CFD modeling of slasher with single and double counter rotating blades using sliding mesh technique

*Lj. Erakovic, A. Subic and J. Tu

RMIT University, Melbourne, Australia

Problems caused by lack of control of Chilean Needle Grass spreading during slashing necessitate the improvement of the slashing process and associated technology. The main objective of such improvements is to reduce seed scattering around the slashing strip and prevent seed accumulating on the slasher. Computational Fluid Dynamics (CFD) is employed here to simulate the flow in grass slashers by simulating rotating blade action. It is assumed that reduced velocities on the slasher outlet would result with reduced scattering of seeds and grass. Two slasher models have been developed and tested by CFD, an existing single blade (SB) model and a new double counter rotating blades (DCRB) model. While SB is currently used for grass slashing DCRB represent a novel approach to the slashing technology. Complex geometry of slasher models is very difficult to be meshed thus uniform mesh is generated by tet/hybrid elements. Advanced meshing quality, which is reflected in smaller spacing of the mesh nodes does not necessarily mean better results, but it slows down the computing process. Hence, optimal solution, which guarantees high quality of results and very small time usage is attained. Both models utilised the time-dependent and computationally intensive "sliding mesh" technique available in the FLUENT commercial CFD package. Velocity profiles of the slasher outlets are recorded for both models showing encouraging results. This paper presents the developed CFD models and the obtained simulation results in greater detail.