

“Working with native CATIA models in SolidWorks”

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Major automotive and aerospace companies use CATIA as main stream MCAD system. This market trend creates new demands on the supplier and sub-contractor chain working with those companies. The general trend is to exchange more and more data using native data formats to maintain data consistency and avoid interoperability problems. The neutral formats such IGES and STEP are alternatives used only in cases where data confidentiality becomes the most important issue (e.g. when working with off-shore engineering and manufacturing companies). Using native data formats reduces engineering costs and simplifies data exchange for large manufactures, but makes life difficult for their suppliers. To comply with those new demands suppliers need to invest in CATIA seats or look for alternative solutions making their MCAD system compatible with native CATIA data formats.

The suppliers using SolidWorks will have to find a solution to comply with these new requirements.

Getting native CATIA models in SolidWorks is not a simple data translation issue. The process involves much more than data format conversion. Both CATIA V4 and V5 use different kernels and modelling concepts than SolidWorks. CATIA V5 feature based modelling is close to SolidWorks solid modelling concepts but kernels are different (SolidWorks uses Parasolid, CATIA V5 proprietary kernel technology).

What does it mean to a SolidWorks user?

- A CATIA solid is not necessary a solid in SolidWorks and vice versa.
- Gaps caused by different model tolerance prevent the formation of consistent topological objects.
- SolidWorks is unable to stitch or sew surfaces into skin or solid.
- Large and unworkable models.
- Surface explosion – SolidWorks is unable to get a 1-to-1 translation.
- Some SolidWorks operations such as scaling or thickening will be impossible to perform on the imported data.

The above list can be further extended, but most of these problems are familiar to SolidWorks users working with automotive and aerospace industry.

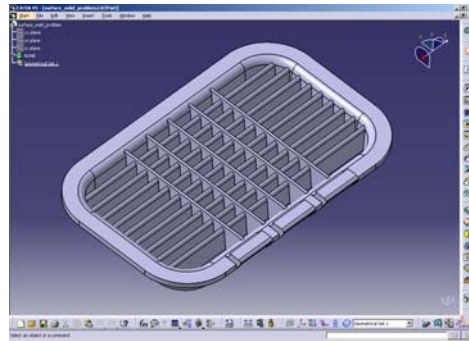
How does Capvidia efficiently deal with these problems?

Capvidia offers specialised data translation products addressing the above problems. Both FormatWorks (SolidWorks Certified Gold Product) and 3D TransVidia include advanced repair and healing capabilities that resolve geometric kernel incompatibilities between different CAD systems. The repair process is tuned for SolidWorks internal requirements to form topological solids with correct model tolerance, without gaps and without missing surfaces. The model correction process is automatic and performed within the original model tolerance without deforming the original model. In cases where the automatic repair fails due to problem complexity (decision can not be taken automatically without a risk to deform the model), extensive specialised manual repair functions are available to the user. The first step in the manual repair process is error classification (gaps, overlaps, T-connections, ledges, open contours, untrimmed surfaces, etc.). Different types of errors are visually indicated on the geometry. The problematic entity together with all the neighbours is automatically transferred into a separate document to simplify the repair process (very useful for large models). The user is prompted with hints (best practices) on how to repair the faulty entity.

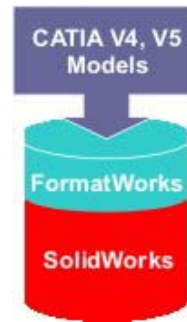
CATIA Parts

For CATIA parts FormatWorks is a fully integrated SolidWorks solution (Certified Gold Partner Product). The automatic import of native CATIA V4 and V5 data is implemented using native CATIA read/write libraries licensed directly from Dassault Systems. Working with original libraries brings many obvious advantages and secures support of the latest releases of CATIA. Both FormatWorks and 3D TransVidia support all CATIA V5 releases up to CATIA V5 R15.

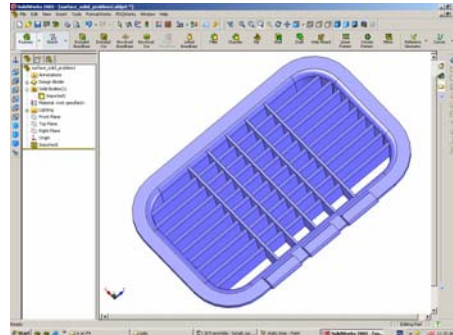
Using original libraries from the manufacturer is a big advantage, but still it does not solve all the data translation issues. The most complex problem is CAD kernels incompatibility. Bringing CATIA (based on proprietary kernel) part into SolidWorks (based on Parasolid kernel) requires applying special healing techniques. The objective of healing is to adopt the original model mathematical definition to the requirements of the receiving CAD system. The healing operation may involve change of an incompatible entity (e.g. NURBS entity is converted into a cylinder representation), but must not allow any change or deformations of the original model (volume and surface should remain the same). In the advanced data translation systems as FormatWorks, 3D TransVidia and others the healing process is implemented in the background and not visible to the user. The system requires from the user only the definition of the receiving CAD system (data format) to which the translation has to be performed.



Model in CATIA V5



FormatWorks SolidWorks Add-in



Model in SolidWorks

CATIA Assemblies

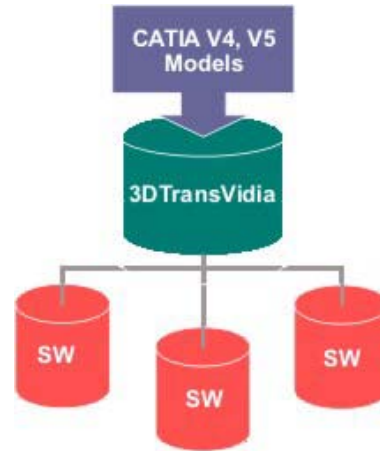
For CATIA assemblies the best solution available is 3D TransVidia to SolidWorks. This product is a stand alone application communicating with SolidWorks over an API. It can be configured to work with multiple SolidWorks seats connected over the local network (using local IP addresses). In this way a user can read a complex assembly into 3D TransVidia without locking any seats during the translation process. The assembly can be further divided into individual parts and send to different users. In this way the work can run in parallel using sub-sets of the imported model on different workstations. This capability is very important in applications related to tooling and fixture design, re-design of limited parts of the original model or design of new related elements using references from the original model, all being typical applications for subcontractors to the automotive and aerospace industry.

Exporting to CATIA from SolidWorks

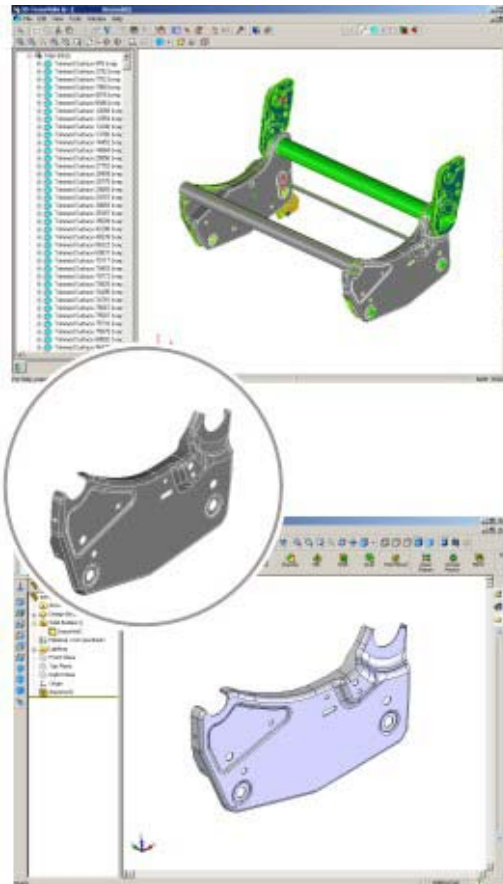
When the user decides to export a SolidWorks part to CATIA, FormatWorks/3D TransVidia will first verify the part compatibility with CATIA, then heal the model if required and only after that translate it into native CATIA format. In cases where the model definition is incompatible with the receiving system and automatic healing cannot be performed the user will receive a warning that the resulting translation may include errors.

Large Data Sets - Batch Translation

Both FormatWorks and 3D TransVidia have options for automatic batch translation. Batch translation is an interesting option for translating large amounts of legacy data. The process is fully automatic and can run overnight. The batch translation mode controls all steps of translation process. The results and potential problems are stored in a log file. Each translated model is read into SolidWorks for verification of the results and stored automatically in SolidWorks file format. If any problems are encountered the system will automatically recover after a user-defined time-out. SolidWorks will also be restarted in case of a crash. All problems are logged with a time stamp for further verification and control.



3DTransVidia configured with multiple SW seats



Selective data import from CATIA to SolidWorks

Conclusion

Working with CATIA V4 and V5 data in SolidWorks is not only possible, it is a welcome pleasure when FormatWorks is used. Interoperability between CATIA and SolidWorks requires specialised import tools that not only convert data formats but also resolve incompatibilities between CAD kernels. Capvidia offers an extensive product range and services helping you to deal with all data translation issues.

About the Author:

T. Luniewski is founder and managing director (CEO) of Capvidia. He holds M.Sc. degree from Royal Technical Collage (KTH) in Stockholm/Sweden and has over 20 years experience in development, sales and marketing of engineering software solutions for virtual prototype simulation, structural dynamics, FEM analysis, CFD and CAD data translation.