Congress on Numerical Methods in Engineering
Lisboa, June 29 to July 02, 2015

RESUMO N° 417

ON THE CALCULATION OF THE FLOW AROUND 2-D SQUARE CYLINDER WITH ROUNDED CORNERS

Hugo Abreu, hugo.abreu.91@gmail.com
IST, Portugal

Luís Eça, luis.eca@ist.utl.pt
IST, Portugal

Guilherme Vaz, G.Vaz@marin.nl
MARIN, Portugal

Keywords: RANS, Unsteady Flow, Eddy-Viscosity Models, Verification, Numerical Errors

Many offshore applications use cylinders that range from circular cross-sections to square cylinders with rounded corners. Typical Reynolds numbers of practical applications are in the range of 1e5 to 1e6 where the so-called "drag crisis" occurs. Flow simulations in such conditions are extremely challenging because the flow exhibits laminar, transitional and turbulent regions. Due to the existence of vortex shedding, the flow is not statistically steady.

Although the Reynolds-Averaged Navier-Stokes (RANS) equations supplemented by eddy-viscosity models have evident shortcomings in such complex flows, there are several attempts published in the open literature to simulate this type of flows with such mathematical model. Due to the periodic nature of vortex shedding, ensemble averaging must be used for the definition of the mean flow and for the averaging of the mass and momentum balance. Therefore, the RANS equations are not statistically steady, which is usually designated by URANS.

In this paper we present a study of the calculation of the flow around a square cylinder with rounded corners assuming two-dimensional and incompressible flow. We have selected the (SST) $k-\omega$ eddy-viscosity model to solve the RANS equations and investigate the influence of the physical settings of the problem, i.e. size of the computational domain and specification of boundary conditions. Grid and time refinement studies are performed to assess the numerical uncertainty of meaningful flow quantities for practical applications. All calculations are performed with the ReFRESCO solver.

The results show that the size of the domain and the specification of pressure and turbulence quantities inlet boundary conditions have a significant influence on the predicted flow. Therefore, any proper Validation procedure must follow the detailed inlet flow conditions. Unfortunately, such information is seldom available and so this hampers the possibility to evaluate correctly the model quality.