Abstract

In the power plant, the steam turbine remains an important component for the efficient production of power. The low-pressure turbine stages are of particular interest since they produce the largest portion of the power (across all of the stages), and yet are sensible to additional losses due to the presence of a second phase. The present work deals with the study of the homogenous condensation of steam flow experimentally and computationally in a convergent – divergent nozzle and to have a clear understanding of the physical phenomena of the real case of the rapid expansion of steam and predict the shock wave that appears in the divergent part due to over-expansion by using optical method. Two operating conditions were studied: the first was to see the effect of decreasing the absolute inlet pressure of (0.69-0.36) bar at constant absolute back pressure of (0.2 bar). The second was to show the effect of increasing the absolute back pressure of (0.145-0.34) bar on the flow at constant absolute inlet pressure of (0.5 bar). All tests were done at saturation temperature of the inlet pressure for each case. The experimental work included manufacturing of the C-D nozzle, adopted from **Hussein and Samir [26]** with area ratio of (0.5) and nozzle length (140 mm). The static pressure distribution was measured along the nozzle, and then Mach No. distribution was obtained. The shadowgraph method was used as an optical technique to capture photos that show the rapid condensation zone and shock wave in the divergent part. The experimental results showed that the rapid condensation zone appears downstream the throat of the nozzle and when the inlet pressure was decreased, the possibility of appearance of the shock wave increased and progressed towards the throat at locations (40,32,28 and 26) millimeters from the nozzle throat.

The maximum Mach number was found to be (1.88) For pressure ratio of (0.29). When the back pressure increases, the shock wave formulated due to over-expansion and progresses towards the throat region at locations of (24 and 16) millimeters from the nozzle throat.

The computational work was carried out by using two-phase flow wet steam model in the well-known package **ANSYS FLUENT code 15.0**. The calculations were performed under the assumption that the flow is: steady, two dimensional, compressible and turbulent. An Eulerian–Eulerian multi-phase approach was used to model in the numerical simulation. The numerical results showed velocity vector, static pressure, Mach No., and droplet growth rate contours. The droplet growth rate contour shows that when the inlet pressure decreases, the droplet growth rate decreases and the maximum value was ($1.02 * 10^3$ micron/s) when pressure ratio of (0.29). While, when the back pressure increases, the droplet growth rate remains constant at (748 micron/s). This approach gave acceptable results when comparing the numerical output results with the experimental ones.