A FULLY PARALLEL COUPLED APPROACH WITH A DISCONTINUOUS GALERKIN SOLVER

O. LABBÉ

ONERA - The French Aerospace Lab
F-92322 Châtillon, France
email: odile.labbe@onera.fr, www.onera.fr

ABSTRACT

Multi-physics simulations require a high performance computing environment in order to perform the simulation in a reasonable amount of computation time. Generally, such applications yield multi-scale problems including different length scales where solving large problems with a monolithic approach would be too expensive. Also, highly sophisticated software codes are available for each single physical phenomena and it is useful to reuse currently available codes for multi-physics simulations. Since different phenomena typically appear spatially separated it is possible to decompose the overall simulation domain into non-overlapping partitions with distinctive treatment and link these partitions via a surface coupling. This allows the use of different numerical methods and tailored grid resolution for each individual domain. Moreover, each partition can be solved by individual software codes [1].

The high order discontinuous Galerkin solver Aghora [2] is based on a polynomial representation within an element and flux calculation between elements. The choice of the polynomial degree controls the spatial discretization order. By choosing a high degree of the polynomial function a higher order method can be constructed. The coupling tool CWIPI used in this study, which is responsible for the data exchange as well as for all interpolations between interfaces, is also performed in parallel and minimizes the amount of communication, since it works locally on solver processes. CWIPI uses a standard domain decomposition approach, since an unstructured mesh is considered. The governing equations are solved in the separate domains and coupled on the boundaries.

In a first step, CWIPI is implemented in the Aghora solver and we consider the convective vortex test case from the HiOCFD4 Workshop [3] in order to check the validity of the coupling method. The main problem is to avoid accuracy loss through the coupling boundary especially with high order method and necessitates the implementation of a 'call back' function which allows calculating the solution at the appropriate order. In the full paper, error estimations on pressure L2-norm will be presented between a single simulation and a coupled one for different meshes and orders.

REFERENCES

[1] O. Labbé, C. Peyret, G. Rahier and M. Huet, A CFD/CAA coupling method applied to jet noise prediction. *Computers & Fluids*, **86** (2013) 1–13.

[2] V. Couaillier, F. Renac, M. de la Llave Plata, J.-B. Chapelier, E. Martin,and M.-C. Le Pape, Turbulent flow simulations with the high-order DG solver Aghora, AIAA paper 2015-0058.

[3] HiOCFD4 4th International Workshop on High-Order CFD Methods, Crete, 4-5 June 2016, https ://how4.cenaero.be/