TOOL Builder

Software documentation

OPEN MIND
THE CAM FORCE

This software documentation applies to TOOL Builder.

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1. Introduction

Many tool suppliers offer 3D models of their tools. You can use the TOOL Builder to interactively prepare these models so they can be used for calculation, collision checking and simulation.

3D data import and tool entity creation is handled by running a pre-specified series of steps from a toolbar in the TOOL Builder. It is also possible to repeat a command from a previous step. Selected entities, such as the solids for a holder or axes, for example, are automatically labelled with tags and saved to an appropriately named layer.

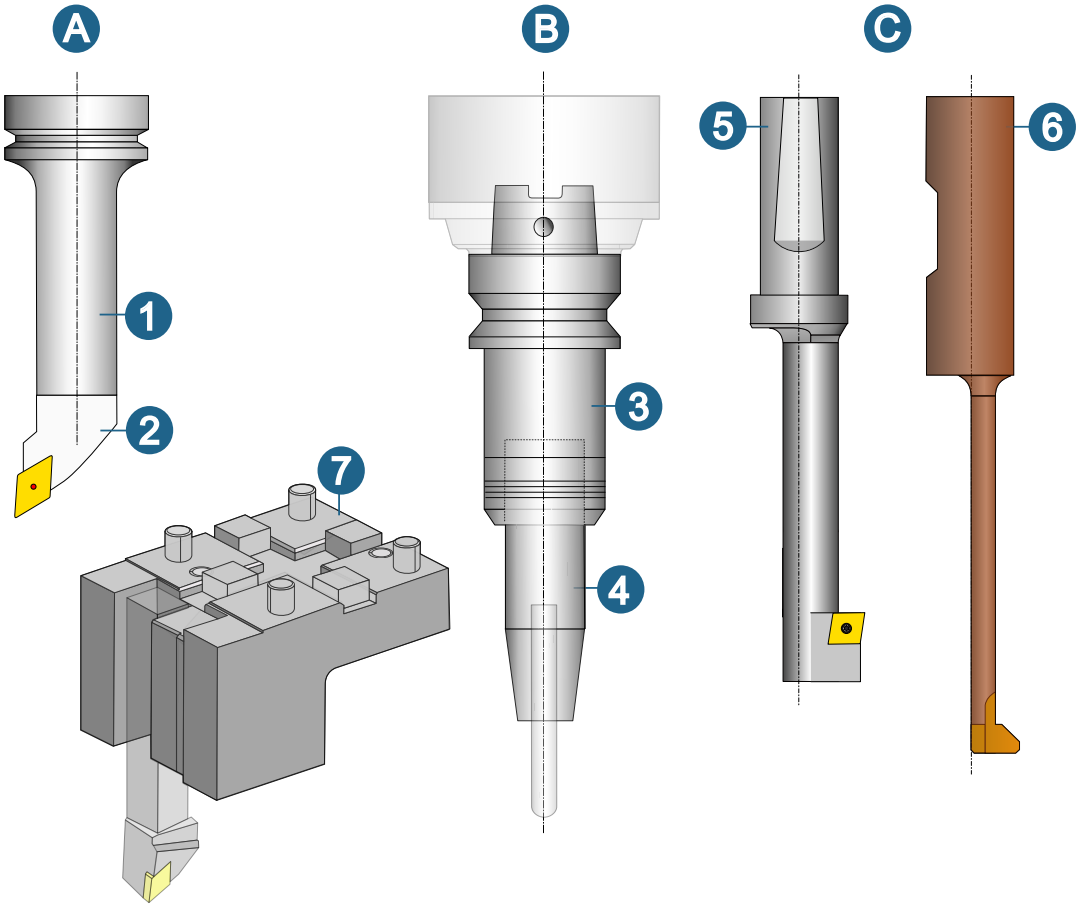
The following tools and tool elements are supported:

Ⓐ **Turning tool**: ① **Holder** (with rotationally symmetrical support in the spindle), ② (Insert) **tool holder**,

⑦ **Static holder** (turning with turret)

Ⓑ **Milling tool**: ③ Holder, ④ Extension.

Ⓒ **Back boring tool**: Geometry type ⑤ Body and insert, Geometry type ⑥ Monobloc



By default, tools are assigned the name of the document with which they were created.

CAD entities, which are used to define a tool holder, holder, static holder or extension, are locked against further editing.

2. Positional orientation

When tools are assembled, a positional orientation is required.

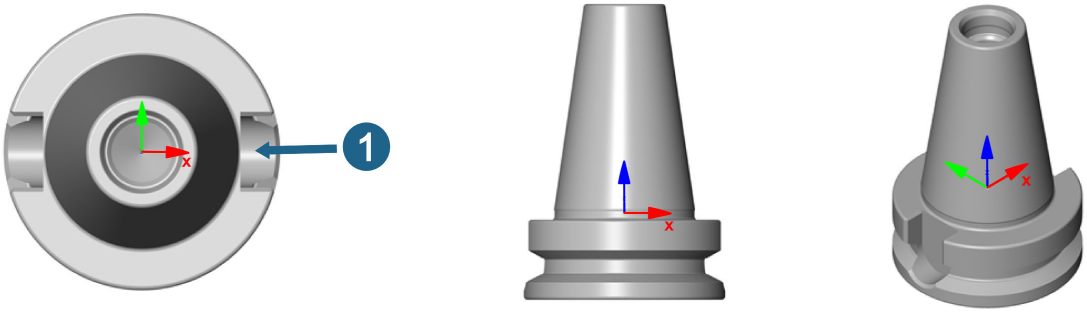
The Z axis lies coaxially to the main flange and points in the direction of the spindle.

Holder/extension positional orientation

The alignment around the Z axis will allow non-symmetrical entities, such as drill heads and angle heads, to be supported in the future. If a holder (for example, HSK, CAPTO, KM and KM4X) is also used for turning on a mill/turn machine, the alignment around the Z axis must be observed. In the case of usual milling applications with circumferentially rotated bodies, the alignment around the Z axis is only a visual/aesthetic issue.

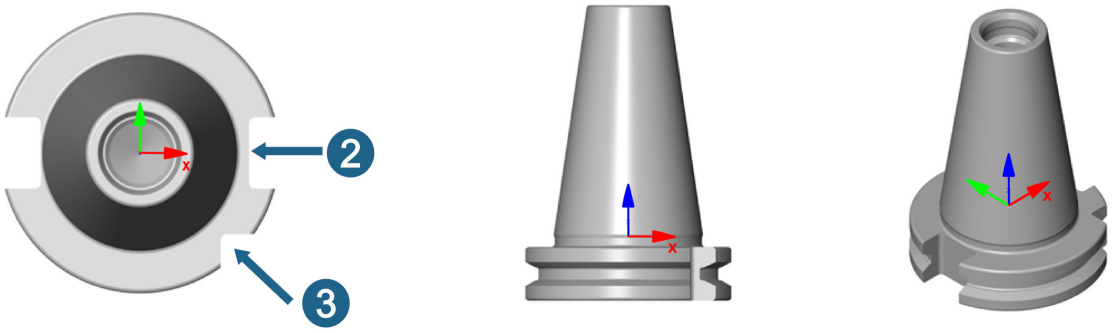
BT

The X axis points towards the center of one of the two slots ①. As the BT is symmetrical, the alignment of the X axis can only be distinguished with other attributes (for example, a screw or another non-symmetrical entity). Check the alignment of this entity before the 3D data import on the machine tool. The upper coupling should be defined in the flange depth.



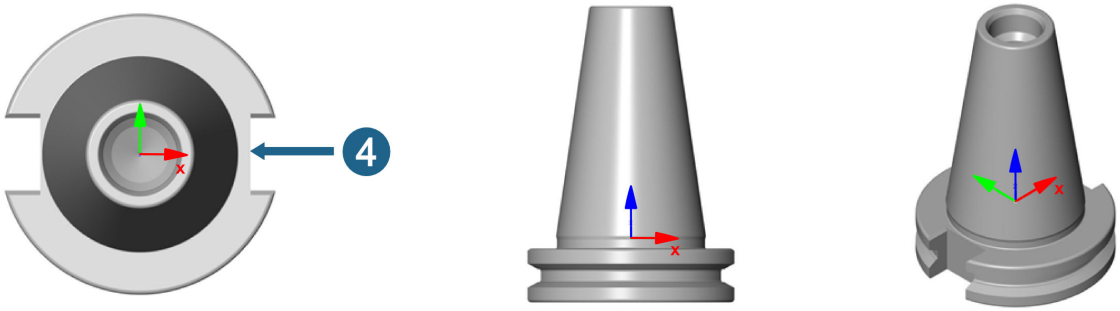
SK

The X axis points towards the center of the flattest slot ② – on the same side as the notch ③. The upper coupling should be defined in the flange depth (3.2 mm above the upper side of the flange).



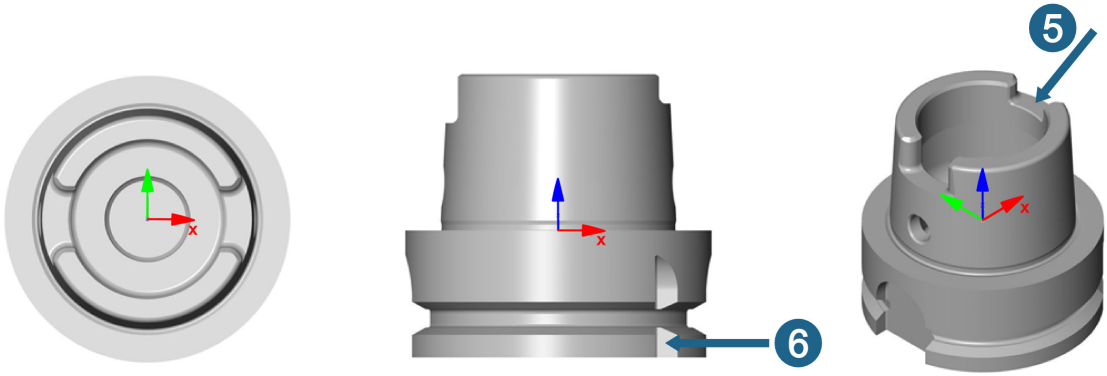
CAT

The X axis points towards the center of the flattest slot ④. The upper coupling should be defined in the flange depth (3.2 mm above the upper side of the flange).



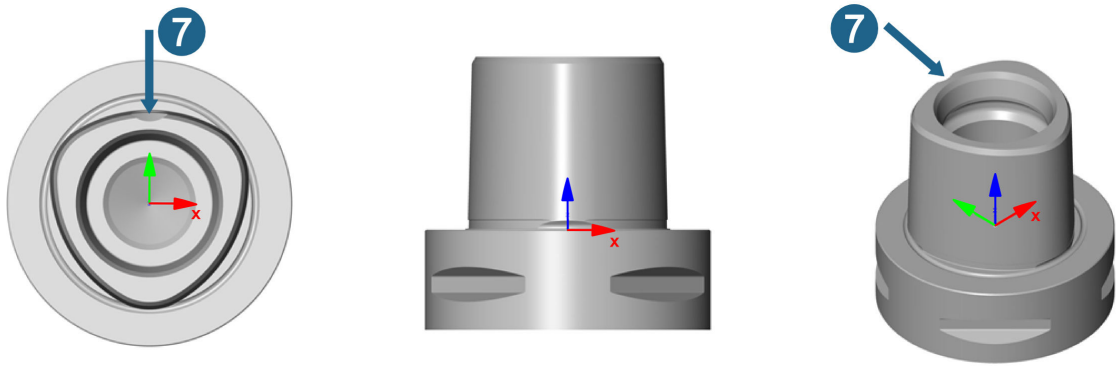
HSK

The X axis points towards the center of the flattest slot ⑤ – on the same side as the notch ⑥. The upper coupling should be defined on the upper side of the flange.



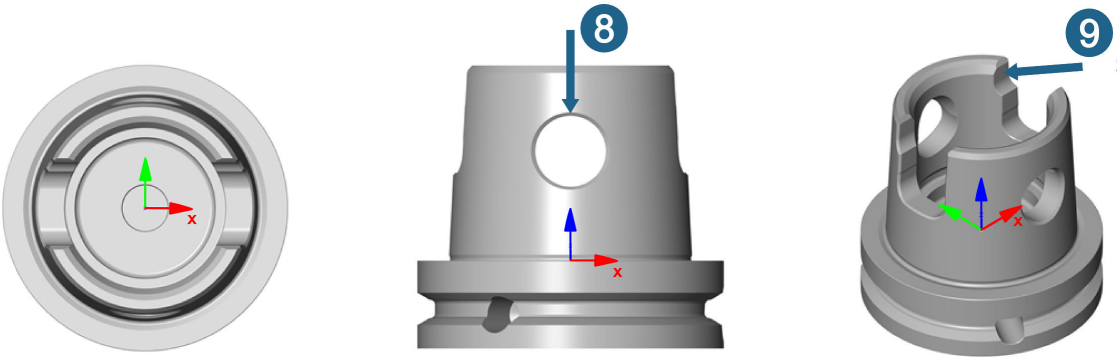
CAPTO

The Y axis points towards the center of the reference notch ⑦. The upper coupling should be defined on the upper side of the flange.



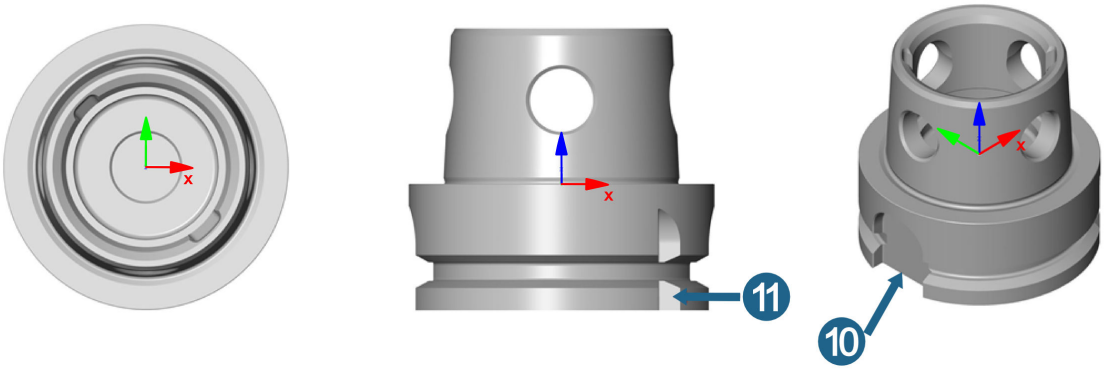
KM

The Y axis is aligned through the axis of the holes ⑧ and points towards the recess ⑨ on the tip (see arrow). The upper coupling should be defined on the upper side of the flange.



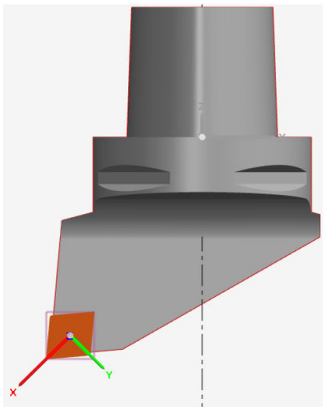
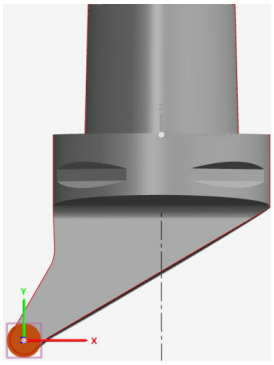
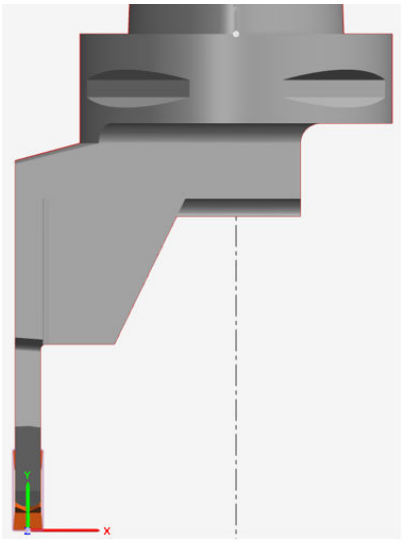
KM4X

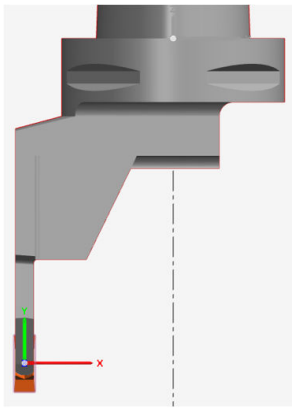
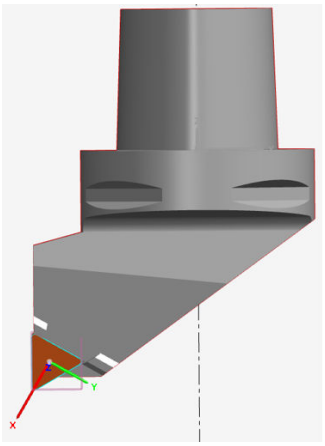
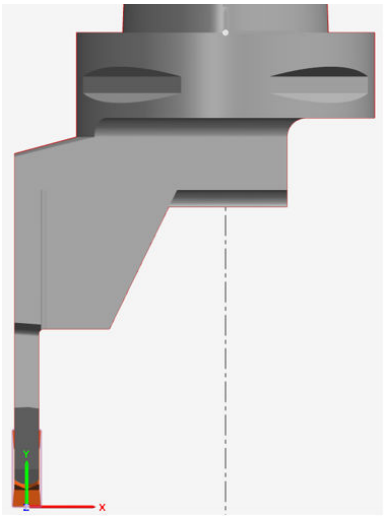
The X axis is aligned through the center of the two lower slots ⑩ and points in the same direction as the notch ⑪. The upper coupling should be defined on the upper side of the flange.



Insert positional orientation

Insert-specific X orientation of the coordinate system

Rhombic insert	Round insert	Rectangular insert
		

Parting insert	Threading insert	Rectangular insert
		

3. Turning tool: Creating an insert tool holder

To create an insert tool holder with the TOOL Builder, proceed as follows.

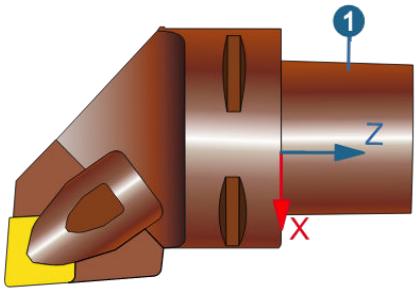
In the tool database, on the **Tool Holders** dialog page, click on the icon for **Geometry** → **3D file** to start the TOOL Builder.

Then use the **File** → **Open** dialog to open a 3D geometry file in `*.step`,`*.stp`, `*.iges` or `*.hmc` format.

Define body

Specify the geometry of the insert tool holder.

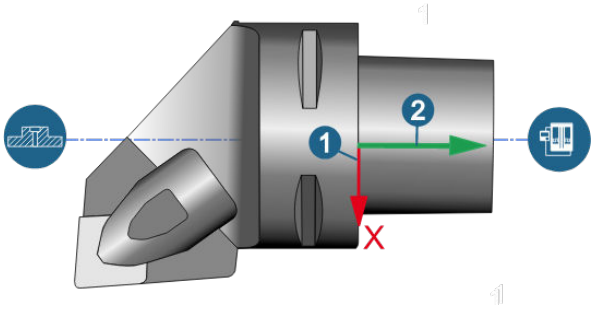
Select Entities. The selected geometric entities ① are highlighted in color and the number of entities is shown. The data is saved on a designated layer and marked with a tag.



Define axis

The geometry of the insert tool holder is recognized automatically based on the coaxial faces of the body and the program automatically suggests the axis alignment (machine side/workpiece side). A reference coordinate system ① is created, whose alignment corresponds to the Z axis of the insert tool holder.

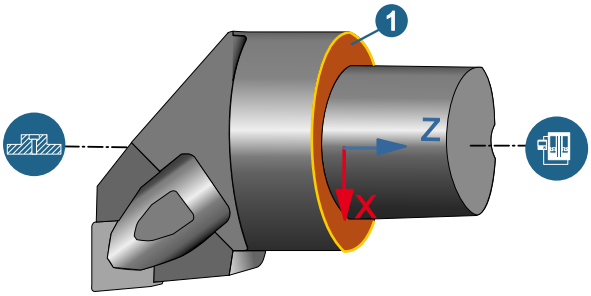
Manually adjust the **Direction** of the axis if necessary. To do so, select entities or **2 points**. The order in which the points are selected defines the direction ② of the axis. Invert the **Direction** by double-clicking the axis arrow ② or use the **Invert** command.



Define coupling

Select a **face** or a **point**. The selected entity ① is highlighted in color.

Specify the **X direction** of the reference coordinate system. To do so, select **2 points** or a single **point**.

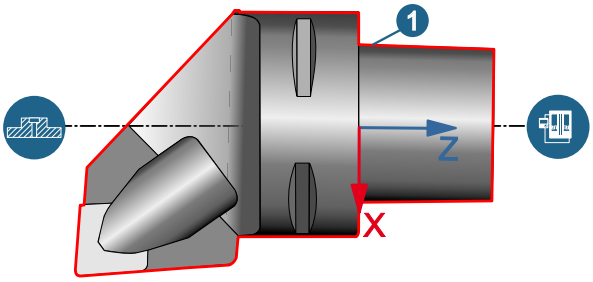


Click the button to rotate the X axis direction by 90°.

Use the **Invert** function to reverse the direction. This can also be adjusted directly for each entity in the corresponding row (red, green, blue). Use **Save settings** to save your changes.

Define outline profile

The program automatically generates an outline profile ① based on the defined body geometry and the alignment of the X axis in the reference coordinate system.



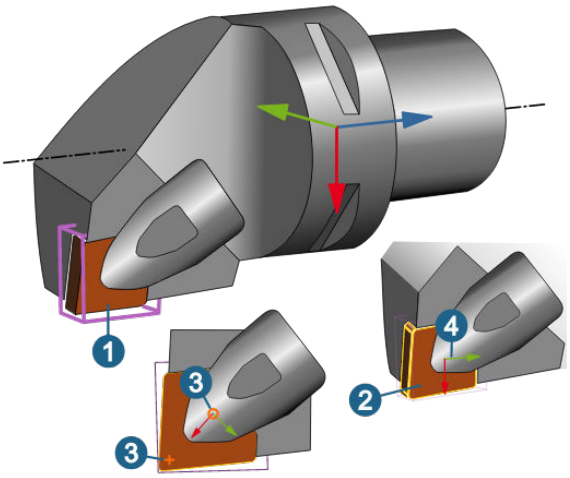
Define insert

Select the **entities**. The selected entities ① are highlighted in color.

It is not absolutely necessary to specify the geometry of the insert in order to define the tool holder.

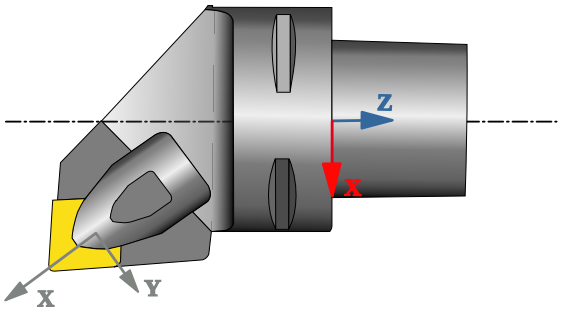
To define the **Mounting position**, specify a **Face** ② or an **Origin**. The selected entities ② are highlighted in color.

Specify the alignment of the insert by defining the **X direction** of the reference coordinate system ④. To do so, select either **Entities**, a **Point** or **2 points** ③.



Generate STL file

The program automatically generates an outline profile ① based on the defined body geometry and the alignment of the X axis in the reference coordinate system.



Export data

Depending on the imported 3D file, **Geometry** parameters of the insert tool holder may be calculated and transferred to the tool definition during data transfer (axis distance, shank distance, rake angle, inclination angle, approach angle, mounting direction, spindle orientation).

The values for the **Rake angle** and **Inclination angle** parameters are calculated exclusively when defining an insert holder with the TOOL Builder software. Manual definition in the tool database is not possible. The **Rake angle** and **Inclination angle** parameters are currently not supported for recessing tools, threading tools and parting tools.

4. Turning tool holder assembly

Assembly the 3D data provided by tool manufacturers to form a tool holder for turning tools.

To assemble a turning tool holder with the TOOL Builder, proceed as follows.

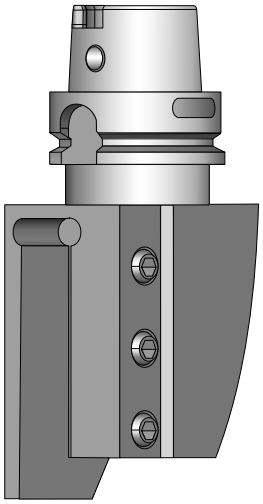
In the tool database on the **Tool holder** dialog page click on the icon for **Geometry** → **3D file** to start the TOOL Builder.

Then use the **File** → **Open** dialog to open a 3D geometry file in `*.step`,`*.stp`, `*.iges` or `*.hmc` format.

Select model for base body

Select model for base body.

Cancel the **Body** command. Then show the preconfigured toolbars. To do this, click  . The background color of the graphics area changes.



Load parts that are attached to the base body

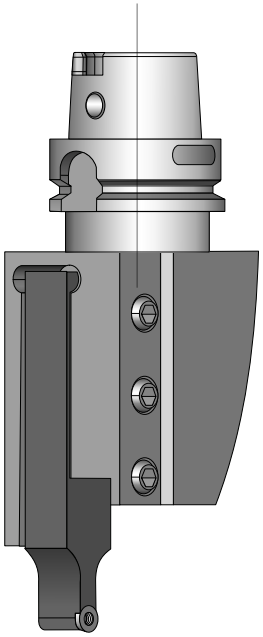
Select the **Group entities** option of the **Merge** command. All imported entities are combined into a group. This makes it easier to attach a part as a whole to the base body. Use **Save as default** to preset the option.

Attach the parts to the base body.

Assemble the various solids and groups using **Mating** and **Move / copy**.

The base body is fixed automatically by loading a first additional part.

If you choose, simplify the tool holder so that it specifies the actually existing tool holder with sufficient accuracy.



Collisions may occur if the outer shape of the tool holder model does not match the tool holder that is really used.

Switch back to the normal mode for defining a tool holder and complete the definition of the tool.

See section **Turning tool: Creating an insert tool holder**.

5. Turning tool: Creating a static holder for machining with a turret

Import the 3D data provided by tool manufacturers and create a static holder for turning operations with a turret.

To create a static holder for machining with a turret with the TOOL Builder, proceed as follows:

In the tool database, on the **Static holders** → **New** → **Turret holder** → **Geometry** dialog page, click the icon to start the TOOL Builder.

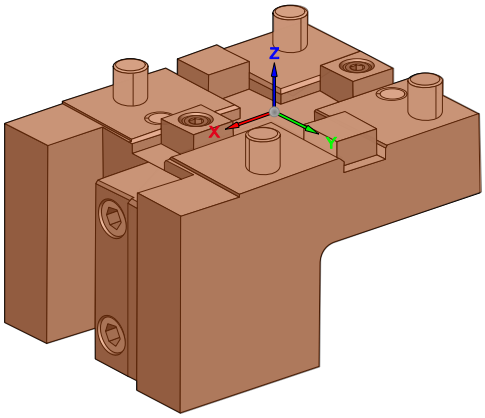
Then use the **File** → **Open** dialog to open a 3D geometry file in `*.step`,`*.stp`,`*.iges`, or `*.hmc` format.

Define the body

Define the geometry of the static holder.

The geometry of the holder is automatically detected, so that after opening the file, the **Elements** of the body are already selected and highlighted in color.

The number of elements is displayed.



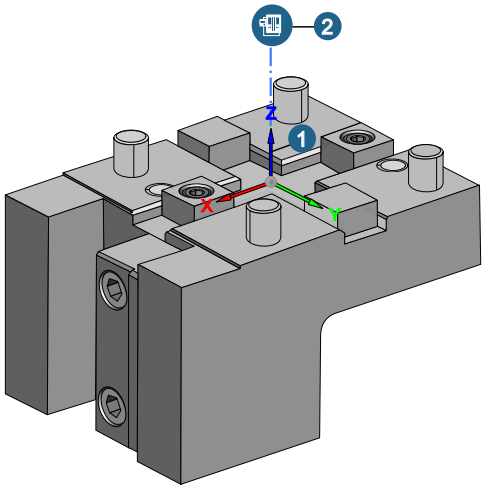
Define the holder axis

To do this, **Select** the required elements (for example, the reference coordinate system ①).

Alternatively, select **2 Points**. The order of selection defines the direction of the axis.

If necessary, invert the direction by double-clicking the arrow of the axis or activate the **Invert** function.

The program attempts to automatically detect the **Machine side** ②. Correct the machine side if necessary.



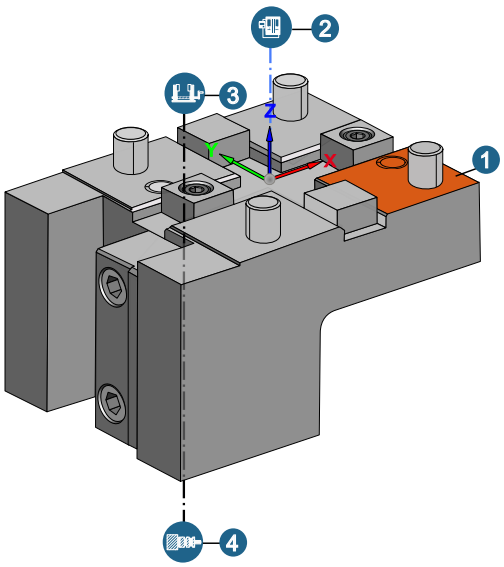
Define the holder coupling

To do this, select a **Face** or a **Point**.

The selected element ① is highlighted in color. Specify the **X direction** of the reference coordinate system. To do this, select the X axis of the reference coordinate system, **2 Points**, or one **Point**.

Machine side ②, **Clamping side** ③, and **Workpiece side** ④ are displayed.

Click the button to rotate the X axis direction by 90°.



Define the tool axis

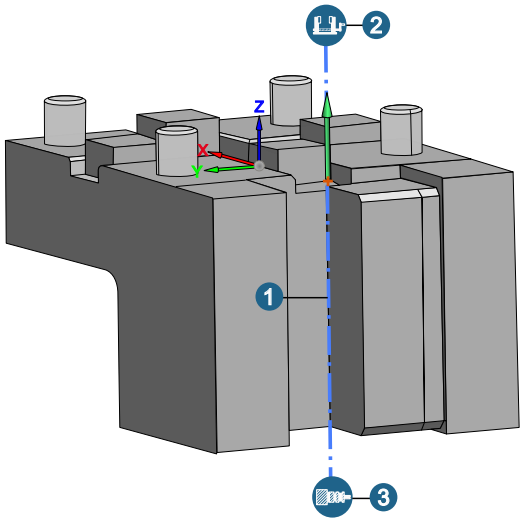
To do this, **Select** the required elements ①.

Alternatively, select **2 Points**.

The order of selection defines the direction of the axis.

Invert the **Direction** by double-clicking the axis arrow or activate the **Invert** function.

Clamping side ② and **Workpiece side** ③ are displayed.



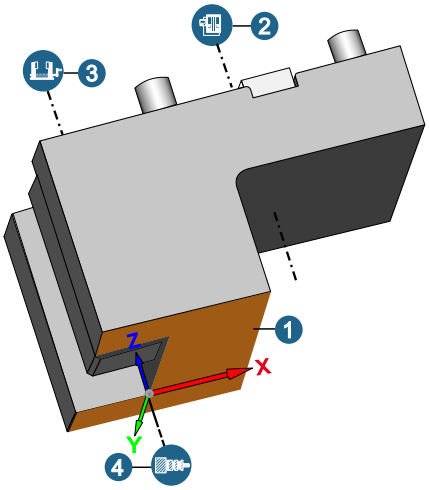
Define the tool coupling

Select a **Face** ① or a **Point** to define the area of the tool coupling.

If necessary, manually adjust the **X direction** of the axis. To do this, **Select** the required elements. Alternatively, select **2 Points** or one **Point**.

The order in which the points are selected defines the direction of the axis.

Machine side ②, **Clamping side** ③, and **Workpiece side** ④ are displayed.



Click the button to rotate the X axis direction by 90°.

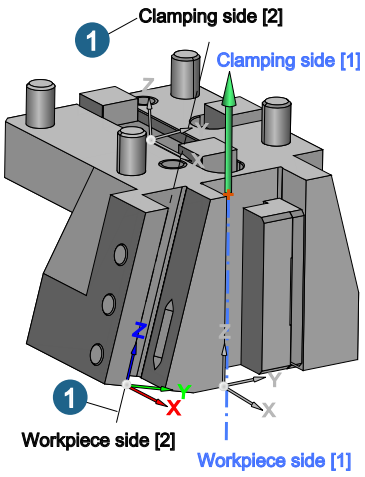
Add a new mounting position

To add an additional mounting position for a turret holder, select the **Add new mounting position** function and carry out the **Define tool axis** and **Define tool coupling** steps again.

Use the **Modify Tool** function on the shortcut menu to adapt a previously defined mounting position.

Then use the **Define tool axis** and **Define tool coupling** functions to correct the selection to be changed.

Use **Remove tool** to delete the mounting position.



Export the data

Transfer the imported 3D file with the previously defined elements to the tool database as a static holder.

6. Milling tool / Drilling tool: Creating a holder / extension

Import 3D data provided by tool manufacturers and create a holder or extension for a milling tool or drilling tool.

To create a holder or an extension with the TOOL Builder proceed as follows.

In the tool database, on the **Extensions** → **Geometry** dialog page click on the icon to start the TOOL Builder.

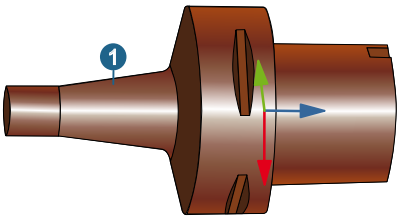
Then use the **File** → **Open** dialog to open a 3D geometry file in `*.step`,`*.stp`, `*.iges` or `*.hmc` format.

The following information and graphics relate to creating a holder. All steps, selections and options in the dialogs also apply to creating an extension.

Define body

Specify the geometry of the holder.

Select **entities**. The selected geometric entities ① are highlighted in color and the number of entities is shown. The data is saved on a designated layer and marked with a tag.



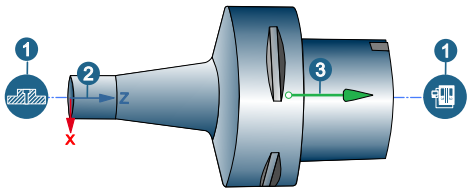
Define axis

The geometry of the holder is recognized automatically based on the coaxial faces of the body and the program suggests an axis alignment ① (machine side/workpiece side).

A reference coordinate system is created, whose Z axis alignment corresponds to the axis of the holder ②.

Manually adjust the **Direction** of the axis if necessary. To do so, select entities or **2 points** ①. The order in which the points are selected defines the direction of the axis. Invert the **Direction** by double-clicking the axis arrow ③ or use the **Invert** command.

The data is saved on a designated layer and marked with a tag.



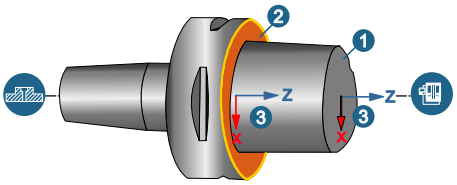
Define coupling

Specify the area for the upper ① and lower ② coupling. Select a **Face** or a **Point**.

The selected entities are highlighted in color. The selected face must be perpendicular to the Z axis. The selected point must be on the holder axis, otherwise it will be projected onto the axis.

The reference coordinate system ③, whose Z axis corresponds to the Z axis of the holder, is shown at the gage point for the upper and lower coupling.

The data is saved on a designated layer and marked with a tag.



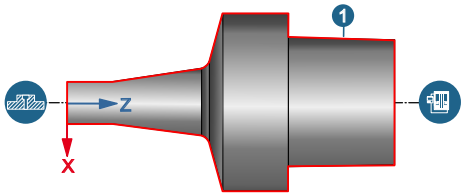
Define rotational profile

The program creates a rotational profile based on the defined body geometry and the alignment of the reference coordinate system axes (Z and X axes). Lines which correspond to the axis are removed automatically.

The rotational profile can be modified if necessary using the **Draw** commands of the TOOL Builder.

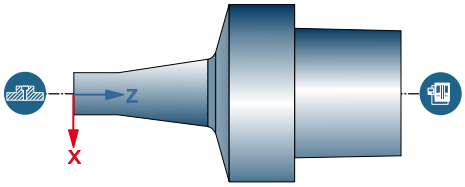
For a rotational profile ① to be created based on the defined body geometry, the geometry must represent a closed solid or a closed face body.

The data is saved automatically on a designated layer and marked with a TAG.



Create rotational shape

The program generates a simplified rotational shape from the defined entities for the holder. It is automatically saved on a designated layer and marked with a TAG.

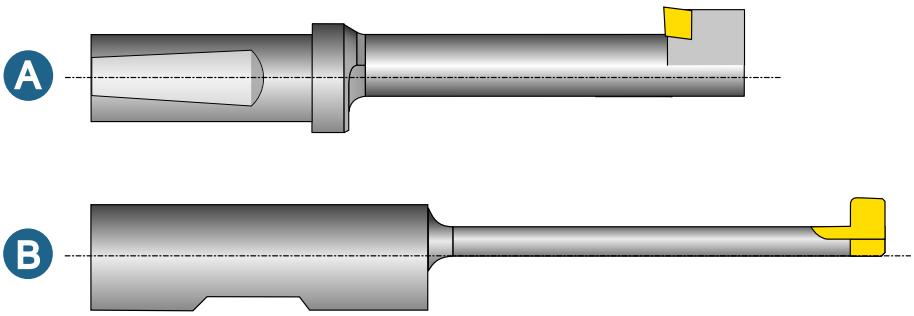


Export data

Export the generated rotational shape to *hyper*MILL and transfer to the tool definition.

7. Create Back boring tool

The two geometry types: **Body and insert** Ⓐ and **Monobloc** Ⓑ are supported.



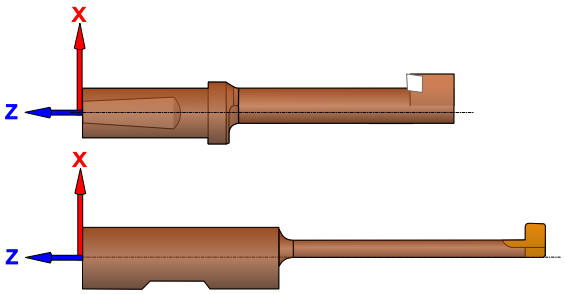
To create a Back boring bool with the TOOL Builder proceed as follows:

In the tool database, on the **Back boring tool** dialog page, select **Body and insert** or **Monobloc** for the geometry type and click on the icon for **3D geometry** to start the TOOL Builder.

Then via the **File** → **Open..** dialog open a 3D geometry file in `*.step`,`*.stp` or `*.hmc` format.

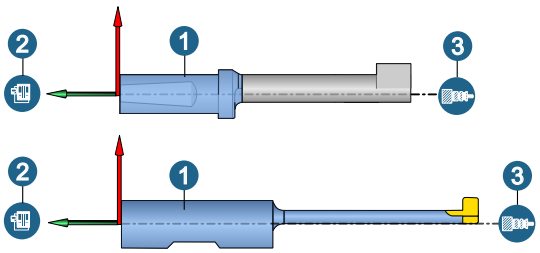
Define the body


Define the geometry of the tool. This is automatically detected, so that after opening the file, the **Elements** are already selected and highlighted in color. The number of elements is displayed.



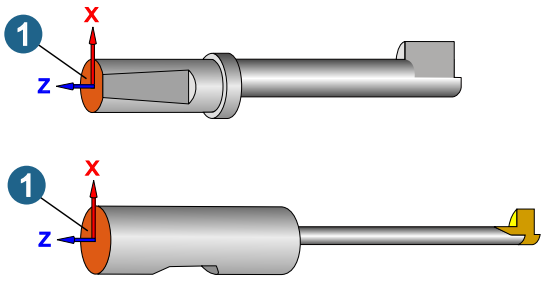
Define the axis


Define the axis by **Select** the desired elements ①. Alternatively, select **2 Points**. The order of selection defines the direction of the axis. If necessary, reverse the direction by double-clicking on the arrow of the axis or activate the **Invert** function. The program tries to detect the **Machine side** ② and the **Workpiece side** ③ automatically.



Define the coupling

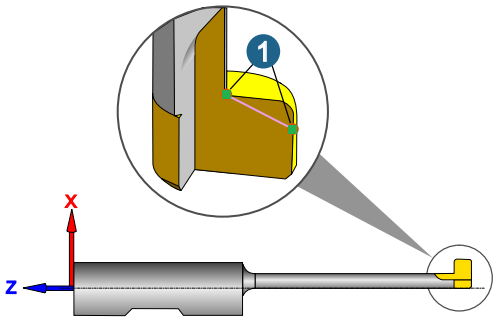
To do this, select a **Face** or a **Point**. The selected element ① is highlighted in color. Set the **X direction** of the coupling. To do this, select the X axis of the reference coordinate system, **2 Points** or one **Point**.



Split body and define the cutting area

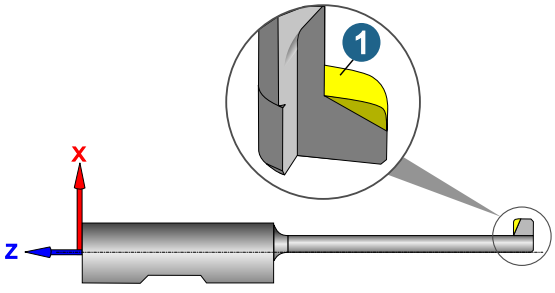
Die **Cutting edge orientation** function is currently not supported.

For the geometry type **Monobloc** split the body and define the cutting area. To do this select **2 Points** ①. The selected element is highlighted in color.



Select the cutting area

For the geometry type **Monobloc** select the cutting area as **Solid**. The selected element ① is highlighted in color.



Export data

Transfer the imported 3D file with the previously defined elements as a back boring tool to the tool database.

8. Options

Open the **hyperMILL TOOL Builder options** dialog to define display and precision options.

File → Options

Mesh tolerance: Specify precision for reading in the geometry data in the *.step, *.stp, *.iges or *.hmc formats.

Assembly mode: Display toolbars for assembling tool holders for turning tools.

Tool holder

Profile tolerance: Specify precision for the outline profile of the insert tool holder.

Holder

Profile tolerance: Specify precision for the rotational profile of the holder.

Colors

Define the colors for the **Axis**, **Machine / workpiece indicators**, **Profile**, **Body mesh**, **Insert mesh** and **Rotational shape**. To do so, select the entity in the **Value** column and click the button to open the color selection dialog and define the desired colors. This can also be adjusted directly for each entity in the corresponding row (red, green, blue).

Display the background of the graphics area in a different color for the **Assembly mode**. To do this, select the **Assembly - bottom color** and **Assembly - top color** options.

Use **Save settings** to save your changes.