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Realize innovation.

What's New in NX 11

Siemens PLM Software
Senior Sales Support Engineer
陈军

NX 11 for Design What's New

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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
Support Remote Display
Support for New Direct Model Version
NX Open Selection Support
Navigator Selection Performance
Circle Selection Method
Selection Tune Up
Mixed Modeling PSM Display Support
Point Cloud - Architecture Support
Plot Support for Colored Drawing Symbols
NX Open Toolkits Upgrades
NX Open Usability
Python UDO Support
NX Open Partner Support
NX Open Support for Spreadsheet Interaction
Timeline Tool
Information Window
Navigator Drag and Drop
Improved Window Management
Automatic Application Change
Upgrade QT

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Design Templates - WAVE link out of date status improvements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 - Spine Curve feature
Wireframe - Scale Curve feature
Wireframe - Isocline curve feature UI & Selection Intent
Surfacing - Variable offset surface (Tailored blanks)
Surfacing - Flattening and Forming
Surfacing - Studio Surface improvements
Surfacing - Section Surface accuracy
Wireframe - Derived curves improvements
Realize Shape - Connect
Realize Shape - Polish
Analysis - Multi-Face Continuity
Mixed NURBS/Facet modeling
Analysis - AO Performance improvements
Analysis - AO Usability improvements
Analysis – AOs in Assembly Context

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

PMI

2D Drawing to 3D PMI
Part Navigator Performance Improvement
PMI Inherit Enhancements
Support for Angular Directed Dimensions
PMI Lightweight Section View Enhancements
PMI Region Enhancements
PMI Resize Enhancements
PMI display parallel to screen
Dimension UI Enhancements
Hole Callout enhancements
Dimension Settings
Limits and Fits
Support for Angular Directed Dimensions
Selection for Routing Anchor Points

Assemblies

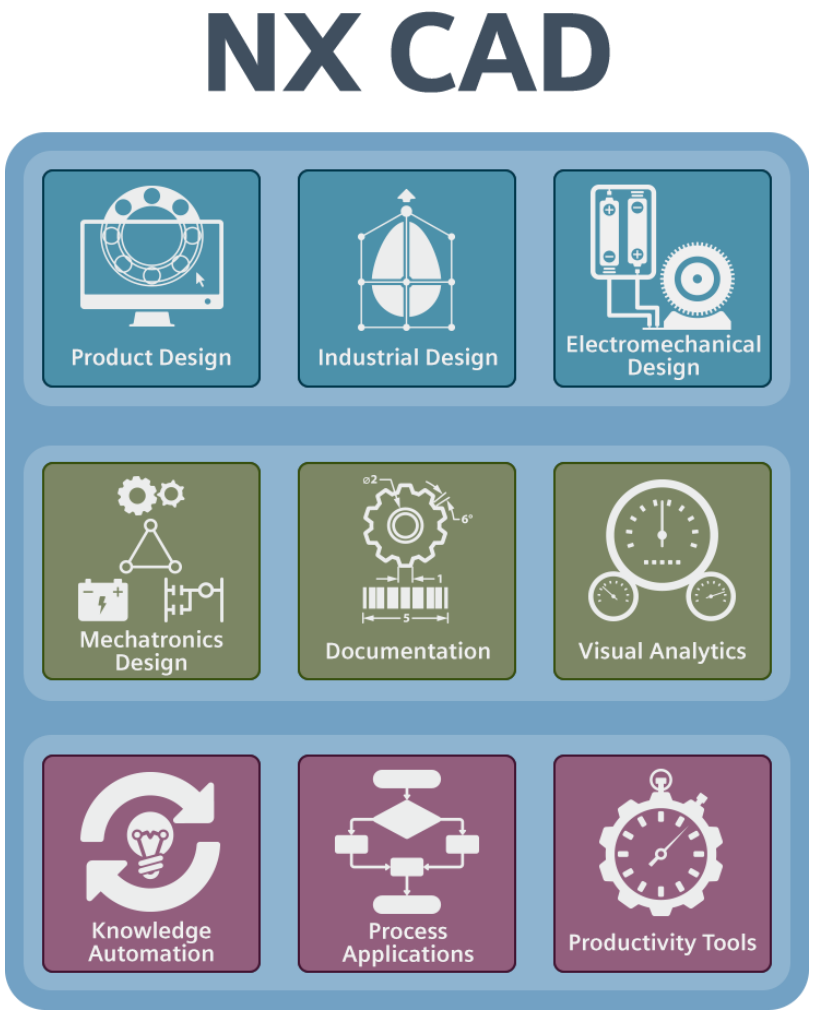
4GD Weld Support (NX 10.0.2)
4GD Constraints (Phase 2)
4GD Support for Change Management
4GD Support for Parts List
Direct Model Integration
Large Assembly Performance
Motion Consolidation Architecture
Component Patterns with Pure Reference Patterns
Support for Multiple Displayed Parts Investigation
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)

NX11 CAD概述

NX 11 提供了一组强大的新工具，并显著增强了现有的功能。旨在帮助您快速设计、测试并制造产品，同时将错误数量降至史上最低。在NX 11 中，多项之前需要冗长手动返工的任务现已经实现了自动化。借助收敛建模，您可以在单个模型中使用小平面几何体和实体/曲面几何体。



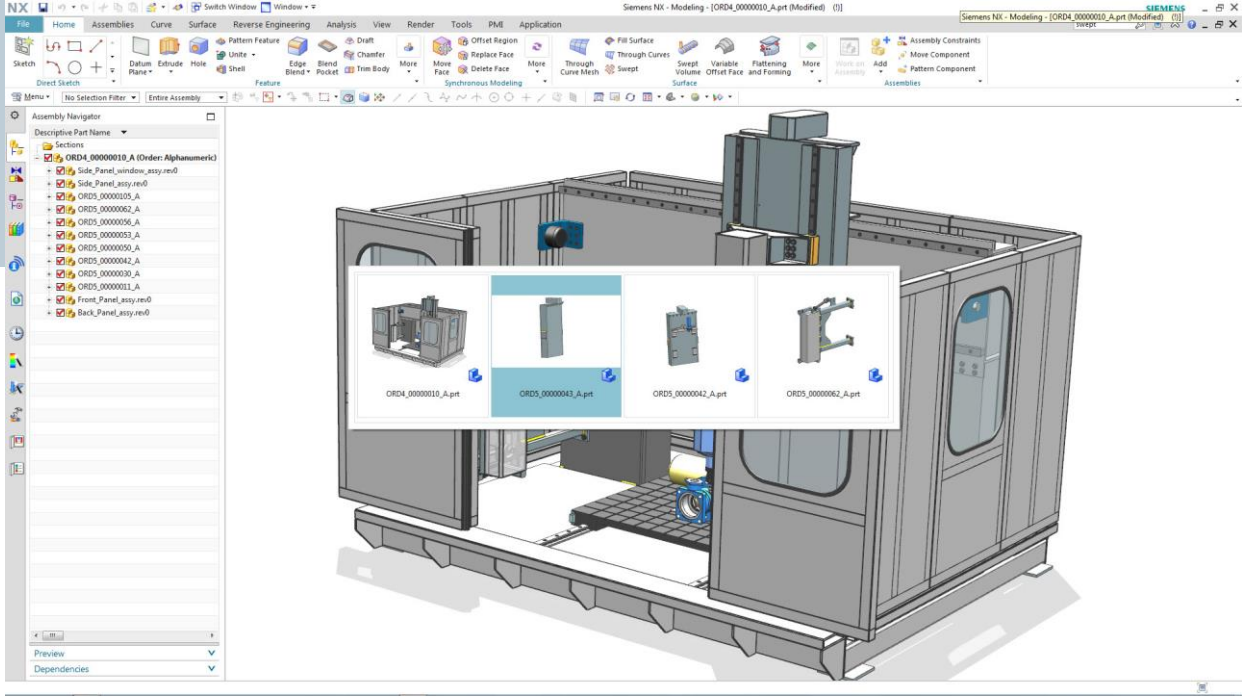
- 通过收敛建模，将扫描的 3D 数据作为小平面导入
- 利用改进后的 3D 打印支持轻松创建原型
- 设计特征的方式与通过扫掠体积进行加工的方式相同
- 改进了草图绘制中的控制、更改和信息访问
- 通过 NX 创意塑型和 NX 布局简化了概念设计
- 自动将图纸对象转换为 3D PMI 对象
- 通过图纸比较减少了图纸检查时间
- 通过 Lightworks Iray+ 可以实现更逼真的可视化和渲染
- 通过直接访问而非重新创建轻松利用点云数据
- 通过在云端管理 NX 提高了灵活性

Usability
易用性

Easier to use so you can get the job done faster

每一版本的 NX 均在上一版本的基础上提高了速度、效率并简化了操作。

- 在 NX 11 中，只需按下 CTRL+Tab组合键即可快速直观地切换窗口，并且窗口中会始终显示所有活动零件的缩略图。
- 通过 HTML 信息窗口，现在可以更方便地读取和导航到所需的零件详细信息。
- 改进后的触屏功能和4K 监视器支持，使 NX 的操作更为简便，帮助您实现高效作业。
- 更简洁实用的新公式界面
- 交互和执行效率的提升



Information

Information listing created by	Iam
Date	26-Aug-2015 10:56:12
Current work part	D:\Parts\Information_Window\NX3_pump\pump assembly.prt
Node name	cyi6w515

All Parts in Session Listing

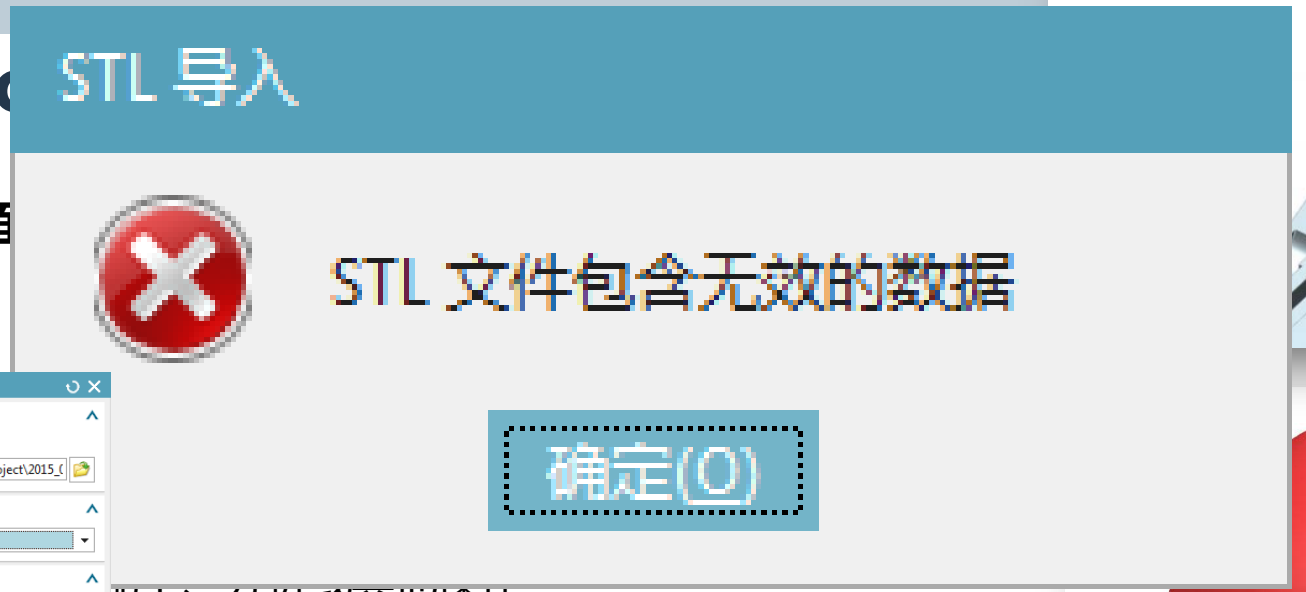
Part File Name	Part Name	Loaded	Modified	Read-only	Level	Directory	Description	Units	Status	Format	Release	Created Time
Info Window Features.prt	Info Window Features	Fully Loaded	✗	✗	Piece Part	D:\Parts\Information_Window		Millimeters	0	NT x64	NX V10.0	29-Apr-2015 14:50
bolt.prt	bolt	Partially(1)	✔	✔	Component	D:\Parts\Information_Window\NX3_pump	cnm-hexagon flange head bolt m8x35 - item:53-09110	Inches	1	NT Intel	NX V2.0	16-Apr-2004 17:21
con_rod.prt	con_rod	Partially(1)	✔	✔	Component	D:\Parts\Information_Window\NX3_pump		Inches	0	NT Intel	NX V2.0	19-Sep-1994 08:30
con_rod_assy.prt	con_rod_assy	Partially(1)	✔	✔	Sub-Assy	D:\Parts\Information_Window\NX3_pump		Inches	0	NT Intel	NX V7.5	16-Apr-2004 15:14
con_rod_cap.prt	con_rod_cap	Partially(1)	✔	✔	Component	D:\Parts\Information_Window\NX3_pump		Inches	0	NT Intel	NX V2.0	16-Apr-2004 15:16
con_rod_pin.prt	con_rod_pin	Partially(1)	✔	✔	Component	D:\Parts\Information_Window\NX3_pump		Inches	0	NT Intel	NX V2.0	16-Apr-2004 15:20
crank_case_assy.prt	crank_case_assy	Partially(1)	✔	✔	Sub-Assy	D:\Parts\Information_Window\NX3_pump		Inches	0	NT Intel	NX V2.0	16-Apr-2004 15:53
crankcase.prt	crankcase	Partially(1)	✔	✔	Component	D:\Parts\Information_Window\NX3_pump		Inches	0	NT Intel	NX V2.0	20-Sep-1994 07:26
crankshaft.prt	crankshaft	Partially(1)	✔	✔	Component	D:\Parts\Information_Window\NX3_pump		Inches	0	NT Intel	NX V7.5	19-Sep-1994

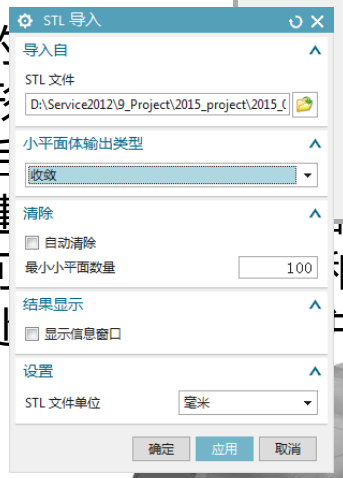
Ctrl-Tab 窗口切换功能可以节省时间和精力。

Convergent Modeling
收敛建模

Intelligent sc

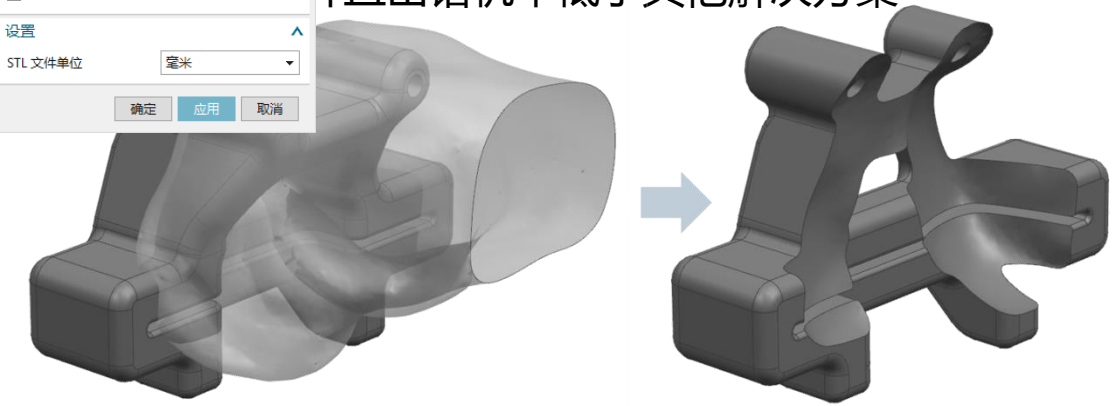
在同一个模型中的小面
易出错的逆向工程

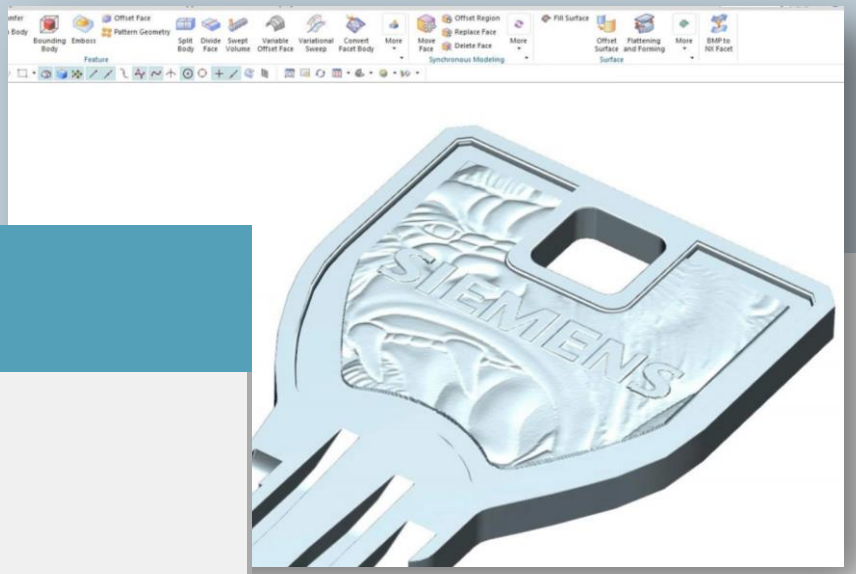


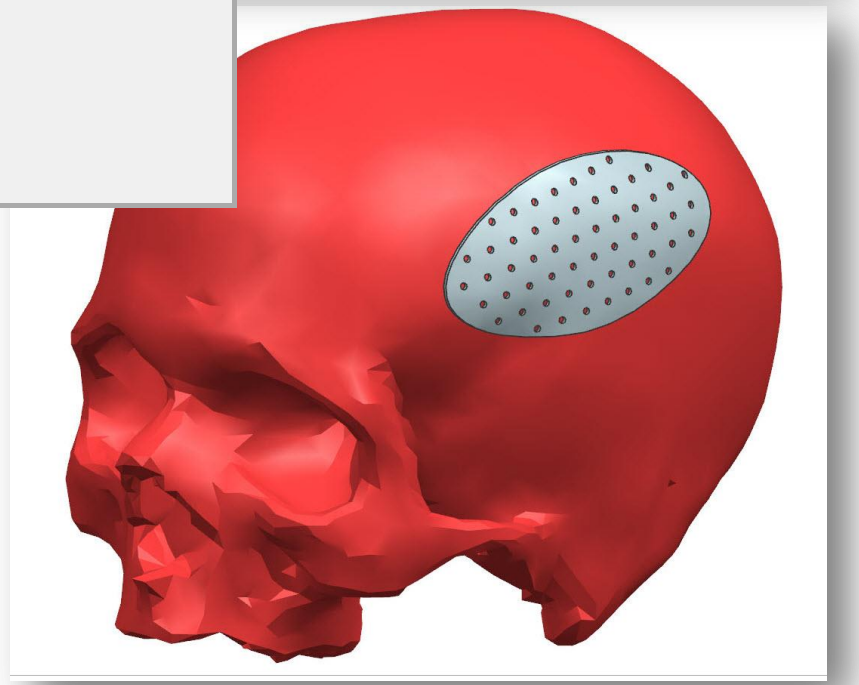


- 是将扫描的
  任何其他形
- 扫描数据后
  据形状创建
- 收敛建模可
  快的速度处

和成本，并消除了容易出错的返工环节，可以更
并且出错机率低于其他解决方案







可以在医疗行业中使用收敛建模，在本例中将其用于颅骨植入。

Mixing facet and solid/surface data in a single model

Convergent Modeling Technology
Revolutionize the way product shapes are developed

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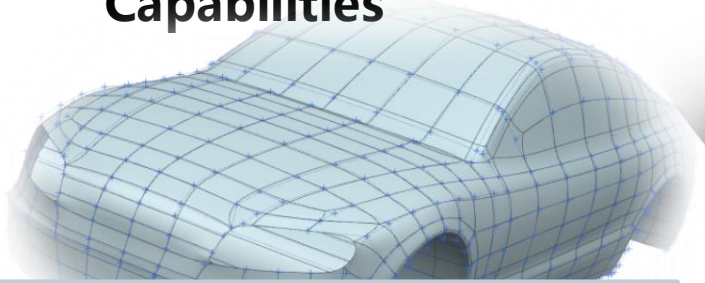
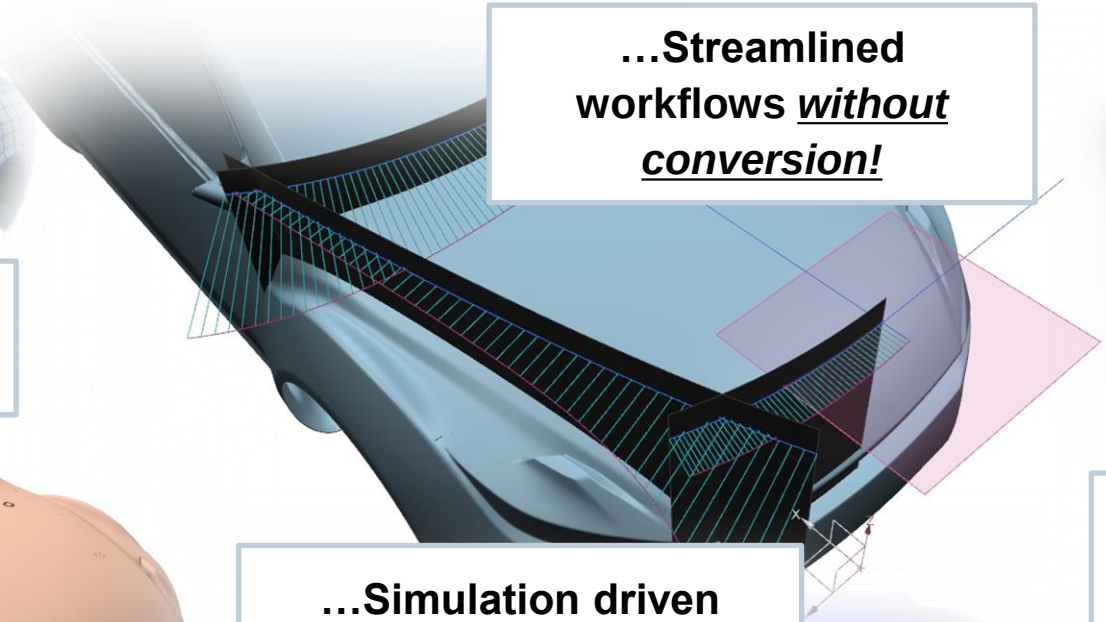
…with the Next Generation Of Advanced CAD Modeling to Production Capabilities

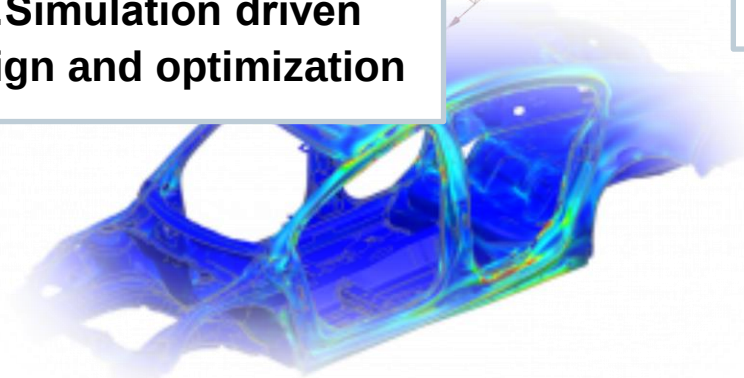


Any design medium: Point Cloud, Polygons, NURBS, …

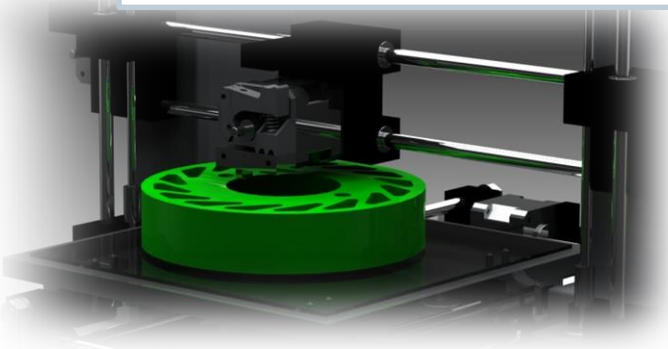
…Streamlined workflows *without conversion!*



…Simulation driven design and optimization



…3D printing of production parts and tooling



…Additive and subtractive manufacturing processes

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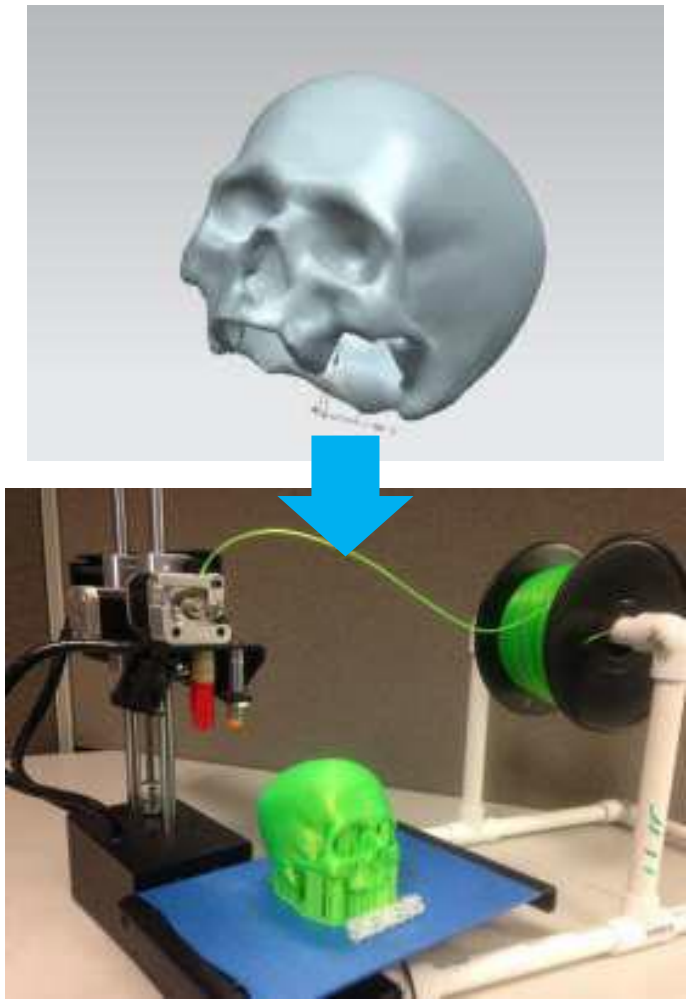
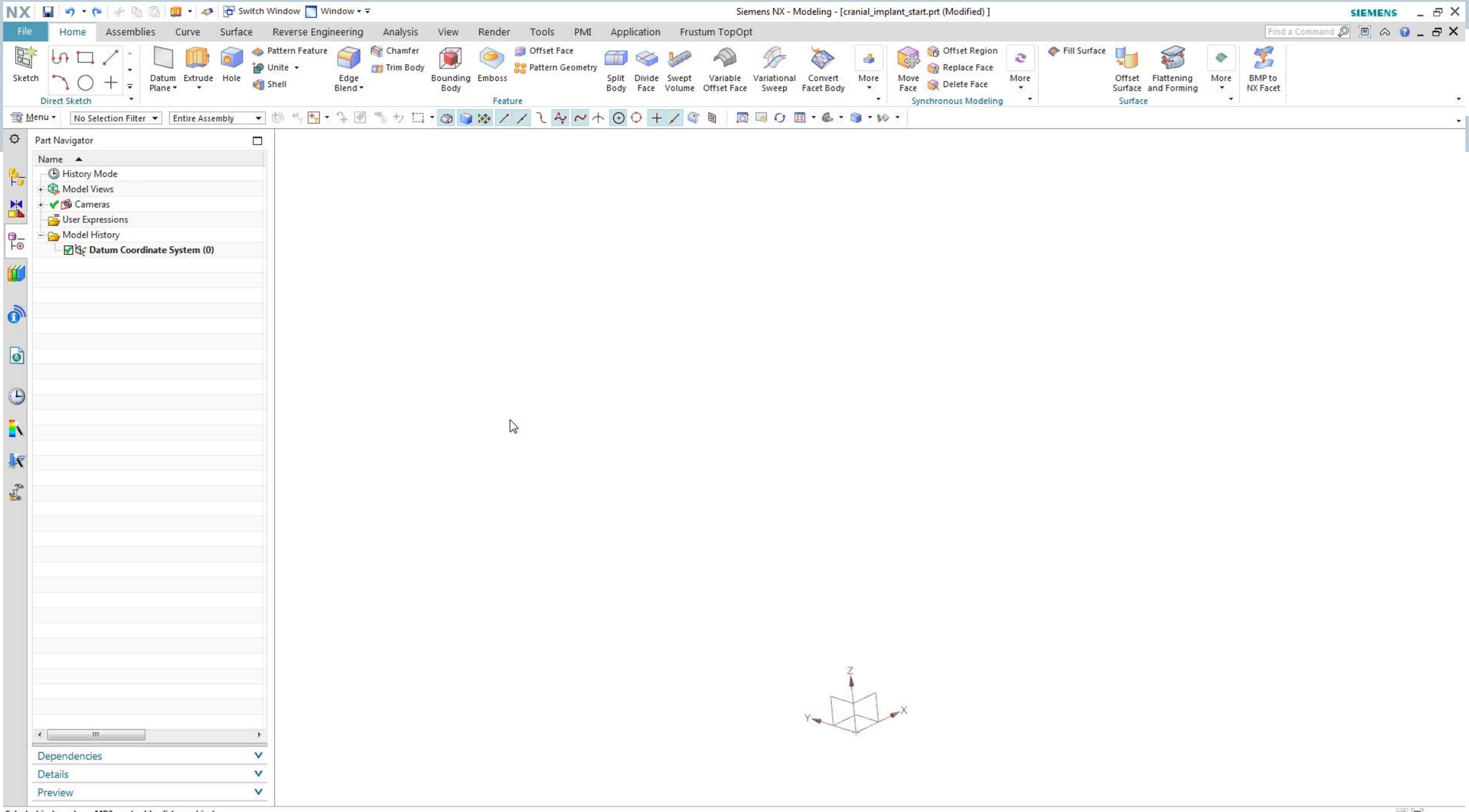
Rapid manufacturing directly from NX

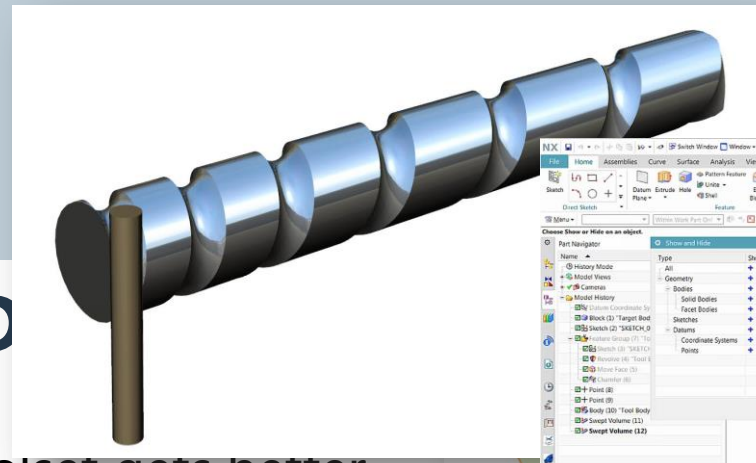
Easily create prototypes or final products

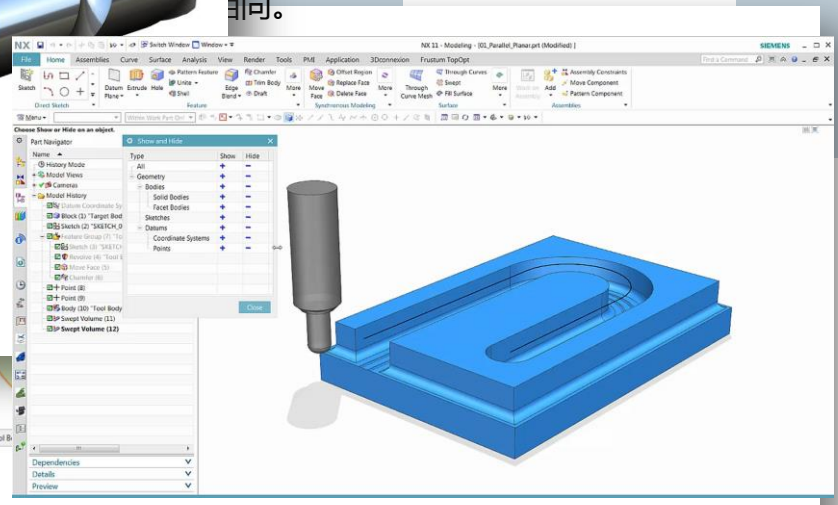
- 可以通过“文件”>“3D 打印”直接从NX 对设计进行 3D 打印
- 此功能使用 Microsoft® Windows® 提供的 3D 打印工具包
- 并采用广受支持的 3D Manufacturing Format(3MF)，因此具有广泛的兼容性。





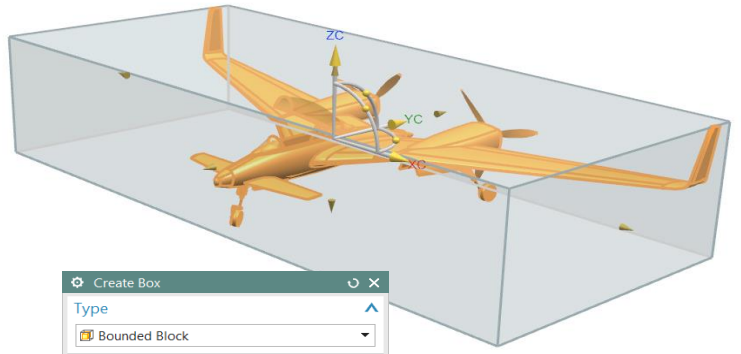
The foundation of modern 3D

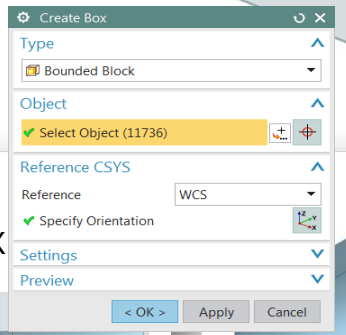


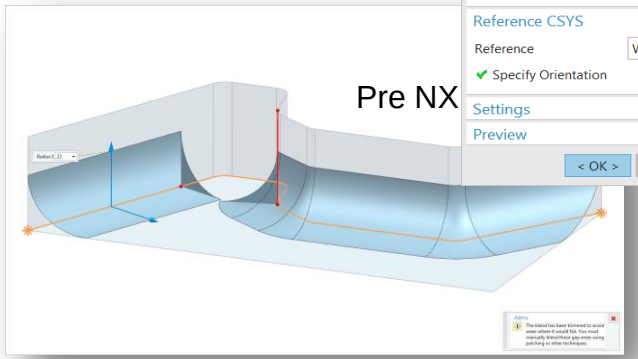


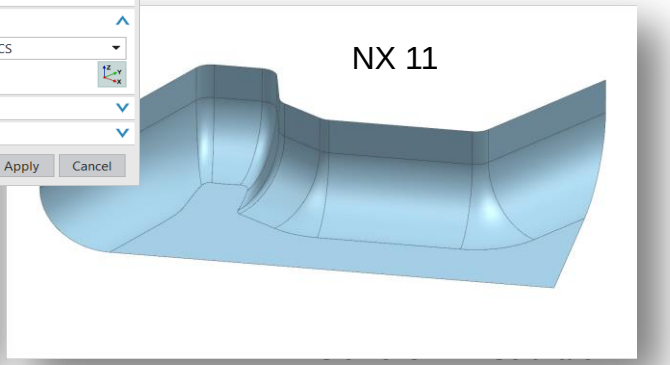
A state-of-the-art parametric solid modeling toolset gets better with new tools and improved usability

- 新增了扫掠体积等工具
  - 用于创建关联包容体的简单新工具尤其适用于工装和夹具设计
- 包络体建立（ Bounding Body ）
- 倒园
  - 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









NX 11 - Modeling - [01_Parallel_Planar.prt]

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File | Home | Assemblies | Curve | Surface | Analysis | View | Render | Tools | PMI | Application | 3Dconnexion | Frustum TopOpt

Find a Command

Sketch | Direct Sketch

Datum Plane | Extrude | Hole | Pattern Feature | Unite | Shell | Edge Blend | Chamfer | Trim Body | Draft | More | Feature

Move Face | Offset Region | Replace Face | Delete Face | More | Synchronous Modeling

Through Curve Mesh | Through Curves | Swept | Fill Surface | More | Surface

Work on Assembly | Add | Assembly Constraints | Move Component | Pattern Component | Assemblies

Menu | No Selection Filter | Within Work Part and

Part Navigator

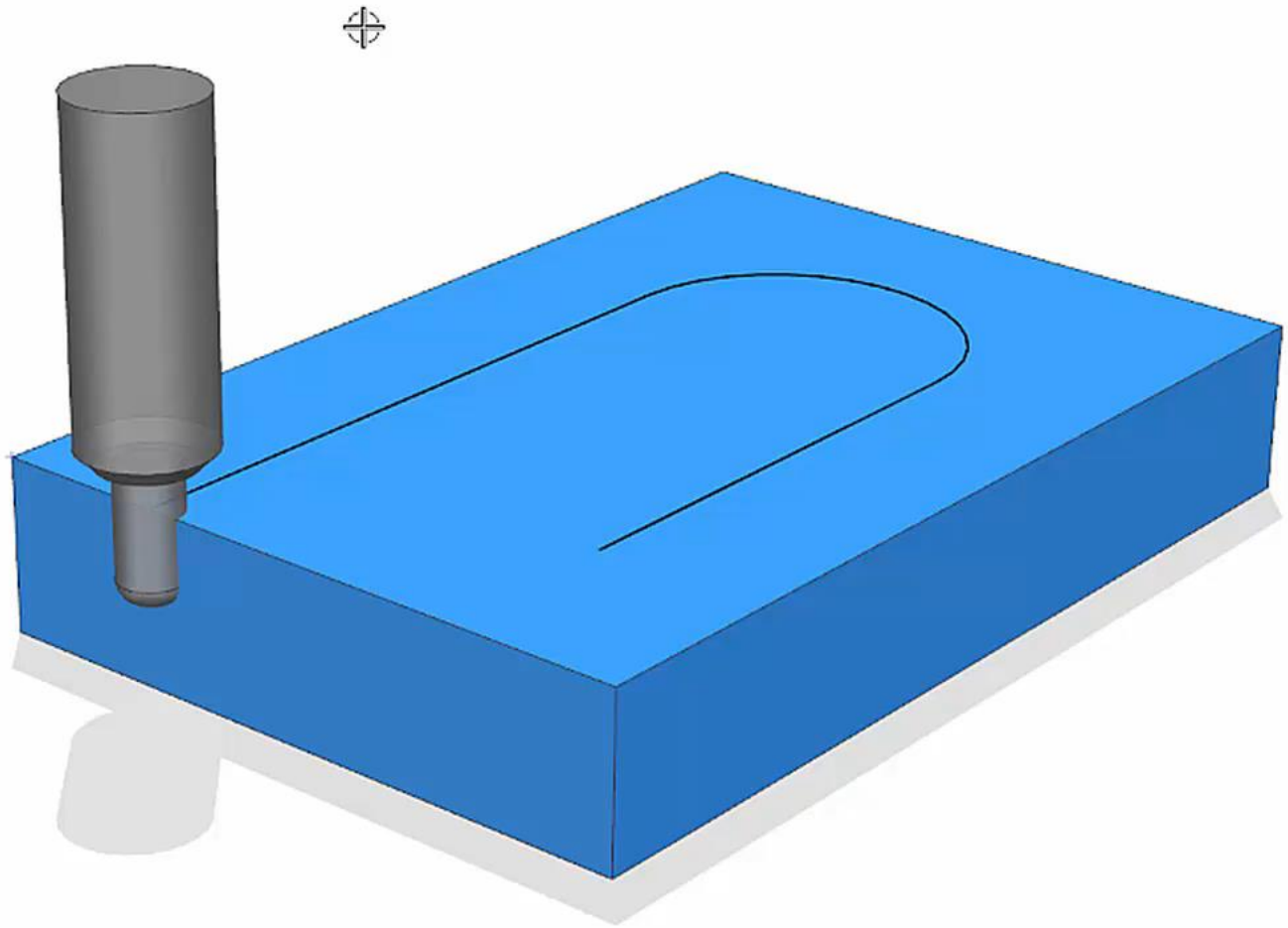
Name

- History Mode
- Model Views
- Cameras
- Model History
  - Datum Coordinate System (0)
  - Block (1) "Target Body or Edges as path"
  - Sketch (2) "SKETCH_000:Tool Path Planar"
  - Feature Group (7) "Tool Body 1"
  - Point (8)
  - Point (9)
  - Body (10) "Tool Body 2"

Dependencies

Details

Preview



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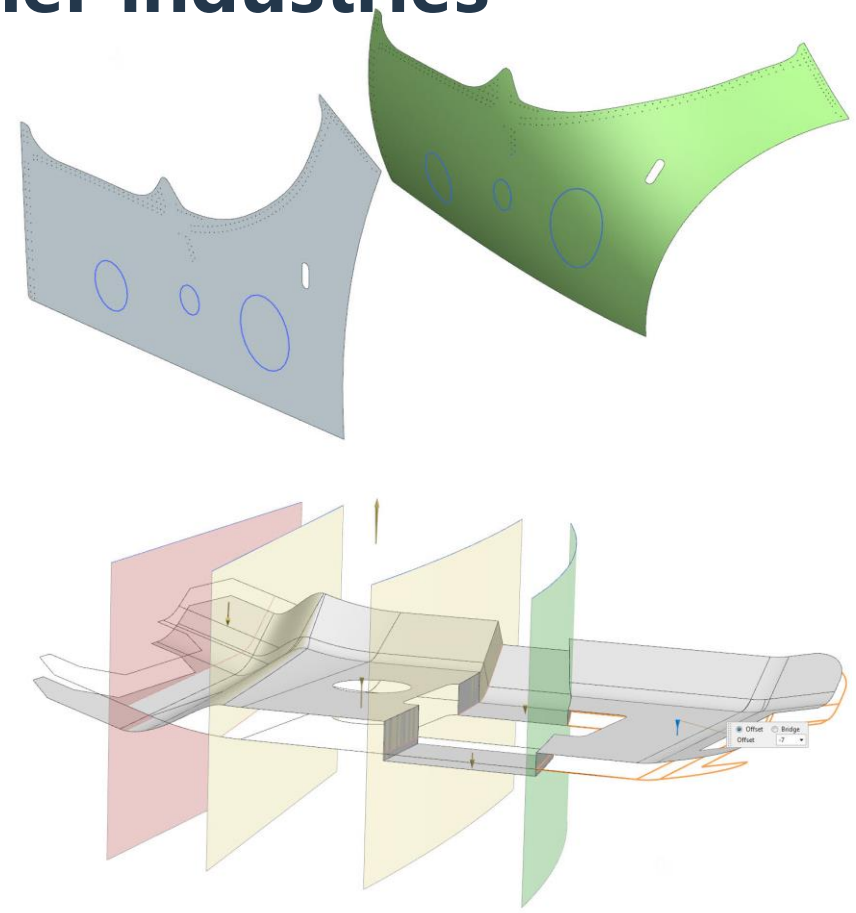
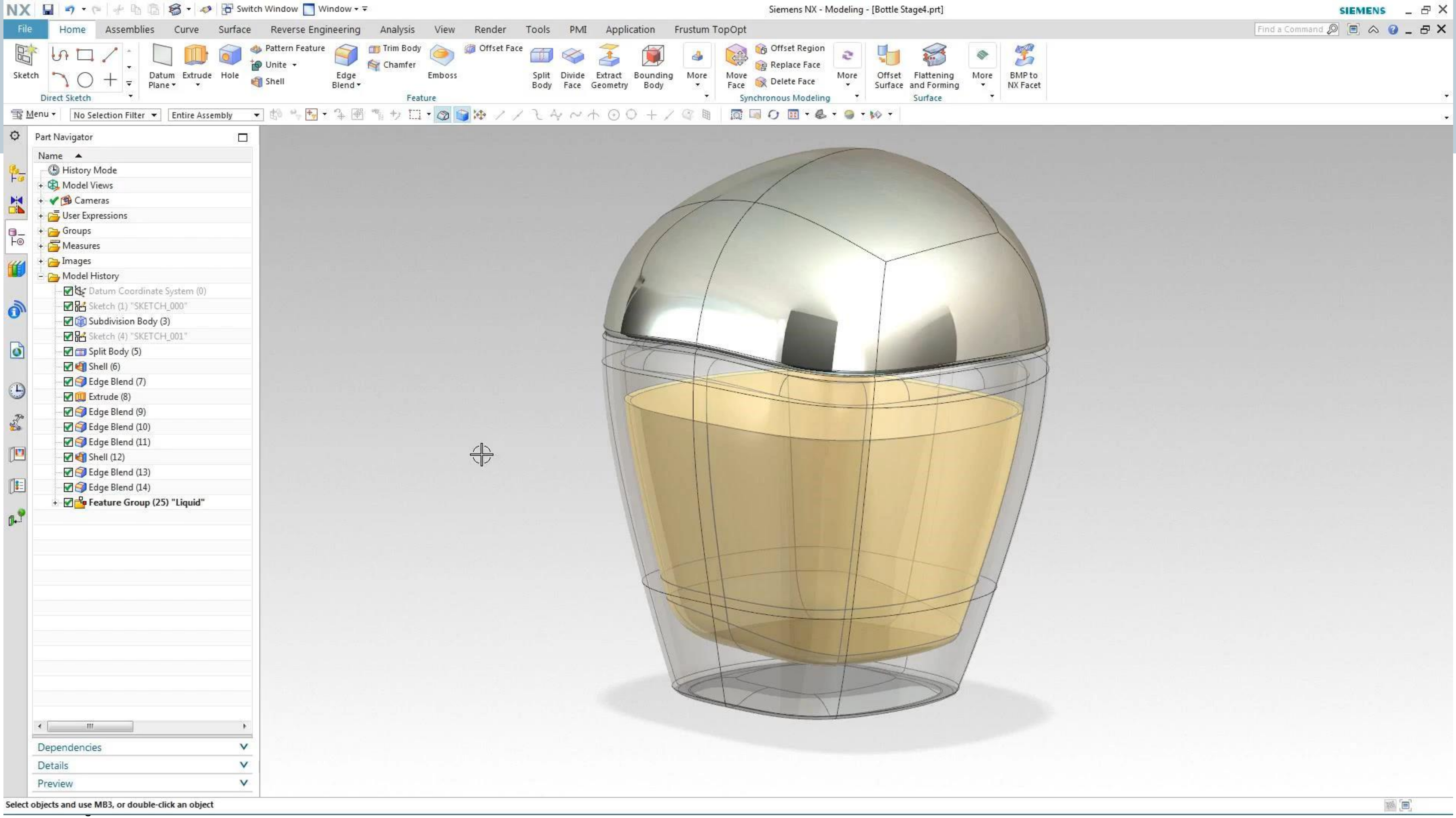
A vital tool in industrial design and other industries

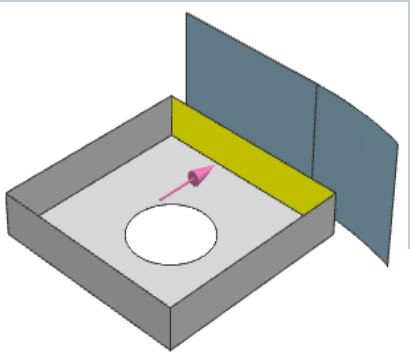
曲面建模仍然是大多数行业中使用的重要工具，尤其适用于工业设计和造型。NX 11 进一步强化了功能强大的曲面设计工具集，可以提供新型和优化功能。

- 曲线缩放：利用曲线缩放功能，您可以将缩放后的曲线、零件边或点副本创建为新特征，并能使其与原始曲线关联。
- 曲面展平：现在您可以展平曲面，对曲面进行更改并将其重塑至原始形状，从而进一步控制所需的工作流。Forming and reuse
- 可偏置曲面：借助 NX 11 中的可变偏置面，您可以在单个操作中创建具有固定厚度和可变厚度的体，这对于轻量级操作尤其有用。
- 此外还改进了变化扫掠，只需几步即可轻松创建复杂形状。
- Variational Sweep improvements
- Trim Sheet improvements
- *New!* Variable Offset Face
- *New!* Swept Volume
- *New!* Flattening and Forming toolset



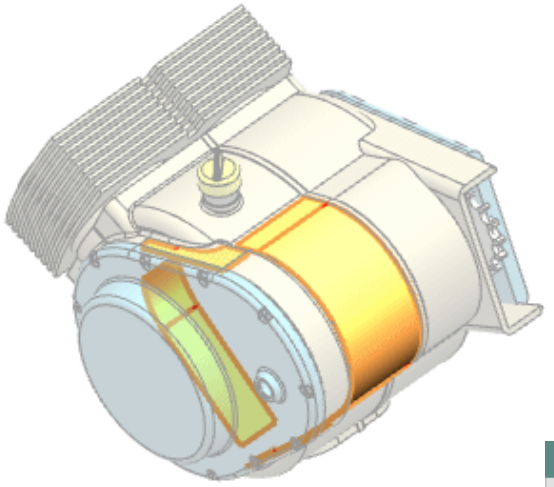
在 NX 11 中展平、修改和重塑曲面。

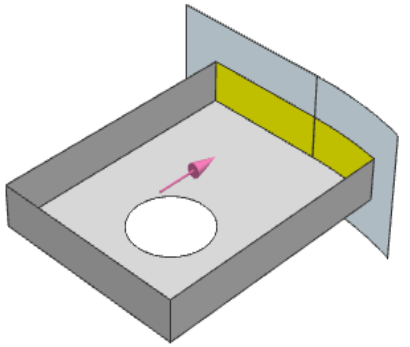

Siemens NX - Modeling - [Bottle Stage4.prt]
SIEMENS
Switch Window
Window
File
Home
Assemblies
Curve
Surface
Reverse Engineering
Analysis
View
Render
Tools
PMI
Application
Frustum TopOpt
Find a Command
Sketch
Direct Sketch
Datum Plane
Extrude
Hole
Pattern Feature
Unite
Shell
Edge Blend
Trim Body
Chamfer
Emboss
Offset Face
Feature
Split Body
Divide Face
Extract Geometry
Bounding Body
More
Move Face
Offset Region
Replace Face
Delete Face
Synchronous Modeling
Offset Surface
Flattening and Forming
Surface
BMP to NX Facet
Menu
No Selection Filter
Entire Assembly
Part Navigator
Name
History Mode
Model Views
Cameras
User Expressions
Groups
Measures
Images
Model History
Datum Coordinate System (0)
Sketch (1) "SKETCH_000"
Subdivision Body (3)
Sketch (4) "SKETCH_001"
Split Body (5)
Shell (6)
Edge Blend (7)
Extrude (8)
Edge Blend (9)
Edge Blend (10)
Edge Blend (11)
Shell (12)
Edge Blend (13)
Edge Blend (14)
Feature Group (25) "Liquid"
Dependencies
Details
Preview
Select objects and use MB3, or double-click an object



Industry-leading technology

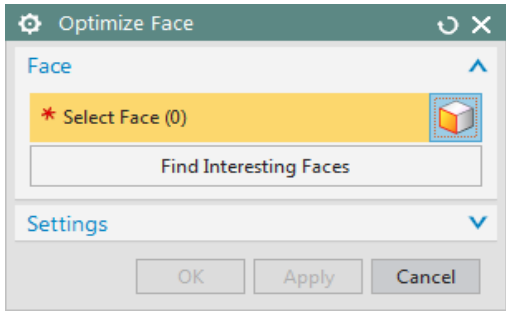
Incremental improvements to the industry’s best tools for modifying models regardless of where they came from, yielding groundbreaking speed and freedom of design

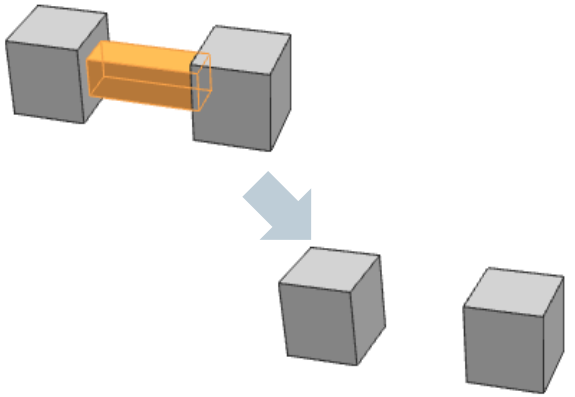


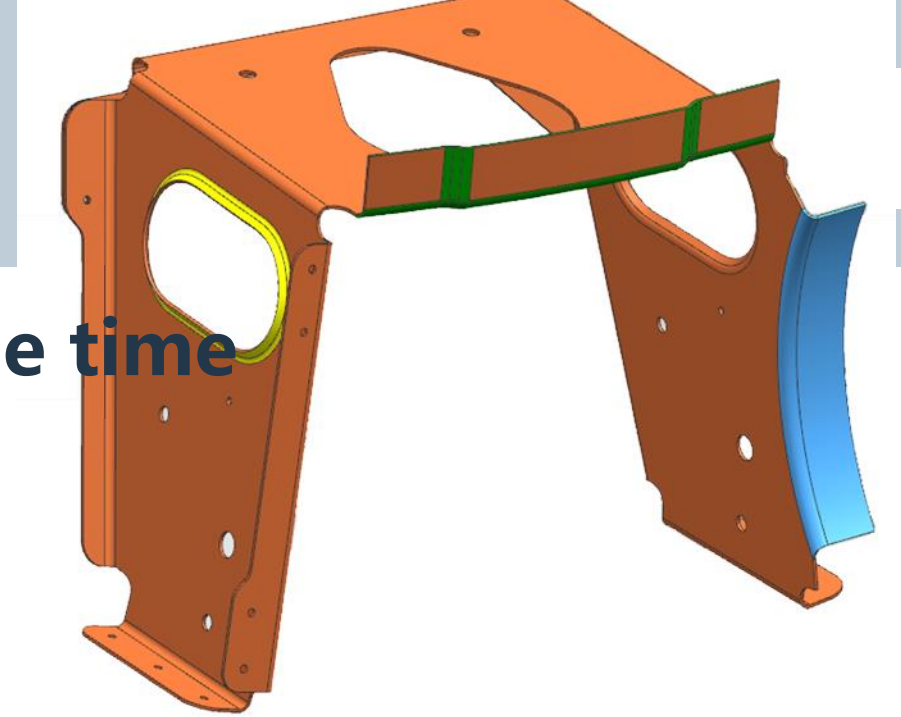


- 利用该项技术，您可以处理任何来源的模型，并能将智能融入生硬的数据，NX 中的同步建模技术处于行业领先水平。

  - NX 11 中的增量改进功能可将面替换为包含开放边的其他面。
  - 使用自动面选择功能轻松对面进行优化。
  - 可以通过删除面将单个体分成多个体。



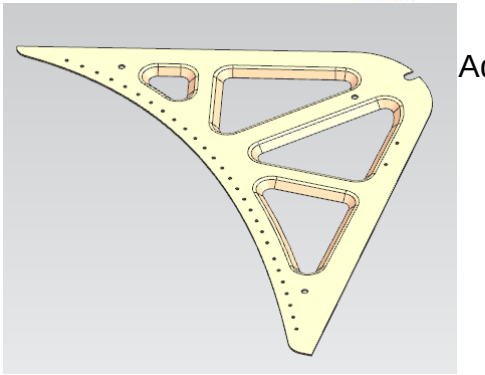




Simpler and more powerful at the same time

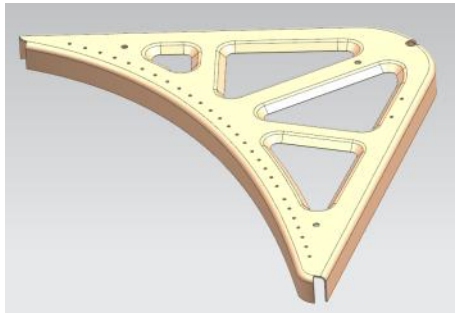
By combining functions and adding new options to existing ones, sheet metal is easier to use and more capable than ever, enabling you to create the sheet metal geometry you need in fewer steps

- 航空航天钣金已退休。在 NX11 中，命令已重新编写、改进并合并到 NX 高级钣金中。
- 基础钣金架构中现已包含高级法兰的创建，提高了可靠性并改善了与其他钣金命令的兼容性。
- 在 NX 11 中，桥接折弯适用于具有标准钣金许可证的所有用户。
- 可以利用新增的对中间腹板面和相邻腹板面的支持功能创建更加复杂的榫接。
- 借助减轻开孔、实体冲压、桥接折弯、凸起和凹坑、法向开孔以及轮廓弯边等的增强功能，您可以提高灵活性并增强控制，减少创建所需几何体的步骤。

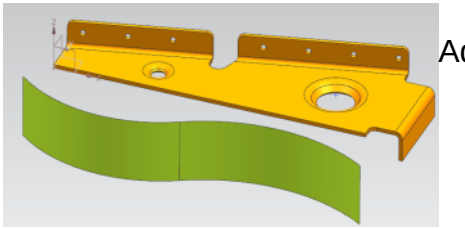


Advanced Flange

By Value

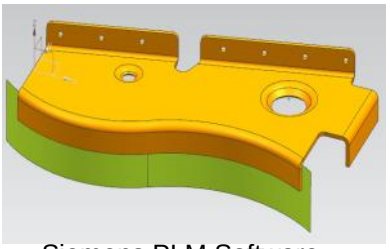


利用 NX 11 的钣金设计功能，可以减少创建所需几何体的步骤。



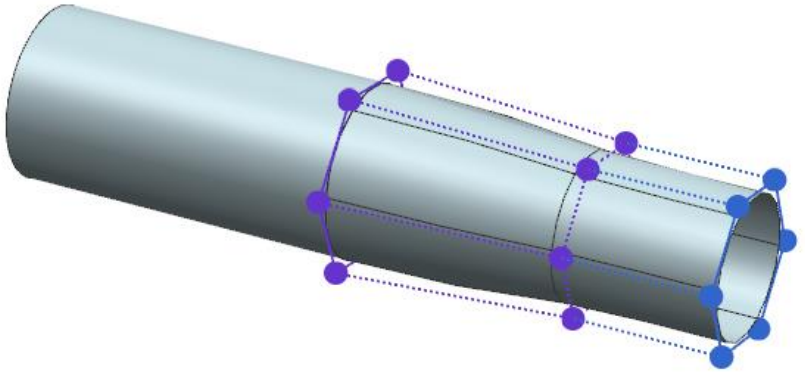
Advanced Flange

To Reference



SIEMENS

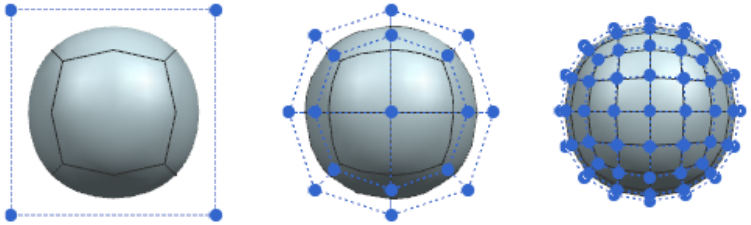
Easy 3D concept design



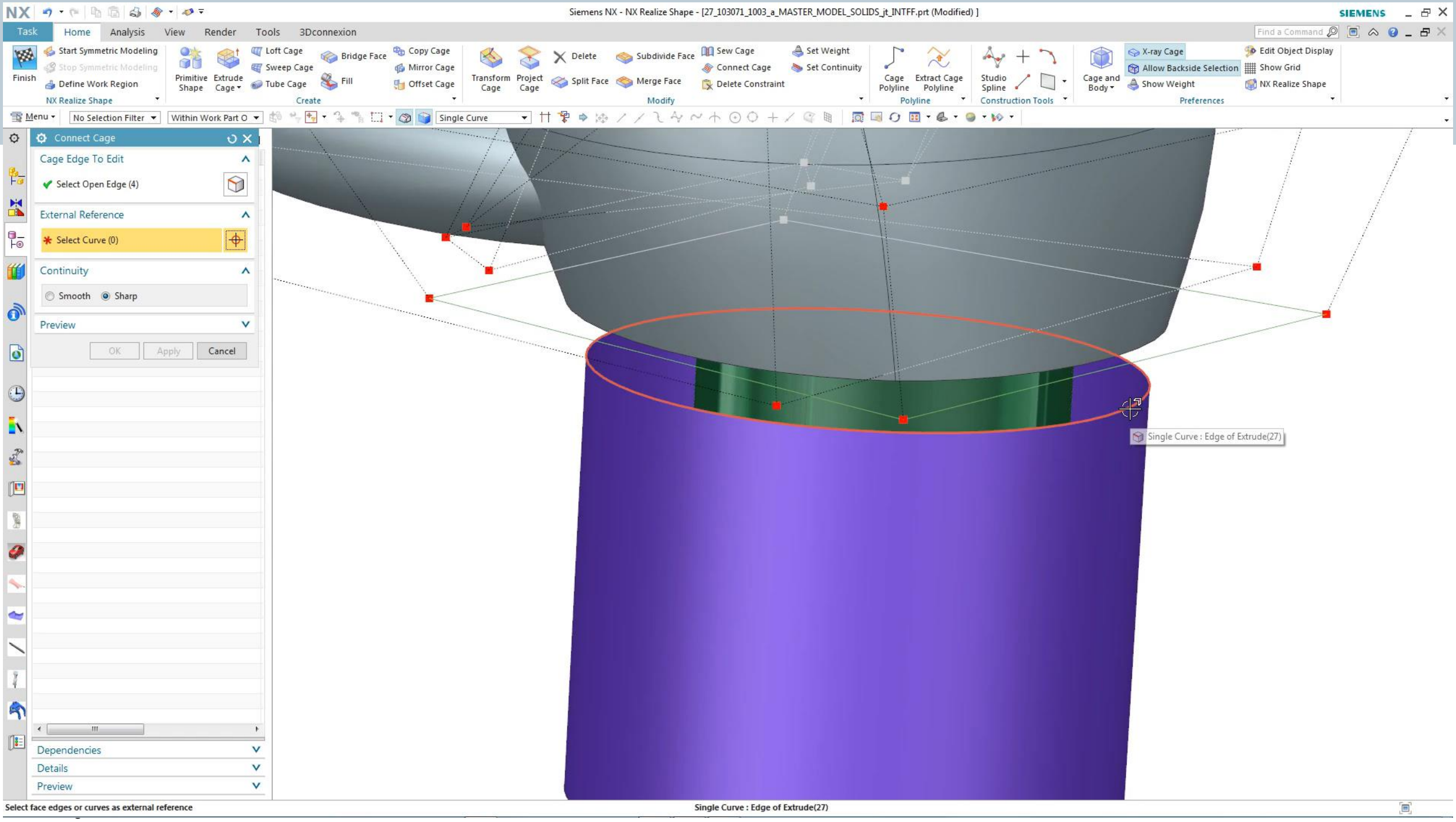
An intuitive subdivision design tool that uses simple shape manipulation to create high-quality B-surfaces with continuous improvements in functionality and ease-of-use

自从在 NX 9 中面世以来，NX 创意塑型™ 不断进行改进和优化。NX 创意塑型是一款易于使用且功能强大的直观型细分设计工具，可通过简单的形状操控创建高质量的 B 曲面。

进一步控制细分模型的元素，以便使用任何模型中的细分形状



- 在 NX 11 中，可以对框架进行拆分和合并体，
- 在创建体素形状时，您现在可以指定框架中的水平段和垂直段的密度与数量
- 凭借删除框架中的某一部分，移动或复制体中单个元素的功能
- 对框架的镜像，偏置
- 对框架的定义工作区域（局部控制）和冻结
- 您还可以利用新增的连接功能，填充体（非框架）之间的任何间隙


Siemens NX - NX Realize Shape - [27_103071_1003_a_MASTER_MODEL_SOLIDS_jt_INTFF.prt (Modified)]
SIEMENS
Task
Home
Analysis
View
Render
Tools
3Dconnexion
Find a Command
Start Symmetric Modeling
Stop Symmetric Modeling
Define Work Region
Finish
NX Realize Shape
Primitive Shape
Extrude Cage
Loft Cage
Sweep Cage
Tube Cage
Bridge Face
Fill
Copy Cage
Mirror Cage
Offset Cage
Create
Transform Cage
Project Cage
Delete
Split Face
Subdivide Face
Merge Face
Sew Cage
Connect Cage
Delete Constraint
Set Weight
Set Continuity
Modify
Cage Polyline
Extract Cage Polyline
Polyline
Studio Spline
Construction Tools
Cage and Body
X-ray Cage
Allow Backside Selection
Show Weight
Edit Object Display
Show Grid
NX Realize Shape
Preferences
Menu
No Selection Filter
Within Work Part O
Single Curve
Connect Cage
Cage Edge To Edit
Select Open Edge (4)
External Reference
Select Curve (0)
Continuity
Smooth
Sharp
Preview
OK
Apply
Cancel
Dependencies
Details
Preview
Single Curve : Edge of Extrude(27)
Select face edges or curves as external reference
Single Curve : Edge of Extrude(27)

Convert to PMI
2D图纸转换到PMI

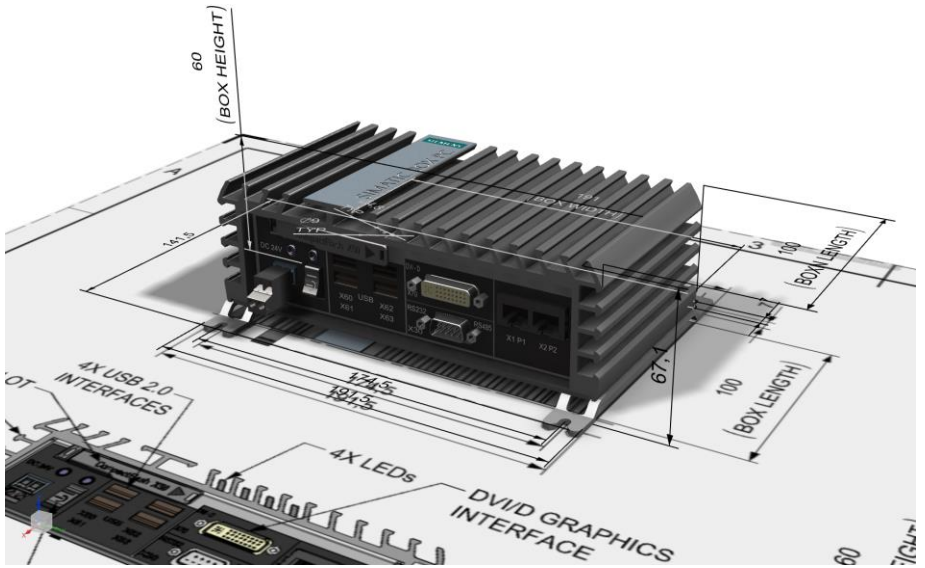
SIEMENS

Create PMI from legacy 2D drawings

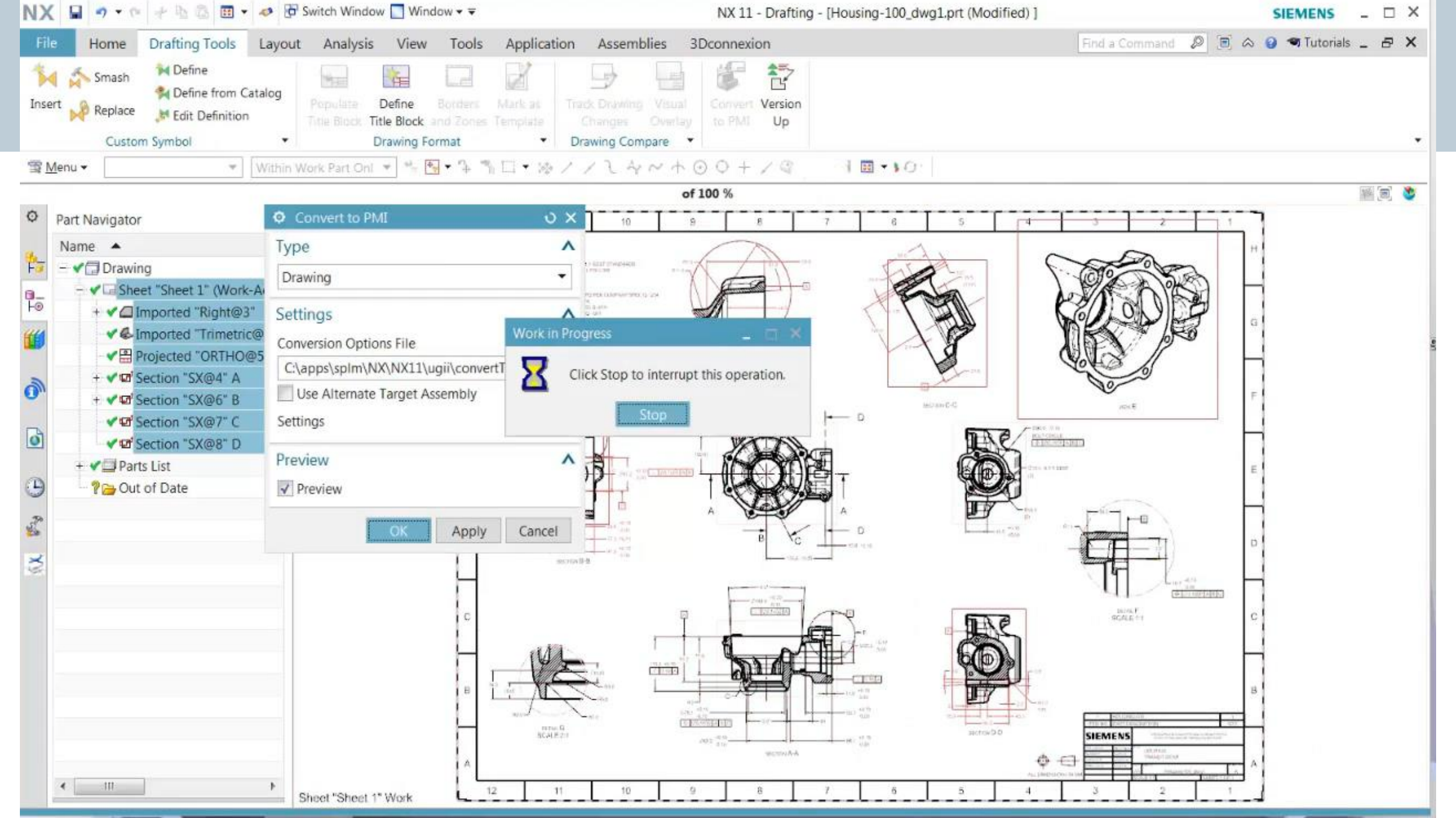
Automatically convert drawing views and objects to model views and PMI objects so you can utilize legacy data without rework by automatically adding intelligence to 3D models

现在，越来越多的公司利用产品和制造信息 (PMI) 来支持其基于模型的定义 (MBD) 战略。与基于图纸的传统流程相比较，PMI 在实现设计和制造下游使用、更好的质量控制和对重要产品信息的更快访问方面的优势明显。许多公司还往往保有大量原有 2D 图纸，为利用这些原有数据，他们必须花费精力将图纸中的信息重新创建为 3D 模型。此过程可能耗时巨大，并且需经过手动验证才能确保相关 3D 信息与图纸中的信息相互匹配。

- 利用 NX 11，您可以将图纸视图和对象自动转换为模型视图和 PMI 对象。这些对象包括尺寸、形位公差 (GD&T) 和注释等。
- 可以通过图纸、工作表、视图和注释对象选项轻松进行自定义。可以交互方式或通过自动化批处理执行转换至 PMI 的操作。
- NX始终坚持在无需返工的情况下利用原有数据。
- 借助 PMI 转换，您可以使用原有 NX 图纸快速、轻松地向3D 模型中添加信息。



将图纸视图和对象自动转换为模型视图和 PMI 对象。


NX
Switch Window
Window
NX 11 - Drafting - [Housing-100_dwg1.prt (Modified)]
SIEMENS
File
Home
Drafting Tools
Layout
Analysis
View
Tools
Application
Assemblies
3Dconnexion
Find a Command
Tutorials
Insert
Smash
Replace
Define
Define from Catalog
Edit Definition
Custom Symbol
Populate Title Block
Define Title Block
Borders and Zones
Mark as Template
Drawing Format
Track Drawing Changes
Visual Overlay
Drawing Compare
Convert to PMI
Version Up
Menu
Within Work Part Only
of 100 %
Part Navigator
Name
Drawing
Sheet "Sheet 1" (Work-A
Imported "Right@3"
Imported "Trimetric@
Projected "ORTHO@5
Section "SX@4" A
Section "SX@6" B
Section "SX@7" C
Section "SX@8" D
Parts List
Out of Date
Convert to PMI
Type
Drawing
Settings
Conversion Options File
C:\apps\splm\NX\NX11\ugii\convertT
Use Alternate Target Assembly
Settings
Preview
Preview
OK
Apply
Cancel
Work in Progress
Click Stop to interrupt this operation.
Stop
SIEMENS
Sheet "Sheet 1" Work

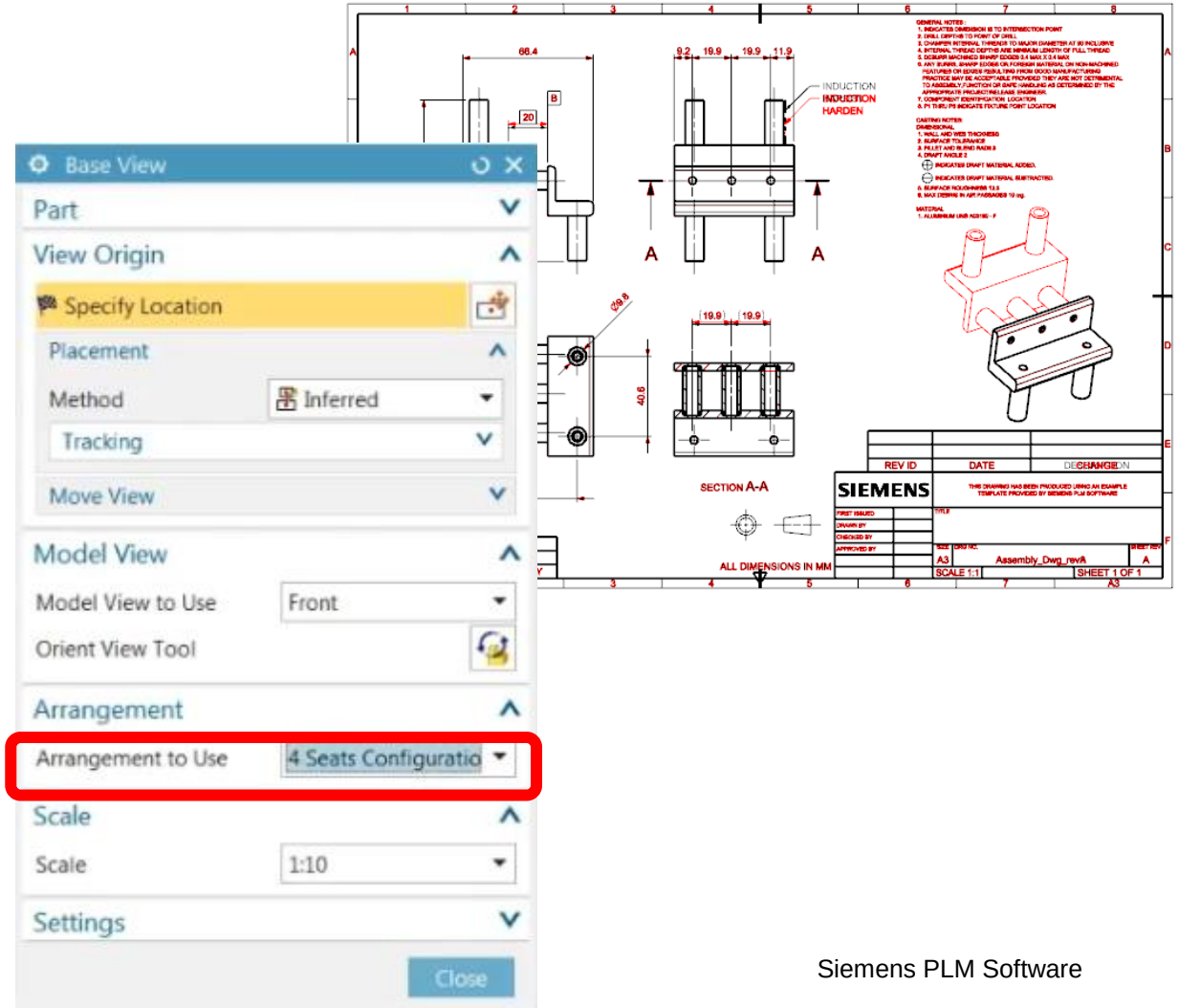
SIEMENS

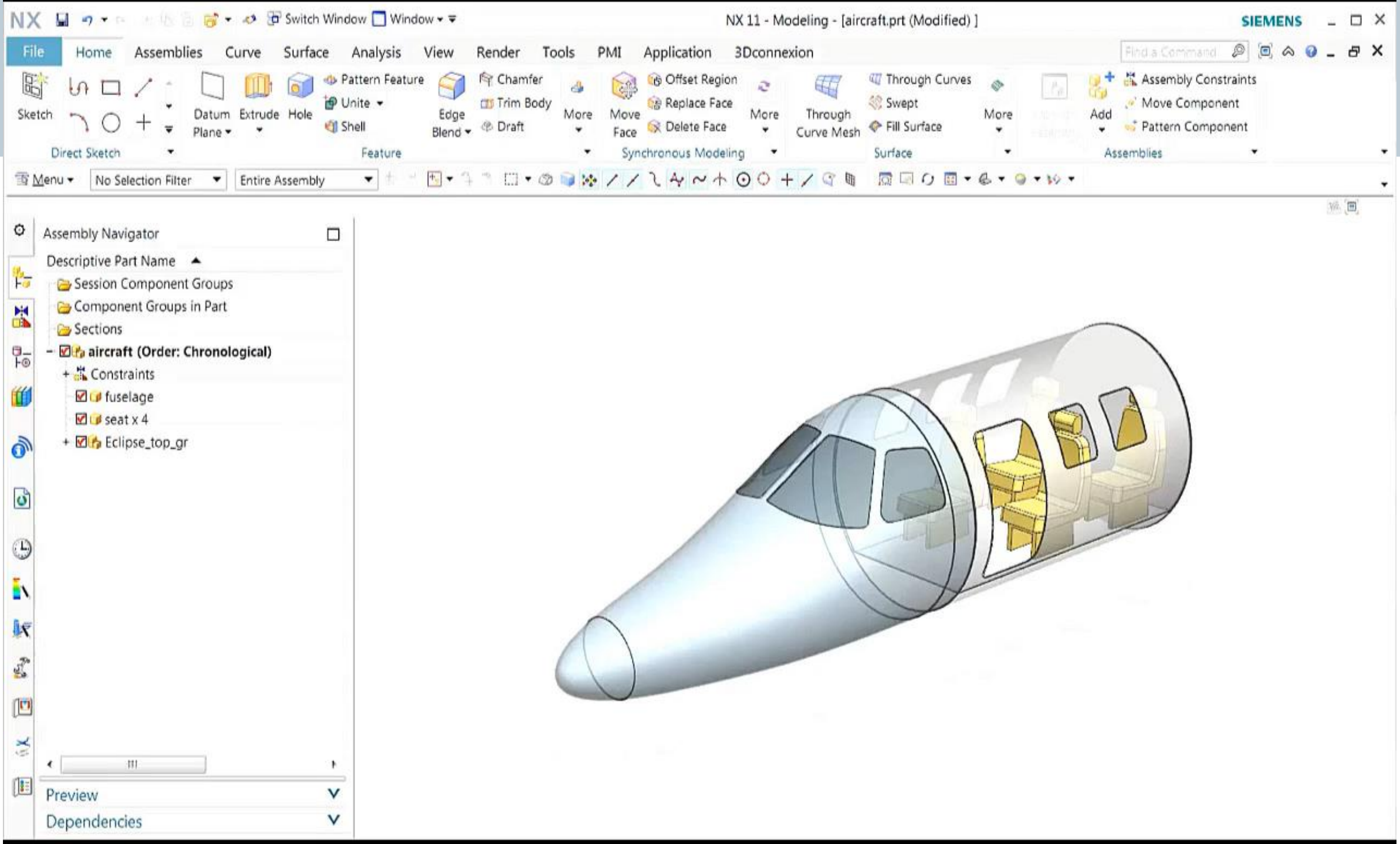
A state-of-the-art tool for drawing creation and modification

通过图纸比较工具减少检查时间。

对于许多公司而言，设计流程中离不开图纸，NX 制图始终是创建和管理图纸的领先工具。在 NX 11 中：

- 智能轻量级视图可以在处理大型装配视图时，利用多线程处理节省时间。
- 新图纸比较工具集可以通过快速、方便地标识更改，大幅缩短图纸检查时间，同时还能消除图纸更改可能导致的制造误差。
- 使用 NX11，您还可以在图纸上显示多种装配布置，不仅能够深入了解产品的装配信息，还能查看内部细节或备选构建方案。




NX
Switch Window
Window
NX 11 - Modeling - [aircraft.prt (Modified)]
SIEMENS
File
Home
Assemblies
Curve
Surface
Analysis
View
Render
Tools
PMI
Application
3Dconnexion
Find a Command
Sketch
Direct Sketch
Datum Plane
Extrude
Hole
Pattern Feature
Unite
Shell
Edge Blend
Chamfer
Trim Body
Draft
More
Feature
Move Face
Offset Region
Replace Face
Delete Face
More
Synchronous Modeling
Through Curve Mesh
Through Curves
Swept
Fill Surface
More
Surface
Add
Assembly Constraints
Move Component
Pattern Component
Assemblies
Menu
No Selection Filter
Entire Assembly
Assembly Navigator
Descriptive Part Name
Session Component Groups
Component Groups in Part
Sections
aircraft (Order: Chronological)
Constraints
fuselage
seat x 4
Eclipse_top_gr
Preview
Dependencies

High-end rende
高性能渲染

Photorealistic

在实际生产前了解
照片级渲染可以在
材料的相关决策。

- NX 11 采用新
- 完全集成到 NX
  供多种材料和
- 并能通过多线程
- 如果需要以更性
  的渲染工作。
- 此外其还具有
  地控制渲染完成

借助 Lightworks
了操作，帮助您以

概述 | IRAY | V-RAY

利用 NVIDIA® Quadro® 视觉计算设备 (VCA) 加速设计与视频特效制作工作流程。网络上的任何人都可以使用它，轻松地把它融入到设计工作流程当中，而且它在处理性能上可以实现线性扩展，从而可造就无噪而互动的全局光照效果。物理效果逼真度的可预测性与准确性现已实现完全互动，从而能够加快创作过程中的决策速度。

VCA规格参数

特性	细节
GPUs	8 颗 NVIDIA 高端 GPU
显存	每颗 GPU 12GB显存
NVIDIA CUDA®核心数量	24,576

Lightworks
Iray+

NVIDIA
NVIDIA Iray

NVIDIA Quadro 视觉计算设备VCA

照片级渲染可以在实际生产前显示产品的外观。

NX11和NX10光纤追踪艺术外观渲染对比（默认设置，材质光照未定义）

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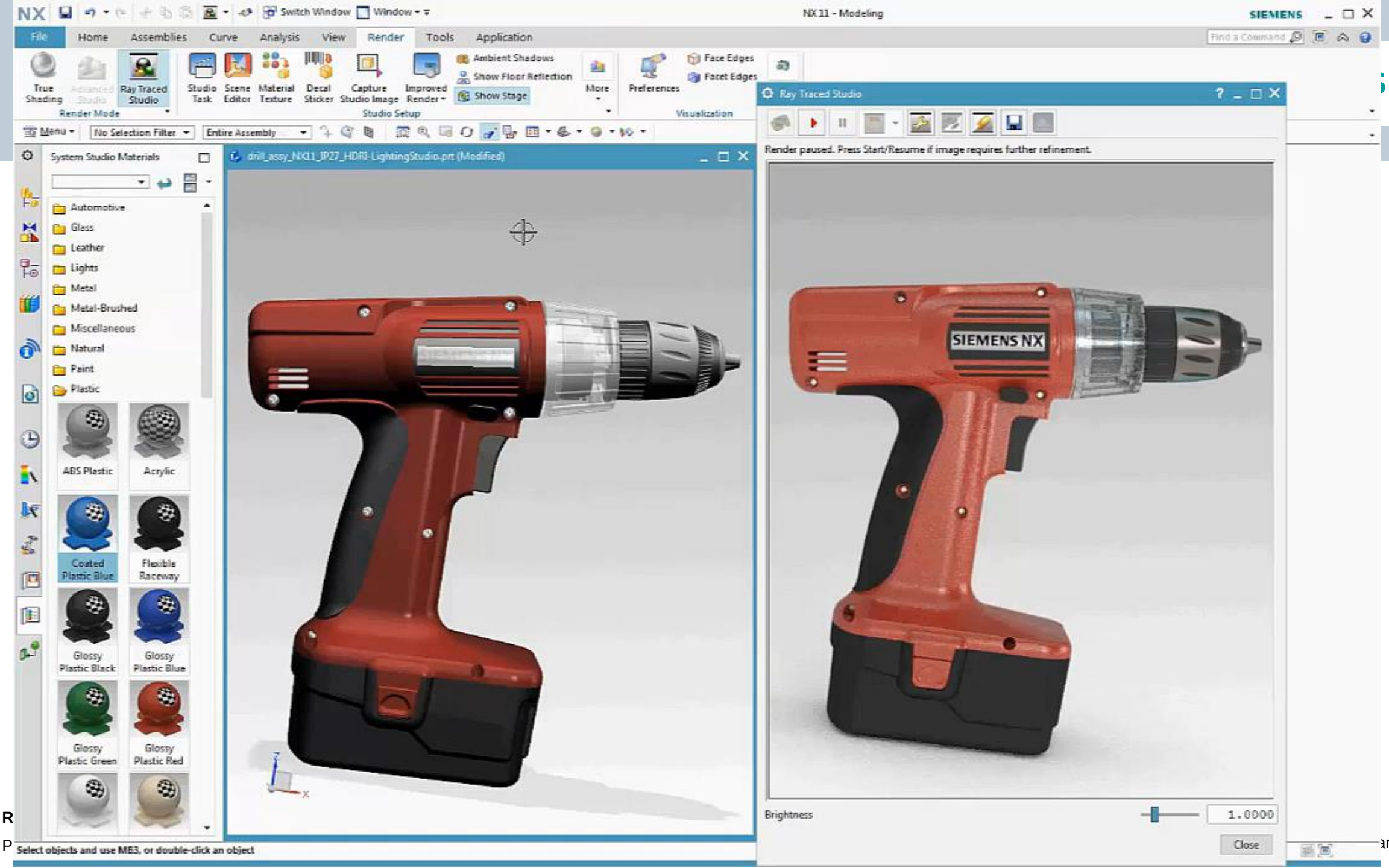

NX
NX 11 - Modeling
SIEMENS
File
Home
Assemblies
Curve
Analysis
View
Render
Tools
Application
True Shading
Ray Traced Studio
Render Mode
Studio Task
Scene Editor
Material Texture
Decal Sticker
Capture Studio Image
Improved Render
Ambient Shadows
Show Floor Reflection
Show Stage
Studio Setup
More
Preferences
Face Edges
Facet Edges
Visualization
Menu
No Selection Filter
Entire Assembly
System Studio Materials
Automotive
Glass
Leather
Lights
Metal
Metal-Brushed
Miscellaneous
Natural
Paint
Plastic
ABS Plastic
Acrylic
Coated Plastic Blue
Flexible Raceway
Glossy Plastic Black
Glossy Plastic Blue
Glossy Plastic Green
Glossy Plastic Red
drill_assy_NX11_JP27_HDRI-LightingStudio.prt (Modified)
Ray Traced Studio
Render paused. Press Start/Resume if image requires further refinement.
SIEMENS NX
Brightness
1.0000
Close
Select objects and use MB3, or double-click an object

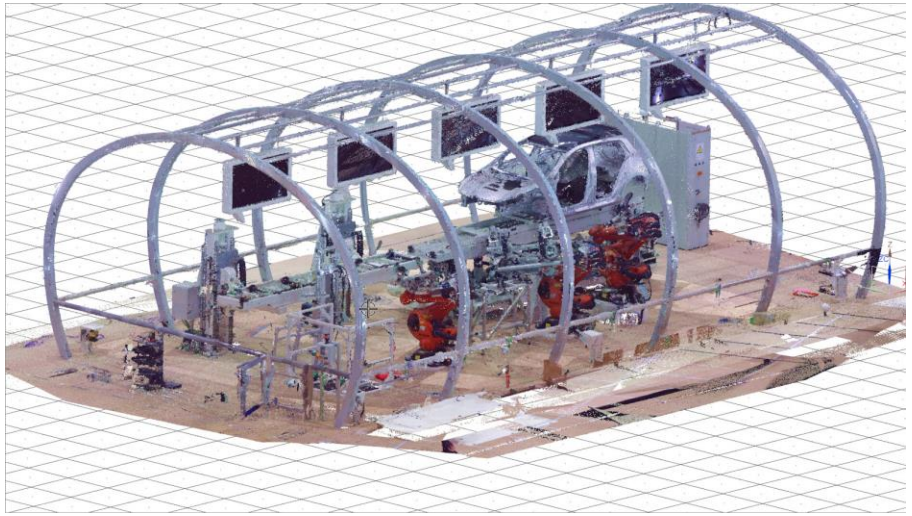
Point Cloud
点云处理

SIEMENS

Point cloud processing with Bentley奔特力 Pointools

许多行业使用点云数据来获取产品、车间、工厂、建筑和基础设施等的现有情况。为此，NX 11 引入了使用 Bentley® Pointools™ 的点云处理功能。

- 现在，您可以导入标准POD 文件格式的点云数据，进行简单测量并进行编辑，例如显示/隐藏、传输、删除以及编辑参数等。
- 由于无需创建 3D 数据，因此可以简化对参考数据的访问，尤其是对包括工厂和生产线设计的应用程序来说更是如此。



点云数据使生产线设计变得更轻松、更准确。

NX on Cloud
云中的 NX

SIEMENS

Take advantage of full NX functionality in a cloud-managed environment

NX on cloud gives you flexibility, scalability, and reliability that helps you matches business demands as they change

- 对于 NX 11 的某些配置，您可以选择在云托管环境中部署并运行 NX。
- 这样既可以利用新增的 NX 功能，又可以享受基于云的产品所带来的各种便利，例如降低在安装、维护和升级过程中的复杂性、时间、成本和风险。
- 同时意味着 NX 将会继续按照既定模式运行并响应。此外，IT 团队可以专注于核心业务功能，而非管理环境。云中的 NX 提供的灵活性、可扩展性和可靠性可以满足您的不同业务需求。

NX 11 for Design What's New

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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
Support Remote Display
Support for New Direct Model Version
NX Open Selection Support
Navigator Selection Performance
Circle Selection Method
Selection Tune Up
Mixed Modeling PSM Display Support
Point Cloud - Architecture Support
Plot Support for Colored Drawing Symbols
NX Open Toolkits Upgrades
NX Open Usability
Python UDO Support
NX Open Partner Support
NX Open Support for Spreadsheet Interaction
Timeline Tool
Information Window
Navigator Drag and Drop
Improved Window Management
Automatic Application Change
Upgrade QT

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 - Spine Curve feature
Wireframe - Scale Curve feature
Wireframe - Isocline curve feature UI & Selection Intent
Surfacing - Variable offset surface (Tailored blanks)
Surfacing - Flattening and Forming
Surfacing - Studio Surface improvements
Surfacing - Section Surface accuracy
Wireframe - Derived curves improvements
Realize Shape - Connect
Realize Shape - Polish
Analysis - Multi-Face Continuity
Mixed NURBS/Facet modeling
Analysis - AO Performance improvements
Analysis - AO Usability improvements
Analysis – AOs in Assembly Context

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

PMI

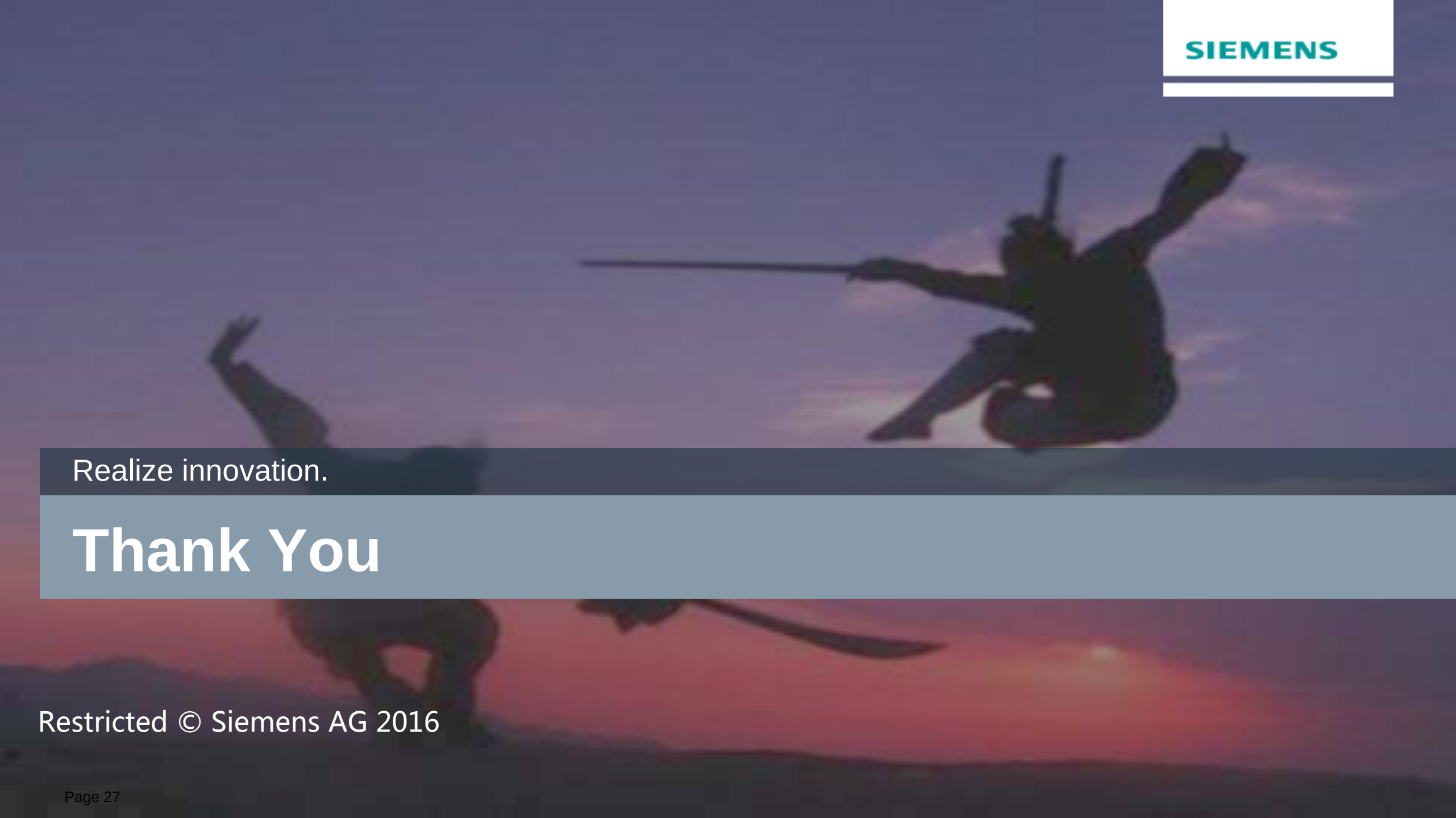
2D Drawing to 3D PMI
Part Navigator Performance Improvement
PMI Inherit Enhancements
Support for Angular Directed Dimensions
PMI Lightweight Section View Enhancements
PMI Region Enhancements
PMI Resize Enhancements
PMI display parallel to screen
Dimension UI Enhancements
Hole Callout enhancements
Dimension Settings
Limits and Fits
Support for Angular Directed Dimensions
Selection for Routing Anchor Points

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
Motion Consolidation Architecture
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)



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Realize innovation.

Thank You