

Lecture 4 Planes and Sketches

14.0 Release

Fluid Dynamics

Structural Mechanics

Electromagnetics

Systems and Multiphysics

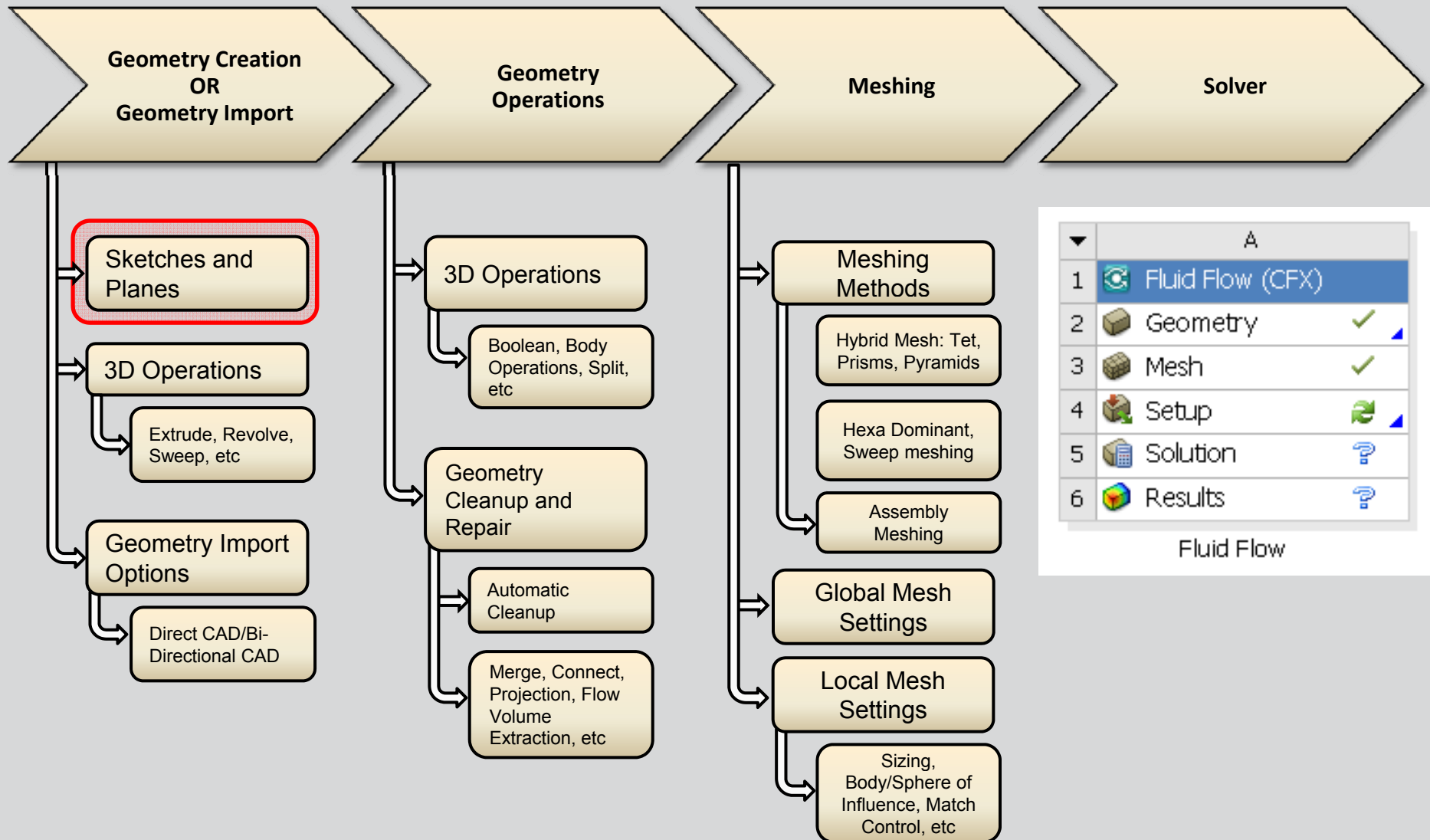
Introduction to ANSYS DesignModeler

Planes and Sketches

What you will learn from this presentation:

- **Concept of Plane and Sketch**
- **How to create Planes and Sketches**
- **Sketching interface**
- **Sketching Toolboxes**
- **How to Draw, Modify, Dimension and Constrain sketches**

Preprocessing Workflow



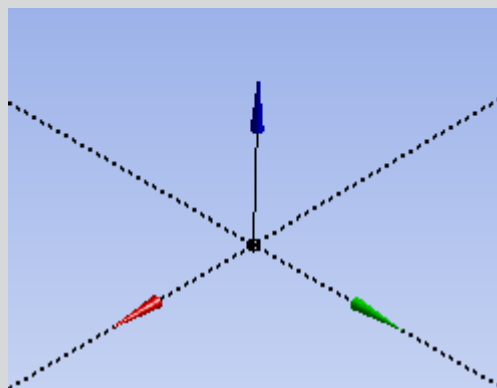
Concept of Planes and Sketches

A plane is required to create a 2D sketch

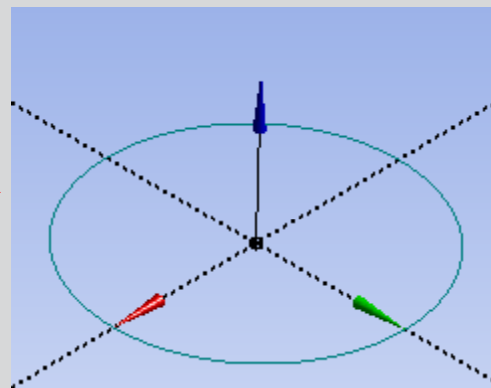
- Also required for some body operations like “move (alignment)”
- 2D/3D geometry can be constructed using a sketch as a base object

Sequence to create a basic 3D geometry :

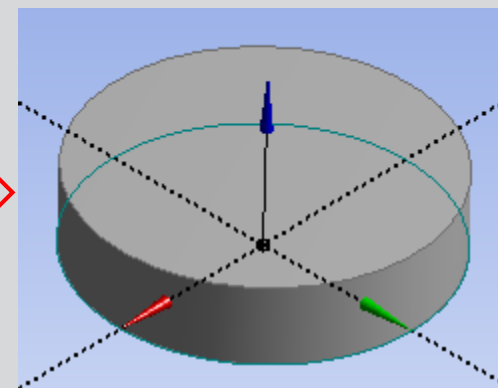
- Create or select a plane
- Create a sketch
- Apply a modeling operation to the sketch



Select desired plane



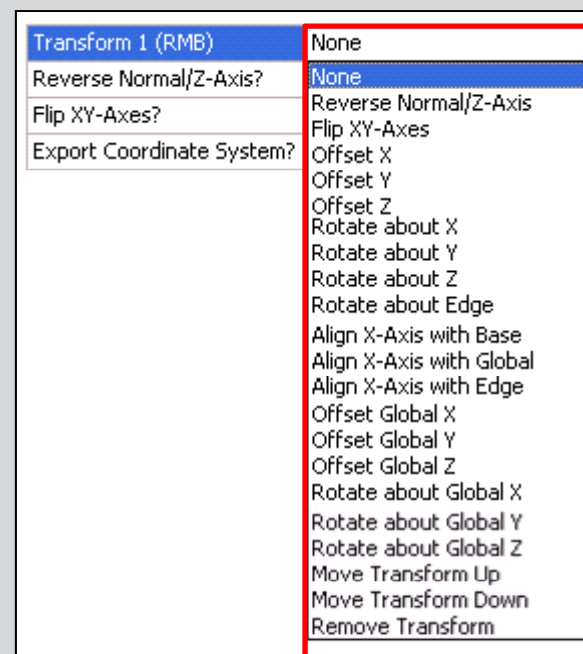
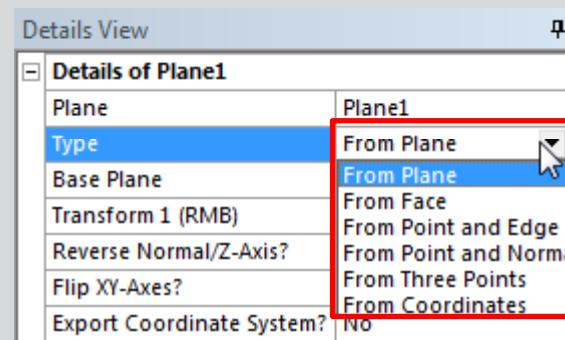
Create sketch on selected plane



Create 3D geometry through Extrusion operation from the sketch

Working with Planes

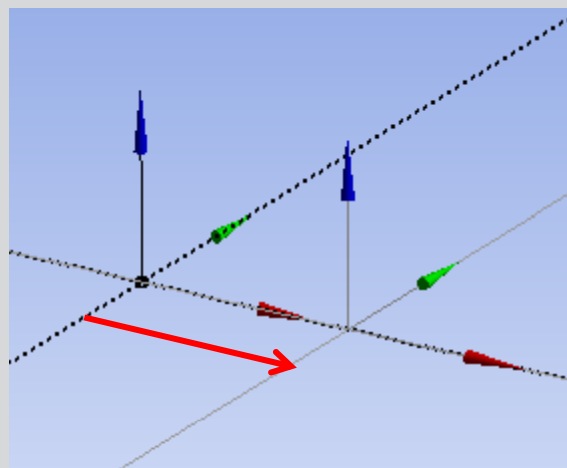
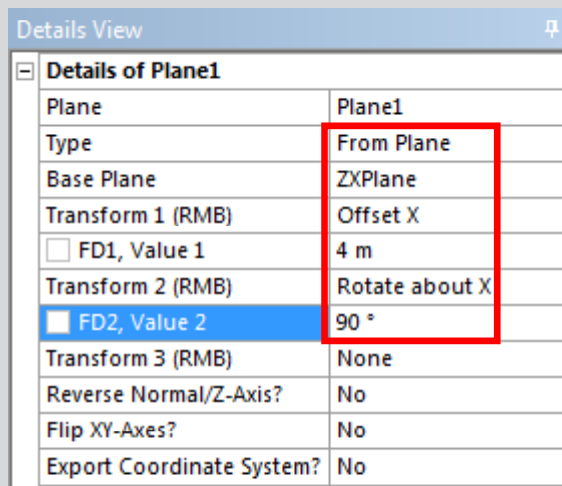
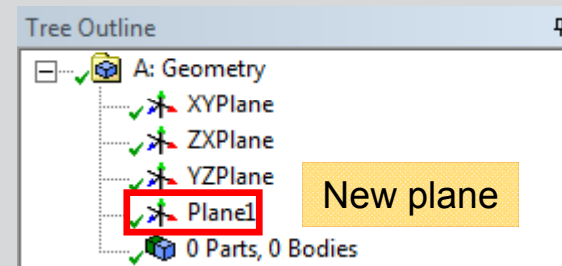
- At start-up, three default planes are available (XY, ZX & YZ) on which sketches can be created
 - New planes can be created to host sketches elsewhere
- Create Plane
 - In the Details View set the Plane Type.
 - This determines how the plane is created, for example, by transforming an existing plane or from a selected face.
 - Optionally apply one or more transformations
- Two examples follow...>



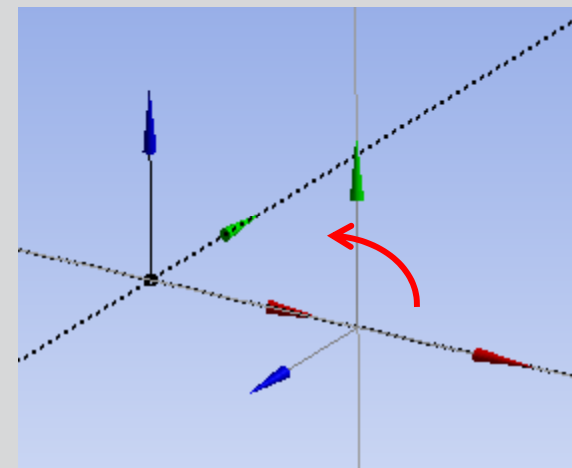
New Plane Creation Example: From Plane

Create new plane by transforming existing plane

- Set Type to From Plane and select Base Plane (ZX Plane)
- Apply Transform 1 (Offset X 4m)
- Apply Transform 2 (Rotate about X 90°)
 - Transforms use local coordinate system
 - Add as many transforms as required



Transform 1



Transform 2

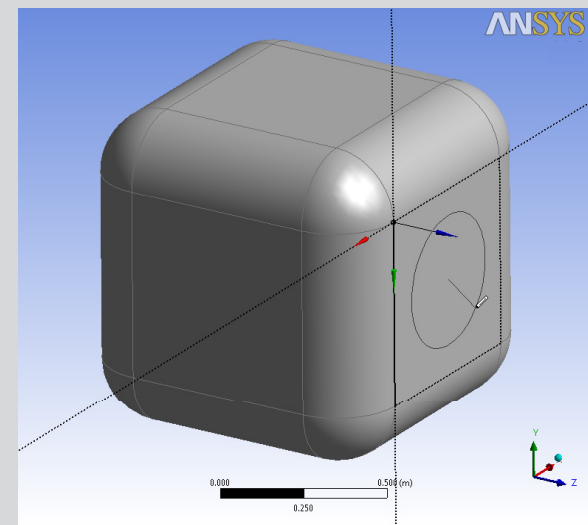
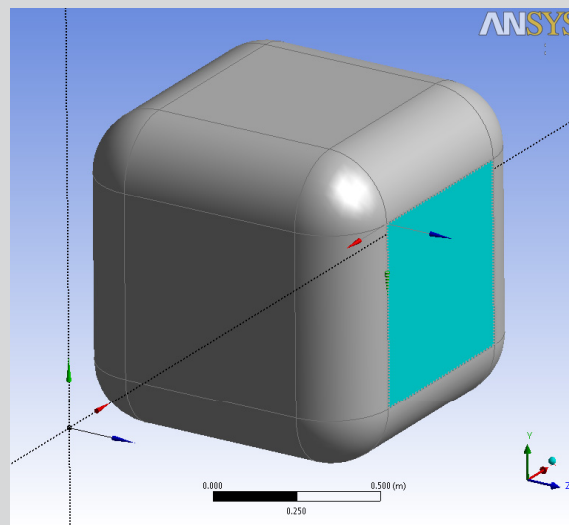


New Plane Creation Examples: From Face

Create new plane on geometry surface

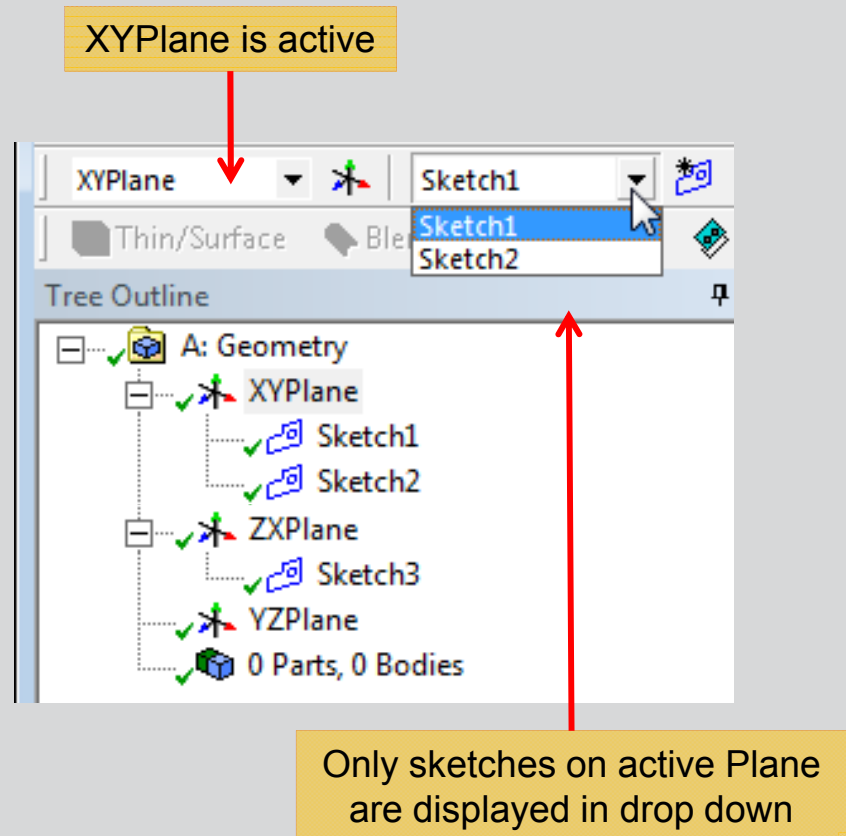
- Set Type to From Face and select Base Face (on geometry)
- Generate plane and create sketches
 - Can be completed in one step
 - Select face and enter sketching mode (plane and sketch are created automatically)

Details View	
Details of Plane2	
Plane	Plane2
Type	From Face
Subtype	Outline Plane
Base Face	Selected
Use Arc Centers for Origin?	Yes
Transform 1 (RMB)	None
Reverse Normal/Z-Axis?	No
Flip XY-Axes?	No
Export Coordinate System?	No



Create New Sketch

- Creates a new sketch (empty pad for graphical sketching) on the active plane
 - New sketches are placed in the Tree Outline beneath their associated plane
- To start creating the Sketch, switch to Sketching mode
 - Selecting a Plane with no sketches and entering sketch mode automatically creates a sketch
 - Multiple sketches can be created in any given plane
 - New sketches are created in the active plane. Check which plane is set to active before creating the new sketch.

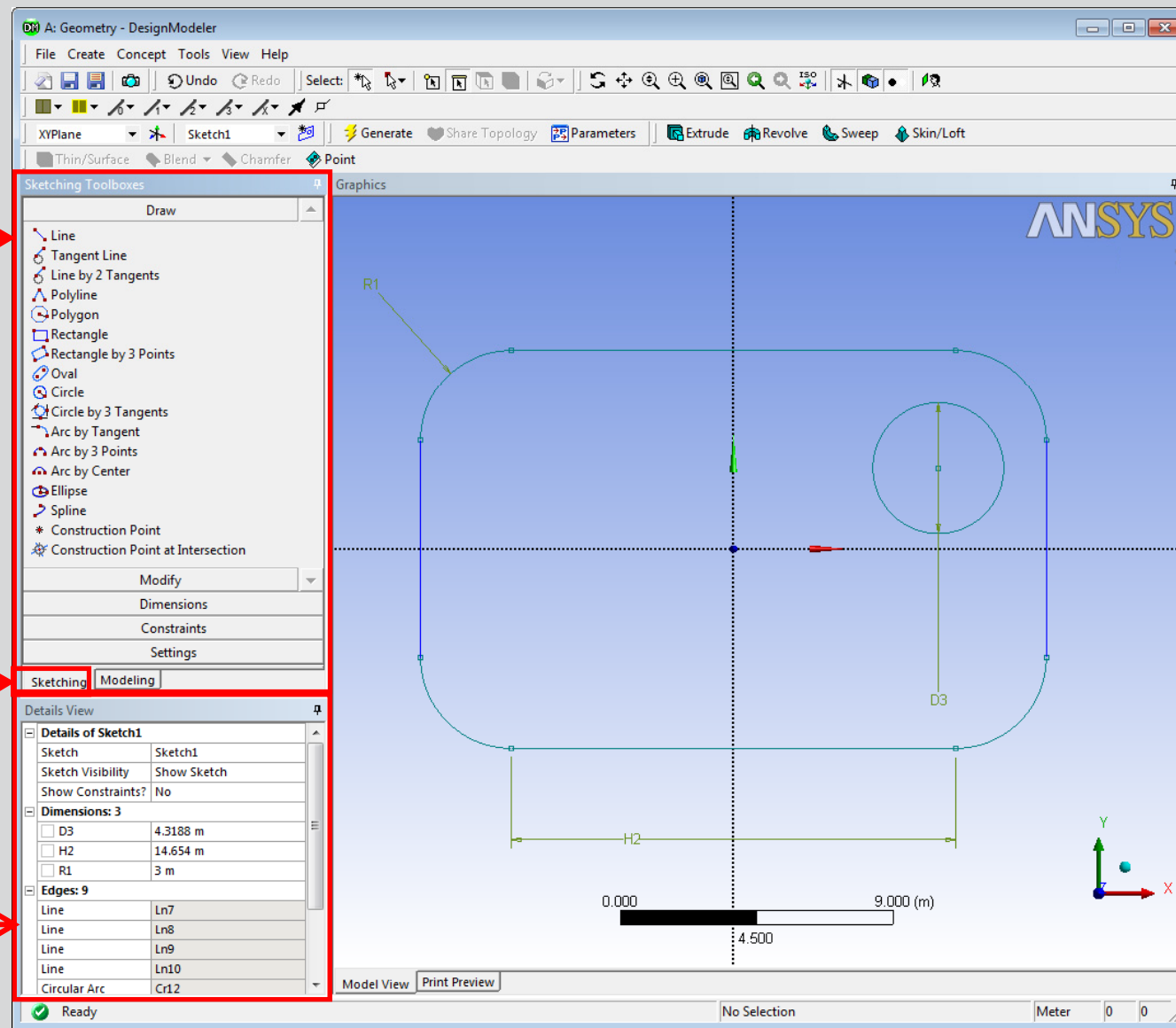


Sketching Interface

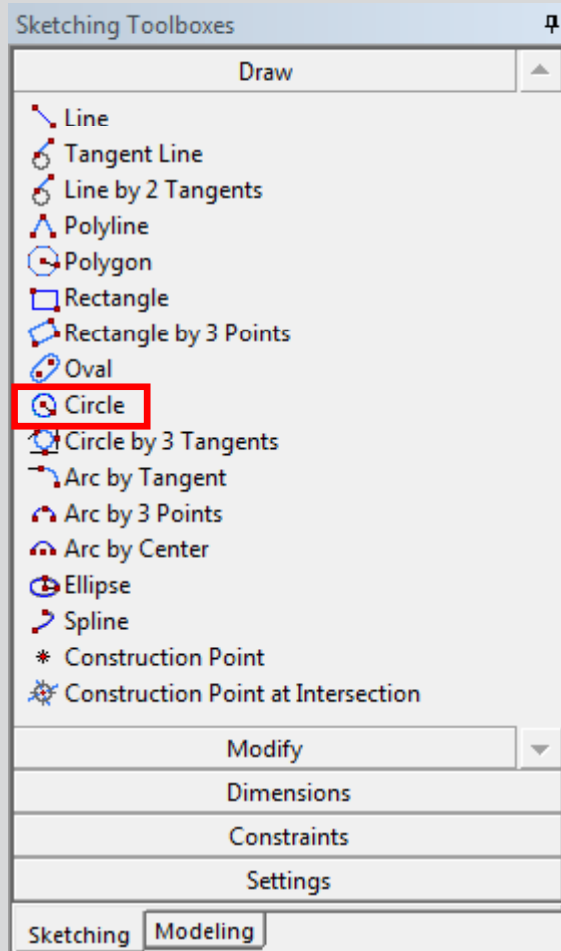
Sketching tools are arranged in five toolboxes - Draw, Modify, Dimensions, Constraints and Settings

Sketching toolboxes are available when you switch to sketching mode

Details of the current sketch (Dimensions, Constraints etc) are displayed in the Details View.



Draw Toolbox Examples: Circle



Many options are straight forward to use

- GUI status bar provides instructions for completing each operation. For example, when Circle is selected...

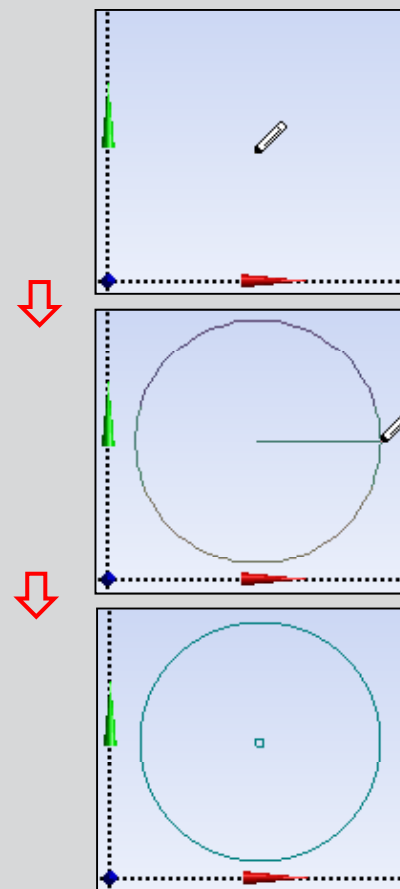
✓ Circle -- Click, or Press and Hold, for center of circle

Follow instructions in the status bar: Choose location for centre point

✓ Circle -- Click, or Release, to define radius

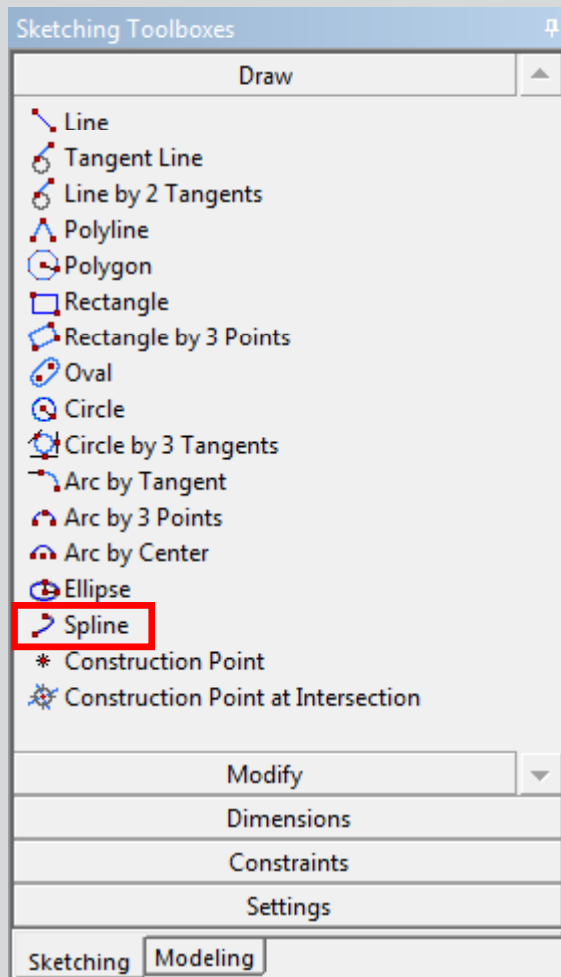
Follow instructions in the status bar: Define Radius

Done!



- Another example, Spline...>

Draw Toolbox Examples: Spline



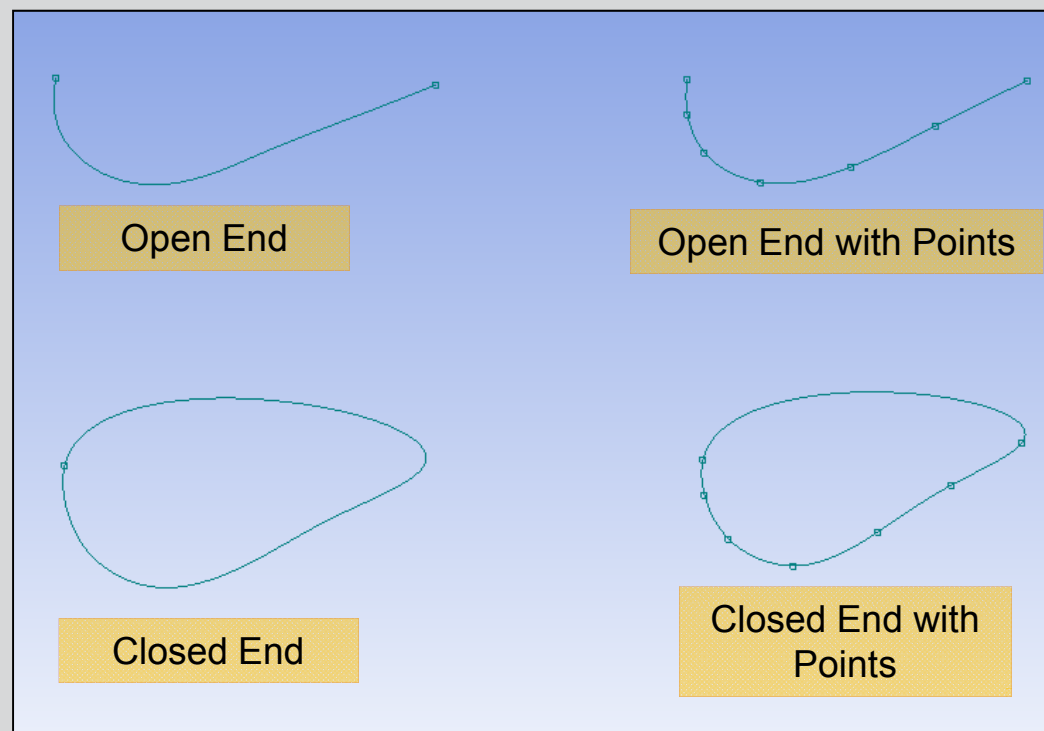
Right click provides additional commands

- Instructions issued in the status bar

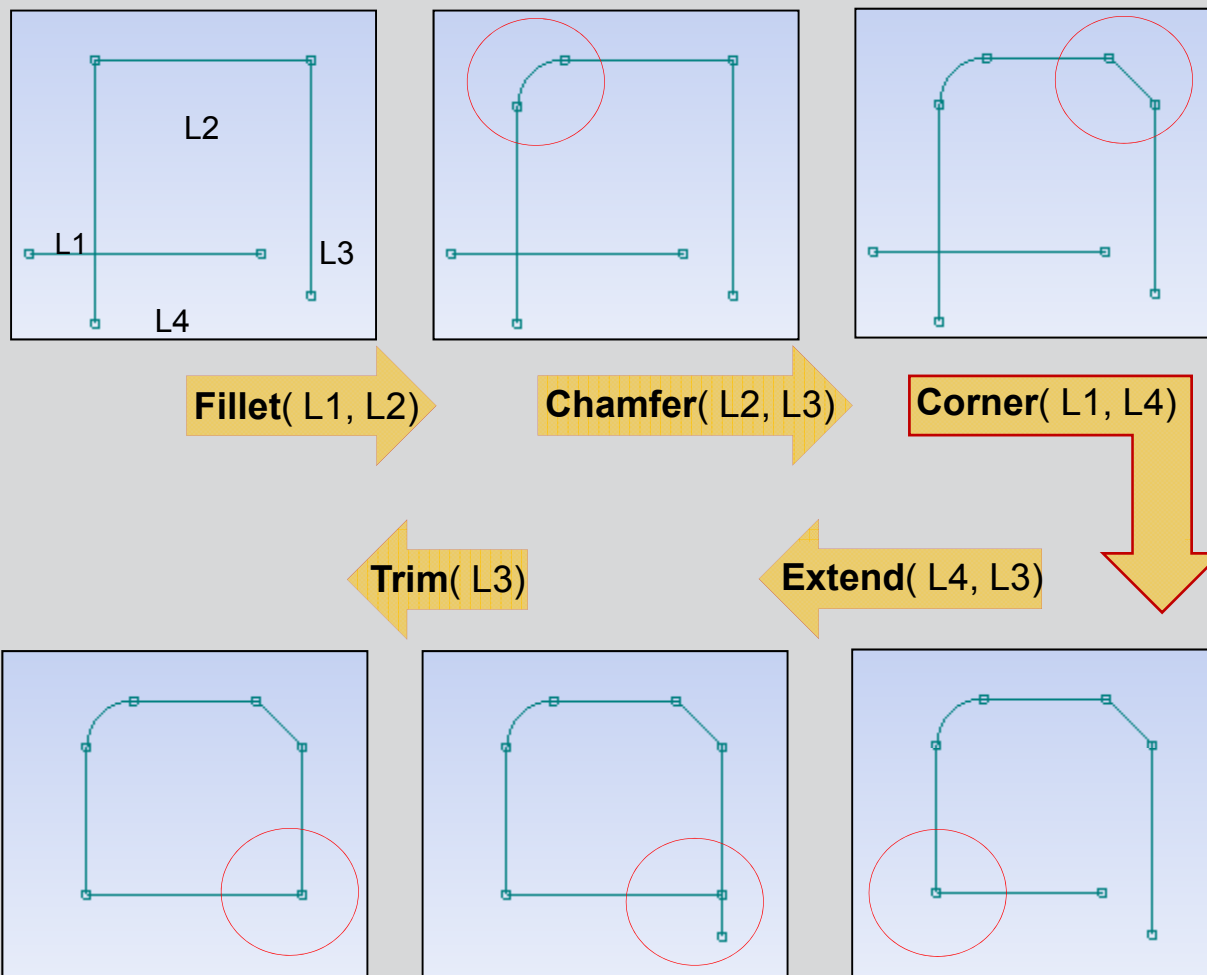
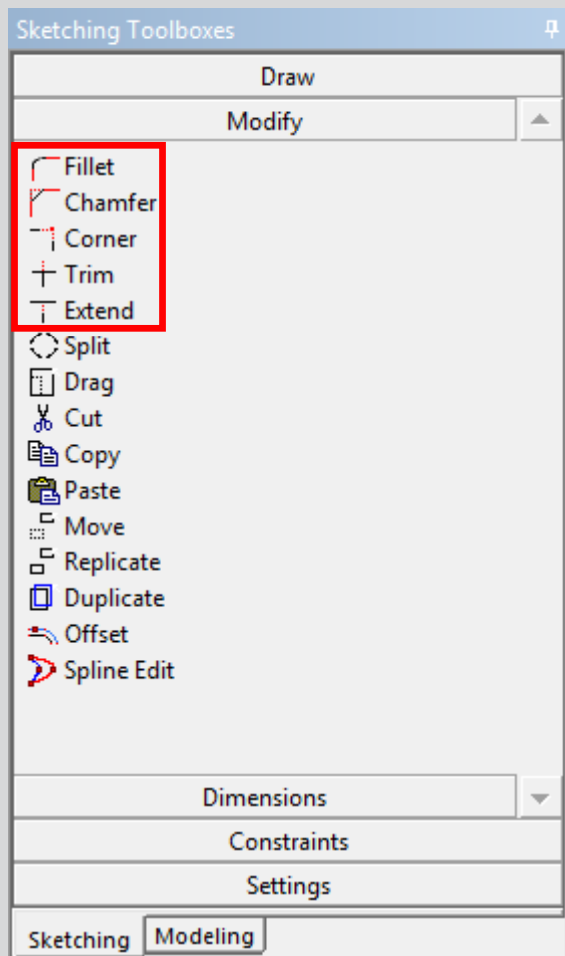
✓ Spline -- Click for start of spline

✓ Spline -- Click for next spline vertex, or use RMB to back up

- Open End
- Open End with Points
- Closed End
- Closed End with Points



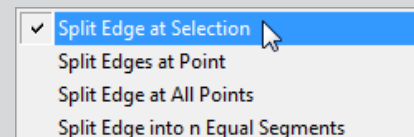
Modify Toolbox Examples: Fillet, Chamfer, Corner, Extend, Trim



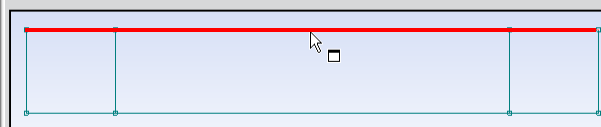
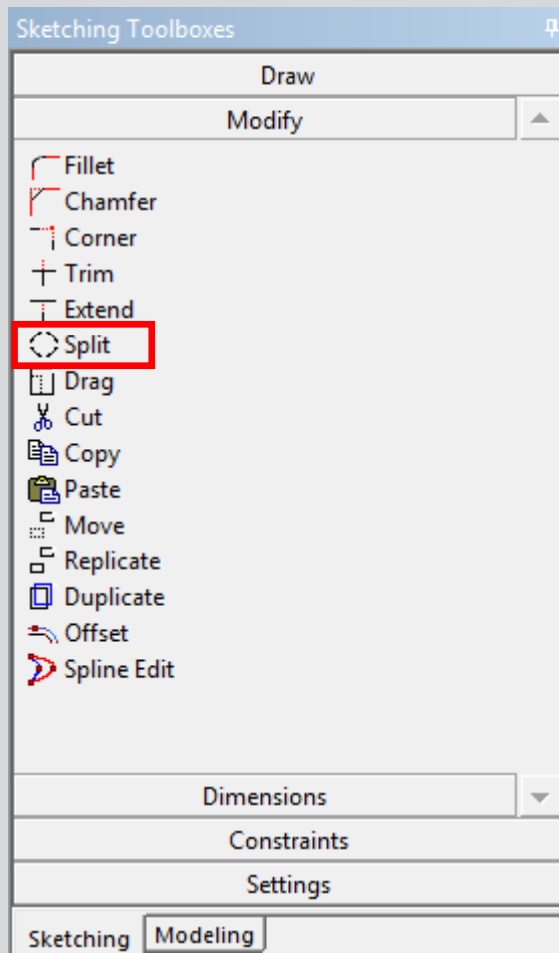
Modify Toolbox Examples: Split

Using Split

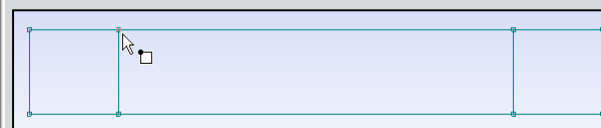
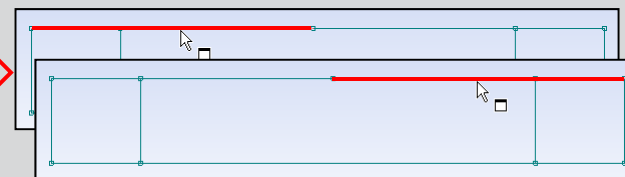
- Right click to specify option
 - For segments option a field will appear in the toolbox



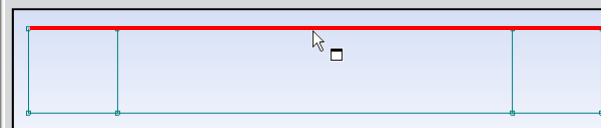
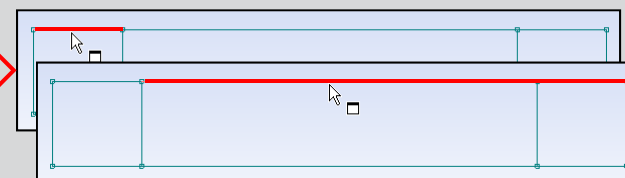
☐ Split Equal Segments n = 3



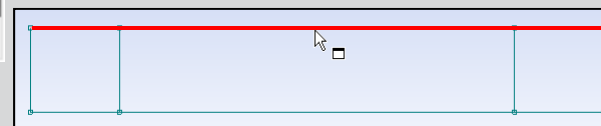
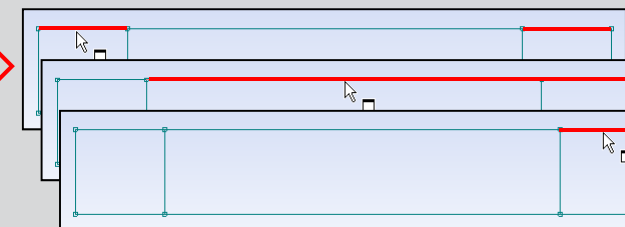
Split Edge at Selection



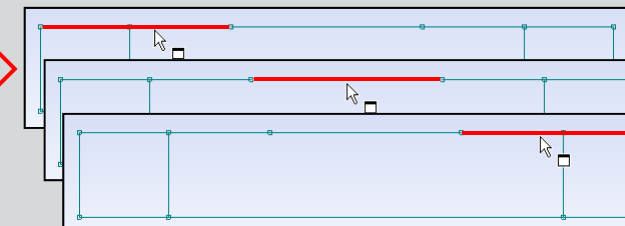
Split Edges at Point



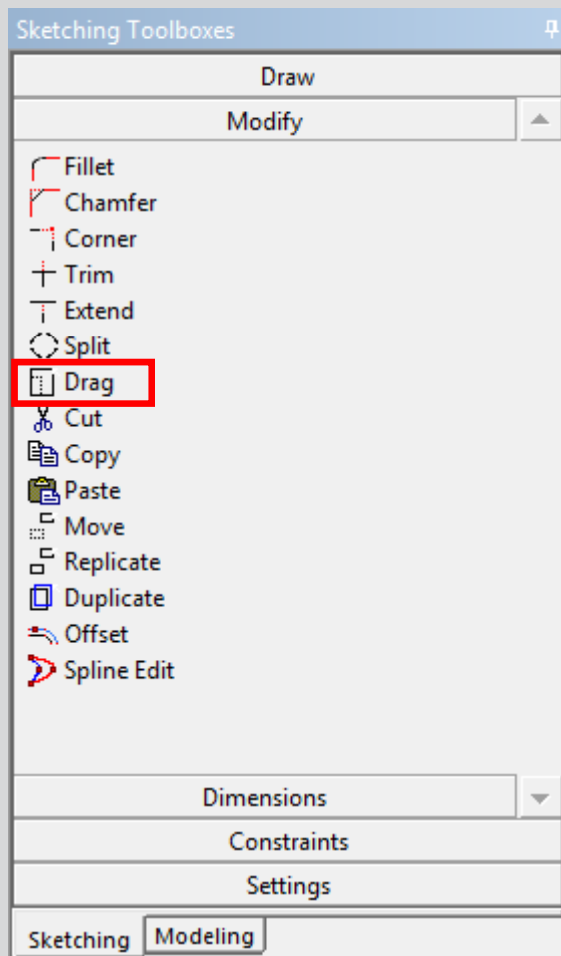
Split Edge at all Points



Split Edge into n Equal Segments

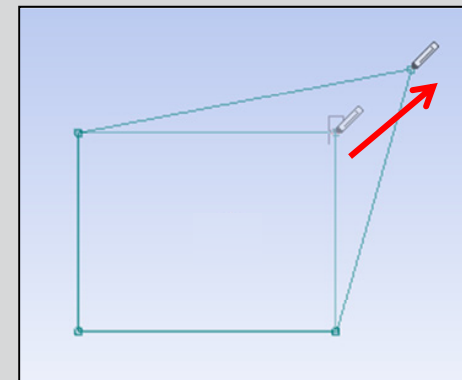
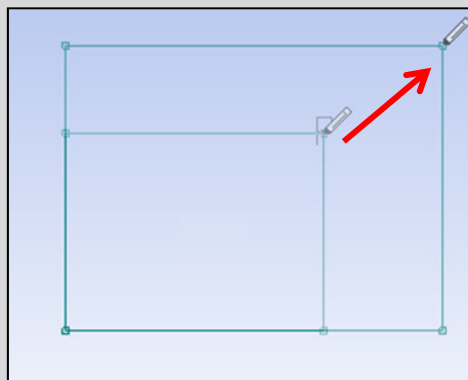


Modify Toolbox Examples: Drag



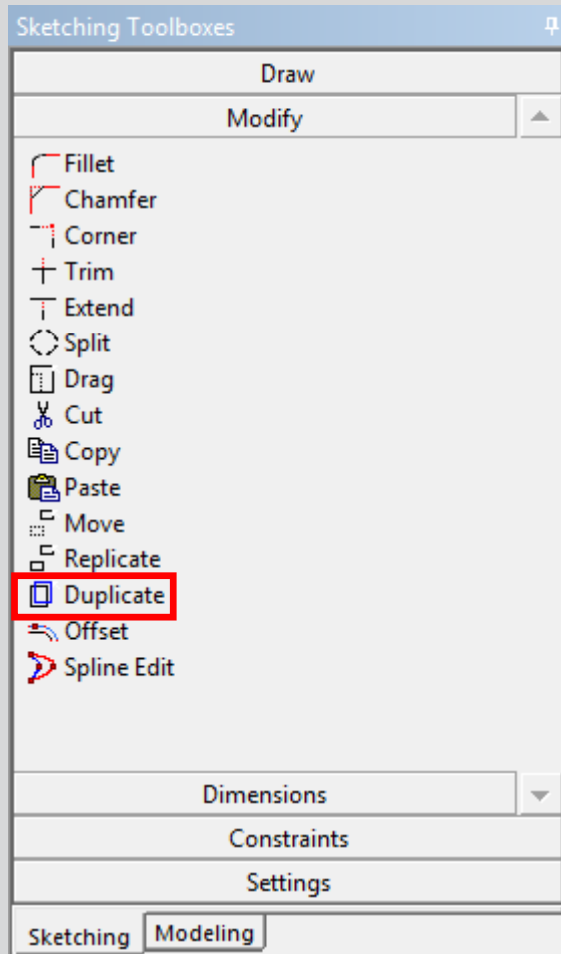
Drag a point or edge to a desired location

- Can also be used for scaling (up or down) the sketch
 - How the model changes depends on what is selected, existing constraints and dimensions
 - Examples here, dragging a corner point connected to horizontally and vertically constrained edges compared to the same where edges are unconstrained.



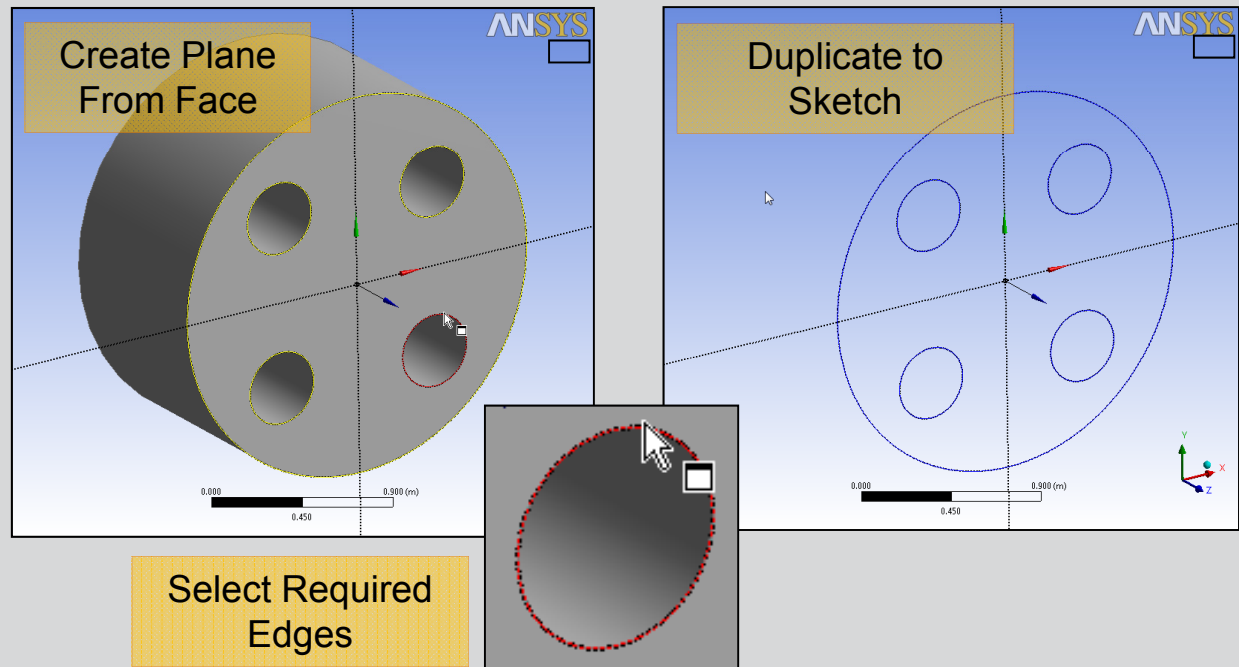
- Can pre-select multiple entities before issuing the Drag function

Modify Toolbox Examples: Duplicate

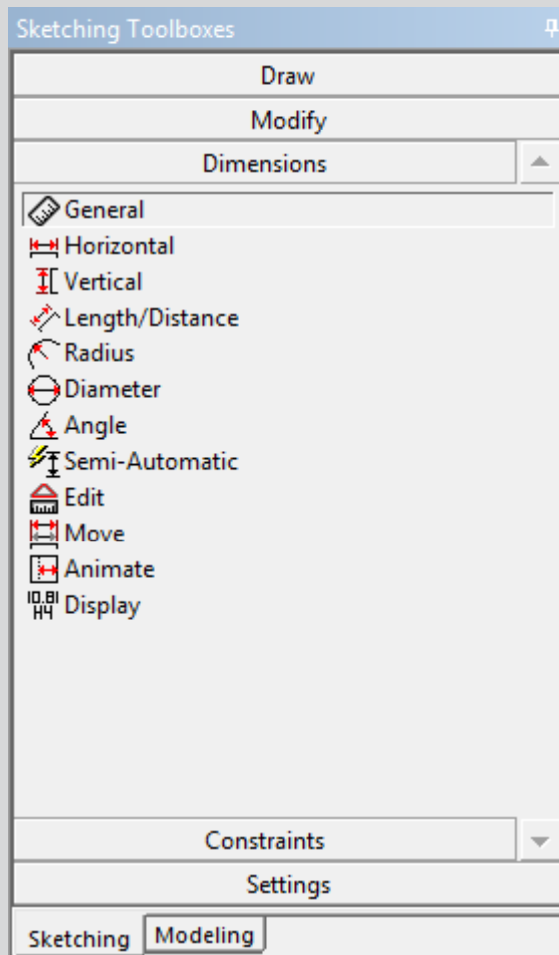


A new sketch can be created by referencing existing geometry

- Allows boundary of plane to be duplicated as new sketch entities
- All sketches have to be in the same plane

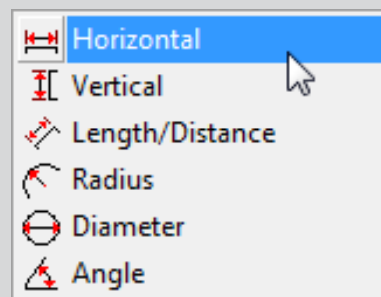


Dimensions Toolbox



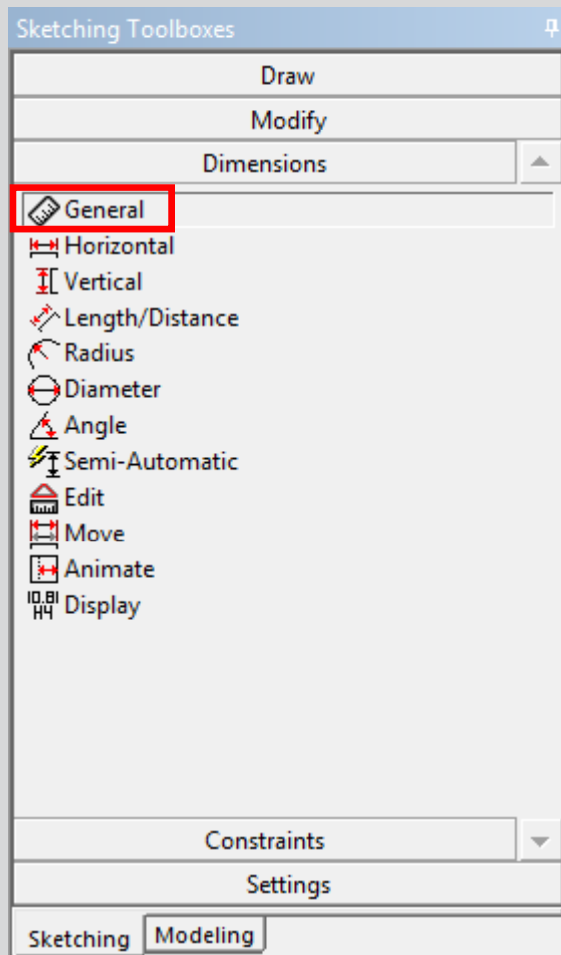
Used for specifying dimensions of sketch entities

- Pick specific dimension types or..
- General
 - Selects appropriate dimension type based on entity selection
 - Right click provide quick access to specific dimension types



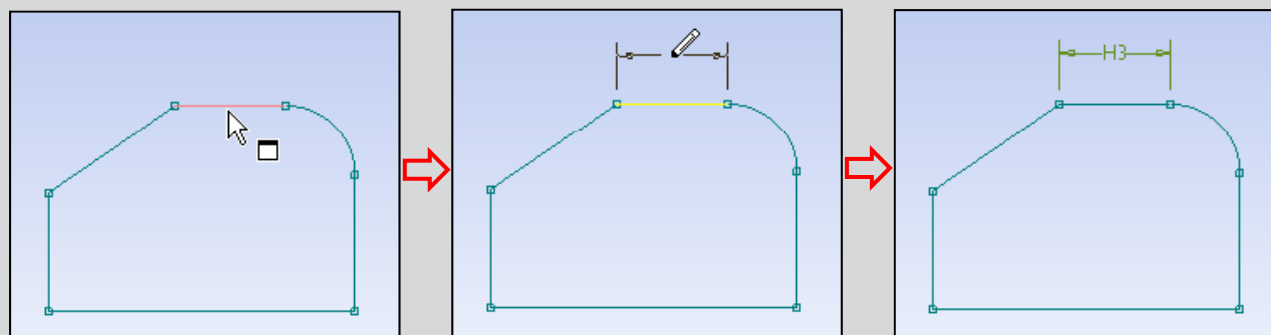
- Semi-Automatic
 - Cycles through entities until sketch is fully constrained (or) user chooses to exit
- Two examples follow...>

Dimensions Toolbox Example 1

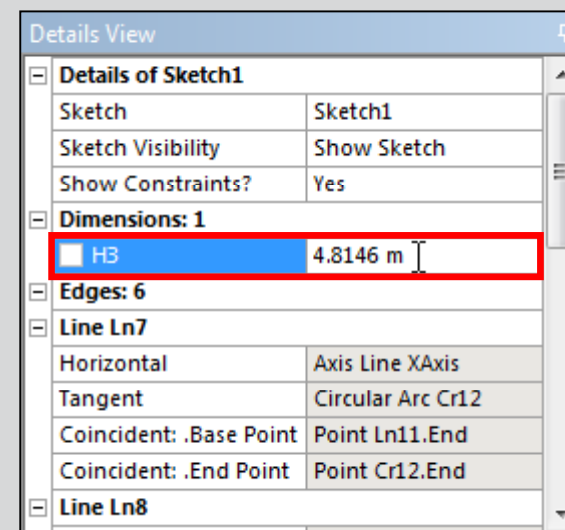


Using the General tool to dimension a horizontal line

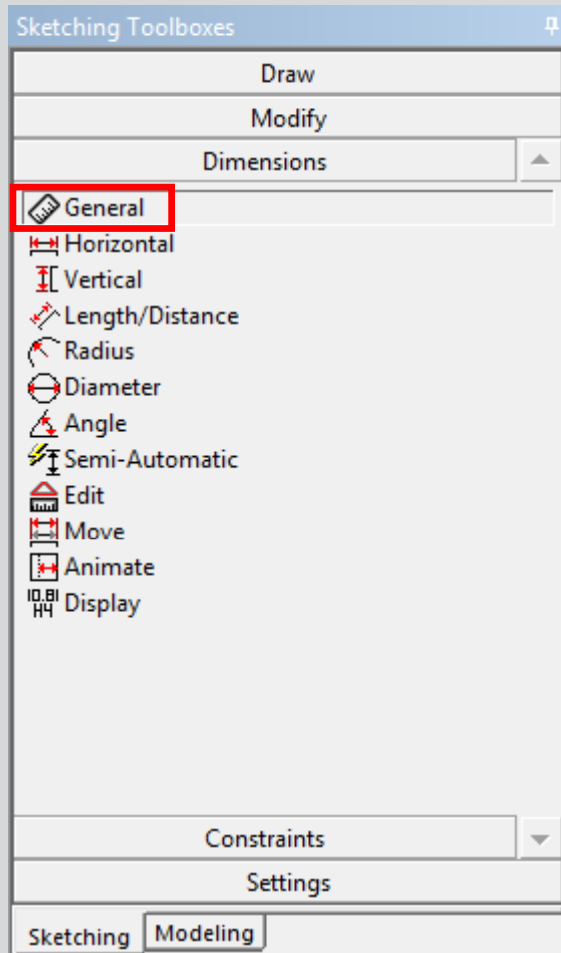
- Click to select entity
- Click to position dimension



- Dimension is automatically numbered
- Dimension value can be edited under the Details View

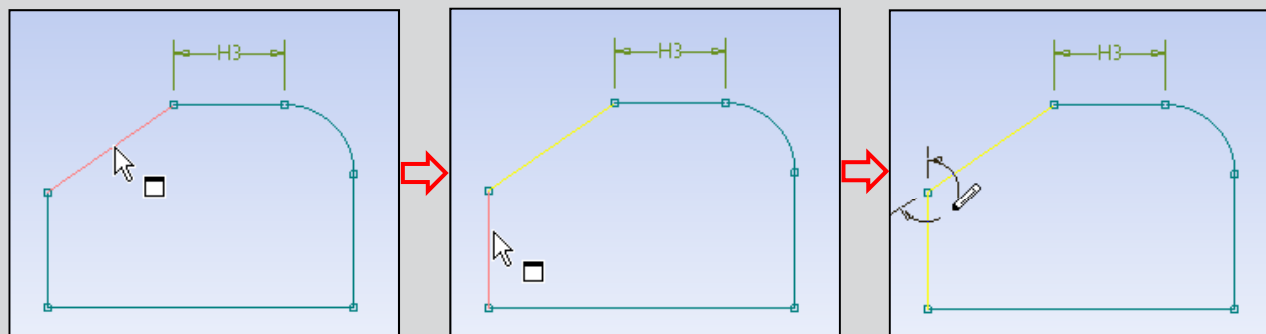
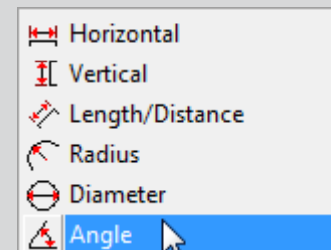


Dimensions Toolbox Example 2

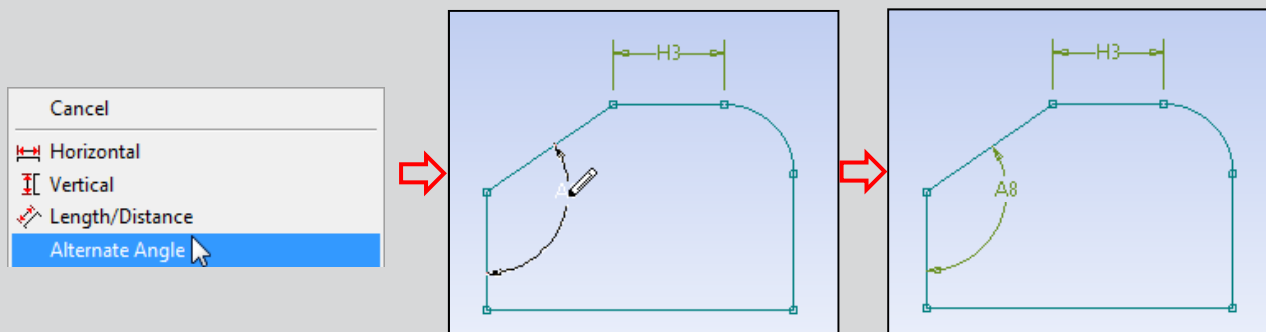


Using the General tool to dimension an angle

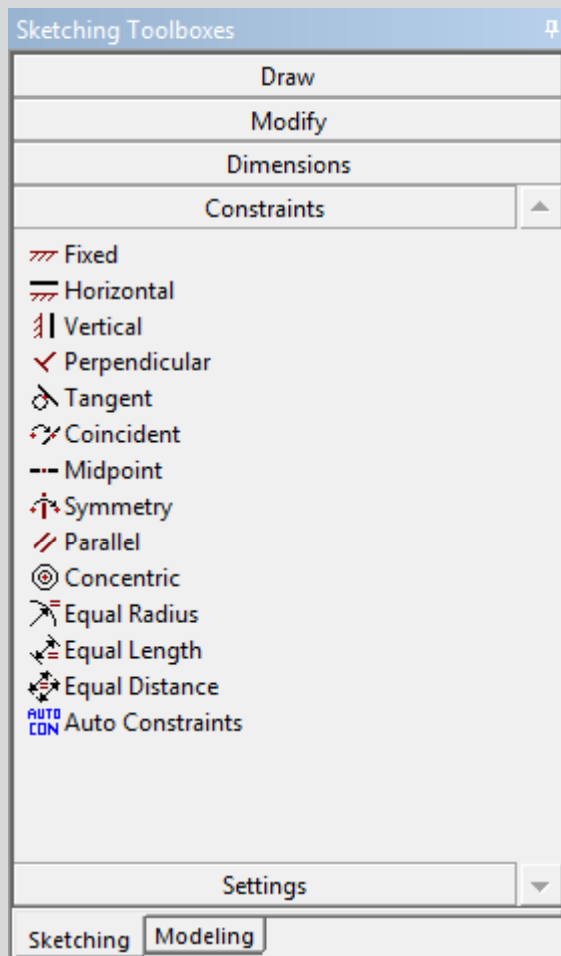
- Right click, specify Angle
- Click to select first entity
- Click to select second entity



- Right click, select Alternate Angle to scroll through options



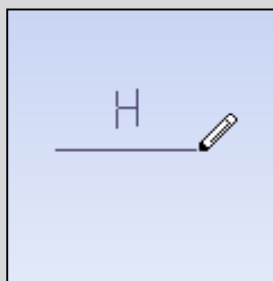
Constraints Toolbox



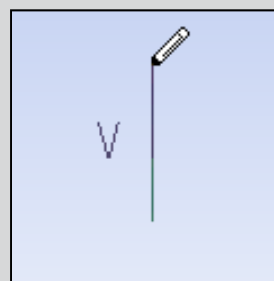
Constraints can be applied on sketch entities

- **Manually** - Pick required constraint, select entity to be constrained
- **Automatically** - Indicates and suggests the constraint and automatically snaps to a location or orientation while you are sketching (automatic on by default).

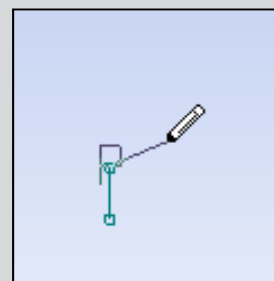
• Four examples



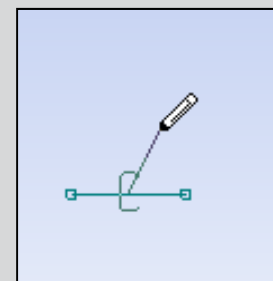
Horizontal



Vertical

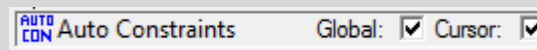


Point



Coincident

- Constraints can be added or removed manually on existing sketch entities
- **Global ON/OFF** applies constraints with respect to all entities in the active plane

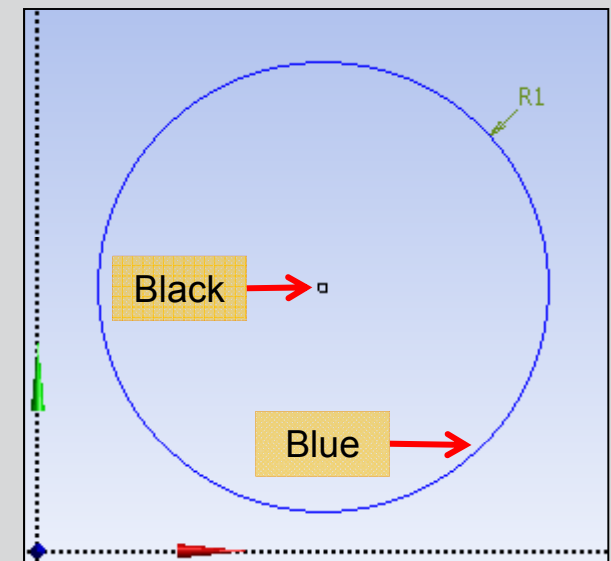
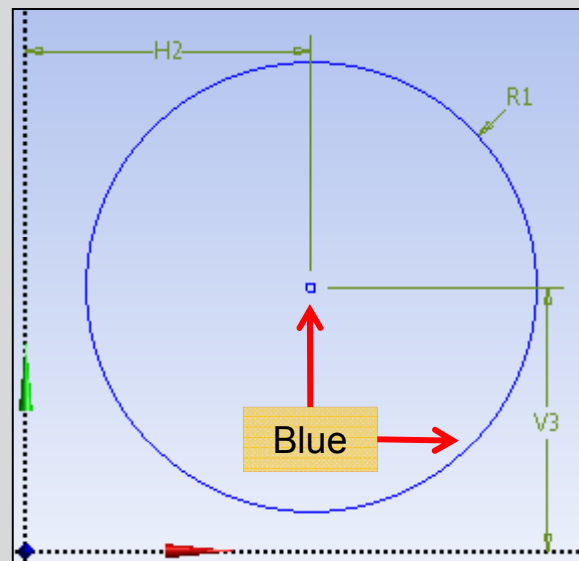
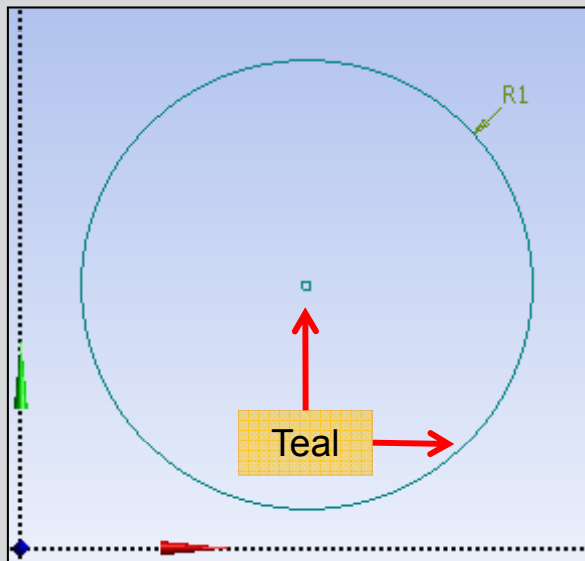


Color Coding:

- Indicates constraint status
 - **Teal:** Under-constrained
 - **Blue:** Well Defined
 - **Black:** Fixed
 - **Red:** Over-Constrained
 - **Gray:** Inconsistent or Unknown

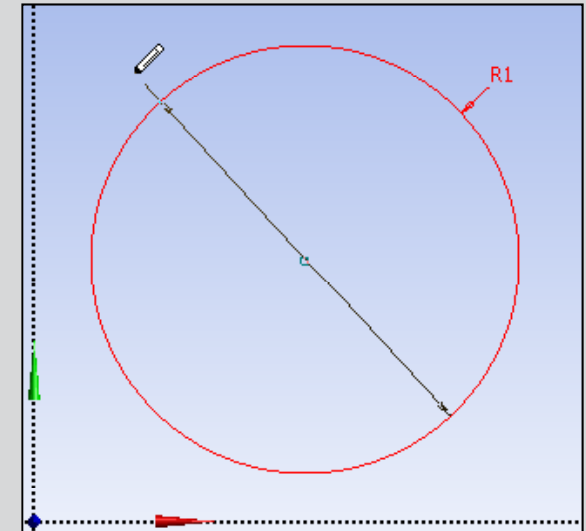
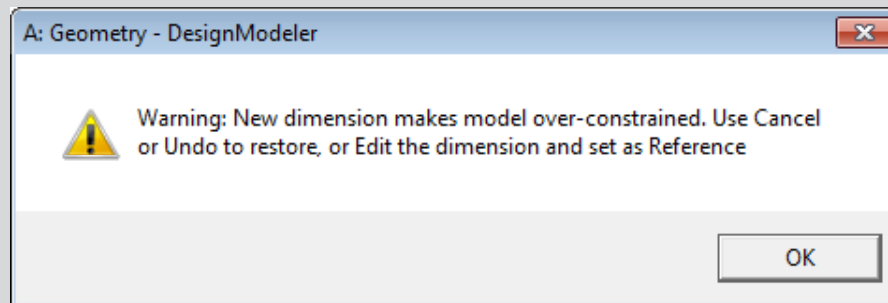
Example:

- Circle with radius dimension
 - Under constrained (centre point is free)
 - Adding 2 more dimensions, fully constrains the sketch, hence all lines turn blue
 - Alternatively, adding a Fixed constraint to the centre point fully constrains the sketch



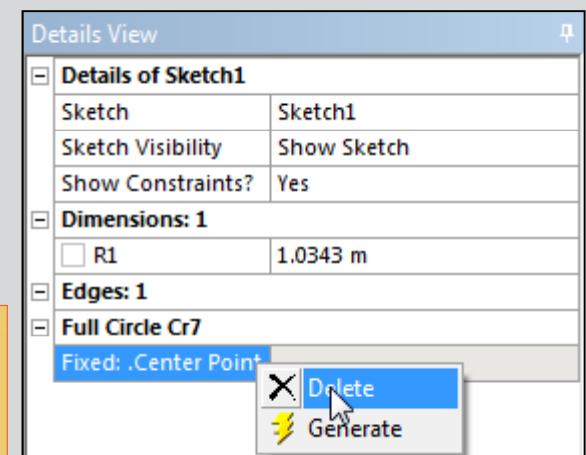
How can sketches become over-constrained?

- Too many dimensions
 - Example – radius & diameter on a circle. Sketch turns red when over-constraining dimension is applied.

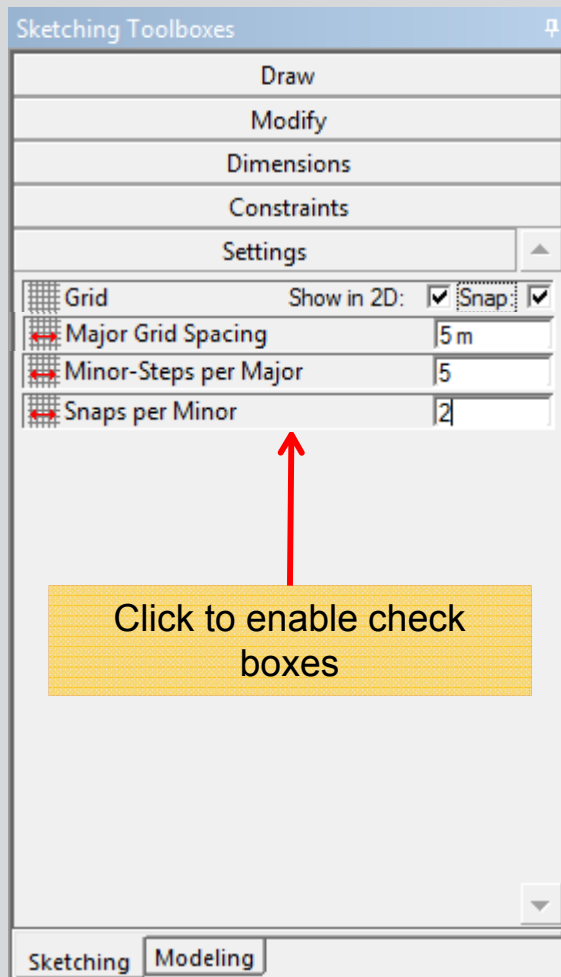


- Viewing & removing constraints
 - Sketch details view, set Show Constraints to Yes
 - Pick entity, constraints are listed. Right click, Delete.

Example - removing
Fixed constraint on
circle centre point



Settings Toolbox



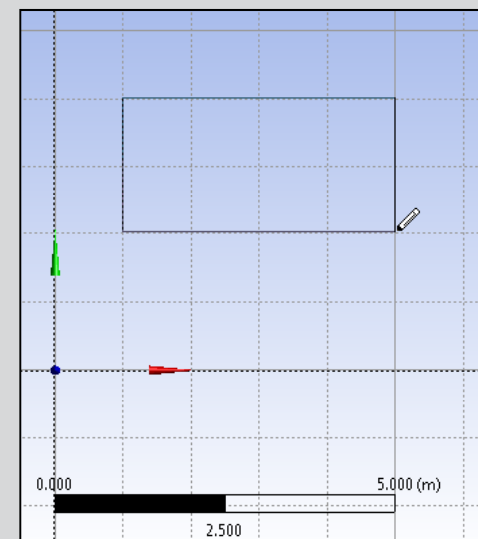
Grid & Snap

- Displays the grid lines in the sketching area, to make sketching easier
- Snap option can be used to snap geometry to grid

Snaps per Minor

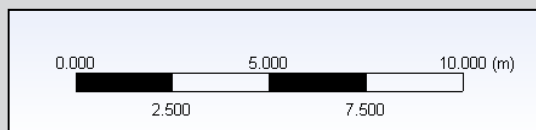
- Allows snapping to points between minor grid lines

The rectangle is snapped halfway between two minor grid lines if Snaps per Minor is set to 2



Sketching Tips

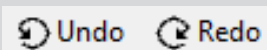
- Ruler allows to get a quick sense of scale while sketching



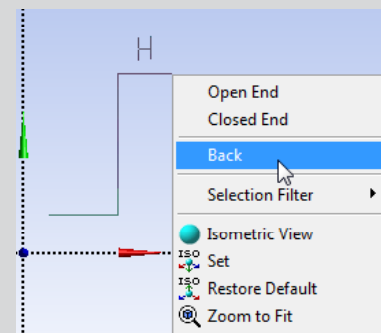
- “Look At” tool will help orient the display such that plane, sketch or selected entity normal to your view



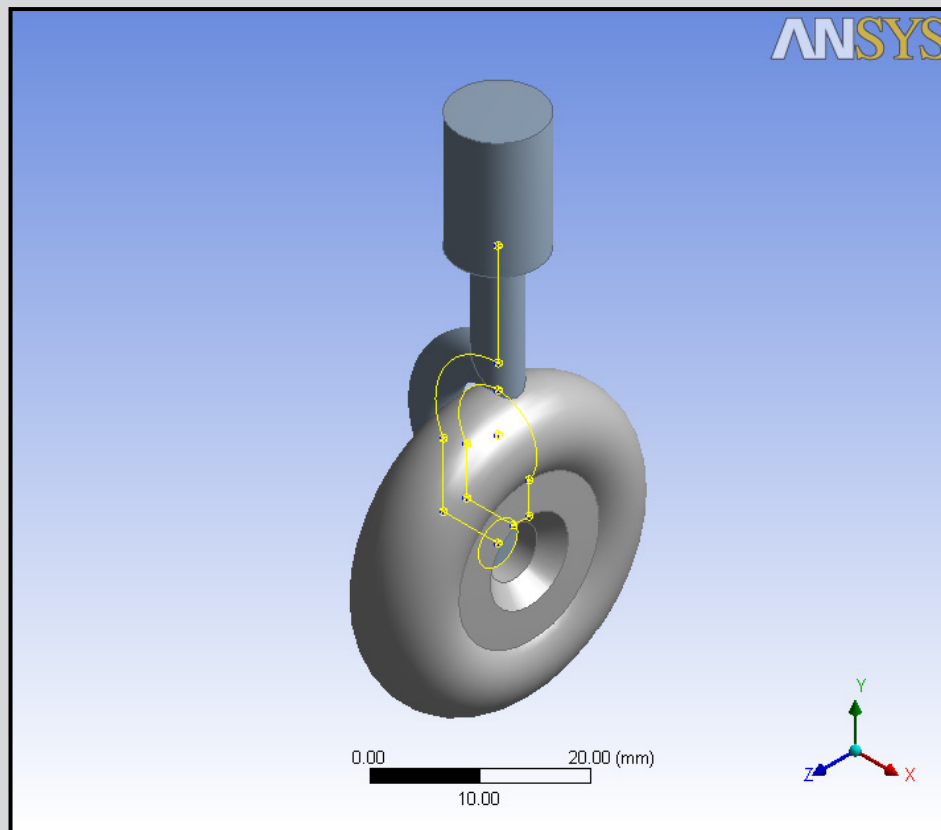
- Many operations become easier via a RMB click context menu on the graphics screen
- Undo/Redo buttons are available in sketch mode only



- Multiple undo's allowed
- IMPORTANT: Each plane stores its own Undo “stack”
- The “Back” operation (available via RMB) acts like a micro undo during multi-step sketching operations (eg polyline)
- Note: Only one sketch is active at a time!



Workshop 2 – Sketching

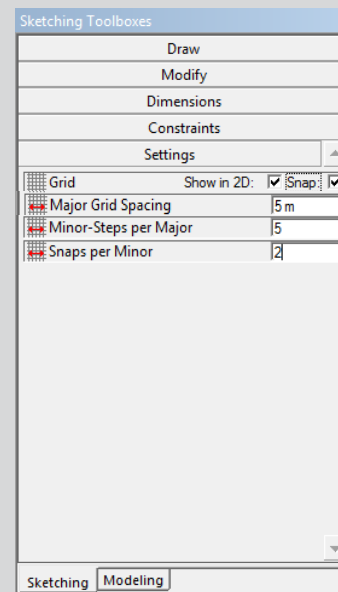
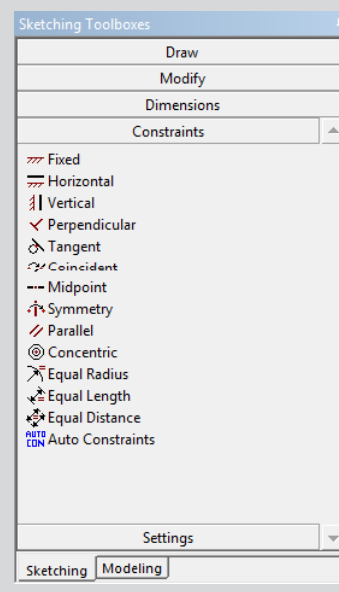
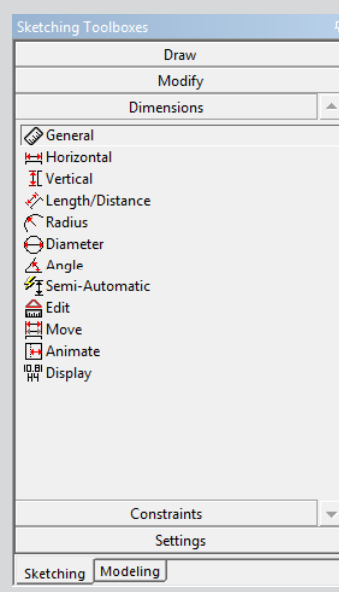
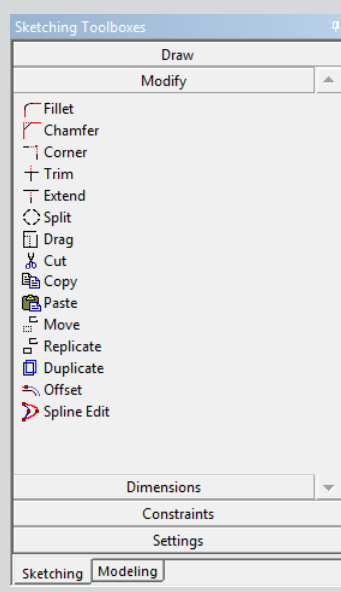
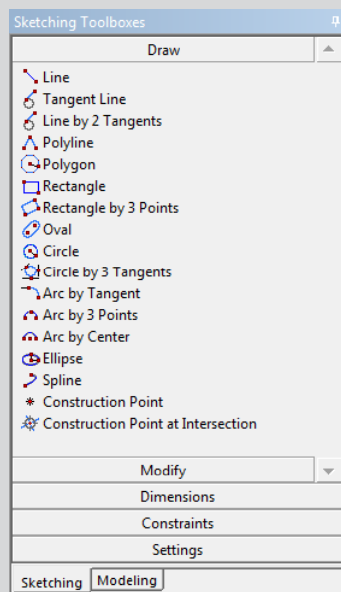


Contents

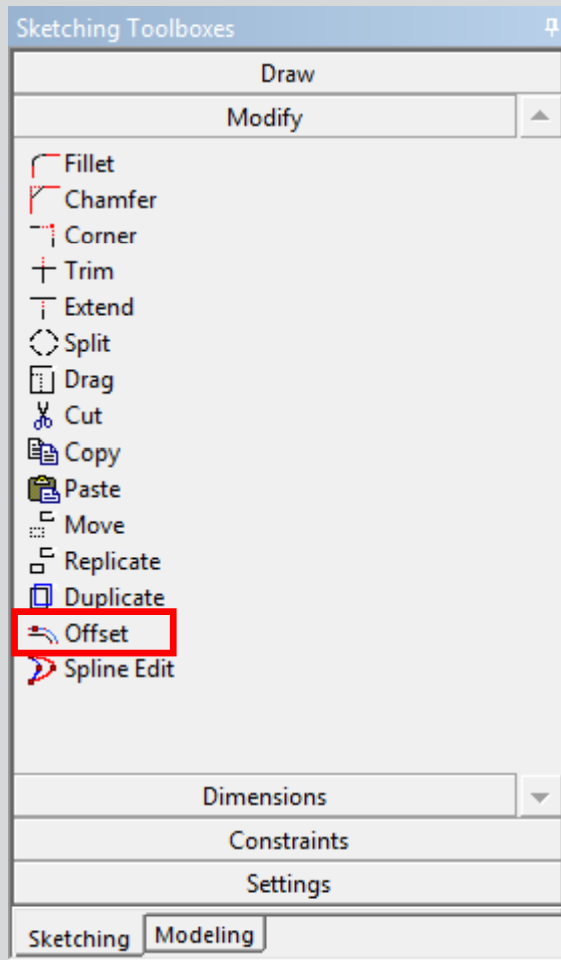
- List of Sketching Tools
- Modify Toolbox: Offset
- Example: Creating Concentric Cylinder
- Modify Toolbox: Cut/Copy
- Modify Toolbox: Paste
- Modify Toolbox: Replicate, Move
- Dimensions Toolbox: Move, Animate, display
- Sketch Instancing
- Sketch Projection

Sketching Tools

- All the sketching tools are segregated into five categories

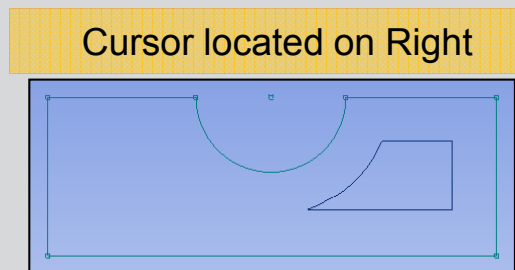
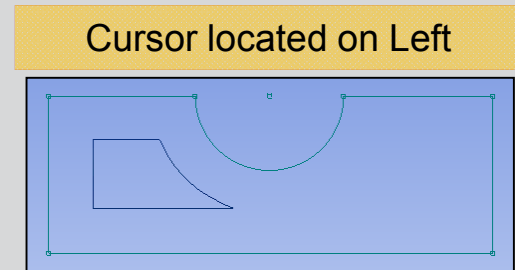
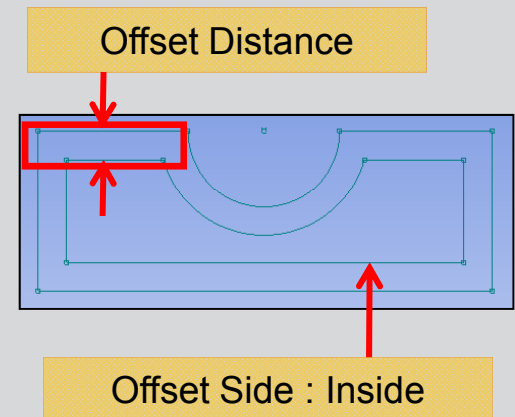


Modify Toolbox: Offset



Offset

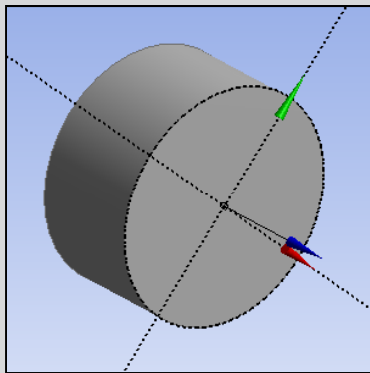
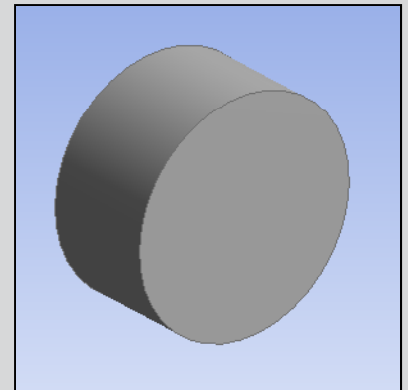
- A sketch can be offset
- Selection options
- End selection
- Place offset
- Three options to perform “offset”
 - Distance
 - Offset side
 - Offset area



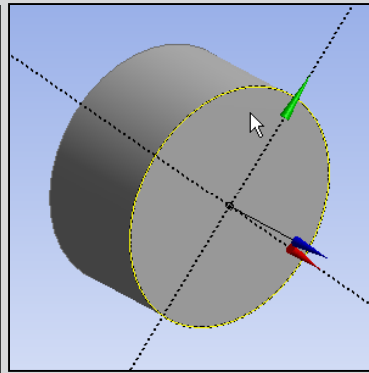
Offset Area : Depends on location of cursor

Example: Creating Concentric Cylinder

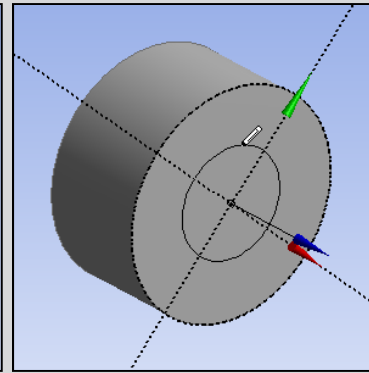
- Select circular face of cylinder and create Outline plane
- Go to sketching mode and select outline of plane i.e. circular edge
- Select Offset option in Modify toolbox and create an offset circle
- Go to modeling mode and select Extrude feature. Select the sketch containing circle as base object for extrude.
- Generate extrude feature to create concentric cylinders as shown



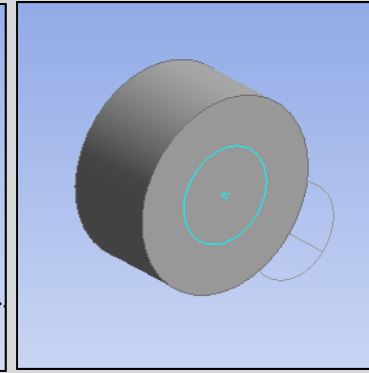
Step 1



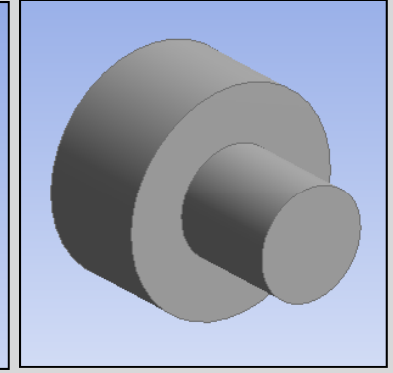
Step 2



Step 3



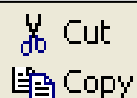
Step 4



Step 5

Modify Toolbox: Cut/Copy

Cut / Copy



- Used to cut or copy selected entities and paste to required location
- Requires the selection of a paste handle (RMB) relative to which the Paste will be performed
- The paste handle is the location to which the cursor is attached while moving the image to new position to paste
- RMB paste handle options:
 - Clear Selection
 - Use Plane Origin as Paste Handle: The 0.0, 0.0 location of the plane will be used as the paste handle
- If Cut or Copy is exited without selecting a paste handle, a default will be used

Copy: RMB options

Clear Selection
End / Set Paste Handle
End / Use Plane Origin as Handle
End / Use Default Paste Handle

Modify Toolbox: Paste

Paste  Paste

- Used to paste entities selected during cut or copy operation
- Entities can be pasted into current or a new sketch, even if it is on different plane
- After copying entities, Paste operation can be performed multiple times
- RMB Paste options:
- Rotate by +/- r Degrees
- Flip Horizontally / Vertically
- Scale by Factor f or 1/f
- Paste at Plane Origin
- Change Paste Handle

Rotate by r Degrees
Rotate by -r Degrees
Flip Horizontal
Flip Vertical
Scale by factor f
Scale by factor 1/f
Paste at Plane Origin
Change Paste Handle

Paste: RMB options

Modify Toolbox: Replicate, Move

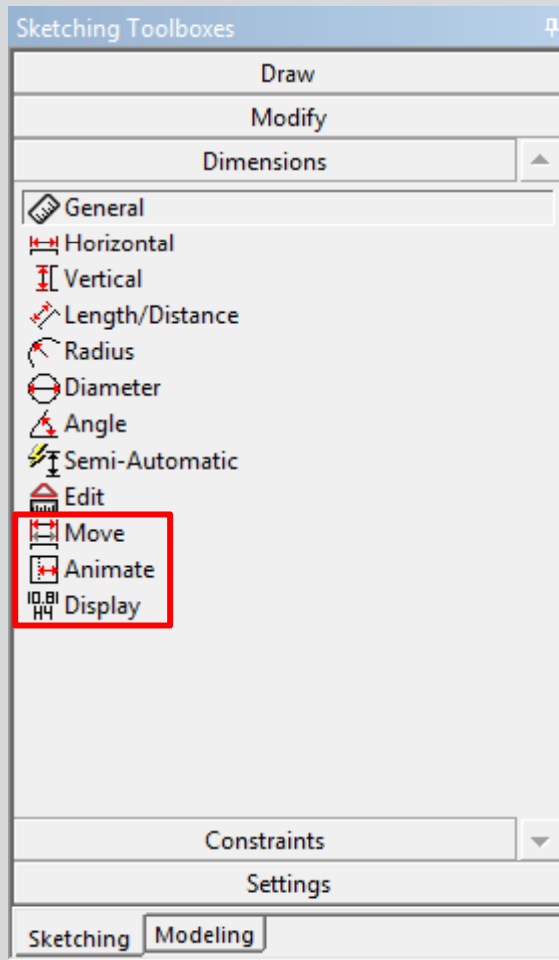
Replicate  Replicate r = 90° f = 2

- The Replicate operation is equivalent to the Copy operation, followed by a Paste
- After selection of entities, the RMB options change to the Paste RMB options
- Note: Replicate option is used to copy sketches. Duplicate option is used instead to copy features of a solid body on to the plane.

Move  Move r = 90° f = 2

- The Move command functions the same as the Replicate operation with the exception that the original selection is moved to a new location instead of being copied

Dimensions Toolbox: Move, Animate, display



- Move function allows placement of dimension to be modified
- Animate function allows to visualize the dynamic changes for a selected dimension within a specified range
- Display function is used to display dimensions in Graphics Window in the form of Name and/or Value
- Dimensions can be modified from Details view or through RMB click

Sketch Instancing

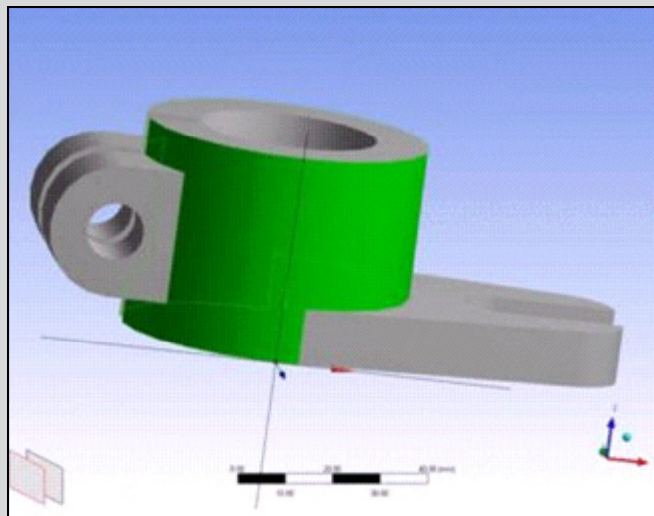
Allows copies of sketches to be added in other planes

- **Features:**
 - Can only be placed in planes lower in the tree outline than the plane with the base sketch
 - Can be scaled and rotated
 - Edges are fixed: cannot be moved, edited, or deleted
 - Sketch is persistent: gets automatically updated when changes in base sketch are made
 - Can be used just like normal sketches for creating other features except:
 - Cannot be used as base sketches for other Instances
 - Not included in the pull-down list of sketches

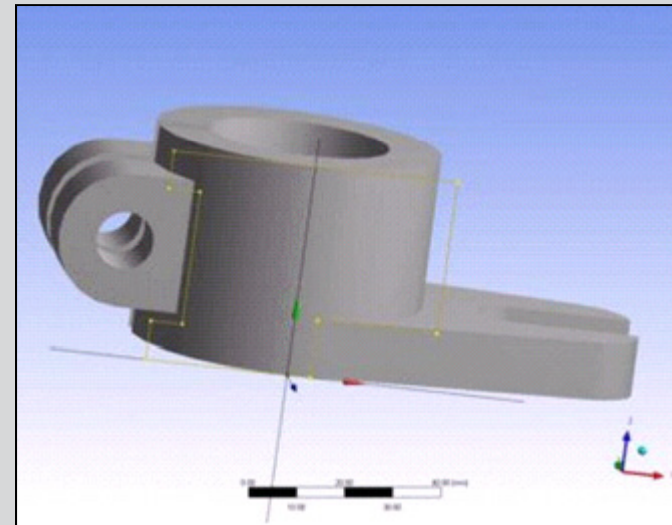
Sketch Projection

Project 3D geometry onto a plane to create new sketch entities

- Select vertices, edges, surfaces or bodies to project
- Cannot be modified using normal sketch tools
- Remain associated to input geometry (updates automatically if 3D geometry changes)



3D Geometry selected for projection



Sketch projected onto existing plane