ABSTRACT

The present paper investigates the turbulent mixing of the heavy precursor gas TEOS in an hot accelerated transonic air co-flow. A mass flux of \( m_{\text{TEOS}} = 0.5 \times 10^{-6} \) TEOS premixed with \( m_{\text{N}_2} = 0.32 \times 10^{-6} \) nitrogen is injected in an air co-flow with a mass flux of \( m_{\text{gas,flow}} = 100 \times 10^{-6} \) at \( Ma_{\text{co-flow}} = 0.66 \) and \( T_{\text{co-flow}} = 1200 \text{K} \) and afterwards accelerated to supersonic speed. High frequency, non equilibrium turbulence phenomena dominated by pronounced free shear layers are expected to significantly affect the mixing process. Therefore, a large-eddy simulation (LES) which is capable of capturing the turbulent behavior on the relevant scales is used to simulate the mixing process. An efficient multi-species LES method based on the MILES technique with an explicit 5-stage Runge-Kutta scheme is applied. The results of the simulations show that the wake is dominated by strong vortex shedding from the blunt injector trailing edge. The mixing process is impeded by these primary vortices.

INTRODUCTION

Nanoparticles are broadly used in industry and science. The preferred current methods to synthesize these particles are flame synthesis and hot-wall synthesis. Studies indicate that high heating and quenching rates as well as a homogeneous flow field are the most important parameters for achieving a narrow size distribution and low aggregation of the particles. (Schidl et al., 1999)

The project “Gasdynamically Induced Nanoparticles” pursues a new approach to realize an improved production process for nanoparticles. A precursor gas is mixed with an accelerated air flow in a Laval nozzle at transonic co-flow speed. The mixed flow is accelerated to supersonic flow speed and afterwards quasi-instantaneously heated by a shock wave. In the reaction chamber the air precursor mixture reacts and nanoparticles are generated. A second Laval nozzle terminates the reaction by accelerating and hence cooling down the flow. Downstream of the second nozzle the total enthalpy of the flow is reduced by injecting water in a quenching system. The schematic of the facility is depicted in figure 1.

As part of this project the mixture of the precursor gas in an accelerated hot air flow is numerically investigated at the Institute of Aerodynamics. A spatially and temporally homogeneous mixture is required to achieve particles of a narrow size distribution and high quality. The high velocity and short mixing time make a rapid mixing necessary. The objectives of the numerical simulations are an improved understanding of the mixing and the parameters defining this process. From this analysis an improvement of the mixing homogeneity and a reduction of the mixing length is to be derived. In addition, based on the ability to predict the mixing process, the impact of changing any part of the overall injection setup on the nanoparticle quality is to be estimated.

The mixing of a gas in a co-flow has been studied for many years due to its relevance for many technical applications. Especially the injection of a propellant in combustion chambers is a frequent application. The research most similar to this project was conducted for RAMJET and SCRAMJET engines. Gerlinger et al. (2008), for example, studied a Mach number \( Ma = 2.0 \) injection of a propellant for an injector which induced longitudinal vortices into the flow. A significant improvement of the mixing quality due to the induced longitudinal vortices was shown. While this study was conducted using a Reynolds-averaged Navier-Stokes (RANS) method Berglund and Fureby (2007) performed an LES for mixing and combustion in a SCRAMJET combustion chamber which showed good agreement with experiments. Many further studies numerically and experimentally investigated similar cases.

While the overall topic of this paper is similar to those studies in the sense that a gas is mixed in a co-flow, there are significant differences to this study. First of all, the work presented in this paper focuses on the mixture of a gas in a co-flow with a negative pressure gradient. This acceleration of the co-flow has an effect on the mixture, which was not studied in the past. A further difference is that the co-flow possesses a higher velocity than the injected gas, whereas in most jet-in-a-co-flow studies the jet had a higher or at least a similar speed as the co-flow.

This paper is organized as follows. After a description of the flow problem governing equations and the numerical method are discussed, the validation of the multi-species mixing by simulation of a propane jet is presented. Subsequently, the grid configuration and some computational details are shown followed by a discussion of the simulation results and the conclusion.

DESCRIPTION OF THE FLOW PROBLEM

The studied problem of the present paper is a Laval nozzle with a width of 15mm and an overall length of about 200mm. The throat has an area of 15mm times 6mm. The total pressure of the main flow is \( p_{0,\text{co-flow}} = 100 \text{bar} \) and...
the total temperature is $T_{0, inflow} = 1300K$. An injector with an airfoil like shape, a length of 20mm and a blunt trailing edge thickness of $D_{inj} = 1.5mm$ injects the precursor gas TEOS at transonic flow speed. It has six injection holes with a diameter of 0.5mm each. The injected mass flux is about 0.8% of the main flow and has a total temperature of $T_{0, inj} = 650K$. The injection flow speed is $u_{0, inj} = 100m/s$, which is approximately 1/3 of the velocity of the main flow. Its Reynolds number is $Re = 16000$. The injected gas mixture of Nitrogen and the precursor TEOS is 3 to 7 times denser than air depending on the mixing ratio of TEOS and Nitrogen.

The x-coordinate of the cartesian frame of reference defines the streamwise direction. The origin of the coordinate system is in the center of the nozzle throat such that the exit of the injector, which is located in the subsonic part of the nozzle is described by a negative x-coordinate. The y-coordinate defines the vertical direction and the spanwise direction i.e., the direction normal to the sidewalls, is denoted by the y-coordinate.

From the illustration of the TEOS concentration in (Fig. 2) it is clear that strong vortices being shed from the blunt injector trailing edge dominate the wake and the mixing area. The mixing strongly depends on the macroscopic turbulence structure which develops in the wake.

GOVERNING EQUATIONS

The Navier-Stokes equations of a multi-species fluid, i.e., the conservation equations for the partial density $\rho_n$ of $N-1$ species with a total number of $N$ species, are included and approximately solved to determine the flow physics of this problem (Peyret and Taylor, (1983)). The dimensionless conservative tensor notation is

$$\frac{\partial \mathbf{Q}}{\partial t} + (\mathbf{F}^C - \mathbf{F}^D)_{ik} = 0,$$

with $\mathbf{Q}$ representing the vector of the conservative variables

$$\mathbf{Q} = [\rho_n, \rho, \rho u_n, \rho E]^T,$$

$\mathbf{F}^C_{ik}$ denoting the vector of the convective, and $\mathbf{F}^D_{ik}$ being the vector of the diffusive fluxes

$$\mathbf{F}^C_{ik} - \mathbf{F}^D_{ik} = \begin{pmatrix} \rho_n u_i \\ \rho u_i \\ \rho u_i u_j + \rho \delta_{ij} \\ \rho u_i (\rho E + p) \end{pmatrix} + \frac{1}{Re} \begin{pmatrix} J_{n\beta} \\ 0 \\ \sigma_{\alpha\beta} + q_{\beta} \\ u_i \sigma_{\alpha\beta} + q_{\beta} \end{pmatrix},$$

For a Newtonian fluid the stress tensor $\sigma_{\alpha\beta}$ can be written as a function of the strain rate tensor $S_{\alpha\beta}$

$$\sigma_{\alpha\beta} = -2\nu \left( S_{\alpha\beta} - \frac{1}{3} S \delta_{\alpha\beta} \right)$$

with

$$S_{\alpha\beta} = \frac{1}{2} (u_{\alpha,\beta} + u_{\beta,\alpha}).$$

The mass diffusion $j_{n\beta}$ for a dilute mixture can be described by Fick’s law

$$j_{n\beta} = \frac{\rho D_n}{Sc_0} Y_{n,\beta},$$

where $Y_n$ is the mass fraction of the species $n$, $D_n$ is the diffusion coefficient computed via mixing rules from the binary diffusion coefficients, $Sc_0 = \nu_0 / D_0$ is the Schmidt number, and the subscript 0 indicates the reference state of the mixture. With Fourier’ law of heat conduction the heat flux $q_{\beta}$ is proportional to the negative gradient of the temperature via

$$q_{\beta} = -\left( \frac{k}{\Pr (\gamma_0 - 1)} \right) T_{\beta}.$$
fraction of TEOS was taken as indicator for areas causing stability problems in this study. The artificial diffusivity term which is added to the physical diffusivity reads

$$D_{art} = c_{art} \left( \Delta x \right)^4 \frac{\partial^3 Y_{TEOS}}{\partial x^3}$$

(12)

where \(c_{art}\) is a constant to adjust the value of the artificial diffusivity \(D_{art}\).

### VALIDATION

The described numerical method has been validated by simulating a propane (C\(_3\)H\(_8\)) jet in an air co-flow, and comparing the numerical data with experimental results of Schefer and Dibble (2001). Laser Rayleigh scattering was used in their experiments to generate time- and space-resolved measurements of the species mixture field. Earlier investigations by Dibble et al. (1987) provide corresponding velocity data.

The jet at a Reynolds number of \(R_{jet} = 68000\) based on the jet diameter and a jet bulk velocity of \(u_{jet} = 53.2\) \(m/s\) is injected into an air co-flow at a velocity ratio of \(u_{co}/u_{jet} = 5.75\). The simulation is done on a structured grid with 13 blocks and 3.5 million grid points. The smallest grid spacing in the streamwise direction is \(\Delta x_{min} = 0.042R_{jet}\), where \(R_{jet}\) is the radius of the jet. The smallest mesh spacing in the shear layer is \(\Delta x_{min} = 0.042R_{jet}\), where \(R_{jet}\) is the radius of the jet. The smallest mesh spacing in the streamwise direction is \(\Delta x_{min} = 0.13R_{jet}\).

Figure 3 shows the time-averaged mixture fraction \(f\). On the centerline the maximum value of unity is preserved for twelve jet radii. This point marks the end of the potential core. The instantaneous mixture fraction is shown in figure 4. Instabilities develop shortly downstream of the inlet. They are triggered by an inflow forcing which induces randomly updated vortical rings into the flow as suggested by Bogey et al. (2002). Downstream of \(x_{inlet} = 12\) the transition to a fully turbulent state is visible.

The experimental data provided by Schefer and Dibble (2001) allows a detailed comparison with the prediction of the turbulent mixing by the jet simulation. The centerline mixture fraction decay for non-reacting turbulent jets can be correlated with the distance from the virtual origin \(x_{0,1}\) using a linear function

$$\frac{1}{f_{cl}} = C_1 \left( \frac{x - x_{0,1}}{R} \right),$$

(13)

where \(C_1\) is the centerline decay constant and \(x_{0,1}\) is a virtual origin, which is defined by the location of the onset of an approximately constant slope. The reciprocal mean mixture fraction \(f\) along the centerline is plotted in Fig. 5. It is evident that the centerline mixture fraction asymptotically approaches the similarity solution defined by equation (13). The constant \(C_1\) is 0.09. This value agrees well with the experimental results given in Table 1.

The spreading rate of the propane jet is characterized by the mixture fraction half-radius \(R_{fb}\) which is defined as the radial location where the mixture fraction scalar \(f\) is half of the value on the centerline. The definition reads

$$\frac{R_{fb}}{R} = C_2 \left( \frac{x - x_{0,2}}{R} \right),$$

(14)

where \(C_2\) is the centerline decay constant and \(x_{0,2}\) is a virtual origin. The present LES predicts the spreading as \(C_2 = 0.06\) (Fig. 5). The spreading rate is compared with the experimental findings in Table 1 and again, the agreement is excellent.

More detailed information on this validation case can be found in Renze et al. (2008).

### COMPUTATIONAL DETAILS AND CONSTRAINTS

The simulations have been conducted on two block structured grids, one for the first part of the nozzle (grid A), covering the region upstream of \(x \approx 3.5mm\), and the other mesh encompasses the part downstream of this point (grid B). Only one out of the six injection holes is taken into account. That is, the spanwise diameter is reduced from 15\(mm\) to 1.8\(mm\), i.e., the distance between two injection holes, and the spanwise boundaries are considered periodic. The upper and lower wall boundary layers are neglected such that an Euler boundary condition is introduced.

These latter measures for the vertical and spanwise direction are taken to significantly reduce computational costs. The purpose of the splitting in two smaller instead of one large grid also is efficiency since downstream of the throat the mesh can be coarsened to reduce the number of grid
Figure 6: Distribution of the velocity components of the turbulent flow field in the injection pipe points. The injector geometry, in particular the injection hole, makes a complex grid structure necessary. This results in a high number of grid points normal to the nozzle-centerline. The supersonic nature of the flow downstream the nozzle throat and the omitted boundary layers allow a simple coupling of two simulations, therefore the second nozzle part can be meshed using a simple H-grid. This saves an immense amount of grid points, resulting in a reduced computational time.

The aforementioned block structured grids consist of 24 blocks. Grid A has 24 million grid points (Fig. 7), grid B 50 million points (Fig. 8). The minimum grid spacing is reached at the upper and lower edges of the injector’s blunt trailing edge with $\Delta x_{\text{min}} = 5 \times 10^{-6} \text{m}$, $\Delta y_{\text{min}} = 1 \times 10^{-6} \text{m}$, $\Delta z_{\text{min}} = 1 \times 10^{-5} \text{m}$, which corresponds to $y^+ = 1$. Near the centerline the grid stretches to $\Delta x^+ = 60$, $\Delta y^+ = \Delta z^+ = 30$.

The flow through the injection pipe at a Reynolds number of almost $Re = 16000$ is considered to be fully turbulent. Therefore, grid A includes a pipe grid with about 2.5 million grid points that is used to generate a turbulent pipe flow. A periodic boundary condition for the velocity field is used to generate a turbulent flow in the injection pipe. Figure 6 shows the instantaneous flow field in the injection pipe which is connected via a slicing method with the main flow (Rütten et al. 2001).

Grid A and grid B are connected using a supersonic outflow boundary condition. The instantaneous flow data is stored and then interpolated to fit the inflow plane of grid B. An expanded supersonic inflow condition allows for the resulting interpolated data. The quadruple time-step of grid A is used for the simulation of grid B.

RESULTS AND DISCUSSION

Dominant Flow Structures
The flow in the area downstream of the injector is dominated by strong vortex shedding at the injector’s blunt trailing edge. The observed Strouhal number is $Sr \approx 0.2$. The simulations show these primary vortices to be preserved throughout the whole nozzle. Unfortunately, the observed vortices confine most of the injected TEOS mass, and hence impede any strong mixing process. The resulting temporal TEOS distribution is very inhomogeneous, whereas the time averaged spatial TEOS distribution is acceptable. A reason for this behavior might be the favorable pressure gradient in the Laval-nozzle. In the case of an adverse pressure gradient the vortices would probably break down and the mixing process would be enhanced.

Turbulence Analysis
Figures 9 to 11 show the development of the turbulence structure using the anisotropy-invariant map of Lumley and Newman (1977). The origin of the map defines isotropic turbulence. The other two vertices represent isotropic two-component turbulence (left vertex) and one-component turbulence (right vertex). The curve connecting the origin and
Figure 9: Lumley-Newman map depicting the turbulence anisotropy along line I, the arrow denotes the order in flow direction.

Figure 10: Lumley-Newman map depicting the turbulence anisotropy along line II, the arrow denotes the order in flow direction.

CONCLUSION

The presented results show that the current injector configuration is not able to perform a homogeneous temporal mixture of TEOS with the co-flow. The strong primary vortices shedding from the blunt injector trailing edge confine most of the injected mass flux and the favorable pressure gradient impedes vortex breakdown. The turbulence structure of the wake is two-dimensional which is a further indicator for poor mixing quality.

To generate a more isotropic turbulence and improve the mixing quality it is necessary to generate turbulence in span-

Figure 11: Lumley-Newman map depicting the turbulence anisotropy along line III, the arrow denotes the order from line I to line II.
wise direction. Ramps, similar to those used at SCRAMJET injectors, or other types of vortex generators, could be used to induce longitudinal vortices and improve mixing quality, as observed by Gerlinger et al. (2008). The effectiveness of this approach for an accelerated nozzle flow will be subject of future investigations, as well as other adjustments to nozzle and injector configuration.

ACKNOWLEDGEMENTS

The support of this research by the Deutsche Forschungsgemeinschaft (DFG), via the project “Gasdynamically Induced Nanoparticles”, and by the High Performance Computing Center Stuttgart (HLRS) is gratefully acknowledged.

REFERENCES


