PLM World 05

Integrated Modeling and Analysis Using TeamCenter Engineering

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Premier Partner:
Microsoft

Topics

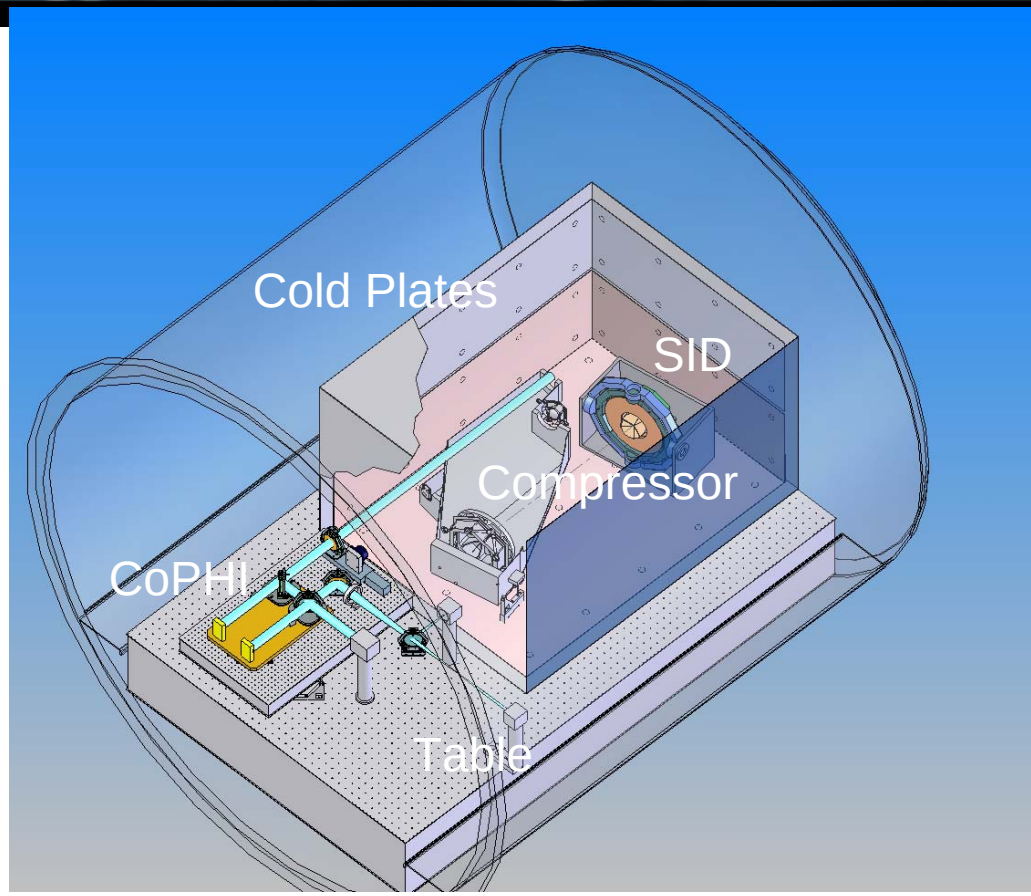
- Integrated Modeling and Analysis Process Pilot
- Overview of TC Eng with UG-NX & I-DEAS
- Constructing the Analysis BOM
- Modeling
- Discipline Interfaces
  - Thermal to Structural
  - Structural to Optical
- Collaboration with Results in TC Engineering
- Process Pilot Accomplishments
- Other Process Activities Not Covered
  - Revising and Version Management
  - Working with External Partners for Design Or Analysis
  - CAE Manager

Analysis Process Pilot Overview

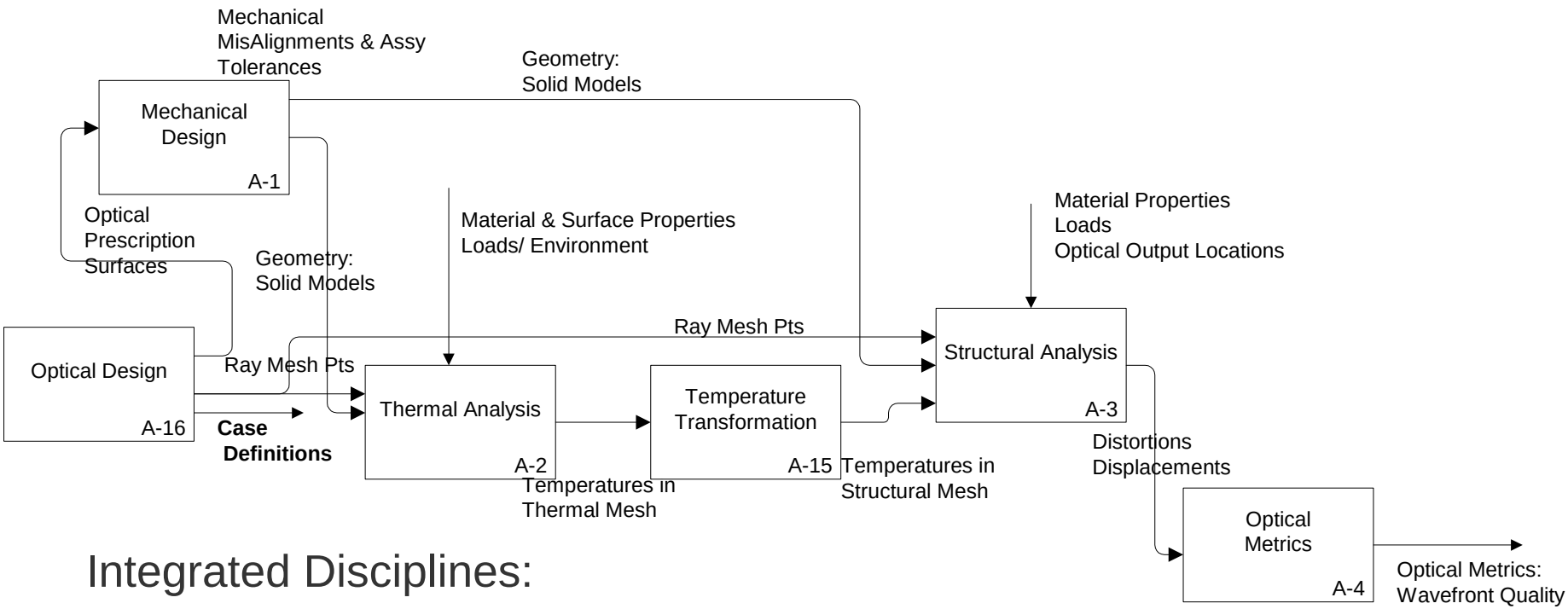
- Pilot Scope
  - Build TOM3 Baseline Models & 1st Measurement Case
  - Integrated Modeling Process Development
  - Start June 2004
  - Complete On or About September 2004
- SIM TOM3 Timeline
  - 8/1/04 Single Optic Case
  - Baseline: 8/30/04 Thermal, 9/15/04 Structural, 10/1/04 Optics
  - Analysis of Data Campaign 10/1/04 – 1/10/05
- Pilot Objectives
  - Reduce technical and schedule risk to SIM TOM3 testbed analysis
  - Establish integrated structural thermal analysis process with deep precision capability for JPL
  - Provide insight to integrated modeling needs and requirements for UGS product development
  - Provide showcase environment for next-stage UGS CAE products

TOM3 Analysis Configuration & Layout

- What is in the model:
  - SID: Geometry, Thermal, Structural, Optical
  - Compressor: Geometry, Thermal, Structural, Optical
  - Newport Table: Simple Geometry, Thermal, Structural
  - Cold Plates: Simple Geometry, Thermal
  - CoPHI: Simple Geometry, Thermal, Optical



Integrated Modeling Process



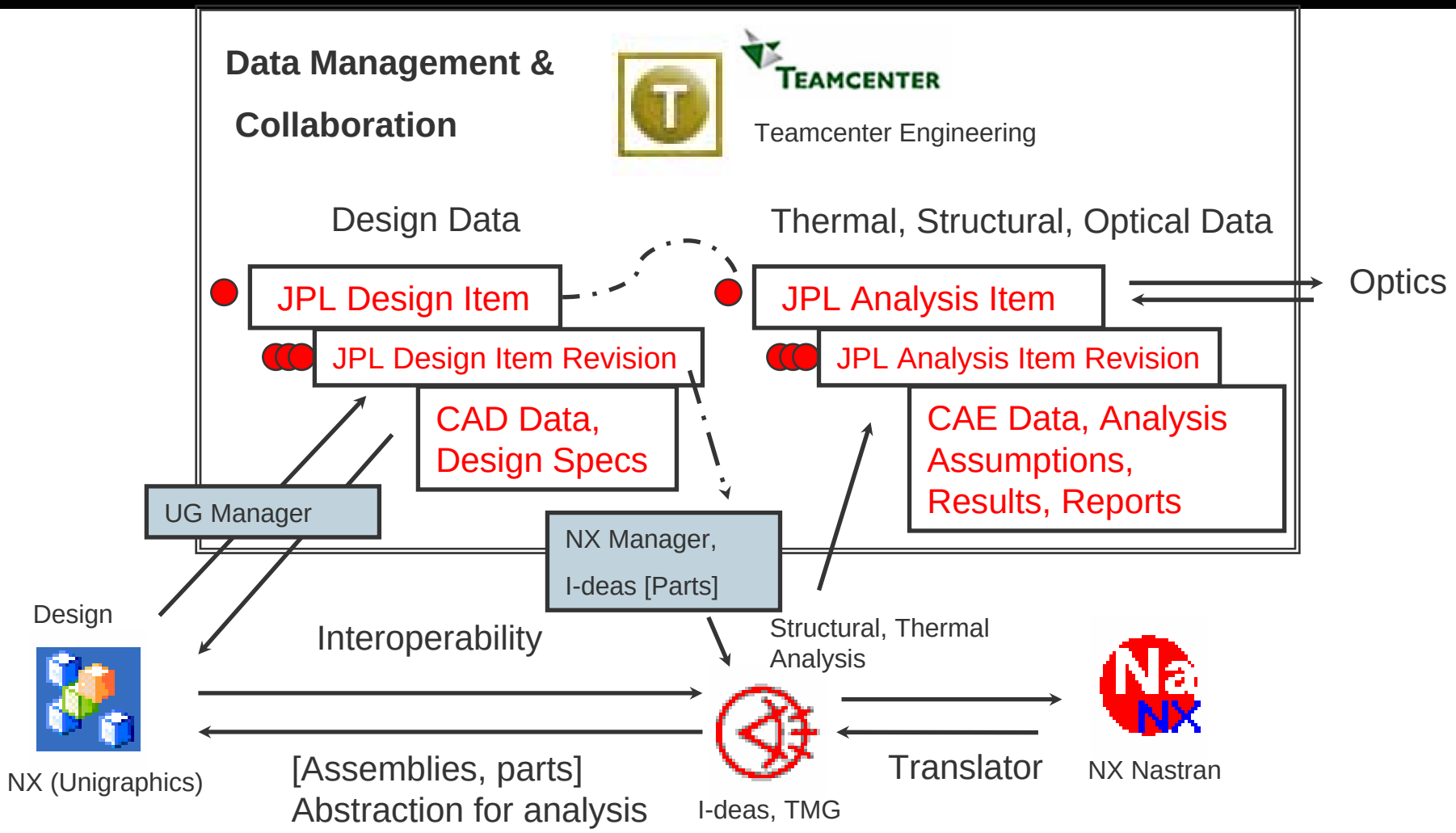
Integrated Disciplines:

- Design, Structural Analysis, Thermal Analysis, Optics

Detailed Design Phase Capabilities

- Large Models
- Configuration Control via TeamCenter Engineering

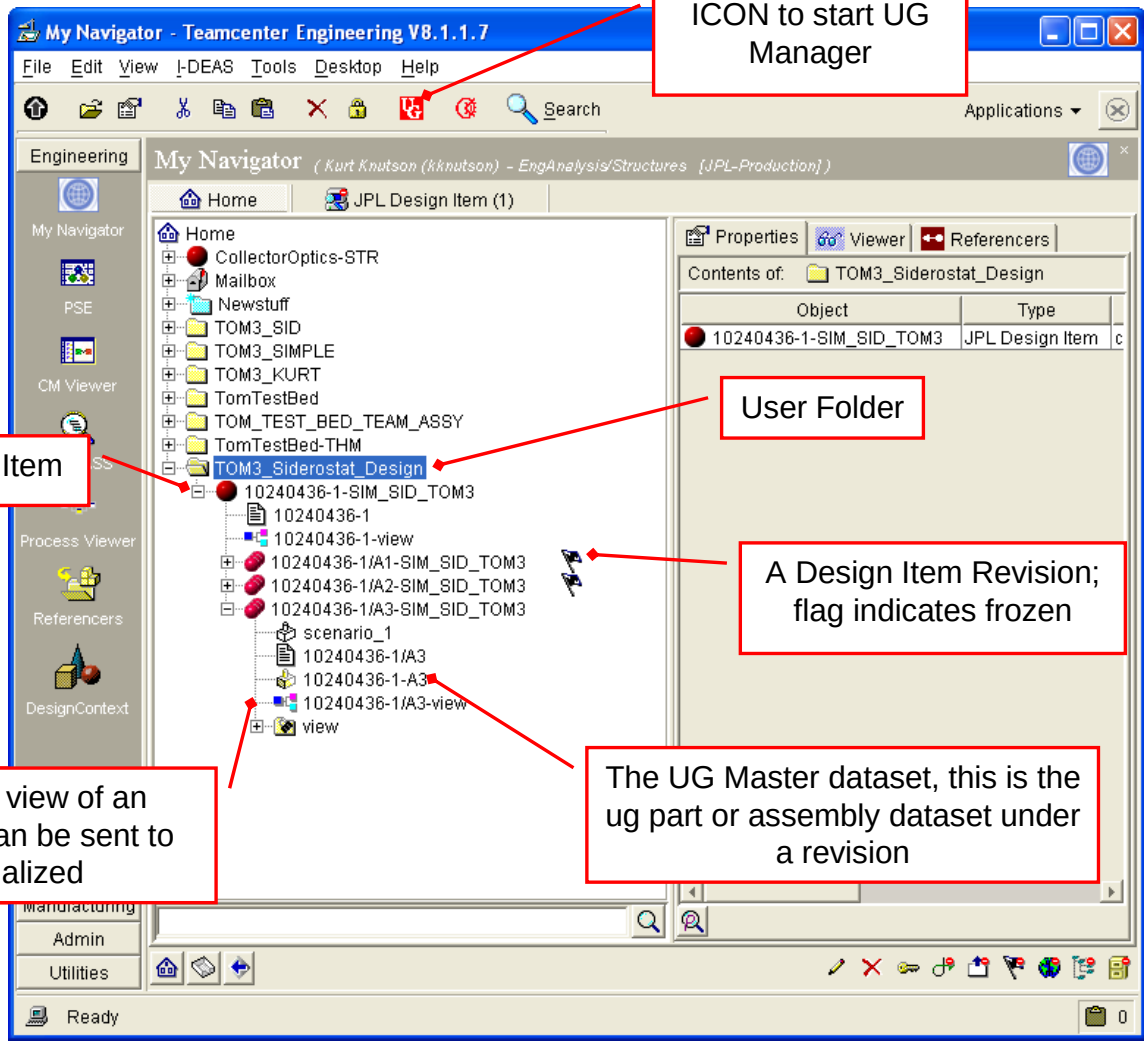
Collaborative Design/Engineering Environment and Software Interaction



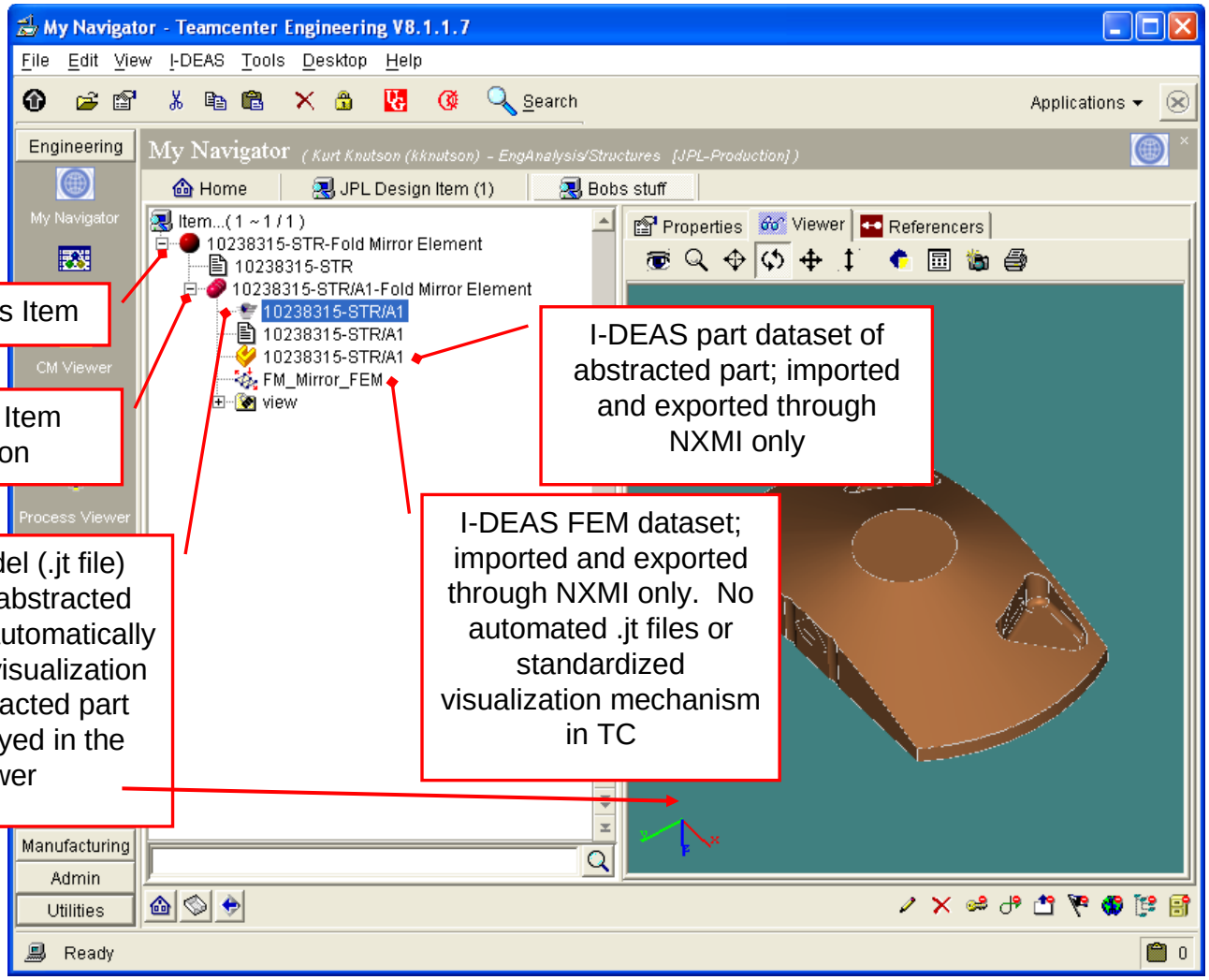
Conventions

- Analysis BOM
  - Parallel Pruned Copy of Design BOM
  - Holds Analyst's Abstracted Models
  - Supports Assembly FEM
  - Holds/Displays Analysis Results
- Naming Convention
  - Design Items: Use Part Number
  - Analysis Items: Use Part Number + "–THM" or "–STR"
  - This ties Analysis BOM to Design BOM
- TC Eng Permissions by Group
  - Design: R/W by All
  - Engineering: R/W Owner Only, R by All
  - Analysis: R/W Owner Only, R by All
- JPL Custom Item Types
  - All Derive from TC Eng Item
  - JPLDesignItem & JPLAnalysisItem Add Metadata in Custom Forms

Design Items and Datasets in TeamCenter Engineering



I-DEAS Analysis Items and Datasets in TC Eng Created Using NXMI

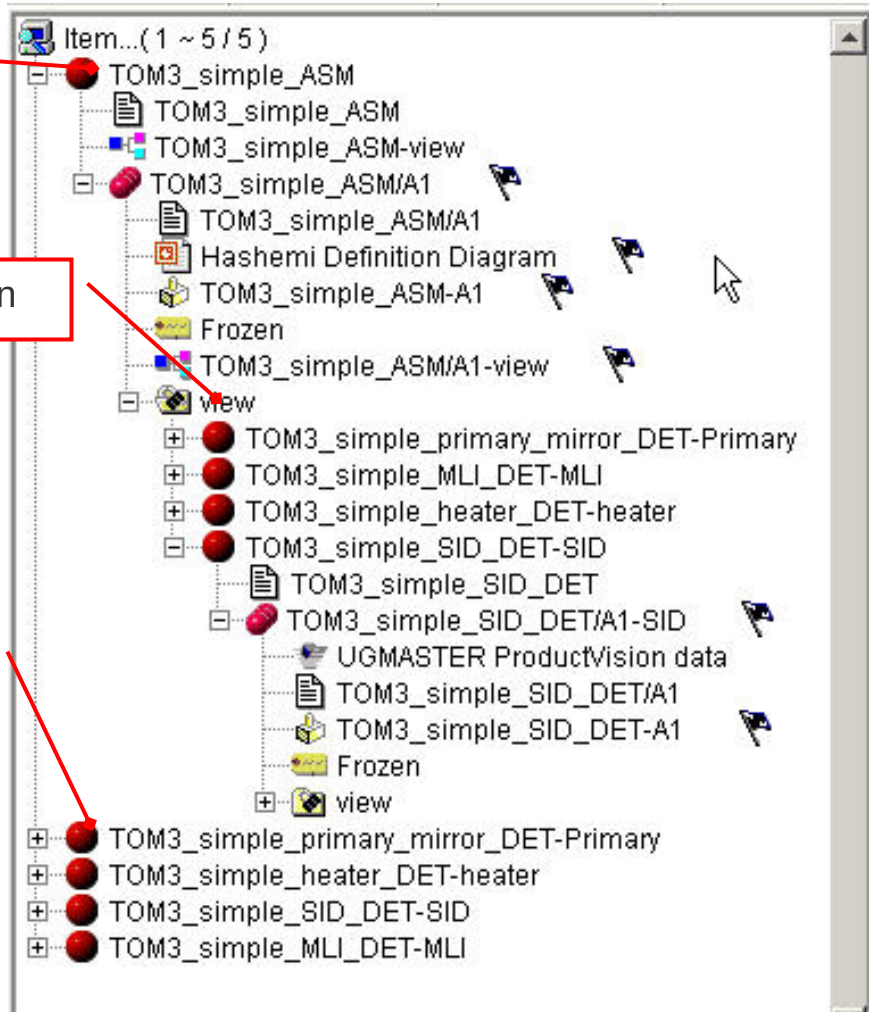


Design Assembly aka Design BOM

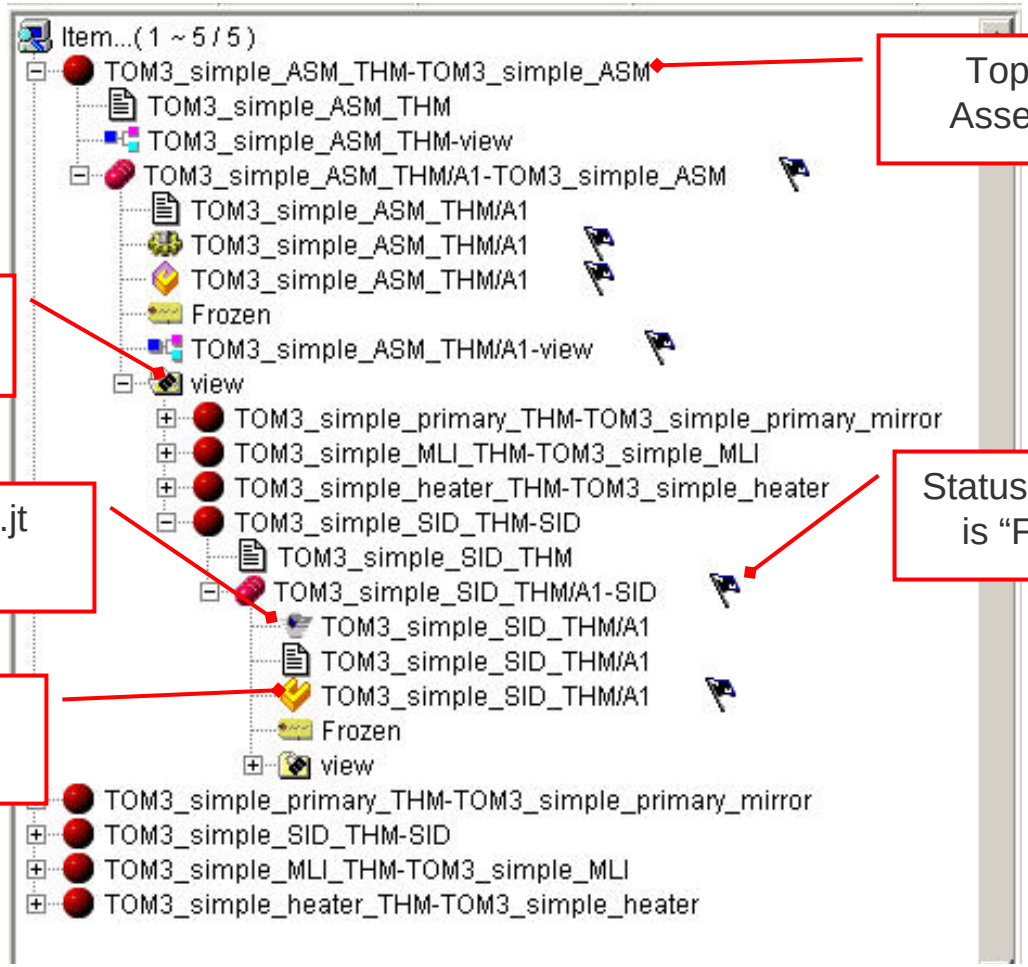
Note Non Standard Naming Convention!

This Assembly's Children

Other Parts that met the Search Criteria "TOM3_simple*"



Thermal Assembly aka Analysis BOM



Top Level Design Assembly + “-THM”

This Thermal Assembly’s Children

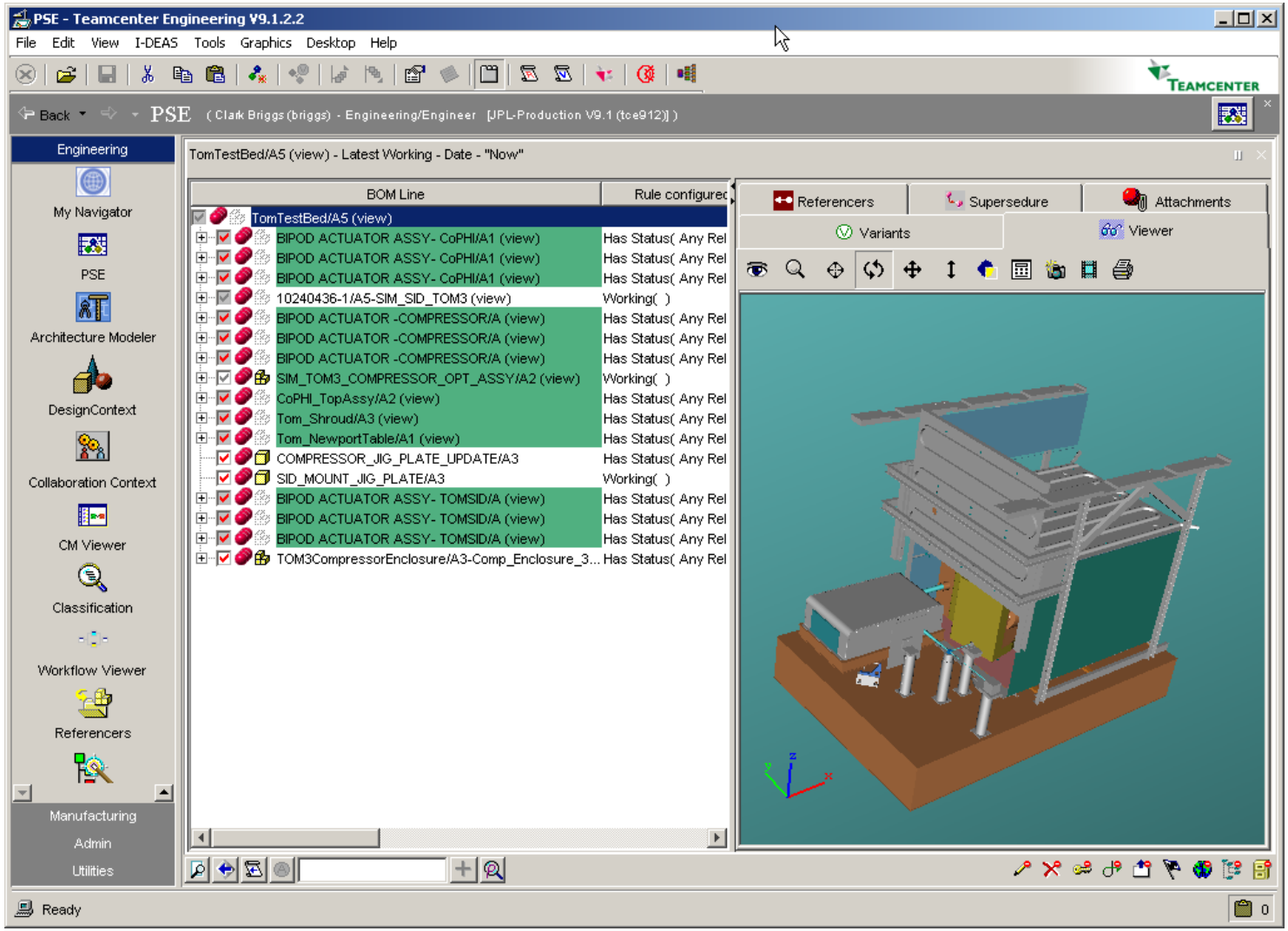
This Thermal Part’s .jt Visualization File

Status Flag Indicates the Part is “Frozen” or Read-Only

This Thermal Part’s I-DEAS Model File

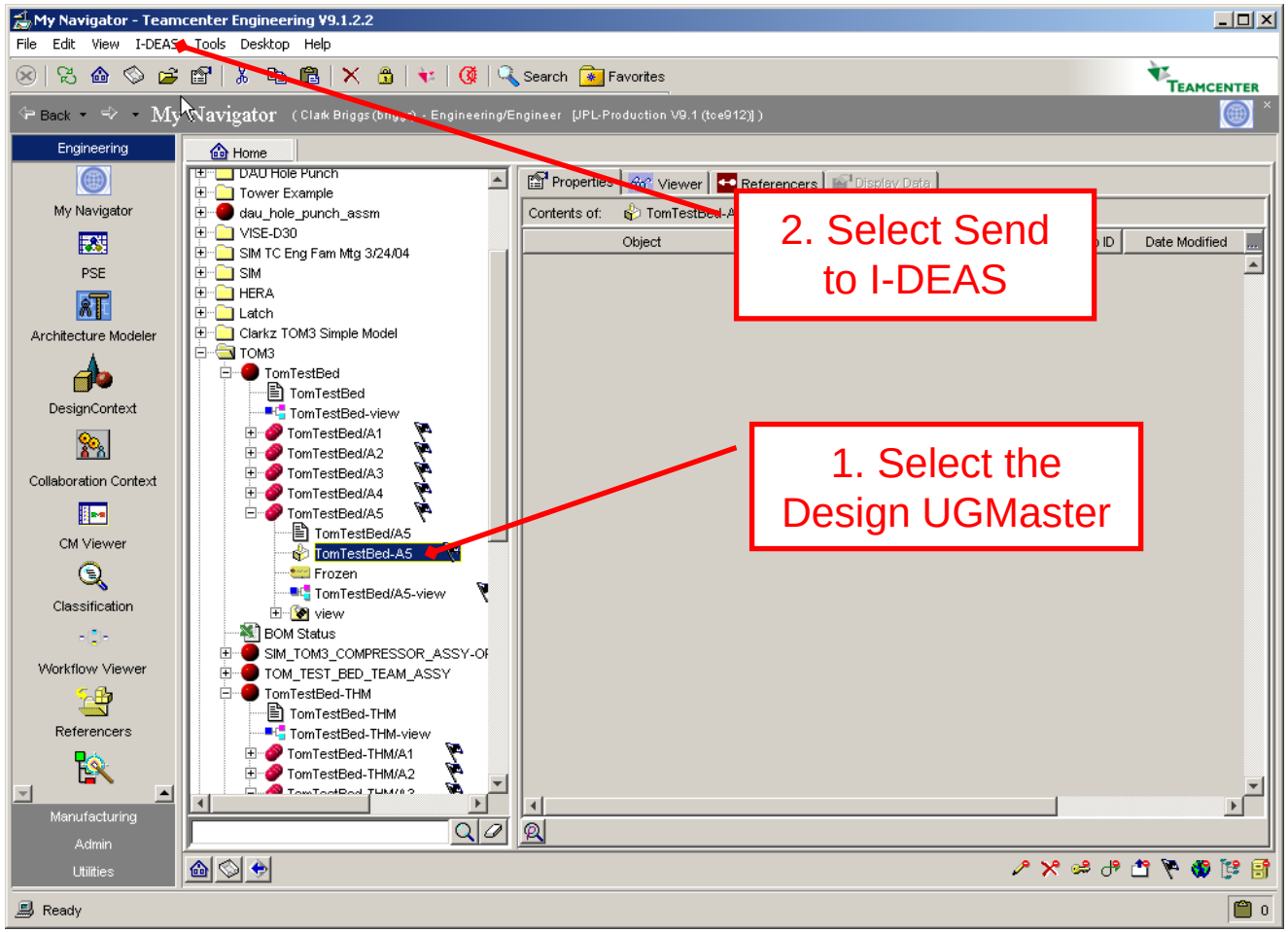
Constructing the Analysis BOM

- Showing Design BOM in PSE
- Objective: Build Analysis BOM for Thermal Model



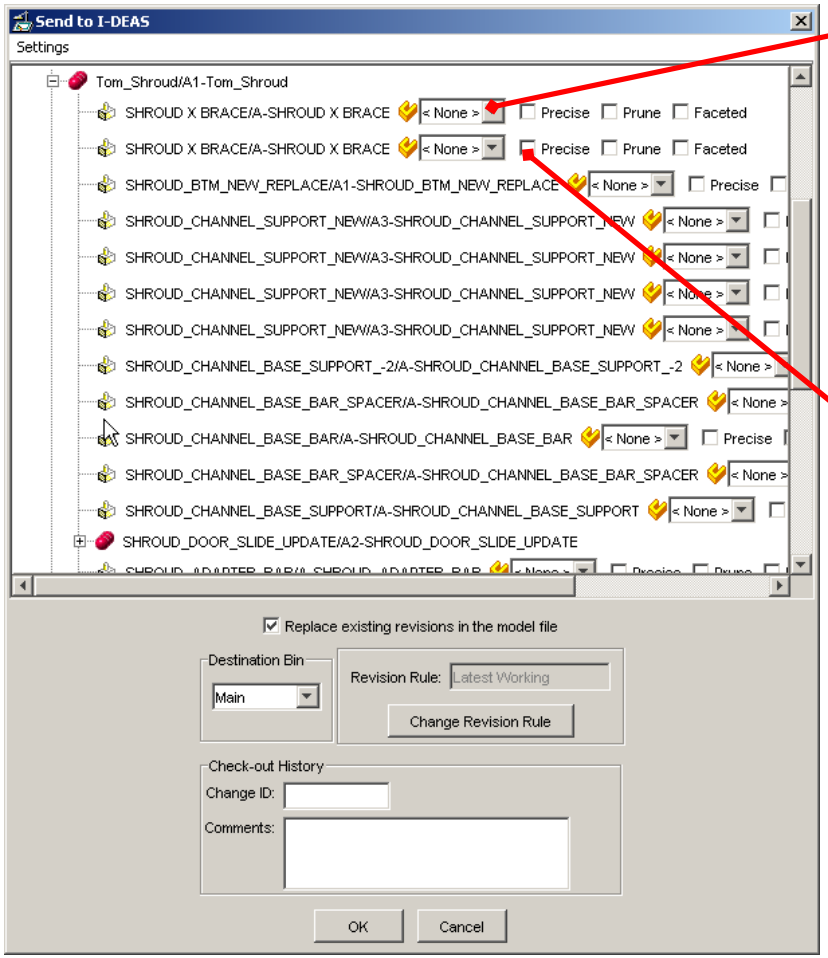
Constructing the Analysis BOM

- Start with Design BOM in Navigator
- Select the Top Assembly UG Master Dataset
- Select I-DEAS -> Send to I-DEAS Menu



Constructing the Analysis BOM

- Select “New” for all Parts to be Analyzed so that Subsequent SaveAs will Create I-DEAS Parts
- Select “Precise” for all Parts to be used in the Analysis Context but not Analyzed

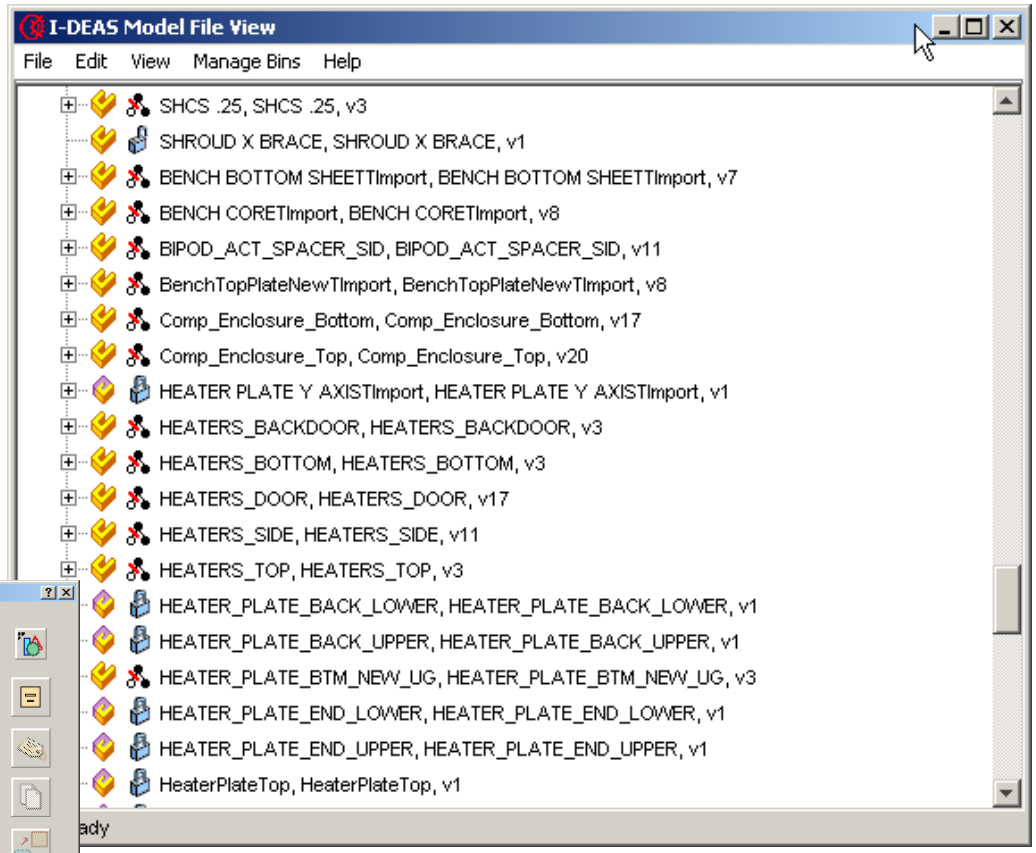


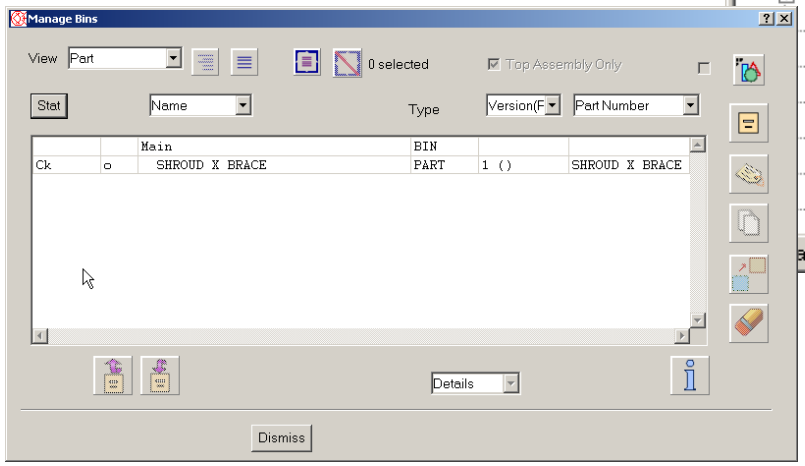
Select New for Parts to become I-DEAS Parts

Select Precise for Design Context Parts

Constructing the Analysis BOM

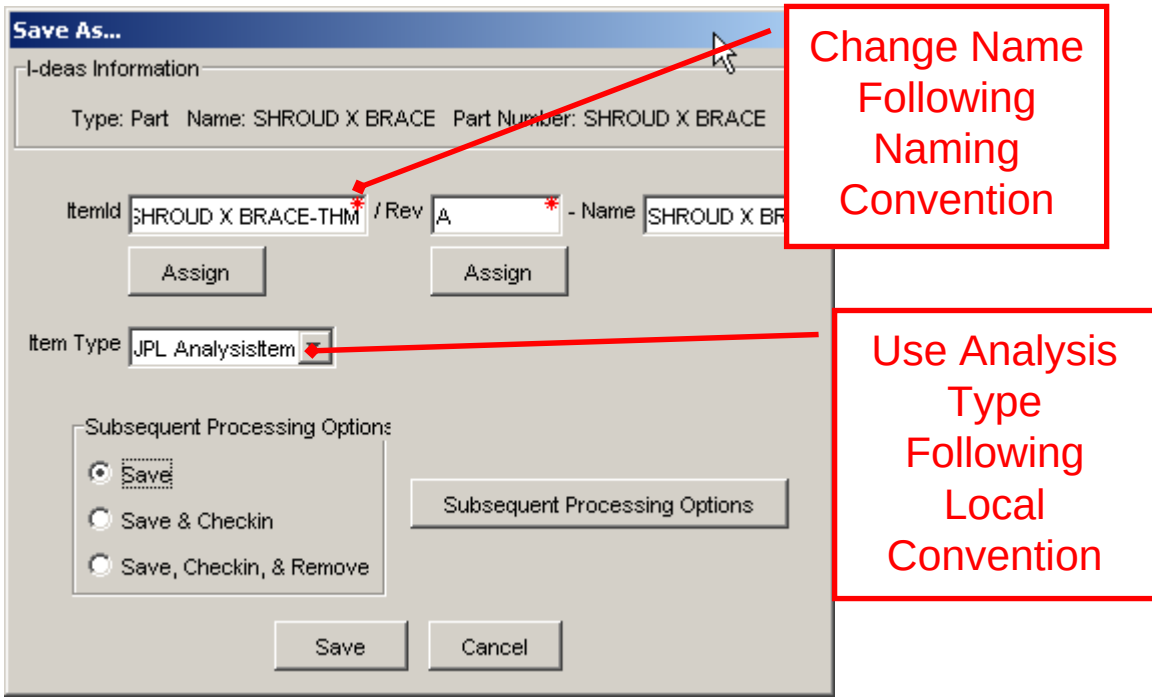
- This shows the Assembly In I-DEAS
- Design NX Parts Sent As Precise Are Pruned
- Design Assemblies Have RFL Permissions
- Analysis Parts are Checked Out





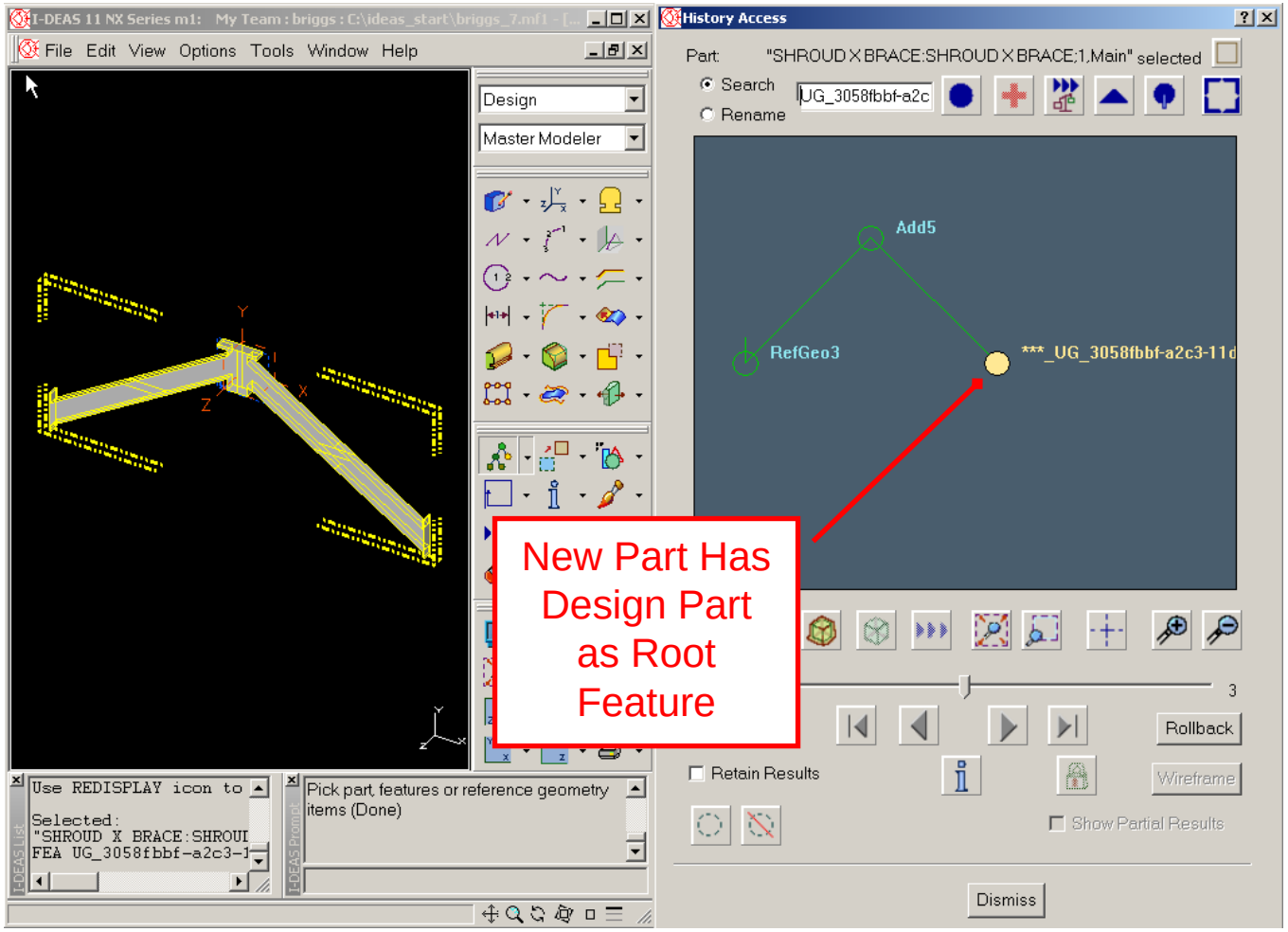
Constructing the Analysis BOM

- Create New I-DEAS Part For Each Part to be Analyzed
- Change the ItemId Following Local Naming Convention
- Change from Design Item to Analysis Item
- Select Save to Create the Initial Analysis Item and I-DEAS Part

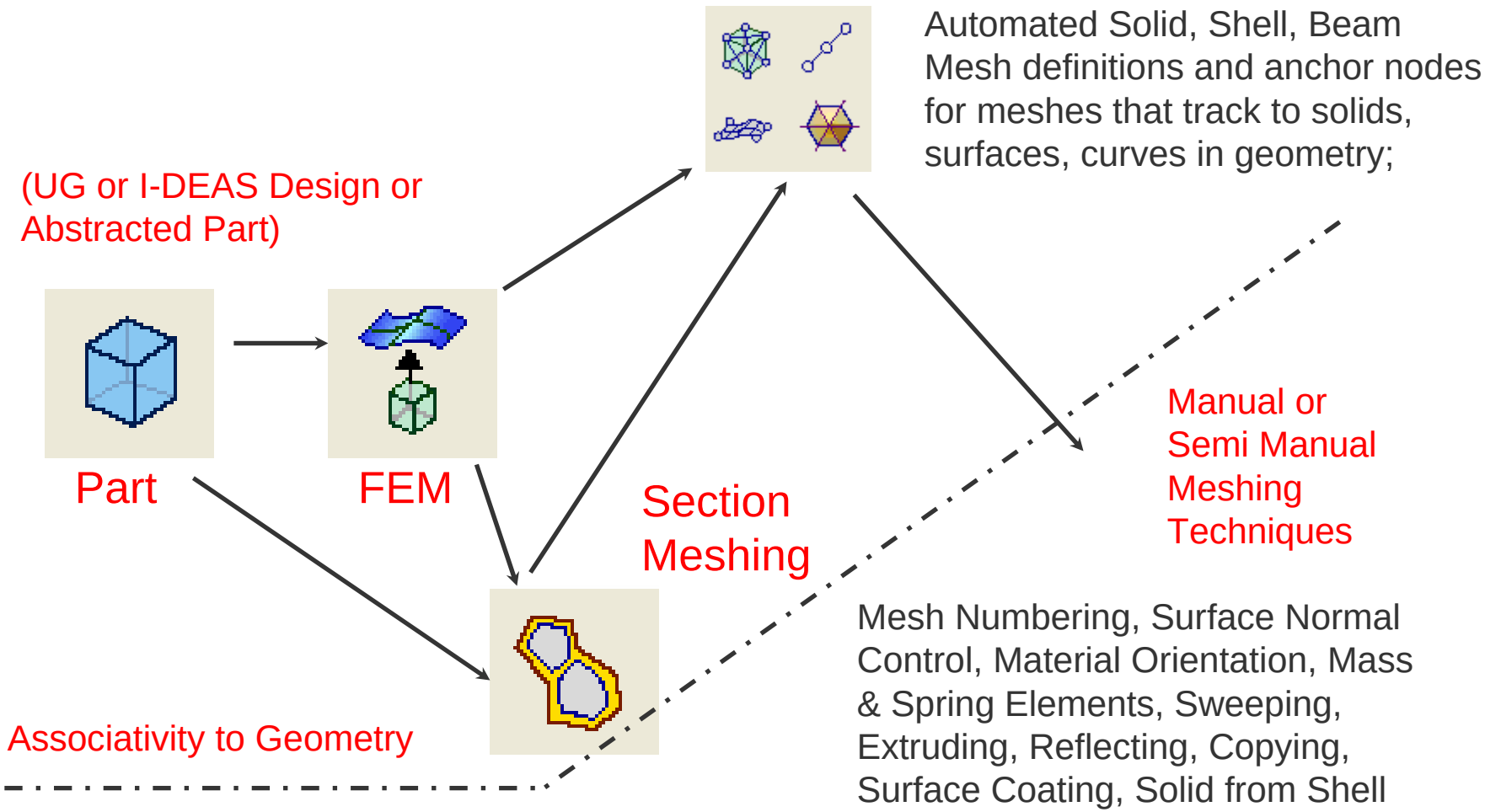


Constructing the Analysis BOM

- Every New I-DEAS Analysis Part Has Its NX Design Part as Root Feature
- I-DEAS Part Geometry Is Associative to the Design Part
- Analysis Can Update With Design Changes



Meshing Methods That Preserve Associativity



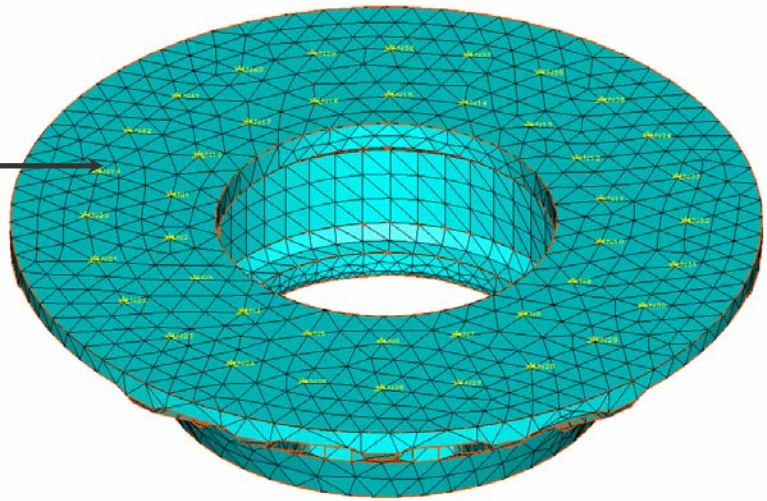
Meshing Methods with Assemblies

- I-DEAS Requires All Meshes to be in a Part
  - Method 1: Use the Assembly to Generate a Part
  - Method 2: Parameterize Assembly as a Part
  - Method 3: Create FEM from Assembly
  - Method 4: Hybrid Methods
- Choice Depends Upon
  - Substantial Use of Instances
  - Modeling and Abstraction Required for Analysis
  - Nature of Design Geometry Changes
  - Associativity of Modeling and Meshing Techniques
- See Back Up Information for More Details

Thermal to Structural Modeling Interface: Mapping TMG Temperature Results

- Thermal results obtained from a TMG run can be mapped from a thermal FEM to a structural FEM within I-DEAS.
- Temperature results can be used to create thermal boundary conditions.
- The thermal analyst needs to provide three universal files.
  - Thermal_Model.unv — The model used for analysis
  - tmgtempn.unv — Nodal temperature data
  - tmgtempe.unv — Elemental temperature data
- Groups or TMG “zones” can be used to help ensure proper mapping.
  - A TMG zone is a specifically named group
  - Temperatures will be mapped from a thermal model zone to its structural model’s counterpart
  - TEZON03 maps to STZON03

Structural to Optical Modeling Interface: Creating Nodes for Ray Points on Geometry



Define Reference Points for ray mesh locations

Modify Mesh definition to match anchor nodes on reference points

Optical Analysis is Based on Bundle of Rays

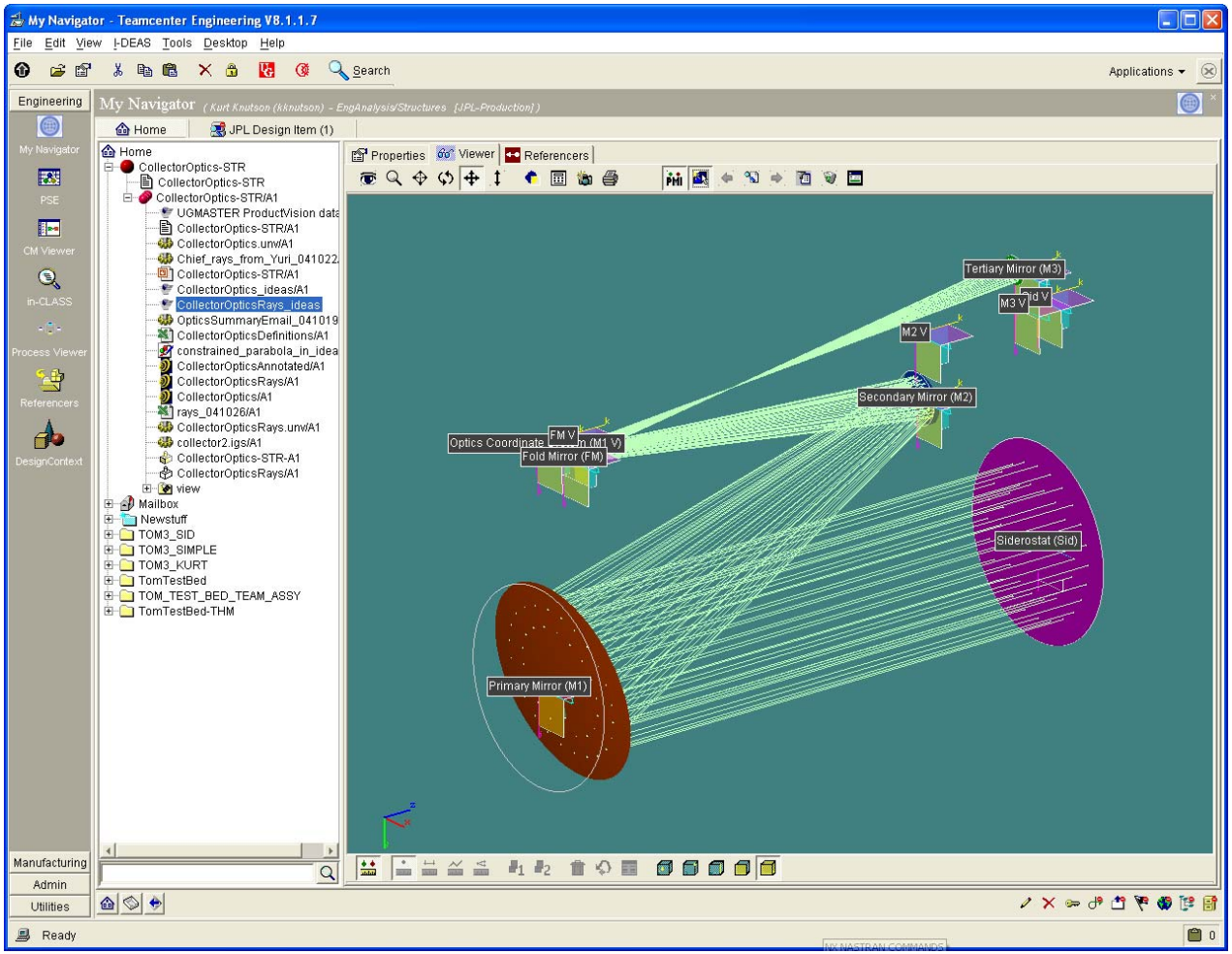
Ray Locations on Mirrors Used to Define Reference Points

FEA Mesh Created with Nodes at Ray Locations

Displacements At Ray Locations Sent to Optical Analysis

Managing Structural/Optical Interface in TeamCenter

- OptoMechanical Data Managed in TC Eng
- Visualization of Rays & Optics for I/F Review
- Distortion DataSets Saved and Managed in TC Eng



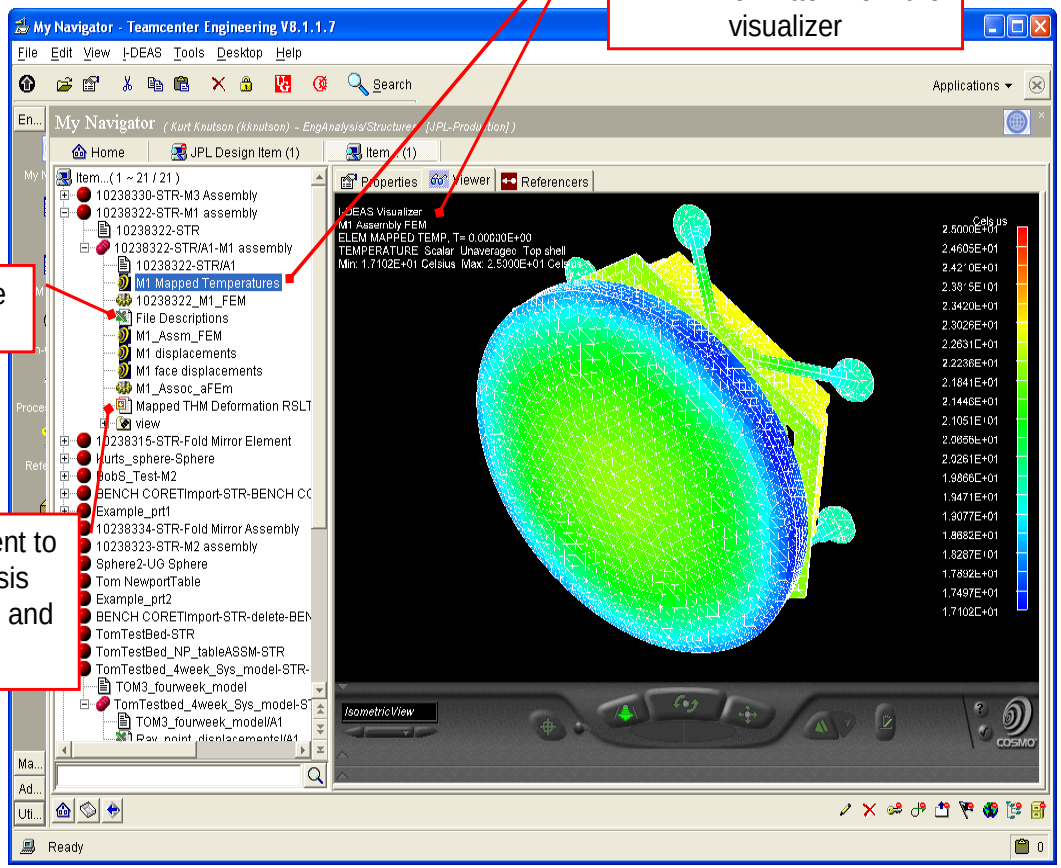
Analysis Results in Collaboration Environment

- Take Advantage of TC Eng Version Control
- Document Analysis Inputs & Outputs
- Take Advantage of TC Eng and MS Office Integration
- Make Results Available to Non-Specialists

Mapped temperature results displayed in a VRML file written from the visualizer

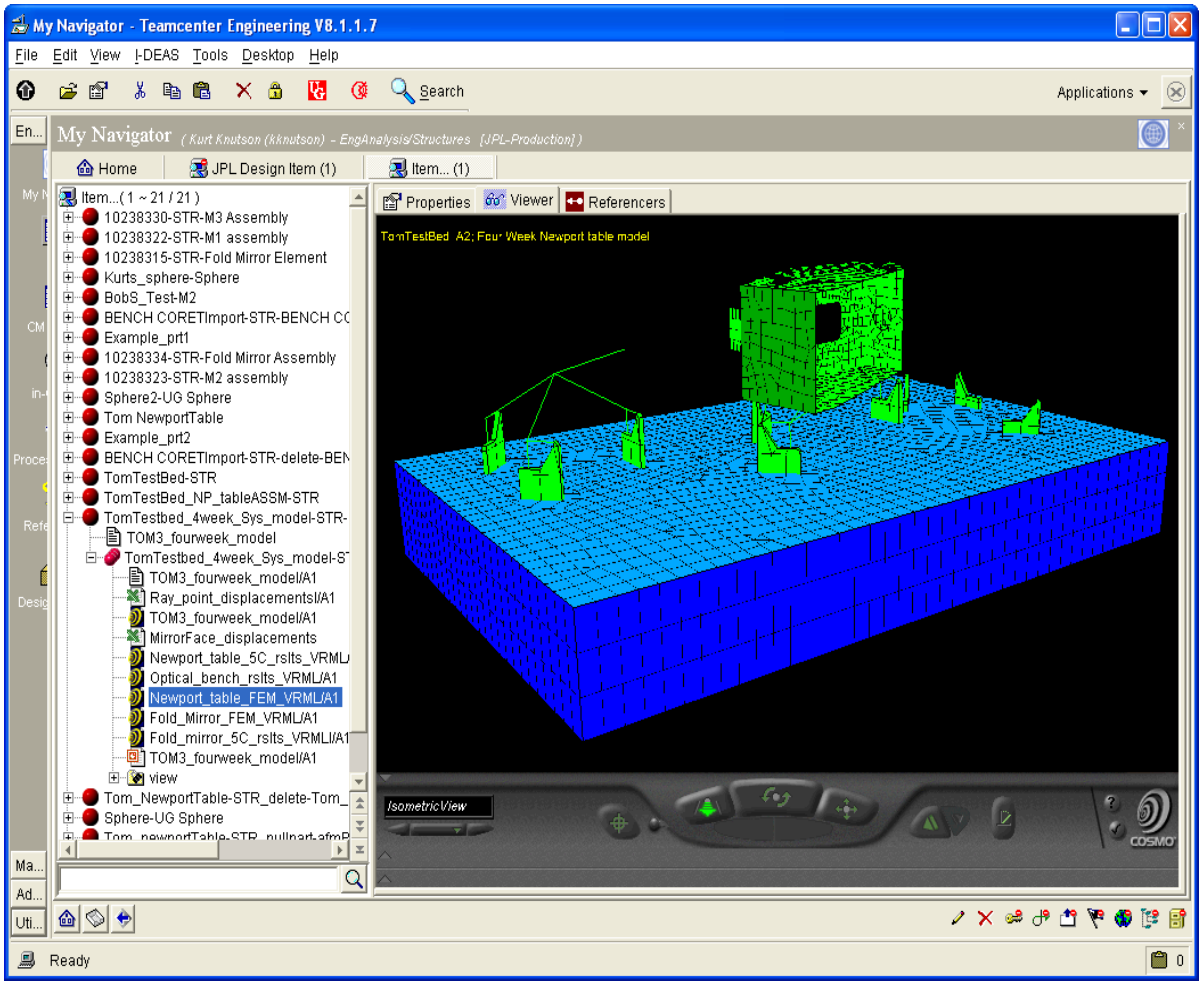
Excel file to describe models and output

Powerpoint document to summarize analysis model assumptions and results



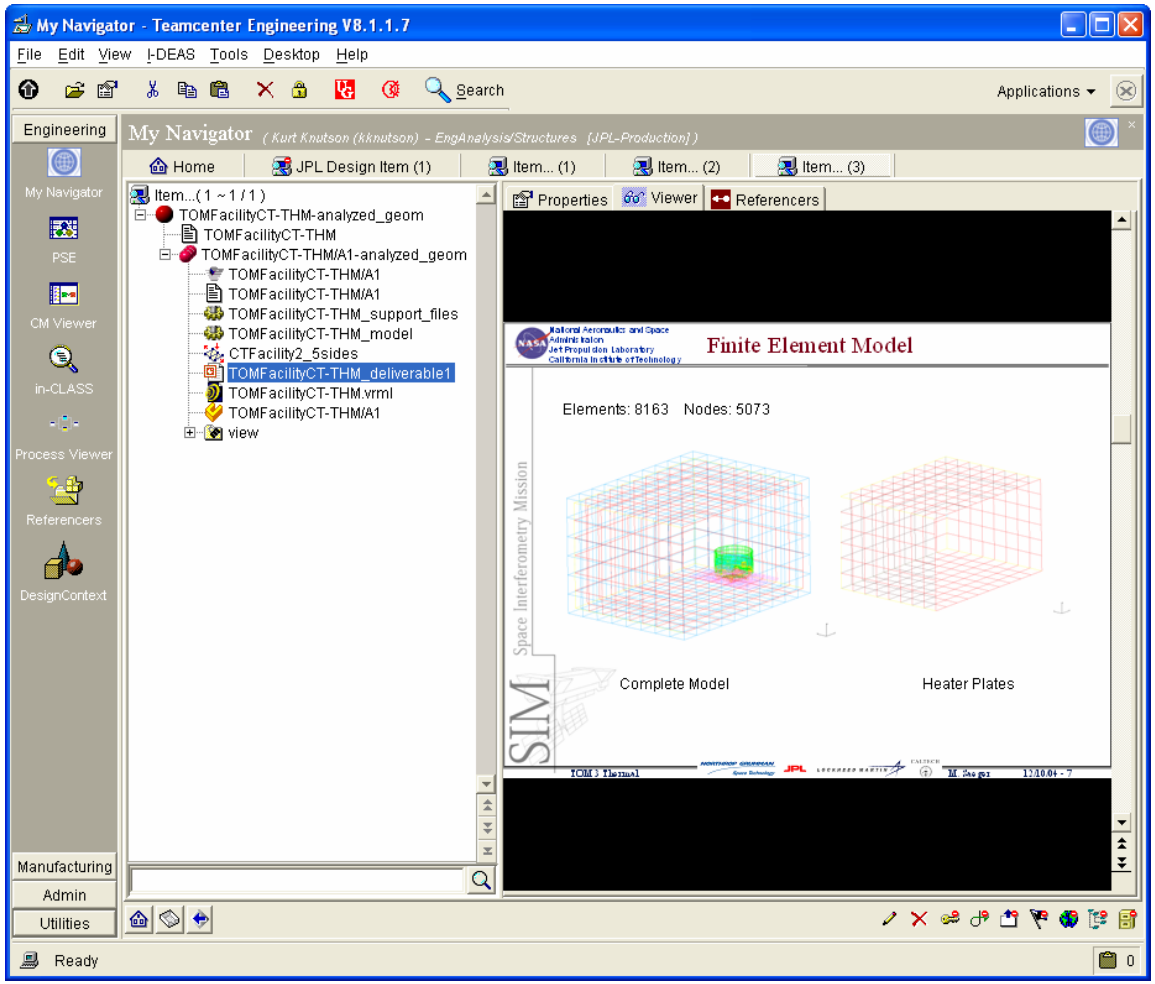
Example of VRML Output

- VRML Created with I-DEAS Visualizer
- Saved in TCEng as VRML DataSet
- Displayed in Viewer with Cosmo Player



Example of Results in PowerPoint

- Summarize Results in PowerPoint
- Save in TC Eng as PowerPoint DataSet
- View In Viewer
- OR Double Click to Launch in PowerPoint
- Use in the Conference Room!



Process Pilot Accomplishments

JPL practices for using TMG, I-DEAS and TC Eng with NX design geometry have been developed, tested and documented.

Integrated modeling of high precision optical systems has been demonstrated using commercial tools which scale for detail design flight hardware teams.

While the initial detailed modeling of high precision systems is still quite time consuming, a basic repeatable capability has been established and case analyses can be run in a few weeks.

Analysis modeling, meshing and solution techniques that assist update and re-use have been identified and demonstrated.

Attention has been drawn to the need for integrating analysis into configuration control and collaboration.

Acknowledgements

- ATA Engineering
  - Robert Schmitz
  - Mary Baker
- UGS
  - P.O. Mehta
- JPL
  - Robin Bruno
  - Kendra Short

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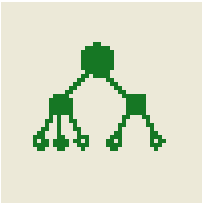
Back Up Information

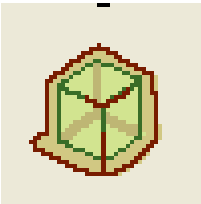
Overview of Meshing Methods with Assemblies: Method 1: Use the assembly to generate a part

I-deas Part that can be meshed with geometry in assembled orientation; Instances are branches on history tree

(UG or I-DEAS Assembly)

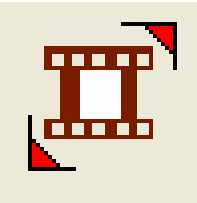


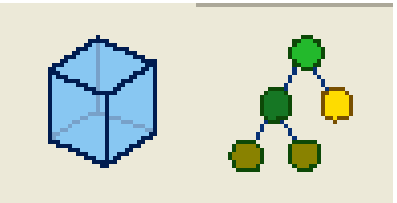
Assembly Hierarchy



Suppression of Instances

Configurations



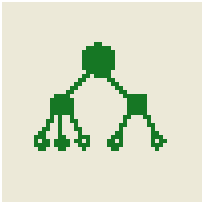


Use Assembly to Join, Join Partition, Add, or combination to get a part with assembly geometry;

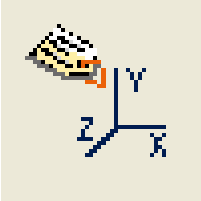
Parts can be updated or replaced as features, but orientation will have to be manually updated

Overview of Meshing Methods with Assemblies: Method 2: Parameterize assembly as a part

Fully parameterized, simplified part of assembly that can be modified geometrically as desired and meshed

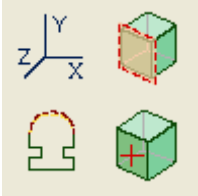


(UG or I-DEAS Assembly)



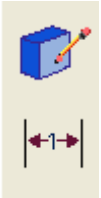
Create an I-deas part

By measuring and interrogating assembly data



Define useful interfaces, references, and coordinate systems

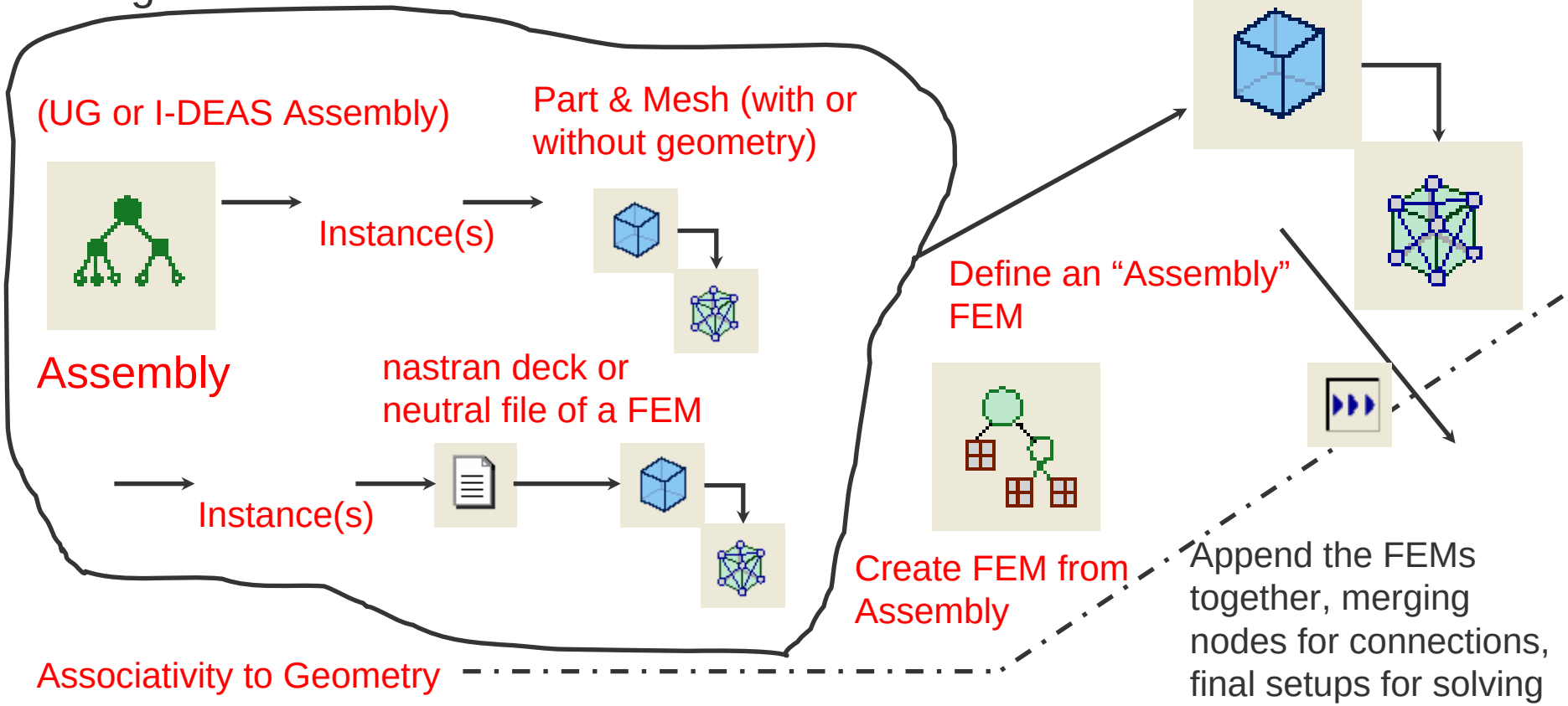
Sketch in place, constrain and dimension to build wireframe and parameterized features



Extrude, Revolve, Surface by Boundary, Partition, Split surface, etc., to get part with solids/surfaces

Overview of Meshing Methods with Assemblies: Method 3: Create FEM from Assembly

The “Create FEM from Assembly” Command defines mapping of instance orientations to FEMS to define and generate an “Assembled FEM”



Overview of Meshing Methods with Assemblies: Method 4: Hybrid methods

- Use any of the previously described methods to build a mesh to represent an assembly
- Considerations for choosing a geometry meshing strategy
  - Meshing detailed parts & Create FEM from Assembly a good choice when
    - Assembly contains multiple instances with chosen interfaces for meshes to match
    - The orientation of the instances is useful to reuse from an assembly
    - There are multiple configurations that can be used from the same part meshes (maybe thermal configurations, for example)
  - Generating a part from an assembly or modeling an assembly as a part can be a good choice when
    - There are certain key geometric variables to be changed and analyzed
    - Meshing of an automatic nature is acceptable
      - mapped volume bricks
      - free volume tetrahedral
      - surface meshing of quads or triangles

In I-DEAS a FEM is ALWAYS Created on a Part

- An I-DEAS FEM requires a part record to exist
- The part does not have to have geometry, but it will have coordinate systems
- When assemblies are used to create a FEM a part is still created
- If you import a bulk data deck to I-deas like from nastran, a part will be created
- Exporting/importing a “Simulation universal file” of just a FEM creates a FEM that references a part with no geometry

Notes on Create FEM from Assembly

- Because a FEM requires a part, this command creates a part for the FEM to reside on.
- A resulting FEM can depend on geometry in Assemblies, Parts, Meshes in Part FEMs, etc.
- If the FEMs are Created with associative techniques to parts, they can be modified at that level and a new “Assembled” FEM can be created for an update. This is typically simpler than updating the Assembled FEM (AFEM) for changes.
- It is a good idea to save a copy of a model file after defining (AFEM) but prior to generating a mesh from “Create FEM from assembly” as a restart point.
- “Create FEM from Assembly” essentially automates “appending” FE models together.
- The update method you use may depend on understanding what the changes are. This is why comparing assembly revisions can be so helpful.
- NXMI & TC Eng do not correctly save the AFEM part. You can manually save the model files as an opaque dataset in TC Eng for backup.