Main Science/Technology results and foregrounds

The main scientific and technological results of the STARLET project consist in the conducting of a proof of concept of fluidic load control approach for reduction or redistribution of high, off-design wing loads. In effect two alternative variants of fluidic load control systems were designed, investigated by numerical flow simulations and wind-tunnel tests (with a third sub-variant investigated solely by flow simulations) that have proven the load-alleviation effectiveness of the fluidic load control approach. The alternative variants of the investigated fluidic load control solutions were designated as Fluidic Spoiler, Double-Trailing Edge Nozzle (DTEN) actuator and a Leaky Wing which is a sub-variant of Fluidic Spoiler.

The concepts were applied on the external part of the half-span wing model in order to obtain high effectiveness of reduction of the wing-root bending moment. The half-span wing model dimensions were: span: 2400mm, root chord: 1450 mm, tip chord: 665mm, reference area: 2.49m^2,aspect ratio 4.6. Both actuators were located at the same spanwise location, between 59% and 92% wing half-span. Both actuators had also the same span, equal to 725mm. The detailed description of the actuators is provided in the following paragraphs. The concepts are described in detail in drawings in the deliverable D4.2 “Final Technical Report” of the project. The schematic of the concepts is presented in Figure 1.

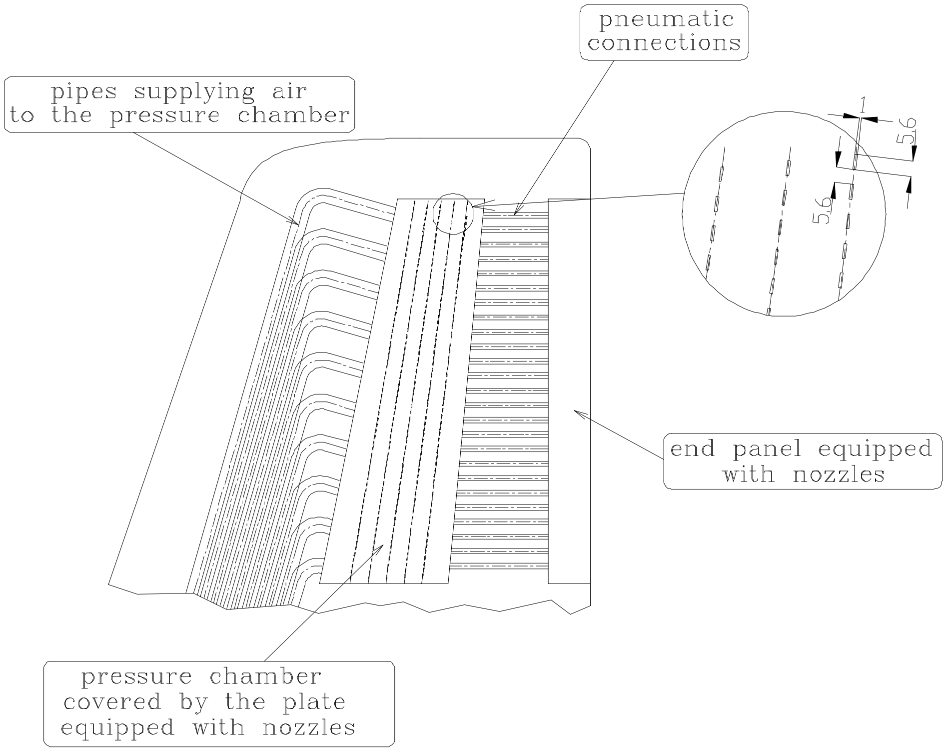

Figure 1. Schematic of the Fluidic Spoiler in the central part of the wing with air supply system and the DTEN actuator in the triling-edge region.

1. Research methodology

The research methodology consisted in designing the fluidic load control actuators, implementing them on a half-span wind-tynnel wing model and conducting numerical flow simulations and wind tunnel investigations to assess the wing load alleviation effectiveness of the concepts.

In numerical flow simulations the unsteady, Reynolds-averaged Navier-Stokes equations with k-ω turbulence model were solved in order to get knowledge about the effectiveness of the devices as well as of the steadiness of the flow pattern and aerodynamic loads with fluidic alleviation system active. In order to increase the fidelity of the solution internal flow in the nozzles of the actuators was also resolved using mass flow boundary condition at the internal end of a nozzle (inlet from pressure chamber inside the model). The boundary conditions used in the external boundaries of the domain were pressure far-field, pressure outlet and symmetry. Most of the simulations were three-dimensional flow simulations around the actual wing model. In initial stages 2.5D (constant-chord wing strip) and 2D flow simulations were also conducted in support of the design of the wing model. This way the most promising values of blowing angle of the Fluidic Spoiler with respect to airfoil chord were determined in the early stage of the project. The load-alleviation effectiveness of fluidic devices was determined in numerical and in experimental investigations by the values of the two parameters, namely: wing-root-bending-moment-alleviation coefficient C_{BMA}:

$C_{BMA} = [M_{B0}-M_B]/ M_{B0})$,

where M_{B0} is reference bending moment with alleviation system off, M_B is the bending moment with alleviation system active. More details of the redistribution of aerodynamic load on the wing were also obtained from the spanwise distributions of the bending moment, measured by tensometer system on six spanwise stations and from the local values C_{BMA} for which the local values of bending moment and reference bending moment at given spanwise station were used. The distributions of local values of C_{BMA} parameter obtained from wind-tunnel measurements and from numerical flow simulations were later compared in order to validate the computational models of the developed flow control actuators. Another parameter used in the literature for comparison of the effectiveness of fluidic devices is blowing momentum coefficient Cμ:

$$C_\mu = \frac{\dot{m} \cdot V_J}{q_\infty \cdot S},$$

where $\dot{m}$ is nozzle mass flow rate, V_j is the velocity of the jet leaving the nozzle, q_∞ is free-stream dynamic pressure and S is reference surface. This parameter is proportional to power needed to supply the system with mass flow. In numerical flow simulations both parameters, mass flow rate and Cμ were computed and the achieved results were presented against both of these parameters. In wind tunnel investigations only mass flow rate was measured. The Cμ used for comparisons of the efficiency of the devices was computed by flow simulations. This way both directions of investigations – numerical and experimental were complementary.

Large proportion of the effort was devoted to the modification of the existing half-span wing model in order to accommodate inside it the elements of fluidic load control devices. The internal wing-box structure was dismantled and made more elastic by removing some material from it. More space was obtained in the half-chord region in order to accommodate the pressure chamber of the Fluidic Spoiler. The inside of the frontal part was occupied by pipes providing air to the Fluidic Spoiler. The aileron was dismantled and replaced with the DTEN actuator. A tensometer system was applied on wing upper side in order to measure the distribution of the bending moment. In addition, the wing-root bending moment was measured by two balances located at the wing spars.

As a reference solution to compare the effects of fluidic devices a classic spoiler of the size of the Fluidic Spoiler actuating plate was used. In the case of classic spoiler its effectiveness can be determined only by the values of the root-bending-moment-alleviation coefficient. Both numerical and wind-tunnel investigations were conducted for the same conditions of angleof attack of 10 degrees, Mach number of 0.1, Reynolds number of 2.4 million. Numerical investigations of the Fluidic Spoiler were restricted to one blowing direction due to constrains in computational resources, but several variants of the device were analyzed, differing in the number of active nozzles, with the same total mass flow in the nozzle system. Wind tunnel investigations of the concepts were conducted in the 5m-diameter wind tunnel of Institute of Aviation, Warsaw. Wind tunnel investigations of Fluidic Spoiler were conducted for two angles of blowing with respect to surface: normal direction and 45 degrees upwind. The DTEN actuator had only one possible blowing direction. The schematic of the test stand is shown in Figure 2.

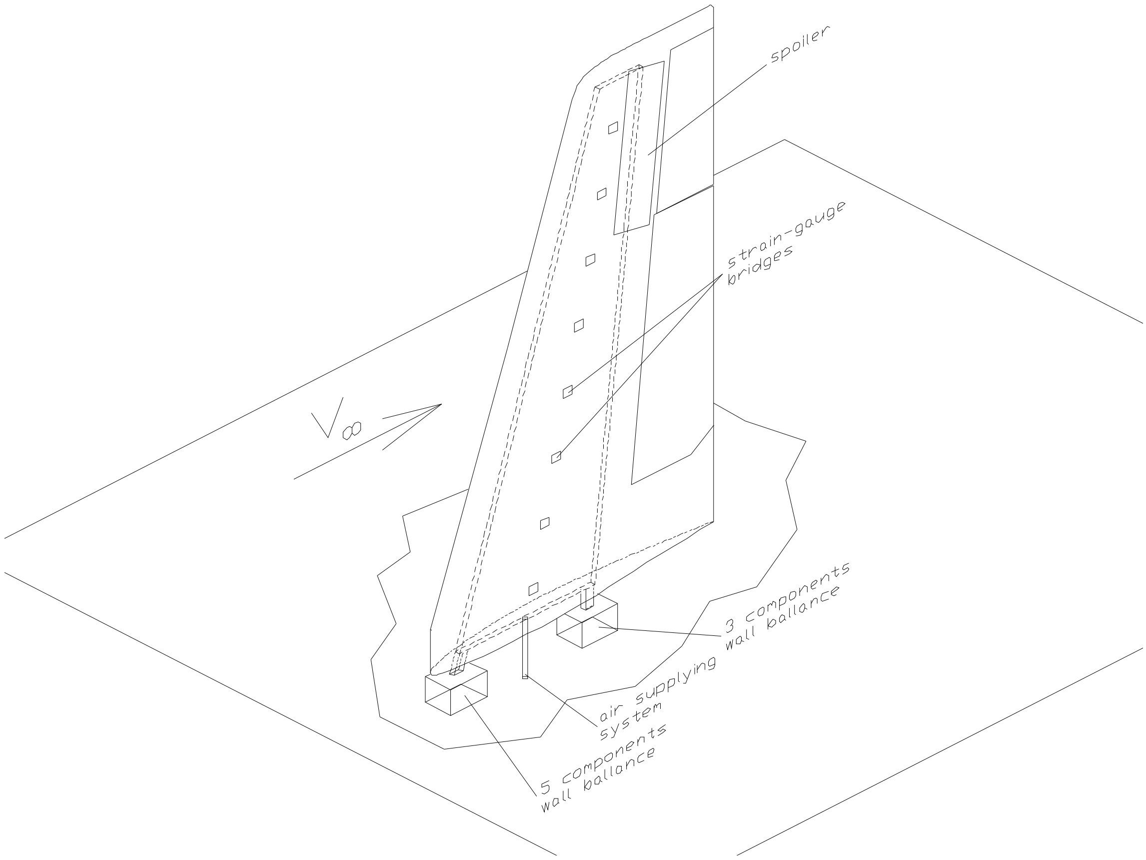

Figure 2. Schematic of the test stand for the investigated concepts of fluidic load control.

2. Description of Fluidic Spoiler and achieved results for this concept

The Fluidic Spoiler actuator consists of an array of mini-nozzles that blow air from a pressure chamber inside the wing model in the direction normal to the suction side of the wing or in direction inclined at a non-zero angle from the normal to the wing surface in the upstream direction. The actuators investigated in wind-tunnel experiments had two variants of inclination: 0° (blowing in the direction normal to the surface) and 45° in the upstream direction. These inclination angles were chosen based on numerical simulations of the effectiveness of the blowing nozzles. The most important elements of the Fluidic Spoiler are pressure chamber inside the wing and the exchangeable pressure-chamber roof plates in which the nozzles are implemented. The nozzles are arranged in 9 spanwise rows in a chequered fashion so as to avoid leaking of air in the spaces between the nozzles. Each of the nozzles had 5.6mm span and 1.0mm width. Each nozzle row consists of 60 nozzles. The schematic drawing of the Fluidic Spoiler concept is shown in Figure 3.

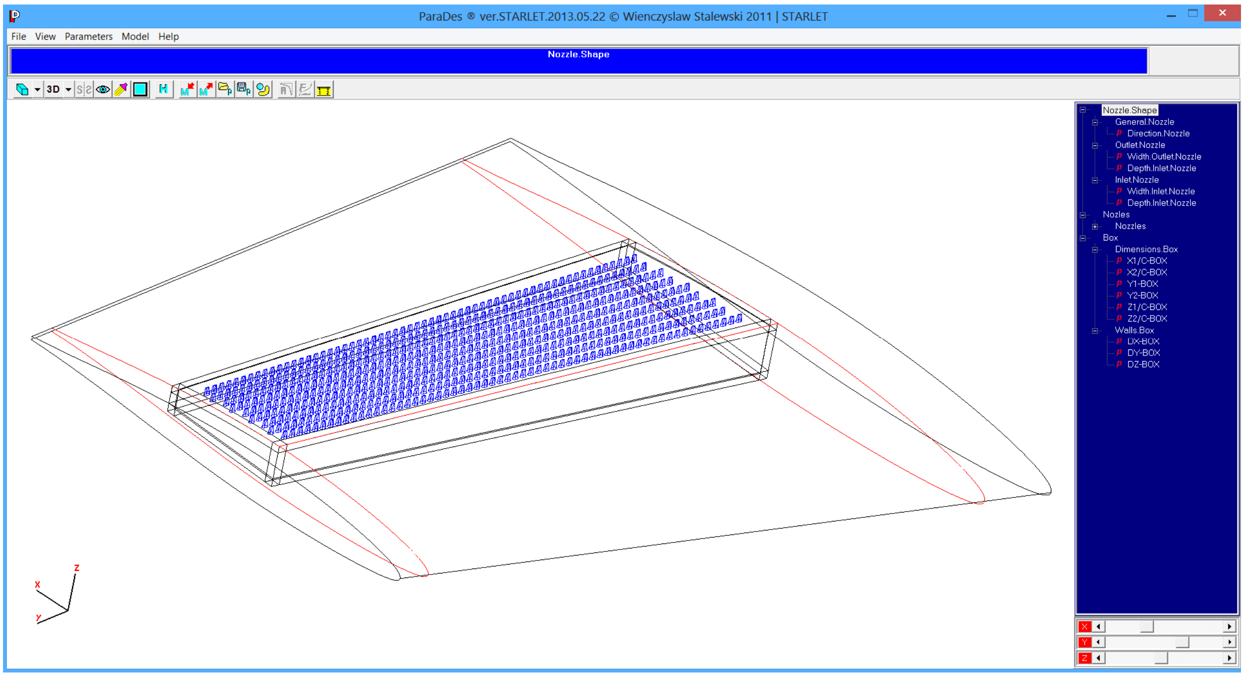

Figure 3. Three-dimensional view of the fluidic spoiler pressure chamber and roof plate

The three-dimensional view on the fluidic spoiler generated by the PARADES optimisation system developed by Instytut Lotnictwa. The nozzles were implemented in the roof plate using three-dimensional printing technology which allowed designing the external side of the roof plate as a part of the suction side of the wing. In the experiments conducted in the Institute of Aviation the nozzle system was provided with high-pressure air from the high-pressure tanks used as second-flow system in the low-speed wind tunnel and the high-pressure source for another, transonic wind tunnel. If such a fluidic flow control system is to be implemented on a real-scale aircraft a suitable source of high-pressure air has to be implemented onboard.

As the reference load alleviation solution a classic spoiler of a length of 10% chord (of the size of the nozzle plate of Fluidic Spoiler) was used. The results - the CBMA coefficient as a function of spoiler deflection is shown in Figure 4.

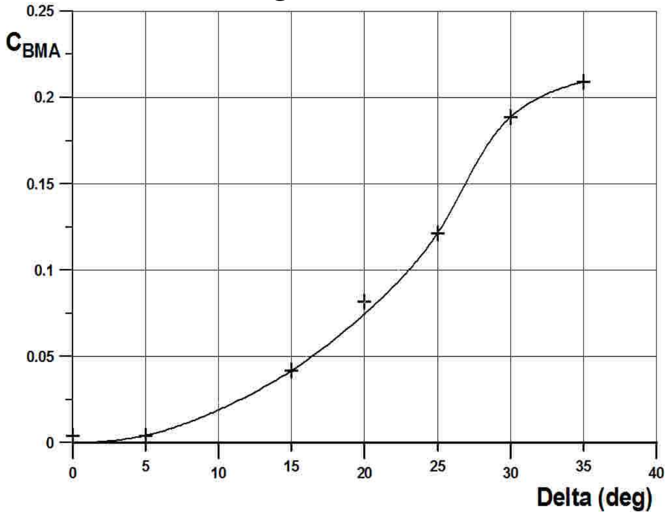

Figure 4. CBMA coefficient as a function of classic spoiler deflection, M = 0.1, Alpha = 10^0

It can be seen that for significant load alleviation effect significant deflections of a spoiler are necessary. At about 30 deg. deflection anhgle saturation effects are visible.

The results of wind tunnel experiment have shown that in blowing normal to wing surface the maximum effectiveness of load alleviation, measured by the reduction of wing-root bending moment, obtained with all rows active was approximately 12% at mass flow of 0.250 kg/s (Figure 5). The most effective configuration was the one with two front nozzle rows active and blowing direction 45 deg. upwind (Figure 6).

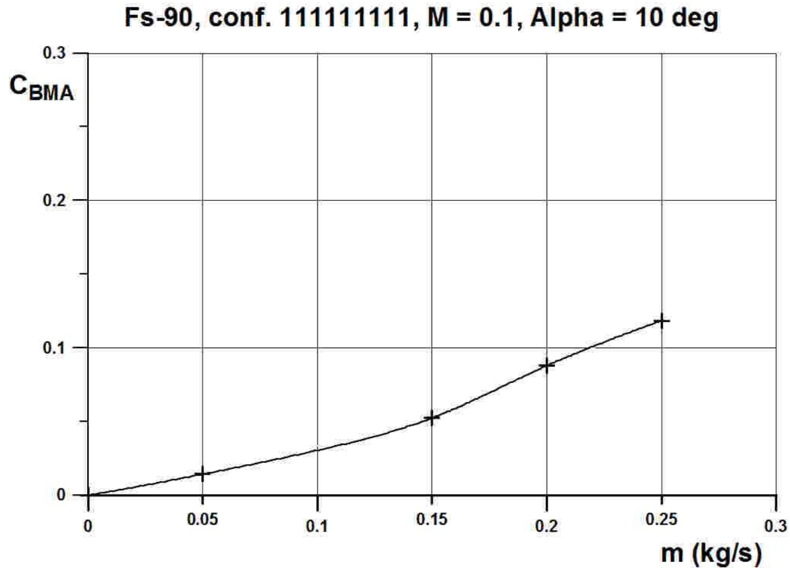

Figure 5. C_{BMA} coefficient as a function of mass flow rate for Fluidic Spoiler with all nozzle rows active, blowing direction normal to surface.

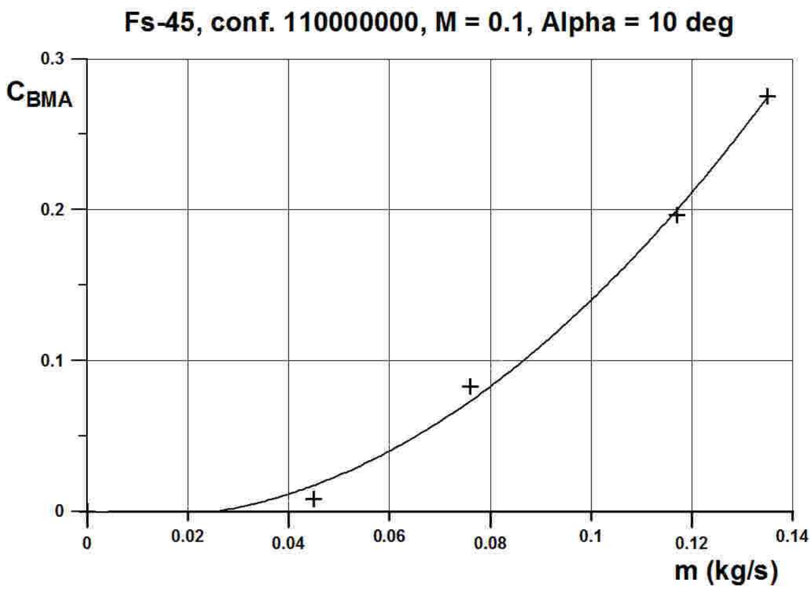

Figure 6. C_{BMA} coefficient as a function of mass flow rate for Fluidic Spoiler with two front nozzle rows active, blowing direction 45 deg. upwind.

Comparison of results of numerical flow simulation and wind tunnel tests is presented in Figure 7. For the Fluidic Spoiler the results of the wind tunnel tests qualitatively confirmed the results of numerical flow simulations conducted for nozzles directed normal to wing surface. The alleviation effect for all active nozzle rows active was more linearly dependent on the mass flow rate than was for lower number of active rows and for two rows active the noticeable effects of load alleviation occurred at nozzle mass flow rate higher than 0.1 kg/s, as in the flow simulations. Maximum values of wing-root bending moment reduction were, however, 30% higher than predicted by the flow simulations. A likely reason for this effect was absence in real flow, or lower intensity of small vortices that appeared near the nozzles in the numerical solution and produced local suction. The values of static pressure measured in the vicinity of the nozzles were higher than predicted in flow simulations. More information could be obtained from PIV scans, however, the PIV technique was not applied in these investigations. In contrast to numerical flow simulations that required large amount of computational time, in wind tunnel experiment larger number of nozzle configurations could be investigated. The experiment confirmed the conclusions of early two-dimensional and two-and-a-half dimensional flow simulations that deflecting blowing direction 45° upwind increases the load alleviation effect. Of the total number of 20 configurations differing in the number of active rows and blowing directions the most effective one was the configuration with first two nozzle rows active, deflected 45° toward the flow from direction normal to surface. The maximum wing-root alleviation level was 28% at 0.135 kg/s nozzle mass flow and at that blowing rate the saturation effect did not yet occur. This was, however, the maximum applied value of nozzle mass flow rate for this configuration due to concerns about the pressure rise in the pressure chamber inside the model and about the strength of the nozzle plate produced by the 3D-printing technology. The noticeable effects of this configuration on wing-root bending moment appeared at nozzle mass flow of 0.4 kg/s and after exceeding this value the dependence of wing-root bending moment alleviation on the nozzle mass flow was almost linear.

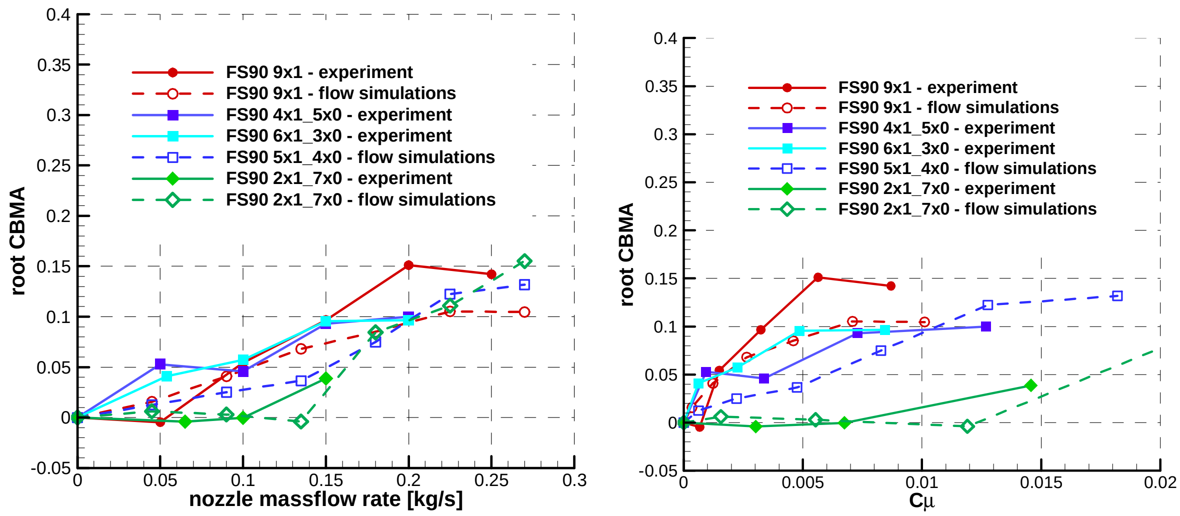

Figure 7. Comparison of wing-root bending alleviation coefficient obtained from flow simulations and from wind-tunnel investigations for the Fluidic Spoiler concept

3. The DTEN concept

The DTEN concept consist of an array of nozzles located in the trailing edge of the external part of the wing at the same span location where the Fluidic Spoiler is located. The DTEN actuator is supplied with air from the same pressure chamber from which the Fluidic Spoiler is supplied. Each nozzle section is connected by a pipe with the pressure chamber, which for the investigations of the DTEN concept is covered with a solid roof plate. The three-dimensional view of the concept is shown in Figure 8 and in Figure 9 In Figure 10 a detail of the trailing edge region is shown demonstrating the outlet of the deflected stream of air.

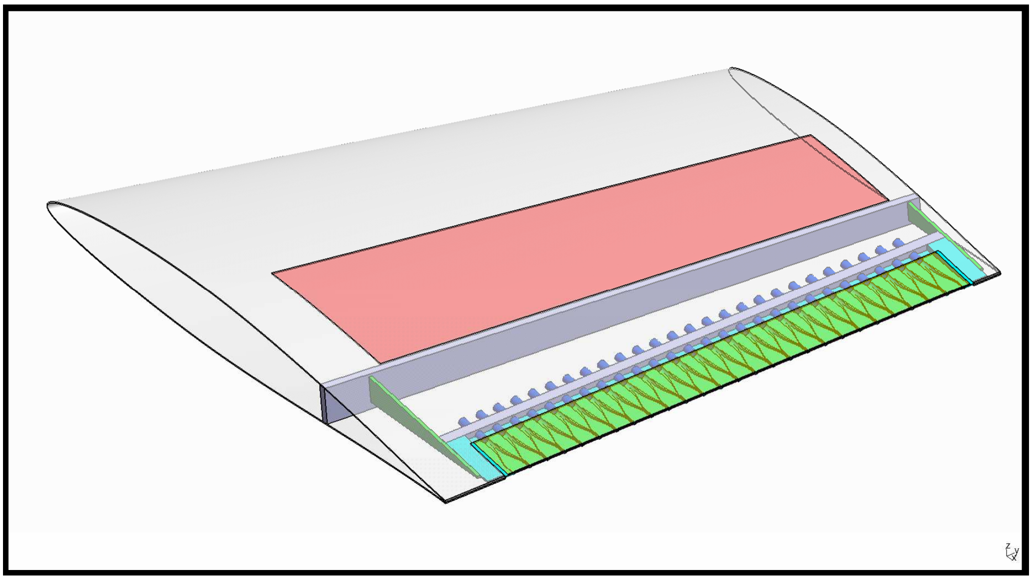

Figure 8. View of the DTEN concept assuming translucent upper plate of the actuator at the trailing edge. Pipes connecting the nozzles with the pressure chamber are not shown

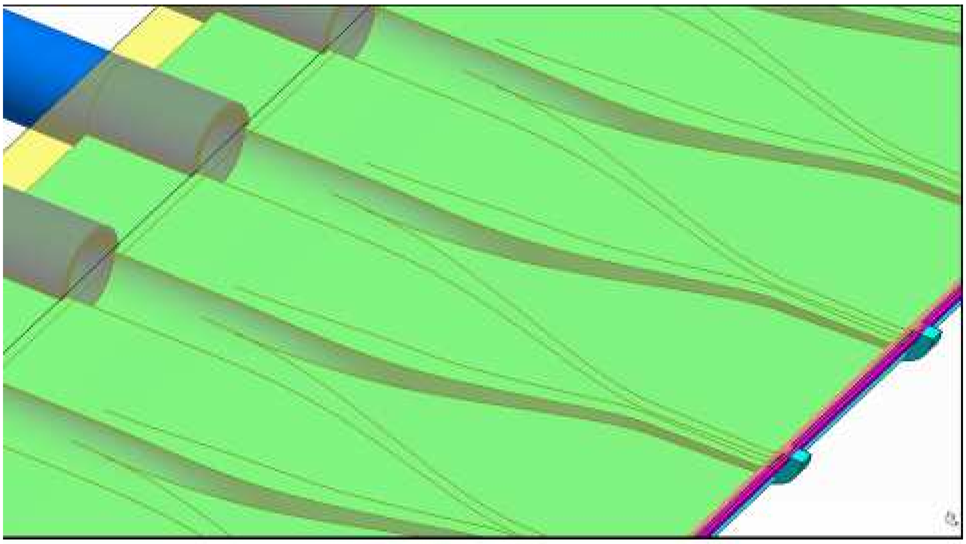

Figure 9. Enlarged fragment of the DTEN actuator.

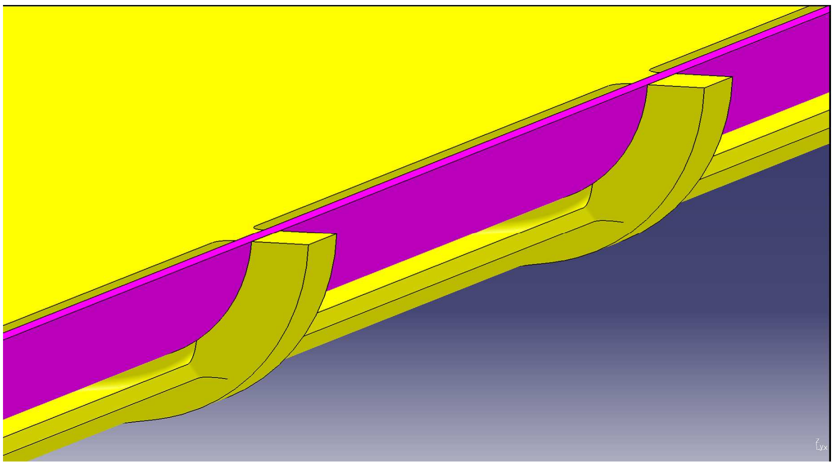

Figure 10. Enlarged fragment of the nozzle region of the DTEN actuator.

The air leaving the pressure chamber flows on either side of the purple surface and the Coanda effect is generated on the lower side of the purple wall and on the lower side of the wing as a result of interaction between the nozzle stream passing on either side the narrow purple surface and the flow passing the wing lower surface. This way the air on the lower surface of the wing is accelerated in the trailing edge region producing area of low pressure there. The rapidly diverted upwards air stream generates stagnation region near the trailing edge on the upper side of the wing. Both phenomena combine to modify lift in the wing segment where the actuator is located. This actuator is one-directional – in the current configuration it may only be used for reduction of lift. The pressure distribution in a control section crossing the DTEN actuator is shown in Figure 11 and comparison of numerical and experimental results of alleviation of wing root bending moment is shown in Figure 12.

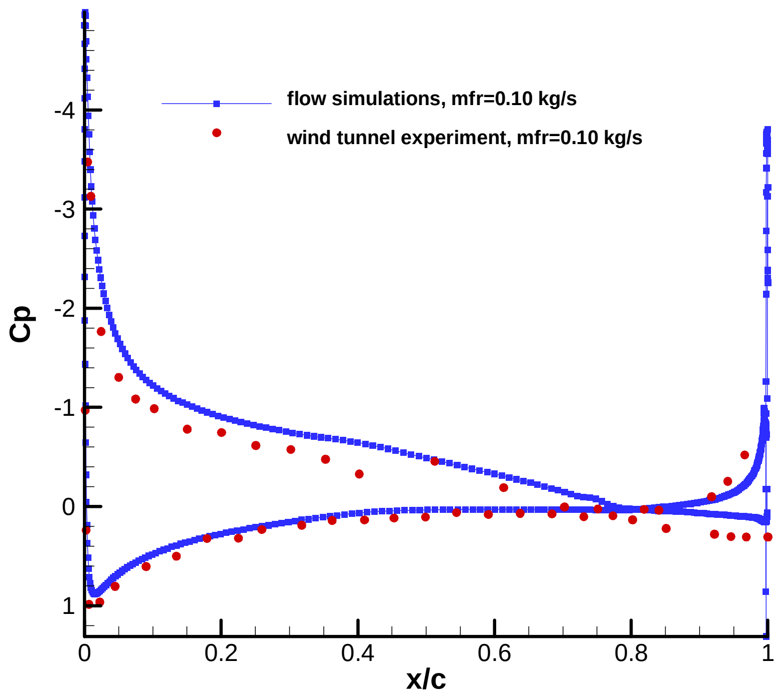

Figure 11. Comparison of pressure distributions in the cross-section at y=1750mm for the DTEN device.

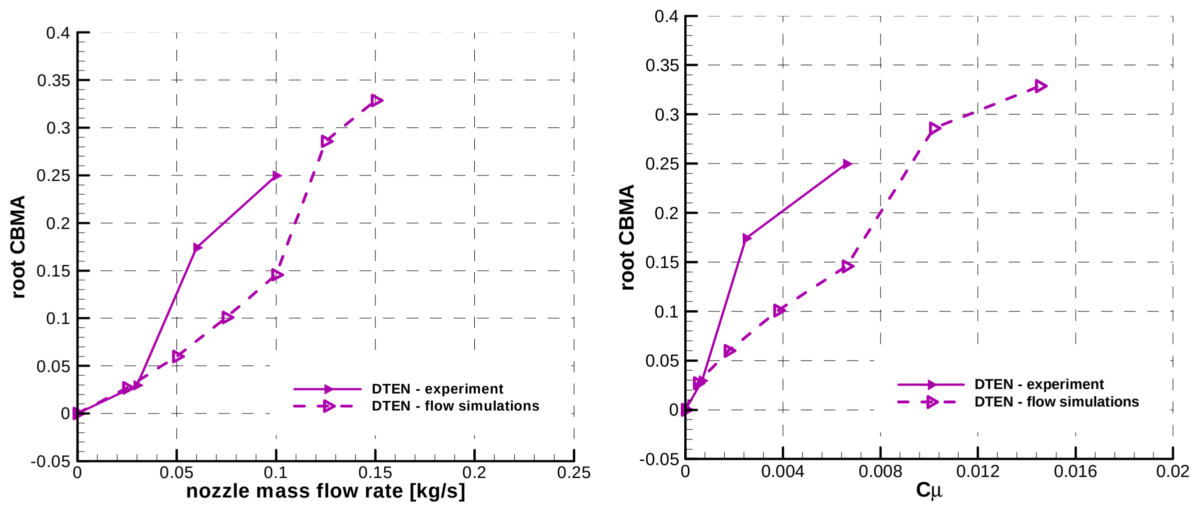

Figure 12. Comparison of wing-root bending alleviation coefficient obtained from flow simulations and from wind-tunnel investigations for the DTEN device.

The results of flow simulations revealed that the DTEN actuator was working in two modes: for lower values of nozzle mass flow rate, up to 0.1 kg/s the nozzles were acting similar to a Gurney tab, deflecting the flow but without producing the Coanda effect. Only after reaching this value of nozzle mass flow rate the Coanda effect appeared and upward deflection of flow behind the trailing edge was accompanied by appearance of a negative pressure region in the lower part of the trailing edge, supporting the load alleviation effect. The maximum wing-root alleviation level was 32% and this was achieved at nozzle mass flow rate of 0.15 kg/s. The side-effect of this concept was a nose-up increase of the pitching moment which was equal to ΔCm=0.068 at the maximum load alleviation effect. For the Fluidic spoiler the pitching moment changes were lower; depending on the location of the number and location of the active rows the pitching moment changes did not exceeded the value of ΔCm=0.001, positive or negative, depending on the chord position of the active nozzles. It must be noted, however, that changes of pitching moment occur also with the traditional load-alleviation systems, such as symmetrical deflections of ailerons on the Lockheed C-5 Galaxy aircraft. They have to be countered by compensatory deflection of elevators. For the DTEN actuator the wind-tunnel investigations confirmed the predictions of numerical simulations that at low values of the nozzle mass flow rate the Coanda effect does not appear or has low intensity, but the strong Coanda effect appeared at the nozzle mass flow rate of 0.055 kg/s which was similar mass flow rate to the rate necessary for activation of the most effective variant of the Fluidic Spoiler (generating flow separation on suction side of a wing). The maximum achieved wing-root bending moment alleviation effectiveness of the DTEN actuator was 31% which was slightly higher than for the most effective variant of the Fluidic Spoiler, but at this maximum effectiveness level the saturation effect was noticeable.

It must be noted also, that both investigated concepts had higher load-alleviation capability than traditional spoiler of 10% wing section chord. For the classic spoiler located in place of the Fluidic Spoiler the wing-root bending moment alleviation level was 21% and this was achieved at a high deflection of 35 degrees. In practical situations achieving such high deflection requires large actuation moments produced by the hydraulic system and generates large increase of drag. For fluidic Spoiler the drag increase is moderate and results from the decreased suction of the leading edge due to decreased velocity circulation. In case of the DTEN actuator there occurs another component of drag – the suction on the rounded trailing edge which acts in the rearward direction. For this reason the drag increase due to activation of the DTEN device is higher than due to deflection of aileron, but lower than due to deflection of spoiler.

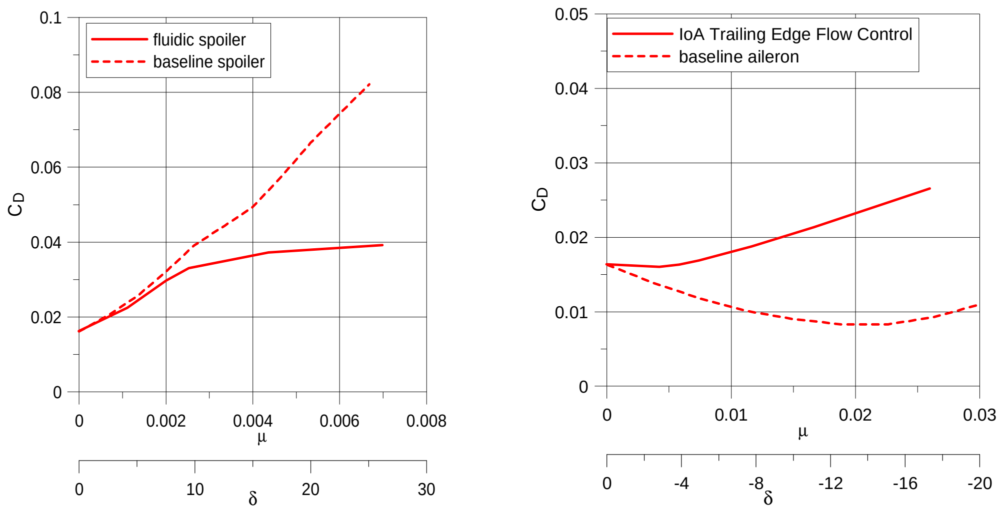

Figure 13. Comparison of drag changes due to tested devices and due to deflection of baseline spoiler and 30% chord aileron (CFD results)

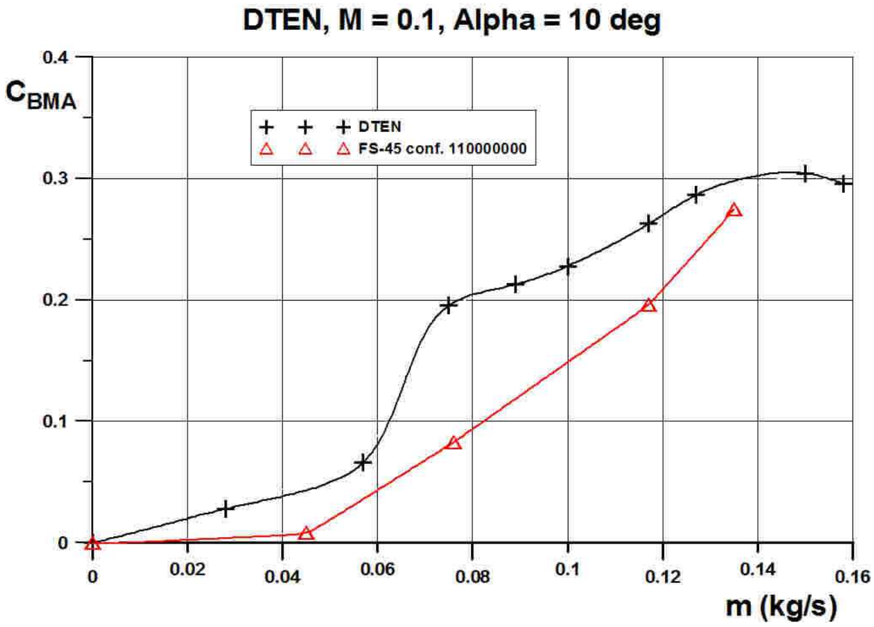

Figure 14. Root-bending-moment-alleviation coefficient as a function of the blowing momentum coefficient for DTEN and FS-45 (config. 110000000), M = 0.1 and Alpha = 10^0.

4. The Leaky Wing concept

As part of complementary actions, not included in the original scope of the project the "Leaky Wing" concept has been developed and investigated by numerical flow simulations. The complementary actions were aimed at increasing chances of further development of the proposed concepts in future projects. They included the development of the Leaky Wing concept and conducting numerical simulations of dynamic changes of aerodynamic loads and load alleviation effects in simulated gust conditions.

The concept of the "Leaky Wing" has been developed as a particular case of Fluidic Spoiler, utilising only natural pressure differences between wing pressure and suction surfaces. During the design process of the concept, a number of requirements and constraints were taken into consideration, including the simplicity and feasibility of the concept, minimal interference with the wing structure (which could weaken the structure) and high efficiency, reliability and response rate of the proposed system in mitigating excessive aerodynamic loads. The proposed concept consists of a matrix of transverse ducts connecting upper and lower surface of the wing. The ducts should be placed rather in outer part of the wing, because this part is a source of largest bending moments acting on the wing structure. On the other hand, such placement of ducts may be risky because of possible unfavourable influence of "fluidic spoiler" on the aileron effectiveness. However this aspect needs further investigations. In normal flight conditions, the ducts are closed, preferably in such a way as not to interfere with the flow around the wing. In extraordinary flight conditions, when excessive bending loads of the wing may occur (e.g. during sudden gusts, accelerated manoeuvres, etc.) the transverse ducts are opened, which should initialise intensive air flow through the ducts, being the effect of considerable difference of static pressure between the suction and pressure sides of the wing. The concept is shown schematically in Figure 15.

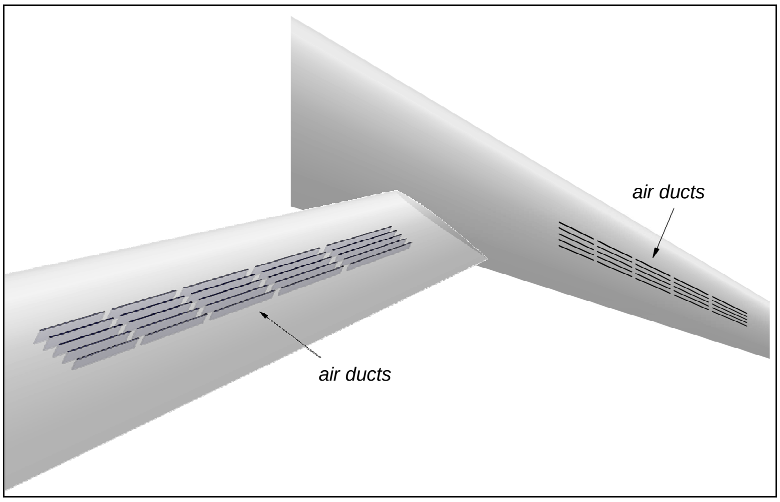

Figure 15. View of the wing with implemented the Leaky Wing concept

Such phenomenon is typical for the flight conditions with occurrence of high aerodynamic loads of the wing. After opening the air ducts, the flow through them is expected to initiate the separation of the main flow on the wing suction side. In situations when main-flow separation already exists before opening the ducts, it is expected that flow through the ducts may enhance effects of existing separation. In both cases, the opening of the ducts in high-wing-load flight conditions, should decrease difference in static pressures on suction and pressure side of the wing, this way decreasing aerodynamic loads acting on the wing structure.

The effectiveness of the "Leaky Wing" concept and its response rate in conditions of sudden gust, were investigated by series of URANS simulations. The simulations were conducted for gust conditions at two flight speeds: M=0.20 and M=0.45. A simple closed-loop control system of the load alleviation was simulated, based on the value of Load Factor (LF=Lift/Weight). The load alleviation system was activated when Load Factor exceeded threshold value of 1.3 and deactivated when it fell below 1.2. In the simulated gust conditions the measure of load-alleviation effectiveness was the relative difference between the wing-root bending moments measured for inactive and active wing-load-alleviation system. Based on conducted flight simulation, the effectiveness was estimated to be around 17-18%. Results of the simulations confirmed that the proposed "Leaky Wing" system quickly responds to increasing aerodynamic loads and its load-alleviation response effect should be quick enough to protect wing structure against fatigue damage. The results of simulations are shown in Figure 18 and Figure 19

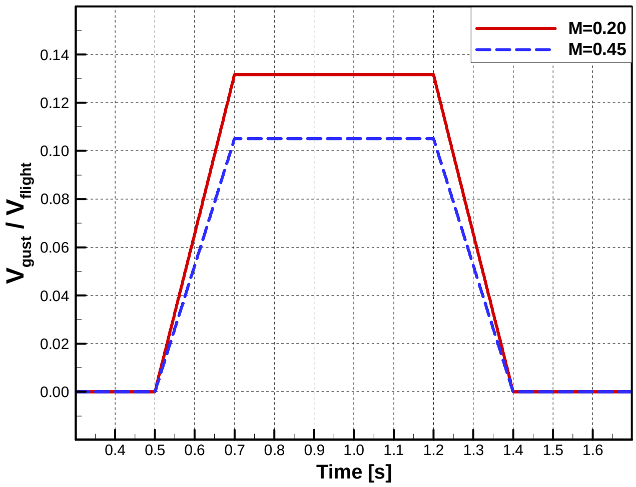

Figure 16. Simulated gust profile.

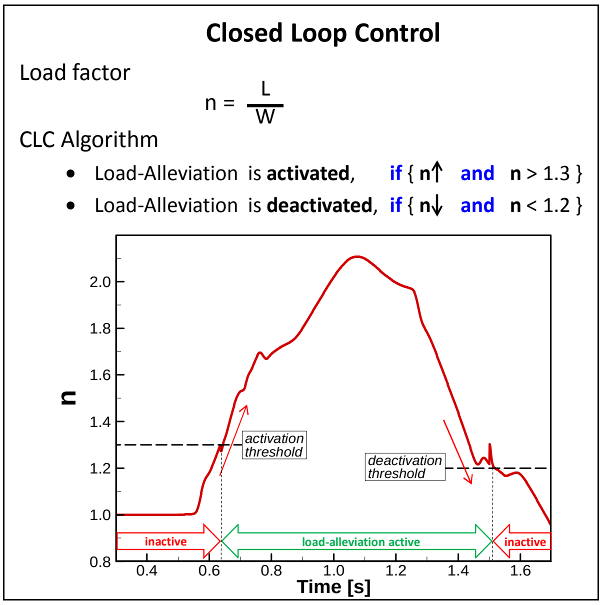

Figure 17. Simulated closed-loop control system.

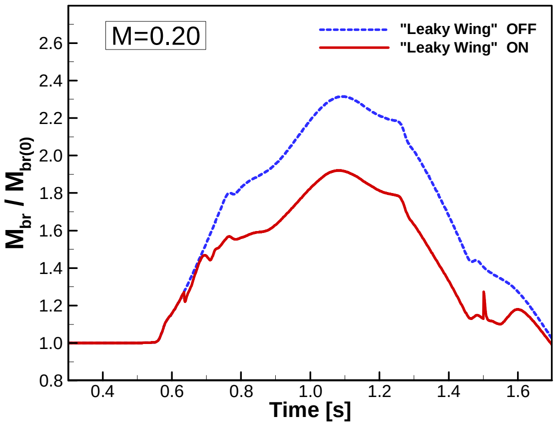

Figure 18. Time histories of wing-root bending moment (M_{br}) related to its value in nominal flight conditions ($M_{br(0)}$), measured for the wing-load-alleviation system OFF and ON, at flight speed M=0.20.

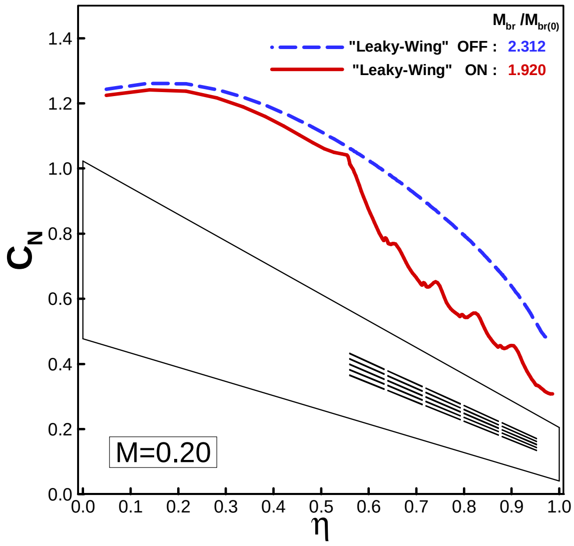

Figure 19. Comparison of spanwise distribution of aerodynamic bending-load coefficient (C_N), for the "Leaky Wing" system OFF and ON at highest values of wing-root bending moment in gust conditions.

At Mach number of 0.45 in non-alleviated case the angle of attack corresponding to maximum lift was reached and exceeded which can be seen as reduction of bending moment after reaching a pek value in Figure 20. Load alleviation with the Leaky Wing system reduced this peak of wing-root bending moment.

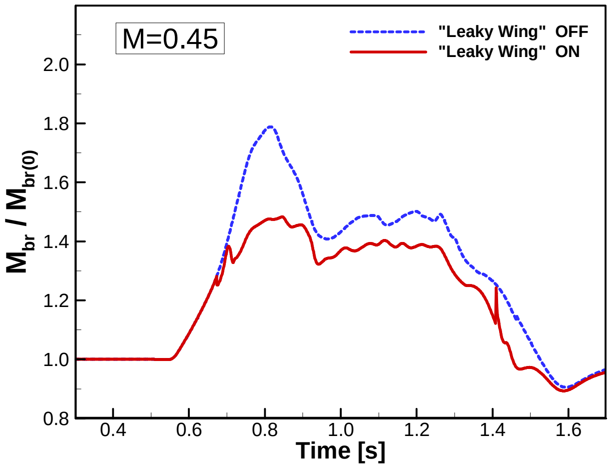

Figure 20. Time histories of wing-root bending moment (M_{br}) related to its value in nominal flight conditions ($M_{br(0)}$), measured for the wing-load-alleviation system OFF and ON, at flight speed M=0.45.

5. Conclusions

As a conclusion it may be stated that the results of the conducted works have proven the load alleviation effectiveness of the developed concepts. The load alleviation effectiveness of Fluidic Spoiler was higher than the effectiveness of a classic spoiler of the size of the actuating plate of Fluidic Spoiler. It must be noted, however, that the presented solutions of fluidic load control are still at very low technology readiness level (TRL2 or TRL3). More research is needed, concentrated on shape, dimensions, positions and characteristics at higher Mach numbers including possibilities of integratiion with wing structure and aeroelastic properties of wing equipped with such systems.