

Lecture 5 Modeling

14.0 Release

Fluid Dynamics

Structural Mechanics

Electromagnetics

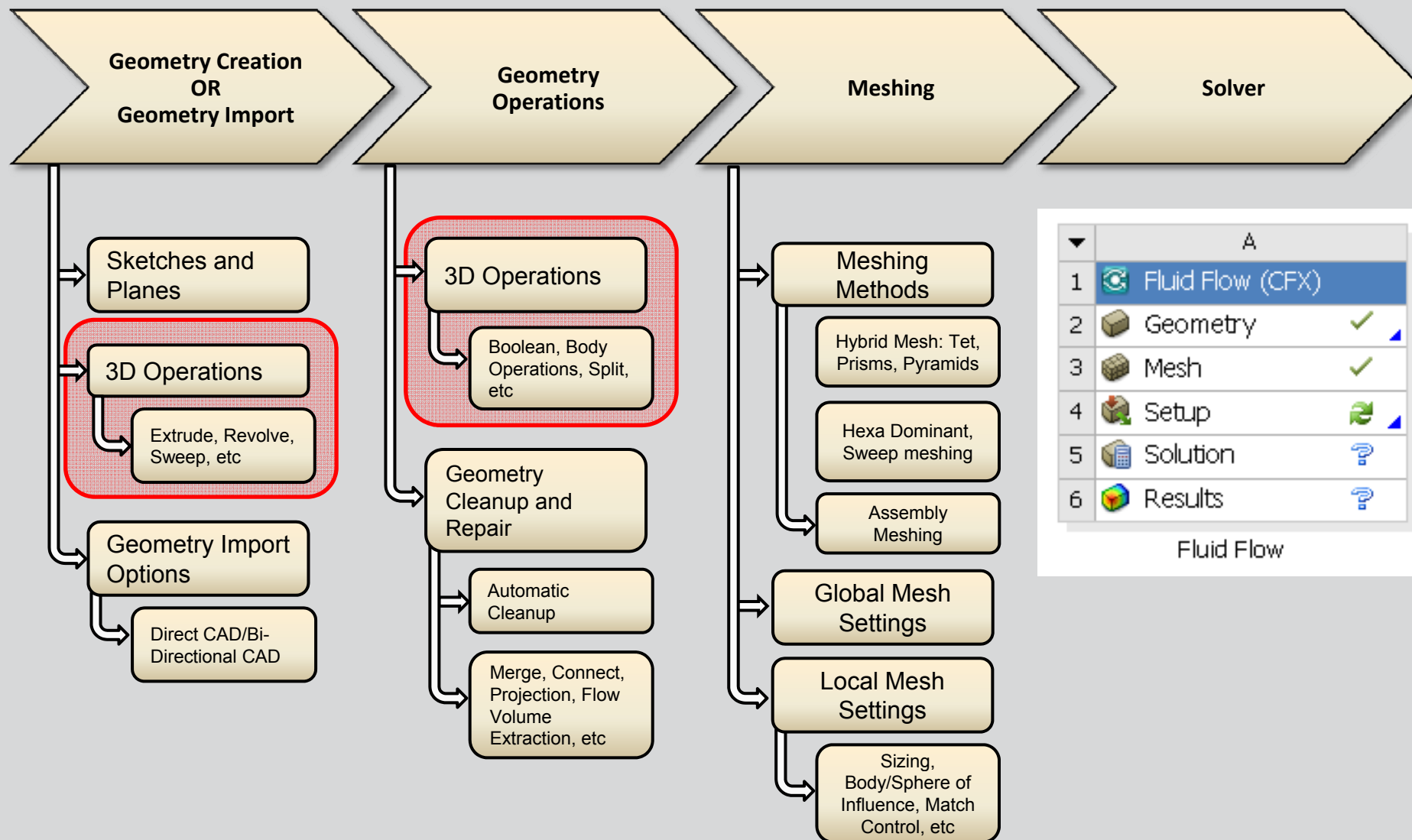
Systems and Multiphysics

Introduction to ANSYS DesignModeler

What will you learn from this presentation:

- **DesignModeler Concepts**
 - **Body Types**
 - **Body States (Frozen & Active)**
- **How to create 3D geometry?**
- **How to modify geometry?**
- **How to create 2D geometry?**
- **Concepts in DM from meshing perspective: Multi-body parts, Share topology**

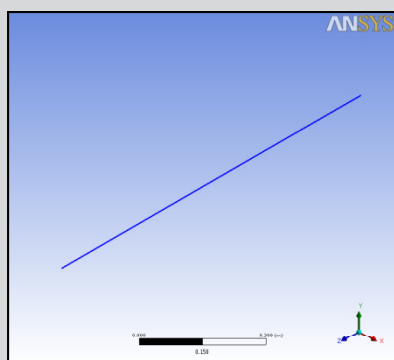
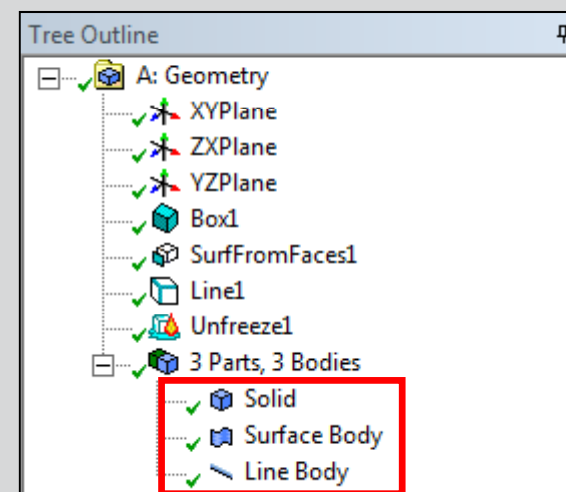
Preprocessing Workflow



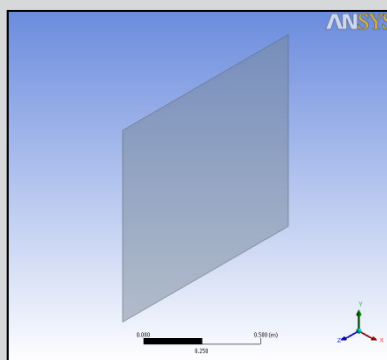
Important DM Concepts: Body Types

In DesignModeler, a body refers to a line, surface or solid

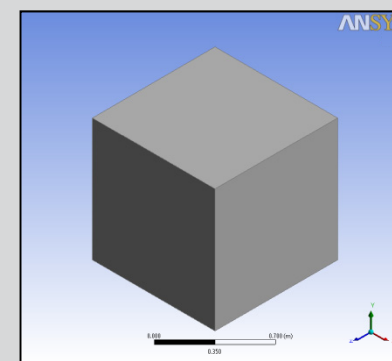
- **Line body**
 - Has length and consists of point lower entities
- **Surface body**
 - Has surface area and consists of point & line lower entities
- **Solid body**
 - Has volume and consists of point, line & surface lower entities
- **Bodies are listed in the Tree Outline identified by default names and icons**



Line body



Surface body



Solid body

Important DM Concepts: Body States

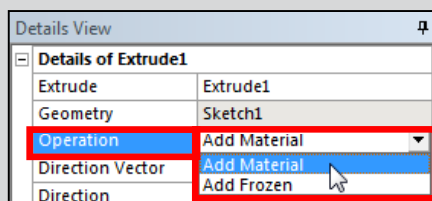
Active & Frozen

- **Bodies can exist in one of two states**
 - Active
 - Frozen
- **Active bodies merge automatically with bodies in contact or overlapping**
- **Frozen bodies remain independent**
- **Why use frozen bodies?**
 - Meshing; It is often more efficient to mesh a series of smaller topologically simpler bodies than one large complex shape
 - Solver; Different physical models and boundary conditions can be applied to different areas of the model if they are defined as separate bodies

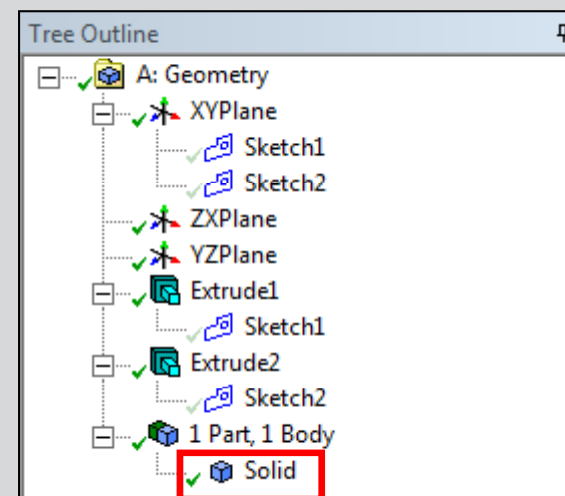
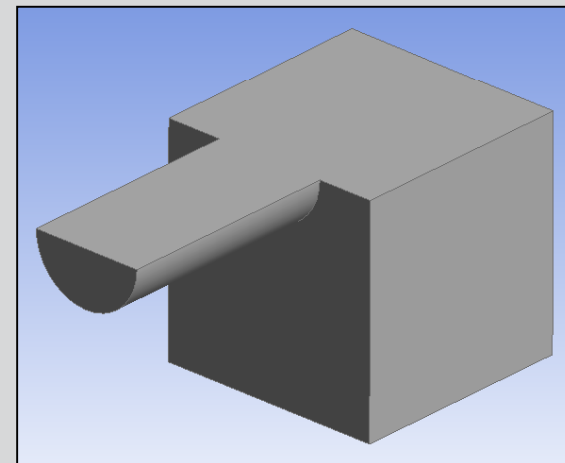
Important DM Concepts: Active Body State

Working with Active Bodies

- Example; Two separate extrusions each creating an active body results in one body
- How to define a body as active?
 - Before a body is created - choose 'Add Material'



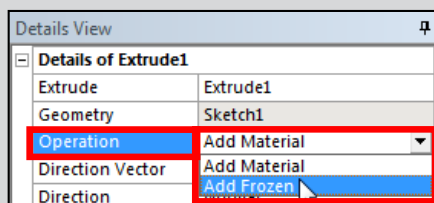
- After a body has been created as frozen - Use 'Unfreeze' operation (available under Tools Menu)
- Displayed as Opaque
- Blue icon under Tree Outline



Important DM Concepts: Frozen Body State

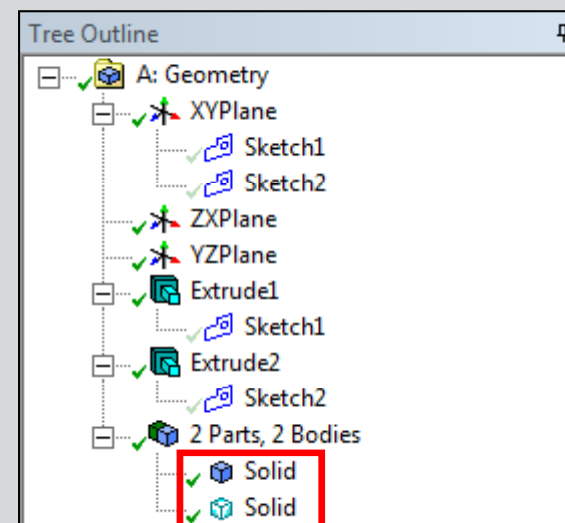
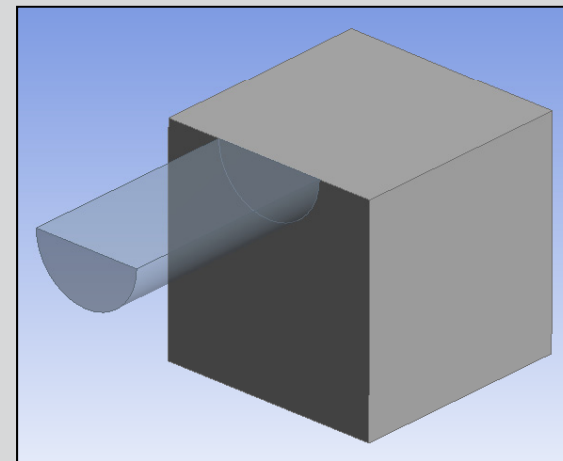
Working with Frozen Bodies

- Example; Two separate extrusions one of which is frozen results in two separate bodies
- How to define a body as frozen?
 - Before a body is created - choose 'Add Frozen'

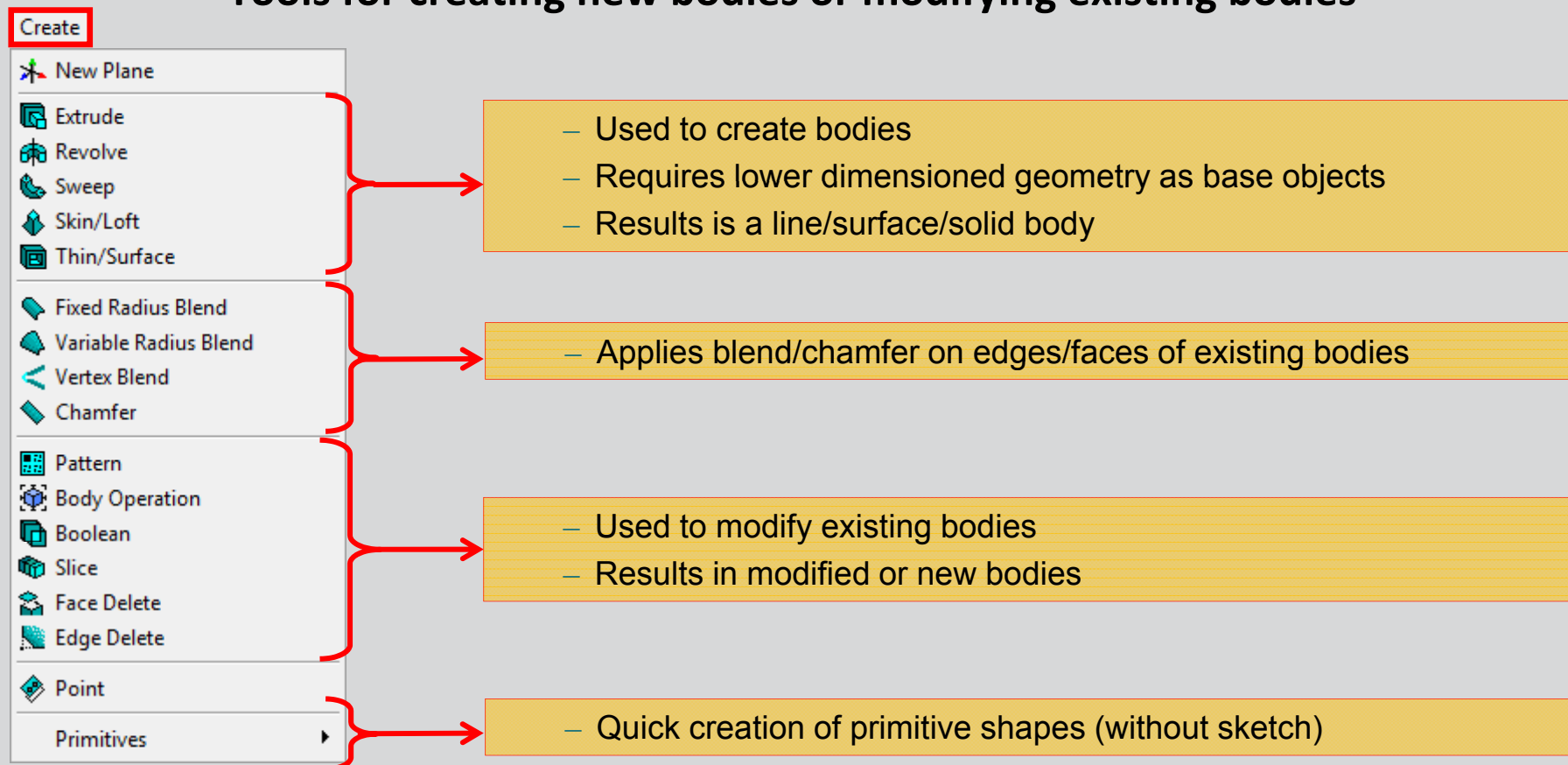


- After a body has been created as frozen - Use 'Unfreeze' operation (available under Tools Menu)
- Slicing an active body will convert it to frozen
- Can be merged manually using Boolean operations or by converting to 'Active' state using 'Unfreeze'

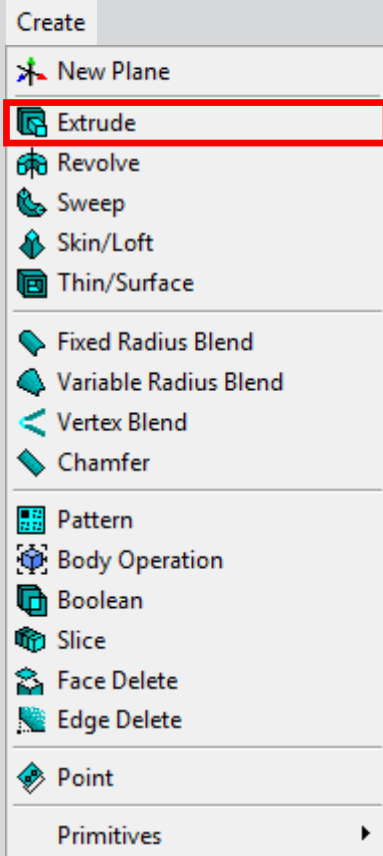
- Displayed as Transparent
- Cyan icon under Tree Outline



Tools for creating new bodies or modifying existing bodies



Extrude Feature



Extrude Details

- **Geometry**

- Can be a valid Sketch, selected face or Named Selection

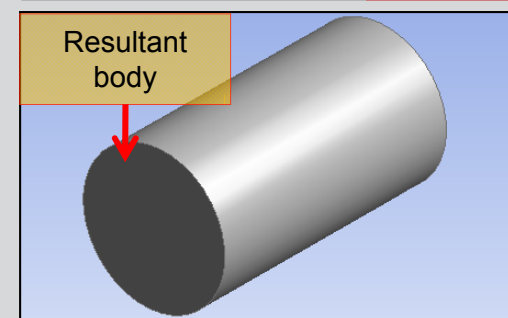
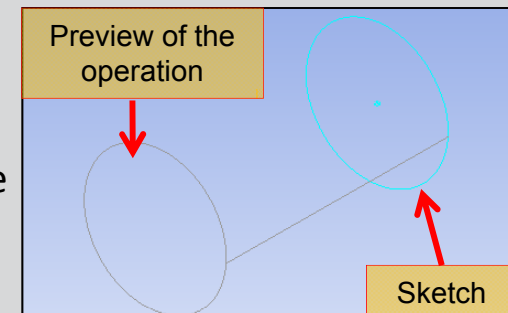
- **Operation**

- Allows Add Material, Cut Material, Imprint Faces, Slice Material, Add frozen
- Discussed in detail in Next 3 Slides

- **Direction Vector**

- Automatically sets direction as normal when base object is set to sketch otherwise needs to be specified
- Plane axes, geometric edges and faces can be used to specify direction

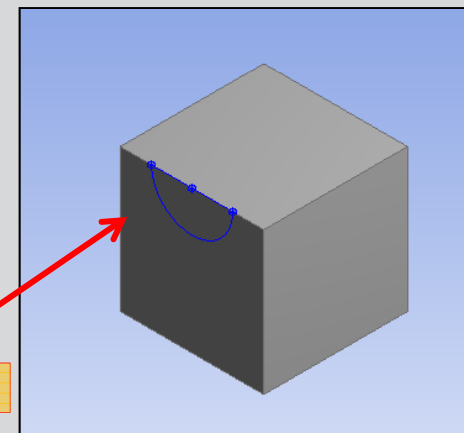
Details View	
Details of Extrude1	
Extrude	Extrude1
Geometry	Sketch1
Operation	Add Material
Direction Vector	None (Normal)
Direction	Normal
Extent Type	Fixed
☐ FD1, Depth (>0)	1 m
As Thin/Surface?	No
Merge Topology?	Yes



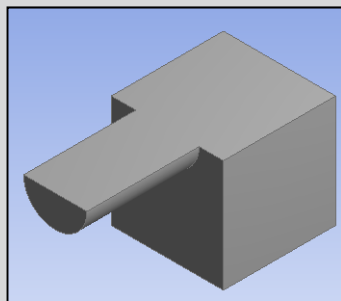
Extrude Feature (cont ...)

- **Add Material**
 - Creates new body and unites with existing active bodies in contact or overlap
- **Add Frozen:**
 - An independent body is created, with separate faces and edges
- **Cut Material:**
 - Create holes or modify existing bodies by removing material
- **Imprint Faces:**
 - Imprints the sketch on the faces of existing body
- **Slice Material:**
 - Slice out resultant body from the existing Frozen body

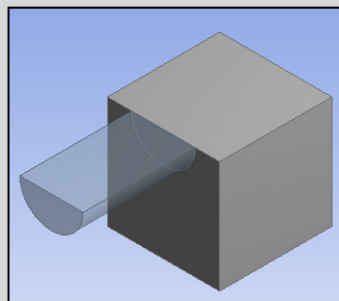
Sketch



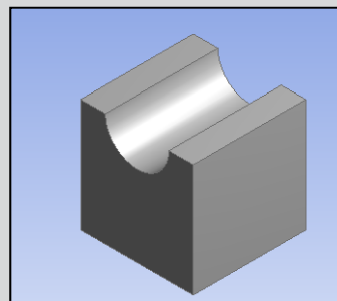
Add Material



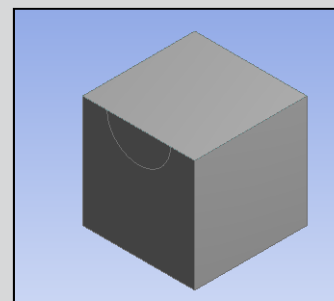
Add Frozen



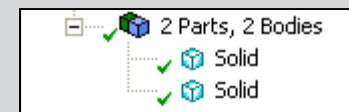
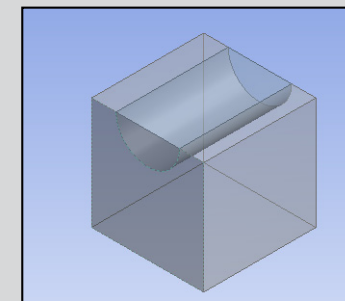
Cut Material



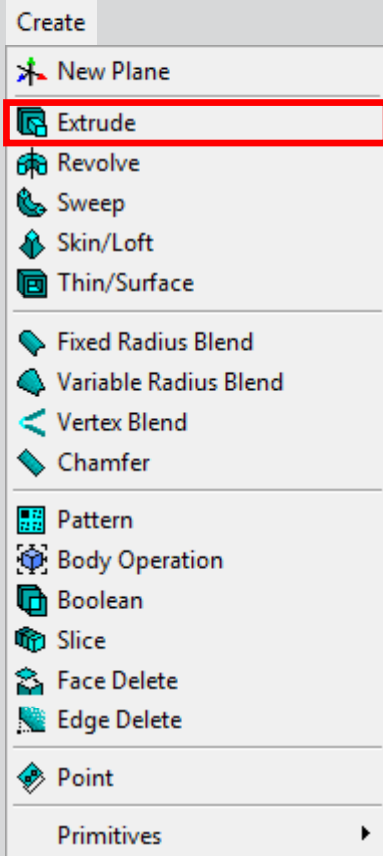
Imprint faces



Slice Material



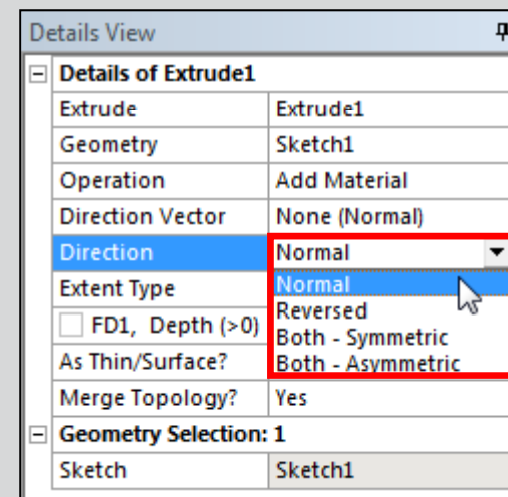
Extrude Feature (cont ...)



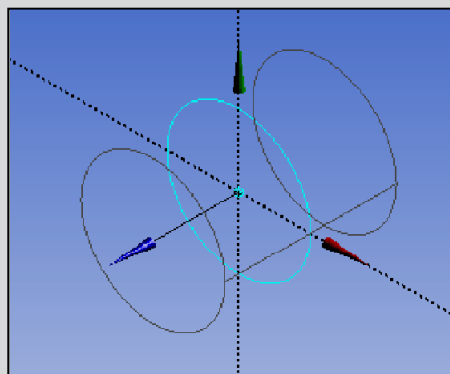
Extrude Details

• Direction

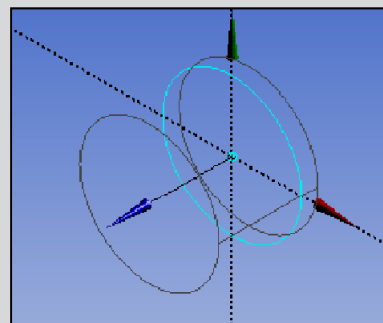
- Normal – Extrude Base object Normal to plane
- Reversed – Extrude Base object in reverse direction of Normal
- Both-Symmetric – Extrude symmetrically in both directions
- Both-Asymmetric – Extrude asymmetrically in both directions



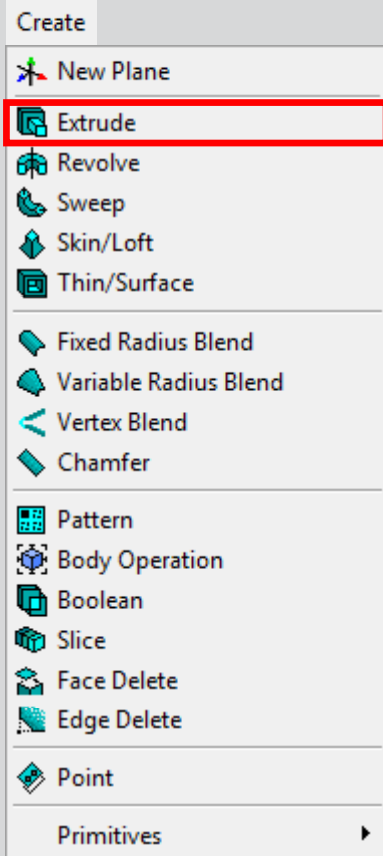
Direction	Both - Symmetric
Extent Type	Fixed
☐ FD1, Depth (>0)	5 m



Direction	Both - Asymmetric
Extent Type	Fixed
☐ FD1, Depth (>0)	5 m
Extent Type 2	Fixed
☐ FD4, Depth 2 (>0)	1 m



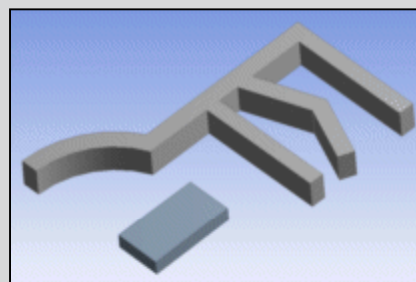
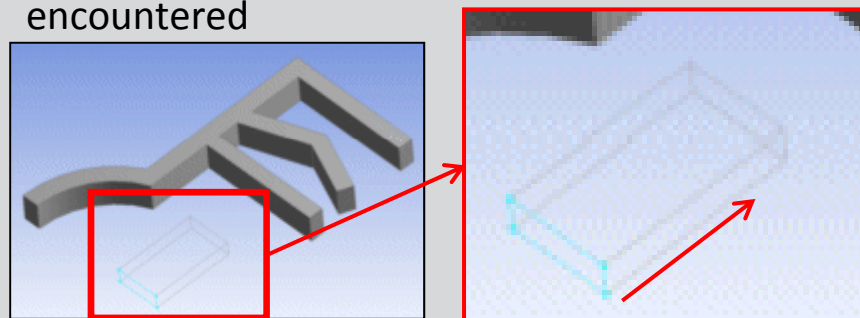
Extrude Feature (cont ...)



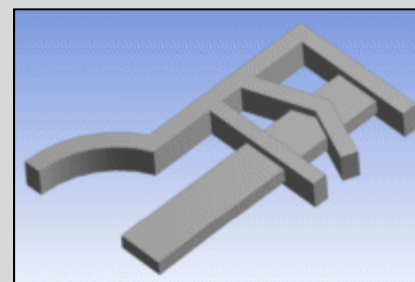
Extrude Details

• Extent Type :

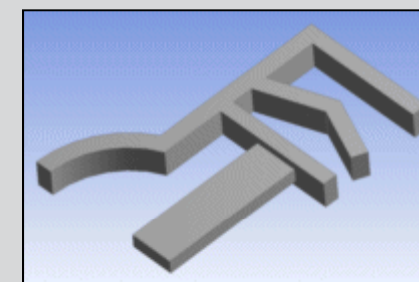
- Fixed – Extrude profile by specified distance
- Through All – Extrude profile through entire model
- To Next – Extrude profile up to first surface encountered



Fixed



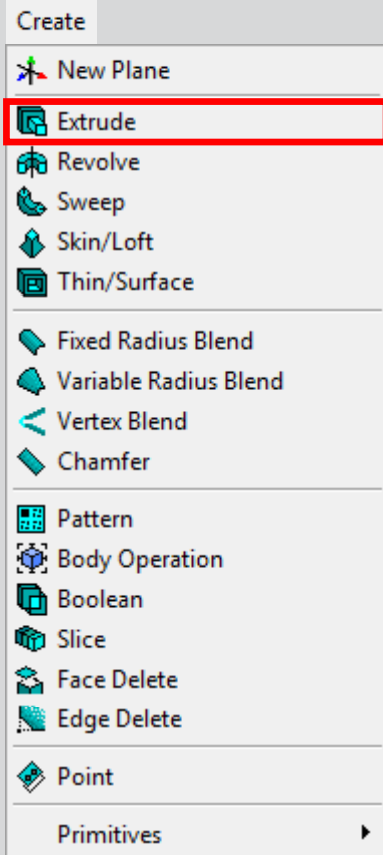
Through All



To Next

Details View	
Details of Extrude1	
Extrude	Extrude1
Geometry	Sketch1
Operation	Add Material
Direction Vector	None (Normal)
Direction	Normal
Extent Type	Fixed
☐ FD1, Depth (>0)	Through All
As Thin/Surface?	To Next
Merge Topology?	To Faces
	To Surface

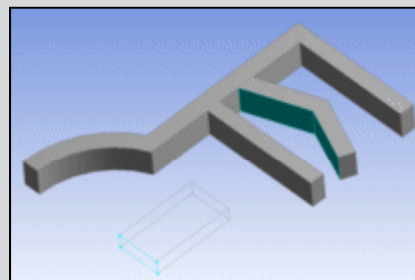
Extrude Feature (cont ...)



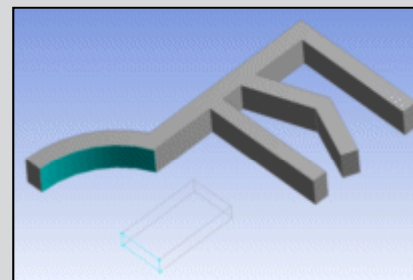
Extrude Details

• Extent Type cont ...

- To Faces – Extrude profile up to one or more selected faces
- To Surfaces – Extrude profile up to surface



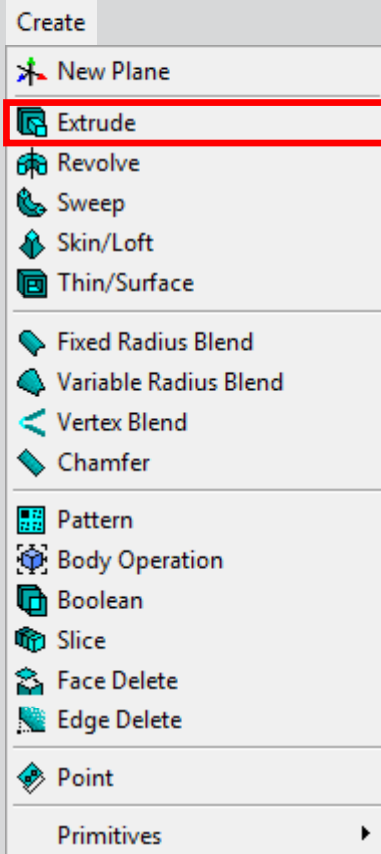
To Faces



To Surfaces

Details View	
Details of Extrude1	
Extrude	Extrude1
Geometry	Sketch1
Operation	Add Material
Direction Vector	None (Normal)
Direction	Normal
Extent Type	Fixed
☐ FD1, Depth (>0)	Through All
As Thin/Surface?	To Next
	To Faces
Merge Topology?	To Surface

Extrude Feature (cont ...)



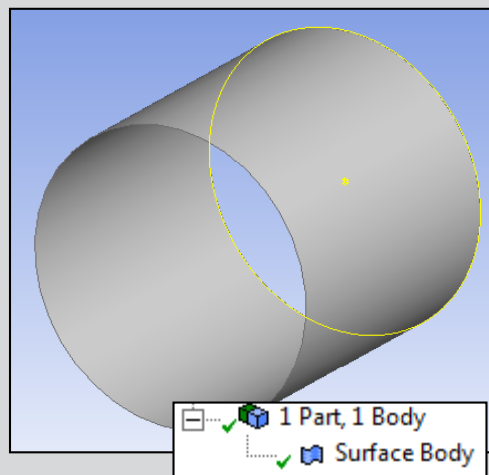
Extrude Details

• Thin/Surface

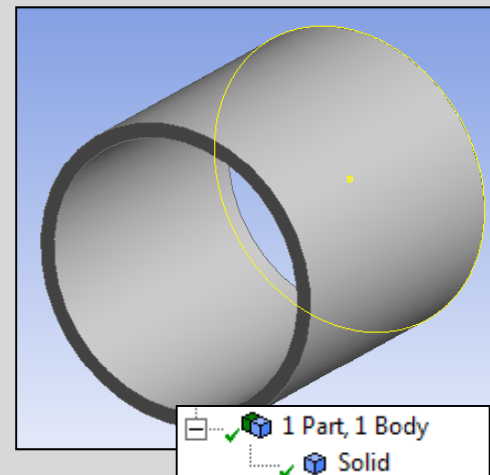
- If enabled, a thin surface is extruded out of the base object with a default thickness of zero.
- Suitable for creating shell bodies
- Options to specify thickness

Details View	
Details of Extrude1	
Extrude	Extrude1
Geometry	Sketch1
Operation	Add Material
Direction Vector	None (Normal)
Direction	Normal
Extent Type	Fixed
☐ FD1, Depth (>0)	1 m
☒ As Thin/Surface?	No
Merge Topology?	Yes

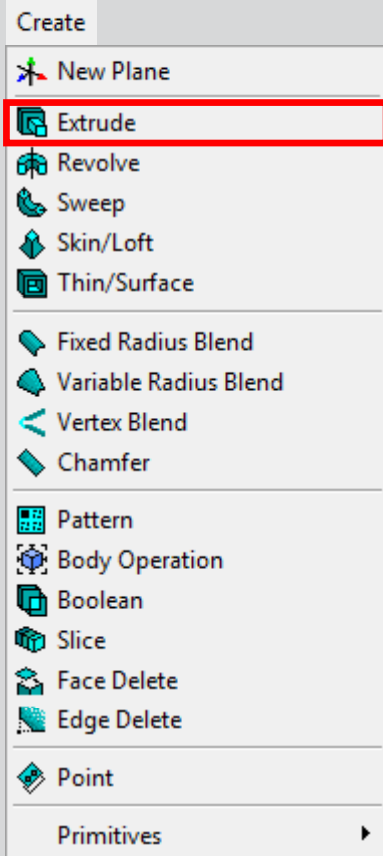
As Thin/Surface?	Yes
☐ FD2, Inward Thickness (>=0)	0.05 m
☐ FD3, Outward Thickness (>=0)	0 m



As Thin/Surface?	Yes
☐ FD2, Inward Thickness (>=0)	0.05 m
☐ FD3, Outward Thickness (>=0)	0 m



Extrude Feature (cont ...)

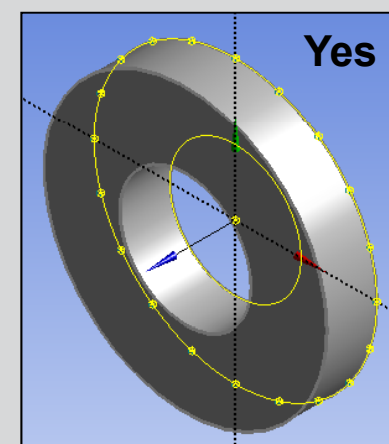
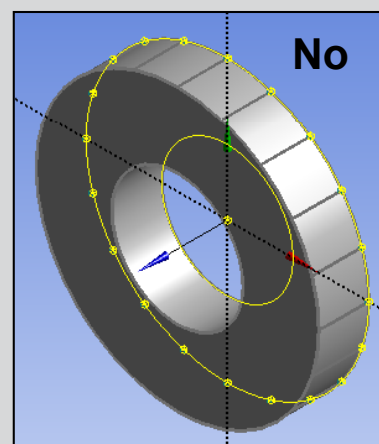
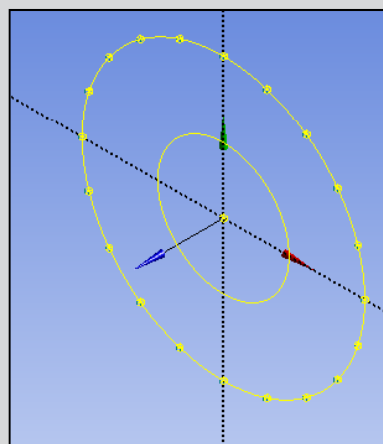


Extrude Details

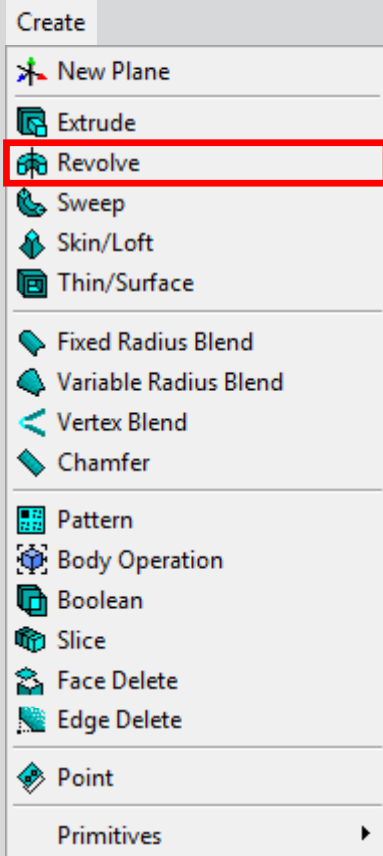
• Merge Topology

- Yes: Optimizes the topology of feature bodies
- No: Leaves the topology of feature bodies unaltered
- Tips from a meshing Perspective:
 - If the small faces are of interest during meshing, then set Merge Topology to 'NO'
 - If the small faces are unnecessarily forcing for fine mesh, then set Merge Topology to 'YES'

Details View	
Details of Extrude1	
Extrude	Extrude1
Geometry	Sketch1
Operation	Add Material
Direction Vector	None (Normal)
Direction	Normal
Extent Type	Fixed
☐ FD1, Depth (>0)	1 m
As Thin/Surface?	No
Merge Topology?	Yes



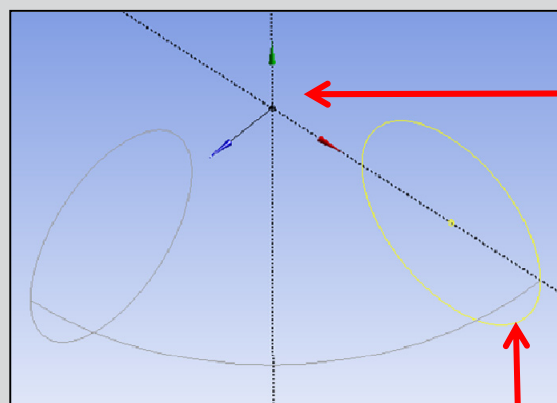
Revolve Feature



Revolve Details

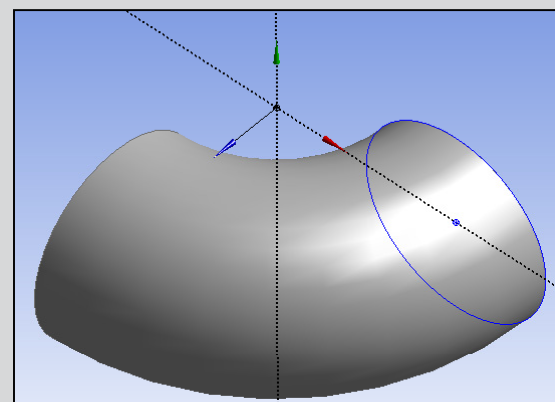
- **Geometry** is rotated to create a body of revolution
- **Axis**
 - Specify Axis of Rotation for selected Base object (plane axis or edge)
- **Angle**
 - Specify Rotation Angle

Details View	
Details of Revolve1	
Revolve	Revolve1
Geometry	Sketch1
Axis	Selected
Operation	Add Material
Direction	Normal
☐ FD1, Angle (>0)	90 °
As Thin/Surface?	No
Merge Topology?	Yes

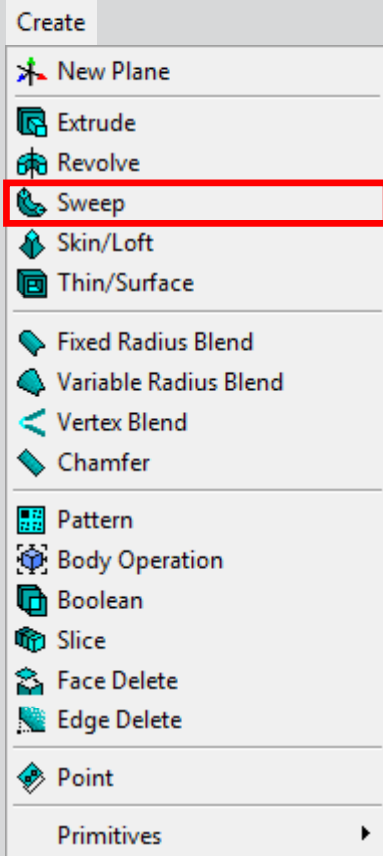


Axis

Base object



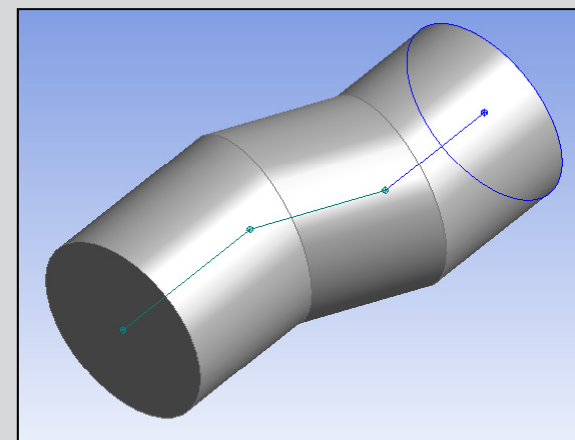
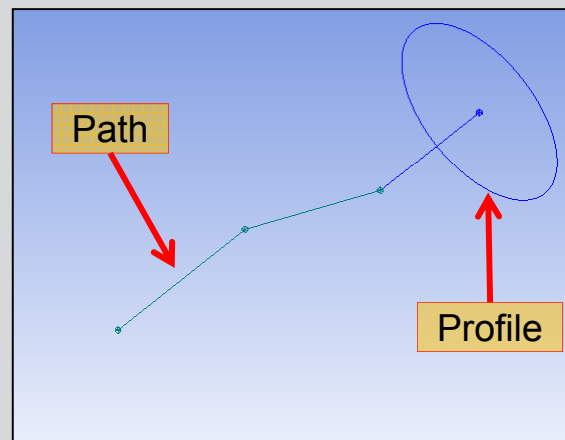
Sweep Feature



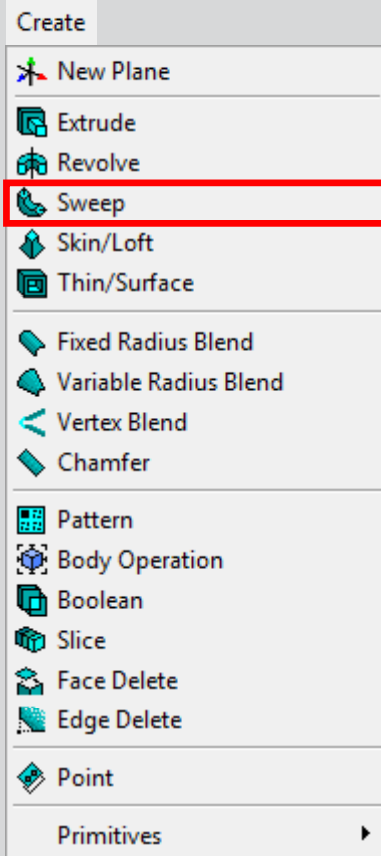
Sweep Details

- Selected profile is swept along selected path to create bodies
- **Profile :**
 - Specify the object to be swept
- **Path :**
 - Specify the path for sweep

Details View	
Details of Sweep1	
Sweep	Sweep1
Profile	Sketch1
Path	Sketch2
Operation	Add Material
Alignment	Path Tangent
☐ FD4, Scale (>0)	1
Twist Specification	No Twist
As Thin/Surface?	No
Merge Topology?	No

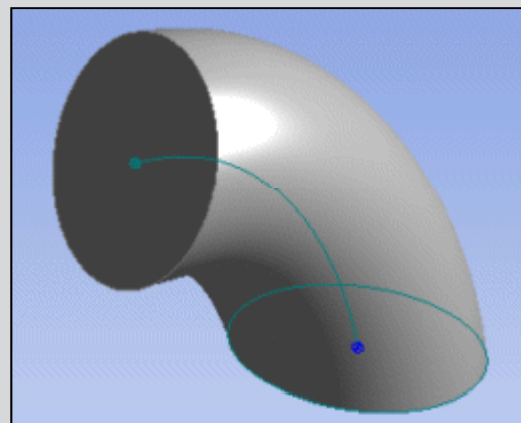
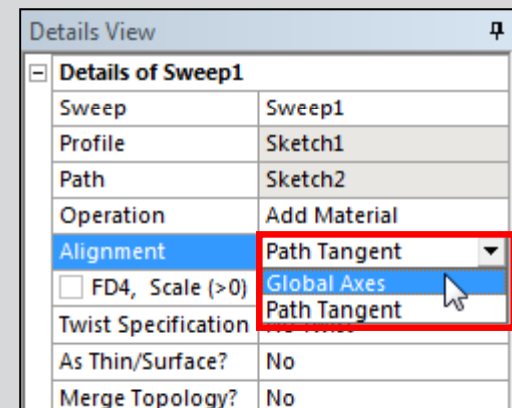


Sweep Feature (cont...)

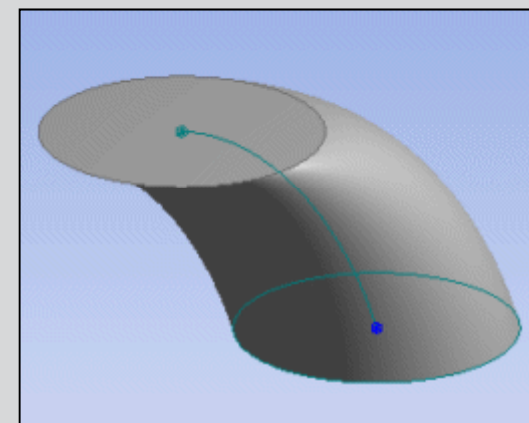


Sweep Details

- **Alignment : Specify orientation for profile with respect to swept path**
 - Path Tangent : Reorients the profile to keep its orientation consistent with respect to the path
 - Global Axes : Profile orientation remains fixed irrespective of shape of the Path

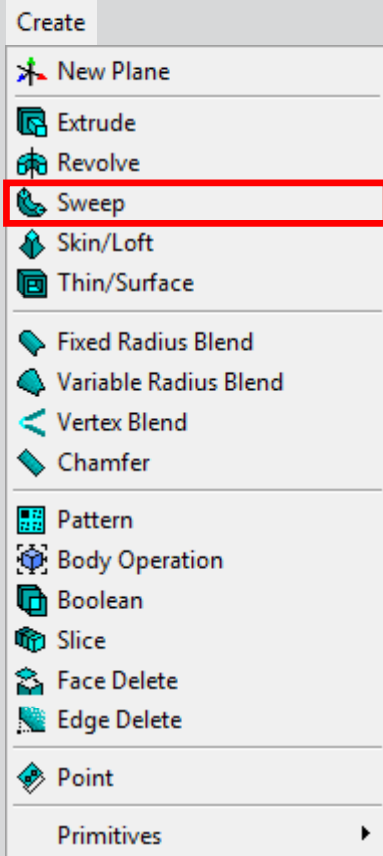


Path Tangent



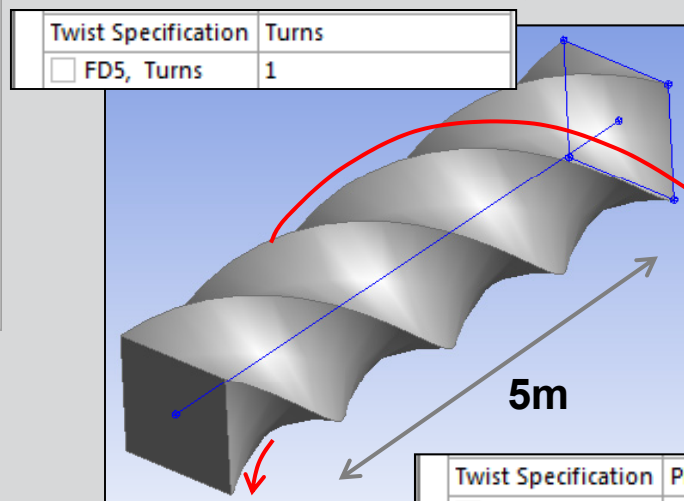
Global Axes

Sweep Feature (cont...)

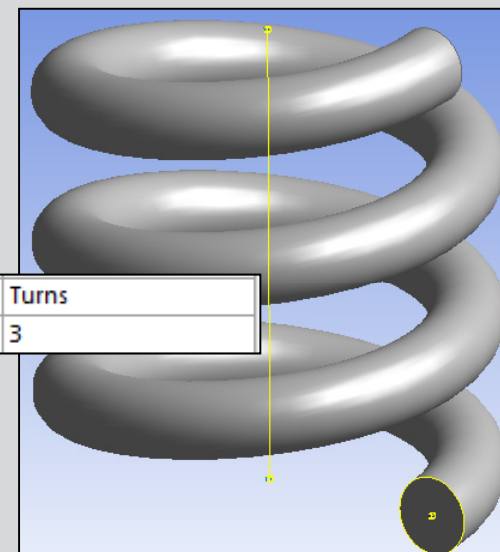
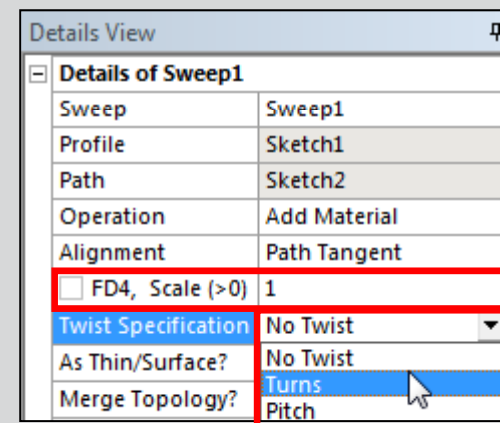


Sweep Details

- **Scale**
 - Taper or expand the Profile as it is swept along the path of Sweep
- **Twist Specification**
 - No Twist : Helical sweep is not created
 - Turns : Specify number of rotations about the path
 - Pitch : Specify pitch length for helical sweep



Twist Specification	Pitch
☐ FD6, Pitch	5 m

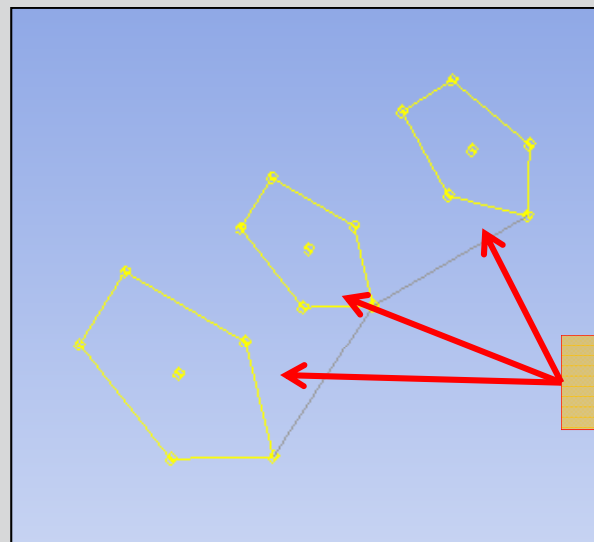


Create

- New Plane
- Extrude
- Revolve
- Sweep
- Skin/Loft**
- Thin/Surface
- Fixed Radius Blend
- Variable Radius Blend
- Vertex Blend
- Chamfer
- Pattern
- Body Operation
- Boolean
- Slice
- Face Delete
- Edge Delete
- Point
- Primitives

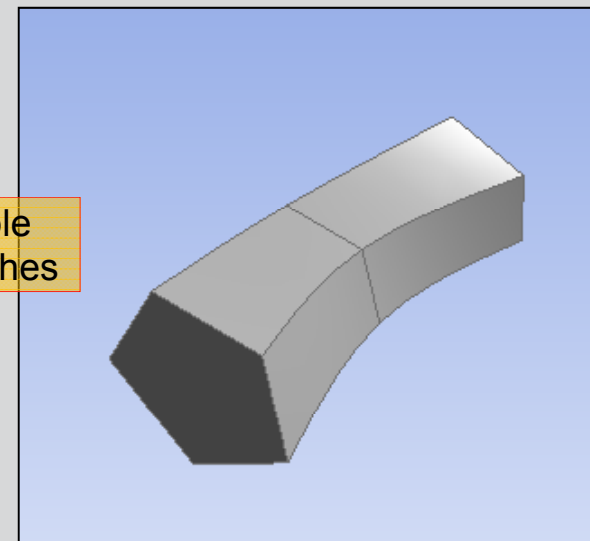
Skin/Loft Details

- Takes a series of profiles and creates a body fitting through them
 - All profiles must have the same number of edges

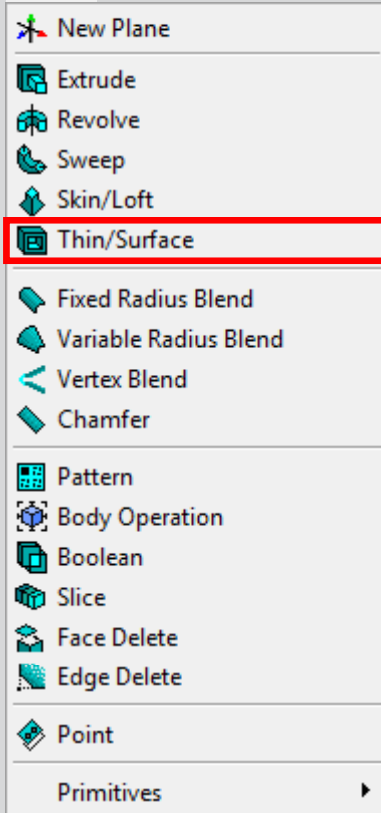


Multiple
sketches

Details View	
Details of Skin1	
Skin/Loft	Skin1
Profile Selection Method	Select All Profiles
Profiles	3 Sketches
Operation	Add Material
As Thin/Surface?	No
Merge Topology?	No
Profiles	
Profile 1	Sketch1
Profile 2	Sketch2
Profile 3	Sketch3

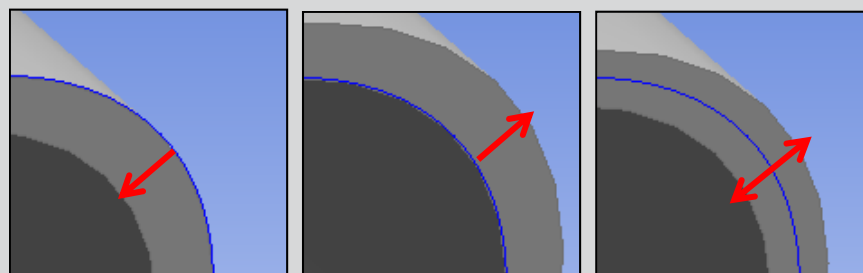


Create



Thin Surface Details

- **Selection Type Faces to Keep/Faces to Remove**
 - Selected faces will be kept while unselected faces will be removed creating a solid of specified thickness
 - Specification of zero thickness will create a surface body (can specify offset)
 - Bodies Only option creates thin solids/offset surface from surface bodies
 - Direction for generation of thickness can be inward, outward or both (midplane)

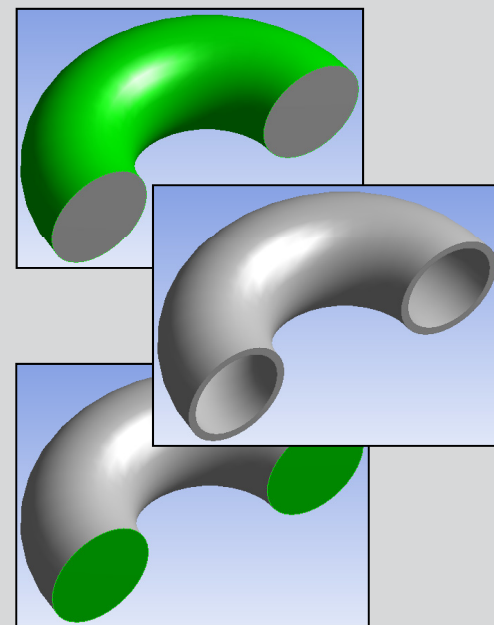


Inward

Outward

Midplane

Details View	
Details of Thin1	
Thin/Surface	Thin1
Selection Type	Faces to Keep
Geometry	1 Face
Direction	Inward
☐ FD1, Thickness (≥ 0)	0.5 m



Selection Type	Faces to Remove
Geometry	2 Faces

Create

New Plane
Extrude
Revolve
Sweep
Skin/Loft
Thin/Surface

Fixed Radius Blend
Variable Radius Blend
Vertex Blend
Chamfer

Pattern
Body Operation
Boolean
Slice
Face Delete
Edge Delete

Point

Primitives

Fixed Radius Blend Details

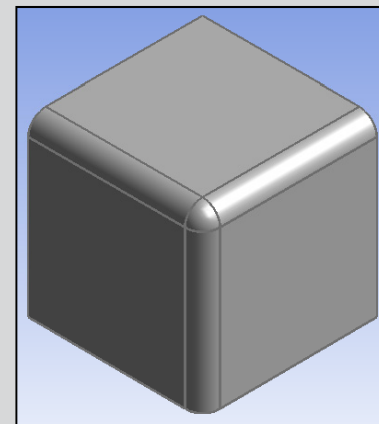
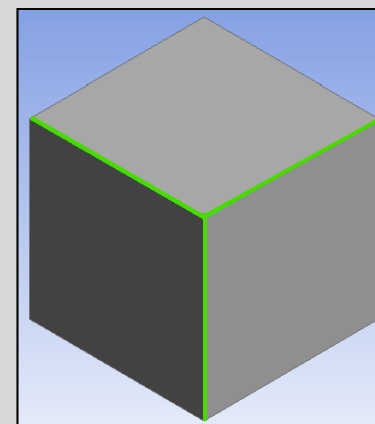
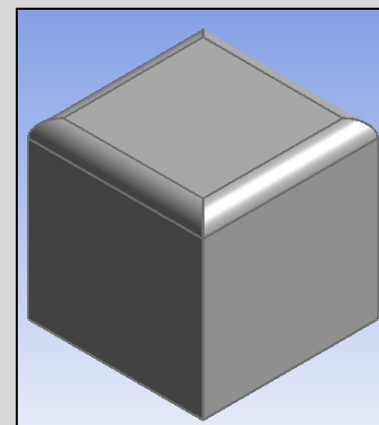
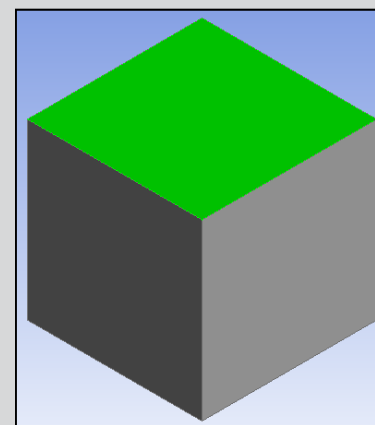
- Creates a blend of specified radius on face/edges

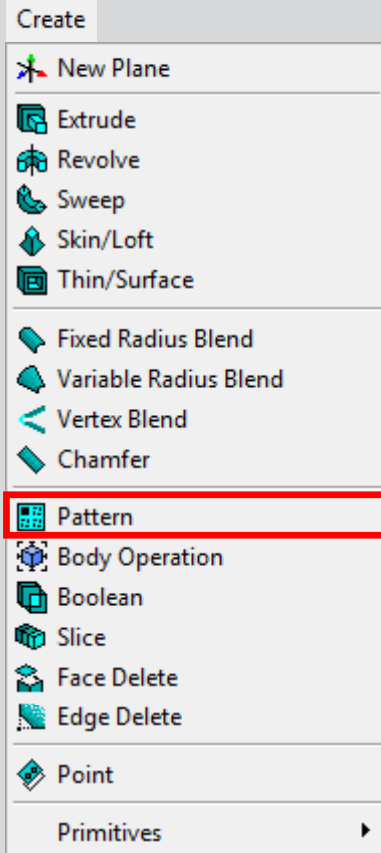
Details View	
Details of FBlend1	
Fixed-Radius Blend	FBlend1
☐ FD1, Radius (>0)	0.1 m
Geometry	1 Face

Face Selection

Details View	
Details of FBlend1	
Fixed-Radius Blend	FBlend1
☐ FD1, Radius (>0)	0.1 m
Geometry	3 Edges

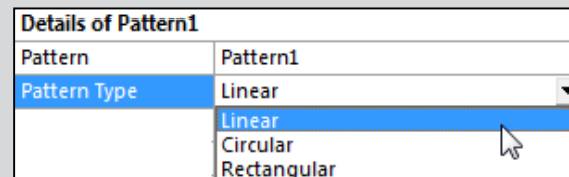
Edge Selection



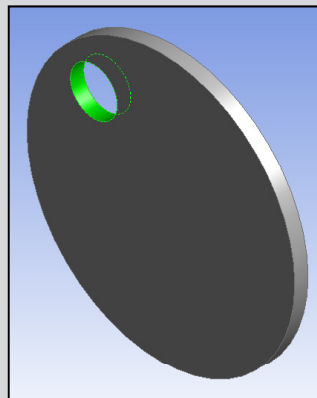


Function

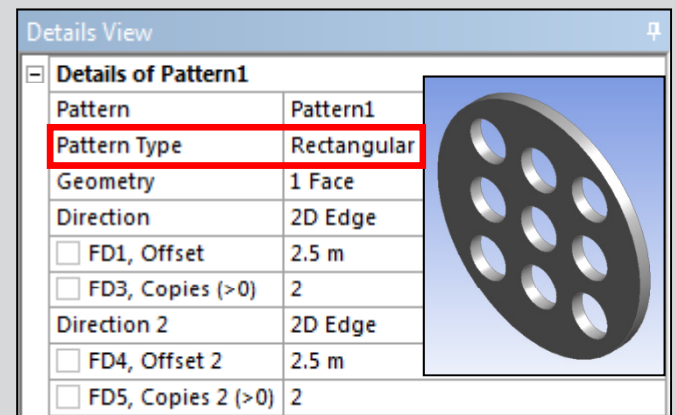
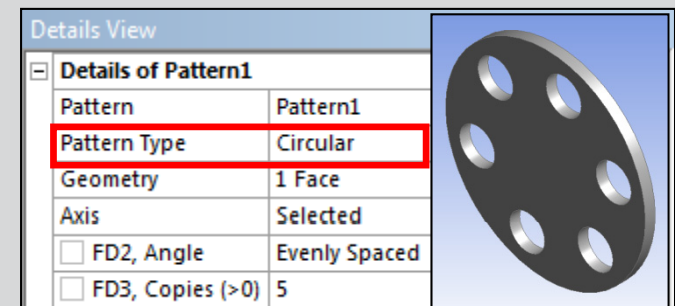
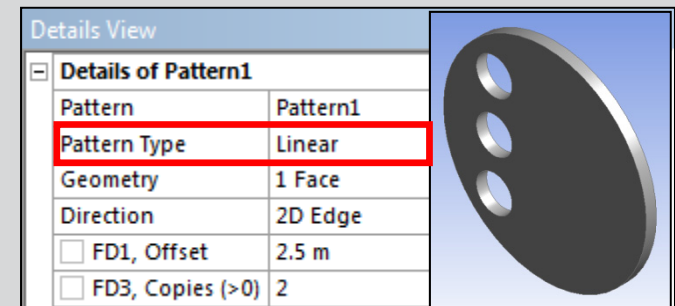
- Creates copies of selected features
- Pattern Type: 3 options
- Linear, Circular or Rectangular



- Select Geometry (features or bodies)



- Specify Direction, Axis, Offset, Copies



Create

New Plane

Extrude

Revolve

Sweep

Skin/Loft

Thin/Surface

Fixed Radius Blend

Variable Radius Blend

Vertex Blend

Chamfer

Pattern

Body Operation

Boolean

Slice

Face Delete

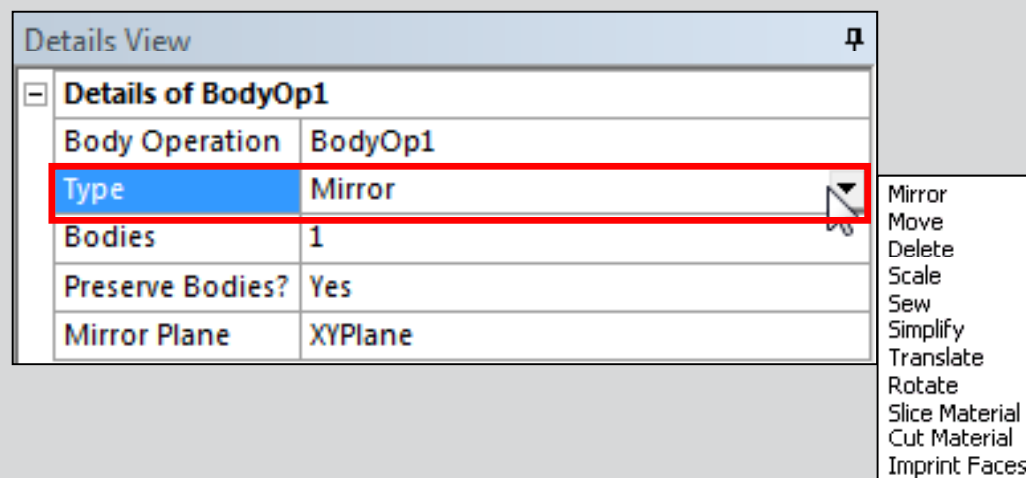
Edge Delete

Point

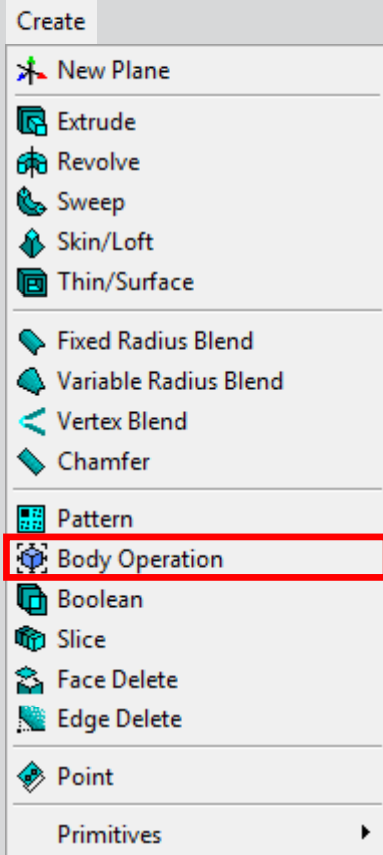
Primitives

Body Manipulation Tools

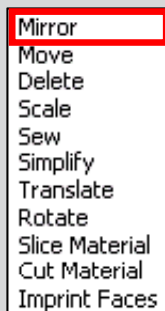
- Contains eleven general body editing tools
- Options vary depending on whether the bodies are Active or Frozen
- Cut and imprint are not available when all bodies are Frozen
- Select Body Operation, set Type



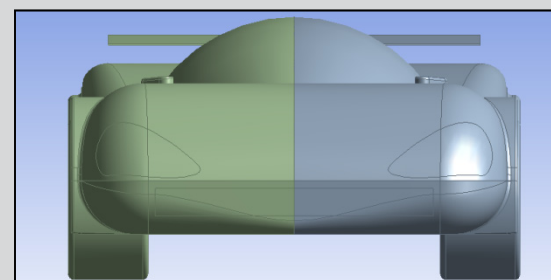
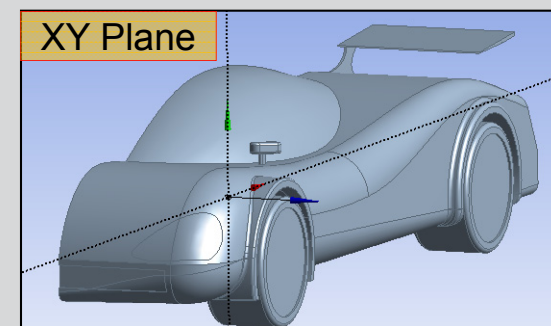
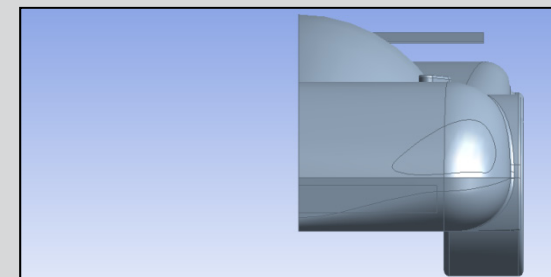
Body Operation: Mirror



- Creates a mirror image body reflected in selected plane
- Select body or bodies
- Set Mirror Plane
- Original body deletion controlled by 'Preserve Bodies' option
- Example – Mirroring car body in the XY Plane preserving the original

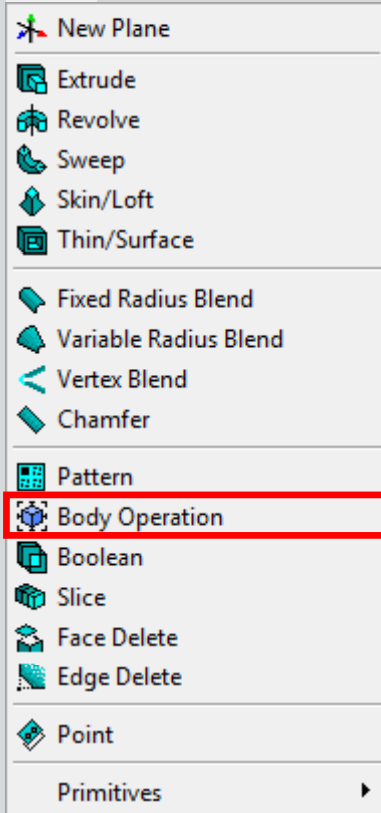


Details View	
Details of BodyOp1	
Body Operation	BodyOp1
Type	Mirror
Bodies	1
Preserve Bodies?	Yes
Mirror Plane	XYPlane

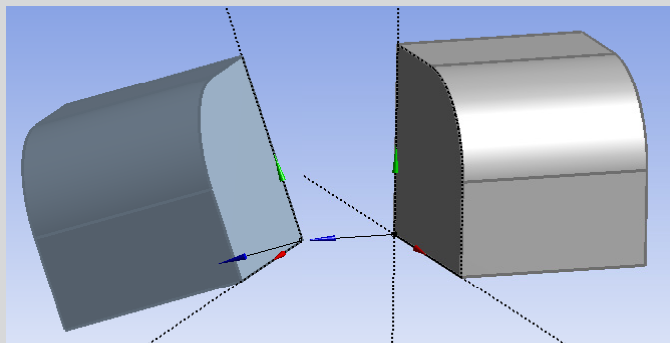
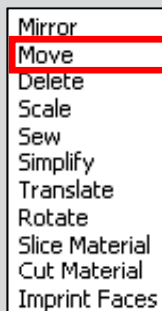


Body Operation: Move

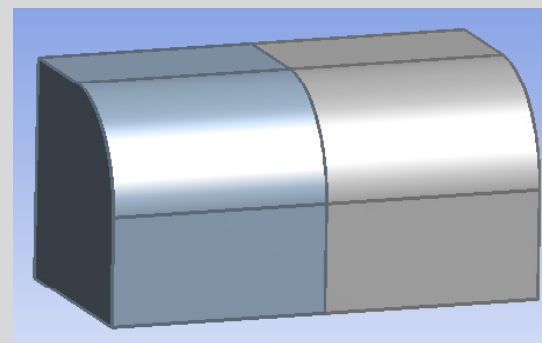
Create



- Moves and aligns selected bodies
- Select body or bodies
- Option to preserve bodies
- Set Source and Destination Planes
- Example – Aligning two bodies using corresponding source and destination planes on each body respectively



Details View	
Details of BodyOp4	
Body Operation	BodyOp4
Type	Move
Bodies	1
Preserve Bodies?	No
Source Plane	Plane6
Destination Plane	Plane4



Body Operation: Scale

Create

New Plane

Extrude

Revolve

Sweep

Skin/Loft

Thin/Surface

Fixed Radius Blend

Variable Radius Blend

Vertex Blend

Chamfer

Pattern

Body Operation

Boolean

Slice

Face Delete

Edge Delete

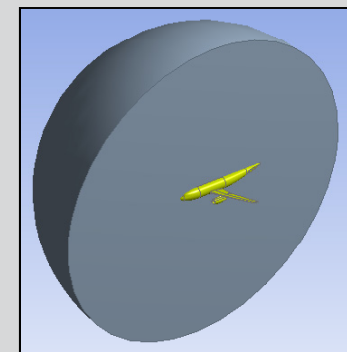
Point

Primitives

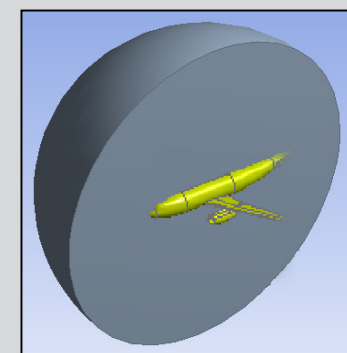
- Scale selected body or bodies
- Select body or bodies
- Set Scaling origin
 - World Origin : Global coordinate origin
 - Body Centroids: Centroid of the bodies selected
 - Point: Selected Point

Mirror
Move
Delete
Scale
Sew
Simplify
Translate
Rotate
Slice Material
Cut Material
Imprint Faces

Details View	
Details of BodyOp3	
Body Operation	BodyOp3
Type	Scale
Bodies	1
Preserve Bodies?	No
Scaling Origin	World Origin
☐ FD1, Scaling Factor (>0)	World Origin Body Centroids Point

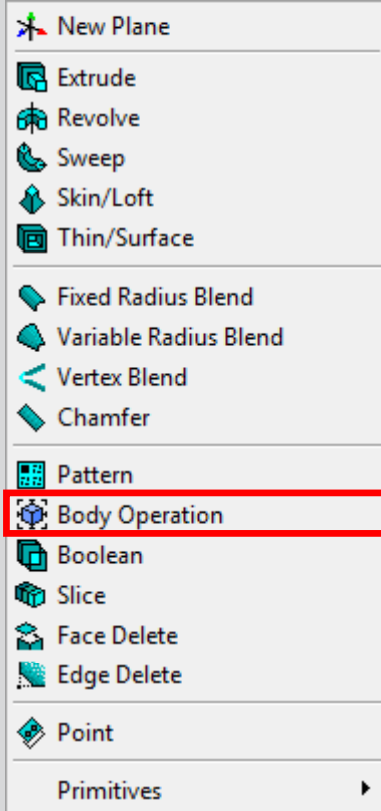


Scale up : 2x



Body Operation: Translate, Rotate

Create



- **Translate the selected body in a specified direction**
 - Direction can be specified as coordinates or as a straight edge and distance
 - Rotate selected bodies about a specified axis by a specified angle
 - Axis can be specified as components or as a straight edge

Details View	
Details of BodyOp1	
Body Operation	BodyOp1
Type	Translate
Bodies	1
Preserve Bodies?	No
Direction Definition	Selection
Direction Selection	2D Edge
☐ FD2, Distance	1 m

Direction Definition	Coordinates
☐ FD3, X Offset	0 m
☐ FD4, Y Offset	0 m
☐ FD5, Z Offset	0 m

Mirror
Move
Delete
Scale
Sew
Simplify
Translate
Rotate
Slice Material
Cut Material
Imprint Faces

Details View	
Details of BodyOp1	
Body Operation	BodyOp1
Type	Rotate
Bodies	1
Preserve Bodies?	No
Axis Definition	Selection
Axis Selection	2D Edge
☐ FD9, Angle	45 °

Axis Definition	Components
☐ FD6, X Component	0
☐ FD7, Y Component	0
☐ FD8, Z Component	0
☐ FD9, Angle	45 °

Body Operation: Slice Material

Create

New Plane

Extrude

Revolve

Sweep

Skin/Loft

Thin/Surface

Fixed Radius Blend

Variable Radius Blend

Vertex Blend

Chamfer

Pattern

Body Operation

Boolean

Slice

Face Delete

Edge Delete

Point

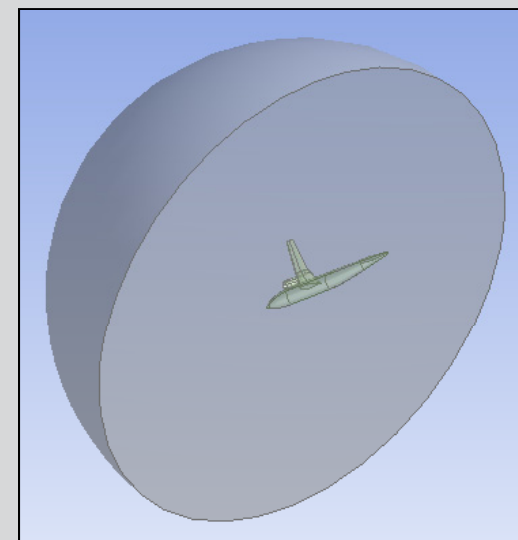
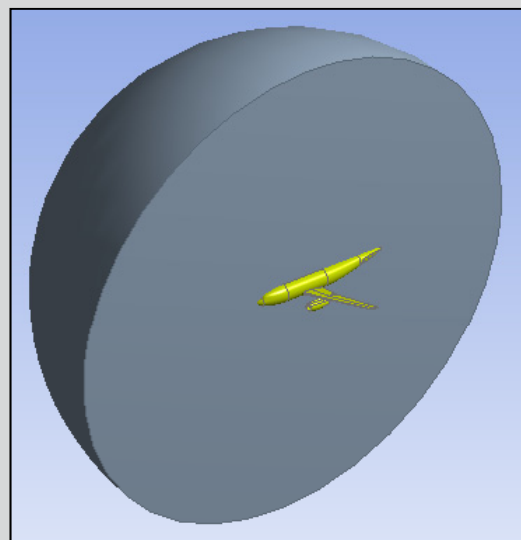
Primitives

- To Slice out one body from the another body

Details View

Details of BodyOp5

Body Operation	BodyOp5
Type	Slice Material
Bodies	1
Preserve Bodies?	No



Mirror
Move
Delete
Scale
Sew
Simplify
Translate
Rotate
Slice Material
Cut Material
Imprint Faces

Create

New Plane

Extrude

Revolve

Sweep

Skin/Loft

Thin/Surface

Fixed Radius Blend

Variable Radius Blend

Vertex Blend

Chamfer

Pattern

Body Operation

Boolean

Slice

Face Delete

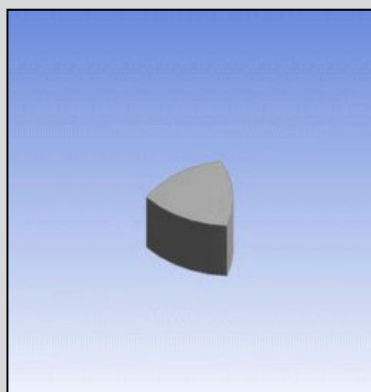
Edge Delete

Point

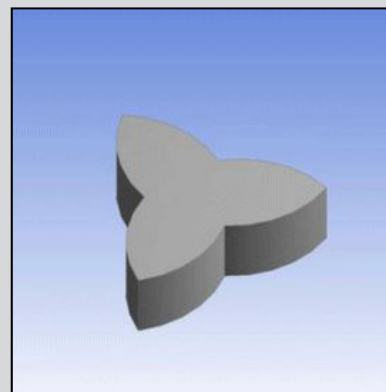
Primitives

Unite, Subtract and Intersect

- **Unite:** Merges two or more bodies
- **Subtract:** Target Body subtract Tool Body
- **Intersection:**
 - Union of All Intersections: Regions shared by two or more Tool Bodies
 - Intersection of All Bodies: Regions shared by all Tool Bodies

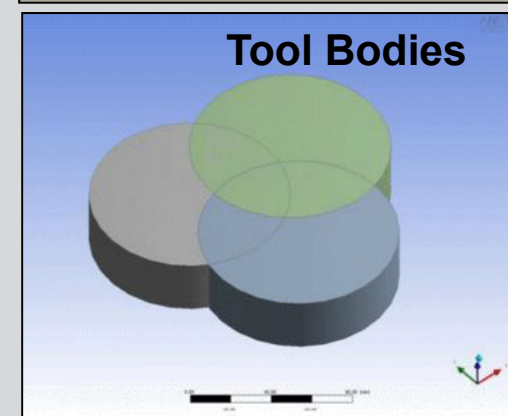


Intersect all bodies

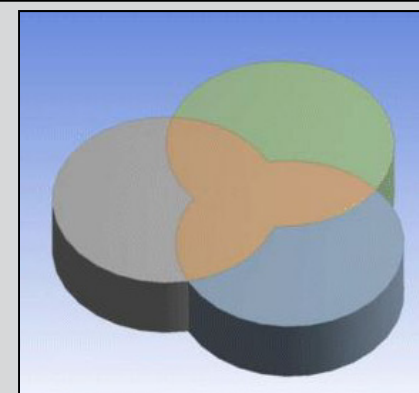


Union of all intersections

Details of Boolean1	
Boolean	Boolean1
Operation	Intersect
Tool Bodies	3 Bodies
Preserve Tool Bodies?	No
Intersect Result	Union of All Intersections



Tool Bodies



Union of all intersections preserving input bodies

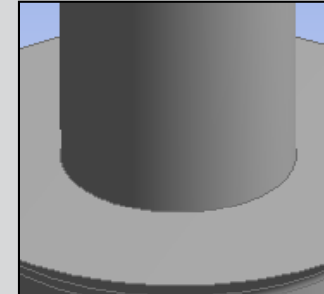
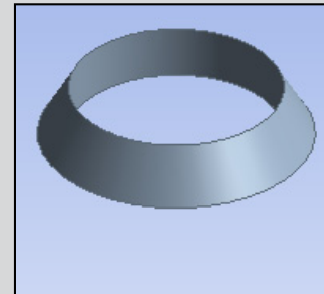
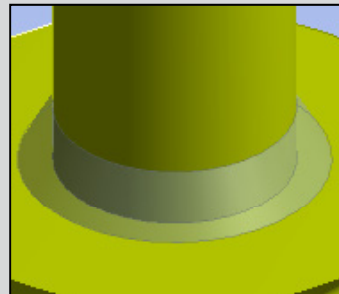
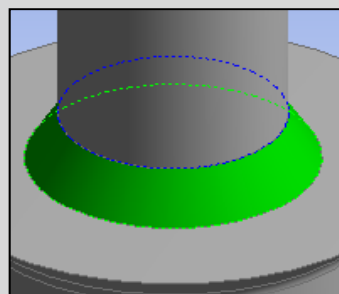
'Slice' Operation under 'Create'

Create

- New Plane
- Extrude
- Revolve
- Sweep
- Skin/Loft
- Thin/Surface
- Fixed Radius Blend
- Variable Radius Blend
- Vertex Blend
- Chamfer
- Pattern
- Body Operation
- Boolean
- Slice**
- Face Delete
- Edge Delete
- Point
- Primitives ▶

Slice Options

- **Slice By Plane**
 - Model is sliced by selected plane
- **Slice Off Faces**
 - Selected faces are sliced off and separate bodies are created from them
- **Slice Off Edges**
 - Selected edges are sliced off and separate bodies are created from them
- **Slice by Surface**
 - Selected surface is used as a slicing tool to slice
- **Slice by Edge Loop**
 - Selected edge loop is used as a slicing tool



Slice Off Faces

Primitive Geometries

Used to create 3D primitives quickly

Create

New Plane

Extrude

Revolve

Sweep

Skin/Loft

Thin/Surface

Fixed Radius Blend

Variable Radius Blend

Vertex Blend

Chamfer

Pattern

Body Operation

Boolean

Slice

Face Delete

Edge Delete

Point

Primitives

Sphere

Box

Parallelepiped

Cylinder

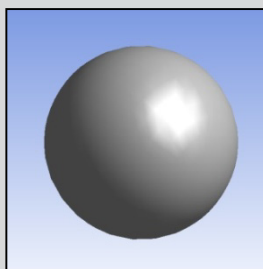
Cone

Prism

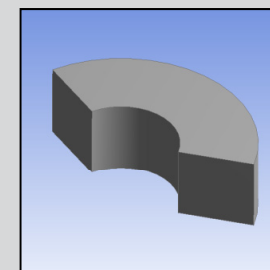
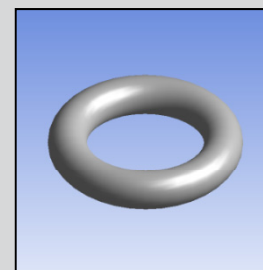
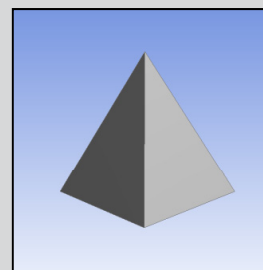
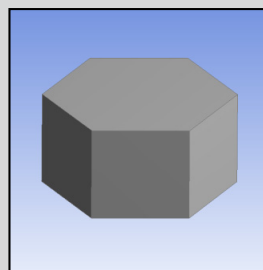
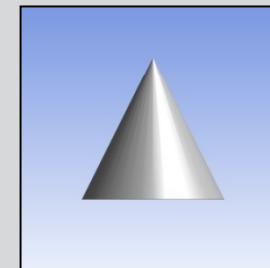
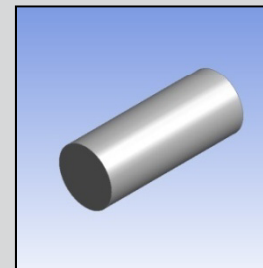
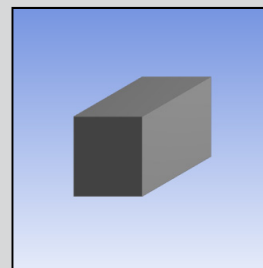
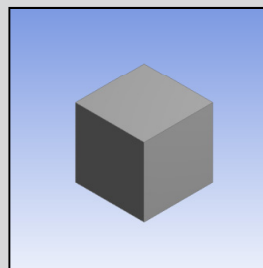
Pyramid

Torus

Bend

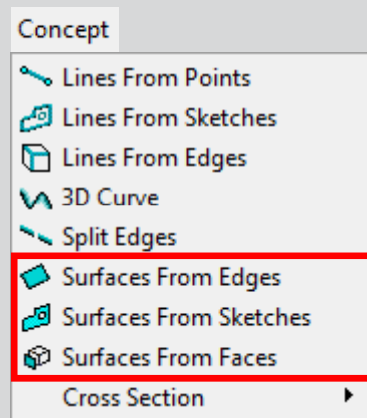


Details View	
Details of Sphere1	
Sphere	Sphere1
Base Plane	XYPlane
Operation	Add Material
Origin Definition	Coordinates
☐ FD3, Origin X Coordinate	0 m
☐ FD4, Origin Y Coordinate	0 m
☐ FD5, Origin Z Coordinate	0 m
☐ FD6, Radius (>0)	1 m
As Thin/Surface?	No



Concept Menu

- Tools for creating line & surface bodies
- 2d Surface Bodies can be created from existing edges, sketches or existing faces



- Further concept tools detailed in separate lecture



For 2D FLUENT models the geometry must be in the XY Plane

Surfaces From Sketches

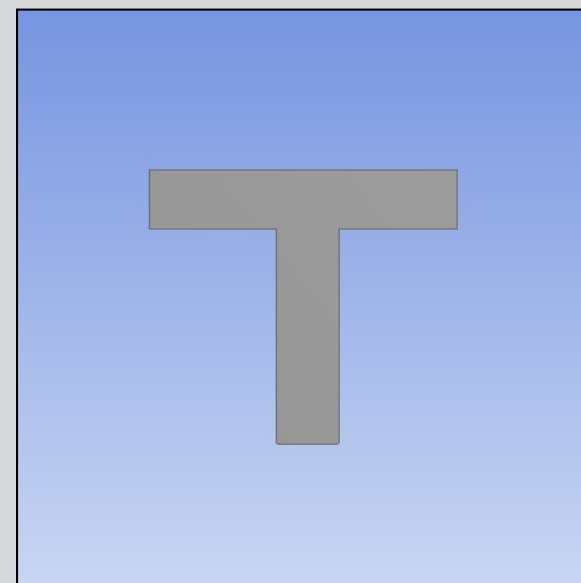
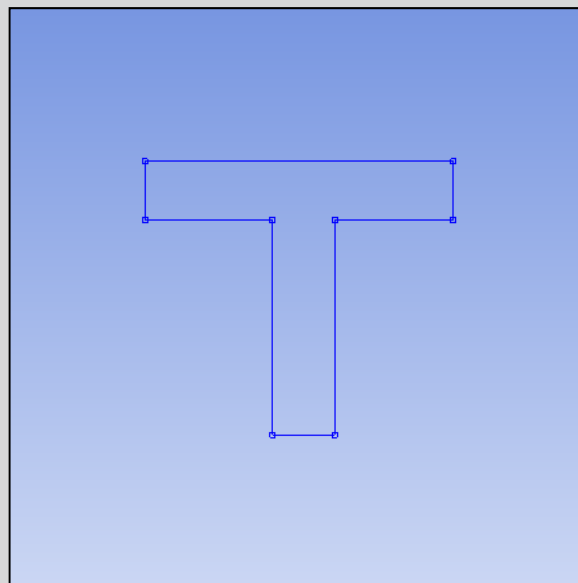
Concept

- Lines From Points
- Lines From Sketches
- Lines From Edges
- 3D Curve
- Split Edges
- Surfaces From Edges
- Surfaces From Sketches**
- Surfaces From Faces
- Cross Section

- Used to create Surface bodies from existing sketches
- Can be used for 2D analysis
- Can be used for creating capping faces required for Fill operation

Details View

Details of SurfaceSk1	
Surface From Sketches	SurfaceSk1
Base Objects	1 Sketch
Operation	Add Material
Orient With Plane Normal?	Yes
Thickness ($>=0$)	0 m



Surfaces From Faces

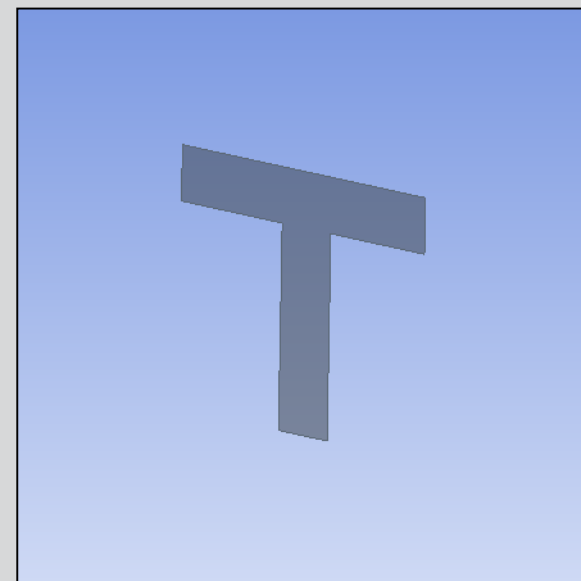
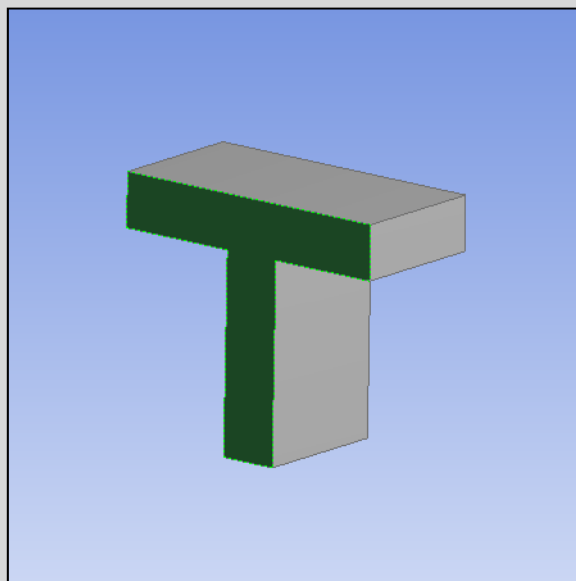
Concept

- Lines From Points
- Lines From Sketches
- Lines From Edges
- 3D Curve
- Split Edges
- Surfaces From Edges
- Surfaces From Sketches
- Surfaces From Faces**
- Cross Section

- Used to create Surface bodies from faces of existing models
- Can be used for 2D analysis

Details View

Details of SurfFromFaces1	
Surfaces From Faces	SurfFromFaces1
Faces	1 Face
Operation	Add Frozen
Holes Repair Method	No Healing

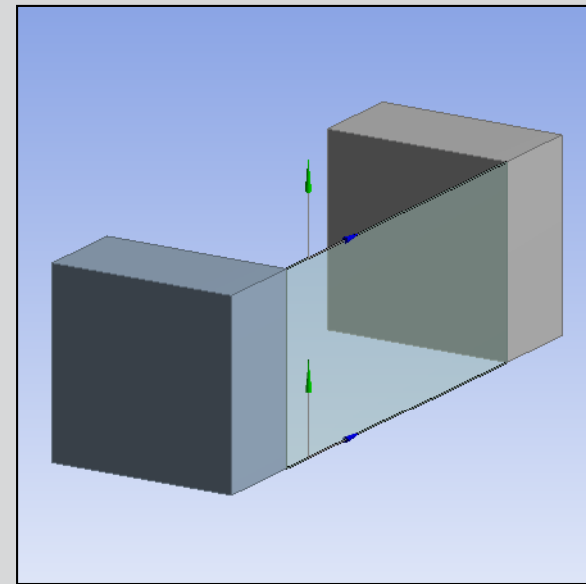
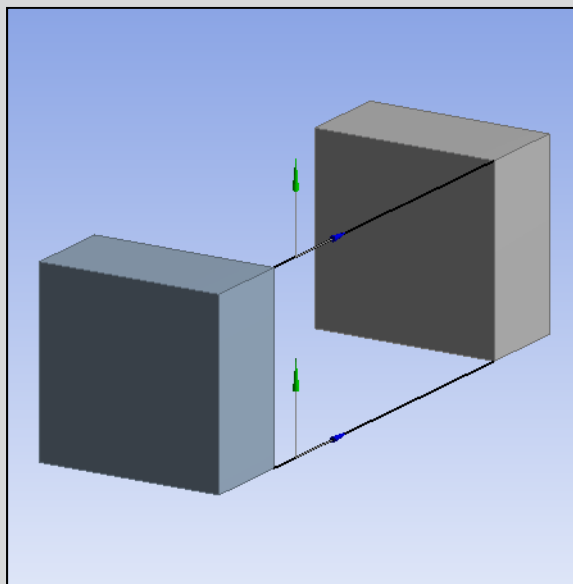


Surfaces From Edges

- Used to create Surface bodies from existing existing edges and Line bodies
- Can be used for 2D analysis
- Can be used for creating capping faces required for Fill operation

Concept

- Lines From Points
- Lines From Sketches
- Lines From Edges
- 3D Curve
- Split Edges
- Surfaces From Edges**
- Surfaces From Sketches
- Surfaces From Faces
- Cross Section



Surface body created using 2 Line bodies and 2 edges

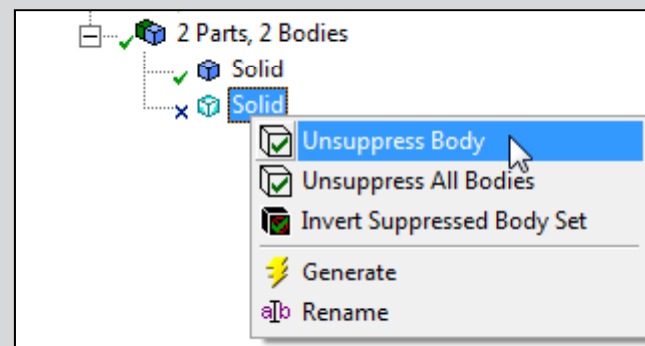
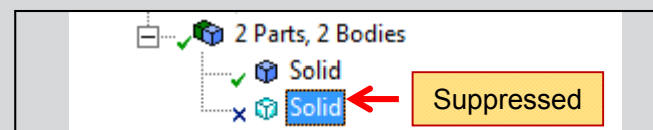
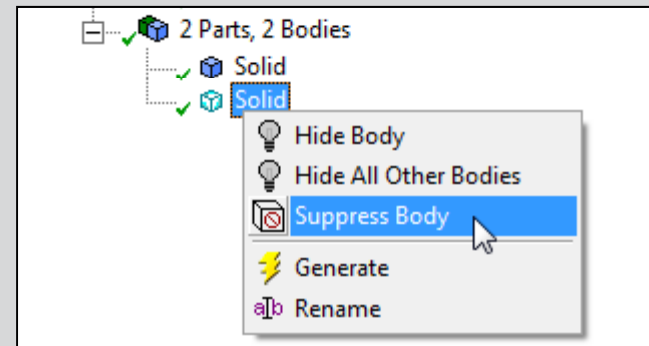
Suppressing Bodies

- **Behaviour of Suppressed Body**

- Will not be exported or transferred to other Workbench modules
- Not visible in Graphics view
- Identified by cross sign in Tree Outline

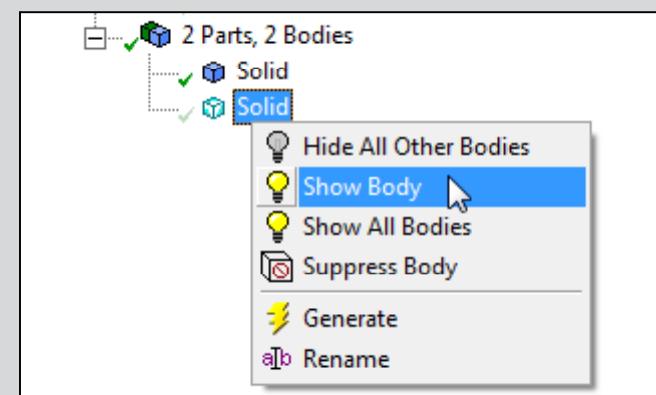
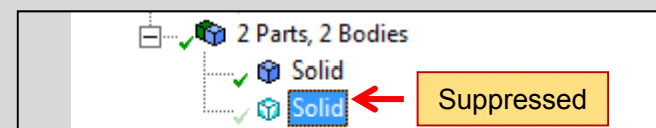
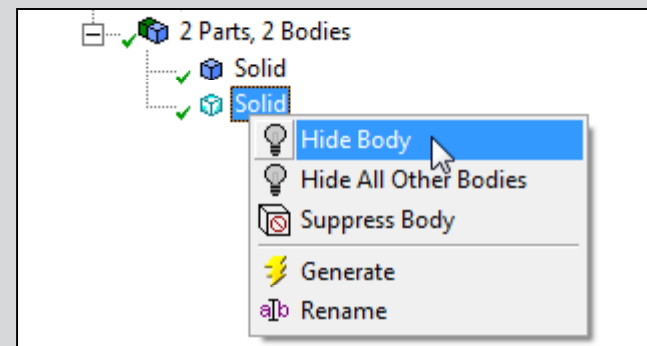
- **How to Suppress or Unsuppress?**

- Select the body to be suppressed, right click and select “Suppress Body”
- Select the body/bodies to be unsuppressed, right click and select “Unsuppress Body” option



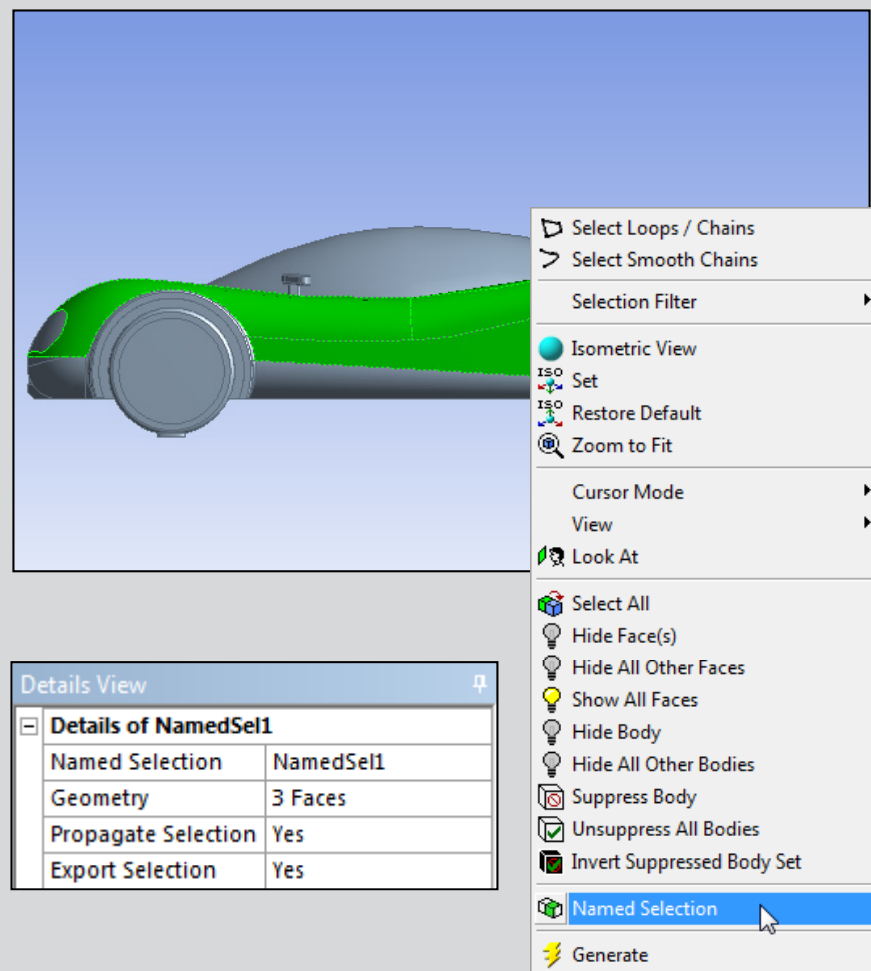
Hiding Geometry

- **Behaviour of Hidden Bodies**
 - In Tree Outline, Hidden body is identified by transparent “tick-mark” sign
 - Hidden bodies are transferred to other Workbench modules and can also be exported
- **Useful when working with complex models**
- **How to Hide or Unhide?**
 - Select the body or face to hide, right click and select “Hide Body”
 - Right click “Show Body” to restore view

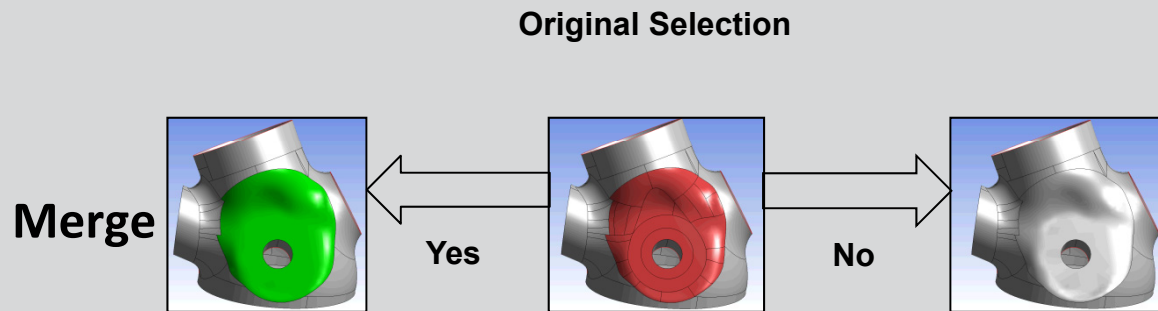


What are Named Selections?

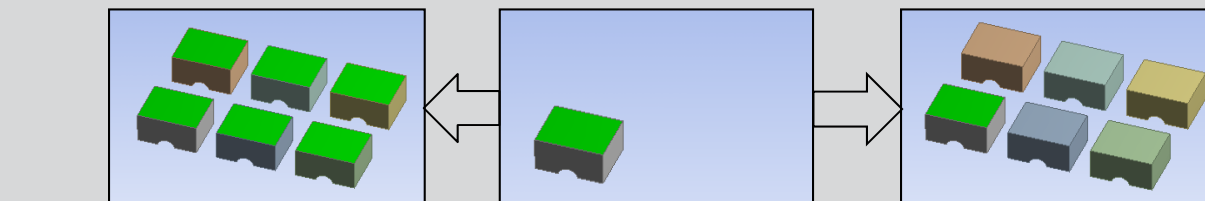
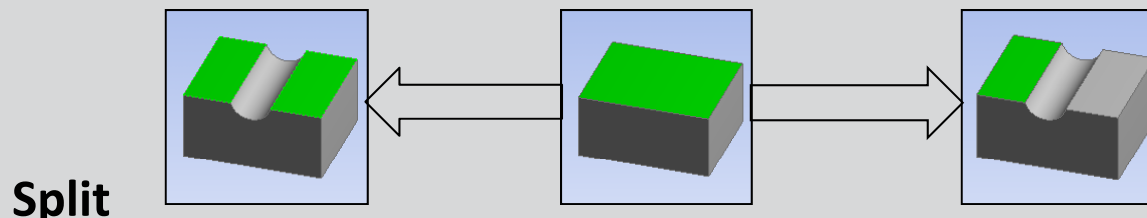
- Any number of entities grouped under one name
- Can be created using one of two ways
 - Select an entity or a group of entities, click RMB, select 'Named Selection' from the context menu or...
 - Select an entity or a group of entities, click on Tools menu and select 'Named Selection'
 - In the details view, enter a name for the 'Named Selection' or continue with the default name
 - Click 'Generate'
- Named selections can be assigned to points, lines, surfaces or solids



Named Selections : Propagation Property



Details View	
Details of NamedSel1	
Named Selection	NamedSel1
Geometry	23 Faces
Propagate Selection	Yes
	No
	Yes

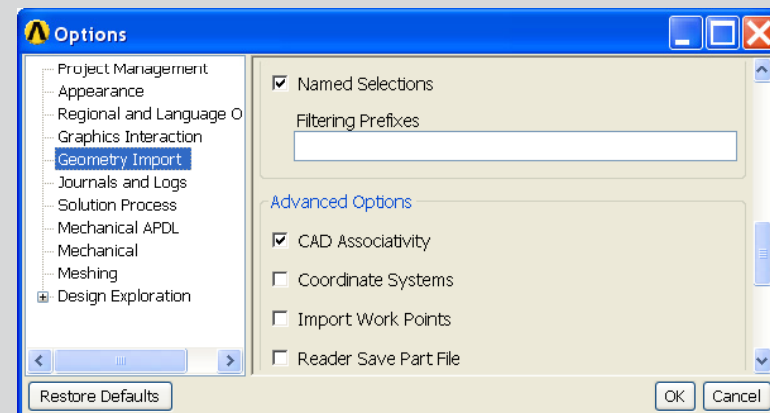
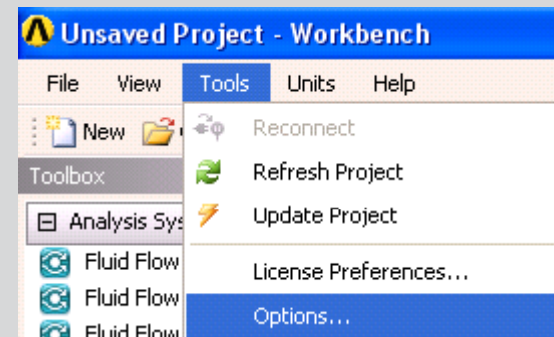


Yes – Selection propagates to all resultant entities
 No – Selection propagates to just one of the resultant entities or destroyed

How to use Named Selections?

Uses

- **Group entities under a single name**
- **Geometry for feature creation**
 - Cannot be solids
- **Define parameters for parametric study**
- **Assign names to boundaries for simulation**
 - When used as boundary names, user must enable the transfer of named selection from DesignModeler to other Workbench modules (like Meshing, Fluent Solver etc.)
- **For more information on boundary naming please see Appendix**



To enable transfer of Named Selections, in the Workbench window, go to Tools → Options... Click on Geometry Import, enable the check box next to 'Named Selections'.

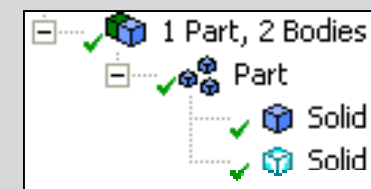
Empty the box under 'Filtering Prefixes' if all named selections are to be transferred, otherwise only the named selections with the prefix in the textbox will be transferred

Working with Parts

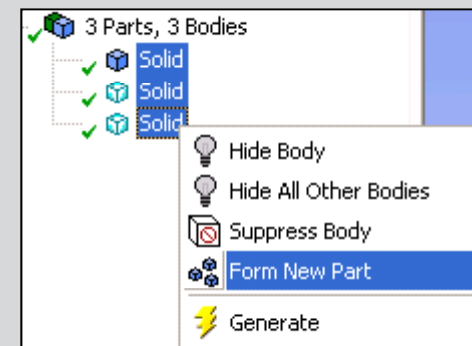
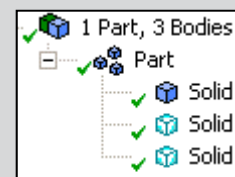
- By default, DM places each body into one part by itself
- **Single-body parts** : Individual parts are meshed separately
 - non-conformal mesh at interface
- **Multi-body part** : Two or more bodies in a single part
 - Conformal mesh at interface
- **Bodies can be grouped into a part in 2 ways :**
 - Select bodies. Click RMB and select Form New Part
 - Select bodies. Go to Tools > Form New Part



Single-body Parts



Multi-body Parts

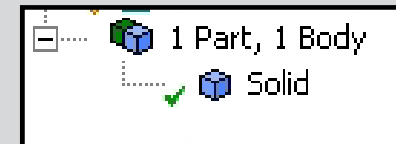


Single and Multi-Body Parts

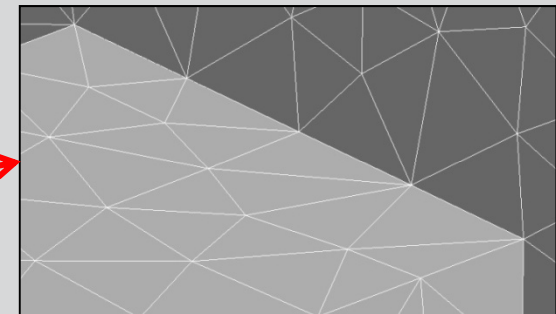
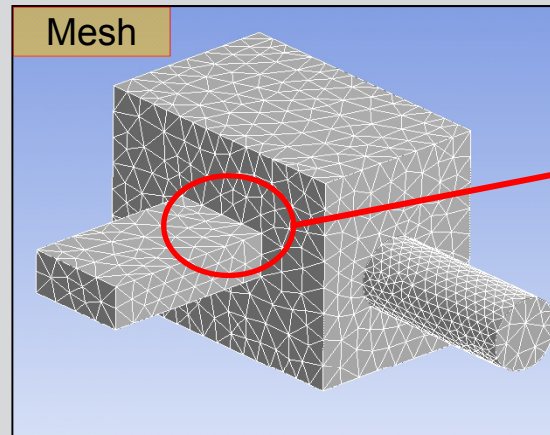
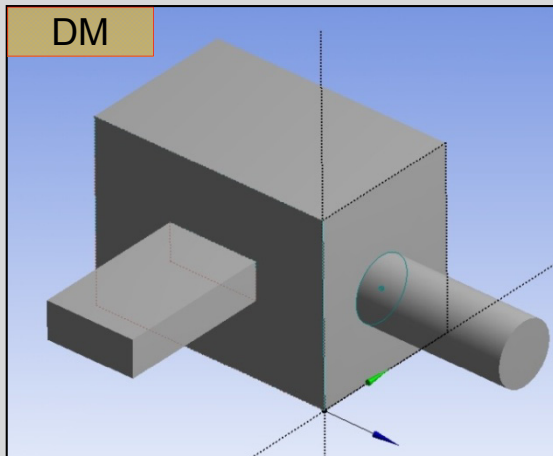
Case 1: In DM: 1 part, 1 body consisting of 1 solid

Result of transferring this geometry to Meshing

- contains: 1 solid ,1 body
- Entire solid meshed as one entity
- No internal faces

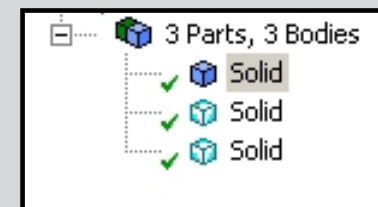


DM

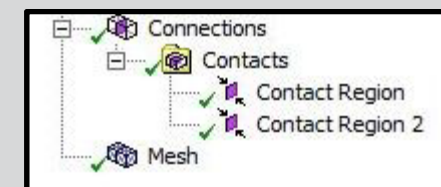


Single, Multi Solid Bodies

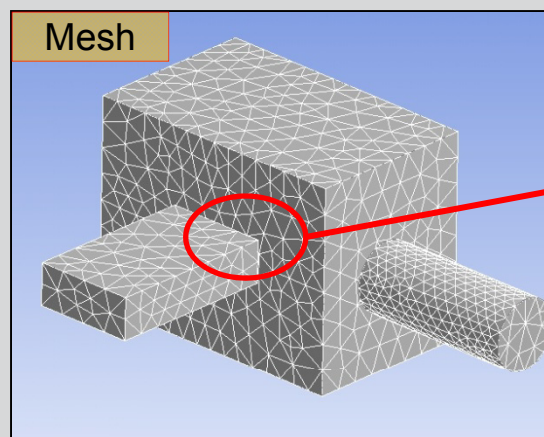
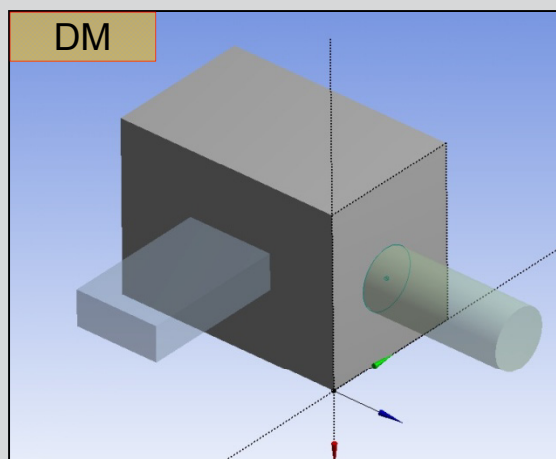
- **Case 2: In DM: 3 parts, 3 bodies consisting of 3 solids**
 - Result of transferring this geometry to Meshing
 - Contains : 3 solids, 3 bodies
 - Each solid is meshed independently
 - Nodes are not shared and do not line-up
 - No connection between the 3 mesh regions for fluid flow and/or heat transfer
 - Can connect using a Grid Interface in FLUENT/CFX



DM

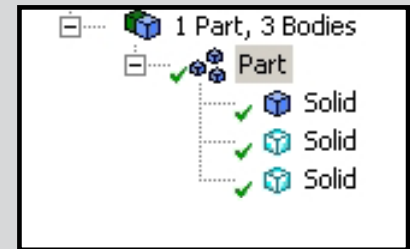


Mesh



Single, Multi Solid Bodies

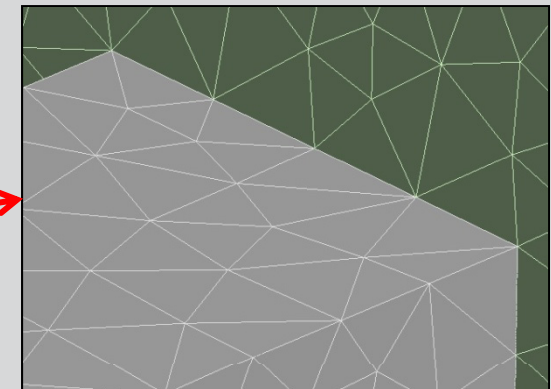
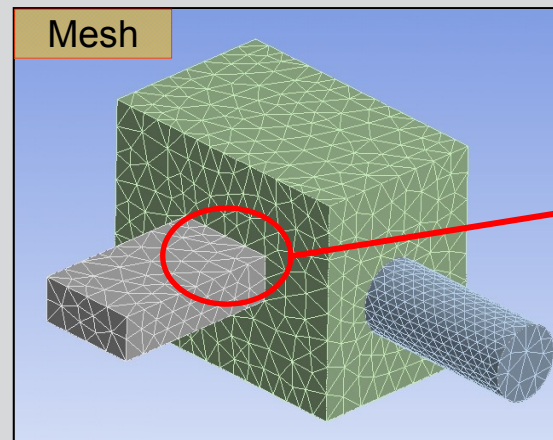
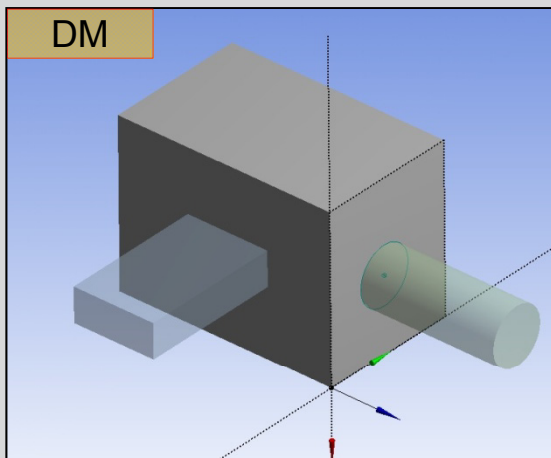
- **Case 3: In DM: 1 multi-body part, 3 bodies/solids**
- **Result of transferring this geometry for Meshing**
 - contains: 1 multi-body part, 3 bodies/solids
 - Each solid meshed independently but node connectivity among solids is preserved
 - Contains internal face which can be used for Post Processing



DM



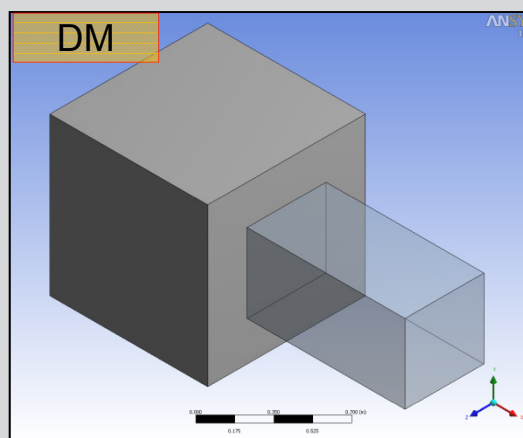
Mesh



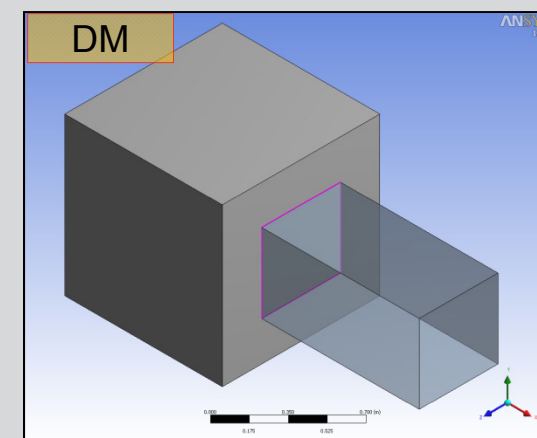
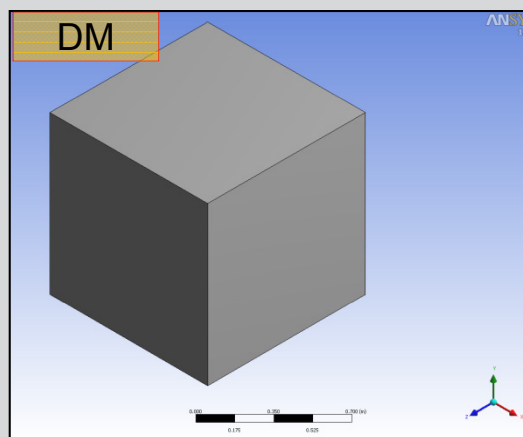
Works on Multi-body parts
only

- Common regions among Multi-body parts are not recognized until model is transferred to different application (AMP, etc.) or **Share Topology** operation is carried out
- If Named Selection is to be specified in DM, first perform Share Topology and then define Named Selection

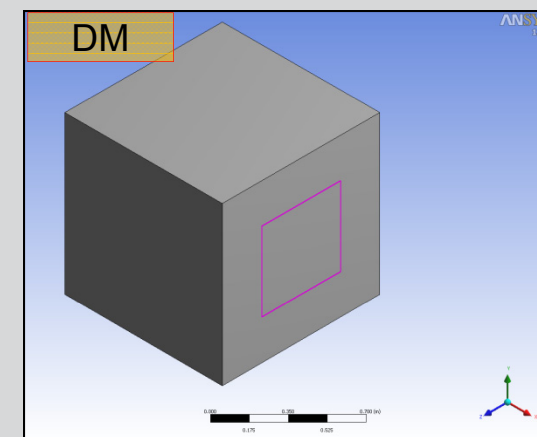
 Share Topology

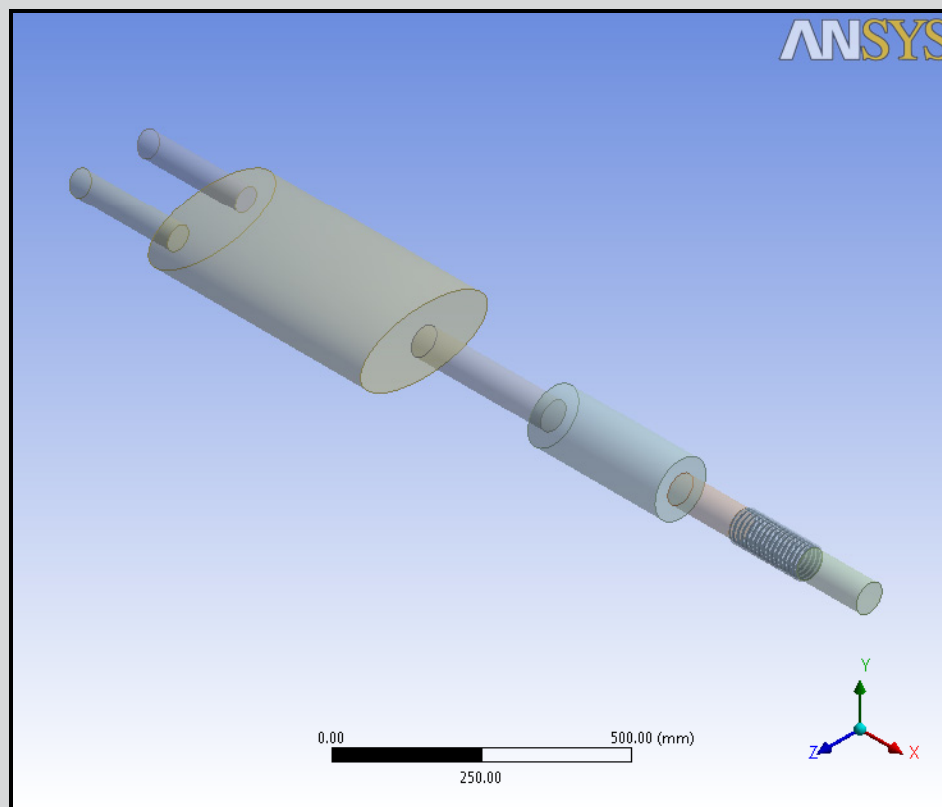


Before Share Topology



After Share Topology



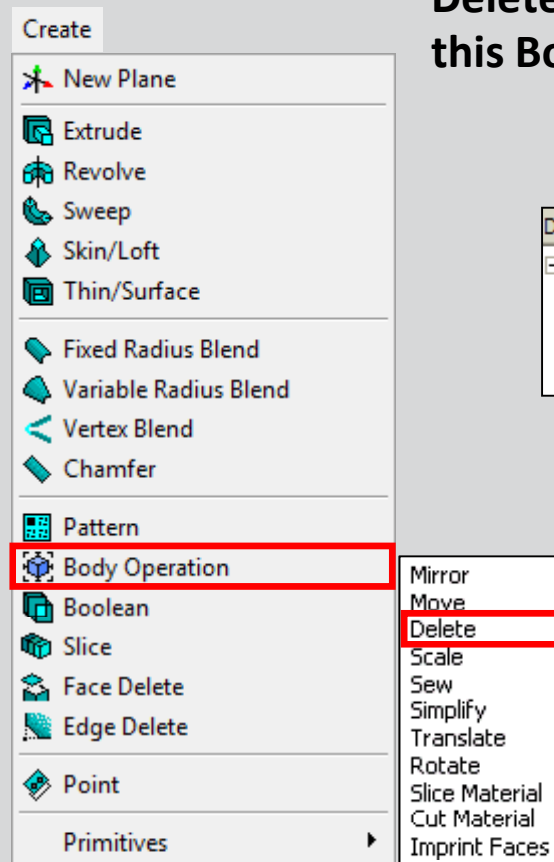


Contents

- **Body Operation Delete**
- **Extrude Merge Topology Example**
- **Variable Radius Blend**
- **Vertex Blend**
- **Chamfer**
- **Create Point**
- **Body Operation Cut Material**
- **Body Operation Imprint**
- **Edge Delete**
- **Named Selections for Boundaries in FLUENT**

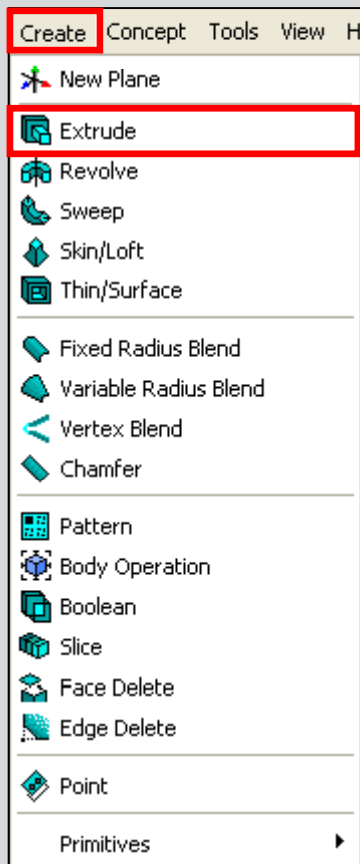
Body Operation: Delete

- Deletes the selected bodies
- Deleted bodies can be recovered by deleting or suppressing this Body Operation

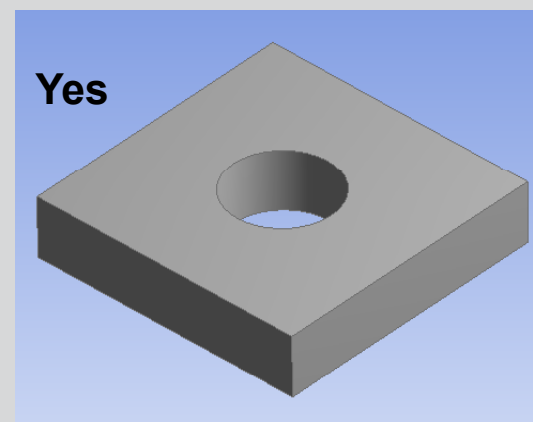
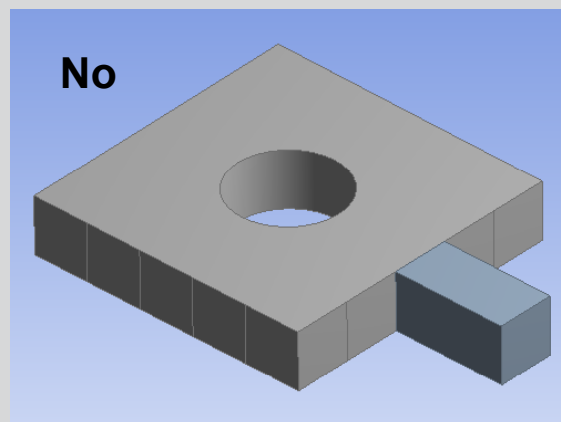
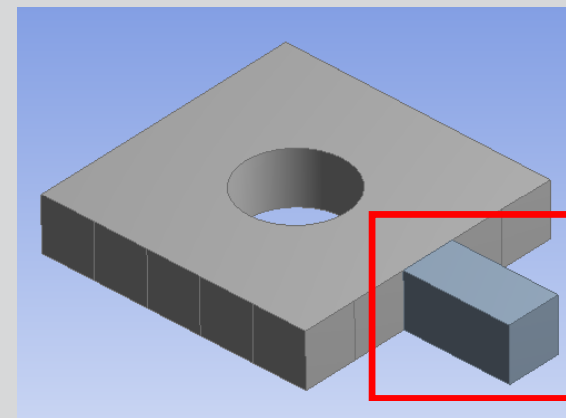


Details View	
Details of BodyOp3	
Body Operation	BodyOp3
Type	Delete
Bodies	1

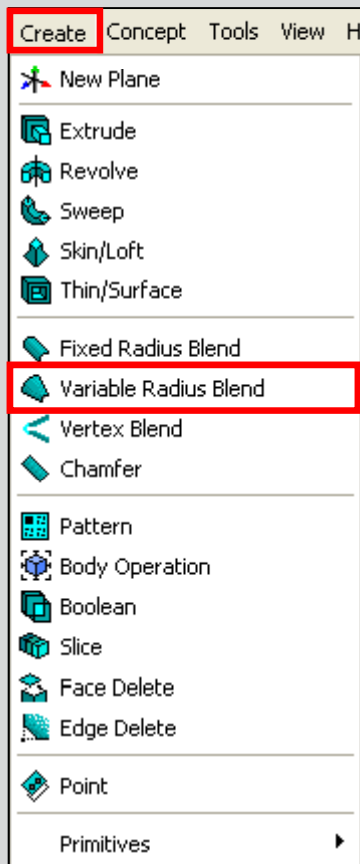
Extrude Feature: Example



- **Merge Topology : Example**
- **Create a rectangular box by extruding one of side faces as shown in adjacent image**
 - In previous Extrude operation, if Merge Topology option is “No”, the rectangular box will be successfully created using sweep operation
 - In previous Extrude operation, if Merge Topology option is “Yes”, the rectangular box cannot be created as there is no appropriate face for extrude

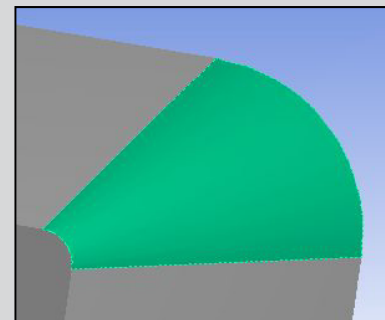


Variable Radius Blend

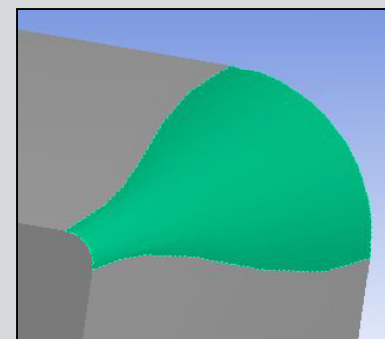


- Creates a blend of variable radius on face/edges
- The Details view is used to specify the start and end radius
- Specify Linear or Smooth Transition for Blend

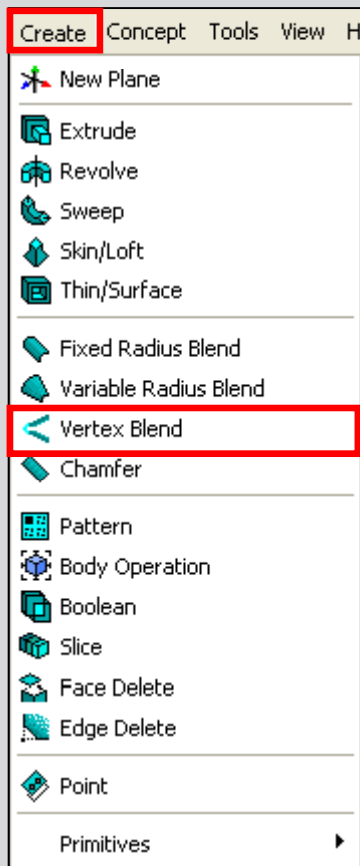
Details of VBlend1	
Variable-Radius Blend	VBlend1
Transition	Linear
Edges	3
Edge 1	
Start Radius (≥ 0)	1 mm
End Radius (≥ 0)	4 mm
Edge 2	
Start Radius (≥ 0)	4 mm
End Radius (≥ 0)	6 mm
Edge 3	
Start Radius (≥ 0)	4 mm
End Radius (≥ 0)	1 mm



Linear Blend

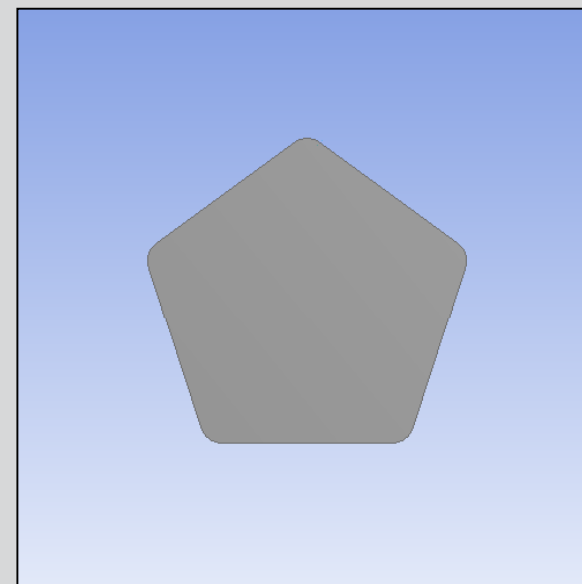
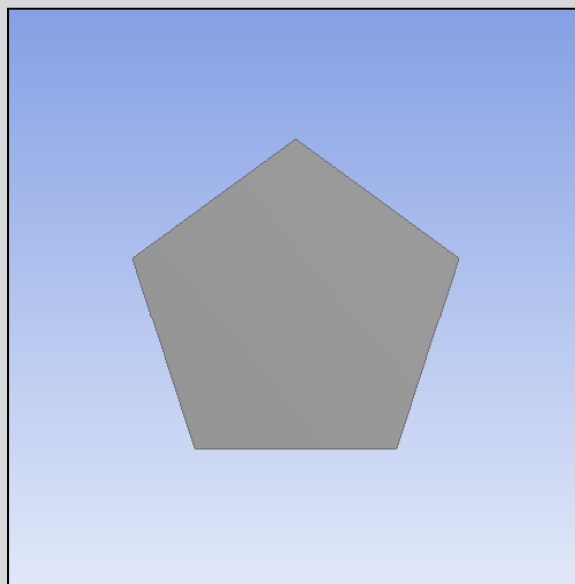


Smooth Blend

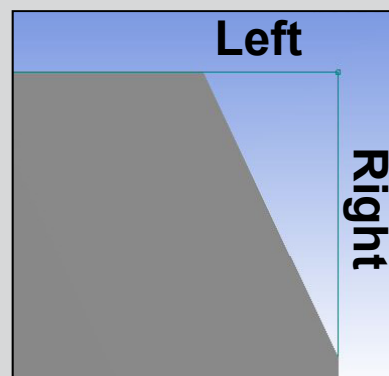
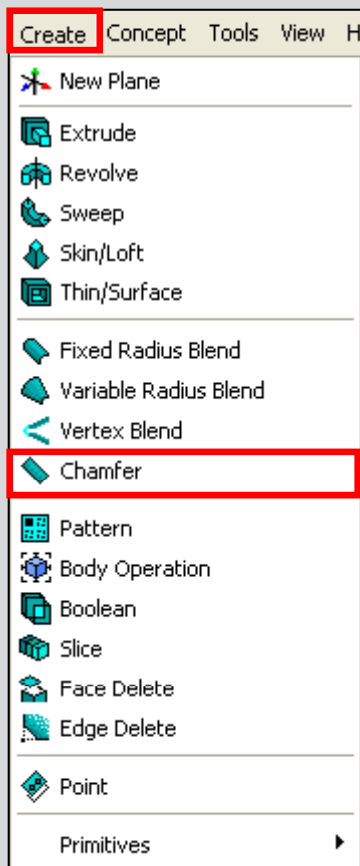


- Creates a blend on vertices
- Used to blend corners of Surface bodies
- The Details view is used to specify the Radius and Vertices

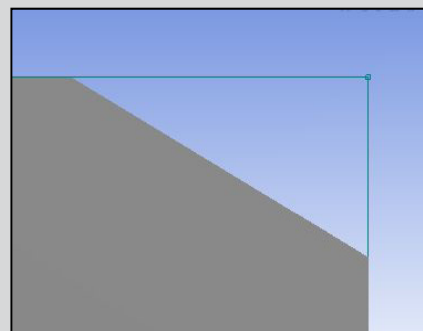
Details of VertexBlend3	
Name	VertexBlend3
☐ FD1, Radius (>0)	1 m
Vertices	5



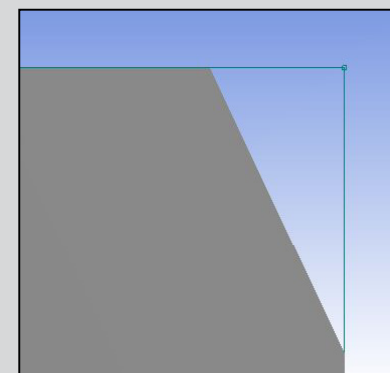
- Creates a chamfer on edge/faces
- The Details view is used to specify the left and right length or length and angle



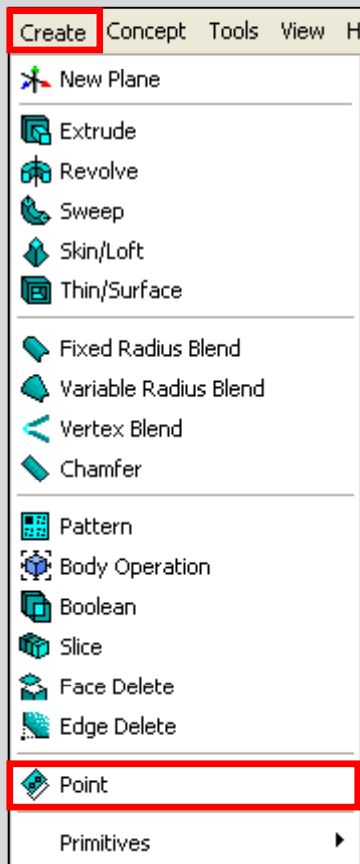
Details of Chamfer1	
Chamfer	Chamfer1
Geometry	1 Edge
Type	Left-Right
☐ FD1, Left Length (>0)	3 mm
☐ FD2, Right Length (>0)	6 mm



Details of Chamfer1	
Chamfer	Chamfer1
Geometry	1 Edge
Type	Left-Angle
☐ FD1, Left Length (>0)	6 mm
☐ FD3, Angle	30 °



Details of Chamfer1	
Chamfer	Chamfer1
Geometry	1 Edge
Type	Right-Angle
☐ FD2, Right Length (>0)	6 m
☐ FD3, Angle	30 °



- Used to create Construction Point, Point Load or Spot Weld
- Can choose definition types for creating Construction point :
- Single, Sequence by Delta, Sequence by N, From Coordinates File, Manual input
- Text file containing external point data in specific format can be imported through From Coordinates File option

```
# List of Point Coordinates

# Format is integer Group, integer ID, then X Y Z all
# delimited by spaces, with nothing after the Z value.

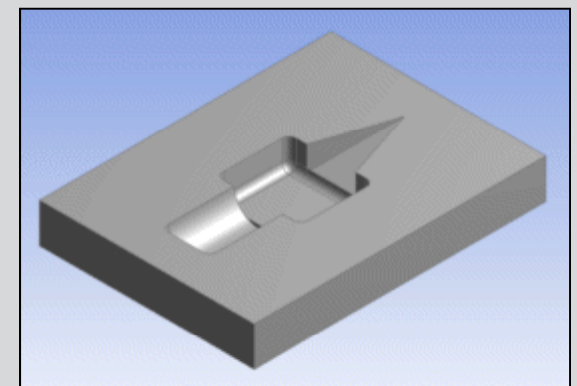
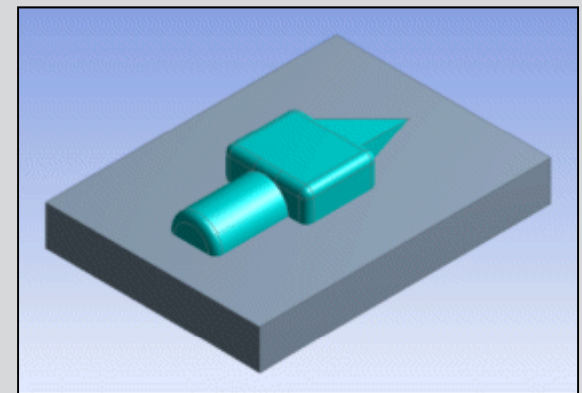
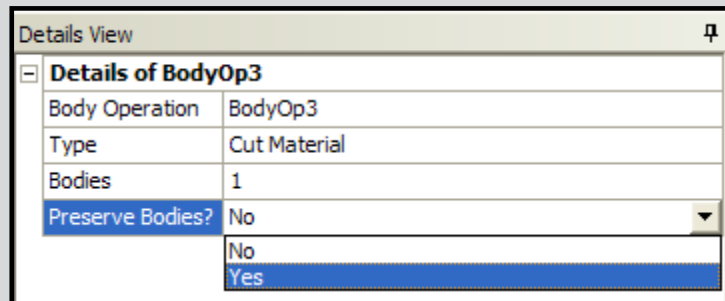
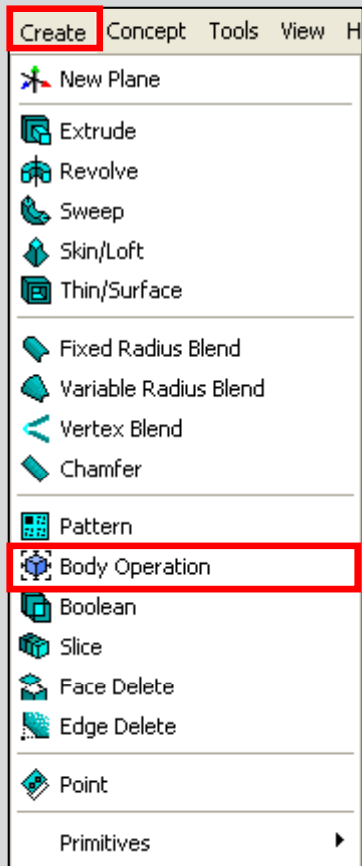
# Group 1
1 1 20.1234 25.4321 30.5678
1 2 25.2468 30.1357 35.1928
1 3 15.5555 16.6666 17.7777

#Group 2
2 1 50.0101 100.2021 7.1515
2 2 -22.3456 .8765 -.9876
2 3 21.1234 22.4321 23.5678
```

External Point Data Format

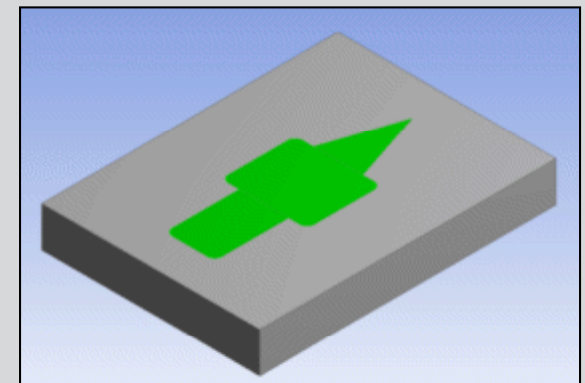
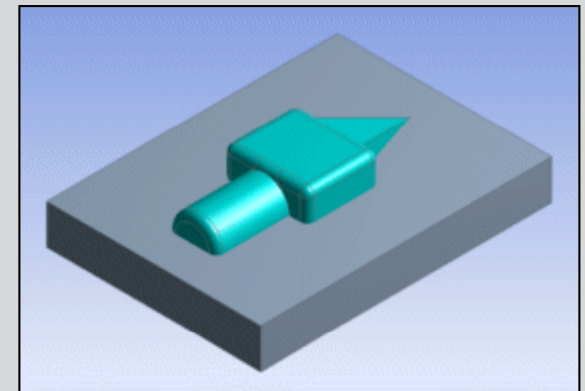
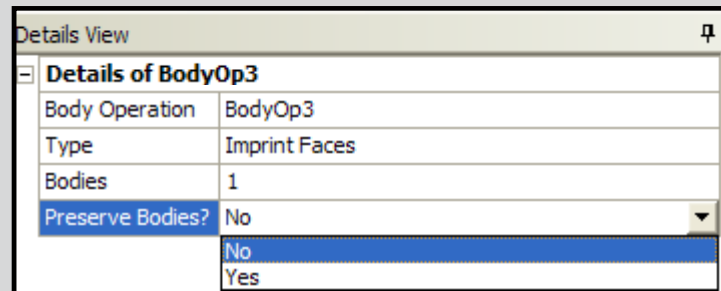
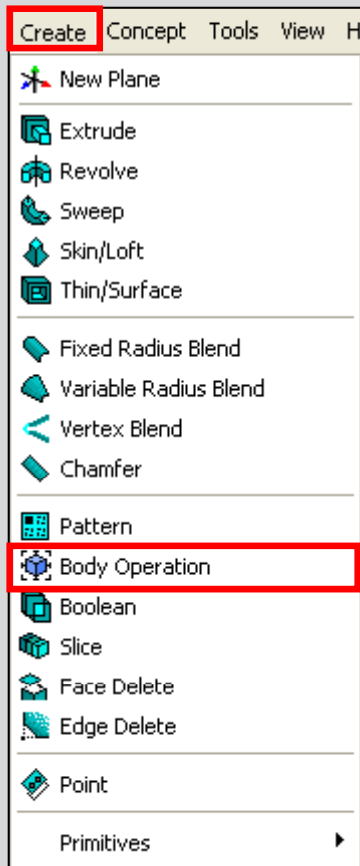
Body Operation: Cut Material

- This option is available when the model has any active bodies
- Uses the selected bodies to cut from other active bodies

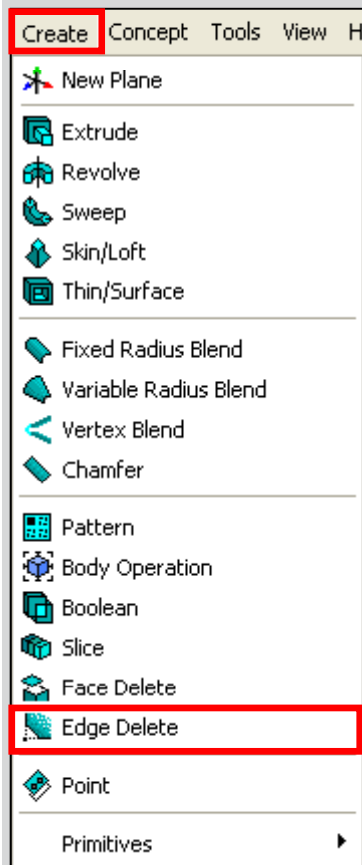


Body Operation: Imprint

- This option is available when the model has any active bodies
- Uses the selected bodies to imprint on the active bodies

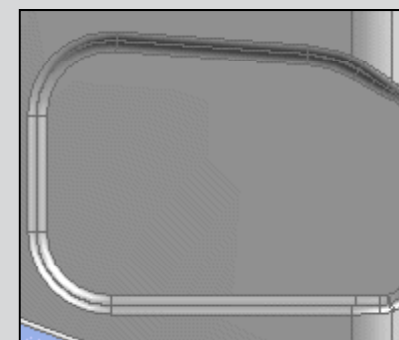
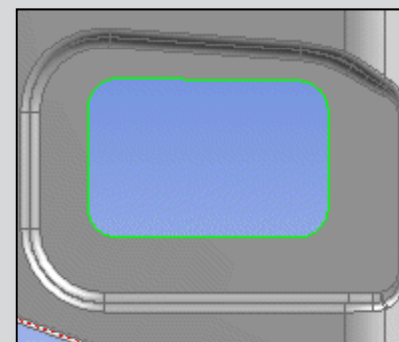


Edge Delete



- Used to remove blends, chamfers, and holes from surface bodies
- Can also be applied to imprinted edges from all body types
- Can choose healing types: Automatic, Natural or No healing

Details of EDelete3	
Edge Delete	EDelete3
Edges	8
Healing Method	Natural Healing



Named Selections for Boundaries in Fluent

Named selections can be assigned a name that corresponds to the desired boundary type

Named selection	Boundary Type
Inletxx	Velocity Inlet
Outletxx	Pressure Outlet
Wallxx	Walls
Pressure_inletxx	Pressure Inlet
(Other names not recognized as boundary type)	Walls

- “xx” in the names stand for any suffix
- Example: one can name a face to be assigned as boundary type of ‘Velocity Inlet’ as ‘inlet1’
- First letter of the names can be capital letters or small letters
- When such a model is transferred to Fluent, the Fluent solver will automatically recognize the boundary types from the names and assign the boundary type to these boundaries based on their assigned names, saving time for setup.