An Implicit Discontinuous Galerkin Chimera Method for Unsteady Laminar Flow Problems with Multiple Bodies

Fabian Genuit1,*, Manuel Keßler1, and Ewald Krämer1

1 Institute of Aerodynamics and Gas Dynamics (IAG), University of Stuttgart, Pfaffenwaldring 21, 70569 Stuttgart

The compressible Navier-Stokes (NS) equations are spatially discretized with the discontinuous Galerkin (DG) method and an implicit backward differentiation formula of second order is used for the temporal discretization. The Chimera method is employed to realize a simple grid generation for complex technical applications consisting of multiple parts. Therefore, non-trivial overlapping grid areas could occur in the numerical setup which require a robust implementation of the Chimera method. The flow around two circular cylinders in tandem arrangement serves as a validation case and reveals promising results for several configurations compared to reference data in literature.

1 Implicit discontinuous Galerkin framework for the Navier-Stokes equations

The compressible NS equations describe the conservation of mass, momentum and energy for any flow problem and serve as the governing equations in this work. The spatial discretization of the NS equations is realized with the DG method. A more detailed derivation of the semi-discrete weak form of the NS equations used in this work has already been shown in [1]. The DG method is characterized by not having any requirements on the continuity properties of the solution at the element boundaries and hence could be discontinuous across the elements. Thus, flux terms of adjacent cells are treated with the Bassi-Rebay 2 method [2] for the diffusive surface flux and with a Harten, Lax and van Leer Riemann solver [3] for the convective surface flux. The method is implemented in the flow solver SUNWinT (Stuttgart University Numerical Wind Tunnel) developed at the IAG at the University of Stuttgart.

SUNWinT uses a second order backward differentiation formula (BDF2) [4] for the temporal discretization. Therefore, the non-linear system is solved applying a Newton-Raphson method. The solution of the linear equation system is obtained by applying an ILU(0) preconditionned GMRES solver at each Newton iteration. Further details on the governing equations and their discretization in this work could be found in [1, 5]. Moreover, curved body surfaces are reconstructed using high-order grids [5, 6].

2 Chimera method for multiple bodies

The Chimera method has already been introduced in [1] as a grid technique allowing a numerical setup to consist of at least two overlapping grids and thus simplifying the grid generation for complex technical applications. While in [1] relative movements of individual grid parts to each other were considered, the focus of this work is on multiple Chimera grids within a numerical setup as it is exemplary illustrated in Fig. 1. It consists of four grids: two body grids, each representing a circular cylinder, shifted in the horizontal direction and two background grids. The cell blanking, i.e. the Chimera specific task of each cell in the setup is colored for each one half of the grid components. The active cells have the same behavior as the cells in a single grid, and in the hole cells the calculation of the solution is deactivated. The interpolation cells (ICs) serve as boundary cells for the Chimera grids. The solution for these cells is interpolated from the active cells in the other Chimera grid parts that overlap with the ICs. Mathematically, the interpolation is achieved by a discrete L2 projection. Since an implicit time integration scheme was chosen, the L2 projection must also be treated implicitly [7, 8].

In case of Chimera setups with multiple grids, further considerations are necessary. The generation of the cell blanking, also known as hole cutting, must be implemented adaptable, flexible and robust. In this work a separate hole grid defines the shape of each hole and it is assigned to its corresponding Chimera grid [9]. It cuts a hole in all other grids except the one it is assigned to and the optionally specified ones. Furthermore, an IC could have more than one possible donor, due to...
multiple overlapping grids. If these multiple donors are neither a hole cell nor an IC the donor with the smallest cell volume is chosen [9].

3 Numerical simulation of two circular cylinders in tandem arrangement

The flow around two cylinders with a diameter $D$ that are shifted by a distance $L$ in the flow direction has extensively been investigated in literature [10–12] and serves as a validation case for the implemented Chimera method. Figure 1 shows a section of the numerical setup used with a distance between the cylinder centers of $L = 1.5D$. The complete Chimera setup consists of six grids: two cylinder grids with 832 curvilinear cells ($y^+ \approx 1$, $p_{geo} = 3$) each and four background grids with uncurved cells, where each of the background grids increases in size and reduces the resolution. The setup has 8040 cells in total which is significantly less than the 24770 cells of the setup used in [10]. The boundary condition for the cylinder surfaces is a no-slip wall and the farfield is located approx. $100D$ away in each direction of the cylinders. Laminar flow with a Reynolds number $Re = 200$ and a Mach number at the farfield $Ma = 0.1$ was considered. The time step size for the BDF2 scheme was $\Delta t = 0.5$. The selected spatial order was four ($p_3$). Furthermore, the distance $L$ was varied.

(a) Mean drag coefficient $c_{D,\text{mean}}$ over the cylinder distance $L/D$ from SUNWinT and from literature [10–12]

(b) Vorticity for $L/D = 1.5$ from SUNWinT (left) and from [10] (right)

(c) Vorticity for $L/D = 4.0$ from SUNWinT (left) and from [10] (right)

Fig. 2: Simulation results for the flow around two cylinders in tandem arrangement with a spatial order of four ($p_3$)

The mean drag coefficients of both cylinders are plotted over the cylinder distance in Fig. 2a and show a promising match with the values found in literature [10–12]. At $L/D = 4.0$ the range of the values is somewhat larger, however the simulation results are well within this range. The vorticity fields for $L/D = 1.5$ (Fig. 2b) and for $L/D = 4.0$ (Fig. 2c) also show identical patterns to the results provided by [10]. In addition, in the flow solution no transition between the Chimera grids is visible.

The DG Chimera method proves valid for applications with multiple body grids and several Chimera grid components. In future, further test cases are planned to expand the validation of the method.

Acknowledgements Open access funding enabled and organized by Projekt DEAL.

References

[1] F. Genuit, M. Kessler, and E. Krämer, PAMM 20(1), e202000172 (2021).
[2] F. Bassi, A. Crivellini, S. Rebay, and M. Savini, Computers and Fluids p. 34 (2005).
[3] E. F. Toro, Riemann Solvers and Numerical Methods for Fluid Dynamics: A Practical Introduction, 2nd edition (Springer, 1999).
[4] E. Süli and D. F. Mayers, An Introduction to Numerical Analysis (Cambridge University Press, 2003).
[5] M. Wurst, Development of a high-order Discontinuous Galerkin CFD solver for moving bodies, PhD thesis, Institut für Aerodynamik und Gasdynamik, Universität Stuttgart, 2016.
[6] F. Hindenlang, Mesh curving techniques for high order parallel simulations on unstructured meshes, PhD thesis, Institut für Aerodynamik und Gasdynamik, Universität Stuttgart, 2014.
[7] M. Wurst, M. Kessler, and E. Krämer, Computers & Fluids 121, 102 – 113 (2015).
[8] M. C. Galbraith, R. D. Knapke, P. D. Orkwis, and J. A. Benek, A Discontinuous Galerkin Chimera scheme with implicit artificial boundaries, in: 51st AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition, (American Institute of Aeronautics and Astronautics (AIAA), jan 2013), p. 23.
[9] T. Schwarz, Ein blockstrukturiertes Verfahren zur Simulation der Umströmung komplexer Konfigurationen, PhD thesis, DLR research report 2005-20, 2005.
[10] J. Meneghini, F. Saltara, C. Siqueira, and J. Ferrari, Journal of Fluids and Structures 15(2), 327 – 350 (2001).
[11] R. Surmas, L. O. dos Santos, and P. C. Philippi, Future Generation Computer Systems 20(6), 951–958 (2004), Computational science of lattice Boltzmann modelling.

[12] H. C. Vu, J. Ahn, and J. H. Hwang, KSCE Journal of Civil Engineering 20(4), 1594–1604 (2016).