LES OF JETS IN CROSSFLOW
AND ITS APPLICATION TO GAS TURBINE BURNERS

Jörg U. Schlüter
C. Angelberger, T. Schönfeld, T. Poinsot
CFD-team, CERFACS
42 Avenue G. Coriolis, 31057 Toulouse Cedex, France

ABSTRACT

LES calculations of jets in cross flow with low spatial discretisation are performed. After elimininating several sources of inaccuracy concerning the point distribution in the mesh, a reproduction of measurements in terms of velocity and scalar fields is obtained. The results of the investigation of the jet in cross flow are used to determine the accuracy required of an LES calculation of a gas turbine burner.

INTRODUCTION

Motivation

The jet in cross flow (JICF) is a common flow configuration in numerous technical applications. Its simple physical design and its good mixing behaviour make it a well suited flow for combustors. A combustor currently under investigation at CERFACS is a gas turbine burner of Siemens (Fig. 1). Here, the JICF can be found on the surface of swirl inducing vanes.

Unsteady flow phenomena, mixing behaviour and combustion processes of the Siemens gas turbine burner are analysed using LES. The application of LES to industrial purposes requires compromises. This led to an investigation of the JICF with about 90000 Mesh points. Such a low resolution was chosen, because the final application in the gas turbine burner has numerous injectors. This means, we have to restrict ourselves to coarse discretizations of the jets to obtain a mesh, where a computation of the complete burner is feasable. The present JICF computations must demonstrate the feasibility and accuracy of LES computations to reproduce flow phenomena and mixing behaviour of a complex flow configuration.

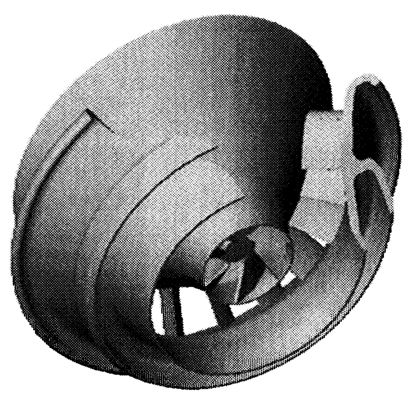

Figure 1: The Siemens V64.3A.HR gas turbine burner currently under investigation at CERFACS

The Jet in Cross Flow

The jet in crossflow has attracted some attention in fluid mechanic research (among others: Chassaing et al., 1974, Moussa et al., 1977, Broadwell & Breidenthal, 1984, Fric & Roshko, 1994, Kelso et al., 1996). A systematic analysis of the large scale coherent structures, which characterize the JICF, began in the 1970's. Up to now, four different types of coherent structures are determined in the JICF (Fig. 2). All of these vortex structures show a strong unsteady behavior.

Reynolds-Averaged-Navier-Stokes (RANS) computations usually have difficulties to reproduce these unsteady vortex structures. An unsteady LES approach is a much more promising way to predict the behaviour of a JICF. Recently Yuan (1997, 1998) made an LES calculation of a JICF. He calculated a JICF with a small Reynolds number (2100) and used 1.3 million

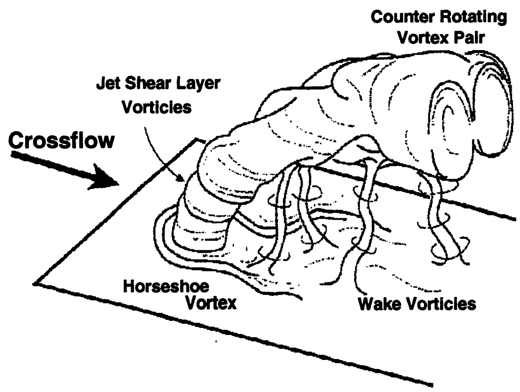

Figure 2: vortex system of a jet in crossflow (from Fric & Roshko, 1994)

mesh points. The high number of points and the low Reynolds number makes this calculation inappropriate to apply it to industrial purposes.

Various experimental investigations on JICF have been reported. Three cases have been chosen to be reproduced. One case is a series of experiments carried out by Andreopoulos and Rodi (1982, 1984, 1985). Their hot-wire measurements of JICF with a Reynolds number Re=81000, based on jet bulk velocity and the jet diameter, and a velocity ratio of $r = 2$ allow a comparison of the momentum field with our calculations.

Because the work on the gas turbine burner focuses on mixing, we have chosen as a second test case the experiments by Smith and Mungal (1993, 1996a, 1996b, 1998). Their LIF measurements of a JICF provide exact measurements of the mixing behaviour in a JICF. Here, we have chosen to reproduce their measurement of a JICF with a Re=16400 and a velocity ratio $r = 5$.

An additional problem in the gas turbine concerns the interaction between adjacent jets. Hence, the third testcase is a measurement of a twin jet from Toy et al. (1993). He measured velocity profiles in the far field of the jets with Re=31800 and a velocity ratio $r = 6$. The most interesting point he observed was the jets connected very soon and formed only one counterrotating vortex pair.

MATHEMATICAL FORMULATION

Gouverning Equations

The basic idea of LES is to resolve the larger scales of motion of the turbulence while approximating the smaller ones. To achieve this, a filter is applied to the continuity equation and the transport equations of momentum, energy and species. For reacting flows, the use of the Favre filtering is helpful, defined as:

$$\bar{\rho}\tilde{Q} = \overline{\rho Q} = \int_{-\infty}^{+\infty} \rho Q(x,t)\, G(x - x')\, dx' \qquad (1)$$

leading to the following equations for momentum u_i and species Y_i:

- *momentum ($j = 1 \ldots 3$)*

$$\frac{\partial \bar{\rho}\tilde{u}_i}{\partial t} + \frac{\partial \bar{\rho}\tilde{u}_i\tilde{u}_j}{\partial x_i} + \frac{\partial \bar{p}}{\partial x_i} = \frac{\overline{\partial \tau_{ij}}}{\partial x_j} + \frac{\partial T_{ij}}{\partial x_j} \qquad (2)$$

- *species mass fraction ($k = 1 \ldots N$)*

$$\frac{\partial \bar{\rho}\widetilde{Y_k}}{\partial t} + \frac{\partial \bar{\rho}\tilde{u}_i\widetilde{Y_k}}{\partial x_i} = \frac{\partial}{\partial x_i}\left(\bar{\rho}D_k \frac{\partial \widetilde{Y_k}}{\partial x_i}\right) + \widetilde{\omega_k} + \frac{\partial \Psi_{kj}}{\partial x_j} \qquad (3)$$

The terms T_{ij} and Ψ_{ik} are resulting from the convective terms $\widetilde{u_i u_j}$ and $\widetilde{Y_k u_i}$, which are split into a resolved part on the left side of the equation, directly delivered by the LES calculation, and an unresolved part, which need to be modeled.

Subgrid scale models

We used an eddy viscosity approach for the subgrid scales:

$$T_{ij} = 2\nu_t \overline{S}_{ij} + \frac{1}{3} T_{ll}\delta_{ij} \qquad (4)$$

with

$$\overline{S}_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \qquad (5)$$

Although the eddy viscosity approach is not valid for free turbulence, its simplicity allows faster computations and by this a higher spatial discretization and an increase of the resolved part of the spectrum.

We used two models to determine the eddy viscosity ν_t. The first one is the Standard Smagorinky Model (Smagorinsky, 1963):

$$\nu_t = (C_1 \Delta x)^2 \sqrt{2\tilde{S}_{ij}\tilde{S}_{ij}} \qquad (6)$$

with $C_1 = 0.18$, which has the advantage of simplicity and speed.

The second model is the Filtered Smagorinsky model (Ducros et al., 1997) defined on a high-pass filter HP:

$$\nu_t = (C_2 \Delta x)^2 \sqrt{2HP(\tilde{S}_{ij})HP(\tilde{S}_{ij})} \qquad (7)$$

and a constant $C_2 = 0.37$. This model offers a better behaviour in transitional flows and was optimized to work in wall boundary layers.

Subgrid mixing is modeled by an eddy diffusity approach with a turbulent diffusity based on the turbulent

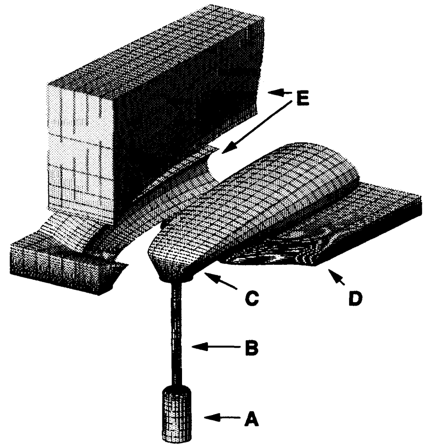

Figure 3: exploded view of grid blocks

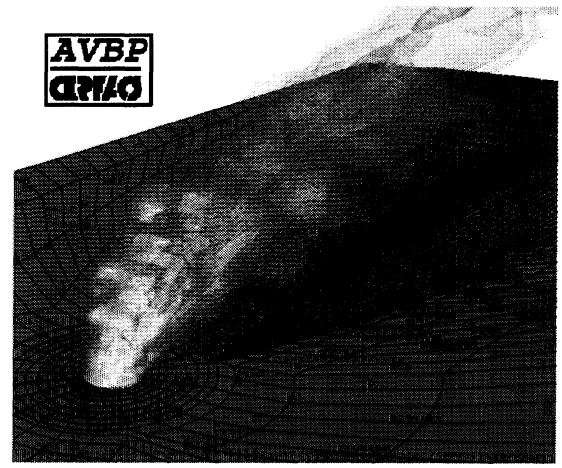

Figure 4: smoke vizualization of the testcase of Andreopoulos & Rodi, $r = 2$, Re=81000

viscosity ν_t of the subgrid stress model and a constant Schmidt number:

$$\Psi_j = \frac{\nu_t}{\text{Sch}} \frac{\partial \tilde{\xi}}{\partial x_j} \tag{8}$$

The AVBP Code

For our LES calculations we used the AVBP code developed at CERFACS and the Oxford University, based on the generic software library COUPL. It is a code which uses unstructured meshes and is able to run on parallel machines. It is second-order in space and third-order in time.

SPATIAL DISCRETISATION

The spatial discretisation of the flow has been made by structured meshes. Although the code allowed unstructured meshes, structured meshes provide better control control of the point distribution in the flow. Fig. 3 shows an exploded view of the mesh. At the bottom is the plenum chamber of the jet (A) passing over into a pipe (B). The jet nozzle is at the upper end of the pipe. An O-grid is put in the jet trajectory and the surrounding of the nozzle (C). A block behind the nozzle (D) describes the flow downstream of the nozzle and several coarse blocks (E) are put around the jet trajectory to mesh the nearly undisturbed outer flow.

Meshing the jet pipe flow is important. Andreopouplos (1982) pointed out, that there is an interaction between the crossflow and the jetflow in the pipe. In the examined case ($r = 2$) the crossflow affects the pipeflow around 2 diameter upstream the pipe. This can be explained by the investigations of Perry and Kelso (1993), who proved the existence of a recirculation zone in the jet pipe at the leading edge of the jet nozzle.

In the LES computation of Yuan (1997) the influence of an extension of the mesh into the jet pipe was examined. He found out, that the flow behaviour is much better represented with a mesh extension. Unfortunately, his mesh extension is only one diameter D long.

We found out, that a simple extension might not be sufficient. Our first calculations have been carried out with a $3D$ long pipe leading to the jet nozzle and the velocity profile u was imposed at the entry of the pipe. This led to strong pressure oscillations in the pipe. As a numerical artefact, the pipe acts as a Helmholtz resonator, because the inlet below the wall forms a velocity node. The frequency of the oscillations are determined by the length of the pipe. The jet shear layer roll-up locks into the the oscillations and the jet acts like a forced jet now. Kelso (1996) found, that the jet trajectory is affected by the forcing. In our computations the trajectory is higher than in the case where the oscillations are supressed.

To avoid pressure oscillations we use a combination of two countermeasures. The first one is to impose the massflux ρu instead of u as a boundary condition to change the acoustic wave reflections at the inlet. The second is to extend the jet pipe mesh into the plenum chamber in front of the jet pipe (Block A in fig. 3). The sudden change in diameter between jet pipe and plenum chamber makes it more difficult for the system pipe/plenum chamber to act as a resonator. This is expensive, but gives as a by-product more certainty on the jet velocity profile.

Furthermore we found, that a refinement of the mesh in the low pressure region downstream of the jet nozzle is necessary. It influences the jet trajectory and we

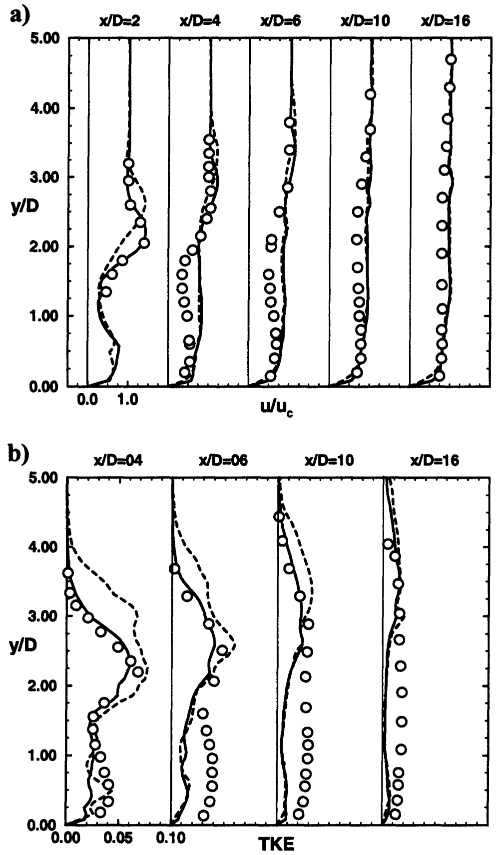

Figure 5: comparison of the a) momentum and b) turbulent kinetic energies on the centerline of the jet, black circles: measured, solid line: LES Filtered Smag. Model, dashed line: LES Standard Smag. Model, averaging time 0.25s

obtain a higher trajectory with an insufficent mesh.

COMPARISON OF EXPERIMENT AND LES

Testcase of Andreopoulos & Rodi

Fig. 4 shows a smoke vizualization of our LES computation of the case of Andreopoulos & Rodi. The u component of velocity is compared in Fig. 5a for different positions downstream on the jet centerline. Regarding the profile at $x/D = 2$ the measurements and the LES computation with the Filtered Smagorinsky model agree well. The LES computation using the Standard Smagorinsky model shows a wrong trajectory, the velocity maximum is $0.4D$ too high, but the right order of magnitude. As mentioned above, the Filtered Smagorinsky model was optimized for boundary layers. Because of that, the oncoming wall boundary layer is better described and the momentum ratio close to the wall is better predicted. This has an influence on the jet trajectory. The u velocity profiles downstream show, that the Filtered model has advantages over the Standard model, although the trajectory is slightly too high.

Fig. 5b shows a comparison of turbulent kinetic energies (TKE) $k^2 = \overline{u'^2} + \overline{v'^2} + \overline{w'^2}$. In the diagram the TKE of the LES calculations represent only the TKE of the resolved part. The profile at $x/D = 4$ shows a good agreement of the LES calculation with the Filtered model, although it can be seen, that the trajectory is slightly too high. The Standard model shows a too high trajectory as well and overestimates the TKE. Obviously the Standard model is not dissipating enough turbulent energy. The profiles downstream show a quite good agreement far off the wall. Close to the wall, the mesh is not fine enough to capture the small scale turbulence of the wall boundary layer, so that the portion of the TKE resolved by the LES approach is smaller.

Testcase of Smith & Mungal

The smoke visualization of the case of Smith & Mungal is shown on Fig 6. Because of the higher velocity ratio, the jet lifts off the wall more clearly and it can be supposed, that the wall boundary layer plays a minor role. The mixing behaviour of a passive scalar is compared in Fig. 7. Despite the simple subgrid mixing model, all profiles of the LES computation with the Filtered model show a good agreement. There is a little deviation of the trajectory at $x/D = 10$.

The influence of the choice of the model is smaller, due to the weaker influence of the wall boundary layer. But even here, it can be seen, that the jet trajectory of the calculation using the Standard Smagorinsky model is too high.

Testcase of Toy

In Fig. 8 the smoke visualization of our LES calculation with the Filtered model of the testcase from Toy. Velocity profiles (Fig. 9) show deviations between measurement and LES computation. Again the trajectory in the LES calculation is too high. The flow on the centerline between the two jets where the profiles are taken, is very unsteady, which might explain the deviations. Nevertheless the major flow characteristic, the merging of the two jets and the development of a single counterrotating vortex pair is reproduced well.

LES OF THE GAS TURBINE BURNER

The lack of measurements in the real geometry to determine the accuracy were compensated by the investigation of the JICF. The similar flow conditions of the

Figure 6: smoke vizualization of the testcase of, Smith & Mungal, $r = 5$, Re=16400

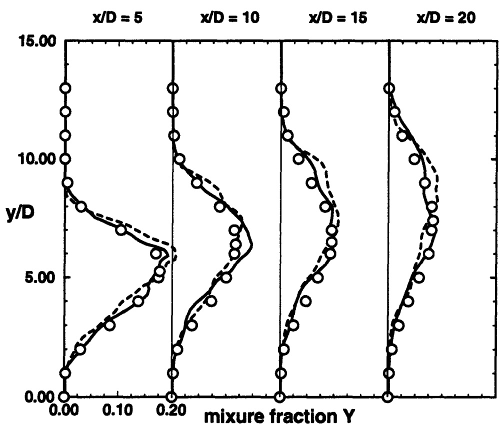

Figure 7: comparison of the scalar field on the centerline of the jet, black circles: measured, solid line: LES Filtered Smag. Model, dashed line: LES Standard Smag. Model, averaging time 0.30s

Figure 8: smoke vizualization of the testcase of Toy, $r = 6$, Re=31800

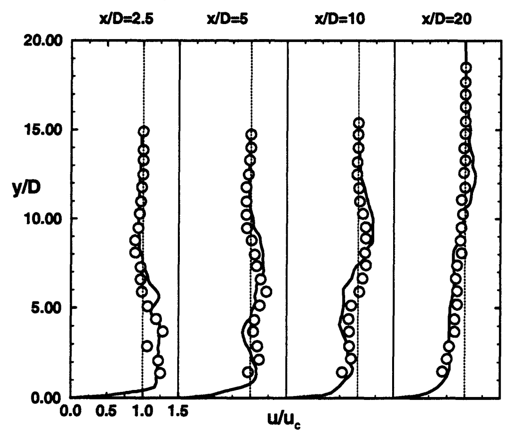

Figure 9: comparison of the momentum field on the centerline of the two jets, black circles: measured, solid line: LES, averaging time 0.235s

JICF and the fuel injectors on the vane in the industrial application gives us confidence, that the computations of the burner geometry predicts the flow behaviour with enough precision to make statements about mixedness and vortex formations inside the burner. A full analysis of our LES computation of the Siemens V64.3A.HR gas turbine burner would be beyond the scope of this article.

Our first approach in examining the burner was to have a look at a pair of injectors on the vane (Fig. 10). This gave us the possibility to determine mass flux rates through the injectors and to investigate separation events on the vane, which interact with the fuel injecting jets. The computation of a full vane in homogeneous surrounding flow (Fig. 11) gave us insight about interactions between adjacent jets, which lead to unmixedness. The unmixedness in the computed domain can be determined by PDFs.

The final step, the computation of a 20° segment of the burner, is planned in the future. Experiments on the burner, carried out at the University of Karlsruhe, will provide possibilities to compare our LES computations with LDV and LIF measurements from the burner exit.

CONCLUSIONS

Our LES computations have proven the ability of reproducing the main features of JICFs with reasonably coarse grids.

The necessarity of including the jet pipe and the

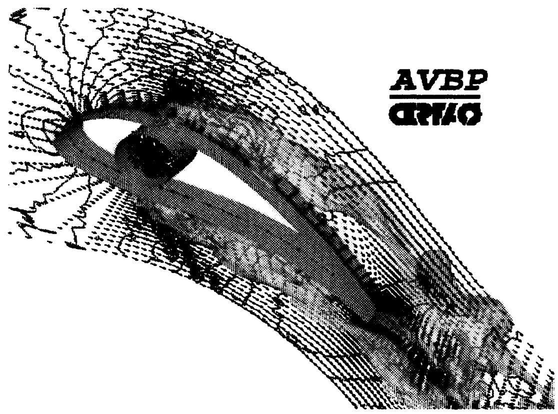

Figure 10: fuel injection on a part of the vane, isosurface: mixture fraction, isocontours: pressure, black arrows: velocity vectors

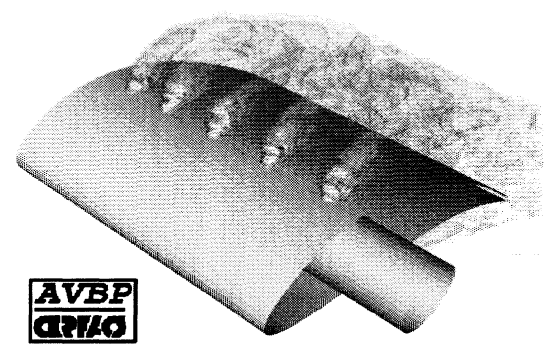

Figure 11: vizualization of fuel injection on the vane

plenum chamber to the computed domain arose. The proper description of the wall boundary layer is required to obtain the correct jet trajectory. The low pressure region downstream of the nozzle influences the jet trajectory as well and needs a high spatial discretization.

The subgrid turbulence model influences the jet trajectory. As long as the flow is strongly affected by the wall boundary layer, the Filtered Smagorinsky model showed better results. The nature of the JICF, to be determined by large scale motions, makes it possible for the LES approach to obtain results with a good agreement.

The capability of applying LES calculation to obtain informations about momentum field and mixing in a gas turbine burner has been shown.

REFERENCES

J. Andreopoulos, 1982, "Measurements in a jet-pipe flow issuing perpendiculary into a cross stream", *J. of Fluids Eng.*, Vol. 104, pp. 493-499

J. Andreopoulos and W. Rodi, 1984, "Experimental investigation of jets in a crossflow",*J. of Fluid Mech.*, Vol. 138, pp 93-127

J. Andreopoulos, 1985, "On the structure of jets in a crossflow", *J. of Fluid Mech.*, Vol. 157, pp 163-197

J. E. Broadwell and R. E. Breidenthal, 1984, "Structure and mixing of a transverse jet in incompressible flow", *J. of Fluid Mech.*, Vol. 148, pp 405-412

P. Chassaing, J. George, A. Claria and F. Sananes, 1974, "Physical characteristcis of subsonic jets in a cross-stream", *J. of Fluid Mech.*, Vol. 62, pp 41-64

F. Ducros, F. Nicoud and T. Schönfeld, "Large Eddy Simulations of Compressible Flows on Hybrid Meshes", *Turbulent Shear Flows 11*, Grenoble, France

T. F. Fric and A. Roshko, 1994, "Vortical structure in the wake of a transverse jet", *J. of Fluid Mech.*, Vol. 279, pp 1-47

R. M. Kelso and T.T. Lim and A. E. Perry, 1996, "An experimental study of round jets in crossflow", *J. of Fluid Mech.*, Vol. 306, pp 111-144

Z. M. Moussa and John W. Trishka and S. Eskinazi, 1977, "The near field in the mixing of a round jet with a cross stream", *J. of Fluid Mech.*, Vol. 80, pp 49-80

J. Smagorinsky, 1963, "General circulation experiments with the primitive equations, i, the basic experiment", *Mon. Weather Rev.*, Vol. 92

S. H. Smith, A. Lozano, M. G. Mungal and R.K. Hanson, 1993, "Scalar Mixing in the Subsonic Jet in Cross Flow", in *Computational and Experimental Assessment of Jets in Cross Flow*, AGARD CP-534

S. H. Smith, E. F. Hasselbrink, M. G. Mungal and R. K. Hanson, 1996, "The Scalar Concentration Field of the Axisymmetric Jet in CrossFlow", *AIAA-Paper 96-0198*, 34th Aerospace Sciences Meeting and Exhibit, Reno, Nevada

S. H. Smith, 1996, "The Scalar Concentration Field of the Axisymmetric Jet in CrossFlow", *PhD Thesis, Stanford University*, HTGL Report No. T-328

S. H. Smith and M. G. Mungal, 1998, "Mixing, structure and scaling of the jet in crossflow", *J. of Fluid Mech.*, Vol. 357, pp 83-122

N. Toy, E. Savory and S. McCusker, 1993, "The Interaction Region associated with Twin Jets and a Normal Cross Flow", in *Computational and Experimental Assessment of Jets in Cross Flow*, AGARD CP-534

L. L. Yuan, 1997, "Large eddy simulations of a jet in crossflow", *PhD Thesis, Stanford University*

L. L. Yuan and R. L. Street and J. H. Ferziger, 1998, "Large eddy simulations of a round jet in crossflow", *J. of Fluid Mech.*