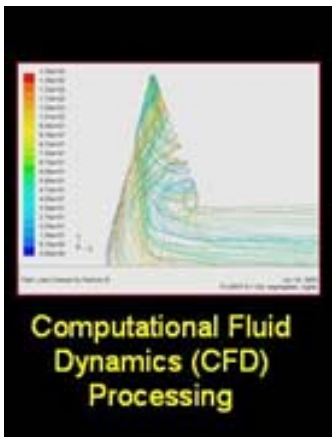
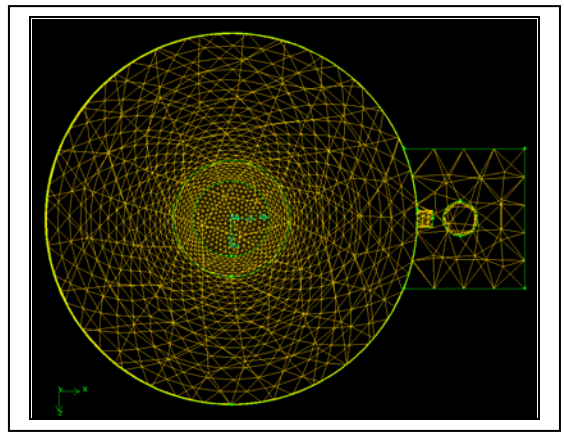
Computational Fluid Dynamics (CFD)

State-of-the-art computational fluid dynamics (CFD) software used for research provides complete mesh capabilities, solving flow problems with supported mesh types including 2-dimensional triangular/quadrilateral, 3-dimensional tetrahedral/hexahedral/pyramid/ wedge, and mixed, or hybrid, meshes. Meshes are computer representations of 2D area and 3D





volumes to which calculations are applied. Using solution-adaptive mesh capability, flow fields can even be modelled in regions with sizeable gradients, such as free shear layers and boundary layers.

The equations used in CFD that govern fluid flow are referred to as the Navier-Stokes equations. Navier-Stokes Equations for an incompressible fluid are given by:

$$\rho\left(\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)=-\frac{\partial P}{\partial x}+\eta\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right)+F_x$$

$$\rho\left(\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)=-\frac{\partial P}{\partial y}+\eta\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right)+F_y$$

$$\rho\left(\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)=-\frac{\partial P}{\partial z}+\eta\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right)+F_z$$

Where *P* is the pressure, η is the viscosity, and F is the force. In many applications, the Navier-Stokes equations can be reduced, based on assumptions, to Bernoulli's equation.