

**SIEMENS**

# NX Advanced FEM (Version NX 5)

# 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

## 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

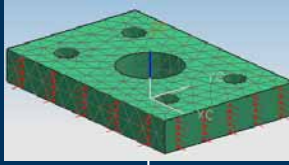
**SIEMENS**

# NX Advanced FEM File Organisation

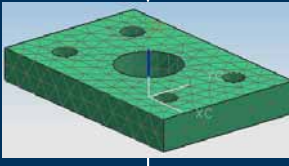
# Basic file structure

- ## 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

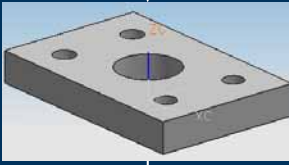
Simulation part



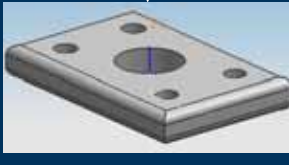
FEM Part



Idealize Part



Master Part



Name	Environment
sim 1	Active: NX NASTRAN - Structural Default: NX NASTRAN - Structural
fem 1	
blade_fem_i	
blade.prt	
Polygon Geometry	
3D Collectors	
Connection Collectors	
Simulation Object Container	
Load Container	
Pressure(1)	
Constraint Container	
Fixed(1)	
Solution 1	NX NASTRAN - Structural
Simulation Objects	
Constraints	
Fixed(1)	
Subcase - Static Loads 1	
Loads	
Pressure(1)	
Results	

Simulation Name: model1\_sim1.sim  
FEM Name: model1\_fem1.fem  
Idealized Part Name: model1\_fem1\_i.prt

Associate to part

Create Idealized Part

model1

**Bodies to use**  
 Use all bodies  Select bodies  No bodies

Geometry Options...

Default Language:  
Solver: NX NASTRAN  
Analysis Type: Structural

Description:

OK Cancel

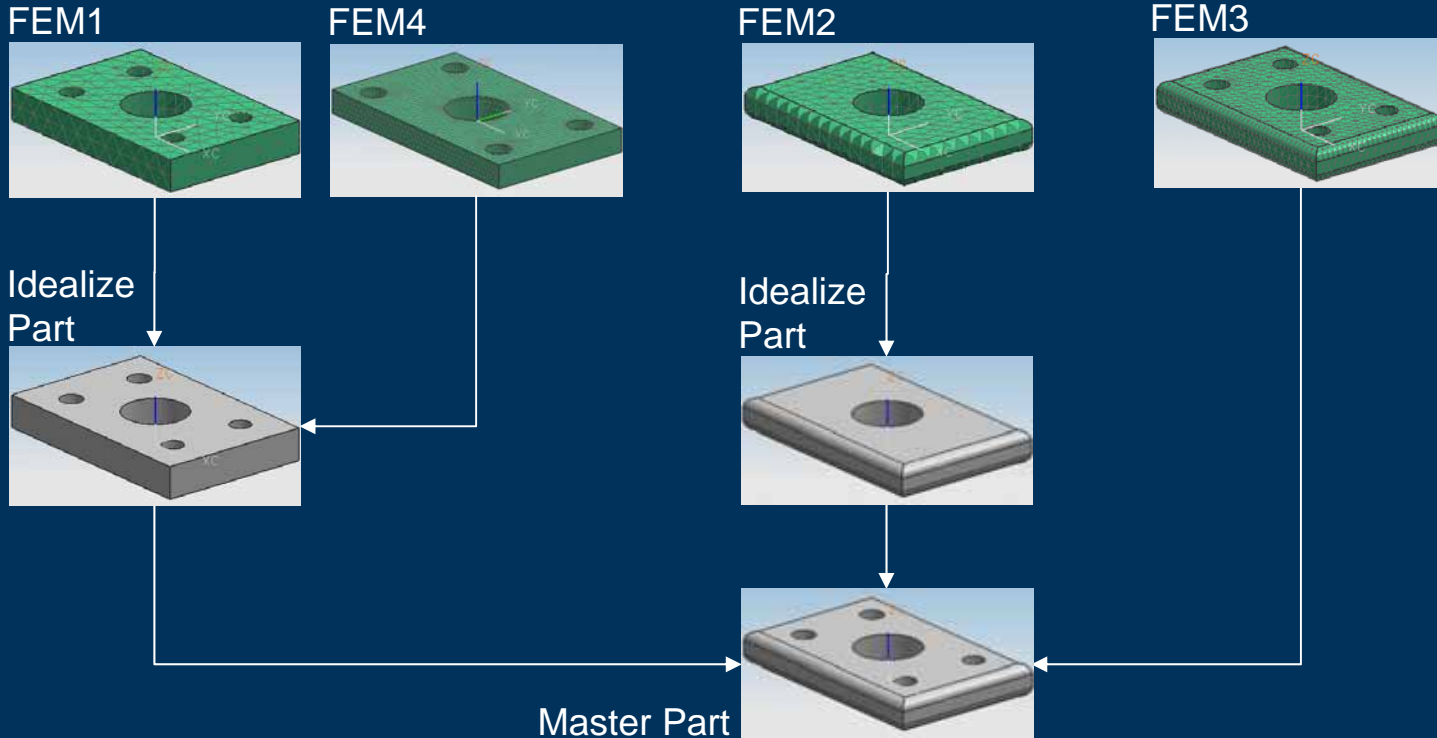
Note – Write Access to Master Part is not required



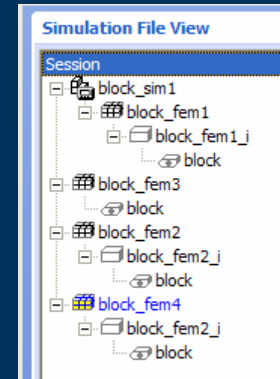
# Multiple Idealize and Multiple FEM's

## Benefits

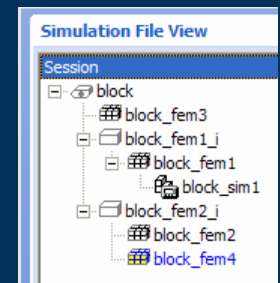
- ▶ Multiple analyses for same CAD part
- ▶ Multiple representations for different analysis needs from same CAD part



## CAE user view



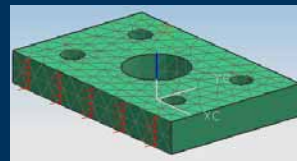
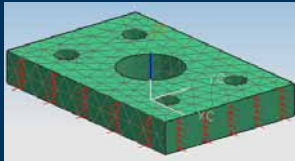
## CAD user view



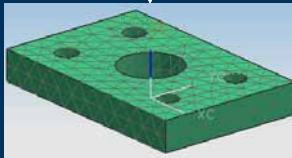
# Multiple SIM's – Physical Property Override

SIM1 – Generic Steel  
(Inherited from Master Part)

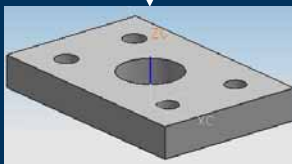
SIM2 – AISI\_STEEL\_1008+HR



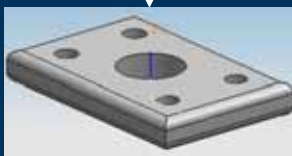
FEM1



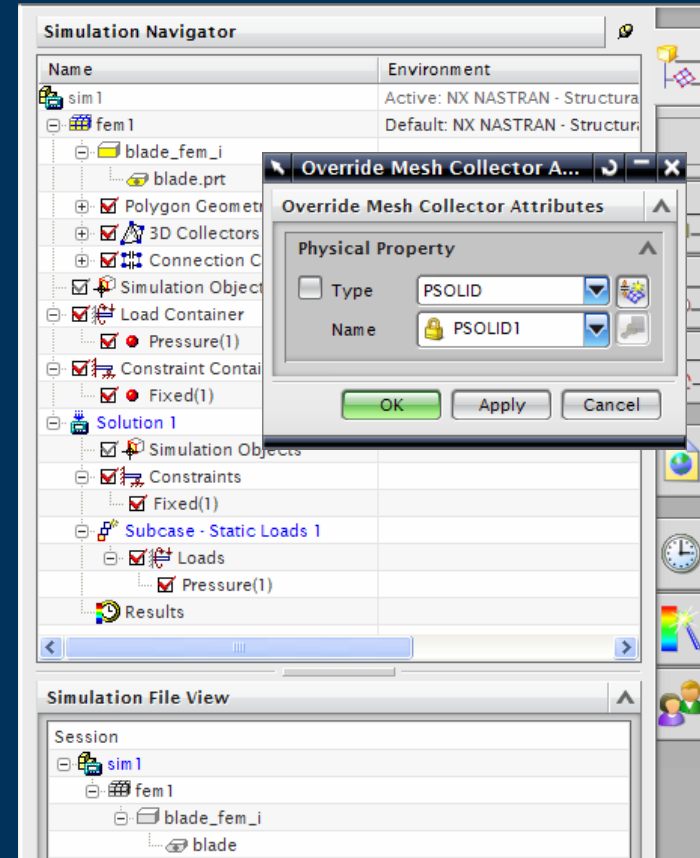
Idealize Part



Master Part

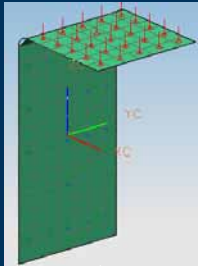


- Benefits**
- ▶ Quickly and easily explore effects of different materials
  - ▶ “What if” and sensitivity studies

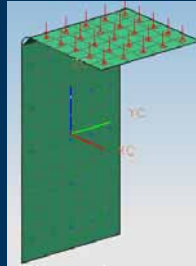


# Multiple SIM's – Properties & Material Override

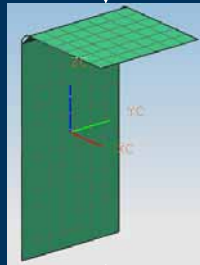
SIM1 – Inherited Properties



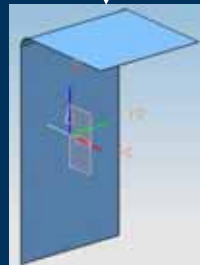
SIM2 – Properties & Material Overrides



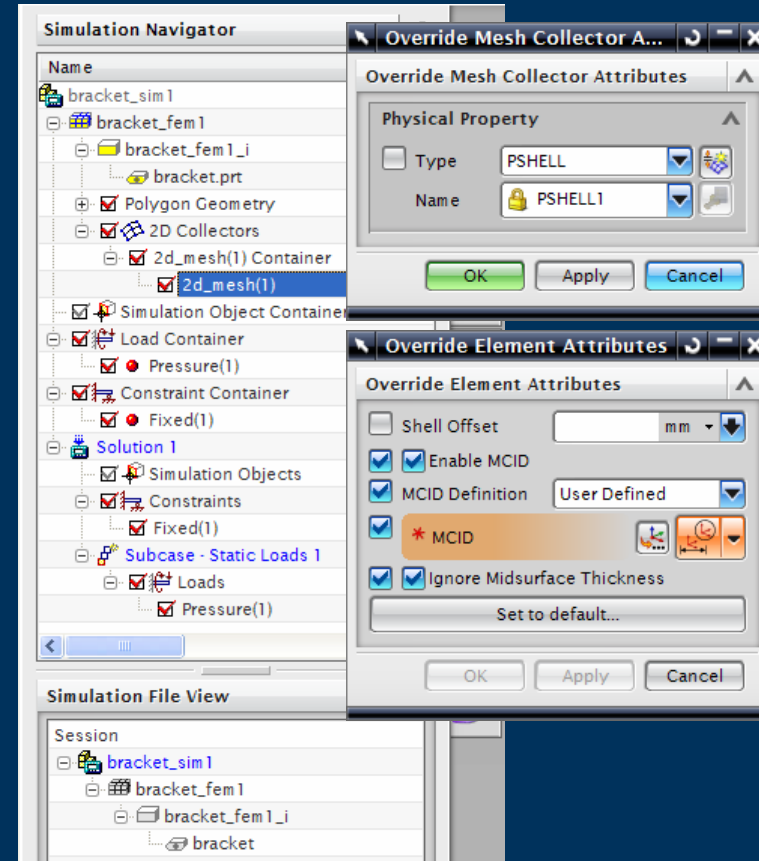
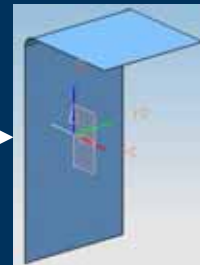
FEM1



Idealize Part



Master Part



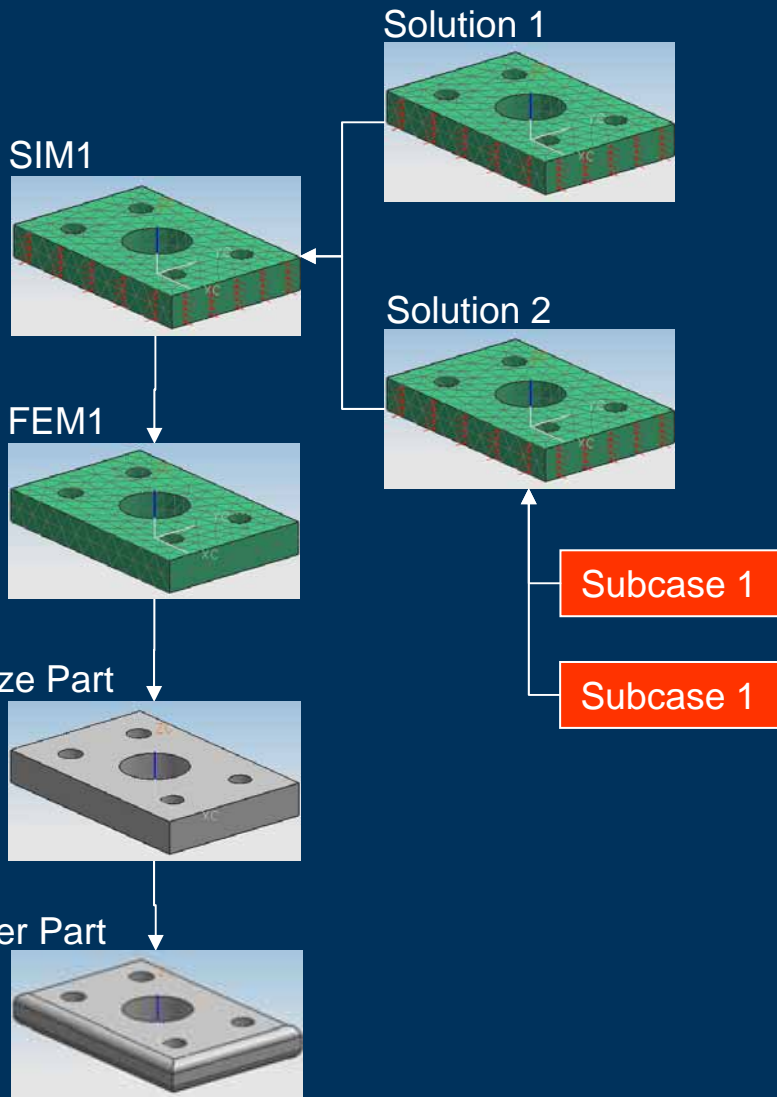
## Benefits

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

# Multiple Solutions and Subcase's

## Benefits

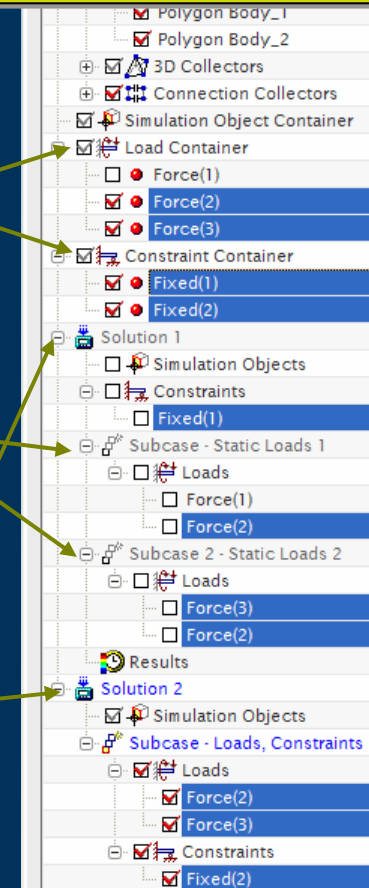
- ▶ Quickly and easily explore effects of different loading conditions
- ▶ Efficient analysis in complex environments



Simulation Containers

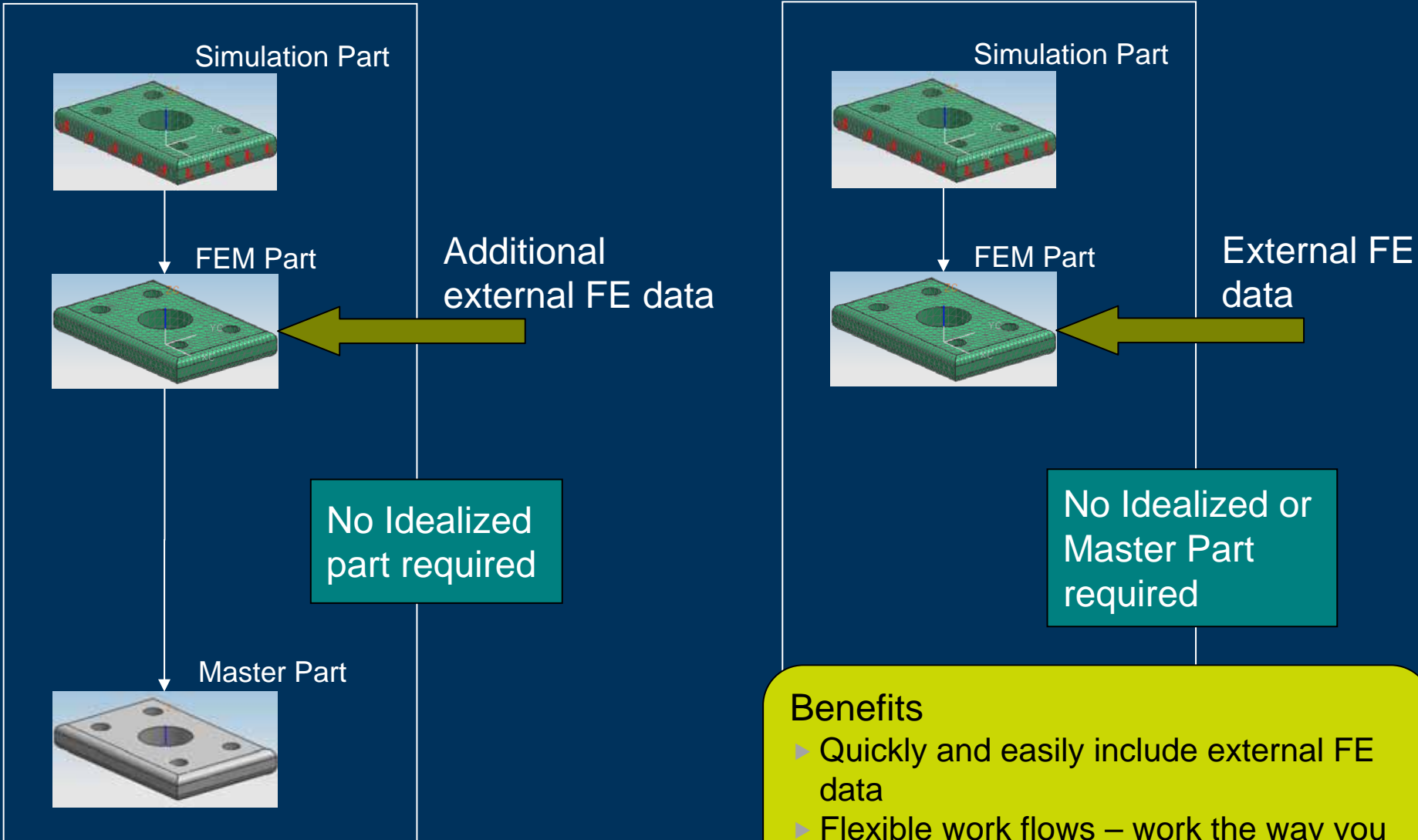
Multiple Subcases

Multiple Solutions



Simulation Container

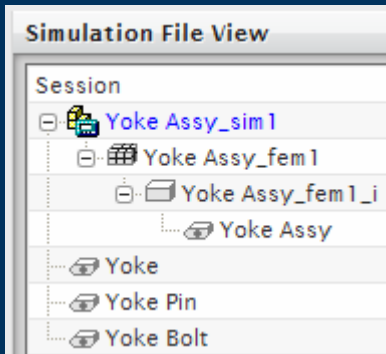
# Variations



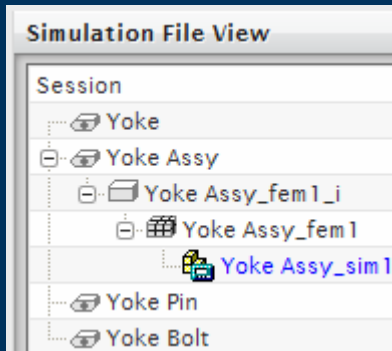
**SIEMENS**

# Model Interaction

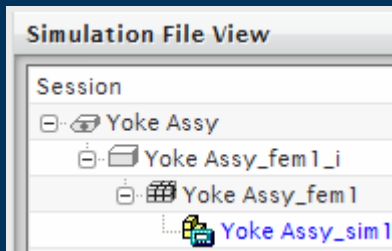
# Simulation Navigator – File View



Simulation Centric View

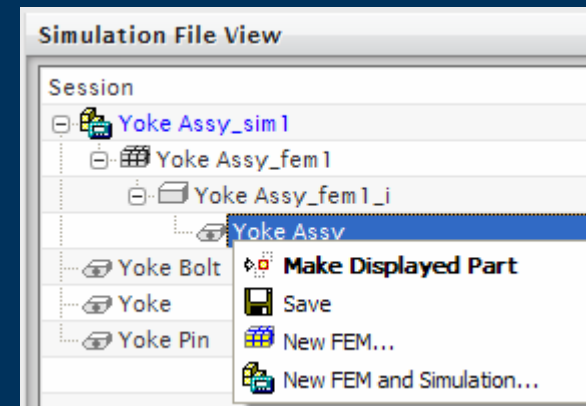


Design Part Centric View



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



# Simulation Navigator – Easy Management

Simulation Centric File View

Active Solver Environment

Name	Environment	Description
block_sim 1	Active: NX NASTRAN - Structural	
block_fem 1	Default: NX NASTRAN - Structural	
block_fem 1_i		
block.prt		
<input checked="" type="checkbox"/> Polygon Geometry		
<input checked="" type="checkbox"/> 3D Collectors		
<input checked="" type="checkbox"/> Simulation Object Container		
<input checked="" type="checkbox"/> Load Container		
<input checked="" type="checkbox"/> Force(1)		
<input checked="" type="checkbox"/> Bearing(1)		
<input checked="" type="checkbox"/> Constraint Container		
<input checked="" type="checkbox"/> Fixed(1)		
<input checked="" type="checkbox"/> Solution 1	NX NASTRAN - Structural	SESTATIC 101 - Single Constraint
<input checked="" type="checkbox"/> Simulation Objects		
<input checked="" type="checkbox"/> Constraints		
<input checked="" type="checkbox"/> Fixed(1)		
<input checked="" type="checkbox"/> Subcase - Static Loads 1		
<input checked="" type="checkbox"/> Loads		
<input checked="" type="checkbox"/> Force(1)		
<input checked="" type="checkbox"/> Bearing(1)		

Mesh Out-of-Date Symbol

Hide/Show of Polygon models and Meshes during selection

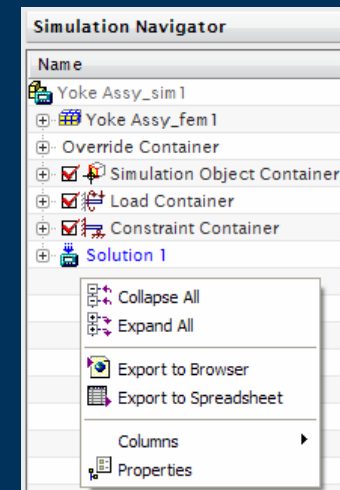
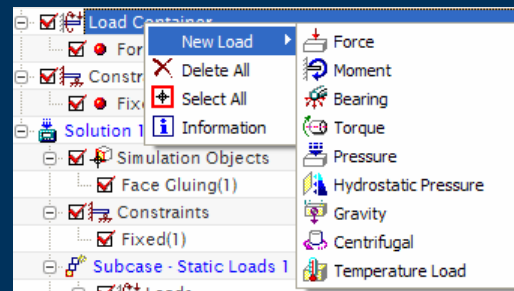
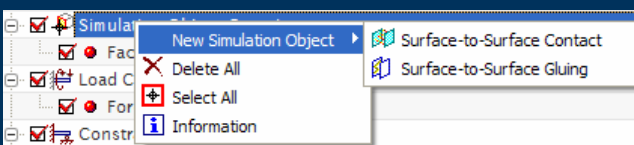
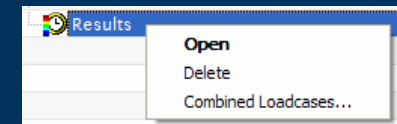
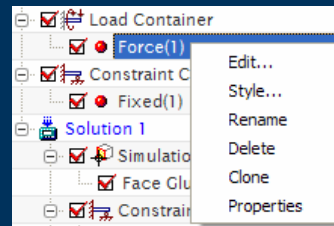
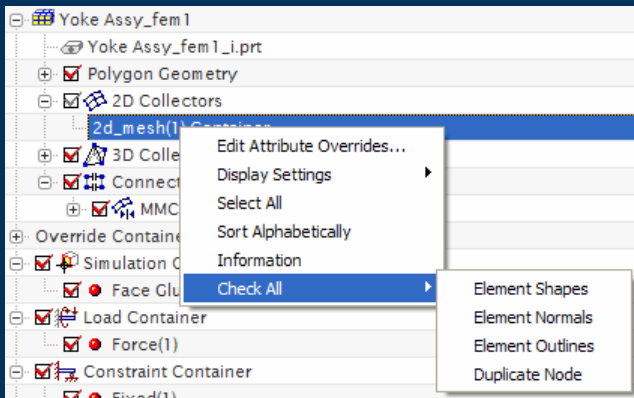
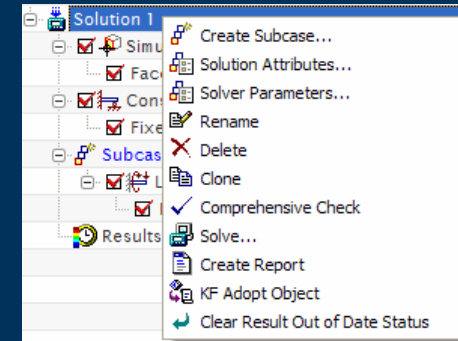
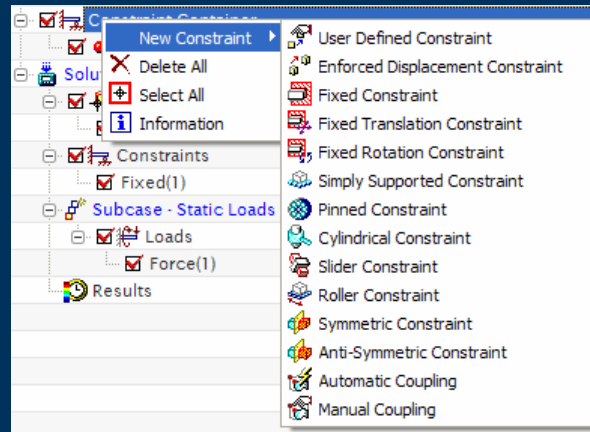
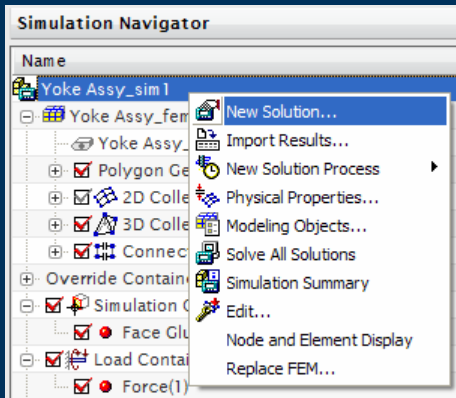
Containers to Organise related CAE Data

Drag 'n' Drop from Containers to Solution

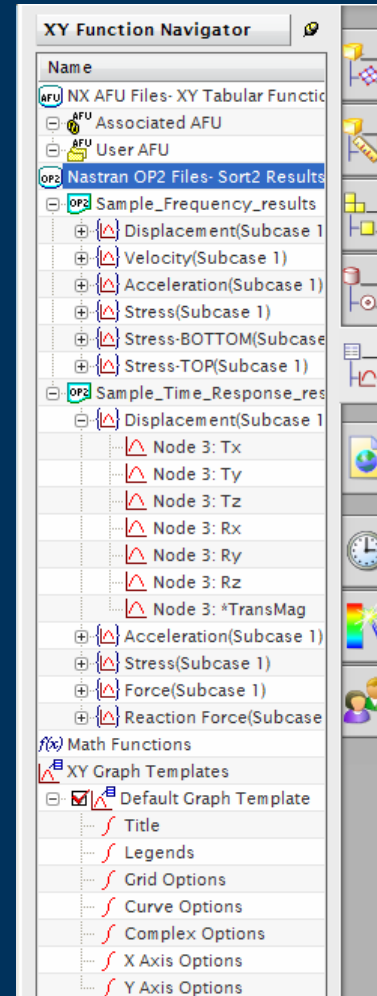
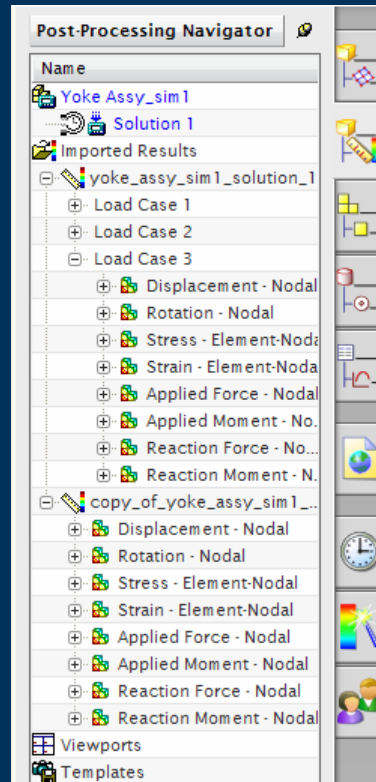
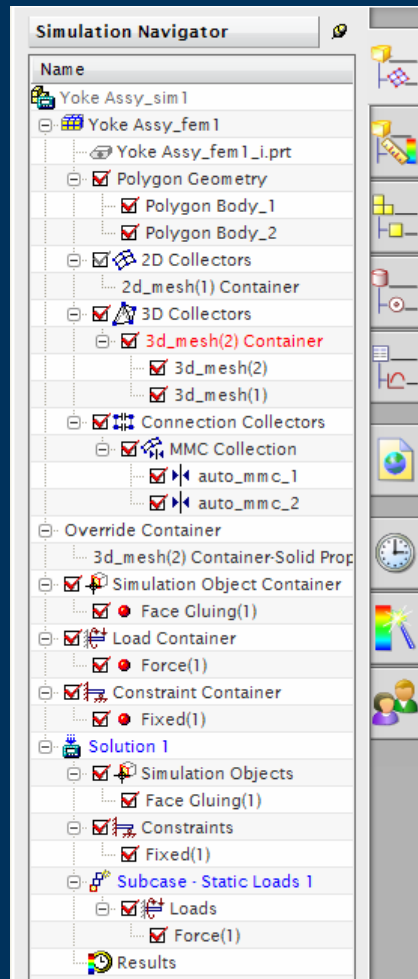


# Simulation Navigator – Easy Management

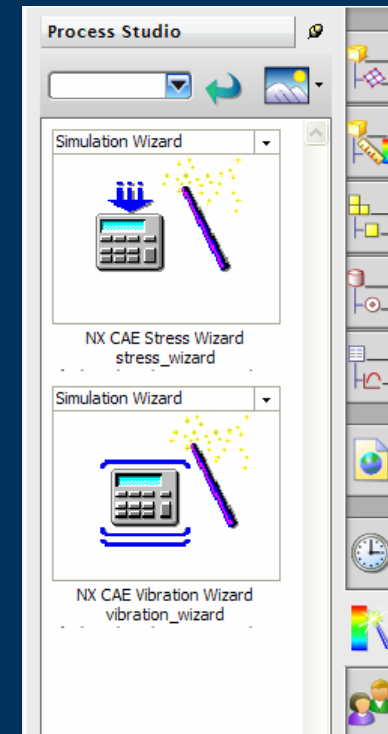
## ► RMB Operations Directly from Navigator



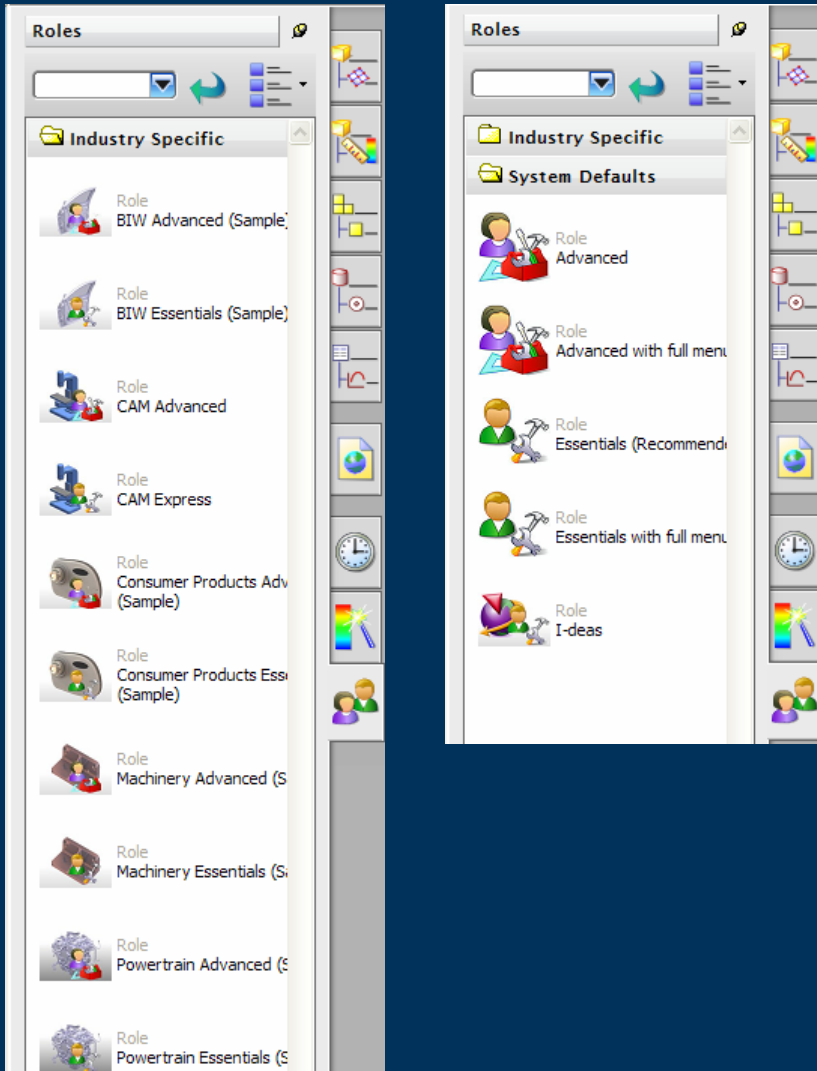
# Simulation Navigator – Resource Bars



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

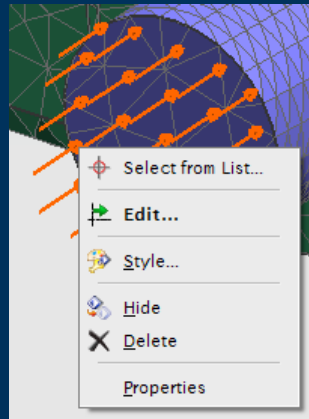
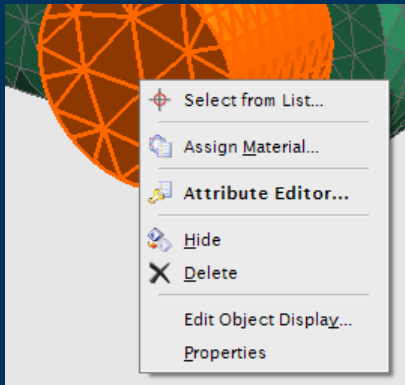


# Simulation Navigator – Resource Bars

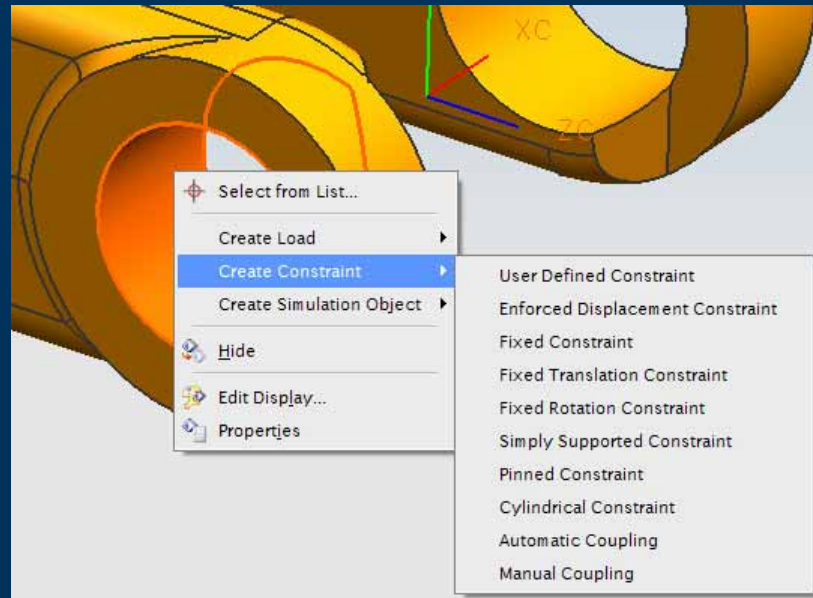
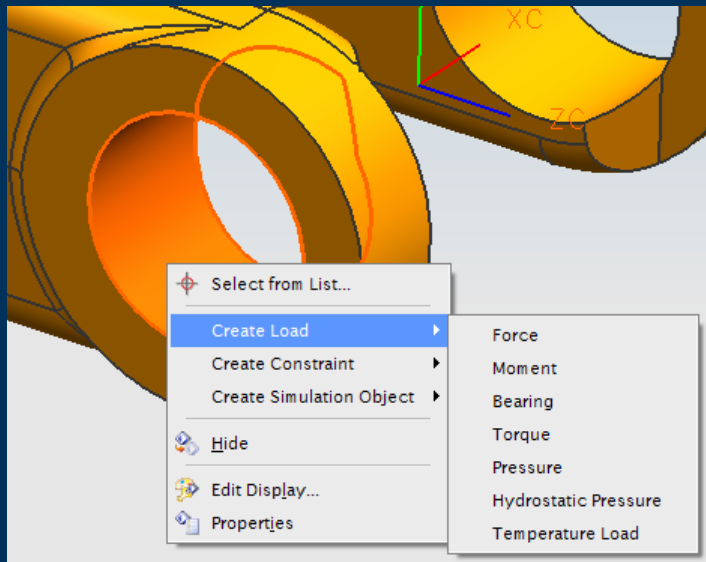


- ▶ Roles
  - ▶ Industry Specific
  - ▶ Advanced & Essentials
- ▶ General
  - ▶ Assembly Navigator
  - ▶ Part Navigator
  - ▶ History
  - ▶ Internet Explorer
  - ▶ On-Line Help

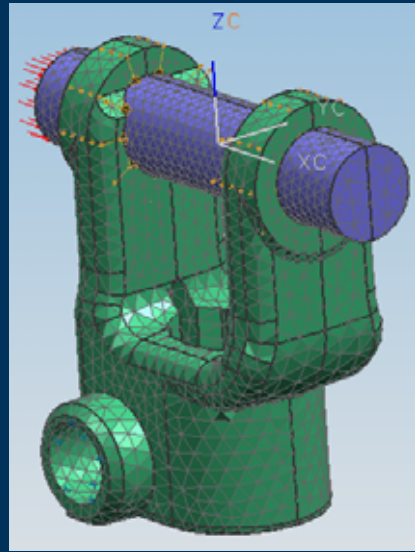
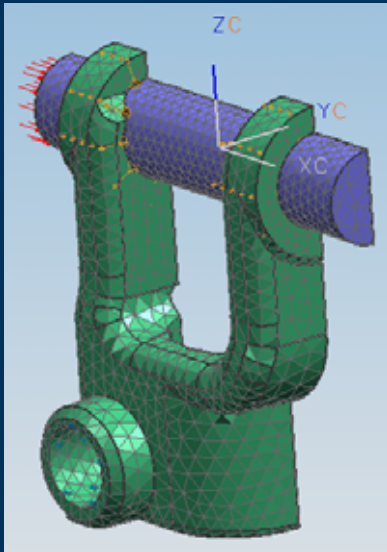
# Interaction – RMB Over Screen Model



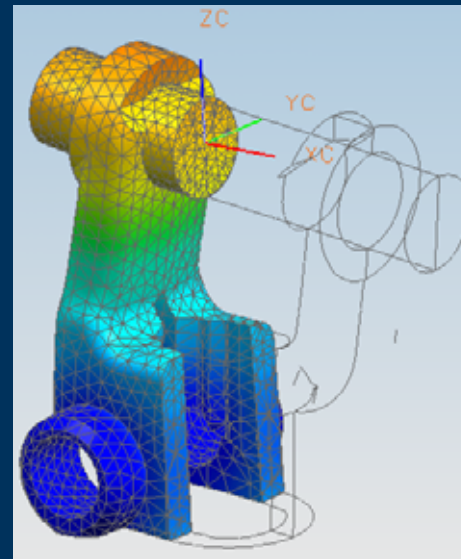
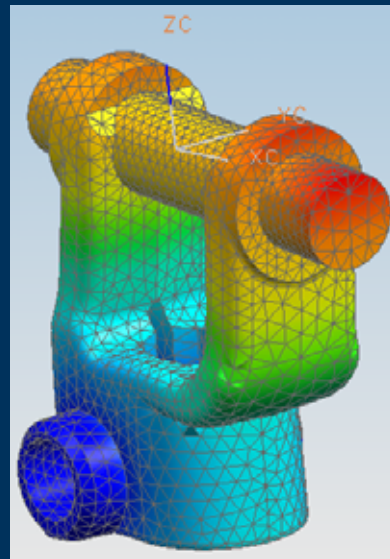
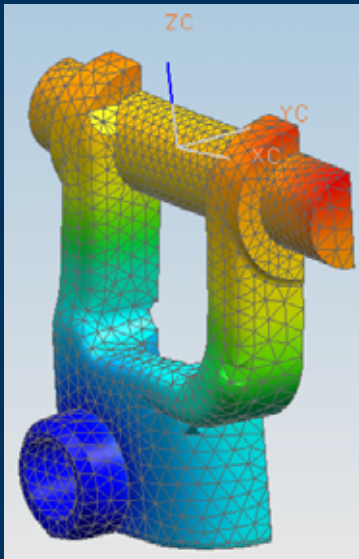
- ▶ RMB Over Screen Entities enables fast access to functionality applicable to the highlighted item



# Mirror Display



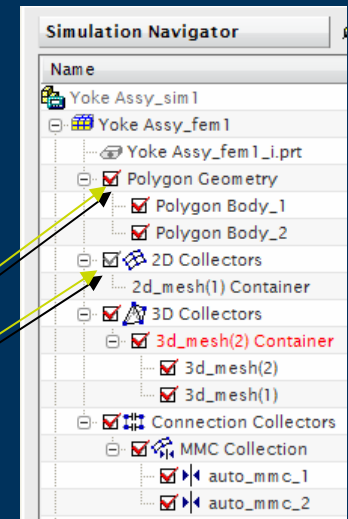
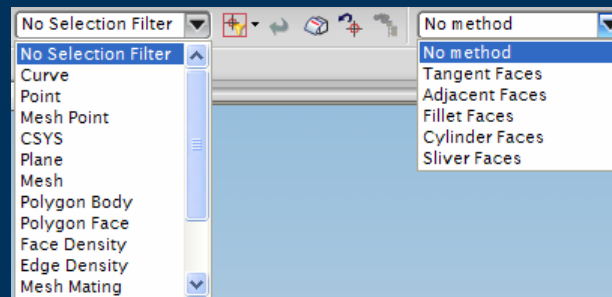
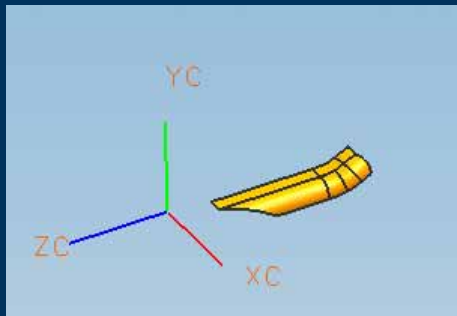
- ▶ Mirror Display is powerful for visualising Symmetric models
- ▶ Mirror Plane can be set anywhere
- ▶ Post View Settings work with the Mirror Display



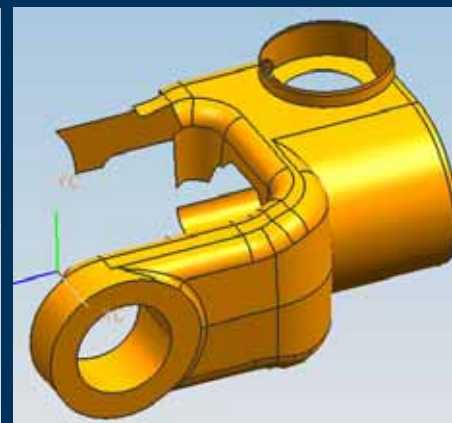
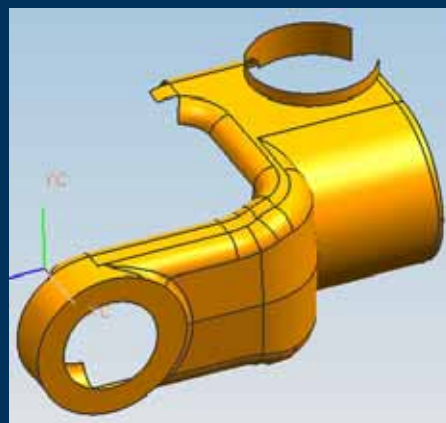
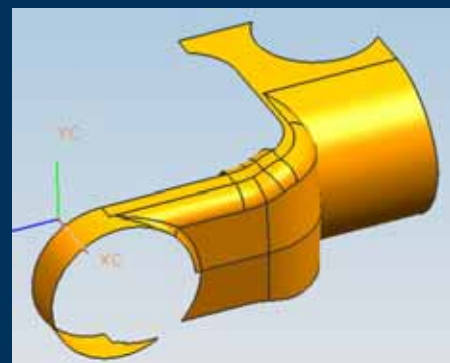
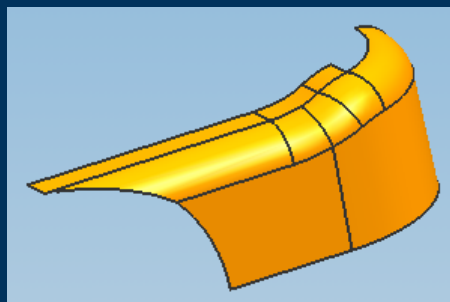
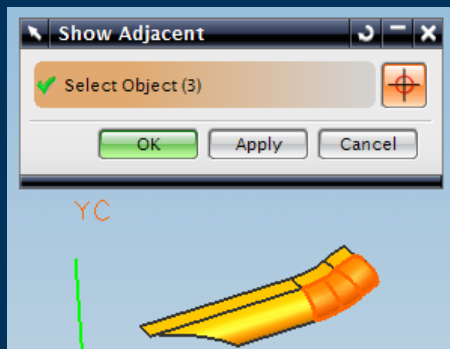
# Model Interaction – Show Only



- ▶ 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

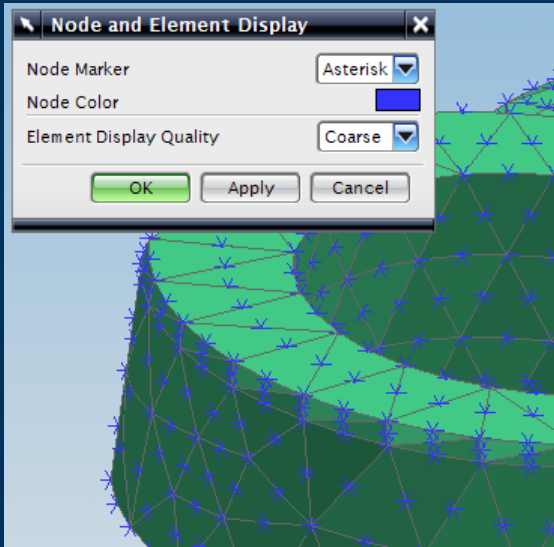


## Model Interaction – Show Adjacent



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

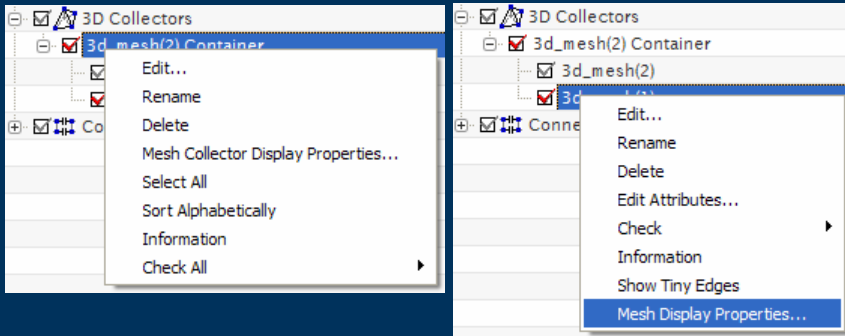
# Model Interaction – Node Display



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

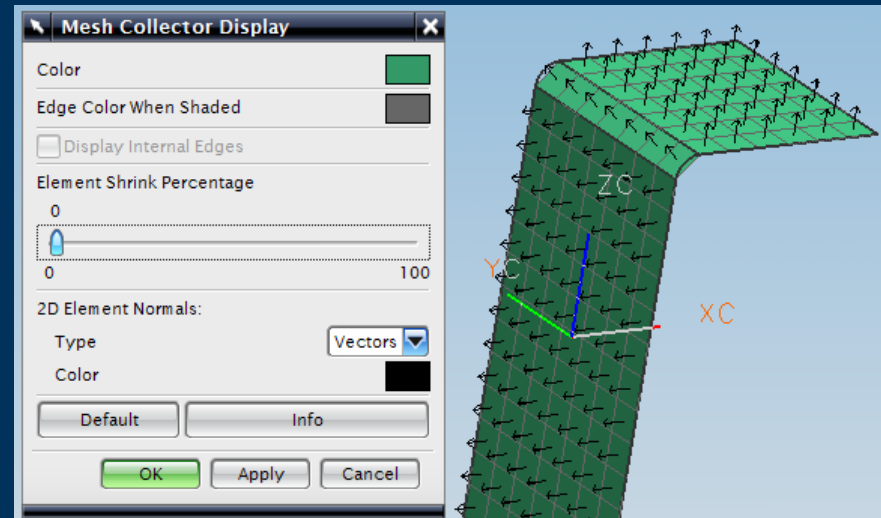
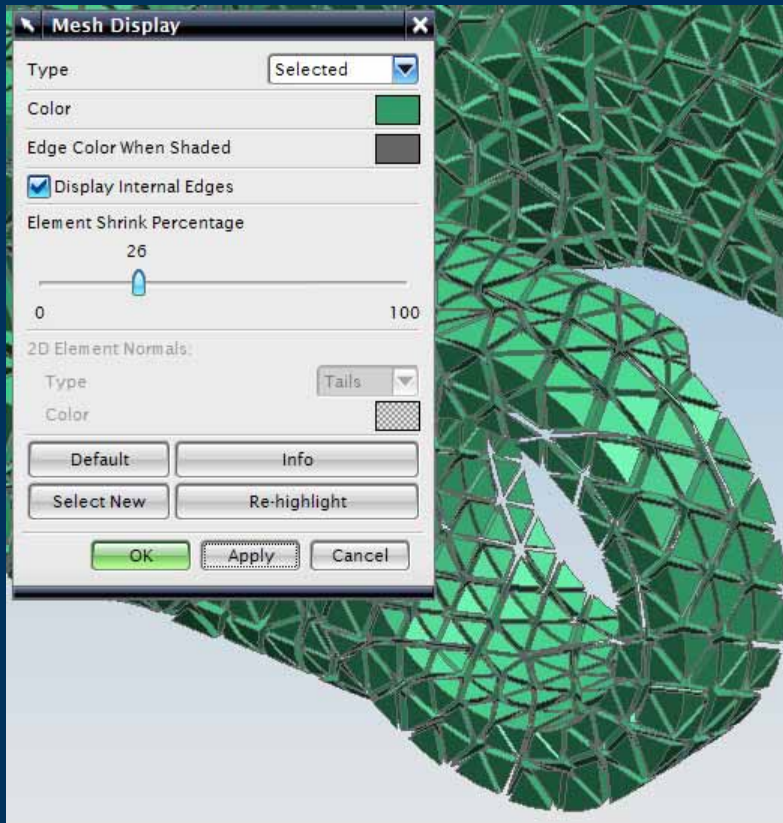


# Model Interaction – Mesh Display

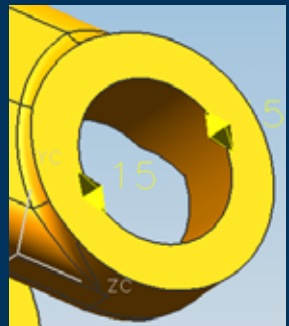
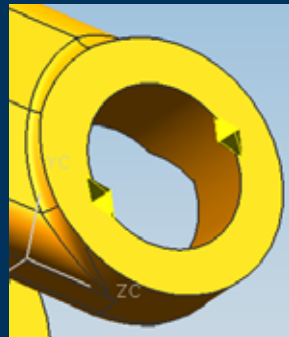
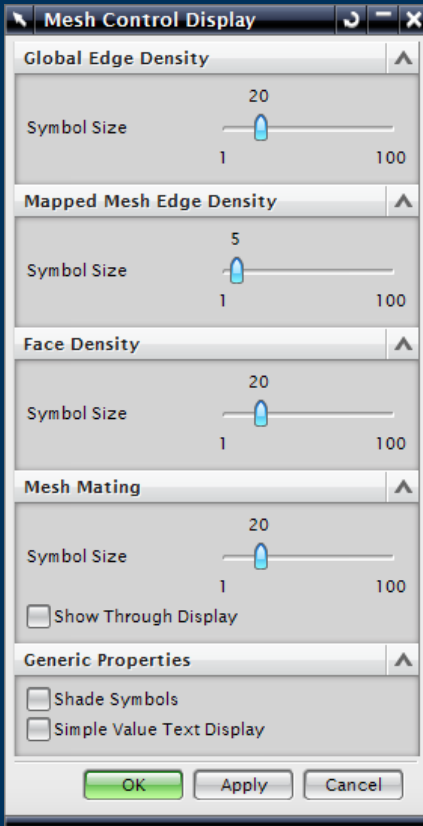


- ▶ Mesh Display
  - ▶ By Collector
  - ▶ By Mesh
- ▶ Mesh Display options

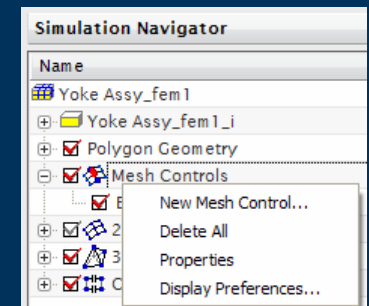
- ▶ Colour
- ▶ Edge Colour
- ▶ Shrink Percentage
- ▶ 2d Element Normals



# Model Interaction – Mesh Control Display



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



**SIEMENS**

# Solver Language Environment

# 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
<b>NX Nastran</b>	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
Axisymmetric Structural	SESTATIC 101 - Single Constraint	
	SESTATIC 101 - Multiple Constraint	
	NLSTATIC 106	
Axisymmetric Thermal	NLSCSH 153	
Solver	Analysis Type	Solution Type
<b>NX Nastran Design</b>	Structural	Linear Statics - Single Constraint
		Therma;
		Linear Buckling
Thermal	Thermal	
Solver	Analysis Type	Solution Type
<b>NX Thermal-Flow</b>	Thermal	Thermal
		Advanced Thermal
		Flow
		Advanced Flow
		Coupled Thermal-Flow
		Thermal-Flow
		Advanced Thermal-Flow
		Complete
		Axisymmetric Thermal
		Axisymmetric Thermal
		Advanced Axisymmetric Thermal
Mapping		
Axisymmetric Mapping		
Thermal-Flow		
Thermal		
Solver	Analysis Type	Solution Type
<b>NX Space Systems Thermal</b>	Thermal	Space Systems Thermal
		Thermal
Solver	Analysis Type	Solution Type
<b>NX Electronic Systems Cooling</b>	Coupled Thermal-Flow	Electronic Systems Cooling
		Advanced Electronic Systems Cooling
		Thermal-Flow

# 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

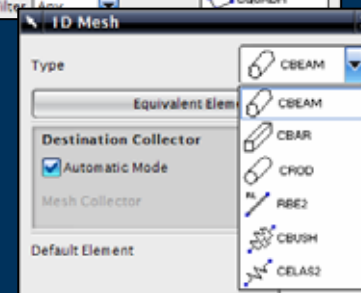
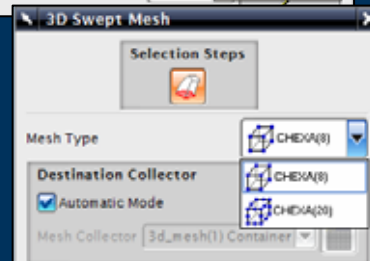
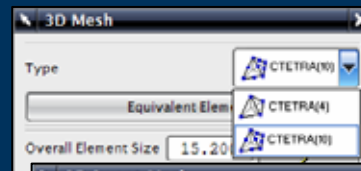
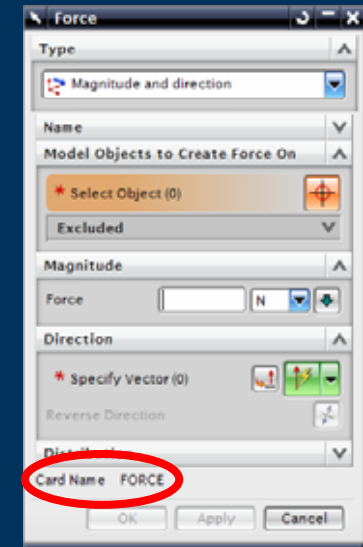
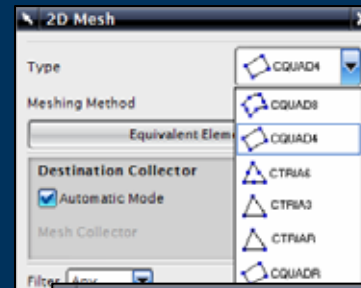
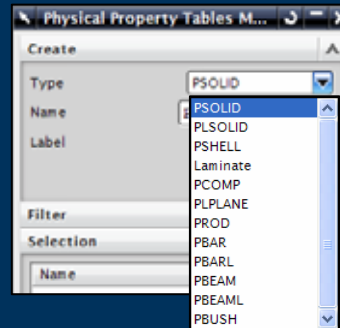
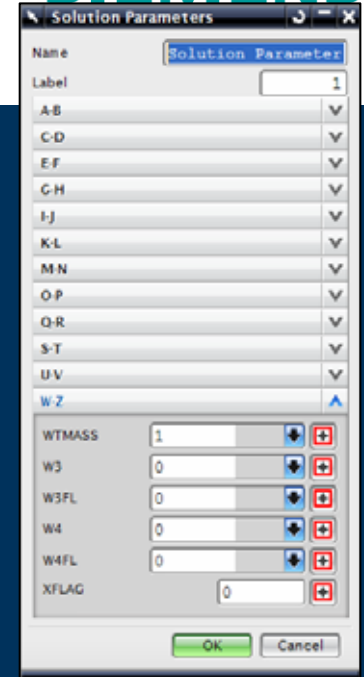
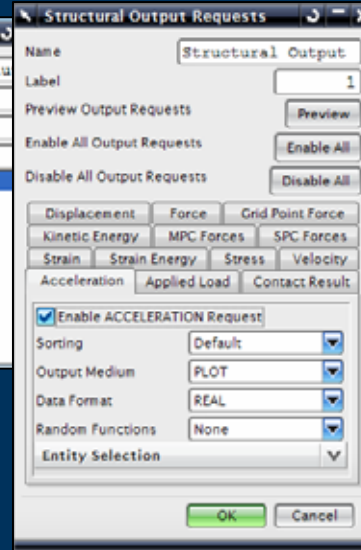
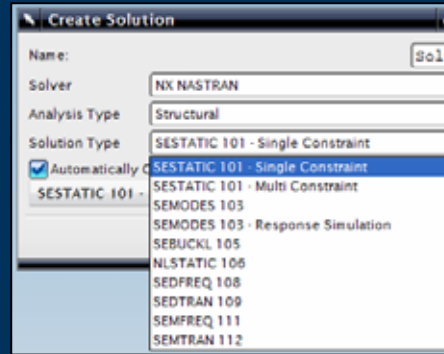
- ▶ Import of I-deas Universal file with CAE data only

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

# “NX Nastran Environment”

## – UI Based on Solver/Solution

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



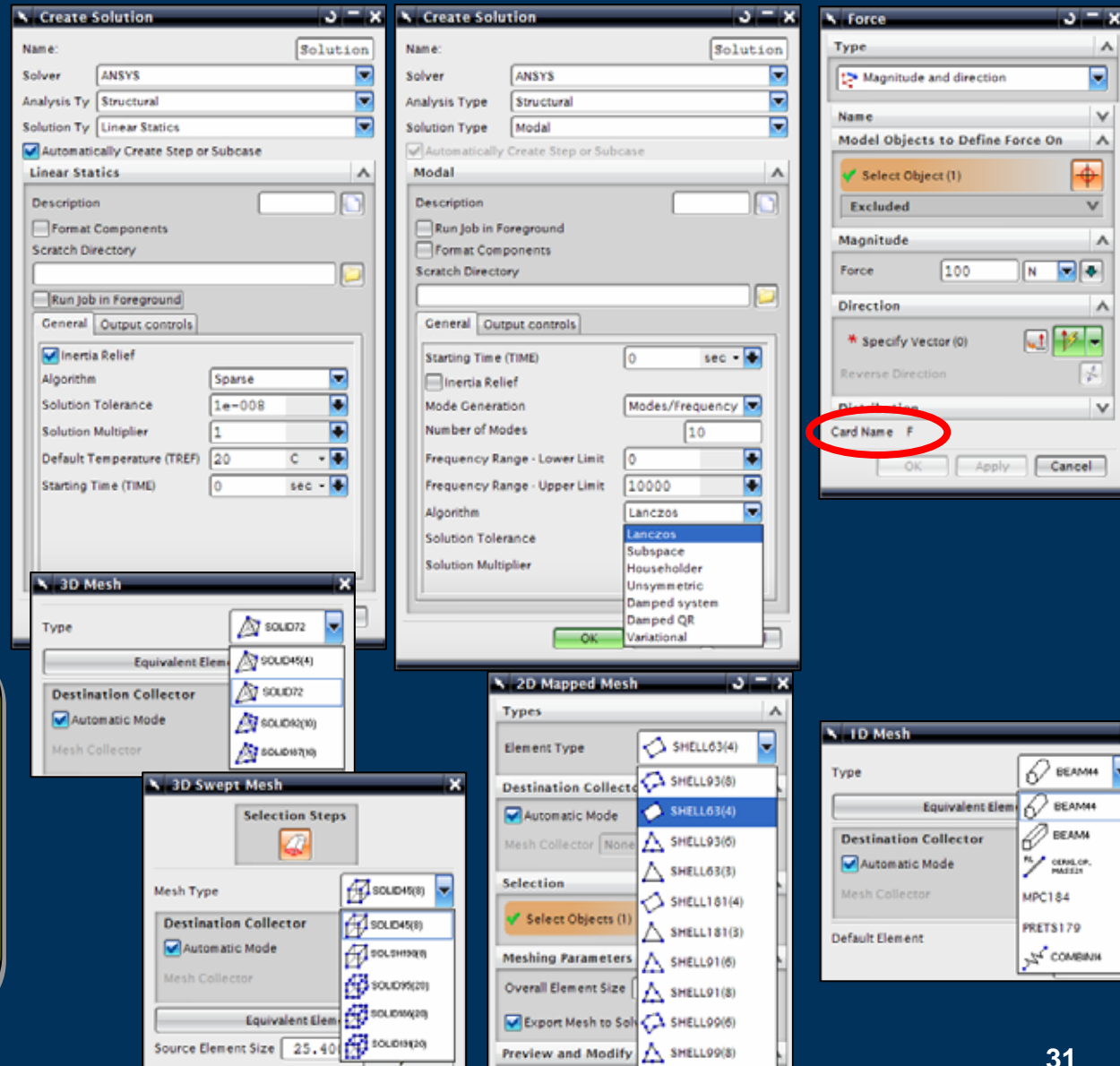
### Benefits

- ▶ 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

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



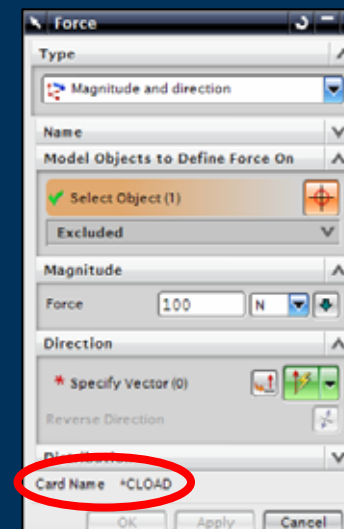
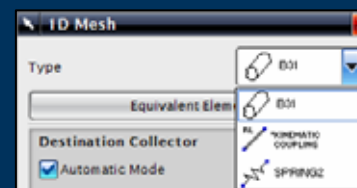
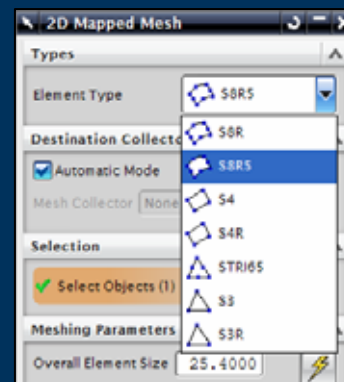
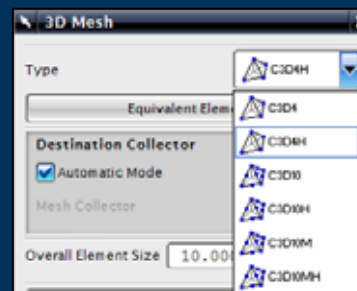
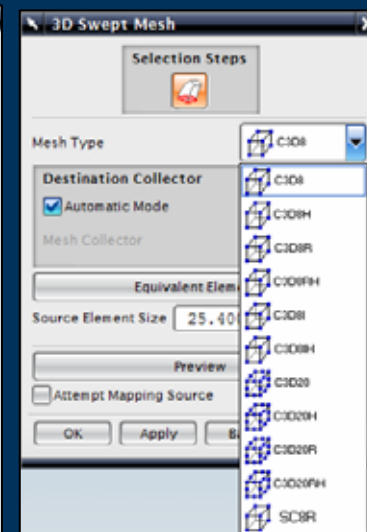
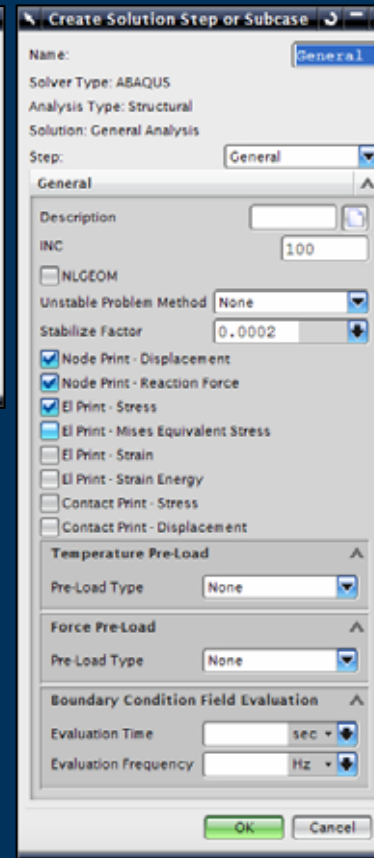
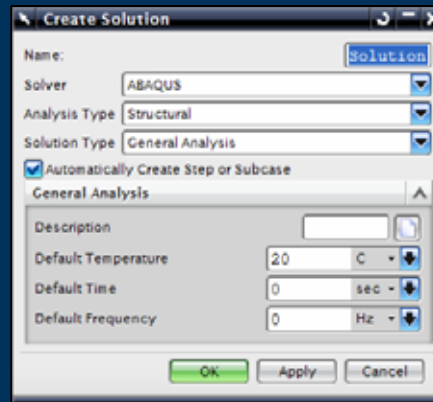
### Benefits

- ▶ 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

- ▶ 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



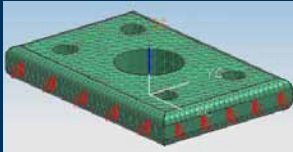
**SIEMENS**

Master Part

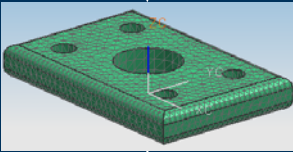
# Master Part

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

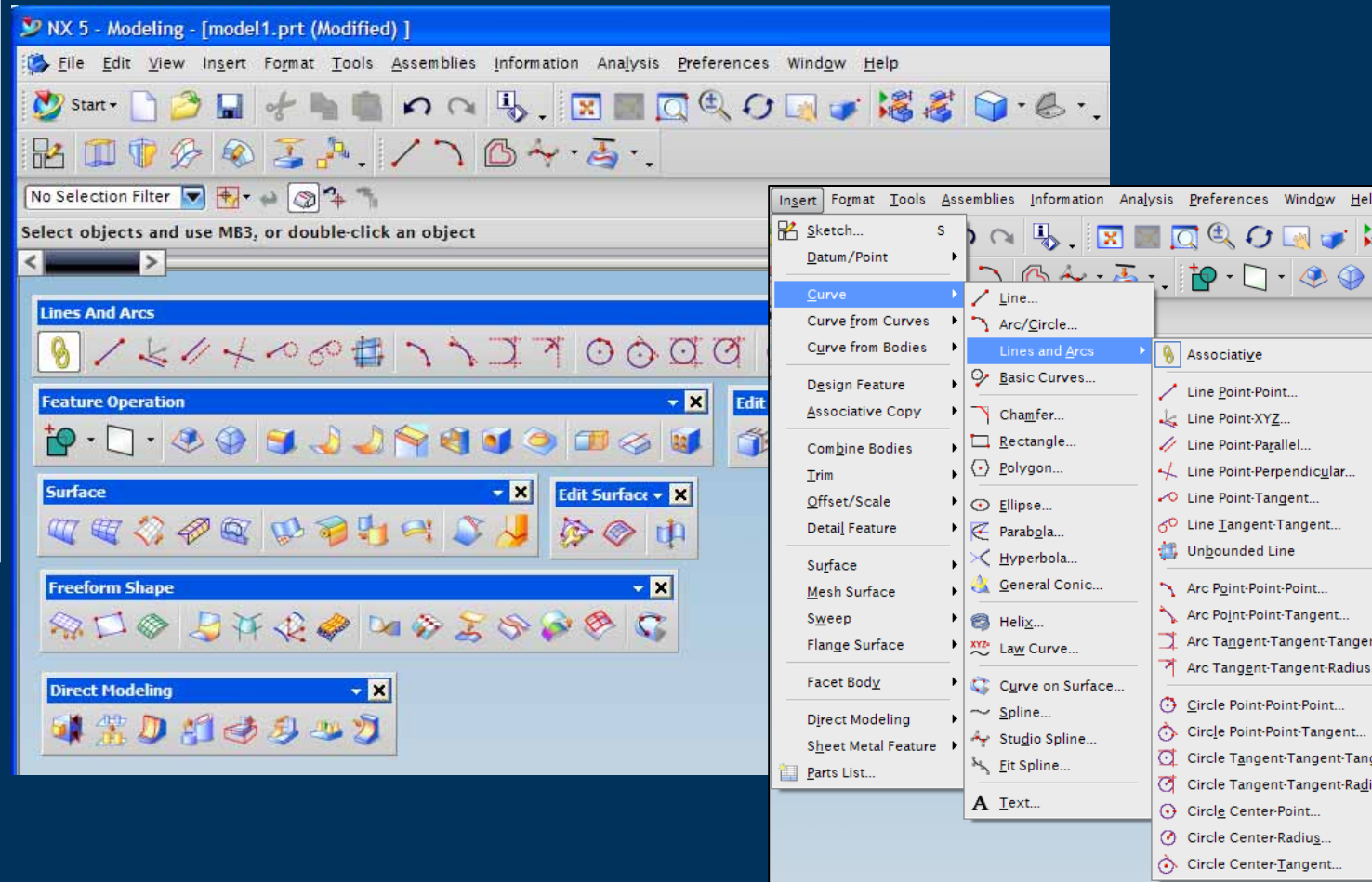
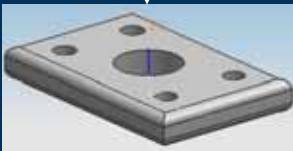
Simulation Part



FEM Part



Master Part



# Material Property – Library

▶ Import from Material Property Library

▶ Metals

▶ Plastics

▶ Isotropic

▶ Orthotropic

▶ Anisotropic

▶ Fluids

▶ Assign to Parts

▶ Create new based on existing materials



**Materials**

Category: [Dropdown]

Material	Category
AISI_310_SS	METAL
AISI_410_SS	METAL
ALUMINUM_2014	METAL
ALUMINUM_6061	METAL
S/STEEL_PH15.5	METAL

Materials Inherited: [Table]

Part: [Table]

Name: AISI\_310\_SS

Category: METAL

Library Reference: 18

Isotropic | Orthotropic | Anisotropic | Fluid

**Basic Structural**

Mass Density: 7.92781e kg/mm<sup>3</sup>

Reference Temperature: [Field] C

Young's Modulus: TABLE mN/mm<sup>2</sup>(kPa)

Poisson's Ratio: TABLE

Shear Modulus: [Field] N/mm<sup>2</sup>(MPa)

Stress/Strain: [Field]

Thermal Expansion Coefficient: TABLE 1/C

OK | Apply | Back | Cancel

**Search Criteria**

Materials

Library Reference: [Field]

Name: [Field]

Category: Metals

Type: Iso Metals

Result info | Count Matches

OK | Back | Cancel

**Search Result**

Lib Ref.	Name
2	Aluminum_2014
3	Aluminum_6061
4	Brass
5	Bronze
8	Iron_Malleable
9	Iron_Nodular
10	Iron_40
11	Iron_60
13	Steel
14	Steel Rolled
16	S/Steel_PH15.5
17	AISI_410_SS
18	AISI_310_SS
20	Titanium_Alloy
21	Tungsten
22	Waspaloy
37	Aluminum_5086
38	Copper_C10100
39	Iron_Cast_G25
40	Magnesium_Cast
41	AISI_Steel_1008-HR
42	AISI_SS_304-Annealed
43	Titanium-Annealed
44	Aluminum_A356
45	Inconel_718-Aged
46	AISI_Steel_1005
47	AISI_Steel_4340
48	AISI_Steel_Maraging
49	Iron_Cast_C40
50	Iron_Cast_C60
51	Titanium_Ti-6Al-4V

**Search Result**

Lib Ref.	Name
23	ABS
24	ABS-CF
25	Acrylic
26	Nylon
27	Polycarbonate
28	Polycarbonate-GF
29	Polyethylene
30	Polypropylene
31	Polypropylene-GF
32	Polyurethane-Soft
33	Polyurethane-Hard
34	PVC
35	SMC

**Search Result**

Lib Ref.	Name
101	Water
102	Air

# 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

Isotropic | Orthotropic | Anisotropic | Fluid

Basic Structural

Mass Density: 7.92781e kg/mm<sup>3</sup>

Reference Temperature: C

Young's Modulus: TABLE mN/mm<sup>2</sup>(kPa)

Poisson's Ratio: TABLE

Shear Modulus: N/mm<sup>2</sup>(MPa)

Stress/Strain: TABLE

Thermal Expansion Coefficient: TABLE 1/C

Strength

Thermal

Electrical

Durability

Isotropic | Orthotropic | Anisotropic | Fluid

Thermal Expansion Coefficient: TABLE 1/C

Strength

Yield Strength: TABLE mN/mm<sup>2</sup>(kPa)

Ultimate Tensile Strength: TABLE mN/mm<sup>2</sup>(kPa)

Max Allowable Stress in Tension: N/mm<sup>2</sup>(MPa)

Max Allowable Stress in Compression: N/mm<sup>2</sup>(MPa)

Max Allowable in plane Shear Stress: N/mm<sup>2</sup>(MPa)

Max Allowable Strain in Tension

Max Allowable Strain in Compression

Max Allowable in plane Shear Strain

Tsai-Wu Interaction Coefficient (F12): mm<sup>4</sup>/N<sup>2</sup>

Thermal

Thermal Conductivity: TABLE microW/mm-C

Specific Heat: 5e+008 microJ/kg-K

Latent Heat: J/kg

Isotropic | Orthotropic | Anisotropic | Fluid

Max in plane Shear Stress: N/mm<sup>2</sup>(MPa)

Max in plane Shear Strain

Tsai-Wu Interaction Coefficient (F12): mm<sup>4</sup>/N<sup>2</sup>

	X	Y	Z	Unit
Young's Modulus				N/mm <sup>2</sup> (MPa)
Poisson's Ratio				
Shear Modulus				N/mm <sup>2</sup> (MPa)
Thermal Expansion Coefficient				1/C
Thermal Conductivity				W/mm-C
Max Stress in Tension				N/mm <sup>2</sup> (MPa)
Max Stress in Compression				N/mm <sup>2</sup> (MPa)
Max Strain in Tension				
Max Strain in Compression				

Mass Density: 7.92781e kg/mm<sup>3</sup>

Reference Temperature: kg/mm<sup>3</sup>

Young's Modulus: TABLE lbf-sec<sup>2</sup>/in<sup>4</sup>

Poisson's Ratio: TABLE lbm/in<sup>3</sup>

Shear Modulus: t/mm<sup>3</sup>

Isotropic | Orthotropic | Anisotropic | Fluid

Mass Density: kg/mm<sup>3</sup>

Reference Temperature: C

Specific Heat: J/kg-K

Material Moduli (N/mm<sup>2</sup>(MPa))

	1	2	3	4	5	6
1						
2						
3						
4						
5						
6						

Thermal Expansion Coefficient: 1/C

	1	2	3	4	5	6
1						

Thermal Conductivity: W/mm-C

Table Property

Temperature - Thermal Expansion Coefficient

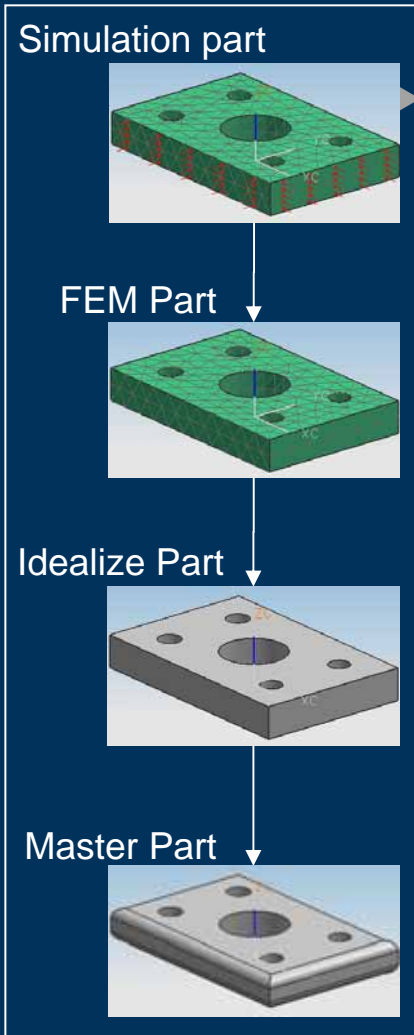
93.33	1.512e-005
107.22	1.5192e-005
121.11	1.5264e-005
135	1.5336e-005
148.89	1.539e-005
162.78	1.5462e-005
176.67	1.5534e-005
190.56	1.5588e-005
204.44	1.566e-005
218.33	1.5732e-005
232.22	1.5786e-005

OK Back Cancel

**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

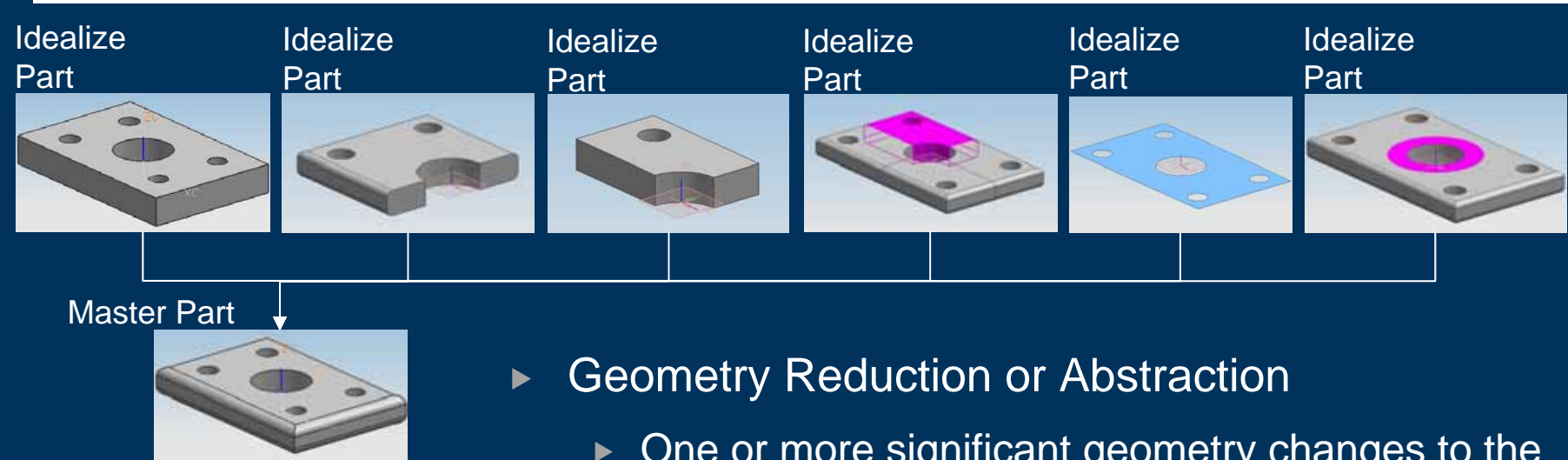
- ▶ Support Concurrent Engineering
- ▶ Associativity to Master Model

# 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

# Uses of the Idealize part

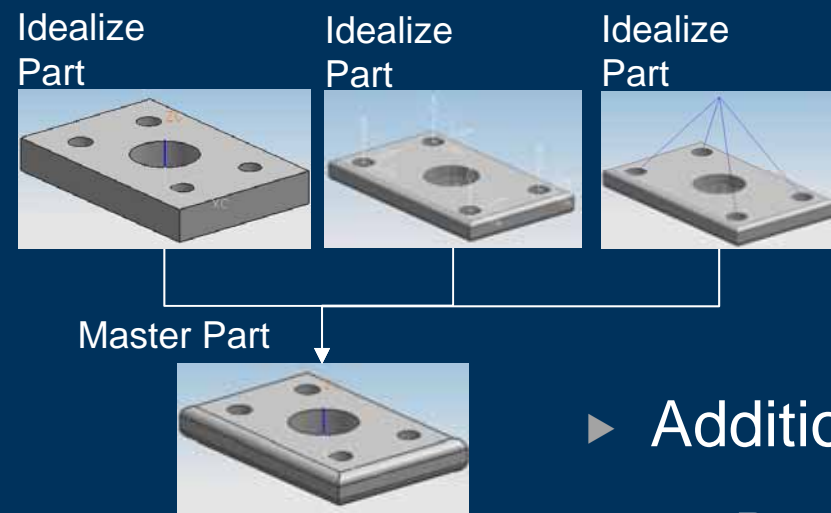


## ► Geometry Reduction or Abstraction

- 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



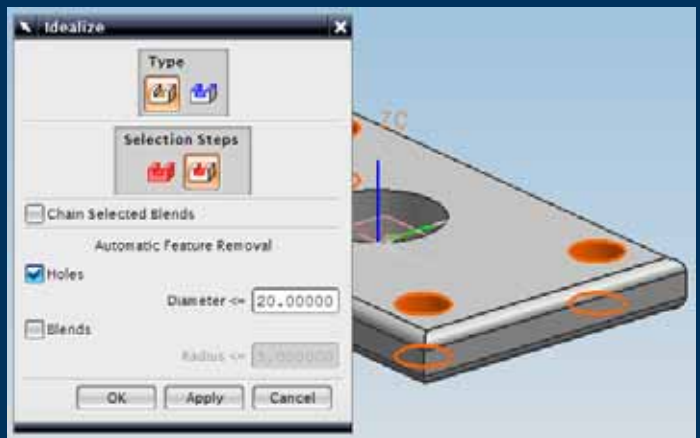
# Uses of the Idealize part



## ▶ Additional Geometry

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

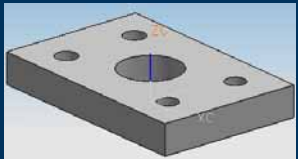
# Idealize Part – Idealize



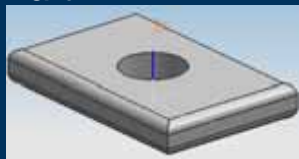
- ▶ Removing Holes and Blends
  - ▶ Based on their size
  - ▶ Manual selection



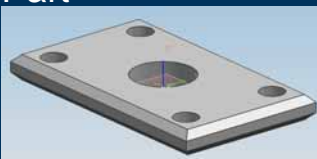
Idealize Part



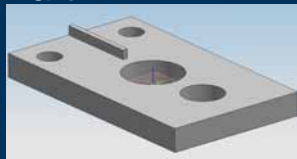
Idealize Part



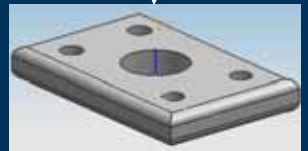
Idealize Part



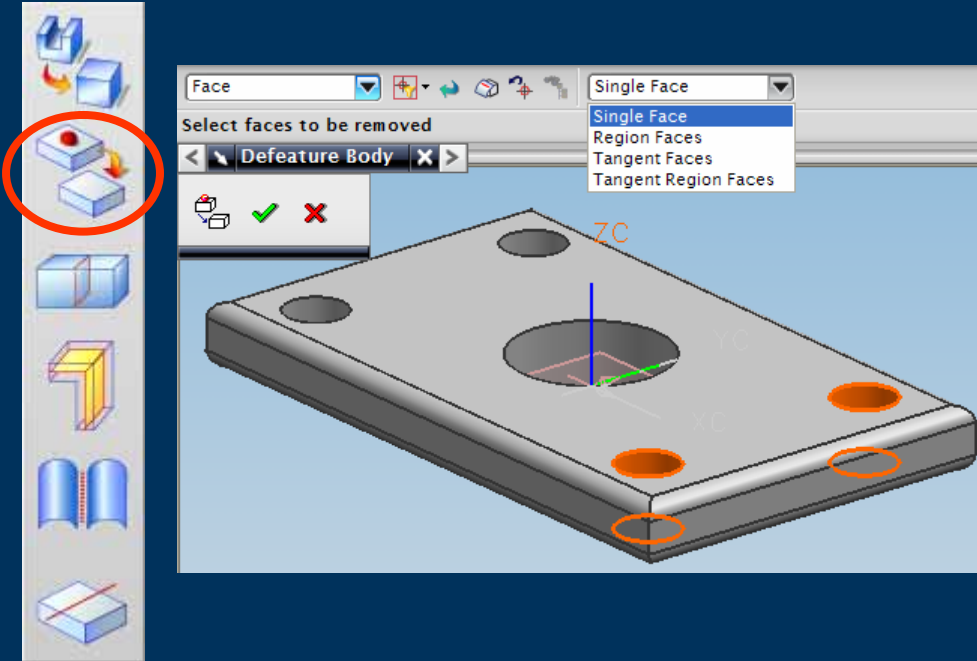
Idealize Part



Master Part

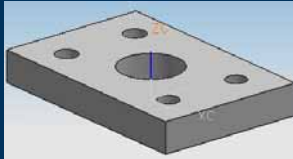


# Idealize Part – Defeature Geometry

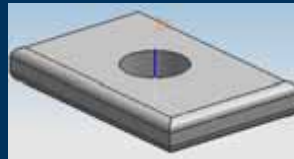


- ▶ Removing Geometry by Selection
- ▶ Method saved for Update replays

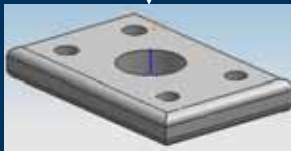
Idealize Part



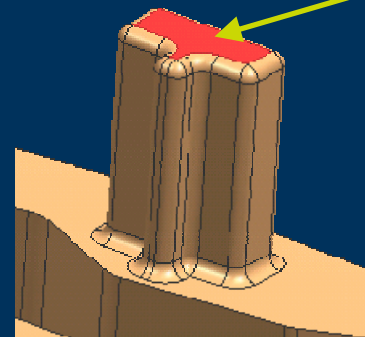
Idealize Part



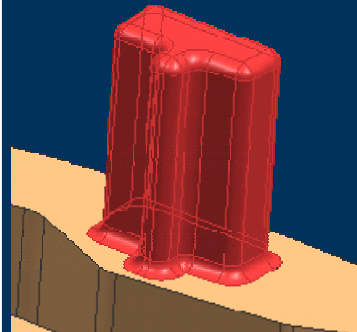
Master Part



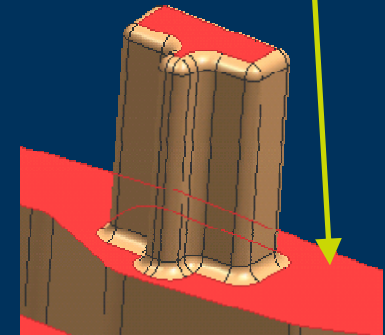
Pick outermost face as seed



Selected faces



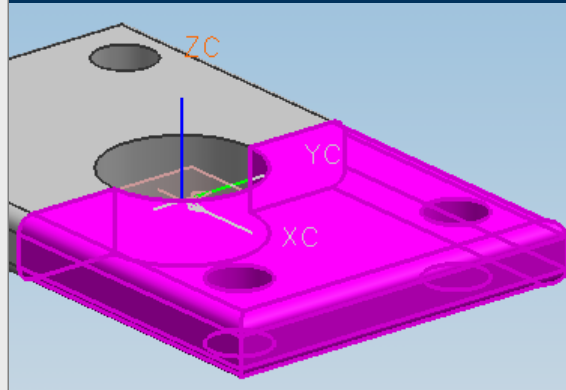
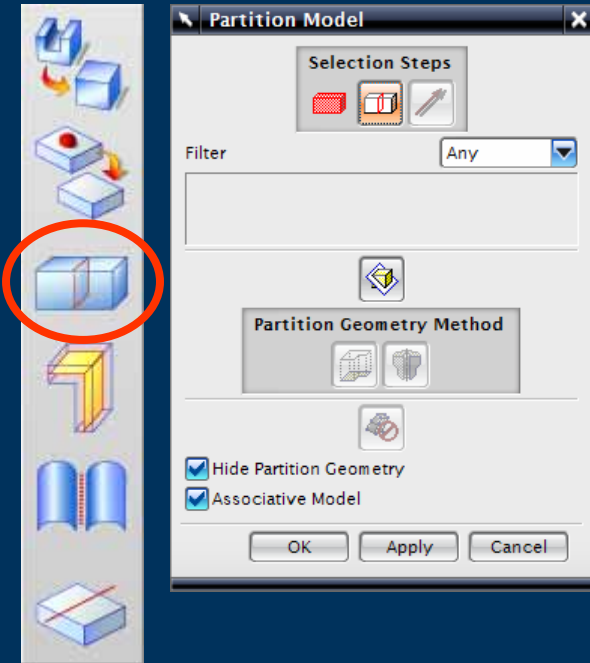
Pick Region boundary faces



Resulting "De-feature"

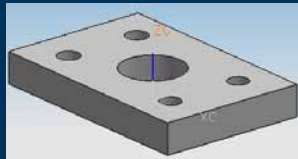


# Idealize Part – Partition

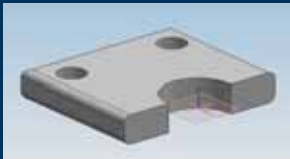


- ▶ Cutting part(s) into multiple volumes that share common faces
- ▶ Cutting part(s) into multiple surface patches that share common edges
- ▶ Load/Restraint Application
- ▶ Local mesh control
- ▶ Mesh Mating condition – common mesh across boundary
- ▶ Associative or Non-Associative to model

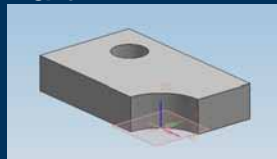
Idealize Part



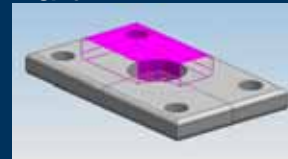
Idealize Part



Idealize Part



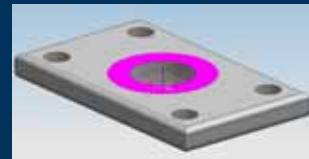
Idealize Part



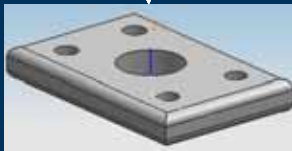
Idealize Part



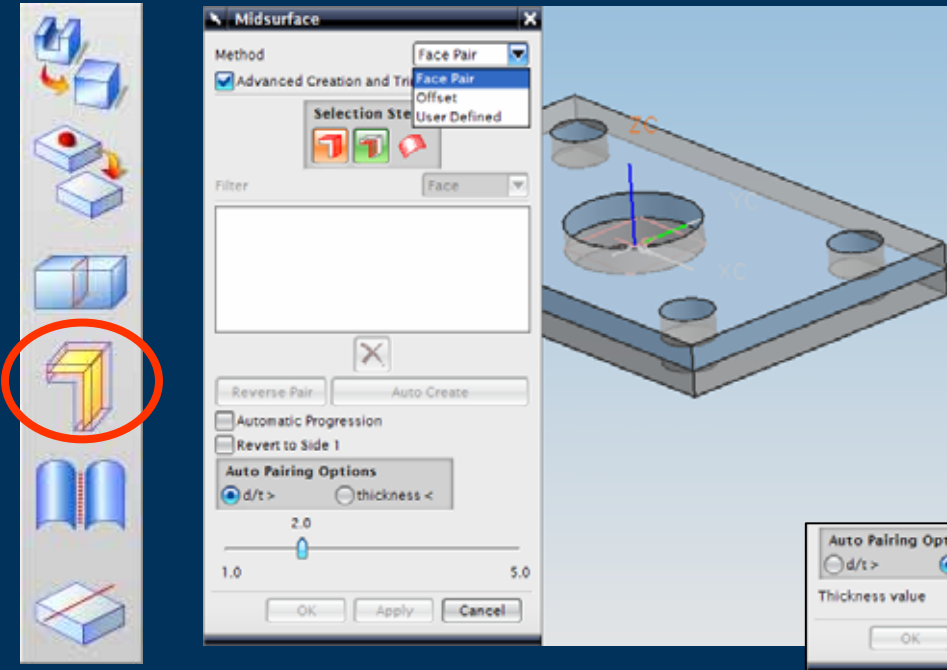
Idealize Part



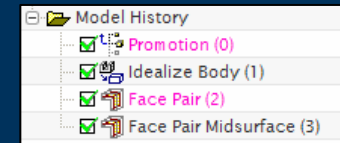
Master Part



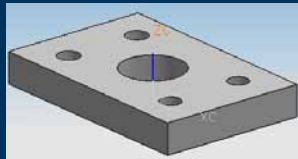
# Idealize Part – Midsurface



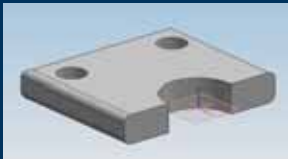
- ▶ Automatic Midsurface generation
  - ▶ Face Pair technique
  - ▶ Offset technique
  - ▶ Using another Sheet Body to define the Midsurface
  - ▶ Proportional Thickness or Specific value



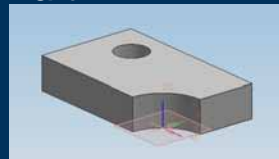
Idealize Part



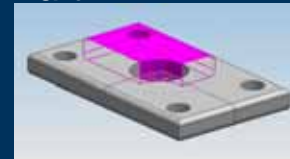
Idealize Part



Idealize Part



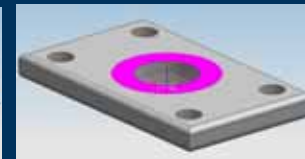
Idealize Part



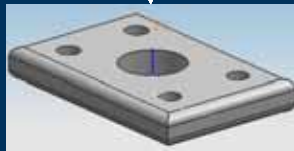
Idealize Part



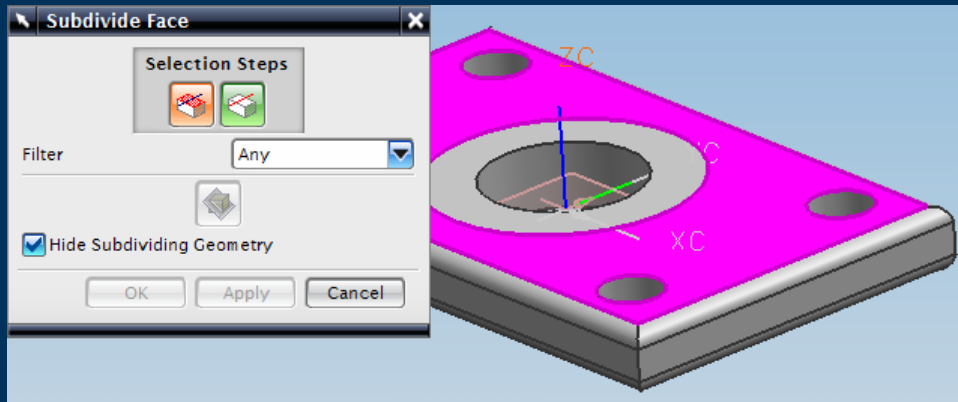
Idealize Part



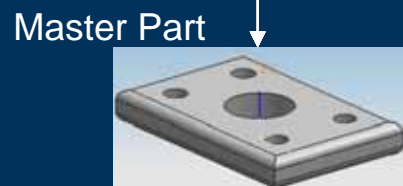
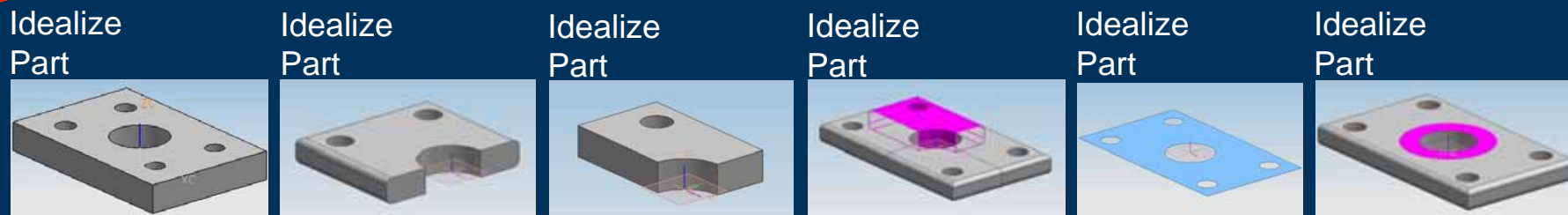
Master Part



# Idealize Part – Subdivide Faces

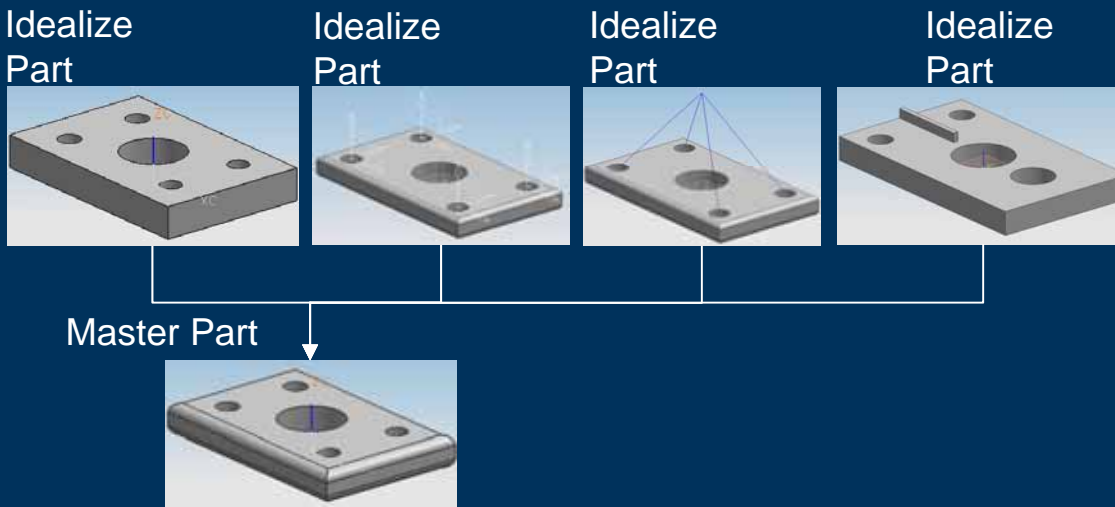
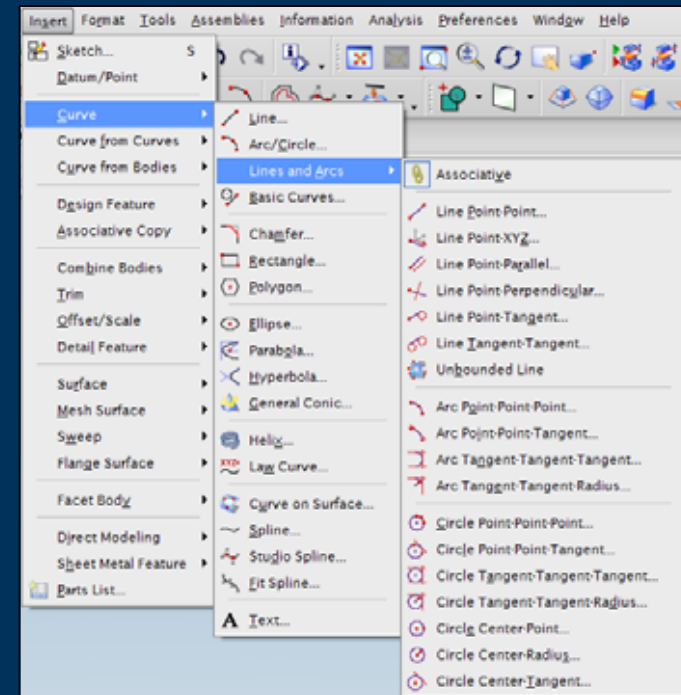
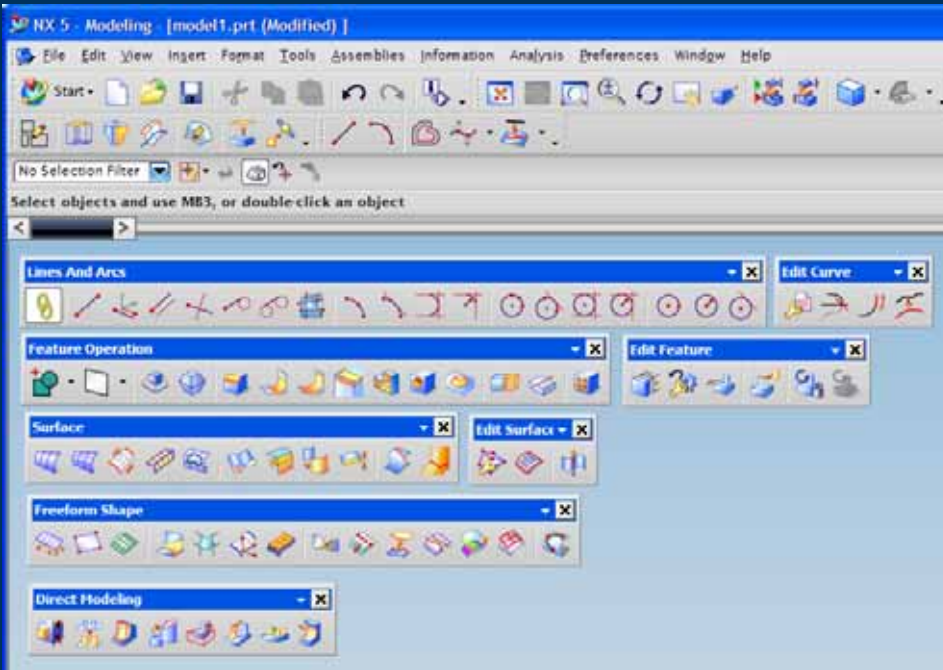


- ▶ Subdividing Faces
  - ▶ Intersection of Datum Planes
  - ▶ Intersection of Faces
  - ▶ Projected Curves and Edges
  - ▶ Projected Line between 2 points
- ▶ Load/Restraint Application
- ▶ Local mesh control



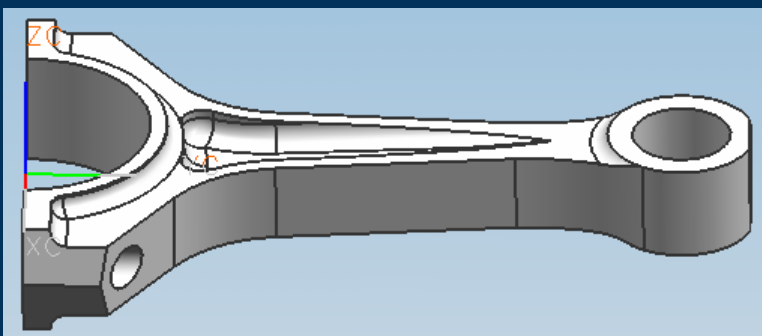
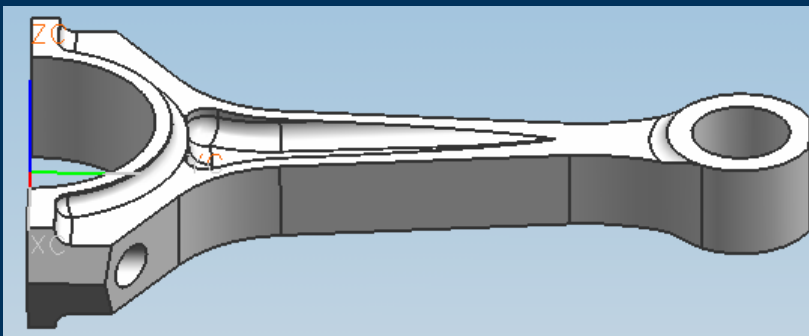
## Idealize Part – Additional Modelling

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

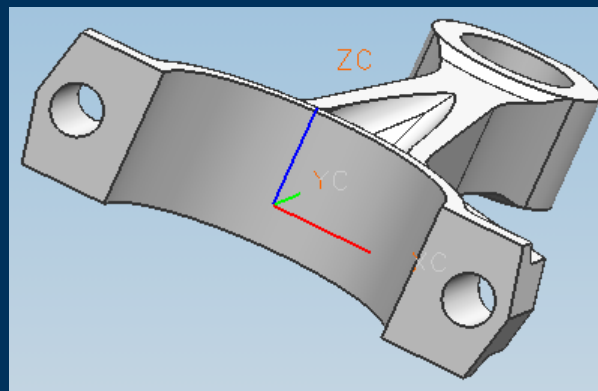
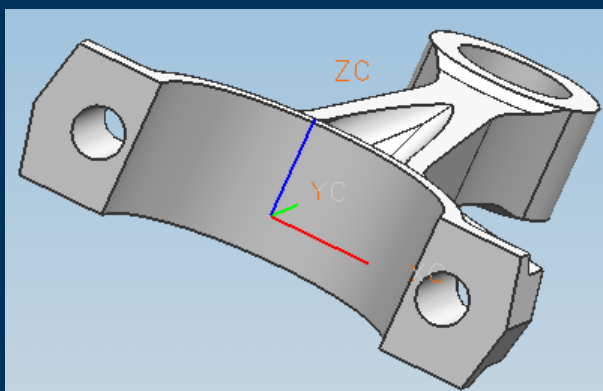


# Idealize Part – Direct Modelling

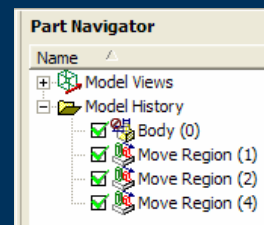
- ▶ Direct Modelling (DMX)
- ▶ Editing parts with no CAD features – Imported geometry
- ▶ Surrounding BREP updated – Tangency maintained
- ▶ Surfaces Resized – Fillets
- ▶ Surfaces Replaced



Little End Moved



Bolt Holes moved





# Idealize Part – Material Properties



**Materials**

Category: +

Material	Category
STEEL-ROLLED	METAL

Materials Inherited

Materials Inherited	Part
STEEL	C:\GuyWills\Demo_Stuff\NX5_CAE\NX5_...

Name: STEEL-ROLLED  
 Category: METAL  
 Library Reference: 14

Isotropic  Orthotropic  Anisotropic  Fluid

**Basic Structural**

Mass Density: 7.85e-00 kg/mm<sup>3</sup>  
 Reference Temperature: C  
 Young's Modulus: 2.06e+00 mN/mm<sup>2</sup>(kPa)  
 Poisson's Ratio: 0.3  
 Shear Modulus: N/mm<sup>2</sup>(MPa)  
 Stress/Strain: TABLE  
 Thermal Expansion Coefficient: 1.728e-0 1/C

Filter: Any

OK Apply Back Cancel

← Override Material

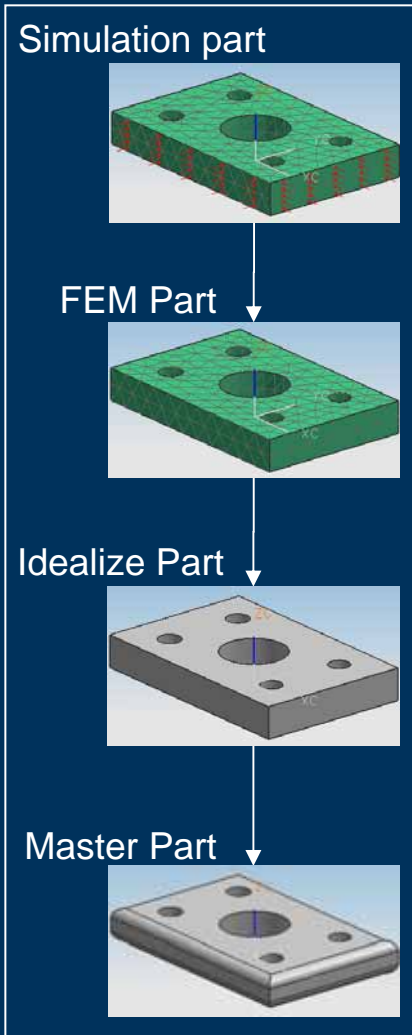
← Master Part Material

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

**SIEMENS**

**FEM Part**

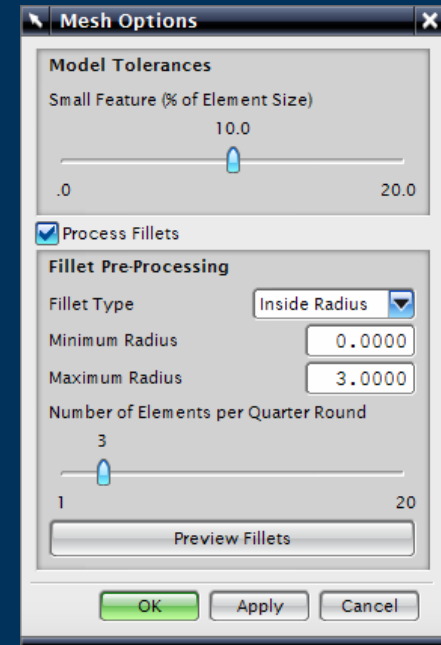
# FEM Part



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

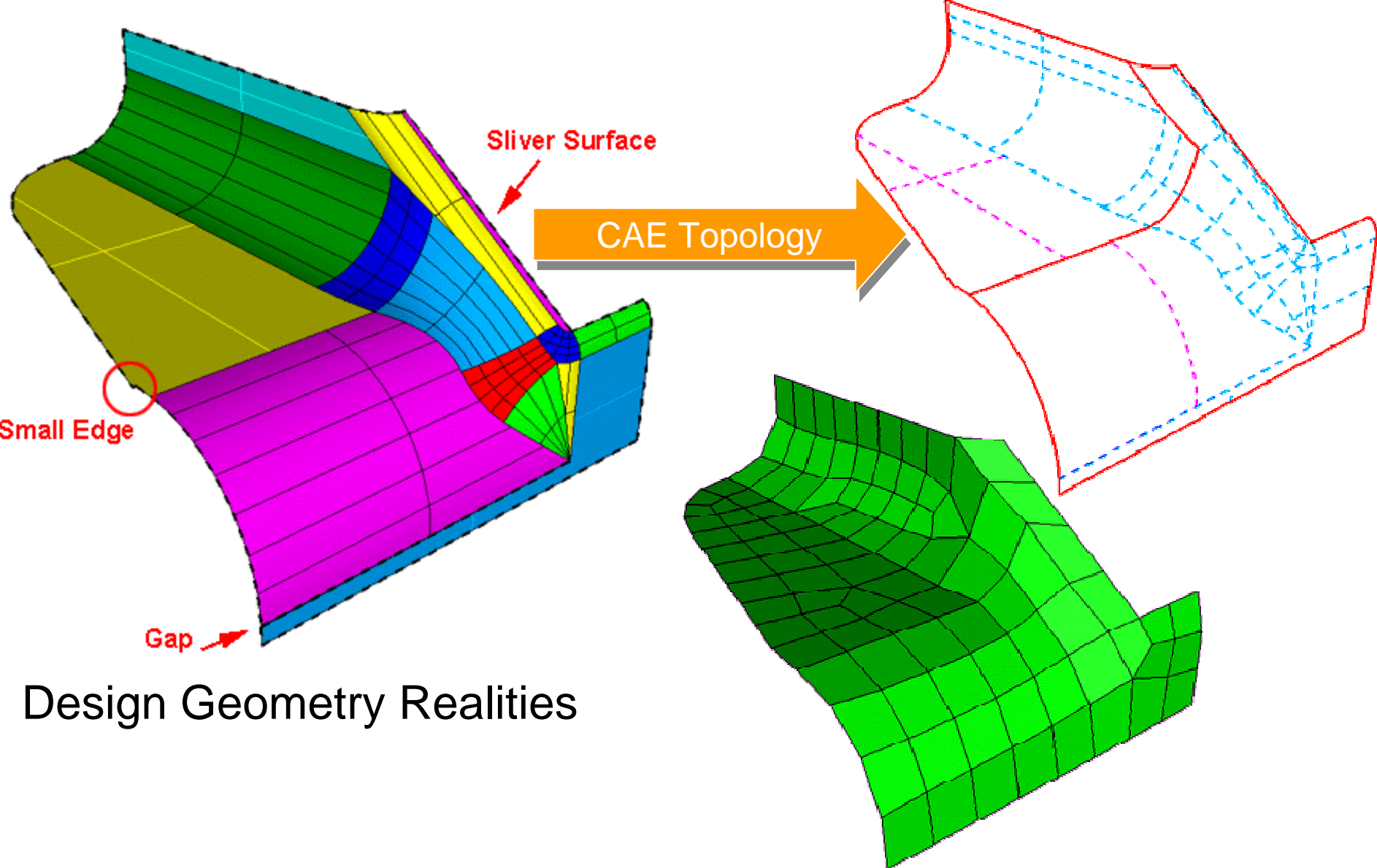
# NX Advanced Simulation : CAE Topology

- ▶ 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

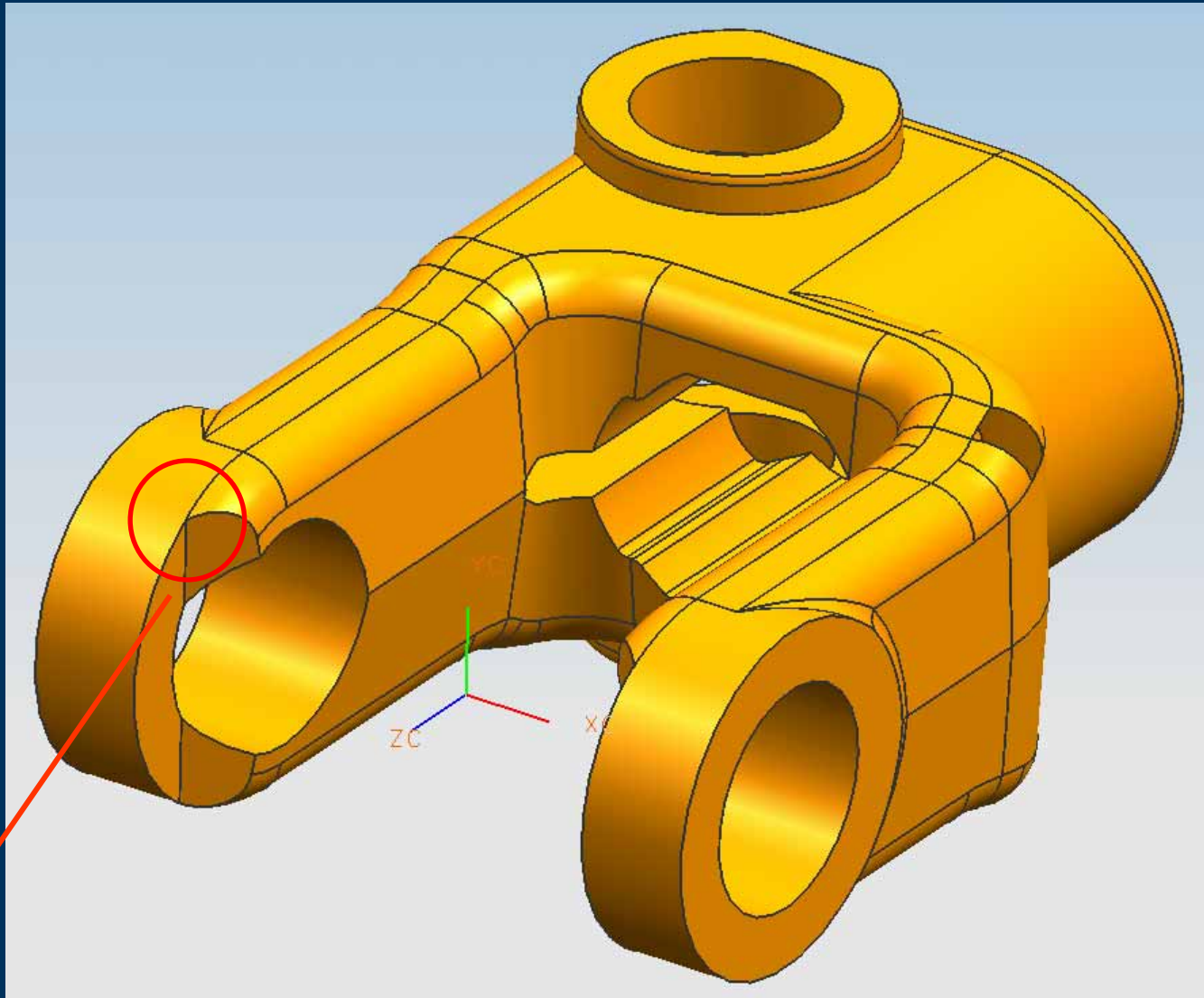


# NX CAE Topology

## – Geometric Abstraction and Meshing

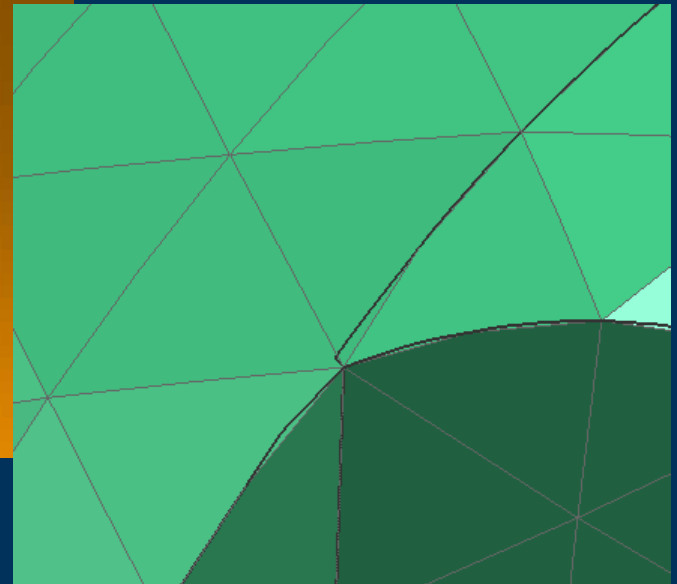
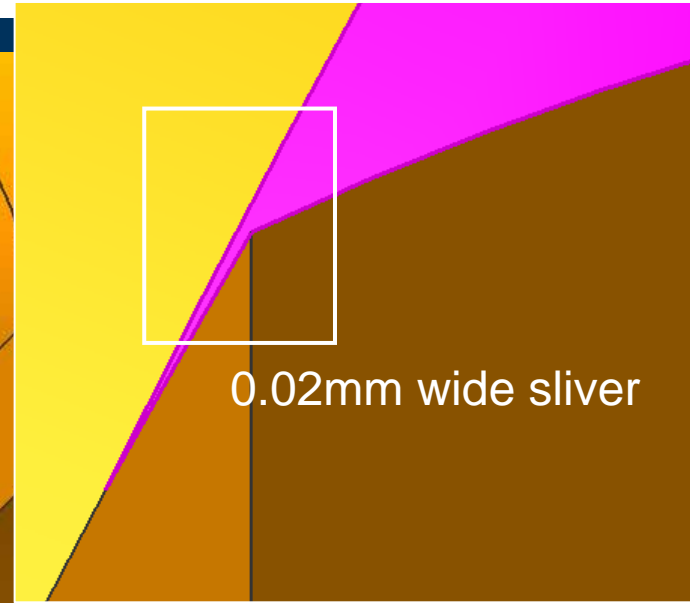
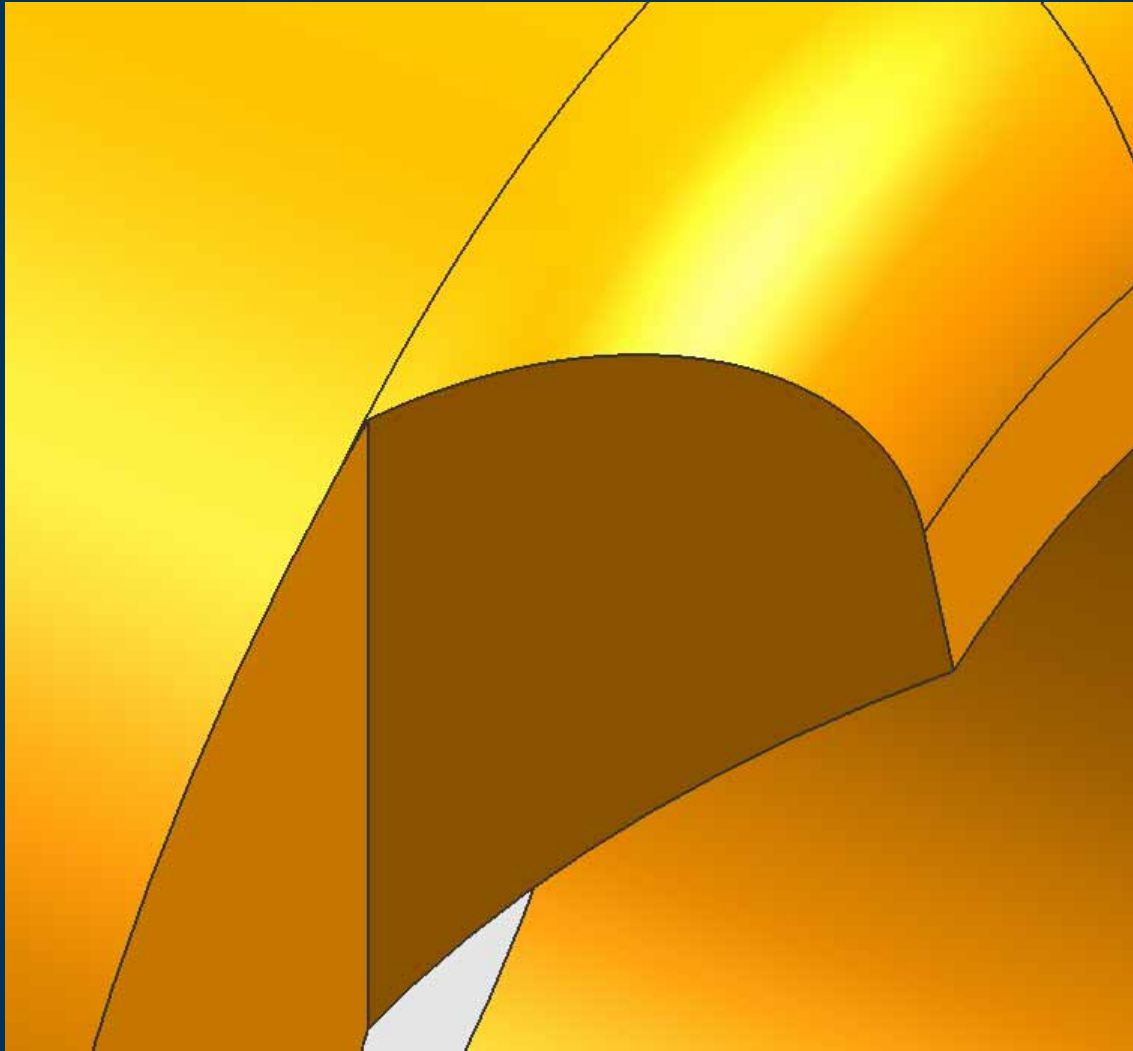


# NX CAE Topology

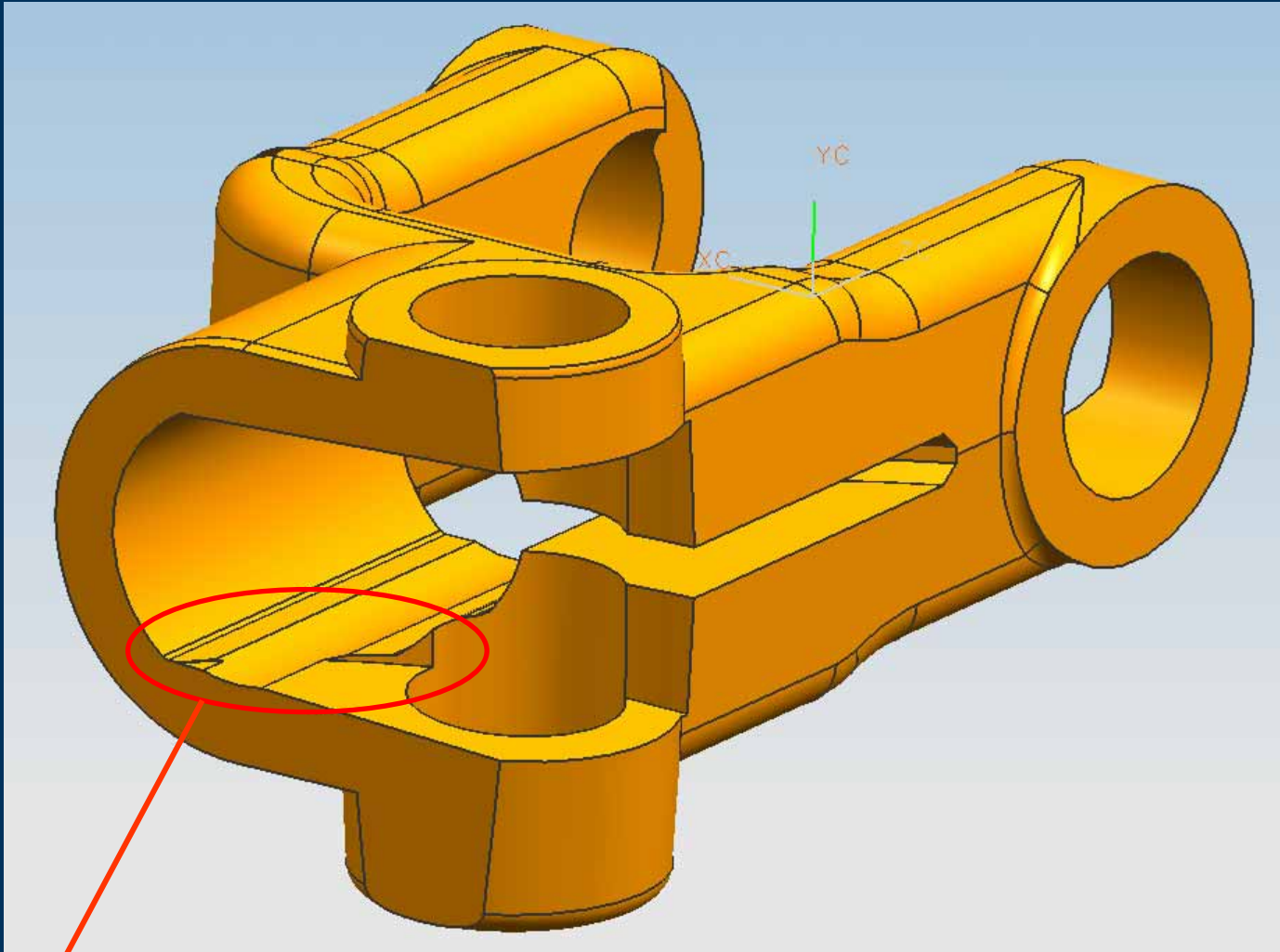


Issues

# NX CAE Topology



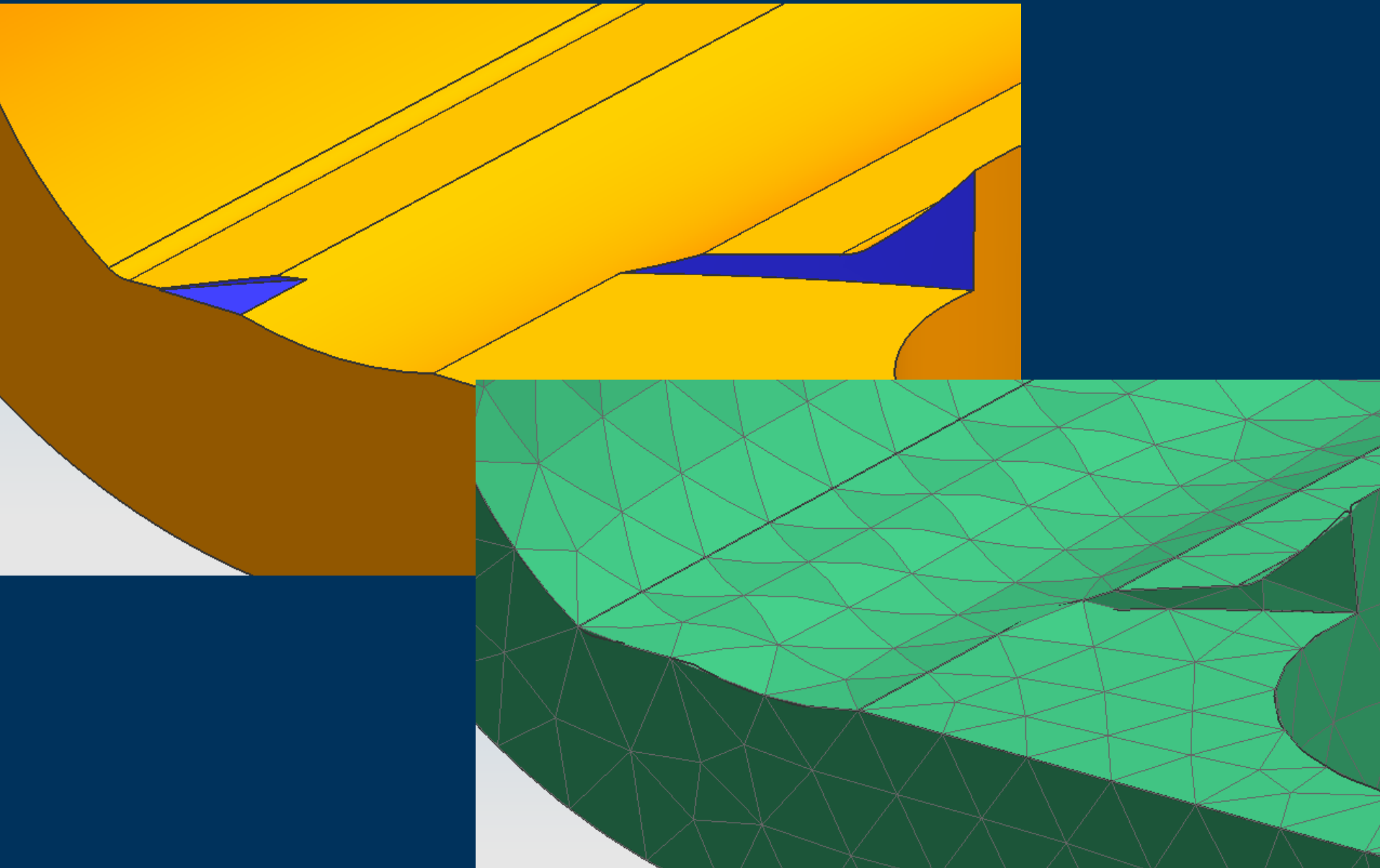
# NX CAE Topology



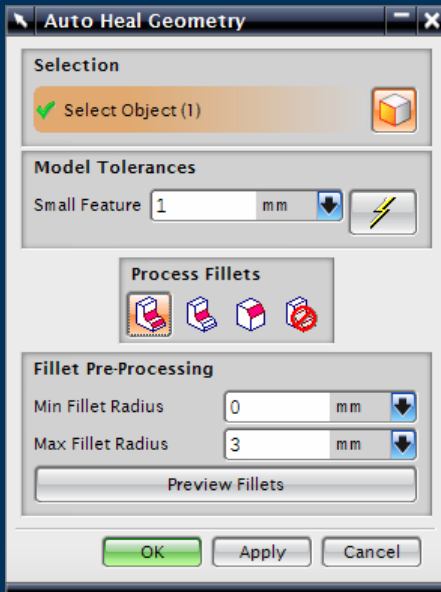
Issues



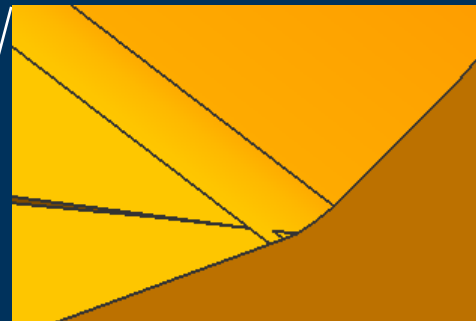
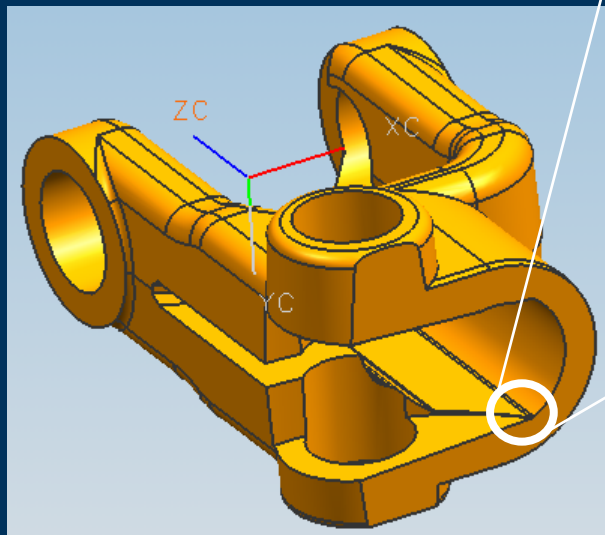
# NX CAE Topology



# 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



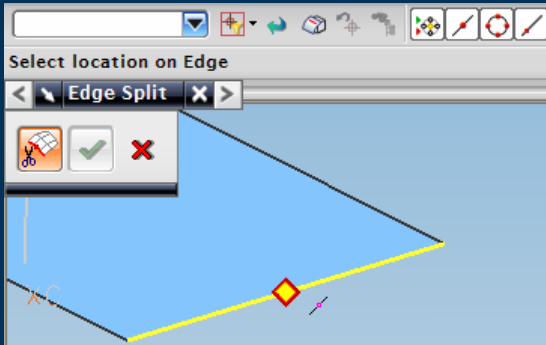
Before Heal



After Heal

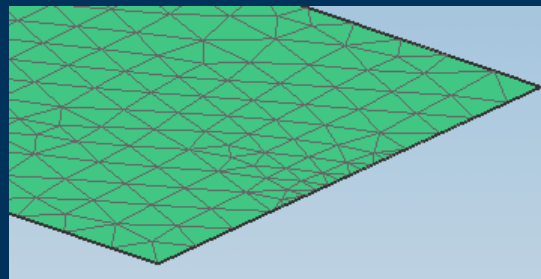
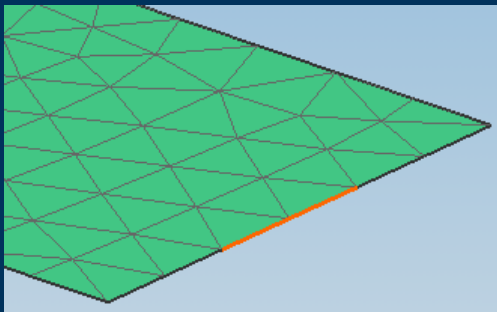


# NX CAE Topology – Split Edge

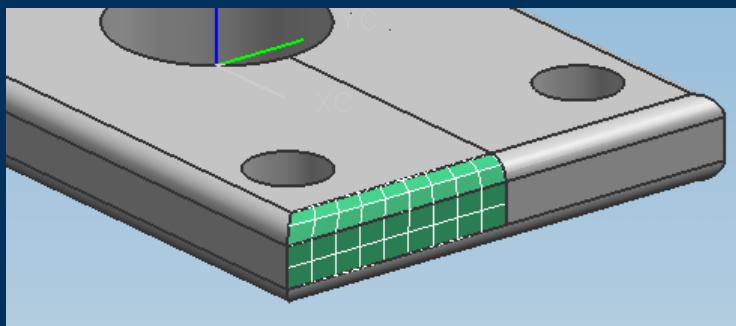
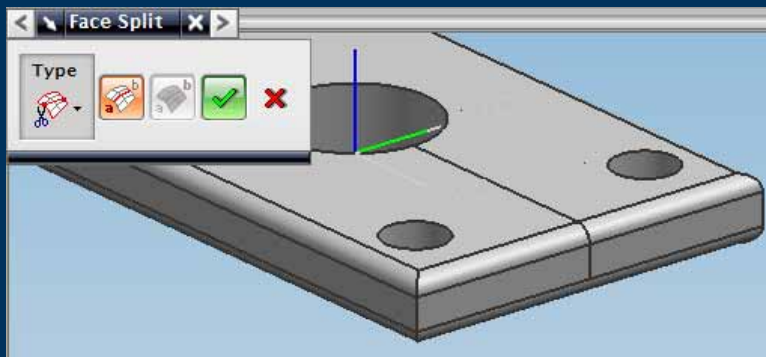
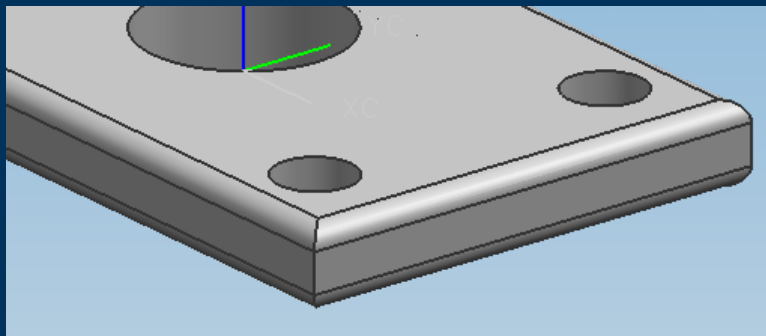


## ► Split an Edge

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



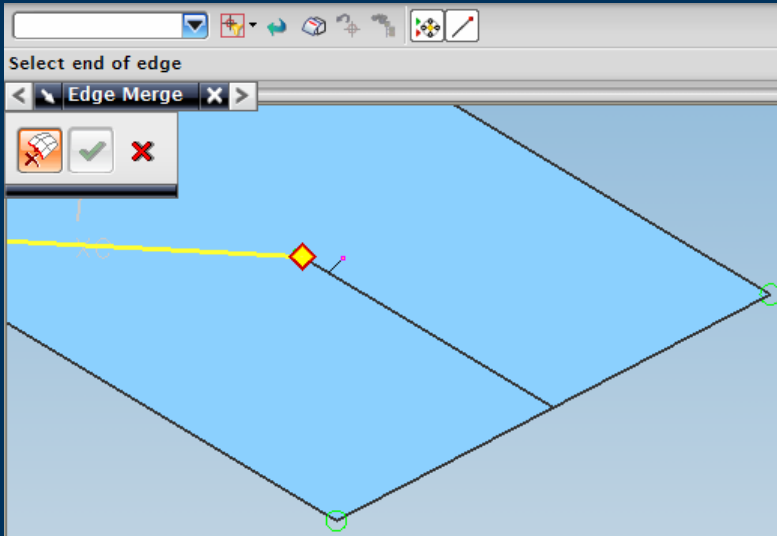
# NX CAE Topology – Split Face



## ▶ Split Face

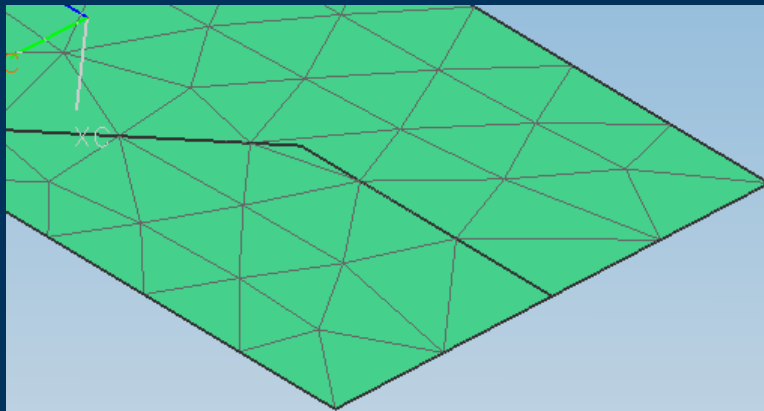
- ▶ Split a polygon face along a projected line
- ▶ Mesh control
- ▶ Boundary Condition control

# NX CAE Topology – Merge Edge

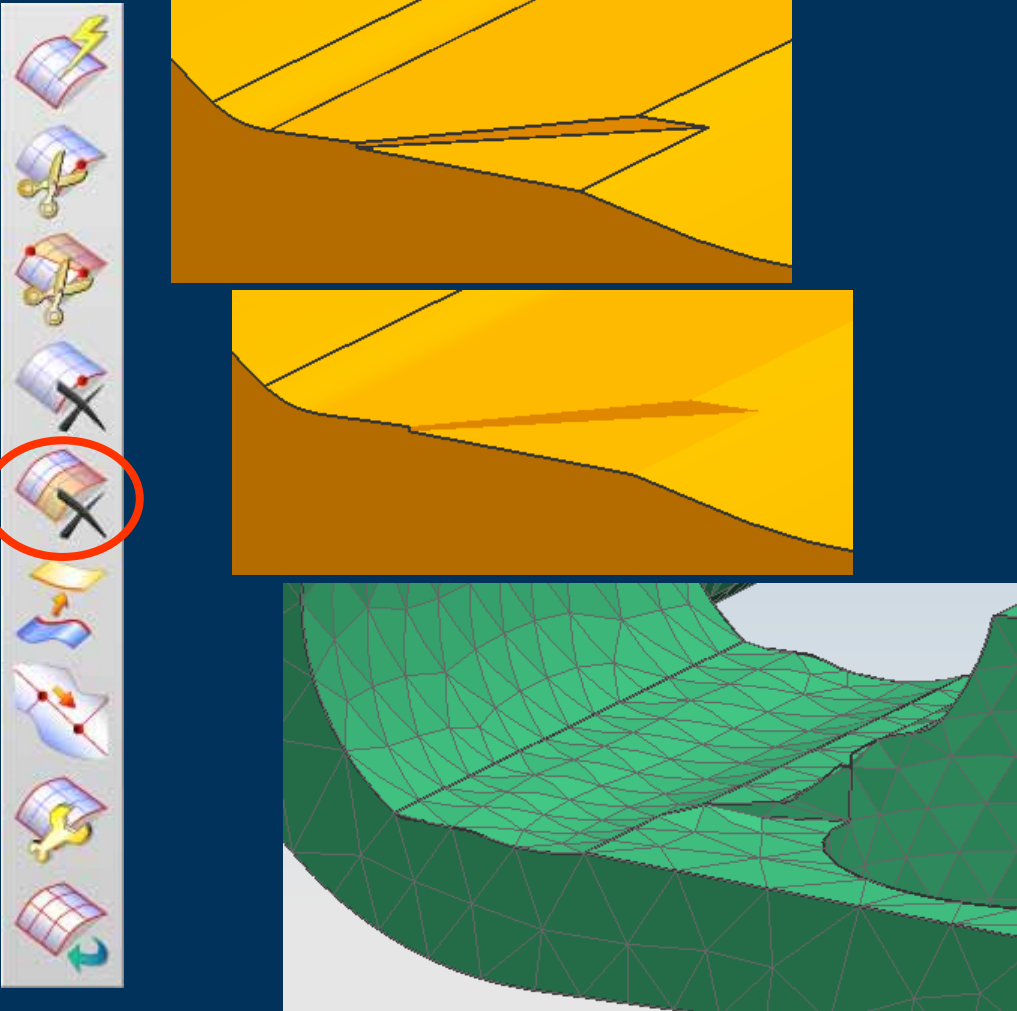


## ▶ Merge Edge

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

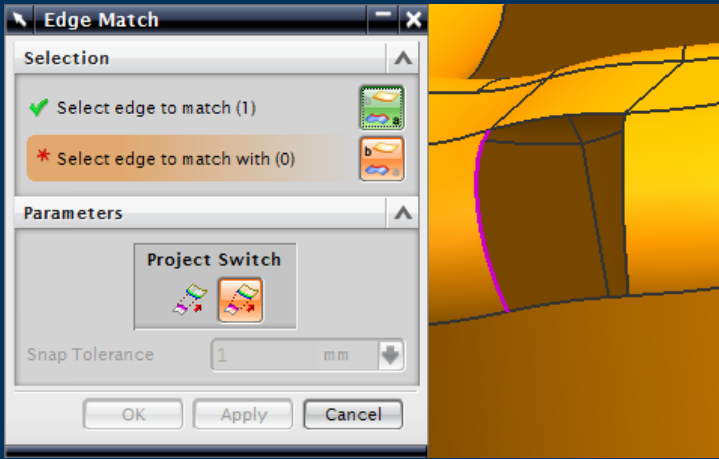


# NX CAE Topology – Merge Face

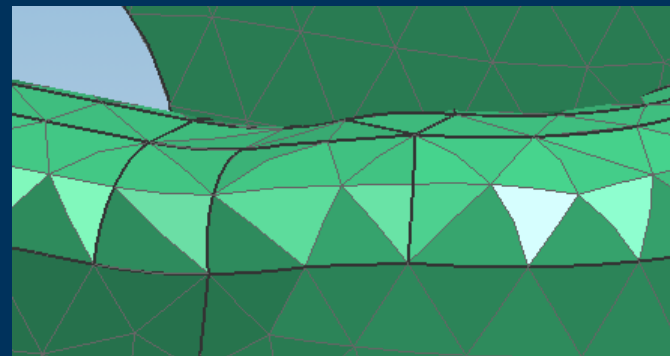
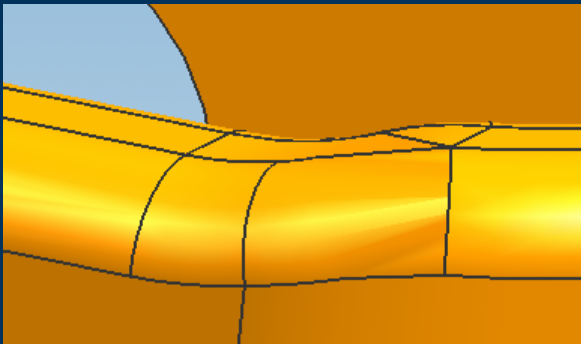


- ▶ Merge Face
  - ▶ Merge two separate polygon faces into a single polygon face along a common polygon edge
- ▶ Commonly before and after an Auto Heal
- ▶ Before or After Meshing
- ▶ Used to remove data to get a better quality mesh

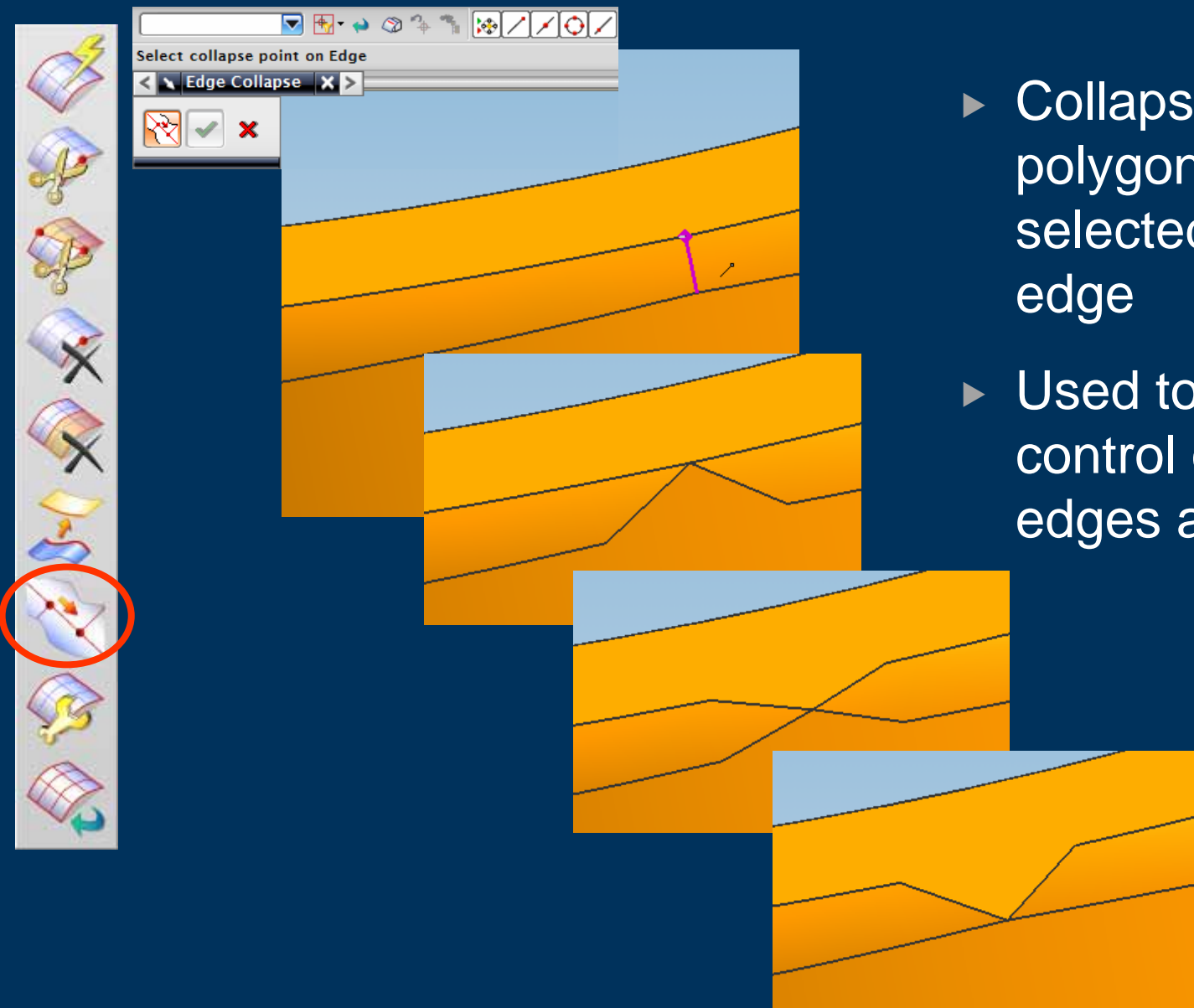
# NX CAE Topology – Match Edge



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



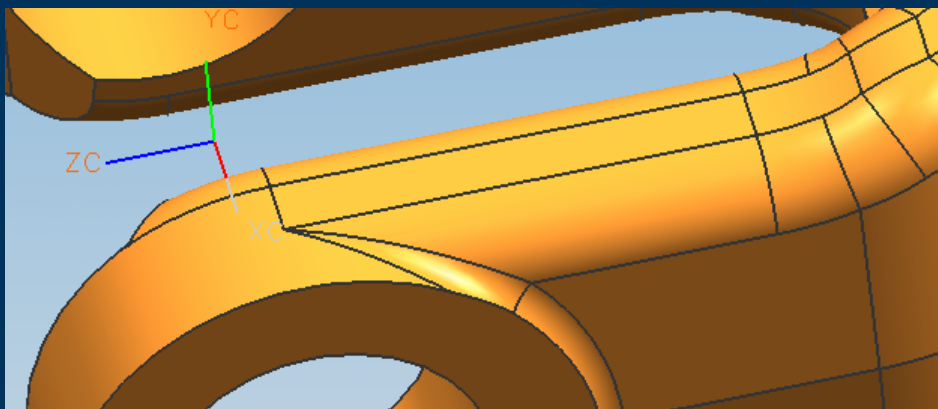
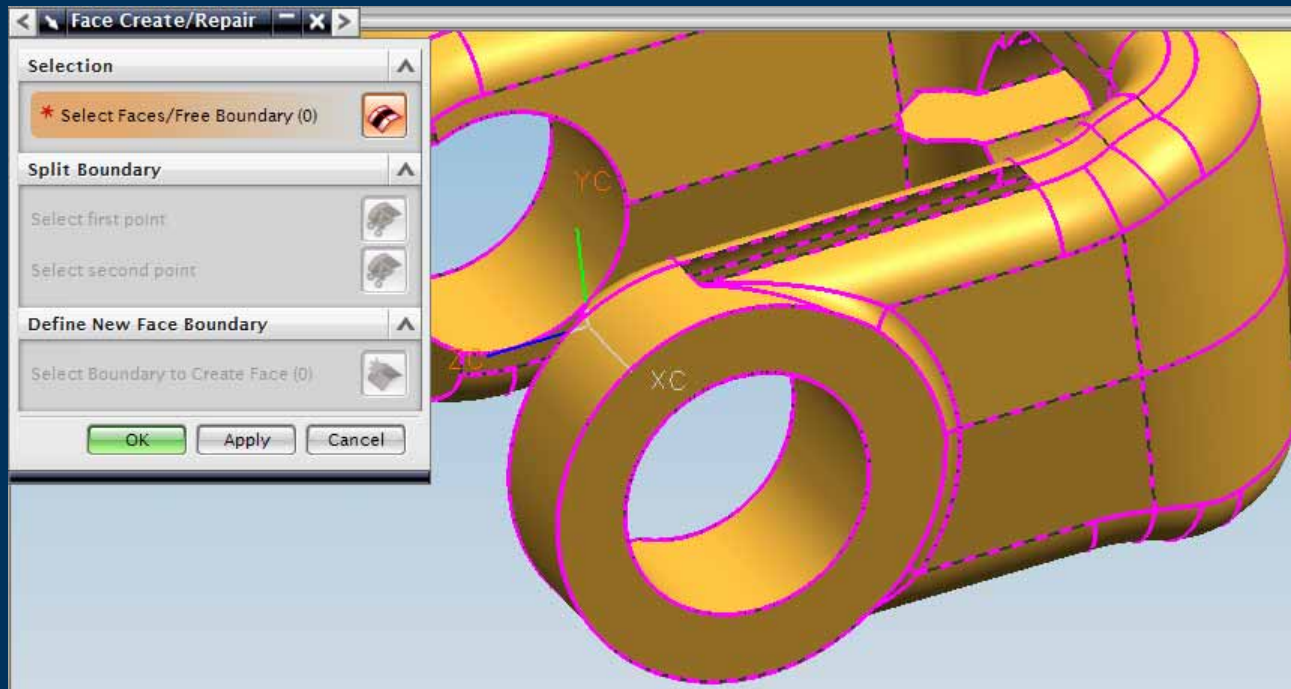
# NX CAE Topology – Collapse Edge



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

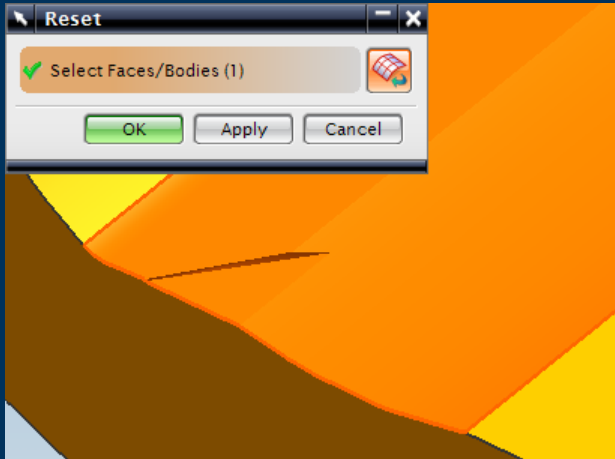


# NX CAE Topology – Face Repair

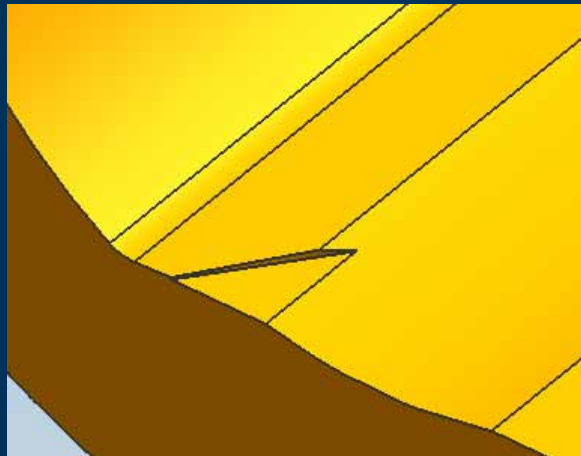


- ▶ Create a new polygon face to fill a hole
- ▶ Repair a poor quality polygon face

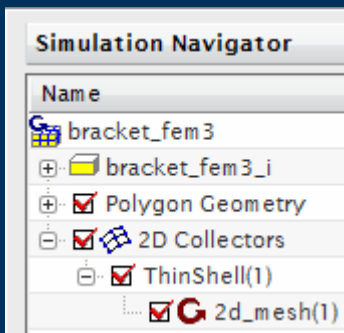
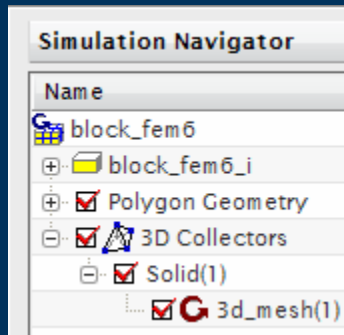
# NX CAE Topology – Reset



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



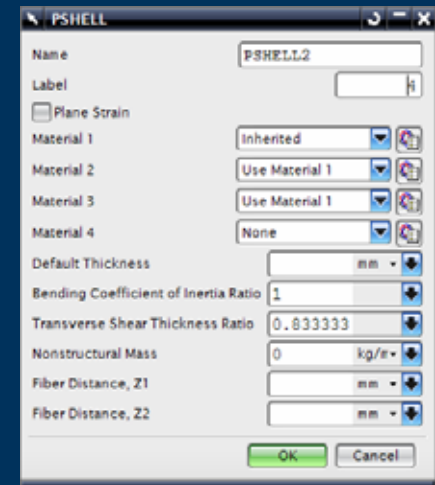
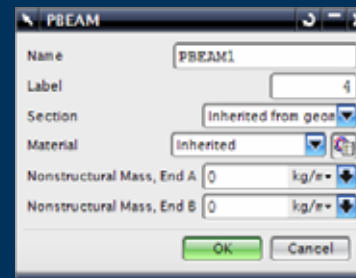
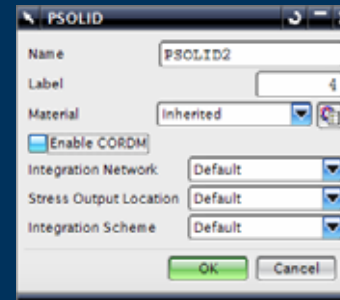
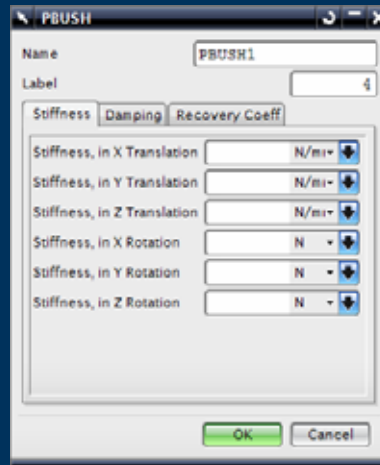
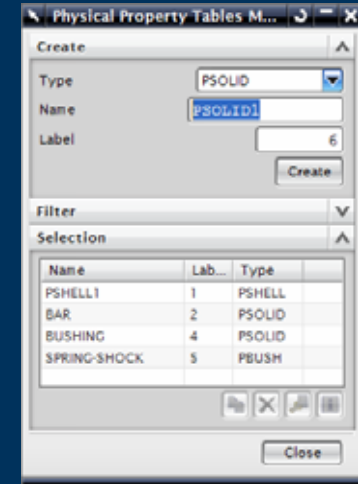
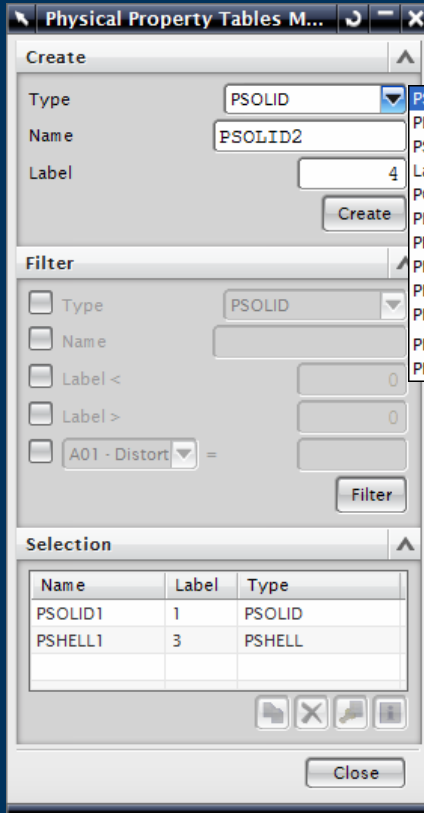
# NX CAE Topology – Mesh Updates



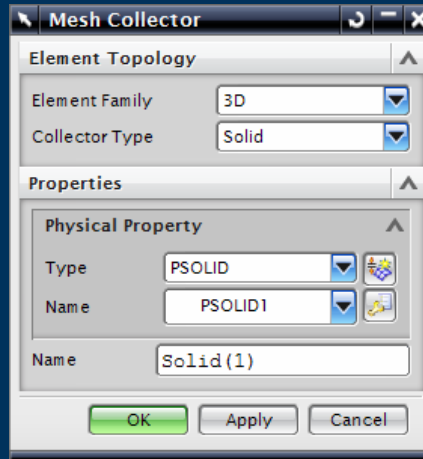
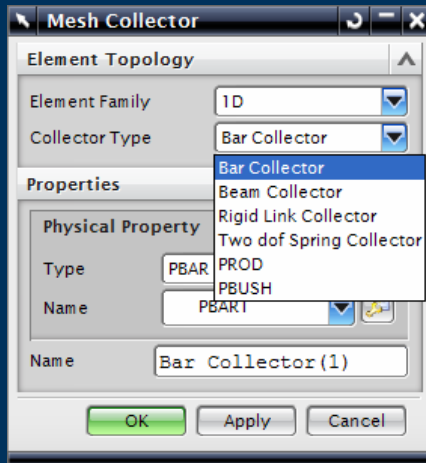
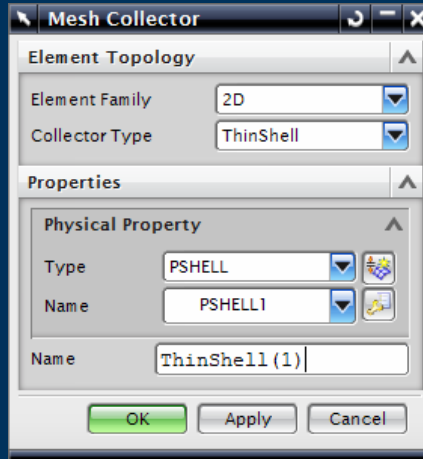
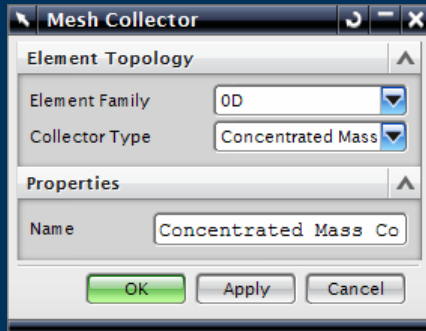
- ▶ CAE Topology changes can be done before and after Meshes are applied
- ▶ After a change (like Merge Face) the Mesh is flagged “out of date”
  - ▶ In the Simulation Navigator
  - ▶ Mesh Update icon
- ▶ Note if multiple meshes exist, only the changed ones are flagged as “out of date” and updated
-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

# Physical Properties

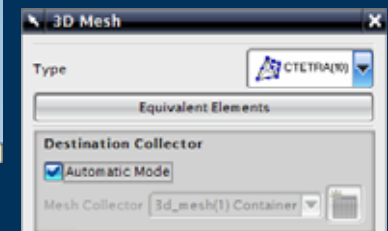
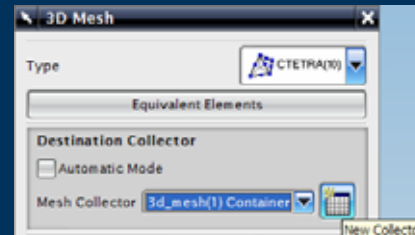
- ▶ Physical Property is Solver and Element dependant
  - ▶ Family of Elements
  - ▶ Material Reference
  - ▶ Commonly referred to as PID
  - ▶ Often used to identify different parts in an Assembly



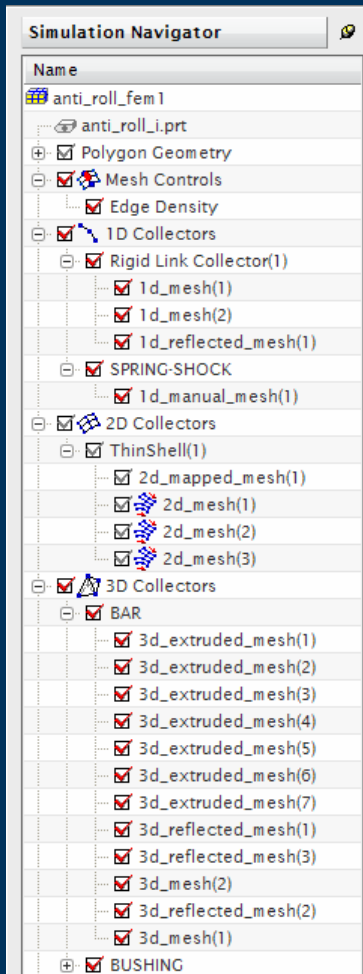
# Mesh Collectors



- ▶ 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

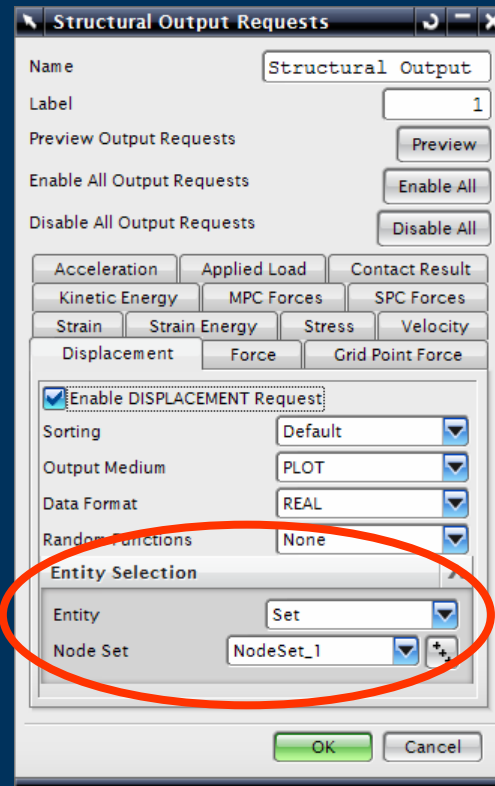
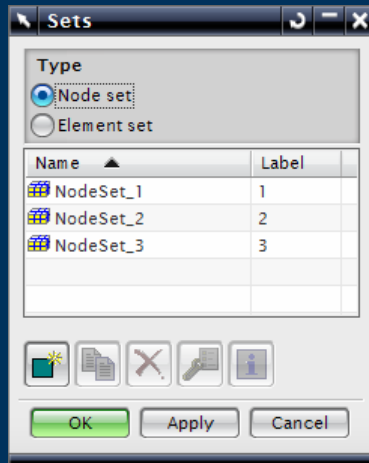
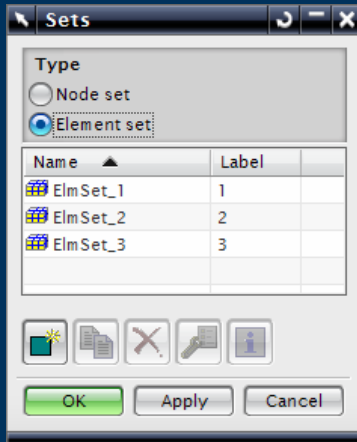
- ▶ Drag 'n' Drop item between collectors
- ▶ Mesh inherits the target Collector properties inc Physical, Material and Display
- ▶ Display control

- ▶ 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

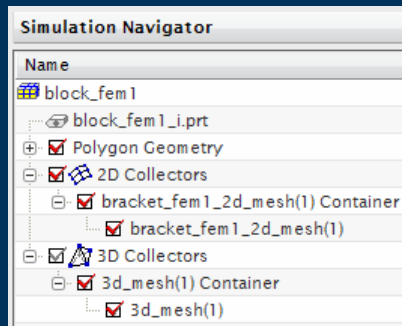
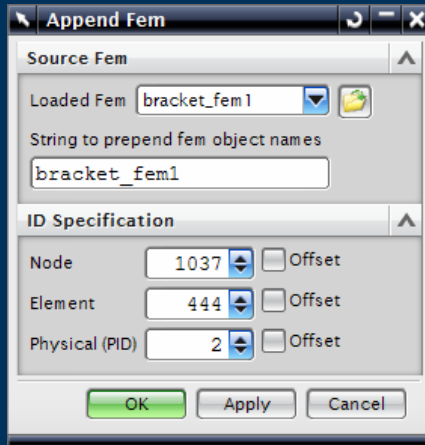
# Node and Element Sets



- ▶ 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

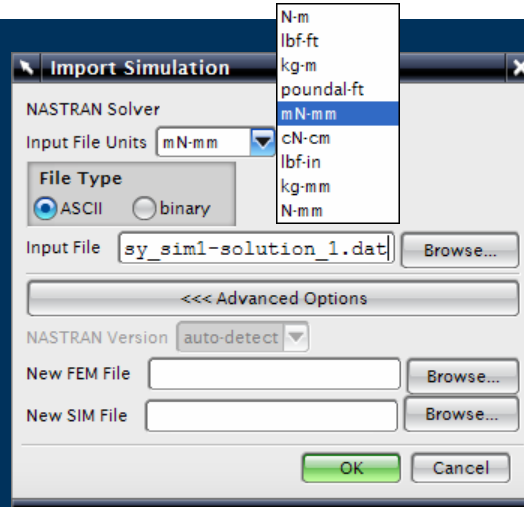
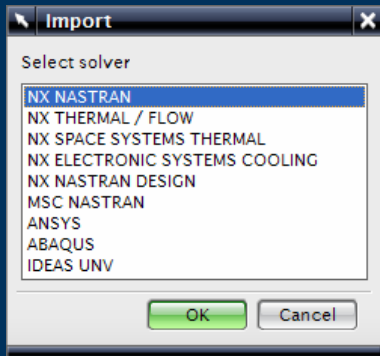
# Mesh Append

- ▶ 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

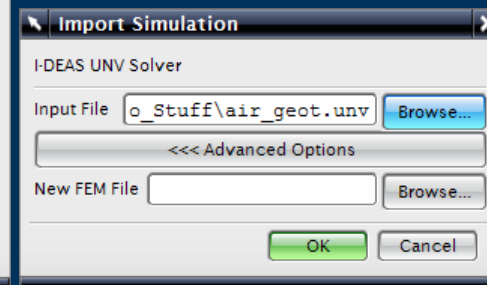
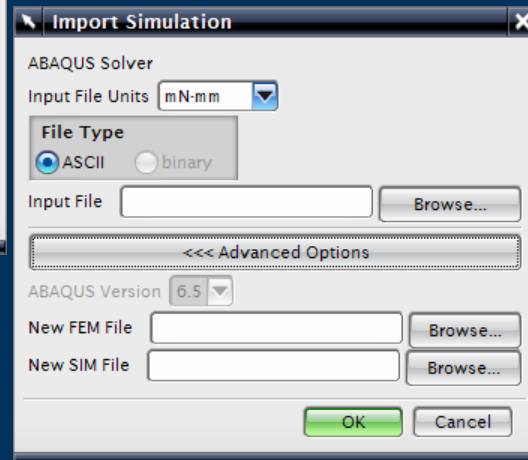
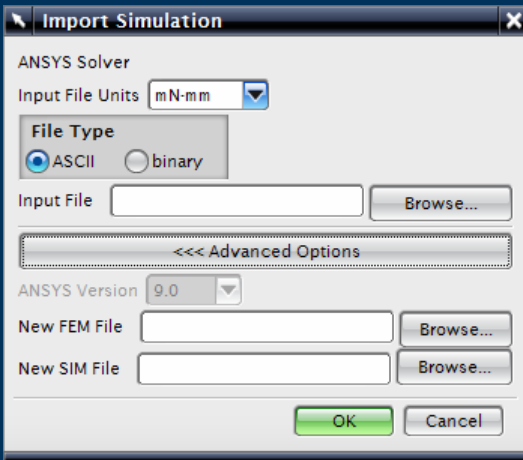




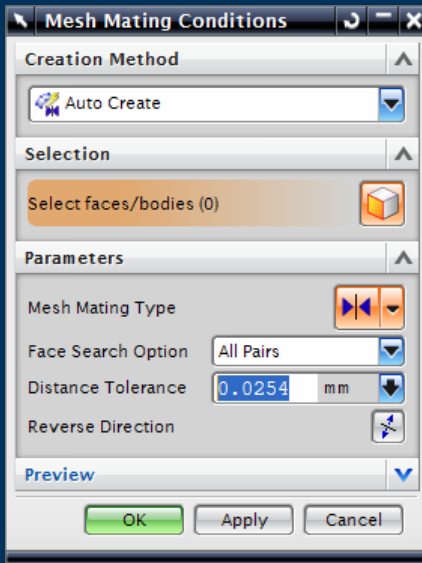
# Mesh Import



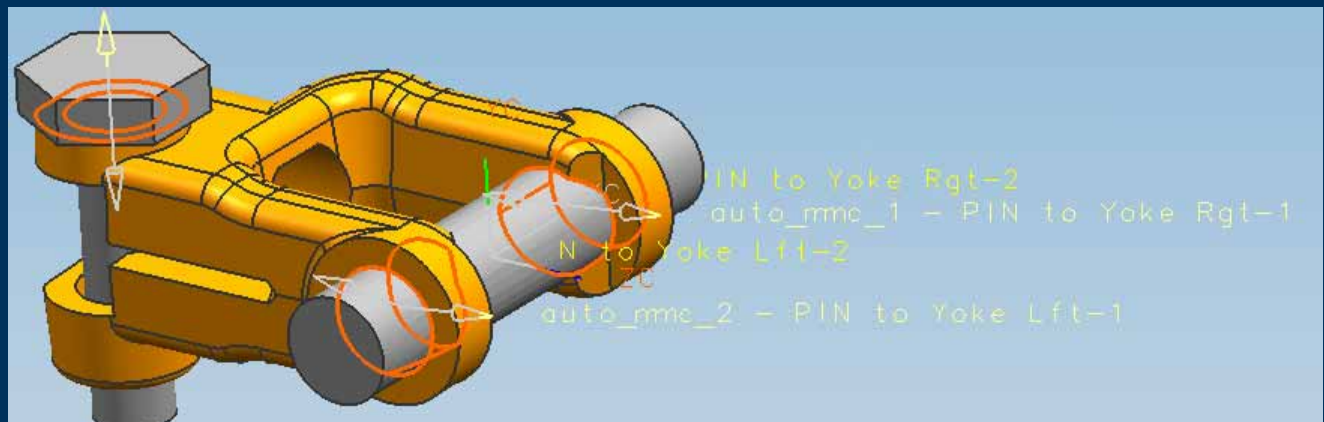
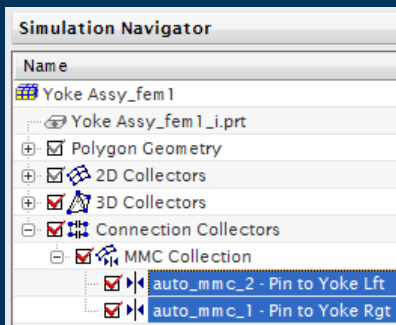
- ▶ Import of a solver deck from an External file
  - ▶ Units selection for the incoming data
  - ▶ Append to existing files or Create New files



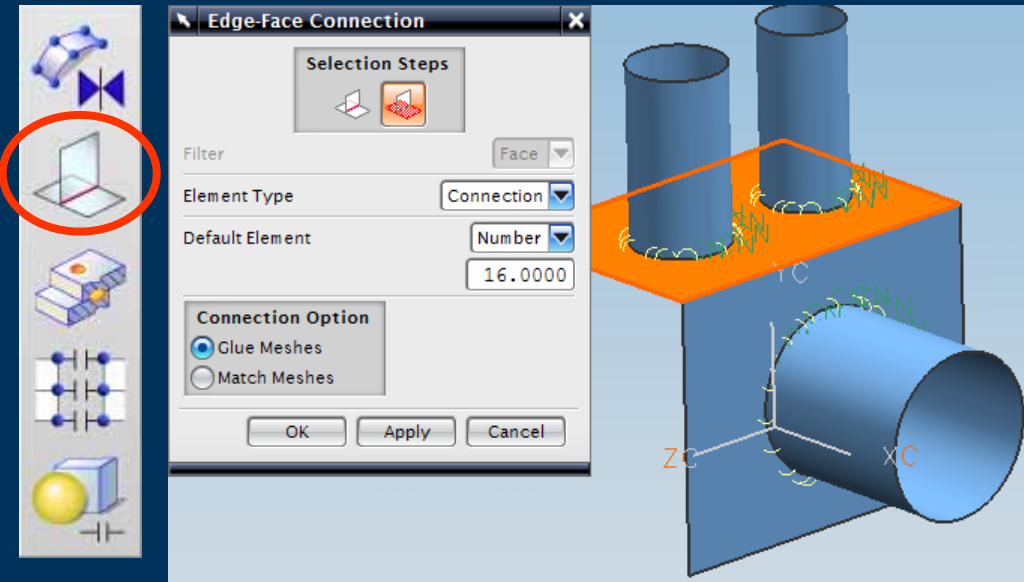
# Mesh Connections – Mesh Mating



- ▶ 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



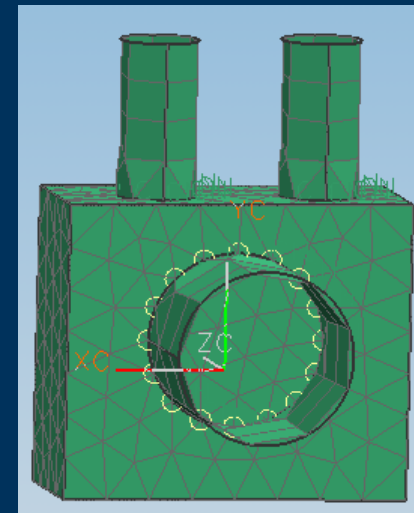
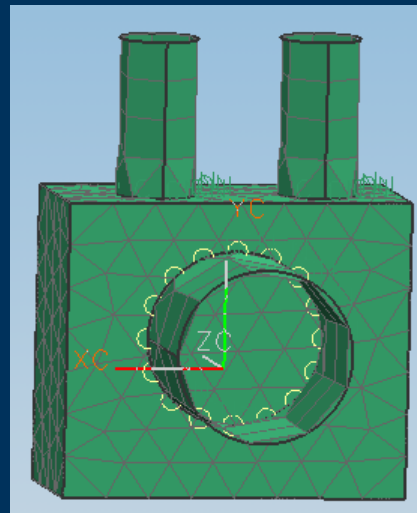
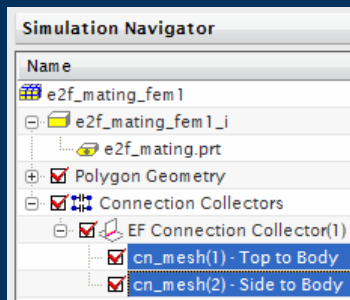
# Mesh Connections – Edge-Face Connection



- ▶ 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

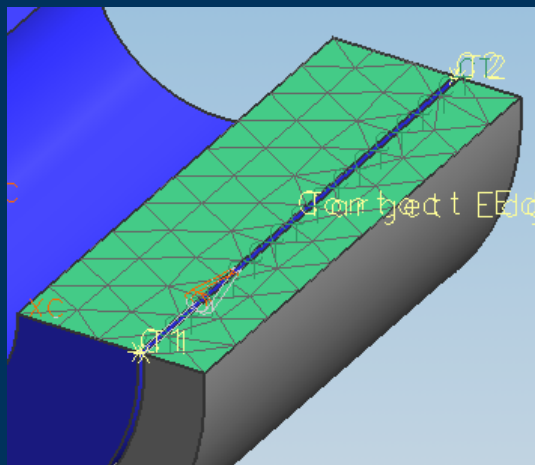
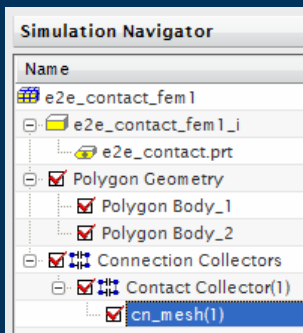
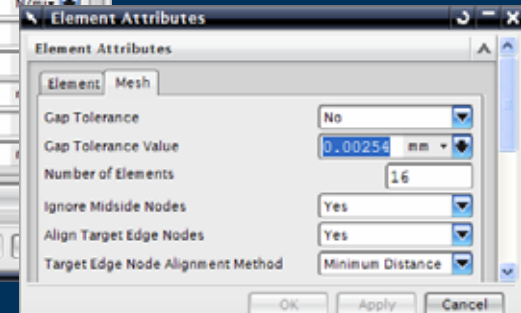
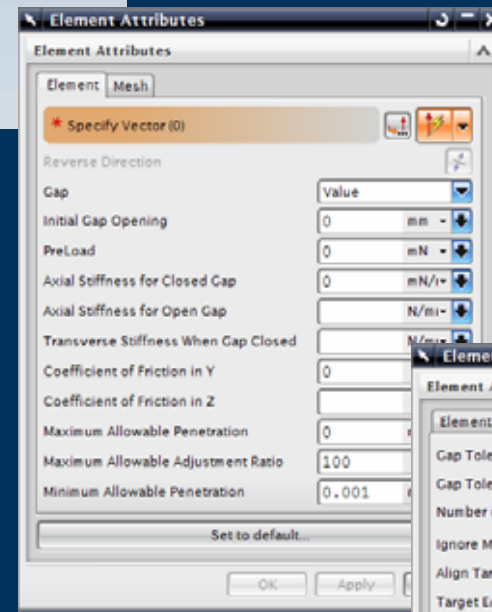
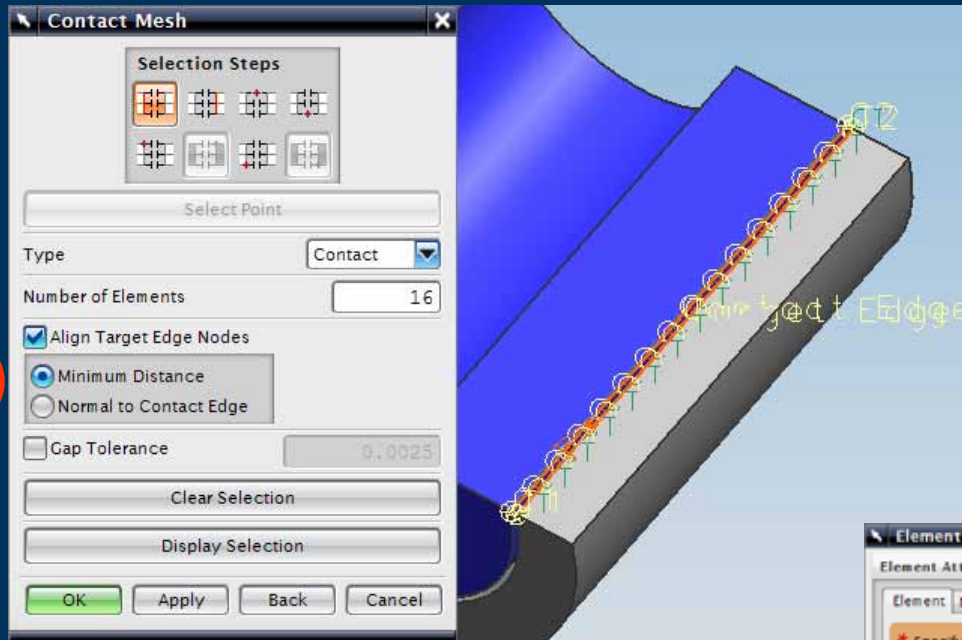
Glue Meshes

Match Meshes



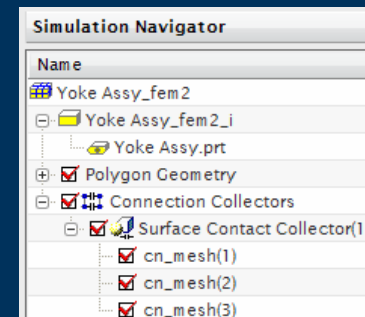
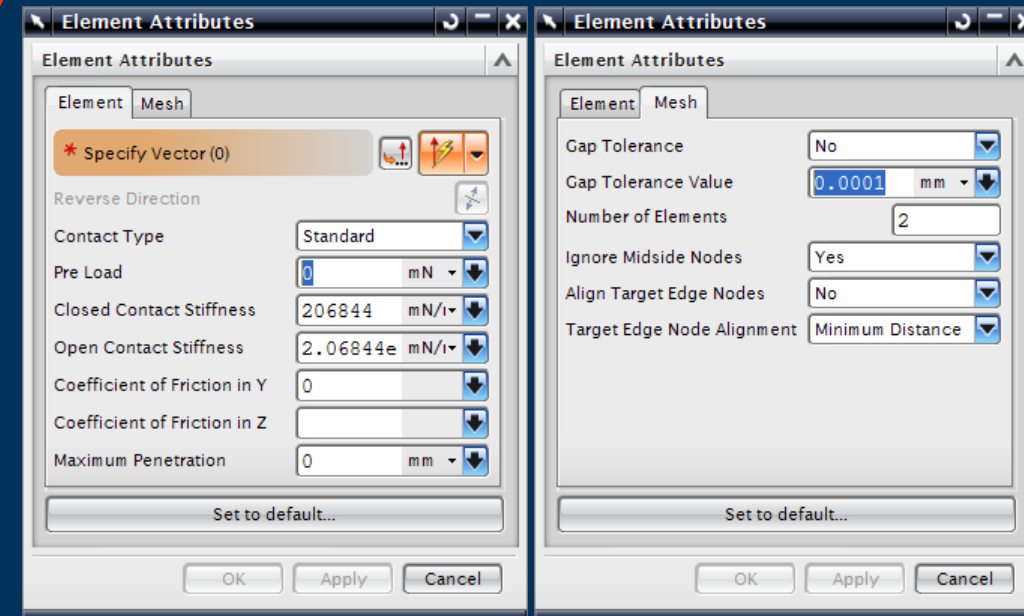
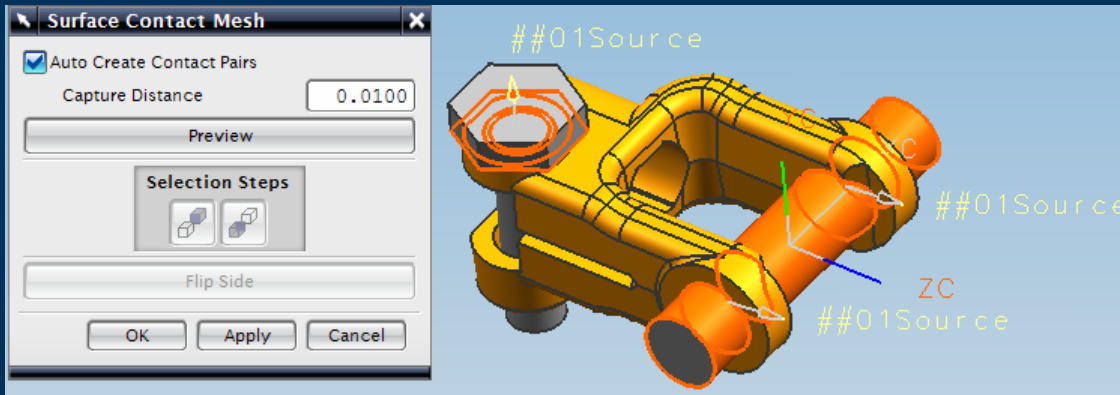
# Mesh Connections – Edge Contact Mesh

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

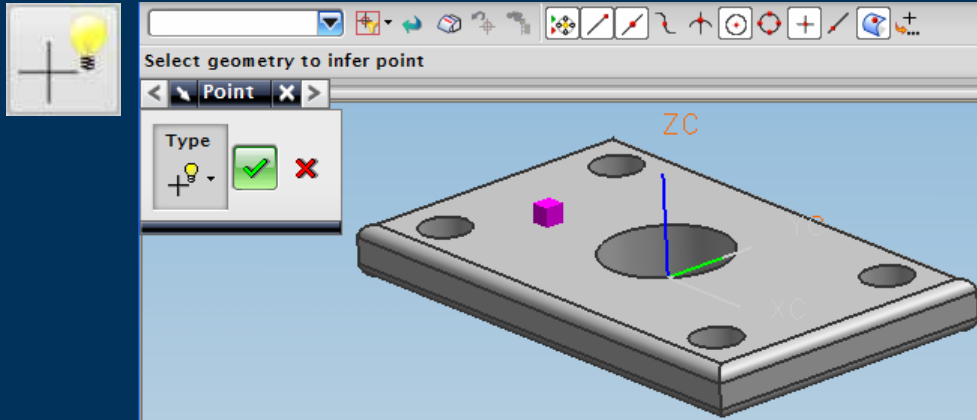


# Mesh Connections – Surface Contact Mesh

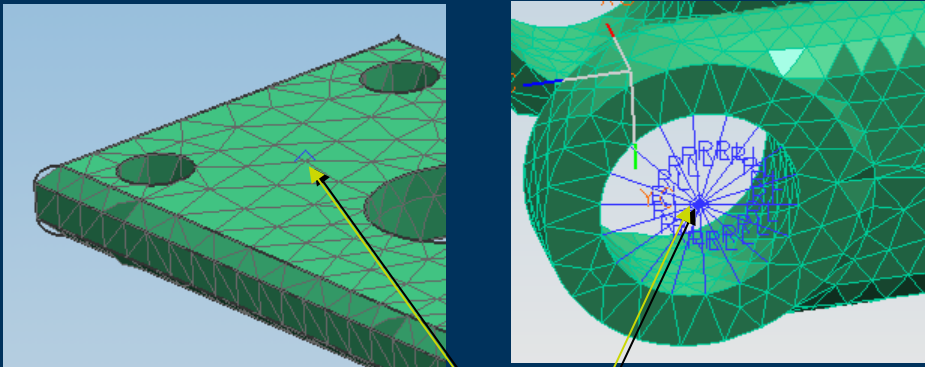
- ▶ Surface Based Contact definition
- ▶ Surfaces are not auto Split or Partitioned
- ▶ Contact Nodes drive meshes on both sides
- ▶ Uses GAP elements to model Contact



# Meshing – Mesh Points

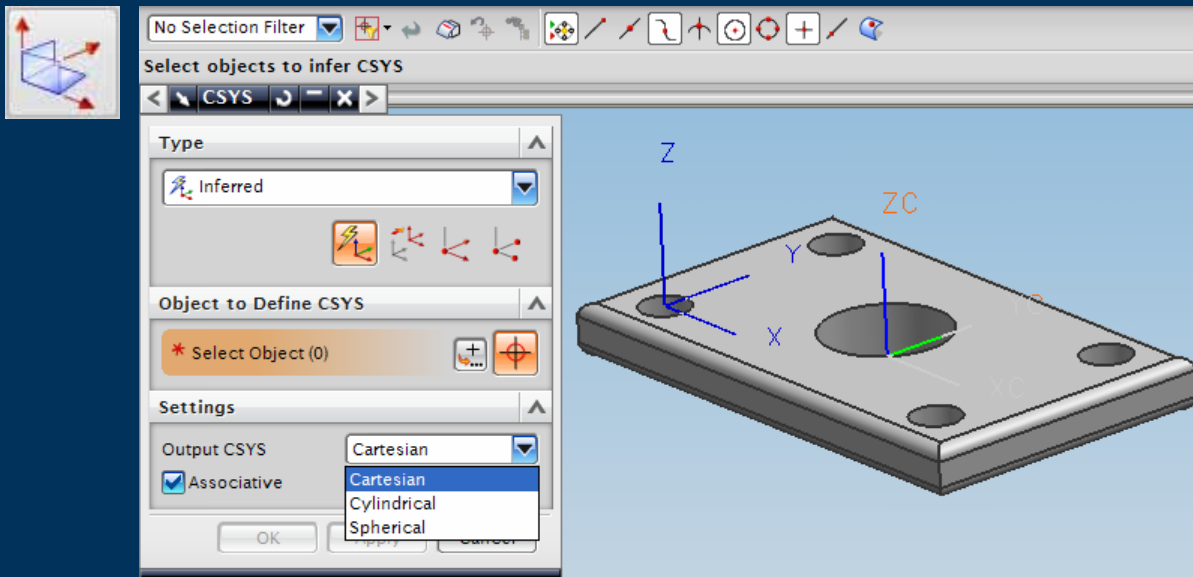


- ▶ Used to create specific location for a node, for example on an Edge or Face
- ▶ Point can be Associative or Non-Associative to Geometry
- ▶ Mesh is associative to the Mesh Point
- ▶ Mesh Point location can be edited
- ▶ Used to create a location for a Load or Boundary Condition, spider for load transfer



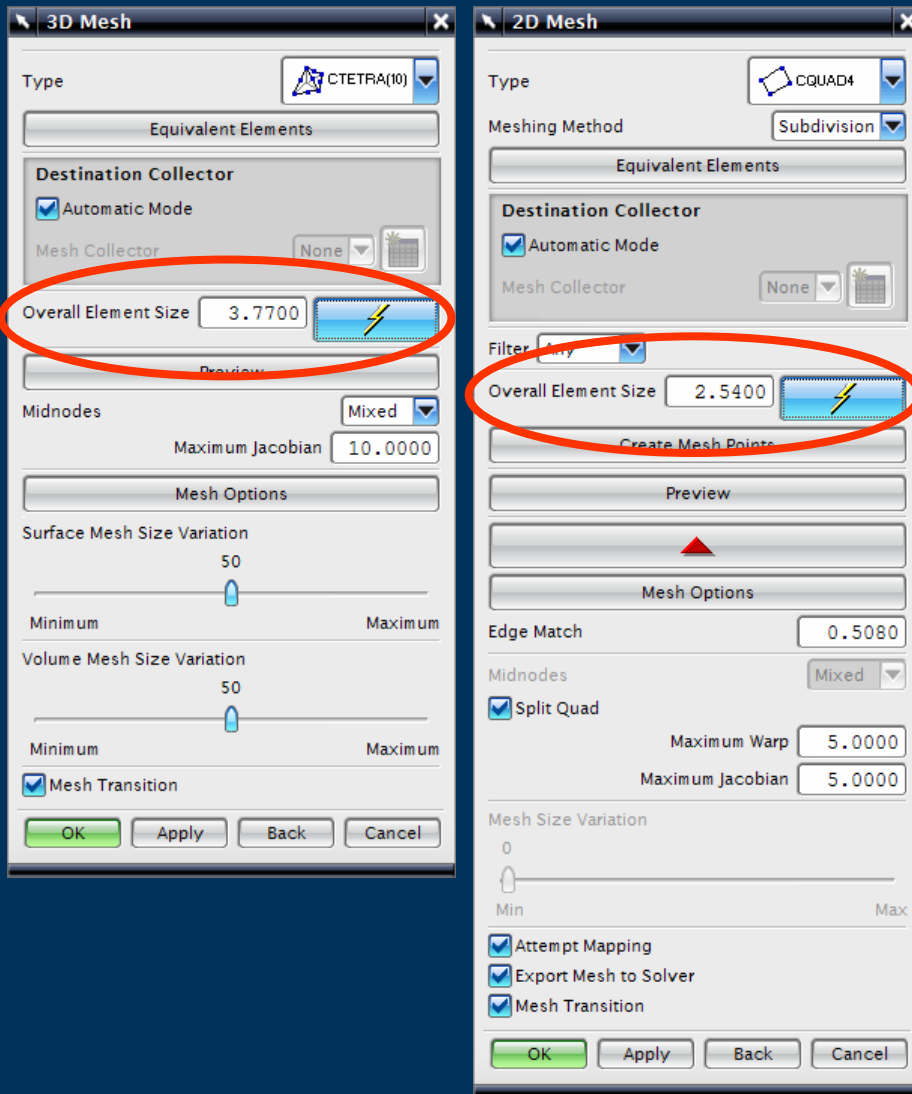
Mesh Point

# Datum Coordinate Systems



- ▶ Datum Coordinate Systems
  - ▶ Cartesian
  - ▶ Cylindrical
  - ▶ Spherical

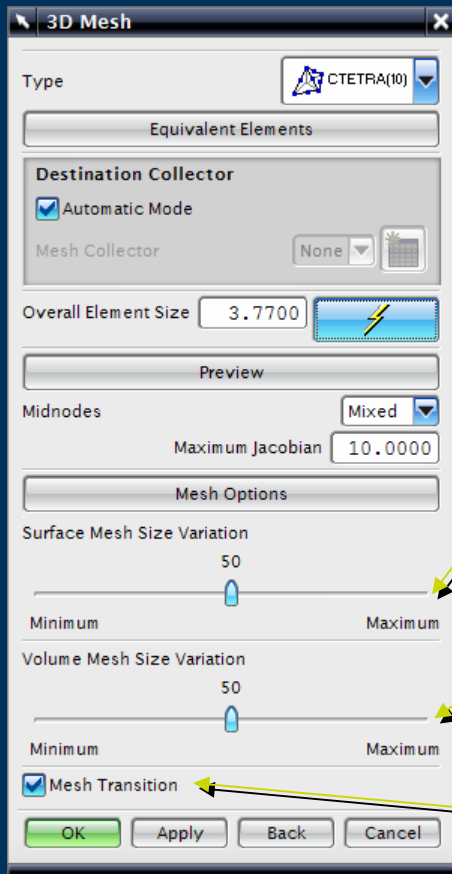
# Mesh Size Selection



- ▶ 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

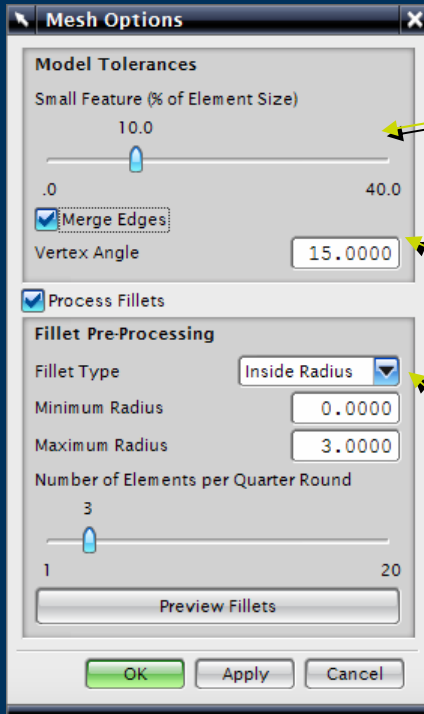
## Volume Mesh Size Variation

- ▶ 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

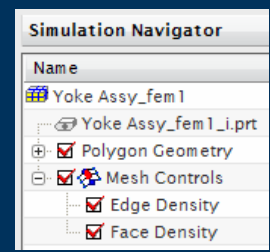
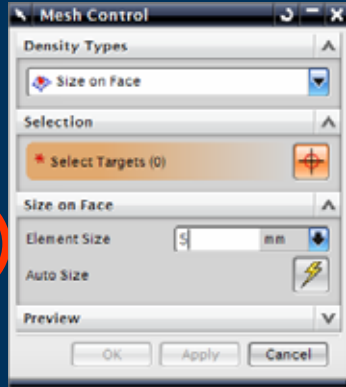
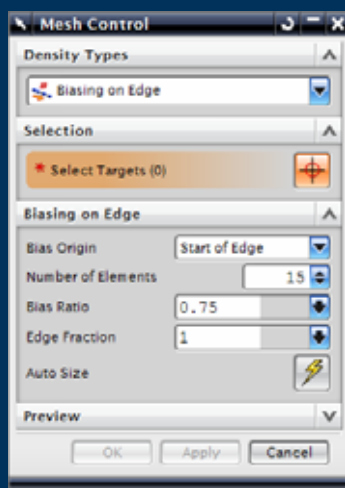
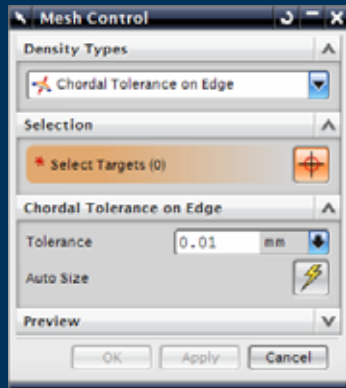
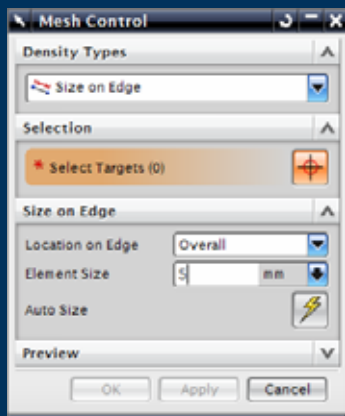
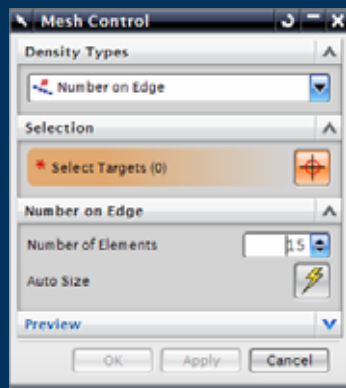
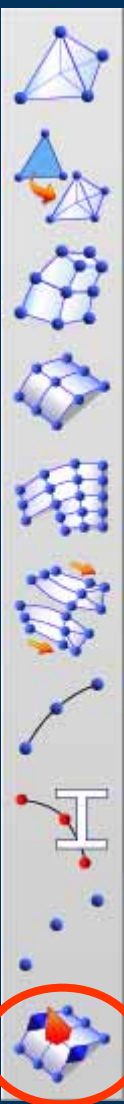
- ▶ Removes the Polygon edge when angle between edges is less than Vertex Angle

▶ Mapped Mesh control of Fillets/Blends faces

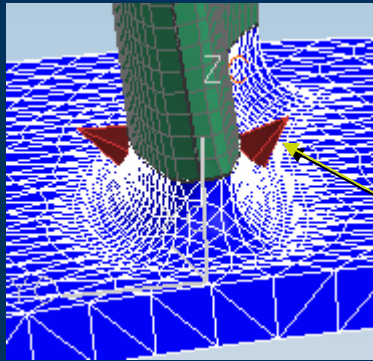
- ▶ Filtered by Inside, Outside or Both
- ▶ Min & Max radius

# Mesh Controls

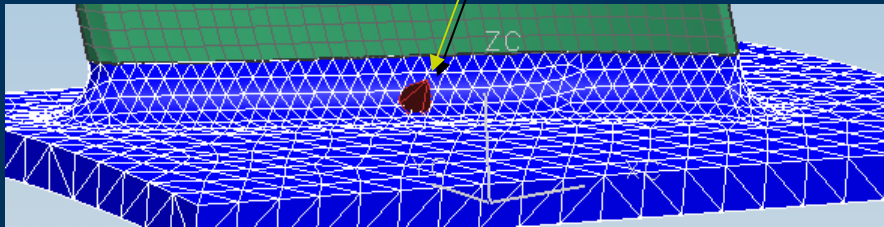
- ▶ Mesh Controls
  - ▶ Number on an Edge
  - ▶ Size on an Edge
  - ▶ Chordal Tolerance on an Edge
  - ▶ Biasing on an Edge
  - ▶ Size on a Face
- ▶ Managed by Mesh Controls Collector



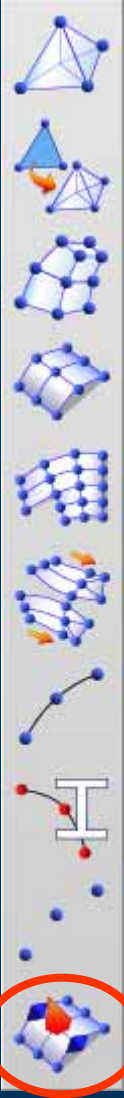
# Mesh Controls



Size on Face  
Mesh Control

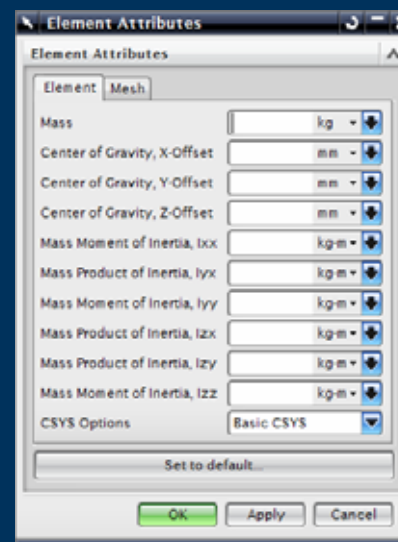
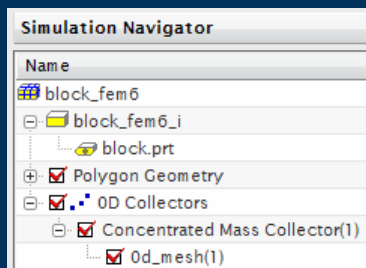
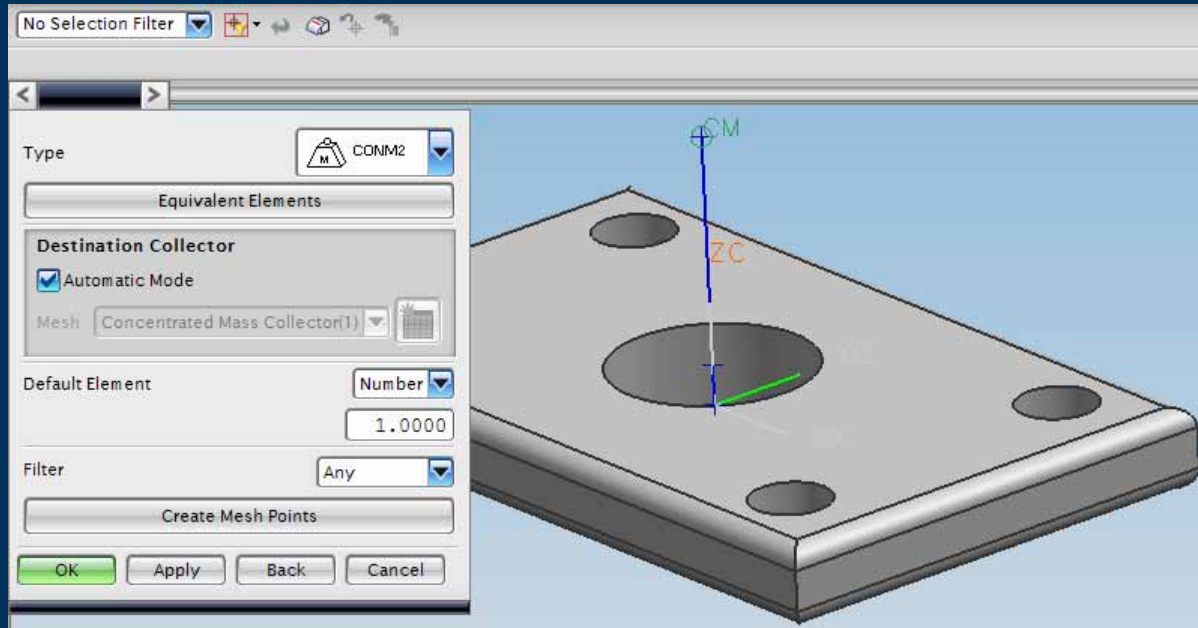


- ▶ Used to control Mesh distribution, quality, mating etc
- ▶ Meshes associative to Mesh Controls



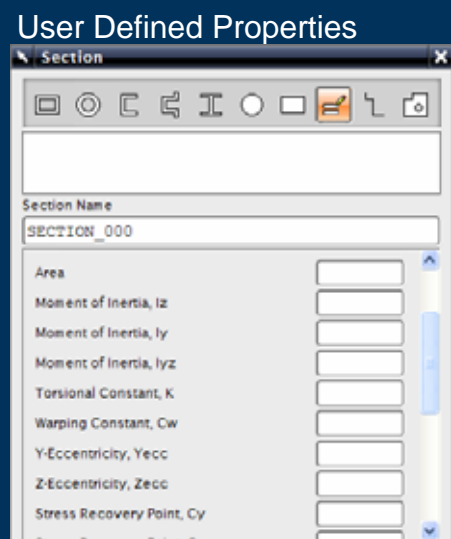
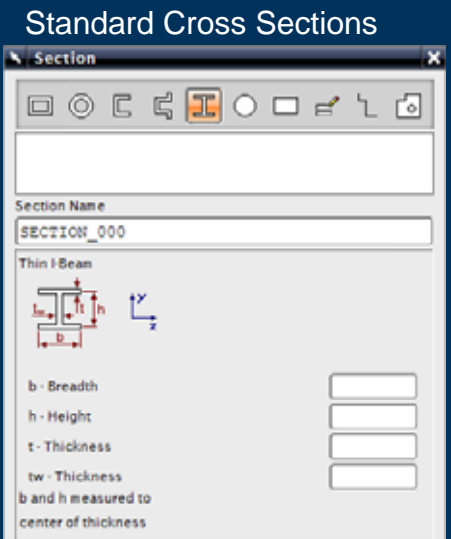
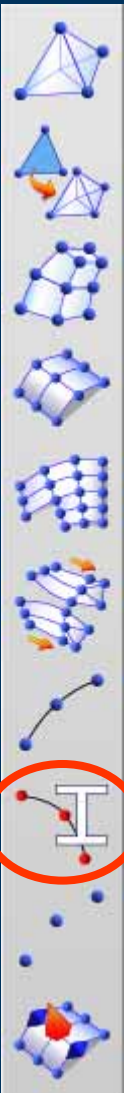
# Meshing – OD Mesh

- ▶ OD or Scalar Elements for Lumped or Distributed Mass

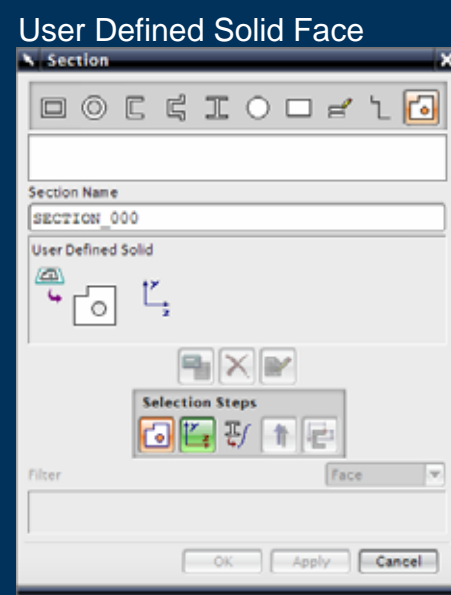
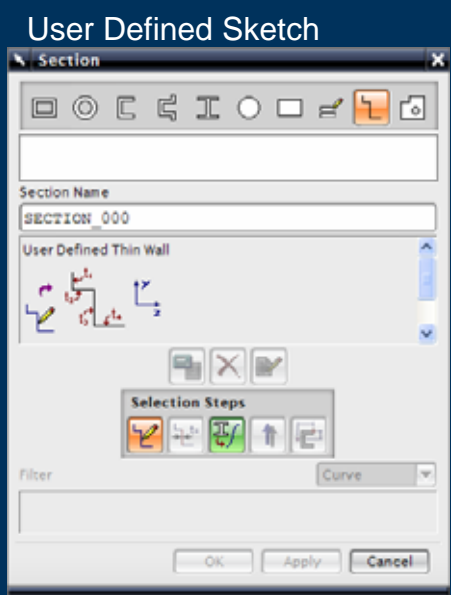


Element Attributes vary according to the Solver Environment & Element Type

# Meshing – 1D Element Cross Sections



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



# Meshing – 1D Mesh

- ▶ 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

**1D Mesh**

Type: CBEAM

Equivalent Elements

Destination Collector: Automatic Mode

Mesh Collector: None

Default Element: Number

Merge Node Tolerance: 1.0000

Selection Steps

Use Mid Nodes:

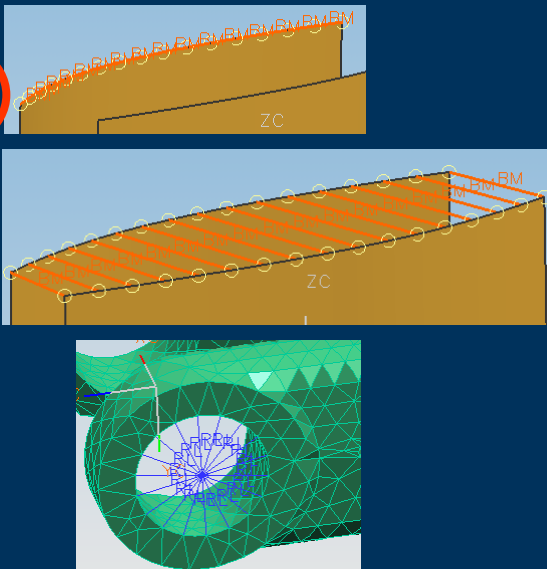
Filter: Any

Buttons: Create Mesh Points, Create Weld Elements, OK, Apply, Back, Cancel

**Simulation Navigator**

Name

- 1D\_elements\_fem1
  - 1D\_elements\_fem1\_i
    - 1D\_elements.prt
  - Polygon Geometry
  - 1D Collectors
    - Beam Collector(1)
      - 1d\_mesh(1)



**Element Attributes**

Element: Mesh

Orientation and Offsets: Specified Values

Orientation Vector: Specify Vector (i)

Reverse Direction

End A Offset

- X (along beam): 0 mm
- Y (along orientation): 0 mm
- Z (sideways): 0 mm

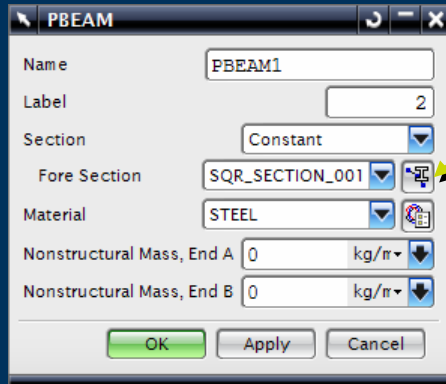
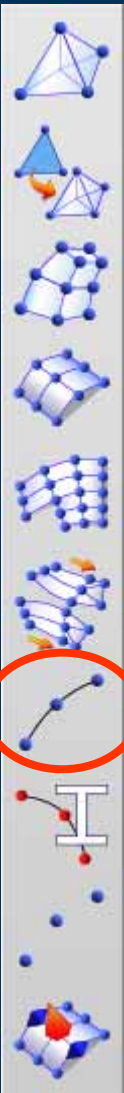
End B Offset

- X (along beam): 0 mm
- Y (along orientation): 0 mm
- Z (sideways): 0 mm

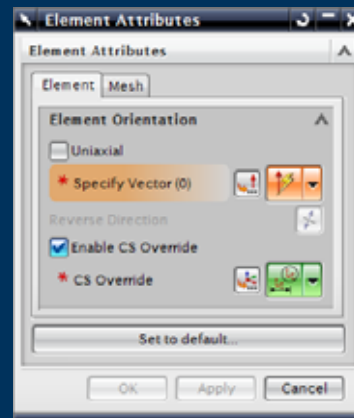
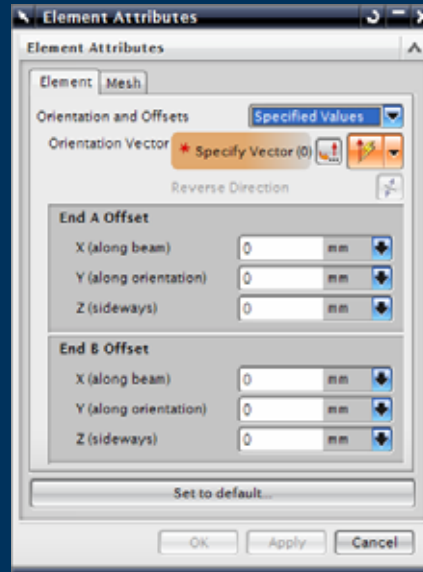
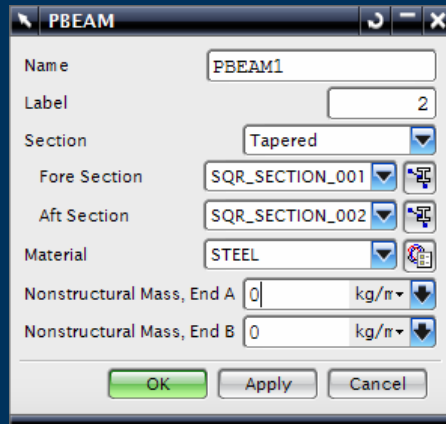
Buttons: OK, Apply, Cancel

Element Attributes vary according to the Solver Environment & Element Type

# Meshing – 1D Mesh – Element Attributes



1D Element Cross Sections

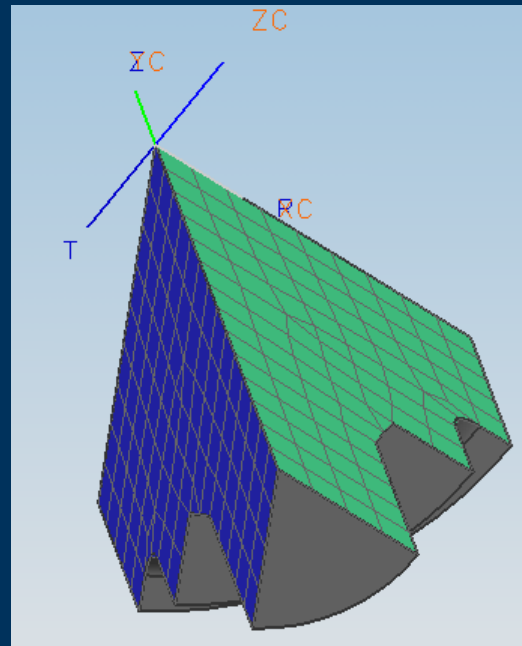
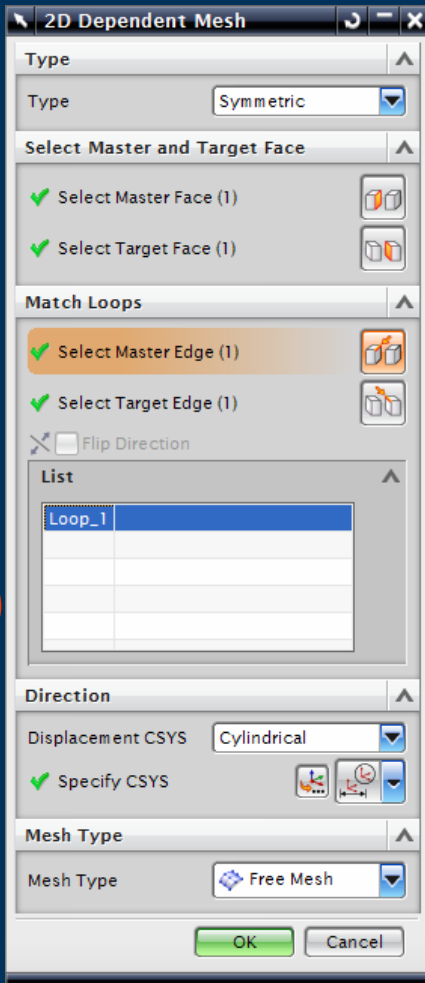


- ▶ 1D Elements reference a Physical Property
  - ▶ Material
  - ▶ Section(s)
- ▶ Element Attributes for Beams, Bars and Rods
  - ▶ Beams & Bars require Orientation vector
    - ▶ Inherited from Geometry
    - ▶ Specific Values
  - ▶ Bushes require Axis Definition

Element Attributes vary according to the Solver Environment & Element Type

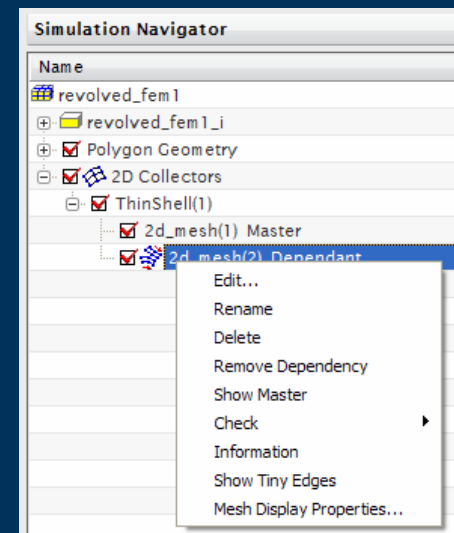


# Meshing – 2D Dependant Mesh

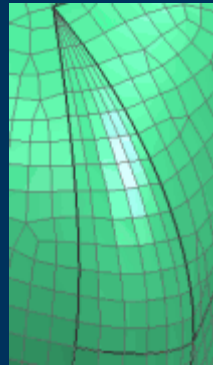
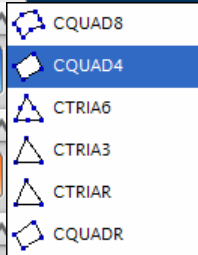
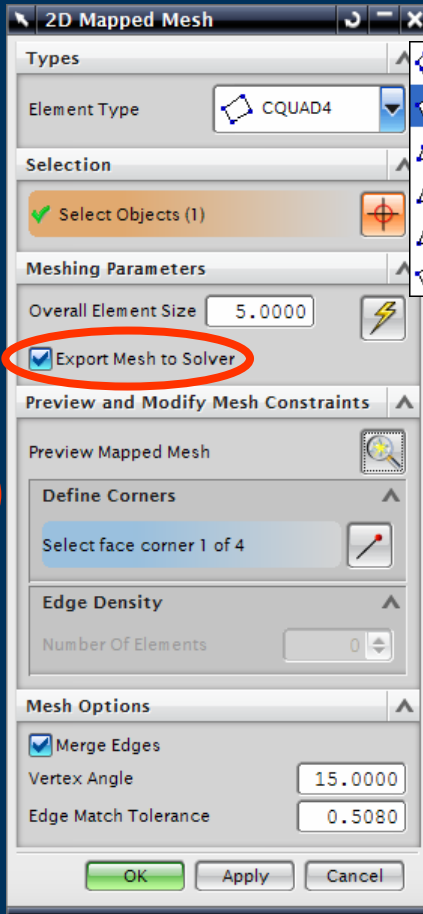


- ▶ Master & Target face selection
- ▶ Topologically Identical Faces
- ▶ Multiple Faces and Loops
- ▶ Coordinate System Selection
- ▶ Mesh Type Selection
  - ▶ Free or Mapped
  - ▶ New Mesh or Use Existing Mesh on Master Face
- ▶ Managed by 2D Collector

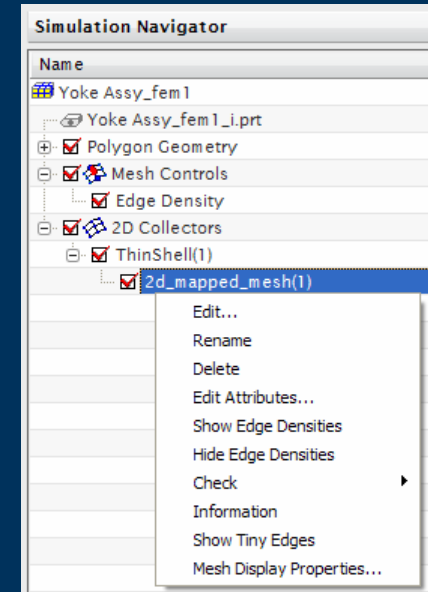
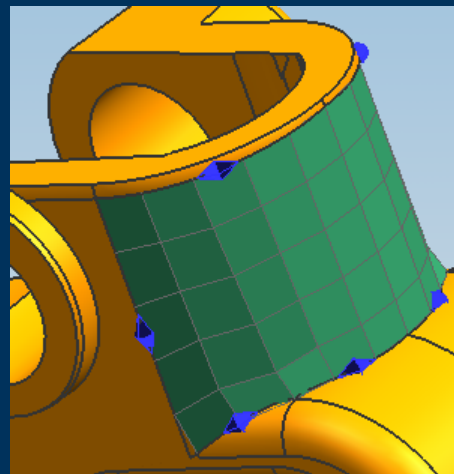
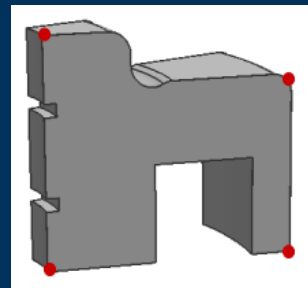
- ▶ Uses
  - ▶ Contact Regions
  - ▶ Flange Mating
  - ▶ Symmetric Faces



# 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



**2D Mesh**

Type: **CTRIA3**

Meshing Method: **Subdivision**

Equivalent Element: **Subdivision Paver**

Destination Collector:  Automatic Mode

Mesh Collector: **None**

Filter: **Any**

Overall Element Size: **1.0000**

Create Mesh Points

Preview

Mesh Options

Edge Match: **0.5080**

Midnodes: **Mixed**

Split Quad

Maximum Warp: **5.0000**

Maximum Jacobian: **5.0000**

Mesh Size Variation: **0** (Min to Max)

Attempt Mapping

Export Mesh to Solver

Mesh Transition

OK Apply Back Cancel

**2D Mesh**

Type: **CQUAD8**

Meshing Method: **Subdivision**

Equivalent Element: **Subdivision Paver**

Destination Collector:  Automatic Mode

Mesh Collector: **None**

Filter: **Any**

Overall Element Size: **1.0000**

Create Mesh Points

Preview

Mesh Options

Edge Match: **0.5080**

Midnodes: **Mixed**

Split Quad

Maximum Warp: **5.0000**

Maximum Jacobian: **5.0000**

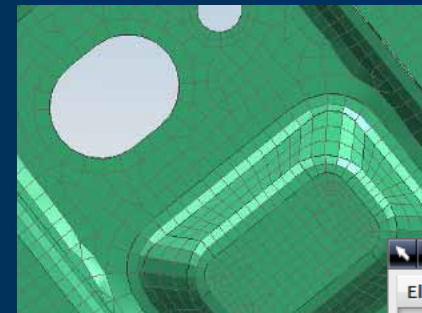
Mesh Size Variation: **0** (Min to Max)

Attempt Mapping

Export Mesh to Solver

Mesh Transition

OK Apply Back Cancel



**Simulation Navigator**

Name

- 1D\_elements\_fem1
- 1D\_elements\_fem1\_i
- Polygon Geometry
- 2D Collectors
- ThinShell(1)
- 2d\_mesh(1)

**Element Attributes**

Element: **Mesh**

Shell Offset: **2** mm

Enable MCID

Ignore Midsurface Thickness

OK Apply Cancel

# Meshing – 2D Mesh Seeding for 3D Mesh



**2D Mesh**

Type: **CTRIA6**

Meshing Method: **Subdivision**

Equivalent Elements

Destination Collector

Automatic Mode

Mesh Collector: **None**

Filter: **Any**

Overall Element Size: **18.2000**

Create Mesh Points

Preview

Mesh Options

Edge Match: **0.5080**

Midnodes: **Mixed**

Split Quad

Maximum Warp: **5.0000**

Maximum Jacobian: **5.0000**

Mesh Size Variation

0

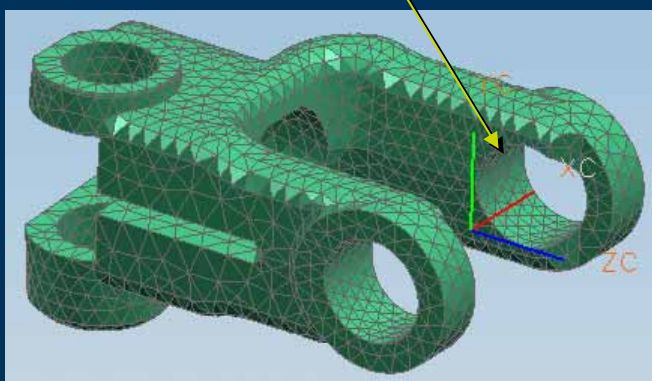
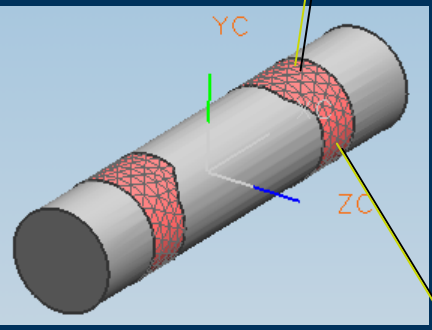
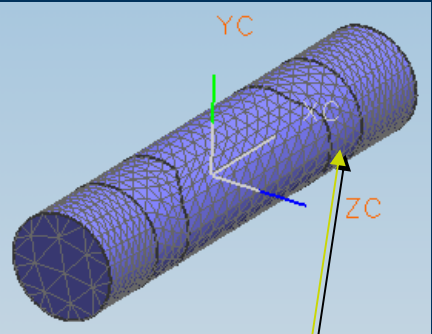
Min Max

Attempt Mapping

**Export Mesh to Solver**

Mesh Transition

OK



- ▶ 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

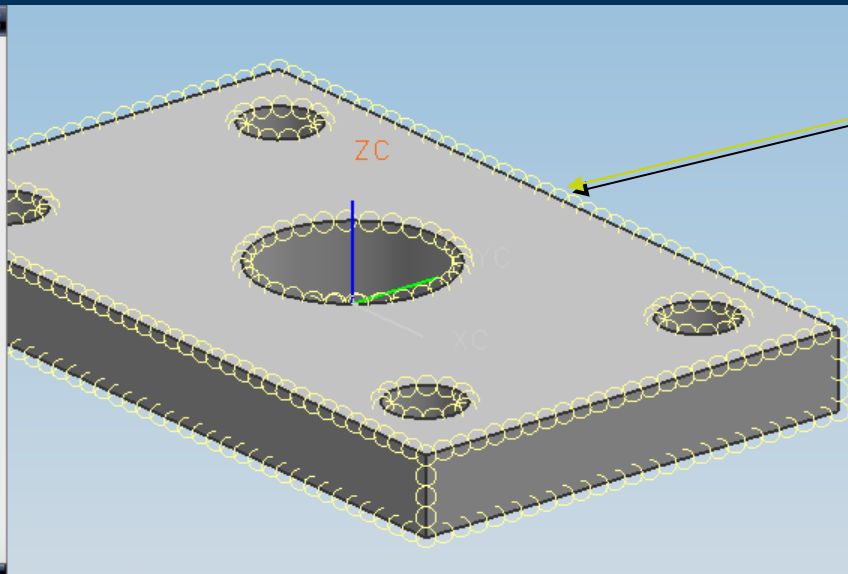
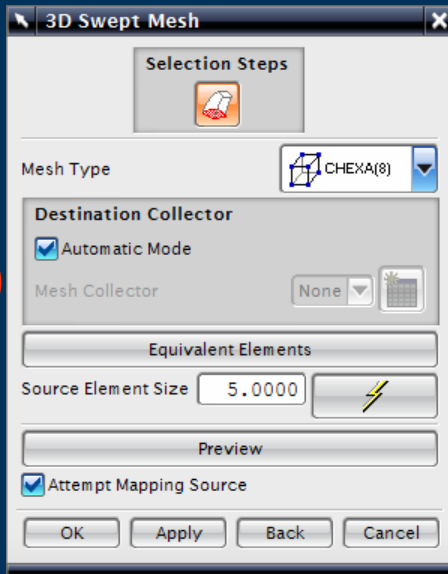
Attempt Mapping

Export Mesh to Solver

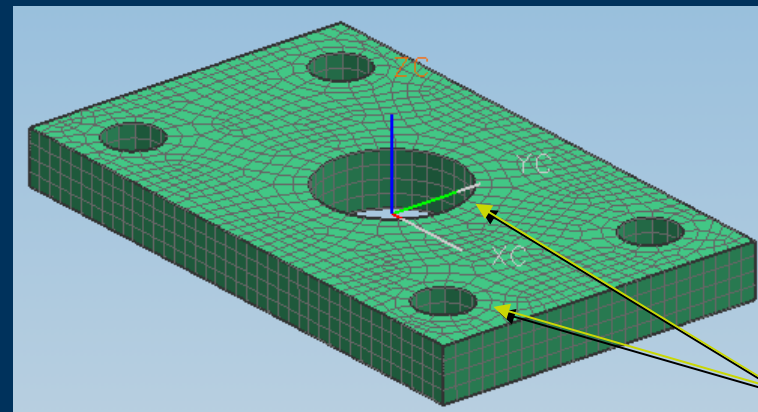
Mesh Transition

OK Apply Back Cancel

# Meshing – 3D Swept Mesh

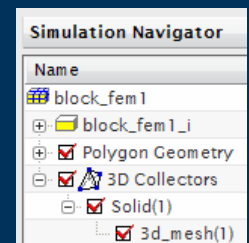


Preview of Mesh distribution along edges

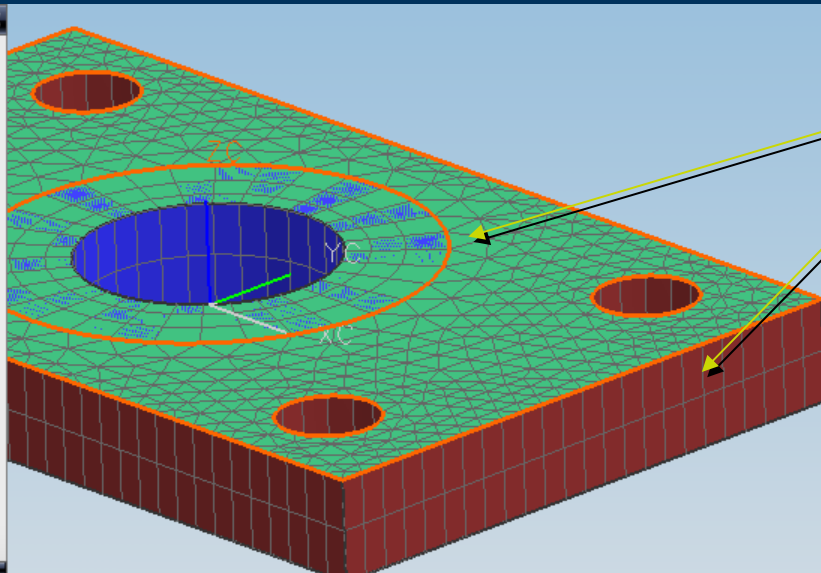
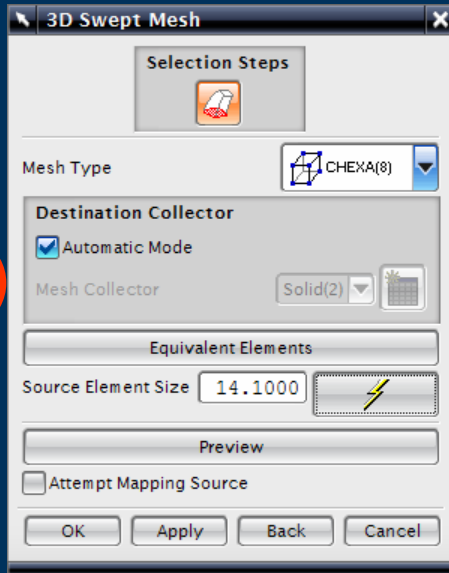


Notice Mapped meshing around holes

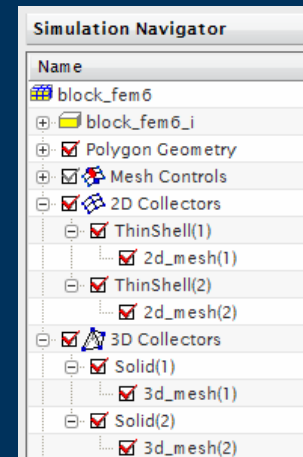
- ▶ 3D Swept Mesh requires source face selection
- ▶ This face is meshed and swept through the volume



# Meshing – 3D Swept Mesh

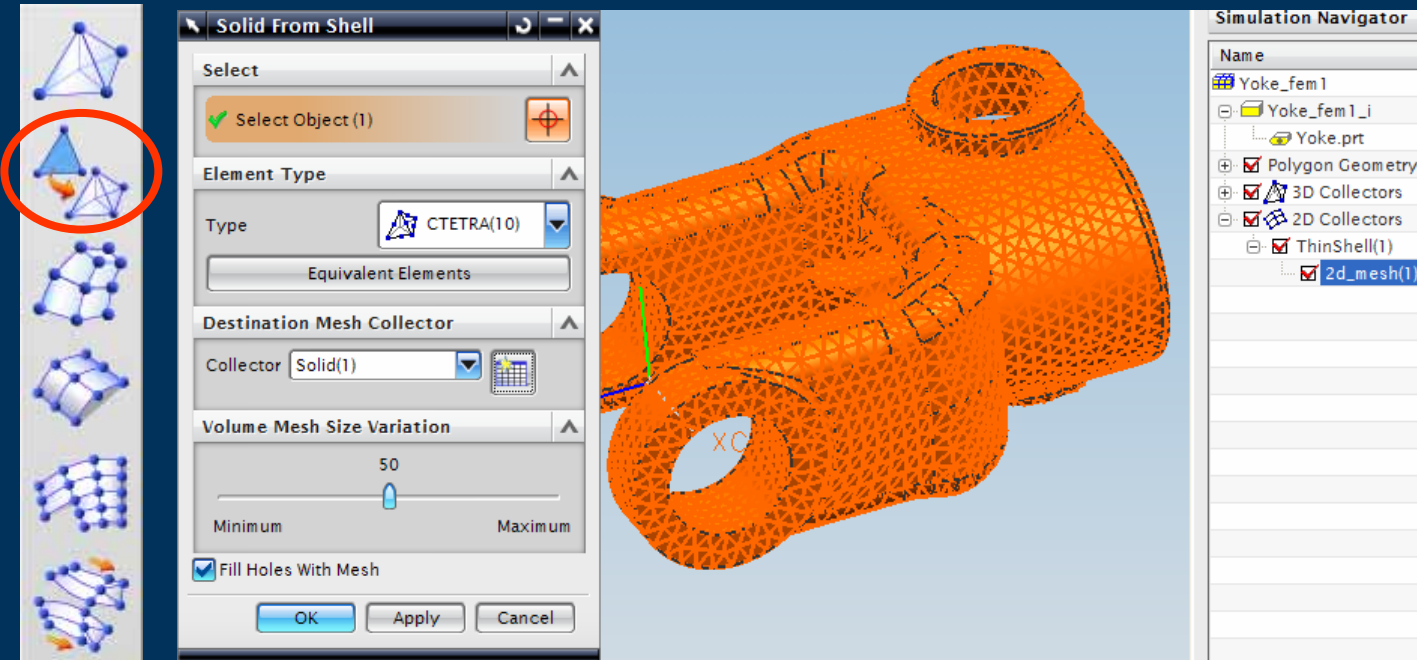


Green – 2D Seed Meshes to define 3D Swept Mesh  
Brown & Blue – 3D Swept Meshes

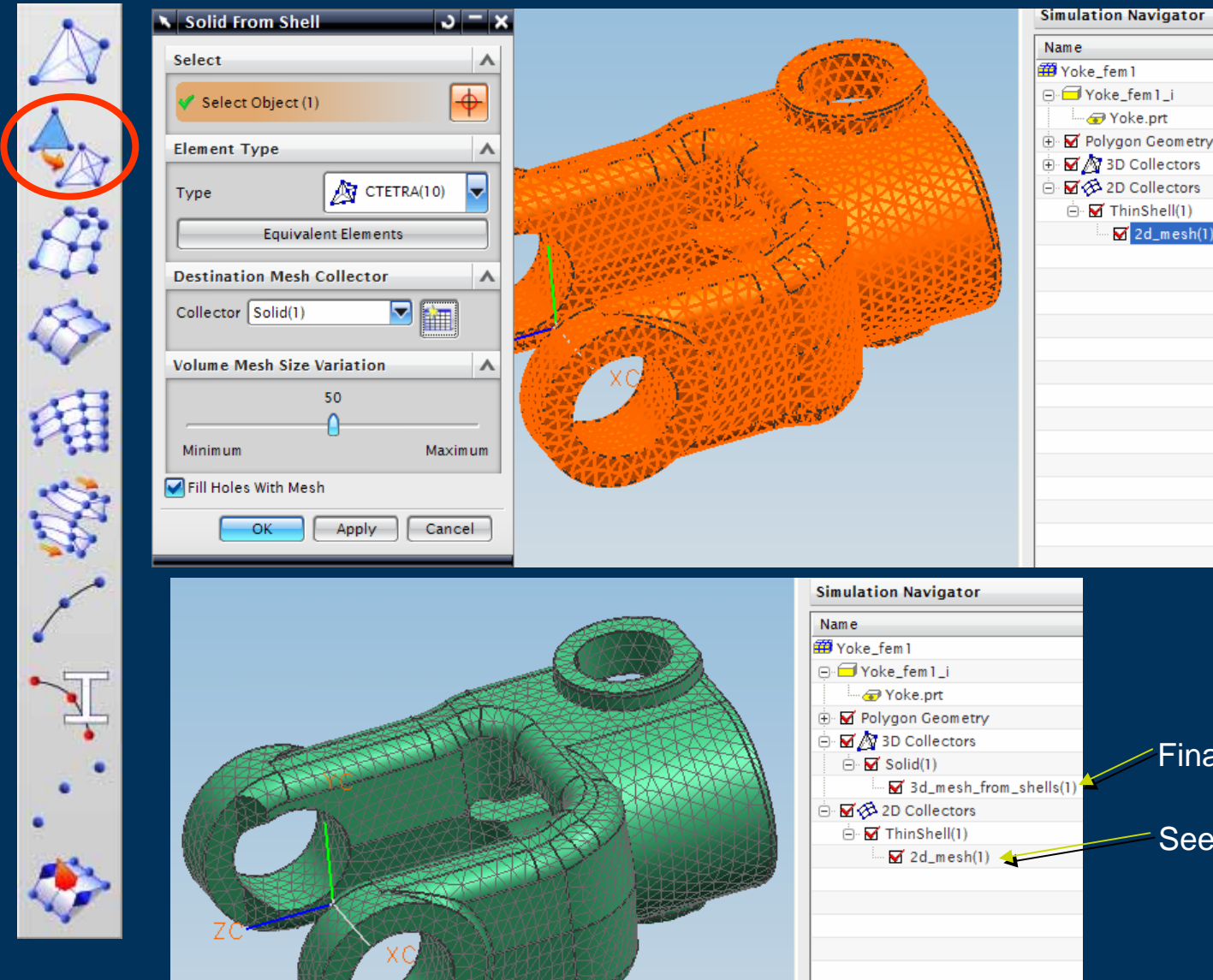


- ▶ 2D Mesh the Seed faces to control mesh type and distribution
- ▶ 2D Mesh – Turn OFF “Export Mesh to Solver” then this mesh is not written to the solver input deck
- ▶ 3D Swept Mesh starts with these Seed meshes

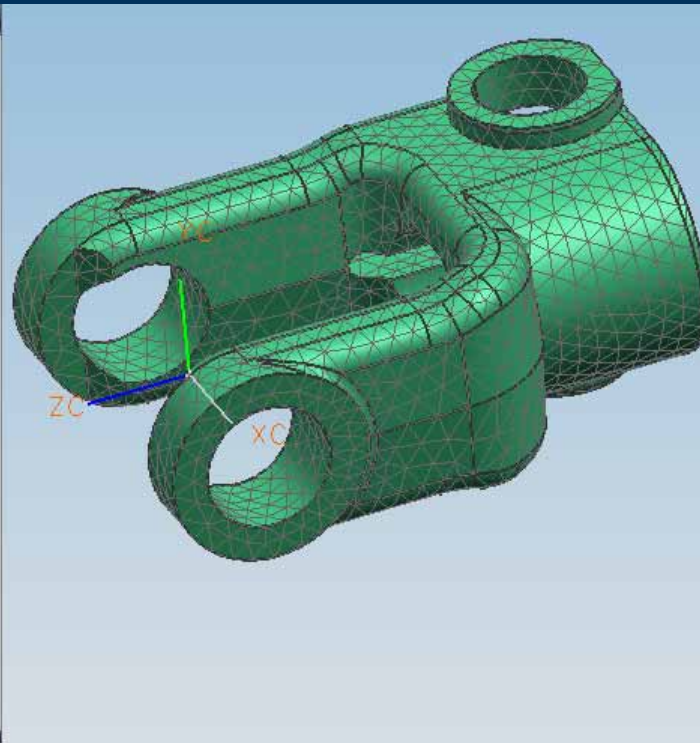
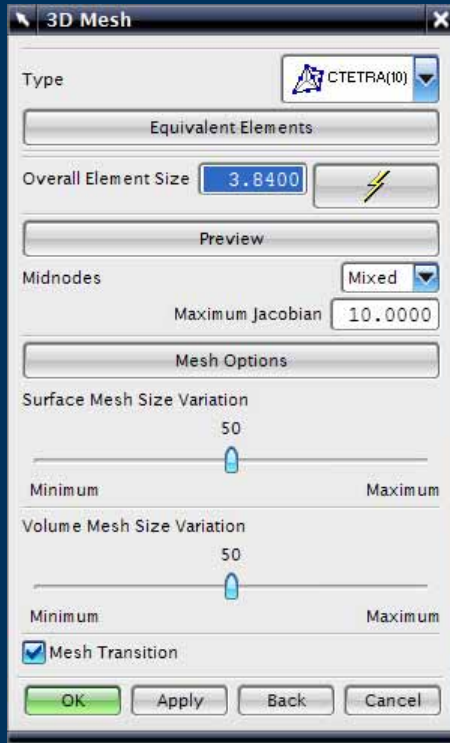
# Meshing – Solid from Shell Mesh



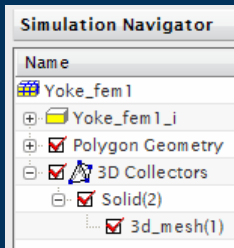
- ▶ Generates a 3D Mesh from a closed surface mesh
- ▶ Used when the importing CAD geometry is not complete and CAE topology is used to close the volume



# Meshing – 3D Tetrahedral Mesh



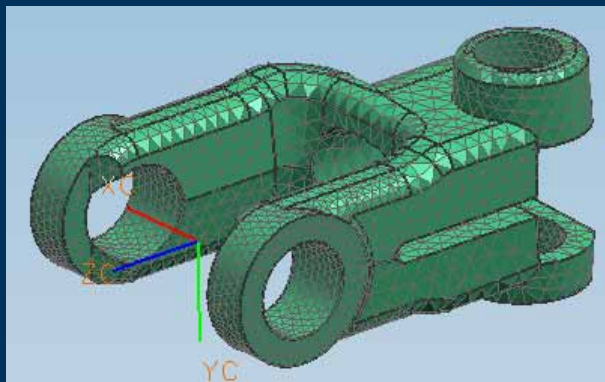
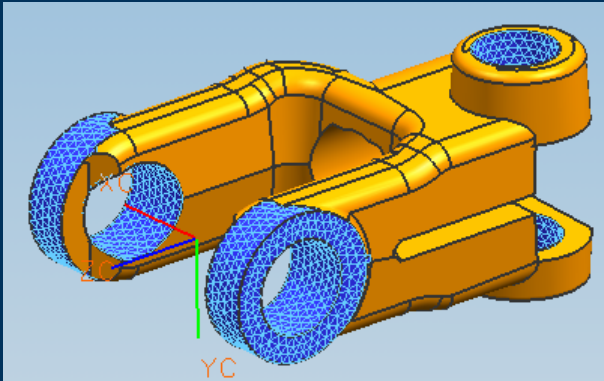
- ▶ 3D Tet Mesh
- ▶ Auto Element Size sets a good value to start the meshing process
- ▶ Same Mesh Options as 2D Mesh



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

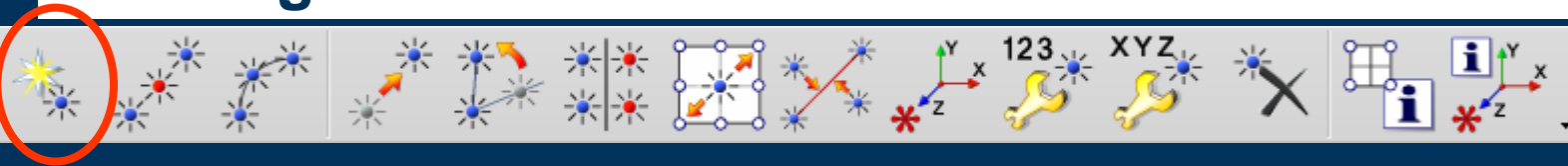


# Meshing – 3D Tetrahedral Mesh

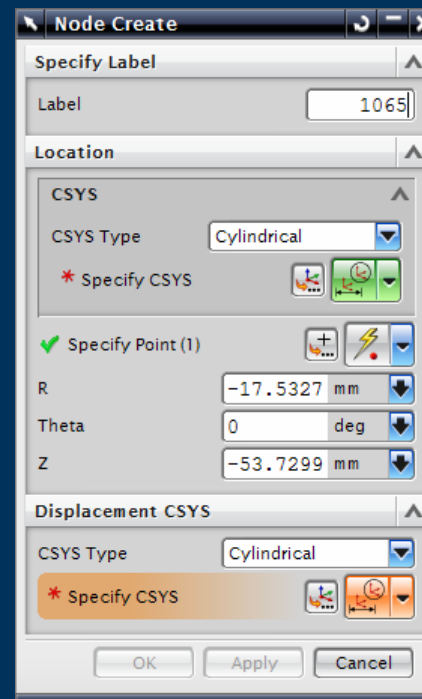
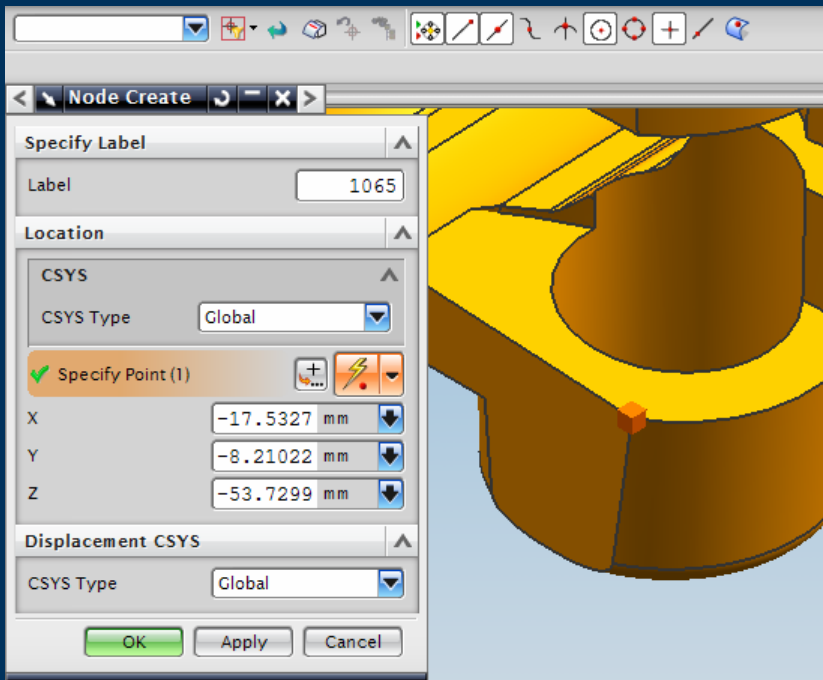


- ▶ Good technique is to “seed” the mesh by applying 2D Mesh to selected faces
- ▶ Add refinement and detail control where required

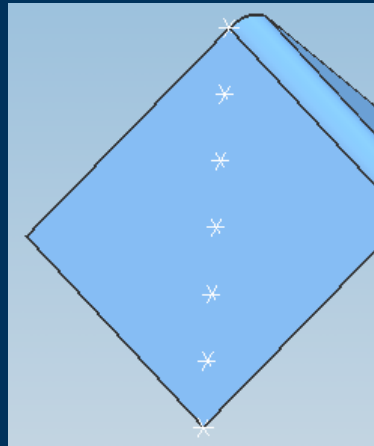
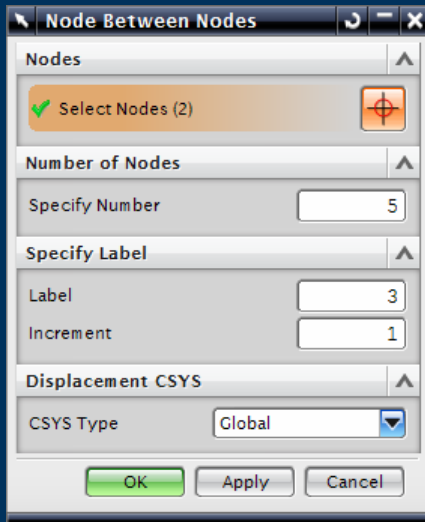
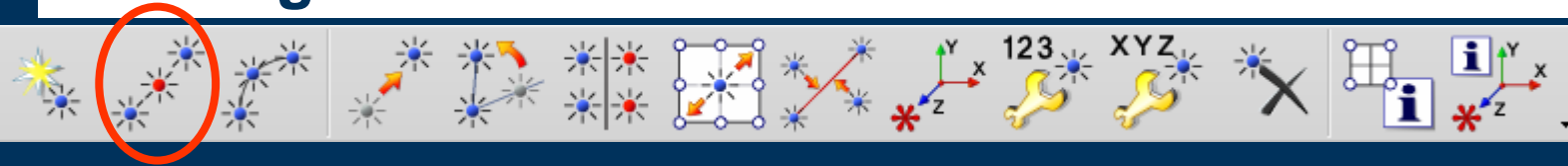
# Meshing – Node Create



- ▶ Location & Displacement by Global or Selected CSYS

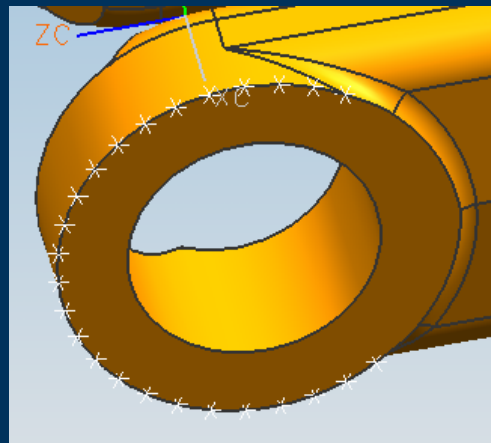
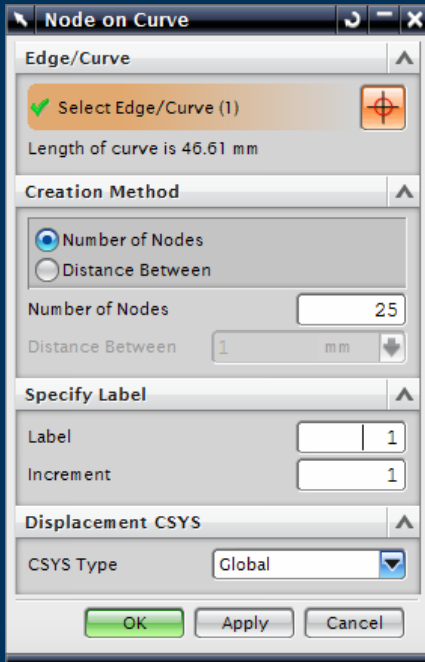
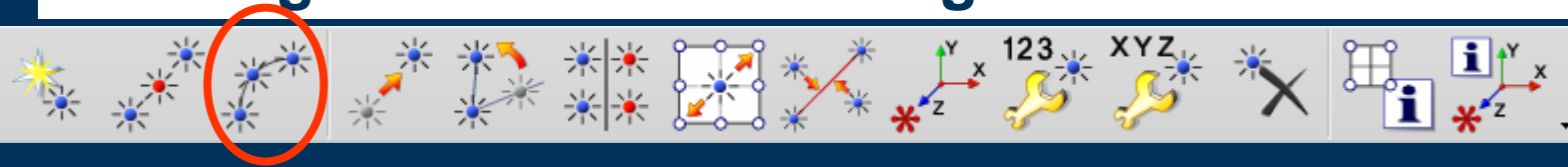


# Meshing – Node Between Nodes



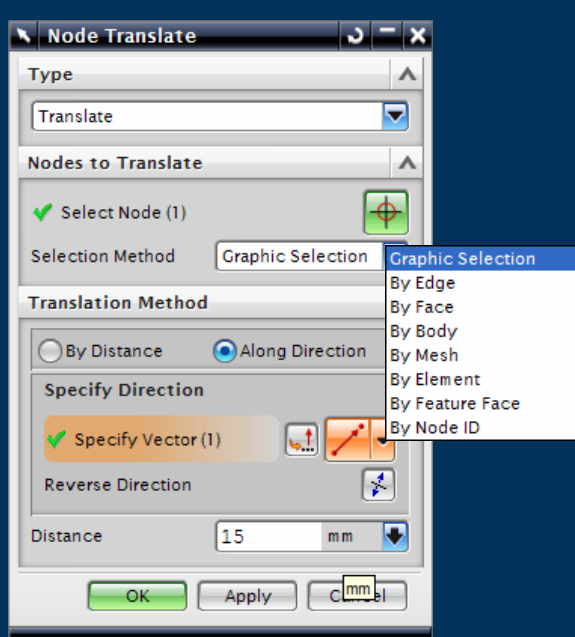
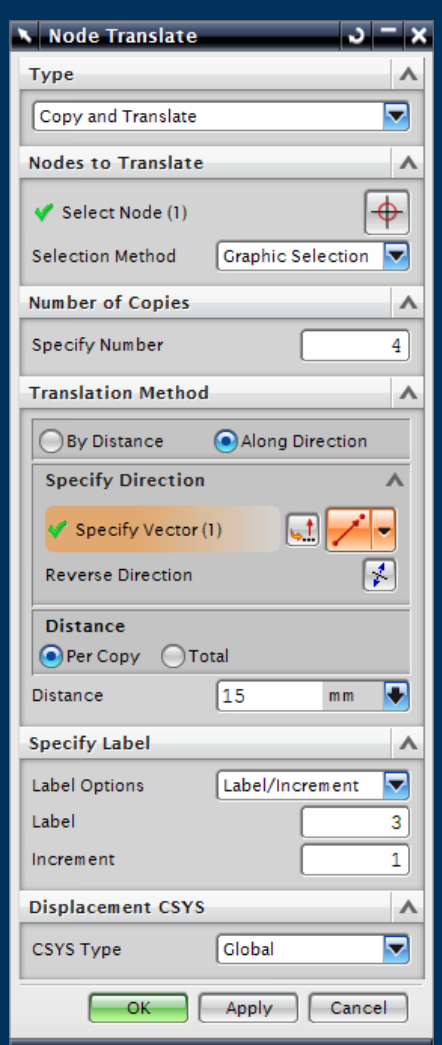
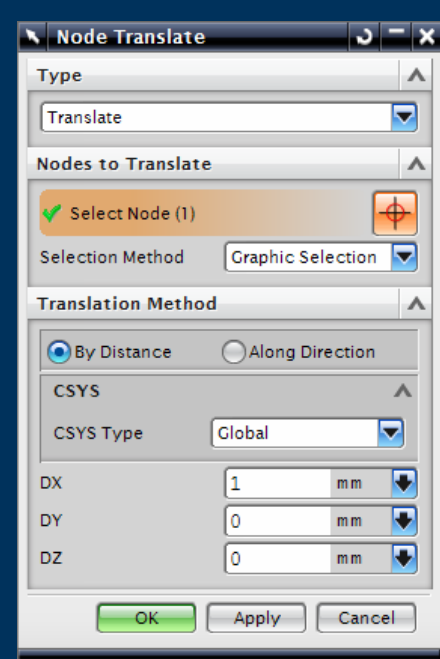
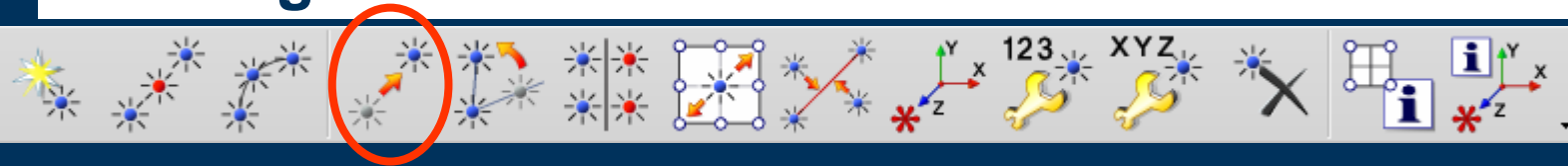
- ▶ Place Nodes equidistant between 2 selected Nodes
- ▶ Geometry independent ie does not track surface(s) between Nodes
- ▶ Displacement by Global or Selected CSYS

# Meshing – Node on Curve/Edge



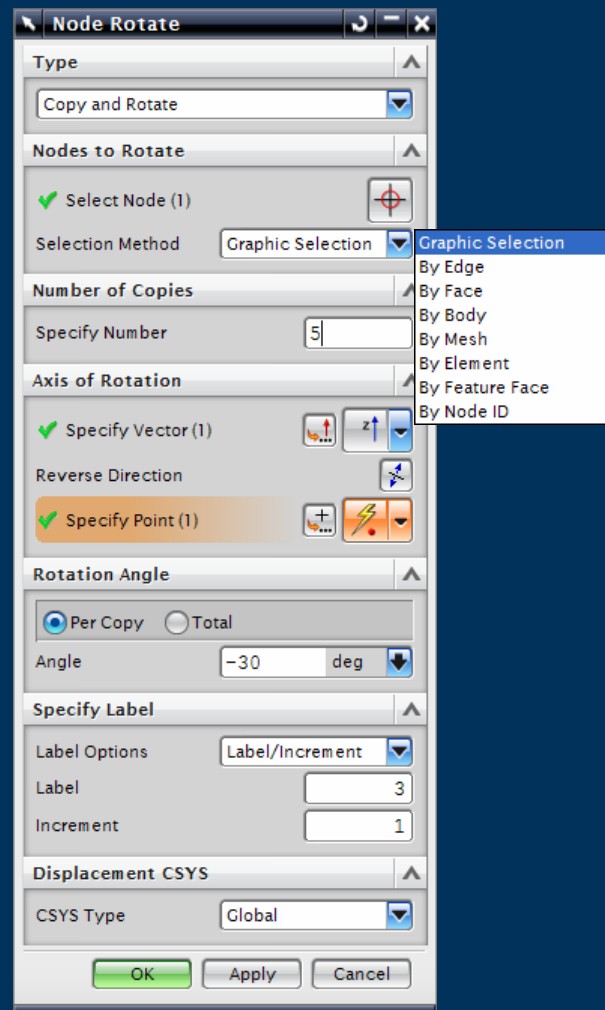
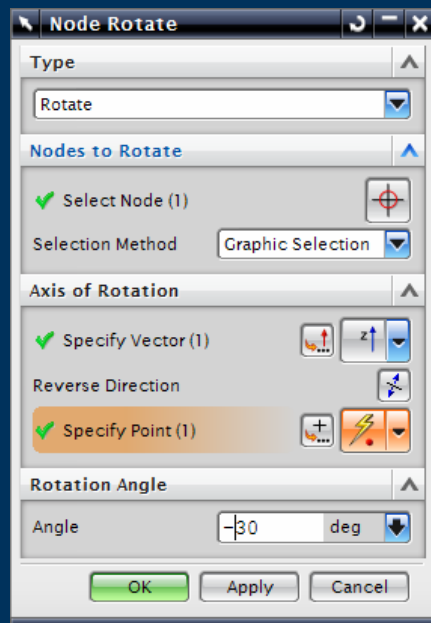
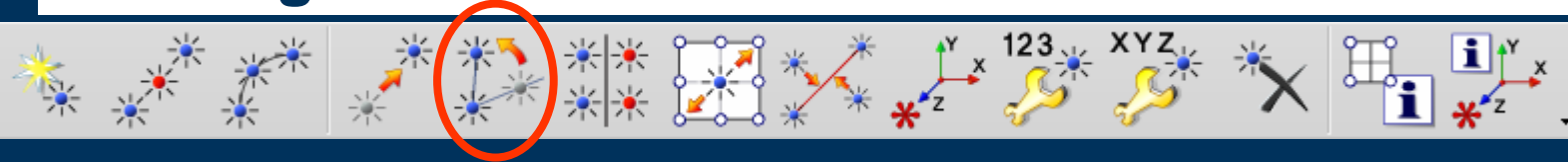
- ▶ Place Nodes equidistant along a selected Edge/Curve
- ▶ Number of Nodes or Distance between Nodes
- ▶ Displacement by Global or Selected CSYS

# Meshing – Node Translate

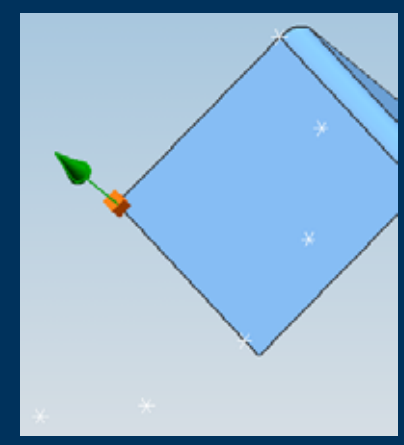


- ▶ Node Translate/Copy
- ▶ Multiple Selection Methods
- ▶ Displacement by Global or Selected CSYS

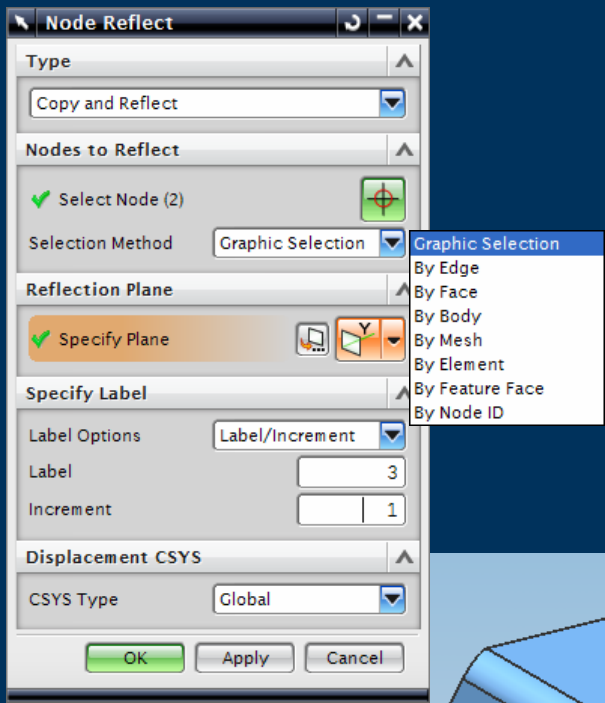
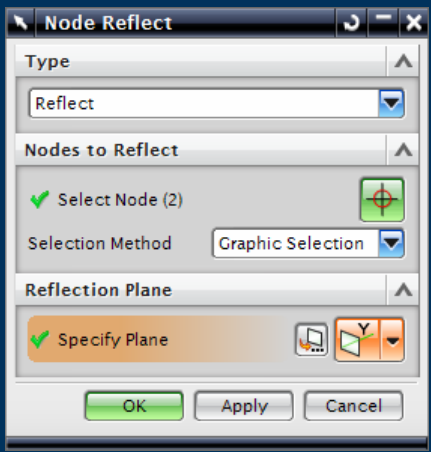
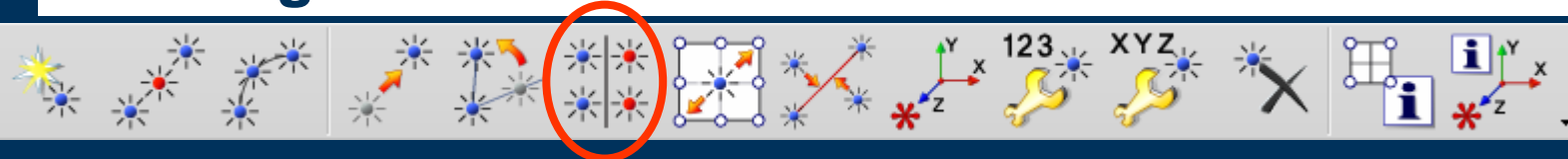
# Meshing – Node Rotate



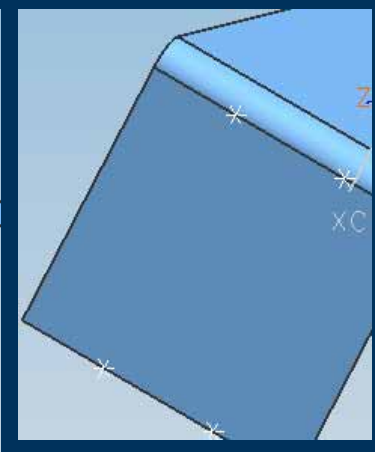
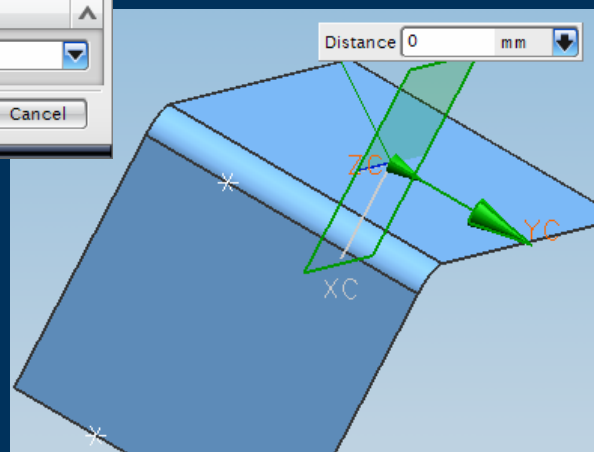
- ▶ Node Rotate/Copy
- ▶ Multiple Selection Methods
- ▶ Displacement by Global or Selected CSYS



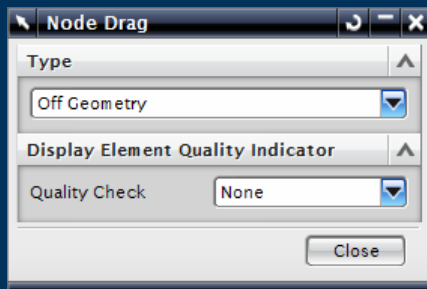
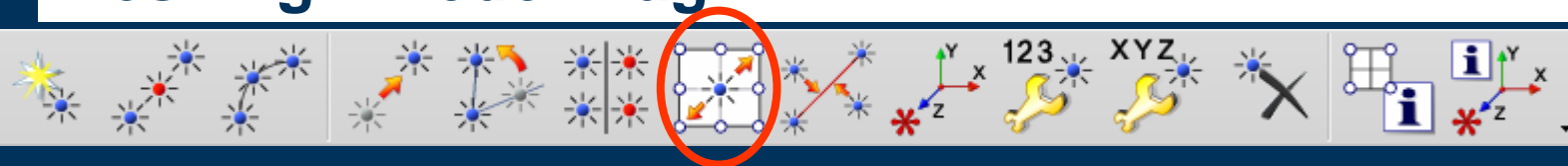
# Meshing – Node Reflect



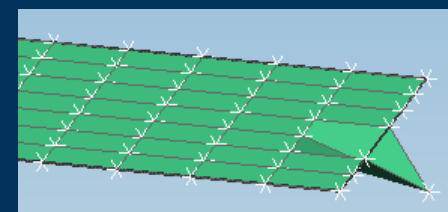
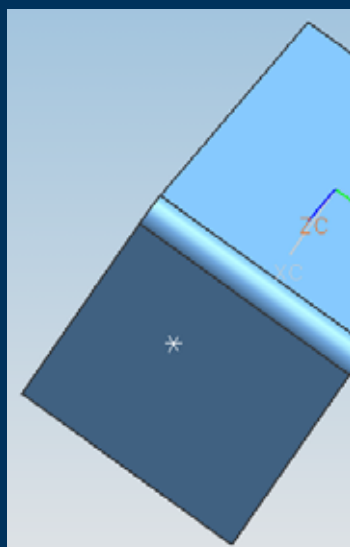
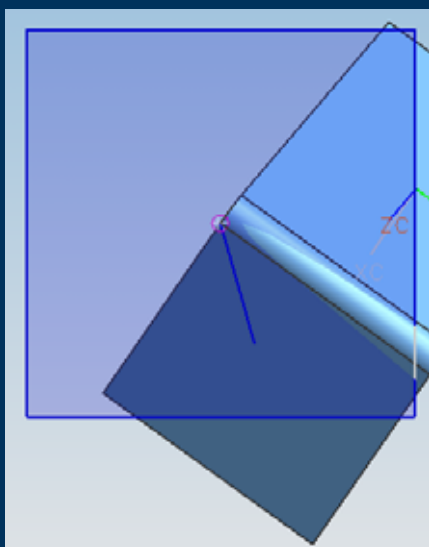
- ▶ Node Reflect/Copy
- ▶ Multiple Selection Methods
- ▶ Displacement by Global or Selected CSYS



# Meshing – Node Drag

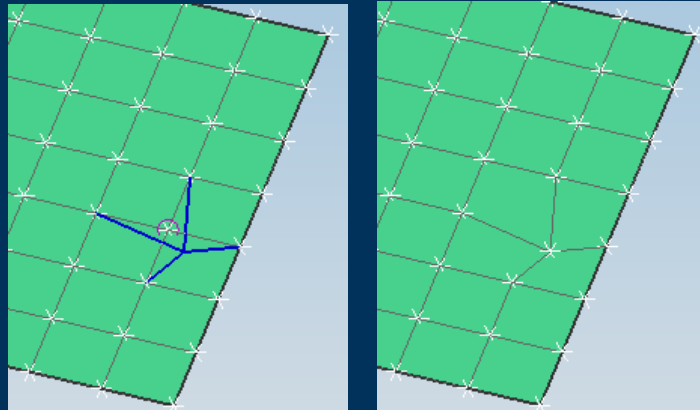
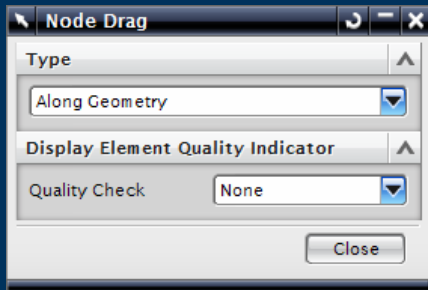
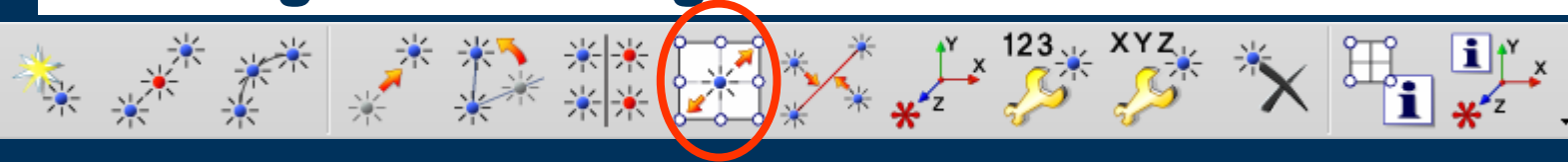


- ▶ Node Dragging Off Geometry
  - ▶ Drags in a plane parallel to screen through start node location





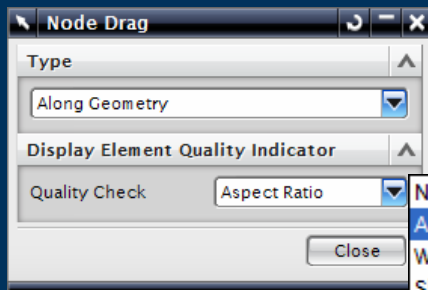
# Meshing – Node Drag



- ▶ Node Dragging On Geometry
- ▶ Drags on associated geometry

- ▶ Edge
- ▶ Face

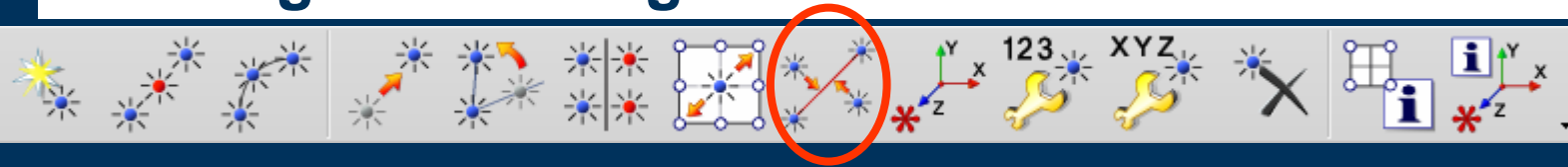
- ▶ Dynamic Display of Element Quality Check



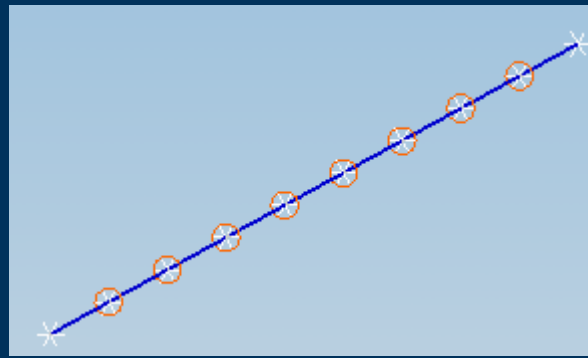
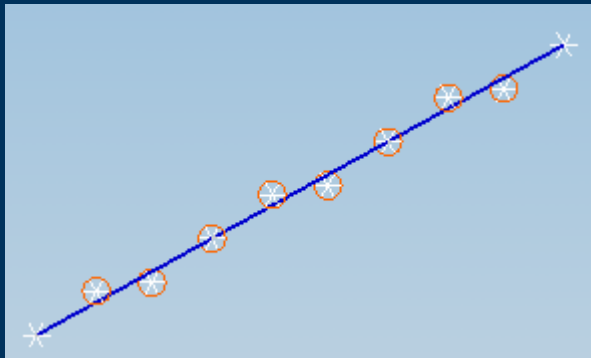
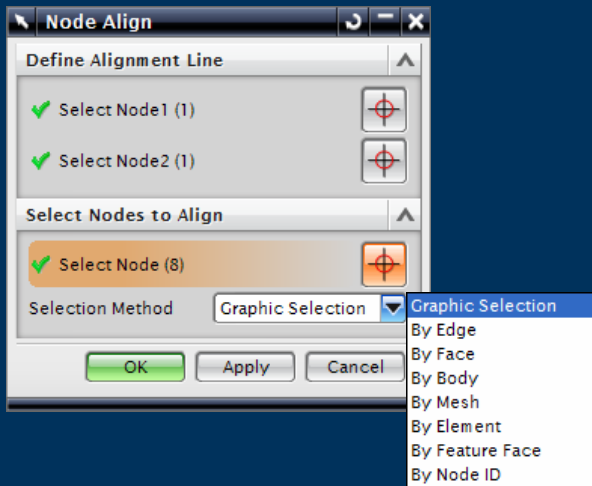
- None
- Aspect Ratio
- Warp
- Skew
- Taper
- Jacobian Ratio
- Minimum Angle
- Maximum Angle



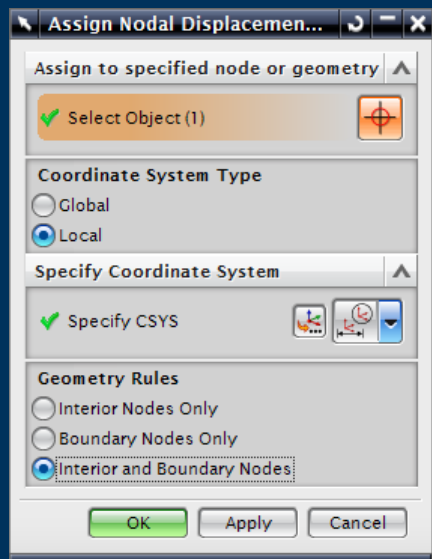
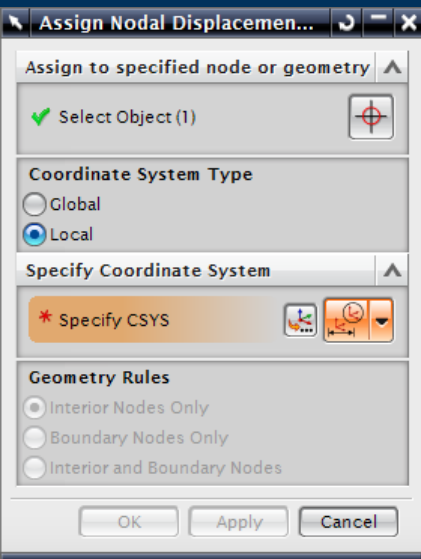
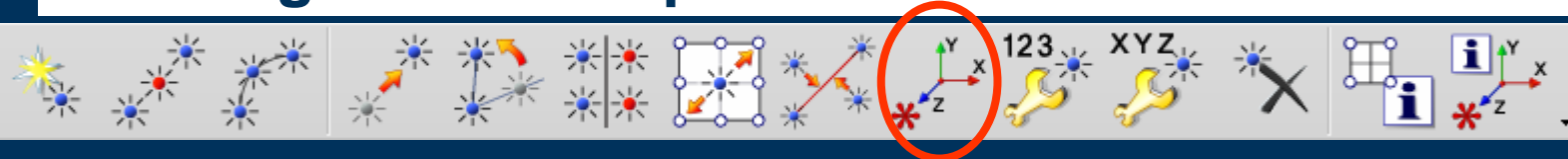
# Meshing – Node Align



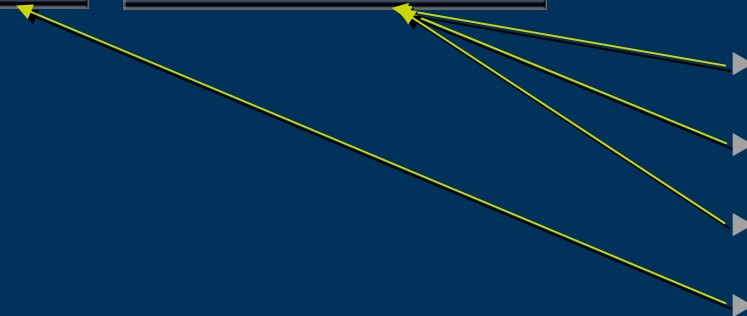
- ▶ Move selected Nodes onto Vector between 2 Nodes
- ▶ Multiple Selection Methods



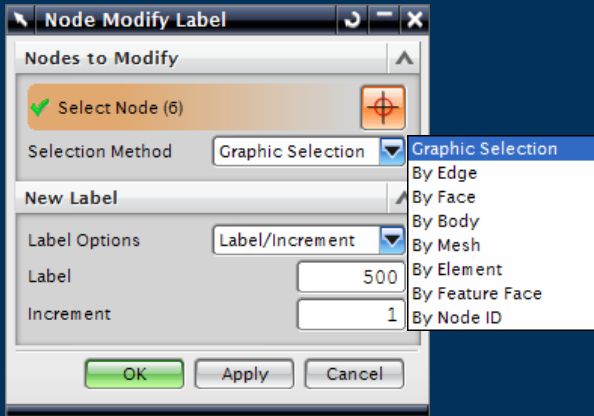
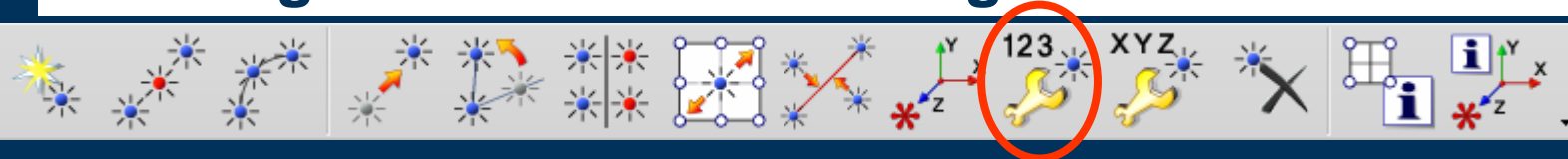
# Meshing – Node Displacement CSYS



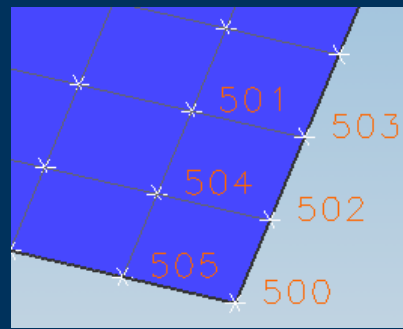
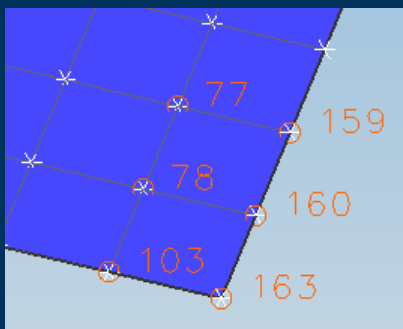
- ▶ Assign Nodal Displacement Coordinate System
  - ▶ Cartesian
  - ▶ Cylindrical
  - ▶ Spherical
  - ▶ Pre-Existing or Created on-the-fly
- ▶ Select Nodes by
  - ▶ Edge
  - ▶ Face
  - ▶ Body
  - ▶ Individual Selection



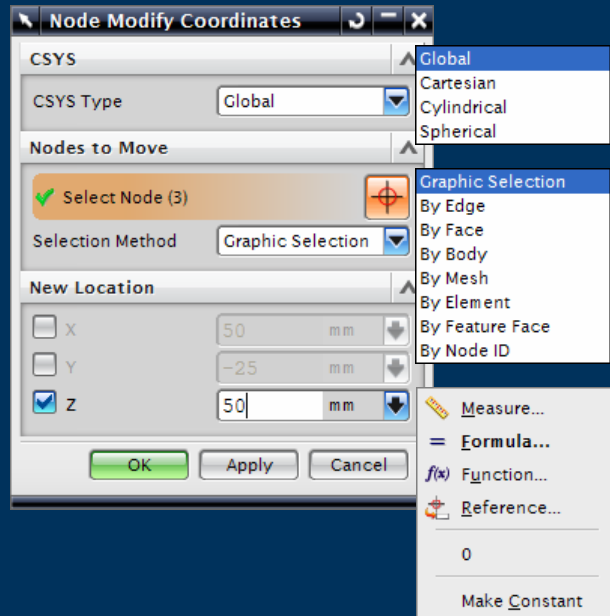
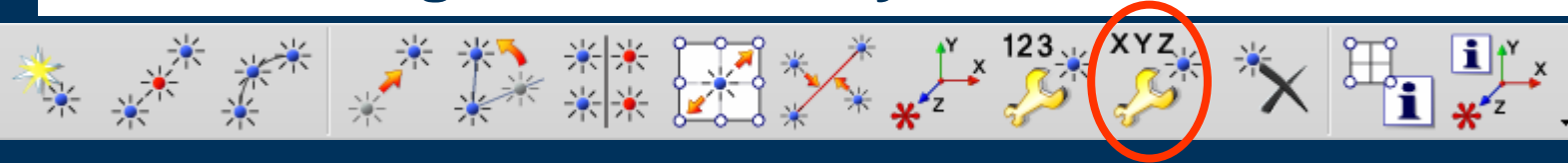
# Meshing – Node Re-Numbering



- ▶ Modify Node Numbering/Label

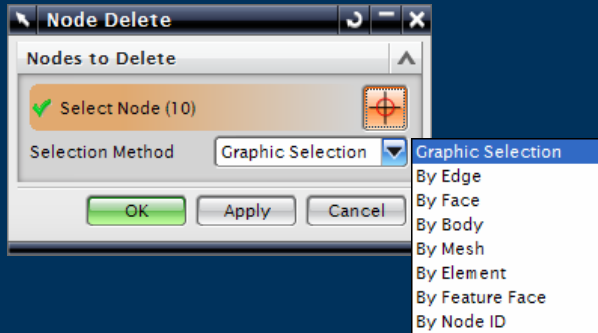
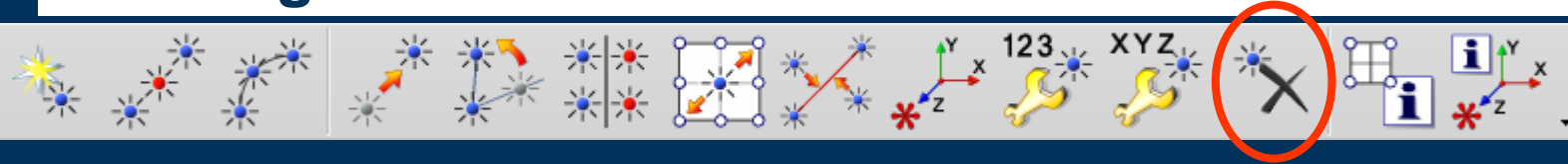


# Meshing – Node Modify Coordinates



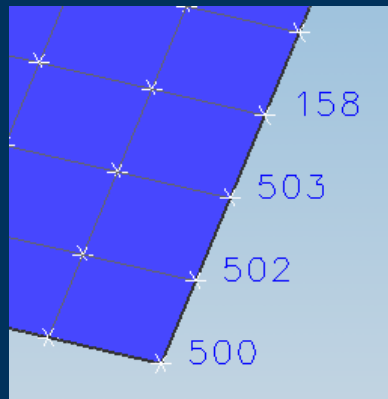
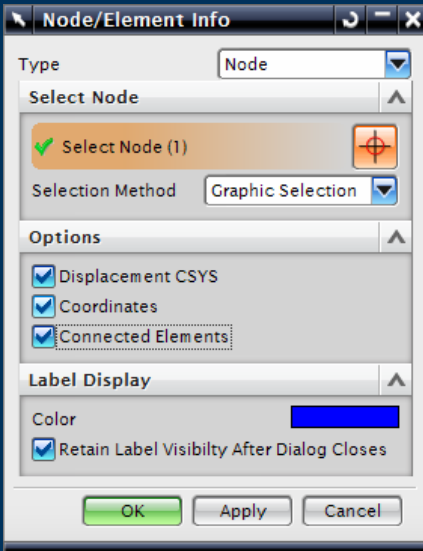
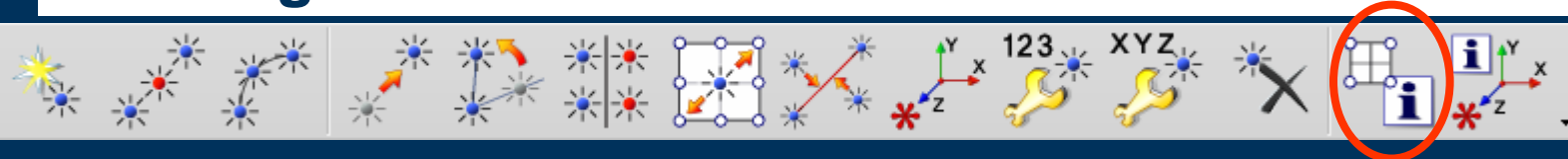
- ▶ Modify the Coordinate(s) of selected Nodes
- ▶ Global or Selected CSYS
  - ▶ X, Y, Z
  - ▶ R, Theta, Z
  - ▶ R, Theta, Phi

# Meshing – Node Deletion

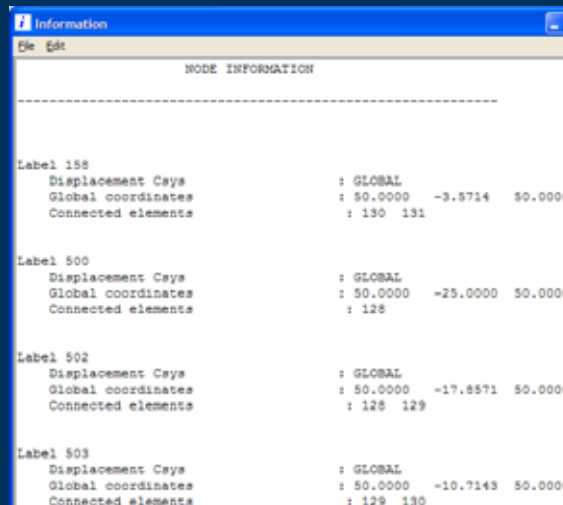


- ▶ Delete Nodes
- ▶ Only Nodes not attached to Elements will be deleted

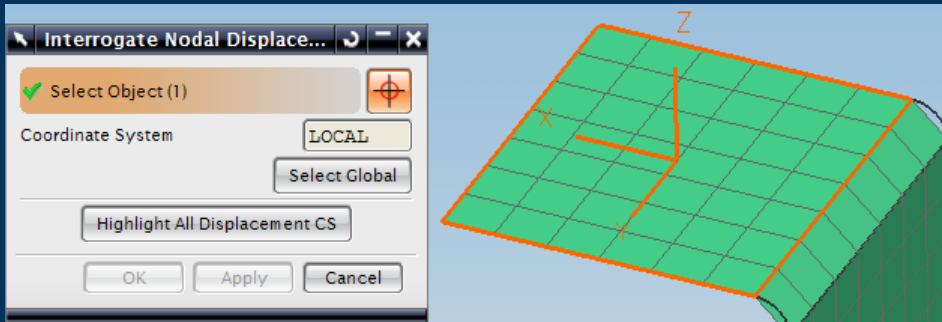
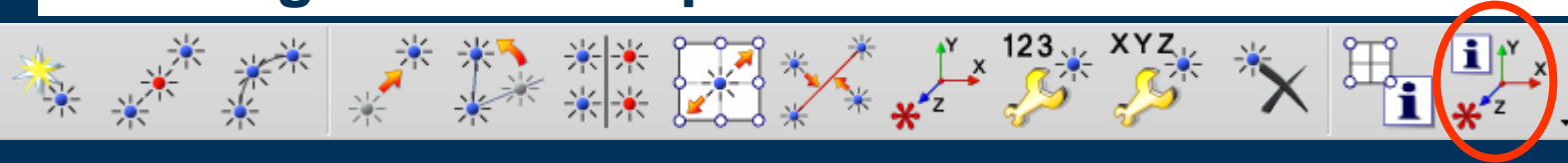
# Meshing – Node & Element Information



- ▶ Node Information
  - ▶ Displacement CSYS
  - ▶ Coordinates
  - ▶ Connected Elements



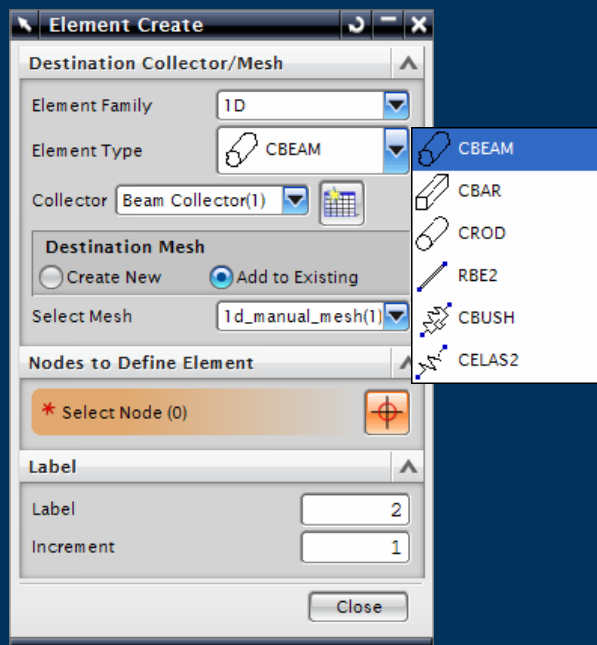
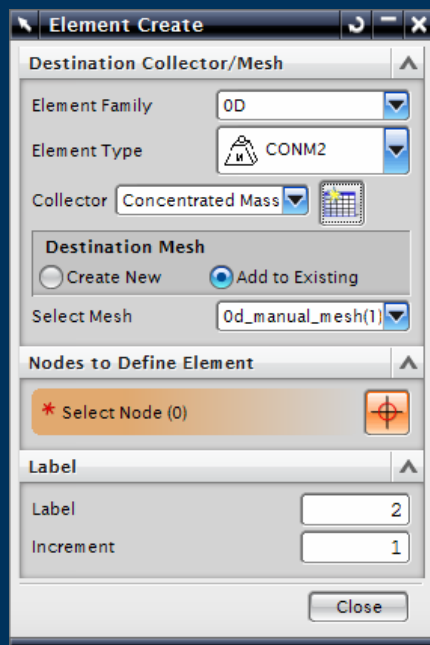
# Meshing – Nodal Displacement CSYS



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

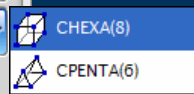
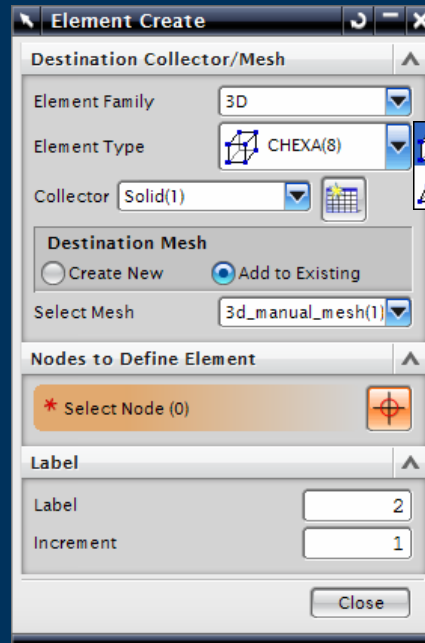
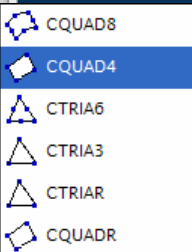
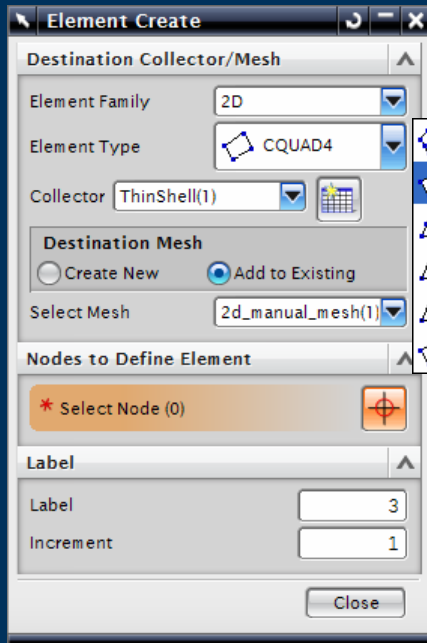


# Meshing – Element Create



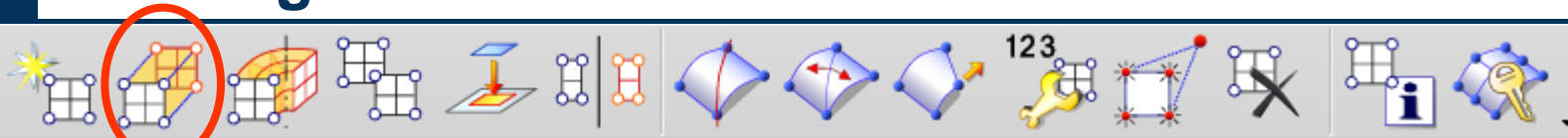
- ▶ Element Creation attached to existing Nodes
- ▶ Mesh Collector selection or Creation on-the-fly
  - ▶ New Mesh
  - ▶ Add to Existing Mesh

# Meshing – Element Create



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

# Meshing – Element Extrude



**Element Extrude**

Elements to Extrude

Selection Mode: Edges

Selection Method: Graphic Selection

- By Edge
- By Mesh
- By Element ID

✓ Select Elements (1)

Number of Copies

Number of Copies: 1

Extrude Options

Method: Along Vector

- Along Vector
- Along Path
- Project to Surface

\* Specify Vector (1)

Reverse Direction

Distance

Per Copy  Total

Distance: 25 mm

Twist Angle

Specify Point (0)

Angle: 0 deg

Destination Collector/Mesh

Element Type: CQUAD4

Mesh Collector: ThinShell(3)

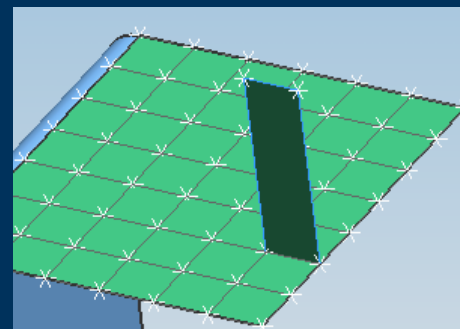
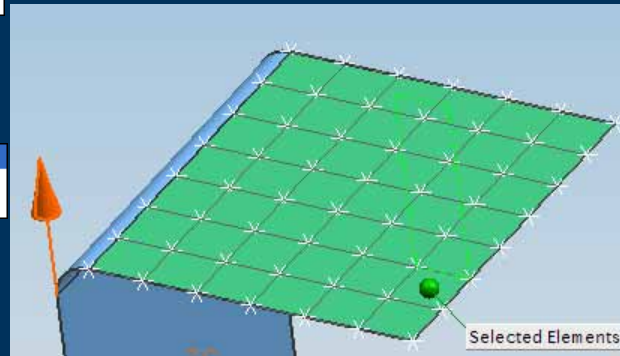
Destination Mesh

Create New  Add to Existing

Label

Preview

OK Apply Cancel

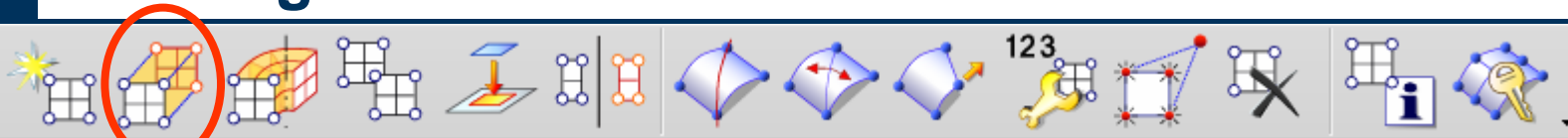


- ▶ Extrude an Existing Element(s) Edge
- ▶ Mesh Collector selection or Creation on-the-fly
  - ▶ New Mesh
  - ▶ Add to Existing Mesh

**Simulation Navigator**

Name	
bracket_fem1	
bracket_fem1_i	
bracket.prt	
Polygon Geometry	<input checked="" type="checkbox"/>
2D Collectors	<input checked="" type="checkbox"/>
ThinShell(1)	<input checked="" type="checkbox"/>
2d_mesh(1)	<input checked="" type="checkbox"/>
2d_extruded_mesh(1)	<input checked="" type="checkbox"/>

# Meshing – Element Extrude



**Element Extrude**

Elements to Extrude

Selection Mode: Faces

Selection Method: Graphic Selection

Select Elements (1)

Number of Copies: 1

Extrude Options

Method: Along Vector

Specify Vector (1)

Reverse Direction

Distance: Total, 25 mm

Twist Angle: 0 deg

Destination Collector/Mesh

Element Type: CHEXA(8)

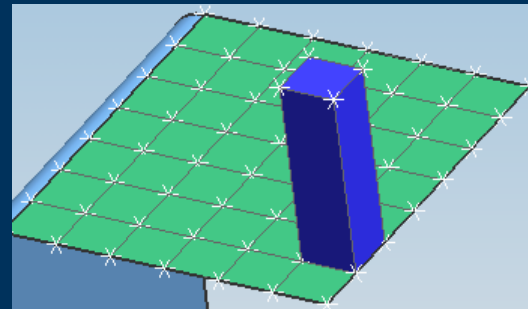
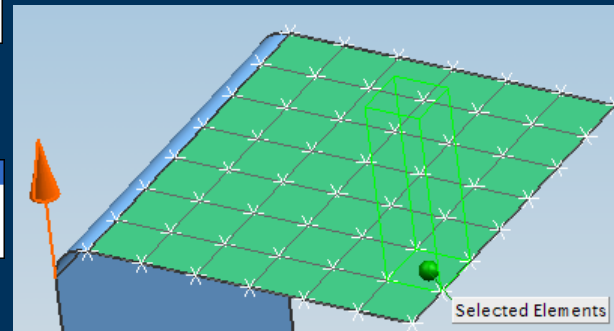
Mesh Collector: Solid(1)

Destination Mesh: Add to Existing

Label

Preview

OK Apply Cancel



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

**Simulation Navigator**

Name	
bracket_fem1	
bracket_fem1_i	
bracket.prt	
Polygon Geometry	<input checked="" type="checkbox"/>
3D Collectors	<input checked="" type="checkbox"/>
Solid(1)	<input checked="" type="checkbox"/>
3d_extruded_mesh(1)	<input checked="" type="checkbox"/>
2D Collectors	<input checked="" type="checkbox"/>

# Meshing – Element Revolve



**Element Revolve**

Elements to Revolve

Selection Mode: Edges

Selection Method: Graphic Selection

- By Edge
- By Mesh
- By Element ID

✓ Select Elements (1)

Number of Copies: 8

Revolve Options

Select Axis

- ✓ Specify Vector (1)
- Reverse Direction
- ✓ Specify Point (1)

Revolution Angle

Per Copy / Total

Angle: 45 deg

Destination Collector/Mesh

Element Type: CQUAD4

Mesh Collector: ThinShell(1)

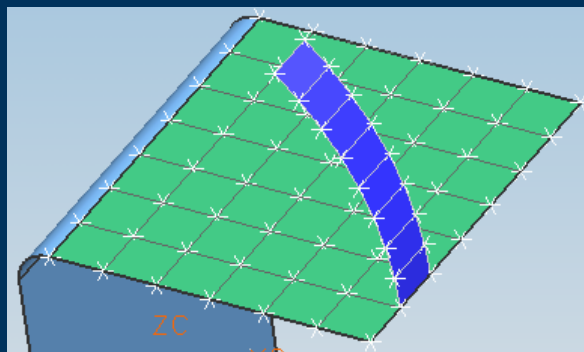
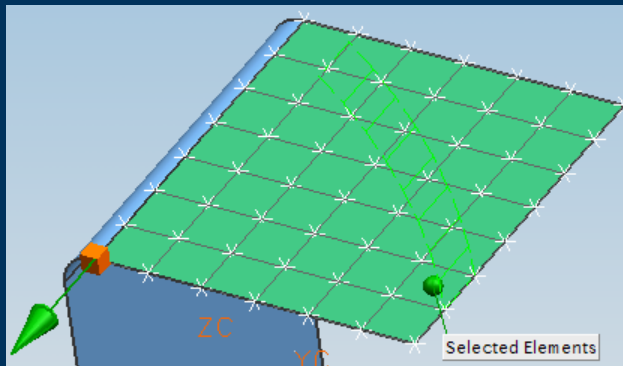
Destination Mesh

Create New / Add to Existing

Label

Preview

OK Apply Cancel



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

**Simulation Navigator**

Name

- bracket\_fem1
  - bracket\_fem1\_i
    - bracket.prt
  - ✓ Polygon Geometry
  - 2D Collectors
    - ✓ ThinShell(1)
      - ✓ 2d\_mesh(1)
      - ✓ 2d\_revolved\_mesh(1)

# Meshing – Element Revolve



**Element Revolve**

Elements to Revolve

Selection Mode: Faces

Selection Method: Graphic Selection

✓ Select Elements (1)

Number of Copies: 8

Revolve Options

Select Axis

✓ Specify Vector (1)

Reverse Direction

✓ Specify Point (1)

Revolution Angle

Per Copy  Total

Angle: 45 deg

Destination Collector/Mesh

Element Type: CHEXA(8)

Mesh Collector: Solid(1)

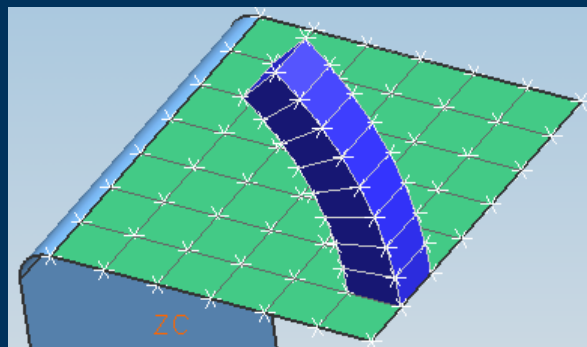
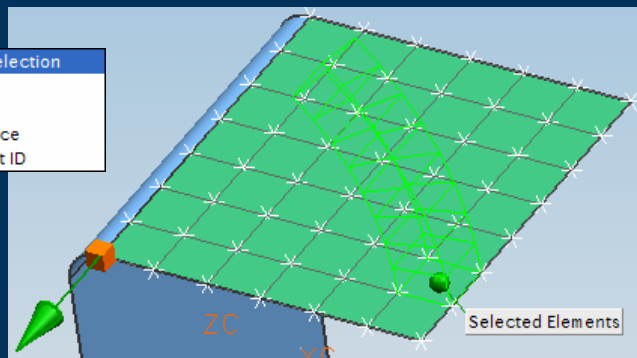
Destination Mesh

Create New  Add to Existing

Label

Preview

OK Apply Cancel

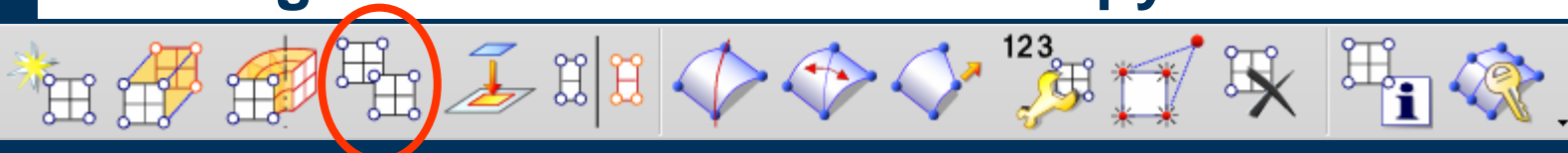


- ▶ Revolve an Existing Element(s) Face
- ▶ Mesh Collector selection or Creation on-the-fly
  - ▶ New Mesh
  - ▶ Add to Existing Mesh

**Simulation Navigator**

Name	
bracket_fem1	
bracket_fem1_i	
bracket.prt	
Polygon Geometry	<input checked="" type="checkbox"/>
3D Collectors	<input checked="" type="checkbox"/>
Solid(1)	<input checked="" type="checkbox"/>
3d_revolved_mesh(1)	<input checked="" type="checkbox"/>
2D Collectors	<input checked="" type="checkbox"/>
ThinShell(1)	<input checked="" type="checkbox"/>
2d_mesh(1)	<input checked="" type="checkbox"/>

# Meshing – Element Translate & Copy



**Element Copy and Translate**

Elements to Translate: Edges, Faces, Solids

Selection Mode: Any

Selection Method: Graphic Selection

Select Elements (1)

Number of Copies: Specify Number: 3

Translation Options:
 

- By Distance  Along Vector
- CSYS Type: Global
- Distance:  Per Copy  Total
- DX: 0 mm
- DY: 0 mm
- DZ: -20 mm

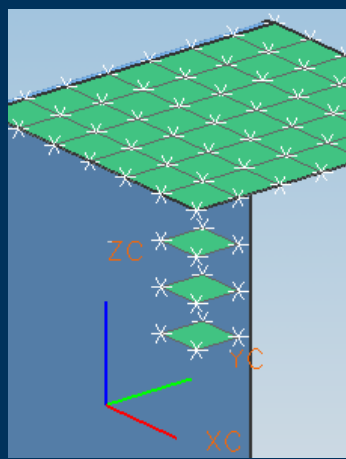
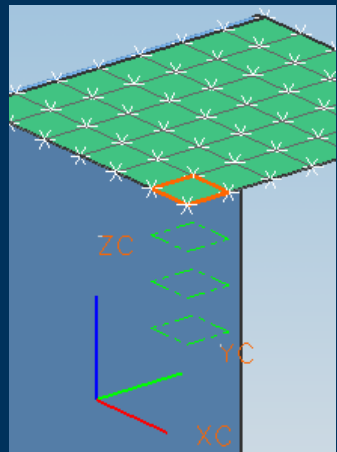
Destination Collector/Mesh:
 

- Element Type: No Selection
- Mesh Collector: No Selection
- Destination Mesh:  Create New  Add to Existing

Specify Label

Preview

OK Apply Cancel



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

**Simulation Navigator**

Name

- bracket\_fem1
  - bracket\_fem1.i prt
  - ✓ Polygon Geometry
  - ✓ 3D Collectors
  - ✓ 2D Collectors
    - ✓ ThinShell(1)
      - ✓ 2d\_mesh(1)
      - ✓ 2d\_translated\_mesh(2)

# Meshing – Element Copy & Project



**Element Copy and Project**

Elements to Project: Any

Selection Mode: Any

Selection Method: Graphic Selection

Select Elements (8)

Target Projection Face: Select Target Face (1)

% Offset to Surface: 50.0000

Define Projection Direction: Specify Vector (1)

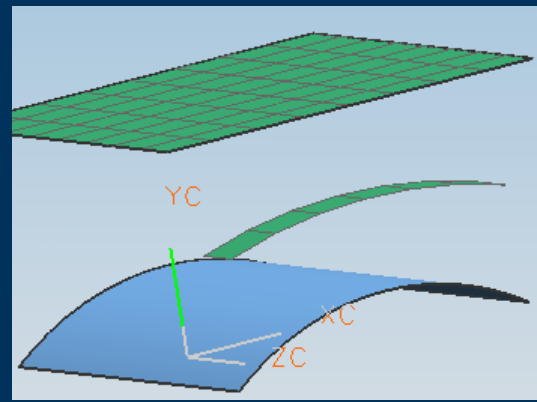
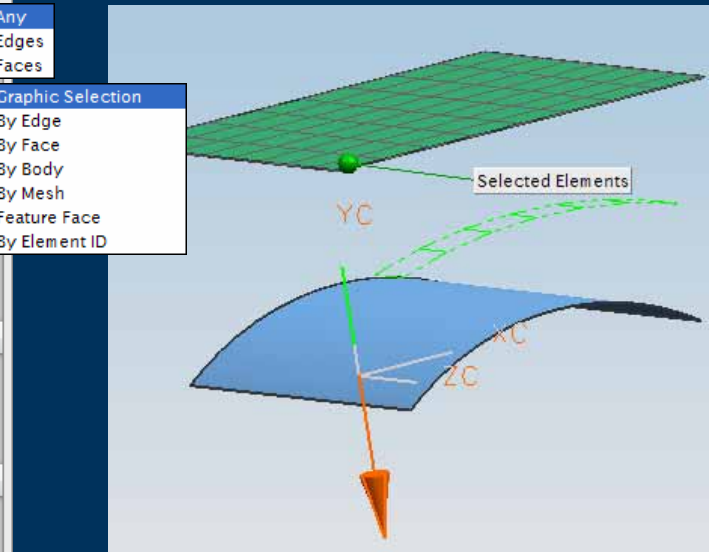
Reverse Direction

Destination Collector/Mesh: Element Type: No Selection, Mesh Collector: No Selection

Destination Mesh:  Create New,  Add to Existing

Label, Preview

OK Apply Cancel



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

**Simulation Navigator**

Name
model1_fem1
model1_fem1_i
model1.prt
Polygon Geometry
2D Collectors
ThinShell(1)
2d_mesh(1)
2d_projected_mesh(1)



# Meshing – Element Copy & Project



**Element Copy and Project**

Elements to Project

Selection Mode: Faces

Selection Method: Graphic Selection

Select Elements (8)

Target Projection Face

Select Target Face (1)

% Offset to Surface: 50.0000

Define Projection Direction

Along Vector (selected) / Element Normal

Specify Vector (1)

Reverse Direction

Destination Collector/Mesh

Element Type: CQUAD4

Mesh Collector: ThinShell(1)

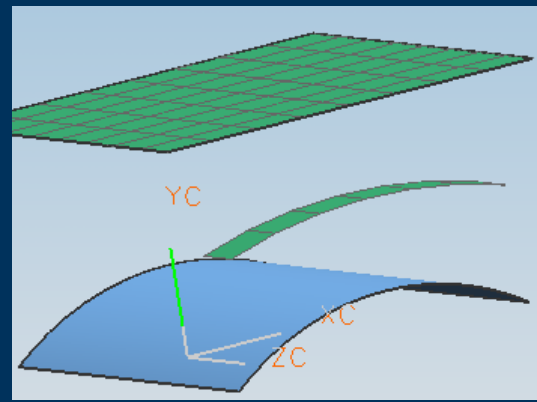
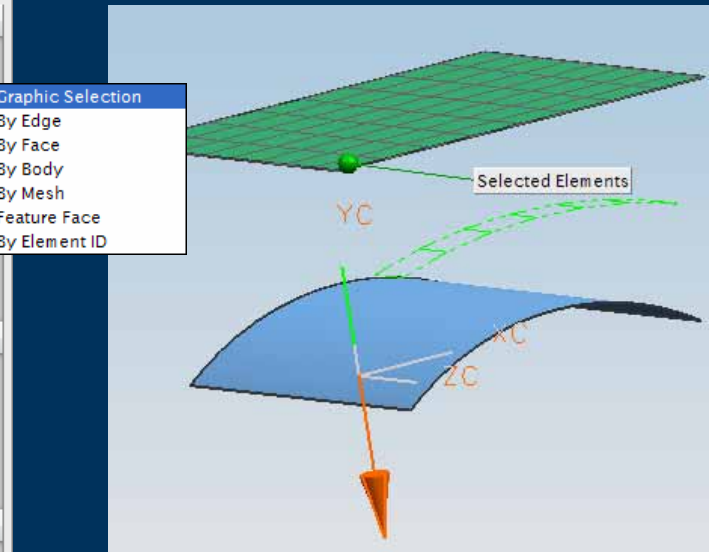
Destination Mesh

Create New (selected) / Add to Existing

Label

Preview

OK Apply Cancel



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

**Simulation Navigator**

Name	
model1_fem1	
model1_fem1_i	
model1.prt	
Polygon Geometry	<input checked="" type="checkbox"/>
2D Collectors	<input checked="" type="checkbox"/>
ThinShell(1)	<input checked="" type="checkbox"/>
2d_mesh(1)	<input checked="" type="checkbox"/>
2d_projected_mesh(1)	<input checked="" type="checkbox"/>

# Meshing – Element Copy & Reflect



**Element Copy and Reflect**

Elements to Reflect: Any

Selection Mode: Any

Selection Method: By Face

✓ Select Elements (56)

Reflection Plane: Specify Plane

Destination Collector/Mesh: Element Type: No Selection

Mesh Collector: No Selection

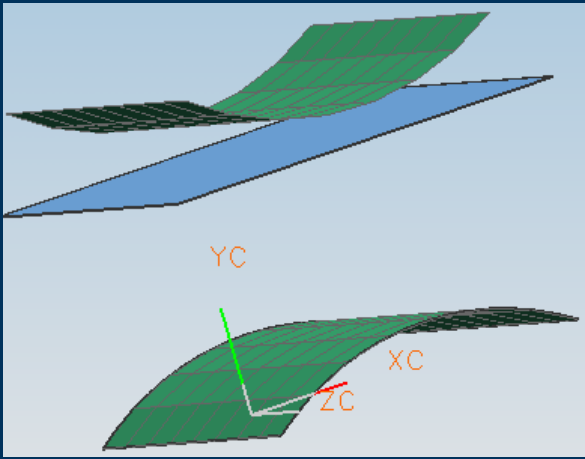
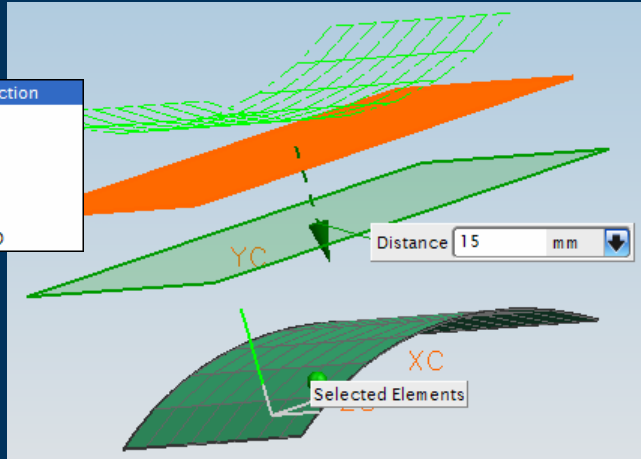
Destination Mesh:  Create New  Add to Existing

Label: Preview

OK Apply Cancel

Graphic Selection

- By Edge
- By Face
- By Body
- By Mesh
- Feature Face
- By Element ID

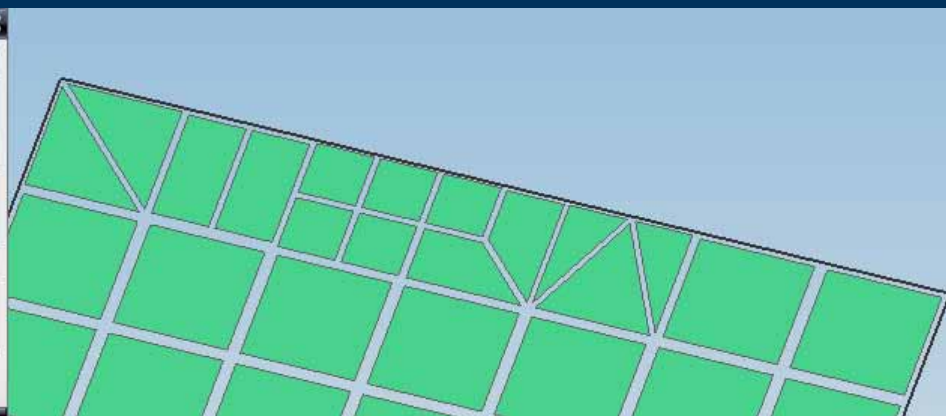
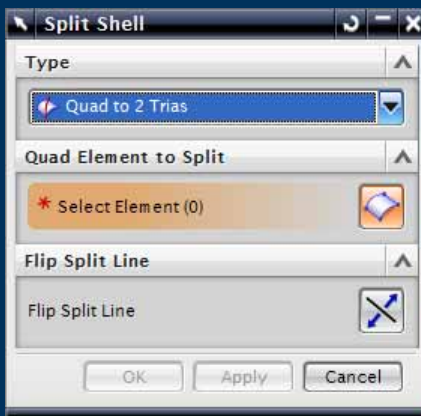
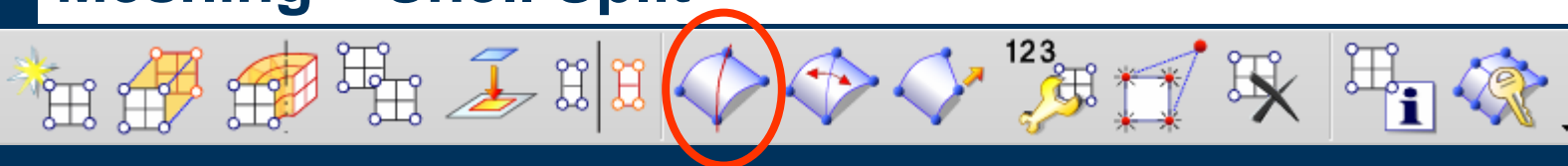


- ▶ Reflect & Copy Element(s) about a Plane
- ▶ Mesh Collector selection or Creation on-the-fly
  - ▶ New Mesh
  - ▶ Add to Existing Mesh

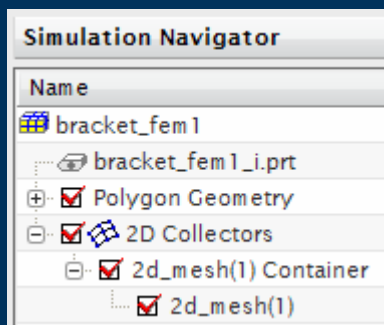
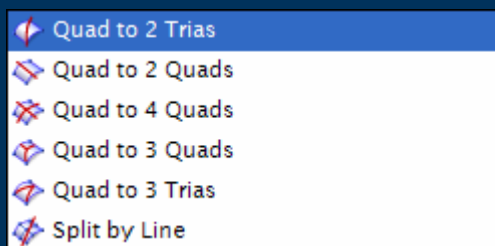
**Simulation Navigator**

Name	
model1_fem1	
model1_fem1_i	
Polygon Geometry	
2D Collectors	
ThinShell(1)	
ThinShell(2)	
2d_mesh(1)	
2d_reflected_mesh(1)	

# Meshing – Shell Split

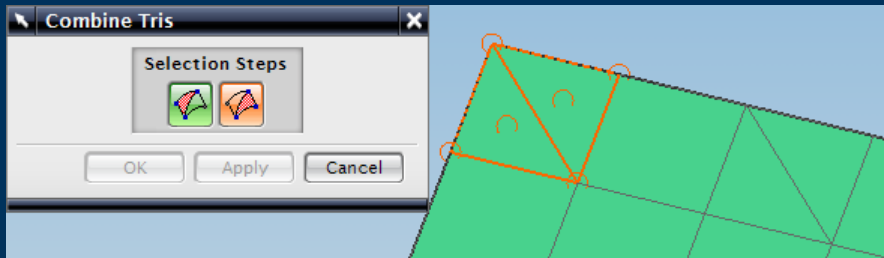
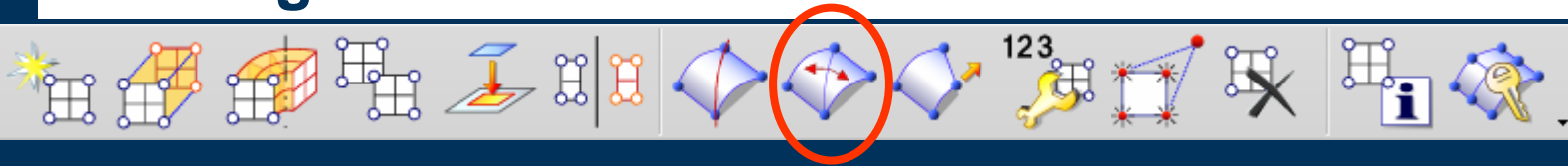


- ▶ Splits Quadrilateral Multiple Elements

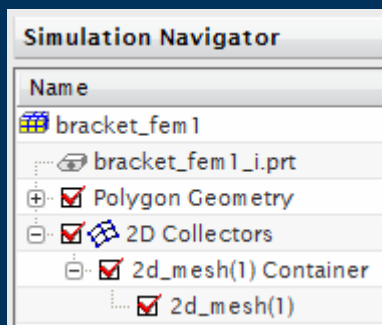


- ▶ Mesh Update will remove Manual changes
- ▶ New Elements remain in Mesh Collector

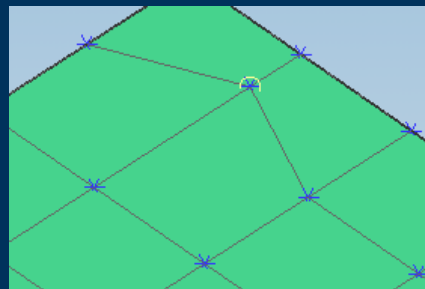
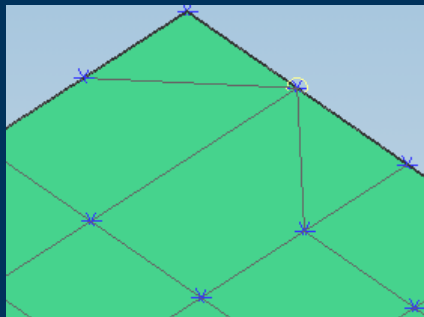
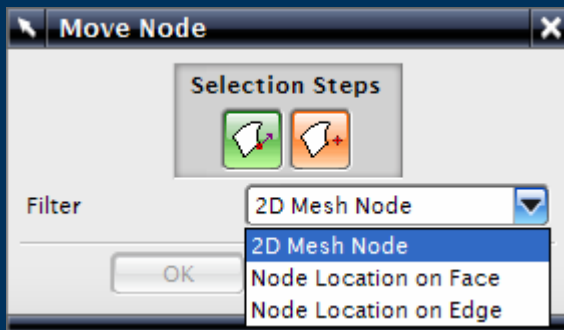
# Meshing – Combine Tris



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

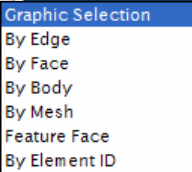
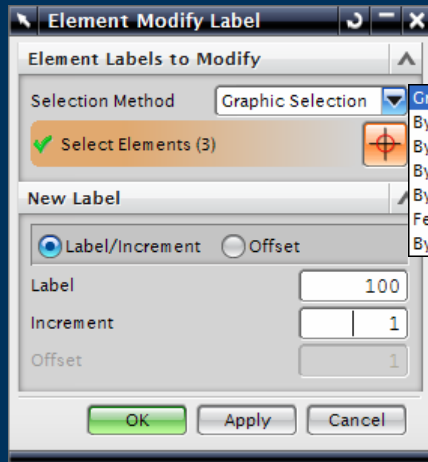
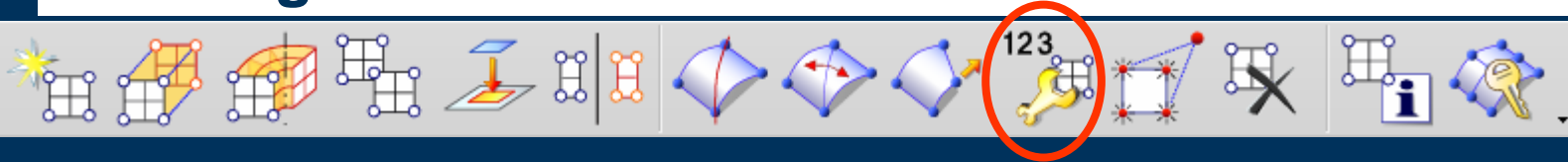


# Meshing – Move Node



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

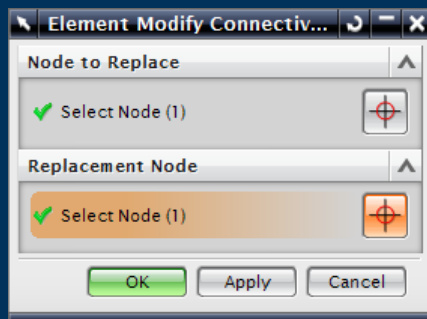
# Meshing – Element Re-Label



► Modify Element Numbering/Label



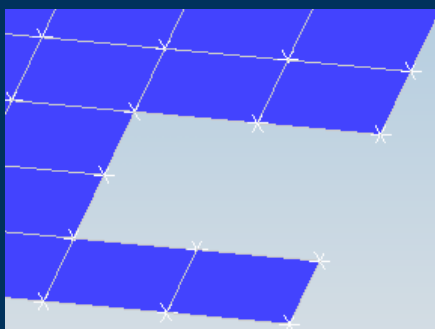
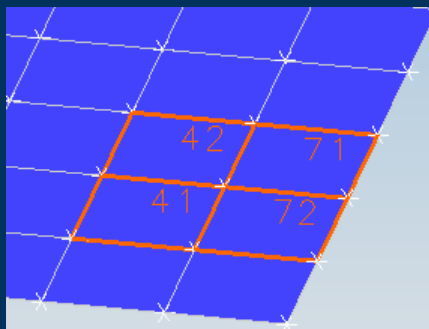
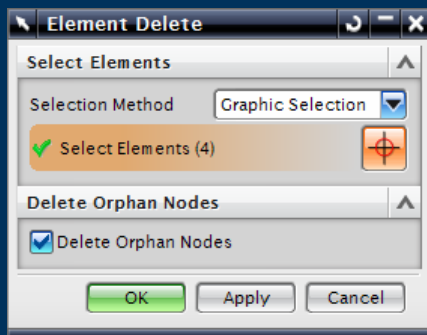
# Meshing – Element Connectivity



- ▶ Replace One Node with another Node
- ▶ Specific Mesh Connections



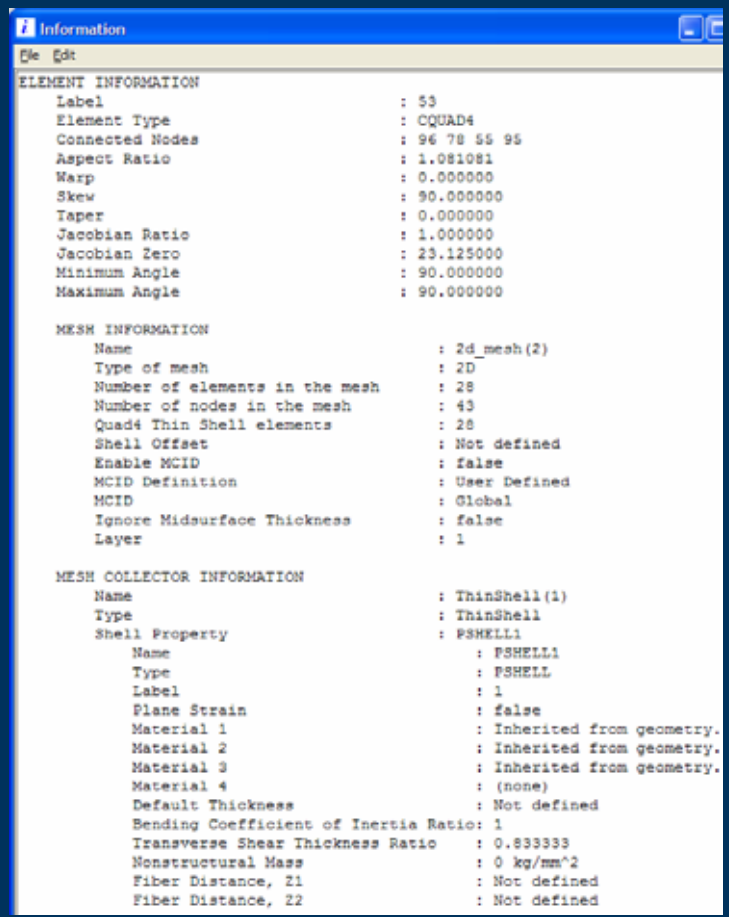
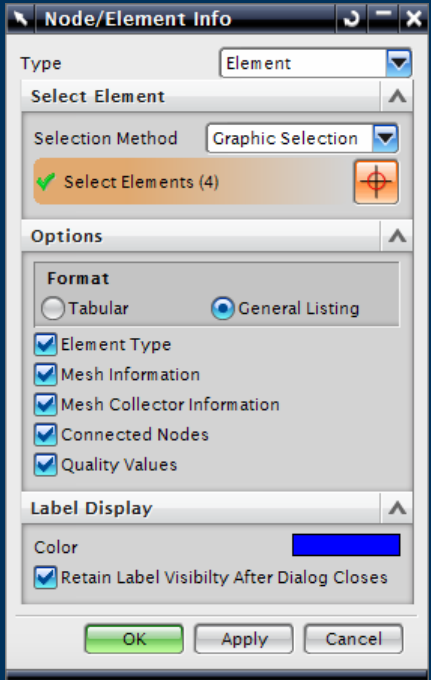
# Meshing – Element Deletion



- ▶ Delete Elements
- ▶ Optionally delete Orphaned Nodes



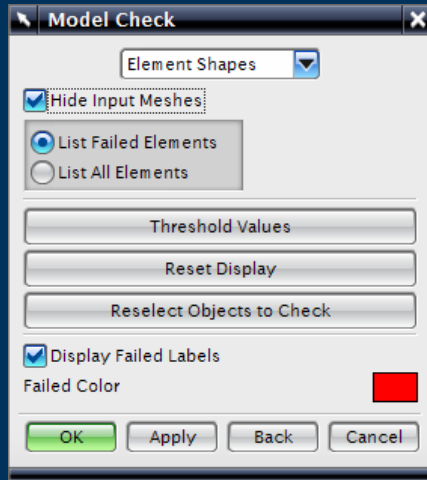
# Meshing – Node & Element Information



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



# 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

Elements	Number failed	Number checked
	0	10727

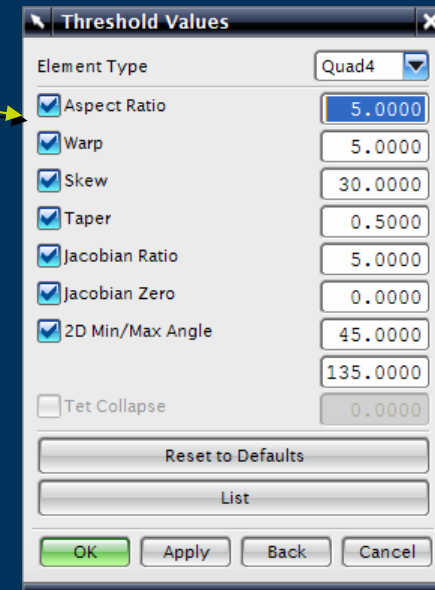
Check	Number failed	Worst value
Aspect Ratio	0	5.702181
Warp	0	-N/A-
Skew	0	-N/A-
Taper	0	-N/A-
Twist	0	-N/A-
Jacobian Ratio	0	7.206479
Jacobian Zero	0	0.437397
Minimum Angle	0	-N/A-
Maximum Angle	0	-N/A-
Tet Collapse	0	43.554548

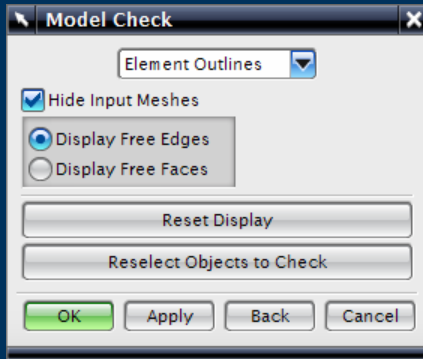
Threshold values

Shape	Aspect Ratio	Warp	Skew	Taper	Jacobian Ratio	Jacobian Zero	Minimum Angle	Maximum Angle	Tet Collapse
Tri3	5.000000	-N/A-	60.000000	-N/A-	5.000000	0.000000	20.000000	120.000000	-N/A-
Tri6	5.000000	-N/A-	60.000000	-N/A-	5.000000	0.000000	20.000000	120.000000	-N/A-
Quad4	5.000000	5.000000	30.000000	0.500000	5.000000	0.000000	45.000000	135.000000	-N/A-
Quad8	5.000000	5.000000	30.000000	0.500000	5.000000	0.000000	45.000000	135.000000	-N/A-
Tetra4	20.000000	-N/A-	-N/A-	-N/A-	10.000000	0.000000	-N/A-	120.000000	100.000000
Tetra10	20.000000	-N/A-	-N/A-	-N/A-	10.000000	0.000000	-N/A-	120.000000	100.000000
Hex8	20.000000	5.000000	100.000000	0.500000	30.000000	0.000000	45.000000	155.000000	-N/A-
Hex20	20.000000	5.000000	100.000000	0.500000	30.000000	0.000000	45.000000	155.000000	-N/A-
Wedge6	20.000000	5.000000	100.000000	0.500000	30.000000	0.000000	45.000000	155.000000	-N/A-
Wedge15	20.000000	5.000000	100.000000	0.500000	30.000000	0.000000	45.000000	155.000000	-N/A-

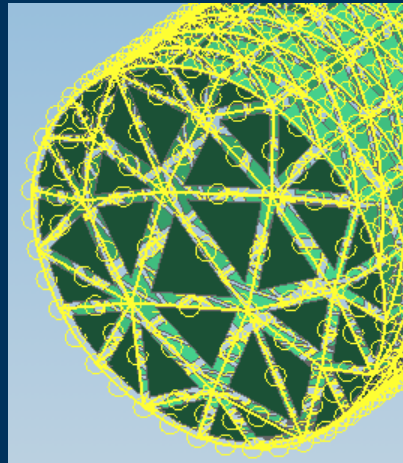
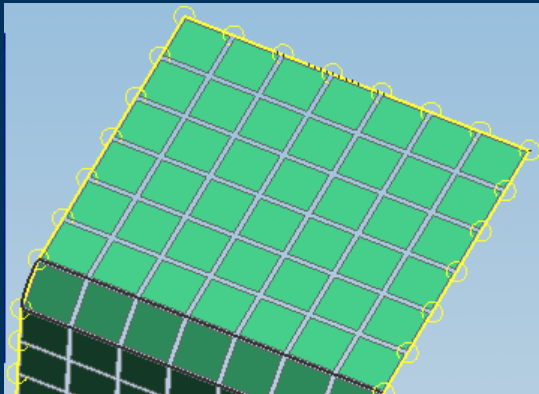
Threshold Values set in Customer Defaults



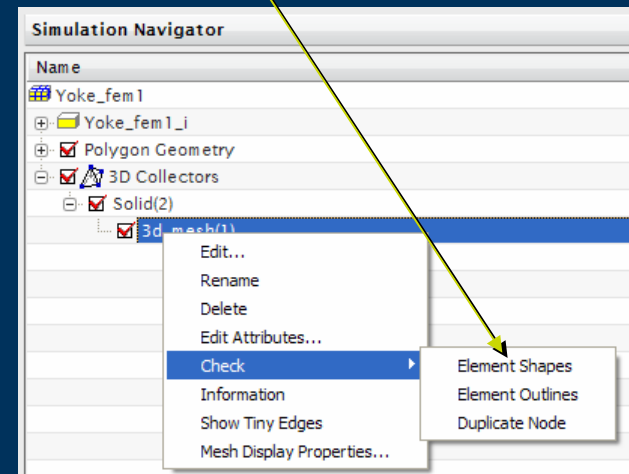
# Model Checking – Element Outlines



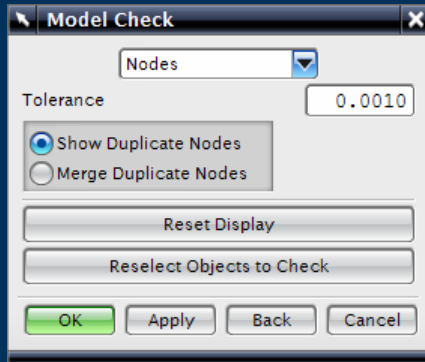
- ▶ Element Outlines show the Free Element Faces or Edges



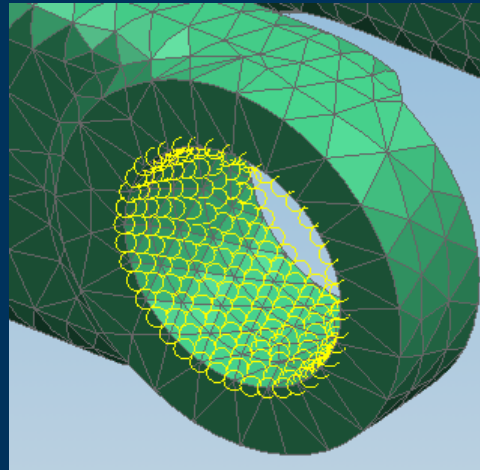
All Checks also available from the Navigator



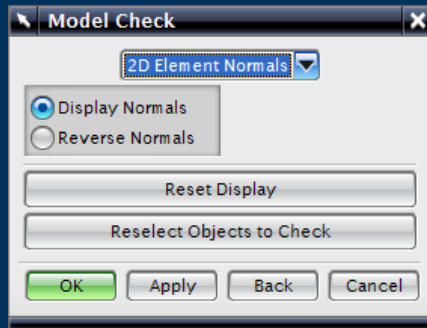
# Model Checking – Duplicate Nodes



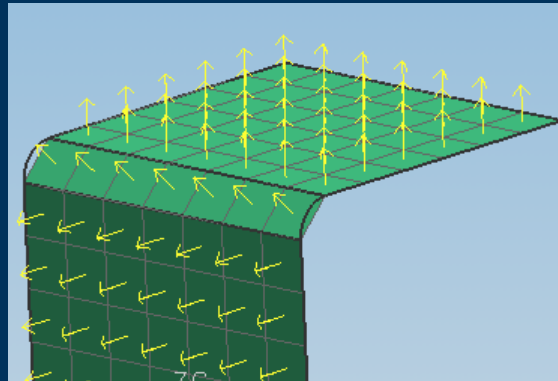
- ▶ Duplicate Nodes
  - ▶ Locate to check model
  - ▶ Merge to correct model



# Model Checking – Element Normals



- ▶ Displays the Element Normals



**SIEMENS**

# SIM Part – Pre-Processing

# Modeling Objects – Manager

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



Name	Lab...	Type
Contact Set Parameters 1	1	Con...
Strategy Parameters 1	2	Stra...
Structural Output Reque...	3	Stru...
Solution Parameters 1	4	Sol...
System Cells 1	5	Sys...

Referenced Modeling Objects



# Modeling Objects – Contact Set Parameters



**Contact Set Parameters**

Parameter	Value / Options
Name	Contact Set Parame
Label	1
Description	[Empty] <b>Constraint Function</b>
Contact Algorithm	Constraint Function <b>Segment Method</b>
Contact Surfaces	Single-Sided <b>Rigid Target</b>
Penetration Depth	0
Birth Time	0 sec <b>Single-Sided</b>
Death Time	0 sec <b>Double-Sided</b>
Initial Penetrations	Eliminated <b>Eliminated</b>
Continuous Segment Normal	Used for Single-Sid <b>Eliminated/Print Penetrating Nodes</b>
Type of Offset	Use Value for Single <b>Ignored</b>
Default Offset	0 mm <b>Overridden</b>
Displacement Formulation	Default <b>Used for Single-Sided Contact</b>
Time to Eliminate Initial Penetrations	0 sec <b>Used</b>
Consistent Contact Stiffness	Not Used <b>Use Value for Single-Sided Contact</b>
Contact Regions in Each Pair	Not Tied <b>Use Value for Single/Double-Sided Contact</b>
Extension Factor	0.001 <b>Half the Shell Thickness</b>
Friction Model Type	0
Delay Friction	No Delay <b>Default</b>
Parameter for Normal Constraint Function	1e-012 <b>Small Displacement</b>
Parameter for Frictional Constraint Function	0.001 <b>Large Displacement</b>
Compliance Factor	0 <b>Not Used</b>
	<b>Used</b>
	<b>No Delay</b>
	<b>Delay</b>

Buttons: OK, Apply, Cancel

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

# Modeling Objects – Strategy Parameters

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

Strategy Parameters dialog box, Analysis Control tab. Fields include: Name (Strategy Parameter), Label (2), Description, Multigrid Solver, Equilibrium, ATS Scheme, LDC Scheme, TLA Scheme, Contact, Restart, Other, Translation, Analysis Control, Analysis Options, Time Integration, Solver to Use (Direct Sparse), Automatic Incrementation Scheme (Not used), Positive Definite Matrix (Analysis may stop), Mass Type (Consistent).

Strategy Parameters dialog box, Analysis Options tab. Fields include: Name (Strategy Parameter), Label (2), Description, Multigrid Solver, Equilibrium, ATS Scheme, LDC Scheme, TLA Scheme, Contact, Restart, Other, Translation, Analysis Control, Analysis Options, Time Integration, Integration Order t-Direction (0), Use Incompatible Modes (CQUAD4) (Yes), Stiffness Matrix Stabilization (Not Used), Matrix Stabilization Factor (1e-012), Element Death Time Delay (0 sec), Shell DOF Angle (5 deg), Shell Drilling Stiffness Factor (0.0001), u/p Formulation (Not Used), Large Strain Formulation (CQUAD4) (ULJ (Rigid Target Al)), Prescribed Displacements Option (Original Configur), Prescribed Loads Option (Affected by Struct), Maximum Displacement Limit (0 mm).

Strategy Parameters dialog box, Time Integration tab. Fields include: Name (Strategy Parameter), Label (2), Description, Multigrid Solver, Equilibrium, ATS Scheme, LDC Scheme, TLA Scheme, Contact, Restart, Other, Translation, Analysis Control, Analysis Options, Time Integration, Mode of Execution (Restart Analysis), Solution Starting Time (0 sec), Frequency of Saving Analysis Results (0).

Strategy Parameters dialog box, Analysis Options tab. Fields include: Name (Strategy Parameter), Label (2), Description, Multigrid Solver, Equilibrium, ATS Scheme, LDC Scheme, TLA Scheme, Contact, Restart, Other, Translation, Analysis Control, Analysis Options, Time Integration, Time Integration Method (ADINA composite), Newmark Alpha coefficient (0.25), Newmark Delta coefficient (0.5).

Strategy Parameters dialog box, Analysis Options tab. Fields include: Name (Strategy Parameter), Label (2), Description, Analysis Control, Analysis Options, Time Integration, Multigrid Solver, Equilibrium, ATS Scheme, LDC Scheme, TLA Scheme, Contact, Restart, Other, Translation, Number of Sub-Groups (1), Stress-Strain Table Extension (Extended), Stress-Strain Table Conversion (No Conversion), Results Coordinate System (Element), Bolt Pre-Load Steps (1).

# Modeling Objects

## – Real Eigenvalue – Lanczos & Householder



Real Eigenvalue - Lanczos

Name: Real Eigenvalue -

Label: 6

Description:

Frequency Options

Frequency Range - Lower Limit: 10 Hz

Frequency Range - Upper Limit: 100 Hz

Number of Desired Modes: 10

Extraction Data

Diagnostic Level: 0

Number of Vectors: 7

Estimate of the First Natural Frequency: Hz

Method for Normalizing Eigenvectors: MASS

OK Cancel

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

Real Eigenvalue - Householder

Name: Real Eigenvalue -

Label: 6

Description:

Frequency Options

Frequency Range - Lower Limit: 10 Hz

Frequency Range - Upper Limit: 100 Hz

Number of Desired Modes: 10

Extraction Data

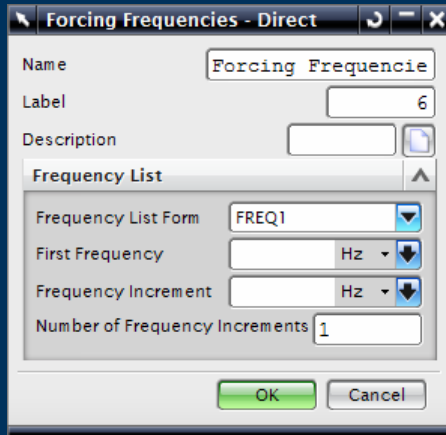
Method for Normalizing Eigenvectors: MASS

OK Cancel

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

# Modeling Objects

## – Forcing Frequencies – Direct & Modal



**Forcing Frequencies - Direct**

Name: Forcing\_Frequencie

Label: 6

Description:

Frequency List

Frequency List Form: FREQ1

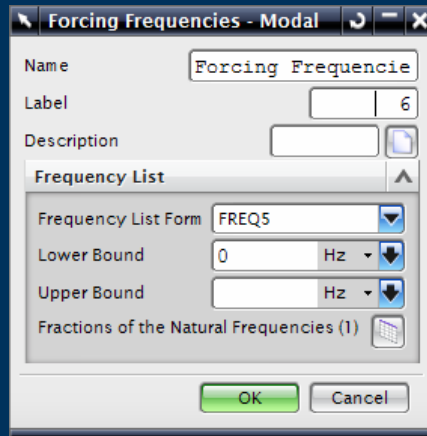
First Frequency: Hz

Frequency Increment: Hz

Number of Frequency Increments: 1

OK Cancel

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



**Forcing Frequencies - Modal**

Name: Forcing\_Frequencie

Label: 6

Description:

Frequency List

Frequency List Form: FREQ5

Lower Bound: 0 Hz

Upper Bound: Hz

Fractions of the Natural Frequencies (1)

OK Cancel

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

# Modeling Objects – Time Step



Time Step

Name: Time Step1

Label: 6

Description: [ ] [ ? ]

Time Step Interval: [ ^ ]

Number of Time Steps: 10

Time Increment: 1 sec [ v ]

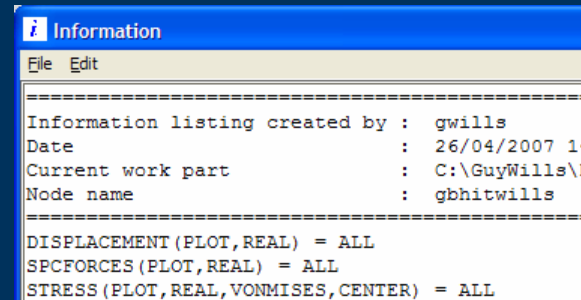
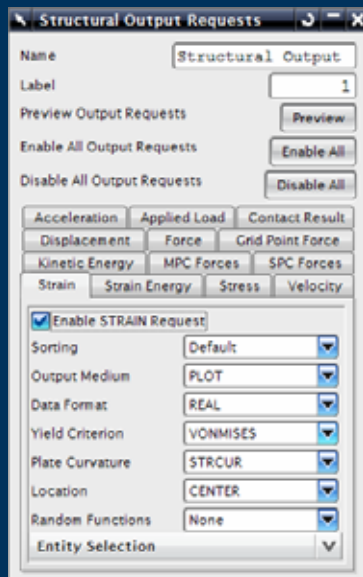
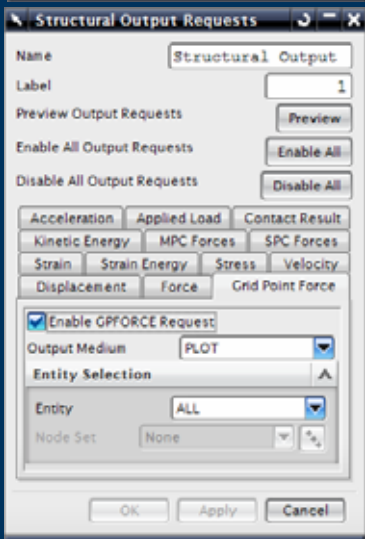
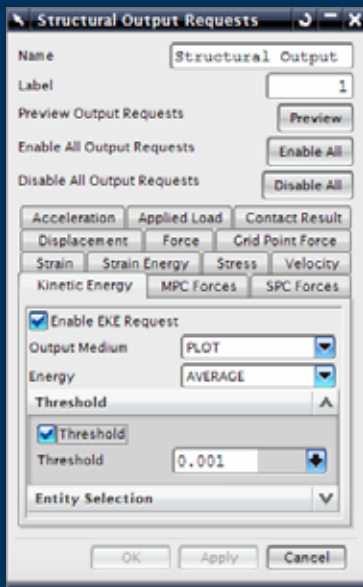
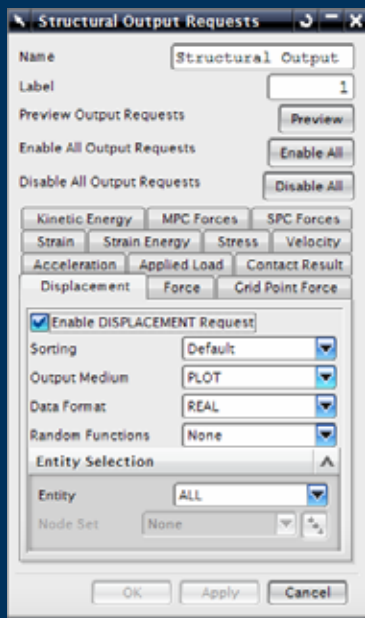
Skip Factor for Output: 1

OK Cancel

- ▶ 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

Parameter	Value
Name	Solution Parameter
Label	6
A-B	
C-D	
E-F	
G-H	
I-J	
K-L	
M-N	
O-P	
Q-R	
Q	0
RANREAL	0
RESLTOPT	8
RESVSALT	NO
RESVEC	NO
RESVINER	NO
RESVPGF	1e-006
RESVSE	NO
RESVSLI	YES
RESVSO	YES
RMSINT	LINEAR
RMXTRAN	NO
ROTCV	
ROTGPF	
RPOSTS1	0
RSPECTRA	-1
RSPRINT	0
S-T	
U-V	
W-Z	

Parameter	Value	Action
IFP		+
IFTM	0	+
INP4FMT	32	+
INREL	0	Remove
IRES	-1	+
ITAPE	-1	Add
IUNIT	11	+

**Parameter Descriptions**

Parameters are used extensively in the solution sequences for input of scalar values and for requesting special features. Parameters values are specified on PARAM Bulk Data entries or PARAM Case Control commands. For more information on the PARAM Bulk Data entry, see [PARAM](#). For more information on the PARAM Case Control command, see [PARAM](#). A complete alphabetical list of PARAMeter names and their functions is given in this section. [Table 7.2](#) and [Table 7.3](#) at the end of this section summarize parameter applicability in the structured and unstructured solution sequences, respectively.

**ACOUT** Default = PEAK  
 ACOUT specifies the type of output to be used with the FORCE Case Control command in coupled fluid-structural analysis (see "Performing a Coupled Fluid-Structural Analysis" in the NX Nastran User's Guide). ACOUT=RMS requests root-mean-square output.  
 To obtain sound pressure level in units of dB and dBA given by the FORCE command, a peak reference pressure must be specified with PARAM, PREFDB. The dB level is defined as:

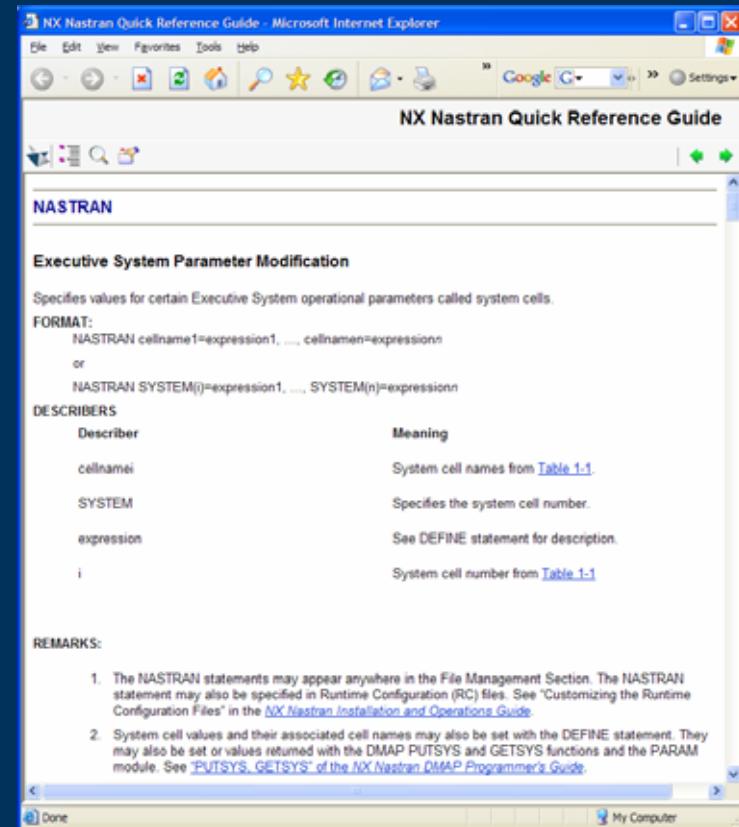
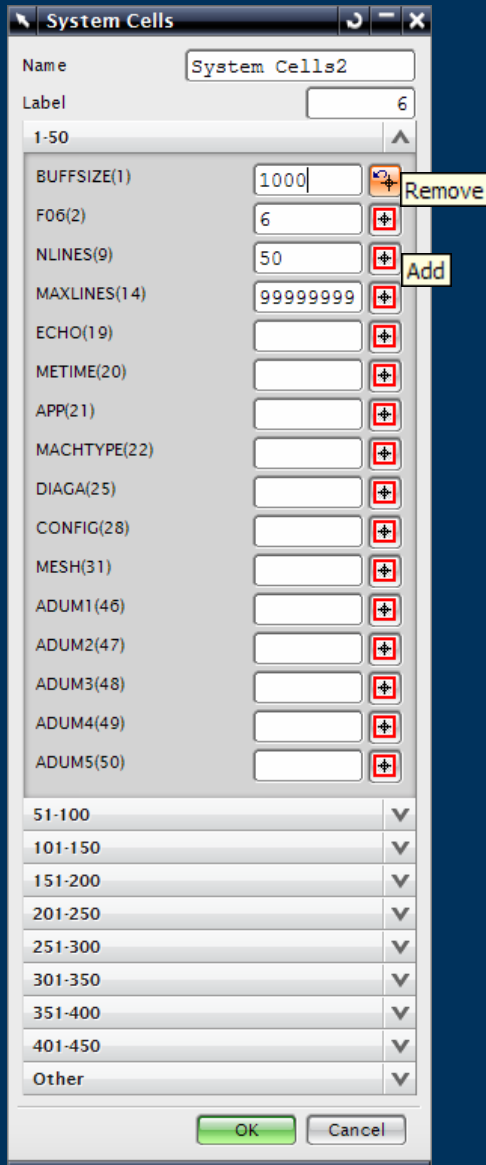
$$dB = 20 \cdot \log_{10} \left( \frac{P}{PREFDB} \right)$$

**ACSYM** Default = YES  
 By default, the dynamic equations for coupled fluid-structure analysis in frequency response are symmetrized for efficiency. PARAM.ACSYM.NO requests the pre-MSC Nastran Version 69 formulation which involves no symmetrization and will require more CPU time. See the "Formulation of Dynamic Equations in SubDMAP\_GMA" in the NX Nastran User's Guide.  
 If the iterative solver is selected (see the ITER=YES keyword on the NASTRAN statement) then the external work diagnostic will be different between ACSYM=YES and ACSYM=NO.

**ADPCON** Default = 1.0  
 Initial penalty values used in contact analysis are calculated automatically by

# Modeling Objects – System Cells

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

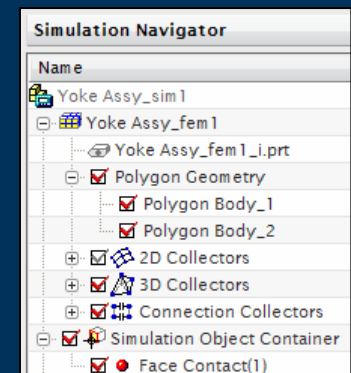
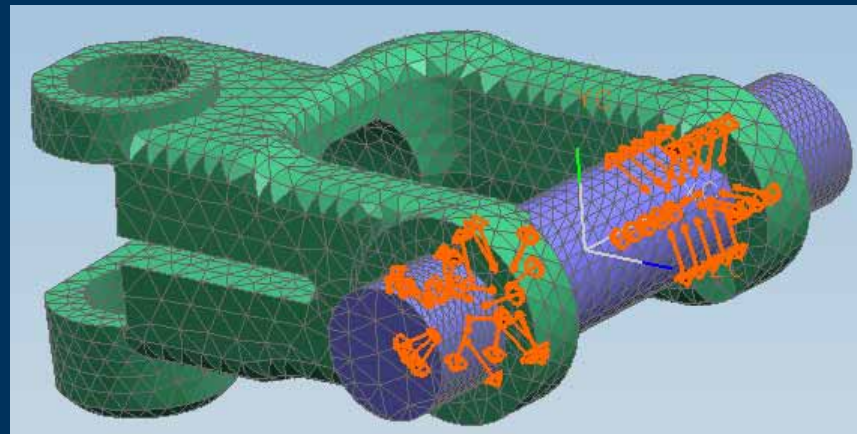
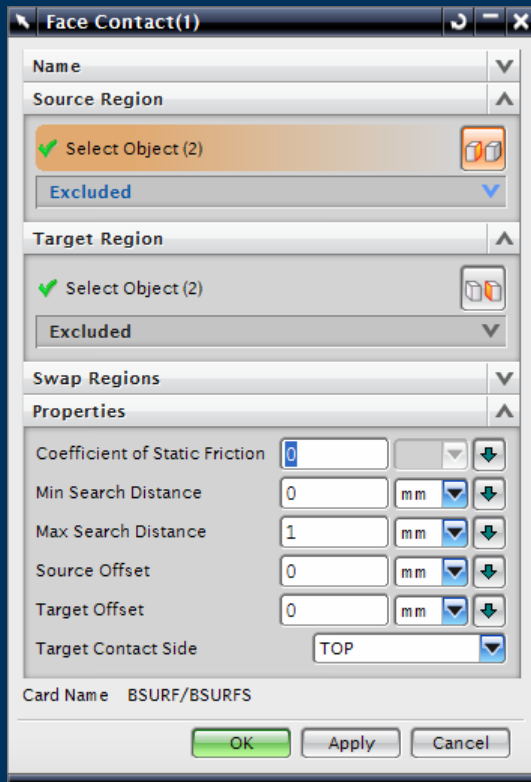




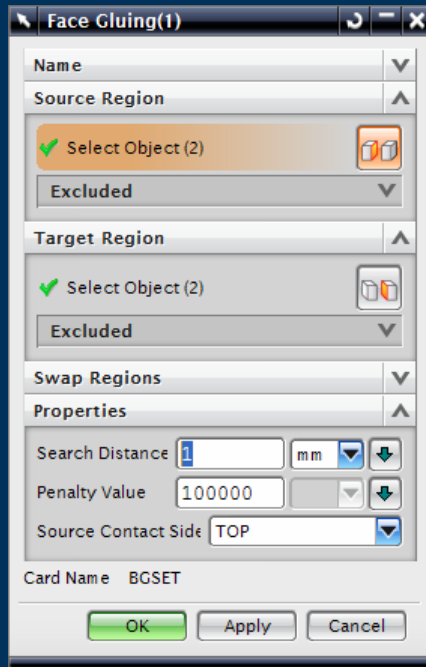
# Surface to Surface – Contact



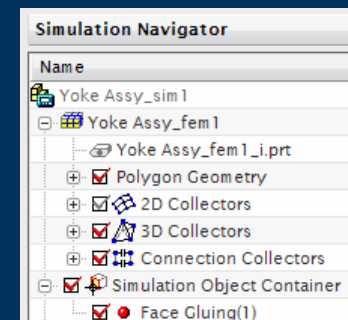
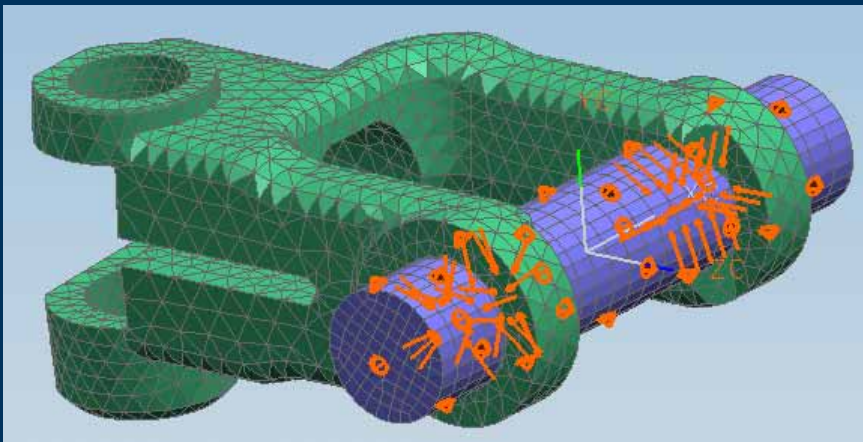
- ▶ Surface to Surface Contact options
  - ▶ Automatic Detection or Manual Selection
  - ▶ Coefficient of Friction
  - ▶ Search distances
  - ▶ Offsets



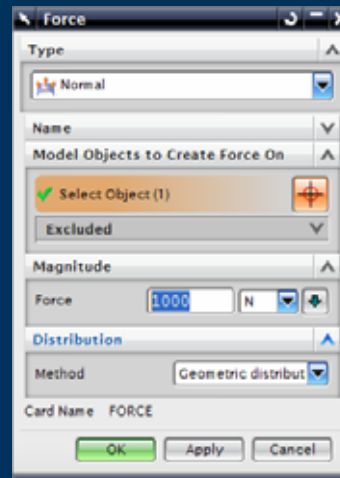
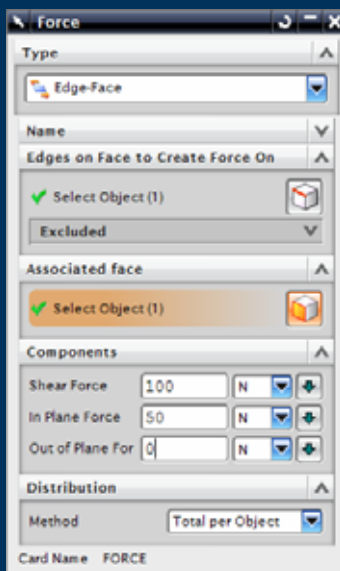
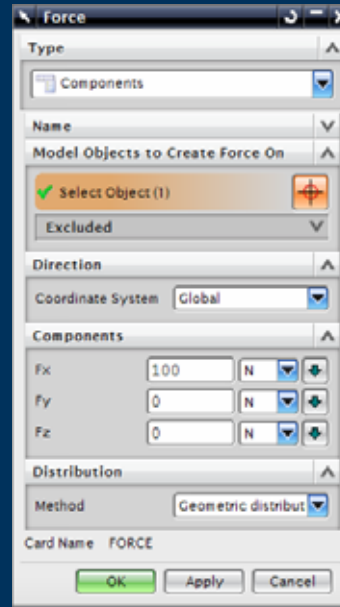
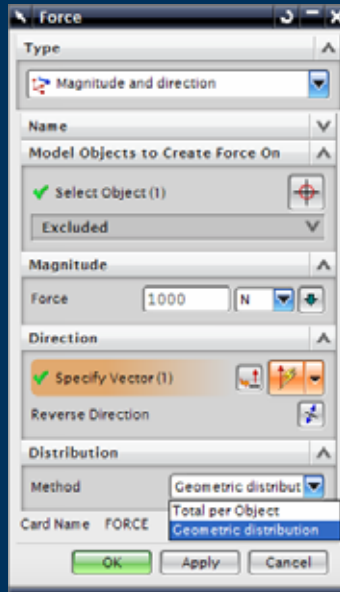
# Surface to Surface – Glue



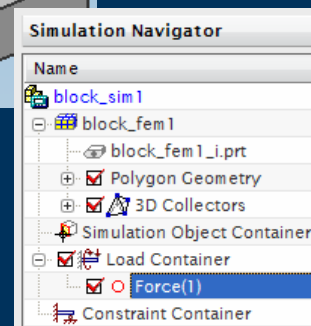
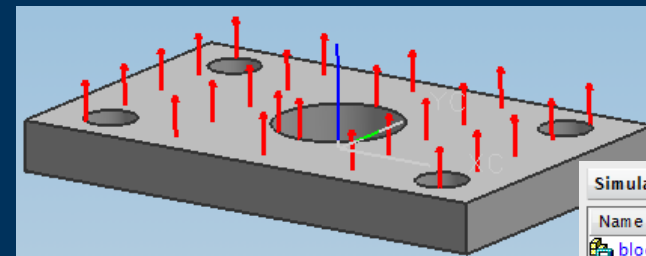
- ▶ Surface to Surface Glue options
  - ▶ Search distance
  - ▶ Penalty Value
- ▶ Does not require similar meshes
  - ▶ For example Tet to Hex
- ▶ Smooth transition of loads across boundaries



## Loads – Force

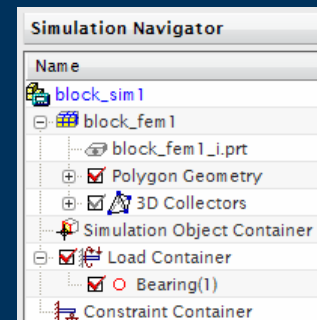
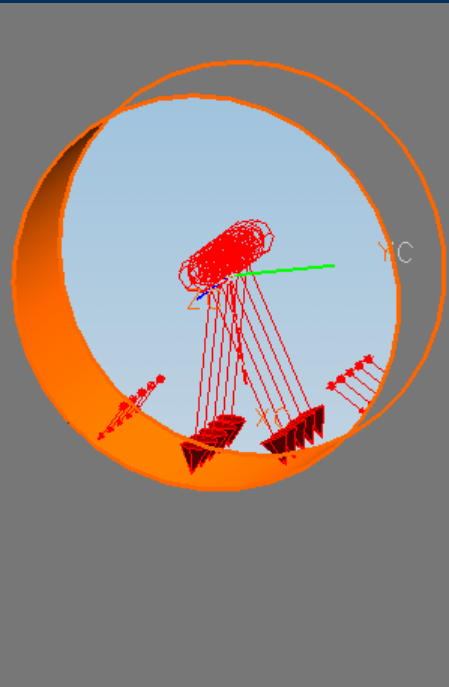
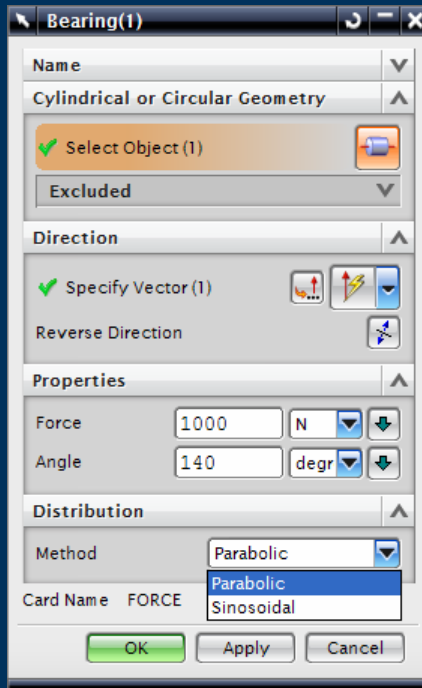


- ▶ Force Load Options
  - ▶ Magnitude and direction
  - ▶ Normal to selected faces
  - ▶ Fx, Fy, Fz Components relative to selected coordinate system
  - ▶ Shear, In/Out plane force
- ▶ Managed in the Load Container

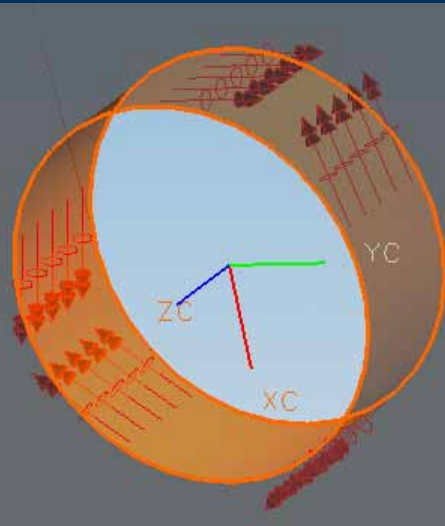
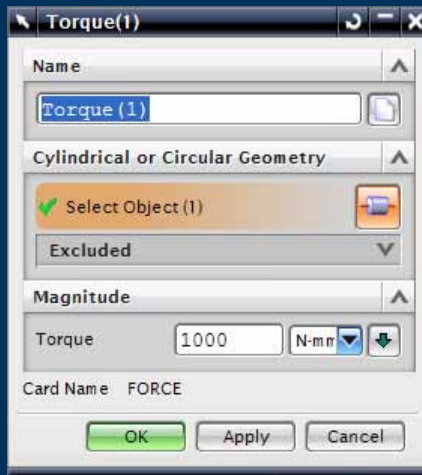
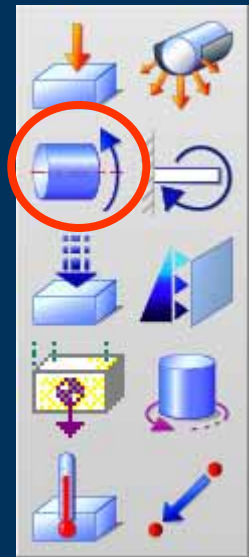


# Loads – Bearing

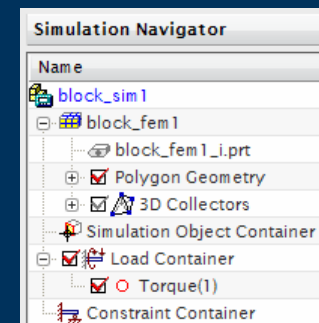
- ▶ Bearing Load
  - ▶ Distributed load across cylindrical curves or faces
  - ▶ Parabolic or Sinosoidal distribution
- ▶ Managed in the Load Container



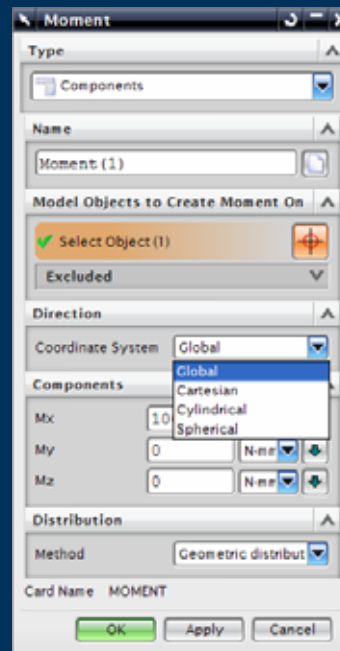
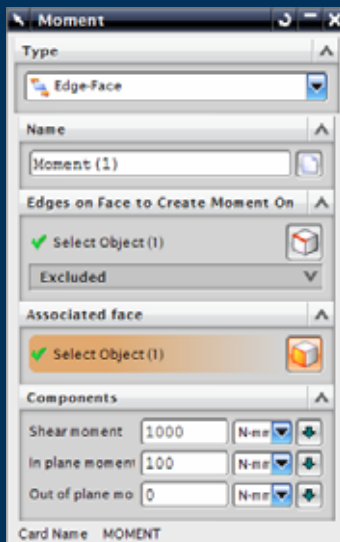
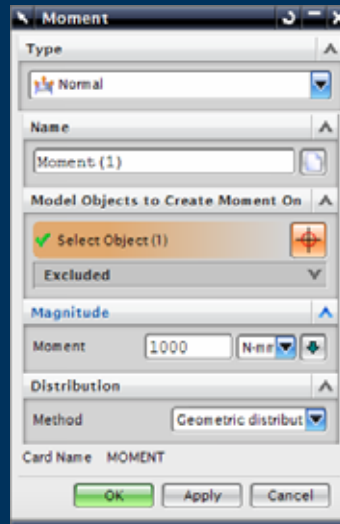
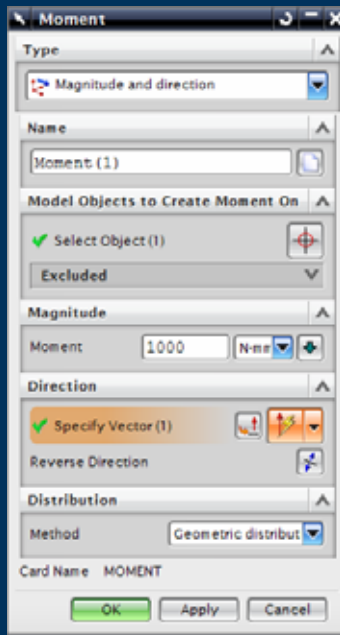
# Loads – Torque



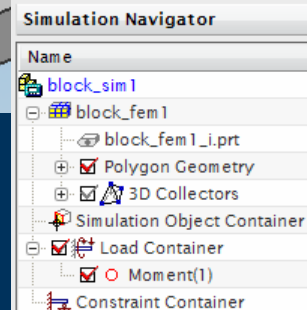
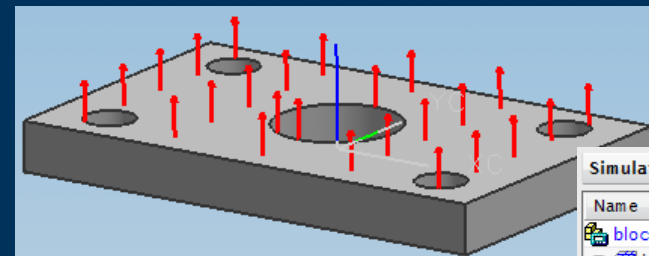
- ▶ Torque Load
  - ▶ Distributed load across cylindrical curves or faces
- ▶ Managed in the Load Container



# Loads – Moment

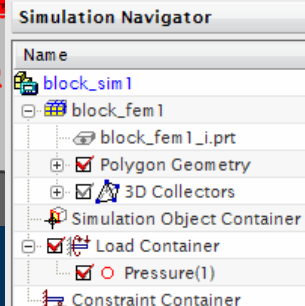
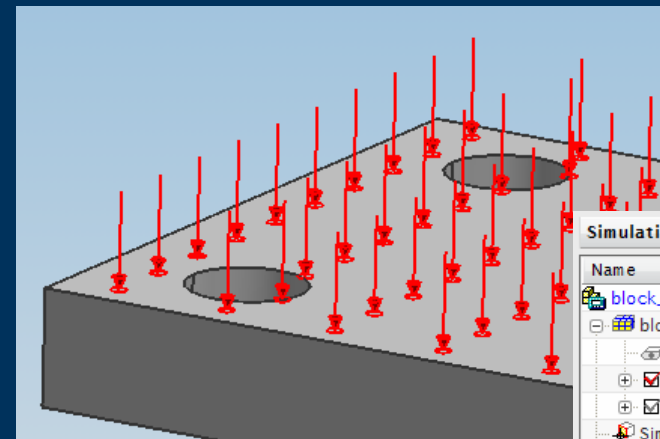
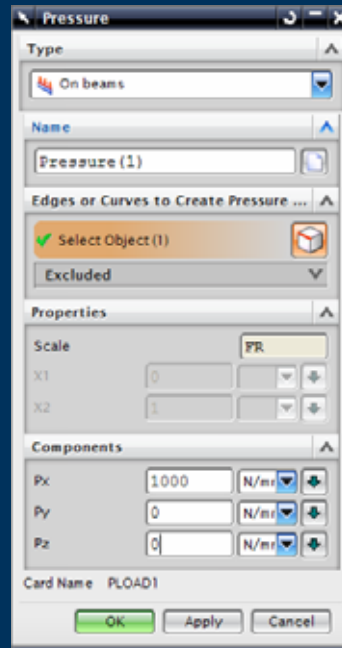
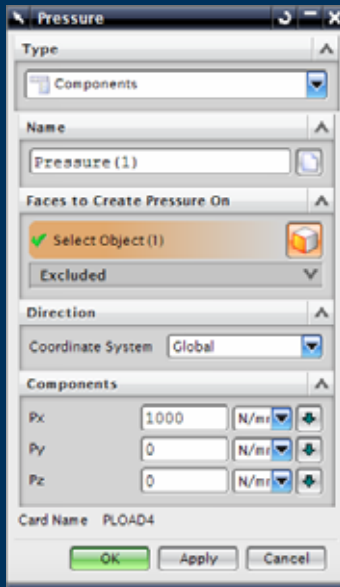
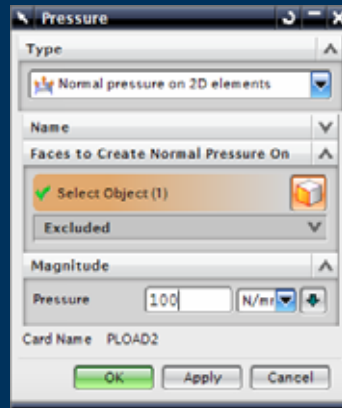
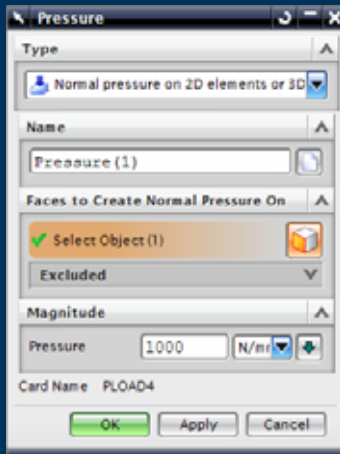
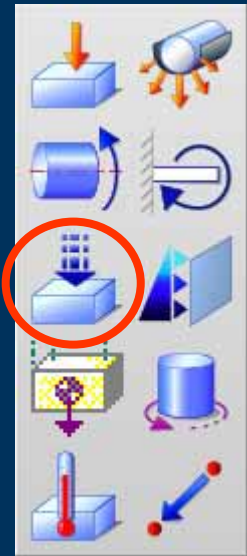


- ▶ Moment Load Options
  - ▶ Magnitude and direction
  - ▶ Normal to selected faces
  - ▶ Mx, My, Mz Components relative to selected coordinate system
  - ▶ Shear, In/Out plane moment
- ▶ Managed in the Load Container



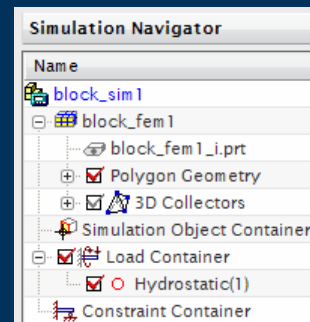
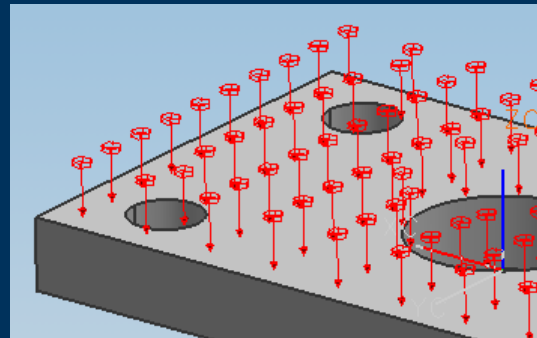
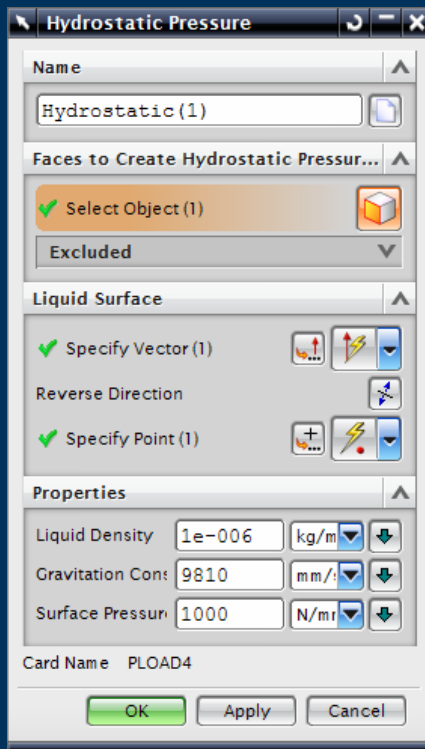
# Loads – Pressure

- ▶ 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



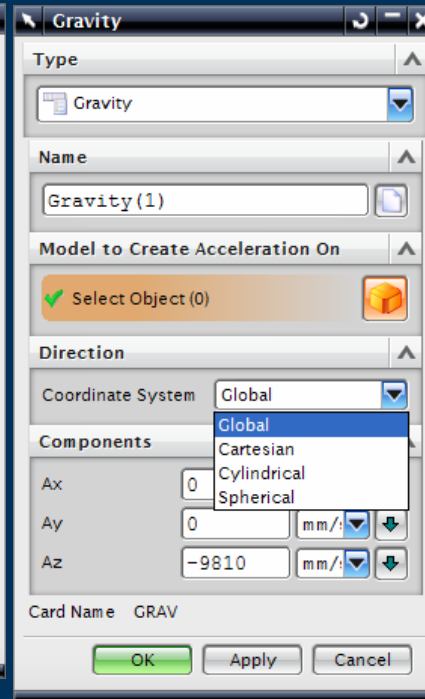
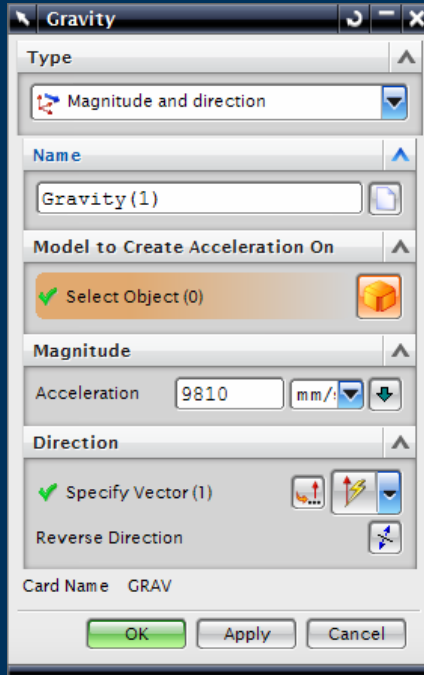
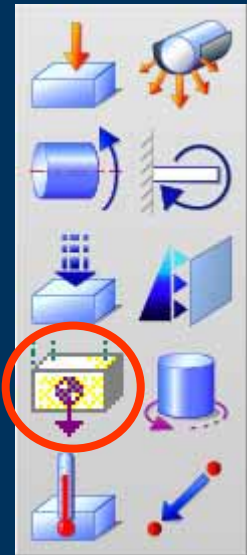
# Loads – Hydrostatic Pressure

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

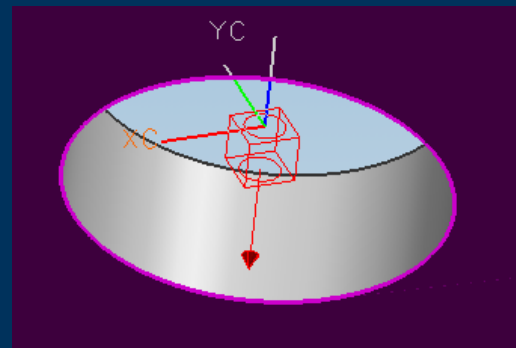




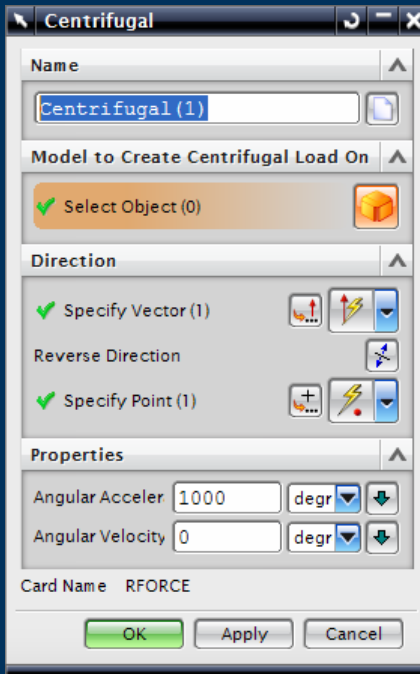
# Loads – Gravity



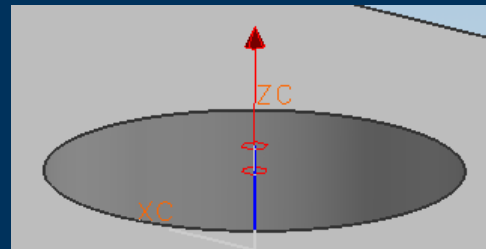
- ▶ Gravity Load
  - ▶ Applied to complete model
  - ▶ Magnitude and Direction
  - ▶ Ax, Ay, Az Component Gravity relative to selected coordinate system
  
- ▶ Managed in the Load Container



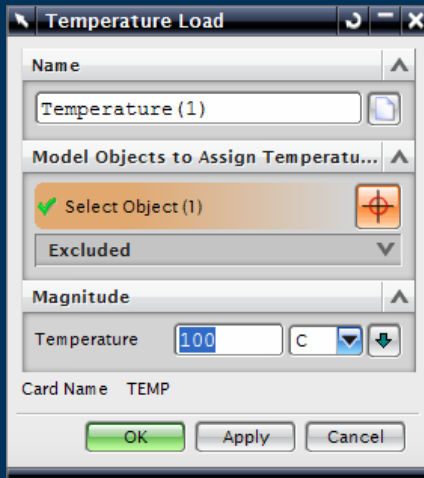
# Loads – Centrifugal



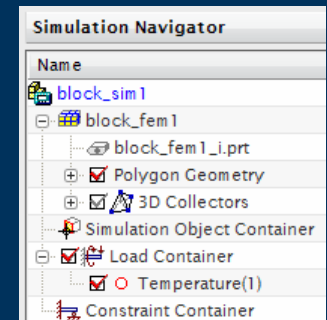
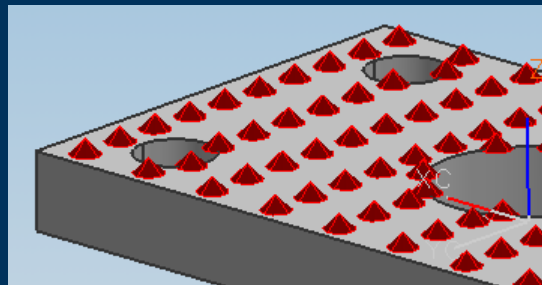
- ▶ Centrifugal Load
  - ▶ Applied to complete model
  - ▶ Direction & centre of rotation
  - ▶ Angular Acceleration
  - ▶ Angular Velocity
- ▶ Managed in the Load Container



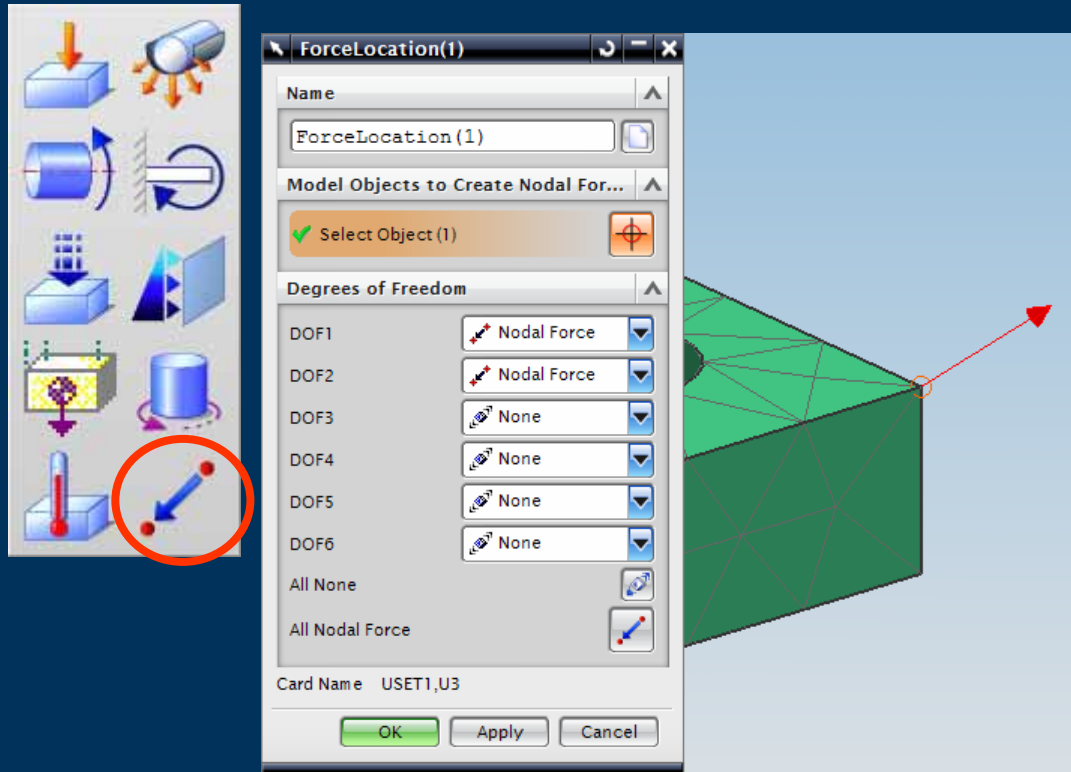
# Loads – Constant Temperature



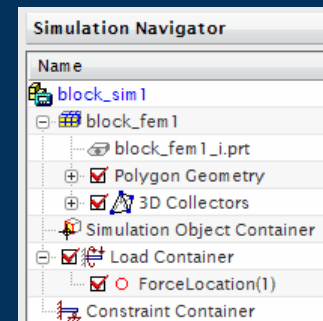
- ▶ Constant Temperature Load
  - ▶ Applied to curves, edges or faces
- ▶ Managed in the Load Container



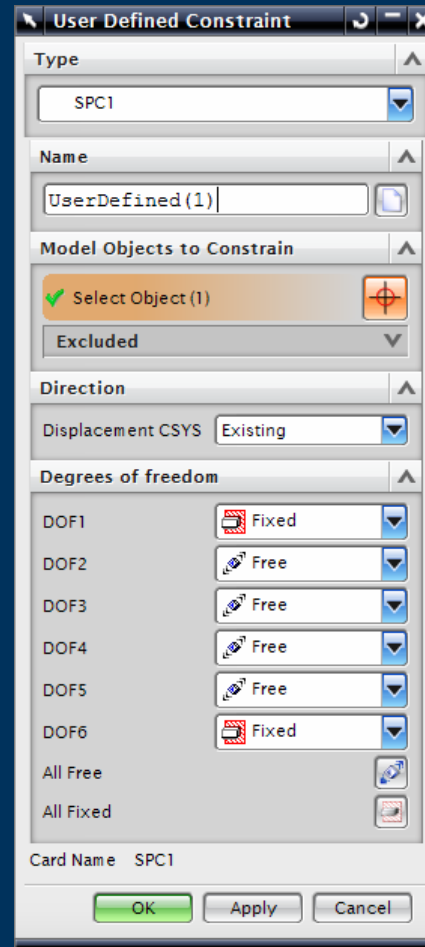
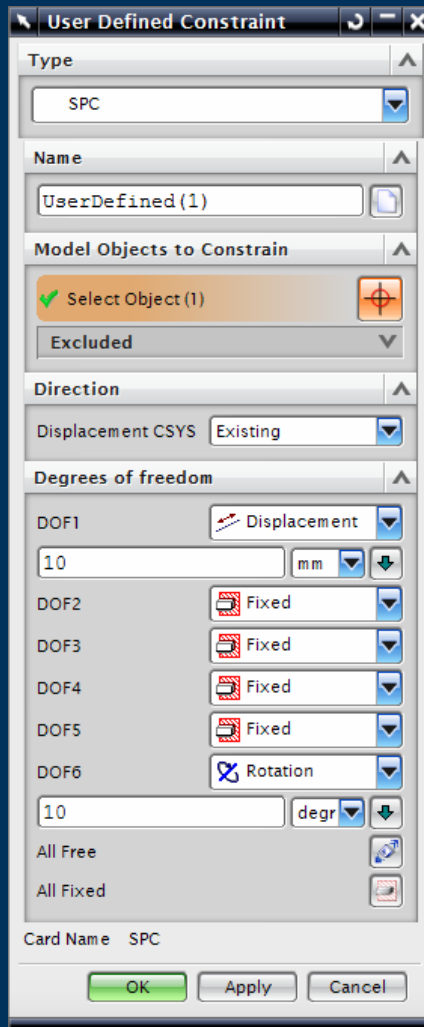
# Nodal Force Location



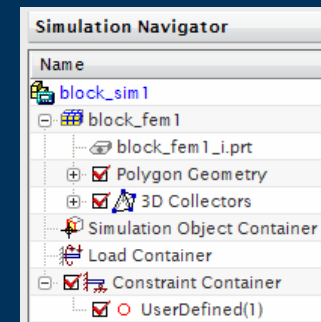
- ▶ Location for a Nodal Force Excitation for the NX Response Simulation application
- ▶ Requires a matching Dynamic Load
- ▶ Managed in the Load Container



# Constraints – User Defined

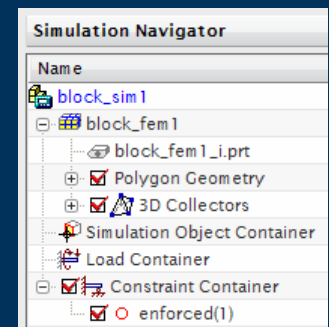
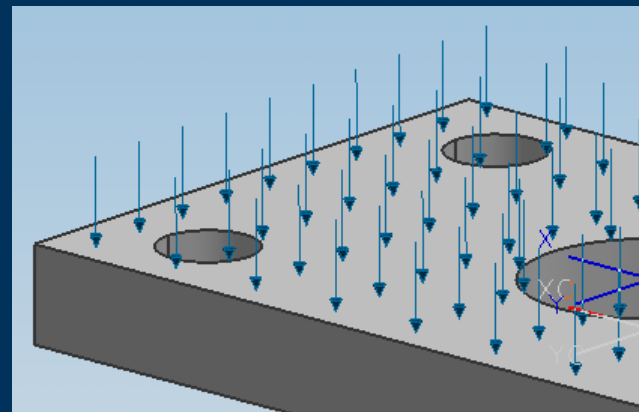
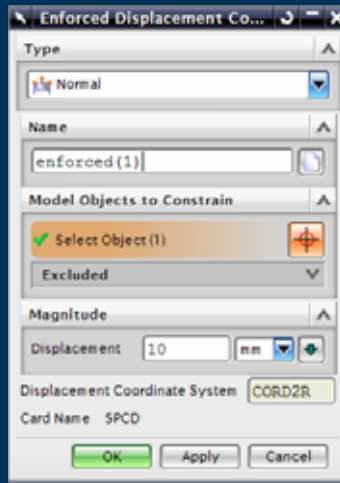
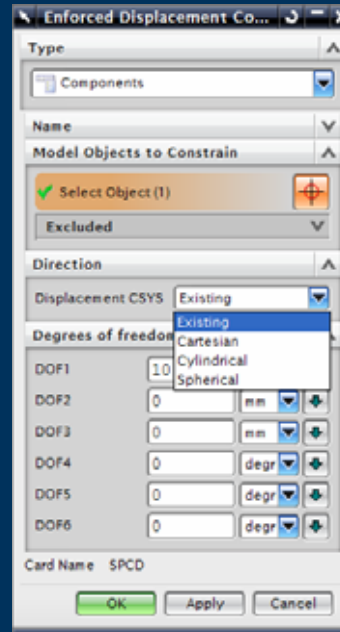
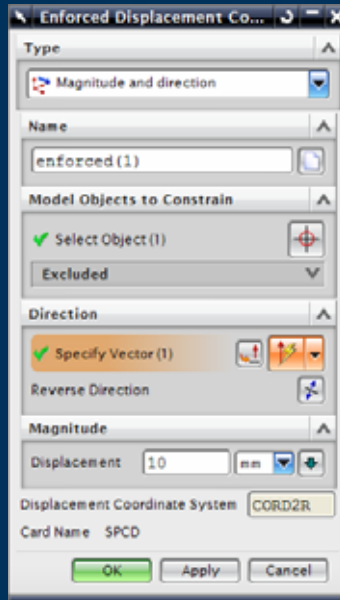


- ▶ User Defined Constraints
  - ▶ Free, Fixed or Displacement
  - ▶ Cartesian, Cylindrical or Spherical coordinate system
- ▶ Managed in Constraint Container

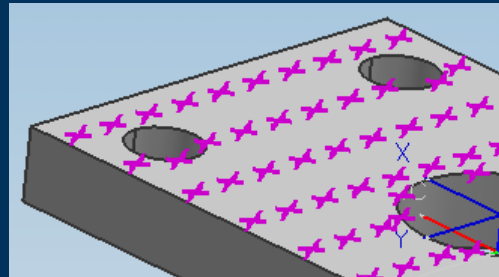
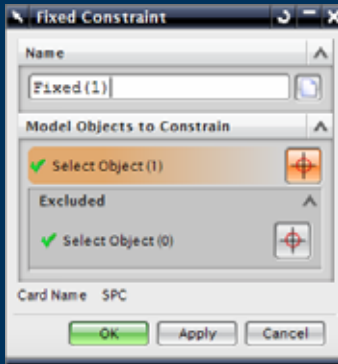


# Constraints – Enforced Displacement

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

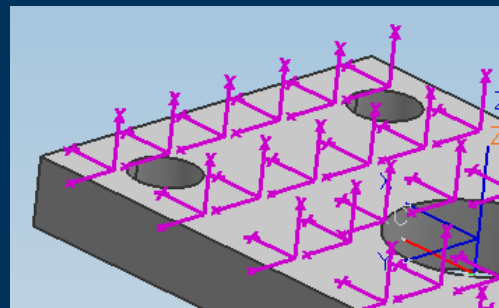
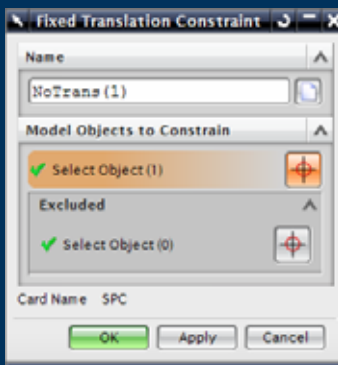


# Constraints – Fixed, Translation & Rotation

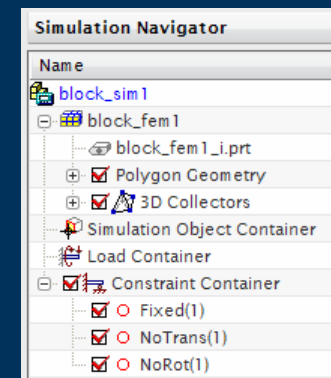
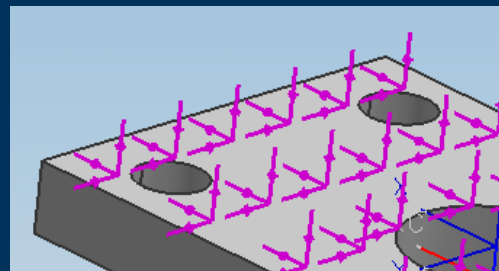
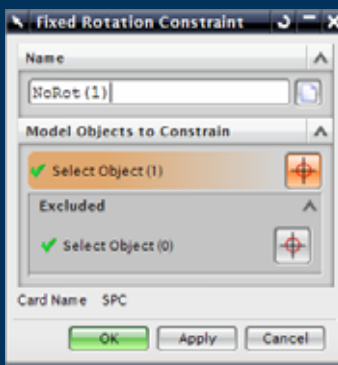


▶ Fixed Constraints (Restrains)

- ▶ All DOF
- ▶ No Translation
- ▶ No Rotation

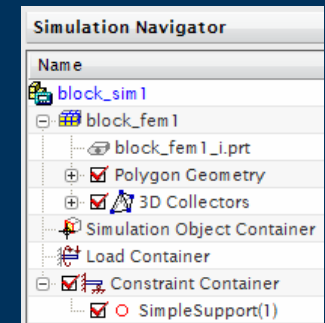
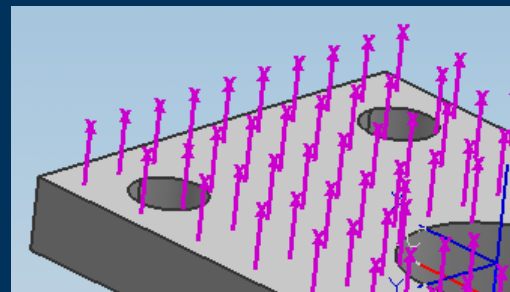
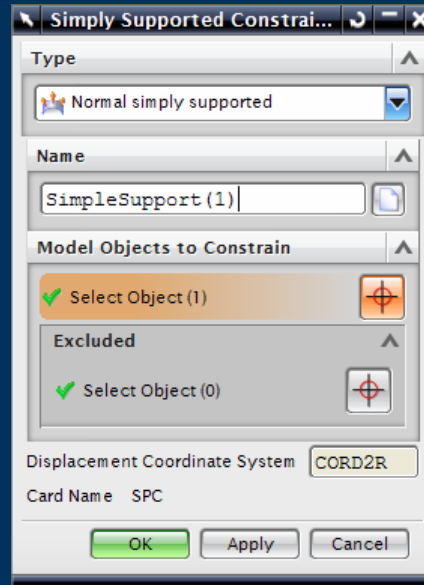
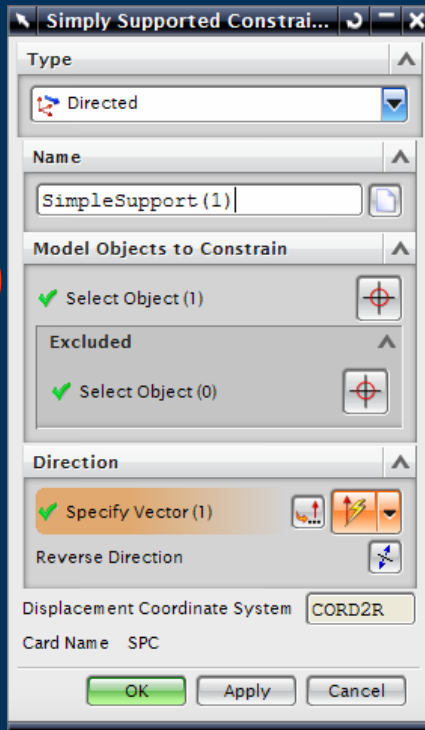


▶ Managed in the Constraints Container



# Constraints – Simply Supported

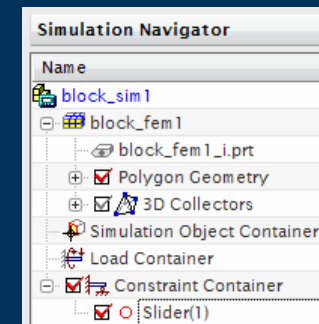
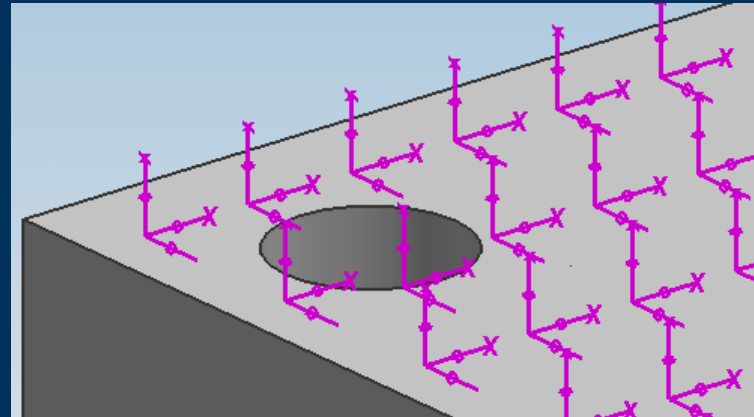
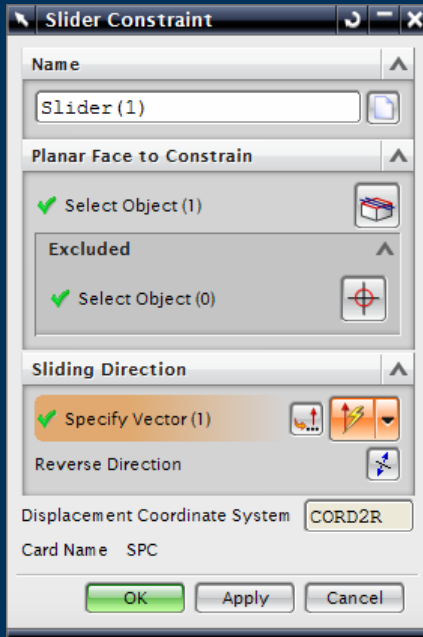
- ▶ Simply Supported Constraint
  - ▶ Magnitude and Direction of Support
  - ▶ Normal to selected surfaces
  - ▶ Managed in the Constraint Container





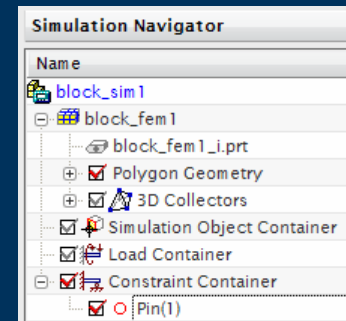
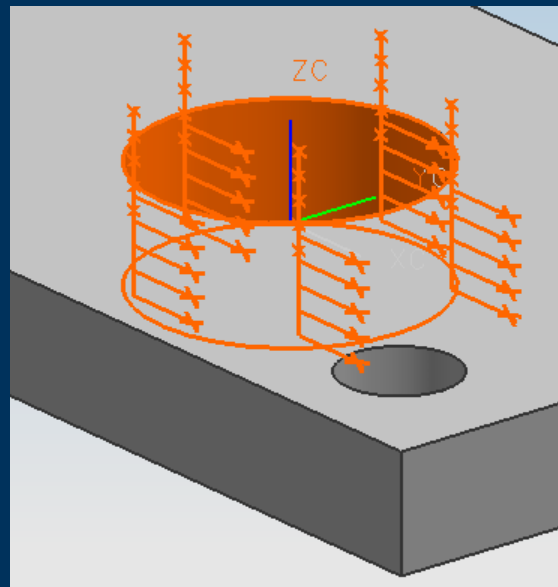
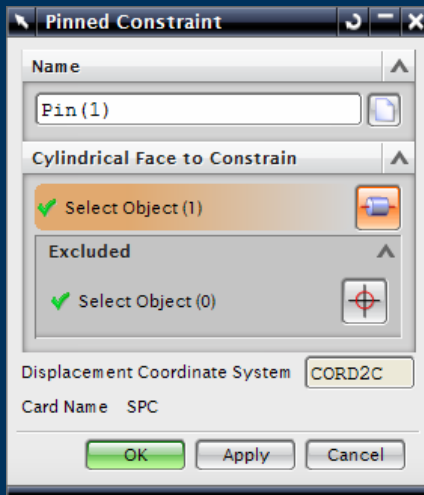
# Constraints – Slider

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



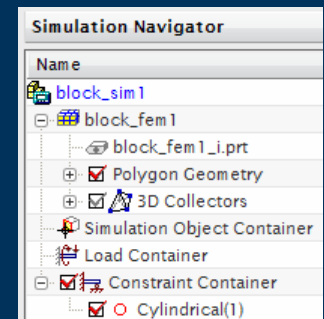
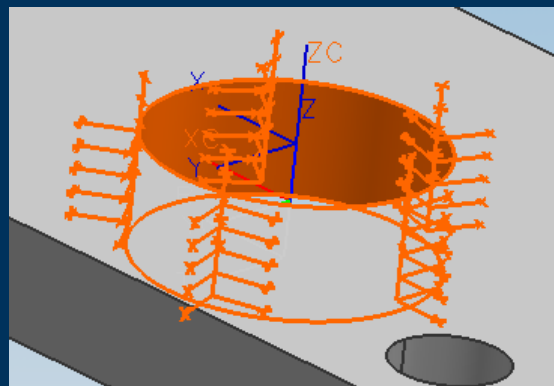
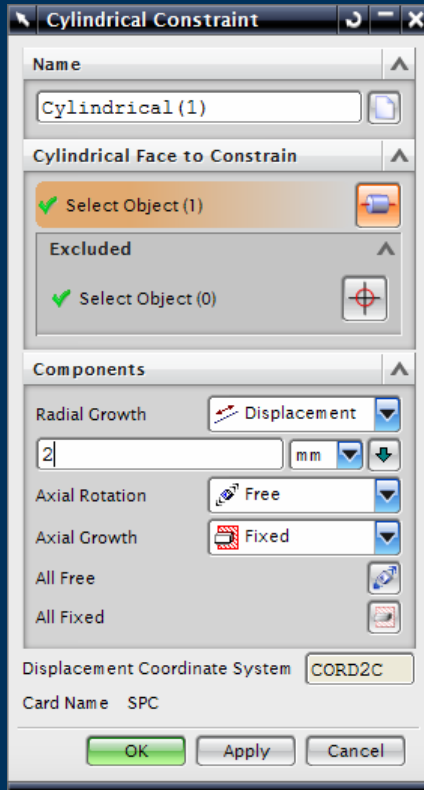
# Constraints – Pinned

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



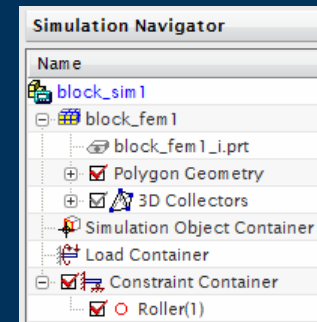
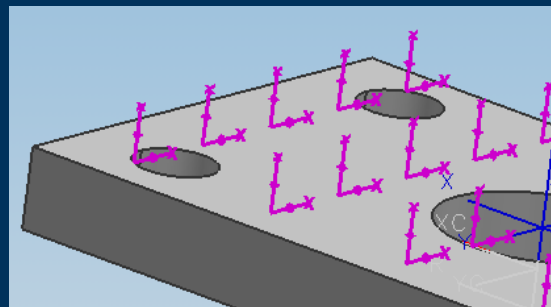
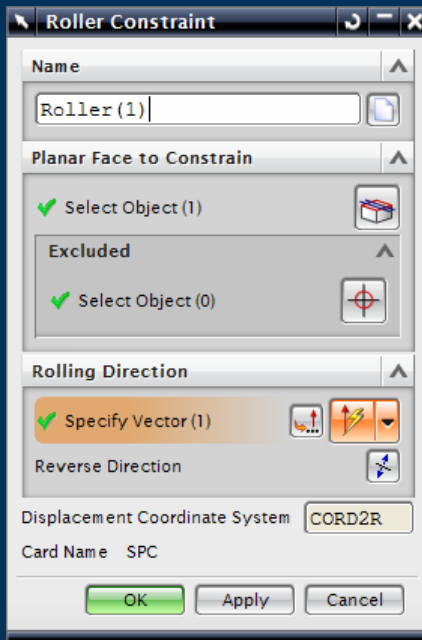
# Constraints – Cylindrical

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



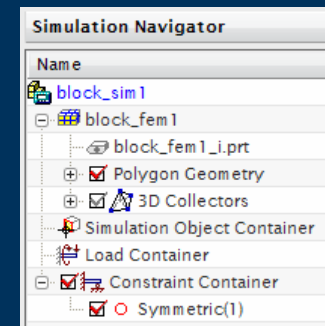
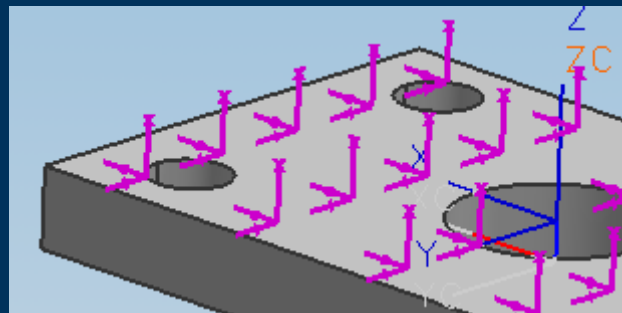
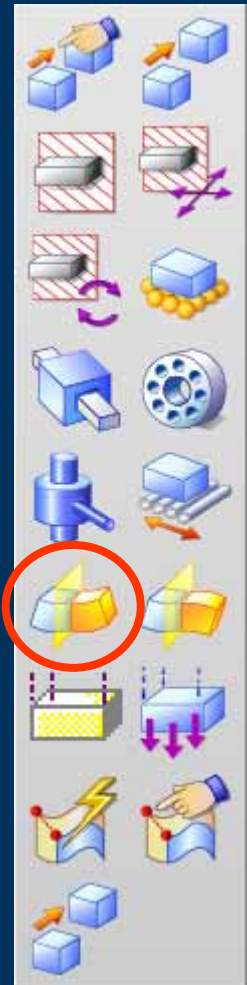
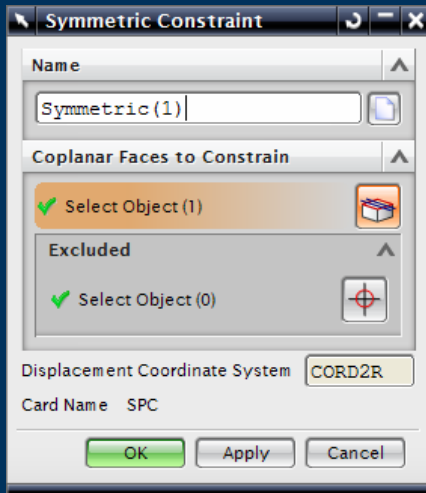
# Constraints – Roller

- ▶ Roller Constraint
- ▶ Managed in the Constraint Container



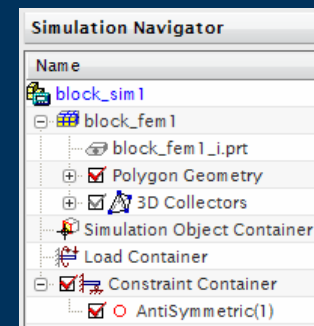
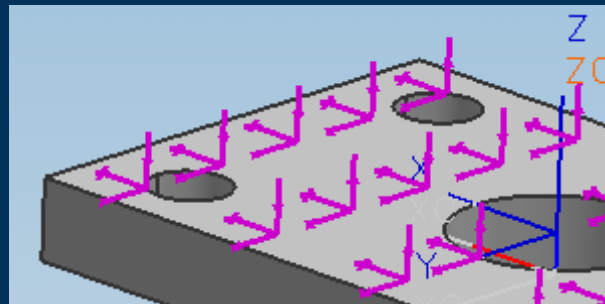
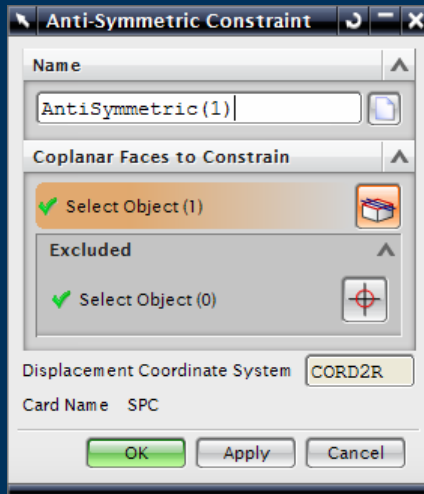
# Constraints – Symmetric

- ▶ Symmetric Constraint
- ▶ Managed in the Constraint Container

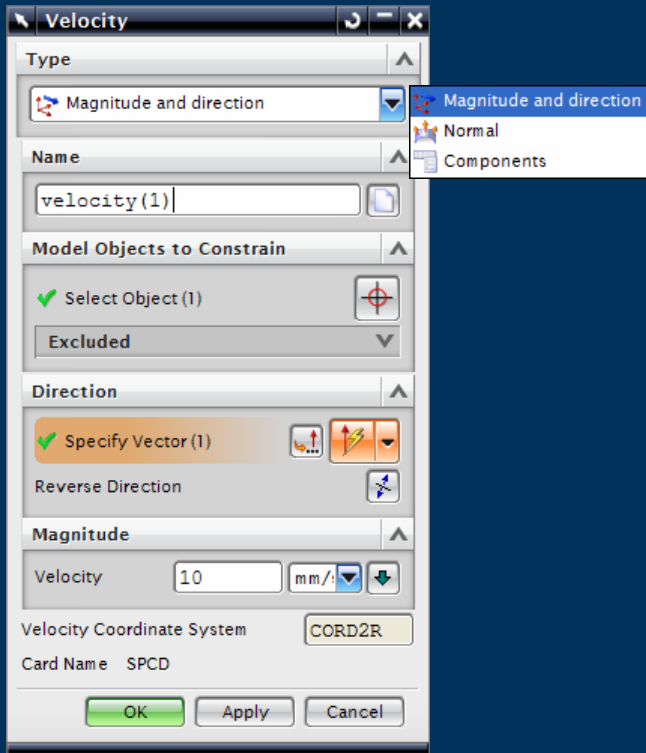


# Constraints – Anti-Symmetric

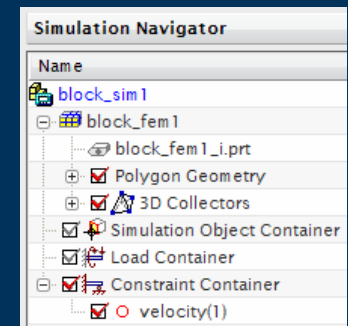
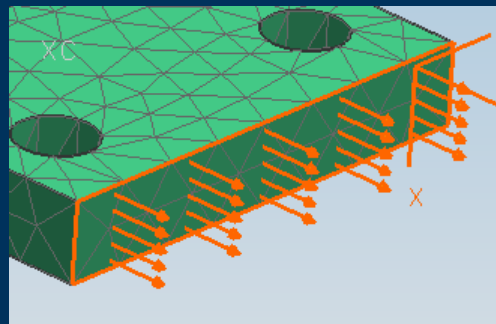
- ▶ Anti-Symmetric Constraint
- ▶ Managed in the Constraint Container



# Constraints – Velocity

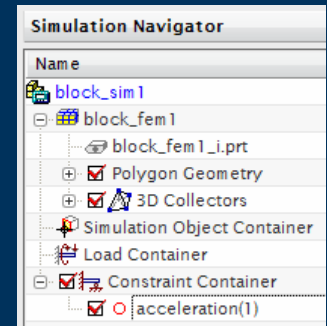
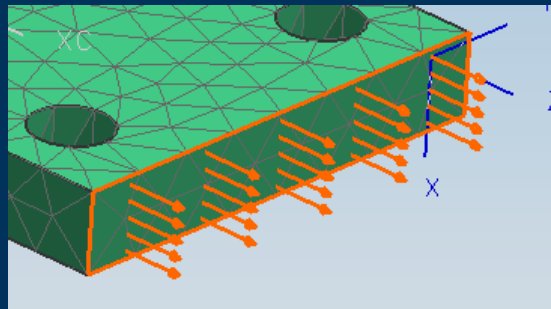
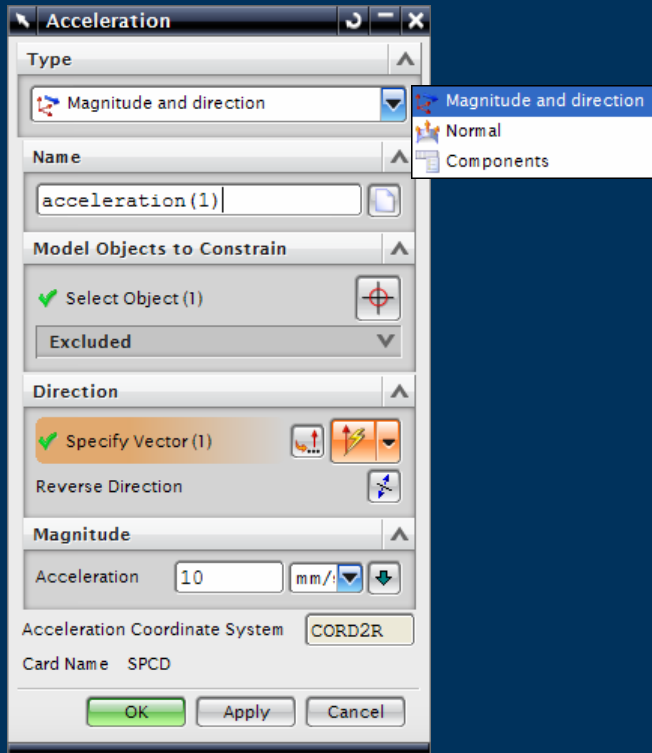


- ▶ Velocity Constraint
- ▶ Specific to these NX Nastran Solutions
  - ▶ SEDFREQ 108 — Direct Frequency Response
  - ▶ SEDTRAN 109 — Direct Transient Response
  - ▶ SEMFREQ 111 — Modal Frequency Response
  - ▶ SEMTRAN 112 — Modal Transient Response
- ▶ Managed in the Constraint Container



# 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



MPC for Rear Axle Bodies

Name: MPC for Rear Axle Bodies

Independent: Automate

Select Object (1)

Excluded

Dependent: Select Object (1)

Excluded

Independent Face / Edge

Reference Coordinate System

Type: Cartesian

Local CSYS

Set Displacement CSYS

Node Match Tolerance

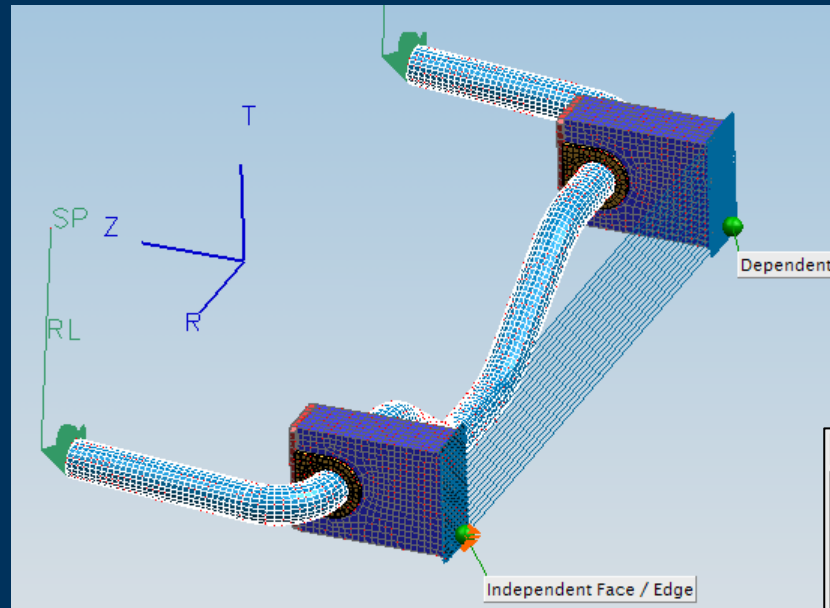
Value: 0.01 mm

Degrees of Freedom

DOF1	On
DOF2	Off
DOF3	On
DOF4	Off
DOF5	Off
DOF6	Off
All Off	
All On	

Card Name: MPC

OK Apply Cancel



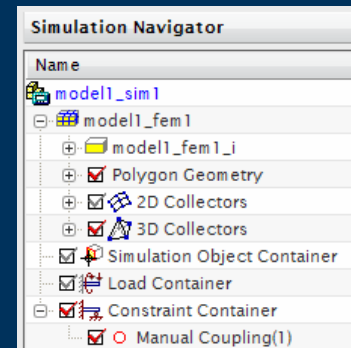
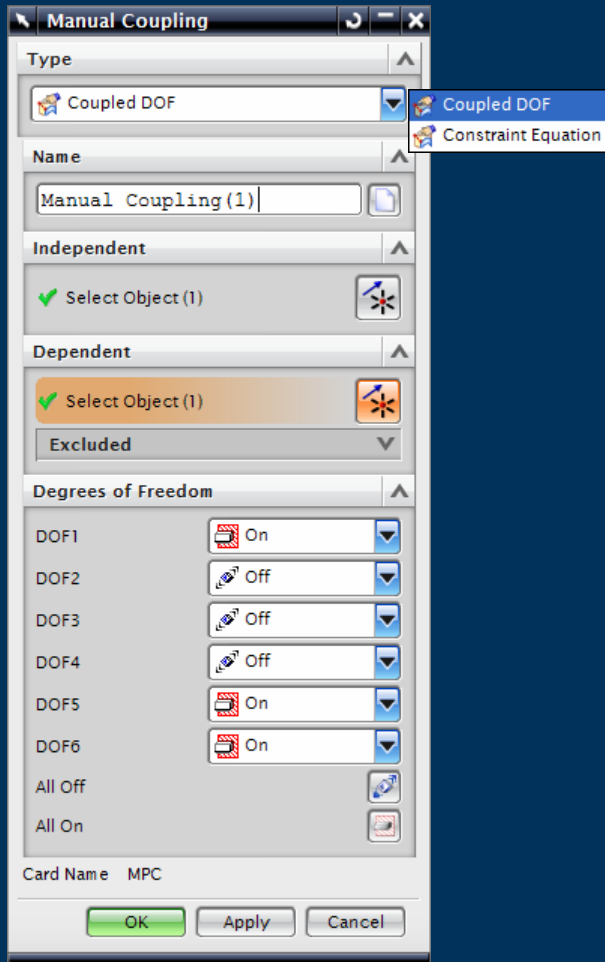
Simulation Navigator

Name

- anti\_roll\_sim
- anti\_roll\_fem 1
- Simulation Object Container
- Load Container
- Constraint Container
  - BUSHING RADIAL CONSTRAINT
  - FIXED SPRING ENDS
  - ENFORCED ROTATION
  - MPC for Rear Axle Bodies

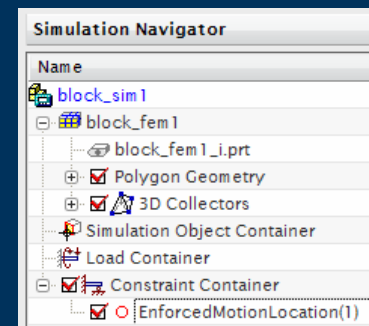
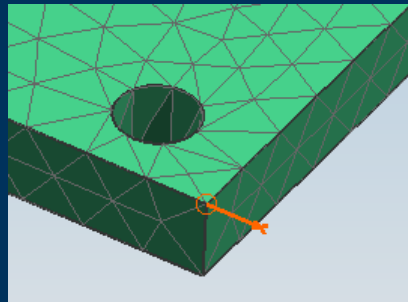
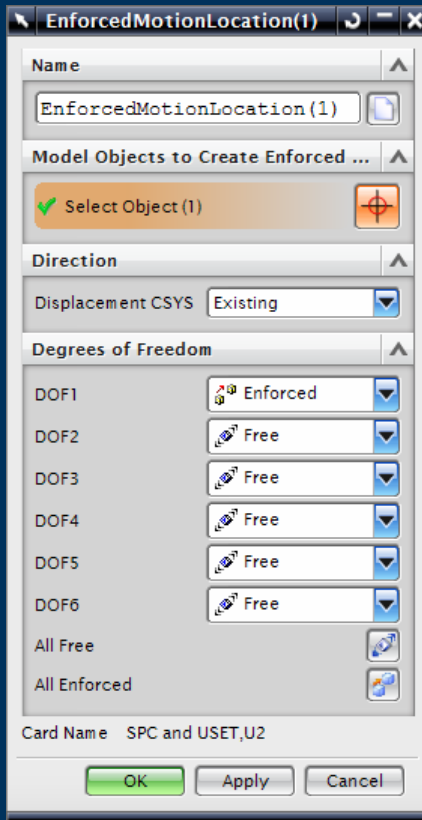
# Constraints – Manual Coupling

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



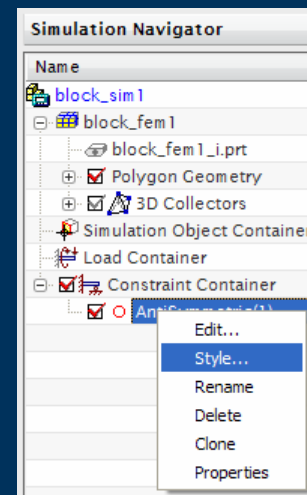
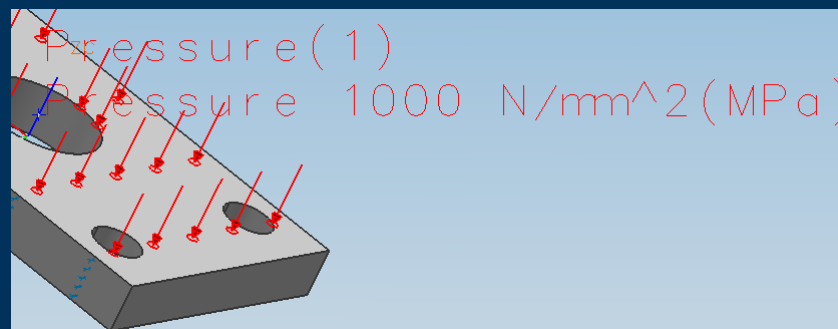
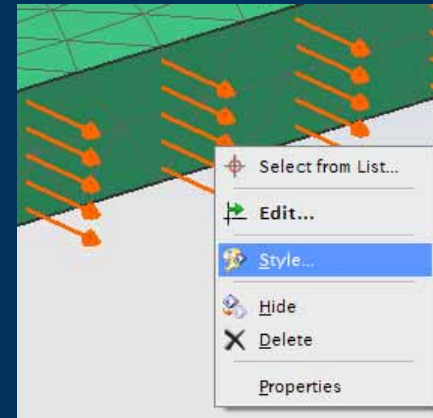
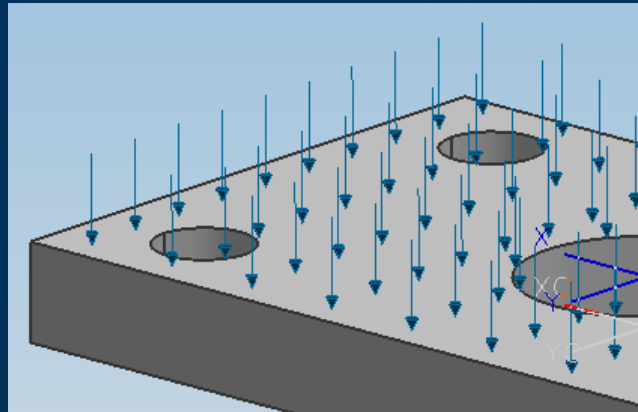
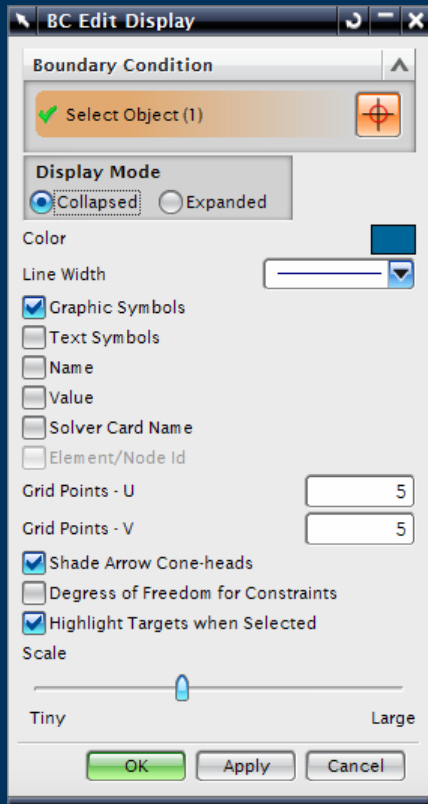
# Constraints – Enforced Motion Location

- ▶ 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

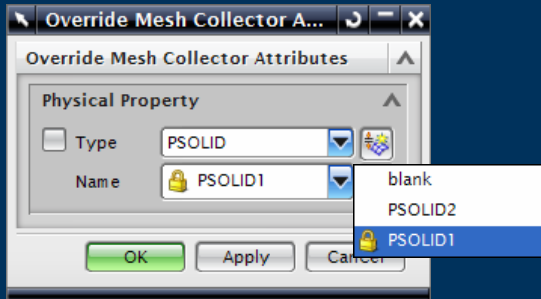


# Boundary Condition Symbol Display Controls

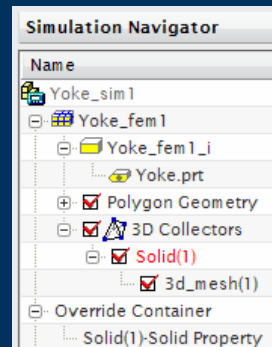
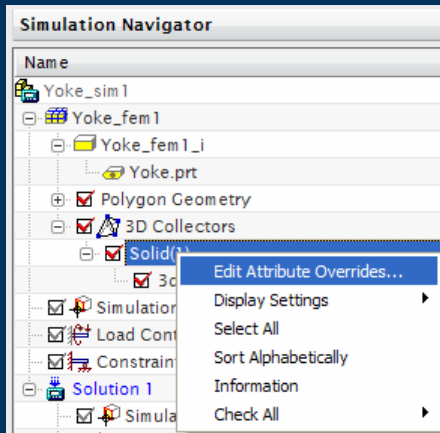
- ▶ All Boundary Conditions have a Symbol associated and the Style can be changed



# 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



# Custom Units & Units Converter

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

**Units Manager**

Measure: Length

Unit Name: MilliMeter

Unit Display Name: mm

Description: millimeters

Conversion Equation: Unit = (a) \* (mm) + (b)

Multiplication Factor (a): 1.0000

Addition Factor (b): 0.0000

Default Unit

Buttons: New Unit, Delete Unit, Update Unit, Close

Available Measures:

- Length
- Area
- Volume
- Mass
- Mass Density
- Fatigue Strength Coefficient
- Time
- Angle
- Velocity
- Acceleration
- Force
- Force Per Unit Length

Available Custom Units:

- Coefficient per Unit Length
- Moment of Inertia (Area)
- Viscous Damping
- Energy
- Power
- Momentum
- Temperature Gradient
- Energy per Unit Mass
- Dissipation Rate of Energy per Unit Mass
- Mass Flux
- Mass per Unit Length
- Mass per Unit Area

**Units Converter**

Quantity: Force Per Unit Length

From: 1.0000 N/mm

To: 68.5217 lbf/ft

Close

**Units Converter**

Quantity: Fatigue Strength Coefficient

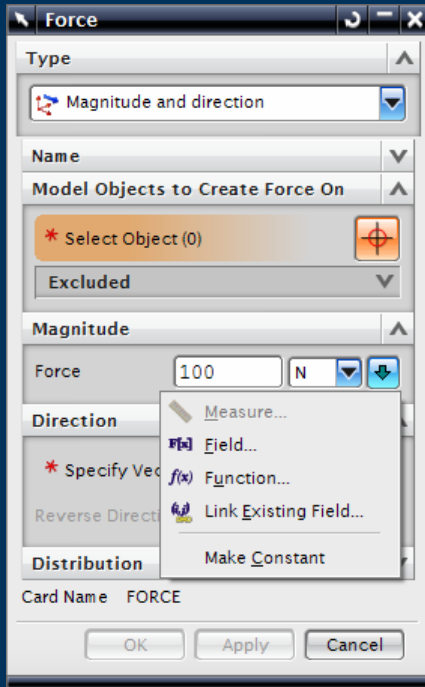
From: 1.0000 N/mm<sup>2</sup>(MPa)

To: 145.0376 lbf/in<sup>2</sup>(psi)

Close

Measure	Unit Name	Display Name	Description
Length	MilliMeter	mm	millimeters
Area	SquareMilliMeter	mm <sup>2</sup>	millimeters squared
Volume	CubiMilliMeter	mm <sup>3</sup>	millimeters cubed
Mass	Kilogram	kg	kilograms
Mass Density	KilogramPerCubiMilliMeter	kg/mm <sup>3</sup>	kilograms per millimeter cubed
Fatigue Strength Coefficient	NewtonPerSquareMilliMeter	N/mm <sup>2</sup> (MPa)	Newtons per millimeter squared
Time	Second	sec	seconds
Angle	Degree	degrees	degrees
Velocity	MilliMeterPerSecond	mm/sec	millimeters per second
Acceleration	MilliMeterPerSquareSecond	mm/sec <sup>2</sup>	millimeters per second squared
Force	Newton	N	Newtons
Force Per Unit Length	NewtonPerMilliMeter	N/mm	Newtons per millimeter
Pressure	NewtonPerSquareMilliMeter	N/mm <sup>2</sup> (MPa)	Newtons per millimeter squared
Moment	NewtonMilliMeter	N-mm	Newtons-millimeters
Stress	NewtonPerSquareMilliMeter	N/mm <sup>2</sup> (MPa)	Newtons per millimeter squared
Strain	Strain_Metric1	mm/mm	millimeters per millimeter
Strain Energy	StrainEnergy_Metric2	N-mm	Newtons-millimeters
Strain Energy Density	StrainEnergyDensity_Metric2	N-mm/mm <sup>3</sup>	Newtons-millimeters per millimeter cubed
Temperature	Celsius	C	Celsius
Heat Flux	WattPerMilliMeter	W/mm <sup>2</sup>	Watts per millimeter squared
Convection Coefficient	ConvectionCoefficient_Metric2	W/mm <sup>2</sup> -C	Watts per millimeter squared per degree Celai
Thermal Conductivity	ThermalConductivity_Metric2	W/mm-C	Watts per millimeter per degree Celsius
Thermal Expansion Coefficient	ThermalExpansion_Metric1	1/C	Expansion coefficient per degree Celsius
Specific Heat	SpecificHeat_Metric2	J/Kg-K	Joules per Kilogram per degree Kelvin
Angular Velocity	DegreePerSecond	degrees/sec	degrees per second
Angular Acceleration	DegreePerSecondSquared	degrees/sec <sup>2</sup>	degrees per second squared
Fatigue Life	DutyCycles	DutyCycles	Duty Cycles
Heat Flow Rate	HeatFlow_Metric2	W	Watt
Thermal Energy	ThermalEnergy_Metric2	J	Joule
Mass Moment of Inertia	KilogramMilliMeterSquared	kg-mm <sup>2</sup>	kilogram - millimeter squared
Dynamic Viscosity	Kg/m-second	kg/m-sec	kilograms per millimeter per second
Heat Generation	HeatGeneration_Metric2	W/mm <sup>3</sup>	Watts per millimeter cube
Thermal Conductance	ThermalConductance_Metric2	W/m-C	Watts per degree Celsius
Conductance per Unit Length	ThermalConductance_Metric2	W/mm-C	Watts per millimeter per degree Celsius
Thermal Resistance	ThermalResistance_Metric2	C/W	degrees Celsius per Watt
Mass Flow Rate	KilogramPerSecond	kg/sec	Kilograms per second
Volume Flow Rate	CubiMilliMeterPerSecond	mm <sup>3</sup> /sec	millimeters cubed per second
Temperature Difference	CelsiusDifference	C	Celsius
Frequency	hertz	Hz	hertz
Coefficient per Unit Length	CoefficientPerMilliMeter	1/mm	coefficient per millimeter
Moment of Inertia (Area)	MilliMeterFourth	mm <sup>4</sup>	millimeters fourth
Viscous Damping	KilogramPerSecond	kg/sec	Kilograms per second
Energy	Joule	J	Joule
Power	Watt	W	Watt
Momentum	KilogramMeterPerSecond	kg-m/sec	kilogram-meter per second
Temperature Gradient	CelsiusPerMilliMeter	C/mm	Celsius per millimeter
Energy per Unit Mass	EnergyPerMass_Metric1	mm <sup>2</sup> /sec <sup>2</sup>	millimeters squared per second squared
Dissipation Rate of Energy per Unit Mass	DissipationRate_Metric1	mm <sup>2</sup> /sec <sup>3</sup>	millimeters squared per second cubed
Mass Flux	MassFlux_Metric1	kg/sec-mm <sup>2</sup>	kilograms per second per millimeter squared
Mass per Unit Length	KilogramPerMilliMeter	kg/mm	kilograms per millimeter
Mass per Unit Area	KilogramPerMilliMeterSquared	kg/mm <sup>2</sup>	kilograms per millimeter squared

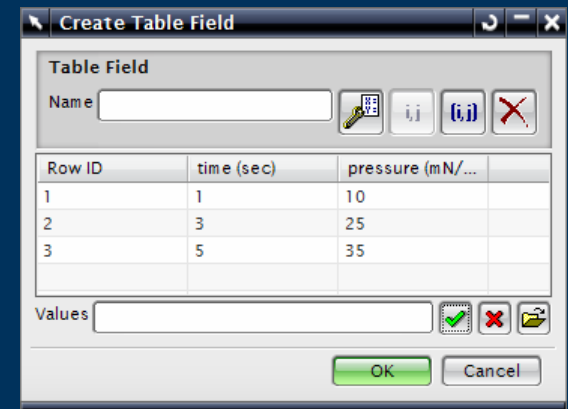
# Unit Selection



- ▶ 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

m	radians	mN	mN/mm <sup>2</sup> (kPa)	kg/m <sup>3</sup>	m/sec <sup>2</sup>
mm	degrees	N	N/mm <sup>2</sup> (MPa)	kg/mm <sup>3</sup>	mm/sec <sup>2</sup>
in		lbf	Pa(N/m <sup>2</sup> )	lbf-sec <sup>2</sup> /in <sup>4</sup>	in/sec <sup>2</sup>
ft	m N-m-m		lbf/in <sup>2</sup> (psi)	Slugs/ft <sup>3</sup>	ft/sec <sup>2</sup>
cm	N-m-m	C	lbf/ft <sup>2</sup>	lbm/in <sup>3</sup>	gs
km	N-m	F	bars	t/m <sup>3</sup>	
mi	lbf-in	K	atmospheres	t/mm <sup>3</sup>	
micron	lbf-ft	R			
nm	rev/sec <sup>2</sup>				
angstrom	rev/min <sup>2</sup>				
	degrees/sec <sup>2</sup>				
	radians/sec <sup>2</sup>				



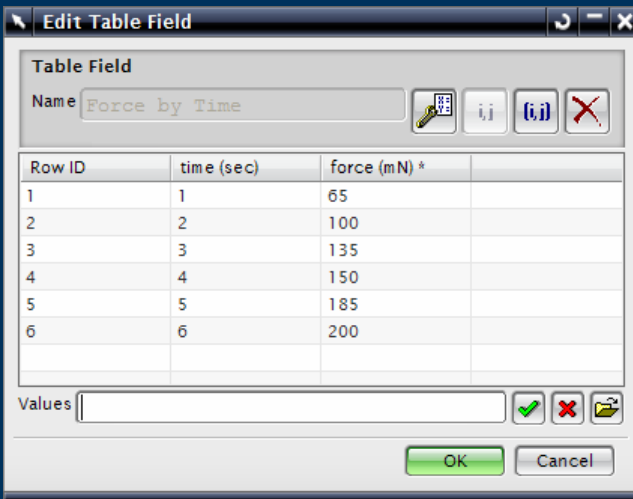
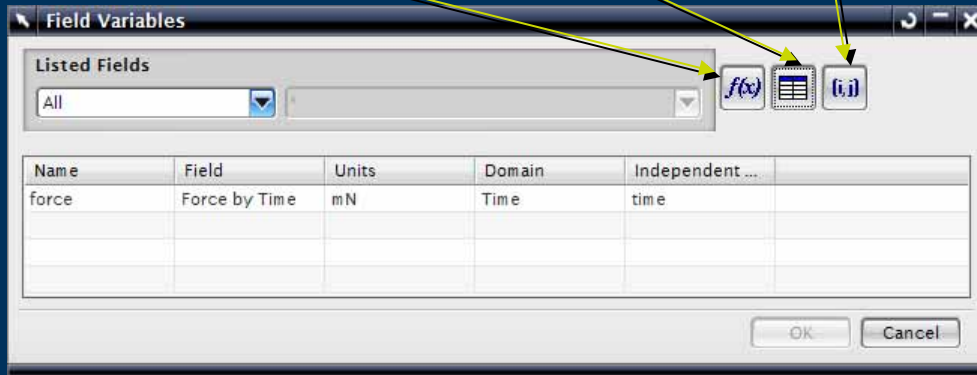
# Boundary Condition Magnitude – Table Field

Maths Expression Field

Table Based Field

Existing Named Variables

- ▶ Tables Field
- ▶ User selected Dependant and Independent columns



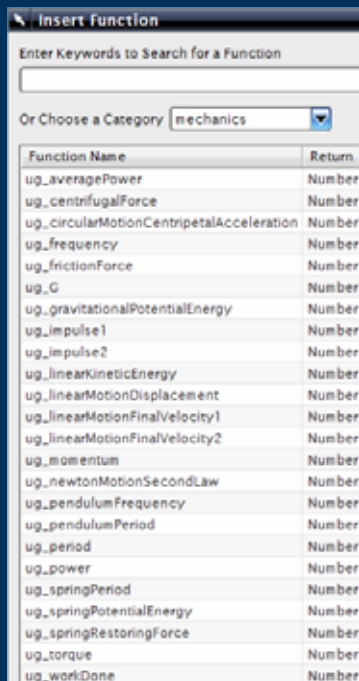
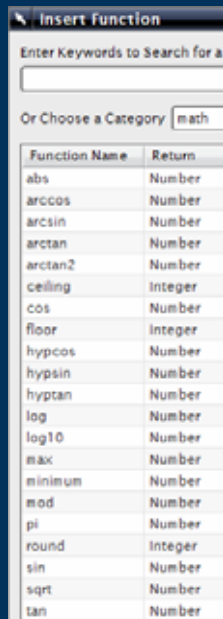
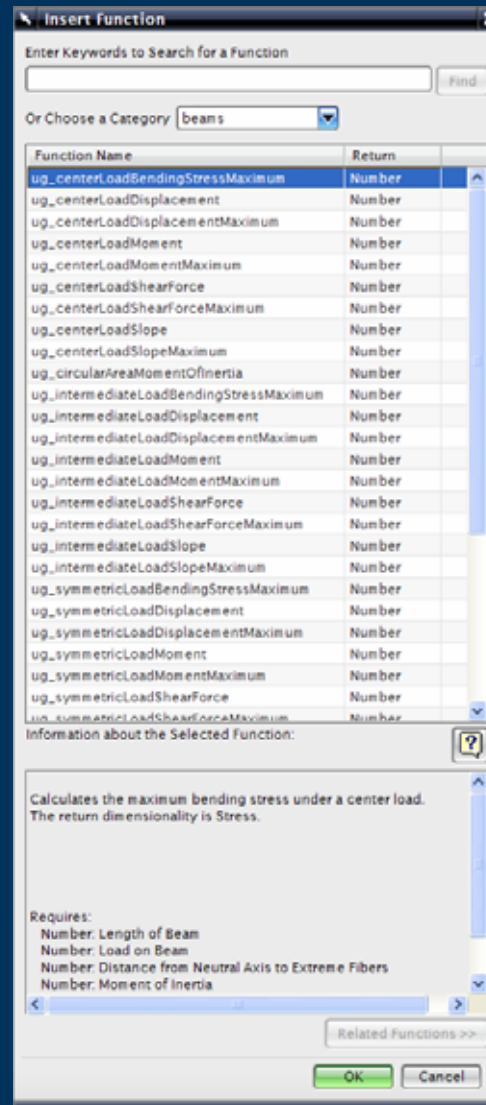
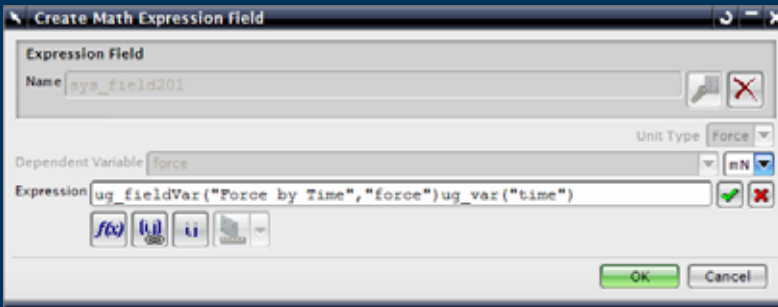


# Boundary Condition Magnitude – Function Field

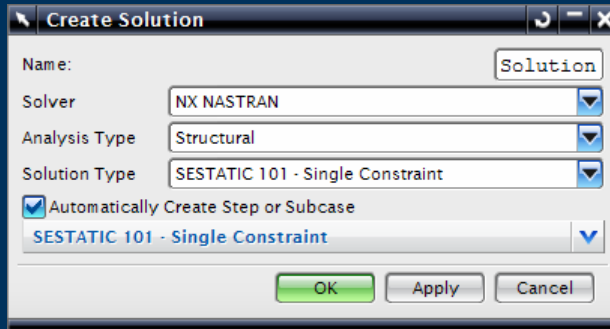
▶ Pre-defined functions that calculate a value

▶ Categories

- ▶ Beams
- ▶ Fluids
- ▶ Gears
- ▶ Geometry
- ▶ Materials
- ▶ Maths
- ▶ Mechanics
- ▶ Misc
- ▶ O Rings
- ▶ Plate
- ▶ Spreadsheet
- ▶ Spring
- ▶ String
- ▶ Units
- ▶ Vibration



# Solution

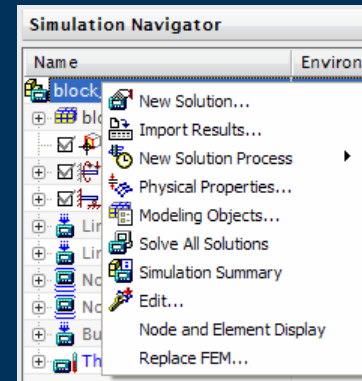


- ▶ Solution is Solver dependant
- ▶ Solution gathers everything together to perform a solve
- ▶ SIM File can contain many Solutions to study different aspects of the design
- ▶ Only one is active

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
Axisymmetric Structural	Structural	SESTATIC 101 - Single Constraint
		SESTATIC 101 - Multiple Constraint
		NLSTATIC 106
Axisymmetric Thermal	Thermal	NLSCSH 153

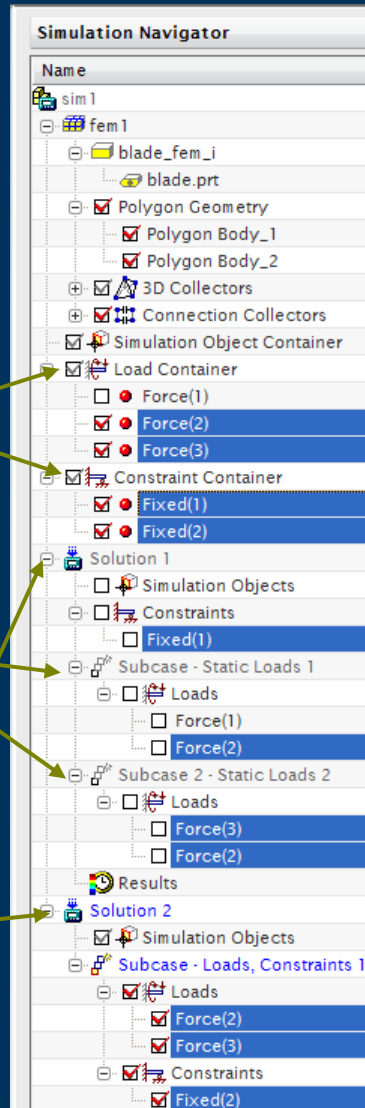
Solver	Analysis Type	Solution Type
ABAQUS	Structural	General Analysis
	Thermal	Heat Transfer
	Axisymmetric Structural	General Analysis
	Axisymmetric Thermal	Heat Transfer

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



Name	Environment	Description
block_sim1	Active: NX NASTRAN - T...	
block_fem1	Default: NX NASTRAN - ...	
Simulation Object C...		
Load Container		
Constraint Container		
Linear Statics 1	NX NASTRAN - Structural	SESTATIC 101 - Single Constraint
Linear Ststics 2	NX NASTRAN - Structural	SESTATIC 101 - Single Constraint
Normal Modes 1 to 25	NX NASTRAN - Structural	SEMODES 103
Normal Modes 26 to 100	NX NASTRAN - Structural	SEMODES 103
Buckling Prediction	NX NASTRAN - Structural	SESTATIC 101 - Multi Constraint
Thermal Constant 75C	NX NASTRAN - Thermal	NLSCSH 153

# Solution – Containers and Re-using Data



Simulation Containers

Multiple Subcases

Multiple Solutions

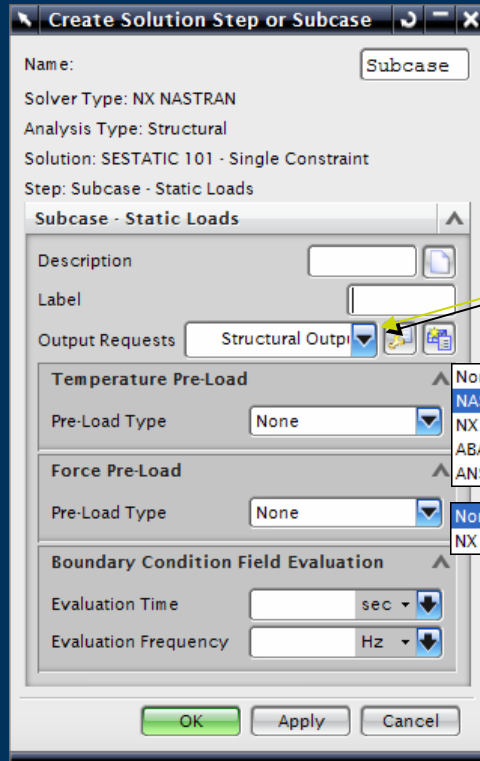
- ▶ All Boundary Conditions, constraints etc are stored in Containers
- ▶ They are then referenced by the Solutions and Subcases

Drag 'n' Drop

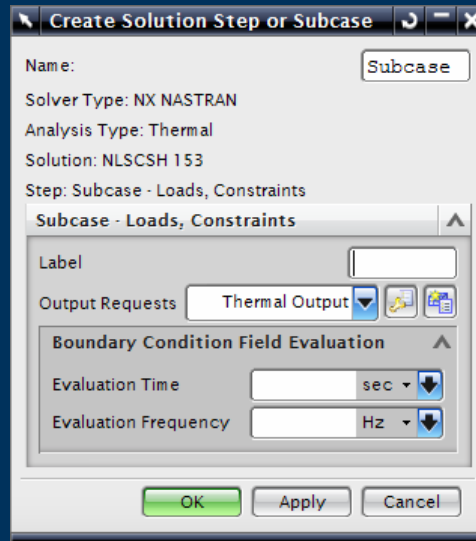
## Benefits

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

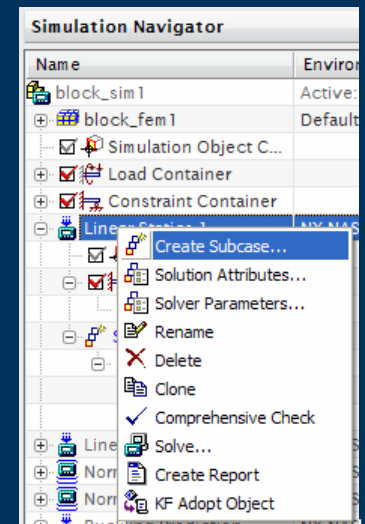
# Solution – Subcase Management



Output Request Object

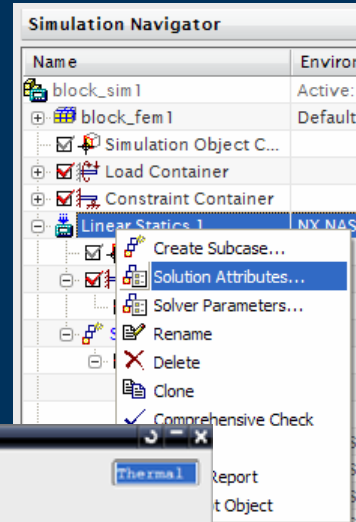
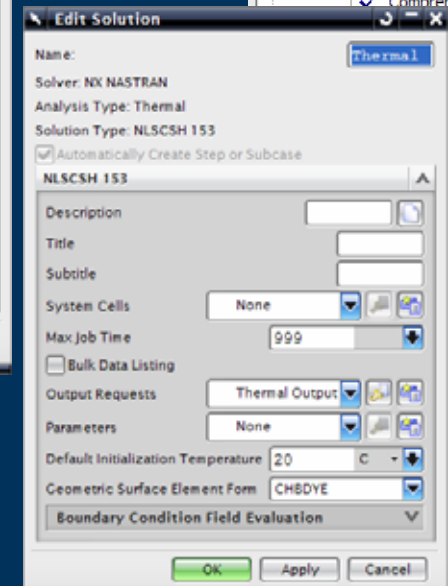
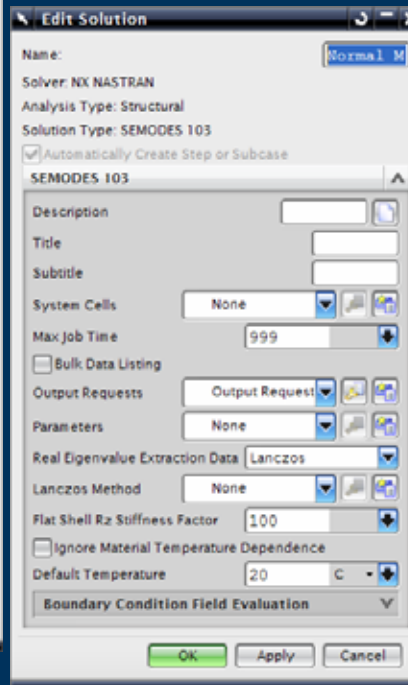
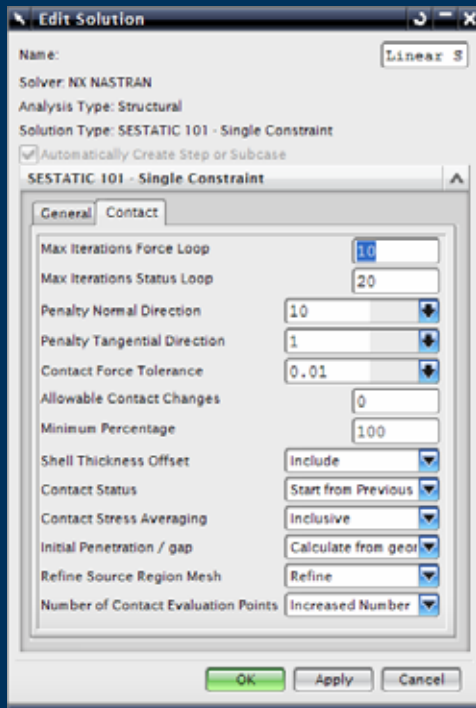
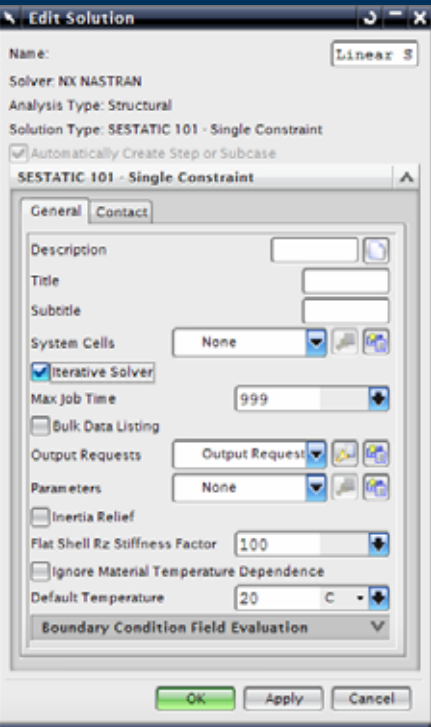


- ▶ Subcase availability and options will vary according to the active Solver and Solution type
- ▶ Each Solution can have multiple Subcases
- ▶ Loads can be used in any combination of Subcases
- ▶ Subcases can include Pre-Loads like Thermal results from a previous solve



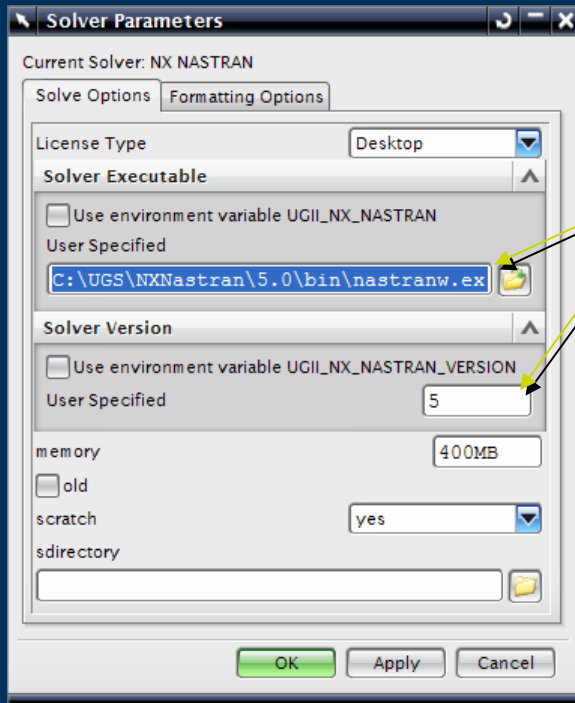
# Solution – Attributes

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

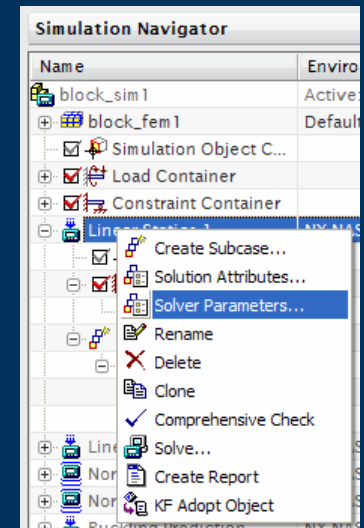
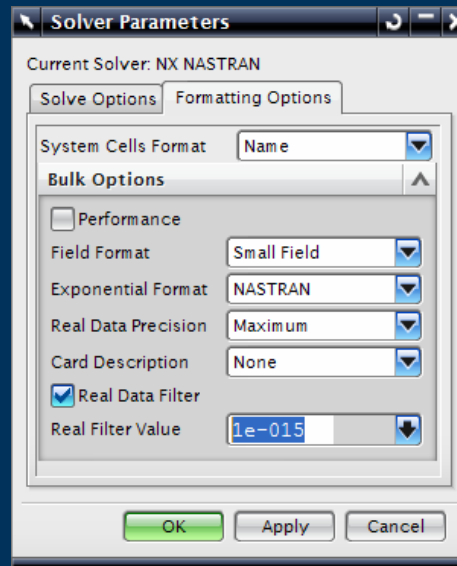


# Solution – Parameters

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



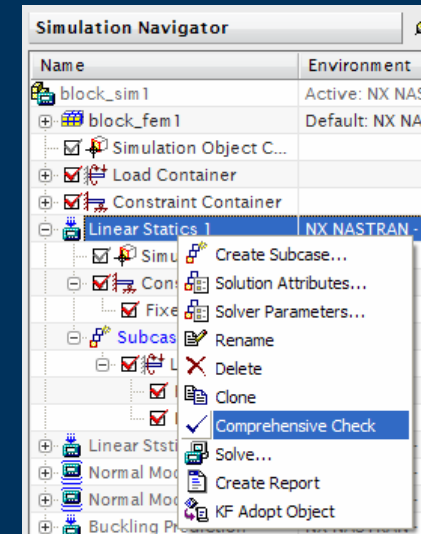
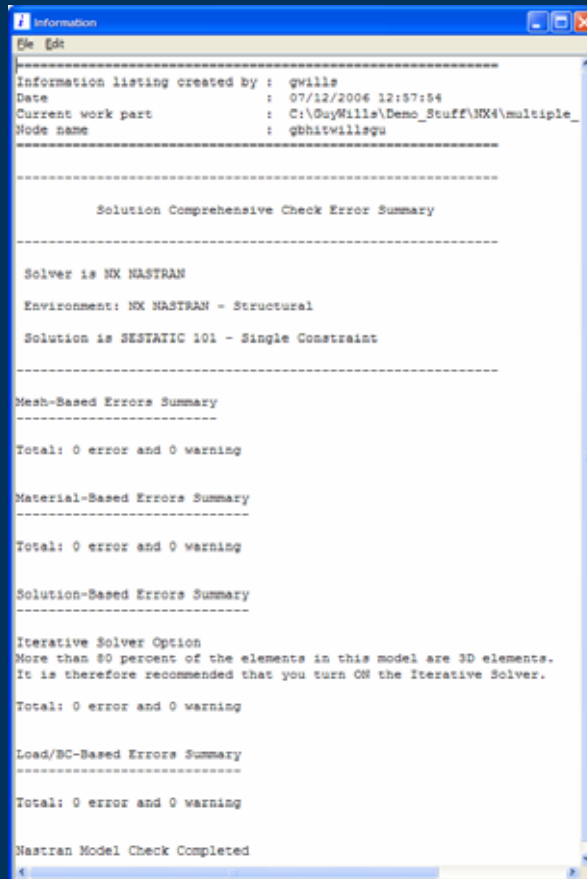
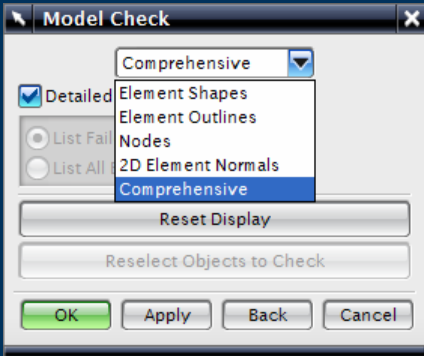
Exact Version of NX Nastran



# Solution – Comprehensive Check

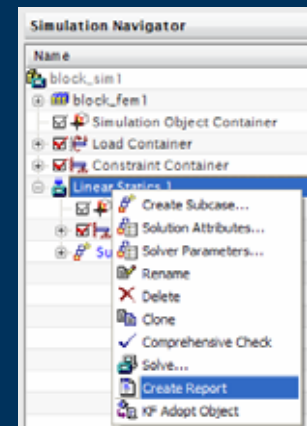
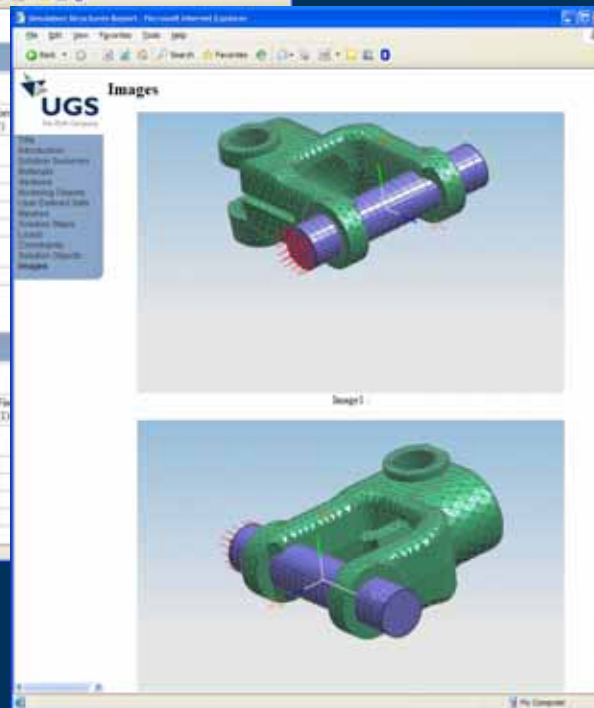
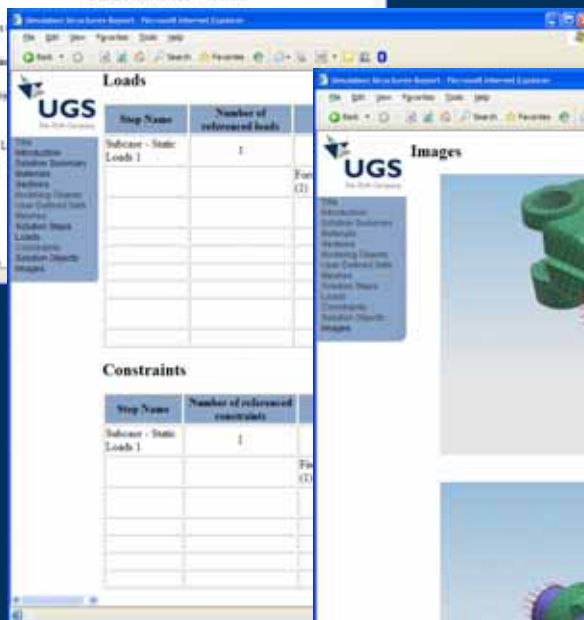
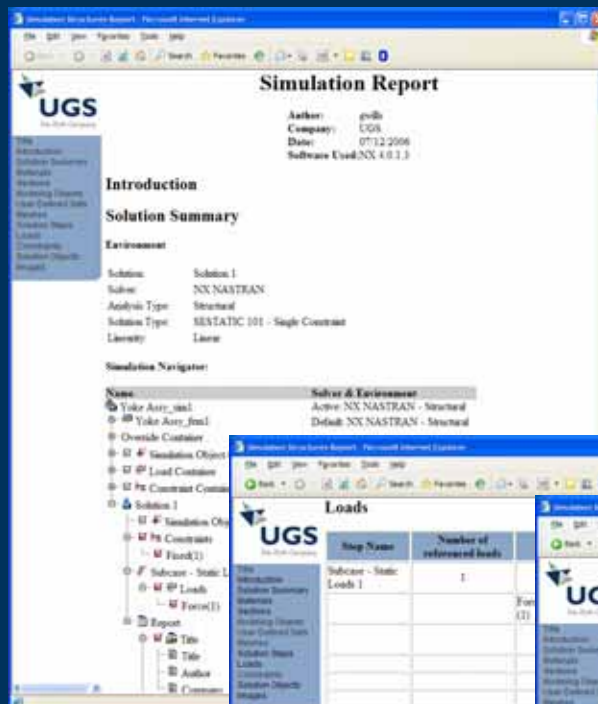
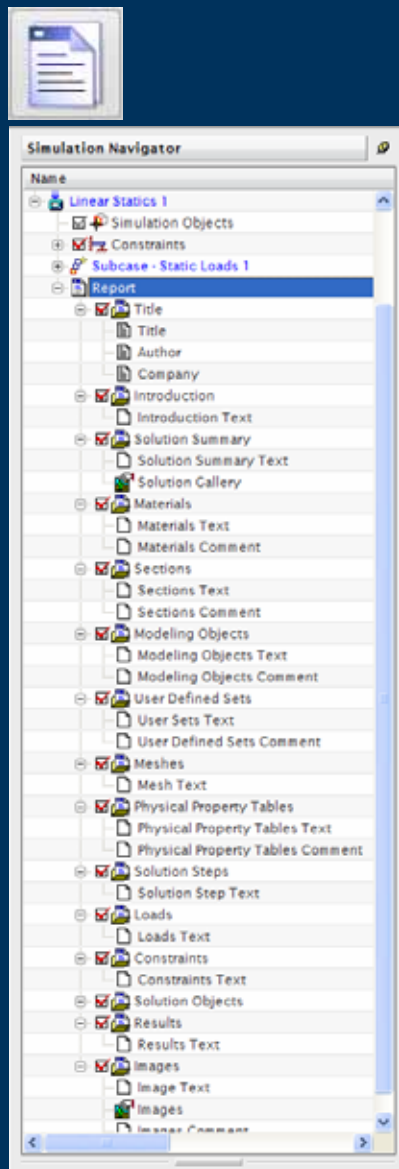


- ▶ Solution Comprehensive Check
- ▶ Warnings & Errors
  - ▶ Mesh
  - ▶ Materials
  - ▶ Boundary Conditions
  - ▶ Solution



## Solution – Report Before Solve

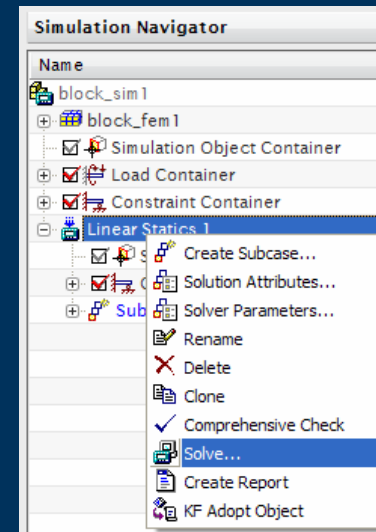
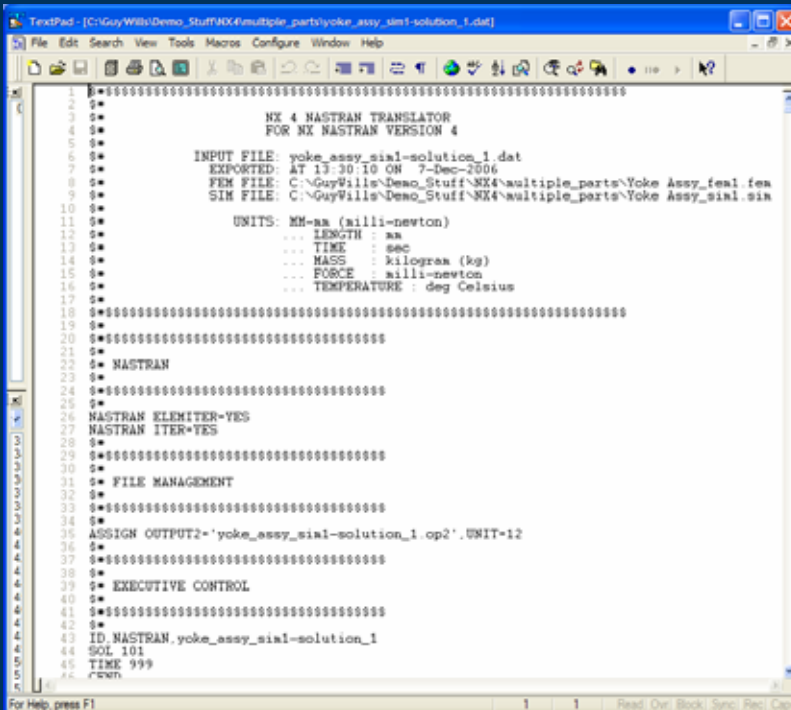
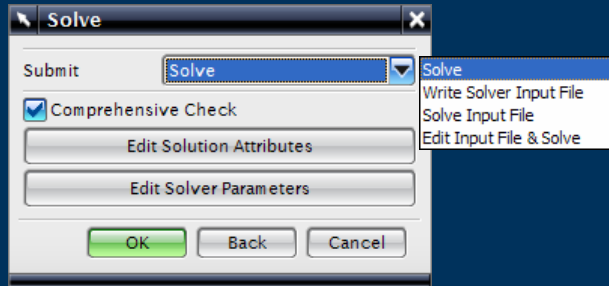
- ▶ Solution Report – Before Solve
  - ▶ User entered text description/documentation
  - ▶ Snapshot screen images
- ▶ HTML Interactive report export





# 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

# NX – Integrated Post Processing

Viewport Synchronization

Viewport Layout

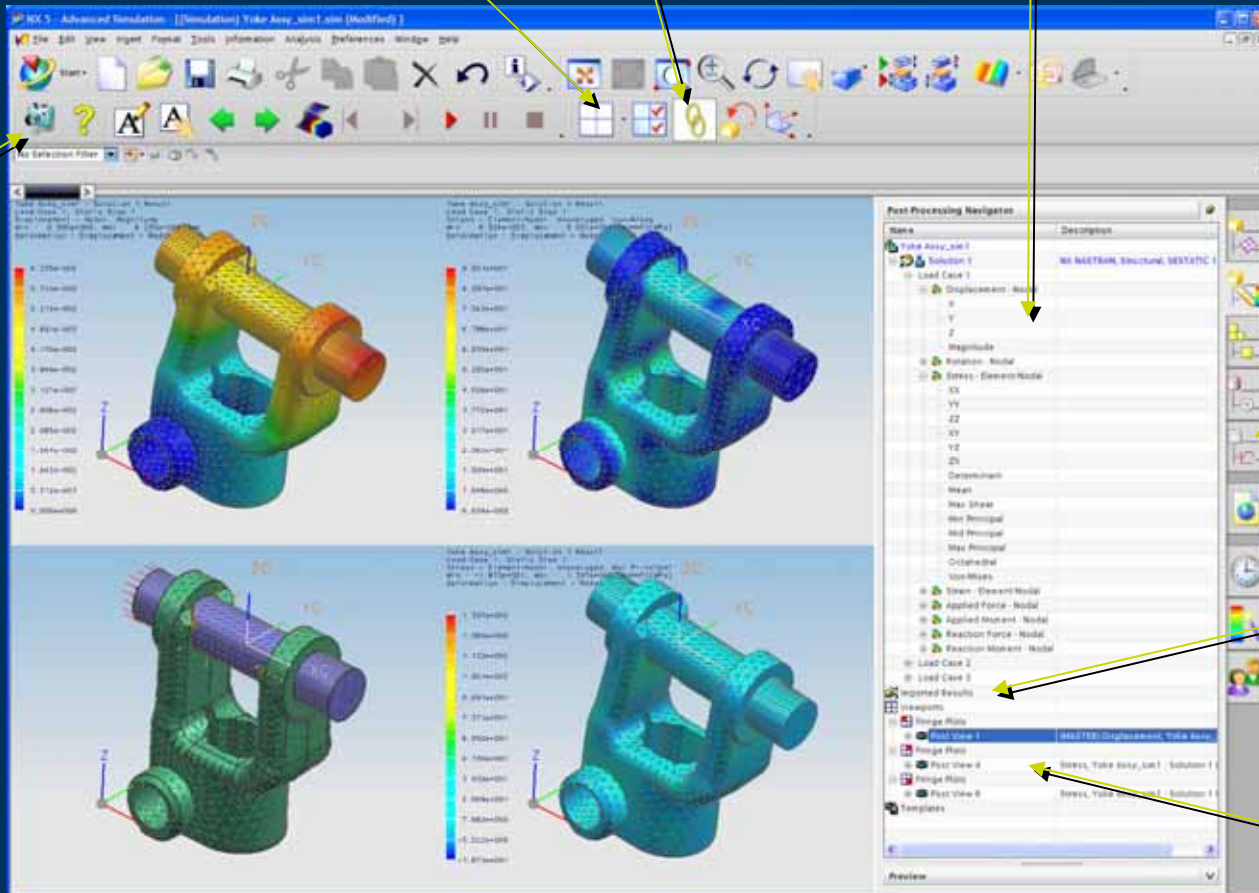
Easy selection of results

Dedicated Post Processing Navigator

Dedicated XY Graphing

Import of Existing Results for comparison

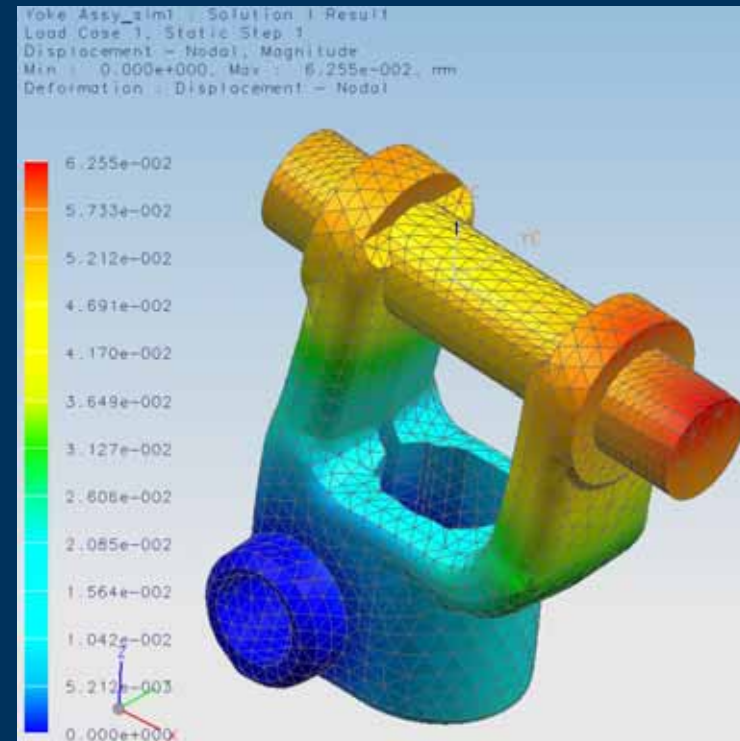
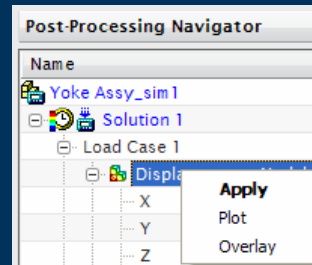
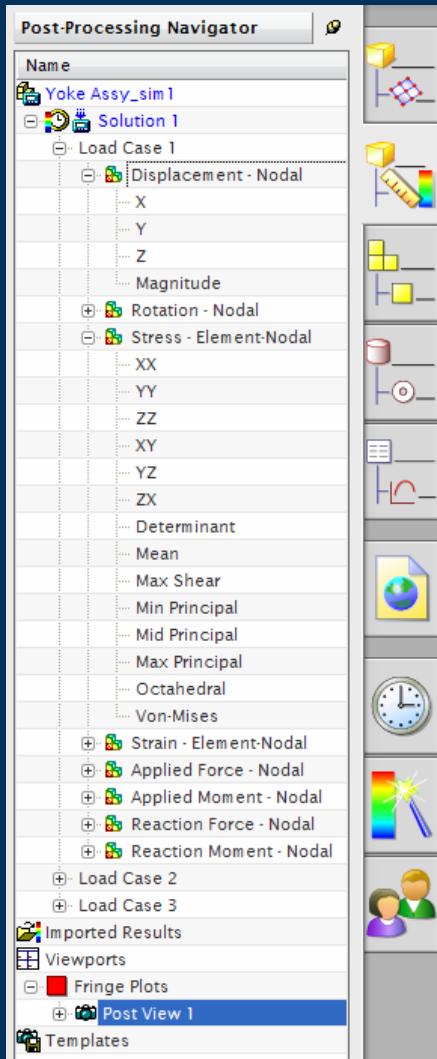
Post Views and Templates



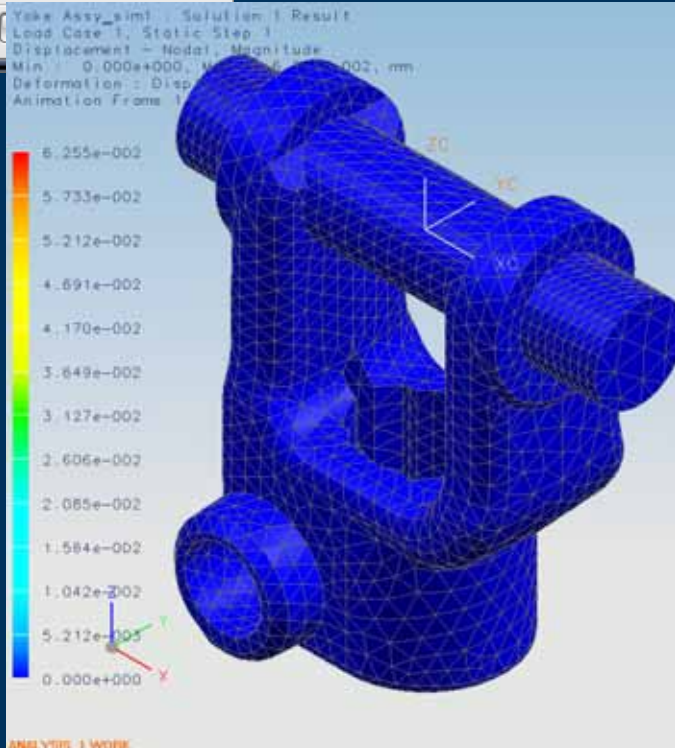
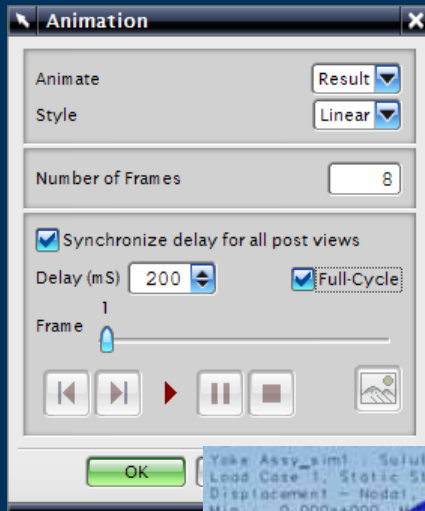
Post Processing Icons

## 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

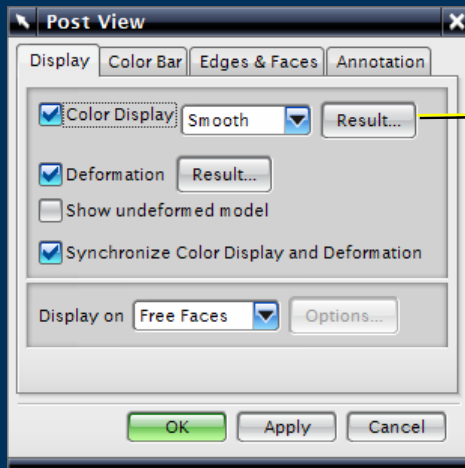
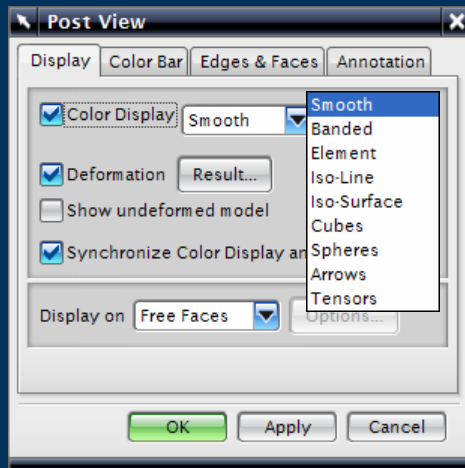
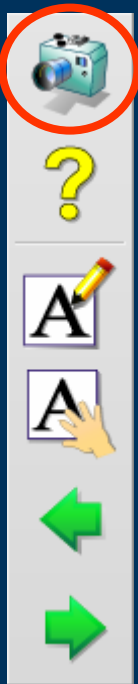


# Results – Animation

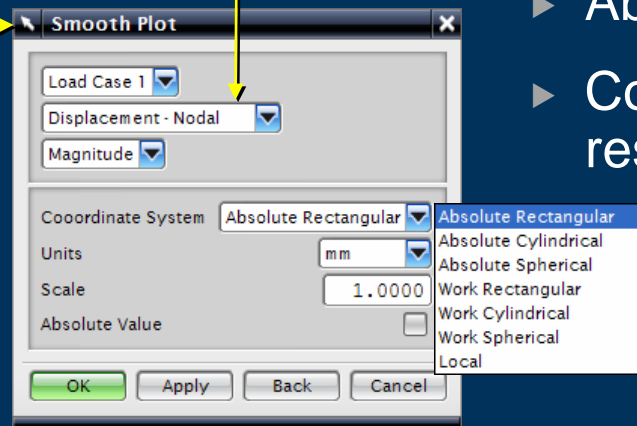


- ▶ Results Animation options
  - ▶ Number of Frames
  - ▶ Delay between frames
  - ▶ Full cycle (“forwards then backwards”)
  - ▶ Play, Pause, Stop
  - ▶ Single frame forwards or backwards
- ▶ During Animation
  - ▶ Change the Results
  - ▶ Screen rotate/pan/zoom
  - ▶ Toggle on/off Meshes
  - ▶ Save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

# Results – Post View Display



Result Set Selection



## ▶ Post View Display Options

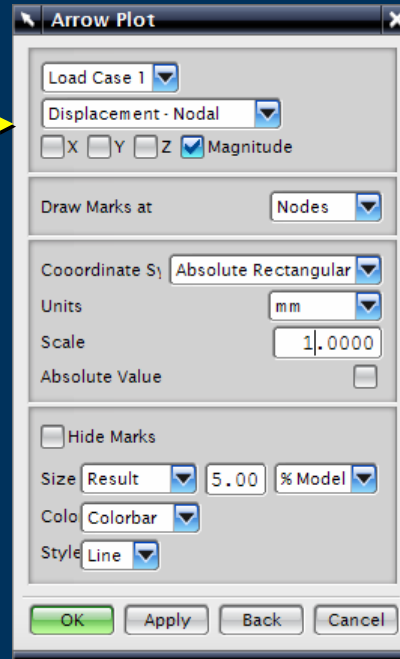
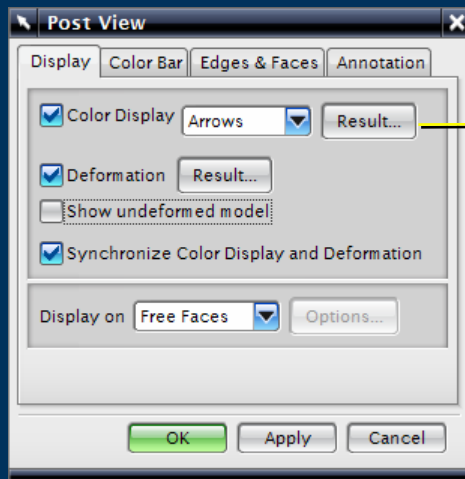
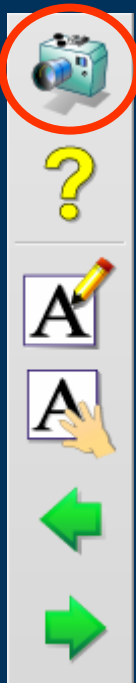
### ▶ Styles

- ▶ Contours
- ▶ Elements
- ▶ Isolines, Isosurfaces
- ▶ Mark – Arrow, Cube, Sphere, Tensor

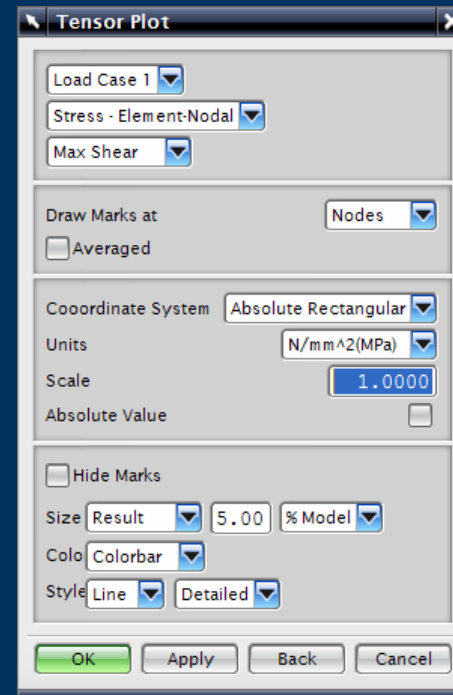
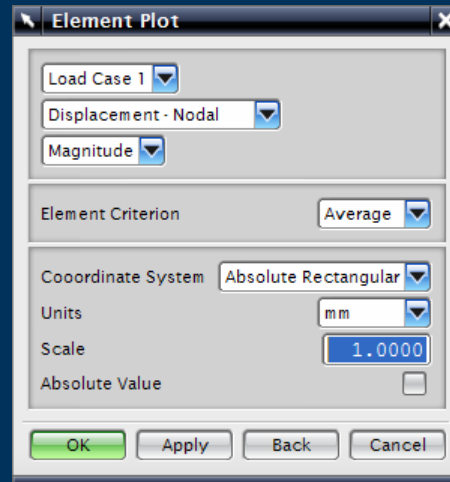
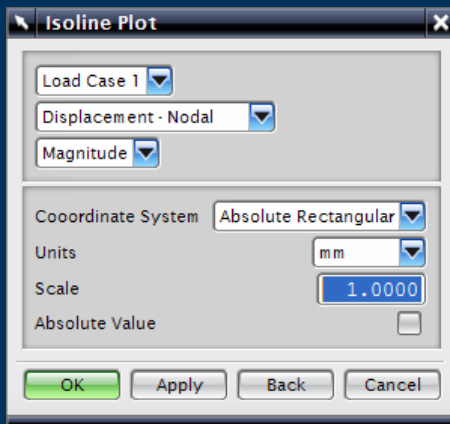
### ▶ Absolute or scaled values

### ▶ Coordinate System for results calculation

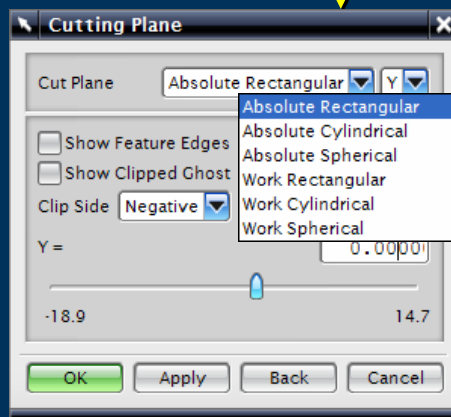
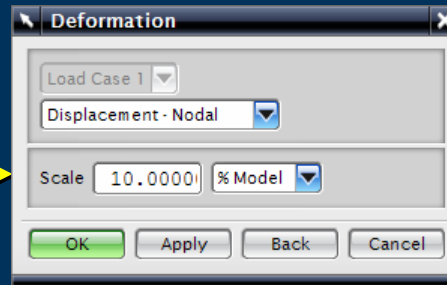
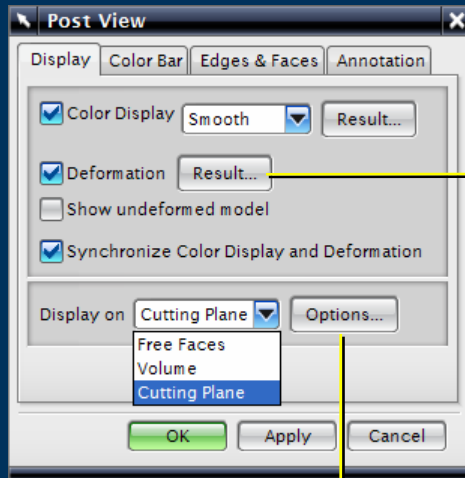
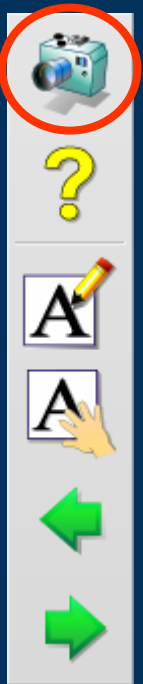
# Results – Post View Display



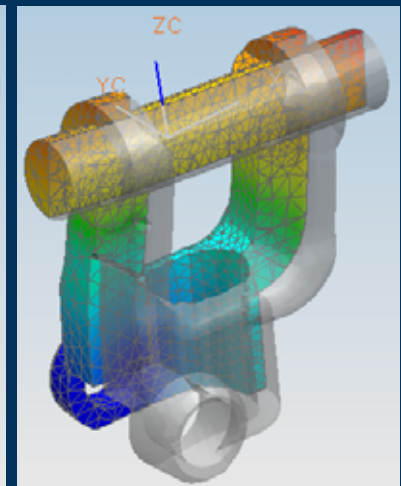
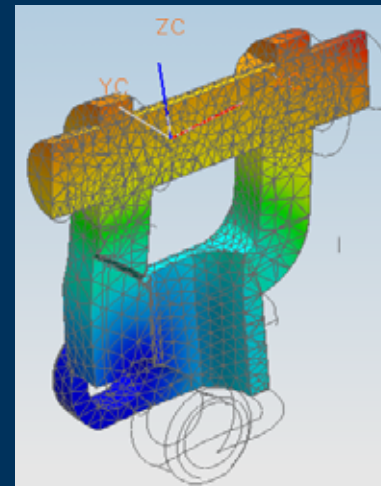
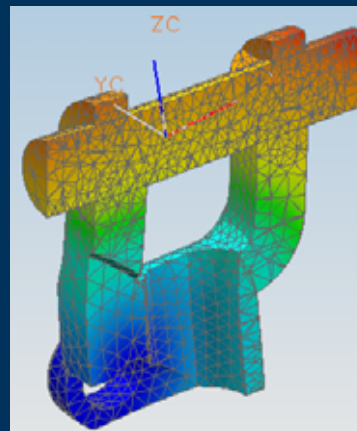
- ▶ Post View Display Options
- ▶ Results Plot options vary according to the Display type



# Results – Post View Display



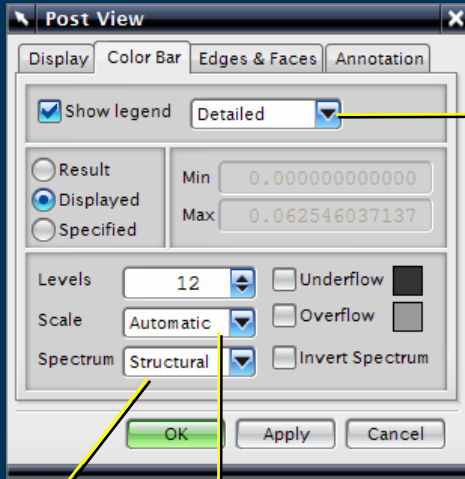
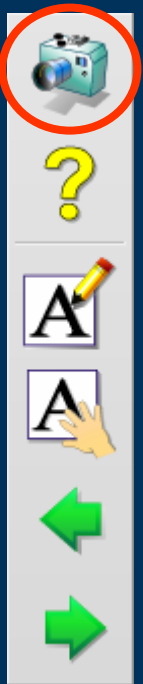
- ▶ Deformed model display
- ▶ Results domain
  - ▶ Free Face
  - ▶ Volume
  - ▶ Cutting Plane





# Results – Post View Color Bar

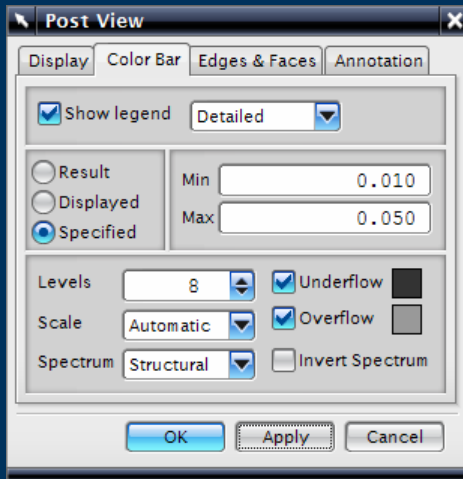
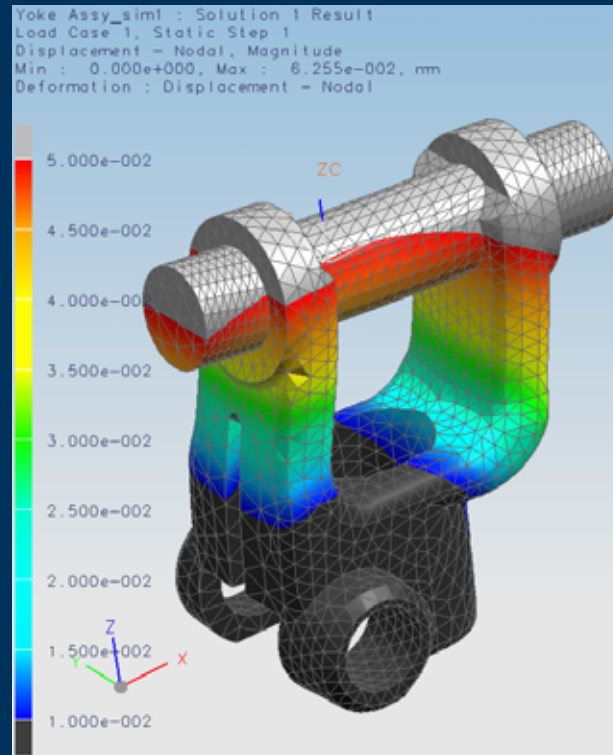
- ▶ Post View Color Bar
  - ▶ Max & Min values
  - ▶ Number of colors
  - ▶ Color scheme
  - ▶ Scale
  - ▶ Legend level of detail



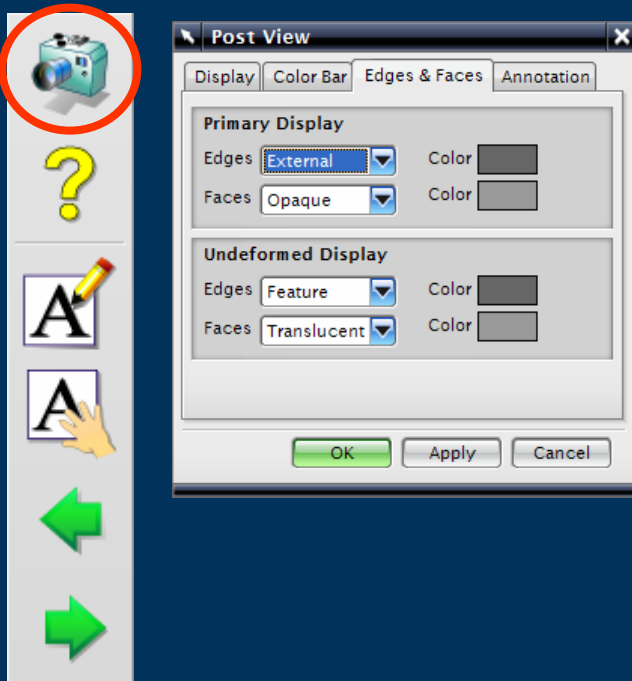
Detailed  
Colorbar Only  
Header Only

Structural  
Temperature  
Gray Scale  
Safety Factor

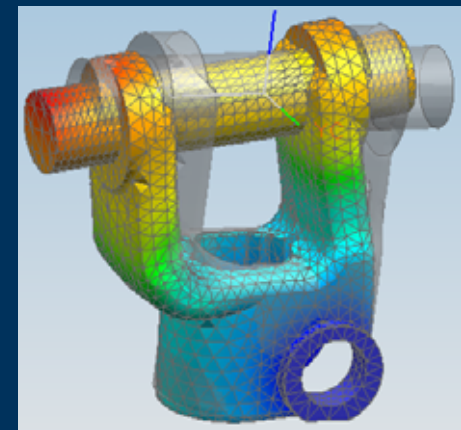
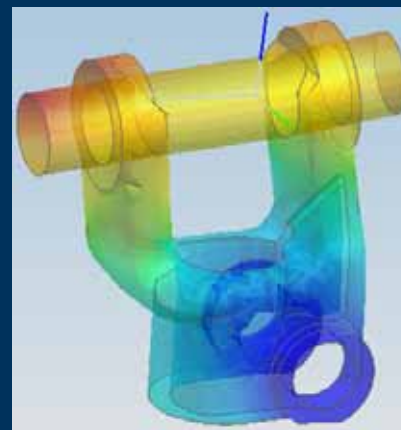
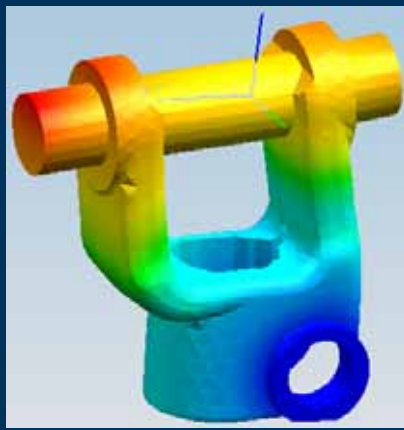
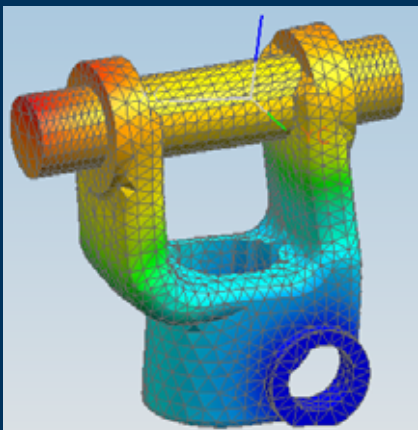
Automatic  
Linear  
Log



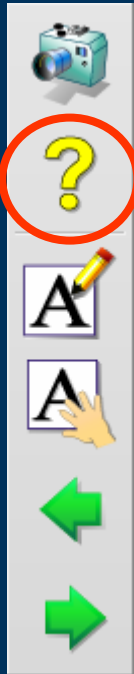
## Results – Post View Edges & Faces



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



# Results – Identify



**Identify**

Nodal Results: Pick from Model

Mark Selection: Mark Result Values

Pick: Single    Dimension: Any

Selection: 1 Item

Values	NodeID
Min 5.551e-002	22663
Max 0.000e+000	22663
Sum 5.551e-002	--

Close

Pick from Model

- Node IDs
- By Result Range
- N Max Result Values
- N Min Result Values

Any

- 1D
- 2D
- 3D

Single

- Mesh
- Feature Face
- Feature Edge

**Identify**

Nodal Results: Pick from Model

Mark Selection: Mark Result Values

Pick: Mesh    Dimension: Any

Selection: 8691 Items

Values	NodeID
Min 3.865e-002	133
Max 0.000e+000	23326
Sum 4.293e+002	--

**Identify**

Nodal Results: By Result Range

Mark Selection: Mark Result Values

Below 0.0000 mm

Above 0.0500 mm

Selection: 5258 Items

Values	NodeID
Min 5.000e-002	19275
Max 0.000e+000	23326
Sum 2.830e+002	--

**Identify**

Nodal Results: N Max Result Values

Mark Selection: Mark Result Values

N = 10

Selection: 10 Items

Values	NodeID
Min 6.211e-002	23345
Max 0.000e+000	23326
Sum 6.231e-001	--

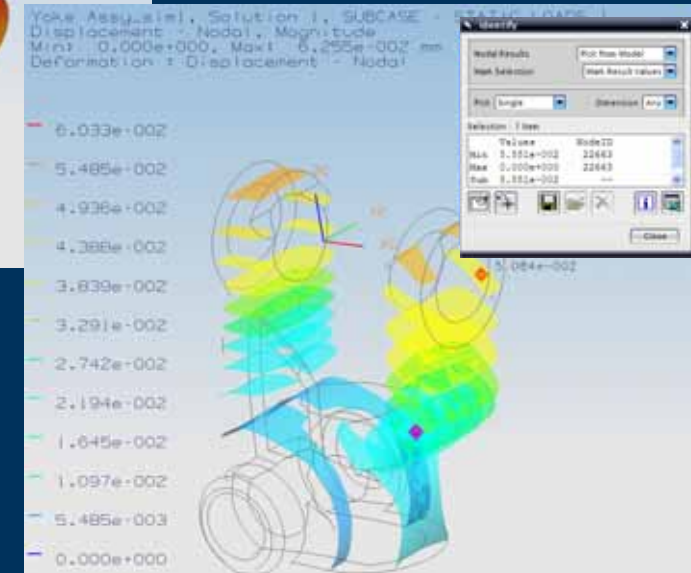
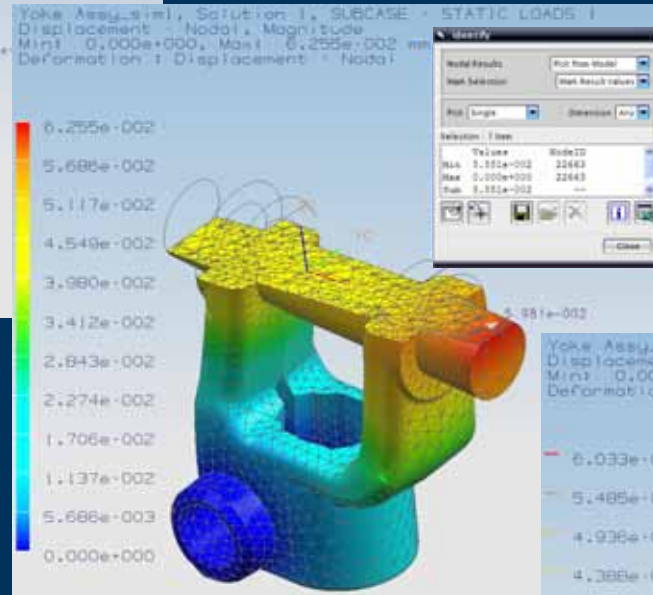
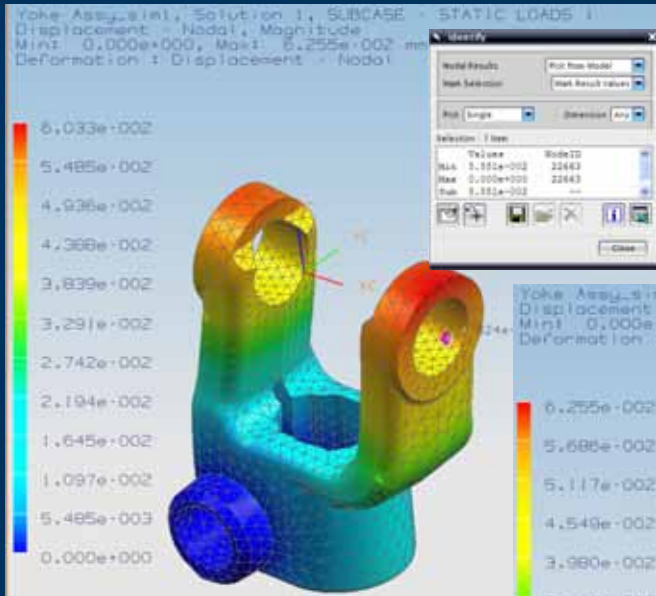
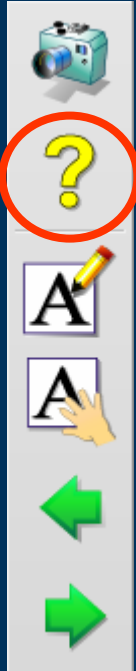
- ▶ **Identify** to probe and display nodal and elemental information
- ▶ Results
  - ▶ n Max, or n Min Result values
  - ▶ Result Range
- ▶ Selected data saved to Excel for further study

Microsoft Excel - Worksheet in IGS PostProcessor Results - Yoke Assy\_sim1.sim

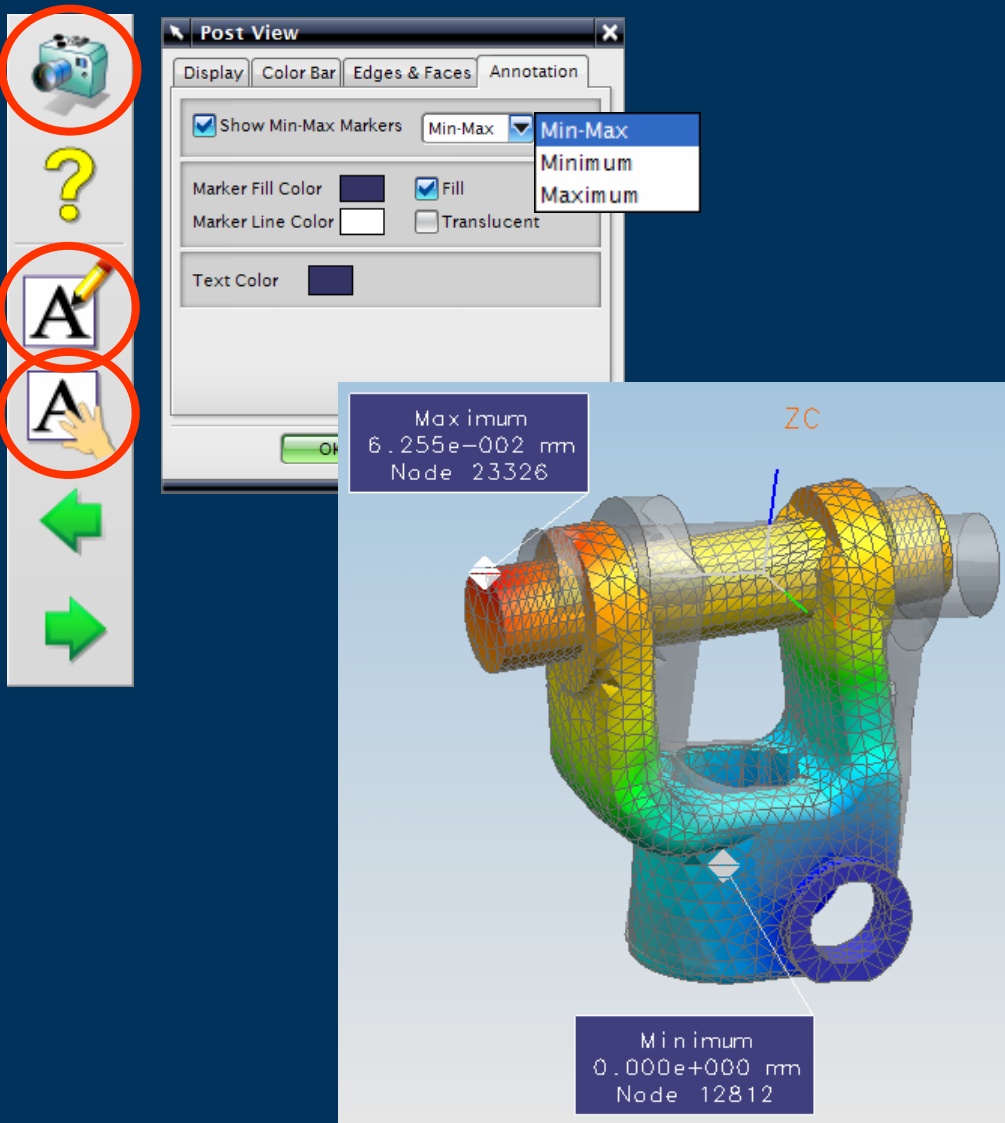
	A1	A2	A3	A4	A5	A6	A7	A8	A9	A10	A11	A12	A13	A14	A15	A16	A17	A18	A19	A20	A21	A22	A23	A24	A25
	B	C	D	E	F	G	H	I	J	K															
2	Result File	C:\Guy\Wills\Demo_Stuff\NX4\multiple_parts\yoke_assy_sim1-solution_1.op2																							
4	Load Case	SUBCASE - STATIC LOADS 1																							
5	Iteration/Model	SUBCASE - STATIC LOADS 1 Loads																							
6	Result	Displacement - Nodal																							
7	Units	mm																							
9	Node ID	X Coord	N Y Coord	Nodal	Z Coord	Nod X	Y	Z	Magnitude																
11	15107	21.688E+0	-1.529E+0	10.733E+0	54.182E-3	-2.673E-6	-26.374E-3	60.260E-3																	
12	15108	21.688E+0	1.502E+0	10.738E+0	54.188E-3	-31.219E-6	-26.376E-3	60.267E-3																	
13	15118	21.688E+0	-3.026E-3	10.431E+0	53.944E-3	10.559E-6	-26.369E-3	60.044E-3																	
14	15125	21.688E+0	-14.089E-3	10.835E+0	54.264E-3	-16.833E-6	-26.377E-3	60.335E-3																	
15	15126	21.688E+0	2.998E+0	10.441E+0	53.957E-3	44.811E-6	-26.373E-3	60.057E-3																	
16	17149	32.667E+0	-3.281E+0	4.655E+0	49.335E-3	-28.664E-6	-35.337E-3	60.685E-3																	
17	17150	32.667E+0	-4.027E+0	3.990E+0	48.795E-3	-28.881E-6	-35.337E-3	60.247E-3																	
18	17176	32.021E+0	-3.318E+0	4.629E+0	49.315E-3	27.952E-6	-34.813E-3	60.365E-3																	
19	18124	30.839E+0	-1.631E+0	5.415E+0	49.954E-3	-26.696E-6	-33.854E-3	60.345E-3																	
20	18125	30.193E+0	-1.671E+0	5.401E+0	49.943E-3	-26.019E-6	-33.329E-3	60.043E-3																	
21	18126	30.839E+0	-643.315E-3	5.568E+0	50.079E-3	-26.929E-6	-33.854E-3	60.449E-3																	
22	18132	30.193E+0	312.674E-3	5.444E+0	49.979E-3	-26.818E-6	-33.330E-3	60.074E-3																	
23	18133	32.021E+0	1.624E+0	5.617E+0	50.121E-3	-28.246E-6	-34.816E-3	61.026E-3																	
24	18134	30.193E+0	-681.138E-3	5.566E+0	50.078E-3	-26.342E-6	-33.330E-3	60.155E-3																	
25	19507	32.667E+0	-2.377E+0	5.081E+0	49.682E-3	-28.555E-6	-35.338E-3	60.968E-3																	

## Results – Identify

- ▶ Identify works on all types of Results display

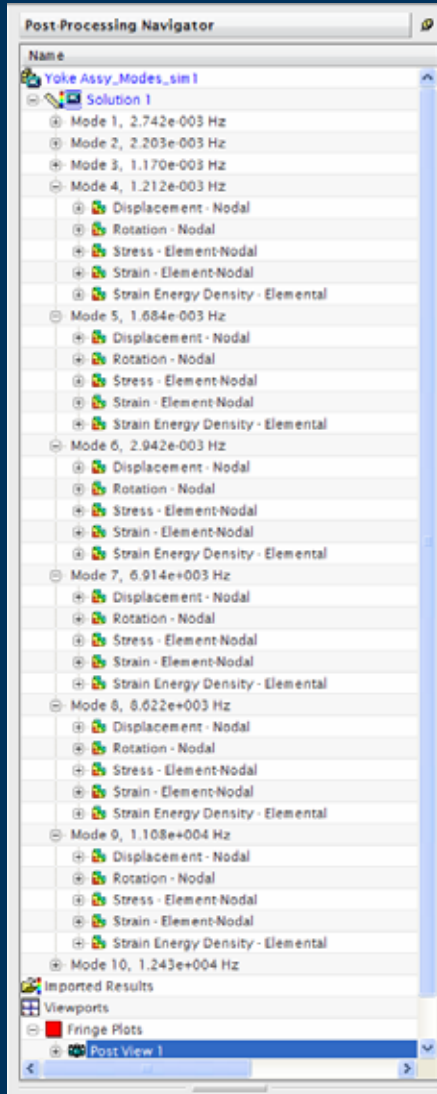
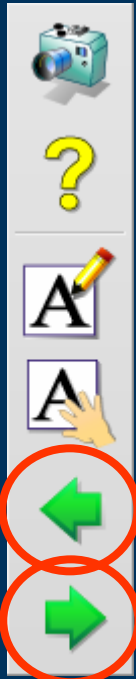


## Results – Annotation Markers

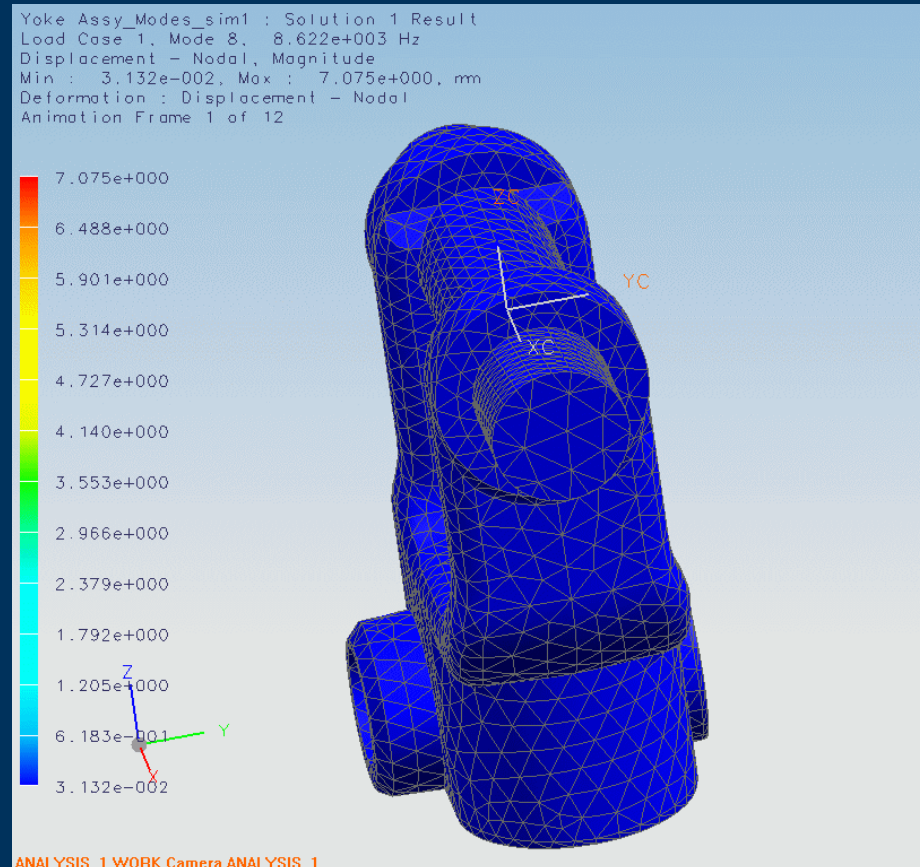


- ▶ Results Marker shows max/min of the current active Result Set
  - ▶ Max & Min
  - ▶ Max only
  - ▶ Min only
- ▶ Drag Marker to reposition

# Results – Previous / Next Mode or Iteration

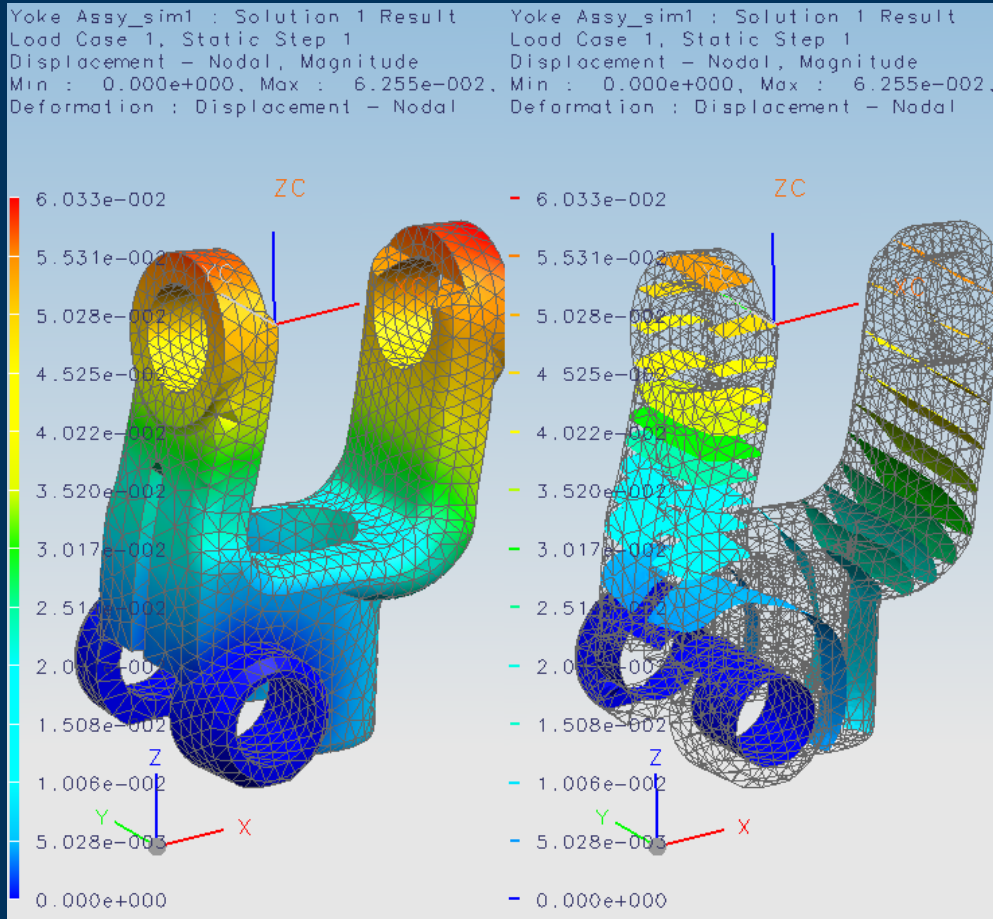


► Quick change to the next Mode or Iteration



ANALYSIS\_1 WORK Camera ANALYSIS\_1

# Results – Post Views & Templates

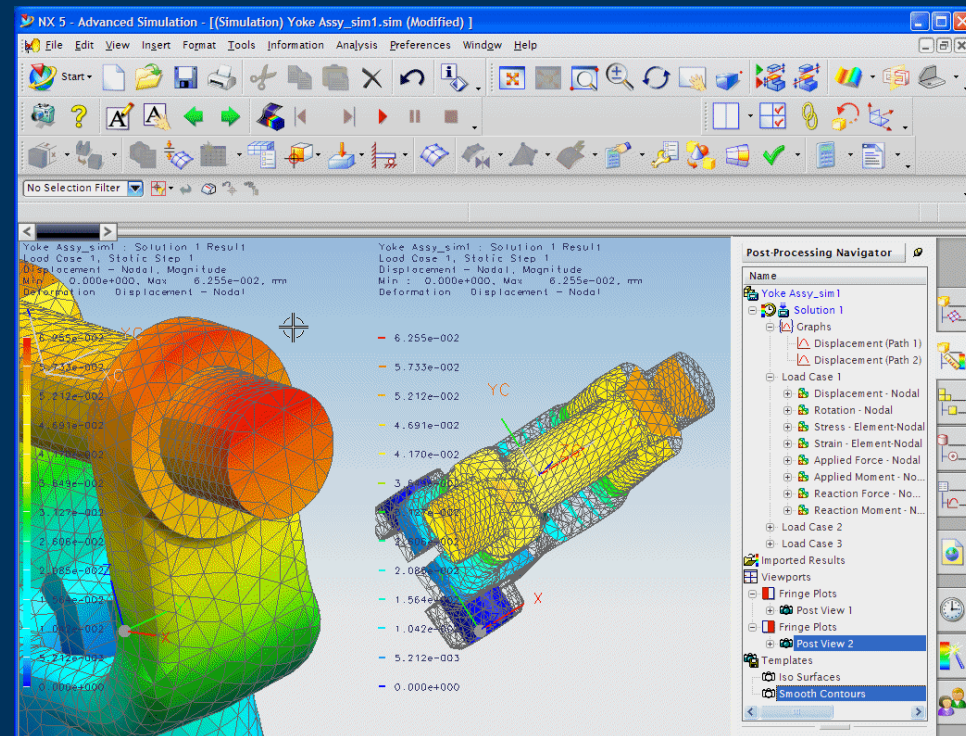
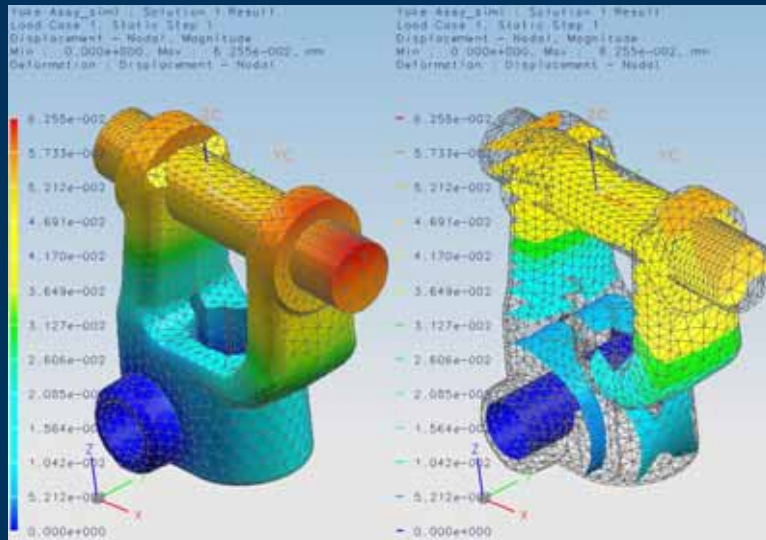


- ▶ Post Views store the Post View setup
- ▶ Provides quick and efficient method of controlling displays in different Views
- ▶ Post Views can be saved and are available for re-use

Name	Description
Yoke Assy_sim1	
Solution 1	NX NASTRAN, Structural
Imported Results	
Viewports	
Fringe Plots	
Post View 1	Displacement, Yoke Ass
3D Elements	
3d_mesh(1)	Material ID = 1, Property
3d_mesh(2)	Material ID = 2, Property
Fringe Plots	
Post View 2	(MASTER) Displacement
3D Elements	
3d_mesh(1)	Material ID = 1, Property
3d_mesh(2)	Material ID = 2, Property
Templates	
Smooth Contours	
Iso Surfaces	

# Results – Multiple Viewports

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

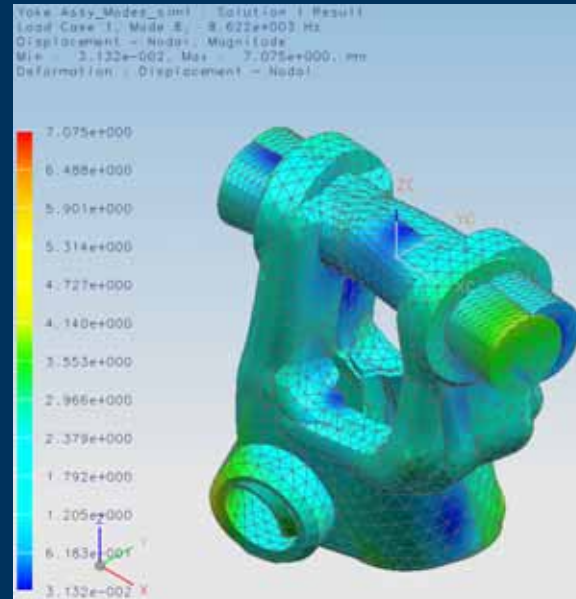
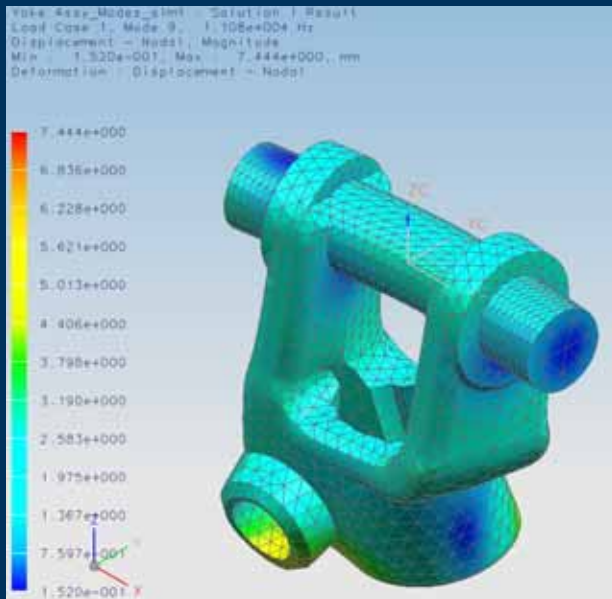




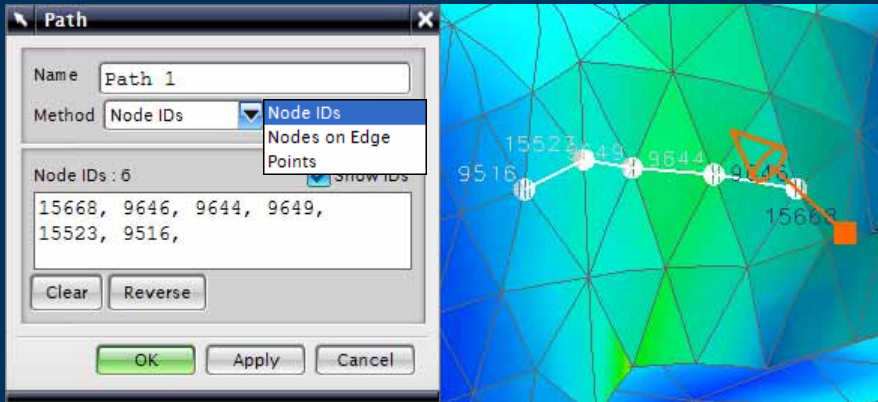
# Results – Post View Overlay

Mode 8, 8.622e+003 Hz	
Displacement - Nodal	
Rotation - Nodal	
Stress - Element-Nodal	
Strain - Element-Nodal	
Strain Energy Densit...	
Mode 9, 1.108e+004 Hz	
Displacement - Nodal	
Rotation - Nodal	
Stress - Element-Nodal	
Strain - Element-Nodal	
Strain Energy Densit...	
Mode 10, 1.243e+004 Hz	
Imported Results	
Viewports	
Fringe Plots	
Post View 2	Displacement, Yoke Assy_Modes...
Post View 4	(MASTER) Displacement, Yoke Ass...

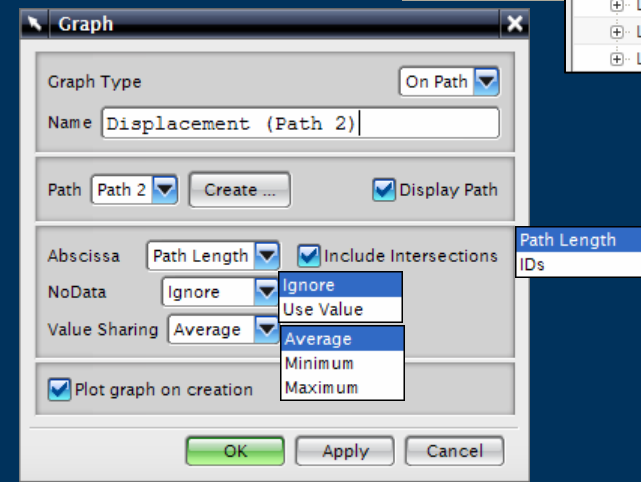
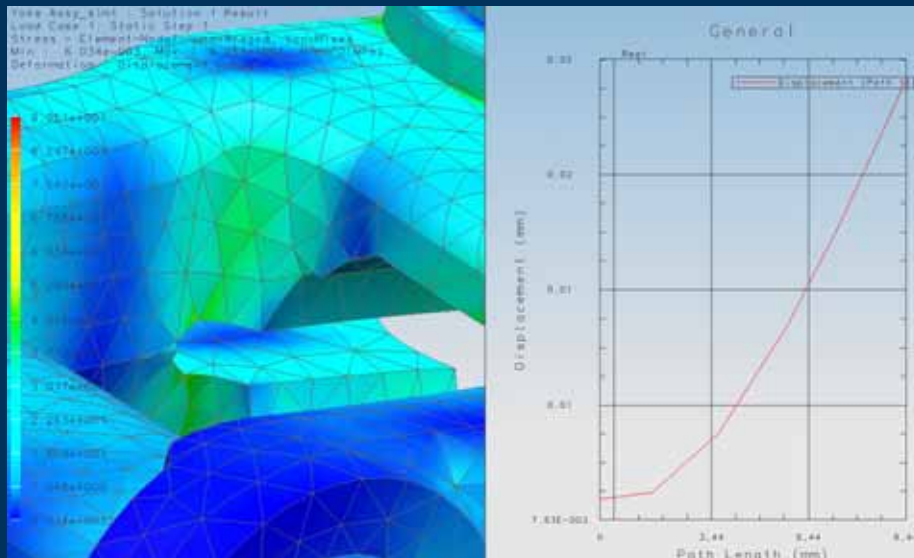
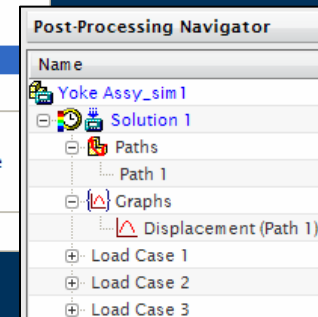
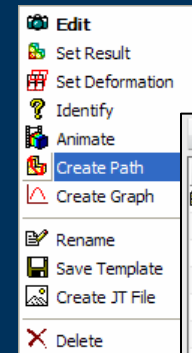
- ▶ Post View Overlay
- ▶ Post View changes can be saved to all or selected Post Views



# Plotting Paths

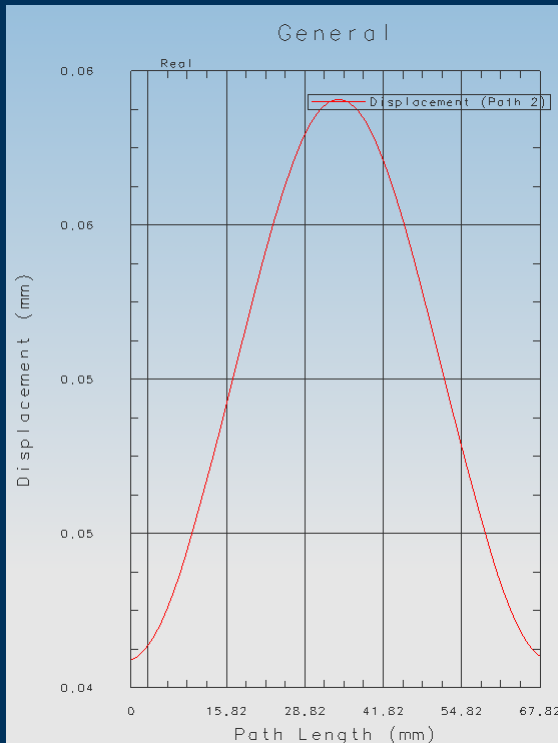


- ▶ Paths defined by
  - ▶ Node labels
  - ▶ Screen selection
  - ▶ Edge selection

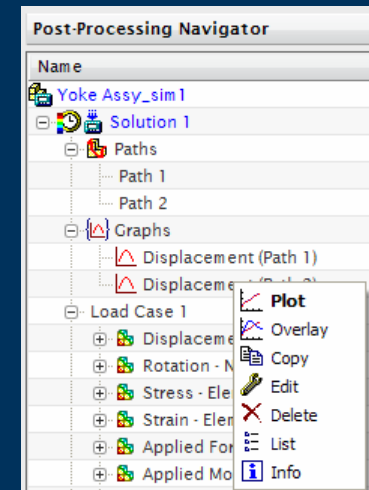


# Plotting Paths

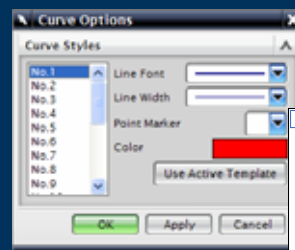
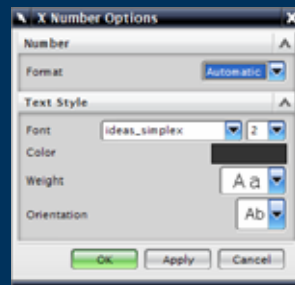
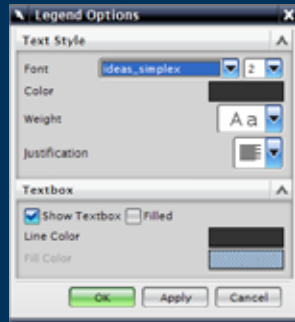
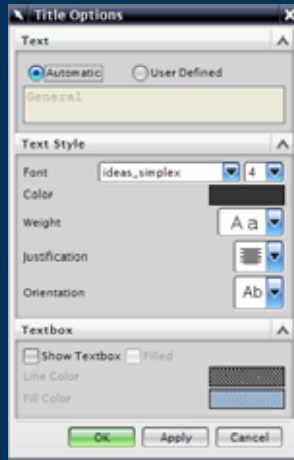
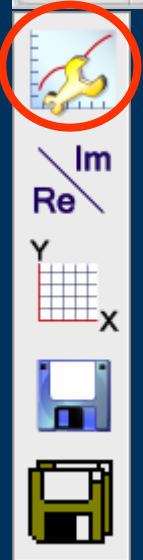
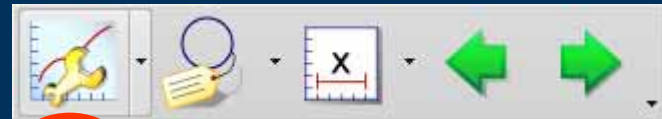
- ▶ Graphs can be Overlaid to compare data using same Axis
- ▶ Path and Graph data stored in external file (\*.afu)



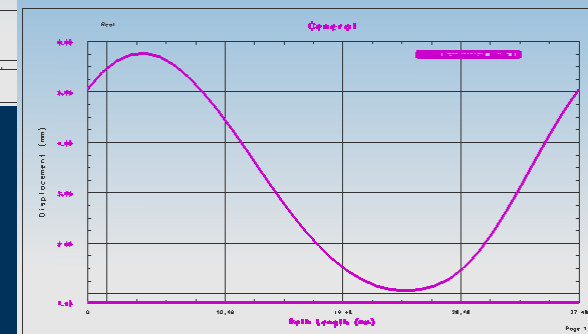
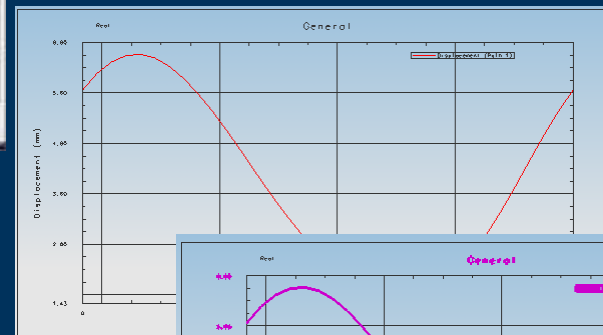
Name	Type
yoke_assy_sim1-solution_1.dat	DAT File
yoke_assy_sim1-solution_1.diag	DIAG File
yoke_assy_sim1-solution_1.f04	F04 File
yoke_assy_sim1-solution_1.f06	F06 File
yoke_assy_sim1-solution_1.log	Text Document
yoke_assy_sim1-solution_1.op2	OP2 File
yoke_assy_sim1-solution_1_PostGraphs.afu	AFU File



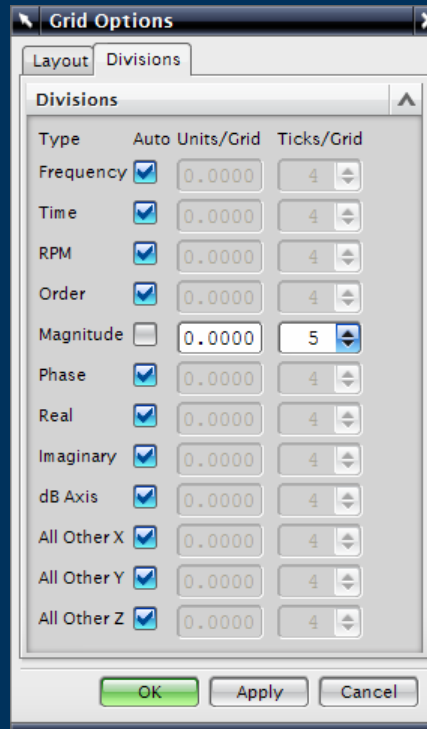
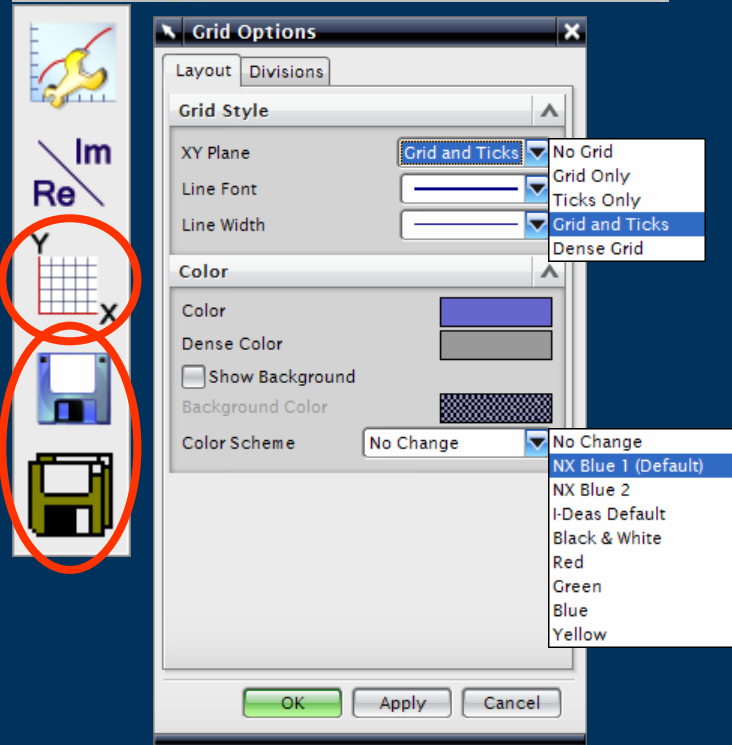
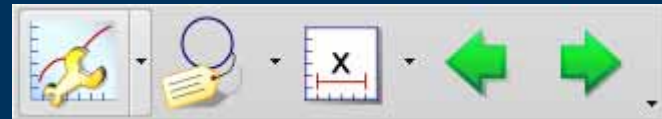
## Graph Style



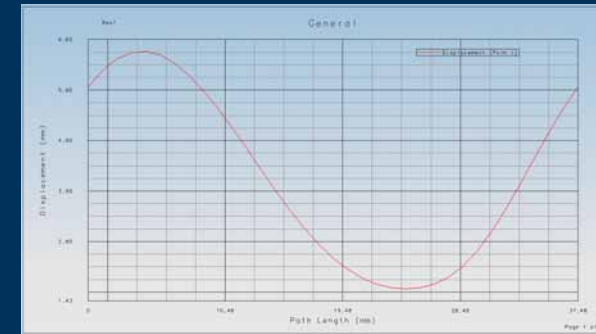
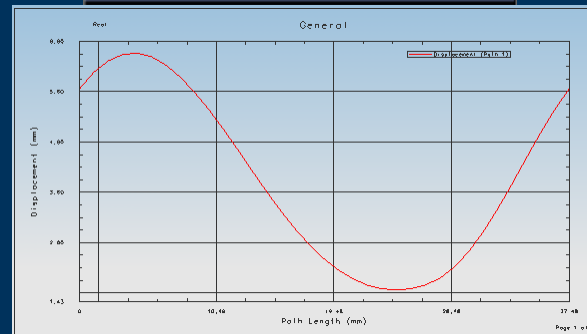
- ▶ Edit Graph Style
  - ▶ Dynamic Selection of Graph element
  - ▶ Graph Title
  - ▶ Graph Legend
  - ▶ Axis style
    - ▶ Axis Labels
    - ▶ Axis Numbers
    - ▶ Axis Type, Values & Units
  - ▶ Curve style



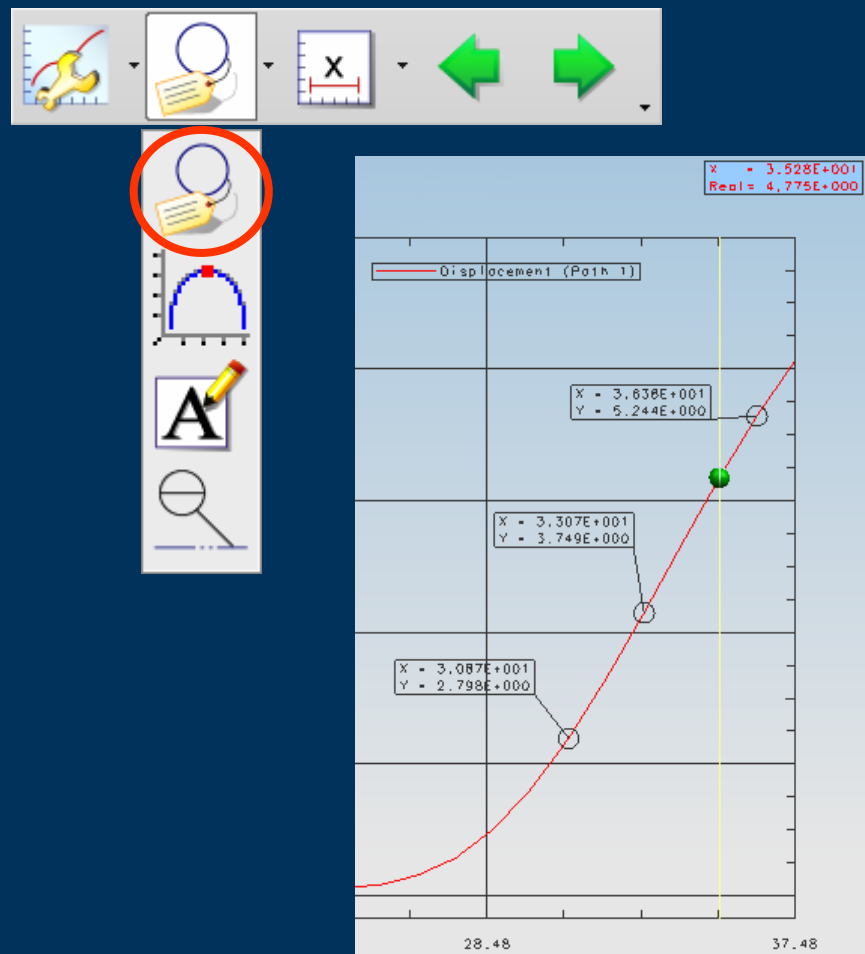
## Graph Style



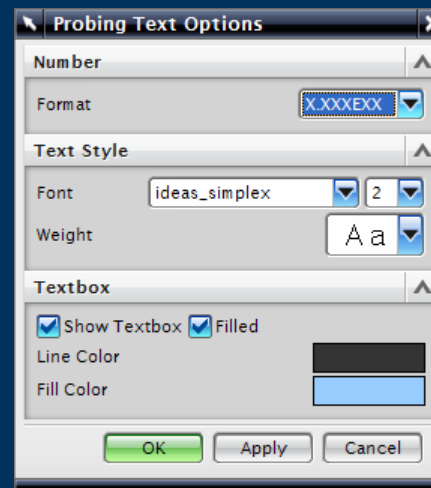
- ▶ Graph Grid Style
  - ▶ Grid Layout
  - ▶ Divisions
- ▶ Save the current Graph Settings to the current Template File
- ▶ Save the current Graph Setting to a new Template File



# Graph Probing

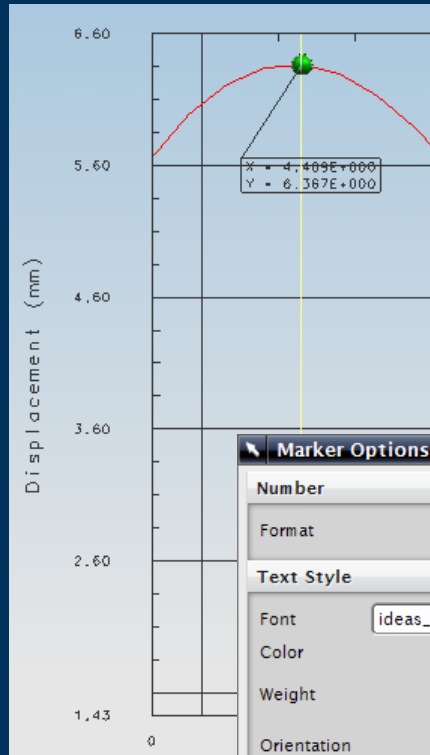
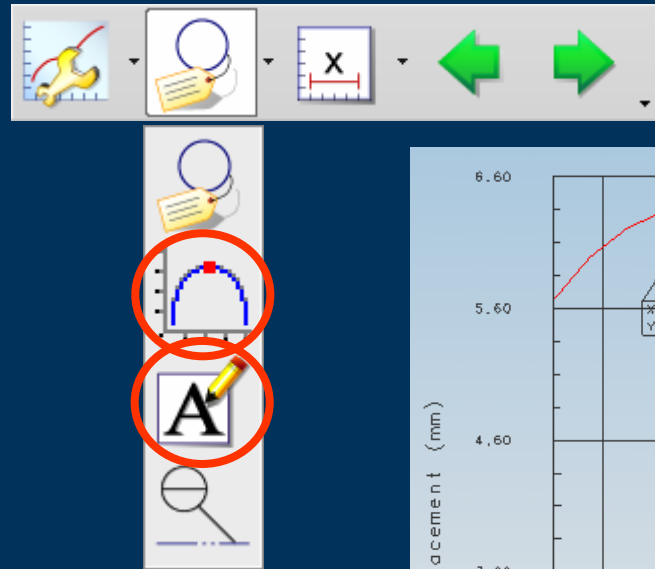


- ▶ Data Point Probing and Marking
- ▶ Dynamic Display of curve data points & values
- ▶ Specific curve location values
- ▶ Probe Text Styles



## Graph Probing

- ▶ Peak and Valley locating and Marking



**Marker Options**

Number:

Format:

Text Style

Font:

Color:

Weight:

Orientation:

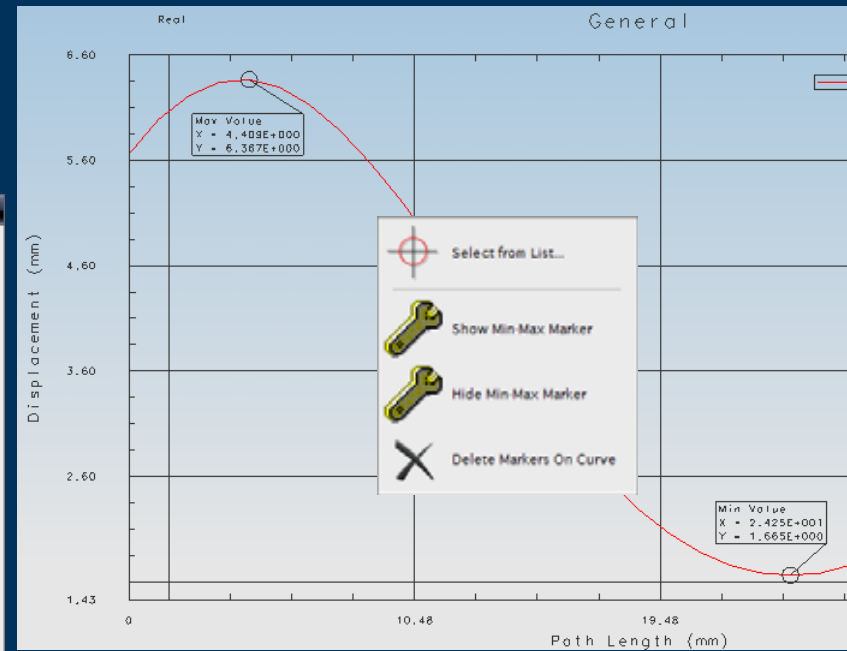
Textbox

Show Textbox  Filled

Line Color:

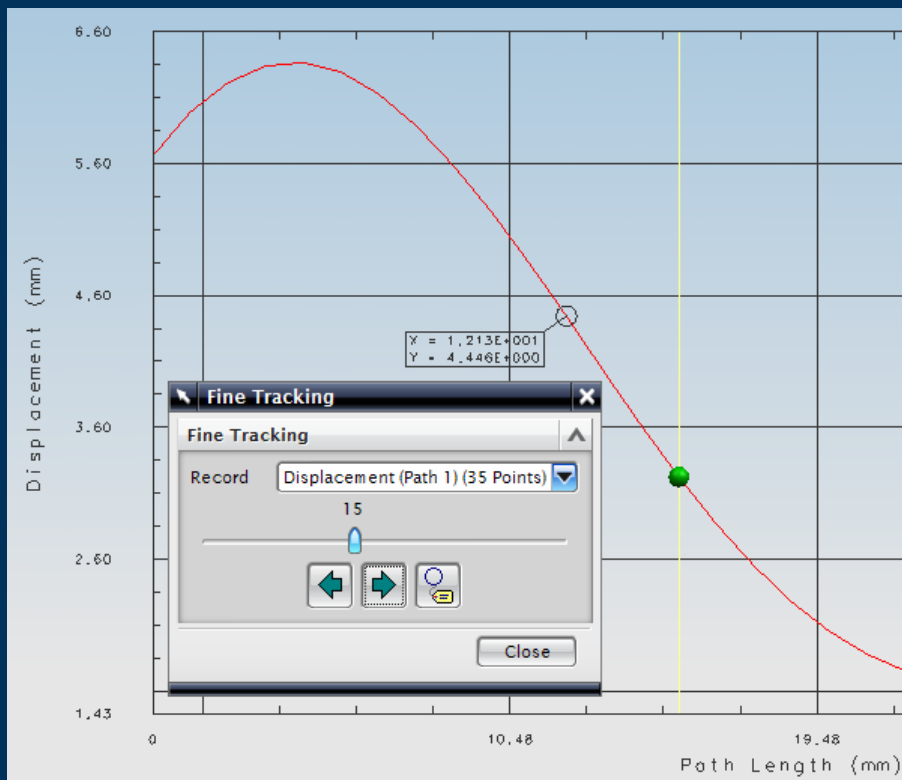
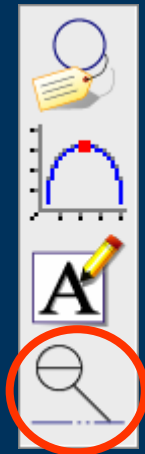
Fill Color:

OK Apply Cancel



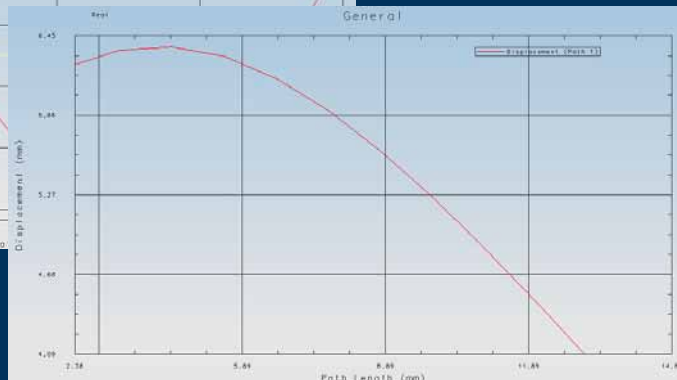
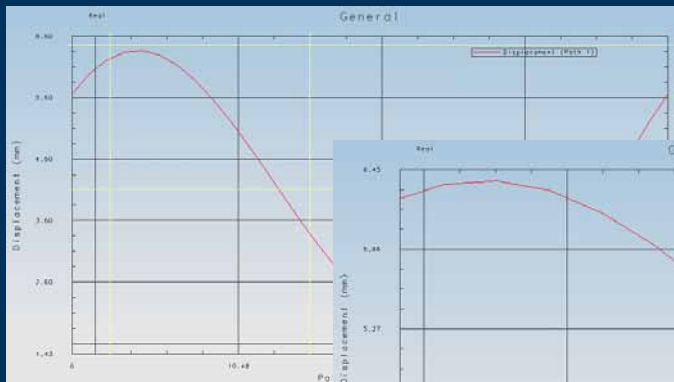
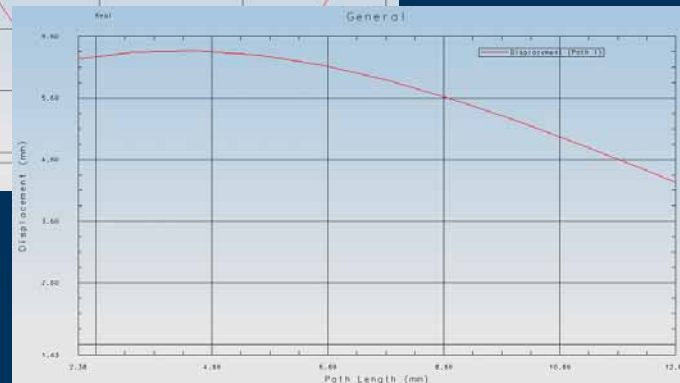
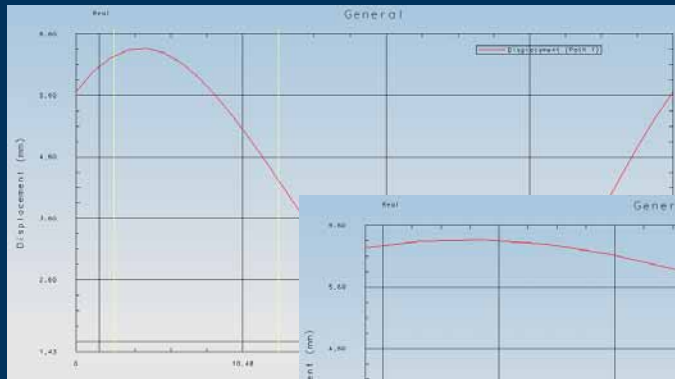
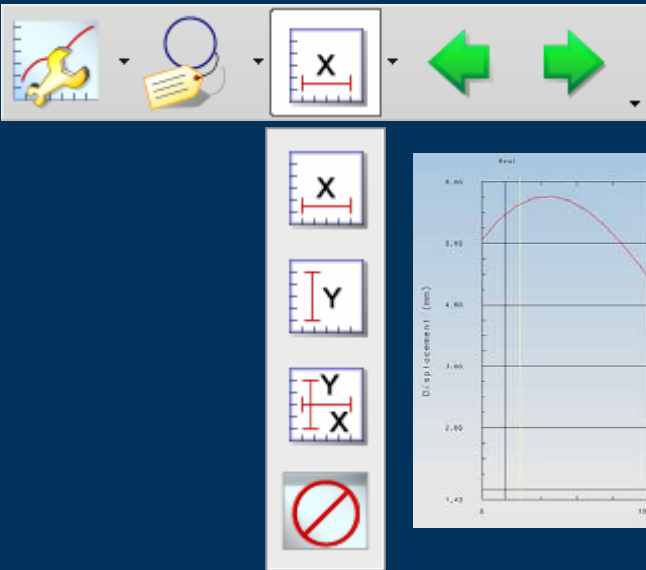
# Graph Probing

► Fine Tracking and Marking





## Graph Windowing



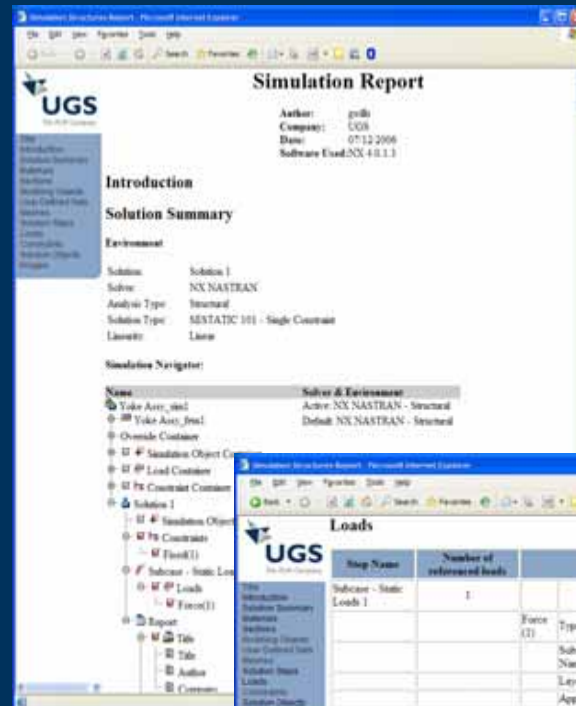
- ▶ Window into the Graph
- ▶ X Window
- ▶ Y Window
- ▶ X-Y Window
- ▶ No Window to return to full graph

## Solution Report – After Solve

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

**Simulation Navigator**

Name	
Yoke Assy_Modes_sim1	
Yoke Assy_fem1	
Override Container	
Simulation Object Container	
Load Container	
Constraint Container	
<b>Solution 1</b>	
Simulation Objects	
Constraints	
Results	
<b>Report</b>	
Title	
Title	
Author	
Company	
Introduction	
Introduction Text	
Solution Summary	
Solution Summary Text	
Solution Gallery	
Materials	
Materials Text	
Materials Comment	
Sections	
Sections Text	
Sections Comment	
Meshes	
Mesh Text	
Solution Steps	
Solution Step Text	
Constraints	
Constraints Text	
Results	
Results Text	
Response Simulation Text	
Images	
Image Text	
Images	
Images Comment	

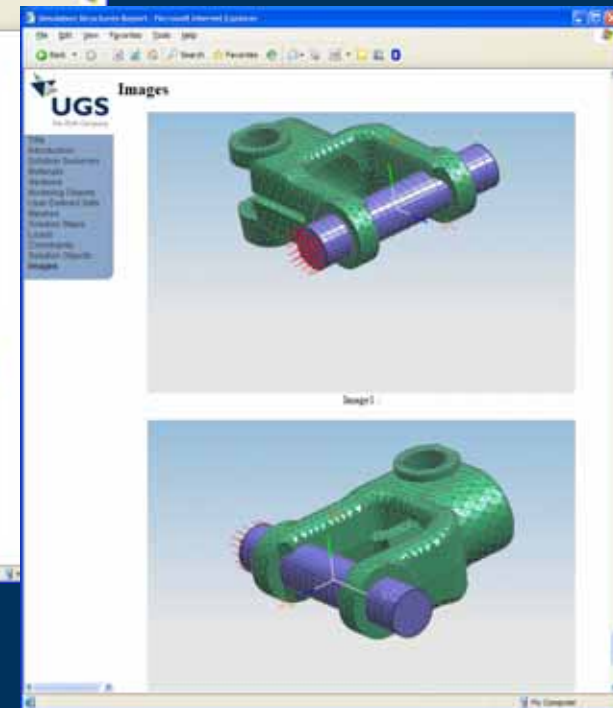


**Loads**

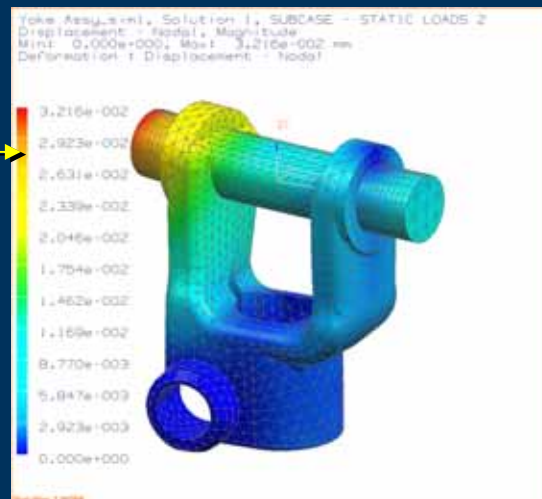
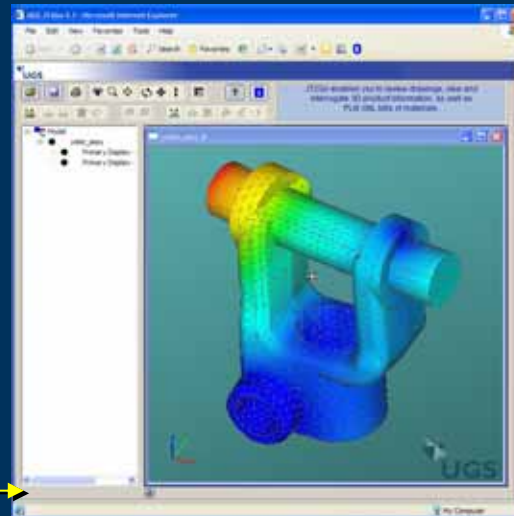
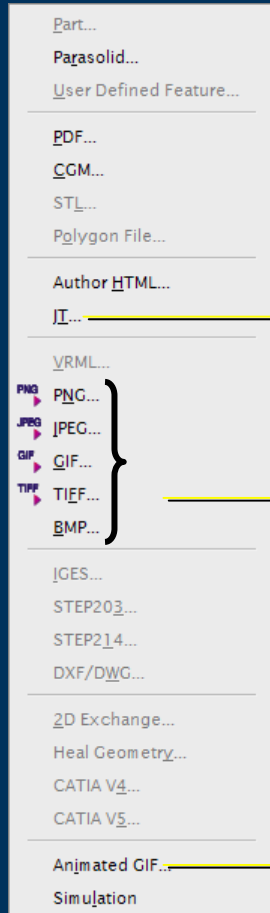
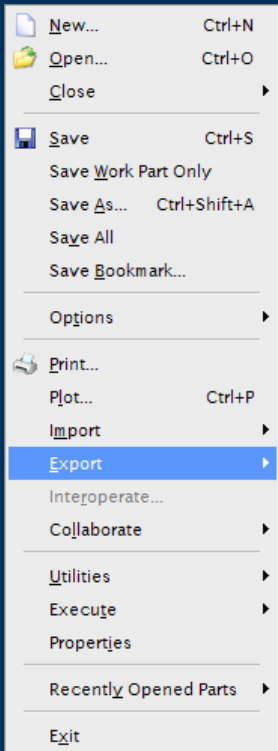
Step Name	Number of referenced loads	Loads	
Solution - Static Loads 1	1	Force (1)	Force - Normal Force
		Solver Card Name	FORCE
		Layer	1
		Applied to	1 Polygon Face
		Description	Force - 1000 N
		Distribution	Geometric
		Type	Illustrative

**Constraints**

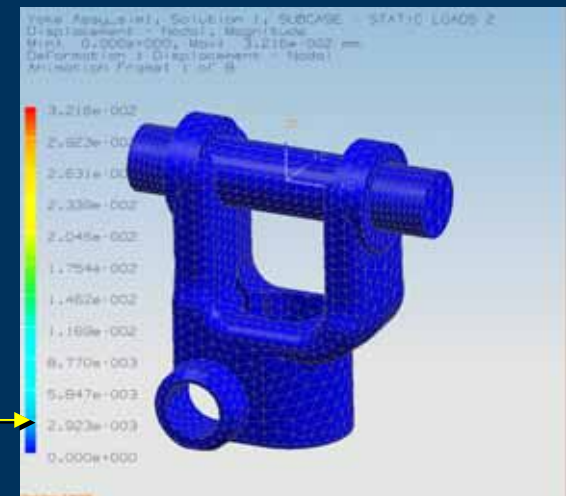
Step Name	Number of referenced constraints	Constraints	
Solution - Static Loads 1	1	Fixed (1)	Fixed - Fixed constraint
		Solver Card Name	SPC
		Layer	1
		Applied to	2 Polygon Face
		Description	



## Export Visualisation Files



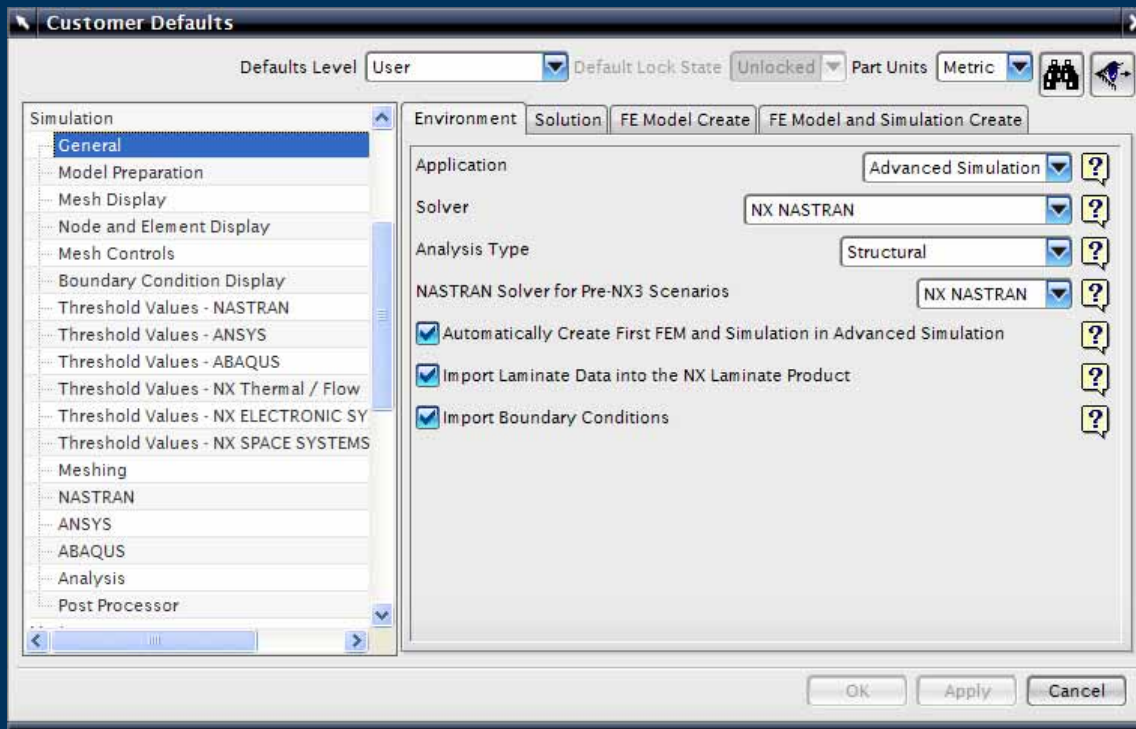
- ▶ Direct export of JT visualisation model
- ▶ Direct export of screen images
  - ▶ PNG
  - ▶ JPEG
  - ▶ GIF
  - ▶ TIFF
  - ▶ BMP
- ▶ Direct export of Animation
  - ▶ Animated GIF



**SIEMENS**

# Customer Defaults for Simulation

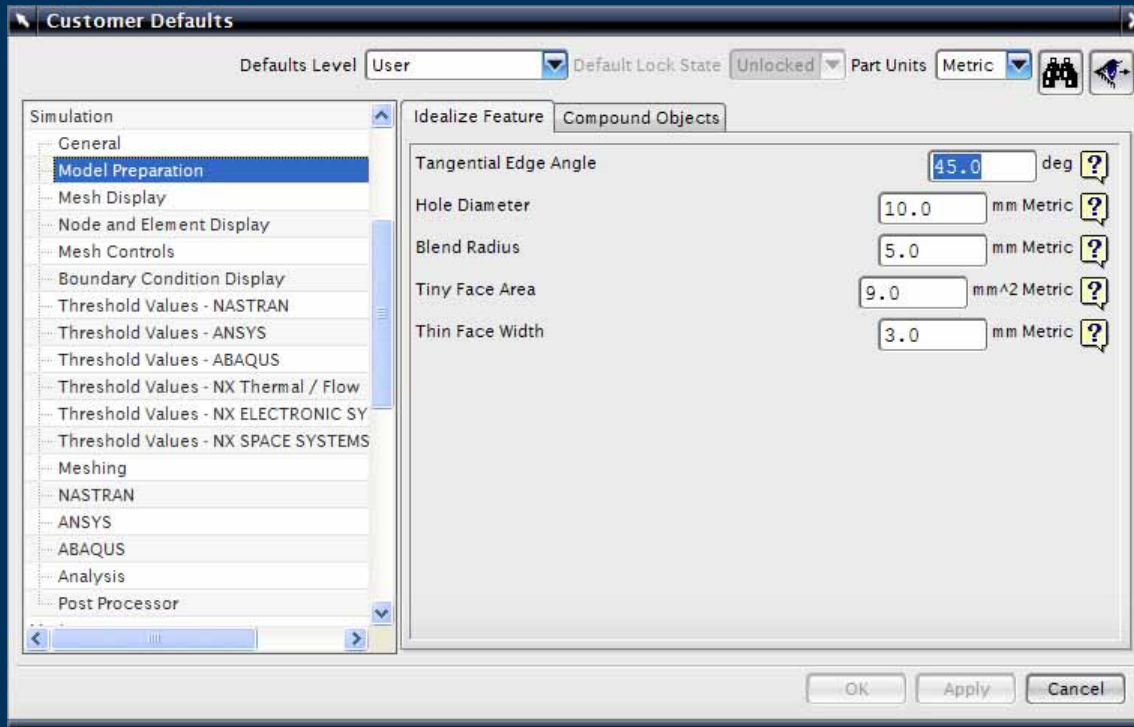
# Customer Defaults – General



## ▶ General

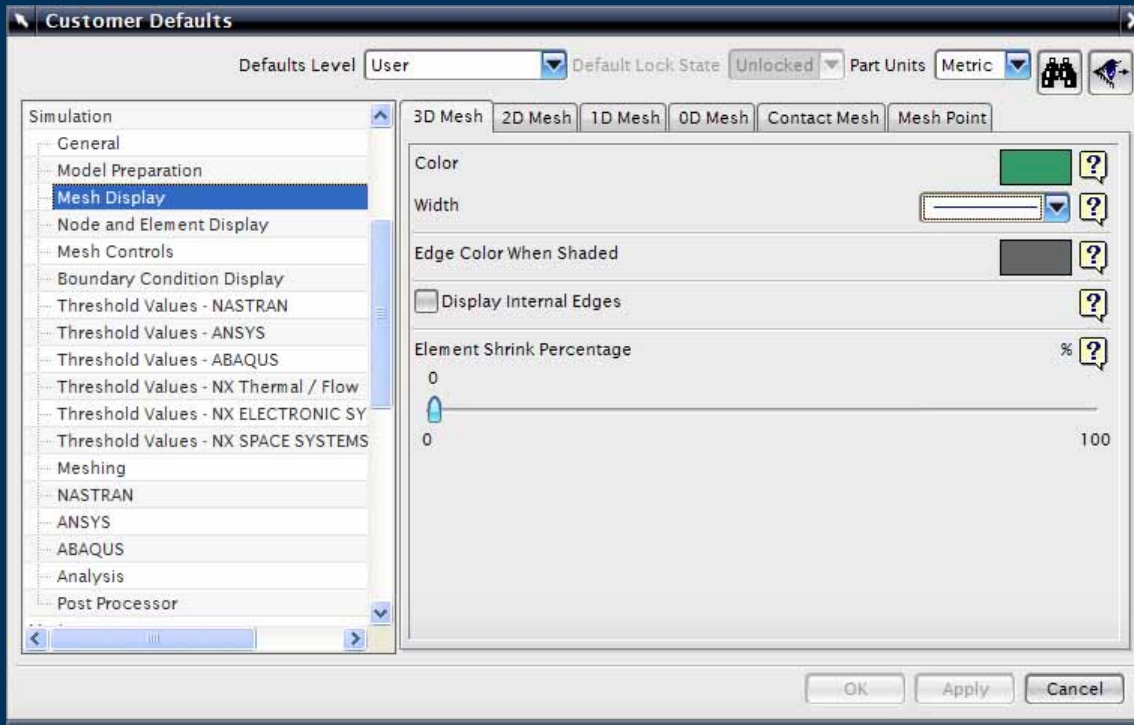
- ▶ Default Solver Language
- ▶ Creation of the four simulation files

# Customer Defaults – Model Preparation



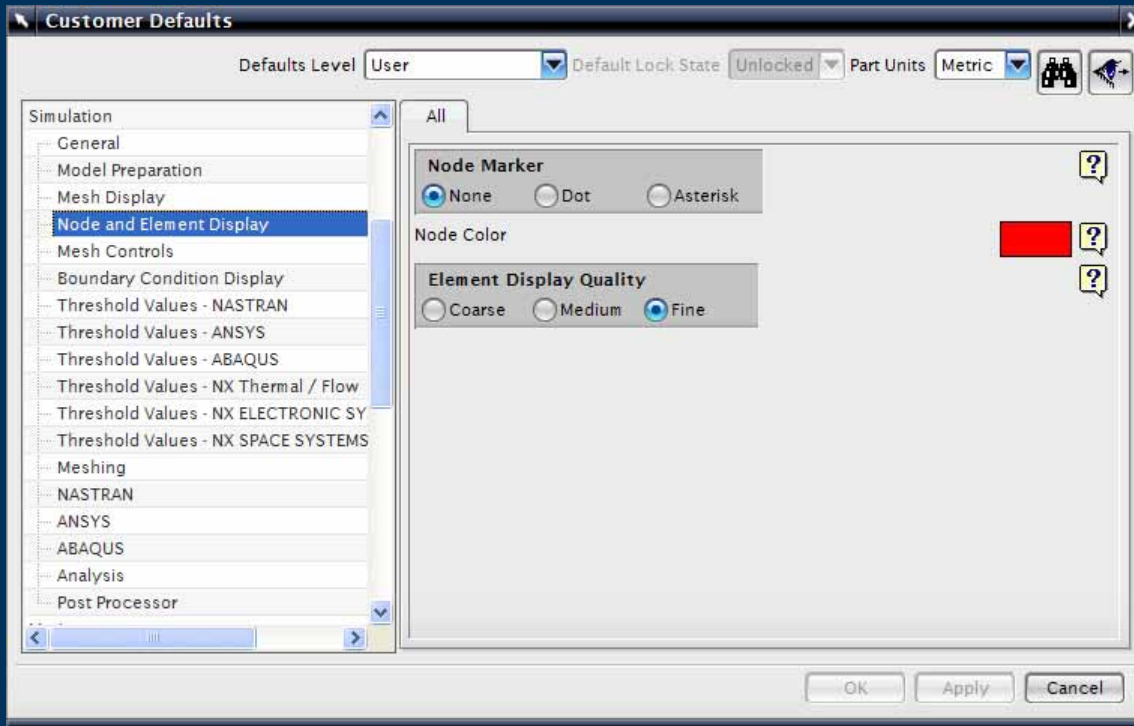
- ▶ Model Preparation
  - ▶ Default values for CAE Topology creation

# Customer Defaults – Mesh Display



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

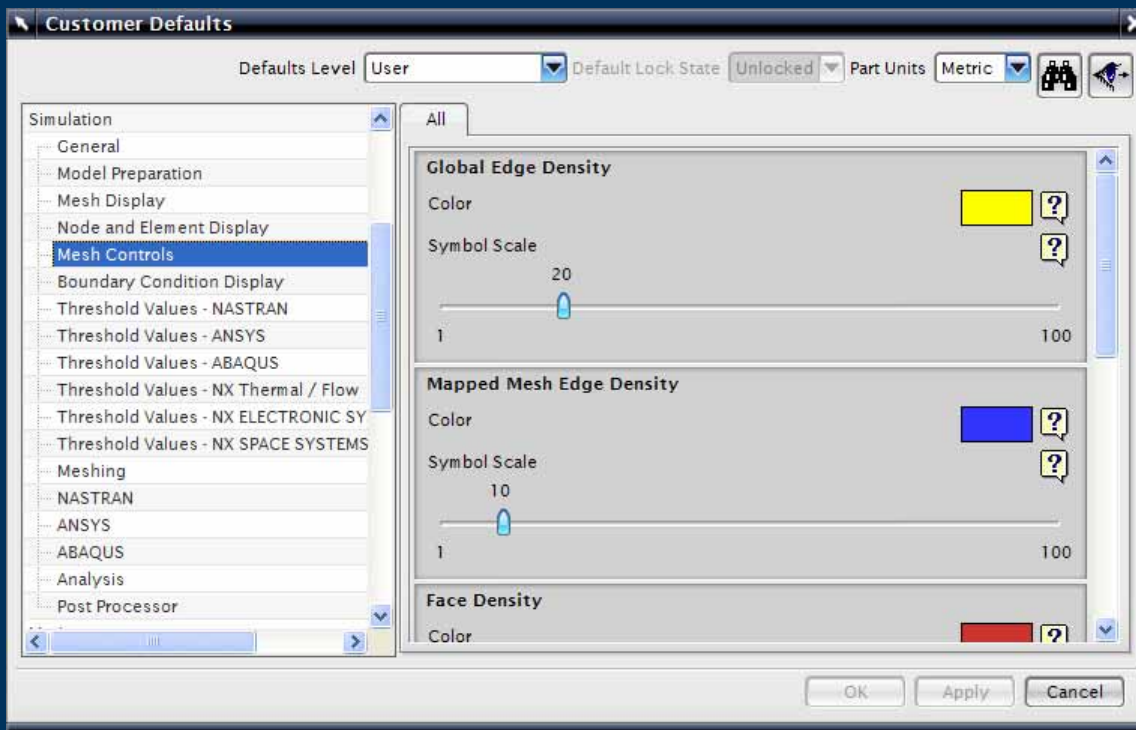
# Customer Defaults – Node & Element Display



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

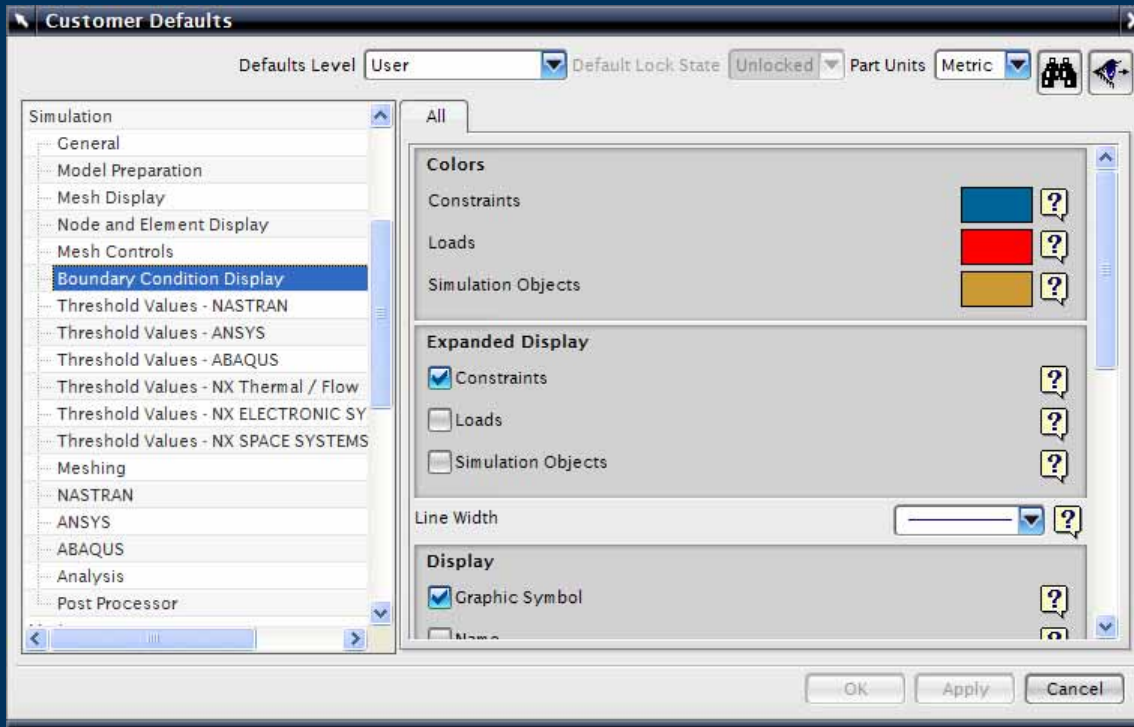


# Customer Defaults – Mesh Controls



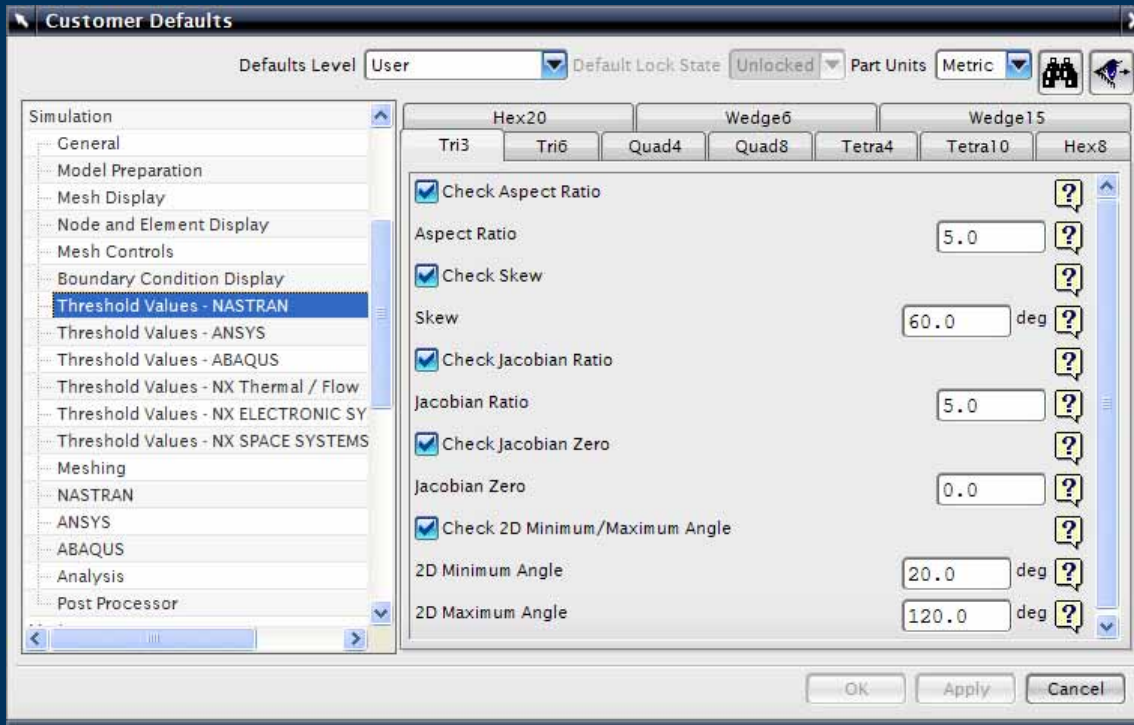
Mesh Control defaults

# Customer Defaults – Boundary Condition Display



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

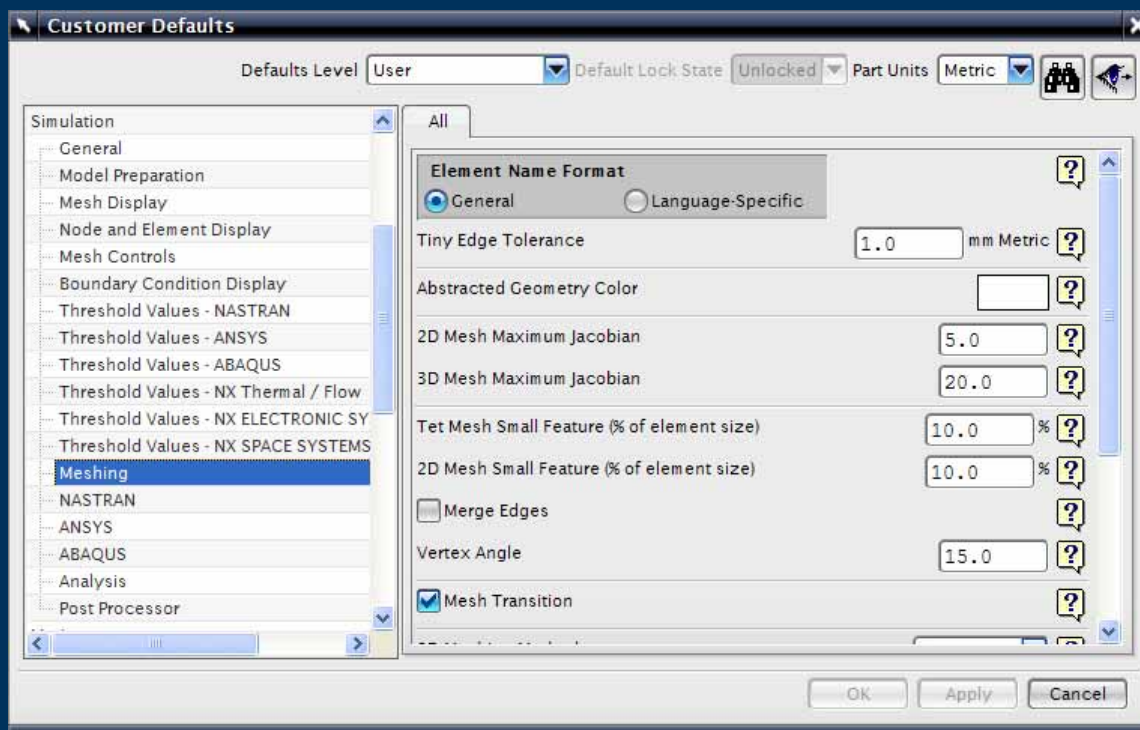
# Customer Defaults – Threshold Values Nastran



## ▶ Threshold Values – Nastran

- ▶ Element quality check threshold values for Tri, Quad, Tet, Hex and Wedge element types

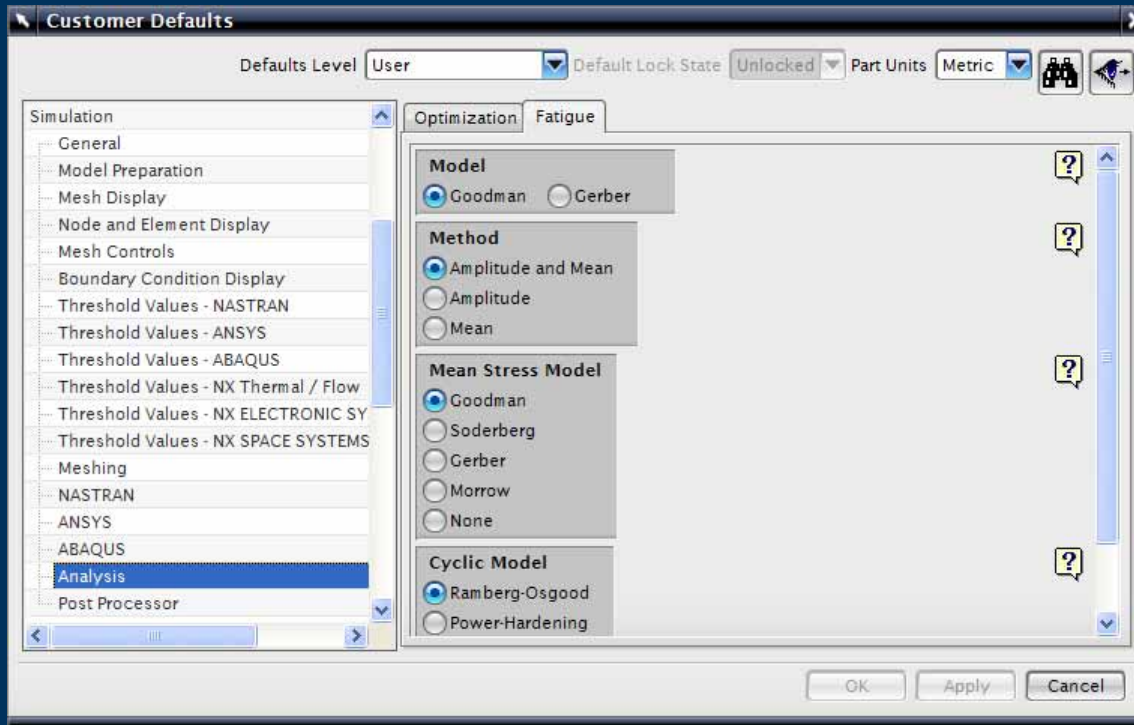
# Customer Defaults – Meshing



## ► Meshing

- General defaults for the meshing task

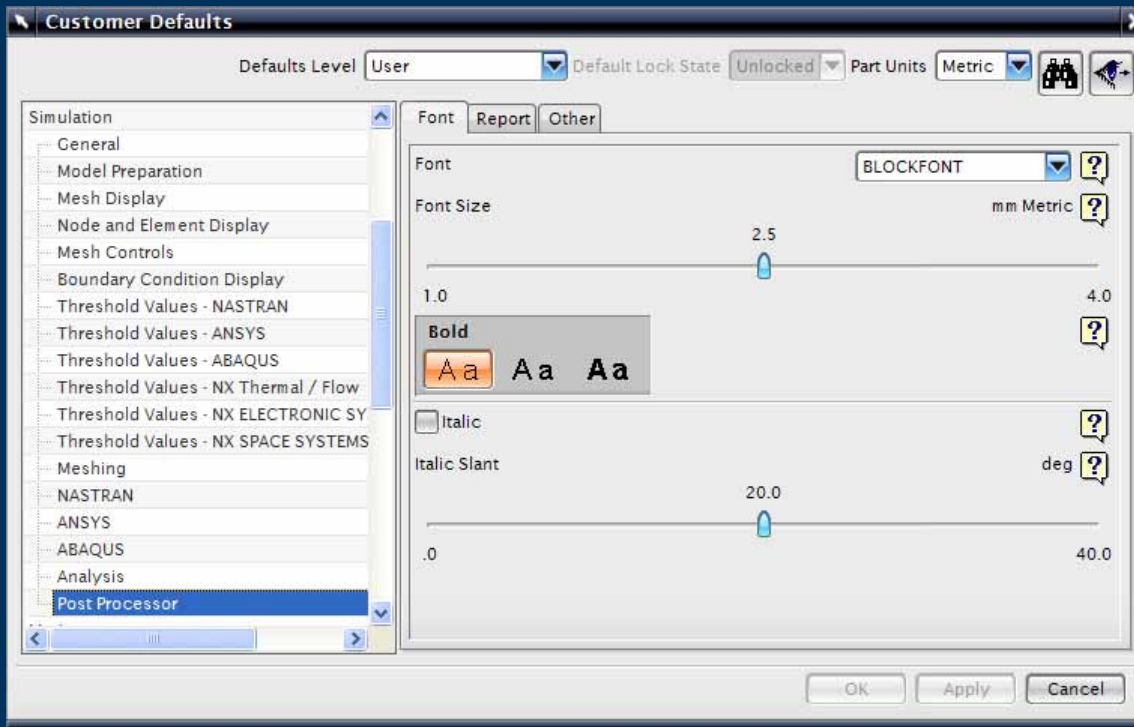
# Customer Defaults – Analysis



## ► 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