



NX Data Exchange Tools Best Practices

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- ▶ Tools
- ▶ Best Practices
- ▶ Helpful Resources
- ▶ Q&A



- ▶ Data Exchange

The process of transmitting data from one CAx application to another



Data Exchange Process



- ▶ Assess Your Needs
 - ▶ Downstream Usage
 - ▶ Data Types (Solids, Assemblies, Drafting, etc.)
- ▶ Common Formats (Parasolid, IGES, STEP, Other?)
- ▶ Determine Translator Options
- ▶ Evaluate the Results



NX Data Exchange Tools



- ▶ IGES
- ▶ STEP AP203/AP214
- ▶ DXF/DWG
- ▶ 2D Exchange
- ▶ CatiaV4 Interface
- ▶ CatiaV5 Interface
- ▶ Theorem Catia V4 CADverter
- ▶ Parasolid X_T
- ▶ STL
- ▶ JT



Entity Support Matrix



	Wireframe	Surfaces	Solids	Faceted Body	Assemblies	Drawings	PMI	Colors/Layers	Part Attributes
<i>IGES</i>	X	X	(1)		X	X		X	
<i>STEP AP203</i>	X	X	X		X			X	X
<i>STEP AP214</i>	X	X	X		X			X	
<i>DXF/DWG</i>	X	X	(2)			X		X	
<i>2D Exchange</i>	X					X		X	
<i>CatiaV4 Interface (3)</i>		X	X						
<i>CatiaV5 Interface (3)</i>		X	X		X				
<i>Theorem CADverter</i>	X	X	X		X	X		X	
<i>JT</i>	X	X	X	X	X		X	X	
<i>Parasolid X_T</i>		X	X						
<i>STL</i>				X					

(1) *IGES exports the faces of solid bodies as trimmed surfaces and imports IGES solids as surfaces into NX.*

(2) *DXF/DWG exports the faces of solid bodies as trimmed surfaces.*

(3) *Windows Only.*



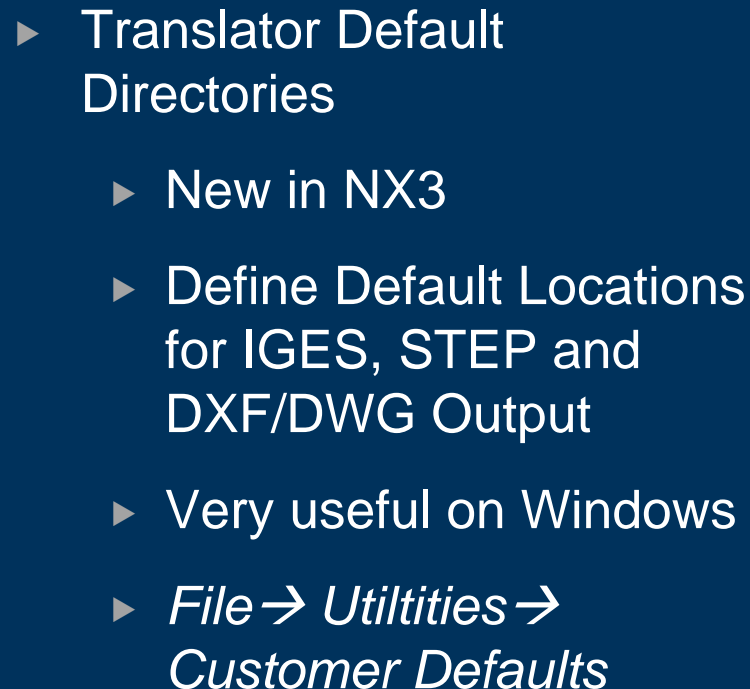
- ▶ *File* → *Open/Save As*
 - ▶ Easy to Use
- ▶ *File* → *Import/Export* (GUI)
 - ▶ Modified Settings Options
- ▶ External (xlatorui)
 - ▶ Multiple File Processing Capability
- ▶ Command Line (CLI)



- ▶ Settings Files
- ▶ Assemblies
- ▶ Entity Filtering and Mapping
- ▶ Drawings
- ▶ 2D Exchange



- ▶ Settings Files
 - ▶ Text file containing Keyword/Value combinations which control translator functionality
 - ▶ Save customized settings for re-use
 - ▶ Customer Preferences
 - ▶ Project Requirements
 - ▶ Naming Considerations





Customer Default Preferences and Imported Data



- ▶ Applies to IGES, DXF/DWG, STEP AP203 & STEP AP214
- ▶ Import ignores Customer Defaults in Some Cases
 - ▶ *File* → *Open*
 - ▶ *File* → *Import* to a New Part
- ▶ Preferences Come from Translator *Base Parts*
 - BASE_PART_IN = igesnullnx40_in.prt
 - BASE_PART_MM = igesnullnx40_mm.prt



Customer Default Preferences and Imported Data



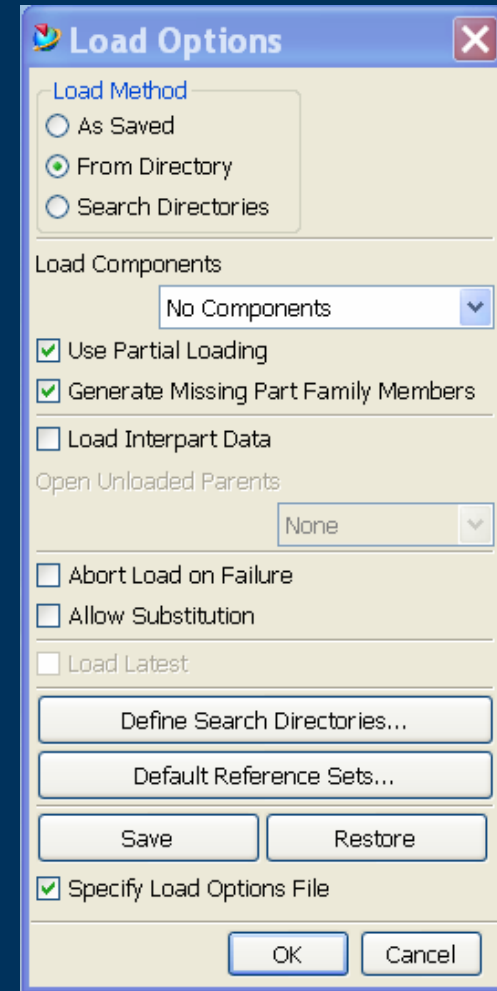
- ▶ Workarounds
 - ▶ Use *File* → *New* to create Inch and Metric part files
 - ▶ P:\Parts\nx40_in.prt, nx40_mm.prt
 - ▶ 2 Approaches
 - ▶ Substitute the existing Base Parts with the New Parts
 - ▶ ingesnullnx40_in.prt → ingesnullnx40_in.prt-orig
 - ▶ P:\Parts\nx40_in.prt → ingesnullnx40_in.prt
 - ▶ Callout New Parts in the Settings File
 - ▶ BASE_PART_IN = P:\Parts\nx40_in.prt
 - ▶ BASE_PART_MM = P:\Parts\nx40_mm.prt



- ▶ Export Assemblies with Mixed Unit Components
 - ▶ IGES, STEP, DXF/DWG
 - ▶ Mixed Units Unsupported in NX Class Selection
 - ▶ Export from Existing Part or via xlatorui



- ▶ Assembly Loading on Export
 - ▶ Default Behavior
 - ▶ Assembly Directory
 - ▶ *Modify Settings → Assembly Load Options*
 - ▶ Save New Values
 - ▶ Naming Considerations
 - ▶ UGII_LOAD_OPTIONS variable
 - ▶ System Variable
 - ▶ Settings File





- ▶ STEP

- ▶ Remember: *Surfaces* and *Wireframe* are turned off by default



- ▶ 8 Standard Colors in IGES Specification vs. 256 Colors in NX
- ▶ Settings File Option: *COLOR_FONT_MAP*
 - ▶ *IGES Color Number*
 - ▶ Maps NX RGB to closest IGES color
 - ▶ *Color Definition*
 - ▶ Uses RGB from current NX color table
- ▶ *GTAC Quarterly, Q1, 2004*



DXF/DWG Import Metric Units



- ▶ Units Default to *Inch*
- ▶ Modify Settings
- ▶ Metric Settings File
- ▶ *File* → *Open* Considerations



- ▶ DXF/DWG Font Mappings
 - ▶ Bi-Directional Control of Character, Color & Line Font Mappings
 - ▶ Set-up

dxfdwg.def

- ▶ *LINEFONT_MAPPING_FILENAME=lf.def*
- ▶ *COLOR_MAPPING_FILENAME=clr.def*
- ▶ *CHARACTERFONT_MAPPING_FILENAME=cf.def*



► Character Font Mapping – *cf.def*

! cf.def -- The following format is used to map fonts on import and export.

! import: DXF font = UG Font, Aspect Ratio

! export: UG font = DXF Font, Aspect Ratio

import : italic.shx = futura, 3.33

import : isocp.shx = cyrillic, 3.33

import : arial.ttf = lubalin, 3.33

export : blockvar = isocp.shx , 0.8

export : blockfont = italic.shx , 0.8

import : times.ttf = raster, 1.00

! The following is used to replace a set of characters with

! user defined characters.

! replace : for font check = checking for font

replace : "abc" = "def"



► Color Font Mapping – *clr.def*

clr.def -- The following is used to map colors from DXF to UG (import) and UG TO DXF (export).

! Color numbers and names can be specified on import.

import:2 = 5 ! This has been added to convert from yellow to magenta(ug)

import : " 4 " = " 11 "

import : "7" = orange

export:11=2 ! Check for export comment too !!



► Line Font Mapping – *lf.def*

! lf.def -- Line font mapping is used to map different fonts on import and

! export. The format is as follows:

! import: DXF_FONT = UG_FONT

! export: UG_FONT = DXF_FONT

import:CONTINUOUS = UF_OBJ_FONT_PHANTOM

export:UF_OBJ_FONT_PHANTOM = CONTINUOUS



Export Drawings to DXF/DWG



- ▶ Export from *Existing Part* (GUI)
- ▶ Turn Off *Surfaces* and *Solids*
- ▶ *Choose Drawings* Option



Export Large Assembly With 2D Exchange



- ▶ Extract Edges in Drawing Views
- ▶ *File → Options → Load Options set Load Components to No Components*
- ▶ Save the settings file, example:
 - ▶ P:\Settings\load_options_multi.def
- ▶ Modify the %UGTO2D_DIR%\usto2d.def
 - ▶ ASSEM_OPTIONS = P:\Settings\load_options_multi.def
- ▶ Export from GUI



2D Exchange Multi File Conversion



ugto2d_multi.def Settings File Options:

INPUT_PARTS_DIR =

OUTPUT_PARTS_DIR = P:\Results\

INPUT_PARTS_LIST = &

P:\Parts\project1\block1.prt,

P:\Parts\project2\block2.prt,

P:\Parts\project3\block3.prt

OUTPUT_PARTS_LIST = &

block1_2d.prt,

block2_2d.prt,

block3_2d.prt

DEFAULT_DRAWING_CNV = _ALL_



2D Exchange Multi File Conversion



NX Command Line:

```
> %UGTO2D_DIR%\ugto2d.exe d="ugto2d_multi.def"
```

Use xlatorui to convert 2D results to DXF,DWG or IGES



Prepare NX Data for Export



- ▶ Isolate Objects to Export
 - ▶ Layers
 - ▶ Extract to a New Part
- ▶ Heal Geometry
 - ▶ Removes Tiny Objects
 - ▶ Removes and Heals Sliver Faces, Spikes and Cuts.
- ▶ Examine Geometry
 - ▶ Repair Data As Needed



Data Validation After Import



- ▶ Review the Log File
- ▶ Visual Inspection
- ▶ Examine Geometry
- ▶ Heal Geometry



When Problems Occur...



- ▶ Note any error messages
- ▶ Check the Translator Log File
- ▶ Check the NX Log File (GUI)
- ▶ Note the steps taken leading to the error



Data Exchange Best Practices



- ▶ Helpful Resources
 - ▶ NX Documentation
 - ▶ UGSolutions Symptom/Solution Search
 - ▶ GTAC Quarterly
 - ▶ NX Translator Newsgroup
 - ▶ GTAC Translator Support
(800) 955-0000, Option 2-1-5



- ▶ Summary
 - ▶ Determine Your Data Exchange Needs
 - ▶ Check Your Data
 - ▶ Choose Your Tool
 - ▶ Choose Your Interface Option
 - ▶ Check Your Results



Questions?



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