Langley Research Center Workshop
CFD Validation of Synthetic Jets and Turbulent Separation Control, March 29-31, 2004

Case 2: Synthetic Jet in a Crossflow

Abbes AZZI
Faculty of Mechanical Engineering
University of Oran, USTO, Algeria

Djamel LAKEHAL
Institute of Energy Technology
ETH Zurich, Switzerland

OUTLINE

- Transport Equations & Turbulence Modelling,
- The Simulation method
- Synthetic Jet in a Crossflow: Results

Transport Equations and Turbulence Modelling

(I): 'Conditional' Phase Averaged Navier-Stokes Equations, using the Reynolds and Hussein's decomposition:

$$f = \underbrace{\overline{f}}_{mean} + \underbrace{\tilde{f}}_{coherent} + \underbrace{f'}_{stochastic} \quad ; \quad \text{with} \quad \langle f \rangle = \overline{f} + \tilde{f}$$

'Conditional' means that the approach reduces to RANS for flows not dominated by a single frequency, e.g. vortex shedding.

The Conditional Phase Averaged continuity and momentum conservation equations:

$$\frac{\partial \langle u_i \rangle}{\partial x_i} = 0,$$

$$\frac{\partial \langle u_i \rangle}{\partial t} + \frac{\partial \langle u_i \rangle \langle u_j \rangle}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \langle p \rangle}{\partial x_i} + \nu \nabla^2 \langle u_i \rangle - \frac{\partial \tau_{ij}}{\partial x_j}$$

where

$$\tau_{ij} \equiv \langle u_i u_j \rangle - \langle u_i \rangle \langle u_j \rangle = \langle u_i' u_j' \rangle + \tilde{u}_i \tilde{u}_j$$

The time–average equations describing the mean flow:

$$\frac{\partial(\overline{u_i}\,\overline{u_j})}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i} + \nu\nabla^2\overline{u_i} - \frac{\partial}{\partial x_j}\left[\overline{u'_i u'_j} + \overline{\tilde{u}_i\tilde{u}_j}\right]$$

where $\overline{\tilde{u}_i\tilde{u}_j}$ denotes the wave-induced stress.

While $\tilde{u}_i\tilde{u}_j$ is embodied into the momentum equations (solved directly), it is the phase average Reynolds stress $\langle u'_i u'_j\rangle$ that requires closure:

In the EVM framework:

$$\langle u'_i u'_j\rangle \;= \frac{2}{3}\,\delta_{ij}\langle k\rangle - 2\,\nu_t\,\langle S_{ij}\rangle$$

or in the ASM framework:

$$\langle u'_i u'_j\rangle \;= \frac{2}{3}\,\delta_{ij}\langle k\rangle - 2\,\nu_t\,\langle S_{ij}\rangle + \nu_t^*\langle k\rangle/\langle\varepsilon\rangle\,[\sum C_n(\langle T_{ij}\rangle)_n]$$

$$(\langle T_{ij}\rangle)_1 = (\langle S_{ik}\rangle\langle S_{kj}\rangle - \tfrac{1}{3}\langle S_{lk}\rangle\langle S_{kl}\rangle\delta_{ij})$$

$$(\langle T_{ij}\rangle)_2 = (\langle \Omega_{ik}\rangle\langle S_{kj}\rangle + \langle \Omega_{jk}\rangle\langle S_{ki}\rangle)$$

(II): The TLV 2-layer $k - \varepsilon$ turbulence model (Lakehal & Rodi,, J. Wind Eng., 1996; Azzi & Lakehal, J. Turbomach. 124(3), 2002)

The core flow region and near-wall region are solved separately then coupled (dynamically, without fixing the grid) according to the damping function (e.g. when the van Driest function $f_\mu \approx 0.95$).

the core flow (2-equation model):

$$\nu_t = C_\mu \langle k \rangle^2 / \langle \varepsilon \rangle$$

solve for k and ε as in the standard model ($k_{wall} = 0$)

the viscosity-affected Layer (1-equation model based on Kim's and Spalart's DNS data):

$$\nu_t \equiv C_\mu \sqrt{\langle k \rangle} \ell_\mu \, ; \quad \langle \varepsilon \rangle = \langle k \rangle^{3/2} / \ell_\varepsilon$$

$$\ell_\mu = C_l \, y_n \, f_\mu \, ; \quad \ell_\varepsilon = \frac{C_l \, y_n}{2 + C_\varepsilon / (C_l \, R_y \, f_\mu)}$$

$$f_\mu = \frac{1}{32} \sqrt{0.116 \, R_y^2 + R_y}; \quad R_y = \langle k \rangle^{1/2} y_n / \nu$$

$$C_l = \kappa / C_\mu^{3/4}; \quad C_\mu = 0.082; \quad C_\varepsilon = 17.29$$

(III): The modified Gatski & Speziale non-linear model (Lakehal and Thiele, Comp. Fluids, 2001) – GSLT –

$$\langle u_i' u_j' \rangle \; = \frac{2}{3}\,\delta_{ij}\langle k\rangle - 2\,\nu_t\,\langle S_{ij}\rangle + \nu_t^* \langle k\rangle / \langle \varepsilon\rangle \,[\Sigma\, C_n(\langle T_{ij}\rangle)_n]$$

$$\nu_t^* = C_\mu^* \langle k\rangle^2 / \langle \varepsilon \rangle$$

as compared to the GS original model:

- sensitizing the production-to-disspation ratio $\mathcal{P}/\varepsilon$ appearing in the C_1 and C_2 coefficients (non-linear group) to the vorticity and strain rate 2nd invariants $S = \frac{k}{\varepsilon}\sqrt{1/2 S_{lk} S_{kl}}$ and $\Omega = \frac{k}{\varepsilon}\sqrt{1/2 \Omega_{kl}\Omega_{kl}}$:

$$\frac{\mathcal{P}}{\varepsilon} = \frac{S^2}{4.8 + \; 1.3\, Max(S,\Omega)}$$

  instead of $\mathcal{P}/\varepsilon = (C_{\varepsilon 2} - 1)/(C_{\varepsilon 1} - 1) \approx 1.89$

- changing the regularization of C_μ^* using a 4th order Padé approximation:

$$\eta^4 \approx \frac{\eta^4}{1+\eta^4}; \qquad \text{or} \qquad \eta^2 \approx \frac{\eta^2}{(1+\eta^4)^{1/2}}$$

instead of (2nd order)

$$\eta^2 \approx \frac{\eta^2}{1+\eta^2}$$

giving rise to

$$C_\mu = \frac{\phi(1+\eta^4)}{3+\eta^4(1+6\xi^2)-2\eta^2+6\xi^2}$$

instead of

$$C_\mu = \frac{\phi}{(3-2\eta^2+6\xi^2)}$$

where

$$\eta^2 = (C_1 S)^2/4;\ \xi^2 = (C_2\Omega)^2/4;\ \phi \approx \frac{1.46}{3.4+\mathcal{P}/\varepsilon - 1}$$

The Simulation Method

1 The solver

- FAST-3D: a Strongly conservative finite-volume code,
- Structured, body-fitted coordinates, centered grid arrangement,
- Multi-grid (FAS), multi-block,
- Convection: 2nd-order schemes HLPA and QUICK,
- Pressure correction: SIMPLE/SIMPLEC algorithms,
- Momentum interpolation (Rhie and Chow),
- Solution: Stone's strongly implicit procedure (SIP),
- High level of vectorization ($> 95\%$).

2 The grid and computational domain

- The computational domain is composed of 03 blocks, connected into a main block.
- only half of the domain was considered.
- dimensions and details are reported in Figure 1.
- the outer-flow grid contains $107 \times 22 \times 72$ nodes for half the domain.
- half hole covered by $15 \times 8 \times 40$ nodes, and half plenum covered by $47 \times 22 \times 32$ nodes.

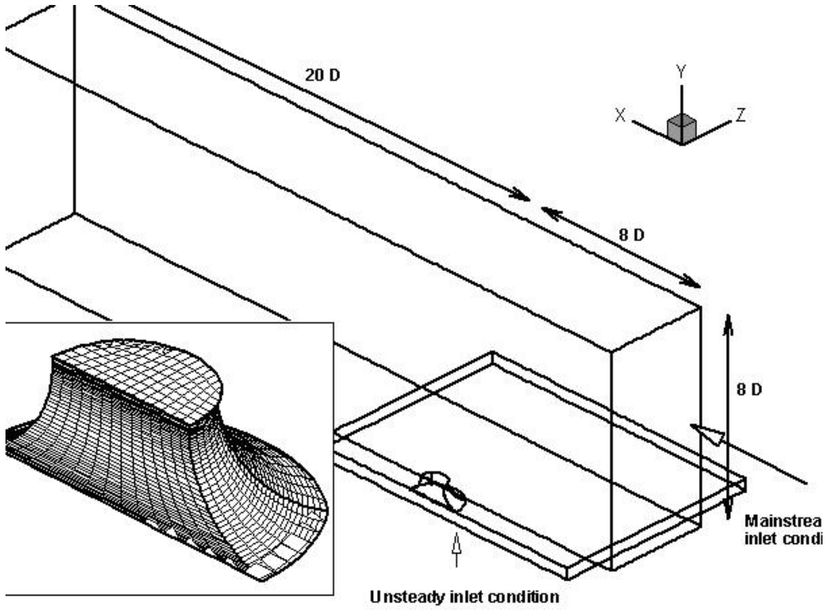

Figure 1: Computational Domain and Hole Grid Details

- total number of nodes: 282.000; 75% active.

3 The boundary conditions

- the diaphragm is simulated using an unsteady BC imposed at the bottom of the cavity.
- injection velocity $V_{inj} = \overline{V}\, cos(2\,\pi\, f\, t)$; $f = 150Hz$.
- inlet BC's are set according to the file 'UpstreamBC.dat', including turbulent kinetic energy (TKE).
- the rate of dissipation of TKE is set assuming equilibrium $\mathcal{P} = \varepsilon = -\overline{u'v'}\frac{\partial \overline{u}}{\partial y}$.
- TKE and ε at the jet exit are set assuming a turbulence intensity of 2% and a ratio of eddy-to-molecular viscosity of 20.
- the time step is set to $\Delta t = 1.825E - 5$ s, i.e. 360 iterations per cycle.