

#### **Introduction to ANSYS Mechanical**

**Realize Your Product Promise®** 

### **ANSYS** 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
- 2 © 2015 ANSYS, Inc. February 27, 2015

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 . . .



Dutline	1
Project  Model (A4)  Geometry  Yoke_B  Yoke_A  Spider  Coordinate Systems	
Details of "Geometry"	1
Definition	
Source	E:\Sample Models\U_Joi.
Туре	DesignModeler
Length Unit	Millimeters
Element Control	Program Controlled
Display Style	Part Color
Bounding Box	42
Properties	
E 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

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.



- 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".





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).





Line Body

6 © 2015 ANSYS, Inc. February 27, 2015

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



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).



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

🕅 A : Static Structural - Mechanical [ANSY	'S Multiphysics]								
File Edit View Units Tools Help 🛛 🥝	🚽 ジ Solve 👻 🏥 🥅	t 🛯 🗛 🐼 -	🕑 Worksheet						
	🚳 •   S 💠 Q		Q 💱 12	🖴   🗖 -					
Geometry Roint Mass 🖄 Thermal Point	Mass 🥪Thickness	BC .							
Outline <b>4</b>	Worksheet								
Project	Geometry								
Geometry	Name	Assignment	Volume (mm <sup>3</sup> )	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
The Shart1	Hex Nut Jam AM	Structural Steel	804.2	6.313e-003	0	0	Not suppre:	Yes	Flex
🛶 🕼 Rotor_compressor	Keeper	Aluminum Alloy	9484.	2.6271e-002	0	0	Not suppre:	Yes	Flex
🛶 🖓 Coupler_3	Packing 1	Copper Alloy	6221.7	5.1641e-002	0	0	Not suppre:	Yes	Flex
🛶 🖓 Coupler_4	Packing 2	Copper Alloy	1927.1	1.5995e-002	0	0	Not suppre:	Yes	Flex
🚽 🖓 Thrust_1	Rotor compressor	Aluminum Alloy	91895	0.25455	0	0	Not suppre:	Yes	Flex
📖 🖓 Coupler_5	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
Keeper	Turbo Housing	Magnesium Alloy	1.9745e+006	3.5542	0	0	Not suppre:	Yes	Flex
Stage Housing	Turbo bushing	Copper Alloy	2126.4	1.765e-002	0	0	Not suppre:	Yes	Flex
Carrier	•								
Coordinate Sustems									
	Graphice Works	heet							
	Graphics Horks	meet							

#### **ANSYS** 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).** 

#### **ANSYS** 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.



#### **ANSYS** 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.





### **ANSYS** ... 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.

February 27, 2015

13

© 2015 ANSYS, Inc.





Definition					
Туре	Cartesian				
Ansys System	Fregram Controlled				
Origin					
Define By	Geometry Selection				
Geometry	Click to Change				
Origin X	4.5 m				
Origin Y	2.5 m				
Origin Z	1 Z 3.5 m				
Principal Axis					
Axis	×				
Define By	Global X Axis				
Orientation About Principal Axis					
Axis	Y				
Define By	Default				
Directional Vectors					
Transformations					
Base Configuration	Absolute				
Offset X	0. m				
Rotate X	0. °				
Transformed Configuration	[ 4.5 2.5 3.5 ]				

#### **ANSYS** 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.).



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.

Insert Go To	► ₩		Selection	
♀ Hide Body ♀ Hide All Other Bodies			C Apply selected geomet	ry
Suppress Body	£			Selection here will
♀ Hide Face(s)				create the first row o
Isometric View			Location Y	the worksheet.
Restore Default			Location Z	
( Zoom To Fit			Apply To Correspon	iding Mesh Nodes
View				
🖉 Look At			OK	Cancel
Create Coordinate System		_		
Create Named Selection			Convert to nodal name	ed selection immediatel

Selection Name

X

15 © 2015 ANSYS, Inc. February 27, 2015

**RMB**:

### **ANSYS** ... 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

De	tails of "Pressur	e"	<b></b>	De	tails of "Pressur	е" <del>7</del>
E	Scope	- Cr.		=	Scope	
	Scoping Method	Geometry Selection	525F		Scoping Method	Named Selection
	Geometry	Geometry Selection			Named Selectio	
	Definition	Named Selection		E	Definition	1 Face
	Define By	Normal To			Define By	blends
	Туре	Pressure			Туре	Preccure
	Magnitude	0. Pa (ramped)			Magnitude	0, Pa (ramped)
	Suppressed	No			Suppressed	No

 A named selection toolbar provides quick access to basic controls "View > Toolbars > Named Selections":



16 © 2015 ANSYS, Inc. February 27, 2015

#### **ANSYS** F. Workshop 3.2

#### Workshop 3.1 – Named Selections



### **ANSYS** 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.





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**

Select tree objects to use as a template, and select geometry to be used as scoping.

Generate From: Current Selection Current Selection Scope To: Top Holes Ianore Oriain Name Prefix: Apply Tag:

New objects will be scoped to the selected geometry. Geometry can be scoped to individual entities or to groups of adjacent entities.

Generate



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.



#### **ANSYS** ... 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.

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

**ANSYS** ... Object Generator

- 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 Gener	ator	
Select tree obje select geometry	cts to use as a templat to be used as scoping	e, and
Contact:		-
Target:		-
Distance:	Between centroids	
Minimum:	0	mm
Maximum:	0	mm
Scope To:	Adjacent Entitie	s 🔻
Ignore Origin	al: 🔽	
Name Prefix:		
Apply Tag:		
Objects will be centroids on a whose distand lower and upp	e created for any pair ( apposite sides of the co ce fails between the sp per bounds.	of onnection ecified
	- 1	
Generati		
Generat		
Generati		
Generati	<u> </u>	
Generati	<u> </u>	

### **ANSYS** ... 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).

adiinto		4	Mechanical Application Wizard 🛛 🕈
Project	22	Circular - Top Plate To Bottom Plate	Object Generator
Model (A	<b>4)</b> etry diasta Sustana	10/16/2012 10:25 AM	Select tree objects to use as a template, and select geometry to be used as scoping.
	anate bystems ections Circular - Top Plate To Bottom Plate		Reference: Top Plate Holes
→ 🥠 Mesh □ 🖓 Name	d Selections	9 9 9	Distance: Between centroids
	Top Plate Holes		Minimum: 18 m
E 🧭 Stati	Bottom Plate Holes ic Structural (A5)		Maximum: 22 m
	Andrew Children (		Scope To: Adjacent Entities 💌
etails of "Circular - To	op Plate To Bottom Plate"	4	Ignore Original:
1 m 1 m m	ties		Name Prefix:
Graphics Proper			and the second sec
Definition			Apply Tag:
Definition	Structural Steel		Apply Tag: Objects will be created for any pair of centroids
Definition Material Cross Section	Structural Steel Circular	-	Apply Tag: Objects will be created for any pair of centroids on opposite sides of the connection whose
Definition Material Cross Section Radius	Structural Steel Circular 4. mm		Apply Tag: Objects will be created for any pair of centroids on opposite sides of the connection whose distance falls between the specified lower and upper builder
Definition Material Cross Section Radius Suppressed	Structural Steel Circular 4. mm No		Apply Tag: Objects will be created for any pair of centroids on opposite sides of the connection whose distance falls between the specified lower and upper bounds.
Graphics Proper     Definition     Material     Cross Section     Radius     Suppressed     Scope	Structural Steel Circular 4. mm No		Apply Tag: Objects will be created for any pair of centroids on opposite sides of the connection whose distance falls between the specified lower and upper bounds. Generate

## **ANSYS** ... Object Generator

#### Scoping the new objects:



<b>Object Gener</b>	ator
Select tree obje geometry to be	ects to use as a template, and select used as scoping.
Generate Fro	m: Current Selection
Scope To:	Adjacent Entities
Relocate:	Each Entity Adjacent Entities
Ignore Origin	All Entities by Part





#### **ANSYS** 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 Coordinat	e System 🛛 🔽 🛛 🔨	Show Individ	lual and Summary		
Entity	Surface Area (mm²)	Centroid X(mm)	Centroid Y(mm)	Centroid Z(mm)	Body	Туре
2 Faces, Summary	9495.3	49.334	25.049	30.		2
Face 1	4747.7	49.334	25.049	55.	SidePlate2	Plane
Face 2	4747.7	49.334	25.049	5.	SidePlate1	Plane

**NNSYS**<sup>®</sup>

# **ANSYS** 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.



# **ANSYS** 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

#### **ANSYS** Named Selections

Worksheet Criteria:

• Insert a new named selection then change the scoping method to "Worksheet".

etails of "Selection 2"	<b>4</b>
Scope	
Scoping Method	Worksheet 🗾 🗾
Geometry	No Selection
Definition	
Send to Solver	Yes
Visible	Yes
Program Controlled Inflation	Exclude
Statistics	
Туре	Manual
Total Selection	No Selection
Suppressed	0
Hidden	0
	etails of "Selection 2" Scope Scoping Method Second by Definition Send to Solver Visible Program Controlled Inflation Statistics Type Total Selection Suppressed Hidden

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

action	2								
ection	2								
Gono	rato								
Gene	rate								
	100								
Action	Geometry T	ype	Criterion	Operator	Units	Value	Lower Bound	Upper Bound	Coordinate System
Action Add	Geometry T Vertex	ype	Criterion Location X	Operator Range	Units mm	Value N/A	Lower Bound 96.	Upper Bound 98.	Coordinate System Global Coordinate Sy



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!).

rksheet Selec	tion								
	Genera	ite							
8 85	Action	Entity Type	Criterion	Operator	Units	Value	Lower Bound	Upper Bound	Coordinate System
	Add	Edge	Location X	Equal	mm	0	N/A	N/A	Global Coordinate System
		Add Ro	W				0		0
		Insert							
		Modify							
		Delete							

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

Action	Entity Type	Criterion
Add	Edge	Location X
 		Size Type Location X Location Y Location Z Named Selection

Generate

Add Row Insert

Modify Delete

#### **ANSYS** ... 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.

	Gene	erate					
	Action	Entity Type	Criterion	Operator	Units	Value	Lowe
$\square$	Add	Edge	Location X	Equal	mm	0.	
$\square$	Remove	Edge	Location Y	Less Than	mm	10.	
	Add	Edge	Radius	Equal	mm	5	



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

#### **ANSYS** ... 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.

Coordinate Systems		Generate								
Mesh		Action	Entity Type	Criterion	Operator	Units	Value	Lower Bound	Upper Bound	Coordinate S
Named Selections     PatchFace     PatchNodes     Static Structural (A)	N	Add Convert To	Face Mesh Node	Named Selection N/A	Equal N/A	N/A N/A	PatchFace N/A	N/A N/A	N/A N/A	N/A N/A
				+						
				IDSERF		100 000	and the second se			
				Select Items in G	iroup Selection					
				Select Items in G Add to Current S Remove from Cu Create Nodal Na	iroup Selection Irrent Selection med Selection					
			Also yo	Add to Current S Careet Nodal Na	iroup Selection Irrent Selection Imed Selection On an exis	ting geo	metry			



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.
- 34 © 2015 ANSYS, Inc. February 27, 2015

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