# USING CFD TO ASSIST FACILITIES COMPLY WITH THERMAL HAZARD REGULATIONS SUCH AS NEW API RP-752 RECOMMENDATIONS

Arnab Chakrabarty, Ph.D. and Arafat Aloqaily, Ph.D.
Baker Engineering and Risk Consultants, Inc., Houston, Texas, USA

Thermal loads produced by fires from process units and equipment can be significant on process and portable buildings, especially if the building is close to the source of fire. While some structures can be good for blast resistance, they might be vulnerable to thermal hazards resulting from different types of fire. The new API RP-752 recommendations require process facilities to analyze thermal hazard to onsite buildings and assess its impact on those structures and personnel working in them. Computational Fluid Dynamics (CFD) techniques were used to assess thermal hazard resulting from horizontal jet fires and their impact on a portable building located downstream of the fire source. The results clearly show that the detailed CFD analysis can be effectively used to simulate jet fires, assess their thermal impact on portable buildings onsite, and help facilities comply with regulations and recommendations, irrespective of the geographic location, such as the new API RP-752 for USA. As expected, thermal impact increases as the distance between the fire source and building shortens. BakerRisk has a comprehensive test program in place to validate and refine the CFD simulation approach, and offer a better understanding of thermal hazard impact on structures and personnel working onsite. This assists in evaluating the protection offered by the building to its occupants.

KEYWORDS: Horizontal Jet Flame, CFD, Computational Fluid Dynamics, PHOENICS, Thermal Hazard, Combustion Modeling, Flame Length, Process Safety, Impinging Fire, Impinging Flame, API RP-752

## INTRODUCTION

To better understand and evaluate risk to personnel occupying process and portable buildings in process facilities, thermal hazard resulting from jet fires/flames should be accurately estimated. While some structures on site can be good for blast and toxic hazard, they could be vulnerable to thermal hazard, especially if the flames impinge on the structure and block evacuation/escape routes. Unfortunately, thermal hazard characteristics are not well understood and predicted, especially in horizontal jet flames, which is the subject of investigation in this paper.

Existing modeling approaches and models used to evaluate thermal hazard resulting from jet fires have been described and evaluated in a previous work (Aloqaily 2010). As pointed out in that study, some of these models such as those discussed by AIChE (CCPS 2000) and NFPA (SFPE 2002) are simple while others such as the model discussed by TNO are more comprehensive and detailed (TNO 2005). A comprehensive model, the geometric solid flame shape model, or Chamberlain's model is also used extensively in process safety applications (TNO 2005). This model is more sophisticated than the approaches presented by CCPS/API RP 521 and the SFPE. It has been developed for flame shape and radiation fields of flares (TNO 2005). Its flame is represented with a frustum of cone radiating energy from solid body with uniform emissive power. The flame length is affected by the release diameter and velocity, and wind speed and direction. A detailed description of the model with illustration examples is given in the TNO Methods for the Calculations of Physical Effects (TNO 2005). Models that focus on predicting flame length for specific materials have also been developed, such as the propane flame investigation presented in reference (Sugawa 1996).

However, these available thermal hazard models have focused on predicting the flame characteristics and its associated thermal field without considering the interaction, as shown in Figure 1, between the flame and structures/buildings affected by the flame and hence may not be suited for problems involving structure/flame interaction. The usage of Computational Fluid Dynamics (CFD) enables a more detailed look into these problems with incorporation of the interaction effects between the flame and affected structures/buildings as illustrated in Figure 2. Figure 1 does not consider the presence of the building and estimates a similar concentration of flammable gas at both the front and back of the building. Since it does not show the effect of buoyancy, the flame remains horizontal. Figure 2 illustrates that the CFD model can incorporate the presence of the building and estimates the flow of gas around the building. The approach also effectively captures the shielding effect owing to the presence of the building. Flame lift off due to buoyancy of the gas as temperature rises is also estimated appropriately in the CFD model. Capturing these dynamics allows better estimation of the effect of shielding, in order to develop exit strategies and other cost effective measures. While a CFD approach is expensive in terms of computational resources, it presents a more reliable and realistic approach to assess the thermal hazard on building/structures.

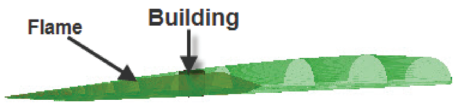


**Figure 1.** Modeling Horizontal Jet Fire in presence of a building using simplified approach

This paper presents CFD analysis results for flame impinging on buildings, and analyzes the impact of jet fires on the target buildings/structures located downstream of the fire source. The interaction between the fire and building, as well as its impact on the flame shape and thermal radiation field, are also discussed in this article.

The new requirements of the API RP-752 explicitly request facilities to assess thermal hazard to building/structures onsite (API RP 752 2009). Since current modeling approaches commonly used in process safety applications do not consider the interaction between jet fires and buildings, it is important to consider using CFD, which allows complex 3-D geometries to be simulated and provides a detailed description of the fire impact on the buildings. CFD techniques can also predict the change of temperature with time at the outer surface of the building (exposed to fire), the inner surface of the building, and inside the building, assessing the protection provided by the building to personnel working inside the building. However, this requires dynamic simulation, which is beyond the scope of this paper, and will be investigated in a future work.

## NUMERICAL STUDY

This section describes the CFD numerical modeling approach used in the simulations presented in this paper. The jet flame results discussed herein were simulated using the commercial CFD software PHOENICS, developed by CHAM, UK (PHOENICS Manual 2009). The characteristic equations defining a jet flame requires solving turbulence and combustion phenomena, simultaneously resulting in a set of complicated Differential Algebraic Equations. This makes it impracticable to solve for both phenomena directly, due to the complex interaction between turbulence and combustion (Aloqaily 2008). As a result, a set of models are used to simplify the calculations. These models are discussed in this section. The geometry used in the simulation presented here is also described in this section.

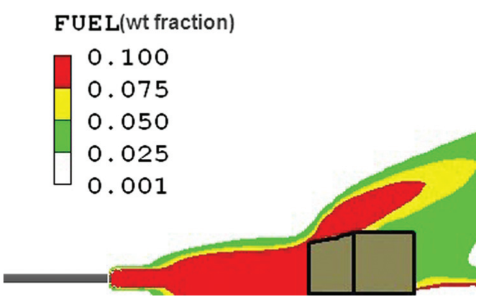


**Figure 2.** Modeling Horizontal Jet Fire in presence of a building using CFD

### NUMERICAL MODEL DESCRIPTION

PHOENICS is a commercial CFD code that can be used to simulate flow, heat, and mass transfer problems. It contains a range of turbulence, combustion and thermal radiation models suitable for simulating turbulent flames. PHOENICS implements the Finite Volume formulation with structured grids to perform the numerical calculations. It uses interpolation between cells and cells' surfaces to evaluate scalar and vector quantities at cells' centers and surfaces, respectively. A special form of SIMPLE algorithm is used in PHOENICS to handle the coupling between continuity and momentum equations, namely SIMPLEST algorithm (PHOENICS Manual 2009). Convergence can be assessed and controlled in different ways. The residuals of all variables and spot values of interest can be monitored for relevant variables in order to assess convergence. PHOENICS mainly implements relaxation techniques to control and promote convergence. Note that an automatic relaxation mechanism is available and was used in most of the calculations, but since parameters can be controlled manually as well, that was used to aid convergence in some cases.

Models for turbulence, combustion and radiation were used in combination to simplify the calculations and solve for the mixing and reaction aspects of the flow. The models are briefly described below. Detailed description of the different models, discussed above, is given in a previous work (Aloqaily 2010).

- The standard $\kappa$-$\varepsilon$ turbulence model (Launder 1982) was used to solve for turbulence. This turbulence model has proven to have accurate predictions of flows with recirculation, boundary layer, and sheer layer type of flows, and is widely used in engineering applications (Aloqaily 2008; Launder 1990; Zhou 1999; Mattick 2005).
- The Simple Chemically-Reacting System (SCRS) model, which implies mixing-controlled and fast reaction, was used to simulate the combustion in this paper. This model was chosen because the combustion reaction between the fuel used in this study (methane) and air is fast, and the combustion phenomena is controlled primarily by the mixing rate of fuel jet and surrounding air (Spalding 1971). This assumption has proven to be valid especially for highly volatile and gaseous fuels such as the one considered in this analysis (i.e. methane). It has been proven that this assumption simplifies the calculations significantly (Spalding 1979; PHOENICS Manual 2009).
- The IMMERSOL radiation model, as implemented by PHOENICS, was used in combination with the combustion and turbulent models to estimate the heat radiation from the jet flame. The model was chosen for this analysis as it can handle the uncertainty in the absorption and

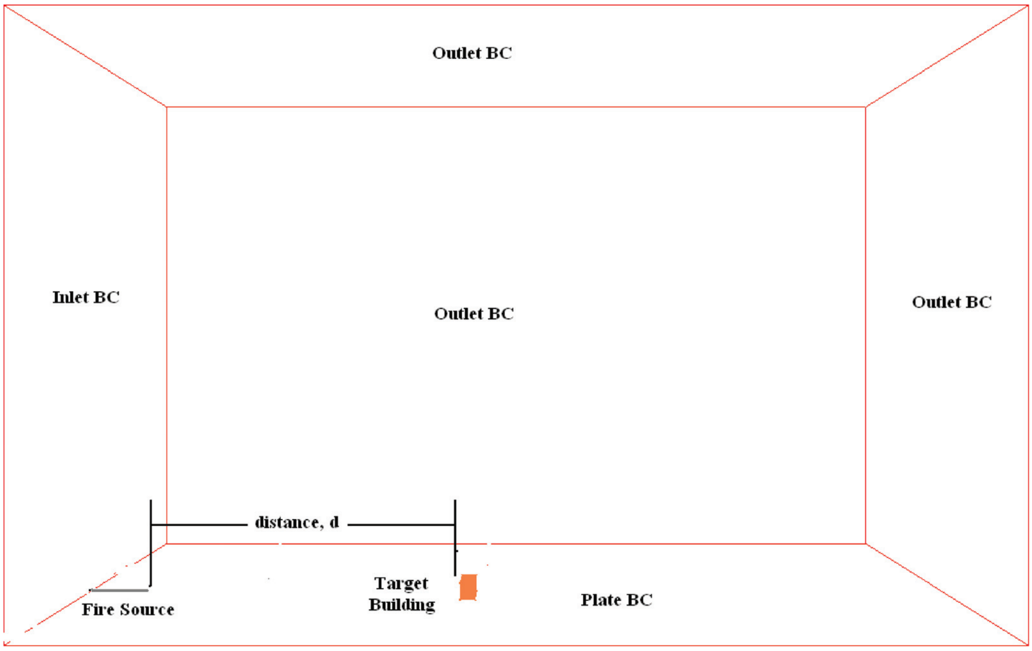


**Figure 3.** Schematic of Geometry and Simulation Domain for Fire Cases Simulated in This Paper

scattering coefficients, and is not computationally demanding (PHOENICS Manual 2009). The combination of standard $\kappa-\varepsilon$, SCRS, and IMMERSOL models provide a robust and efficient solution methodology for simulating horizontal jet flames that combines good accuracy with reasonable demand for computational resources. The results presented in this paper are generated using these three models, as implemented in PHOENICS.

## GEOMETRY AND BOUNDARY CONDITIONS

This paper focuses on the simulation of horizontal non-premixed methane jet flame, which can result from burning of methane subsequent to its accidental release from flammable processing units/sources. The fire is impacting a target building located downstream of the fire source. The formulation of the problem is given in this section.

Figure 3 illustrates the simple geometry and the simulation domain for the impinging flame. Figure 4 is a closer look at the geometry of the target building, showing its dimensions. The target building represents typical Blast Resistant Modular Building (BRMB) used onsite, which is common in process facilities. Three cases have been modeled in this study, covering a building located at different distances downstream of the fire source.

The jet fire simulations presented in this paper were all performed for methane releases from 2 inch release size at 1,000 psig and atmospheric temperature. As a result a fixed burning rate was used in all simulations. Three cases were analyzed for this fixed burning rate, and the distance between the release source and target building was varied to represent three different levels of thermal radiation on building surfaces (high, medium and low thermal impact). The high, medium and low thermal loads are represented with thermal radiation levels of 37.5 $kW/m^2$, 12.5 $kW/m^2$ and 4.0 $kW/m^2$, respectively. These thermal levels are capable of causing different levels of damage to structures and injury to personnel (as will be explained later in this paper).

The type of boundary conditions (BC) used in the simulations was identical for all cases. The jet fire is

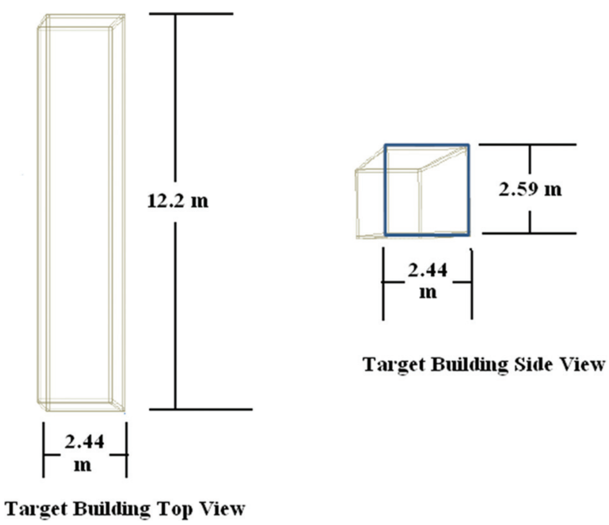


**Figure 4.** Schematic of Target Building used in Simulations Presented in this Paper

represented with a release of methane from a horizontal pipeline, where fixed mass flow rate BC was set at the exit of the pipeline. The pipeline was assumed to be at a height of 1.0 m above the ground level. A fixed velocity inlet BC was defined for surrounding air, which was assumed to flow uniformly with low turbulent intensity of 5%. The released fuel (methane) was assumed to be flowing in the same direction as the wind (no crosswind effect was considered). This results in the flame being symmetrical in shape around the plane that goes through the center of the simulation domain. This in turn implies that it is sufficient to solve the problem using half of the domain and assuming the other half as a mirror image (i.e. applying symmetry boundary condition across the calculation domain shown in Figure 3 to divide it into two). Note that although Figure 3 and Figure 4 show a complete domain, the majority of cases analyzed in this paper were simulated using half of the domain due to symmetry, which reduces the time needed for calculations and allows for higher grid resolution.

An outlet BC was set for the four other edges of the simulation domain shown in Figure 3 (in the case of symmetry, one of these edges was set to symmetry BC). Outlet BC, as defined in PHOENICS, is a fixed-pressure boundary that allows mass to leave or enter the solution domain as needed. The ground level is defined as a Plate BC with adiabatic conditions, thus permitting no material to flow through. The logarithmic law is used for the wall function close to the ground surface, and for the purpose of this simulation, the ground roughness was ignored.

The temperature of the surrounding atmosphere was set to 25°C, and the fuel content in the atmosphere was naturally set to zero. These parameters applied to the air inlet BC and all outlet BCs as shown in Figure 3. The conditions at the exit of the release source were assumed to be pure methane at 25°C, and sensitivity analysis showed that temperature has insignificant impact on the simulation results as the jet temperature rises quickly due to combustion.

The CFD analysis presented in this paper was performed using a standard numerical approach that was well described in a previous work performed by the authors (Aloqaily 2010). As described in this previous work, the approach was validated against experimental tests performed by BakerRisk, and the results were used to verify the simulation output.

## CFD AND MODELING RESULTS

It is important to mention that while the analysis that is usually performed for non-impinging flames (usually performed using screening models in Process Safety Applications) provides information about thermal hazard of horizontal flames, it still does not take into account the interaction between the flame and the affected structures/buildings. In this paper, three different cases have been modeled with the building located at distances chosen to produce high, medium and low thermal impact on the target building. These analyses will show the effect of the flame on the target building, thus providing some of the details not covered in the typical non-impinging flame analyses currently performed in process safety applications.

CFD results of jet flame simulations can be extracted in several formats. Of interest to this work is the temperature, thermal, and fuel distribution profiles. These profiles characterize the flame and describe its thermal hazard and impact. The results are presented in this section.

All cases were simulated for a 2 inch release size from a methane source at a pressure of 1000 psig. The

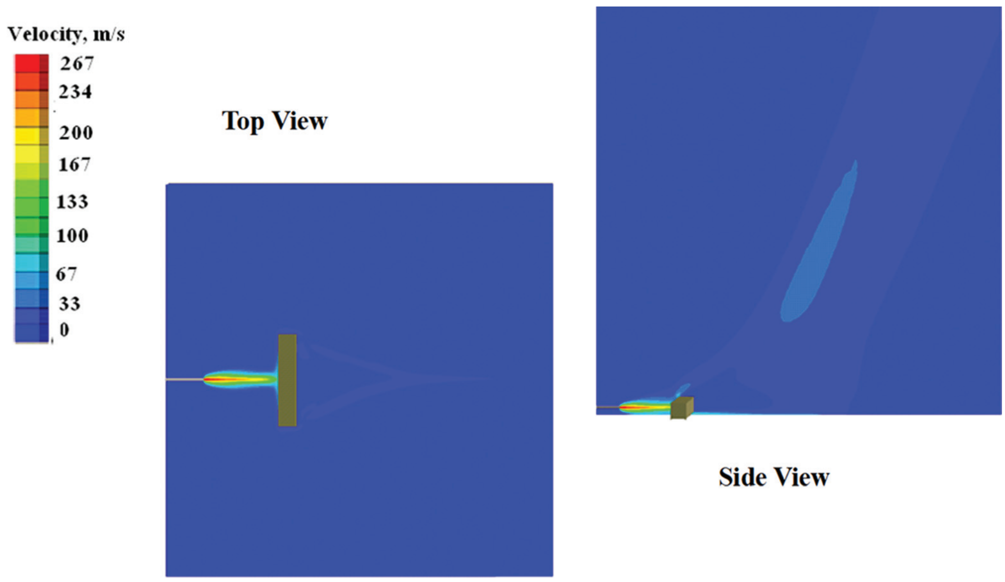


**Figure 5.** Velocity Distribution for Flame Impinging on Target Building Located 10 m Downstream of Fire Source

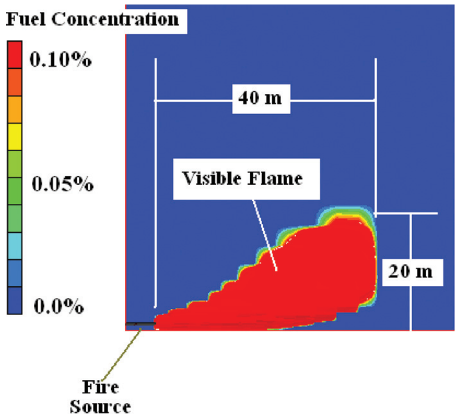


**Figure 6.** Visible Flame for Fire

release flow rate was estimated to be 23.7 kg/s. The target building was located 10 m, 65 m and 95 m downstream of the fire source. These distances were chosen through an iterative process to represent the required thermal thresholds intended in this study (high, medium and low thermal loads).

Figure 5 shows velocity profiles for the first case (building located 10 m downstream of fire source). Both side view and top view (at 1 m above the ground) show that the target building is impacted with high momentum from the jet fire because it is too close to the source of jet fire. The jet fire did not have enough time and space to disperse and lose momentum before it impacts the building. However, momentum impact is not the main concern for this case. Figure 6 shows a distribution of the unburned fuel (in terms of mass fraction), which represents the flame envelope. The figure clearly shows that the building located at 10 m from the source is engulfed in flame.

The building is predicted to be exposed to high temperatures from the jet fire, as shown in Figure 7 (top view and side view of flame temperature around the target building). The temperature at the surface of the building is expected to exceed 1000 k, as shown in Figure 8, which represents the predicted temperatures at the surface of the target building. Thermal radiation levels at the building are predicted to exceed 37.5 $kW/m^2$, an intensity high enough to cause serious damage to process equipment (AIChE CCPS 2000).

It is important to mention that the flame impingement case, discussed above, is not a typical case in Process Safety (PS) applications. Buildings are usually not located that close to sources of hazard, and therefore this case should not be of concern from a PS point of view. The case was simulated in this paper for comparison purposes only, and to represent an extreme reference case for high thermal loads. Results should not, therefore, be alarming to PS management/personnel.

For the second flame impinging case modeled in this paper, the target building was located 65 m downstream of the fire source to produce medium thermal impact on the building. The building was found to be outside of the visible flame zone at this distance, as shown in Figure 9, which presents the flame temperature distribution for this case. The temperature at the surfaces of the target building is predicted to be between 400 and 460 K, as shown in Figure 10.

It has been shown, in a previous work by the authors, that in order to accurately describe the horizontal flame, both horizontal length and vertical height of the visual flame (average height to flame tip) need to be estimated (Aloqaily 2010). The horizontal flame length (referred to

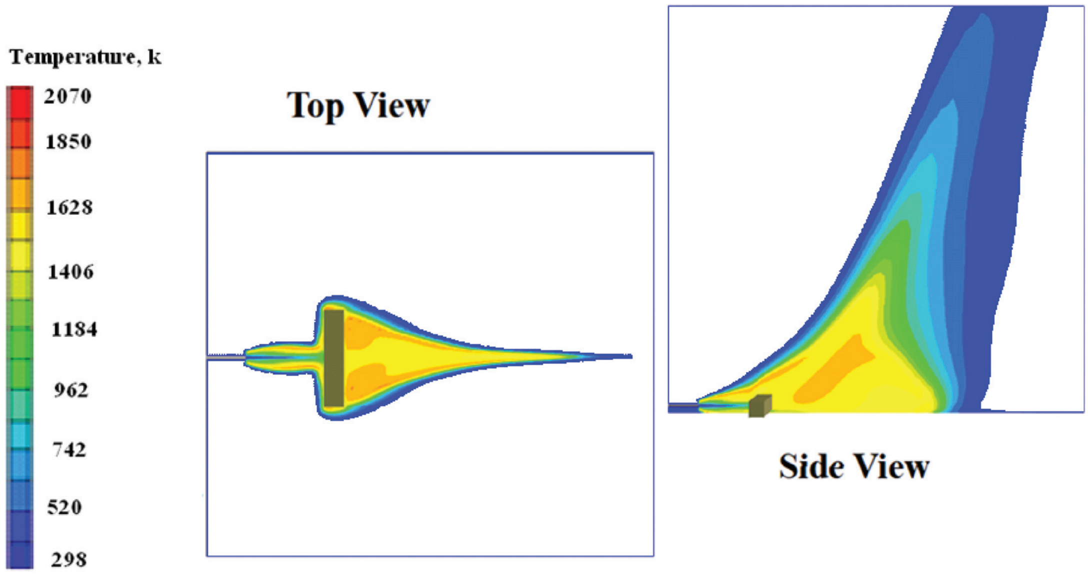


**Figure 7.** Temperature Distribution for Flame Impinging on Target Building Located 10 m Downstream of Fire Source

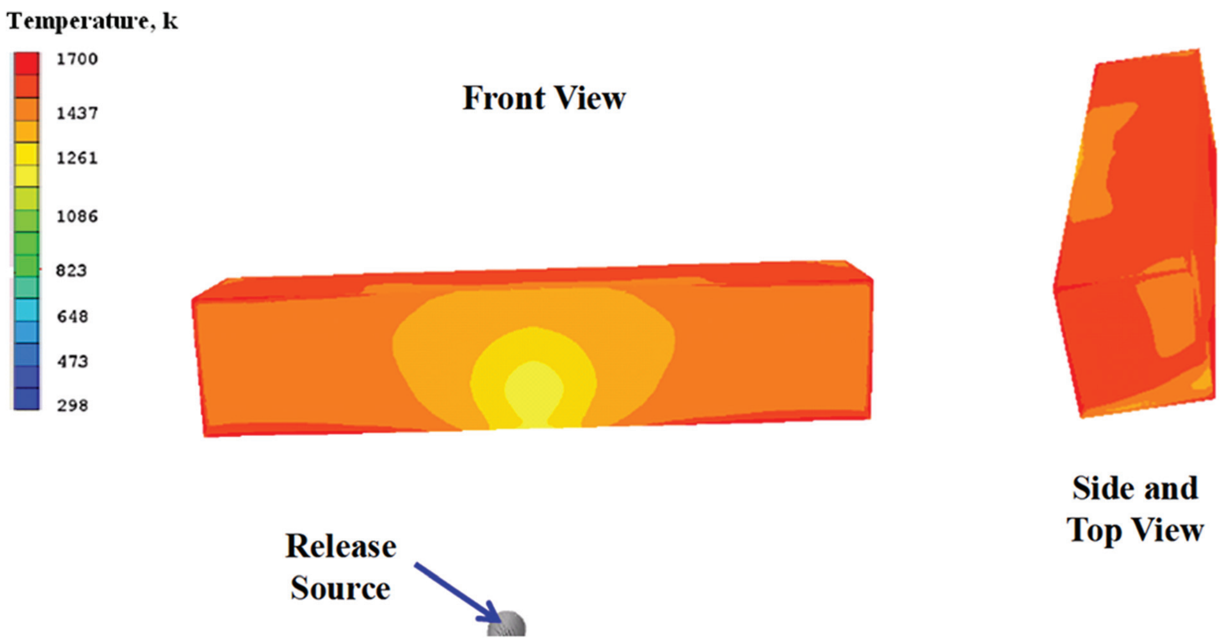


**Figure 8.** Temperature Distribution at Building Surface for Flame Impinging on Target Building Located 10 m Downstream of Fire Source

herein as flame length) is defined as the horizontal distance from the fire source to the tip of the flame. The vertical flame height (referred to herein as the flame height) is the vertical distance from the fire source to the tip of the flame. Figure 6 shows that the flame length is around 40 m and the flame height is around 20 m. As long as the building is located outside the 40 m zone, it would be expected to be outside of the flame and no flame/jet fire impingement is expected.

Inspecting Figure 11, we observe that the target building at 65 m is located within the range of 12.5 kW/m$^2$, which is characterized as a medium thermal impact for the purposes of this paper. This thermal radiation level is sufficient for piloted ignition of wood and plastic tubes melting (AIChE CCPS 2000). The results of this case show that moving the building further from the source of hazard can reduce the thermal impact significantly compared to the first case discussed above.

It should also be noted that the temperature values presented in Figure 10 are for the outer surface of the building. If the building is well-insulated, the temperature inside is expected to be lower and will rise slowly with time, providing protection to building occupants. The efficiency of the building in providing such protection will be the subject of a future investigation, as it will require dynamic modeling, which is beyond the scope of this paper.

The third impinging case was simulated for a target building located 95 m downstream of fire source. This distance was chosen to result in low thermal impact on the building. As the flame temperature distribution was similar to that of Figure 9 it is not shown here.

The building is predicted to be impacted with temperatures in the range of 340–400 k. Thermal radiation contours at 2 m above ground are shown in Figure 11. The building is predicted to receive an average thermal load of 4.0 kW/m$^2$. This is a low thermal load sufficient only to cause pain to personnel if they are unable to reach cover within 20 seconds. An insulated building should be capable of providing protection to personnel occupying it, thus avoiding the impact from thermal radiation. The efficiency of such protection will be the subject of a future investigation, as mentioned above.

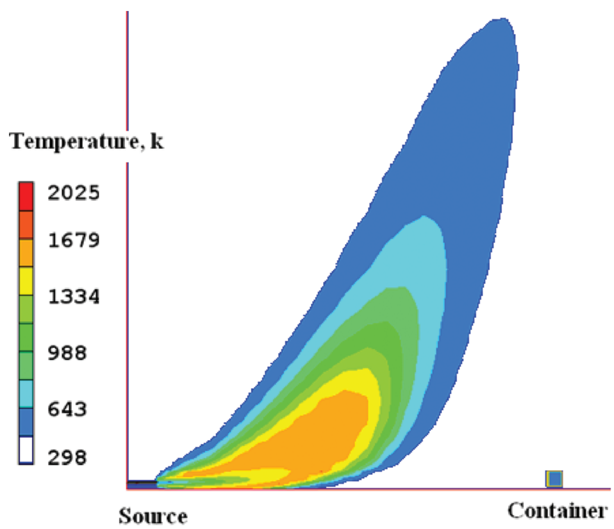


**Figure 9.** Temperature Distribution for Flame Impacting Target Building Located 65m Downstream of Fire Source

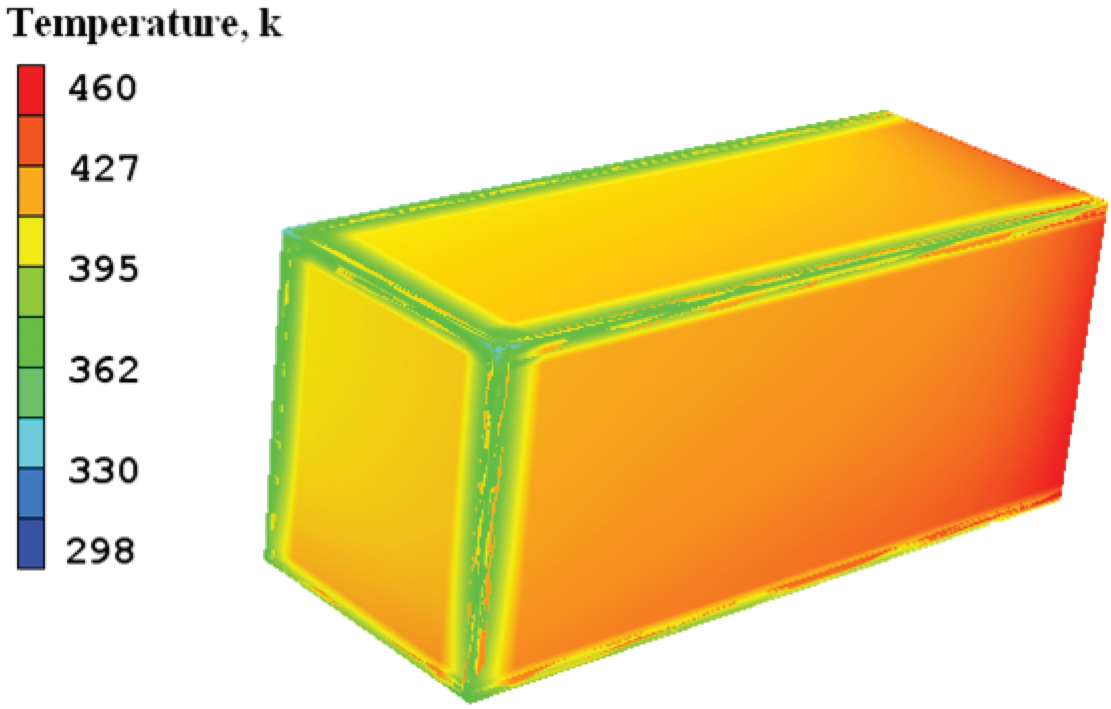


**Figure 10.** Temperature Distribution at Building Surface for Flame Impacting Target Building Located 65 m Downstream of Fire Source

The impinging flame simulation results presented in this paper clearly show that there is a need to locate buildings (especially portable and modular buildings) away from thermal hazards. The cases presented in this paper are used to illustrate this, and should not be taken as absolute values, because they are limited to the situation discussed here and are subject to the assumptions used in this paper. Such assumptions and limitations may not be applicable in all cases, and it is important to analyze any case by taking its unique conditions and geometry into account. The analysis results presented here still need to be verified experimentally before they are considered final. However, the trend is still valid. The distance and size of fire and building affect thermal impact from fires on buildings. Table 1 summarizes the results and compares the thermal impact for the three cases.

## SUMMARY AND CONCLUSIONS

Horizontal jet flame resulting from accidental methane release was simulated numerically using the commercial CFD software, PHOENICS. Three cases of a methane release impinging on a target building located at different distances downstream of the fire source were simulated in this paper. The flame thermal characteristics and impact on the building were evaluated using PHOENICS.

The CFD results showed that the horizontal flame lifts up when the jet loses its momentum due to buoyancy, which results from the heat generated inside the flame. This is an important characteristic of the horizontal jet fire, which helps reduce the distance to high thermal radiation loads, and hence thermal impact on buildings located downstream of the fire source. Simplified Common modeling approaches currently used in process safety applications do not account for that. CFD analysis, as performed in this paper, can be used to assist facilities evaluating thermal hazard to specific buildings and structures while taking into account a comprehensive range of details, which cannot be done using these simplified models. CFD analysis can therefore help facilities better understand and estimate thermal hazard, and evaluate

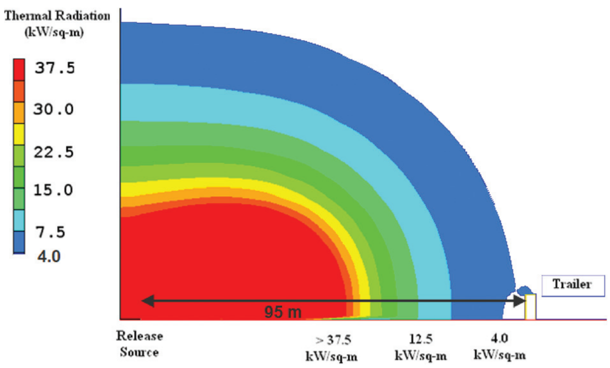


**Figure 11.** Thermal Radiation Distribution at 2 m above Ground for Flame Impacting Target Building Located 95 m Downstream of Fire Source

**Table 1.** Impinging Flame Simulation Results Summary

| Distance between Container & Fire Source | Thermal Impact Level | Average Surface Temp. K | Average Surface Radiation $kW/m^2$ | Comments |
|---|---|---|---|---|
| 10 m | High/ Severe | 1500 | >37.5 | 1. Building engulfed in flame<br>2. Process equipment damaged |
| 65 m | Medium | 450 | ≈12.5 | 1. Medium Thermal Radiation Level<br>2. Piloted ignition of wood<br>3. Plastic tubing melt |
| 95 m | Low | 360 | ≈4.0 | 1. Low Thermal Radiation Level<br>2. Pain to personnel in 20 s<br>3. Second-degree burns |

the ability to meet their commitment to operate safely and recommended practices such as API RP-752. The fire cases presented in this paper show an example of reaching that goal.

The simulation results show that the distance between the target building and the fire source significantly affects thermal impact from fires on affected buildings. Buildings (especially portable ones) should not be located too close to thermal hazard sources. Table 1 summarizes the results for the fire cases simulated in this study. The buildings can provide some protection against thermal impact to personnel staying inside. The evaluation of such protection will be the subject of future work.

The work presented in this paper also shows potentially significant benefits that can be gained by using CFD modeling techniques in PS applications that appropriately take into account the effect of flame lifting due to buoyancy, and it also characterizes thermal hazards in a more accurate and defensible manner. Traditional models incorrectly apply buoyancy, which can result in highly conservative estimates of thermal hazards (Aloqaily 2010). This may lead to incorrect conclusions that expensive mitigation measures should be implemented, whereas more accurate calculations (such as the ones performed by CFD analysis and presented in this paper) may show that these actions are not necessary.

## REFERENCES


AIChE CCPS, 2000, "Guidelines for Chemical Process Quantitative Risk Analysis", 2nd edition, New York, NY.

Aloqaily, A. M., and Chakrabarty, A., March 2010, "Jet Flame Length and Thermal Radiation: Evaluation with CFD Simulations", 2010 Global Congress on Process Safety, AIChE, San Antonio, TX.

Aloqaily, A., 2008, "A Study of Aerodynamics in Rotary Kilns with Two Burners", PhD thesis, University of Toronto.

API RP-752, 2009, "Management of Hazards Associated with Location of Process Plant Permanent Buildings", Washington, DC, Third Edition.

Launder, B.E., and Spalding, D.B., 1990, "The Numerical Computation of Turbulent Flows", Computer Methods in Applied Mechanics and Engineering, 269–289.

Launder, B.E., and Spalding, D.B., 1982, Mathematical Models of Turbulence, Academic Press, New York.

Mattick, S.J., and Frankel, S.H., 2005, "Numerical Modeling of Supersonic Combustion: Validation and Vitiation Studies Using Fluent", AIAA, 41st AIAA/ASME/SAE/ASEE joint propulsion conference & exhibit, Tucson, AR.

NFPA, 2002, "The SFPE Handbook of Fire Protection Engineering", 3rd edition, Quincy, Massachusetts.

PHOENICS Manual and lectures, CHAM, UK, 2009.

Spalding, D.B., 1979, "Combustion and Mass Transfer", Pergamon Press.

Spalding, D.B., 1971, "Mixing and Chemical Reaction in Confined Turbulent Flames", 13th International Symposium on Combustion, 649–657, The Combustion Institute.

Sugawa, O., and Sakai, K., 1996, "Propane Length and Width Produced by Ejected Propane Gas Fuel from a Pipe", Thirteenth Meeting of the UJNR Panel on Fire Research and Safety, Volume 1.

TNO, 2005, "Methods for the Calculation of Physical Effects – due to release of hazardous materials (liquids and gasses)", Yellow Book, PGS 2 (CPR 14E) – Part 2, Committee for the Prevention of Disasters, 3rd edition, 2nd revised print.

Zhou, X. Sun, Z., Durst, F., and Brenner, G., 1999, "Numerical Simulation of Turbulent Jet Flow and Combustion", Int. J. Computer and Mathematics with Applications, **38**, 179–191.