Design Innovation Through High-Fidelity Simulations

Parker

ENGINEERING YOUR SUCCESS.

Introduction

Jagan Gorle, Ph.D.

Principal R&D Engineer at Hydraulic & Industrial Process Filtration Division of Parker Hannifin in Finland. He owns the technology roadmap and leads the innovation process. His years of industry and research experience were established in the engineering of various fluidic systems.

Determining a better structural design for engineering systems is a never-ending problem. It is imperative to explore all possible dimensions of associated physics for the best design solution to realize the reliable operation while maintaining structural integrity. The effectiveness of virtual engineering, including computer simulations in this context, has been demonstrated in several applications, thanks to *high-performance computing* capabilities. One such example is the design of hydraulic tanks, which receive foreign matter of different physical and chemical properties through the return line flow. Although the solid contamination can be separated from the oil stream using a return line filter, the air bubbles inevitably pass through the filter and enter the tank, and then through the suction line. Also, a sudden drop in oil pressure in the tank can lead to cavitation and increases the chance of bubbles in the tank. On the other hand, the return line fluid usually has the temperature corresponding to the system operating conditions which is imparted to the tank's internal structure, while the external surface of the tank is exposed to ambient conditions. This creates considerable temperature gradients in the tank's structure. The structural behavior of the tank in response to the flow pressure as well as the variable thermal loads thus constitutes a multiphysics problem with the aspects of sloshing, bubble motion, turbulence, heat transfer, and structural dynamics. In this context, virtual engineering practices during designing and redesigning phases have become decisive in reaching the final outcome.

Case Study

The inlet configuration of a hydraulic tank has a direct influence on the overall flow pattern and affects the force acting on the internal surface of the tank. To examine this, a tank made of Polyamide 66 is considered. The tank is filled with ISO VG 32 oil in 2/3rd of volume. A mixture of oil at 0.96 kg/s and air at 5E-7 kg/s from the return line enters the tank through the funnel. The suction line of the tank which is assimilated as constant pressure zone serves as the outlet of the flow domain. Due to the spacing and operational constraints, the geometry of the tank is fixed with a possibility to alter the funnel positioning. Therefore, the following five funnel configurations are considered to investigate the effect of inlet properties on the flow pattern.

Case 1 - Funnel @ 0º: Reference design, which has an angled open bottom so that the flow enters the tank in the downward direction.

Case 2 - Funnel @ 90º: Funnel is turned by 90º in the clockwise direction viewed from the top view.

Case 3 - Funnel @ 180º: Funnel is turned by 180º.

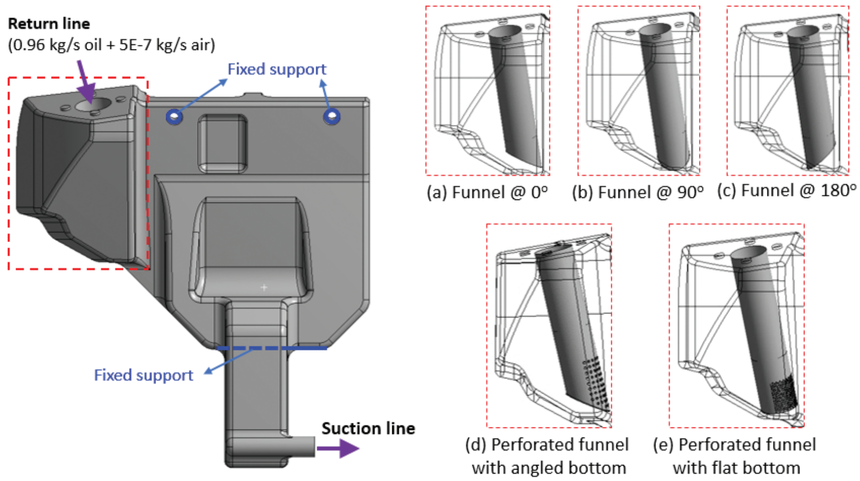


Figure 1. Tank design under study. Five different funnel configurations were examined. (a-c) The funnel with angled opened bottom, turned in the clockwise direction by 0°, 90° and 180° respectively.
(d) The funnel has a closed bottom with the perforations on one side.
(e) The funnel has a flat bottom with compacted perforations.

Case 4 - Perforated funnel with an angled bottom: Funnel with the same orientation as in **Case 1** has closed bottom. Flow enters the tank normal to the funnel's axis through the perforations on the funnel.

Case 5 - Perforated funnel with a flat bottom: Same as **Case 4** but with a flat bottom. Perforations are clustered in a smaller surface area.

Fluid-Structure Interaction (FSI) Model

While the tank's geometry is retained for the analysis of computational solid dynamics (CSD), the internal volume of the tank is extracted for the study of computational fluid dynamics (CFD). The finite volume discretization for the CFD study comprises unstructured tetrahedral cells with a fine resolution on the geometrical curvatures and propinquity regions. Homogeneity of the distribution of mesh nodes and undistorted cell creation is ensured by thorough control over the quality parameters. The finite element model for structural analysis considers the tank's geometry without the funnel as the objective is to evaluate the response of only the tank's structure. The mesh consists of fine tetrahedral solid elements to capture the small and complex geometrical features. The structure of the tank is assumed to be isotropic and homogeneous linear elastic in nature.

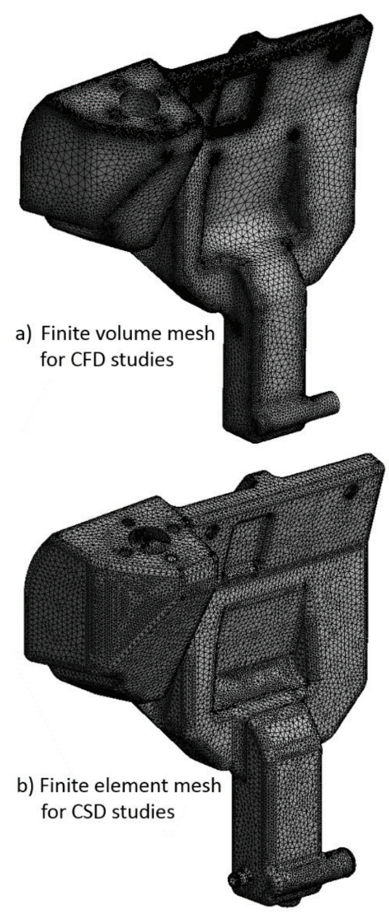


Figure 2. Computational meshes for CFD (top) and CSD (bottom) analysis.

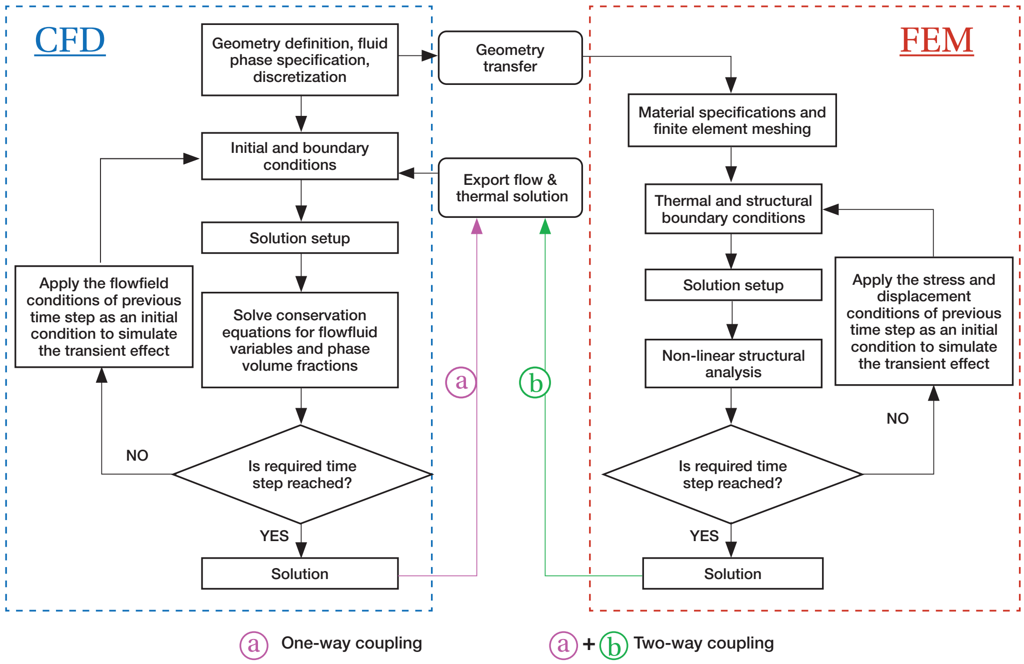


Figure 3. Workflow of fluid-structure interaction

The fluid temperature and flow conditions in the tank has a considerable effect on the structural response. The resulting stresses and deformation of the tank walls are directly governed by the interaction of turbulent two-phase flow and the structure. While the fluid flow is simulated by defining URANS solver, the structural deformation is computed by means of the mechanics of flexible solids, assuming a linear elastic behavior of the material. The interaction between the fluid and solid mechanics is modeled by a constraint boundary condition for CSD solver, the numerical value of which is given by CFD solver. Because the coupling is completely localized on the internal skin of the tank, this surface acts as an interface between the fluid and solid domains. The Volume of Fluid (VOF) method is used for modeling the two-phase flow with the definition of turbulence using Standard k-ε model. The pressure on the walls of the fluid domain is mapped, interpolated and exported to the structural mechanics' solver as the pressure, statically acting on the internal surface of the tank.

Results

Effect of funnel positioning

With 0° turned funnel, the oil stream undergoes a swirling action in the downstream of the funnel to make a quick turn through the vertical channel '*c*' and then to the suction line. This passage is characterized by a streamline cluster, while the right end of the domain '*e*' has fewer streamlines. The swirling becomes more profound as the funnel is turned in the clockwise direction by 90° and 180°, and possibly enhances mixing in the dead zones. This promotes the recirculation of flow in region '*r*'. One of the requirements of hydraulic tank design is to maximize the use of the tank's volume without static zones. To this end, the funnel at 90° is more effective than the other two designs. The preliminary qualitative observations from the steady-state contour plots reveal that there is no appreciable sloshing in the tank. The amount of air reaching the region '*e*' is reduced with the turning angle of the funnel. When the funnel is at 0°, the confined flow passage at the bottom of the funnel causes the oil stream to impinge on the solid surface, which involves an abrupt energy exchange. The air retains the momentum to move wider in the tank. With the funnel at 90° and 180°, the maximum volume fractions of the air appear near the funnel with a reduced void in the downstream locations.

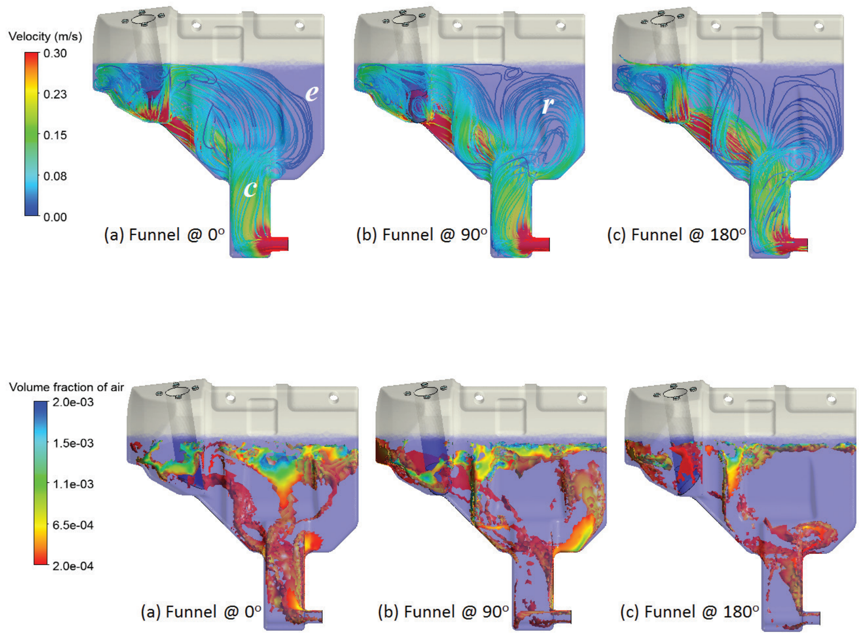


Figure 4. Oil flow streamlines (top) and air volume distribution (bottom) in the tank with different funnel positions.

The strength analysis based on the CFD-predicted pressure which is imparted to the internal walls of the tank includes the magnitude of the resulting stresses and deformation of the structural material, which enables us to identify the critically loaded/displaced locations and evaluate the structural safety. Irrespective of the design, the location l2 is the most vulnerable, followed by l3, as these are the farthest away from the fixed supports. The designs, however, differ significantly from each other in terms of the magnitude of displacement.

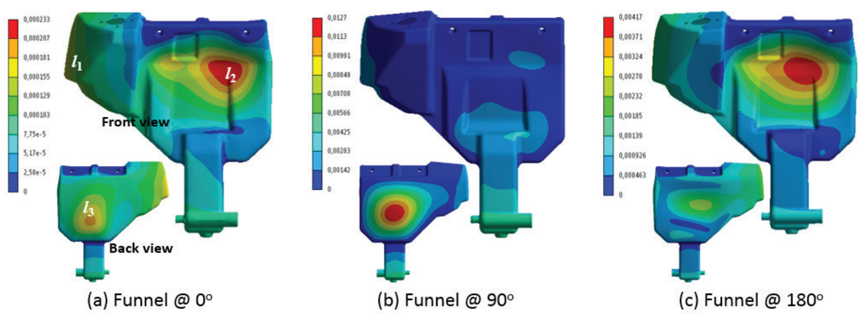


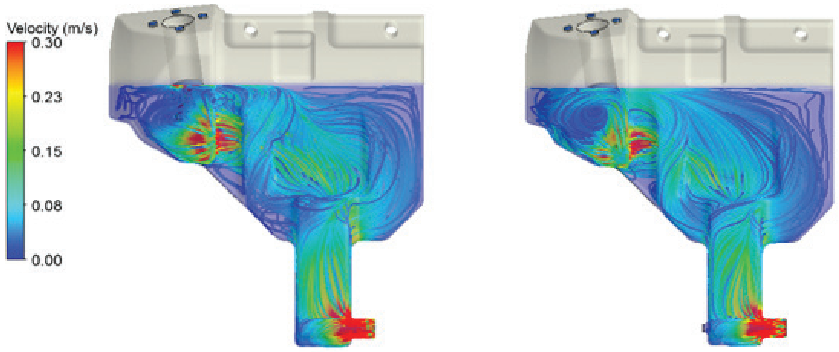
Figure 5. Structural deformation in m resulting from the flow dynamics in the tank.

The effect of funnel position on the shear stress distribution is computed from the surface integral of wall shear stress (WSS). Although the effective flow sectional area before entering the tank is the same in **cases 1-3**, the angular orientation of the funnel opening has a significant impact on the skin friction of tank walls. The half-turned funnel in **case 3** has the lowest stress, which is approximately 19% less than the unturned case and 22% less than the quarter-turned case. Furthermore, the heat transfer is computed from the total heat transfer rate across the fluid-solid contact, which follows the same trend as surface integrated WSS.

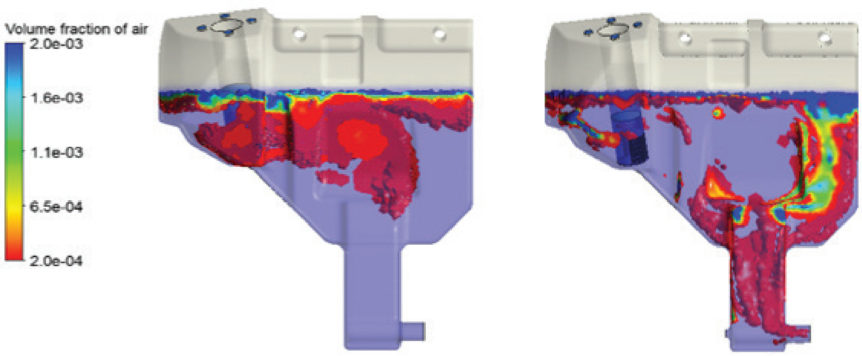
The funnel positioning in **cases 1-3** has a considerable effect on the thermal convection phenomenon. Funnel turned by 180° exhibits the lowest rate of heat transfer of 235 W, which is approximately 18% lower than that in **cases 1 and 2**.

Effect of Perforation

Figure 6. Oil flow streamlines (top) and air volume distribution (bottom) in the tank with the perforated funnels



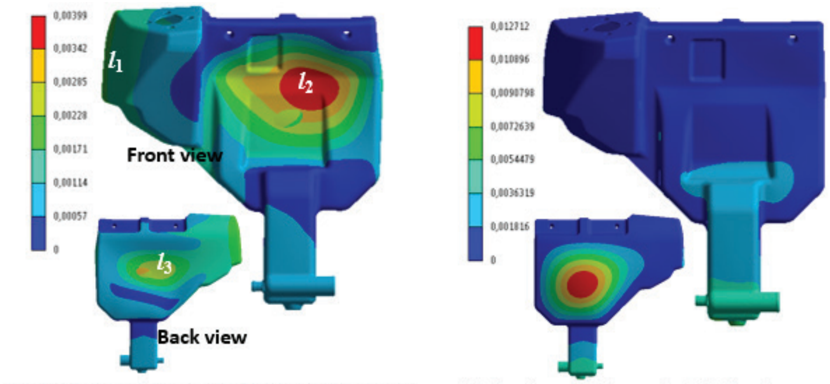
(a) Perforated funnel with angled bottom (b) Perforated funnel with flat bottom



(a) Perforated funnel with angled bottom (b) Perforated funnel with flat bottom

The funnel designs with the flow exiting through perforations have a significant impact on the flow momentum, which results in an improved oil stream distribution in the tank. Unlike the bottom opened funnel **(cases 1-3)**, the uniformity of the flow pattern certainly increases with the perforated design, e.g., **cases 4 and 5**. It is evident that the funnel design in **case 4** causes the secondary phase to reach the oil surface most efficiently compared to all other cases, resulting in only 13% of air going into the suction line. In **case 5**, closely spaced nozzles lead to a quicker mixing of emanating jets and hence a faster momentum exchange. This effect is also visible in the surface integrated wall shear stress and heat transfer characteristics, which are substantially higher than all other cases. The contracted flow injection through the funnel nozzles and hence increased mass flux in this case results in a higher heat transfer rate. This is consistent with the penetration theory. In spite of lower open area through perforations in the case of the angled bottom versus that of flat bottom, the difference of the nozzles' size and their spatial distribution leads to both designs having almost identical heat transfer rates.

The angular pattern of the nozzles in case 5 creates much different structural dynamics than in case 4. With more evenly spread flow in the tank, the maximum deformation with the angled-bottom funnel is three times less than that in the case of the flat-bottom funnel. Another major difference between the two designs is that the location of maximum displacement is on the opposite sides of both cases, e.g., in l2 with the angled-bottom funnel and at l3 with the flat-bottom funnel. The peak stress occurring in any case is, however, less than the ultimate strength of the material, which ensures the operation of the tank is in the elastic region. Although the factor of safety below 3 is observed in all cases, this is identified at streaks of certain corners of the tank. The overall safety factor is 15 or above, which confirms the structural safety of the tank.



(a) Perforated funnel with angled bottom (b) Perforated funnel with flat bottom

Figure 7. Structural deformation of the tank with perforated funnel on the return line

Conclusions

Major engineering trends in recent times such as hybridization, 5G, autonomous systems, etc. are transforming the products and processes in many disciplines. Take for instance mobile hydraulic systems, where the manufacturers aim at reliable, efficient and yet cost-effective solutions. The predictive capabilities of state-of-the-art computational tools at real-world accuracy enable the designers to understand the coherent phenomena and make a rationale-based decision in developing the systems. These physics-based simulation technologies are likely to promote collaborative product development practices between CAD, PDM and supplier management systems, and hence realize the innovation in response to engineering requirements. The case study presented in this document shows how the computer simulations could be used as design instruments where the explicit models of fluid and structural mechanics are critical to drawing scientific conclusions. Such virtual reality offers much faster and cheaper of several orders of magnitude than conventional prototyping and testing. In addition, these simulations also suggest suitable materials, appropriate tolerances and adequate manufacturing methods that would lead to the efficient management of engineering resources.

Parker Hannifin Corporation
Hydraulic & Industrial Process-Filtration Division EMEA
Industriepark Kleefse waard Westervoortsedijk 73 LB Arnhem,
The Netherlands 6827 AV
phone +31(0) 26 376 0376
filtrationinfo@parker.com