SIEMENS

# NX Advanced FEM (Version NX 5)



UGS PLM Software

SIEMENS

# Aims

- There are 225 slides in this file, it is NOT the expectation that you show all of them to the prospect/customer
- The aim here is to provide a deck of slides that you can choose to “pick and mix” from to show a workflow or solution that is appropriate to your requirements
- These slides describe the “Core” functionality. I have skipped some icons/functions as they are only applicable for one of the add-on applications.
- These can be used as the build up for a presentation on one of the add-on applications like Laminates, Response Simulation, Thermal, Flow etc
- As these slides are all built to a consistent style, doing a “pick and mix” will still result in a clean looking presentation
- Please note that after the Solver Language Environment slides, everything is NX Nastran specific
- Please provide any enhancements or suggestions to

Guy.Wills@Siemens.com (+44 1462 44 5029)

Slides 2 – 6 are not intended for Public use

# Slide Organisation (1)

- NX Advanced FEM File Organization
  - Basic file structure
  - Idealize and Multiple FEM's
  - Multiple SIM's – Physical Property Override
  - Multiple SIM's – Physical Property & Thickness Override
  - Multiple Solutions and subcase's
  - Variations

- Model Interaction
  - Simulation Navigator – File View
  - Simulation Navigator – Easy Management
  - Simulation Navigator – Resource Bars
  - Interaction – RMB Over Screen Model
  - Mirror Display
  - Model Interaction – Show Only
  - Model Interaction – Show Adjacent
  - Model Interaction – Node Display
  - Model Interaction – Mesh Display
  - Model Interaction – Mesh Control Display

- Solver Language Environment
  - Solver Language Environment
  - "NX Nastran Environment" – UI Based on Solver/Solution
  - "ANSYS Environment" – UI Based on Solver/Solution
  - "ABAQUS Environment" – UI Based on Solver/Solution

- Master Part
  - Master Part
  - Material Property – Library
  - Material Properties

- Idealize Part
  - Part Idealize Part
  - Uses of the Idealize part
  - Idealize Part – Idealize
  - Idealize Part – Defeature Geometry
  - Idealize Part – Partition
  - Idealize Part – Midsurface
  - Idealize Part – Subdivide Faces
  - Idealize Part – Additional Modelling
  - Idealize Part – Direct Modelling
  - Idealize Part – Material Properties

Slides 2 – 6 are not intended for Public use

SIEMENS

# Slide Organisation (2)

- FEM Part
  - FEM Part
  - NX Advanced Simulation : CAE Topology
  - NX CAE Topology – Geometric Abstraction and Meshing
  - NX CAE Topology
  - NX CAE Topology – Auto Heal
  - NX CAE Topology – Split Edge
  - NX CAE Topology – Split Face
  - NX CAE Topology – Merge Edge
  - NX CAE Topology – Merge Face
  - NX CAE Topology – Match Edge
  - NX CAE Topology – Collapse Edge
  - NX CAE Topology – Face Repair
  - NX CAE Topology – Reset
  - NX CAE Topology – Mesh Updates
  - Physical Properties
  - Mesh Collectors
  - Node & Element Sets
  - Mesh Append
  - Mesh Import
  - Mesh Connections – Mesh Mating
  - Mesh Connections – Edge-Face Connection
  - Mesh Connections – Edge Contact Mesh
  - Mesh Connections – Surface Contact Mesh
  - Meshing – Mesh Points
  - Datum Coordinate Systems
  - Mesh Size Selection

- FEM Part (cont)
  - Mesh Controls
  - Meshing – OD Mesh
  - Meshing – 1D Element Cross Sections
  - Meshing – 1D Mesh
  - Meshing – 1D Mesh – Element Attributes
  - Meshing – 2D Dependant Mesh
  - Meshing – 2D Mapped Mesh
  - Meshing – 2D Mesh
  - Meshing – 2D Mesh Seeding for 3D Mesh
  - Meshing – 3D Swept Mesh
  - Meshing – Solid from Shell Mesh
  - Meshing – 3D Tetrahedral Mesh
  - Meshing – Node Create
  - Meshing – Node Between Nodes
  - Meshing – Node on Curve/Edge
  - Meshing – Node Translate
  - Meshing – Node Rotate
  - Meshing – Node Reflect
  - Meshing – Node Drag
  - Meshing – Node Align
  - Meshing – Node Displacement CSYS
  - Meshing – Node Re-Numbering
  - Meshing – Node Modify Coordinate
  - Meshing – Node Deletion
  - Meshing – Node & Element Information
  - Meshing – Node Displacement CSYS

- FEM Part (cont)
  - Meshing – Element Create
  - Meshing – Element Extrude
  - Meshing – Element Revolve
  - Meshing – Element Translate & Copy
  - Meshing – Element Copy & Project
  - Meshing – Element Copy & Reflect
  - Meshing – Shell Split
  - Meshing – Combine Tris
  - Meshing – Move Mode
  - Meshing – Element Re-label
  - Meshing – Element Connectivity
  - Meshing – Element Deletion
  - Meshing –Node & Element Information
  - Meshing – Mesh Unlock
  - Model Checking – Element Shape
  - Model Checking – Element Outlines
  - Model Checking – Duplicate Nodes
  - Model Checking – Element Normals

Slides 2 – 6 are not intended for Public use

# Slide Organisation (3)

- SIM Part – Pre-Processing
  - Modeling Objects – Manager
  - Modeling Objects – Contact Set Parameters
  - Modeling Objects – Strategy Parameters
  - Modeling Objects – Real Eigenvalue, Lanczos & Householder
  - Modeling Objects – Forcing Frequencies – Direct & Modal
  - Modeling Objects – Time Step
  - Modeling Objects – Structural Output Requests
  - Modeling Objects – Solution Parameters
  - Modeling Objects – System Cells
  - Surface to Surface – Contact
  - Surface to Surface – Glue
  - Loads – Force
  - Loads – Bearing
  - Loads – Torque
  - Loads – Moment
  - Loads – Pressure
  - Loads – Hydrostatic Pressure
  - Loads – Gravity
  - Loads – Centrifugal
  - Loads – Constant Temperature
  - Loads – Nodal Force Location
  - Constraints – User Defined
  - Constraints – Enforced Displacement
  - Constraints – Fixed, Translation & Rotation
  - Constraints – Simply Supported

- SIM Part – Pre-Processing (cont)
  - Constraints – Slider
  - Constraints – Pinned
  - Constraints – Cylindrical
  - Constraints – Roller
  - Constraints – Symmetric
  - Constraints – Anti-Symmetric
  - Constraints – Velocity
  - Constraints – Acceleration
  - Constraints – Automatic Coupling
  - Constraints – Manual Coupling
  - Constraints – Enforced Motion Location
  - Boundary Condition Symbol Display Controls
  - Physical Property Overrides
  - Custom Units & Units Converter
  - Unit Selection
  - Boundary Condition Magnitude – Table Field
  - Boundary Condition Magnitude – Function Field
  - Solution
  - Solution – Containers and Re-using Data
  - Solution – Subcase Management
  - Solution – Attributes
  - Solution – Parameters
  - Solution – Comprehensive Check
  - Solution – Report Before Solve
  - Solution – Solve the Active Solution

Slides 2 – 6 are not intended for Public use

# Slide Organisation (4)

- SIM Part – Post-Processing
  - NX – Integrated Post Processing
  - Results – Selection
  - Results – Animation
  - Results – Post View Display
  - Results – Post View Color Bar
  - Results – Post View Edges & Faces
  - Results – Identify
  - Results – Annotation Markers
  - Results – Previous / Next Mode or Iteration
  - Results – Post Views & Templates
  - Results – Multiple Viewports
  - Results – Post View Overlay
  - Plotting Paths
  - Graph Style
  - Graph Probing
  - Graph Windowing
  - Solution Report – After Solve
  - Export Visualisation Files

- Simulation Customer Defaults
  - Customer Defaults – General
  - Customer Defaults – Model Preparation
  - Customer Defaults – Mesh Display
  - Customer Defaults – Node & Element Display
  - Customer Defaults – Mesh Controls
  - Customer Defaults – Boundary Condition Display
  - Customer Defaults – Threshold Values Nastran
  - Customer Defaults – Meshing
  - Customer Defaults – Analysis
  - Customer Defaults – Post Processor

Slides 2 – 6 are not intended for Public use

SIEMENS

# NX Advanced FEM File Organisation



UGS PLM Software

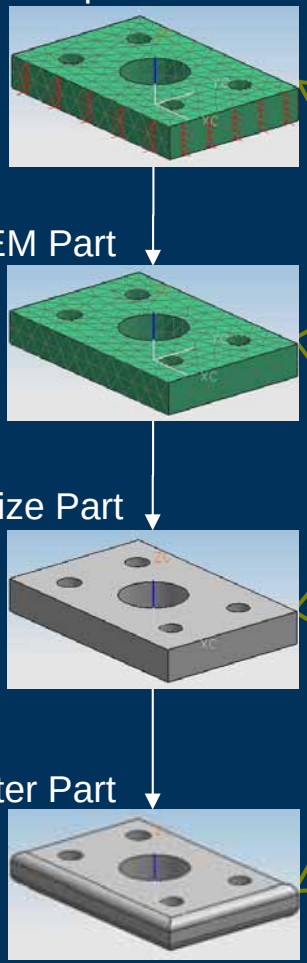
# Basic file structure

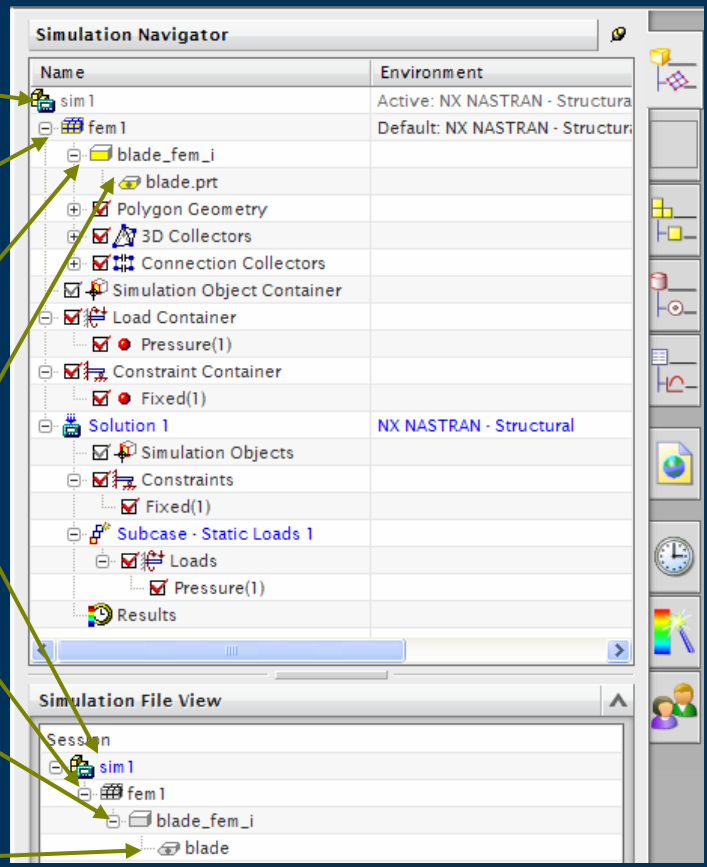
Simulation part

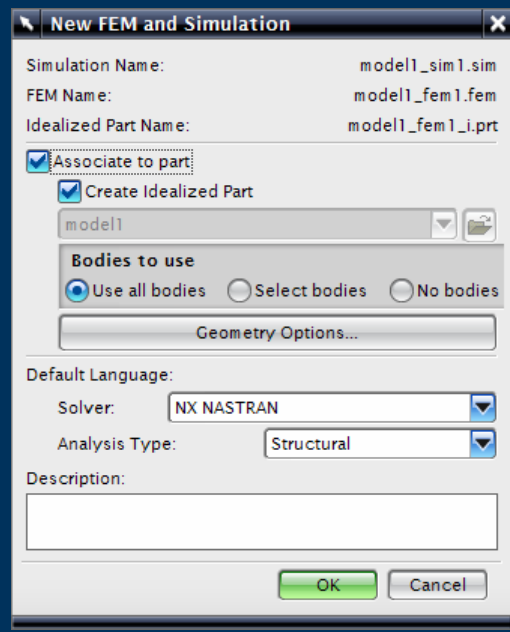
FEM Part

Idealize Part

Master Part






## 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

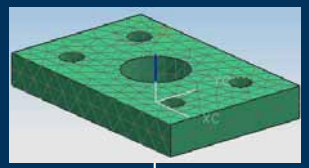
Note – Write Access to Master Part is not required

SIEMENS

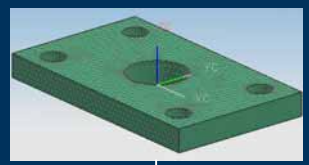
# Multiple Idealize and Multiple FEM's

Benefits
- Multiple analyses for same CAD part
- Multiple representations for different analysis needs from same CAD part

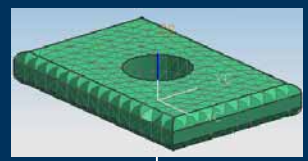
FEM1



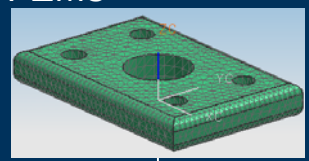
FEM4



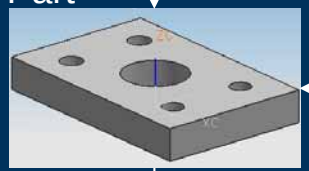
FEM2



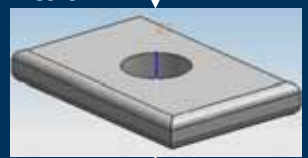
FEM3

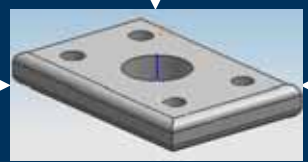


Idealize Part



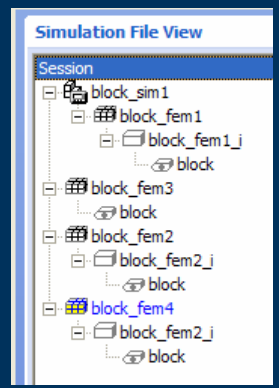
Idealize Part



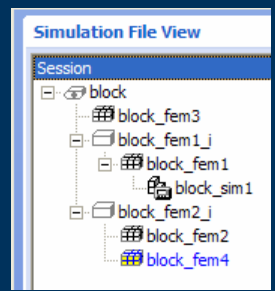


Master Part

CAE user view



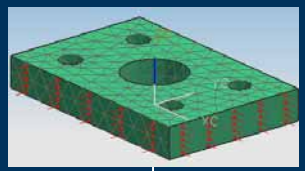

CAD user view



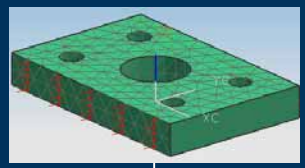
SIEMENS

# Multiple SIM's – Physical Property Override

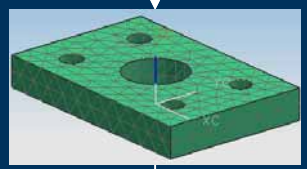
SIM1 – Generic Steel (Inherited from Master Part)



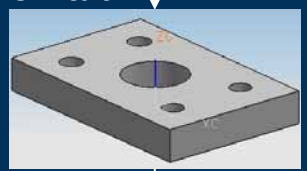
SIM2 – AISI_STEEL_1008+HR



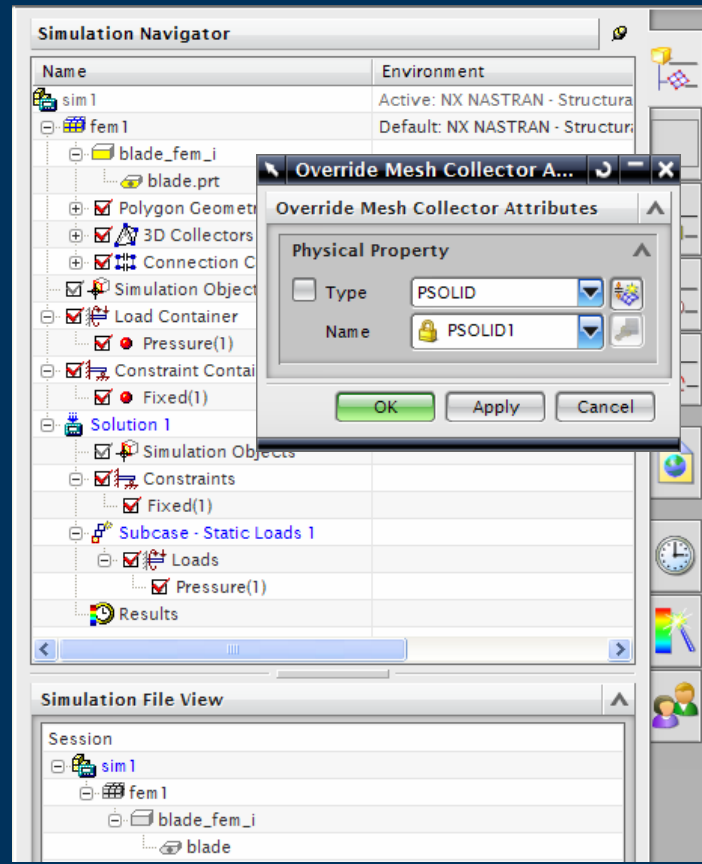
FEM1



Idealize Part



Master Part



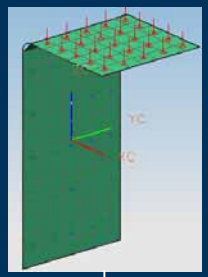

## Benefits

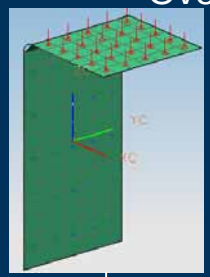
- Quickly and easily explore effects of different materials
- "What if" and sensitivity studies

# Multiple SIM's – Properties & Material Override

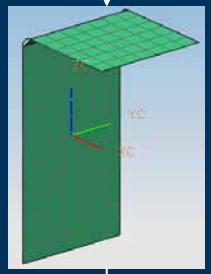
SIM1 –Inherited Properties

SIM2 – Properties & Material Overrides

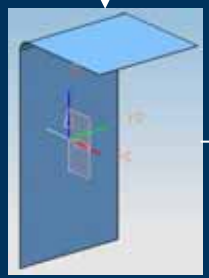




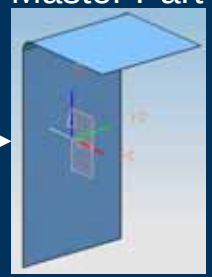
FEM1

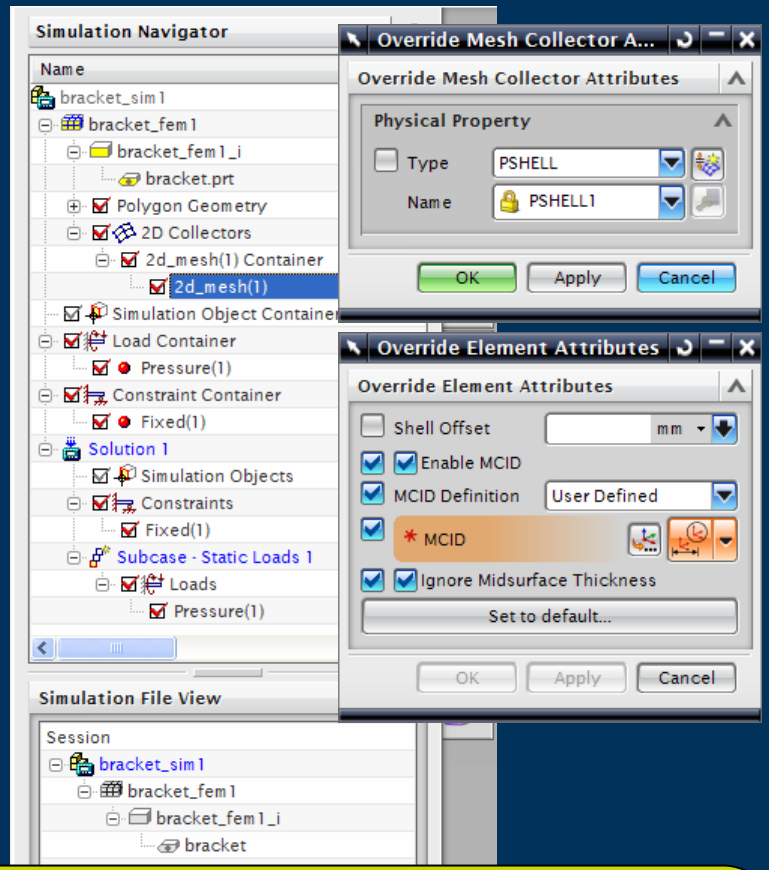


Idealize Part



Master Part

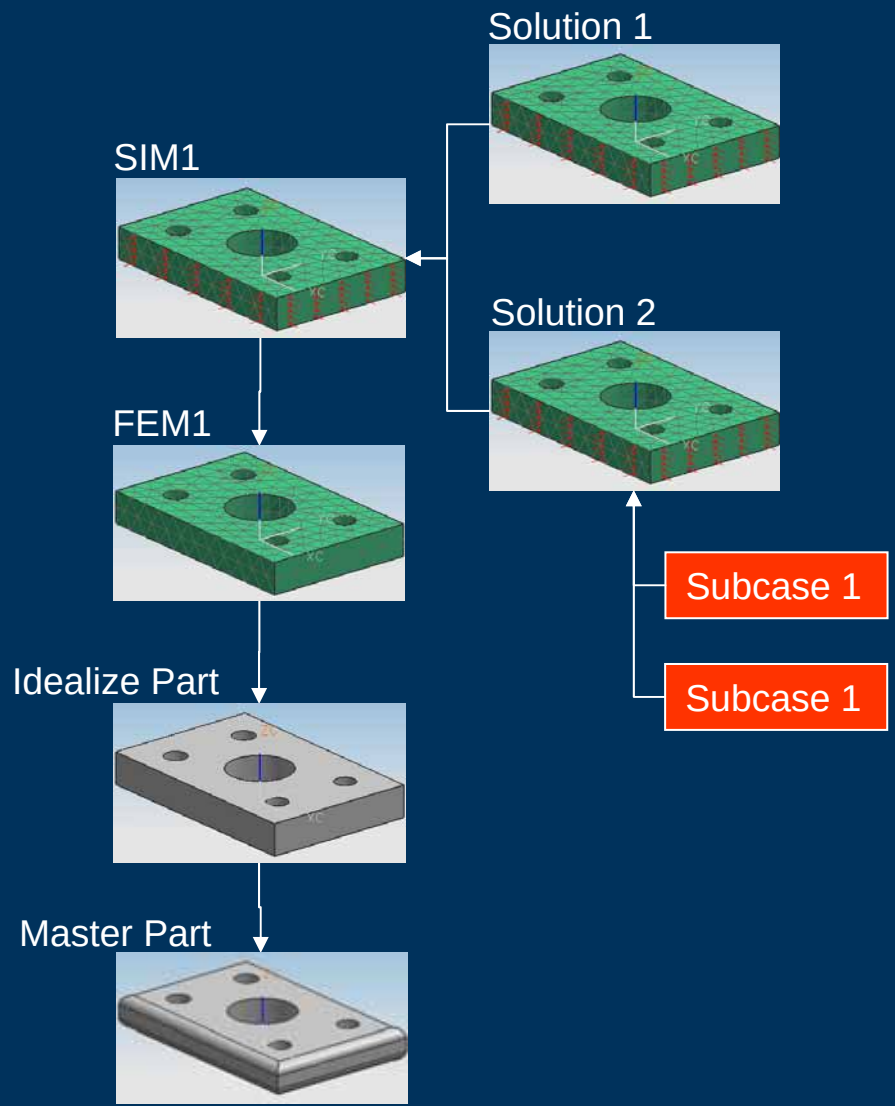





## Benefits

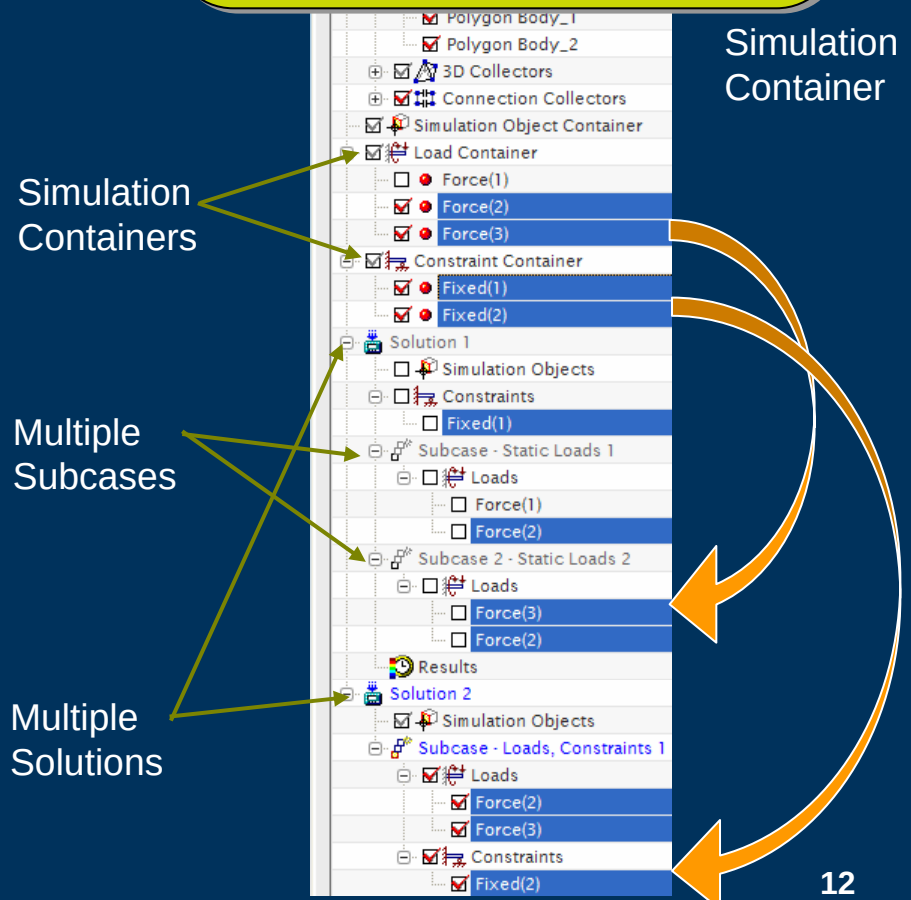
- Quickly and easily explore effects of different materials, thicknesses, shell offsets etc
- "What if" and sensitivity studies

# Multiple Solutions and Subcase's

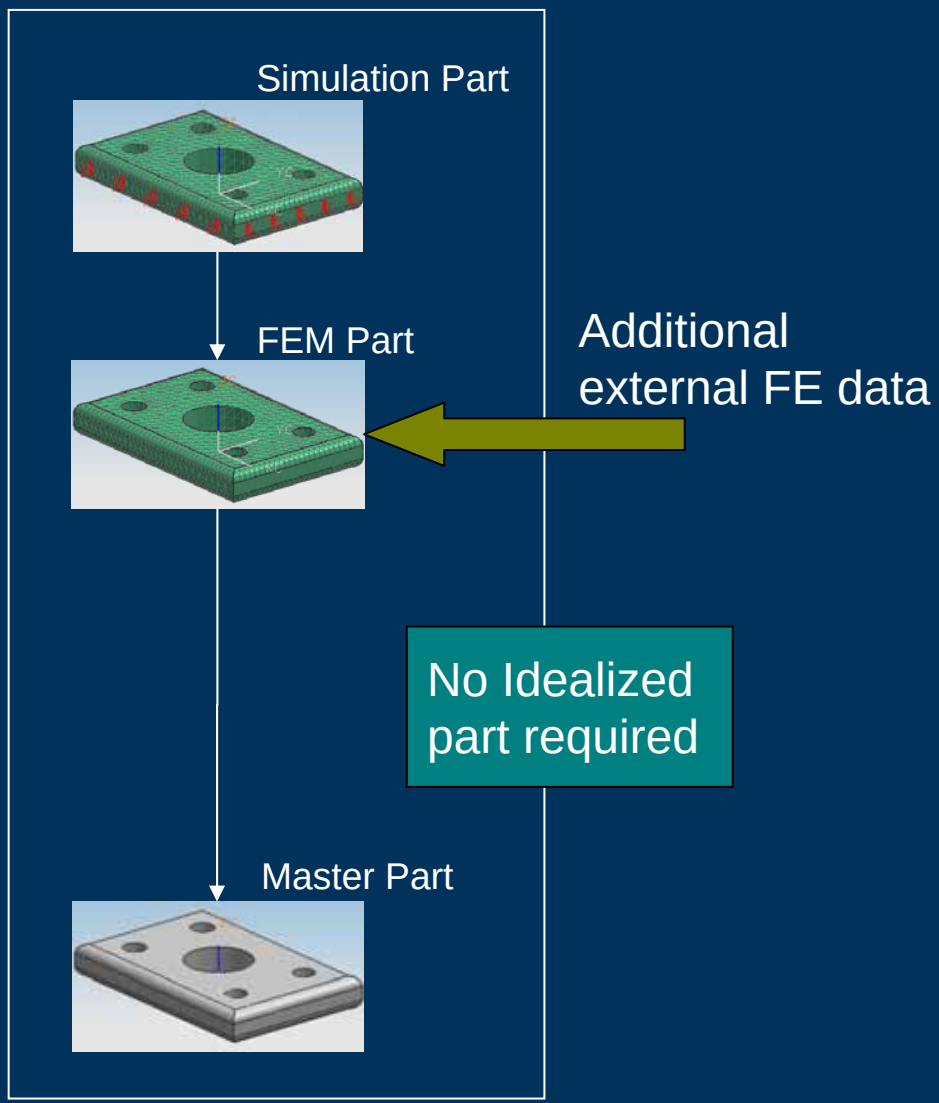


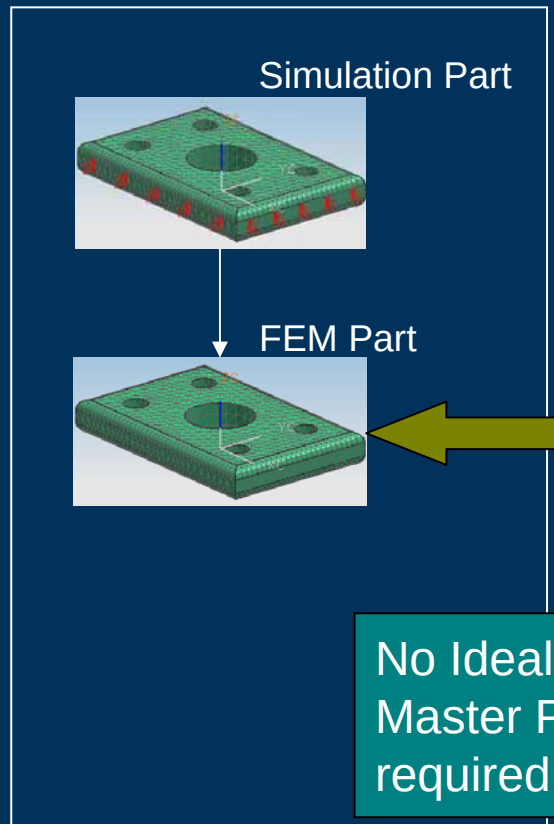

## Benefits

- Quickly and easily explore effects of different loading conditions
- Efficient analysis in complex environments



# Variations




Benefits

- Quickly and easily include external FE data
- Flexible work flows – work the way you want to work

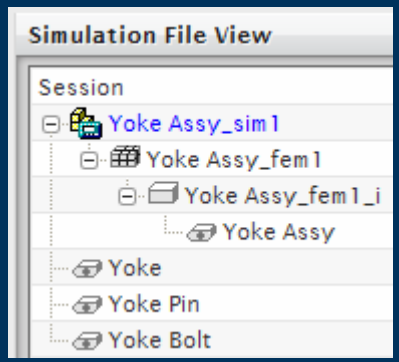
SIEMENS

# Model Interaction

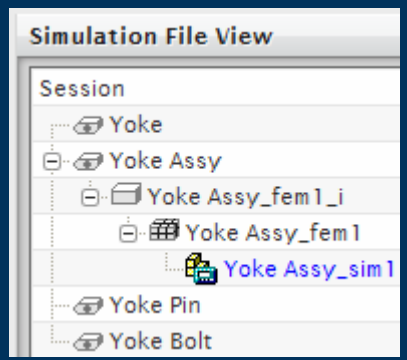


UGS PLM Software

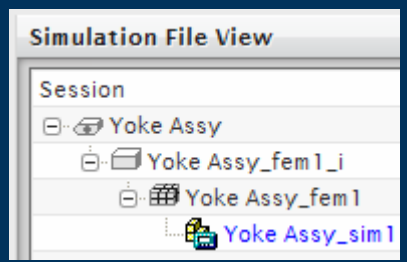
# Simulation Navigator – File View




Simulation Centric View

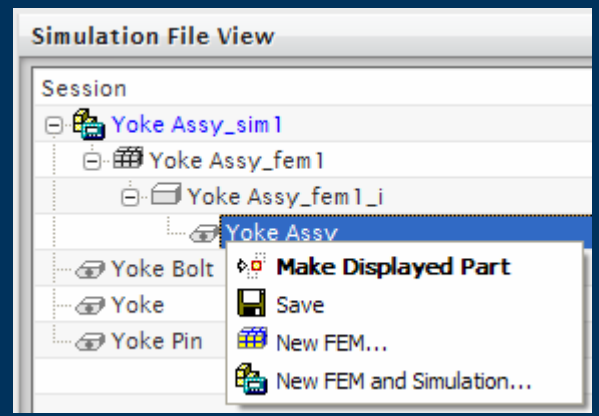



Design Part Centric View




Design Part Centric View with Unused Parts Hidden

- Simulation File View
  - Simulation Centric View for the analysts
  - Design Part Centric View for the inverted view
  - Active File shown in Blue
  - Optionally any unused and open parts can be hidden
- Easy to understand the file relationships
- Fast method of “Switching” active file – Double Click
- Fast method to create new Simulation files



SIEMENS

# Simulation Navigator – Easy Management

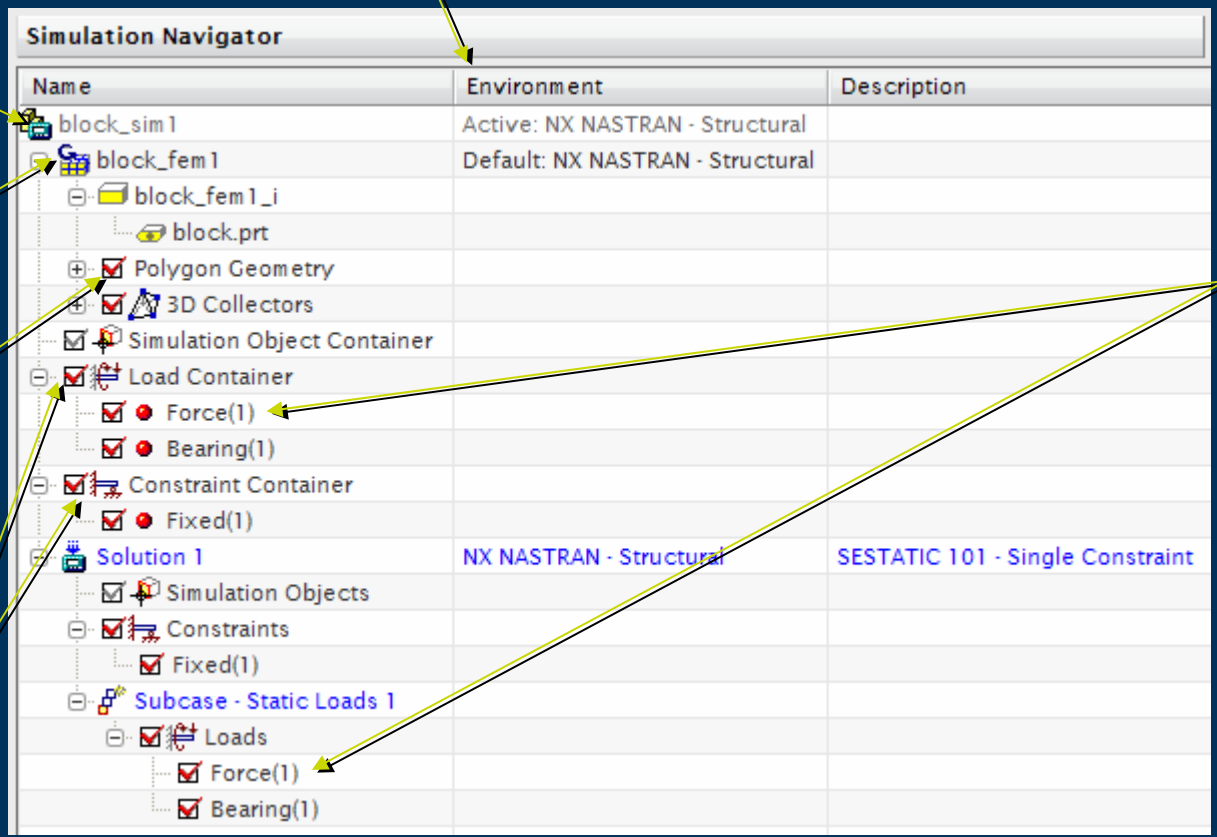
Simulation Centric File View

Active Solver Environment

Mesh Out-of-Date Symbol

Hide/Show of Polygon models and Meshes during selection

Containers to Organise related CAE Data

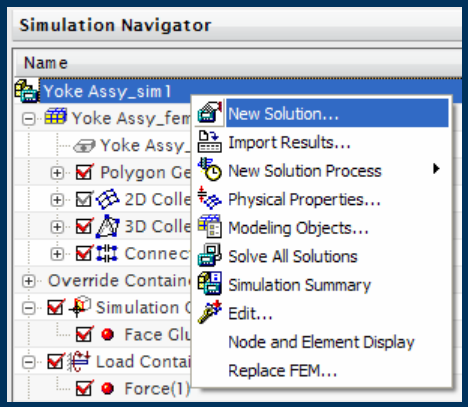


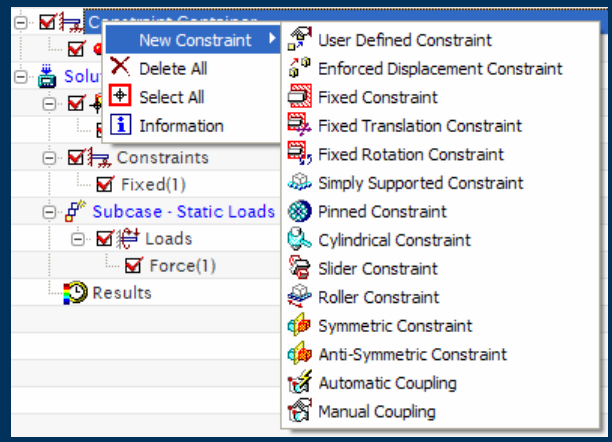

Drag ‘n’ Drop from Containers to Solution

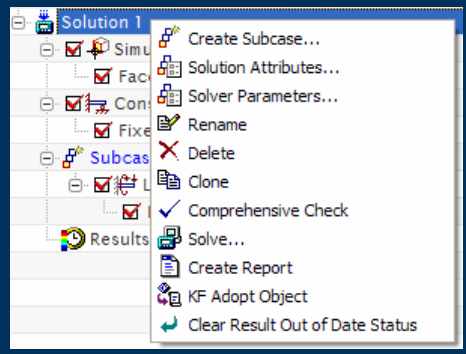
SIEMENS

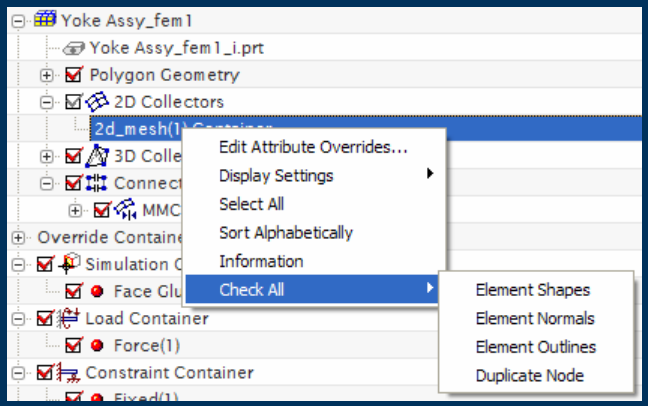
# Simulation Navigator – Easy Management

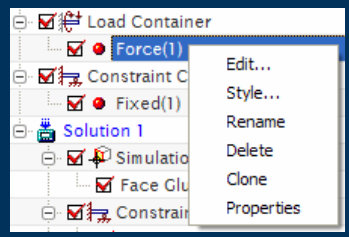
- RMB Operations Directly from Navigator

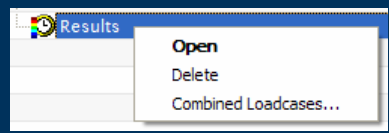


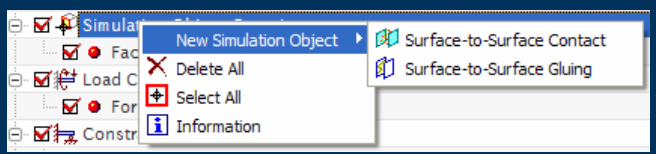



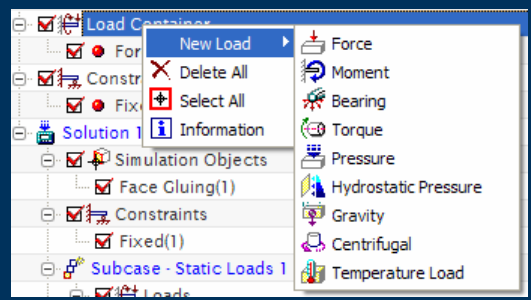



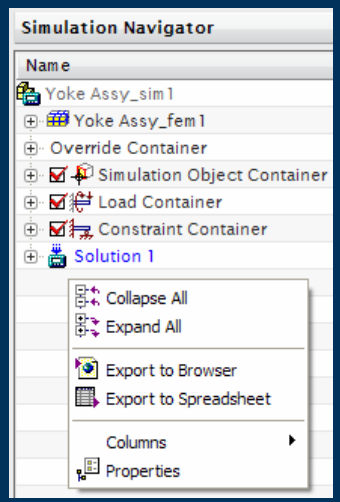



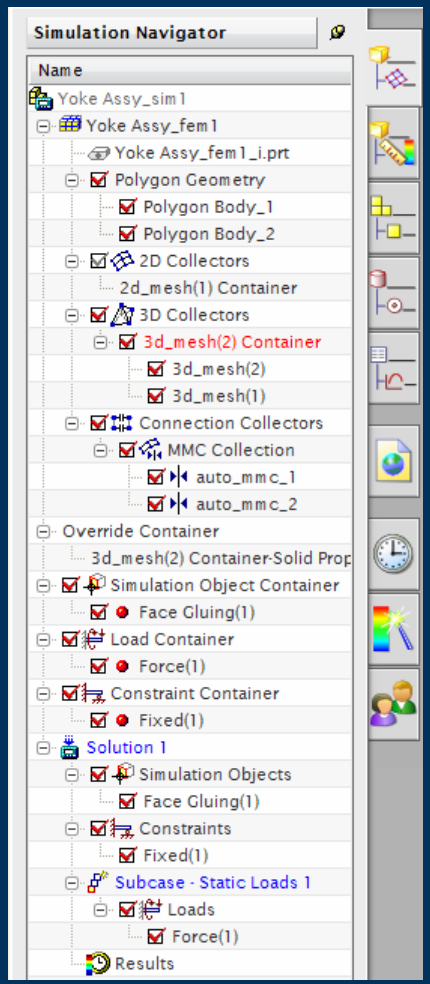



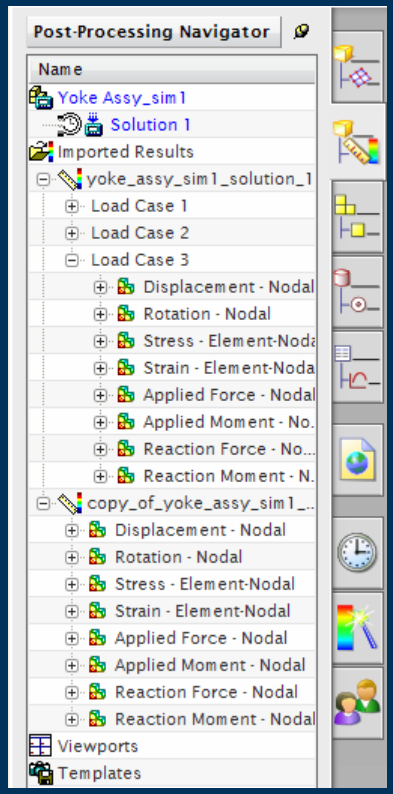



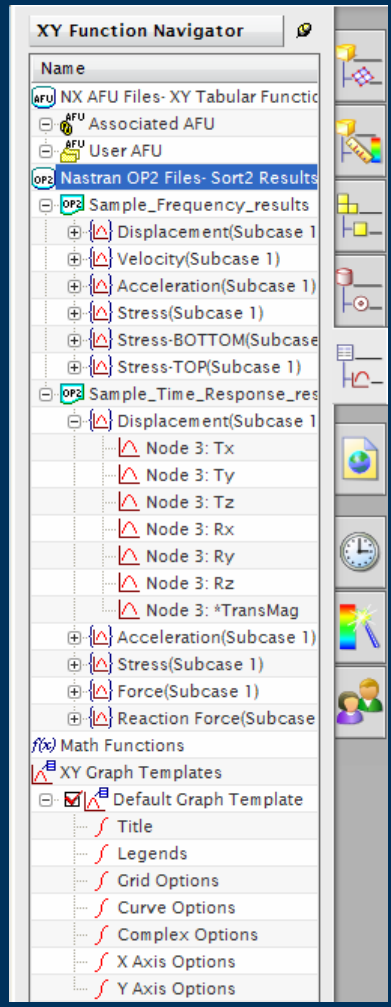



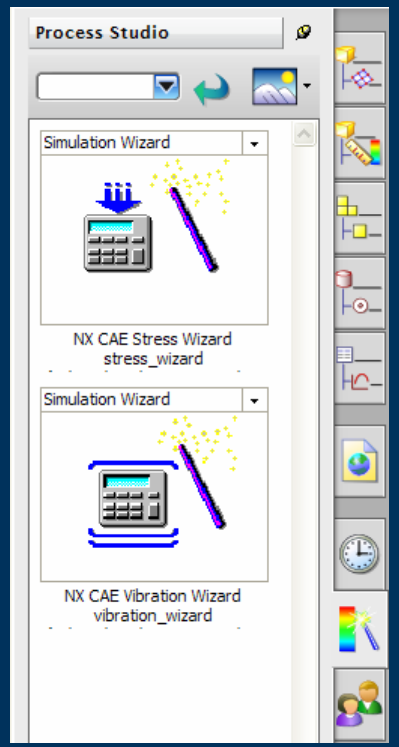
# Simulation Navigator – Resource Bars



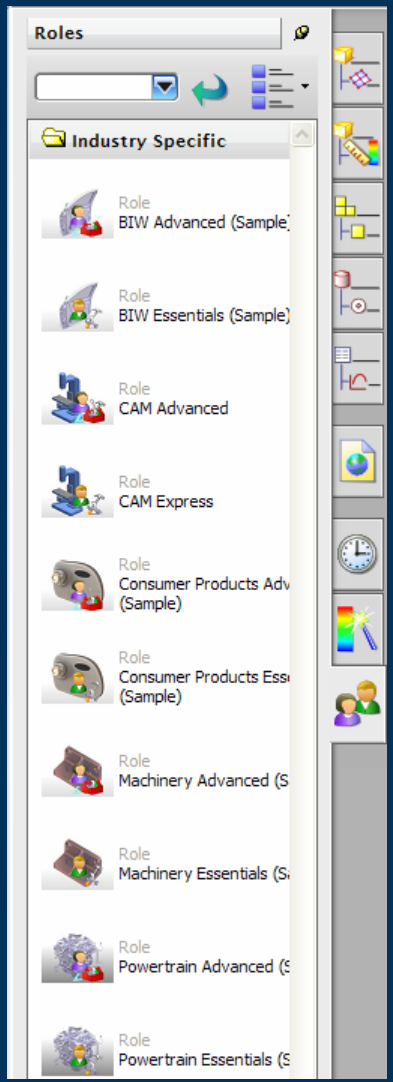



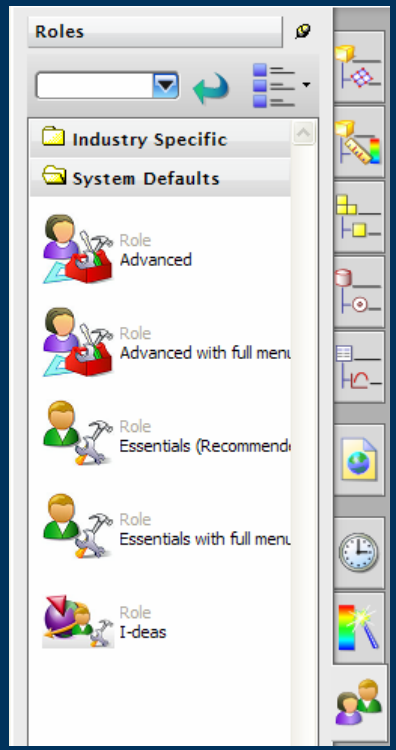




- Resource Bars for Simulation
  - Simulation
  - Post Processing
  - XY Functions



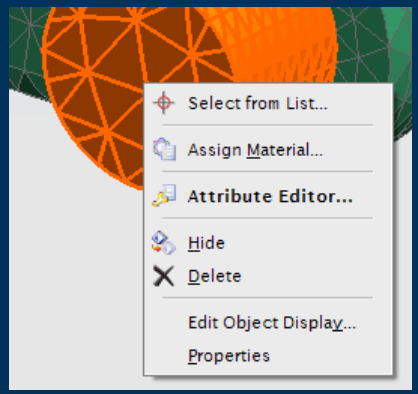
# Simulation Navigator – Resource Bars

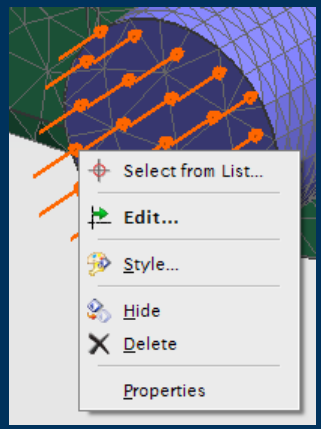


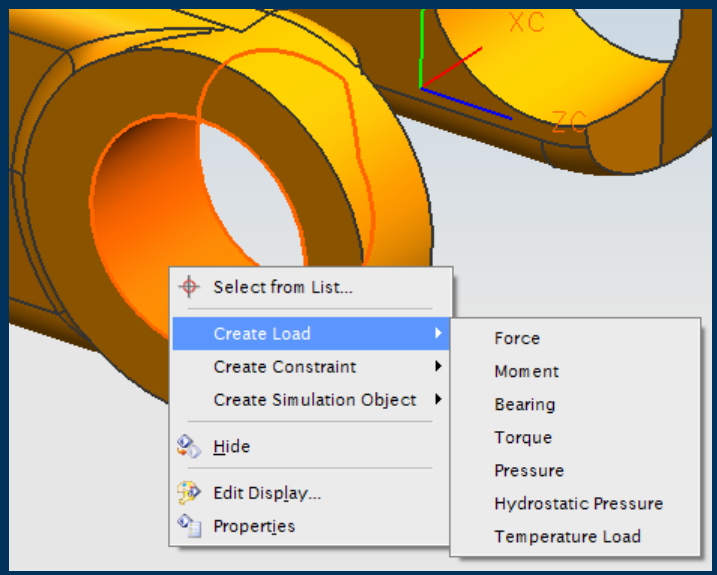

- Roles
  - Industry Specific
  - Advanced & Essentials
- General
  - Assembly Navigator
  - Part Navigator
  - History
  - Internet Explorer
  - On-Line Help

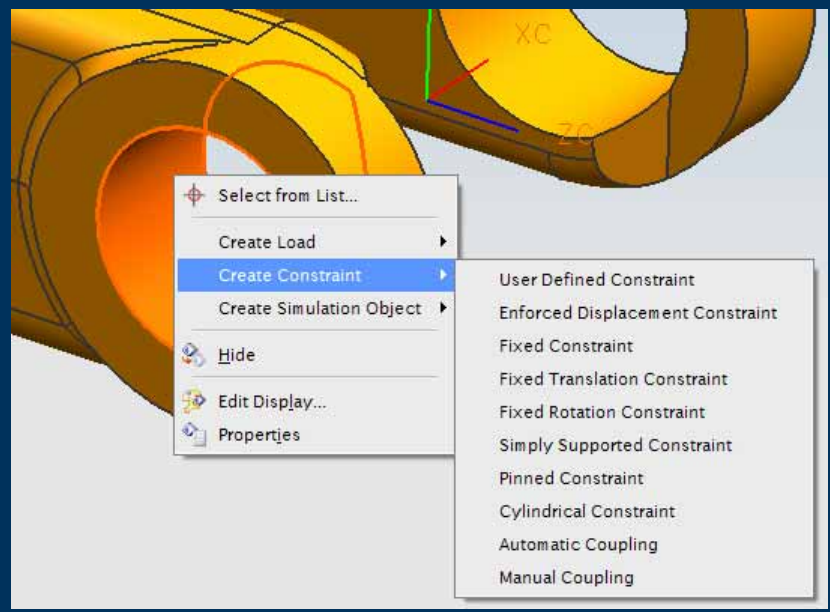
# Interaction – RMB Over Screen Model



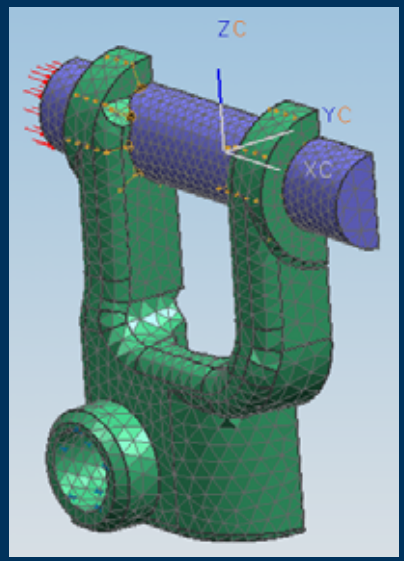

- RMB Over Screen Entities enables fast access to functionality applicable to the highlighted item

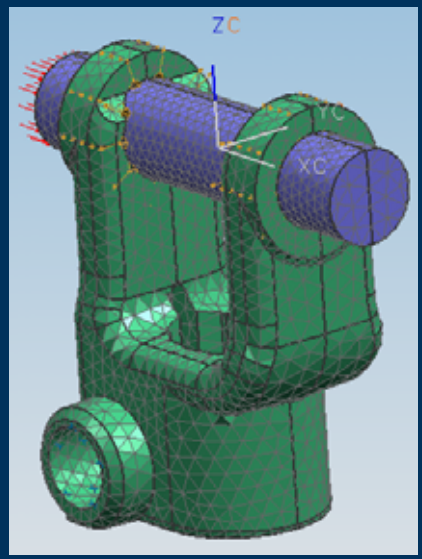


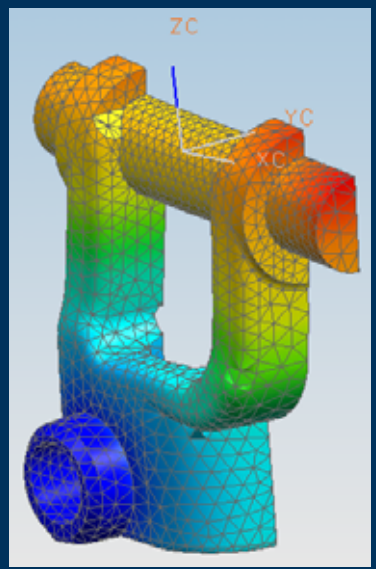
SIEMENS

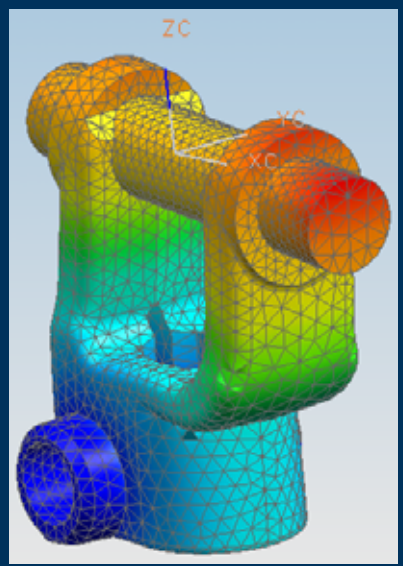
# Mirror Display

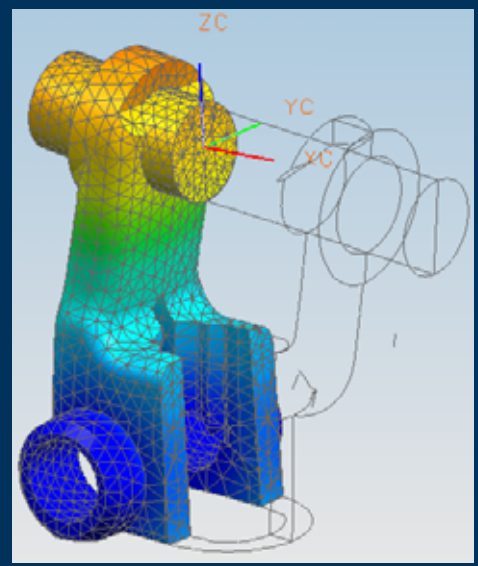



- Mirror Display is powerful for visualising Symmetric models
- Mirror Plane can be set anywhere
- Post View Settings work with the Mirror Display

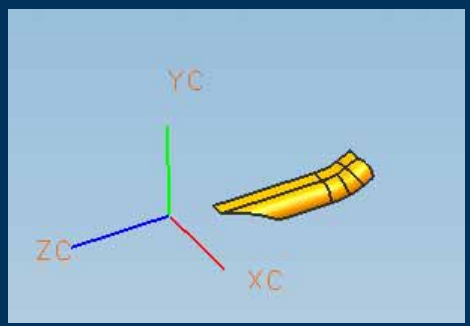


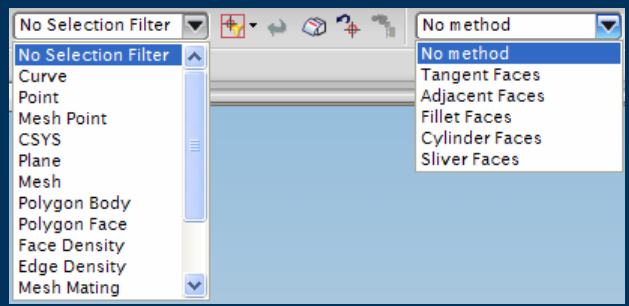



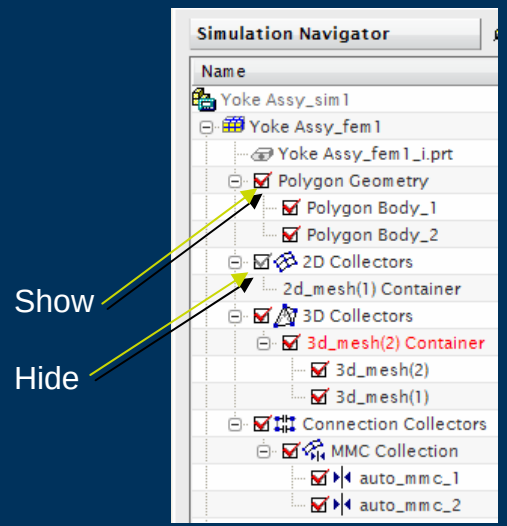



# Model Interaction – Show Only


- Reduce the complexity of geometry on the screen
- Focus on a sub-set of the model
- Selection Methods
  - Tangent Faces
  - Adjacent Faces
  - Fillet Faces
  - Cylindrical Faces
  - Sliver Faces
- Hide/Show from the Simulation Navigation
  - Used during other commands to simply the screen

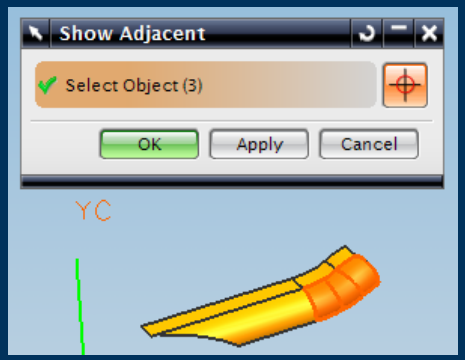


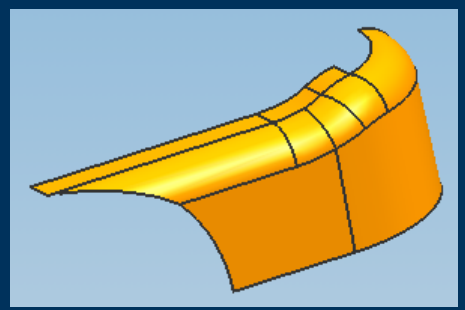



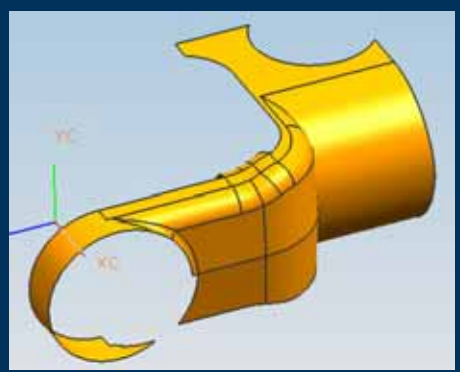
SIEMENS

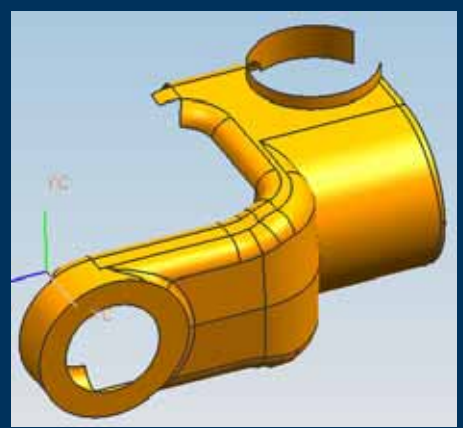
# Model Interaction – Show Adjacent

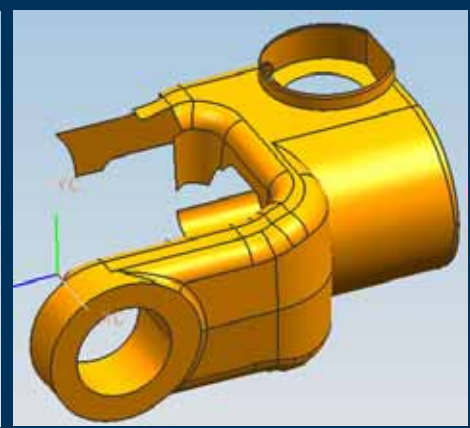



- Show Adjacent to “grow” visible related geometry
- Selection Methods
  - Tangent Faces
  - Adjacent Faces
  - Fillet Faces
  - Cylindrical Faces
  - Sliver Faces



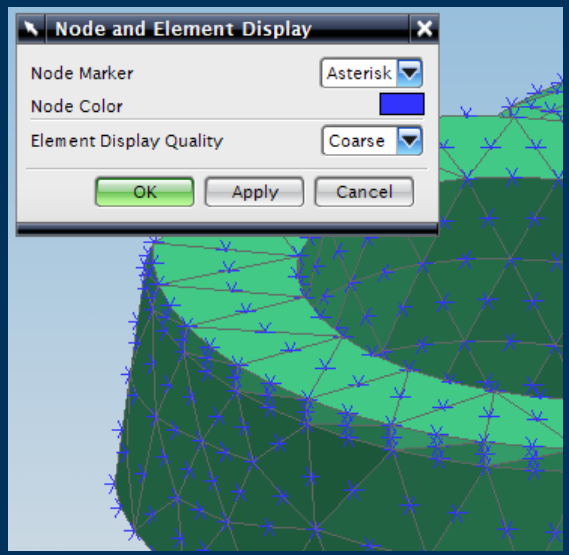






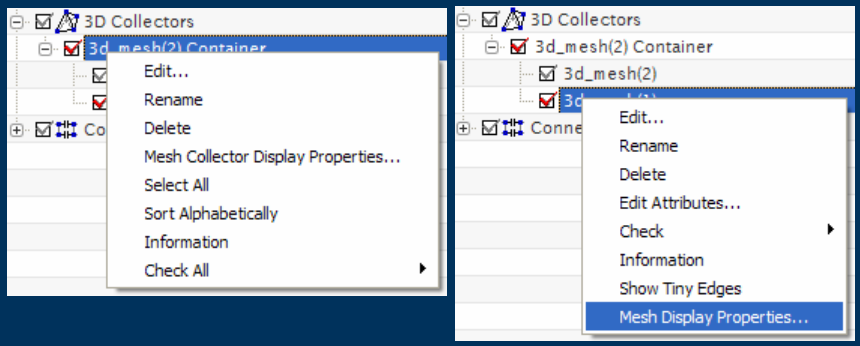
SIEMENS

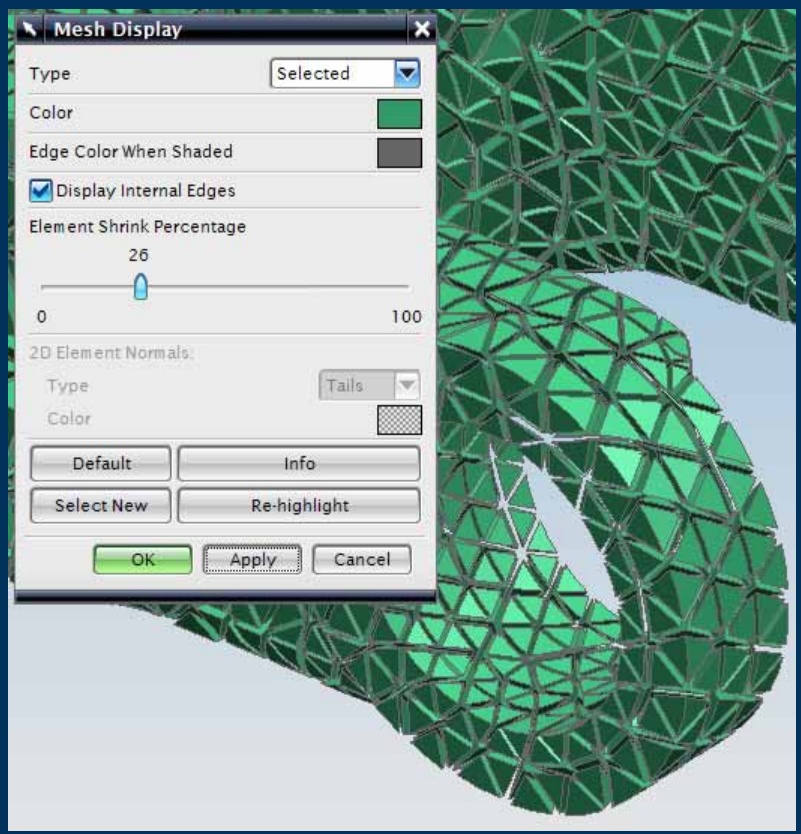
# Model Interaction – Node Display



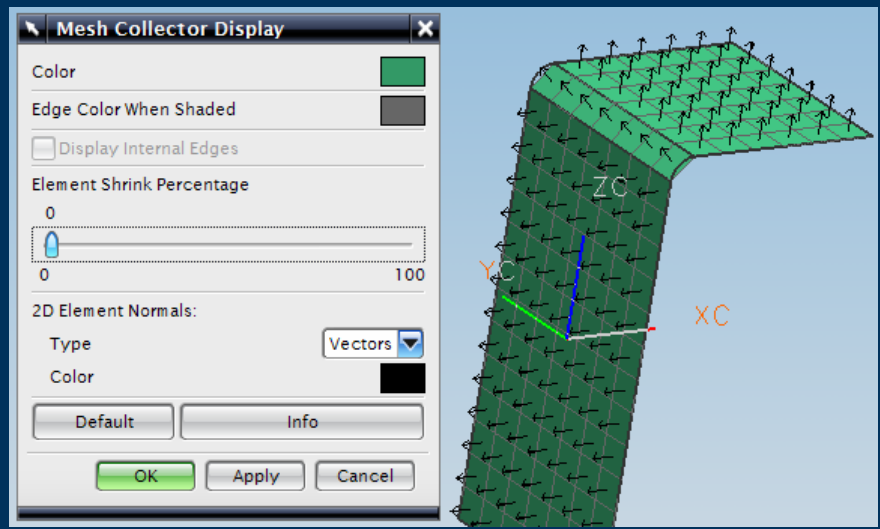

- Node Display options
  - None (default)
  - Dot
  - Asterisk
  - Color
- Element Display options
  - Coarse (default)
  - Medium
  - Fine

# Model Interaction – Mesh Display

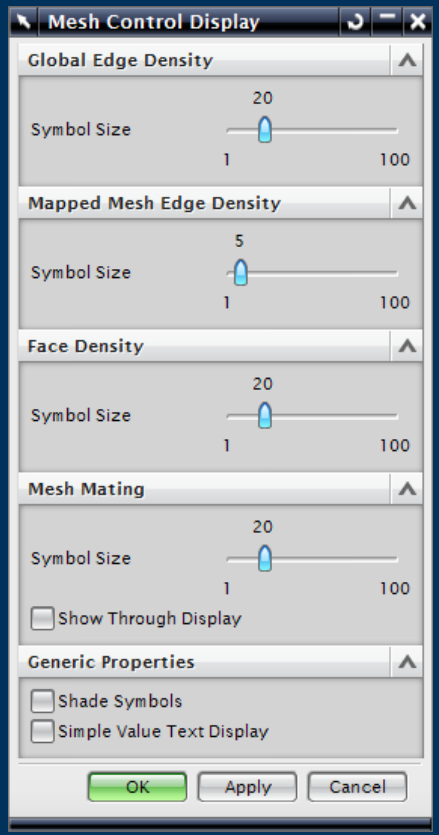


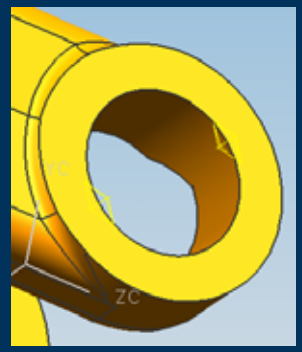



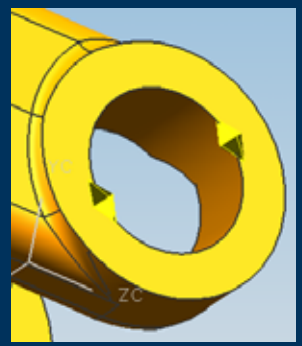

- Mesh Display
  - By Collector
  - By Mesh
- Mesh Display options
  - Colour
  - Edge Colour
  - Shrink Percentage
  - 2d Element Normals

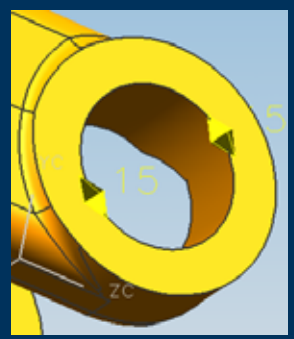


# Model Interaction – Mesh Control Display

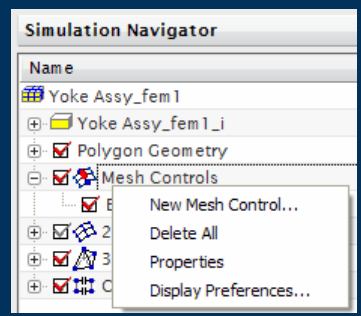






- Mesh Control Symbol display
  - Size
  - Shaded
  - With Text Value



SIEMENS

# Solver Language Environment



UGS PLM Software

SIEMENS

# Solver Language Environment

- PLM XML definitions enable rapid change & addition of solver languages
- All Loads, Boundary Conditions, Element Types, and Solver Inputs reflect selected solution environment

| Solver | Analysis Type | Solution Type |
|---|---|---|
| **NX Nastran** | Structural | SESTATIC 101 - Single Constraint |
| | | SESTATIC 101 - Multiple Constraint |
| | | SEMODES 103 |
| | | SEMODES 103 Response - Simulation |
| | | SEBUCKL 105 |
| | | NLSTATIC 106 |
| | | SEDFREQ 108 |
| | | SEDTRAN 109 |
| | | SEMFREQ 111 |
| | | SEMTRAN 112 |
| | | ADVNL 601, 106 |
| | | ADVNL 601, 129 |
| | Thermal | NLSCSH 153 |
| | Axisymmetric Structural | SESTATIC 101 - Single Constraint |
| | | SESTATIC 101 - Multiple Constraint |
| | | NLSTATIC 106 |
| | Axisymmetric Thermal | NLSCSH 153 |

| Solver | Analysis Type | Solution Type |
|---|---|---|
| **NX Nastran Design** | Structural | Linear Statics - Single Constraint |
| | | Therma; |
| | | Linear Buckling |
| | Thermal | Thermal |

| Solver | Analysis Type | Solution Type |
|---|---|---|
| **NX Thermal-Flow** | Thermal | Thermal |
| | | Advanced Thermal |
| | Flow | Flow |
| | | Advanced Flow |
| | Coupled Thermal-Flow | Thermal-Flow |
| | | Advanced Thermal-Flow |
| | | Complete |
| | Axisymmetric Thermal | Axisymmetric Thermal |
| | | Advanced Axisymmetric Thermal |
| | Mapping | Thermal-Flow |
| | Axisymmetric Mapping | Thermal |

| Solver | Analysis Type | Solution Type |
|---|---|---|
| **NX Space Systems Thermal** | Thermal | Space Systems Thermal |
| | Mapping | Thermal |

| Solver | Analysis Type | Solution Type |
|---|---|---|
| **NX Electronic Systems Cooling** | Coupled Thermal-Flow | Electronic Systems Cooling |
| | | Advanced Electronic Systems Cooling |
| | Mapping | Thermal-Flow |

SIEMENS

# Solver Language Environment (cont)

- Non-UGS Solver support

| Solver | Analysis Type | Solution Type |
|---|---|---|
| MSC Nastran | Structural | SESTATIC 101 - Single Constraint |
| | | SESTATIC 101 - Multiple Constraint |
| | | SEMODES 103 |
| | | SEBUCKL 105 |
| | | NLSTATIC 106 |
| | | SEDFREQ 108 |
| | | SEDTRAN 109 |
| | | SEMFREQ 111 |
| | | SEMTRAN 112 |
| | Thermal | NLSCSH 153 |
| | Axisymmetric Structural | SESTATIC 101 - Single Constraint |
| | | SESTATIC 101 - Multiple Constraint |
| | | NLSTATIC 106 |
| | Axisymmetric Thermal | NLSCSH 153 |

| Solver | Analysis Type | Solution Type |
|---|---|---|
| ABAQUS | Structural | General Analysis |
| | Thermal | Heat Transfer |
| | Axisymmetric Structural | General Analysis |
| | Axisymmetric Thermal | Heat Transfer |

| Solver | Analysis Type | Solution Type |
|---|---|---|
| ANSYS | Structural | Linear Statics |
| | | Modal |
| | | Buckling |
| | | Nonlinear Statics |
| | Thermal | Thermal |
| | Axisymmetric Structural | Linear Statics |
| | | Nonlinear Statics |
| | Axisymmetric Thermal | Thermal |

- Import of I-deas Universal file with CAE data only

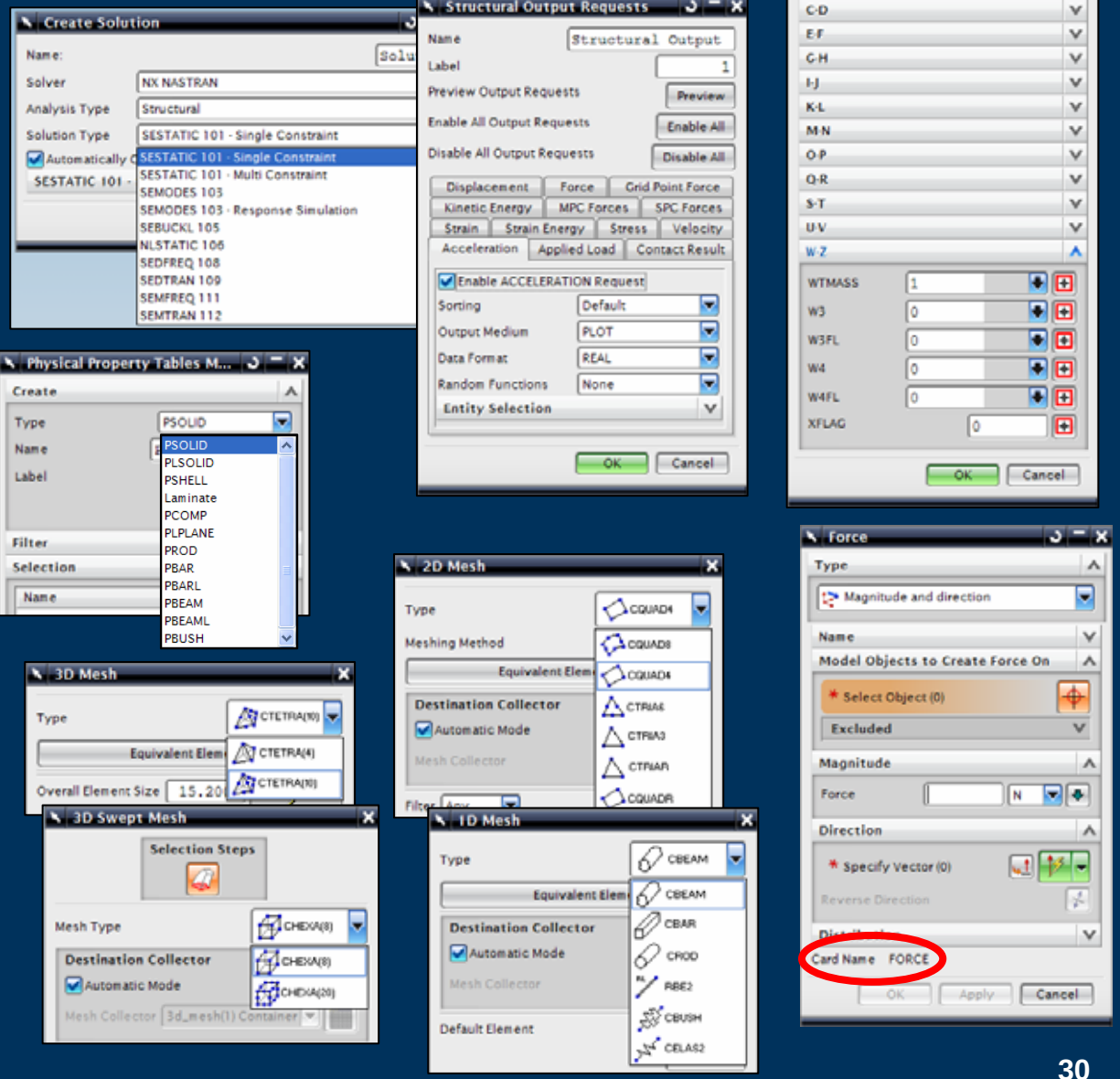
| Solver | Analysis Type | Solution Type |
|---|---|---|
| I-DEAS UNV | Universal | I-DEAS UNV |

# “NX Nastran Environment” – UI Based on Solver/Solution

SIEMENS

- Selected at FEM part file creation
  - Mesh creation
- Selected at Solution creation in SIM part
  - Solution creation and editing
  - Defines Sub-Case options

Benefits
- User interface words are familiar
- Elements, Loads, Boundary Conditions etc are all in the words of the selected Solver

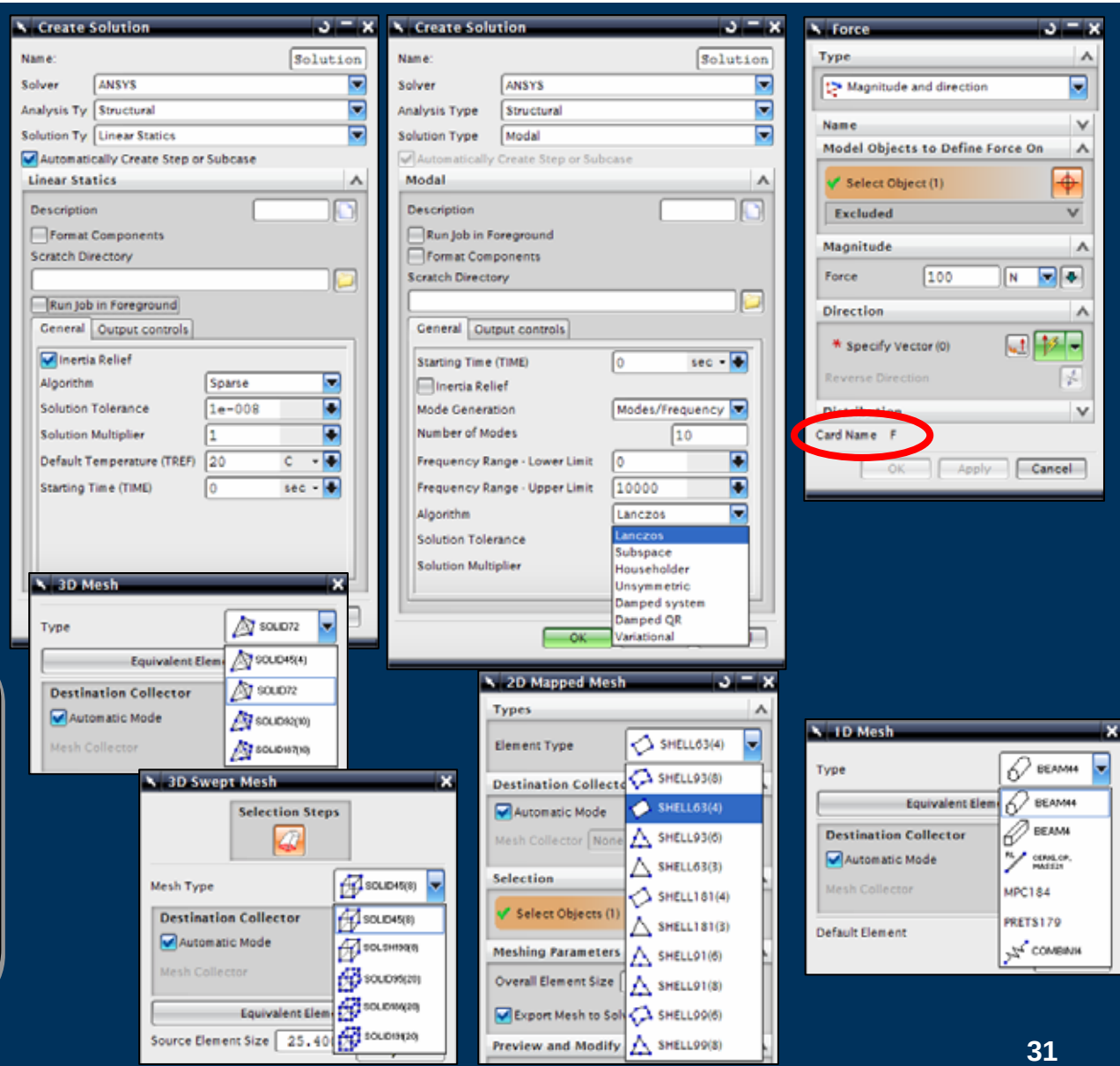


# “ANSYS Environment” – UI Based on Solver/Solution

SIEMENS

- Selected at FEM part file creation
  - Mesh creation
- Selected at Solution creation in SIM part
  - Solution creation and editing
  - Defines Sub-Case options

Benefits
- User interface words are familiar
- Elements, Loads, Boundary Conditions etc are all in the words of the selected Solver

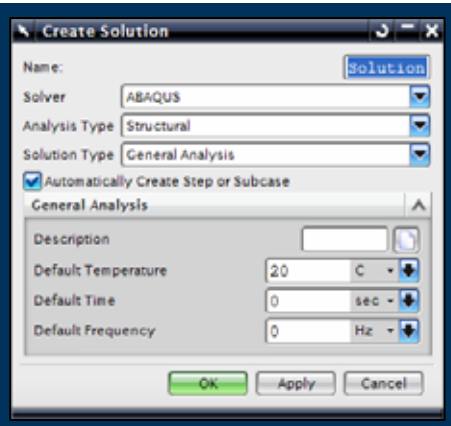


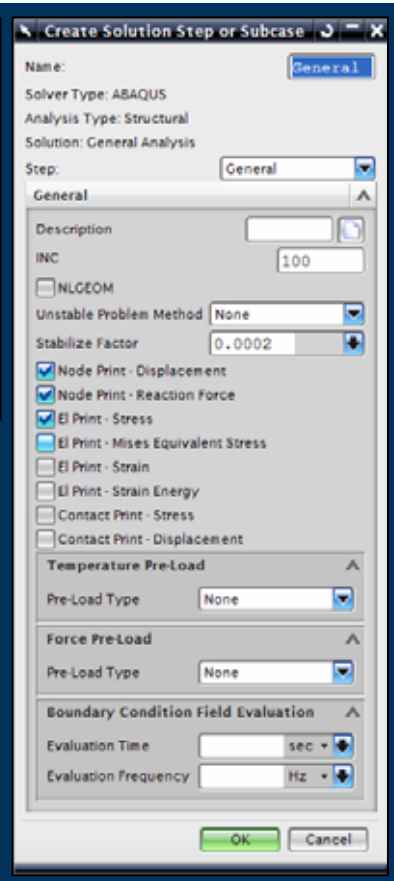
# “ABAQUS Environment” – UI Based on Solver/Solution

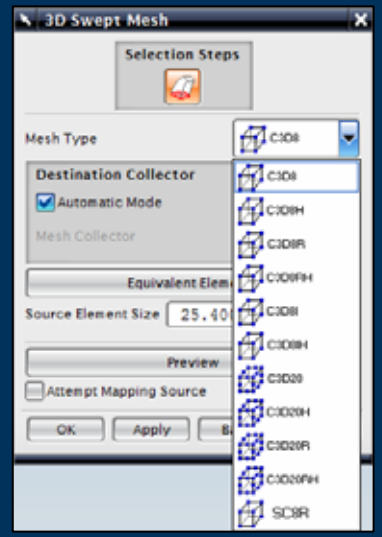
SIEMENS

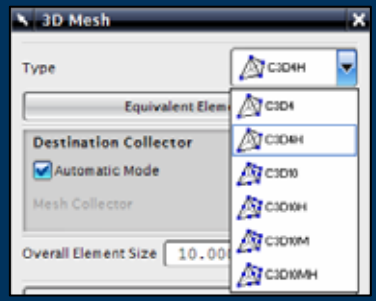
- Selected at FEM part file creation
  - Mesh creation
- Selected at Solution creation in SIM part
  - Solution creation and editing
  - Defines Sub-Case options

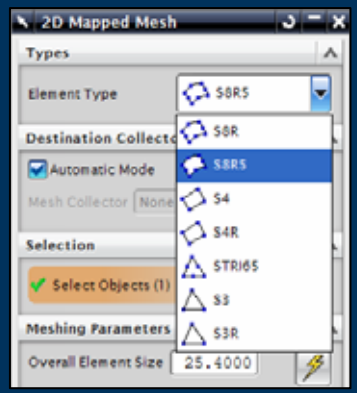
Benefits
- User interface words are familiar
- Elements, Loads, Boundary Conditions etc are all in the words of the selected Solver

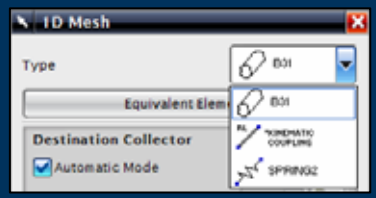


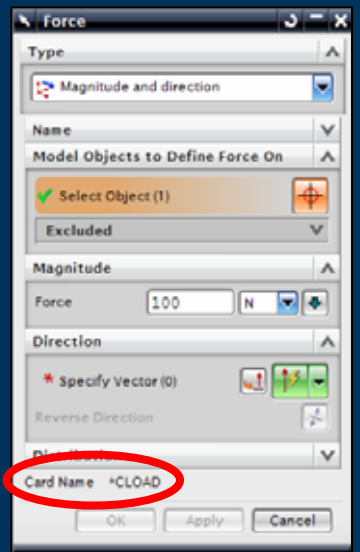



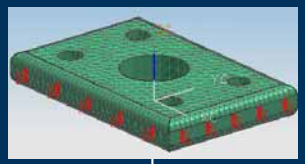



SIEMENS

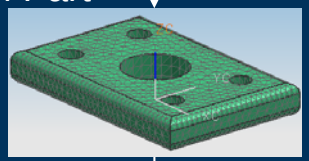
# Master Part

# Master Part

Simulation Part

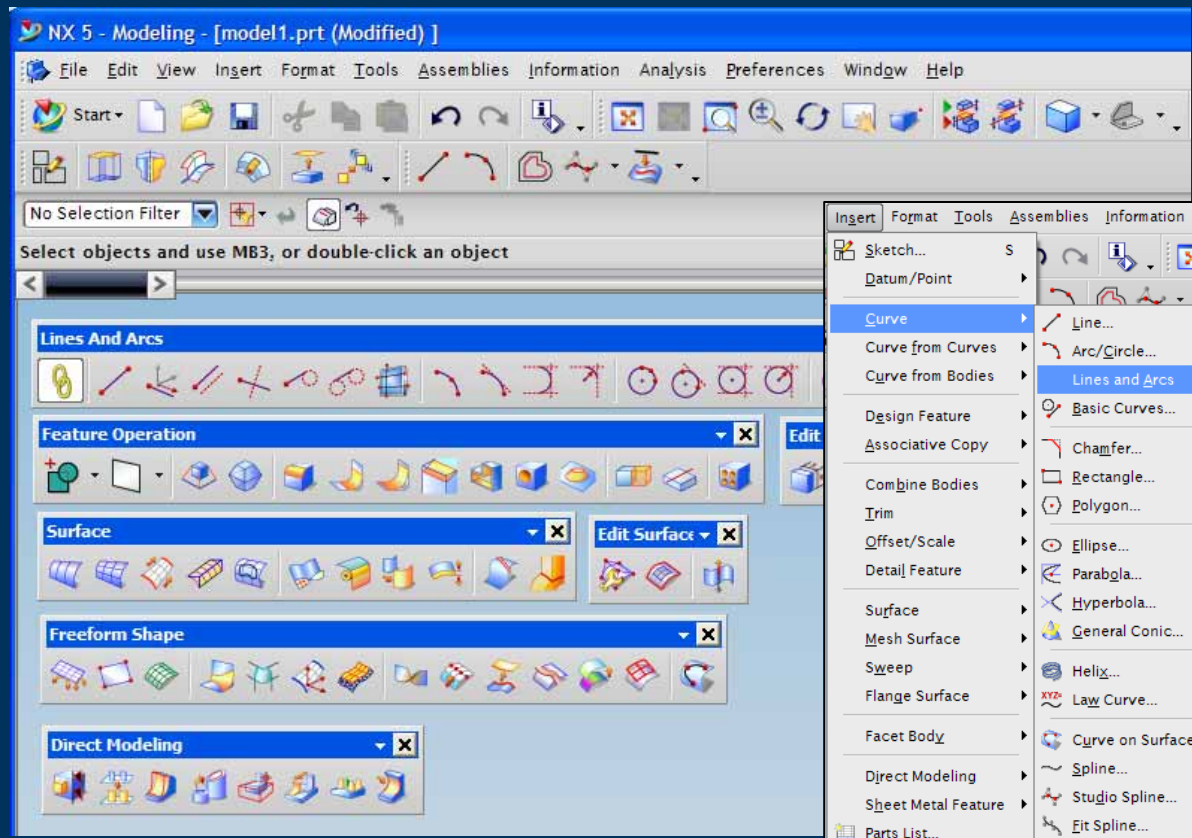


FEM Part



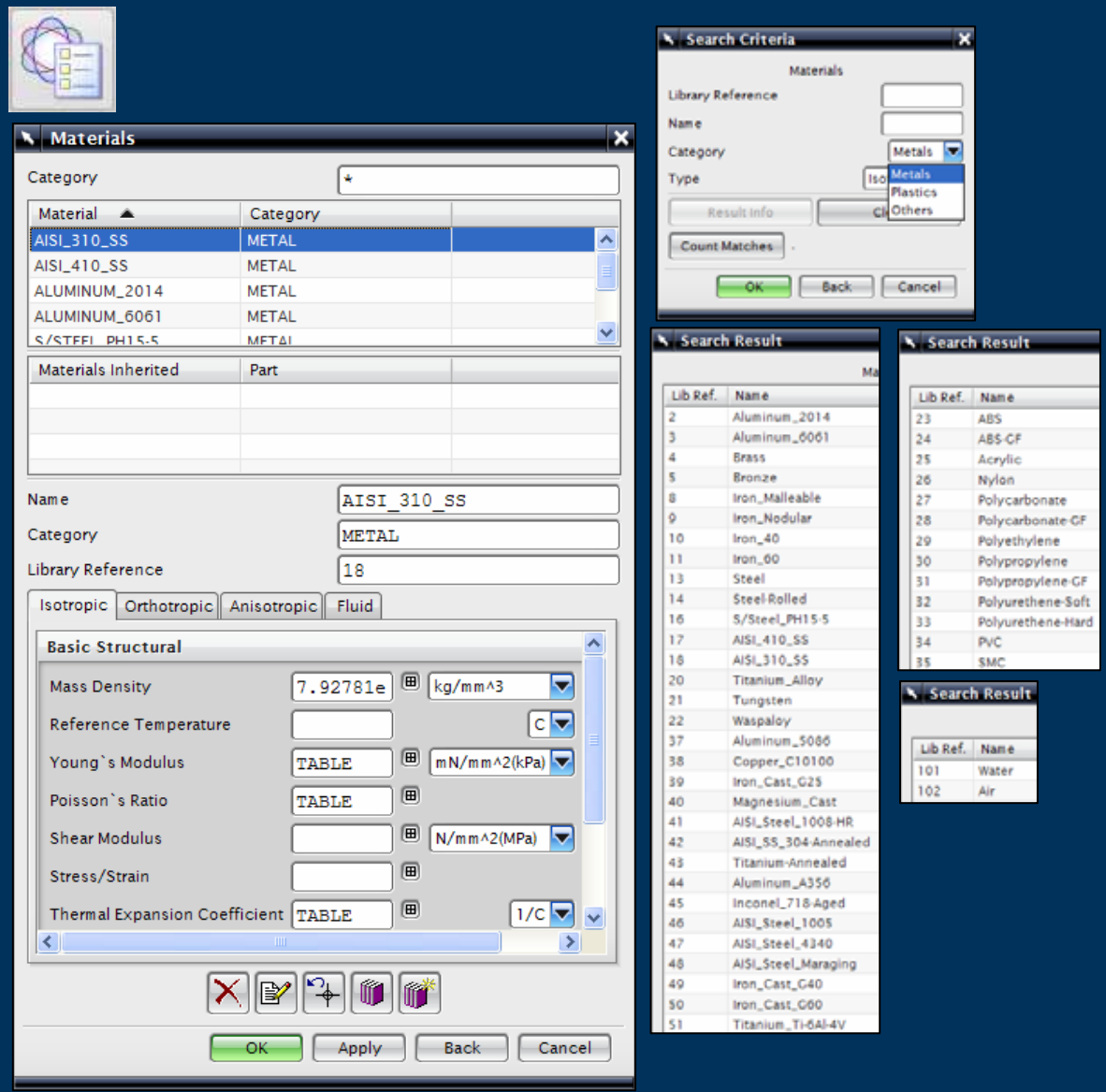
Master Part

- Idealized part is NOT a requirement
- Full NX CAD modelling functionality is available for building models for CAE purposes



SIEMENS

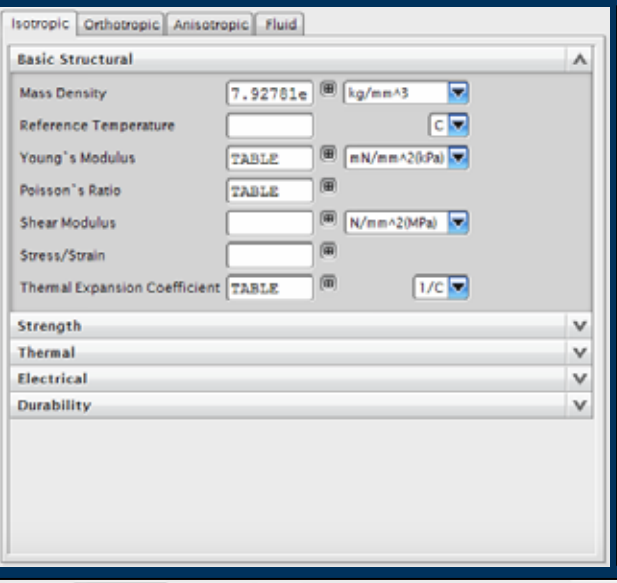
# Material Property – Library

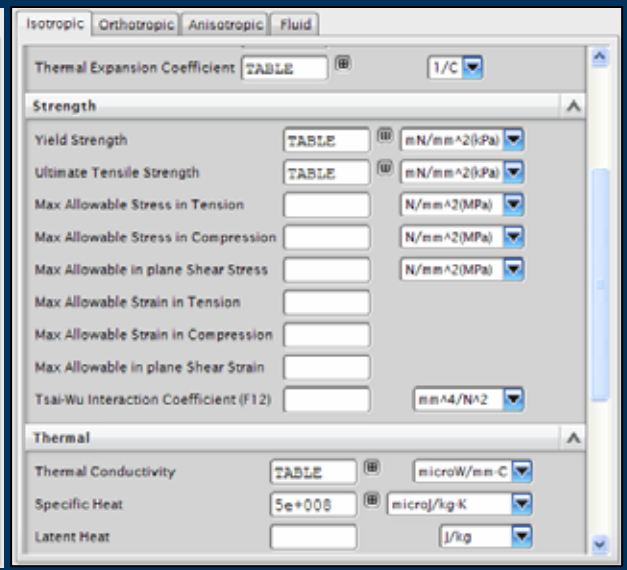



- Import from Material Property Library
  - Metals
  - Plastics
  - Isotropic
  - Orthotropic
  - Anisotropic
  - Fluids
- Assign to Parts
- Create new based on existing materials

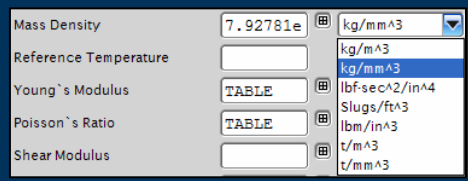
SIEMENS

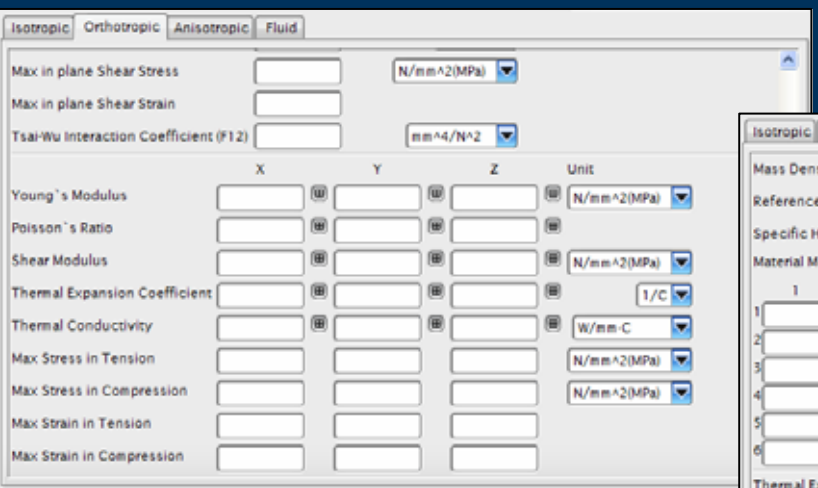
# Material Properties

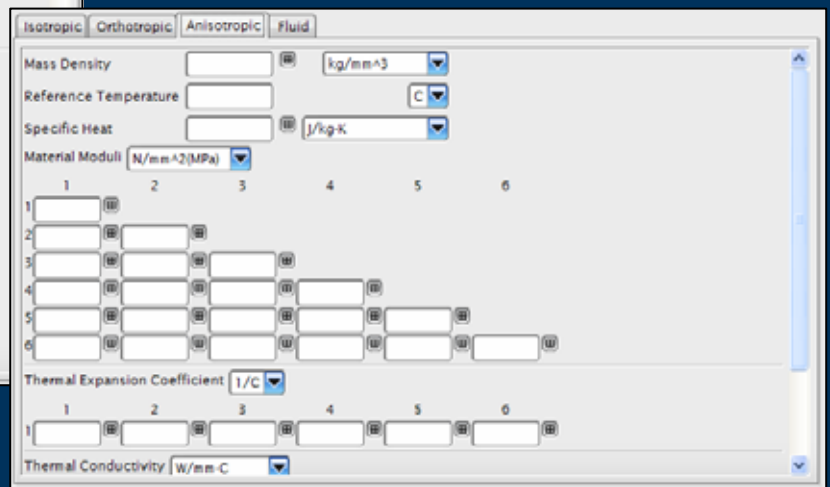


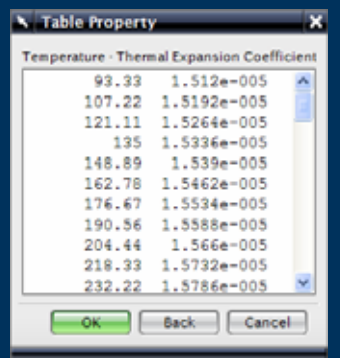




- Constant Values
- Variable Values defined by a Table
- Units selection
- Adding new materials to the Library is documented in the on-line help



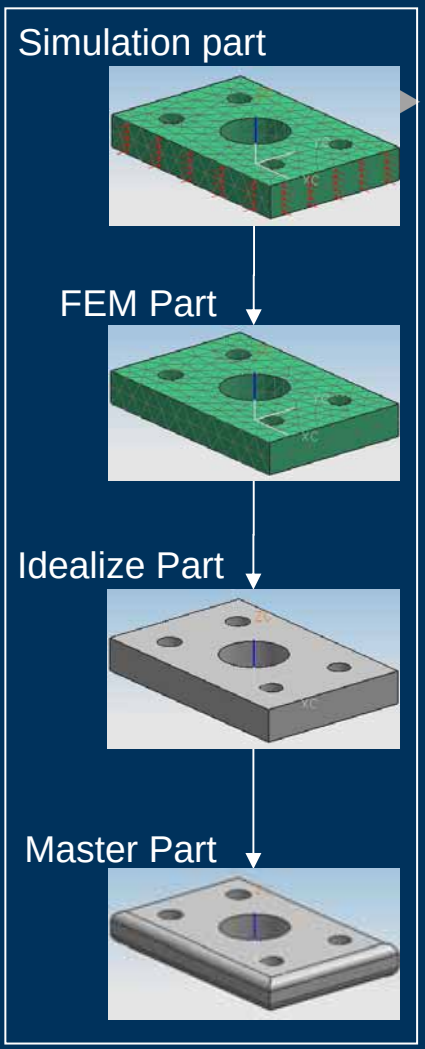



SIEMENS

# Idealize Part

# Idealize Part




## Uses of the Idealize part

- Read only Master Part
  - Therefore can not change the Master Part geometry
  - Vital in a Managed Environment
- “ What If ” exploration or studies based on the same Master Part
- Geometry Reduction or Abstraction
- Additional Geometry or Datums

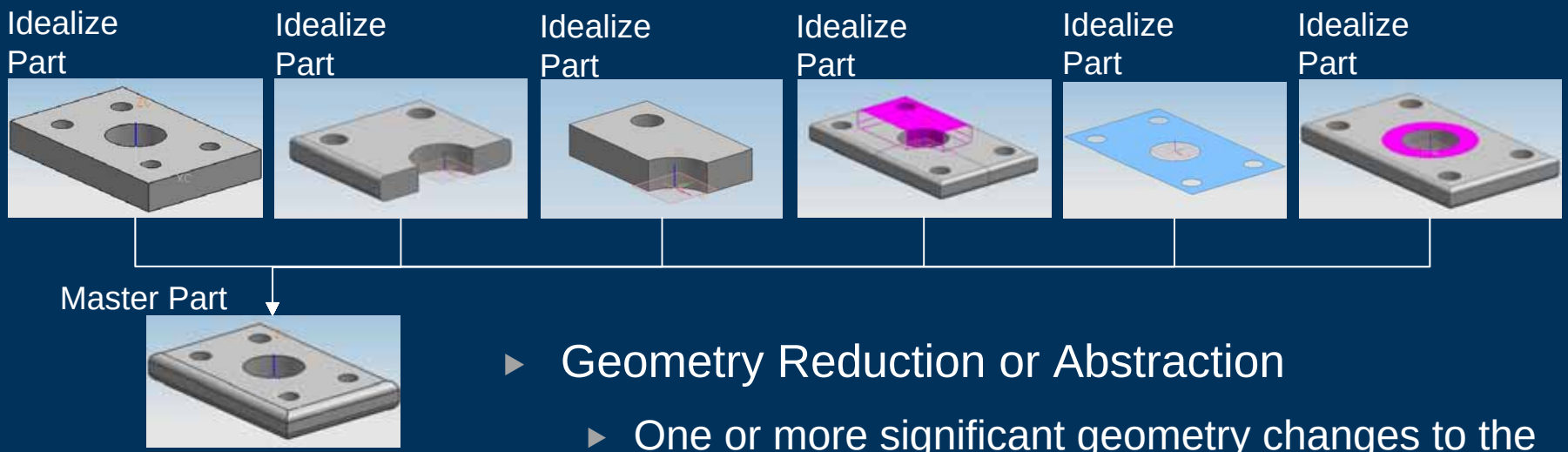
Benefits
- Support Concurrent Engineering
- Associativity to Master Model

# Uses of the Idealize part


- " What If " exploration or studies based on the same Master Part
  - Removing geometry
    - By type and size – Holes and Blends
    - By selection – Auto saved methods for updates
  - Adding Additional Modelling features, holes, blends, chamfers, ribs, bosses etc
  - Different materials from the Master Part

SIEMENS

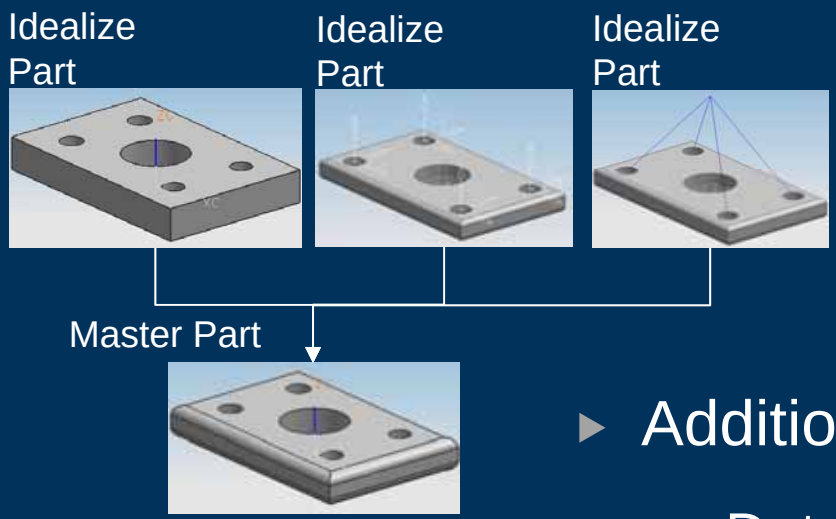
# Uses of the Idealize part




- Geometry Reduction or Abstraction
  - One or more significant geometry changes to the Master Part
  - Symmetric, Asymmetric or Axisymmetric models
  - Mid-Surface
  - Partition or Surface Splitting
    - Load/Restraint Application
    - Local mesh control
    - Mesh Mating condition – common mesh across boundary

SIEMENS

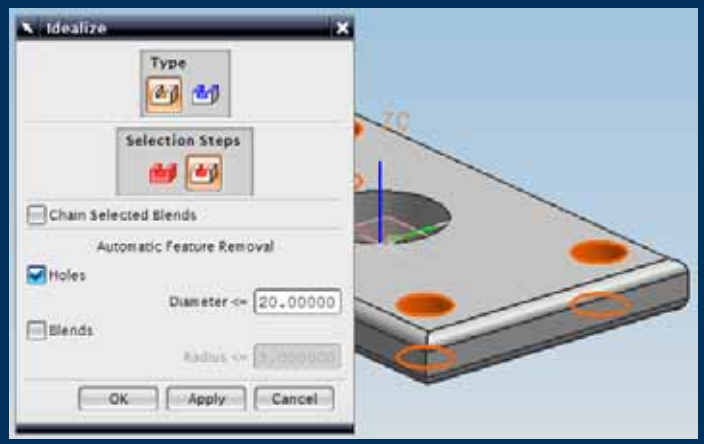
# Uses of the Idealize part



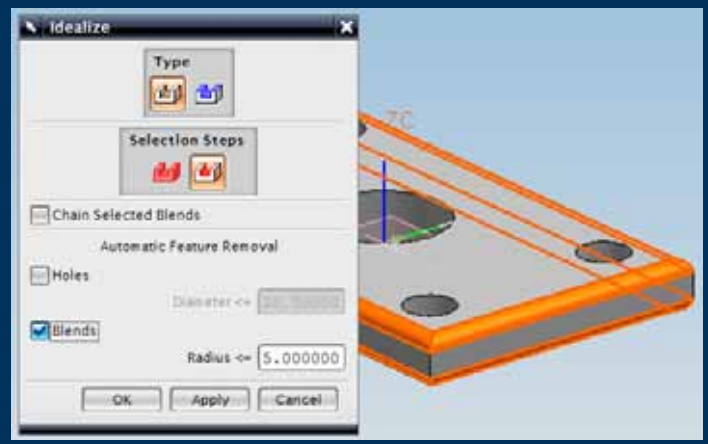

- Additional Geometry
  - Datums like Coordinate Systems
  - Curves and points to place FEM entities
    - Lumped Mass
    - Rigid Elements

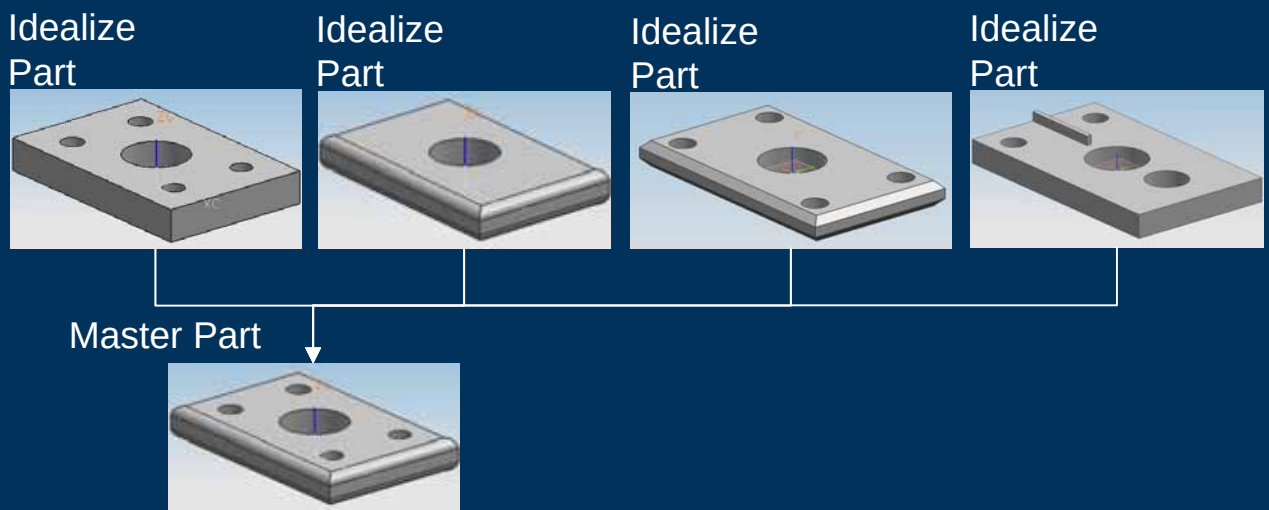
SIEMENS

# Idealize Part – Idealize

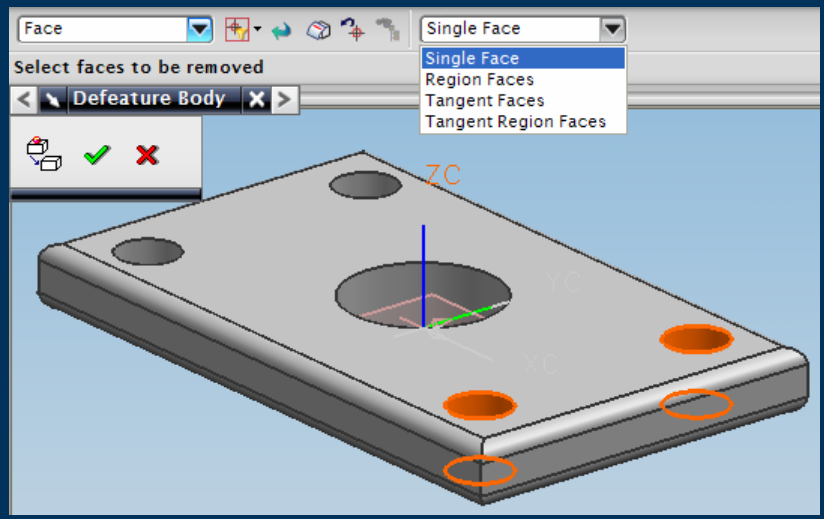



- Removing Holes and Blends
  - Based on their size
  - Manual selection



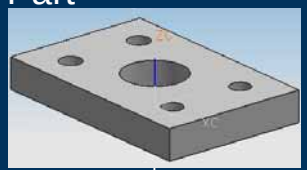



# Idealize Part – Defeature Geometry

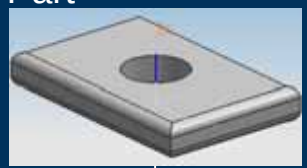



- Removing Geometry by Selection
- Method saved for Update replays

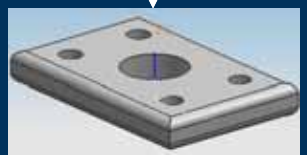
Idealize Part



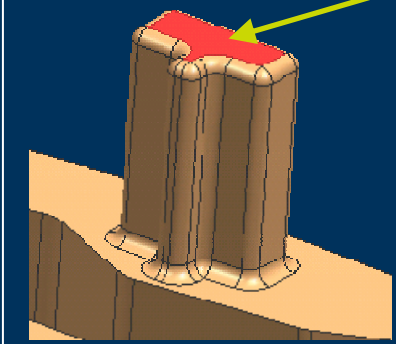
Idealize Part



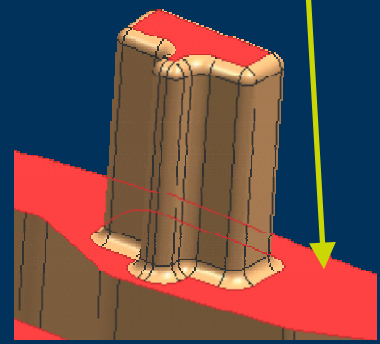
Master Part



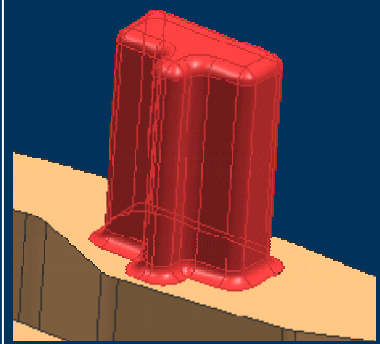
Pick outermost face as seed



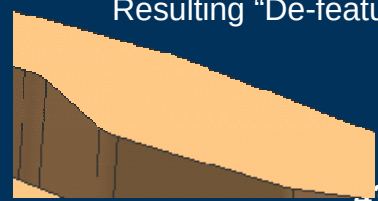
Pick Region boundary faces



Selected faces

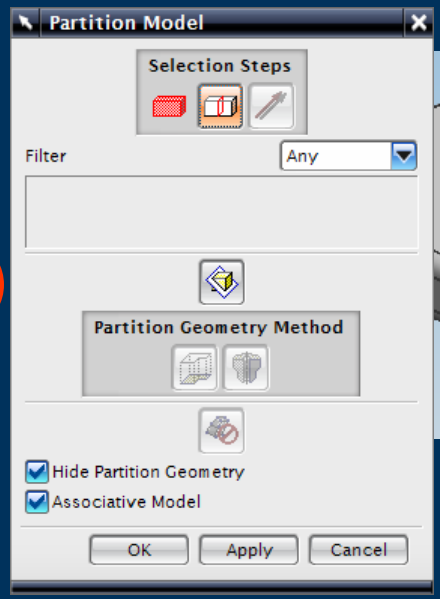


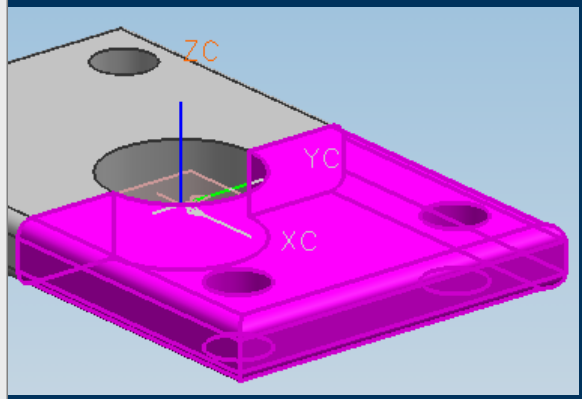
Resulting “De-feature”



SIEMENS

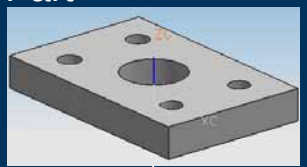
# Idealize Part – Partition



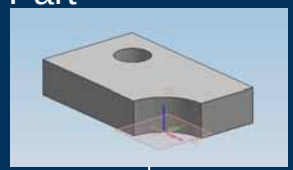

- Cutting part(s) into multiple volumes that share common faces
- Cutting part(s) into multiple surface patches that share common edges
- Load/Restraint Application
- Local mesh control
- Mesh Mating condition – common mesh across boundary
- Associative or Non-Associative to model

Idealize Part

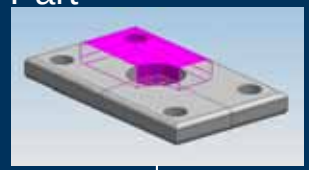


Idealize Part

Idealize Part

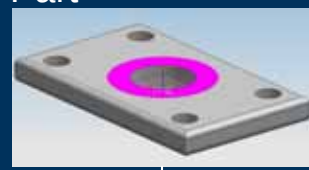


Idealize Part



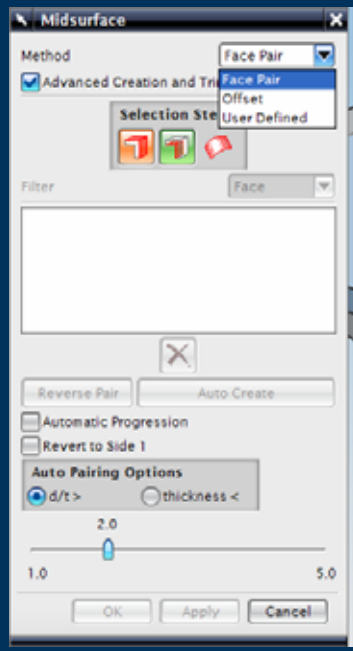
Idealize Part

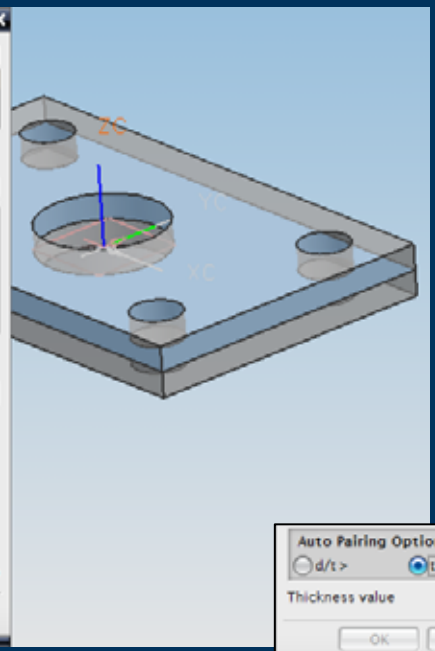
Idealize Part

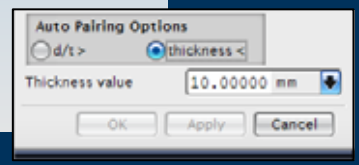


Master Part

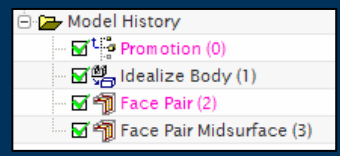
# Idealize Part – Midsurface



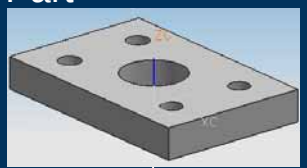



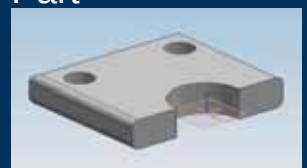

- Automatic Midsurface generation
  - Face Pair technique
  - Offset technique
  - Using another Sheet Body to define the Midsurface
  - Proportional Thickness or Specific value



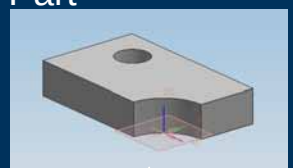

Idealize Part



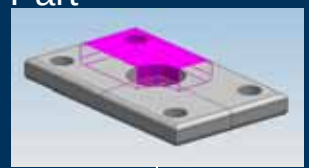
Idealize Part



Idealize Part

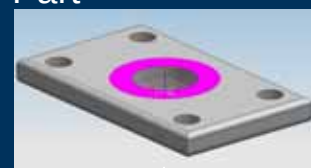


Idealize Part



Idealize Part

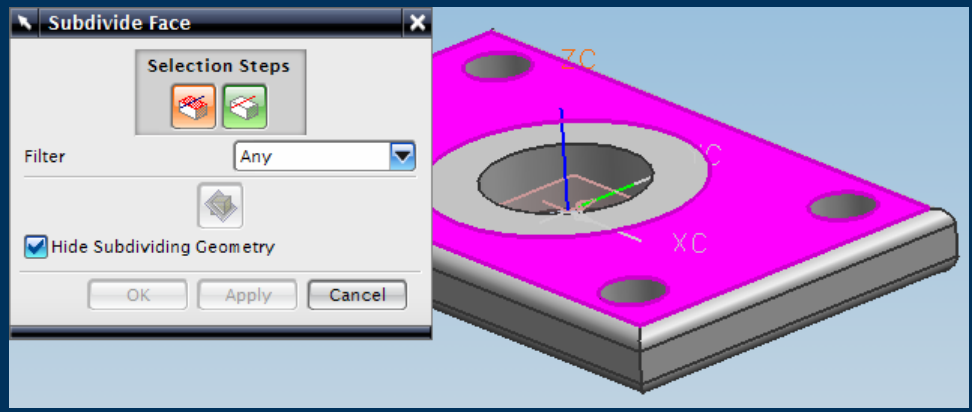
Idealize Part



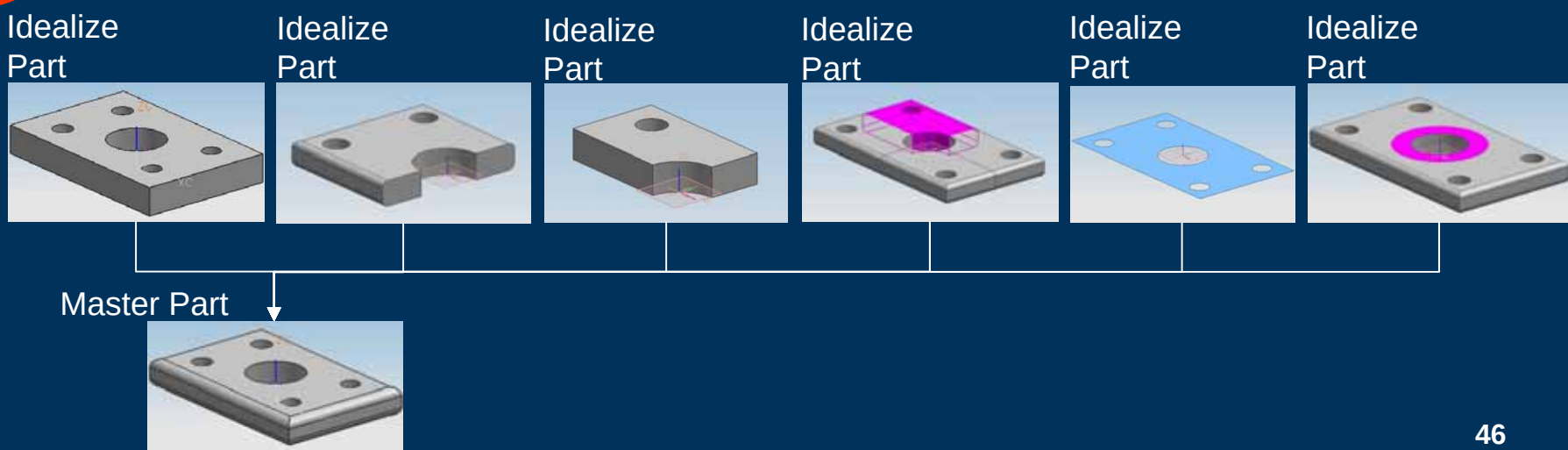
Master Part

SIEMENS

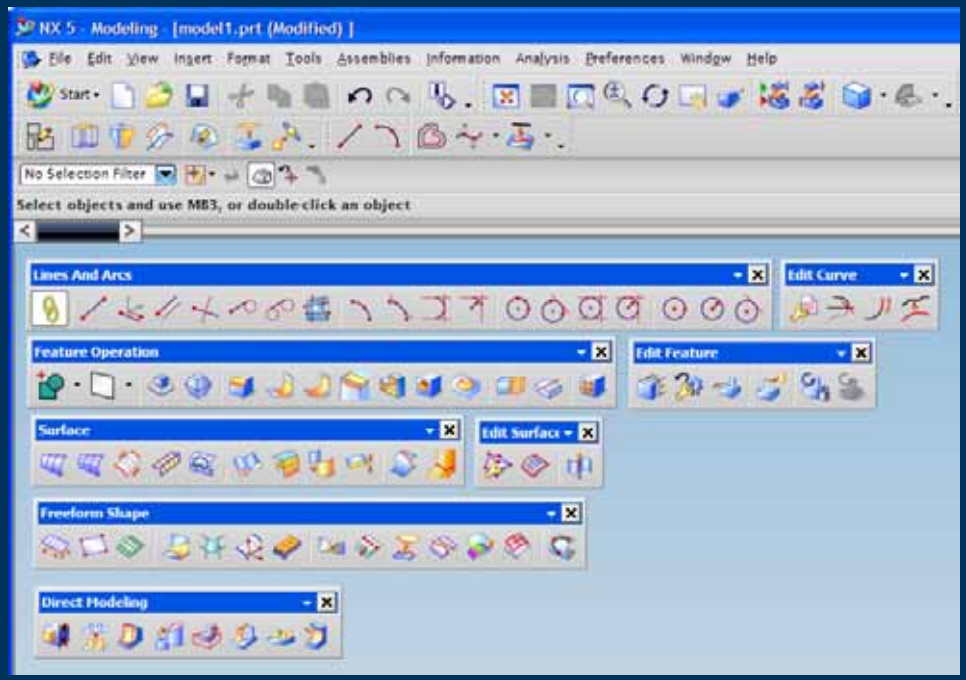
# Idealize Part – Subdivide Faces



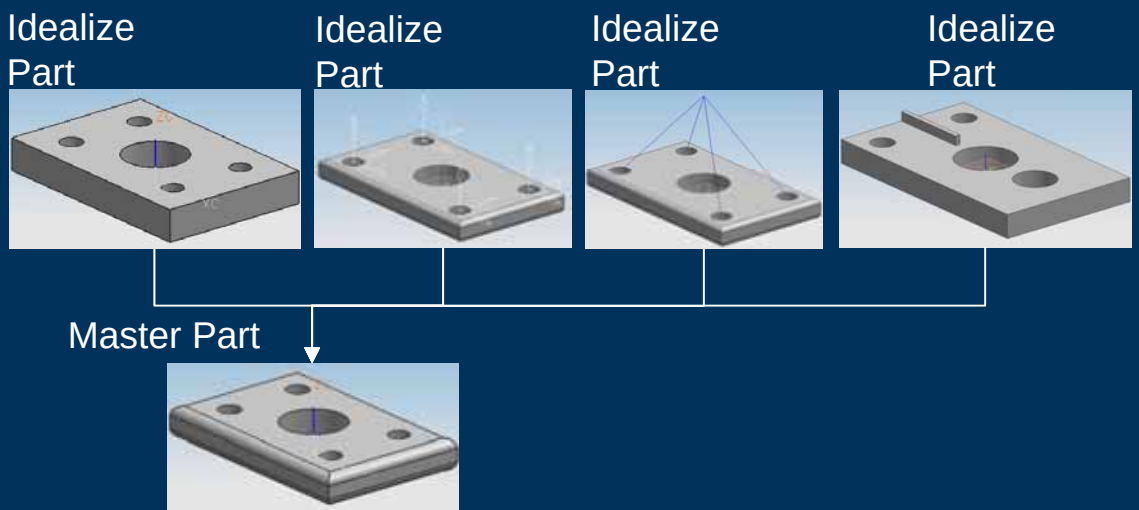

- Subdividing Faces
  - Intersection of Datum Planes
  - Intersection of Faces
  - Projected Curves and Edges
  - Projected Line between 2 points
- Load/Restraint Application
- Local mesh control

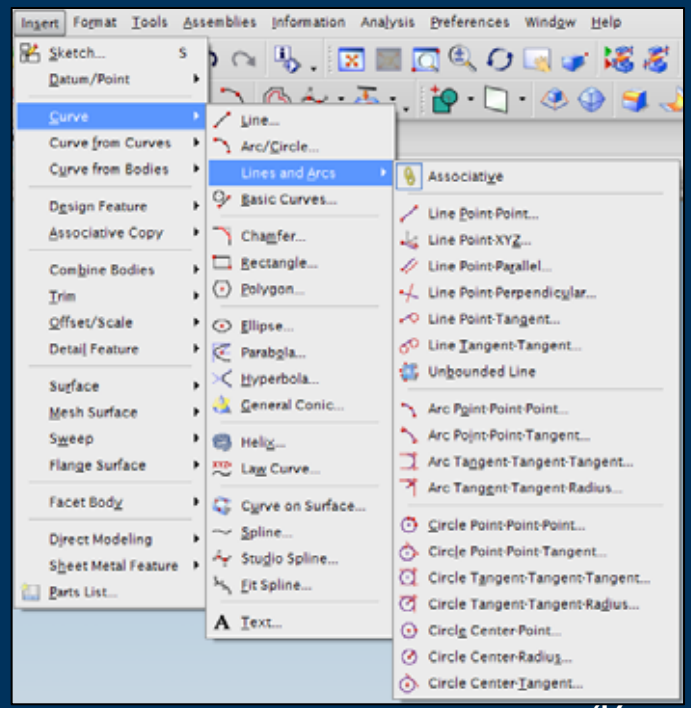


# Idealize Part – Additional Modelling



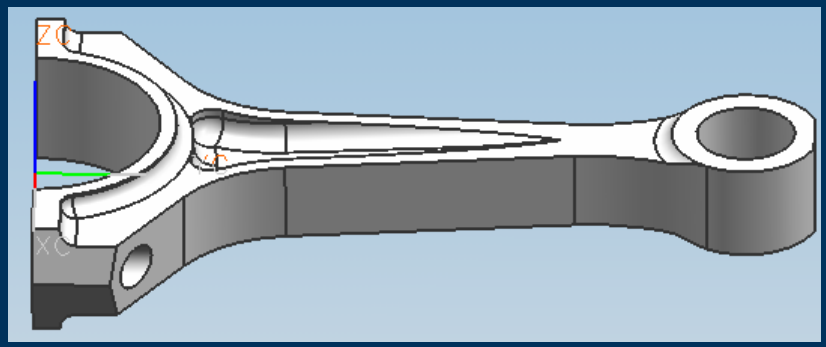

- All modelling functionality is available
- Datums, curves, holes, blends, chamfers, ribs, bosses, surfaces, solids etc etc

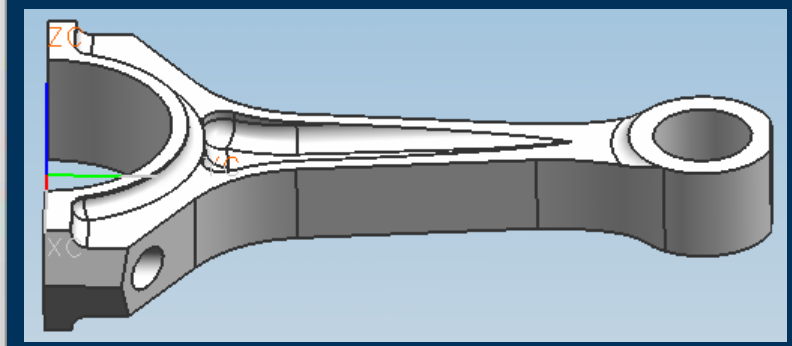


SIEMENS

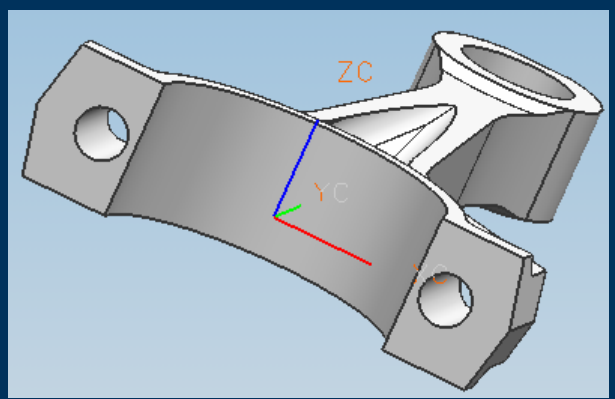
# Idealize Part – Direct Modelling

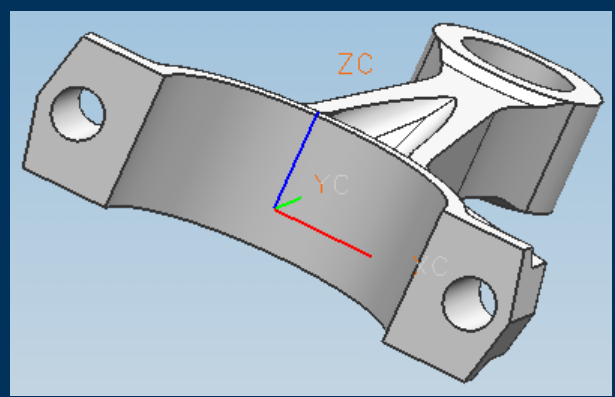


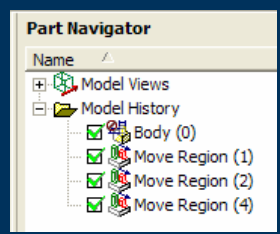




- Direct Modelling (DMX)
- Editing parts with no CAD features – Imported geometry
- Surrounding BREP updated – Tangency maintained
- Surfaces Resized – Fillets
- Surfaces Replaced

Little End Moved

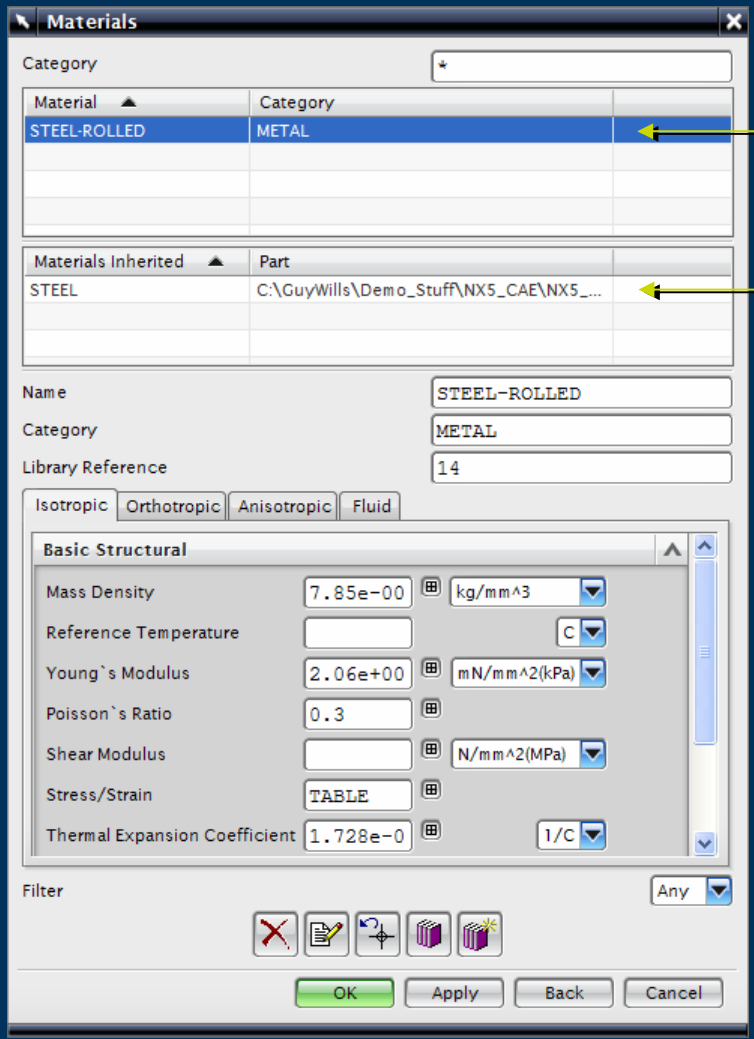



Bolt Holes moved

# Idealize Part – Material Properties




Override Material

Master Part Material

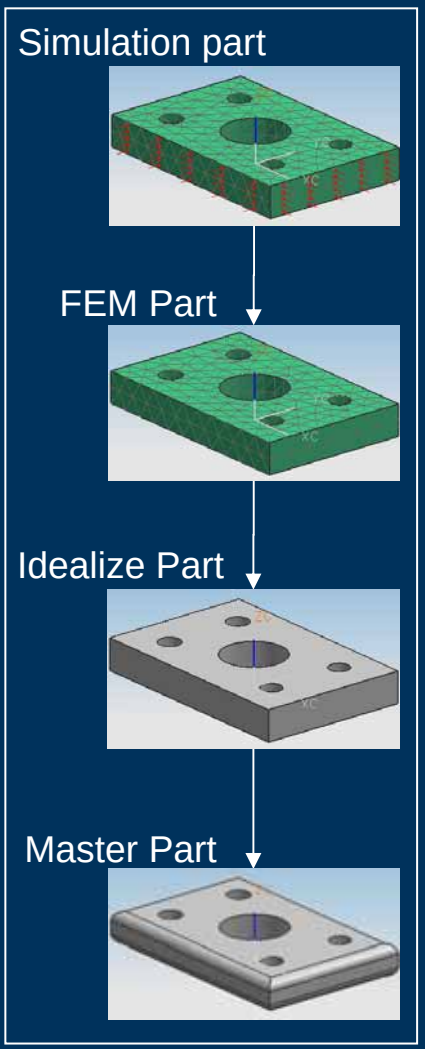
- Set the Material Properties of the body(s) different from the Master Part
- “ What If ” studies of different materials

SIEMENS

# FEM Part

SIEMENS

# FEM Part

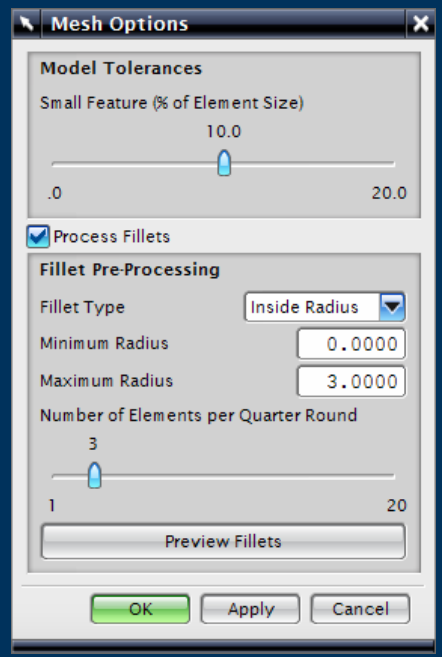



- Uses of the FEM part
  - Geometry Abstraction – CAE Topology
  - Model Organisation using Collectors
  - Meshing
    - Automatic
    - Manual
  - Mesh Connections
  - Model Checking

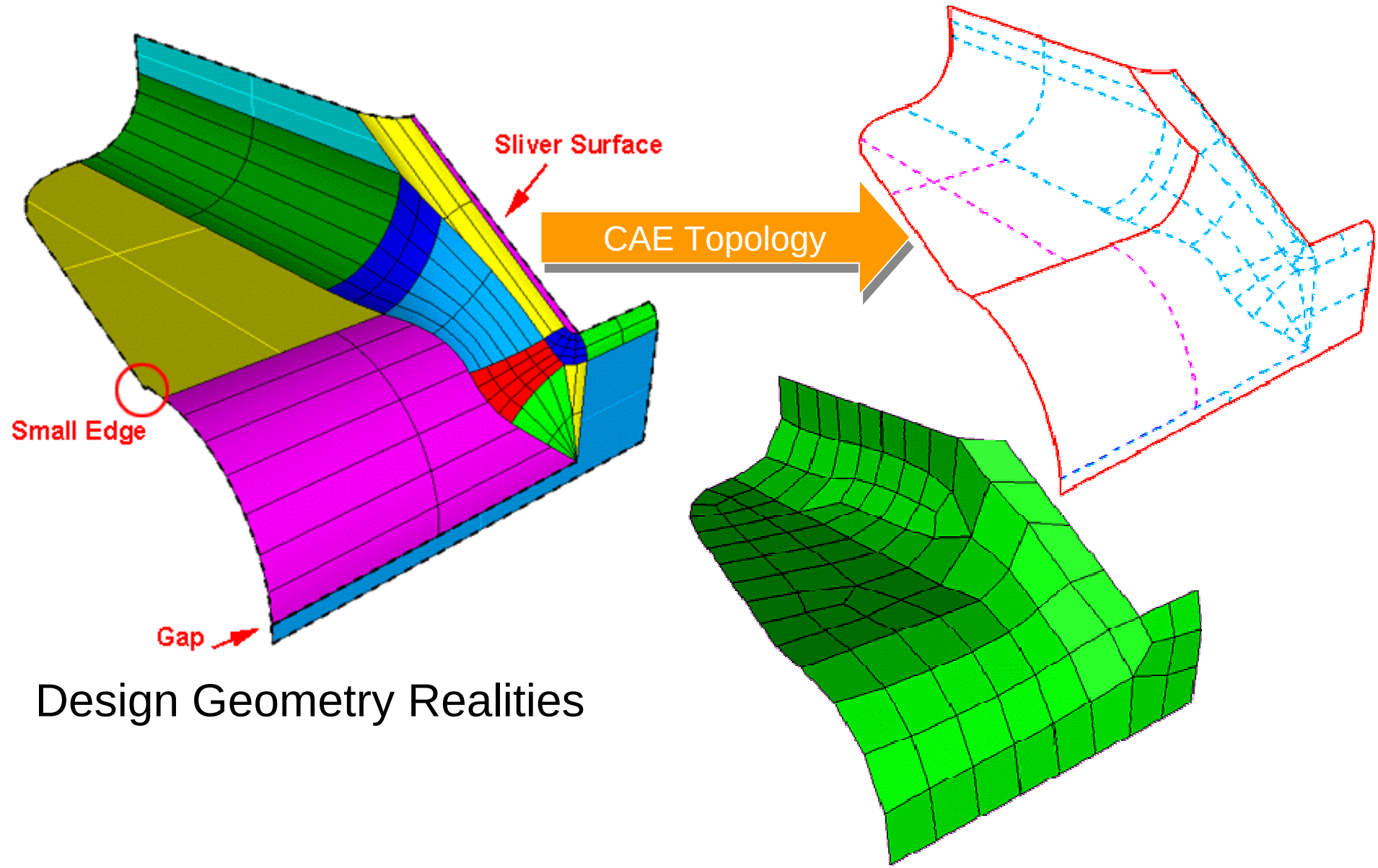
SIEMENS

# NX Advanced Simulation : CAE Topology

- CAE Topology
  - What is it?
    - An abstracted layer of CAE specific topology with CAE specific modeling tools, over and above that provided by CAD
    - Initially one polygon face is created for each CAD face
  - What does it do?
    - Automatically simplifies geometry by removing irregular and tiny features to allow effective CAE meshing
    - Fully Manual through to a Fully Automatic process. Best practise is a mix of Manual and Automatic simplification
  - Why is it valuable?
    - Reduces the time to mesh and the number of elements generated (reducing solve time) while improving element quality and results accuracy



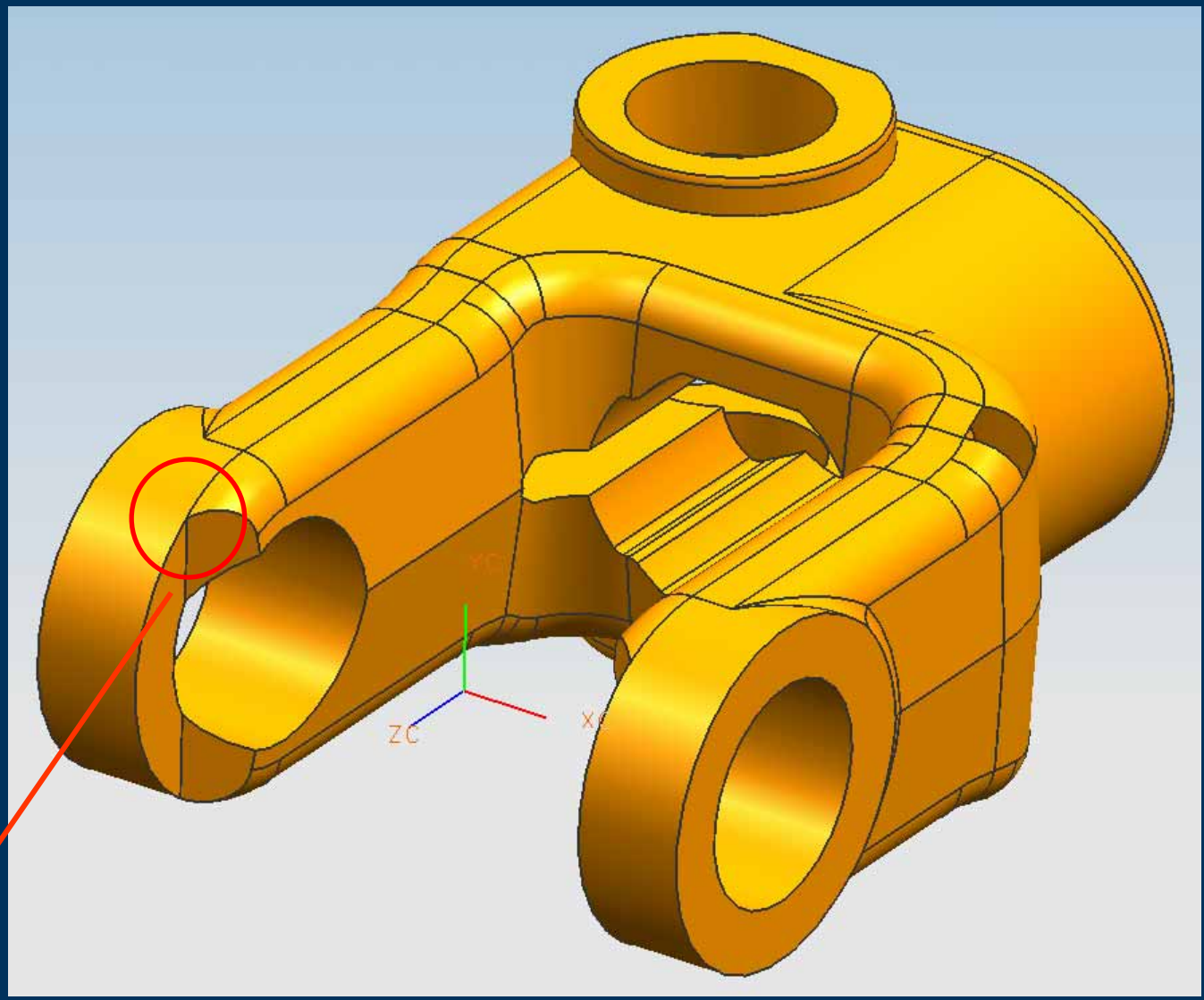
# NX CAE Topology
# – Geometric Abstraction and Meshing




Design Geometry Realities

SIEMENS

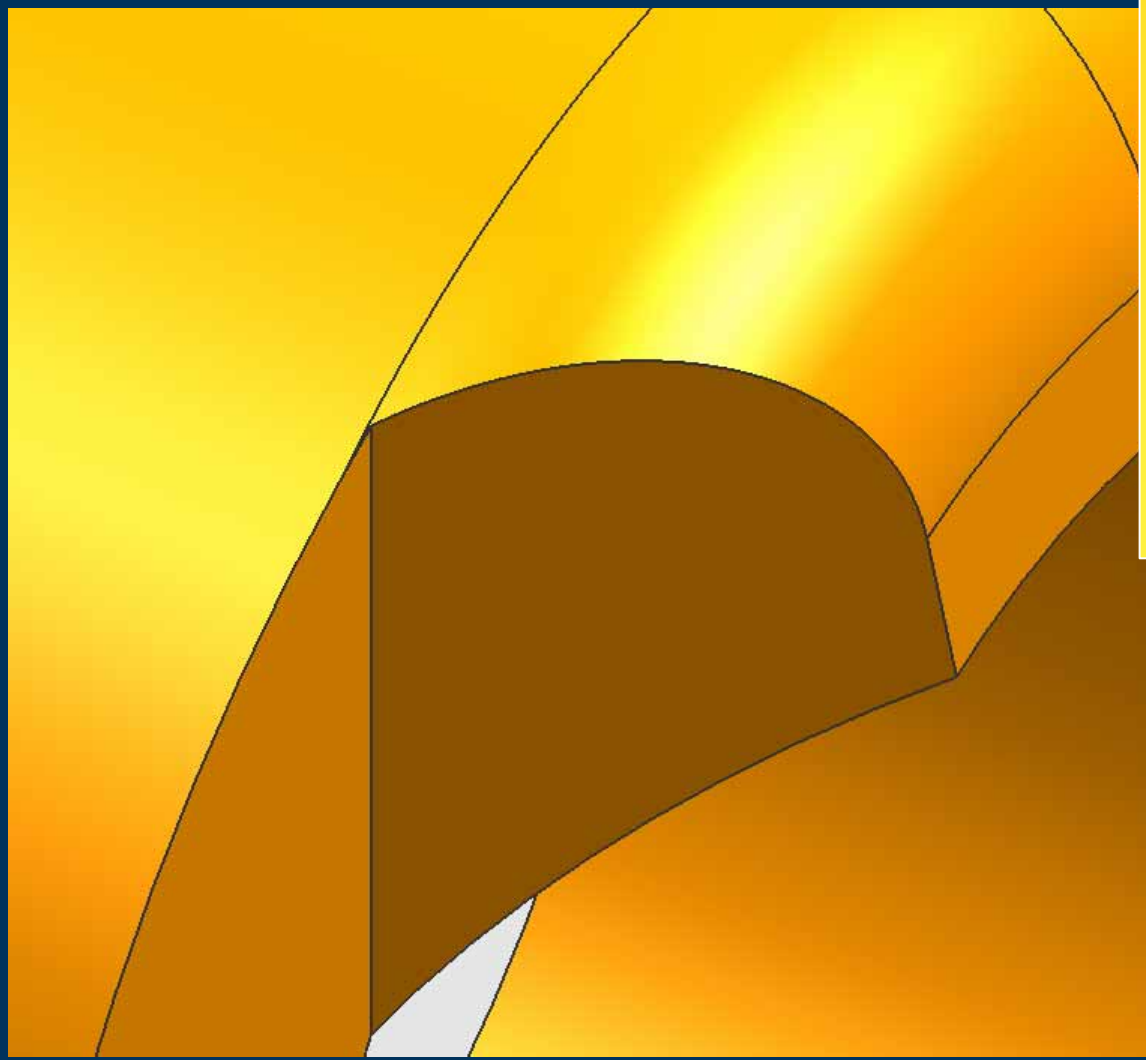
# NX CAE Topology

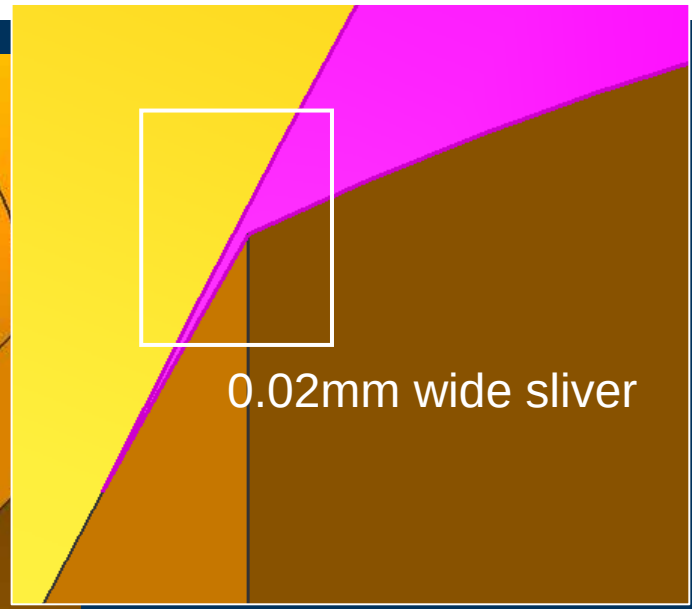



Issues

SIEMENS

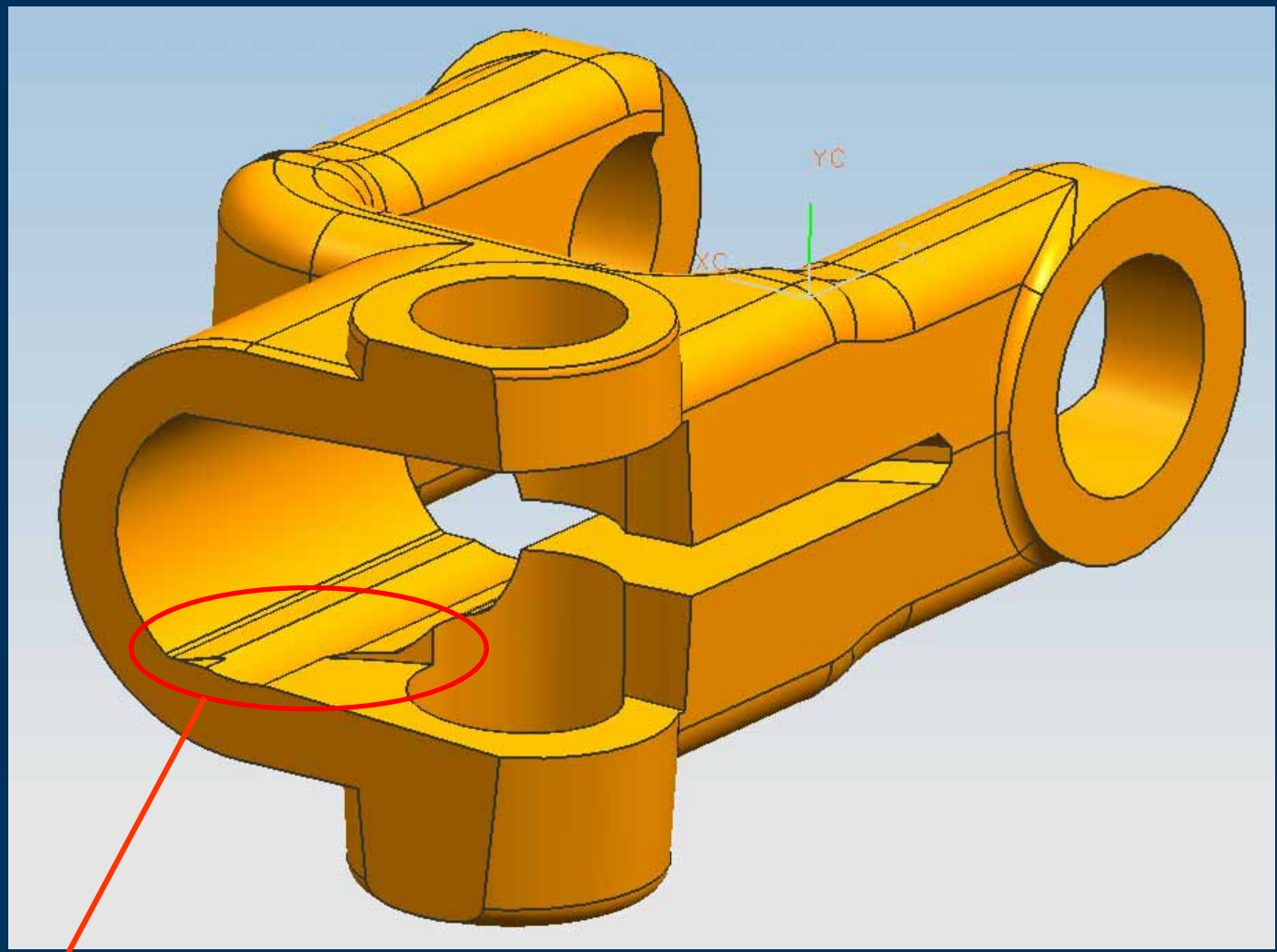
# NX CAE Topology






SIEMENS

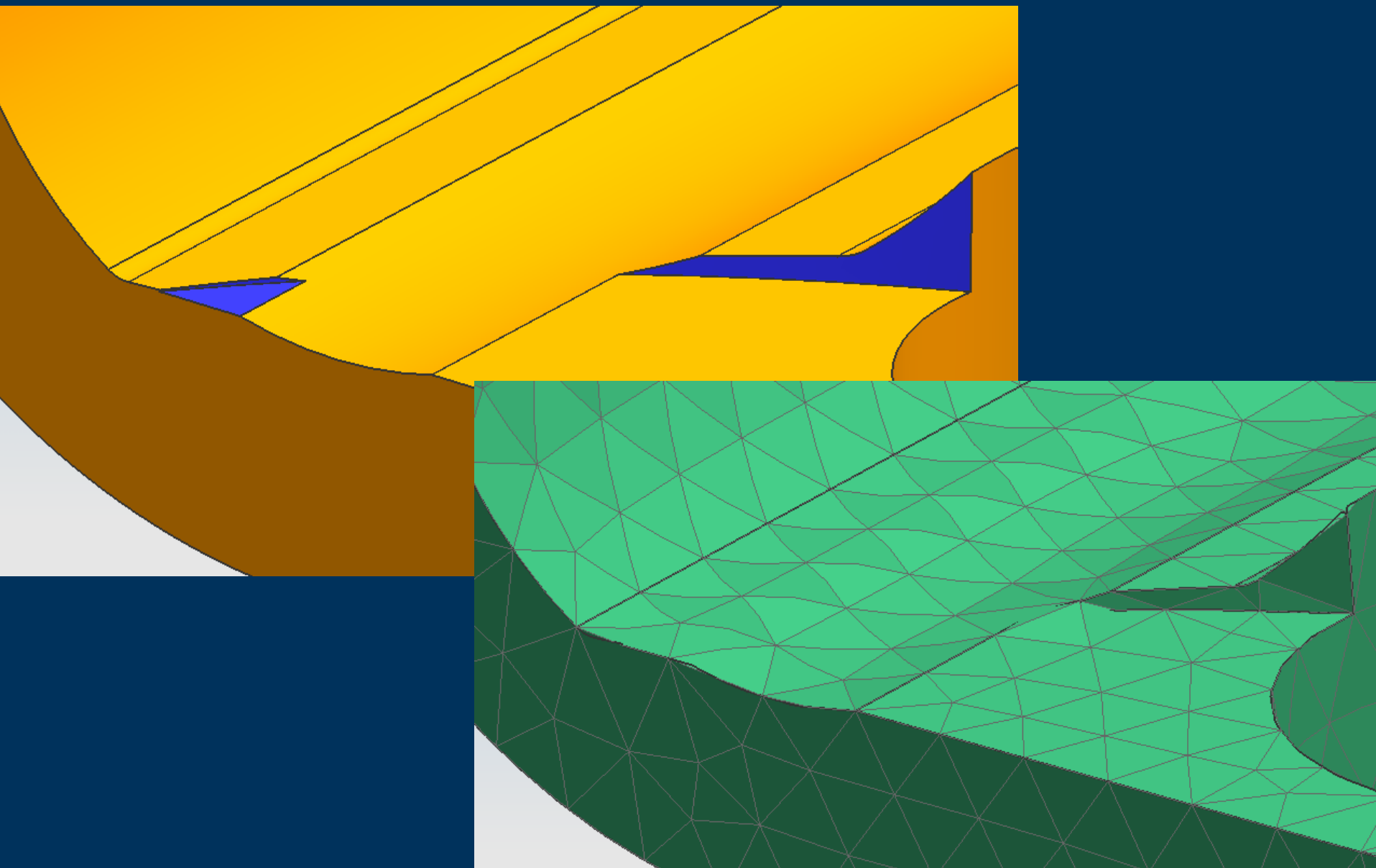
# NX CAE Topology




Issues

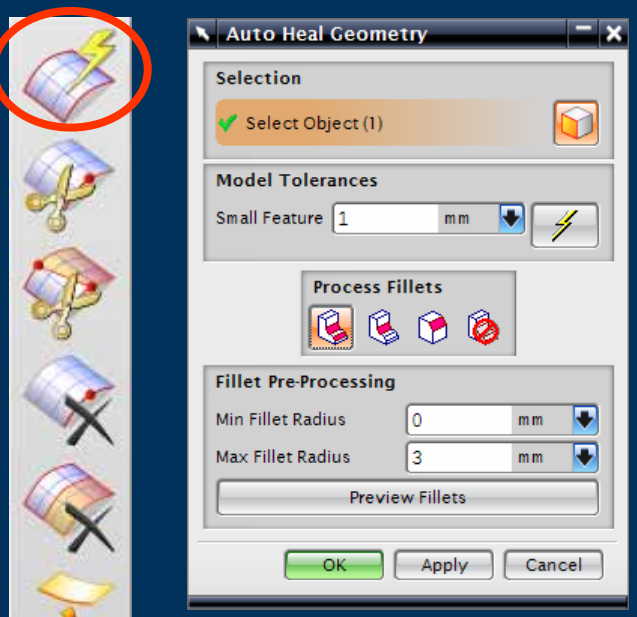
SIEMENS

# NX CAE Topology

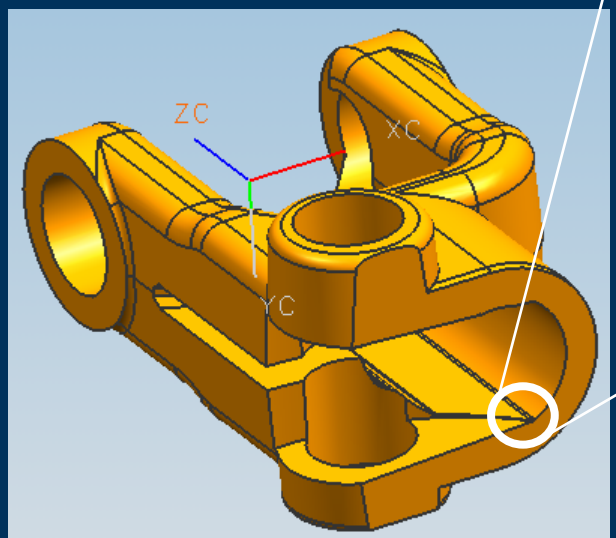


SIEMENS

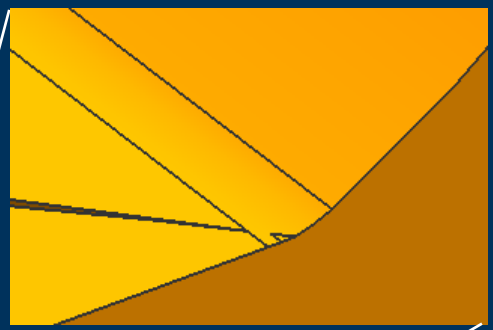
# NX CAE Topology – Auto Heal

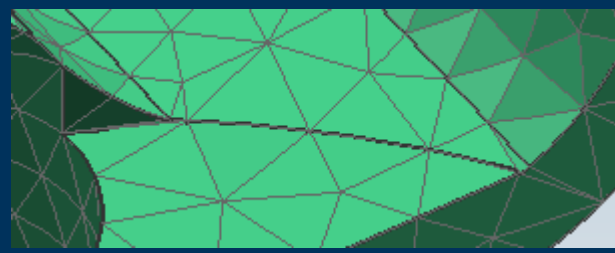



- Healing of CAE Topology
  - Selected faces
  - Complete model
  - Auto calculation of “Small Feature” value
  - Also removes sharp sliver like corners




Before Heal

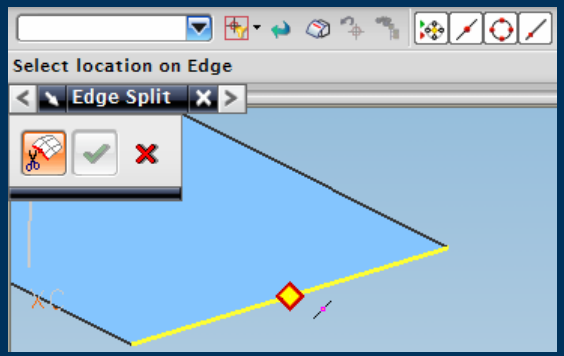




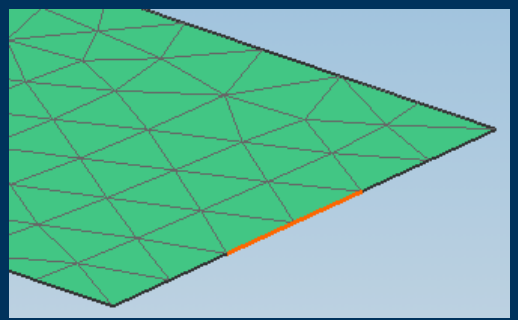
After Heal

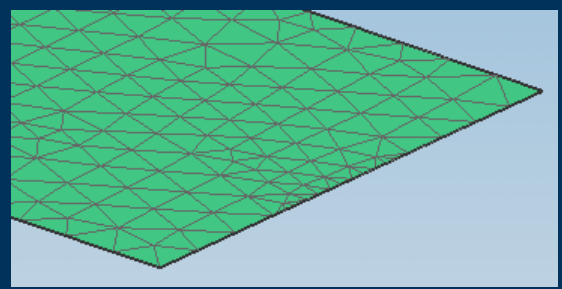
SIEMENS

# NX CAE Topology – Split Edge



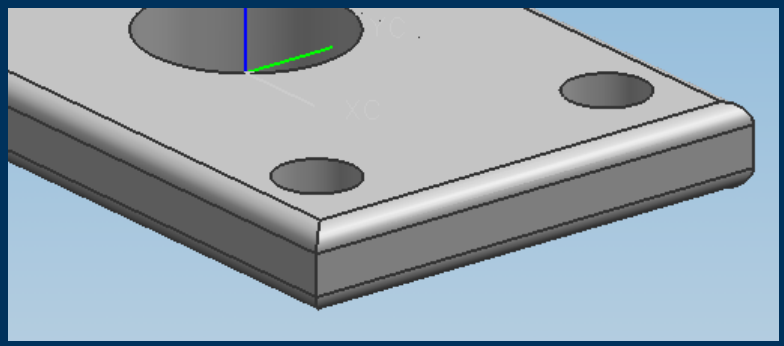

- Split an Edge
  - To define separate Boundary Conditions along a Polygon edge
  - Point Selection
  - Control mesh density

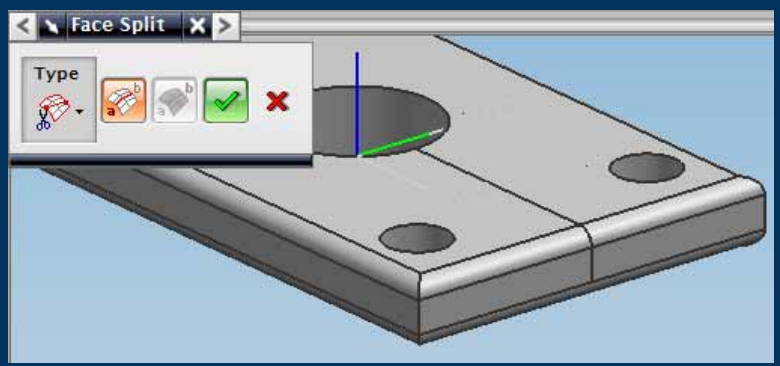


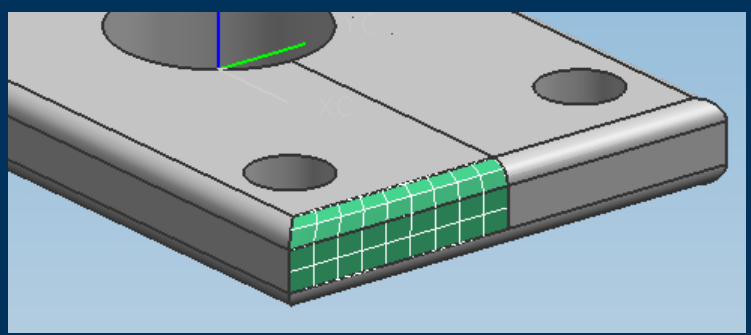


SIEMENS

# NX CAE Topology – Split Face



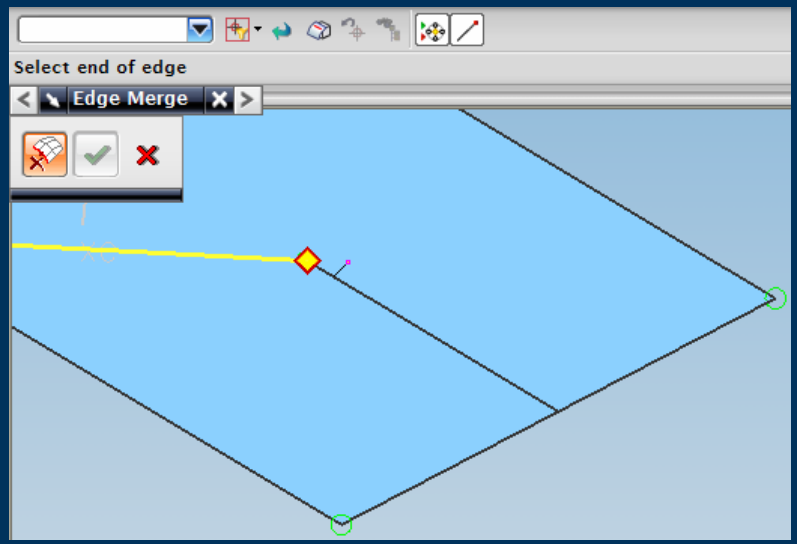


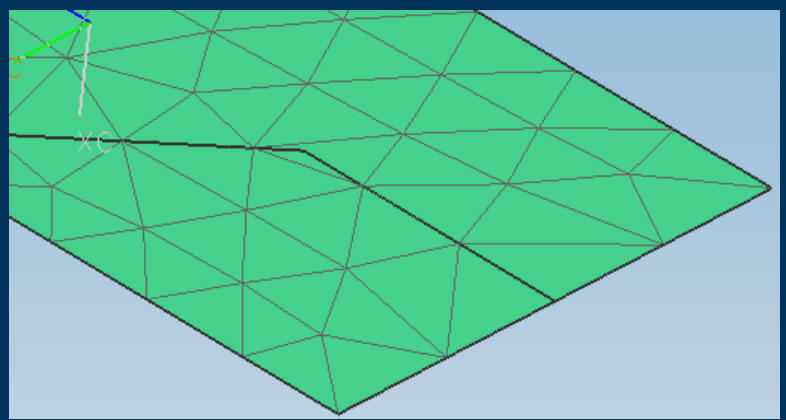
- Split Face
  - Split a polygon face along a projected line
  - Mesh control
  - Boundary Condition control

SIEMENS

# NX CAE Topology – Merge Edge

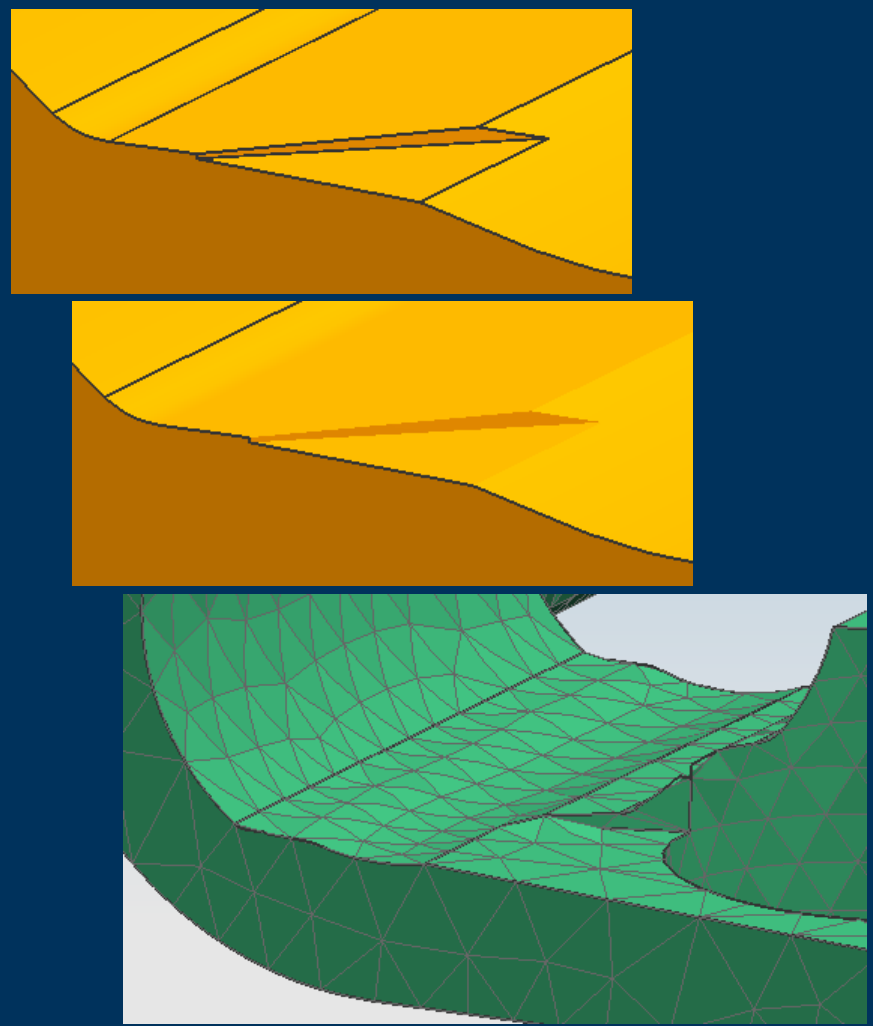



- Merge Edge
  - Merges 2 polygon edges that share a vertex into one polygon edge
  - Used to “recover” Split Edges



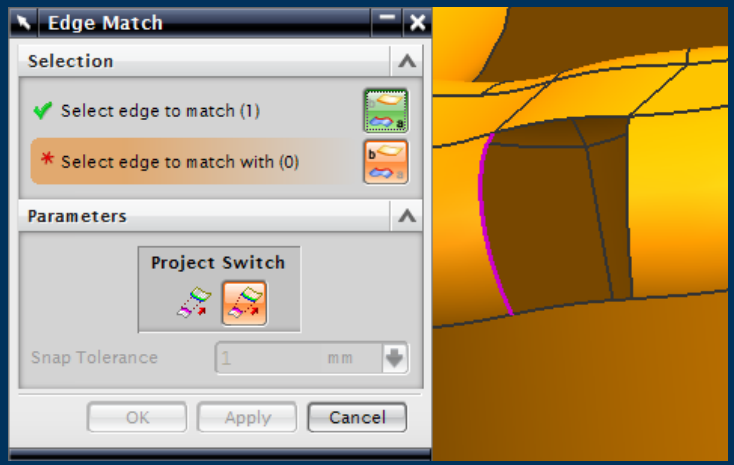
SIEMENS

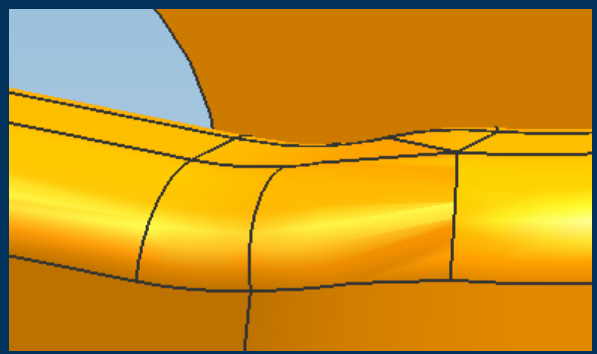
# NX CAE Topology – Merge Face



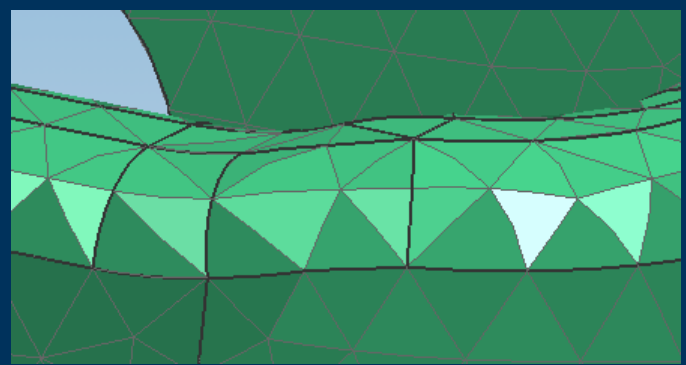
- Merge Face
  - Merge two separate polygon faces into a single polygon face along a common polygon edge
- Commonly before and after an Auto Heal
- Before or After Meshing
- Used to remove data to get a better quality mesh

# NX CAE Topology – Match Edge



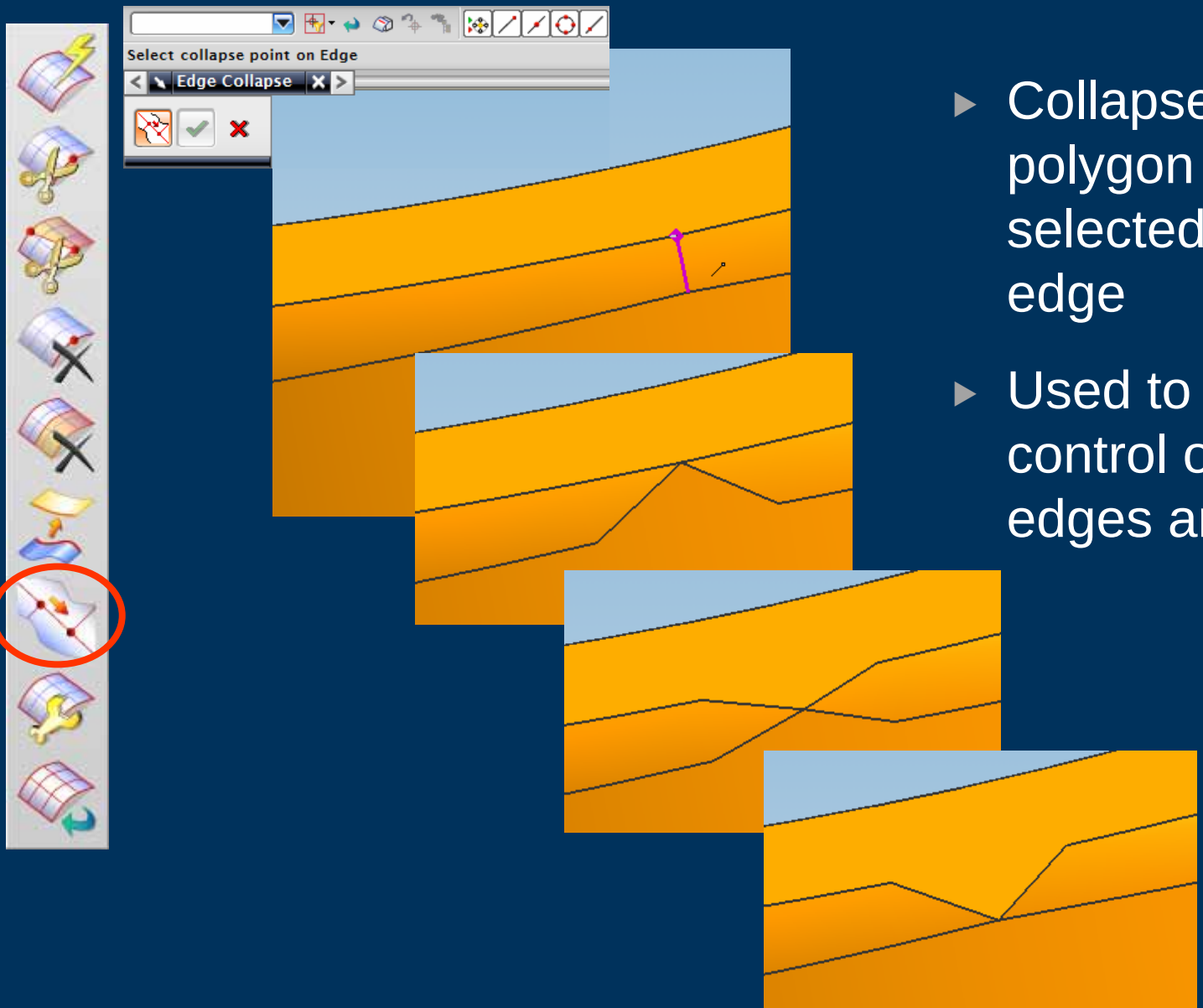



- Match one polygon edge to a second polygon edge
- Result is a single polygon edge
- Used to “tidy up” or repair poor quality geometry



SIEMENS

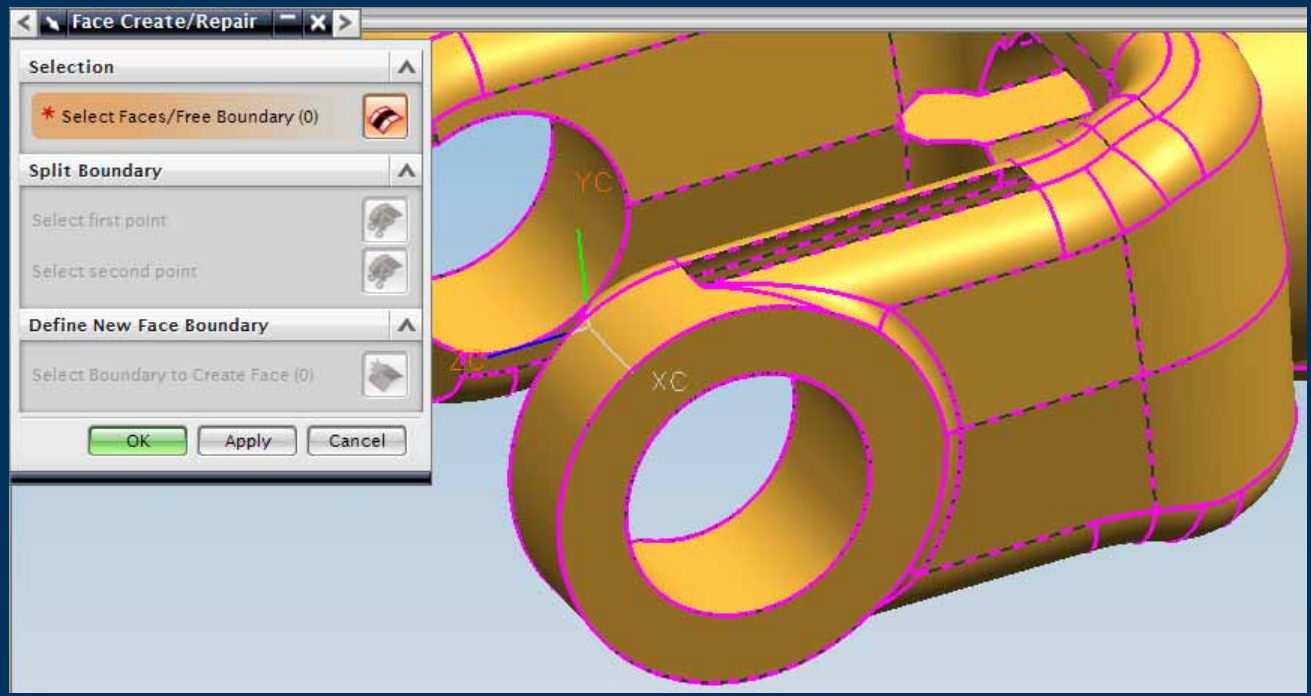
# NX CAE Topology – Collapse Edge

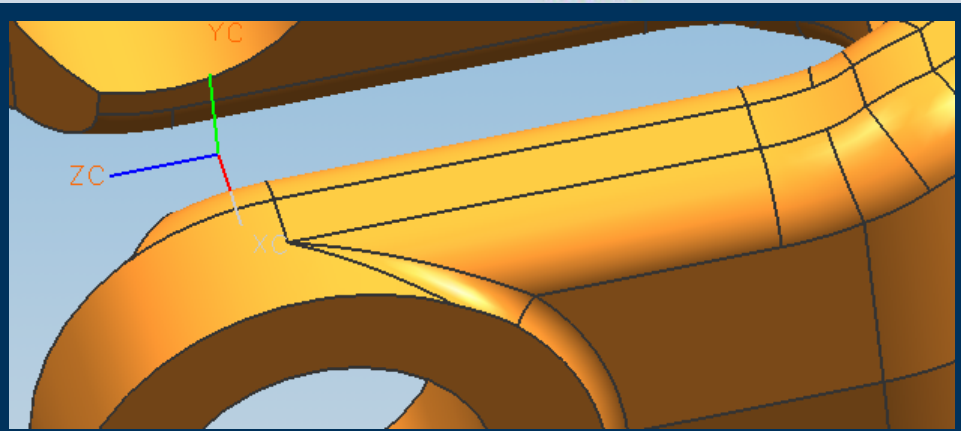



- Collapses the selected polygon edge to a selected point on the edge
- Used to get manual control over how small edges are collapsed

SIEMENS

# NX CAE Topology – Face Repair

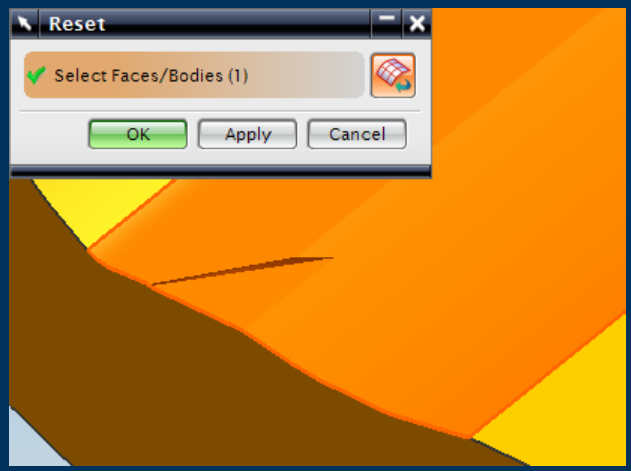


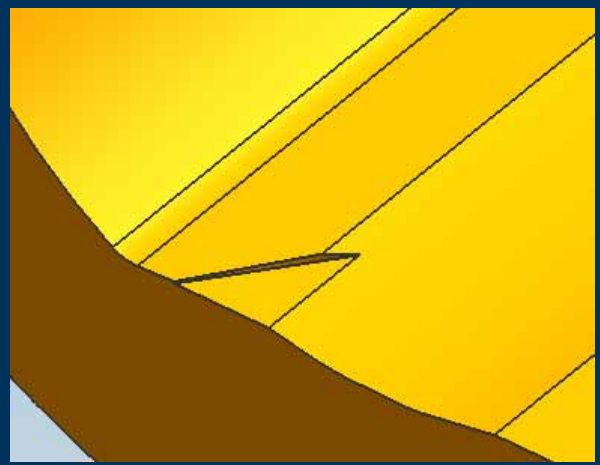




- Create a new polygon face to fill a hole
- Repair a poor quality polygon face

SIEMENS

# NX CAE Topology – Reset

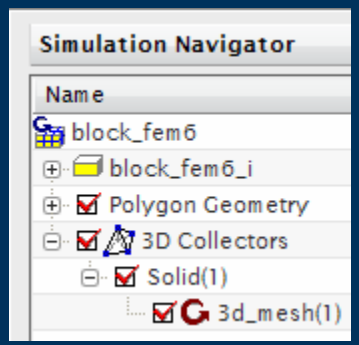


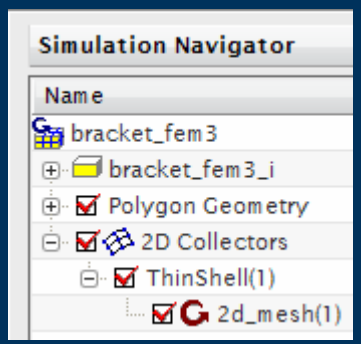



- Resets the selected polygon geometry to it's original state – ie one for one with the CAD surfaces
- Recover data for including in Mesh

SIEMENS

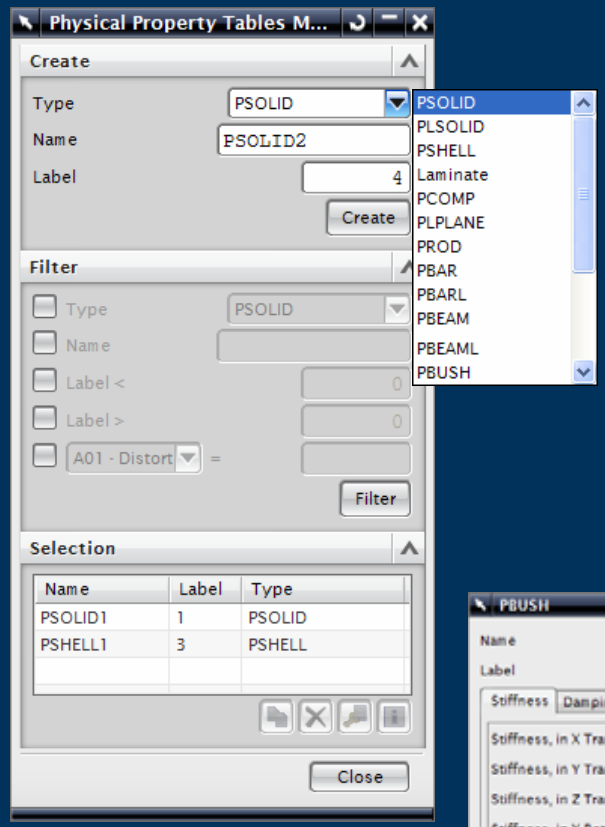
# NX CAE Topology – Mesh Updates



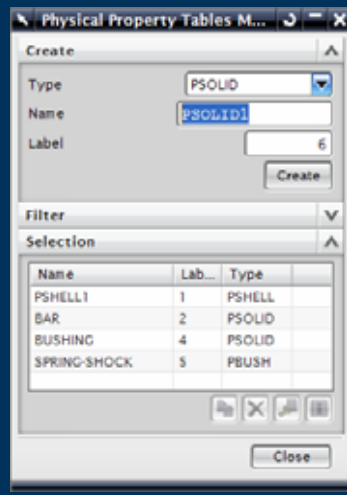



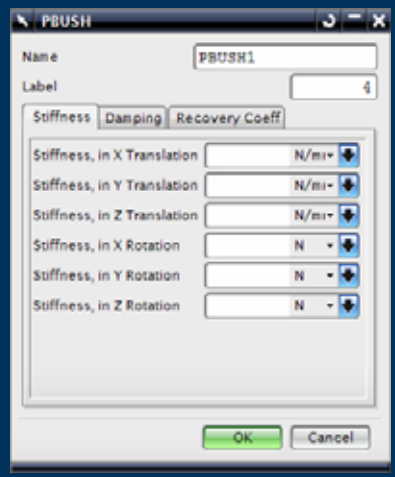

- CAE Topology changes can be done before and after Meshes are applied
- After a change (like Merge Face) the Mesh is flagged “out of date”
  - In the Simulation Navigator
  - Mesh Update icon
- Note if multiple meshes exist, only the changed ones are flagged as “out of date” and updated
- Allows for multiple CAE Topology changes and one mesh update

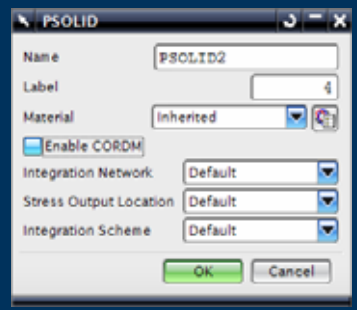
# Physical Properties

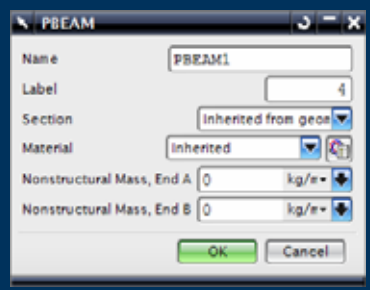


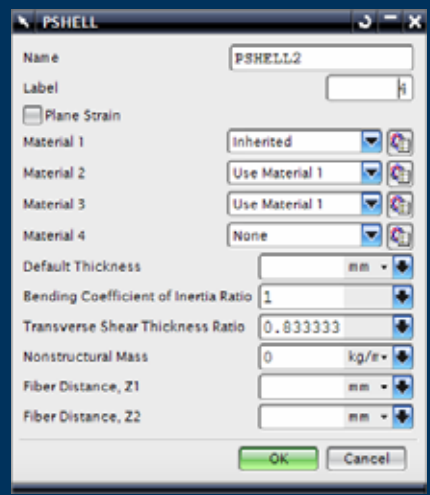

- Physical Property is Solver and Element dependant
  - Family of Elements
  - Material Reference
  - Commonly referred to as PID
  - Often used to identify different parts in an Assembly



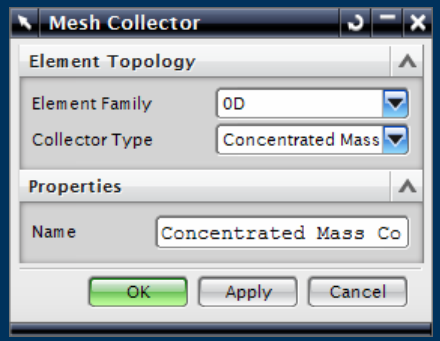



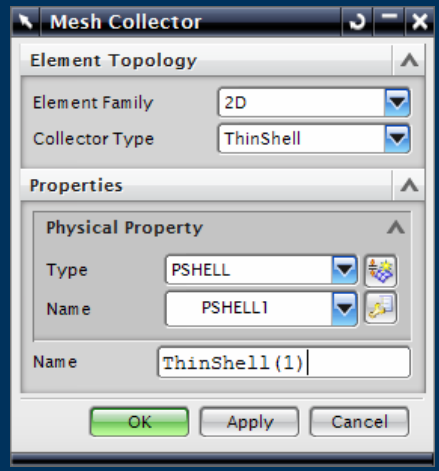



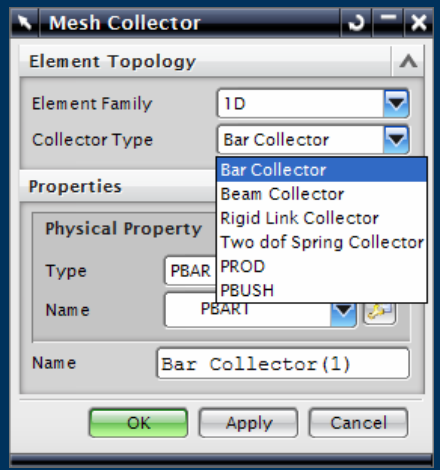



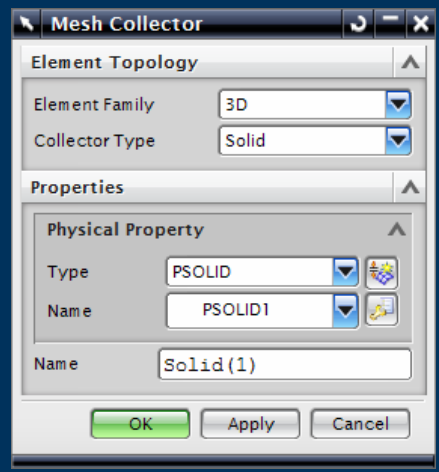



# Mesh Collectors

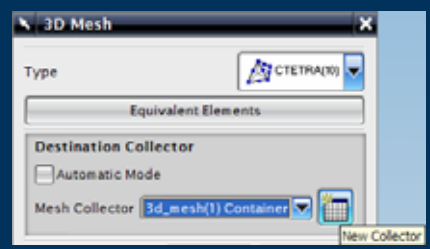


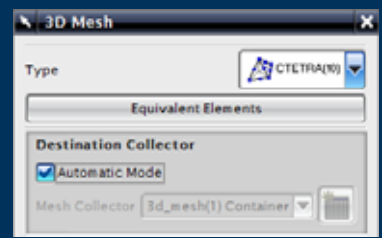



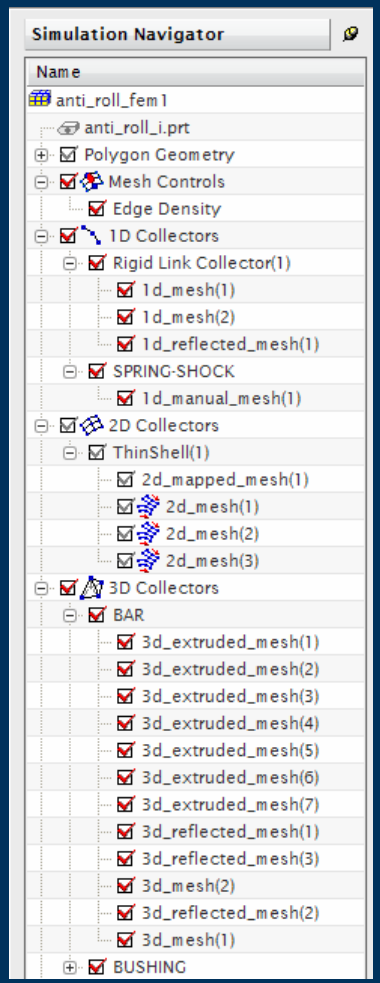




- Mesh Collectors are a method for multiple Meshes to reference the same Material, Physical Properties and Display Properties
- Organised by
  - Element Family
  - Element Type
  - Physical Property (inc Material)
- Workflow Creation Options
  - Prior to Meshing
  - On-the-fly During Meshing
  - Post Meshing



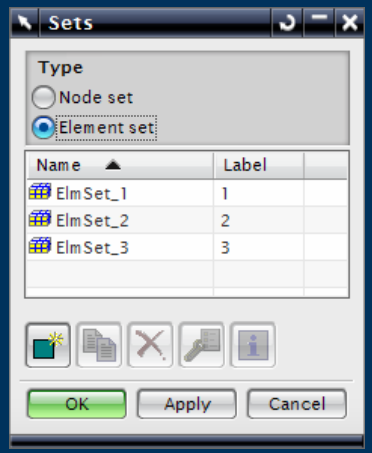



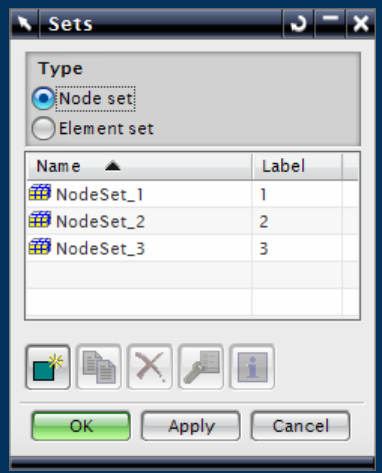
# Mesh Collectors

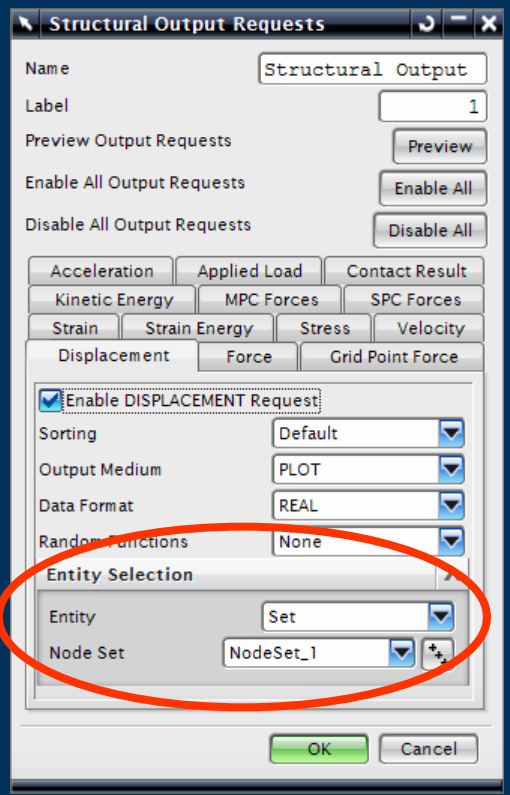



- Model Management
  - Drag ‘n’ Drop item between collectors
  - Mesh inherits the target Collector properties inc Physical, Material and Display
  - Display control
    - Hide/Show all Meshes in Collector
    - Hide/Show Individual Meshes
- Benefits
  - Model management
  - Visible model organization
  - Fast and easy to use for detail or global changes

# Node and Element Sets

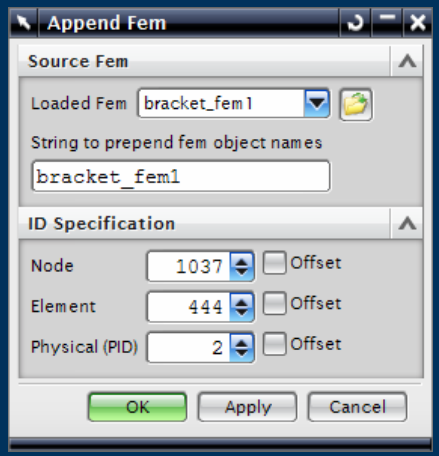


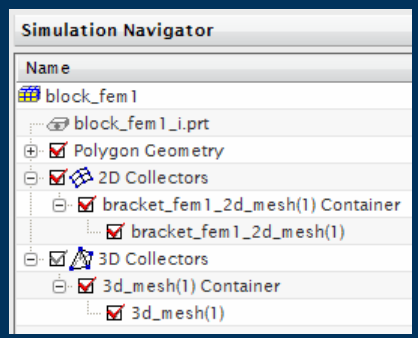




- Named Collection of Nodes or Elements
- Used for defining output for a solution
- FEM Based Sets can be used by any referencing SIM
- SIM Based Sets are only available within that SIM file

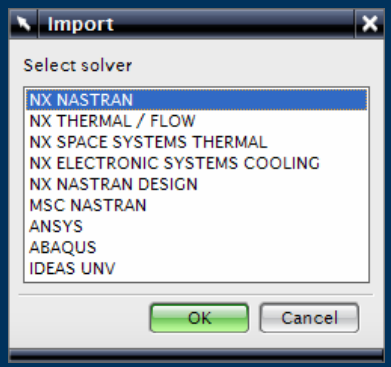
# Mesh Append

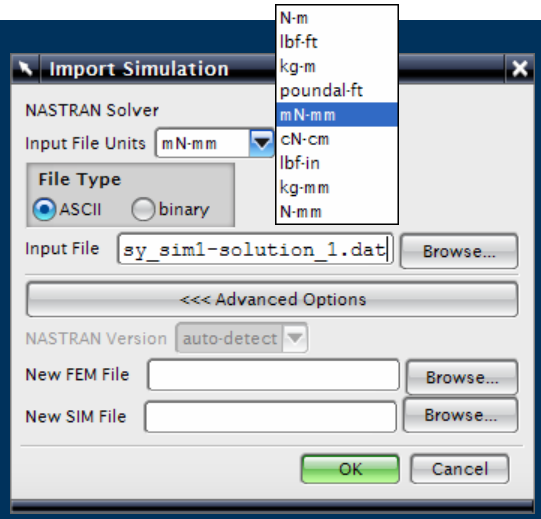


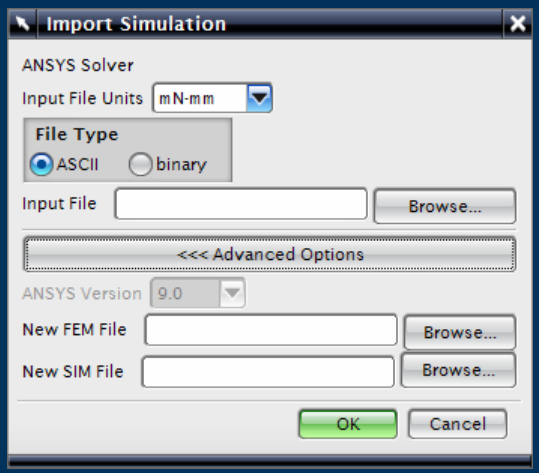



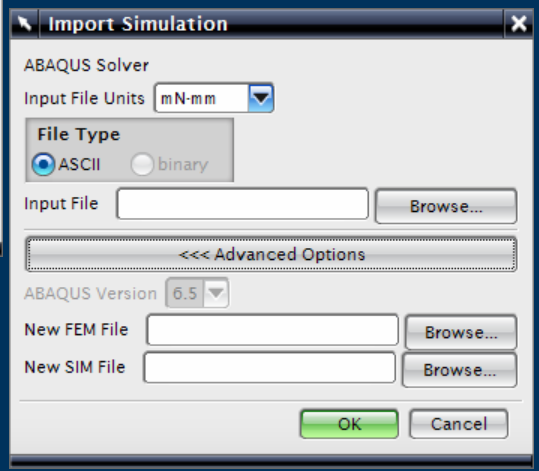

- Mesh Append copies Mesh from one FEM file into the current FEM file
  - Optional Prefix to Imported object Names
  - Node, Element & PID number Start and Offsets

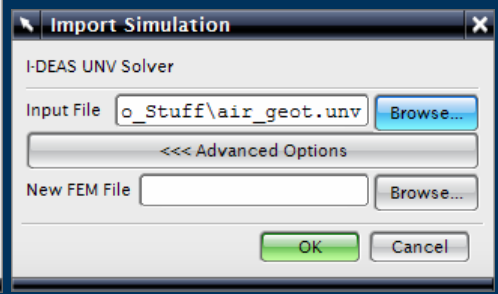
# Mesh Import



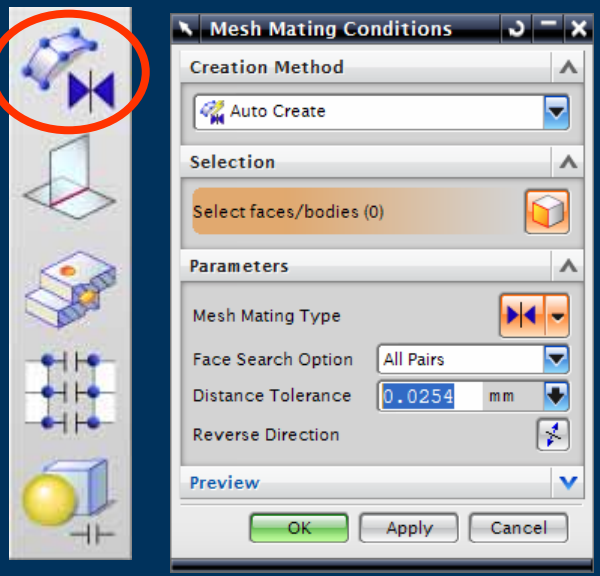



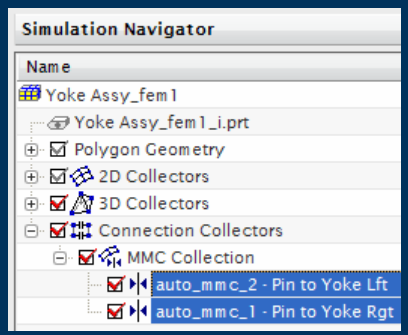



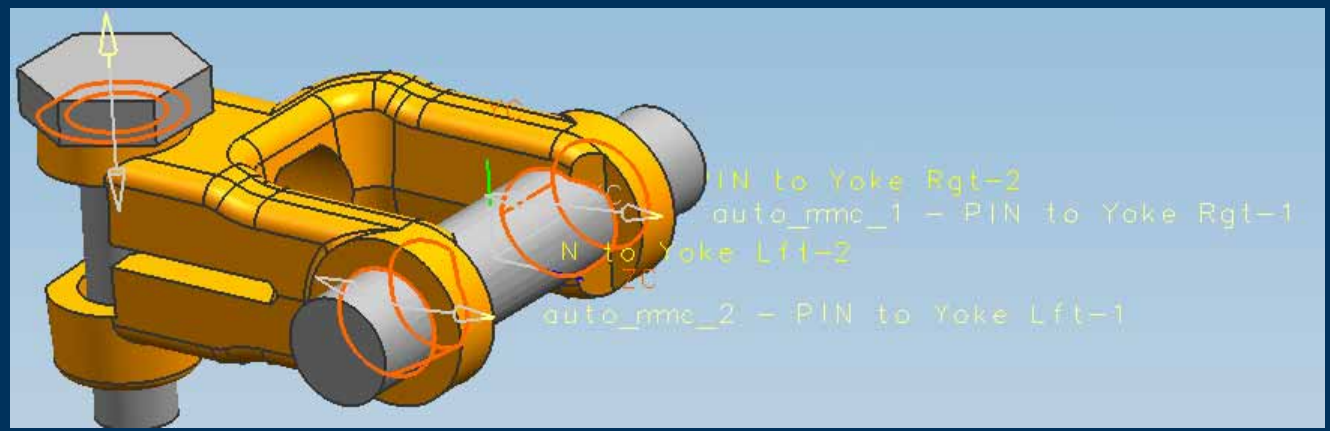

- Import of a solver deck from an External file
  - Units selection for the incoming data
  - Append to existing files or Create New files

# Mesh Connections – Mesh Mating



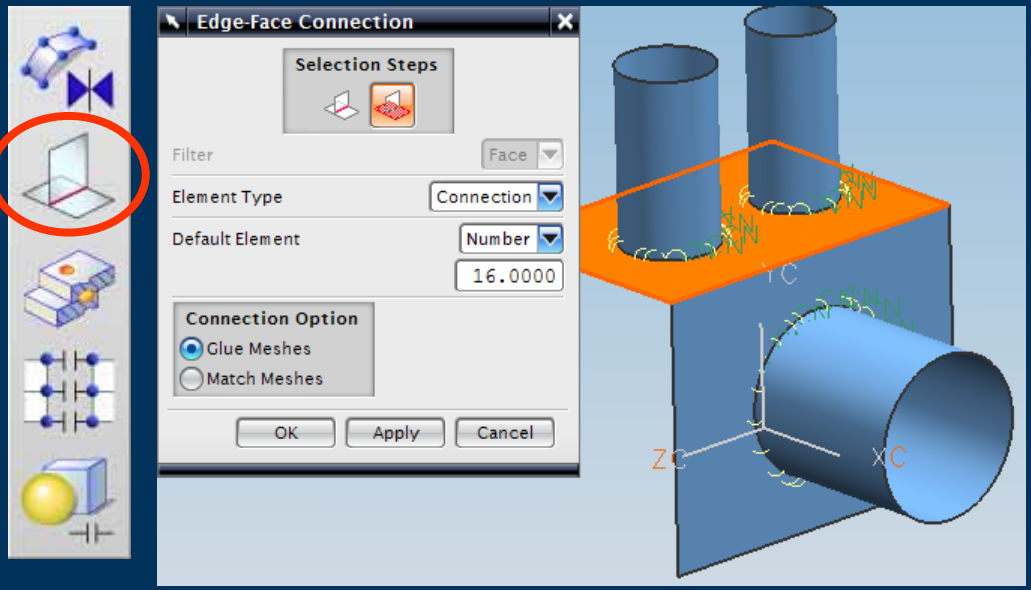

- Mesh Mating Conditions aligns the mesh on Source and Target
  - Glue Coincident condition
    - 2 faces share same nodes
  - Glue Non-Coincident condition
    - Multi-Point Constraints (MPC's) to connect the meshes
  - Free Coincident condition
    - No mesh connection
- Auto Detection or Manual Selection of mating faces
  - Search all possible pairs or only for identical pairs of faces



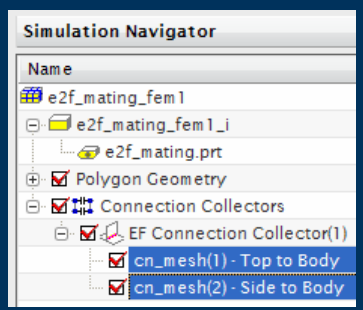



SIEMENS

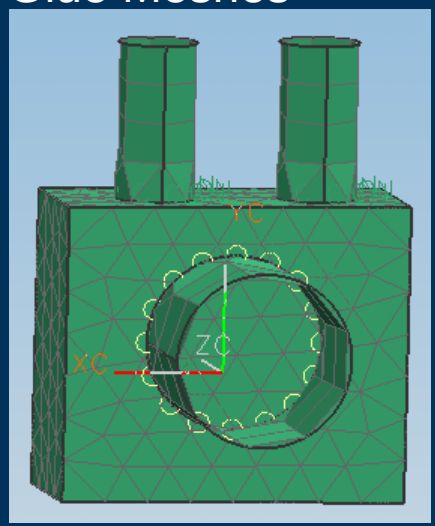
# Mesh Connections – Edge-Face Connection



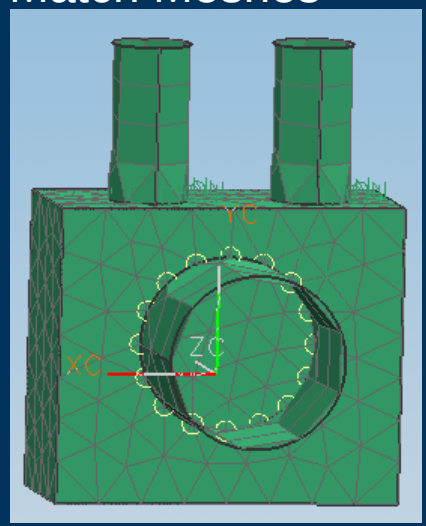

- Connection between a set of edges and a set of faces
- Contact Node drive Meshes in both sides if Match Meshes used
- Uses Rigid Links and MPC's to connect the meshes



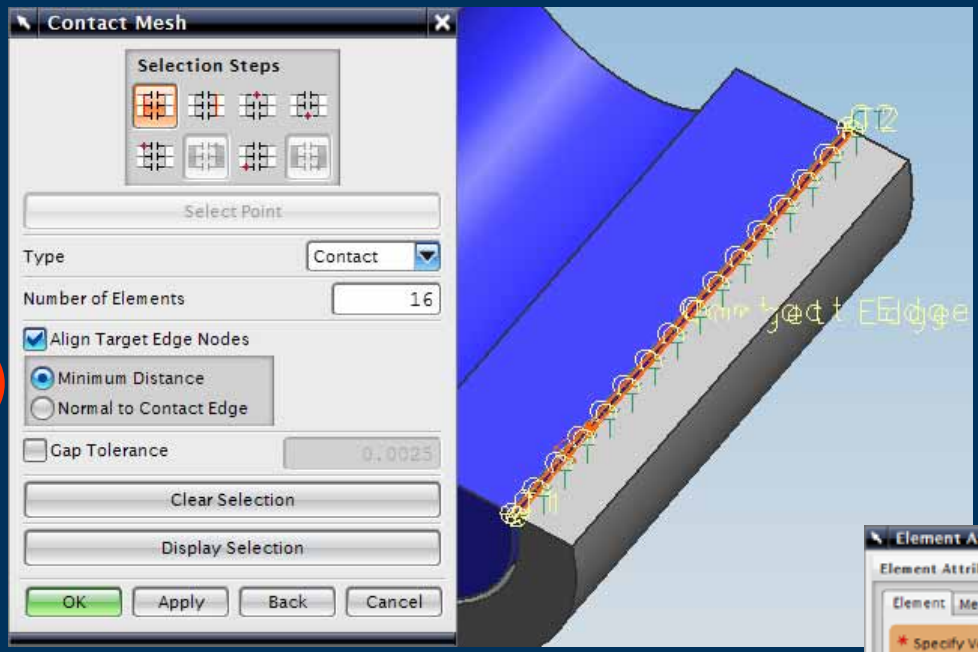

Glue Meshes



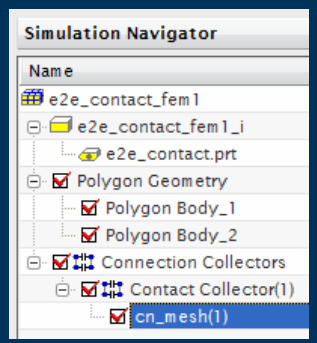

Match Meshes

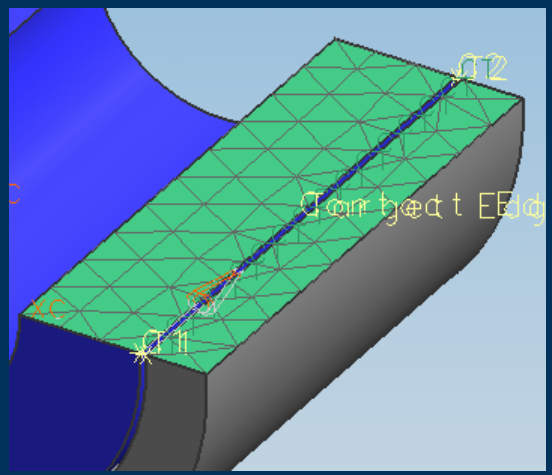


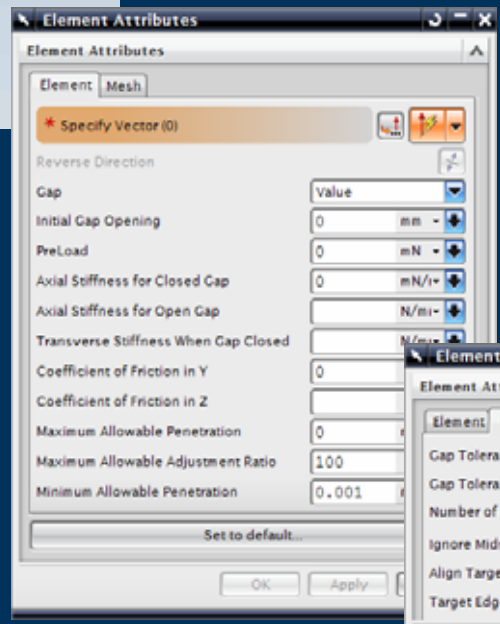
# Mesh Connections – Edge Contact Mesh

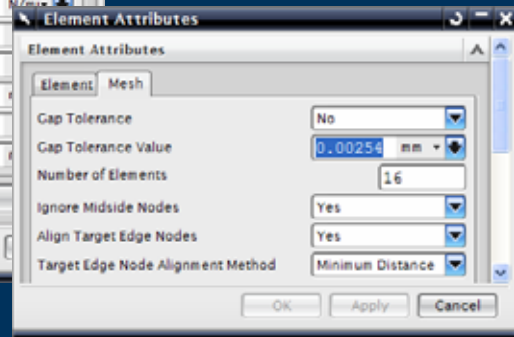



- Edge Based Contact definition
- Contact Nodes drive meshes on both sides
- Uses GAP elements to model Contact



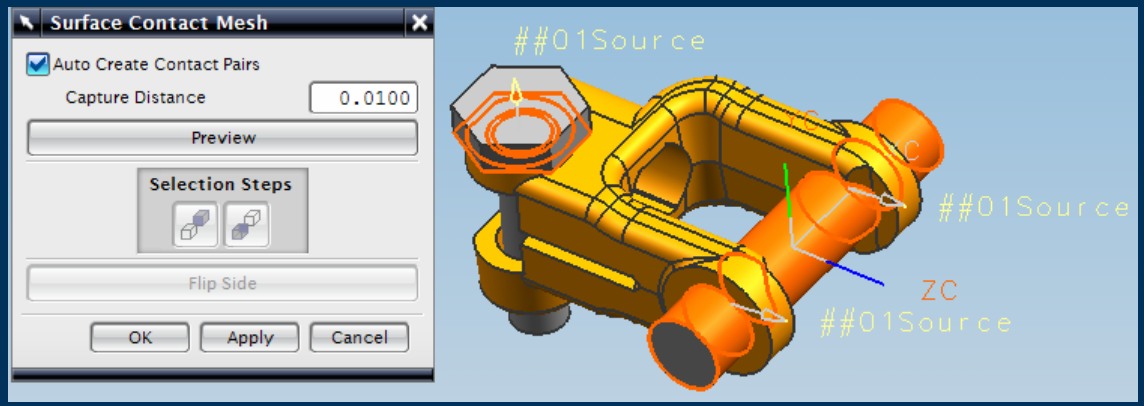





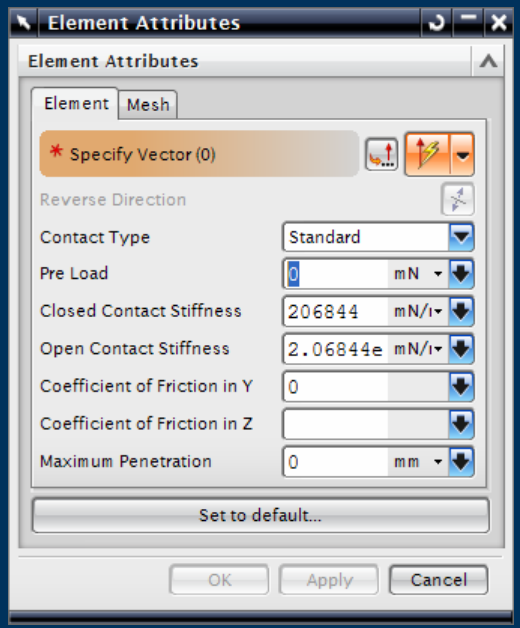



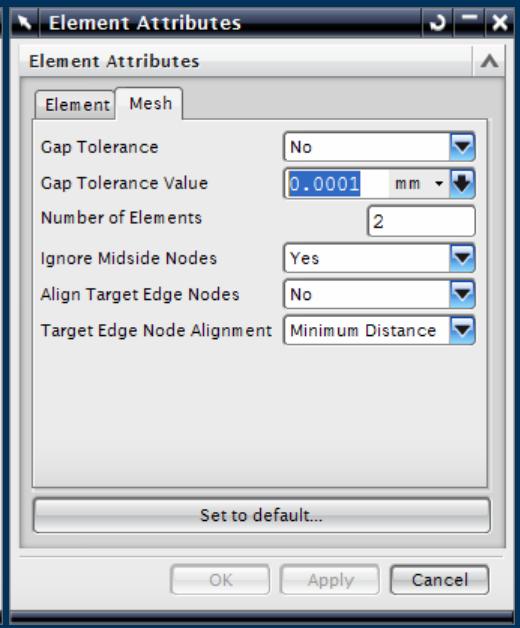
SIEMENS

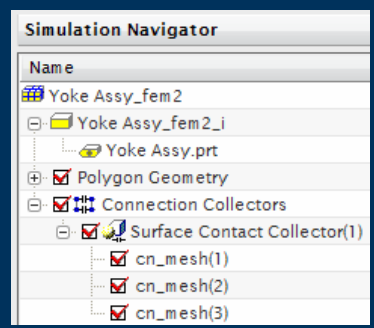
# Mesh Connections – Surface Contact Mesh




- Surface Based Contact definition
- Surfaces are not auto Split or Partitioned
- Contact Nodes drive meshes on both sides
- Uses GAP elements to model Contact

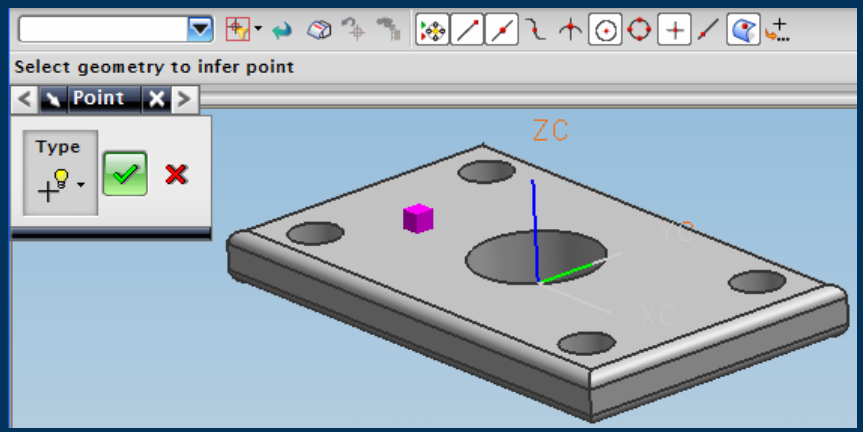


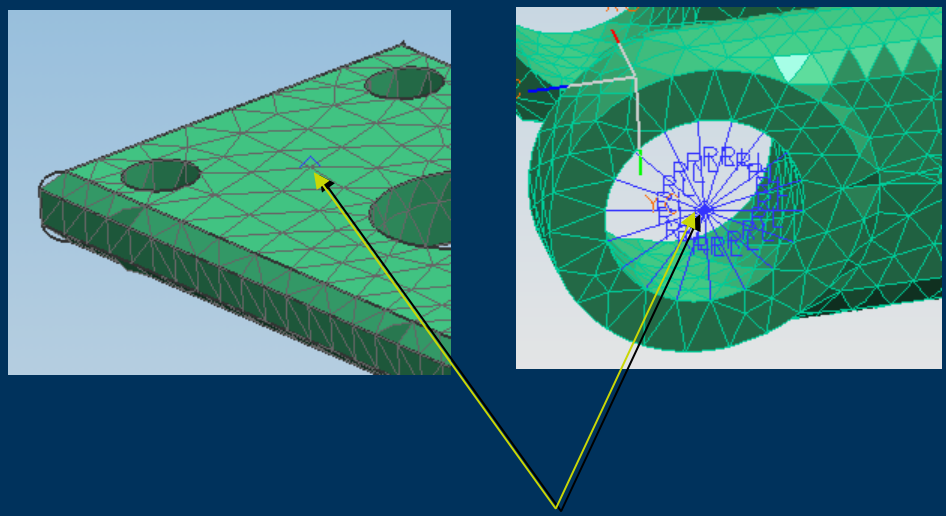



SIEMENS

# Meshing – Mesh Points

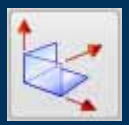


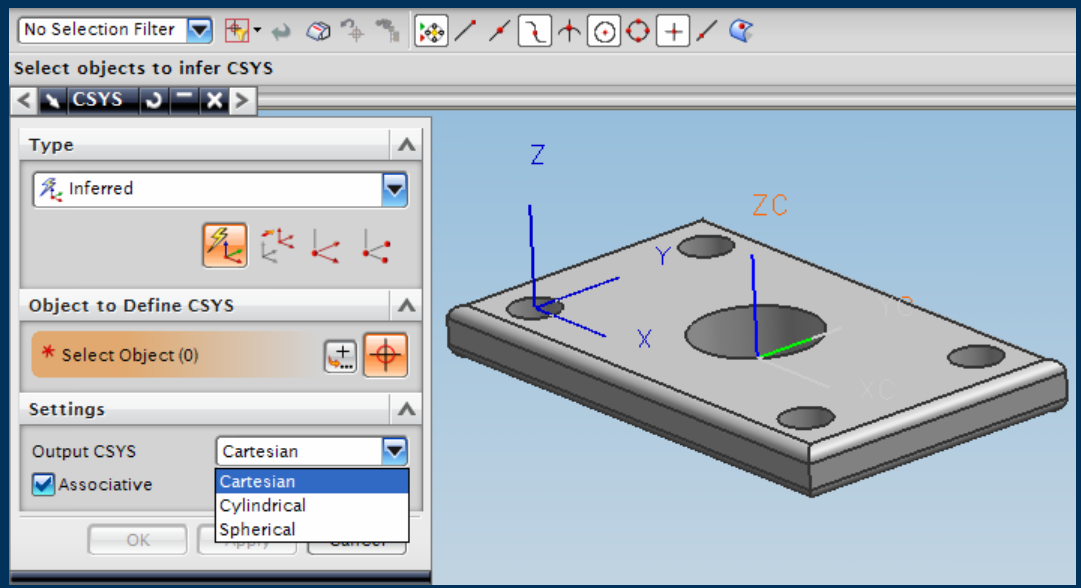




- Used to create specific location for a node, for example on an Edge or Face
- Point can Associative or Non-Associative to Geometry
- Mesh is associative to the Mesh Point
- Mesh Point location can be edited
- Used to create a location for a Load or Boundary Condition, spider for load transfer

SIEMENS

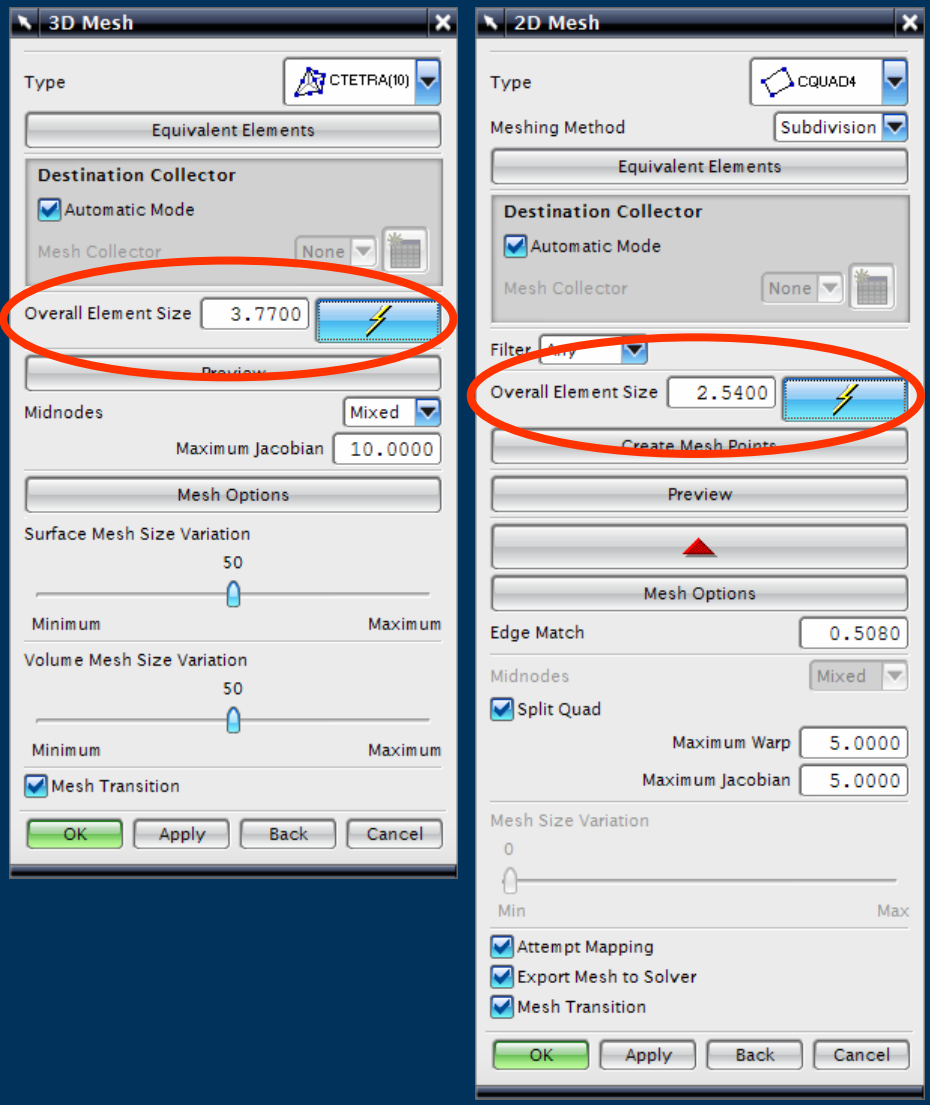
# Datum Coordinate Systems





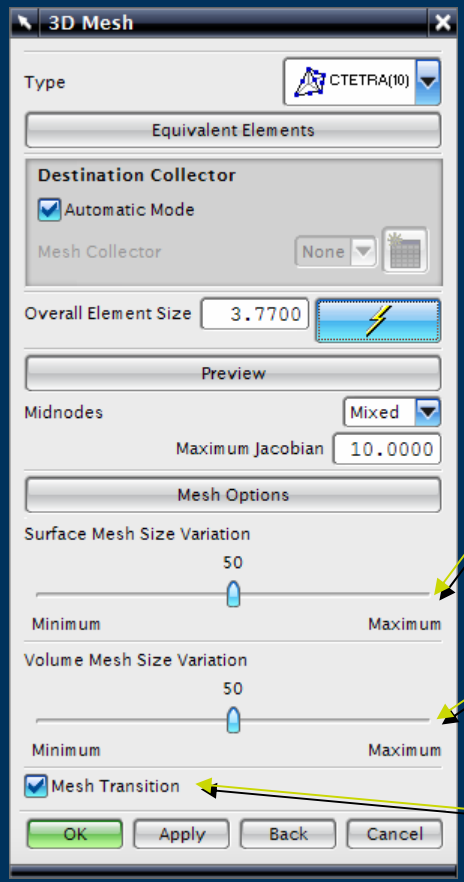

- Datum Coordinate Systems
  - Cartesian
  - Cylindrical
  - Spherical

# Mesh Size Selection



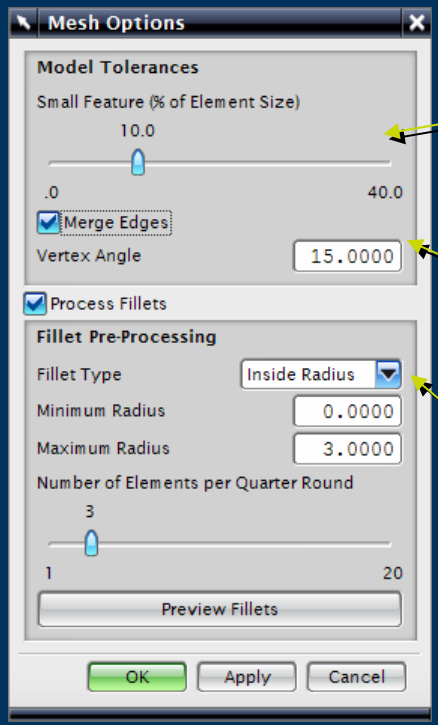

- The “Lightening” symbol will suggest an Overall Element Size based on examination of the selected geometry
- User can set a value appropriate to their task
- Default settings for everything else will give a “good mesh” for most geometry

# Mesh Size Selection



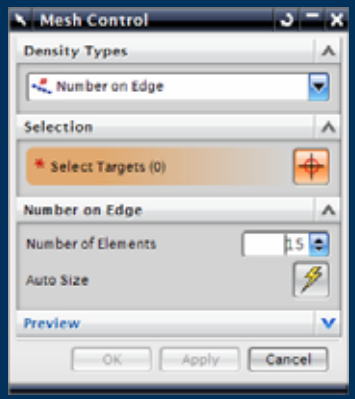

- Surface Mesh Size Variation
  - Min – less curvature refinement to follow geometry
  - Max – more curvature refinement to follow geometry
- Volume Mesh Size Variation
  - Min – elements remain approx constant in size throughout the body
  - Max – elements expand rapidly towards the center of the body
- Mesh Transition
  - Gradually transitions the size of elements in the mesh from any defined local element sizes back to the global element size

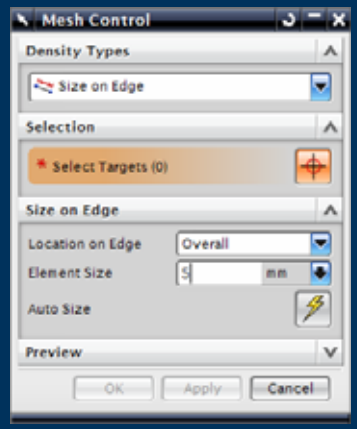
# Mesh Size Selection

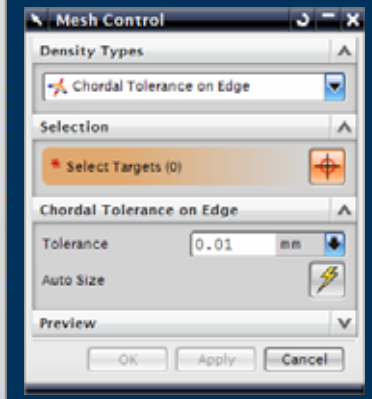


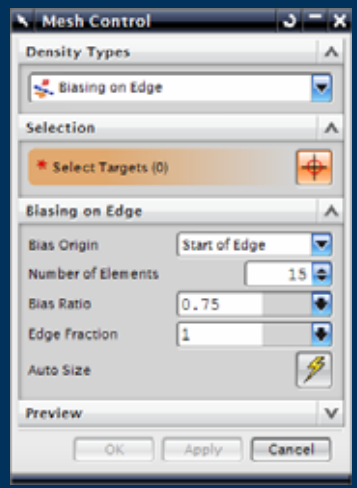

- Small Feature tolerance defines size of geometry that will be abstracted
  - Element size of 10mm & 10% setting will abstract out 1mm sized faces
- Merge Edges
  - Removes the Polygon edge when angle between edges is less than Vertex Angle
- Mapped Mesh control of Fillets/Blends faces
  - Filtered by Inside, Outside or Both
  - Min & Max radius

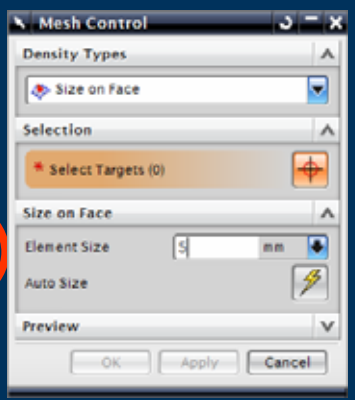
# Mesh Controls



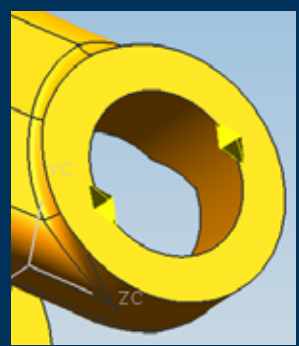



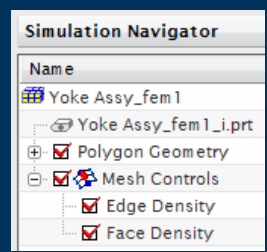



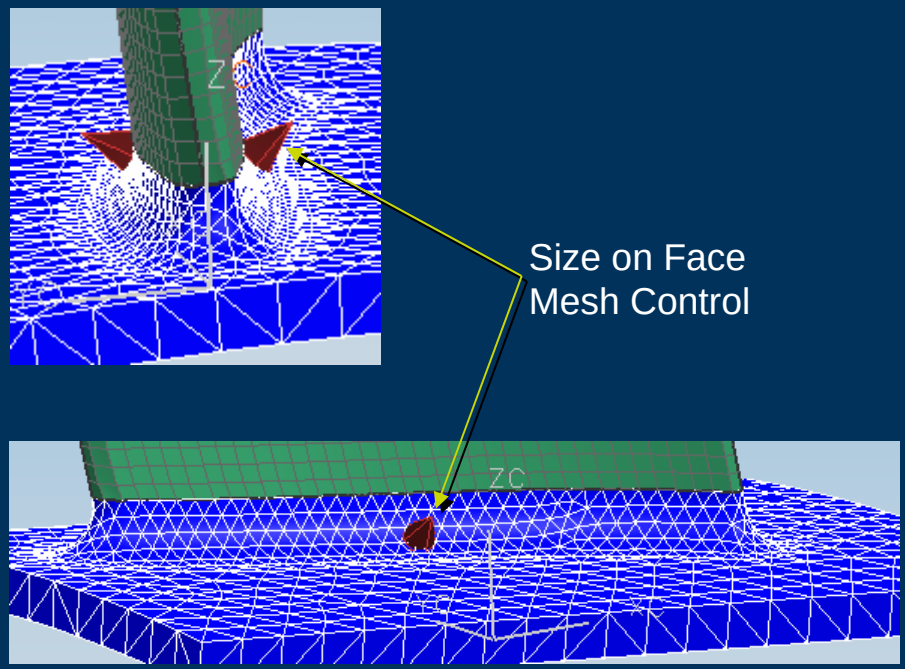




- Mesh Controls
  - Number on an Edge
  - Size on an Edge
  - Chordal Tolerance on an Edge
  - Biasing on an Edge
  - Size on a Face
- Managed by Mesh Controls Collector



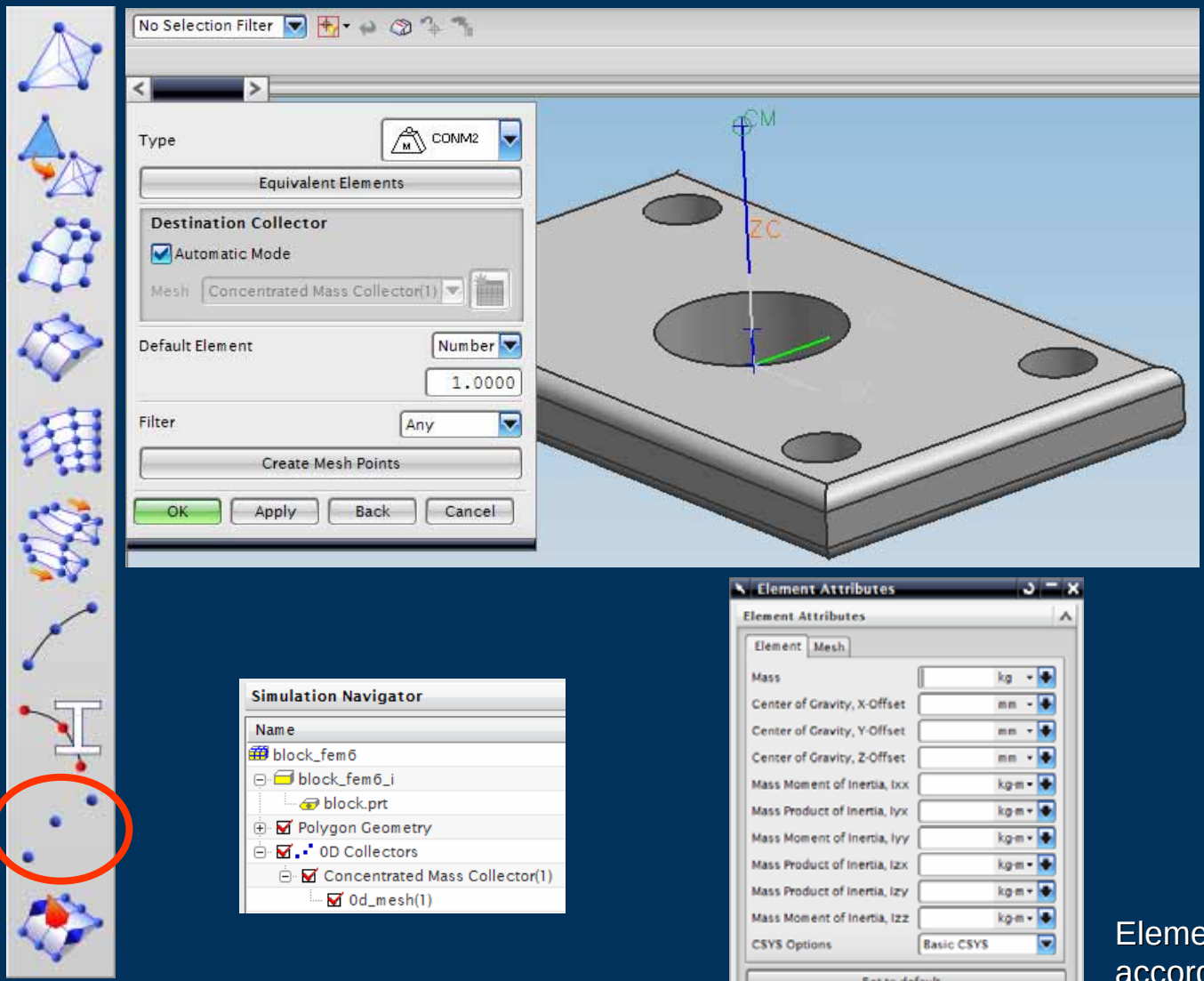


# Mesh Controls




- Used to control Mesh distribution, quality, mating etc
- Meshes associative to Mesh Controls

# Meshing – OD Mesh




- 0D or Scalar Elements for Lumped or Distributed Mass





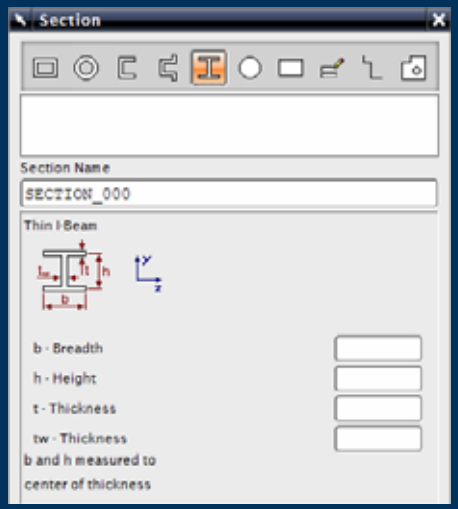


Element Attributes vary according to the Solver Environment & Element Type

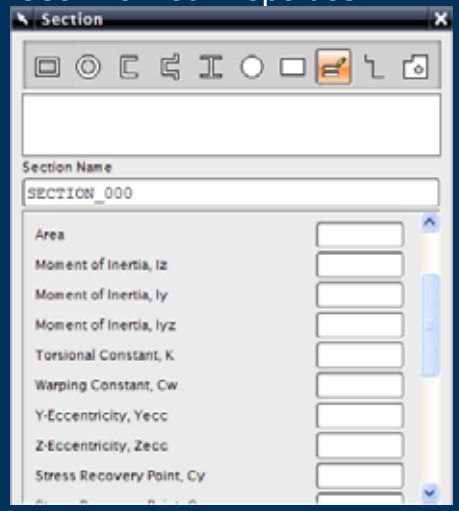
SIEMENS

# Meshing – 1D Element Cross Sections

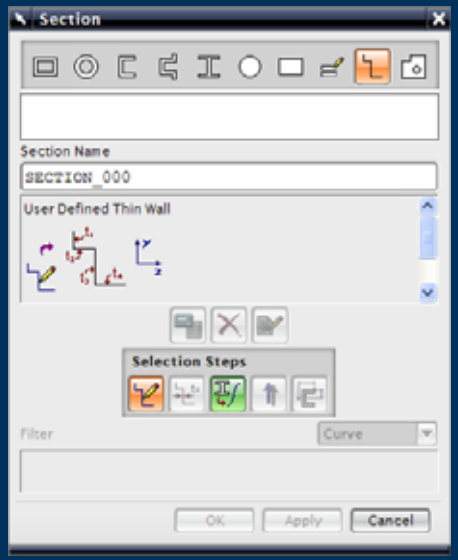
Standard Cross Sections



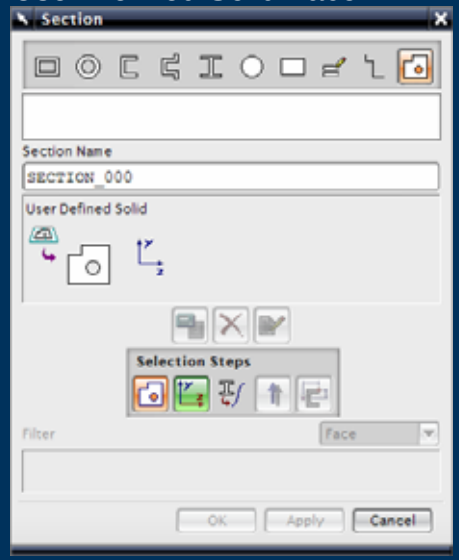

User Defined Properties




User Defined Sketch

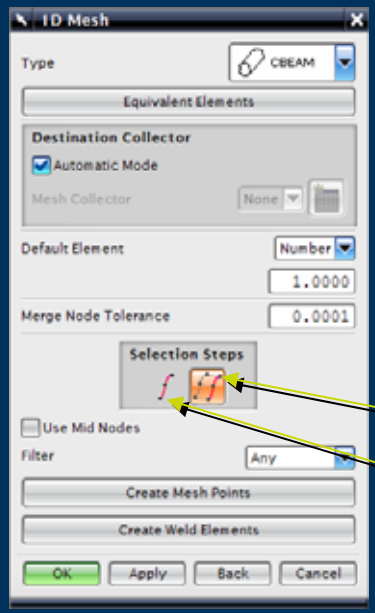


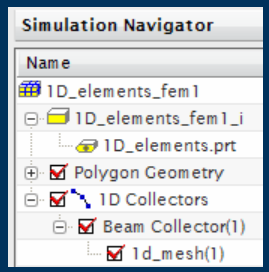

User Defined Solid Face




- Dialog to define and Manage Cross Sections
- 1D Element Attributes will Reference Stored Sections

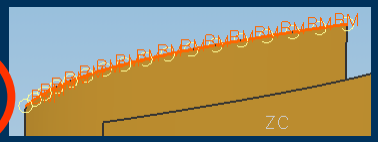
# Meshing – 1D Mesh

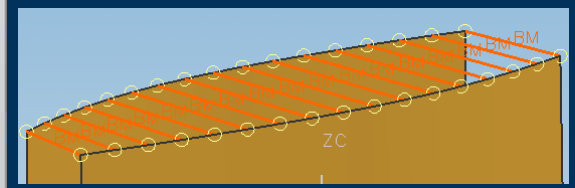


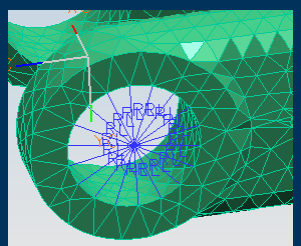




Group 2

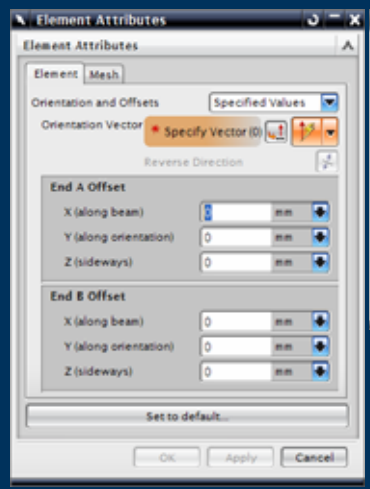
Group 1

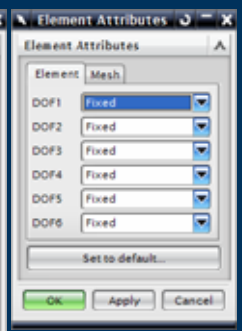


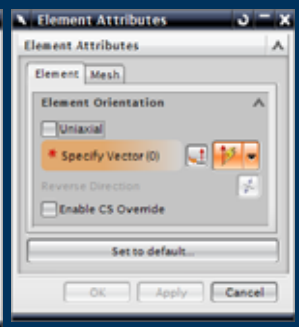


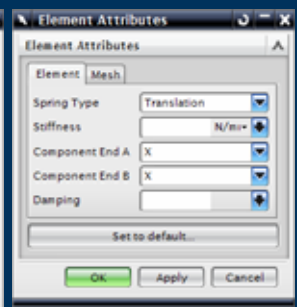


- Multiple options depending on the selection of Group 1 and 2
- Along an edge, around a face, between curves or edges, point to curve/edge etc



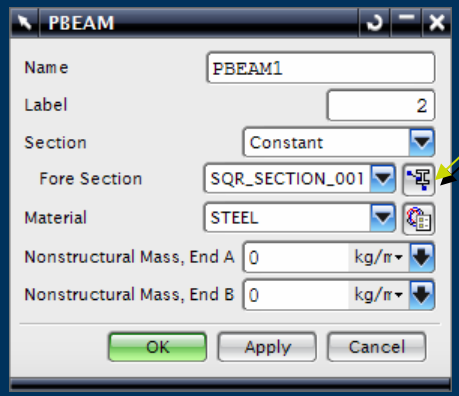



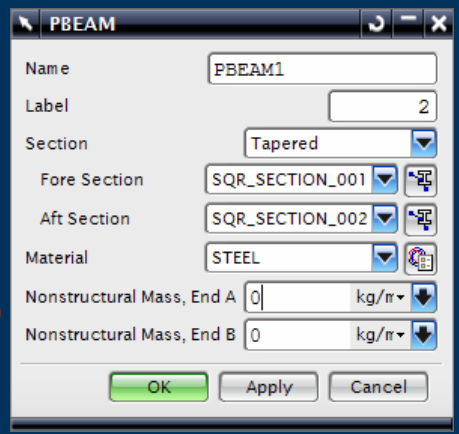



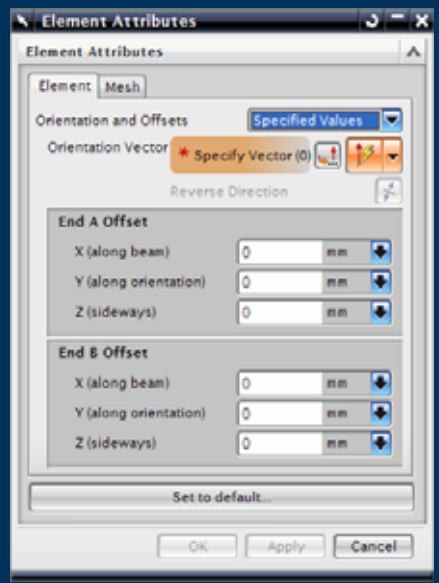

Element Attributes vary according to the Solver Environment & Element Type

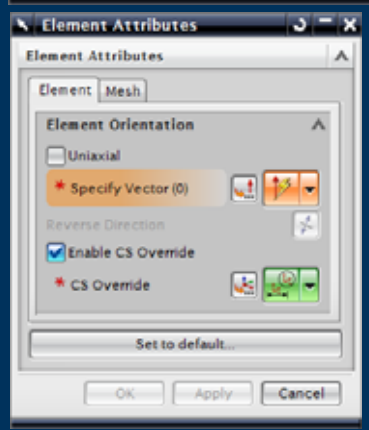
# Meshing – 1D Mesh – Element Attributes




1D Element Cross Sections

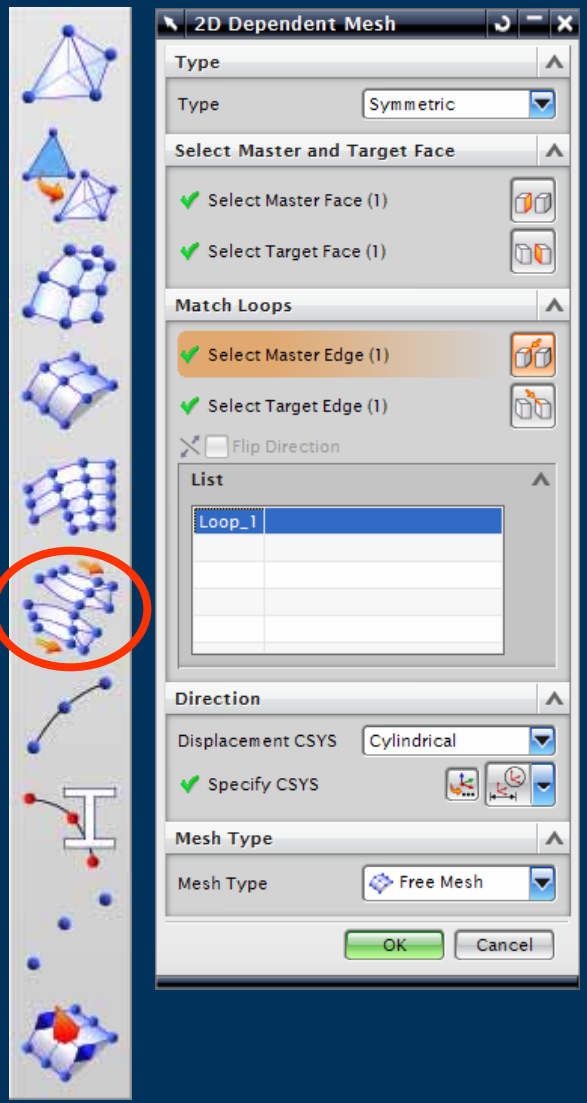


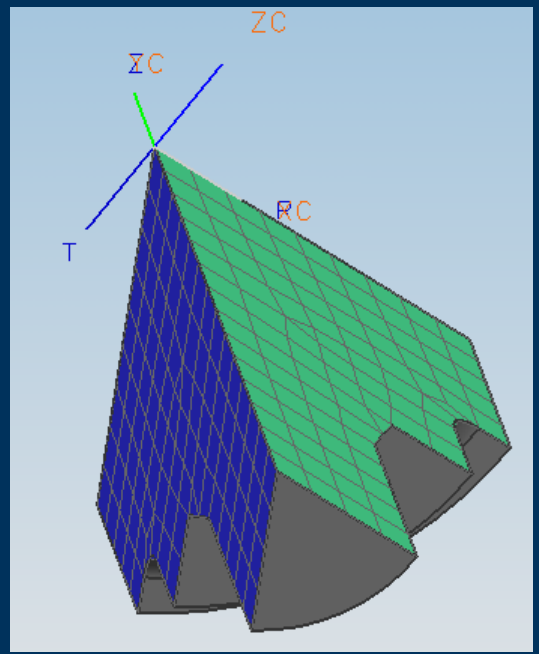




- 1D Elements reference a Physical Property
  - Material
  - Section(s)
- Element Attributes for Beams, Bars and Rods
  - Beams & Bars require Orientation vector
    - Inherited from Geometry
    - Specific Values
  - Bushes require Axis Definition

Element Attributes vary according to the Solver Environment & Element Type

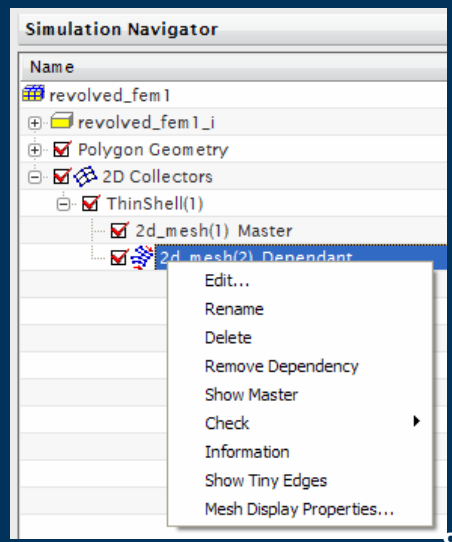
# Meshing – 2D Dependant Mesh



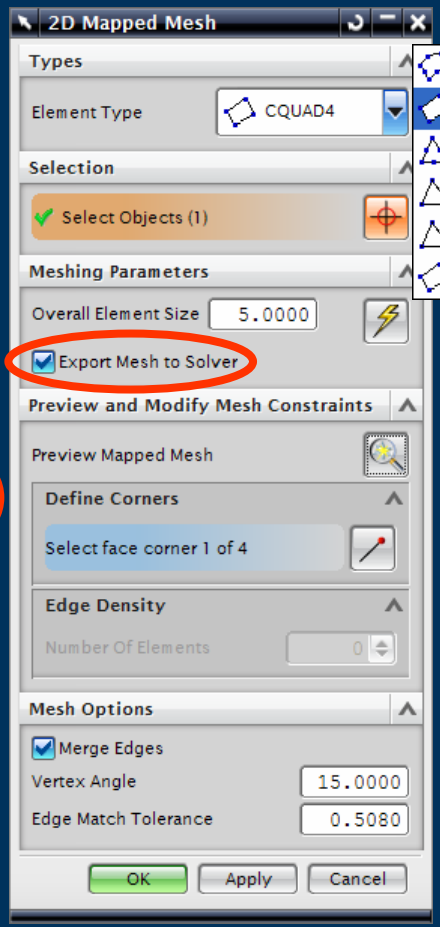



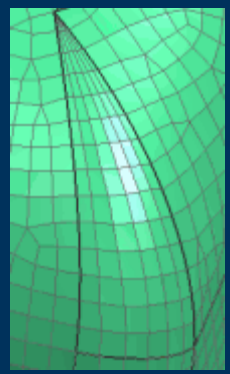

- Master & Target face selection
- Topologically Identical Faces
- Multiple Faces and Loops
- Coordinate System Selection
- Mesh Type Selection
  - Free or Mapped
  - New Mesh or Use Existing Mesh on Master Face
- Managed by 2D Collector

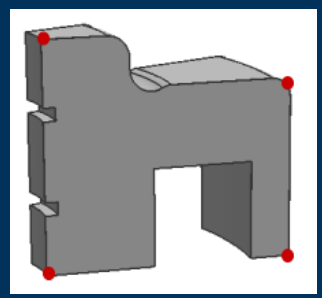
- Uses
  - Contact Regions
  - Flange Mating
  - Symmetric Faces

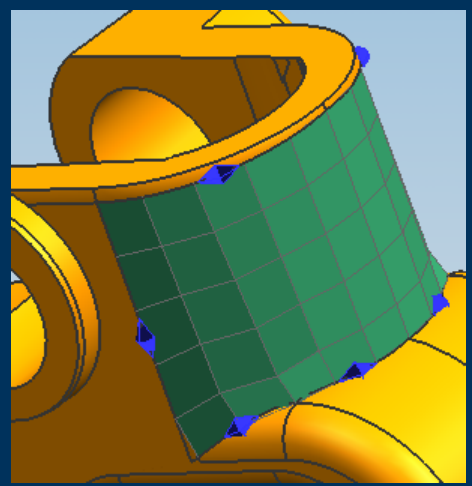


# Meshing – 2D Mapped Mesh

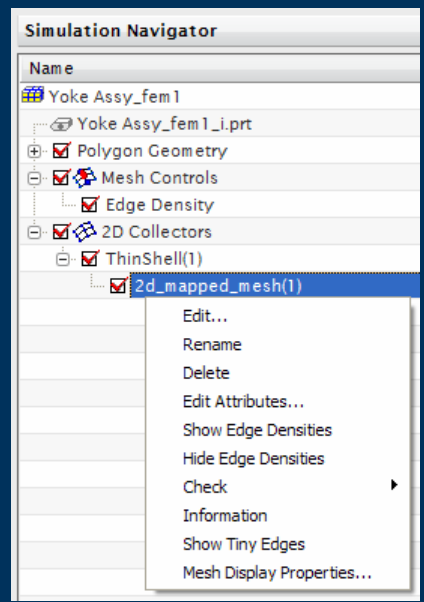




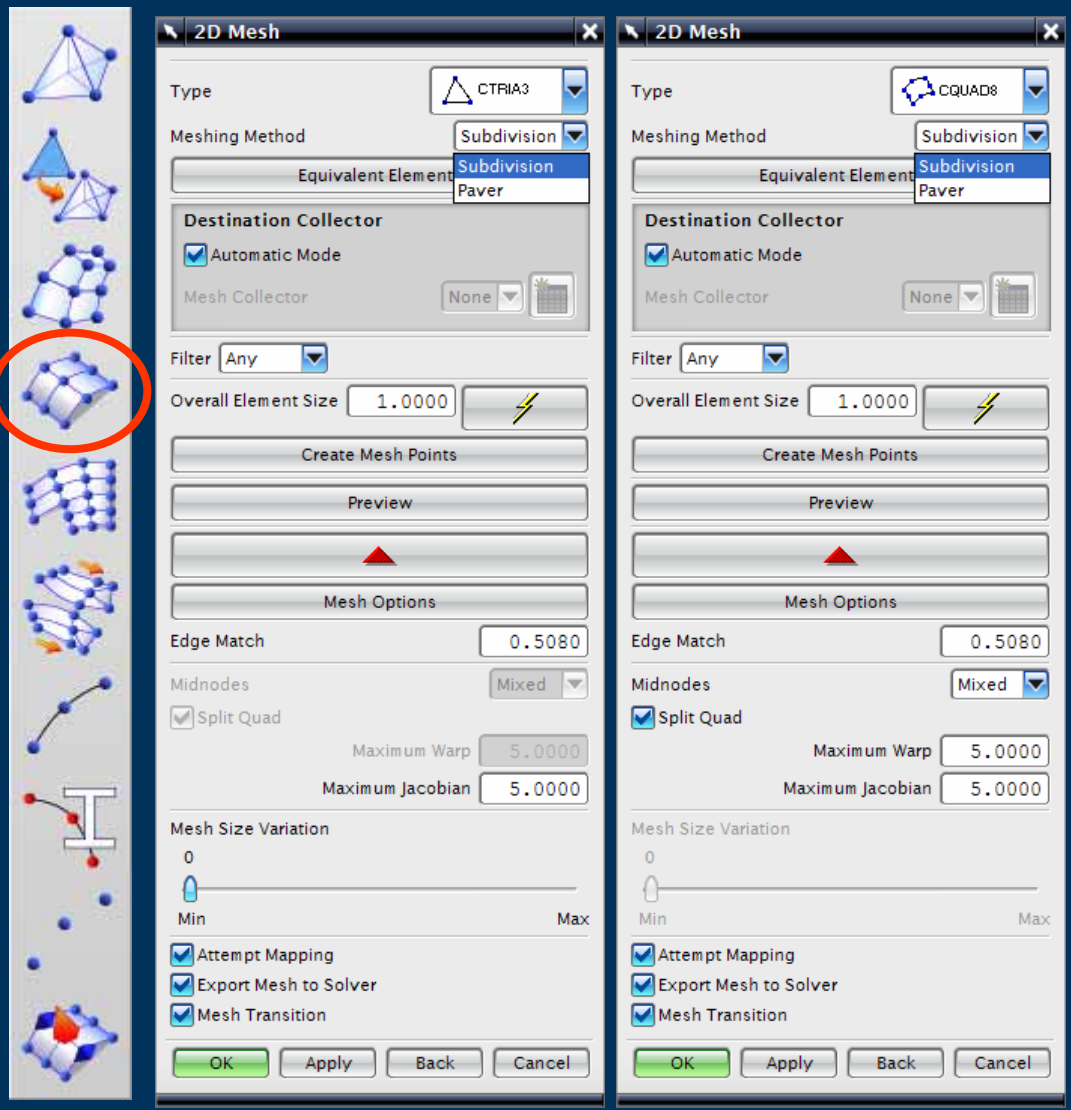


- 3 or 4 Topological Sided Face (Single Loop)
- Mesh Controls automatically added if not pre-defined
- Can be used as Seed for 3D Meshing

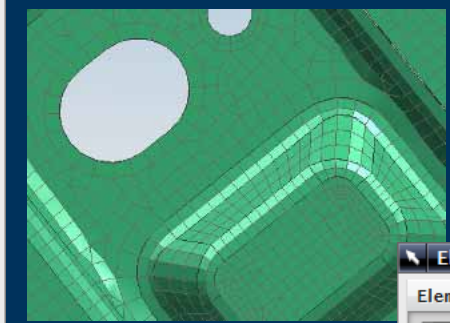


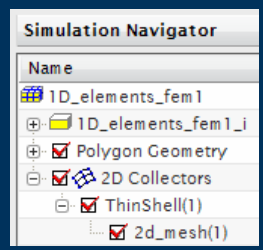

Element Attributes vary according to the Solver Environment & Element Type

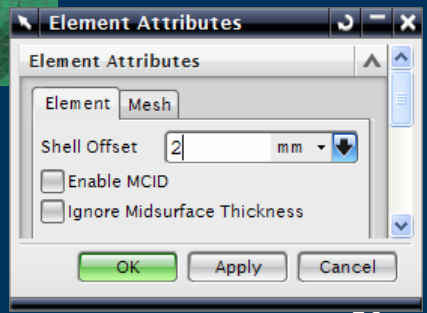
# Meshing – 2D Mesh




- Creation of 2D Shell or Plate elements on selected faces
- Mesh will be also driven by
  - Mesh Points
  - Mating Conditions
  - Contact Definitions
  - Mesh Controls

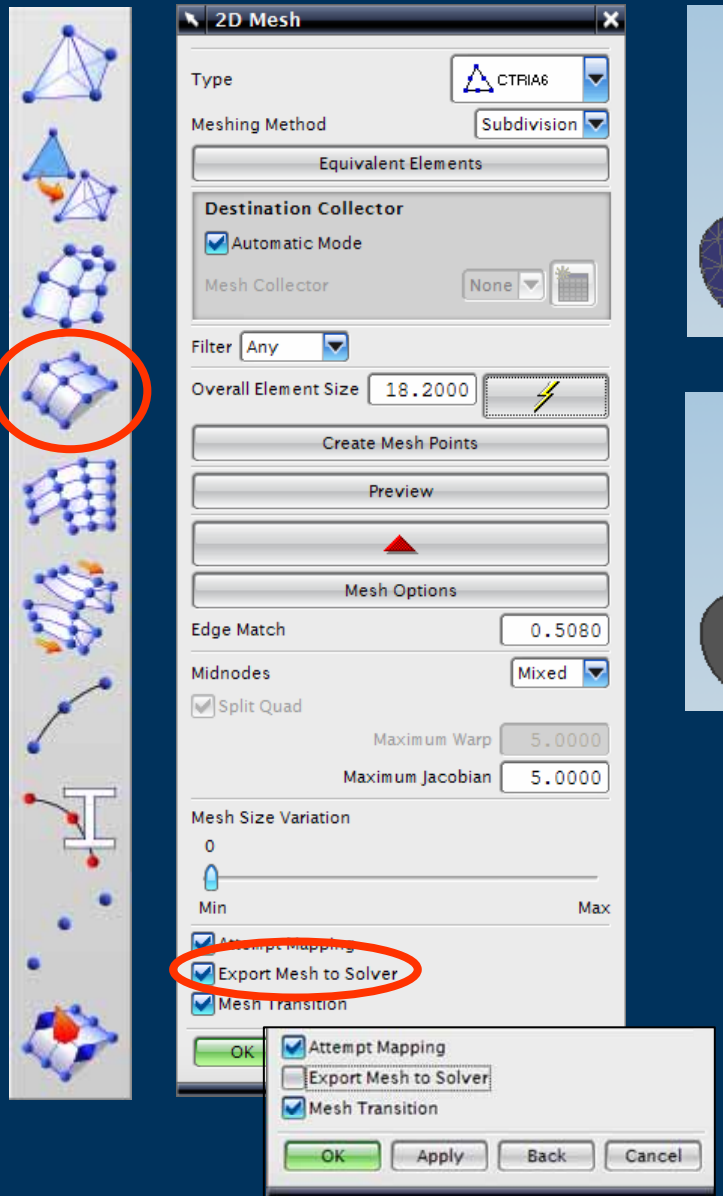


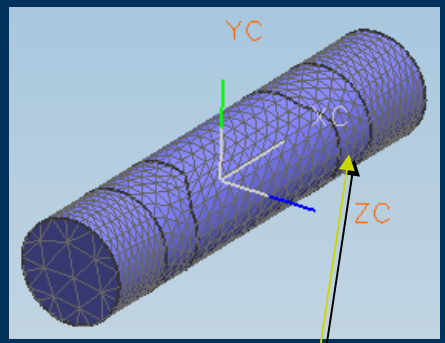


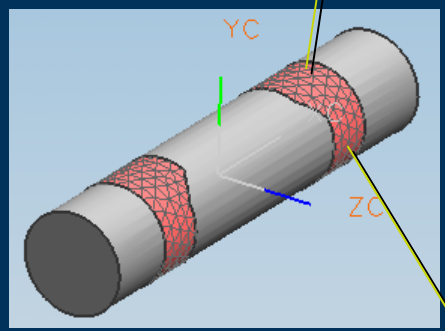



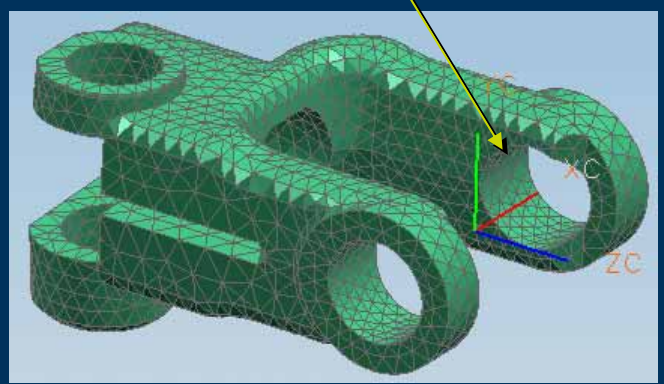
SIEMENS

# Meshing – 2D Mesh Seeding for 3D Mesh



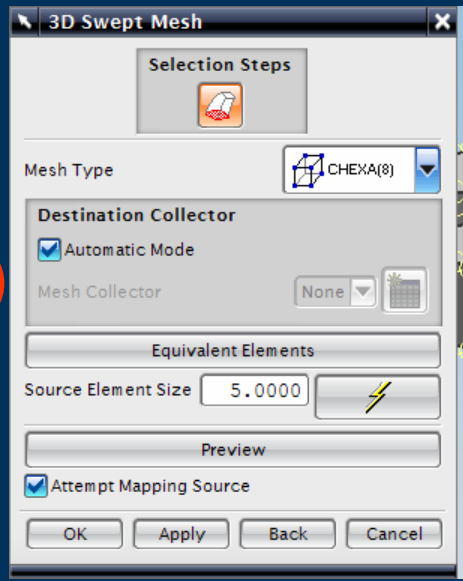



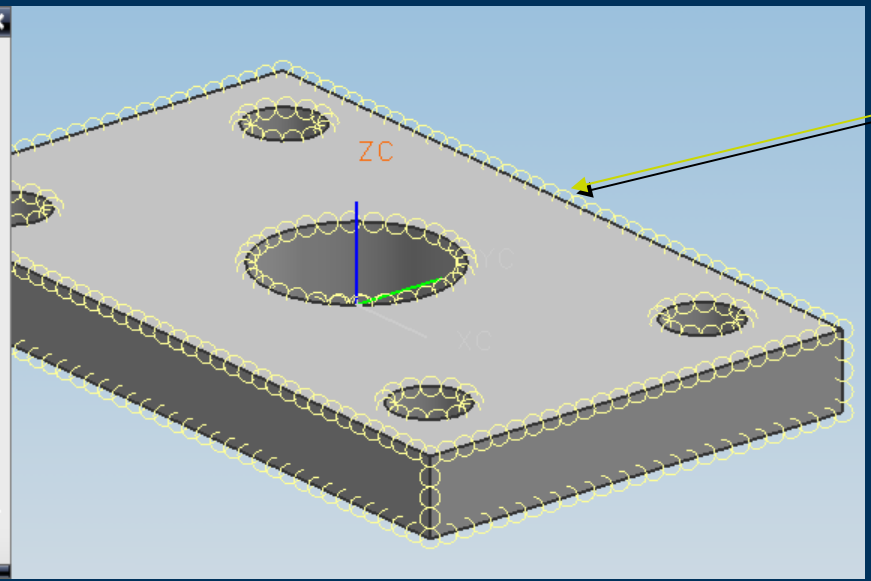




- 2D Mesh can be used to "Seed" or define a 3D Mesh
- Turn OFF Export Mesh to Solver
  - 2D Mesh does NOT get written to the solver
  - Also does not appear in the SIM file

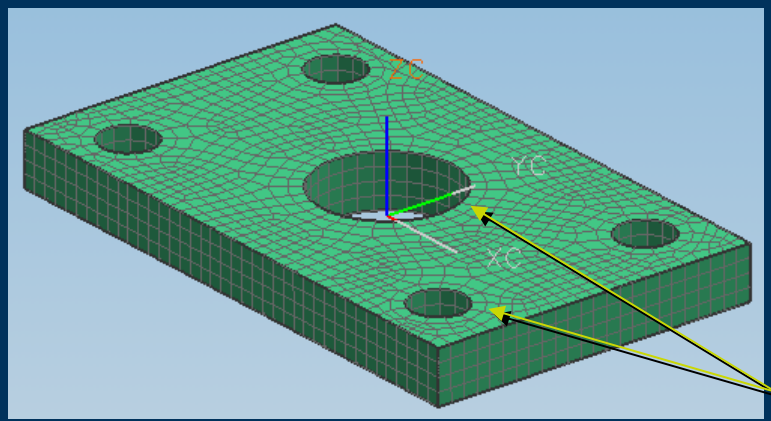
SIEMENS

# Meshing – 3D Swept Mesh



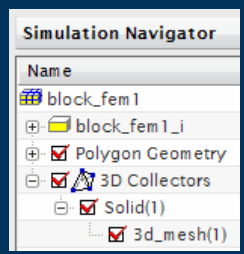




Preview of Mesh distribution along edges

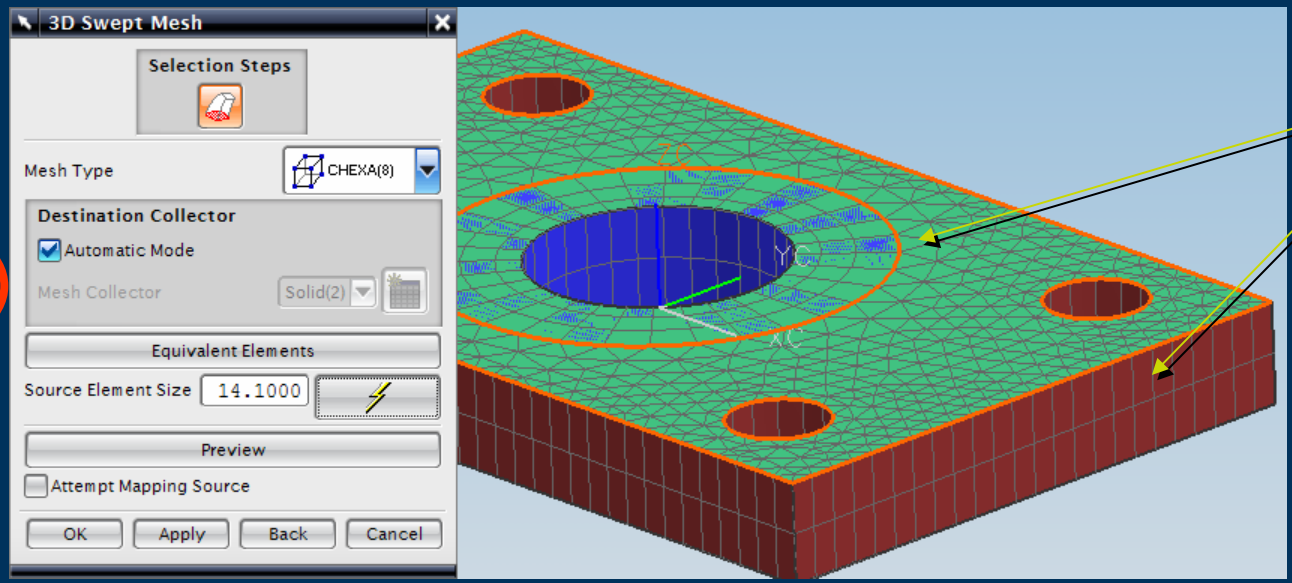



Notice Mapped meshing around holes

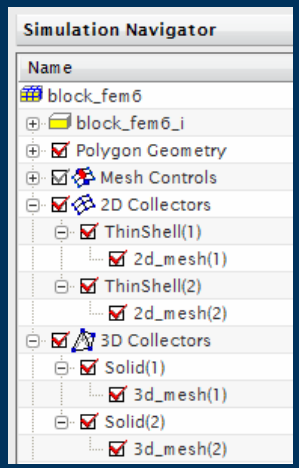
- 3D Swept Mesh requires source face selection
- This face is meshed and swept through the volume



# Meshing – 3D Swept Mesh



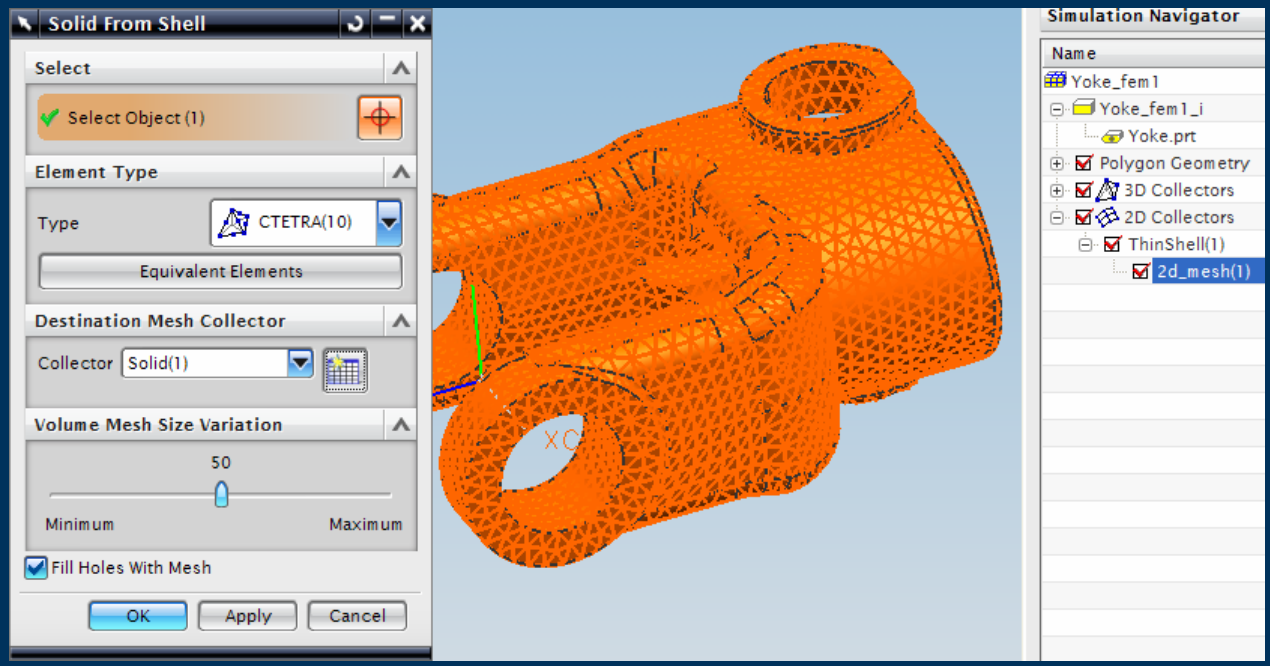

Green – 2D Seed Meshes to define 3D Swept Mesh
Brown & Blue – 3D Swept Meshes



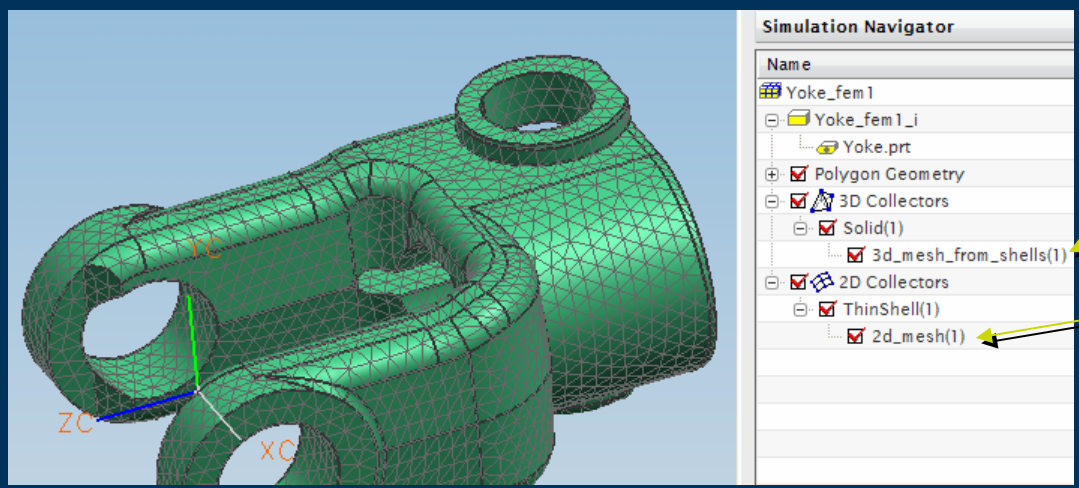

- 2D Mesh the Seed faces to control mesh type and distribution
- 2D Mesh – Turn OFF “Export Mesh to Solver” then this mesh is not written to the solver input deck
- 3D Swept Mesh starts with these Seed meshes

SIEMENS

# Meshing – Solid from Shell Mesh

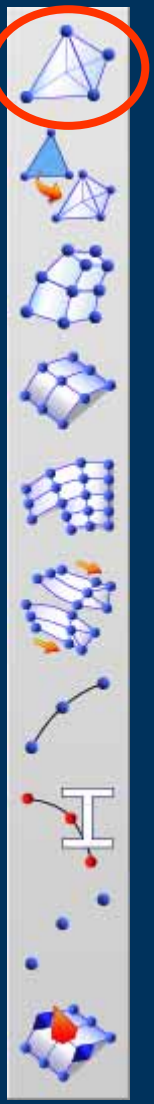


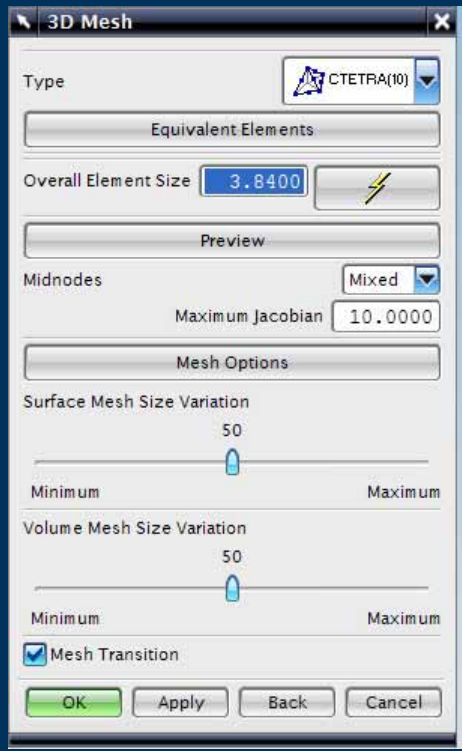

- Generates a 3D Mesh from a closed surface mesh
- Used when the importing CAD geometry is not complete and CAE topology is used to close the volume

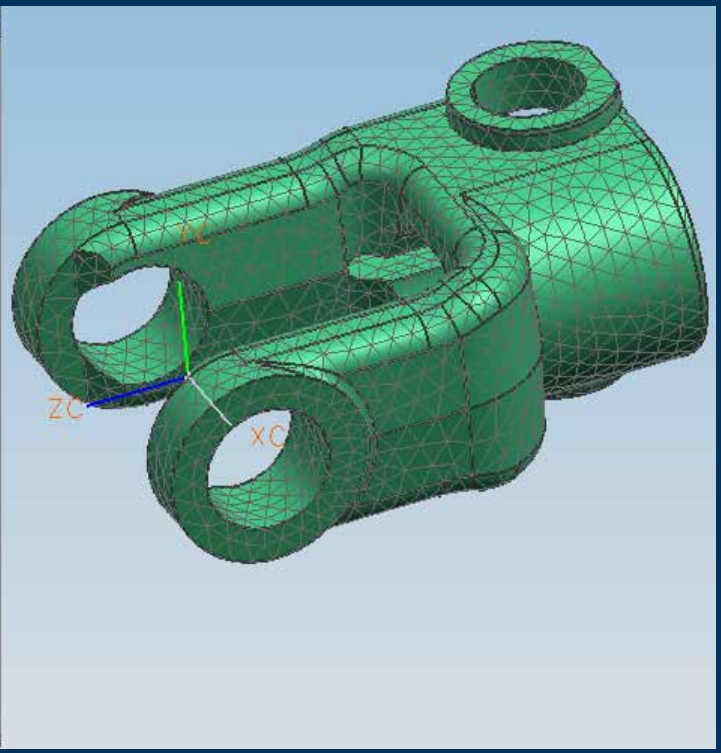


SIEMENS

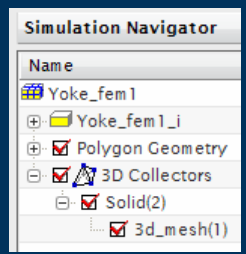
# Meshing – 3D Tetrahedral Mesh





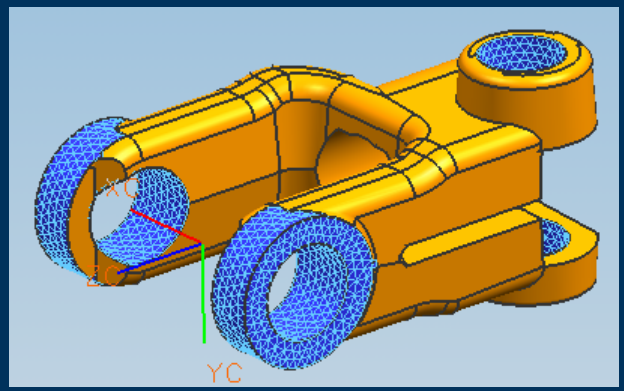

- 3D Tet Mesh
- Auto Element Size sets a good value to start the meshing process
- Same Mesh Options as 2D Mesh

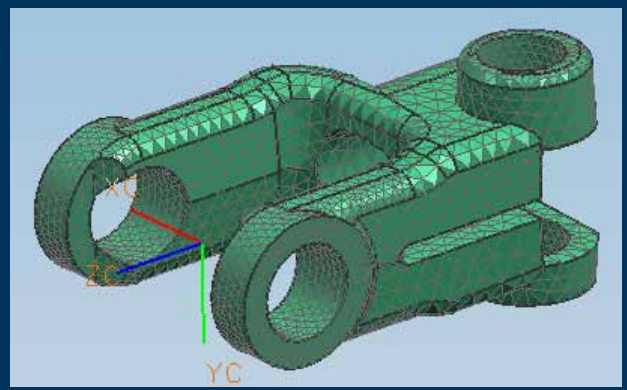



Mesh quality

- CAE Topology Editing
- Mesh Controls
- 2D Surface Seed Meshes
- Volume Mesh Size Variation

# Meshing – 3D Tetrahedral Mesh

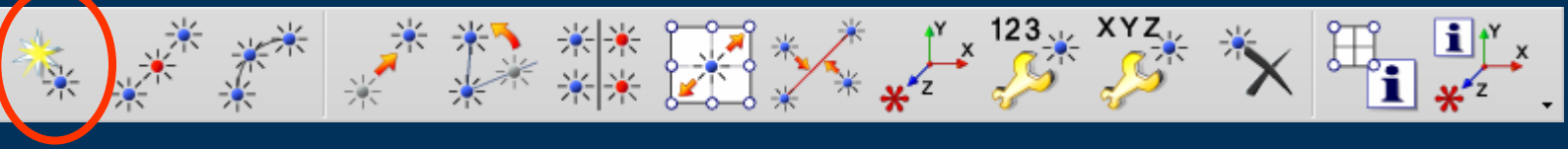


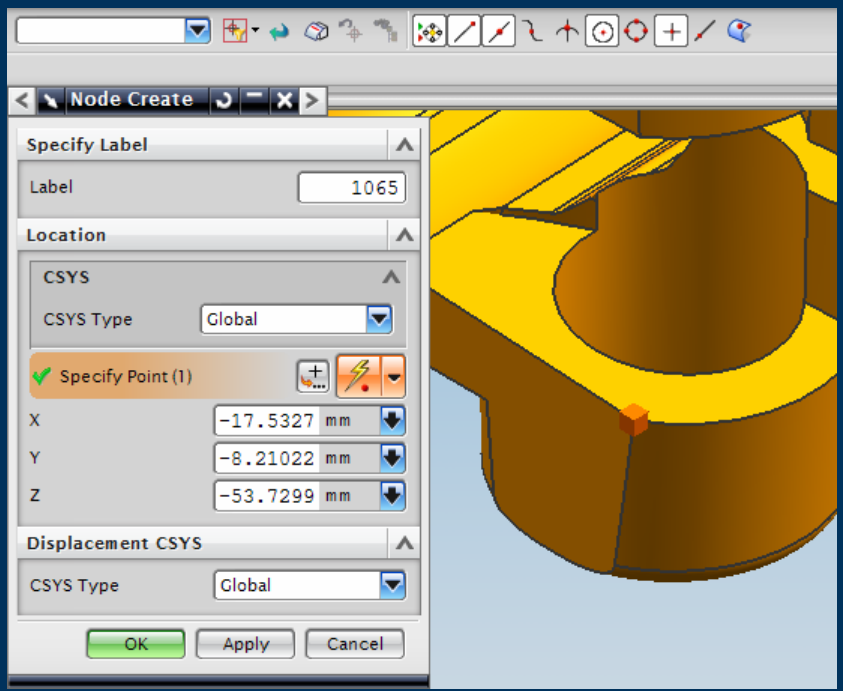




- Good technique is to “seed” the mesh by applying 2D Mesh to selected faces
- Add refinement and detail control where required

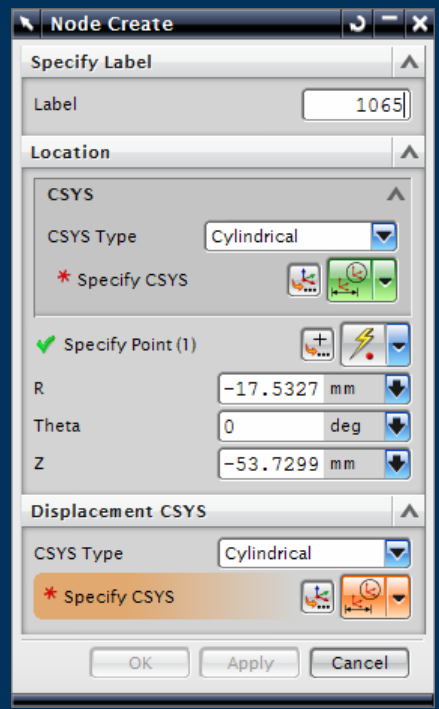
SIEMENS

# Meshing – Node Create



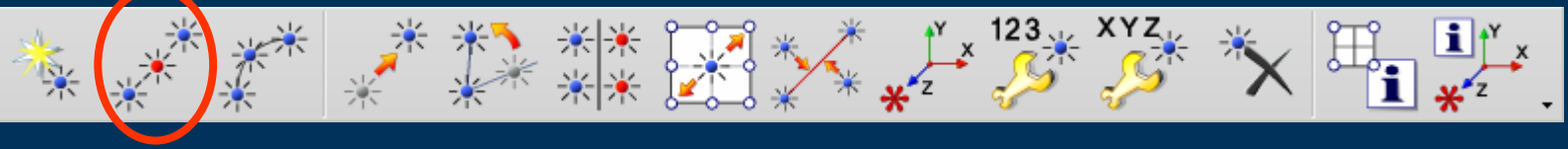



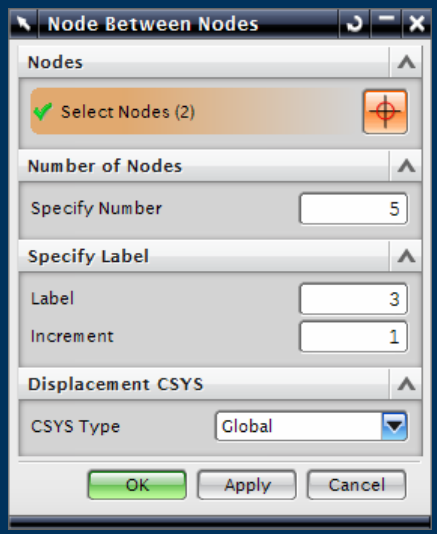

- Location & Displacement by Global or Selected CSYS

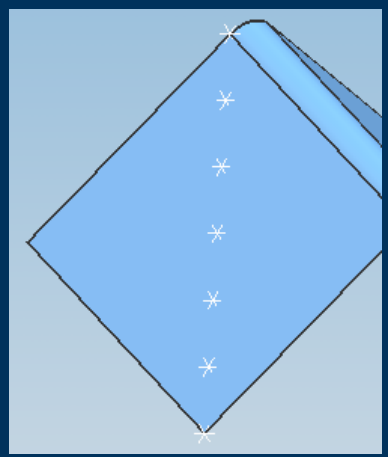


SIEMENS

# Meshing – Node Between Nodes



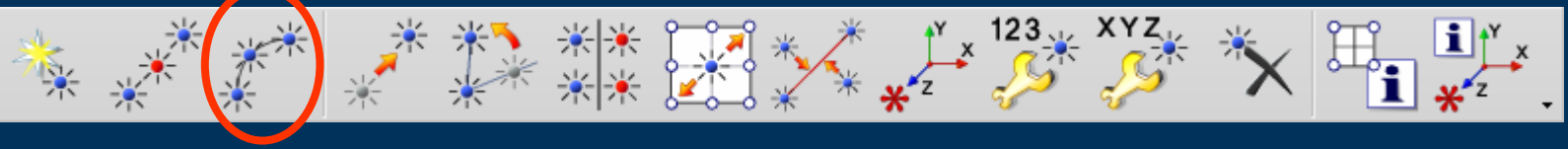


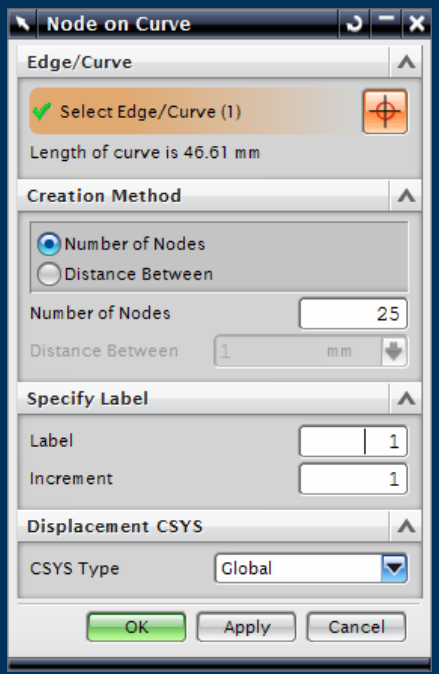



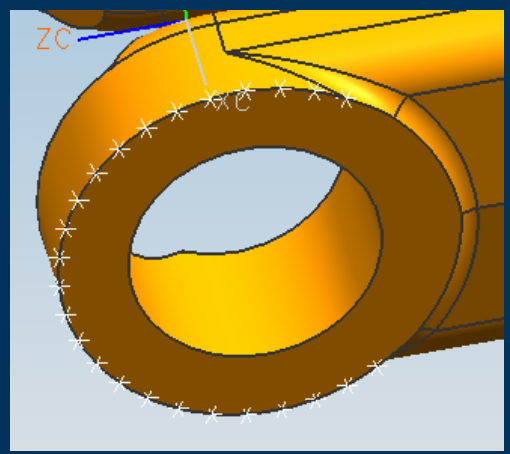
- Place Nodes equidistant between 2 selected Nodes
- Geometry independent ie does not track surface(s) between Nodes
- Displacement by Global or Selected CSYS

SIEMENS

# Meshing – Node on Curve/Edge



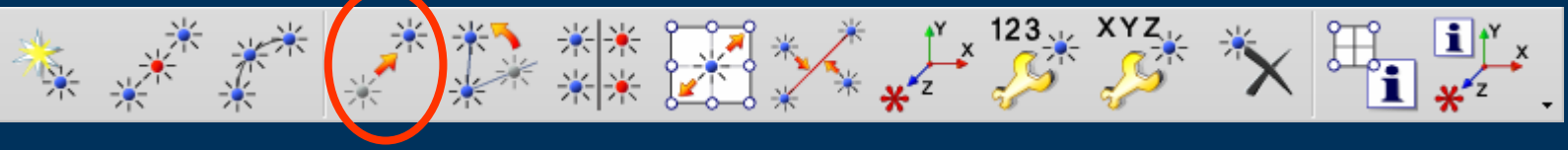


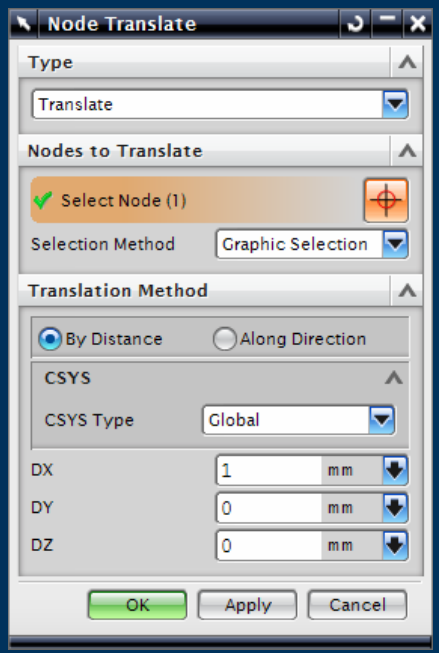



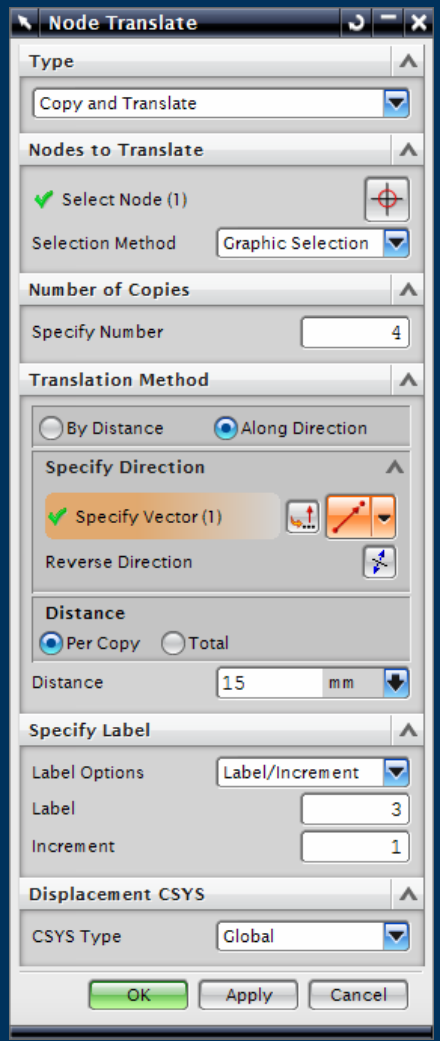

- Place Nodes equidistant along a selected Edge/Curve
- Number of Nodes or Distance between Nodes
- Displacement by Global or Selected CSYS

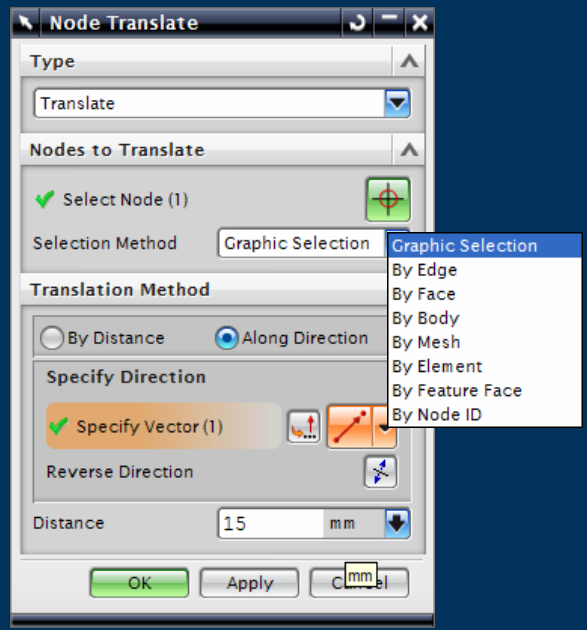
SIEMENS

# Meshing – Node Translate





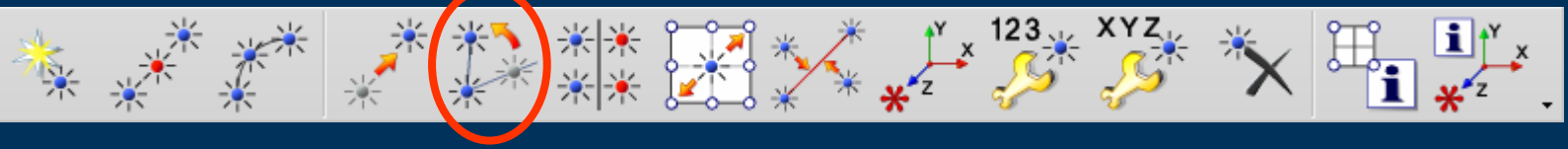



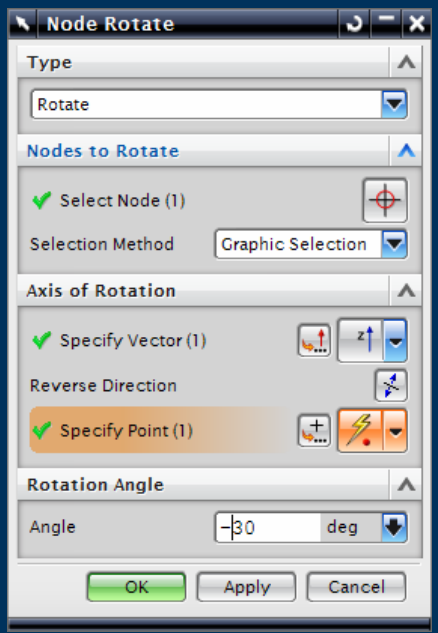



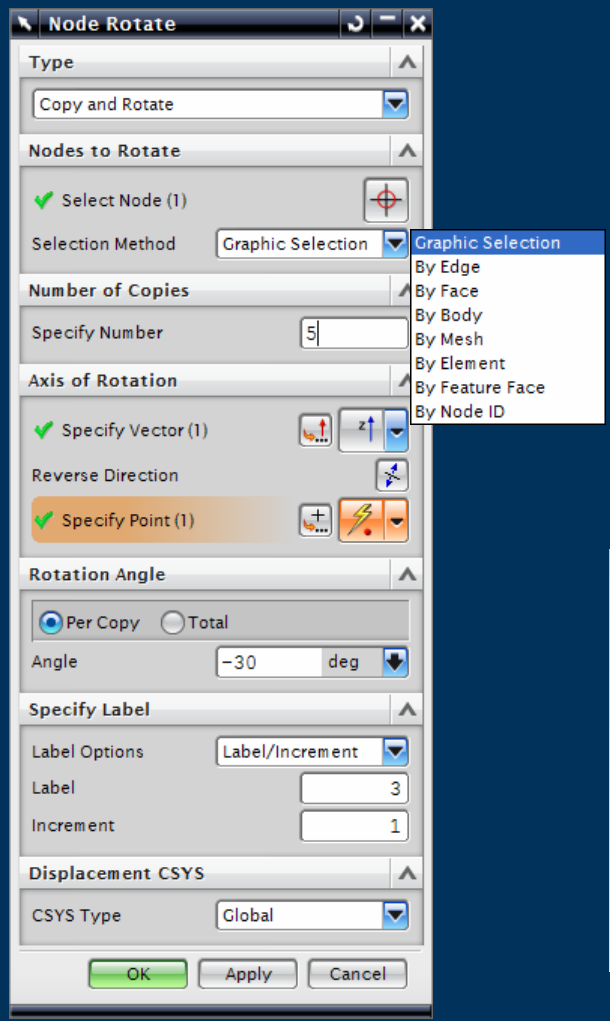

- Node Translate/Copy
- Multiple Selection Methods
- Displacement by Global or Selected CSYS

SIEMENS

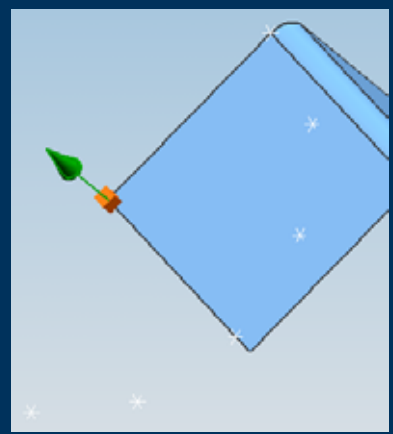
# Meshing – Node Rotate





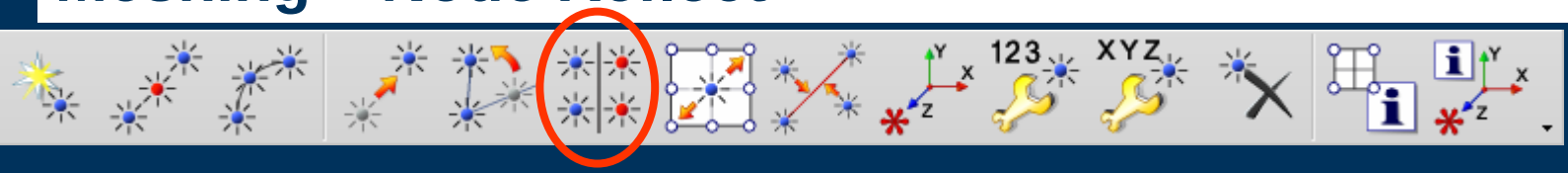



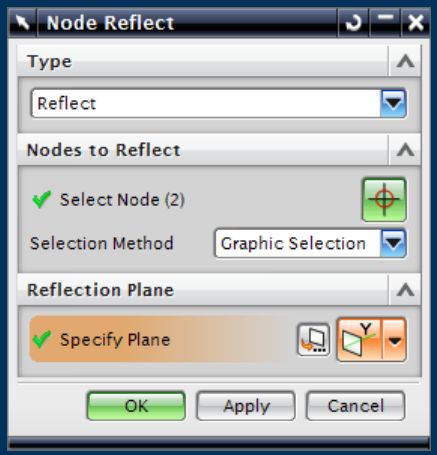

- Node Rotate/Copy
- Multiple Selection Methods
- Displacement by Global or Selected CSYS

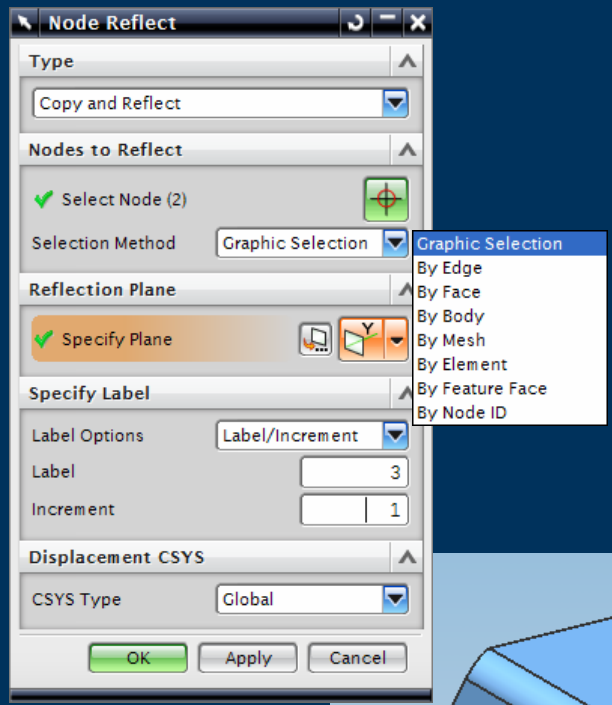


SIEMENS

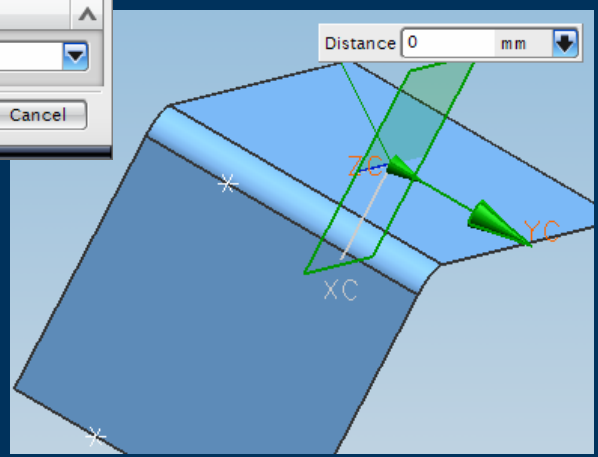
# Meshing – Node Reflect

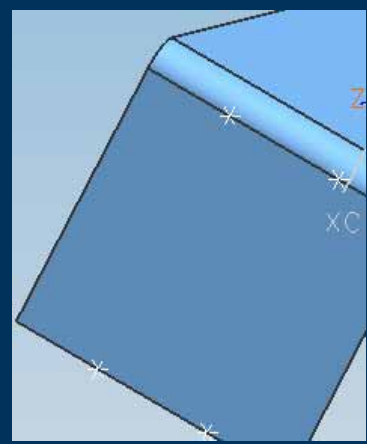




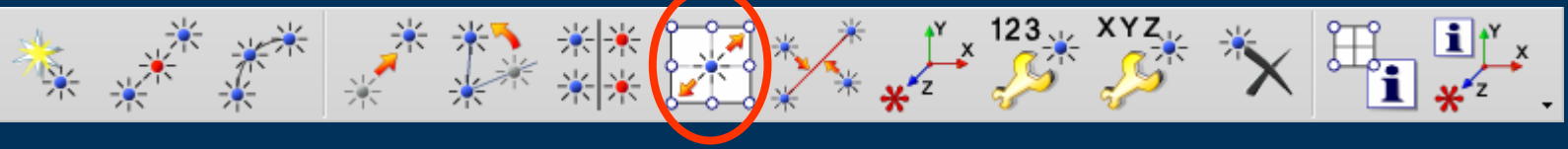

- Node Reflect/Copy
- Multiple Selection Methods
- Displacement by Global or Selected CSYS

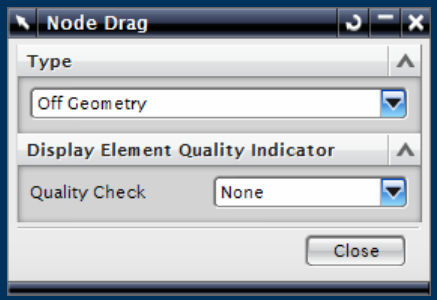


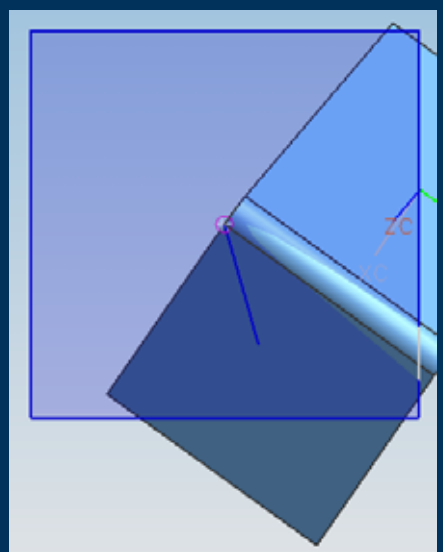
SIEMENS

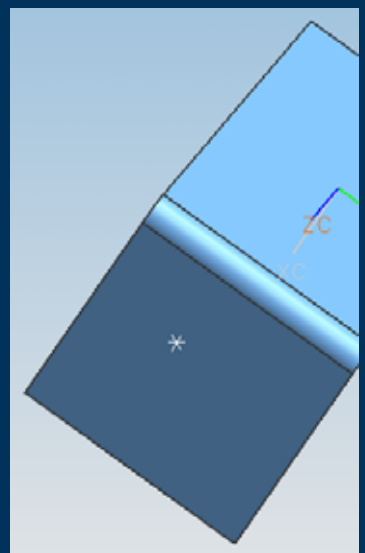
# Meshing – Node Drag

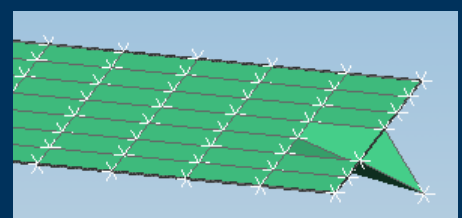



- Node Dragging Off Geometry
  - Drags in a plane parallel to screen through start node location

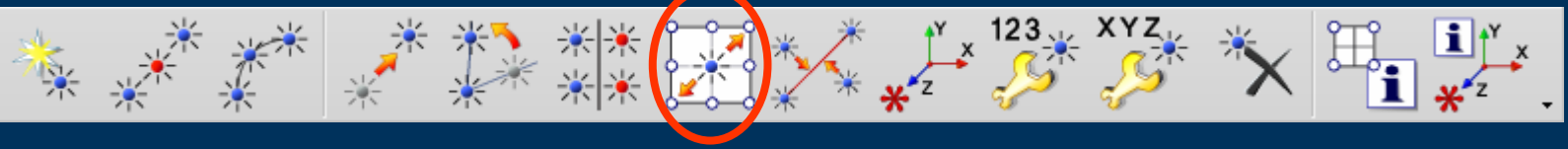


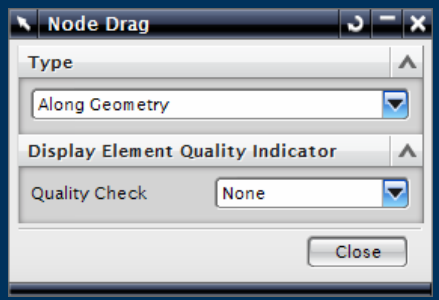


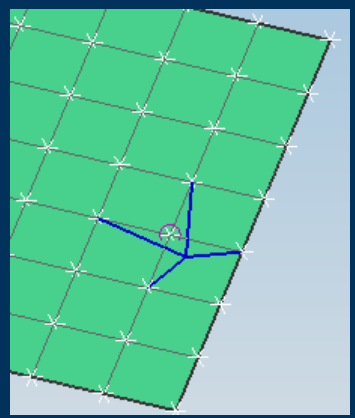


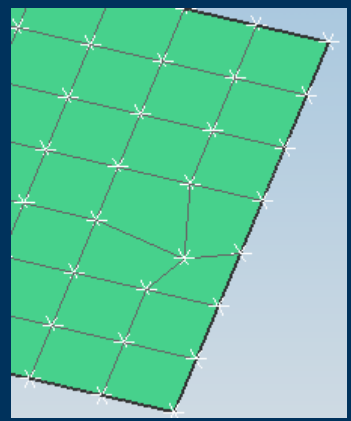
SIEMENS

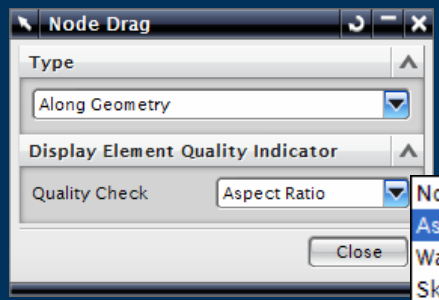
# Meshing – Node Drag

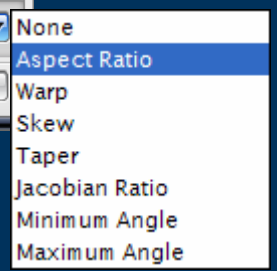


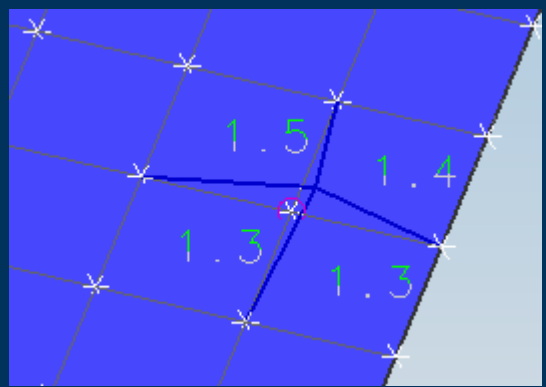






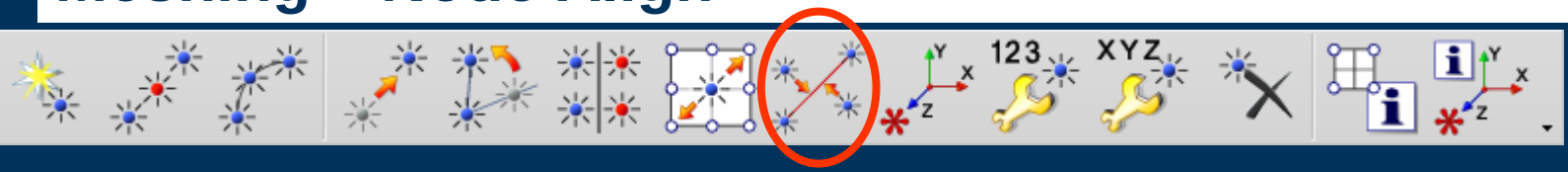



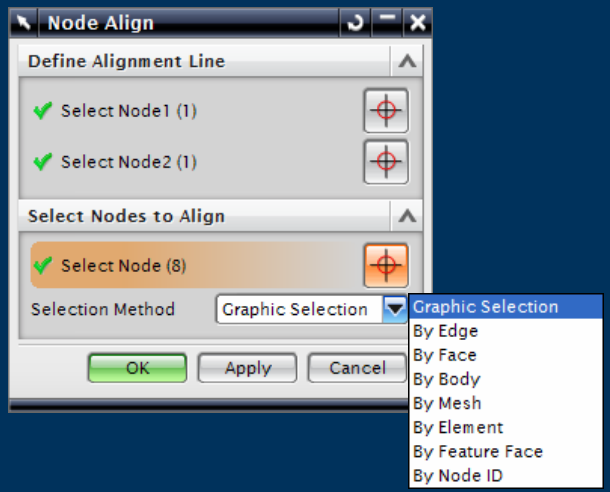




- Node Dragging On Geometry
- Drags on associated geometry
  - Edge
  - Face
- Dynamic Display of Element Quality Check

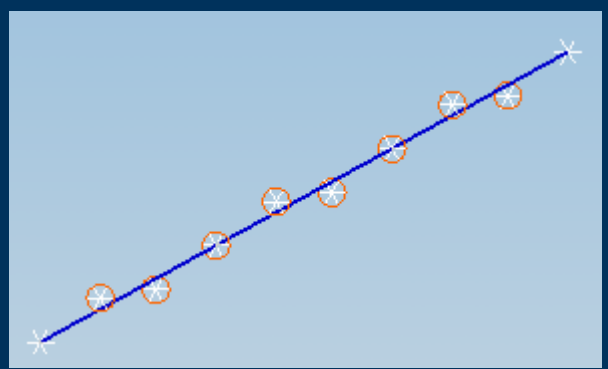
SIEMENS

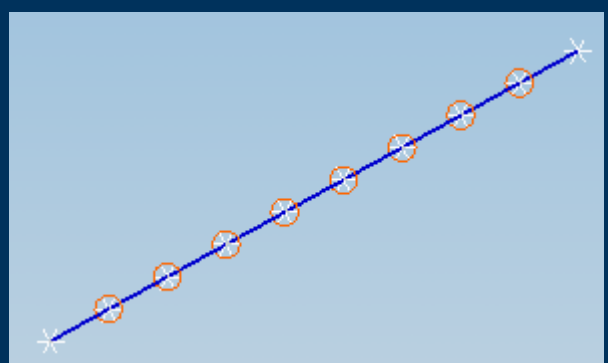
# Meshing – Node Align





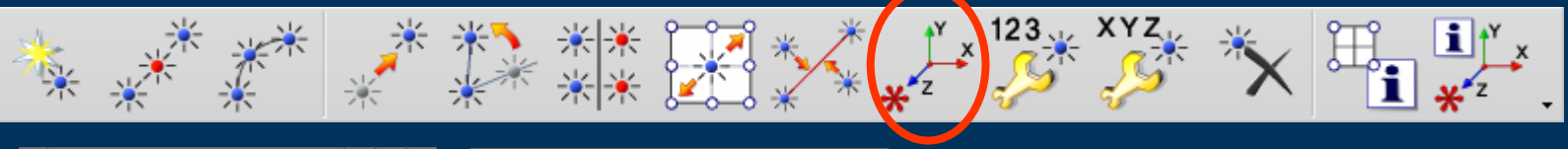

- Move selected Nodes onto Vector between 2 Nodes
- Multiple Selection Methods

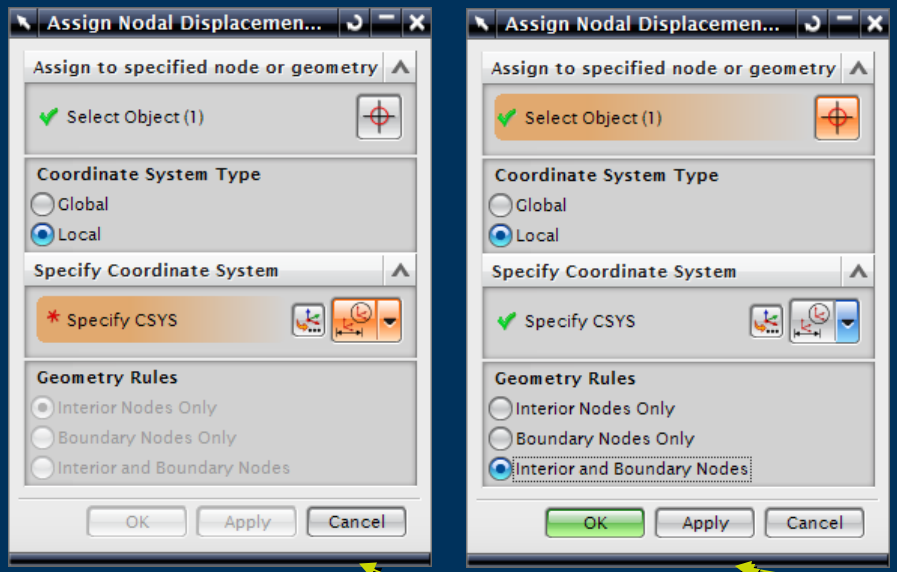




SIEMENS

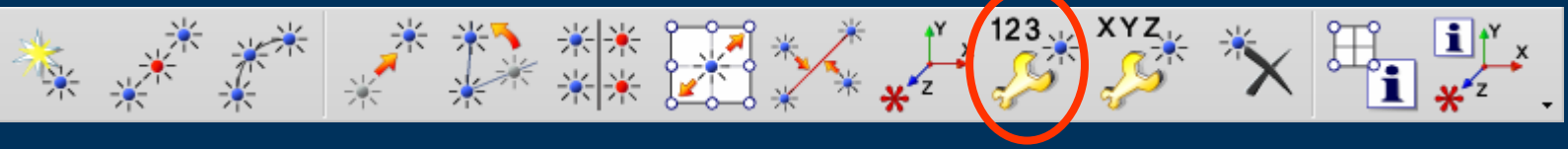
# Meshing – Node Displacement CSYS

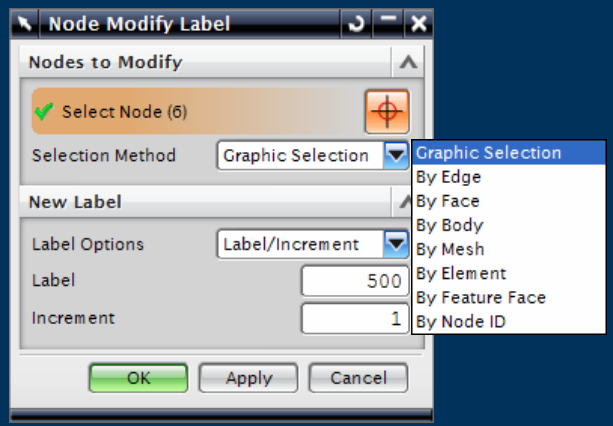




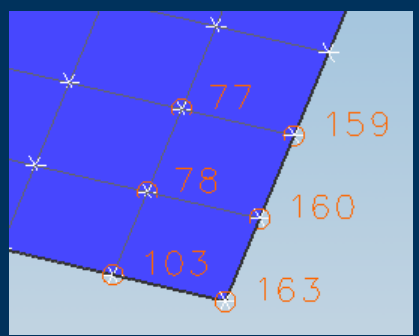

- Assign Nodal Displacement Coordinate System
  - Cartesian
  - Cylindrical
  - Spherical
  - Pre-Existing or Created on-the-fly
- Select Nodes by
  - Edge
  - Face
  - Body
  - Individual Selection

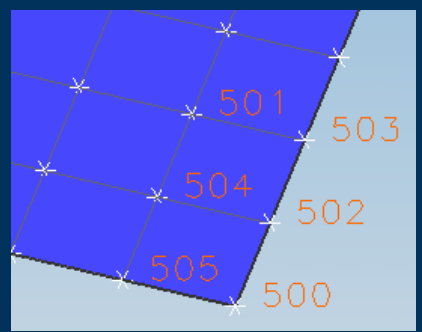
# Meshing – Node Re-Numbering



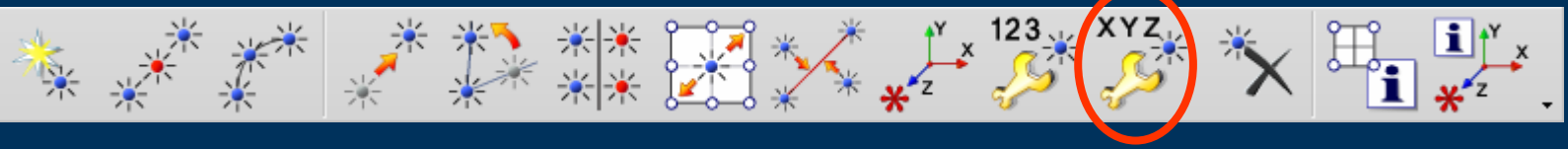

- Modify Node Numbering/Label

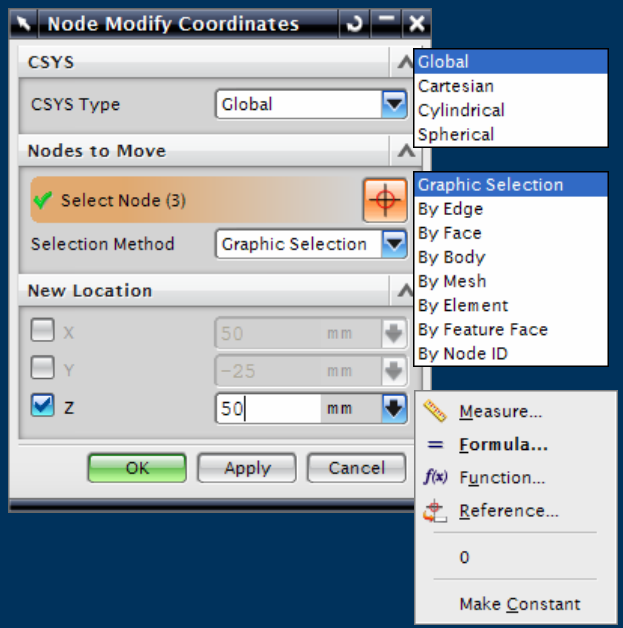


SIEMENS

# Meshing – Node Modify Coordinates

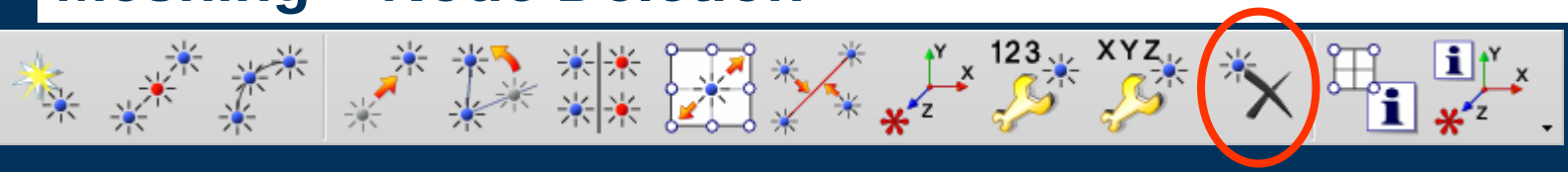


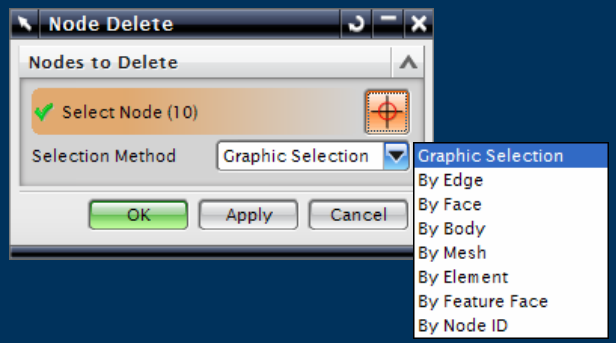




- Modify the Coordinate(s) of selected Nodes
- Global or Selected CSYS
  - X, Y, Z
  - R, Theta, Z
  - R, Theta, Phi

SIEMENS

# Meshing – Node Deletion

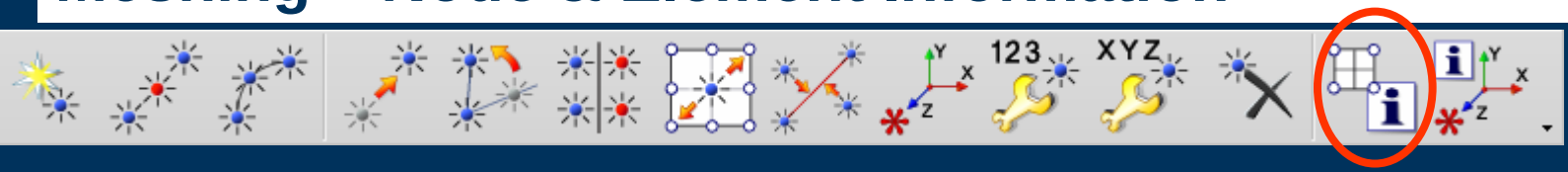


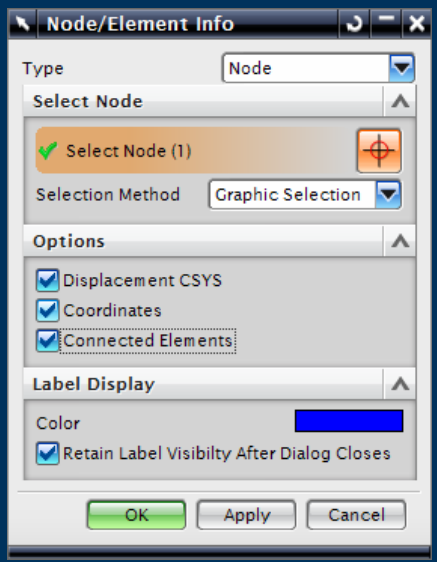



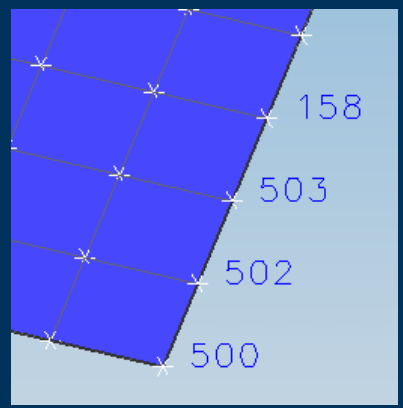

- Delete Nodes
- Only Nodes not attached to Elements will be deleted

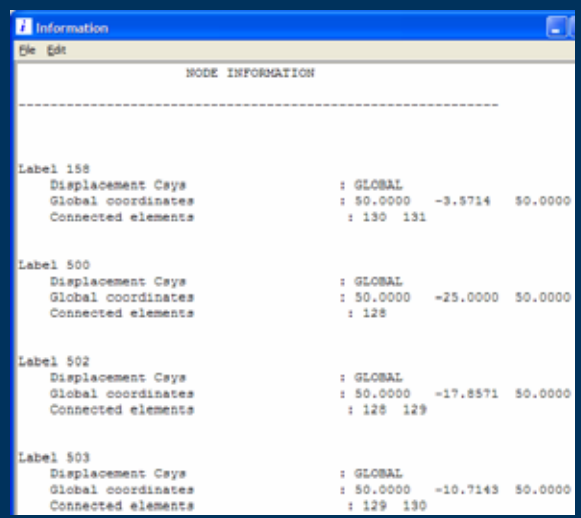
SIEMENS

# Meshing – Node & Element Information



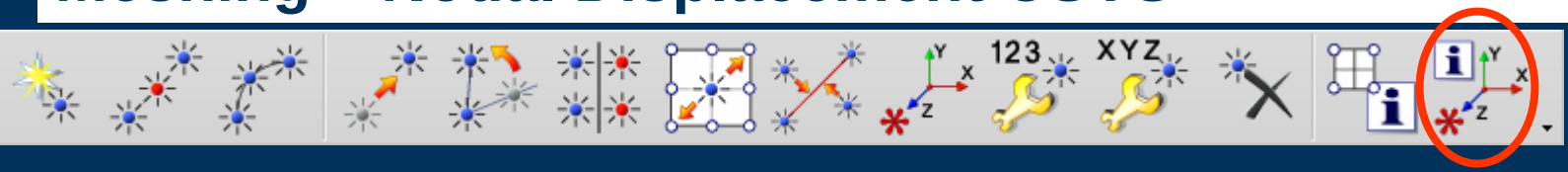



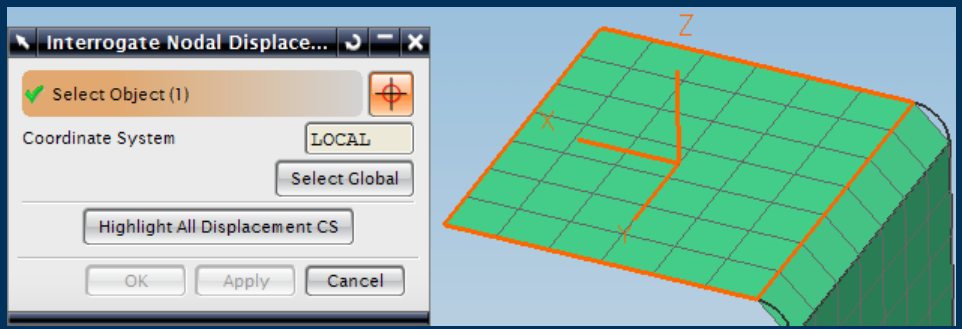




- Node Information
  - Displacement CSYS
  - Coordinates
  - Connected Elements

SIEMENS

# Meshing – Nodal Displacement CSYS

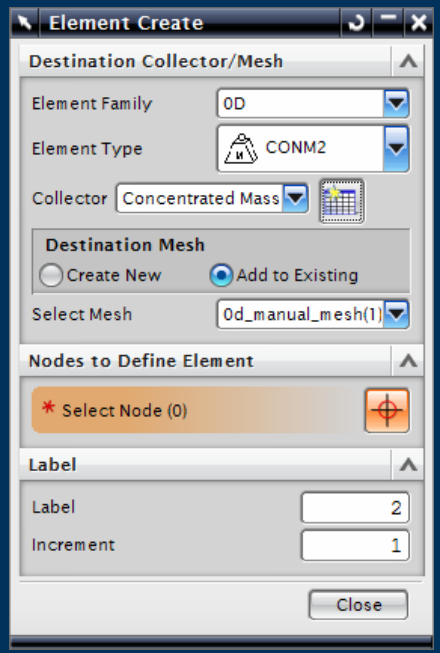


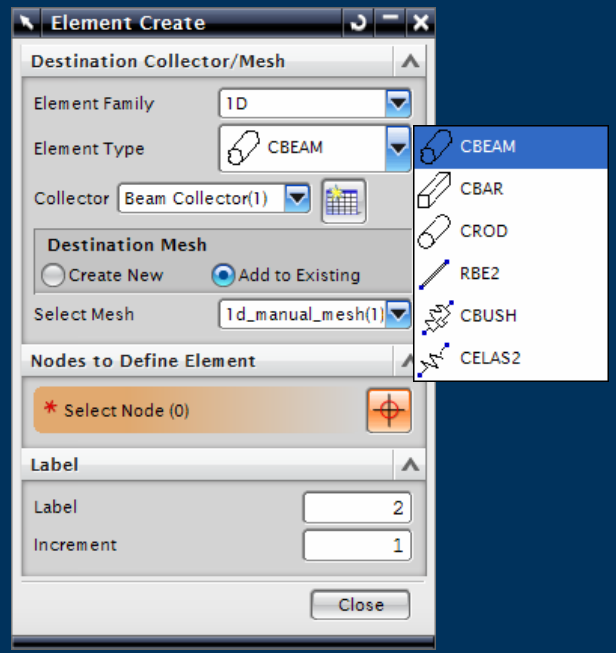




- Display Assigned Displacement Coordinate System for selected Nodes
- Display Related Nodes or Geometry to a Displacement Coordinate System

SIEMENS

# Meshing – Element Create

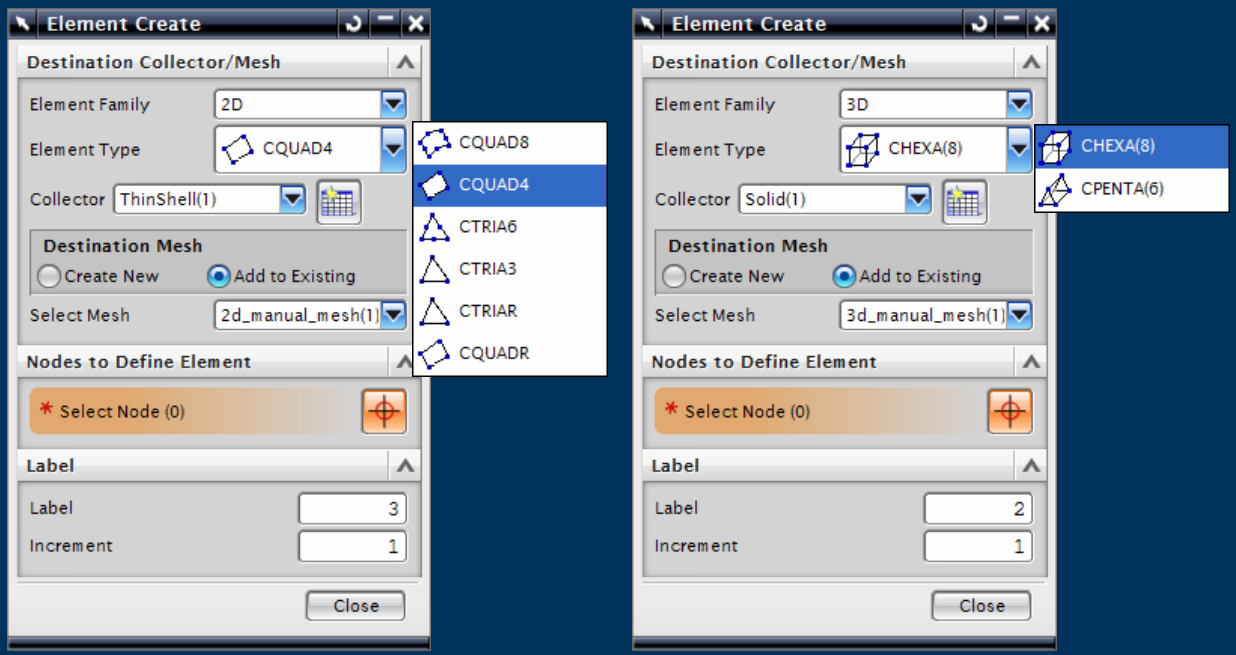



- Element Creation attached to existing Nodes
- Mesh Collector selection or Creation on-the-fly
  - New Mesh
  - Add to Existing Mesh

SIEMENS

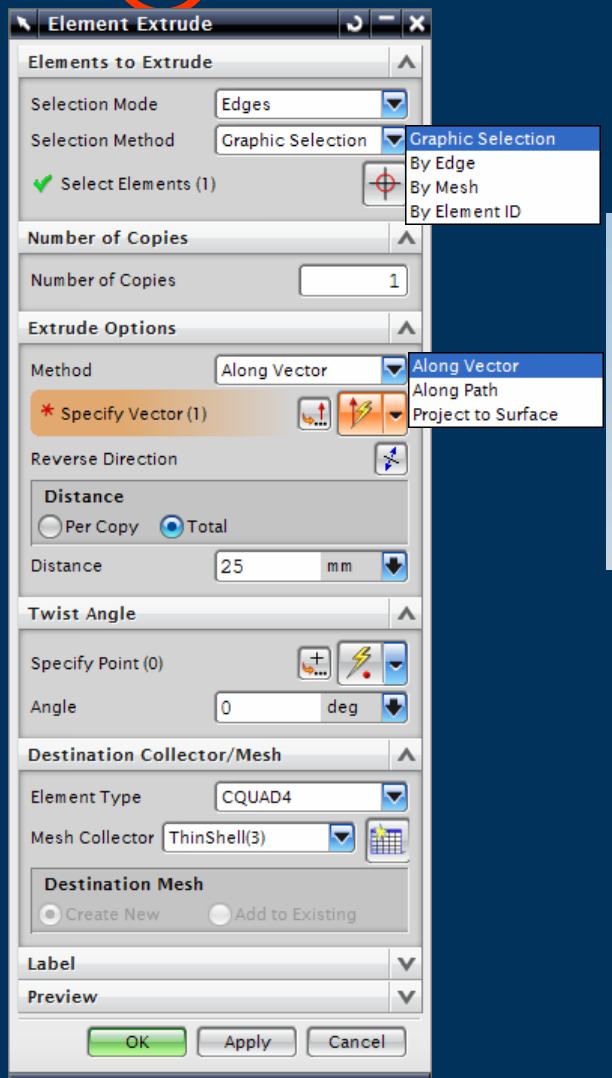
# Meshing – Element Create

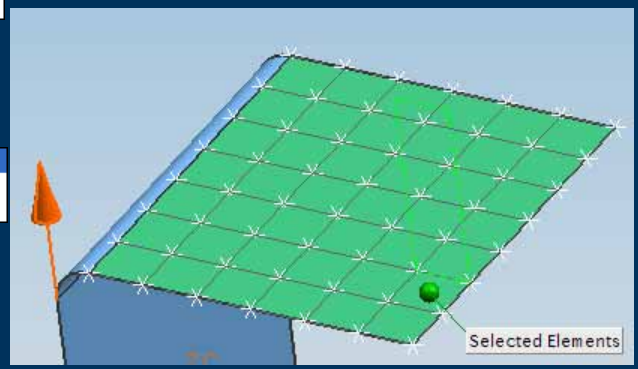


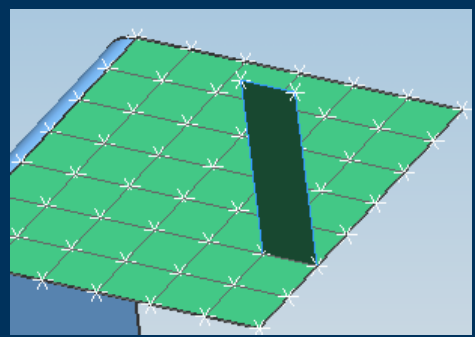

- Element Creation attached to existing Nodes
- Mesh Collector selection or Creation on-the-fly
  - New Mesh
  - Add to Existing Mesh

SIEMENS

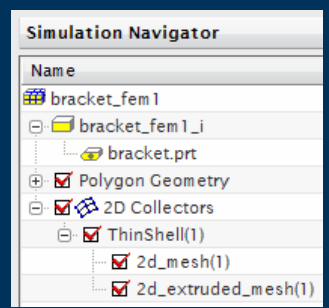
# Meshing – Element Extrude



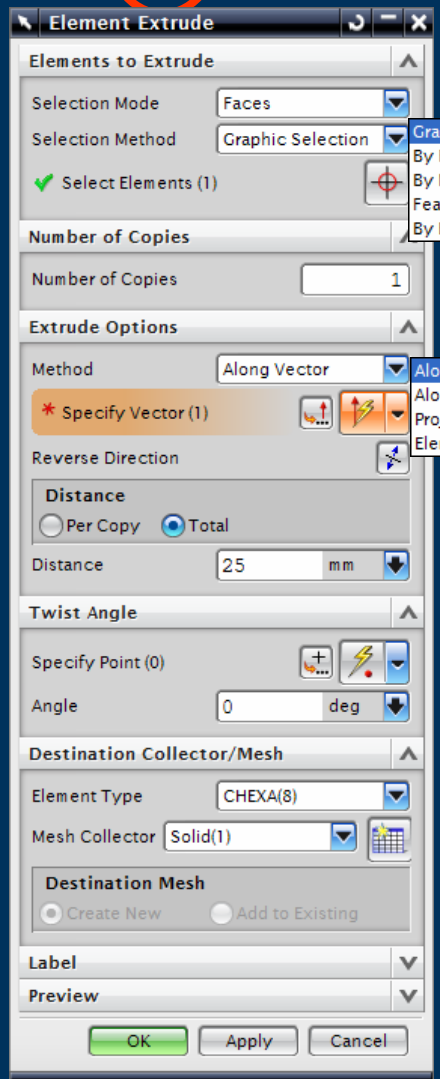



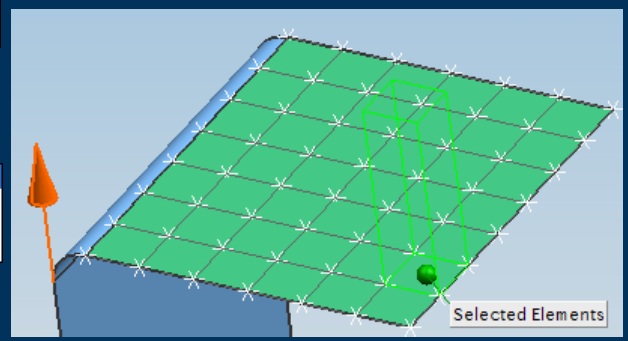



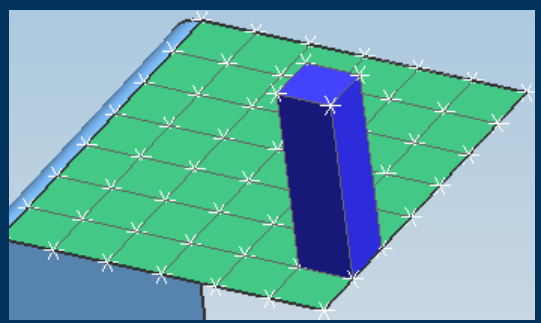
- Extrude an Existing Element(s) Edge
- Mesh Collector selection or Creation on-the-fly
  - New Mesh
  - Add to Existing Mesh



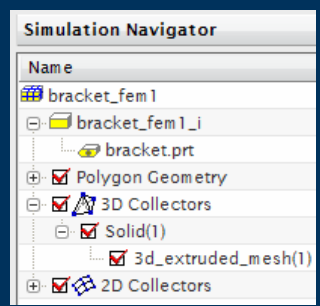
# Meshing – Element Extrude



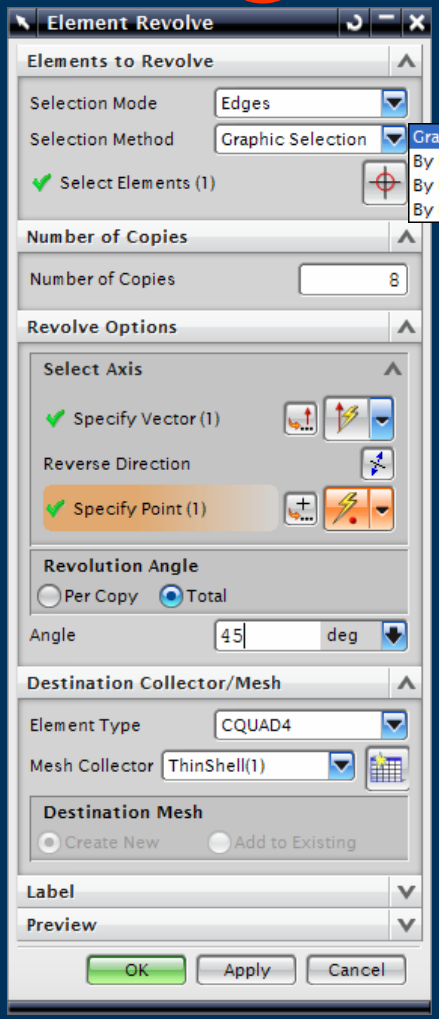



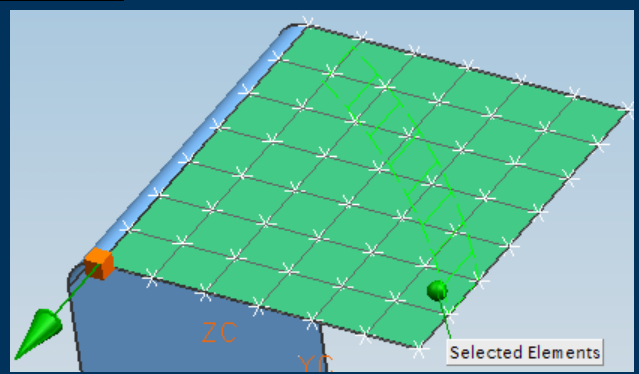



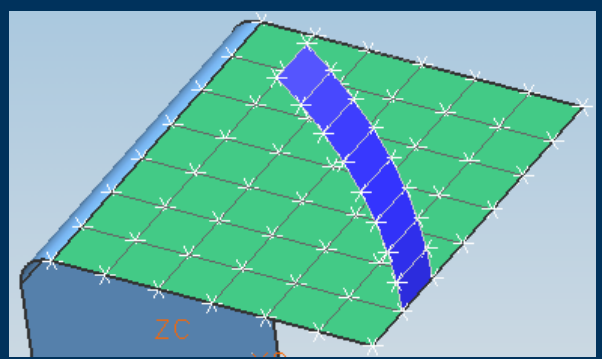
- Extrude an Existing Element(s) Face
- Mesh Collector selection or Creation on-the-fly
  - New Mesh
  - Add to Existing Mesh



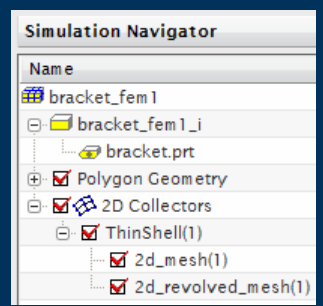
# Meshing – Element Revolve



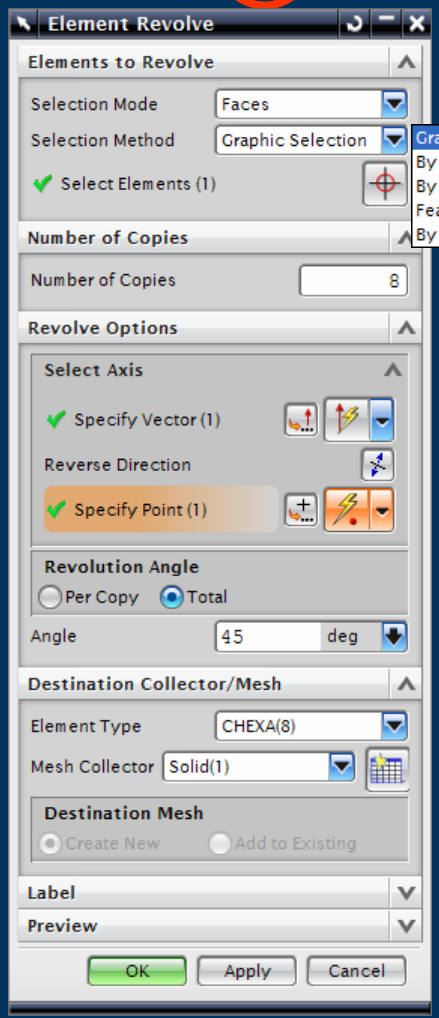



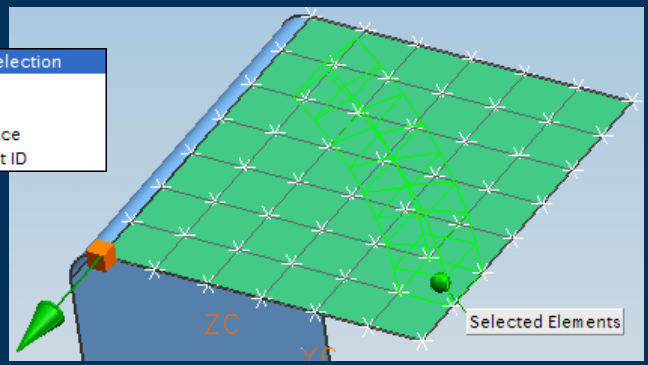



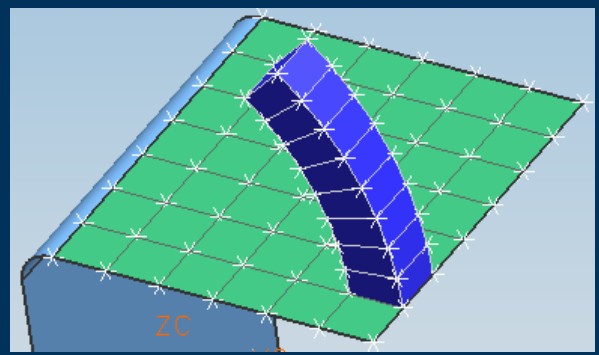

- Revolves an Existing Element(s) Edge
- Mesh Collector selection or Creation on-the-fly
  - New Mesh
  - Add to Existing Mesh



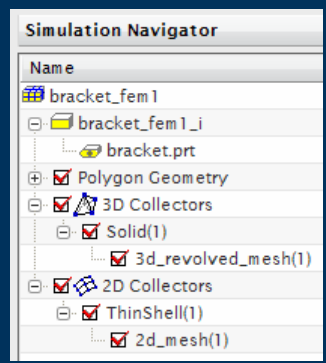
# Meshing – Element Revolve



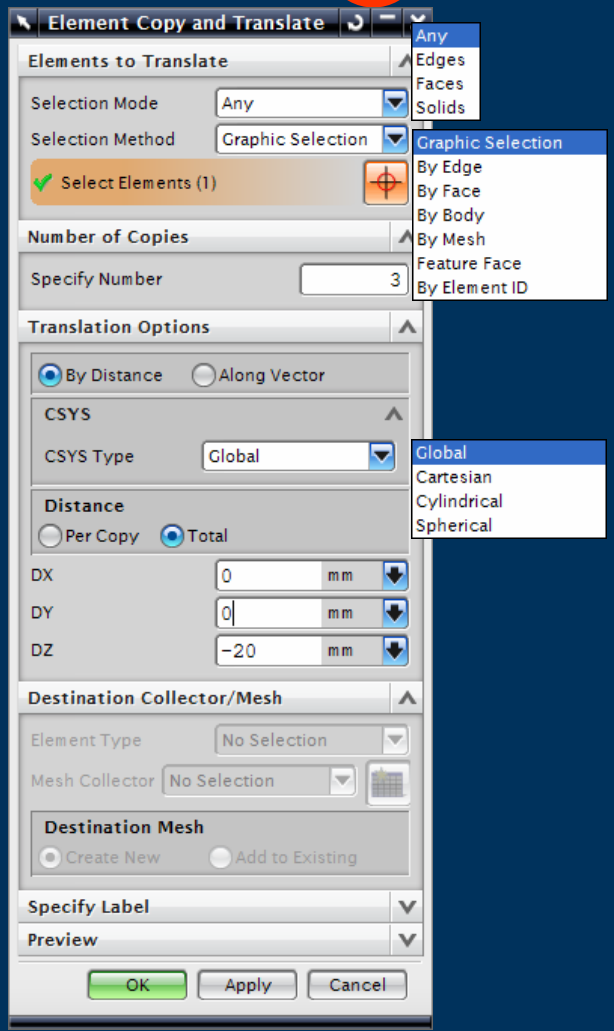



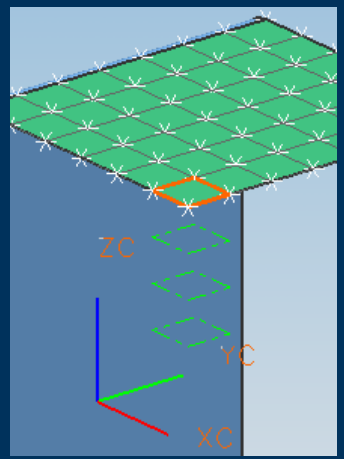



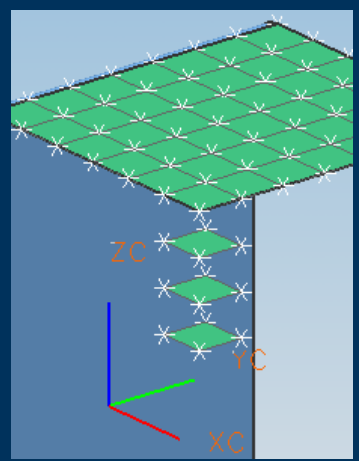

- Revolve an Existing Element(s) Face
- Mesh Collector selection or Creation on-the-fly
  - New Mesh
  - Add to Existing Mesh



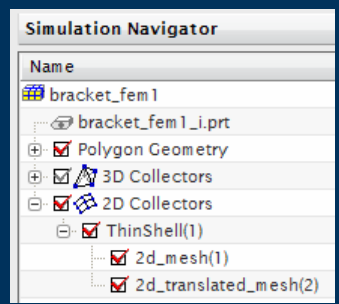
# Meshing – Element Translate & Copy



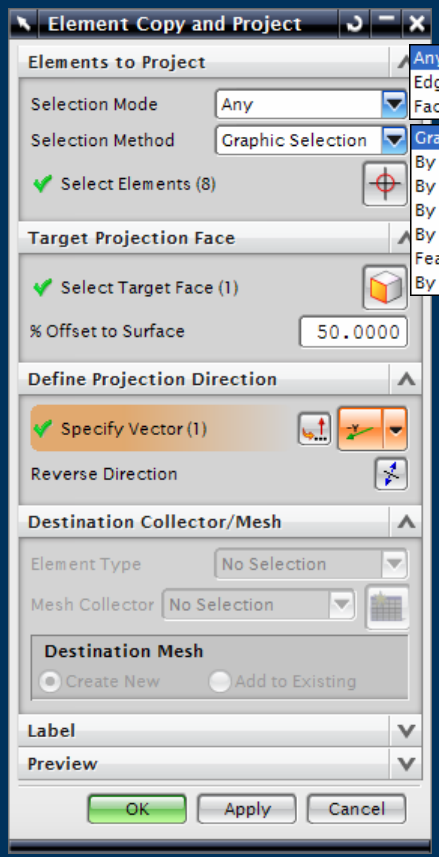



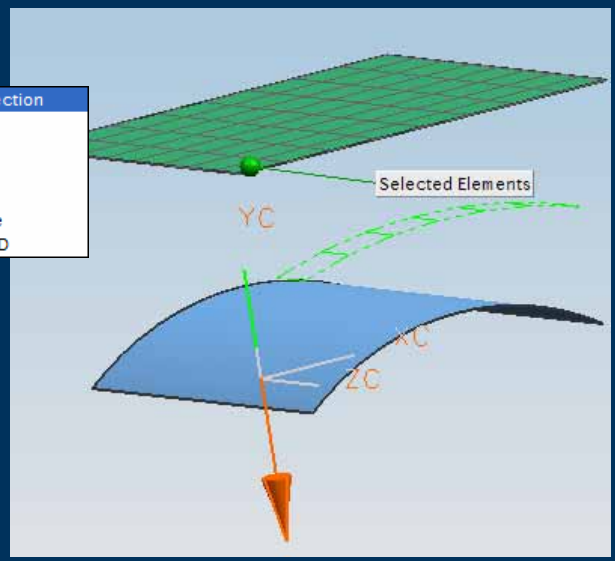

- Translate & Copy Element(s) relative to CSYS or a Vector
- Mesh Collector selection or Creation on-the-fly
  - New Mesh
  - Add to Existing Mesh

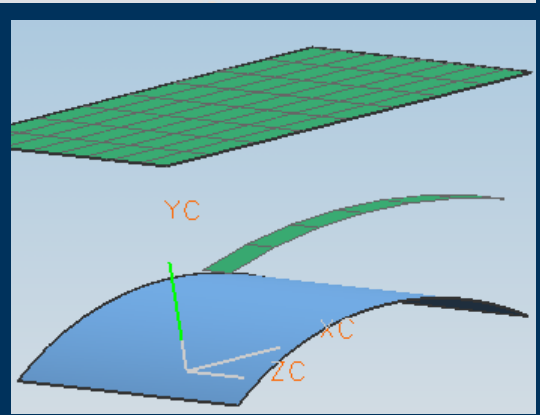


SIEMENS

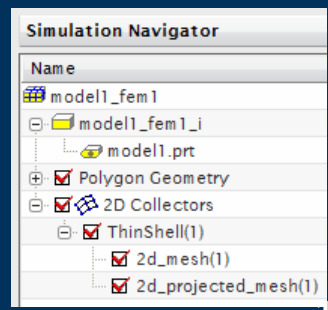
# Meshing – Element Copy & Project



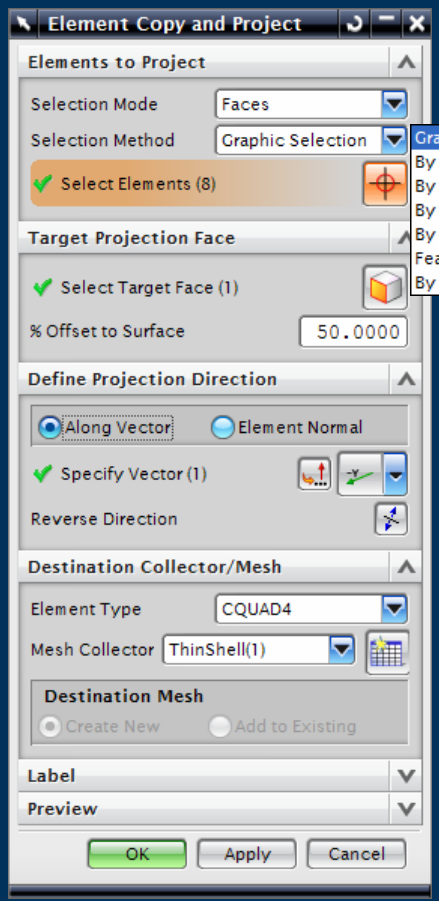



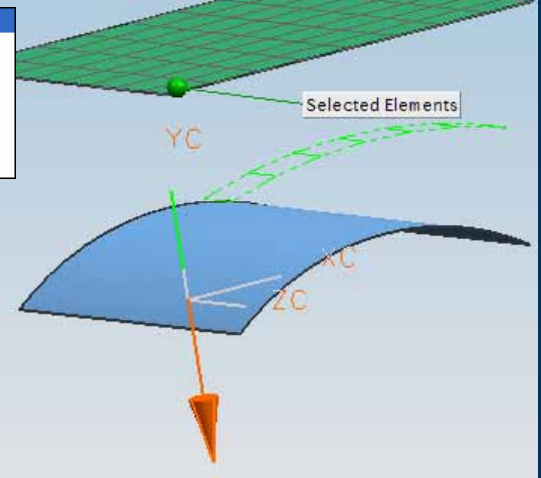

- Project & Copy Element(s) onto a Target Surface(s)
- Mesh Collector selection or Creation on-the-fly
  - New Mesh
  - Add to Existing Mesh

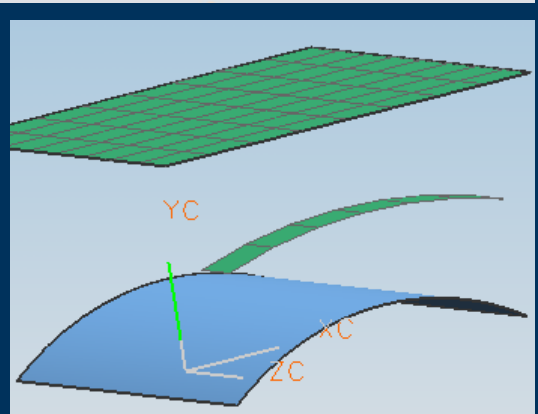


SIEMENS

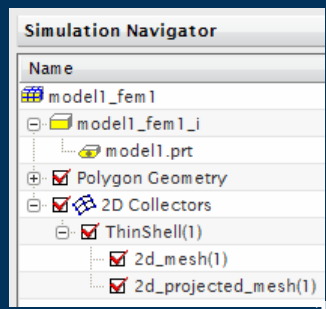
# Meshing – Element Copy & Project



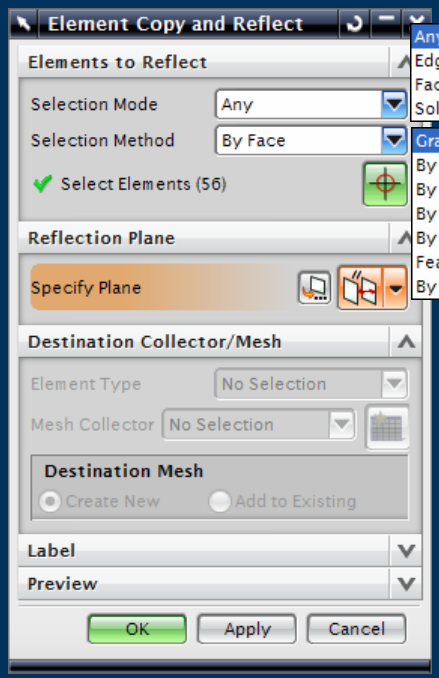



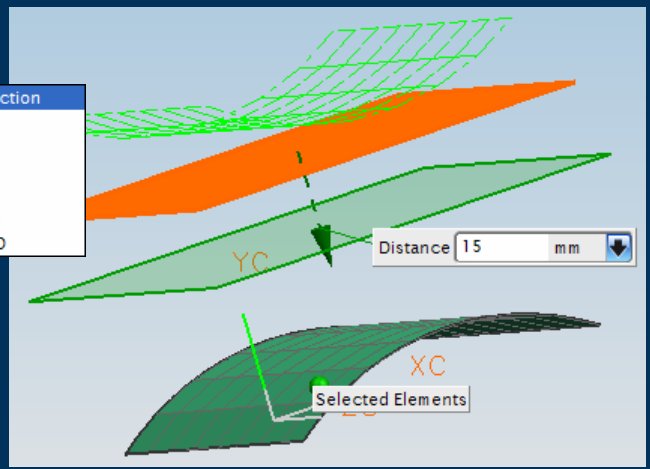



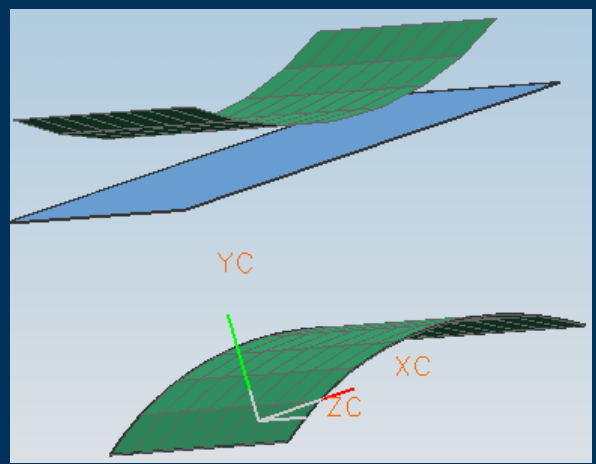

- Project & Copy Element(s) onto a Target Surface(s)
- Mesh Collector selection or Creation on-the-fly
  - New Mesh
  - Add to Existing Mesh



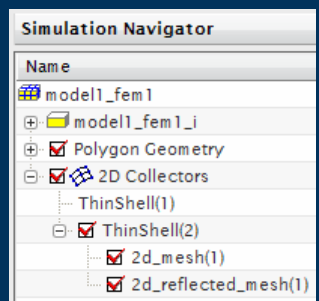
# Meshing – Element Copy & Reflect



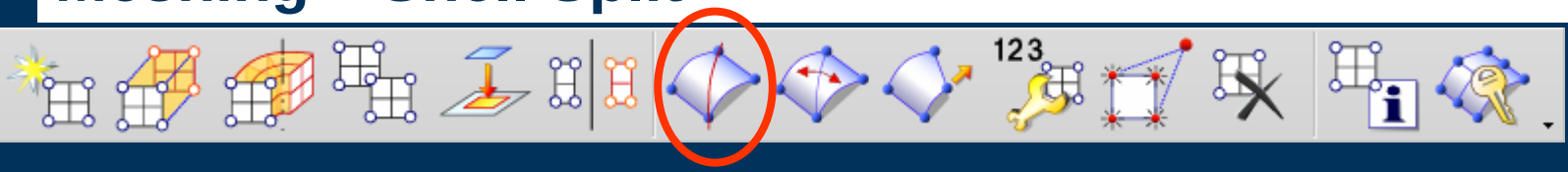



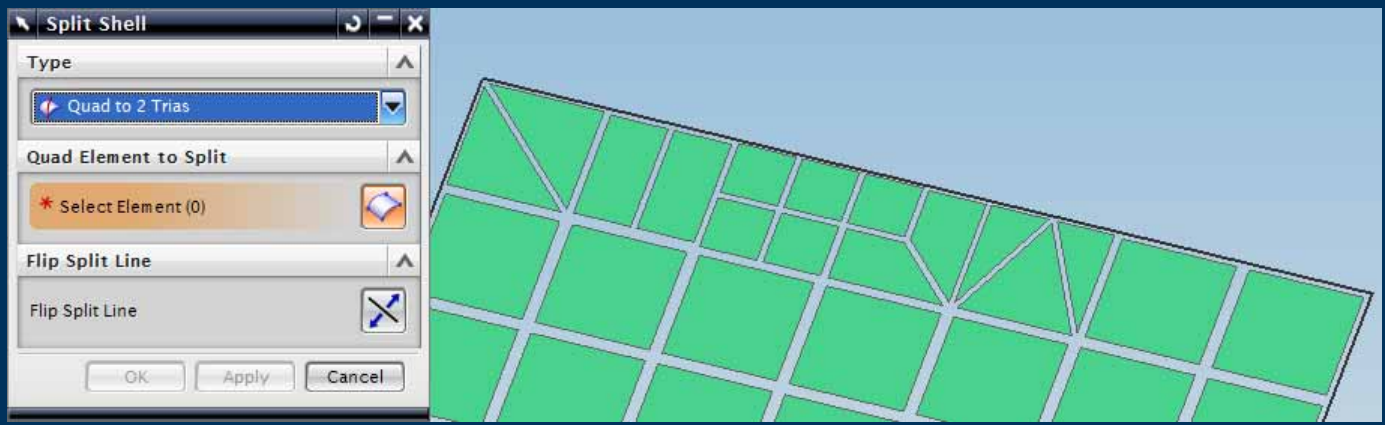

- Reflect & Copy Element(s) about a Plane
- Mesh Collector selection or Creation on-the-fly
  - New Mesh
  - Add to Existing Mesh



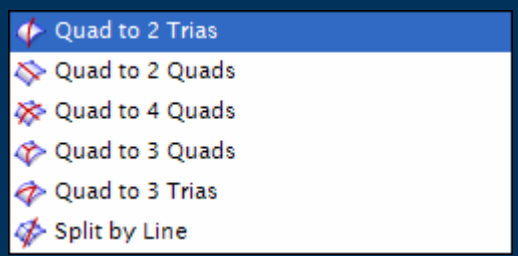
SIEMENS

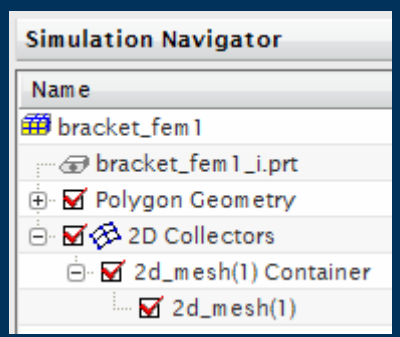
# Meshing – Shell Split






- Splits Quadrilateral Multiple Elements

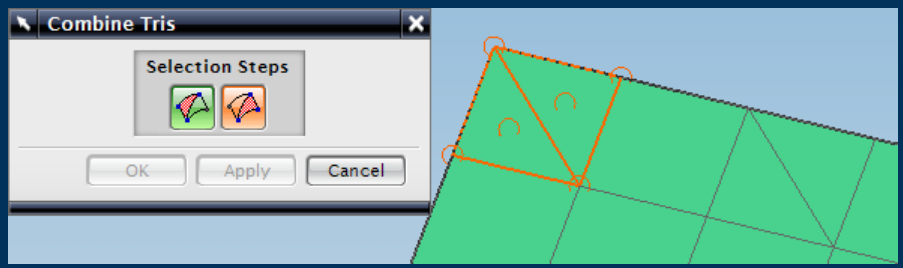


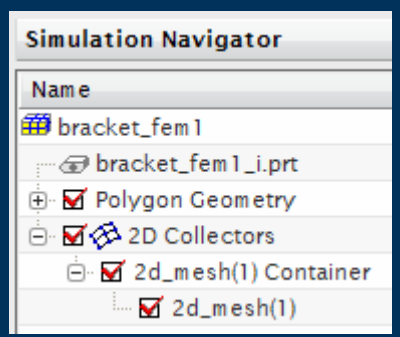

- Mesh Update will remove Manual changes
- New Elements remain in Mesh Collector

SIEMENS

# Meshing – Combine Tris

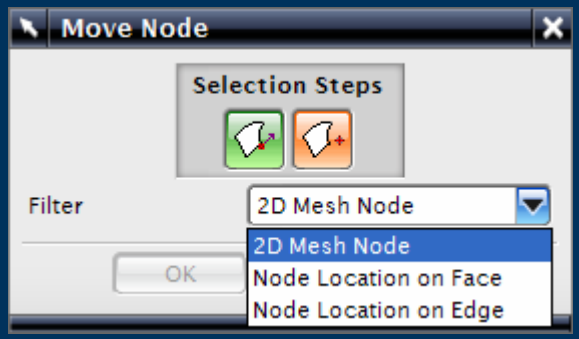



- Combine Triangular elements into Quadrilaterals
  - Linear to Linear
  - Parabolic to Parabolic
- Mesh Update will remove Manual changes
- New Element remains in Mesh Collector

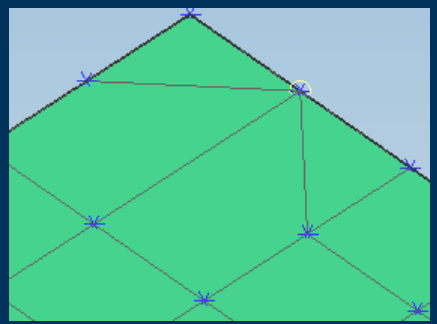


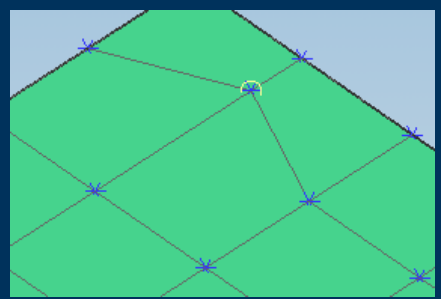
SIEMENS

# Meshing – Move Node

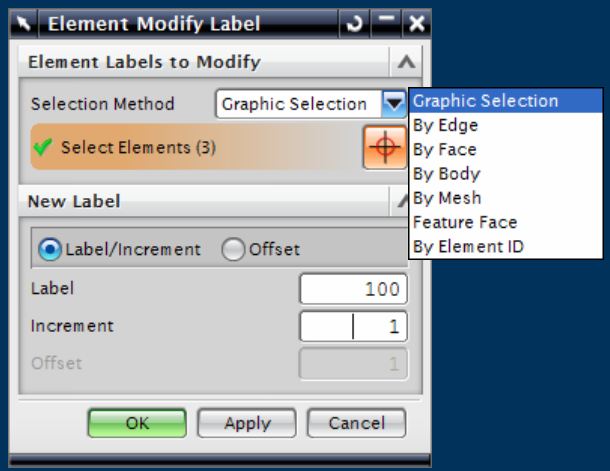



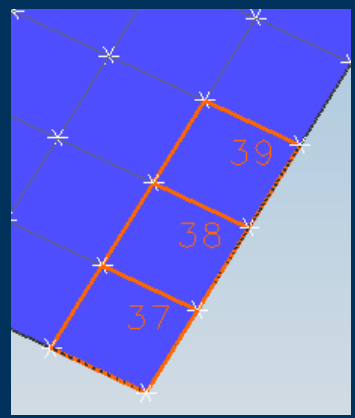

- Move a Node (and it's connected elements)
- Converts Quads to Tris if required & removes duplicate nodes
- Mesh Update will remove Manual changes

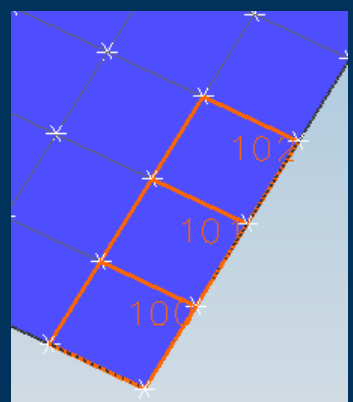




# Meshing – Element Re-Label

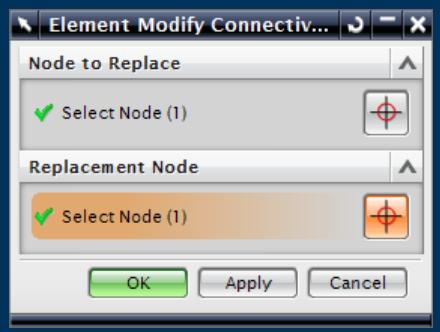



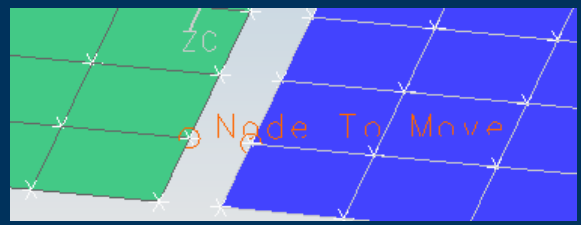




- Modify Element Numbering/Label

SIEMENS

# Meshing – Element Connectivity

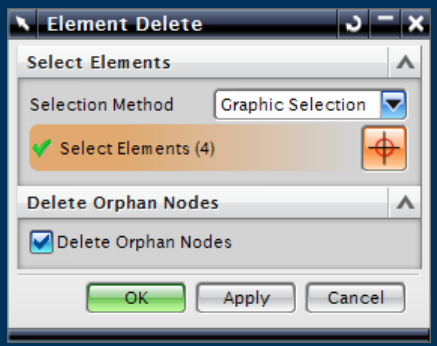



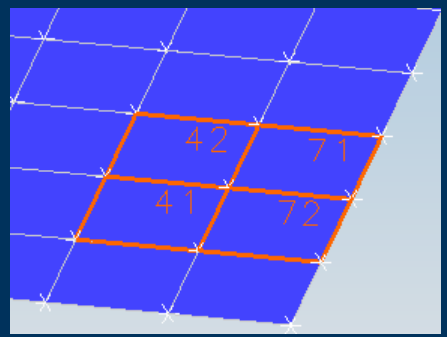


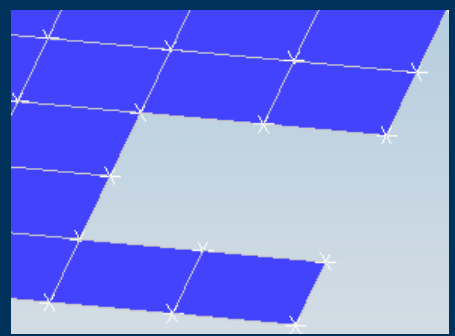
- Replace One Node with another Node
- Specific Mesh Connections

SIEMENS

# Meshing – Element Deletion

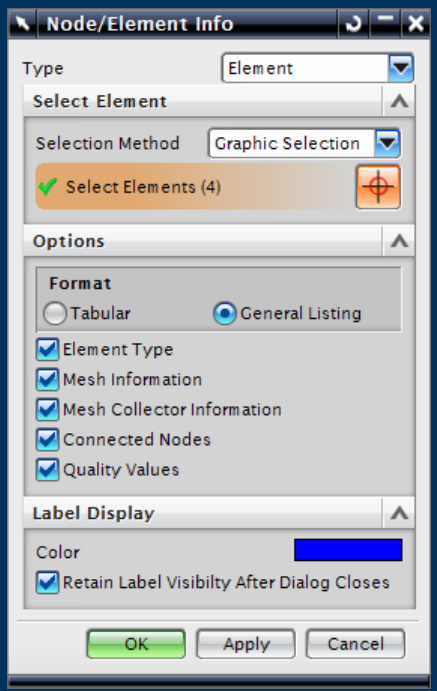


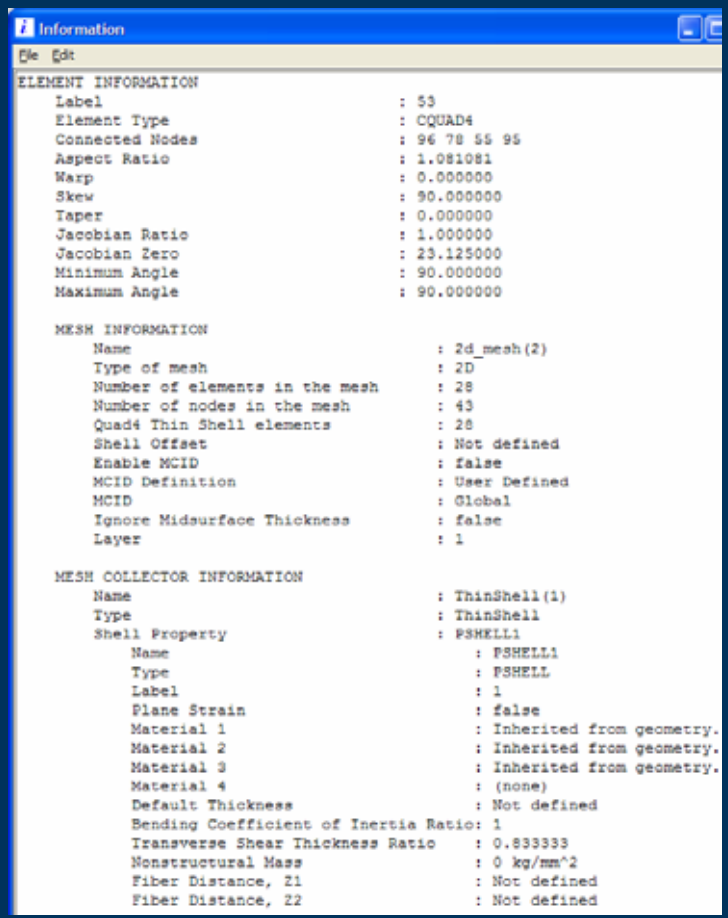



- Delete Elements
- Optionally delete Orphaned Nodes

# Meshing – Node & Element Information

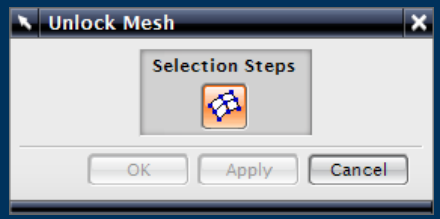



- Element Information
  - Type
  - Mesh
  - Collector
  - Nodes
  - Quality

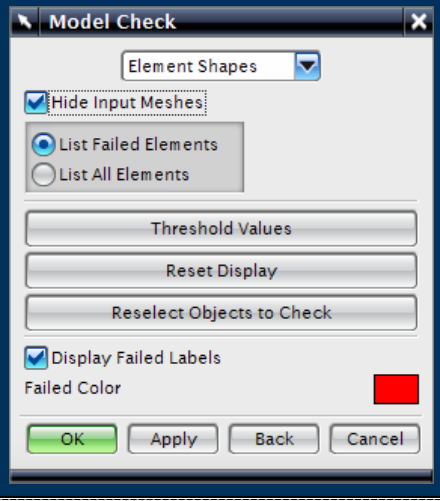
SIEMENS

# Meshing – Mesh Unlock


- Unlock a Mesh for Manual Operations to be carried out

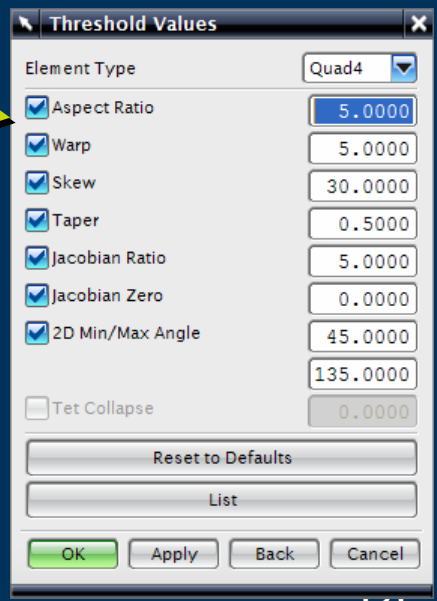
# Model Checking – Element Shape



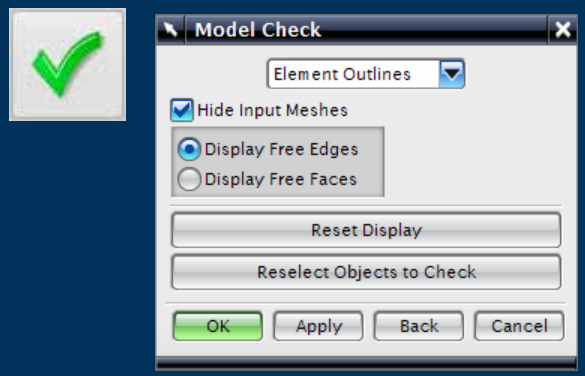

- Element Shape tests the elements against a series of Threshold Values for different element types
- User can set these values in the Preference dialog

```
================================================================================
                    Results of Element Shape Check
================================================================================

Overview

                                Number failed     Number checked
Elements                                0              10727

   Check                        Number failed      Worst value
    Aspect  Ratio                       0               5.702181
    Warp                                0               -N/A-
    Skew                                0               -N/A-
    Taper                               0               -N/A-
    Twist                               0               -N/A-
    Jacobian Ratio                      0               7.206479
    Jacobian   Zero                     0               0.437397
    Minimum   Angle                     0               -N/A-
    Maximum   Angle                     0               -N/A-
    Tet  Collapse                       0              43.554548


                                Threshold values
================================================================================
                    Aspect                                Jacobian  Jacobian  Minimum   Maximum   Tet
Shape               Ratio      Warp     Skew      Taper    Ratio     Zero      Angle     Angle     Collapse
=========           =========  =======  ========  =======  ========  ========  ========  ========  ========
Tri3                 5.000000  -N/A-    60.000000  -N/A-     5.000000  0.000000 20.000000 120.00000  -N/A-
Tri6                 5.000000  -N/A-    60.000000  -N/A-     5.000000  0.000000 20.000000 120.00000  -N/A-
Quad4                5.000000  5.000000 30.000000  0.500000  5.000000  0.000000 45.000000 135.00000  -N/A-
Quad8                5.000000  5.000000 30.000000  0.500000  5.000000  0.000000 45.000000 135.00000  -N/A-
Tetra4              20.000000  -N/A-     -N/A-     -N/A-    10.000000  0.000000  -N/A-    120.00000 100.00000
Tetra10             20.000000  -N/A-     -N/A-     -N/A-    10.000000  0.000000  -N/A-    120.00000 100.00000
Hex8                20.000000  5.000000 100.00000  0.500000 30.000000  0.000000 45.000000 155.00000  -N/A-
Hex20               20.000000  5.000000 100.00000  0.500000 30.000000  0.000000 45.000000 155.00000  -N/A-
Wedge6              20.000000  5.000000 100.00000  0.500000 30.000000  0.000000 45.000000 155.00000  -N/A-
Wedge15             20.000000  5.000000 100.00000  0.500000 30.000000  0.000000 45.000000 155.00000  -N/A-
```

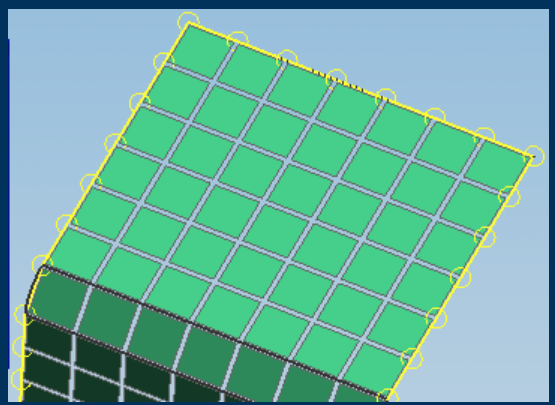
Threshold Values set in Customer Defaults

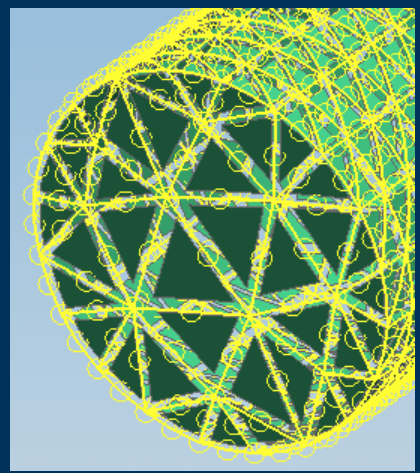


# Model Checking – Element Outlines

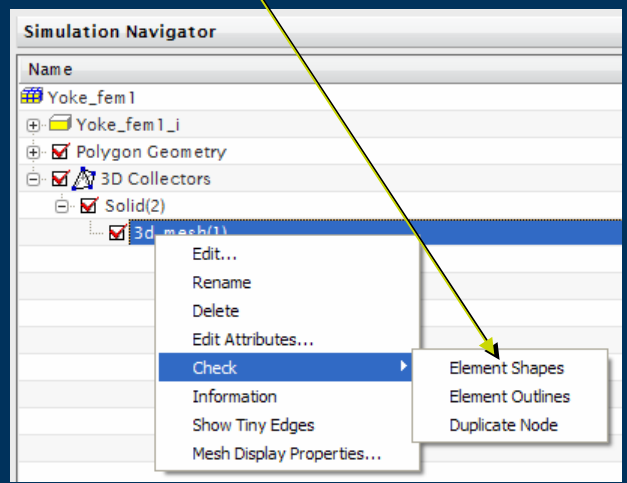



- Element Outlines show the Free Element Faces or Edges



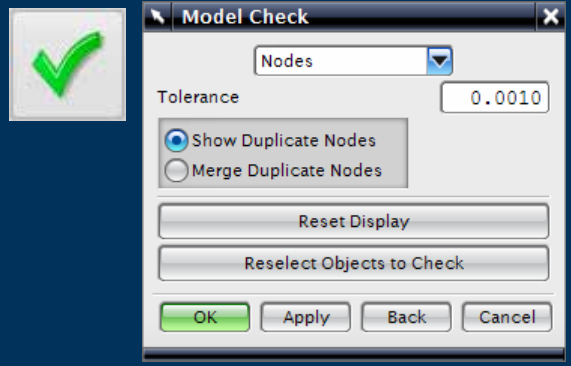


All Checks also available from the Navigator

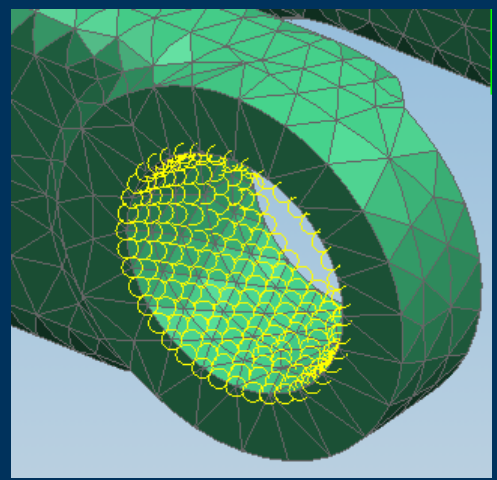


SIEMENS

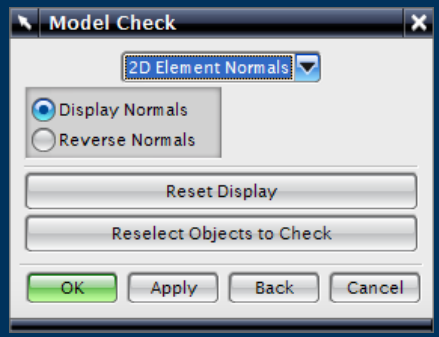
# Model Checking – Duplicate Nodes



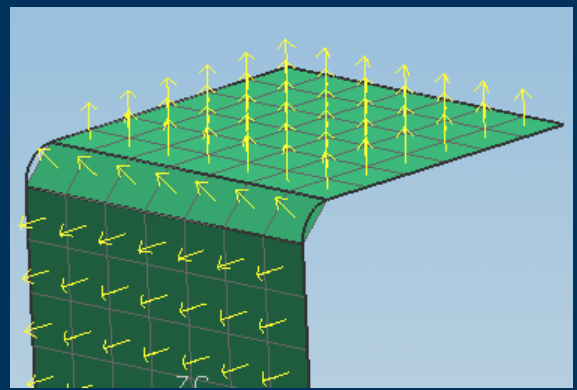

- Duplicate Nodes
  - Locate to check model
  - Merge to correct model



# Model Checking – Element Normals




- Displays the Element Normals



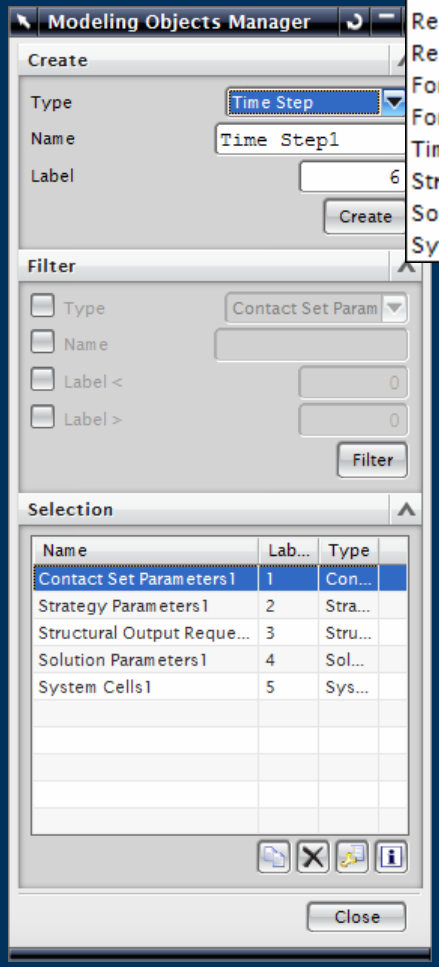
SIEMENS

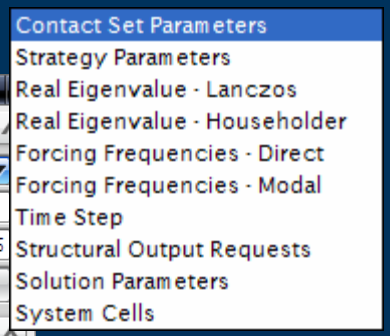
# SIM Part – Pre-Processing

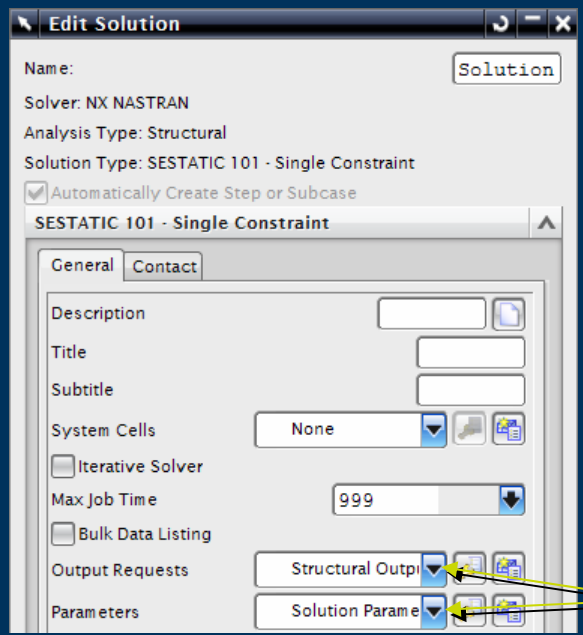


UGS PLM Software

# Modeling Objects – Manager



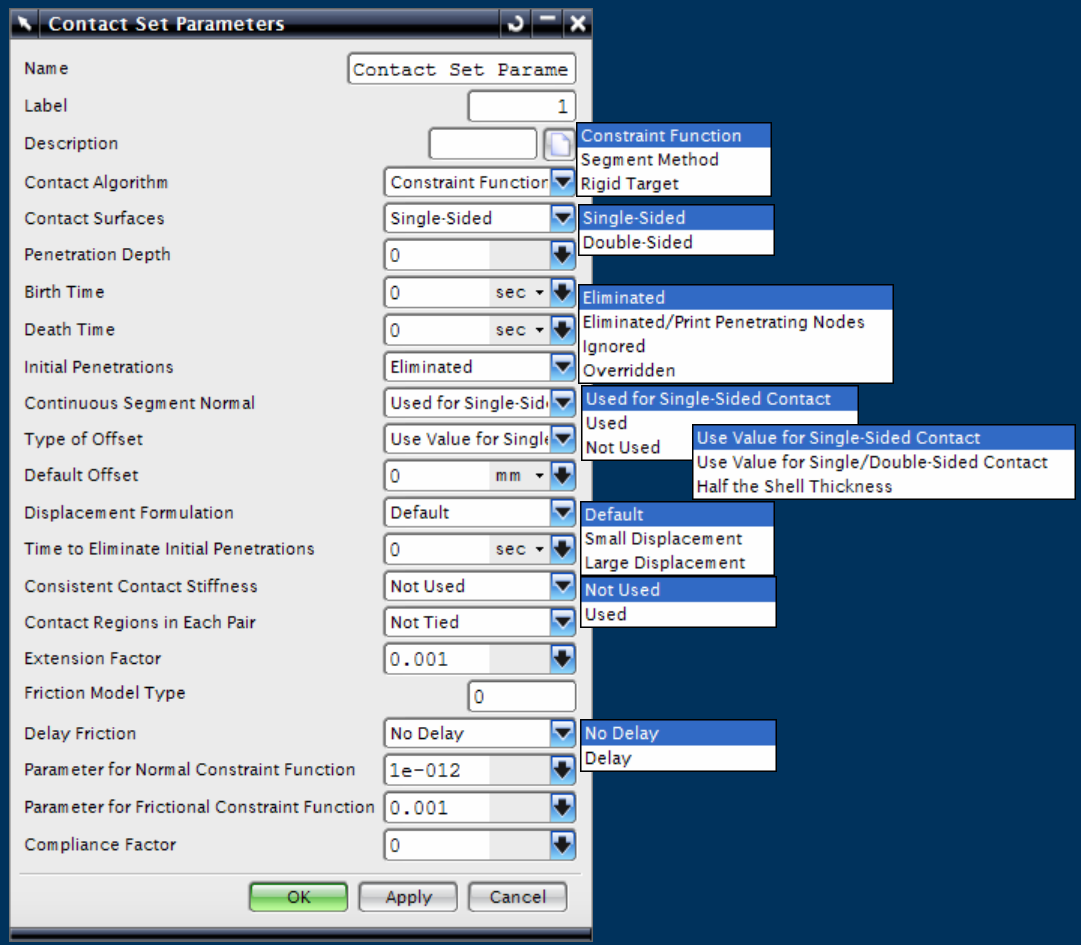




- Modeling Objects
- For re-use by multiple solves
- Solver and Solution Type dependant

Referenced Modeling Objects

SIEMENS

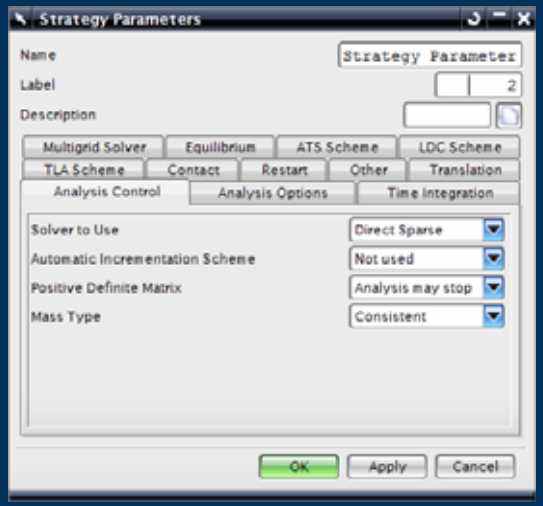
# Modeling Objects – Contact Set Parameters

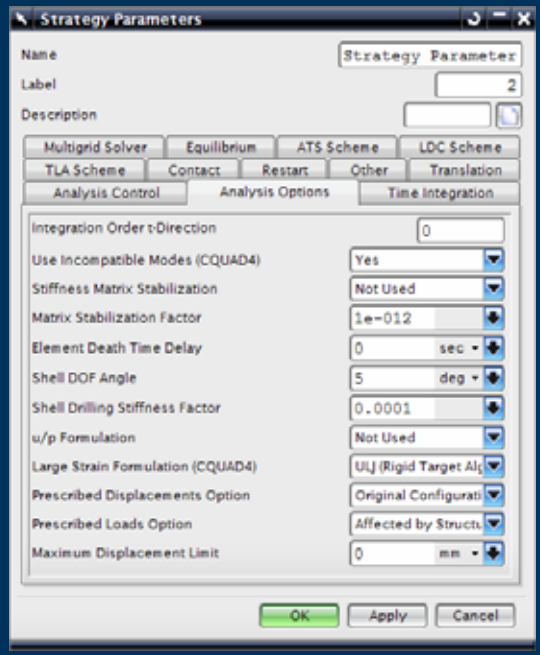


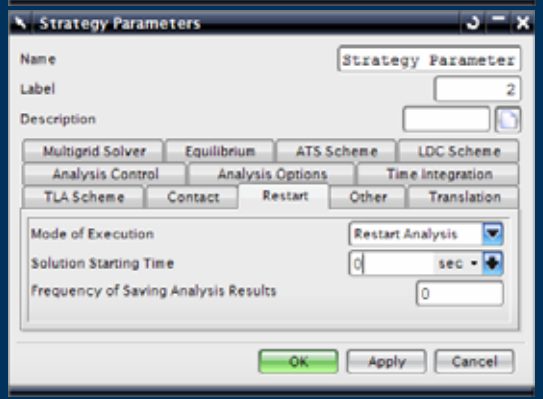

- Parameters to define the Contact conditions
- Solver and Solution Type dependant
- Options shown for NX Nastran

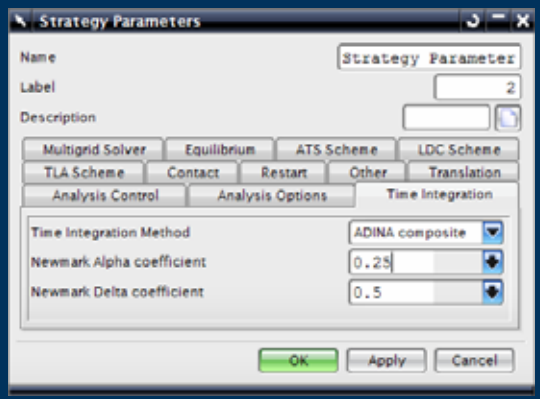
SIEMENS

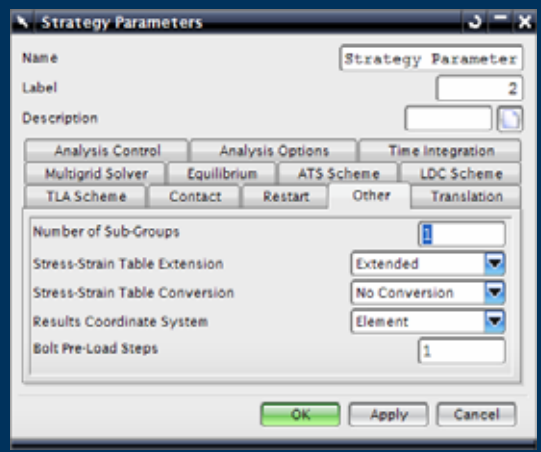
# Modeling Objects – Strategy Parameters



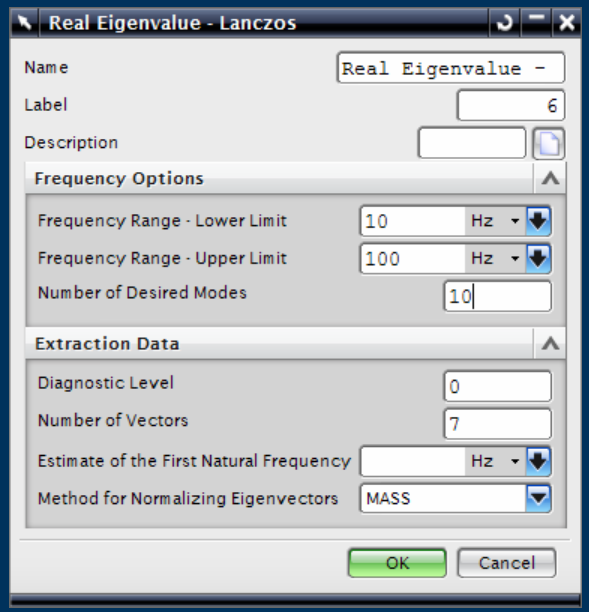



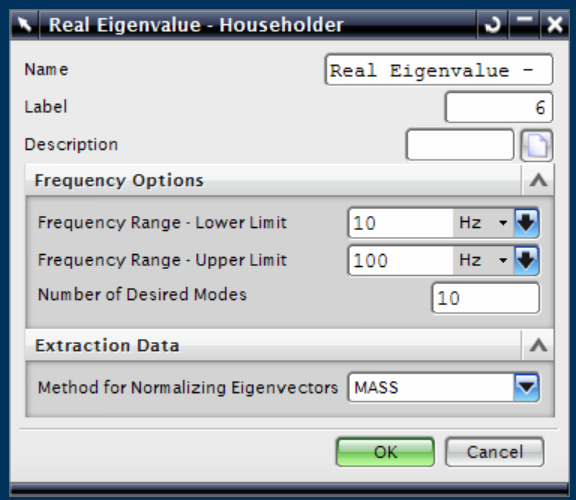

- Parameters to define the Non-Linear Strategy
- Solver and Solution Type dependant
- Options shown for NX Nastran

# Modeling Objects – Real Eigenvalue – Lanczos & Householder

SIEMENS



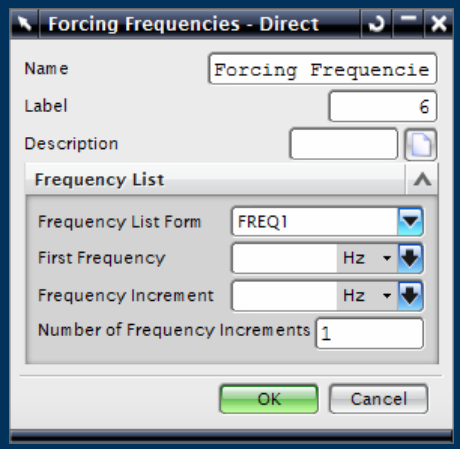

- Parameters for a Lanczos run
- Solver and Solution Type dependant
- Options shown for NX Nastran



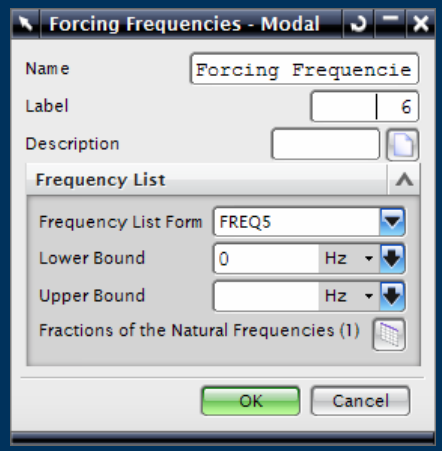

- Parameters for a Householder run
- Solver and Solution Type dependant
- Options shown for NX Nastran

# Modeling Objects – Forcing Frequencies – Direct & Modal

SIEMENS



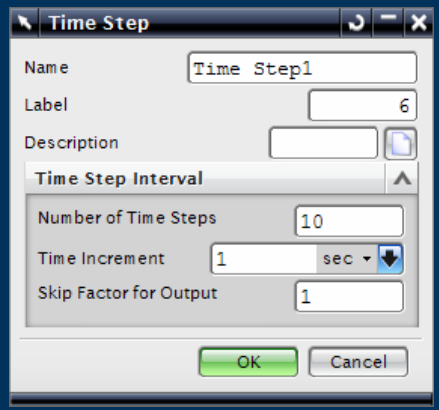

- Parameters for a Direct Forced Frequency run
- Solver and Solution Type dependant
- Options shown for NX Nastran




- Parameters for a Modal Forced Frequency run
- Solver and Solution Type dependant
- Options shown for NX Nastran

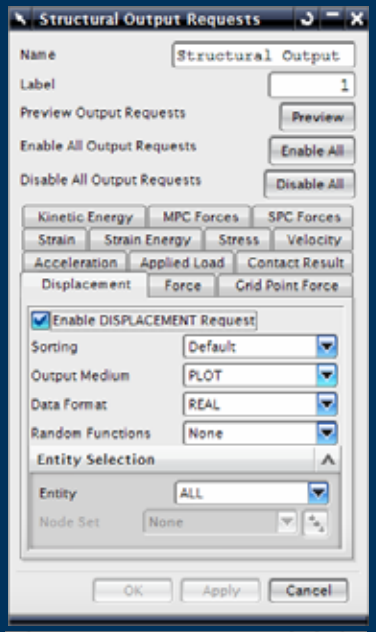
SIEMENS

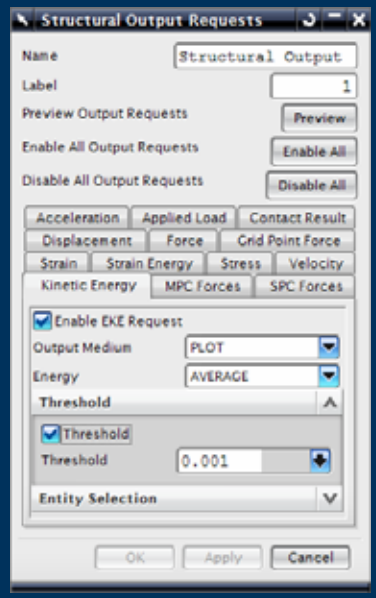
# Modeling Objects – Time Step

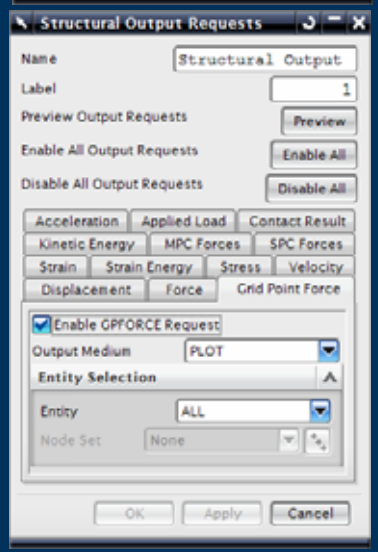


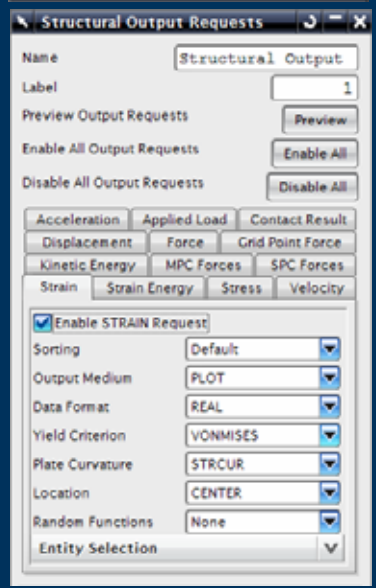

- Parameters to define a Time Step

# Modeling Objects – Structural Output Requests

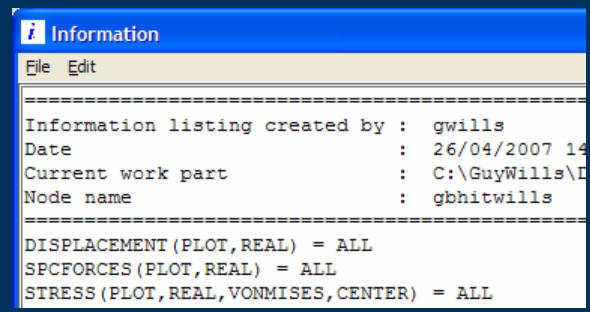


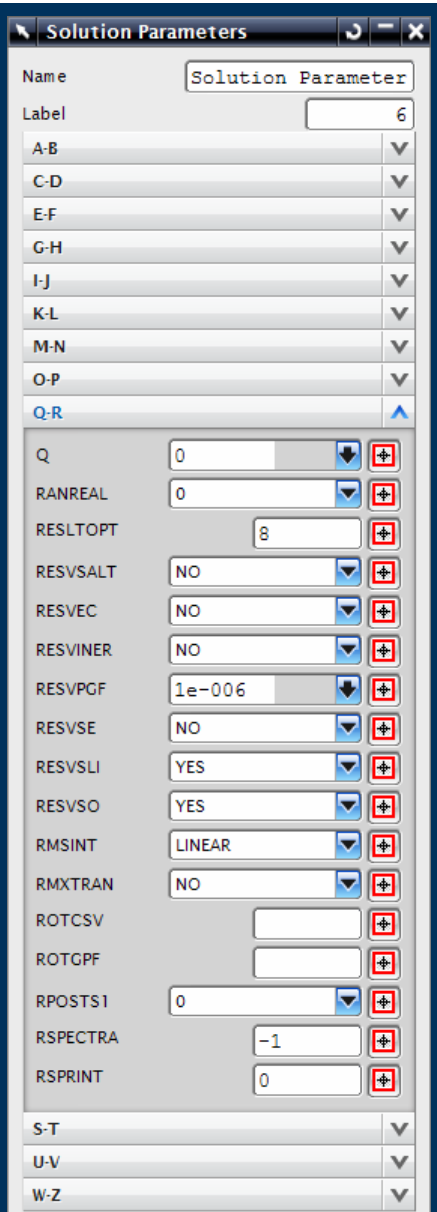



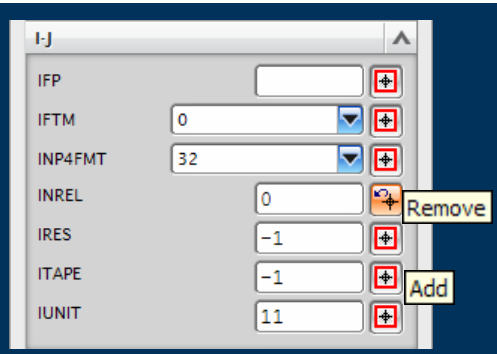




- Parameters to define Structural Output Requests
- Grouped according to function
- Preview to see what will be written to the solver

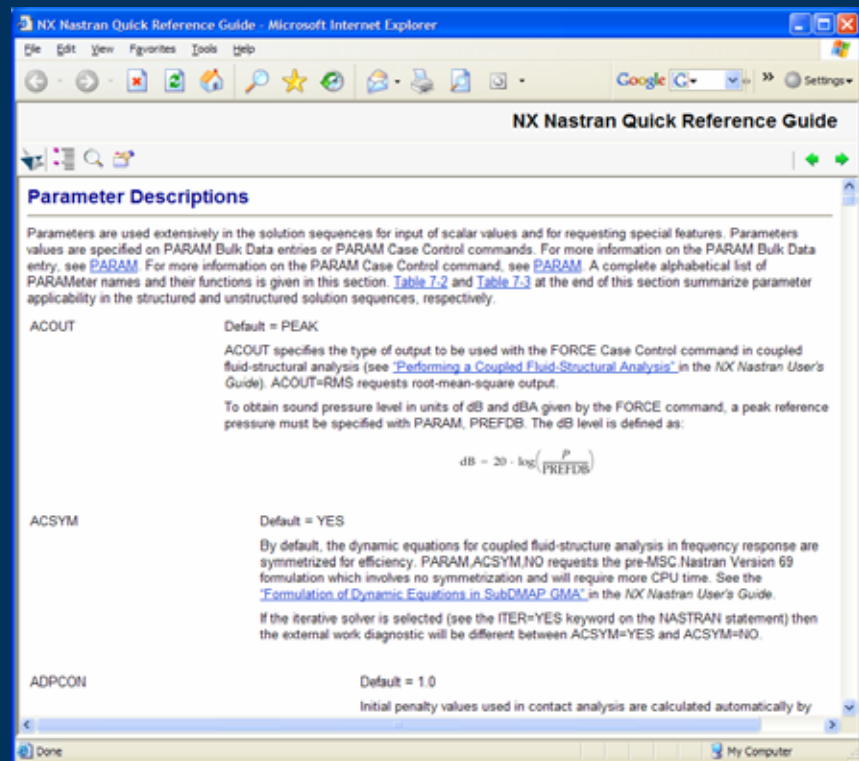


# Modeling Objects – Solution Parameters



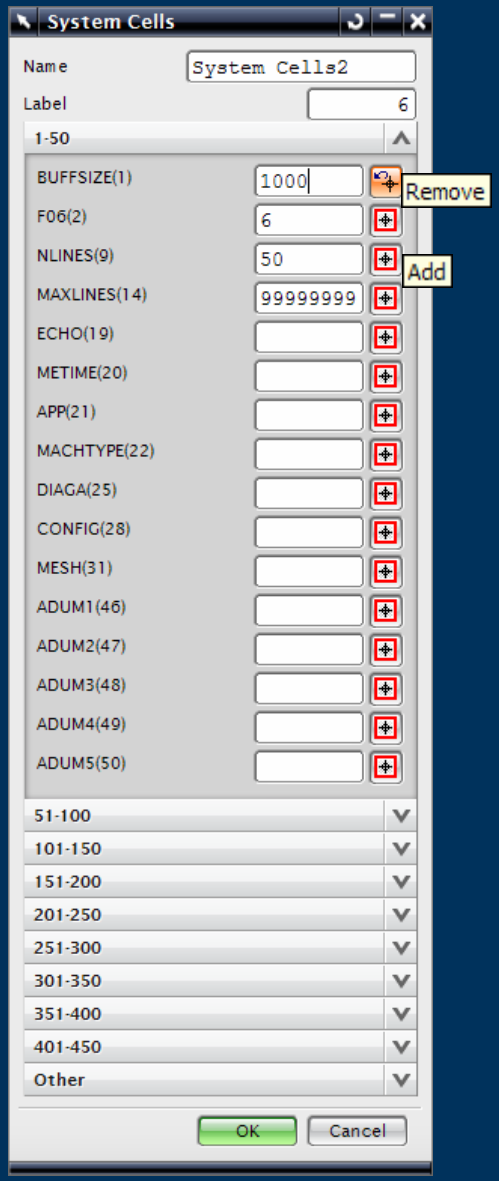




- Solution Parameters
- Solver and Solution Type dependant
- Options shown for NX Nastran
- See Quick Reference Guide for details

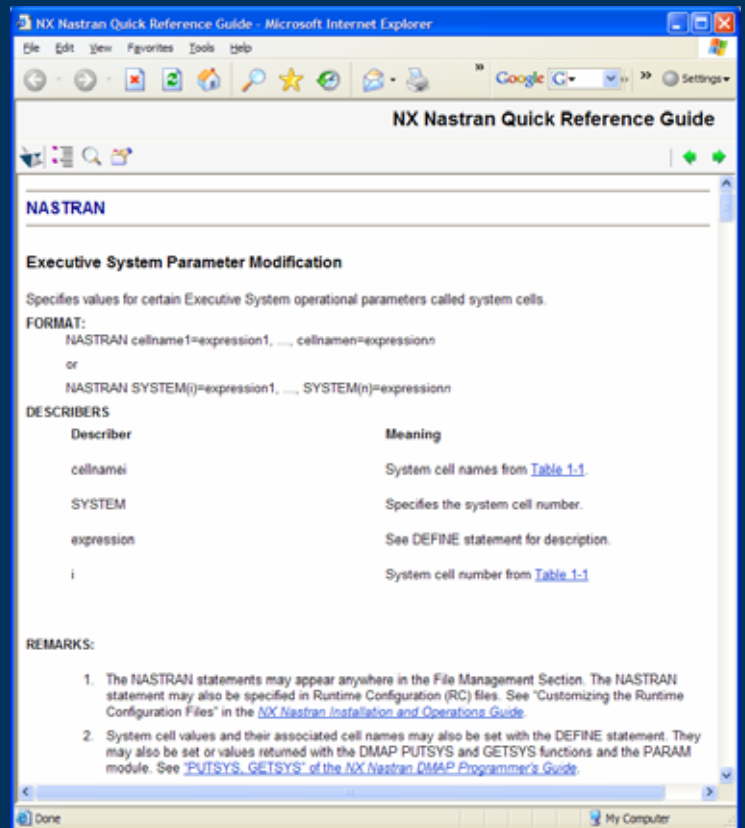


SIEMENS

# Modeling Objects – System Cells

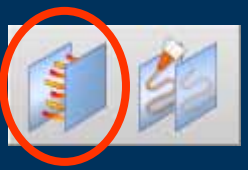


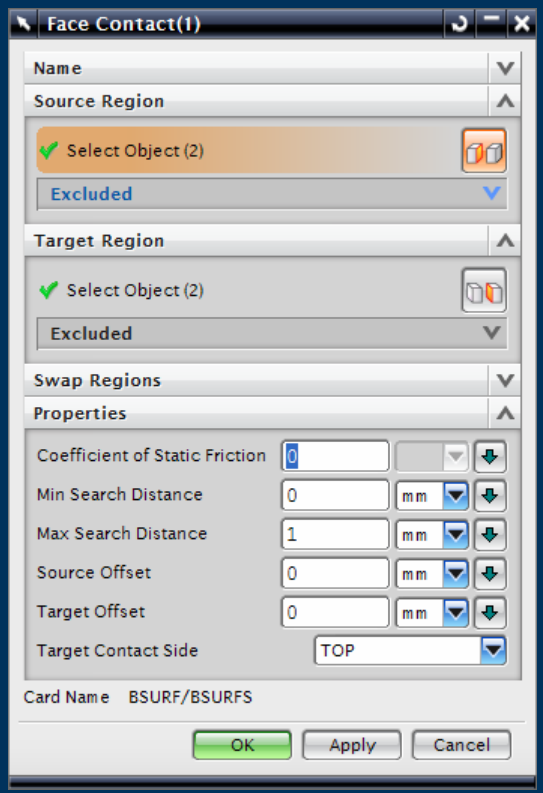

- Solution Parameters
- Solver and Solution Type dependant
- Options shown for NX Nastran
- See Quick Reference Guide for details



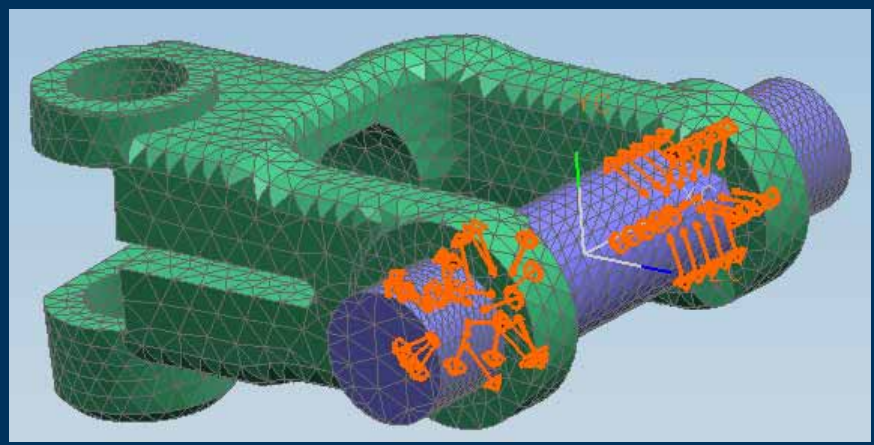
SIEMENS

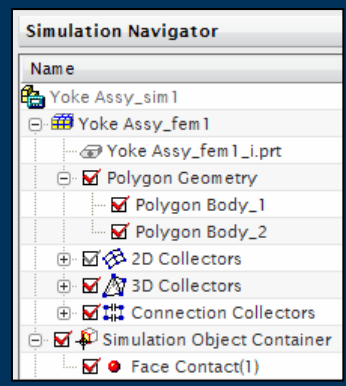
# Surface to Surface – Contact





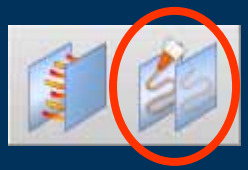

- Surface to Surface Contact options
  - Automatic Detection or Manual Selection
  - Coefficient of Friction
  - Search distances
  - Offsets

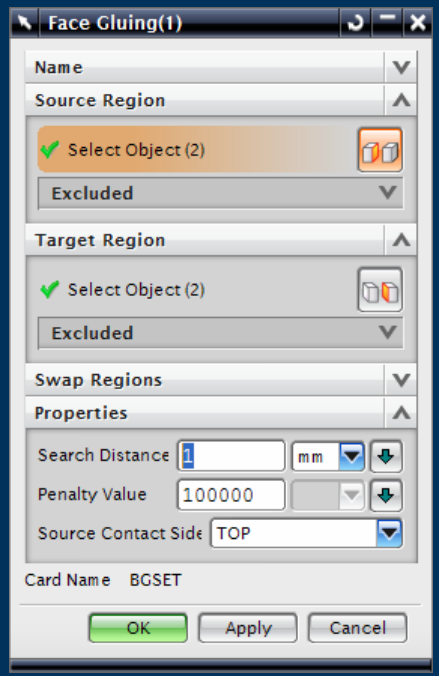




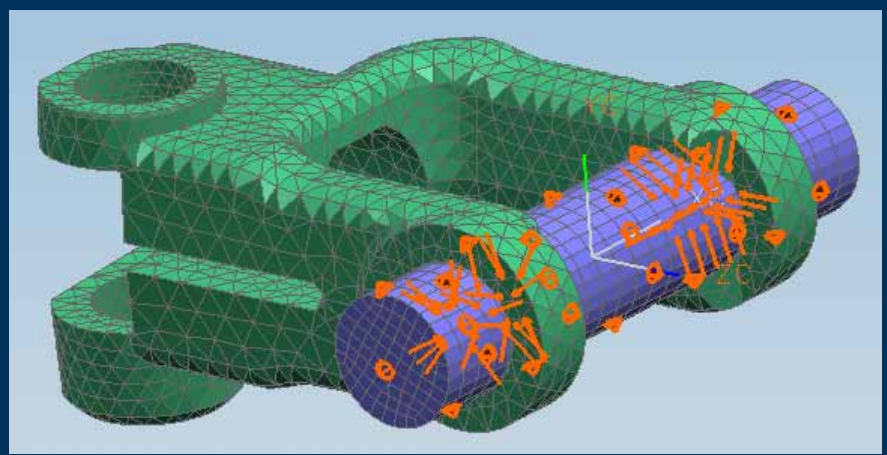
SIEMENS

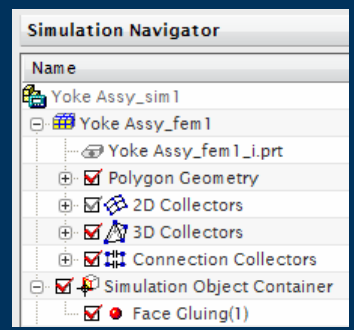
# Surface to Surface – Glue



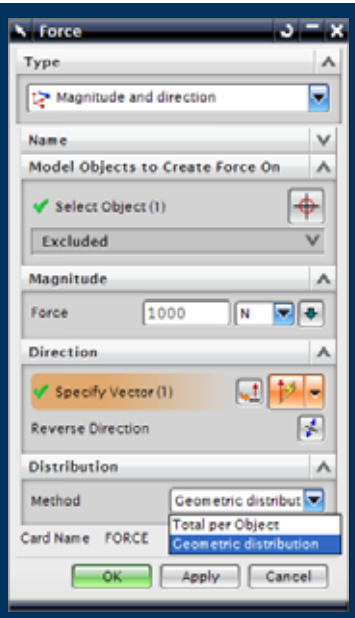


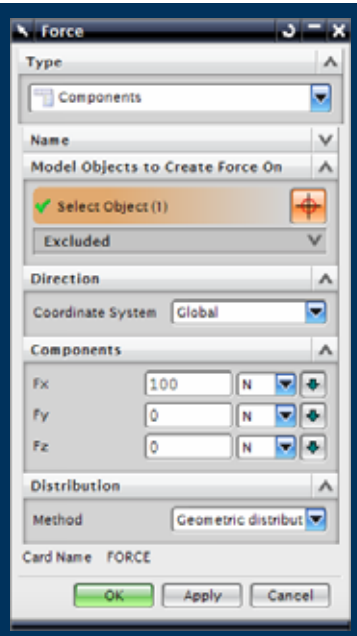

- Surface to Surface Glue options
  - Search distance
  - Penalty Value
- Does not require similar meshes
  - For example Tet to Hex
- Smooth transition of loads across boundaries

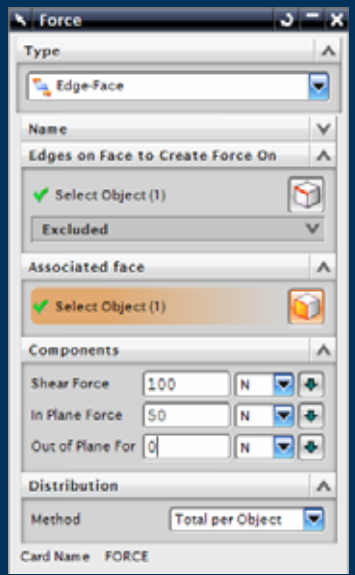


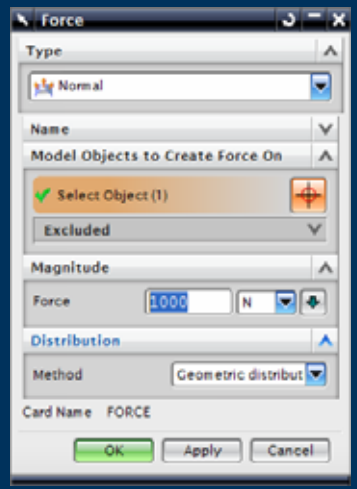


# Loads – Force

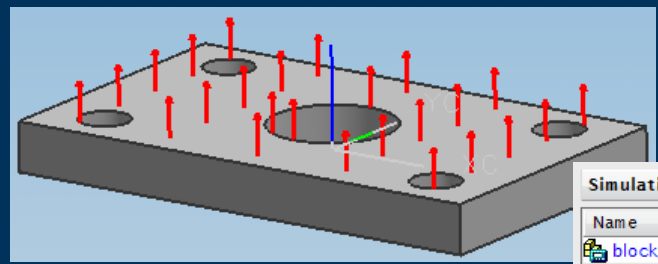


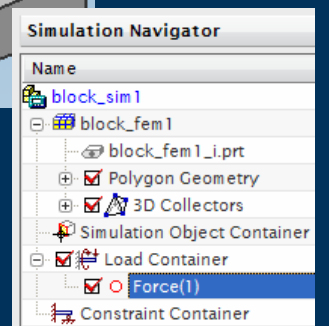



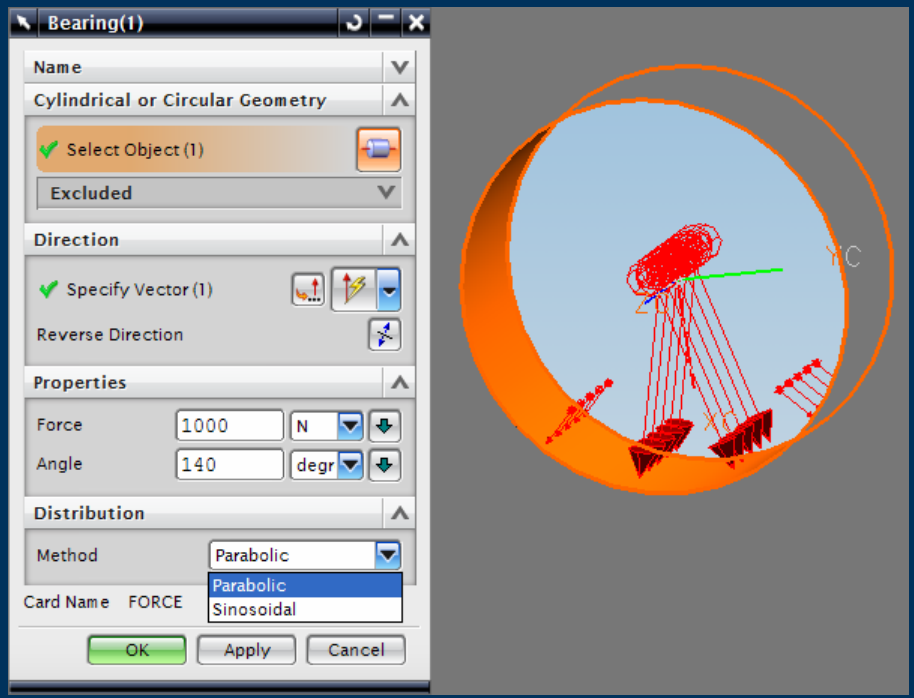




- Force Load Options
  - Magnitude and direction
  - Normal to selected faces
  - Fx, Fy, Fz Components relative to selected coordinate system
  - Shear, In/Out plane force
- Managed in the Load Container

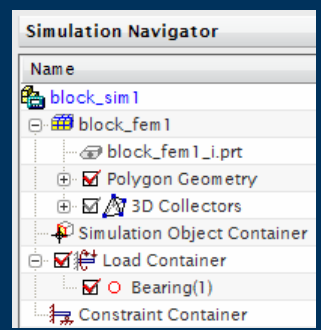




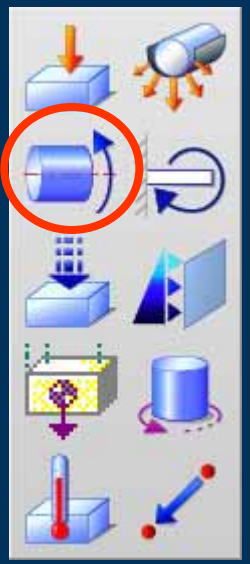
# Loads – Bearing

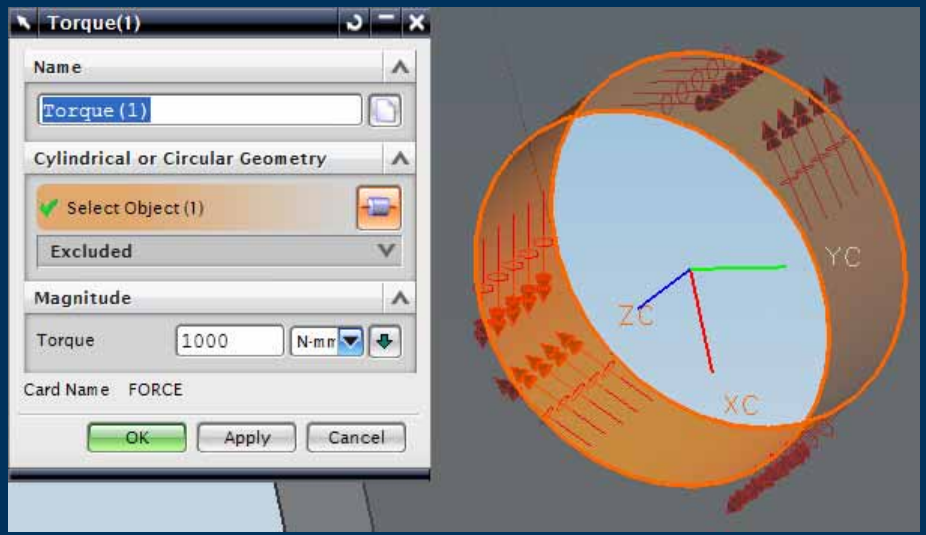



- Bearing Load
  - Distributed load across cylindrical curves or faces
  - Parabolic or Sinosoidal distribution
- Managed in the Load Container

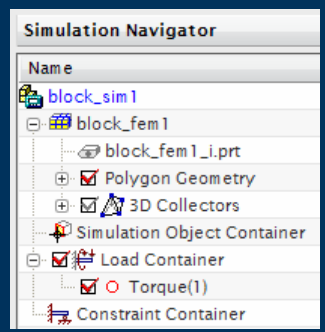


# Loads – Torque

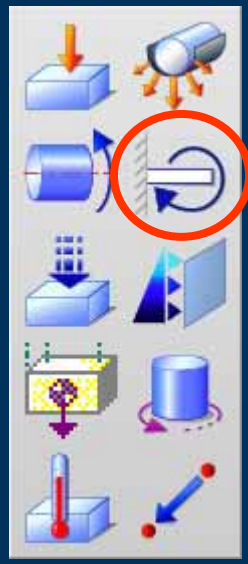


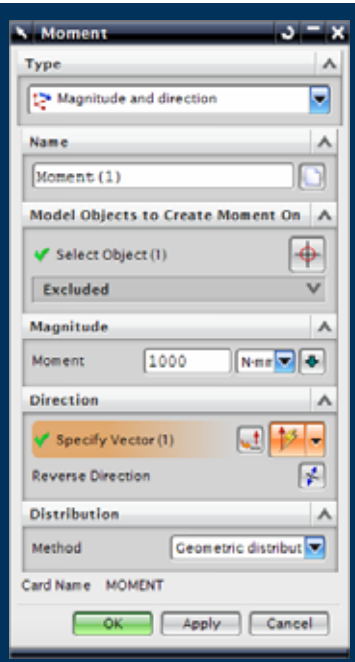


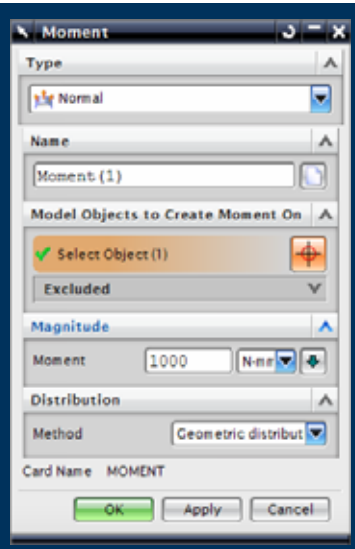

- Torque Load
  - Distributed load across cylindrical curves or faces
- Managed in the Load Container

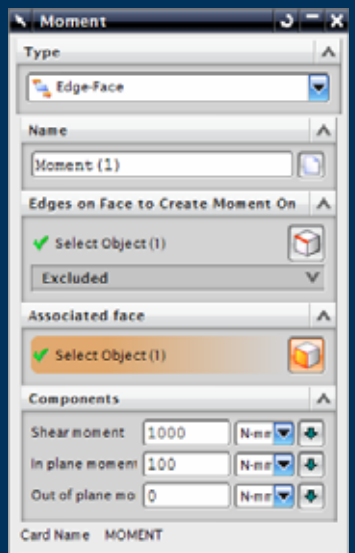


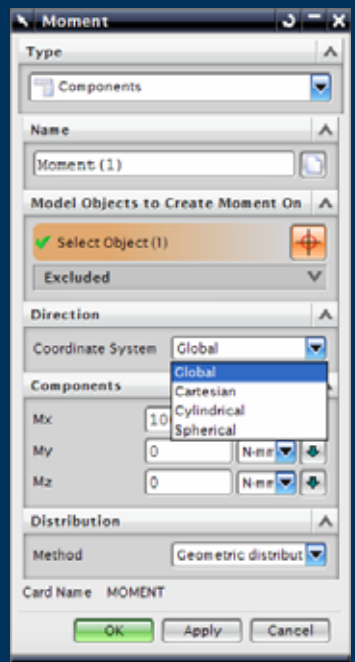
# Loads – Moment



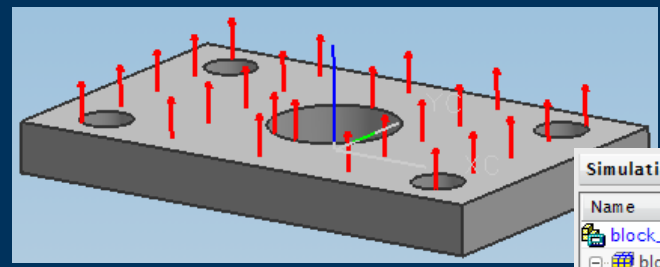


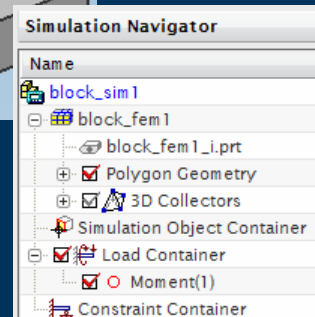



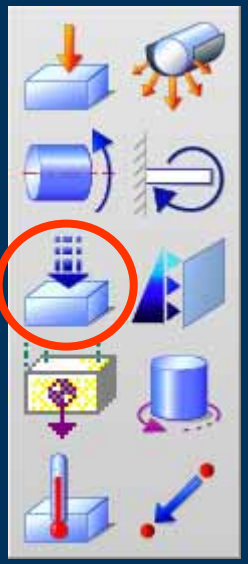



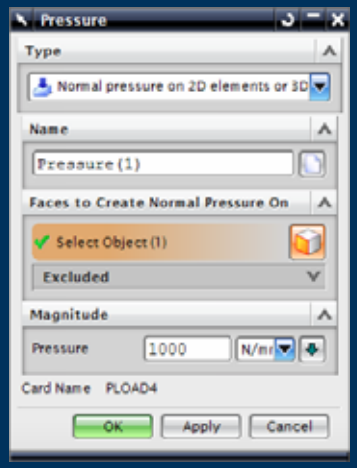

- Moment Load Options
  - Magnitude and direction
  - Normal to selected faces
  - Mx, My, Mz Components relative to selected coordinate system
  - Shear, In/Out plane moment
- Managed in the Load Container

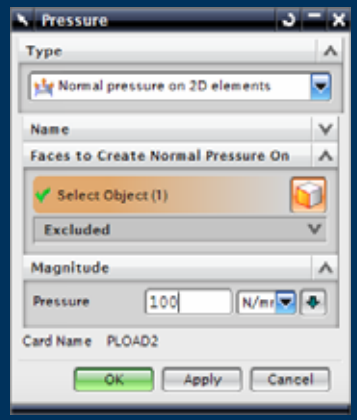


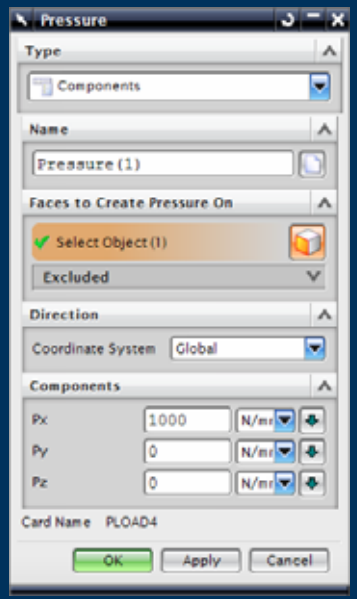


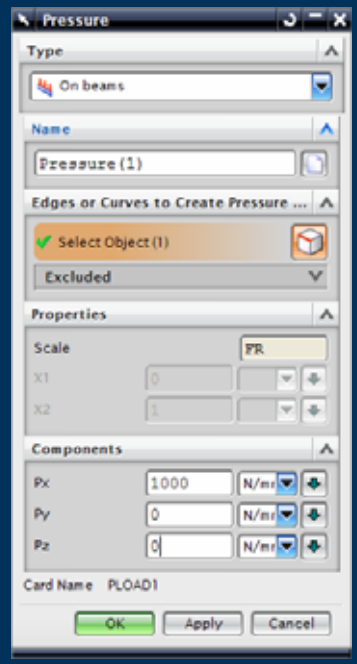
# Loads – Pressure



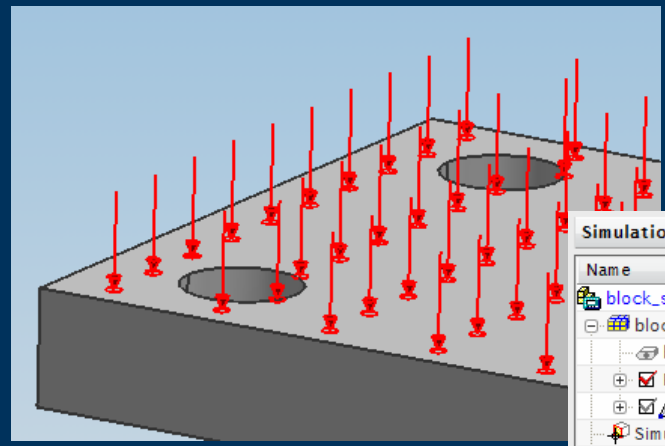


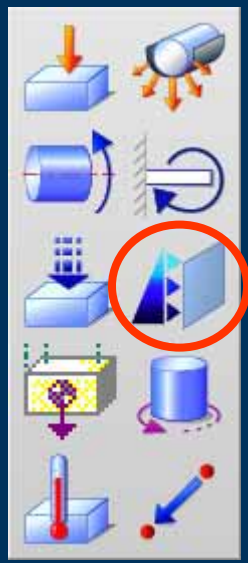



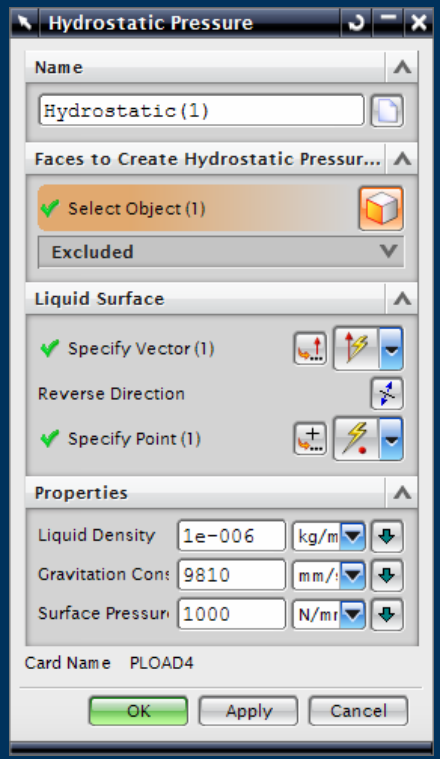




- Pressure Load Options
  - Normal to 3D faces
  - Normal to 2D faces only
  - Px, Py, Pz Component Pressure
  - Px, Py, Pz Component Pressure on Beams
- Managed in the Load Container

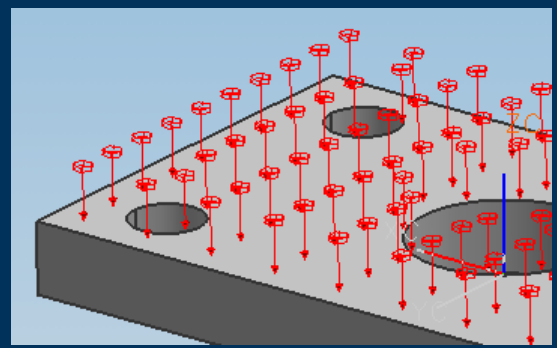


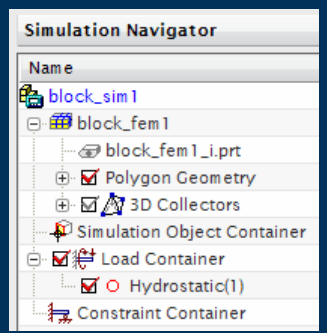
# Loads – Hydrostatic Pressure



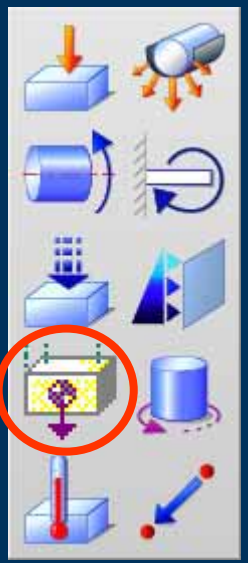


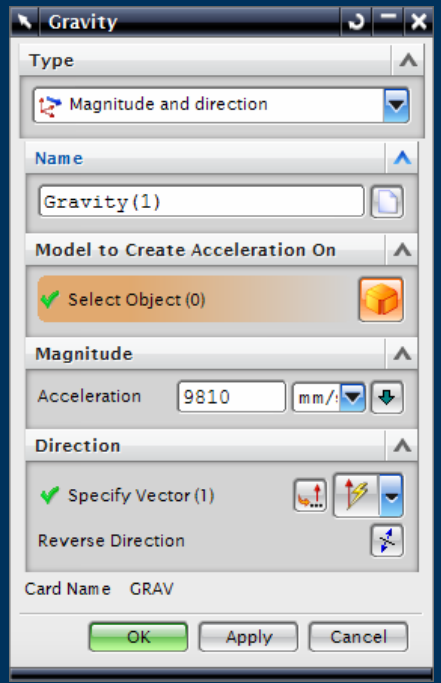

- Hydrostatic Pressure
  - Distributed pressure across selected faces
- Managed in the Load Container

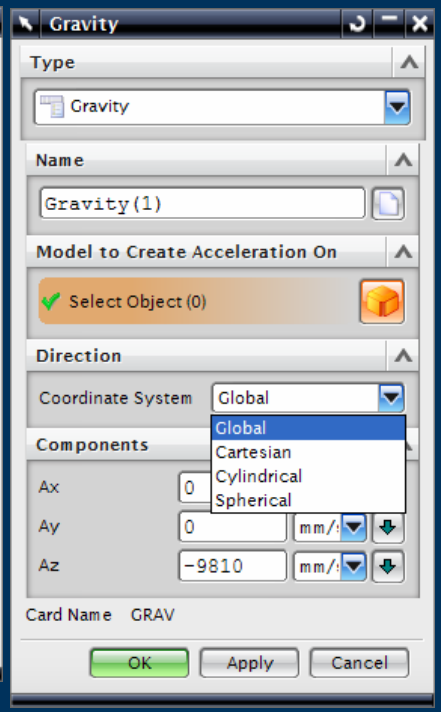


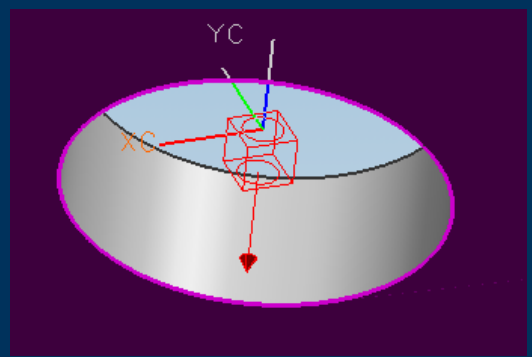


# Loads – Gravity



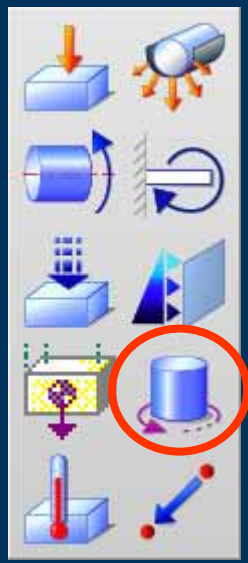


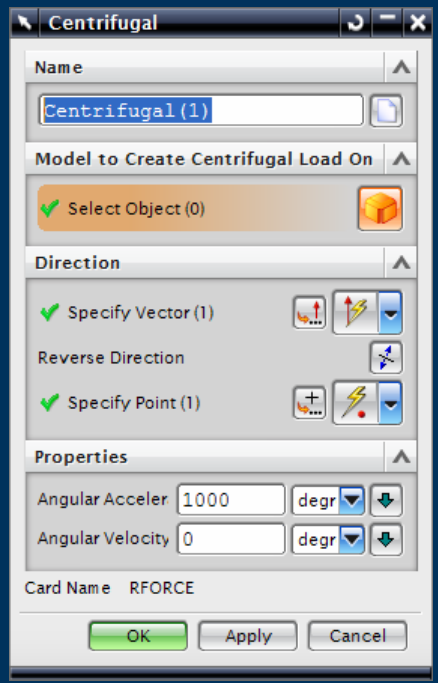



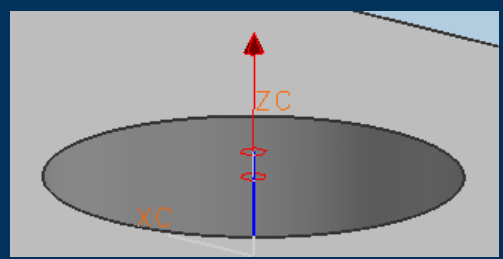

- Gravity Load
  - Applied to complete model
  - Magnitude and Direction
  - Ax, Ay, Az Component Gravity relative to selected coordinate system
- Managed in the Load Container

# Loads – Centrifugal



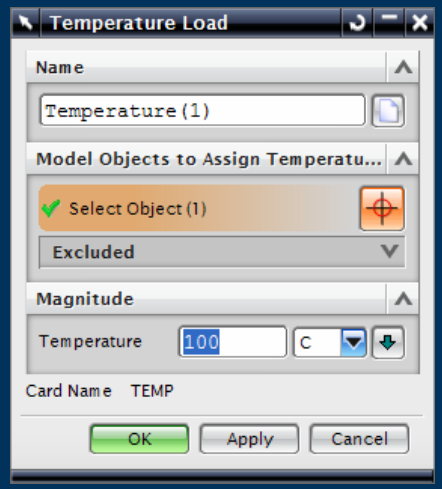



- Centrifugal Load
  - Applied to complete model
  - Direction & centre of rotation
  - Angular Acceleration
  - Angular Velocity
- Managed in the Load Container

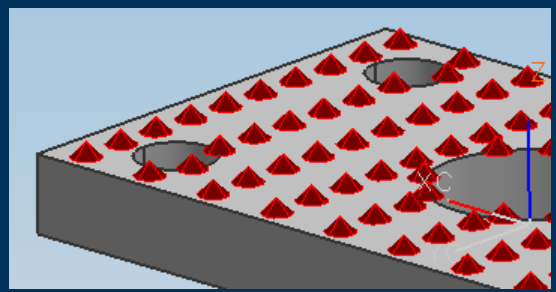


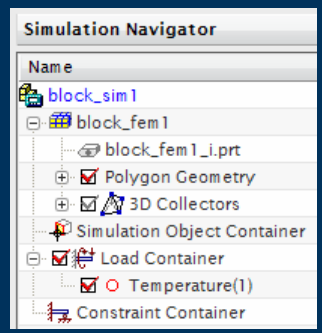
SIEMENS

# Loads – Constant Temperature

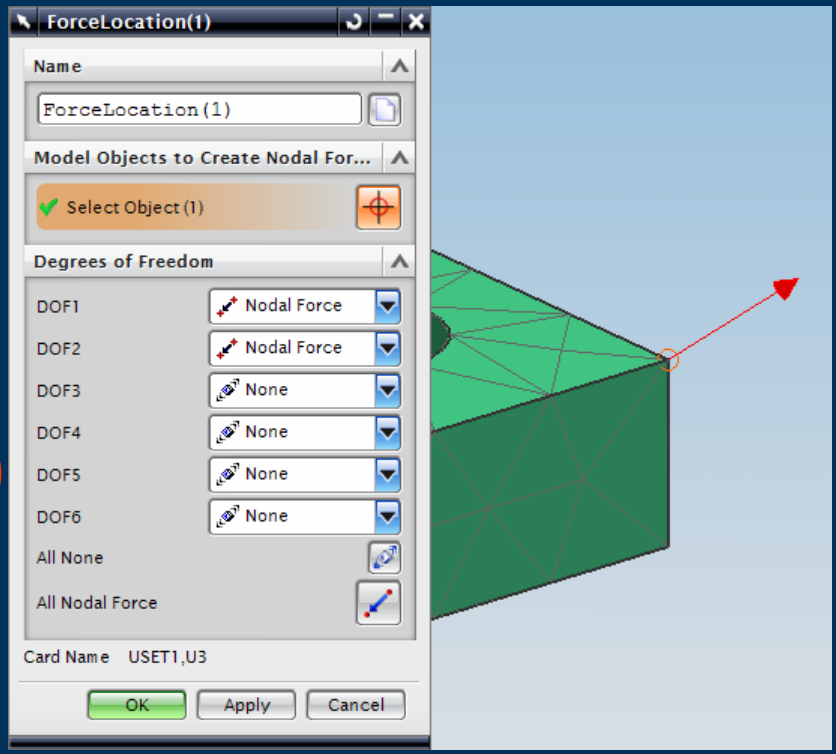



- Constant Temperature Load
  - Applied to curves, edges or faces
- Managed in the Load Container

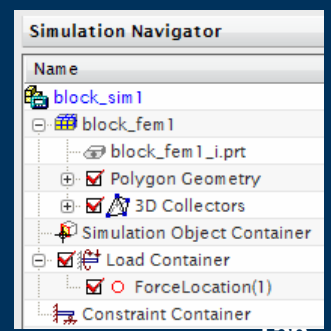




# Nodal Force Location

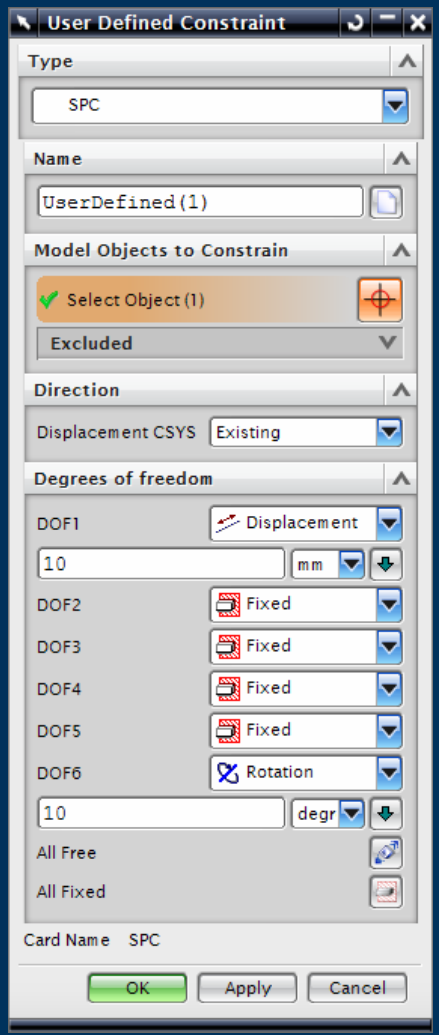


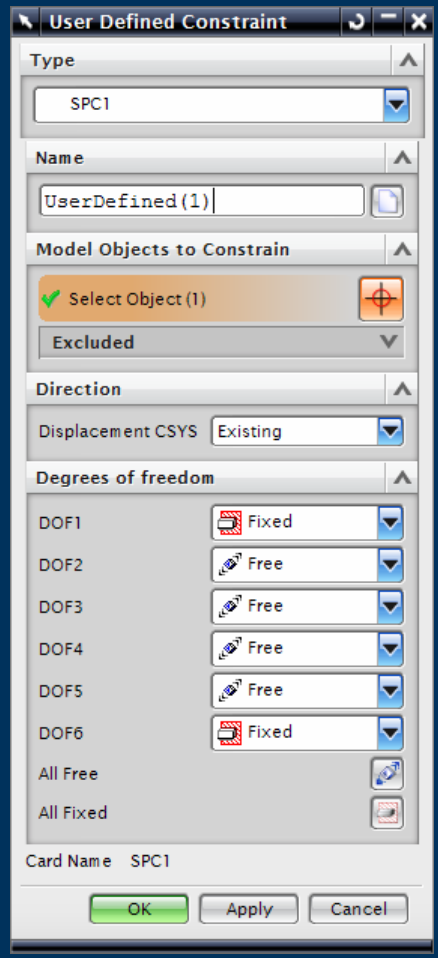

- Location for a Nodal Force Excitation for the NX Response Simulation application
- Requires a matching Dynamic Load
- Managed in the Load Container



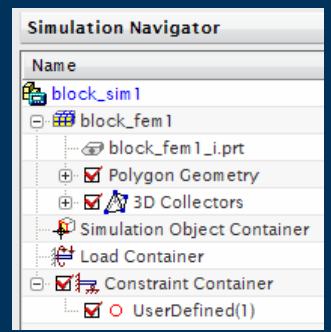
SIEMENS

# Constraints – User Defined

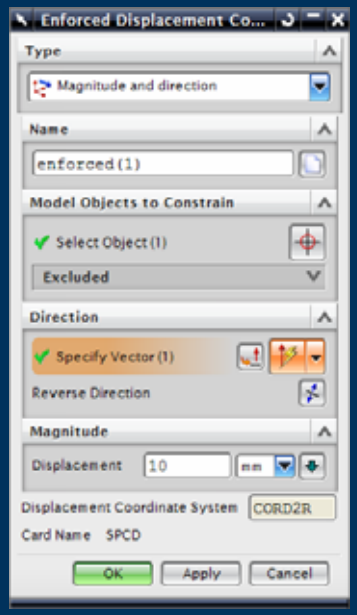


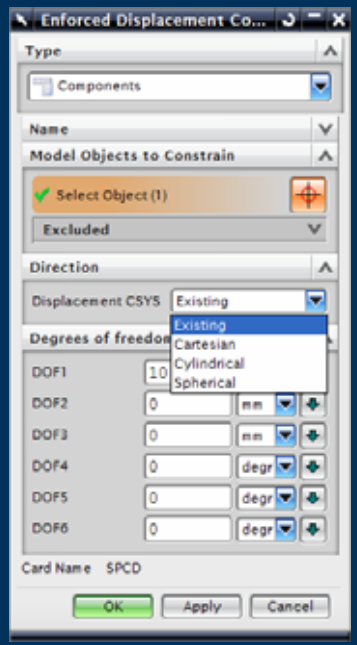



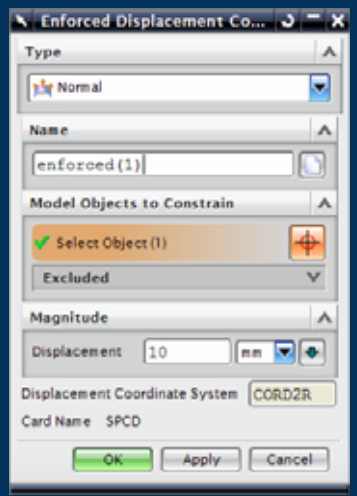

- User Defined Constraints
  - Free, Fixed or Displacement
  - Cartesian, Cylindrical or Spherical coordinate system
- Managed in Constraint Container



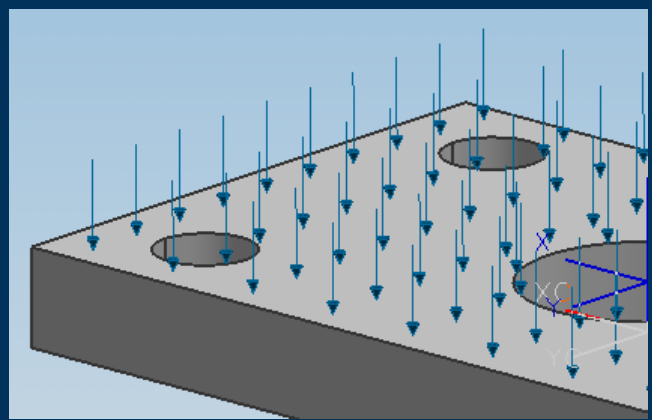
# Constraints – Enforced Displacement

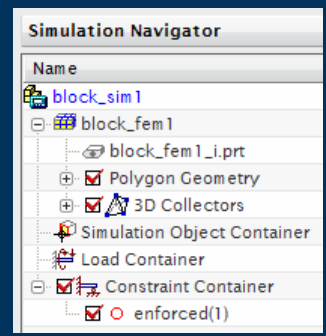


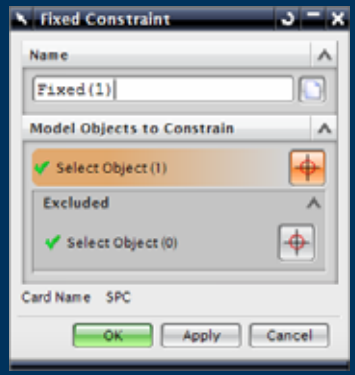



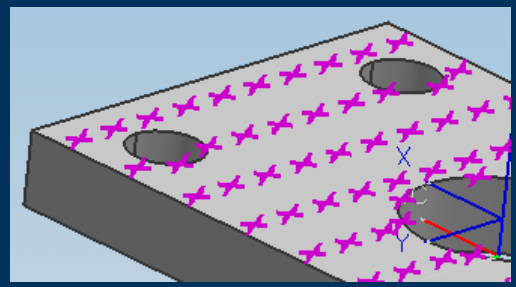

- Enforced Displacement Options
  - Magnitude and direction
  - Normal to selected faces
  - Component Displacement relative to selected coordinate system
- Managed in the Constraint Container

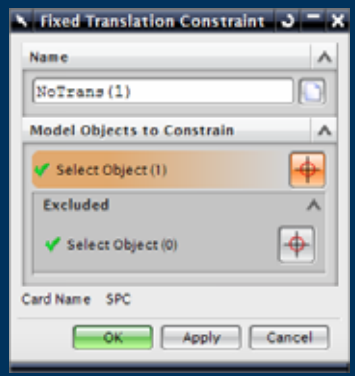


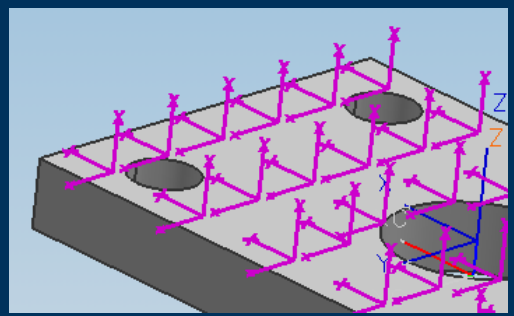


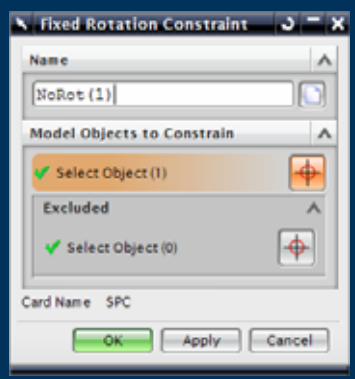
# Constraints – Fixed, Translation & Rotation

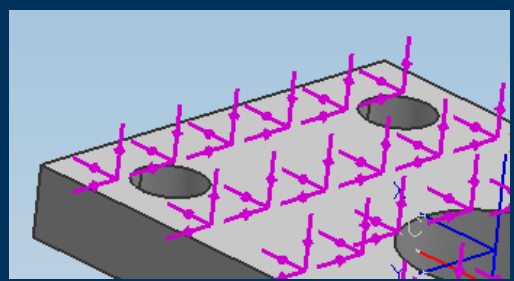


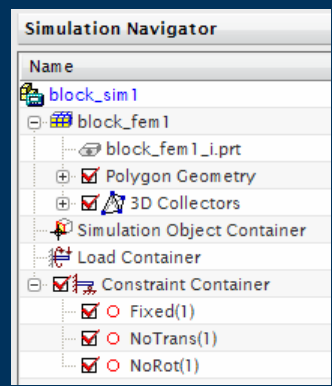


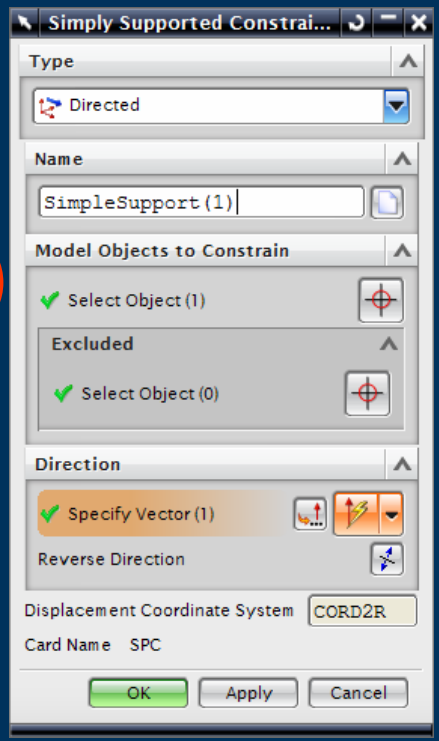


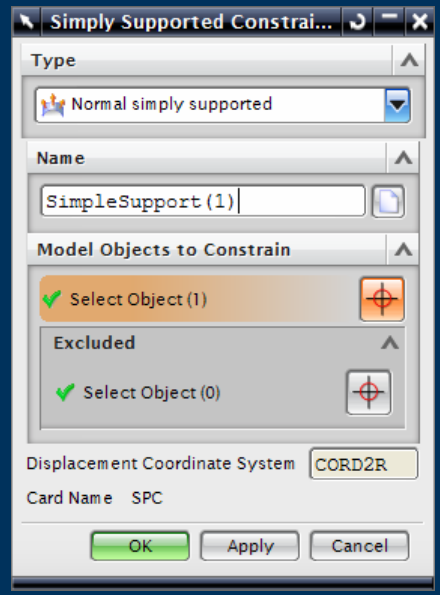



- Fixed Constraints (Restraints)
  - All DOF
  - No Translation
  - No Rotation
- Managed in the Constraints Container

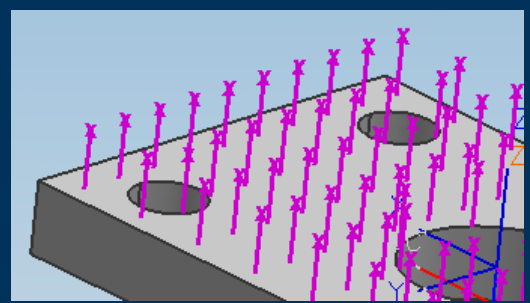


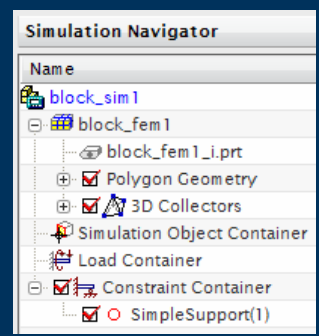
# Constraints – Simply Supported



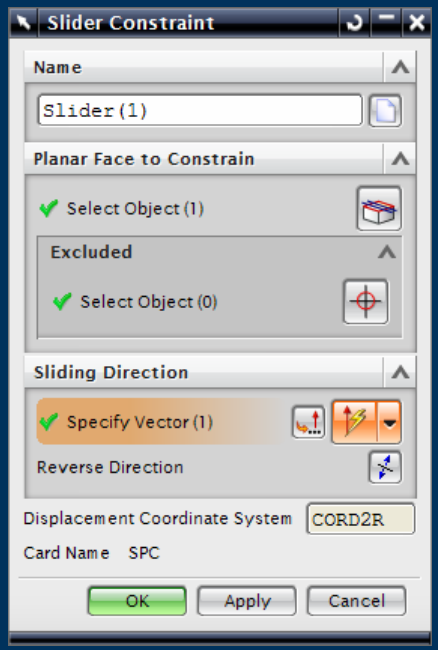




- Simply Supported Constraint
  - Magnitude and Direction of Support
  - Normal to selected surfaces
  - Managed in the Constraint Container

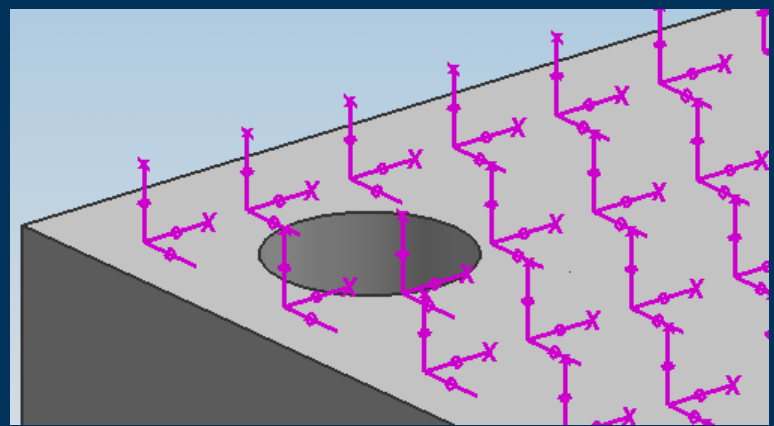


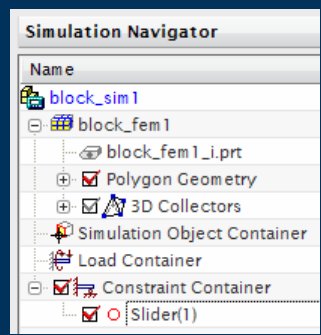


# Constraints – Slider

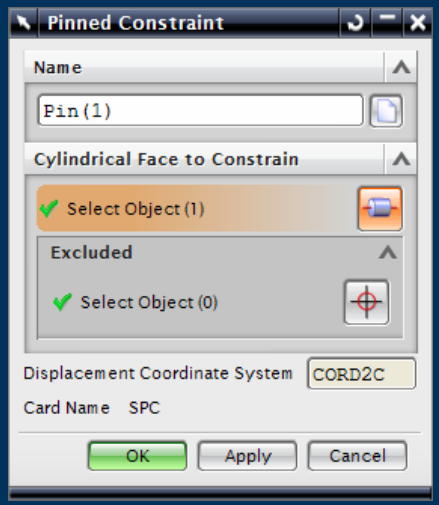



- Slider constraint
  - Planar sliding face
  - Sliding direction
- Managed in the Constraint Container

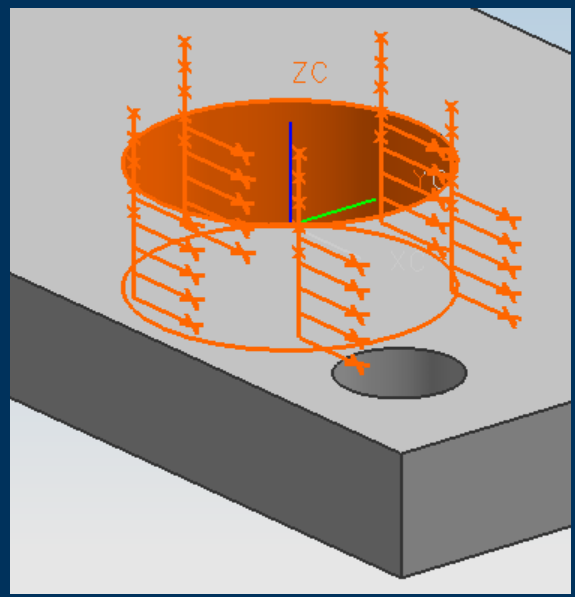


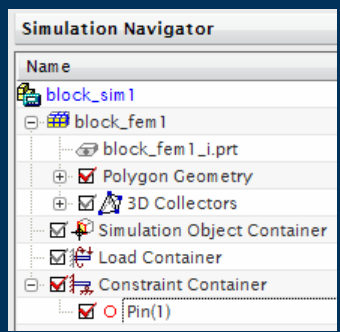


# Constraints – Pinned

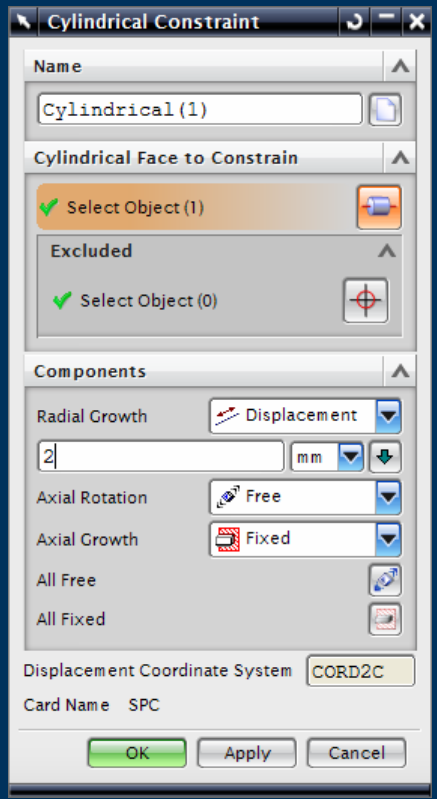



- Pinned Constraint for Cylindrical Surfaces
- Managed in the Constraint Container

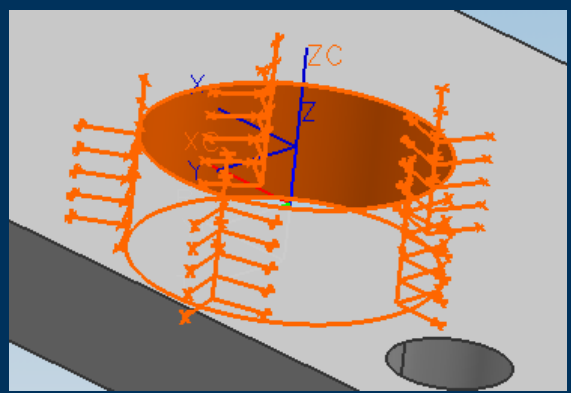


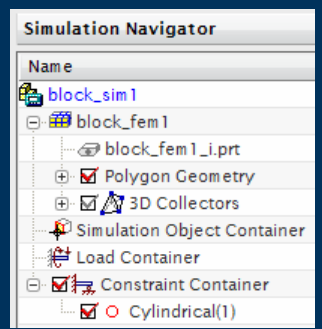



# Constraints – Cylindrical

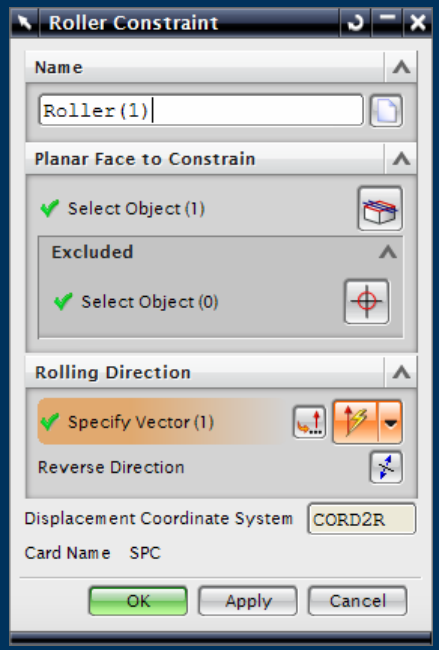



- Cylindrical Constraint
  - Radial Growth
  - Axial Rotation
  - Axial Growth
  - Relative to selected cylindrical surface
- Managed in the Constraint Container

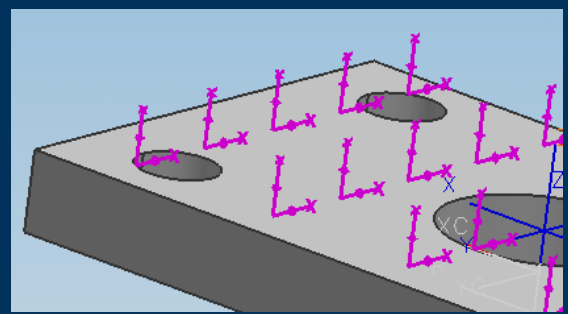


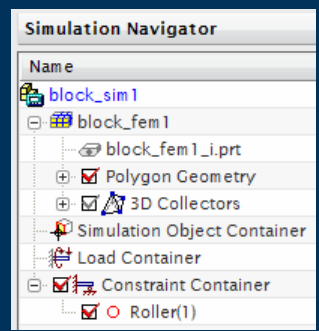



# Constraints – Roller

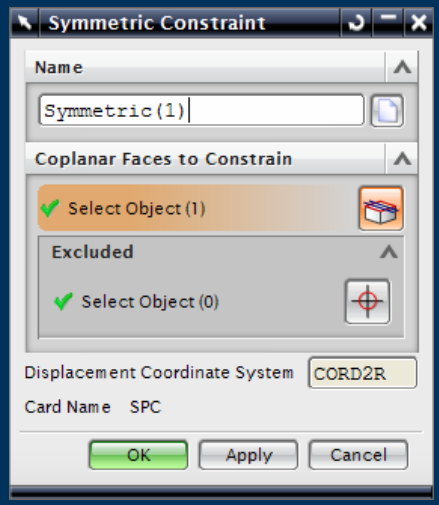



- Roller Constraint
- Managed in the Constraint Container

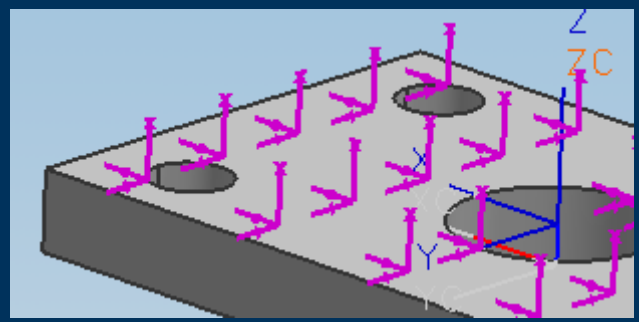


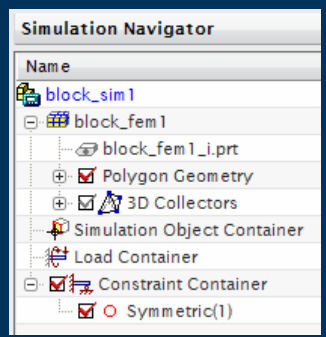


# Constraints – Symmetric

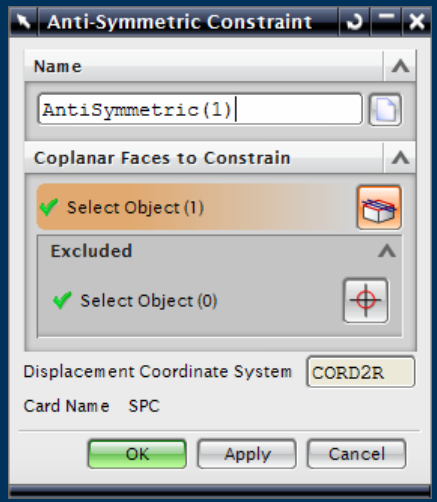



- Symmetric Constraint
- Managed in the Constraint Container

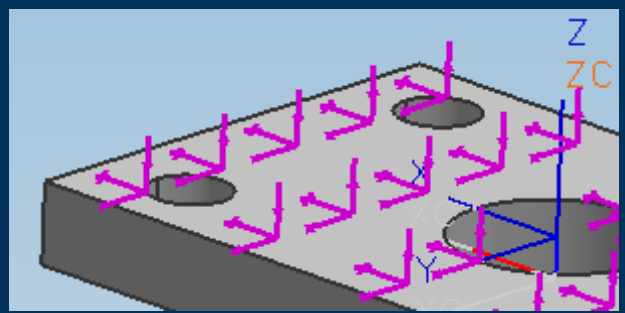


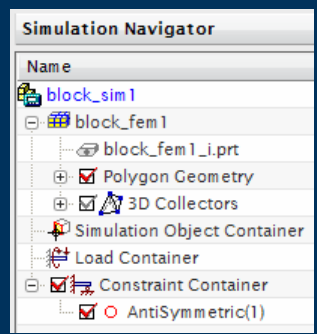



# Constraints – Anti-Symmetric

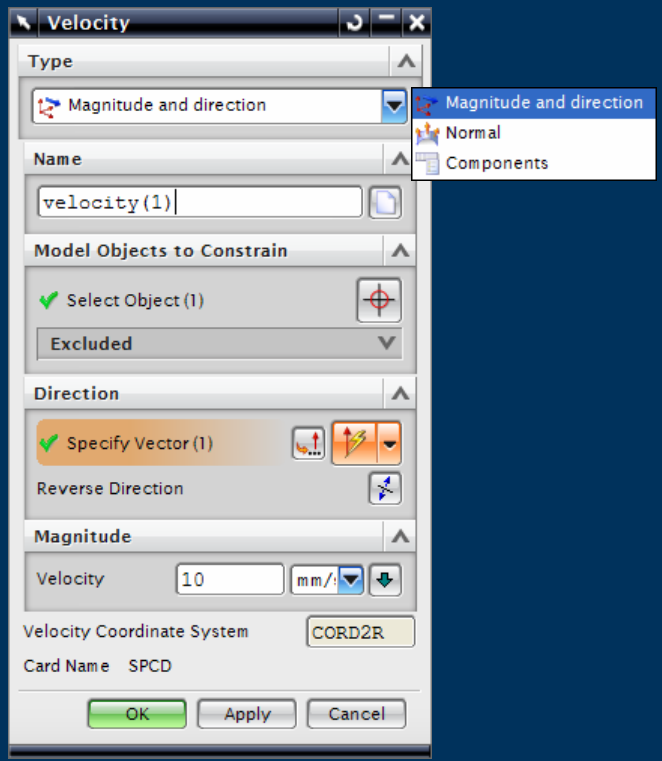



- Anti-Symmetric Constraint
- Managed in the Constraint Container

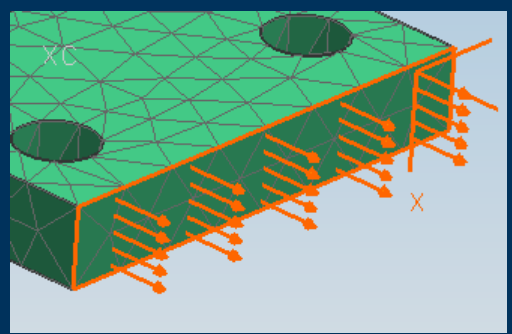


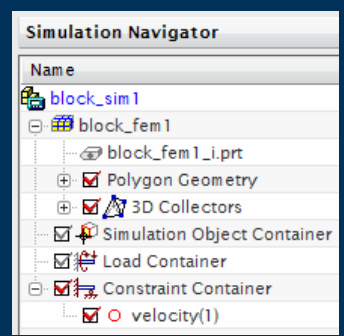



# Constraints – Velocity

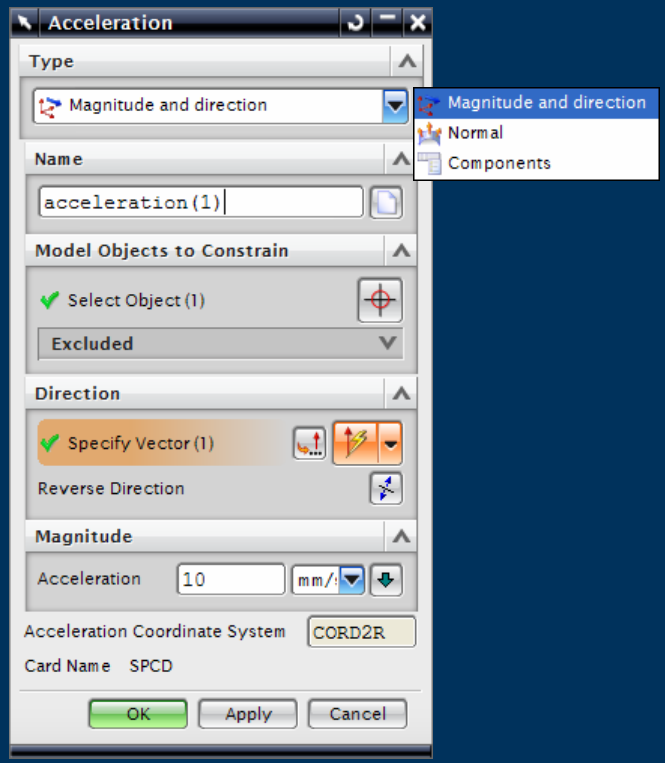



- Velocity Constraint
- Specific to these NX Nastran Solutions
  - SEDFREQ 108 — Direct Frequency Response
  - SEDTRAN 109 — Direct Transient Response
  - SEMFREQ 111 — Modal Frequency Response
  - SEMTRAN 112 — Modal Transient Response
- Managed in the Constraint Container

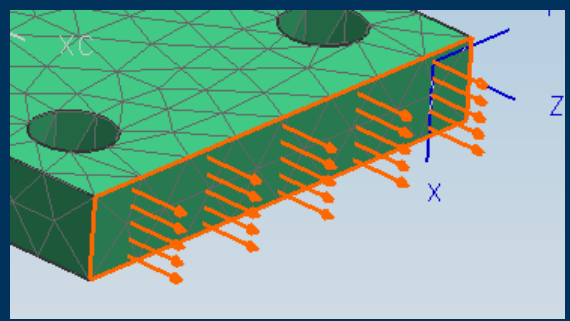


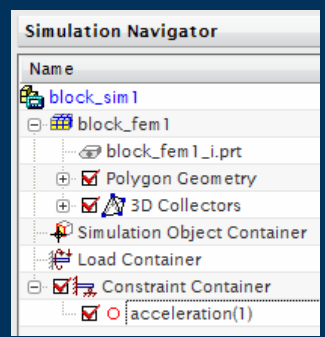



# Constraints – Acceleration

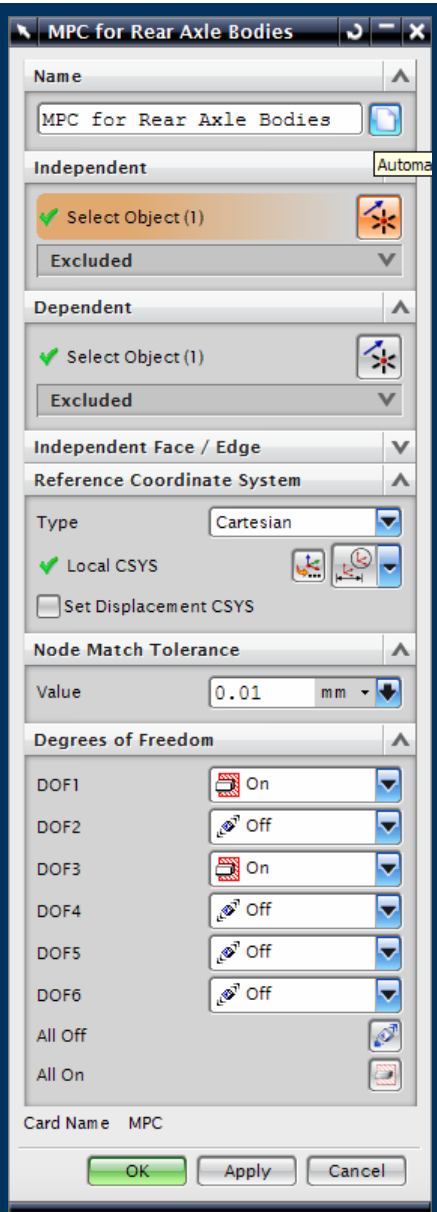



- Velocity Constraint
- Specific to these NX Nastran Solutions
  - SEDFREQ 108 — Direct Frequency Response
  - SEDTRAN 109 — Direct Transient Response
  - SEMFREQ 111 — Modal Frequency Response
  - SEMTRAN 112 — Modal Transient Response
- Managed in the Constraint Container

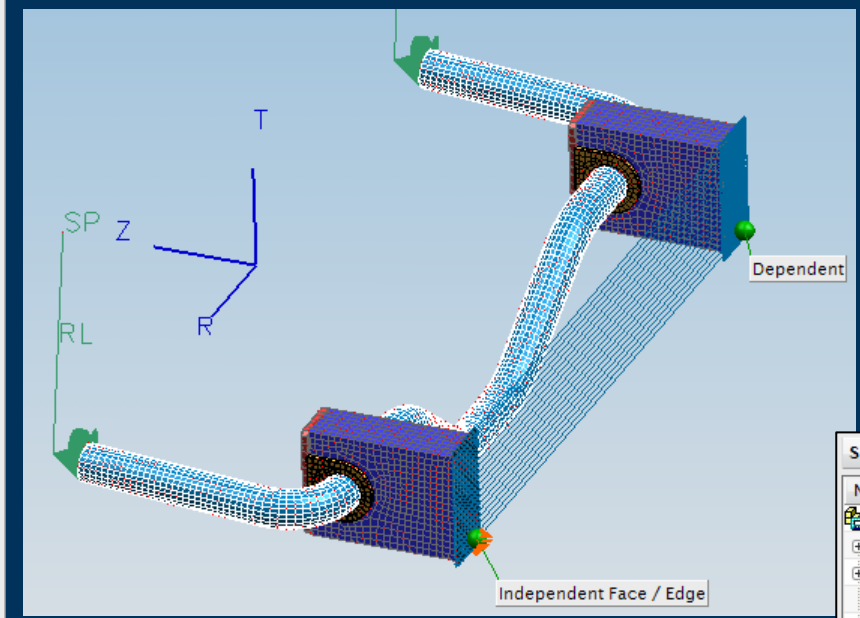


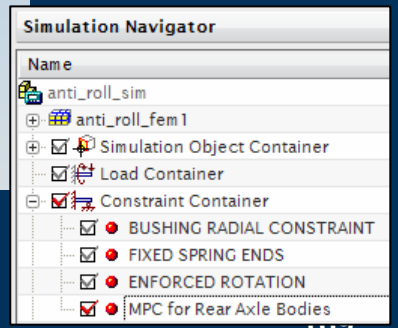



# Constraints – Automatic Coupling

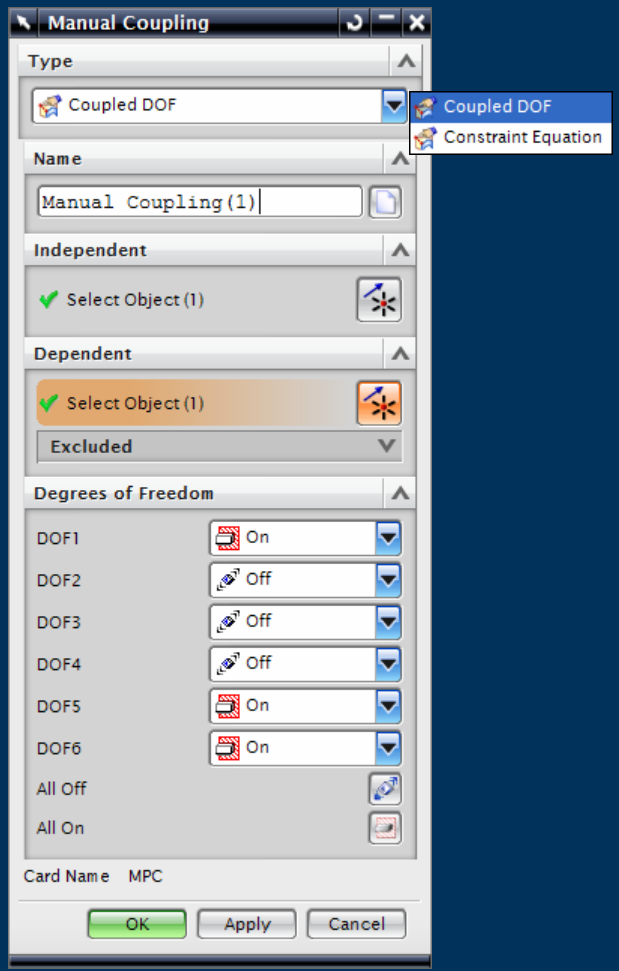



- Coupled degrees of freedom between offset or symmetric meshes
- Managed in the Constraint Container

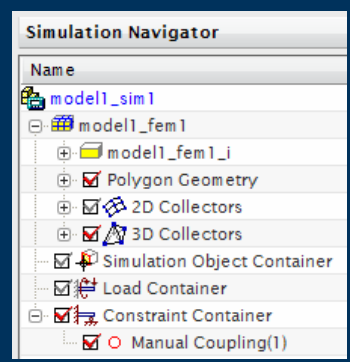


# Constraints – Manual Coupling

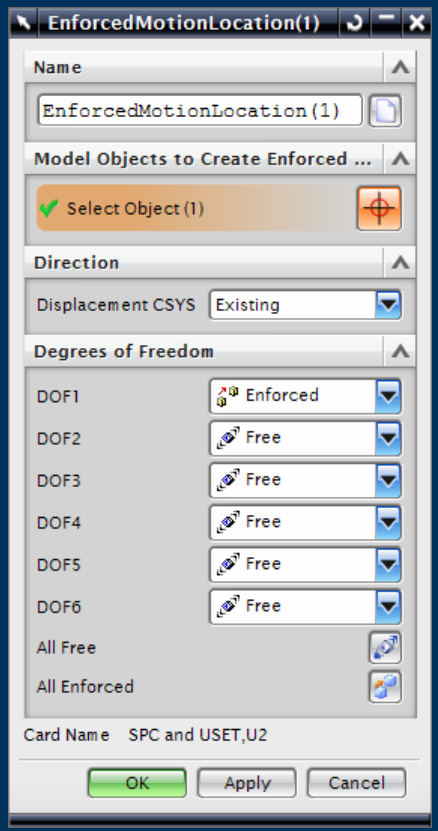



- Create either Coupled DOF or Constraint Equations between selected nodes
- Managed in the Constraint Container

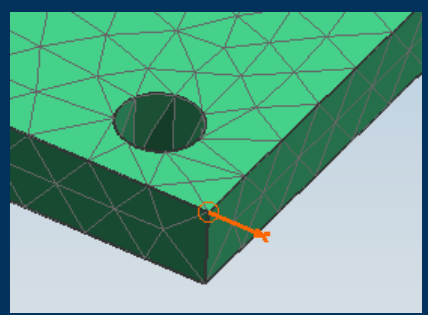


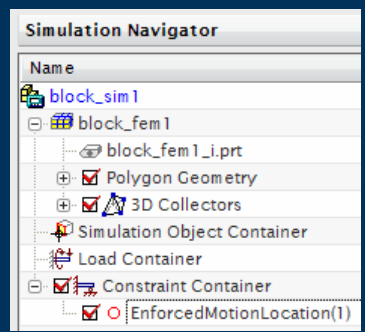
SIEMENS

# Constraints – Enforced Motion Location



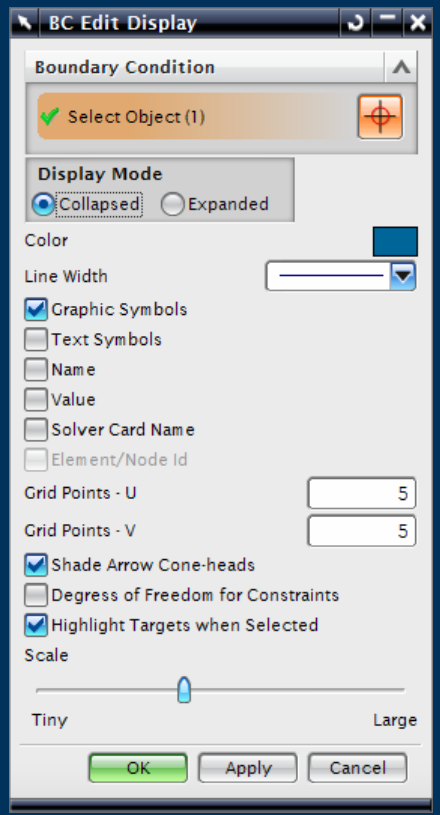

- Enforced Motion Location Constraint
- Specific to these NX Nastran Solutions
  - SEDFREQ 108 — Direct Frequency Response
  - SEDTRAN 109 — Direct Transient Response
  - SEMFREQ 111 — Modal Frequency Response
  - SEMTRAN 112 — Modal Transient Response
- Managed in the Constraint Container



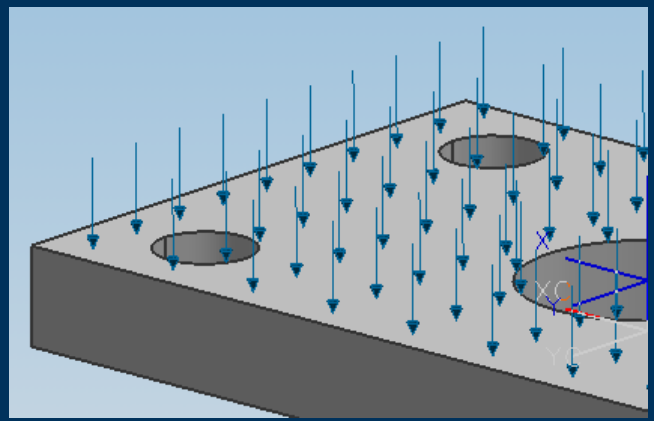


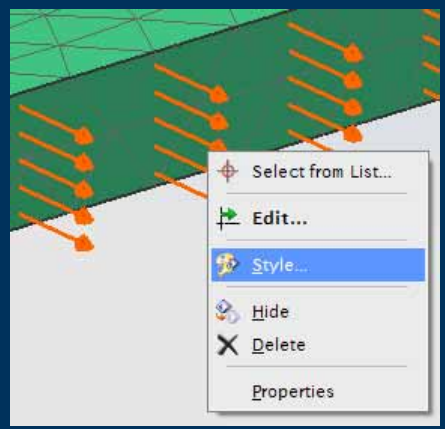
SIEMENS

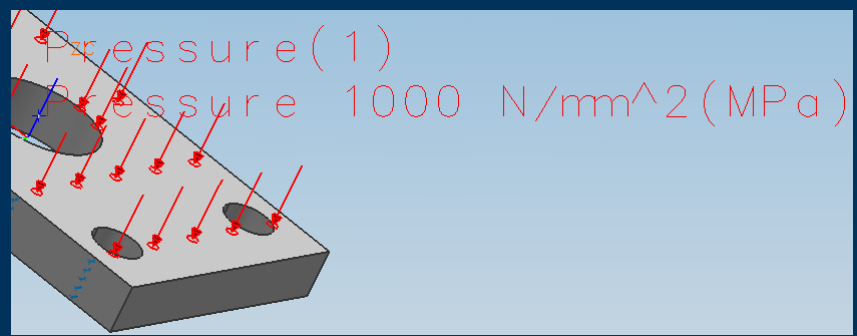
# Boundary Condition Symbol Display Controls

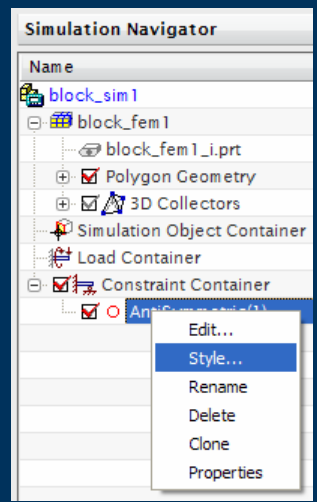



- All Boundary Conditions have a Symbol associated and the Style can be changed

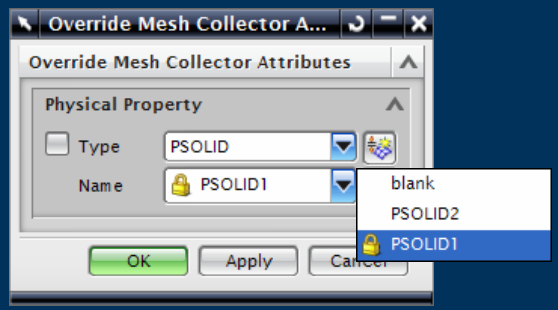


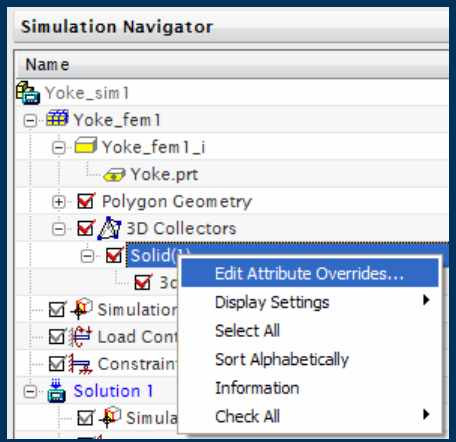


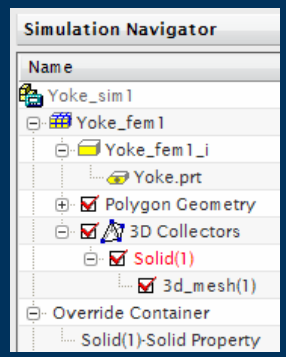



# Physical Property Overrides



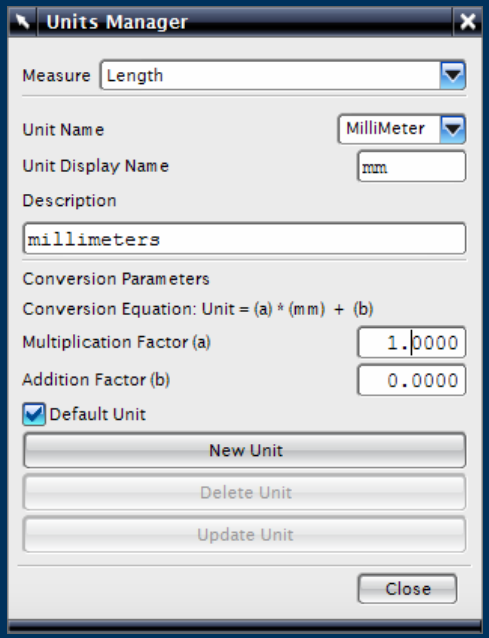



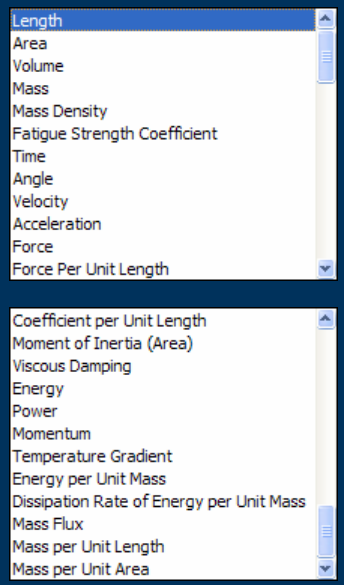



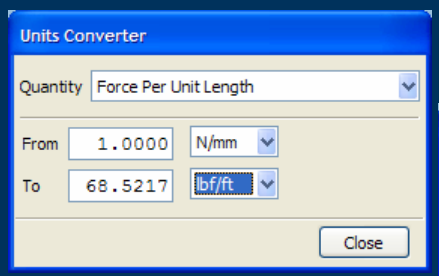

- Allows the SIM file to override the Physical properties defined in the FEM file
- Mesh with Overridden property shown in Red
- “What-if” studies

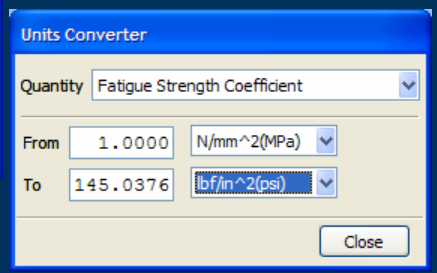
SIEMENS

# Custom Units & Units Converter

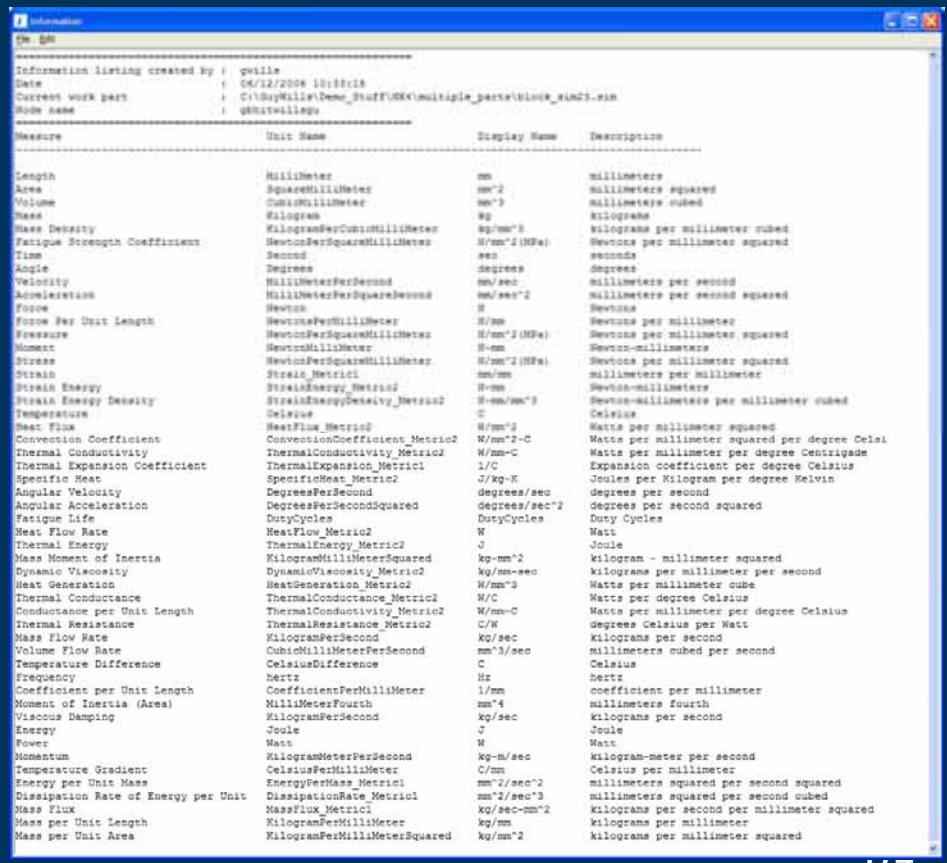


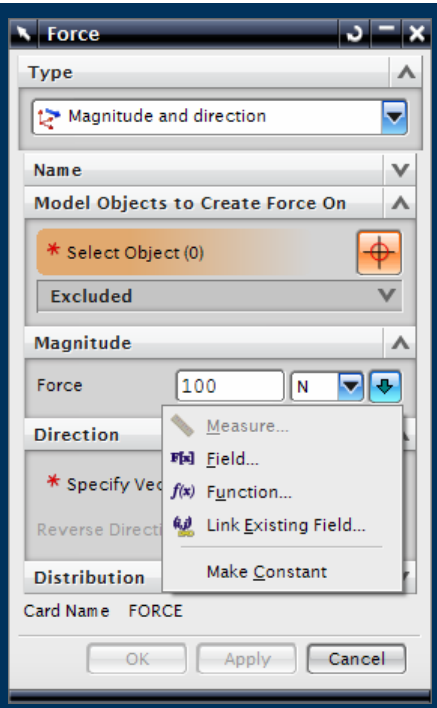




- Units Manager create Custom Units for various Measures
- Units Converter

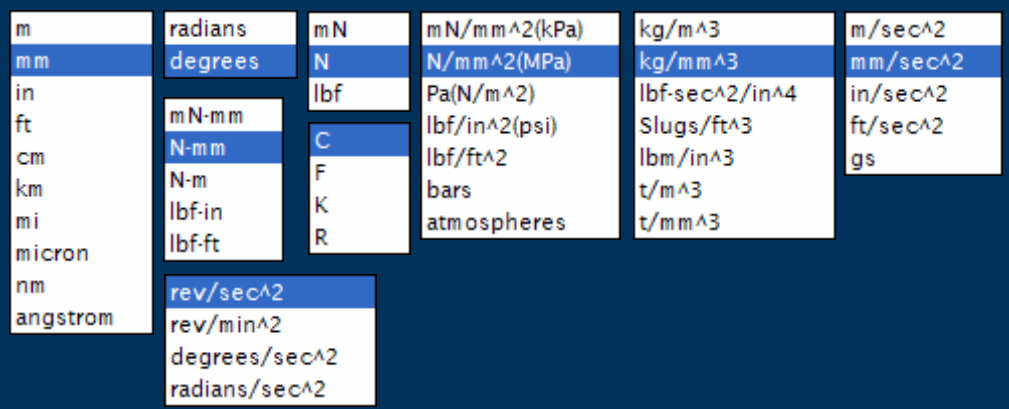


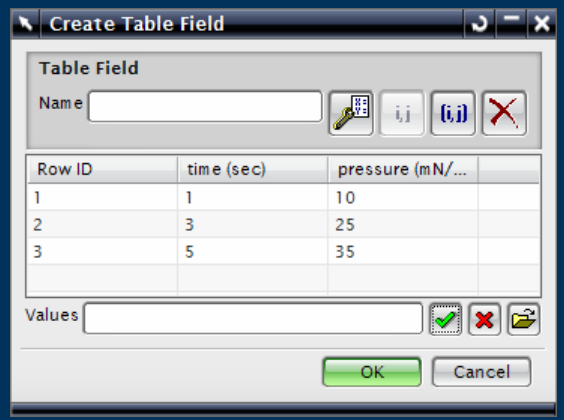
# Unit Selection



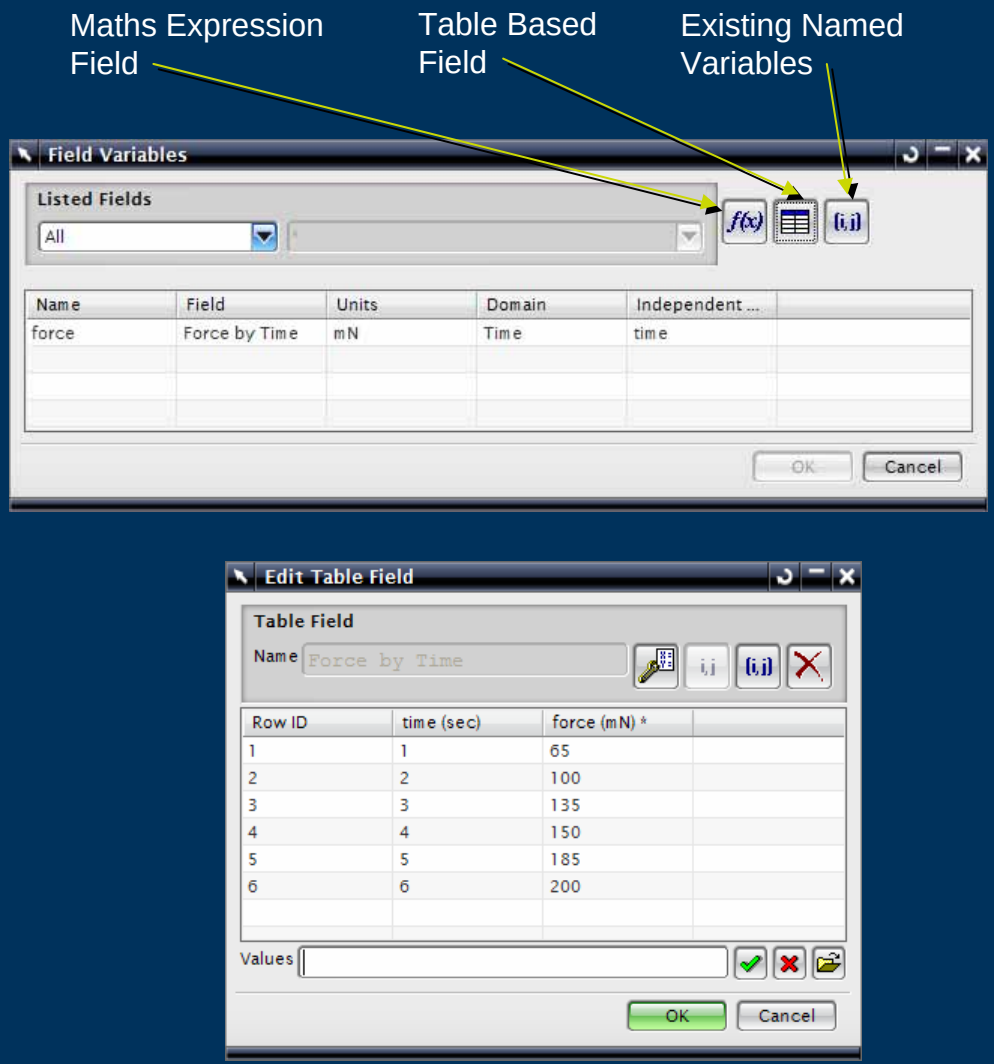

- Numeric entry for a value can be entered in different units
  - Measure
    - "on the fly" measurement from existing geometry
  - Field
    - Define the magnitude as a constant or variable (eg time dependant)
  - Function
    - Define the magnitude as a function that calculates a single value
  - Link Existing Field
    - Link to an existing Field Variable
  - Make Constant
    - Converts an expression to a constant value

Examples



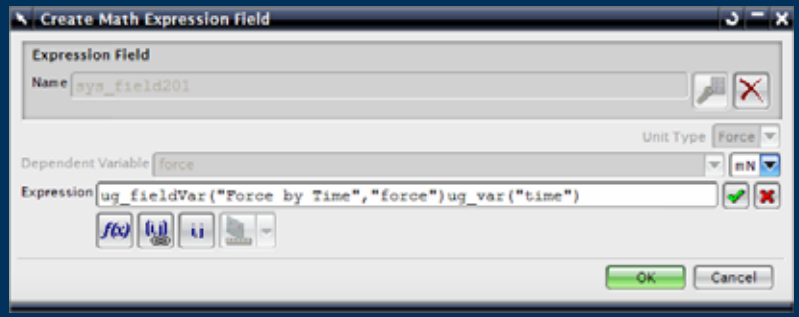
# Boundary Condition Magnitude – Table Field

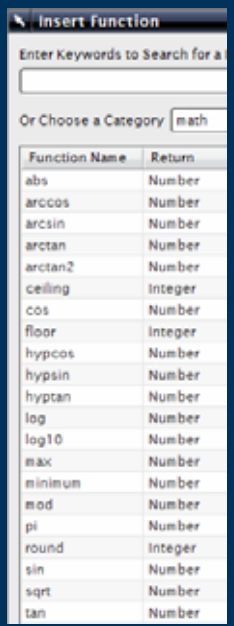


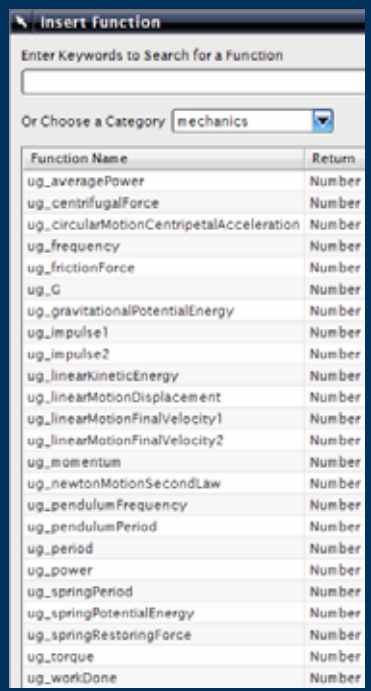

- Tables Field
  - User selected Dependant and Independent columns

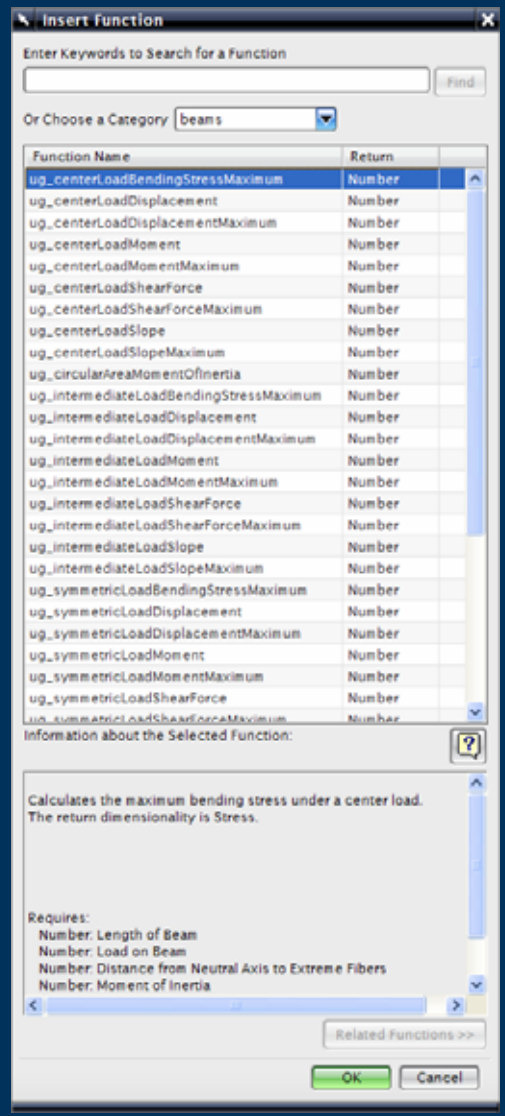
SIEMENS

# Boundary Condition Magnitude – Function Field




- Pre-defined functions that calculate a value
- Categories
  - Beams
  - Fluids
  - Gears
  - Geometry
  - Materials
  - Maths
  - Mechanics
  - Misc
  - O Rings
  - Plate
  - Spreadsheet
  - Spring
  - String
  - Units
  - Vibration

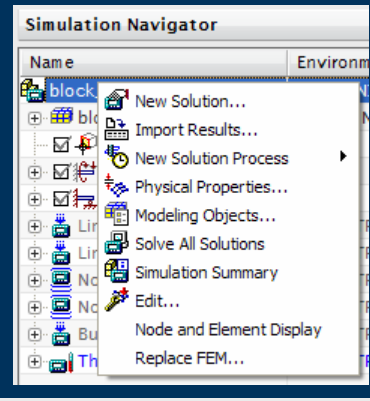
# Solution

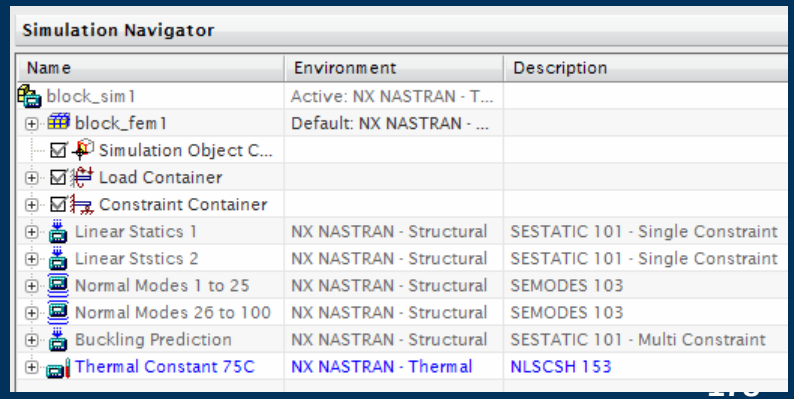

| Solver | Analysis Type | Solution Type |
|---|---|---|
| NX Nastran | Structural | SESTATIC 101 - Single Constraint |
| | | SESTATIC 101 - Multiple Constraint |
| | | SEMODES 103 |
| | | SEMODES 103 Response - Simulation |
| | | SEBUCKL 105 |
| | | NLSTATIC 106 |
| | | SEDFREQ 108 |
| | | SEDTRAN 109 |
| | | SEMFREQ 111 |
| | | SEMTRAN 112 |
| | | ADVNL 601, 106 |
| | | ADVNL 601, 129 |
| | Thermal | NLSCSH 153 |
| | Axisymmetric Structural | SESTATIC 101 - Single Constraint |
| | | SESTATIC 101 - Multiple Constraint |
| | | NLSTATIC 106 |
| | Axisymmetric Thermal | NLSCSH 153 |

| Solver | Analysis Type | Solution Type |
|---|---|---|
| ABAQUS | Structural | General Analysis |
| | Thermal | Heat Transfer |
| | Axisymmetric Structural | General Analysis |
| | Axisymmetric Thermal | Heat Transfer |

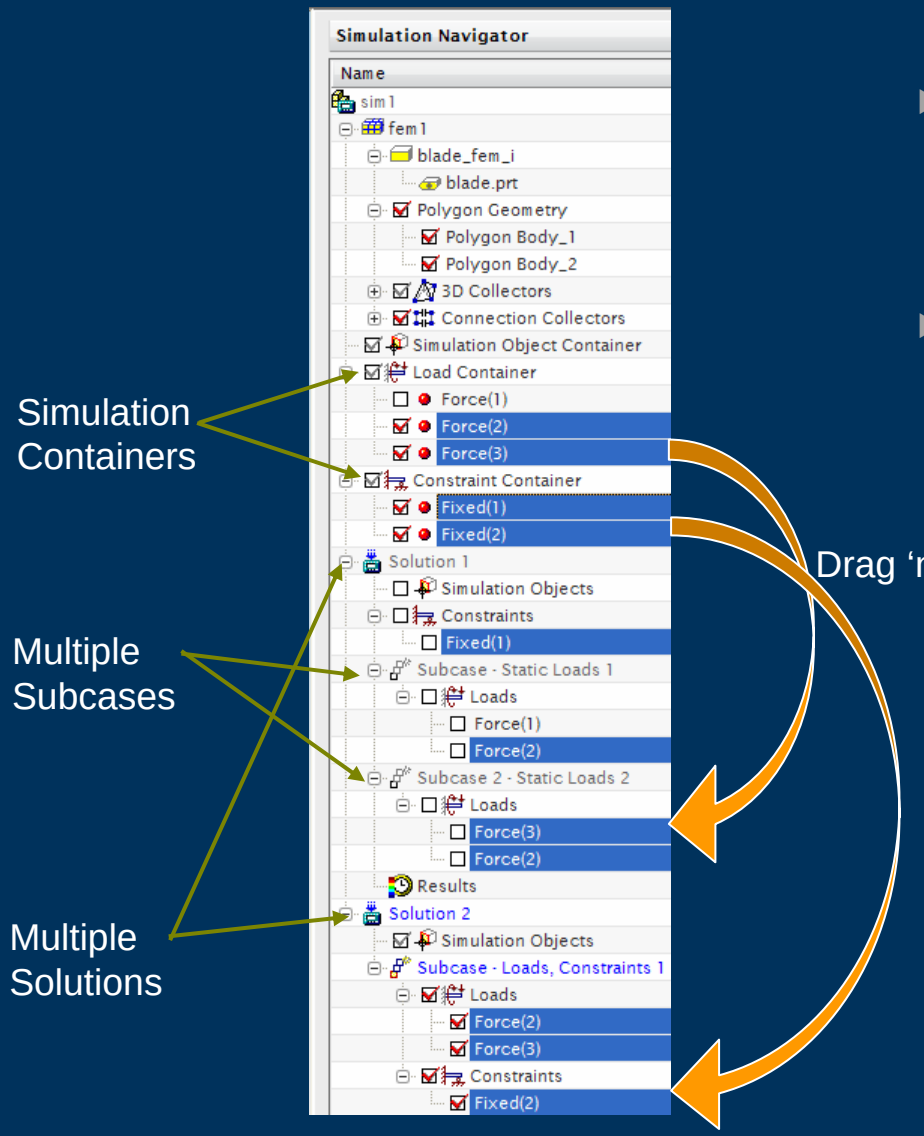
| Solver | Analysis Type | Solution Type |
|---|---|---|
| ANSYS | Structural | Linear Statics |
| | | Modal |
| | | Buckling |
| | | Nonlinear Statics |
| | Thermal | Thermal |
| | Axisymmetric Structural | Linear Statics |
| | | Nonlinear Statics |
| | Axisymmetric Thermal | Thermal |

- Solution is Solver dependant
- Solution gathers everything together to perform a solve
- SIM File can contain many Solutions to study different aspects of the design
- Only one is active



# Solution – Containers and Re-using Data

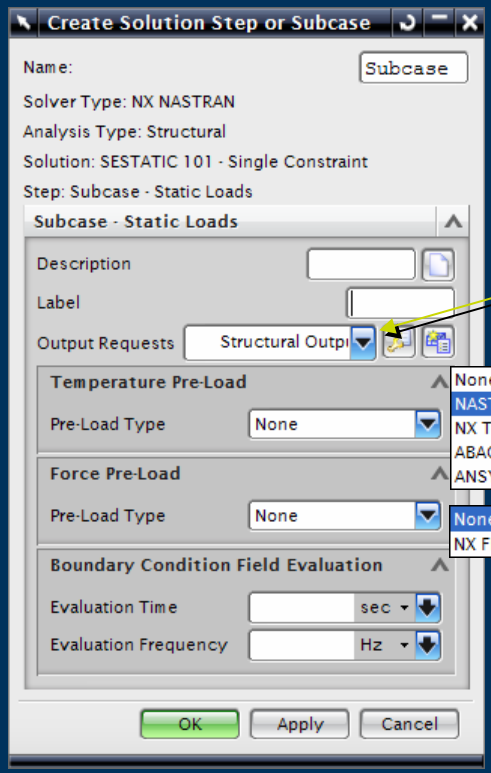



- All Boundary Conditions, constraints etc are stored in Containers
- They are then referenced by the Solutions and Subcases

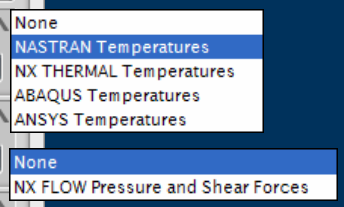
**Benefits**

- Re-use of data
- Quickly and easily explore effects of different loading conditions
- Efficient analysis in complex environments

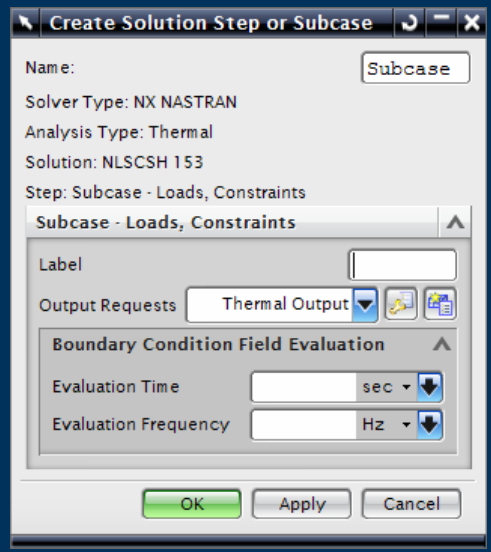
# Solution – Subcase Management

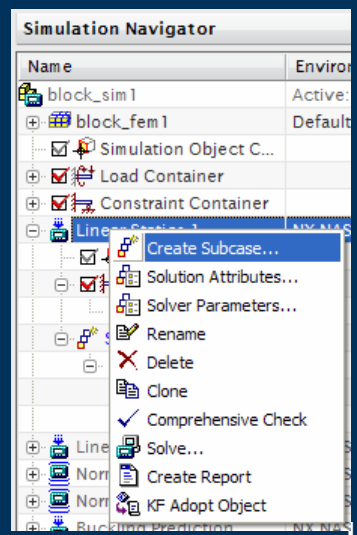



Output Request Object



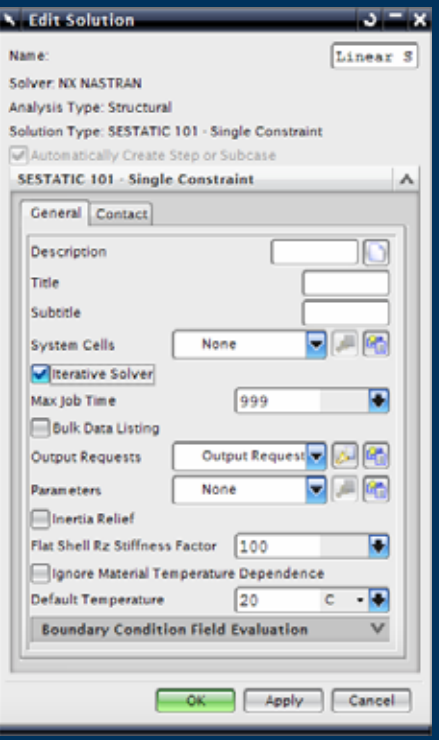

- Subcase availability and options will vary according to the active Solver and Solution type
- Each Solution can have multiple Subcases
- Loads can be used in any combination of Subcases
- Subcases can include Pre-Loads like Thermal results from a previous solve

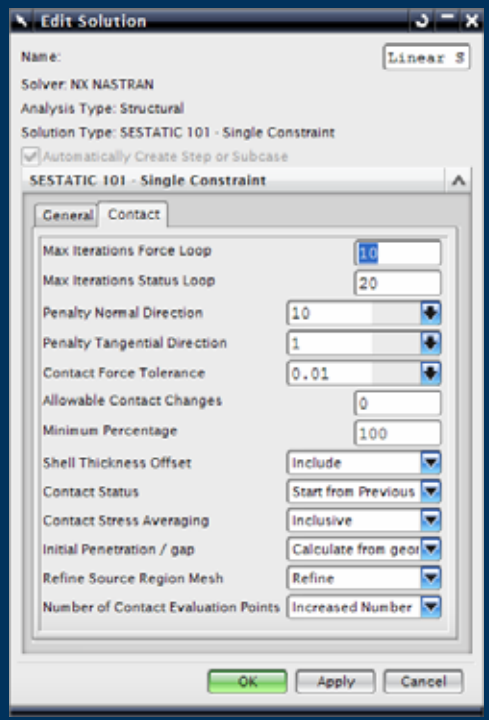


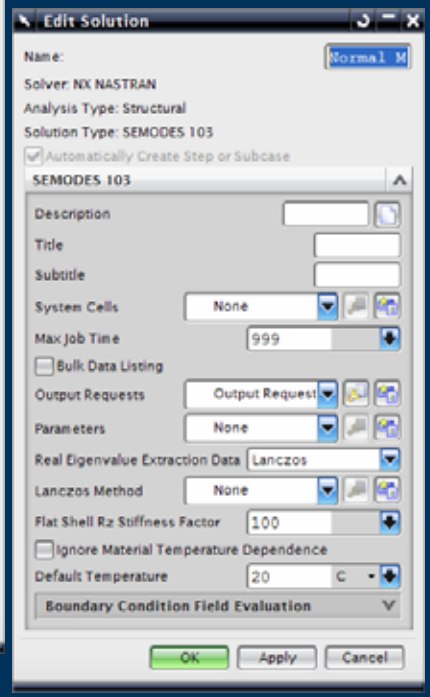



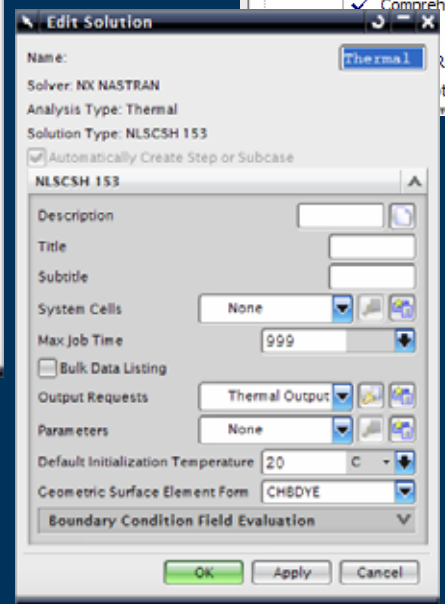
# Solution – Attributes

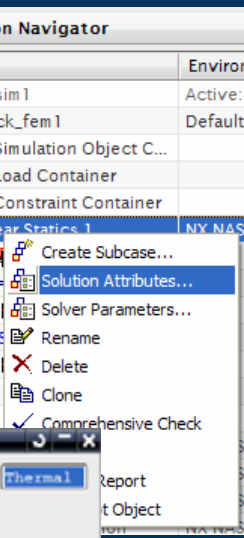
- Solution Attributes availability and options will vary according to the active Solver and Solution type



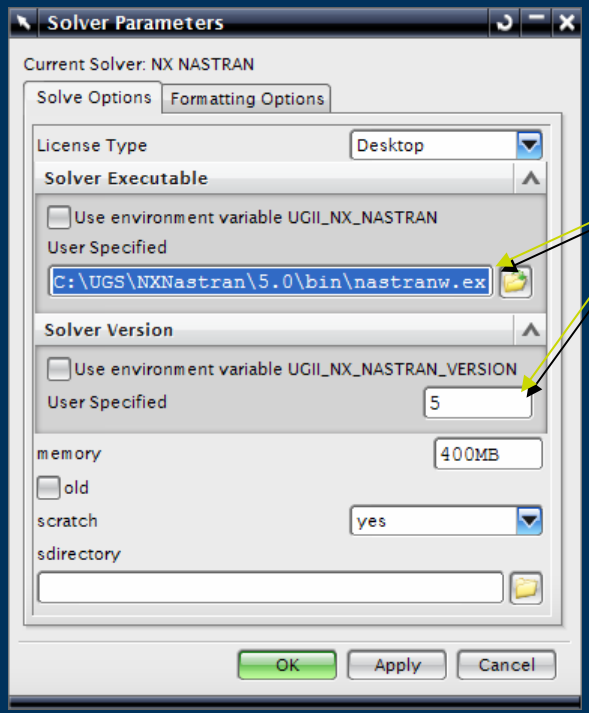



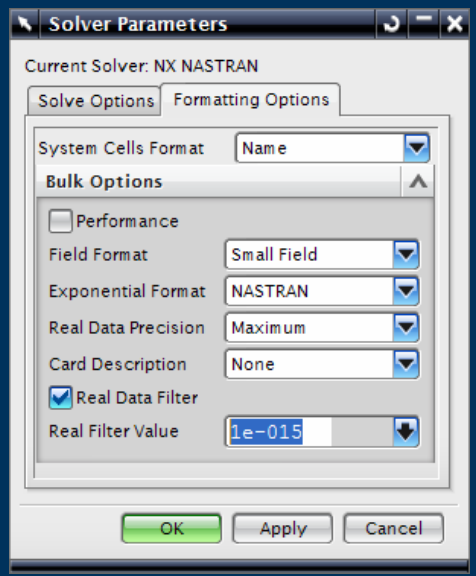



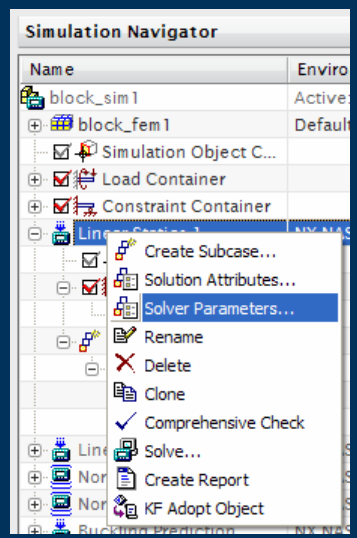
# Solution – Parameters

- Solution Attributes availability and options will vary according to the active Solver and Solution type

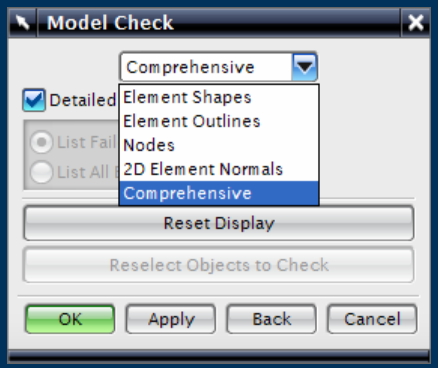


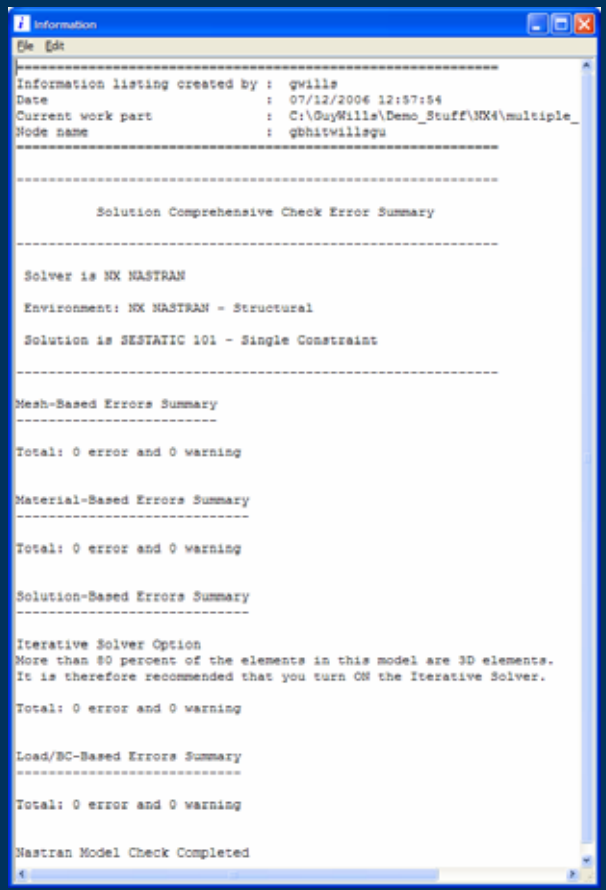

Exact Version of NX Nastran



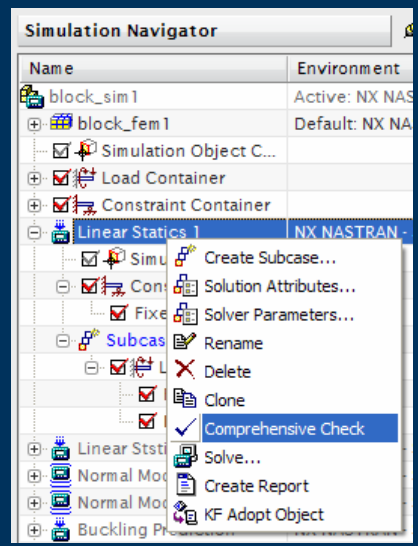



# Solution – Comprehensive Check



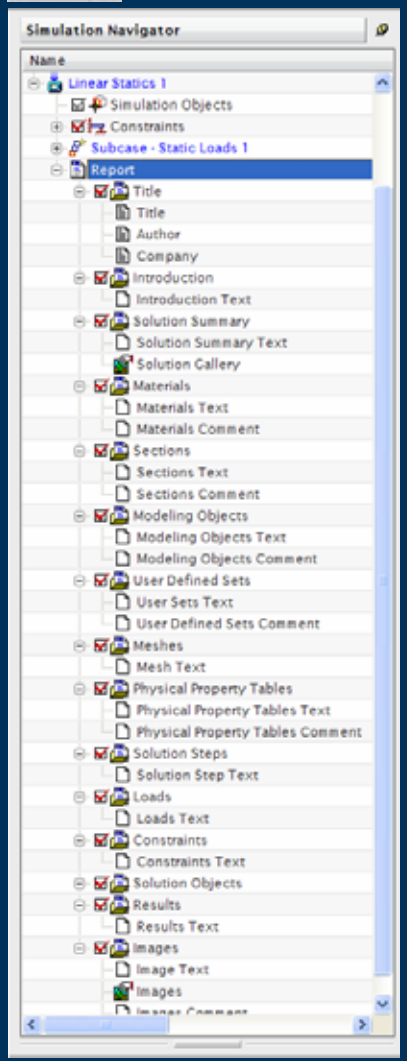



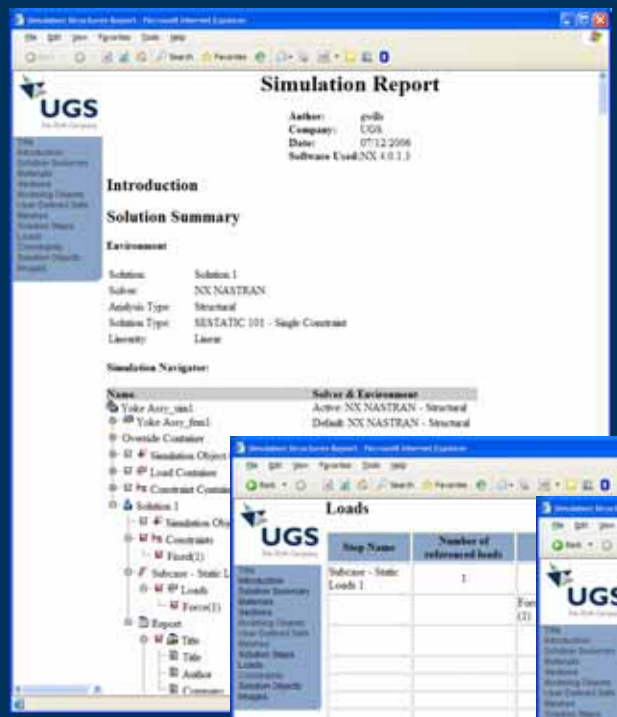

- Solution Comprehensive Check
- Warnings & Errors
  - Mesh
  - Materials
  - Boundary Conditions
  - Solution

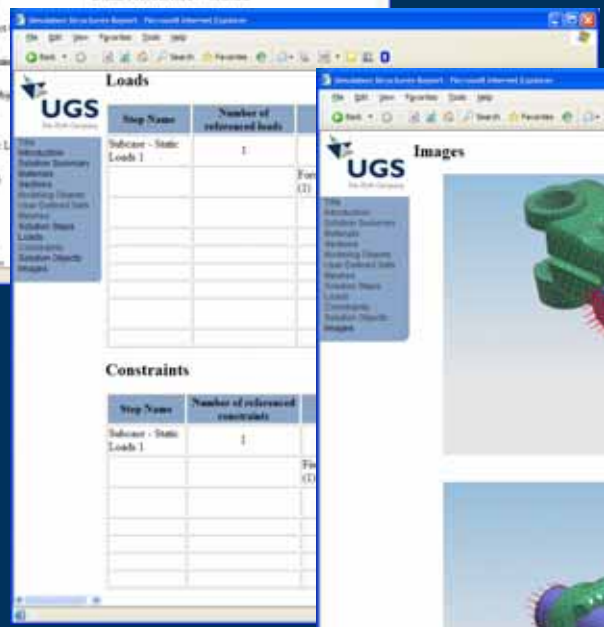


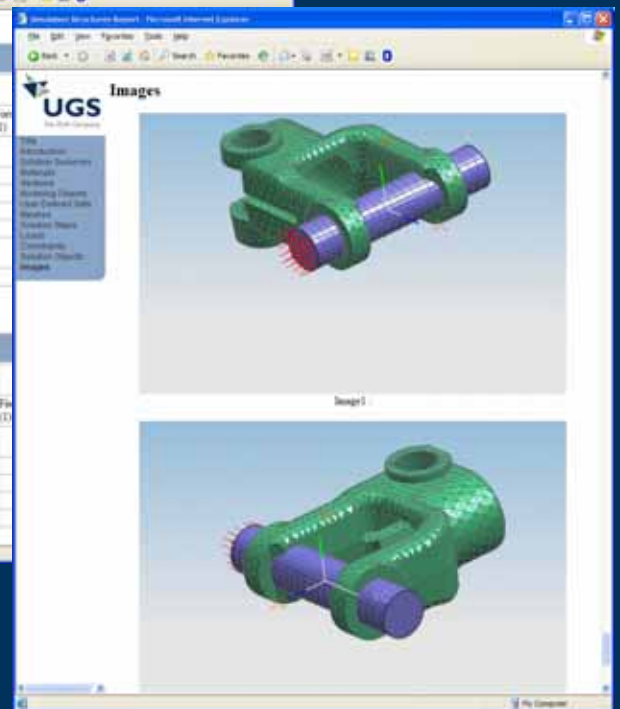
SIEMENS

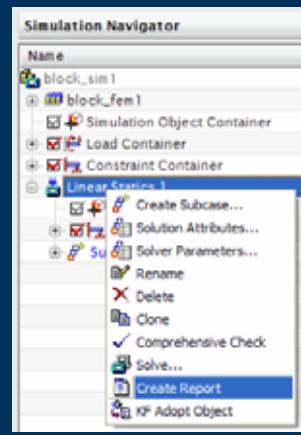
# Solution – Report Before Solve



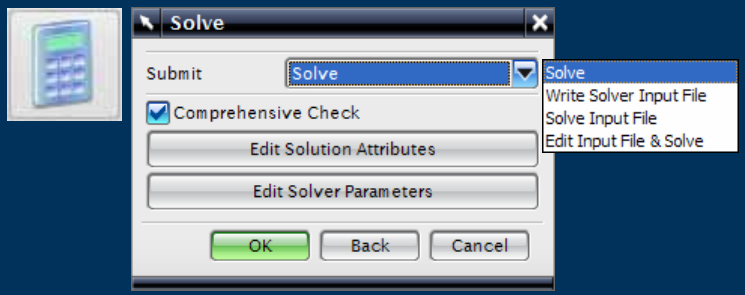



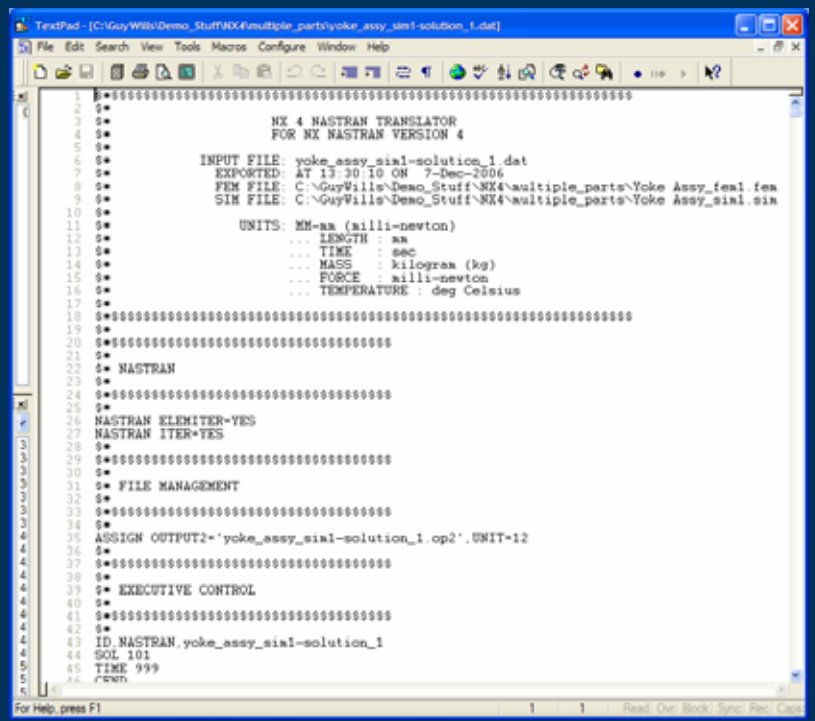



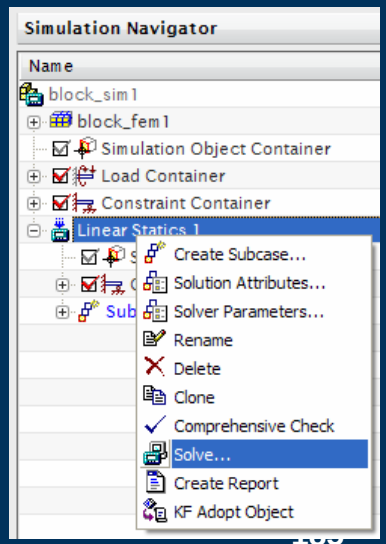

- Solution Report – Before Solve
  - User entered text description/documentation
  - Snapshot screen images
- HTML Interactive report export

# Solution – Solve the Active Solution




- Solve the Active Solution options
  - Solve interactively (eg using NX Nastran Desktop)
  - Write (export) the solver input file for further job editing or queue management
  - Solve interactively an existing Input file (ie Restart)
  - Edit the Input file and solve directly (useful at add extra Solver specific entries)



SIEMENS

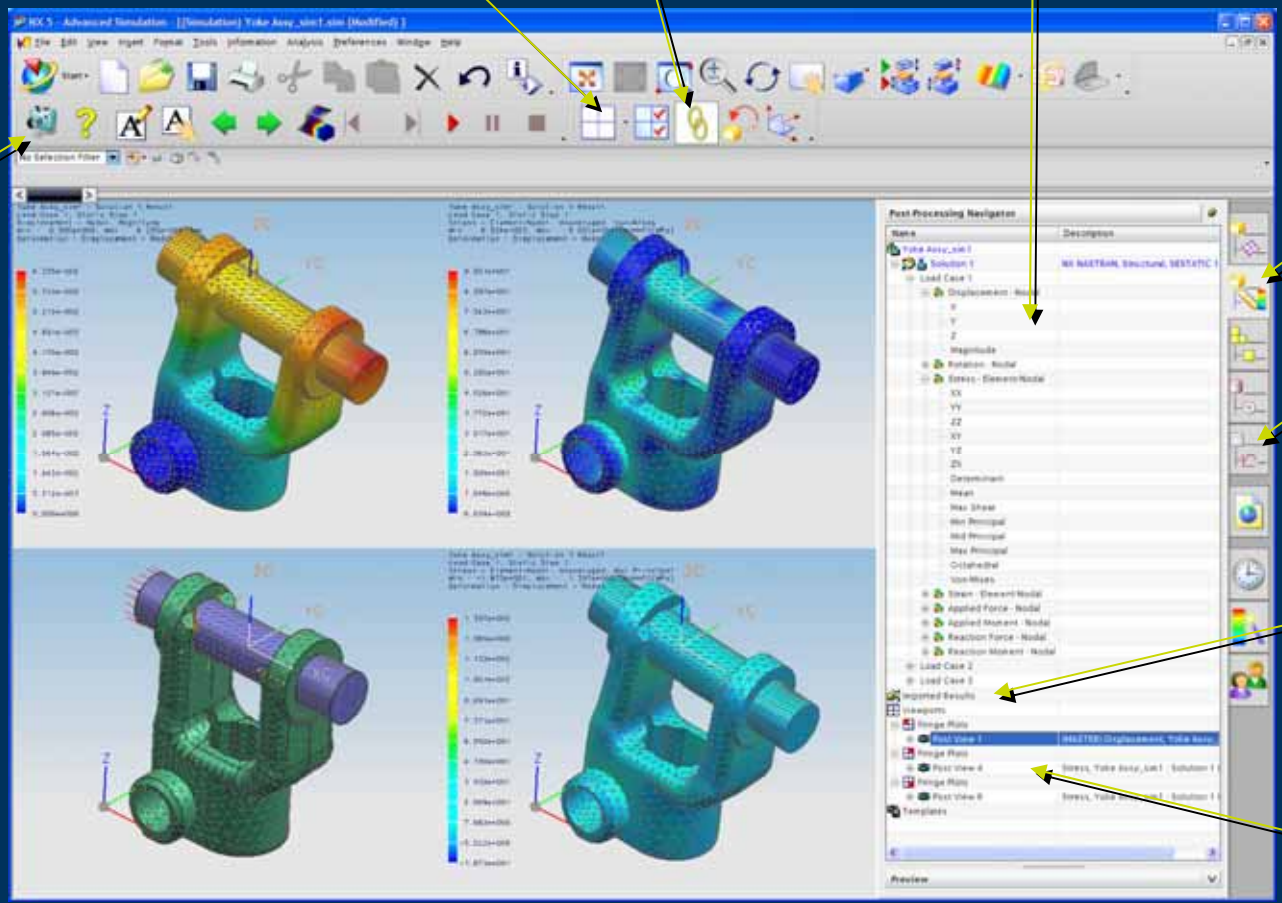
# SIM Part – Post-Processing



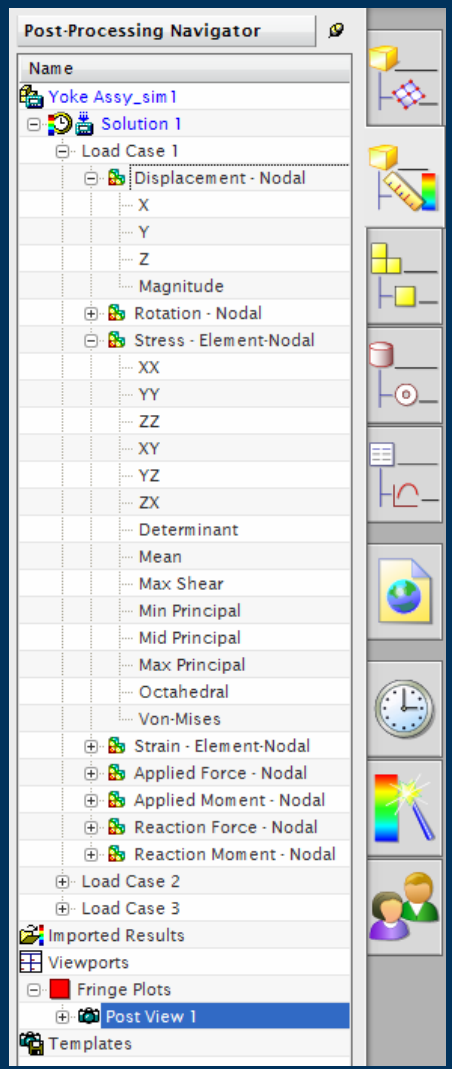
UGS PLM Software

SIEMENS

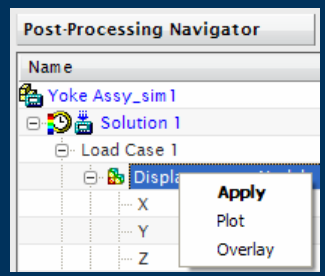
# NX – Integrated Post Processing

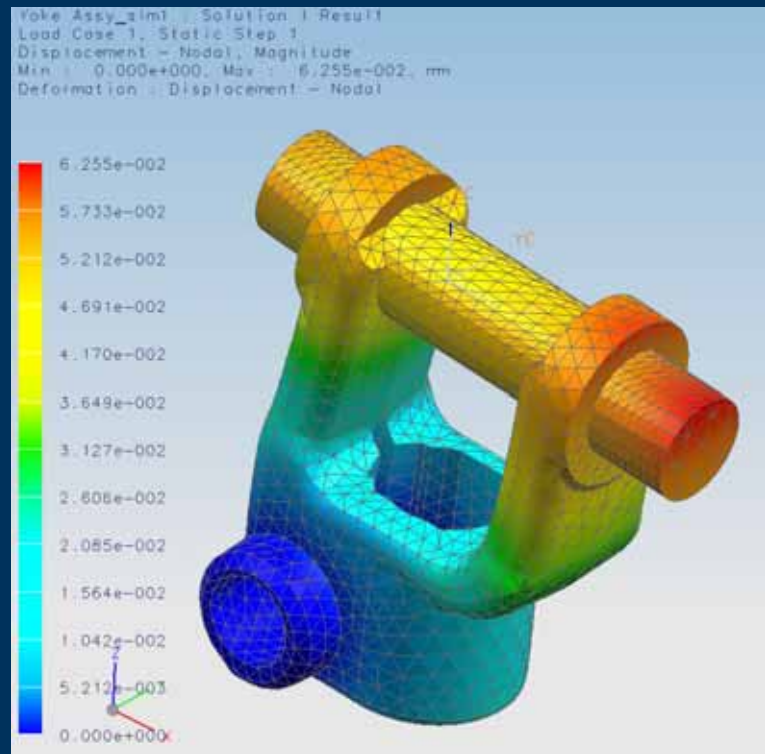


# Results – Selection

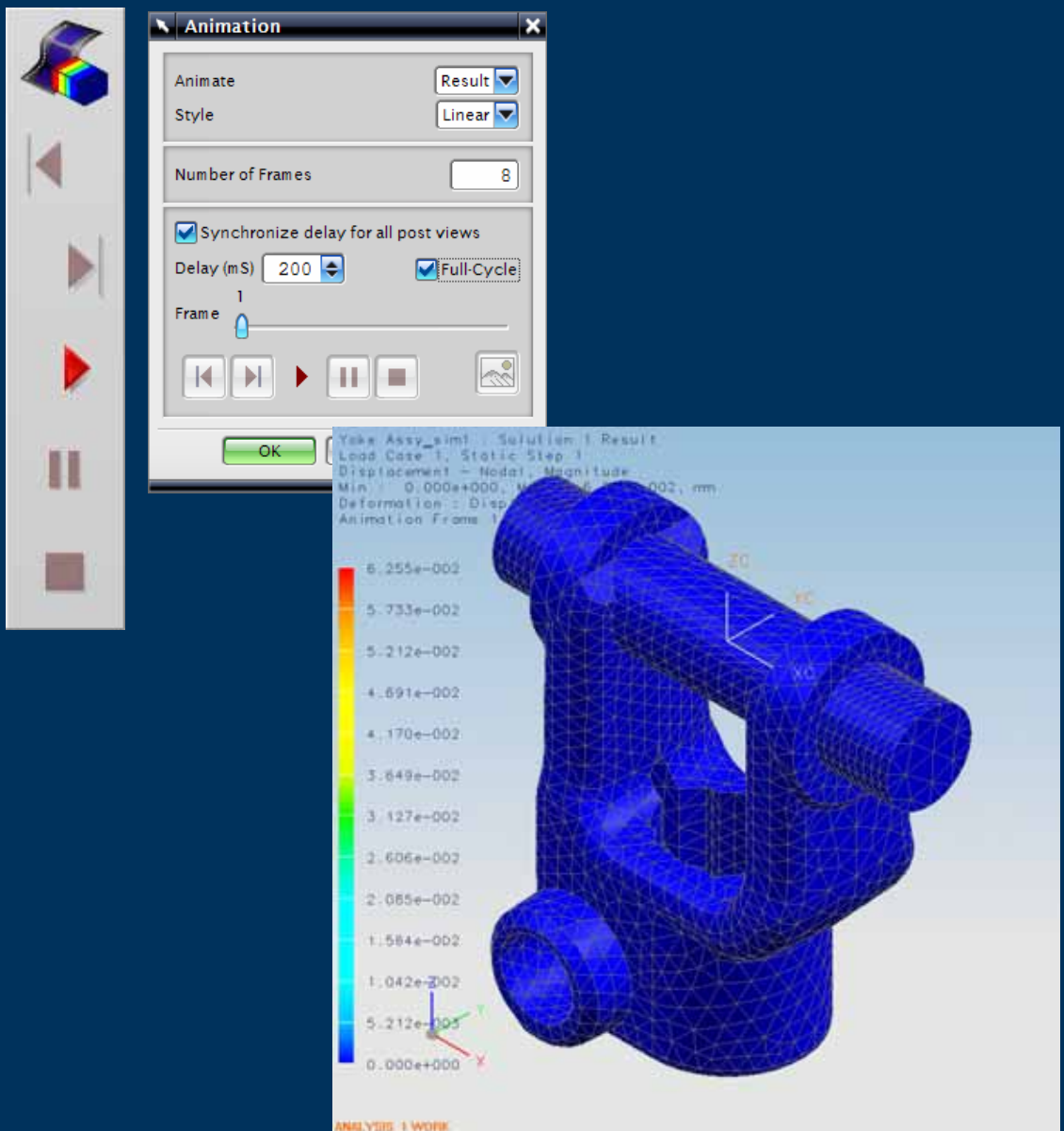



- Results Selection from the Post Processing Navigator
- Ease of navigation through available Results
- Double mouse click to change the Results display
- Plot to a Existing Viewport, to a New Viewport or Overlay (combine) Existing display



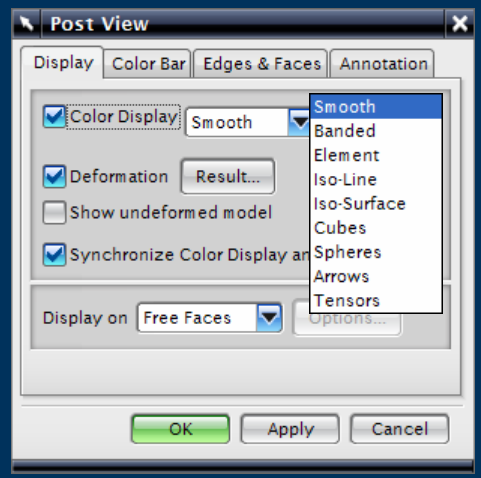



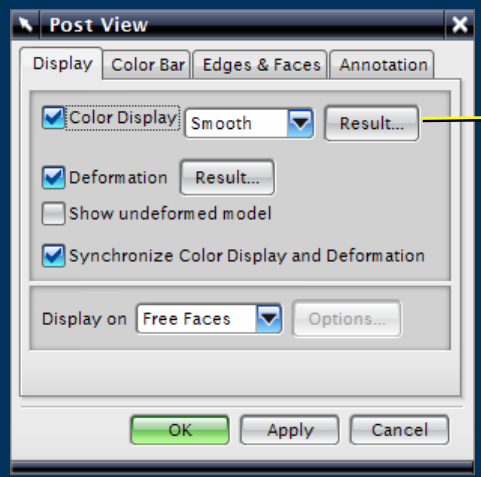
# Results – Animation



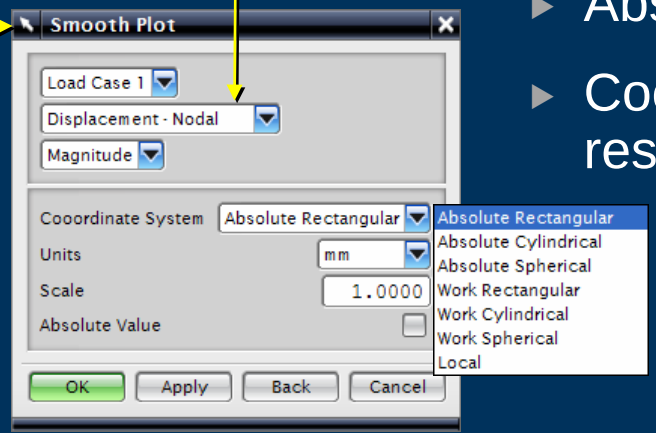

- Results Animation options
  - Number of Frames
  - Delay between frames
  - Full cycle (“forwards then backwards”)
  - Play, Pause, Stop
  - Single frame forwards or backwards
- During Animation
  - Change the Results
  - Screen rotate/pan/zoom
  - Toggle on/off Meshes
  - Save to Animated GIF

# Results – Post View Display



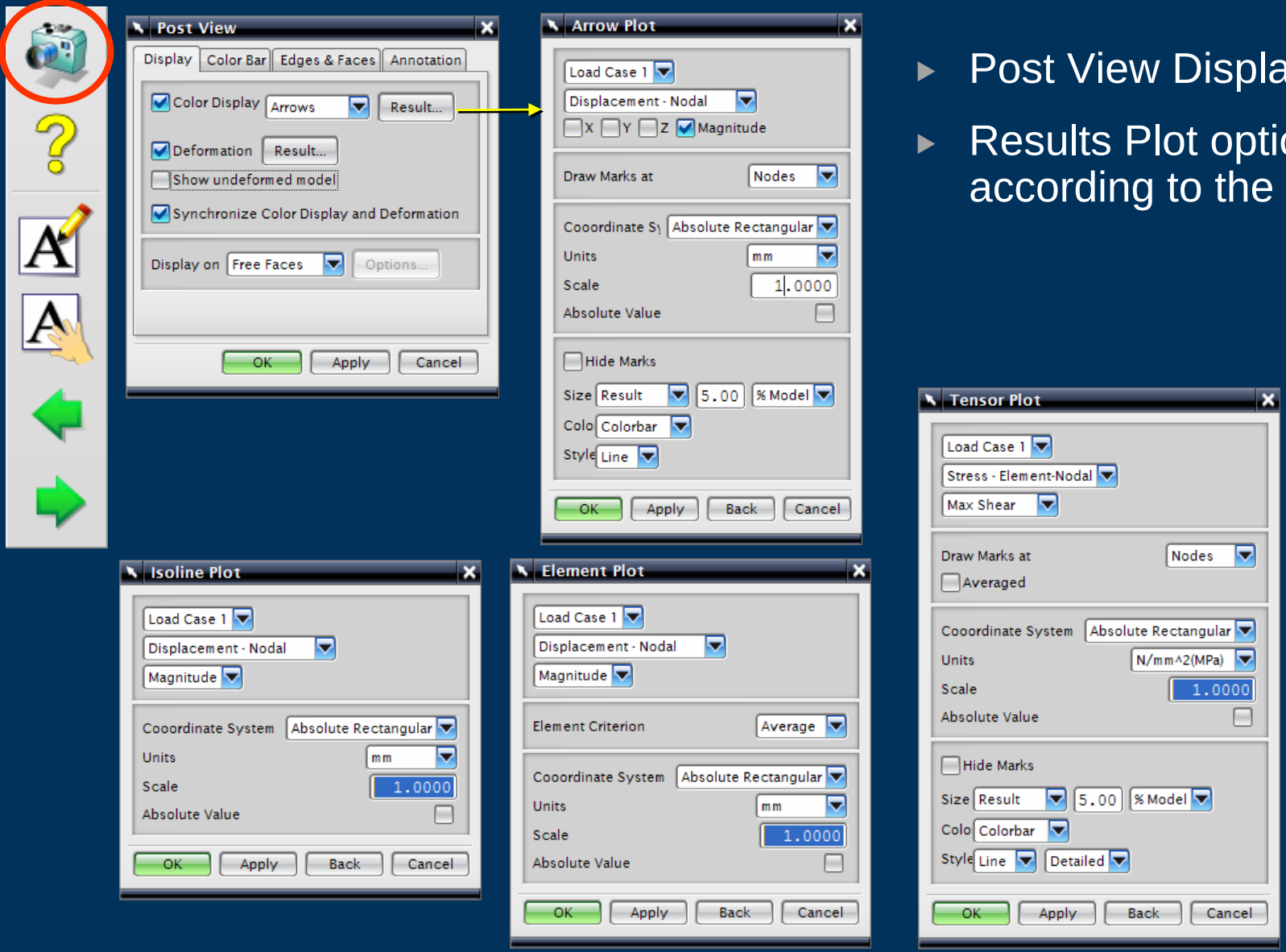




Result Set Selection



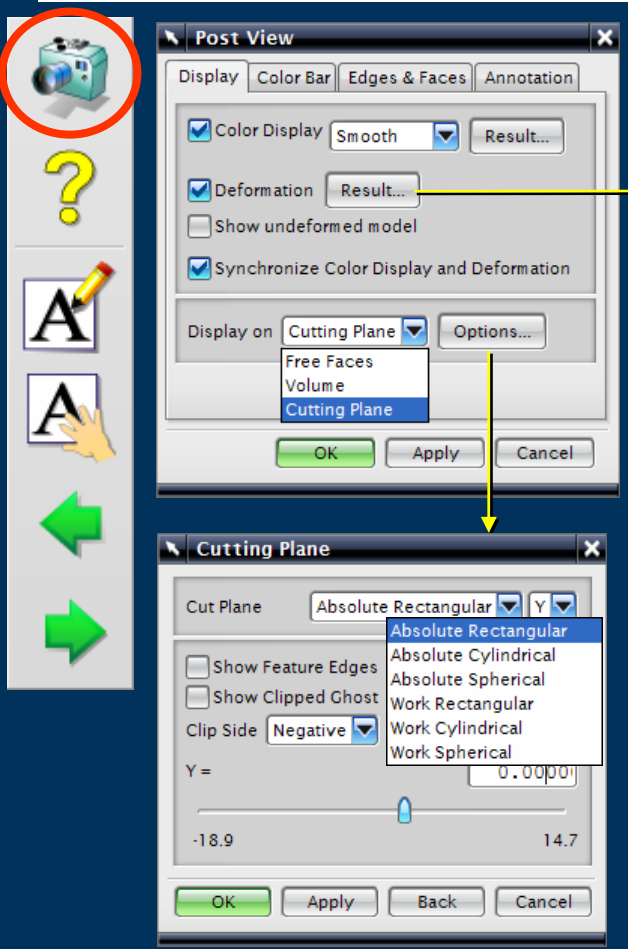

- Post View Display Options
  - Styles
    - Contours
    - Elements
    - Isolines, Isosurfaces
    - Mark – Arrow, Cube, Sphere, Tensor
  - Absolute or scaled values
  - Coordinate System for results calculation

# Results – Post View Display

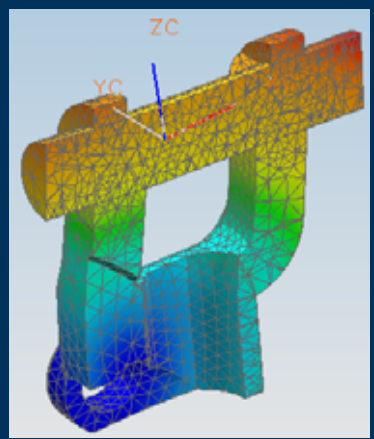


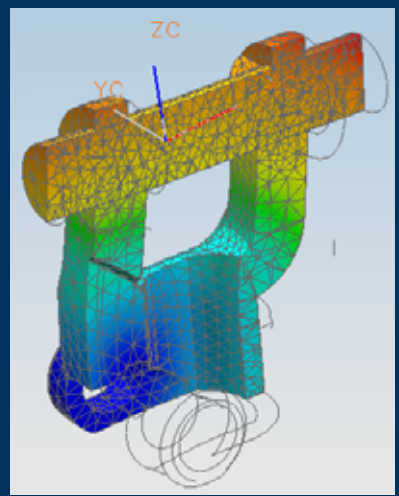

- Post View Display Options
- Results Plot options vary according to the Display type

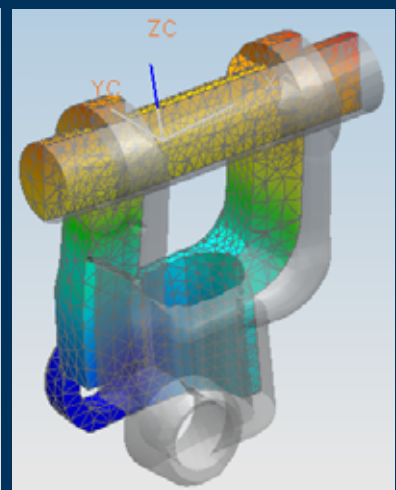
# Results – Post View Display



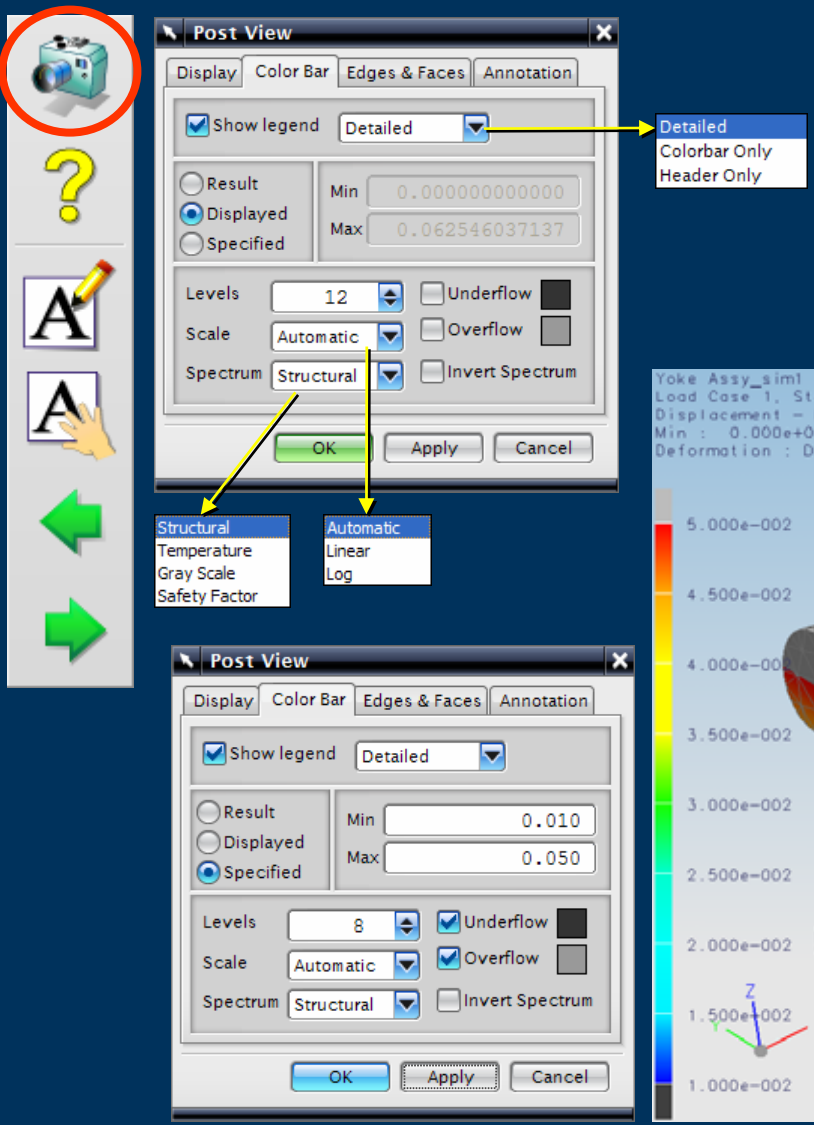

- Deformed model display
- Results domain
  - Free Face
  - Volume
  - Cutting Plane

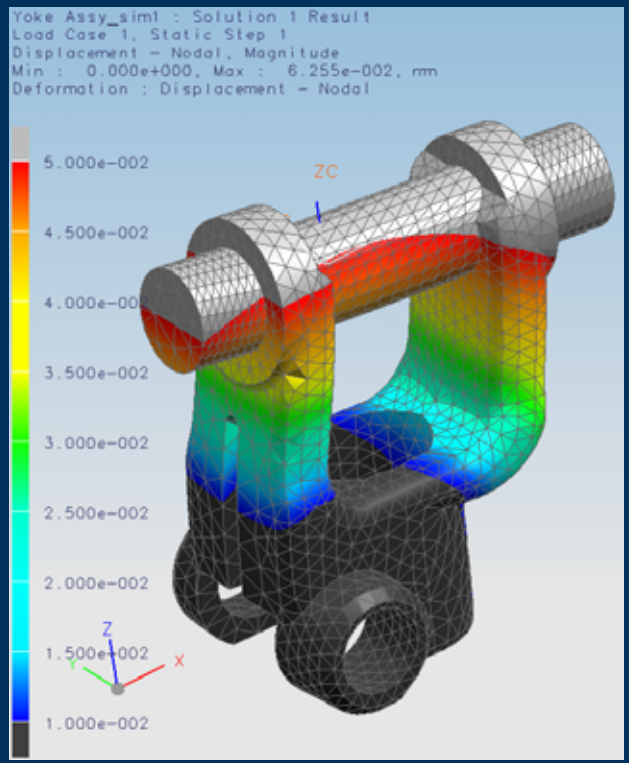


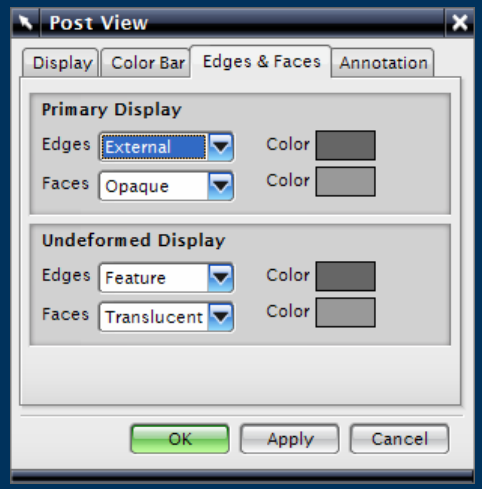
# Results – Post View Color Bar



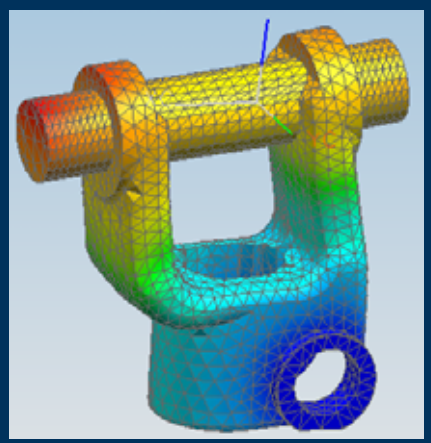



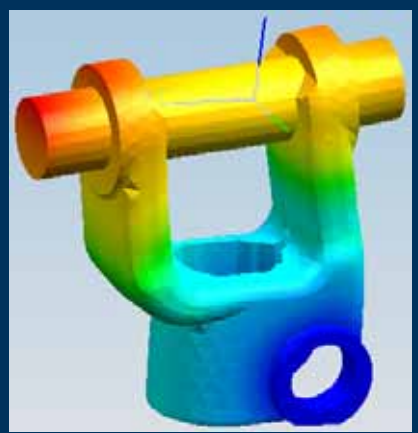

- Post View Color Bar
  - Max & Min values
  - Number of colors
  - Color scheme
  - Scale
  - Legend level of detail

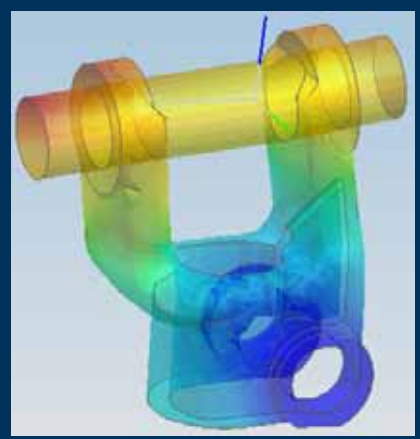
# Results – Post View Edges & Faces

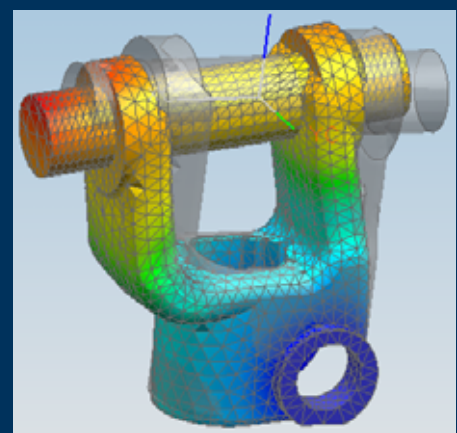



- Post View Edges & Faces display & color options
  - Primary Display for Element Edge & Faces
  - Undeformed Display

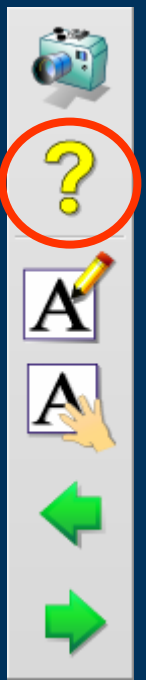


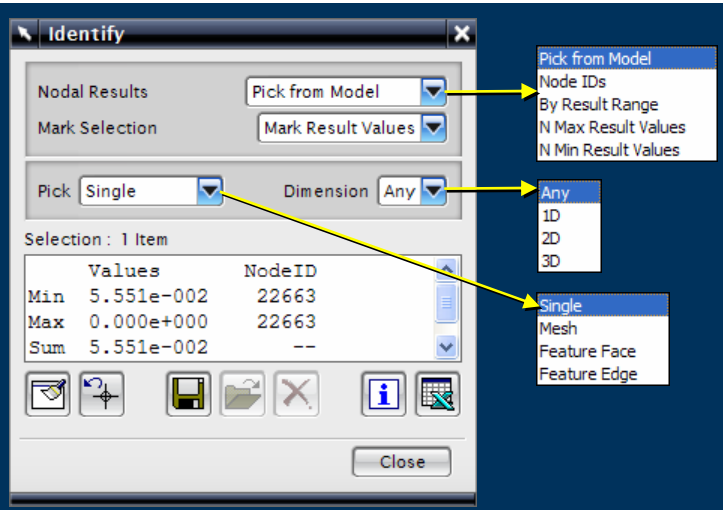


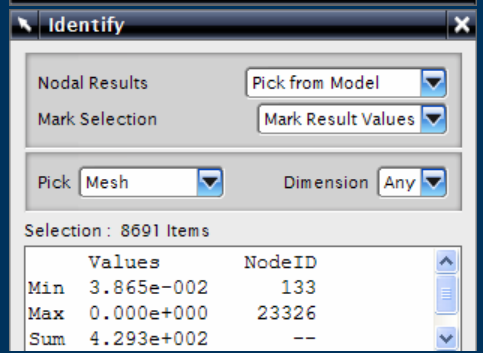


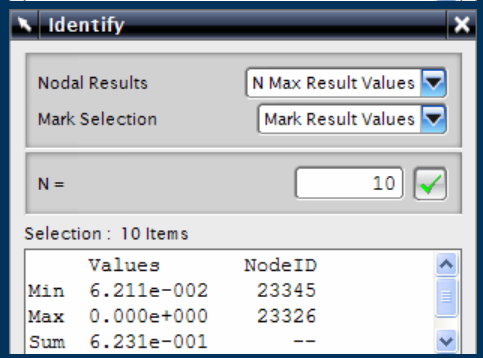


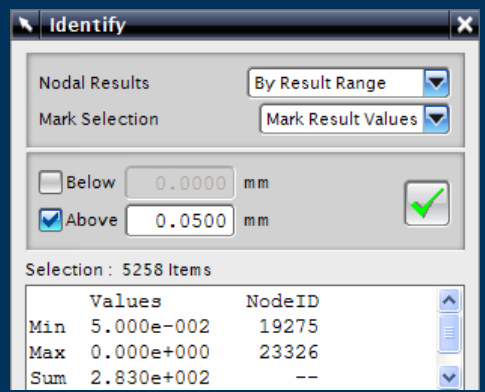
# Results – Identify



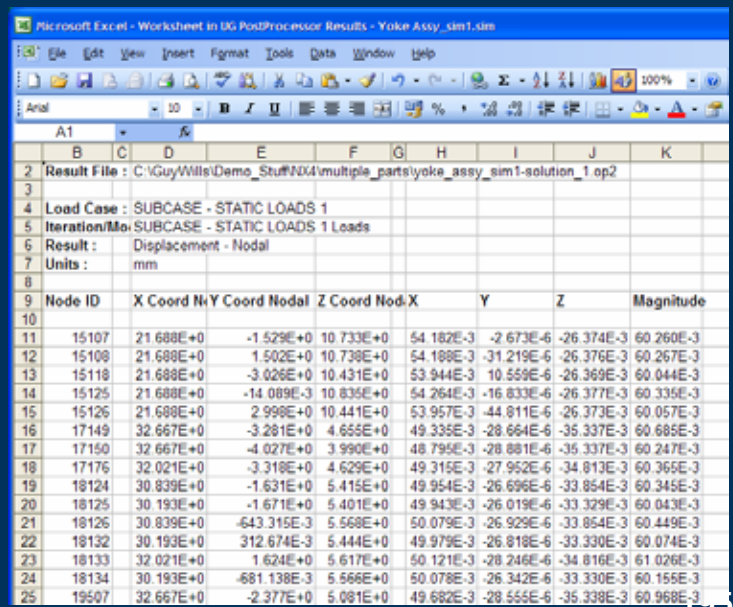


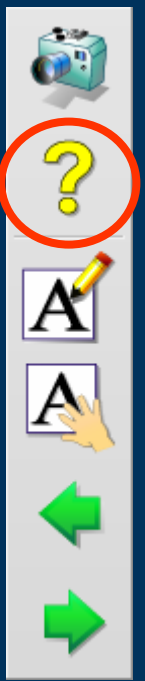




- **Identify** to probe and display nodal and elemental information
- Results
  - n Max, or n Min Result values
  - Result Range
- Selected data saved to Excel for further study

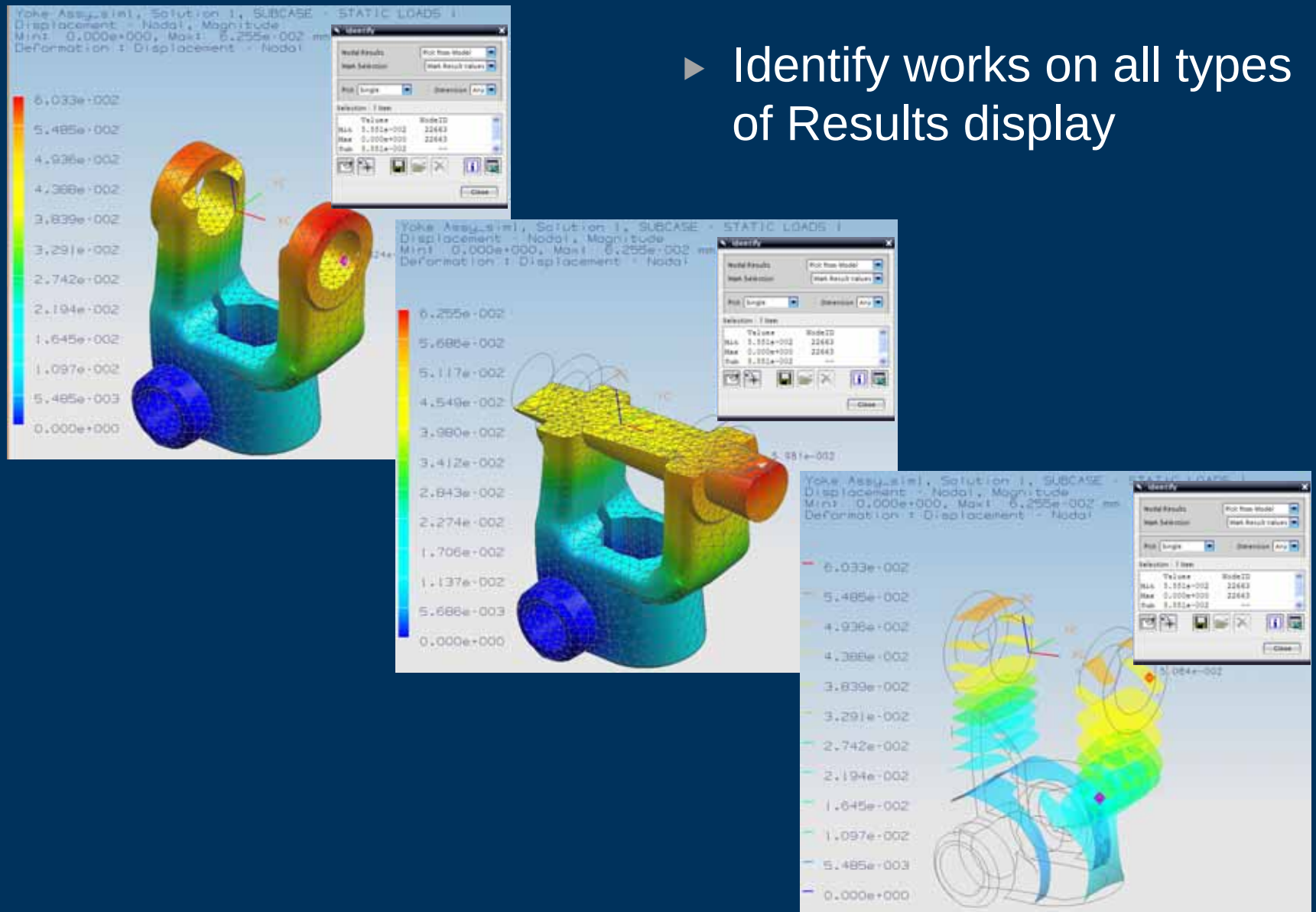


SIEMENS

# Results – Identify

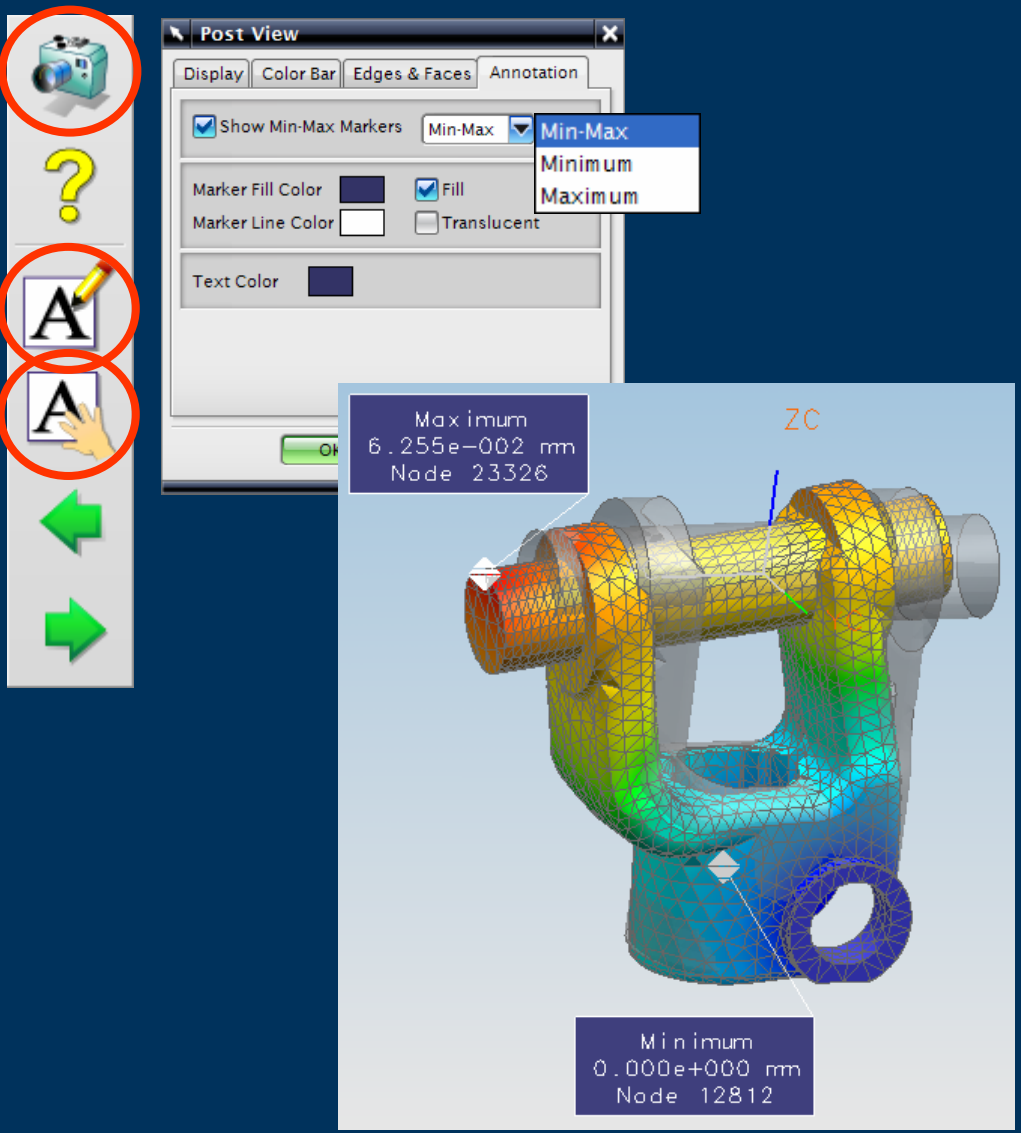


- Identify works on all types of Results display



SIEMENS

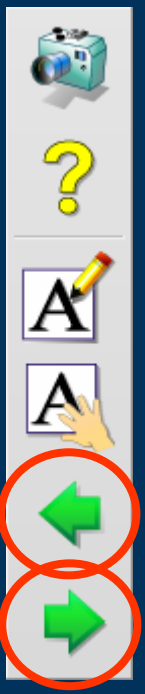
# Results – Annotation Markers

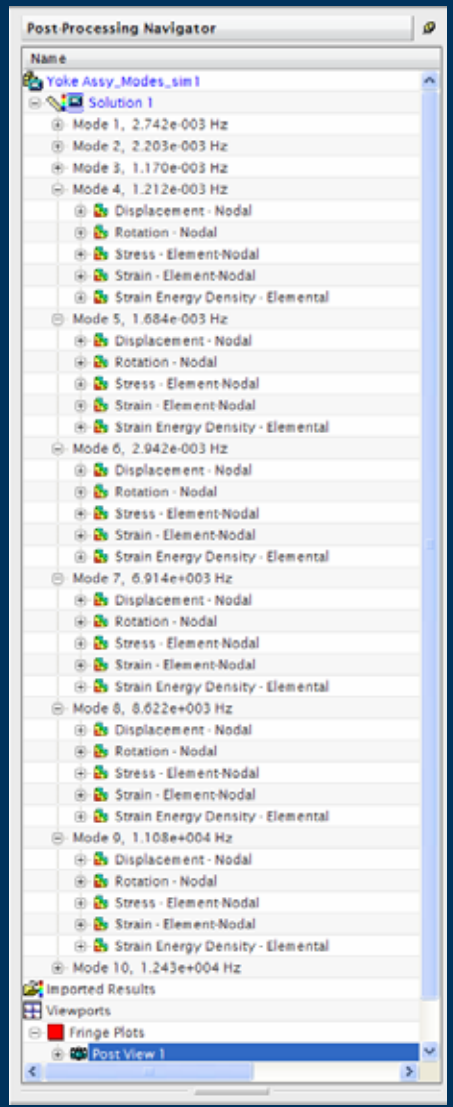



- Results Marker shows max/min of the current active Result Set
  - Max & Min
  - Max only
  - Min only
- Drag Marker to reposition

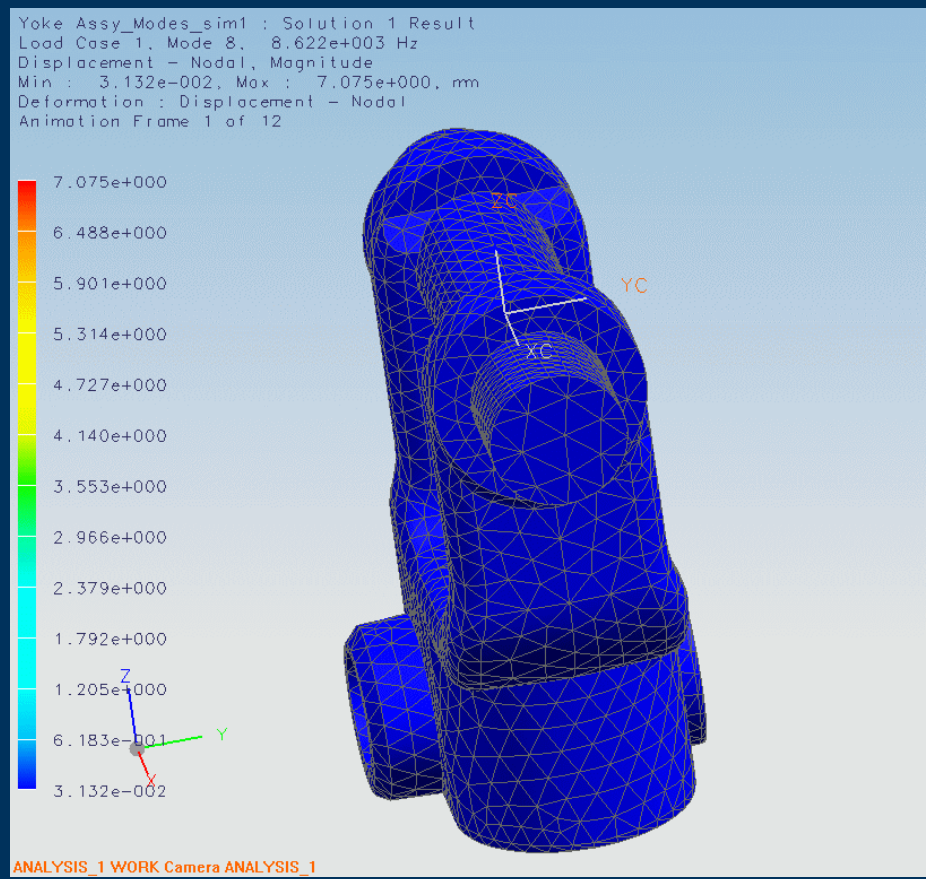
SIEMENS

# Results – Previous / Next Mode or Iteration



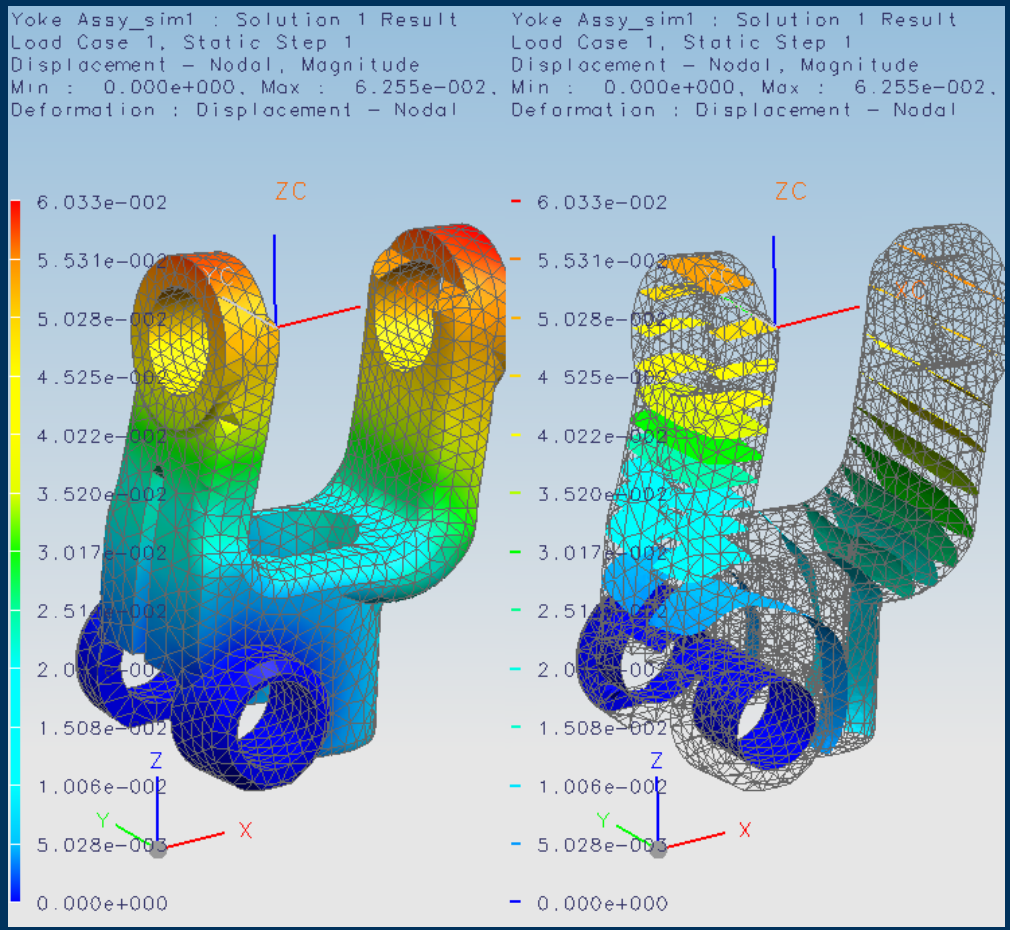



- Quick change to the next Mode or Iteration

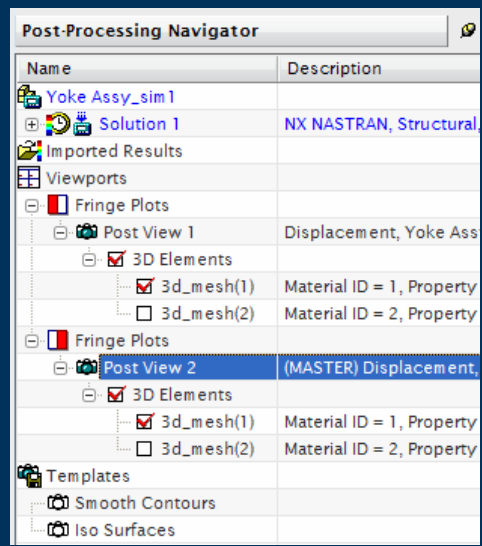


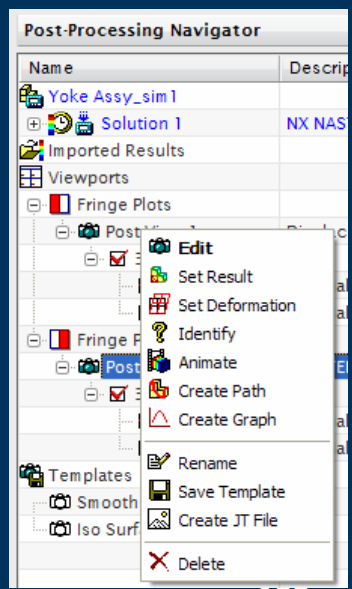
SIEMENS

# Results – Post Views & Templates



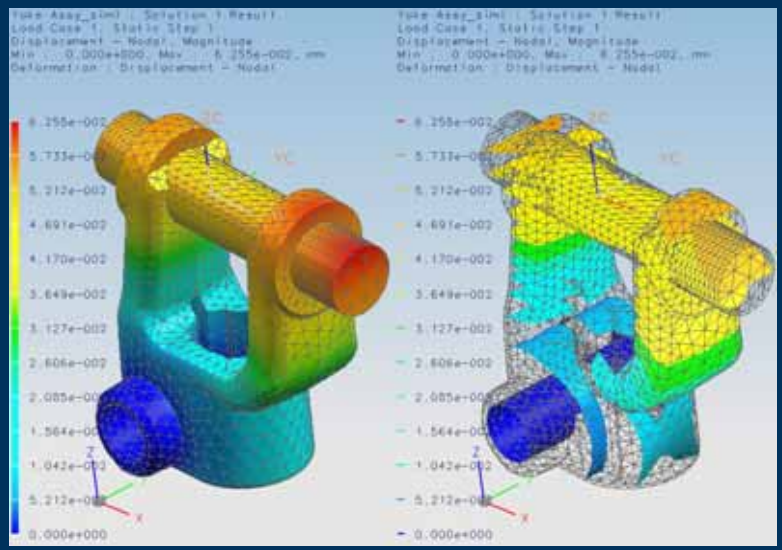

- Post Views store the Post View setup
- Provides quick and efficient method of controlling displays in different Views
- Post Views can be saved and are available for re-use

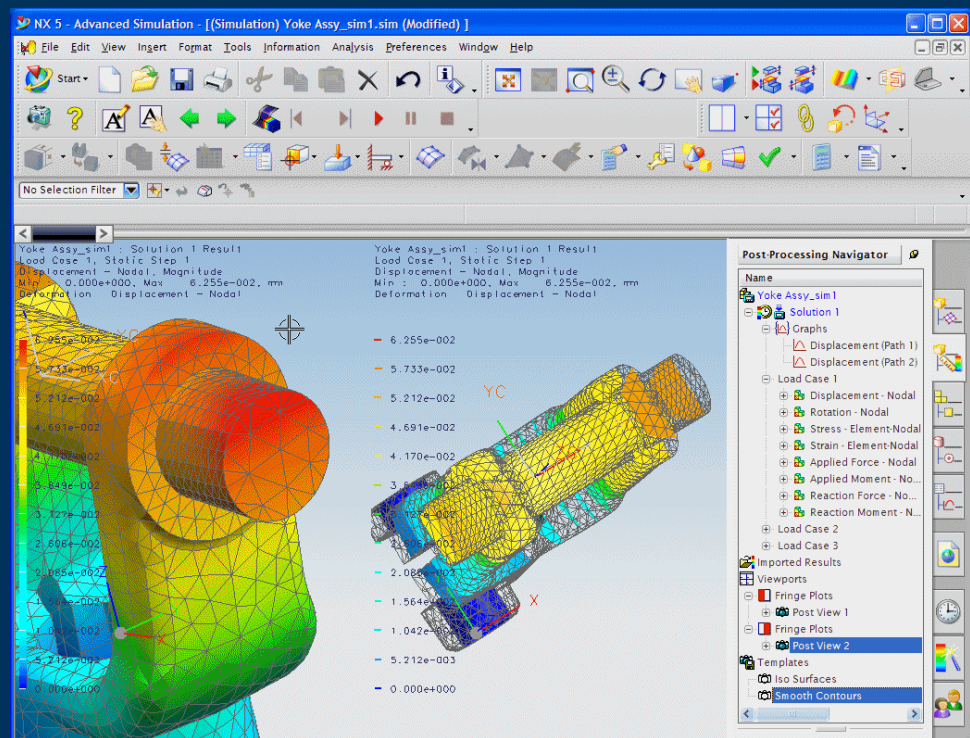


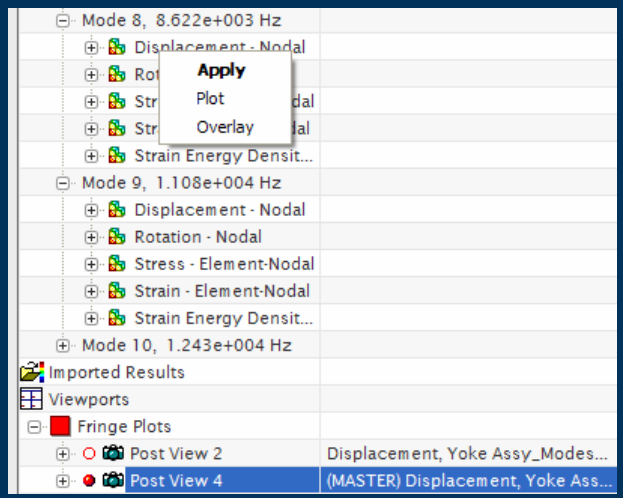
# Results – Multiple Viewports

- Multiple Viewports
- Different Post Views per Viewport
- Select all views ports for Synchronised screen rotate/pan/zoon
- Return the view to Model display

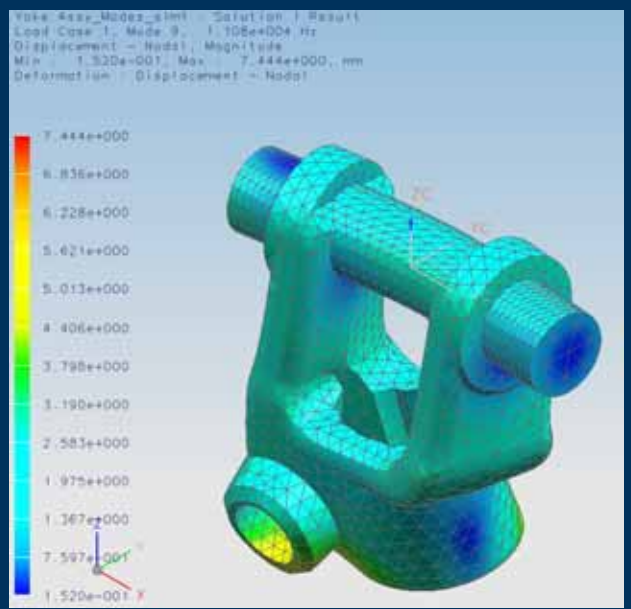


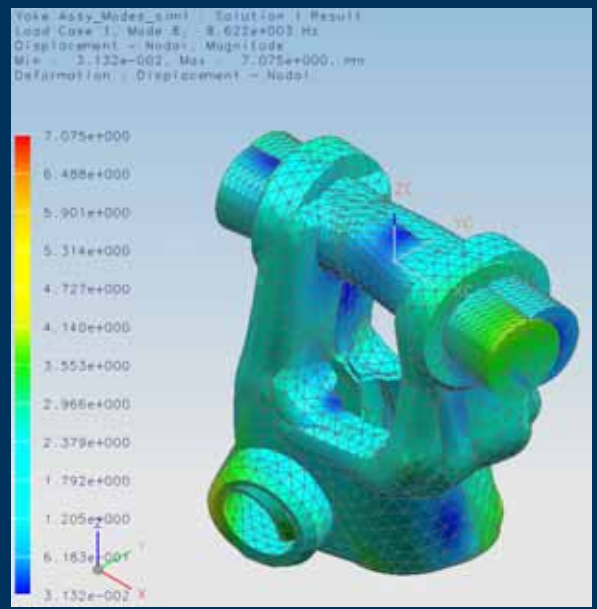


# Results – Post View Overlay

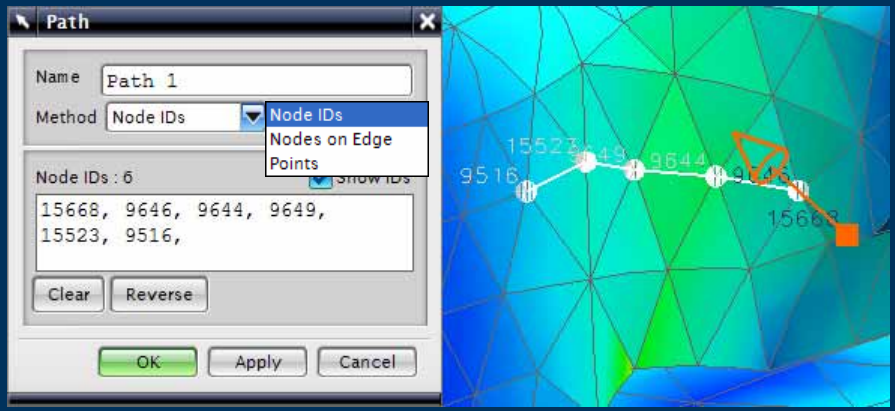



- Post View Overlay
- Post View changes can be saved to all or selected Post Views

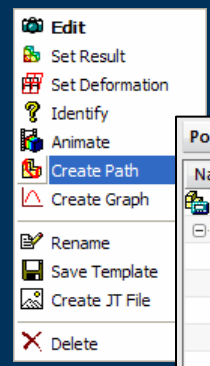


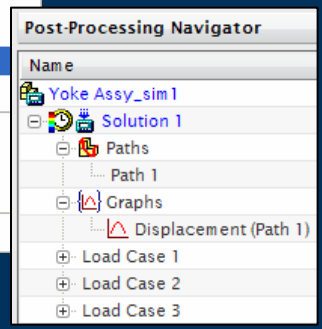


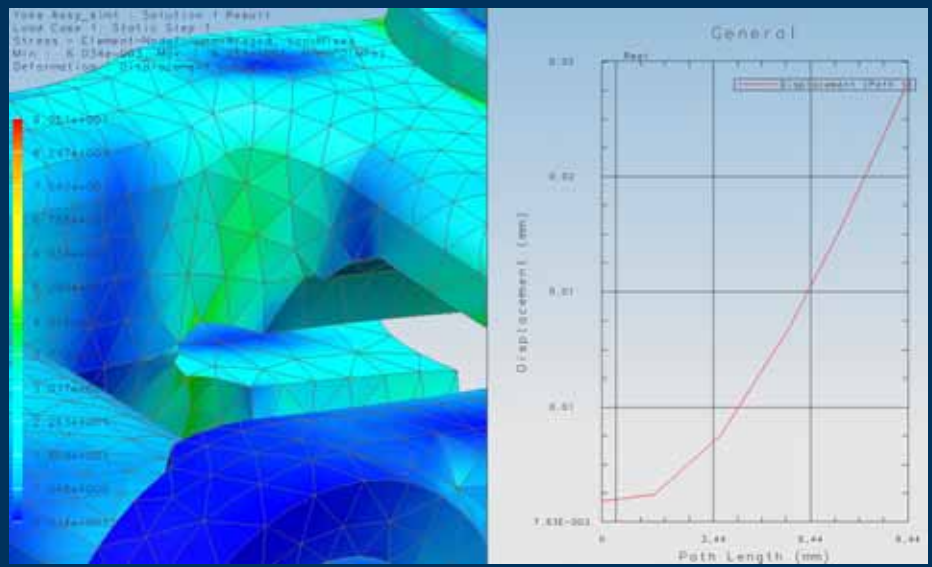
# Plotting Paths

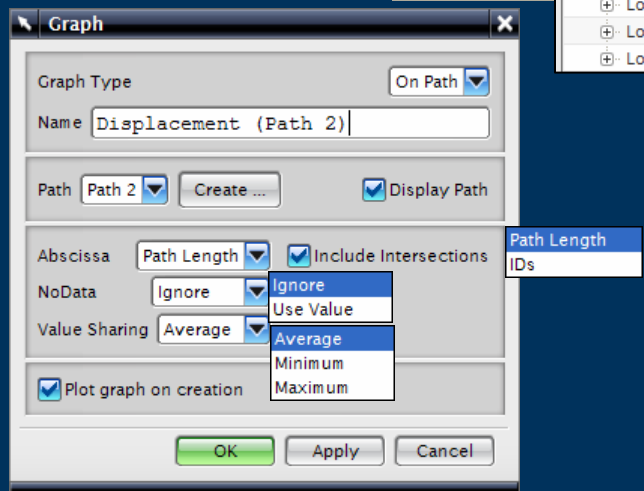



- Paths defined by
  - Node labels
  - Screen selection
  - Edge selection

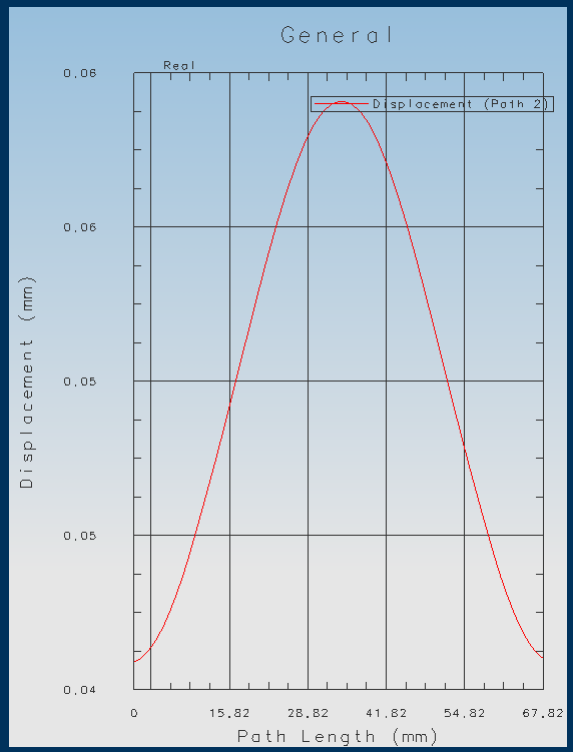


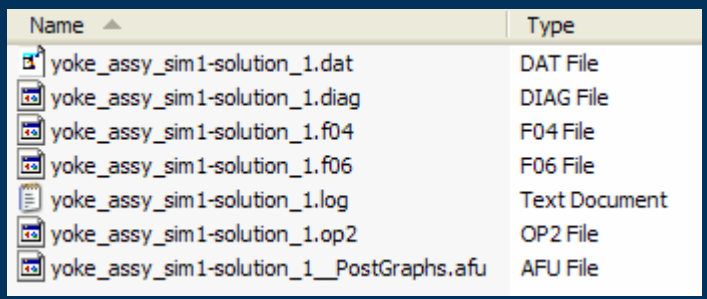



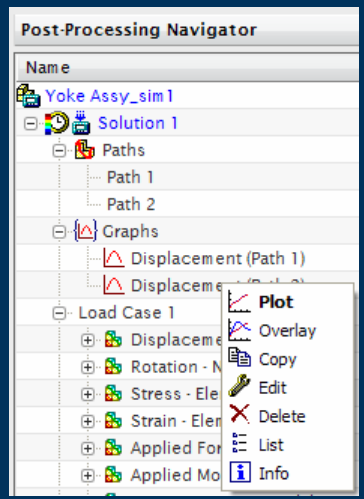



# Plotting Paths

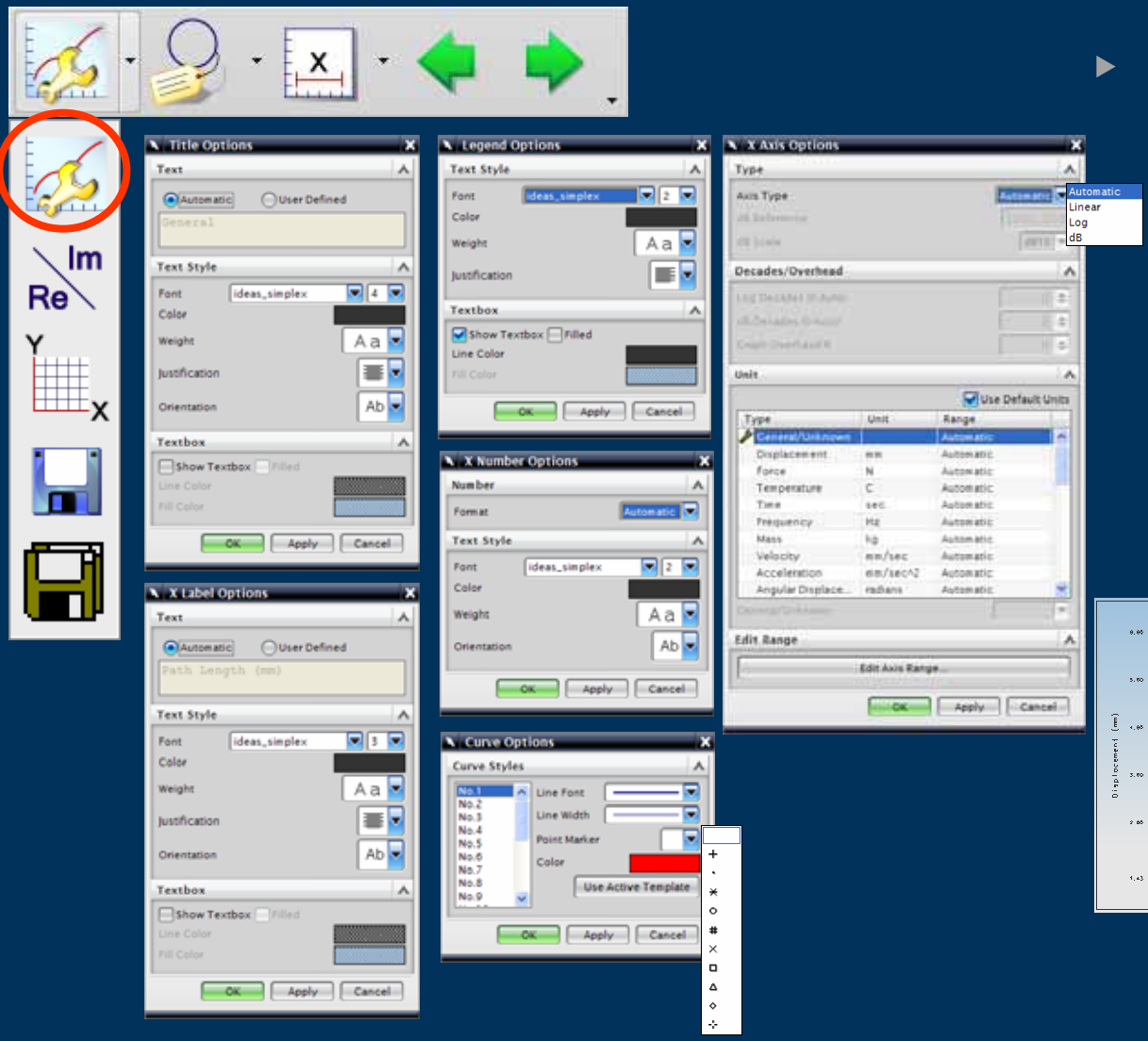


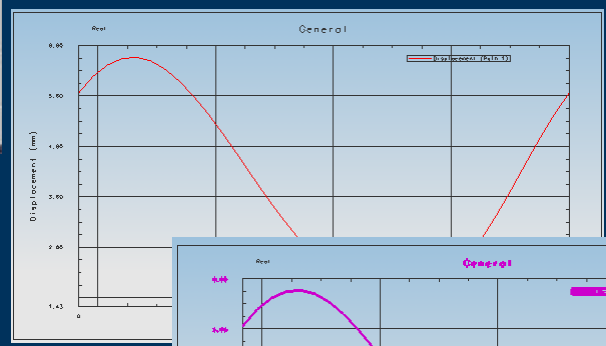

- Graphs can be Overlaid to compare data using same Axis
- Path and Graph data stored in external file (*.afu)

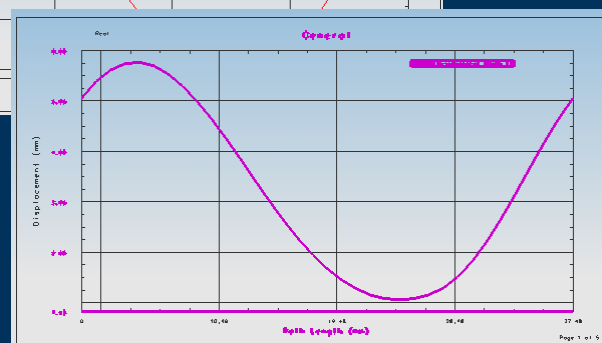


# Graph Style

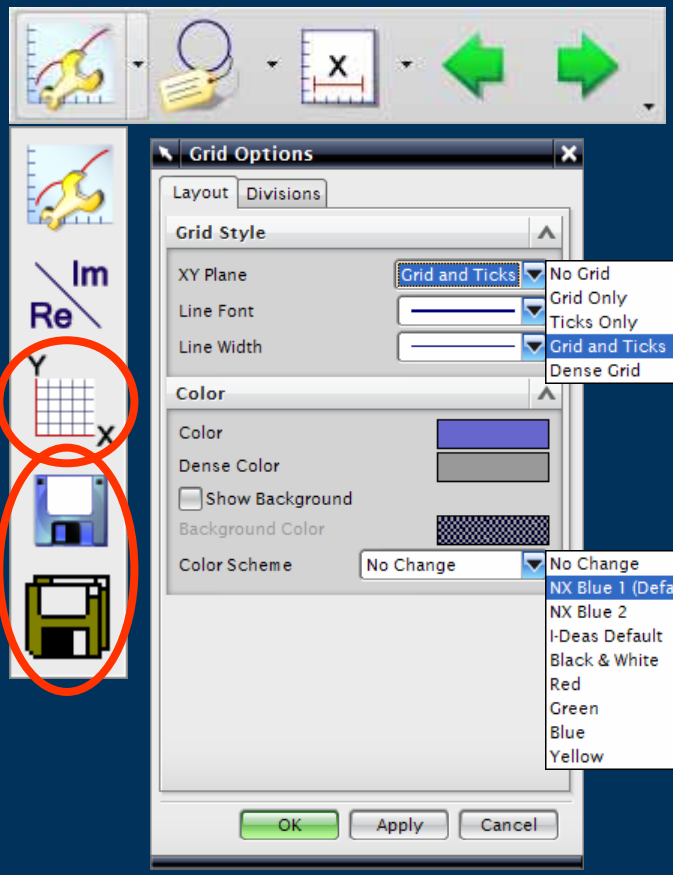


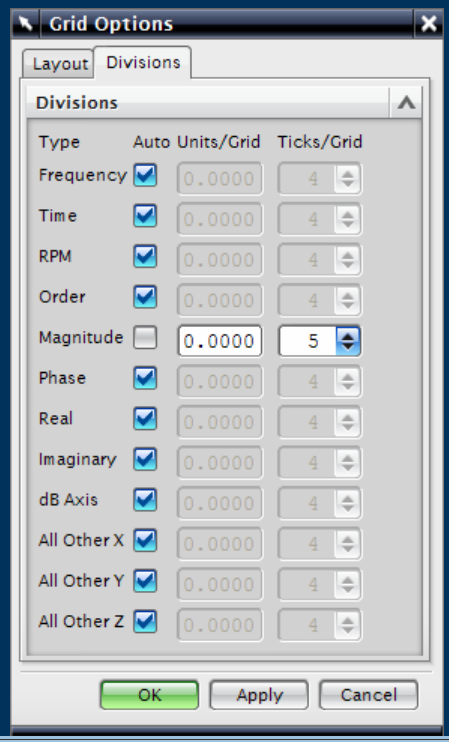



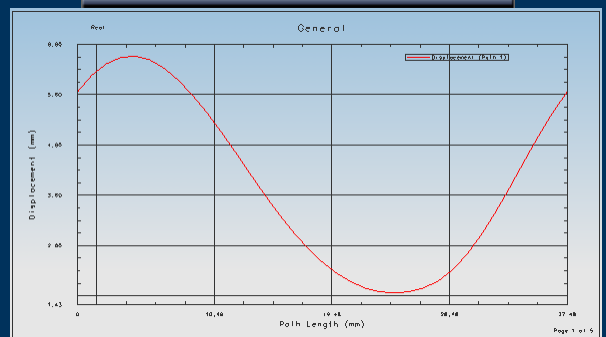



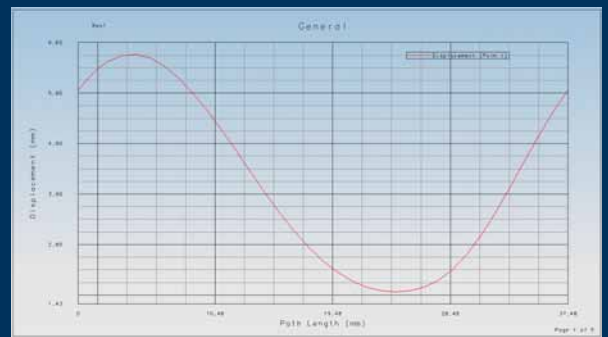

- Edit Graph Style
  - Dynamic Selection of Graph element
  - Graph Title
  - Graph Legend
  - Axis style
    - Axis Labels
    - Axis Numbers
    - Axis Type, Values & Units
  - Curve style

# Graph Style



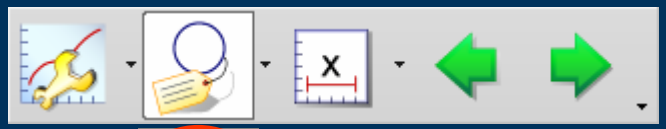



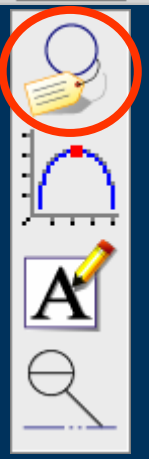



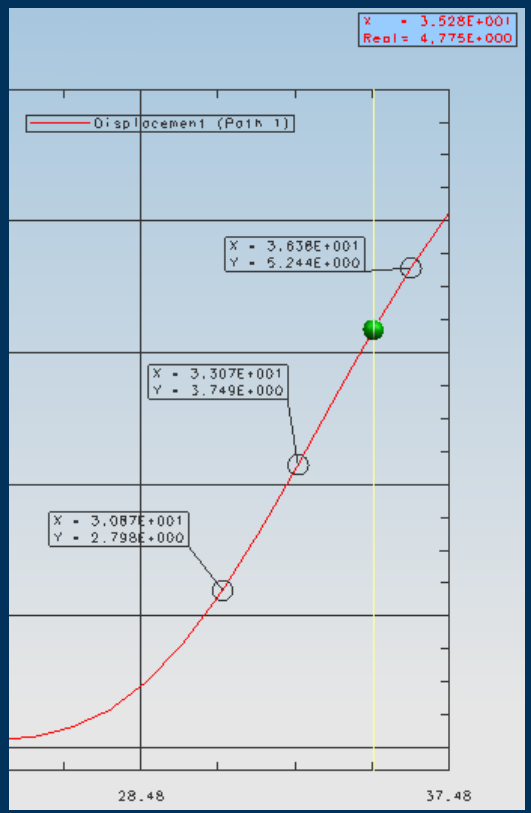

- Graph Grid Style
  - Grid Layout
  - Divisions
- Save the current Graph Settings to the current Template File
- Save the current Graph Setting to a new Template File

SIEMENS

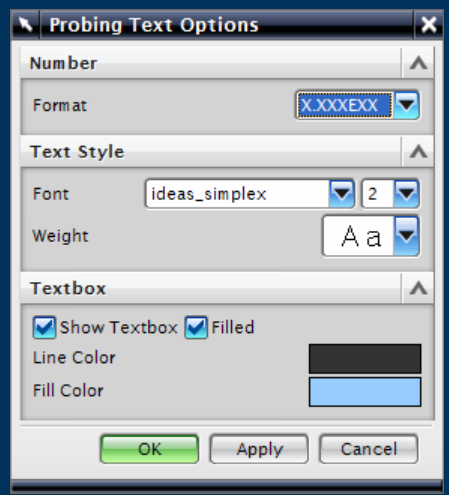
# Graph Probing





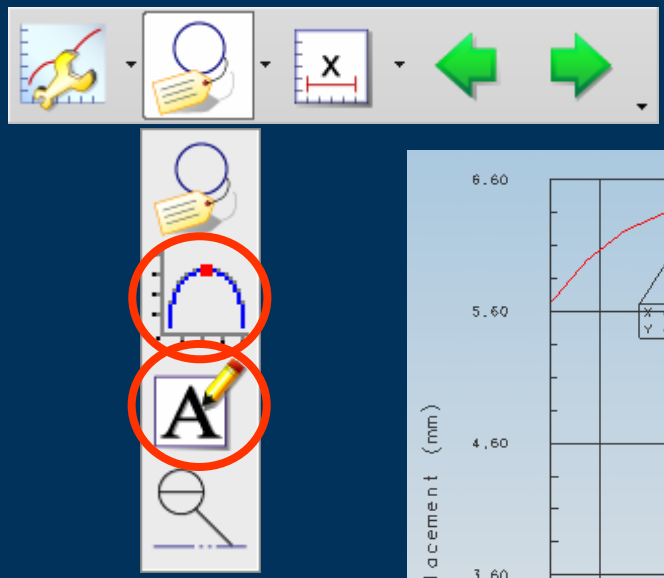


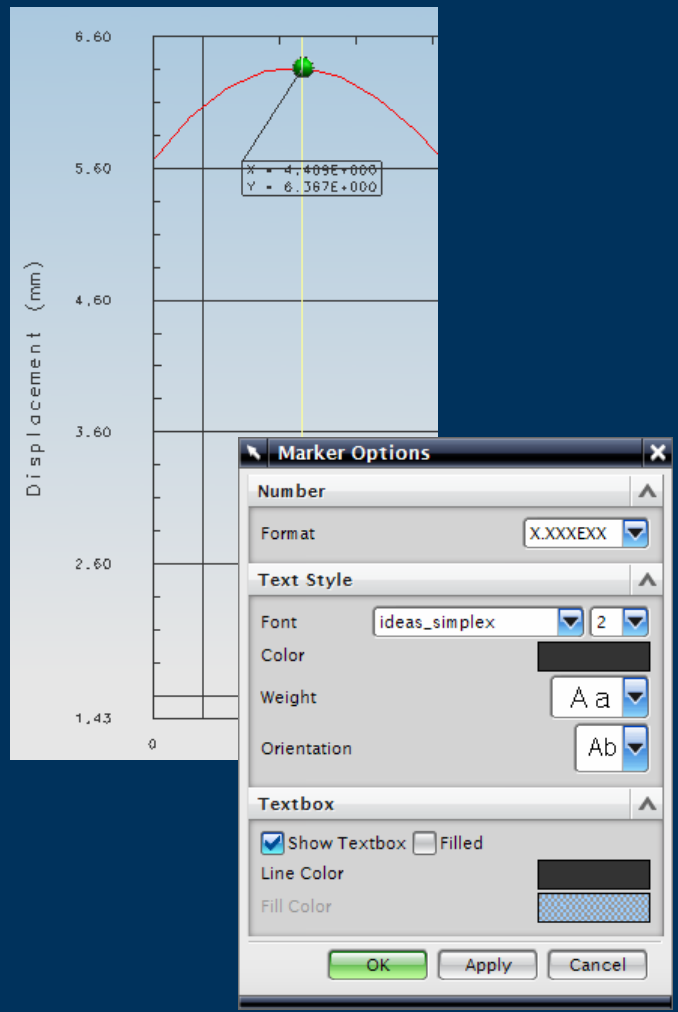

- Data Point Probing and Marking
- Dynamic Display of curve data points & values
- Specific curve location values
- Probe Text Styles



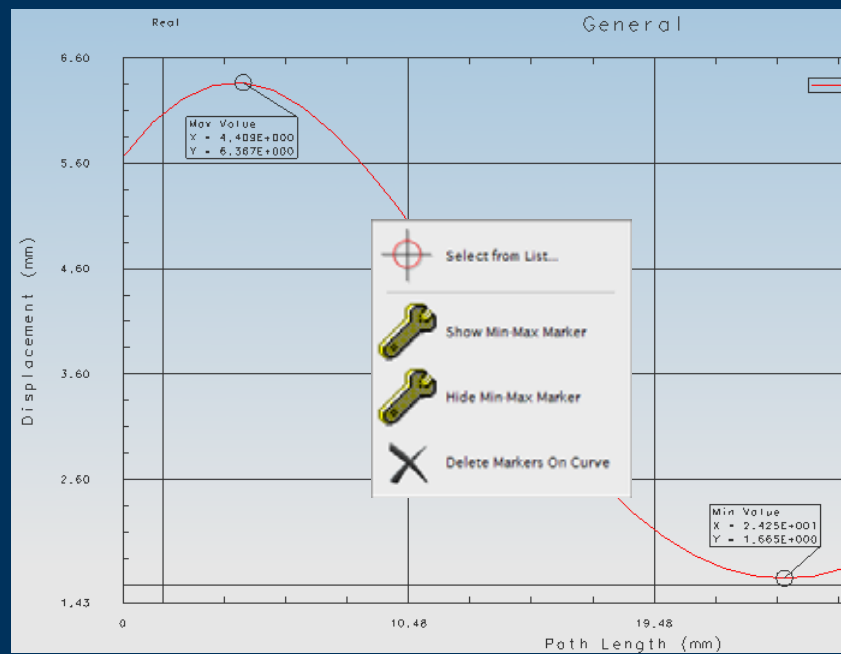
SIEMENS

# Graph Probing



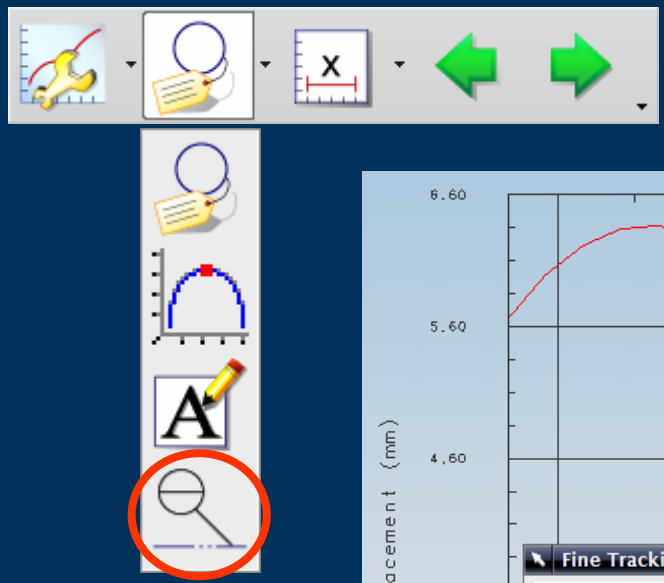



- Peak and Valley locating and Marking

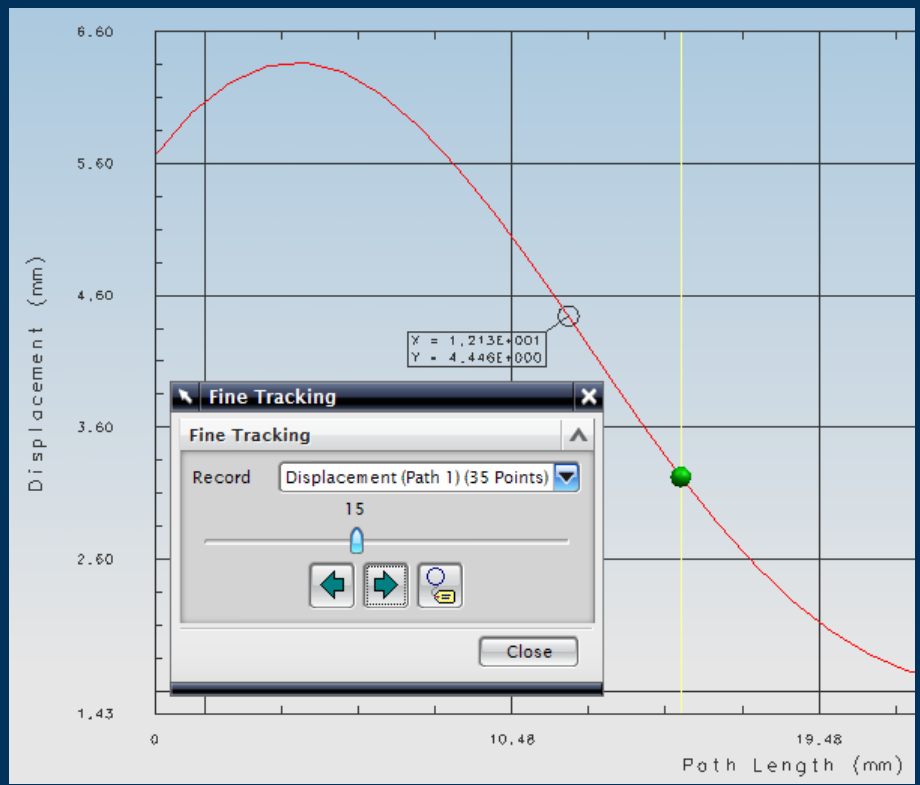


SIEMENS

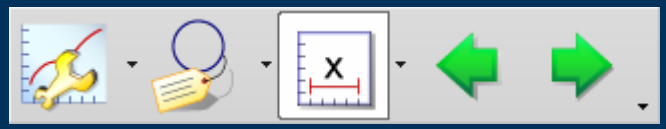
# Graph Probing

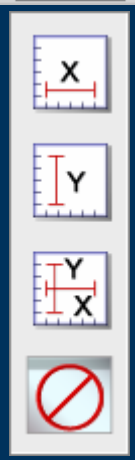


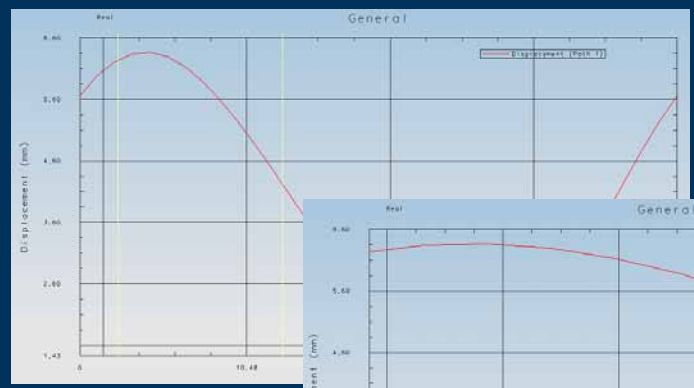
- Fine Tracking and Marking

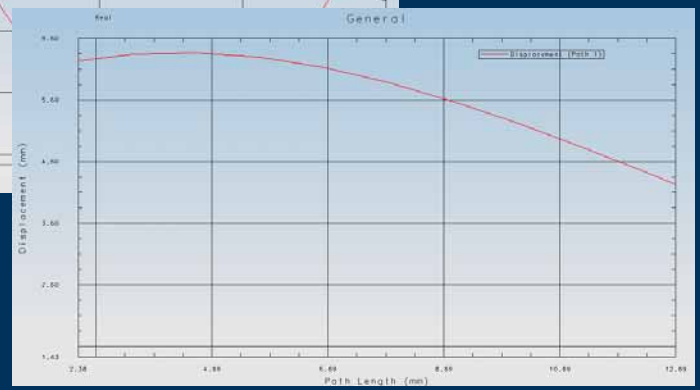


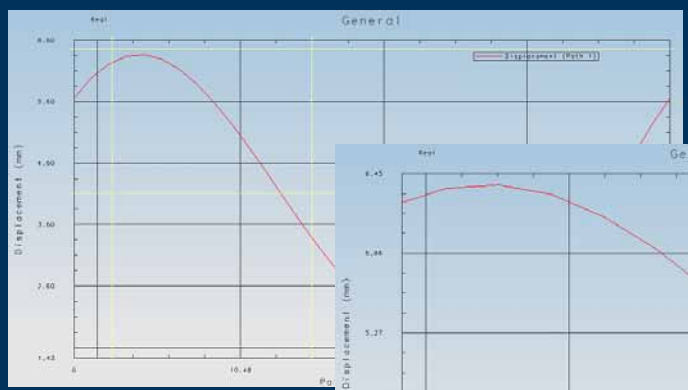
# Graph Windowing

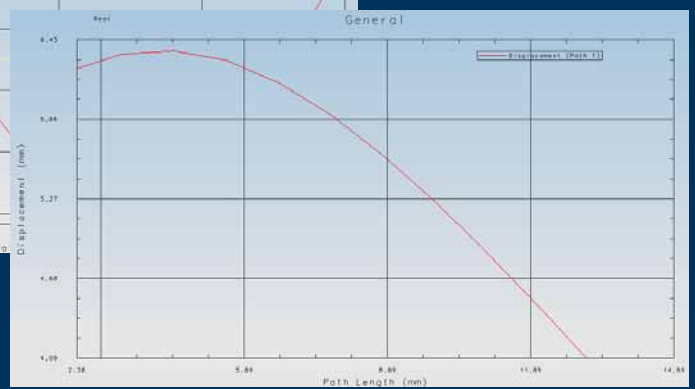






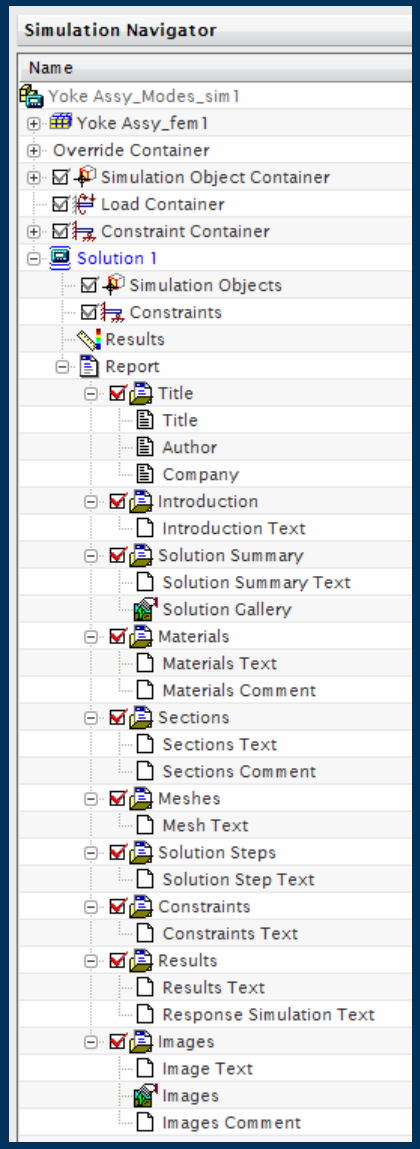



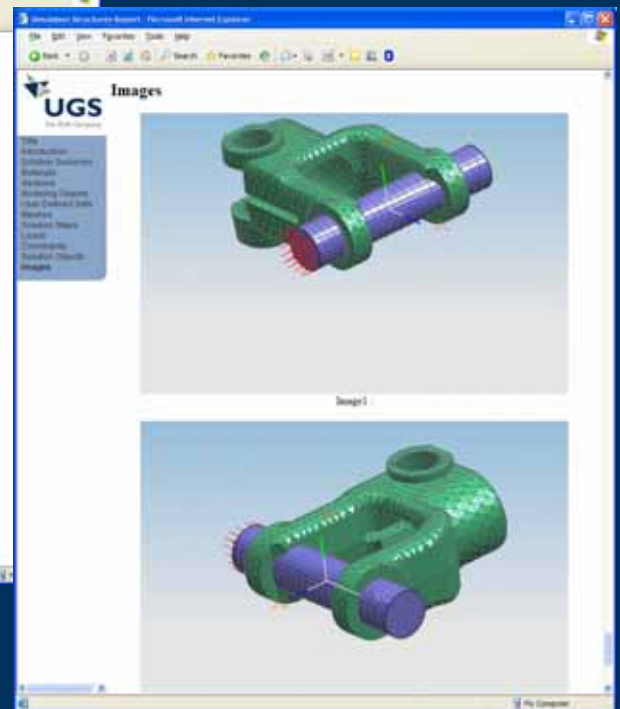

- Window into the Graph
  - X Window
  - Y Window
  - X-Y Window
  - No Window to return to full graph

SIEMENS

# Solution Report – After Solve

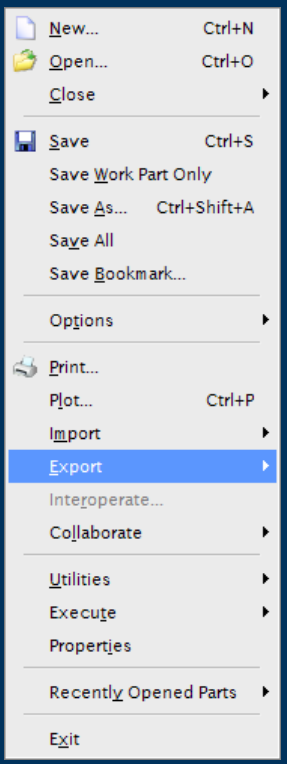


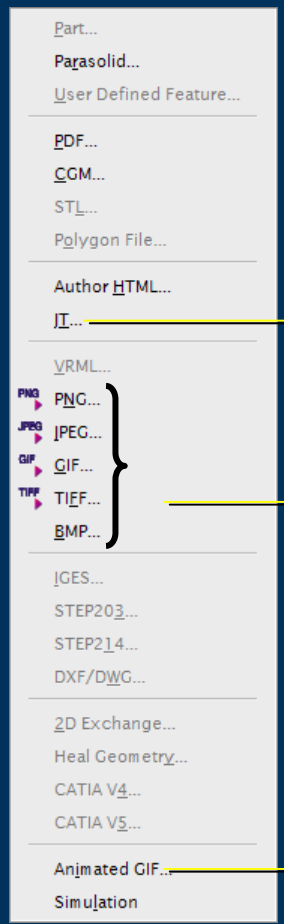


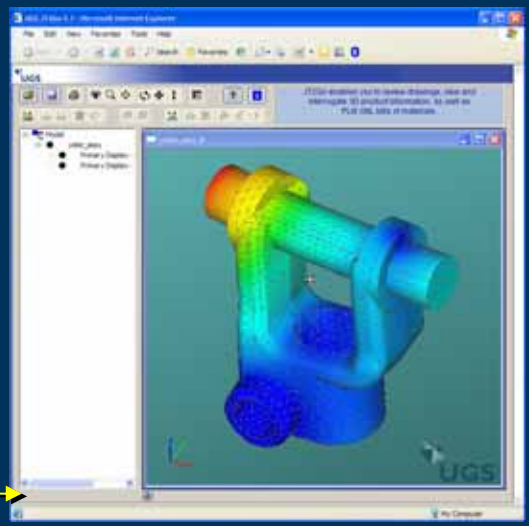



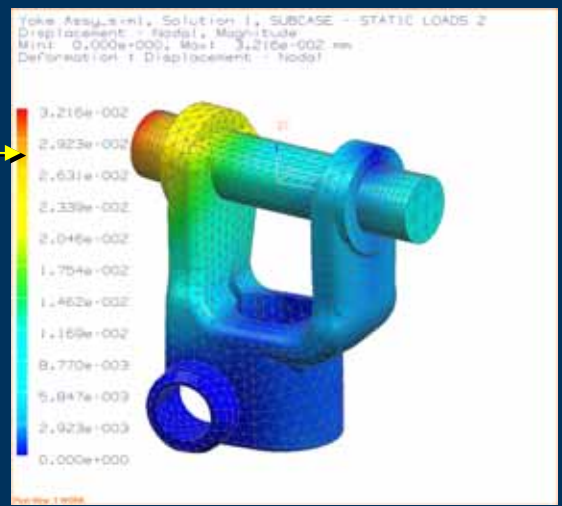

- Solution Report – After Solve
  - User entered text description/documentation
  - Snapshot screen images
- HTML Interactive report export

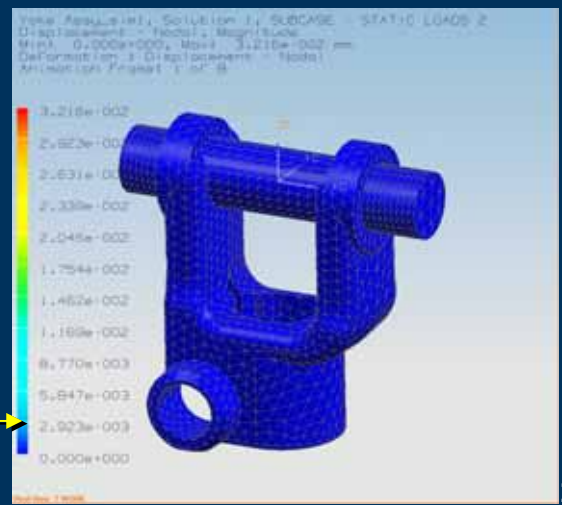
# Export Visualisation Files





- Direct export of JT visualisation model
- Direct export of screen images
  - PNG
  - JPEG
  - GIF
  - TIFF
  - BMP
- Direct export of Animation
  - Animated GIF

SIEMENS

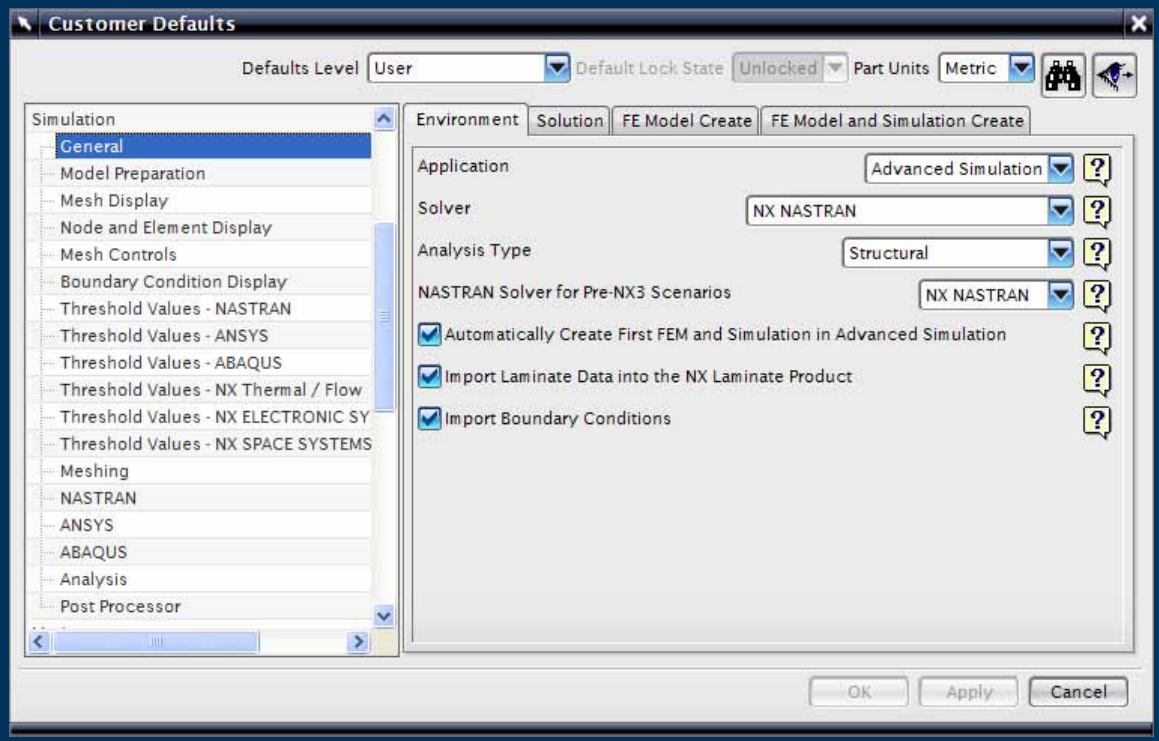
# Customer Defaults for Simulation



UGS PLM Software

SIEMENS

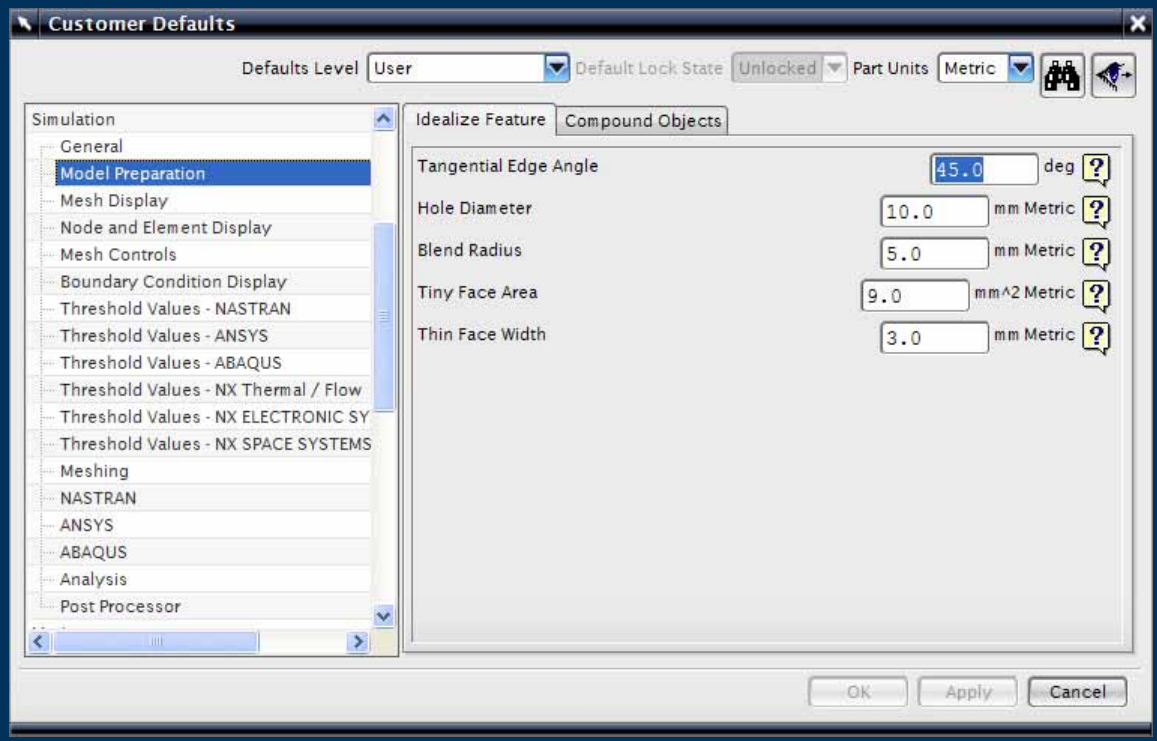
# Customer Defaults – General




- General
  - Default Solver Language
  - Creation of the four simulation files

SIEMENS

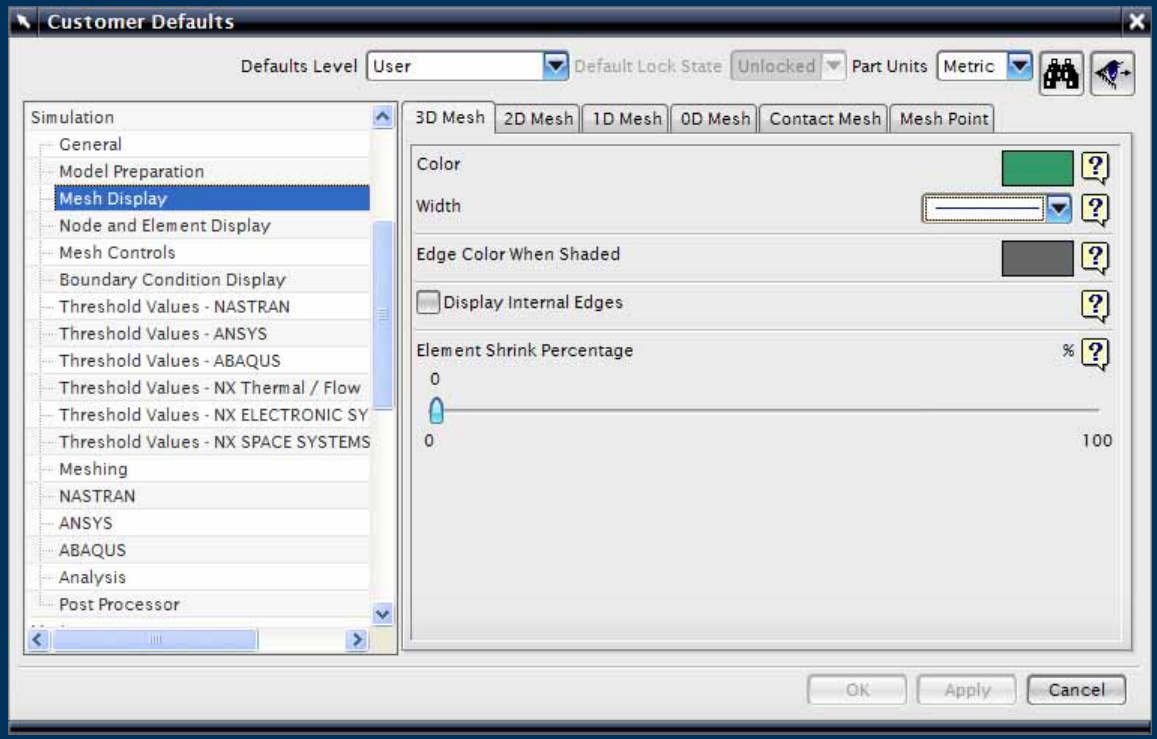
# Customer Defaults – Model Preparation




- Model Preparation
  - Default values for CAE Topology creation

SIEMENS

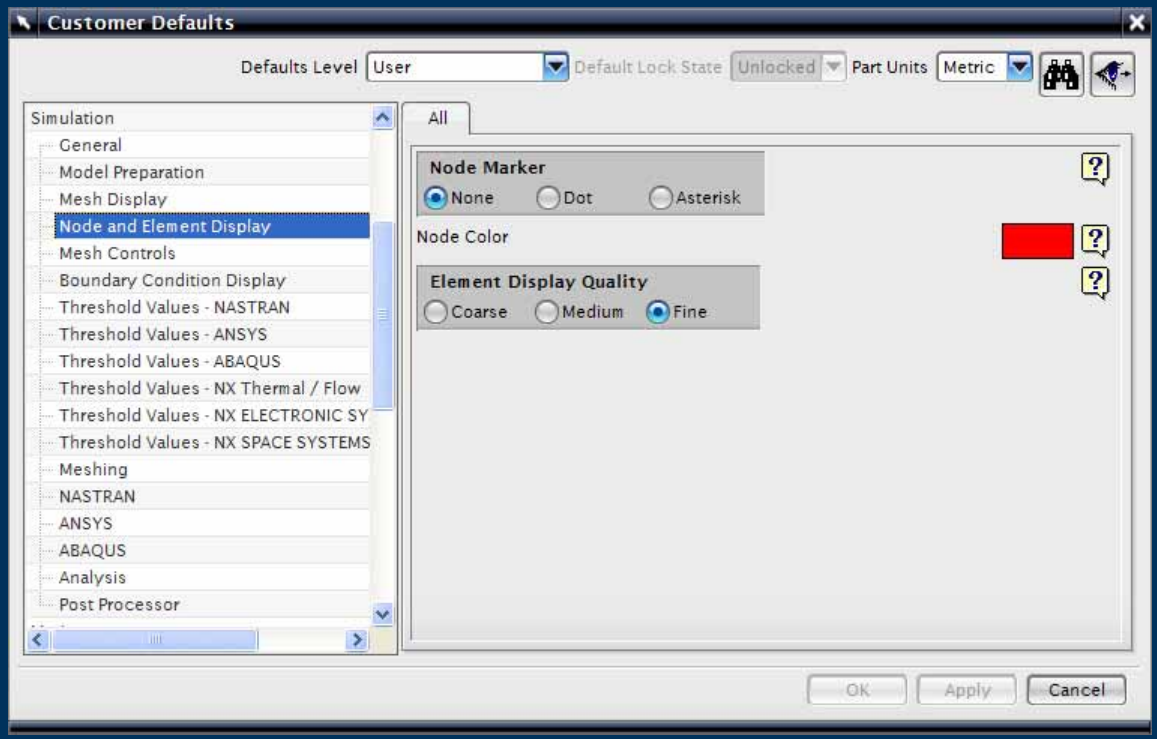
# Customer Defaults – Mesh Display




- Mesh Display
  - Default Mesh displays for different mesh types
  - Color, Line width, Shrink etc

SIEMENS

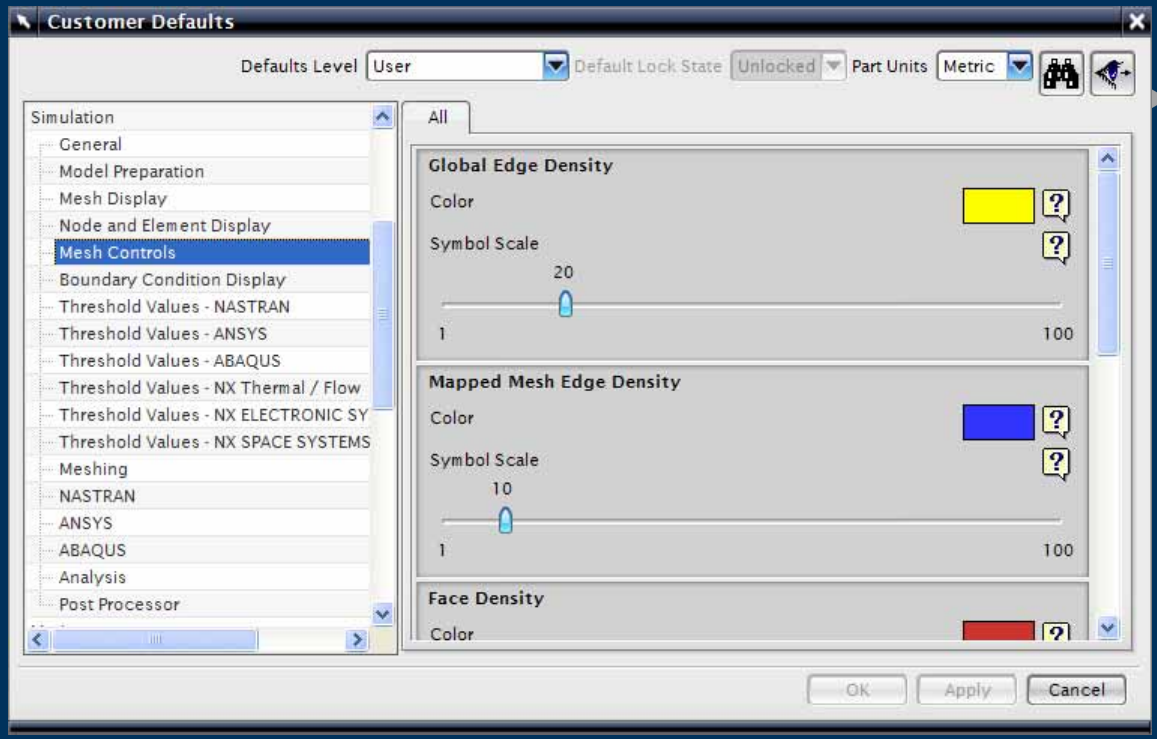
# Customer Defaults – Node & Element Display




- Node & Element Display
  - Node style
  - Element display quality

SIEMENS

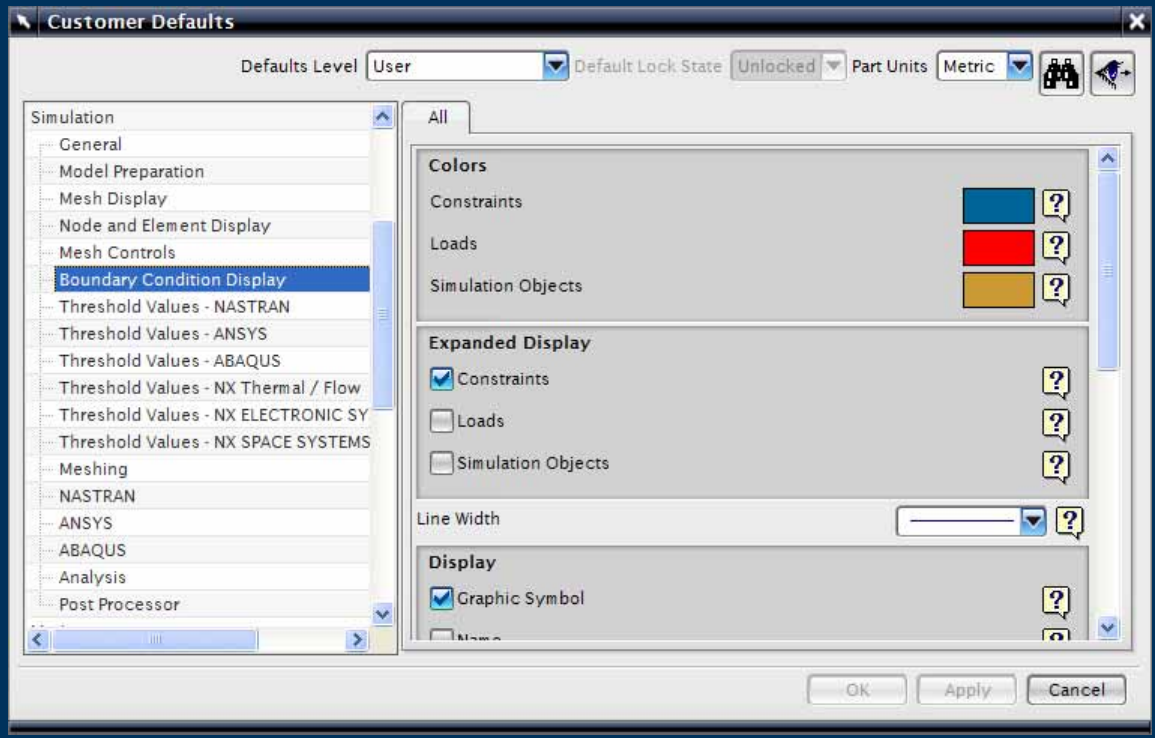
# Customer Defaults – Mesh Controls




Mesh Control defaults

SIEMENS

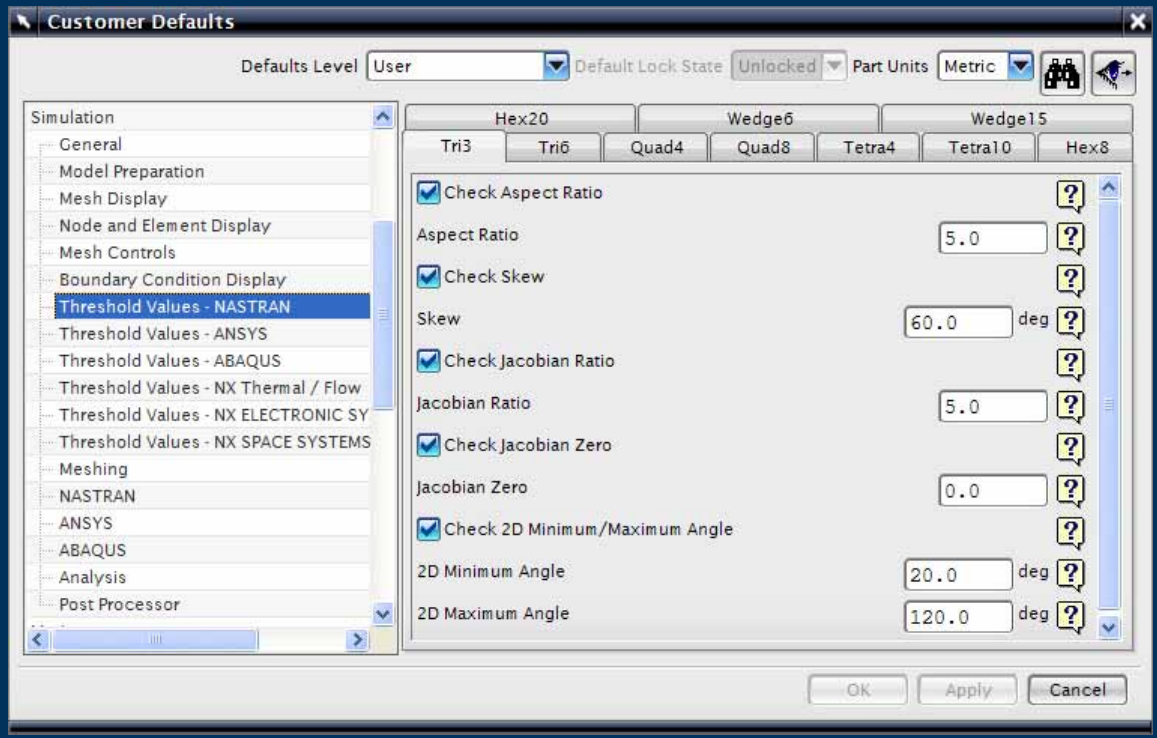
# Customer Defaults – Boundary Condition Display




- Boundary Condition Display
  - Defaults for color, line width and style

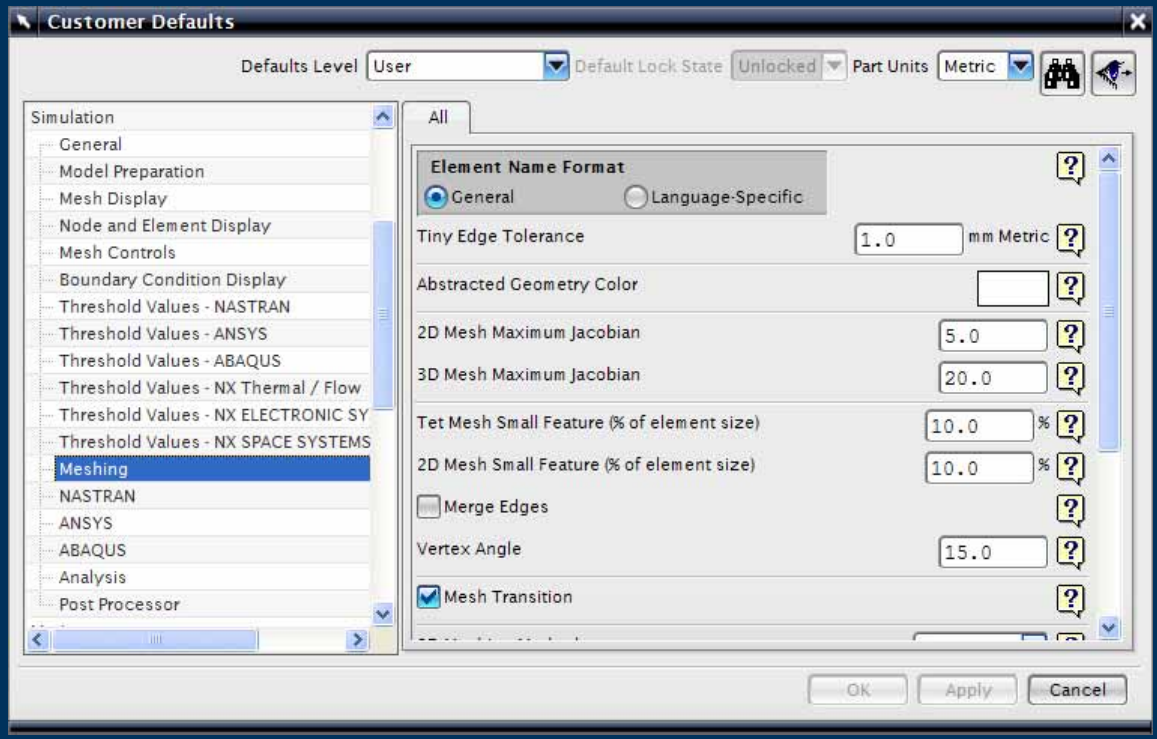
SIEMENS

# Customer Defaults – Threshold Values Nastran




- Threshold Values – Nastran
  - Element quality check threshold values for Tri, Quad, Tet, Hex and Wedge element types

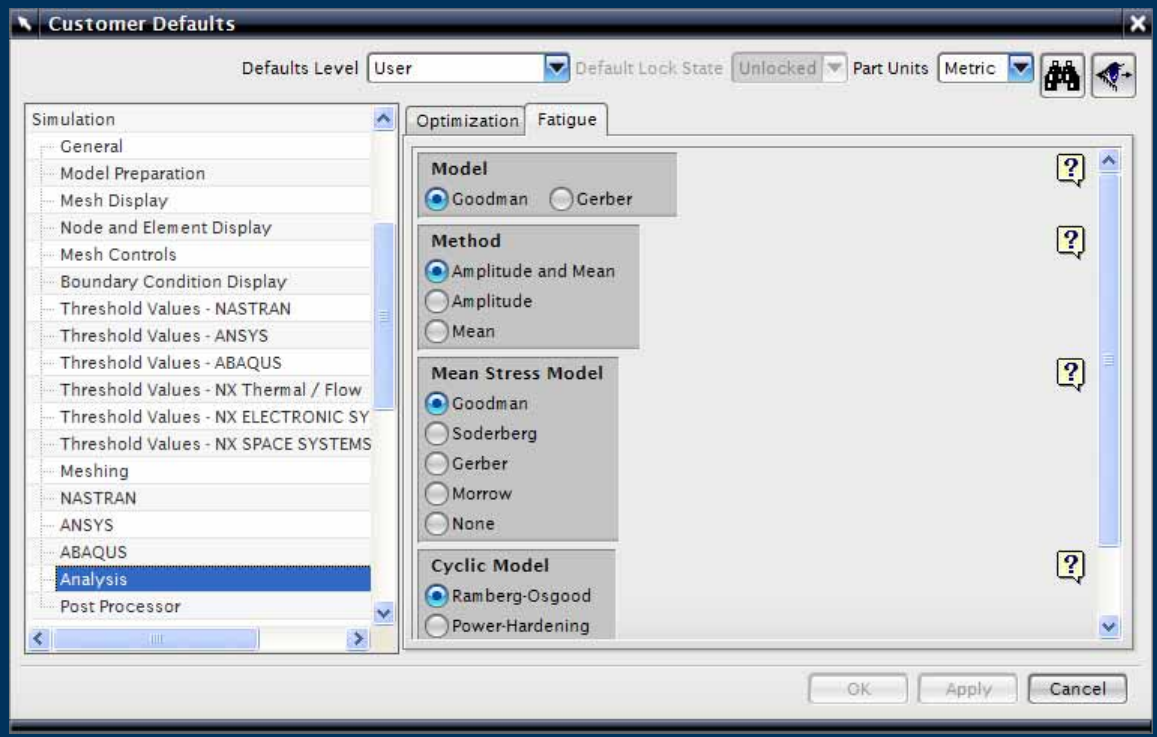
# Customer Defaults – Meshing




- Meshing
  - General defaults for the meshing task

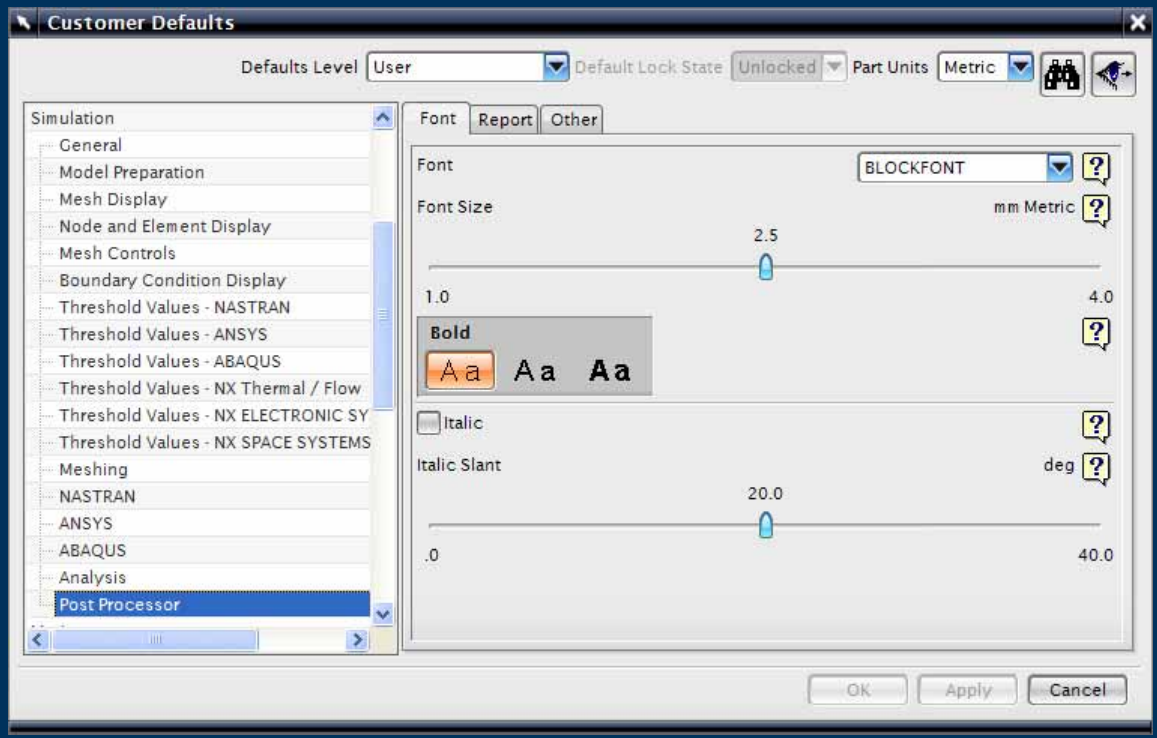
SIEMENS

# Customer Defaults – Analysis




- Analysis
  - General and specific for Optimization and Fatigue (Durability)

# Customer Defaults – Post Processor




- Post Processor
  - Defaults for text display
  - Report file names

SIEMENS

# Thank You