A partitioned solution approach for the fluid-structure interaction of wind and thin-walled structures

A. Halfmann, E. Rank
Lehrstuhl für Bauinformatik, Technische Universität München
M. Glück, M. Breuer, F. Durst
Lehrstuhl für Strömungsmechanik, Universität Erlangen-Nürnberg

1 Introduction

In many engineering applications two or more different interacting systems require the numerical solution of so-called multifield problems. In civil engineering the interaction of fluid and structures plays an important role, i.e. for fabric tensile structures of light and flexible materials often used for large roof systems, capacious umbrellas, tents, canopies or pavillions. Whereas powerful numerical simulation techniques have been established in structural engineering as well as in fluid mechanics, only relatively few approaches to simulate the interaction of fluids with civil engineering constructions have been presented. To determine the wind loads on complex structures, it is still state-of-the-art to apply semi-empirical, strongly simplifying methods or to perform expensive experiments in wind tunnels. In this paper an approach of a coupled fluid-structure simulation will be presented for membrane and thin shell structures. The interaction is described by the structural deformation as response to wind forces, resulting in a modification of the fluid flow domain. Besides a realistic determination of the wind loads, information on the structural stability can be obtained. The paper is organized as follows: Section 2 includes some remarks concerning the applied simulation codes integrated in a coupled application which is described in the third section. The partitioned solution approach realized by a so-called frame algorithm for controlling the simulation process and transferring the grid based data is focused in section 4. An application for quasi static explicit coupling is presented in section 5.

2 Simulation codes

The numerical simulation of fluid-structure interaction for wind loaded real-life structures is a challenge concerning the algorithmic complexity as well as the computational costs. Dynamic and nonlinear effects must be considered on both sides of the coupled problem. So it seems to be reasonable to perform a loose coupling strategy based on highly specialized and well evaluated simulation codes, each developed for the special field of the interacting system.
The Computational Fluid Dynamics (CFD) problem is described by the Reynolds-Averagered Navier-Stokes equations (RANS) for incompressible fluids. The system is closed by a $k - \epsilon$ turbulence model and solved by the Finite Volume Method. To extend the description for moving grids effected by dynamic fluid-structure interaction, an Arbitrary Langrangian Eulerian (ALE) formulation [4] is used. For the Computational Structure Dynamics (CSD) the characteristics of thin walled structures can be specified by state variables acting in the middle plane of the structure. The dynamic nonlinear response of membrane and thin shell structures is described by the equations of motion solved by the Finite Element Method.

3 System architecture

To preserve the advantages of the different simulation codes and to realize an effective coupling algorithm a loose coupling is performed by a partitioned solution approach [11, 13, 2]. The simulation is separated into several parts. In a first step, the structural model is defined, using a classical CAD environment on a PC or a workstation. All geometric information is stored in a database describing a b-rep (boundary representation) model completed by information concerning material properties and boundary conditions. For the structural part this project database [12] represents a central module. All programs used for the CSD simulation have access to the information stored in the database, e.g the preprocessor generating a typical finite element surface mesh for the thin walled structure. To consider different stress distributions in a membrane, the geometric shape can first be computed as a result of a formfinding process [1]. The geometric model for the fluid dynamic part is derived from the CAD model completed by boundary conditions and discretized as a block-structured three-dimensional grid being adequate to the finite volume technique. Due to extremely high computational requirements, the numerical simulation of the dynamic fluid-structure interaction is performed on a vector parallel computer in the next step. Finally, the results can be evaluated and visualized by powerful postprocessors on each side of the coupled application on a workstation or on a PC.

4 Partitioned solution approach

The solution of the partitioned approach is based on an iterative frame algorithm, integrating the CFD and CSD simulation codes in an explicit or implicit time-stepping procedure [8]. The data transfer between the two simulation codes is performed via a neutral geometric model using the *GRISSLi* coupling interface developed by the GMD [9].

4.1 Coupling interface

The coupling library supports the data exchange between different grid based applications. Each simulation code runs on its own processor(s) after being spawned by a main process. The interprocess communication is performed by the Message Passing Interface (MPI). The definition of process overlapping state variables allows the collective use of grid based data by different processes. Futhermore, the coupling library supports a grid data structure for defining the geometric model and control sequences as synchronization and branch points for controlling the coupled iteration process. Caused by the different discretization techniques for the CFD and CSD simulation, the fluid-structure interface is described by different numerical grids, e.g. a block-structured volume grid for the CFD and a non-matching unstructured surface mesh for the CSD simulation. Therefore, the data transfer between the two grids has to be combined with the interpolation of the grid based data. Besides a predefined standard interpolation, the interface allows to implement user-defined interpolation techniques via a programming interface [9].

4.2 Interpolation

Using the *GRISSLi* standard interpolation, the value of a source point is assigned to the closest target point with regard to the geometrical distance. It is also possible to consider several neighbouring nodes and a weighting procedure based on the distance for calculating the value for the target point ('Next-Neighbour Interpolation'). Usually fluid-structure interaction requires a finer mesh for the fluid part than for the structural part. It is therefore not guaranteed that, e.g., the pressure field is transferred correctly from the CFD to the CSD grid by this standard interpolation.

In the following the extension of the coupling library by a conservative load projection described by Farhat et al. [7] is focused. The discretization method corresponding to the finer mesh is used on both sides of the interface for computing the loads. Instead of transferring the pressure and stresses, equivalent nodal loads are exchanged by the following algorithm:

- perform a loop over all nodes S_j of the CFD grid belonging to the interface and search for each node the closest structural element $\Omega^{(e)}_{CSD}$
- determine the local coordinates ζ_j, η_j of the CFD node S_j concerning the CSD element $\Omega^{(e)}_{CSD}$
- compute the structural load f_{CSD_i} by distributing the value of the CFD node S_j with the finite-element shape functions N_i of the structural element $\Omega^{(e)}_{CSD}$ using

$$f_{CSD_i} = \sum_{j=1}^{j_{CFD}} N_i(\zeta_j, \eta_j)\, f_{CFD_j} \quad i \in \Gamma_{CSD},\, j \in \Gamma_{CFD} \tag{1}$$

  where j_{CFD} is the number of fluid nodes belonging to the interface elements.

The first step describing the so-called pairing of CFD nodes and CSD elements is performed in a very efficient and fast way using an octree based search algorithm [10]. Notice that, as for each quadrilateral finite element $\sum_{i=1}^{i=4} N_i = 1$, it is obvious that the described procedure is conservative and that therefore the sum of the nodal loads based on the different grids (CFD/CSD) is equal:

$$\sum_{i=1}^{i_{CSD}} f_{CSD_i} = \sum_{i=1}^{i_{CSD}} \sum_{j=1}^{j_{CFD}} N_i(\zeta_j, \eta_j)\, f_{CFD_j} = \sum_{j=1}^{j_{CFD}} f_{CFD_j} \tag{2}$$

4.3 Coupling algorithm

The coupling algorithm controlling the time-stepping procedure in the iterative solution based on [5] is shown in Figure 1. The outer loop describes the time discretization of the problem. Within each time-step an outer iteration between the CFD and CSD simulation is performed until convergence is reached. Thereby, the load for the CSD simulation is computed from the pressure and stresses as a result of the CFD computation and the boundary geometry is modified by the structural displacements computed by the CSD simulation. Significant structural deformations can be taken into account by an under-relaxation of the boundary geometry. To reduce the number of outer iterations within each time-step of the dynamic coupling procedure, this strategy is extended by a predictor-corrector scheme. At the beginning of each time-step the boundary geometry is estimated from the results of previous time-steps. Based on this geometry a CFD simulation is performed followed by a CSD computation which corrects the predicted interface geometry used in the next FSI (Fluid-Structure-Interaction) iteration as shown in Figure 1.

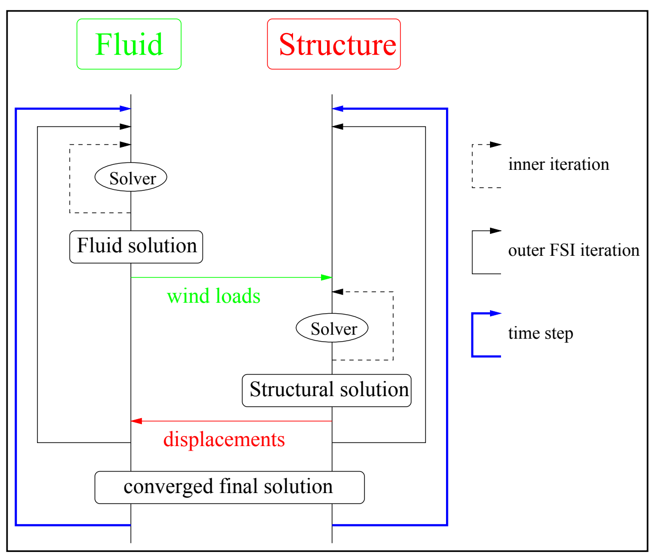

Figure 1: Coupling algorithm

5 Application example

The following results of a quasi-static fluid-structure simulation for the textile roof shown in Figure 2 corresponding to one time step in the coupling algorithm of section 4.3 is presented. The simulations are carried out with the CFD code *FASTEST-3D* [6] and the CSD code *ASE* [12]. A wind velocity of 30 m/s is assumed in positive x-direction. Streamlines around the tent roof and the pressure distribution as a result of the CFD simulation are shown in Figure 3. The structural displacements in z-direction are depicted in Figure 4.
Table 1 shows the sum of the loads and the computed maximum displacements of three iteration steps as a comparison of the two load projection schemes 'Next-Neighbour' and 'Conservative Interpolation' as described in section 4.2. The projection of the CSD grid shown in Figure 6 and two different CFD discretizations on the interface (Figure 5) are used for the simulations. An incremental displacement tolerance of 2 mm was chosen as convergence criterion.

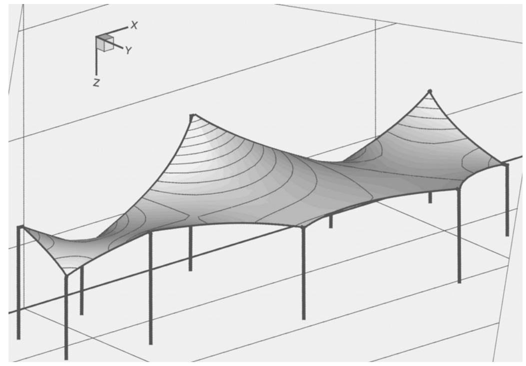

Figure 2: Geometry of a tent roof

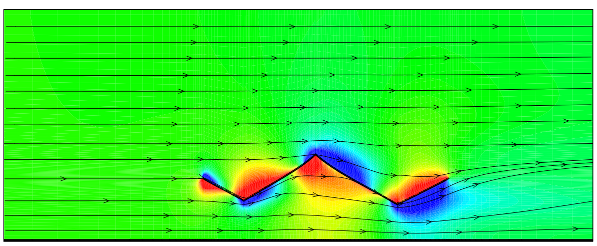

Figure 3: Streamlines and pressure distribution around the tent roof at west wind

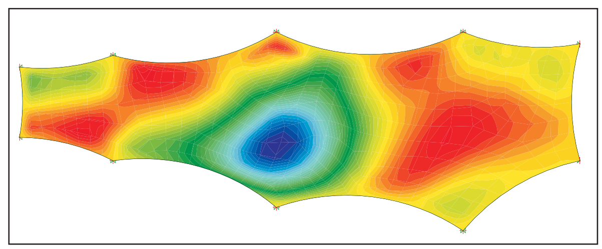

Figure 4: Displacement in z-direction

Variant	Global iteration steps	Total wind force [N] CFD	CSD	Difference of total wind force [%]	Maximum displacement of the structure [mm]
A	I	5709.2	5613.5	1.70	103.5
	II	5805.3	5803.4	0.03	116.9
	III	5808.9	5809.2	0.01	118.3
B	I	5702.8	5708.7	0.10	101.9
	II	5802.0	5870.5	1.17	116.0
	III	5809.1	5884.8	1.29	117.2
C	I	5709.2	5709.2	0.00	102.2
	II	5804.2	5804.2	0.00	115.8
	III	5807.5	5807.5	0.00	117.4

Table 1: Variant A: Results based on a coarse CFD grid (surface: 1008 nodes and 935 elements) and CSD grid (Figure 6) using 'Next-Neighbour Interpolation'. Variant B: Results based on fine CFD grid (Figure 5) and CSD grid (Figure 6) using 'Next-Neighbour Interpolation'. Variant C: Results based on fine CFD grid (Figure 5) and CSD grid (Figure 6) using 'Conservative Interpolation'.

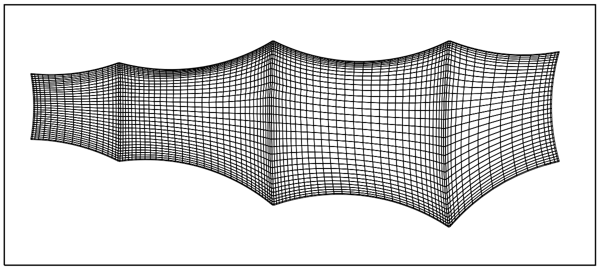

Figure 5: CFD surface mesh (3536 nodes, 3400 elements)

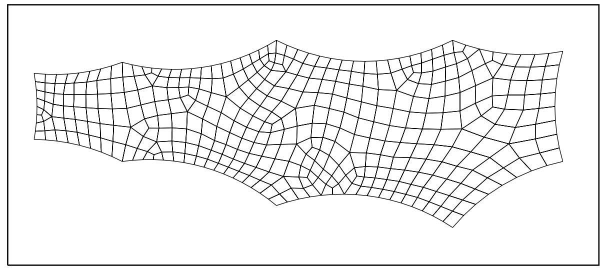

Figure 6: CSD mesh (437 nodes, 377 elements)

The sum of the total wind force on the CFD grid is almost the same in all three variants. Comparing the force for the CSD simulations, a deviation of the total wind load up to 1.7 % caused by the different CFD discretizations (variant A and B) is obvious. It is also visible that only a refinement of the CFD grid in case of 'Next-Neighbour Interpolation' does not guarantee a better load projection. For the 'Conservative Interpolation' (variant C) the sum of the total wind load is by definition exactly the same on both sides of the interface. Continuing the iteration process, the displacement of the structure behaves as shown in Figure 7 for variant C. After the third iteration step the maximum displacement u remains unchanged. Verifying the convergence criterion not only for the maximum displacement u but also for each node individually, oscillations at some nodes of the structure could be observed. Further investigations are necessary to determine the reason for this special behavior of variant C.

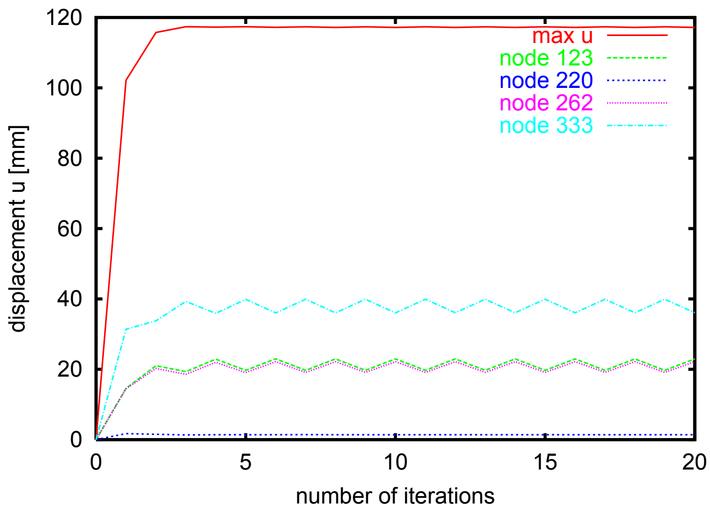

Figure 7: Displacements in z-direction at different nodes

6 Conclusions

A fluid-structure interaction for a non-trivial membrane construction was simulated using a partitioned solution approach. Two interpolation schemes were investigated and it turned out that the 'Next-Neighbour Interpolation' behaves reasonable well, if CFD- and CSD-grids use elements of approximately equal size. If, however, the CFD-grid is significantly finer than the finite element mesh of the structural simulation, the 'Conservative Interpolation' scheme should be used.

7 Acknowledgements

The authors gratefully acknowledge the support provided by Dr. Jürgen Bellmann from SOFiSTiK AG München. This research was funded by the Bavarian Consortium for High Performance Scientific Computing, FORTWIHR.

References

[1] J. Bellmann: *Membrantragwerk und Seifenhaut - Unterschiede in der Formfindung.* Bauingenieur 73, Nr. 3, S.118–123, 1998.

[2] H.-J. Bungartz, A. Frank, F. Meier, T. Neunhoeffer, S. Schulte: *Fluid Structure Interaction: 3D Numerical Simulation and Visualization of a Micropump.* In Computation and Visualization of 3D Vortical and Turbulent Flows, NNFM 64, 350-368, 1998.

[3] J.R. Cebral: *Loose Coupling Algorithms for Fluid-Structure Interaction.* Dissertation, George Mason University Fairfax, Virginia, 1996.

[4] I. Demirdžić, M. Perić: *Space Conservation Law in Finite Volume Calculations of Fluid Flow.* Int. J. for Numerical Methods in Fluids, vol. 8, pp. 1037–1050, 1988.

[5] H. Dütsch, A. Melling, F. Durst: *Coupled Numerical Computations of the Fluid Damped Oscillations of a Lamina.* Lecture Notes in Computational Science and Engineering, vol. 8, pp. 103–112, 1999.

[6] F. Durst, M. Schäfer: *A Parallel Block–Structured Multigrid Method for the Prediction of Incompressible Flows,* Int. J. Num. Methods Fluids, vol. 22, pp. 549–565, 1996.

[7] C. Farhat, M. Lesionne, P. LeTallec: *Load and Motion Transfer Algorithms for Fluid/Structure Interaction Problems with Non-Matching Discrete Interfaces: Momentum and Energy Conservation, Optimal Discretization and Application to Aeroelasticity.* Comput. Methods Appl. Mech. Engrg., 157, pp. 95–114, 1998.

[8] M. Glück, M. Breuer, F. Durst, A. Halfmann, E. Rank: *Computation of Fluid–Structure Interaction on Lightweight Structures.* Fourth Int. Colloq. on Bluff Body Aerodynamics & Applications, Bochum, Germany, Sept. 11–14, 2000.

[9] GRISSLi–Abschlußbericht: *GRISSLi–Projektdokument*, Fin–Rep–01/1999_3_15–1094 im Schwerpunktprogramm "Softwaretechnologie" des BMBF, 1999.

[10] R. Krause, E. Rank: *A Fast Algorithm for Point-Location in a Finite Element Mesh*, Computing 57/1, 49-63, 1996.

[11] H.G. Matthies, J. Steindorf: *Coupling Procedure in Partitioned Methods for Fluid-Structure Interaction.* Private communications, 1999.

[12] SOFiSTiK GmbH, Katz+Bellmann: *ASE Handbuch*, 1988–1995.

[13] W.A. Wall, E. Ramm: *Fluid-Structure Interaction based upon a Stabilized (ALE) Finite Element Method.* Conference Proceedings of fourth WCCM; E. Onate, S.R. Idelsohn (eds.), Buenos Aires, Argentina, June 29 - July 2, 1998; CIMNE, Barcelona, Spain 1988–1995.