Drag reduction in gas export pipelines; a fundamental numerical approach

F. Nygård[a], H.I. Andersson

Fluids Engineering Division, Department of Energy and Process Engineering, Norwegian University of Science and Technology (NTNU), Trondheim, Norway

[a]*E-mail:fnygaard@ntnu.no*

Only a few percent reduction of friction in gas export pipelines would form vast annual savings of energy. Motivated by this, the present work commenced with an aim of performing fundamental numerical research on the mechanisms responsible of friction in turbulent pipe flows. In wall-bounded flows (e.g. pipe or channel flow) drag seems to be linked to so called streaks. Such streaks were first discovered by different visualization techniques around 1970, [1, 2]. The structures might be described as elongated pockets of high- and low-speed fluid and are located in the wall-boundary-layer. The structures come into being by inward flow (sweep) or outward flow (ejection) of fluid in the wall-boundary. Such mixing of wall-layers will create (turbulent) friction. The turbulent friction in turbulent flows dominates the viscous friction-forces that acting as the only drag-contributor in laminar flows. Another reported characteristic of wall-bounded flows is the bursting process. The bursting process comprehends low-speed streaks starting to oscillate while moving from the wall (ejection) and breaking up. The instability of low-speed streaks is believed to be important in maintaining the turbulence as described by Swearingen & Blackwelder [3]. The described turbulent features are claimed to make a considerable subscription to the turbulent kinetic energy production in the near-wall region [4]. Therefore, controlling these mechanisms might be a key in taming the turbulence. Drag reduction of polymers in wall-bounded turbulent liquid flows is today a well-known phenomenon and have been reported to reduce drag by up to 80% [5]. However, drag reduction in wall-bounded gas flows has shown it self as a greater challenge. Nevertheless, drag reduction by up to 10% is reported in turbulent wall-bounded flows with the wall covered by elongated riblets [6]. Drag reduction achieved in numerical [7] and experimental [8] studies of rotating pipelines lead to the present idea of investigate swirl in pipe flow with regard to illuminate potential modifications of the turbulence. However, instead of rotating the pipe, the fluid is rotated inside a stationary pipe. The present work presents direct numerical simulation (DNS) results of low Reynolds number (Re = 4900) turbulent pipe flow. Naturally, the results from the present study are not directly applicable to the flow in gas export pipelines ($Re \sim 10^7$) but have to be regarded as fundamental research on the turbulent mechanisms which are assumed to be similar in the two cases. Two different methods of setting up the swirl have been pursued. First, an academic approach has been investigated. Here, a numerical applied body-force is used to induce the swirl. Secondly, a more practical case with a 0.5R helical wall-fin has been simulated. Drag reduction was only achieved in the first case. Results from the two cases are presented and compared. The modifications of turbulence due to the swirl are pointed out i.a. by use of Reynolds stress profiles and visualization plots of fluctuating velocities in the near-wall region.

Preprint submitted to European Gas Technology Conference 2011 *November 10, 2010*

[1] E. R. Corino and R. S. Brodkey, J. Fluid Mech., **37**, 1 (1969).

[2] H. T. Kim and S. J. Kline and W. C. Reynolds, J. Fluid Mech., **50**, 133 (1971).

[3] D. Swearingen and R. Blackwelder, J. Fluid Mech., **182**, 255 (1987).

[4] S. J. Kline and W. C. Reynolds and F. A. Schraub and P. W. Rundstadler, J. Fluid Mech., **30**, 741 (1967).

[5] S. L'vov and R. Benzi, Rev. of Modern Phys., **80**, 225 (2008).

[6] D. W. Bechert, M. Bruse, W. Hage, J. G. T. Van Der Hoeven and G. Hoppe, J. Fluid Mech., **338**, 59 (1997).

[7] P. Orlandi and M. Fatica, J. Fluid Mech., **343**, 43 (1997).

[8] M. Murakami and K. Kikuyama, Trans. ASME J. Fluids Engng., **102**, 97 (1980).