Application
of a
Continuous Adjoint Flow Solver
for
Geometry Optimisation
of
Automotive Exhaust Systems

Christof Hinterberger, Mark Olesen

FLOWHEAD Workshop, Varna • Sept. 2010
Workshop on industrial design optimisation for fluid flow
Abstract

Meeting backpressure and flow uniformity requirements within severe packaging constraints presents a particular challenge in the layout of catalyst inlet cones. In these cases, a parameterized optimization of the potentially complex cone geometries is inefficient (and inappropriate). Even assuming that a parameterization of the complex surface forms is possible, the choice of parametric shapes invariably affects the achievable results. Additionally, the long computation time for solving the flow fields limits the number of shape parameters that can be considered.

To overcome these restrictions, an optimization tool has been developed at Faurecia Emissions Control Technologies [1] that is based on the continuous adjoint formulation derived and implemented by C. Othmer et al [2, 3]. The open source CFD toolbox OpenFOAM® is used as the platform for the implementation. Since the geometry itself is modelled using an immersed boundary method, no geometry parameterization is required. The method allows computation of the sensitivity of flow uniformity and energy dissipation (or other target quantities) based on the instantaneous geometry. After the calculated surface sensitivities are combined and corrected for manufacturing and topological constraints, the location of the immersed boundary is automatically adjusted. It is thus possible to automatically determine a feasible catalyst cone geometry starting from an amorphous box (representing the packaging constraints) that is supplemented by definitions of inflow boundaries (for the flow coming from different manifold runners) and the outflow boundary (the catalyst surface). The calculation time associated with the process is on the same order of magnitude as the solution of the RANS equations itself. The optimization tool and some practical results will be presented.

"Implementation of a continuous adjoint for topology optimization of ducted flows", AIAA-2007-3947

OpenFOAM® is a registered trademark of OpenCFD Ltd.
Outline

Optimization of Catalyst-Cone

- reduce back pressure
- Improve flow uniformity at catalyst

OpenFOAM® based CFD solver

- CAGO (Continuous Adjoint Geometry Optimisation)

Application Examples
Motivation

Challenge for the CFD-engineer:

„find optimal geometry of catalyst inlet cone!“

- good flow distribution + low pressure drop
- suitable for production, costs, development time, …

Mathematical problem description:

„optimize objective function \( J (\gamma, \Delta p, ...) \) “

\[ \Delta p \ldots \text{pressure drop} \]

(fuel consumption)

\[ \gamma \ldots \text{flow uniformity index} \]

(noble metal cost, emission standards)

\[
\gamma = 1 - \frac{1}{2A\bar{V}_n} \int_{cat} |v_n - \bar{v}_n| dA
\]

\( \gamma = 0.8 \rightarrow 80\% \text{ efficency of noble metal utilization} \)
Optimization of Catalyst-Cone

Basic configuration:

- $\gamma = 0.70$
- $\Theta = 0.72$
- $v_{\text{max}} = 117 \text{ m/s}$
- $dp/dx_{\text{max}} = 11 \text{ bar/m}$
Optimization of Catalyst-Cone

\[ \gamma = 0.87 \quad \Theta = 0.93 \]

optimized configuration

\[ v_{\text{max}} = 100 \text{ m/s} \]
\[ dp/dx_{\text{max}} = 7 \text{ bar/m} \]
Main problem: „How to change the geometry?“

- **Analytical solution**
- **Experience**
- **Trial and error**
- **Geometry modification**
  - Basis Optimization
  - Sensitivities $\frac{\partial J}{\partial \text{geom}}$
  - Finite Differences
  - Automatic Differentiation

- **Geometry parameterization**
  - + DOE, RSM
  - $\Rightarrow$ automatic optimizer

- **Adjoint Method**
OpenFOAM® based solver CAGO  
(Continuous Adjoint Geometry Optimisation)

Starting Point

- OpenFOAM-solver from C. Othmer, H. Weller, E. de Villiers
- Theory paper from C. Othmer

Modifications

- topology conserving geometry variations
- discrete fluid/solid distinction
- wall functions at fluid/solid-interface
- multiple adjoint flow fields (pressure drop, uniformity & centricity)
- multiple flow fields (different ports of naturally aspirated engine)
- details can be found in SAE paper 2010-01-1278
CFD model

- Wall functions for high Re k-ε model

- Packaging space

- Manifold

- Catalyst

- Inflow BC

- Outflow BC

- Immersed boundary

- Solid

- Fluid
Flow equations

**NSE**

momentum: \((u \cdot \nabla) u = -\nabla p + \nabla \cdot (\nu (\nabla u + (\nabla u)^T))\)

continuity: \(\nabla \cdot u = 0\)

**Adjoint**

Adj. momentum: \(- (\nabla u^* + (\nabla u^*)^T) \cdot u = -\nabla p^* + \nabla \cdot (\nu (\nabla u^* + (\nabla u^*)^T))\)

Adj. continuity: \(\nabla \cdot u^* = 0\)

**Implementation of Momentum Eqn. in OpenFOAM**

**NSE:**
\[\text{fvm::div( } \phi, \nu) + \text{turbulence->divDevReff(u) } = \text{-fvc::grad}(p)\]

**Adjoint:**
\[\text{fvm::div( } - \phi, u^*) - \text{fvc::grad}(u^*) \& u + \text{turbulence->divDevReff}(u^*) = \text{-fvc::grad}(p^*)\]

Examples

Simple 2D Cases
example 1)
flow from the side -> analytical solution

Bernoulli-flow

\[ p_{\text{total}} = p_{\text{static}} + p_{\text{dynamic}} = \text{const} \]
\[ p_{\text{dynamic}} = \frac{\rho}{2} v^2 \]

Catalyst Cone

\[ v_{\text{cat}} = \text{const.} \Rightarrow p_{\text{static}} = \text{const.} \Rightarrow p_{\text{dynamic}} = \frac{\rho}{2} v_{\text{cone}}^2 = \text{const.} \Rightarrow v_{\text{cone}} = \text{const.} \]

\[ \dot{m}_{\text{cone}} = \dot{m}_{\text{cat}} \]
\[ \dot{m}_{\text{cone}} = \rho A_{\text{cone}} v_{\text{cone}} \]
\[ \dot{m}_{\text{cat}} = \rho A_{\text{cat}} v_{\text{cat}} \]

\[ A_{\text{cone}} = A_{\text{cat}} v_{\text{cat}} / v_{\text{cone}} \]

\[ A_{\text{cone}} \sim A_{\text{cat}} \]
example 2) unsuitable inflow

→ solution: Adjoint Method
Adjoint Optimization Method

\[ \gamma = 0.9455 \]
Optimisation for energy dissipation

Iteration 24

flow velocity $v$

adjoint velocity $u_1$

sensitivity $s_1$
energy dissipation

$v \cdot u_1$

1.00
0.10
0.01
0.00
-0.01
-0.10
-1.00
Optimisation for uniformity

Iteration 460

flow velocity \( v \)
adjoint velocity \( u_2 \)
sensitivity for flow uniformity
\[ s_2 = v \cdot u_2 \]

Current geometry
New geometry
New blocked cells
Solid region
Area kept free
Catalyst

Uniformity \( \gamma = 0.93 \)
Final geometry
Application Examples

(pages with proprietary information have been removed)
Complex Design space

- design space meshed with 300,000 polyhedral cells
- < 10 hr simulation runtime on single CPU (3.3 GHz)
- achieved excellent uniformity $\gamma = 0.956$
Summary

CAGO (Continuous Adjoint Geometry Optimisation)

- Innovative form optimization tool
- not restricted by predefined shape functions
- robust and very fast
- helps us to find solutions for a given packaging
Technical perfection, automotive passion

faurecia