

Transmagic Release 8 SP4 Enhancement Overview

New Features:

XL-64 – 64-bit Support

- Native 64-bit version of TransMagic is available in EXPERT and ADVANTAGE products. TransMagic XL-64 allows for visualization and translation of virtually any file size.

Full Support for 64-bit Windows

- Supports Windows: XP, Vista, or Win7 64-bit systems. Adequate RAM is recommended for large files!

PowerPack for Inventor (Add-In)

- Now certified by Autodesk for Inventor 2011
- Now supports 64-bit version of Inventor (Version 8 - 2011)

PowerPack for SolidWorks (Add-In)

- Now includes IGES/STEP Read and Write
- Re-architected and includes various new features, including:
 - New TransMagic menu item
 - New Command Bar
 - Open With TransMagic Function
 - Save With TransMagic Function
 - Send To TransMagic Function

New Fast Bounding Box

- The new Fast Bounding Box feature in TransMagic is now the default setting. (See File -> Settings, Menu, General Tab - check the box at the bottom.) This option greatly improves the Bounding Box calculation and rendering speed. "Un-checking" the Fast Bounding will enable the super-precise calculation of a Bounding Box. However, this calculation can result in longer processing time on complex parts. The Bounding Box feature is a useful job quoting tool and is typically used to check gross part dimensions or determine the minimum stock size required to make a part.

New accelerated graphics technology

- Under the hood of TransMagic is new, accelerated graphics technology for faster rendering of large assemblies. This is evident when changing the view (rotate, pan, zoom, etc.) of very large models.

RAM Threshold Setting for the Auto Repair Wizard

- This feature provides control over how much RAM will be allocated to the Auto Repair Wizard for pre-processing a model to find errors.
 - Note: This setting can turn off the automatic error checking process if the threshold is reached and avoid a time consuming process. The RAM Threshold setting makes the Auto Repair Wizard more versatile and allows a streamline workflow while optimizing RAM usage when dealing with a wide range of datasets. If the Auto Repair Wizard is stopped on a large dataset by the RAM Threshold setting, one suggestion is to hid sub assemblies that you are not working on and then, run the Wizard. The Wizard will run on everything in the display. Another option is to run long processes at a later time or overnight.

Added "Render in Wire-Frame Mode"

- This option check-box to the "Un-Repaired Geometry Remains" dialog. This makes the highlighted geometry easier to find when working with large datasets.

New Enhancements:

Improved File -> Settings layout

- New ToolTips reduce text, simplify options and improve readability.

Enhanced Handling of Large Model Dataset

- New Level of Detail (LoD) technology facilitates high performance rendering and manipulation. Numerous function-level performance enhancements to handle large datasets when using: Lite Repair, Full Repair, Bounding Box, Scaling, Geometry Checking, and Selecting.

Added OpenGL 2.0 Support

Added DirectX 9 Support

- (Check your video card or experiment with this new setting to see if you get better graphics performance.)

The Default rendering setting is OpenGL (See File -> Settings Menu, Appearance Tab)

New Support Formats

CGR Read format supported: CATIA V5 Graphics files

- Limitations:
 - Reading: Supports the CATIA V5 scene graph data saved out to the CGR file.
 - Note: The CGR file is not associated to the CAD model, so subsequent transforms will not appear correct if parts are transformed again within the assembly. The CGR file will show the original assembly transforms as this is how the scene graph is stored in the CGR file.
 - CGR Write will be supported in a future release.

New Supported Versions

- CATIA V5 R20
 - Default output version moved up to R17 (See File -> Settings Menu, CATIA V5 Write)
- Siemens UG/NX 7
- Pro/E Wildfire 5.0
- JT Version 9.3
 - Default output version setting is now version 9.0 (See File -> Settings Menu, JT Write)
- SolidWorks 2010
- Inventor 2011
- ACIS R21
- Parasolid v22
 - Default output version moved up to 19.0 (See File -> Settings Menu, Parasolid Write)
- HOOPS 18.0