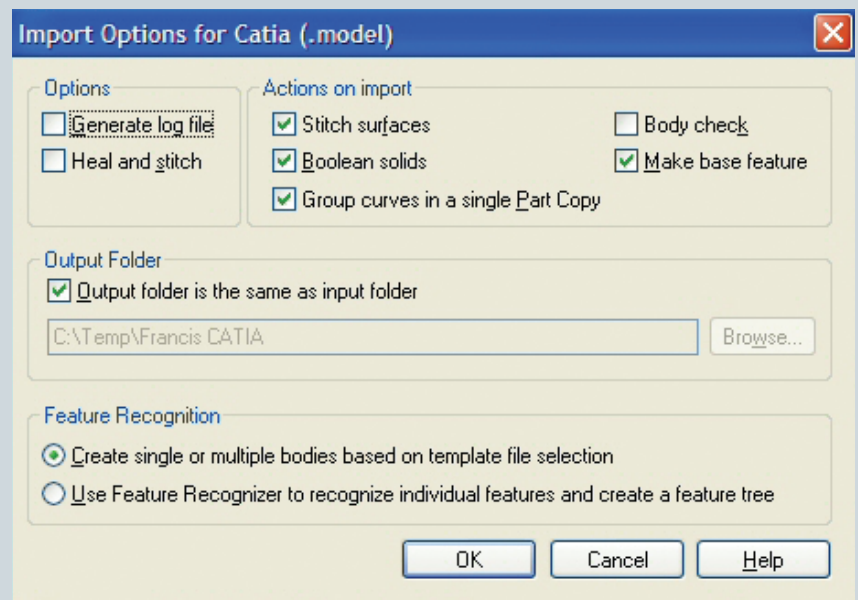# Solid Edge/Catia translator

Benefits
- Translate Catia data in an efficient and effective manner
- Work with suppliers or customers in a more cohesive manner
- Create better designs, enhance communications, increase customer satisfaction and deliver significant cost savings

Features
- Users can easily leverage the translator's bidirectional capabilities when handling Catia files
- The heal and sew option allows you to "heal" free surfaces and solids when importing them into Solid Edge
- Solid Edge automatically specifies a stitch tolerance range, so you are not required to specify a stitch range or tolerance
- The Solid Edge/Catia translator is a separately purchasable item available in both node locked and floating licenses
- The batch processor provides the ability to import one or more Catia .model files

Summary
Solid Edge®/Catia translator software is a powerful migration tool for Solid Edge users who need to reference Catia data from their customers and/or suppliers in their product designs. By translating Catia data from and to Solid Edge, companies can expect an increase in design communication, quality and productivity, as well as significant cost savings. Whether it's designing tooling or components for an assembly, it no longer matters if customers or suppliers are using Catia data. Solid Edge provides the right tools to help you stay ahead of the competition and provide data in a timely, cost effective manner.

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Solid Edge/Catia translator

The Solid Edge/Catia translator provides Solid Edge users with the ability to translate Catia data in an efficient and effective manner. Importing or exporting Catia model files with the Solid Edge/Catia translator allows design engineers to work with suppliers or customers in a more cohesive manner. With the Catia translator, users can design parts or assemblies in Solid Edge and then translate data into the required Catia format – or read in Catia data and continue designs in Solid Edge. The benefits of using the Solid Edge/Catia translator are clear – better designs, enhanced communication and an increase in customer satisfaction.

The Solid Edge/Catia translator provides bi-directional translation capabilities for Catia files, extending Solid Edge translation capabilities for opening foreign data to help you in markets where Catia translation is necessary.

The translator is capable of reading Catia 4.1.9 and 4.2.X (up to 4.2.4) and will write Catia 4.1.9. A separate module is also available for reading and writing Catia 5 files. Both translators support single part files, or multiple bodies can be opened into a Solid Edge assembly.

Templates supported in Solid Edge are .asm, .prt, .psm. Data can have several options applied together, such as stitch surfaces and Boolean solids. Solids can also be feature recognized if Feature Recognizer also has been purchased. The Solid Edge/Catia translator is a separately purchasable item available in both node locked and floating licenses.

This batch processor provides the ability to import one or more Catia .model files, or export one or more x_t, part or sheetmetal files to .model format.

On Import, the Solid Edge/Catia translator has an option to:
- *Heal and stitch* – This option specifies that you want to heal and stitch the free surfaces to create a solid body. Solid Edge automatically specifies a stitch tolerance range, so you are not required to specify a range.

  This option allows you to "heal" free surfaces and solids when importing them into Solid Edge. Healing involves preparing the free surfaces for stitching and the cleaning of solid bodies. Solid Edge cleans the faces to resolve underlying problems such as self-intersection, multiple intersections or edges equal to or smaller than the minimum stitching tolerance. Once the faces are cleaned, Solid Edge identifies and removes sheets that are invalid due to bad trimming curves, or sheets that result in slivers.

- *Stitch surfaces* – The Solid Edge/Catia translator reads and translates surface data from the Catia model file. When stitching is enabled (default setting) all surfaces will be stitched. Solid Edge automatically specifies a stitch tolerance range, so you are not required to specify a range. If the stitching operation creates a valid volume, the volume is converted to a solid.

- *Boolean solids* – This option specifies that you want to Boolean all solid bodies together to form a solid or a disjoint solid body and then insert these bodies into Edge Bar as a part copy. This option is on by default. If the option is off, all solid bodies are added as individual part copies to Edge Bar.

When exporting data from Solid Edge to Catia, you can filter what type of data you want to export:

Export solids (constructions) Specifies that you want to export any construction geometry that exists as solid bodies.

Export sheets (constructions) Specifies that you want to export any construction geometry that exists as sheet bodies.

Export wires (constructions) and general Specifies that you want to export any construction geometry that exists as wire bodies.

Export displayed only Specifies that you want to export only the active model and construction geometry that is visible in the part or assembly.

Contact
Siemens PLM Software
Americas   800 807 2200
Europe   44 (0) 1202 243455
Asia-Pacific 852 2230 3308

www.siemens.com/velocity