BACKGROUND AND RELEVANCE

Cavitation is the major source of degradation of the suction performance, reliability, useful life and power density of the high power density turbopumps for space application (Brennen, 1994) and the cause of other equally undesirable effects, such as the reduction of the overall efficiency and the drastic increase of the noise generation (Stripling and Acosta, 1962).

Even more important for space applications, cavitation can provide the necessary flow excitation and compliance for triggering dangerous rotodynamic and/or fluid mechanic instabilities of the turbopump (d’Agostino et al., 1995, Jery et al., 1985, Franz et al., 1989), or even, through the coupling with trust generation, of the entire propulsion system (POGO auto-oscillation of liquid propellant rockets).

The occurrence of rotating cavitation has been extensively reported in the development of the most high performance liquid propellant rocket fuel feed systems, including the Space Shuttle Main Engine (Ryan et al., 1994), the Ariane 5 engine (Goirand et al., 1992) and the LE-7 engine (Kamijo et al., 1993).

The occurrence of cavitation represents to-day one of the most important limitations in pump design. For many applications, in particular in rocket pump technology, weight reduction is an essential feature. For rocket turbopump, the lowest possible suction head is requested to minimize tank pressurization at rocket starting and directly the weight of the structure. Typical gain of 100 kg on the total weight is obtained with a gain of 0.1 bar on the minimal suction pressure (Kueny, 1993).

This brief overview illustrates the importance of the prediction of cavitation. It also indicates that the successful development of a model for simulating cavitation must be based on careful consideration of the final objectives in order to exploit all opportunities to simplify the formulation of the problem including only the essential physical phenomena.

OBJECTIVES

This Ph.D. Thesis, carried out at C.I.R.A., the Italian Aerospace Research Center, was focused on the development of a physical and numerical model for the simulation of the macroscopic aspects of cavitating flows more directly affecting the suction and dynamic performance of high performance turbomachines in typical space propulsion applications. The model was intended to be capable of providing information on the extent of cavitation, its dependence on the geometric and fluid dynamic parameters of the flow, the induced modification on the boundary forces, and ultimately the unsteady evolution of the flow.

General applicability and independence on specific hypotheses concerning the nature and form of cavitation have been considered as prominent objectives in the development of the model, in order to maximize its predictive value. For the same reason the use of empirical parameters has been limited as much as possible to those commonly accepted in the description of cavitating flows.

In the following a survey on the physical and numerical development will be given. Finally the main numerical results obtained will be shown and discussed.

PROPOSED APPROACH

FLOW MODEL

In cavitating liquids with relatively high vapor pressures (like most cryogenic propellants), pressure and velocity differences between the two phases can safely be neglected (Brennen, 1995) and the development of an homogeneous flow model seems to be an efficient approach to the simulation of cavitating flows for performance predictions in space propulsion applications.

These approaches are characterized by a set of equations valid for a mixture, one-fluid model, which is assumed to be in thermodynamic equilibrium. The ‘void fraction’ variable, defined as the ratio between the vapor volume and the whole volume, is employed to quantify the extent of cavitation.

Within the frame of homogeneous approach, a continuous variation of density between liquid and vapor values, in a range of pressure centered at the vapor pressure, is assumed. Such a variation can be represented by a sinusoidal curve (Fig. 1), by a polynomial curve, etc. In all cases, the maximum slope of the curve occurs at the vapor pressure. Alternatively, the barotropic model can be derived by an isentalphic model (Avva, 1995), which leads to a non continuous slope of the barotropic curve at the saturation point (Fig. 2).

For the present study a sinusoidal law, following Delannoy and Keuny (1998), has been chosen.

In the homogeneous approach and in the hypothesis of thermodynamic equilibrium the numerical problem is described by the Navier-Stokes equations with a barotropic law of state.
\[ \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{V}) = 0 \quad (1.0) \]

\[ \frac{\partial \rho \mathbf{V}}{\partial t} + \nabla \cdot (\rho \mathbf{V} \mathbf{V}) = -\frac{\partial P}{\partial x} + \nabla \cdot (\mu \nabla \mathbf{V}) \quad (2.0) \]

\[ \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{V} \mathbf{V}) = \frac{\partial P}{\partial x} + \nabla \cdot (\mu \nabla \mathbf{V}) \quad (3.0) \]

where the viscosity of the mixture is calculated from (Kubota et al., 1992):

\[ \mu = \mu_l + \mu_v (1 - \alpha) \quad (5.0) \]

The barotropic variation in the two-phase zone is represented by the following relation, as shown in Fig. 1:

\[ \rho = \rho_l + \frac{1}{a_{\min}^2} \pi (p_l - p_v) \sin \left( \frac{p - p_v \pi}{p_l - p_v} \right) \quad (6.0) \]

The minimum speed of sound, \(a_{\min}\), is the parameter that determines the maximum slope of the barotropic curve. In this approach, it is necessary to specify three constants: the minimum speed of sound, the liquid density and the vapor density. For the last two properties one can choose the density values corresponding to the equilibrium conditions (for a fixed temperature). A reasonable value of \(a_{\min}\) could be assumed in the range limited by the values obtained by the dynamic approach of Jakobsen (1964) and the isenthalpic transformation approach (Avva and Shingai, 1995), which gives considerably different values of minimum speed of sound. The effects of the minimum speed of sound on the flow solution are discussed in Pascarella et al., 2000a.

**NUMERICAL APPROACH**

The simultaneous presence of two phases produces dramatic changes of the speed of sound, which can be as low as a few meters per second even at moderately high void fractions (on the order of 20%), and therefore much lower than the flow speed. Besides, the rapid response of the cavities to changes of the external pressure generates steep gradients of the void fraction in the transition region from the pure liquid to the cavitation zone. Hence, these flows are characterized by extremely rapid changes of the Mach number from negligible values (corresponding to the zone of pure liquid or pure vapor) to supersonic values in the two-phase cavitation region. Usual numerical approaches are not capable of handling in a unified manner the simultaneous presence of extremely compressible and practically incompressible regions separated by very rapid shock-like transitions. In addition, the large density difference between the liquid and the vapor further contributes to making the numerical problem extremely ill-conditioned.

The simulation of cavitating flows requires therefore the application of specialized methods capable of overcoming these difficulties. In view of the above considerations, the most suitable scheme to solve the Reynolds Averaged Navier Stokes equations for barotropic cavitating flows is a pressure-based method with a finite volume discretization in conservative form based on the SIMPLE (Semi Implicit Method for Pressure Linked Equations) algorithm (Patankar, 1980). With the compressibility correction proposed by Karki (1986) this method is capable to solve flow fields at all Mach numbers. Furthermore, due to the conservative form of the discretized equations, it is also capable to capture shock discontinuities.

**VALIDATION**

The numerical method has been validated using the experimental data available from Rouse and McNown (1948), consisting in pressure measurements on the surface of some head forms at zero angle of yaw, in water under steady state conditions. Although these experiments are very old, the data are quite reliable.

Among the several tests reported in Rouse and McNown (1948) two different head form shapes have been chosen. Both head forms belong to the so-called “Rounded Series”; in particular, they are called “2in.-Caliber Ogival” and “Hemispherical” head forms (Fig. 3). The comparison between numerical and experimental data has been made for each geometry at two different flow conditions corresponding to incipient cavitation and fully developed cavitation (Pascarella et al., 2000a). The simulations have been performed neglecting the viscous effects (Euler approximation) and assuming a minimum speed of sound of 0.12 m/s. The calculations have been carried out assuming a temperature of 320 K; the corresponding density ratio is equal to 13305 (989/0.072) and the vapor pressure is equal to 10530 Pa. Both the very high density ratio and the very low speed of sound represent quite severe conditions: in most of the works found in the open literature on the numerical modeling of cavitation using a barotropic law of state, a maximum value of density ratio of 1000 and a minimum speed of sound of 0.5-1 m/s is reported (Delannoy and Keuny, 1998, for example).
RESULTS AND DISCUSSION

NACA0015 AIRFOIL

Simulations of a NACA0015 airfoil have been performed at different angles of attack and cavitation numbers to verify the capability of the computer code to resolve the main flow features and the cavitation bubble.

Water at 320 K has been assumed as working fluid. An inlet velocity of 3 m/s and a pressure at the exit of 17000 Pa have been fixed as boundary conditions. A C-grid with 162x61 nodes has been employed.

Fig. 7 is a density contour plot for the airfoil at 8 degree angle of attack. A large cavitation bubble develops on the suction side of the airfoil, originating from a point close to the leading edge and extending up to about half chord length.

The flow is unstable, with a periodic detachment of vapor bubbles. The detailed description of the flow evolution is rather complicated and is discussed in Kubota et al. (1992). However, it is interesting to show that the current model is capable to capture the so-called reentrant jet, a flow inversion downstream of the main bubble, with a liquid jet traveling upstream along the surface of the airfoil, which eventually breaks the cavity into two or more smaller bubbles that are then transported downstream in the main flow.

The change in the dynamic response of the cavitating flow field around a NACA0015 airfoil to a variation of the minimum speed of sound,  \( a_{\text{min}} \), has been also investigated. Water at temperature of 323 K, that corresponds to an enthalpy equal to 200 kJ/kg, has been chosen as working fluid.

Under these conditions, the isenthalpic model (Avva, 1995) and the dynamic approach (Jackobsen, 1964) give the following values: isenthalpic model: \( a_{\text{min}} = 0.17 \) m/s; Dynamic approach: \( a_{\text{min}} = 8.16 \) m/s.

A parametric study of the cavitating flow for a variation of minimum speed of sound between the above two limits has been performed. In particular four cases have been investigated: \( a_{\text{min}}=3.0 \) m/s, 2.0 m/s, 1.0 m/s and 0.8 m/s. These values are representative of two different extreme behaviors (Pascarella et al., 2000b), i.e. sheet cavitation with a quasi-steady field (\( a_{\text{min}}=3.0 \) m/s ) and an unsteady sheet cavitation formation with a very fast variation of both bubble size and cavitation intensity (\( a_{\text{min}}=0.8 \) m/s).

Two main conclusions can be drawn. First, at equal cavitation number, the flow steadiness decreases with the diminution of speed of sound and the field goes from a quasi-steady flow for the case with a minimum speed of sound of 3.0 m/s to an almost periodic response for the case at minimum speed of sound of 0.8 m/s (Fig. 8).

The other, significant conclusion is that the dimension of cavitation region, in the case in which the flow is steady or quasi-steady, tends to decrease at higher values of  \( a_{\text{min}} \) (Fig. 9). For all the calculations performed, due to the stiffness of the phenomenon, very low under-relaxation parameters and very low time step increments, and therefore a large amount of computer time is required.
have been needed.

**INDUCER**

The numerical model has been applied to the simulation of a typical inducer geometry. In particular, a 9° Helical Inducer referred to as Impeller III in Brennen (1994) has been selected. The numerical analysis has regarded 2D simulations of the blade-to-blade flow at a fixed radial position. The particular inducer considered in the present analysis is an axial, helical inducer, with three blades having straight, radial leading edges; since the pitch of the helix is constant, the development of a cylindrical section leads to a cascade representation with straight blades and constant blade angle $\beta_b$ (Fig. 10). The main geometrical parameters of the selected inducer are the following: Tip radius: $R_T = 37.90$ mm; Hub radius: $R_H = 15.16$ mm; Axial length: $d = z_{in} - z_{ex} = 30.68$ mm; Blade tip angle at inlet: $\beta_{b1}$ = 9 degree.

The computations have been carried under the following conditions: rotational speed: $\Omega = 10000$ rpm = 1047.19 rad/sec; flow coefficient: $\phi = 0.10$; absolute inlet velocity: $V_{in} = \Omega R_T/\phi = 3.97$ m/s; relative inlet velocity: $V_r = \sqrt{V_{in}^2 + (\Omega x)^2}$ = 28.06 m/s; flow angle: $\beta_1 = \tan(V_{in}/\Omega x) = 8.13^\circ$; incidence angle: $\alpha = \beta_b - \beta = 4.62^\circ$.

Water at a temperature of 323 K has been chosen as working fluid. A minimum speed of sound of 1.0 m/s has been assumed. This seems to be a reasonable value, within the range previously defined and limited by the isenthalpic model (Avva, 1995) and the dynamic approach (Jackobsen, 1964) values.

Under these conditions, six test cases have been carried out varying the exit pressure between the two limits corresponding to non-cavitating flow and fully choked cavitating flow, as summarized in Tab. 1. Fully choked cavitation is attained when the whole length of the inducer blade airfoil is covered by a cavitation stream. The cavitation number at which such event occurs is called the choked cavitation number, and indicated with $\sigma_c$, as in Brennen (1994).

The values of the choked cavitation number, $\sigma_c$, are available for the case under considerations, Brennen (1994). To reach gradually the choked condition and, at the same time, to analyze the intermediate behavior, the exit pressure has been progressively reduced starting from a non-cavitating condition. The onset of cavitation occurs when the exit pressure is decreased below 200000 Pa, corresponding to a cavitation number of 0.19.

The further decrease of the exit pressure produces as a result an enlargement of the cavitation region. This is visible in Fig. 11, where the density contour plots for the four cases a-d are reported. The thickness of the bubble for the cases b, c, and d can be estimated from the figure. The rotation of the streamlines around the bubble is also clearly visible; the liquid-vapor mixture zone represents a sort of an obstacle that displaces the streamlines, thus creating vorticity and consequently dissipation. It should be noted that Fig. 11 is not to scale, i.e. the axial dimensions are greater of one order of magnitude.

The exit pressure at which the choked condition is reached is about 157000 Pa and the associated value of $\sigma$ is 0.0105. For this case, Brennen (1994) reports a value obtained by means of an empirical correlation, equal to 0.009. The experimental value of the breakdown cavitation number, that is the value of the cavitation number at which a choked condition is reached at any radial station, is also reported; for Impeller III, this value is equal to 0.012. Taking into account all the approximations made in the present analysis, the agreement is considered to be excellent. The behavior of the choked cavitating flow is shown in Fig. 12 by means of the pressure distribution. The vapor cavity now extends over the entire foil surface, as shown by the flat segment between $x=0$ and $x=0.13$. It is interesting to note that in this condition, as in the case of choked supersonic flow, no changes occur upstream of the choked field zone by further decreasing the exit pressure. This is revealed by the fact that the pressure curves at 150000 Pa and at 157000 Pa are exactly overlapped upstream of the cavitating zone. In both cases, the re-condensation occurs by a normal shock much stronger than in the non-choked cases. The density contour plot for the choked condition, case e, reported in Fig. 13, impressively shows the behavior of flow when the whole suction side of the inducer foil is wetted by a cavitating stream.

![Fig. 10 9 degree Helical Inducer Impeller III](image-url)

![Fig. 11 Density contour plots for different values of the exit pressures (not to scale)](image-url)

![Fig. 12 Pressure plots for different choked conditions](image-url)

![Fig. 13 Density contour plots for choked condition (not to scale)](image-url)
<table>
<thead>
<tr>
<th>Case</th>
<th>$a_\infty$ [m/s]</th>
<th>Inlet velocity [m/s]</th>
<th>Outlet pressure</th>
<th>$\sigma = \frac{p_1 - p_2}{0.5\rho L^2 R_i^2}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>a</td>
<td>1.0</td>
<td>28.06</td>
<td>250000</td>
<td>0.319</td>
</tr>
<tr>
<td>b</td>
<td>200000</td>
<td>0.190</td>
<td></td>
<td></td>
</tr>
<tr>
<td>c</td>
<td>175000</td>
<td>0.115</td>
<td></td>
<td></td>
</tr>
<tr>
<td>d</td>
<td>160000</td>
<td>0.041</td>
<td></td>
<td></td>
</tr>
<tr>
<td>e</td>
<td>157000</td>
<td>0.0105</td>
<td></td>
<td></td>
</tr>
<tr>
<td>f</td>
<td>150000</td>
<td>0.0105</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

*Table 1: Run matrix for 9 degree Impeller III*

**CONCLUSION**

A physical and numerical model for the simulation of the macroscopic aspects of cavitating flows more directly affecting the suction and dynamic performance of turbomachines in typical space propulsion applications has been developed. A barotropic cavitation model has been implemented within an unsteady 2-D planar and axysymmetric Euler/Navier Stokes flow solver. Several simulations have been carried out to validate the model and to verify the capability of the computer code to capture the vapor cavity and the relevant flow features. A parametric study of the influence of speed of sound variation, using a NACA0015 airfoil, has been performed. Simulation carried out on the Helical Impeller III are in excellent agreement with available data. Ultimately, the developed code can be considered as an useful tool in the quantitative prediction of cavitating flows as well as it can provide rocket engineers with the essential performance information on a more efficient design of cavitating turbomachinery for space applications.

**REFERENCES**


Pascarella C., Ciucci A. and Salvatore V., 2000a, “Numerical Study on the Effects of Speed of Sound Variation on Unsteady Cavitating Flows”, Submitted to JFE, Transaction from ASME


