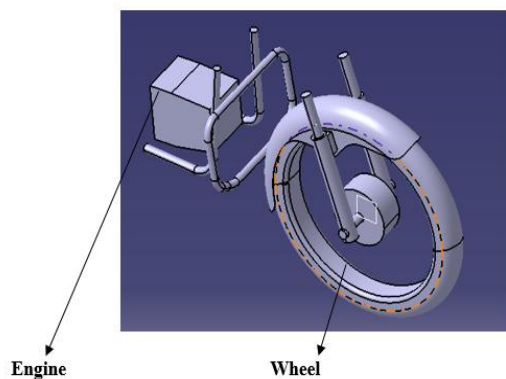


## Analysis of Effective Cooling Method for Spark Ignition Engine Using Air Channel

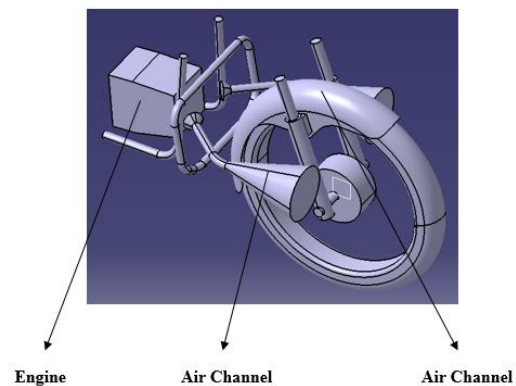
The main aim of the proposal is doing fluid analysis in Hero Honda 4 stroke S.I Engine. Finding where are all fluid is not get contacted on engine due to front wheel. This front wheel hides engine front cover and decreasing the air flow on it. It diverts the air flow. So less amount of air is falling on engine especially very less air is falling on engine front. So when long travel doing the engine is gets heated more and it may leads to crack .To avoid that first the following procedure adopted

1. Fluid speed is calculated (normal  $u=4/s$ )
2. Air channel is designed and fixed in the front wheel to increase velocity( $u=12$  m/s)
3. Drag force is calculated to ensure air channel is cable of resisting the drag force after fixed in front wheel

Finally water is sprayed in air channel hole to increase more heat transfer from engine when subjected to high thermal stresses on long travel (steady state analysis). In this proposal I am going to calculate only engine heat transfer by Openfoam CFD package. No air channel analysis (air channel velocity, air channel drag force analysis not taken into account). Models are created in CATIA. Meshing is done in Ansys ICEM CFD for the accurate results as the mesh generated by using this software is more precise. Later the mesh is imported to Openfoam for thermal analysis. Boundary conditions ( $U=12$  m/s,  $T=363$  K,  $P=1$  atm, back  $P=0$ ) are applied for the regions selected and required details given. The result obtained in the analysis is explained after. **(Only engine part is going to be analyzed in Openfoam)**



Geometric model plot without air channel



Geometric model plot with air channel