Cadimage PLM
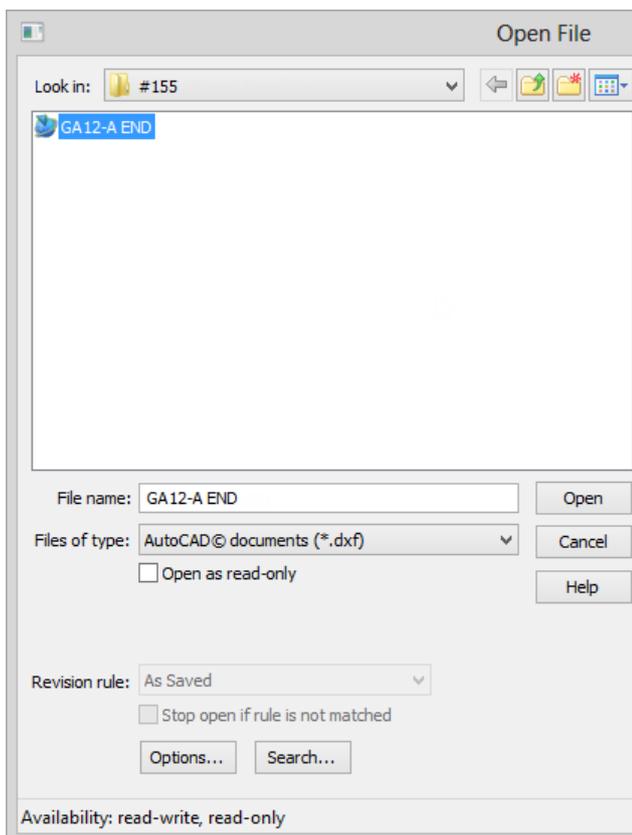product lifecycle management solutions

Tech Tip

Working with DWG and DXF Files
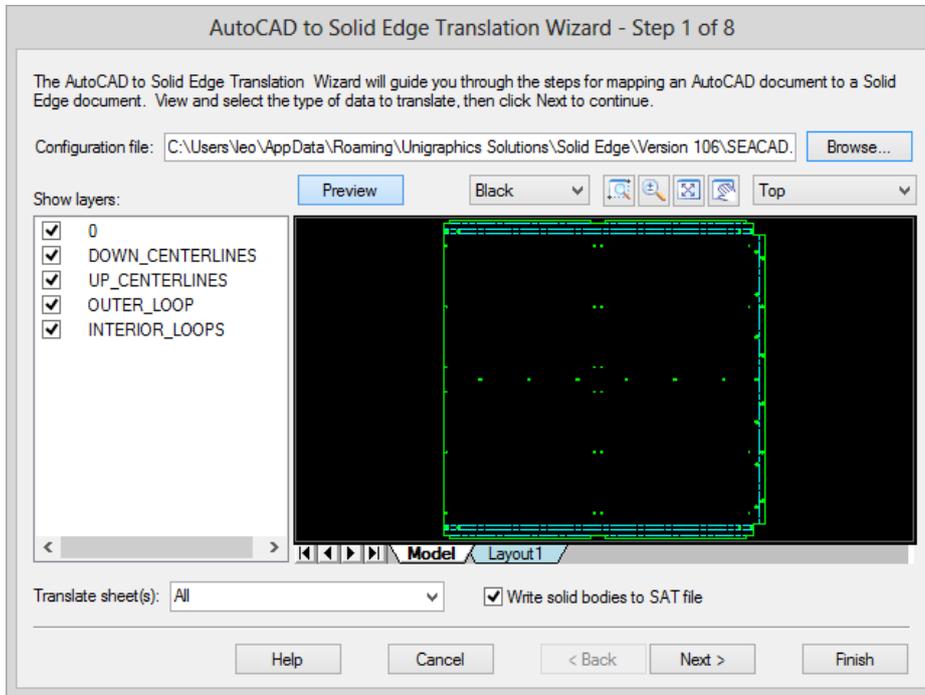
Working with DWG and DXF Files

When opening or exporting files (including the Save as Flat option from the sheet metal environment) to the DWG or DXF format, you need to ensure that the settings are set up correctly.

By default, Solid Edge uses the configuration file SEACAD.ini which is stored in the \Program Files\Solid Edge ST6\Program directory. The configuration file is a text file, that can be opened in Notepad, and holds all of your settings. An important fact to point out is that the default units held in this file is set to inches. You will therefore notice that if you open a metric dwg/dxf file, the sizes will be out by a factor of 25.4. The configuration file allows you to not only define the units in use, but it also allows you to map line and text styles.

Apart from modifying this file in Notepad, the only place to change these settings is via the Options button on the File Open Dialog box. For the button to become active, you need to change the "Files of type" box to be either dwg or dxf and then find a file of this type. When you single click on the file, the Options button will become active.
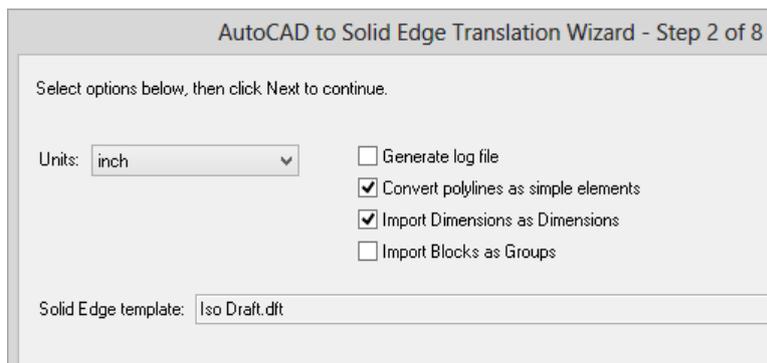

Once you have selected the Options button, Solid Edge launches the AutoCAD to Solid Edge Translation wizard. The first step displays which configuration file is being used and, using the Preview button, allows you to see what is stored in the file. There are some basic tools to display/hide layers, change the background colour and pan and zoom around the preview window. Note that you can select the Finish button at any stage to skip past steps not required.
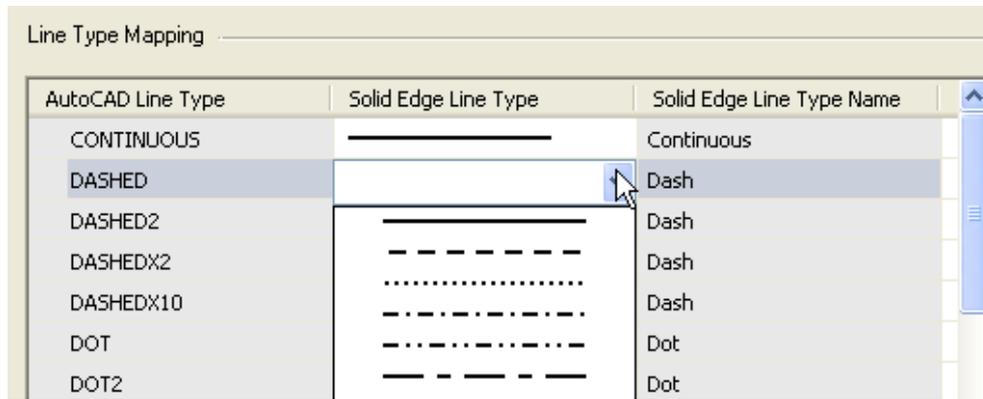
Step 2 is defines what units it will read and write. As stated earlier, the default is set to inches. The other options in this stage allow you to define how the AutoCAD lines, dimensions and blocks are converted. You can also specify what is the default template used when opening these files.

Step 3 of the process defines the sheet sizes used and how you wish to handle the AutoCAD model views.
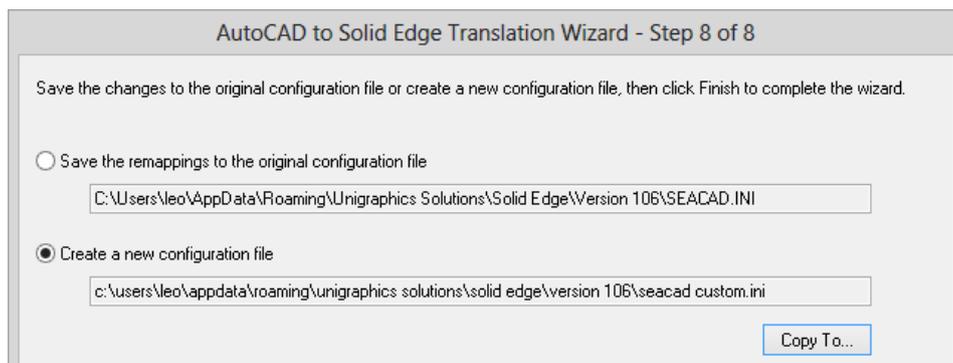

Step 4 allows you to specify how autoCAD linestyles are mapped to the available Solid Edge linestyles.

Step 5 allows you to map AutoCAD line colours to line widths, step 6 maps fonts from one product to the other and step 7 does the same with the hatching styles.

The final stage allows you to save your settings to a new file (the delivered file is locked out from modification.

Click the "Create a new configuration file" option and the click on the copy to button to define the name of the new file.

Once you have done this once, this configuration file will be used for all import and export of dwg and dxf files.

Just remember to redo it when you load a new version of Solid Edge!