Transactions of the VŠB – Technical University of Ostrava, Mechanical Series
No. 2, 2018, vol. LXIV
article No. 2047

Michal KOVÁČIK*

BOUNDARY MOVEMENT IN STATIC EULERIAN MESH

POHYB HRANICE NA STATICKÉ EULEROVSKÉ SÍTÍ

Abstract

This work is focused on a procedure to treat the boundary movement associated to fluid-structure interaction. In the beginning, the theory of the procedure is described. The procedure described here is an alternative to the deformation of the Eulerian mesh bounded by the solid phase. This alternative procedure is then applied to a specific one-dimensional problem, where results are visualized and discussed. Alas, the cons and pros of the method are considered as well as alternatives to improve the method.

Abstrakt

Tato práce se zabývá postupy pro chování hranice při interakci tekutých a pevných těles. Nejprve je popsáno teoretické zázemí. Dále je popsána metoda jako alternative deformace eulerovské sítě ohraničené pevným tělesem. Tato problematika je pak osvětlena na jednoduché jednorozměrné úloze s přednesením výsledků a diskuzí. Nakonec jsou vyhodnoceny výhody i nevýhody včetně návrhů pro zlepšení metody.

Keywords

Fluid-structure interaction, Eulerian mesh, Boundary movement, Finite Volume Method, Euler equations

1 INTRODUCTION

The fluid-structure interaction problem is quite common in nowadays calculations. It is also often a feature of commercial simulation software. The algorithms behind may seem however somehow hazy. There are many procedures of treating such functionality. It is default to use the Eulerian mesh type for fluid computations, which tracks the quantities through the domain, while the domain itself remains often fixed in space. However, the Lagrangian mesh type deforming together with the domain is default to solve the solid mechanics. Numerous observations on this matter were published by Hou, Wang and Layton [1]

A common procedure to solve interaction with those two types of meshes is to deform the Eulerian mesh bounded by the outer Lagrangian mesh during the simulation as it happens for instance in coupling using ANSYS CFX [2] and ANSYS mechanical. As the mesh stretches or shrinks, it may happen that some Euler elements distort and form a negative volume, which results in a computational failure. Therefore, care must be taken while meshing to avoid such scenarios.

Another procedure, which is used by MSC.Dytran [3] might be using a static Euler mesh while the Lagrangian mesh bounding the domain intersects it. The intersection forms another set of

* Ing., Department of applied mechanics, Faculty of Mechanical Engineering, VŠB–Technical University of Ostrava, 17. Listopadu 15/2172, 708 33 Ostrava, tel. (+420) 59 732 5273, e-mail michal.kovacik@vsb.cz

faces in the Euler elements, while the outside of the bound is not considered. The stability may be analysed by the Courant-Friedrichs-Lewy condition (eq. 9) associated to the time-step being proportional to the element edge length. While in the previous procedure, the deformation may influence the edge of all Fluid elements, in this procedure it affects only the elements coinciding with the boundary.

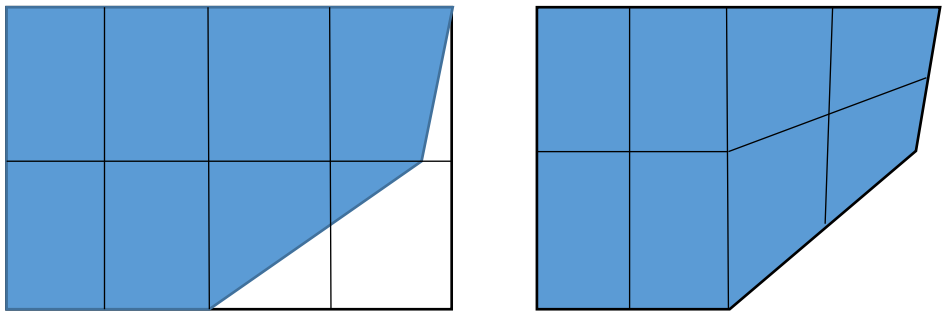

Fig. 1 Comparison of both procedures, pure Eulerian mesh with boundary intersecting the volumes on the left, while the mix of Lagrangian and Eulerian mesh deforming with the boundary on the right

2 FLUID-STRUCTURE INTERACTION IN ONE DIMENSION

This work focuses on the latter procedure simulated in one dimensional fluid domain. The Fluid within the domain is modelled using the time-dependent isothermal Euler equations for a compressible inviscid fluid. The equations conserve two main quantities-density ρ and momentum P:

$$\frac{\partial \rho}{\partial t} = -\frac{\partial P}{\partial x} \tag{1}$$

$$\frac{\partial P}{\partial t} = -\frac{\partial}{\partial x}\left(\frac{P^2}{\rho} + p(\rho)\right) \tag{2}$$

With time t and position x, the p accounts for the pressure being a function of the density. The modelled fluid is water, so the viscosity is known and the pressure may be described by linearized Bulk equation of state with reference density ρ_{ref} and bulk modulus K:

$$p = K \cdot \ln\frac{\rho}{\rho_{ref}} \approx K \cdot \left(\frac{\rho}{\rho_{ref}} - 1\right) \tag{3}$$

The conservation of the variables is computed using a one-dimensional finite volume scheme for volumes of length L and constant cross-section. The conservation equations 1 and 2 are rewritten into matrix form with vector of conserved variables $w = \begin{pmatrix} \rho \\ P \end{pmatrix}$ and corresponding fluxes $f = \left(\frac{P^2}{\rho} + K \cdot \left(\frac{\rho}{\rho_{ref}} - 1\right)\right)^{P}$. Notice the pressure is already substituted from eq. 3:

$$\frac{\partial}{\partial t}\int_0^L w dx = f(0) - f(L) \tag{4}$$

The flux interpolation at the boundaries is done using upwind differencing scheme and time integration carried out using forward Euler integration. The upwinding is enforced using central differencing scheme with numerical dissipation proportional to the local maximum of characteristics of the flow. The non-decreasing entropy condition is as well treated by the numerical dissipation. [4]. The characteristics are found as the eigenvalues λ of the characteristic determinant:

$$\left|\begin{pmatrix} \frac{\partial f_0}{\partial w_0} - \lambda & \frac{\partial f_0}{\partial w_1} \\ \frac{\partial f_1}{\partial w_0} & \frac{\partial f_1}{\partial w_1} - \lambda \end{pmatrix}\right| = 0 \tag{5}$$

There is only one type of boundary condition – the wall, which is enforced using a formula proposed by JST [5]. The momentum has to be zero at the wall for all the time, therefore the time derivative is zero as well:

$$\left.\frac{\partial P}{\partial t}\right|_{wall} = 0 \tag{6}$$

Substituting the term into the conservation equation (2) yields an equation for the wall static pressure. Since the term $\frac{p^2}{\rho} + p$ is often referred to as the total pressure, it can be used to extrapolate the static pressure from two cells closest to the boundary denoted by indices n and n-1. The index $n + \frac{1}{2}$ denotes the boundary. The variable x represents the position:

$$p_{static_{n+\frac{1}{2}}} = \frac{p_{total_n} - p_{total_{n-1}}}{x_n - x_{n-1}}\left(x_{n+\frac{1}{2}} - x_n\right) + p_{total_n} \tag{7}$$

The static boundary pressure is used as well to compute the movement of the boundary. The boundary acceleration a is proportional to the resultant force which is the external force *F* subtracted from boundary pressure multiplied by the area *A* of the boundary, the mass of the boundary *m* accounts as well:

$$a = \frac{p_{static_{n+\frac{1}{2}}} A - F}{m} \tag{8}$$

The acceleration is integrated in time and the resultant displacement is used to move the boundary. The integration scheme used for the boundary is the leap-frog scheme. The displacement is used to update the volume of the element closest to the boundary. This change of volume is accounted to update the density in the cell.

The integration timestep needs to be accounted for both the solid and fluid domain. The stability of the bounding domain is not difficult to satisfy in this case, but the fluid domain requires the Counrant-Friedrichs-Lewy condition which states that the timestep cannot exceed the time required to pass the waves through one cell, with Δx being the cell length, the speed of the waves is again the characteristics of the system solved in eq. 5. The minimal required timestep Δt is then:

$$\Delta t = \frac{\Delta x}{\lambda_{max}} \tag{9}$$

As the cell shrinks, the timestep becomes smaller, to overcome critical shrinkage of the time step. The volumes in this model are allowed only to shrink to half their initial size, if the boundary reaches more than half the cell volume the cell is deactivated and the neighboring cell is stretched to enclose the volume, while the mass is added from the deactivated element to the neighboring element. The momentum is interpolated in a linear sense.

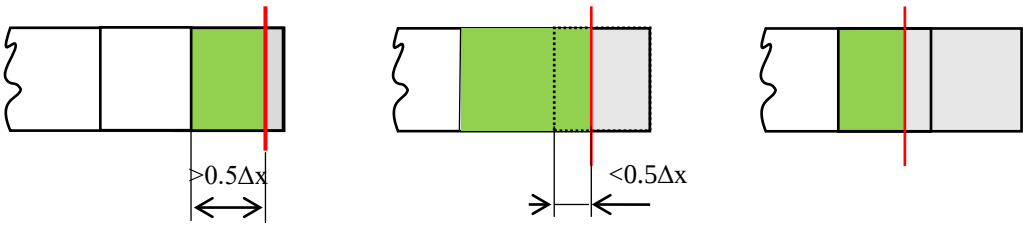

Fig 2. Left - If the volume of the boundary cell is over half its initial length, it is still used in the analysis. center – for the opposite case, the boundary cell is merged to its neighbor. Right if the boundary moves through the whole cell, the whole element is deactivated

On the other hand if the volumes expand past the half of a deactivated cell volume, the cell is reactivated (again). The boundary cell is now a neighboring cell and the deactivated cell is now an active cell at the boundary. The masses from the previous cell split and the momentum is extrapolated from two neighboring cells.

3 DEMONSTRATION

The boundary is simulated as a single degree of freedom intersecting the Euler domain, where one side is kept stationary and the other is moving. With no initial displacement, a constant external force is exerted on the boundary. The volume length in the Eulerian mesh is uniform for all elements at the start.

Fig. 3 Computational domain with the boundary which is pushed by a constant force

To be able to demonstrate the feature of deactivating and reactivating elements requires significant boundary displacements. The domain has been split into 50 volumes. That means the boundary needs to change the volume at least by 2%, which requires enormous amount of force exerted at the boundary. The main issue is that the bulk equation model is suitable only for liquids, which are nearly incompressible. The consequence of that is that any change of volume results in a severe change of pressure. However, the purpose of the problem is a benchmark of the method, so although the scales of the problem seem unrealistic, it serves the purpose of testing.

Since this is a boundary-initial problem both boundary and initial conditions are to be specified. Boundary conditions for the fluid domain are already specified in eq. 6. The initial conditions for both solved variables are uniform reference density and zero momentum. No movement is initiated from the fluid domain. However, the outer force is exerted to the boundary and it is kept constant in time.

4 RESULTS

For the results, several variables were focused on. The boundary displacement and the distribution of the static pressure in time.

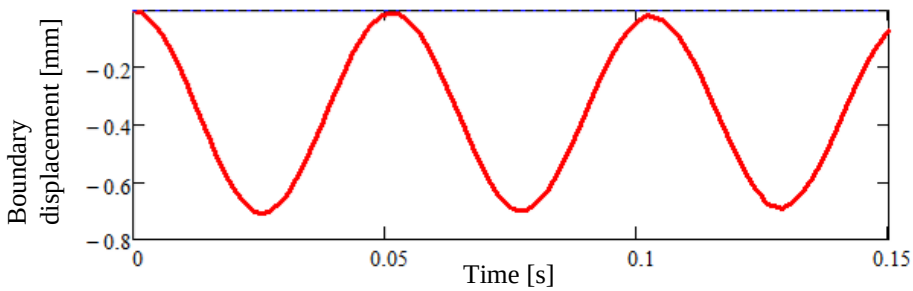

Fig.4 The resulting displacement for 100 time steps. Displacement in meters on the vertical axis, time in seconds on the horizontal axis.

It can be observed that the response of the boundary displacement is almost harmonic with no observable damping.

Fig 5. The pressure distribution in the domain in the first time step in the computation (initial conditions are the 0-th timestep). The blue line represents the boundary. Each bar represents each finite volume value.

The pressure at the boundary clearly corresponds to the displacement of the boundary. The sampling of the results is good enough to observe the pressure waves travelling through the domain. However, the shockwave is not as sharp, because the velocity of the boundary is comparable to the wave speed. It can be computed as the characteristic of the equation set, which is in fact the velocity of the fluid summed with the speed of sound:

$$c = u \pm \sqrt{\frac{K}{\rho_{ref}}} \qquad (9)$$

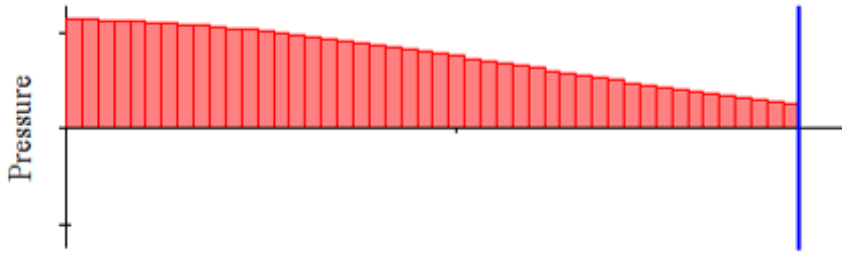

Fig 6. The pressure wave bouncing back as the boundary is slowed down to its maximal displacement. Notice the small black vertical line showing the original position of the boundary.

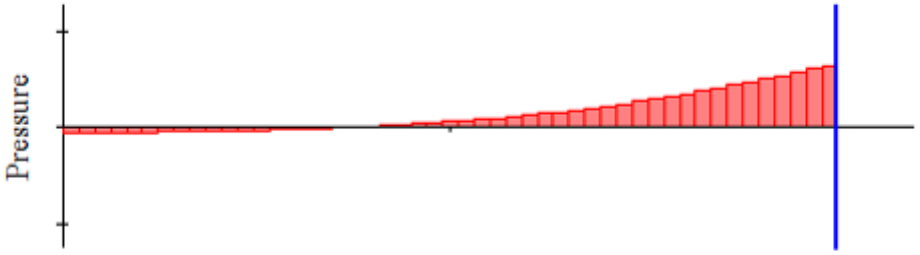

Fig 7. The boundary almost reaching its original position. The pressure is minimal at this moment (not considering the 0-th time-step). Notice there is slight negative pressure on the left caused by slight diffusion together with convection.

The mass of the boundary has naturally a dominant influence on the period of the boundary movement. However, the frequency is still affected by the waves travelling through the domain both ways.

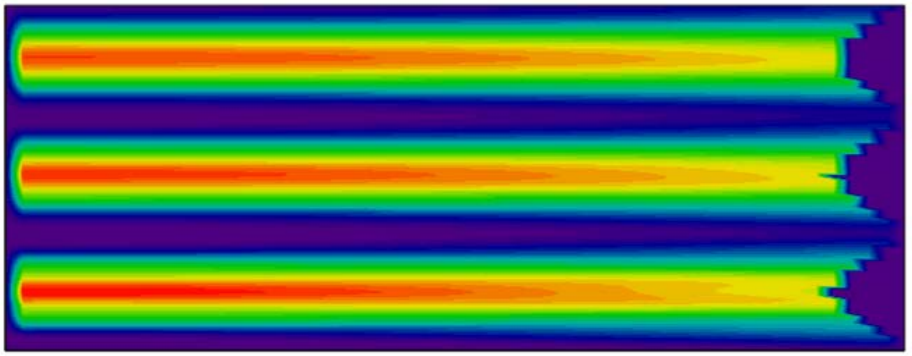

Fig 8. The contour representing the pressure distribution in a space-time plot. The position is represented by the horizontal axis and the time in vertical. Red colour indicates the highest pressure, while violet the lowest. The sharp edges on the right represent the deactivated elements. The waves are clearly observable travelling together with the boundary displacement.

5 CONCLUSIONS

It can be observed that the deactivating and activating mechanism acts flawlessly and brings no additional errors to the analysis. It can be observed that the fluid domain interacting with the solid structure acts as a spring damper system.

As observed, the activation and deactivation of elements including merging neighbor elements is robust in terms of treating the fluid structure interaction. The half of the element length as a margin for deactivation is however individual and can be chosen higher in order not to slow down the computation by reducing the time step because of single element size. However such merging can be considered inaccurate since the element closest to the boundary would have half the resolution of others.

In comparison to the Lagrangian-Eulerian formulation of the interaction, this procedure is quite complex to implement, however, it is not so unstable with the boundary movement and the mesh quality is not as important for the stability. Problems can arise if the boundary reaches out of the edge of the Eulerian mesh.

REFERENCES

[1] HOU, G., WANG, J. & A. LAYTON. Numerical Methods for Fluid-Structure Interaction — A Review. *Communications in Computational Physics* [online]. 2012, **12**(02), 337-377 [cit. 2018-06-19]. DOI: 10.4208/cicp.291210.290411s

[2] ANSYS Inc. *ANSYS CFX-Solver Theory Guide*., Release 15.0, 2013

[3] MSC Software *MSC Nastran Explicit Nonlinear (SOL 700) User's Guide*, 2013

[4] JAMESON, A. *Artificial Diffusion, Upwind Biasing, Limiters and their Effect on Accuracy and Multigrid Convergence in Transonic and Hypersonic Flows.* AIAA paper 93-3359, Jul. 1993

[5] JAMESON, A., SCHMIDT, W. & TURKEL E. *Numerical solutions of the Euler equations by finite volume methods with RungeKutta time stepping schemes*. AIAA paper 81-1259, January 1981