Convergent Modeling and Reverse Engineering
25 October 2017
Siemens is investing in future modeling

- Continuous investment in current and future modeling technologies
- Single application environment for product to production design using any form of geometry
Convergent Modeling™ technology
Revolutionizing product development...
...with the Next Generation of Modeling to Production Capabilities

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

...Streamlined history based workflows without conversion!

...3D printing of production parts and tooling

...Additive and subtractive manufacturing processes

...Simulation driven design and optimization
Convergent Modeling™ technology

Design without conversion

10x faster than traditional methods!
Convergent Modeling™ technology
NX 11.0.0 investment overview

Capabilities
• NX design can work with meshes (facets) as any other geometry!
• Direct use of scan/mesh data without conversion
• History based modeling with mesh geometry
  • Associative wireframe and feature operations (Boolean, Mirror, Scale, …)
  • Analysis (Measure, Weight, FEA, …)
  • Assembly/DMU

Business value
• Accelerate concept to production workflows
• No need for reverse engineering
• No expert knowledge required
Convergent body

- Convergent body = Body of mesh type
- Closed body = Convergent solid body
- Open body = Convergent sheet body
- Comprised of Convergent mesh faces and edges
- Body, Faces and Edges have full properties and attributes
  - Mass, Volume, Area, Length, …
  - Color, Object properties
- Convergent body lives in Part History and partakes in full history based modeling (feature) operations
## Roadmap

**NX 11.0.0 capabilities**

<table>
<thead>
<tr>
<th>import/open</th>
<th>export/save as</th>
</tr>
</thead>
<tbody>
<tr>
<td>JT, STL, VRML + various CAD and Neutral file format that support facets</td>
<td></td>
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</tbody>
</table>

<table>
<thead>
<tr>
<th>conversion</th>
</tr>
</thead>
<tbody>
<tr>
<td>Convert Facet Body: Convert between NX, JT and Convergent mesh formats</td>
</tr>
<tr>
<td>Facet Body from Body: Convert a std. b-rep CAD model to an NX or Convergent mesh body</td>
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</tbody>
</table>

<table>
<thead>
<tr>
<th>cleanup</th>
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<tbody>
<tr>
<td>Snip, Decimate, Subdivide, Smooth, Fill Holes,</td>
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<table>
<thead>
<tr>
<th>std. interactions</th>
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</thead>
<tbody>
<tr>
<td>Selection, Highlight, View Section, Transform, Snap To, Cut/Copy/Paste, Information, Wireframe / Shaded / Rendered Display Modes, Assign Material</td>
</tr>
<tr>
<td>Extrude, Offset Surface, Thicken Body, Extract, WAVE Link, Pattern / Mirror, Projected / Intersected / Sectioned Curves,</td>
</tr>
<tr>
<td>Boolean (Unite / Subtract / Intersect), Scale, Trim / Split, Trim Sheet, Divide Face, Extend Sheet, Offset Face, Shell, Merge (Disjoint / Overlapping / Touching)</td>
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</tbody>
</table>

<table>
<thead>
<tr>
<th>creation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Detect primitives, Paint Facets, Rapid Surfacing, Fit Curves, Fit Surfaces, Refit Face, Extrude (to be retired),</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>operations</th>
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</thead>
<tbody>
<tr>
<td>Measurement – Distance, Angle, Face, Body, …, Mass Properties, Facet Body Curvature,</td>
</tr>
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</table>

<table>
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<tr>
<th>analysis</th>
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<tbody>
<tr>
<td>Object naming and display, Component selection, Find by size, Open by proximity, Replace, Product outline, Reference sets, Advanced weight management, Mirror, Make unique, Clearance analysis, Limited Collision detection (Move component &amp; Assembly sequence, Assembly cut</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>assemblies</th>
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</thead>
<tbody>
<tr>
<td>Send model directly to 3D printer via File &gt; 3D Print (using Microsoft 3D Printing toolkit)</td>
</tr>
</tbody>
</table>
Consider the possibilities
Allowing you to go further with polygons in the early stages of design
Roadmap
NX 11.0.1 accelerated capabilities

- Import/Export
  - STL import / export dialog and functional improvements
- Snap to facet vertex to support design and applications
- Associative Facet Body from Body
- New feature support: Delete Body, New! Combine, Divide Face extension
- Replace Feature accepts convergent input
- Facet alignment commands
  - Best fit
  - Point Set to Point Set
  - Multi-patch
- CAE – Mesh on convergent
- MCD application
- Performance fixes
- Robustness of operators – Imprint, Offset, Extension
- NX Open support
- Analysis tools
  - Draft Analysis
  - Wall Thickness Analysis
  - Deviation Analysis

CAE Analysis on a Convergent Body

Check Wall Thickness
Consider the possibilities
Scan to FEA and back to CAD without reverse engineering!

Facet / mesh geometry

History based modeling on facets

FE mesh for Analysis
Impact of additive design to production processes

- End-to-end support to design, optimize and produce parts using additive manufacturing processes is revolutionizing the way future structures are developed
- Siemens is leading in the development of next generation software and processes to enable this shift
Roadmap

NX 11.0.1 Design for additive checkers

Analysis > Design Validation for Additive Manufacturing
- Check Minimum Wall Thickness
- Check Maximum Overhand Angle
- Check Model within Printable Volume
- Check Wholly Enclosed Volume

NX 11.0.1
- nx_additive_design license
Consider the possibilities
Scan to print

Scan ➔ Analyze / Optimize ➔ Create / Modify ➔ Print
Roadmap
NX 11.0.2 Lightweight structure design

Fill a solid volume with lattices for part light weighting and material reduction while supporting the required loads

License
• nx_additive_lattice...
NX 12.0.0
Workflow focus

Mesh prep for molding / 3D printing
New modeling / Design with meshes
Lightweight structure design
Workflow integration

Modeled in hours / versus days

10x faster with Polygons
10x faster with B-rep
Convergent Modeling™ technology
Update NX 12.0.0 Capabilities

- **Import / Export**: JT, STL, VRML + various CAD and Neutral file format that support facets

- **Conversion**: Convert Facet Body (Convert between NX, JT and Convergent mesh formats), Facet Body from Body – **Associative** (Convert a std. b-rep CAD model to an NX or Convergent mesh body)

- **Cleanup**: **Clean-up Facet Body**, Snip, Decimate, Subdivide, Smooth, Fill Holes

- **Mesh Operators**: Facet Modeling Task Environment, Facet Selection Block, Create Transition, Local Offset, Facet Offset, Fix Undercuts, Adjust Minimum Radius, Create Topology (Divide Facet Faces, Merge Facet Faces)

- **Std. Interactions**: Selection, Highlight, View Section, Transform, Snap To, Cut/Copy/Paste, Information, Wireframe / Shaded / Face Analysis / Rendered Display Modes, **Perspective**, Assign Material

- **Creation**: Extrude w/Offsets, Offset Surface w/Extension, Thicken Body, Extract, WAVE Link, Pattern / Mirror, **Curve on Surface**, Projected / Intersected / Sectioned Curves, **Hole**, **Body by Equation**, Text

- **Operations**: Boolean (Unite / Subtract / Intersect) w/Region Selection, Scale, Trim / Split Body w/Extrusion type, Trim Sheet w/Imprint Completion, Divide Face, Extend Sheet, Offset Face, Shell, Merge (Disjoint / Overlapping / Touching), **Delete Body**, **Combine w/Mixed mode support**, Sew, Edge Blend, **Global Shaping**, **Trim and Extend**

- **Reconstruct CAD**
- **Lattice Design**: Rectangular unit cell lattice, **Surface conformal lattice**

- **Analysis**: Measurements – Distance, Angle, Face, Body, …, Mass Properties, Facet Body Curvature, Draft Analysis, Check Wall Thickness Analysis, Deviation Gauge, **Face Analysis (Radius, Slope, Distance, Reflection)**, AM Wall Thickness, AM Overhang Angle, AM Printable Volume, AM Wholly Enclosed Volumes
Convergent Modeling™ technology
Update NX 12.0.0 Capabilities

Assemblies
Object naming and display, Component selection, Find by size, Open by proximity, Replace, Product outline, Reference sets, Advanced weight management, Mirror, Make unique, Clearance analysis, Limited Collision detection (Move component & Assembly sequence, Assembly cut

Send model directly to 3D printer via File > 3D Print (using Microsoft 3D Printing toolkit)

PMI

Drafting
View Creation Wizard, Base Views, Projected Views, Detail Views, View Break, Section Line, Section View, Linear & Angular Dimensions, Annotation, …

Mesh a Convergent Body, …

CAE

Section View, Linear & Angular Dimensions, Annotation, …

3D Printing

Siemens PLM Software
Polygon Modeling task environment

On the Reverse Engineering tab, select 1 or multiple bodies.

**Capability**

- Dedicated environment for history free editing of convergent/facet geometry.
- Will roll back to timestamp of convergent body if downstream history exists.
- To make history free edits on a history based convergent body, first create a non-associative copy.
Polygon Modeling task environment

Value
• Easy to use, dedicated environment for history free editing within a history based modeler (very much like NX Realize Shape)
Cleanup Facet Body (Analyze)

- 5 Degenerate Facets
- 0 Edges with Folded Facets
- 0 Thin Facets
- 0 Non-Manifold Edges
- 0 Non-Manifold Vertices
- 3 Intersection Facets
- 0 Inconsistent Normals
- 0 Laminar Slits

Use Show Results to Preview the Analysis
Cleanup Facet Body (Repair)

Use Show Results to Preview the Repair

0 Degenerate Facets
0 Edges with Folded Facets
0 Thin Facets
0 Non-Manifold Edges
0 Non-Manifold Vertices
0 Intersection Facets
0 Inconsistent Normals
0 Laminar Slits
Facet Selection

**Single Facet**
- Single
- Rectangle
- Circle
- Lasso
- Polygon

**Rough Brush**
Selects within and crossing the brush

**Fine Brush**
Selects within the brush

**Color Region**
Fills within a selected boundary

**Face Facets**
All facets of face

**Employed in:**
- Snip
- Decimate
- Subdivide
- Smooth
- Paint Facet Body
- Divide Facet Faces
- Adjust Minimum Radius
- Local Offset
Mesh Prep for Molding / 3D Printing

Create Transition

Fix Undercuts

License
- nx_polygon_modeling license
Mesh Prep for Molding / 3D Printing

Local Offset

Adjust Minimum Radius

License
• nx_polygon_modeling license
New Modeling / Design with Meshes

Global Shaping
New Modeling / Design with Meshes

Hole
Lightweight structure design

Surface conformal lattice
• Create lattices that follow the design form and supports the required loads

License
• nx_additive_lattice...
Workflow Integration

PMI and Drafting Documentation
Engineering driven Design with Symbolica
Workflow Integration

Creation of Topology for FEA and CAM

License
- nx_polygon_modeling license
Divide / Merge Facet Faces

License
- nx_polygon_modeling
Apply organic shape onto product CAD model
Apply textures and graphics onto product CAD model

Not Texture Mapping: 3D Shape
Morph large, complex CAD model

Huge, Normal CAD Model
Morphing needs long time.

Convert CAD Model to JT Facet

Morph JT Facet

Apply Same Morphing onto Normal CAD Model

Continue “Normal” CAD Operation

Convert JT Facet to Convergent Model
Demo
Reverse Engineering Workflow

- STL
- Align
- Fit Surface
- Outside Body
- Inside Body
- Details
- Compare
- Add Blends
- Edge Model
- Done!
Underlying Technology – Shape Detection

Analytic Based
- Plane, Cylinder, Cone, Sphere Curvature
- Concave, Convex
Underlying Technology – Surface Fitting

- Analytic Fitting
- Freeform Fitting
- N-Sided Fitting

Rapid Surfacing
Leverage the power of NX Modeling in the RE workflow

Combining Sheets

Synchronous Rotate

Replace Face

Corner Blending

Synchronous Offset
Deviation Gauge – Compare As Modeled to As Scanned / Printed

± 0.05 [mm]  ± 0.02 [mm]  ± 0.01 [mm]
Morphing

Product CAD Shape Morphing
• Create Compensated CAD Model to Polygon.
• Fit CAD Model to Polygon.
• Based on CAE Analysis of Spring back or Die Shrinkage or Real Physical Product Scan Data

Business Value
• Modify “As Designed” to reflect changed needed to account for manufacturing process variation
• Modify “As Designed” to reflect how parts are being successfully produced on the shop floor
## NX Reverse Engineering – Working Time on Before/After NX 10

<table>
<thead>
<tr>
<th>Model Shape Overview</th>
<th>Model A</th>
<th>Model B</th>
<th>Model C</th>
<th>Model D</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>210 Features</td>
<td>240 Features</td>
<td>295 Features</td>
<td>2700 Features</td>
</tr>
<tr>
<td></td>
<td>286 Face</td>
<td>286 Faces</td>
<td>179 Faces</td>
<td>2852 Faces</td>
</tr>
<tr>
<td>Working Time, Before</td>
<td>1 Week</td>
<td>1 Week</td>
<td>1 Week</td>
<td>3 Weeks</td>
</tr>
<tr>
<td>Customer Estimation</td>
<td></td>
<td></td>
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</tr>
<tr>
<td>Working Time w/NX RE</td>
<td>Half Day: 2 or 3 hours</td>
<td>Half Day: 2 or 3 hours</td>
<td>Half Day: 3 or 4 hours</td>
<td>4 days, 30 hours</td>
</tr>
</tbody>
</table>
Demo
Thank you.