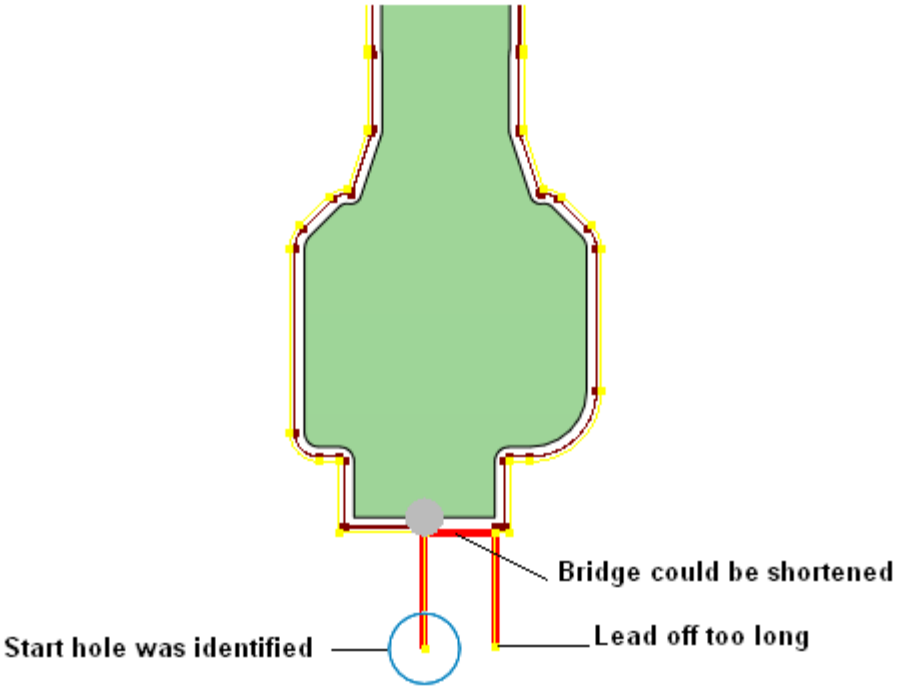


Adjusting the Lead on/off

Click the first feature in the OPTICAM Feature Manager. In the model, tool paths of the selected feature are highlighted.



If you extend the tool paths and look at them more closely, you will note several things. The start position is in the center of the cylinder. This is correct and the result of the automatic start hole recognition of the OPTICAM Feature Recognition.

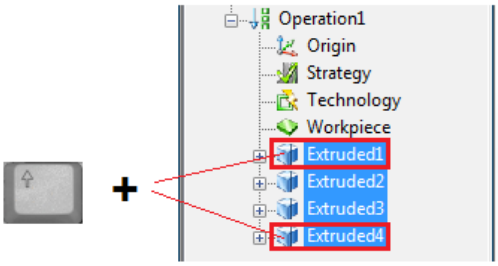


The lead off is too long and should be shortened. In the following, we will shorten the lead off.

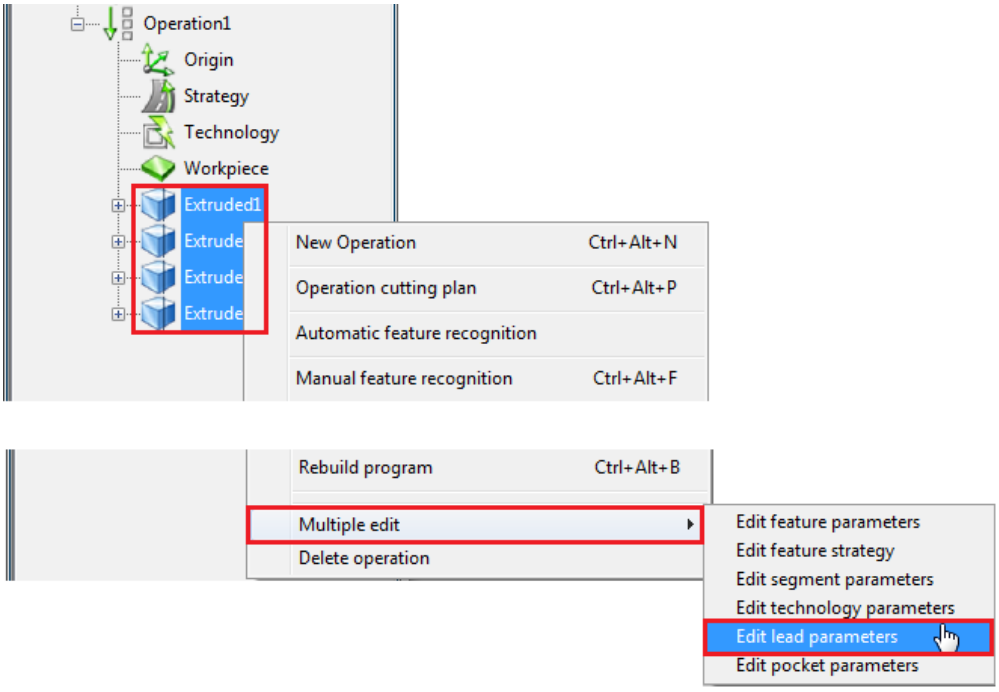
The **Size of the Bridge** is slightly too big and could be adjusted.

Shortening the Length of the Lead off

Holding down the **Shift** key, click the first feature and then the last feature in the OPTICAM Feature Manager. This way you can very easily select all four features.

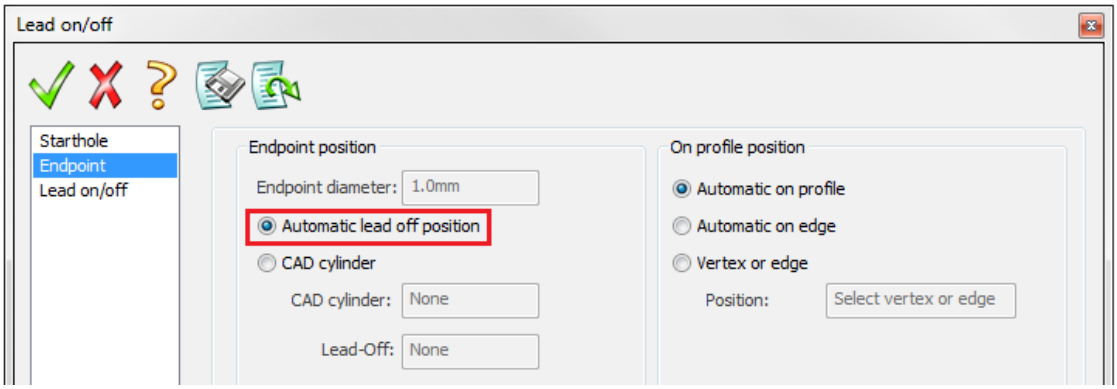


Now right-click one of the selected features and, on the context menu, click **Multiple edit > Edit lead parameters**.

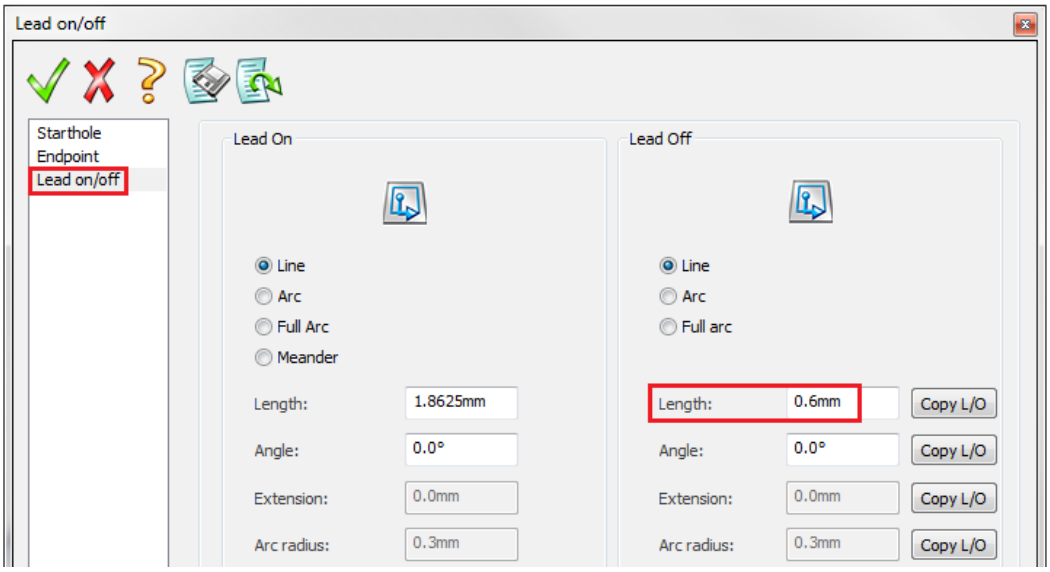


This opens the **Lead on/off** dialog.

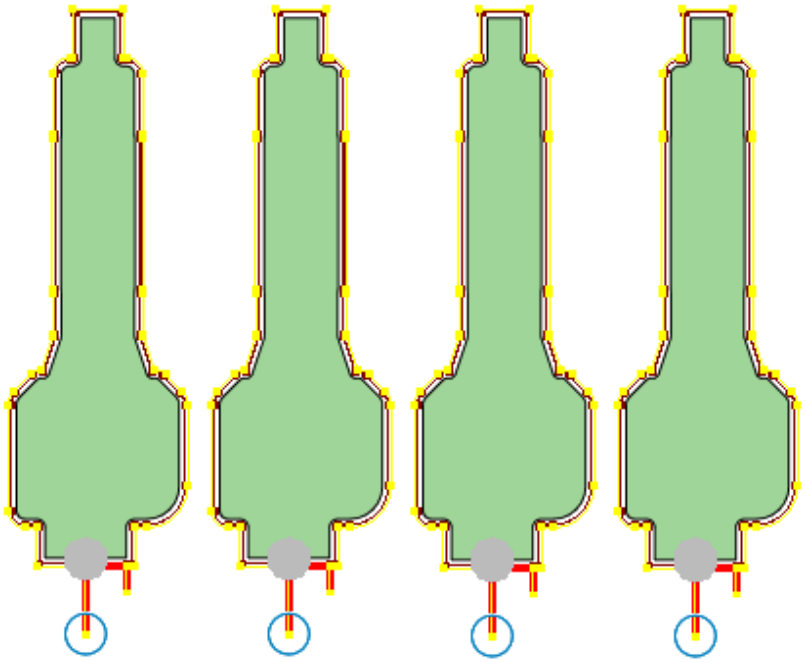
On the left-hand side, click **Endpoint** and choose option **Automatic lead off position**.



Click **Lead on/off** and modify the length of the **Lead Off** to **0.6** mm.

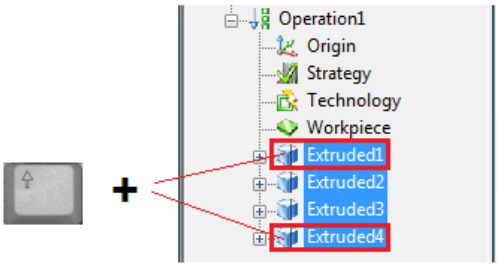


Accept changes by clicking  . The lead off adjustment is applied to all selected features.

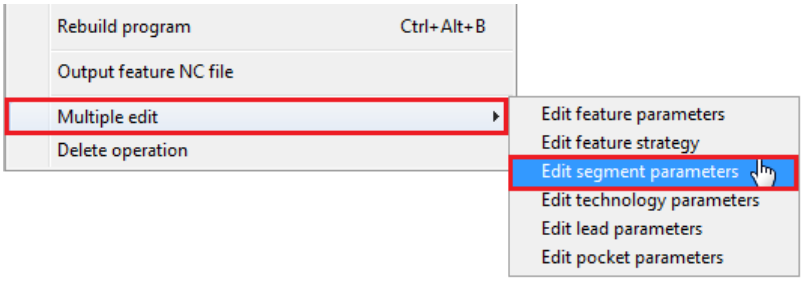


Adjusting the Size of the Bridges

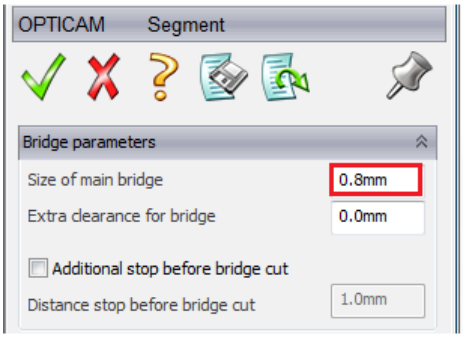
To select all four features, click the first feature and then hold down the **Shift** key while clicking the last feature in the OPTICAM Feature Manager.



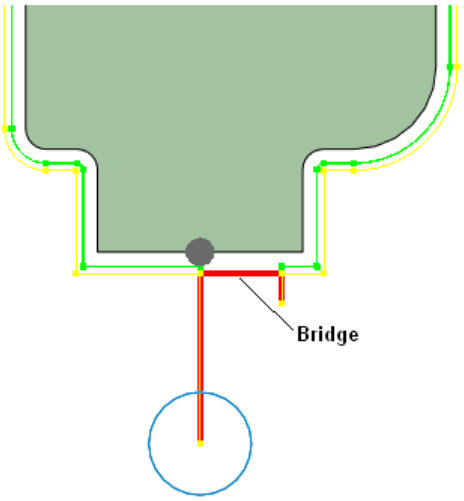
Now right-click one of the selected features and, on the context menu, click **Multiple edit > Edit segment parameters**.



In the **Segment** dialog, modify the **Size of main bridge** from 1 mm to **0.8** mm.



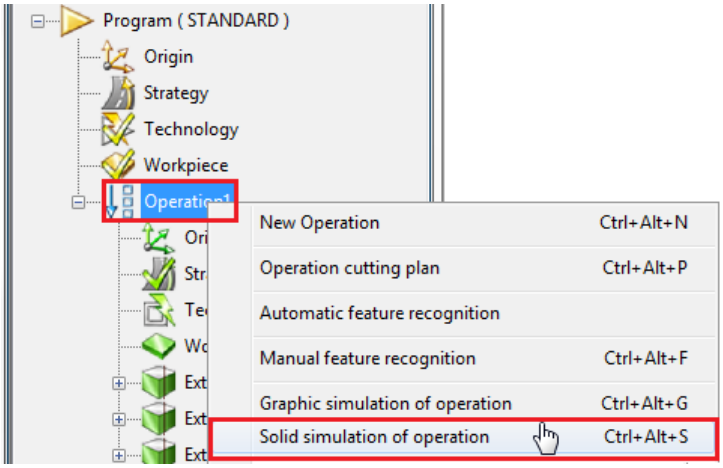
The size of all bridges is adjusted to **0.8** mm.
The following figure shows a magnification of the adjusted bridge:



Accept changes by clicking  .

Machining Simulation (Solid Simulation)

Right-click **Operation1** and, on the context menu, click **Solid simulation of operation**.

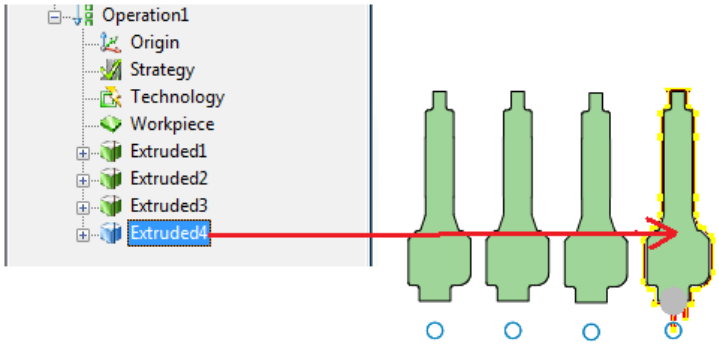


Modifying the Feature Order

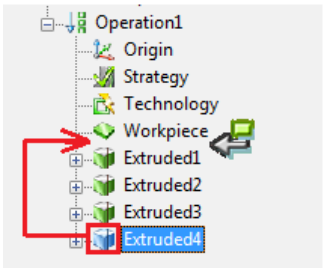
During the machining simulation it may happen that you do not like the cut sequence.

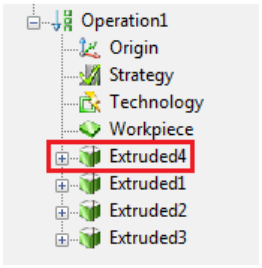
With the help of the OPTICAM Feature Manager, you can modify the cut sequence however you like, using drag-and-drop.

Click the last feature in the OPTICAM Feature Manager. The associated tool path graphic will be highlighted.

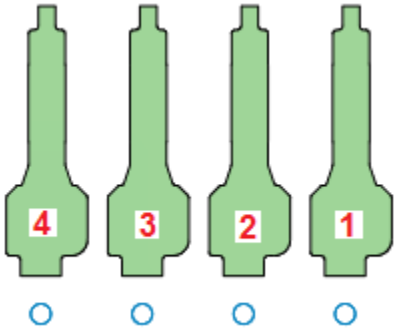


In the Feature Manager, left-click the symbol in front of the feature Extruded4. Hold down the left mouse button and position the feature between the feature Workpiece and Extruded1. Then release the mouse button.



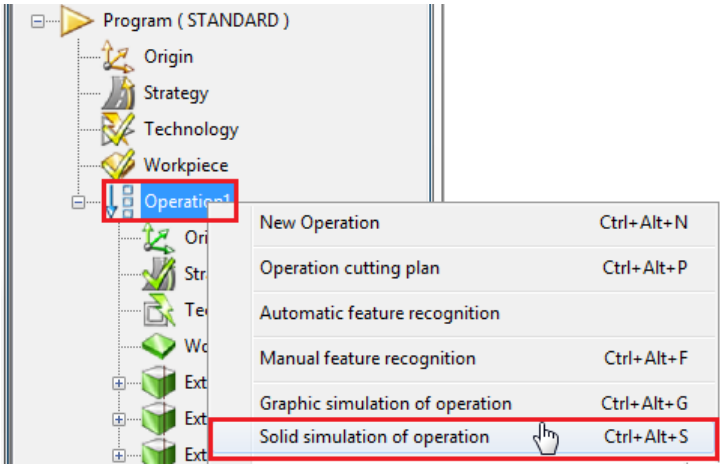


The last operation is moved to the top. Move the remaining features until the punches are machined from the left to the right side in the order **4,3,2,1** (see following figure).



Machining Simulation (Solid Simulation)

Check the adjusted feature order through a second **Solid Simulation**.

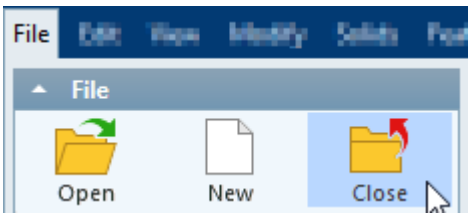


Saving the OPTICAM Program

At the end, you should save your work.

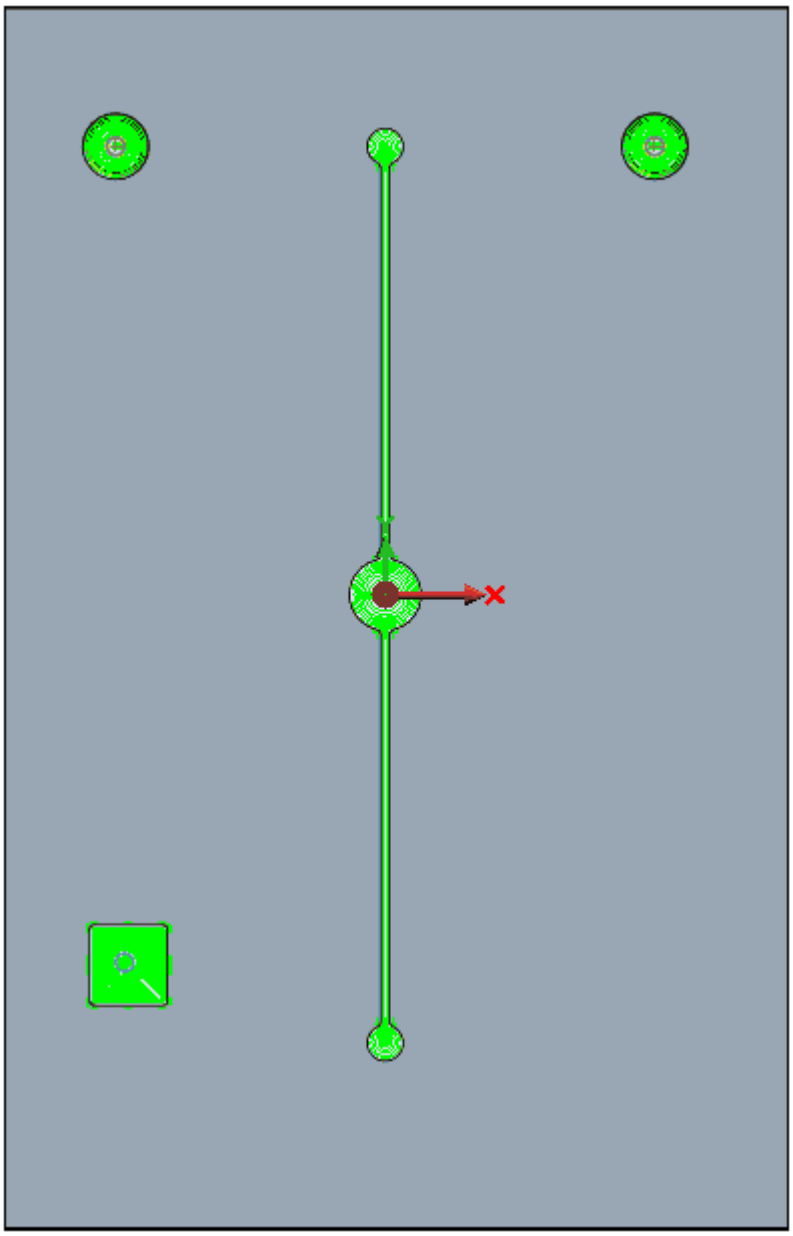
User advice: In addition to the GibbsCAM file **GC14_Opticam_example_7.vnc**, a second file containing the OPTICAM machining information for the current program will be saved. This OPTICAM file has the file extension .**wire.opticam**.

Close the file **GC14_Opticam_example_7.vnc**.



This exercise is now completed.

Example 8: Pocketing Apertures



Demonstrated in This Example

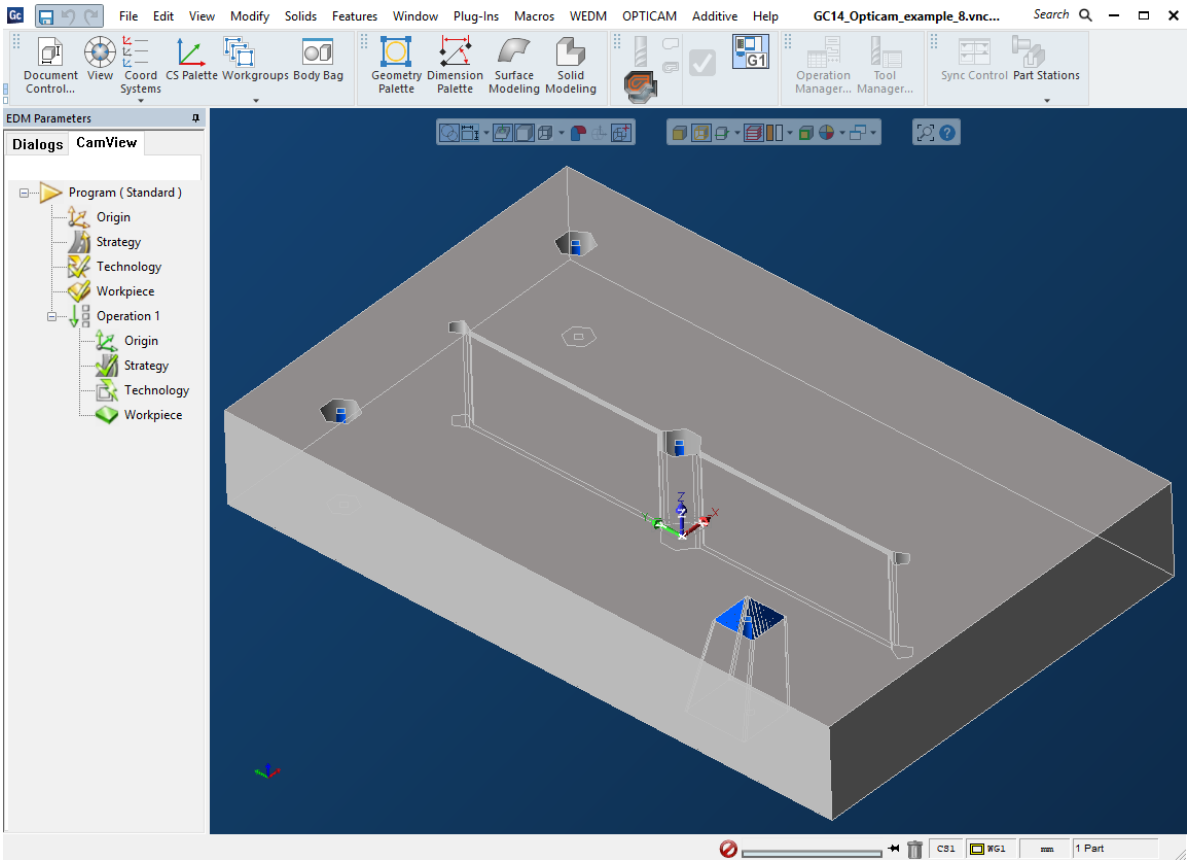
Opening a file: GC14_Opticam_example_8

Pocketing Cylindrical, Tapered and Four Axis Geometries

Turning on the Display of Slugs (Solid Simulation)

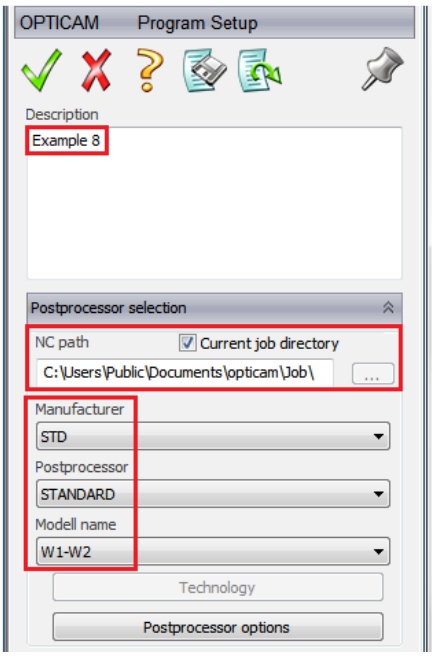
Open GC14_Opticam_example_8

In GibbsCAM, open the example file **GC14_Opticam_example_8.vnc**.



Using the Program Dialog for Basic Settings

Double-click **Program (STANDARD)** and fill in the dialog as shown to the right:

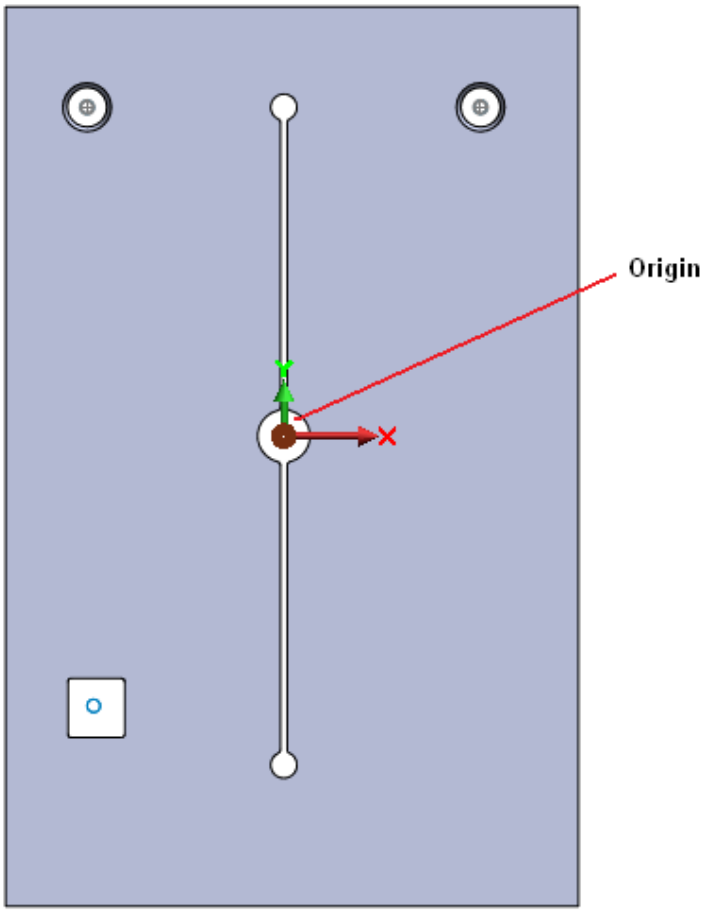


Accept changes by clicking  .

.

Defining the Machine Origin

In this example, the origin is in the center of the center of the workpiece. As this is okay, we do not have to define an origin.

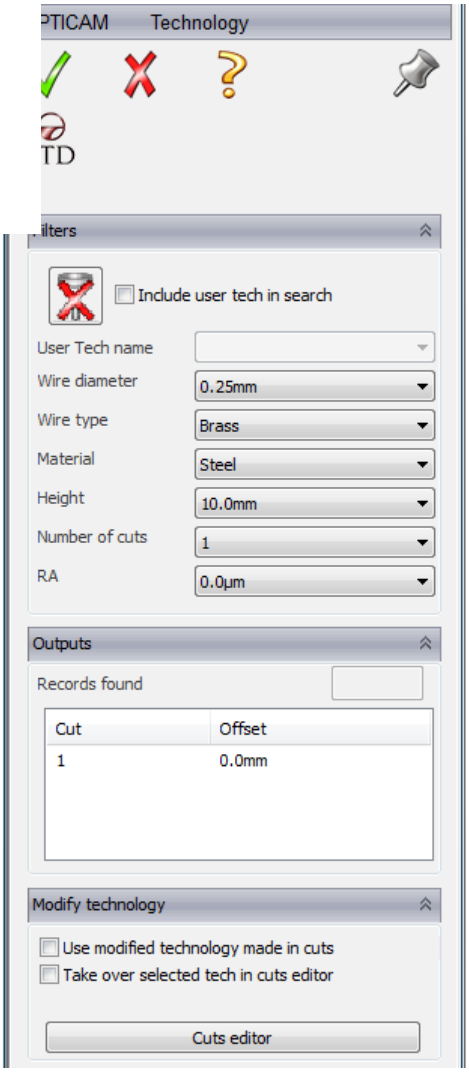


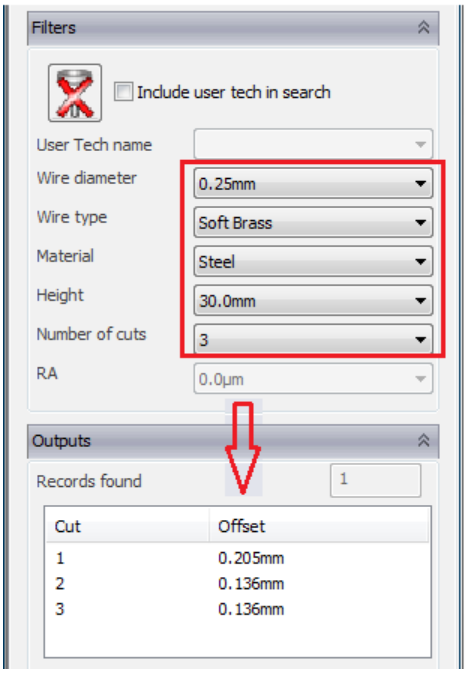
Defining the Cutting Technology (from Database)

Double-click **(Program) Technology** to open the **Technology** dialog to define the cutting technology.

We will select the cutting technology from the Technology Database.

Select suitable technology parameters from the listboxes **Wire Diameter**, **Wire Type**, **Material** and **Height**.





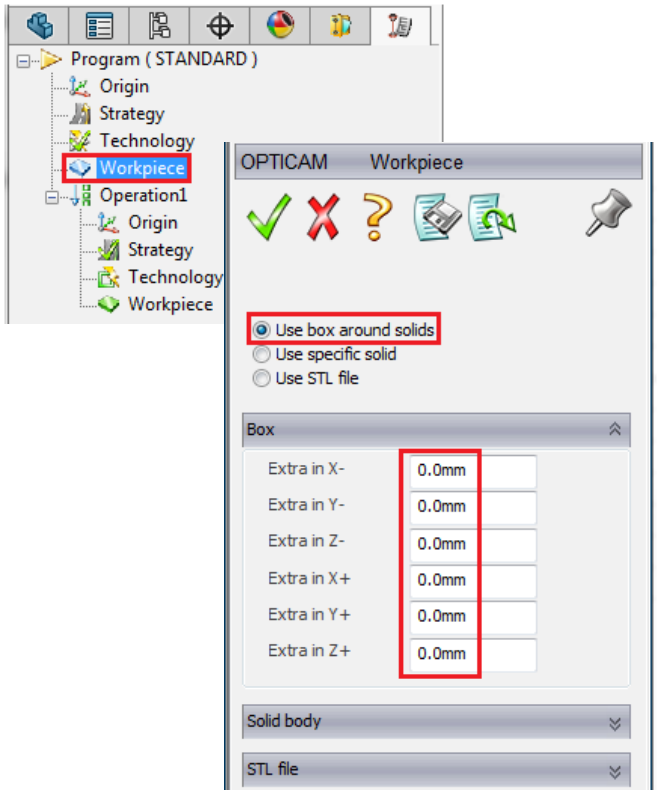
The selected cutting technology will be displayed in the dialog area **Records found**.

Accept the selection of the technology parameters by clicking  .

Creating the Workpiece (Billet) Geometry

In the next dialog, you can define the geometry of the workpiece (billet). The workpiece is to be derived from the solid body; additionally, no allowance values are to be defined.

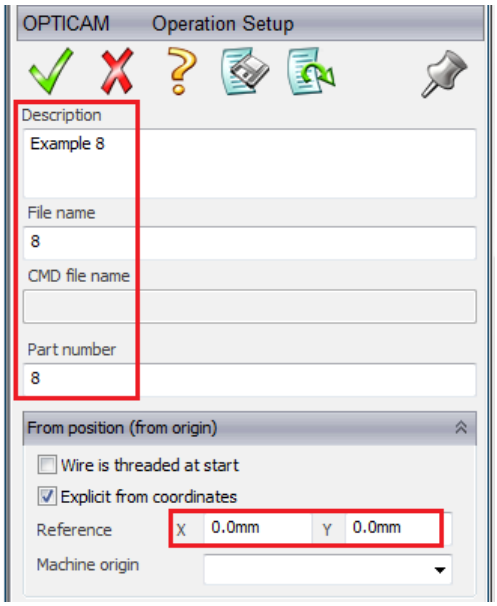
Double-click **(Program) Workpiece**:, and then fill in the dialog as shown to the right



Accept changes by clicking  .

Setting the Cutting Heights

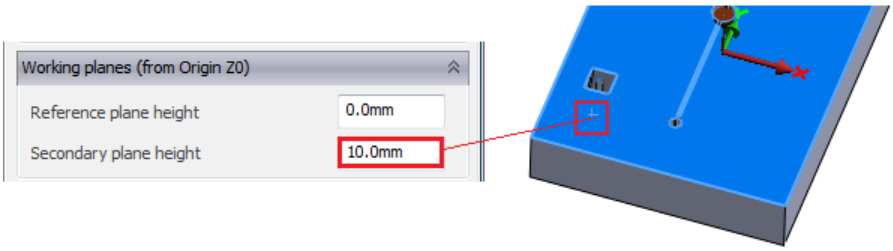
The following dialog sets the name of the NC output file and the Z-height of the Reference and the Secondary Plane Height.



Double-click **Operation1** and fill in the fields **Description**, **File name**, **Part number** and **Position** as shown to the right:

To identify the Secondary plane height in the model. Position the cursor in the input field **Secondary plane height** and click the top plane face of the model.

OPTICAM calculates the Z-height of the selected element and passes the value on to the dialog box **Secondary plane height**.

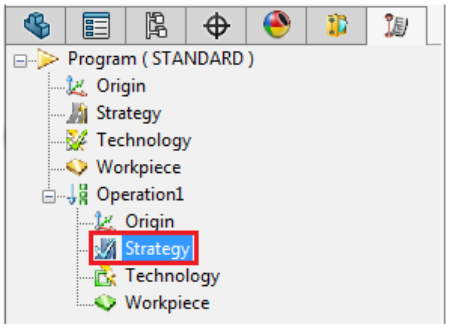


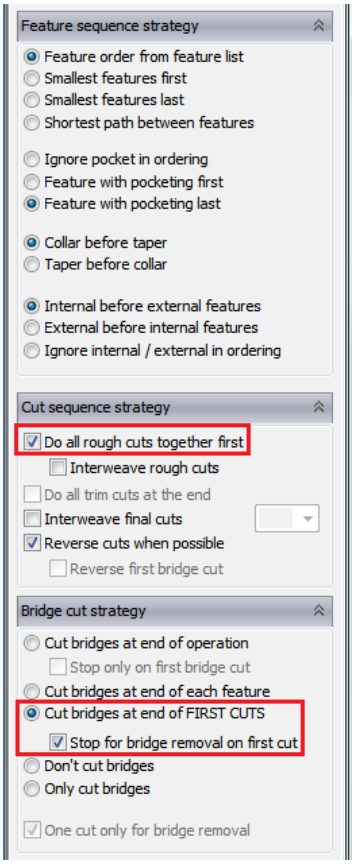
Accept changes by clicking  .

Setting the Cutting Strategy

In the next dialog, you can define with which strategy the workpiece will be machined. First, all apertures are to be roughed in succession. The trim cuts follow after.

Double-click **(Operation) Strategy** and then set the dialog as shown to the far right.





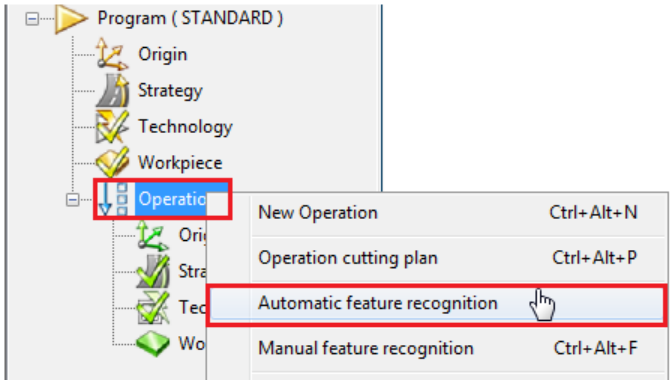
Accept changes by clicking  .

Creating the Cutting Machining

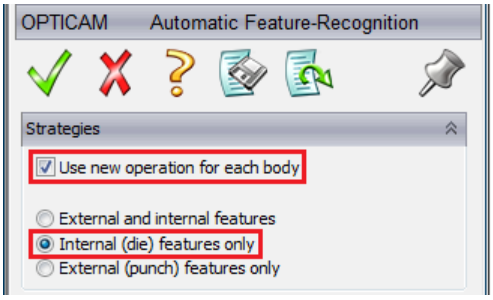
In this example, the four internal apertures are machined with the automatic Feature Recognition. As the start holes exist as geometries, they will be automatically identified.

Automatic Creation of the Cutting Machining

Right-click **Operation1** and, on the context menu, click **Automatic feature recognition**.

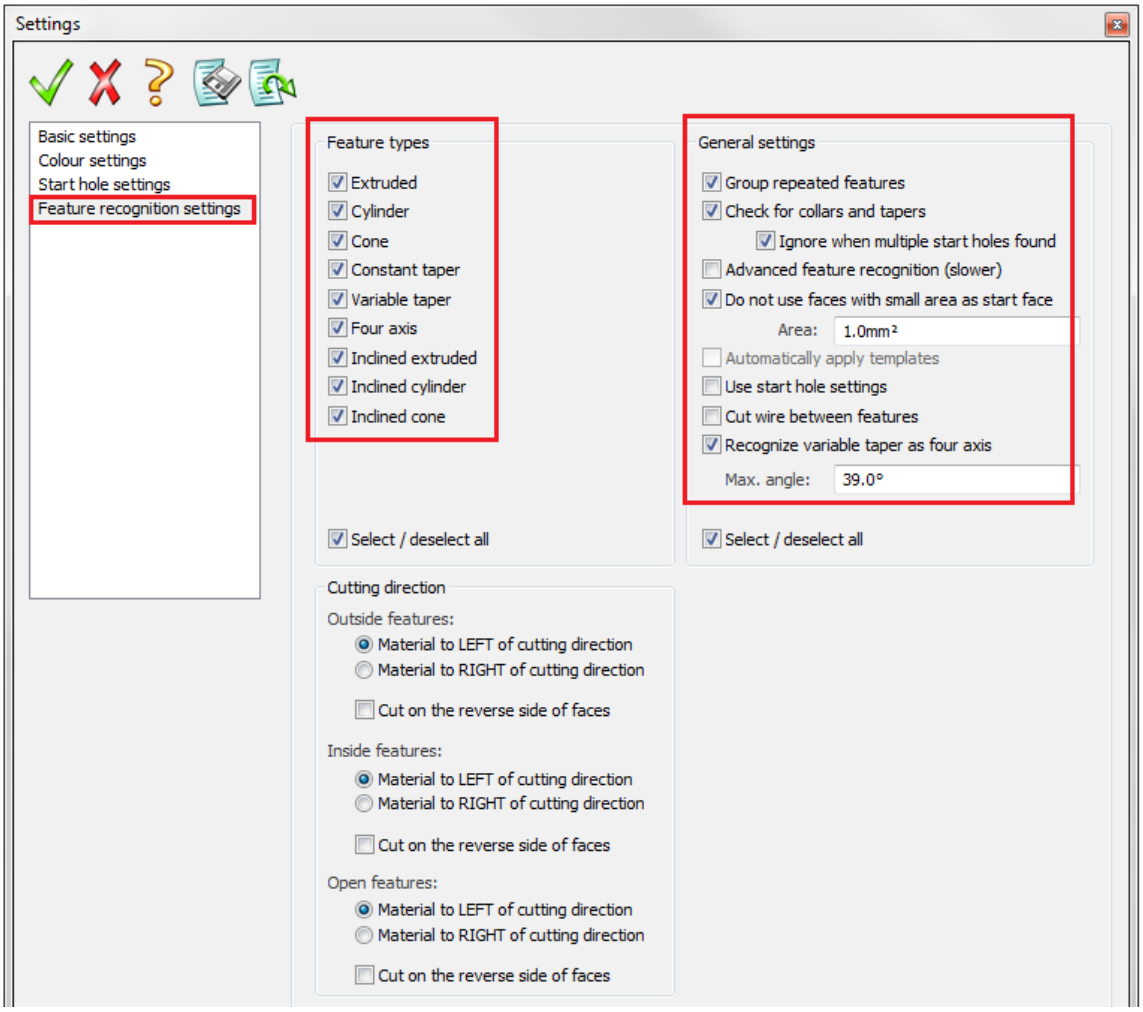


Set the **Automatic Feature Recognition** dialog as shown:



To set even more precisely what exactly the **Feature Recognition** is supposed to recognize, click the Feature recognition settings button.

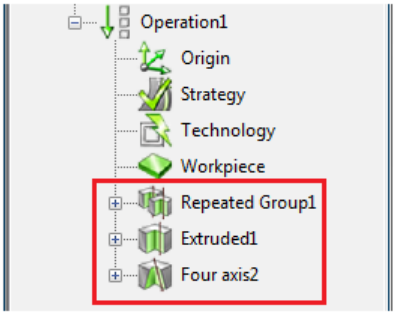
Set the dialog as follows:



Accept changes by clicking  . Then close the **Feature Recognition** dialog:  .

The system will now search the solid body for all geometries that can be wire cut (feature recognition) and create a machining suggestion with the cutting parameters previously set.

For each geometry that can be wire cut, OPTICAM creates a separate feature in the Feature Manager:

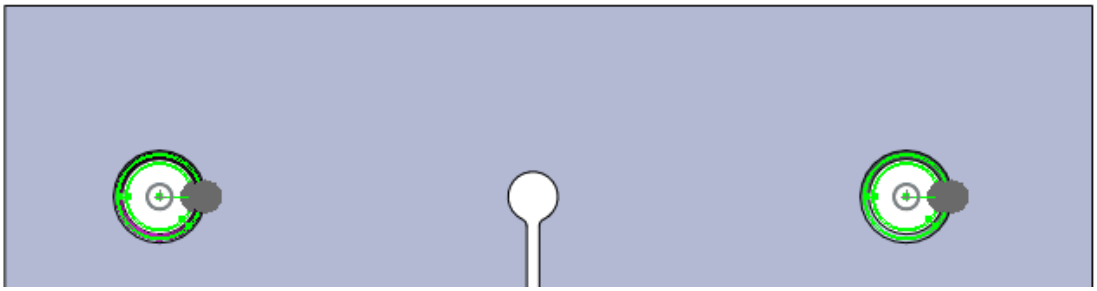


Automatic Start Hole Recognition

Note how the OPTICAM feature recognition identifies start holes in a 3D model and uses them as threading position for the cutting machining.

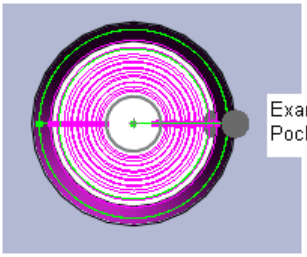
Therefore, the start holes in the 3D model have to be drawn as GibbsCAM surfaces of the type Extruded Circle. As this is the case in this example, the start holes were automatically identified.

The following figure is a part of the upper area of the model and shows the start hole recognition for the two tapered holes.



Pocketing Apertures

The apertures in this example have to be pocketed, instead of being machined with one or several cuts.

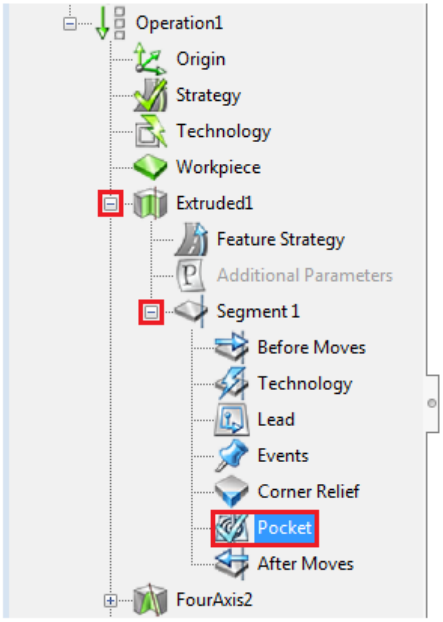


Example:
Pocketing of a tapered hole

OPTICAM is able to machine cylindrical and tapered geometries as well as four axes by means of pocketing. The advantage of the pocketing strategy is that no slugs are formed and the operator does not have to intervene to remove the slugs. However, the process only makes sense for smaller apertures as it leads to longer running times.

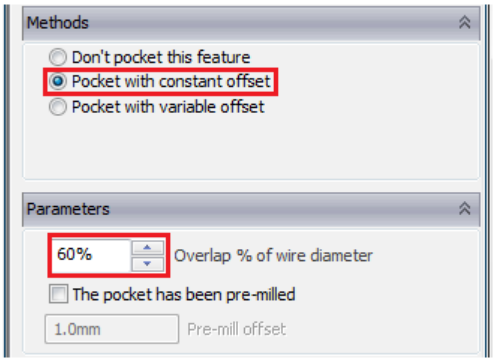
First, we will program the **pocketing** for the **cylindrical aperture** in the center of the model.

In Feature Manager, open the feature Extruded1 (the cylindrical aperture in the center) and double-click the branch Pocket.



The **Pocketing** dialog opens. Choose the option **Pocket with constant offset**. The selected option is a strategy which is primarily used for the **pocketing of cylindrical geometries.**

By modifying the setting **Overlap % of wire diameter** to **60%**, you set the distance of the tool paths.

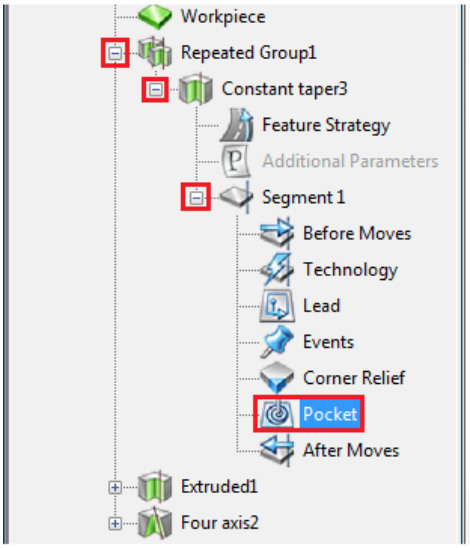


User advice: Do not define the value in the field **Overlap % of wire diameter** too big. When dealing with complex geometries, the definition of values of over 60% can lead to the formation of slugs. Use **Solid Simulation** to check the tool path for slugs.

Accept changes by clicking .

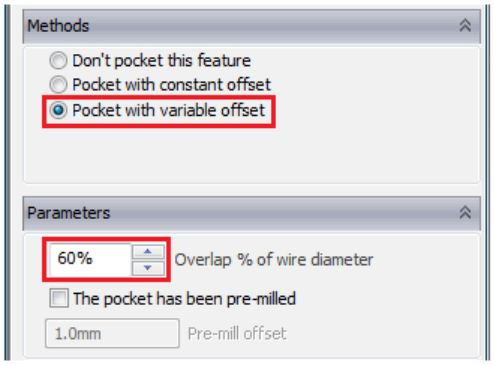
Next, you have to pocket the two tapered apertures.

In Feature Manager, open the feature Constant taper3 (the two tapered apertures at the top) and double-click the branch Pocket.



The **Pocketing** dialog opens. Choose option **Pocket with variable offset**. This option is a strategy for the pocketing of Tapers and Four Axis Geometries.

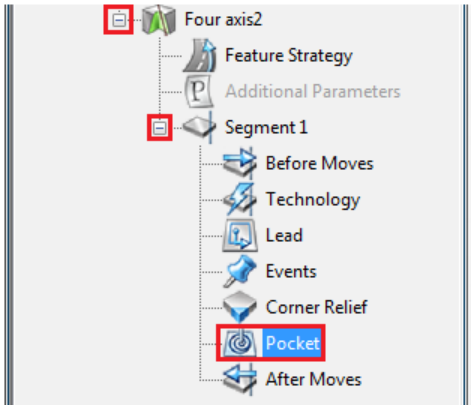
Modify the setting **Overlap % of wire diameter** to **60**%. This sets the distance of the tool paths.



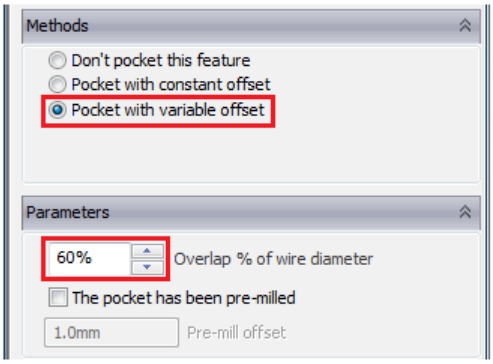
Accept changes by clicking .

At the end, you have to pocket the four axis.

In Feature Manager, open the feature FourAxis2 and double-click the branch Pocket.



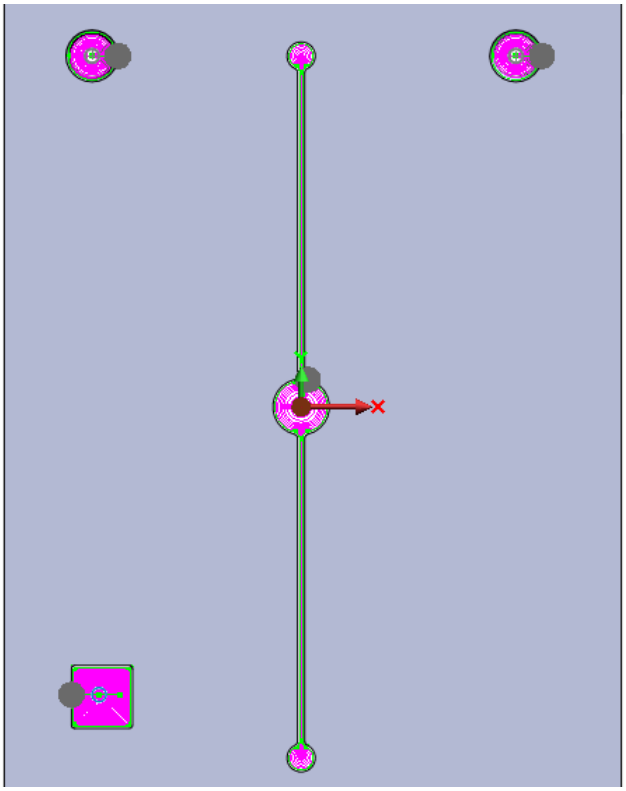
The **Pocketing** dialog opens. Choose option **Pocket with variable offset**. This option is a strategy for the pocketing of tapers and four axis geometries.



Modify the setting **Overlap % of wire diameter** to **60**%.

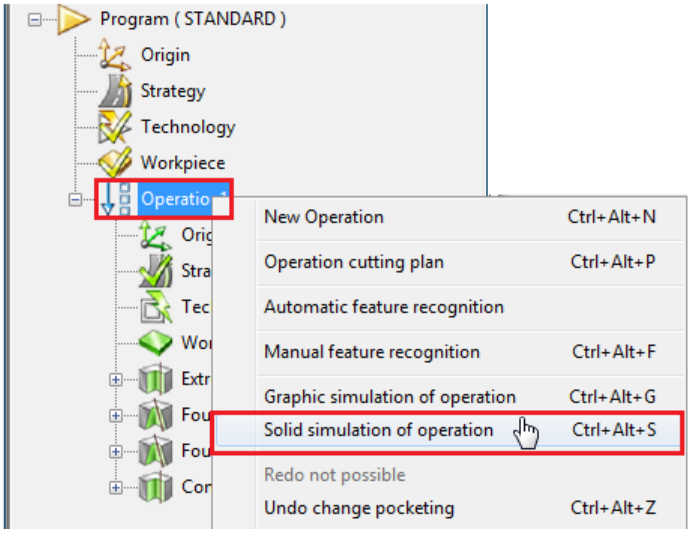
Accept changes by clicking  .

The following figure shows what the tool paths should look like:



Machining Simulation (Solid Simulation)

Right-click **Operation1** and, on the context menu, click **Solid simulation of operation**.

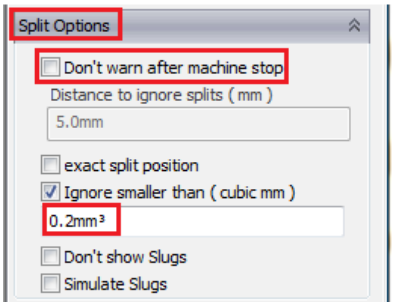


Display of Slugs (Solid Simulation)

When dealing with pocketing, it is recommendable to turn on the display of slugs before starting the solid simulation. If slugs are formed, a warning is displayed there.

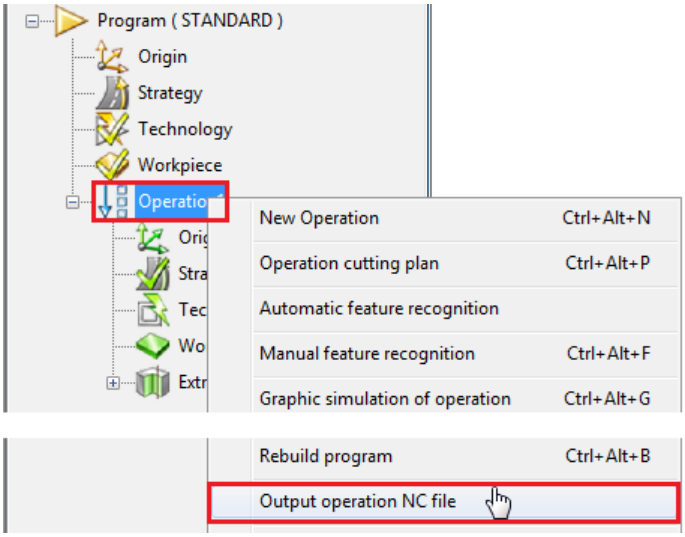
Open the field **Split Options** in the simulation dialog. *Deactivate* the option **Don't warn after machine stop**.

Then enter the value **0.2** mm³ in the field **Ignore smaller than (cubic mm)**. This setting ignores slugs whose volume is smaller than 0.2 cubic mm.



Creating the NC Program

Right-click **Operation1** and, on the context menu, click **Output operation NC file**.

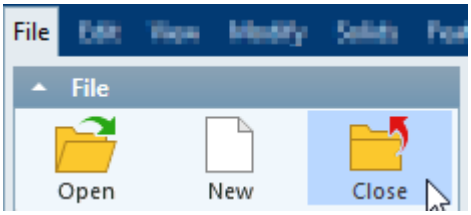


Saving the OPTICAM Program

At the end, you should save your work.

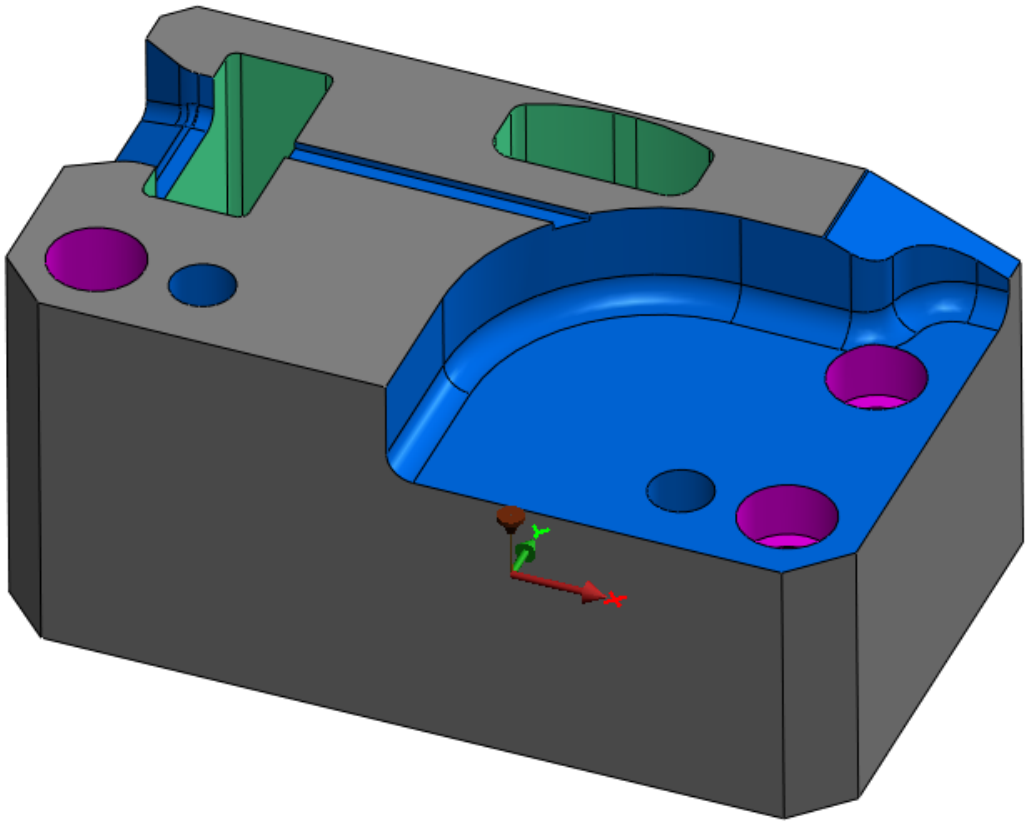
User advice: In addition to the GibbsCAM file **GC14_Opticam_example_8.vnc**, a second file containing the OPTICAM machining information for the current program will be saved. This OPTICAM file has the file extension .**wire.opticam**.

Close the file **GC14_Opticam_example_8.vnc**.



This exercise is now completed.

Example 9: Feature Recognition by Colors



Demonstrated in This Example

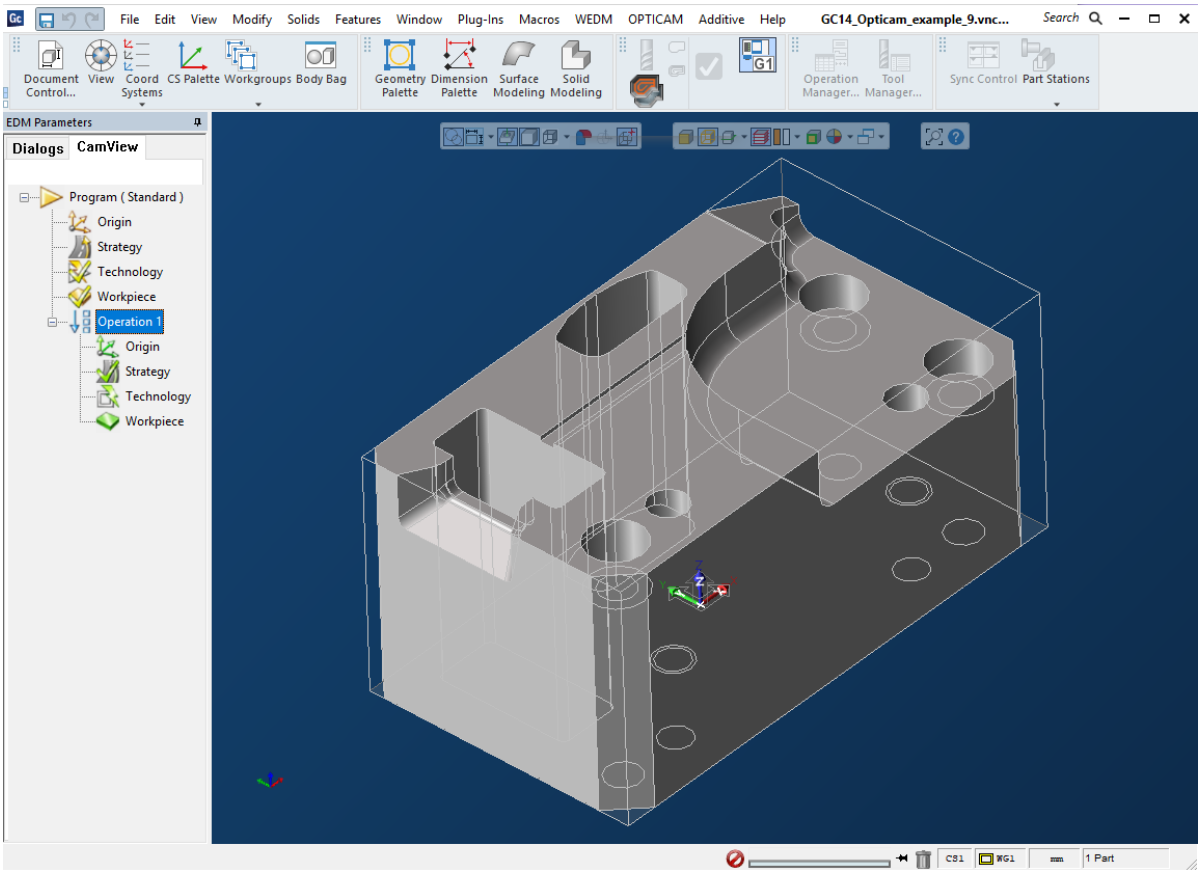
Opening a file: GC14_Opticam_example_9

Creating a Color Search List

Recognizing Features by Their Colors

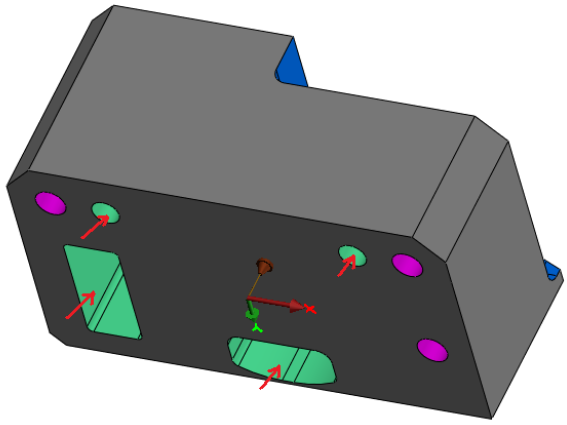
Open GC14_Opticam_example_9

In GibbsCAM, open the example file **GC14_Opticam_example_9.vnc**.



The model contains several apertures that differ from each other in color. Many companies use this color distinction to mark the required machining process. In our case, all **light green faces** are to be **wire cut**, the **blue faces** are to be **milled** and the **purple faces** are to be **drilled**.

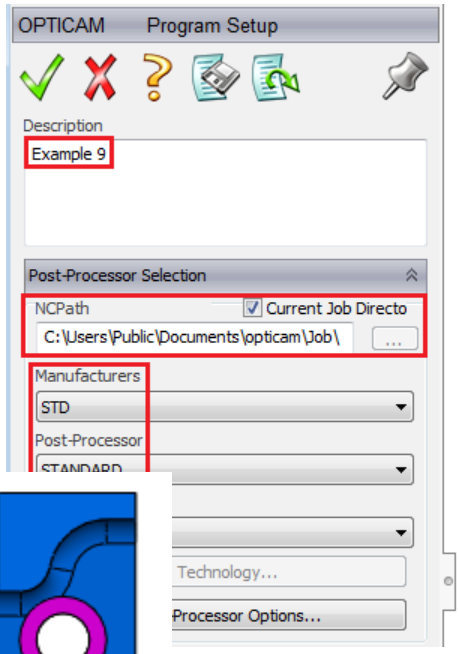
The following model shows the faces which are to be wire cut (see the four arrows).



Using the Program Dialog for Basic Settings

Double-click the branch **Program (STANDARD)** in the OPTICAM Feature Manager.

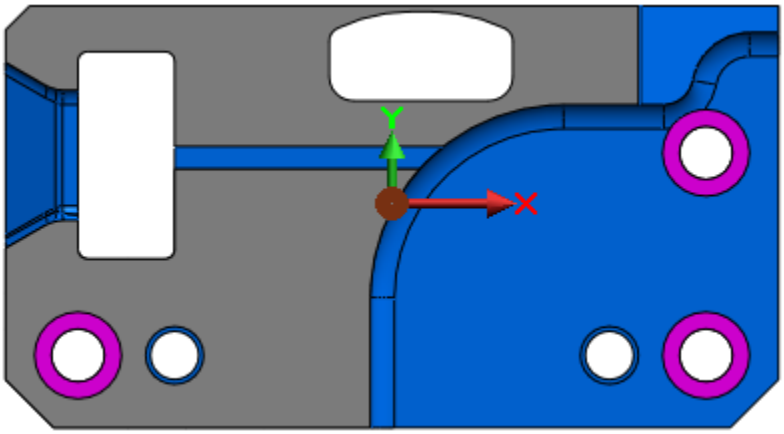
Fill in the dialog as shown to the right:



Accept changes by clicking  .

Defining the Machine Origin

In this example, the origin is in the center of the center of the workpiece again. As this is okay, we do not have to change the origin.



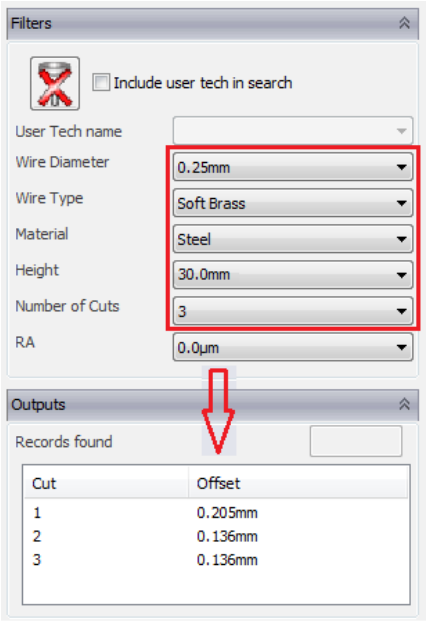
Defining the Cutting Technology (from Database)

Double-click **(Program) Technology** to open the **Technology** dialog to define the cutting technology. We will select the technology parameters from the technology database.

Select the appropriate technology parameters from the listboxes **Wire Diameter**, **Wire Type**, **Material** and **Height**.

The selected technology will be displayed in the dialog area **Records found**.

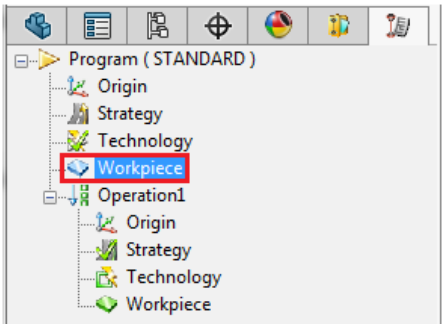
Accept the selection of the technology parameters by clicking  .

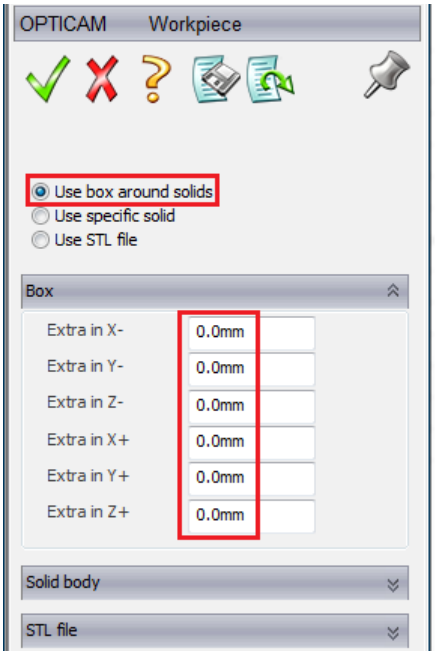


Creating the Workpiece (Billet) Geometry

In the next dialog, you can define the geometry of the workpiece (billet). The workpiece is to be derived from the solid body; additionally, no offset values are to be defined.

Double-click **(Program) Workpiece** and fill in the dialog as shown:

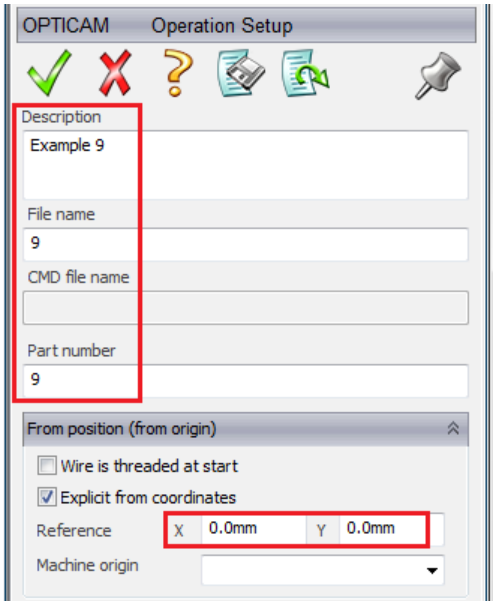




Accept changes by clicking  .

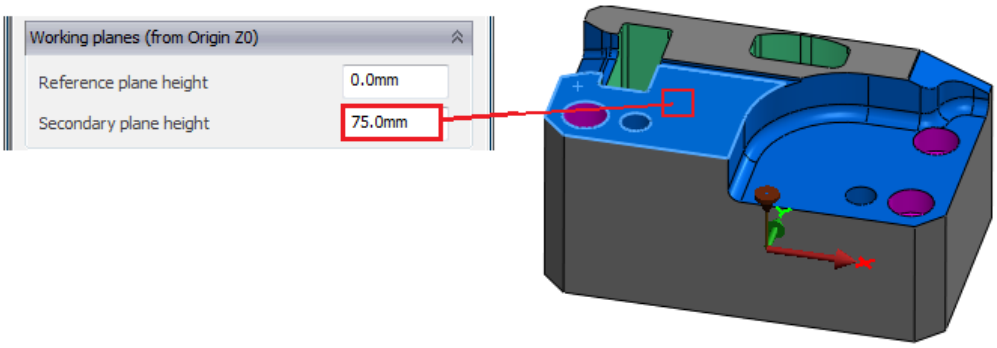
Setting the Cutting Heights

Double-click **Operation1** and fill in the fields Description, File name, Part number and Position as follows:



To identify the Secondary plane height, position the cursor in the input field **Secondary plane height** and click on one of the top plane faces of the model.

OPTICAM calculates the Z-height of the selected element and passes the value to the dialog box **Secondary plane height**.

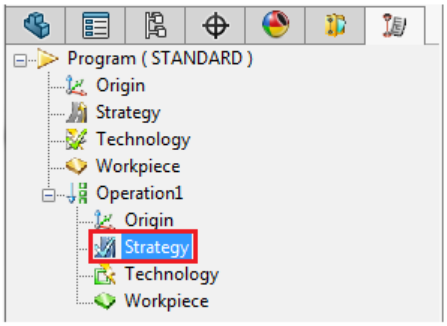


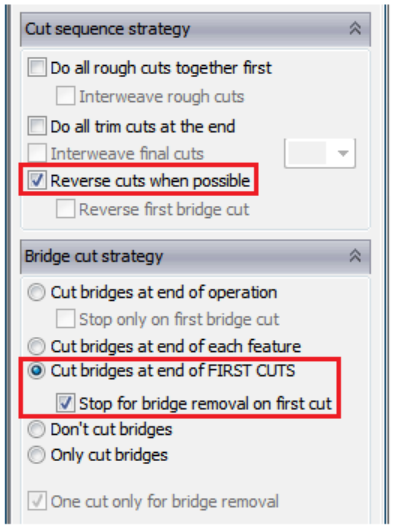
Accept changes by clicking  .

Setting the Cutting Strategy

In the next dialog, you set the strategy for machining the workpiece. The bridge has to be removed directly after each rough cut, and immediately afterwards the trim cuts are to follow.

Double-click the branch **(Operation) Strategy** and set the dialog as shown:

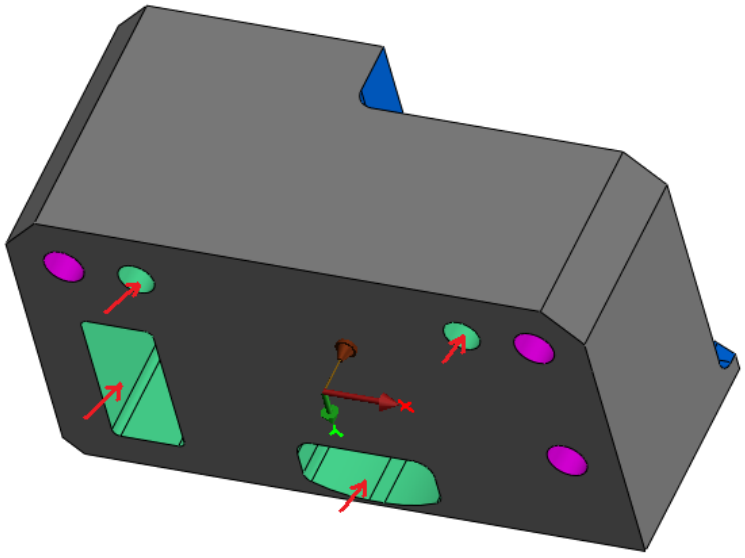




Accept changes by clicking on  .

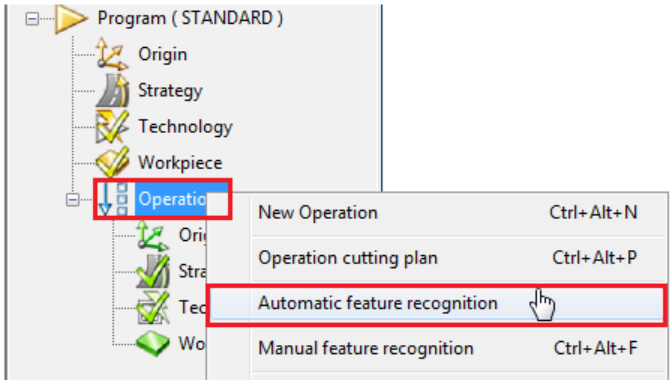
Recognizing Features by Their Colors

In this example, we want the automatic feature recognition to recognize all light green faces and to machine only these faces. (The desired apertures are marked with arrows in the figure below).

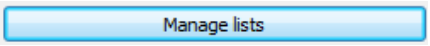


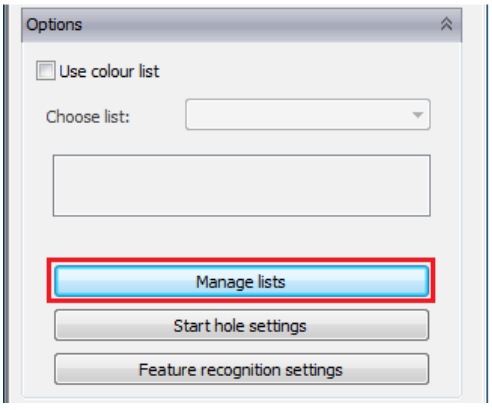
Creating a Color Search List

Right-click **Operation1** and, on the context menu, click **Automatic feature recognition**.

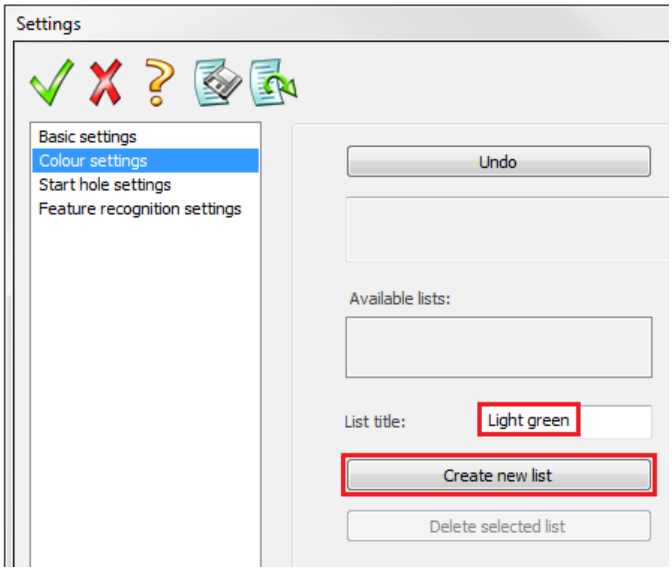


Initially, we will create a **color search list** in which we determine the color(s) to be recognized.

To create the **color search list**, select the  button.



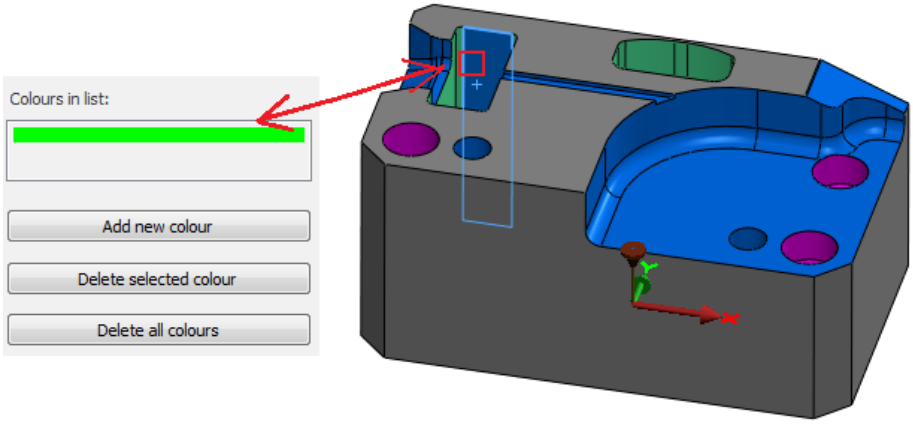
In the **Color settings** dialog, for **List Title**, enter the name of the color list: **Light green**. Then click the button **Create new list**.



Now click on a light green face in the model.

The color of the desired face is passed on to the dialog and entered in the field **Colours in List**.

User advice: More faces with other colors could be selected to create a color search list with several colors.

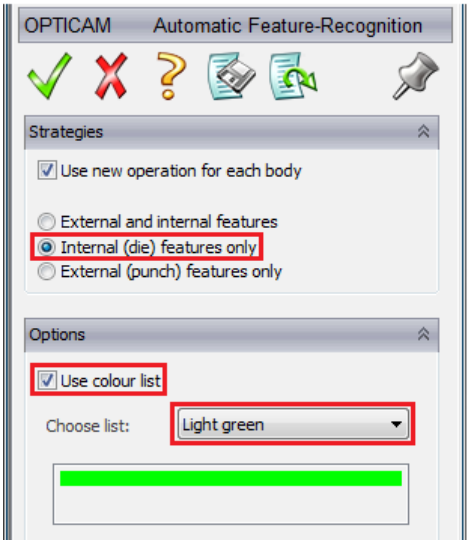


Accept the color settings by clicking .

The system returns to the dialog for the automatic feature recognition.

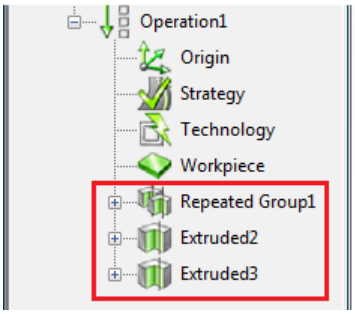
Select checkbox **Use colour list** so the system searches for colored faces only—in this case, only light green faces. Then, in the **Choose list** area, select the just-created list item: **Light green**.

Make sure that the dialog looks as follows:



Accept changes by clicking  .

The automatic feature recognition searches the solid body for all light green geometries which can be wire cut and creates a machining suggestion with the cutting parameters previously set. As always, OPTICAM creates a separate feature in Feature Manager for any geometry that can be wire cut:

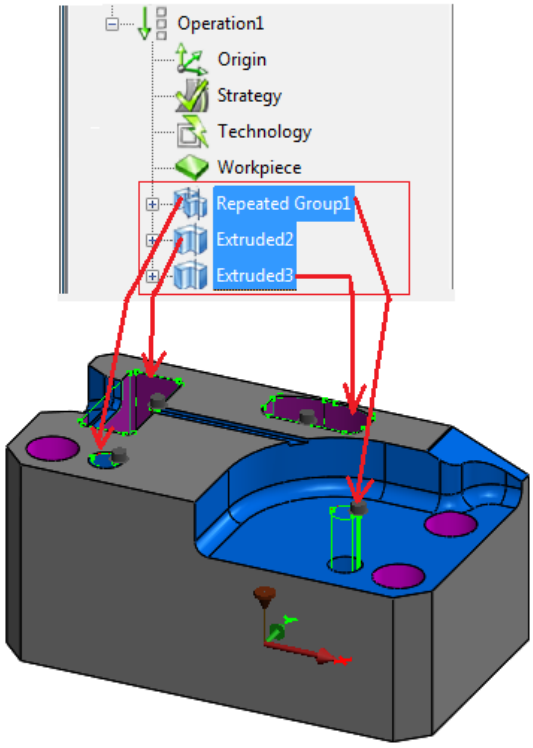


Checking the Created Machining Features

The easiest way to check the created features is the visual check.

To do this, click the existing features in the Feature Manager that you want to check. This shows the tool paths belonging to the features highlighted so that we can check them graphically.

To select all four features, click the first feature and hold down the **Shift** key as you select the last feature. The corresponding tool path is highlighted.

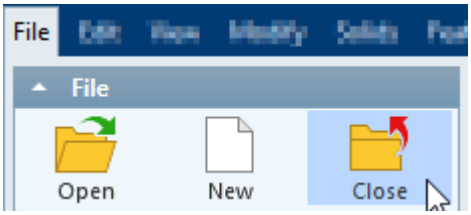


Saving the OPTICAM Program

At the end, you should save your work.

User advice: In addition to the GibbsCAM file **GC14_Opticam_example_9.vnc**, a second file containing the OPTICAM machining information for the current program will be saved. This OPTICAM file has the file extension .**wire.opticam**.

Close the file **GC14_Opticam_example_9.vnc**.



This exercise is now completed.