CFD study on high-pressure water-spray cleaning system for CSP mirrors

Francesco Anglani1, Willem Dekkers1, Raad Aljawi1, Zhiqiang Guan2

1Science and Engineering Faculty, Queensland University of Technology, Brisbane, 4000, Australia
2School of Mechanical and Mining Engineering, University of Queensland, Saint Lucia, 4072, Australia

Introduction

This study outlines that the effectiveness of water-jet cleaning processes depends on parameters such as nozzle diameter, jet impingement angle, standoff distance, water velocity and nozzle pressure [1, 2]. Findings shows that the presence of jet rebounds due to the interaction of adjacent flows affect the cleaning ability of the central nozzle in some scenarios. Finally, the CFD model proposed in this analysis has been validated with empirical data from literature.

Methodology

Two domains have been created to model the water-spray cleaning systems equipped with an interface on points of contact between nozzles and control volume. An inhomogeneous multiphase problem approach has been considered to model the two phase gas-liquid flows. Water at 25°C has been treated as dispersed fluid with droplet size of 10µm while air as a continuous medium at ambient temperature. The continuity and momentum equations describes the state of any type of flow are reported down below, respectively.

\[
\frac{\partial p}{\partial t} + \nabla \cdot (p\nu) = 0 
\]

\[
\frac{\partial p}{\partial t} + \nabla \cdot (p\nu u) = -\nabla p + \nabla \cdot \tau + p g 
\]

Nozzles chosen for this study are full cone convergent types 54mm long with inlet internal diameter of 8mm and outlet internal diameter of 2mm. The contraction has been modeled with an angle set to 22.5° to the axis of symmetry (Figure 1). Nozzle interaxial distance has been set to 60mm to minimise interaction between adjacent flows.

Results

Results confirm an increase in shear force magnitude as a function of increasing inlet pressure. In figure 4, the water-spray behavior impinging normally onto the flat target with three different standoff distances is shown. For low inlet pressures, there is not much difference in the shear force peak. More significant differences occur above 20 Pa of inlet pressure, especially between 5D and 25D standoff. The shear forces are increased by tilting the jet 15° from the normal (angle of impingement θ=75°) (Figure 5). The tilt effect prevents adjacent flows of the outermost nozzles from obstructing the central flow with a subsequent generation of a higher shear stress peak registered than in the normal impingement (Figure 6 and 7). The increase is most marked for the 5D standoff distance with modest enhancement of 3% and 10% for 15D and 25D respectively. The turbulence due to back flows is strongly influenced by the amplitude of the speed prior to hitting the planar surface. At closer standoff distances the jet impingement is more concentrated than for larger standoff, and interaction between jets less more significant.

Validation

The following CFD model has been compared with empirical experiments conducted by Young et al. (Figure 8). Moreover, we conducted a dimensionless analysis to compare our findings with Smedley’s experimental results (Figure 9).

Conclusions

In this study a detailed analysis of water-jet impingement on a flat surface has been performed with multiple nozzle configuration in line. Three different standoff distances in conjunction with the variation of inlet pressure and angle of impingement have been considered in the CFD simulations with ANSYS v15 software.

Findings confirm a significant increase of shear stress as the standoff distance decreases. By varying the angle of impingement from θ= 90° to θ=75° for both configurations, an enhancement of shear forces occurs for all standoff distances but is more effective at 5D. Fluid flow and fluid shear may be significantly disrupted due to interaction between adjacent jets. Varying the angle of impingement from θ= 90° to θ=75°, an enhancement of shear forces occurs for all standoff distances with discontinuance of back flows.

Nozzle design and arrangement should minimise turbulent flow in order to maximise cleaning efficiency.