



# User Guide

## CADDS - NX

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## Overview of TRANSLATE

### About Theorem



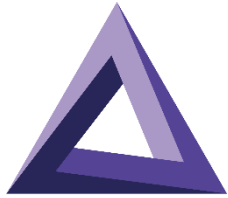
Theorem Solutions is a world leader in the field of Engineering Data Services and Solutions. This leadership position stems from the quality of our technology and the people in the company. Quality comes not only from the skills and commitment of our staff, but also from the vigorous industrial use of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

## Theorem's Product Suite

Theorem have 3 main Product brands. These are:



**CADTranslate**

### **CADTranslate**

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

See our [website](#) for more detail.



**CADPublish**

### **CADPublish**

The creation of documents enriched with 3D content

See our [website](#) for more detail.



**TheoremXR**

### **TheoremXR**

Visualization for [Augmented \(AR\)](#), [Mixed \(MR\)](#) and [Virtual \(VR\)](#) Reality applications

See our [website](#) for more detail.

# The CADD5 Bi-directional NX Translator

## 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.

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

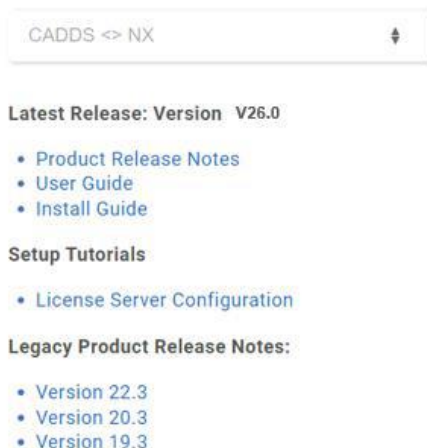
The installation is run from the .msi file download provided. For full details of the installation process, visit [www.theorem.com/documentation](http://www.theorem.com/documentation) and select UI from the product selection list.

### 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](http://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](http://www.theorem.com/documentation)



CADD5 <math>\leftrightarrow</math> NX

**Latest Release: Version V26.0**

- [Product Release Notes](#)
- [User Guide](#)
- [Install Guide](#)

**Setup Tutorials**

- [License Server Configuration](#)

**Legacy Product Release Notes:**

- [Version 22.3](#)
- [Version 20.3](#)
- [Version 19.3](#)

## Using the Product

### Default Translations

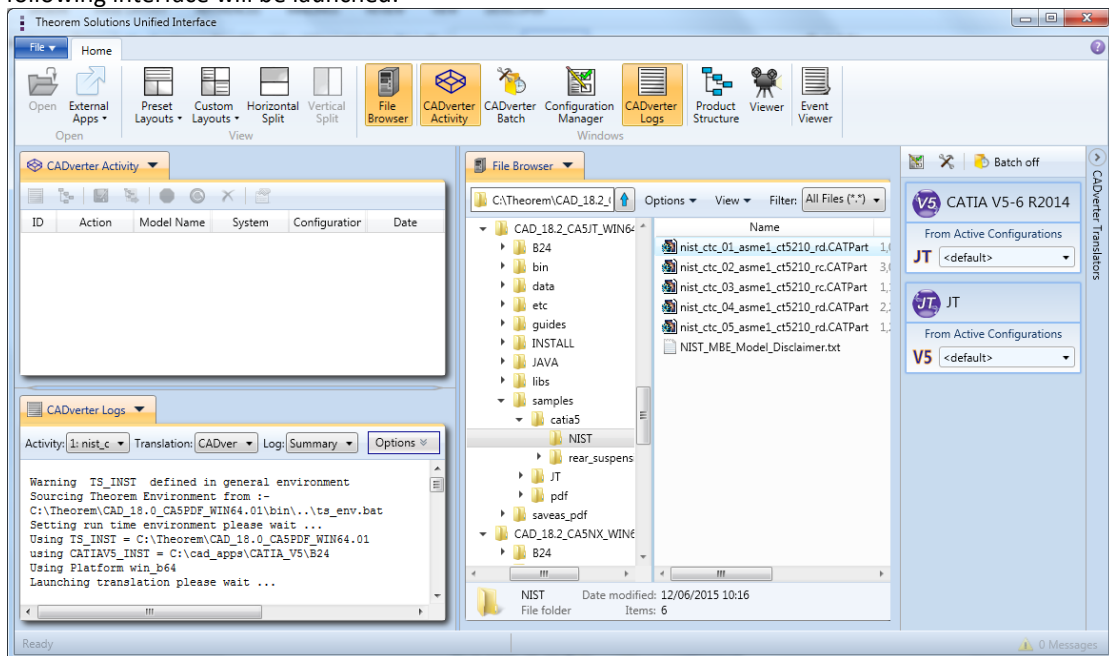
#### Default Translation – via the Unified Interface

The Unified Interface can be started via the Start Menu – if a shortcut was added during installation.

Alternatively, the Unified Interface can be run via a Windows Explorer selection in:

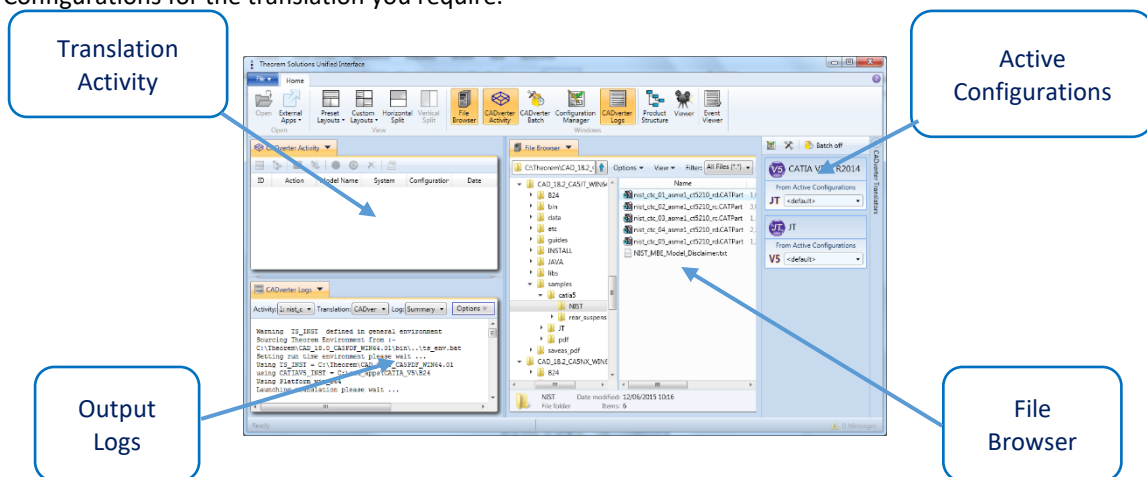
**<UI\_installation\_directory>\bin\Unified\_Interface.cmd**

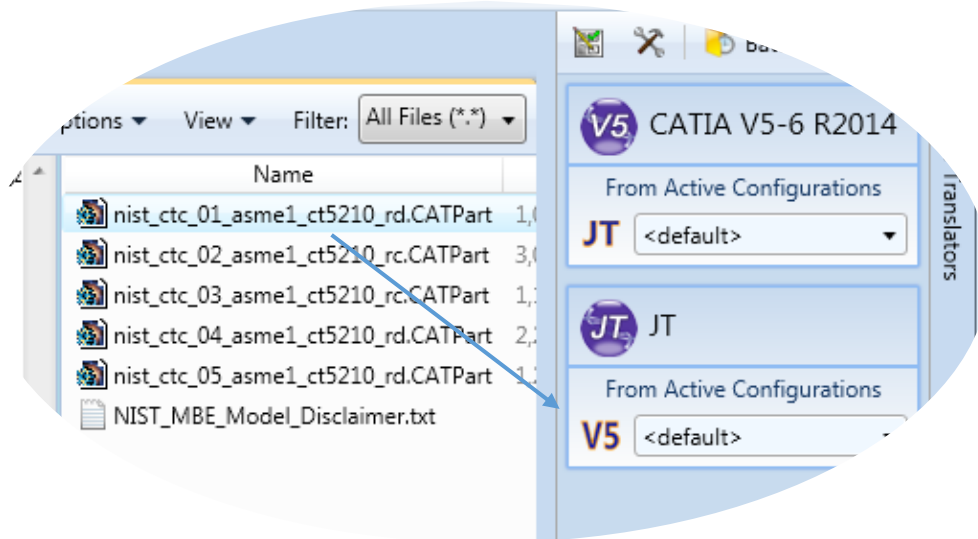
The following interface will be launched:



The default layout is split into 4 primary areas, which can be altered to the users prefer:

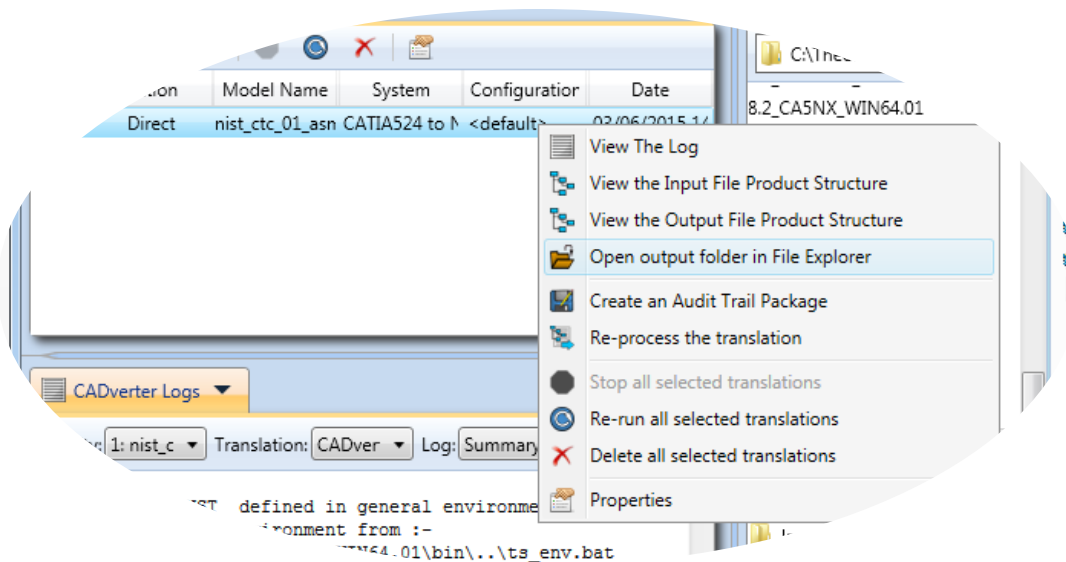
The simplest way to translate from CATIA V5 or JT is to drag a file from the file Browser Pane on to the Active Configurations for the translation you require.





On completion, the Unified Interface will display the activity information and details from the log file created during the translation, if requested, in the Translation Activity and Output Log panes, respectively.

The generated output data can be located by selecting the translation from the Activity pane and opening the output folder:



## 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 CADD5 to NX:

```
<Translator_installation_directory>\bin\cad_run.cmd CADD5_NX[XX] <input_file>  
<output_file>
```

The format of the command is as follows when translating from NX to CADD5:

```
<Translator_installation_directory>\bin\cad_run.cmd NX[XX]_CADD5 <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 CADD5 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:



## CADD5 to NX Arguments List

### CADD5 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 <ul style="list-style-type: none"> <li>○ Command Line Syntax               <ul style="list-style-type: none"> <li>▪ <i>maintain_blancked</i></li> </ul> </li> </ul>
Use CADD5 Entity Colours	Use CADD5 entity colours rather than part colours <ul style="list-style-type: none"> <li>○ Command Line Syntax:               <ul style="list-style-type: none"> <li>▪ <i>ecol</i></li> </ul> </li> </ul>
Assembly Search Path	Specify the search paths that contain assembly parts <ul style="list-style-type: none"> <li>○ Command Line Syntax:               <ul style="list-style-type: none"> <li>•</li> </ul> </li> </ul> <p>SEARCH_PATH "PATH1;PATH 2;PATH3"</p> <p>OR</p> <p>SEARCH_PATH &lt;Filename&gt;</p> <p>Where the file contains per line the paths to search i.e.</p> <p>PATH1 PATH 2 PATH3</p>
Name Assembly Nodes from Associated Geometry File	<ul style="list-style-type: none"> <li>○ Command Line Syntax: Default is OFF               <ul style="list-style-type: none"> <li>▪ <i>mapitem</i></li> </ul> </li> </ul>
Process Part Revision Information	Read assembly revision info from _ps file <ul style="list-style-type: none"> <li>○ Command Line Syntax:               <ul style="list-style-type: none"> <li>▪ <i>read_rev</i></li> </ul> </li> </ul>
Assembly Units	Specify the units when reading an assembly <ul style="list-style-type: none"> <li>○ Command Line Syntax:               <ul style="list-style-type: none"> <li>▪ <i>assy_units</i> &lt;mm/inch&gt;</li> </ul> </li> </ul>

### NX Write Arguments

CMD LINE Option	Purpose	Data Type	Default
<b>poly_sol/no_poly_sol</b>	For gco Fsolids produce Facetted bodies (else attempt brep)	Flag	off
<b>heal_ug &lt;tol&gt; [def tol = 0.0095/units]</b>	attempt a UG heal on the created body (if nocheck on)	Flag	off
<b>keep_all_bodies/no_keep_all_bodies</b>	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)
<b>nocheck</b>	Don't check created Parasolid geometric entities	Flag	off

<b>no_brep_prep</b>	Prepare solids switched off	Flag	on (surfs read as nurbs+prep )
<b>pstolmodel &lt;num&gt;/nopstolmodel [def num = 3]</b>	Enable Parasolid tolerant modeling	Flag	on
<b>pssew &lt;tol&gt;/nosew</b>	Sew failed breps and opensols	Flag	on
<b>csg_prep &lt;tol&gt; [def tol = 0.000001*scale]</b>	Prepare CSG Primitives	Flag	off
<b>csg_shift &lt;tol&gt; [def tol = 0.000001*scale]</b>	Change CSG Shift Distance	Flag	off
<b>csgfix</b>	Fix CSG Primitives	Flag	off
<b>ps_fix_small/no_fix_ps_small</b>	Remove small edges, sliver and spike faces in breps	Flag	off
<b>ps_fix_osol/no_ps_fix_osol</b>	Remove small edges, sliver and spike faces in opensolids	Flag	off

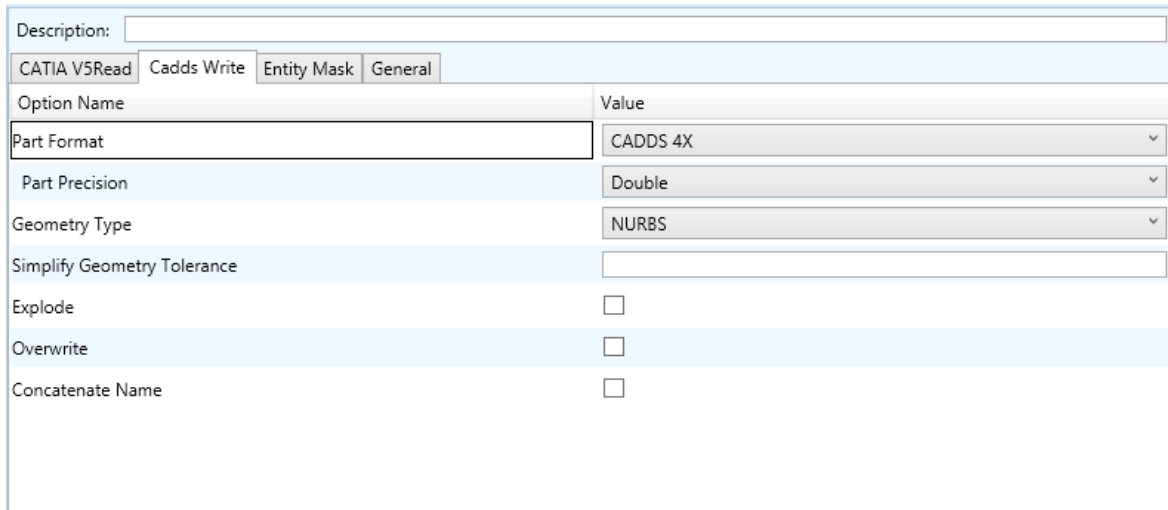
## NX to CADD5 Arguments List

### NX Read Arguments List

<b>CMD LINE Option</b>	<b>Purpose</b>	<b>Data Type</b>	<b>Default</b>
<b>read_name</b> <b>no_read_name</b>	Read UG entity names (if they exist)	Flag	off
<b>part_layer</b>	Process As Saved part layers, else All	Flag	ALL
<b>read_pmi</b>	Read PMI as stroked data	Flag	off
<b>noprep/prepsol</b>	Prepare solids switched off / on	Flag	on (surfs read as nurbs+prep )
<b>rd_native_edge/no_read_native_edge</b>	Read native edge curves	Flag	off (read as nurbs curves)
<b>trim_face_surfs/no_trim_face_surfs</b>	Trim surface to face	Flag	off (don't trim)
<b>ugdiags</b>	Switch on validate read to progress file	Flag	off
<b>read_diags</b>	Switch on read diagnostics to progress file	Flag	off
<b>no_mergen</b>	No Parasolid merging of entities	Flag	on (merge)
<b>checksol/nochecksol</b>	Check Parasolid entities before read	Flag	off (don't check)
<b>noprep/prepsol</b>	Prepare solids switched off / on	Flag	on (surfs read as nurbs+prep )
<b>mprops</b>	Read Mass Props	Flag	off
<b>draft</b>	Process 2D drawings	Flag	off

## CADD5 Write Arguments

The image below shows the Write CADD5 arguments that are available, with their default settings.



The screenshot shows a dialog box with a 'Description' field at the top. Below it are four tabs: 'CATIA V5Read', 'Cadds Write', 'Entity Mask', and 'General'. The 'Cadds Write' tab is active. The dialog contains several options with their corresponding values:

- Part Format:** CADD5 4X
- Part Precision:** Double
- Geometry Type:** NURBS
- Simplify Geometry Tolerance:** (empty text field)
- Explode:**
- Overwrite:**
- Concatenate Name:**

Each of these options is described below:

Option	Description
Part Format	<p>Defines the format of the output file to be cadds4x or cadds5 - Default is cadds4x</p> <ul style="list-style-type: none"> <li>○ Command Line Syntax <ul style="list-style-type: none"> <li>▪ cadds5</li> </ul> </li> </ul>
Part Precision	<p>Specifies the output part to be written in single or double precision Default is double</p> <ul style="list-style-type: none"> <li>○ Command Line Syntax <ul style="list-style-type: none"> <li>▪ <i>single</i></li> </ul> </li> </ul>
Geometry Type	<p>Defines whether NURBS or ASD geometry is written – default is NURBS</p> <ul style="list-style-type: none"> <li>○ Command Line Syntax <ul style="list-style-type: none"> <li>▪ <i>asd</i></li> </ul> </li> </ul>
Simplify Geometry Tolerance	<p>Tolerance value for CADD5 simplify of psurfs Default tol =0.001 in part units</p> <ul style="list-style-type: none"> <li>○ Command Line Syntax <ul style="list-style-type: none"> <li>▪ c4simplify &lt;tol&gt;</li> </ul> </li> </ul>
Explode	<p>Explode brep to faces Default off</p> <ul style="list-style-type: none"> <li>○ Command Line Syntax <ul style="list-style-type: none"> <li>▪ split_brep</li> </ul> </li> </ul>
Overwrite	<p>Overwrite existing parts default=use existing parts</p> <ul style="list-style-type: none"> <li>○ Command Line Syntax <ul style="list-style-type: none"> <li>▪ <i>no_overwrite – use existing parts</i></li> <li>▪ <i>overwrite – overwrite existing parts</i></li> </ul> </li> </ul>
Concatenate Name	<p>concatenate top level assy name to all subcomponents default= no_concat_assy</p> <ul style="list-style-type: none"> <li>○ Command Line Syntax <ul style="list-style-type: none"> <li>▪ concat_assy/no_concat_assy</li> </ul> </li> </ul>

