A 3D, PARALLEL FLUID–STRUCTURE INTERACTION CODE

Oliver Oxtoby⋆ and Arnaud G. Malan⋆⋆

Advanced Computational Methods, Council for Scientific and Industrial Research,
PO Box 395, Pretoria 0001, South Africa
⋆ooxtoby@csir.co.za; ⋆⋆amalan@csir.co.za

SUMMARY

We describe the development of a 3D parallel Fluid–Structure–Interaction (FSI) solver and its application to benchmark problems. Fluid and solid domains are discretised using and edge-based finite-volume scheme for efficient parallel computation, with a hybrid of node- and element-based strains calculated in the solid model for accuracy. The Pressure-Projection Artificial Compressibility Split (PACS) algorithm [1] is used in the fluid domain and re-derived in an Arbitrary-Lagrangian–Eulerian (ALE) reference frame. A preconditioned GMRES algorithm is developed for matrix-free solver acceleration. The fluid and structural domains are strongly coupled with a fast mesh-movement technique employed in the fluid domain. The solver is parallelised for distributed-memory architectures.

Key Words: Fluid–Structure Interaction, preconditioned GMRES, ALE

1 INTRODUCTION

Many physical systems involving fluid flows are in truth strongly dynamic systems where there exists an intimate coupling between fluid and structural or solid domains. Examples are flutter in aircraft wings and cardio-vascular and respiratory systems in the human body. Phenomena of interest range from large-deformation structural response to dynamic induced stresses and non-linear flow behaviour. Fluid–Structure-Interaction (FSI) modelling is a branch of Computational Mechanics which aims at accurately calculating these effects in a quantitative manner. Though recent years have seen much research going into the development of FSI modelling technology, the efficient and robust modelling of large-scale, strongly-coupled multiphysics systems which involve complex geometries is still some way off. In this work, we develop and evaluate a new fully-coupled, matrix-free methodology as a first step to addressing this challenge.

2 METHOD

Our FSI strategy is fully coupled in the sense that information is transferred at solver sub-iteration level, leading to a fully-converged solution at each timestep where both dynamic and kinematic continuity – i.e. continuity of forces and velocities – is satisfied at the fluid/solid interface. Spatial discretization of the fluid and solid domains is entirely independent, although in this work we have chosen to use a finite-volume scheme for both, in the interests of coding efficiency and

simplified parallelisation. That is, we are able to use a single solver code in a manner which allows independence in terms of both discretization as well as solution strategy for the fluid and solid domains, while ensuring strong coupling via a simple interface data transfer method. In the case of the solid, large non-linear deformation is allowed for via a total-Lagrangian formulation. The fluid governing equations are written in an Arbitrary-Lagrangian–Eulerian coordinate system to allow efficient and unified modelling of the entire fluid-solid system.

The non-linear unified governing equations are spatially discretized via a hybrid-unstructured edge-based finite volume method whose spatial accuracy is formally of second order. In the interest of both computational and programming efficiency, the chosen spatial discretisation algorithm should be naturally applicable to any part of a fluid or solid mesh. This is achieved by employing a purely edge-based compact discretisation methodology, which holds the additional advantage of being computationally considerably more efficient than element-based approaches while being ideally applicable to massively parallel distributed memory machines. In the case of the solid domain, a hybrid elemental/nodal strain approach was implemented in the interests of accuracy. In this method, shear strains are calculated by integrating around elements to obtain derivatives, while longitudinal strains are obtained in the standard node-based finite volume approach of integrating around dual-cells. The former prevents large errors at boundaries while the latter prevents odd-even decoupling from occuring.

Second-order accuracy is also attained in temporal discretization and results in an implicit solution algorithm. This is of critical importance in order to ensure a fully coupled solution procedure, which is effected in a matrix-free manner via the use of dual-timestepping. The pseudotime-integration required for the solid domain to calculate displacement from acceleration is done via a second-order accurate single-step procedure [2]:

$$
\begin{aligned}
w_i^{\tau+1} &= w_i^\tau + \Delta t_\tau \left(v_i^\tau + \tfrac{1}{2} \frac{\Delta t_\tau}{\rho^{n+1} \mathcal{V}_{n+1}} R_i(\overline{w}^\tau) \right) \\
v_i^{\tau+1} &= v_i^\tau + \frac{\Delta t_\tau}{\rho^{n+1} \mathcal{V}_{n+1}} R_i(\overline{w}^\tau)
\end{aligned}
\tag{1}
$$

where $\overline{w}^\tau$ denotes a projected displacement which is calculated as $\overline{w_i}^\tau = \overline{w_i}^\tau + \Delta t_\tau v_i^\tau$.

In the case of the fluid, the Pressure-Projection Artificial Compressibility Split (PACS) algorithm [1] is used. As mentioned, the three-step scheme is extended for moving meshes as follows.

$$
\frac{\Delta W_i^*}{\Delta t_\tau} V^\tau = - \int_{\partial \mathcal{V}(t)} \left[\rho u_i (u_j - v_j) - \sigma_{ij} \right] n_j dS \bigg|^\tau + \mathbf{S} V |^\tau
\tag{2a}
$$

$$
\frac{1}{c^2} \frac{\Delta p}{\Delta t_{\tau 2}} V^\tau = - \int_{\partial \mathcal{V}(t)} \left[\rho u_k + \Delta t_\tau \left(\frac{\Delta W_k^*}{\Delta t_\tau} - \frac{\partial p}{\partial x_k} \right) \right] n_k dS \bigg|^\tau
\tag{2b}
$$

$$
\frac{\Delta W_i}{\Delta t_\tau} V^{\tau+1} = \frac{\Delta W_i^*}{\Delta t_\tau} V^\tau - \int_{\partial \mathcal{V}(t)} p n_i dS \bigg|^{\tau+1}
\tag{2c}
$$

where u denotes the flow velocity, v the mesh velocity, and V the cell volume. The τ superscript denotes the current pseudo time-step and Δt_τ the psedo-timestep size, c^2 is the artificial-compressibility pseudo-acoustic velocity [3]. The sourceterm $\mathbf{S}$ contains the second-order accurate dual-timestepping term.

The solver is accelerated using a GMRES algorithm with preconditioning based on [4], providing matrix-free operation at a much lower cost than Jacobi iterations. The algorithm is parallelised for distributed-memory architectures using MPI.

Finally, the coupled FSI solver is applied to strongly-coupled large-displacement FSI benchmark problems from literature.

3   CONCLUSIONS

A high-performance solver has been developed to accurately predict flows and stresses in strongly-coupled fluid–solid systems involving large structural deflections.

REFERENCES

[1]   P. Nithiarasu, An efficient artificial compressibility (AC) scheme based on the characteristic based split (CBS) method for incompressible flow, *International Journal for Numerical Methods in Engineering*, 56:13, 1815–1845, 2003.

[2]   O. C. Zienkiewicz and R. L. Taylor. *The Finite Element Method*, 5th Edition, Vol. **I**, Butterworth-Heinemann, 2000.

[3]   A. G. Malan, R. W. Lewis and P. Nithiarasu, An Improved Unsteady, Unstructured, Artificial Compressibility, Finite Volume Scheme for Viscous Incompressible Flows: Part I. Theory and Implementation, *International Journal for Numerical Methods in Engineering*, 54:5, 695–714, 2002.

[4]   H. Luo, J. D. Baum and R. Löhner, A Fast, Matrix-Free Implicit Method for Compressible Flows on Unstructured Grids, *Journal of Computational Physics*, 146, 664–690, 1998.