

Lecture 3

Introduction to ANSYS Meshing

14.0 Release



Fluid Dynamics

Structural Mechanics

Electromagnetics

Systems and Multiphysics

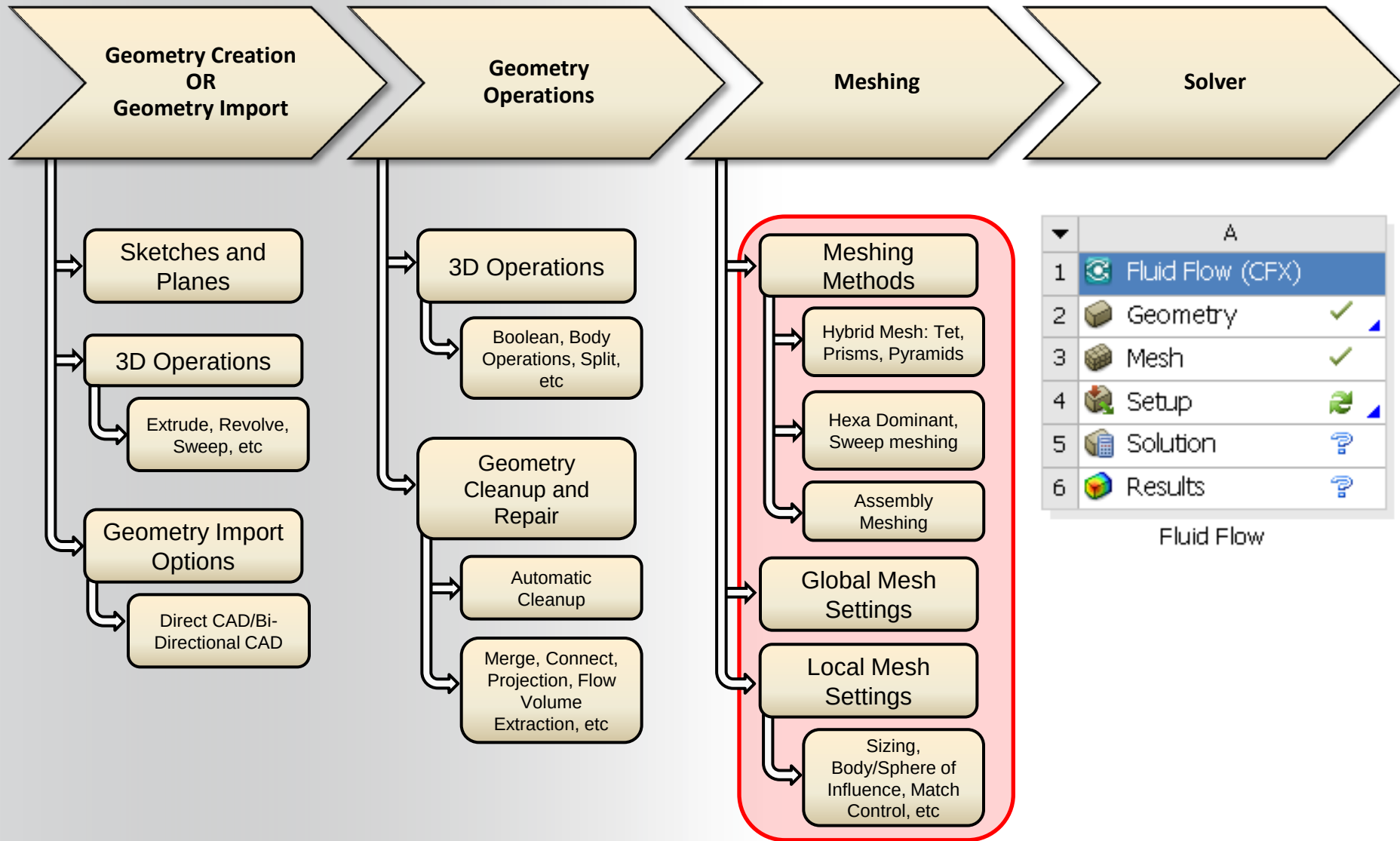
Introduction to ANSYS Meshing

Introduction to ANSYS Meshing

What you will learn from this presentation

- Process for preprocessing using ANSYS tools
- What is the ANSYS Meshing?
- Meshing Fundamentals
- How to launch ANSYS Meshing?
- ANSYS Meshing interface
- Geometry concepts
- Meshing methods
- Mesh controls

Preprocessing Workflow



What is ANSYS Meshing?

ANSYS Meshing is a component of ANSYS Workbench

- **Next generation meshing platform**
- **Combines and builds on strengths of preprocessing offerings from ANSYS: ICEM CFD, TGRID, CFX-Mesh, Gambit**

Creates Meshes for different Physics and Solvers

- **CFD: Fluent, CFX and POLYFLOW**
- **Mechanical: Explicit dynamics, Implicit**
- **Electromagnetic**

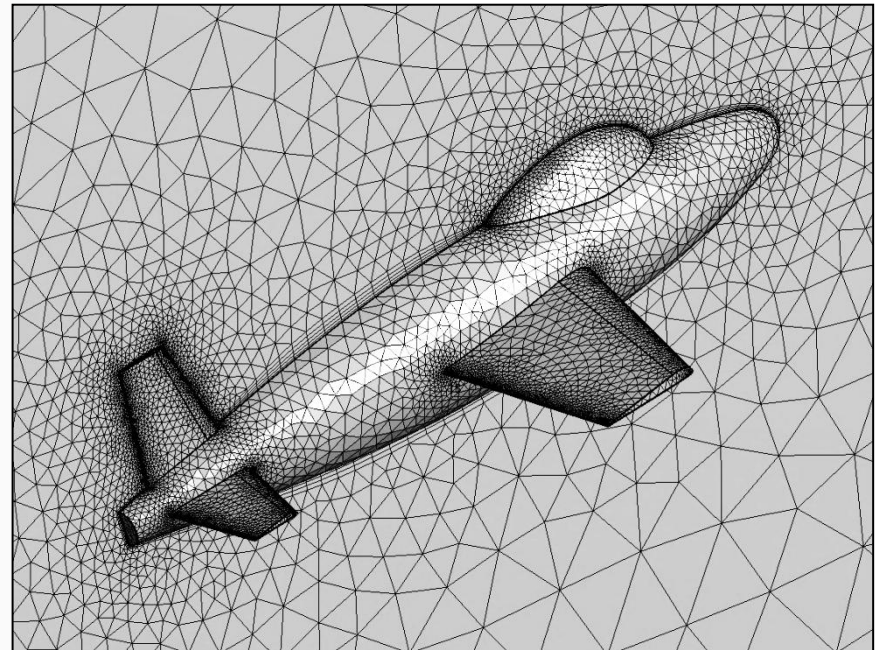
Integrates directly with other WB systems

Purpose of the Mesh

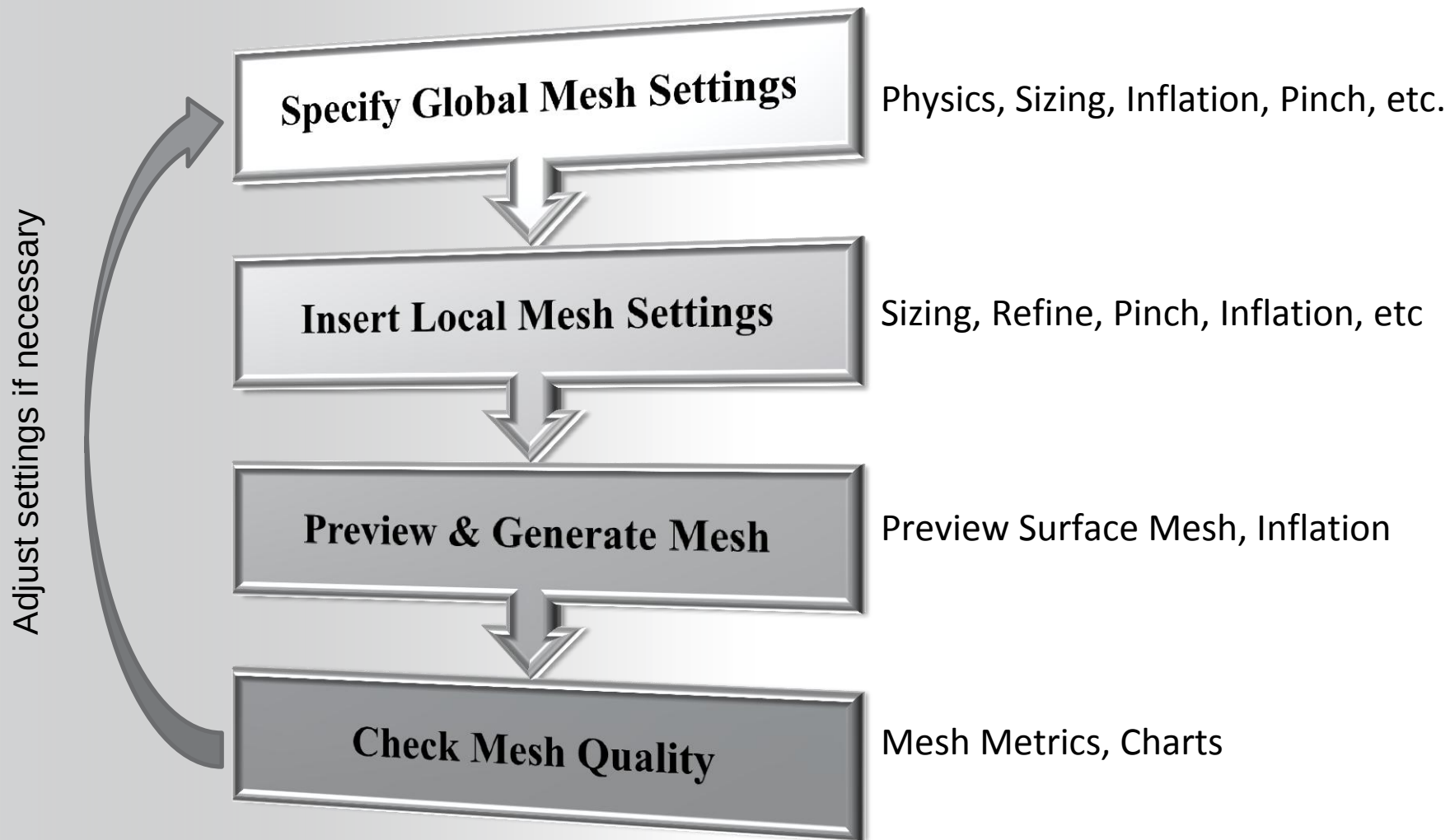
- Equations are solved at cell/nodal locations
 - Domain is required to be divided into discrete cells (meshed)

Requirements

- Efficiency & Accuracy
 - Refine (smaller cells) for high solution gradients and fine geometric detail. Coarser mesh (larger cells) elsewhere.
- Quality
 - Solution accuracy & stability deteriorates as mesh cells deviate from ideal shape



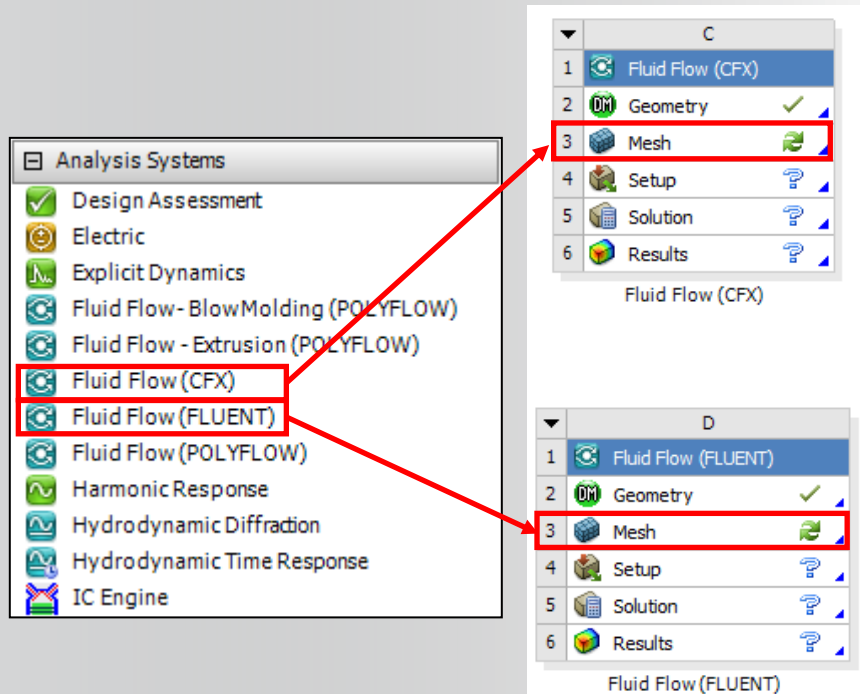
Meshing Process in ANSYS Meshing



Launching ANSYS Meshing

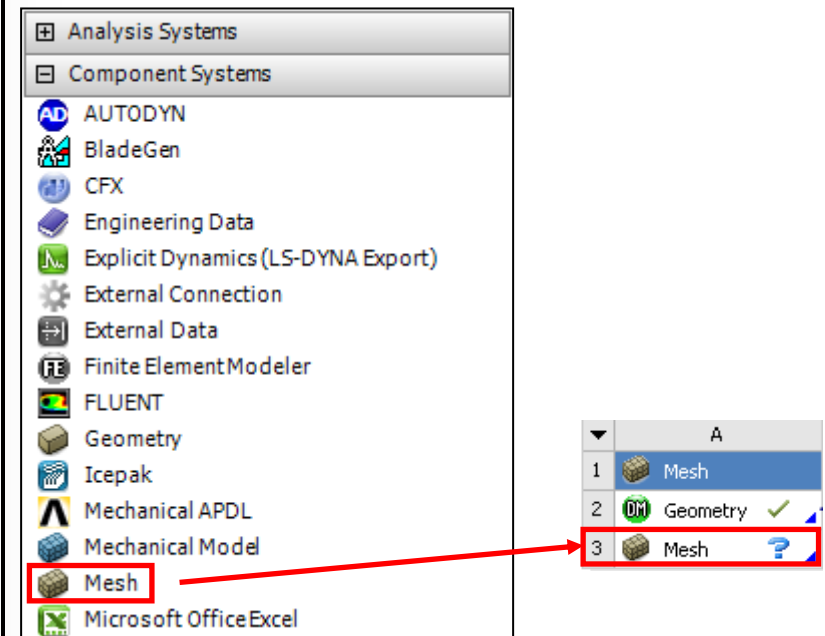
From Analysis Systems

Ex: Fluid Flow (Fluent), Fluid Flow (CFX)



From Component Systems

Ex: Mesh



- ANSYS Meshing is launched within Workbench
- Double click Mesh in the System or right click and select Edit

ANSYS Meshing Graphics User Interface

Toolbars

Outline

Details View

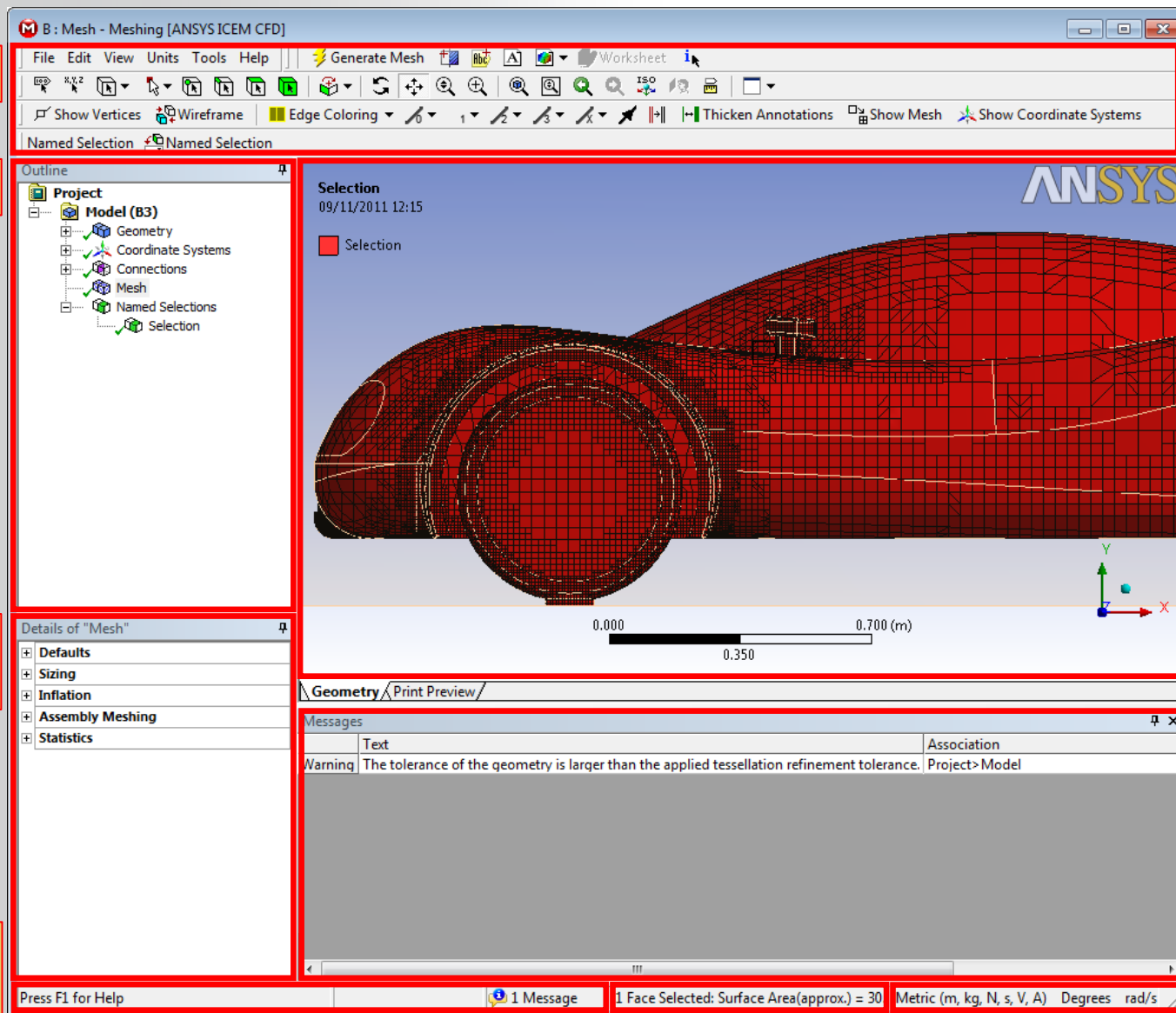
Status/Info Bar

Graphics Window

Message Window

Units Bar

Entity Details Bar

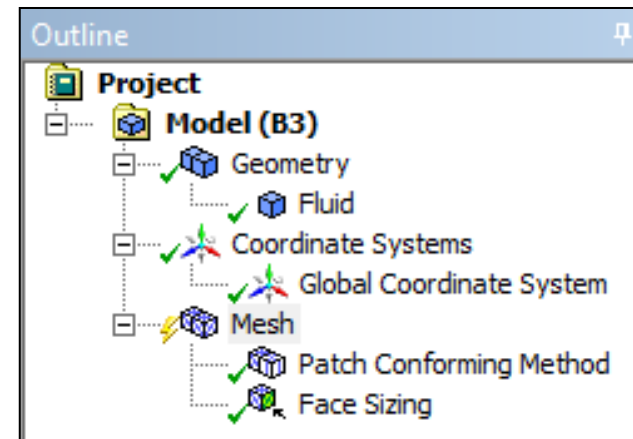


The Outline contains three default sections

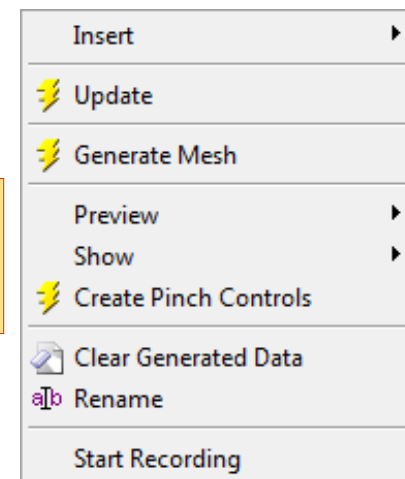
- **Geometry**
 - Bodies
- **Coordinate Systems**
 - Default global & user defined systems
- **Mesh**
 - Meshing operations (controls & methods) displayed in the order in which they are inserted

Right clicking on any object in the tree launches a context sensitive menu

- Example: Context menu for mesh object contains commands to generate, preview, clear mesh etc.

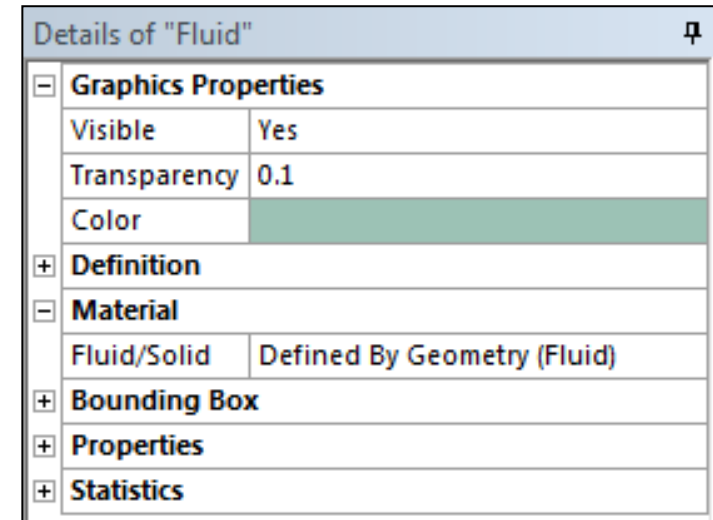
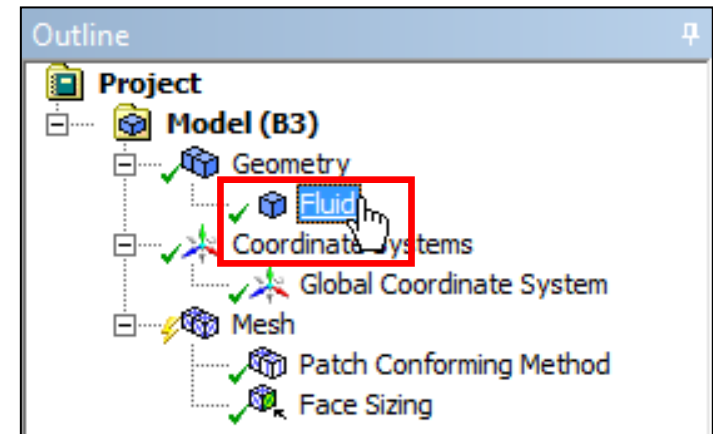


Right click context menu for Mesh object



Accessing Object Details

- If an object in the Outline is selected details relating to that object are displayed in the Details View below
 - Example: Selecting a body (“Fluid”) in the Outline displays Details of “Fluid” which contains graphical and geometric details
 - For access to meshing details click the Mesh object or any of the inserted objects
- The Details View provides options to review, edit or input values for every object in the Tree



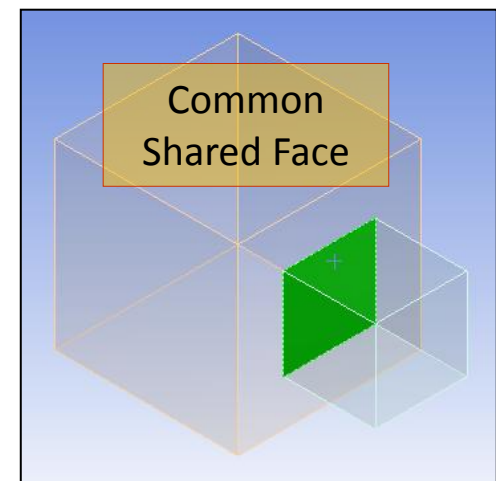
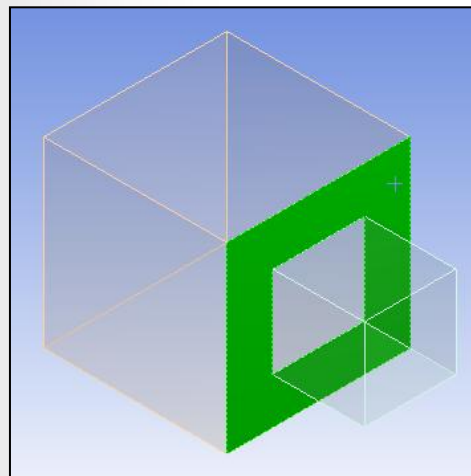
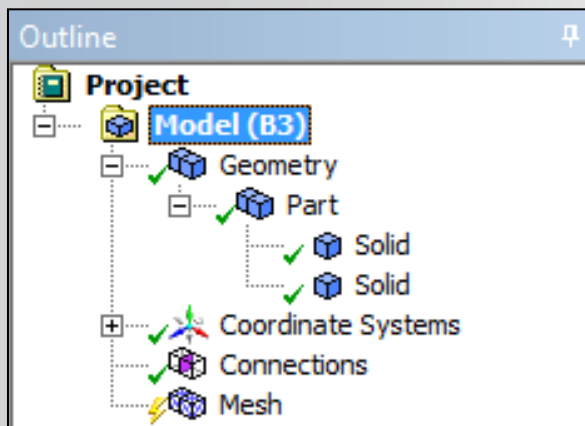
Geometry Configuration – Multi-Body Parts

Handling of Multi-Body Parts

- In DM, where Shared Topology is set to Automatic (default)
 - Faces in contact imprinted & fused to form a single face shared between the two bodies
 - Results in Conformal mesh
 - Common face acts as 'Interior' zone in Fluent

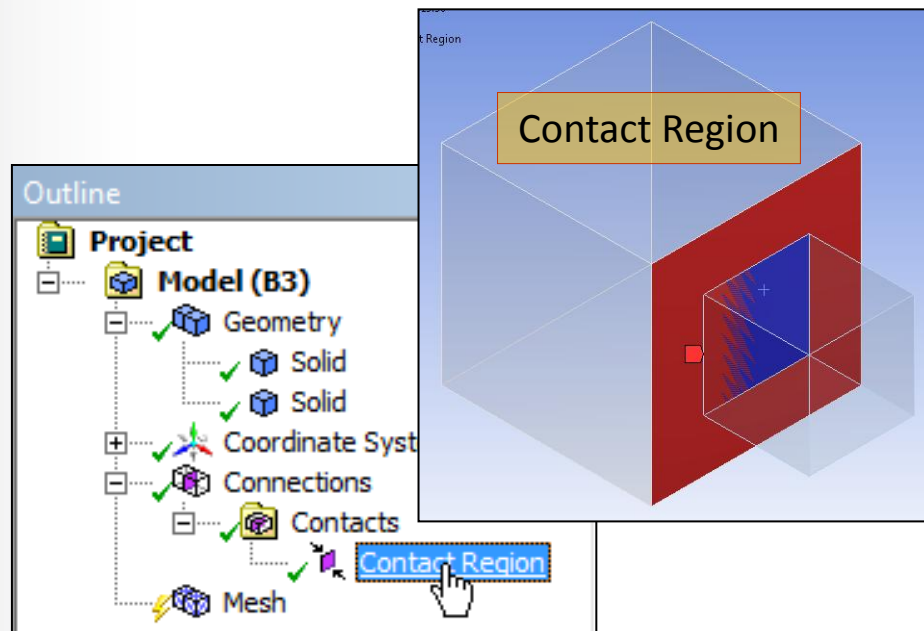
DM Part
Details

Details View	
Details of Part	
Part	Part
Volume	1.125 m ³
Surface Area	7.5 m ²
Bodies	2
Faces	12
Edges	24
Vertices	16
Fluid/Solid	Fluid
Shared Topology Method	Automatic

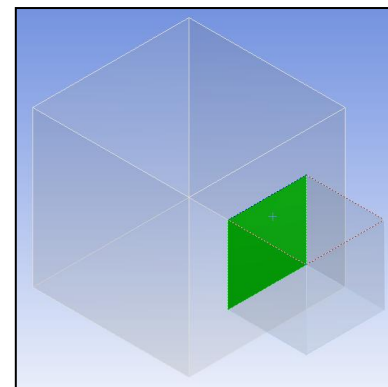
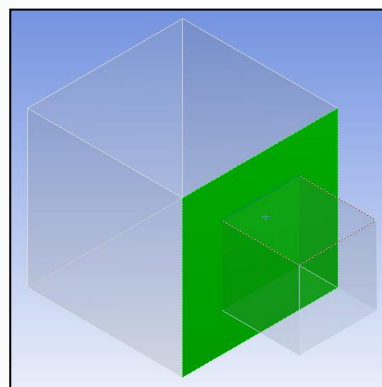


Handling of Multiple Parts

- When multiple parts are attached to a Meshing session a 'Contact Region' is automatically created between those parts
- 2 Faces at Contact Region
- Each part meshed independently
 - Results in a non-conformal interface between the two parts (mesh nodes on bodies do not line up and are not connected)
 - Grid Interface in Fluent or GGI in CFX can be defined for the faces at the contact region



Independent
Faces

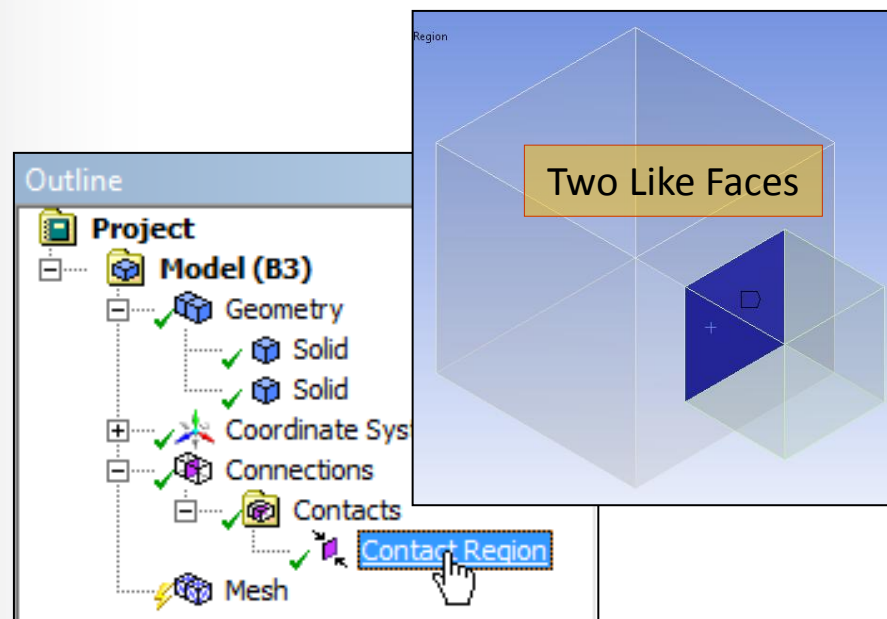


Geometry Configuration – Multi-body Parts

Handling of Multi-Body Parts

- In DM, where Shared Topology is set to Imprints
 - The boundaries of the faces at contact region are imprinted on each other resulting in 'like' faces
 - Contact Region is automatically created at 'like' faces
 - Results in Non-conformal interface
 - Grid Interface in Fluent or GGI in CFX can be defined for the faces at the contact region
 - To create identical mesh on the 'like' faces use 'Match Control' option (mesh will remain unconnected)

Details View	
Details of Part 2	
Part	Part 2
Volume	1.125 m ³
Surface Area	7.5 m ²
Bodies	2
Faces	12
Edges	24
Vertices	16
Fluid/Solid	Fluid
Shared Topology Method	Imprints



Two Meshing Approaches

- **Part/Body based**
 - Meshing occurs at part or body level.
 - Meshing Methods are scoped to individual bodies.
 - Method assignment can be automatic or manual. Bodies contained in one part are conformally meshed.
- **Assembly Meshing**
 - Meshes an entire model (assembly of parts) in one process.
 - Capable of performing boolean operations including volume filling, intersection & combination.
 - Does not require prior fluid body definition or shared topology.
 - Conformal mesh created across parts.
- **Part/Body Meshing & Assembly Meshing not interoperable.**

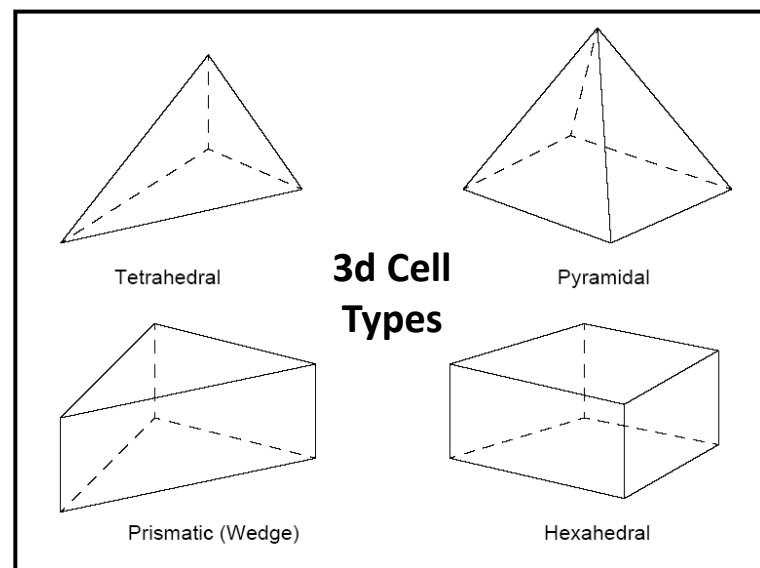
Meshing Approaches (3d Models)

Part/Body based Approach (“Methods”)

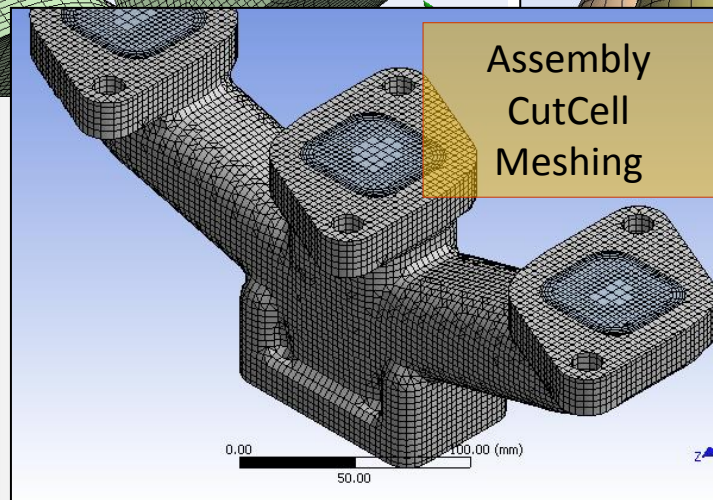
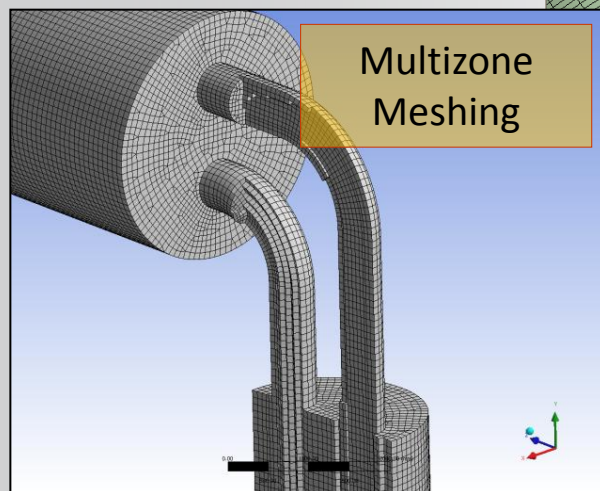
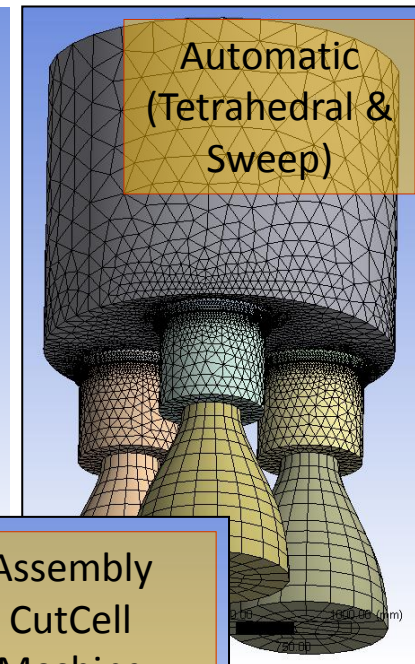
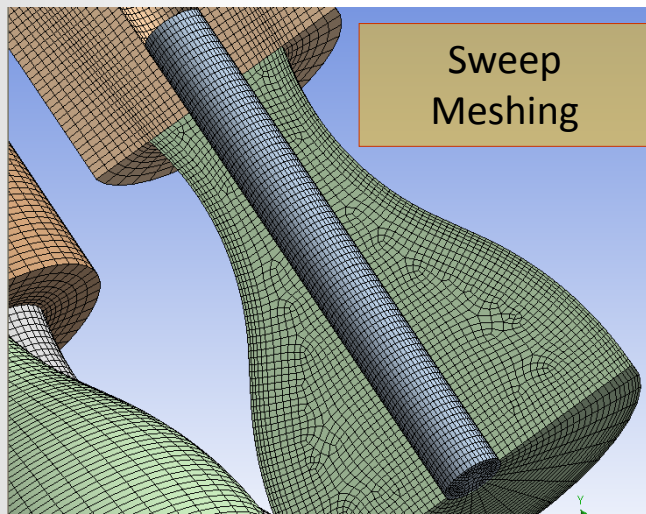
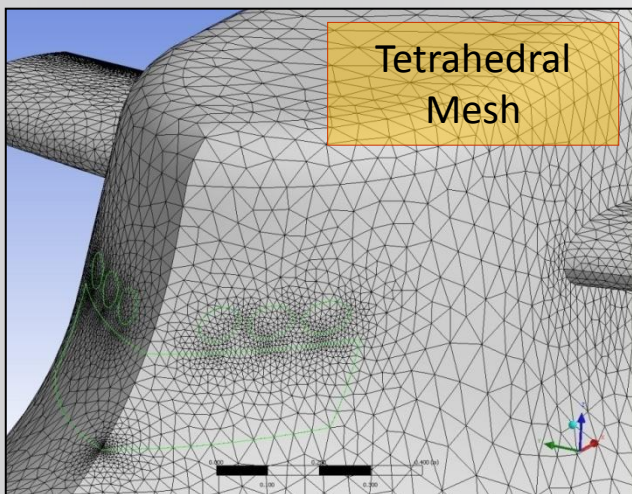
- **Tetrahedrons**
 - Generates tetrahedrons only
- **Sweep**
 - Generates prisms or hexahedrons
- **MultiZone**
 - Mainly hexahedron, can contain prisms, tetrahedrons & pyramids
- **Hex Dominant** (not recommended for CFD)
- **Automatic**
 - Combines Tetrahedrons & Sweep based on complexity of the geometry
- **Interoperability between different meshing methods in Part/Body approach**

Assembly Meshing Approach

- Generates mainly hexahedrons or tetrahedrons



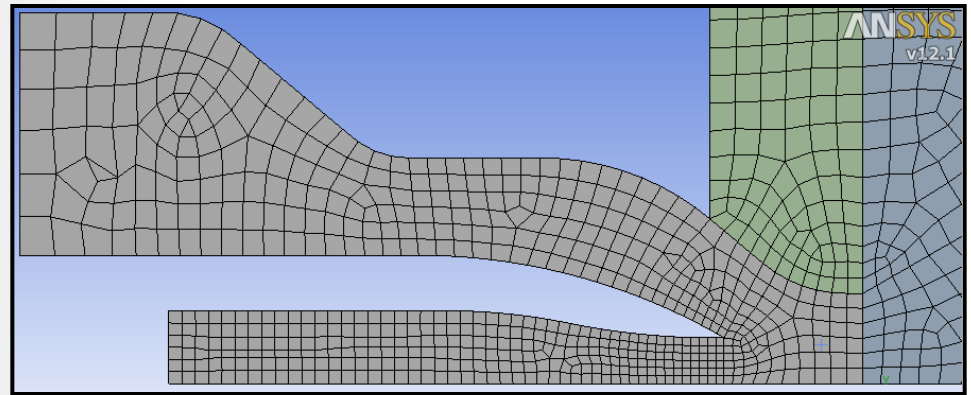
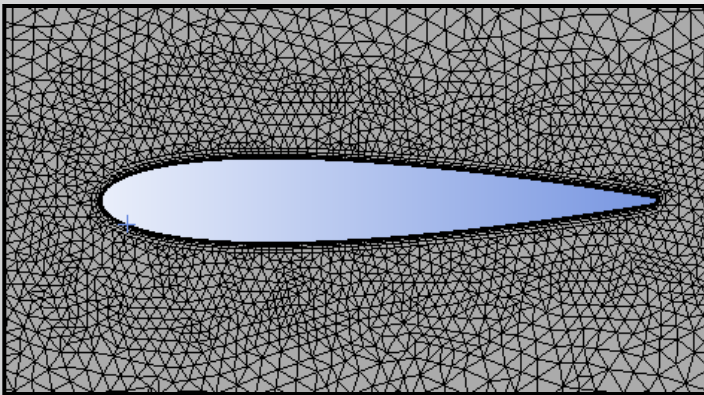
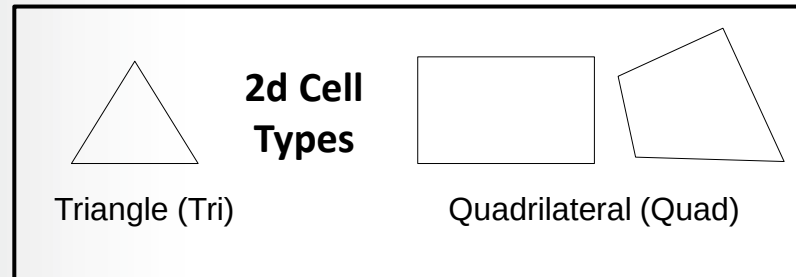
Meshing Methods for 3D Geometry - Examples



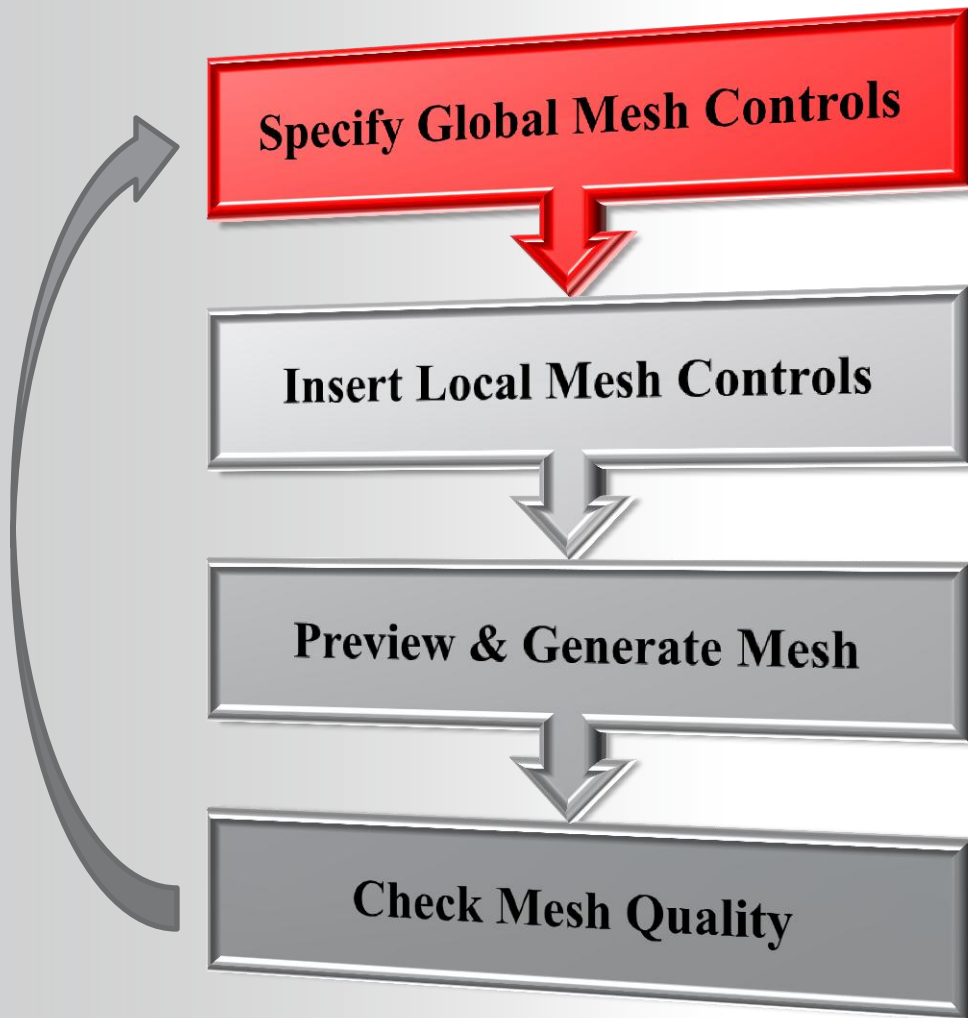
Meshing Methods for 2D Geometry

Four meshing methods available for 2D geometries:

- Automatic Method (Quad Dominant)
- Triangles
- Uniform Quad/Tri
- Uniform Quad



Meshing Process in ANSYS Meshing



Global Mesh Controls

Controls Acting Globally

- **Defaults** : Set Physics and Solver preferences
- **Sizing** : Specify sizing function (curvature, proximity, fixed), mesh sizes, growth rate, etc.
- **Inflation** : Prism layer growth
- **Assembly Meshing** : Activates Assembly meshing approach (None/CutCell/Tetrahedrons)
- **Patch Conforming Options** : Tri Surface Mesher
- **Advanced** : Advanced mesh parameters
- **Defeaturing** : Ignore small features in geometry for improving mesh quality
- **Statistics** : View mesh count and mesh quality

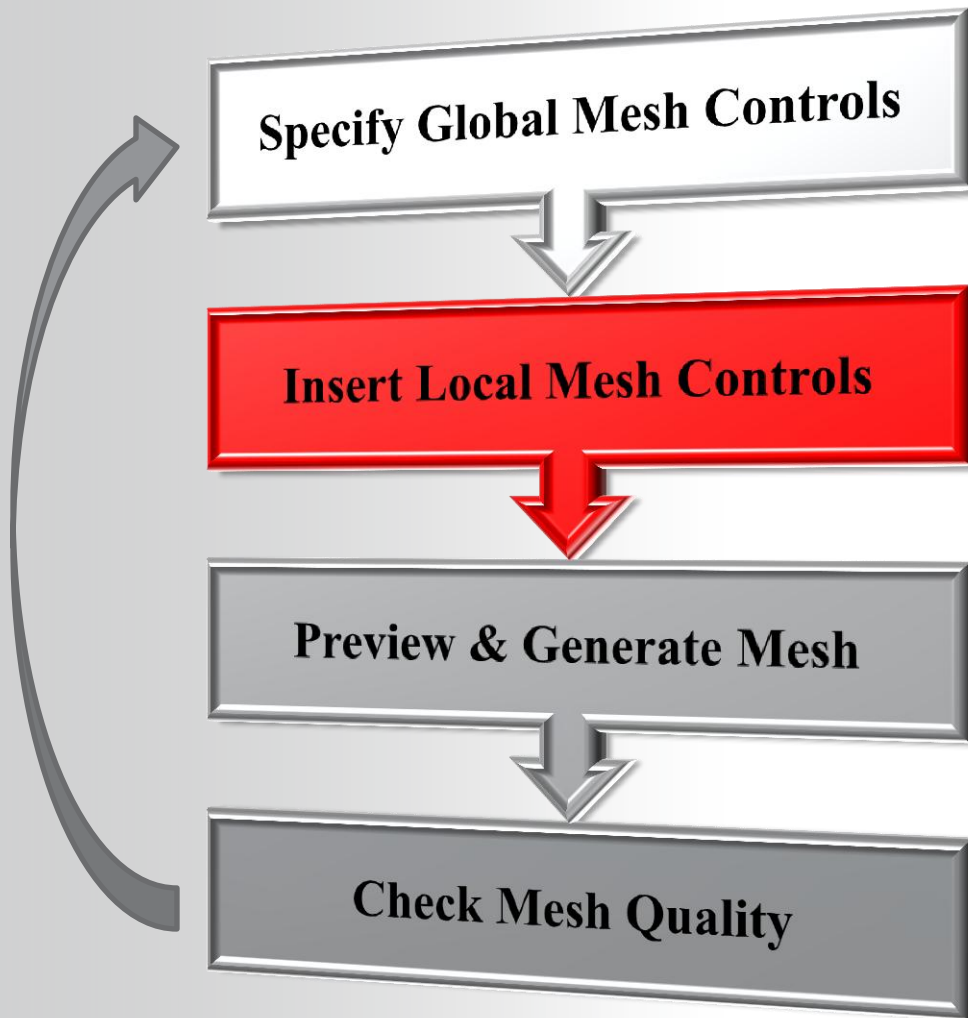
How to Access

- Displayed under Details of “Mesh” when the Mesh object is selected in the Outline

The screenshot shows the ANSYS software interface. The **Outline** panel on the left displays the project hierarchy: **Project** > **Model (B3)** > **Geometry** > **Coordinate Systems** > **Mesh**. The **Mesh** object is highlighted with a red box and a mouse cursor. To the right, the **Details of "Mesh"** panel is open, showing various meshing controls. The **Defaults** section includes Physics Preference (CFD), Solver Preference (Fluent), and Relevance (0). The **Sizing** section includes Use Advanced Size Function (On: Curvature), Relevance Center (Coarse), Initial Size Seed (Active Assembly), Smoothing (Medium), Transition (Slow), Span Angle Center (Fine), and Curvature Normal Angle (Default (18.0 °)). The **Assembly Meshing** section includes Program Controlled, Dropped, 0, Yes, Dimensionally Reduced, and Disabled. The **Advanced** section includes Default (7.7699e-004 m), No, and On. The **Defeaturing** section includes Defeaturing Tolerance (Default (4.3166e-004 m)). The **Statistics** section includes Nodes, Elements, and Mesh Metric (None).

Details of "Mesh"	
Defaults	
Physics Preference	CFD
Solver Preference	Fluent
☐ Relevance	0
Sizing	
Use Advanced Size Function	On: Curvature
Relevance Center	Coarse
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Slow
Span Angle Center	Fine
☐ Curvature Normal Angle	Default (18.0 °)
☐ Min Size	Default (8.6332e-004 m)
	Default (8.6332e-002 m)
	Default (0.172660 m)
	Default (1.20)
	1.0 m
	None
	Smooth Transition
	0.272
	5
	1.2
	Pre
	No
	None
	Program Controlled
	CFD
	Dropped
	0
	Yes
	Dimensionally Reduced
	Disabled
	Default (7.7699e-004 m)
	No
	ing On
☐ Defeaturing Tolerance	Default (4.3166e-004 m)
Statistics	
☐ Nodes	
☐ Elements	
Mesh Metric	None

Meshing Process in ANSYS Meshing

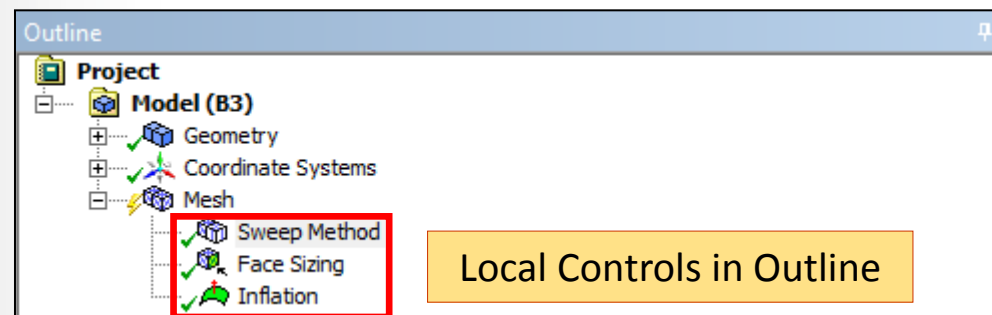
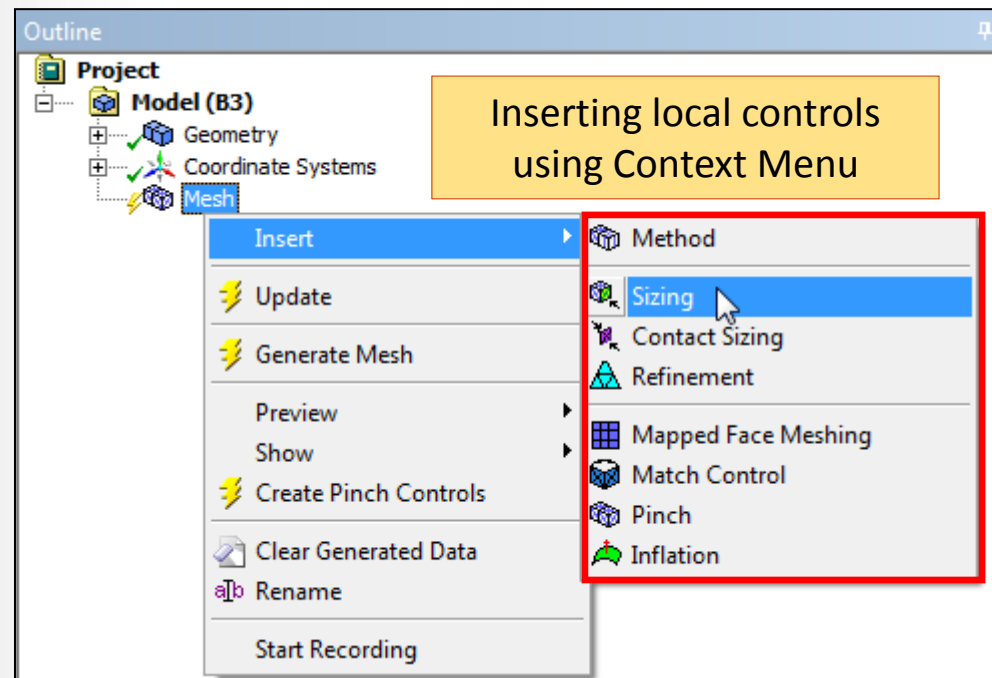


Controls Acting Locally

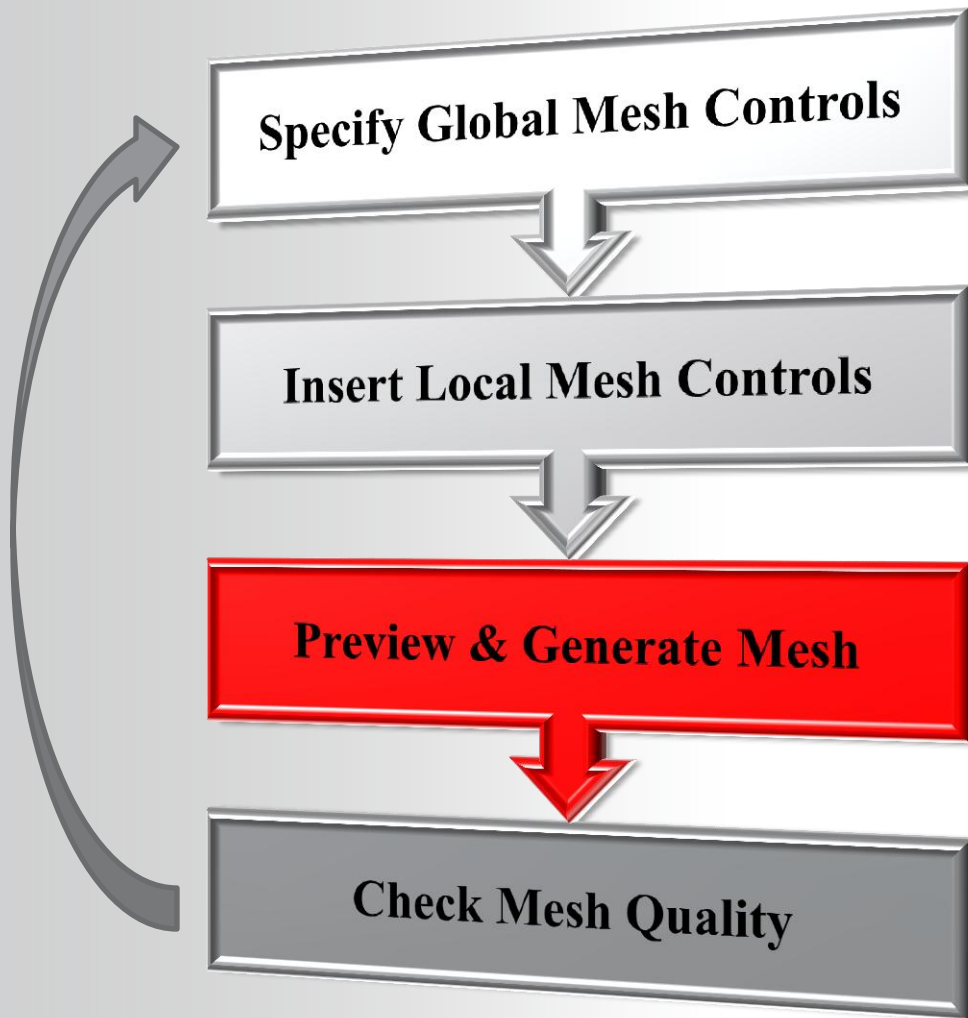
- Used to control the mesh locally by scoping the following options to geometric entities (bodies, faces, edges) as required
 - Sizing
 - Contact Sizing
 - Refinement
 - Mapped Face Meshing
 - Match Control
 - Pinch
 - Inflation

How to Access

- Context menu accessed by right clicking Mesh in Outline

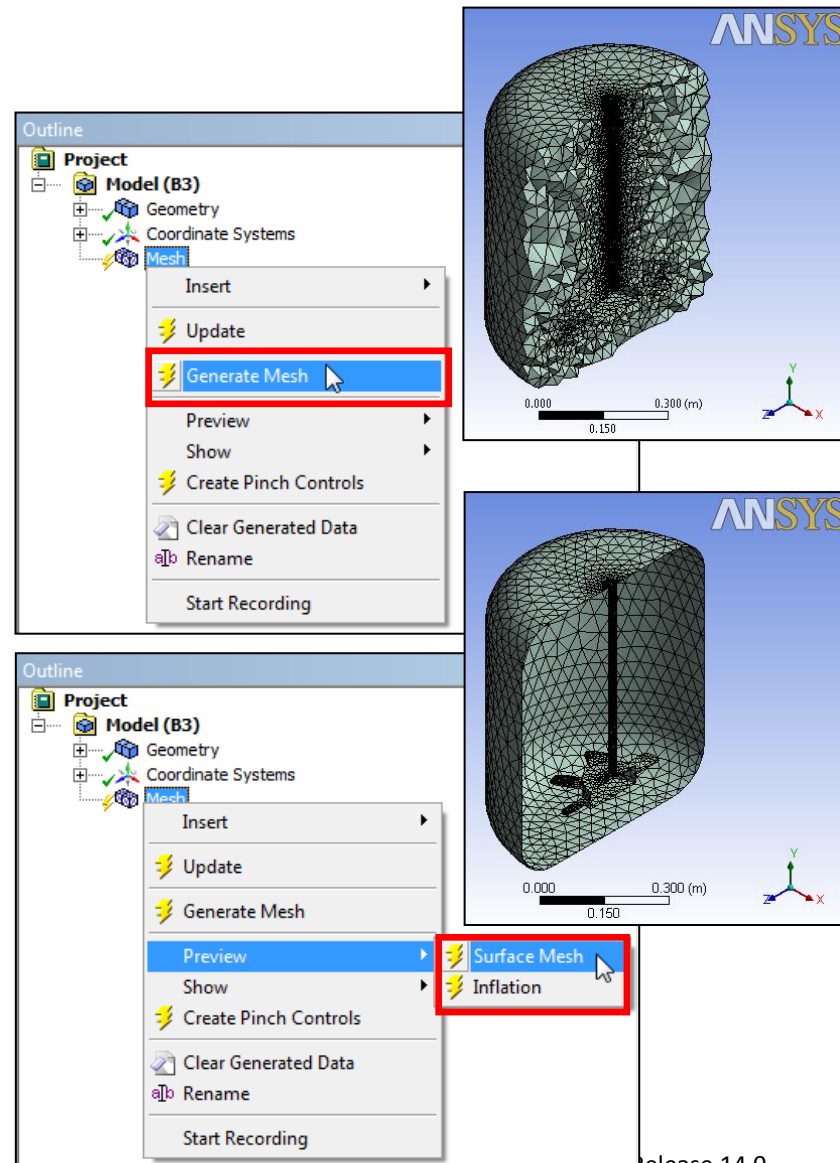


Meshing Process in ANSYS Meshing



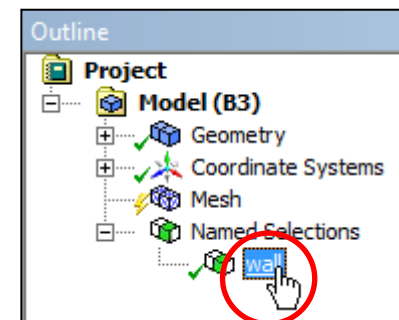
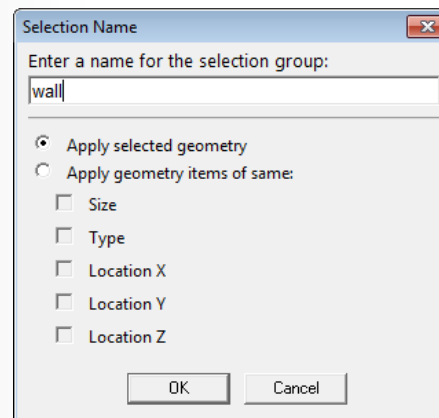
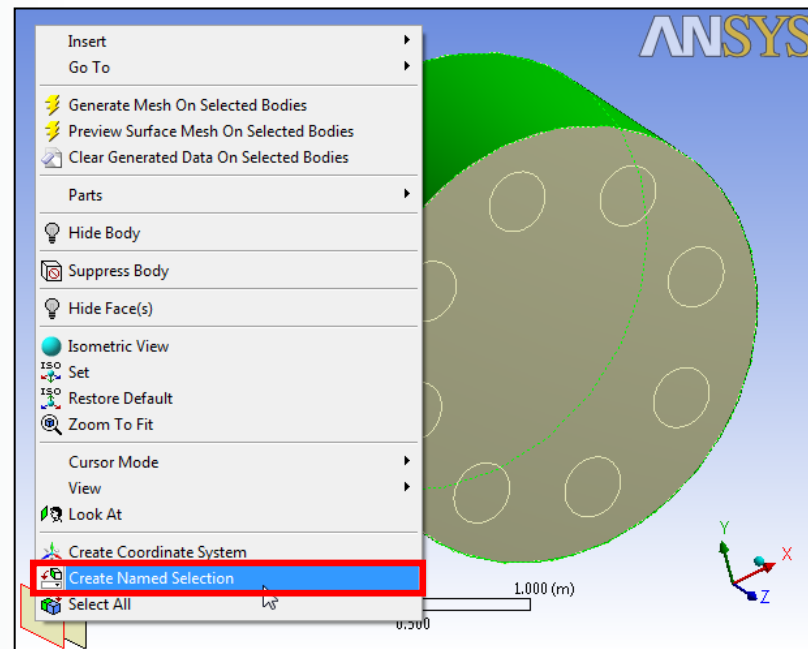
How to Generate a Mesh

- Right click Mesh in the tree for context menu
- Generate Mesh
 - Generates entire volume mesh on all bodies
 - Right click on individual bodies to generate mesh on selected body
- Preview
 - Generates surface or inflation mesh only
 - Allows surface or inflation mesh quality to be checked before volume meshing
 - Not available when using Patch Independent Tetrahedron, MultiZone or Assembly Meshing.



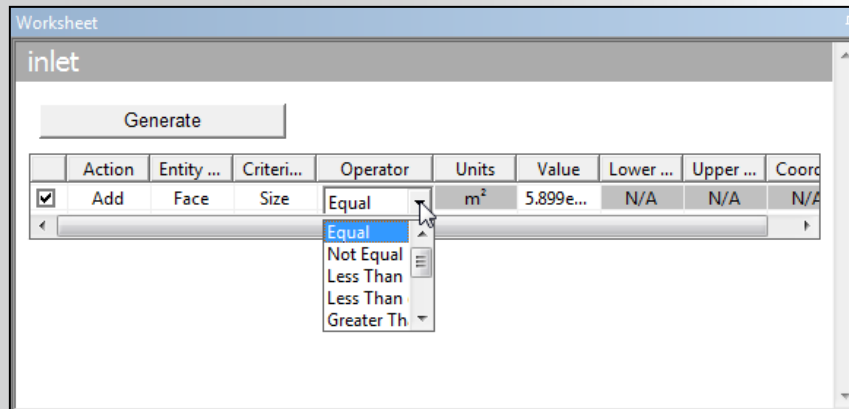
Named Selections

- **Used to assign a name to one or more selected entities**
 - Can be created in Meshing by selecting entity(s) and right click or those created in DM or CAD can be transferred to Meshing
 - Entities within a Named Selection must be of the same topology (edge/surface/volume)
 - Easy to reselect groups that will be referenced often
 - Named selections are automatically transferred to solver modules like Fluent or CFX-Pre (transfer can be deactivated)
 - Listed under Named Selections object in Outline

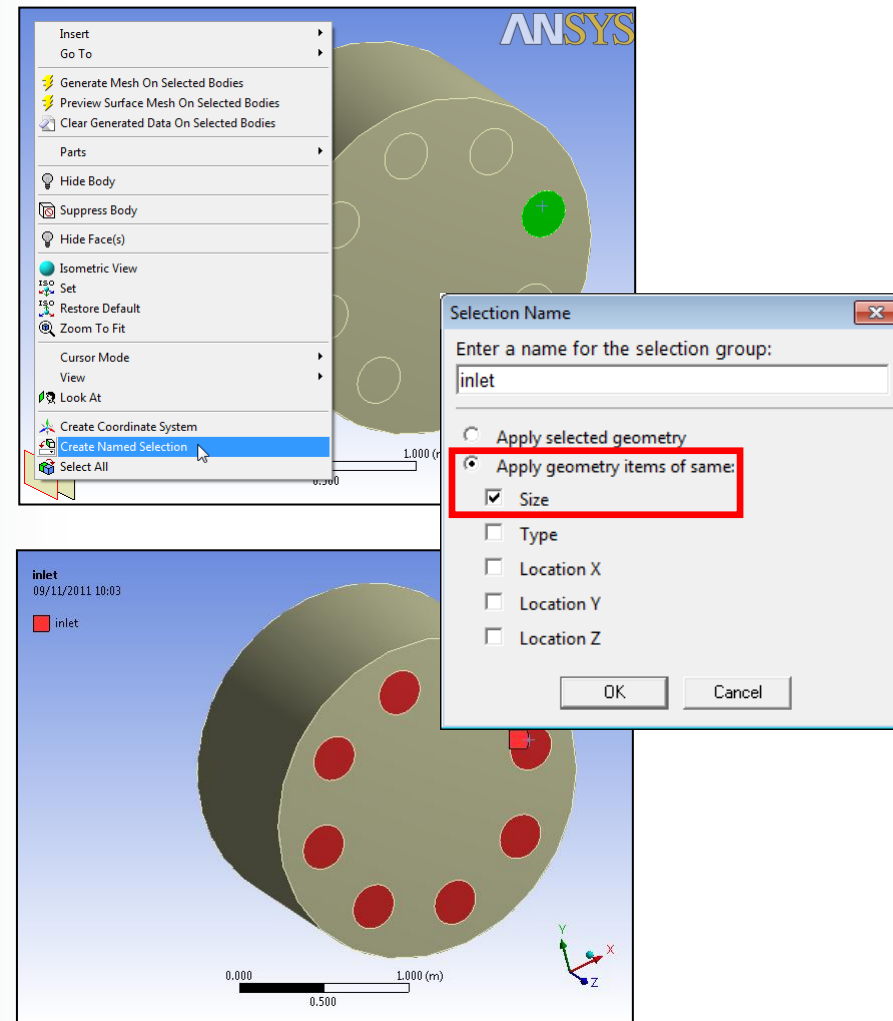


Named Selections

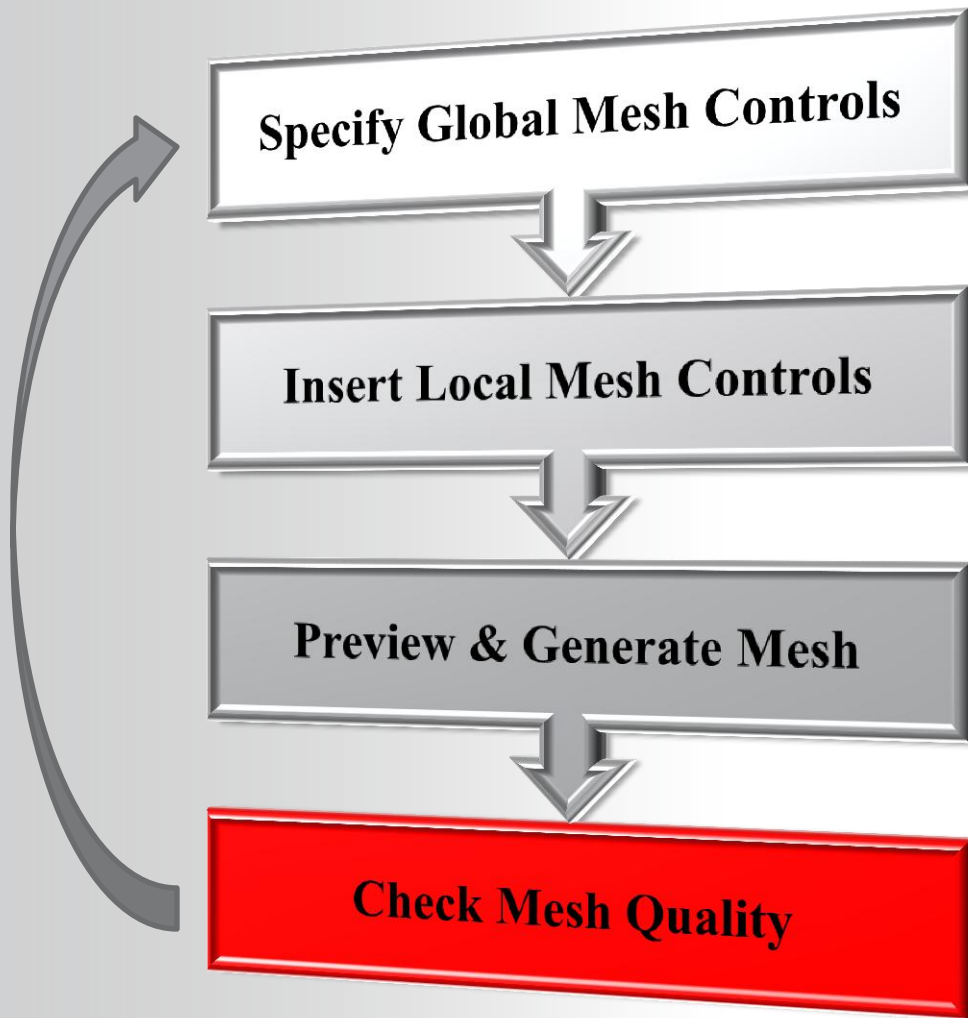
- Named Selections can be applied to the entities of the same size, type or location, by using selection options
- A worksheet enables selection criteria to be modified and Named Selections regenerated using new criteria if required



- Drop down boxes display options
- Worksheet toggle button  Worksheet

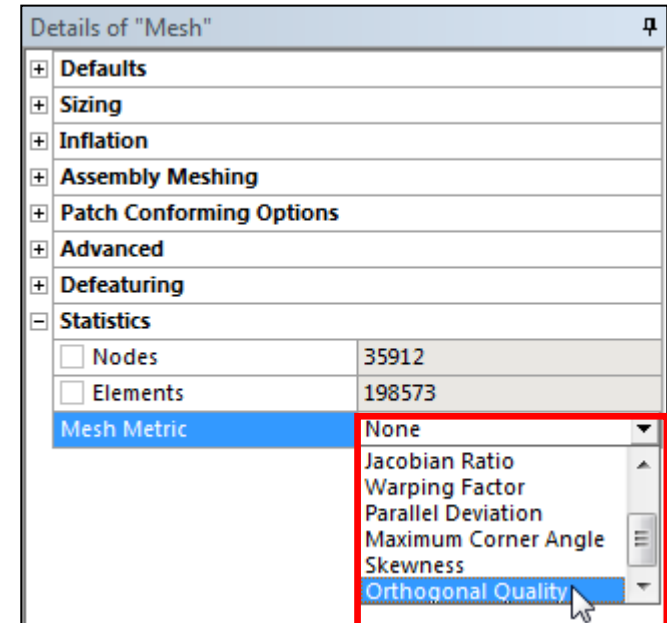


Meshing Process in ANSYS Meshing



Mesh Statistics and Mesh Metrics

- **Displays global Node/Element count and quality**
 - For per-body statistics select body in Tree
- **Quality defined by Metrics;**
 - Element Quality
 - Aspect Ratio
 - Jacobean Ration
 - Warping Factor
 - Parallel Deviation
 - Maximum Corner Angle
 - Skewness
 - Orthogonal Quality
- **Shows min, max, average and standard deviation**
- **Different physics and different solvers have different criteria for mesh quality**

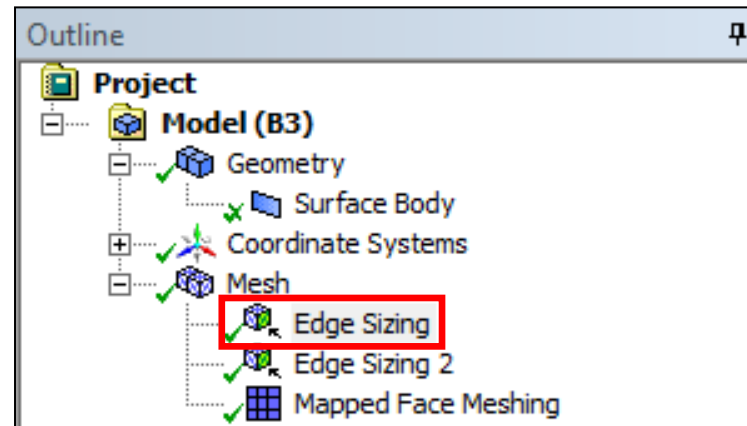
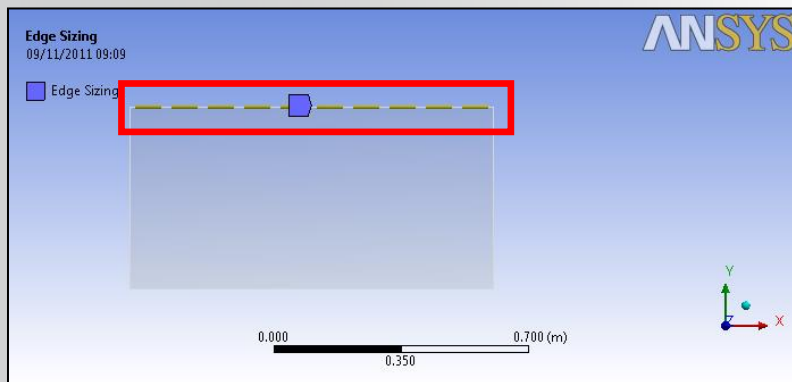


Statistics	
☐ Nodes	35912
☐ Elements	198573
Mesh Metric	Orthogonal Quality
☐ Min	0.279779307354839
☐ Max	0.99534728249954
☐ Average	0.857022319935802
☐ Standard Deviation	8.65382492188541E-02

Parameterization in ANSYS Meshing

Parameterized Meshing controls

- Global controls and local controls
- Selection of parameter promotes the parameter to the WB project page
 - Same process as DM
- Geometry and Meshing parameters can be related using expressions in the parameter manager
- Example – Parameterized edge sizing



Details of "Edge Sizing" - Sizing	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge
Definition	
Suppressed	No
Type	Number of Divisions
P Number of Divisions	10
Behavior	Hard
Bias Type	No Bias

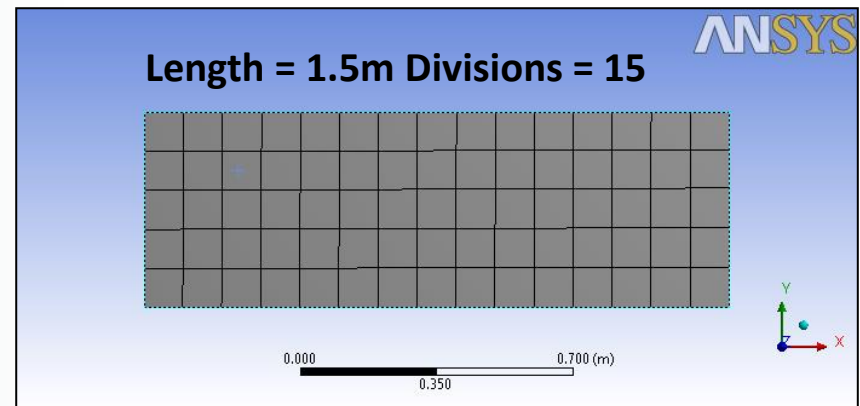
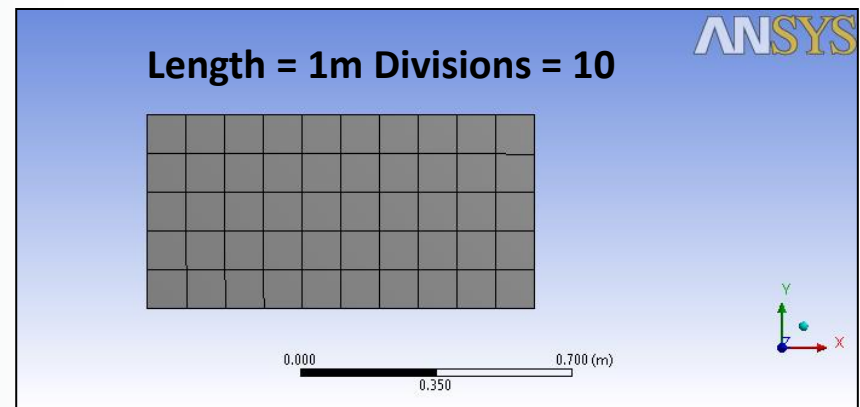
Parameterization Example

- In this example the parameterised edge sizing is expressed as a function of the length parameter (from DM). Expressions created in Workbench.

Outline of All Parameters				
	A	B	C	D
1	ID	Parameter Name	Value	Unit
2	Input Parameters			
3	Geometry (A1)			
4	P1	Length	1	
5	Mesh (B1)			
6	P2	Edge Sizing Number of Divisions	10	
*	New input parameter	New name	New expression	
8	Output Parameters			
*	New output parameter		New expression	
10	Charts			

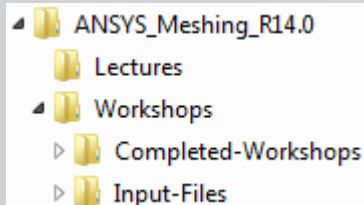
Properties of Outline C6: P2		
	A	B
1	Property	Value
2	General	
3	Expression	10*P1
4	Description	
5	Error Message	
6	Expression	
7	Usage	
8	Quantity Name	Dimensionless

Number of Divisions = 10 x Length

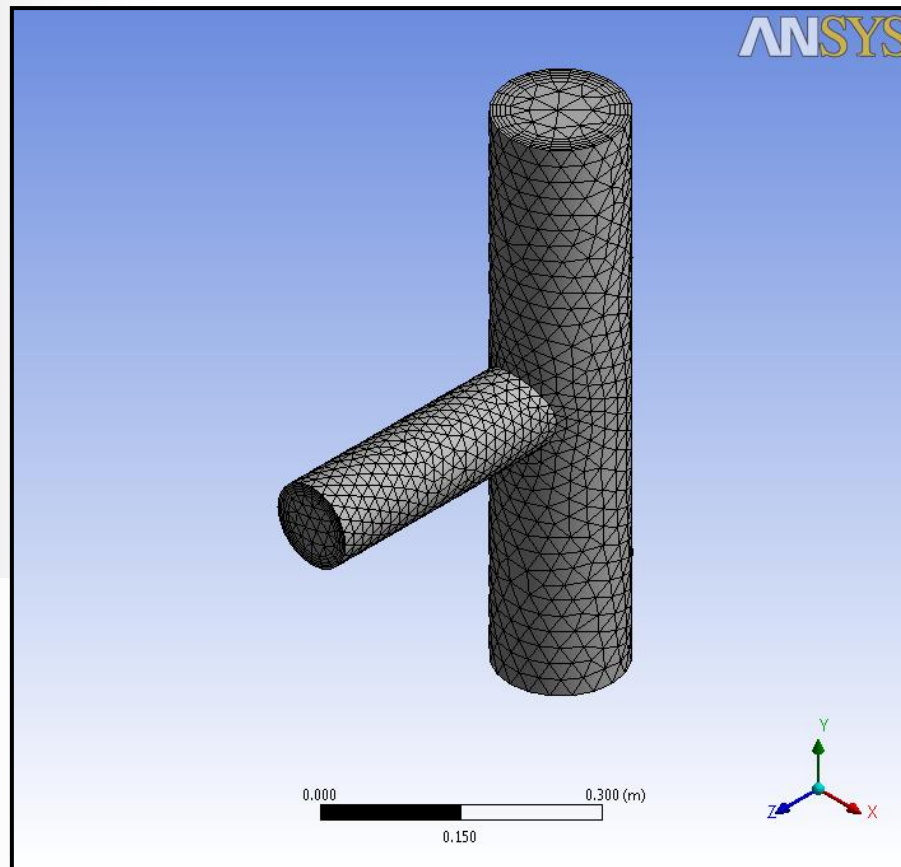


Workshop 1 – ANSYS Meshing Basics

- Workshops are contained within the ANSYS_Meshing_R14.0 folder supplied.
- Some Workshops require supporting files (found in the Input-Files Folder).
- Pre-completed Workshops are supplied in the Completed-Workshops folder for your convenience.



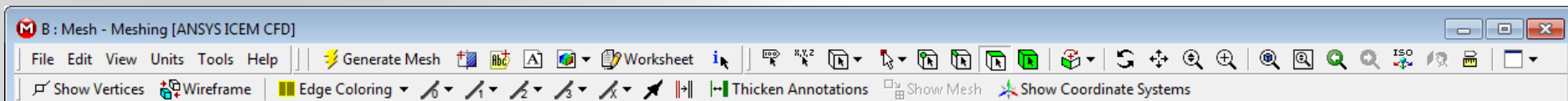
Mesh-Intro_14.0_WS-01_ANSYS_Meshing_Basics.pdf
Mesh-Intro_14.0_WS-02_Meshing_Methods.pdf
Mesh-Intro_14.0_WS-03_Global_Mesh_Controls.pdf
Mesh-Intro_14.0_WS-04_Local_Mesh_Controls.pdf
Mesh-Intro_14.0_WS-05a_Applications_Mixing_Tank.pdf
Mesh-Intro_14.0_WS-05b_Applications_Auto_Aero.pdf
Mesh-Intro_14.0_WS-05c_Applications_Chamber.pdf
Mesh-Intro_14.0_WS-05d_Applications_Assembly_Meshing.pdf
Mesh-Intro_14.0_WS-05e_Applications_Manifold.pdf



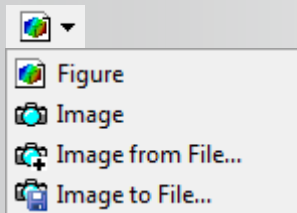
Contents

- **Standard Toolbar**
- **Model Toolbar**
- **Mesh Toolbar**
- **Coordinate Systems**
- **Named Selections as Boundary Conditions**
- **Connections**

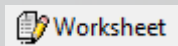
Standard toolbar



- Create sectional plane to view cut section of geometry and mesh
- Add a text comment (tag) in Graphics window
- Add comments (notes) within currently highlighted branch in Outline Tree
- Capture Figures and Images of graphics view.

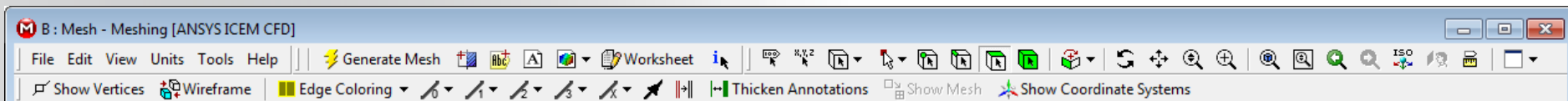


- ‘Image’ is a static screenshot of model ‘Figure’ is dynamic screenshot



- Access Worksheet for recorded mesh operations or Name Selection criteria

Standard toolbar

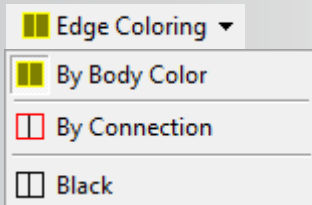


Show Vertices

- Show all vertices on the model

Wireframe

- Toggle wireframe mode on or off
- Edge colouring by body colour, by connection or in black colour

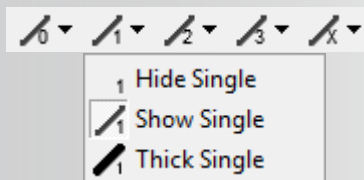


- Show edges/faces with free (0, blue), single (1, red), double (2, black), triple (3, magenta) and multiple (x, yellow) connectivity

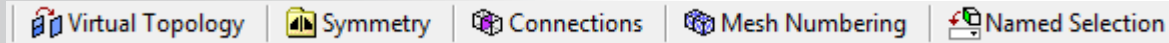


- Display edge direction sense

- Edge thickening by connection

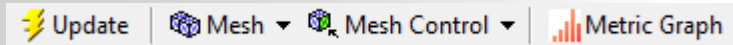


Model toolbar



- **Active when 'Model' object is selected in Outline tree**
- **Virtual Topology : Create virtual entities by merging small faces and/or edges to improve meshing**
- **Symmetry : Define Symmetric portions of geometry with symmetry planes**
- **Connections : Define Contact regions, Interfaces, etc.**
- **Mesh Numbering : Renumbering the nodes and elements**
- **Named Selections : Define named selections**

Mesh toolbar



- **Active when 'Mesh' object is selected in Outline tree**
- **Update** : Update geometry, generate mesh or write output data
- **Mesh** : Generate or preview mesh in ANSYS Meshing
- **Mesh Control** : Contains list of meshing controls like Mesh method, Sizing, Inflation, etc. that can be inserted in the outline tree
- **Metric Graph**: Display or hide the mesh metrics graph
- **Options** : Open Meshing Options panel for defining Physics preference and Mesh method

Coordinate Systems

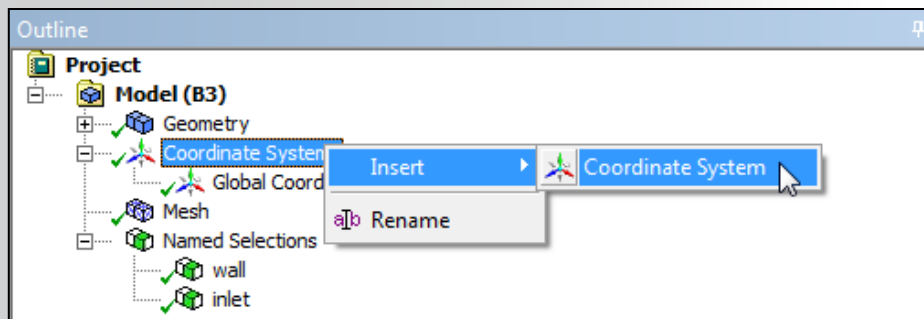


- Create new Coordinate systems
- Required during 'Match Control' operation
- Global coordinate system is available by default
- Following transformations can be performed on new coordinate system

 – Offset (x)

 – Rotate (x)

 – Flip (x)



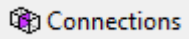
Details of "Coordinate System"	
Definition	
Type	Cartesian
Coordinate System	Program Controlled
Origin	
Define By	Global Coordinates
Origin X	1. m
Origin Y	2. m
Origin Z	3. m
Location	Click to Change
Principal Axis	
Axis	X
Define By	Global X Axis
Orientation About Principal Axis	
Axis	Y
Define By	Default
Directional Vectors	
Transformations	
Base Configuration	Absolute
Transformed Configuration	[1. 2. 3.]

Geometry Selection
Global Coordinates

Named Selections as Boundary Conditions

- Named Selections are also used to assign names to Boundary Zones and Cell Zones and reference those in the Solvers like Fluent or CFX. Particularly the Named Selections can be used to group regions for CFX-Pre
 - In Fluent, appropriate boundary type is automatically assigned if the Named Selection label contains the keywords pertaining to the boundary type
- Examples of how the Named Selections can be used to automatically assign boundary condition types in Fluent:
 - A 3D body can be assigned a named selection of “fluid_1” if that body represents a fluid domain. Fluent will assign a zone type of fluid to that cell zone when the mesh is transferred to Fluent
 - If a set of faces are assigned a Named Selection label of “pressure_far_field_2”, Fluent will assign the boundary type of pressure-far-field when the mesh is imported to Fluent. Note that the name can be any variation containing the keywords “pressure”, “far”, and “field”, like “far_pressure_field_2”, or “farpressurefield_2” or “pressure-farfield_2” and it will still be recognized as pressure-far-field boundary type when transferred to Fluent
 - A Named Selection label containing the keyword “inlet” and no other keywords pertaining to other boundary condition types will be assigned a Velocity-Inlet boundary condition type. Similarly, a Named Selection label containing the keyword “outlet” and no other keywords pertaining to other boundary condition types will be assigned a Pressure-Outlet boundary condition type
 - Wall boundary type is assigned to a Named Selection label containing “Wall” or if it doesn’t contain keywords matching other boundary types

Connections



- **Connections are used to establish surface to surface contact between two parts, mainly used for Mechanical Analysis**
- **These contact regions are automatically resolved during a Mechanical analysis, but will not be resolved in a CFD analysis**
- **To resolve them, the faces in the contact regions should be made into a Named Selection and defined as Interface zones in ANSYS Fluent or General Grid Interface in ANSYS CFX.**
- **Whether a mesh is conformal or non-conformal depends on how the geometry was configured in DM**
 - **Multiple Parts (= Non Conformal Mesh)**
 - **Multi-body with Shared topology as Imprint (= Non Conformal Mesh)**
 - **Multi-body part with Shared Topology as 'Automatic' (= Conformal Mesh)**
 - **However, Patch Independent tetrahedron meshing method with Match Mesh Where Possible option may generate conformal grids whatever is the Shared Topology Method used in DM**