Shape Optimization for Aerodynamic Efficiency Using Adjoint Methods

Adjoint solvers take a Computational Fluid Dynamics (CFD) flow solution and calculate the sensitivity of performance indicators (for example, drag or heat transfer) for all of the design variables. Obtaining an adjoint solution gives designers key information on how to alter the shape of the design to improve the quantity of interest. Mesh morphing is an easy and powerful tool that allows designers to alter the geometry at the mesh level and within the parallel solver to evaluate effects of the design alterations. Designers can then iterate on this process to achieve an optimum design without the need to return to the original CAD geometry. For complex geometries, the aerodynamic interactions become complex and the insight gained by adjoint solvers can take the design in unexpected directions that are difficult to represent using parametric geometry.

INTRODUCTION

Transportation is a major consumer of the world’s energy. Nearly a quarter of the total United States’ energy use is in transportation, and a significant portion is spent by aerodynamic drag. The aerodynamic drag of a vehicle plays a major role in determining the overall efficiency that the vehicle can achieve. For example, nearly 22% of a typical truck’s fuel is spent on overcoming aerodynamic drag. Consequently, any improvements in aerodynamics can have a significant improvement in the world’s total energy consumption, and yield savings to vehicle operators.

Stringent government regulations — such as the Corporate Average Fuel Economy (CAFE) norms for automobiles in the U.S. — and a strong consumer demand for cheaper, greener transportation has increased the urgency for aerodynamic improvements. For instance, airline operators work with very fine profit margins, typically around 1%. Recent volatility in fuel price makes their business environment even more challenging. In parallel, the aerospace industry has committed to aggressive environmental targets — such as the CleanSky initiative, which mandates a 50% reduction in CO2 emissions and an 80% reduction in NOx emissions by 2020. Consequently they demand the most fuel efficient aircraft possible from the aircraft OEMs and their suppliers. Using Computational Fluid Dynamics (CFD) to assess the efficiency of a design is common practice. By combining with an adjoint solver and the latest mesh morphing technology, designers can optimize designs quickly in unexpected directions that are often beyond those obtained from parameter-based optimization.

This paper presents an overview of adjoint technology and explains the benefits, including recent advances allowing multi-objective design assessments and methods for solving large models.
Shape Optimization for Aerodynamic Efficiency using Adjoint Methods

It also presents a case study that uses these recent advances to optimize the aerodynamic efficiency of a passenger vehicle over the New European Driving Cycle (NEDC). While this case focuses on an automotive application, the techniques discussed here work equally as well for aerospace simulations.

ADJOINT SOLVER TECHNOLOGY

Overview

An adjoint solver is a specialized tool that extends the scope of a conventional flow solver by providing detailed sensitivity data for the performance of a fluid system. This section gives an overview of the technology; more in-depth details of the calculations are have been presented previously [1, 2].

To perform a simulation using a flow solver, you supply system geometry in the form of a computational mesh and specify the problem, physics and boundary conditions. The conventional flow solver provides a detailed data set that describes the flow state governed by the flow physics specified and various post-processing steps can be taken to assess the performance of the system.

Making a change to any of the inputs that define the problem can cause the results of the calculation to change. The degree of that change depends on how sensitive the system is to the particular parameter that is being adjusted. Indeed, the derivative of the solution data with respect to that parameter quantifies this sensitivity to first order. Determining these derivatives is the domain of sensitivity analysis.

There is a large collection of derivative data that can be computed for a fluid system, given the extensive set of input data that is required, and the extensive flow data that is produced. The matrix of derivatives for output data with respect to input data can be vast and — depending upon the goal of the analysis — only a portion of this derivative data may be of interest.

An adjoint solver accomplishes the remarkable feat of calculating the derivative of a single engineering observation, with respect to a large number of input parameters, simultaneously in a single computation. The engineering observation could be a measure of the system performance such as the lift or drag on an object, the heat flux from a surface or the total pressure drop through a system. However, the key principle is that the solver finds derivatives with respect to the geometric shape of the system, allowing the sensitivities to be evaluated.

Understanding these sensitivities in a fluid system can provide extremely valuable engineering insight. A system that is highly sensitive may exhibit strong variability in performance due to small variations in geometry and/or variations in the flow condition. Additionally, calculating the sensitivities of a fluid system satisfy a central need in gradient-based shape optimization, making an adjoint solver a unique and powerful engineering tool for design optimization.
Shape Optimization for Aerodynamic Efficiency using Adjoint Methods

Once you compute an adjoint solution, the derivative of the observable with respect to the position of each and every point on the surface of the geometry is available, and you can find the sensitivity of the observation to specific boundary condition settings.

The power of this methodology is highlighted when you consider alternatives for assembling the same information. Imagine a sequence of flow calculations in which you move the points on an aerofoil surface, set a small distance in the surface-normal direction, then recompute the flow and drag. If there are N points on the surface, then N flow calculations are required to build the data set. Considering that the same data is provided by a single adjoint computation, the adjoint approach has an enormous advantage, as for even modest 3-D flow computations there may be many thousands of points or more on a surface.

You can then use the computed adjoint sensitivities to guide intelligent design modifications to the system because the adjoint sensitivity data provides a map across the entire surface of the geometry. Design modifications can be most effective if made in regions of high sensitivity since small changes will have a large effect upon the engineering quantity of interest. Iteratively applying this principle of making changes to a system in proportion to the local sensitivity is the foundation for the simple gradient algorithm for design optimization as illustrated in Figure 1. This workflow has the advantage over the traditional optimization workflow; it is conducted entirely with the parallel solver, and therefore does not have the overhead of modifying geometry, remeshing and reading/writing files between the different stages.

2-D Cylinder example
Consider a 2-D analysis of a circular cylinder subjected to a cross flow and bounded top and bottom by symmetry planes. The flow is laminar and incompressible with a Reynolds number of 40 based on the cylinder diameter. The problem is posed at a Reynolds number where the flow is steady, as can be seen in the velocity plot shown in Figure 2.

If we consider the corresponding adjoint solution for the drag in the X direction on the cylinder, we can see in Figure 3 how sensitive the drag on the cylinder is to the application of a body force in the X-direction in the flow. Here, the figure shows that the effect of applying a body force downstream of the cylinder is minimal. If a body force is applied directly upstream of the cylinder, however, the disturbed flow is incident on the cylinder and modifies the force that it experiences.

Figure 4 shows how sensitive the drag on the cylinder is to changes in the surface shape. The drag is affected more significantly if the cylinder is deformed on the upstream rather than the downstream side. As could be expected in this simple example, the maximum effect is achieved by narrowing the cylinder in the cross-stream direction.
If the sensitivity for the lift force in the y direction is considered as a second objective in addition to the drag and the design change required to reduce the drag and increase the lift can be calculated, the surface of the cylinder can be morphed as shown in Figure 5 to improve these objectives.

Recent Advances
In the past, the adjoint solver has been primarily used by the Motorsport arena [3-5]. Slower adoption of this technology in the automotive community has been due primarily to the difficulty in obtaining convergence in the adjoint solution for more unsteady problems with highly separated wake regions. However, there have been a number of recent advances in adjoint solver technology that have improved the landscape for this class of flow.

Stabilization
The most recent development is the dissipation scheme, which provides stabilization for the solution advancement of the adjoint solution by introducing nonlinear damping strategically into the calculation domain. The strategy provides minimal intervention to damp the growth of instabilities that lead to adjoint solution divergence. A marker is tracked and, based on the state of the adjoint solution, a damping is applied directly to the adjoint solution in regions where the marker becomes large.

The dissipation scheme handles both weak and strong instabilities with low memory and computational requirements and no user intervention. However, unlike the modal and spatial schemes, the dissipation scheme can slightly affect the adjoint solution, although the spatial order of the damping is chosen to be one order larger than of the adjoint calculation. This means that the formal order of accuracy of the adjoint solver is unaffected by the addition of the dissipation scheme. When applied to problems with large cell counts and complex geometry the adjoint solver sometimes experiences stability issues. These instabilities can be associated with small scale modes of unsteadiness in the flow field and/or strong shear, and tend to be restricted to small and isolated regions of the flow domain. Despite the spatial localization of these instabilities, the linearity of the adjoint problem provides no intrinsic limit on their growth during solution advancement. Therefore their presence can disrupt the entire adjoint calculation even though the problem may occur in just a few cells.

This means that solution stabilization is required to obtain adjoint solutions for problems at high Reynolds number in which there is strong shear and/or complex geometry.

Three stabilization schemes have been developed to overcome these stability difficulties when complex cases are being solved, and are designed to intervene only when the standard advancement scheme is experiencing instability.
The first stabilization scheme to be developed was the spatial scheme. It identifies parts of the domain where unstable growth occurs and applies a more direct and stable solution procedure in those regions.

While this can work well for some local, weak or unstable behavior, it can have a high memory overhead.

The next scheme is the modal scheme. This involves a process of identifying the particular details of the unstable growth patterns or modes. These patterns are localized in space and are used to split the solution into parts that have stable and unstable characteristics when advanced. The stable part is advanced as usual, while the algorithm is designed to compensate for the unstable part so that the overall calculation is stabilized.

The advantages of the modal scheme are that it handles strong instabilities, formally solves the adjoint equations and has a low memory requirement. However, it may have a large number of modes to identify and manage during the solution and any changes to solution parameters (under-relaxation, etc.) can change the modes, therefore there is computational overhead associated with this method.

The dissipation scheme is considered the most effective scheme so far for solving adjoint problems, and opens the doors in ANSYS CFD to optimize a wider variety of complex CFD simulations.

**Sub-Modeling**

The adjoint solution assumes that the turbulence is frozen, standard wall functions are used and the solution is valid in the local region of the design point flow solution. Combining this with the smoothing effect of the relatively low resolution of the control points used to morph the design allows the assumption that sub-modeling can be used.

The assumption is that if a coarse model flow field is a good approximation of a detailed model flow field, you can expect that the sensitivity field on the coarse model is a good approximation of the one we would obtain from the detailed model.

Reducing the cell count of a model has the benefit of not only improving the stability of the adjoint solution but also reducing the computational effort of the solution. Therefore, it is additionally a useful tool to speed up the design process when applied to external aerodynamic problems, as is described in more detail in the case study below.
PASSENGER VEHICLE OPTIMIZATION CASE STUDY

This section details a case study that optimizes the aerodynamic performance of a passenger vehicle over the NEDC. The study uses a weighted multi-objective approach to optimize the vehicle over the speeds of the drive cycle and sub-modeling approach to reduce computational overhead.

Vehicle geometry

The vehicle selected as the basis for the case study was the DrivAer Fast-back Detailed underbody with Mirrors with Wheels (F_D_wM_wW) provided courtesy of TU Munich, Institute of Aerodynamics [5]. This was selected as it has previously been the subject of a validation study comparing the analysis results of ANSYS Fluent and ANSYS CFX to experimental data [6].

To reduce the computational overhead, the designers conducting the study created a half-car symmetric model, shown in Figure 6. This method prevents the final design from becoming asymmetric.

Drive Cycle

The aim of the study was to optimize the shape of the vehicle to reduce the Coefficient of Drag (Cd) and therefore the CO2 emissions of the vehicle over the drive cycle. Although Cd is usually quoted as a single value it will vary at different speeds as the flow rate over the vehicle changes.

The first stage of defining the boundary conditions was to assess the influence of each of the vehicle speeds over the NEDC. As can be seen from the graphical representation in Figure 7, the drive cycle contains various speeds and durations. To assess their influence, the designers assumed a representative Cd and calculated the force acting on the vehicle for each speed and then weighted the Cd based upon the duration.

This assessment showed that 90% of the force on the vehicle occurred at speeds 50, 70, 100 and 120 km/h. Therefore, they assumed that the influence of the lower speeds would be negligible for the purpose of the optimization.

Baseline Detailed Model

The work performed by Frank et al [6] showed that using a steady state K-omega SST model with a boundary layer mesh giving a Y-plus below 1 agreed with the experimental data.

The model used in that study consisted of a tetrahedral mesh with 20 prismatic boundary layers on the surfaces of the car and road with an initial height of 0.02mm. The model also had refinement regions surrounding and downstream of the car to capture the flow structures away from the boundary layer.

These mesh sizings were used as the basis for the detailed model used here. However, in this study the designers created a hexcore mesh using ANSYS Fluent Meshing which uses a similar prismatic boundary layer joined to a Cartesian hexahedral outer domain with a layer of tetrahedrons. The hexcore mesh was used in preference to tetrahedrons as it reduced the overall cell count and improves the capture of vertical flow structures.
The resulting mesh contained 42 Million cells, a reduction of 20 million cells over the tetrahedral mesh even with an extended wake refinement region. A plot of the mesh on the symmetry plane highlighting the refinement regions and the prismatic boundary layer is shown in Figure 8.

The designers then solved the model with inlet flow velocities of the four vehicle speeds of interest and corresponding moving wall boundary velocities for the road and wheels. They calculated the solution using the k-omega turbulence model and the pressure-based coupled solver within ANSYS Fluent. The designers then adopted a solution strategy of varying the Courant number to accelerate convergence was adopted. By varying the Courant number between initially 50 and then increasing 400 before finally reducing to 50 convergence is achieved in 300 iterations. They assessed the Cd over a further 100 iterations to account for any oscillations in the solution. The monitor plots and average values of Cd over the final 100 iterations for the four speeds can be seen in Figure 9.

Coarse Model Adjoint Sensitivities and Optimization

As discussed above, to improve the convergence and reduce the computational overhead, a separate coarsened model was used for calculating the adjoint sensitivities.

To create this model the designers used the same Fluent Meshing surface mesh and refinement regions as they used for the detailed model, but meshed the model using polyhedral elements and boundary layer settings they changed to give four layers and first layer thickness for high Y-plus. This resulted in the mesh profile shown in Figure 10 which contains 7.1 Million polyhedral elements, giving a sixfold reduction in the mesh size.

In addition to reducing the size of the mesh, they further simplified the turbulence model by changing it to the k-epsilon model with enhanced wall functions. This change was facilitated by the high Y-plus boundary layer matching the standard wall functions of the adjoint solver and had the added benefit that the k-epsilon model can reduce the unsteadiness in the flow solution, thus improving the convergence of the adjoint solution.

They set all boundary conditions as per the detailed model and solved the model using the same solver parameters as the detailed model. From the results shown in Figure 11, you can see that there is a reduction in the oscillation of the solution, but also a reasonable difference in the Cd calculated. On further investigation, this was found to be due to a difference in the resolution of the flow separation point along the fastback area with the coarse model.

The designers then subjected each of the flow solutions to an adjoint solver calculation with the observable for the sensitivities set to be the drag force acting on the non-rotating surfaces of the vehicle. The adjoint solution settings were left as the default for automatic Solution-Based Controls Initialization and Auto-Adjust Controls. They set the modes to solve for 500 iteration to ensure convergence and that the resulting sensitivity fields were...
Once the sensitivity field for each of the speeds had been calculated, they were combined and a design change calculated with a weighting based upon their contribution to the drive cycle. The plot of in Figure 12 shows contours of normal optimal displacement, showing in red where the geometry should be “pulled” outwards and blue where the geometry should be “pushed” inwards in order to improve the objective. From this, it can be seen that there are a number of regions on the geometry that exhibit sensitivity to the drag force. As areas such as the wheel arches and the underbody often have other consideration and functions than aerodynamic efficiency, the area selected for modification here is the rear portion of the bodywork behind the rear wheel arch.

As these regions are away from the area in which the flow separates from the fastback, it can be considered that the adjoint sensitivities will be similar between the two models and thus the sub-modeling approach used.

The next phase is to follow the iterative workflow shown in Figure 1 to solve the flow solutions, then adjoint solutions and morphing the geometry accordingly. For this case study the optimization loop was repeated for 7 iterations and in each case retaining the sensitivity field obtained for use in the detailed model optimization. Figure 13 shows the output of the drag force observable for the rear of the car during the optimization. This highlights that although the rear of the car exhibits high sensitivity to drag force, the magnitude of the force is only approximately 15% of the total drag force on the vehicle.

The table below shows the Cd values over the final 100 iterations for the baseline geometry and the final iteration of the modified geometry. You can see a comparison of the initial geometry and the final modified geometry in Figure 14.

<table>
<thead>
<tr>
<th>Vehicle Speed</th>
<th>Initial geometry Cd</th>
<th>Modified geometry Cd</th>
</tr>
</thead>
<tbody>
<tr>
<td>50 km/h</td>
<td>0.259</td>
<td>0.251</td>
</tr>
<tr>
<td>70 km/h</td>
<td>0.257</td>
<td>0.251</td>
</tr>
<tr>
<td>100 km/h</td>
<td>0.256</td>
<td>0.248</td>
</tr>
<tr>
<td>120 km/h</td>
<td>0.256</td>
<td>0.247</td>
</tr>
</tbody>
</table>

Detailed Model Optimization

The process for optimizing the detailed model is similar to that for the sub model. However, we already have the sensitivities calculated and therefore it is simply a matter of importing the sensitivities and morphing the geometry iteratively. The resulting modified geometry is also shown in the comparison in Figure 14.

We can then run the modified detailed model using the k-omega settings as before and the results can be seen in Figure 15 which shows a comparison between the detailed baseline and modified models. From these plots we can see that the modified model gives a reduction in Cd of between 3.9% and 6.6%. The plot in Figure 16 showing the iso-surface of total pressure highlights the reduction of the wake structure that resulted in the improvements in Cd.
Conclusions

The adjoint solver in ANSYS CFD provides a powerful tool that can optimize even complex flow systems in non-intuitive directions with little user input. Using sub-modeling reduces the complexity and computational overhead of using the adjoint solver on external aerodynamics flow problems. Combining an adjoint solver and the latest mesh morphing technology can help you optimize your product designs for energy efficiency, satisfying federal regulations and international and customer demand for better, greener products.

REFERENCES