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Work with Legacy Data or Data from Other CAD Systems in Different Ways

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Learning Objectives

- Learn different possibilities for working with data from other systems
- Know how to address challenges associated with CAD conversion projects
- Get to know ITI's GoToINVENTOR solution for automated feature-based CAD consolidation

Description

You get nonnative Inventor data and you must work with it?

Or you've made the decision to standardize on Autodesk INVENTOR, but your current design data is in a different CAD format. What can you do now?

Learn how to work with data from other CAD Systems in Inventor using AnyCAD functionality or importing data to Inventor. Feature out how BREP data can be used downstream or can be edited. See how GoToInventor from ITI can support you in a feature-based converting process.

Speaker(s)

Manuela Zelinka – Autodesk GmbH Munich
Manuela Zelinka has a degree in mechanical engineering. She started her career at a SolidWorks reseller in 1997. She joined Autodesk in 2002 as a Technical Specialist in the Technical Sales Team in Munich. She has been in the CAD industry for over 20 years and brings in a lot of experience for her Presales role. Her focus is in Autodesk PD&M Collection.

Rene Bosiaki – ITI – International TechneGroup

Rene has been with ITI for over four years in the role of the Account Manager for European Business activities with key focus on the DACH region. Rene has extensive experience in supporting the sales process, including new business development and managing existing accounts across ITI's CAD interoperability software solution and services product line.
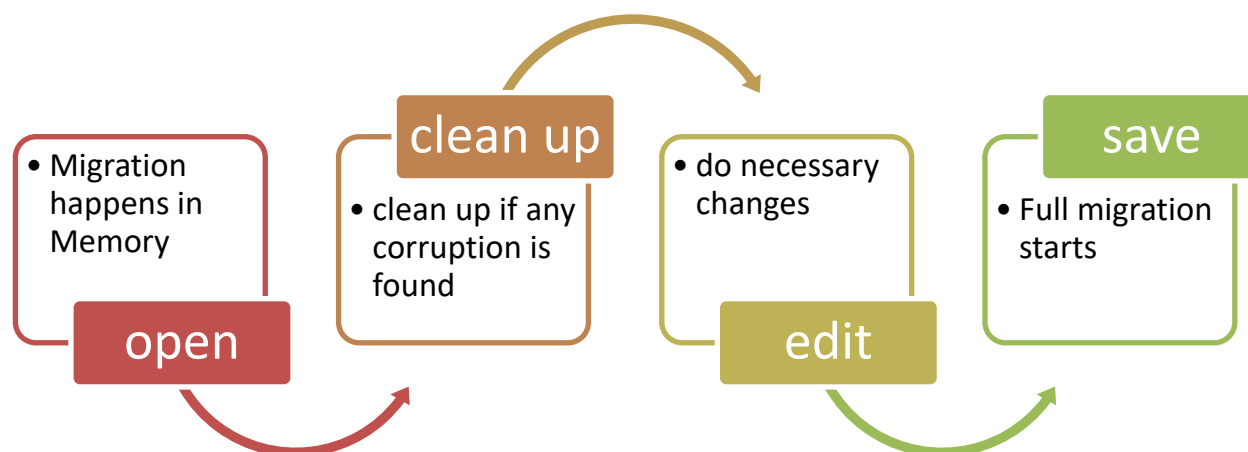
Working with legacy data or data from other systems

In every company there is legacy data available. This is either from the current system or from other systems and platforms.
Surely everyone wants to continue using this data. But what is the best way to do this?

Migration of the existing inventory data:
A migration of data from an old Inventor release was always necessary until a few years ago. Since release 11 we speak of a ZeroMigrationImpact for Inventor. This means that a migration is no longer mandatory. The data can simply be opened. The following process takes place when opening old data.

clean up
• clean up if any corruption is found

open
• Migration happens in Memory

edit
• do necessary changes

save
• Full migration starts

This avoids time-consuming migration of legacy data.
When using a 3rd party data management system, a migration is often necessary due to the technology used.

Data-migration
https://knowledge.autodesk.com/support/inventor/learn-explore/caas/CloudHelp/cloudhelp/2021/ENU/Inventor-Install/files/GUID-FB9B4422-2348-46FA-964B-7845D9D96018-htm.html?st=zero%20migration

Migration tips and tricks
https://knowledge.autodesk.com/support/inventor/learn-explore/caas/CloudHelp/cloudhelp/2021/ENU/Inventor-Install/files/GUID-BF6C7C26-F29E-4E4C-A18B-862EA4EEAD81-htm.html

Migration enhancements
https://knowledge.autodesk.com/support/inventor/learn-explore/caas/CloudHelp/cloudhelp/2021/ENU/Inventor-WhatsNew/files/GUID-7782AAA5-FD93-49A1-A2DE-F4977E7C83A3-htm.html

Task Scheduler

Autodesk also offers tools for migration. The Task Scheduler can be used to execute specific tasks in batch mode.

Task Scheduler is a separate application that performs automated tasks and acts as a batch processor in Autodesk Inventor and other applications.
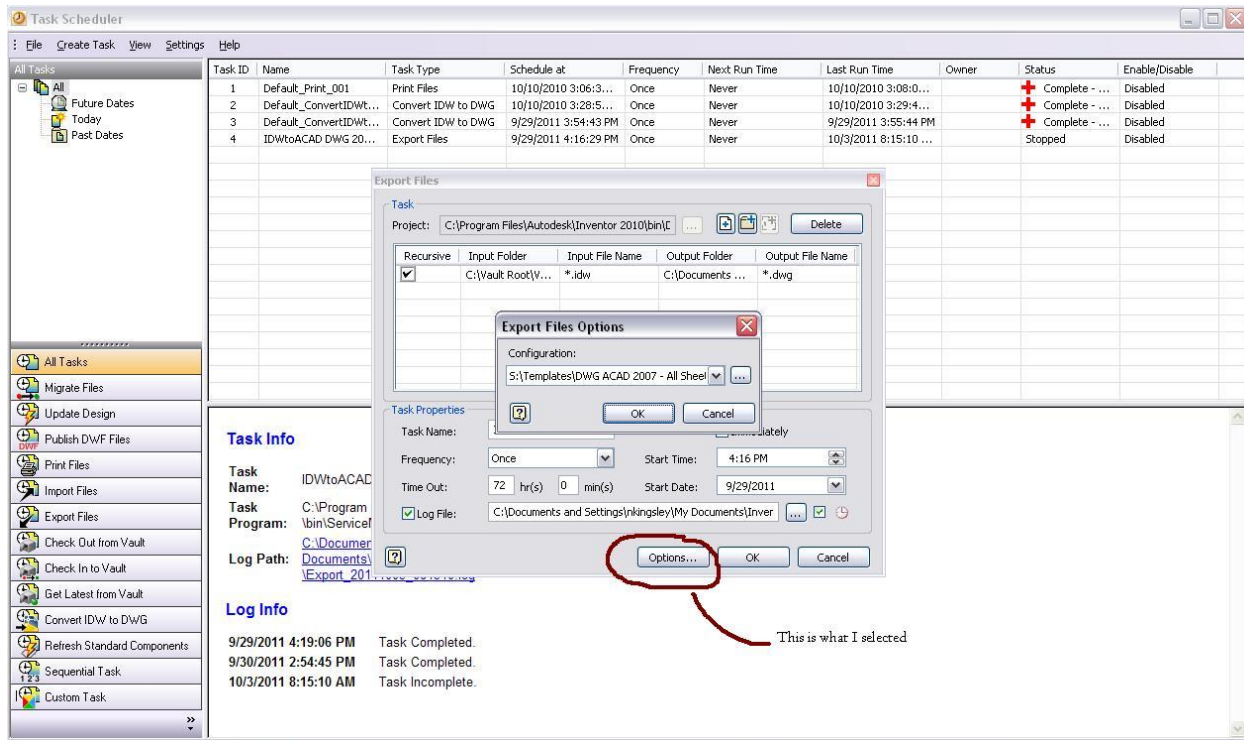
To open Task Scheduler, click Start ➤ All Programs ➤ Task Scheduler.

Use the Task Scheduler to organize and define one or more time-consuming tasks from different types of programs. Close the Task Scheduler window, and the tasks you scheduled run in the order and at the time that you specified.

Task Scheduler contains predefined task managers for executing common tasks, and a custom task manager for defining your own tasks. A Sequential Task manager is provided to set up multiple predefined and custom tasks. The Migrate Files task can migrate files directly from the Vault.

In addition to Custom and Sequential tasks, the predefined tasks include:

- Migrate Files
- Update Design
- Publish DWF Files
- Print Files
- Import and Export Files
- Check Into and Check out from Vault
- Get Latest Version from Vault
- Convert IDW to DWG
- Refresh Standard Components in Assemblies
- Shrinkwrap Assemblies
- Purge styles and materials from files

Task Scheduler
https://knowledge.autodesk.com/support/inventor/learn-explore/caas/CloudHelp/cloudhelp/2021/ENU/Inventor-Help/files/GUID-B5F1F5EC-8145-492D-9411-7E02AB519243-htm.html

https://knowledge.autodesk.com/support/inventor/learn-explore/caas/CloudHelp/cloudhelp/2021/ENU/Inventor-Help/files/GUID-95DE0A26-EAAC-49EB-A907-3F3BC29707A8-htm.html

Best Practise
https://knowledge.autodesk.com/support/inventor/learn-explore/caas/sfdcarticles/sfdcarticles/Inventor-best-practices-for-Migrating-to-a-new-release.html?st=task%20scheduler

Data exchange with other systems

Collaboration is currently a big keyword. Collaboration also takes place in the construction department, where data interoperability is a prerequisite.
This can be achieved by providing translators. In this way data can be transferred from system A to system B. This data can be exchanged in native formats or in neutral formats.
Both ways have their own charm. If you use a neutral format, you do not have to pay attention to the release status in the CAD system, everyone can read it and therefore an exchange is guaranteed.
Native data has its advantages. In addition to the geometry, meta data can also be included.

3D: Inventor AnyCAD

In Inventor there are further possibilities to work with native data. These data can not only be imported, but also allow referencing. The technology AnyCAD makes this possible. With AnyCAD, foreign 3D CAD data as well as 2D AutoCAD data can be referenced.
You can import the following files as an AnyCAD reference model: Alias, Revit, CATIA, DWG, PTC Wildfire, Solidworks, NX, STEP, Solid Edge, Fusion 360***, and Pro-E/Creo.
Note: To share data between Inventor and Revit, see About Using Revit Models.

If the design is changing, take control and improve performance by importing the design as a Reference Model. The Reference model option maintains a link to the selected file which enables you to monitor and update Inventor as the model changes. If changes are made to the file in its respective product after import, the Inventor file reflects the changes and downstream references update as well.

supported files for import and AnyCAD
https://knowledge.autodesk.com/support/inventor/learn-explore/caas/CloudHelp/cloudhelp/2021/ENU/Inventor-Help/files/GUID-AF41FA87-7588-4698-9C41-756A01EBE7F4-htm.html?st=translator%20for%20anycad

enhancements Translator 2021
https://knowledge.autodesk.com/support/inventor/learn-explore/caas/CloudHelp/cloudhelp/2021/ENU/Inventor-WhatsNew/files/GUID-AF3FF305-03D8-4C6B-BDF2-6744006BA01F-htm.html?v=2021&st=translator

AnyCAD for Inventor
AnyCAD for Inventor allows you to reference a part or assembly file that is one year newer than the version you are on.
https://knowledge.autodesk.com/support/inventor/learn-explore/caas/CloudHelp/cloudhelp/2021/ENU/Inventor-Help/files/GUID-CEB8F9D8-C6DB-4BBC-A034-8BFDC8385D0C-htm.html?st=anycad%20translaotr

AnyCAD FAQ
https://knowledge.autodesk.com/support/inventor/learn-explore/caas/CloudHelp/cloudhelp/2021/ENU/Inventor-Help/files/GUID-88E98285-7D43-436D-BCFF-BB49EDB44002-htm.html

AnyCAD technology can also be used for the neutral format Step. By referencing this external data can be integrated into the Inventor work process. However, there is a small difference when referencing step data. If the data is updated, an update must be run and the data is read again. This results in a new numbering of the surface IDs. This can lead to the fact that the assembly mates already made in an assembly become faulty.
In the case of neutral formats, the surfaces are not renumbered again. Therefore these data are more stable during the update.

Besides the referencing (Anycad) data can still be imported. This means that new Inventor Files were created for the data sets.
The data is read as BREP body and has no features and parameters. You can, however, change the geometry at any time by adding further features (fillet, chamfer, extrusion...) or by using the feature **direct edit** to do changes.

Direct Edit allows you to adjust size, shape, and/or location of model features by directly manipulating the geometry.

https://knowledge.autodesk.com/support/inventor/learn-explore/caas/CloudHelp/cloudhelp/2021/ENU/Inventor-Help/files/GUID-96463E7B-A7E1-4DA5-B579-C4F346015AD7-htm.html?st=direct%20edit

https://www.youtube.com/watch?v=G8lUVstccrs

3D: Inventor Feature Recognition

Another way to bring life back to these parts is to use the ADSK App Feature Recognition. The Feature Recognition application converts neutral 3D CAD models, such as STEP, SAT, or IGES solids, into full-featured Autodesk® Inventor® models. Feature mapping could be executed automatically or interactively as needed to maintain design intent.

Feature Recognition for Inventor is an App available to subscription customers. It can be found by following the below steps:
1. Go to http://apps.exchange.autodesk.com/en.

2. Click on "Select a store to begin".

3. Choose "Autodesk Inventor" from the list.

4. Search "Feature Recognition" at the top of the page.

5. Sign in using subscription log in credentials before the download button will appear. The sign in button is in the upper right hand corner of the page.

Notes:
- The latest version of Feature Recognition add-ins compatible with all previous version of Inventor.

- A list of compatible versions is displayed in the product page

2D: DWG Underlay

Another beauty of AnyCAD technology is the pssobility to to link 2D drawings into an inventor file - DWG Underlay. This enables the construction of the 3D model from the 2D geometry, but can also be used for the positioning of assemblies within a layout.
Your 2D data is not lost and can be used for specific purposes.

Feature based conversion - ITI

The king discipline under the data conversion is a feature based translation of the 3D data.