Introduction to Mesh Generation with ANSYS Workbench

MECH479

Sean Delfel
2013-11-14
Overview

• Background
• Workbench Overview
• ANSYS DesignModeler (DM) Overview
• ANSYS Meshing (AM) Overview
• Tutorial #1: 2D Cylinder
Background

• What is Workbench?
  – Platform for integration of ANSYS analysis tools.
    • Solid mechanics, fluid dynamics, EM, optimization, etc.
  – Entire project contained in common platform: geometry creation, meshing, analysis, and post-processing.
Background

• A brief history of ANSYS:
  – ANSYS originally a finite element analysis (FEA) code for structural mechanics.
  – Began acquiring other analysis codes ~2000.
  – Most relevant for CFD:
    • CFX purchased in 2003
    • Fluent purchased in 2006
  – With ANSYS 10.0, began combining all acquired preprocessing tools into DM and AM.
Background

• Pros of ANSYS DM/AM:
  – Robust collection of meshing algorithms (AM)
  – Easily integrated with CAD packages for complex geometries (DM)
  – Geometry can be parameterized (DM/AM)

• Cons of ANSYS DM/AM:
  – Blocking of geometry can be difficult (DM)
  – Less control of mesh specifics, structured meshing more difficult (AM)
Workbench Overview

- Project Schematic
- Component Links
- Properties

Component Systems
Workbench Overview

• Project built by dropping component systems into schematic.
• Links connect different components, show flow of project/data.
DesignModeler (DM) Overview

- Geometry creation application for Workbench.
- Solid-based, parameterizable
- Integrates well with CAD platforms
- General geometry creation procedure:

  2D Sketch → 3D Body → Manipulation of Bodies → Combine Bodies in Part
DesignModeler (DM) Overview

- Operations List
- Modeling/Sketching Tabs
- Info. Panel
- Graphics
ANSYS Meshing (AM) Overview

- Meshing generation application for Workbench.
- Incorporates algorithms from multiple legacy packages
- General meshing procedure:

  - Import Geometry
  - Select Method
  - Add Sizing Controls
  - Generate Mesh
  - Refine
ANSYS Meshing (AM) Overview

Operations List

Info. Panel

Graphics
Tutorial #1: Cylinder

• Objectives:
  – Use DM/AM to create geometry for a 2D cylinder and domain.
  – Create and Refine Mesh.
Tutorial #1: Cylinder

• Launch Workbench.
• Ensure that the “Properties” pane is active (View -> Properties)
Tutorial #1: Cylinder

1) Drag “Mesh” component to project schematic
2) Select “Geometry”
3) Toggle Analysis Type to “2D”
4) Launch DM (Double click/right click Geometry)
Tutorial #1: Cylinder

• Create Sketch

1) Select Plane

2) New Sketch Button

3) Sketching Tab

Note: For 2D simulations, all geometry must be created in the XY Plane
Tutorial #1: Cylinder

- Create Sketch (cont.)

4) Under “Draw”, select circle, create in graphics window

5) Repeat with rectangle
Tutorial #1: Cylinder

- Create Sketch (cont.)

6) Select “Dimensions”, add the diameter, horizontal (x2) and vertical (x2) dimensions shown

D = 1 m, H1 = 10 m, H2 = 20 m, V1 = 10 m, V2 = 10 m

Note: Lines will change color as constraints are added. Cyan = under constrained, Dark Blue = Fully Constrained, Red = Over Constrained
Tutorial #1: Cylinder

- Create Surface from Sketch

1) Return to “Modeling”

2) Concept -> Surfaces from Sketches

3) For Base Objects, select “Sketch1”, click apply.

4) Toggle “Add Material” to “Add Frozen”

5) Generate

Note: “Add Material” = new body will unite with existing bodies. “Add Frozen” = No booleans performed
Tutorial #1: Cylinder

- Review Body, set to “Fluid”

1) Expand Part menu
2) Select Surface Body
3) Toggle Fluid/Solid to “Fluid”

Note: For multiple bodies, must be combined into a Part to make up a single domain.
Tutorial #1: Cylinder

- Return to Workbench, launch Meshing

If preference menu pops up, select CFD for physics preference.
Tutorial #1: Cylinder

- Set Mesh Preferences

1) Click on Mesh

2) Ensure Physics Preferences are set to CFD and Fluent
Tutorial #1: Cylinder

- Insert Mesh Method

1) Right click on Mesh
2) Insert -> Method
3) Select the body for Geometry Selection, click Apply
4) Click “Update” or “Generate Mesh”
Tutorial #1: Cylinder

- Try out different Mesh Methods

Quad. Dominant

Triangles

Multizone Quad/Tri
Tutorial #1: Cylinder

- Improve Mesh: Insert Edge Sizing

1) Return Method to Quad Dominant
2) Right click Mesh. Insert -> Sizing
3) Click Edge Selection, Select top and bottom edges (hold control for multiple)
4) Change Type to “Number of Divisions”, Input 50 divisions, Change Behavior to “Hard”
5) Re-Generate Mesh
Tutorial #1: Cylinder

• Improve Mesh: Insert Inflation for B.L.

1) Right click Mesh. Insert -> Inflation
2) Select Face for Geometry
3) Select circle for Boundary
4) Input 10 for Max. Layers
5) Re-Generate Mesh
Tutorial #1: Cylinder

- Create Boundary Conditions w/ Named Selections

1) Select edge, then right click -> Create Named Selection
2) Specify BC names for all edges
3) BC types can be adjusted in Fluent
Tutorial #1: Cylinder

• Export Mesh (if using standalone Fluent)

1) File -> Export
2) Specify Fluent Input File (.msh)
3) Enter filename and save