M: Capturing and Designing for Aeroelastic Effects

Lead: Mehdi Vahdati, Loic Salles, Norbert Hoffmann (Imperial College London)

Overview. This technical pillar focuses on developing a new approach for aero-structural analysis of turbomachinery to provide the large gas generated loads for complex system vibration (WP3) and impact and failure cases (WP2). The aim is to develop a new approach that is able to overcome the key barriers in terms of numerical capability that current methods used for aeroelasticity analysis in turbomachinery industry are facing. Today's methods are based on finite volume discretisation for fluid dynamics and finite element methods for structural analysis, but a key deficiency is their inability to deal with very large deformations or changing topology. For example, during high vibration amplitude, the gaps between rotors and stators close and change the topology of the mesh, or in extreme events the damaged components may be severely deformed or even become separated. The demands of large scale analyses leads to poor numerical efficiency, especially due to a lack of poor scalability to parallel computer architectures. To overcome these difficulties, meshless techniques will be developed to provide a step change in capability and better performance on anticipated computing architectures.

The background and motivation. The development of the fluid-structure interaction capability at the Rolls-Royce Vibration University Technology Centre started as early as in the early 1990s. The main research challenges were to understand and predict the complex vibration mechanisms which can occur within a gas turbine due to the air flow. An example of this is flutter of fan blades in which the blade motion causes changes to the aerodynamics, which in turn affects the structural response. A CFD methodology (known as AU3D and based on unsteady compressible flow in rotating geometries) was developed to model details of the interaction between the air flow and the structure. It calculates the instantaneous state of the fluid and the structure and allows each to interact as the solution proceeds. On the structural side, another team has focussed on non-linear contact and friction and their effect on overall system response. A unique computer based prediction method (called FORSE) has been developed to model structural interfaces with emphasis on the overall dynamic response. The research on fluid-structure interaction has revolutionised the approach to predicting the vibration behaviour and is now in routine use at Rolls-Royce across their major development sites throughout the world. Processes and software (AU3D/FORSE) developed by the Vibration UTC have been applied to the design of the majority of Rolls-Royce civil & military aero-engines, including all versions of the Trent family for the large civil engine market. Despite the successes of the above numerical procedure, there are still areas which cannot be modelled numerically and require experimental testing. Typically, these areas involve large deformations of the structure or contact between different parts of the structure, and will result in damage to the engine (or the test rig), and hence are very costly. For industries such as aero-space, who are looking to impact the current and the future designs through numerical simulation to reduce the cost of testing alternative numerical models are required.

The major shortcoming of traditional numerical discretization techniques, (such as AU3D) used in computational fluid dynamics (CFD) comes from the fact that they rely on the use of grids as the underlying structures on which the Navier-Stokes equations are discretized For such methods 'good quality' grids are essential for stability and accuracy. The term 'good quality grid' means a grid which is fine enough to model all the geometric and flow features, while it is smooth and uniform enough so that the CFD solver can operate. For the problems involving large deformations or contact, the quality of the grid in the CFD domain will degenerate (or collapse in case contact) which results in numerical stability and accuracy issues. To overcome the sketched deficiencies of mesh-based methods, a unified approach integrating geometry description and

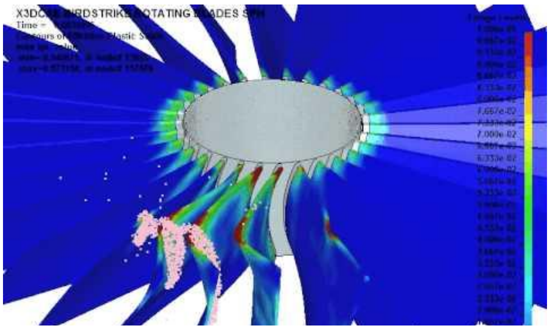

Figure 1. Fully coupled fluid-structure interaction event. [MSC Software.]

solution of the underlying physical problem needs to be achieved.

To outline and initiate such an approach for fluid-structure interaction in turbomachinery is the objective of this work.

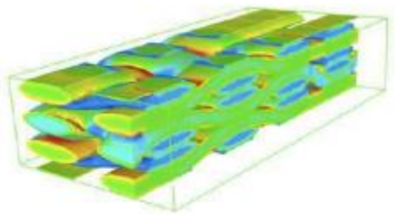

Figure 2. Coupled fluid-structure modelling of complex multi-scale composite materials. [SNECMA.]

Aerodynamic Meshless Method.

A large part of the benefit in terms of numerical efficiency in meshless CFD lies in relieving the burden of mesh generation. Meshless schemes only require clouds of points, from which the Navier-Stokes equations may be discretized. Local clouds for each point in a domain are proximity-based subsets of the global set of points. Local clouds of points replace the more traditional forms of connectivity found in FEM, FDM, and FVM. It is evident that removing the need for a mesh (connectivity) will make the discretization of the computational domain much easier and will ease capturing more complex geometries. Also for non-stationary and transient problems involving moving geometries, such methods clearly have striking advantages.

Fluid-Structure Interaction Meshless Method.

The main issue of the current Fluid Structure interaction approach is the difference between the modelling and approximation methods used for the fluid and the one for the structure. In this part of the project we propose to develop improved fluid-structure coupling approaches to leverage the developments on WP2 & 3 and a new approach to unify the two methods by using a common meshless technique for both fluid dynamics and structural dynamics. Early work towards such a unified approach has already been explored for damage due to foreign objects, as e.g. shown in Figure 1. This unified approach will permit the study of aerodynamic effects during fan blade off or the change of flow during blade tip rub. Meshless methods will allow the study of large amplitude displacement of the blade in the aeroelastic analysis. New engine architectures will also extensively make use of composite materials, and the internal material damping will add an extra degree of complexity to the structural dynamics. Meshless methods avoid the expensive preprocessing process and make it possible to capture the fine-structure of composite elements directly, see Figure 2. In terms of novel computer architectures, the use of meshless techniques will permit to take advantage of the new computer generations based on multicore architecture, especially graphical processing units (GPU) and new cluster generations.

Main elements of work.

Task #1 (months 1-6): Development and validation of a strategy to implement meshless methods in fluid-structure interaction of turbomachinery for new parallel computer architectures. Task #2 (months 7-30): A CFD solver using meshless grids is developed and validated. This Code will form the basis of the aeroelastic solver. Task #3 (months 7-30): Develop the meshless method for structural non-linear dynamics. Task #4 (months 24-36): Couple both meshless solvers for aeroelastic analysis. Task #5 (months 24-36): Integrate the spectral time domain method in the CFD code.

Overlaps with other parts of the programme.

The present work-package is intimately intertwined with the other project partners at Oxford and Nottingham. With Oxford work on contact mechanics and wear will be conducted to improve structural contact modelling relevant to capture and predict structural vibration damping. The Oxford work on impact dynamics will complement the Imperial work on vibration and aero-elastic interaction. The joint work with Nottingham on bearings, rotordynamics, and thermal effects will play a key role for the present modelling of large assembled systems, and the component models developed in Nottingham will be used in the vibration and fluid-structure interaction approach.

Interactions with other Universities

The work will also build on existing and proven collaborations with leading experts on mesh-less computational techniques, including Paul Tucker from the Whittle Lab at Cambridge, and Robert E. Kielb from Duke University (USA).