Siemens PLM
Connection Japan 2016
Welcome.
**NX 11 CAD Projects**

### User Experience
- Multi-Core extension
- Cloud Licensing Support
- Simple Part File Password
- Support for Multiple Managed NX Configurations
- STEP 242 Adapter Support
- Close Subassembly Performance
- Improve Units Handling
- High End Rendering Engine
- Multiple Displayed Parts Investigation
- Multi-Core Lightweight Drawing View Completion

### Modeling
- Feature replay performance improvements
- Part Modeling version up improvements
- Feature algorithm update controls
- Edge Blend improvements
- Face Blend improvements
- Part Modules - Usability & Performance
- Interrupt feature
- Sketch - Usability Improvements
- Sketch - Performance improvements
- Sketch - Scale constraint
- ST2D - Local Scale
- ST2D - Rigid Sets
- Sketch - Constraint to string of curves
- Sketch - Splines improvements
- Sketch - Notes within feature
- Sketch - Display dependencies to external geometry
- Sketch - Position and reattach improvements
- Sketch - Horizontal and Vertical constraints of points and endpoints of lines
- Synchronous Modeling - Delete Face supports multi body output
- Synchronous Modeling - Replace face at laminar sheet boundary
- Synchronous Modeling - Delete Notch/Cliff blends
- Synchronous Modeling - Optimize Face Selection improvement
- Selection - Select from navigator consistency
- Design Templates - Relinking a product interface
- Design Templates - Product interface expressions
- Design Templates - WAVE link out of date status improvements
- SDPD: Modular Interface Design
- Variational Sweep improvements
- Feature Modeling Swept Volume
- Wireframe - Spline Curve feature
- Wireframe - Scale Curve feature
- Wireframe - Isocline curve feature UI & Selection Intent
- Surfacing - Variable offset surface (Tailored blanks)

### Drafting
- Multi-thread support for Smart Lightweight views
- Working with arrangements on drawings
- View Break workflows
- Settings Enhancements
- Track Drawing Changes Enhancements
- Drawing Compare
- Dimension UI Enhancements
- Hole Callout enhancements
- Dimension Settings
- Limits and Fits
- Support for Angular Directed Dimensions
- 4GD-Parts List Support
- 4GD-Relations and Where-Used
- 4GD-View of Part Support
- NX Layout-enhanced workflows
- NX Layout-leveraging legacy data

### Assemblies
- 4GD Weld Support (NX 10.0.2)
- 4GD Constraints (Phase 2)
- 4GD Support for Change Management
- 4GD Support for Parts List
- Direct Model Integration
- Large Assembly Performance
- Component Patterns with Pure Reference Patterns
- Support for Multiple Displayed Parts Investigation
- Associative References to Multi-CAD Override Geometry
- Support for Mixed/Facet Modeling Initiative

### Routing
- Routing support of attributes
- Formboard Architecture
- Update Bolting Options
- Quick Path enhancements (NX 10.0.1)
NX 11 Architecture

- Drag and Drop from Embedded Active Workspace in NX 🎉
- Enhanced Window Management – e.g. Ctrl+Tab 😊
- Automatic Application Change
- High Density Display Support – 4K role
- **New!** Reference Point Cloud
- Iray+ High End Rendering 😄
- Information Window – HTML format 😊
NX 11 Design

Sketcher

- Enhanced Usability
- Large Sketch Performance Improvements
- **New!** Scale on first dimension 😊
- **New!** ST2D Scale Curve 😊
- **New!** Scalable Sketch Group
- Enhanced Constraints 😊
- **New!** Sketch Browser 😊
- Reattach Improvements
NX 11 Design

Wireframe

- **New!** Scale Curve feature
- **New!** Spine Curve feature
- Improved Isocline Curve UI & Selection
- Improved curve fitting and accuracy
- Combined Curve Projection
- Composite Curve
Demonstrations
Surfacing

- Variational Sweep improvements
- Guided Extension feature – NX 10
- Trim Sheet improvements
- New! Variable Offset Face 😊
- New! Swept Volume 😊
- New! Flattening and Forming toolset 😊
Blending

• Edge and face blending works more robustly for complex cases
• Improved blend shape and flow
• Apply large radius blends over small
• Handling steps in the part
• Blend self intersection patch quality
• Edit trim and attach options in Face blend
Features

- **New!** Bounding Body available in the Modelling and Shape Studio Applications
- Select bodies and features from Navigator as inputs to feature operations
NX 11 Design

Synchronous modelling

• Replace Face support sheet bodies with open edges 😊
• Optimize Face automatic selection of faces or interest
• Delete Face multi-body output 😏
Demonstrations
NX 11 Design

NX Layout

- **New!** Create from 3D 😊
- Edit in Isolation
- Improved colour scheme workflow
- Control + Press/Drag support
NX 11 Design

Assemblies

- Display Parent: List Parents in Order 😊
- Constraint Limits: Distance and Angle
- Pattern Component: Enhanced Reference Patterns
- Extraction Path: Manual Guide Path Enhancements
PMI

- New! Convert to PMI 😊
- PMI Resize Enhancements
- Dimension UI Enhancements
- Drafting section line derived from PMI
Drawing

- Multithread processing for Smart Lightweight views
- Drawing Compare with tracking 😊
- Arrangements on Drawings
- View Break Enhancements 😊
Demonstration
Expressions

• New “Table” interaction model
  • No more separate edit area
  • Edit-in-place for formula, units, etc.
• Edit Dialog
  • Extended Text
  • Alternate Entries
• New multi-filtering mechanism
• Immediate Update OR Future change notification
  • “Display Only Pending Changes”
  • Automatic return post-update
• Error Checking Display
  • Highlight on mismatch
  • Same name checking
  • Auto-revert on Apply
  • Auto-correct on Create
NX 11 Design

Product Template Studio, (PTS)

- Java no longer needed at all for PTS. Visual Rules, can now be defined and edited inside NX
Sheet Metal

- The Aerospace Sheet Metal application has been retired in NX11.
- Aerospace features re-mapped to Advanced Sheet Metal.
- Features enhanced in NX 11:
  - Advanced Flange
  - Joggle
  - Lightening Cutout
  - Solid Punch
  - Normal Cutout
  - Contour Flange
Subdivisional modelling

- **New!** Copy & Paste
- **New!** Split & Merge 😊
- Accelerator Keys
- Primitive segmentation control
- **New!** Mirror & Offset
- Continuity control in all creation commands
- Transform – Polylines & within Extrude
- **New!** Define working regions 😊
- Consistent cage selection intent
- Rationalized selection methods
- **New!** Select Sharp / Weighted edges selection intent
- Consistent de-select automatically
- Polyline preference settings

- Polylines only bodies 😊
- Snapping to cage objects
- Symmetric edge support in fill
- Bridge between faces with open edges
- Delete supports all entity types including edges and vertices

- **New!** Connect to geometry 😊
Demonstration
**New! Convergent modelling😊😊**

- NX design can *work with facets as any other geometry!*  
  - Output from operations is facet

- Enabler of new workflows  
  - Scan 2 Print

- Expand possibility to use NX in more or the process  
  - Direct use of early polygon models  
  - Direct use of scans from tooling of MRO
Revolutionizing the way product shapes are developed

...With the Next Generation Of Advanced CAD Modeling to Production Capabilities

Any design medium: Point Cloud, Polygons, NURBS, ...

...Streamlined workflows *without* conversion!

...3D printing of production parts and tooling

...Additive and subtractive manufacturing processes

...Simulation driven design and optimization

Single Solution from Concept to Production
Topology Optimization for the NX CAD User/Designer

Workflow is designed for the NX CAD user

- Familiar dialogs and workflow.
- Working in component part, or in the context of an assembly (Work Part) making use of the surrounding geometry to define
  - keep in or keep out spaces
  - connections (eg bolt holes)
  - load directions
- Has it’s own boolean tree, so the user creates NX bodies and features without having to do NX boolean operations, even re-ordering for the optimization. These are then identified as Optimization Features.
  - Any mix of solid, facet or primitive bodies can be used.
  - NX SIMPLE HOLE & COUNTERBORED HOLE are treated as special features.
  - Each Optimization Feature has it’s own params, eg blending to control the result.
- NX Material library to assign material properties.
Multiple Loadcases can be applied.

Loads & Constraints are applied to the Optimization Features, for example the SIMPLE HOLE features shown.

To account for loads that need to be applied to a face:

- Use Synchronous cmd Copy Face to create a feature of the required face(s).
- Add this new feature as an Optimization Feature and add an offset thickness.
- A load can now be applied to this feature.
Optimization Setup & Result

Resolution
- Use the slider bar to achieve a balance between Fast/Coarse result and Slow/Fine result.

Optimization Mass Target
- Design Space Mass shown to give the user an indication of a sensible target value.

Overhang Prevention
- Manufacturing constraint to prevent overhang or undercuts in the specified direction.

Result
- Facet body to be used as-is or for guidance in further CAD work.
New! 3D Printing – Requires Windows 8.1 / 10 to test

Project Detail

Planned for NX 11
- Send model directly to 3D printer from NX!
- File > 3D Print
- Uses Microsoft 8.1/10 toolkit

Customer Value
- Supports customer’s needs for 3D printing NX designs
- Supports NX 3D Printing message to the market
Thank you

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