Metal Fume Control – An IH Engineering Case Study

Clyde J. Porter, PE, CIH

Air Science & Engineering, LLC   Dayville, CT

ABSTRACT

Metal fumes from a large torch cutting operation were escaping into adjacent work areas bypassing an existing side-draft hood. The process also included a high-velocity push jet which was used to prevent the buildup of gases under the work piece. The project objectives were to develop a hood design that would capture and contain the process fumes while minimizing the required exhaust flow rate.
Field testing was performed to quantify key process variables such as heat output, fume generation rate and push jet performance. Using this data, a computational fluid dynamic (CFD) model of the existing process was developed and the output for the baseline condition validated.
Alternative hood configurations likely to improve fume capture were developed and a preferred option was selected by the customer. A CFD model of the selected configuration was developed and utilized to predict fume capture at different exhaust flow rates and at different torch positions. Results led to a final hood design and the required exhaust rate. After installation, the hood performed as predicted and the process fumes are now adequately controlled.
This case study demonstrates how CFD modeling technology, combined with traditional industrial hygiene engineering, can yield effective and practical results.

PROBLEM

Fumes from a large torch cutting operation were contaminating the work area; existing side-draft exhaust hood was ineffective.

OBJECTIVES

- Develop new hood design to effectively capture process fumes.
- Minimize exhaust volume to reduce capital and operating costs for pollution control equipment.
- New hood must continue to allow parts to be loaded by overhead crane.
- New hood must accommodate existing high-velocity push jet necessary for process safety.

Process Information

Large slabs of titanium are cut using automated torch. Cutting over water is used to reduce fume generation, but a high-velocity push jet is necessary to prevent the buildup of flammable gases under the workpiece. Despite these measures significant fume levels escape into surrounding work areas.

In operation, fumes drift in direction of exhaust hood, but mostly rise up and over hood into surrounding work areas.

Process is large; 10' x 20' tank; parts up to 6' x 14'.

Existing 8,000 cfm exhaust hood mounted on end of tank.

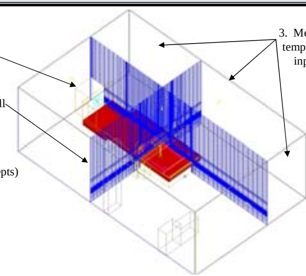
Air Science & Engineering, LLC
www.airscience-engineering.com

ANALYSIS

Approach – Traditional IH engineering, coupled with CFD (computational fluid dynamics) analysis. Since the process was unique, field testing was necessary to characterize the fume source, the high-velocity push jet and the cross drafts in the open bay, shop environment. A baseline CFD model of the existing condition was developed and validated using the field data. The model was then utilized to evaluate the likely effectiveness of possible new hood configurations and to optimize the design of the final selected configuration.

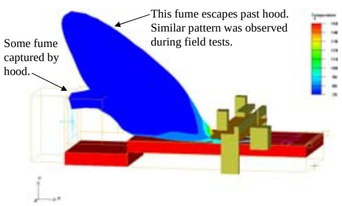
Baseline CFD Model Setup

1. Geometry of actual process and surrounding area was constructed.
2. Model space is divided up into small cells. (~800,000 cells in this case.)
3. Measured data, e.g. flow direction, speed, temperature, and heat and fume sources are input into the model at each boundary.
4. Using mass & energy balances, computer calculates flow direction, speed, temperature and fume concentration at each cell.
5. Graphic images are prepared to display the model output of interest. Results usually reflect trends or patterns better than absolute values.



Field Testing Data:
(for model input and for developing hood concepts)
- Heat generated during cutting.
- Cross drafts.
- Fume generation rate.
- Push-jet flow velocities and distribution.
- Part loading/unloading envelope.
- Operators' access requirements.

Baseline CFD Model Output
(Cloud of Constant Fume Concentration.)



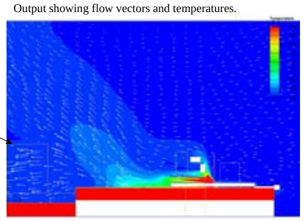
Some fume captured by hood.

This fume escapes past hood. Similar pattern was observed during field tests.

Model Conditions: Exhaust at 8,000 cfm. Existing push jet on. Cross draft at 50 fpm.

Validating the Baseline CFD Model

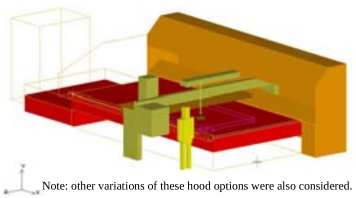
Output showing flow vectors and temperatures.



Existing exhaust hood

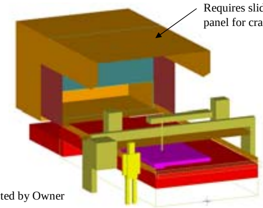
Model predicted temperatures and smoke patterns were consistent with the observed conditions. Model assumptions valid.

Design Option A – Side Draft Hood



Note: other variations of these hood options were also considered.

Design Option B – Large, Back-Draft Hood



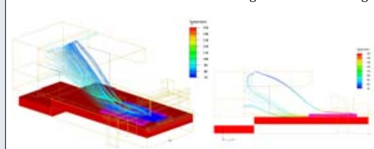
Requires slide-back top panel for crane access.

Option Selected by Owner

SOLUTION

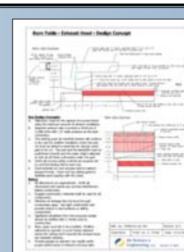
After the basic configuration of the new hood was selected, modeling and engineering work continued to determine the required exhaust rate and the design details. Various combinations of exhaust rate, cutting position and environmental conditions were modeled leading to a final design exhaust rate of 11,500 cfm. This optimization process also indicated that a second push jet would likely be necessary for worst case cutting conditions. Detailed designs for the exhaust hood and the secondary push jet were developed. The owner installed the new exhaust hood and the associated air pollution control device. New hood has been operational for nearly two years. Excellent fume capture has been reported. Owner's project manager has indicated that the hood performs "just like the model!"

Model Predicted Hood Performance during Worst Case Cutting



Pathlines from upper push jet. Pathlines from fume sources.

Model Setup: 11,500 cfm exhaust. Upper push at 450 cfm. Cross draft at 50 fpm.



New Hood Installed

Retractable roof

CONCLUSIONS

- New exhaust hood, designed using CFD modeling and traditional IH engineering, was installed and effectively captures the process fumes.
- Estimated savings of $50,000 in capital costs and $ 5,000 in annual operating costs because CFD analysis allowed hood design and exhaust rate to be optimized before construction.
- New hood performed as predicted from startup; field prototype development and associated rework costs and production delays were avoided.
- Effective and practical results can be achieved using CFD airflow modeling technology combined with traditional IH engineering.