CAD to Mesh in Fluent

1 Steps

1.1 Set up the working directory

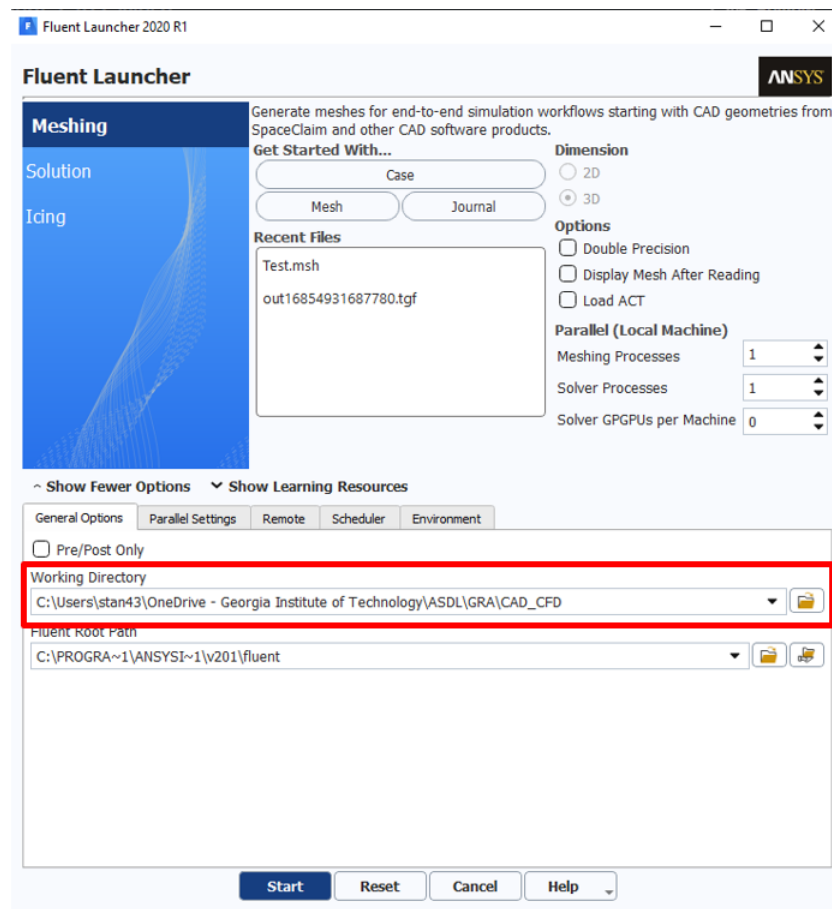Choose the directory including the CAD file, then click **start**.

Figure 1: Step 1: set up the working directory.

1.2 Import CAD file

1. Click **Select Workflow Type**

2. Click **Watertight Geometry**

3. Click **Import Geometry**

4. Click **File Name**. Notice here we can choose the units of the dimensions for the CAD file.
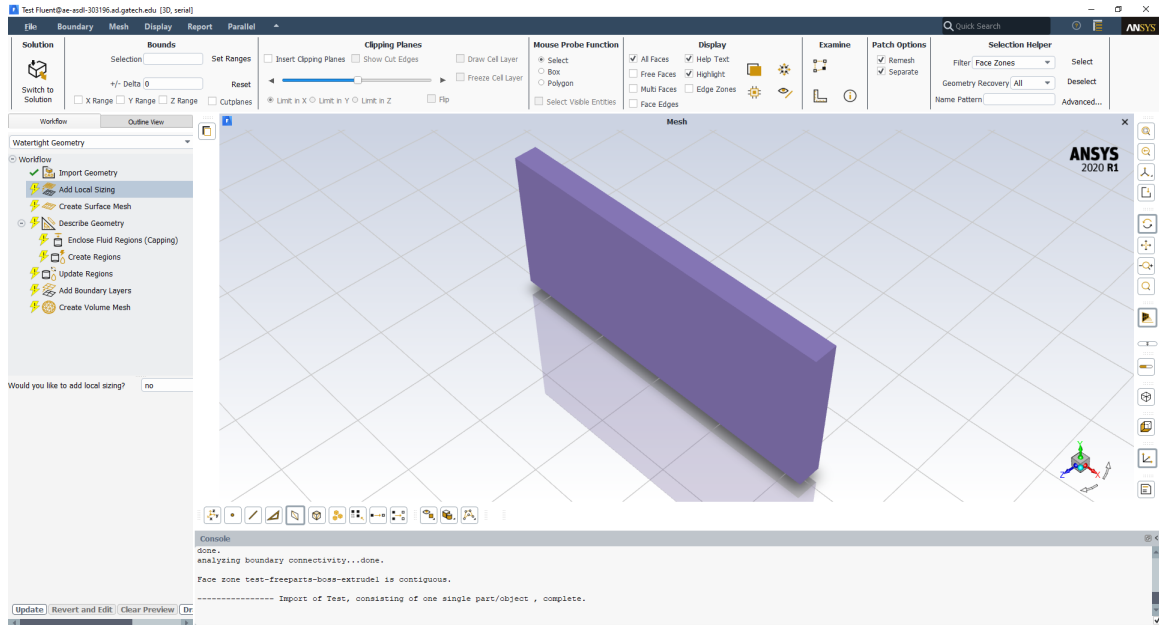
5. Import the CAD file

Figure 2:   Step 2: Import the CAD file.

1.3   Mesh setting

1. Click **Add Local Sizing**. Normally we don't add it. Then click **Update**

2. Click **Create Surface Mesh**. There are many options in this step, we can choose the size and the curvature of the mesh.

3. Click **Describe Geometry**. Here we need to specify whether the mesh is for fluid or solid. This is important.

4. Click **Update Boundaries**. Specify each boundary's type. See the next section.

5. Click **Update Regions**. Specify the region as solid or fluid.

6. Click **Add Boundary Layers**.

7. Click **Create Volume Mesh**.

Figure 3: Step 3: Surface Mesh.

Figure 4: Step 3: Describe Geometry.

1.4 Export the Mesh File

1. Click **File**

2. Click **Write**

3. Click **Mesh**

4. Save it in .msh format

2 Boundary Type

2.1 Velocity-Inlet

Velocity inlet boundary conditions are used to define the flow velocity, along with all relevant scalar properties of the flow, at flow inlets. In this case, the total (or stagnation)

pressure is not fixed but will rise (in response to the computed static pressure) to whatever value is necessary to provide the prescribed velocity distribution.

2.2 Pressure-Outlet

Pressure outlet boundary conditions require the specification of a static (gauge) pressure at the outlet boundary. The value of the specified static pressure is used only while the flow is subsonic. Should the flow become locally supersonic, the specified pressure will no longer be used; pressure will be extrapolated from the flow in the interior. All other flow quantities are extrapolated from the interior.

2.3 Pressure-Inlet

Pressure inlet boundary conditions are used to define the fluid pressure at flow inlets, along with all other scalar properties of the flow. They are suitable for both incompressible and compressible flow calculations. Pressure inlet boundary conditions can be used when the inlet pressure is known but the flow rate and/or velocity is not known. This situation may arise in many practical situations, including buoyancy-driven flows . Pressure inlet boundary conditions can also be used to define a "free" boundary in an external or unconfined flow.

2.4 Pressure-Far-Field

Pressure far-field conditions are used in ANSYS FLUENT to model a free-stream condition at infinity, with free-stream Mach number and static conditions being specified. The pressure far-field boundary condition is often called a characteristic boundary condition, since it uses characteristic information (Riemann invariants) to determine the flow variables at the boundaries.

2.5 Mass-Flow-Inlet

Mass flow boundary conditions can be used in ANSYS FLUENT to provide a prescribed mass flow rate or mass flux distribution at an inlet. As with a velocity inlet, specifying the mass flux permits the total pressure to vary in response to the interior solution. This is in contrast to the pressure inlet boundary condition, where the total pressure is fixed while the mass flux varies. However, unlike a velocity inlet, the mass flow inlet is equally applicable to incompressible and compressible flows.

2.6 Outflow

Outflow boundary conditions in ANSYS FLUENT are used to model flow exits where the details of the flow velocity and pressure are not known prior to solving the flow problem. You do not define any conditions at outflow boundaries (unless you are modeling radiative heat transfer, a discrete phase of particles, or split mass flow): ANSYS FLUENT extrapolates the required information from the interior. It is important, however, to understand the limitations of this boundary type.

2.7   Symmetry

Symmetry boundary conditions are used when the physical geometry of interest, and the expected pattern of the flow/thermal solution, have mirror symmetry. They can also be used to model zero-shear slip walls in viscous flows. This section describes the treatment of the flow at symmetry planes and provides examples of the use of symmetry. You do not define any boundary conditions at symmetry boundaries, but you must take care to correctly define your symmetry boundary locations.

2.8   Wall

Wall boundary conditions are used to bound fluid and solid regions. In viscous flows, the no-slip boundary condition is enforced at walls by default, but you can specify a tangential velocity component in terms of the translational or rotational motion of the wall boundary, or model a "slip" wall by specifying shear.

2.9   Internal

Mesh cells are attached to both sides of the boundary faces.