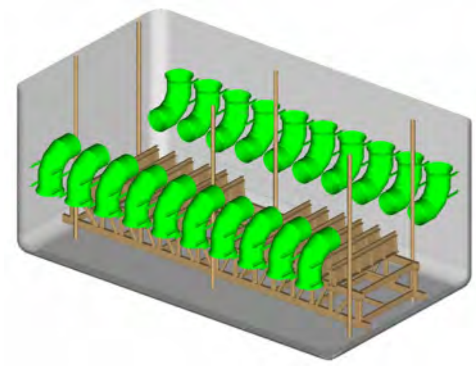
Development of Quench Tank Agitation Design Using CFD Modeling

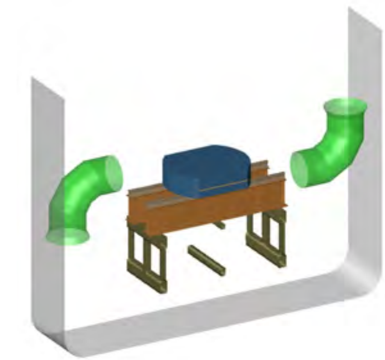
John D. Nitz, PE
Project Engineer

Airflow Sciences Corporation

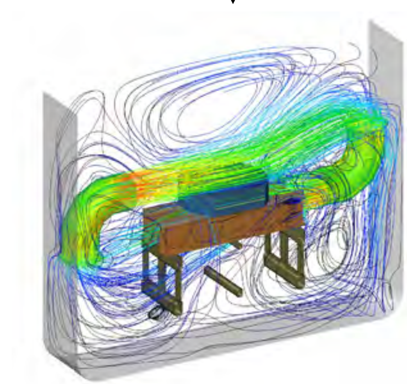
Iterative Design Process - Overview



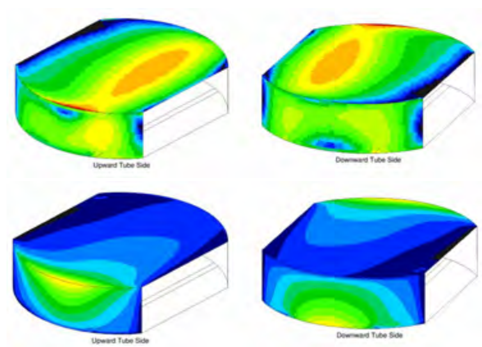
System Design
(CAD Geometry)



CFD Model
(Symmetry Section)

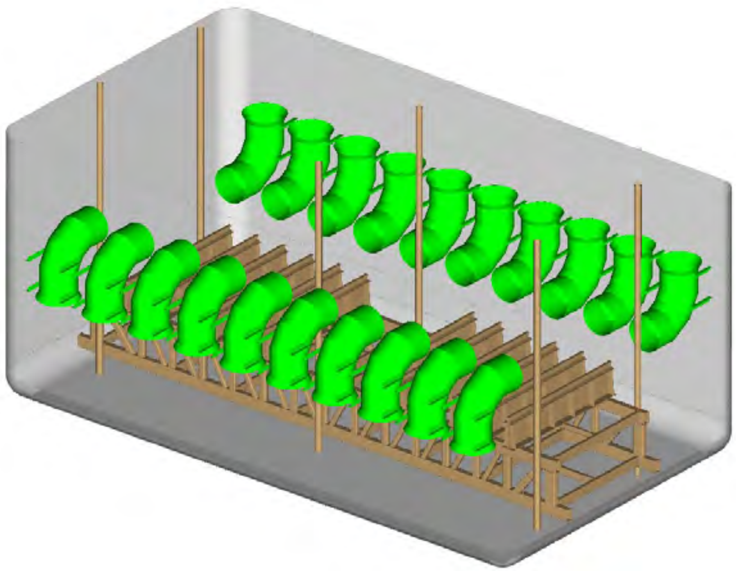


Predicted Flow Field



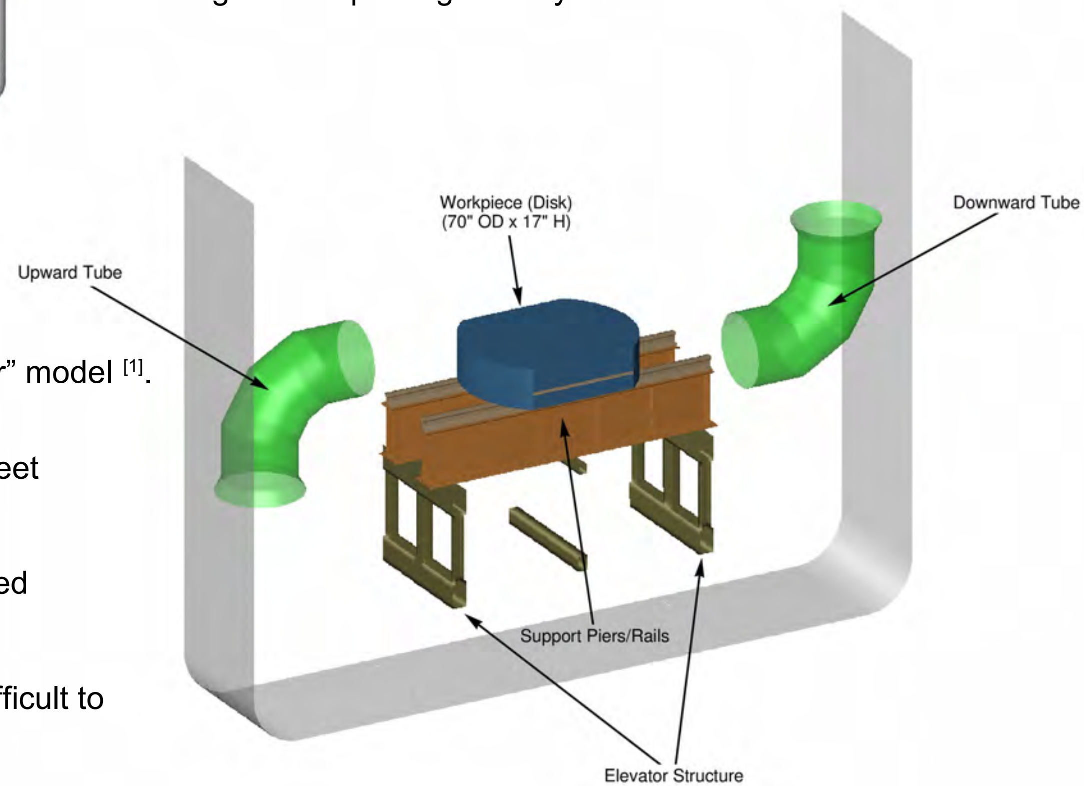
Calculated Velocity Profile & Heat Transfer Coefficients
(Surrogate Part)

Baseline Design – CFD Model Geometry



Baseline CAD Design

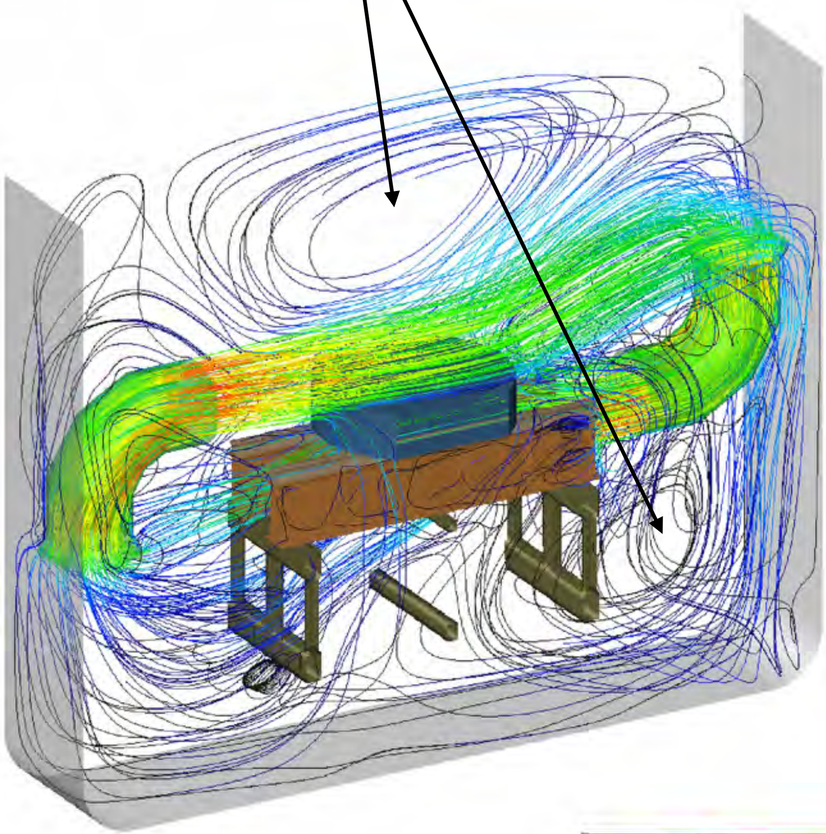
- Discrete, opposing draft tube concept (upward / downward).
- Variable vertical working envelope.
- Forced circulation across width of tank (minimal lengthwise flow).
- CFD model represents symmetry section (one tube pair).
- Surrogate workpiece geometry for evaluation.



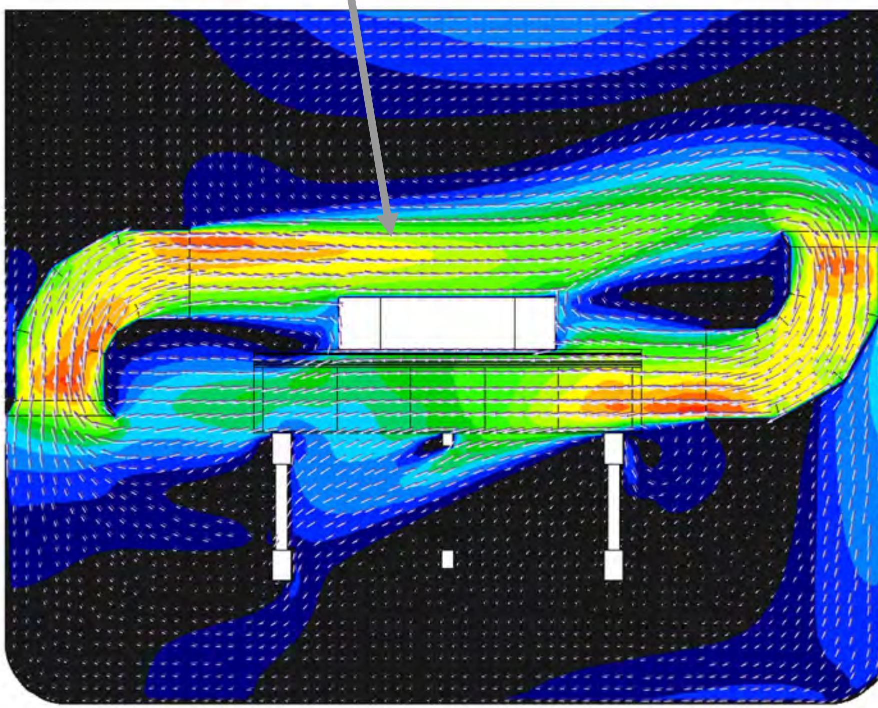
- Impellers simulated using computational “propeller” model [1].
- Fluid momentum and pressure rise specified to meet target flow rate (21,500 GPM).
- Rotational component specified based on estimated impeller efficiency.
- Actual impeller geometry is preferred, but more difficult to obtain.

Baseline Design – CFD Results

Low velocity re-circulation zones above and below workpiece

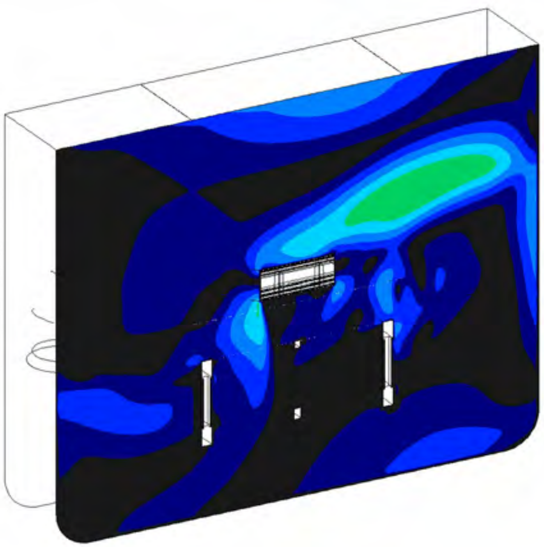


High velocity jets vertically localized in work zone.

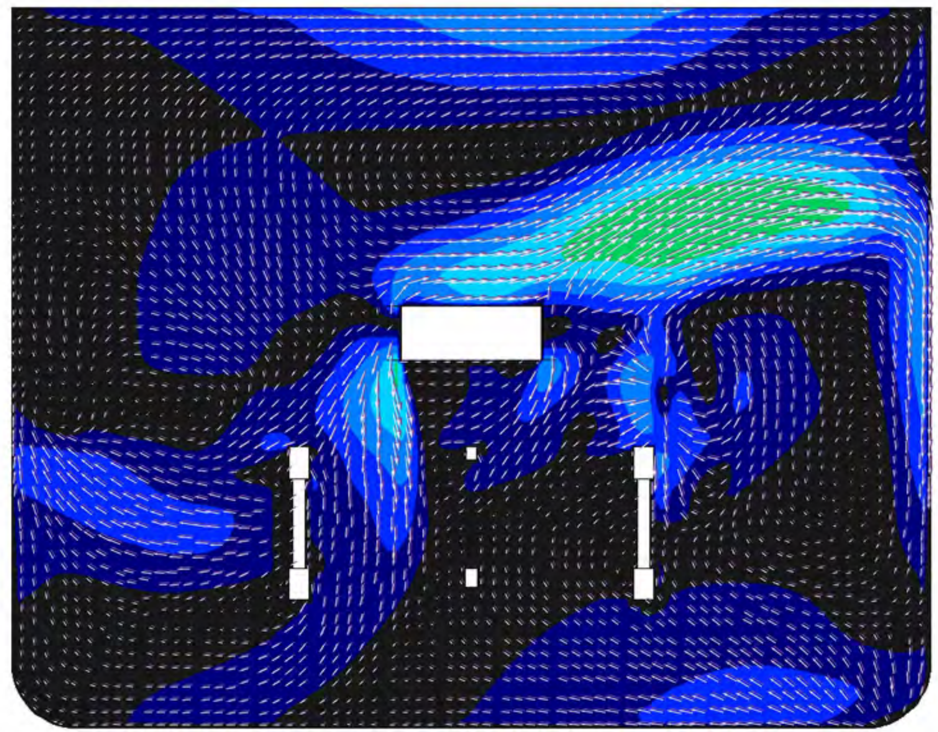


Impeller Center Line Plane

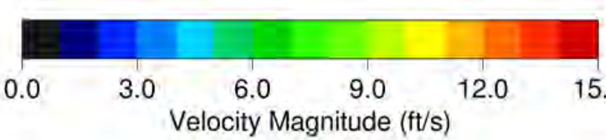
Baseline Design – CFD Results



Low velocities between impellers / draft tubes.



Plane Between Impellers

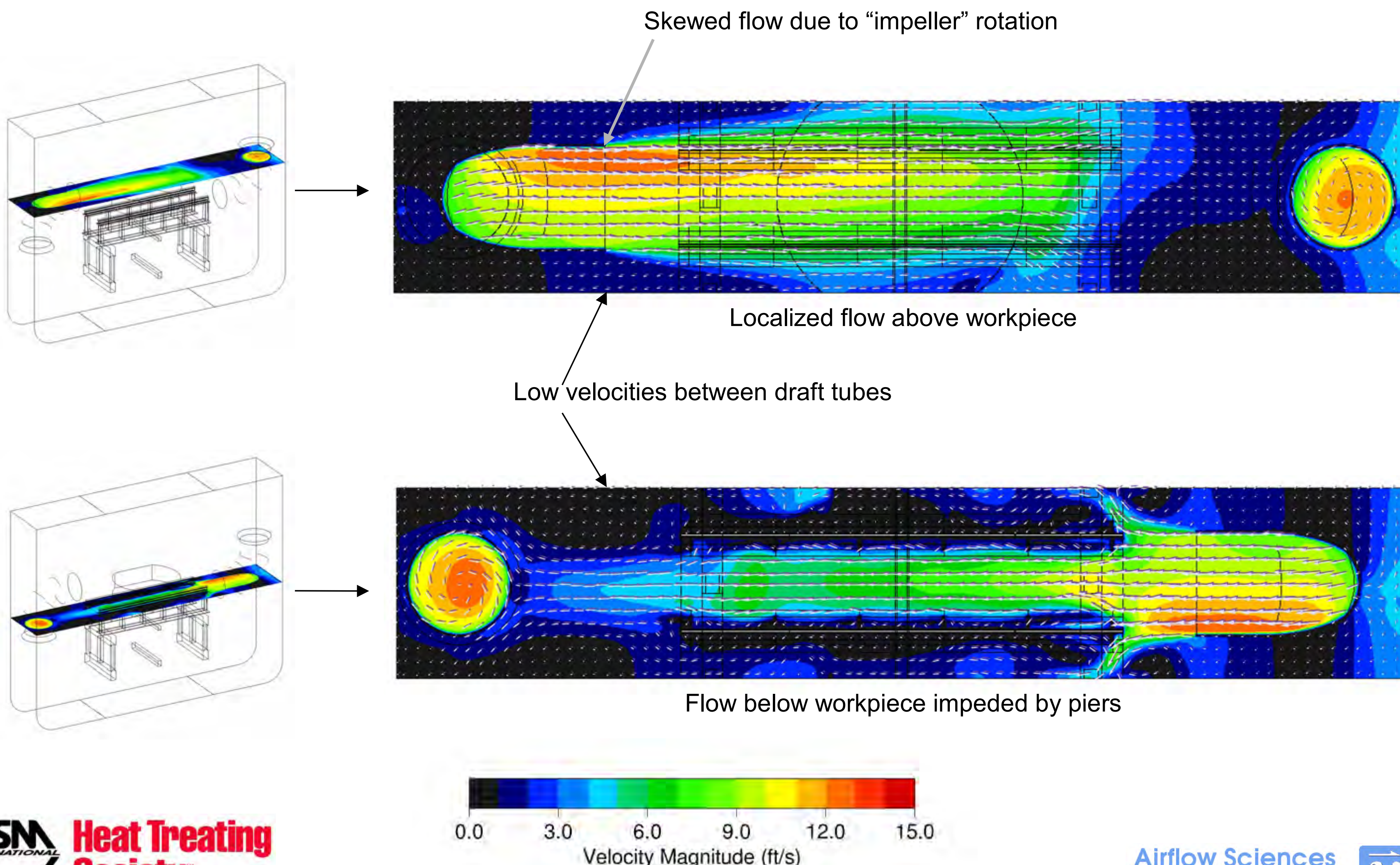


ASM INTERNATIONAL Heat Treating Society
An Affiliate Society of ASM International®

Airflow Sciences Corporation

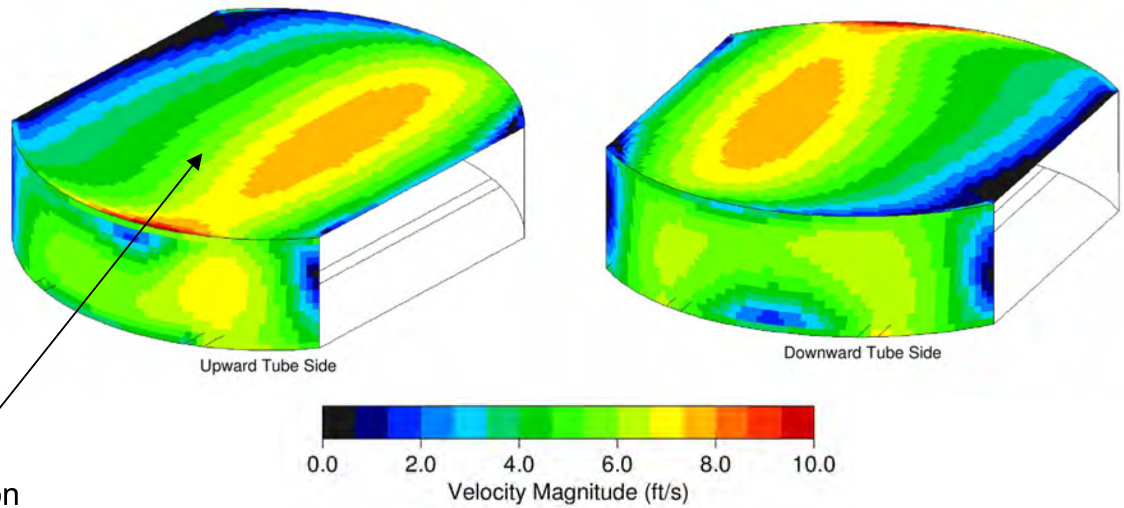
Baseline Design – CFD Results

Horizontal section cuts



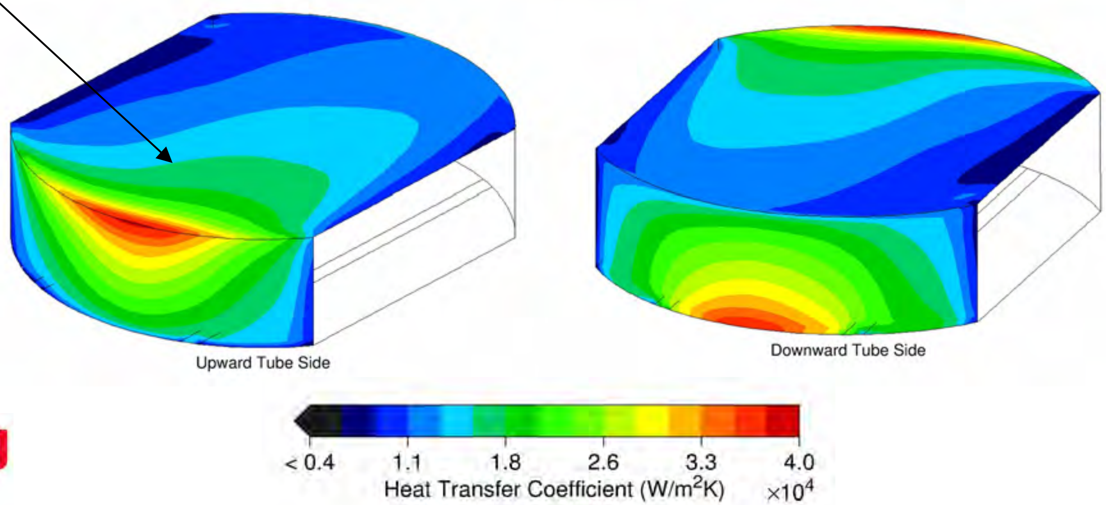
Baseline Design – CFD Results

Near-surface velocity

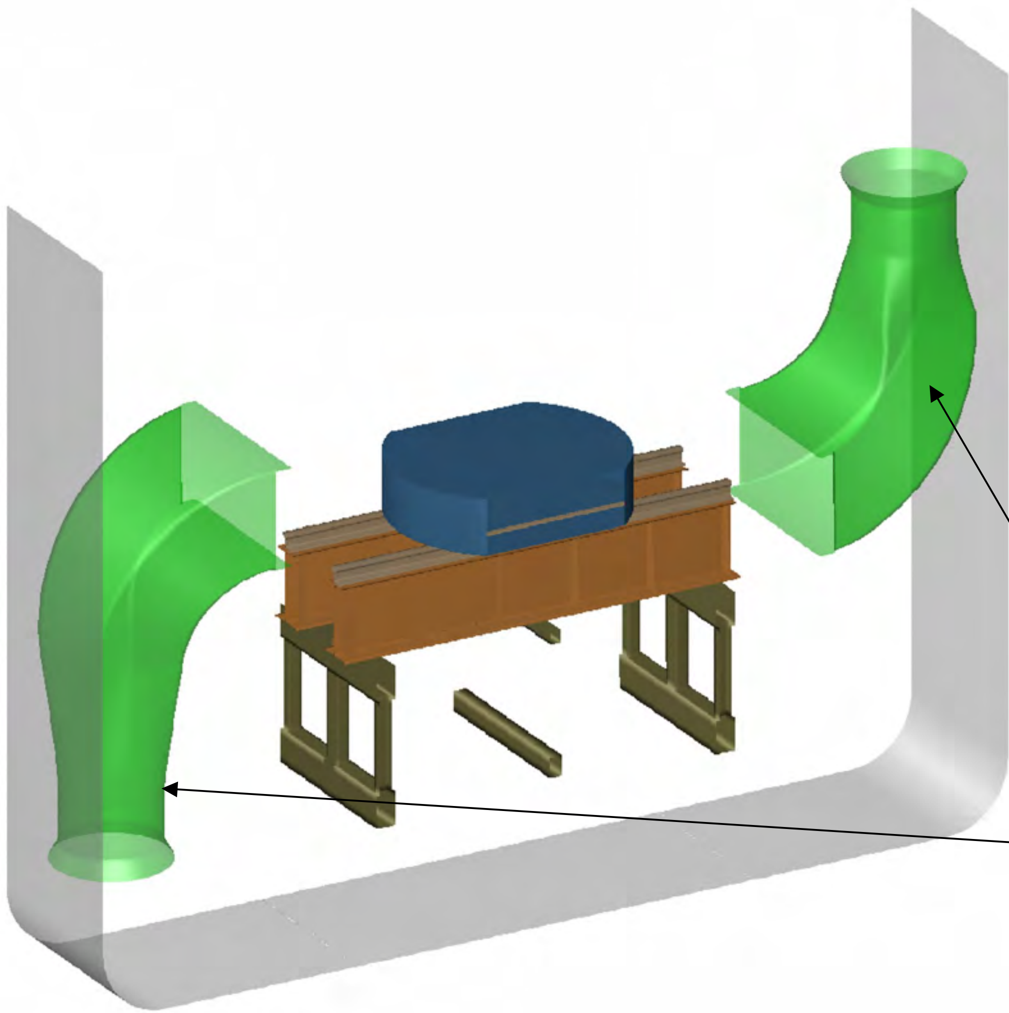


Skewed distribution
due to impeller rotation

Surface heat transfer coefficient

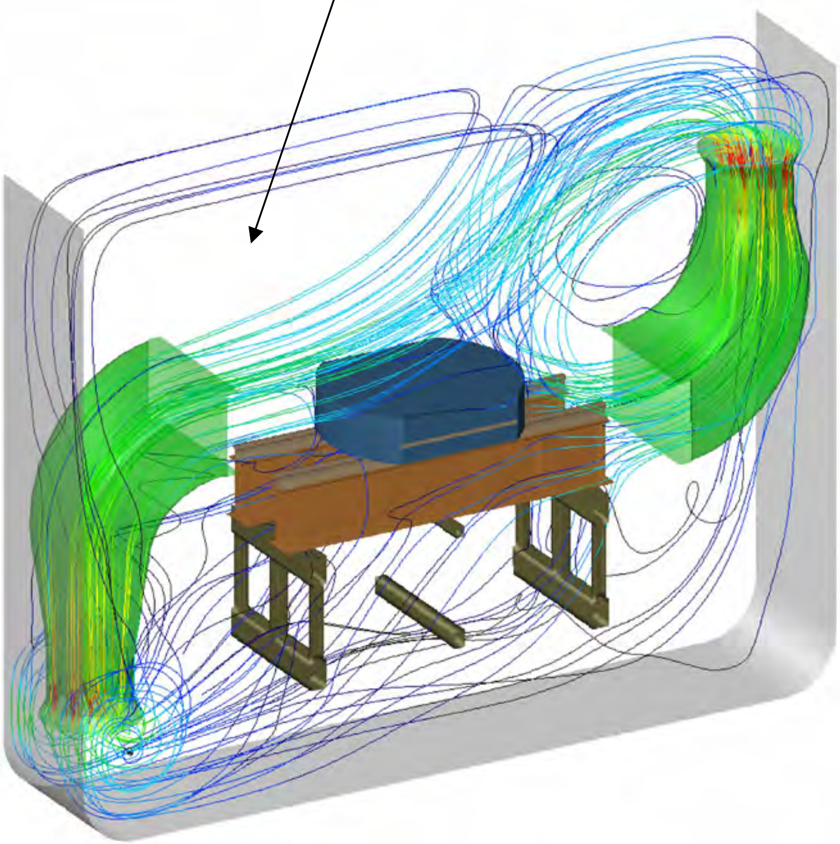


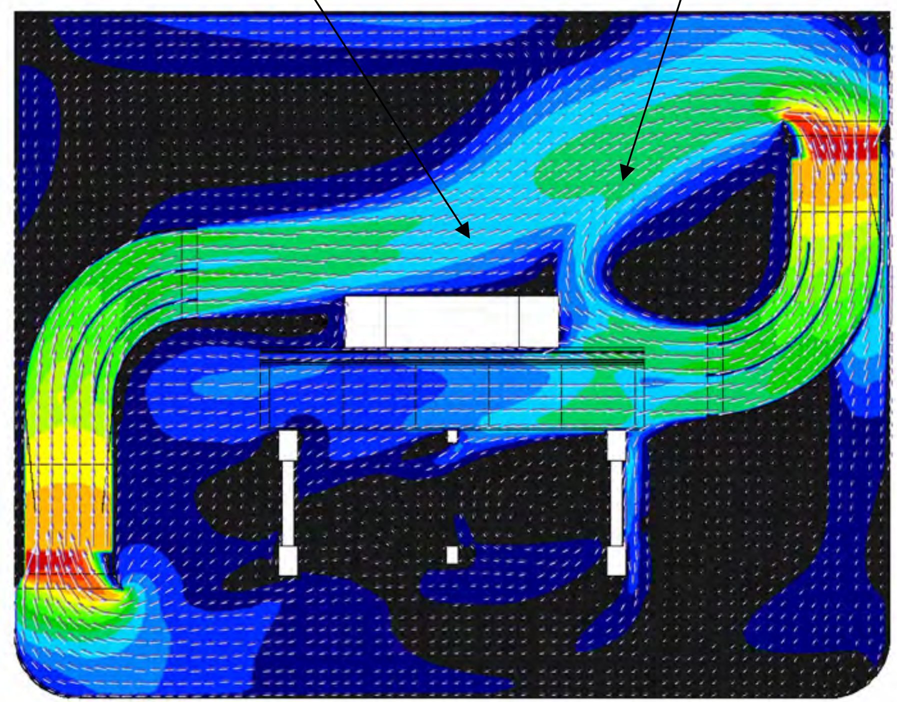
Intermediate Design – CFD Model Geometry



- Transition to rectangular outlets.
- Taller intake tubes.
- Internal turning vanes (not shown).

Intermediate Design – CFD Model Geometry





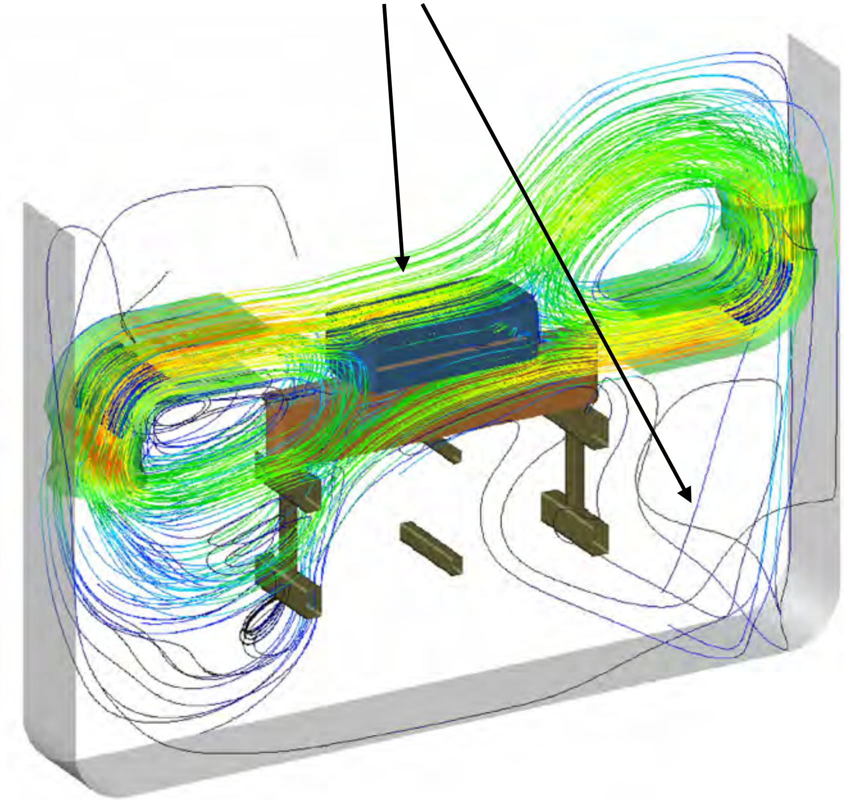
Impeller Center Line Plane

ASM INTERNATIONAL Heat Treating Society
An Affiliate Society of ASM International®

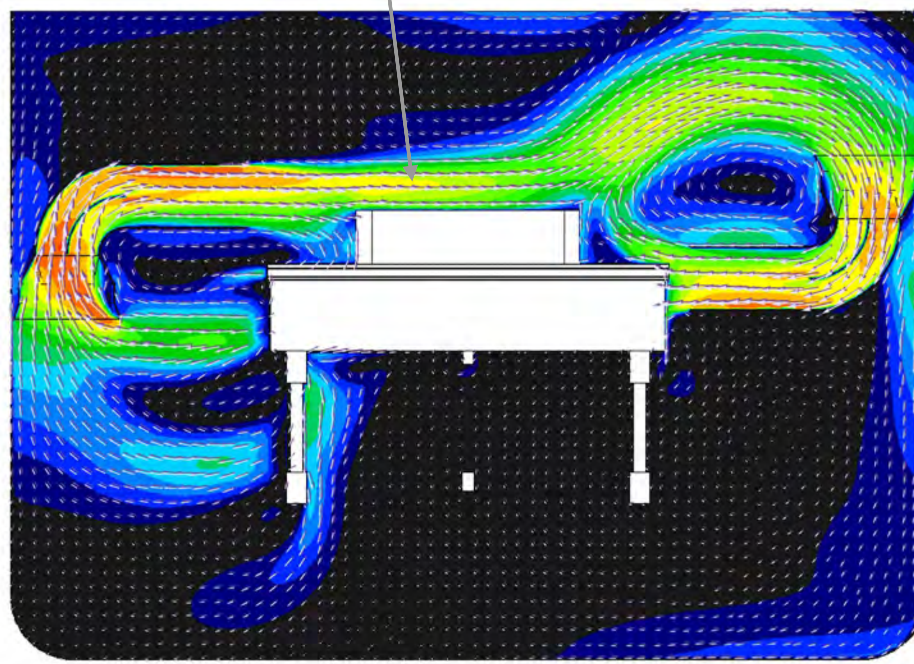
Airflow Sciences Corporation

Final Design – CFD Results

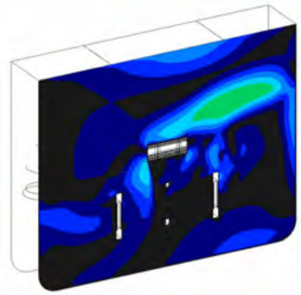
Stonger flow through work-zone.
Reduced re-circulation at top and bottom of tank.



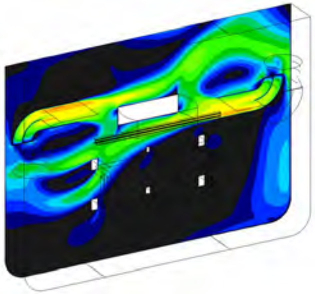
Fluid jets localized near part surface.

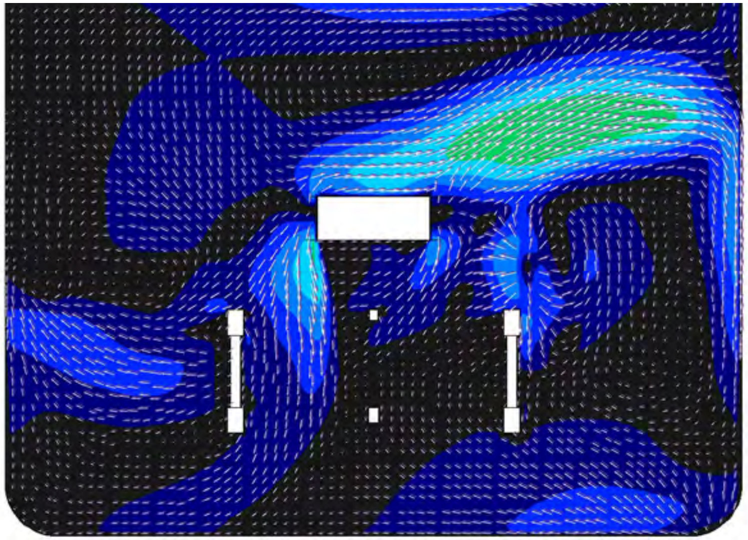


Final Design – CFD Results

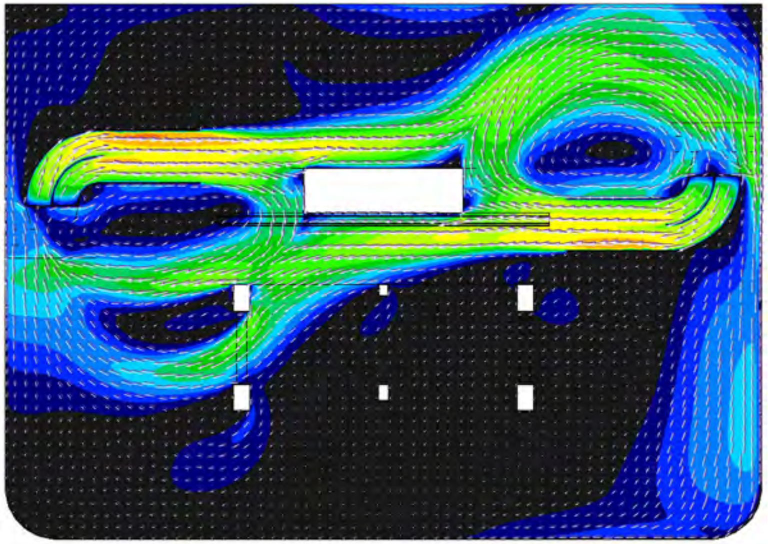


Planes Between Impellers

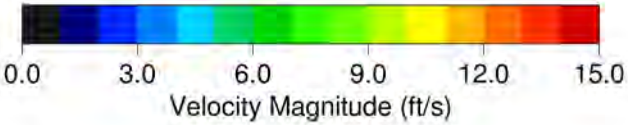




Original Design
Low velocities between impeller tubes.

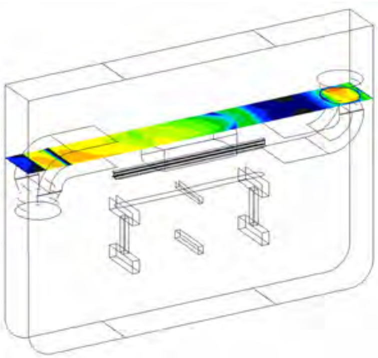


Final Design
Improved velocities between impeller tubes.

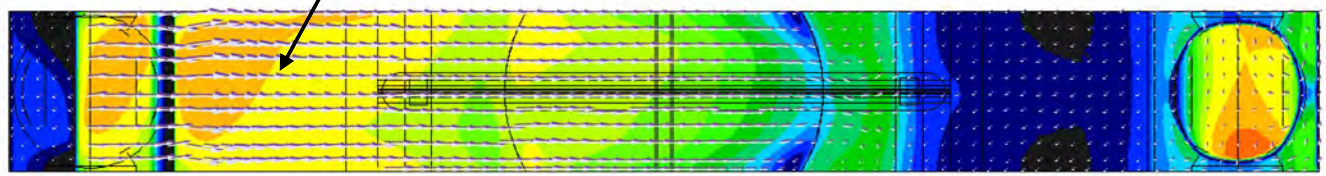


Final Design – CFD Results

Horizontal section cuts

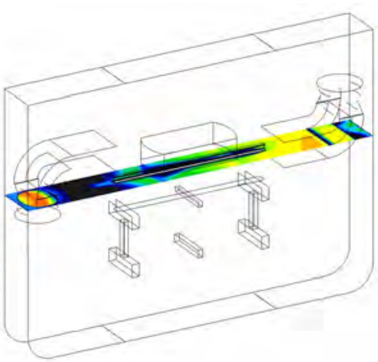


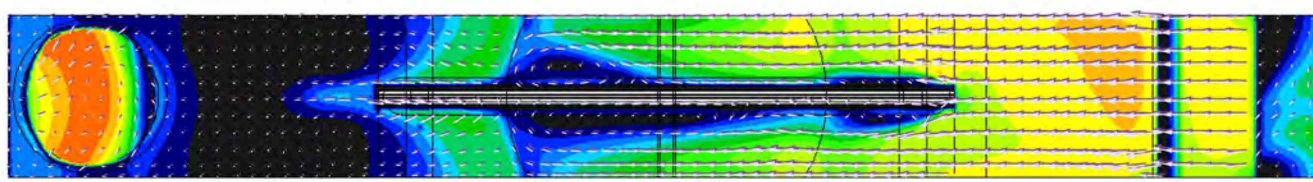
Turning vane minimizes flow skew.



Flow distribution above workpiece

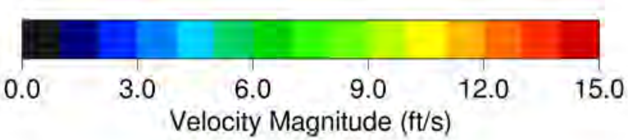
Higher velocities between impeller tubes





Improved uniformity between piers

Velocities still lower on average compared to top due to support structure

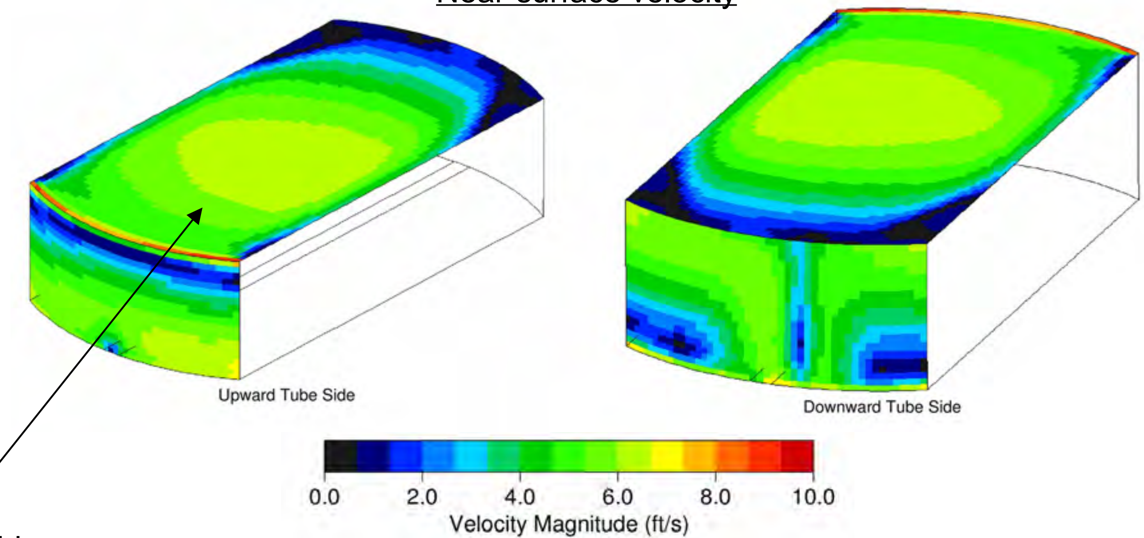


ASM INTERNATIONAL Heat Treating Society
An Affiliate Society of ASM International

Airflow Sciences Corporation

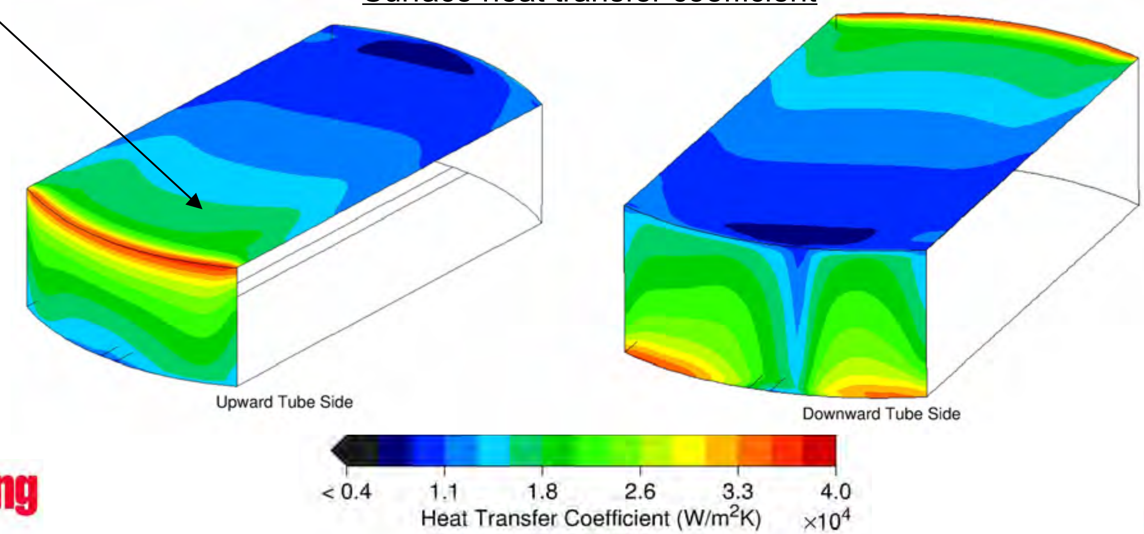
Final Design – CFD Results

Near-surface velocity

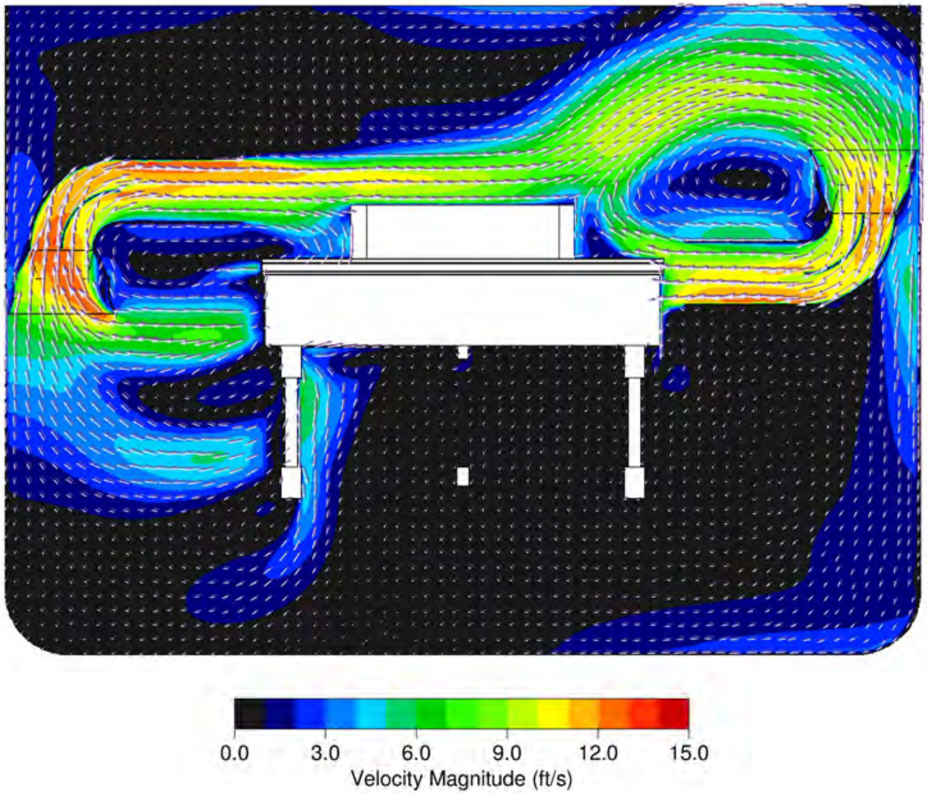


Improved side-to-side uniformity

Surface heat transfer coefficient



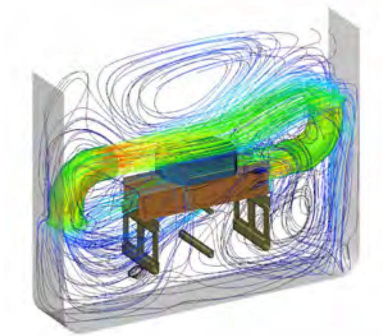
Design Validation



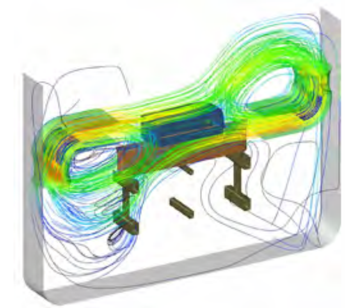
- Baffles in neutral position.
- 19.5 Ton load (taller than surrogate “disk” in CFD model).
- Measured velocities at outlet: 8 – 13 ft/s.
- Measured velocities near upper surface: 1.1 – 6.5 ft/s over range of elevator height.
- Measured velocities near lower surface: 0.6 – 2.2 ft/s over range of elevator height.
- All measured data per reference [2].

ASM INTERNATIONAL Heat Treating Society
An Affiliate Society of ASM International

Airflow Sciences Corporation



Conclusions



- Opposing draft tube concept creates strong circulation across width of tank.
- Spacing of the discrete tubes creates “striping” effect with low velocities between tubes.
- Flow rotation imposed by impeller creates skewed draft tube outlet profile.
- Intermediate design showed non-intuitive results with taller inlet tubes.
- Common rectangular duct with decreased impeller tube spacing improves velocity distribution between impeller tubes (along length of tank).
- Turning vane in elbow dampens flow rotation imposed by impeller.
- Variable baffle allows flow to be focused on work-zone for shorter or taller workpieces.
- Measured velocities compare favorably with predicted results.
- Detailed representation of impeller geometry could provide better prediction of actual performance curves.

References

[1] Ansys, Fluent 6.3 User's Guide, Sect. 7.20.

[2] Alexander, Jared C., “Trial to Determine the Suitability of the New Heat Treat Facility at Corry Forge Company for Processing Blowout Preventer Bodies”, Corry Forge Company, Corry, PA, 2013.