Direct Calculations of Cavitating Flows by the Space-Time CE/SE Method

Jian-Rong Qin¹, S.T. John Yu²
Zeng-Chan Zhang,³ Ming-Chia Lai⁴
Mechanical Engineering Department
Wayne State University
Detroit, MI 48202

Abstract

This paper reports one- and two-dimensional simulations of cavitating flows by the Space-Time Conservation Element and Solution Element (CE/SE) method. A continuum cavitation model based on specifying the speed of sound of two-phase flows is employed. The CE/SE method is a viable CFD method for flows at wide range of Mach numbers. The method is explicit and is suitable for time accurate simulations. Moreover, without using a Riemann solver or a reconstruction procedure, the logic and operation count is simple and efficient for sharp resolution of evolving liquid/vapor interfaces. To validate the present model, three cavitating flows are simulated: the one-dimensional simulation of the water-hammer effect, flows over a hydrofoil, and flows through a high-pressure fuel injector. Numerical results show salient features of cavitations commonly observed in experiments, including, reentrant jet, hydraulic flip, and cyclical cavitations. The numerical results compare favorably with previously reported data.

1. Introduction

Many researchers investigated cavitations by numerical simulations [4-10]. The challenges are twofold: (1) viable modeling for complex flow physics involved, and (2) development of robust and accurate numerical methods for the unsteady two-phase flows.

Owing to the complex physics involved in the cavitating flows, in spite of many excellent studies, the underpinning flow physics of cavitation is still not fully understood. Flow features of cavitating flows are liquid/vapor phase change, high gradients of flow variables, and unsteadiness. When cavitations take place, bubbly clouds are often observed in the lee of the cavity. The clouds interact with vortices and form vortex cavitations. Both cavitation cloud and vortex cavitation consist of many bubbles of different sizes. The bubble clusters grow, collapse, interact with each other, and exchange mass, momentum, and heat with the surrounding liquid, and the physical processes are extremely complex.

To simulate cavitations by resolving each tiny bubble is beyond the computational capabilities currently available. Therefore, viable cavitation models, which simulate salient features of cavitations macroscopically, must to be employed. Many approaches exist. In this present paper, a simple continuum method is adopted.

We treat cavitating fluid as a homogeneous mixture of liquid and vapor rather than identifying the liquid/vapor interface. The interface is inferred from the value of the mixture density (or a void fraction). No wake model is required in the lee of cavities. Since we don’t need to calculate for a distinct interface, time-accurate simulation of evolving cavitations is efficient.

We use a pseudo-density in the model. The value of the pseudo-density varies between the liquid and vapor extremes. To close the system, a constitutive relation is

¹ Senior Engineer, Siemens Automotive, Email: Michael.qin@at.siemens.com
² Associate Professor, AIAA member, Email:styu@me1.eng.wayne.edu, http://141.217.13.61/
³ Research associate, AIAA member, Email:zhangzc@me1.eng.wayne.edu
⁴ Professor, AIAA member, Email:lai@me1.eng.wayne.edu
employed to relate the pseudo density to other variables. Delannoy and Kueny [7] used a sine function between pressure and pseudo density, and obtained encouraging results. Chen and Heister [6] suggested a model based on the equilibrium condition between the two phases by introducing a source term to the constitutive relation. The stiff source enforces the equilibrium indirectly. In more complex modeling works, Kubota et al. [8] constructed a sub-grid-scale model to connect pseudo-density to size distributions of bubbles. This model considered bubble dynamics by using the Rayleigh-Plesset [3] equation. Their results showed a strong interaction between cavitations and large-scale vortices. However, the assumption of a constant number of bubbles in the model is difficult to be justified. In applying the continuum methods, most researchers assumed that liquid and vapor share the same momentum field, i.e., no velocity difference between liquid and vapor is allowed.

With regards to numerical methods, several difficulties exist in predicting cavitating flows. In an engineering device where cavitations occur, most of fluid is incompressible. However, the density of the bubbly mixture changes greatly with respect to the pressure fluctuations. Thus, the employed numerical method must be versatile to calculate both incompressible flows and highly compressible flows. This is a major problem for many numerical methods.

The tremendous density difference between the vapor and liquid requires high spatial resolution by the numerical schemes employed. Since cavitationss are in general transient, the numerical method must be time-accurate. In addition, the numerical method must be robust to accommodate complex cavitation models, which usually are sub-modules in the flow solver.

Although advanced CFD methods have been successfully used to simulate single-phase flows, great difficulties exist in applying these methods to cavitations. Kubota et al. [8] used a finite-difference projection method. Although a fourth-order derivative term was added to stabilize the calculation, very small time steps have to be used, such that CFL < 0.25, to ensure numerical stability. Schmidt [9] employed a finite-volume upwind scheme, i.e., the Cavalry code. Numerical instability was also an issue.

In the present work, we use the CE/SE method [1,2], which is substantially different from traditional CFD methods in both concept and methodology. It is based on enforcing space-time flux conservation directly without using any derived property of the conservation laws, e.g., the characteristics decomposition. The introduction of different definitions for solution element (SE) and conservation element (CE) is novel. The computational domain is divided into many SEs, while the space-time flux conservation is enforced locally and globally over a single CE and clusters of CEs. Moreover, the spatial derivatives of flow variables are treated as unknowns and march hand in hand with the primitive flow variables. The resultant method has remarkable abilities in resolving discontinuity interfaces and wave motions.

The CE/SE method also possesses unique features important to model cavitations. The most notable is the method’s capabilities to handle flows at wide range of Mach numbers, i.e., from incompressible flow limit to high-speed compressible flows. Note that no preconditioning to the governing equations is needed. Moreover, the CE/SE method is a time-accurate method for unsteady flows, which is imperative for cavitations. Because the method is explicit, parallel computation is straightforward. In this conference, we have another paper AIAA 2001-0140, in which we report our experience of parallel computation on a low-cost Beowulf cluster using the CE/SE method.

We remark that a successful CFD simulation for cavitations must be a synergy of a robust and accurate CFD method and a viable cavitation model. Owing to the dual requirements, in an unsatisfying CFD calculation, the source of errors cannot be clearly identified.

The objective of the present paper is to assess the envelop of necessary complexity of the cavitation models employed, based on the use of a very accurate CFD method. To this end, we adopted a simple homogeneous equilibrium model based on a specified function for the speed of sound of the liquid/vapor mixture.

The rest of the paper is organized as follows. The mathematical model is illustrated in Section 2. In Section 3, a brief account of the CE/SE method for the Navier Stokes equations on unstructured meshes is provided. Section 4 reports numerical results of one- and two-dimensional cavitations. The numerical accuracy is validated through comparison with previously reported experimental results. We then offer concluding remarks and provide cited references.

2. Mathematical Model

The two-dimensional governing equations for cavitations flows can be written in the following conservative form:
\[
\frac{\partial U}{\partial t} + \frac{\partial E}{\partial x} + \frac{\partial F}{\partial y} = 0,
\]

(2.1)

where the flow variable vector \( U \) and flux vectors \( E \) and \( F \) are given by

\[
U = \begin{bmatrix} \rho \\ \rho u \\ \rho \nu \end{bmatrix}, \quad E = \begin{bmatrix} \rho u \\ \rho u^2 + p - \tau_{uv} \\ \rho \nu u - \tau_{uv} \end{bmatrix}, \quad F = \begin{bmatrix} \rho \nu \\ \rho \nu u + p - \tau_{uv} \end{bmatrix},
\]

(2.2)

\( \rho, u, \nu, \) and \( p \) are mixture density, \( x \)-component velocity, \( y \)-component velocity, and pressure, respectively. The \( \tau \)'s are viscous stresses:

\[
\tau_{ux} = \frac{2\mu}{3} \left( 2 \frac{\partial u}{\partial x} - \frac{\partial v}{\partial y} \right), \quad \tau_{vy} = \mu \left( \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} \right).
\]

(2.3)

Here \( \mu \) is viscosity of the liquid/vapor mixture, which is assumed to be

\[
\mu = (1 - \alpha)\mu_l + \alpha\mu_g,
\]

(2.4)

\( \mu_l \) and \( \mu_g \) are, respectively, liquid and gas viscosity, and \( \alpha \) is the void fraction of the mixture. Equation (2.4) shows that the mixture viscosity is proportion to the void fraction. We note that Eq. (2.4) is not a comprehensive model for mixture viscosity. This is an unresolved issue in formulating the viscosity of cavitating flows in the continuum approach.

The Jacobian matrices of the inviscid convection terms in Eq. (2.1) are

\[
\frac{\partial E}{\partial U} = \begin{bmatrix} 0 & 1 & 0 \\ 0 & 2u & 0 \\ -u \nu & \nu & u \end{bmatrix},
\]

(2.5)

\[
\frac{\partial F}{\partial U} = \begin{bmatrix} 0 & 0 & 1 \\ -u \nu & \nu & u \\ 0 & 2v & 0 \end{bmatrix}
\]

(2.6)

where \( a \) is the acoustic speed of the mixture and we assume

\[
\frac{dp}{d\phi} = a^2.
\]

(2.7)

A homogeneous equilibrium cavitation model is employed. The mixture density in the above equations is defined as

\[
\rho = \alpha \rho_v + (1 - \alpha)\rho_l,
\]

(2.8)

where \( \alpha, \rho_v, \) and \( \rho_l \) are the void fraction of the mixture, gas density, and liquid density, respectively.

Since pressure is not an unknown, to close the model, we need to adopt a constitutive relation for pressure to be related to density. To this end, we specify the acoustic speed of the two-phase fluid in cavitations based on Wallis’ formulation [11]:

\[
a = \sqrt{\frac{\alpha \rho_v + (1 - \alpha)\rho_l}{\rho_v a_v^2 + \rho_l a_l^2}}
\]

(2.9)

where \( a_v \) and \( a_l \) are the acoustic speed of vapor and liquid, respectively. Figure 2.1 shows the speed of sound as a function of the void fraction. We remark that the speed of sound reach a minimum when void fraction is about 50%.

Fig. 2.1: The Speed of sound of bubbly flows.

In the CFD calculations, constant acoustic speed is assumed for pure liquid and gas because temperature doesn’t change much in most cavitation conditions. For gas flows, constant acoustic speed may not be a good assumption. However, for cavitations over hydrofoils and through fuel injectors, only small regions are fully occupied by gas. In these regions, we assume constant acoustic speed to simplify the model.

Substituting Eqs. (2.8) and (2.9) into (2.7), we obtain an ordinary differential equation for pressure as a function of the void fraction. Integrating the ODE, we obtain the solution of pressure as:

\[
p = p^s + \beta \cdot p^s \cdot \log \left[ \frac{\rho_v a_v^2 \cdot (\rho_v + \alpha \cdot (\rho_g - \rho_v))}{\rho_v a_v^2 \cdot (\rho_v + \alpha \cdot (\rho_g a_v^2 - \rho_v a_v^2))} \right]
\]

(2.10)

where \( p^s \) is a constant given by

\[
p^s = \rho_v a_v^2 \cdot \rho_l a_l^2 \cdot (\rho_g - \rho_v) \]

(2.11)

Figure 2.2 shows the constitutive relation \( p = p(\rho) \) specified by Eqs. (2.10,11), which are used to determine \( p \) based on the calculated \( \rho \) in time marching. Note that
pressure is plotted in logarithmic scale, i.e., small change in density incurs significant pressure variation, which in turn drives fluid motions. In CFD calculations, the \( p = p(\rho) \) relation must be carefully treated for numerical stability.

We rewrite Eq. (2.1) by using subscript indices:

\[
\frac{\partial U_m}{\partial t} + \frac{\partial (f_m - F_m)}{\partial x} + \frac{\partial (g_m - G_m)}{\partial y} = 0
\]  

(3.1)

where \( m = 1, 2, \) and \( 3 \) for the continuity equation and the two momentum equations. Here \( f_m \) and \( g_m \) are the inviscid fluxes, which are functions of \( U_m, F_{vm} \) and \( G_{vm} \) are the viscous fluxes, which are functions of \( U_{vm}, U_{vm} \) and \( U_{ym} \). Let \( x_1 = x, x_2 = y, \) and \( x_3 = t \) be the coordinates of a three-dimensional Euclidean space \( E_3 \). The integral counterpart of Eq. (3.1) is

\[
\oint_{S(V)} H_m \cdot ds = 0
\]  

(3.2)

where \( H_m = (f_m, g_m, U_m) \) are the space-time current density vectors of mass, \( x \)-momentum, and \( y \)-momentum, respectively. \( S(V) \) is the boundary surface of a space-time region \( V \) in a Euclidean space \( E_3 \). The above flux vector \( H_m \) can be decomposed into inviscid and viscous parts:

\[
H_m = h_m - H_{vm}
\]  

(3.3)

Here \( h_m = (f_m, g_m, U_m) \) and \( H_{vm} = (F_{vm}, G_{vm}, 0) \) are the inviscid and viscous fluxes, respectively.

In two spatial dimensions, triangular mesh is used to perform space-time integration, as shown in Fig. 3.1(a). The grid points are located at centers of triangles. At each mesh node, three conservation elements (CEs) and one solution element (SE) are defined in connection with its three neighbors. For example, at point \( G \), three CE\((l) (l = 1, 2, 3) \) are the quadrilateral cylinders  
\[\text{EFGDE'}', \text{ABGF'A'B'}', \text{CDGBC'D'G'}', \text{and CDGBC'D'G'}, \text{as shown in Fig. 1(b)}. \]  

The SE is the union of four planes ABCDEF, \( G'G'G' \), \( G'G'G' \), \( G'G'G' \), and \( G'G'G' \) and their immediate neighborhood.

By using the first-order Taylor series expansions, the inviscid flux variables \( h_m = (f_m, g_m, U_m) \) are assumed to be linearly continuous inside each \( SE(j, n) \) in the space-time domain, where \( j \) indicates the \( j^{th} \) element and \( n \) is the time step. Note that there is only one spatial index because the mesh is unstructured. We then enforce the local flux balance over each conservation element by

\[
\int_{S(V)} h_m \cdot ds = 0
\]  

(3.4)

The detail derivations and formulations can be found in the following references on the CE/SE methods: Chang [6], Wang and Chang [25], Yu and Chang [26], and Zhang et al. [27].

The procedure of integration of viscous flux through surface \( \text{ABA'B'} \) is similar. However, more effort is needed for the calculation of the viscous flux through the surfaces \( \text{GBG'B'} \) and \( \text{GF'G'F'} \). This is because the two surfaces belong to the solution element associated with point \( G \), where flow variables are the unknowns. Therefore, we have to solve a set of coupled nonlinear equations.

For the CE/SE method using an unstructured mesh, a dual mesh is used to march in time. In this case, the surfaces \( \text{GBG'B'} \) and \( \text{GF'G'F'T} \) also belong to the SE of point \( G \). Thus we can use the flow variables \( U_m, U_{mx}, U_{my}, \) at point \( G' \) instead of \( G \) to calculate the viscous flux.

Since point \( G \) is located at previous time step, we can avoid solving nonlinear equations for \( U_{mx} \) and \( U_{my} \). In our calculation, we also tried to use the flow solutions at \( A' \) and \( G' \) to represent the variable values at the centroids of the lateral surfaces \( \text{AFF'A'}, \text{ABB'A'}, \text{GBB'G'} \) and \( \text{GFF'G'} \) for calculation of viscous fluxes.
This treatment further simplifies the marching procedure, and also stabilizes the computation. The last treatment was used in the following simulations.

![Diagram](image)

Fig. 3.1 A schematic of the CE/SE scheme: (a) triangle mesh in two spatial dimensions; (b) definitions of the CEs and SEs

### 4. Results And Discussions

Three cases will be presented: (1) one-dimensional calculations of the water hammer effect, (2) flows over a NACA0015 hydrofoil, and (3) flows through a fuel injector.

#### 4.1 Water Hammer Effect

This flow was originally reported in [10]. Here, the calculation is initialized by a pipe flow at a steady state. When $t = 0$, an upstream valve is suddenly closed. Due to the flow inertia, the liquid continues to flow in the same direction, and a vacuum region is formed near the closed valve. The low pressure of the vacuum region imposes an adverse pressure gradient to the pipe flow. Eventually, the liquid flows in the reverse direction back to the valve. The collapse of cavitations creates a pressure surge. As a result, fluid flow changes the direction and bounces away from the valve. The back and forth oscillations of the pipe flows is the classical water hammer effect.

Figure 4.1 shows snapshots of pressure distributions along the pipe with an interval of 0.185 seconds. The horizontal axis is the pipe length and the vertical axis is the pressure head in meters. A logarithmic (base 10) scale is used for the vertical axis from 0.1m to 100m. The arrows in this figure indicate the wave propagation directions.

Note that wave propagation speed varies considerably during the process because the speed of sound of two-phase flows is sensitive to the value of the void fraction. Here the speed of sound can be as low as 20m/s. In Fig. 4.1, nine snapshots similar to the one just above them have been omitted, when the wave is moving at the slowest speed.
Fig. 4.1 Snapshots of the pressure distribution along the pipe (interval time = 0.185 sec.)

Figure 4.2 shows the pressure history on the valve surface with (a) as the experimental data and (b) the numerical results by the CE/SE method. The numerical results of the pressure pick and the oscillation period compared favorably with the experimental data. Note that in the later stage of the flow development, experimental data showed more damped condition. This is due the use of the one-dimensional model in the simulations, which is inadequate to represent the flow friction largely caused by the boundary layer effects.

4.2 Cavitating Flows Over a Hydrofoil

Figure 4.3 shows the development of cavitations over a NACA0015 hydrofoil at an angle of attack ($\alpha$) of $8^\circ$. In the free stream, Reynolds number ($Re$) = $3 \times 10^5$, and the cavitation number is $\sigma = 1.5$, which is defined as

$$\sigma = \frac{p_{\infty} - p_*}{\frac{1}{2} \rho \bar{v}^2}$$

where $p_{\infty}$, $p_*$, $\rho$, and $\bar{v}$ are free-stream pressure, vapor pressure, liquid density, and free-stream velocity, respectively. Figure 4.3(a) and (b) are experimental and numerical results by Kubota et al. [8]. Figure 4.3(c) is our solutions by the CE/SE method. Shown in Fig. 4.3(b) and (c) are the contours of void fraction. The contour levels are from 0.1 to 0.9 with an interval of 0.1 except for the most outer line, which corresponds to void fraction of 0.01. The shape of the cavitating regions predicted by the CE/SE method compares well with Kubota’s results.
Figure 4.3. Cavitation over NACA0015 hydrofoil with an angle of attack $\alpha = 8^\circ$: (a) experimental observation [13]; (b) previous numerical results [13]; (c) present numerical results by the CE/SE method.

A reverse flow is predicted by the CE/SE method at the trailing edge of the bubble cloud. In this case, this re-entrant jet is weak and cannot cause the cavity to be separated. Nevertheless, the aft part of the cavity is highly unsteady and oscillates up and down at high

Fig. 4.4: Cavitation development over NACA0015 hydrofoil with the angle of attack $\alpha = 20^\circ$: (a) experimental observation, (b) numerical results by Kubota et al. (1992); (c) present numerical results by the CE/SE method.
Fig. 4.5: Cavitation development over NACA0015 hydrofoil with the angle of attack equal to 20°.
frequency. The entire cavity also shrinks and grows cyclically. Although not shown in this paper, the predicted unsteadiness agrees well with the experimental observations.

In experiments, a bubble cloud is observed at the downstream of the cavity. Refer to Fig. 4.3(a). This phenomenon cannot be predicted by the present model, Because of the equilibrium assumption, bubbles collapse immediately as the ambient pressure recovers.

Figure 4.4 shows snapshots of void fraction contours over the hydrofoil with the angle attack at $20^\circ$. In this case, the Reynolds number $Re = 3 \times 10^5$, cavitation number $\sigma = 1.5$. The void fraction contour levels are from 0.1 to 0.9 with an interval of 0.1 except for the most outer line, which corresponds to void fraction of 0.01. Three bubble clouds are shown in Fig. 4.4: one at the leading edge, one at middle of the upper edge, and the last one near the trailing edge. The CE/SE results compared favorably with Kubota’s experimental data.

Figure 4.5 show a sequence of snapshots for the cyclic cavitations. From the trailing edge, a strong reverse flow can be seen along the top surface of hydrofoil. This re-entrant jet is an important phenomenon for cavitating flows and is believed to be the reason of cavity breakup.

In the middle of the cycle, a large-scale vortex coincides with the cavitation cloud can be observed. This clockwise vortex was originally separated from the upper leading edge. The vortex stagnates above the hydrofoil. The low pressure at the vortex center sustains the cavitations. Another small counterclockwise vortex at the trailing edge can be seen in the CE/SE results. This smaller vortex has been elusive when the projection method with very sophisticated cavitation model [8].

4.3 Cavitation Inside a Nozzle Injector

In modern fuel injection system for diesel engine, fuel cavitates due to high injection pressures. For these fuel injection systems, the following parameters are important: the needle lift (H/D), orifice inlet curvature (R/D), injection angle ($\beta$), injection pressure ($P_{in}$), and nozzle aspect ratio (L/D). Refer to Fig. 4.6 for the definitions. In this numerical test, we used 22,183 triangles for the computational domain as that in Fig. 4.7. The upper boundary is the inlet and the middle-right boundary is the nozzle exit. Since the geometry is symmetric, half of the injector is used as the computational domain. Accordingly, we impose symmetric boundary conditions to the left vertical boundary. The rest of the boundaries are non-slip wall. The diameter of the nozzle orifice is $D = 0.184$ mm. The cavitation number is near unity ($\sigma = 1$), defined as

$$\sigma = \frac{P_{in} - P_c}{P_{in} - P_b} \tag{4.2}$$

where $P_{in}$, $P_c$, $P_b$ are injection pressure, vapor pressure, and back pressure, respectively.

Fig. 4.6: A schematic of the VCO nozzle tip and the design parameters investigated in this study

Figure 4.7 shows void fraction contours inside the nozzle with sharp orifice inlet. The injection pressure is 100 bar. The ambient pressure is 5 bar. The contour interval levels are 0.1 to 0.9 with the interval of 0.1. The outer most contour of 0.01 is also drawn. As shown in Fig. 4.7, cavitation incepts at the sharp inlet corner and extends downstream. Simultaneously, a re-entrant jet occurs in the lee of the cavity. After the cavity develops to its maximum size and reaches the nozzle exit, the re-entrant jet forces the cavity to separate. As a result, a bubble cloud is generated downstream of the cavity. At the nozzle exit, the bubble cloud merges with a large vortex, rotates with the vortex, and exits the injector. When the re-entrant jet reaches the orifice inlet and occupies the entire upper part of the orifice, the external pressure propagates into the orifice. As a result, the nozzle flow flips back to a non-cavitation mode. Subsequently, The internal flow separates at the nozzle inlet and does not reattach. No cavitation can be observed afterwards. This phenomenon is refereed to as “hydraulic flip”, which has been observed by many researchers.

6. Conclusion

Based on the CE/SE method, a Navier-Stokes solver has been successfully developed to model cavitations. A homogeneous equilibrium cavitation model is employed for the two-phase flows. The present method and the computer code have been validated by favorable
comparison between the CE/SE solutions and previously reported experimental and numerical data.

The solver was also used to study cavitations inside a nozzle injector. The “hydraulic flip” has been clearly simulated. This exercise shows that the CE/SE is a viable CFD method for cavitations. Perhaps, one does no need to use very sophisticated cavitation model when a highly accurate CFD method is employed.

References


