CHAPTER 6

COMPUTATIONAL FLUID DYNAMICS

6.1 INTRODUCTION

Computational fluid dynamics (CFD) is the simulation of fluids engineering systems using modeling (mathematical physical problem formulation) and numerical methods discretization methods, solvers, numerical parameters, grid generations and so on). A commercial Computational Fluid Dynamics (CFD) software is used in the design procedure. The main aim of this study by means of applying numerical experimentation to the designed pump is CFD code integration into the design procedure and verification of the design before the pump is produced.

It is but conventional to use the CFD code in plotting the pump characteristics curves, which includes plotting head against flow rate and efficiency against flow rate.(Ozgen 2006). In fact, studies relating to the internal flow structure are also carried out to improve the design and achieve better performance characteristics of the designed pump. By exploring the capabilities of CFD code, the flow in the impeller and bowl is investigated using features such as the path line traces, velocity vector representations as well as physical quantities like pressure and velocity distribution in the pump. But, a condition that permits the integration of the CFD software into the design procedure is that the verification of the code should be done by applying CFD to the previously designed pumps. The CFD results are compared with the calibrated and certified test stand of the company (Ozgen,
Adopted from the work of Ozgen (2006), the conventional design and CFD integrated design procedures are revealed in Figure 6.1.

![Figure 6.1 Conventional and CFD integrated design procedures (Ozgen 2006)](image)

6.2 IMPORTANCE OF COMPUTATIONAL FLUID DYNAMICS

Basically, three methods exist for the study of fluids: (1) theoretical analysis (Daily and Harleman 1966), (2) experiments and (3) simulation (CFD). These are listed in terms of some common criteria, as stated by Zuo (2005): in Table 6.1.
Table 6.1 Comparison of simulation and experiment (Zuo 2005)

<table>
<thead>
<tr>
<th></th>
<th>Simulation (CFD)</th>
<th>Experiment</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cost</td>
<td>Cheap</td>
<td>Expensive</td>
</tr>
<tr>
<td>Time</td>
<td>Short</td>
<td>Long</td>
</tr>
<tr>
<td>Scale</td>
<td>Any</td>
<td>Small</td>
</tr>
<tr>
<td>Information</td>
<td>All</td>
<td>Measured</td>
</tr>
<tr>
<td>Repeatable</td>
<td>Yes</td>
<td>Some</td>
</tr>
<tr>
<td>Safety</td>
<td>Yes</td>
<td>Some dangerous</td>
</tr>
</tbody>
</table>

6.3 METHODOLOGY

In all of these approaches the same basic procedure is followed.

1. During preprocessing
   - The geometry of the problem is defined.
   - The volume occupied by the fluid is divided into discrete cells (the mesh). The mesh may be uniform or non-uniform.
   - Boundary conditions are defined. This involves specifying the fluid behaviour and properties at the boundaries of the problem. For transient problems, the initial conditions are also defined.

2. The simulation is started and the equations are solved iteratively as a steady-state or transient.

3. Finally, a postprocessor is used for the analysis and visualization of the resulting solution.
6.4 NUMERICAL MODELING

6.4.1 Governing Equations

The basic governing equations for fluid flow are the conservation of mass and momentum. These equations describe the behavior of fluids. The flow behavior is assumed to be incompressible and turbulent in nature. The rotating impeller is solved using moving reference frame with constant rotational speed. Whereas, the stationary parts like bowl is solved using inertial frame of reference.

For three-dimensional incompressible and steady flow, the continuity equation and momentum equation:

Continuity Equation

\[ \frac{\hat{c}u}{\hat{c}x} + \frac{\hat{c}v}{\hat{c}y} + \frac{\hat{c}w}{\hat{c}z} = 0 \] (6.1)

For Stationary part:

X-momentum:

\[ \rho \left[ u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} \right] = -\frac{\partial p}{\partial x} + \mu \left[ \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right] \] (6.2)

Y-momentum:

\[ \rho \left[ u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} \right] = -\frac{\partial p}{\partial y} + \mu \left[ \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right] \] (6.3)
Z-momentum:
\[
\rho \left[ u \frac{\partial w}{\partial x} + v \frac{\partial w}{\partial y} + w \frac{\partial w}{\partial z} \right] = - \frac{\partial p}{\partial z} + \mu \left[ \frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right]
\] (6.4)

For Moving part:

X-momentum:
\[
\rho \left[ u_r \frac{\partial u_r}{\partial x} + v_r \frac{\partial u_r}{\partial y} + w_r \frac{\partial u_r}{\partial z} \right] = - \frac{\partial p}{\partial x} + \mu \left[ \frac{\partial^2 u_r}{\partial x^2} + \frac{\partial^2 u_r}{\partial y^2} + \frac{\partial^2 u_r}{\partial z^2} \right]
\] (6.5)

Y-momentum:
\[
\rho \left[ u_r \frac{\partial v_r}{\partial x} + v_r \frac{\partial v_r}{\partial y} + w_r \frac{\partial v_r}{\partial z} \right] = - \frac{\partial p}{\partial y} + \mu \left[ \frac{\partial^2 v_r}{\partial x^2} + \frac{\partial^2 v_r}{\partial y^2} + \frac{\partial^2 v_r}{\partial z^2} \right]
\] (6.6)

Z-momentum:
\[
\rho \left[ u_r \frac{\partial w_r}{\partial x} + v_r \frac{\partial w_r}{\partial y} + w_r \frac{\partial w_r}{\partial z} \right] = - \frac{\partial p}{\partial z} + \mu \left[ \frac{\partial^2 w_r}{\partial x^2} + \frac{\partial^2 w_r}{\partial y^2} + \frac{\partial^2 w_r}{\partial z^2} \right]
\] (6.7)

6.5 MESH GENERATION

Fluid structure is formed through the placement of the extensions to the inlet and outlet of the pump and it is divided into finite elements (Ozgen 2006). Before running a CFD design analysis, there is a need to breakdown the geometry into elements, which are small and manageable pieces. Calculations are performed at each mode. Let it be noted that the corner of each element is referred to as a node (CFdesign 2004; Ozgen 2006). For reliable solutions in CFD utilization, it is required to increase the number of elements to as many as possible. However, let it be noted that in increasing the number of elements the tune to obtain solution will be longer. On the other hand, in using a 64 bit computer, the user may not be allowed to
use more than 8GB of memory in the computer hardware. The mesh independence cannot be studied in the turbo machinery CFD analyses (Hirsch 1994). The strategy for meshing is to use smaller elements for the impeller and the surrounding rotating regions. The elemental size is then increased for the other parts. Meshing in the impeller is commenced by attaching a minimum thickness to the elemental size closer to the impeller blade. The bowl is also meshed by considering an elemental size of the minimum bowl rare thickness (CFdesign 2004; Ozgen 2006).

6.6 BOUNDARY CONDITION

In solving the equation system, there is also a need to establish the boundary conditions. Typically, boundary conditions are of four classes in CFD: no-slip, axisymmetric, Inlet-outlet and periodic. An example will demonstrate some of these boundary conditions as illustrated by Zuo (2005) in Figure 6.2.

![Figure 6.2 Boundary conditions of pipe flow (Zuo 2005)](image)

In this example, a pipe is shown with fluid flowing in it from left to right. The left part represents the input, while the right part shows the output of the system. Since the inlet is at the left, the velocity can be set manually. However, on the right hand side (outlet), the outlet boundary condition may be used to keep all the properties constant. The implication is that all the
gradients are zero. It is possible to set the velocity to zero at the wall of the pipe, and this is known as the no-slip boundary condition. But at the center of the pipe, axisymmetric boundary conditions may be obtained in Fox (1985), Shames (1992) and White (1999).

6.7 MOVING REFERENCE FRAME (MRF)

The principal reason for employing a moving reference frame is to render a problem which is unsteady in the stationary frame steady with respect to the moving frame. It should also be noted that one can run an unsteady simulation in a moving reference frame with constant rotational speed. A mixed-flow submersible pump stage has both rotating and stationary components coupled together in a CFD model by one or two interface models. The rotating component, such as the impeller, is modeled after a rotating frame of reference, while the stationary components are assigned to a stationary frame of reference.

In ANSYS Fluent, the multiple reference frame (MRF) option was developed as an alternative to the sliding mesh approach for modeling flow field in geometry where there are parts that rotate relative to each other. Fluid motion in a rotating sub domain is solved in a rotating frame, and the solution is matched at the interface between the rotating and stationary regions via velocity transformations from one frame to the other. The “velocity matching” step involves the assumption of steady flow conditions at the interface, but permits multiple fluid (not grid) regions to rotate relative to each other.