



Understanding the flow behavior around Krueger flaps through the means of CFD simulations

Apurva Hasabnis¹ · Jochen Wild¹ 

Received: 13 April 2026 / Revised: 31 July 2026 / Accepted: 24 September 2026
© The Author(s) 2026

Abstract

This study aims to establish the feasibility of Krueger flaps acting as high lift devices on a natural laminar wing aircraft configuration. It is part of the German funded project ‘ULTIMATE’, aiming an aircraft wing design providing fuel savings up to 10% as compared to aircrafts today at the next short to medium range aircraft expected to enter the market in 2030. The current paper aims to outline the status of implementing Krueger flaps as leading edge high-lift system, arranged in distributed manner and the ability to achieve handling quality requirements in high-lift configuration, and taking into consideration failure cases, which allows compact kinematic and system designs fitting into the restricted space of the aerodynamic profile. The current work focuses on the evaluation of the achieved low-speed performance by Computational Fluid Dynamics as well as first insights into critical failure cases with Krueger flaps jammed in unfavorable positions.

Keywords Aerodynamics · Unsteady · High-lift systems

1 Introduction

Krueger flaps have experienced augmented attention and research in the recent years. They are increasingly proving to be the high-lift system of choice at the leading edge for wings incorporating laminar flow technology. Stored in a cavity at the lower side of the wing, they deploy from the lower surface of the wing and extend forward into the airstream, modifying the wing’s leading-edge flow to delay flow separation and increase lift coefficient. The primary advantage of Krueger flap is its property to not disturb laminar flow over the wing’s leading-edge while being comparable to conventional slats in terms of lift coefficient augmentation. It also helps to protect the wing leading edge from contamination and pollution by dusts, dirt or insects.

The ‘Krueger flaps’ – or ‘nose split flaps’ – were first invented by Werner Krueger [1] in year 1943. They have since evolved from a simple hinged panel mounted on the leading edge of the wing to a more advanced and detailed

high-lift device of choice for commercial aircrafts. Its early application can be seen on Boeing’s jet liners B707 to B747 [2].

Significant research has since improved the flaps into the modern version. The Krueger flaps used on transport aircraft applying laminar wing technology were further investigated during the B757 laminar wing flight test campaign [3]. At that time there was a significant performance gap to conventional slat devices. The performance of the Krueger flaps has since been improved significantly to match that of the slats, while providing added advantage of delayed flow separation and shielding properties. In current context, the previous EU funded project DeSiReH [4] laid the foundation of new design methodology of Krueger flaps using numerical optimization. The project dealt with a design parameterization guaranteeing both kinematical feasibility and contamination shielding capabilities. In a wind tunnel test it was proven that the developed slotted Krueger devices are competitive with conventional systems, avoiding a need in wing area increase for low-speed considerations. The follow-on EU project AFLoNext [5] matured the concept toward full-scale wing integration on a hybrid laminar flow control (HLFC) wing demonstrating the kinematics feasibility even in a more constraint space. Within the previous EU project UHURA [6] the dynamic characteristic of such Krueger devices was analysed in detail. The

✉ Apurva Hasabnis
apurva.hasabnis@dlr.de

Jochen Wild
jochen.wild@dlr.de

¹ DLR – German Aerospace Center, Brunswick, Germany

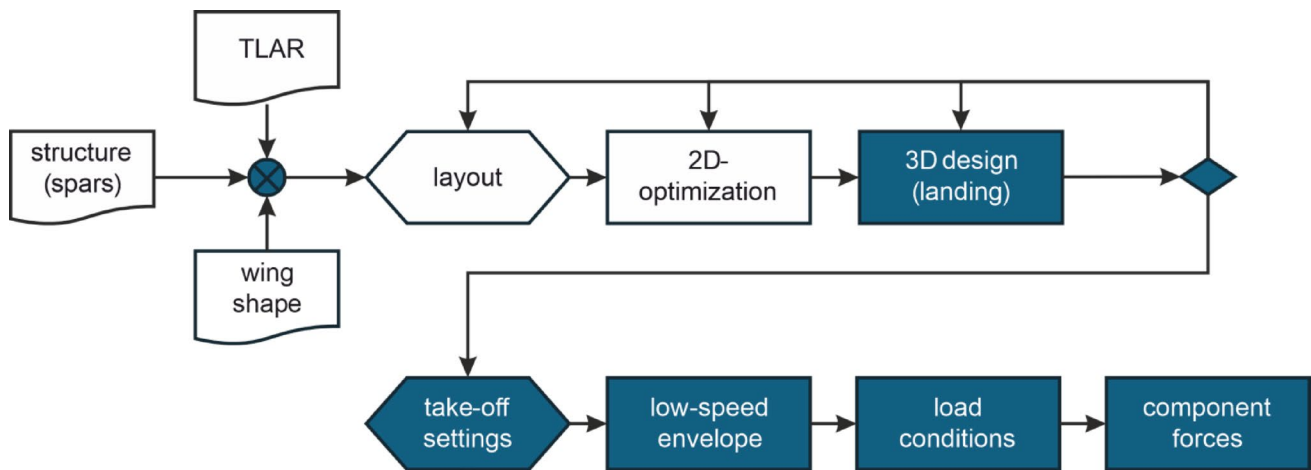


Fig. 1 Design and evaluation process of high-lift system design

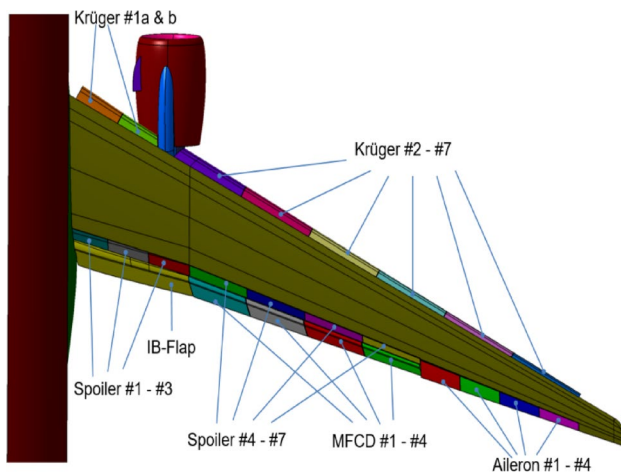


Fig. 2 High-Lift system layout of the wing

results of these projects lay the foundation for the current project ULTIMATE [7].

This paper in particular, focuses on the numerical results of steady Computational Fluid Dynamics (CFD) simulations of such Krueger panels as high-lift devices. The panels are mounted on the leading edge of transport aircraft in distributed configuration and assumed to deploy sequentially. A test matrix of possible configurations was laid out. The Krueger panels were intermittently deployed and the flow behavior was captured through CFD simulations. One important task was to understand the impact of flow-critical positions, aka the so-called “barn-door” positions of various panels at set deployment angles on the aircraft lift. The barn-door position is defined as the instance when a panel stands directly perpendicular to the flow and offers the maximum aerodynamic drag and flow disturbance. There is a quite significant risk that in such position the flow not only around the Krueger but all over the impacted wing segment is negatively affected. It is essential to understand also the

interference with the neighboring Krueger elements set at individual deployment angles during the deflection motion. The results of these simulations are further to be used to carry out failure analysis for possible failure cases.

2 Design process

The process followed for design and aerodynamic analysis of the high-lift system followed in the current project is depicted in Fig. 1. Within this contribution, mainly the steps in the colored boxes are outlined in more detailed. The process starts with a given clean wing, some structural information, mainly regarding the wing-box dimensions for structural integrity and fuel volume, and the top-level aircraft requirements (TLAR), where for the low speed mainly two specific goals are set: the desired runway length and the envisaged ICAO approach category of the aircraft.

The basic wing for the high-lift design is a natural laminar wing design for significantly high cruise Mach numbers [7]. The wing shows only slightly-reduced leading edge sweep compared to comparable today’s transport aircraft. As mentioned before, the control surface layout has been transposed from a highly adaptive wing concept investigated in a previous national project [8]. Figure 2 shows the layout of the final configuration. A total of seven Krueger panels can be seen on the leading edge. Panel 1a and 1b on the inboard side, while panels 2 through 7 on the outboard side. The trailing edge comprises of an inboard flap, four multi-functional control devices (MFCD) along with seven spoilers and 4 Aileron surfaces.

The chosen Krueger actuation concept is shown in Fig. 3. As kinematics type, the goose-neck type kinematics as already investigated in the previous projects has been implemented on the new wing. The figure depicts from left

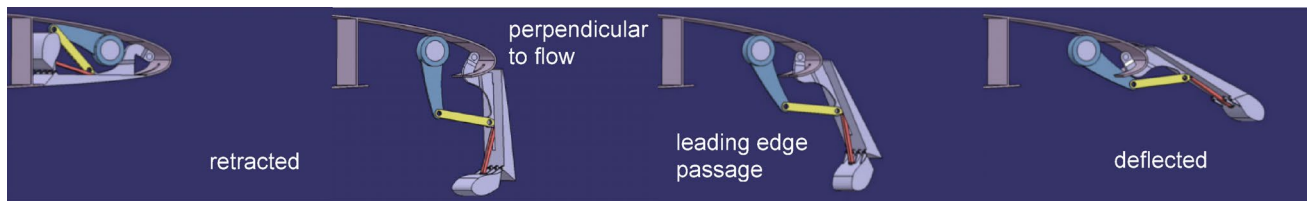


Fig. 3 Krueger flap kinematic concept and specific positions during deployment (from [6])

to right the path from retracted to fully deployed, indicating the two most critical conditions for further analysis.

The entire underlying actuation system is developed by project partners Liebherr Aerospace and the Technical University of Hamburg-Harburg. Establishing the kinematic relationships and a clear understanding of the motion is essential to ensure the panels follow the intended trajectories and for modelling the system's positions along its motion path. The respective positions of all the control surfaces have been accurately simulated for various scenarios for capturing the flow behavior accurately. The resulting loads are considered for sizing the kinematics and actuation, keeping in mind the failure scenarios of single or multiple panel failures and critical positions encountered by the panels throughout deployment. Physical stops are introduced to limit maximum panel deployment.

3 CFD simulation methodology

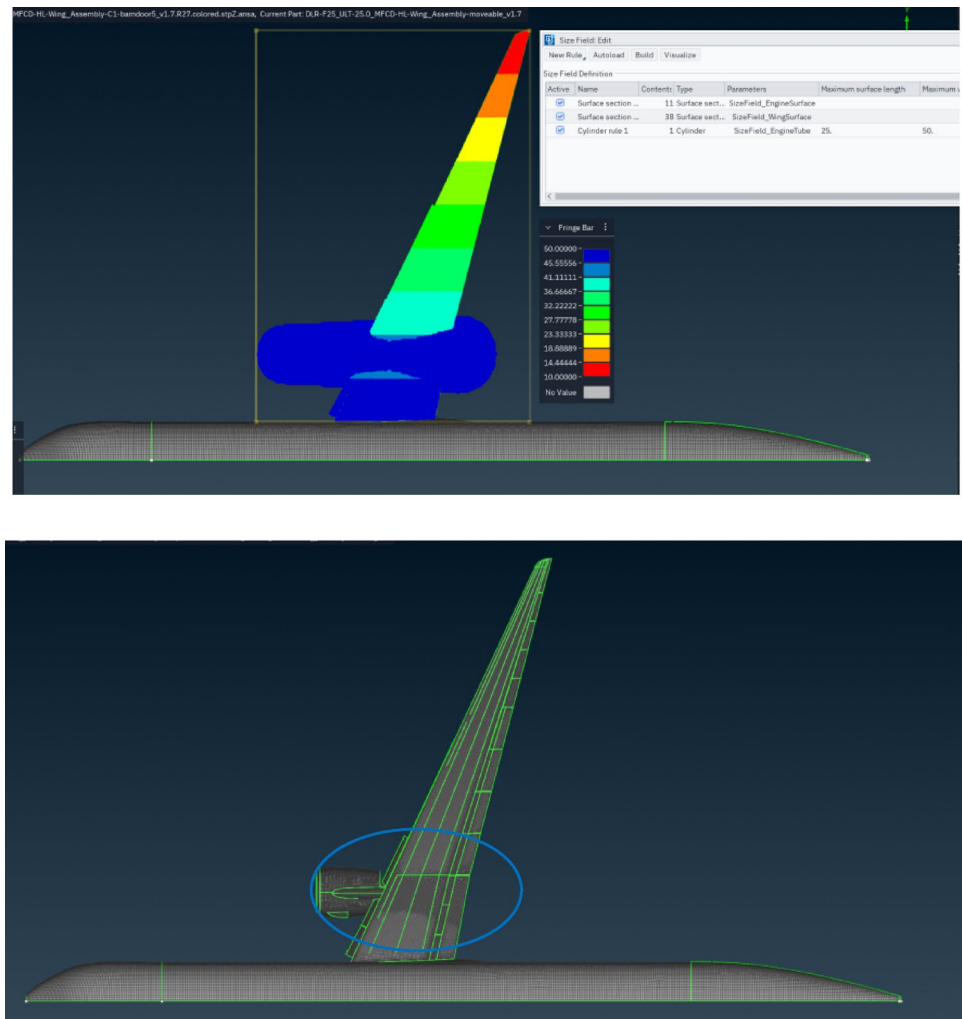
First, the space discretization by the computational mesh establishes the foundation for the future CFD analysis. The accuracy and reliability of a solution is a direct result of a good mesh. The super-fine mesh does not directly imply a good mesh. As it is well established, CFD analysis is essentially resolving the partial differential equations representing the corresponding flow field numerically. This requires discretizing the flow field into element volumes. The array of such discrete points and volumes represents the grid. The way these points are identified and arranged ensures the efficiency of the solution. As explained by J.D. Anderson [9], a grid essentially transforms a physical space into a computational domain. Depending upon the problem at hand, the most suitable method can be adapted for the distribution. The discretization should conform to the boundary of the region in such a way that it can represent the boundary conditions or it is possible to assign the boundary conditions accurately. A structured mesh divides the volume in quadrilateral cells, whereas an unstructured mesh divides it into tetrahedrons. An unstructured mesh offers more flexibility when it comes to complex geometries. On the other hand, a structured mesh offers more accuracy due to uniformity. To derive from the best of both approaches, a hybrid mesh is adapted for the geometry at hand.

After establishing a detailed CAD model in CATIA® representing the layout, kinematics and the system design, the next step was to import this geometry to a meshing tool to make it ready for CFD analysis. The meshing tool selected for current purpose is the software ANSA by BETA CAE Systems as it provides an integrated environment to clean-up the geometry, define topology and create high-quality surface and volume meshes in order to ensure accuracy of the CFD simulations. Its robust topology handling and automated meshing algorithms make it particularly suitable for complex aerospace geometries such as the one at hand.

ANSA applies geometric feature characteristics like leading and trailing edges or sharp corners together with a 'size-field' to refine meshes in crucial locations for accurately simulating the flow. These size-fields act as controls where finer elements are generated, enabling better resolution of flow featured such as boundary layers, wakes etc. without unnecessarily increasing the overall mesh size. Figure 4 shows the major characteristics of the applied refinement at the top and the resulting surface mesh at the bottom. The targeted refinement improves numerical accuracy and solution stability while maintaining computational efficiency. The color-coded distribution indicates the degree of local refinement, with larger element sizes shown in cooler colors and highly refined regions are shown in warmer colors. In the current mesh a size rule refines the maximum cell size consistently with the local wing chord to keep aspect ratio of cells together with the number of cells in circumferential direction similar along the span. A cylindrical size-rule is applied around the engine and its wake to refine the impact of the presence of the engine. The final meshes consist of about 70 to 80 million elements, slightly depending on the configuration. Such targeted mesh refinement ensures increased solution accuracy, reduced simulation times, in turn keeping the computation costs in check.

Once the desired mesh was established, it was exported to the DLR TAU code for simulations [10]. DLR's in-house CFD simulator, the TAU code, is a finite-volume flow-solver for hybrid unstructured meshes, specifically designed for complex aircraft geometries. It is dedicated to maintain high standards and accuracy of the solver to meet aerospace industry standards. The TAU code solves the compressible Reynolds-averaged Navier–Stokes (RANS) equations. From a wide selection of RANS turbulence models, for the

Fig. 4 Mesh refinement using ANSA size-fields



current study the Spalart–Allmaras model in its negative formulation [11] has been chosen for its improved robustness in the design context. TAU uses a vertex-centered volume discretization with a second order central scheme. A semi-implicit LU-SGS time-stepping enables efficient convergence to steady state. More details about the underlying concepts can be found in the paper [10].

During the course of the project, wind tunnel experiments have been conducted with the clean wing configuration (retracted high-lift system) in the European Transonic Wind-tunnel facility (ETW) to validate the laminar wing properties. It was possible to add measurements even at subsonic velocities as a basis to validate the computational setup used for the low-speed regime and the high-lift properties. Figure 5 shows the lift-coefficient plotted against the AoA for the clean wing configuration, in subsonic as well as transonic flow regime. The experimental values are plotted along with the CFD results. The experiments were conducted in two versions, one with free transition and one with a transition fixated by tripping close to the leading edge. The bandwidth of experiments represents the uncertainty by

having laminar or fully turbulent flow. As it can be observed, in the subsonic region, the CFD values agree well with the experimental values up to the onset of flow separation indicated by the deviation from the linear relationship, which is predicted more severely by the CFD analysis. In the linear range, the deviation of experimental and simulation data is evaluated to be less than 5% at angles of attack between 1° and 7° for the subsonic cases that are here of most interest. In the transonic region however, the stall evolution fits well between experimental and CFD values. These are represented by the peak of the curve for each data set. The lift curve leading up to the onset of flow separation is steeper for transonic speeds, representing the compressibility of air. These graphs establish a good confidence into the validity of the numerical approach. This approach can thus be replicated to undertake design and evaluation for the high-lift configurations as needed.

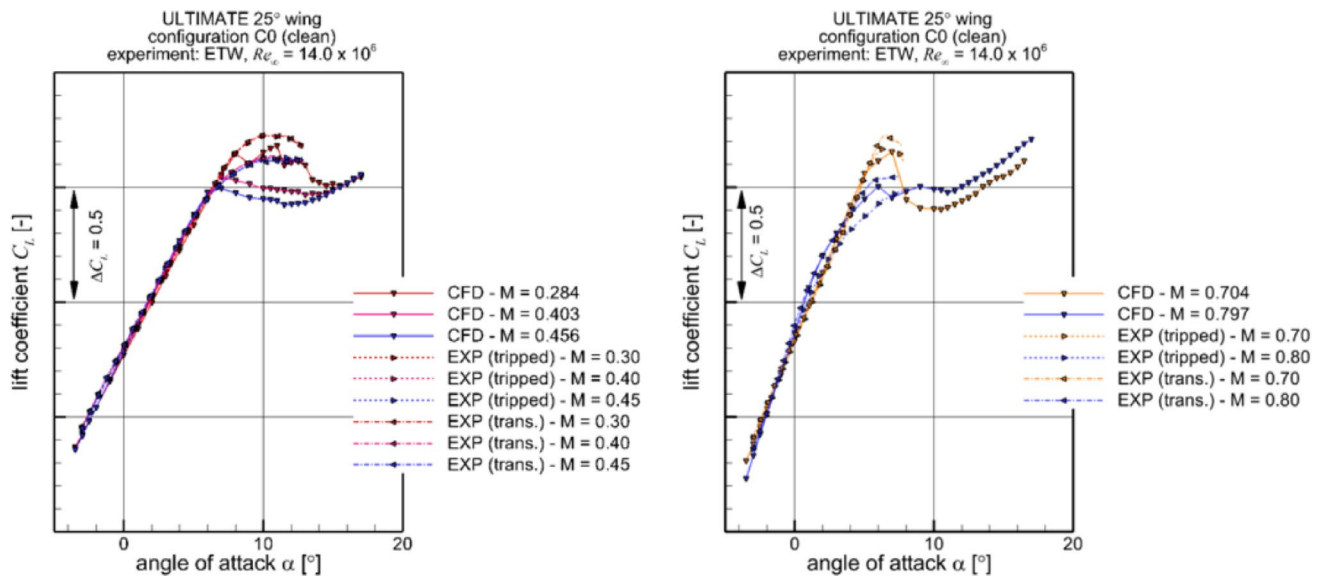


Fig. 5 Comparison of aerodynamic performance of clean wing configuration between experiment (free transition and transition tripped) and CFD: (left) subsonic flow; (right) transonic flow

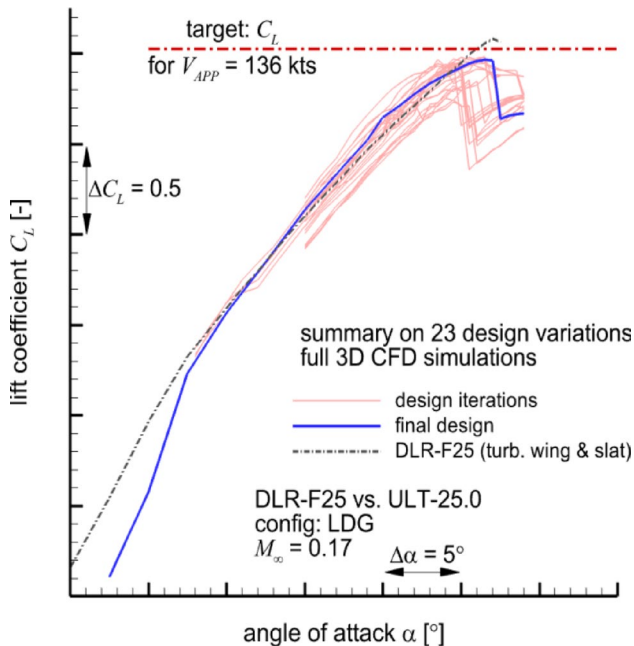


Fig. 6 Lift coefficient vs. angle of attack for the design iterations on the ULTIMATE laminar wing with Krueger flaps, compared to reference aircraft DLR-F25 with turbulent wing with slats

3.1 High-lift design

The first step in establishing the high-lift system was to aerodynamically design the shape and positions of the different high-lift flaps. As the major target is to achieve a sufficiently low approach speed, the design was mainly driven by the landing configuration, characterized by the highest flap deflection. The design started by transferring

the Krueger flap concept and a classical trailing edge single-slotted dropped-hinge flap with spoiler droop concept to the new natural laminar flow (NLF) wing.

An initial design of the geometric characteristic parameters of the Krueger flap were achieved by 2D numerical optimization with the same approach as referred to in [4]. This parametric set has been propagated to the full 3D wing. For the final 3D design, in total 23 design iterations have been simulated by full 3D CFD simulations to compensate local 3D effect, mainly at the trailing edge. The designs were mainly compared with regards to the flow characteristics, namely for achieving sufficient maximum lift coefficient but also achieving attached flow at the flap at lower angles of attack. A summary showing the lift coefficient versus the angle of attack (AoA) is shown in Fig. 6. It represents the final design (blue solid line) in comparison to the DLR-F25 design (black dotted line) with turbulent wing with slat. It was observed that the achievable maximum lift stays little below the requirement for a target approach speed of 136 kts and only gets close to the reference DLR-F25 capabilities [12]. However, after reassessing the designs, it was concluded that this requirement could only be accurately achieved with either a planform change with increased flap size, or by a double-slotted flap. This would entail added design and kinematics effort. At the end, the resulting approach speed of about 2 kts above the target was judged to be acceptable for the existing design, as it still allows to keep the aircraft in the same approach category.

Table 1 Overview of the configurations of the various flap positions for the definition of the ULTIMATE high-lift system

Configuration	Krueger	Flap
	[°]	[°]
C0	0	0
C1	135	0
C1+F	135	12
C2	135	20
C3 (take-off)	135	28
C3 (approach)	135	28
C4	135	37.5

3.2 High-lift performance evaluation

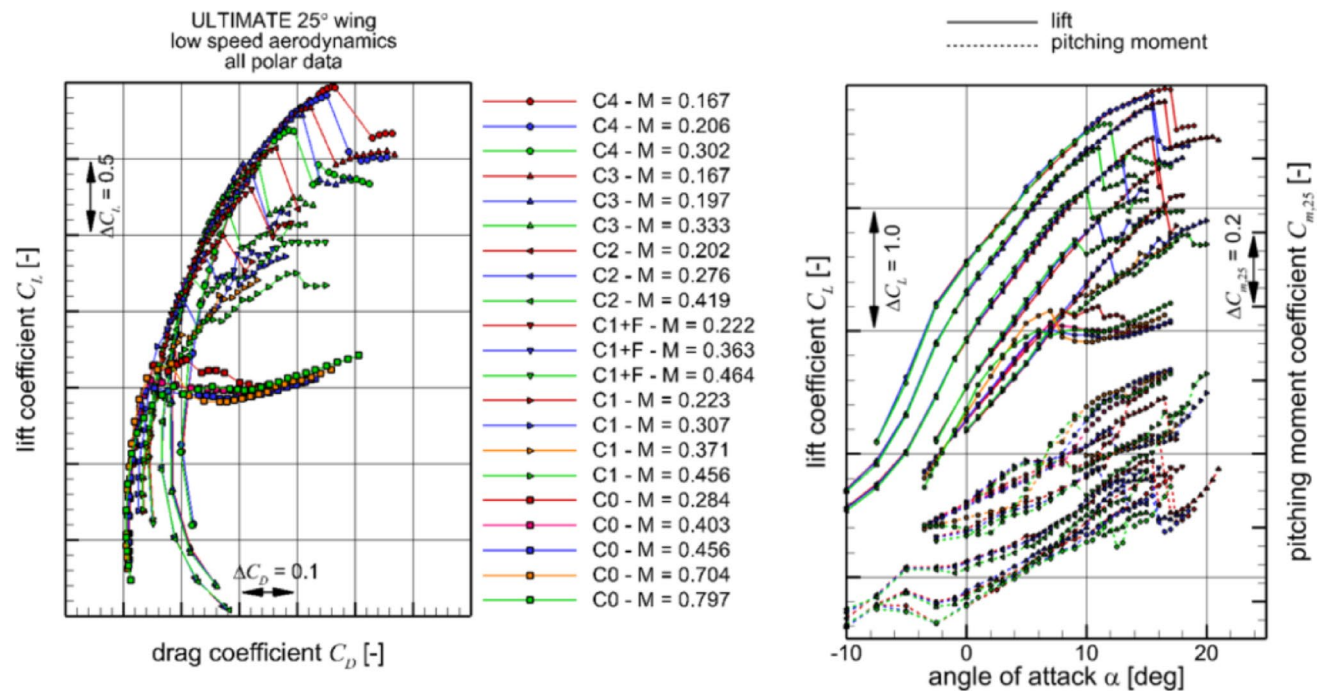
Once the design was finalized, all intermediate settings of the flap system were derived. Table 1 lists the selected deflection angles for the intermediate positions. The nomenclature of device settings follows the terminology of C0 as the clean wing and moving on to configurations C1 (only Krueger flap deflected), C1+F (low take-off) through C2 (normal take-off), C3 (high take-off & approach) up to C4 (landing), with increasing deflection of contributing high-lift devices including aileron, spoiler and flaps. They all were meshed and subjected to CFD simulations (Fig. 7).

All configurations are simulated for at least three Mach numbers to cover their operational range. Figure 1 (left) shows the Lilienthal polar graph of all simulated configurations from C0 up to C4. Figure 5 (right) shows the lift coefficient and the pitching moment coefficient over the angle of attack. A keen observation of the map on the left shows a

broad and smooth distribution of maximum lift coefficients along the path for subsonic speeds as well as transonic speeds for the clean configuration. Significant increase in C_L for the take-off, approach and landing configurations, compared to the clean-wing configuration are testament to the functionality of the high-lift system. At low incidences, a noticeable drag rise is observed from C3 to C4. This in-turn is intended to facilitate steeper descent paths.

Coming to Fig. 5 (right), the pitching moment diagram, the graph corresponds to the stability of the aircraft. It represents the ability of the aircraft to attain equilibrium through restoring pitching moment in case of disturbances causing variation in the angle of attack. Changes in aircraft configuration, center of gravity location or the control surface deflection will shift or rotate this curve. The C_L vs angle of attack graph is also incorporated in the same for reference. At the onset of stall, the drop in pitching moment correlates to the consequent drop in the C_L value. For negative angle of attack, the nonlinearity of the curve can be observed, which entails instability. Above zero angle of attack however, the curve is linear for all Krueger configuration. Which confirms the stability of the aircraft for all variations of control surfaces. Beyond the angle of attack of about 10 deg, the tendency of the aircraft to move away from the trimmed position is evidenced by the sudden drop in the pitching moment curve.

From the 3D CFD analysis of the ULTIMATE configuration maneuvering envelopes and speed ranges are derived for all high-lift settings and the clean configuration, and for

**Fig. 7** Summary of aerodynamic characteristics of all configurations simulated by CFD

a mass range of operational empty weight (OEW), maximum landing weight (MLW), and maximum take-off weight (MTOW). They essentially represent a relationship between the load factor (n) and airspeed (V) for the ULTIMATE aircraft, depending on the aerodynamic performance analyzed before. It defines the structural and aerodynamic limits within which the aircraft can be safely operated during maneuvers. Figure 8 shows the maneuvering envelope for MLW condition. The lower envelop speed boundary is governed by stall speed V_S limits, where the configuration can no longer generate enough lift at a given load factor, while the upper speed limit is set by the speed flexibility required to define the flap design speed V_F . The upper bound of the load factor results from the required maximum allowable structural load factor. The envelope covers the design range for load assessment as specified by certification regulation CS-25 §25.333, §25.337, and §25.345. This data is fundamental in determining potential critical load cases with the high-lift system extended, along with operational load cases where the transition between flap positions occur.

3.3 High-lift component aerodynamic load evaluation

Once the envelope of aircraft operation is defined, the simulated data can be further used to derive the aerodynamic forces acting on the different high-lift system components for structure and systems sizing. Given the aerodynamic

behavior and the operational points, the existing data is screened for determining the aerodynamic loads limiting the structural strengths requirements.

Figure 9 depicts exemplarily, for the landing configuration C4, the maneuvering envelopes as a function of aircraft mass with the critical load cases under consideration plotted as points. As the aircraft weight shifts from OEW to MTOW, the stall speed shifts to higher values, effectively narrowing the usable maneuvering region. Thus, this diagram highlights the safe flight domain bounded by stall, limit load factors and maximum design speed. As it can be seen, all but two of the values plotted for the configuration C4 lay within the defined limits of flight. The two others consider overspeed conditions according to CS-25 §25.697(d) where it is defined that in this case the system must be designed to be able to retract. This is a limiting design case for the actuation system.

Figure 10 outlines the coordinate system adopted to represent the aerodynamic surface forces at component level derived from the CFD calculations. The figure on the left corresponds to the coordinate system on the Krueger panel, while the one on the right for the trailing edge flaps.

The graphical representation of the individual component forces depending upon the described coordinate systems are covered in this section. It provides a summary of max. positive and negative aerodynamic forces depending on the high-lift setting, explicitly shown for Krueger flap no. 2 in Fig. 11, panel no. 7 in Fig. 12 and trailing edge

Fig. 8 Maneuvering envelopes of the ULTIMATE configuration for all flap settings at max. landing weight

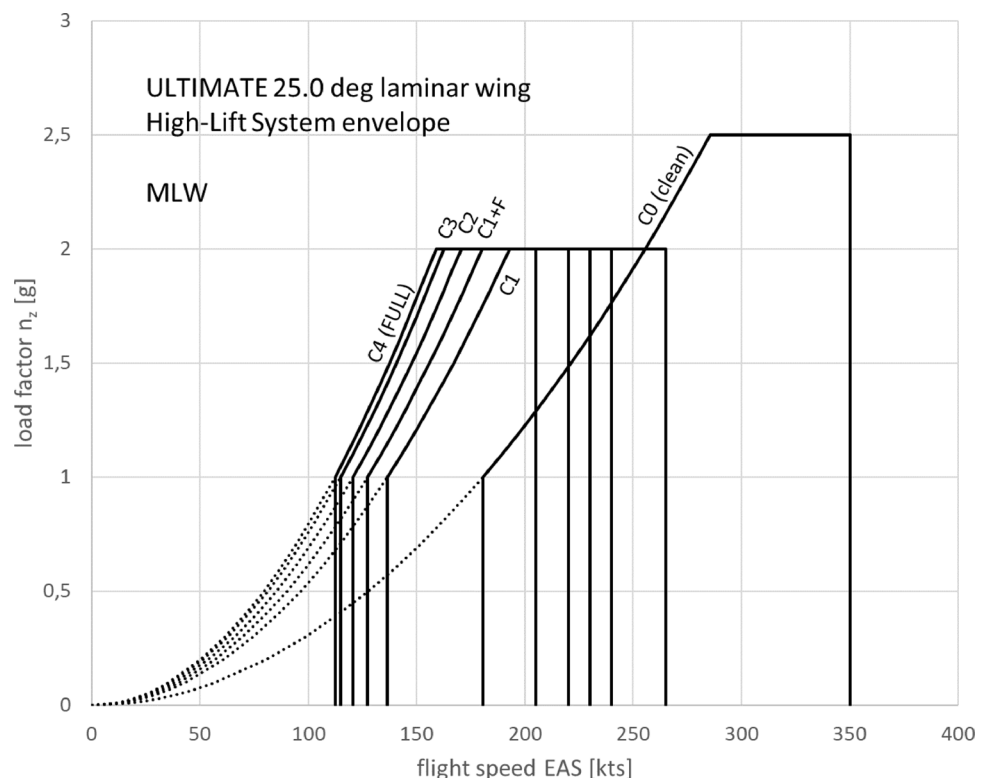


Fig. 9 Load conditions selected for load assessment, exemplarily shown for C4 high-lift setting

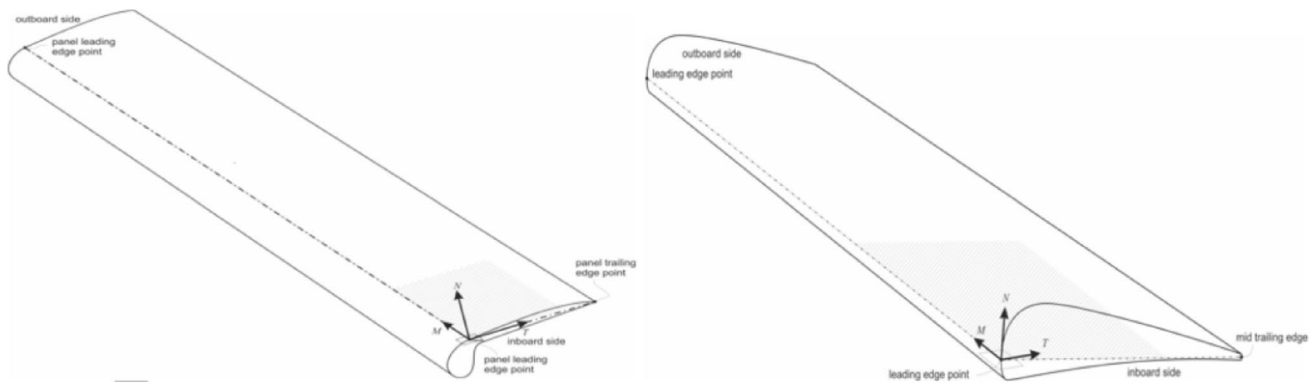
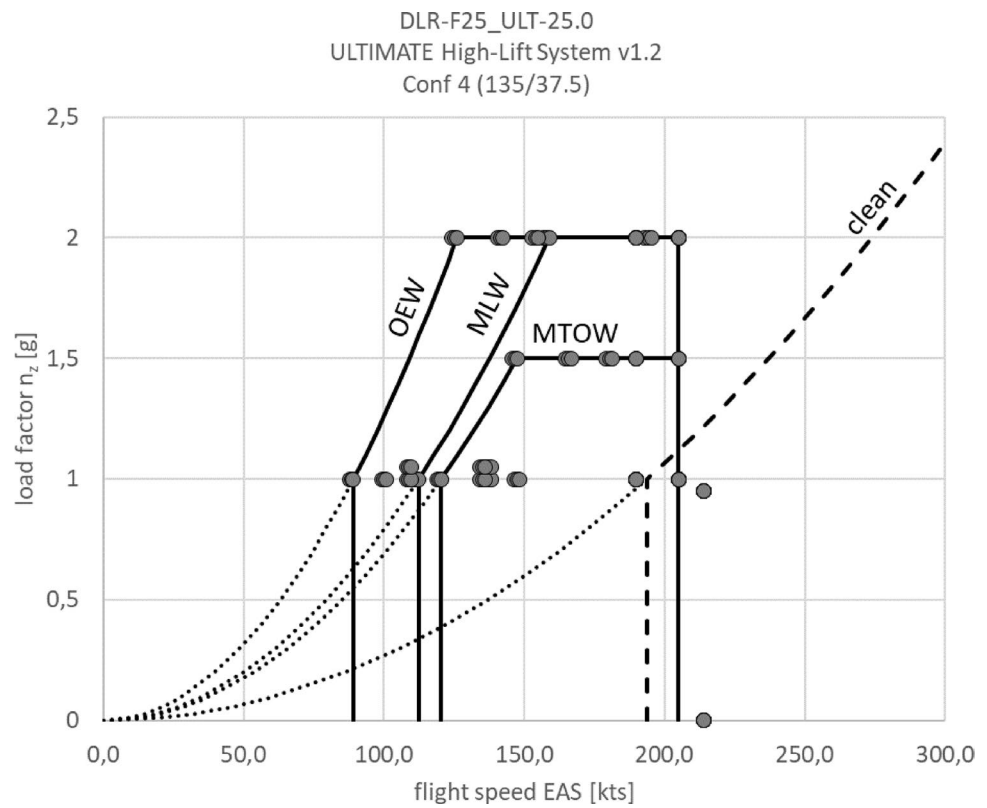


Fig. 10 Definition of local component body coordinate system at a Krueger panel (left) and trailing edge flap

flap no. 3 in Fig. 13. As mentioned, forces are given in a local coordinate system fixed to the component-body. N is the force acting normal to the panel with positive orientation to the outward and upper direction in retracted and deflected position, respectively. T is the force acting tangential to the panel surface. c and η are the relative chordwise and spanwise locations of the center of pressure.

The outer Krueger flap no.7 appears to have a maximum load that depends less on the flap position as compared to the inner Krueger flap no.2. Furthermore, for panel no.7, the maximum load in the positive direction when retracted is caused by suction forces acting on the leading edge of the lower wing surface at high speeds, which act to pull the Krueger panel out of the wing. It represents the maximum

force for keeping the Krueger in stowed position during cruise flight. These results necessitate additional attention to the kinematics and structural integration with regard to stowing and pre-tensioning of the Krueger panel. The next force level is related to low speeds, namely the condition of stall at high load factors. These are the cases when the Krueger experiences the highest lifting forces. Maximum negative forces are identified for the deflected Krueger flap for low flap settings (C1 and C1+F). They are observed at high flight speeds when the flow pushes onto the Krueger flap from above. These forces in turn dictate the power demanded from the actuation system, and shape the design and development of the kinematics to serve the requirement. On the other hand, Krueger panel no.2 shows significantly

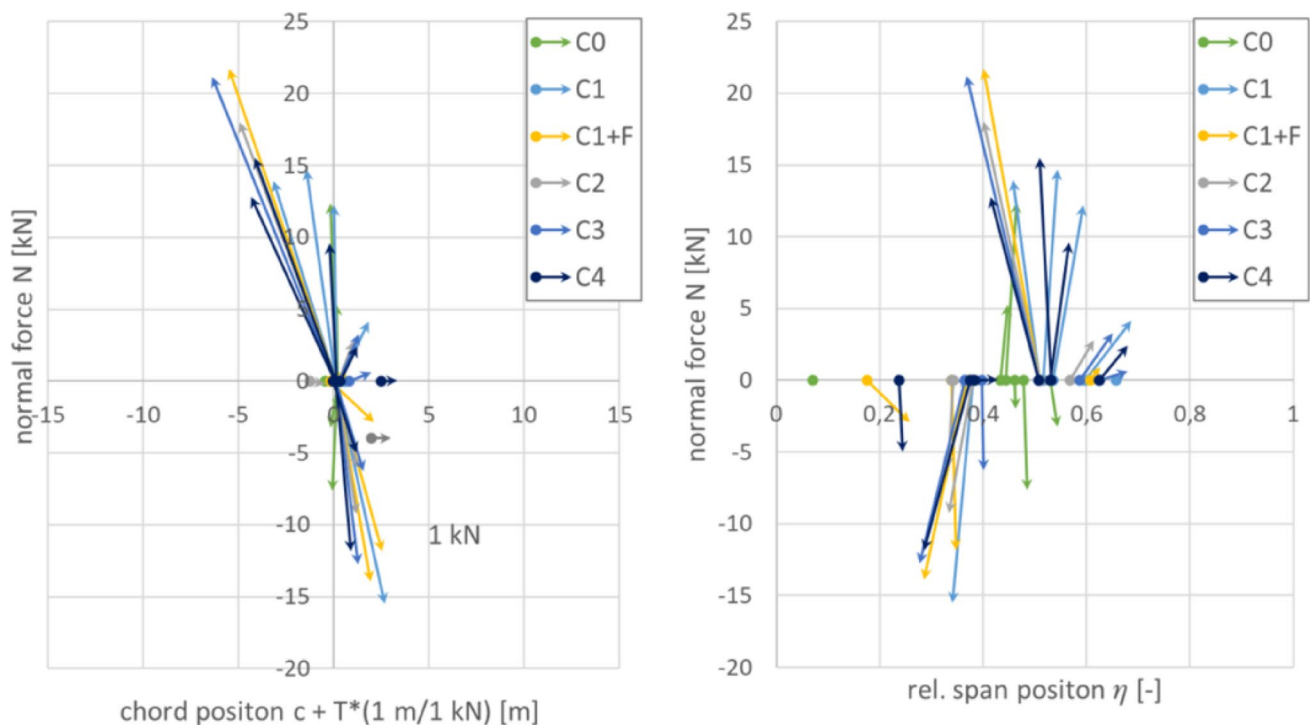
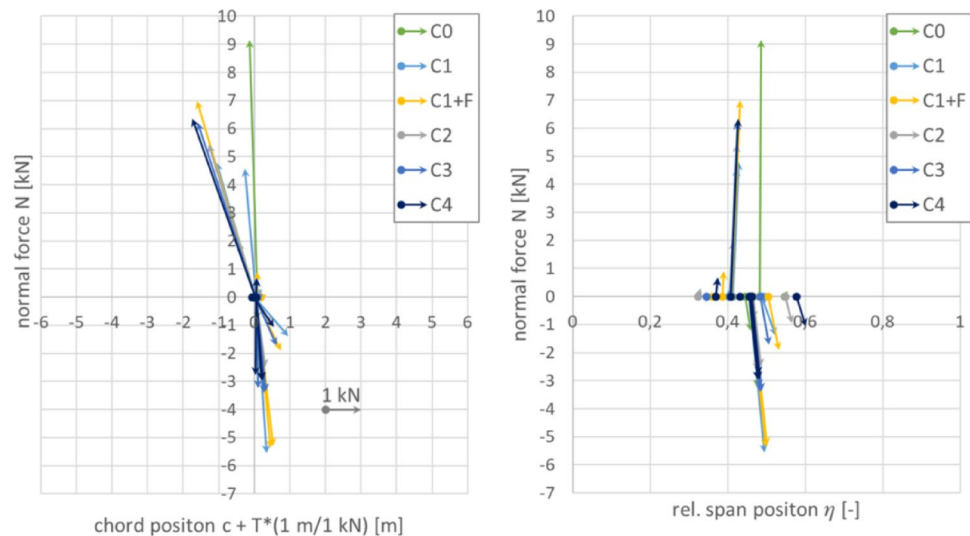


Fig. 11 Summary of aerodynamic forces on Krueger flap #2 (outboard of engine) for different high-lift setting configurations; depicted force vectors capture envelope in terms of min and max values of all force components; forces are given in component-body coordinate system

Fig. 12 Summary of aerodynamic forces on Krueger flap #7 (most outboard close to wing tip) for different high-lift setting configurations; depicted force vectors capture envelope in terms of min and max values of all force components; forces are given in component-body coordinate system



greater variation in maximum aerodynamic forces across different positions, which could be attributed to the vicinity of the nearby engine nacelle.

Figure 13 showcases forces acting on one of the multifunctional surfaces on the trailing edge of the wing, MFCD no.3 as a representation, during various flight stages. Referring to Table 1 for the deflection angle of flap for the corresponding setting. The highest positive normal force in Fig. 13 is observed for the phase C3 (high take-off/

approach), as is to be expected. No significant variation can be seen in spanwise direction.

4 “Barn-door” position evaluations

The Krueger movement, as also mentioned before, is peculiar. When the Krueger is actuated, it moves around the leading edge, against the flow and extends forward into the oncoming flow. During the motion, it passes a position

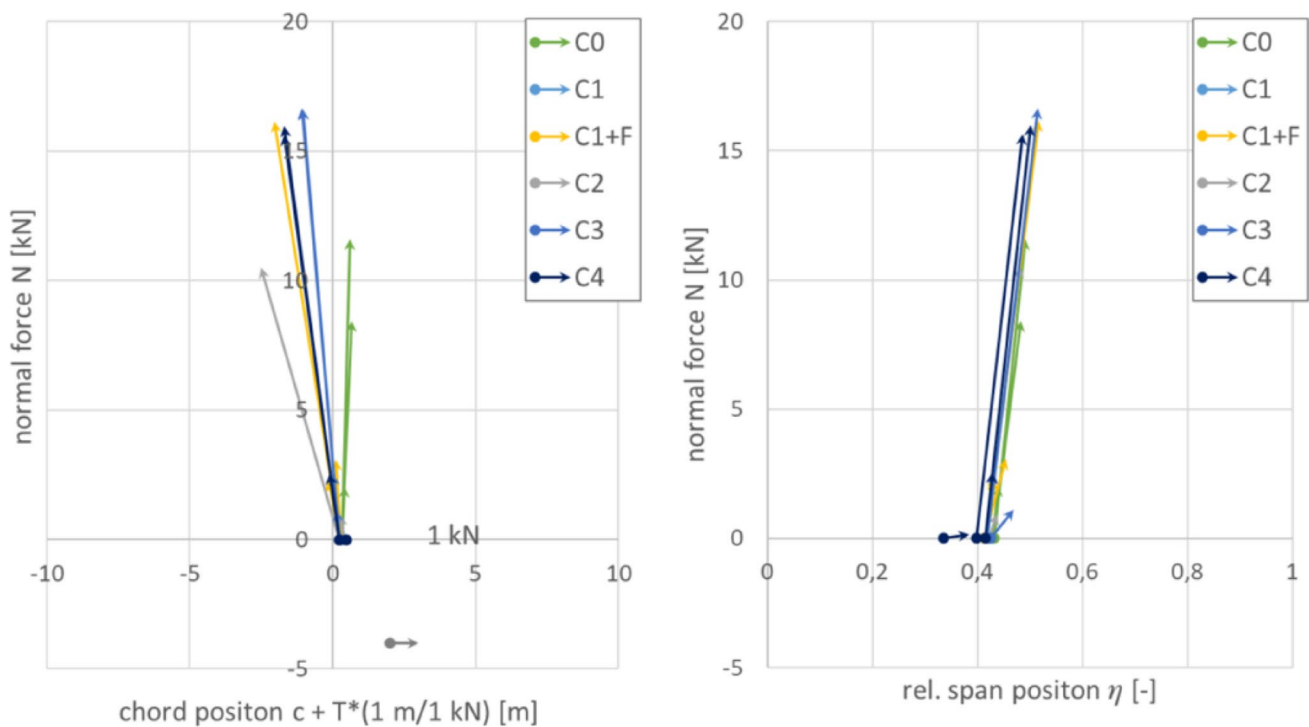


Fig. 13 Summary of aerodynamic forces on multi-functional flap (MFCD) #3 for different high-lift setting configurations; depicted force vectors capture envelope in terms of min and max values of all force components; forces are given in component-body coordinate system

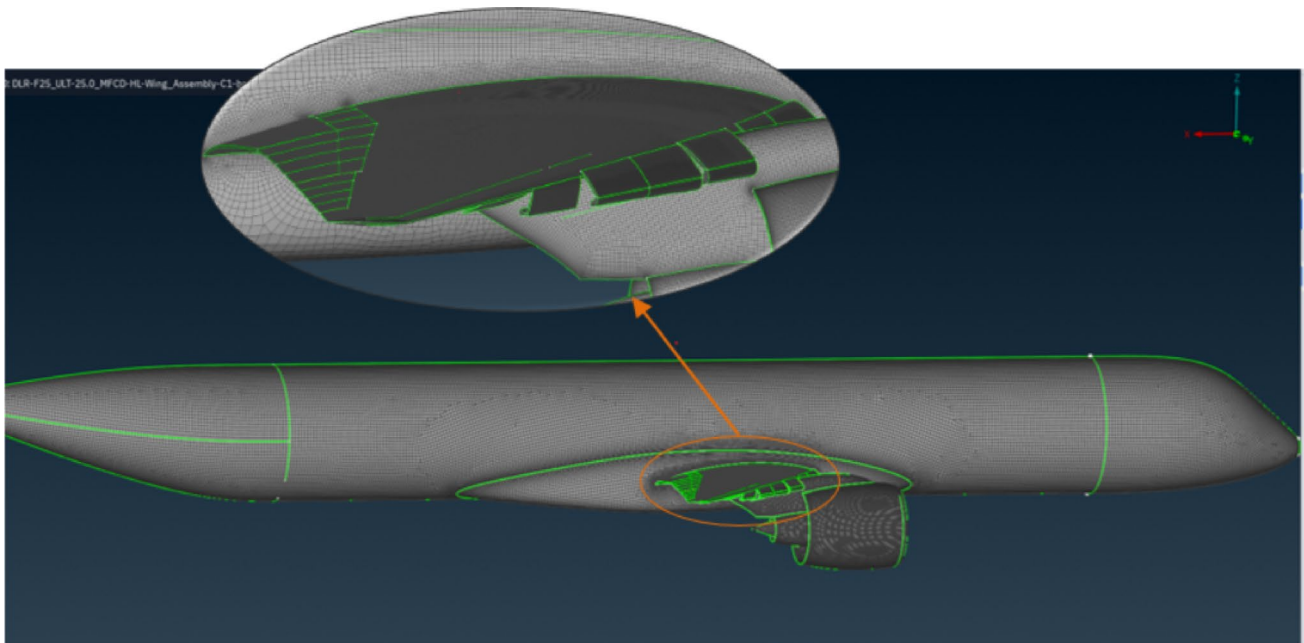


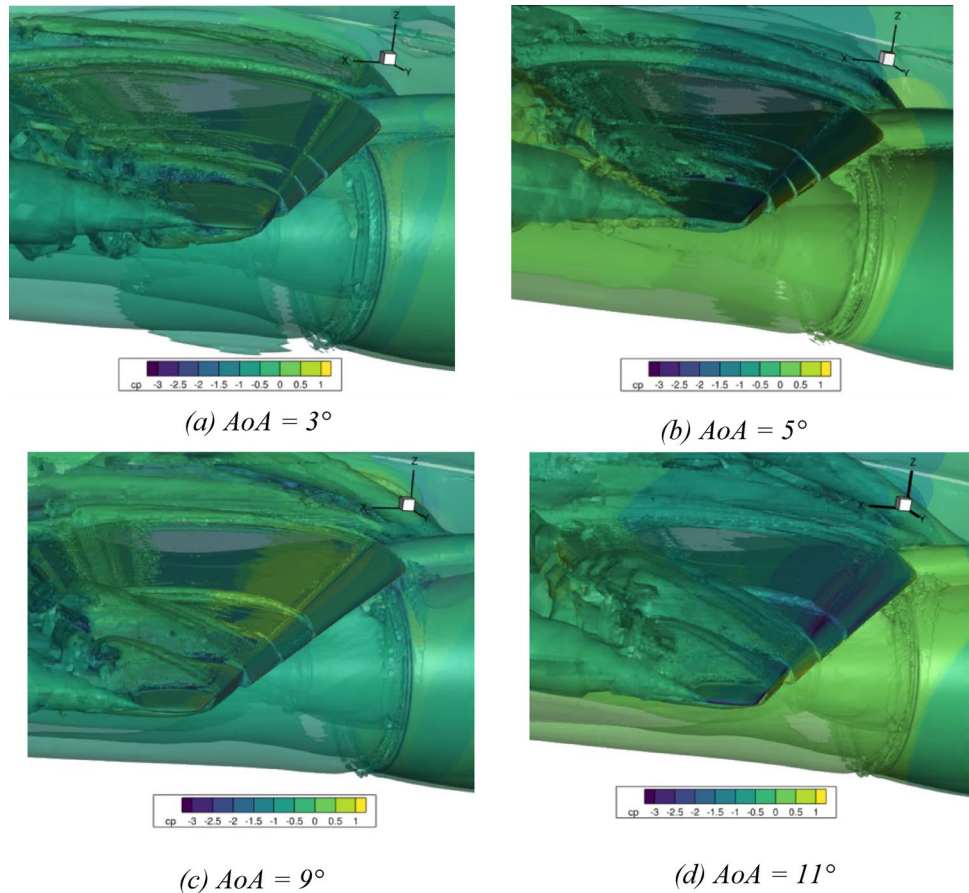
Fig. 14 Mesh around the Krueger panel in deployed position

where it stands perpendicular to the flow, the so-called “barn-door” position. To avoid all panels getting jammed in such an unfavorable position, Krueger panels are assumed to be operated in sequence. This situation is depicted in Fig. 14. It shows Krueger no.5 in barn-door position, i.e. perpendicular to the flow, The Krueger no.3 & no.4 have not

yet fully reached their final position and Krueger no. 6 has just started the travel around the leading edge.

As it can be seen, in the barn-door position the panel stands against the oncoming flow, and thus generates significant disturbance to the flow. To understand the impact of this position on the flow as well on the overall aircraft

Fig. 15 Flow around the wing with Krueger panel no. 7 in “barn-door” position



performance, it is vital to simulate and study the flow around the panels with precision. To achieve this, the mesh around the panels is further refined to resolve the flow physics with higher accuracy and achieve high-fidelity results. Refining the mesh locally, or limiting the refinement to critical areas avoids unnecessarily increasing the mesh size and thereby increasing the computing time, effort and costs. These refined meshes are then subjected to CFD simulations to resolve the flow.

The CFD results obtained from the simulations of the barn-door position for Krueger no.7 yields in the flow field shown in. The surface is colored by the pressure coefficient and the semi-transparent iso-surface of the Q-criterion is colored in the x-velocity component. The flow is being studied at an angle of attack (AoA) in the range of 2° to 15° . In the adjoining image, the Krueger panel no.7 is considered in the barn-door position. It can be seen standing against the flow, while the remaining panels have either completely or partially deployed. Four angles of attack are represented, 3.0° , 5.0° , 9.0° and 11° .

Figure 15 depicts the flow around the Krueger panels and over the wing during one such deflection scenario for intermittent angles of attack varying from 3° to 11° . As it can be seen, the flow field shows a stagnation line on the

upper side of the bull-nose, and a region of accelerated flow develops downstream, depicted by the pressure coefficient distribution. A flow separation is visible on the upper side of the wing behind Krueger flap no. 6 at the two lower angles of attack.

As the angle of attack increases to 9° , the onset of flow separation spreads outward to the region behind Krueger flap no. 7. In addition, a strong side vortex develops at the edge of the more deployed Krueger flaps no. 4 & no. 5. The turbulence in the Krueger cavities is also visible. As the angle of attack approaches 11° , the flow separating over panels 6 & 7 is increasing in volume. Further studies of flow around the other Krueger panels in barn-door positions are still to be established to complete studies on critical flow conditions during flight.

5 Summary

Krueger panels have been designed and analyzed for application on a highly swept natural laminar wing transport aircraft. The recent design shows an aerodynamic performance nearly comparable to established turbulent wing characteristics. The high-lift system has been simulated for all

relevant high-lift settings. A comparison made on the clean configuration with experimental data showed a good agreement, especially in the linear range and for the onset of separation, while stall development was predicted too severe. From the data set, aerodynamic forces for load calculations were derived to gain insight on the flow conditions where high loads in different orientations act on the Krueger flap itself. Finally, initial CFD simulation have been obtained for critical “barn-door” positions. The CFD results confirm that even in “barn-door” Krueger position, the leading-edge flow topology over the rest of the wing shows stable flow.

Acknowledgements The national project ULTIMATE is funded by German Federal Ministry for Economic Affairs and Energy within the German Aeronautical Research Program LuFo VI-2/2022- 2025 under grant agreement no 20A2101A. The authors gratefully acknowledge the scientific support and HPC resources provided by the German Aerospace Center (DLR). The HPC system CARA is partially funded by “Saxon State Ministry for Economic Affairs, Labour and Transport” and “Federal Ministry for Economic Affairs and Climate Action”.

Author's contribution The manuscript is a result of combined contribution of Apurva Hasabnis and Jochen Wild of the German Aerospace Center. All authors have reviewed and accepted the manuscript.

Data availability No datasets were generated or analysed during the current study.

Declarations

Conflict of interest Jochen Wild, co-author of this paper, is also an editor of this journal, but was not involved in the review process. All other authors have no competing interests to declare that are relevant to the content of this article.

Open Access This article is licensed under a Creative Commons Attribution 4.0 International License, which permits use, sharing, adaptation, distribution and reproduction in any medium or format, as long as you give appropriate credit to the original author(s) and the source, provide a link to the Creative Commons licence, and indicate if changes were made. The images or other third party material in this article are included in the article's Creative Commons licence, unless indicated otherwise in a credit line to the material. If material is not included in the article's Creative Commons licence and your intended use is not permitted by statutory regulation or exceeds the permitted use, you will need to obtain permission directly from the copyright

holder. To view a copy of this licence, visit <http://creativecommons.org/licenses/by/4.0/>.

References

1. W. Krüger: Über eine neue Möglichkeit der Steigerung des Höchststauflusses von Hochgeschwindigkeitsprofilen, AVA-Bericht 43/W/64 (1943).
2. Rudolph, P. K. C.: High-Lift Systems on Commercial Subsonic Airliners, NASA CR-4746 (1996).
3. Boeing Commercial Airplane Group: High Reynolds Number Hybrid Laminar Flow Control (HLFC) Flight Experiment - II. Aerodynamic Design, NASA CR-1999-209324 (1999).
4. Strüber, H., Wild, J.: Aerodynamic Design of a High-Lift System Compatible With a Natural Laminar Flow Wing Within The DeSiReH Project, 29th Congress of the International Council of the Aerospace Sciences (ICAS), ICAS-2014-0300 (2014)
5. Wild, J., Palazzo, S., Brinza, I., van der Biest, K., Vervliet, A., Schmitz, H.: Design and Testing a Full Scale Laminar Wing Leading Edge High-Lift System. Springer, Cham (2022)
6. Hasabnis, A., Wild, J., Strüber, H., Moens, F., van Rooijen, B., Maseland, H.: The UHURA project at a glance – motivation and objectives. J. Phys. Conf. Ser. **2526**, 012005 (2023). <https://doi.org/10.1088/1742-6596/2526/1/012005>
7. Streit, T., Ruberte Bailo, J., Büscher, A., Seitz, A.: Design of a fast highly efficient passenger NLF aircraft with a backward swept wing and a long single-aisle fuselage, 34th Congress of the International Council of the Aeronautical Sciences (ICAS), Florence, Italy (2024).
8. Wunderlich, T.: Optimal wing aspect ratio of highly efficient long-range aircraft. 34th Congress of the International Council of the Aeronautical Sciences (ICAS), ICAS-2024-1225(2024).
9. Anderson, J.D.: Computational Fluid Dynamics: The Basics with Applications. McGraw-Hill, New York, NY (US) (1995)
10. Schwamborn, D., Gerhold, T., Heinrich, R.: The DLR TAU-code: recent applications in research and industry. ECCOMAS CFD 2006, Egmond aan Zee, Netherlands (2006).
11. Allmaras, S.R., Johnson, F.T., Spalart, P.R.: Modifications and clarifications for the implementation of the Spalart-Allmaras turbulence model. 7th International Conference on Computational Fluid Dynamics (ICCFD7), ICCFD7-1902 (2012)
12. Eberle, B., Stefes, D., Reckzeh: Clean Aviation Ultra-Performance Wing (UP Wing), AIAA SCITECH 2024 Forum, AIAA paper 2024-2109 (2024), <https://doi.org/10.2514/6.2024-2109>.

Publisher's Note Springer Nature remains neutral with regard to jurisdictional claims in published maps and institutional affiliations.