11th World Congress on Computational Mechanics (WCCM XI)
5th European Conference on Computational Mechanics (ECCM V)
6th European Conference on Computational Fluid Dynamics (ECFD VI)
July 20–25, 2014, Barcelona, Spain

# NODE-BASED VS CAD-BASED APPROACH IN CFD ADJOINT-BASED SHAPE OPTIMISATION

**Mateusz Gugala*[1], Shenren Xu[1] and Jens-Dominik Müller[1]**

[1] Queen Mary, University of London The School of Engineering and Material Science
Mile End Road, London, E1 4NS, UK,
*m.gugala@qmul.ac.uk

**Key words:** *Optimisation, Adjoint, Parametrisation, CAD-based, Node-based.*

Adjoint-based optimisation makes the cost of computing the gradient independent of the number of design variables, which in turn removes the need to construct a small design space with few parameters that hopefully expresses significant design modes. Current adjoint solvers are becoming more and more capable for steady flow optimisation cases. i.e. flows with negligible time dependence of the cost function, which is handled effectively by the use of a RANS solver. However, many industrial cases have an unsteady nature which has a significant impact on the quantities of interest. That implies a need of developing an unsteady adjoint solver for unsteady optimisation problems. For both flow types the choice of design space parametrisation follows the same considerations. A number of automatically derived parametrisations with large numbers of degrees of freedom have been proposed. Two interesting ones are the node-based and CAD-based approaches.

The node-based approach [1] uses the movement of every surface point of the CFD mesh as a design variable and hence offers the richest design space that the CFD solver can possibly analyse. However this parametrisation produces oscillatory shapes, which requires regularisation for either gradient or shape to damp high frequency shape modes. On the other hand this smoother gives some flexibility on deciding how much dumping we want by changing smoothing parameters or number of smoothing iterations. Furthermore, the result of node-based optimisation is the shape of a surface mesh and additional work is required to transform this back to CAD.

The CAD-based approach as proposed by Xu et al. [2] uses the control points of the NURBS surface patches as design variables. It hence represents the richest design space that the CAD can express. The CAD description is kept inside the design loop. Additional constraints have to be imposed on control point displacements to maintain geometric continuity between patches. Additionally the process of creating geometry in CAD system usually requires more attention from an engineer, so that the geometry is not only smooth for a humans eye, but also the mathematical description and representation is clean and tidy within the specified and required for given case tolerances. As the CAD parametrisation typically is much coarser than the CFD mesh, the projection onto a coarser design space typically provides adequate high-frequency filtering. The output of optimisation is directly in CAD form, which allows to reduce time of design process.

The two approaches are discussed and compared. Using an in-house adjoint solver the approaches are demonstrated on a number of test cases such as pressure loss minimisation of an S-Bend duct and lift constrained and geometrically constrained drag minimisation of the M6 wing.

**Acknowledgement**
This work is part of aboutFlow project - http://aboutflow.sems.qmul.ac.uk/
This project has received funding from the European Unions Seventh Framework Programme for research, technological development and demonstration under grant agreement no [317006].

## REFERENCES

[1] Faidon Christakopoulos. *Sensitivity computation and shape optimisation in aerodynamics using the adjoint methodology and automatic differentiation.* Ph.D. thesis, Queen Mary University of London, 2012.

[2] Shenren Xu, Wolfram Jahn, Jens-Dominik Müller. *Cad-based shape optimisation with cfd using a discrete adjoint.* International Journal for Numerical Methods in Fluids (2013).