Lecture 4
Planes and Sketches

Introduction to ANSYS
DesignModeler
Preprocessing Workflow

Geometry Creation OR Geometry Import

- Sketches and Planes
- Direct CAD/Bi-Directional CAD
- 3D Operations
  - Extrude, Revolve, Sweep, etc
- Geometry Import Options

Geometry Operations

- 3D Operations
  - Boolean, Body Operations, Split, etc
- Geometry Cleanup and Repair
  - Automatic Cleanup
  - Merge, Connect, Projection, Flow Volume Extraction, etc

Meshing

- Meshing Methods
  - Hybrid Mesh: Tet, Prisms, Pyramids
  - Hexa Dominant, Sweep meshing
  - Assembly Meshing
- Global Mesh Settings
  - Sizing, Body/Sphere of Influence, Match Control, etc
- Local Mesh Settings
  - Assembly Meshing

Solver

Fluid Flow

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Concept of Planes and Sketches

A plane is required to create a 2D sketch
- Also required for some body operations
- 2D/3D geometry can be constructed using a sketch as a base object

Sequence to create a basic 3D geometry
- Create or select an existing plane
- Create a sketch on the plane
- 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...>
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

New Plane Creation Example: From Plane
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)
Sketch Creation

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.

XYPlane is active

Only sketches on active Plane are displayed in drop down
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...

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

  Follow instructions in the status bar: Define Radius

  Done!

- Another example, 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
Modify Toolbox Examples: Fillet, Chamfer, Corner, Extend, Trim

Fillet( L1, L2)  \[\text{Corner}( L1, L4)\]

Chamfer( L2, L3)  \[\text{Trim}( L3)\]

Extend( L4, L3)
Modify Toolbox Examples: Split

Using Split

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

- 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
A new sketch can be created by referencing existing geometry

- Allows selected boundaries 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
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 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
Constraints Toolbox – cont....

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
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 right 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!
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)
Workshop 2 – Sketching
Appendix

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

- All the sketching tools are segregated into five categories
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: 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
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
Modify Toolbox: 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

Paste: RMB options
Modify Toolbox: Replicate, Move

Replicate

• 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

• 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