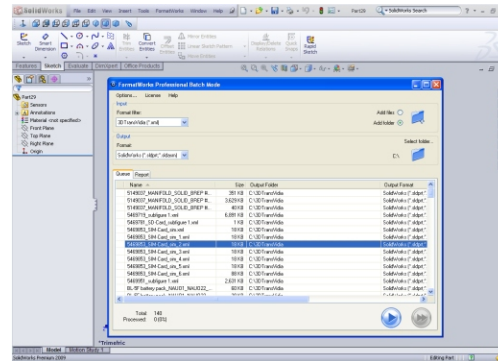


Preview, Select, Translate

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 base 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

Batch Translation

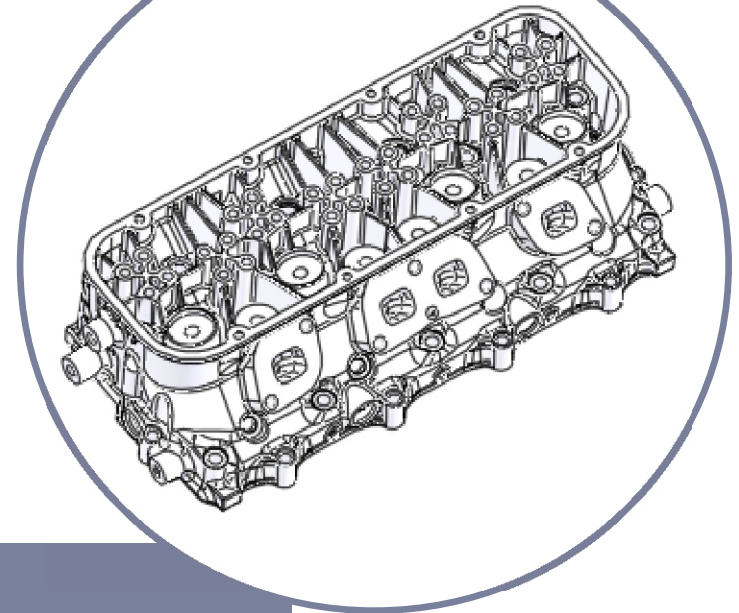
FormatWorks batch translation is designed to simplify and automate translation of large assemblies and multiple CAD data sets (e.g. large CATIA models). Batch translation is performed automatically. It is controlled from a single window allowing you to manage the translation process, monitor the progress, 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).

FORMATWORKS

PROFESSIONAL 2009

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

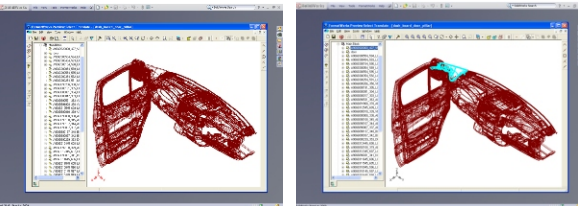
CATIA V5
CATIA V4
PRO/E
UGS
INVENTOR
ACIS
STEP
IGES



Data Translation Add-In for Optimal Engineering Workflows in SolidWorks™

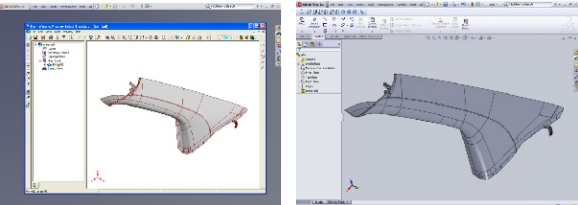
FormatWorks extends the SolidWorks data translation capability to support native CATIA V4 and V5 data as well as other formats. FormatWorks provides automatic, accurate and reliable bi-directional translation for SolidWorks.

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 while maintaining original model integrity and preventing model deformation.



Assembly 1.5GB. Preview

Select



Copy into to separate document

Part in SolidWorks

Feature Translation

Complex CAD models imported through FormatWorks can be simplified into entities that are recognizable 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 original CAD model and results in 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.

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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.

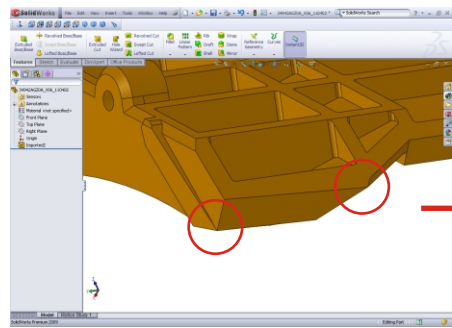
Data Formats

| | Import | Export |
|---------------------|--------|--------|
| CATIAV5 up to R19* | + | + |
| CATIAV4 up to 4.2.4 | + | + |
| ProE Wildfire 4.0* | + | - |
| Unigraphics NX5.0* | + | - |
| Inventor up to V12 | + | - |
| IGES 6.0 | + | + |
| STEP A203/A214 | + | + |
| ACIS V.16 | + | + |
| VDA-FS | + | + |

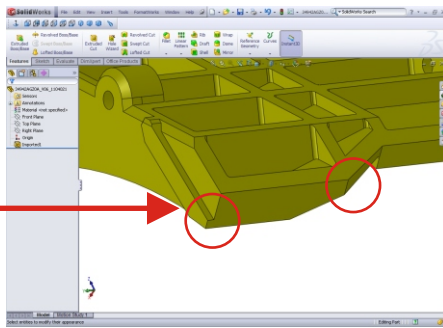
*) Also Product Manufacturing Information

Automatic Repair

Automatic repair & healing is a unique and 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 maximize your productivity

Manual Repair

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.

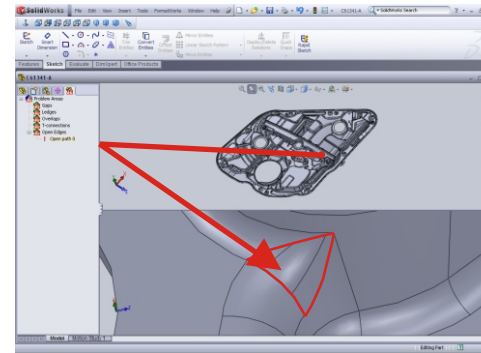
Errors categories:

- 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.

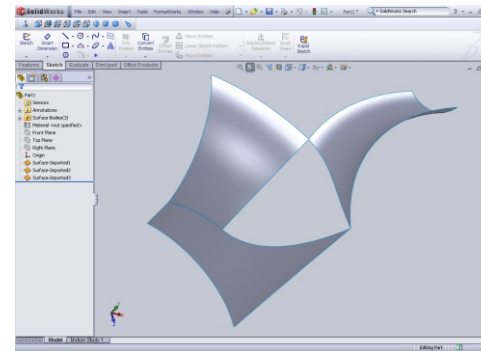
Manual Repair Workflow

1 Error Classification



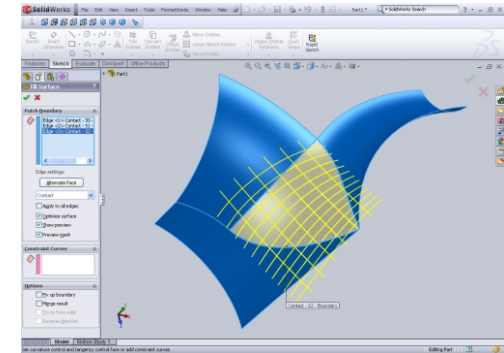
Error classified as an open contour

2 Export to separate document



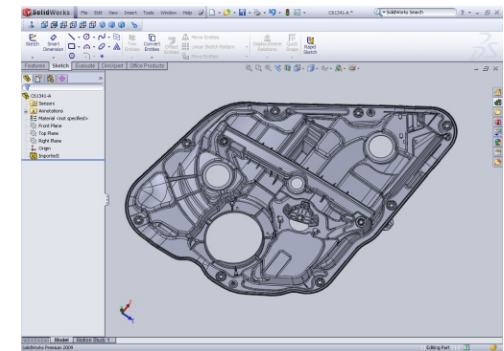
Error extracted with neighbors into a separate document

3 Repair



Repair

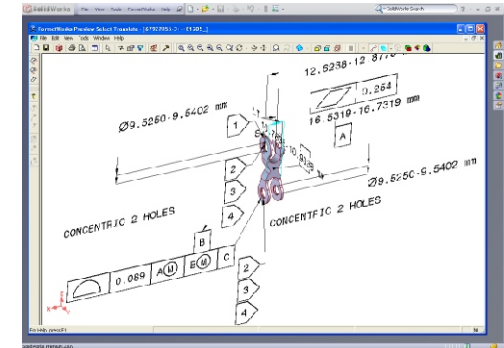
4 Update of original geometry



Original model updated with the repaired part

Product Manufacturing Information

The Product Manufacturing Information (PMI) stored with a 3D model is now easily accessible from CATIA V5, UGX and Pro/E native files. PMI text and dimensions provide essential manufacturing information about tolerances, surface quality, tolerance for parallel walls or axis, etc. PMI information is displayed on the 3D model and accessible from the project tree. You can activate/hide PMI layers, change the font type, alter the letter size, edit dimensions, and print the PMI information for documentation purposes.



PMI from CATIA V5 native file