USING COMPUTATIONAL FLUID DYNAMICS TO SUPPORT BLUE GRASS CHEM DEMIL PLANT THERMAL TREATMENT PROCESS EQUIPMENT DESIGN

JON BERKOE
BECHTEL NATIONAL, INC.
SAN FRANCISCO, CA
415-768-2149 (P); 415-768-1794 (F)
jberkoe@bechtel.com

YO CHAN KIM
BECHTEL NATIONAL, INC.
FREDERICK, MARYLAND
(240) 379-3101 (P)
Ykim2@bechtel.com

JOHN URSILLO
PARSONS INFRASTRUCTURE AND TECHNOLOGY, INC.
PASCO, WASHINGTON
(509) 547-7724 (P)
john.ursillo@parsons.com

Introduction

The chemical agent pilot plant to be built at Blue Grass Army Depot in Richmond, Kentucky (BGCAPP) is a non-incineration process using water-based neutralization reactors and a first of a kind piece of process equipment known as a Metal Parts Treater (MPT) to decontaminate drained and flushed projectile bodies to a level that would be suitable for unrestricted release. The MPT is essentially an oxygen depleted, superheated steam purged autoclave-like device measuring 6 feet 6 inches in diameter and approximately 17 feet in length, and is designed to decontaminate a chemical agent inside steel munitions bodies (referred to in this paper as munitions) using a thermal treatment process. The chamber's energy source is a pair of external induction coils that heat the outer surface of the Inconel shell to an operating temperature of approximately 1,450°F. Trays of munitions (as well as other metal materials or contaminated solid waste) are processed in a batch continuous mode to achieve decontamination temperatures. The design requirement for the MPT operation is that the munitions need to maintain a minimum temperature of 1,000°F for 15 minutes to meet regulatory criteria for decontamination. The decontaminated munitions are collected and are shipped offsite for disposal.

Due to the first-of-a-kind nature of the MPT, a design strategy was employed in which state-of-the-art computational fluid dynamics (CFD) modeling was integrated with pilot-scale unit testing to produce a validated CFD model that could be used to predict the performance of the full-scale MPT prior to construction. The challenge for the design team was to confirm that all locations on the munitions met the “unrestricted release” design criteria, which is difficult to verify with instrumentation due to the complex geometry, with many munitions on the tray obstructed from the radiative heat emitted by the wall.

The scope of the CFD modeling study took the effort through two distinct phases:

1. The initial phase of CFD modeling involved the simulation of a pilot-scale known as the Technical Risk Reduction Project (TRRP) test unit, which was essentially a fully

operational half scale unit. This model was intended for comparison to scale-model testing in order to demonstrate the predictive accuracy (i.e., validation) of the CFD model and to calibrate the CFD model against representative measurements obtained from the TRRP scale-model unit operation. The CFD model was designed to simulate the heating operation for 155mm munitions and tray. These are the heavier weight design between the two types of similar-sized projectiles (the other are 8 inch) and thus result in longer heat up times.

2. Convert the validated CFD model of the TRRP design into a scaled-up model to be used to simulate the full-scale First-Of-A-Kind (FOAK) MPT design. The FOAK unit is essentially a 2:1 scale-up from the TRRP with nearly identical geometry. This model was intended to be used to calibrate the performance of the full-scale MPT for the 155mm munitions trays as well as the faster heating 8 inch munition trays.

<u>Methodology</u>

The main components of the MPT are the inlet airlock, main chamber, discharge airlock and cooling chamber. Figure 1 shows the Baseline CAD model for the main chamber MPT unit without trays and munitions, along with the scaling comparison between the cross sections of the TRRP and FOAK MPT units. The CFD model consisted of the full MPT Main Chamber and simulated a continuous operation with the trays positioned in Zone 1 (input or feed half) and Zone 2 (outlet or discharge half) without transfer between zones. The simulation assumes Zone 2 contains a tray of munitions that has already been heated to temperature while Zone 1 contains a tray that is the subject of the model simulation, entering the MPT with ambient temperature as its initial condition.

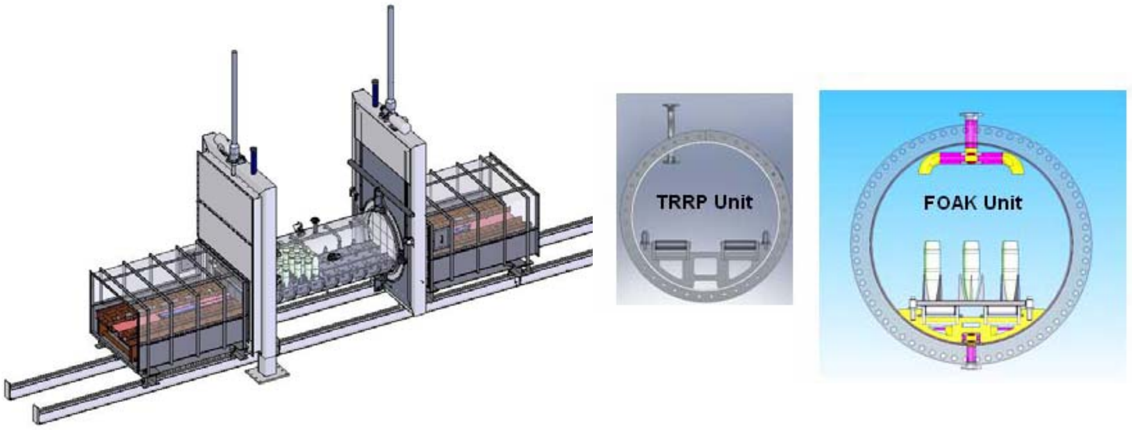

Figure 1 – MPT Configuration; TRRP and FOAK Unit Scaling

The TRRP tested tray designs for two configurations – one with the munitions nose (ogive) up, and one with the nose (ogive) down. The CFD model validation was performed for the nose-down configuration. Figure 2 shows the CFD model geometry of the MPT Test Unit chamber with forty 155mm projectiles on two trays. The geometry of the system is modeled in detail in order to accurately simulate the effect of radiation shadowing caused by the “fin-shaped” munition supports (referred to as “holding fins”) and munitions support plates. To simplify the munitions model the burster well located inside the munitions is not included (although its mass is accounted for by adjusting the density of the munition wall).

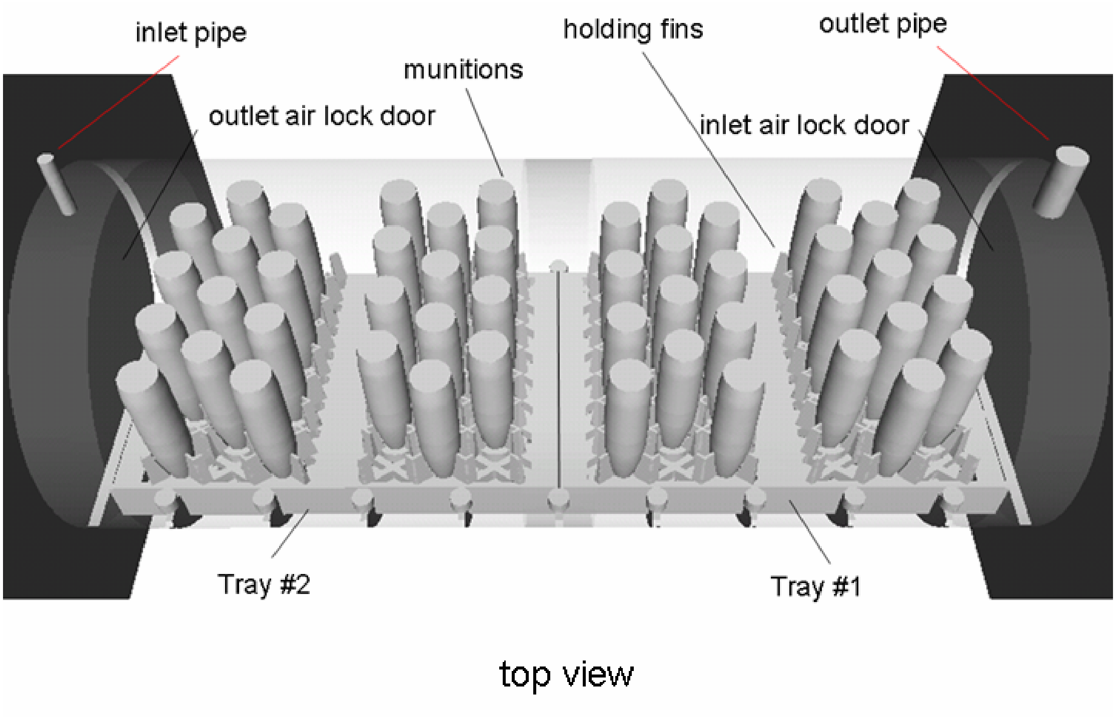

Figure 2 - CFD Model Geometry for TRRP Model Validation Study

The physical properties used in the models were based on the actual material properties of the TRRP test unit for the munitions, tray, steam, etc. The key thermo-physical properties from a transient heat-up and radiation heat transfer perspective are the specific heat and emissivity of the munitions and trays. These properties for the munitions vary with temperature and were obtained from experiment.

The model strategies and methodology employed for the CFD analysis simulates the turbulent steam flow, convective surface heat transfer, conduction heat transfer in the tray and munitions, and radiation heat transfer between all reflecting surfaces, including surface-to-surface shadowing. The commercial CFD solver AcuSolve (www.acusim.com) was used. AcuSolve is a general-purpose finite element based incompressible flow solver. Its equal-order pressure/velocity coupling results in fast convergence and it was developed to run very efficiently on parallel platforms - ideally suited for large models with complex geometries and the need for high accuracy solutions. The computational grids were typically on the order of 3 million cells. The simulations were run on a Hewlett Packard 35 parallel computing cluster, and typically required about 24 hours to complete. This is extremely fast performance for a model of such size and complexity.

The model boundary condition temperature for the MPT wall was initially set using a constant 1,350°F wall temperature, as this was the basic operating condition for the test unit during experimental operation. However, this was subsequently adjusted during the early testing to be a two-zone wall temperature - one for Zone 1 (1,250°F) and one for Zone 2 (1,350°F) based on the measured values obtained during preliminary testing.

Temperature history data was plotted for all simulation runs and compared to test measurements for locations where test thermocouples were employed. The thermocouples were placed strategically based on knowledge as to where the “cold spot” would occur, i.e., the surface location on the munition that was last to reach 1,000°F in the entire tray. One of the key early findings from preliminary CFD modeling studies was that the cold spot location consistently occurs in the same localized region in the most isolated spot on the most shadowed munition. Additionally, one additional representative munition away from the "coldest" munition was equipped with thermocouples for comparison. Figure 3 shows a TRRP tray loaded with 155 mm munitions alongside a close-up view of the slowest heatup munition with thermo-couple installation in the vicinity of the "cold spot". Additionally, a thermocouple was also placed on the tray near the base of this munition in order to compare the tray temperature to the munition.

Figure 3 – TRRP Munition Test Tray with Coldest Munition Thermocouple Installed

Results

The cold spots in the model were localized around tray support fins caused by shielding of radiative heat transfer to the base of the munition. Figure 4 shows the instantaneous temperature profile on the tray and munitions in the coldest heatup region taken from a typical transient simulation. Note the pronounced shadowing effect of the munition support fins. Figure 5 shows a temperature time history comparison between the CFD model and TRRP for one of the thermocouple locations on the munition nose. Note the shape of the heatup curve is similar for both, and the time to reach the 1,000°F temperature compared very well, a difference of less than 10%.

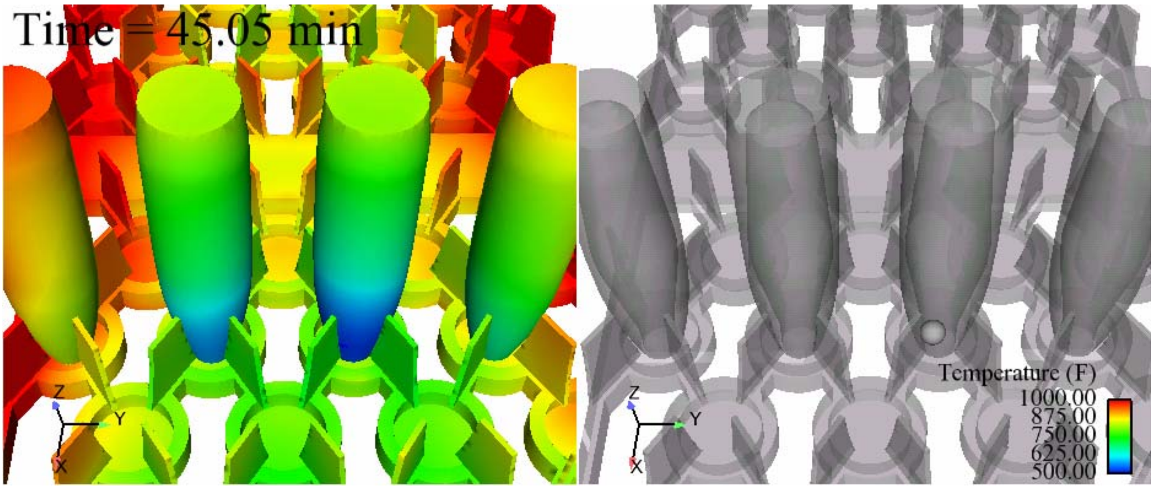

Figure 4 – Temperature Contours from a CFD Simulation Showing Slowest Cold Spot Location

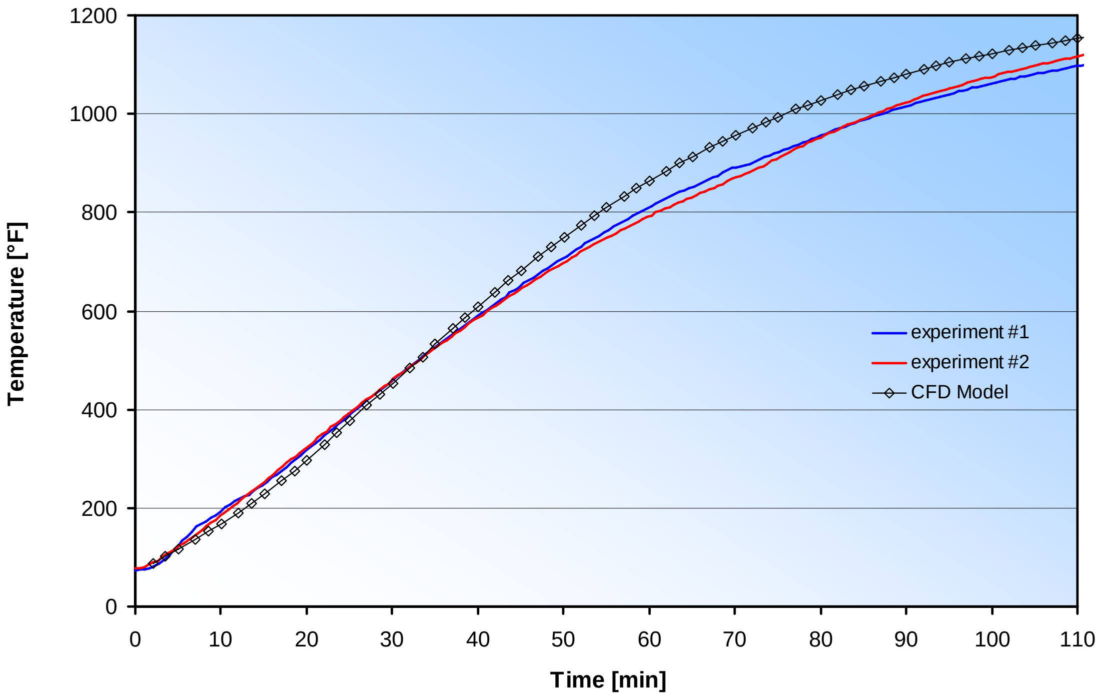

Figure 5 – Temperature History Comparison between CFD Model and TRRP Experiment at Munition Nose Thermocouple

It was observed that the heatup rate of the tray (and fins) significantly influences the heatup rate of the coldest areas of the munition, because it absorbs emitted heat first, and then emits radiation heat transfer to the shielded portion of the munition. Thus the munition "cold spot" is heated indirectly - it turns out that the thermo-physical properties of the tray (in the vicinity of the munitions support fins) play an important role in the process. This factor was considered in optimizing the final FOAK design configuration.

The finding that the CFD model heatup was faster than the experimental measurements in the "cold spot" location indicates that likely some of the actual thermo-physical properties may have varied from what was used in the model, and also that some of the geometry simplifications may have played a role in leaving out potential obstructions to the local radiant heat transfer. But overall – based on a maximum discrepancy on the order of 15% - the results of the study indicate a generally successful validation, and thus it was agreed to proceed with using CFD for calibration of the FOAK unit performance.

Figure 6 shows the temperature profiles for the FOAK unit (cold spot) including a comparison of a range of MPT wall and steam temperatures that can be obtained in the full-scale unit operation. The model shows that the TRRP-recommended increase in wall temperature setting to 1,450°F at the design steam injection temperature of 1,100°F provides a significant boost in reducing the heatup time of the "cold spot" region, which provides the design team with an operational adjustment to add to the safety margin that more than compensates for the margin of discrepancy observed in the CFD analysis for the TRRP.

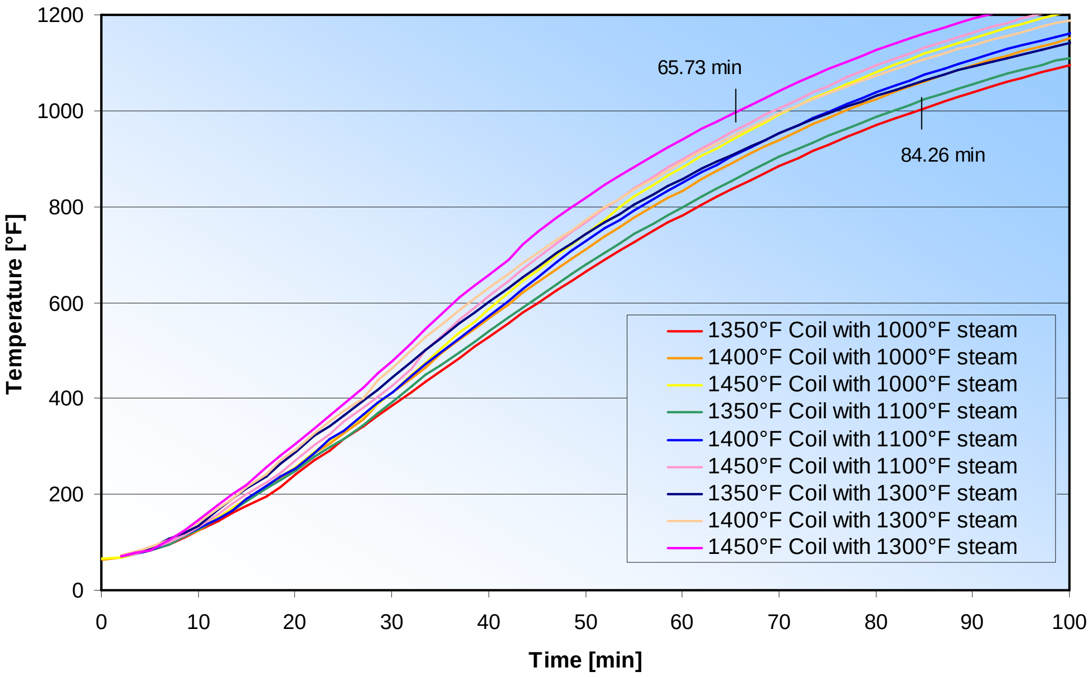

Figure 6 – CFD-predicted Temperature History for FOAK (155 mm munition tray) model

Conclusion

The CFD study showed that simulation, when effectively validated against scale-model experiments, can help in directing the design optimization process to a successful result. The model validation is essentially accomplished by obtaining acceptable agreement in a scientific and statistical comparison between simulation and experiment. However, just as important as the data comparison are the key insights that are learned during the process, insights that allow equipment designers to hone in on ways to improve operational performance more effectively, and reduce the amount of "trial-and-error" re-design and rework that inevitably occur.

ABOUT THE AUTHOR

Jon Berkoe is a Senior Principal Engineer and Manager of Bechtel's Advanced Simulation and Analysis Group, Mr. Berkoe has over 20 years of experience in the engineering/construction, nuclear, and aerospace industries with extensive expertise in the fields of heat transfer, fluid dynamics, and computational fluid dynamics. He has pioneered the use of CFD on large, complex projects encompassing a wide variety of process and environmental applications. Prior to joining Bechtel, he worked for the Spacecraft Division of Lockheed Corporation and the Nuclear Division of General Electric Company. Jon holds Bachelor of Science and Master of Science degrees in Mechanical Engineering from the Massachusetts Institute of Technology.