User Guide

CADDS - NX

Product Category	CADTranslate
Product Group	CADDS <> NX
Product Release Version	26.3

Document Type	User Guide
Document Status	Released
Document Revision	1.0
Document Author	Product Manager
Document Issued	16/01/2024

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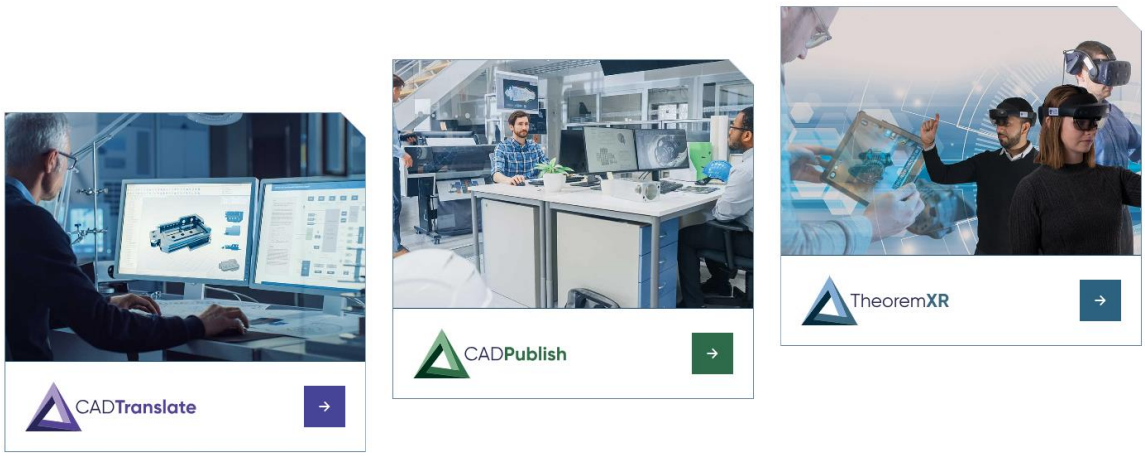
Overview of TRANSLATE

About Theorem



Theorem Solutions is a world leader in the field of Engineering Data Services and Solutions. This leadership position stems from the quality of our technology and the people in the company. Quality comes not only from the skills and commitment of our staff, but also from the vigorous industrial use of our technology & services by world leading customers.

We are proud that the vast majority of the world's leading Automotive, Aerospace, Defense, Power Generation and Transportation companies and their Supply chains use our products and services daily. Working closely with our customers, to both fully understand their requirements and feed their input into our development processes has significantly contributed to our technology and industry knowledge.

Theorem Solutions is an independent UK headquartered company incorporated in 1990, with sales and support offices in the UK and USA. Theorem has strong relationships with the major CAD and PLM vendors, including; Autodesk, Dassault Systemes, ICEM Technologies (a Dassault company), PTC, SolidWorks, Spatial Technology and Siemens PLM Software. These relationships enable us to deliver best in class services and solutions to engineering companies worldwide.

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Theorem's Product Suite

Theorem have 3 main Product brands. These are:

CADTranslate

Direct translation of 3D data to or from an alternate CAD, Visualization or Standards Based format.

See our website for more detail.

CADPublish

The creation of documents enriched with 3D content

See our website for more detail.

TheoremXR

Visualization for Augmented (AR), Mixed (MR) and Virtual (VR) Reality applications

See our website for more detail.

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The CADDS Bi-directional NX Translator

The CADDS to NX translator may be installed on a number of machines each accessing a central network-floating license.

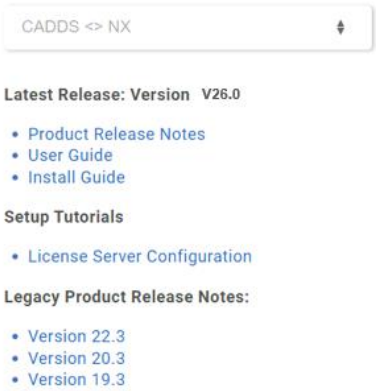
Getting Started

Documentation & Installation Media

The latest copy of the User Guide documentation can be found on our web site at:

http://www.theorem.com/Documentation

Each product has a specific link that provides user documentation in the form of PDF and Tutorials.



Each product has a specific link that provides user documentation in the form of PDF and Tutorials.

The latest copy of Theorem software can be found via the link above and by searching for the specific product. Each product has a specific link to the Product Release Document, which contains a link to the download location of the installation CD.

Alternatively, you can request a copy of the software to be shipped on a physical CD.

Installation

The installation is run from the .msi file download provided. For full details of the installation process, visit www.theorem.com/documentation and select UI from the product selection list.

License Configuration

To run any product a valid license file is required. The Flex License Manager is run from the .msi file download provided. For full details of the installation process, visit www.theorem.com/documentation

Using the Product

To use the product, follow the documented steps found in this document or follow the online video tutorials which can be found from www.theorem.com/documentation

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Using the Product

Default Translations

Default Translation – via the Command Line

Running a translation via the command line can be carried out via the ***cad_run.cmd*** file located in the ***<installation_directory>\bin*** directory. The format of the command is as follows when translating from CADDS to NX:

<Translator_installation_directory>\bin\cad_run.cmd CADDS_NX[XX] <input_file> <output_file>

The format of the command is as follows when translating from NX to CADDS:

<Translator_installation_directory>\bin\cad_run.cmd NX[XX]_CADDS <input_file> <output_file>

(Note! Replace the [XX] seen in the example with the version of NX you are using. E.g. for NX11 change to UnigraphicsNX11):
Customizing Translation Output
The following sections describe, in outline, available command line arguments to customize the output of the CADDS to NX translator.

Some of these arguments are available via the Unified Interface. All can be used as additional arguments on the default command line:

CADDS to NX Arguments List

CADDS Read Arguments

Each of these options is described below.

Option	Description
Maintain Blanked Entities	Read blanked entities and maintain their show/hid state Default is OFF ○ Command Line Syntax ▪ *maintain_blanked*
Use CADDS Entity Colours	Use CADDS entity colours rather than part colours ○ Command Line Syntax: ▪ *ecol*
Assembly Search Path	Specify the search paths that contain assembly parts ○ Command Line Syntax: • SEARCH_PATH “PATH1;PATH 2;PATH3” OR SEARCH_PATH <Filename> Where the file contains per line the paths to search i.e. PATH1 PATH 2 PATH3
Name Assembly Nodes from Associated Geometry File	○ Command Line Syntax: Default is OFF ▪ *mapitem*

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Process Part Revision Information	Read assembly revision info from _ps file ○ Command Line Syntax: ▪ *read_rev*
Assembly Units	Specify the units when reading an assembly ○ Command Line Syntax: ▪ *assy_units <mm/inch>*

NX Write Arguments

CMD LINE Option	Purpose	Data Type	Default
poly_sol/no_poly_sol	For gco Fsolids produce Facetted bodies (else attempt brep)	Flag	off
heal_ug <tol> [def tol = 0.0095/units]	attempt a UG heal on the created body (if nocheck on)	Flag	off
keep_all_bodies/no_keep_all_bodies	If input solid gets created as a solid after sewing, plus one or more tiny sheet bodies, keep or delete these	Flag	on (keep all)
nocheck	Don't check created Parasolid geometric entities	Flag	off
no_brep_prep	Prepare solids switched off	Flag	on (surfs read as nurbs+prep)
pstolmodel <num>/nopstolmodel [def num = 3]	Enable Parasolid tolerant modeling	Flag	on
pssew <tol>/nosew	Sew failed breps and opensols	Flag	on
csg_prep <tol> [def tol = 0.000001*scale]	Prepare CSG Prinitives	Flag	off
csg_shift <tol> [def tol = 0.000001*scale]	Change CSG Shift Distance	Flag	off
csgfix	Fix CSG Primitives	Flag	off
ps_fix_small/no_fix_ps_small	Remove small edges, sliver and spike faces in breps	Flag	off
ps_fix_osol/no_ps_fix_osol	Remove small edges, sliver and spike faces in opensolids	Flag	off

NX to CADDS Arguments List

NX Read Arguments List

CMD LINE Option	Purpose	Data Type	Default
read_name no_read_name	Read UG entity names (if they exist)	Flag	off
part_layer	Process As Saved part layers, else All	Flag	ALL
read_pmi	Read PMI as stroked data	Flag	off
noprep/prepsol	Prepare solids switched off / on	Flag	on (surfs read as

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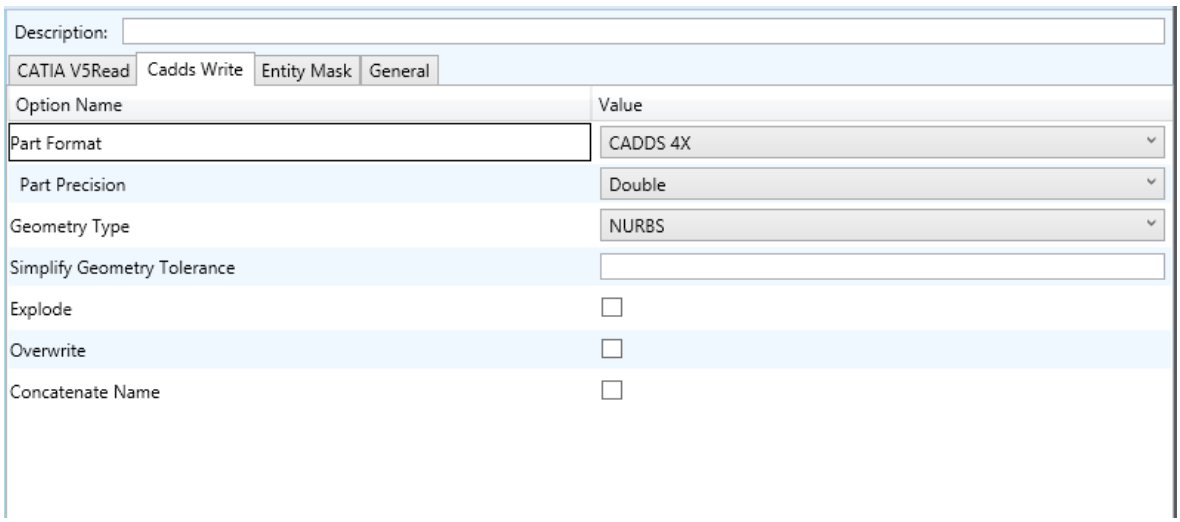
			nurbs+prep)
rd_native_edge/no_read_native_edge	Read native edge curves	Flag	off (read as nurbs curves)
trim_face_surfs/no_trim_face_surfs	Trim surface to face	Flag	off (don't trim)
ugdiags	Switch on validate read to progress file	Flag	off
read_diags	Switch on read diagnostics to progress file	Flag	off
no_mergen	No Parasolid merging of entities	Flag	on (merge)
checksol/nochecksol	Check Parasolid entities before read	Flag	off (don't check)
noprep/prepsol	Prepare solids switched off / on	Flag	on (surfs read as nurbs+prep)
mprops	Read Mass Props	Flag	off
draft	Process 2D drawings	Flag	off

CADDS Write Arguments

The image below shows the Write CADDS arguments that are available, with their default settings.



Each of these options is described below:

Option	Description
Part Format	Defines the format of the output file to be cadds4x or cadds5 - Default is cadds4x ○ Command Line Syntax ▪ cadds5
Part Precision	Specifies the output part to be written in single or double precision Default is double ○ Command Line Syntax ▪ *single*
Geometry Type	Defines whether NURBS or ASD geometry is written – default is NURBS ○ Command Line Syntax ▪ *asd*
Simplify Geometry Tolerance	Tolerance value for CADDS simplify of psurfs Default tol =0.001 in part units

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	○ Command Line Syntax ▪ c4simplify <tol>
Explode	Explode brep to faces Default off ○ Command Line Syntax ▪ split_brep
Overwrite	Overwrite existing parts default=use existing parts ○ Command Line Syntax ▪ *no_overwrite – use existing parts* ▪ *overwrite – overwrite existing parts*
Concatenate Name	concatenate top level assy name to all subcomponents default= no_concat_assy ○ Command Line Syntax ▪ concat_assy/no_concat_assy

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