

Investigation Of Lid Driven Cavity

Prepared by: Ng Voon Shen

ABSTRACT

As aircraft is slowly become a part of human needs of transportation, the types/shape of lid-driven cavity at the opening structure of the airplane tyres during take-off and landing become important so that the fluid motion such as noise, vibration, and premature failure can be avoided without affect the comfort and safety of aircraft passenger. This undergraduate project presents an investigation lid driven cavity using numerical method with the shape of rectangular, triangular, and semi-circular with aspect ratio of 2. The research is focus on the flow mode, two-dimensional cavity flows, flow visualizations, computational investigation of a cavity. The numerical analysis use ANSYS FLUENT software as an interface. The Reynolds number used in the investigation is 467893 (270km/h). The flow behaviour inside a lid-driven cavity has determined. Besides that, the best shape of cavity has been determined from the shapes of streamline. On the other hand, the pressure distribution and the position at the centre of vortex for developing flow has been identify. Based on the observed result, the best shape of the cavity is semi-circular. The cavity contours obtained was in good agreement. Thus, the findings are concluded and recommendations for future work are suggested.