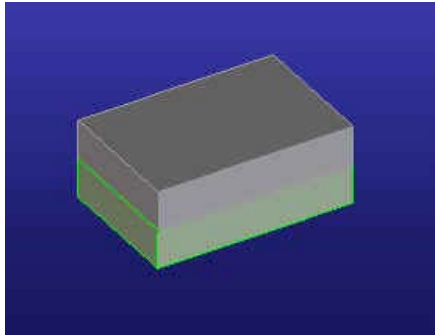# SolidGenius Tutorials

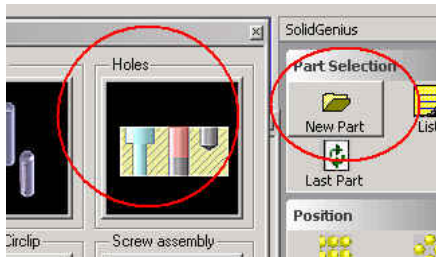<u>Example 1</u>

<u>Subject:</u> How to design countersunk-hole-pattern with screws and nuts.

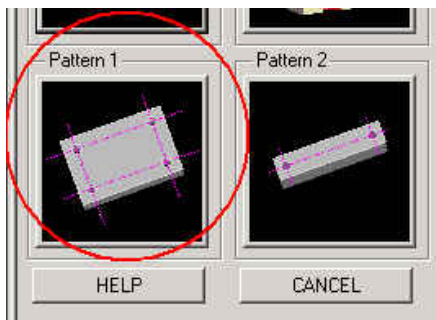- Start SolidGenius

- Load package file "tutorial1_start.pkg" from the "Tutorials" folder under the SolidGenius installation directory
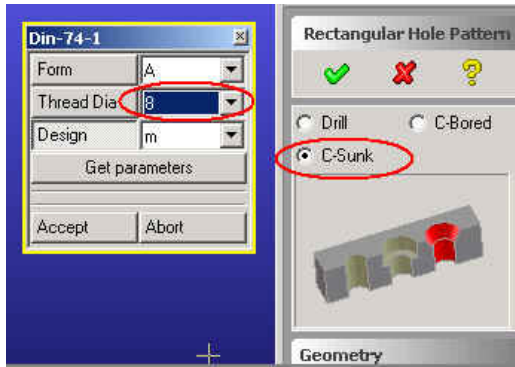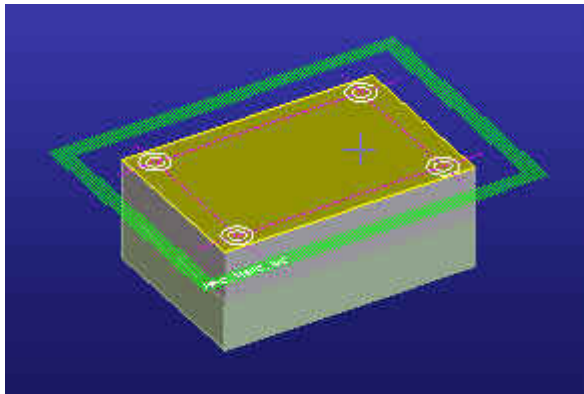

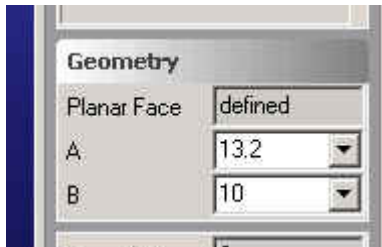- Select [New Part] and [Holes]

- Select [Pattern1]

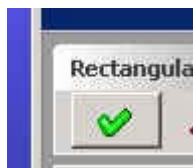- Select [C-Sunk] and thread diameter 8 and [Accept]

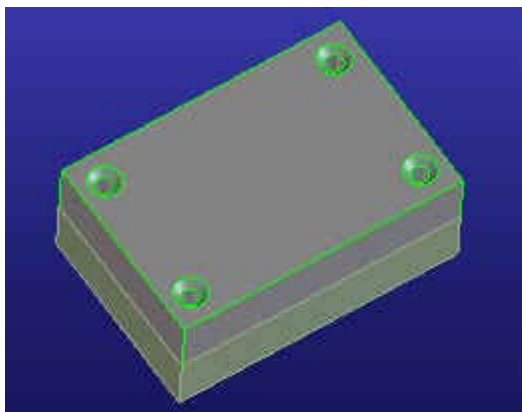- Select the planar face - You should see the preview

- Change the distance values [A] and [B] to see the different possibilities
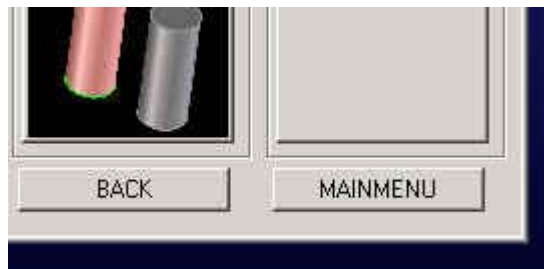
 [The values are measured from the edges !]

- Select [OK] to see the result.

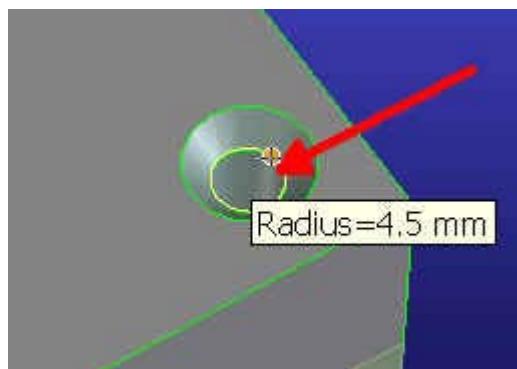- Go back directly to main menu and ….

.. select [Screws] …

… select [Countersunk Head] …


… select as an example [DIN EN ISO 2009]

- Without setting any parameters, simply select the lower circular edge of the countersunk.
You **don't have** to catch the midpoint of the edge by using [CTRL-SHIFT] !
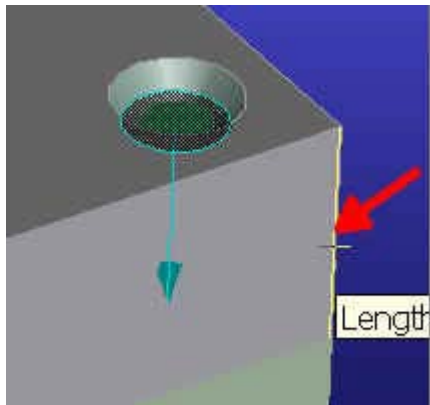It is enough simply click the circular edge !

- Notice how SolidGenius took the center and the direction automatically and selected the fitting screw diameter fitting to the hole !
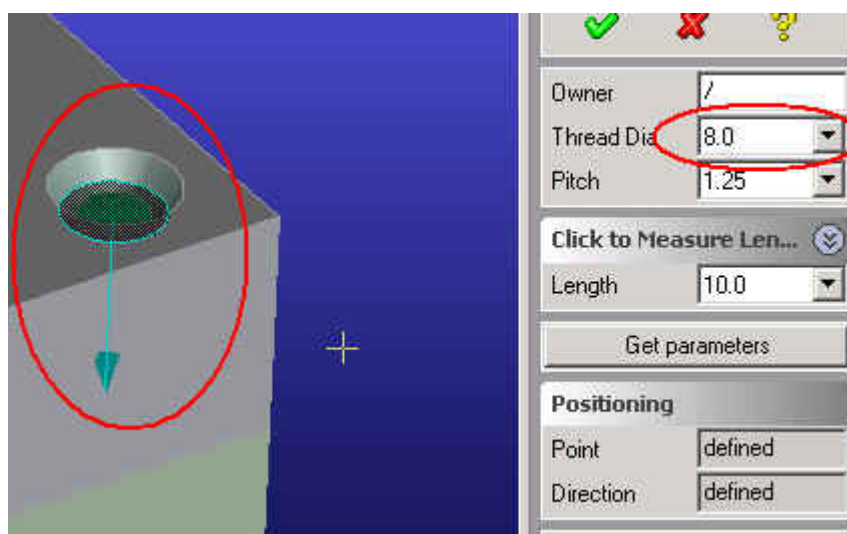
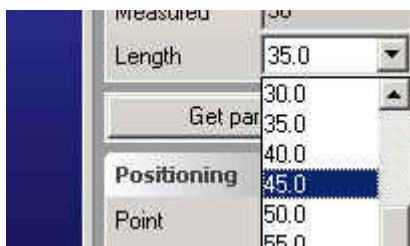- Open the submenu [Click to measure Length] and select [Edge Length]
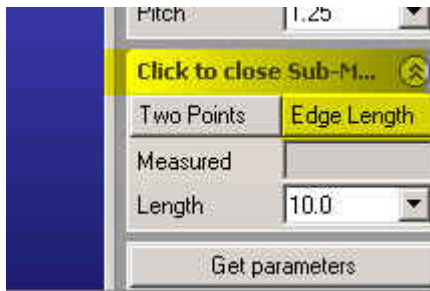
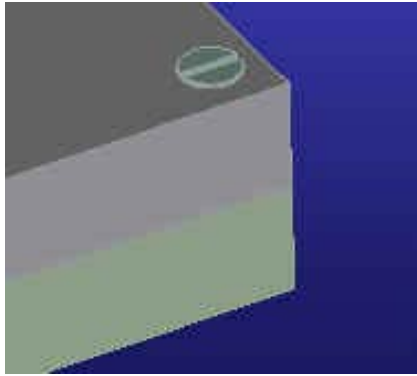- Select the vertical edge – its length is 30 mm

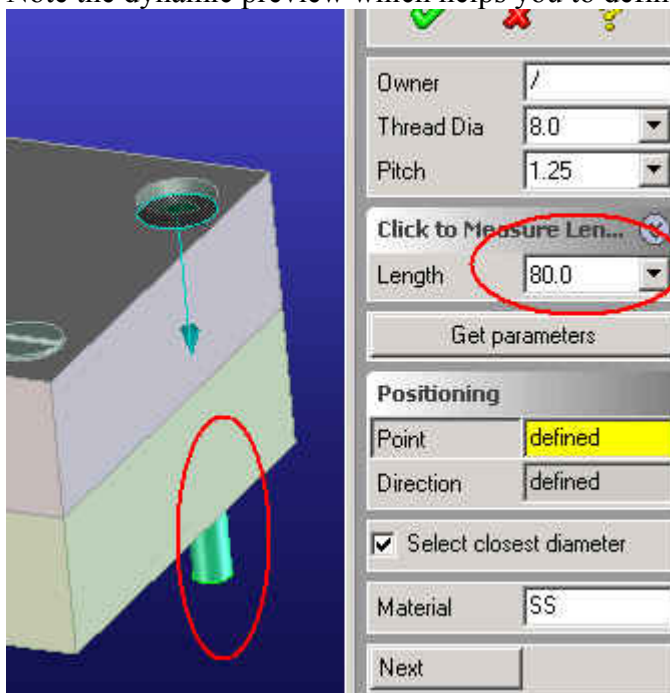- Notice how SolidGenius selects the next larger length value from the list of available lengths.

- Correct the selected value to 45 mm

- Select [NEXT] to create the screw.
(Please note that if you use the [OK] button instead of [NEXT] , screw dialog will close – but we want to keep it open to position another screw)

- Position the next screw by selecting another circular edge.
Change the length value to 80 mm to make sure that screw is coming out of the hole.
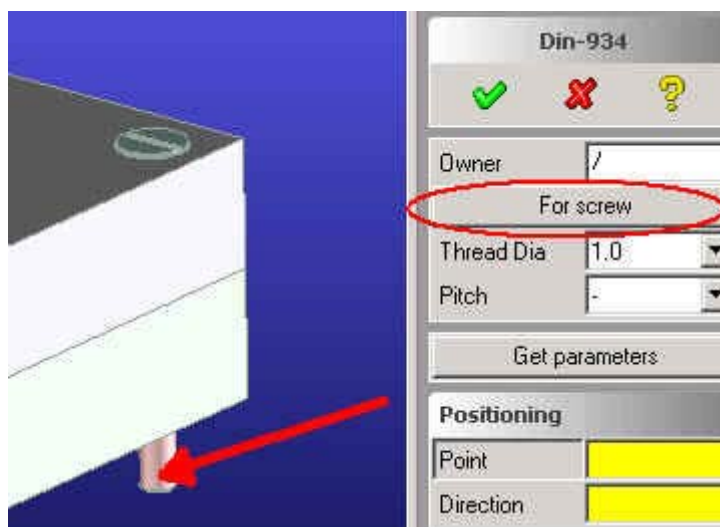Note the dynamic preview which helps you to define correct parameters.

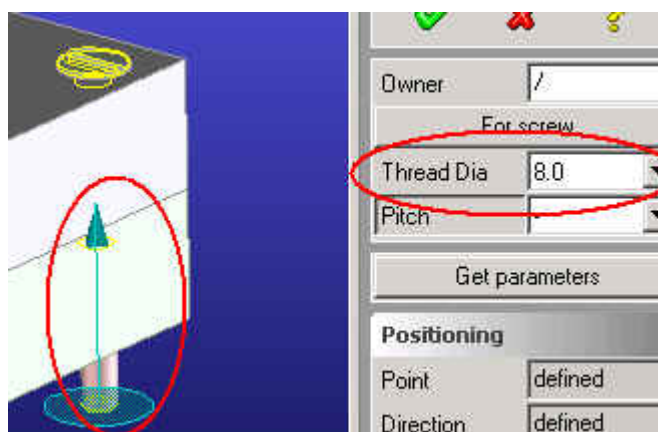- Go back to [MAINMENU] ..select [Nuts]
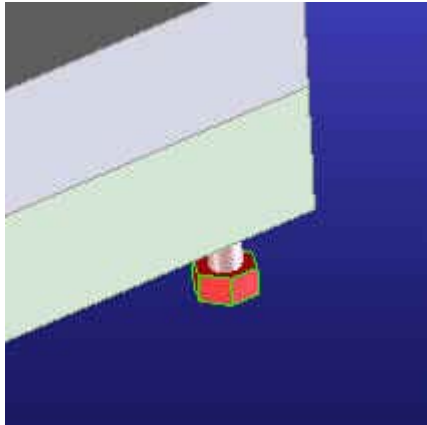
… and select [Hex] .. and [DIN 934]

- Without making any settings, select option [For Screw] and select the screw end which is looking out of the body ..
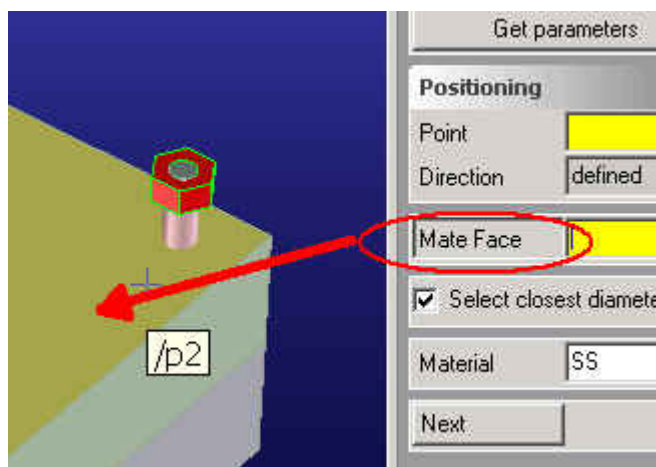

- Notice how SolidGenius selects the correct nut parameters and positions the nut at the end of the screw.
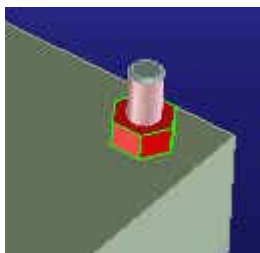

- Select [Next] to see the nut … (don't close the dialog yet !)

- Select [Mate Face] and select the lower face of the part ..

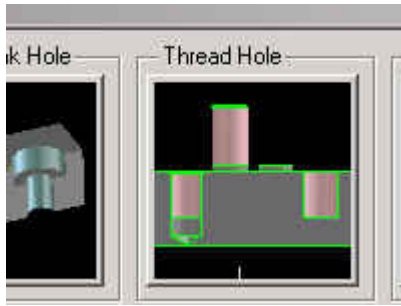- Notice how the nut is moved along its axis to mate the face !

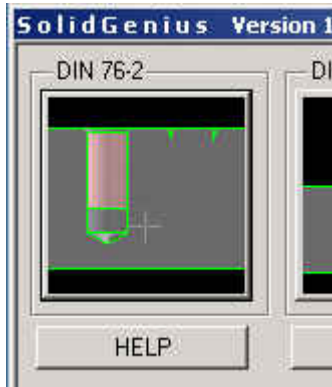- Close the dialog by canceling it.

- Now we should automatically create a threaded blind hole for the first screw we have already placed.
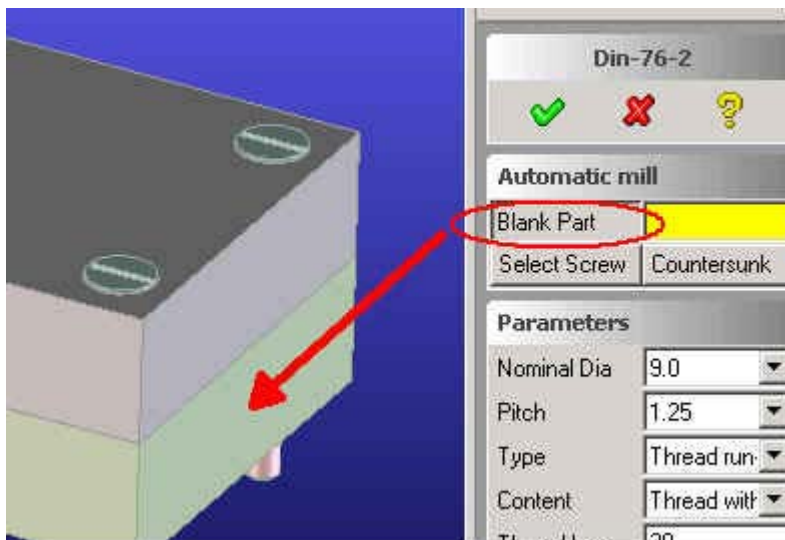
- Select [MAINMENU] and [HOLES]

- Select [Thread Hole]

- Select [DIN 76-2]

- We do not want to define the position and length of the threaded hole – the screw is already clashing with the lower part – let SolidGenius calculate the position and length by analyzing the clash of the screw with the blank part.
Hit [Blank Part] and select the lower part as blank object.

- Hit [Select Screw] and select the first screw which is ending in the blank part.

- Notice how SolidGenius sets the correct parameters.
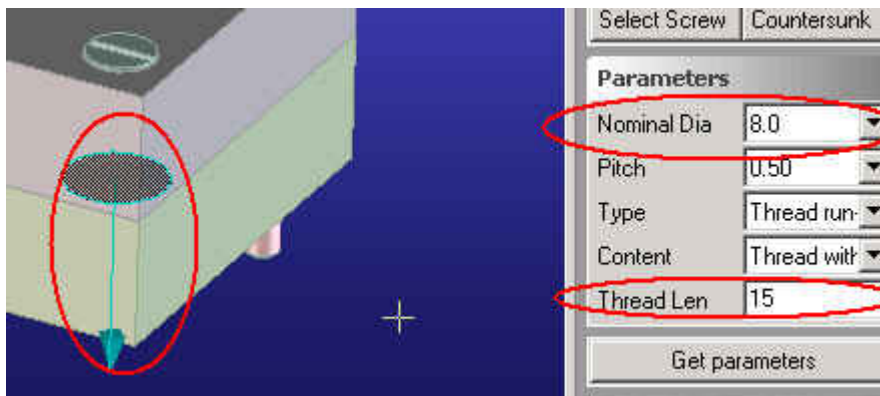
- Select [OK] to create the threaded blind hole automatically.

- Blank all the parts to see the result.