# 1 Introduction

1.1 Motivation

The dynamic interaction of a fluid flow with an elastic structure plays an important part in engineering science. Interesting applications can also be found in biomechanical engineering, where for example the blood flow interacting with the abdominal aorta is investigated to prevent an aneurysm, [FVJ+06, SF07]. In civil engineering, wind induced vibrations of bridges need to be suppressed, [BS06]. A fluid-structure interaction (FSI) analysis, may help to find an engineering solution for this demand. A further application of FSI in civil engineering is the computation of a dam failure caused by a seismic excitation [RFPS08, RSFP09]. Such applications are of interest due to the enormity of the destructive power of the water flood wave, which is released after the failure of a dam.

In engineering science the field of *aeroelasticity* mainly driven by aeronautical engineering has been developed for such FSI problems. Quoting [WC07], "Aeroelasticity is the subject that describes the interaction of aerodynamic, inertia and elastic forces for a flexible structure and the phenomena that can result". Aircraft structures in particular are subjected to massively deflect under aerodynamic and inertia loads. Modern lightweight structures and increasing flight speeds amplify the aeroelastic behavior. Aeroelastic problems can mainly be divided into two classes [CD04, Foe74, WC07]:

1. static aeroelasticity
2. dynamic aeroelasticity

Inertia forces can be neglected for the first class and typical problems are: torsional divergence, control reversal, static stability. Typical problems for the second class are: *flutter*, dynamic flight stability, buffeting. Thereby, flutter is a critical aeroelastic problem, which can lead to the damage of the structure. A typical flutter phenomenon is the *panel flutter problem*, Figure 1.1(a), which shows - depending on the flow conditions - a limited flutter amplitude. Such *limit cycle oscillation* (LCO) is caused by a non-linear structural deflection. Furthermore, simplified linear aerodynamic models, in particular strip theory or panel method aerodynamics, are unable to predict shocks in the flow field. Thus, the prediction of the stability in the transonic flow regime becomes inaccurate as shown in Figure 1.1(b), where the flutter speed versus the Mach number for a typical panel flutter problem is plotted. The flutter speed is defined as the lowest flow speed at which flutter occurs. In the transonic flow regime, a significant reduction of this characteristic aeroelastic value can be observed from Figure 1.1(b). Such transonic dip cannot be predicted with linear aerodynamic models and modern fluid dynamics modeling techniques need thus to be used, which solve the Euler or Navier-Stokes equation of fluid motion. Moreover, depending on the flow conditions other long term system responses of the panel might be static deflection, i.e. divergence or an undeflected panel. Thus, the panel flutter problem can serve as a significant model problem for aeroelastic applications to study several numerical schemes, which are going to be discussed in this thesis.
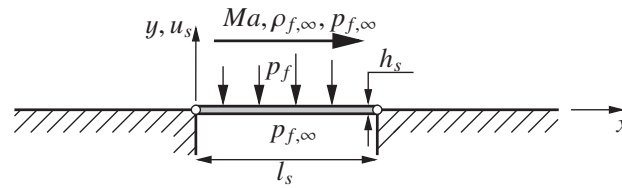
(a) panel flutter problem
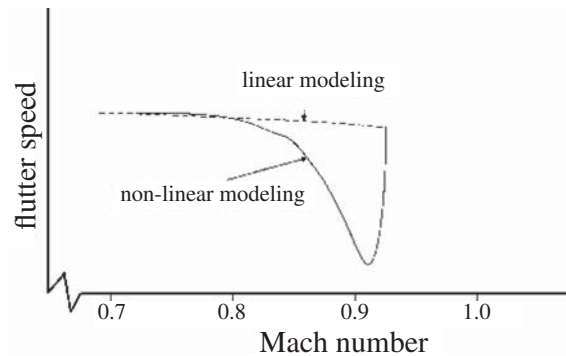
(b) typical flutter behavior of a panel in a transonic flow regime
(taken from [WC07])

Figure 1.1: Description of the panel flutter problem and the typical flutter behavior in a transonic flow regime

Although several non-linear *structural* approaches for the panel flutter problem can be found in literature, [Dow70, ACM99], only some attention has been paid to the non-linear *fluid* part of the panel flutter problem, [DB93]. To consider shock waves in the transonic flow regime, often the Euler equations of fluid motion are solved, [GM00, Mas02]. In [Dow73], a simplified shear layer fluid model is used to incorporate the effect of viscosity. A full Navier-Stokes fluid solver was used in [GV02] for the panel flutter problem, but the flow was assumed to be laminar at a Reynolds number of 10^5 based on the panel length. Some results for the panel flutter problem with a turbulent Navier-Stokes fluid solver can be found in [GV02, Hur01, HAN09, BS08]. That a turbulent boundary layer affects the flutter behavior depending significantly on a supersonic Mach number of the overflowing fluid could be shown in [HAN09].

1.2 Methodology

In the previous section, it could be shown, that aeroelasticity describes a subset of fluid-structure interaction problems and this multidisciplinarity is an important aspect for aeronautical engineering. In this thesis, *computational aeroelasticity* (CA) is understood as the numerical treatment of aeroelastic problems with accurate simulation methods [Far04, Ben04]. Several approaches exist here. In the so-called monolithic coupling approach, one computational algorithm is developed for both - the fluid and the structural subsystem - and the governing equations are treated simultaneously, [Blo98]. Such a method often requires a new code development and is frequently applied only for academic problems, [Hei04]. Since the coupled physical system is treated with one numerical scheme, the *monolithic* approach might show better and more robust computational behavior, [BC10]. However, the most common method to numerically treat an aeroelastic problem is the so-called *partitioned* approach, which is discussed in the next subsection.

1.2.1 Partitioned coupling approach

Applying the partitioned coupling approach each participating subdomain is computationally solved by its own solver [FLM95, Pip97, FL00, MS03, GBD⁺03, vZB05, DP06, VLDV07, SMU⁺10, DSVP⁺10]. Thus, a framework is necessary, into which the fluid and structural solver are integrated. Numerical data have to be transferred from one to the other simulation code to ensure the computational coupling. Moreover and due to the use of separate solvers, the aerodynamic forces and the structural deformations need to be intrinsically in an equilibrium, which requires an iteration procedure.

Frequently, for the second demand of the *equilibrium iteration*, a straightforward Richardson iteration is applied, where the coupled problem is treated as a Dirichlet-Neumann-decomposition, i.e. the structural deformations are set as boundary conditions on the fluid and the aerodynamic forces are imposed on the structural side. However, this simple Richardson iteration procedure might show an unstable behavior or slow convergence rate for strong coupled problems, where a high density fluid acts on a high flexible structure, [CGN05, FWR07]. Thus, high-order iteration schemes, like Netwon-Raphson, Newton-GMRES, etc. are applied for these kinds of applications, which is an active research field, [vZB05, VLDV07]. The most typical aeroelastic problems are weakly coupled, i.e. a relative stiff structure interacts with an airflow. For those weakly coupled applications, a *simple staggered iteration* procedure, where the force and deformation data transmission is accomplished only once in a time step, is generally sufficient [FL00]. The accuracy of this simple staggered scheme can be improved by the usage of *structural predictors*, where the structural deformation of the next time level is estimated based on the structural solution at the actual time level [Pip97, FvdZG06].

While the equilibrium iteration is extensively studied in literature, the topic of data transmission over computational meshes is often treated without significant attention. A comprehensive overview of this essential coupling aspect can be found in [JH04, JJGL05, dBvdSB07, Kim10]. The problem of the numerical data transfer arises due to the usage of different computational grids for the fluid flow and the structural simulations. Thus, the interface is represented by different surface grids which are generally *non-matched*, i.e. the nodes and elements are non-coincident. Several methods exist and an appropriate transfer scheme should not only satisfy accuracy but also load and energy conservation, i.e. neither artificial load nor energy should be generated when transmitting data from one to the other interface grid, [dBvZB08, dB08]. Furthermore, the accuracy of the structural displacements transfer is frequently disregarded. In [dB08], an example of an unsatisfied deformation transfer and its effect on the accuracy of the solution are given by a problem of a flexible flap coupled with an incompressible fluid. For multiphysical problems involving a compressible fluid, an inaccurate displacement transmission can lead to unintentional artificial shocks close to the interface when the fluid interface grid is much better resolved than the structural counterpart [UHH07b].

The most common transfer method is the nearest neighbour interpolation, [TBU00], where the datum of a node from mesh one is simply set to the nearest node of mesh two. Obviously, such a procedure is neither load nor energy conservative. A further class of transfer schemes are projection methods like the conservative node interpolation, [FLL98] or *quadrature point interpolation*, [CL97]. A variation of the quadrature point interpolation was proposed within the field of contact mechanics by Puso in [Pus04], which uses *dual-Lagrange multipliers* to obtain more efficient transfer schemes. Furthermore, geometric splines are frequently used for the data transmission across non-matching interface meshes, [SCH00].

1.2.2 Finite-element methodology for the fluid flow

Another difficulty of using a partitioned coupling approach arises due to the application of inconsistent numerical schemes to simulate aeroelastic phenomena. For example, often a *finite volume* scheme for the computation of the fluid flow, [Bla06, FP01], is utilized in conjunction with a *finite element* methodology (FEM) for the structural part, [Hug00, ZTZ05, Bat96]. Due to this fact, difficulties might be encountered to design a mathematically correct transfer scheme. Therefore and for reasons of numerical consistency, a numerical coupling scheme is employed, where the fluid subdomain is also discretized with finite elements. In the context of the *discontinuous Galerkin* method such finite element approximation becomes a key method for solving the governing fluid equations in the near future, [BO99, ZTSP03, Coc03, BCRS05, Har06, LBL08, HHLP10].

The finite element method for fluid flow problems has been established in the last twenty years, while first attempts were made by Zienkiewicz, [ZC65], Oden, [Ode72], or Chung, [CC76] with a *continuous Galerkin* method. To use the standard Petrov-Galerkin form (weight functions are equal to interpolation functions), stability terms need to be added to suppress instabilities caused by the convective terms of the fluid equations. Such stability terms are introduced naturally by the *characteristic based split* (CBS) scheme proposed by Zienkiewicz and Codina [ZC95, ZMS$^+$95], which uses a local Taylor expansion to design a computational effective scheme. A comprehensive overview of this scheme is provided by Nithiarasu in [NCZ06] and nowadays the CBS scheme is widely used to solve the compressible and incompressible fluid flows. In [Nit03, NMWM04] incompressible flows are treated via an *artificial compressibility* method. Turbulent incompressible flow with the aid of the CBS scheme is investigated in [NHM$^+$08]. Inviscid compressible flows treated with the CBS scheme is analyzed for a wide range of Mach numbers in [TN05].

In the context of FSI problems, the concept of artificial compressibility is advantageous, due to the observation of the so-called *added mass effect* when simulating an elastic structure coupled to a pure incompressible fluid flow, [CGN05, FWR07]. With an incompressible fluid, Poisson's equation is solved for the pressure unknown and thus a disturbance caused by the elasticity of the structure can propagate with infinite velocity, i.e. the speed of sound is infinity. This infinite velocity is responsible for the added mass effect, which cause numerical difficulties. Recently, it could be proven, that this fact is responsible for the requirement of an equilibrium iteration with slow convergence rates, [vB09]. In the same paper, it was revealed, that with a finite speed of sound a simple staggered iteration procedure is possible and the added mass effect can be reduced by a smaller time step. With the artificial compressibility, the infinite value for the speed of sound is replaced by a finite numerical value and thus the CBS scheme can be used for incompressible fluid without any restrictions. This is motivated by the findings in [FRWB10], where a stable and robust simple staggered scheme could be designed as long as the fluid is basically modeled as a compressible fluid.

The discontinuous Galerkin method has been developed mainly for problems in fluid mechanics and combines features of the finite element and the finite volume schemes. Indeed, the finite element method is frequently criticized to violate local conservation of the primary fluid quantities, [HSBB06]. The discontinuous Galerkin method can ensure such *local conservation* by a proper treatment of the element edge flux. A good overview of this methodology can be found in [Coc03, Li06, BCRS05] and the references therein. One noteworthy advantage of the discontinuous Galerkin method is the possibility of easy parallelization of the algorithm since this method allows an element-by-element solution procedure. However, each node belongs to several elements and therefore multiple solutions for each node need to be stored, which results in a large memory requirement. Further, additional edge fluxes for each element have to be computed, which makes the discontinuous methodology usually more computationally expensive than its continuous Galerkin counterpart. On the other hand, solving a large system of linear algebraic equations

necessary for the continuous Galerkin CBS scheme is needless for the explicit discontinuous version, proposed in this thesis. In [TNB08], the discontinuous Galerkin method to the CBS scheme for laminar incompressible fluids was applied and the accuracy for typical testcases shown.

Finally, a consistent usage of finite elements as the spatial discretization method is motivated by the geometric flexibility of this method and the inherent possibility to impose physical boundary conditions, [GSE98, Loe08]. A further advantage of the FEM is the straightforward development of higher order spatial discretizations. In this context and from the mathematical point of view, a rigorous convergence theory is available, e.g. [Hug00, ZTZ05, Bat96]. Furthermore, in [GSE98, Chu02] the finite element method is seen as a generalized finite volume method, which underlines the general character of the FEM.

1.3   Research aim

Two different points of view are considered for the research aim in this thesis.

From the numerical standpoint, consistency is an important requirement for a system's convergence, especially when a system should be analyzed as accurately as possible with modern computational tools. Consistency of a numerical scheme is usually defined as the property, that the discretized equations converge to the underlying differential equations if the time step as well as the element size approach zero [Bla06]. Therefore, a unified coupling approach is developed in this thesis, which is in its discrete form spatially as well as temporally consistent. The question arises whether such unified coupling approach can be designed in a partitioned way so that existing and well-established algorithms can furthermore be used. Thus, a continuous energy based, variational principle for coupled problems is employed, which serves as the fundament for the consistent spatial discretization with finite elements. The choice of the FEM is motivated by a long history of this scheme in structural mechanics and by recent numerous research activities of Galerkin methods in fluid dynamics. The partitioning of the system leads to the consideration of several coupling aspects, e.g. data transfer, fluid grid deformation, equilibrium iteration and time integration.

With the requirement of keeping the consistency of the discrete system and with the variational principles as the fundament, some new ideas regarding these coupling aspects are considered and existing schemes are revisited and improved. Especially with the three-field approach, an attractive smooth transfer of the interface values can be employed to overcome unphysical numerical effects in the fluid flow solution. Further, the development of the CBS scheme for moving and deforming fluid domains is a topic to be answered in this thesis. To obtain a temporal consistency of the simulation, a classical fluid time integration scheme is adapted to the structural subdomain of an aeroelastic system. Finally, the question arises whether with higher order finite elements, an improvement of a simulation's accuracy can be reached. In this context, a discontinuous Galerkin form of the CBS scheme is required to implement, which yields a matrix-free scheme for the fluid part of the coupled simulation.

From the aeroelastic point of view, the transonic panel flutter problem is still an interesting and important model problem, which is characterized by structural as well as aerodynamic non-linearities. As already mentioned, many of the simulations found in literature have been conducted with an inviscid fluid flow model. The effect of a boundary layer is the topic to be answered in this thesis and therefore, the panel flutter problem is investigated with a modern CFD algorithm, which solves the fluid equations in the framework of the Reynolds-averaged Navier-Stokes model. The proposed numerical methods (three-field approach, consistent time integration) are used to ensure accurate simulations and a turbulence model needs further to be applied for compressible flows in the context of the CBS scheme. Some insights into the importance of a viscous fluid boundary

layer on the stability boundary should be provided and thus the panel's LCO is also intensively investigated with an inviscid flow.

1.4   Thesis outline

To draw the introduction to a close, this thesis is divided into two parts: in a first, the continuous coupling approach and its discretization with finite elements are presented. Within the second part numerical examples and verification are shown. More precisely, a unified coupling approach based on a classical variational principle of stationary action is introduced in chapter 2. This principle is known as Hamilton's principle and it is used to consistently express the structural and fluid subsystem by a variational principle. The fluid subsystem itself is described by Hamiltonian fluid dynamics in an Arbitrary-Lagrangian-Eulerian frame of reference. The interaction of both subsystems is then further expressed via a weak formulation of the interface displacement equality.

The spatial and temporal discretization of the fluid and structural subsystem as well as the interface displacement transfer are discussed in chapters 3 and 4. Thereby, the discretization of the fluid subsystem is dedicated to an extra chapter (chapter 3), where the characteristic based split scheme is used to numerically treat the Navier-Stokes equation for the ALE frame of reference. A discontinuous version of this scheme is furthermore proposed in this chapter. The discretization of the whole coupled aeroelastic system is discussed in chapter 4, where an operational view of the system is introduced. This operational view is used to express the equilibrium iteration and the time integration of the coupled system. Further, data transfer methods over non-matching interface grids as well as a constistent time integration for the structural subsystem are proposed.

Within the second part of this thesis, the fluid solver is verified for several flow problems ranging from inviscid, viscous and turbulent as well as compressible and incompressible fluid flows, chapter 5. The whole coupling environment is verified and assessed in terms of the fluid grid deformation algorithm, the data interface approach in chapter 6 and the equilibrium iteration procedure. A smooth data transfer is proposed here, which use the three-field approach in conjunction with a higher order spatial discretization of the interface frame.

Finally, the methods proposed in this thesis are examined for the panel flutter problem, which shows a limit cycle oscillation at various flow conditions in chapter 7. Moreover, the panel flutter behavior is investigated in a transonic flow regime and additionally for turbulent flows.

In chapter 8, this thesis is summarized with a conclusion and an indication of future work is discussed. In the appendix A, some further informations are provided, where some mathematical notations are explained and the turbulence modeling with the aid of the CBS scheme are discussed in detail.