

3D-Tool Version 13 - Specifications

3D-Tool is a professional CAD-Viewer for 3D-models and 2D-drawings. You do not need any special CAD skills to view models in 3D, to measure distances, angles, radii and wall thicknesses as well as to create cross sections and exploded views.

The 3D-NativeCAD Converter lets you convert 3D-CAD files to common 3D exchange formats.

Available Program Versions

- 3D-Tool Free Viewer
- 3D-Tool Basic
- 3D-Tool Advanced
- 3D-Tool Premium inclusive 3D-NativeCAD Converter

The difference between the Basic, the Advanced, and the Premium version is the supported 3D file formats. Apart from that, they offer the same functions for viewing and analyzing 3D-models and 2D-drawings. Additionally, Premium Version of 3D-Tool contains the 3D-NativeCAD Converter.

The 3D-Tool Free Viewer supports 3D-Tool EXE and DDD files plus STL files. The import of multiple files into one scene is not possible. The Free Viewer cannot publish/save data and has limited features.

Languages

- Installation and documentation: English, German
- User interface: English, German, French, Korean

Free Test

You can request a one-time 14-day trial of the software with all its features. To do so, install the latest program version from our website. Then run 3D-Tool, and click on the [Get Trial-Key] button in the licensing window.

Licensing and Activation

For each license ordered, you will receive a 3D-Tool License Certificate by e-mail. The software is provided by download from our website.

After installation, the software needs to be activated with a License Key. The License Keys can be requested directly through out of the software. This requires the Authorization-Key from the 3D-Tool License Certificate and an active internet connection. The License-Key will be delivered by E-Mail within 60 minutes.

If this online request fails, the License-Key can be requested by e-mail to team@3d-tool.de. The processing of a request by e-mail can take up to two business days.

Software Requirements

Supported Operating Systems

3D-Tool Basic / Advanced	3D-Tool Premium
Windows Vista (32/64bit)	Windows 7 (32/64bit)
Windows 7 (32/64bit)	Windows 8.1 (32/64bit)
Windows 8 / 8.1 (32/64bit)	Windows 10 (64bit*)
Windows 10 (32/64bit)	

*Premium Interfaces under Windows 10 x86 (32bit)

We intensively tested the Premium interfaces under Windows 10 x86 and have not found any noticeable problems. But the interface supplier does not offer support for Windows 10 x86, so we cannot officially release 3D-Tool Premium for Windows 10 x86.

Software Requirements to use the Premium Import

The following software will be installed together with 3D-Tool, as it is required to use the Premium Import:

- Microsoft Visual C++ 2015 [x86/x64] Redistributable

64bit Support

3D-Tool is a 32bit and 64bit software. The 64bit version of the program is supported by Windows 7 x64, Windows 8 x64 and Windows 10 x64.

Hardware Requirements

We recommend a Desktop-PC, Laptop, or Tablet-PC with a standard processor, 4GB of RAM or more, and a screen resolution of 1024 x 768 pixels or more. For bigger and more complex CAD models, we recommend a graphics card which supports OpenGL 3.3 or higher.

The speed of opening and converting CAD files using the 3D-Tool Advanced and Premium interfaces depends on the processor speed and the amount of RAM available. With increasing size CAD models require more computing power and main memory. For big and complex CAD models, we recommend a fast modern processor and at least 8GB of RAM or more.

Supported File Formats - 3D-Tool Viewer (Version 13. 30)

3D File Formats	Supported by 3D-Tool Version
CATIA V6 , V6R2013 to V6R2018, PMI display support (* .catpart, * .catproduct, * .cgr, * .3dxml)	Premium
CATIA V5 , V5R8 to V5-6R2018, PMI display support (* .catpart, * .catproduct, * .cgr, * .3dxml)	Premium
CATIA V4 , 4.1.9 to 4.2.4 (* .model, * .exp, * .session)	Premium
Pro/E, Creo , 16 to Creo 5.0, PMI display support (* .prt, * .asm, * .xpr, * .xas)	Premium
Inventor , V6 to 2019 (* .ipt from Inventor 6, * .iam from Inventor 11)	Premium
SolidWorks , 98 to 2019, SW2015 and newer require 3D-Tool 64bit (* .sldprt, * .sldasm)	Premium
SolidEdge , 18 to 2019 (* .par, * .psm, * .asm)	Premium
UG/Siemens NX , 11 to NX 12.0.0, PMI display support (* .prt)	Premium
JT , V8.x – V10.2 (* .jt)	Premium
Parasolid files (* .x_t, * .x_b)	Advanced, Premium
STEP files (* .stp)	Advanced, Premium
IGS files (* .igs)	Advanced, Premium
VDA files (* .vda)	Advanced, Premium
SAT files (* .sat)	Advanced, Premium
STL files (* .stl)	Free Viewer, Basic, Advanced, Premium
VRML1, VRML2 files (* .wrl)	Basic ,Advanced, Premium
Render files (* .slp)	Basic, Advanced, Premium
PLY files (* .ply)	Basic, Advanced, Premium
XGL files (* .xgl * .zgl)	Basic, Advanced, Premium
OBJ files (* .obj)	Basic, Advanced, Premium
3DS files (* .3ds * .prj * .pli)	Basic, Advanced, Premium
ASC files (* .asc)	Basic, Advanced, Premium
Open-Inventor files (* .iv)	Basic, Advanced, Premium
2D File Formats	Supported by 3D-Tool Version
DXF files (* .dxf)	Basic, Advanced, Premium
DWG files, to AutoCAD 2015 (* .dwg)	Basic, Advanced, Premium
HPGL files (* .plt, * .plo, * .hpg)	Basic, Advanced, Premium
3D-Tool Formats (3D/2D Scenes)	Supported by 3D-Tool Version
3D-Tool EXE files (* .exe)	Free Viewer, Basic, Advanced, Premium
3D-Tool DDD files (* .ddd)	Free Viewer, Basic, Advanced, Premium

3D-CAD Input (CAD/BREP data only)

CATIA V6 (as *.catpart,*.catproduct) V6R2013 to V6R2018
CATIA V5 (*.catpart,*.catproduct) V5R8 to V5-6R2018
CATIA V4 (*.model,*.exp,*.session) 4.1.9 to 4.2.4
Pro/E (*.prt,*.asm,*.xpr,*.xas) V16 to WF 5 to Creo 5.0
Inventor (*.ipt) V6 to 2017 (*.iam) V11 to 2019
SolidWorks (*.sldprt, *.sldasm) 98 to 2019
SolidEdge, 18 to 2019 (*.par, *.psm, *.asm)
UG/Siemens NX (*.prt) 11 to NX 12.0.0
JT (*.jt) V8.x – 10.2
Parasolid-Dateien (*.x_t,*.x_b) V10 to V31
STEP-Dateien (*.stp,*.step)
IGS-Dateien (*.igs,*.iges)
VDA-Dateien (*.vda)
SAT/SAB-Dateien ASIC text (*.sat), ASIC binar (*.sab)



3D - Output

CATIA V5 (*.catpart,*.catproduct) V5R15 to V5-6R2018
CATIA V4 (*.model)
STEP files (*.stp)
IGS files (*.igs)
VDA files (*.vda)
SAT files (*.sat) V7, V8, from V10 to R2019
SAB files (*.sab), V18 to R2019
Parasolid files (*.x_t) V12 to V31
STL files (*.stl)

3D-Tool Viewer Functions (* = not available in Free Viewer)

- Hide and show parts
- Adjust graphical display of parts: color, transparency, shade mode
- Move, rotate, scale and copy parts
- Display of Product Manufacturing Information (PMI) for native models out of CATIA V5/V6, Pro/Engineer, Creo and Siemens/UG NX
- Property Editor to edit names, colors and transparency *

- Create cross sections
- Explode assemblies
- Create animations
- Place models in the coordinate system and align models*
- Repair bad edges and twisted surfaces *
- RP-Layout tool to place models on a RP-platform *
- Support of 3D-mouses made by 3Dconnexion

- 3D-dimensions: distance, angle, radius, diameter, wall thickness, clearance
- 2D-dimensions: distance, angle, radius, diameter
- Model compare: highlight differences graphically *
- Tooling analysis: drafts, draft angles, projected area, undercuts
- Wall thickness analysis
- Information about dimensions, volume, surface area and weight

- Save Custom Views
- Custom Views Editor to manage Custom Views *
- Add 3D-annotations
- Add 2D-annotations and redline markups
- Add pictures and text

- Publish 3D-Tool EXE/DDD files to share your designs *
- Publish 3D-PDF files to be viewed with the Acrobat Reader *
- Save models as STL, VRML, PLY, 3DS, OBJ and*
- Print
- Create PNG/BMP/JPG pictures
- Capture screen section to clipboard
- Export animations as AVI-video *

Common File Limitation

Due to the complexity of CAD files and despite intense testing, particular files may fail to open/convert completely or partially.

Limitations of 3D-Tool Viewer Functions

Common

- **Free Points**
The Viewer does not support free points contained in 3D-models

Tools and Functions

- **Accuracy of measurement and analysis**
3D-CAD files will be triangulated during the import into, which means they are broken down to triangles. Dependent on the quality of the triangulation some imprecision may appear in measuring and analyzing the 3D models.
- **Tooling Analysis**
The projected area of a model is calculated by graphical projection which could cause some imprecision.
- **Wall Thickness Analysis**
The calculation of wall thicknesses is computationally intensive and can take hours for big models and high quality analysis settings. Open edges and flipped surfaces may distort the results of the analysis.
- **Model Compare**
The 3D-Tool Model Compare is a graphical/visual comparison by dyeing the models in different colors and then superimposing the models. Differences located inside the models can only be seen by using cross section or by hiding parts.
- **Information Tool**
Information on volume and weight is only accurate for closed parts and models. Open edges and flipped surfaces distort the volume calculation and may lead to wrong results. Particularly, IGES files may be prone to this type of error.
- **Repair Tool**
The automatic repair may not completely close models with open edges or flipped surfaces. To get a "watertight" model manual repairs are needed.
- **Limited Touch-Screen/ Tablet-PC support**
Some 3D-Tool functions use by mouse-over/hover effects. Not all mouse-over effects are available on a touch-screen. The usability of these functions may be more or less limited by the inaccuracy of the touch points.

Publish and Save

- **No downward compatibility of 3D-Tool files**
3D-Tool EXE and DDD files published by a particular major version of 3D-Tool can only be opened by 3D-Tool or the 3D-Tool FreeViewer in this or a newer major version.
- **Publish 3D-PDF**
When publishing 3D-PDF files only the geometry and structure of the 3D models will be published but not the notices, dimensions, Custom Views, or animations created with 3D-Tool.
- **Save STL files**
To ease the editing of STL files, the "Save" function will automatically and without warning update open STL files with the state present in 3D-tool. To avoid the automatic update of existing STL files, the "Save as ..." function must be used.

Function Limitations of the 3D-Premium Import

The following limitations apply to CATIA, Pro/E, Creo, Inventor, SolidWorks, SolidEdge, UG/Siemens NX, STEP, IGES, VDA, SAT, JT and Parasolid files during the Premium-Import into the 3D-Tool Viewer and during conversion using the 3D-NativeCAD Converter.

Common

- **Assembly attributes**
Attributes assigned on the assembly level are not read by the viewer or the converter, e.g. colors assigned on the assembly level get lost and elements hidden on the assembly level will be loaded.
- **Assembly features**
Features (e.g. cuts, bodies, holes) added on the assembly level are not supported and will be ignored by the viewer or the converter. The support of patterns added on the assembly level is limited in the viewer or the converter.
- **Properties and attributes**
The viewer and the converter have limited support of common properties such as color and name. Further properties (e.g. material properties, user defined properties) are not supported by the viewer or the converter.
- **Product Manufacturing Information (PMI)**
The viewer supports the display of most PMI-data for native models out of CATIA V5/V6, Pro/Engineer, Creo and Siemens/UG NX. Limitation of PMI display can be found in the file format specific limitations. The converter does not support the conversion of PMI data, not even to output formats that support PMI.
- **2D-data / 2D-sketches**
2D-data and 2D-sketches are not supported by the viewer or the converter.
- **Layers**
Layers are not displayed in the viewer. In the converter, the translation of layer information is limited.

- **Free parts, faces, and curves**
The converter supports free parts, faces, and curves only on the top assembly level and not within sub-assemblies.
- **Axes, planes, and local coordinate systems**
The support of axes, planes, and local coordinate systems is limited in the converter.
- **User-defined views**
User-defined views and component views are not supported by the viewer or the converter.
- **Graphical data (visualization data)**
The converter does not support graphical visualization data that is contained in 3D-CAD files in addition to the CAD/BREP data. The viewer supports the display of this graphical data when the import settings are set to "CAD - graphical data". The viewer will read the highest level of display (LOD) from the part level. Graphical data from the assembly level is not supported by the viewer. The support of curves in graphical data is limited. In the viewer the colors of graphical data may differ from the original.
- **Embedded triangulated data**
Triangulated data embedded in 3D-CAD files is not supported by the converter. The viewer will only read embedded triangulated data via the import setting "CAD - graphical data".
- **Suppressed elements**
Suppressed element will not be read by the viewer or the converter, not even by activating the "Load hidden elements" or "Convert hidden elements" option.
- **Hidden elements**
Hidden elements will be read by the viewer and the converter when the "Load hidden elements" or "Convert hidden elements" option is activated. If such hidden elements are converted to file formats that do not support hidden elements (e.g. STEP) the hidden elements will be visible.

Catia V5/V6

- The converter does not support 3DXML files. The viewer will read 3DXML files featuring static tessellation. XML tessellation as well as XML files in Authoring Mode are not supported by the viewer.
- CGR files can be displayed in the viewer but are not supported by the converter.
- All parts and sub-assemblies of an assembly file (*.CATProduct) have to be in the folder of the assembly file or sub-folders, otherwise they will not be read by the viewer or the converter.
- File names may only contain ASCII characters. During the conversion to CATIA V5, all non ASCII characters in file and part names will be replaced by an underscore. Additionally, in part names the characters ! : / \ \ will be replaced by an underscore.
- The viewer and the converter require a complete file path to read and write CATIA V5 files in batch mode.
- During the conversion to CATIA V5, the attributes line-type, and line-thickness will only be translated for free wires and curves but not for edges.
- The viewer does not support the display of PMI-data without geometry references, for hole features in user defined patterns, and hole features on assembly level.
- The viewer does not support the display of PMI-data from 3DXML files.
- CATPart files created using the geometry scale "Small Scale" or "Big Scale" are not supported by the viewer and the converter.
- Due to hardware requirements, it may not be possible to view or convert CATIA V5 files using an older computer (approx. before 2003). If you plan to use 3D-Tool Premium on such a system, request a free Trial Key to test the import and conversion.

STEP

- The viewer supports PMI-data only for STEP AP242 and only from graphical CAD data (visualization data).
- The viewer does not support monolithic STEP assembly files, which contain only graphical CAD data.

IGES

- Binary and compressed IGES files are not supported by the viewer or the converter.
- IGES files often do not contain information on face normals (inside/outside). This may lead to flipped surfaces when IGES files are imported into 3D-Tool. That means, the inside of surfaces is turned outside which can have negative effects especially on volume calculation and other analyses of the model.

Inventor

- All parts and sub-assemblies of an assembly file (*.iam) have to be in the same folder, otherwise they will not be read by the viewer or the converter.
- Attributes, such as color and layer, are not read by the viewer or the converter.
- Some special Inventor features, such as "Lofting" and "Weld Symbols", are not supported by the viewer or the converter.
- Free form surfaces (T-splines) are not supported by the viewer and the converter.
- Sheet metal bodies are not supported by the viewer and the converter prior to Inventor version 11.

JT

- The viewer and the converter only support versions 8.x, 9.x and 10.x.
- "Big Endian" files are not supported by the viewer or the converter.
- The Name and Layer attributes are not supported by the viewer or the converter. The support of the Color attribute is limited.
- Free points are not supported by the converter.
- Free curves within graphical data are not supported by the viewer.

Pro/Engineer, Creo

- All parts and sub-assemblies of an assembly file (*.asm) have to be in the same folder otherwise they will not be read by the viewer or the converter.
- The viewer and the converter do not support Simplified Representations at part level.
- Even though instances in family tables are only optional in Pro/E, they are read by the viewer or the converter only if the corresponding XPR and XAS files are present. Without the XPR and XAS files always the generic parts are read.
- The converter translates local coordinate systems only to file formats that support assemblies (CATIA V5, STEP, IGES, ASAT).
- The curves "using equation" and "local push" are not supported by the viewer or the converter.
- Cosmetic features are not supported by the viewer or the converter.
- The viewer supports product manufacturing information (PMI) starting with Pro/Engineer WF3. There is no support of PMI without geometry reference, PMI text attributes (e.g. font and color), PMI set to "Remove from State", unicode text, and manually overwritten dimension values. There is only limited support of PMI based on 3D-dimensions, for hole features and patterns, as well as for combined geometric tolerances.
- Hidden "merge features" may not always be read correctly by the viewer or the converter.
- The viewer and the converter do not support flexible assemblies. Thus, positioning and/or sizing of these components might be incorrect.
- In certain complex BREP scenarios the viewer and the converter may not show the expected results for bodies with multiple lumps or for lumps with multiple shells.

UG / Siemens NX

- All parts and sub-assemblies of an assembly file (*.prt) have to be in the folder of the assembly file or sub-folders, otherwise they will not be read by the viewer or the converter.
- 2D drawing included in a 3D model will not be identified as a 2D drawing by the viewer and will be loaded partly as 3D curve elements together with the 3D model.
- For body cuts, the viewer and the converter do not distinguish between the cutting tool body and the body to be cut. Both will be loaded or converted as bodies.
- The viewer does not support Product manufacturing information (PMI) prior to NX6. PMI without geometry reference is not supported. PMI text attributes (e.g. font and color) are not supported. PMI associated with datum plane, datum axis and Annular Region is not supported.
- PMI in graphical CAD data (visualization data) is always loaded, even if they are hidden. Graphical PMI marked as "Assorted Parts" is not supported.

Parasolid X_T

- The converter creates an empty Parasolid body for various types of elements such as material properties, axis systems and user defined attributes. Some applications based on the Parasolid kernel have issues while opening such files. This is not a limitation and we suggest that you contact the support for that particular Parasolid-based application.

VDA

- Errors can occur when reading VDA files with the viewer or the converter, especially if the accuracy of the VDA data is insufficient. If such errors occur, as much data as possible is read.

SolidWorks

- All parts and sub-assemblies of an assembly file (*.sldasm) have to be in the folder of the assembly file or its sub-folders, otherwise they will not be read by the viewer or the converter.
- Colors are supported by the viewer and the converter starting with SolidWorks 2004.
- Colors assigned to instances of parts and assemblies are not read by the viewer or the converter.
- The units of a model are read by the viewer and the converter starting with SolidWorks 2001. With earlier versions the units are always assumed as meters. This also applies to unsupported units such as feet and inch.
- Hidden bodies and features within parts are not supported by the viewer or the converter.
- The attributes "Show", "No-show" and "Hidden" will be read by the viewer and the converter starting with SolidWorks 2004.
- Hidden elements can be read by the Viewer and the Converter starting with SolidWorks 2009.
- Configurations are supported starting with SolidWorks 98. In order to display and convert a part within an assembly in its correct configuration, the according configuration must be saved in the part file. This is not necessarily the case, if older parts have been opened and saved with a newer version of SolidWorks. If configurations are missing, open the part in SolidWorks, activate each configuration, and save the part. Faulty or missing part configurations within SolidWorks assemblies may cause the viewer and the converter to read the assembly only partially or even not at all. The graphical data within SolidWorks files does not provide configurations, thus when selecting an import configuration into the viewer the viewer will always read the CAD/BREP-data even if the import settings are set to "CAD - graphical data".
- Coordinate systems, work planes, free curves, and free points are not supported by the converter.
- Curves within graphical CAD data are not supported by the viewer.
- Only CAD-models out of SolidWorks major versions will be supported by the viewer and the converter. Alpha and beta versions cannot be read.
- SolidWorks 2015 and newer is only supported by the 64bit versions of the viewer and the converter.
- Only unicode text from the "Basic Multilingual Plane" (Plane 0, BMP) is supported. Filenames from the "Supplementary Ideographic Plane" (SIP) are not supported by the viewer or the converter.

SolidEdge

- All parts and sub-assemblies of an assembly file (*.asm) have to be in the folder of the assembly file or its sub-folders, otherwise they will not be read by the viewer or the converter.
- Coordinate systems (WCS) are not supported by the converter.
- Simplified views will not be read by the viewer or the converter.
- Assemblies with family tables will always be loaded with the master representation from the assembly file and not with the actual version of the assembly used in the main assembly.
- SolidEdge sub-assemblies can have multiple family tables. In the root assembly one can select one of the tables for each instance of the sub-assembly. The viewer and the converter do not support this, so that the instances may have unwanted or wrongly transformed parts.

Limitations of the 3D-Advanced Import

The following limitations apply when opening models using the Advanced interfaces.

- **Open Edges**
The Advanced-Import does not provide Healing for the generation of "watertight" models. Due to the triangulation of the models during import generally there will be some open edges. In rare cases there can be missing surfaces.
- **STEP AP242**
The Advanced-Import does not support graphical visualization data from STEP files (STEP AP242). Files containing only visualization data cannot be opened. From files containing both, visualization and CAD data, only the CAD data will be loaded.
- **IGES**
IGES files often do not contain information on face normals (inside/outside). This may lead to flipped surfaces when IGES files are imported into 3D-Tool. That means the inside of surfaces is turned outside which can have a negative effect especially on volume calculation and other analyses of the model.

Limitations of the 3D-Basic Import

The following limitations apply when opening models using the Basic interfaces.

- **VRML**
The viewer will only load triangulated data (IndexedFaceSets) and has only limited support of transformations.
- **Open Inventor 2.0**
The viewer will only load triangulated data (IndexedFaceSets) and has only limited support of transformations.

Limitations of the 2D-Import

The following limitations apply to DXF, DWG, and HPGL files when loaded into the 3D-Tool viewer.

- **Embedded pictures**
The display of embedded pictures in the viewer is limited.
- **Filled polylines**
Filled polylines (Trace entities) are not supported by the viewer.
- **AEC objects**
AEC (Architecture, Engineering, and Construction) objects are not supported by the viewer.

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