

A Study on an Efficient Duct Component Evaluation Method Using Fluid Topology Optimization

Yoshio Itoh ¹⁾ Satoshi Ito ¹⁾ Fumiya Shiratsu ²⁾ Masahiko Kono ²⁾ Toshiki Watanabe ²⁾

*1) TOYOTA MOTOR CORPORATION
1, Toyota-cho, Toyota, Aichi, 471-8572, Japan*

*2) TOYOTA SYSTEMS CORPORATION
1-1-1 Meieki, Nakamura-ku, Nagoya, Aichi, 450-6332, Japan*

KEY WORDS: Vehicle Development, CAD/CAM/CAE, Design Optimization, Heat • Fluid, CFD, Air Conditioning, Experimental Validation [B2]

1. Introduction

In automotive HVAC development, designers reduce duct pressure loss by repeatedly performing CFD analysis and CAD modification. This iterative process depends heavily on the designer's experience and requires substantial man-hours. In addition, the influence of the outlet boundary condition—particularly downstream elements such as register fins—on the validity of the CFD result has not been sufficiently examined. This study proposes an efficient duct-shape generation technique using fluid topology optimization that designers can readily apply, and verifies the effect of the outlet boundary condition through CFD and component testing.

2. Method

Fluid topology optimization progressively removes low-velocity (low-contribution) regions from a predefined design domain based on the flow field obtained by CFD, and automatically generates the flow path. Under an inlet volumetric flow of 120 m³/h, an atmospheric outlet pressure (101,325 Pa) and 293.2 K, two optimizations were performed: (1) Optimization #1, computed for the duct alone without the register, and (2) Optimization #2, in which the register fins were modeled at the outlet. The design shape, Optimization #1 and Optimization #2 were fabricated by 3D printing and evaluated on a component test bench under register-off and register-on conditions (five patterns in total).

3. Results and Discussion

The CFD and experimental pressure losses agreed well. Optimization #1 (optimized without the register) reduced pressure loss under the register-off evaluation but conversely worsened it under the register-on evaluation, showing a reversal (Fig. 1). In contrast, Optimization #2 (optimized with the register) reduced pressure loss under the realistic register-on condition. This indicates that optimization should be performed with an outlet boundary condition close to the actual operating state, i.e., including downstream elements. Sectional analysis showed that Optimization #2 markedly reduced the loss near the outlet, with the (6)–(7) section decreasing from 16.65 to 2.98 Pa (82% reduction). This is attributed to the flow being distributed so as to reduce the mixing loss, which uniformized the outlet velocity distribution and lowered the system-level loss.

Furthermore, the proposed workflow eliminated the CAD/CFD iteration and, even including the man-hours to incorporate manufacturing and supplier requirements, reduced the design study effort by about 24 h (54%) per development phase (Fig. 2).

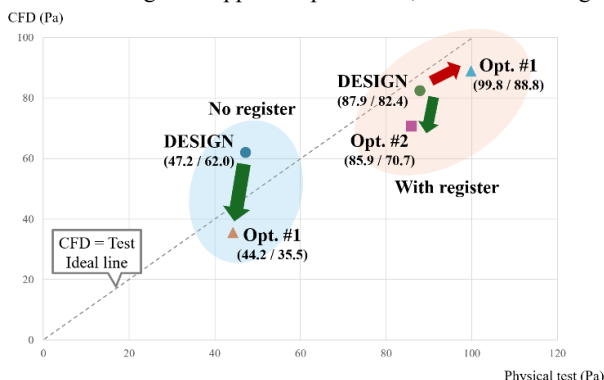


Fig.1 CFD and Experimental Results of Duct Pressure Loss

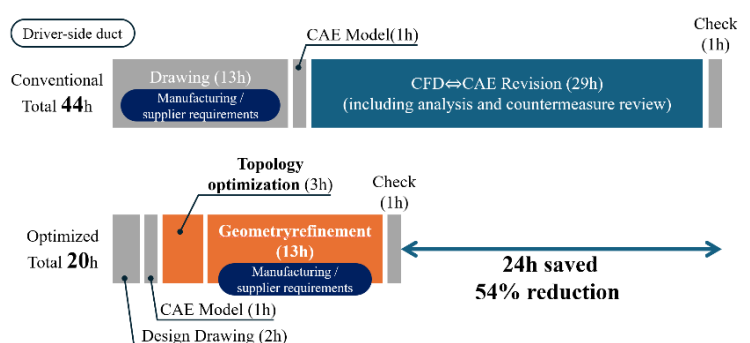


Fig.2 Man-Hour Comparison Between Existing and Proposed Approach

4. Conclusion

- (1) The proposed optimization technique reduced the drawing/CFD iteration and saved about 24 h per development phase.
- (2) The outlet boundary condition (whether downstream elements are modeled) fundamentally affects the optimized shape, even reversing whether the pressure loss improves or worsens.
- (3) Optimizing under realistic register conditions uniformizes the outlet velocity distribution and reduces the system-level loss, providing useful knowledge for HVAC duct design.