

## EVALUATION OF LAGRANGIAN, EULERIAN, ARBITRARY LAGRANGIAN-EULERIAN METHODS FOR FLUID-STRUCTURE INTERACTION PROBLEMS IN HCDA ANALYSIS

Y. W. CHANG, H. Y. CHU, J. GVILDYS, C. Y. WANG

*Argonne National Laboratory, 9700 South Cass Avenue, Argonne, Illinois 60439, U.S.A.*

The analysis of fluid-structure interaction involves the calculation of both fluid transient and structure dynamics. In the structural analysis, Lagrangian meshes have been used exclusively, whereas for the fluid transient, Lagrangian, Eulerian, and arbitrary Lagrangian-Eulerian (quasi-Eulerian) meshes have been used. This paper performs an evaluation on these three types of meshes. The emphasis is placed on the applicability of the method in analyzing fluid-structure interaction problems in HCDA analysis.

First, these three methods are evaluated and compared on the basis of their treatment of the fluid-structure interfaces, internal structures, and free surfaces. Sample calculation of a reactor vessel response under a postulated core disruptive accident condition is used to illustrate the application of the method and its advantages and disadvantages. It is shown that the treatment of the fluid-structure interaction with the Lagrangian method is very simple and the difficulty is in the elimination of the mesh distortions when fluid undergoes large displacements. Sliding lines can alleviate some of the distortion problems, but a multi-dimensional sliding is very difficult to be implemented into a Lagrangian code. Thus, for a typical reactor in the Lagrangian calculation, the upper wall deformation is often slightly overpredicted while the lower wall deformation is underpredicted. The treatment of fluid-structure interaction in the Eulerian method is more complex. Complicated procedure and flagging scheme for treating and identifying the irregular meshes at the fluid-structure interfaces are needed. This complicates the logic of the programming immensely, but the Eulerian method can predict the wall deformations rather accurately. In the arbitrary Lagrangian-Eulerian method, the fluid nodes at the fluid-structure boundary can be programmed to remain in contact with the structural surface while the interior fluid meshes can be moved in such a way to minimize the mesh distortions. Thus, both the mesh distortions and complicated flagging scheme at the fluid-structure interface are eliminated.

Next, spatial resolution, ease of coding, and the computer program CPU time are compared. Results of a shock tube problem are used to study the resolution of the numerical solution, the broadening of the wave front, and the dispersion of waves. Although Lagrangian solution gives a better resolution in describing the wave front, artificial viscosities are often needed for elimination of spurious oscillation at the discontinuities. Since the Eulerian formulation has internal dissipation terms, its calculation can be performed with no artificial viscosities. Thus, the Eulerian calculation can give a better resolution than the Lagrangian calculation when the problem involved discontinuities and the use of artificial viscosities.

## 1. Introduction

The analysis of fluid-structure interaction is a complex subject. Not only it involves the calculation of fluid transient and structure dynamics, but also the equations which govern the motion of the fluid and structure are coupled together. In reactor safety analyses, the problem is further complicated by the facts (1) that the structures in the reactor primary containment during a hypothetical core disruptive accident (HCDA) are subject to a series of shock waves emanating from the core where the rate of HCDA energy release depends on the motion of the structures, particularly for those which are adjacent to the core; (2) that the problem is highly nonlinear in both geometry and material laws; and (3) that the fluid region has finite boundaries in all directions. Thus, solutions of such fluid-structure interaction problems are often obtained with numerical analysis through the use of computer programs.

However, the arithmetic operation in the computer is both discrete and finite. It is necessary in the computer program to formulate the equations of interest in a discrete form and in the analysis to replace the physical system by a discretized system. In discretizing a physical system, the continuum is replaced by a group of meshes.

In the structural analysis, because of history dependence of the constitute equations, Lagrangian meshes have been used exclusively. However, in the fluid analysis, both Lagrangian and Eulerian meshes are used. In treating problems involved large displacements, both methods have some shortcomings. For example, excessive mesh distortions in the Lagrangian calculation can deteriorate the accuracy of the numerical result, while in the Eulerian analysis, complex computation procedures are needed for treating the material and boundary interfaces. Recently, an arbitrary Lagrangian-Eulerian (quasi-Eulerian) method has been suggested by many workers [1-4] as an alternate for the Lagrangian or Eulerian method. However, this method is not without drawbacks. For once the mesh point moves differently from the material particle, the convective terms appear in the formulation and the difficulties associated with the treatment of free surface may also appear.

The aim of this paper is to examine the salient features of the three types of meshes and their application to fluid-structure interaction problems in reactor safety analysis. The finite difference and finite element methods, which commonly used in the numerical analysis, will not be discussed. As pointed out by Belytschko [5] that both the finite element equations and Wilkins' equations for hydrodynamics may be considered as weak forms of the partial differential equations which can be lead to identical difference equations. The differences between these two methods appear to be in the formulation of the equations of motion and in the numbering of mesh points. In the finite difference method, the equations of motion are usually expressed in terms of the pressure gradients of the neighboring meshes and the mesh points are not numbered, whereas in the finite element method the equations of motion are formulated through the use of the intermediate nodal forces and the mesh points are numbered where the connectivity information is explicitly stored. However, these distinctions are no longer visible. For example, the finite difference technique used in YAQUI code [6] is similar to the nodal force technique in the finite element method and the mesh points in some recent developed finite difference programs are numbered.

In the following sections, Lagrangian, Eulerian, and Arbitrary Lagrangian-Eulerian methods are evaluated and compared on the basis of their effectiveness in the treatment of the fluid-structure interfaces, internal structures, and free surfaces. Spatial resolution, ease of coding, and the computer program CPU time are also compared.

## 2. Treatment of the Fluid-Structure Interfaces

Since the Lagrangian mesh moves with the material particle, the presence of a fluid-structure interface does not complicate the computation procedure. In general, two sets of mesh points must be used at the fluid-structure interface to account for the different motion between the fluid and structure. The motion of the two sets of mesh points are assumed to be independent to each other in the direction tangential to the interface, but in the normal direction they are assumed to move together, i.e. no separation of the fluid particle from the structural member is allowed. When an internal shell structure is present, it is usually placed at the Lagrangian mesh lines, and three sets of mesh points are employed. One set of mesh points belongs to the shell and the other two to the fluids, one on each side of the shell. Again, the motions of the three sets of mesh points are assumed to be independent to each other in the direction tangential to the shell surface, but in the normal direction they are forced to move together.

Analysis of fluid-structure interactions using the Eulerian meshes is more complex. The major difficulty is in the treatment of the irregular meshes at the fluid-structure interfaces. As the structure is displaced under the applied pressure loads, its boundaries will intersect the Eulerian mesh lines, creating irregular meshes of various shapes. Figure 1 illustrates some possible shapes of the irregular mesh in a two-dimensional Eulerian cell where velocities are defined at the center of the cell boundary. For convenience, we have chosen to use  $i, j$  subscripts to identify the variables.

One way of dealing with the irregular meshes is by use of the control volume technique to derive the conservation equations with respect to the partial cell which is actually occupied by the fluid. However, because of the structure movement, the shape of the irregular mesh is changing continuously. The procedure of the control volume technique becomes too cumbersome. An alternative to the control volume method is to treat the irregular meshes as if they were regular cells and to express the unknown interaction pressure at the fluid-structure interface in an equation form to satisfy the physical boundary condition required at the fluid-structure interface. If the fluid at the fluid-structure interface is assumed to be inviscid, the mathematical equation to satisfy the physical boundary condition is  $(\bar{V}_f - \bar{V}_s) \cdot \bar{n} = 0$ , where  $\bar{V}_f$  is the velocity vector of the fluid particle located at the midpoint of the interface,  $\bar{V}_s$  is the velocity of the structure at that point, and  $\bar{n}$  is the unit normal vector of the interface, pointing into the fluid region. This equation together with other equations obtained from the regular cells provide a sufficient set of equations for the determination of pressures in the fluid region.

Another difficulty is in the calculation of fluid velocities where fluid pressures of the neighboring cells may not always exist. At the fluid-structure interface cell, a part of the cell is occupied by the structure; so are the neighboring cells. For example, the pressure in cell  $(i+1, j)$  shown in Fig. 2 is undefined and the velocity  $u_{i+\frac{1}{2}, j}$  cannot be obtained from the momentum equation. Here one must use the mass equation to determine the velocity of the fluid at the cell boundary. If two neighboring cell pressures are undefined, one has to make a further assumption that one of the velocity components can be found from the fluid velocities inside the fluid region. Similarly, if three neighboring cell pressures are undefined, two of the velocity components are computed from the fluid velocities inside the fluid region. Thus, complex computation procedures must be developed for calculating velocity components of all types of irregular meshes as shown in Fig. 1.

In the arbitrary Lagrangian-Eulerian method, the mesh point of the computing cell can be made to move with the fluid, fixed in space, or to move in any arbitrary way. Thus, the computation procedures at the fluid-structure interface can be made simplified by allowing the computing cells to move together with the structure. However, this method is not without drawbacks. Because once the mesh point of the computing cell moves differently from the material particle, the convective terms appear in the formulation and the major characteristics of an Eulerian formulation retain.

### 3. Treatment of Internal Structures

Treatment of internal structures presents a problem in the Lagrangian analysis. This is because the fluid in the vicinity of an internal structure usually undergoes large distortions, particularly at the end of the structure where geometry discontinuity occurs. Although rezoning of meshes can reduce the severity of the mesh distortions, there is an upper limit on how far one can push this rezoning technique in the Lagrangian analysis. If rezoning is done at every few time steps, it means, essentially, that the fixed mesh of a Eulerian discretization is more appropriate. Sliding lines have been used quite successfully for treating fluid sliding on vessel wall. However, applying to internal structures, the sliding line must be able to change its direction at the end of the structure, which is very difficult to implement in a computer program.

Eulerian equations of hydrodynamics are ideal for treating problems involving large material distortions. Therefore, for problems containing internal structures and involving large distortions, the Eulerian mesh becomes more attractive. Since the arbitrary Lagrangian-Eulerian method has the option of using the fixed Eulerian meshes in the regions where the fluid is expected to move extensively, it is also suitable for treating problems involving internal structures. However, for both methods the computation procedures for treating internal structures are rather complex. This is because the fluids on both sides of the internal structure have different motions, and the field variables which are used to calculate the fluid motion must also have different values. Similarly at the end of the internal structure, another set of field variables is needed for the motion of the fluid located above or below the structure. In general, a cell can have up to four sets of field variables.

The advantages of using the Eulerian and arbitrary Lagrangian-Eulerian methods in the analysis of fluid-structure interactions involving internal structures can be seen from the following comparisons where computer code predictions obtained by Lagrangian, Eulerian, and arbitrary Lagrangian-Eulerian methods are compared with the experimental measurement. The experiment chosen for this purpose is the SRI flexible vessel test FV102 [7]. The apparatus used in the experiment is shown in Fig. 3. It consists of a flexible vessel, a core canister, a flexible core barrel (pure lead contained in a thin aluminum cylinder), a vessel cover, and a support platform. The profiles of the final vessel wall and core barrel deformations obtained with the computer code prediction and the experimental measurement are given in Fig. 4. As can be seen the agreement between the Eulerian and arbitrary Lagrangian-Eulerian calculations with the experiment is very good. However, in the Lagrangian calculation; the calculated deformations of the upper vessel wall are higher than the experimental data, whereas at the lower vessel wall, the Lagrangian results are lower than the experimental data. These differences can be attributed to the lack of a two-dimensional sliding capability in the Lagrangian code. Also, the use of sliding line allows the fluid to slide more freely in the code than in the experiment along the designated surfaces. As a result, more energy was

diverted to the axial direction in the form of coolant kinetic energy and less energy was directed to the radial direction. For the case studied, the sliding line at the inner surface of the core barrel was extended all the way from the bottom of the platform to the top of the water surface.

#### 4. Treatment of Free Surface

Free surface position in the Lagrangian calculation can be accurately determined if a Lagrangian mesh line is initially placed at the free surface. However, this is not the case in the Eulerian calculation where one must rely on the marker particle to determine the position of the free surface. This is also true in the arbitrary Lagrangian-Eulerian method where the mesh points have already been prescribed to move with the structural nodes. They can no longer move with the free surface in a Lagrangian manner. Therefore, marker particles are often needed in the arbitrary Lagrangian-Eulerian calculations.

The field variables are not defined outside the fluid region, but they are needed to carry out numerical calculations. For example, the velocity field in an empty cell adjacent to the surface cell is used to move the marker particles. Figure 5 shows the velocity field of a two-dimensional Eulerian mesh in the vicinity of a surface cell, where  $u_{i-\frac{1}{2},j+1}$ ,  $v_{i,j+\frac{1}{2}}$ , and  $u_{i+\frac{1}{2},j+1}$  are the unknown velocities. The basic idea in calculating the unknown velocities is to satisfy the mass equation and the tangential stress condition. For the case shown in Fig. 5, the unknown axial velocity  $v_{i,j+\frac{1}{2}}$  in the surface cell can be computed from the mass equation; the unknown radial velocities,  $u_{i-\frac{1}{2},j+1}$  and  $u_{i+\frac{1}{2},j+1}$  in the empty cell are computed from the condition that tangential stress vanishes near the free surface. The orientations of the free surface are similar to those fluid-structure interfaces shown in Fig. 1. Thus, the computation of the unknown velocities and the other field variables requires a complex procedure.

#### 5. Spatial Resolution

As mentioned earlier, once the mesh point moves differently from the material particle, the convective terms appear in the formalation. Thus, in the Eulerian and arbitrary Lagrangian-Eulerian methods, mass, momentum, and energy equations have convective terms. As is well known, the mass convection tends to introduce additional smoothing of solutions. Therefore, results obtained by the Eulerian and arbitrary Lagrangian-Eulerian methods can be expected to be somewhat more dispersive than the Lagrangian solution. However, in the analysis of problems involving discontinuities, artificial viscosities are often used in the Lagrangian analysis for elimination of the spurious oscillation at the discontinuities. On the other hand, the Eulerian and arbitrary Lagrangian-Eulerian methods already have internal dissipation terms built in the formulation, their calculations can be performed with no additional artificial viscosities. As a result, the Eulerian and arbitrary Lagrangian-Eulerian calculations can give a better resolution than the Lagrangian calculation if the problem to be solved involved the use of artificial viscosities.

To illustrate this, a shock tube problem is presented. The shock tube is 100-cm-long and 10-cm-dia with rigid walls and end caps. The tube was filled with ideal gas and divided into equal regions by a diaphragm. The internal energies and temperatures were the same on both sides; the density and pressure on the left side were twice the values on the right side. This problem can provide a sensitive test of the amount of broadening of discontinuities, and of the numerical noise which is introduced by the numerical method.

Figure 6 shows two pressure profiles obtained by the Lagrangian method at  $t = 500 \mu s$  after the diaphragm is removed: one without artificial viscosity and one with artificial viscosity. As can be seen the use of artificial viscosity in the Lagrangian method is necessary, if the spurious oscillation is to be eliminated. Figure 7 is the pressure profile obtained with the Eulerian method, where the Lagrangian solution with the use of artificial viscosity is also given. It can be seen that the Eulerian solution gives better spatial resolution than the Lagrangian solution.

#### 6. Computer Programming and CPU Time

Lagrangian formulation does not have convective terms and the fluid-structure interface is clearly delineated. Therefore, programming procedure is relatively simple. The treatment of the fluid-structure interface in the Eulerian method is rather complex. Complicated procedure and flagging scheme for treating and identifying the irregular meshes at the fluid-structure interface must be developed. This complicates the logic of the programming immensely. In the arbitrary Lagrangian-Eulerian method, the complex flagging scheme at the fluid-structure interfaces has been eliminated. But at the core gas-fluid interface and free surface, it still needs a flagging scheme to identify the orientation of the interface and a computational procedure to compute the movement of marker particles. Since an arbitrary Lagrangian-Eulerian calculation is usually performed in two steps: first Lagrangian and then Eulerian, considerable amount of programming work is needed.

The complexity of computer programming can be further seen from the CPU times required for one cycle of computation given below. For the shock tube problem, the CPU times for Lagrangian, Eulerian, and arbitrary Lagrangian-Eulerian methods are 0.34, 0.76, and 1.00 ms per mesh cycle, respectively. The ratio is approximately 1:2:3. For the flexible vessel problem, because of extensive calculations were needed at the fluid-structure interfaces, the CPU time for the Eulerian method becomes considerably longer. The CPU times are 0.97, 3.29, and 2.61 ms per mesh cycle, respectively, for the Lagrangian, Eulerian, and arbitrary Lagrangian-Eulerian methods. All computer runs were performed on IBM 370 computer model 195.

#### 7. Concluding Remarks

It has been demonstrated that the Eulerian and arbitrary Lagrangian-Eulerian methods are more versatile than the Lagrangian method in the analysis of the fluid-structure interaction problem which contains internal structures. They can give a better solution than the Lagrangian method. It has also been shown that in the determination of fluid-structure interfaces, the Eulerian method needs a complex flagging scheme and complicated computing procedures. Therefore, it needs a longer CPU time. However, in the selection of method to use in the reactor safety analysis, other factors, such as the type of problem to be analyzed, the type of solution desired, and the time available to develop a computer program, must also be taken into consideration. If the problem to be analyzed involving coolant spillage, fluid cavitation, heat transfer, structure with perforated holes, and two-phase flows, it is advantageous to use an Eulerian formulation.

#### 8. Acknowledgments

This work was performed in the Engineering Mechanics Section of the Reactor Analysis and Safety Division at Argonne National Laboratory, under the auspices of the U. S. Department of Energy.

References

- [1] HIRT, C. W., AMSDEN, A. A., COOK, J. L., "An Arbitrary Lagrangian-Eulerian Computing Method for All Flow Speeds," *J. of Comp. Physics*, pp. 227-253 (1974).
- [2] CHU, H. Y., "ALICE: A Hybrid Lagrangian-Eulerian Code for Calculating Fluid-Structure Interaction Transients in Fast Reactor Containment," ANL Report, to be published.
- [3] DONEA, J., FASOLI-STELLA, P., GUILIANI, S., "Lagrangian and Eulerian Finite Element Techniques for Transient Fluid-Structure Interaction Problems," Fourth Intl. Conf. on Structural Mechanics in Reactor Technology, San Francisco, California, USA, August 15-19, 1977, paper B 1/2.
- [4] BELYTSCHKO, T. B., KENNEDY, J. M., SCHOEBERLE, D. F., "Quasi-Eulerian Finite Element Formulation for Fluid-Structure Interaction," ASME paper 78-PVP-60, ASME/CSME Pressure Vessel and Piping Conference, Montreal, Canada (June 1978).
- [5] BELYTSCHKO, T. B., KENNEDY, J. M., SCHOEBERLE, D. F., "On Finite Element and Difference Formulations of Transient Fluid Structure Problems," Proc. Conf. Comp. Methods in Nucl. Eng., Charleston, South Carolina (April 1975).
- [6] AMSDEN, A. A., HIRT, C. W., "YAQUI: An Arbitrary Lagrangian-Eulerian Computer Program for Fluid Flow at All Speeds," LA-5100 (March 1973).
- [7] CAGLIOSTRO, D. J., ROMANDER, C. M., "Experiments on the Response of Rigid and Flexible Reactor Vessel Models to a Simulated Hypothetical Core Disruptive Accident," SRI report (November 1976).

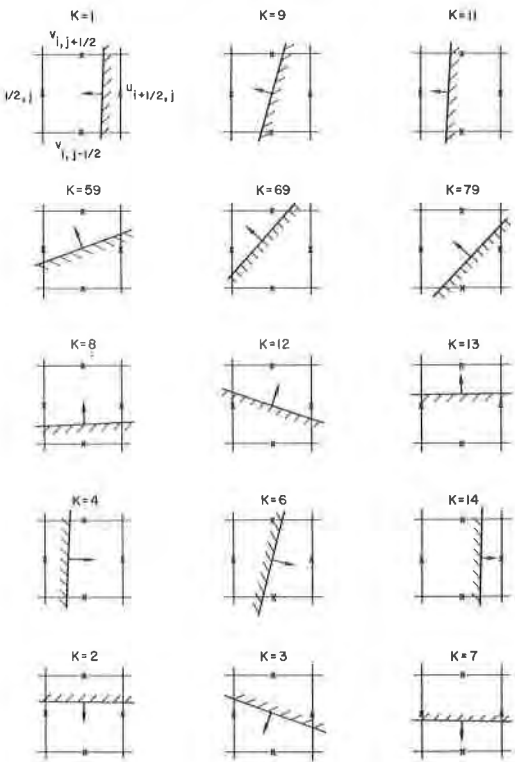


Fig. 1. Different Types of Irregular Meshes.

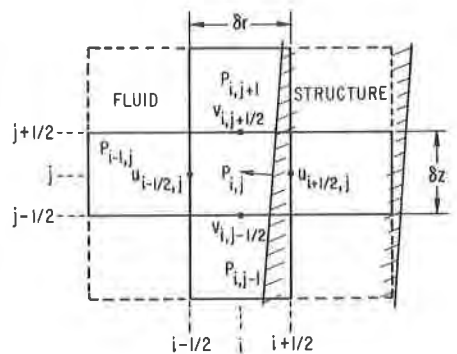
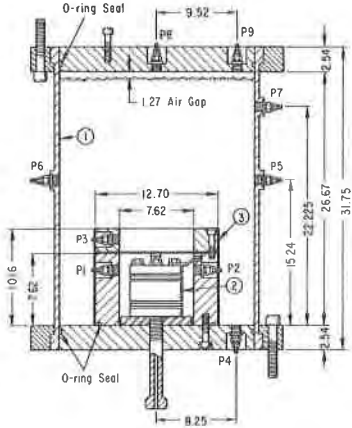


Fig. 2. Structure Boundary Cell with One Neighboring Pressure Undefined.



- ① FV102 Outer Cylinder, 20.0 I.D x 21.0 O.D Forged and Annealed Ni 200
- ② 8g 65/35 PETN-Microballoon Mix
- ③ 0.127 cm Aluminum Shell and 2.413 cm Lead Liner

All dimensions in cm

Fig. 3. Details of the SRI Flexible Vessel Test FV102.

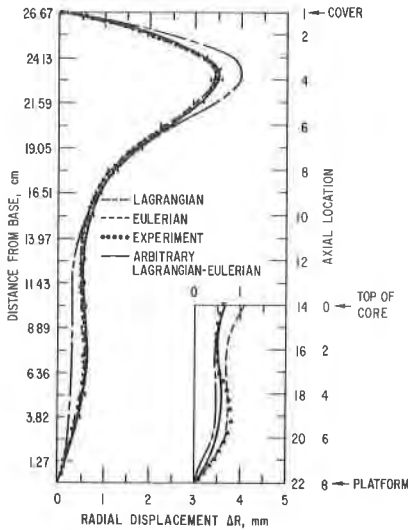


Fig. 4. Profiles of the Vessel Wall and Core Barrel Deformations.

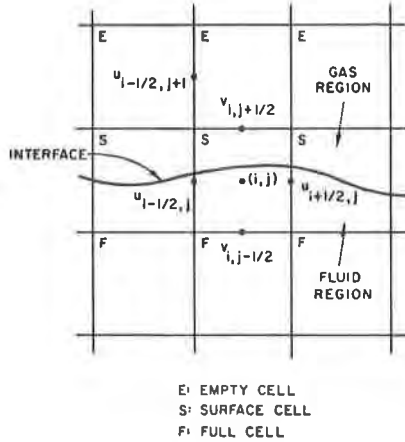


Fig. 5. Velocity Field in the Vicinity of a Surface Cell.

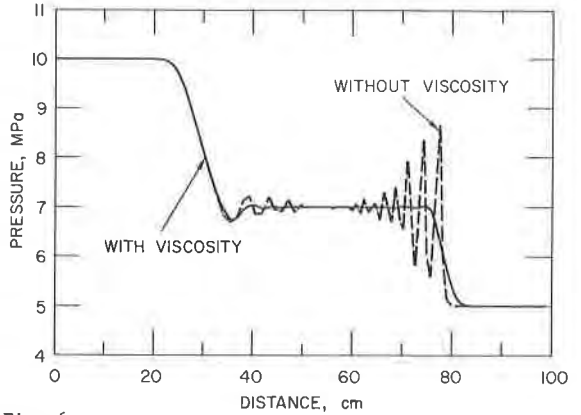


Fig. 6. Pressure Profiles Obtained by the Lagrangian Method at  $t = 500 \mu s$  After the Removal of the Diaphragm.

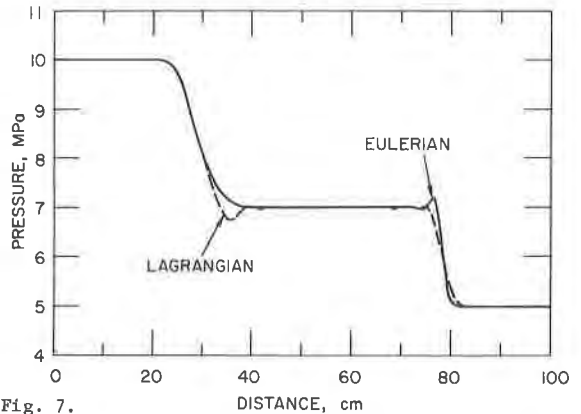


Fig. 7. Comparison of the Eulerian and Lagrangian Method Predicted Pressure Profiles at  $t = 500 \mu s$  After the Removal of the Diaphragm.