Introduction

- Data Exchange Overview
- Tools
- Best Practices
- Helpful Resources
- Q&A
Data Exchange Definition

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

<table>
<thead>
<tr>
<th></th>
<th>Wireframe</th>
<th>Surfaces</th>
<th>Solids</th>
<th>Faceted Body</th>
<th>Assemblies</th>
<th>Drawings</th>
<th>PIM</th>
<th>Colors/Layers</th>
<th>Part Attributes</th>
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<tr>
<td>IGES</td>
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<tr>
<td>DXF/DWG</td>
<td>X</td>
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<td>(2)</td>
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<td>Theorem CADverter</td>
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<td>JT</td>
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<td>X</td>
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</table>

(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.
NX Data Translator Interfaces

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

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

Settings Files

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

■ Translator Default Directories
  ■ New in NX3
  ■ Define Default Locations for IGES, STEP and DXF/DWG Output
  ■ Very useful on Windows
  ■ File → Utilities → Customer Defaults
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
Assemblies

- 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
Entity Filtering

- **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.

```plaintext
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
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_
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
Data Exchange Best Practices

Questions?
Data Exchange Best Practices

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