CAS

CONTRACT ANALYTICAL SERVICES

CONFIDENTIAL REPORT

HEAT HERO SYSTEM FLUID FLOW AND THERMAL ANALYSIS

CISD

CENTRE FOR INDUSTRIAL SOLUTIONS AND DESIGN

Client:AMCA Ltd
PO No: IV 2015 1203

Report Date: 22nd February 2016
Contact: Art McArdle

***Introduction*:** A number of CFD and thermal simulations have been carried on various SFI heating systems. This was to ascertain what performance benefits could be otained from installing the Heat Hero manifold. Studies were carried out using both ANSYS/Fluent and Solidworks Flow Simulation to achieve the relevant results.

The primary theoretical test results indicated:

- Heat Hero on an open vented heating system improved ciculation in the system by 48%
- Heat Hero would not restrict the circulation of water as it passes through the system which demonstrates that it is safe to install.
- Water passing through a heat source at higher velocity results in an improved heat transfer rate

---

CFD Analysis 1 Solidworks Flow Simulation – Circulation Velocities

Initlally for benchmarking reasons a basic heating system was developed. This would allow a comparison between various heating systems with Heat Hero installed and without. The basic systems would include for reference:

- Stove
- Coiled tank
- Radiator
- Piping
- Heat Hero Manifold

As the system is focusing on fluid flow only and not heat transfer the stove and volume of the cylinder are of no significance and are for reference only. The piping howerver is studied fully with pump on and off.

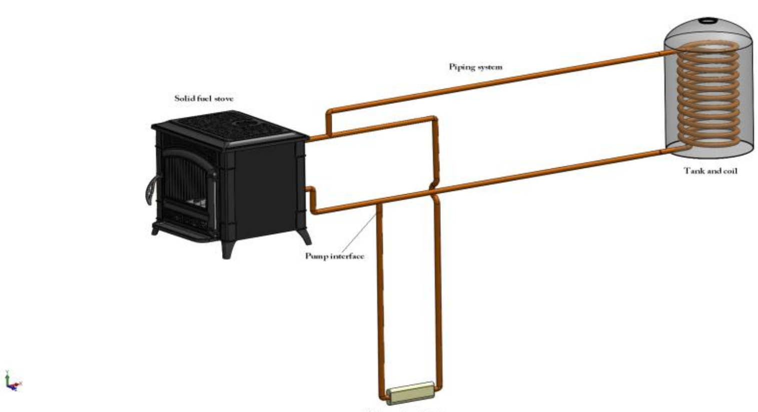


Fig. 1 Basic No Manifold Solid Fuel Heating System

Three primary system configurations were developed fully for comparison:

1. Pump on Return System
2. Pump on Flow System
3. SFI Injector System

The goal of these tests were to compare average velocities of water fluid flow into the radiators. The velocities will vary depending on the degree of balancing between the cylinder flow and radiator flow. This study assumes all balancing valves are open to the same degree in all three systems. This eliminates any false readings which may yield inaccurate velocities measurements.

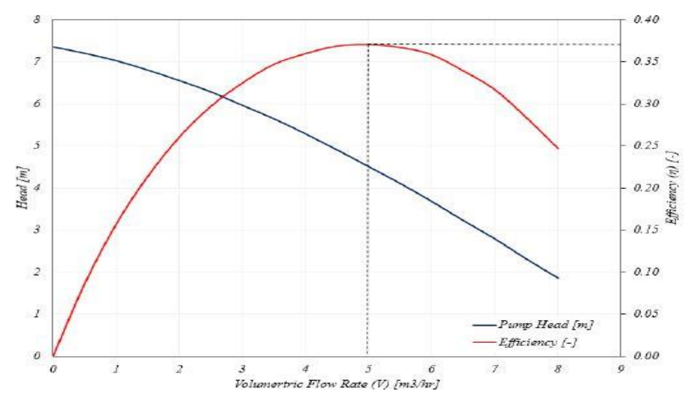
Boundary Conditions

Under normal operation conditions a Grundfos UPS 25 80 pump has a maximum efficiency of approximately 37% when the volumetric flow rate is 5m3/hr with a gauge pressure of 44145Pa. Therefrore the driving pressure available is 101325Pa plus the gauge pressure; hence 145.47Kpa. Since the pump in this system will typically be on the radiator return the only boundary conditions are:

- A driving pressure entering the system of 145.47Kpa
- A return pressure of 101.325Kpa
- Surface roughness of copper pipes is approximately 1.5µm.

1.3.1 Grundfos UPS 25 80 (Numerical Study of Pump Characteristics):

Pump Characteristics					
V [$m^3.h^{-1}$]	Pump Head [m]	Power in [W]	Pressure [Pa]	Power out [W]	η [-]
0	7.35	107	72103.5	0.00	0.00
0.5	7.2	114	70632	9.81	0.09
1	7.025	122	68915.25	19.14	0.16
1.5	6.8	130	66708	27.80	0.21
2	6.55	137	64255.5	35.70	0.26
2.5	6.28	144	61606.8	42.78	0.30
3	5.96	150	58467.6	48.72	0.32
3.5	5.64	155	55328.4	53.79	0.35
4	5.28	160	51796.8	57.55	0.36
4.5	4.9	163	48069	60.09	0.37
5	4.5	165.5	44145	61.31	0.37
5.5	4.1	167.5	40221	61.45	0.37
6	3.67	167.5	36002.7	60.00	0.36
6.5	3.215	168	31539.15	56.95	0.34
7	2.78	167.5	27271.8	53.03	0.32
7.5	2.3	166	22563	47.01	0.28
8	1.85	163.5	18148.5	40.33	0.25



4 | Page

Fig. 2 Pump Characteristics

1.2.1 System Evaluation (Reynolds's Transport Theorem):

$$\sum \dot{Q} - \sum \dot{W} = \frac{\partial}{\partial t}\iiint_{Vol} (\rho e) dV + \oiint_{Sur} (e + Pv)\rho(\vec{V} \cdot \vec{dA})$$

$$e = u + \frac{V^2}{2} + gz \qquad\qquad Note, h = u + Pv$$

$$\sum \dot{Q} - \sum \dot{W} = \frac{dE_{Cv}}{dt} + \dot{m}(h + \frac{V^2}{2} + gz)_e - \dot{m}(h + \frac{V^2}{2} + gz)_i$$

Assumptions:

- No heat transfer is accounted for, hence $(\sum \dot{Q})$ is negligible.
- The system is in steady flow, hence $(\frac{dE_{Cv}}{dt})$ is negligible.
- No enthalpy changes are accounted for, hence (h) is negligible.
- Mass is conserved, hence the mass flow rates $(\dot{m})$ in and out of the system are constant.

$$\dot{W} + \dot{m}(\frac{V^2}{2} + gz)_i = \dot{m}(\frac{V^2}{2} + gz)_e$$

Where;

$\dot{W}$ is the power required by the pump

$\dot{m}$ is the mass flow rate of the water in the circuit

V is the velocity of the water in the pipe

g is the gravitational acceleration

z is the vertical height of the inlet and outlet pipe relative to each other

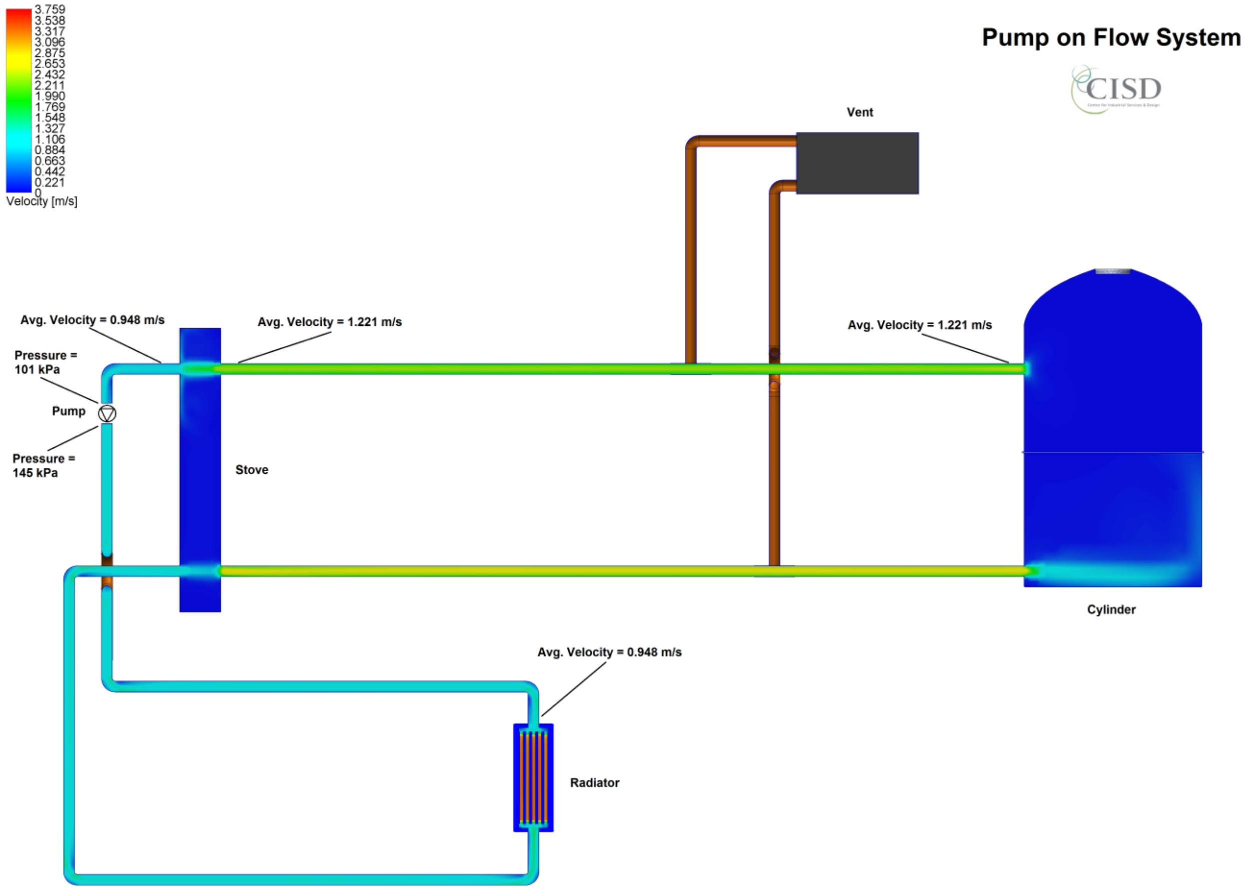


Fig. 3 Pump on Flow System

Primary Components

- Stove
- Pump
- Vent
- Radiator
- Cylinder
- Piping

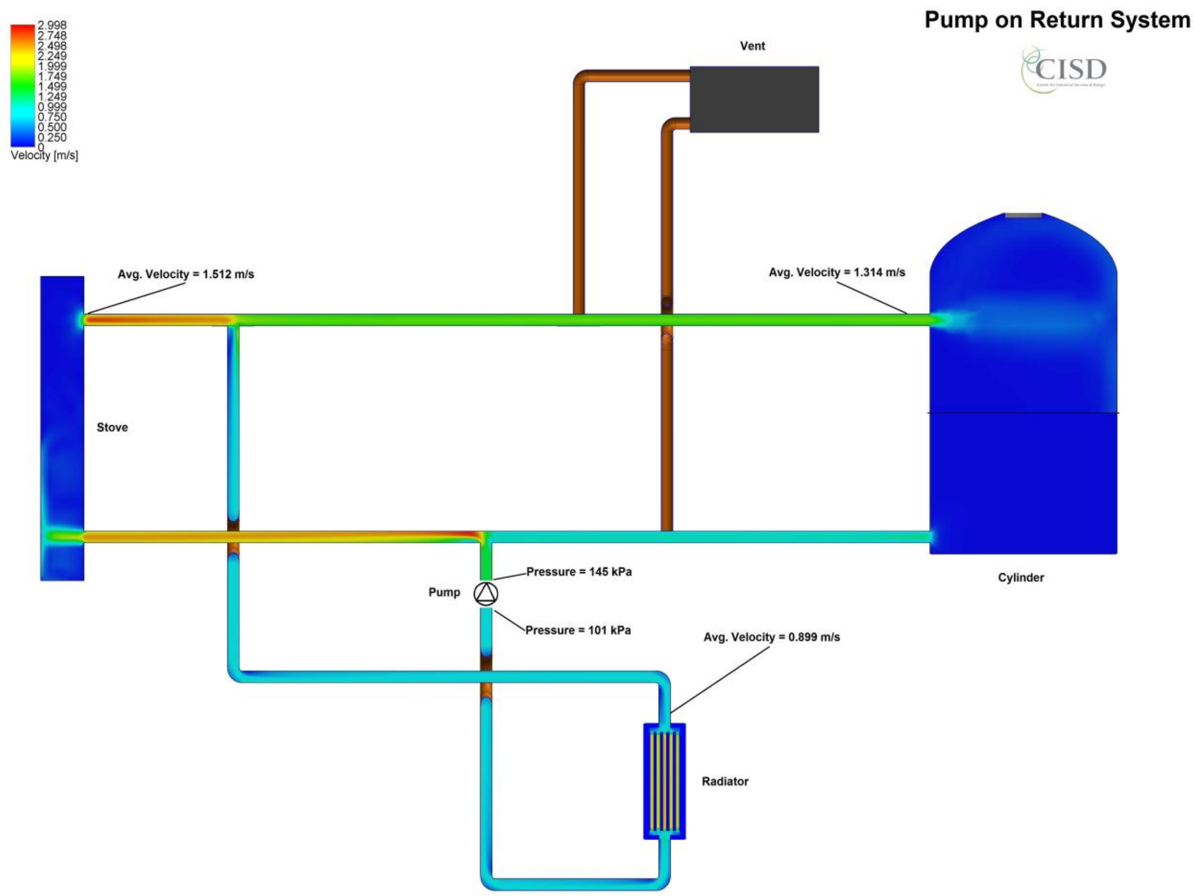


Fig. 4 Pump on Return System

Primary Components

- Stove
- Pump
- Vent
- Radiator
- Cylinder
- Piping

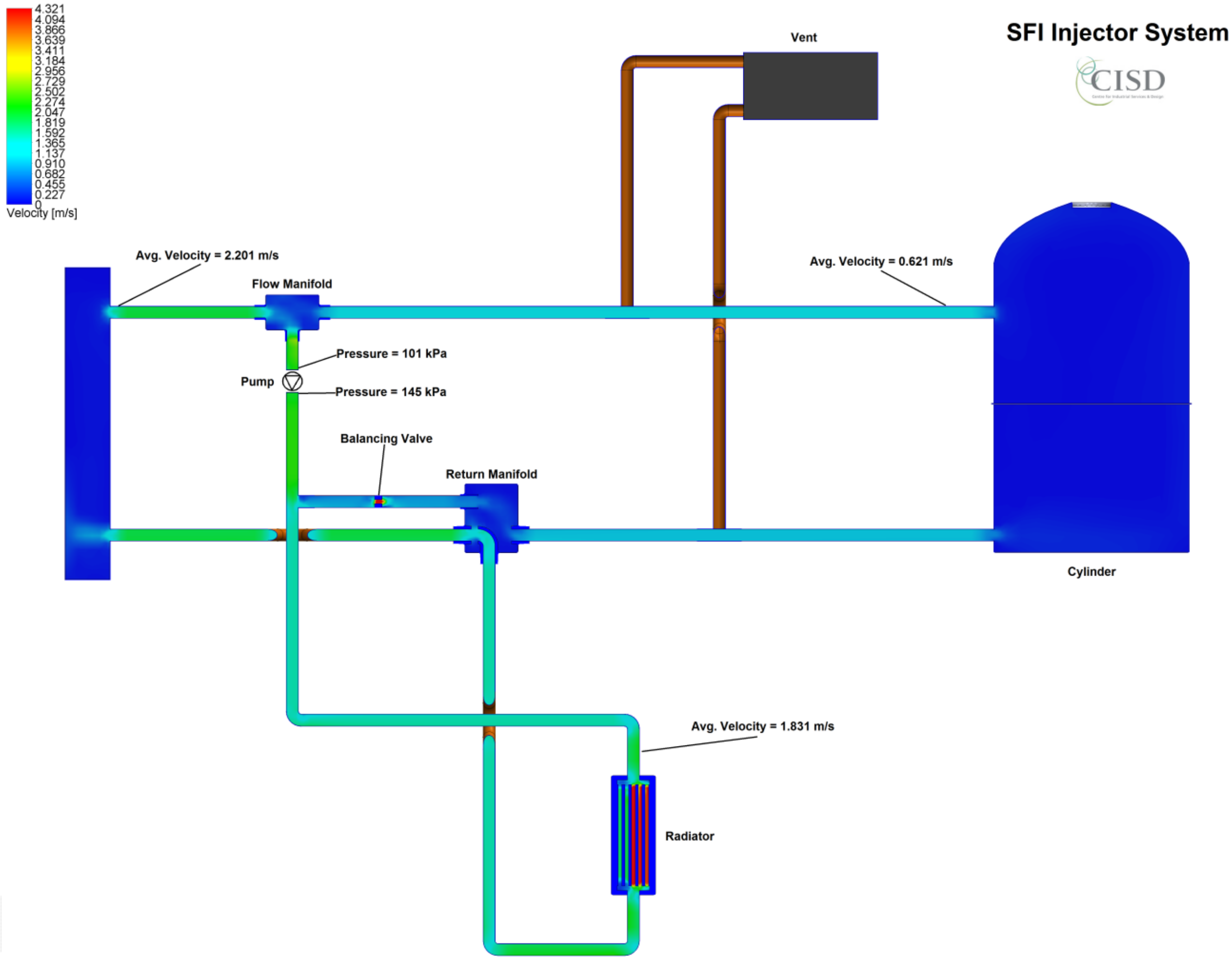


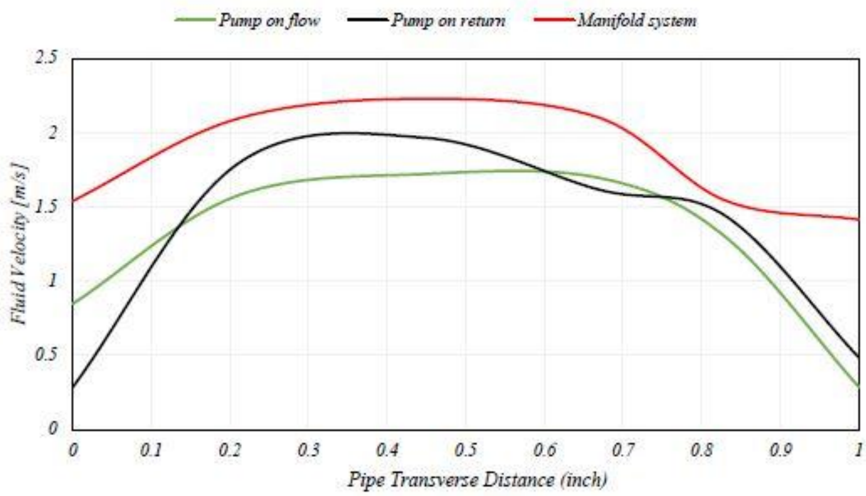
Fig. 5 Heat Hero SFI Injector System

Primary Components

- Stove
- Pump
- Vent
- Radiator
- Cylinder
- Piping
- Heat Hero Return Manifold
- Balancing Valve
- Flow Manifold

CFD Analyss 1 Results

It was seen that a higher average velocity at the radiator was found at the SFI Injector system with Heat Hero was found to be approximately 48% higher than both Pump on Return System and Pump on Flow System.



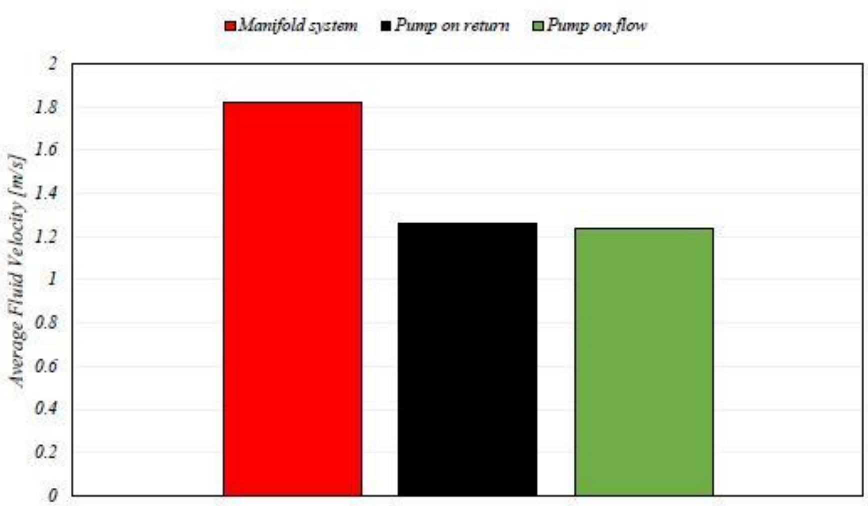


Fig. 6 Fluid Velocity Results

CFD Analysis 2 Fluent Fluid Flow Simulation – Thermal vs Velocity

A basic CFD thermal analysis system was developed in Fluent to demonstrate the effects of increased fluid flow on heat transfer from the heated surface to the fluid. The basic system was simplified to a fluid passing through a 20mm pipe at 19 C through 1.6 metres of pipe length. A heat source of 250 C was assigned at the initial cycle. Through conjugate heat transfer the water temperature would increase after each cycle passing through the heat source. There were two analyses completed at a water velocity of 1.6m/s and 3.2m/s to equate to 60 cycles per minute and 120 cycles per minute. Average water temperatures in the centre of the pipe were noted after each cycle. The increase in temperature of both cycles was noted and extrapolated to increase over a longer period of time i.e. 2 hours.

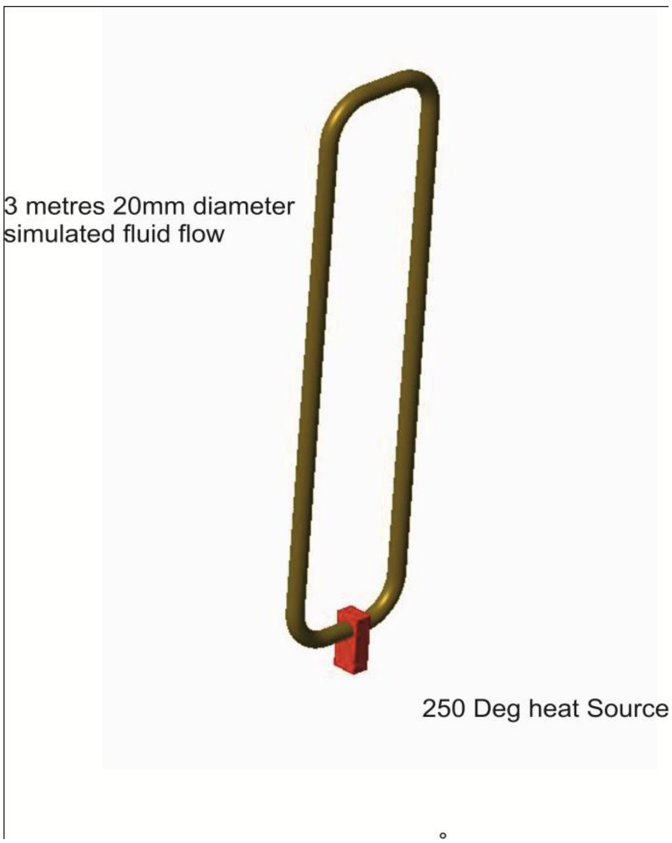


Fig. 7 Basic CFD Heat transfer Test Set up

Fluent Test Settings and Parameters:

Pipe Diameter: 20mm
Pipe Length: 1600mm
Pipe Wall Thickness: 3mm
Pipe Material: Copper
Heated Surface Area: 1256 mm3
Domain: Continuous Fluid (Water initially at 19 C)
Heat transfer Model: Isothermal
Turbulence Model: K epsilon
Flow Regime: Subsonic
Mass and Momentum: No Slip Wall
Wall Roughness: Smooth wall

Fluent Heat Transfer Calculations

For the 250 C fixed temperature condition applied at the wall the heat flux to the wall from the water was computed:

$$q = h_f(T_w - T_f) + q_{\text{rad}}$$

h_f = fluid-side local heat transfer coefficient

T_w = wall surface temperature

T_f = local fluid temperature

q_{rad} = radiative heat flux

The fluid side heat transfer coefficient was computed based on the local flow field conditions e.g. turbulence level temperatuire and velocity profiles. For turbulent flow the law-of-the-wall for temperatures derived using the analogy between heat and momentum transfer.

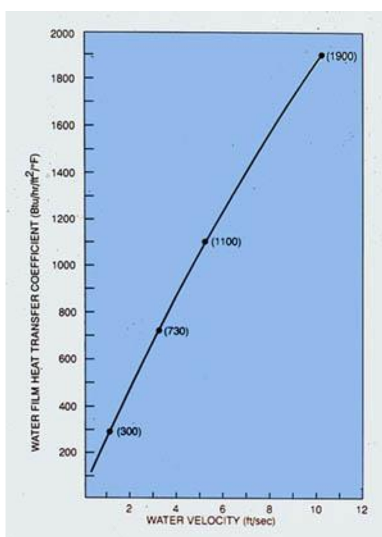


Fig. 8 Heat Transfer Coefficient vs Water Velocity Graph

CFD Analysis 2 Results

From the results it was found that with a 2 hour cycle the maximum temperature reached at 60 cycles per minute was 36 C, for 120 cycles per minute the maximum temperature reached was 49 C. This indicates that for this system a 36% increase in heat transfer between the heated surface to moving fluid was found.

The main point of this test was to demonstrate that water passing through a heat source at a higher velocity results in a better heat transfer due to the lower losses in heat through natural conduction after each cycle. It was found that the water temperature was at a higher temperature after 1 cycle at 3.2 m/s versus 1.6 m/s.

It is also known that a higher heat transfer occurs between a fluid and solid when fluid flows at higher velocities. This is due to higher heat transfer coefficients in turbulent flow, see fig. 6. This effect would aid the heat transfer resultant also in practice.

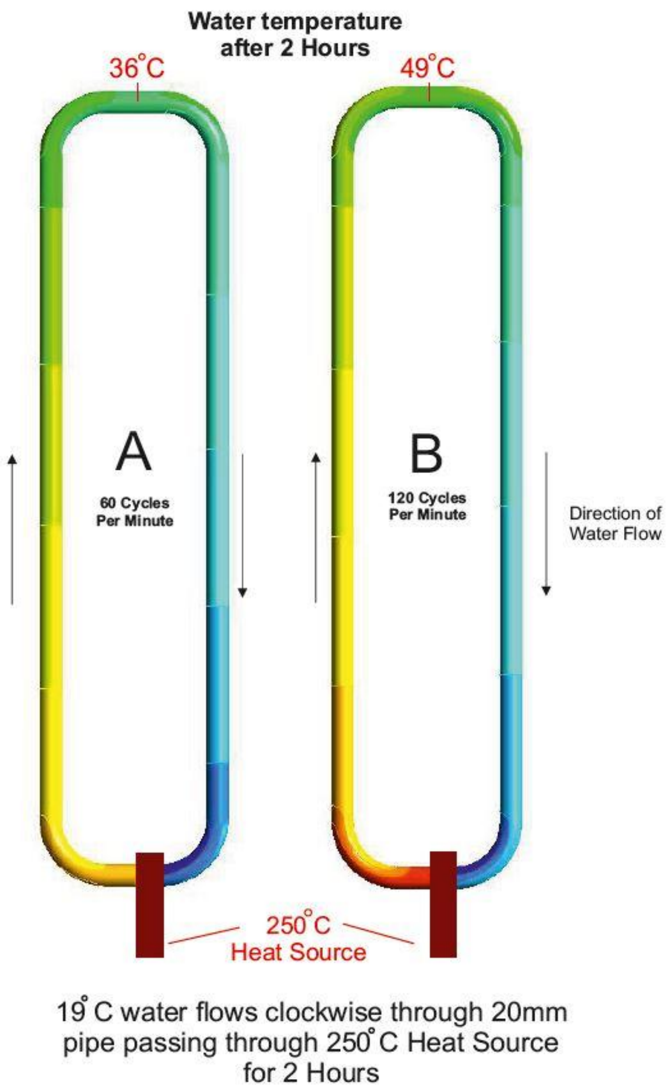


Fig. 8 Water Temperature Results after 2 Hours

Conclusion and Discussion

Two main CFD studies were carried out using Solidworks Fluid Flow and ANSYS Fluent computer simulation software.

Using Solidworks Fluid Flow, simulations were set up and carried out on three different simplified solid fuel heating systems:

- Pump On Flow System
- Pump on Return System
- SFI Injector System with Heat Hero

Fluid flow velocities were determined for each analysis and comparisons made between average fluid flow velocities at the radiator. It was seen that an increase of 48% of fluid flow was found with the Heat Heros system over both Pump On Flow System and Pump on Return.

For the Fluent analyses a basic pipe and heat source system study was set up and carried out with the fluid flow at two different velocities.

Temperatures of the fluid after each cycle for both velocities were noted and theoretical final temperatures after a two hour period were calculated. It was seen on this system that a 36% increase in temperature of the fluid was found when the fluids velocity was doubled.

From the Analyses it can be said that:

– Heat Hero could improve circulation around a heating system by potentially up to 48% when installed
– This increased circulation of a heating system could improve heat transfer in that system, potentially up to 36% according to this analyses. This is primarily due to the reduced amount of heat energy lost by the volume of water returning to the heat source at a faster rate.
– It indicates that Heat Hero would be safe to install in an open venting system as it can be seen that there are no increased pressures or restrictions in such a system.
– Ultimately from these tests Heat Hero can improve the overall efficiency of a solid fuel central heating system.

Approved By:

Conor Hayes
Senior Design Engineer