

# High Reynolds flows DDES over a square rounded cylinder using OpenFOAM

Marcello Meldi<sup>(1)</sup>, Eric Serre<sup>(2)</sup> & Matthieu Minguez<sup>(3)</sup>

(1) Institut Pprime

(2) M2P2, Ecole Centrale Marseille

(3) SEAL Engineering (Technip Subsidiary)

The work presented in this talk fits into a global collaborative R&D program between SEAL Engineering (Technip subsidiary) on the one hand and both, the laboratory of Modeling in Mechanics of Marseille (the M2P2, Ecole Centrale Marseille) and the Institut Pprime of Poitiers on the other hand. It more particularly fits into the frame of an international collaborative numerical benchmark for high Reynolds which is subjected to a special session at the OMAE2016.

As requested for the benchmark, the efforts have been spent onto the CFD modelling of the high Reynolds flow that develops over a square rounded shape cylinder. The proposed analysis follows the one already presented in Nantes at the OMAE2013, refer to OMAE2013-10200 and which highlighted very encouraging comparisons for 'relatively high' Reynolds conditions ( $Re=22000$ ). The 'in-house' CFD capabilities are, for the proposed work, used to increase the investigated Reynolds range up to reach flow regime representative of full scale offshore conditions (i.e.  $Re=10^7$ ).

Presentation of the numerical solver

The CFD analysis consequently focuses on the square rounded cylinder flow for the  $45^\circ$  heading angle. The numerical simulations have been performed using the open source code *OpenFOAM* on a mesh of  $27 \cdot 10^6$  elements. In order to highlight the emerge of different physical mechanisms due to the sensitivity of the flow to the Reynolds number, two configurations for  $Re=10^5$  and  $Re=10^7$  have been investigated. Additionally, three-dimensional effects have been magnified considering a very large domain in the spanwise direction, which measures approximately 8 characteristic lengths. The strategy adopted for the turbulence closure is the well known hybrid DDES approach proposed by Spalart, which has been combined with a Spalding wall function for the higher Reynolds simulation. Several physical quantities have been analyzed, which include the pressure coefficient, the distribution of  $y^+$  over the surface of the cylinder and the classical bulk flow coefficient such as  $C_D$ ,  $C_L$  and the Strouhal number. Isocontours of the vorticity field are shown in Figure 1. The results underline a characteristic drag behavior which put in mind the well-known drag crisis of the classical circular cylinder. At the conference time, the talk will aim at characterizing the two different flow regimes addressing mainly the first order analysis of the hydrodynamic properties of the bluff body. In a second time, complementary analyses will be proposed to try to make the light on the mechanism of such possible turbulent transition.

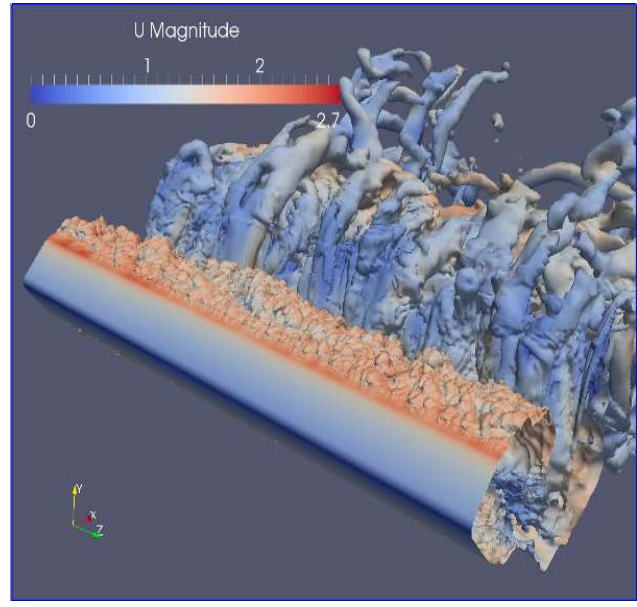
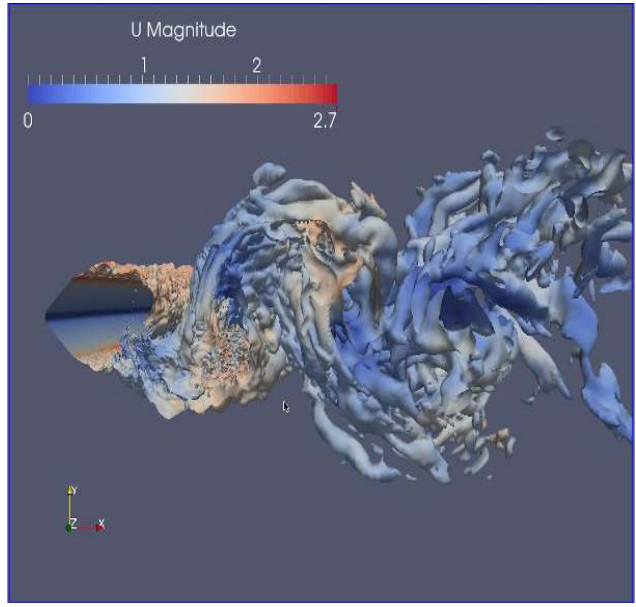


Figure 1 : isocontours of the vorticity field, colored by the magnitude of the velocity, for the flow configuration at  $Re=10^5$