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
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

Data Management & Collaboration

Teamcenter Engineering

Design Data
- JPL Design Item
- JPL Design Item Revision
- CAD Data, Design Specs
- NX Manager, I-deas [Parts]

Thermal, Structural, Optical Data
- JPL Analysis Item
- JPL Analysis Item Revision
- CAE Data, Analysis Assumptions, Results, Reports

Interoperability
- UG Manager
- NX Manager, I-deas [Parts]
- Structural, Thermal Analysis
- Translator

CAE Data, Analysis Assumptions, Results, Reports
- NX Nastran
- Thermal, Structural, Optical Data

Design
- NX (Unigraphics)
- Abstraction for analysis
- Translator

[Assemblies, parts]
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

- JPL Design Item
- User Folder
- A Design Item Revision; flag indicates frozen
- ICON to start UG Manager
- The UG Master dataset, this is the ug part or assembly dataset under a revision

This is the BOM view of an assembly which can be sent to PSE or visualized.
I-DEAS Analysis Items and Datasets in TC Eng
Created Using NXMI

Direct Model (.jt file)
- Dataset of abstracted part; this is automatically created as visualization of an abstracted part and displayed in the viewer.

I-DEAS part dataset
- I-DEAS part dataset of abstracted part; imported and exported through NXMI only.

I-DEAS FEM dataset
- I-DEAS FEM dataset; imported and exported through NXMI only. No automated .jt files or standardized visualization mechanism in TC.
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
- This Thermal Part’s I-DEAS Model File
- Status Flag Indicates the Part is “Frozen” or Read-Only
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

1. Select the Design UGMaster
2. Select Send to I-DEAS
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

Change Name Following Naming Convention
Use Analysis Type Following Local Convention
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

New Part Has Design Part as Root Feature
Meshing Methods That Preserve Associativity

Automated Solid, Shell, Beam Mesh definitions and anchor nodes for meshes that track to solids, surfaces, curves in geometry;

Manual or Semi Manual Meshing Techniques

Mesh Numbering, Surface Normal Control, Material Orientation, Mass & Spring Elements, Sweeping, Extruding, Reflecting, Copying, Surface Coating, Solid from Shell
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
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.

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
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!
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.
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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

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)

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

Create an I-deas part

By measuring and interrogating assembly data

Define useful interfaces, references, and coordinate systems

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
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.