Designing High-Lift Systems for Low Drag

J. Wild*, U. Herrmann†
DLR – German Aerospace Center
Institute of Aerodynamics and Flow Technology
Lilienthalplatz 7, D-38108 Braunschweig, Germany

Abstract

The purpose of this lecture is to give examples for the application of numerical methods for low drag design of high-lift configurations. First the importance of drag reduction for high-lift configurations is outlined. Part one of this lecture deals with the automated design of high-lift wing sections for subsonic transport aircraft by means of numerical optimisation. The second part outlines and describes the low drag design for a high-lift system of a supersonic transport configuration as well as its validation.

1. Introduction

The design of high-lift devices is one of the challenging items in aerodynamic research. Although the augmentation of lift by using additional wing devices has been discovered in the 1920s by Handley-Page [1] and Lachmann [2] independently it has taken over 50 years until Smith [3] discovered the aerodynamic principles of the lift increasing behaviour. Another milestone in the understanding of high-lift flows occurred another 20 years later, documented as the proceedings of the AGARD conference on "High-Lift System Aerodynamics" in 1993. Especially the introductory lecture by Woodward and Lean [4] gave an overview on the results of a very large UK research program, where a wide range of high-lift configurations has been investigated experimentally. In the US also a large wind-tunnel campaign has been conducted in the 1990's based on the well-known McDonnell Douglas multi-element airfoils and was summarized by Valarezzo et al. [5]. Nowadays high-lift devices based on these principles are commonly used for transport aircrafts. A nearly complete overview of the types of high lift devices in use on subsonic transports has been collected by Rudolph [6].
While looking at the procedures how high-lift devices are currently designed within industry, it is found, that main aspects of the work are still carried out by wind tunnel tests. Only the preliminary design of the device shapes is assisted by lower order CFD tools like panel-boundary layer methods. The reason is, that in the past the development and application of higher order methods like RANS solvers were mainly targeted to transonic airfoil and wing design. First attempts for a validation and application of these solvers for high-lift configurations at low mach numbers just started about 10 years ago [7] and did not yet come to an end.

To give an example, in 2000-2003 there have been two European EC-funded research projects addressing this topic. First to mention is EUROLIFT, a project dealing with the validation of RANS solvers for high-lift configurations. A parallel effort is undertaken in the EU-project EPISTLE (2000-2003), where a CFD based design method for low drag high-lift systems is developed and validated. To date, only few attempts to use CFD methods for the design of high-lift configurations were made in industry up to now. Most efforts were located at research institutes and universities. A summary of this work excluding SCT aspects was collected by van Dam. But in these days CFD is taking the step to become usable for practical design purposes.

---

* jochen.wild@dlr.de
† ulrich.herrmann@dlr.de