Scale-adaptive simulation of a hot jet in cross flow

B M Duda¹, F R Menter², T Hansen² and M-J Esteve¹

¹ AIRBUS Operations S.A.S., Toulouse, France
² ANSYS Germany GmbH, Otterfing, Germany
E-mail: benjamin.duda@airbus.com

Abstract. The simulation of a hot jet in cross flow is of crucial interest for the aircraft industry as it directly impacts aircraft safety and global performance. Due to the highly transient and turbulent character of this flow, simulation strategies are necessary that resolve at least a part of the turbulence spectrum. The high Reynolds numbers for realistic aircraft applications do not permit the use of pure Large Eddy Simulations as the spatial and temporal resolution requirements for wall bounded flows are prohibitive in an industrial design process. For this reason, the hybrid approach of the Scale-Adaptive Simulation is employed, which retains attached boundary layers in well-established RANS regime and allows the resolution of turbulent fluctuations in areas with sufficient flow instabilities and grid refinement. To evaluate the influence of the underlying numerical grid, three meshing strategies are investigated and the results are validated against experimental data.

1. Introduction

Jets in cross flow have been studied both experimentally and numerically because of their frequent occurrence in technical applications. A special area of interest are auxiliary air system outlets on aircrafts, such as discharge locations of anti-icing systems. A generic system is illustrated in figure 1, where hot air circulates inside the nacelle’s lip to prevent the formation of ice on the leading edge. The air is collected in a scoop and then blown out through an ejector grid into the main flow, where it interacts with the downstream structural components. As high temperatures can damage composite parts of the nacelle, special care has to be taken in designing the system and a proper simulation of the flow and temperature field is required. Due to the appearance of large scale turbulent structures and the inherent dynamics of the jet, RANS simulations using standard statistical two-equation turbulence models fail in predicting the correct time averaged quantities. Accounting for anisotropy of turbulence by employing higher order turbulence closure models does not yield considerably improved results (Acharya et al., 2001). Even though Large Eddy Simulations of jets in cross flow can be found in literature e.g. (Yuan et al., 1999), they usually concentrate on small Reynolds numbers and reduced computational domains. Since currently available computational resources impose restrictions on temporal and spatial resolution, Large Eddy Simulations of wall bounded, high Reynolds number flows will not be feasible within the near future. Additionally, as only a local resolution of turbulence scales is required and RANS capabilities should be retained for attached boundary layers, the Scale-Adaptive Simulation (SAS) (Menter & Egorov, 2010; Egorov et al., 2010) is investigated for the considered case. For turbulence model validation, the influence of different mesh types on scale resolution, dynamical behavior of the jet and time statistics is investigated.
2. Test case description

Albugues (2005) experimentally investigated a generic jet in cross flow configuration, which is based on the nacelle anti-icing system exhaust and shown in figure 2. The configuration consists of two pipes feeding hot air symmetrically into a plenum, which is integrated inside a three-dimensional airfoil with a pressure distribution characteristic for a nacelle. As the hot air exits the plenum through a square shaped ejector, a jet in cross flow forms on the upper side of the wing. To characterize the establishing flow field, a cross flow Reynolds number \( \text{Re}_{cf} = U_\infty D/\nu_\infty \) is built using the free stream velocity of the cross flow \( U_\infty \), its kinematic viscosity \( \nu_\infty \) and a characteristic length of the ejector \( D \), which in this case is the square’s edge length. The large value of \( \text{Re}_{cf} = 90000 \) implies the broad range of turbulent scales that appear in the jet and cross flow interaction region and therefore highlights the necessity of proper temporal and spatial resolution. Another important similarity parameter is the blowing ratio \( C_R = (\rho_j U_j)/(\rho_\infty U_\infty) \) (Callaghan & Ruggeri, 1948), which quantifies the momentum ratio of jet and cross flow. The small value of \( C_R = 0.69 \) characterizes an attached jet wake, which consequently leads to a strong thermal impact on the wall behind the orifice. The temperature difference between jet and cross flow for the wind tunnel set-up is \( \Delta T = 62 \, \text{K} \), which can be used to build a cross flow Richardson number \( \text{Ri}_{cf} = g\beta D\Delta T/U_\infty^2 \) to characterize the ratio of free to forced convection, with acceleration through gravity \( g \) and thermal expansion coefficient \( \beta \). As \( \text{Ri}_{cf} \ll 1 \) for the considered case, buoyancy effects can be neglected and temperature is a passive scalar.

3. Simulation methodology

3.1. Turbulence modeling

Background of the SAS turbulence model is the transport equation for \( kL \), with \( k \) being the turbulence kinetic energy and \( L \) the integral length scale of turbulence. As this correlation equation is exact, a term-by-term modeling leads to a more rigorous approach finally introducing the von Kármán length scale \( L_{vK} \) into the transport equation. Transforming the \( k \) – \( \omega \) SST framework (Menter, 1994), an additional source term \( Q_{\text{SAS}} \) enters the turbulence scale equation for the specific dissipation rate \( \omega \).

\[
Q_{\text{SAS}} = \max \left[ \frac{\rho \eta k S^2}{L_{vK}^2} \left( \frac{L}{L_{vK}} \right)^2 - C_{\text{Q}} \frac{2 \rho k}{\sigma_\phi} \max \left( \frac{1}{\omega^2} \frac{\partial \omega}{\partial x_i} \frac{\partial \omega}{\partial x_j}, \frac{1}{k^2} \frac{\partial k}{\partial x_i} \frac{\partial k}{\partial x_j} \right) \right], \quad (1)
\]

This source is activated when the flow exhibits sufficient inherent instabilities and the numerical mesh is sufficiently refined. Consequently, the local eddy viscosity is reduced allowing a resolution of turbulent fluctuations. As the model is based on the \( k \) – \( \omega \) SST turbulence model, boundary layers are simulated using its RANS capabilities including automatic wall treatment.
3.2. Meshing strategies

As the utilized turbulence model requires meshes with the ability to locally resolve turbulent fluctuations and to properly resolve boundary layers, different meshing strategies are considered: a) pure hexahedral, b) hybrid tetrahedral and c) hybrid Cartesian mesh, which are illustrated in figure 3. The first mesh is based on a multi-block approach, which allows very accurate boundary layer resolution and smooth transition inside the volume. The downside however is that mesh refinement cannot be kept locally, therefore increasing the number of cells in areas where it is not needed. The second approach is a hybrid strategy combining prismatic cell layers to accurately resolve the boundary layer and tetrahedral cells in areas away from walls. This allows on the one hand a local refinement confined to the desired areas but on the other hand yields a highly increased number of cells in these areas compared to hexahedral elements. The last approach is also a hybrid strategy, which employs a prismatic and hexahedral inflation layer for boundary layer resolution and a Cartesian mesh in the open domain. Tetrahedral and pyramidal cells are used for the transition from the inflation layer to Cartesian cells. The advantages of this method lie in the use of mostly hexahedral elements and locally confined grid refinement through the use of hanging nodes with the ratio 2:1.

Figure 3. Mesh detail for a) hexahedral, b) hybrid tetrahedral and c) hybrid Cartesian mesh

As on the one hand attached boundary layers will be kept in RANS regime and a heat transfer calculation is considered, the near wall mesh requires a non-dimensional wall distance $y^+$ smaller than one. On the other hand, recalling the ability of the SAS model to resolve turbulent scales, the question of the required temporal and spatial resolution in the jet interaction region needs to be considered. Therefore, an estimation of the large, energy containing and geometry dependent vortices is necessary, which can be achieved by recalling the idea of the energy cascade and respectively the definition of the inertial subrange. For a jet in cross flow, the size $l_0$ of large eddies is in the same order of magnitude as the jet diameter $D$ and their characteristic velocity $u_0$ is in the order of $U_\infty$. As stated by Pope (2000), the demarcation size $l_d$ between geometry dependent vortices and those within the inertial subrange can be defined as $l_d = 1/6l_0$. A characteristic time $t_d$ can then be estimated by

$$t_d \sim \frac{l_0}{u_0} \left( \frac{l_d}{l_0} \right)^{2/3} > \Delta t$$

(2)

and the numerical time step $\Delta t$ is chosen to be smaller than $t_d$. Once the time step $\Delta t$ has been specified, the corresponding grid spacing $\Delta x$ can be estimated through

$$\Delta x = l_0 \left( \frac{u_0 \Delta t}{l_0} \right)^{3/2}$$

(3)
with \( 2\Delta x \) representing the size of the smallest resolvable vortices with the characteristic time \( \Delta t \). These choices ensure the spatial and temporal resolution of all energy containing and geometry dependent vortices, whereas the more universal turbulent fluctuations within the inertial subrange will be accounted for by the statistical turbulence model.

The meshes have been created using ANSYS 13 Meshing applications and the resulting statistics are shown in table 1. The number of inflation layers for the hybrid approaches is 20, with the same height for the wall adjacent cell and the same expansion ratio (\( \sim 1.2 \)) within the boundary layer as for the hexahedral mesh.

|              | Elements | Min. cell angle | Max. aspect ratio | Max. volume change |
|--------------|----------|-----------------|-------------------|--------------------|
| Mesh a)      | 12.9 \( \cdot \) \( 10^6 \) | 28.1°           | 3500              | 10                 |
| Mesh b)      | 21.0 \( \cdot \) \( 10^6 \) | 20.0°           | 7600              | 8                  |
| Mesh c)      | 13.1 \( \cdot \) \( 10^6 \) | 6.0°            | 6000              | 16                 |

### 3.3. Numerical scheme and boundary conditions

The CFD solver ANSYS FLUENT 13 is used to solve the resulting set of equations with a pressure based segregated algorithm, where the SIMPLEC algorithm (Vandoormaal & Raithby, 1984) ensures pressure velocity coupling. A bounded central differencing scheme is used to discretize convective fluxes, whereas an implicit second order central difference scheme is employed for temporal discretization with a physical time step size \( \Delta t = 5 \cdot 10^{-5} \)s. Double precision for numerical accuracy is needed since all three meshes have high aspect ratio cells for boundary layer resolution. Computational domain and boundary conditions are chosen to match the experimental set-up. At the domain inlet a uniform cross flow velocity \( U_\infty = 47.2 \)m/s at \( T_\infty = 291 \)K is specified and the mass flow rate at the each pipe is fixed to \( \dot{m} = 0.01771 \)kg/s with a temperature \( T_j = 353 \)K. At the domain outlet a constant pressure of \( p = 101325 \)Pa is specified. Boundary conditions for turbulence include an eddy viscosity ratio of 10 and a turbulence intensity of 0.5% for both inlet types. All walls of the generic configuration are adiabatic and have no-slip. Considering the small cross flow Mach number and avoiding pressure reflections, an incompressible ideal gas law is used, where density is only a function of temperature. Sutherland’s law is employed to account for temperature influence on viscosity.

### 4. Results and validation

Instantaneous iso-surface of the Q-Criterion (Hunt et al., 1988) are illustrated in figure 4 in order to judge the scale resolvability. All three meshing strategies allow the resolution of turbulent fluctuations at different length scales. As the element edge length is identical for tetrahedral and Cartesian cells in the wake, more and finer structures are resolved for the hybrid tetrahedral approach. A horseshoe vortex, which is a characteristic flow feature of a jet in cross flow, can be found upstream of the ejector and is mostly pronounced for the hybrid tetrahedral and hybrid Cartesian mesh. Additionally, hairpin like vortices form in the wake, which have also been described for jets in cross flow at low blowing ratios (Fric & Roshko, 1994).

Time statistics have been collected for a total of 7000 time steps. As the mean surface temperature behind the ejector is of prime interest, the thermal efficiency \( \eta = (\bar{T} - T_\infty) / \Delta T \) is plotted along the symmetry line and compared to experimental data in figure 5a). In contrast to the far field, where only small differences are visible, the near field shows a stronger mesh
Figure 4. Q-Criterion for a) hexahedral, b) hybrid tetrahedral and c) hybrid Cartesian mesh dependence. The best agreement is achieved with the hybrid tetrahedral mesh, while the hexahedral mesh follows the trend of the experimental data with a slight underestimation. The hybrid Cartesian mesh shows a stronger temperature gradient, which can be explained by the larger cell size leading to lesser scale resolution and hence thermal mixing. Figure 5b) shows the lateral distribution of $\eta$ at a location of $X/D = 8$ downstream of the ejector. The thermal spreading is generally in good agreement with experimental data and only the solution on the hexahedral mesh shows a tendency to underestimate the overall temperature distribution. As the fluctuating field of transient simulations need to be validated too, root mean square values of the $X$-velocity component along a wall normal line are shown in figure 6a). Even though the maximum value is underestimated, a qualitatively good agreement is achieved with only minor dependence on the mesh. A sample power spectral density for the $Y$-velocity component in the jet wake is illustrated in figure 6b). The dominant frequency at $St_D = fU_\infty/D = 0.14$ is predicted well for all three meshing strategies, but with a stronger overestimation of the peak for meshes a) and b). Results from a standard unsteady RANS calculation using the $k-\omega$ SST turbulence model on the hexahedral mesh have been included in these figures to highlight the necessity of scale resolution for a correct simulation of a jet in cross flow.

Figure 5. Thermal efficiency: Exp $\Diamond$, SAS mesh a) $\cdots\cdots$, SAS mesh b) $\cdot\cdot\cdot\cdot\cdot$, SAS mesh c) $\cdot\cdot\cdot$, URANS mesh a) $\cdots\cdots$

5. Summary

Scale-Adaptive Simulations have been carried out for a hot jet in cross flow at a high Reynolds number and a low blowing ratio. It was shown that all applied meshing strategies allow the local resolution of turbulent scales. Due to the characteristics of the turbulence model, more and finer turbulent structures are resolved as the cell size decreases. A good agreement with experimental
data for time averaged temperature is achieved on all meshes. Sample results of root mean square values and spectra show the capability of the SAS turbulence model to correctly reproduce the fluctuating field and the necessity of scale resolution for the proper simulation of a jet in cross flow.

References

ACHARYA, S., TYAGI, M. & HODA, A. 2001 Flow and Heat Transfer Predictions for Film Cooling. *Annals of the New York Academy of Science* 934, 110–125.

ALBUGUES, L. 2005 Analyse expérimentale et numérique d’un jet débouchant dans un écoulement transverse. PhD thesis, L’Ecole Nationale Supérieure de l’Aéronautique et de l’Espace, Toulouse, France.

CALLAGHAN, E. & RUGGERI, R. 1948 Investigation of the penetration of an air jet directed perpendicularly to an air stream. *Tech. Rep.* 1615. National Advisory Committee for Aeronautics.

EGOROV, Y., MENTER, F. R., LECHNER, R. & COKLJAT, D. 2010 The Scale-Adaptive Simulation Method for Unsteady Turbulent Flow Predictions, Part 2: Application to Complex Flows. *Flow, Turbulence and Combustion* 85, 139–165.

FRIC, T. & ROSHKO, A. 1994 Vortical Structure in the Wake of a Transverse Jet. *Journal of Fluid Mechanics* 279, 1–47.

HUNT, J. C., WRAY, A. A. & MOIN, P. 1988 Eddies, stream, and convergence zones in turbulent flows. *Tech. Rep.* CTR-S88. Center for Turbulence Research.

MENTER, F. R. 1994 Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications. *AIAA Journal* 32(8), 269–289.

MENTER, F. R. & EGOROV, Y. 2010 The Scale-Adaptive Simulation Method for Unsteady Turbulent Flow Predictions, Part 1: Theory and Model Description. *Flow, Turbulence and Combustion* 85, 113–138.

POPE, S. 2000 *Turbulent Flows*. Cambridge University Press.

VANDOORMAAL, J. P. & RAITHBY, G. D. 1984 Enhancements of the SIMPLE Method for Predicting Incompressible Fluid Flows. *Numerical Heat Transfer* 7, 147–163.

YUAN, L., STREET, R. & FERZIGER, J. 1999 Large-Eddy Simulations of a Round Jet in Crossflow. *Journal of Fluid Mechanics* 379, 71–104.