----------Version: 14.00.000 - 05/22/2023----------

Updated Format Versions:

- ACIS 2023
- NX 2206
- JT 10.8
- SOLIDWORKS 2023
- AutoCAD 2022
- Inventor 2023
- Solid Edge 2023
- Parasolid Version 35
- 3D PDF up to 2023

New Functionality:

Enhancement: Assembly Explode

- Using this tool, the assembly needs to be flattened. If the assembly is not yet flattened, you will be prompted to flatten the assembly. Click on Flatten Assembly, and an Assembly Explosion Percentage slider will appear. Drag the slider to the percentage you want to explode the assembly and click OK. Assembly Explode is located on the right side of the Operations toolbar.

Enhancement: Colorize with industrial color

- Models without specific colors will be given industrial colors by default, and others can be forced to industrial colors by selecting the geometry and clicking Colorize. If you want to change components to random colors, there is a drop down for Colorize With Random Colors. Colorize is located on the left side of the Operations toolbar.

Enhancement: Convert TEdges to Edges

- This button will convert any tolerant edges to regular edges, which can aid in the repair process. Convert TEdges to Edges can be found on the Home toolbar to the right of Full Repair and Show T-Edges.

Enhancement: Add Centerline.

- If a cylindrical face is selected, this button will add a centerline to that cylinder. There is also a Render On Top option if you want the centerline to be shown over the surrounding geometry. The centerline entity can be accessed and deleted via the Assembly Browser. Add Centerline can be found on the Operations toolbar to the right of Dynamic Dim.

Enhancement: Add Centerpoint

- If an arc or circle edge (such as can be found at the top of a hole) is selected, Add Centerpoint will create a vertex at that location. The centerpoint entity can be accessed and deleted via the Assembly Browser. Add Centerpoint can be found on the Operations toolbar to the right of Add Centerline and Dynamic Dim.

Enhancement: Export BOM

- This button will export a Bill Of Material spreadsheet. Default BOM includes Part No., Part Name and Quantity. For other data, see Settings > BOM. There are checkboxes for Part Image, Material, Weight, Surface Area, Bounding Box, Address and Company Logo. Check the boxes you want active in the BOM before clicking on Export BOM. This tool can be found via the Home button, at the far right under Special Functions.

Enhancement: Show Origin

- This button will make the file origin visible as a red, green and blue graphic where X=red, Y=blue and Z=green. A vertex will also be placed at the origin for snapping purposes. If Show Origin is clicked again, the origin will turn off, and the origin vertex will go away. So Show Origin is an On/Off toggle. This tool is located on the View toolbar, to the right of Show Axes in the Rendering section.

Enhancement: Surface Normal

- If a model, face or edge is selected, this button will add normal vectors to that geometry. If the same model or surface is selected and the button is clicked again, the vector will go away. There is also a Vertex Normal option which will generate vectors at all vertices on the selected geometry. This tool can be found on the Operations toolbar.

EXPERT and MagicHeal Enhancement: Feature Recognition.

- You can now recognize features (holes, fillets and chamfers) and remove those features. If the feature is being recognized, it will turn the feature red.

MagicCheck Enhancement: PMI Compare

- This functionality has three buttons: Open PMI Parts prompts you to open the two PMI parts you want to compare. Compare PMI runs the comparison of their PMI and gives you a brief result; pass or fail. PMI Compare Report creates a spreadsheet that compares all of the PMI elements. Note: This tool can currently compare PMI between CATIA, NX, Creo and SOLIDWORKS PMI, and can compare STEP to STEP PMI, but cannot yet compare CAD native format PMI to STEP PMI.