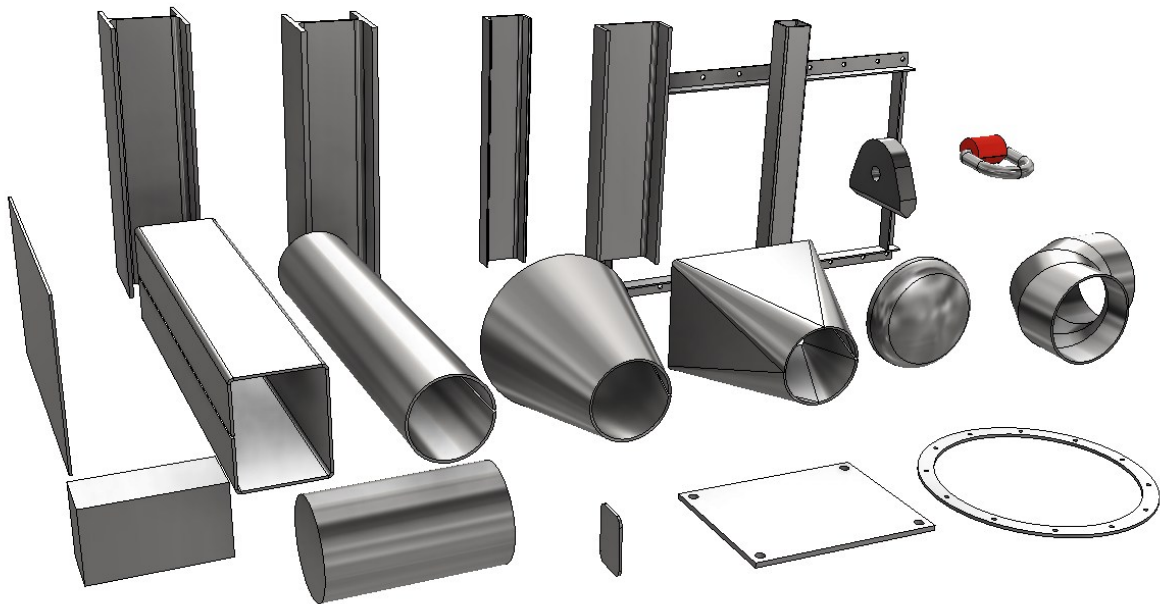


e-STEEL

For



AUTODESK
Inventor Professional



Plates, blocks, shafts, Structural profiles, folded plates, rolled plates, section elbow, lifting lugs and much more.

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1. Introduction.

Thank you for using our software.

e-Steel is an add-in that has been built specifically for Autodesk Inventor. It is therefore necessary that Autodesk Inventor Professional is already installed on your workstation before you can use the software. More specifically, this version of e- Steel is suitable for the version of Inventor 2025. Because Inventor 2025 works with the .NET 8.0 framework, this has the disadvantage that this version of e- Steel will not work with all previous versions of Inventor.

e- Steel was conceived and programmed in Belgium and works according to the metric system and international standards.

With e- Steel, it is possible to configure and then insert Plates, blocks, shafts, Structural profiles, folded plates, rolled plates, section elbow, lifting lugs and more within the working environment of the Inventor Assembly.

e- Steel creates sheet metal parts for the plates and normal parts for blocks, shafts and lifting lugs.

2. Install the software.

Before you can start installing e-Steel, you need to make sure that Autodesk Inventor Professional is installed on your workstation and that the version corresponds to the version of e-Steel.



Make sure that Inventor is completely shut down.

The files of the Add-in are in a directory called: "eSteel". This directory contains at least the files "eSteel.dll" and "Autodesk.eSteel.Inventor.addin".

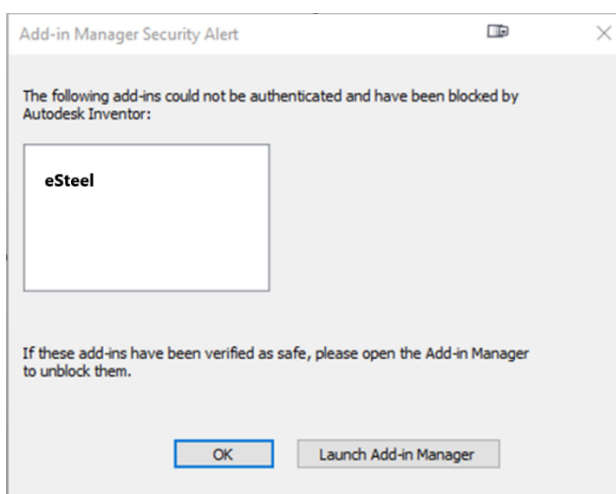
There is also a subdirectory "ButtonResources" which are all the images of the buttons. It goes without saying that you should not change anything in this structure. Any adjustment will result in the Add-in no longer working.

Copy the entire directory of the add-in to the following location:

C:\Users\ »YourName»\AppData\Roaming\Autodesk\ApplicationPlugins.

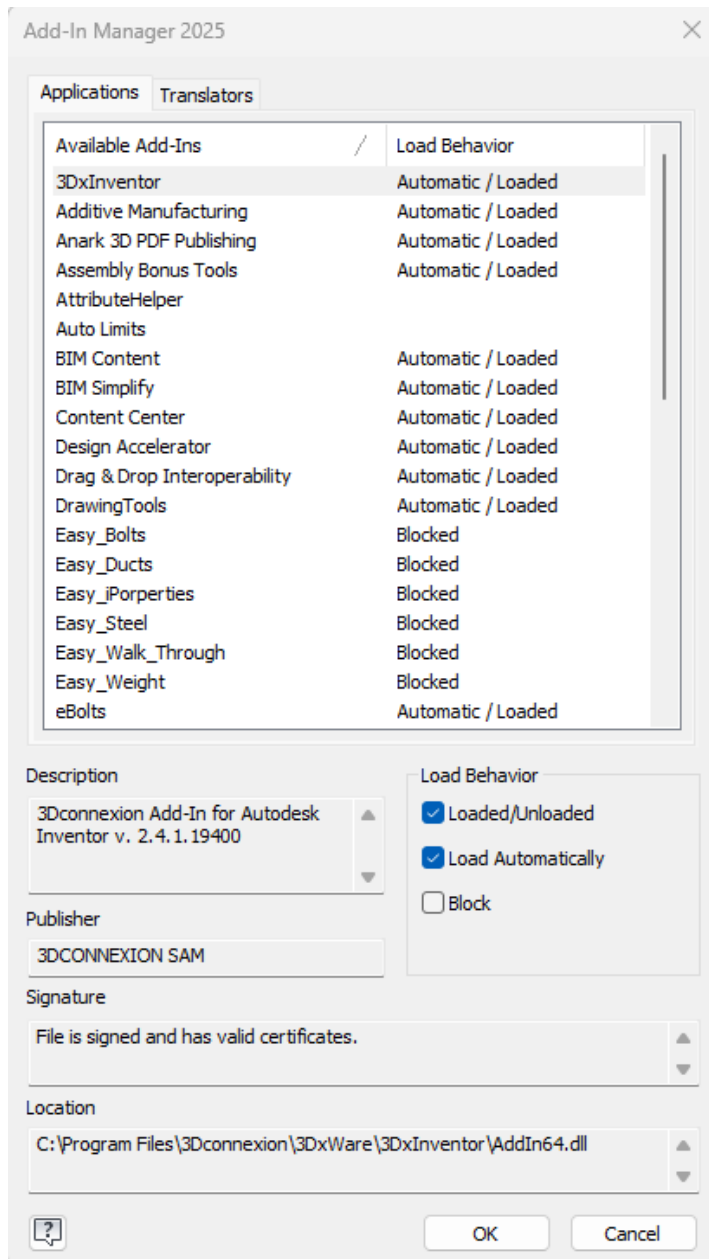
On the place »YourName» you must enter your username of windows.

Start Autodesk Inventor Professional, a window will appear during startup with the message that a new add-in has been found.



Accepted with OK, and Inventor will continue to be started.

On the **"Options" panel**, select the **"Add-in"** field and the "Add-in Manager" window will open. Select the **"Applications"** tab, if it is not selected, and search for "eStairs" in the list of applications.



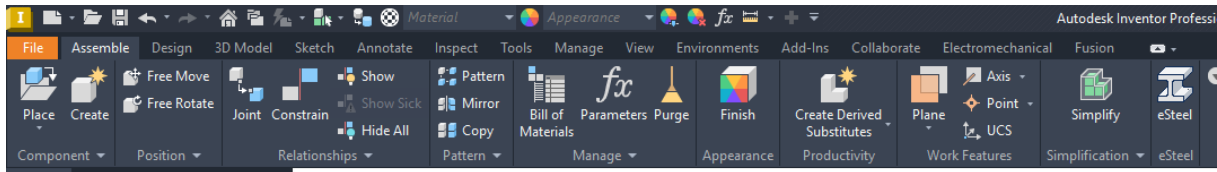
Select "Steel" and you will notice that at the bottom the "Load Behavior" is set to "Block". This is a safety for new applications.

Uncheck "Block" and check the "Load/Automatically" and "Loaded/Unloaded" options.

Click OK to exit the Add-in Manager and the e-Steel add-in is installed.

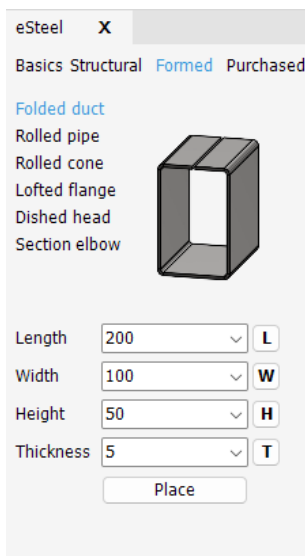
If you create a new assembly file or open an existing assembly file, the eSteel add-in panel will appear in the "Assemble" tab of the GUI (graphical user interface) of Inventor.

3. The graphical interface.

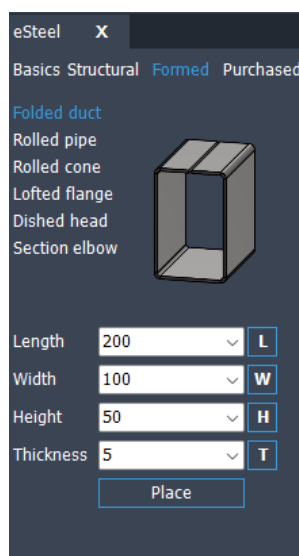


As you could already notice, the eSteel panel can only be reached via the Assemble tab of the Assembly work environment. This makes sense since the parts generated by eSteel must be placed in an assembly file.

Depending on which UI theme is selected, the eSteel panel will look like the one shown below. All other windows of the application will follow the UI theme used.

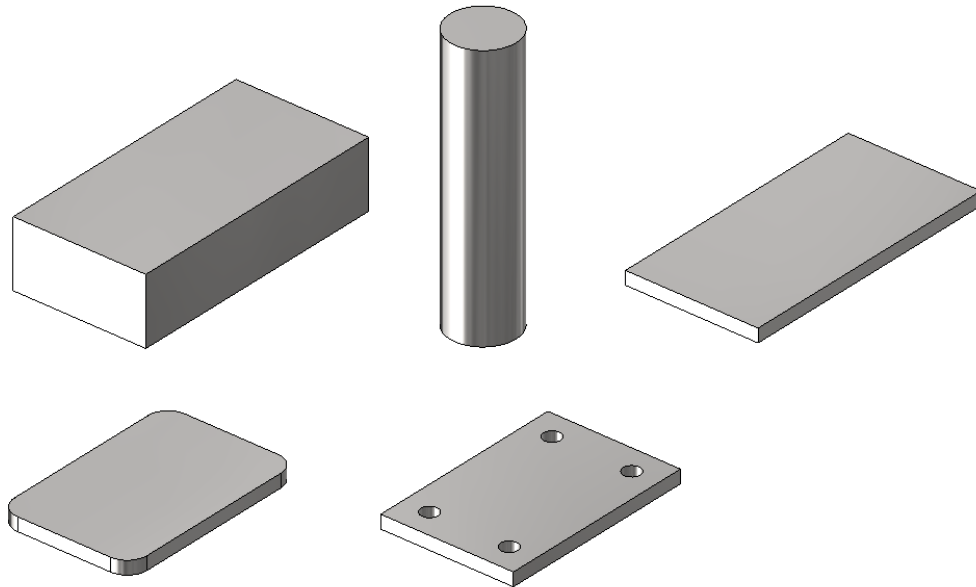


For the light UI theme

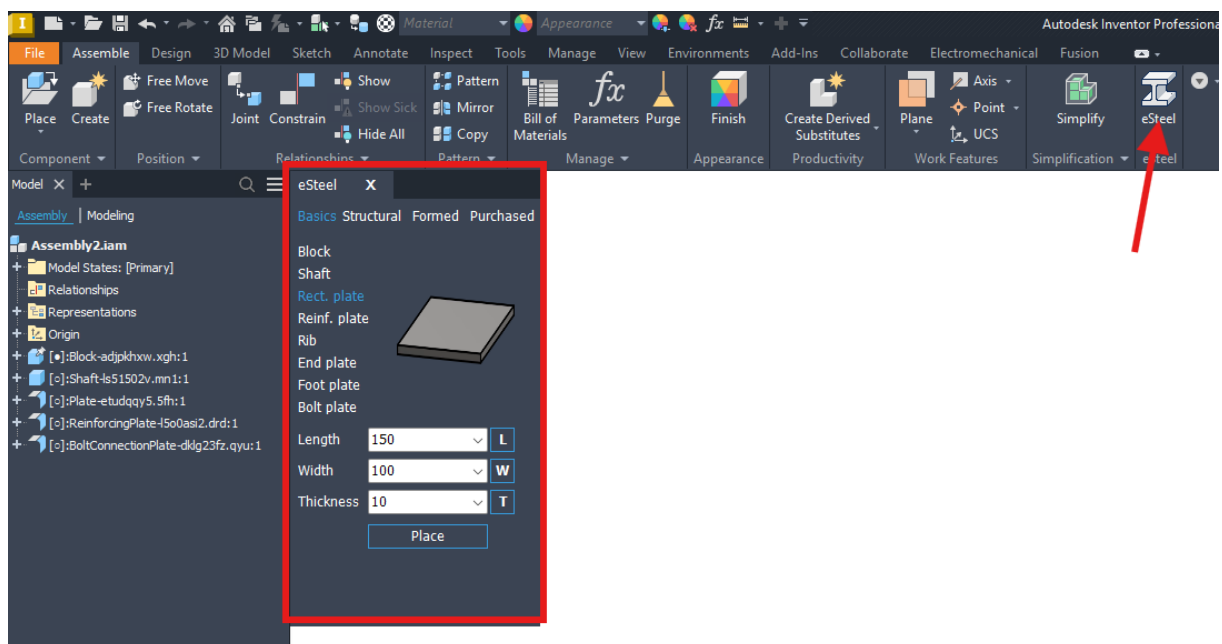


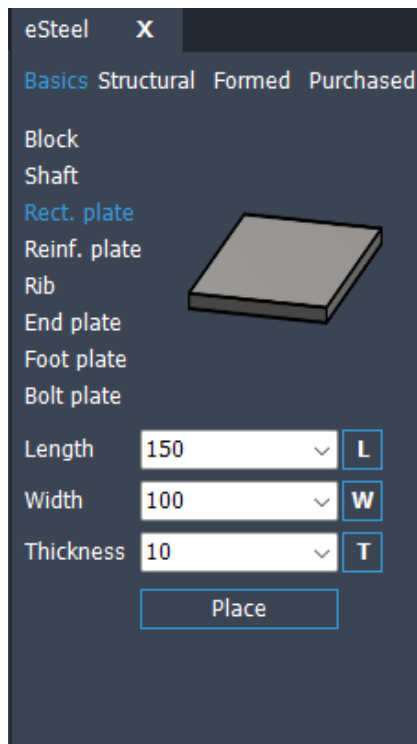
for the dark UI theme

4. Inserting Basics parts.



Click on the button of the eSteel panel to start inserting steel parts. The window to configure it will open at the top left corner of the graphical screen.





At the very top next to the title is a cross. If you move the cursor over it, it will turn red. With a click on the cross you can close the window.

You can also close the window by clicking on the escape key.

Below the title are four labels: Basics, Structural, Formed and Purchased.

If the Label Basics is selected the following vertical selection of items will appear. Block, Shaft, Rect. Plate, Reinf. Plate, Rib, End plate, Foot plate and Bolt plate.

In this case the rectangular plate is selected.

If you hover over the labels, the items will be preselected, and the picture will be shown of the preselected item.

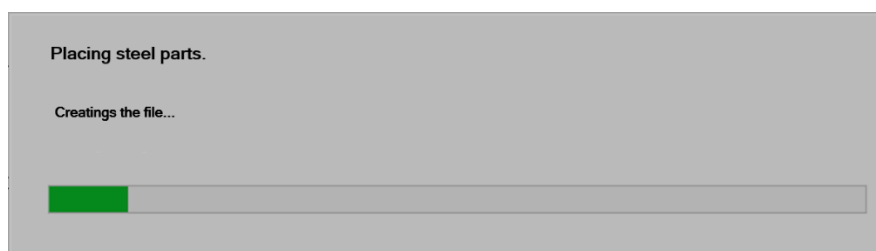
If you click on the label, the label text will change to blue the picture will be of the new item and the combo boxes will be modified according to the new item.

Using combo boxes is self-explanatory. You can select a number or enter a number yourself. However, one condition must be met. We do not use commas, so we round the measurements to one millimeter. However, you can use the arithmetic expression plus and minus.

For example, if you enter $154+10+2$ and press enter, the combo box will make it 166.

On the right of the combo boxes are buttons with a letter. These buttons start the measurement command. Select two entities and that rounded distance will be used in the combo box.

With a click on the PLACE Button, the routine that will place the configured part starts. A progress bar gives the progress and a description of the actions that are being carried out in the background during the creation of the part file.



Here are the tasks that are run in the background:

Creating a new part file.

Modeling the part.

Filling in the i-Properties.

The steel part is given the material "Steel, Mild" with the appearance "Semi-Polished".

The file is written on the project in the subdirectory **Steel**, subdirectory **according to the item**. If this directory does not exist, it is created. The filename is given the prefix "Name of the item_" followed by a unique number and the extension: ipt.

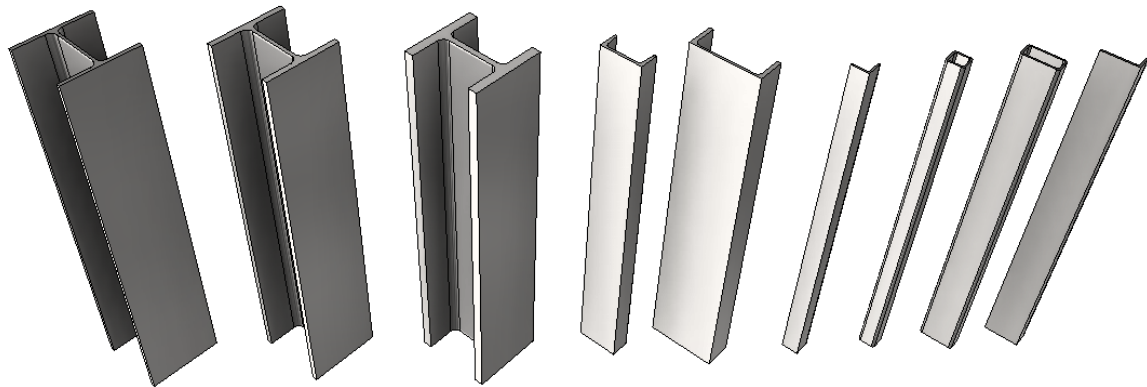
After the file has been saved and depending on the item, you will be asked to indicate a surface to attach the new part to. You may then be asked to connect the part to another surface. After you have designated this face, the part is attached to this face with one constraint.

The routine is terminated, and the window of eSteel is opened again, ready to install a new part. You must perform the constraints in the other two directions manually.

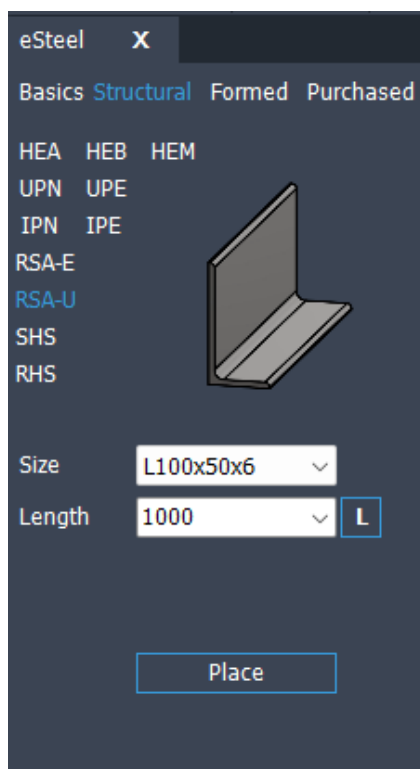
If you don't want to connect the part to a face, you can also continue with an "ENTER" click. Autodesk Inventor will then execute the regular "Insert" command with the new part.

Please note that after the first part is placed, the insert command will insert the same part again, and it has the same filename. If one of these two parts are modified with a regular Inventor command, the other part will change also.

5. Inserting Structural parts.



Click on the button of the eSteel panel to start installing a steel part. The window to configure the part will open at the top left corner of graphics display.



At the very top next to the title is a cross. If you move the cursor over it, it will turn red. With a click on the cross you can close the window.

You can also close the window by clicking on the escape key.

Below the title are four labels: Basics, Structural, Formed and Purchased.

If the Label Structural is selected the following vertical selection of items will appear. HEA, HEA, HEM, UPN, UPE, IPN, IPE, RSA-E, RSA-U, SHS and RHS.

In this case the RSA-U (Rolled steel angle – Unequal) is selected.

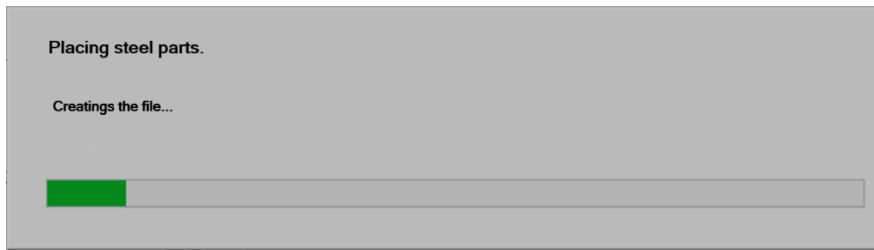
If you hover over the labels, the items will be preselected, and the picture will be shown of the preselected item.

If you click on the label, the label text will change to blue the picture will be of the new item and the combo boxes will be modified according to the new item.

In the combo boxes we do not use commas, so we round the measurements to one millimeter. However, you can use the arithmetic expression plus and minus.

On the right of the combo boxes are buttons with a letter. These buttons start the measurement command. Select two entities and that rounded distance will be used in the combo box.

With a click on the PLACE Button, the routine that will place the configured part starts. A progress bar gives progress and a description of the actions that are being carried out in the background during the creation of the part file.



Here are the tasks that are run in the background:

Retrieving the file from the content center.

Filling in the i-Properties.

The structural part is given the material "Steel, Mild" with the appearance "Semi-Polished".

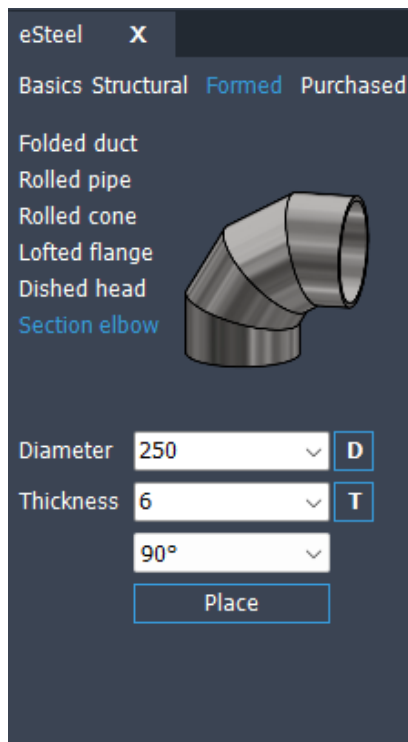
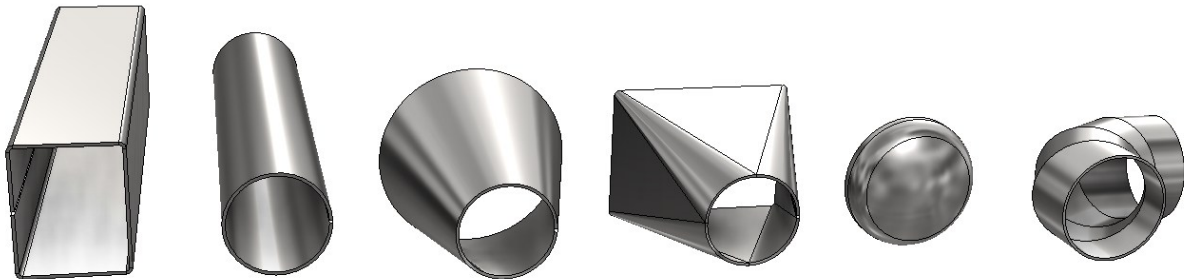
The file is written on the project in the subdirectory **Steel**, subdirectory **according to the item**. If this directory does not exist, it is created. The filename is given the prefix "Name of the item_" followed by a unique number and the extension: ipt.

The structural will always be inserted with the regular "Insert" command of inventor.

Please note that after the first part is placed, the insert command will insert the same part again, and it has the same filename. If one of these two parts are modified with a regular Inventor command, the other part will change also.

If you want the same structural and modify it later, you better place a new structural with the same values in the combo boxes so the new structural is unique.

6. Insert Formed parts.



Below the title are four labels: Basics, Structural, Formed and Purchased.

If the Label Formed is selected the following vertical selection of items will appear. Folded duct, Rolled pipe, rolled cone, lofted flange, Dished head and Section elbow

In this case the Section elbow is selected.

If you hover over the labels, the items will be preselected, and the picture will be shown of the preselected item.

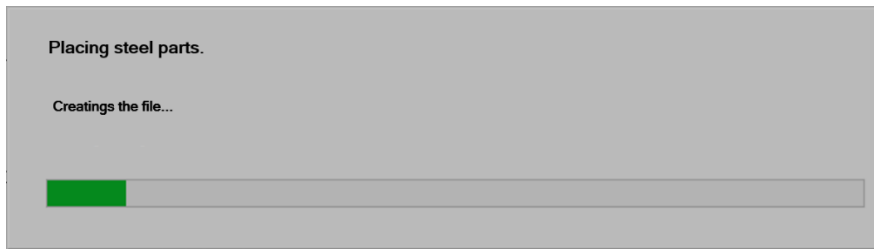
If you click on the label, the label text will change to blue the picture will be of the new item and the combo boxes will be modified according to the new item.

In the combo boxes we do not use commas, so we round the measurements to one millimeter. However, you can use the arithmetic expression plus and minus.

On the right of the combo boxes are buttons with a letter. These buttons start the measurement command. Select two entities and that rounded distance will be used in the combo box.

Depending on the item there could be more combo boxes, in the case of the section elbow you can choose the angle of the elbow from a list.

With a click on the PLACE Button, the routine that will place the configured part starts. A progress bar gives progress and a description of the actions that are being carried out in the background during the creation of the part file.



Here are the tasks that are run in the background:

Creating a new part file.

Modeling the part.

Filling in the i-Properties.

The formed part is given the material "Steel, Mild" with the appearance "Semi-Polished".

The file is written on the project in the subdirectory **Steel**, subdirectory **according to the item**. If this directory does not exist, it is created. The filename is given the prefix "Name of the item_" followed by a unique number and the extension: ipt.

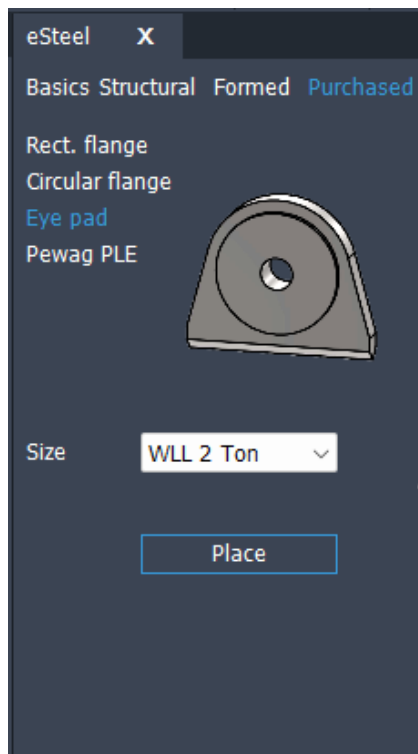
After the file has been saved and depending on the item, you will be asked to indicate a surface to attach the new part to. You may then be asked to connect the part to another surface. After you have designated this face, the part is attached to this face with one constraint.

The routine is terminated, and the window of eSteel is opened again, ready to install a new part. You must perform the constraints in the other two directions manually.

If you don't want to connect the part to a face, you can also continue with an "ENTER" click. Autodesk Inventor will then execute the regular "Insert" command with the new part.

Please note that after the first part is placed, the insert command will insert the same part again, and it has the same filename. If one of these two parts are modified with a regular Inventor command, the other part will change also.

7. Insert Purchased parts.



Below the title are four labels: Basics, Structural, Formed and Purchased.

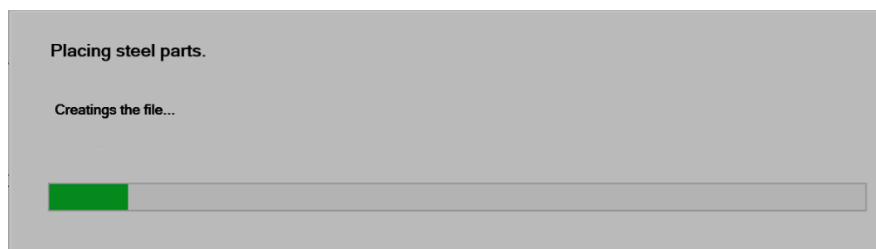
If the Label Purchased is selected the following vertical selection of items will appear. Rect. flange, Circular flange, Eye pad and Pewag PLE.

In this case the Eye pad is selected.

If you hover over the labels, the items will be preselected, and the picture will be shown of the preselected item.

If you click on the label, the label text will change to blue the picture will be of the new item and the combo boxes will be modified according to the new item.

With a click on the PLACE Button, the routine that will place the configured part starts. A progress bar gives progress and a description of the actions that are being carried out in the background during the creation of the part file.



Here are the tasks that are run in the background:

- Creating a new part file.

- Modeling the part.

- Filling in the i-Properties.

The formed part is given the material "Steel, Mild" with the appearance "Semi-Polished".

The file is written on the project in the subdirectory **Steel**, subdirectory **according to the item**. If this directory does not exist, it is created. The filename is given the prefix "Name of the item_" followed by a unique number and the extension: ipt.

After the file has been saved and depending on the item, you will be asked to indicate a surface to attach the new part to. You may then be asked to connect the part to another surface. After you have designated this face, the part is attached to this face with one constraint.

The routine is terminated, and the window of eSteel is opened again, ready to install a new part. You must perform the constraints in the other two directions manually.

If you don't want to connect the part to a face, you can also continue with an "ENTER" click. Autodesk Inventor will then execute the regular "Insert" command with the new part.

Please note that after the first part is placed, the insert command will insert the same part again, and it has the same filename. If one of these two parts are modified with a regular Inventor command, the other part will change also.

8. Finally.

The use of the application allows you to create a whole series of files in a short time and insert them into an assembly. As a result, Inventor itself can sometimes have problems with its RAM. It is advisable to save regularly.