Direct Numerical Simulation of separated turbulent flow in axisymmetric diffuser

A S Stabnikov¹, D K Kolmogorov¹, A V Garbaruk¹ and F R Menter²

¹ Peter the Great St.Petersburg Polytechnic University (SPbPU), Politechnicheskaya 29, Saint-Petersburg 195251, Russia
² Ansys Germany GmbH, Staudenfeldweg 20, Otterfing 83624, Germany

E-mail: an.stabnikov@gmail.com

Abstract. Direct numerical simulation (DNS) of the separated flow in axisymmetric CS0 diffuser is conducted. The obtained results are in a good agreement with experimental data of Driver and substantially supplement them. Along with other data, eddy viscosity extracted from performed DNS could be used for RANS turbulence model improvement.

1. Introduction

Precise prediction of flow separation from smooth surfaces is crucial for different CFD applications because an inaccuracy in prediction of the separation point position could lead to significant changes in the overall flow structure and errors in pressure distribution and aerodynamic forces. Since the position of the separation point is highly sensitive to prediction of turbulence properties in the vicinity of the separation area and shape of the velocity profile this task is one of the most complex problems for both RANS and hybrid methods of turbulent flow computation (see, for example, [1], [2]). Therefore, the improvement of the turbulence models for accurate description of flow separation is highly important.

For that purpose, reliable data about the structure and details of the flow field and turbulent characteristics in the vicinity of the separation are required. One of the cases most frequently used for turbulent model verification is the flow in CS0 diffuser, studied experimentally by Driver [3]. However commonly used numerical setup of this flow ([1], [4], [5]) differs from the experimental setup which makes numerical and experimental results not fully comprehensive. The purpose of the current Direct Numerical Simulation (DNS) is to check if the numerical setup corresponds to experimental conditions and supplement the experimental data by DNS results. To the authors’ knowledge the DNS of CS0 flow has not been presented in the literature yet.

2. Flow description

In the experiment of Driver [3] a turbulent boundary flow around circular cylinder with adverse pressure gradient was studied. The cylinder with diameter of 0.14m was placed along a wind tunnel with a rectangular cross section. Reynolds number based on a length scale of $L_0=1m$ and inlet velocity $U_0=30 \text{ m/s}$ is equal to $Re=2.0 \times 10^6$. Separation of the boundary layer on the cylinder wall occurred due to strong adverse pressure gradient that was imposed on the downstream portion of the flow by diverging all four outer tunnel walls as seen on Figure 1. The distance from the cylinder axis in the narrow part is 0.125m and 0.196m in the wide part. To avoid the flow separation on the tunnel walls boundary layer was thinned using suction spots.
Figure 1. Schematic experimental setup of D. Driver [3].

3. Numerical problem setup and Solution Method

Identically recreating experimental setup numerically with complex, large domain and suction would be a hard and unnecessary task to take on. The common practice to approach numerical setup of this flow ([1], [4], [5]) is an axisymmetric formulation and using upper boundary aligned with the experimental streamline at approximately half height of the diffuser (see Figure 2 with schematic comparison of the experimental and numerical setup) and impose slip boundary condition on it. Even though the setup is similar to experimental, they are not identical, so some errors may be introduced to the results. Firstly, in the experimental setup contrary to the numerical setup the pressure gradient normal to the upper boundary is not zero. Second, the turbulent structures downstream from the separation may interact with the upper boundary of the computational domain.

Figure 2. Schematic CFD setup compared to experimental configuration

Based on this approach, computational domain in the current study has streamwise dimensions of \( X_{\text{min}} = -0.6L_0 \) and \( X_{\text{max}} = 1.0L_0 \). Preliminary computations with domain width angles of 30°, 60° and 90° were performed to select the spanwise domain size. Spectral and correlation analysis of the results shows that domain width of 60° is sufficient for the CS0 diffuser simulation. At inlet boundary, velocity and turbulent characteristic profiles from precursor SST RANS simulation were used along with a synthetic turbulence content generator to add velocity fluctuations to inlet velocity. The fluctuations were also based on turbulence characteristics from the inlet profile. Constant pressure was set at outlet, no-slip wall at the bottom wall and slip wall condition at the top wall. Periodic conditions were used on the side borders of computational domain. Computational mesh was uniform in streamwise and spanwise directions and condensed to the wall so that \( \Delta y^+ = \Delta z^+ = < 1 \) (here and below wall units are based on the maximum friction velocity taken from the attached part of the flow upstream of separation). Grid steps in streamwise and spanwise directions are \( \Delta x^+ = 20, \Delta z^+ = 10, \) and \( \Delta y < \Delta z_{\text{max}} < \Delta z \) which satisfy requirements for DNS of wall bounded flows [6] with good resolution. The size of the used computational mesh (5109 x 208 x 586) is about 0.6 billion of cells. Computations were also held on a two times coarser mesh (under resolved DNS) and results suggest that grid convergence was achieved.
Time step was chosen so that the maximum CFL number is less than 1.0 to ensure that the solution is time step independent. Period of flow sampling, $T_s$, was set to approximately 3.5 flow passes based on maximum streamwise velocity $U_0$.

Since experimental Mach number is low ($M < 0.1$), for incompressible form of the Navier-Stokes equations were used. Computations were held in an in-house code NTS [7]. This is a structured finite-volume high-order CFD code accepting multi-block overlapping grids of Chimera type. The incompressible branch of the code is based on an implicit flux-difference-splitting numerical scheme of Rogers-Kwak [8]. The convective terms approximated with blend of 4th order CD scheme and 3% portion of Bounded CD [9]. Complimentary DNS computations carried out on the same mesh in ANSYS Fluent commercial code give virtually the same results which confirms independence of obtained solution on the code used.

4. Results

The flow in the CS0 diffuser has three distinguishable zones (figure 3). The initial part is the attached boundary layer in the constant width part of the tunnel ($x < -0.3L_0$). Further downstream the attached boundary layer is subjected to adverse pressure gradient due to tunnel expansion until it finally separates from the cylinder surface at approximately $x = 0.0$. The flow then reattaches at about $x = 0.3L_0$ and continues further downstream to the outlet plane. Beginning with the separation, massive turbulent structures develop, the size of which are about the magnitude of angular size of the domain (figure 4a). Starting from about $x = 0.3-0.4L_0$ (figure 4b) these structures reach the upper slip wall boundary, which results in their deformation and an introduction of error to the flow structure downstream. Therefore, the DNS results are reliable only up to the point of approximately $x = 0.3-0.4L_0$. It should be noted that reason for this problem is insufficient distance between upper boundary (experimental streamline) and the cylinder wall. Unfortunately, this boundary could not be shifted further from the wall because no wider streamlines are available from the experimental data. The same problem takes place when approaching this flow using any other Scale Resolving Method. This circumstance also means that there is no point in increasing the angular size of the domain since from this point downstream the flow is constrained with wall normal domain size, not angular size.

![Figure 3. Time- and span- averaged $U/U_0$ and $k/U_0^2$](image)

![Figure 4. Isosurface of normalized helicity, $H \cdot L_0/U_0^2 = 20$, coloured with x-velocity in the area around the flow reattachment point ($x/L_0 \approx 0.35$) and instant vorticity field at middle XY plane](image)
Overall, the presented DNS results agree favorably with experimental data of Driver [3] (figures 5-8). It can be seen that the separation point location is almost identical, however in the DNS calculations the separation zone is marginally longer (figure 5, left and figure 6). The pressure coefficient (figure 5, right) is overall higher than experimental in the separation region while they almost match in attached regions. Mean velocity and kinetic energy profiles of DNS reasonably match experimental in all parts of the flow (figures 7 and 8).

---

**Figure 5.** Comparison of the computed and experimental $C_f$ and $C_p$ distribution

**Figure 6.** Streamlines obtained with DNS in comparison with experimental data

**Figure 7.** Comparison of the computed and experimental mean streamwise velocity profiles normalized by the local maximum
Presented DNS offers a substantial expansion to available experimental data on the studied flow. Apart from obvious ways to use the new data to assess and improve turbulence models, such as comparing velocity profiles, skin friction and pressure distributions, one can make use of other statistical data gathered.

One of the approaches is to extract eddy viscosity from DNS and to use it for the RANS equations closure. If the obtained results agree with DNS data the viscosity could be used as a target value for the turbulence models improvement. Using the approach from [10] based on the Boussinesq formula, the eddy viscosity was extracted from the DNS computation. Though it is a bit noisy (for the smoother
distribution significantly longer averaging period is required) it represents the required level of eddy viscosity in the considered flow. Comparison with SST RANS [11] (figure 9) shows that upstream the separation ($x < 0.0$) SST generates a reasonable level of eddy viscosity whereas downstream the separation it is significantly overpredicted.

Results of the computations with RANS equations closed by the frozen eddy viscosity extracted from the DNS are shown on figure 10. One can see that the obtained results are in a good agreement with the DNS data which justifies using of the Boussinesq assumption for this flow. These results confirm that the eddy viscosity extracted from the DNS are applicable for turbulence model improvement.

5. Conclusions

Direct Numeric Simulation (DNS) of the CS0 diffuser flow using mesh of about 0.6 billion nodes was performed. It was shown that commonly used setup based on the experimental streamline is suitable for DNS and other SRS only upstream of about $x \approx 0.3-0.4L_0$. Further downstream turbulent structures start to interact with upper slip boundary which affects the results. The presented DNS offers a substantial expansion to available experimental data on the studied flow. In particular, the results of the current DNS study have been used in LES validation study [12]. Finally, eddy viscosity was extracted from the DNS and it is shown that it could be used as a target to develop, calibrate and even fine-tune RANS turbulence models.

Acknowledgments

The reported study was funded by RFBR, project number 19-31-90046. The results have been obtained with use of computational resources of Supercomputer Center at Peter The Great Saint-Petersburg Polytechnic University (www.spbstu.ru).

References

[1] Menter F R 1992 Performance of popular turbulence models for attached and separated adverse pressure gradient flows AIAA J. 30(8), 2066–72
[2] Frère A, Hillewaert K, Chatelain P and Winckelmans G 2018 High Reynolds number airfoil: from wall-resolved to wall-modeled LES Flow Turbulence Combust. 101(2), 457–76
[3] Driver D M 1991 Reynolds shear stress measurements in a separated boundary layer flow AIAA Paper 91–1787
[4] Dudek J C, Georgiadis N J and Yoder D A 1996 Calculation of turbulent subsonic diffuser flows using the NPARC Navier-Stokes code AIAA Paper 96–0497 1–11
[5] Rumsey C L NASA Langley Turbulence Modeling Resource http://turbmodels.larc.nasa.gov/
[6] Coleman G N, Garbaruk A V and Spalart P R 2015 Direct numerical simulation, theories and modelling of wall turbulence with a range of pressure gradients Flow Turbulence Combust. 95(2–3), 261–76
[7] Shur M, Strelets M and Travin A High-order implicit multi-block Navier-Stokes code: ten-years experience of application to RANS/DES/LES/DNS of turbulent flows https://cfd.spbstu.ru/agarbaruk/doc/NTS_code.pdf
[8] Rogers S E and Kwak D 1988 An upwind differencing scheme for the time accurate incompressible Navier-Stokes equations AIAA Paper. 88-2583
[9] Jasak H, Weller H G, Gosman A D 1999 High resolution NVD differencing scheme for arbitrarily unstructured meshes Int. J. Numer. Methods Fluids 31(2) 431–49
[10] Spalart P R, Belyaev KV, Garbaruk AV, Shur ML, Strelets M K and Travin A K 2017 Large-eddy and direct numerical simulations of the Bachalo-Johnson flow with shock-induced separation Flow Turbulence Combust. 99(3), 865–85
[11] Menter F R, Kuntz M and Langtry R 2003 Ten years of industrial experience with the SST turbulence model Turbulence, Heat and Mass Transfer 4 625–32
[12] Kolmogorov D K, Garbaruk A V, Stabnikov A S and Menter F R 2021 Large-Eddy simulation of CS0 diffuser on resolved and under-resolved meshes 13th International ERCOFTAC Symposium (ETMM13) (accepted for publication)