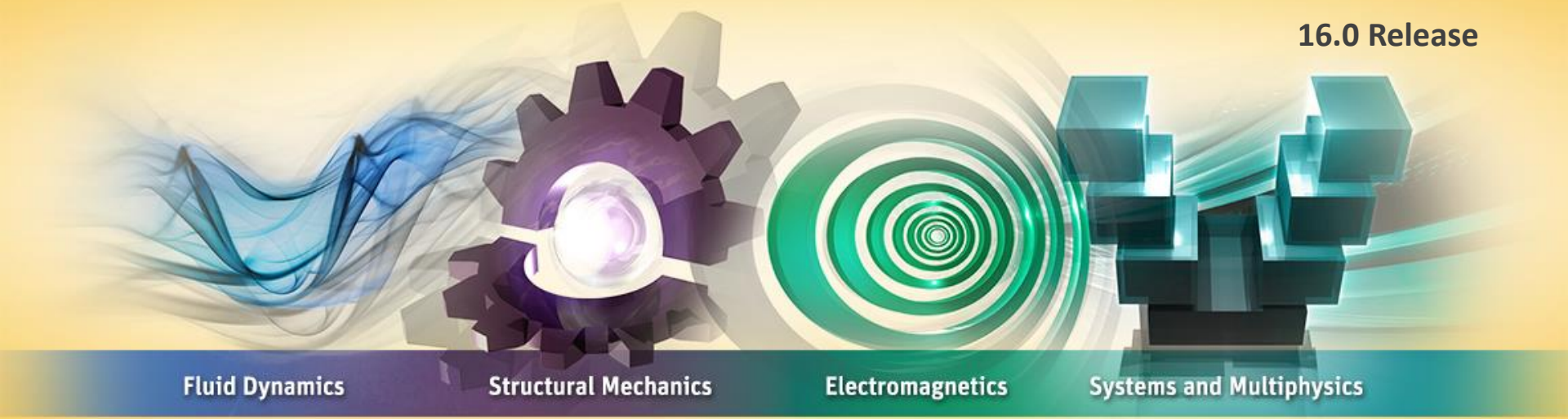
Lecture 3 : General Preprocessing

16.0 Release



Introduction to ANSYS Mechanical

Realize Your Product Promise®

Chapter Overview

In this chapter we cover basic preprocessing operations that are common to all disciplines.

Topics:

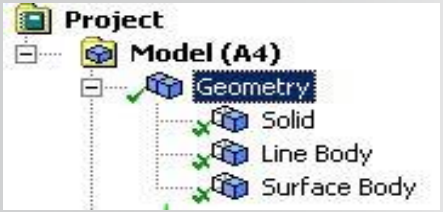
A. Geometry
B. Contact
C. Workshop 3-1, “2D Gear and Rack Analysis”
D. Coordinate Systems
E. Named Selections
F. Workshop 3-2, “Named Selections”
G. Object Generator
H. Selection Information
I. Workshop 3-3, “Object Generator 1”
J. Workshop 3-3, “Object Generator 2”
K. Appendix

A. Geometry

The Geometry branch contains the part(s) that make up the model.

In Mechanical, there are three types of *bodies* which can be analyzed:

- **Solid bodies are 3D or 2D volumes or areas**
- **Surface bodies are only areas**
- **Line bodies are only curves**
- **Each is explained next . . .**



Outline

- Project
  - Model (A4)
    - Geometry
      - Yoke_B
      - Yoke_A
      - Spider
    - Coordinate Systems

Details of "Geometry"

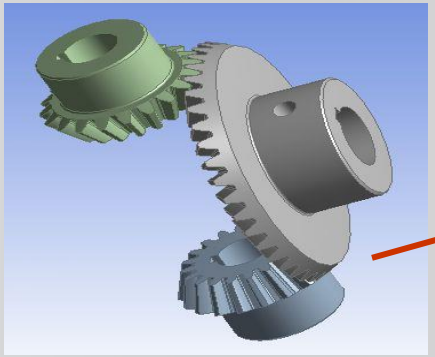
Definition	
Source	E:\Sample Models\U_Joi...
Type	DesignModeler
Length Unit	Millimeters
Element Control	Program Controlled
Display Style	Part Color
Bounding Box	
Properties	
Statistics	
Preferences	
Import Solid Bodies	Yes
Import Surface Bodies	Yes
Import Line Bodies	Yes
Parameter Processing	Yes
Personal Parameter Key	DS
CAD Attribute Transfer	No
Named Selection Processing	No
Material Properties Transfer	No
CAD Associativity	Yes
Import Coordinate Systems	No
Reader Save Part File	No
Import Using Instances	Yes
Do Smart Update	No
Attach File Via Temp File	No
Analysis Type	3-D
Mixed Import Resolution	None
Enclosure and Symmetry Processing	Yes

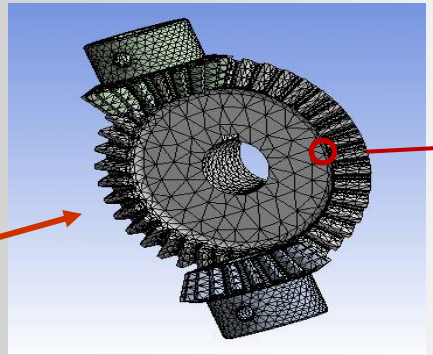
… Geometry

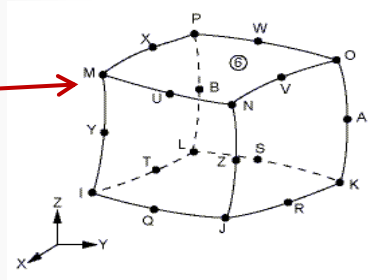
Solid bodies are geometrically and spatially 3D or 2D:

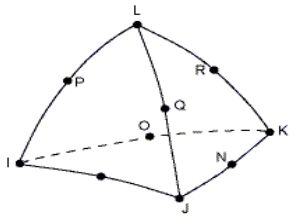
- **3D solids are meshed by default with higher-order tetrahedral or hexahedral solid elements with quadratic shape functions.**

- **Each node in a 3D element has three translational degrees of freedom (DOF) for structural or one temperature DOF for thermal.**





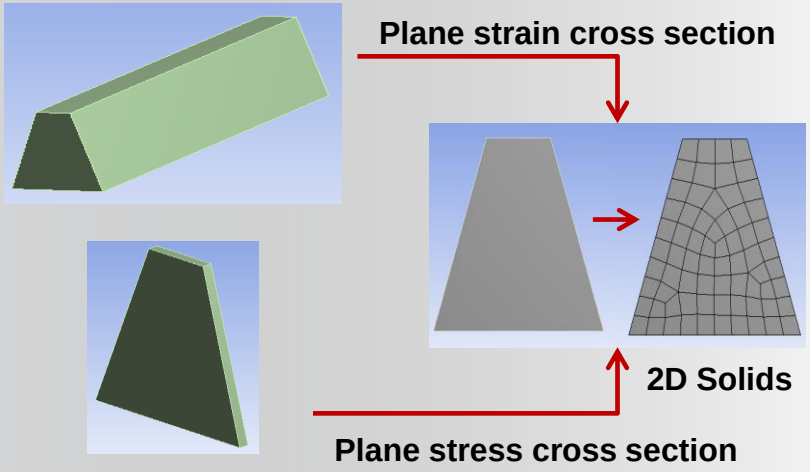


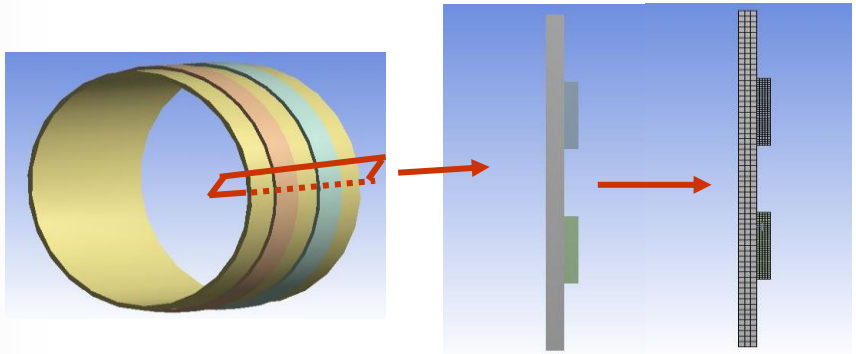


3D Solids → **3D Element** **Hex Element** **Tet Element**

… Geometry

- 2D solids are meshed by default with higher order trianglular or quadrilateral solid elements with quadratic shape functions.
  - The “2D” switch must be set on the Project page prior to importing geometry.
- Each node in a 2D element has two translational degrees of freedom (UX and UY) for structural or one temperature DOF for thermal.
- 2D solids are used to represent three types of 3D geometry, “Axisymmetric”, “Plane stress” and “Plane strain”.





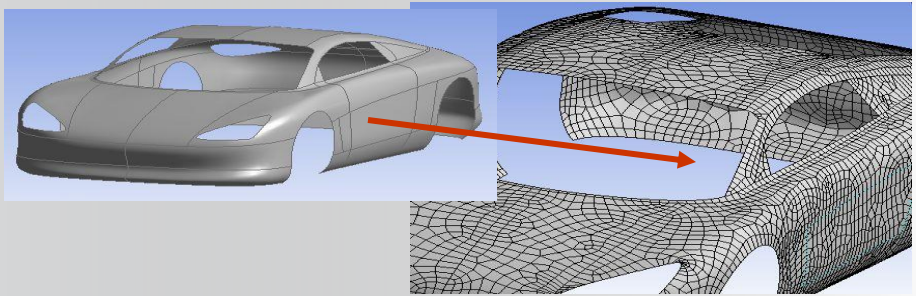
… Geometry

Surface bodies are geometrically 2D but spatially 3D:

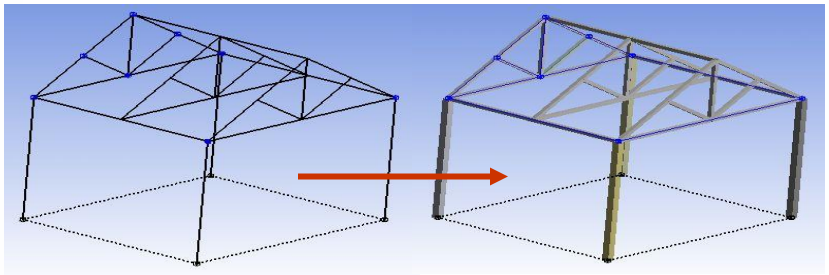
- **Surface bodies represent structures which are thin in one dimension (through the thickness). Thickness is not modeled but *supplied as an input value*.**
- **Surface bodies are meshed with shell elements having six DOF (UX, UY, UZ, ROTX, ROTY, ROTZ).**

Line bodies are geometrically 1D but spatially 3D:

- **Line bodies represent structures which are thin in two dimensions. The cross-section is not modeled, it is mapped on to the line body.**
- **Line bodies are modeled with beam elements having six DOF (UX, UY, UZ, ROTX, ROTY, ROTZ).**



Surface Body



Line Body

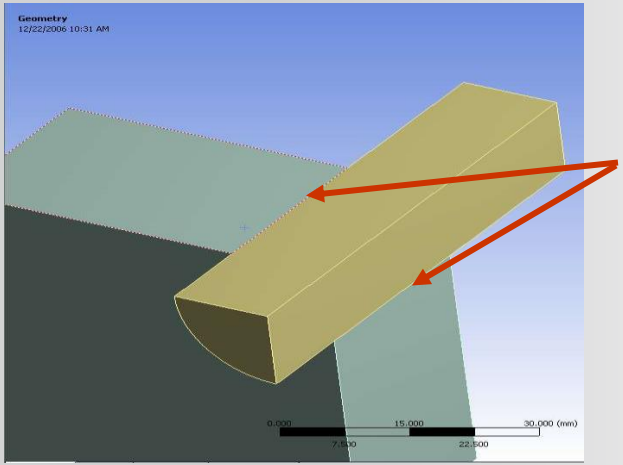
… Geometry

In general, *bodies* and *parts* are the same. In DesignModeler or SpaceClaim however, multiple bodies may be grouped into *multibody parts*.

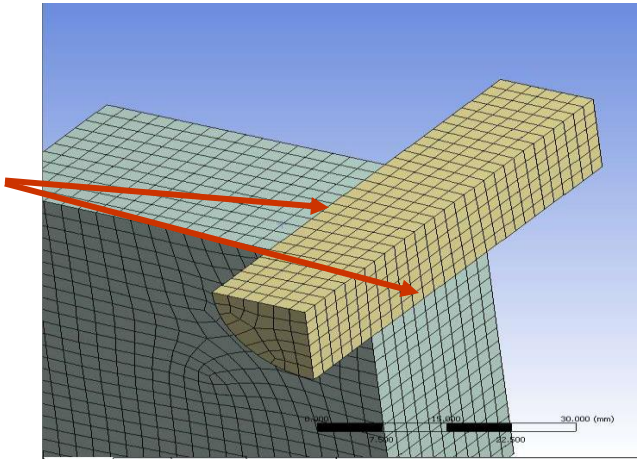
Multibody parts share common boundaries so nodes are shared at that interface.

- **No contact is needed in these situations.**

Example:



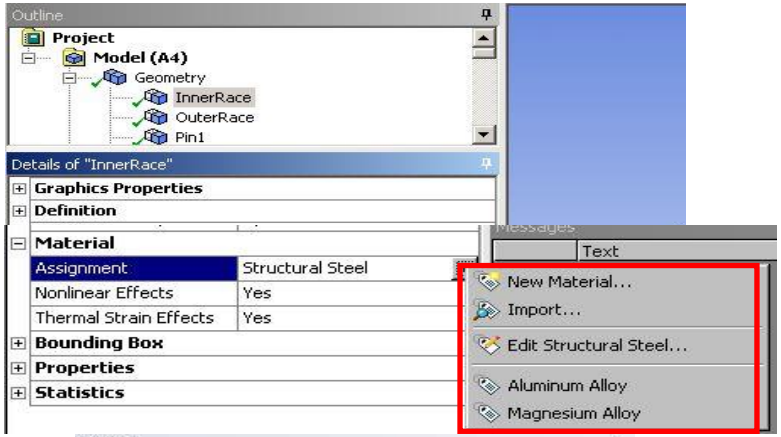
Common nodes are shared by adjacent bodies

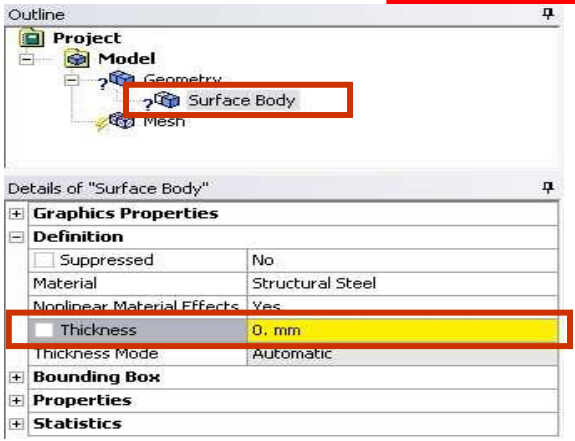


… Geometry

To assign material properties to a part, highlight it and select from the available properties in the "Assignment" field :

- The only materials appearing in the list will be materials added using the "Engineering Data" application (see chapter 2).
- The user can also access Engineering Data directly for creating, editing or importing new materials.



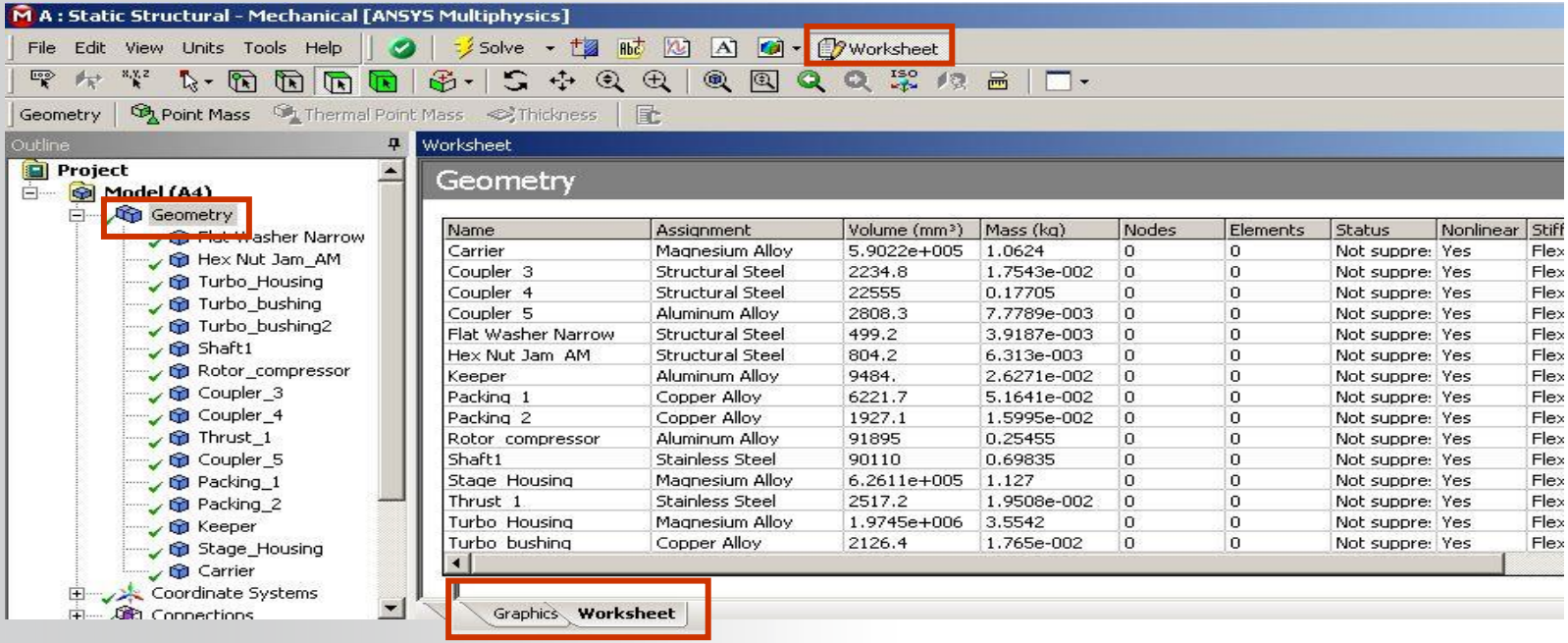


For surface bodies a thickness needs to be supplied as well (thickness can also be imported).

… Geometry

A summary of all parts along with assigned materials, mesh statistics, etc..

- **Select “Geometry” branch and toggle the “Worksheet” icon.**
- **Toggle between graphics or worksheet via tabs at bottom**



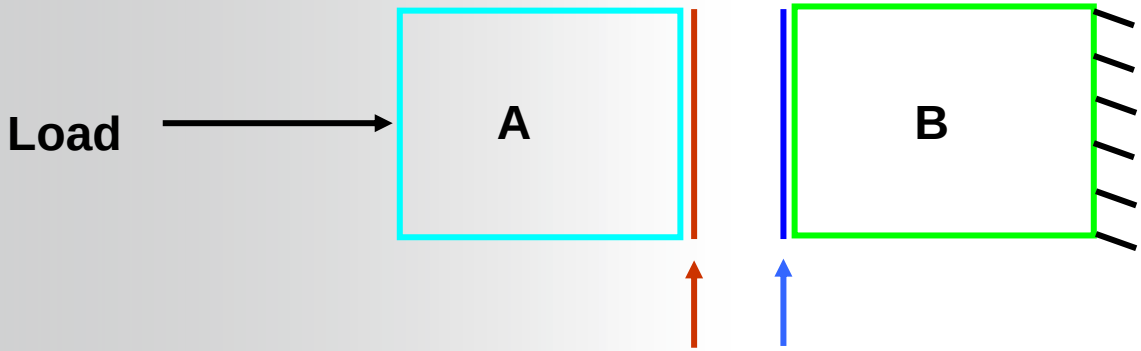
Name	Assignment	Volume (mm³)	Mass (kg)	Nodes	Elements	Status	Nonlinear	Stiff
Carrier	Magnesium Alloy	5.9022e+005	1.0624	0	0	Not suppre:	Yes	Flex
Coupler_3	Structural Steel	2234.8	1.7543e-002	0	0	Not suppre:	Yes	Flex
Coupler_4	Structural Steel	22555	0.17705	0	0	Not suppre:	Yes	Flex
Coupler_5	Aluminum Alloy	2808.3	7.7789e-003	0	0	Not suppre:	Yes	Flex
Flat Washer Narrow	Structural Steel	499.2	3.9187e-003	0	0	Not suppre:	Yes	Flex
Hex Nut Jam_AM	Structural Steel	804.2	6.313e-003	0	0	Not suppre:	Yes	Flex
Keeper	Aluminum Alloy	9484.	2.6271e-002	0	0	Not suppre:	Yes	Flex
Packing_1	Copper Alloy	6221.7	5.1641e-002	0	0	Not suppre:	Yes	Flex
Packing_2	Copper Alloy	1927.1	1.5995e-002	0	0	Not suppre:	Yes	Flex
Rotor_compressor	Aluminum Alloy	91895	0.25455	0	0	Not suppre:	Yes	Flex
Shaft1	Stainless Steel	90110	0.69835	0	0	Not suppre:	Yes	Flex
Stage_Housing	Magnesium Alloy	6.2611e+005	1.127	0	0	Not suppre:	Yes	Flex
Thrust_1	Stainless Steel	2517.2	1.9508e-002	0	0	Not suppre:	Yes	Flex
Turbo_Housing	Magnesium Alloy	1.9745e+006	3.5542	0	0	Not suppre:	Yes	Flex
Turbo_bushing	Copper Alloy	2126.4	1.765e-002	0	0	Not suppre:	Yes	Flex

B. Contact

When multiple parts are present contact elements define the relationship between parts.

- **Are parts bonded together, sliding, transferring heat, etc.?**

Without contact or spot welds, parts will not interact with each other.

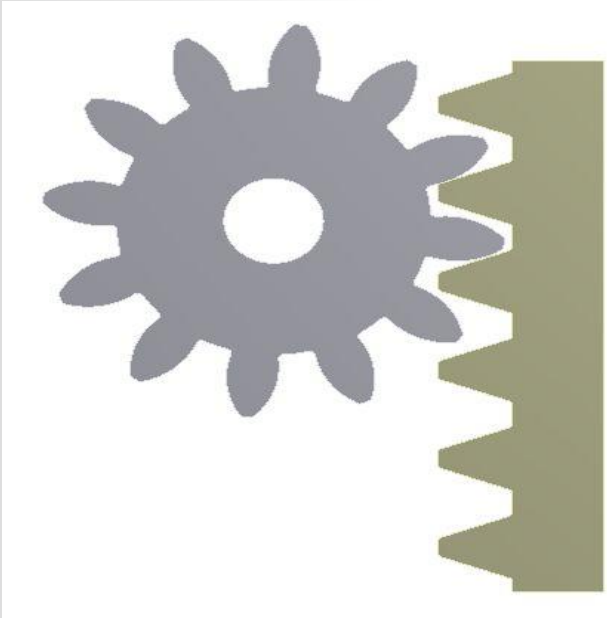


Contact elements can be visualized as a "skin" covering the regions where contact will occur.

Contact details are covered in Connections lecture (L05_Connections).

C. Workshop 3.1 – 2D Gear and Rack Analysis

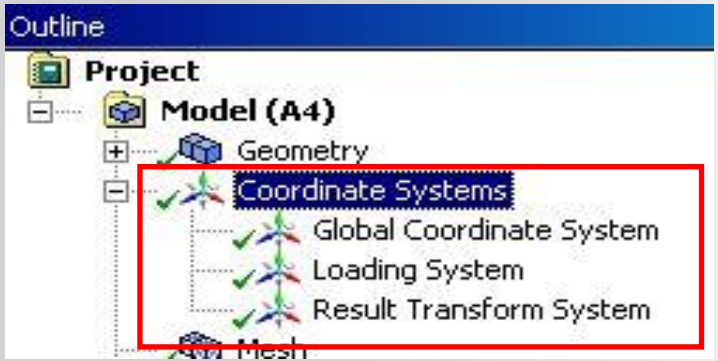
- **Workshop 3.1 – 2D Gear and Rack Analysis**
- **Goal:**
  - **Determine the torque required in the gear to produce the desired output.**

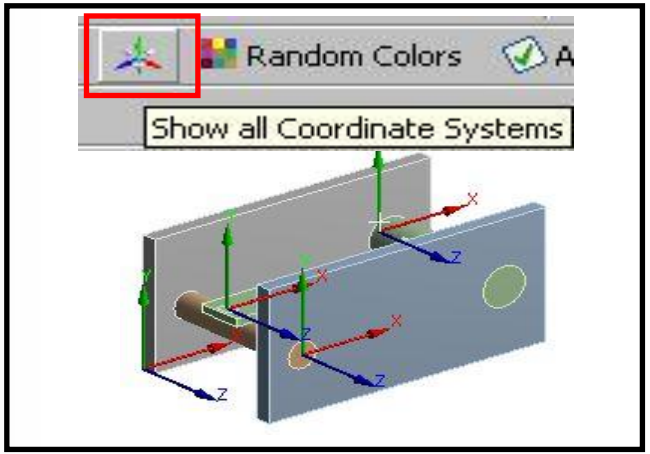


D. Coordinate Systems

The Coordinate Systems branch initially contains only the global Cartesian system:

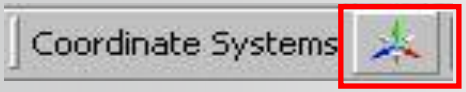
- **User coordinate systems can be Cartesian or cylindrical and can be renamed.**
- **User coordinate systems can be added and used for mesh controls, point masses, directional loads, results, etc..**
- **An icon in the Graphics Option toolbar allows all coordinate systems to be viewed simultaneously.**

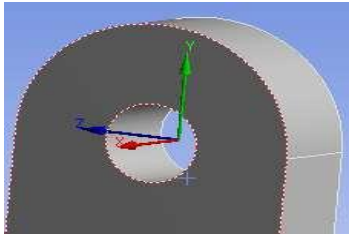




… Coordinate Systems

Coordinate Systems are defined by selecting "Coordinate System" icon from the Context toolbar.

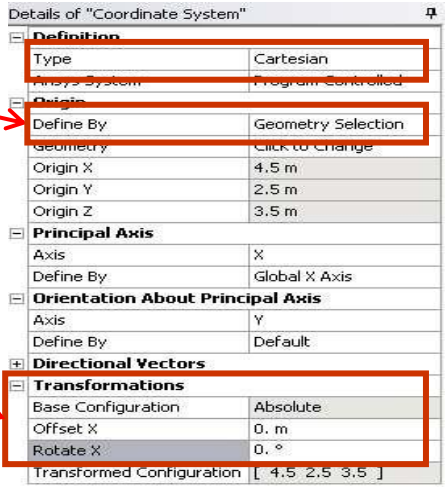


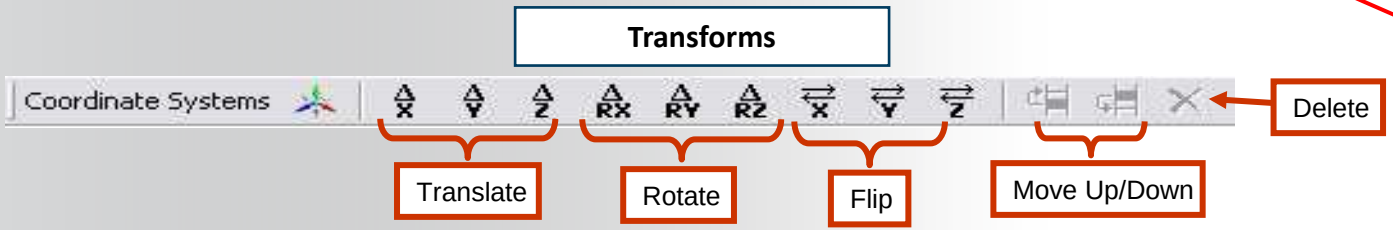


Associative coordinate systems can be defined by selecting geometry (surfaces, edges, etc.). Associative CS update their origin to match geometry updates.

Non-associative coordinate systems are defined by entering its location in global coordinates.

The location and orientation can then be further modified using the various transforms from the context toolbar.

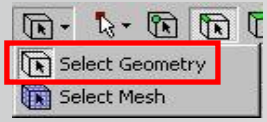


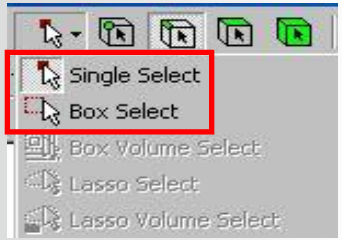


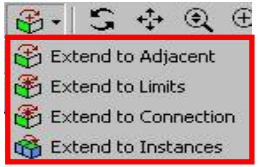
E. Named Selections

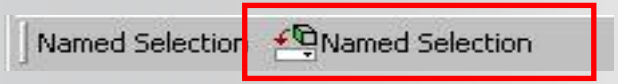
Named Selections are groups of geometric or finite element entities:

- **Named selections can be created either by selecting the desired items and clicking the "Named Selection" icon in the context toolbar or RMB > Named Selection OR using the named selection worksheet (shown later).**









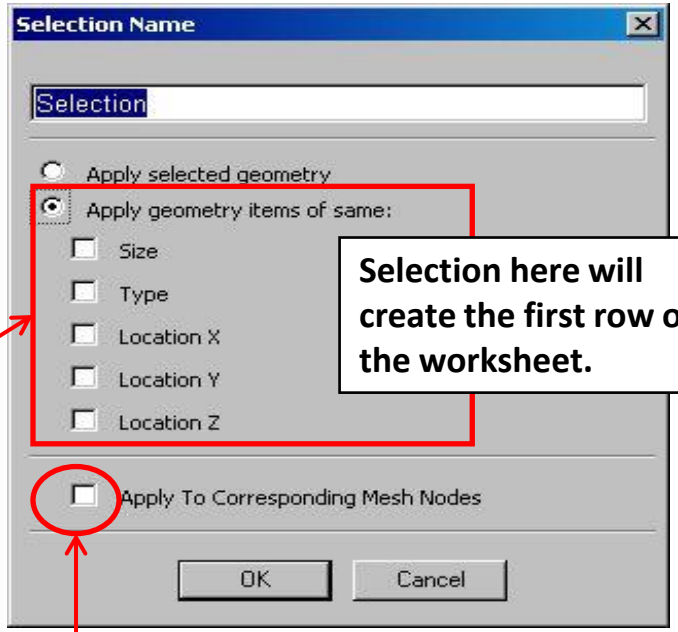
- **Named selections must be composed of "like" entities (all surfaces or all edges, all nodes, etc.).**

… Named Selections

A new criteria selection can be based on an initial selection:

- **Make an initial selection followed by a RMB > “Create Named Selection”.**
- **Note, initial selection must be a single entity.**

RMB >



Selection here will create the first row of the worksheet.

Convert to nodal named selection immediately.

… Named Selections

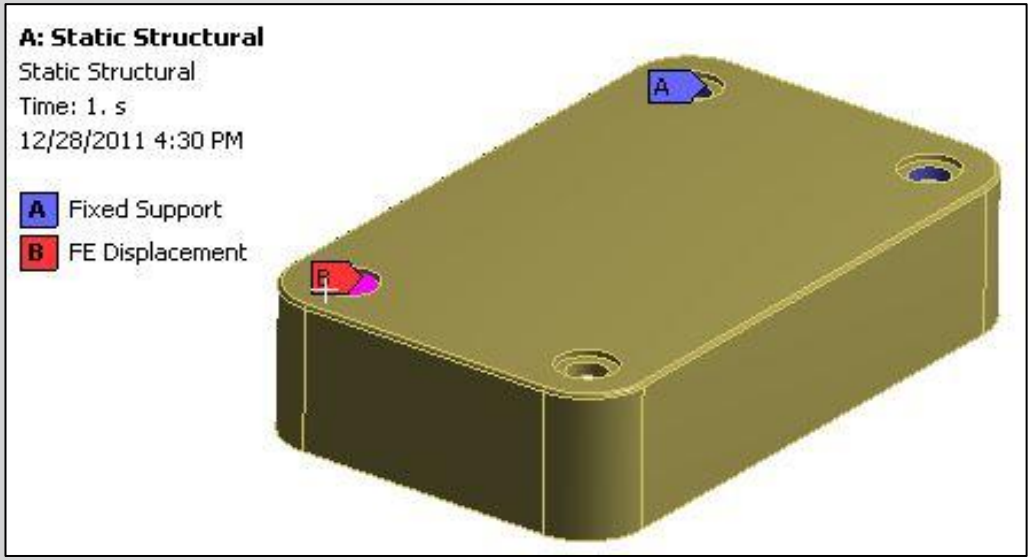
In many detail window fields Named Selections can be referenced directly:

- **In the Details view, change “Scoping Method” from “Geometry Selection” to “Named Selection”**
- **Select the “Named Selection” from the pull-down menu**

- **A named selection toolbar provides quick access to basic controls “View > Toolbars > Named Selections”:**

F. Workshop 3.2

Workshop 3.1 – Named Selections



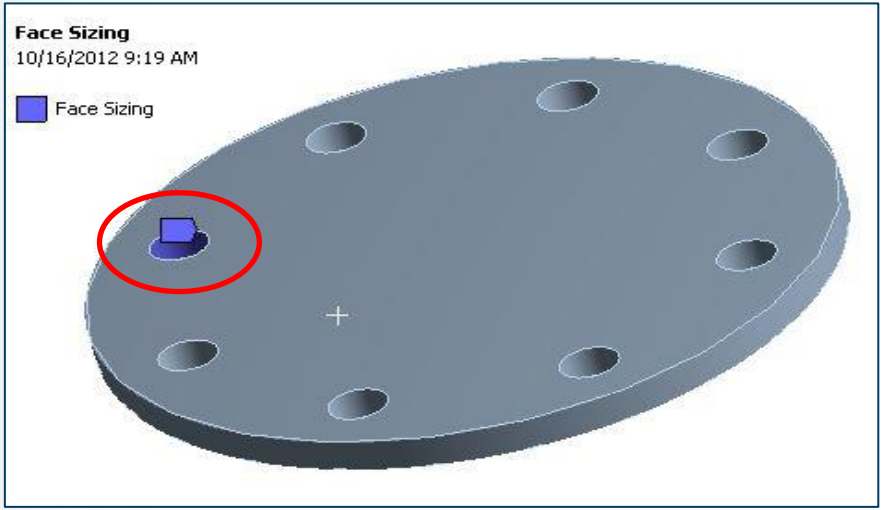
G. Object Generator

The Object Generator uses an existing object in the tree as a template for replication.

Almost any tree object that supports “RMB > Duplicate” can be used as a template.

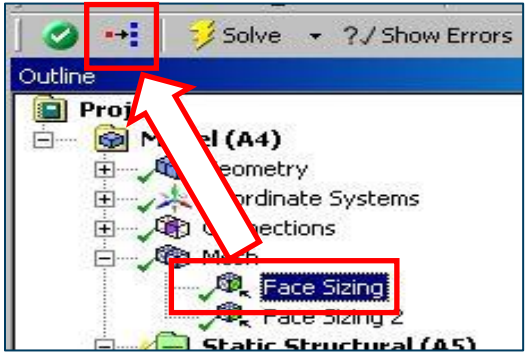
An example:

In the model shown, we have set up a mesh size control on one of the hole faces. We would like to have this same control applied to all other holes.

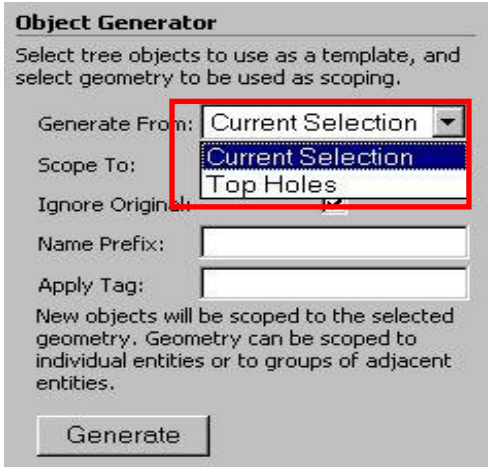


. . . Object Generator

Highlight the tree object to be replicated (the Face Sizing in this case) and activate the Object Generator using the icon.

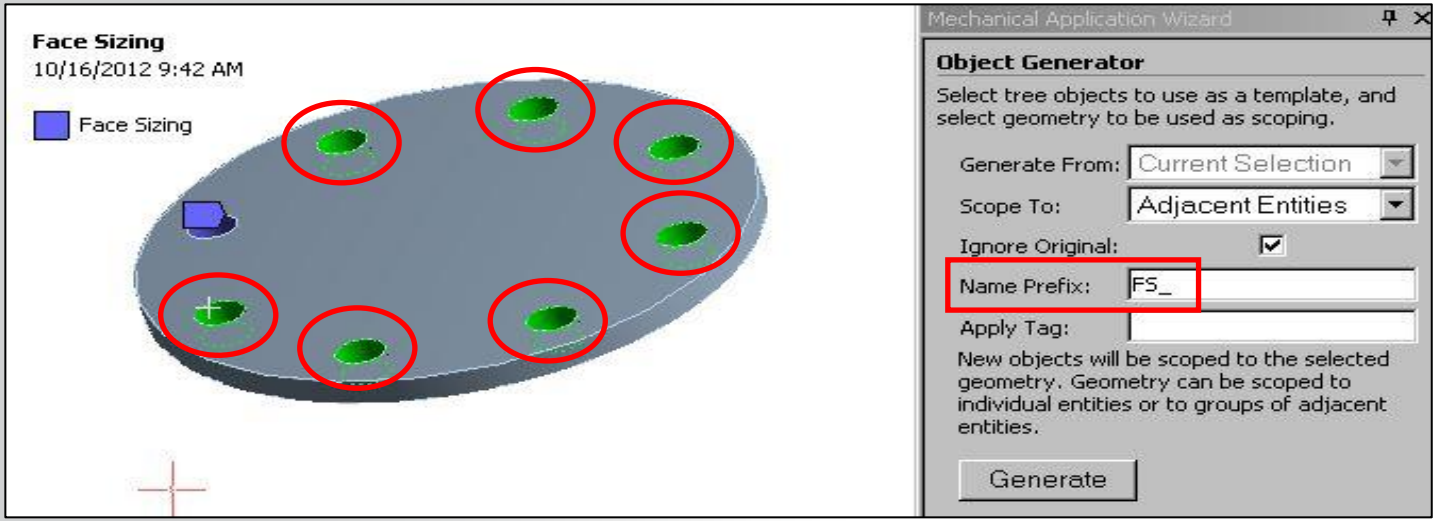


The object generator opens and indicates objects will be generated from the “Current Selection” (graphical selection) or from a named selection we’ve called “Top Holes”.



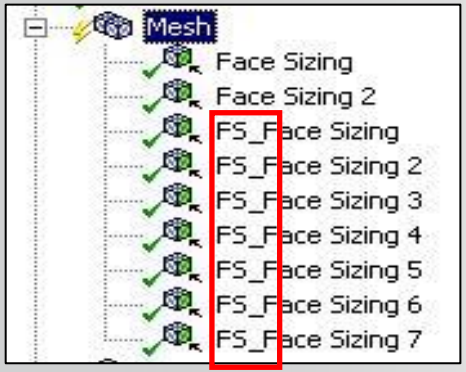
. . . Object Generator

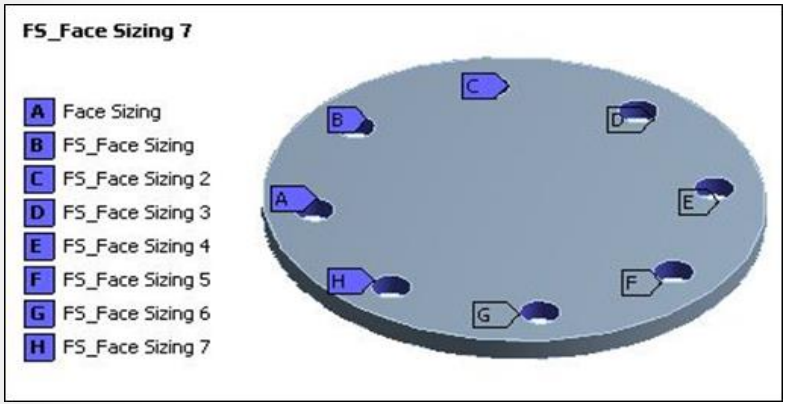
We choose to graphically highlight the remaining hole faces and add a Name Prefix (“FS_”) so the new objects can be easily identified in the tree.



. . . Object Generator

After generating, the new objects can be displayed graphically and they appear in the tree. The object names reflect that of the parent object along with the prefix chosen (if any).



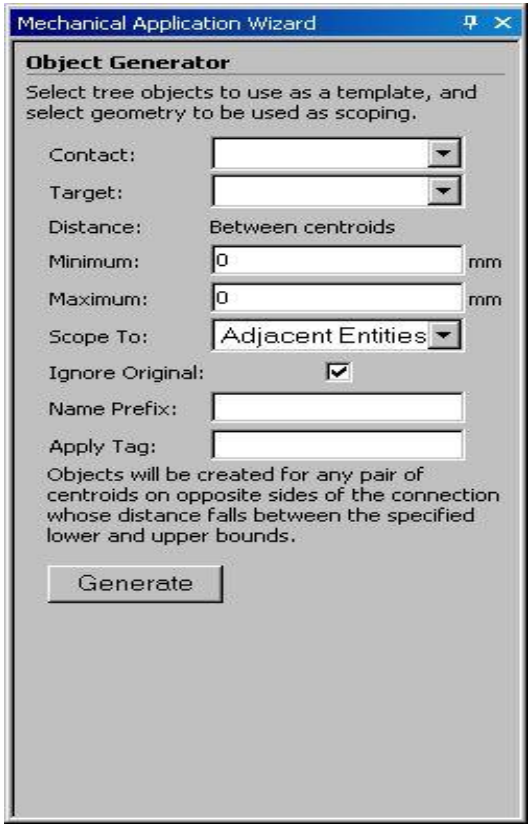


In this example the object to be generated (the mesh size control) required only a single geometry selection, a face, to define its scope. Some objects that can be duplicated require multiple scoping selections (e.g. contacts, beam connections, etc.). We'll look at this next.

. . . Object Generator

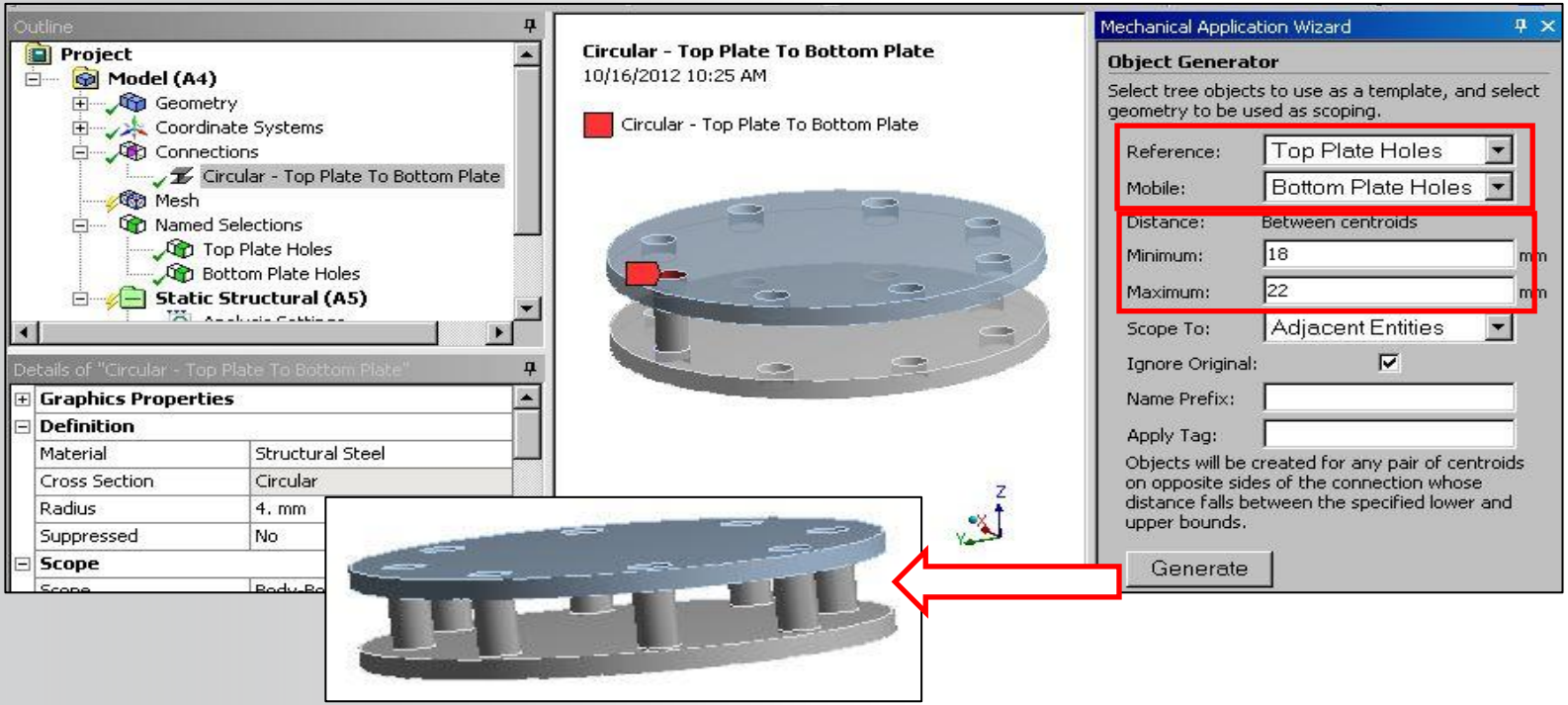
When an object in the tree that requires multiple scoping is selected, the object generator reflects this (a contact selection is shown here).

- Fields for contact and target scoping are visible however there is no longer a “current selection” choice. Instead, named selections must now be chosen from the drop down fields.
- A “Distance Between Centroids” range must now be set to indicate which parts of the named selections are to be paired to one another.
- In addition to an optional prefix, new items can be tagged when generated.



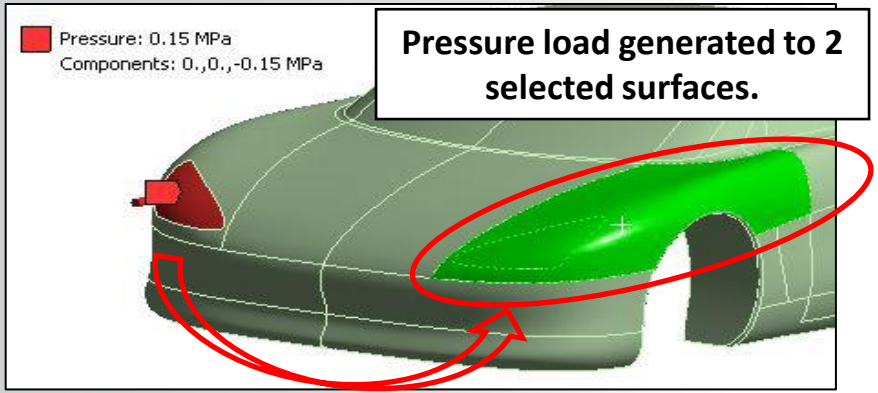
. . . Object Generator

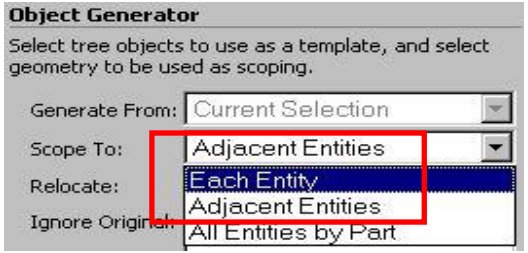
Here a bolt connection has been defined between the plates and named selections created for the holes in each plate. The range chosen is based on the distance between the plates (20 mm).

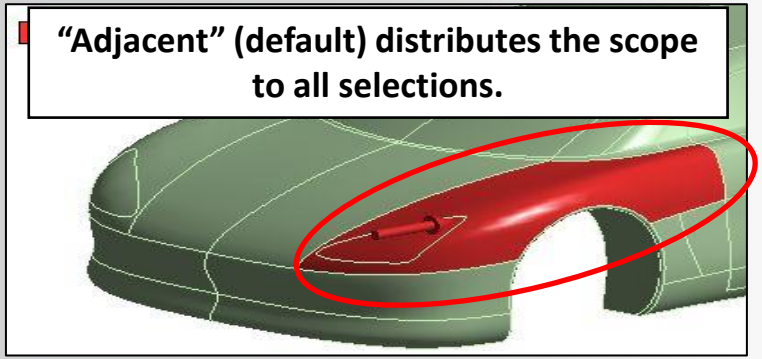


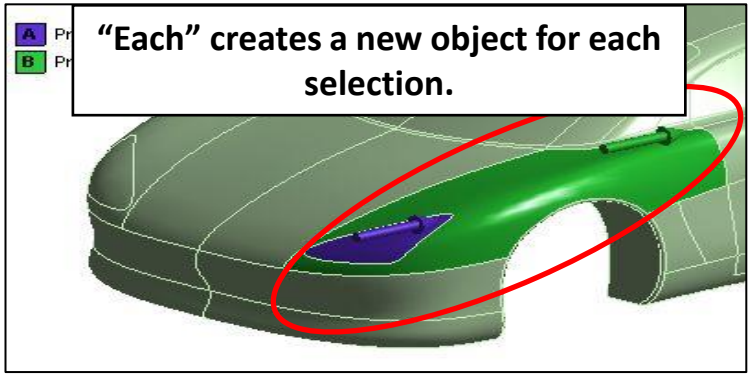
. . . Object Generator

Scoping the new objects:





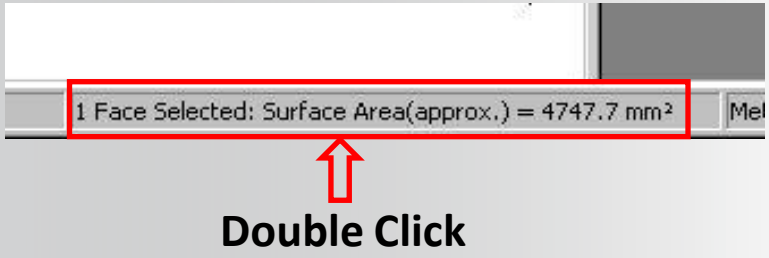


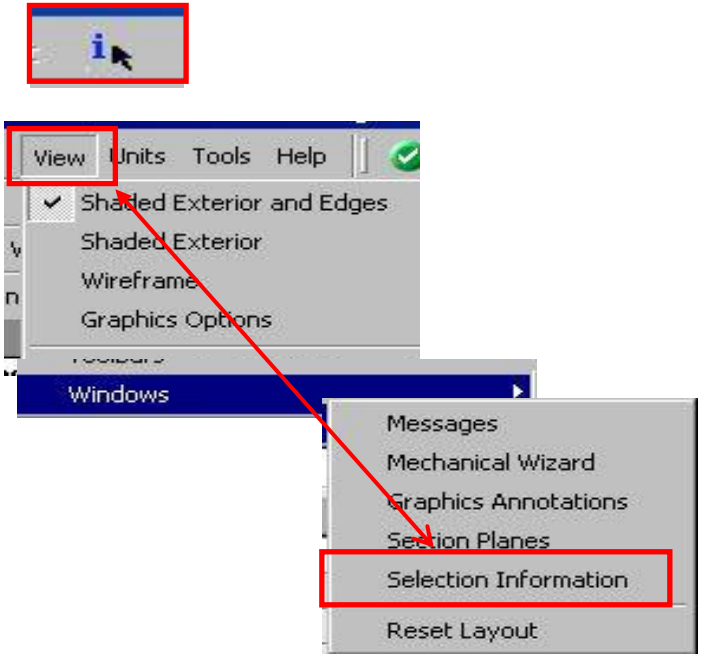


H. Selection Information

When a selection is made in the graphics window (node, vertex, face, etc.), the status bar lists basic information (e.g. line length, surface area, etc.). Additional information can be obtained by using the selection information window.

- **Activate (3 ways) by:**
  - **Icon**
  - **View > Windows > Selection Information**
  - **Double click the status bar selection.**





. . . Selection Information

The selection information window provides a summary of all selections and/or a list of individual selections (or both as shown).

Select: vertex, edge, face, body, node or xyz coordinate location.

Coordinate selection returns information on the nearest node to the coordinates.

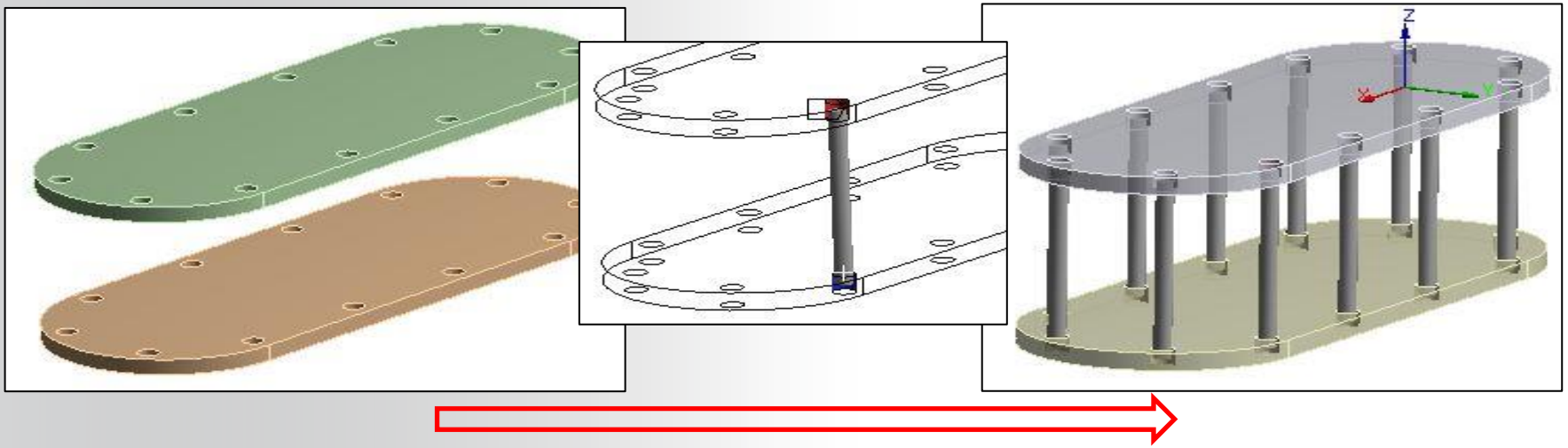
Selection Information

Coordinate System: Global Coordinate System | Show Individual and Summary

Entity	Surface Area (mm²)	Centroid X(mm)	Centroid Y(mm)	Centroid Z(mm)	Body	Type
2 Faces, Summary	9495.3	49.334	25.049	30.		
Face 1	4747.7	49.334	25.049	55.	SidePlate2	Plane
Face 2	4747.7	49.334	25.049	5.	SidePlate1	Plane

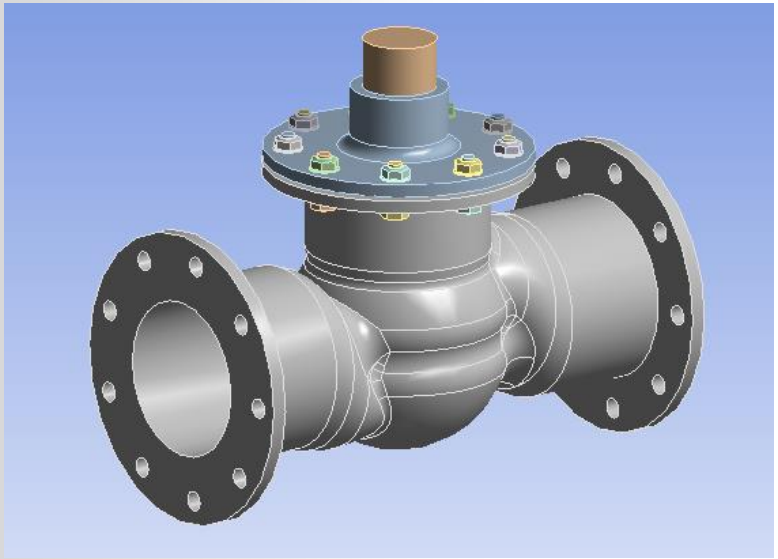
I. Workshop 3.3 – Object Generator 1

- **Workshop 3.3 – Object Generator 1**
- **Goal:**
  - **Become familiar with the operation of the object generator in the Mechanical application.**



J. Workshop 3.4 – Object Generator 2

- **Workshop 3.4 – Object Generator 2**
- **Goal:**
  - **Become familiar with the operation of the object generator in the Mechanical application.**



K. Appendix

- **Named Selection**

Named Selections

Worksheet Criteria:

- **Insert a new named selection then change the scoping method to “Worksheet”.**

Details of "Selection 2"

Scope	
Scoping Method	Worksheet
Geometry	No Selection
Definition	
Send to Solver	Yes
Visible	Yes
Program Controlled Inflation	Exclude
Statistics	
Type	Manual
Total Selection	No Selection
Suppressed	0
Hidden	0

- **Selections are created using various criteria.**
- **Add, remove, filter, etc. to “stack” criteria for complex selections.**
- **Generate selections after the criteria choices are complete.**

Worksheet

Selection 2

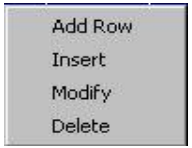
Generate

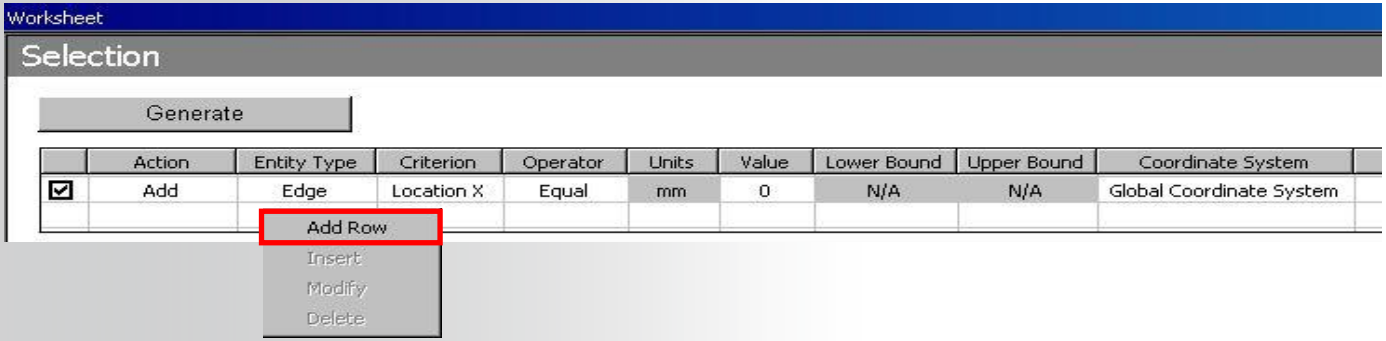
Action	Geometry Type	Criterion	Operator	Units	Value	Lower Bound	Upper Bound	Coordinate System
Add	Vertex	Location X	Range	mm	N/A	96.	98.	Global Coordinate Sy...
Filter	Vertex	Location Y	Greater Than	mm	30.	N/A	N/A	Global Coordinate Sy...

… Named Selections

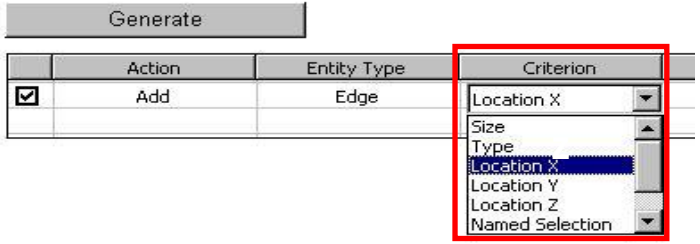
Worksheet Criteria is entered in rows and columns in the worksheet.

- **When a new worksheet opens the first step is to RMB to “Add Row”.**
- **Subsequent rows can be added to the list or inserted mid list (order matters!).**





- **As rows are added to the worksheet columns are configured using drop down menus available by clicking the desired cell.**



… Named Selections

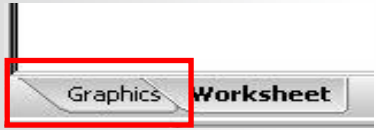
After entering the criteria (various actions, entity types, etc.) the named selection is created by clicking the "Generate" button.

- **Rows can be made temporarily inactive using the check box column on the left.**
- **Following named selection generation, the details show the result.**
- **To view the named selection toggle to the "Graphics" tab at the bottom of the worksheet.**

Selection

Generate

	Action	Entity Type	Criterion	Operator	Units	Value	Lowe
☑	Add	Edge	Location X	Equal	mm	0.	
☑	Remove	Edge	Location Y	Less Than	mm	10.	
☑	Add	Edge	Radius	Equal	mm	5	



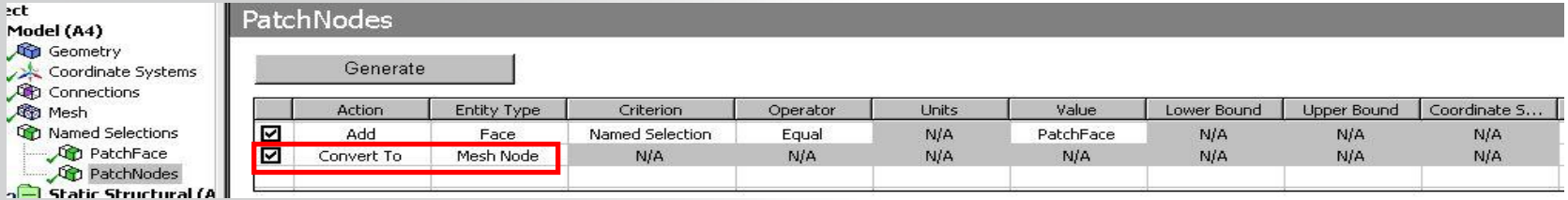
Details of "Selection"

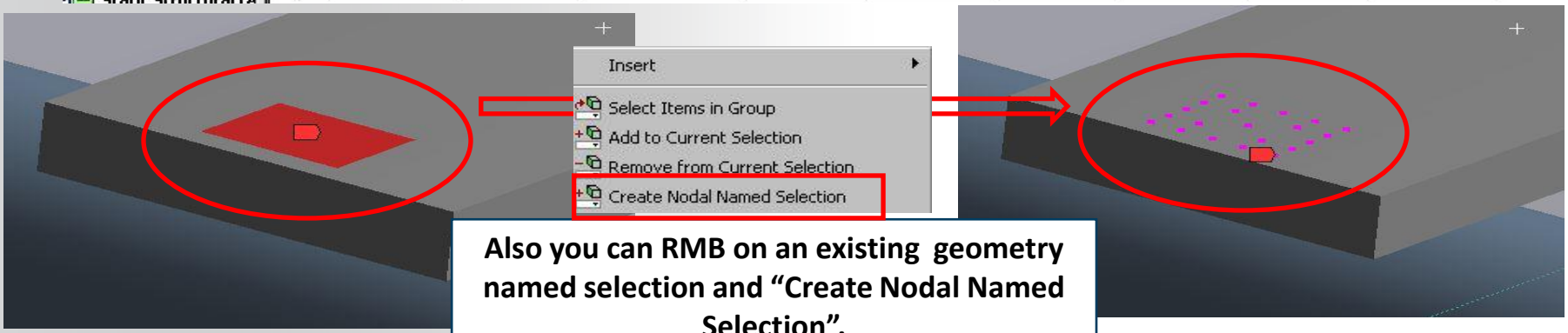
Scope	
Scoping Method	Worksheet
Geometry	37 Edges
Definition	
Send to Solver	Yes
Visible	Yes
Program Controlled Inflation	Exclude
Statistics	
Type	Manual
Total Selection	37 Edges
Suppressed	0
Used by Mesh Worksheet	No
Tolerance	
Tolerance Type	Program Controlled
Zero Tolerance	1.e-008
Relative Tolerance	1.e-003

… Named Selections

It is often convenient to convert a geometric named selection into a nodal named selection in order to obtain the underlying nodes

- **Create a geometry named selection (face for example).**
- **Create a second named selection using the “Worksheet”.**
- **Convert the geometry selection to nodes.**



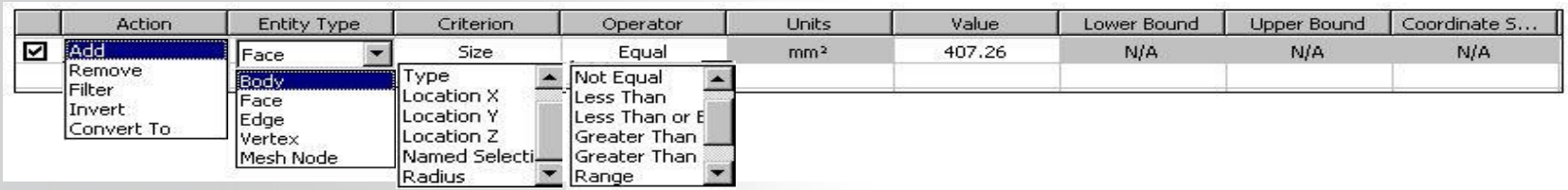


Also you can RMB on an existing geometry named selection and “Create Nodal Named Selection”.

… Named Selections

Worksheet Summary:

- **As can be seen, numerous controls are available in the worksheet. See the documentation for a complete discussion of each.**
- **As discussed elsewhere in this course the "Convert To" action is used to change a geometry named selection (face, edge, etc.) to a nodal named selection.**



When creating named selections using worksheet criteria a set of tolerances is available if needed:

- **Tolerance adjustment is usually only necessary in special circumstances like models in micro units or node selections in dense meshes.**

Tolerance	
Tolerance Type	Program Controlled
Zero Tolerance	1.e-008
Relative Tolerance	1.e-003