Boychev, K., Barakos, G.N. and Steijl, R. (2021) Numerical Simulations of Multiple Shock Wave Boundary Layer Interactions. In: 2021 AIAA SciTech Forum, 11-15 & 19-21 Jan 2021, (doi:[10.2514/6.2021-1762](https://doi.org/10.2514/6.2021-1762))

The material cannot be used for any other purpose without further permission of the publisher and is for private use only.

There may be differences between this version and the published version. You are advised to consult the publisher's version if you wish to cite from it.

<http://eprints.gla.ac.uk/226058/>

Deposited on 10 November 2020

Enlighten – Research publications by members of the University of
Glasgow

<http://eprints.gla.ac.uk>

Numerical simulations of multiple shock wave boundary layer interactions

K. Boychev*, G.N. Barakos[†] and R. Steijl[‡]

CFD Laboratory, School of Engineering, University of Glasgow, G128QQ, Glasgow UK

Shock wave boundary layer interactions occur in many aerospace applications, and of particular interest are the interactions occurring in high-speed intakes. The high-speed intakes aim to decelerate the flow with minimum losses using a series of oblique shocks followed by a weak normal shock. Depending on the state of the boundary layer and on the upstream Mach number, multiple shocks can form in the throat of the intake. Often, they are referred to as shock trains, or pseudo-shocks and can have a significant impact on the intake performance. The in-house CFD solver of the University of Glasgow is used here, to investigate an isolated multiple shock interaction and quantify the effect of different non-linear turbulence models. The non-linear models, and their ability to account for the Reynolds stress anisotropy, resolve the corner flows and give favourable agreement with experiments. As a second step, shock train simulations in a geometry more representative of a high-speed intake are performed. Three different pitot intakes are considered and performance metrics based on the total pressure recovery and flow distortion are evaluated at different free-stream conditions. The predicted shock trains are highly asymmetric and the strong sensitivity of the total pressure recovery and flow distortion to the intake geometry is observed which reduces at higher incidence angles.

Nomenclature

Latin

b_{ij}^{QCR}	=	QCR stress tensor
H	=	Height
$h = H/2$	=	Half height
I	=	Turbulence intensity
k	=	Turbulent kinetic energy
L	=	Length
M	=	Mach number
p	=	Static pressure
P_k	=	Turbulent kinetic energy production limiter
$\mathbf{R}$	=	Flux residual vector
Re	=	Reynolds number
$S_{ij} = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)$	=	Strain rate tensor
S_{ij}^*	=	Strain rate tensor
t	=	Time
$\mathbf{V} = (u, v, w)^T$	=	Velocity vector
$V = \mathbf{V} $	=	Velocity vector magnitude
$V_{i,j,k}$	=	Volume of cell i,j,k
$\mathbf{W}$	=	Vector of conservative flow variables
$\mathbf{x} = (x, y, z)^T$	=	Position vector

*PhD Student, CFD Laboratory, School of Engineering, Email: k.boychev.1@research.gla.ac.uk

[†]Professor, MAIAA, MRAS, CFD Laboratory, School of Engineering, Email: George.Barakos@glasgow.ac.uk

[‡]Senior Lecturer, CFD Laboratory, School of Engineering, Email: Rene.Steijl@glasgow.ac.uk

Greek

δ	=	Boundary layer thickness
δ_{ij}	=	Kronecker delta
η	=	Pressure ratio
λ	=	Roll angle
μ	=	Dynamic viscosity
μ_t	=	Turbulent (Eddy) viscosity
ρ	=	Density
σ	=	Incidence angle
τ_{ij}	=	Viscous stress tensor
$\omega = (\omega_x, \omega_y, \omega_z)^T$	=	Vorticity vector
$\Omega_{ij} = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i} \right)$	=	Vorticity tensor

Subscripts and superscripts

$(.)'$	=	fluctuation of a quantity
$\overline{(.)}, < (.) >$	=	Reynolds averaged quantity
$(.)_u$	=	inlet quantity
$(.)_r$	=	quantity at the start of the interaction (initial pressure rise)
$(.)_\infty$	=	free-stream quantity

Acronyms

CFD	=	Computational Fluid Dynamics
EARSM	=	Explicit Algebraic Reynolds Stress Model
LES	=	Large Eddy Simulation
QCR	=	Quadratic Constitutive Relation
RANS	=	Reynolds Averaged Navier Stokes
RSM	=	Reynolds Stress Model
SST	=	Shear Stress Transport
WMLES	=	Wall-modelled Large Eddy Simulation

I. Introduction

SHOCK wave boundary layer interactions inside ducts where flow confinement effects can have a significant influence on the separation and on the resulting interaction are difficult to simulate and predict accurately. Supersonic or high-speed intakes are representative examples where such flows occur. Confined shock wave boundary layer interactions have been studied experimentally and numerically, and the general conclusion is that the interaction is affected by the corner separations [1, 2]. Reduction or increase of the corner separations through blowing and suction has a direct impact on the separation at the centreline [3]. Downstream of a confined normal shock interaction, the local streamline curvature re-accelerates the flow. Depending on the upstream conditions the curvature might be sufficient to re-accelerate the flow to supersonic speeds and promotes the formation of multiple shocks. Multiple shock wave boundary layer interactions are often referred to as shock trains or pseudo-shocks. Although the terms are often used interchangeably the shock train term refers to the pressure rise across the region where shocks are present and the pseudo-shock to the entire pressure rise region. Figure 1 (a) shows a schematic of a multiple shock wave boundary layer interaction. As the flow goes across the first shock, the local streamline curvature creates an aerodynamic nozzle which re-accelerates the flow. Depending on how strong the re-acceleration is, the flow can again become supersonic and promote the formation of subsequent shocks. Matsuo et al. [4] states that the main parameters that govern the formation of shock trains are the upstream or pre-shock Mach number M_r and the flow confinement δ_r/h where h is the duct half-height and δ_r is the upstream boundary layer thickness.

Fig. 1 Schematic of a multiple shock wave boundary layer interaction (a) and of the numerical setup (b)

At lower M_r the confinement has a stronger effect whereas at higher M_r shock trains are predominantly observed. Shock trains that have a shock with a λ -foot are often referred to as λ -foot shock trains. At higher upstream Mach numbers, the Mach stem disappears and χ -shaped shock trains are exclusively observed. The χ -shape is formed by two crossing oblique shocks. The shock trains featuring λ -foot can often be identified by their distinct supersonic "tongues". These are regions below the slip line where the flow remains supersonic for longer distances downstream. The slip line itself emanates from the triple point formed by the leading and trailing legs of the λ -foot. Many of the past numerical works on shock trains used linear-eddy viscosity models and do not consider the three-dimensional flow. Numerical work by Giglmaier et al. [5] is one of the few that show in detail the benefits of using RSM-based turbulence models for shock train simulations. Scale resolving methods such as LES [6, 7] and WMLES [8] show promising results, however, the Reynolds number must often be reduced for the former, and the latter relies on near-wall modelling which may not be applicable in separated regions. Previous investigations performed by Boychev et al. [9] showed that the use of the $k-\omega$ EARS model improves the shock train predictions of the experiment by Carroll et al. [10] by reducing pressure oscillations occurring at the wall. The sensitivity to symmetry boundary conditions was investigated and no asymmetry in the corner separations was observed. Simulations of single shock interactions performed in the UFAST project [11] show that the use of certain turbulence models can lead to asymmetric corner separations on diagonally opposing walls. Such asymmetries were not observed in the numerical simulations of the experiment by Carroll et al. [10]. The present work investigates the effect of non-linear turbulence model closures on the interaction and whether extensions to existing linear turbulence models such as the quadratic constitutive relation (QCR) can lead to improved flow predictions. In a second step, simulations of high-speed pitot intakes were performed. The performance metrics of three intakes, each having a different geometry, were evaluated at different free-stream conditions.

II. Numerical method

The Helicopter Multi-Block (HMB3) [12, 13] code is used in the present work. HMB3 solves the Unsteady Reynolds Averaged Navier-Stokes (URANS) equations in integral form using the Arbitrary Lagrangian Eulerian (ALE) formulation for time-dependent domains, which may include moving boundaries. The Navier-Stokes equations are discretised using a cell-centred finite volume approach on a multi-block grid. The spatial discretisation of these equations leads to a set of ordinary differential equations in time

$$\frac{d}{dt} (\mathbf{W}_{i,j,k} V_{i,j,k}) = -\mathbf{R}_{i,j,k} (\mathbf{W}_{i,j,k}), \quad (1)$$

where i,j,k represent the cell index, $\mathbf{W}$ and $\mathbf{R}$ are the vector of conservative flow variables and flux residual respectively, and $V_{i,j,k}$ is the volume of the cell i,j,k . To evaluate the convective fluxes Osher [14] approximate Riemman solver is used, while the viscous terms are discretised using a second order central differencing spatial discretisation. The Monotone Upstream-centered Schemes for Conservation Laws, which is referred to in the literature as the MUSCL approach and developed by Leer [15], is used to provide high-order accuracy in space. The HMB3 solver uses the alternative form of the Albada limiter [16] being activated in regions where large gradients are encountered mainly due to shock waves, avoiding the non-physical spurious oscillations. An implicit dual-time stepping method is employed to perform the temporal integration, where the solution is marching in pseudo-time iterations to achieve a fast convergence, which is solved using a first-order backward difference. The linearized system of equations is solved using the Generalised Conjugate Gradient method with a Block Incomplete Lower-Upper (BILU) factorisation as a pre-conditioner [17]. To allow an easy sharing of the calculation load for a parallel job, multi-block structured grids are used. Various turbulence models are available in the HMB3 solver, including several one-equation, two-equation, three-equation, and four-equation turbulence models. Furthermore, Large-Eddy simulation (LES), Detached-Eddy Simulation (DES), and Delayed-Detached-Eddy Simulation (DDES) are also available. In the present work the fully-turbulent $k - \omega$ SST [18] and $k - \omega$ EARSM [19, 20] turbulence models are used. Further modifications to the $k - \omega$ SST model were made which included the addition of two different quadratic constitutive relations (QCRs). The quadratic constitutive relations by Spalart [21] and by Sabnis [22] are considered in the present work.

A. The quadratic constitutive relation

The quadratic constitutive relation (QCR) was developed by Spalart [21] as an extension to the one-equation Spalart-Allmaras linear eddy viscosity model which better predicts the anisotropy of the Reynolds stresses. The relation considers terms that are quadratic in the mean vorticity and strain tensors. There have been two versions of the quadratic constitutive relation developed by Spalart. The original version - QCR-2000 [21] and a more recent QCR-2013 version. The latter version features an additional term which accounts for the $\frac{2}{3}\rho k\delta_{ij}$ term in the Boussinesq approximation, equation 2, since the Spalart-Allmaras model does not include k . The QCR can also be applied to two-equation eddy viscosity models. In this case, the additional term in the QCR is redundant since the two-equation eddy viscosity model provides k . An additional QCR stress tensor b_{ij}^{QCR} is added to the existing stress tensor τ_{ij} as:

$$\tau_{ij}^{RANS} = 2\mu_t \bar{S}_{ij}^* - \frac{2}{3}\rho k\delta_{ij} - b_{ij}^{QCR}, \quad (2)$$

where the definition of b_{ij}^{QCR} depends on the QCR version. In the original QCR-2000 [21] version b_{ij}^{QCR} is given by:

$$b_{ij}^{QCR} = C_{cr1} (O_{ik}\tau_{jk} + O_{jk}\tau_{ik}), \quad (3)$$

where $\tau_{ij} = 2\mu_t \bar{S}_{ij}^* - \frac{2}{3}\rho k\delta_{ij}$ is the stress tensor given by the Boussinesq approximation, $O_{ij} = 2\Omega_{ij}/\sqrt{\frac{\partial u_m}{\partial x_n} \frac{\partial u_m}{\partial x_n}}$ is the antisymmetric normalized rotation tensor, and $C_{cr1} = 0.3$ is a model coefficient, calibrated in the outer part of an equilibrium boundary layer. Another QCR version, developed by Sabnis [22], was also considered. This QCR differs from the QCR-2000 [21] version by the addition of one additional term, quadratic in the strain tensor, and is given by:

$$\begin{aligned} b_{ij}^{QCR} = & \frac{4C_{cr1}\mu_t}{\sqrt{\bar{S}_{kl}\bar{S}_{kl} + \bar{\Omega}_{kl}\bar{\Omega}_{kl}}} (\bar{\Omega}_{ik}\bar{S}_{kj}^* - \bar{S}_{ik}^*\bar{\Omega}_{kj}) \\ & + \frac{4\tilde{C}_{cr3}\mu_t}{\sqrt{\bar{S}_{kl}\bar{S}_{kl} + \bar{\Omega}_{kl}\bar{\Omega}_{kl}}} \left(\bar{S}_{ik}^*\bar{S}_{kj}^* - \frac{1}{3}\bar{S}_{kl}^*\bar{S}_{kl}^*\delta_{ij} \right), \end{aligned} \quad (4)$$

where $C_{cr1} = 0.7$ and $\tilde{C}_{cr3} = 0.8$ are model coefficients. Channel, boundary layer, and pipe flows at moderate Reynolds numbers were used to calibrate the model coefficients. The QCR-2000 version by Spalart [21] and the QCR by Sabnis [22] are referred to as QCR V1 and QCR V2 in this work.

III. Numerical setup and simulations

A. Shock train simulations

For the shock train simulations presented here, the multiple SWBLI experiments by Carroll and Dutton [10], are targeted for comparison. In the experiment, a 753.8 mm long rectangular test section was used with LDV measurements beginning at $x_r = 264.8$ mm, the approximate location of the initial shock, and extending downstream, at variable intervals over 400 mm. The top and bottom wall of the rectangular test section had a divergence angle of 0.13 deg. From the boundary layer measurements the height of the rectangular test section at $x_r = 264.8$ mm was $2h_r = 33.75$ mm ($\delta_r/h = 0.32, \delta_r = 5.4$ mm). The width of the section is $2w = 76.2$ mm and is constant. The Mach number before the interaction was $M_r = 1.61$ and the unit Reynolds number was $Re = 3.0 \times 10^7 \text{ m}^{-1}$. The Reynolds number based on the half-height of the duct at $x_r = 264.8$ mm is, therefore, $Re_h \approx 5.06 \times 10^5$. A numerical domain of length $L_x/h = 46.32$ was used for the shock train simulations of the experiment by Carroll et al. [10]. The domain was made dimensionless by the duct half-height at $x/h = 0$ ($h = 16.275$ mm). Figure 1 (b) shows a qualitative sketch of the numerical domain. The numerical domain included the divergence angle of 0.13 deg, however, the angle was not sufficient to keep the Mach number constant. Therefore to match the correct experimental Mach number upstream of the interaction the Mach number at the inlet M_u was increased to take into account the downstream reduction due to the boundary layer growth. Our previous numerical investigations [9] of the experiment by Carroll et al. [10] revealed the sensitivity of the solution to grid refinement and other RANS modelling assumptions. Only a quarter of the numerical domain was simulated with symmetry boundary conditions applied at the $x - y$ and $x - z$ planes. Extrapolation was performed at the outlet of the domain where the flow is supersonic, otherwise, a specified pressure value was imposed. Based on [9], the grid employed here was made of 1199, 76, and 98 points in the x-,y-,and z-direction and consisted of 8.93×10^6 points. Hyperbolic stretching is used away from the wall. The first cell height above the wall is equal to 1×10^{-5} and ensures a y^+ value less than 1. The maximum cell size in the refinement region was $\Delta x/h = 0.02$ and $\Delta y/h = \Delta z/h = 0.07$. The simulation parameters are listed in table 1.

Table 1 Shock train simulation parameters

I_u %	M_u	M_r	δ_r mm	δ_r/h	x_r/h	p/p_u	p/p_r	Model
1.0	1.69	1.6036	6.1324	0.3768	25.3988	2.4776	2.1787	k- ω SST QCR V1
1.0	1.69	1.6039	6.2261	0.3826	25.8962	2.4776	2.1803	k- ω SST QCR V2
1.0	1.69	1.6230	5.0083	0.3077	26.4258	2.4776	2.2357	k- ω EARSM
1.0	1.69	1.6036	6.1441	0.3775	25.2981	2.4776	2.1741	QCR V1 $C_{cr1} = 0.4$
1.0	1.69	1.6054	6.1571	0.3783	25.2981	2.4776	2.1805	QCR V1 $C_{cr1} = 0.4$ P_k lim
1.0	1.69	1.6185	5.1769	0.3181	20.9316	2.6015	2.3183	QCR V1 $C_{cr1} = 0.4$
		1.6100	5.4000	0.3200			2.2309	Experiment [10]

The flow was initialised with an ideal normal shock placed at the end of the numerical domain. The steady-state solutions are fully-implicit and were obtained with an implicit CFL of 4. Flux-residuals were allowed to converge to at least 5 orders of magnitude.

B. High-speed pitot intake simulations

To investigate the shock trains in the context of high-speed intakes a fore-body intake geometry was created. Figure 2 shows the intake geometry which comprises of a spherically blunted tangent ogive followed by a cylinder of constant diameter D . A total of three intake configurations were considered - a square one, one resembling a kidney shape, and one with rounded corners. They are referred to as configurations A, B, and C in this work. The height and the width of the intake for configuration A are $h = w = 0.5H$ where $H = 0.5D$ and the thickness $t = 0.03D$. The thickness reduces towards the lips of the intake which are filleted. The intake geometry of configuration B was obtained by filleting two concentric circles with radiuses $R_1 = D$ and $R_2 = 0.6D$. Configuration C is similar to A apart from the slightly increased width and height $h = w = 0.5H = 0.2525D$ and the filleted corners. The cross-sectional area was kept approximately the same between the three configurations $A \approx 0.0025L$. The diverter width and intake placement were driven by the findings of Goldsmith [23] which showed that the total pressure recovery increases if the diverter width is $0.1D$ and the intake is placed at 45 deg to the symmetry plane. All intakes are self-starting since the ratio of the intake

throat to capture areas is 1 and it is above the Kantrowitz limit [24]. Multi-block structured grids of approximately 890 blocks were used to discretise the numerical domain. The grids were non-dimensionalised with the fore-body length of $L = 12D$. Figure 3 shows the blocking topology on the domain boundaries and the employed boundary conditions. The free-stream Mach number of $M_\infty = 2$ and $Re_L = 2.5 \times 10^7$ were selected as to obtain similar conditions based on the half-height h of the intake as the ones in the shock-train simulations of the experiment by Carroll et al. [10]. A minimum wall distance of $2.5 \times 10^{-7}D$ was selected to give a minimum wall distance of 1×10^{-5} based on h . This ensured a $y^+ < 1$. Table 3 lists the simulation parameters and table 2 the grid parameters.

Fig. 2 Fore-body intake geometries; configuration A (top), configuration B (middle), configuration C (bottom)

Table 2 Fore-body intake grid parameters

Configuration	Min dist.	Intake points	Total points	Total cells
A	$2.5 \times 10^{-7}D$	26.82×10^6	60.04×10^6	57.40×10^6
B	$2.5 \times 10^{-7}D$	15.65×10^6	60.07×10^6	54.41×10^6
C	$2.5 \times 10^{-7}D$	15.65×10^6	60.07×10^6	54.41×10^6

Table 3 Fore-body intake cases; colon separates port and starboard quantities

Configuration	Symmetry	σ (deg)	λ (deg)	TPR	FD	η	Model
A	Y	5	0	0.7122	0.6854	3.825	k- ω EARS
A	N	5	3	0.7136 : 0.7095	0.6799 : 0.6778	3.825	k- ω EARS
B	Y	5	0	0.7166	0.6873	3.825	k- ω EARS
B	N	5	3	0.7191 : 0.7147	0.7074 : 0.7109	3.825	k- ω EARS
C	Y	5	0	0.7172	0.6820	3.825	k- ω EARS
C	N	5	3	0.7177 : 0.7140	0.6944 : 0.6938	3.825	k- ω EARS

The k- ω EARS turbulence model was used in the simulations as it showed favourable agreement for the idealised

Fig. 3 Fore-body intake boundary conditions (left) and topology (right)

shock train case, presented in the previous section. The angles σ and λ correspond to the incidence and roll-angle of fore-body intake with respect to the free-stream velocity vector. The simulations were initialised with an inviscid shock of strength $M = 2.0$ at the end of the intake and were run until the aerodynamics loads stabilised. The solutions are fully-implicit, using a MUSCL third-order scheme, and are performed with a CFL of 1.

IV. Results and discussion

A. Shock train results

Figure 4 shows the wall pressure and the Mach number contours at the mid-plane. The solid black line indicates $M = 1$. The QCR extension reduces the pressure oscillations at the wall, however, the wall pressure is still slightly overpredicted. The overprediction appears at $(x - x_r)/h \approx 1$. The distance between the first and second shock in the shock train is smaller for the $k-\omega$ SST QCR V1 and V2 models and the supersonic flow is farther away from the wall (indicated by the sonic $M = 1$ line). The size of the corner separations, shown by the iso-surfaces in figure 5, is considerably reduced by the introduction of the QCR extension. Both models reduce the streamwise and spanwise extent of the corner separations. As the size of the corner separations and the centreline separation are dependent in confined flows, the centreline separation increases. Figure 6 shows wall shear stress visualisation using friction lines just above the surface. The increase of the separated region at the centreline with respect to the reference is 3.15 and 3.5 times for the $k-\omega$ SST QCR V1 and V2 respectively. The onset of the corner separation begins at approximately $(x - x_r)/h \approx -1.2$ for the $k-\omega$ SST model. For the $k-\omega$ SST QCR V1 and V2 models the corner separation begins at approximately $(x - x_r)/h \approx -0.4$. Figure 7 shows the comparison of the $k-\omega$ EARSIM wall shear stress visualisation and the experimental oil flow visualisation. Red lines indicate the surface flow features. The non-linear model overpredicts the centreline separation size, however, the size of the corner separations qualitatively agrees with the experiment. Downstream of x_r , in the duct corners, a flow pattern is observed where the oil flow lines roll-up. Similar roll up is observed in the $k-\omega$ EARSIM simulation and the simulations employing the QCR extensions.

Fig. 4 Wall pressure and Mach number contours for the $k-\omega$ SST, $k-\omega$ SST QCR V1, $k-\omega$ SST QCR V2, and the $k-\omega$ EARSM models

Fig. 5 Separation $u/V_u = 1 \times 10^{-3}$ and sonic $M = 1$ iso-surfaces for the $k-\omega$ SST, $k-\omega$ SST QCR V1, $k-\omega$ SST QCR V2, and the $k-\omega$ EARSM models

The turbulent kinetic energy profiles, shown in figure 8, agree well with the experiment, but closer inspection of the Reynolds stresses shows that the $\overline{u'u'}/V_u^2$ component is underpredicted and the $\overline{v'v'}/V_u^2$ component is overpredicted. The addition of the QCR extension does not affect the $\overline{u'u'}/V_u^2$ and $\overline{v'u'}/V_u^2$ components. The $\overline{v'v'}/V_u^2$ component is the most affected by the addition of the QCR extension. Both QCR V1 and V2 overpredict the $\overline{v'v'}/V_u^2$ component.

Fig. 6 Wall shear stress visualisation using friction lines just above the surface for the $k-\omega$ SST, $k-\omega$ SST QCR V1, $k-\omega$ SST QCR V2, and the $k-\omega$ EARSM models

Fig. 7 Wall shear stress visualisation using friction lines just above the surface for the $k-\omega$ EARSM model (top); experimental oil flow (bottom)

The simulation with the QCR V1 model having a coefficient of $C_{cr1} = 0.4$ further reduced the corner separations. No significant differences in the modelled Reynolds stresses were observed for $C_{cr1} = 0.4$. Only a slight increase of the peak $\overline{v'v'}/V_u^2$ values at $y/h \approx 0.18$ was present. Similarly to the original QCR V1 model for which $C_{cr1} = 0.3$, the model with increased C_{cr1} showed a slight overprediction of the wall pressure at the beginning of the interaction. The difference in the interaction onset x_r between the models amounted to 0.2%. In comparison to the SST solution where $C_{cr1} = 0$ the movement of x_r amounted to approximately 6.2% of the domain length. The movement downstream was accompanied by approximately 12% increase in confinement levels. For the models predicting an initial shock with a Mach stem, higher values of k are present near the centreline. Since the employed models do not limit the P_k term they result in increased values of k/V_u^2 , $\overline{u'u'}/V_u^2$, and $\overline{v'v'}/V_u^2$ near the centreline downstream of the first and second shocks. This can be seen in figures 8, 9, and 11. The effect of limiting P_k was investigated separately, and no significant differences in the onset of the interaction x_r and in the Reynolds stress profiles near the wall were observed. The high values near the centreline were removed by limiting P_k .

Fig. 8 Turbulent kinetic energy k/V_u^2 contours and profiles for the four turbulence models

Fig. 9 Reynolds stress $\overline{u'u'}/V_u^2$ contours and profiles for the four turbulence models

Fig. 10 Reynolds stress $-\overline{u'v'}/V_u^2$ contours and profiles for the four turbulence models

Fig. 11 Reynolds stress $-\overline{v'v'}/V_u^2$ contours and profiles for the four turbulence models

The shock spacings of the QCR V1 $C_{cr1} = 0.4$, QCR V1 $C_{cr1} = 0.4$ with the P_k limiter and QCR V1 $C_{cr1} = 0.4$ with increased backpressure simulations were compared against the experiment. The spacings were obtained either from the density gradient magnitude or from the experimental Schlieren, and were normalized with respect to the spacing between the first and second shocks in the shock train, Δ , shown in figure 12. The normalization was performed in order to reduce potential errors arising from the scaling of the experimental Schlieren image. Table 4 lists the spacings and figure 13 compares the spacings. Empty cells correspond to no measurement due to the difference in the number of downstream shocks between the experiment and the simulations. The effect of the P_k limiter on the d_1 is small. Without the limiter $d_1 = 1.7766h$ and with the limiter $d_1 = 1.7662h$. The relative reduction of d_1 is approximately 0.5%. The d_2/Δ distance is also weakly affected by the limiter. As the last shock for the case without the limiter is very weak, comparison of d_3/Δ is difficult, nevertheless, d_2/Δ and d_3/Δ are overpredicted by approximately 11% and 25% when compared to the experiment. Increase of the backpressure to $p/p_r = 2.3183$ relocated the shock train upstream where the confinement level was in closer agreement in the experiment. As the confinement level, δ_r/h is more important at lower Mach numbers, the M_r reduction resulted in smaller distances between the shocks. The d_2/Δ and d_3/Δ distances were still overpredicted, however, the overprediction was reduced by approximately 6%.

Fig. 12 Definition of the distance Δ used to normalize the distances between the shocks; $\Delta = d_1$

Table 4 Distances between the shocks in the shock trains

Case	d_1/Δ	d_2/Δ	d_3/Δ	d_4/Δ	d_5/Δ	d_6/Δ
$C_{cr1} = 0.4$	1.0000	0.8132	0.7033			
$C_{cr1} = 0.4 P_k$ lim	1.0000	0.8109	0.6667	0.5871		
$C_{cr1} = 0.4 p/p_r = 2.3183$	1.0000	0.7661	0.6422			
Experiment	1.0000	0.7319	0.5483	0.4313	0.3681	0.3083

The improved predictions show that adjustment of the confinement level upstream of the shock train to match the experiment is important. However, every turbulence model will predict the boundary layer development differently

which would require iterative adjustment of the inlet/outlet conditions to match the confinement. This is one of the difficulties in simulating the idealised shock train interactions.

Fig. 13 Distances between the shocks in the shock trains

B. High-speed pitot intake results

Figures 14 to 16 show the top and bottom wall pressures and the Mach number contours of the A,B, and C configurations at a pitch angle of 5 deg and a roll angle of 0 deg. All configuration result in multiple shock wave boundary layer interactions inside the intake, however, the structure of the interactions differs from the idealised one presented in the previous section. There is an absence of symmetry and the structure of the initial shock is not as well defined as in the isolated interactions. For the same pressure ratio and far-field conditions, the onset of the interaction x_r begins at $x/h \approx 13.8$. The onset of the interaction for configuration A is similar on the top and bottom walls and so are the wall pressure profiles. The top wall exhibits slight pressure oscillations, suggesting that the shocks are stronger in the top wall region. The largest difference in x_r is observed for configuration B where the onset of the interaction begins at $x/h \approx 10$ for the bottom wall and $x/h \approx 12$ for the top wall. This shows that the initial shock is weaker in the bottom wall region of the intake. Configuration C shows a smaller relative difference in x_r which amounts to approximately 5.8%. Compared to configuration B, which has a relative difference of 17.6% the decrease is approximately 67%.

Fig. 14 Configuration A top and bottom intake wall pressures and Mach contours

Figure 17 shows the Mach number contours for the three configurations at $x/L = 0.9$. Configuration A does not feature separations at the walls, however, large corner separations are observed in two opposing corners. The asymmetry in the corner separations can be seen from the figure. Configurations B and C both feature a large separation at the

Fig. 15 Configuration B top and bottom intake wall pressures and Mach contours

Fig. 16 Configuration C top and bottom intake wall pressures and Mach contours

bottom wall of the intake and smaller separations on the top and sidewalls. The separations are not symmetric, however, the separations on the sidewalls of configuration C were very similar in size. The effect of the incidence angle on the separations was to increase the asymmetry. At a zero incidence angle, $\sigma = 0$, configuration A resulted in relatively symmetrical corner separations. The separation on the bottom intake wall of configurations B and C were observed to increase as the incidence angle was increased. The opposite was true for the separations on the side and top walls. Figure 19 compares the Mach number contours of the three configurations at zero and non-zero roll angles. The incidence angle is fixed at $\sigma = 5$ deg. The non-zero roll moves the interaction onset downstream. The greatest streamwise movement is for configuration C, suggesting that the effect of non-uniform confinement is to increase the movement.

The total pressure recovery, TPR, and the flow distortion, FD, were evaluated for each configuration at the engine face. These efficiency metrics are commonly used for high-speed intakes and are given by

$$FD = \frac{p_{0,max} - p_{0,min}}{p_{0,average}}, \quad (5)$$

$$TPR = \frac{p_{0,average}}{p_{0,\infty}}, \quad (6)$$

where $p_{0,average}$ is the total averaged pressure at the engine face and $p_{0,\infty}$ is the total pressure of the free-stream. The effect of pseudo-shock length on the total pressure recovery was investigated by Mahoney [25] who showed that maximum total pressure recovery is achieved when the throat length of the intake equals or is slightly greater than the pseudo-shock length. For maximum total pressure recovery at off-design conditions e.g. increase/decrease of the

Fig. 17 Mach number contours at $x/D = 0.9$; configuration A (left), configuration B (middle), configuration C (right)

design Mach number or non-uniform flow due to change in the angle of attack or sideslip, it was concluded that the length of the throat section should be sufficient to account for changes in the pseudo-shock length. The average total pressure recovery for the three configurations is $TPR \approx 0.7153$ and the flow distortion is $FD \approx 0.6849$. Figure 18 shows the total pressure recovery for the three configurations for zero and non-zero roll and incidence angles. The TPR for all three configurations increases by approximately 6.3% with increasing incidence angle. Configurations B and C have slightly higher TPR than configuration A. The same is also true when the roll angle is non-zero. The TPR in the port intake is consistently higher than the TPR in the starboard intake. Compared to the zero roll case, the TPR in the starboard intake is on average lower by approximately 0.5%. The flow distortion shown in figure 20 is more sensitive to the incidence angle. A significant increase for configurations B and C is observed at a zero incidence angle. Configuration B results in the highest FD at the engine face, approximately 14% higher than configuration A. As the incidence angle is increased, the flow distortion of the intakes is relatively similar. Configuration C results in the smallest FD amongst the three. When a non-zero roll angle is introduced, the FD for configuration A decreases by approximately 1%. For both configurations B and C the FD increases, with configuration C having the smallest increase of FD - approximately 1.7%. The relative increase with respect to configuration A is approximately 2%. The onset of the interaction x_r moves downstream for all three configurations as the roll angle is increased. Higher flow distortion values are expected if the supersonic core flow is closer to the engine face.

Fig. 18 Total pressure recovery, TPR, for configurations A, B, and C

Fig. 19 Mach number contours for configuration A (top), B (middle), C (bottom), effect of roll angle

Fig. 20 Total pressure recovery, TPR, for configurations A, B, and C

V. Conclusions and future work

Improved agreements in the wall pressure between the simulations and the experiments of Carroll et al. [10] were observed when the $k-\omega$ EARSM turbulence model was used. The $k-\omega$ EARSM model which uses the constitutive relation by Wallin and Johansson [19] accounts for the secondary flows present in duct corners. The vortices in this region supply high momentum fluid from the core flow and reduce the size of the corner separations. Since the corner separations affect the separation at the centreline in confined flows, reduction of the corner separation increased the centreline separation. Two additional models - the $k-\omega$ SST QCR V1 and the $k-\omega$ SST QCR V2 based on the QCR by Spalart [21] and the QCR by Sabnis [22] were implemented in the HMB3 flow solver. Both models reduced the separations at the corner and showed improved agreements in the wall pressure compared to the $k-\omega$ SST model. The $k-\omega$ SST QCR V2 model was observed to predict higher levels of normal stress anisotropy compared to the QCR V1 and the EARSM models. Variation of the C_{cr1} coefficient of the $k-\omega$ SST QCR V1 model showed that a coefficient of $C_{cr1} = 0.4$ improves the prediction by reducing the corner separations even further. Inclusion of the P_k limiter did not have a significant effect on the wall pressure and onset - the wall pressure and the onset of the interaction were almost identical. The higher values of k/V_u^2 at the centreline, downstream of the first and subsequent shocks were removed by the limiter. The spacing of the shocks did not vary significantly and it was overpredicted when compared to the experiment. In summary, the addition of QCR extensions to the linear $k-\omega$ SST turbulence model improved the wall pressure predictions. However, further investigations of other shock train experiments with this model are necessary to determine its suitability for such flows. The fore-body intake geometry, created to couple the internal shock train flow with the more realistic external missile flow showed that the total pressure recovery and flow distortion is less sensitive to the intake geometry at higher angles of incidence. A positive roll angle did not introduce significant changes in the total pressure recovery - the total pressure recovery in the port intakes was consistently higher than the starboard one. The flow distortion of the particular kidney and filleted intakes was more sensitive to the roll angle. Increased values were observed at positive roll angles. Although the pitot intake is not efficient in terms of the performance metrics, the investigation showed that shock trains were present in all three intake configuration and their structure was significantly different from the idealised case.

Acknowledgments

The authors would like to acknowledge J. Dutton for providing the experimental data. The computational resources provided by ARCHIE-WeSt and the financial support from MBDA UK are also gratefully acknowledged.

References

- [1] D.M.F., B., Babinsky, H., and Bruce, P., "Experimental Investigation into Parameters Governing Corner Interactions for Transonic Shock Wave/Boundary Layer Interactions," *48th AIAA Aerospace Sciences Meeting Including the New Horizons Forum and Aerospace Exposition*, 2010.

- [2] Bruce, P., H. B., Tartinvill, B., and Hirsch, C., "Corner Effect and Asymmetry in Transonic Channel Flows," *AIAA Journal*, Vol. 49, No. 11, 2011, pp. 2382–2392.
- [3] Bruce, P., Burton, D., Titchener, N., and Babinsky, H., "Corner effect and separation in transonic channel flows," *Journal of Fluid Mechanics*, Vol. 679, 2011, p. 247–262.
- [4] Matsuo, K., Y. M., and Kim, H.-D., "Shock train and pseudo-shock phenomena in internal gas flows," *Progress in Aerospace Sciences*, Vol. 35, 1999, pp. 33–100.
- [5] Giglmaier, M., Quaat, J., Gawehn, T., Gulhan, A., and Adams, N., "Numerical and experimental investigations of pseudo-shock systems in a planar nozzle: impact of bypass mass flow due to narrow gaps," *Shock Waves*, 2014.
- [6] Morgan, B., Duraisamy, K., and S.K., L., "Large-Eddy and RANS Simulations of a Normal Shock Train in a Constant-Area Isolator," *50th AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition*, 2012.
- [7] Morgan, B., Larsson, J., Kawai, S., and K., L. S., "Large-Eddy Simulations of a normal shock train in a constant-area isolator," *AIAA Journal*, Vol. 52, No. 3, 2014, pp. 539–558.
- [8] Vane, Z., Bermejo-Moreno, I., and Lele, S., "Simulations of a Normal Shock Train in a Constant Area Duct Using Wall-Modeled LES," *43rd Fluid Dynamics Conference*, 2013.
- [9] Boychev, K., Barakos, G., and Steijl, R., "Flow physics and sensitivity to RANS modelling assumptions of a multiple shock wave/turbulent boundary layer interaction," *Aerospace Science and Technology*, Vol. 97, No. 1, 2020.
- [10] Carroll, B., and Dutton, J., "Multiple normal shock wave/turbulent boundary-layer interactions," *Journal of Propulsion and Power*, Vol. 8, No. 2, 1992, pp. 441–448.
- [11] Doerffer, P., Hirsch, C., Dussauge, J., Babinsky, H., and G.N., B., "Unsteady Effects of Shock Wave Induced Separation," *Notes on Numerical Fluid Mechanics and Multidisciplinary Design*, 2011.
- [12] Steijl, R., Barakos, G. N., and Badcock, K., "A framework for CFD analysis of helicopter rotors in hover and forward flight," *International Journal for Numerical Methods in Fluids*, Vol. 51, No. 8, 2006, pp. 819–847.
- [13] Steijl, R., and Barakos, G. N., "Sliding mesh algorithm for CFD analysis of helicopter rotor-fuselage aerodynamics," *International Journal for Numerical Methods in Fluids*, Vol. 58, No. 5, 2008, pp. 527–549.
- [14] Osher, S., and Chakravarthy, S., "Upwind schemes and boundary conditions with applications to Euler equations in general geometries," *Journal of Computational Physics*, Vol. 50, No. 3, 1983, pp. 447–481.
- [15] van Leer, B., "Towards the ultimate conservative difference scheme. V. A second-order sequel to Godunov's Method," *Journal of Computational Physics*, Vol. 32, No. 1, 1979, pp. 101–136.
- [16] van Albada, G. D., van Leer, B., and Roberts, W. W., "A Comparative Study of COmputational Methods in Cosmic Gas Dynamics," *Astronomy and Astrophysics*, Vol. 108, No. 1, 1982, pp. 76–84.
- [17] Axelsson, O., *Iterative Solution Methods*, Cambridge University Press, 1994.
- [18] Menter, F., "Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications," *AIAA Journal*, Vol. 32, No. 8, 1993, pp. 1598–1605.
- [19] Wallin, S., and Johansson, A., "An Explicit Algebraic Reynolds Stress Model for Incompressible and Compressible Turbulent Flows," *Journal of Fluid Mechanics*, Vol. 403, 2000, pp. 89–132.
- [20] Hellsten, A., "New Advanced k-omega Turbulence model for High-Lift Aerodynamics," *AIAA Journal*, Vol. 43, No. 9, 2005, pp. 1857–1869.
- [21] Spalart, P., "Strategies for turbulence modelling and simulations," *International Journal of Heat and Fluid Flow*, Vol. 21, No. 2, 2000.
- [22] Sabnis, K., "Supersonic Corner Flows in Rectangular Channels," Ph.D. thesis, University of Cambridge, 2020.
- [23] Goldsmith, E., and Seddon, J., *Practical Intake Aerodynamic Design (AIAA Education Series)*, 1st ed., American Institution of Aeronautics and Astronautics, 1993.
- [24] Kantrowitz, A., and Donaldson, C., "Preliminary investigation of supersonic diffusers," Tech. Rep. NACA-WR-L-713, Langley Memorial Aeronautical Laboratory, 1945.
- [25] Mahoney, J., *Inlets for supersonic missiles (AIAA Education Series)*, 1st ed., American Institute of Aeronautics and Astronautics, 1990.