Parametric Modifications and Spreadsheet

This task helps you to repair the given Roman lock bolt and retrieve the connecting rod. First, we shall modify the existing dimensions and set a new target position. Next we shall hide and unhide the solid and then edit the profile.

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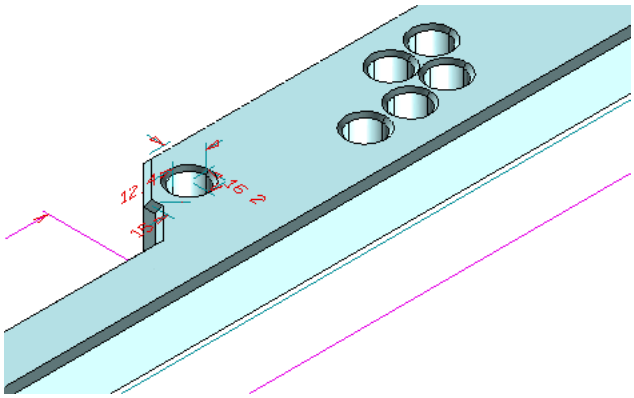
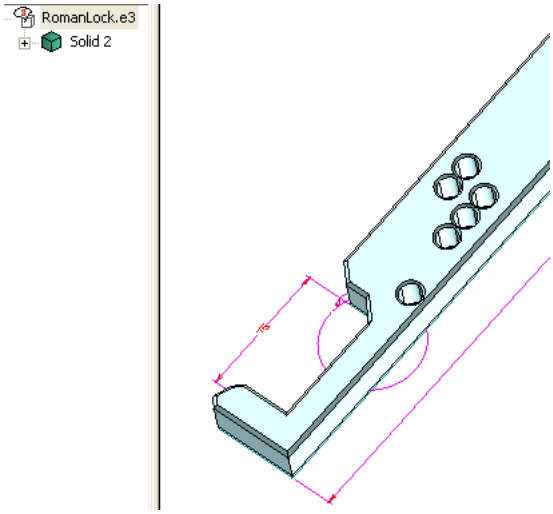
1. Step 1: Show and Change Dimensions

In this step, we shall change one of the holes by modifying the value of the driving dimension that controls its location. But before we can change the dimensions, we ought to be able to see them.



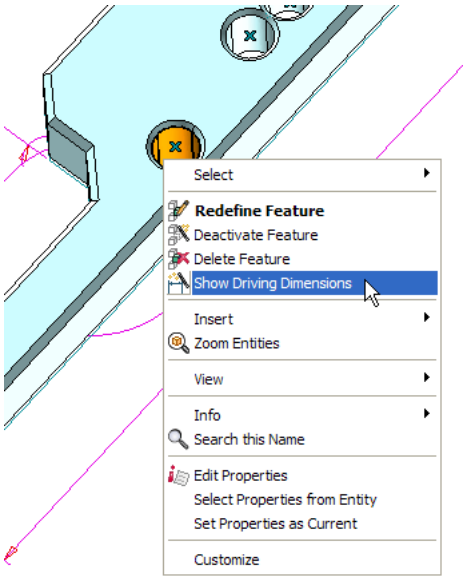
Since we will be making changes to the dimensions and the solid itself, we will leave the History Tree open.



Now we can start with the changes.

- Zoom in using the **Zoom Window**, closer to the holes for a better look.
- Right click on the lower hole (not the chamfer).
- Select **Show Driving Dimensions**.

Notice that when you select a feature in the Graphics Area, the History Tree expands to display the associated feature.

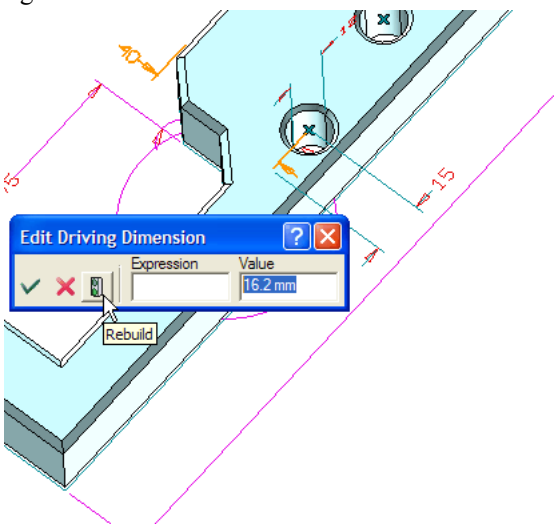


Let's change a dimension.

- Double click on the 40 dimension.

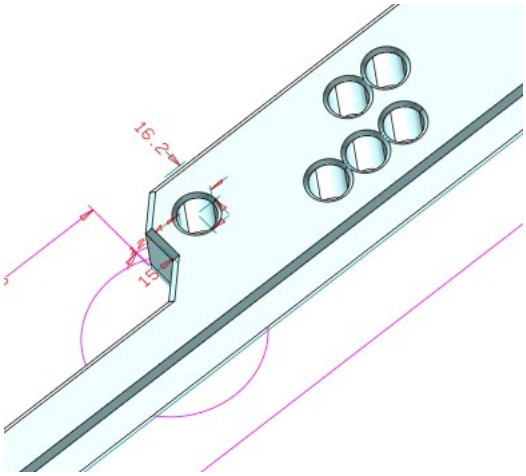
OR

right click and select **Edit Dimension...**.



- Change the Value to 16.2.

- Click on **Rebuild Model** to update the model with the new dimension value.



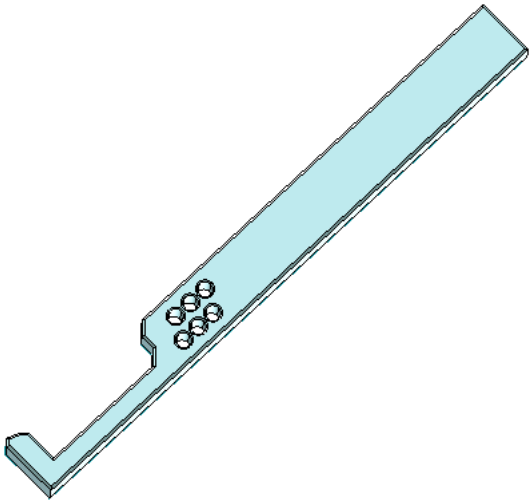
For your information: Hitting OK or Enter when you are changing the driving dimensions records the changes to the value, but does not update the model. This allows you to change several values prior to rebuilding. This change was painless, but complex models can take longer to rebuild.

If you hit OK or Enter by accident, do not worry. There is a **Rebuild Model** button on the Standard toolbar too, but it will be active (not grayed out) only when the model actually needs rebuilding.

One change done and a couple to go...on to Step 2!

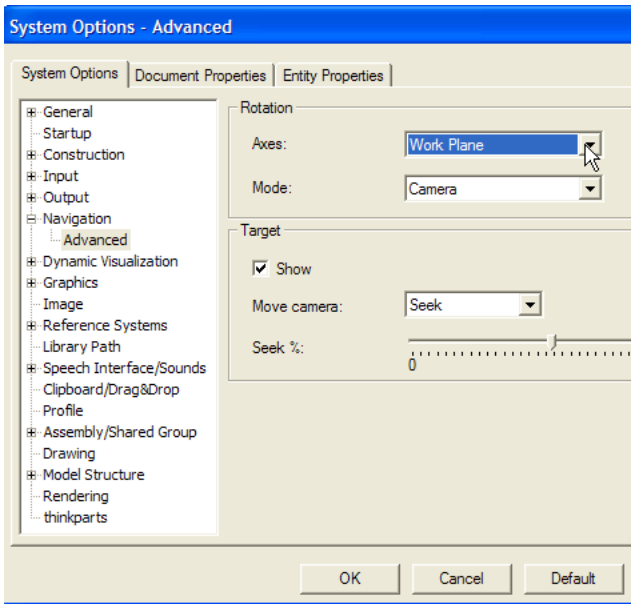
2. Step 2: Parametric Changes to the Model

In this step we shall change a few more dimensions to match the bolt to the key. We will learn how to set the dynamic rotation target position and also to rotate the view of the model around a specific point, as we go along.

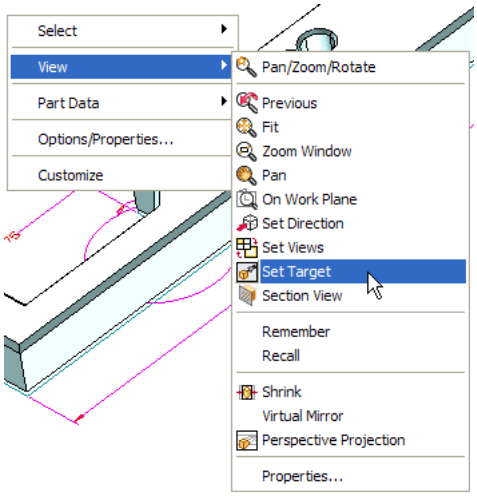


Since most of our changes are near the holes, let's make it easier to rotate the bolt.

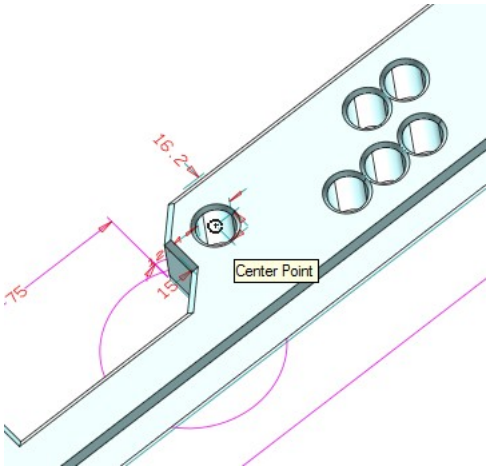
- Select **Options/Properties**
- Choose Work Plane as the option from the Rotation axes: drop-down list under the Advanced section of the Navigation category in the System options tab.



- Right click in the Graphics Area.
- Select **View**➪**Modify**➪**Set Target Position**.

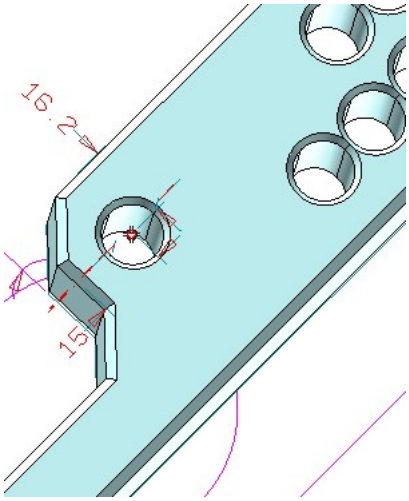


- Click on the Center Point of the hole we just modified.



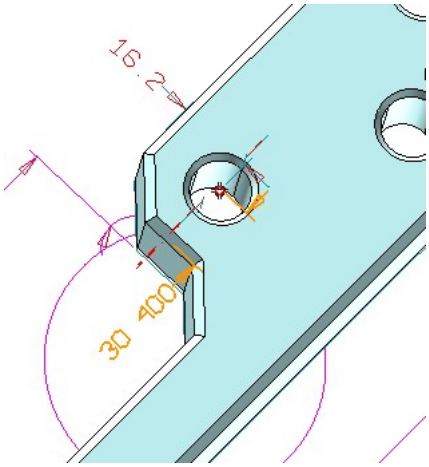
- Dynamically rotate the bolt.

Notice that the model now rotates around the point you just selected.



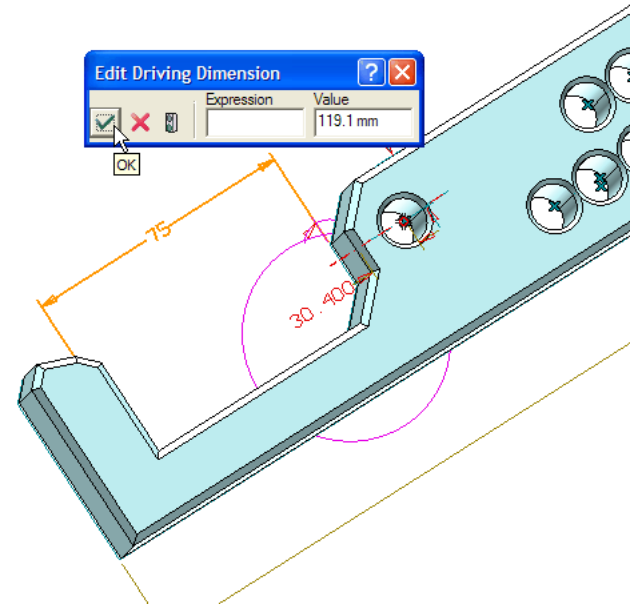
Let's change some dimensions. We shall put a hold on rebuilding the model, until we are done with all the changes.

- Double click the 15 dimension.
- Change the Value to 30.4.



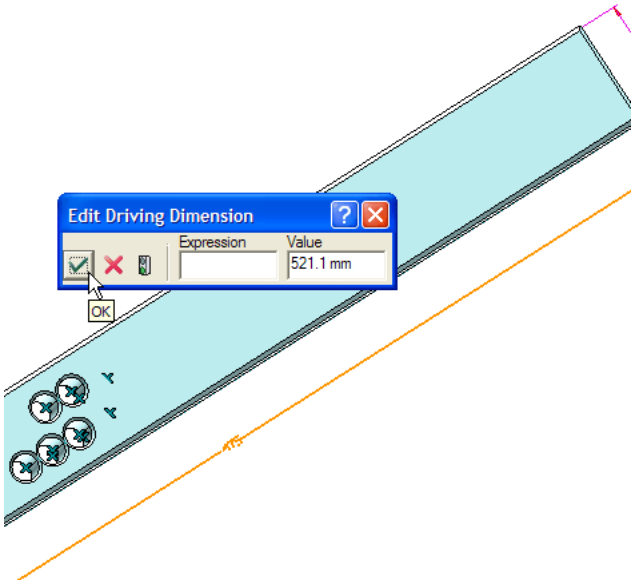
- OK.

The dimension you just changed is now displayed in a different color, and the value has trailing zeros (30.400) to let you know it has been modified but not rebuilt.

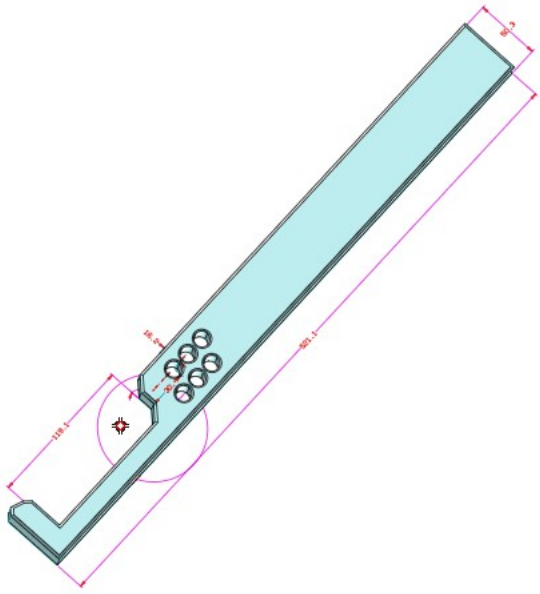


Now we shall change the shape of the bolt, starting with the overall length.

- **Zoom Out** until you can see the length dimension.
- Double click the 475 dimension.
- Change the Value to 521.1.
- OK.



- Double click the 75 dimension.
- Change the Value to 119.1.
- OK.

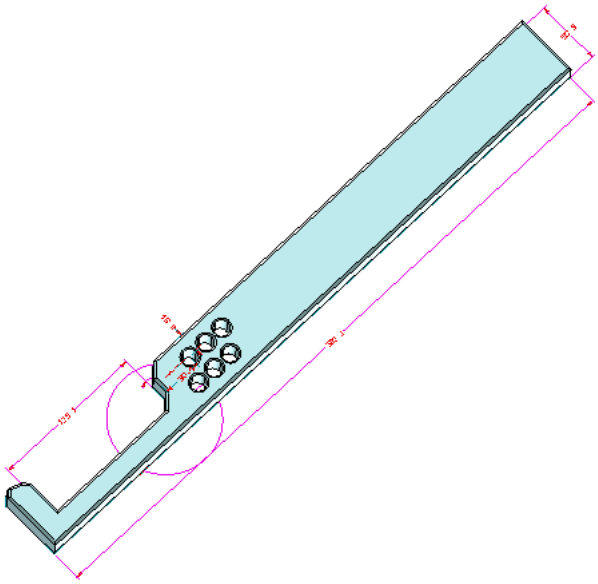


Now we shall **Rebuild Model** the model.

- Click on the **Fit View** button, so that you will be able to see the changes.

Note that **Fit View** always resets the target position to the center of the display.

- Hit **Rebuild Model**.



Great job, but let's clean it up.

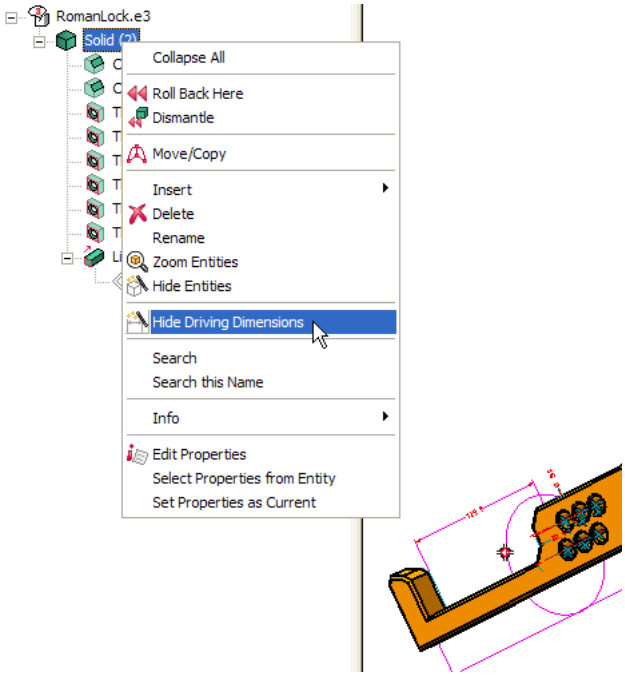
Let`s hide the dimensions on the entire solid. We can do that using the History Tree.

- Right click the Solid in the History Tree.
- Select **Show Driving Dimensions**.

Showing the dimensions this way displays all the dimensions on the entire Solid. Since some dimensions

were visible and some were hidden, we must show all the dimensions before we can hide all the dimensions.

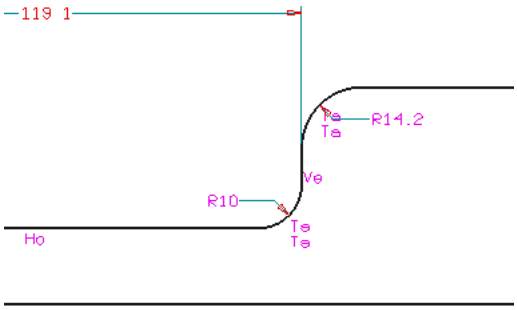
- Right click the Solid in the History Tree again.
- Select **Hide Driving Dimensions**.



Nicely done! In the next step, we shall change the shape of the bolt by editing the profile.

3. Step 3: Editing a Profile

In order to make the key fit in the bolt, we need to modify the shape of the bolt by changing the chamfers to fillets (or rounds). We will do this by editing the profile of the bolt.



Profiles define the shape of solids, and every protrusion or a slot uses a profile to define its shape. Here is the cool part: if you change a profile, the associated solid or feature changes, too! You have been creating profiles all along, you just may not have known it. Every time you selected lines and arcs for a **Linear Protrusion** or a **Linear Slot**, ThinkDesign turned them into a profile.

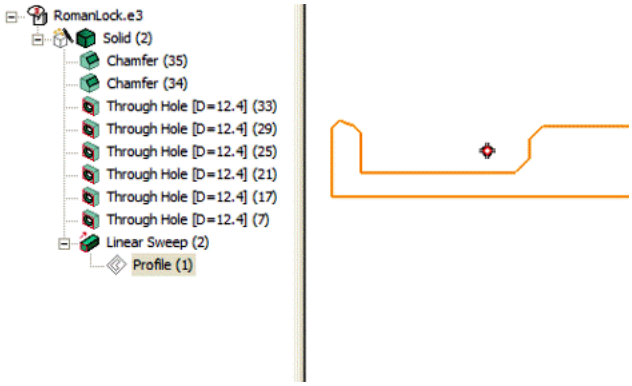
Before we edit the profile, we need to see it clearly.

- Right click on the Solid in the History Tree.
- Select **Hide Entities** from the context menu.

The solid is hidden, leaving just the profile.

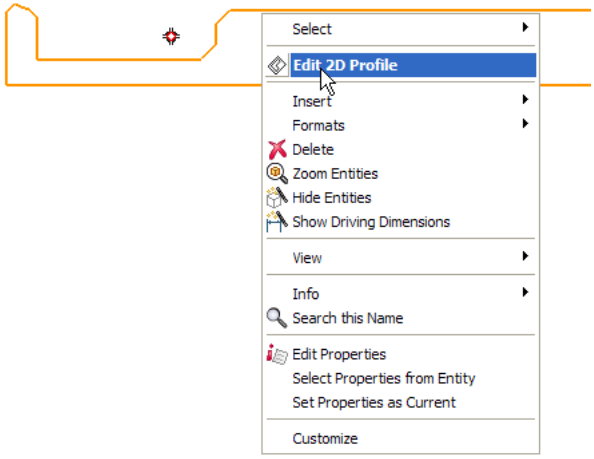
- Click on the profile.

Notice that you cannot select individual lines.



You cannot select individual lines because ThinkDesign combines all the lines into a single profile object. So, if a profile is a single object, how do we modify the individual lines?

To edit the individual lines, arcs and circles that make up a profile, you have to use Profile Mode. **Edit Profile** to edit a profile, and **Switch to Model** when you are done.

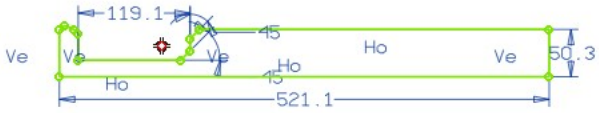


- Right click on the Profile.
- Select **Edit Profile**.

NOTE

Starting from Version 2006.2, think3 is introducing a New Sketcher environment. When you Try to edit a profile created in the older versions(2006.1 and earlier), ThinkDesign displays a warning message as shown in the picture below.

Do not be alarmed by the change in the background! This color change lets you know you are in Profile Mode. You will get used to it, and it may even start to grow on you.

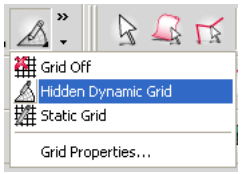


Now let's look at another way to switch between profile and model modes, simply click on the Profile tab below the work space. Remember, ThinkDesign will assume that it is your intention to convert any selected entities into a profile.

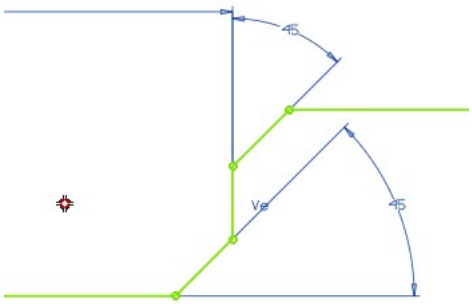
Click the Model tab below the work space to switch back to modeling. Simple.

So let's work on those changes. We shall start by setting up the view.

- Turn off grid visibility. Select **Hidden Dynamic Grid** from the Grid drop down button, in the status bar.



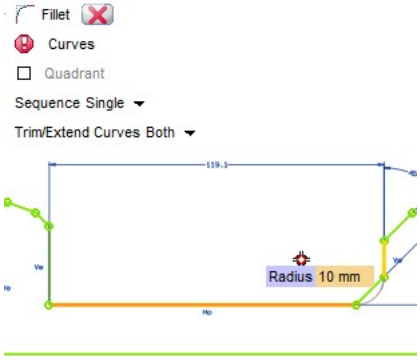
- **Fit View** .
- **Zoom Window** closer for a better view of the chamfers.



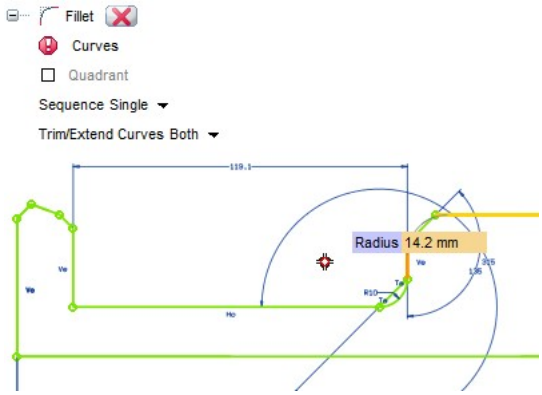
Now we will replace the two chamfers with two fillets.

- Start the **Insert Fillet** command.
- Set the radius to 10. Radius10

- First click on the bottom horizontal line.
- Now click on the vertical line as shown below.

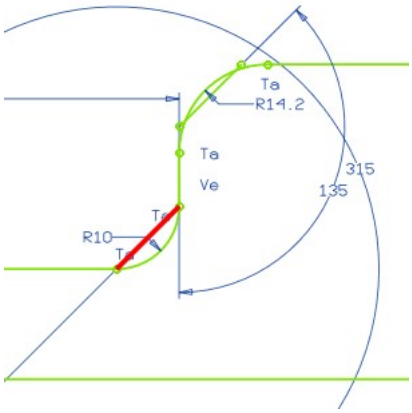


- Set 14.2 for the radius value. Radius14.2
- Now click on the top horizontal line.
- And then click on the vertical line.



That change was easy enough, but the chamfers are still there. Let's get rid of them.

- Start **Smart Delete**.
- Select the two chamfer lines.
- Esc



With the changes complete, **Switch to Model**.

NOTE :

After switching to model, the hole gets shifted to other place due to change in reference dimension(after adding fillet of 14.2 mm) so kindly redefine the hole & bring it back to its original place by keeping the 1st line 16.2 & 2nd line as 30.4mm & rebuild.

- Hit the Model tab

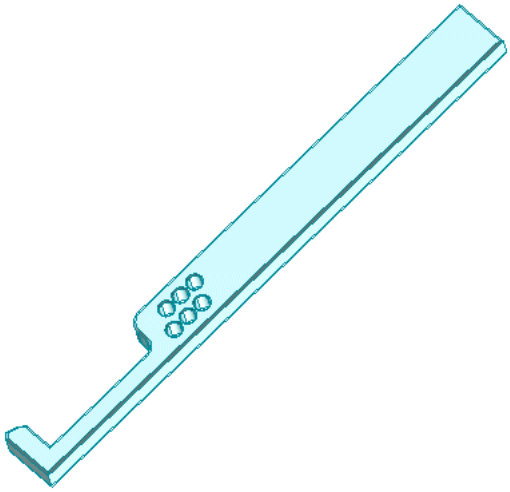
OR

right click in the workspace to **Switch to Model**.

Next, we shall see how the changes to the profile effect the model.

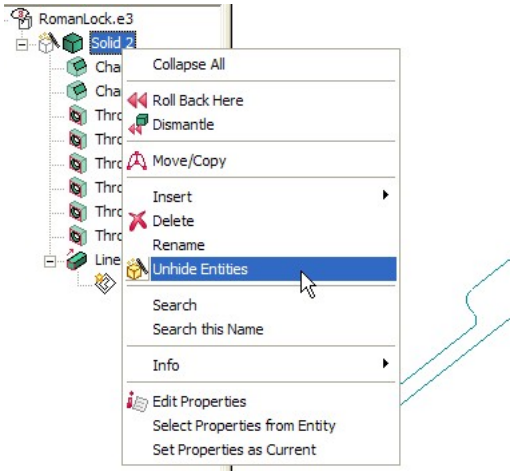
4. Step 4: Update the Solid

Let us check the effect of modifying the profile on the solid.

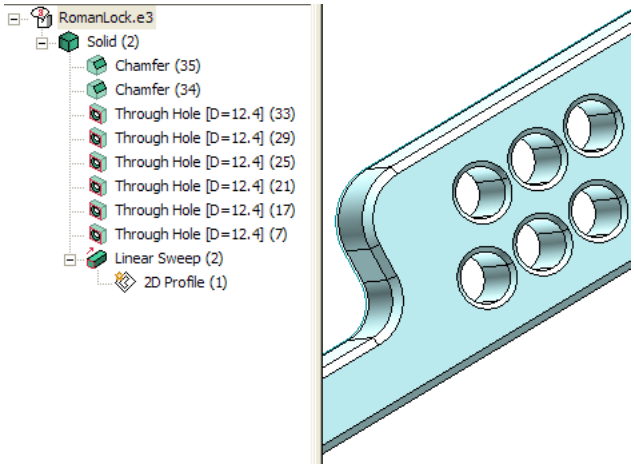


Let's **Unhide Entities** the Solid and take a look.

- Rotate the view a little, if you like.
- Right click on the Solid in the History Tree.
- **Unhide Entities** .



- Right click on the Solid again.
- **Hide Driving Dimensions** .

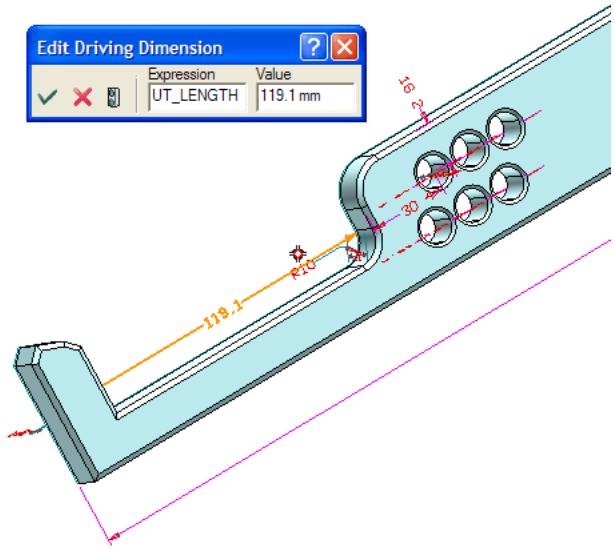


Now, the solid (including the chamfered edge) is automatically updated!

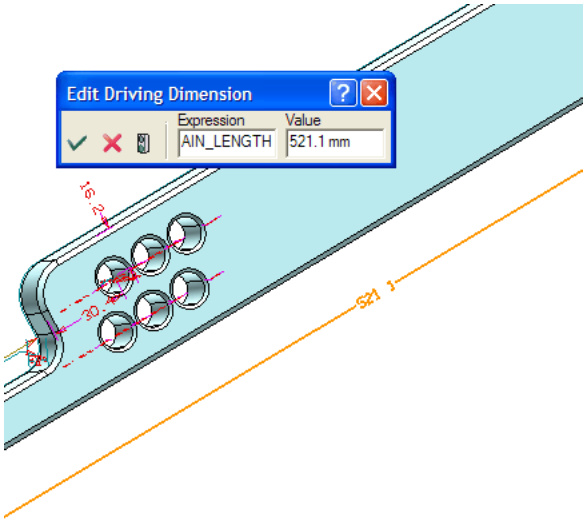
5. Step 5: Spreadsheet variables.

Now let's assign variable names to some of the dimensions in order to modify solid's dimension directly from spreadsheet.

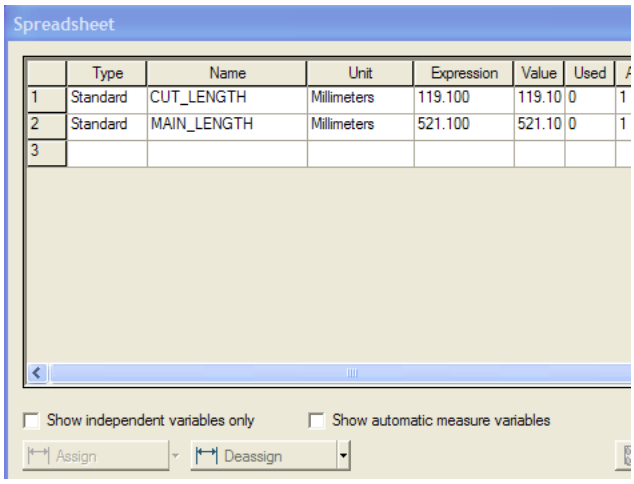
- First Right click on the Solid in the History Tree and choose Show Dimensions.
- Double click on dimension 119.1
- Insert expression as CUT_LENGTH.
- Click **Rebuild Model** .



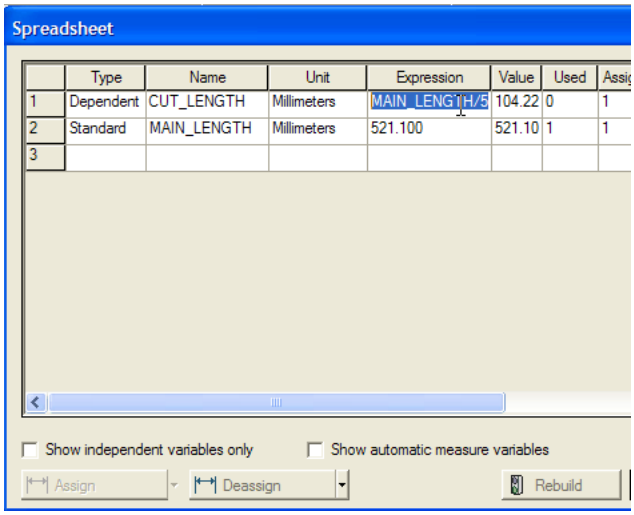
- Double click on dimension 521.1 and write the expression as MAIN_LENGTH.
- Click **Rebuild Model** .



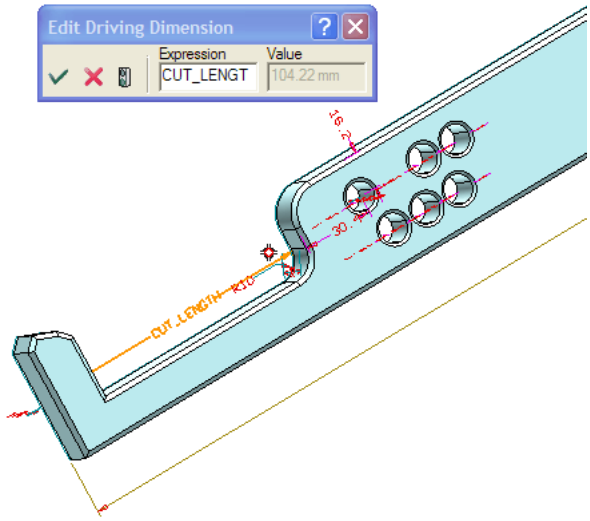
Open **Spreadsheet**, we can see these two variables.



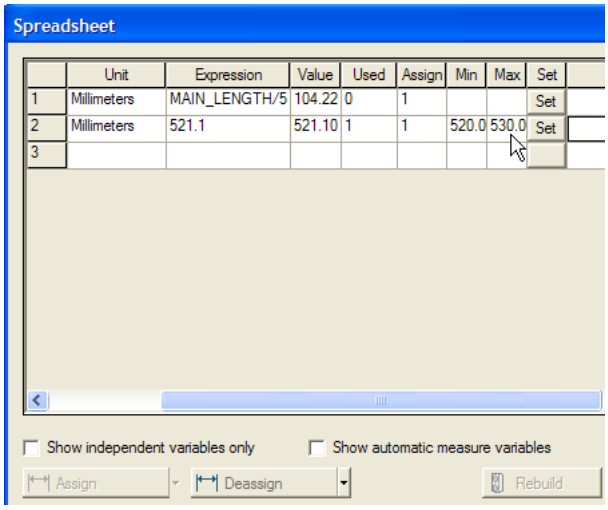
- Modify the expression of the first variable to MAIN_LENGTH/5.
- Apply Rebuild.



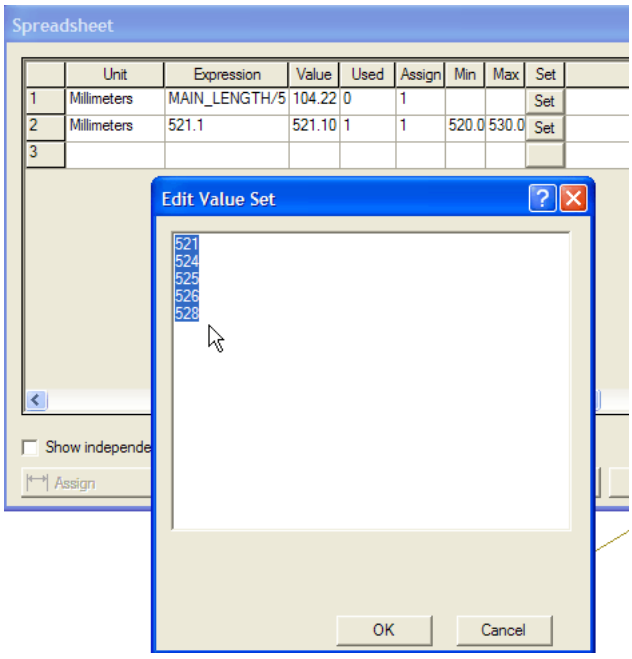
Double click on CUT_LENGTH. The value field has been grayed out. This indicates that dimension is linked.



We can also set a continuous range of values by adding Min and Max values in the spreadsheet.



OR Click on the Set button to enter a set of separate values.



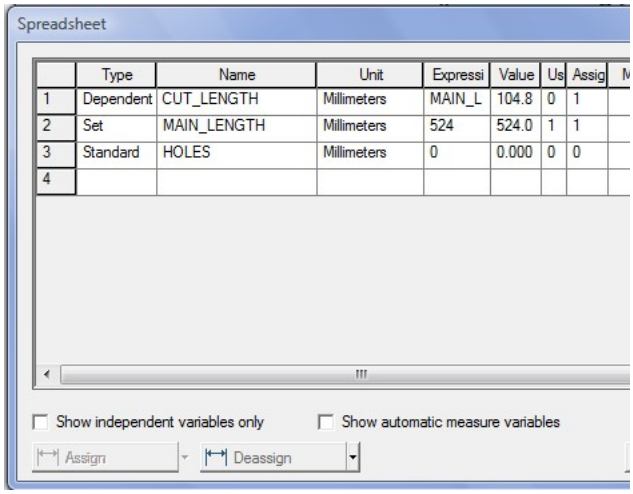
use Enter, for each new value.

As you can see MAIN_LENGTH value is automatically updated to 521.000(closer value to 521.1 is updated).

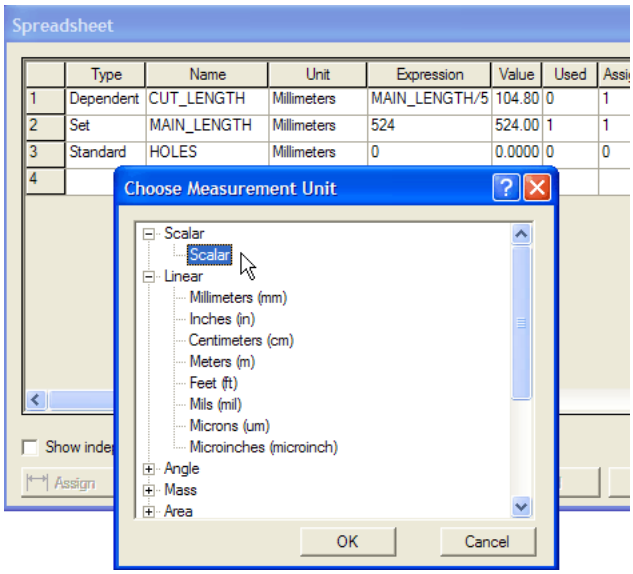
Now try to change the MAIN_LENGTH to 523: value is automatically updated to 524(closer value).

Update the model.

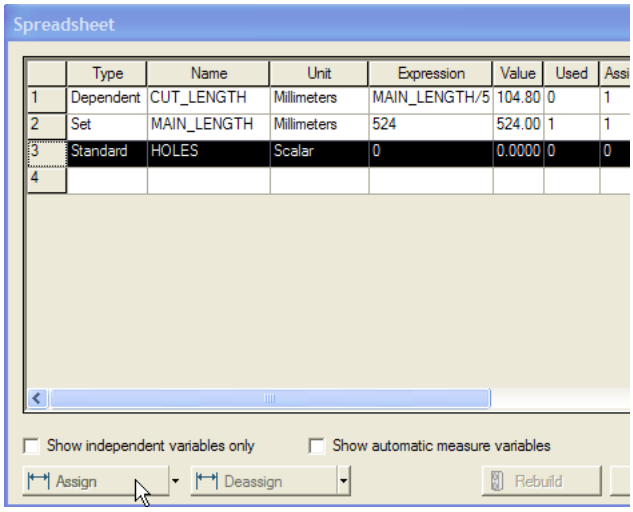
Okay, now let's create a new variable called HOLES.



Click on Unit - Millimeters box and select Scalar from the box.

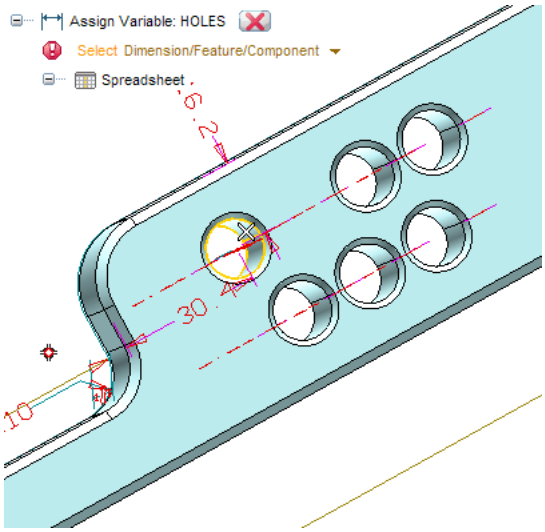


Now select the entire row and say Assign.



Now, we need to select either a feature or dimension or a component for assigning variable.

- Select one of the holes from the model.
- Hit Esc.



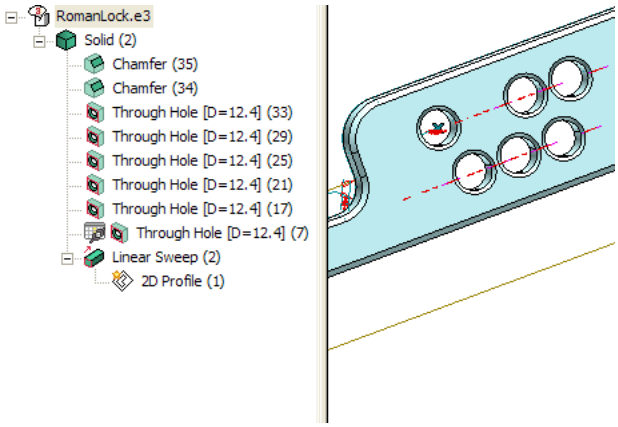
In the next step let's see how we can activate/deactivate hole feature using **Spreadsheet** only.

6. Step 6: Activate/Deactivate using Spreadsheet.

Open **Spreadsheet**.

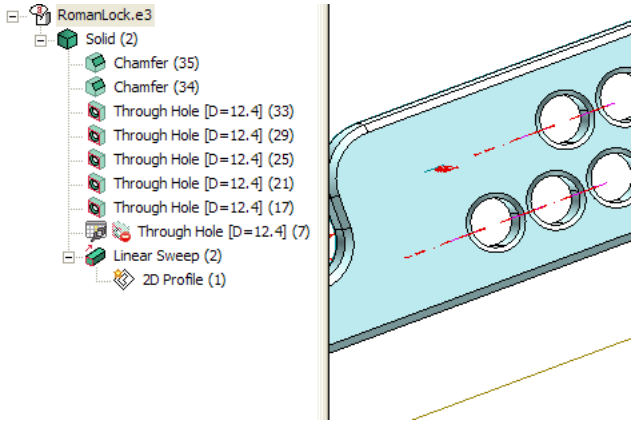
Change the expression to 1 in spreadsheet and rebuild. We can see the hole has been activated.

Observe that in the history tree we can see a spreadsheet icon besides the hole feature. It means the hole feature is related to a variable.

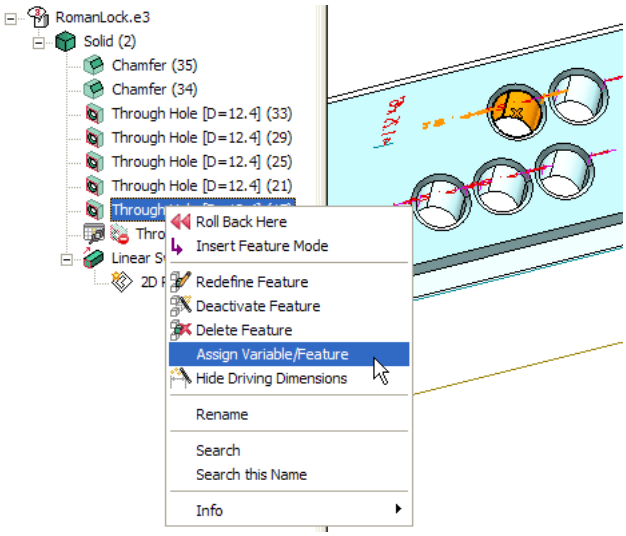


Now change the value to 0 and rebuild.

The hole gets deactivated.

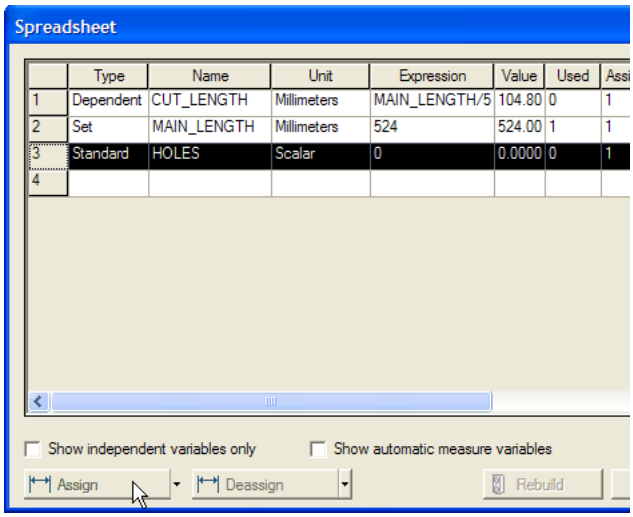


Right click on other hole features in the history tree and say Assign Variable/Feature.

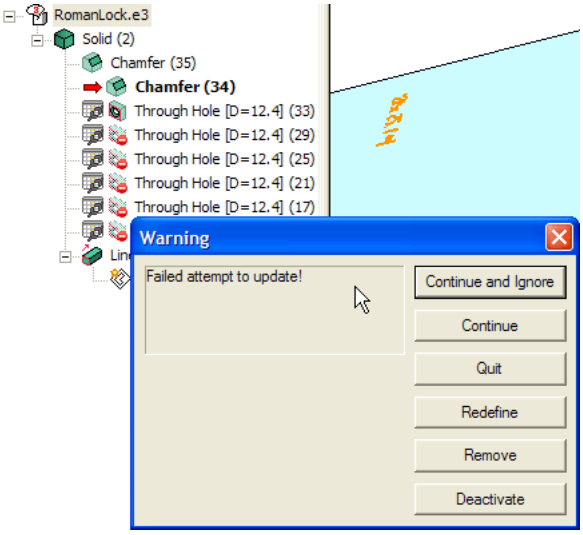


Again select the entire row in **Spreadsheet** and hit Assign.

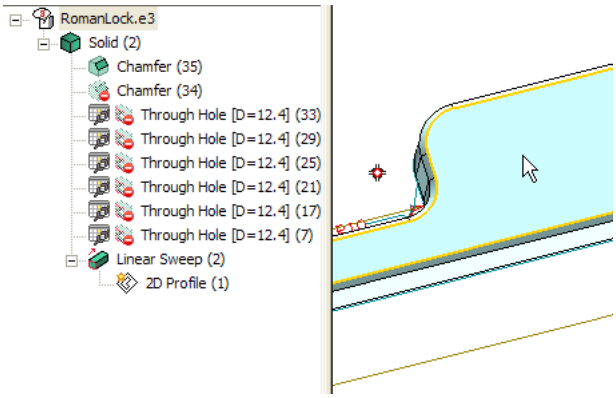
Follow the same steps for all the hole features.



Apply Rebuild.

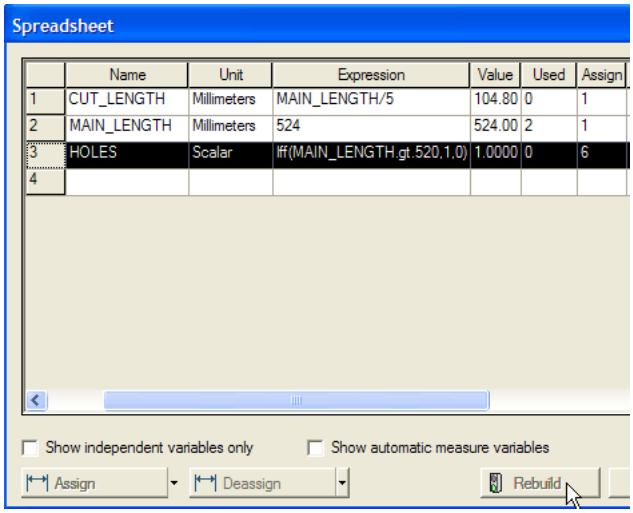


We get error message saying that chamfer fails. Deactivate it.

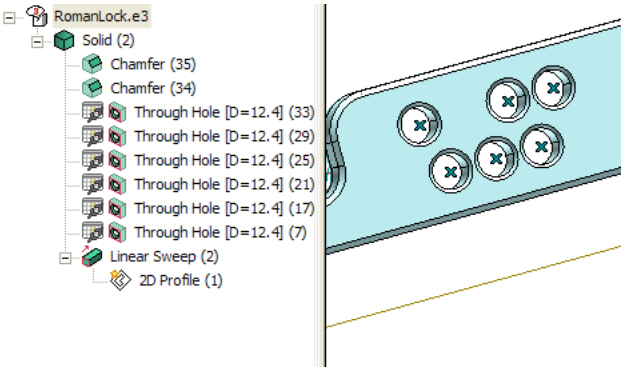


Open **Spreadsheet** again and write a new expression for the HOLES if(MAIN_LENGTH.gt.520,1,0).

It means, if MAIN_LENGTH is greater than 520, set variable to 1(Activate) otherwise 0(Deactivate).



Rebuild now and test for different values.



Nice work, Congratulations.