7. ANKARA INTERNATIONAL AEROSPACE CONFERENCE
11-13 September 2013 - METU, Ankara TURKEY

AIAC-2013-018

FLOWFIELD DEPENDENT VARIATION (FDV) METHOD FOR FLUID-STRUCTURE INTERACTION PROBLEMS

Mohd Fadhli[1], Ashraf A. Omar[2] and Waqar Asrar[3]
International Islamic University Malaysia
Kuala Lumpur, Malaysia

ABSTRACT

In this study, Flowfield Dependent Variation (FDV) method is coupled with Arbitrary Lagrangian-Eulerian (ALE) method in order to solve one-dimensional fluid-structure interaction problems. FDV method is a mixed explicit-implicit numerical scheme where its implicitness is determined by several parameters that are dependent on the physical properties of the local flow. The advantages of FDV method in dealing with transition and interaction of complex flow motivated us to extend its capability to fluid-structure interaction problems. The combination of FDV and ALE method is discretized using finite volume method in order to give flexibility in dealing with complicated geometries. Several numerical tests have been conducted and the results are in good agreement with exact and available numerical solutions in the literature.

INTRODUCTION

In dealing with a domain which contains all speed flows with various physical properties, where the equation of state for compressible and incompressible flows are different, and where the transitions between laminar and turbulent flows are involved, very special and powerful numerical treatments are needed. The Flowfield Dependent Variation (FDV) method has been developed by Chung et al. (1999) in order to resolve such complex flow problem with a single numerical scheme. This method is a mixed explicit-implicit numerical scheme that changes its implicitness due to the physical properties of the local flow regions. This is done by several parameters dependent on the physical flow parameters (such as Mach number and Reynolds number) that act as weight factors between explicit and implicit scheme in the formulation of the FDV method.

The parameters, which are also called FDV parameters, are characterized into two categories, namely first and second order parameters. Besides determining the implicitness of the method, these parameters also have other significant characteristics. The first order parameters ensure the solution accuracy and second order parameters assist in the solution stability. Because of these parameters, the terms of fluctuation variables in the FDV formulation are automatically generated following the current physical phenomena and adequate numerical controls (artificial viscosity) are automatically activated according to the current flowfield. The numerical scheme of FDV equation itself will then adjust accordingly for every node based on the flow properties of different regions that coexist in the computational domain [Chung, 1999; 2002].

On the other hand, fluid-structure interaction problems such as airfoil oscillations, wing flutter, rotating propellers, reciprocating engines, flapping wings, etc. require dynamic meshes to be solved. This has attracted many researches to develop various kinds of moving grid interpolation techniques. Arbitrary Lagrangian-Eulerian (ALE) method [Donea, 2004] is probably the most popular moving grid interpolation technique in fluid and solid mechanics. Implementation of ALE method requires re-meshing formulation to update the computational meshes at each time step while at the same time avoiding severe mesh distortion and mesh entanglement. Due to the influence of re-meshing technique in the stability and accuracy of ALE method, many researches have enforced the Geometric Conservation Law (GCL) [Thomas, 1979; Guillard, 2000] when using this method.

[1] PhD student in a Mechanical Department, Email: fadhli@uthm.edu.my
[2] Prof. in a Mechanical Department, Email: aao@iium.edu.my
[3] Prof. in a Mechanical Department, Email: waqar@iium.edu.my

The advantages of FDV method in dealing with the transition and interaction of complex flows motivated us to extend its capability into the flow-structure interaction problems. The main objective of this study is to propose a new technique based on a combination of the FDV and ALE methods to solve such problems. Its application is focused for one-dimensional compressible inviscid flow problems in this paper. To the authors' knowledge, FDV theory has not been fully incorporated into dynamic mesh applications although some researches have utilized the FDV parameters as error indicator for adaptive mesh refinements [Heard, 2007]. The finite volume method is used for spatial discretization to add flexibility to the proposed method in dealing with complicated geometries. It is expected that the proposed method would provide a new technique of resolving accurately the interaction of arbitrary structures in arbitrary flow fields.

ARBITRARY LAGRANGIAN-EULERIAN (ALE) FORM OF FDV METHOD

The FDV formulation was derived by substituting Navier-Stokes equations into a special form of Taylor series that includes implicitness parameters (see for instance [Chung, 2002] for further details). Then, the finite volume form of FDV method for dynamic mesh application can be obtained by volume integration of the conservative variables ***U*** over a moving control volume $\Omega(t)$ as follows,

$$\int_{\Omega(t)} \frac{\Delta \boldsymbol{U}^{n+1}}{\Delta t} \mathrm{d}\Omega = -\int_{\Omega(t)} \left[\frac{\partial}{\partial x} (\boldsymbol{E}\Delta \boldsymbol{U})^{n+1} - \frac{\Delta t}{2} \frac{\partial^2}{\partial x^2} (\boldsymbol{EE}\Delta \boldsymbol{U})^{n+1} + \frac{1}{\Delta t} \frac{\partial \boldsymbol{Q}^n}{\partial x} \right] \mathrm{d}\Omega \tag{1}$$

where $\Delta \boldsymbol{U}^{n+1} = \boldsymbol{U}^{n+1} - \boldsymbol{U}^n$ and terms ***E***, ***EE*** and ***Q*** are

$$\begin{aligned} \boldsymbol{E} &= \Delta t (s_1 \boldsymbol{a} + s_3 \boldsymbol{b}) \\ \boldsymbol{EE} &= -\frac{\Delta t^2}{2} (\boldsymbol{a} + \boldsymbol{b})(s_2 \boldsymbol{a} + s_4 \boldsymbol{b}) \\ \boldsymbol{Q}^n &= \Delta t (\boldsymbol{F}^n + \boldsymbol{G}^n) - \frac{\Delta t^2}{2} (\boldsymbol{a} + \boldsymbol{b}) \left(\frac{\partial \boldsymbol{F}^n}{\partial x} + \frac{\partial \boldsymbol{G}^n}{\partial x} \right) \end{aligned} \tag{2}$$

respectively. In equation (2), ***a*** and ***b*** are convection and diffusion Jacobian matrices while ***F*** and ***G*** are the convection and diffusion flux vectors, which are known at time step *n*. Parameters s_1 to s_4 are the so-called FDV parameters. For inviscid flow problems, diffusion flux and Jacobian matrix can be ignored, hence the ***E*** and ***EE*** terms as well as the ***Q*** term in the above equations are reduced to,

$$\begin{aligned} \boldsymbol{E} &= \Delta t s_1 \boldsymbol{a} \\ \boldsymbol{EE} &= -\frac{\Delta t^2}{2} s_2 \boldsymbol{a}^2 \\ \boldsymbol{Q}^n &= \Delta t \boldsymbol{F}^n - \frac{\Delta t^2}{2} \boldsymbol{a} \frac{\partial \boldsymbol{F}^n}{\partial x} \end{aligned} \tag{3}$$

In particular, s_1 and s_2 are called the first and second order convection FDV parameters, respectively and are dependent to the minimum and maximum local Mach number, *M* as defined below.

$$\begin{aligned} s_1 &= \begin{cases} \min(r,1) & r > \alpha \\ 0 & r < \alpha \quad M_{\min} \neq 0 \\ 1 & M_{\min} = 0 \end{cases} \quad \text{with } r = \frac{\sqrt{M_{\max}^2 - M_{\min}^2}}{M_{\min}} \\ s_2 &= \tfrac{1}{2}\left(1 + s_1^{\eta}\right) \end{aligned} \tag{4}$$

Parameters α and η are user-specified small constants depending on the problem being solved. Typically α is chosen about 0.01, while η is chosen appropriately between 0.05 and 0.2. In addition, parameters s_3 and s_4 are evaluated similarly as in equation (4) except Reynolds number is used instead of Mach number. This study proposes to combine the FDV method with the Arbitrary Lagrangian-Eulerian (ALE) method. ALE method is a technique that combines Lagrangian and Eulerian descriptions of a continuum (i.e. fluid and solid) in one numerical scheme. By combining both kinds of description, a computational mesh can follow the moving structures in an arbitrary movement, while the fluid is still seen in a Eulerian way. The starting point in deriving an ALE type numerical scheme is the implementation of Reynolds transport theorem to the control volume $\Omega(t)$ where its boundary $\Gamma(t)$ moves with velocity V_m [Donea, 2004]. Applying the theorem to equation (1) yields,

$$\frac{1}{\Delta t}\Delta\left[\int_{\Omega(t)} \boldsymbol{U}\,\mathrm{d}\Omega\right]^{n+1} = -\int_{\Gamma(t)}\left[(\boldsymbol{E}\Delta\boldsymbol{U})^{n+1} - \frac{\Delta t}{2}\frac{\partial}{\partial x}(\boldsymbol{EE}\Delta\boldsymbol{U})^{n+1} + \left(\frac{1}{\Delta t}\boldsymbol{Q}^{n} - \boldsymbol{U}^{n}V_{m}\right)\right]\cdot \boldsymbol{n}\,\mathrm{d}\Gamma \qquad (5)$$

where the volume integral on the right hand side of equation (1) has been changed into a surface integral using the Gauss divergence theorem. Equation (5) is linearized as below,

$$\Delta\boldsymbol{U}_i^{n+1}[\Delta x_i]^{n+1} + \sum_{j=1}^{2}\left[\left(\boldsymbol{E}^{n} + \boldsymbol{EE}^{n}\frac{\partial}{\partial x}\right)\Delta\boldsymbol{U}_i^{n+1}\cdot\boldsymbol{n}\right]_j = -\sum_{j=1}^{2}\left(\boldsymbol{Q}^{n} - \Delta t\boldsymbol{U}^{n}V_{m}\right)\boldsymbol{n} - \boldsymbol{U}_i^{n}\Delta[\Delta x_i]^{n+1} \qquad (6)$$

where subscripts, i and j is the spatial index for the cells and its boundaries in the computational domain while Δx is defined as the volume for individual cell in one-dimensional space. The Geometric Conservation Law (GCL) is satisfied by evaluating the volume as below,

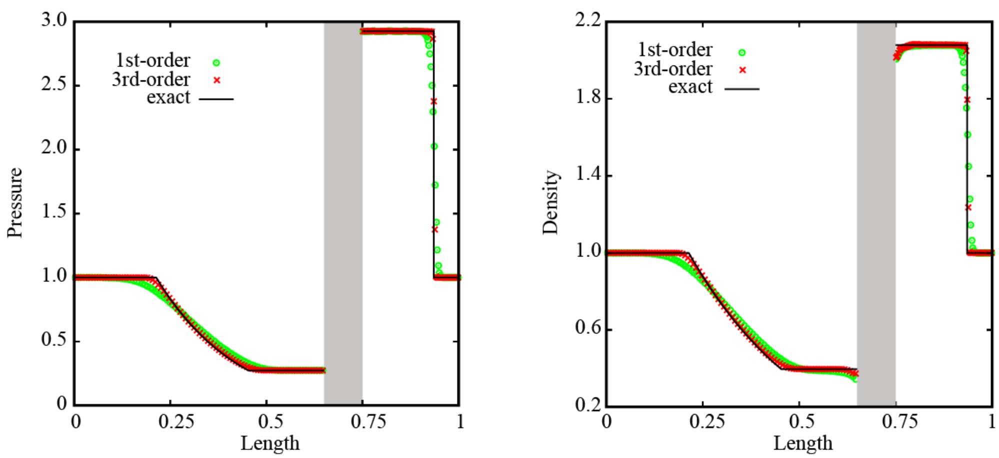
$$\Delta[\Delta x]^{n+1} = \Delta t\left(\sum V_{m}\cdot\boldsymbol{n}\right)^{n+1/2} \qquad (7)$$

Higher-order accurate solution can also be obtained by reconstructing the flux variables using MUSCL scheme. In this study, flux variables in the ***Q*** term are approximated with a third-order MUSCL scheme with minmod limiters.

RESULTS AND DISCUSSION

Solution of half-Riemann problem

This is a problem equivalent to the complete Riemann problem in stationary mesh, thus exact solutions exist [Nkonga, 1994]. Consider a tube containing quiescent fluid with its density and pressure equal to unity everywhere and a 0.1 unit long rigid wall initially located at the center of the tube. At time instant t = 0.0, the wall is suddenly moved with a velocity u_w = 1.0 resulting the development of a rarefaction wave behind the wall, while at the same time creating a shock ahead of the wall. Pressure and density profile at time instant t = 0.5 are shown in Figure 1. First order accurate numerical solutions matched with the exact solutions even with high dissipation error in the discontinuity regions. These errors can be minimize using higher order approximation, where as shown in the same figure, third order accurate solutions give better resolution at the discontinuity regions.



a) Pressure profile b) Density profile

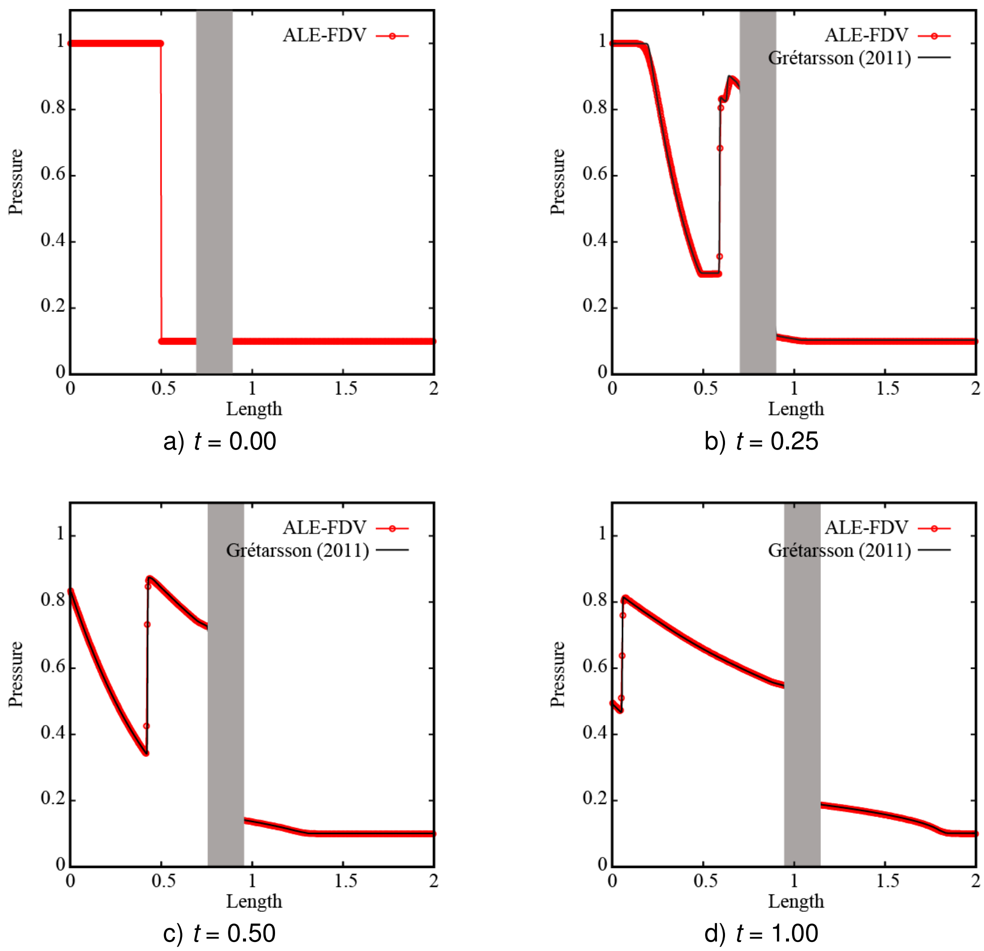
Figure 1: Numerical results for solution of half-Riemann problem (t = 0.5)

Sod shock coupled with a rigid wall

This problem has been investigated earlier by Grétarsson et al. (2011). Consider a movable rigid wall inside a 2.0 unit long tube filled with a quiescent fluid with the initial condition same as the Sod shock tube.

$$(\rho, u, p) = \begin{cases} (1.0, \quad 0, \quad 1.0) & x \le 0.5 \\ (0.125, \quad 0, \quad 0.1) & x > 0.5 \end{cases} \tag{8}$$

Periodic boundary condition is applied at the left and right boundary of the tube, while reflecting wall condition is applied on the wall surfaces. The wall has a mass 1.0 unit and is 0.2 unit long with its center initially located 0.8 unit from the left boundary of the tube. The simulation begins when the shock begins to move rightward. When it hits the wall, the wall begins to move due to the pressure difference between both sides of the wall, while the shock is reflected leftward. Figure 2 shows the pressure profile at various times and its comparison with the results by Grétarsson et al. (2011).



a) t = 0.00 b) t = 0.25

c) t = 0.50 d) t = 1.00

Figure 2: Pressure profile at several time instants

Sod shock interacting with fluid piston

Consider a closed boundaries tube with a movable wall located in the center. The tube is 3.0 units long while the wall is 0.2 units long with 1.0 unit mass. Initial and boundary condition are similar to the previous test except with the addition of reflective wall boundary conditions applied at both tube boundaries. The simulation begins with the shock located at a distance 1.0 unit from the left boundary of the tube moving towards the wall. When the wall is hit by the shock, it moves rightward because of the pressure difference while the shock is reflected back. Pressure in the region ahead of the wall rises at the same time because the wall is compressing the fluid in that region. This in turn, pushes the wall back making the wall decelerate. The position of its center of mass as a function of time is shown in Figure 3. As shown in this figure, wall location peaks roughly at t = 2.87, before going back towards its initial location. The pressure profile for several time instants are shown in Figure 4. The results are in good agreement with the simulation done by Grétarsson et al. (2011).

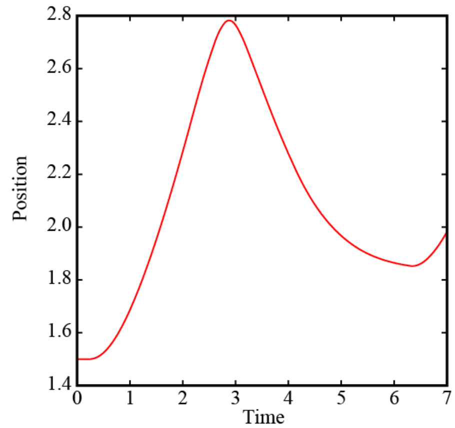
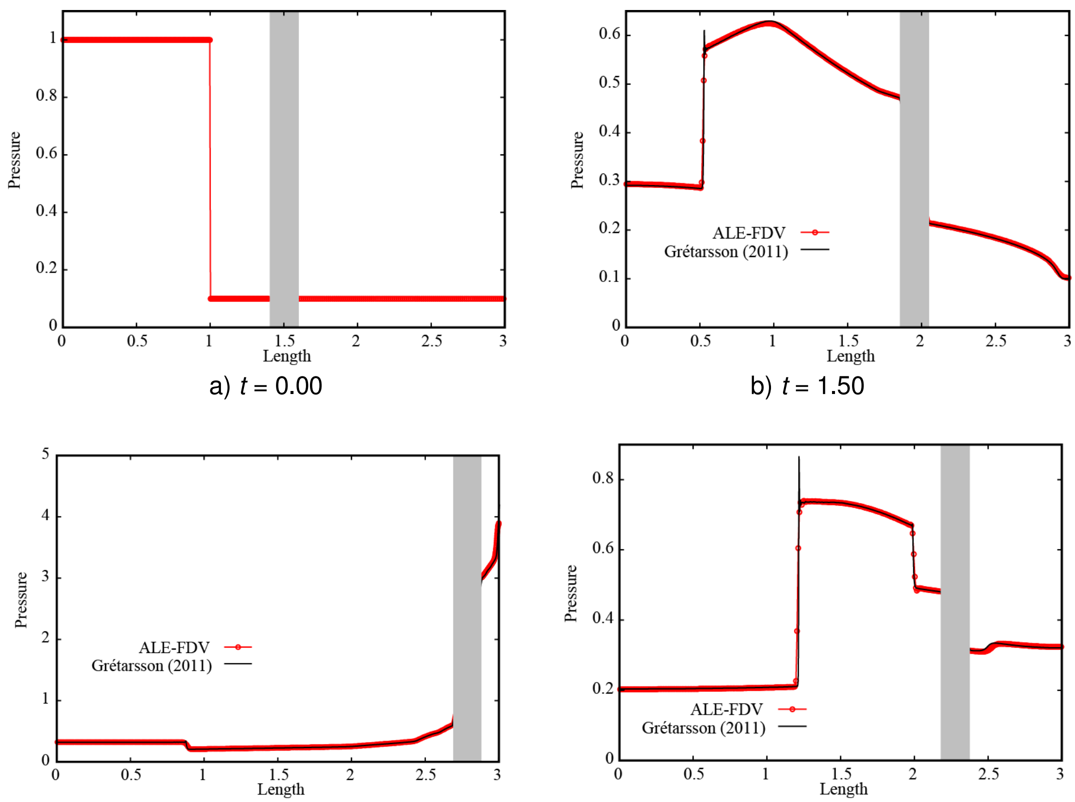


Figure 3: Position of the wall



a) t = 0.00

b) t = 1.50

c) t = 2.87

d) t = 4.00

Figure 4: Pressure profile at several time instants

CONCLUSION

The FDV method has been formulated in ALE form for dynamic mesh applications, motivated by our interest to extend its capability to fluid-structure interaction problems. Several one-dimensional fluid-structure interaction problems have been chosen as the test problems for validating the ALE form of FDV method. The results obtained are in good agreement with other numerical methods thus confirming its applicability to dynamic mesh applications. Although only one-dimensional inviscid flow was considered in this paper, the method showed great promise for viscous flow and higher dimensional fluid-structure interaction problems.

References

Chung, T.J. (1999) *Transitions and interactions of inviscid/viscous, compressible/incompressible and laminar/turbulent flows*, International Journal for Numerical Methods in Fluids, Vol. 31, p:223-246, Sep 1999.

Chung, T.J. (2002). *Computational Fluid Dynamics*. Cambridge University Press, 1st edition.

Donea, J., Huerta, A., Ponthot, J. and Rodríguez-Ferran, A. (2004) *Arbitrary Lagrangian–Eulerian Methods*. Encyclopedia of computational mechanics, p:1–38, Nov 2004.

Grétarsson, J.T., Kwatra, N., and Fedkiw, R. (2011) *Numerically stable fluid–structure interactions between compressible flow and solid structures*, Journal of Computational Physics, Vol. 230, p:3062-3084, Apr 2011.

Guillard, H. and Farhat, C. (2000) *On the significance of the geometric conservation law for flow computations on moving meshes*, Computer Methods in Applied Mechanics and Engineering, Vol 190, p:1467-1482, Dec 2000.

Heard, G.W. (2007) *Flowfield-Dependent Variation (FDV) Method for Compressible, Incompressible, Viscous, and Inviscid Flow Interactions with FDV Adaptive Mesh Refinements and Parallel Processing*, PhD thesis, University of Alabama, Huntsville.

Nkonga, B., and Guillard, H. (1994) *Godunov type method on non-structured meshes for three-dimensional moving boundary problems*, Computer methods in applied mechanics and engineering, Vol. 113, p:183-204, Mar 1994.

Thomas, P.D. and Lombard, C.K. (1979) *Geometric conservation law and its application to flow computations on moving grids*, *AIAA Journal*, Vol. 17, p:1030-1037, Oct 1979.