

Part 2

The first step in each CFD analysis is creation of the geometry, meshing the domain and defining the suitable boundary conditions for this means. What was available at that preliminary stage of this work was an initial direct numerical simulation (DNS) solution of this case which consisted of the DNS of flow field before the bubble area and the Euler solution in the rest of the domain which didn't catch any separation or bubble and only it provided us with the top wall distribution of rectangular domain of DNS analysis. The DNS domain was a rectangular shape which the favorable pressure gradient of deceleration part of the experiment has been added by some numerical means. To be more accurate in DNS calculation the streamwise pressure gradient has been imposed by prescribing the potential velocity distribution for streamwise velocity at constant distance $y=0.12$ (Topwall) from the wall while the displacement effect of the boundary layer is captured by boundary layer interaction model.

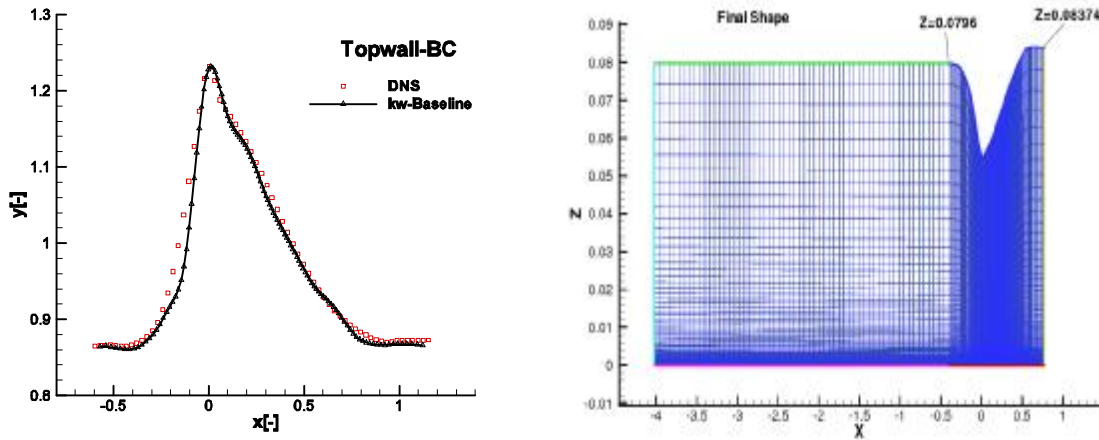
This implementation of top wall pressure is not available in common RANS simulation code and there is not such an option that you can simply prescribe a pressure distribution in one of your boundary. So this can be only done by changing the shape of your boundary itself in a way that it produces the favourable pressure distribution.

Having the velocity field in advance and using mass conservation law in each cross section, the most preliminary area of each cross section has been found. Adding the boundary layer thickness to this length at each cross section gives a more accurate estimate of the cross sectional area that eventually has to lead to that favourable pressure gradient. This is not the end of story and the produced geometry has to be meshed and run in the code and has to be compared with top wall velocity distribution of DNS to validate the goodness and accuracy of the created geometry. If this is not the case, points in deceleration or even acceleration part should be changed manually in order to get close to the top wall velocity step by step. Curve fitting methods should also be recruited to avoid wiggles in numerical solution. Yes! It is a very frustrating and laborious task.

Moreover the grid convergence analysis must be done as usual.

This is the final geometry of top wall boundary and also the comparison of potential velocity with the initial DNS data. The term potential velocity refers to condition where the transition location is prescribed far upstream to surpass the bubble for the sake of

comparing the result with initial DNS solution. As it can be seen in deceleration area from neck to the right which is the most important part of the domain, good matching of top wall velocities (or pressure distribution) has been obtained.



The inflow boundary condition should have the same boundary layer thickness of the DNS data. This can only be achieved by adding an extension part in the beginning of our geometry to yield that boundary layer thickness of DNS at inflow. Several more iterative processes need to be done to get the same inflow boundary condition even by considering the Blasius formulation for calculation of boundary layer thickness.

Now the geometry is created.

The numerical code which has been implemented and modified and generally is the basis of computations in this project is an open code of DLR TAU-code which firstly developed and employed almost a decade ago by German aerospace institution, DLR. Since a decade ago the code has been involved in numerous national and European projects like MEGAFLOW, FASTFLO, TAURUS, FLOMANIA and DESider.

The very short description of the code can be found [here](#) and [here](#) and more in detail in its technical document. More description of this very complicated C language written code is out of scope of this report.