FORMATWORKS
PROFESSIONAL

Proven in Automotive and Aerospace Industry for optimal work with imported CAD models in SolidWorks™

Preview, Select, Translate

FormatWorks preview is especially useful for importing large models or complex assemblies. The imported part is first opened in a graphical preview window providing zoom, rotate and pan operations on the model. A project tree shows the entire model structure with direct access to each model entity. Select the model entities you want by picking them from the 3D model, using the project tree, or color based filtering. The selected parts are directly translated into SolidWorks leaving the rest behind. This allows you to translate only the information needed for your work, saving significant time and computer resources. Your models are light and easily manageable in SolidWorks.
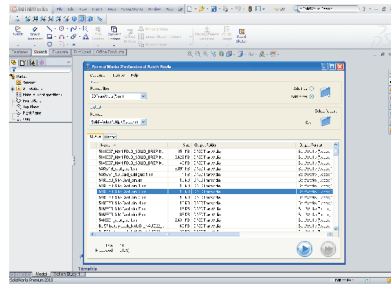
Batch Mode Control Window

Assembly 1.5GB. Preview

Select

Copy into to separate document

Part in SolidWorks

Batch Translation

FormatWorks batch translation is designed to simplify and automate translation of large assemblies and multiple CAD data sets (e.g. large CATIA models). Batch translation is performed automatically. It is controlled from a single window allowing you to manage the translation process, monitor the progress, and analyze results. Batch translation is fully autonomous and will automatically recover in case of an unexpected error or crash. A report shows the translation results classified into the following categories: solid, non-solid and model with errors (specifying amount and type of error).

Feature Translation

Complex CAD models imported through FormatWorks can be simplified into entities that are recognized as SolidWorks features. For example, a complex NURBS representation can be automatically converted to an analytical definition such as plane, cylinder, sphere, cone or SOR (Surface of Revolution). This significantly reduces the size of the original CAD model and results in a much higher success rate for automatic feature recognition. FormatWorks gives you the possibility to recognize more features automatically to make imported CAD models fully editable in SolidWorks.
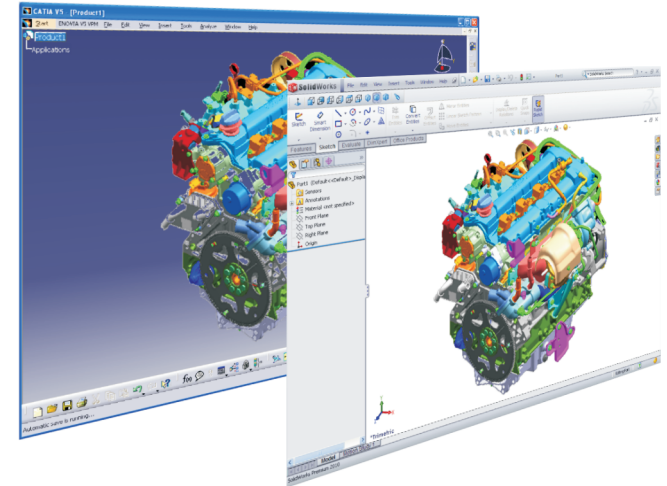
Auxiliary Geometry and LCS

FormatWorks gives you the possibility to import from supported CAD formats an Auxiliary Geometry such as, 2Dsketches, featurebased sketches and Local Coordinate Systems to make a fully editable model in SolidWorks.

CATIA V6
CATIA V5
CATIA V4
PRO/E
UGS/NX
INVENTOR
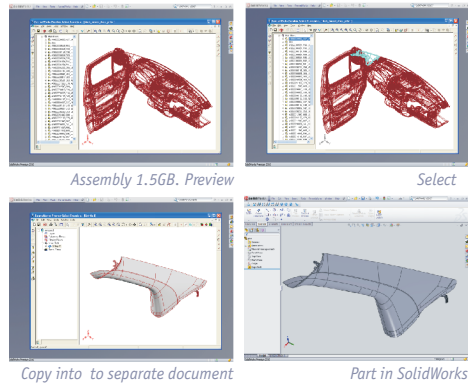SolidEdge
ACIS
STEP
IGES

Data Translation Add-In for SolidWorks™

FormatWorks extends the SolidWorks data translation capability to support native CATIA V4 and V5 data as well as other formats. FormatWorks provides automatic, accurate and reliable bi-directional translation for SolidWorks.
Automatic repair and healing ensures that imported CAD models meet the criteria set by SolidWorks to form valid solids. Repair and healing are always performed within the model tolerance to ensure original model integrity and preventing model deformation.

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FormatWorks

FormatWorks solves frustrating data incompatibility problems occurring during import of 3D models into SolidWorks such as: 1000+ surfaces that do not knit, model deformation, surface explosion. It seamlessly imports native data formats not supported in SolidWorks and performs necessary repairs to form valid and editable solid models.

Data Formats

|	Import	Export
CATIAV5 *)	+	+
CATIAV4	+	+
ProE Wildfire*)	+	-
ProE Creo *)	+	-
UGS, NX*)	+	-
Inventor 2014	+	-
IGES	+	+
SolidEdge	+	-
STEP A203/A214/242*)	+	+
ACIS	+	+
VDA-FS	+	+
3DPDF	-	+

*) Also Product Manufacturing Information

Automatic Repair

Automatic repair & healing is an integral part of the FormatWorks data translation process. FormatWorks identifies over 70 different errors and repairs them automatically using specialized proprietary algorithms. Geometry and topology definitions are verified and corrected so that the imported model can form a watertight solid in SolidWorks. This process is controlled and always performed within the model tolerance to ensure that the model is not deformed or altered from the original definition.
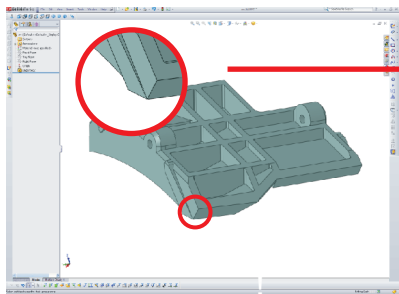
Direct import into SolidWorks
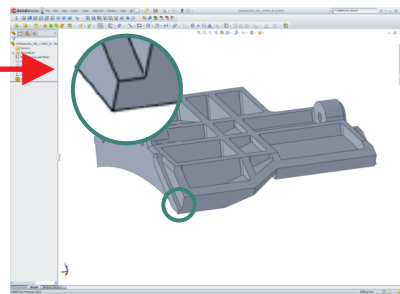
Import over FormatWorks

FormatWorks saves time and maximizes your productivity

Manual Repair

The manual repair process has never been simpler. FormatWorks provides an optimal workflow to correct errors that cannot be repaired automatically. The model is first analyzed and errors are classified into different types and grouped in the error tree.

Errors categories:
- Gaps
- Ledges
- Overlaps
- T-connections
- Open contours

FormatWorks gives you tools to isolate faulty entities and neighbors in a separate document. All manual operations are intuitive and easy to use. FormatWorks special functions such as: surface recreation, surface creation, surface merge, surface trimming, and surface splitting extend the SolidWorks tools allowing you to repair very complex geometry or topology errors. After the repair is completed, FormatWorks inserts the repaired entities back into the original model and automatically updates it.
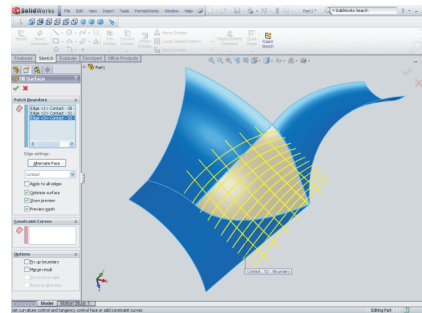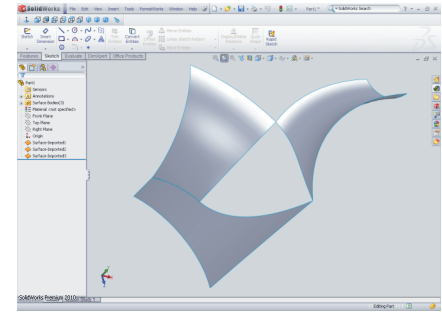
Manual Repair Workflow

① Error Classification
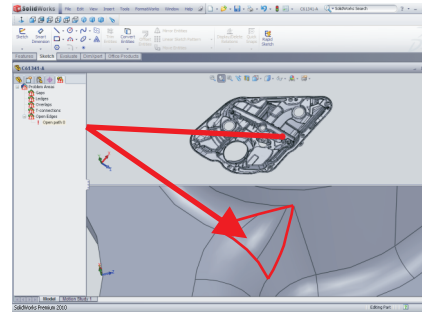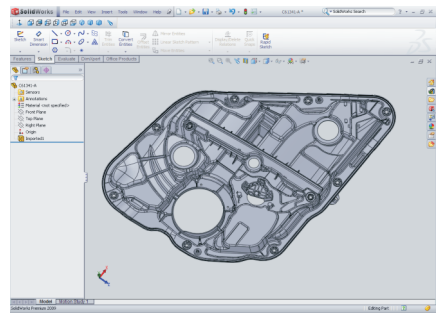

Error classified as an open contour

② Export to separate document

Error extracted with neighbors into a separate document

③ Repair

Repair

④ Update of original geometry

Original model updated with the repaired part

MBD (Model Based Definition) Support

The PMI (Product Manufacturing Information) stored together with 3D models native CAD models (CATIA V5, UGX and Pro/E) is now easily accessible for SolidWorks users. FormatWorks will import the 3D model together with all related annotations and dimensions providing essential manufacturing information about tolerances, finishing surface quality, and other manufacturing specific parameters. The PMI information is displayed directly on the imported into SolidWorks 3D model and can be stored together with the 3D CAD data in native SolidWorks file format.
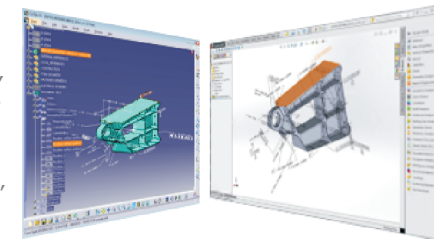
PMI from CATIA V5 native file displayed directly on imported SolidWorks model