

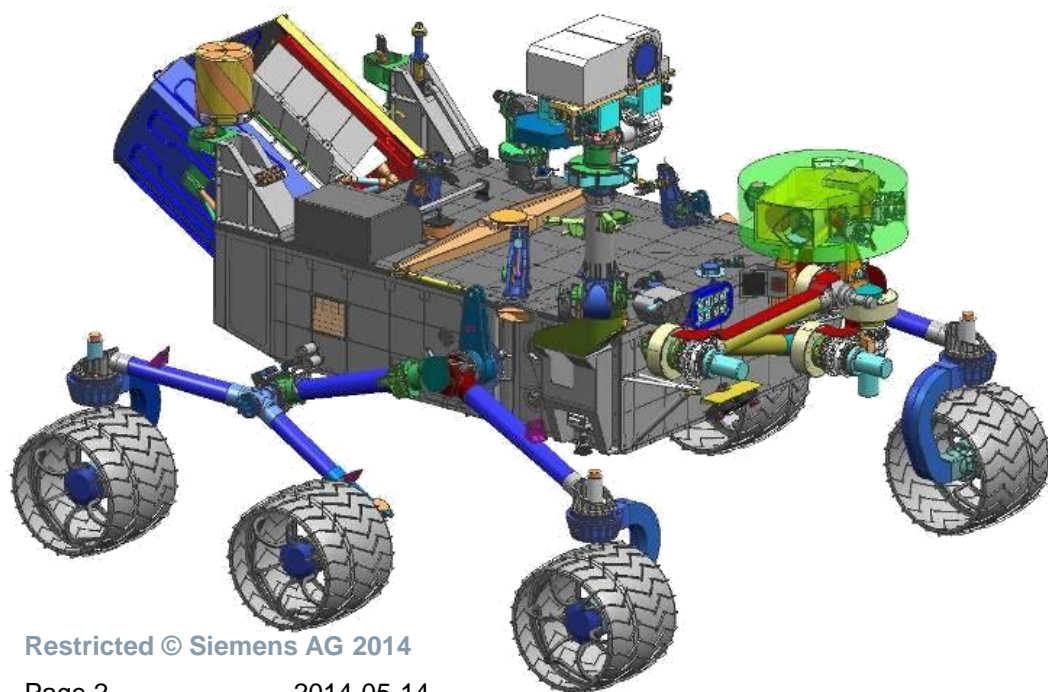


Christophe Vandavelde


NX CAE - NX Nastran Tips & Tricks

Agenda

- Reuse
- Assembly Fem
- Connection examples
- Debugging some related issues



Restricted © Siemens AG 2014



NX CAE

Elements of a Modern Open CAE Environment

Integrated Modeling and Solutions

- Delivers easy-to-use geometry editing and advanced FE capabilities for a range of physics solutions within a single, consistent user paradigm

System Level Modeling and Simulation

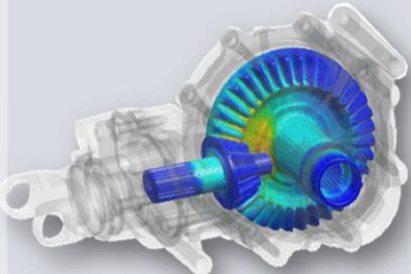
- Enables systems-level simulation with capabilities for building and managing FE assemblies, correlating system behavior with test data, and efficiently solving large models

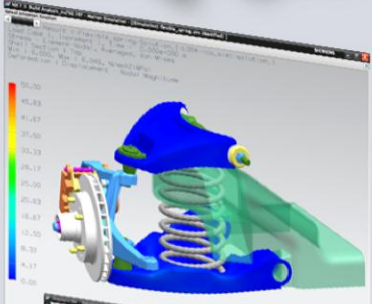
Multi-Discipline Simulation and Optimization

- Streamlines workflows for both individual analysts and CAE workgroups by coupling best-in-class solvers and solutions for multi-discipline and multi-physics simulation and optimization

Simulation Data and Process Management

- Enables CAE data, knowledge, and processes to be captured, shared, and re-used by more people within the organization

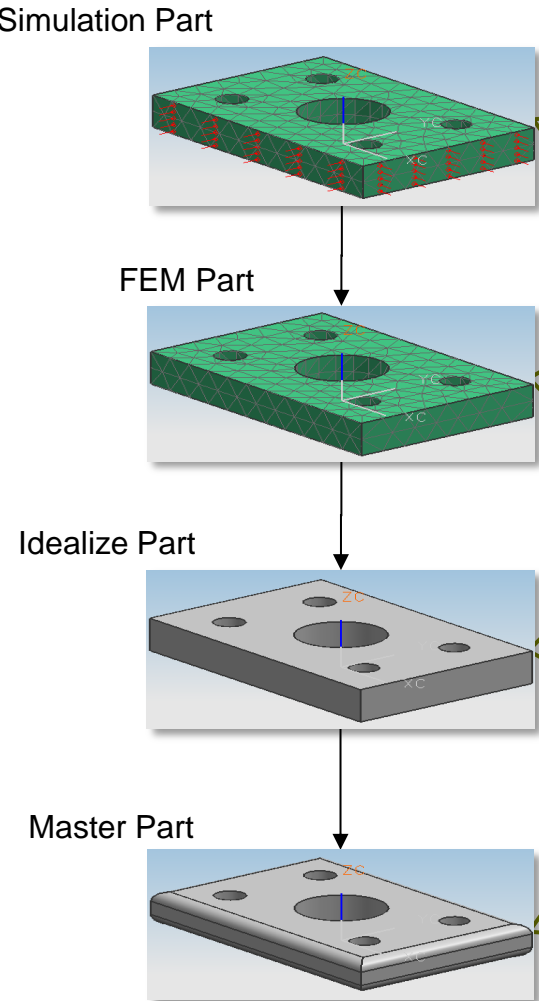




© Siemens AG 2011. All Rights Reserved.
Siemens PLM Software

NX CAE: Basic File Structure – Single FEM

- Benefits**
- Working in a concurrent environment
 - Efficient use of model and data reuse
 - Efficient use of local memory – not all files need to be loaded



Simulation Navigator

Name	Status
block_sim1.sim	Displaye
[-] block_fem1.fem	
[-] block_fem1_i.prt	
[-] block.prt	
[x] Mesh Controls	
[+] Polygon Geometry	
[+] 3D Collectors	
[] Fields	
[+] Groups	
[] Fields	
[] Simulation Object Container	
[+] Load Container	
[+] Constraint Container	
[-] Solution 1	Active
[x] Simulation Objects	
[+] Constraints	
[-] Subcase - Static Loads 1	Active
[x] Loads	
[x] Pressure(1)	
[] Results	Inferred

Simulation File View

Name	Status
Session	
[-] block_sim1	Displayed & ...
[-] block_fem1	
[-] block_fe...	
[-] block	

Part: block

Idealized Part

Create Idealized Part

Name: block_fem1_i.prt

Bodies

Geometry

Solver Environment

Solver: NX NASTRAN

Analysis Type: Structural

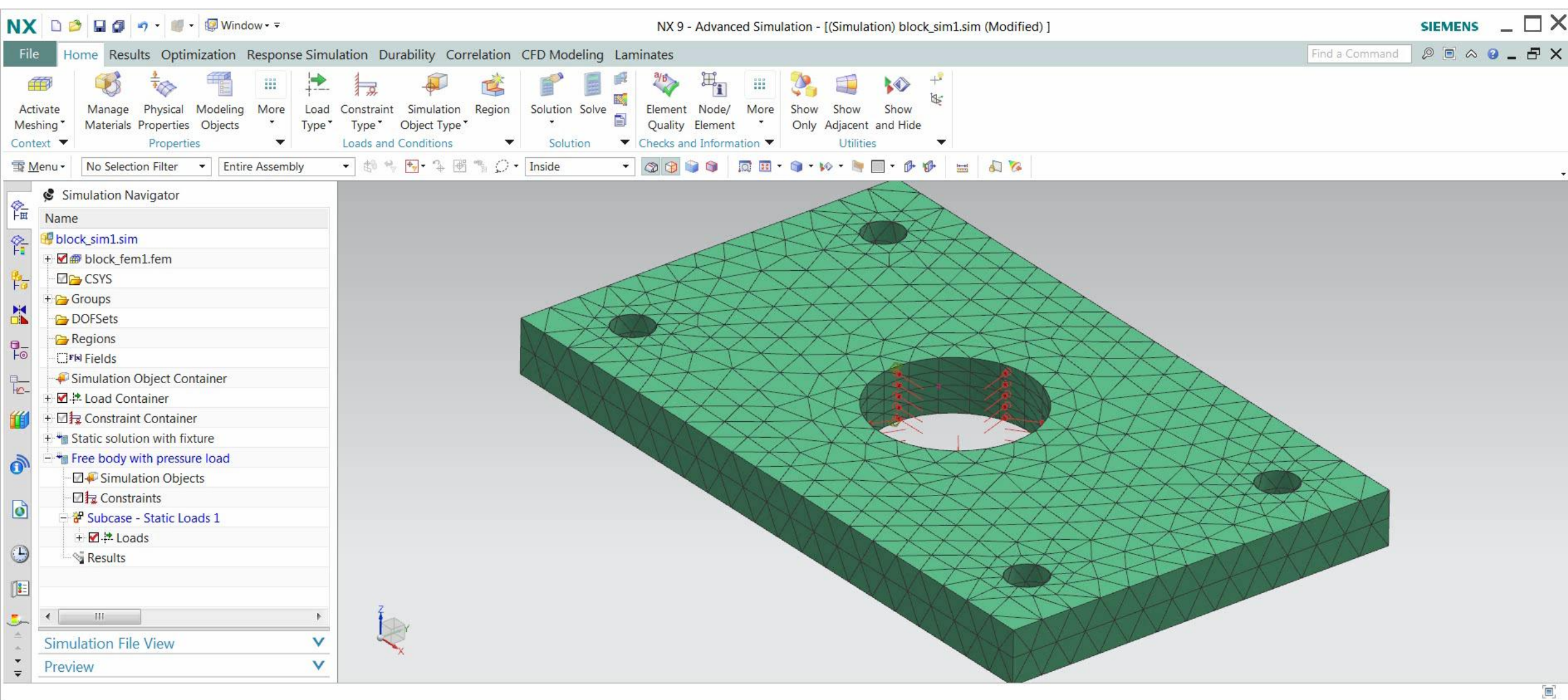
2D Solid Option: None

Description

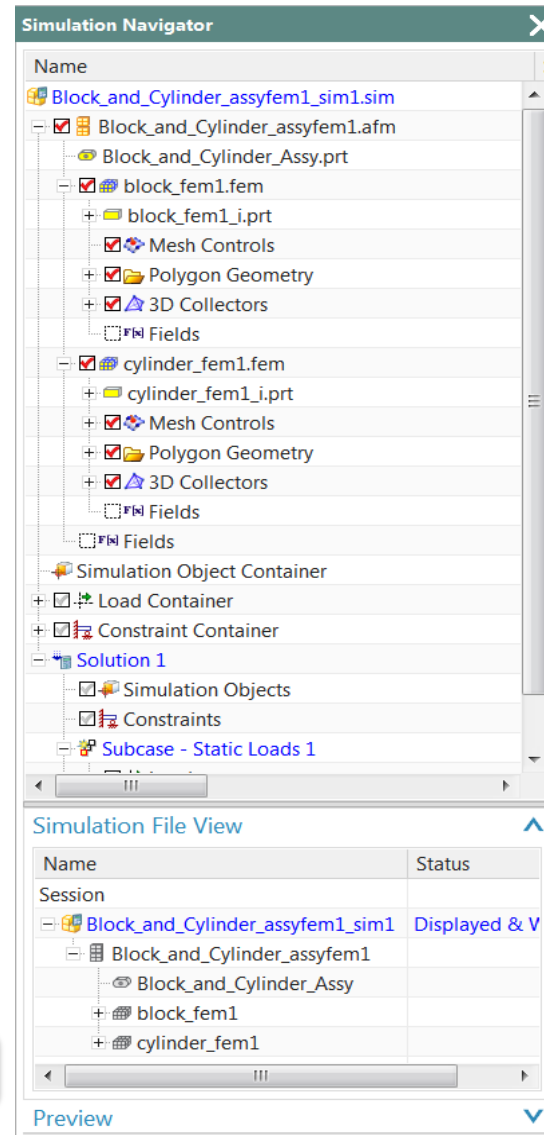
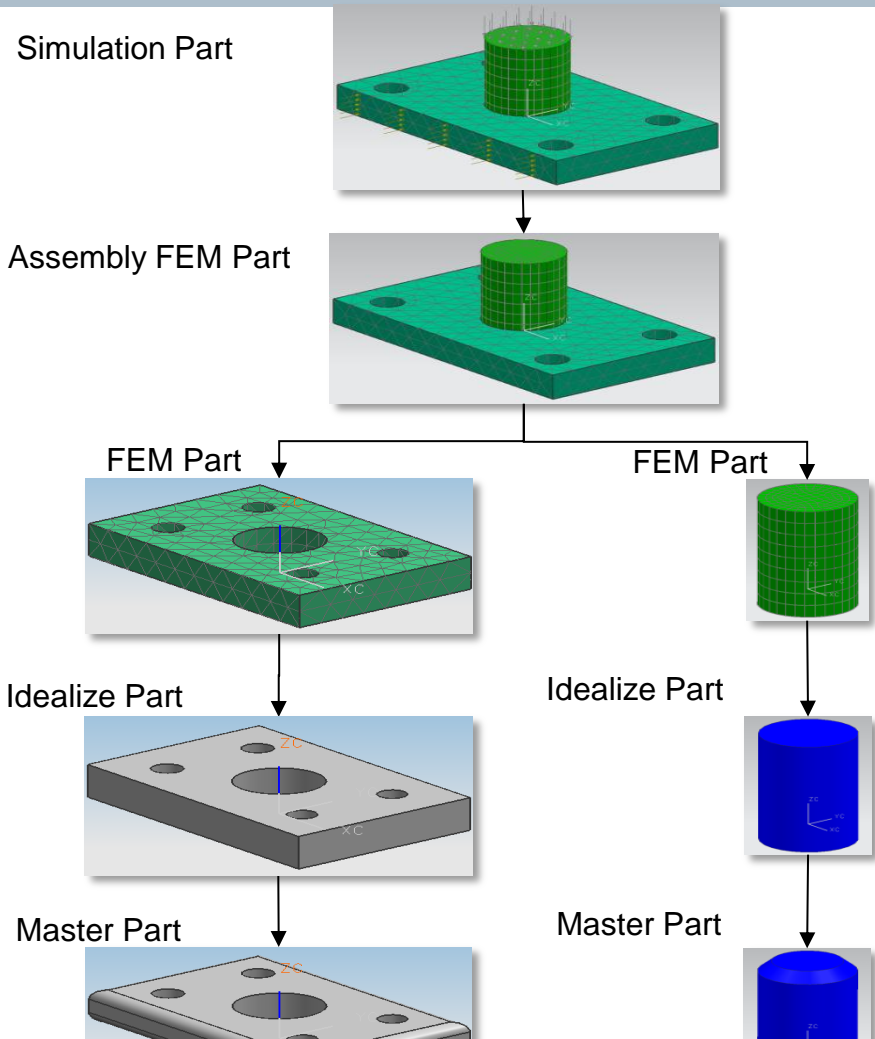
OK Cancel

Note – Write Access to Master Part is not required

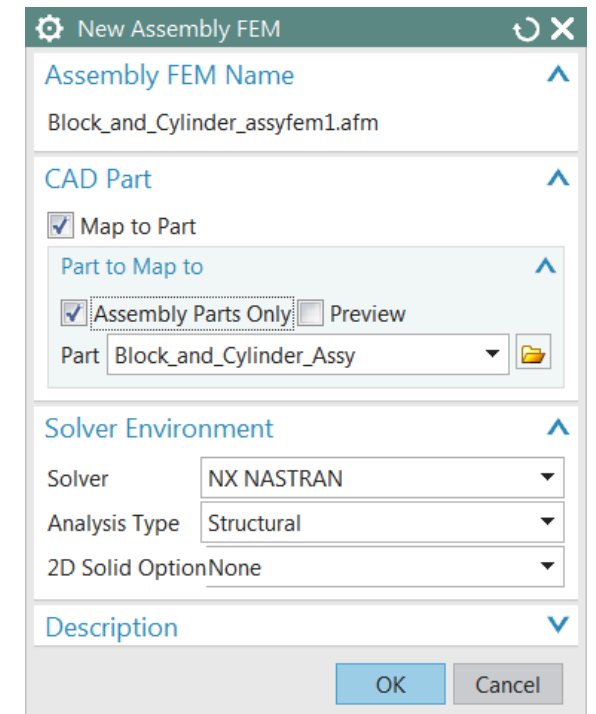
Reuse Fem/Sim + templates



NX Advanced Simulation: Basic File Structure – Assembly FEM



Benefits
 Working in a concurrent environment
 Efficient use of model and data re-use
 Efficient use of local memory – not all files need to be loaded



Note – Write Access to Master Part is not required

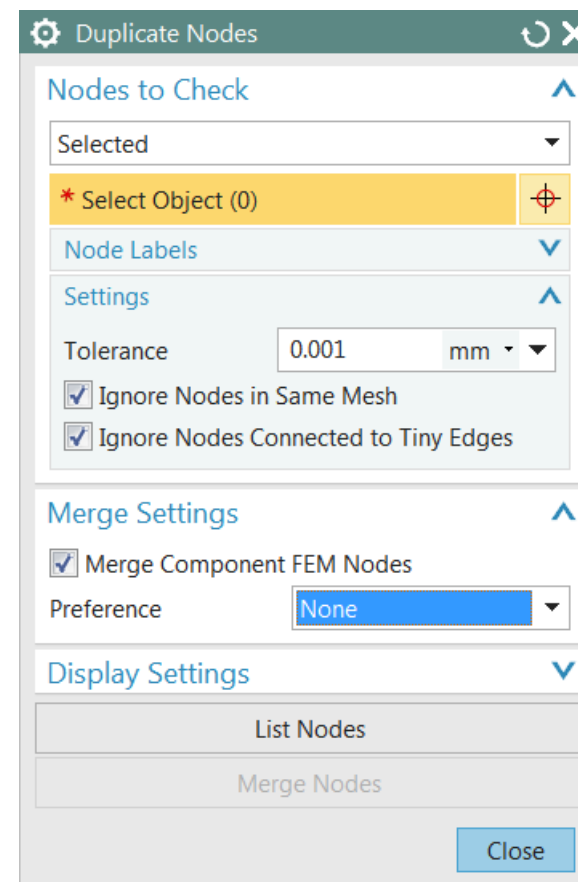
NX Advanced Simulation: Basic File Structure – Assembly FEM

- Assembly FEM can reuse the CAD assembly locations defined by the designer.
- Assembly FEM support single and multi level structures
- Distribute work among members of a team
- Improve the documentation and management of component meshes using assembly FEMs.
- Use and reuse existing component FEMs.
- Replace individual component FEMs with alternate mesh or geometry representations.
- Support workflow for external super elements.
- Less methods to make assy fem level connection between component compared to ‘single fem above assembly’ workflow
 - Operators that change component mesh like Mesh Mating, Stitch Edge,... not possible.

Connecting Component FEMs

If the FEMs brought into the AFEM have coincident nodes, the nodes can be merged at the assembly FEM level without modifying the individual FEMs.

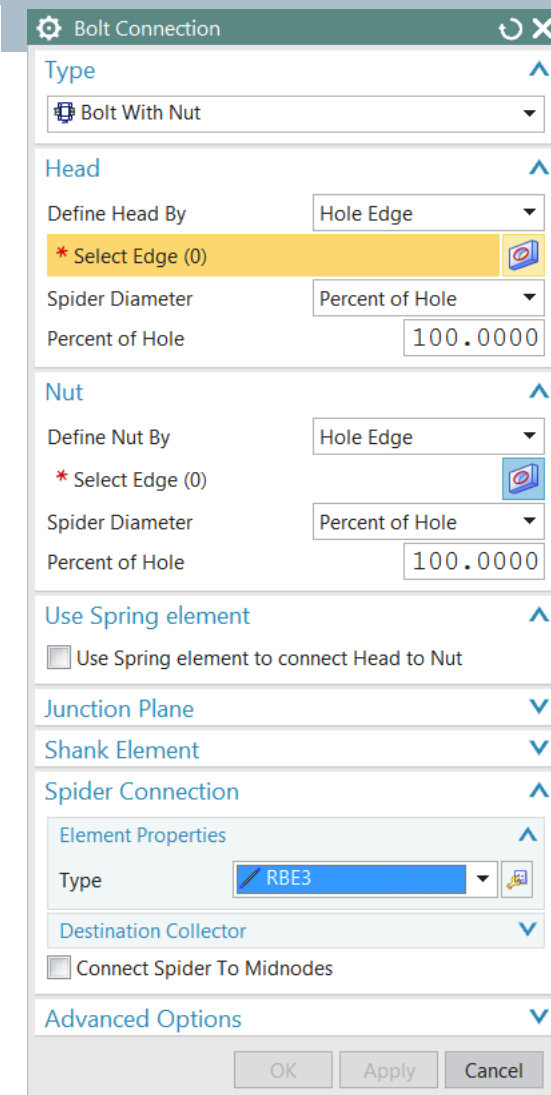
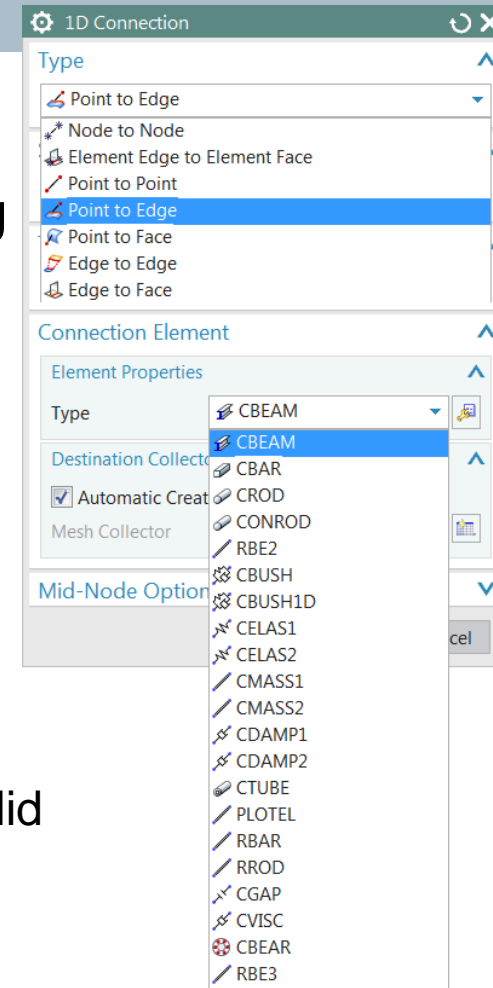
- The user may select the nodes or meshes to be checked or the software investigates the displayed model
- It is recommended that the user verify the duplicate nodes before merging them.



Connecting Component FEMs

In the assembly FEM file, you can define connection elements to join component FEMs into a system using the following tools:

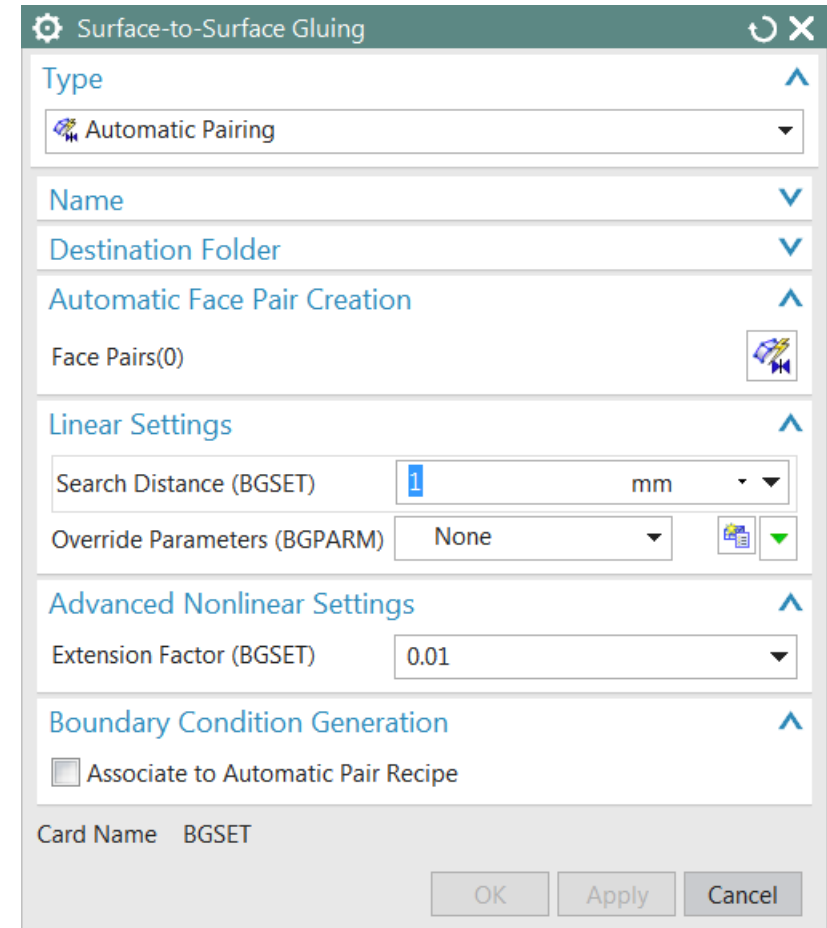
- Use the **1 D Connection** command to define 1 D connection meshes, spider elements, or connecting structures such as pins, bolts, or struts.
- Use **Bolt connection** method
- **CFAST – CWELD** connections
- Use **manual** node and element operations to create individual elements, including lumped mass, shells, or solid elements.
- Automation using **NX.Open** to support manual methods connection methods



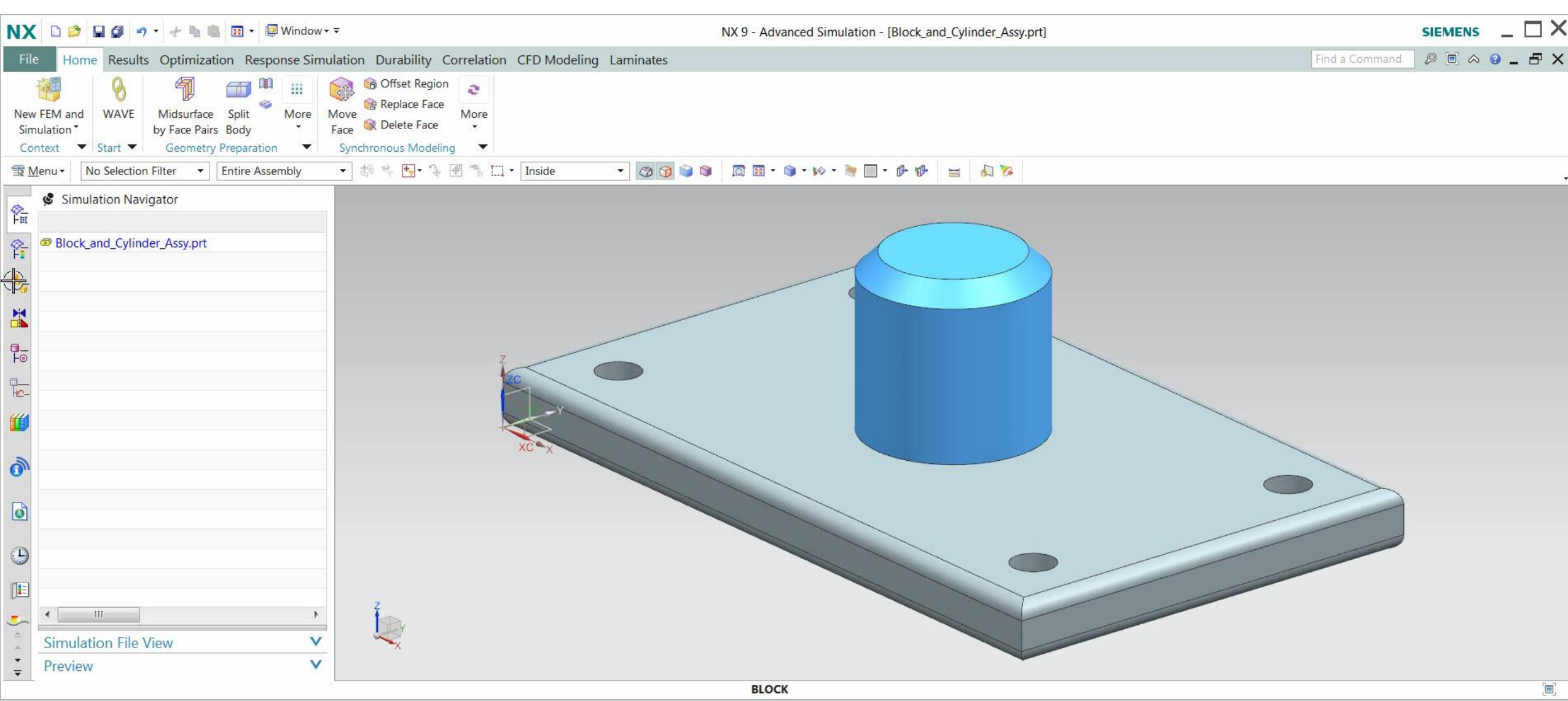
Connecting Component FEMs

In the Simulation file, you can use the Boundary conditions like:

- Surface-to-Surface Contact
- Surface-to-Surface Gluing
- Edge to Surface Gluing
- Edge to Edge Gluing
- Edge to Edge Contact (axisymmetric, plane stress and plane strain elements)
- MPC



Creating Assembly FEMs – Reuse LBC



NX CAE: An extensive set of connection tools

The decision on which tool to use depends on the model requirements and the project needs.

Connection techniques for NX Nastran

- 1D Connections
 - Node-to-Node
 - Element Edge to Element Edge
 - Point-to-Point
 - Point-to-Edge
 - Point-to-Face
 - Edge-to-Edge
 - Edge-to-Face
- Mesh – 3D Sweep Between
- Node Merge
- Mesh Mating
 - Glue Non-coincident (MPC's)
 - Glue Coincident
 - Free Coincident
- Edge Contact
- Surface Contact
- Simulation Regions
- Contact
 - Surface-to-Surface Contact
 - Linear Contact
 - Non-Linear Contact
- Glue
 - Surface-to-Surface Glue
 - Edge-to-Surface Glue
 - Edge-to-Edge Glue

- Bolt
 - Bolt with Nut
 - Bolt in Tapped Hole
 - Bolt with Spider at Junction
- Constraints
 - Coupling
- Weld Techniques
 - CFAST/CWELD
 - LOHR technique

Connection techniques in NX Thermal, Flow, ESC & Space Thermal

- 1D Connections
 - Node-to-Node
 - Point-to-Point
 - Point-to-Edge
 - Point-to-Face
 - Edge-to-Edge
 - Duct for thermal/flow transfer
- Surface-to-Surface Contact
- Surface-to-Surface Glue
- Thermal Coupling
- Articulation Joints for transient radiative heat exchange
- Recirculation Loop
- Interference Resistance
- Peltier Cooler
- Radiation Enclosure & Heating
- Convection and Radiation to Environment

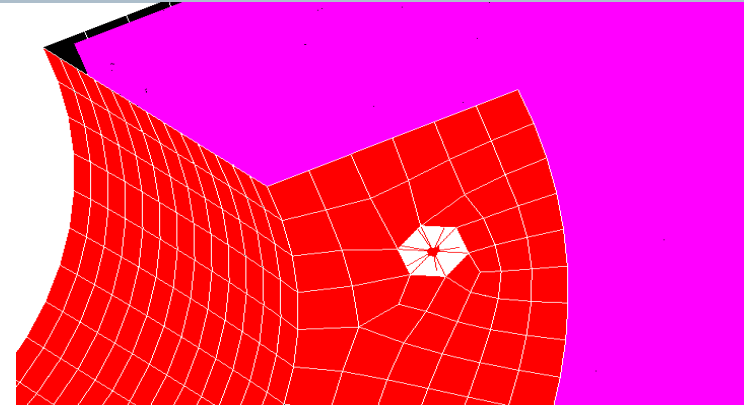
RBE2 - Rigid Elements

Considerations:

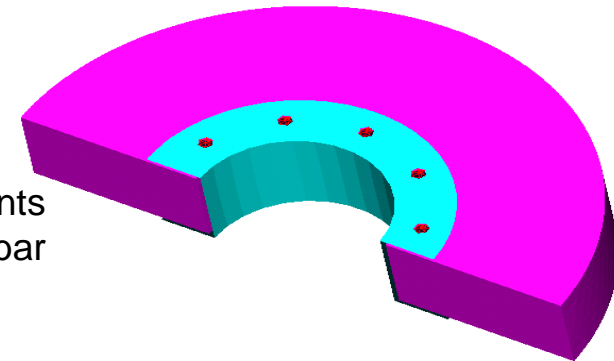
- All forces will be transferred
- No localized behavior considered
- Maintains shape of hole

Concerns:

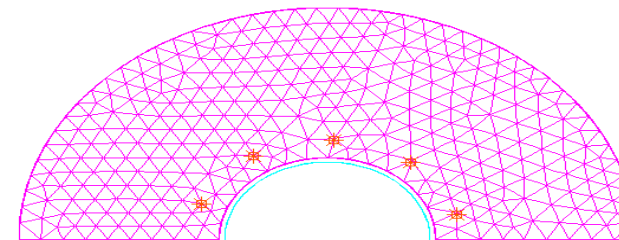
- Consider DOF of connected elements
- May add numeric stiffness
- If the nodes are not coincident, the resulting moment will be included
- Depended nodes: No constraints or other depended nodes connected



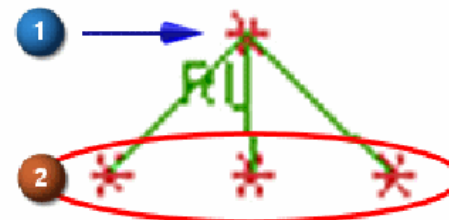
One rigid element connecting all the nodes to one central node



Two rigid elements connected with one bar



One rigid element connecting a node at the center of the hole



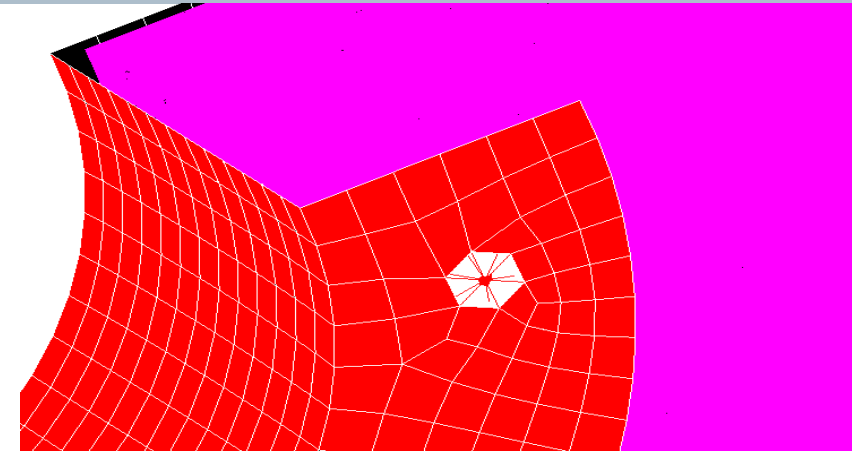
(1) independent node (2) depended nodes

RBE3 – Constraint Relation Elements

Considerations:

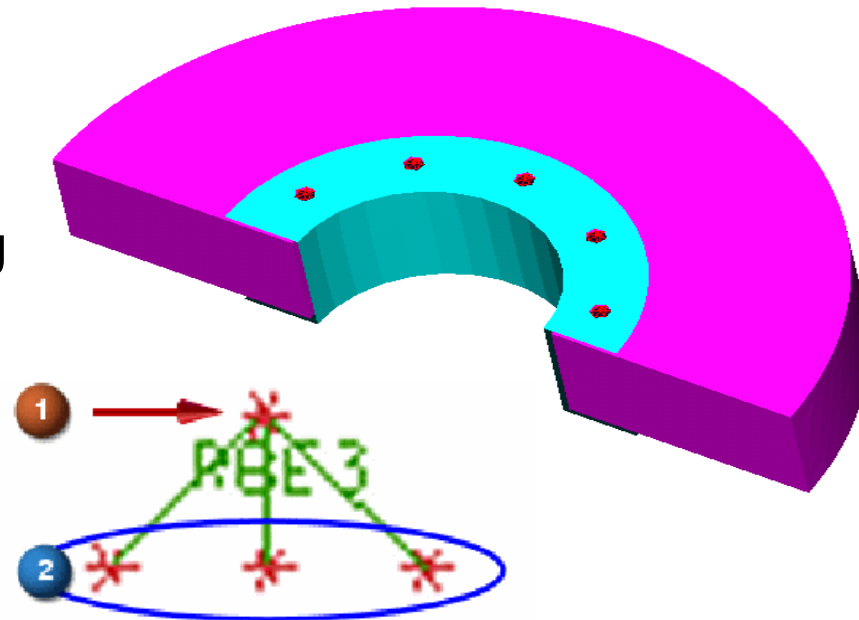
- All forces will be transferred
- No localized behavior considered
- Allows ovalization of hole

One constraint element connecting all the nodes to one central node



Concerns:

- Consider DOF of connected elements
- If the nodes are not coincident, the resulting moment will be included
- Depended node: No constraints or other depended nodes connected



Two constraint elements connected with one rigid bar or beam element

(1) dependent node (2) independent nodes

RBE2 example: DOF 3D Elements

The screenshot displays the Siemens NX 9 Advanced Simulation interface. The main window shows a 3D model of a green cube with a blue vertical bar on top, representing a RBE2 example with DOF 3D elements. The interface includes a menu bar (File, Home, Nodes and Elements, Results, Optimization, Response Simulation, Durability, Correlation, CFD Modeling, Laminates), a toolbar with various simulation tools, and a Simulation Navigator on the left. The Simulation Navigator shows the following structure:

- connection2_sim1.sim
 - connection2_fem1.fem
 - CSYS
 - Groups
 - DOFSets
 - Regions
 - Fields
 - Simulation Object Container
 - Load Container
 - Constraint Container
 - Solution 1
 - Simulation Objects
 - Constraints
 - Fixed(1)
 - Subcase - Static Loads 1
 - Loads
 - Gravity(1)
 - Results

The 3D model shows a green cube with a blue vertical bar on top. The coordinate system (XC, YC, ZC) is visible at the bottom left of the model.

RBE2 example: Depended Node

The screenshot displays the Siemens NX 9 Advanced Simulation environment. The main window shows a 3D model of a green cube with a blue vertical rod attached to its top surface. The rod is supported by a fixed constraint at its base. The cube is supported by a fixed constraint at its bottom-left corner. The software interface includes a menu bar, a toolbar, and a Simulation Navigator panel on the left.

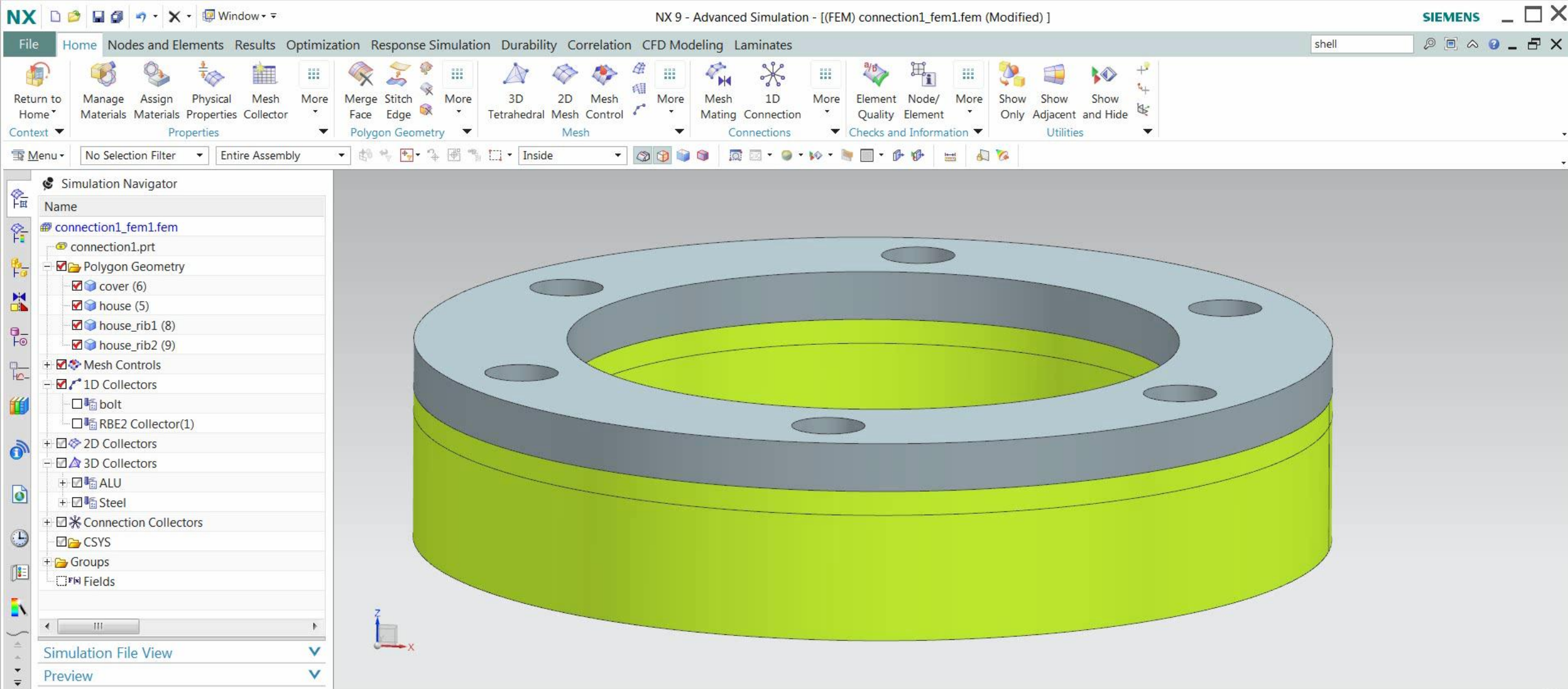
Simulation Navigator

- Name
- connection2_sim1.sim
 - connection2_fem1.fem
 - CSYS
 - Groups
 - DOFSets
 - Regions
 - Fields
 - Simulation Object Container
 - Load Container
 - Fixed(1)
 - Constraint Container
 - Fixed(1)
 - Solution 1
 - Simulation Objects
 - Constraints
 - Fixed(1)
 - Subcase - Static Loads 1
 - Loads
 - Gravity(1)
 - Results

Simulation File View

- Preview

RBE2 example: Bolt connection



Questions





SIEMENS



A stylized globe with a grid of latitude and longitude lines, rendered in a light blue/teal color against a dark background. The globe is centered on the Atlantic Ocean, showing parts of North America, South America, Europe, and Africa.

Christophe Vandavelde