CFD analysis of a Planetary Gearbox

Parth Tawarawala

Abstract

The idea of the present study is to simulate the fluid flow of lubricant inside a planetary gearbox. OpenFOAM solver simpleFoam will be used for the same, since we can reasonably assume steady-state in a gearbox of a car in motion, and simpleFoam provides the best options for the same. The geometry is relatively complex, with a curved square pipe with rotating walls and a rotating cylinder placed within the flow. As part of an initial analysis, the geometry has been simplified to a 2D geometry. Since the geometry may not be exact, the emphasis will not be on exact flow parameter values but rather on visualisation of flow and look for any accumulation of fluid in the current complex geometry and recording it. This study is flexible because it can be expanded further to study the heat transfer within such geometries since the flow parameters and pattern will be well established.

Problem Statement

The goal is to do the following:

1. Plot the velocity vectors and streamlines to visualise and study the fluid flow

2. Plot the spatial variation of velocity, and find the location and value of the highest velocity.

3. Look for any accumulation in the gearbox flow pattern.
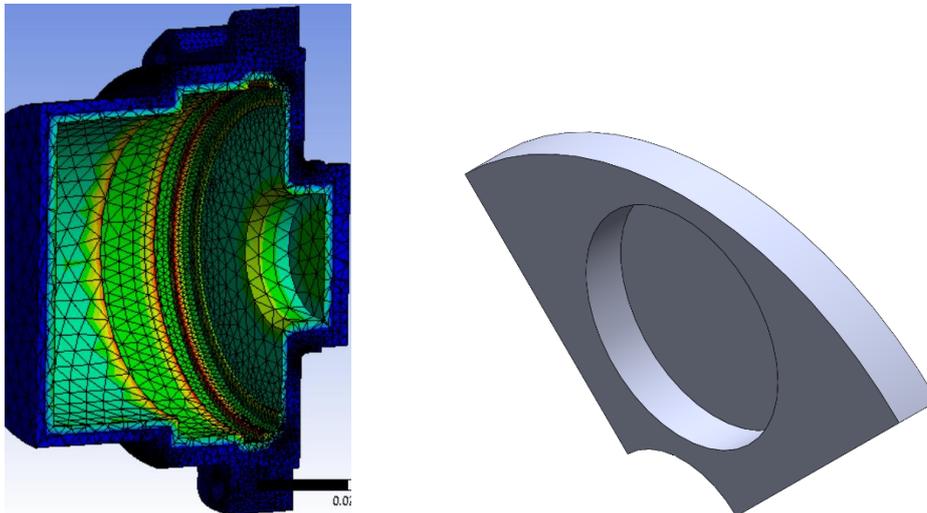
Figure: (i) Representative image of thermal analysis as done in research papers (ii) An isometric, cross-sectional view of the CAD of the domain, with sun and ring gears and bottom and top walls, and planet gear in the centre