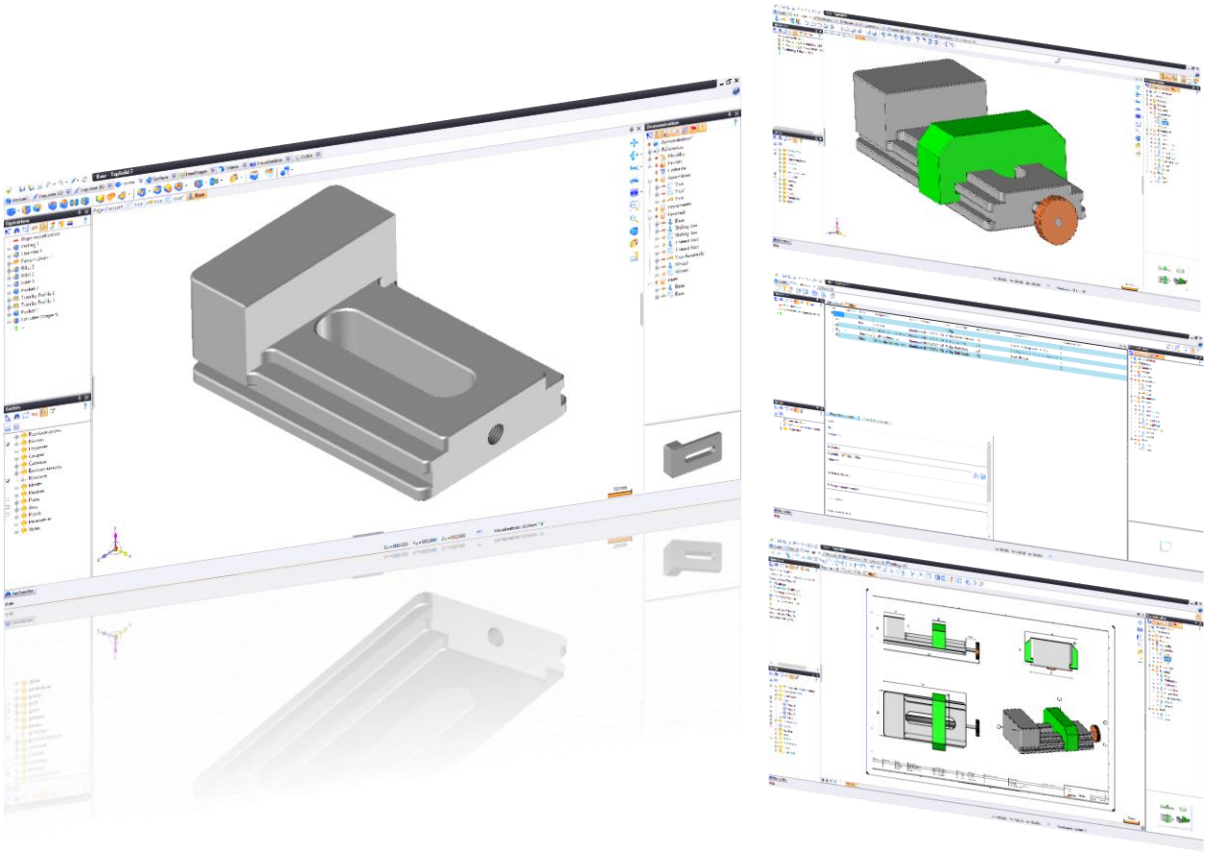
TopSolid Design

Tutorial

TopSolid'Design 7 Basics



MASTER YOUR MANUFACTURING PROCESS

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Introduction

The goal of this tutorial is to walk you through some basic design steps in **TopSolid 7**. Let's get started!

TopSolid 7 is a revolutionary product that has been rebuilt from the ground up to meet the needs of manufacturing companies, big and small. The following are some key features regarding **TopSolid 7** that will be highlighted during this tutorial:

- **TopSolid 7** is built on a PDM foundation
- **TopSolid 7**'s PDM manages ALL documents
- **TopSolid 7** is the only CAD/CAM software to fully support Windows 10
- Quicker design by simply using the drag-and-drop technique
- Commands adapted to the task at hand are easily accessible with an intuitive contextual menu
- Everything is managed automatically regardless of the user
- Easy to build and manage bill of materials
- Drafting documents created and managed within seconds

Requirements

This tutorial requires that you use a demonstration project. By default, this project comes with the software located in the directory: "C:\Program Files\TOPSOLID\TopSolid 7.14\Samples\en\TopSolid'Design Tutorial.TopPkg". You can also access this project from our website once you become a registered member. To register, please click on the following link: http://www.topsolid.com/download/login.htm. If you cannot click on the link, please either copy and paste it, or type it into your internet browser address bar.

To complete the registration, click on the link that was automatically sent to the email address with which you registered. Please note that some spam filters may block the registration email. To avoid this, please add the "topsolid.com" domain to your white list or check your spam folder if you do not see the email from us within a few minutes. If you have problems registering, please send an email to support@topsolid.com with a brief message explaining the difficulty you are having.

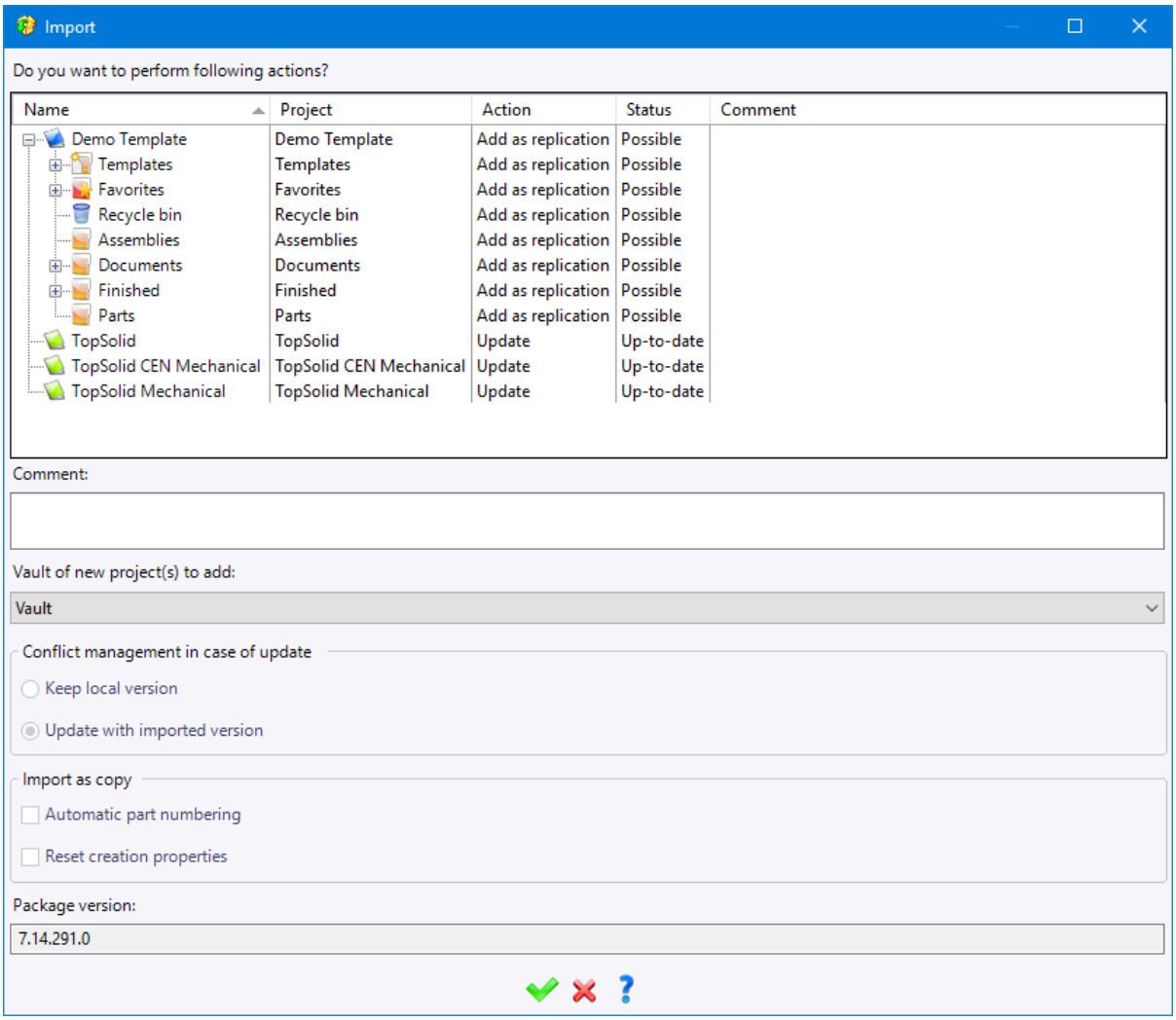
Once you have registered, you can then log in and go to the "Download" section of our website to download the required components for this tutorial. In addition, you will have instant access to downloadable content on our site. They are available at the following address: http://www.topsolid.com/download/login.htm.

Step 1: Creating a Project

Creating a project template

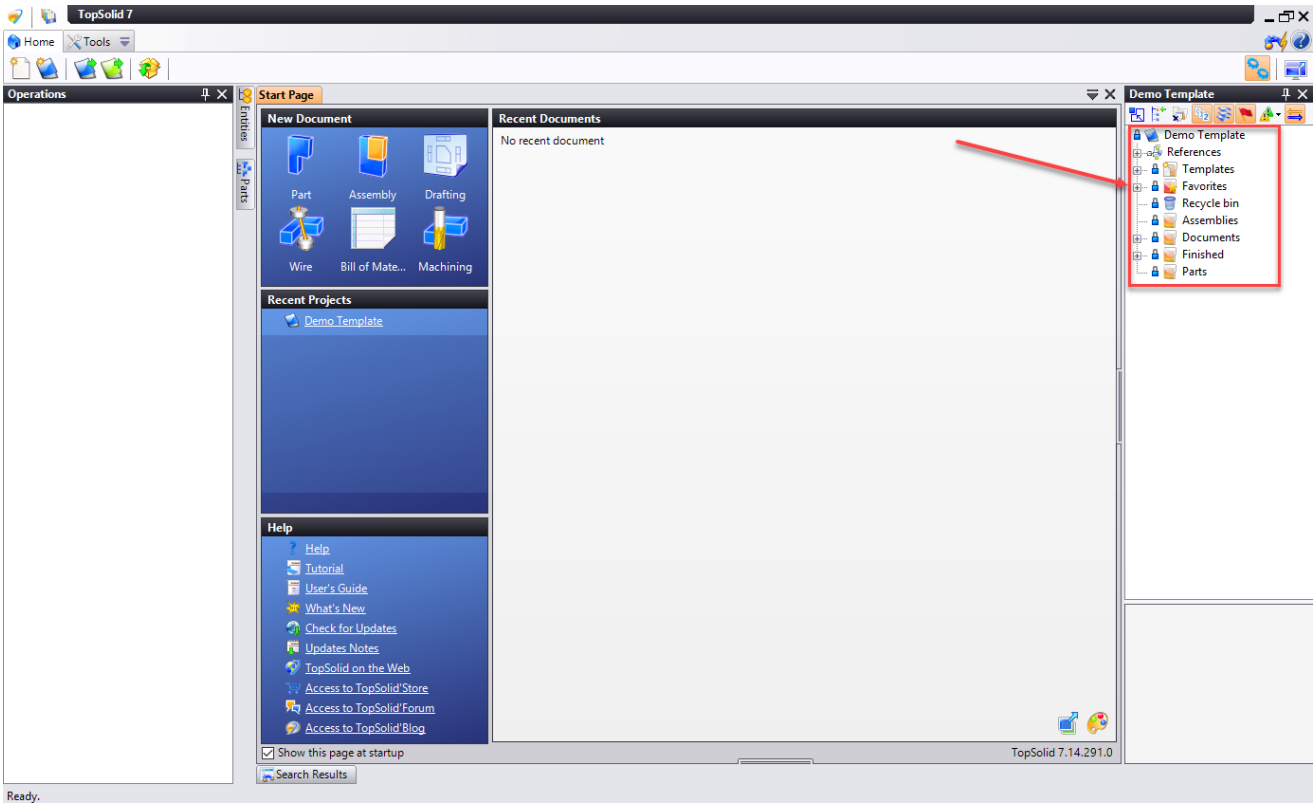
To begin, we will add the *TopSolid'Design Tutorial.TopPkg* package as a project template. A project template is a project which provides a basis for newly created projects. If a project template contains folders and files, these will automatically be created in a new project using this template.

- Import the *TopSolid'Design Tutorial.TopPkg* package by clicking on the **Import Project** icon from the **Home** tab (during installation, this file is installed in the "C:\Programs Files\ TOPSOLID \TopSolid 7.14\Samples\en" folder).

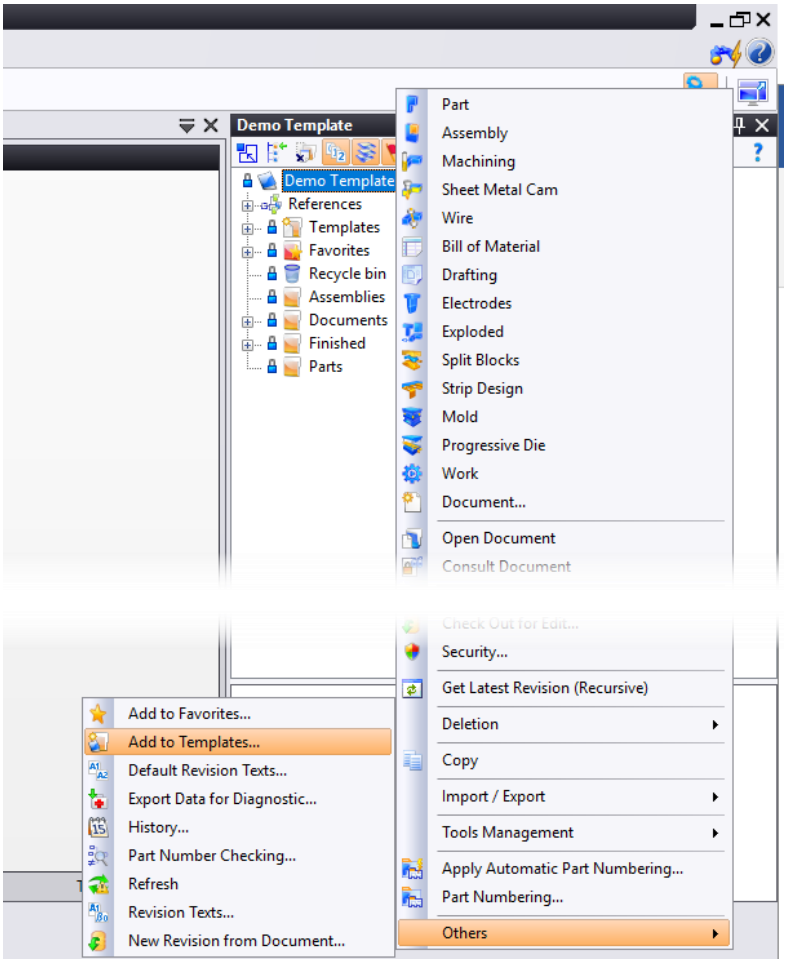


- Click on the green check mark to **confirm** the import.

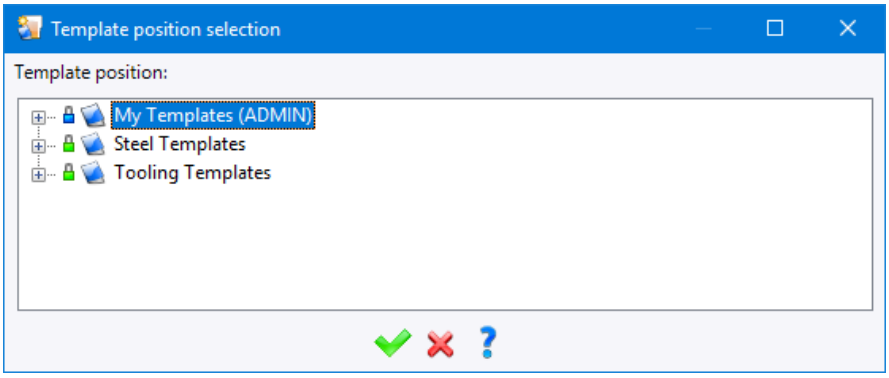
The *Demo Template* project is displayed on the left or right of the screen.



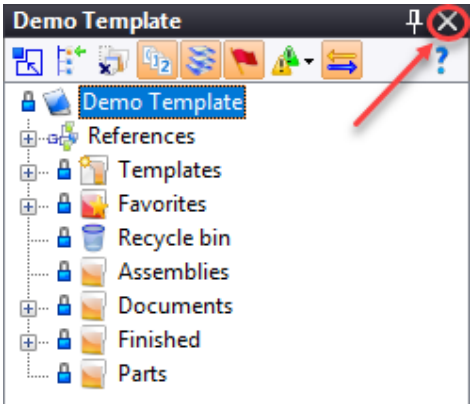
- Right-click on the project name and select the **Others** > **Add to Templates** command from the contextual menu.



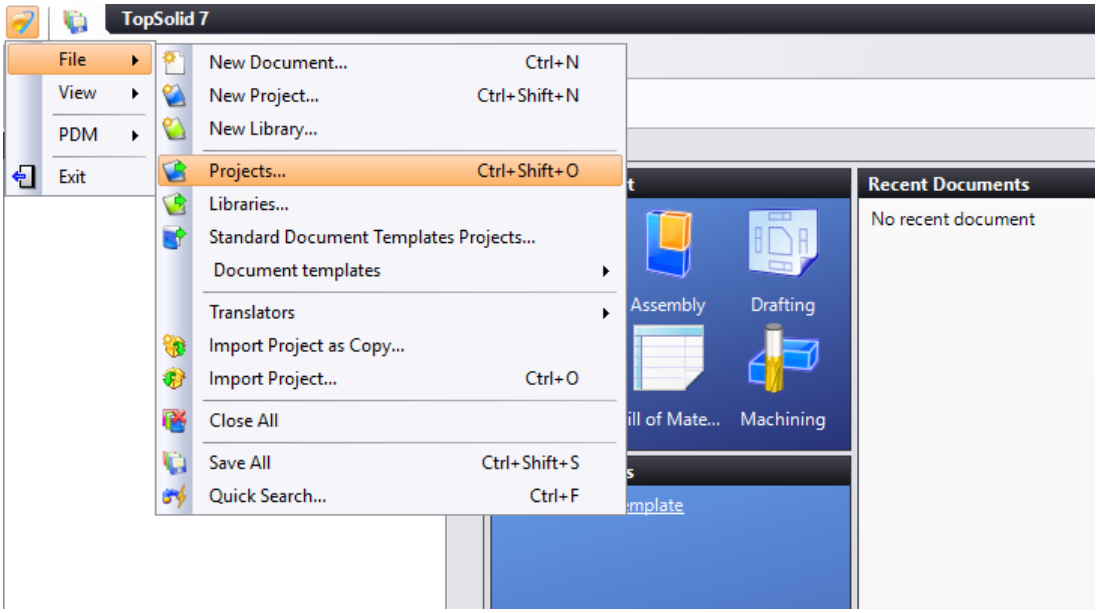
- Select where the template is to be saved as shown below and then click to **confirm** the operation.



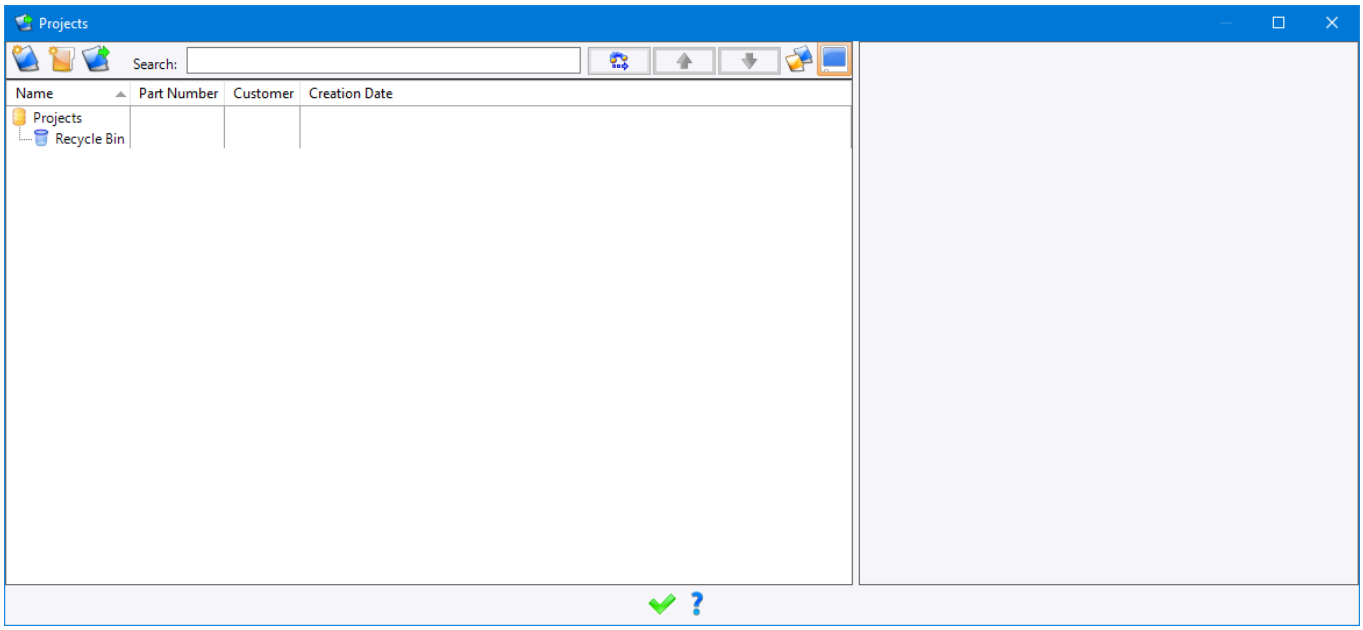
- **Close** the project tree dialog box by clicking on the icon.



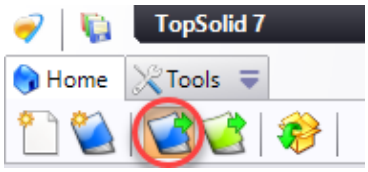
- Click on the **TopSolid** icon at the top left of the screen and then select the **File** > **Projects** command.



The **Projects** dialog box opens.

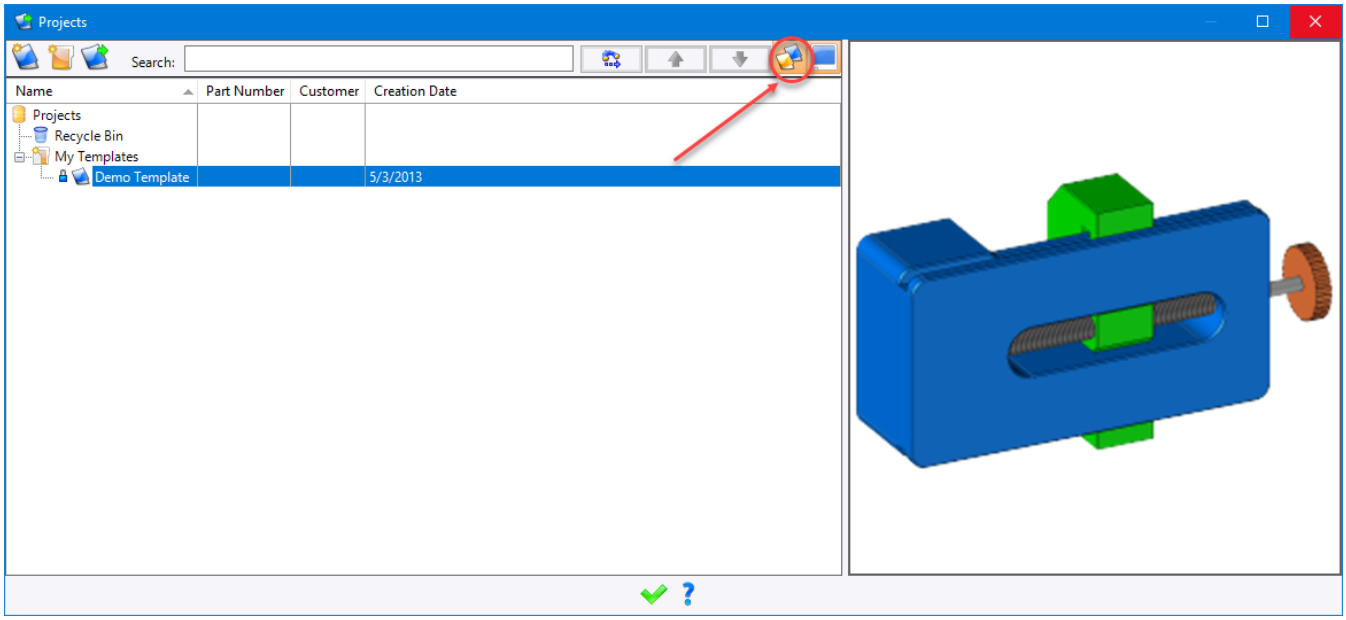


Note: You can also access the **Projects** dialog box by clicking the icon from the **Home** tab or using the **Ctrl + Shift + O** keyboard shortcut.



- From the **Projects** dialog box, click on the **Show Template Projects** icon.

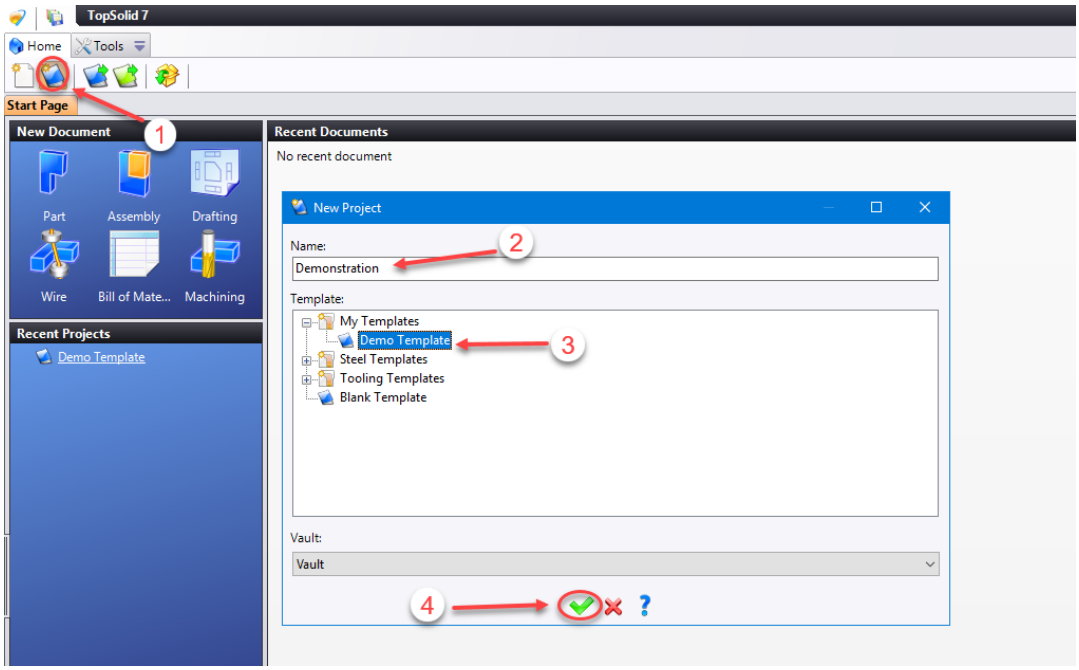
The *Demo Template* project has been added to the project templates.



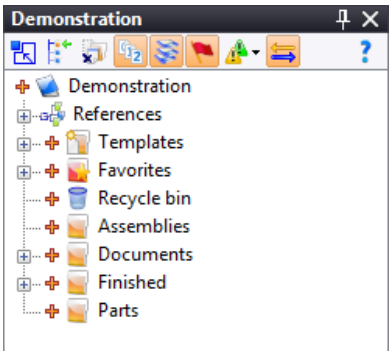
- **Close** the **Projects** dialog box by clicking on the icon.

Creating a new project

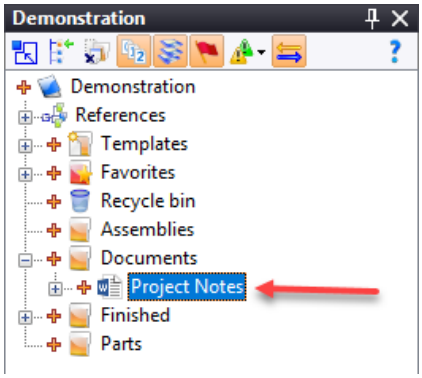
- From the **Home** tab, select the **New Project** command.
- In the dialog box that opens, rename the project *Demonstration.*
- Select the **Demo Template** project template.
- **Confirm** the operation by clicking on the icon.



Once confirmed, the following new project is created and appears automatically in the Project tree.

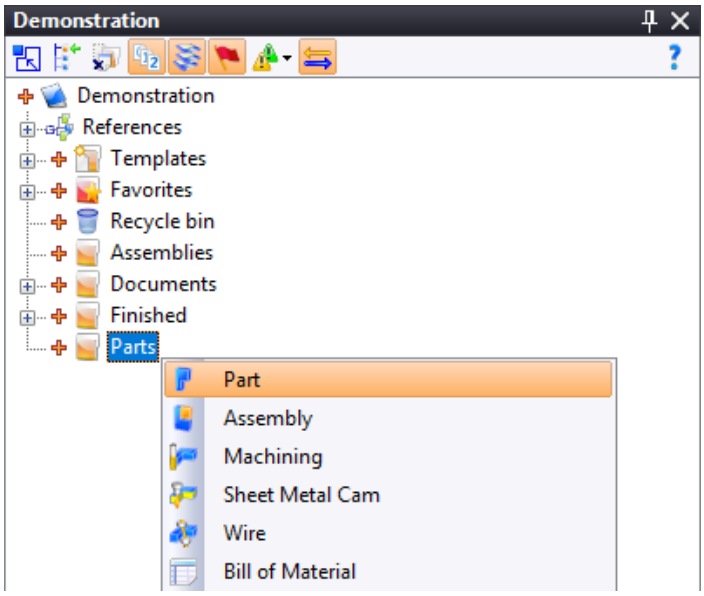


TopSolid'Pdm manages all documents related to the project. For example, in the picture below, a Microsoft Word document is listed within the project. You can double-click on it to open it. You can add additional files to be managed by dragging and dropping them into a PDM project.



Step 2: Creating a Part Document

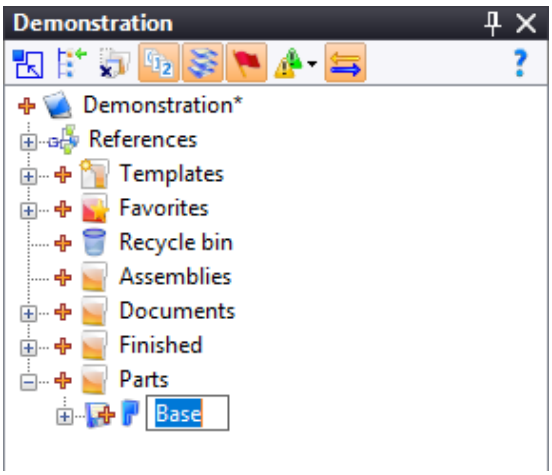
- In the Project tree, right-click on the *Parts* folder.
- Select the **Part** command from the contextual menu.



Note that the contextual menu is intuitive. It changes based on what you select. It is the fastest way to create standard documents in **TopSolid 7**.

The part document automatically opens.

- Rename the document *Base.*



As soon as the document is created, you can name it. However, you can rename it later if you like. The PDM will always manage it correctly regardless of what you do.

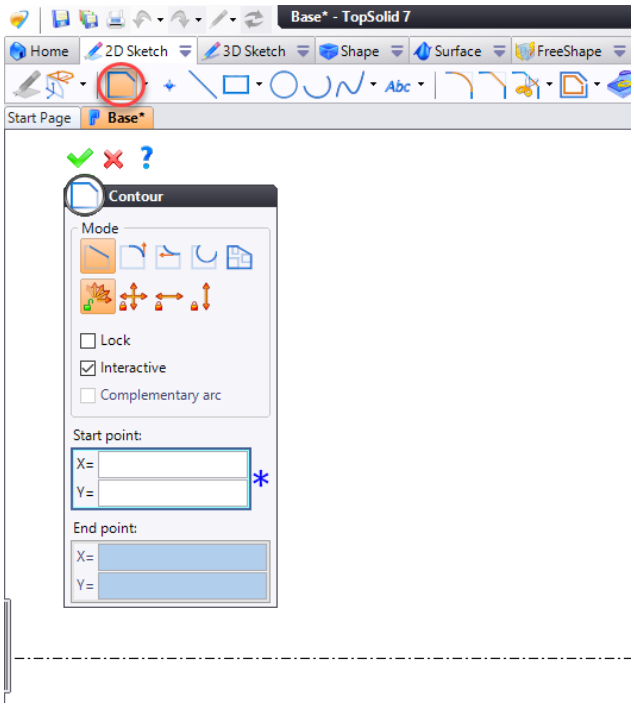
When modifying the name or making changes to a file, it will be preceded by the icon and followed by an asterisk *, indicating that the document has been changed (or created). Saving the document will remove the icon and asterisk.

Step 3: Designing the Base Support

In this section of the tutorial, you will learn how to design a simple part.

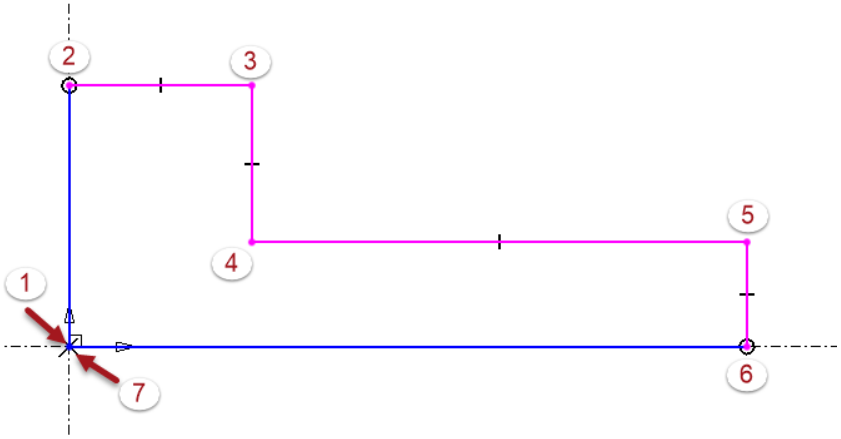
Creating the first sketch

- Go to the **2D Sketch** tab.
- Select the **Sketch** command.
- Select the **Contour** command.



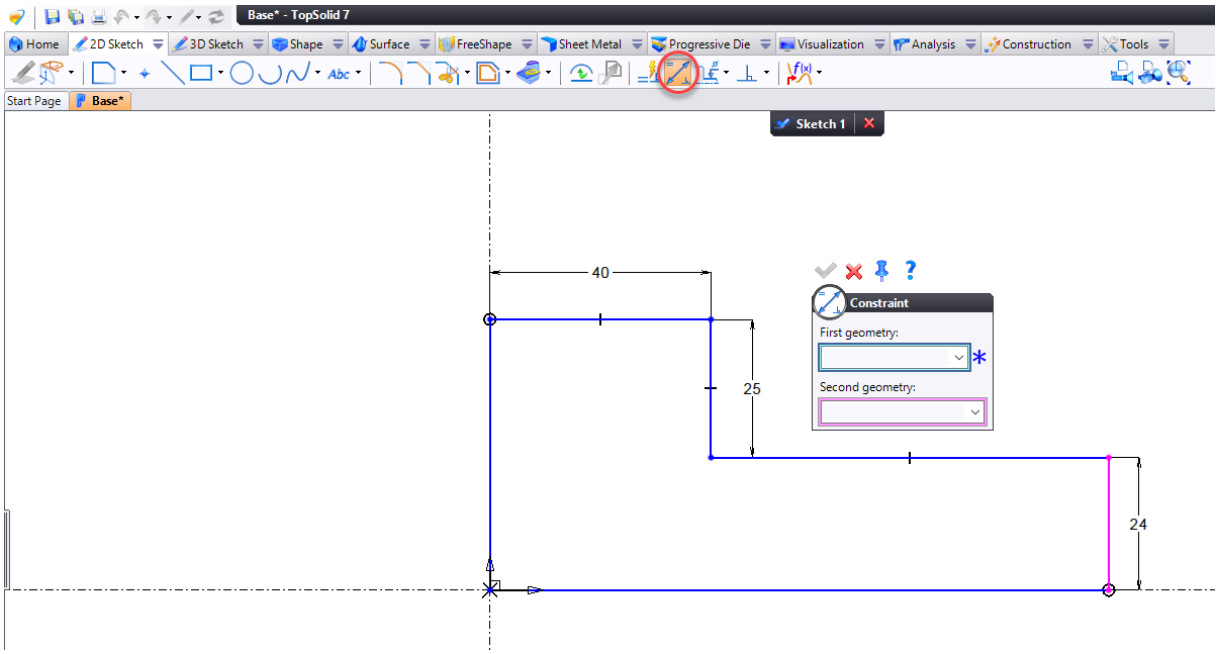
The **TopSolid 7** interface is designed to be as simple as possible. All curve creation tools are located in the **2D Sketch** tab while all shape tools are in the **Shape** tab, etc.

- Draw the contour by following the order shown below. Start at the frame origin (0,0) and continue clockwise to create the sketch.
- Click to **confirm** the contour.



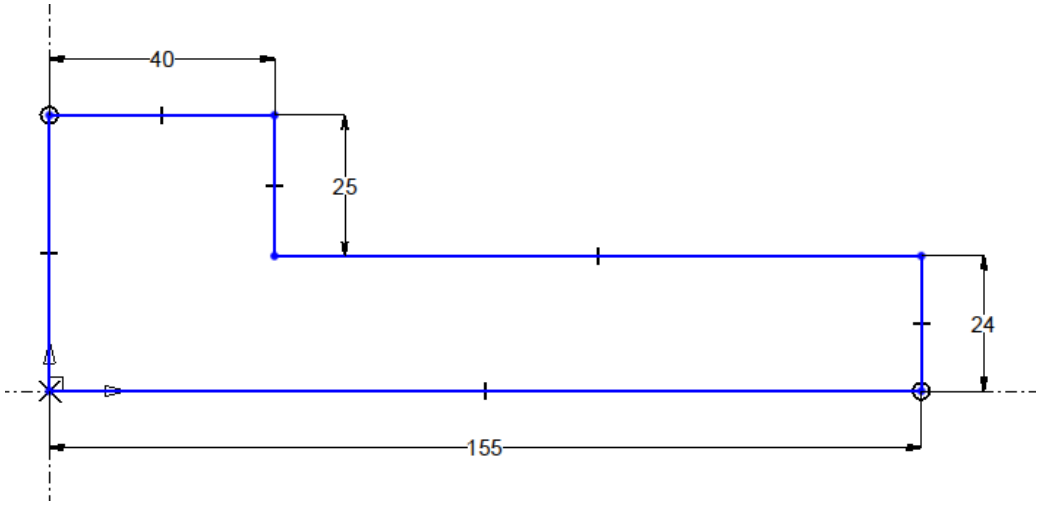
Note: The segments of the contour have two distinct colors. The blue color represents fully constrained geometry while the purple color represents under-constrained geometry. Knowing what is under-constrained will help you to determine the missing information.

- Select the **Constraint** command and then add the following dimensional constraints.



At this point, there is only one under-constrained segment left (shown in purple on the image above). If you select the segment with your mouse (left click and hold), you will see that there is still a degree of freedom here and you can drag the segment from the left to the right.

- Add the following last dimensional constraint.



The sketch should now be fully constrained.

- **Close** the **Constraint** dialog box by clicking on the icon.

Extruding the solid model

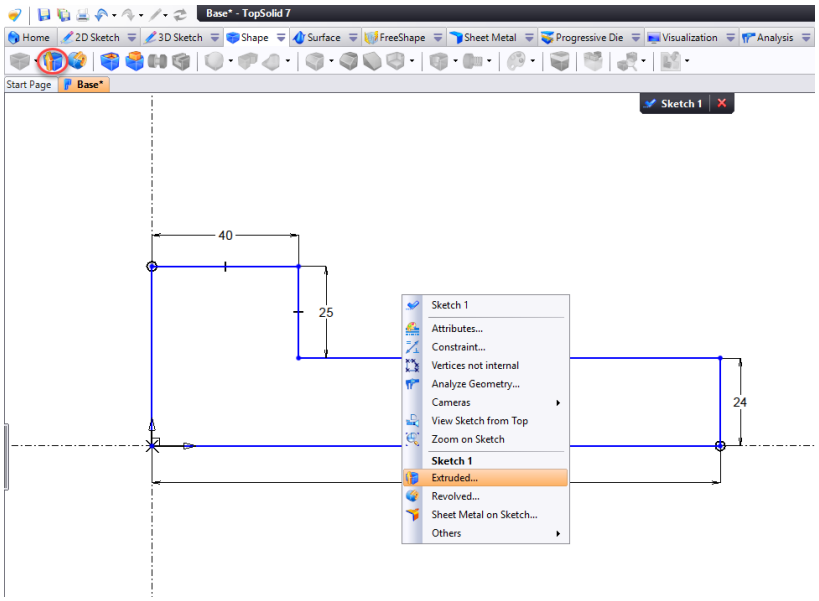
Now you will create your first extrusion. To access the extrusion function, you can either select the **Extruded** command from the **Shape** tab or right-click in the graphics area to select the **Extruded** command from the contextual menu. This is another great example of the dynamic intelligence of **TopSolid 7**.

The contextual menu will change based on the operation you are currently doing and will provide you with commands adapted to the task at hand.

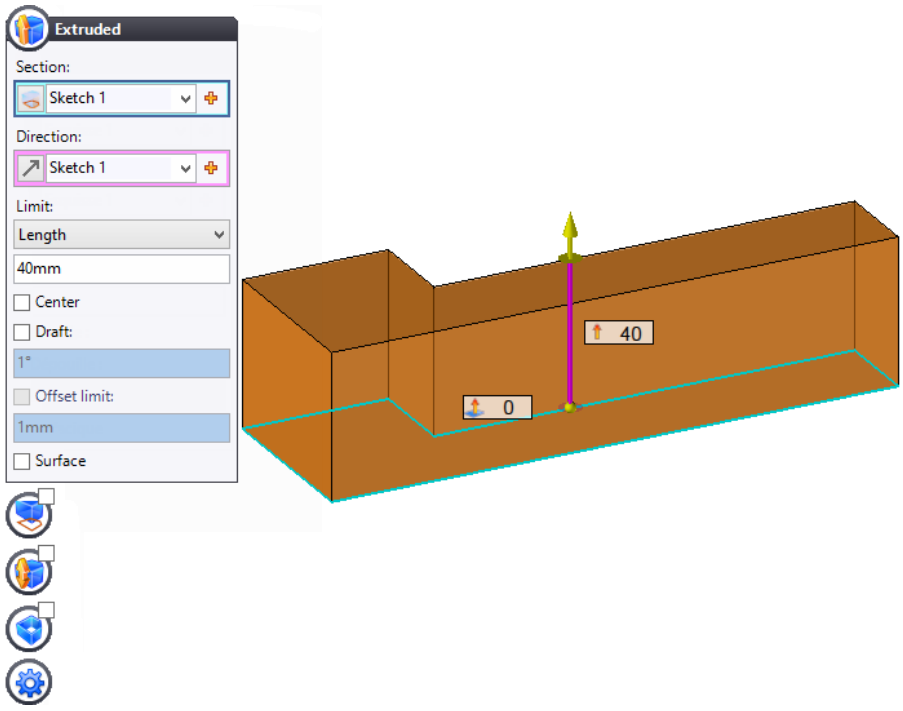
- Go to the **Shape** tab.
- Select the **Extruded** command from the icon bar.

Or

- Right-click in the graphics area and select the **Extruded** command from the contextual menu.

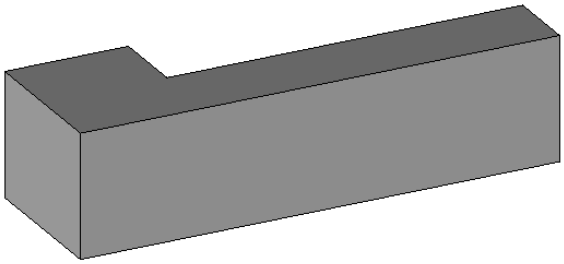


- Select the yellow arrowhead and move it up or down while holding down the left mouse button until the extrusion value is *40mm*.



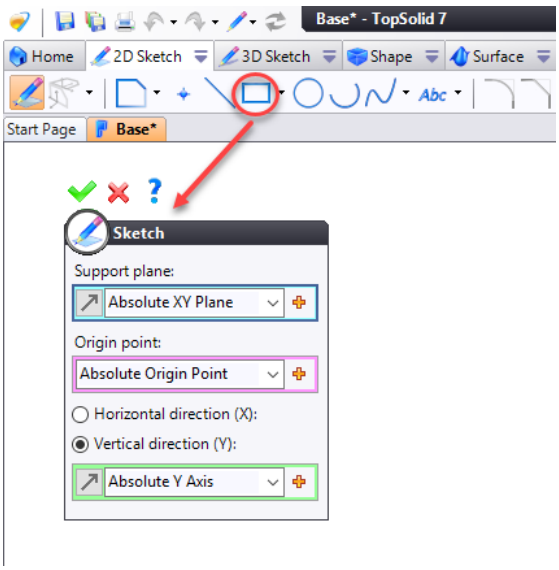
Notice how dynamic everything is in **TopSolid 7**. For example, you instantly get a dynamic preview of the modifications made to an object before applying them.

- **Confirm** the extrusion by clicking on the icon.



Creating a rectangle

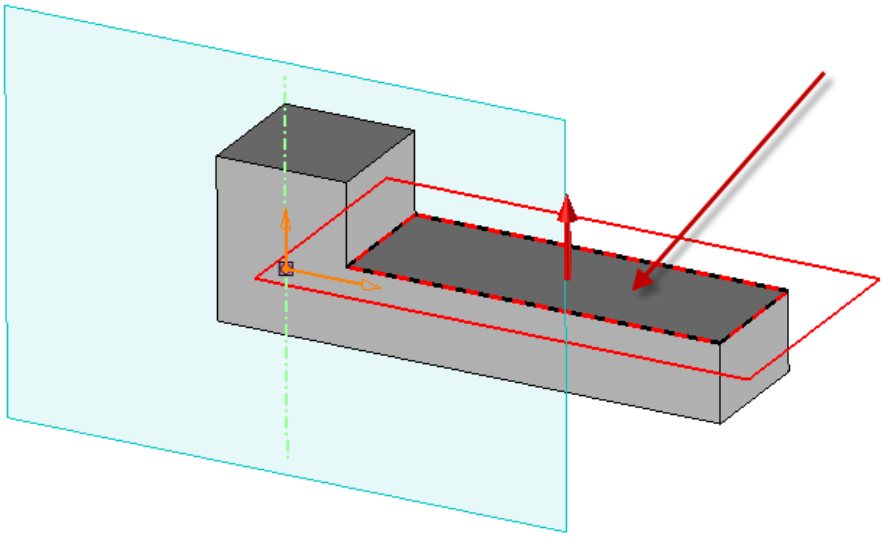
In this section, you will create another sketch. To do this, you must first indicate the support plane on which **TopSolid** must create the sketch.



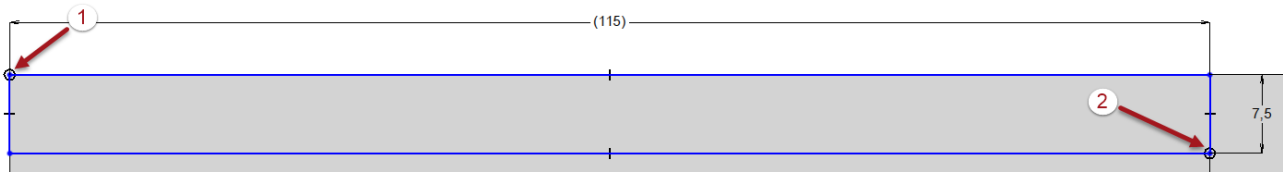
- Go to the **2D Sketch** tab.
- Select the **Rectangle** command.

As the part document already contains a sketch, **TopSolid** asks for a support plane for the second sketch.

- Select the face shown below as the **support plane**.



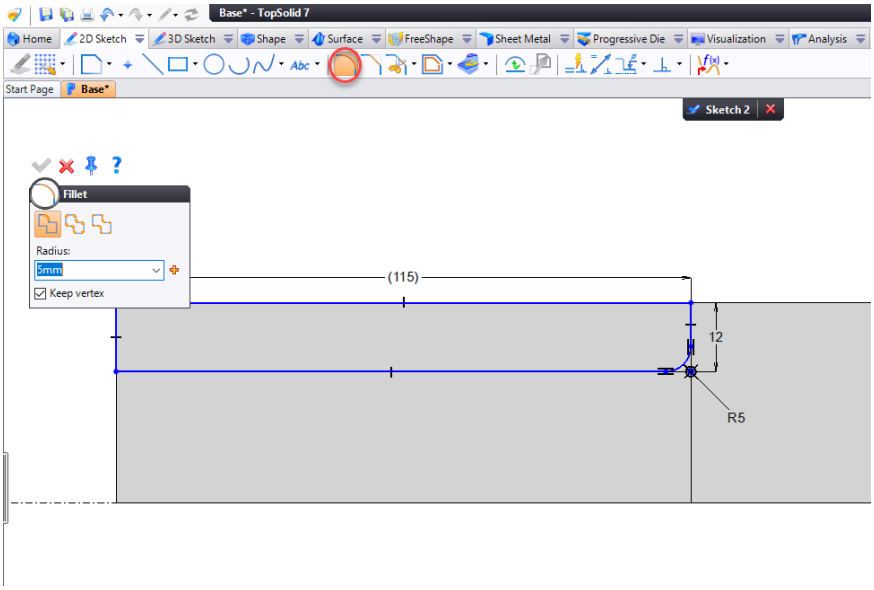
- **Confirm** the operation by clicking on the icon.
- Create the rectangle by selecting the points shown below, in the order specified. Be sure to correctly connect the points as shown in the model below so that the sketch will be fully constrained by default.
- **Close** the **Rectangle** dialog box by clicking on the icon.



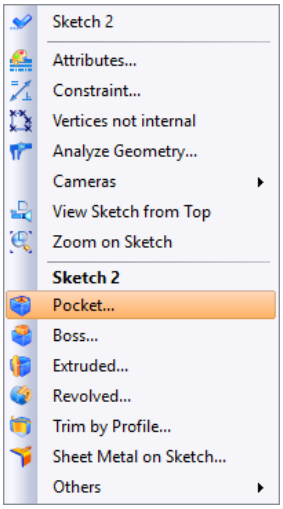
- Double-click on the smaller dimension to edit it, adjust its value to *12mm*, and then click on the icon to **confirm** the operation.

Creating a pocket

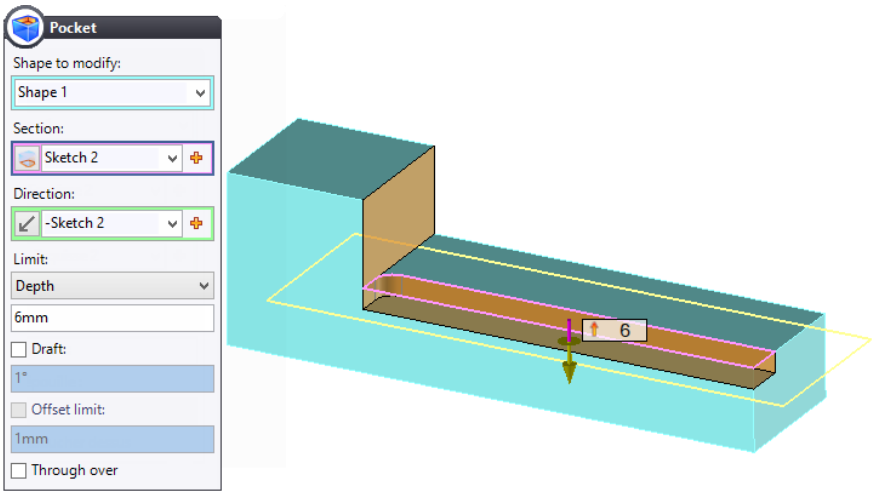
- Select the **Fillet** command from the **2D Sketch** tab.
- Adjust the radius value to *5mm*.
- Click on the lower right corner of the sketch to apply the fillet.



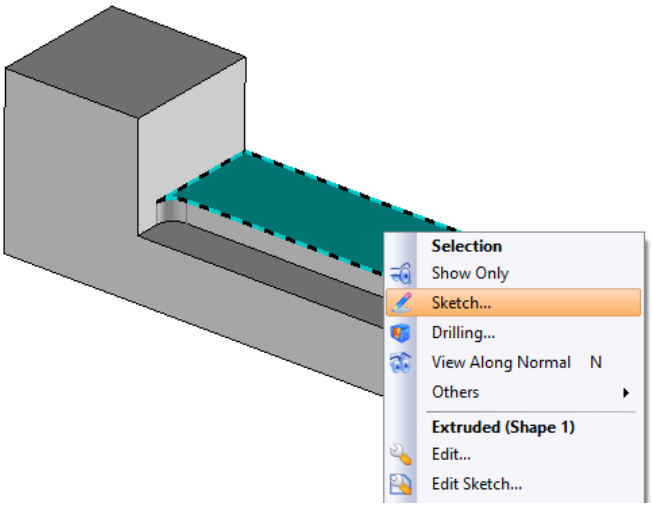
- **Close** the **Fillet** dialog box by clicking on the icon.
- Right-click in the graphics area and then select the **Pocket** command.



- Select the yellow arrowhead and move it up and down so that the pocket is 6mm deep, and then click to **confirm** the operation. You can also double-click on the label next to the arrow to change its value.



Creating a contour

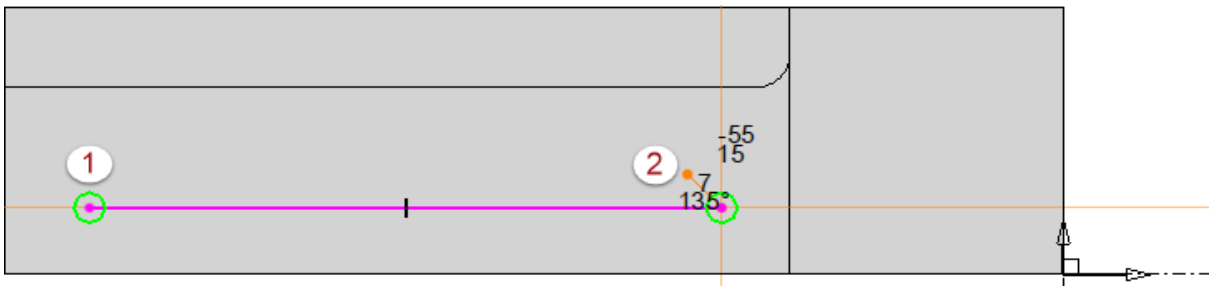


- Right-click on the face shown opposite.
- Select the **Sketch** command.

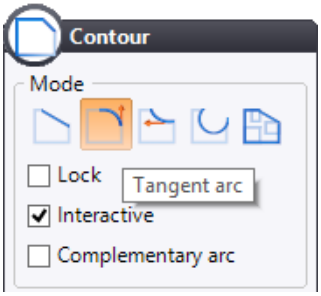
Here is yet another way to create a sketch dynamically.

Also note that the **Contour** command was selected automatically, which is perfect because that is the tool we are going to use in the next step.

- Create the contour by selecting the points shown below, in the order specified.

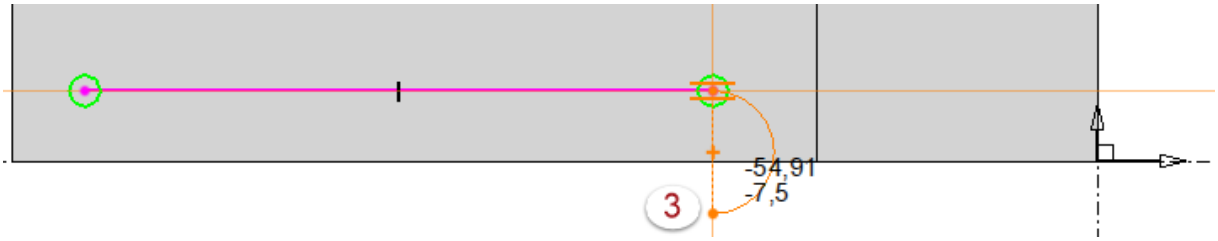


- Select the **Tangent arc** mode as shown here.



Note: You can also switch to the **Tangent arc** mode via the contextual menu.

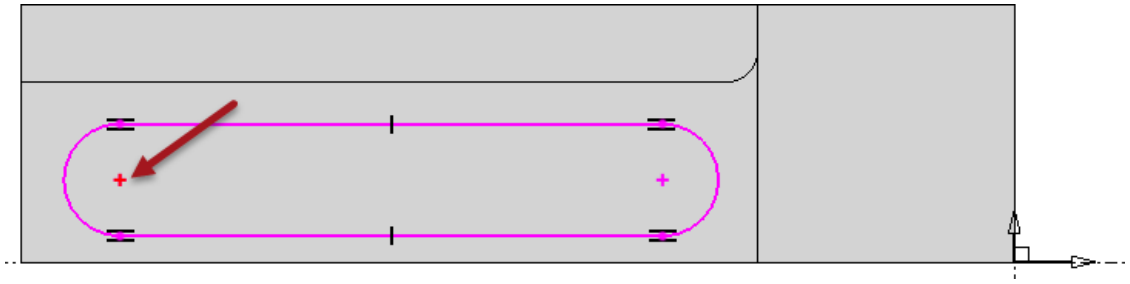
- Select the point shown below to swing a 180-degree tangent arc.



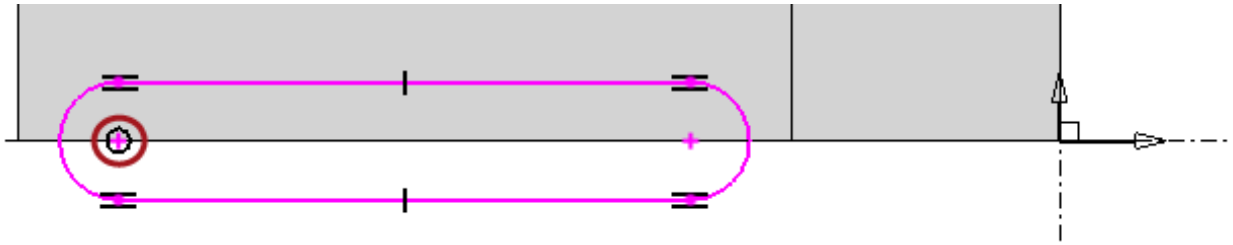
- Place the cursor directly under the first point (as shown by point 4 below). By doing this, a dynamic reference should appear. Left-click to position the point and dynamically add the relation.

- Select the **Tangent arc** mode again and select the top horizontal line to close the sketch as shown below. Click to **confirm** the contour.

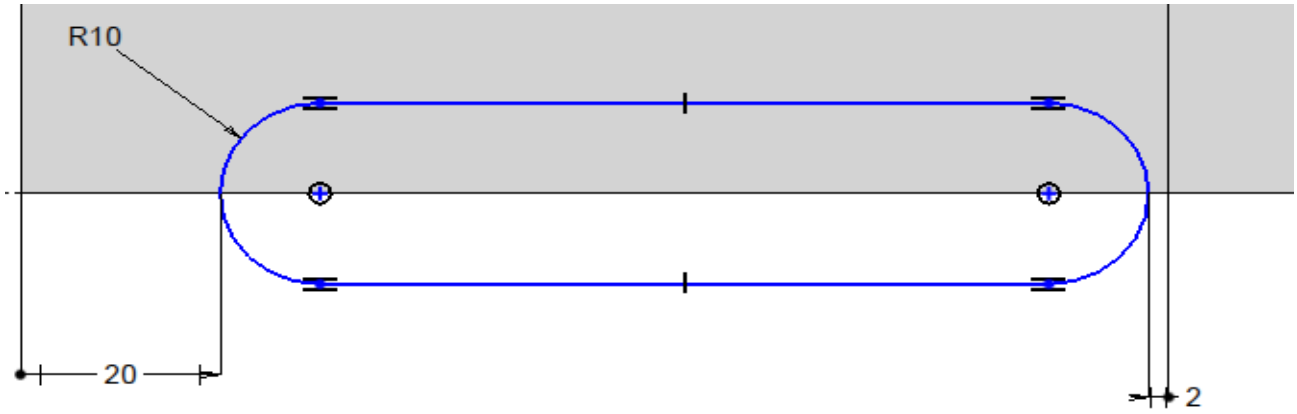
- Select the center point of the circle arc to the left and drag your mouse up and down without releasing the left mouse button. This shows the automatic constraints in action.
- Drag the sketch until it looks like the image shown below.



- Select the same point and drag it down to the bottom edge of the model and hover a second before you release the left mouse button. If done correctly, you will create an automatic coincident constraint between the center point of the circle arc and the edge of the model as shown below.

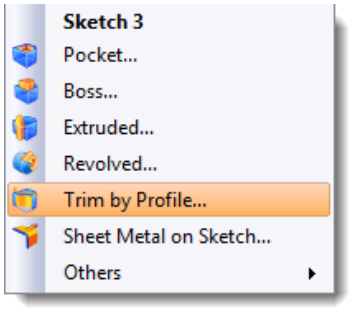


- Follow the same procedure to create the second coincident constraint between the other center point of the circle arc and the edge of the model.
- Add the following three dimensional constraints.

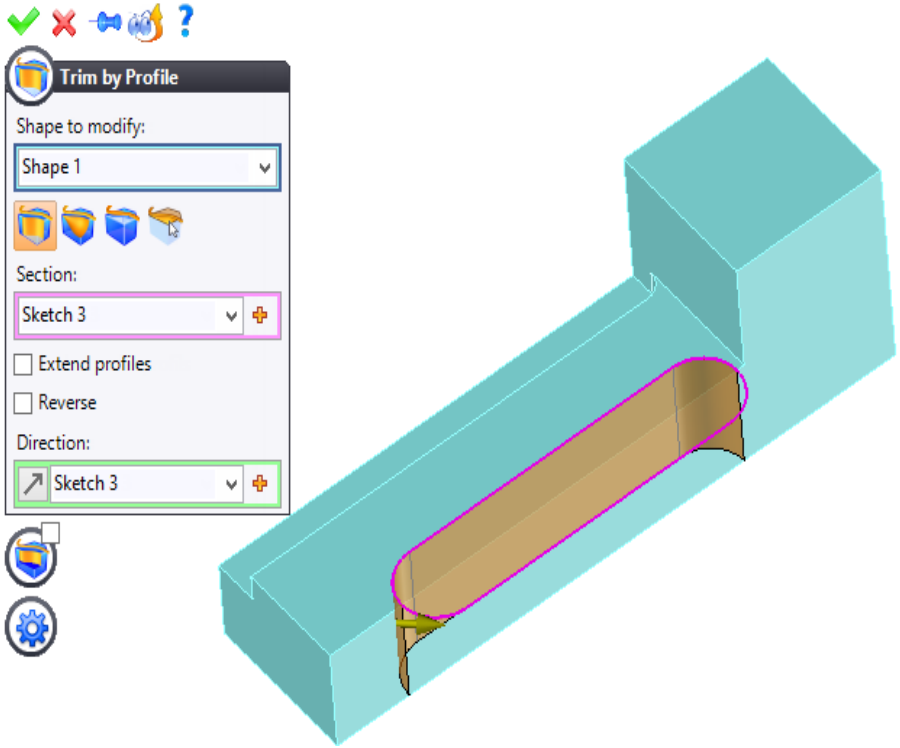


- **Close** the **Constraint** dialog box by clicking on the icon.

Trimming by profile

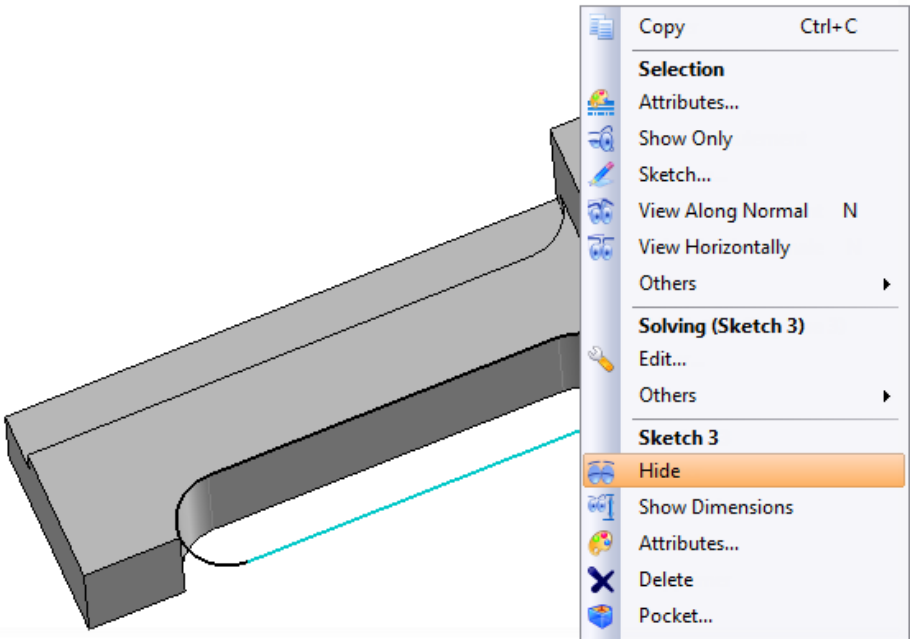


- Right-click in the graphics area and select the **Trim by Profile** command from the contextual menu.
- Make sure the trimming arrow is pointing towards the inside of the sketch, and then click to **confirm** the trimming operation.

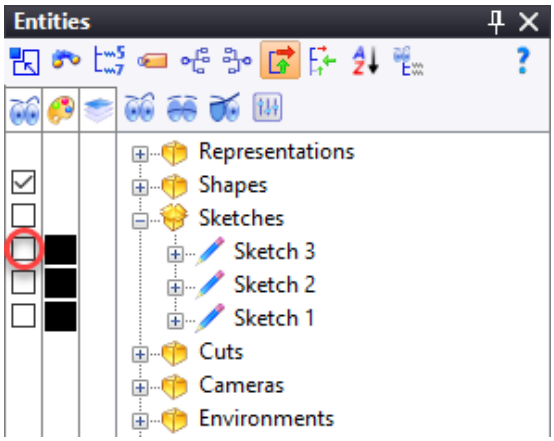


Note: If the sketch is still shown on the screen after the operation has been confirmed, you can hide it by doing one of the following:

- From the graphics area, right-click on the sketch and select the **Hide** command.

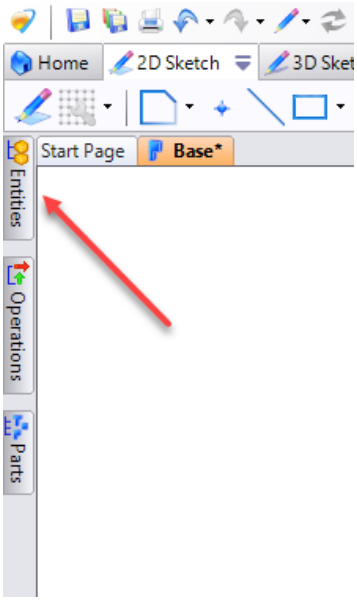


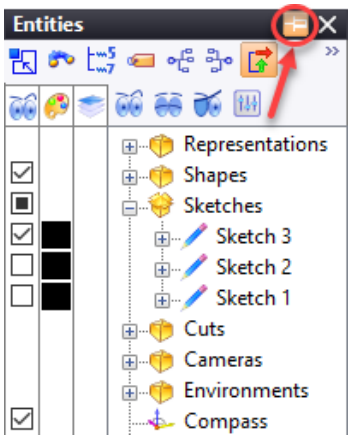
- From the Entities tree, open the **Sketches** folder and uncheck the box in front of the sketch you want to hide.



Note:

- If the Entities tree is hidden, click on the corresponding tab on the left or right of the screen and then click on the icon to pin the tree.

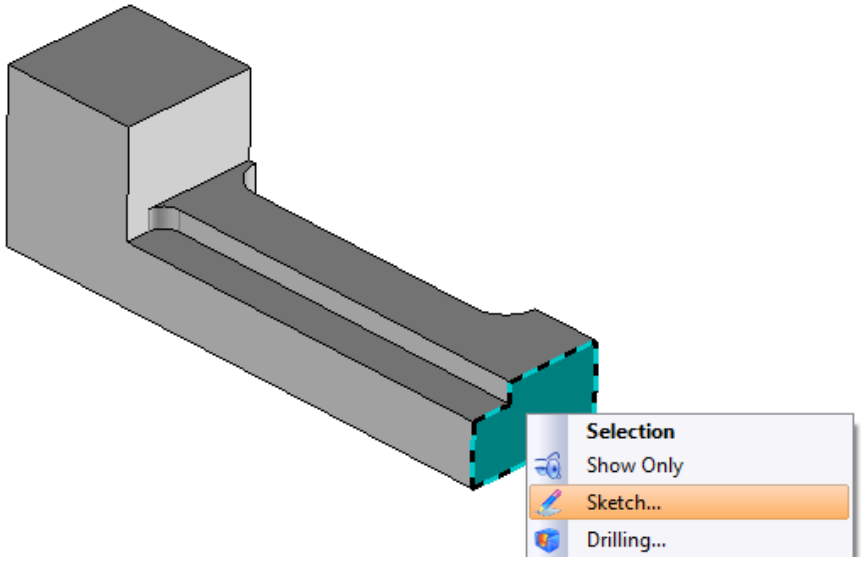




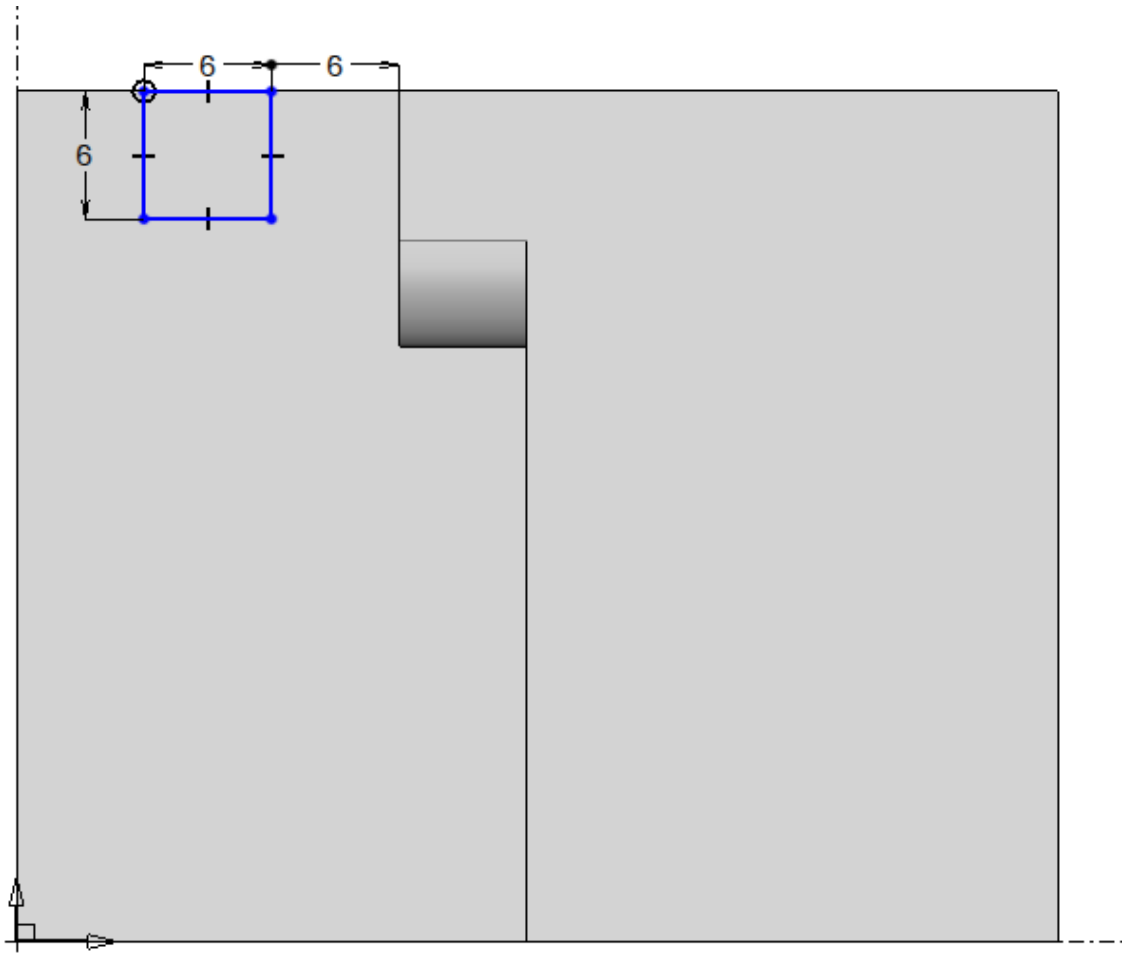
- If the Entities tree is closed, click on the **TopSolid** icon at the top left of the screen and then select **View** > **Entities** to open the tree. The Entities tree appears on the left or right of the screen.

Adding another rectangle for trimming

- Right-click on the face shown below and create a new **sketch** on it.

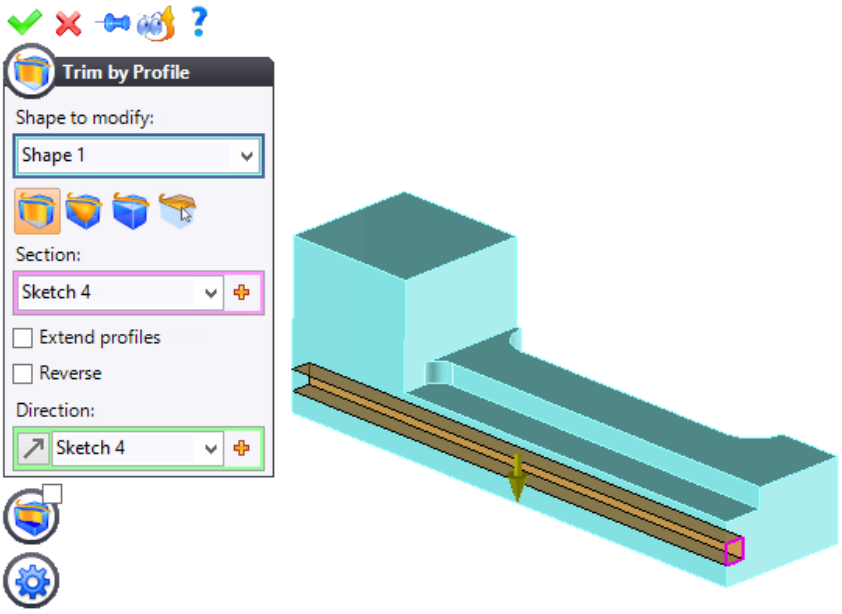


- Draw the following **rectangle** and constrain it as shown below and then **close** the dialog box.



- Right-click in the graphics area and select the **Trim by Profile** command.

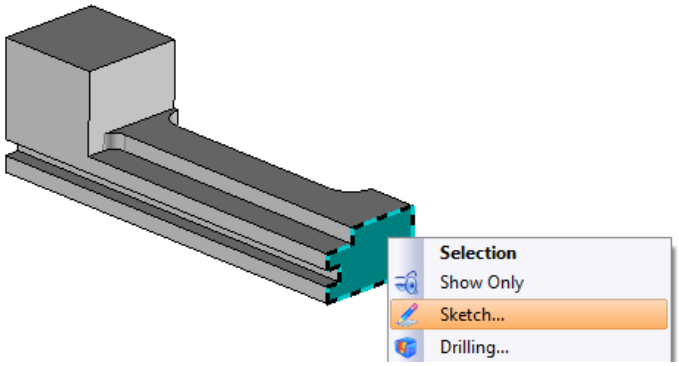
- Make sure that the trimming arrow is pointing toward the inside of the sketch, and then click to **confirm** the operation.



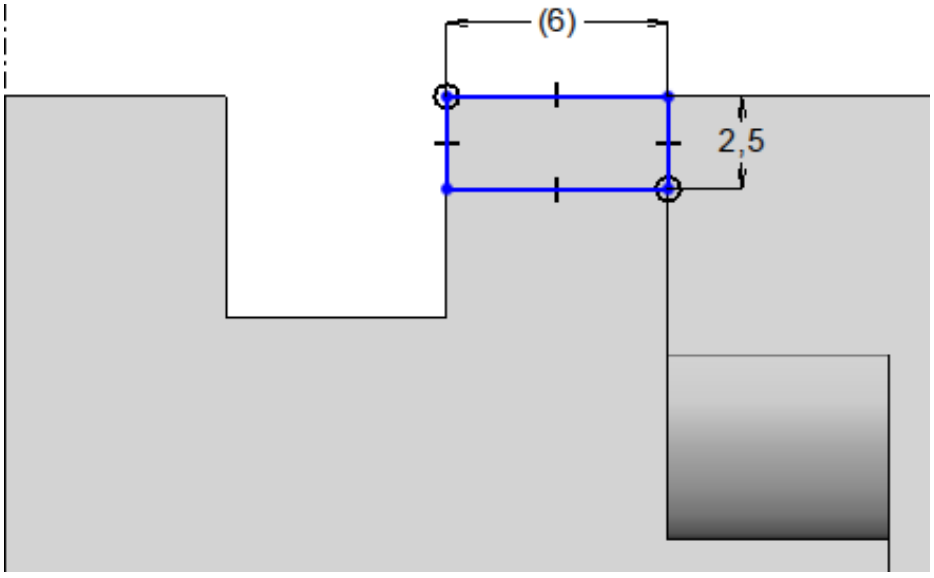
- Display the Entities tree. If it is hidden or closed, proceed in the same way as before to hide sketch 4.

Creating a new sketch and adding a pocket operation

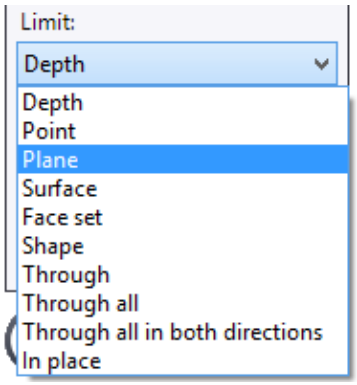
- Right-click on the face shown opposite to create a new **sketch** on it



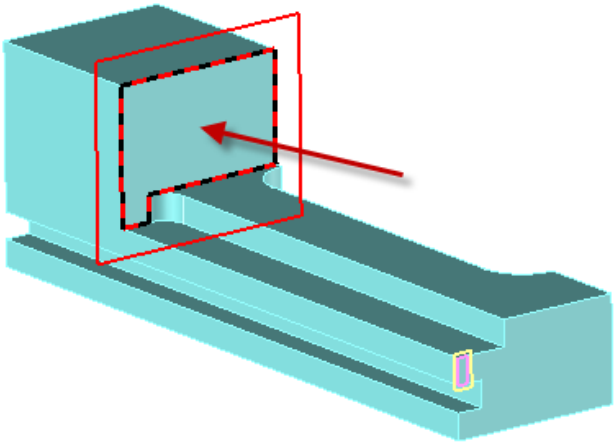
- Draw the following **rectangle** and constrain it as shown in the image below.



- Right-click in the graphics area and select the **Pocket** command. Change the **limit** type to **Plane**.



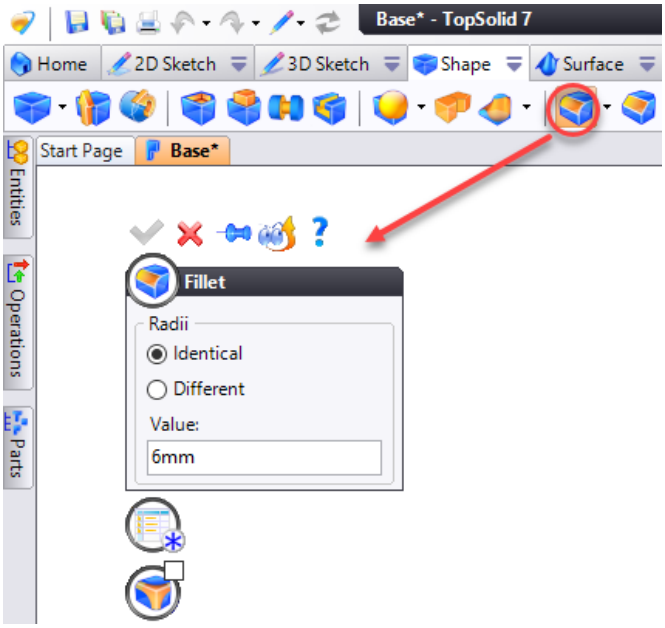
- Select the planar face of the model as shown below, and then click to **confirm** the operation.



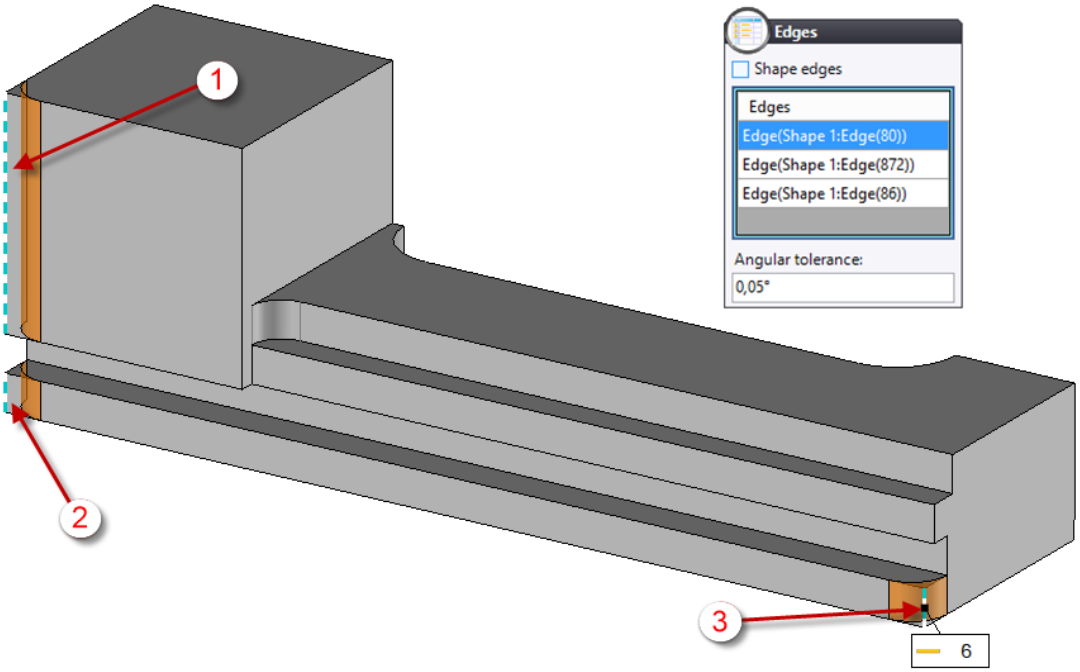
The depth of the pocket will therefore always be limited to the location of that specific face.

Adding some fillets

- Go to the **Shape** tab and select the **Fillet** command.
- Enter the radius value of the fillet as *6mm*.



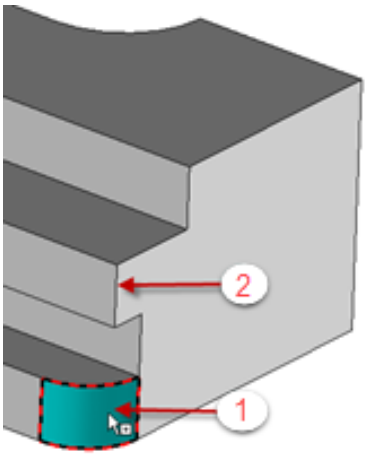
- Open the **Edges** dialog box and then select the three edges shown on the model below.
- Click to **confirm** the fillets.



Dragging and dropping a fillet

In the next step, you are going to use the drag and drop technique in **TopSolid 7**. With this technique, you are able to complete your modeling tasks quickly and easily. To use drag and drop, hold down the **Ctrl** button on your keyboard, and with the left mouse, select the element of the model that you want to copy and paste in a new area. In this case, you will drag the 6mm fillet that you added in the previous step and drop it onto the vertical edge located above the fillet.

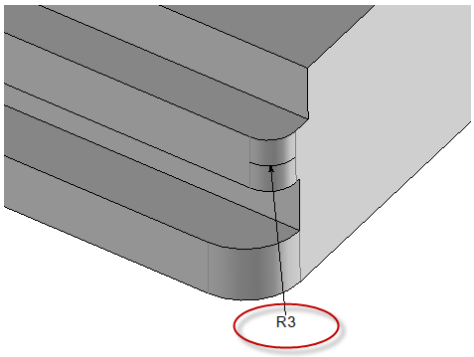
- Hold down the **Ctrl** key and then click on the fillet shown opposite (1).
- While still holding down the **Ctrl** key, drag and drop the fillet on the edge shown opposite (2).
- After selecting the edge, release the mouse button.

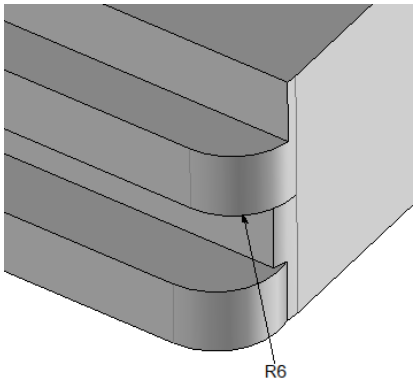


Note: When several elements are close to each other or overlap each other, or when the selected element does not correspond to the one you want (selecting a face instead of an edge for example), you might need to use "rotary picking". To do this, just put the mouse cursor near the element that you want to select. Then click and hold down your left mouse button. Now while you are holding down the left mouse button, start tapping your right mouse button. Keep tapping the right mouse button until the edge or face that you are trying to select is highlighted. Then, release the left mouse button.

Editing an operation

- Double-click on the face of the fillet that you just added. This will bring up the driving dimension (radius value).
- Double-click on the dimension and adjust it to *3mm*.

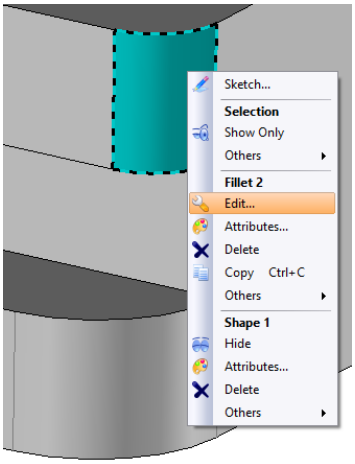




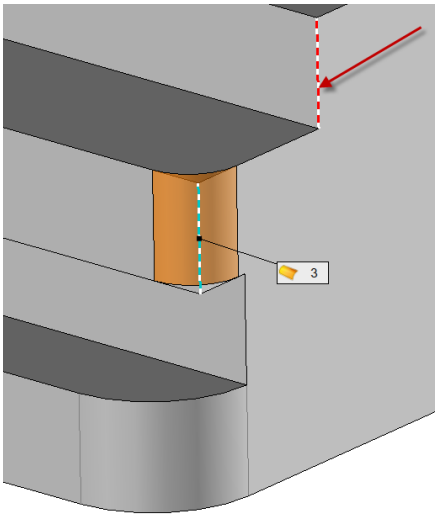
- Double-click on the fillet face again to hide the driving dimension.

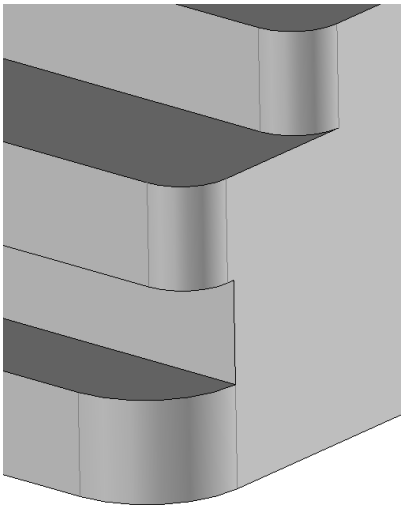
There are many ways to work in **TopSolid 7**. Now you will edit the definition of a fillet and add an additional reference edge to the same fillet.

- Right-click on the face of the 3mm fillet that you created in the previous step and select the **Edit** command.



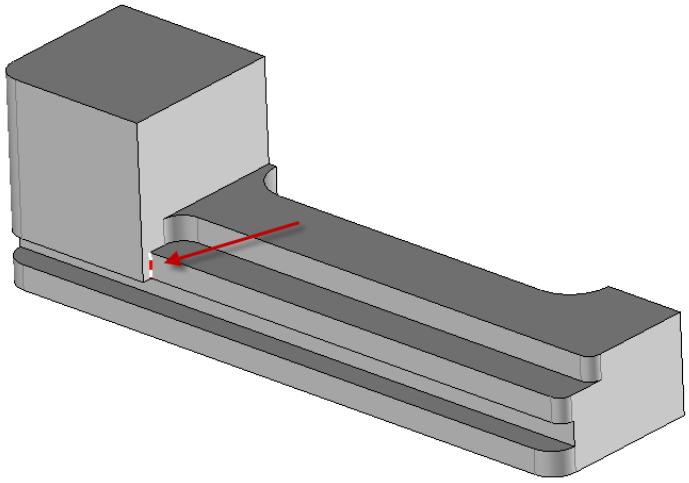
- Select the edge shown below to add it to the definition of this fillet. Once selected, click to **confirm** the fillet.





Using the contextual menu to add a fillet

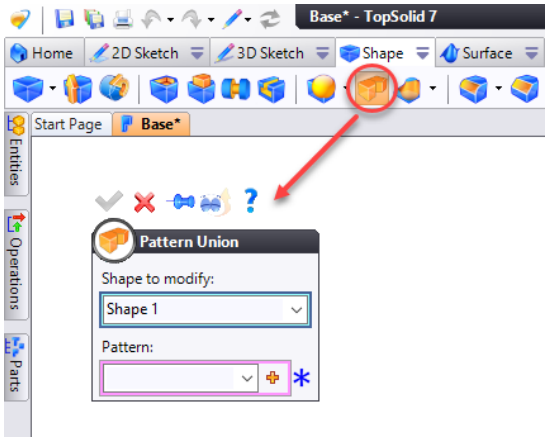
- Right-click on the edge shown below, select the **Fillet** command from the contextual menu, and then add a *5mm* fillet to the edge of the model.



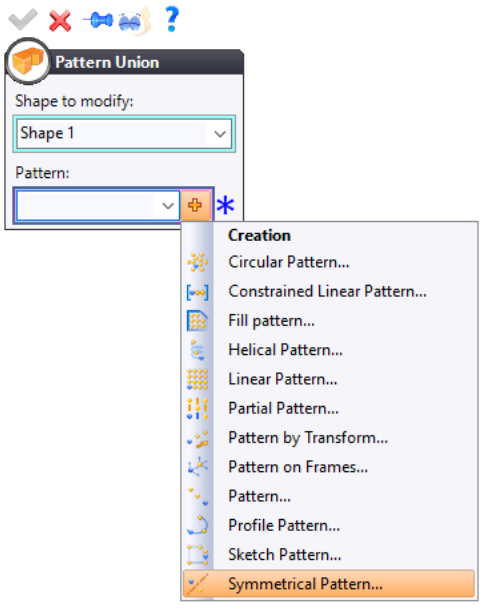
Creating a pattern union

Design intent can be useful in making a design easier and quicker to complete. So far, we have designed half of the part. In the following step, you will learn how to mirror what is already designed.

- Go to the **Shape** tab and select the **Pattern Union** command.



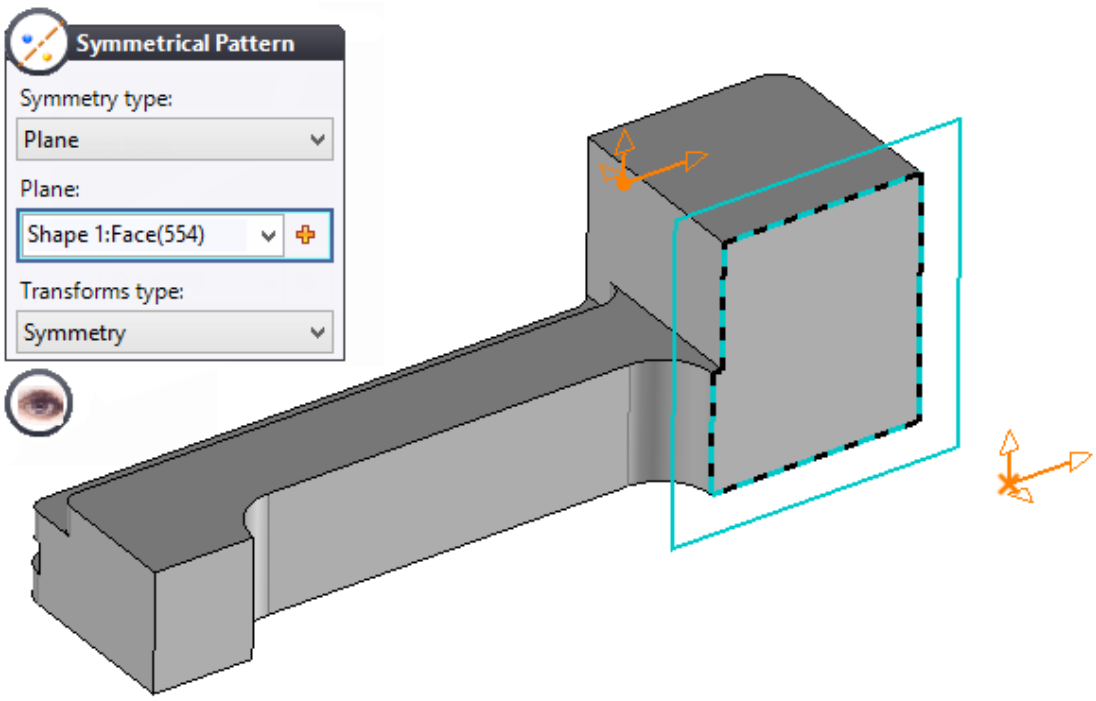
- Click on the icon to the right of the **Pattern** field in the dialog box.
- Select **Symmetrical Pattern** from the drop-down menu.



Note: Since you have not previously created any pattern in this document, you have to define a new pattern here. Note that you can dynamically create a new pattern within the **Pattern Union** command itself. Moreover, you can reuse the created pattern at a later date by selecting it directly from the drop-down list if needed.

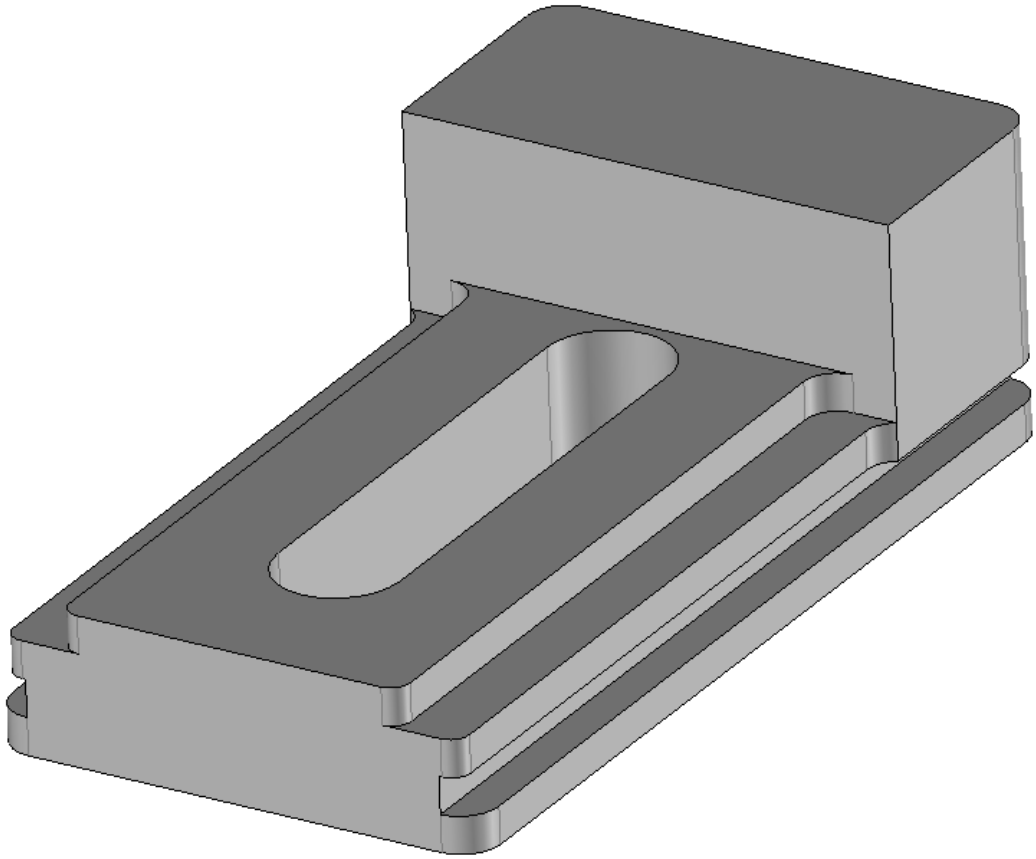
- From the drop-down menu, select **Plane** as the symmetry type.

- Select the planar face shown below as your symmetry plane and select **Symmetry** as the transform type.



- Click to **confirm** the symmetrical pattern.
- Click to **confirm** the pattern union.

You should now see the entire part on your screen. Note that **TopSolid 7** is smart enough to simplify the common faces on the completed part. This means that there are not any unnecessary edges on the model.

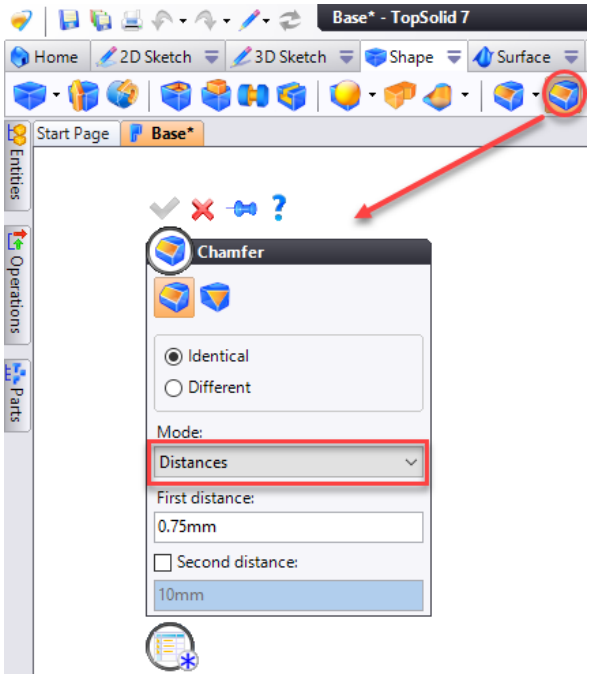


Creating a chamfer

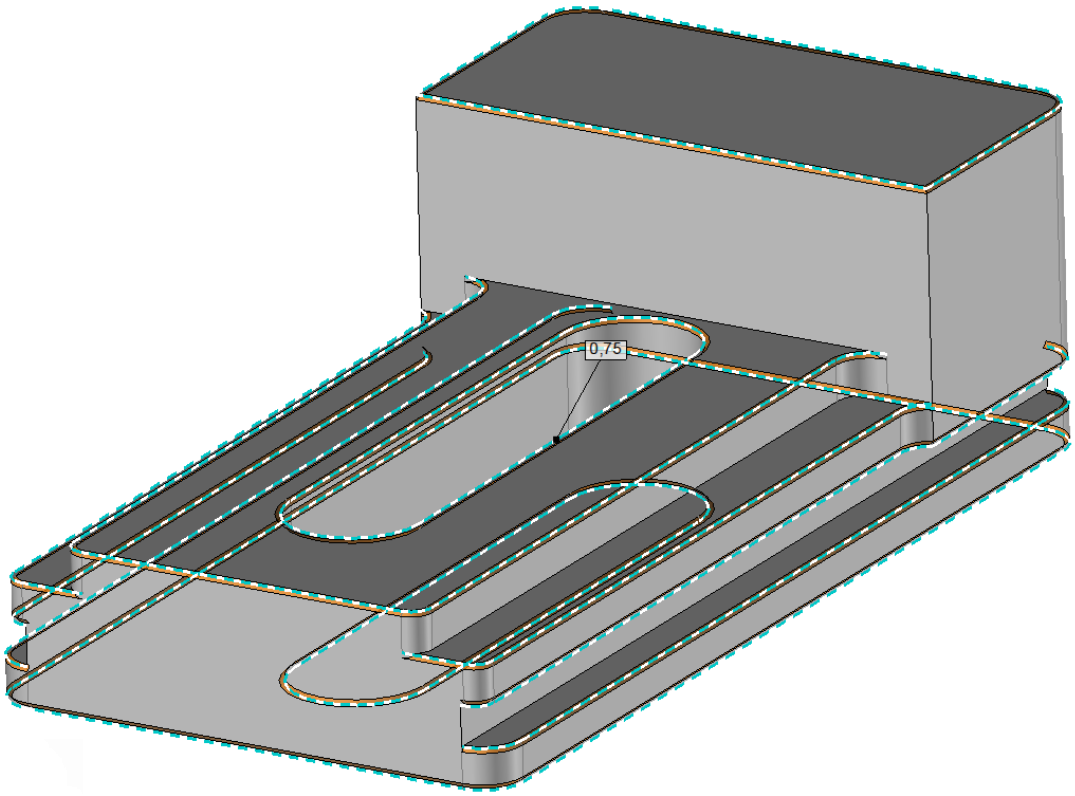
Now you will learn how to break some edges using the **Chamfer** command.

- Go to the **Shape** tab.
- Select the **Chamfer** command.
- Change the current mode to **Distances**.
- Adjust the chamfer size to *0.75mm*.

Note: You can also right-click in the graphics area or on an edge of the model to directly have access to the **Chamfer** command from the contextual menu.



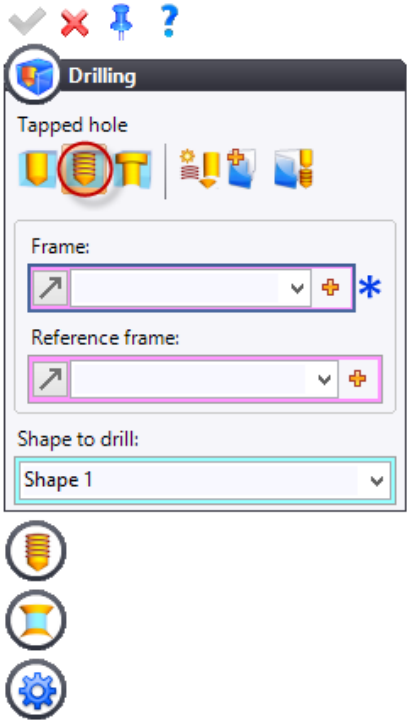
- Select all of the edges shown below to apply the chamfers. Once selected, click to **confirm** the operation.



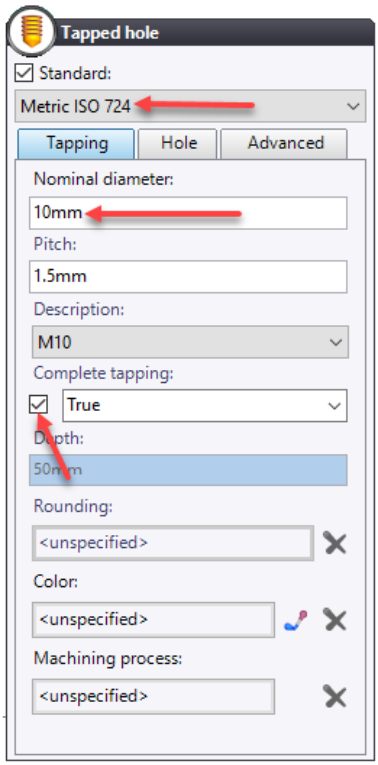
Adding a drilling

To complete this design, we need to add a single tapped hole.

- From the **Shape** tab, select the **Drilling** command.
- Click on the **Tapped hole** icon in the dialog box.

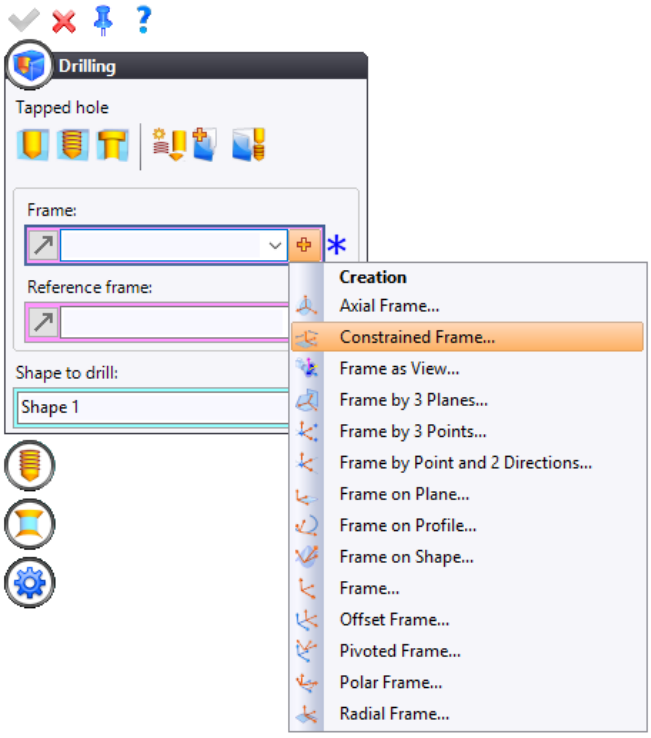


- Expand the tapping options by clicking on the icon.
- Set the tapping options as shown below.

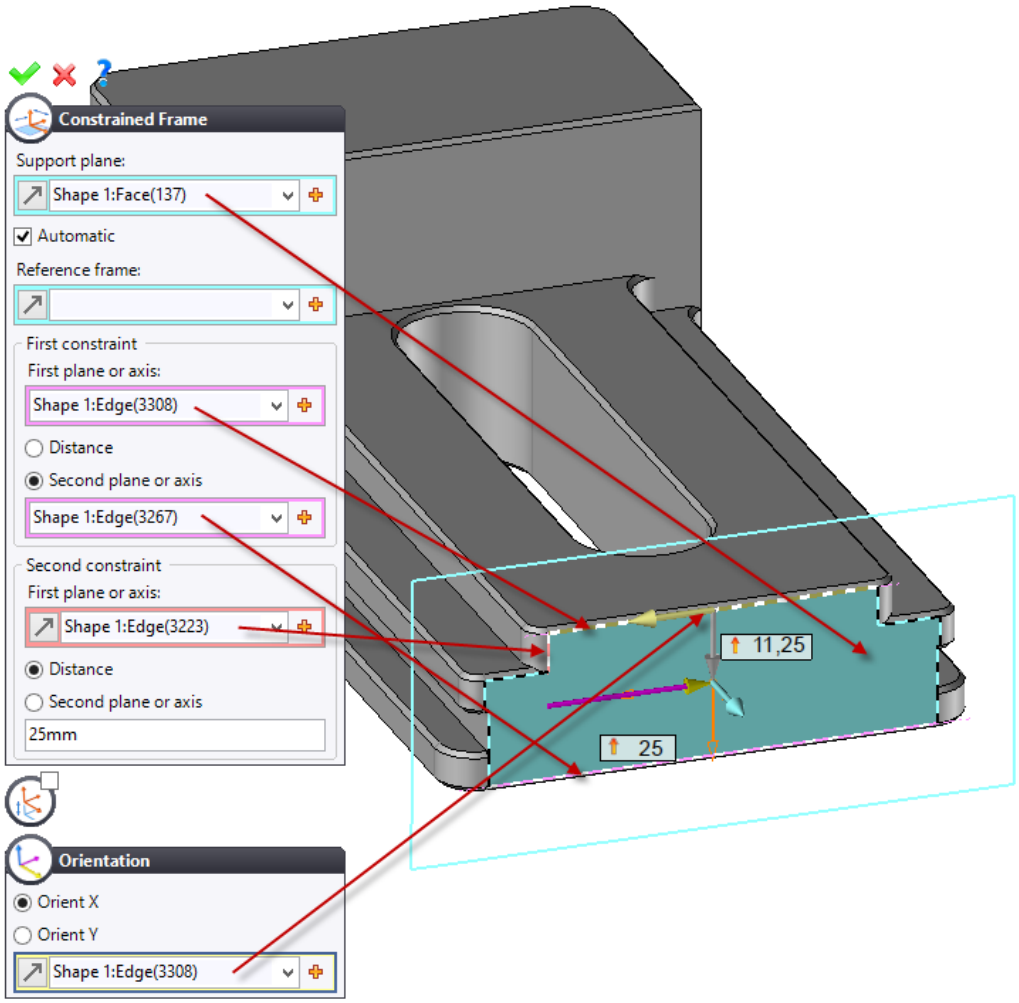


Notice that all you need to do is set the nominal diameter and **TopSolid** will select the correct tap for you.

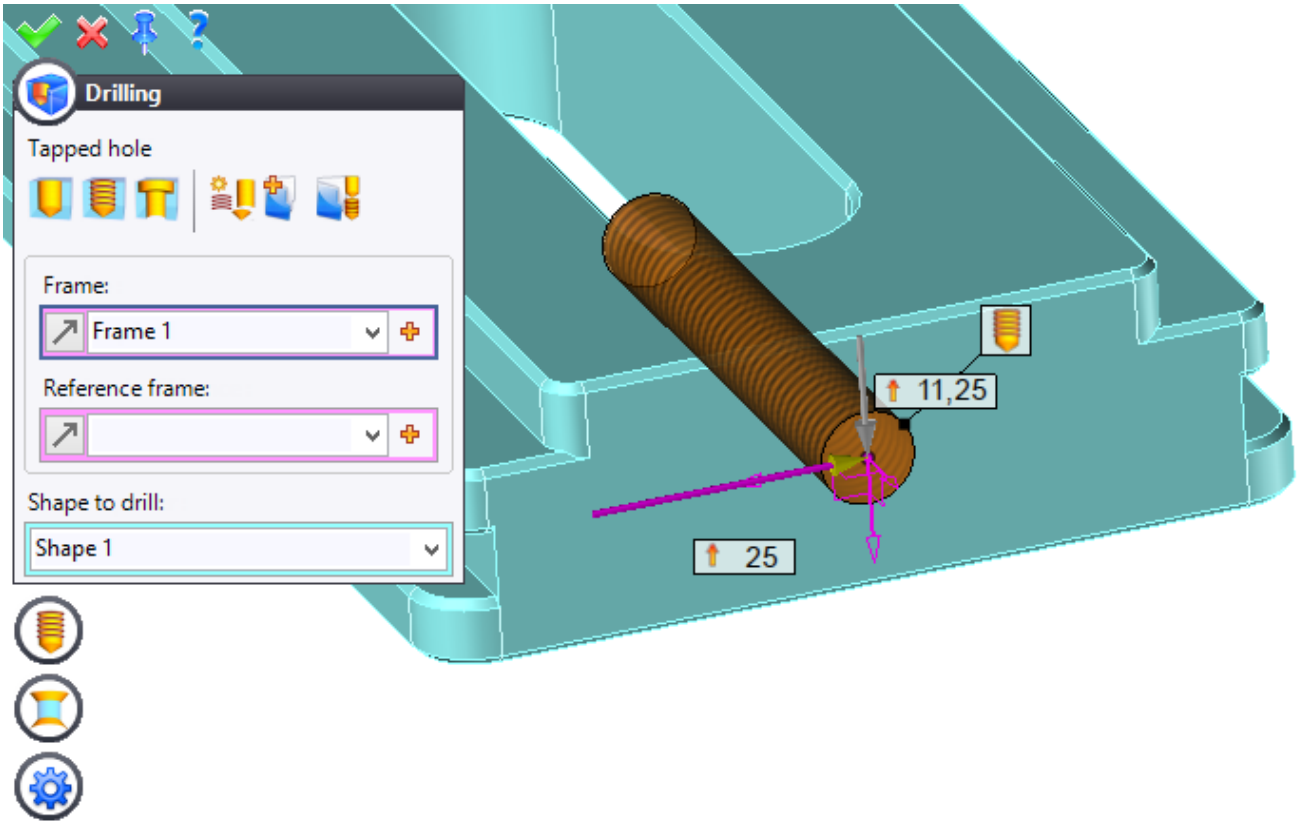
- Create a frame to position the drilling. To do this, click on the icon to the right of the **Frame** field and select **Constrained Frame**.



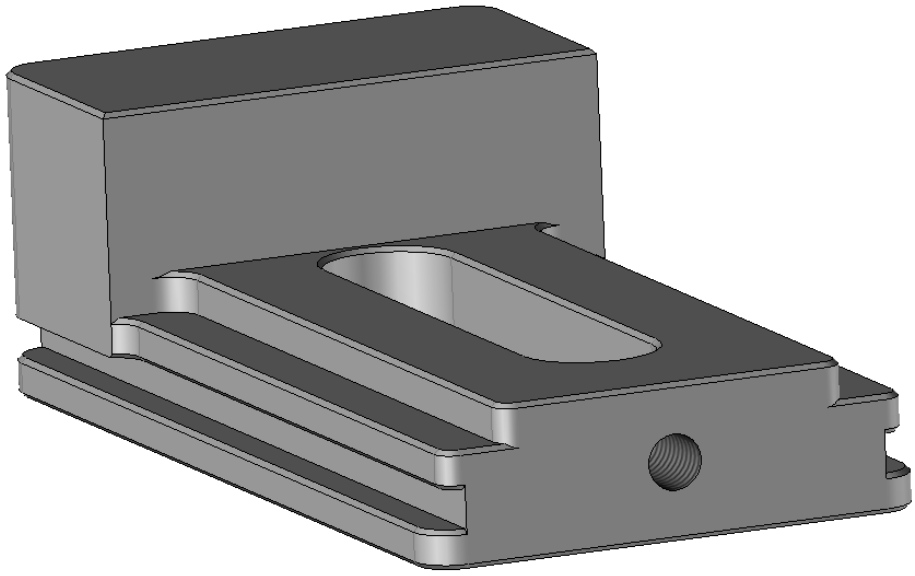
- Modify the dialog box as shown below.



- Click to **confirm** the frame.



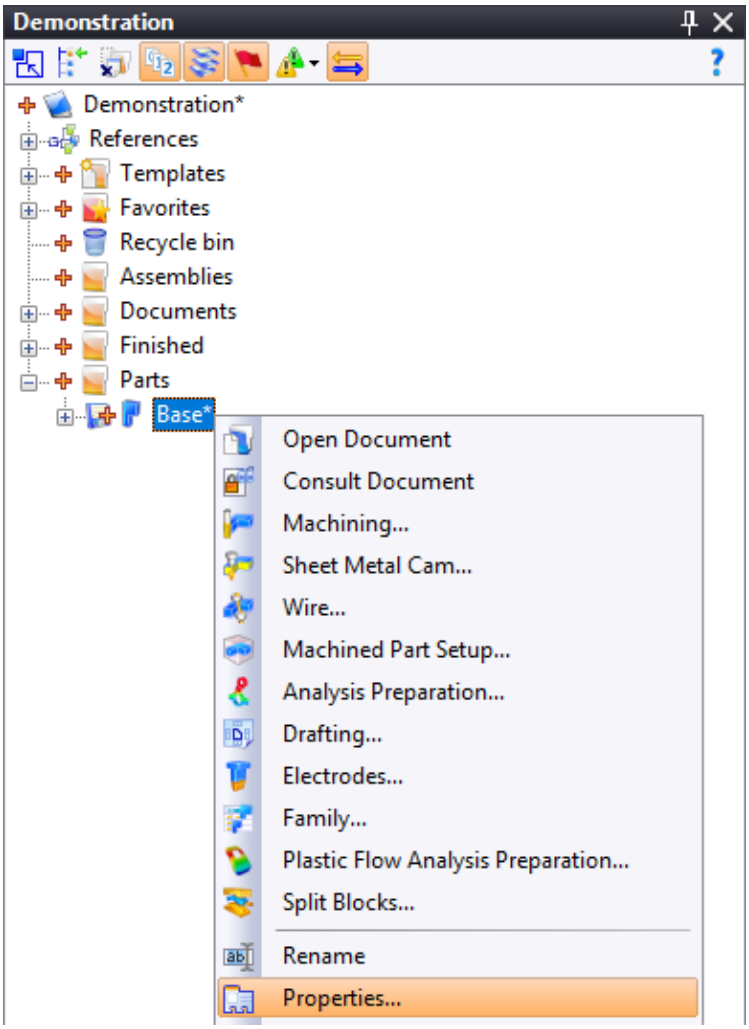
- Click to **confirm** the drilling and **close** the dialog box.



Defining the document properties

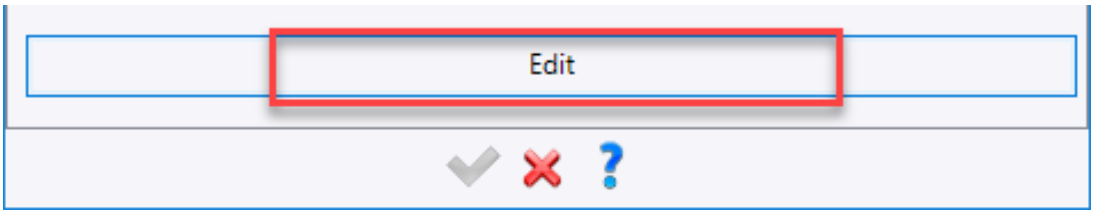
The last step is to define the basic properties for the part.

- In the Project tree, right-click on the *Base* part document.
- Select the **Properties** command.

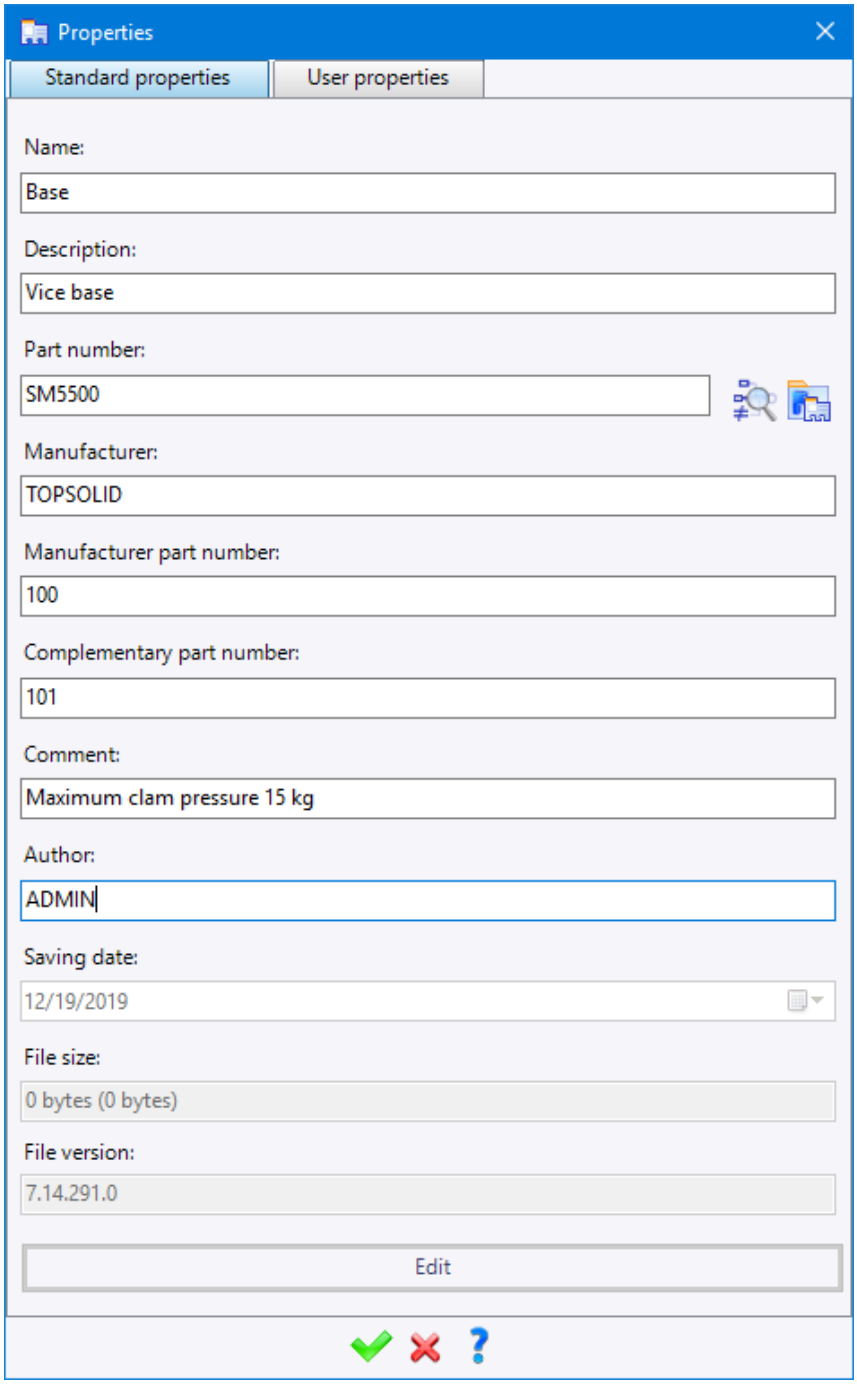


In this section, you are asked to provide certain information for documentation purposes. Note that this information can be filled in at any time. Later in this tutorial, you will be producing some drafting documents with a complete bill of materials. It is the BOM that will use this information.

- Click on the **Edit** button at the bottom of the dialog box to edit its content.

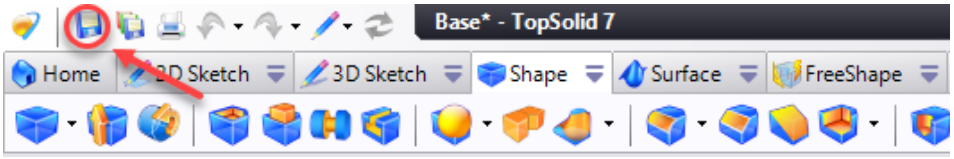


- Fill in the information as shown below or with your own information. When finished, **confirm** the operation.



Saving your work

You have now completed the design of the base for the vice part. Please be sure to save your work. You can save your work by clicking on the **Save Document** icon shown in the image below.

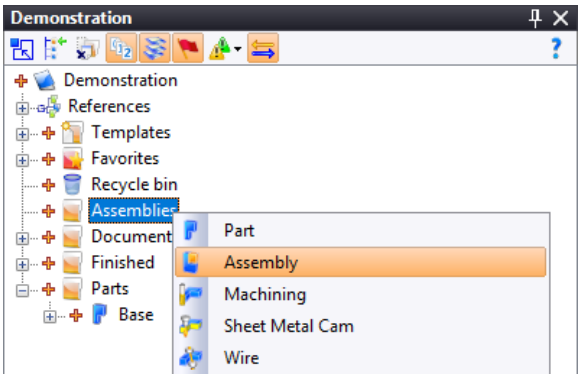


In the next section, you will learn how to create a bottom-up assembly in **TopSolid 7**.

Step 4: Creating an Assembly Document

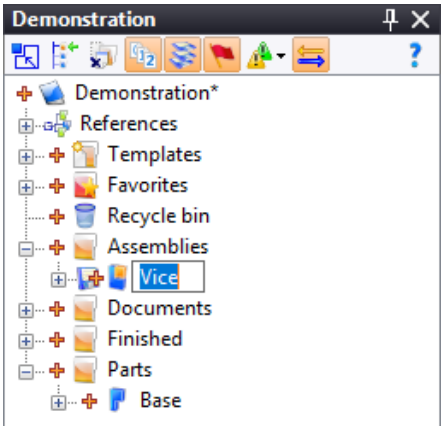
- Right-click on the *Assemblies* folder.
- Select the **Assembly** command.

Keeping everything organized is important in any company. Isn't it nice to have a software that does this automatically?



- Rename the assembly document *Vice*.

You can rename this assembly as often as you like without affecting any of the associativity. This is because the PDM keeps everything managed and up-to-date for you.

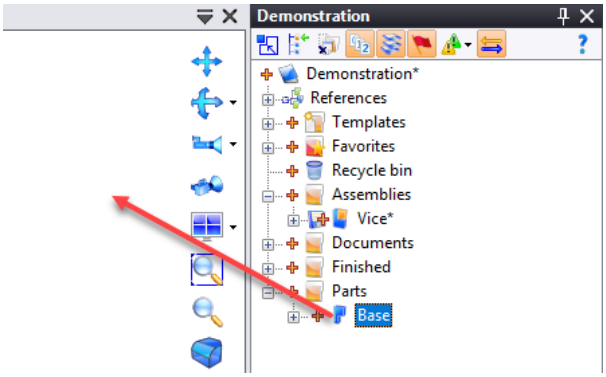


Inserting a part document using drag-and-drop

Once the new assembly document is created, all you need to do is drag the parts you want to use in this document.

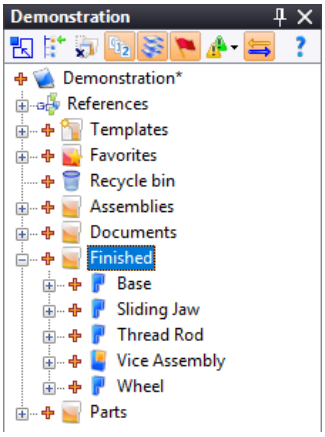
- From the Project tree, select the *Base* part document and hold down the left mouse button.
- Drag this document into the *Vice* assembly's graphics area.

Are you sure that you have selected the correct part? Remember that **TopSolid 7** shows a preview of the selected part at the bottom of the Project tree.



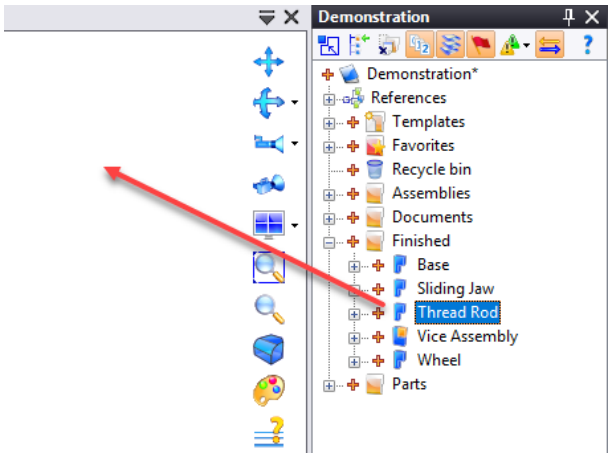
- From the Project tree, open the *Finished* folder to show the rest of the parts that will be used in this sample.

Note that there is another assembly document called *Vice Assembly* and another part document called *Base* in this folder. Both documents have been provided for reference purpose but they will not be used during this tutorial.



Applying assembly constraints

- Select the *Thread Rod* part document.
- Hold down the left mouse button and then drag this document into the assembly's graphics area.

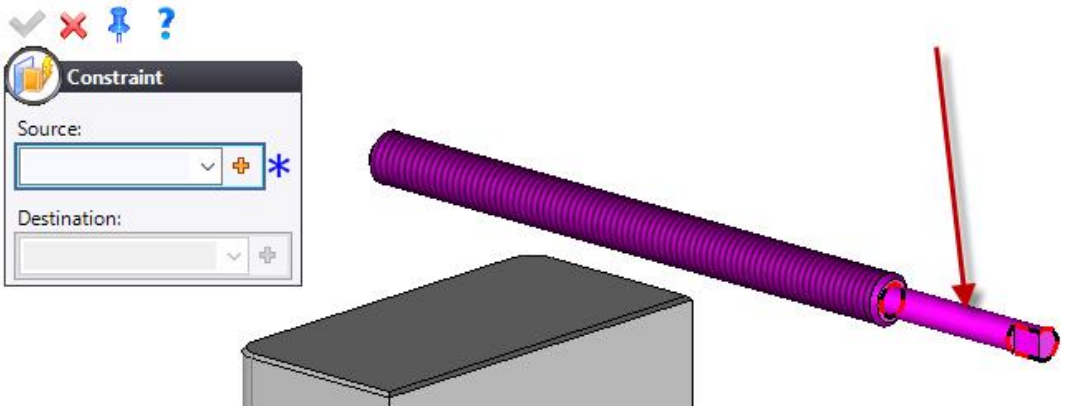


The *Thread Rod* appears purple in the graphics area because the part is not fully constrained. **TopSolid 7** is the only software to give users a visual feedback regarding constrained parts within assemblies.

Note: The **Constraint** command is automatically launched.

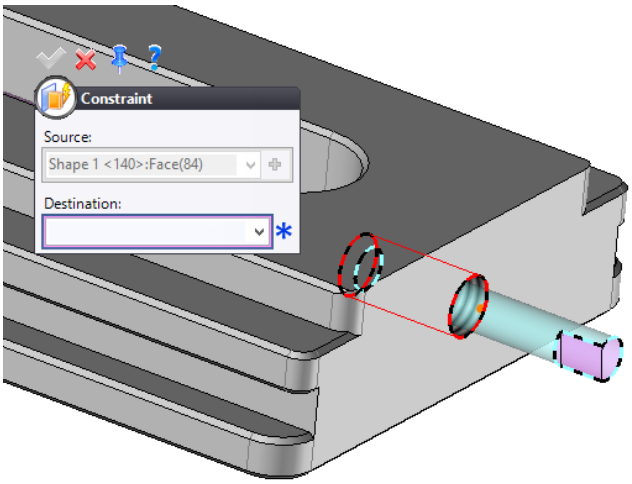
The **Constraint** positioning command is a dynamic command that automatically selects the correct assembly constraint for the user based on the source and destination geometries that have been selected.

- Click on the cylindrical face of the *Thread Rod* part as shown below.



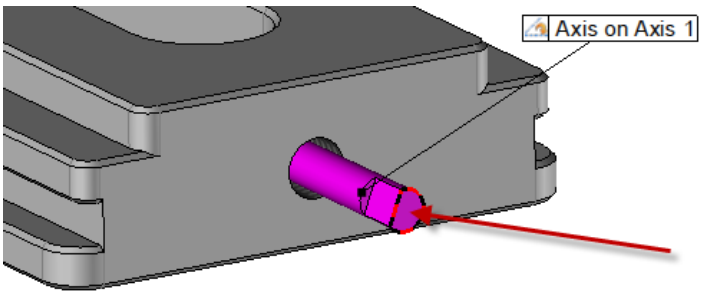
Once you have selected the face, the *Thread Rod* part follows your cursor around the screen.

- Click on the tapped hole on the *Base* part as shown below.

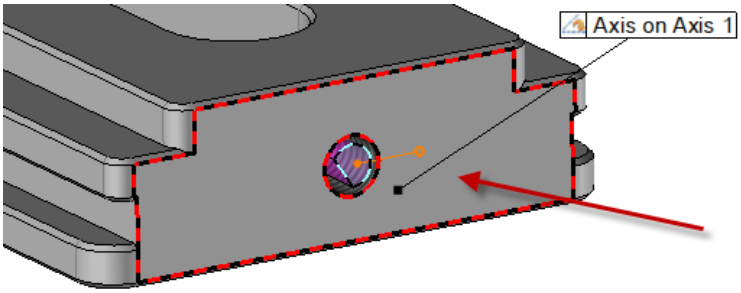


This creates an **axis-on-axis** constraint automatically.

- Select the front planar face of the *Thread Rod* part as shown here.



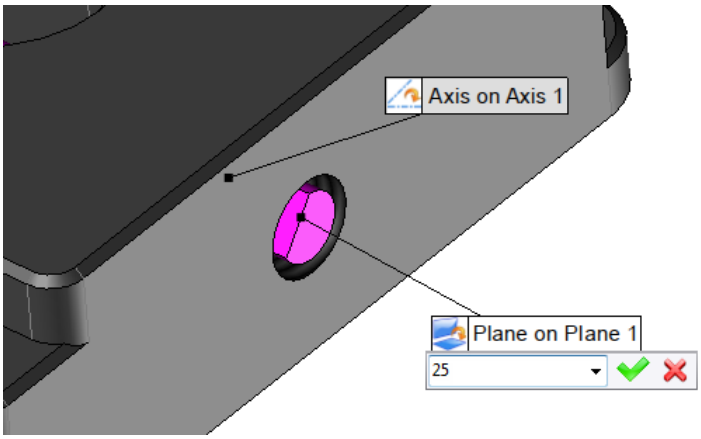
- Select the front planar face of the *Base* part as shown here.



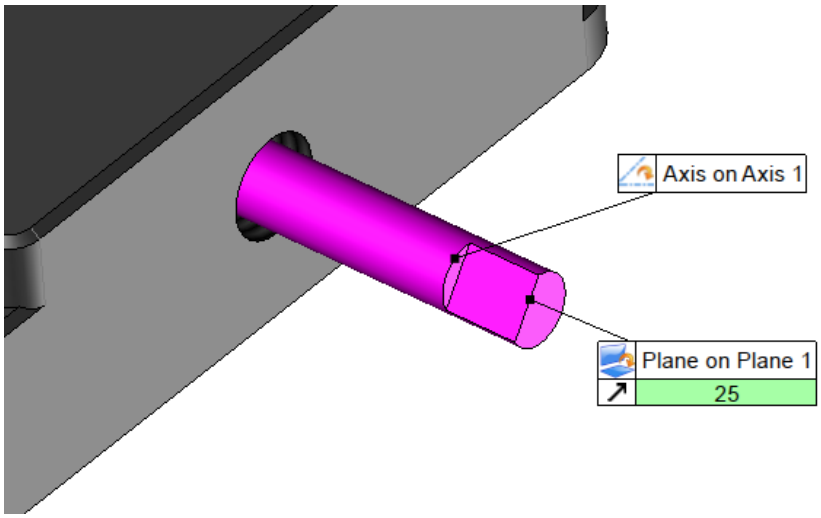
This creates a **plane-on-plane** constraint automatically.

You are now going to apply an offset to the plane-on-plane constraint.

- Move the cursor to the bottom of the **Plane on Plane 1** constraint label where you see the 0 field, then double-click on it.
- Adjust the value to *25* and click to **confirm** the operation.
- **Close** the **Constraint** dialog box.



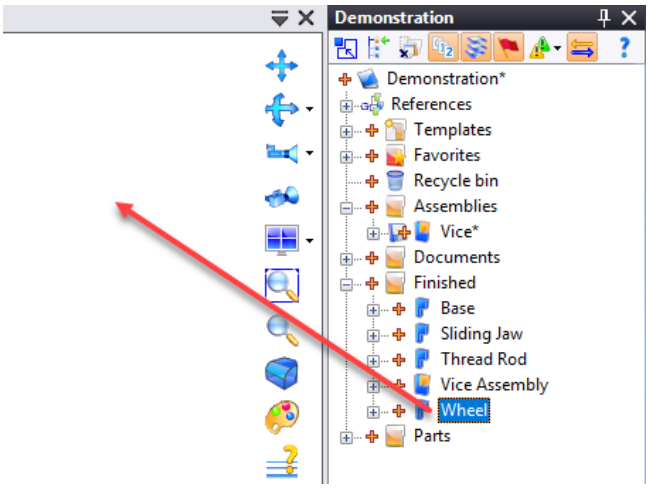
Note that this procedure has allowed you to quickly and easily offset the flat front face of the *Thread rod* part away from the flat front face of the *Base* part. You did not need to find an additional "special" command to allow an offset.



Including and constraining the wheel

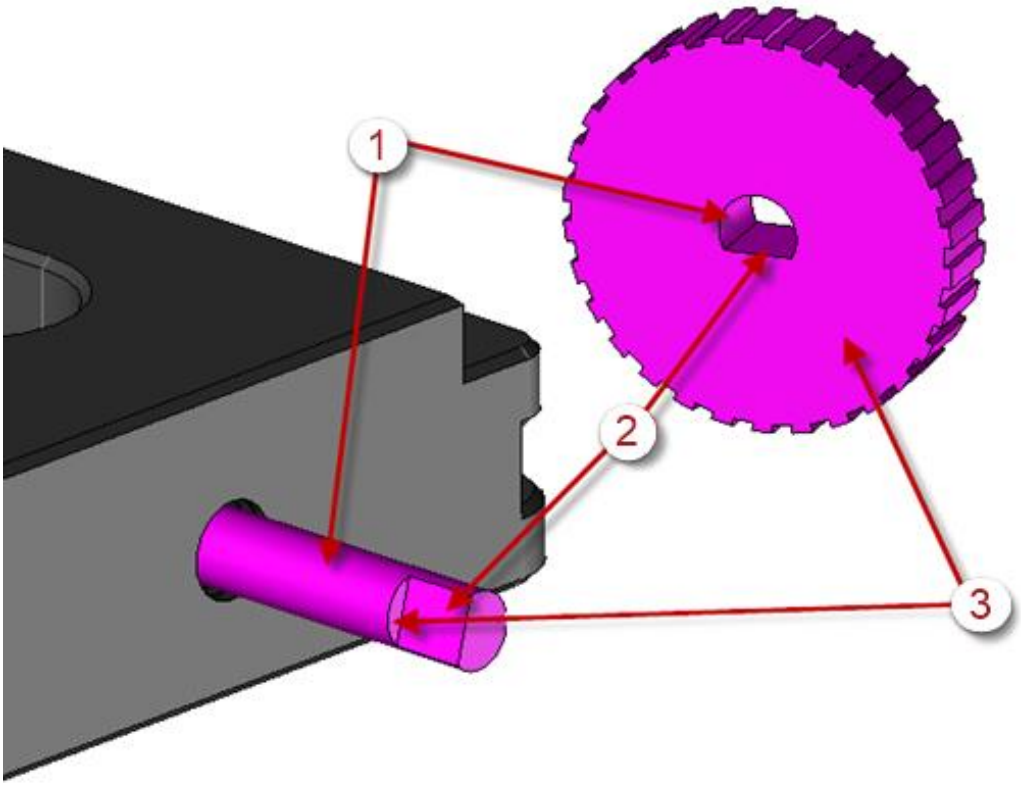
- From the Project tree, select the *Wheel* part document.
- Drag and drop it into the assembly's graphical area.

Note that you do not need to close the constraint positioning dialog box to add an additional component. **TopSolid** will do this automatically for you.

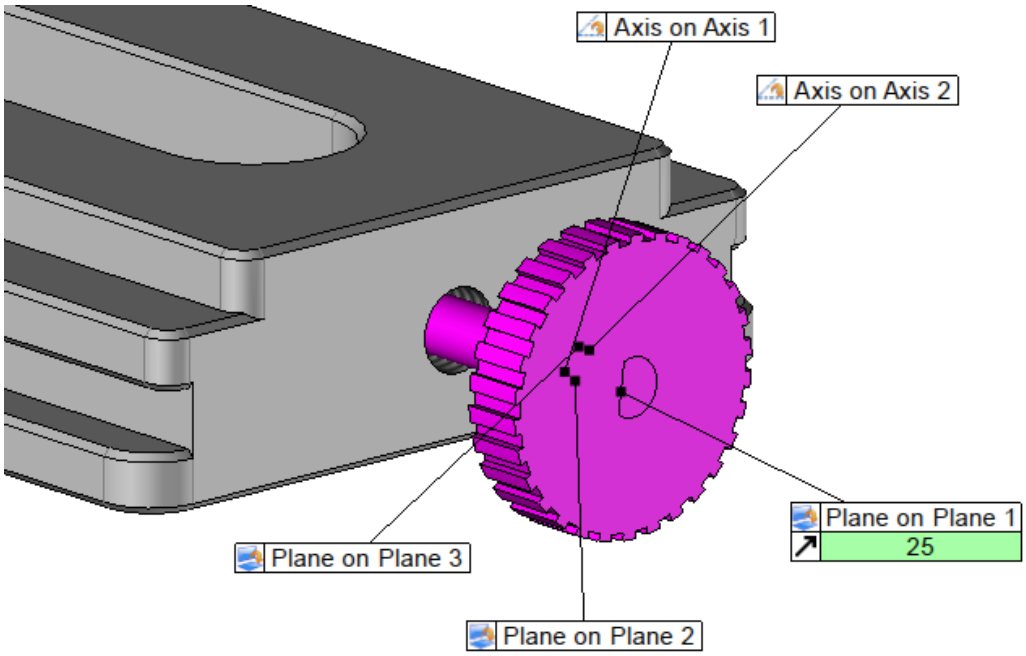


In this next example, you will be positioning the *Wheel* onto the *Thread Rod*, and then applying the following constraints. You can use the **Constraint** command like before, or you can use each specific type of constraint.

1. **Axis on axis**
2. **Plane on plane**
3. **Plane on plane**

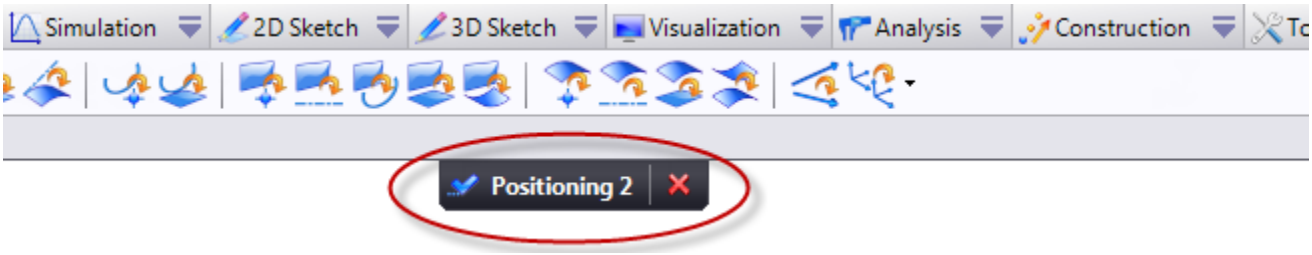


When finished, your assembly should look like the image below.



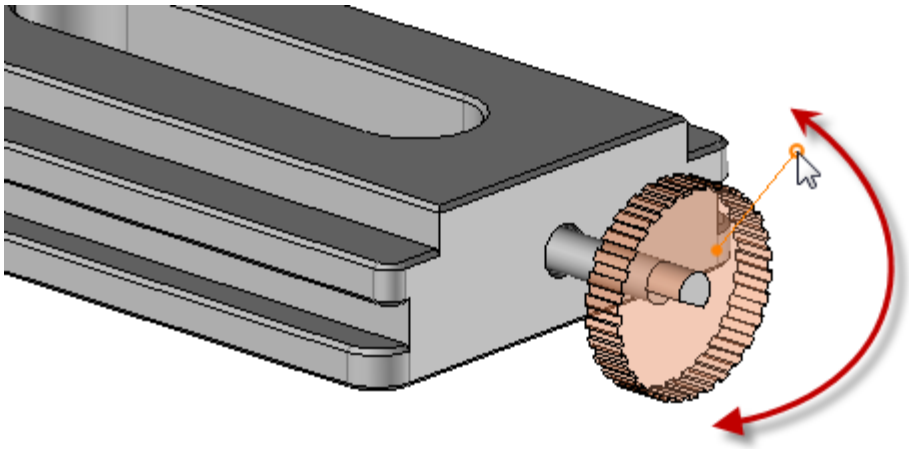
Using dynamic assembly motion

- **Confirm** the positioning by clicking the **Positioning** button at the top center of the screen.



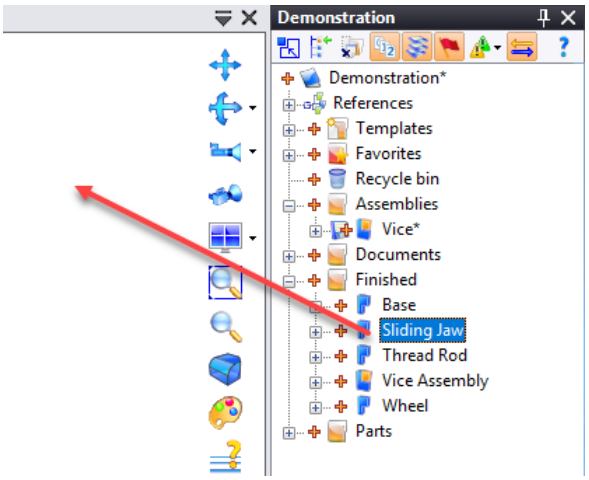
At this point, you can check the degree of freedom that the wheel has left.

- Using the left mouse button, click and hold down the wheel and move your mouse. You should notice that the assembly is dynamically updating so that you can test it.



Including and constraining the sliding jaw

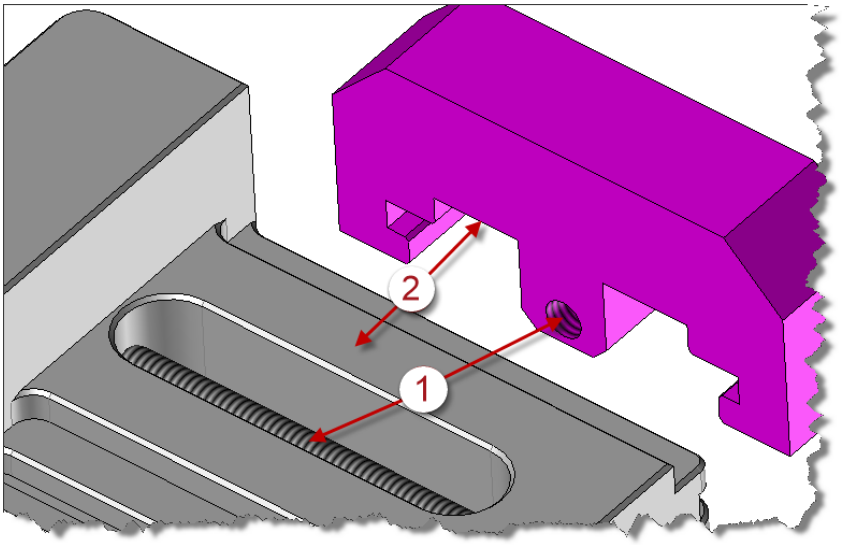
You will now position the *Sliding Jaw* part into the *Vice* assembly. You can use the **Constraint** command like before or you can use the constraint specific commands.



- Select the part document *Sliding Jaw*.
- Drag and drop it into the assembly's graphical area.

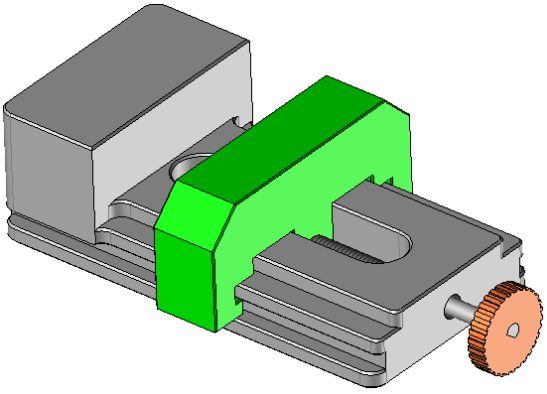
- Apply the following constraints.
  1. **Axis on axis**
  2. **Orientation**

Note: For the orientation constraint, select the underside of the sliding jaw near the edge as shown below. Make sure that the arrows are pointing in the correct directions.



- Once the constraints are applied, **confirm** the positioning by clicking on the **Positioning** button.

When finished, your vice should now look like the image shown here. Note that you can move the *Sliding Jaw* part by selecting it with your mouse.



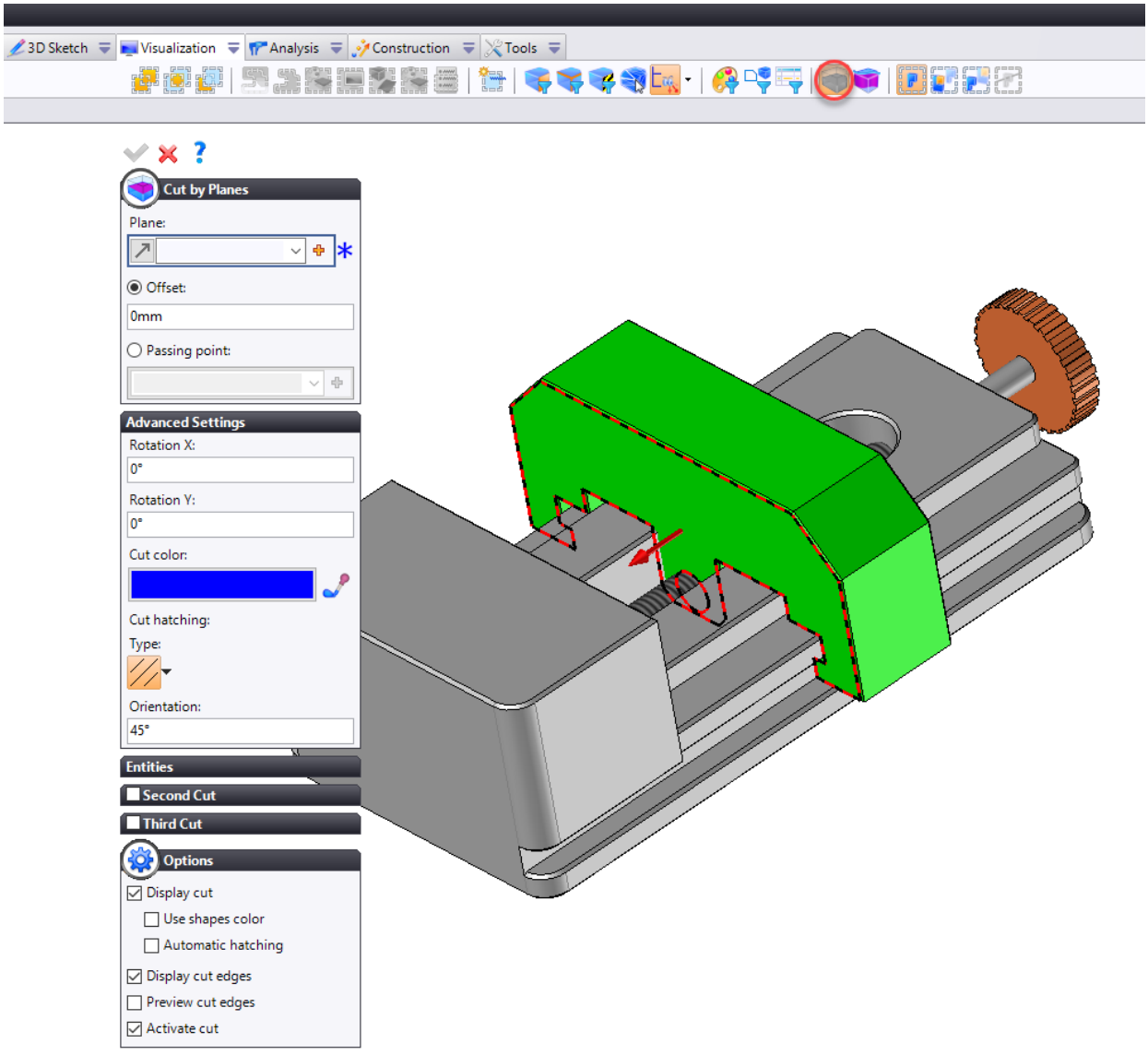
Step 5: Editing a Part Within the Assembly

Now that the assembly is complete, it is time to learn how to modify one of the parts within the assembly. Firstly, you need to identify if there are any problems.

Dynamic 3D section

Sometimes it is difficult to see certain details when examining an assembly. Because of this, **TopSolid 7** gives you the ability to create dynamic 3D sections to help you visualize your assembly.

- Go to the **Visualization** tab and select the **Cut by Planes** command.
- In the **Plane** field, select the face of the sliding jaw in the graphics area as shown below.
- To customize the cut, uncheck the **Use shapes color** and **Automatic hatching** boxes in the **Options** dialog box.
- Open the **Advances Settings** dialog box, select a color for the cut and then select the **Solid** hatching type.



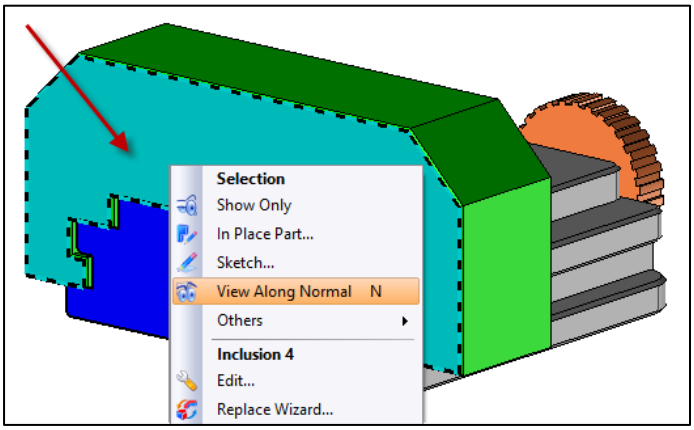
- Click to **confirm** the cut.

This will create a dynamic section of the entire assembly. It will make it significantly easier to see some of the inner details of the design.

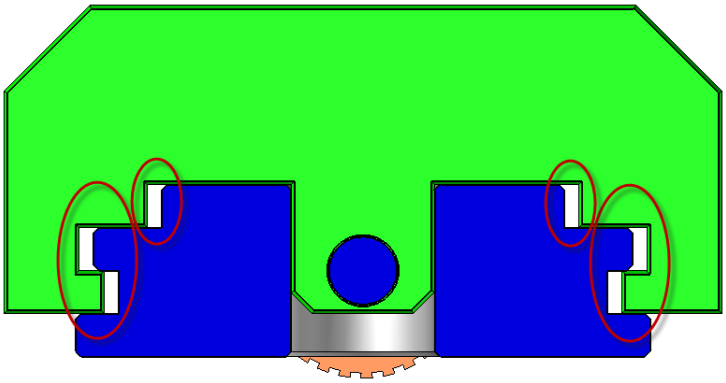
View along normal

- Right-click on the face of the sliding jaw as shown here.
- Select the **View Along Normal** command.

This is a handy trick to switch to a plan view that is normal to the planar face that you select.



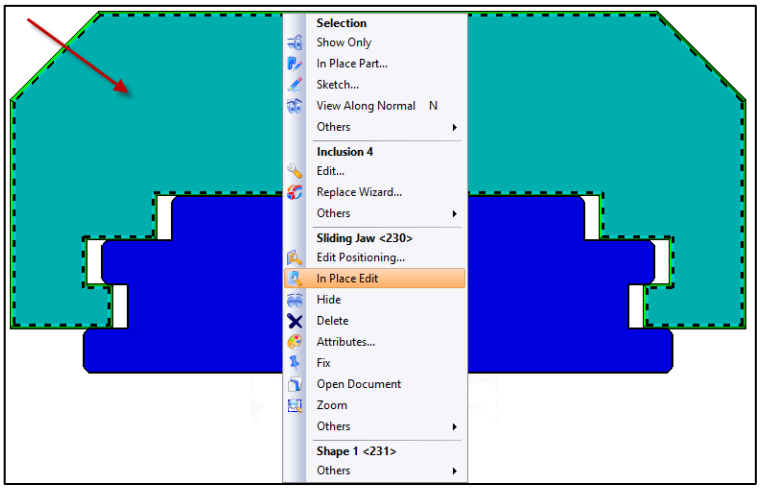
Now that you are looking directly at the *Sliding Jaw* part, you should easily see where the values are not correct on the design. Refer to the picture below.



In-place editing of the sliding jaw

From here, you will do an in-place edit of the *Sliding Jaw* part. This way, you will be able to instantly see the results of the change.

- Right-click on the face of the sliding jaw shown below and select the **In Place Edit** command.



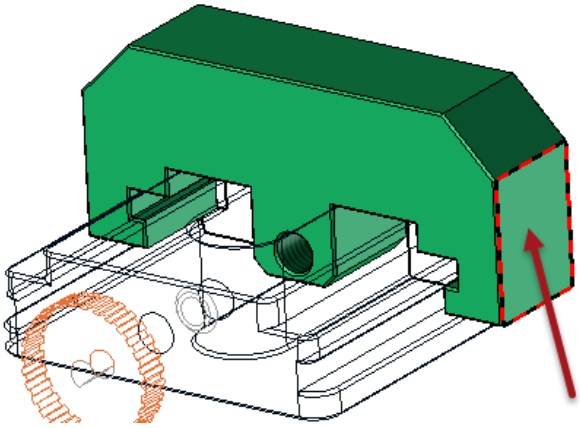
Note: Another way to edit a part within the assembly is to double-click on the part that you want to edit.

Now that you are in an in-place editing mode, you will notice that the part that you are editing is shown as a shaded solid model, while all of the other parts in the assembly are shown in wireframe.

Displaying and editing the driving sketch

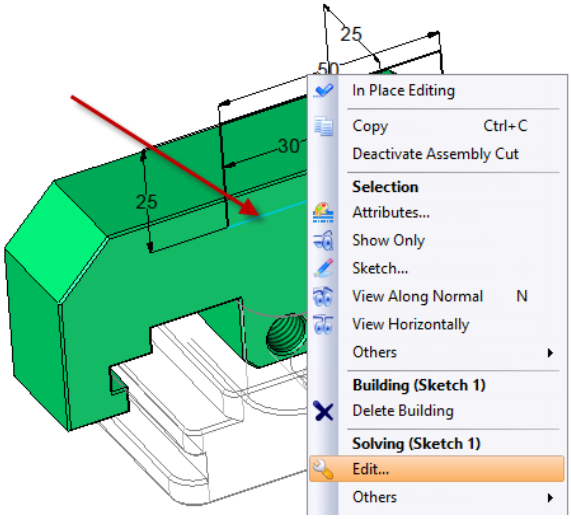
- Double-click on the face shown below to display the driving sketch for it.

Double-clicking on a face of a feature will either show or hide the driving elements of that feature.



- Right-click on the sketch as shown below and select the **Edit** command.

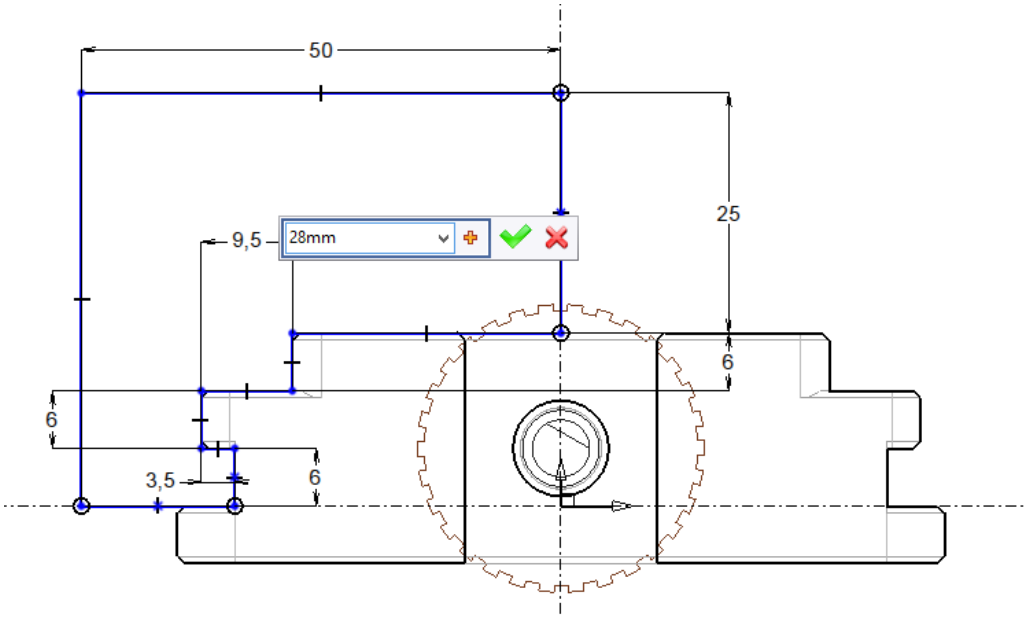
Note that you are now editing the sketch that was used to create the main extrusion of the sliding jaw. And all from within the assembly document without needing to return to the part document. The changes that you make here will automatically update the part document thanks to the PDM.



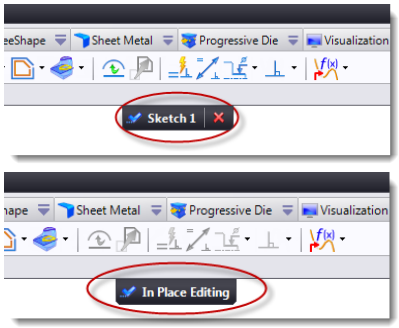
Editing a driving dimension

You will now modify one of the dimensions in the sketch.

- Double-click on the 30mm dimension, change it to *28mm*, and then click to **confirm** the operation.

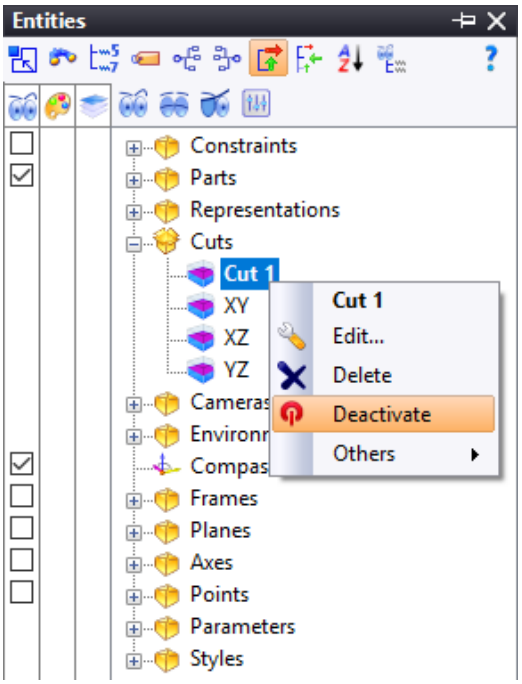


Finish editing and return to the assembly



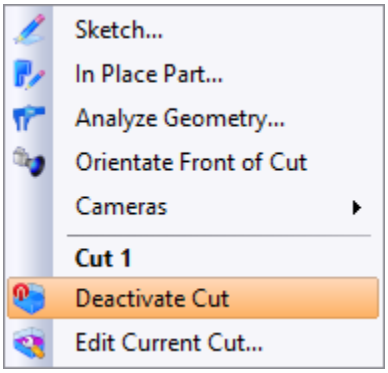
- **Confirm** the sketch by clicking the **Sketch** button at the top center of the screen.
- Double-click on the face of the part (same one as before) to hide the sketch.
- **Confirm** the in-place editing by clicking on the **In Place Editing** button at the top center of the screen.

Deactivating the 3D cut



- From the Entities tree, open the **Cuts** folder.
- Right-click on **Cut 1** and select the **Deactivate** command.

The 3D cut can also be disabled directly within the graphics area by right-clicking in the graphics area (not an object) and selecting the **Deactivate Cut** command.



You can always come back to the Entities tree and reactivate this cut at any time.

- **Save** the document.

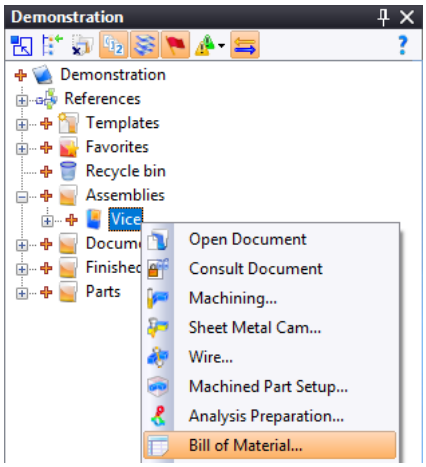
You have now completed the **Assembly** section of this tutorial. Therefore, in the next section, you will learn how to create a bill of materials and all of the related drafting documents.

Step 6: Creating a Bill of Materials

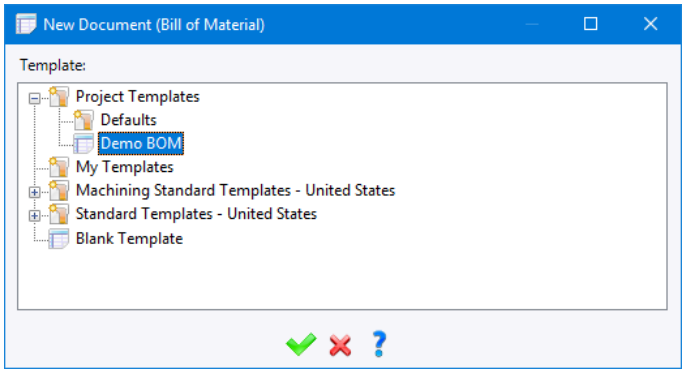
In out-of-date CAD/CAM systems, you typically create the drawing first, and then create your bill of materials. That is ok, but does it really make sense? Nope, we did not think so either. A BOM is used to manage an entire project, so why not manage a complete set of drawings based on one as well? This is all possible thanks to **TopSolid 7**. Here is how.

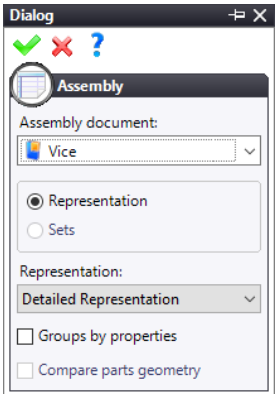
- Right-click on the *Vice* assembly document from the Project tree.
- Select the **Bill of Material** command.

It seems that **TopSolid 7** always knows what you are looking for with its "intuitive" menu technology.



- Select **Demo BOM** from the project templates and then click to **confirm** the operation.





- Click to **confirm** the assembly document to reference.

Start Page | Vice | Vice*

Index	Quantity	Name	Description	Material Name	Mass	Manufacturer	Manufacturer Part Number	Comment	Major Revision
	1	Vice			1.06kg				A
1	1	Base	Vice base	Aluminium AB-21000 EN 1780	0.81kg	TOPSOLID	100	Maximum clam pressure 15 kg	A
2	1	Sliding Jaw	Small soft Jaw for small vice	Aluminium AB-21000 EN 1780	0.22kg		200	All edges require .75mm min edge break	A
3	1	Thread Rod	Small Vice thread rod	Aluminium AB-21000 EN 1780	0.03kg		300	Break all edges	A
4	1	Wheel	Clamp wheel for small vice	Aluminium AB-21000 EN 1780	0.01kg		400		A

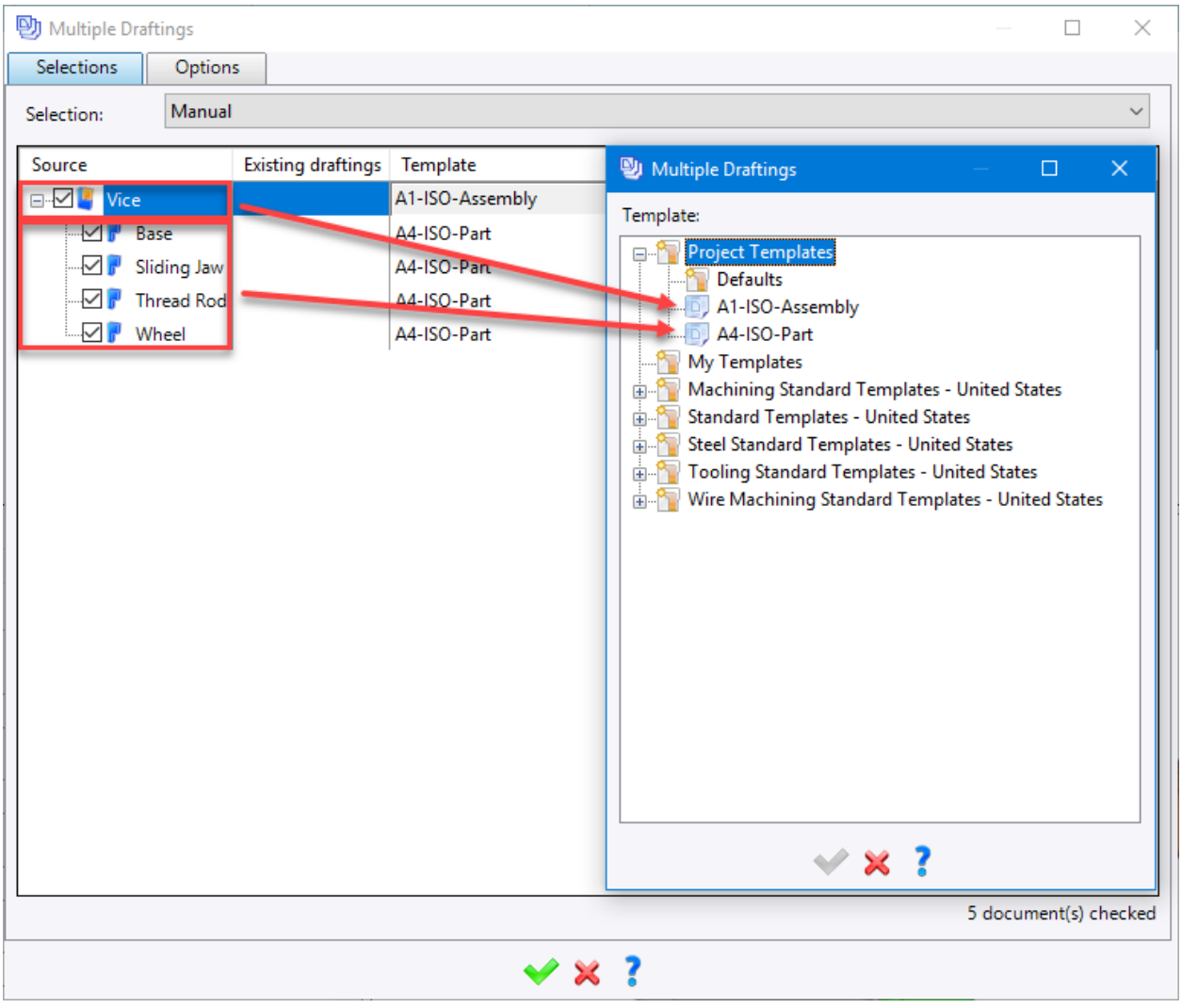
Take a minute to look at the information automatically completed in the sample BOM above. You will notice that the BOM even includes the mass of each of the parts. What is impressive is that **TopSolid** automatically calculated the mass for you and will make updates based on your changes. To open a document from the BOM line, right-click on this document in the Project tree and select the **Open Document** command.

- **Save** the BOM document by clicking on the icon.

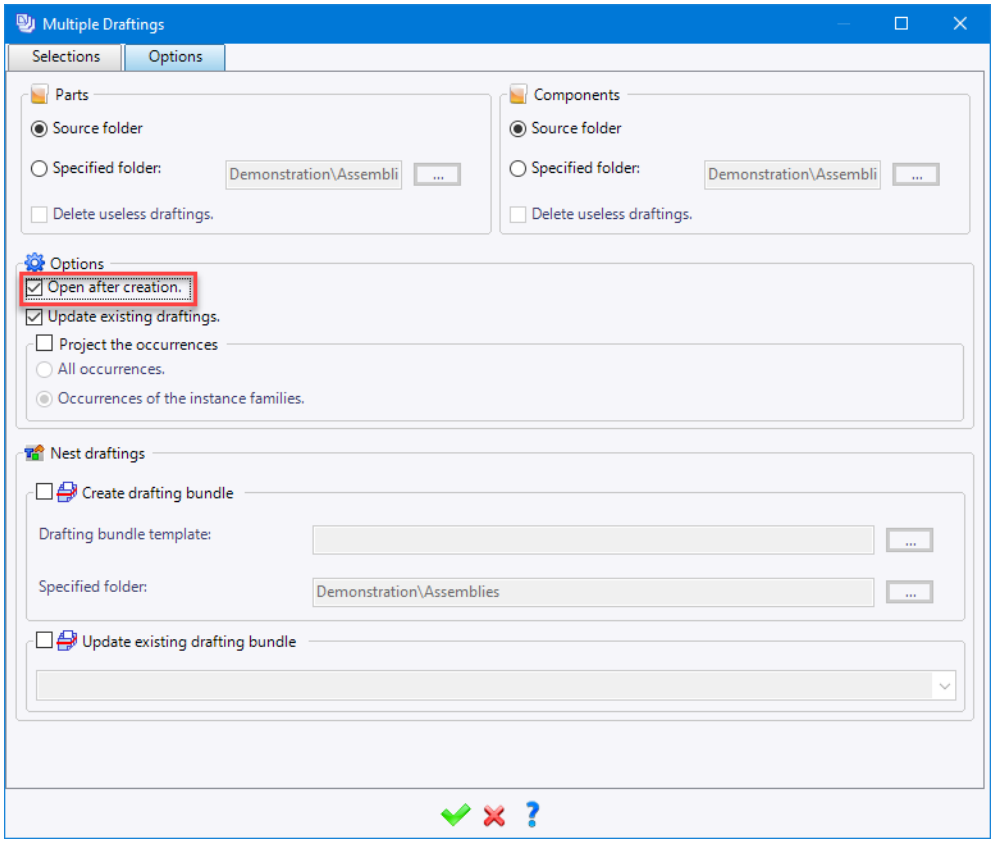
Step 7: Creating the Draftings from the BOM

You will now create all of the draftings needed for the *Vice* assembly. The greatest part is that TopSolid creates all of the necessary documents automatically.

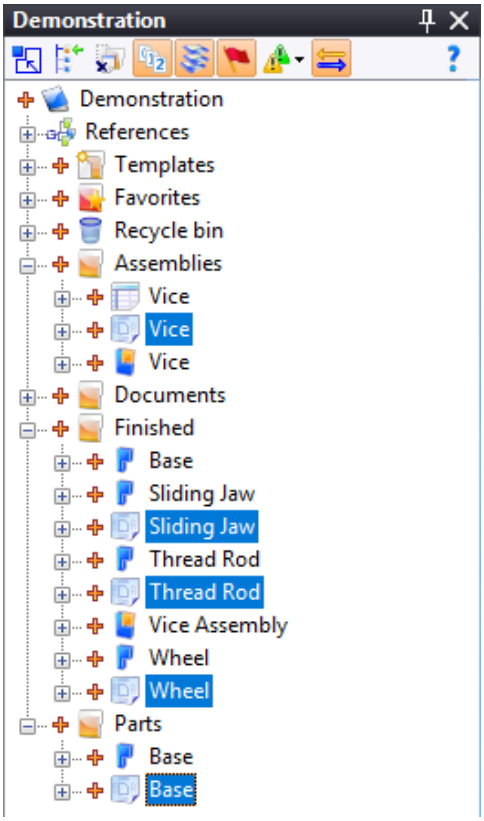
- Go to the **Bill of Material** tab and select the **Multiple Draftings** command.
- From the **Selections** tab, double-click on the text next to the assembly document in the **Template** column to define a drafting template.
- In the **Project Templates** folder, select the **A1-ISO-Assembly** drafting template for the assembly and then click to confirm the operation.
- Double-click on the text next to a part document in the **Template** column.
- In the **Project Templates** folder, select the **A4-ISO-Part** drafting template for the part.
- Right-click on the name of the part template (**A4-ISO-Part**) in the **Template** column and select **Apply the template to this type**. The **A4-ISO-Part** drafting template is then applied to all parts.



- Open the dialog box's **Options** tab, make sure that the **Open after creation** option is checked and then click to **confirm** the operation.



Once confirmed, **TopSolid 7** creates all of the draftings. Moreover, if you look in the Project tree, you will see that all of the new drafting documents are located directly under each of the parts or assemblies that they were built from.



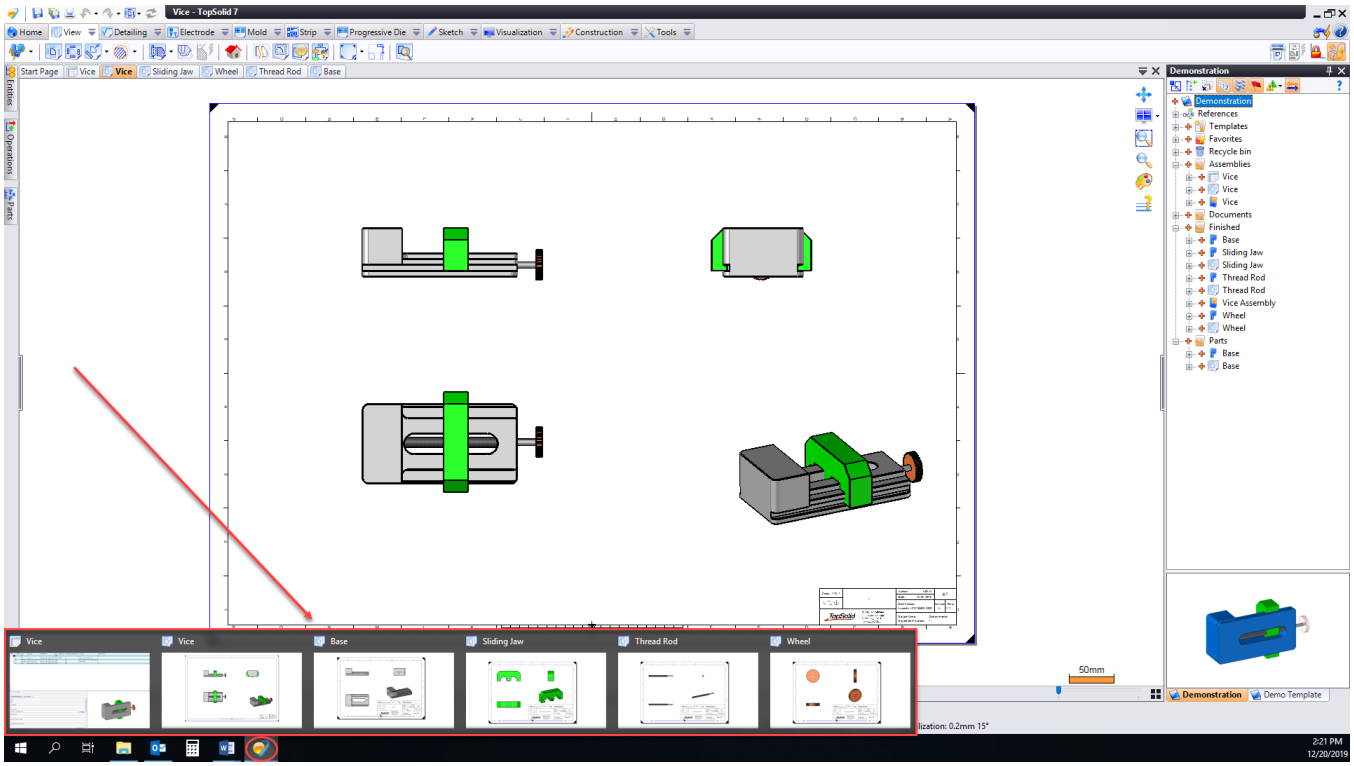
Using Windows 8 or 10 to switch between documents

Now that all the draftings have been created and are opened, we will switch to the assembly drafting by using the Windows 8 or Windows 10 Aero interface.

- Hover over the **TopSolid 7** taskbar icon.

When you do this, all open **TopSolid 7** documents will appear in preview mode.

- Drag your mouse over each of the documents to show how **TopSolid** is integrated to Windows 8 or 10.
- Select the assembly drafting as shown below.

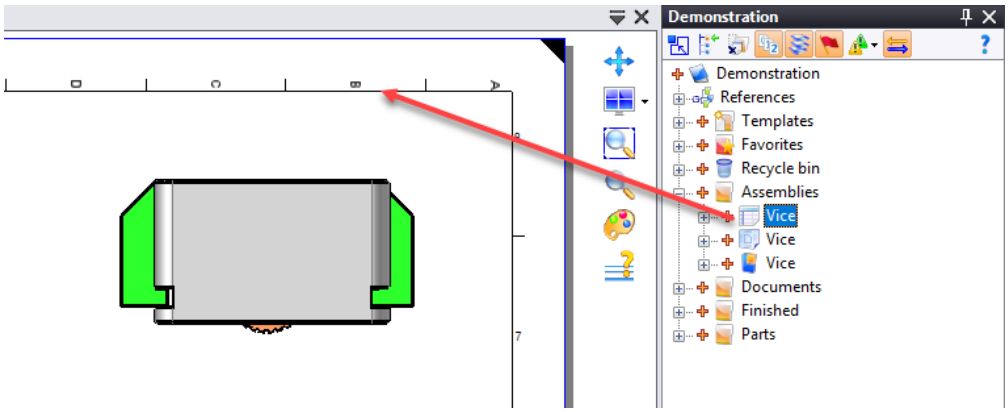


If you do not have Windows 8 or Windows 10, you can switch between the documents using the tabs located across the top of the drawing area.

Including the BOM into the drafting using drag-and-drop

You are now going to add the BOM that has already been created to the assembly drafting by dragging and dropping it from the Project tree.

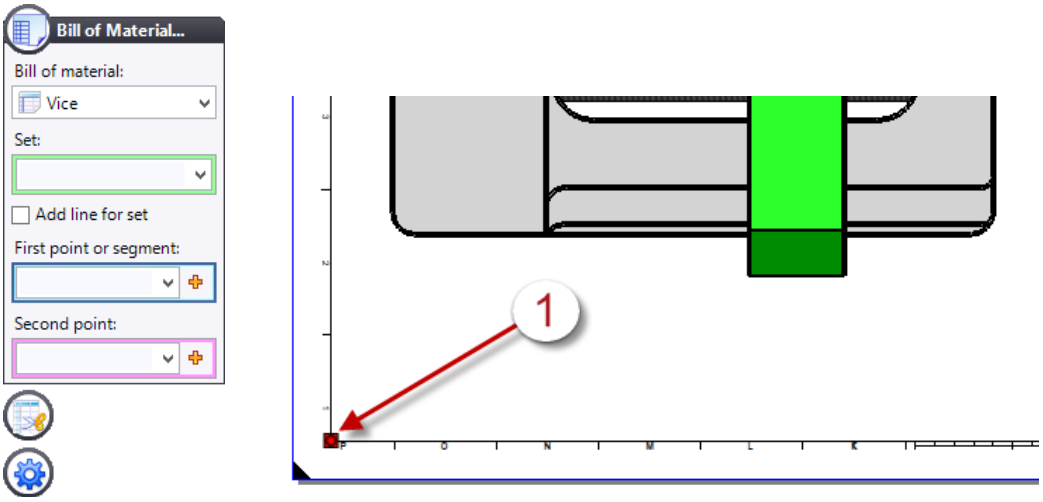
- Select the *Vice* BOM from within the *Assemblies* folder of the Project tree.
- Drag-and-drop it onto the screen of the assembly drafting.



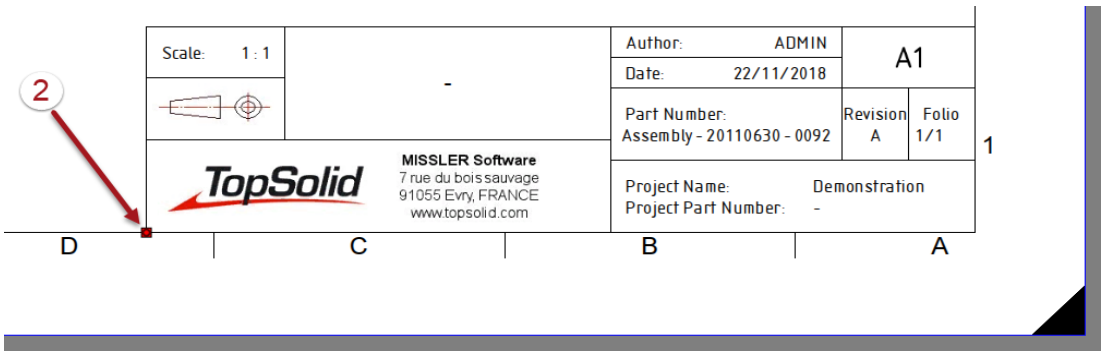
Take a look at the various documents in the Project tree. You should notice that every document created and managed by **TopSolid 7**'s PDM has a special icon that represents it. This makes it easy to visually know what is happening in the Project tree.

Positioning the BOM

- Click the bottom left point of the border of the drawing as shown in the image below.



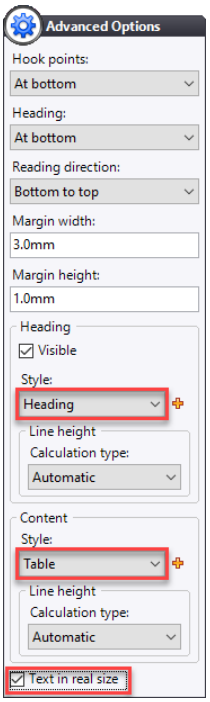
- Click on the bottom left point of the title block as shown in the image below.

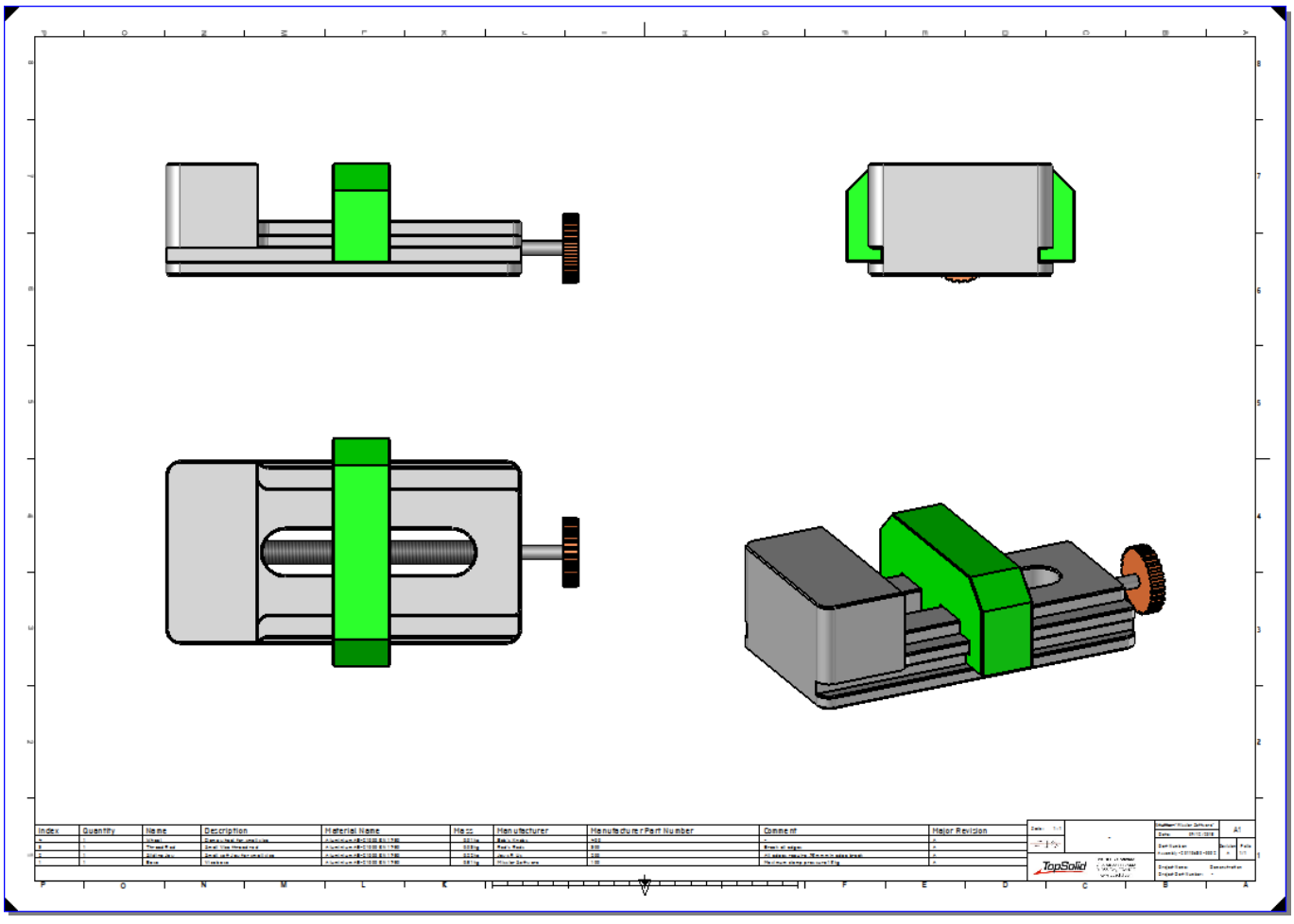


Sometimes the formatting of a table can be a bit tricky, especially when you change paper size, or the size of the table itself. Thankfully, **TopSolid 7** has solved that as well: you now have user definable styles.

Choosing and applying styles to the BOM

- To modify the text styles used in this table, open the **Advanced Options** dialog in the **Bill of Material** dialog box.
- Change the **heading style** to **Heading**.
- Change the **content style** to **Table**.
- Check the **Text in real size** box.
- Click to **confirm** the operation.

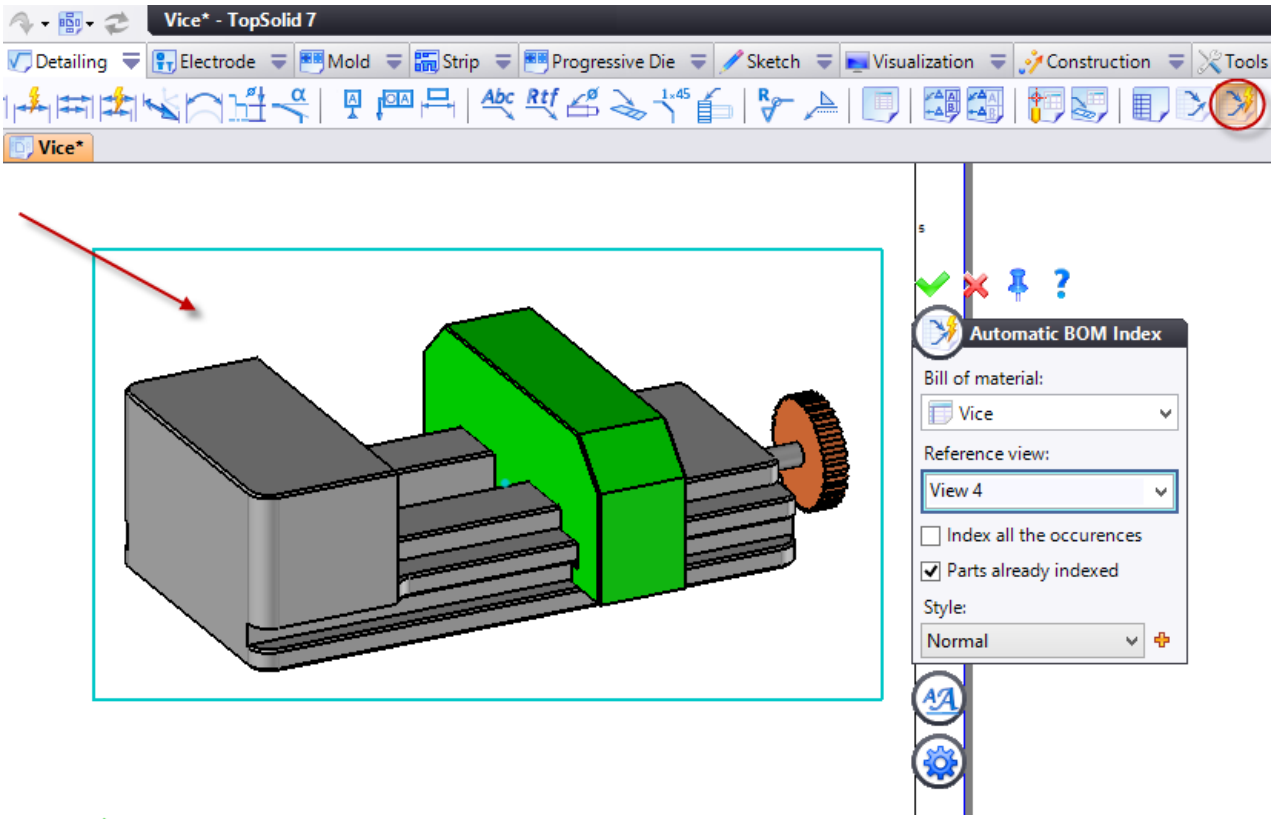




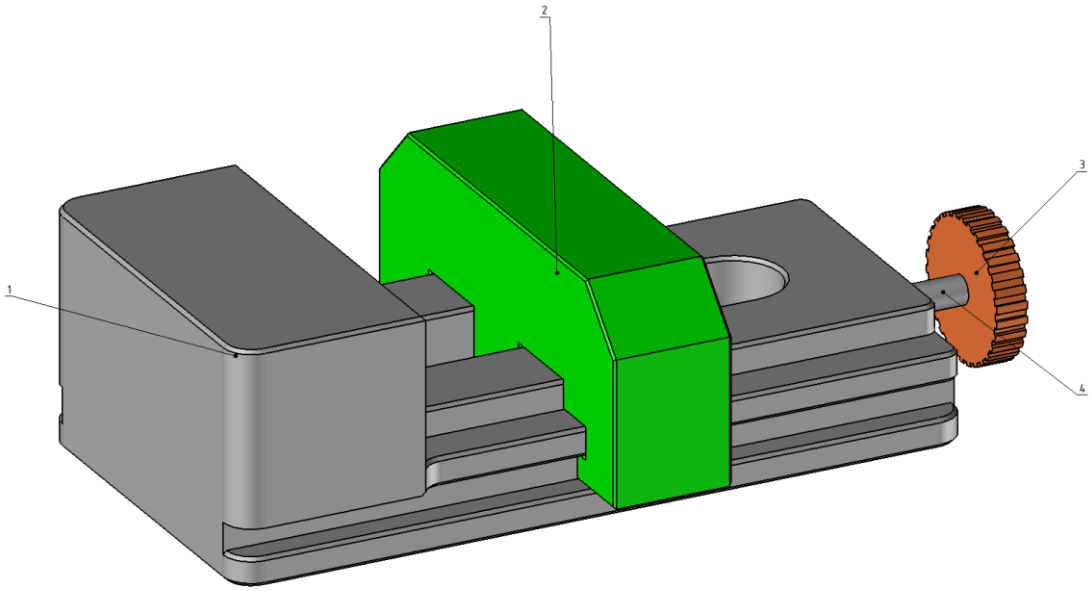
Adding automatic BOM indexes

You will now add BOM indexes to the isometric view using an automatic function.

- Go to the **Detailing** tab and select the **Automatic BOM Index** command.
- Select the isometric view of the *Vice* assembly.



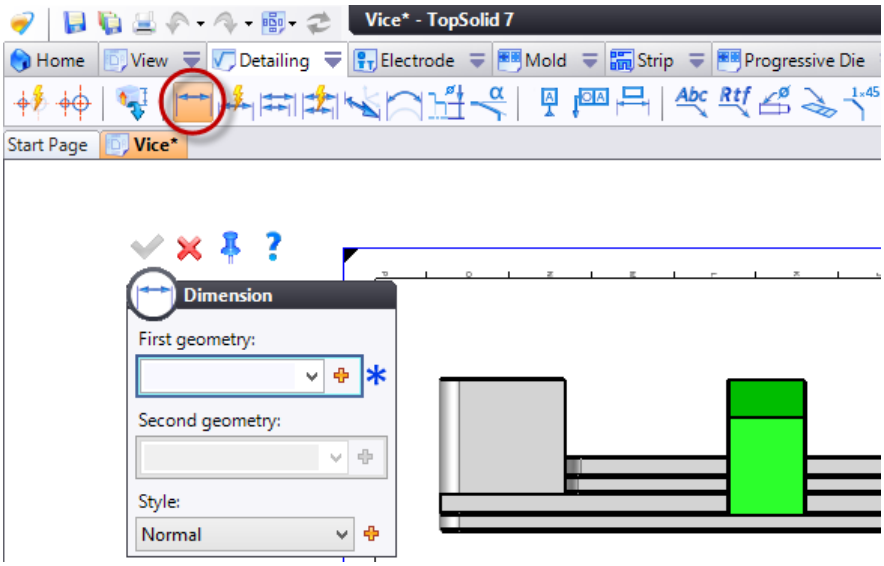
- Click to **confirm** the operation.



Step 8: Adding Dimensions to the Drafting

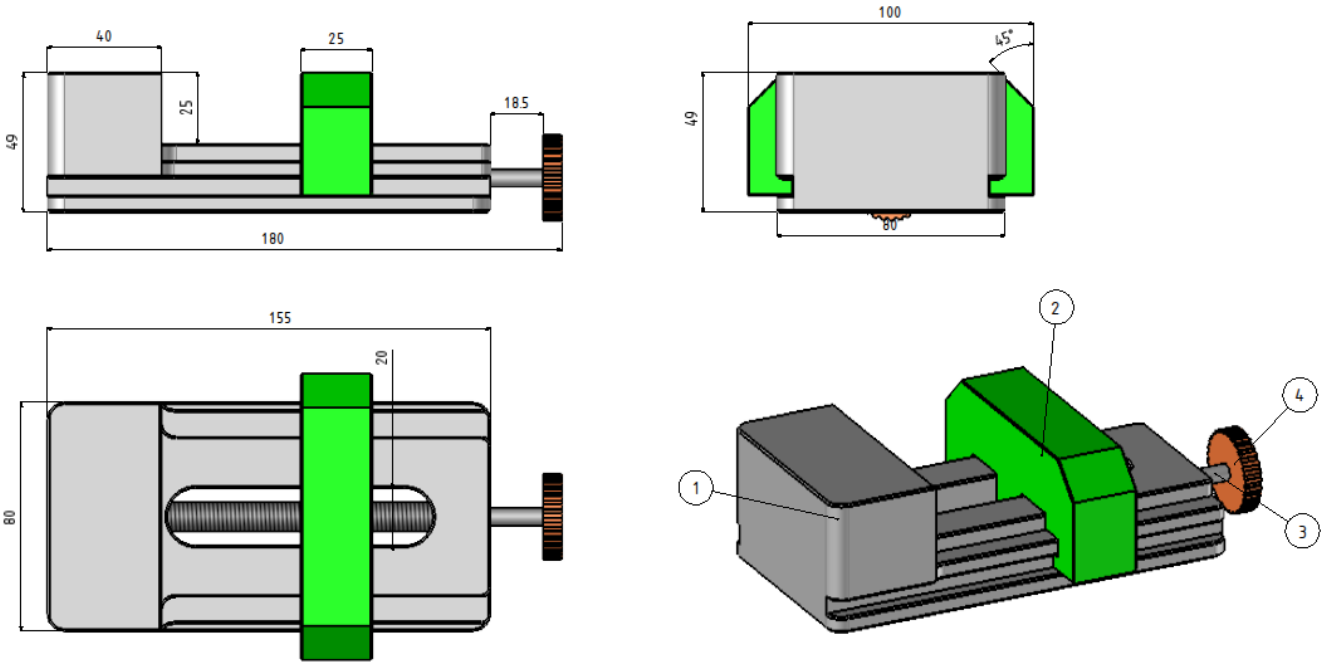
Now you will learn how easy it is to add some basic dimensions to your drafting document.

- Go to the **Detailing** tab and select **Dimension**.



The dimension tool in a **TopSolid 7** drafting acts just like the one in design. This means that it is an "intuitive" dimension tool. For example, if you select one edge, and then a parallel edge you will get a linear dimension. On the contrary, if you select two edges that are not parallel with each other you will get an angular dimension. If you select a circular edge, you will get either a diameter or a radius. For more information on dimensioning, consult the online help by clicking the icon and selecting **Help**.

- Add the following dimensions to each of the views.



You have now completed the BOM and drafting sections of this tutorial. The goal of this tutorial was to introduce you to some of the key concepts of **TopSolid 7**. We hope that you have enjoyed this tutorial and learned some useful points about **TopSolid 7** as well along the way.