

Transient, Three Dimensional CFD Model of the Complete Engine Lubrication System

Sujan Dhar
Simerics Inc.

Homa Afjeh
Caterpillar Inc.

Chiranth Srinivasan, Raj Ranganathan, and Yu Jiang
Simerics Inc.

ABSTRACT

This paper reports on a comprehensive, crank-angle transient, three dimensional, computational fluid dynamics (CFD) model of the complete lubrication system of a multi-cylinder engine using the CFD software Simerics-Sys / PumpLinX. This work represents an advance in system-level modeling of the engine lubrication system over the current state of the art of one-dimensional models. The model was applied to a 16 cylinder, reciprocating internal combustion engine lubrication system. The computational domain includes the positive displacement gear pump, the pressure regulation valve, bearings, piston pins, piston cooling jets, the oil cooler, the oil filter etc... The motion of the regulation valve was predicted by strongly coupling a rigorous force balance on the valve to the flow. The results show that the majority of the system pressure drop and flow rates occur in bearings, lifters and piston cooling jets, confirming the importance of a three-dimensional treatment for these components versus a relatively empirical zero or one-dimensional treatment. The newly developed CFD capability is timely because it can also drive the development of system-level models in adjacent design areas of topical interest such as engine oil warm up, interaction between lube oil and coolant temperatures, engine oil temperature distribution (which can impact oil life), uncovering of oil pick up tube, oil pump priming, bearing cavitation and dry spots, oil aeration, piston cooling jets, variable displacement vane pump and other related topics.

CITATION: Dhar, S., Afjeh, H., Srinivasan, C., Ranganathan, R. et al., "Transient, Three Dimensional CFD Model of the Complete Engine Lubrication System," *SAE Int. J. Engines* 9(3):2016, doi:10.4271/2016-01-1091.

INTRODUCTION

The growing role of analysis in the design and development of the internal combustion engine lubrication system, over the last couple of decades, has been facilitated by continuous capability improvements of system-level and component-level models driven by advances in numerical techniques [1,2,3] as well as modeling of cavitation and aeration [4].

System-level analyses include both transient and steady state one-dimensional (1D) flow models. These models interconnect bearing and other component models with one 1D treatment for ducts. Bearing models are zero-dimensional expressions based on experimental or other data [5], thus, introducing empiricism into the analysis. This is unfortunate, because the vast majority of the pressure drop between the pump exit and the crankcase occurs across the bearings. Heat transfer considerations can also be included into one-dimensional flow models [6].

For bearings, the state of the art methods use finite volume or finite element models based on the Reynolds equation [7, 8, 9, 10, 11, 12]. Positive displacement oil pumps have also been studied extensively using 0 and 1 dimensional models such as those presented in [13,14,15]. The relative scarcity in available literature on full 3D CFD methodologies and analyses in positive displacement pumps is because of the extremely complex geometry of such machines feature double curved contiguous surfaces and extremely small gaps between solids. Over the past several years the commercial CFD software Simerics-Sys/ PumpLinX has been developed to address this challenging problem and studies on positive displacement devices [16,17,18,19,20] have been published using this platform. For other components, 3-dimensional CFD models are increasingly used in product development. Porous media formulations are commonly invoked for the oil filter and oil cooler; however, such models depend upon experimental measurements of flow versus pressure drop information to generate the porous media model constants.

In summary, the state-of-the-art of engine lubrication system analysis is 1D modeling at the system-level and 3D modeling at the component-level. The authors are not aware of any work reported on 3D modeling at the system-level of the complete engine lubrication system, and this is the subject of the present paper. In the present work a full 3D CFD model was developed for the entire lubrication system of a V-16 engine. The model development was performed within the commercial CFD tool Simerics-Sys/ PumpLinX. The components of the model included the oil gear pump and associated pressure regulation spool valve (along with all pilot line connections), 9 main (crankshaft) main bearings (MB), 16 con-rod bearings (RB), 16 piston pins, 7 camshaft bearings (CB), 16 piston cooling jets, 32 valve lifters, oil cooler and oil filter. A view of the entire computational domain can be seen in Figure 1.

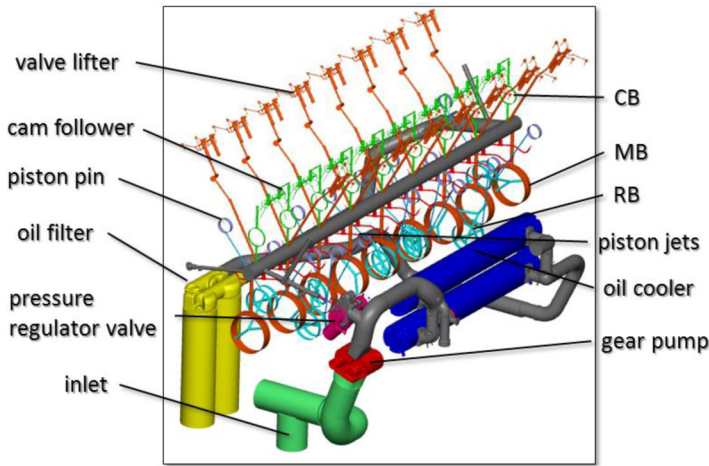


Figure 1. Domain of the 3D CFD model of a V-16 lubrication system

Thus, the 3D CFD model reported in this paper represents a significant progress for engine lubrication system modeling from the current state of the art of one-dimensional network models. Introduction of new products to the engine market can benefit from optimization for performance, durability, development time and cost driven by detailed CFD results of components as well as interactions between the components of the system.

Mathematical Model

Governing Equations for Flow

The governing equations for the mass and momentum conservation laws for the system of liquid, non-condensable gas and vapor may be expressed, in the integral form, as follows:

$$\frac{\partial}{\partial t} \int_V \rho dV + \oint_S \rho(u - u_g) \cdot ds = 0 \quad (1)$$

$$\begin{aligned} \frac{\partial}{\partial t} \int_V \rho u dV + \oint_S \rho u(u - u_g) \cdot ds \\ = \oint_S \tau \cdot ds - \oint_S p ds \\ + \int_V F dV \end{aligned} \quad (2)$$

The terms on the right side of the momentum equation (2) represent, respectively, the viscous shear force, the pressure force and the body force. For oil pump flows cavitation is often encountered and is an important factor of design considerations. In the present work the vapor mass fraction f_v is solved via

$$\begin{aligned} \frac{\partial}{\partial t} \int_V \rho f_v dV + \oint_S \rho f_v(u - u_g) \cdot ds \\ = \oint_S D_f \nabla f_v \cdot ds + \int_V (R_e \\ - R_c) dV \end{aligned} \quad (3)$$

with the evaporation and the condensation source terms expressed as [4]:

$$R_e = C_e \rho_l \rho_v \left[\frac{2(P_v - P)}{3 \frac{\rho_l}{\rho_v}} \right]^{0.5} (1 - f_v - f_g) \quad P < P_v \quad (4)$$

$$R_c = C_c \rho_l \rho_v \left[\frac{2(P - P_v)}{3 \frac{\rho_l}{\rho_v}} \right]^{0.5} f_v \quad P > P_v \quad (5)$$

The non-condensable gas mass fraction f_g follows its own conservation equation with no volumetric sources (sinks):

$$\begin{aligned} \frac{\partial}{\partial t} \int_V \rho f_g dV + \oint_S \rho f_g(u - u_g) \cdot ds \\ = \oint_S D_g \nabla f_g \cdot ds \end{aligned} \quad (6)$$

The mixture density ρ in the above equations are calculated from

$$\frac{1}{\rho} = \frac{f_v}{\rho_v} + \frac{f_g}{\rho_g} + \frac{(1 - f_v - f_g)}{\rho_l} \quad (7)$$

The above set of equations essentially describes a system of homogeneous multiphase flows. In addition, the standard $k - \epsilon$ two-equation model [21] is used to account for turbulence. The above equations are discretized using the finite volume approach, and solved using a pressure based solution algorithm.

Dynamic Equation for the Pressure Relief Valve

In the present model, the spool position of the pressure relief valve is solved as a fluid - structure interaction problem using the following equation for the spool dynamics:

$$m \frac{d^2 x}{dt^2} = F_{pres} + F_{fric} + F_{spr} \quad (8)$$

where m is the mass of the spool. F_{pres} , F_{fric} , and F_{spr} are the forces due to the pressure force, the friction force and the spring force respectively which are given by:

$$F_{pres} = \int_A p dA \quad (9)$$

$$F_{fric} = \int_A -(\tau dA) + \mu \cdot N \quad (10)$$

$$F_{spr} = (-kx - F_{preload}) \quad (11)$$

Where A is the area of the spool, p is the pressure, τ is the shear stress, μ represents the coefficient of friction (both static and sliding friction are considered) and N is the normal force. For the spring, both the spring constant (k) and the preload force ($F_{preload}$) are provided as inputs.

The convergence of the predicted solution is automatically ensured by the solution algorithm at every time step and is specified to be one order of magnitude decrease in the residuals. In addition, to obtain a periodic solution, the certain predicted quantities such as pump outlet pressure, valve position and other outputs are monitored from one crankshaft cycle to the next.

MODEL ATTRIBUTES

The flow was assumed to be transient (crank angle time scales), three dimensional (3D) and turbulent. The crank angle time dependence stems from the fact that relevant lube system events such as journal orbits and lifter reciprocating motions follow the time scales of crank shaft rotation. Importantly, the bearing models were 3D CFD models in which the complete Navier-Stokes equations (mass, momentum and energy) were solved without resorting to simplified formulations of these equations such as the Reynolds equation. The orbital paths of the journals within the bearings as well as bearing deformation due to combustion and other loads were inputs to the model. The journal bearing film thicknesses in the CFD simulations are load dependent and change through the rotation of the crankshaft. The crank angle dependent eccentricity of the journal bearings are provided as an input to the model in the form of the X and Y displacement of the shaft axis from the concentric position. The deformation profile of the bearings was provided in a table format which included the radial deformation (dr) of the bearing as a function of the degree along the circumference (θ) of the bearings.

Leakage flows through micron scale, dynamic clearances (seen in bearings, pumps, valves etc.) are rigorously modeled. The clearances are meshed using a structured mesh and then connected to the rest of the mesh using the mismatched grid interface (MGI). The pulsatile motion of the pressure regulation valve was predicted by strongly coupling the flow equations to the force balance on the plunger. The motions of piston pins are provided by the kinematic equation for piston motion.

$$x = c \cdot \cos(\omega t) + \sqrt{l^2 - [c \cdot \sin(\omega t)]^2} \quad (12)$$

Where c is the crank radius, l the con-rod length and ω is the rotational speed of the engine in rad/s.

The flow through bearings is particularly complex, influenced by geometry, physics and numerical challenges. The geometry challenges include (a) the combined orbital motion and rotation of the journal within the main and cam bearings, (b) complicated journal motions within the con-rod bearings and piston pins, (c) the deformation of bearings due to combustion and other loads, and d) transient flow through very narrow leakage paths of the order of microns within bearings. From a physics standpoint, the challenges include (a) cavitation of lube oil, (b) non-condensable gases going in and out of solution (c) reverse flow of crankcase gases back into bearing clearances and (d) non-Newtonian considerations at low temperatures to name a few. From a numerical standpoint the CFD model should meet the requirements of industrial product development, namely, (a) transient, moving, deforming meshes of good quality, (b) give predictions of acceptable engineering accuracy which at a minimum should give reliable design A versus design B distinctions, (c) produce converged solutions within reasonable run times and (d) do the above in a robust manner without the simulations diverging or blowing up. Each of the above listed items represents a challenge by itself. Thus the inherent complexities in bearing flows would clearly point to the importance of a three-dimensional treatment for bearings.

Mesh Creation

The fluid regions were extracted within the CAD software and imported into the CFD software, Simerics-Sys/PumpLinx. The meshing process involved creating a volume mesh and projecting it on to the CAD surfaces within Simerics-Sys/ PumpLinx. Such an approach is tolerant to CAD surface imperfections such as unwanted gaps, overlaps, holes etc., unlike the typical meshing processes in which the CAD surfaces need to be cleaned and a surface mesh created prior to creating the volume mesh.

Good quality, transient, moving, deforming meshes of predominantly hexahedral cells were automatically created using the body-fitted binary-tree approach.

The model was filled with (predominantly) hexahedral cells as shown in Figure 2. Figures 3a and 3b show the mesh used in the different components of the model. A total of 8 million cells were required for grid independent results.

Stand-alone CFD models (steady state and transient) were created for individual components in order to closely assess the dependence of the results to the grid size. While all the grid independence studies were not included in this paper for brevity, a couple of studies are explained in greater detail. For meshing the bearing film thicknesses a comparison was performed with an analytical solution for volumetric flow through thin gaps between two plates and the results from this have been published in [22]. The results become mesh independent as long as the film thickness is resolved by three cells or more. In the present work five cells were used for the bearing film thickness (as shown in Figure 5), which is well within the mesh independent domain. Similarly grid independence studies were also performed for the gear pump mesh. The volumetric flow ripple for three different meshes is shown in the Figure 2. Documents