Aeroacoustics of a canonical jet/nozzle configuration

Richard Sandberg, Victoria Suponitsky, Neil Sandham (School of Engineering Sciences, University of Southampton), Jason Beech-Brandt, Tom Edwards (Cray CoE)

In this project, significant computing resources were utilized to perform preliminary direct numerical simulations of fully turbulent high subsonic Mach number jets including the nozzle, allowing for realistic turbulent inflow conditions of the jet. The simulations of the turbulent flow in the nozzle were completed and the feasibility of conducting simulations of the canonical jet/nozzle configuration was established.

Introduction

Aero-engine manufactures have so far been able to considerably reduce jet noise by increasing bypass ratios. Further progress on noise reduction now depends on a detailed understanding of the exact noise generation mechanisms. It is widely recognised that both large-scale structures and fine-scale turbulence contribute to the overall sound radiated from subsonic jets and that here is an additional noise source associated with the flow separation over the sharp corner that occurs in the region near the nozzle exit. However, due to limitations of available computational power, previous simulations of jet noise have not been able to include all possible noise generation mechanisms.

In the current project massive computing resources were employed to perform preliminary direct numerical simulations (DNS) of fully turbulent high subsonic Mach number jets. Crucially, the simulations included the nozzle, allowing for realistic turbulent inflow conditions of the jet, and the computational domain was large enough to directly compute part of the far-field sound. This more complex (compared with previous studies) simulation has been made possible by the recent increase in computer power. With such a direct simulation approach we take into account all possible noise generation mechanisms and capture both hydrodynamic and acoustic fields without invoking any form of modelling. A prerequisite for the very large simulations is the ability to generate fully turbulent and reproducible flow conditions at the nozzle exit. Therefore, precursor simulations of compressible turbulent flow in a pipe, a canonical representation of a nozzle, were also conducted.

The Scientific Achievements

The first three months of the project were devoted to running preparatory simulations in order to establish the set-up (domain size, grid resolution, flow parameters and geometry) for the direct hydrodynamic and noise calculations of a round high-subsonic jet exiting from a fully turbulent pipe. During this period we focused on the following tasks: (i) simulations of the spatially developing turbulent compressible pipe flow; (ii) modifying the code to set up a test case for the canonical nozzle/jet problem; and (iii) code optimization.

Simulations of spatially developing turbulent compressible pipe flow have not, to our knowledge, been computed to date and are a crucial step in the current project. The main goal is to achieve a fully developed turbulent compressible flow close to the pipe exit to have well defined (reproducible) turbulent upstream condition for the jet calculations. From this perspective the following issues are of interest: (i) the effect of the inflow boundary conditions and the length of the pipe required to obtain fully developed turbulent pipe flow (development length); and (ii) the effect of compressibility on the temperature field. The former is needed to specify the length of the nozzle for the full jet calculations, while the latter is related to the correct prescription of the ambient temperature. A prior knowledge of the temperature distribution inside the pipe (in particular near the pipe exit) is necessary for the correct setting of the ambient temperature depending on the type of jet being simulated (isothermal, cold, hot).

In this part of the project both periodic and spatially developing pipe flows were simulated. The periodic pipe calculations with Mach number M=0.4 and Reynolds numbers of Re=4929 and Re=5268 (based on pipe diameter and bulk velocity) were first carried out to validate the code by comparison with the recent incompressible results of Wu&Moin[1]. The data obtained from the periodic calculations were later used to specify mean streamwise velocity inflow profiles and parameters (integral length scale etc.) required for the digital filter turbulence inflow technique in the novel spatial calculations. A large number of spatially developing pipe flow simulations were conducted to determine the required length of the pipe and the appropriate parameters for the turbulence inflow generation to achieve statistically converged turbulence in good agreement with the results from periodic simulations. Figures 1 & 2 show how our present DNS results for the periodic and spatially developing pipes compare with those of [1]. Excellent agreement between our periodic simulations and the reference data can be observed. Furthermore, the graphs show that good agreement with the periodic simulations and the reference data can be achieved with spatially developing simulations for pipe lengths exceeding 30 radii. Thus, this approach appears capable of producing the desired turbulent nozzle exit conditions for full jet simulations. The spatially developing pipe flow simulations were conducted using up to 300 cores and required up to 30,000 AUs each to record sufficient data to obtain converged statistics. Additional simulations with finer grids demonstrated grid-convergence of the results, as shown by the red line in figure 2.

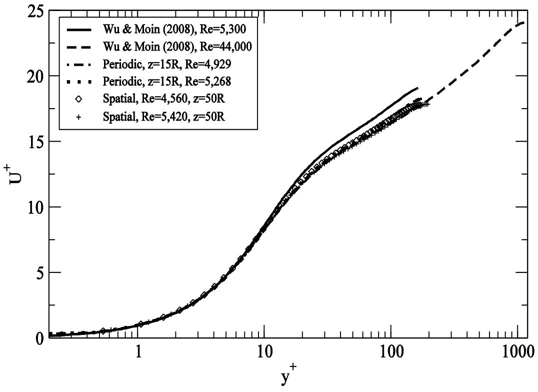

Figure 1: Mean axial velocity component U^+ as a function of $y^+=1-r^+$. Present DNS: Periodic pipe - Standard-central differences scheme, 201x72 grid points in the streamwise and radial directions, respectively, 64 Fourier modes in the azimuthal direction. Spatial calculations – compact-difference scheme, 500x56 grid points in the streamwise and radial directions, respectively. Pipe length L_s=50R.

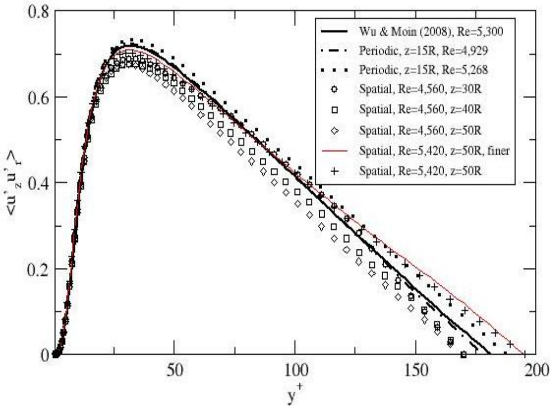

Figure 2: Turbulent shear stress component as a function of y+=1-r+. Shown are several locations in the streamwise direction compared with the results from the periodic pipe simulation and the reference data by Wu&Moin[1].

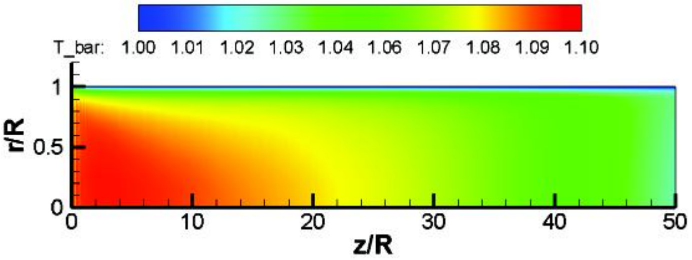

Figure 3: Mean temperature contours (spatial calculations); Re=4560, M=0.8.

To examine compressibility effects, in particular temperature variations, the pipe Mach number (based on bulk velocity) was increased to M=0.8. This pipe Mach number is anticipated to be used in the full jet calculations. The distribution of the temperature inside the pipe is shown in figure 3. The results indicate that for this Mach number the increase in the temperature (near the pipe exit) is insignificant, approximately 3%. Therefore, isothermal jets can be computed without requiring modifying the ambient temperature.

Following the successful completion of the precursor pipe simulations, the case of an axisymmetric jet exiting from a fully turbulent pipe into a strong co-flow (40% based on bulk velocity) was set up. Based on the results of the isolated pipe calculations, the pipe length was set to 50R. The isolated pipe data was also used to specify turbulent inflow conditions. The parameters were chosen such that the relative jet Mach number was M_{jet}*=0.72 (based on the relative velocity between pipe exit and co-flow) and the Reynolds number was* Re_{jet}*=7700 (based on the maximum velocity at the pipe exit), exceeding by far the highest Reynolds number DNS published to date. For this case, initially 26 Million points were used to discretise the computational domain and the case was run on 512 cores, requiring approximately 50,000 AUs. The jet flow field in the vicinity of the nozzle was sufficiently resolved, as was shown in the intermediate report. However, far downstream of the nozzle the grid was too coarse to adequately resolve the flow and to ensure minimal damping of the accoustic waves. Therefore, a much finer grid was produced, now using 70 Million grid points for the same domain size. With the much greater number of points in the streamwise and radial directions, the directions of MPI parallelisation, the finer grid case could be conducted efficiently on 4096 computing cores and more than 400,000 AUs were used. The preliminary results are shown in Figures 4 & 5. The turbulent flow inside the pipe and in the developing turbulent jet is shown in figure 4 (top) by contours of the axial vorticity. The presence of a streamwise pressure gradient inside the pipe (without being explicitly imposed) such that the pressure at the pipe exit equals that of the ambient is illustrated in figure 4 (bottom). Figure 5 shows that the simulation has progressed to a well developed state, with the initial axisymmetric flow structures having broken down to fine scale turbulence. Sound waves can be observed emanating from the nozzle corner and from the jet core. Importantly, the resulting sound field is ‘clean’, i.e. no interference from boundary reflections is present. This simulation demonstrates the ability of our code and the current set-up to conduct the originally proposed large-scale DNS of canonical nozzle/jet flows. However, for the initial simulations during this project only 16 azimuthal Fourier modes were utilised. For fully resolving the jet in all spatial directions, 64 modes will be required. This will result in a total grid count of more than 300 Million points. To achieve statistical convergence at two different jet Mach numbers that can be compared with each other, our original estimate of 20 Million AUs to complete our study appears still valid.*

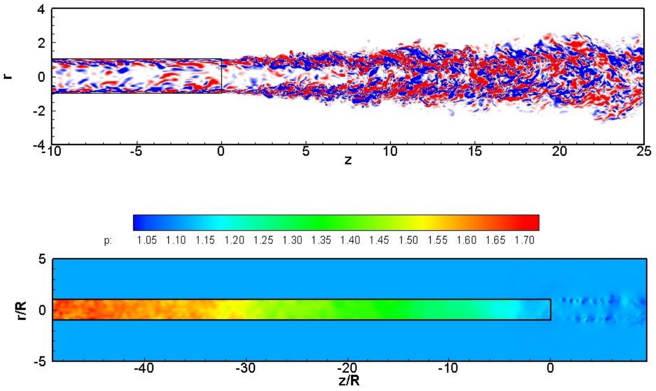

Figure 4: Instantaneous contours of axial vorticity (top) and pressure (bottom) for preliminary DNS of canonical nozzle/jet configuration.

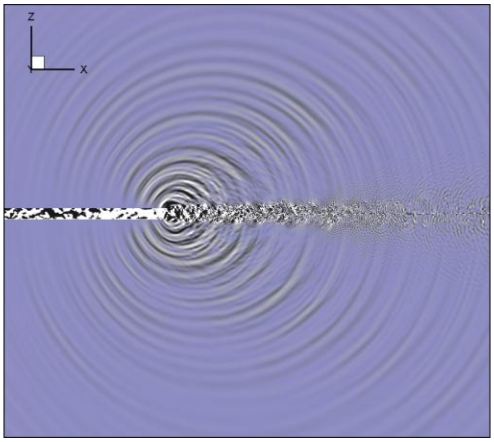

Figure 5: Instantaneous contours of dilatation field for preliminary DNS of canonical nozzle/jet configuration.

The Technical Achievements

HiPSTAR already exhibits excellent weak scaling up to 2048 processors on HECToR. The CoE has focused on tuning the code to further optimize the performance at the largest processor counts. Serial code optimisations have concentrated on restructuring how data is packed and unpacked before and after being passed to the FFTW library. Reordering loops so that read and write operations are performed in sequential bursts makes better use of the cache hierarchy and memory bandwidth available to the application. Parallel performance tuning concentrated on allowing HiPSTAR to scale further to even larger numbers of processors. Converting synchronous MPI calls to asynchronous calls optimised the MPI communication pattern, allowing receive buffers to be posted well in advance of corresponding send messages. This has improved the overlap between communication and computation which should improve the overall scalability of the code.

Overall these optimizations led to an observed 8% improvement in runtime performance on 512 processors.

Conclusions

Preliminary work required to set up a DNS simulation of a round jet exiting from a fully developed turbulent pipe has been completed. It was shown that fully developed turbulent pipe flow can be generated using a spatially developing flow approach. The pipe length and turbulent inflow parameters have been determined from the pipe simulations. Several initial full nozzle/jet configuration DNS have been conducted on up to 4096 cores demonstrating the feasibility of the chosen approach and giving a first indication of the presence of multiple noise sources. Grid resolution and domain size estimates for the full-scale simulations were made using the results of the preliminary simulations. In total, 1.8 Million AUs have been used for the pipe and preliminary jet simulations. A further 20 Million AUs will be needed to complete the scientific goals of this project.

An abstract entitled "DNS OF A CANONICAL NOZZLE FLOW" was submitted for the DLES8 Conference in Eindhoven, The Netherlands, July 2010 and has been accepted.