CFD Model to Reduce Automobile Development Costs Related to Refueling System

T. McKay Stoker, Mangesh Dake, Bret Windom – *CSU Mechanical Engineering Department*

Marc Henderson – *Honda R&D Americas, Inc.*

mckay.stoker@colostate.edu

ABSTRACT

Testing procedures for automobile refueling systems can be costly. To reduce the amount of testing during the design of refueling systems, car manufacturers desire a CFD tool predictive of system performance. The potential of such a method is demonstrated here.

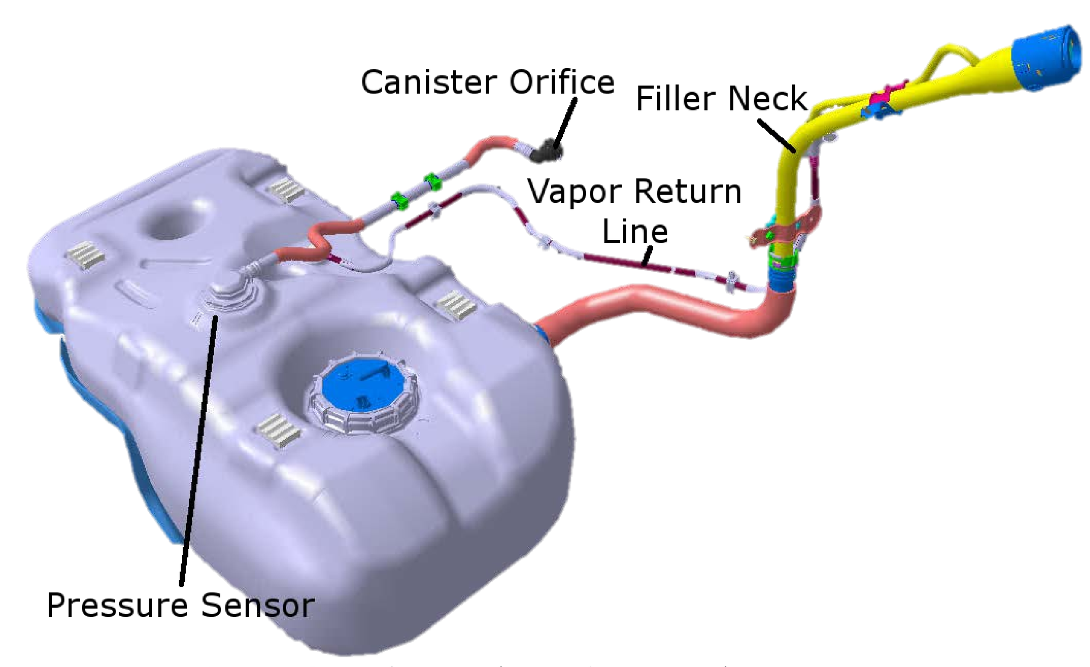

Figure 1: CAD of tank used in this study, with critical components labeled

SPECIFIC AIMS

1. Develop a CFD (Computational Fluid Dynamics) model in commercial software
   a. Begin with just the filler neck
   b. Study most appropriate physics models to use
2. Check capability of model by comparing results with well-controlled experiments
   a. Correlation with flow through just the filler neck
   b. Correlate to full system testing by filling tank from empty at three flow rates (4, 10, and 14 GPM), recording tank pressure. Compare with CFD calculated pressures.

Phase I: Correlation of Filler Neck Flow

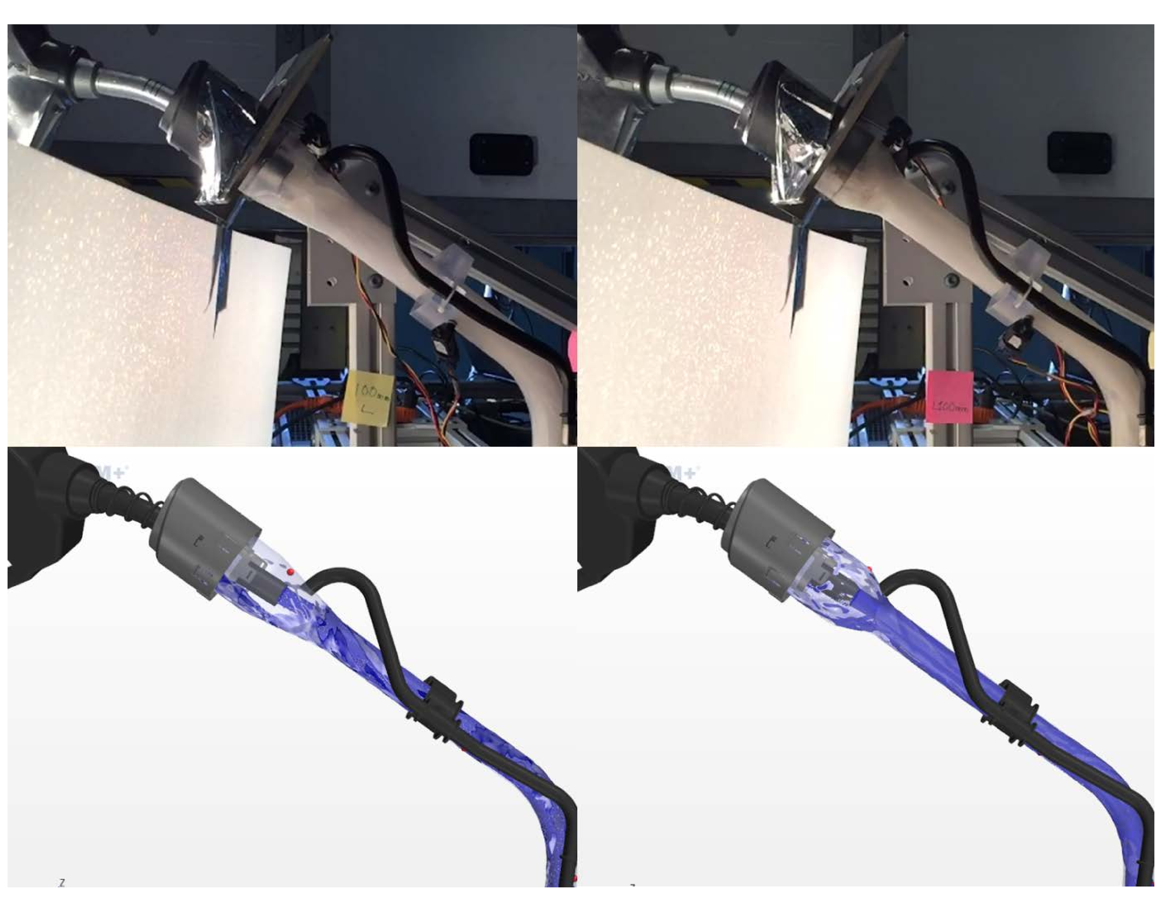

Figure 2: Visual comparison. Left two images are from the Passing pipe; right two are from the Failing pipe.

- Simulated and tested two different filler necks at several nozzle orientations
- One showed instances of very early-clickoff (called Failing pipe or NoGo) while the other did not (called Passing pipe or Go)
- Good visual correlation showing recirculation in Failing pipe
- Also obtained simulation metrics showing greater amount of gasoline hitting the nozzle for the Failing pipe, responsible for clickoff

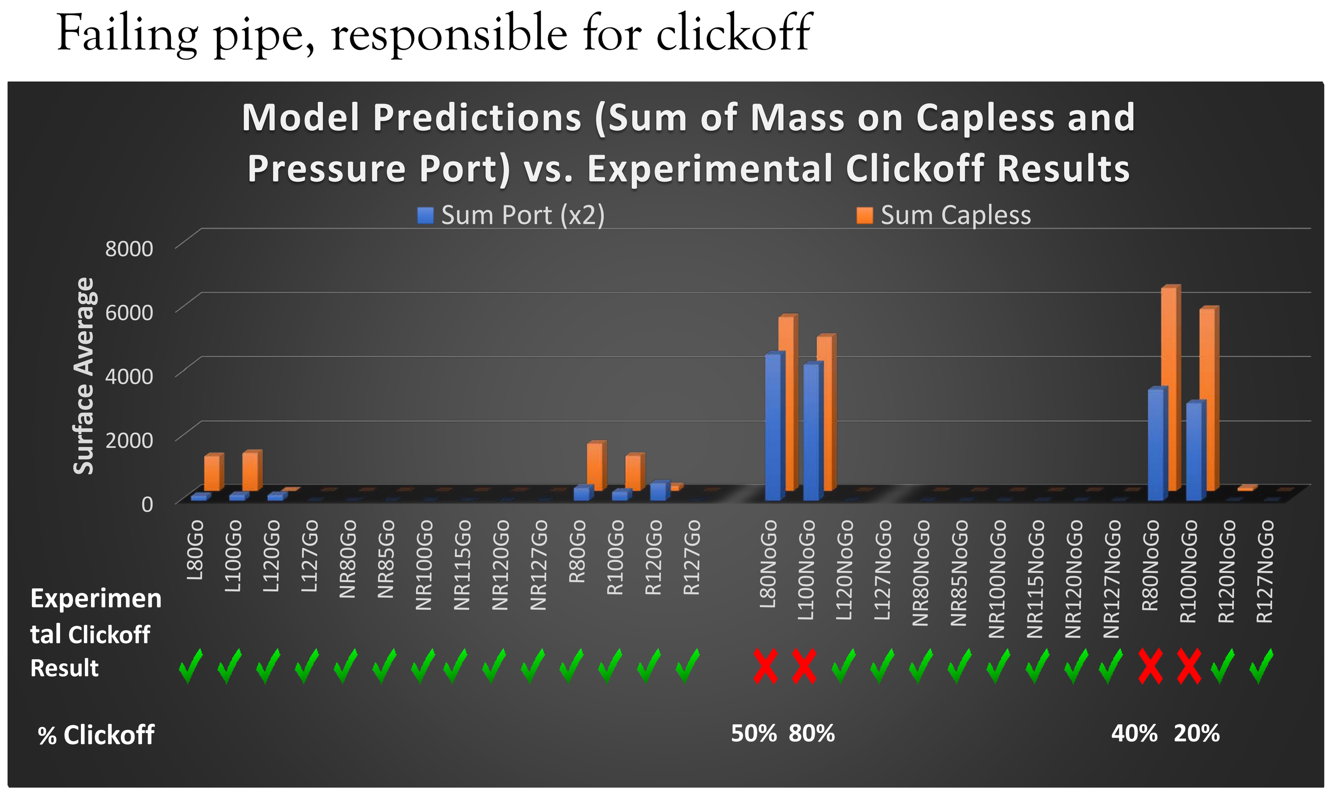

Figure 3: CFD metrics of fuel mass hitting critical components versus experimental clickoffs

Phase II: Correlation of Tank Pressure During Full System Filling

- To neglect evaporation in the full tank model, Stoddard solvent was used instead of gasoline due to its much lower volatility in experiments and simulations
  - Reid Vapor Pressure of 0.3 psi for Stoddard fluid instead of 7 psi for gasoline
- Experiments used to develop boundary conditions for vapor return line and canister orifice in CFD
- CFD pressures are higher than experiment by: **154 Pa (4 GPM), 101 Pa (10 GPM), and 150 Pa (14 GPM)**
- The offset of CFD pressure is not a percentage of the measured value but rather a consistent value
- The constant offset means simulation can provide a good estimate of tank pressure
- Simulation is computationally expensive. The fastest fill takes approximately one week to run on 1024 cores.

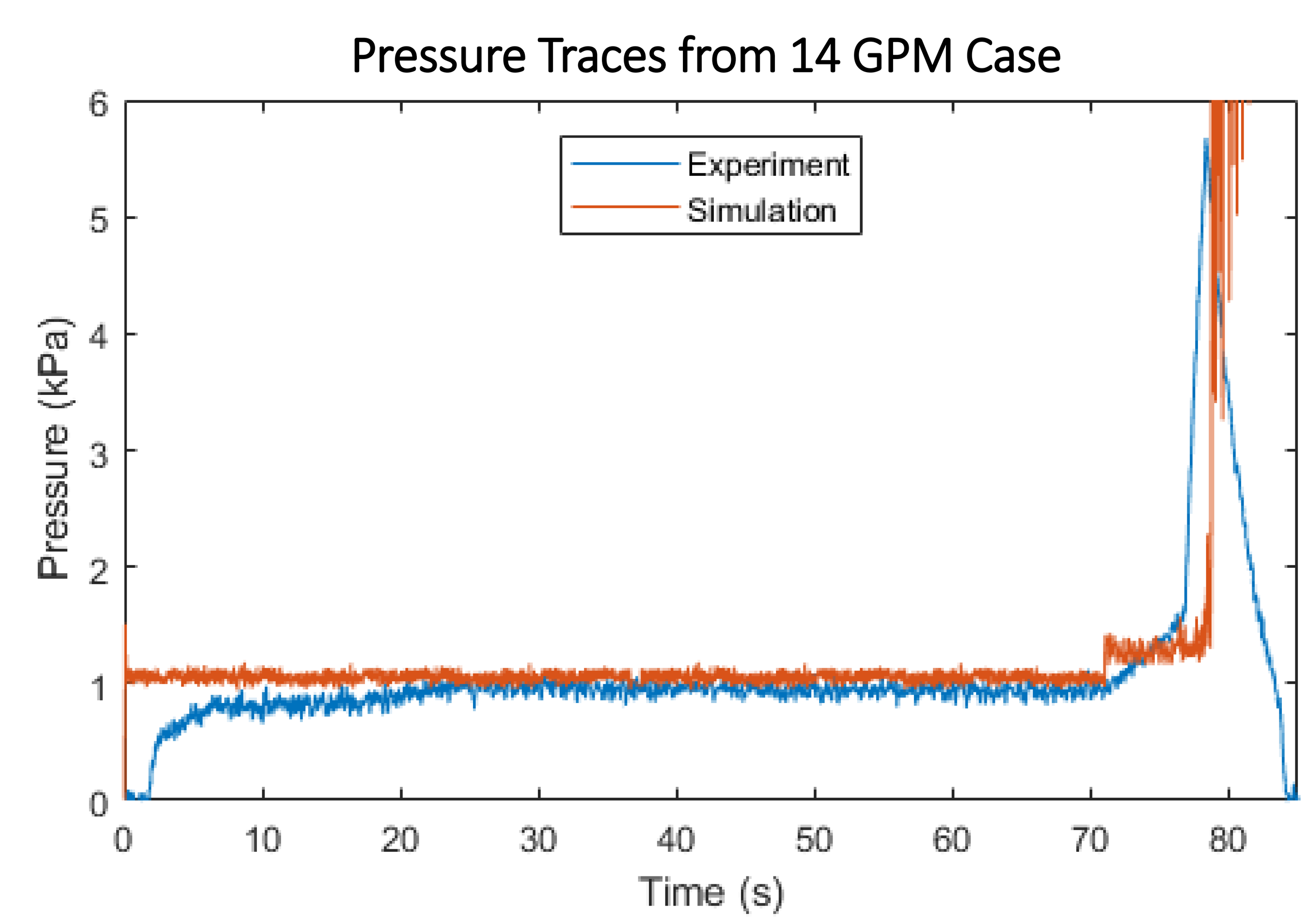

Figure 4: Comparison of tank pressure traces from experiment and simulation for the 14 GPM case. CFD takes a few more seconds to fill due to slightly larger volume in CAD. Initial pressures differ but steady state shows good agreement.

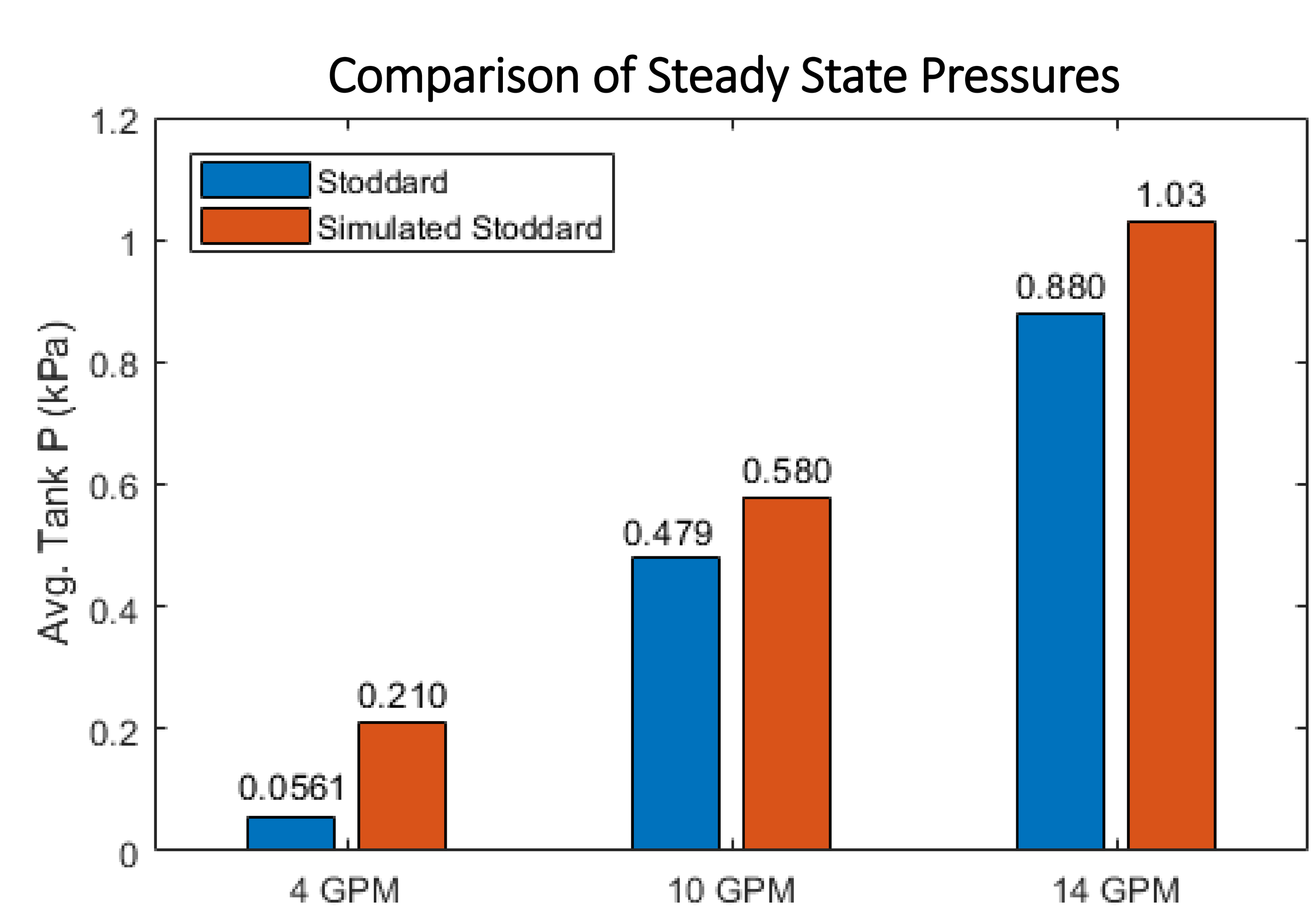

Figure 5: Steady state tank pressures from simulation and experiment. Pressures from each method trend the same as flow rate increases. CFD pressures are higher but the offset is fairly constant and not a percentage of the measured value.

CONCLUSION

- Model can predict tank pressure when evaporation is neglected
- Pressure is critical to system performance. Therefore, CFD has potential to predict performance of a new design.
- Future work will focus on adding evaporation physics to simulate gasoline while ensuring accuracy continues
- Also, the model must be simplified to see significant time benefits over testing

HONDA

Honda R&D Americas