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Translator Update

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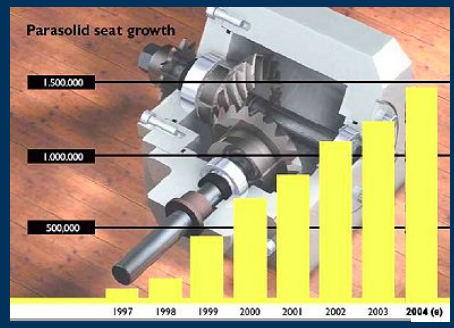
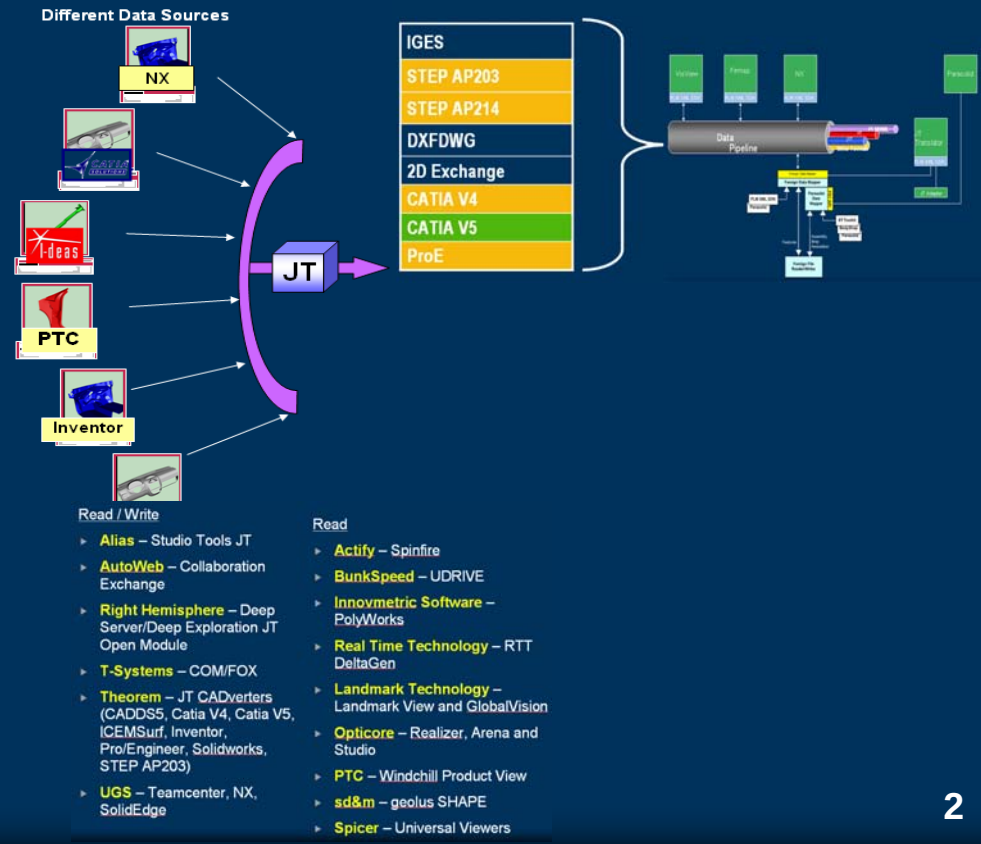
PLM World 2006

Components of NX/UGS Data Interoperability

- Parasolid based systems exchange
- Internally developed translators for NX
- JT Translators
- 3rd party partners
- 3rd party JT Open
- PLM XML



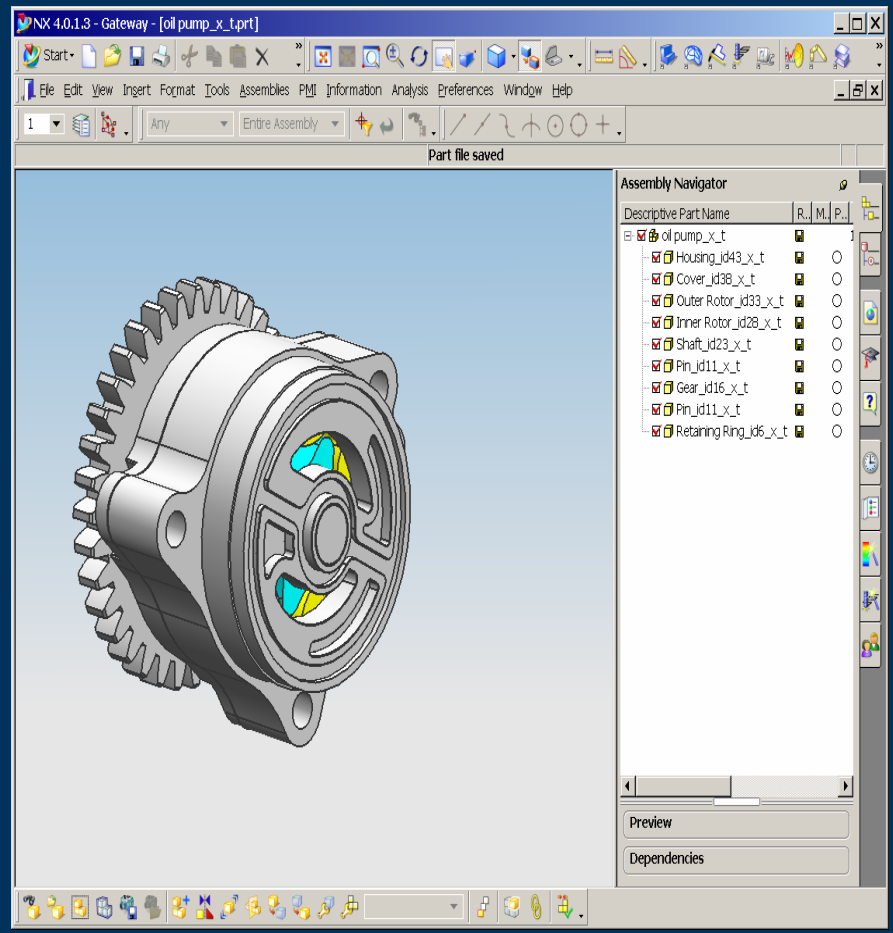


Parasolid Usage

- CAD
  - NX
  - Solid Edge
  - Solidworks
  - Bentley Systems
  - IronCAD
  - ImpactXoft
  - Fujitsu
  - ……
- CAM
  - NX CAM
  - CNC/Mastercam
  - Missler
  - Vero
  - Gibbs
  - DP Technology
  - Pathtrace
  - ……
- **CAE**
  - NX CAE
  - FEMAP
  - Ansys
  - MSC.Software
  - Delmia
  - SRAC
  - CD-Adapco
  - ……
- **Visualisation**
  - Teamcenter Visualisation
  - Cimmetry Systems (Agile)
  - Lattice Technology
  - Right Hemisphere
  - Seemage
  - ……
- **Translation**
  - Theorem Solutions
  - ITI
  - Elysium
  - Datakit
  - Lumiscaphe
  - ……

New in NX 4.0.1 File Open Parasolid

- Parasolid text and binary formats supported
  - * .x_t
  - *.xmt_txt
  - *.x_b
  - *.xmt_bin
- If assembly structure is written to Parasolid it will be recreated in NX
- Parts saved to disk on File Save
- Parasolid versions are upward compatible so the source version should not be an issue



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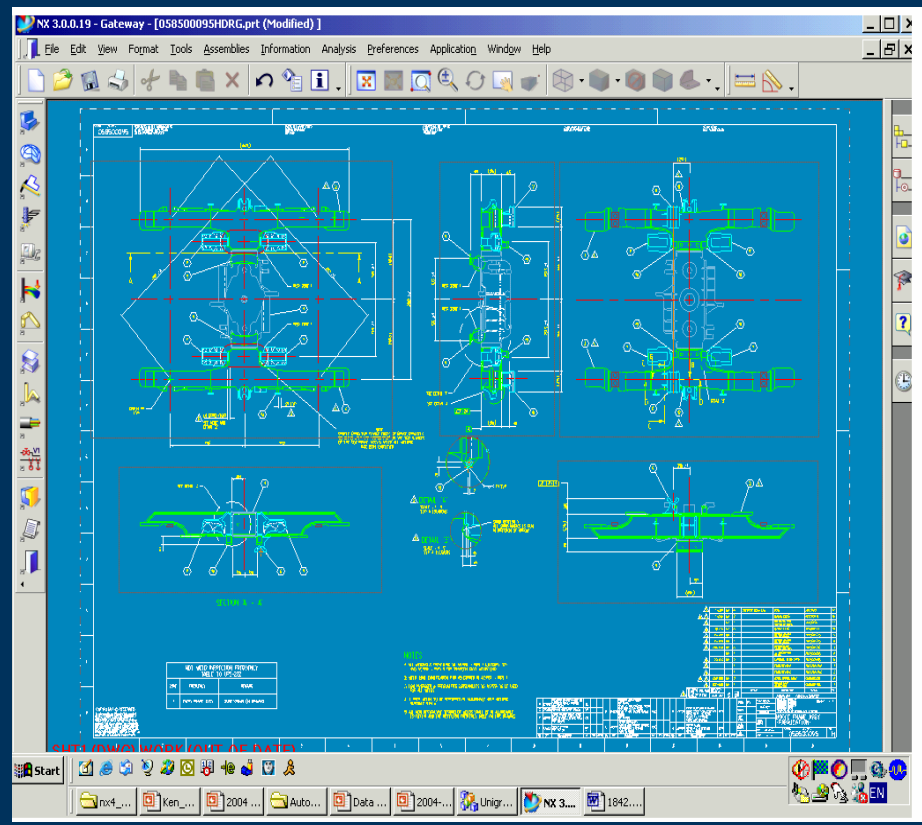
NX Translators

NX Translators Update

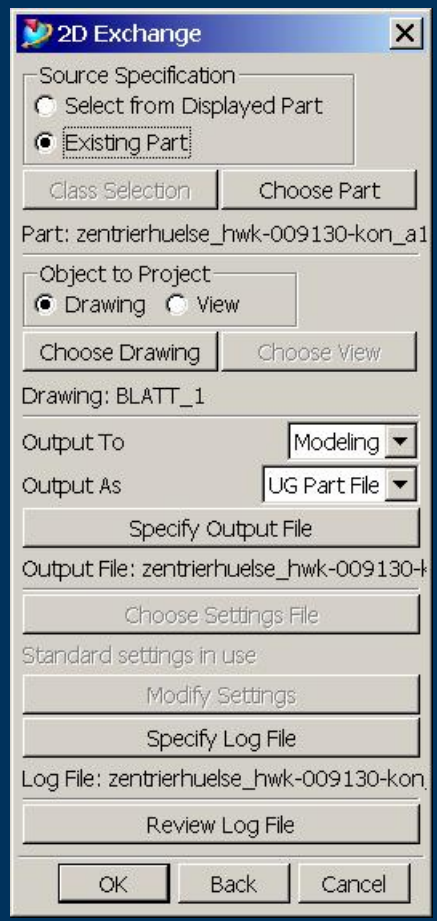
Hundreds of improvements to DXFDWG, IGES, STEP, 2D exchange CATIA Translators

- Drawing exchange improvements
- Improved robustness of CATIA V4 exchange and V5 exchange, better failure recovery
- IGES and STEP Geometry Improvements
- Most changes applied to
  - NX 3.0.4 and NX 4



2D Exchange

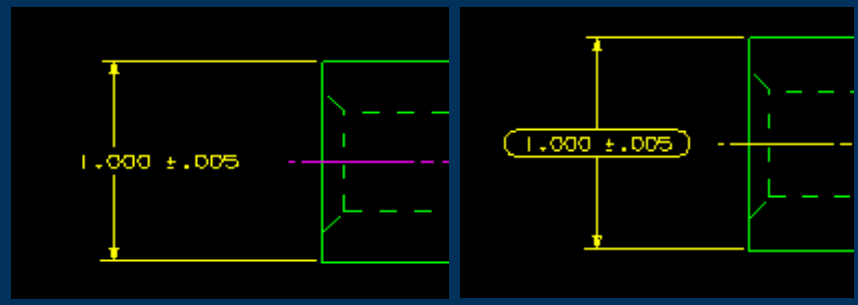
- Converts UG parts to 2D parts
  - Projects drawings or views to 2D plane, removes redundant edges
  - Used mainly as a preprocess for DXF/DWG but also used for IGES translations
  - Accuracy of 2D exchange directly impacts the quality of DXFDWG translations and many improvements and fixes have been made
  - No major functionality changes planned but continued maintenance



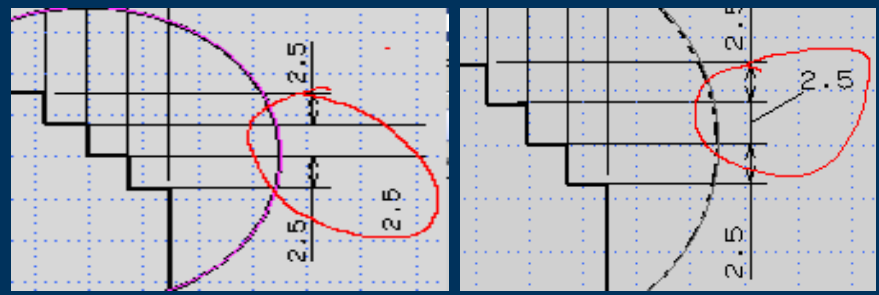
2D Exchange Improvements

- User can suppress (hide) the Dimensions using expressions, 2D exchange translator now checks the value of controlling expression.
- Preferences/Display properties: The blanked out arrows are not converted, the display properties of dimensions such as colors are maintained
- Inspection dimension added in NX3
- Dimensions between linear centerlines are now converted as real
- Improved narrow dimensions support

Inspection Dimension Before/After



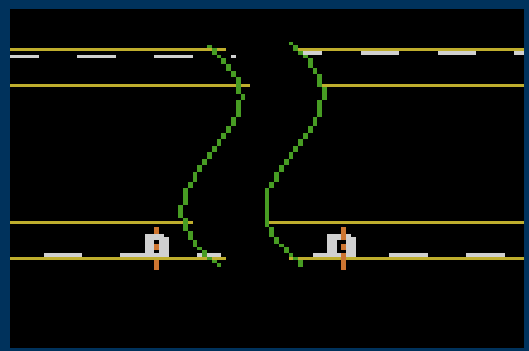
Narrow Dimension Before/After



2D Exchange Improvements

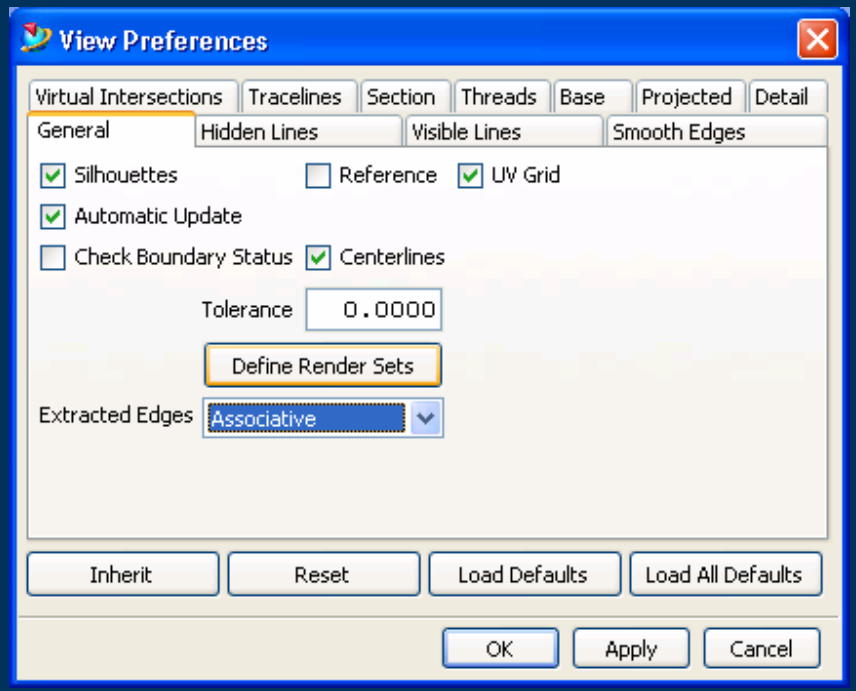
- Fonts of the notes and labels are maintained in 2d part for older parts
- Colors of the notes and labels are better maintained
- Text position and angle improvements
- Notes which are suppressed using expressions in 2D part
- Improved support of leaders of weld symbols converted to 2d part.
- Grow direction of parts list maintained

- Z-Clipping: objects outside the clipping are no longer converted.
- View border display preferences are honored.
- Views with unrecognized characters: Views with characters (“@, :, <,>) are converted correctly
- Improved ID, User defined, custom and GD&T symbol support
- Better section line and center line support
- Improved broken view support



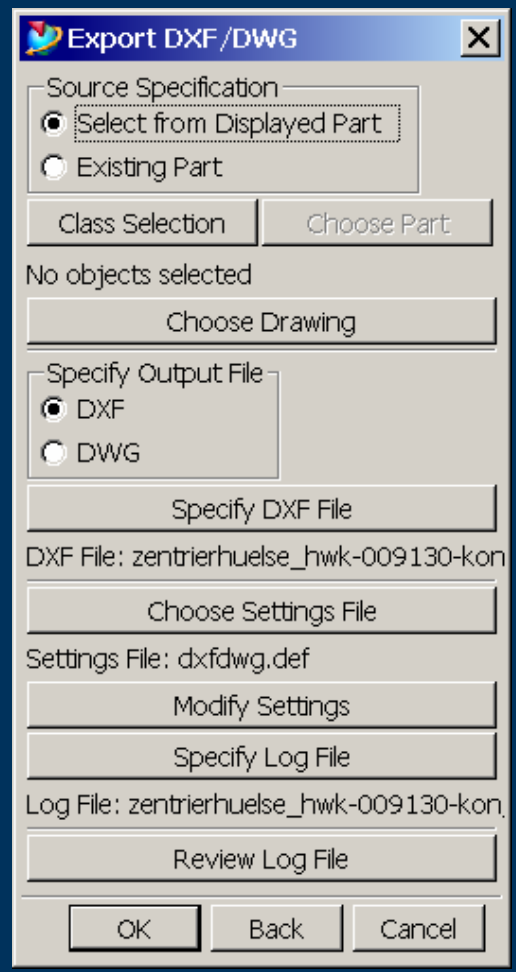
Extracted Edges Best Practice

- Benefits
  - Faster and more accurate translation to 2D format
  - Faster view update for larger assemblies
  - Drawing can be viewed without opening component parts
  - Drawings views do not change until updated
  - If all drawing views have Extracted Edges ON, then the translator does not need to modify or update the views to perform the translation.



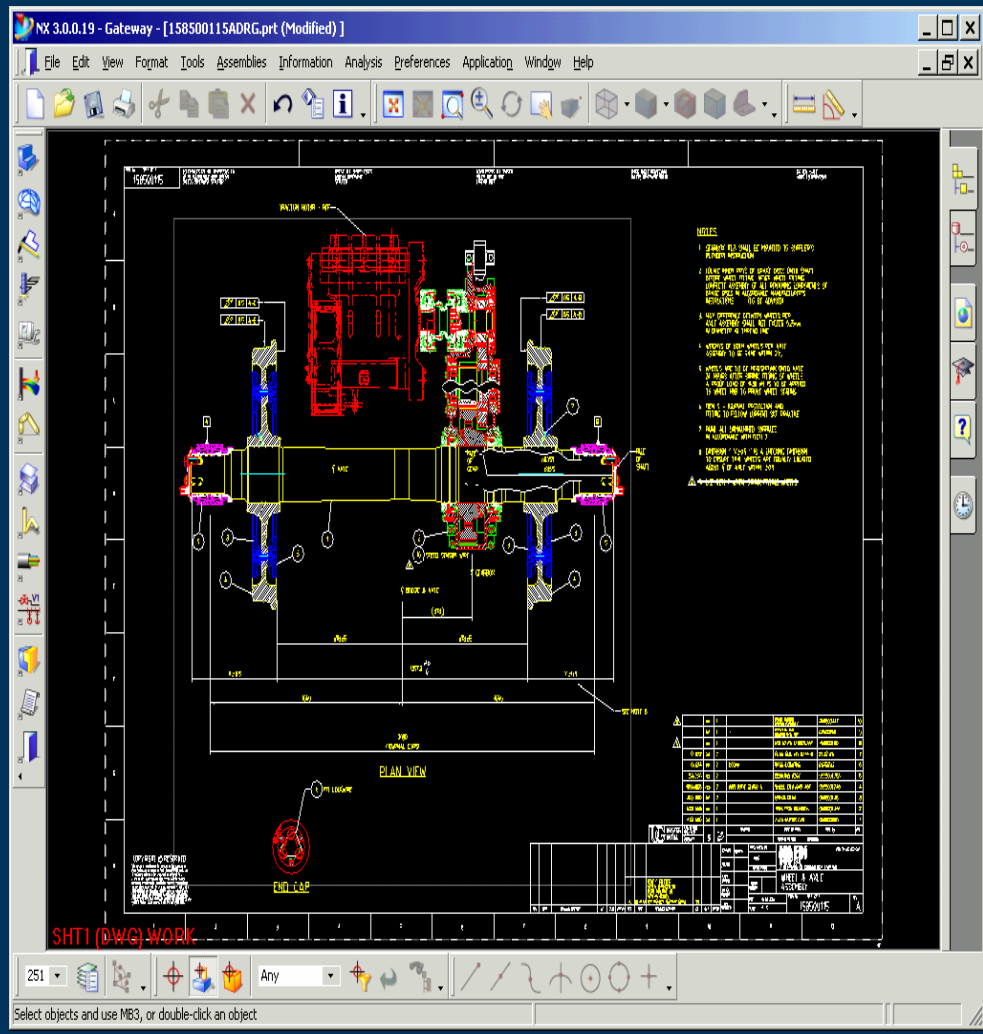
DXFDWG

- Not only translation format for AutoCAD but many other 2D systems
- AutoCAD is the standard that we work to for read and write of DXF files
- Plans are to continue to improve read and write capabilities
  - Improve intelligence of dimensions both read and write
  - Improve quality
  - Support new versions as needed



AutoCad 2004/2005 in NX 4

- Read of AutoCAD 2004 and 2005
- Write of AutoCAD R13-2005
- Based on new set of API's causing a rewrite of much of the translator
- Supports all the same entities as the existing DXF/DWG translator

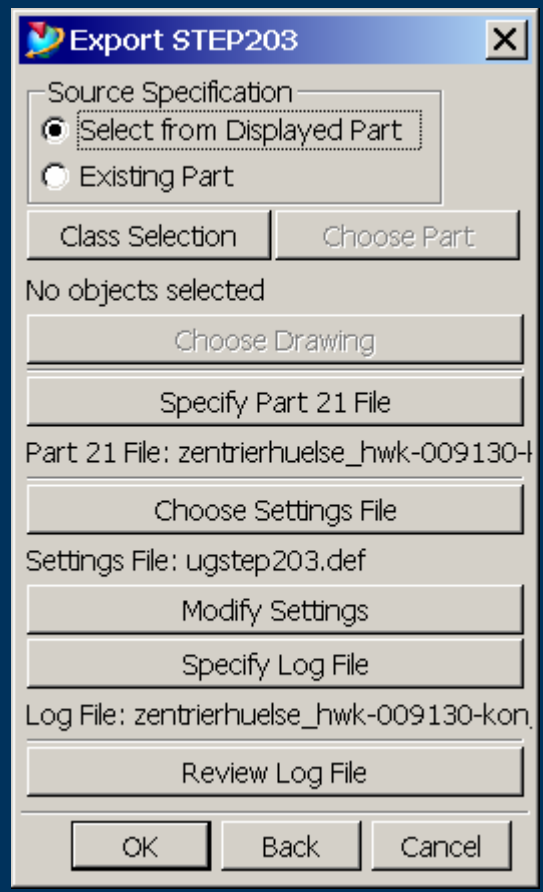


DXFDWG Improvements

- Improved Layer mapping
- Assembly component visibility
- Sub component positioning
- Improved font support and font mapping
- Better support for MTEXT, special characters and control characters
- Improved edit of dimensions from NX to AutoCAD
- Poly-lines are imported as splines reducing file size
- Export of sketch curves allowed
- Better tabular note support
- Improved hatch import and hatch boundary
- Improved user defined symbol support
- Import of view ports to NX
- …

STEP

- NX supports STEP AP203 and STEP AP214
- Improvements in performance release of NX 4.0.1
- Added coordinate system support
- Improved error recovery

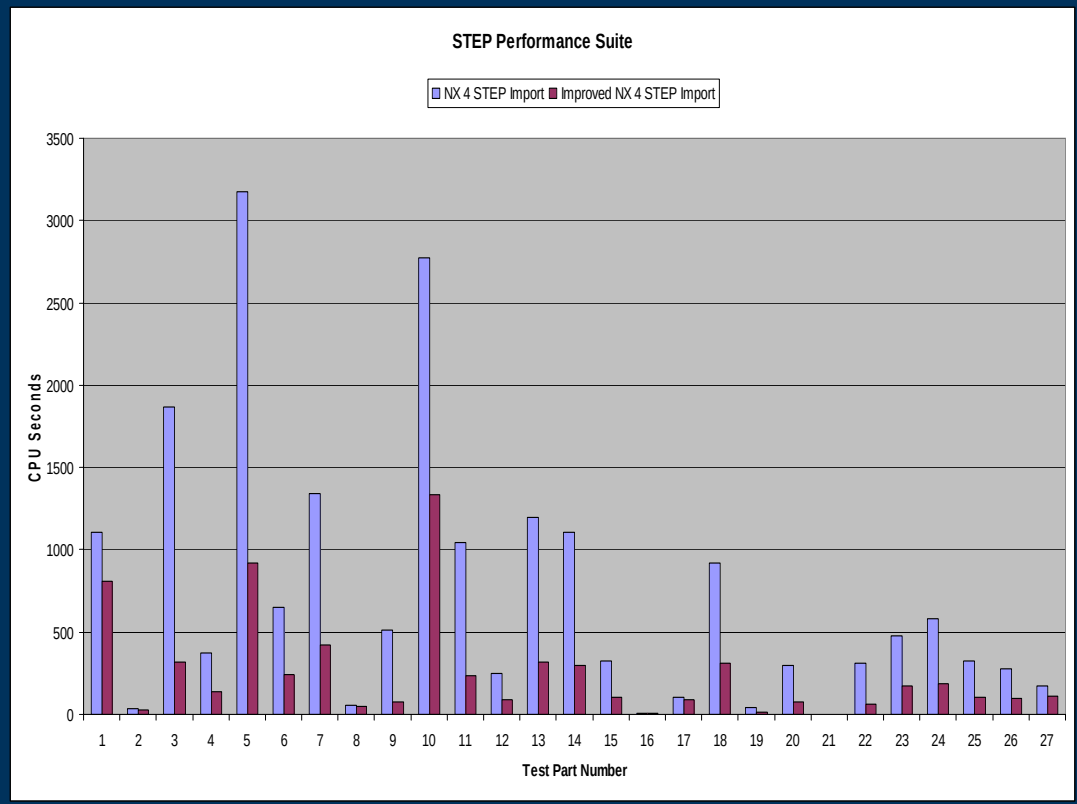


Value Add of NX STEP Translator

- The NX STEP translator adds value to the STEP data by creating the best data it can for downstream application usage including:
  - Maintaining analytic data from analytic data
  - Creating analytic data from b-surface data where possible
  - Creating exact edges from tolerant edges
- Extensive data validation and repair:
  - Edge of surface
  - Vertex off surface
  - Face to face inconsistencies
  - Self intersections …

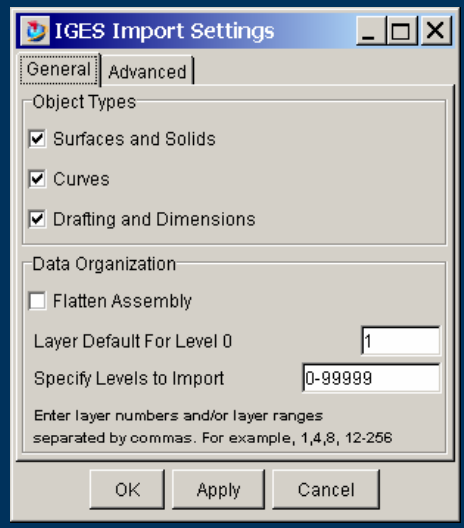
STEP Performance

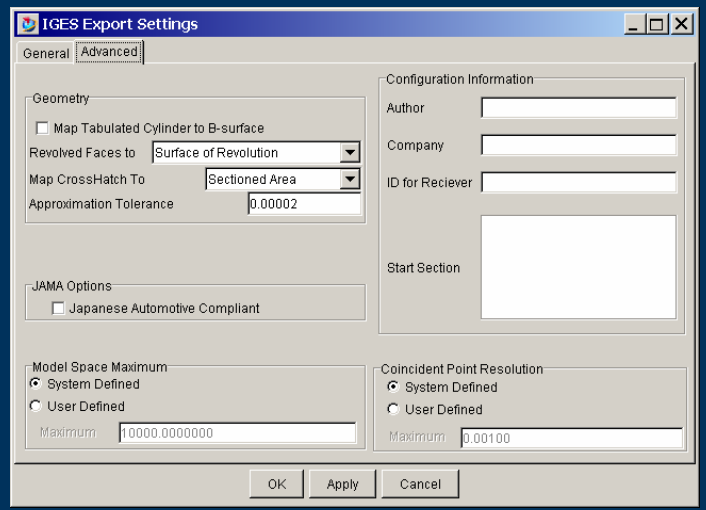
- Performance improvements
- Overall the test suite showed a 65% faster translation across the suite over NX 4 phase 25 in NX 4.0.1
- The Average improvement was 58%
- The Average improvement for CATIA V5 STEP files and NX STEP files into NX were nearly identical at 58%



IGES

- Still Widely used by many applications as the preferred exchange format, machining, forming, and other applications
- Specification is fixed
- IGES from a functionality point of view is in maintenance mode
- Improvements made to default settings to improve performance
- Code changes to speed performance on export
- Quality





Recent NX IGES Export Performance Improvements

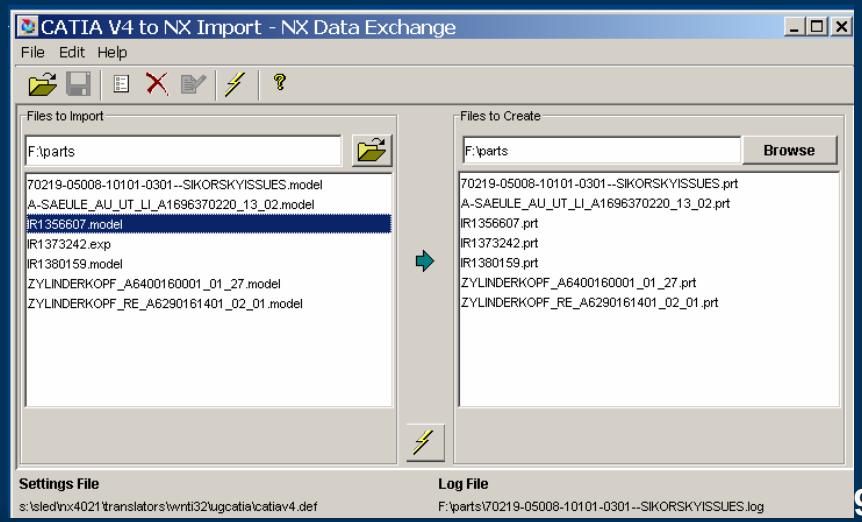
- Timing studies have led to several recent improvements:
- Used a more reasonable tolerance value of .025 mm.
  - An NX IGES tolerance setting controls the IGES file output approximation accuracy, e.g., for blend faces. The default tolerance, .0005 mm, required more computation time and creates more data. The new tolerance cut the translation times in half.
  - Improved algorithms
  - removed unnecessary updates
  - removed unnecessary marks

Overall results in some parts taking 1/10 of the time of the original translation

CATIA V4 NX Interface

- Internal CATIA V4 translator available from file open, file import, file export and file save as
- Bidirectional support of solids and surfaces, does not support wire frame or assembly data (.model assemblies are read but flattened on import)
- Translates CATIA V4 *.model, and *.exp files
- Geared toward the machine shop, or model shop that needs access to solid geometry
- Not intended to replace Theorem translator, but to give users easier access to the solid geometry
- Windows only

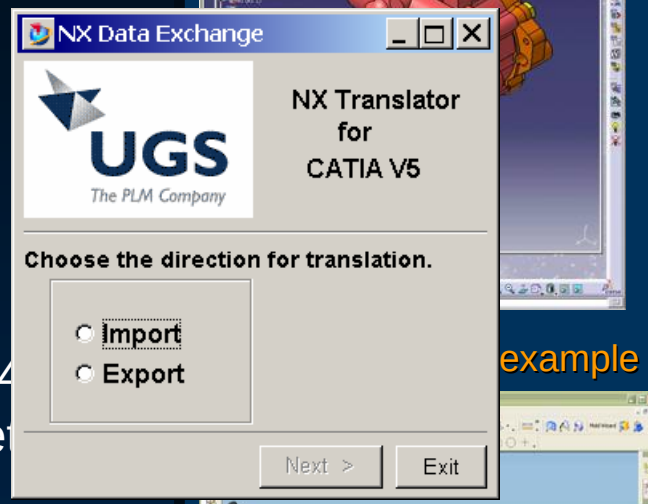
- Environment controlled options
- Export: SolidE support
  - UGII_CATIA_EXPORT_AS_SOLIDE = 1
- Import: Healing and simplification
  - UGII_CATIA_HEAL_GEOMETRY = 1
  - UGII_CATIA_PS_BODYSHOP = 1
- External interface available in NX4



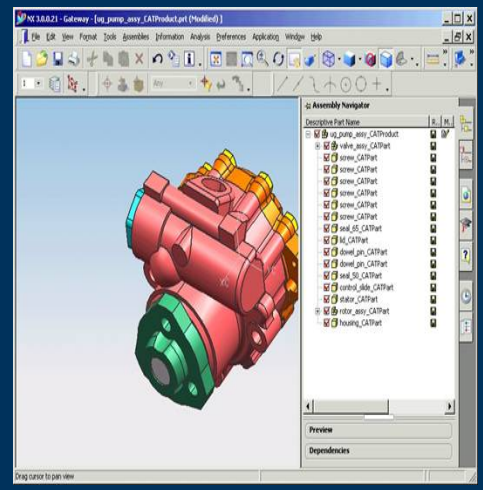
CATIA V5 Support

CATIA V5 to NX translator introduced in NX 2.0.4.2 and NX 3

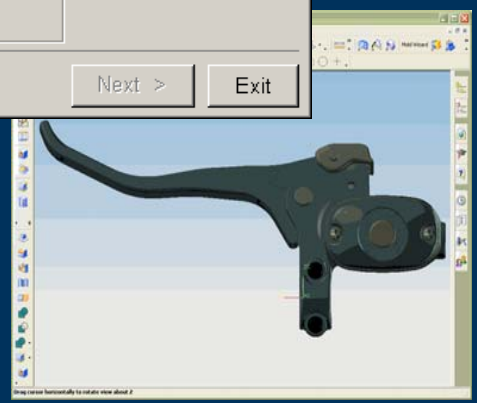
- Read
  - CATPart (component)
  - CATProduct (assembly)
- Maintains Assembly
- Geometry support for
  - Solids, Surfaces, and
  - External user interface
- Added to File Open in 3.0.1
- NX to CATIA V5 write in NX 4
  - Solid and surface geomet
  - Flattened assemblies
  - Added to File Export in NX 4
  - Added to external user interface

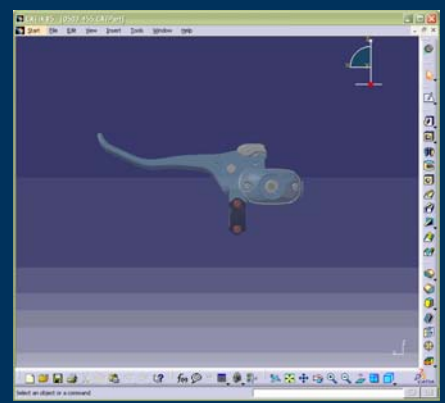


CATIA V5 to NX example



example



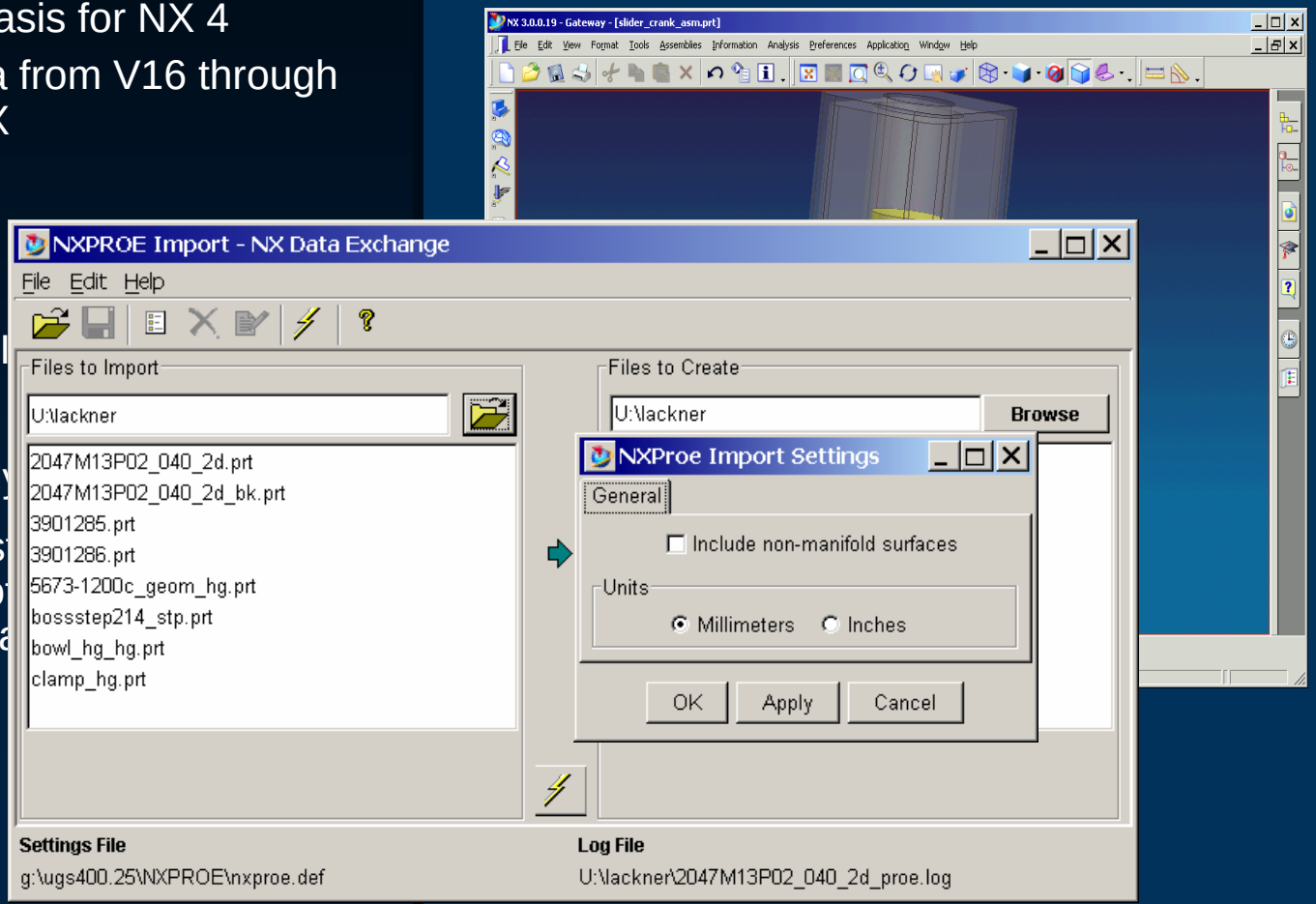


Pro/E Read

NX 4

- Available on trial basis for NX 4
- Read of Pro/E data from V16 through Wildfire 2.0 into NX
- Support for:
  - Geometry
  - Assemblies
  - External User I
  - File Import
- Windows 32 bit only
- Fewer translation s to machining and o usage of Pro/E data

Pro/E Part in NX



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JT Translators

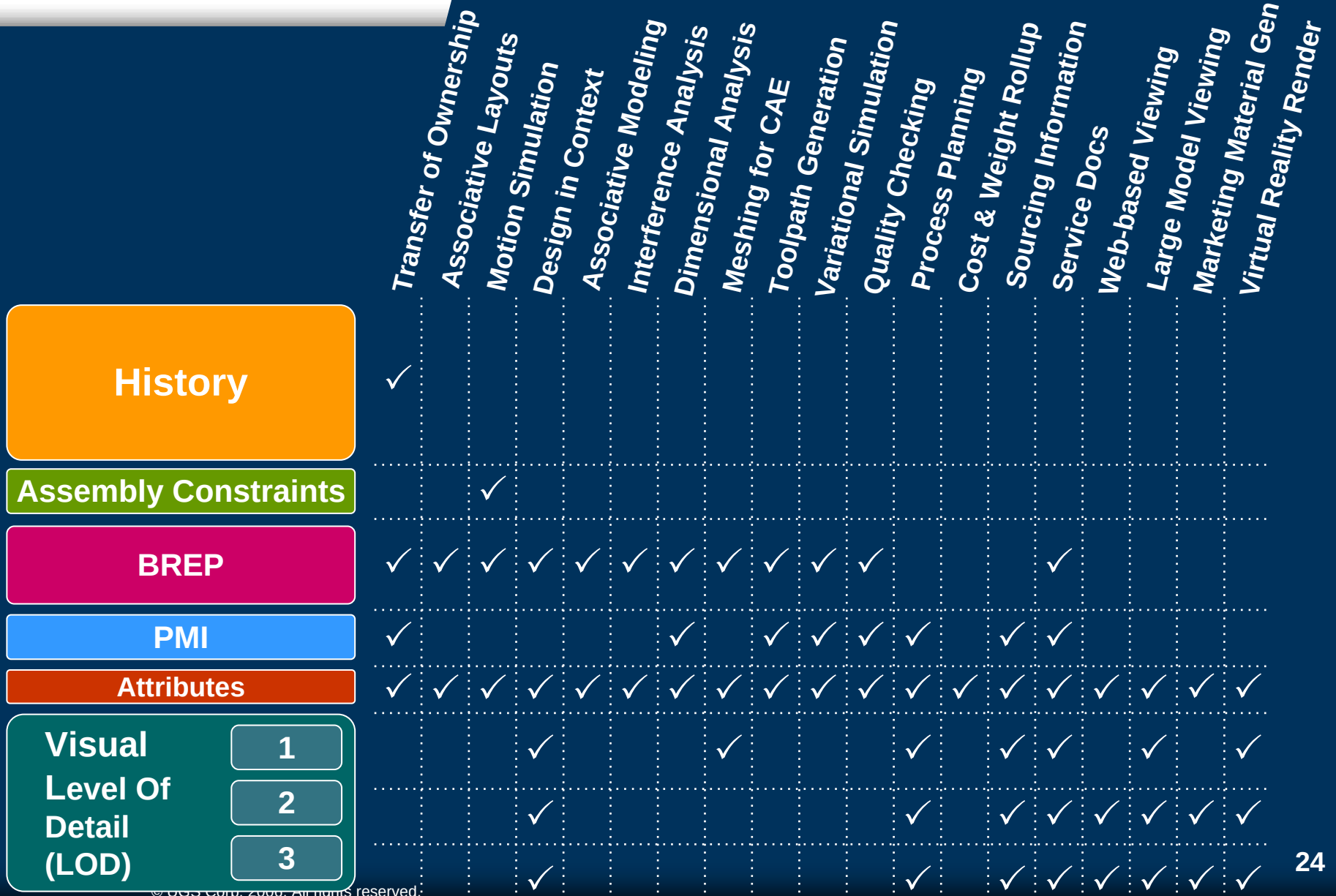
JT

- Using common collaboration format (JT) optimizes for viewing cases

Activity		Use of data by receiver	SW used by receiver	% of Cases	JT file	CAD file	Comments
Reviewing		Review, understanding, comment (often by large numbers of people)	JT domain	60%			CAD data is too big, and CAD systems are too expensive
Analyzing		Inclusion into assembly. Measuring, checking clearances.		30%			Precise data in JT files allows accurate measurements
Referencing		Design use… new data… NC toolpa… mating parts)	JT based design in context.				JT data imported into receiver's CAD system
Addition	Transfer of Ownership	Additions made to design itself (e.g. additional features)	Native CAD Req'd	1%			JT data imported into receiver's CAD system
Editing	Transfer of Ownership	Transfer of ownership. Extensive changes to shape of part	Native CAD Req'd	1%			Data transferred via JT only allows certain types of shape editing (no history replay)

Key: O = Good requirement coverage.
△ = Marginal functionality.
× = Not well supported.

JT Use Cases



CAD TO JT 1 of 2

CAD	Marketed By	Facets	Brep	PMI	Assembly Structure	Assoc -iativity	Comments
NX	UGS	☑	☑	☑	☑	☑	
Ideas-NX	UGS	☑	☑	☑	☑	☑	
SE	UGS	☑	☑	NA	☑	☑	
STEP	UGS	☑	☑	-	☑	-	
IGES	UGS	☑	☑	-	☑	-	
VRML	UGS	☑	NA	NA	☑	NA	
ProE	UGS	☑	☑	☑	☑	-	
S/works	UGS	☑	☑	-	☑	-	
AutoCAD	UGS	☑	☑	-	☑	-	
STL	UGS	☑	NA	NA	NA	NA	
Robface	UGS	☑	☑	-	☑	-	
V4	UGS	☑	☑	☑	☑	-	
V5	UGS	☑	☑	☑	☑	☑	

☑ Available ☑ Under development **NA** – Not applicable

CAD TO JT 2 of 2

CAD	Marketed By	Facets	Brep	PMI	Assembly Structure	Assoc -iativity	Comments
V5	Theorem	☑	☑	☑	☑	-	
V5	T-Systems	☑	☑	☑	☑	-	Available as part of a service
V5	Spatial (Dassault)	☑	☑	-	☑	-	Toolkit based on JT Open
V4	Theorem	☑	☑	-	☑	-	
ICEM	Theorem	☑	☑	-	-	-	
Inventor	UGS	☑	☑	-	☑	-	Theorem developed
Inventor	Theorem	☑	☑	-	☑	-	Same product
CADDS5	UGS	☑	☑	-	☑	-	Theorem developed
CADDS5	Theorem	☑	☑	-	☑	-	Same product
ProE	Theorem	☑	☑	-	☑	-	
S/Works	Theorem	☑	☑	-	☑	-	
STEP	Theorem	☑	☑	-	☑	-	
Alias	Alias	☑	☑	NA	☑	-	

☑ Available ☑ Under development **NA** – Not applicable

CAD formats created from JT

CAD	Developed by	Facets	Brep	PMI	Assembly Structure	Assoc -iativity	Comments
STEP	UGS	NA	☑	-	☑	-	
Robface	UGS	☑	-	-	-	-	
Nastran	UGS	☑	-	-	-	-	
VRML	UGS	☑	NA	NA	☑	-	

☑ Available ☑ Under development **NA** – Not applicable

Summary of Capabilities

	Sol	Surf	ASM	DRW	ATT	PMI	FEAT
IGES	☑	☑	☑	☑*	☑		
STEP 203	☑	☑	☑				
STEP214	☑	☑	☑				
DXF/DWG				☑*			
ProE to NX direct	☑	☑			☑		
CATIA V4 Direct	☑	☑			☑		
CATIA V4 Theorem	☑	☑	☑	☑*	☑		
CADDS4/5 Theorem	☑	☑	☑	☑*			
3rd party Feature Exchange	☑	☑	☑		☑		☑
JT	☑	☑	☑		☑	☑	

Questions