SWISS / FINNISH COMPUTATIONAL FLUID DYNAMICS SIMULATION ON THE F/A-18

M. Guillaume*[&], A. Gehri*, P. Stephani*, J. Vos**, T. Siikonen***, E. Salminen***, G. Mandanis[+]
*RUAG Aviation; Switzerland, **CFS Engineering; Switzerland, ***Finflo Ltd; Finland, [+]M&M GmbH Mandanis; Switzerland
[&]School of Engineering ZHAW; Switzerland, michel.guillaume@zhaw.ch

Keywords: *Computational Fluid Dynamics, Simulation, Loads*

Abstract

Since 1995 the Swiss and Finnish Airforces have been operating the F/A-18C/D Hornet as their leading fighter aircraft. Both countries decided to invest in Computational Fluid Dynamics (CFD) simulation tools to study and analyze the flow over the F/A-18. The goal was to develop a modern simulation environment to support the engineering and maintenance of this aircraft, and in particular its structural integrity. In 2005 at the first meeting between RUAG/CFS from Switzerland and Finflo Ltd. from Finland it was decided to perform common research on CFD to improve the grids and solver technology used to simulate the flow over the F/A-18 aircraft. The main objective of these CFD simulations is to provide steady and unsteady loads for engineering investigation as a supplement to the very expensive flight test program.
A first series of CFD calculations were carried out in Switzerland for Swiss design load conditions and aerodynamic forces and pressure distributions were compared to the data provided by Boeing from St. Louis.
To compare the two non-commercial flow solvers NSMB from Switzerland and FINFLO from Finland the test case M6 wing from ONERA was used. Some differences were observed, which were attributed to the different grids, solution methods and turbulence modeling approaches.
In 2010 load cases from the Finish operational loads monitoring program (MINIHOLM) were selected to run CFD calculations. From the CFD results the structural component loads at reference locations were computed. The results obtained for both the Swiss and Finnish grids were excellent and only small differences in component loads were observed.

The Swiss-Finnish collaboration is very unique and provides a great opportunity to improve the complex CFD calculations on the F/A-18. Both countries will profit from this effort which is much more than just research in the field of advanced CFD calculations. The advantage provides a considerable improvement in mesh grid strategies and in numerical simulation for accurate maneuver loads prediction.

1 Nomenclature

AoA	=	Angle of Attack
ASIP	=	Aircraft-Structural-Integrity-Program
BM	=	Bending Moment
CAD	=	Computer Aided Design
CFD	=	Computational Fluid Dynamics
FEM	=	Finite Element Model
FSI	=	Fluid Structure Interaction
LC	=	Loadcase
LEX	=	Leading Edge Extension
MI	=	Modal Integration
MPI	=	Message Passing Interface
NSMB	=	Navier Stokes Multi Block
OEM	=	Original Equipment Manufacturer
Q	=	Dynamic pressure
RANS	=	Reynolds Averaged Navier Stokes
SFH	=	Service Flight Hour

TEF = Trailing Edge Flap
TFI = Trans Finite Interpolation
TQ = Torque or torsion moment

2 Modeling Development

Both solvers, NSMB for Switzerland [1, 2] and FINFLO [2, 3] from Finland, are Navier-Stokes codes based on the cell-centered Finite Volume method using multi block structured grids. In this study only symmetrical load conditions are considered therefore half models were used for CFD calculations. The Swiss grid consists of approximately 14 million cells and was generated using ANSYS ICEMCFD (3 000 blocks). The Finnish gridh features approximately 15 million cells and was generated using Gridgen (76 blocks, 81 blocks with Sidewinder). FINFLO used the Chimera technology for different flap positions. In the Swiss grid all gaps on moving control surfaces were closed, while they were all open in the Finnish grid. A first series of calculations was made for Swiss design load conditions and aerodynamic forces and pressure distributions were compared. Some differences were observed, which were attributed to the different grids, solution methods and turbulence modeling approaches. The agreement in results was good, especially if one keeps in mind the complex geometry and the complex flow physics.

From the CFD results the structural component loads at reference locations were computed to assess the quality of the simulations. These reference locations correspond to the locations used during the Swiss ASIP study performed by Boeing in St. Louis. Four symmetrical load cases at different points in the sky were selected for this study. The results obtained on both the Swiss and Finnish grids were excellent; some differences were observed at the leading edge and trailing edge flaps due to the different grid strategies in this area (open versus closed gaps). Also the surface grid of both were compared, which showed small differences at several locations. For the next series of calculations the same surface model was used. On both the Swiss and Finnish side improved grids were developed, which consists in the order of 25 million cells for half a model. The old calculations were redone and showed an improvement in results.

2.1 The Swiss Approach

2.1.1 NSMB Solver

The calculations of the Swiss F/A-18 flow field are carried out using the NSMB Structured Multi Block Navier Stokes Solver. NSMB was developed from 1992 until 2003 in a consortium composed of two universities, namely EPFL (Lausanne) and KTH (Stockholm), one research establishment CERFACS (Toulouse) and two industrial companies Airbus France (Toulouse) and SAAB Aerospace (Linköping). Since 2004 NSMB has been developed in a new consortium lead by CFS Engineering and composed of RUAG Aviation (Emmen), Astrium Space Technologies (Les Mureaux), EPFL (Lausanne), ETHZ (Zürich), IMFT (Toulouse), IMFS (Strasbourg), the Technical University of Munich and the University of the Army in Munich.

NSMB employs the cell-centered Finite Volume method using multi block structured grids to discretize the flow field. Various space discretization schemes are available to approximate the inviscid fluxes, among them the 2nd and 4th order centered scheme with artificial dissipation, and 2nd, 3rd and 5th order upwind schemes.

The space discretization leads to a system of ordinary differential equations, which can be integrated in time using either the explicit Runge Kutta scheme or the semi-implicit LU-SGS scheme. To accelerate the convergence to steady state the following methods are available:

- local time stepping
- implicit residual smoothing (only with the Runge Kutta scheme)
- multigrid and full multi grid (grid sequencing)
- pre-conditioning for low Mach number
- artificial compressibility for incompressible flows

The ALE approach is available to simulate the flow on deforming grids. Recently, a re-meshing algorithm was implemented in NSMB to permit the simulation of the flows on deforming grids, as found for example in Fluid Structure Interaction problems. NSMB has no limit on the number of blocks used in a calculation. Block interfaces do not need to be continuous since a sliding grid block interface treatment is available.

2.1.2 F/A-18 Grid Generation

The most time-consuming process in a CFD simulation is the generation of the grid. This involves different steps. First (if required) the CAD surface needs to be cleaned up, then a multi block topology needs to be set up, and finally the grid is generated.
The F/A-18 single seat model was the basic configuration for this study. The gaps between the control surfaces were closed for simplification. Important antennas were incorporated. No engine model was used. The turkey feathers of the engine were omitted. Pylons, tanks and AIM-9 and AMRAAM missiles are part of the store configurations.
The latest grid for the F/A-18 fighter was generated by the RUAG Department of Aerodynamics in collaboration with Mindware, using ANSYS ICEM CFD software. The half model grid has 3377 blocks and 14.5 million cells see Fig. 1.

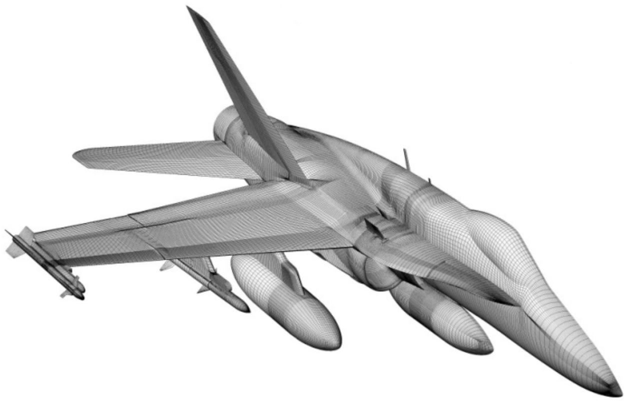


Figure 1: Detail of the F/A-18 grid (half model).

2.1.3 Fluid Structure Coupling

To predict accurate loads the stiffness of the structure has to be considered especially on the wing. Therefore a static structure coupling procedure was developed. The geometric coupling techniques implemented in the Fluid Structure Interaction (FSI) belongs to the class of scattered data interpolation methods between the structural grid and the computational fluid gid [4].

The Swiss load case corresponds to an 8.25 g steady-state pull-up maneuver at an angle of attack AoA = 15.9°. In this condition the wing tip deforms due to the high loads up to 0.5m (see Fig. 2), and one can expect that this change in wing shape will influence the flow over the wing, and thus on the aerodynamic loads. To investigate this effect an iterative CFD calculation on a flexible F/A-18 wing (with control surfaces) is made. Four iteration steps are needed to reach the equilibrium between aerodynamic and structural forces. During this simulation the fuselage, horizontal stabilizer, vertical tail and rudder are considered as rigid.

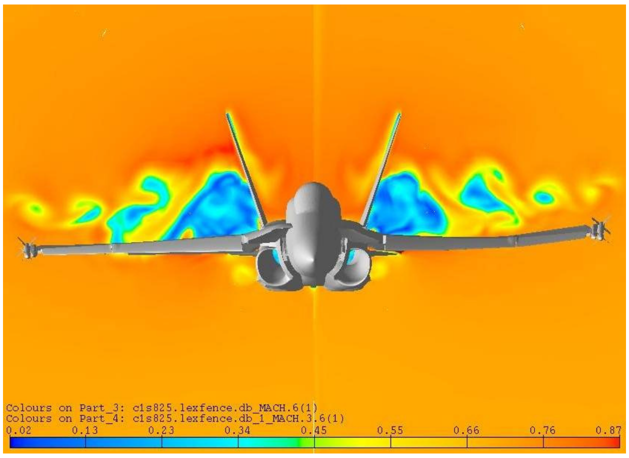


Figure 2: Impact of deformed wing on left side.

2.2 The Finish Approach

2.2.1 FINFLO Solver

The development of FINFLO flow solver dates back to 1987, when a CFD research project was started at the Helsinki University of Technology. One of the initiators of CFD research was the Finnish Air Force, which needed the right tools and expertise to study aerodynamic loads. Today the Air Force is still one of the main users of the code via Patria and Finflo Ltd. Since 2001, the software has been maintained and developed by Finflo Ltd., a company founded by the developers of the original code. The code is also developed at Aalto University, Lappeenranta University of Technology and VTT Technical Research Centre of Finland.

In FINFLO a structured multiblock grid topology and on a finite-volume technique are used [5, 6]. Geometry modeling is enhanced by a Chimera technique and discontinuous block interfaces, while a multigrid technique is utilized to accelerate convergence. The FINFLO code has been parallelized using the MPI standard to allow it to be run on both servers and multicore workstations. Turbulence modeling using FINFLO can range from k–ε [7] and SST k–ω models [8, 9] to full Reynolds stress closure. For the solution the following methods are utilized:

- local time stepping
- implicit residual smoothing (only with the Runge Kutta scheme)
- multigrid and full multigrid (grid sequencing)
- pre-conditioning for low Mach number
- artificial compressibility or pressure correction for incompressible flows

The ALE approach is also available to simulate the flow on deforming grids.

One important field where FINFLO has been applied is that of rotating machinery. The code can be used to simulate flows e.g. in pumps, in high-speed compressors, and around ship propellers. The same approach can be utilized in aircraft pull-up simulations by setting the external flow field to rest and putting the grid into a circular motion [3]. The aircraft can be considered to be attached to the end of a whirling arm pivoted at a point somewhere above the aircraft. The angle of attack cannot be modeled in the traditional way, i.e. by manipulating the direction of the external flow. It must be taken into account in the determination of the pivot point location as shown in Fig. 3. Note that the angle of attack is not constant, but increases from nose to tail.

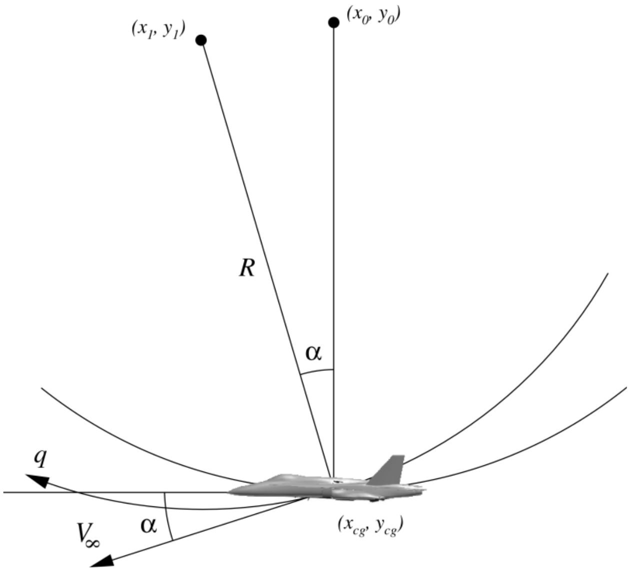


Figure 3: A definition of the angle of attack as the aircraft is in a pull-up motion.

The radius of the pull-up circle is obtained from

$$R = \frac{V_\infty^2}{g(n-1)} \tag{1}$$

where n is a load factor, g the acceleration of gravity and V_∞ the free stream velocity. The angular velocity q is

$$q = \frac{V_\infty}{R} = \frac{g(n-1)}{V_\infty} \tag{2}$$

In the present solution system a steady-state simulation can be performed. As a result of the grid motion the aircraft sees the steady flow field as if it were curved (see Fig. 3).

2.2.1 F/A-18 Grid Generation

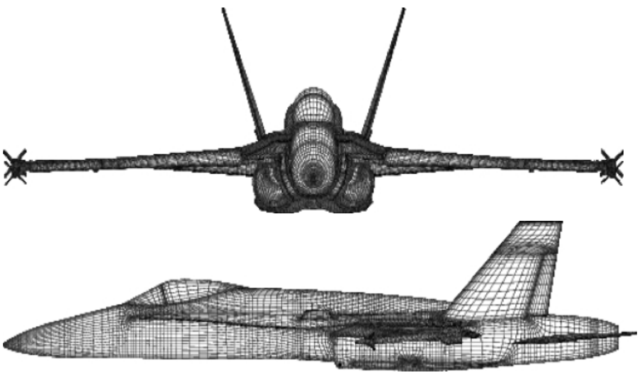
The Finnish half plane F-18C grid consist of 76 computational blocks and the number of cells in these blocks is 13’760’000. With the AIM-9M Sidewinder missile in place, the half plane grid contains 15’873’536 cells in 81 blocks (see Table 1).

	Blocks	NOF cells	Surface elements
Base	76 (12)	13 760 000	191 136
AIM-9M	5 (5)	2 113 536	33 024
Total	81 (17)	15 873 536	224 100

Table 1: F/A-18C computational grid details. Number of Chimera blocks shown in parentheses.

The surface grid with the wing tip missile is shown in Fig. 4. The overlapping grid structure (Chimera) around movable control surfaces in a wing section is illustrated in Fig. 5. The grid was created using the Gridgen software.

Neither the Finnish F/A-18C CFD model nor the FINFLO flow solver contains any engine models. However, the effects of the engine on the surrounding flow field can be described by defining the flow conditions at locations where the engine (General Electric F404-GE-402) would be connected to the CFD model (see Fig. 6).



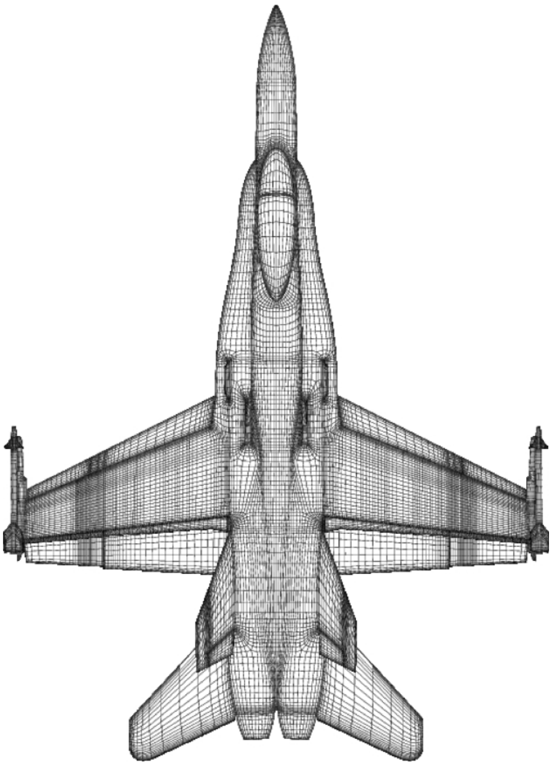


Figure 4: Finnish F-18C surface grid with AIM-9M (only every other grid line is shown).

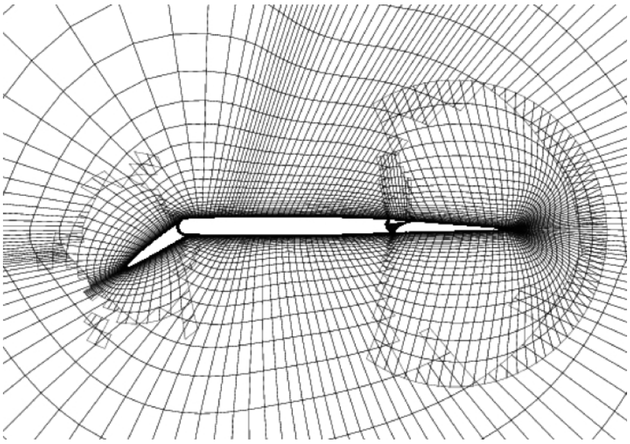


Figure 5: Chimera blocks around the control surfaces.

Figure 6: Engine boundary values are defined at the compressor location and at the nozzle inflow location.

The flow conditions through the engine are obtained using a separate computer program provided by the engine manufacturer. In this case the flow situation is interesting only at stations 1, 7, 8 and 9 (see Fig. 7). Flow situations at stations 1 and 7 are used as boundary conditions in the CFD model. The flow solver handles station 1 as an outlet, since the flow comes out of the grid, and station 7 as an inlet, since the flow direction is into the grid. From stations 8 and 9 only the nozzle throat diameter and the engine exhaust diameter are used when manipulating the computational grid. The nozzle adjustment is done in the same grid manipulation program that is used for adjusting the control surfaces. In most cases only the flight altitude and the free stream Mach number are needed for a good approximation of the engine mass flows.

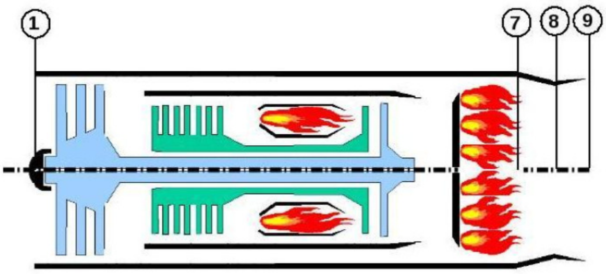


Figure 7: Engine station diagram.

3 Results

3.1 Geometry Comparison

Before the generation of the grid it was decided to compare the F/A-18 geometry used for the CFD simulations in Finland and Switzerland. Several differences were observed as can be seen in Fig. 3 which shows the two geometries. For example the canopy position was not the same, a difference in LEX fence height was observed and the position of the wing tip missile was different. Several other smaller differences were found. These geometrical differences were corrected so that the same geometry was used in Finland and Switzerland.

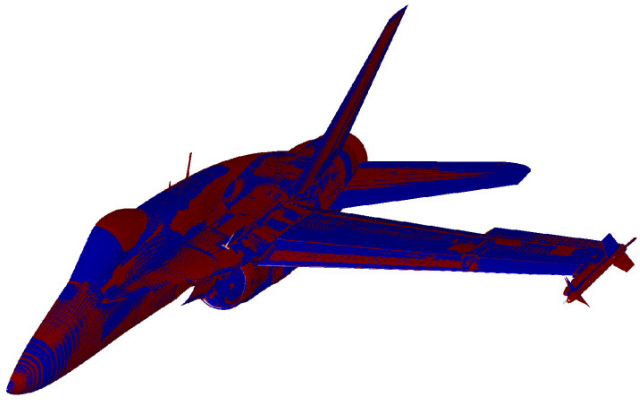


Figure 3: Red Swiss model, blue Finland model.

The Swiss F-18C CFD grid is fully structured while the Finnish grid contains overlapping blocks (Chimera). The volume grid resolution of the Swiss grid is better than the resolution of the Finnish grid. The nominal first cell height of the Swiss grid is smaller but the cell height stretching is stronger. The radius of the Swiss volume grid is about 250 m while the radius of the Finnish grid is about 500 m.

In the Finnish model also the Sidewinder is modeled using overlapping blocks which makes grid modification very easy.

Several other smaller differences were found. The use of overlapping blocks in the Finnish model requires small gaps between the main wing and the control surfaces. On the leading edge side the gap between the wing and the leading edge flaps is larger than on the real aircraft. On the trailing edge side the gap geometry in the Finnish model is more realistic than the closed geometry in the Swiss model.

However, also in the Finnish model the geometry of the gaps around the trailing edge flaps and shrouds is strongly simplified.

In the Finnish model the engine nozzle shape is adjusted according to the power setting. The difference in the after body modeling will lead to different pressure distributions thus complicating the comparison of after body loads. In general, all the geometry differences found are unimportant when the overall accuracy of numerical flow simulations is considered.

3.2 Flow Calculations & Comparison

3.2.1 Load Cases

The load cases were selected from an instrumented Finish aircraft of the MINIHOLM test campaign for structural loads evaluation.

AIM9M	Mach	Altitude	AoA
Yes	0.73	3150 ft	9.6°
No	0.44	8750 ft	25.3°

Table 2: Load cases studied.

The Swiss (CH: NSMB) CFD calculations employed the k-ω-SST and Spalart-Allmaras turbulence models whereas the Finnish (FI: FINFLO) CFD simulations used only the k-ω model. The following values for C_L (lift coefficient) were obtained:

Mach	AoA	Spalart CH	k-ω-SST CH	k-ω FI
0.73	9.6°	0.750	0.794	0.696
0.44	25.3°	1.331	1.352	1.336

Table 3: Computed lift coefficient C_L.

C_p plots were made to understand the differences, and these are shown in Figs. 9 and 10. It should be mentioned that the gaps between control surfaces are modeled in the Finnish CFD model, while they are closed in the Swiss model. Analyses of the results show a small difference in the position of the canopy shock. But larger differences can be observed on the wing. Fig. 9 shows that the results obtained using the Swiss approach show a larger low pressure region than the results obtained using the Finnish approach. This is probably due to the modeling of the gaps in the Finnish approach. Fig. 10 shows an opposite behavior (a larger low pressure region on the wing with the Finnish approach), but it should be kept in mind that the angle of attack as well as the trailing edge flap deflection angle are much larger for this case.

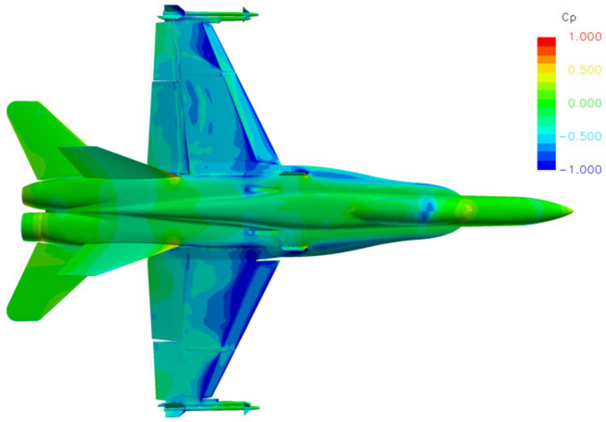


Figure 9: Top Finland, bottom Swiss load case with AIM-9M.

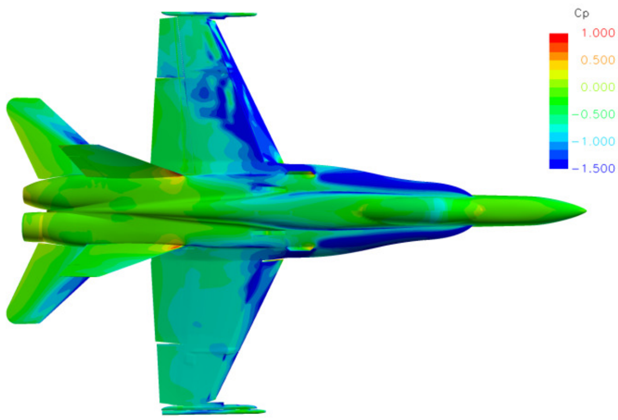


Figure 10: Top Finland, bottom Swiss load case without AIM-9M.

3.2.2 Sensitivity Analysis of CFD Calculations

All calculations discussed in the previous sections were made using the k-ω turbulence model. This model was developed for aerospace applications, and in general provides satisfactory results. For highly separated flows, the k-ω model, and in particular the Menter Shear Stress (MSS) variant has received much attention recently.

Due to the high angle of attack for the 8.25 g steady-state maneuver, large regions of unsteady and separated flow are present. For this reason the C_L convergence histories showed oscillations. One of these 8.25 g manoeuvre was calculated using the k-ω MSS model, and Table 4 summarizes the aerodynamic coefficients and the N_Z for the 2 computations. The differences are small, with the computation using the k-ω model yielding a slightly higher N_Z which is closer to the expected value.

Case	C_L	C_D	C_M	com Nz	exp Nz
Spalart	1.287	0.561	0.154	8.06	8.25
k-ω	1.294	0.563	0.170	8.11	8.25

Table 4: Aerodynamic coefficients for Spalart and k-ω turbulence models, 8.25g maneuver.

Small differences in the pressure contours (p – p∞) can be observed on the upper side of the horizontal stabilizer, on the vertical fin, and on the fuselage downstream of the wing attachment. On the lower side differences can only be observed on the horizontal stabilizer. In the plane at x = 16 m (the reference position of the vertical tail) large separated flow regions could be observed, and differences in computed results were apparent, see Fig. 11. However, it should be noted that the flow is unsteady, and differences may come not only from the turbulence model, but also from the unsteadiness of the flow.

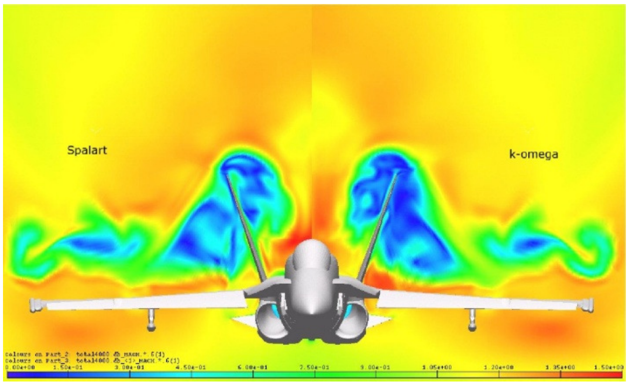


Figure 11: Comparison Mach contour plot of Spalart and k-ω-SST turbulence model.

Besides the influence of the turbulence model, the influence of the Mach number, of the angle of attack (AOA), and the deflection of all control surfaces on the F/A-18 were analyzed. The change of the angle of attack (AOA) was very remarkable because it affects the lift of the aircraft. A difference of only 1° in angle of attack may change the C_L value by 20%. The same chance in the deflection angle of the control surfaces showed only a small influence on the aerodynamic coefficient C_L, C_D, and C_M.

The influence of a change of the Mach number in the order of 0.02 showed for the aerodynamic coefficients a very small impact of 2% which is within the order of the accuracy of the CFD computation.

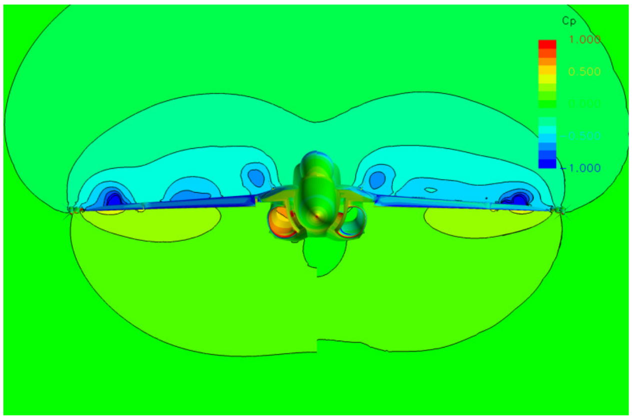


Figure 12: Pressure coefficient obtained using the FINFLO solver on different grids (Swiss grid on the left, Finnish grid on the right), Mach=0.73, x=12.5m.

The Swiss grid and the Finnish grid were processed with the FINFLO solver to study the differences. In general a fairly good agreement between CFD results concerning the component loads and even differences in the flow field were observed, see Fig. 12.

3.2.3 Structural Component Loads

For the structural analysis the loads of the aircraft on the different components and the different sections are of prime interest. Figure 13 shows the sections, the reference axes and points where these loads are calculated for comparison. The aircraft was split into two sections on each wing, two sections on the fuselage, one section at the root of each control surface and a hinge axis for each flap.

In the Fig. 14 the partition of the aircraft surface used for the CFD calculations is represented. Every structural element has been taken into account, so that accurate quantitative comparisons between different simulations and between CFD simulation and flight testing can be done. Of course for the flight loads the inertia loads have to be subtracted before the comparison can be made.

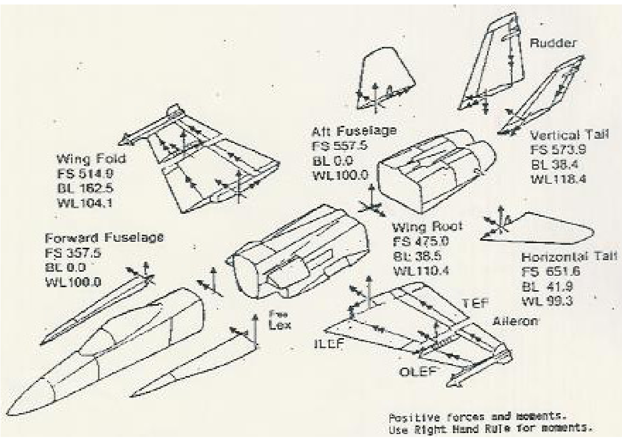


Figure 13: Reference locations and sign convention.

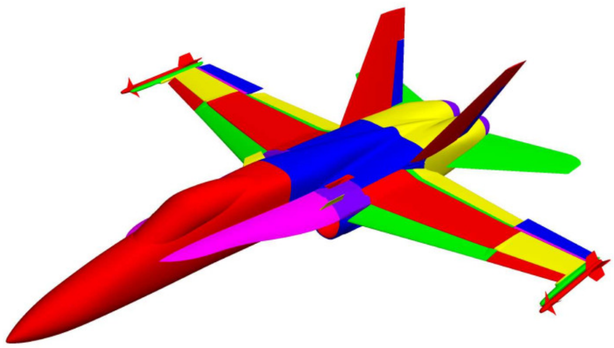


Figure 14: F-18 surface division into components.

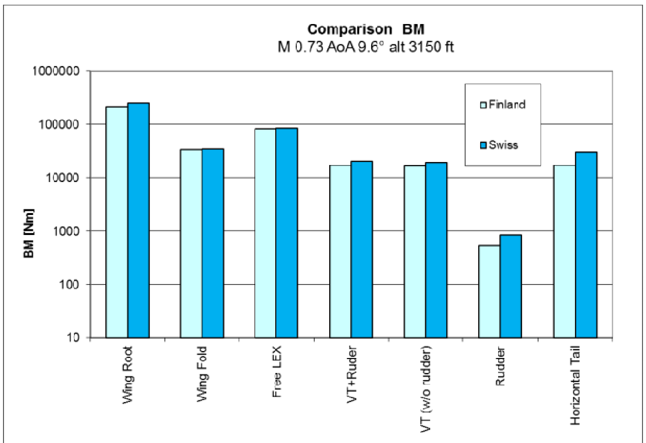


Figure 15: Bending moment on aircraft components.

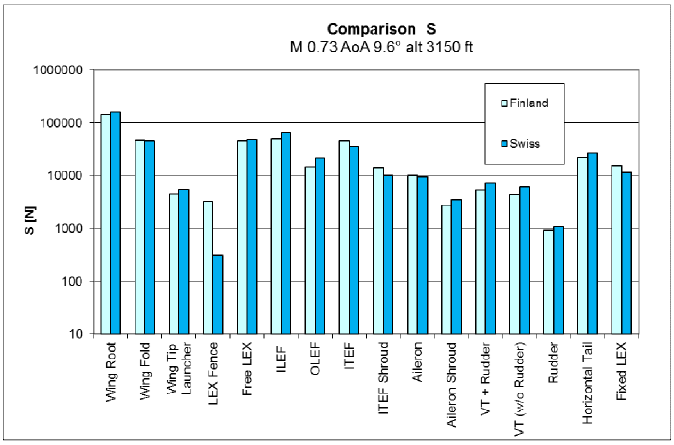


Figure 16: Shear forces on aircraft components.

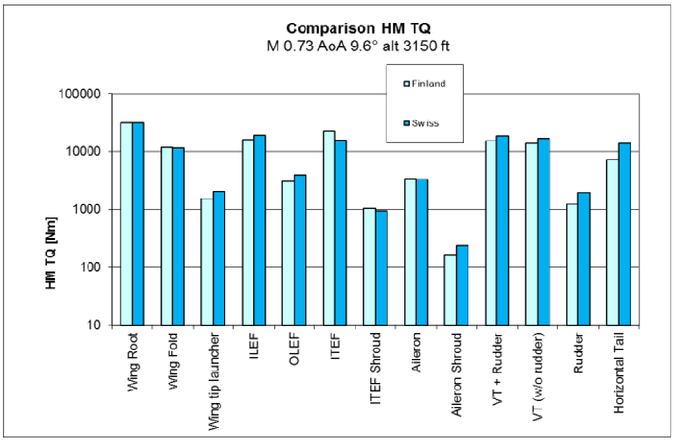


Figure 17: Hinge moment or torque on aircraft components.

Fig. 15 to 17 show the comparison between the Finnish (FINFLO) and Swiss (NSMB) grid/solver for the different component loads. A logarithmic scale on the absolute values is used in order to be able to put all the components in the same diagram.

Both simulations give bending moments in very good agreement for the wing, quite good for the vertical tail, but they show differences for the horizontal tail (40%). We also observe that Swiss (NSMB) bending moments and the corresponding shear forces are systematically higher than those of Finland (FINFLO), with the exception of the wing fold, where the shear is lower. This fact indicates that the center of pressure of the outer wing is more outboard in the Swiss CFD calculation.

For the vertical forces the agreement is also quite good to good. We observe that the big differences concern mostly the small components like the LEX fence, the rudder and the shroud, which are very sensitive to the local flow conditions and to details of the grid generation (see Fig. 18). The same remark holds for the comparison of hinge moment and torque.

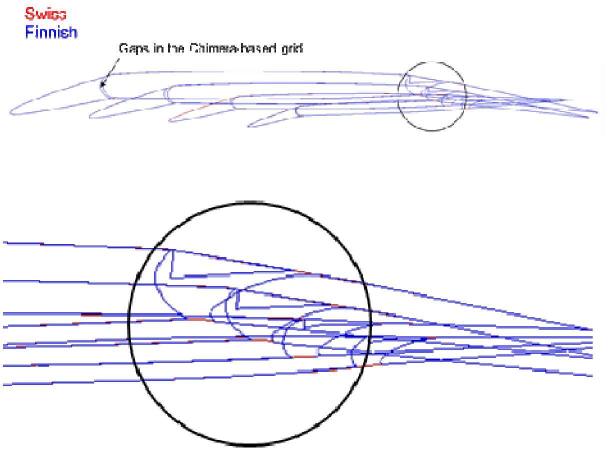


Figure 18: Geometry comparison in four wing sections

Other sources of differences worthwhile to mention are:

1) The position of the vortex generated by the free LEX and of the flow separation, which are detrimental for the forces of the components in the rear part of the aircraft, can be principally attributed to the higher angle of attack.
2) The use of different turbulence models and different flow solvers also has an influence on the results.

4 Conclusion & Outlook

In addition to the open cooperation, a spirit of healthy international competition has contributed to model and method improvements in both countries. Both sides have improved their models and developed their post-processing capabilities. Both sides have benefited from geometry and control surface hinge line comparisons. In the newest Finnish model the cabin (canopy) location is fixed according to the information received from Switzerland. Similarly, the Swiss side is improving the Sidewinder location and the LEX fence size.

The collaboration is very unique and provides a great opportunity to improve the complex CFD calculations on the F/A-18 aircraft.

Both countries will profit from this effort which is much more then just research in the field of advanced CFD calculations. The advantage provides a considerable improvement in grid strategies and numerical simulation for accurate maneuver loads prediction including unsteady flows.

The aim of the cooperation is a situation where all numerical simulations done in either country could be used by both sides.

Acknowledgement

The Finnish work was funded as part of Finnish Air Force Material Command research program on aircraft structures.

The Swiss activities were partly funded by the armasuisse for the support of the F/A-18 aerodynamic know how.

References

[1] Vos J. B., Rizzi A. W., Corjon A., Chaput E., and Soinne E.; *Recent Advances in Aerodynamics inside the NSMB (Navier Stokes Multi Block) Consortium.* AIAA 36th Aerospace Sciences Meeting, January 1998.

[2] Vos J. B., Rizzi A. W., Darracq D., and Hirschel E. H., *Computational Aerodynamics for Industrial Airframe Design using Navier Stokes Solvers.* Progress in Aerospace Sciences, Vol. 38, 2002, pp. 601-697.

[3] Siikonen, T., Rautaheimo, P. and Salminen, E., *Numerical Techniques for Complex Aeronautical Flows.* European Congress on Computational Methods in Applied Sciences and Engineering, Barcelona, Spain, Sept. 11-14, 2000.

[4] Guillaume M., Gehri A., Mandanis G., Vos J. B., *Calculation of F/A-18 Fatigue Loads and Wing Deformations using Computational Fluid Dynamics.* Proceedings 2006-10.5.2, ICAS Hamburg, Sept

2006Roe, P.L., *Approximate Riemannn Solvers, Parameter Vectors, and Differences Schemes.* Journal of Computational Physics, Vol. 43, No. 2, pp. 357-372, 1981.

[5] Lombard, C.K., Bardina, J.; Venkatapathy, E. and Oliger, J., *Multi-Dimensional Formulation of CSCM – An Upwind Flux Difference Eigenvector Split Method for Compressible Navier Stokes Equations.* AIAA Paper 83-1895-CP, 6th AIAA Computational Fluid Dynamics Conference, Danvers, Massachusetts, pp. 649-664, 1983.

[6] Chien, K.-Y., *Predictions of Channel and Boundary-Layer Flows with a Low-Reynolds-Number Turbulence Model.* AIAA Journal, Vol. 20, January 1982.

[7] Menter F.R., *Zonal Two Equation k- ω Turbulence Models for Aerodynamic Flows.* 24th Fluid Dynamics Conference, Orlando, July, 1993.

[8] Menter F.R., Two Equation Eddy-Viscosity Turbulence Models for Engineering Applications. AIAA Journal, Vol. 32, August 1994.

[9] Roe, P.L., *Approximate Riemannn Solvers, Parameter Vectors, and Differences Schemes.* Journal of Computational Physics, Vol. 43, No. 2, pp. 357-372, 1981.

Copyright Statement