Chapter 6

Types of cavitation: SHEET CAVITATION

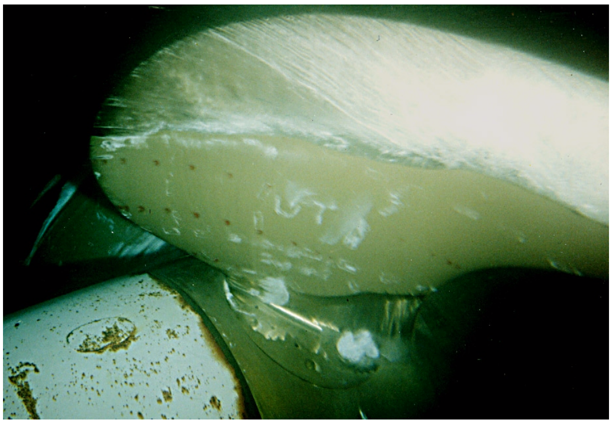

Figure 6.1: Sheet cavitation on a propeller blade at full scale

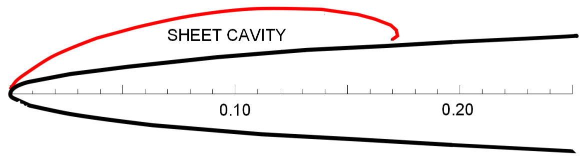

Figure 6.2: Sketch of 2-dimensional sheet cavitation

Objective: *Description of the appearance and behavior of sheet cavitation*

An example of sheet cavitation on a ship propeller is given in Fig. 6.1.

Sheet cavitation is a region of vapor which remains approximately at the same position relative to the profile or propeller blade. In this way it seems attached to the foil. The surface of sheet cavitation can be very glossy, at least at model scale, but at full scale the surface is often not transparent, as in the outer radii on Fig. 6.1. In this Figure the sheet cavity is attached to a strongly cavitating tip vortex, which will be treated later. Without diffusion the pressure in the sheet cavity will be close to the equilibrium vapor pressure and the surface of the cavity can be considered as a free surface.

6.1 2 Dimensional sheet cavitation

In two dimensions a sheet cavity can be sketched as in Fig. 6.2

Sheet cavitation occurs when there is a strong low pressure peak at the leading edge of the foil and sheet cavitation therefore has its leading edge close to the leading edge of the foil.

6.1.1 The cavity leading edge

The fact that the surface of a sheet cavity is a constant pressure surface has consequences

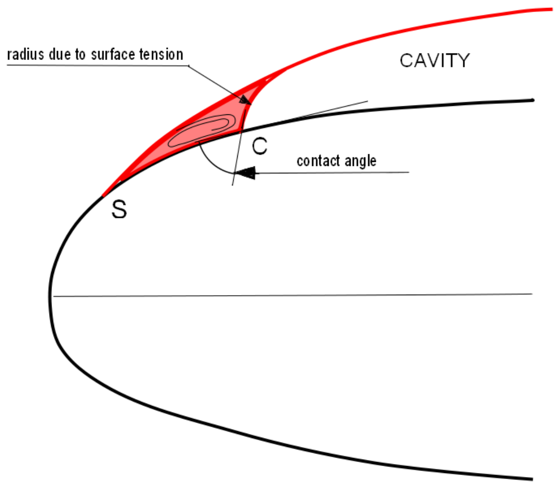

Figure 6.3: Leading edge structure of a 2D sheet cavity

for the streamlines at the beginning and at the closure of the cavity, as shown in Fig 6.3. At the beginning or leading edge of the cavity the constant pressure means that the streamlines separate tangentially from the surface of the foil in point S(assuming that the foil is a smooth surface). Tangential separation, however, means that there is a region just downstream of the separation location where the cavity is very thin. So thin that the surface tension becomes recognizable and results in a curved surface, making the leading edge of the cavity in point C instead of S.

The curvature at the leading edge of the cavity depends on the surface tension between vapor and water, and on the location of point C. The angle at the separation point depends on the contact angle between water, vapor and the surface material. Due to viscosity of the fluid a circulation will occur inside the red region.
As a consequence of the surface tension the pressure in the red region upstream of the separation location will be below the vapor pressure. This would mean that there is fluid between point S and C with a pressure below the vapor pressure. This is only possible when there are no nuclei in that region. Although some measurement indicate the existence of such a small region, it is more probable that the location of C is such that the radius of the cavity leading edge is large and the pressure outside the cavity remains very close to the vapor pressure. This makes the region S-C a constant pressure region. Hoekstra [20] found from RANS calculations that in this region a significant production of vapor takes place, which could make that the streamlines are not coinciding with the free surface. It is not clear yet if this is physical or a consequence of the cavitation model in the RANS calculations. Williams et al [64] point out that the definition of cavity length is different when the leading edge is taken from points S or C. For thin foils such as used in ship propellers and airplanes the distance S-C is very short and the precise structure of the leading edge of the cavity is unimportant.
Note that the flow separation at the leading edge of the cavity (point S) is *cavity induced.* At a higher pressure without cavitation no separation may occur!

6.1.2 The trailing edge of the cavity

More important is the structure at the trailing edge of the cavity. A constant pressure along the free surface requires a smooth attachment of the streamline to the foil. This does not happen in general. Instead some fluid will reenter the cavity, creating a so-called *re-entrant jet.*Fig 6.4

6.1.3 Shedding of clouds by sheet cavitation

As a result the two-dimensional cavitating flow over a foil is never steady, as is illustrated in Fig. 6.5.

When the cavity becomes shorter at a higher pressure the shedding becomes less violent,

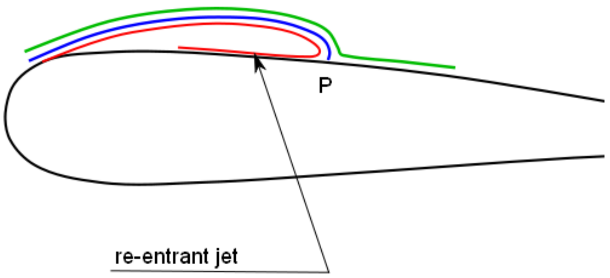

Figure 6.4: Leading edge structure of a 2D sheet cavity

Figure 6.5: High Speed Video:Shedding of cloud cavitation behind a long two-dimensional sheet cavity.

but remains present. (Fig. 6.6)

In principle the red streamline in Fig 6.4 is the cavity surface and it is a constant pressure surface at the equilibrium vapor pressure. The existence of a re-entrant jet means that there is a streamline just outside the re-entering fluid which ends in a stagnation point P (the blue streamline in Fig. 6.4). Outside that streamline the flow continues over the foil surface (the green streamline).

Figure 6.6: High Speed Video:Shedding of cloud cavitation behind a short two-dimensional sheet cavity.

The presence of a re-entrant jet makes that this cavity cannot remain stable. It gradually fills with fluid. As a result part of the cavity, or sometimes even the whole cavity, is separated from the foil and is shed as *cloud cavitation* with the fluid, as sketched in Fig. 6.7

When the re-entrant jet is 2-dimensional it means that the jet is directed upstream. A strong upstream re-entrant jet can even reach the leading edge and in such a case the whole sheet cavity detaches from the foil,as is shown in Fig. 6.8, while a new sheet is formed at the leading edge. Note that this occurs in *steady* inflow!. This is a very violent unsteady behavior of the cavity, which leads to a violent implosion of the cloud.

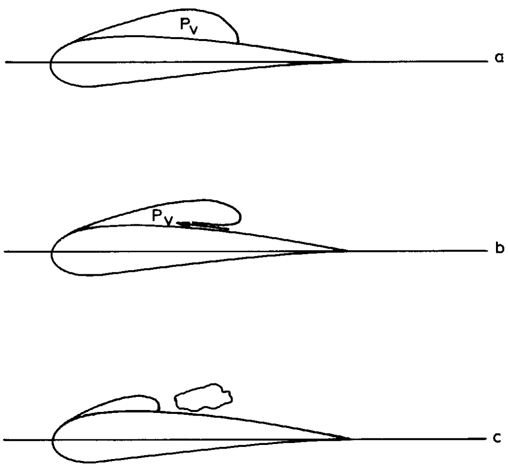

Figure 6.7: Sketch of shedding of cloud cavitation in 2D

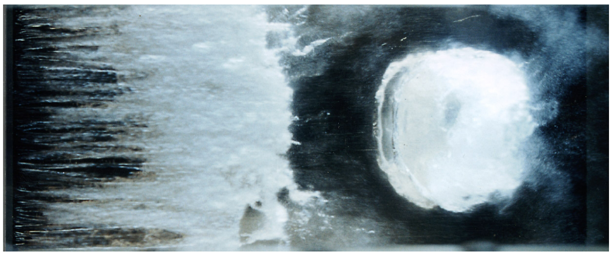

Figure 6.8: Shedding of cloud cavitation on a 2-dimensional foil

6.2 Three-dimensional shedding of cloud cavitation

In general the re-entrant jet is not 2-dimensional, even not on a 2 dimensional foil, as can be seen in Fig. 6.6. Sometimes there is a large part of the sheet which detaches, but generally the shedding has a smaller scale. This is because the trailing edge of the cavity is not a straight line. There are disturbances in the cavity length and this leads to curvatures in the trailing edge. In such a case the re-

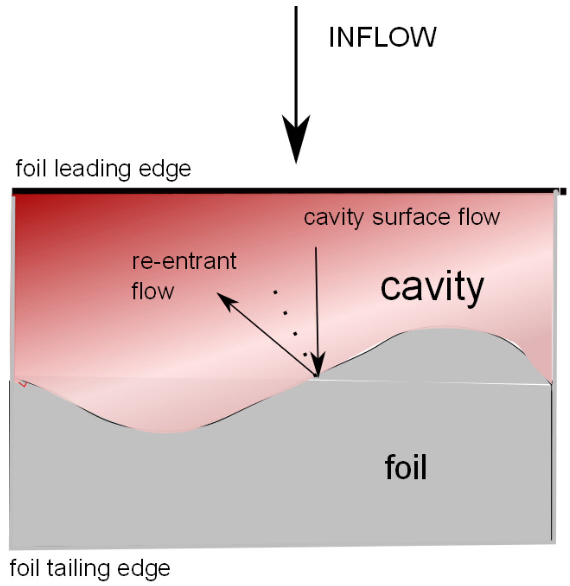

Figure 6.9: Cavity surface flow and reflected re-entrant flow

entrant flow the re-entrant flow is "reflected" against the cavity trailing edge, as is sketched in Fig 6.9. This reflexion can be understood, because the trailing edge of the cavity is a constant pressure line. There is no reversion of the flow along the trailing edge. So only the flow component perpendicular to the trailing edge of the cavity is reversed.

The reflection of the re-entrant flow against the trailing edge of the cavity has serious consequences for shedding. When the cavity trailing edge is convex the re-entrant flow will converge, as is sketched in Fig. 6.10.

This leads to a strong upward flow in the centerline of the convergent re-entrant flow, with shedding as a result. A schematic form of the shedding is that a wedge shaped opening is formed in the cavity. Although the structure of the re-entrant flow is unknown, it seems that the collision at the centerline cuts off the downstream part of the cavity and this cut-off spreads quickly in spanwise direction, leading to the wedge-shaped hole in the cavity.
The part of the cavity downstream of the opening is shed with the flow. It often contains ro-

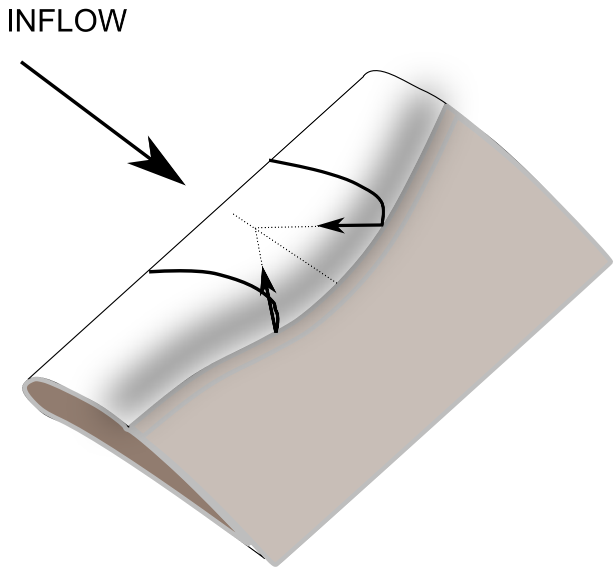

Figure 6.10: Converging re-entrant flow due to a concave cavity trailing edge

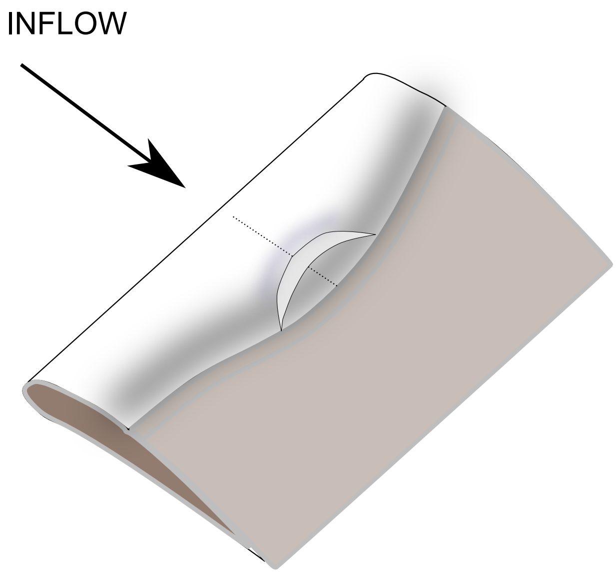

Figure 6.11: Schematic shedding of a vortical cavity due to converging re-entrant flow

tation, so it has the characteristic of a short cavitating vortex (Fig 6.11).

A picture of such a shedding is shown in Fig. 6.12.

In this figure the cavity was strongly 3-dimensional, which was caused by the twisted shape of the foil ([16]). The remaining sheet

Figure 6.12: Shedding of a vortical cavity behind a three-dimensional sheet cavity in steady inflow

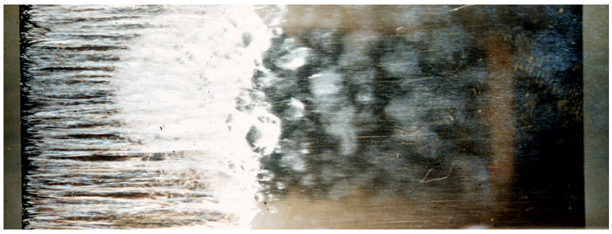

Figure 6.13: Shedding behind a two-dimensional cavity in steady inflow

cavity has a concave trailing edge at the location of the shedding and the re-entrant flow will not converge, but spread out inside the cavity. However, at both ends of the concave region there is a convex location again and the shedding will repeat itself on a different scale. This mechanism is also present in "two-dimensional" cavities, where smaller oscillations of the trailing edge of the cavity cause local shedding. An example is given in Fig. 6.13.

Such small scale shedding was also observed in the video of Fig 6.6. Cavitation frees dissolved air from the fluid and after collapse of a cavity there is always free air left. A stream of free air bubbles is therefore always present in the wake of cavitation. The three-dimensional; shedding on a small scale behind a 2 dimensional sheet cavity can explain that the stagnation pressure (at the location P of Fig. 6.4 is not present (although it is also very difficult to measure).

6.2.1 Shedding frequencies

A feature that can be measured in this complex flow is the shedding frequency. This shedding frequency is generally more pronounced at strong , more or less two-dimensional, cavitation, where the re-entrant flow is reaching the leading edge. When the shedding is caused at the leading edge the relevant length scale is the cavity length l_{cav}. The shedding frequency is therefore often made non-dimensional as the Strouhal number:

$$St = \frac{f * l_{cav}}{V} \tag{6.1}$$

where f is the shedding frequency (in HZ) and V is the incoming velocity.
When the re-entrant jet reaches the leading edge of the cavity and causes shedding there, the situation is as given in Fig. 6.4. In that case some sort of theoretical shedding frequency can be derived. The flow velocity at the cavity surface can be estimated assuming potential flow as $V\sqrt{1+\sigma}$. Assuming that the re-entrant flow has the same velocity, the time required for one shedding is $2 * \frac{l_{cav}}{V(\sqrt{1+\sigma})}$. The theoretical Strouhal number in that case is $St = 0.5\sqrt{1+\sigma}$. [1] A weak point is the assumption that the shedding takes place over the maximum cavity length. When the real shedding occurs over a fraction of the maximum cavity length, this theoretical Strouhal number is not changed when the length scale of the cavity length l_{cav} is replaced by an arbitrary shedding length l_{shed}, as long as in the definition of the Strouhal number the same length scale l_{shed} is used. However, when the Strouhal number remains based on the maximum cavity length and the relevant length is the shedding length, this results in an *increase* of the theoretical Strouhal number by the ratio $\frac{l_{cav}}{l_{shed}}$.

Another questionable assumption in the determination of the theoretical Strouhal number is the velocity of the re-entrant flow. Very little is known experimentally about this value. It seems reasonable that viscosity will slow the re-entrant flow down. In that case the time required for one shedding increases and the Strouhal number *decreases.*

For two dimensional configurations a value of the Strouhal number of around 0.25 has been reported (e.g. [10], [3]), but values of 0.11 to 0.40 have also been reported [2]. A difficulty is the definition of the cavity length, which is inherently unsteady when shedding occurs. In most cases the maximum cavity length was taken. Foeth ([16]) found no relation between the cavity length and the cavitation index and for his cavity shape (Fig. 6.12). The Strouhal number (again based on the maximum cavity length) he found was less than 0.2. However, that was on a three-dimensional cavity , where multiple sheddings occurred during a cycle, as will be discussed next.

6.2.2 Shedding behind a three-dimensional sheet cavity

In three dimensional cavities the shape of the trailing edge affects the convergence of the re-entrant flow. This leads to a strong spanwise component of the re-entrant flow (called side entrant jets by Foeth in [16]). The mechanism of shedding, as described above, may become more complex, as is shown in the high speed

[1] There is something to say for the assumption that the velocity of growth after the shedding goes with velocity V instead of with $V\sqrt{1+\sigma}$. In that case the shedding time is $T = \frac{l_{cav}}{V} * (1 + \frac{1}{\sqrt{1+\sigma}})$

[2] Note that Arndt mentions a theoretical coefficient of 0.25 instead of 0.5, but the argument was not given in [3]

Figure 6.14: High Speed Video:Shedding of cloud cavitation behind a three-dimensional sheet cavity in steady inflow.

video of Fig. 6.14.

In this case the cavity was made three dimensional, so that the shape of the trailing edge was no longer a line, but a curve. This was done using a twisted foil, which had the heaviest loading in the centerline and very little or no loading at the tunnel walls. This arrangement shows more clearly the effect of converging re-entrant flow. The shedding mechanism as described above occurs repeatingly. After a shedding in the cenbterline the two resulting cusps also generate shedding and so on. At the same time the shedding does not involve the whole re-entrant flow. A diverging re-entrant flow persists after a shedding, so the complete shedding period of the cavity contains multiple local sheddings. The fact that the shed cavity contains vorticity is illustrated by the complex vortex pattern, which is visible in between of the local sheddings. These connecting vortices become also visible because they tend to cavitate in the core.

Figure 6.15: High Speed Video:Shedding of cloud cavitation at the tip of a rectangular foil. Courtesy: Gongzheng Xin, CSSRC, Wuxi,China

This generally results in a very complex shedding pattern, in which repeatable structures can only distinguished with difficulty. An example is the shedding pattern of a sheet cavity at the rectangular tip of a foil. (Fig. 6.15). The inflow is steady.

The regular pattern is sharply illustrated by a single picture out of this collapse: Fig. 6.16. This picture also shows that in this case roughness has been used to generate sheet cavitation. At the leading edge the sheet still consists of local spots, which merge further downstream. Due to this merging the cavity behaves as a single sheet (for a discussion on these phenomena see also [59].

A three-dimensional cavity can be made completely steady at the trailing edge when the re-entrant flow is deflected and does not cause shedding. This is e.g. the case on a propeller blade in Fig. 6.17. At the inner radii of the sheet a triangle is visible. This is the re-entrant flow, which reflects against the trail-

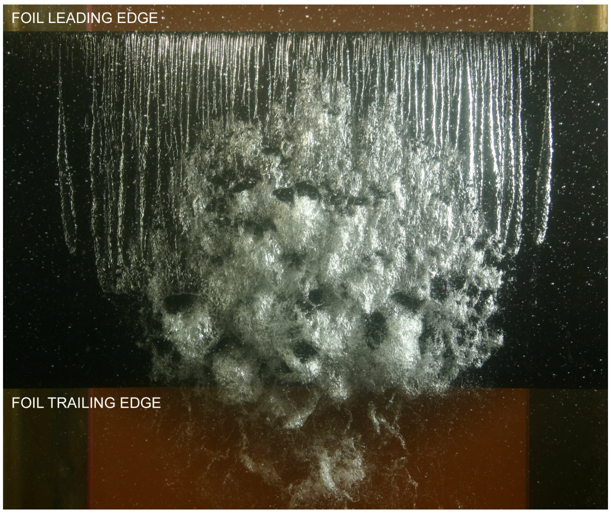

Figure 6.16: Regular but complex three dimensional shedding of a sheet cavity on a foil in steady inflow

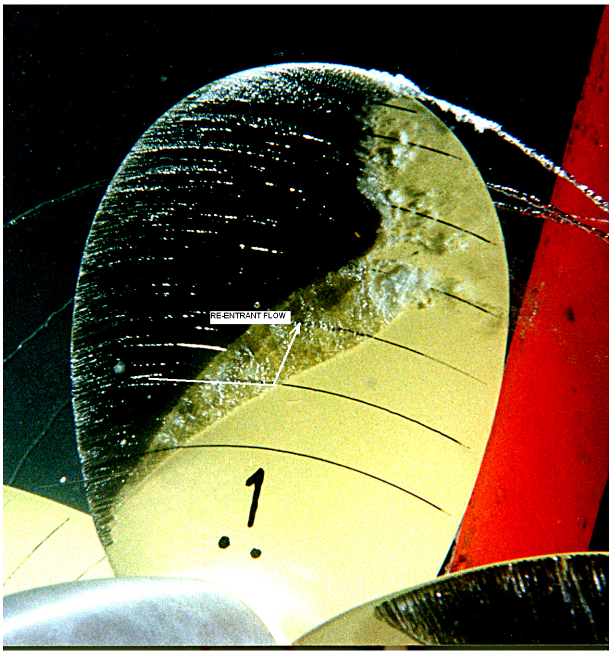

Figure 6.17: Steady sheet cavitation at the inner radii of a model propeller with visible reentrant flow

ing edge and therefore moves outward. At the outer radii the collected re-entrant flow causes shedding, but at the inner radii the sheet is stable. //

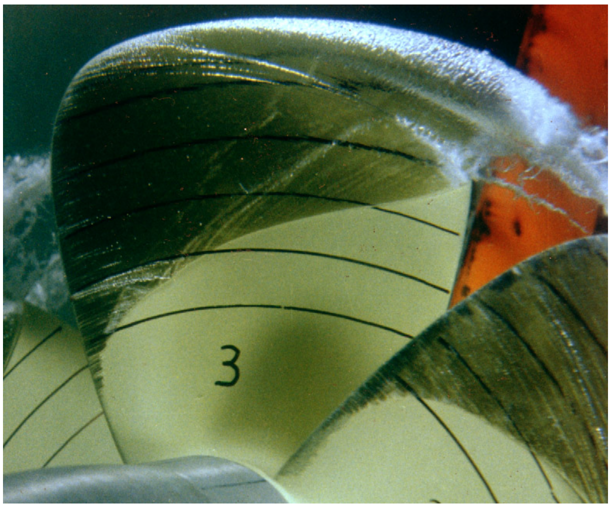

Figure 6.18: Smooth and steady sheet cavity on a model propeller blade in uniform flow

An extreme example of a stable sheet without shedding is shown in Fig. 6.18, where the re-entrant flow is fully absorbed by the cavitating tip vortex.

6.2.3 Unsteady Sheet cavitation

The shedding of sheet cavitation generally is very complex. Fortunately the natural shedding of sheet cavitation in steady conditions is not very erosive in many conditions. An example is the shedding of cloud cavitation behind an irregularly shaped sheet cavity on a ship propeller (Fig. 6.19

The sheet cavity on this propeller extends from the leading edge, which is not completely visible. The trailing edge of the sheet has some strong curvatures due to imperfections of the propeller geometry. The sheet varies in time with the blade position, but the sheet is relatively steady in comparison to the natural shedding behind the sheet. Thee natural shedding at the corner of the sheet trailing edge is strong and almost periodical. This is due to a local converging re-entrant flow at the location of the maximum length of the sheet. However, this type of natural shedding is generally

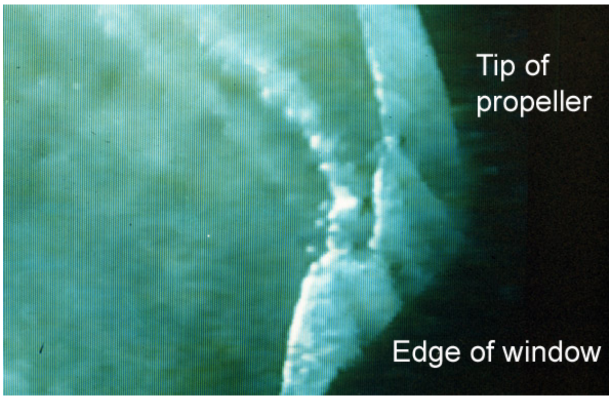

Figure 6.19: Shedding behind an irregular sheet cavity on a full scale propeller

not erosive. Erosivity is strongly enhanced by unsteady conditions, which can enhance the shedding process and the implosion. This will be discussed in a separate chapter. Now only some physical features of unsteady behavior will be mentioned.

A varying angle of attack causes an unsteady cavity on the foil. This means that the sheet cavity has a period of growth and a period of decline. Consider the situation as in Fig. 6.4. When the cavity grows and the velocity of point P is about equal to the flow velocity, the re-entrant jet disappears, because the pressure in the moving stagnation point is also equal to the vapor pressure. In that case the re-entrant jet disappears. Similarly, when the cavity contracts, the strength (thickness) of the re-entrant flow will increase and the shedding process will become more violent.

This is illustrated in the following case of an unsteady sheet cavity on a foil. The angle of attack of the inflow of this observed foil was varied using another oscillating foil upstream of the observed foil, which remained steady. The cavity observed is a very smooth sheet cavity on the steady foil. The top view is shown in the video of Fig 6.20 (Chalmers University, Department of Naval Achitecture and Ocean Engineering, project 2096).

Figure 6.20: High Speed Video:Oscillating sheet cavity. Top view. (courtesy Goran Bark, Chalmers University, Sweden)

The sheet cavity grows when the angle of attack increases and in this stage the re-entrant flow is absent or weak. The trailing edge of the cavity remains smooth and transparent. When the angle of attack begins to decrease the growth of the cavity stops and the re-entrant flow emerges. The re-entrant appears when the growth stops and becomes strong when the cavity begins to shrink. This results in a very strong re-entrant flow neat the trailing edge of the sheet, which is converging because of the shape of the trailing edge. The sharp convergence in the center of the sheet is clearly visible. Note that the collision of re-entrant flow begins in the rear end of the cavity. The re-entrant flow at the sides has a *downstream* component due to the angle of the cavity trailing edge. this flow will also end up in the rear end of the cavity. The colliding re-entrant flow cuts through the cavity, separating the two halves of the cavity.// It is remarkable that the implosion of these two halves is not very violent. Violent implosion occurs in the two clouds which are formed at the centerline. The violence of the implosion is evident

Figure 6.21: High Speed Video: Oscillating sheet cavity. Side view. (courtesy Goran Bark, Chalmers University, Sweden)

from the rebound which occurs. This implosion is assumed to be erosive, but the precise mechanism there is not yet well understood.

The collision of the re-entrant jet causes an upward flow in the centerline of the sheet cavity. The strength of this upward flow is illustrated by the side view of the same oscillating cavity, as shown in Fig. 6.21.

The vertical velocity is highest near the trailing edge of the sheet. This is also the region of the violent collapse. So it seems that the main collapse, and subsequent risk of erosion, comes from the bubble cloud which is generated by the colliding re-entrant flow. This has to be investigated further. In this simplified case the cloud generated by the upwards directed re-entrant flow is separated from the implosion of the two glassy parts on each side of the cavity. It makes the events involved more clear, but it should be kept in mind that generally the phenomena are much more complex.