SIEMENS

Solid Edge/CATIA V4 translator

Enabling more effective collaboration with suppliers and customers

Benefits
- Translate CATIA V4 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
- Easily leverage the translator's bidirectional capabilities when handling CATIA V4 files
- The heal-and-sew option allows you to heal free surfaces and solids when importing them into Solid Edge

Summary
Solid Edge® software includes the Solid Edge/CATIA V4 translator, which is a powerful migration tool that enables Solid Edge users to reference CATIA V4 product design data that is provided by customers and suppliers. By translating CATIA V4 data from and to Solid Edge, companies can expect improved 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 V4 data. Solid Edge provides the right tools to help you stay

ahead of the competition and provide data in a timely, cost-effective manner.

The Solid Edge/CATIA V4 translator provides Solid Edge users with the ability to translate CATIA V4 data in an efficient and effective manner. Importing or exporting CATIA V4 model files with the Solid Edge/CATIA V4 translator allows design engineers to work with suppliers or customers in a more cohesive manner. With the Solid Edge/CATIA V4 translator, users can design parts or assemblies in Solid Edge and then translate data into the required CATIA V4 format, or read in CATIA V4 data and continue designs in Solid Edge. By using the Solid Edge/CATIA V4 translator, companies achieve better designs, enhanced communication and increased customer satisfaction.

The Solid Edge/CATIA V4 translator provides bidirectional translation capabilities for CATIA V4 files, extending Solid Edge translation capabilities for opening foreign data to help you in markets where CATIA V4 translation is necessary.

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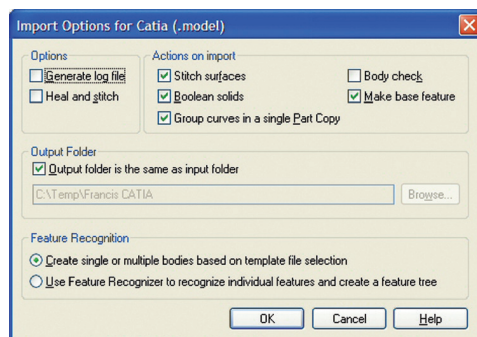
Solid Edge/CATIA V4 translator

The translator is capable of reading CATIA V4.1.9 to V4.2.4 and will write CATIA V4.1.9. A separate module is also available for reading and writing CATIA V4 V5 files. Both translators support single part files, or multiple bodies that can be opened into a Solid Edge assembly.

Templates supported in Solid Edge are .asm, .prt and .psm. Data can have several options applied together, such as stitch surfaces and Boolean solids. The Solid Edge/CATIA V4 translator can be purchased separately in both node-locked and floating licenses.

This batch processor provides the ability to import one or more CATIA V4 .model files, or export one or more x_t, part or sheet metal files to .model format.

On import, the Solid Edge/CATIA translator provides the 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 can be used to clean the faces to resolve underlying problems, such as self-intersection, multiple intersections or edges equal to or smaller than the minimum stitching tolerance.

- *Stitch surfaces* – The Solid Edge/CATIA V4 translator reads and translates surface data from the CATIA V4 model file. When stitching is enabled, which is the 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 the feature tree as a part copy. This option is the default setting. If the option is off, all solid bodies are added as individual part copies to the feature tree.

When exporting data from Solid Edge to CATIA V4, you can filter the 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 an assembly.

Features *continued*
- Automatically specifies a stitch tolerance range

- Can be purchased separately in both node-locked and floating licenses

- The batch processor provides the ability to import one or more CATIA V4 .model files