ENE 505 – Applied Computational Fluid Dynamics in Renewable Energy Technologies

WEEK 9: CFD PROBLEM DESCRIPTION

DESCRIPTION:

- **Flow Problem**

➢ The present computational study is conducted using a moving mesh technique which is incorporated with a FVM based fluid flow solutions through the commercial software package FLUENT [1] (ANSYS Inc., Canonsburg, Pennsylvania).

➢ The computational domain is comprised of two zones namely; rotating zone in the wake region and stationary fluid zone outside.

➢ The unsteady 2-D and 3-D governing flow equations i.e. continuity equation and the Navier-Stokes equations are solved together with the transport equations for the turbulence kinetic energy and its dissipation rate to fully resolve the turbulence flow characteristics.

- **Governing Equations**

➢ The fluid flow is assumed to be incompressible, Newtonian and non-isothermal. The flow equations are conservation equations for mass and momentum. The mass or continuity equation in the general form is described below:
It can be examined through three ways:

$$\frac{\partial \rho}{\partial t} + \nabla\left(\rho \vec{v}\right) = 0$$

- Where is density, ρ is velocity vector field, *t* is time, and the equation is applicable for compressible and incompressible flows. The momentum equation for an inertial reference frame in conservation form and neglecting the curvature and surface tension is described by:

$$\frac{\partial}{\partial t}\left(\rho\vec{v}\right)+\nabla\left(\rho\vec{v}\vec{v}\right)=-\nabla p+\nabla\left(\bar{\tau}\right)+\rho\vec{g}+\vec{F}$$

- Where, *p* is the static pressure, $\bar{\tau}$ is the stress tensor (as described below), $\rho\vec{g}$ and $\vec{F}$ are the gravitational body forces and external body forces respectively. The stress tensor is expressed as:

$$\bar{\tau}=\mu\left[\left(\nabla\vec{v}+\nabla\vec{v}^{T}\right)-\tfrac{2}{3}\nabla\vec{v}I\right]$$

- Where, μ is the molecular viscosity, and *I* is the unit tensor.

- In addition to conservation equations for mass and momentum, sliding mesh technique is incorporated with the present FVM based fluid flow solver [1] to better visualize and capture the unsteady flow characteristics in a rotating frame on the blades surfaces.

- The equations of motion are modified to include moving motion of the blades and to incorporate the acceleration terms which occur due to the transformation from the stationary to the moving mesh.

- The sliding mesh technique is a special case of general dynamic mesh motion where the nodes move rigidly in a given dynamic mesh motion (ANSYS 2009).

- The grid is divided into two main regions; one attached to a rotating geometry and the other attached to the stationary boundaries of the flow, which slides relative to one another along a slip plane [2]

- In this case all the boundaries in a mesh zone and nodes move together in a rigid fashion without any deformation of the cells.

- The general conservation equation formulation for a general scalar, Φ, for dynamic meshes is applied in sliding mesh and described as:

$$\frac{d}{dt}\int_{v}\rho\Phi dV+\int_{\partial V}\rho\Phi\left(\vec{u}-\vec{u}_{g}\right)d\vec{A}=\int_{\partial V}\Gamma\nabla\varphi d\vec{A}+\int_{V}S\varphi dV$$

Where ρ is the fluid density, $\vec{u}$ is the flow velocity vector, $\vec{u}_g$ is the mesh velocity of the moving mesh, Γ is the diffusion coefficient, S_Φ is the source term of ϕ and ∂V is used to represent the boundary of the control volume, *V*.

References:

1. Ansys, 2009. Fluent Theory Guide, V.l3., Ansys Inc..

2. Bakker, A., LaRoche, R. D., Wang, M.-H. and Calabrese, R. V. (2000), “Sliding mesh simulation of laminar flow in stirred reactors”, The Online CFM Book.