

Comparison of Measured and Block Structured Simulation Results for the F-16XL Aircraft

O. J. Boelens*

National Aerospace Laboratory, 1006 BM Amsterdam, The Netherlands

K. J. Badcock†

University of Liverpool, Liverpool, England L69 7BZ, United Kingdom

A. Elmilgui‡

Analytical Services and Materials, Inc., Hampton, Virginia 23666

K. S. Abdol-Hamid§

NASA Langley Research Center, Hampton, Virginia 23681

and

S. J. Massey¶

Eagle Aeronautics, Hampton, Virginia 23666

DOI: 10.2514/1.35064

This paper presents a comparison of the predictions of three Reynolds-averaged Navier–Stokes codes for flight conditions of the F-16XL aircraft that feature vortical flow. The three codes, ENSOLV, parallel multiblock, and propulsion aerodynamics branch 3-D unsteady Reynolds-averaged Navier–Stokes, solve on structured multiblock grids. Flight data for comparison were available in the form of surface pressures, skin friction, boundary-layer data, and photographs of tufts. The three codes provided predictions that were consistent with expectations based on the turbulence modelling used, which was k - ϵ , k - ω with vortex corrections, and an algebraic stress model. The agreement with flight data was good, with the exception of the outer wing primary vortex strength. The confidence in the application of the computational fluid dynamics codes to complex fighter configurations increased significantly through this study.

Nomenclature

C_f	=	skin friction coefficient, $ \tau_{nt} /(1/2\rho_\infty u_\infty^2)$
C_p	=	pressure coefficient, $(p - p_\infty)/(1/2\rho_\infty u_\infty^2)$
h	=	airplane altitude, ft
k	=	turbulent kinetic energy, ft^2/s^2
L_{ref}	=	reference length, in
M	=	Mach number
P_k	=	production of turbulent kinetic energy
P_k^u	=	unlimited production of turbulent kinetic energy
p	=	pressure, psia
p_t	=	total pressure, $p(1 + 1/2(\gamma - 1)M^2)^{\gamma/(\gamma-1)}$, psia
p_∞	=	freestream pressure, psia
r	=	ratio of the magnitude of the rate-of-strain and vorticity tensors
Re, R_n	=	Reynolds number
S_{ij}	=	mean flow strain rate, s^{-1}
T	=	temperature, °R
T_t	=	total temperature, $T(1 + 1/2(\gamma - 1)M^2)$, °R
u	=	velocity vector, ft/s

u_∞	=	freestream velocity, ft/s
V/V_{re}	=	ratio of the velocity magnitude in the boundary layer to that at the rake extreme total-pressure tube
x/c	=	fractional distance along the local chord, positive aft
y/s	=	fractional distance along the local span, positive outward
y^+	=	Re -like term for the flat plate turbulent boundary layer
z	=	normal distance above the surface at a rake location, mm
α	=	angle of attack, deg
β	=	sideslip angle, deg
γ	=	specific heat ratio, 1.4
ρ_∞	=	freestream density, slugs/ft^3
τ_{nt}	=	tangential component of the total stress tensor, psia
Ω_{ij}	=	vorticity, s^{-1}
ω	=	specific turbulent dissipation rate, $1/\text{s}$

I. Introduction

THE application of computational fluid dynamics (CFD) methods to support the operation of fighter aircraft requires realism and cost effectiveness compared with alternatives such as wind-tunnel tests or flight tests. Two envisaged applications of CFD are for the assessment of stability and control characteristics and the calculation of changes in aircraft loads due to new store configurations. By using CFD, the number of flight conditions in a flight-test certification program can be optimized, and potentially dangerous cases can be identified beforehand. To enable the application of CFD methods for such purposes, the methods should be well validated and evaluated against state-of-the-art wind-tunnel and/or flight-test data.

The Cranked-Arrow Wing Aerodynamics Project (CAWAP) [1] provided the CFD community with an excellent database for validation and evaluation purposes. This project focused on the understanding of flow phenomena encountered on the cranked-arrow wing of the F-16XL aircraft. The CAWAP database contains both

Received 9 October 2007; revision received 25 March 2008; accepted for publication 25 March 2008. Copyright © 2008 by National Aerospace Laboratory NLR, The Netherlands. Published by the American Institute of Aeronautics and Astronautics, Inc., with permission. Copies of this paper may be made for personal or internal use, on condition that the copier pay the \$10.00 per-copy fee to the Copyright Clearance Center, Inc., 222 Rosewood Drive, Danvers, MA 01923; include the code 0021-8669/09 \$10.00 in correspondence with the CCC.

*Research and Development Engineer, Applied Computational Fluid Dynamics, Department of Flight Physics and Loads, Aerospace Vehicles Division, Post Office Box 90502; boelens@nlr.nl.

†Professor, Computational Fluid Dynamics Laboratory, Department of Engineering; K.J.Badcock@liverpool.ac.uk.

‡Senior Research Scientist. Member AIAA.

§Aerospace Engineer, Configuration Aerodynamics Branch. Associate Fellow AIAA.

¶Senior Research Scientist. Member AIAA.

subsonic and transonic data at flight Reynolds numbers. The data obtained during the flight tests comprised surface pressure measurements along both the butt line (BL) and fuselage stations, boundary-layer measurements at four positions on the left wing, skin friction measurements at the FS330 station on the left wing, and surface flow visualizations using tufts.

Initiated by NASA, the Cranked-Arrow Wing Aerodynamics Project International (CAWAPI) [2–6] was started as a follow-on project. Along with the Vortex Flow Experiment 2 (VFE-2) [7], this project was incorporated under the NATO Research and Technology Organization (RTO) task group Applied Vehicle Technology (AVT) 113. This paper reports on the contributions to CAWAPI made by the National Aerospace Laboratory (NLR), the University of Liverpool, and NASA Langley, using structured multiblock flow solvers.

This paper continues with a summary of the flow solvers used. The generation of a common grid is summarized and then results are compared for several flight conditions that feature vortical flow. Finally, the performance of the solvers is evaluated and conclusions are drawn.

II. Flow Solvers

A. National Aerospace Laboratory Solver (ENSOLV)

The flow solver ENSOLV, which is part of NLR's flow simulation system ENFLOW [8], is capable of solving the Euler and Navier–Stokes equations on multiblock structured grids for arbitrary configurations. The configuration can be either fixed or moving relative to an inertial reference frame and can be either rigid or flexible. The equations in full conservation form are discretized in space by a second-order accurate, cell-centered, finite volume method, central differences, and matrix artificial diffusion. The artificial diffusion consists of a blending of second- and fourth-order differences with a Jameson-type shock sensor for the basic flow equations and a total variation diminishing (TVD) discontinuity sensor for the turbulence model equations. For steady flow simulations, the discretized time-dependent system of equations is integrated toward the steady-state solution using a five-stage explicit Runge–Kutta scheme. Local time stepping, implicit residual averaging, and multigrid acceleration techniques are applied.

B. Liverpool Solver (Parallel Multiblock)

The parallel multiblock (PMB) solver [9] has been under development at the Universities of Glasgow and Liverpool during the past decade. The Euler and Reynolds-averaged Navier–Stokes (RANS) equations are discretized on curvilinear, multiblock, body-conforming grids using a cell-centered finite volume method that converts the partial differential equations into a set of ordinary differential equations. The convective terms are discretized using Osher's upwind method. A monotone upwind scheme for conservation laws variable extrapolation is used to provide second-order accuracy with the Van Albada limiter to prevent spurious oscillations around shock waves. Following Jameson, the spatial residual is modified by adding a second-order discretization of the real time derivative to obtain a modified steady-state problem for the flow solution at the next real time step, which is solved through pseudotime. This pseudotime problem is solved using an unfactored implicit method based on an approximate linearization of the residual. The linear system is solved in unfactored form using a Krylov subspace method with block incomplete upper lower preconditioning. The preconditioner is decoupled between blocks to allow a high efficiency on parallel computers with little detriment to the convergence of the linear solver. For the Jacobian matrix of the CFD residual function, approximations are made that reduce the size and improve the conditioning of the linear system without compromising the stability of the time marching.

C. Propulsion Aerodynamics Branch 3-D Unsteady Reynolds-averaged Navier–Stokes Solver

The propulsion aerodynamics branch 3-D unsteady Reynolds-averaged Navier–Stokes (PAB3D) CFD code [10] is a structured,

multiblock, parallel, implicit, finite volume solver of the three-dimensional RANS equations. Advanced turbulence models are available in the code. Viscous models include coupled and uncoupled Navier–Stokes and thin layer Navier–Stokes options. Roe's upwind scheme is used to evaluate the explicit part of the governing equations, and van Leer's scheme is used for the implicit part. Diffusion terms are centrally differenced. PAB3D uses either a two- or three-factor numerical scheme to solve the governing equations. For unsteady calculations, PAB3D is second-order in time with subiterations. For steady-state calculations, local time stepping and grid sequencing are applied to accelerate convergence. PAB3D is widely used for internal and external flow applications by NASA and by the U.S. aerospace industry. PAB3D has several built-in time saving routines, including grid sequencing and distributed computer memory requirements, that permit the user to quickly obtain a converged solution. There are several state-of-the-art two-equation and algebraic Reynolds stress turbulence models implemented. In an attempt to increase the fidelity and accuracy, multiscale-type (hybrid) turbulence models such as unsteady Reynolds-averaged Navier–Stokes/large eddy simulation (URANS/LES) [11] and partially averaged Navier–Stokes (PANS) [12] have been added to the code. PAB3D has been well tested and documented for the simulation of aeropropulsive and aerodynamic flows involving separation, mixing, and other complicated phenomena. PAB3D is ported to a number of platforms and offers a combination of good performance and low memory requirements. In addition to its advanced preprocessor, which can handle complex geometries through multiblock general patching, PAB3D has a runtime module capable of calculating aerodynamic performance on the fly as well as a postprocessor used for follow-on data analysis.

For this study, PAB3D used a third-order monotone upwind scheme for conservation laws variable extrapolation of the fluxes with the minimal model (MINMOD) limiter and the upwind flux difference splitting of Roe for evaluating the convective terms. Viscous diffusion terms were modeled as uncoupled in the flow direction and fully coupled in the crossflow direction. A three-factor scheme was used for the approximation of implicit terms. Local time stepping and grid sequencing were applied to accelerate convergence.

D. Turbulence Models

1. Basic Models

The test cases that were computed for this paper feature the flow over the F-16XL aircraft, with extremely complex geometry, at flight Reynolds numbers. The flows themselves are complex, with multiple vortices present. The complex geometry provides a grid generation challenge, which is considered in Sec. III. The complex flow field provides a challenge to the turbulence modeling. In the current section, the options used for this modeling are summarized. All calculations were made assuming fully turbulent flow.

The basic Boussinesq models were of the two-equation type. For ENSOLV, the turbulent nonturbulent (TNT) k – ω model, which is a variant of the Wilcox k – ω model, is employed. The equations of the model are slightly modified by the introduction of a “cross diffusion” term [13]. This modification has been introduced to resolve the dependency of the freestream value of ω . In addition, to remove the singular behavior of ω at solid boundaries, the equations of the k – ω model are reformulated such that, instead of ω , the quantity $\tau = 1/(\omega + \omega_0)$ is used. Here, ω_0 is a positive constant (default value $\omega_0 L_{\text{ref}}/u_\infty = 20$, where u_∞ is the freestream velocity and L_{ref} is the reference length). At the solid boundaries, both k and τ are set to zero. To prevent unphysically high values of k near stagnation points, the production term in the k equation has been limited to a maximum of 20 times the dissipation term in the k equation. At the “inflow” parts of the far-field boundary, the freestream values of the turbulent variables are computed from the freestream turbulent Reynolds number (0.01 in the present simulations) and the freestream dimensionless turbulent kinetic energy ($k/u_\infty^2 = 10^{-6}$ in the present simulations). For PMB, the standard k – ω model was used. Finally, for PAB3D the standard k – ε model was used. In addition, the

PAB3D code also presented results using the Shih–Zhu–Lumley (SZL) algebraic stress model [14], in which the stress equation is given by

$$-\overline{u_j u_i} = -\frac{2}{3}\delta_{ji}k + 2C_\mu(k^2/\varepsilon)S_{ji} - 2\beta(K^3/\varepsilon^2)(\Omega_{ik}\bar{S}_{kj} - \bar{S}_{ik}\Omega_{kj})$$

where

$$C_\mu = \left(A_o + A_s^* \frac{U^* k}{\varepsilon}\right)^{-1}, \quad \Omega_{ij} = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i} \right)$$

$$\bar{S}_{ij} = S_{ij} - \frac{1}{3} S_{kk} \delta_{ij}, \quad S_{ij} = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)$$

Note the higher-order nonlinear terms involving the deviatoric mean flow strain rate (\bar{S}_{ij}) and the vorticity (Ω_{ij}) tensors in the stress equation. In the C_μ equation, $A_o = 6.5$ and A_s^* , and U^* involve tensor products of S_{ij} and Ω_{ij} and their variants. Algebraic stress models give inherently better results than the linear stress model because of the explicit modeling of effects such as relaxation and the specific inclusion of nonlinear anisotropic effects from the mean flow strain and vortices.

2. Enhanced Vortex Correction Model

Boussinesq models overpredict the eddy viscosity within the vortex core, leading to an exaggerated diffusion of vorticity. As a consequence, the details of the vortex core are lost, and low suction peaks with wide vortex bases are a characteristic of the solution. For this reason, an enhanced vortex correction model [15,16] is used that controls the production of turbulent kinetic energy and, hence, eddy viscosity through either an increase in the production of dissipation (ω) or a decrease in the production of turbulent kinetic energy within regions of highly rotational flow. To illustrate, in the second variant, the production of turbulent kinetic energy k is limited as

$$P_k = \min\{P_k^u, (2.0 + 2.0 \times \min\{0, r - 1\})\rho\beta^*k\omega\}$$

Here P_k^u is the unlimited production of k , and r is the ratio of the magnitude of the rate-of-strain and vorticity tensors. This approach has proven to be effective in producing surface pressure profiles on simple delta wings in good agreement with those of the experimental data [15,16]. The results presented in this paper for the ENSOLV solver used the ω -enhanced version of this method based on the TNT k - ω model already described. The PMB results used the k -limited variant based on the k - ω model.

3. Summary

In summary, the ENSOLV results have been obtained using the TNT k - ω model with a ω enhancer vortex correction. The PMB calculations used the standard k - ω model with a k -limited vortex correction. PAB3D used the k - ε model and the SZL algebraic Reynolds stress model.

III. Grid Generation

A common grid for the block structured solvers was generated by NLR. The approach to grid generation is described in [3] and is not repeated here. Two small modifications to the surface description were made to further facilitate the generation of a structured grid.

First, a small “step” or “plate” on the wing upper surface was removed. Second, the end part of the vertical tail base was slightly rounded off. Apart from these changes, the original definition of the geometry as given in the CAD model was respected. The final grid was produced in less than 4 weeks, including some development of the grid generation tools. This time compared favorably with that required to produce the unstructured grids in CAWAPI.

The following family of structured grids has been used: 1) the baseline structured grid around the half-span full-scale model of the F-16XL consisting of 1903 blocks, 14,750,720 grid cells, and 17,014,119 grid points; 2) the baseline structured grid with a reduced number of blocks, in which the merging step resulted in a reduction of the number of blocks from 1903 to 216, and 3) a structured grid around the full-scale model of the F-16XL consisting of 3806 blocks, 29,501,440 grid cells, and 34,028,238 grid points. This final grid has been generated by mirroring the baseline structured grid around the half-span full-scale model of the F-16XL with respect to the symmetry plane.

The upper surface grid and the resulting y^+ distribution over the upper surface are shown in [3], Fig. 7. From this figure, it is evident that the grid spacing normal to the surface has a desired value of y^+ of less than one (based on an ENSOLV solution), except for the regions below the vortical structures.

IV. Results for Vortical Flow Flight Conditions

A. Flight Test Cases

Initially, four flight conditions were selected for computation in CAWAPI, of which three featured vortical flow. Later, three additional flight conditions were added to the original set. These additional flight conditions also featured vortical flow, with flight condition 25 at a higher incidence and the other two conditions (flight conditions 50 and 51) exhibiting sideslip. Only the vortical flow conditions are of interest in the current paper and these are summarized in Table 1. Conditions 7, 19, 25, and 46 have zero sideslip and were computed by all three codes using the baseline structured grid around the half-span full-scale model of the F-16XL. Conditions 50 and 51, which have nonzero sideslip, were computed using ENSOLV and PAB3D only using the structured grid around the full-scale model of the F-16XL. Results for all flight conditions can be found in [6]. In the current paper, results for flight conditions 7, 19, and 46 were evaluated in detail.

The engine flow conditions were specified in CAWAPI, and the engine face and jet boundary conditions, based on this data, were used in the ENSOLV and PAB3D solutions. In ENSOLV, at the inlet duct exit plane (engine inlet) a prescribed normalized static pressure p/p_∞ was used, whereas at the mixing plane (engine exit) a boundary condition with a prescribed normalized total pressure $p_t/p_{t,\infty}$ and total temperature $T_t/T_{t,\infty}$ was applied. For PAB3D, constant total values were specified for the nozzle inlet, and a constant pressure boundary condition was specified at the nozzle exit. Preliminary calculations using ENSOLV and all the PMB calculations used a simple flow through condition, in which the engine face and jet plane were treated as far-field boundaries. This treatment did not noticeably influence the predictions of the vortical flow on the upper surface of the wing; therefore, no further discussion of the engine boundary conditions is made in this paper.

Table 1 CAWAPI flight conditions

Flight condition	Nominal					Actual				
	α , deg	β , deg	h, ft	M	$10^6 R_n/\text{ft}$	α , deg	β , deg	M	Re , $\times 10^6$	
7	13	0	5000	0.29	1.79	11.89	−0.133	0.304	44.40	
19	13	0	10,000	0.32	1.71	11.85	0.612	0.360	46.80	
25	20	0	10,000	0.24	1.28	19.84	0.725	0.242	32.22	
46	10	0	24,000	0.51	1.77	10.40	0.310	0.527	46.90	
50	13	5	24,000	0.42	1.46	13.56	5.310	0.434	38.41	
51	13	−5	24,000	0.42	1.46	12.89	−4.580	0.441	38.95	

B. Overview of Flow Topology

Before looking at the results in detail, the large-scale vortical flow structure above the wing is discussed. Figure 1 shows an isosurface of the vorticity magnitude for flight condition 7. The flow structure consists of several vortices: 1) the inner wing primary vortex originating from the wing leading edge inboard of the crank, 2) the outer wing vortex originating from the wing leading edge outboard of the crank, 3) the air dam vortex originating from the air dam at wing upper surface, 4) the missile vortices originating from the missile fins, and 5) the fuselage vortex. In addition to these vortices, other vortical structures, such as the inner wing secondary vortex, are present. All flight conditions characterized by vortical flow exhibit a similar vortical flow structure, although the strength and location of the vortices may differ.

C. Comparison with Flight Data

The predictions are now compared against available flight data. Note that the common flight conditions chosen in CAWAPI did not match exactly the conditions of the flight data and, in particular, the angle of incidence is lower in the computations than the measurements for the comparisons shown here. This is likely to have the effect of making the primary vortices weaker in the computational results, with lower suction in the footprint on the wing. The computational results were available from the vortex correction variants of the $k-\omega$ models in ENSOLV and PMB and from the $k-\omega$ and ASM models in PAB3D.

The comparison for the surface pressure coefficient is shown for flight condition 7 in Fig. 2 for three butt lines, BL80, BL95, and BL153.5, going progressively outboard. The general agreement for the two BLs inboard of the crank is good. The ENSOLV and PAB3D ASM predictions show similar suction levels for the primary vortex. The PMB and PAB3D $k-\varepsilon$ predictions have weaker vortices, with the PMB solution showing a more compact vortex footprint. The comparisons at BL80, showing the lack of a flat region in the C_p distribution close to the leading edge, indicate that the strength of the secondary inboard vortex is underpredicted. The secondary vortex appears to have developed more fully by BL90. The comparisons outboard of the crank, at BL153.5, are less favorable. The suction in the primary outboard vortex is significantly less than the measurements in all predictions.

The boundary-layer profiles are compared in Fig. 3 for flight condition 7. Three inboard rake locations are used, with one inboard of the primary vortex, the second underneath the primary vortex, and the third in the region of the secondary vortex. The general agreement with flight measurements is again good. For the rake closest to the leading edge, the ASM model and the vortex correction models give a fuller profile than the measurements and the $k-\varepsilon$ predictions.

The comparison of the skin friction values along fuselage station FS330 for flight condition 19 is shown in Fig. 4. The PMB prediction,

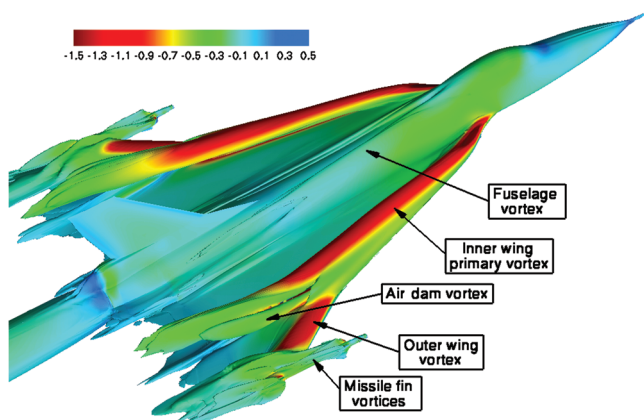


Fig. 1 Isosurface (level equals 250 Hz) of vorticity magnitude for flight condition 7 ($\alpha = 11.89^\circ$, $M = 0.304$, and $Re = 44.40 \times 10^6$). The vortices are colored by the pressure coefficient C_p .

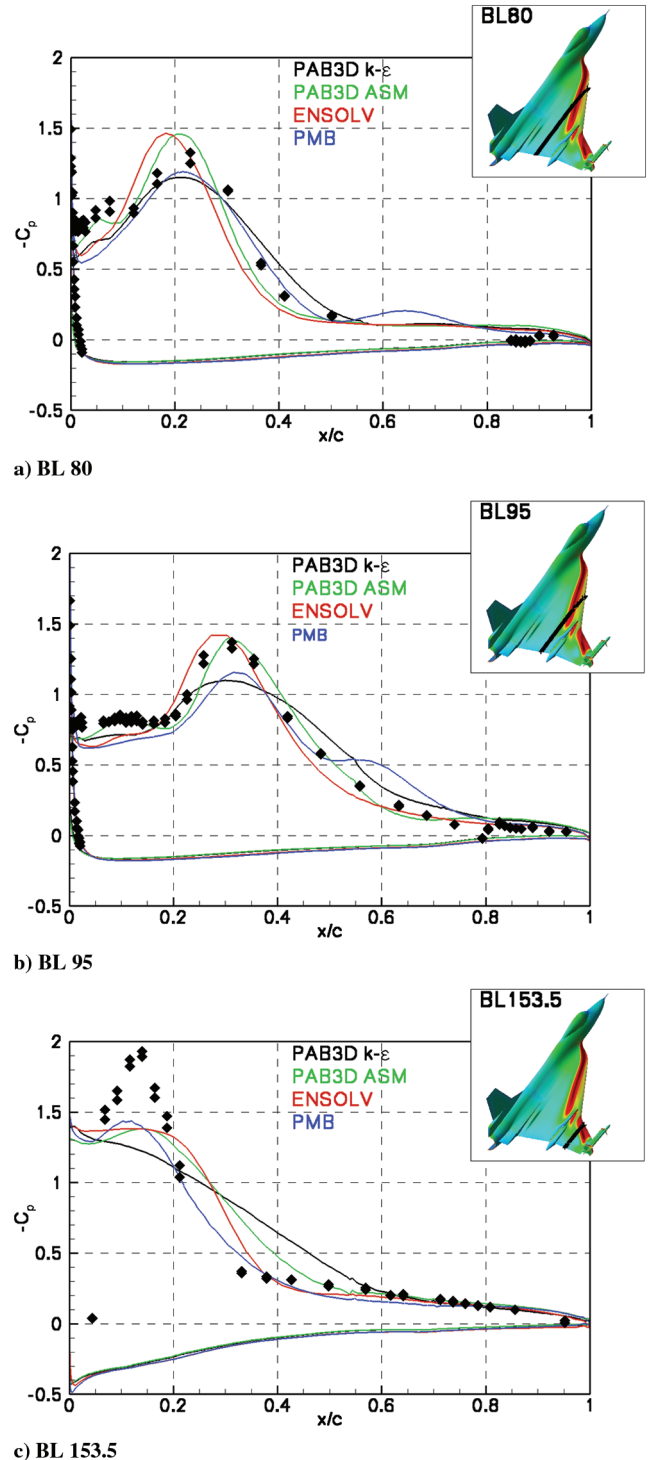


Fig. 2 Surface pressure coefficient for flight condition 7 ($\alpha = 11.89^\circ$, $M = 0.304$, and $Re = 44.40 \times 10^6$) and flight condition 19 ($\alpha = 11.85^\circ$, $M = 0.360$, and $Re = 46.80 \times 10^6$) along the butt lines: a) BL80, b) BL95, and c) BL153.5.

which has not been extracted, is not included. The comparisons show that the levels under the primary vortex agree well with the measurements, with the PAB3D predictions lower than those from ENSOLV. Again, there is more discrepancy under the secondary vortex; the levels from ENSOLV are too high, whereas those from PAB3D are more in line with the measurements. The $k-\varepsilon$ predictions show the wrong shape in this region but the ASM predictions are closest to the measurements.

Finally, the comparison between surface streamlines from the ENSOLV solution and tufts from the flight tests is shown in Fig. 5 for

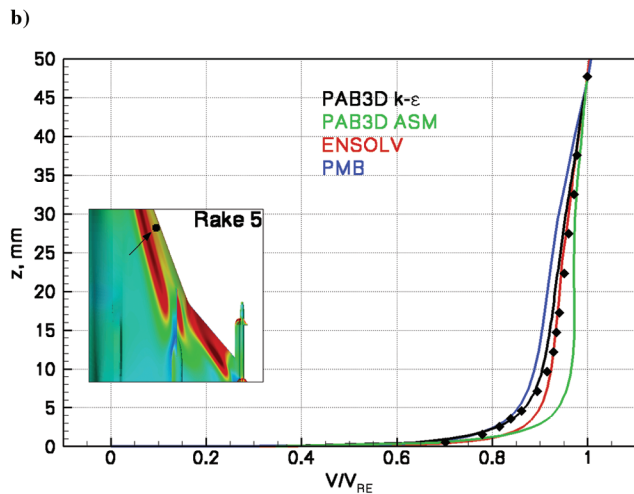
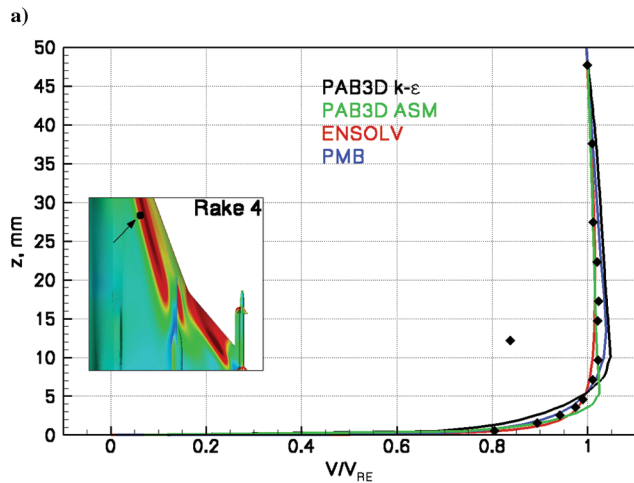
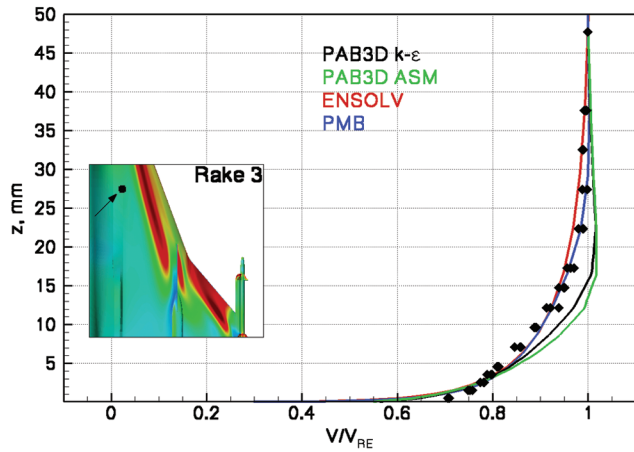


Fig. 3 Boundary-layer profiles for flight condition 7 ($\alpha = 11.89^\circ$, $M = 0.304$, and $Re = 44.40 \times 10^6$).

flight condition 7. The black dots visible are calibration targets. Inboard of the crank, the agreement in the flow direction as indicated by the tufts and surface streamlines is good. The reattachment line of the inner wing primary vortex and the separation line of the inner wing secondary vortex are clearly visible. The agreement in the flow direction outboard of the crank is less good. It is noted that here the tufts have a blurred character, indicating local unsteadiness of the flow.

D. Evaluation

The comparisons with flight data indicate that the CFD predictions are generally good. However, some flow details are not well

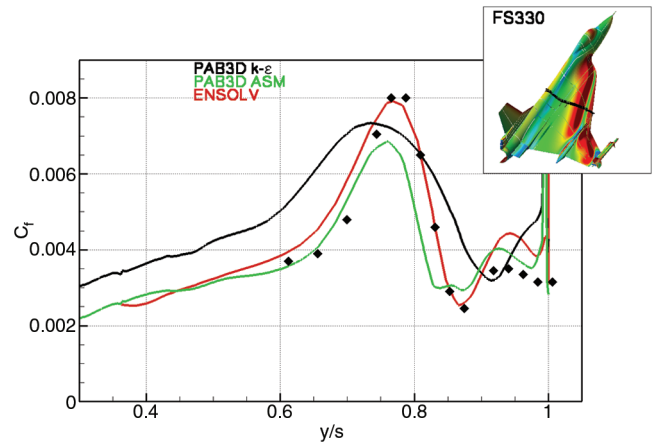


Fig. 4 Local skin friction coefficient for flight condition 19 ($\alpha = 11.85^\circ$, $M = 0.360$, and $Re = 46.80 \times 10^6$) at FS330.

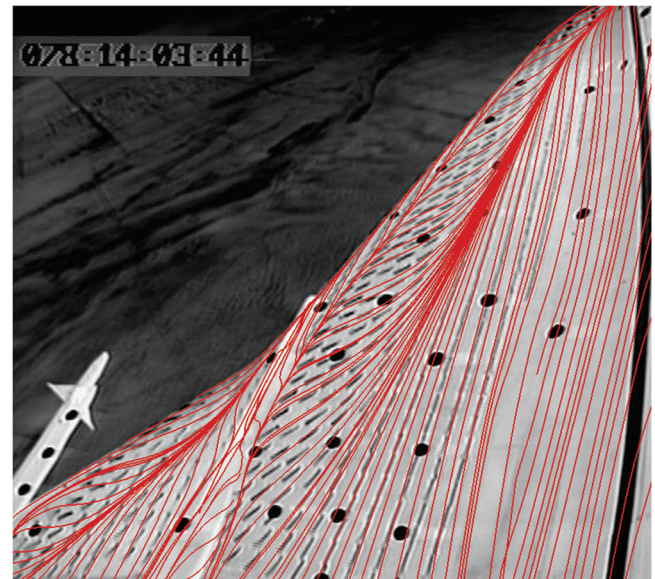


Fig. 5 Surface streamlines superimposed on the negative of the tuft image for flight condition 7 ($\alpha = 11.89^\circ$, $M = 0.304$, and $Re = 44.40 \times 10^6$), flight 145, run 16b, video (1996), 078:14:03:44 (day: hour:minute:second).

represented and these are considered in the current section. It is observed that the wing leading-edge geometry and the high Reynolds number are both helpful to the CFD in the sense that the prediction of the initial flow separation around the leading edge is made easier. First, the geometry toward the fuselage S blend contains a flat strip at the section leading edge, which has the effect of producing a sharp corner. This corner fixes the separation point and so the problem of predicting a smooth body separation, which is a challenge for CFD, is not present, at least when the inner wing primary vortex is becoming established. The leading edge at the crank is very sharp, meaning the same situation holds at the formation of the primary outboard vortex. Further, the high Reynolds number means that the leading-edge flow is likely to be fully turbulent, avoiding any influence of transition. Flow separation in all solutions was observed to be right at the section leading edge, both inboard and outboard of the crank, as indicated in Fig. 6.

The upper wing flow is very complex with a number of separations and reattachments, as illustrated in Fig. 6. The surface streamline pattern in this figure is composed of the following main elements:

1) *The primary separation on the inner and outer wing.* The flow separates on the wing leading edge. On the part of the wing inboard of the crank this separation results in the inner wing primary vortex,

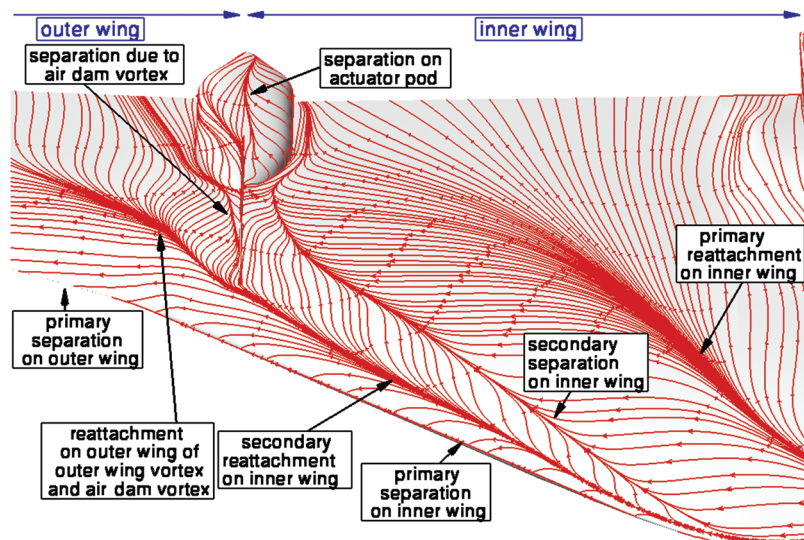


Fig. 6 Overview of the surface streamline pattern for flight condition 7 ($\alpha = 11.89$ deg, $M = 0.304$, and $Re = 44.40 \times 10^6$).

whereas on the part of the wing out board of the crank the outer wing vortex is formed.

2) *The primary reattachment on the inner wing.* At this line, the vortical flow associated with the inner wing primary vortex reattaches on the wing surface. Note that for the present case the inner wing primary vortex lifts off of the surface, resulting in a fanning out of the reattachment line.

3) *The secondary separation on the inner wing.* Underneath the inner wing primary vortex, the flow separates resulting in the inner wing secondary vortex. This separation line starts some distance downstream of the apex of the wing and is present up to the wing trailing edge.

4) *The secondary reattachment on the inner wing.* At this line, the vortical flow associated with the inner wing secondary vortex reattaches to the wing surface. This reattachment line extends to the leading edge of the air dam.

5) *The reattachment on the inner wing side of the air dam.* After reaching the air dam, the vortical flow resulting from the inner wing secondary vortex reattaches to the inner wing side of this air dam. Note that this reattachment line is a continuation of the secondary reattachment line on the inner wing. When the air dam changes to the actuator pod, this reattachment line ceases to exist, and the vortical flow associated with the inner wing secondary vortex reattaches in the junction between the actuator pod and the wing.

6) *The separation from the edge of the air dam.* The flow separates from the upper edge of the air dam. This separation results in the air dam vortex.

7) *The separation on the actuator pod.* The flow separates from the upper side of the actuator pod. Note that this separation starts before the intersection of the air dam and the actuator pod. The air dam vortex is fed further by the flow coming from this separation.

8) *The reattachment on the outer wing of the outer wing vortex and the air dam vortex.* At this line, the vortical flows associated with both the outer wing vortex and the air dam vortex reattach.

9) *The separation due to the air dam vortex.* Because of the air dam vortex, the flow separates just outboard of the air dam. This separation results in a small vortex in the outboard junction region between the air dam and the wing surface.

10) *The reattachment on the outer wing side of the air dam.* At this line, the vortical flow associated with the small vortex described in element 9 reattaches to the air dam.

The different turbulence options provided results that followed a pattern. The $k-\epsilon$ model predicted weaker vortices due to the excessive levels of turbulence produced. The ASM produced a stronger inner wing primary vortex, as shown in Fig. 7 for flight condition 46. The vortex correction models also fixed this problem, and realistic levels of suction under the primary inboard vortex were obtained with a simple correction to the underlying $k-\omega$ models.

Inboard of the crank, problems were noted with the strength of the inner wing secondary vortex. The grid in the region concerned was observed to be somewhat coarse, meaning that no conclusions could be drawn about the performance of the physical modelling there.

The most important discrepancy arising from the comparisons is the very weak primary vortex predicted outboard of the crank. The

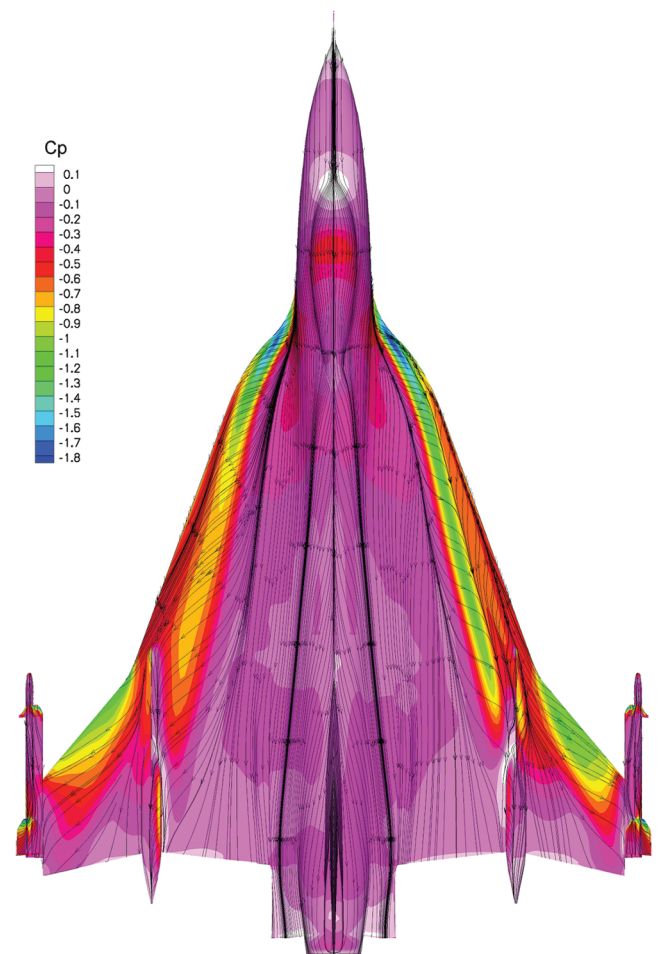


Fig. 7 Upper surface pressure distribution with streamlined flight condition 46 ($\alpha = 10.4$ deg, $M = 0.527$, and $Re = 46.9 \times 10^6$). The left side shows the PAB3D $k-\epsilon$ result and the right side shows the PAB3D ASM result.

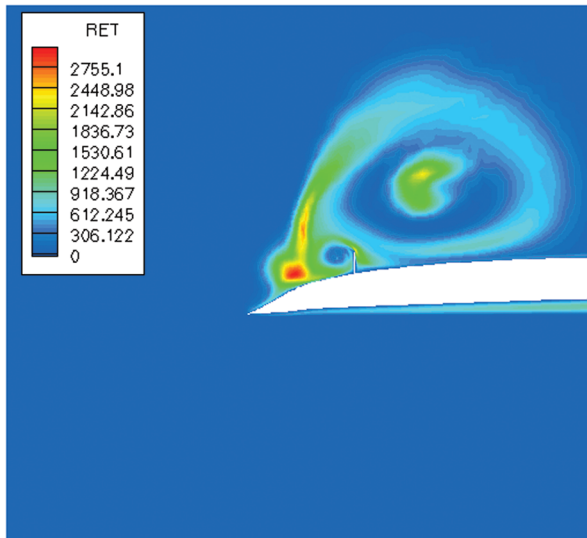


Fig. 8 Contours of turbulent Reynolds number for flight condition 7 ($\alpha = 11.89^\circ$, $M = 0.304$, and $Re = 44.40 \times 10^6$) on a spanwise slice just downstream of the crank. Here, the turbulent Reynolds number is the ratio of the eddy viscosity to the molecular viscosity.

flow in this region is very complex, featuring the primary and secondary inner wing vortices, the outboard primary and secondary vortices, and also vortices that have arisen from the interaction of the inboard vortices with the air dam. There are two main possibilities for the weakness of the predicted outboard primary vortex. First, the levels of turbulence predicted for the PMB solution are shown in Fig. 8. The turbulent Reynolds number plotted is defined as the ratio of the eddy viscosity to the molecular viscosity and is an indication of how much more viscous the flow is from the contribution of the turbulence model. Low levels of turbulence are indicated in blue and high levels in red. The influence of the vortex correction on the inner wing vortex and the vortex arising from the interaction with the air dam can be seen. However, the outer wing primary vortex is filled with red, indicating high levels of turbulence and explaining its weakness. The flow in this region is complex, with regions of shear between different vortices. The simple vortex correction appears to fail because the levels of turbulence in the outer wing vortex do not only depend on production terms, but also on the convection of turbulence. Interestingly, the more general ASM also fails to predict the outboard primary vortex strength close to the measurements, and this may be due to the coarse nature of the computational grid in the area of interest. To better resolve the outer primary vortex, one needs a finer grid along the full trajectory of the vortex core or one needs to use a grid adaption methodology that will cluster points in areas of high flow gradients.

The second consideration in this region is the likely presence of unsteady flow when several vortices are in close proximity. The evidence for this comes from the photographs of the tufts from the flight tests, which are blurred in this region. All the computations presented in this paper assumed steady-state flow.

E. Assessment of Solver Performance

The ENSOLV calculations used grid sequencing on three levels and a two-level full approximation storage multigrid scheme to solve on the two finer grid levels. The simulations were performed on two processors of NLR's NEC SX5/8B vector computer. Four orders of convergence were obtained for the root mean square norms. Larger block dimensions resulted in an increase of the vector length, and merging the blocks (see Sec. III) resulted in a reduction of the required computational time by a significant factor of around 4. The final calculation of a flight condition required around 16 h and 1200 fine grid multigrid cycles.

The PAB3D solution also used two levels of grid sequencing. The calculations were run on a cluster with 2.8 GHz processors. A typical

calculation on 56 processors gave a converged solution in 51 h and 10,000 fine grid iterations.

The PMB calculations were made on the Liverpool University CFD Laboratory cluster, which has 192 processors with a speed of 3.4 GHz. The calculations required around 2 days on 96 processors, with up to 20,000 iterations required to reach convergence. No grid sequencing was used. The main difficulty with the calculations was the small Courant–Friedrichs–Lewy (CFL) number required to avoid divergence. For difficult cases involving large gradients, it is usual to for PMB to run with a CFL number of 5. However, the current calculations required a CFL number of 1 or 2, leading to a large number of iterations to convergence. One possible reason for the relatively poor performance is the flow behind the tip missile, which appeared to be unsteady. The probability of unsteady flow outboard of the crank could also be a reason.

V. Conclusions

This paper presented and evaluated results obtained from three multiblock RANS solvers for the F-16XL aircraft. The following conclusions are drawn:

- 1) The generation of a good quality grid was achieved by NLR in a time comparable to that required for the generation of unstructured grids within CAWAPI.
- 2) The agreement between the three codes was generally good, with the expected dependence on the turbulence modelling observed.
- 3) The inner wing primary vortex was well predicted by the vortex correction based turbulence models, offering an easy way to implement a correction to standard two-equation Boussinesq models.
- 4) The outboard primary vortex was not predicted well by any code, perhaps due to unsteady flow in this region.
- 5) A computational time of one to two days was observed for all three codes.

Acknowledgments

This ENSOLV study has been conducted under NLR's programmatic research funding. The Engineering and Physical Sciences Research Council supported the work at Liverpool through the provision of travel grant GR/S16485. The authors would like to thank all the members of the NATO RTO task group AVT-113, "Understanding and Modeling Vortical Flows to Improve the Technology Readiness Level for Military Aircraft," and particularly J.E. Lamar for interesting and fruitful discussions during the course of this project. The authors would also like to acknowledge the following papers that have been summarized herein: "Comparison of Measured and Simulated Flow Features for the Full-Scale F-16XL Aircraft" by O.J. Boelens, S.P. Spekreijse, H.A. Sytsma, and K.M.J. de Cock (AIAA-2007-489); "Evaluation of Results from a Reynolds Averaged Multiblock Code Against F-16XL Flight Data" by K.J. Badcock (AIAA-2007-490); and "PAB3D Simulations for the CAWAPI F-16XL" by A. Elmilgui, K.S. Abdol-Hamid, and S. J. Massey (AIAA-2007-491).

References

- [1] Lamar, J. E., Obara, C. J., Fisher, B. D., and Fisher, D. F., "Flight, Wind-Tunnel, and Computational Fluid Dynamics Comparison for Cranked Arrow Wing (F-16XL-1) at Subsonic and Transonic Speeds," NASA TP-2001-210629, 2001.
- [2] Obara, C. J., and Lamar, J. E., "Overview of the Cranked-Arrow Wing Aerodynamics Project International," *Journal of Aircraft*, Vol. 46, No. 2, 2009, pp. 355–368. doi:10.2514/34957
- [3] Boelens, O. J., Badcock, K. J., Görtz, S., Morton, S., Fritz, W., Karman, S. L., Jr., Michal, T., and Lamar, J. E., "Description of the F-16XL Geometry and Computational Grids Used in CAWAPI," *Journal of Aircraft*, Vol. 46, No. 2, 2009, pp. 369–376. doi:10.2514/34852
- [4] Goertz, S., Jirasek, A., Morton, S. A., McDaniels, D. R., Cummings, R. M., Lamar, J. E., and Abdol-Hamid, K. S., "Standard Unstructured

- Grid Solutions for CAWAPI F-16XL,” *Journal of Aircraft*, Vol. 46, No. 2, 2009, pp. 385–408.
doi:10.2514/35163
- [5] Fritz, W., Davis, M. B., Karman, S. L., Jr., and Michal, T., “RANS Solutions for the CAWAPI F-16XL Using Different Hybrid Grids,” *Journal of Aircraft*, Vol. 46, No. 2, 2009, pp. 409–422.
doi:10.2514/35106
- [6] Rizzi, A., Jirasek, A., Badcock, K. J., Lamar, J. E., Boelens, O. J., and Crippa, S., “What Was Learned from Numerical Simulations of F-16XL (CAWAPI) at Flight Conditions,” *Journal of Aircraft*, Vol. 46, No. 2, 2009, pp. 423–441.
doi:10.2514/35698
- [7] Hummel, D., and Redeker, G., “A New Vortex Flow Experiment for Computer Code Validation,” *RTO-AVT Symposium on Advanced Flow Management; Part A—Vortex Flow and High Angle of Attack Papers on Disc* [CD-ROM], Paper 8, NATO, Neuilly sur Seine, France, May 2001.
- [8] Boerstoel, J. W., Kassies, A., Kok, J. C., and Spekrijse, S. P., “ENFLOW, a Full-Functionality System of CFD Codes for Industrial Euler/Navier-Stokes Flow Computations,” National Aerospace Laboratory TP-96286U, Amsterdam, 1996.
- [9] Badcock, K. J., Richards, B. E., and Woodgate, M. A., “Elements of Computational Fluid Dynamics on Block Structured Grids using Implicit Solvers,” *Progress in Aerospace Sciences*, Vol. 36, No. 5–6, Aug. 2000, pp. 351–392.
doi:10.1016/S0376-0421(00)00005-1
- [10] Abdol-Hamid, K., Pao, S. P., Hunter, C., Deere, K. A., Massey, S. J., and Elmiligui, A., “PAB3D: Its History in the Use of Turbulence Models in the Simulation of Jet and Nozzle Flows,” AIAA Paper 2006-489, 2006.
- [11] Nichols, R. H., and Nelson, C. C., “Application of Hybrid RANS/LES Turbulence Models,” AIAA Paper 2003-83, 2003.
- [12] Abdol-Hamid, K., and Girimaji, S., “A Two-Stage Procedure Toward the Efficient Implementation of PANS and Other Hybrid Turbulence Models,” NASA TM-213260, 2004.
- [13] Kok, J. C., “Resolving the Dependence on Free-Stream Values for the $k-\omega$ Turbulence Model,” National Aerospace Laboratory TP-99295, Amsterdam, 1999.
- [14] Shih, T. H., Zhu, J., and Lumley, J. L., “A New Reynolds Stress Algebraic Equation Model,” NASA TM-106644, Aug. 2004.
- [15] Brandsma, F. J., Kok, J. C., Dol, H. S., and Elsenaar, A., “Leading Edge Vortex Flow Computations and Comparison with DNW-HST Wind Tunnel Data,” National Aerospace Laboratory TP-2001-238, Amsterdam, 2001.
- [16] Dol, H. S., Kok, J. C., and Oskam, B., “Turbulence Modeling for Leading-Edge Vortex Flows,” AIAA Paper 2002-0843, 2002.