ANALYSIS OF FLOW IN MOVING BLADE SYSTEM USING COMPUTATIONAL FLUID DYNAMICS
## CHAPTER-6

**ANALYSIS OF FLOW IN MOVING BLADE SYSTEM USING COMPUTATIONAL FLUID DYNAMICS**

<table>
<thead>
<tr>
<th>S.No.</th>
<th>Name of the Sub Title</th>
<th>Page No.</th>
</tr>
</thead>
<tbody>
<tr>
<td>6.1</td>
<td>Computer aided simulations</td>
<td>135</td>
</tr>
<tr>
<td>6.2</td>
<td>strategy of CFD</td>
<td>136</td>
</tr>
<tr>
<td>6.3</td>
<td>Solving the problem using CFD</td>
<td>127</td>
</tr>
<tr>
<td>6.4</td>
<td>Compressible flow and incompressible flow</td>
<td>138</td>
</tr>
<tr>
<td></td>
<td>6.4.1 Mesh</td>
<td>139</td>
</tr>
<tr>
<td></td>
<td>6.4.2 Boundary conditions in the analysis</td>
<td>140</td>
</tr>
<tr>
<td></td>
<td>6.4.3 Analysis of single moving blade System</td>
<td>142</td>
</tr>
<tr>
<td></td>
<td>6.4.4 Simulation output Parameters for multi blade system</td>
<td>145</td>
</tr>
<tr>
<td></td>
<td>6.4.5 Simulation Input Parameters</td>
<td>145</td>
</tr>
</tbody>
</table>
CHAPTER 6

ANALYSIS OF FLOW IN MOVING BLADE SYSTEM USING COMPUTATIONAL FLUID DYNAMICS

6.0 INTRODUCTION

The computer aided design (CAD) is software that is primarily used on the engineering, design. The main function of this tool is to aid the design and elaboration of technical drawings of products. This software is capable of modeling 2D and 3D drawings, surfaces and solids that virtually represents products or parts. The 2D model creation is commonly driven by basic shapes as lines and arcs that are placed together in a plane. On the other hand to create 3D solids, different kind of features like extrusions, revolves or sweeps are applied to the mentioned basic shapes. Then those solids are detailed with features like rounds, chamfers, Boolean operations, among others [8].

6.1 COMPUTER AIDED SIMULATION

This technology is another of the computer aided field. It is a tool that helps the engineering teams to simulate, analyze, and manufacture products virtually. This can save money and time to the development of new products because it decrease errors and helps to design robust products and processes.

The C.A.E. computer-aided engineering includes areas like F.E.A., C.F.D. and Multibody methods. To use CAE software it is needed to follow three steps, the first is the pre-processing, the second
is the analysis solver and the third is the post-processing: On the pre-processing stage the virtual model is defined and discretized dividing it in small Sections or elements. The group of elements is defined as a mesh. Then the initial and boundary conditions are set. These conditions are the forces applied to the model, material properties, and movement of parts, among others. Finally the solver is defined. This involves the selection of equations of motion, turbulence, and species, among others.

The second stage involves the analysis of the model defined on the first stage. This stage can be highly demanding with the workstation. Some analyses are restricted to very powerful computers. On the third stage the results are analyzed using visualization and plotting tools. These tools can show the loads, power, energy, temperature or whatever the engineer is analyzing:

6.2 STRATEGY OF CFD

The strategy of CFD is to replace the continuous domain with a discrete domain using a grid. In the discrete domain, variable is defined at grid points. In CFD, the flow can be analyzed for the relevant flow variables only at grid points. The values at other locations are determined by interpolating the values occurred at the grid points. Governing equations define the variables in the discrete form. The discrete system is the largest set of coupling algebraic equations in the form of discrete variables. Setting up the discrete system and solving it involves an extraordinarily large number of iterations to converge the solution.
6.3 SOLVING THE PROBLEM USING CFD

The three principal stages of solving the problem are stated below.


Pre-Processor: Pre-processing steps are discussed in this section. The software Geometry And Mesh Building Intelligent Tool (GAMBIT) is used in the study. The version 2.3.16 is used in the analysis. The main pre-processor steps are:

- Geometry creation
- Meshing Method and its type
- Identifying and specifying the boundary conditions
- Specifying the continuum type

The mesh for 2-dimensional analysis includes quadrilateral and triangular mesh. Quadrilateral type meshing indicates the square or rectangle cell. But, the triangular meshing indicates the triangular cell. Out of them, quadrilateral meshing is preferred due to more accuracy. Triangular meshing yields better results when the skew angle of the triangle is 60°. Continuum type may be either solid or fluid.

The time taken for solving the problem may be increased due to increase in number of cells. But, it is compensated by increasing the accuracy levels. The main disadvantage in solving the fluid flow problems through CFD is the ram capacity of the computer system. It also depends upon the shape of the object. It may require the capacity of 4 GB. For complex components, the ram capacity...
may exceed 8GB also.

Solver:

At the outset, the numerical methods form the basis of the solver to converge the solution. Solver performs the following events.

- Approximation of the unknown flow variables by means of elementary functions
- Meshing by substitution of the approximations into the governing flow equations and subsequent mathematical manipulations
- Solution of the algebraic equations

Post-Processor:

Packages of computational dynamics are now equipped with versatile data visualization tools: they include,

- Domain geometry and grid display
- Vector plots
- Line and shaded contour plots
- 2D and 3D surface plots
- Particle tracking
- Animation view
- Colour postscript output

### 6.4 Compressible Flow and Incompressible Flow

The flow of fluid with invariant density is called incompressible fluid flow. An ideal gas behaves like incompressible flow. If, the density of flowing fluid is varied then such fluid flow is called compressible fluid flow. Flow can also be classified based on
Mach number. If Mach number is more than 0.3, then the flow can be treated as compressible fluid flow. If it lies in the range of 0 to 0.3, then the flow can be treated as incompressible fluid flow. Incompressible fluid flow is governed mainly by the conservation of mass and conservation of momentum equations. But compressible fluid flow is governed by conservation of energy equation. The flow of gas through open pipe system either internally or externally can be treated as compressible fluid flow if Mach number exceeds 0.3. Compressible flow includes flow of air around bodies such as the wings of an airplane. Results may vary by 5%, if compressible

6.4.1 Mesh

As the previous section mentioned, the mesh is the discretization of the model that is being analyzed. This forces us to create a detailed mesh to obtain an accurate representation of the model in order to obtain good results. Depending on the model, every mesh is composed by several 2D or 3D elements. Every element is defined by nodes positioned on each vertex and the second order elements have one more node positioned on the center of every edge. To create a mesh first it is needed to define the size, number or density of elements desired on each edge. Then the surfaces are meshed with tri, quad or mixed elements according to the mesh of the edges. The next step (only for 3D models) is to mesh the solids using any kind of elements. The mesh creation is made automatically by the software according to the defined mesh size. This automatic process is
very simple when using tri elements. But in the case of the quad/hex, it can be very time consuming and require some experience. The reason of continuing using this kind of elements is its superior accuracy.

6.4.2 Boundary conditions in the Analysis

The following boundary conditions are considered in analyzing the fluid flow through moving blade system. They are:

(i) Incompressible fluid flow (ii) Turbulent fluid flow
(iii) Atmospheric pressure at the inlet of blade system
(iv) Axis symmetry
(v) Neglecting wall viscous forces

Incompressible fluid flow:

Experimentally, the velocity of air at the outlet of the moving blade system is measured using rotating disc type anemometer. It is 5m/s at a minimum and 14m/s at maximum in all modules for single and multiple moving blade systems. The compressibility is defined based on Mach number. It is the ratio of velocity of an object to velocity of sound in the surrounding medium. Velocity of sound at sea level is 340 m/s. Thus, minimum and maximum Mach number becomes 0.0249 and 0.0596 respectively. It is less than 0.3. If Mach number is more than 0.3, the flow can be treated as compressible fluid flow. Hence, the flow chosen is incompressible in the analysis.

In the above two cases, Reynolds number lies above 4000. For a fully developed flow through a circular pipe, the flow becomes
turbulent since, Reynolds number exceeds 4000. Hence, in the analysis the flow is treated turbulent. Inlet velocity, inlet Pressure and inlet temperature.

This tool is used to model and measure the flow of fluids. To this flow, it can be added heat, mass transfer, phase changes, chemical reactions and interaction with moving solids in order to accurately represent the analyzed system or device. The CFD codes are based on two main aspects, the physical modeling and the numerical methods [3]. The physical model translate the information of the analyzed system or device contained on the mesh into a set of equations that relate the governing equations of mass, force (Navier-Stokes equations) and energy conservation (thermodynamics 1st law). There are different approaches to compute this job. The majority of engineers used methods are finite volume, finite difference and the method of finite element. From this approaches the FVM is the most extensively used in commercial CFD software such as Fluent, STAR-CD, CFX, Open FOAM, Floworks, Phoenics, Numeca etc.

The transformation of the differential equations creates a linear system of equations that can be solved using iterative methods. Typical direct methods cannot be used because they are computationally expensive, inefficient for large sparse matrices and the non-linearity of the coefficient of the system’s matrix forces to use iterative methods. This iterative method compute the results of the CFD analysis until it converges, another way saying it is, that it will evaluate the outcome of the new computation with the outcome of the
last computation and when the error stays lower than the user defined value, the solution is reached. When the analysis is time dependent it must be added one more consideration, the time. The pre-processing stage of a common CFD analysis consists in the following steps.

1. Import the mesh file.
2. Solver selection this step consists in the selection of the type of analysis, discretization Method for the time and conservation equations, turbulence models.
3. Operating conditions such as gravity, operating temperature and operating pressure.

**6.4.3 Analysis of single moving blade System**

The main objective of the present analysis is to show the increase in Power co efficient (Cp) of a VAWT with moving blade arrangement. Cp of the turbine is determined by using CFD analysis software i.e., Ansys CFX. In Ansys, the following input parameters which are taken into consideration [9].

Material = Air at 25 C and 1 atm.

Thermodynamic State = Gas,

Turbulence = 1 %

Density = 1.185 [kg/m³]

Molar Mass = 28.96 [kg/kmol]

Inlet Speed = 5 [m/s]

Flow type = Isothermal and steady state

After analysis of the test rig, the following results are observed
Fig: 6.1: Velocity streamline of VAWT

Fig: 6.2: Velocity counters of VAWT.
Inlet velocity = 5 m/s

Velocity at the outlet of concave side = 3.1

Velocity at the outlet of convex side = 4.7


\[
m = \rho \frac{A}{2} \left( \frac{V_1 + V_2}{2} \right)
\]

where \( m \) is the mass per second, \( \rho \) is the density of air, \( A \) is the swept rotor area and \( \left( \frac{V_1 + V_2}{2} \right) \) is the average wind speed through the rotor area. The power extracted from the wind by the rotor is equal to the mass times the drop in the wind speed squared (according to Newton's second law):

Substituting \( m \) into this expression from the first equation we get the following expression for the power extracted from the wind:
\[ P = \frac{1}{2} m (V_1^2 - V_2^2) \]
\[ P = \left( \frac{\rho}{4} \right) (V_1^2 - V_2^2) (V_1 + V_2) A \]

Total power in the undisturbed wind (Po)
\[ P_o = \left( \frac{\rho}{2} \right) V_1^3 A \]

Power co efficient (Cp) = \( \frac{P}{P_o} \)

\[
\text{Power co efficient (Cp)} = \frac{P}{P_o} = \frac{1}{2} \left[ 1 - \left( \frac{V_2}{V_1} \right)^2 \right] \times \left[ 1 + \left( \frac{V_2}{V_1} \right) \right]
\]

\[ C_p = 0.468. \]

### 6.4.4 Simulation output Parameters for multi blade system

CFD analysis is a advanced mathematical tool for fluid flow analysis. The wind flow effects are evaluated using CFD package i.e., Ansys CFX Software used- Ansys CFX

### 6.4.5 Simulation Input Parameters

- Input Velocity - 14.0 m/s
- Flow type - isothermal, steady flow, turbulence 5%
- Fluid Type - air at 25°C
- Test Rig - Vertical axis wind turbine

In symmetrical airfoils also there are different types of airfoils available but to get more accurate results I taken reference from the energy report released by Sandia National Laboratories on Lift, Drag forces and momentum of different to National advisory Committee for Aeronautics (NACA) airfoils for an angle of attack from 0 to±180°
After optimizing these results I used NACA-0009 airfoil for my project this is because of its high lift and low drag values in the range of angle of attack I used in this project.

Lift and Drag Values For +180 degrees angle of attack of NACA 0009 airfoils are given below.

Velocity at exit of turbine on concave side - 9.0976 m/s
Velocity at exit of turbine on convex side - 14.263 m/s

Fig: 6.4 Velocity counters of (Multiple blade) VAWT

After the completion of the simulation it has been utilized the values of Wind speeds to calculate the Cp of the turbine using the formula [172].

\[ m = \rho \frac{A}{2} \frac{(V_1 + V_2)}{2} \]

where \( m \) is the mass per second, \( \rho \) is the density of air, \( A \) is the swept rotor area and \( \frac{(V_1 + V_2)}{2} \) is the average wind speed through the rotor area. The power extracted from the wind by the rotor is equal to
the mass times the drop in the wind speed squared (according to Newton's second law):

Substituting m into this expression from the first equation we get the following expression for the power extracted from the wind:

\[ P = \frac{1}{2} m (V_1^2 - V_2^2) \]

\[ P = \left( \frac{\rho}{4} \right) (V_1^2 - V_2^2) (V_1 + V_2) A \]

Total power in free air (the undisturbed wind) \( P_o \)

\[ P_o = \left( \frac{\rho}{2} \right) V_1^3 A \]

Power coefficient \( C_p \) = \( \frac{P}{P_o} \)

Power co efficient \( C_p \) = \( \frac{P}{P_o} = \frac{1}{2} \left[ 1 - \left( \frac{V_2}{V_1} \right)^2 \right] \times \left[ 1 + \left( \frac{V_2}{V_1} \right) \right] \)

\[ C_p = \frac{P}{P_o} = \frac{1}{2} \left[ 1 - \left( \frac{9.0976}{14.263} \right)^2 \right] \times \left[ 1 + \left( \frac{9.0976}{14.263} \right) \right]. \]

Cp of VAWT at 0° AOA with reference to NACA airfoil = 0.4854