SOLUTION OF PARTICULATE VISCOUS FLOW IN A RADIAL INFLOW TURBINE

B.V.R. Vittal and W. Tabakoff
Department of Aerospace Engineering and Applied Mechanics
University of Cincinnati
Cincinnati, Ohio 45221

ABSTRACT

The presence of solid particles in turbomachinery flow affects the component performance as well as its life. The subject of particulated flows can be broadly divided into three parts, namely, particle trajectories, the effect of particles on the aerodynamics of flow and material erosion. The first two aspects are investigated in this paper taking into account the viscosity of the carrier fluid.

The Lagrangian formulation is adopted for the particles, whereas the Eulerian approach is used for the continuous phase. The effect of particles is incorporated as interphase force terms in the fully incompressible stream function-vorticity form of the Navier-Stokes equations. The field analysis is based on the numerical integration of this equation over the rotor blade to blade stream channels. The numerical code used to solve the governing equations employs a non-orthogonal boundary fitted coordinate system that suits the most complicated blade geometries. The trajectories of the solid particles are determined including particle impacts with the blades. The particle rebounding velocity and direction after each impact is determined using semi-empirical correlations for the restitution ratios obtained experimentally. The method of analysis is applied to a radial inflow turbine. The effect of particles on the aerodynamics of the flow is studied by analyzing the fluid streamline pattern in the rotor blades with and without solid particles. The analysis is carried out for various particle concentrations.

NOMENCLATURE

\( a \) Acceleration of the particle, m/sec
\( d \) Particle diameter, m
\( F \) Interaction parameter, \((F_0 G_1)\)
\( C_0, C_1 \) Denoting source terms in the flow governing equations
\( m \) Meridional distance, m
\( \dot{m} \) Mass flow, kg/sec
\( \Phi_p \) Mass fraction of the particles carried by a trajectory, kg/sec
\( r \) Radius from axis of rotation, m
\( \rho \) Reynolds number
\( \vec{R} \) Force per unit volume
\( \vec{R}_m, \vec{R}_s \) Interphase force terms
\( t \) Time, sec
\( V \) Absolute velocity, m/sec
\( \omega \) Stream function
\( w \) Vorticity, 1/sec
\( \gamma \) Number of blades
\( \alpha \) Angle between meridional and axis, rad, (Fig. 1)
\( \mu_e \) Effective viscosity, kg/m/sec
\( \rho \) Gas density, kg/m\(^3\)
\( \rho_i \) Particle material density, kg/m\(^3\)
\( \tau \) Particle residence time, sec
\( \theta \) Angular coordinate, rad

INTRODUCTION

The study of particulate flow is of considerable importance in a number of fields. The fine metallic powder present in many solid rocket propellants, causes erosion in the rocket nozzle. Further, as the particles lag during high flow accelerations, they may cause...
deterioration in the performance of the nozzle. In such flows, the particles may have concentrations as high as 40% of the total mass flow (1) and affect the fluid flow properties considerably. The inability of solid particles to follow sudden flow changes, is made use of in homogenous gas-particle separators and cyclones.

Many gas turbine engines operate in environments where the ingestion of particles is very common. For example, in aircraft and naval installations, the particles encountered can be sand, dust, salt or water droplets. Solid particles are also ingested by gas turbine engines in ground vehicles, auxiliary power units and tanks. Industrial gas turbines burning synthetic fuels or coal, may have problems with ash particles.

The presence of solid particles in turbomachinery flow, not only affects the life of the components, but also their performance. The problem of particulate flows in turbomachinery can be divided into three parts. The first part consists of the particle trajectories. The second part is the effect of the presence of the particles on the flow behavior. The third part consists of the nature of the solid surface impact and the material erosion. The first two aspects are investigated in this paper, for particulate flow in viscous fluid.

Particle trajectories have been studied extensively (2,3), in most cases neglecting the viscosity of the carrier fluid. There are a number of published reports (4) available on the prediction of erosion of materials due to particle impacts. The studies concerning erosion are predominantly empirical, requiring extensive experimental work. However, there is very little work published regarding the second part of the problem concerning the effect of the presence of the particles on the carrier fluid properties. Tabakoff and Hussein (5) developed an approximate method for calculating the flow properties of gas-particle mixture flowing over blades in a cascade, assuming inviscid fluid. They assumed that two stream tubes exist in the flow field around the blade; one at the suction side and the other at the pressure side. The gas flow without particles is used to determine the nondimensional area of the streamtube as a function of the given pressure distribution and inlet gas conditions of the non-particulate gas flow. The stream tubes are considered to be one dimensional nozzles and the governing equations of the particulate flow are then solved numerically.

In the present paper, the solid particle path and its effect on the carrier fluid flow properties are investigated, taking into account the viscosity of the fluid. The particle impact with the solid boundaries and the consequent deflection of the particle paths are taken into consideration. The analysis is applied to calculate viscous fluid flow with solid particles in a radial inflow turbine. The fluid flow is considered to be incompressible and having a constant effective viscosity, \( \mu^e \).

**FLOW ANALYSIS**

In the case of solid particle-gas flow, where the volume fraction of the particles is low, the Lagrangian formulation of the equations of motion is appropriate for the particulate phase, while the Eulerian approach is convenient for the continuous phase. Further, if the Eulerian approach is to be used for the particulate phase, it becomes extremely difficult to define the boundary conditions at the solid boundaries, where the particles get deflected after impact. This approach has been successfully used to analyze the viscous fluid-particle flow over a two dimensional cylinder (6).

Usually the complex three dimensional flow problem in a turbomachine is analyzed by reducing it into a combination of several two-dimensional solutions on intersecting families of stream surfaces. Most of the two dimensional solutions are either on a blade-to-blade surface of revolution \( S_1 \) or on the meridional or midchannel stream surface between two blades \( S_2 \).

The blade-to-blade stream surface, \( S_1 \), may be described by the annulus that would extend from the pressure surface of a blade to the suction surface of the next blade, as shown in Fig. 1. For the purpose of this present discussion, the stream surface \( S_2 \) will be considered to represent the mean geometric properties of the annulus. Considering the flow annulus shown in Fig. 1, the curvilinear distance along the intersection of the mid-surface of the annulus with a meridional plane is denoted by \( m \). The distance normal to the mid-surface is represented by \( n \). The thickness of the filament \( b_m \) is assumed to be small compared to the radius, \( r \). Hence, the \( n \) component of the velocity vector and all variations in the \( n \) direction are neglected. Khali et al. (7) presented the flow governing equations in terms of the stream surface annulus coordinate system \( (m,n) \). Using the well known stream function-vorticity approach, one obtains the vorticity transport equation as:

\[
\frac{\partial}{\partial m} \left( \frac{\dot{m}}{\partial m} \right) + \frac{\partial}{\partial n} \left[ \frac{\dot{m}}{\partial n} \right] = \frac{\dot{m}}{\partial m} (\mu \frac{\partial \omega}{\partial m})
\]

where

\[
\dot{m} = \frac{\dot{m}}{\partial m} \left[ \frac{3}{\partial m} \left( m \sin \frac{3}{\partial m} - \frac{1}{\partial m} \left[ \frac{3}{\partial m} (m \omega) \right] + G_0 - \frac{1}{\partial m} \left( \frac{3}{\partial m} (m \omega) \right) \right] + \frac{1}{\partial m} \left( \frac{3}{\partial m} (m \omega) \right) + G_0
\]

and \( R_m \) and \( R_n \) are the interphase force terms depicting the presence of the particles. The force terms can be included as source terms, if the volume fraction of the particles is low as in the case of solid particle-gas flow (8). The vorticity is expressed as follows:

\[
\omega = -\frac{1}{\partial m} \left( \frac{3}{\partial m} \left( m \sin \frac{3}{\partial m} - \frac{1}{\partial m} \left[ \frac{3}{\partial m} (m \omega) \right] \right) + \frac{1}{\partial m} \left( \frac{3}{\partial m} (m \omega) \right) \right)
\]

where

\[
\frac{\dot{m}}{\partial m} = \frac{1}{\partial m} \left( m \sin \frac{3}{\partial m} - \frac{1}{\partial m} \left[ \frac{3}{\partial m} (m \omega) \right] \right)
\]

A) BLADE ROW INTERSECTION WITH A STREAM SURFACE.
BOUNDARY CONDITIONS

The flow region of interest is shown in Fig. 2. It contains the blade row and the segments of the stream surface, $S_1$, extending upstream and downstream of the row. Due to the circumferential periodicity in turbomachine passages, the selected domain needs to encompass only that fraction of the flow annulus containing a single blade to blade passage. The upstream and downstream boundaries (AN, GH) are located sufficiently far from the blade so that the tangential variation along them is ignored. The flow properties are consequently considered to be uniform along the boundaries AN and GH.

Upstream Flow Boundary AN

Along the boundary AN, the known magnitude of the inlet relative velocity and its direction, is shown in Fig. 2, and specifies the values of $\frac{v_n}{m}$ and $\frac{v_t}{m}$; according to the relations

$$
\frac{v_n}{m} = \frac{b_n}{m} \quad \frac{v_t}{m} = \frac{b_t}{m} \quad \frac{v}{m}.
$$

Since the inlet stream of the gas is considered to be uniform, the absolute value of vorticity, $\omega$, has to be zero along the boundary AN. In a rotating frame of reference, as it is in the present case, the relative value of $\omega$ is given by $\omega = -[2\pi \sin \theta]$.

The Periodic Flow Boundaries AB, NM and FG, TH

The periodicity condition requires that the magnitude and direction of the flow velocity as well as other fluid properties be equal at every two corresponding points along AB and NM and similarly on FG and GH.

The Blade Surfaces Boundary MN and BF

For the laminar flow, two boundary conditions over the blades surfaces are usually specified. These are the no-slip and the impermeability of the surface in the case of blades with no injection. Therefore, the blade surfaces are treated as streamlines with the $\psi$ values specified as zero on the MN surface and unity on the BF surface. The boundary condition for $\omega$ is determined by a modified evaluation of Eq. (2) (7,9). The modification is introduced in such a way to satisfy the no slip condition.

The Downstream Boundary GH

For the vorticity $\omega$, the absolute value is taken to be zero, hence,

$$\omega = -[2\pi \sin \theta] \quad \text{exit}.$$

The estimated exit flow angles, $\beta_{\text{exit}}$, along GH are used to specify the values of $\psi$ derivatives in $m$ direction through the relation:

$$\frac{\partial \psi}{\partial m} = \frac{1}{2 \pi} \frac{\tan \beta_{\text{exit}}}{\beta_{\text{exit}}}.$$

The flow field equations are then solved for the above boundary conditions, to obtain the velocity and the pressure distribution throughout the turbomachine channel. An evaluation of the torque developed by the channel is obtained through the integration of the difference in pressure and shear forces acting on the blade surfaces. The change in the angular momentum between the known inlet and the estimated exit flow conditions is calculated. If the value of the predicted torque was not equal to the rate of change of the total angular momentum, then the direction of the exit velocity is altered. This procedure is outlined in reference (9).
LAGRANGIAN FORMULATION FOR THE PARTICLES

For large ratios of particle material density to the gas density, the force acting on a spherical particle due to the flow pressure gradient can be neglected, when compared to the drag force on it. Other forces that act on the particles such as the force to accelerate the apparent mass of the particle relative to the fluid, the Magnus force, the Basset force, and the force that causes particle motion due to shear flow, may be neglected. In addition, particle-particle interactions are neglected. Considering the drag as the only force of interaction between the two phases, the equations of motion of a single particle in m-λ coordinates are given as follows:

\[
\frac{d^2m}{dt^2} = F(W_{m} - \frac{dm}{dt}) \quad \text{and} \quad \frac{d^2p}{dt^2} = F(W_{p} - \frac{dp}{dt}).
\] (4)

In the trajectory calculations the particles are assumed to be spherical, leading to the following expression for the interaction parameter:

\[F = 3 \frac{\rho_c}{\rho_b} \frac{C_d}{d} \left[\left(W_{m} - \frac{dm}{dt}\right)^2 + \left(W_{p} - \frac{dp}{dt}\right)^2\right].\] (5)

where \(\rho_c, \rho_b\) are the gas and solid particle densities, \(d\) is the particle diameter, and \(C_d\) is the drag coefficient. The drag coefficient is dependent on the Reynolds number, which is based on the relative velocity between the particle and the gas. Empirical relations are used to fit the drag curve over a wide range of Reynolds numbers.

PARTICLE REBOUND MODEL

After impacting a solid boundary, the magnitude and direction of the particle rebounding velocity is dependent on the particle material, the surface material and on the aerodynamics of impact. The following empirical relations for the rebound to impact restitution ratios are used in the trajectory calculations (10):

\[
\frac{V_{N_2}}{V_{N_1}} = 1.0 - 2.12 \frac{\beta_{p}}{\rho_{p}} + 3.0775 \frac{\rho_{p}}{\rho_{p}}^2 - 1.1 \frac{\rho_{p}}{\rho_{p}}^3 \quad \text{and} \quad \frac{V_{T_2}}{V_{T_1}} = 1.0 - 0.4159 \frac{\beta_{p}}{\rho_{p}} - 0.4994 \frac{\rho_{p}}{\rho_{p}}^2 - 0.292 \frac{\rho_{p}}{\rho_{p}}^3.
\] (6)

\(V_{N}\) and \(V_{T}\) represent the particle velocity components normal and tangent to the solid surface, and the subscripts 1 and 2 refer to the conditions before and after impact, respectively. \(\beta_{p}\) is the angle between the impact velocity and the tangent to the surface in radians.

NUMERICAL PROCEDURE

The solution of Eqs. (1-2) subjected to the appropriate boundary conditions discussed earlier is carried out numerically. In order to reduce the complexity of handling the numerics near the curved boundaries of the blade surface, a coordinate transformation of the \((m,\lambda,n)\) system to a contracted boundary fitted coordinate system is carried out using Thomson transformation (11). The overall effect of this transformation is to produce a square field in which the blade surfaces become straight and parallel. The details of this procedure are given in reference (9).

The stream function and vorticity transport form of the equations of motion of fluid including the interphase force terms, are solved using the point SOR method. The flow domain has been divided into \((35 \times 37)\) grids with greater number of grids distributed in the meridional direction. The interphase force terms in the fluid flow equations are evaluated separately using particle trajectory calculations as employed by Crowe (12). Particles are introduced upstream of the blades at various pitch locations. Uniform entry of the particles with the flow is approximated by 100 discrete entry locations, each location is assumed to carry a fraction of the total particle mass. First, the flow field is established by solution without solid particles. Using this flow field, the particle flow path and its properties along the path are calculated by the numerical integration of Eqs. (4) and (5). Referring to Fig. 3, the force per unit volume, \(R\), at the grid point \((i,j)\) is given by

\[R_p = \frac{m_p \rho_p}{Vol(i,j)},\]

where

\[m_p = \frac{MF \tau}{Vol(i,j)},\]

MF is the mass fraction of the particles carried by the particular trajectory, and \(\tau\) is its residence time in volume, \(Vol(i,j)\). Using the above procedure, the interphase force terms are calculated and then incorporated in the numerical solution of the fluid flow equations.

RESULTS AND DISCUSSION

The viscous fluid-particle flow is investigated for a radial inflow turbine. The rotor consists of eight straight blades. The description of the rotor geometry is given in Fig. 4. The primary reason for the selection of this specific rotor is that a substantial amount of experimental data is available for it (13).
The operating conditions used for the turbine in the present analysis are as follows:

- Turbine Inlet Total Pressure, $P_t = 10,750 \text{ kg/m}^2$
- Turbine Inlet Total Temperature, $T_t = 288\degree\text{K}$

The calculated fluid streamline pattern, in the absence of particles, for the given conditions is shown in Fig. 5. The streamlines are plotted for the region between a pair of blades, represented by the heavy thick lines. The streamlines are designated by a stream function ratio $\Psi/\Psi_{\text{total}}$, such that the distance between any two streamlines indicates the amount of the flow between them. Thus, for the given channel configuration, the streamline spacing is indicative of the velocity relative to the rotor, with close spacing indicating high velocities and with wide spacing indicating low velocities. From the inspection of Fig. 5, it can be observed that there is a recirculating zone near the blade pressure surface. After obtaining the flow solution the next step was to calculate the particle trajectory for various particle sizes ranging from 10 to 100 microns in diameter.

A time step of 0.0001 sec. is used in all the trajectory calculations, except preceding the solid surface impact. Lower time steps require higher computer time. The optimum time step is obtained after a few trails, in such a way that any further reduction in the time step do not change the magnitude of the interphase forces. The results are presented for various...
particle sizes to illustrate the generally different trajectory characteristics. Figures 6, 7 and 8 show the representative particle trajectories for 10, 50, and 100 micron diameter particles. The smaller particles follow the fluid streamlines more closely and the bigger particle deviate from the streamlines because of their inertia. The particle impacting on the suction blade surface are slowed down considerably after repeated impacts. Particles with larger diameter have more repeated impacts when compared to the smaller particles. Some particles travel through the blade channel without any impact, on either of the blade surfaces. From these figures, it can be observed that, after repeated impacts on the blade suction surface, the particles velocity decreases and they follow the blade suction surface to the exit. The flow pattern present in the rotor channel is such, that there are not many particle impacts on the blade pressure surface. Larger particles in the range of 100 microns are the only ones which impact on the blade pressure surface near the channel exit. The smaller diameter particles, because of their lower inertia, enter into the recirculation zone as shown in Fig. 6. The trajectory of a particle in a moving fluid is governed by the vector balance of its rate of momentum change and the external forces (viscous drag) acting upon it. The particle motion causes changes in the fluid flow properties due to the particle-gas interactions. Figure 9 shows the effect of particle concentration on the fluid flow pattern inside the radial inflow turbine blade channel, for a mean particle diameter of 10 microns. In the rotor blade channel entrance region, the particles do not have too much influence on the fluid, because the relative velocities between them are small. After the solid particles impact on the blade surfaces, their velocities differ considerably from that of the gas flow, both in magnitude and direction, consequently affecting the fluid flow properties. Figure 10 is obtained by superposing streamlines of Figs. 9C on Fig. 5. Comparing Figs. 5 and 9, it can be observed that the fluid streamlines are shifted with the presence of the particles. Near the blade exit, the streamlines are shifted in the direction of the blade pressure surface. This shift becomes more dominant at higher particle concentrations. In the vicinity of the blade suction surface, towards the exit, the streamlines are less closely spaced, indicating that the fluid has slowed down considerably due to the particles momentum loss. Since ten micron diameter particles follow the fluid streamlines more closely (Fig. 6), they enter into the recirculation zone and cause further flow disturbances. Figure 9 shows the case were as the particle concentration increases, the recirculation zone grows larger. All these flow changes will affect the turbine performance. The numerical solution is obtained on an AMDahl 470 computer. The computer code requires about 500 k in storage and 6 minutes of CPU time.

CONCLUSIONS

The dynamic behavior of solid particles entrained by the viscous gas flow in a inward radial flow turbine rotor blade channel is determined. The particle impact
with the blades and the consequent deflection of the
particle paths are considered. The true characteristics
of the particle trajectories very near to the blade
surfaces are brought out clearly in a viscous flow
analysis and also the path of the particles in the
recirculation zone.

In general, it may be stated that the effect of the
presence of solid particles in the fluid flow is better
demonstrated in the analytical results when the analysis
is performed with viscous flow. The difference between
using viscous or inviscid flows is very much pronounced
in certain blade regions.

FIG. 9A EFFECT OF PARTICLE CONCENTRATION ON FLUID
STREAMLINE PATTERN. (PARTICLE SIZE = 10 MICRONS,
MASS FRACTION = 0.00).

FIG. 9B EFFECT OF PARTICLE CONCENTRATION ON FLUID
STREAMLINE PATTERN. (PARTICLE SIZE = 10 MICRONS,
MASS FRACTION = 0.08).

FIG. 9C EFFECT OF PARTICLE CONCENTRATION ON FLUID
STREAMLINE PATTERN. (PARTICLE SIZE = 10 MICRONS,
MASS FRACTION = 0.10).

FIG. 10 EFFECT OF PARTICLES ON FLUID STREAMLINE
PATTERN.

ACKNOWLEDGEMENT

This research was sponsored by the U.S. Army

REFERENCES

1. Rudinger, C., "Fundamentals and Applications of

2. Clevenger, W.B. and Tabakoff, W., "Dust Particle
Trajectories in Aircraft Radial Turbines," Journal


