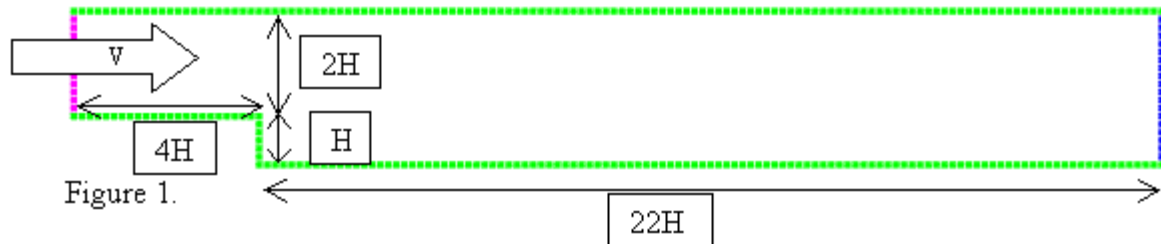


Additional Validation Examples

All these additional validation samples are two dimensional and are included as part of the Ambiens software release.

Backward Facing Step

The backward facing step (Figure 1.) is a classical flow configuration used by many CFD practitioners to validate their codes. In this example we use the following geometry under three very different conditions.

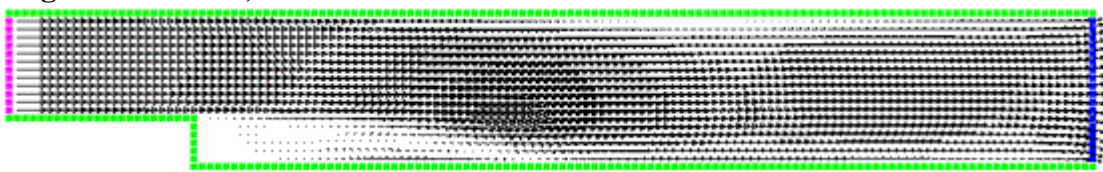


The main interest is in the reattachment point of the re-circulation caused by the step. For fully developed turbulent flow this should be roughly $7H \pm 0.5H$, where H is the height of the step. The example follows from the work performed by Kim,

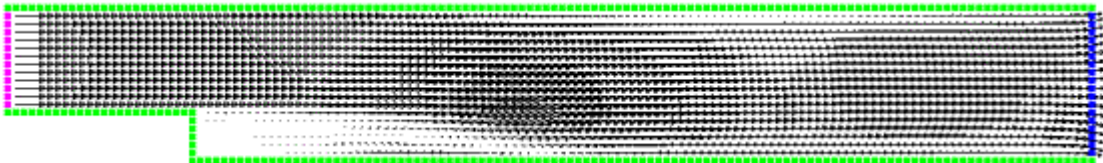
J J Kim, "Investigation of separation and reattachment of a turbulent shear layer: flow over a backward facing step", PhD Thesis, Stanford University (1978).

The following web page has more detail and references,
<http://staffweb.cms.gre.ac.uk/~ct02/research/thesis/node72.html>.

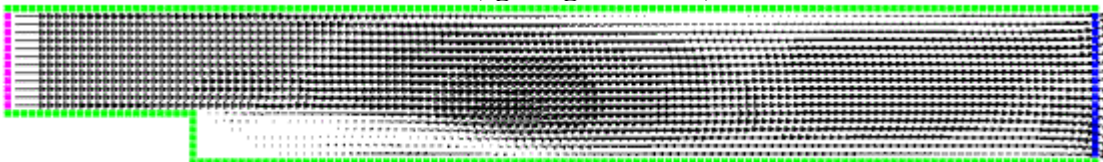
Case 1: $H = 0.5\text{m}$ and $V=1.0\text{m/s}$, giving a Reynolds Number based on the step height of $Re_H = 33,333$.



Case 2: $H = 0.1\text{m}$ and $V=5.0\text{m/s}$, giving once again $Re_H = 33,333$.



Case 3: $H = 0.132\text{m}$ and $V=1.25\text{m/s}$, giving $Re_H = 11,000$.



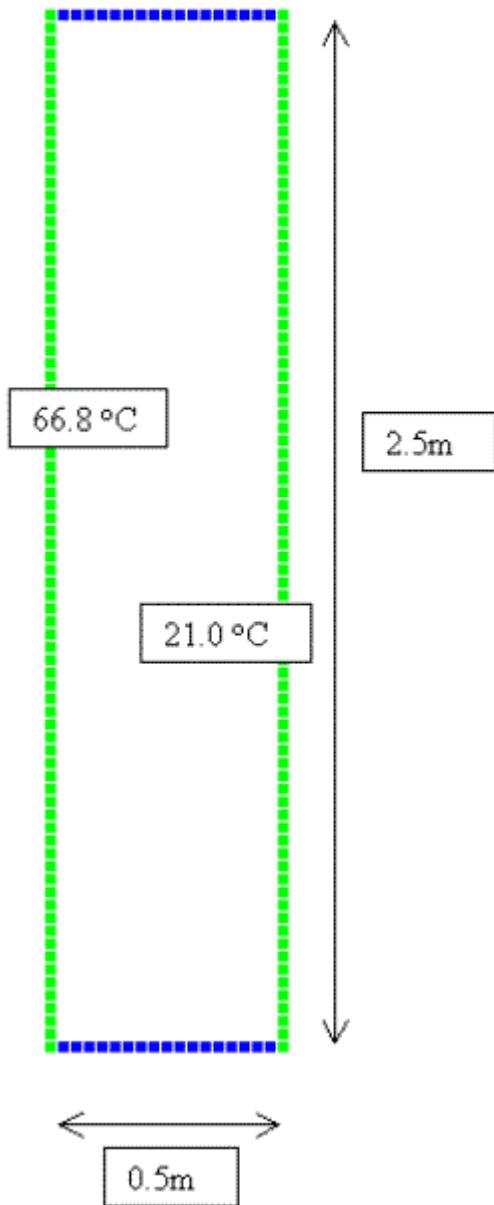
The simulations have correctly predicted the reattachment point in each of the cases. They have also managed to capture the secondary re-circulation, just over half way along the top surface. This secondary re-circulation varies as expected under the different conditions. The main re-circulation reattachment point is very close to $7H$ for each of the cases and is inside the experimental range. It should be noted that the arrow size has been rescaled for each image to aid clarity.

Heated Cavity

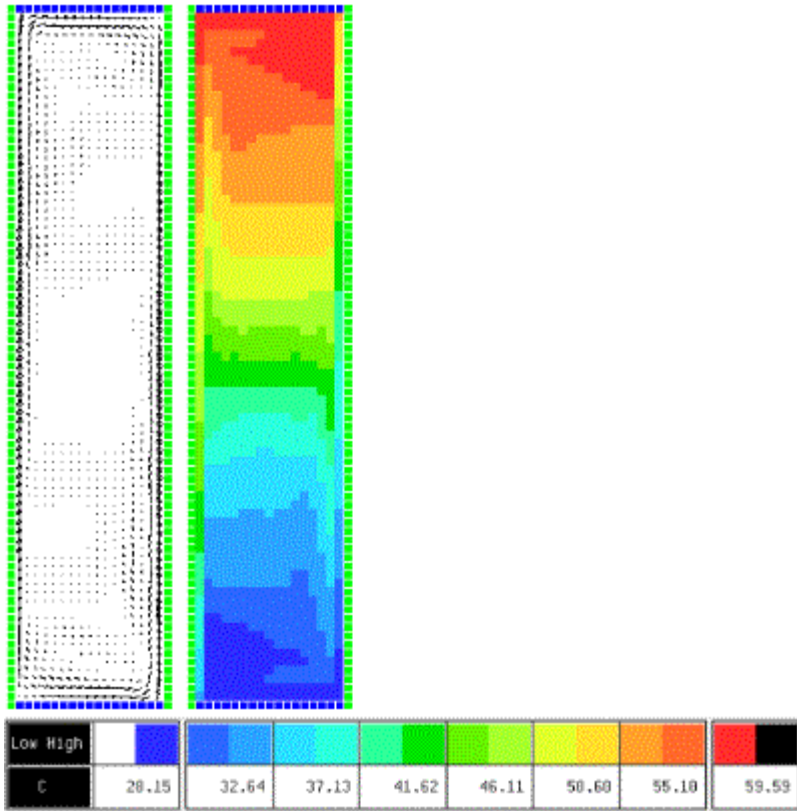
Cavity experiments were carried out by Cheeswright et al and the results can be found in the following paper.

Cheeswright, R., King, K.J. and Ziui, S., 'Experimental Data for the Validation of Computer Codes for the Prediction of Two-Dimensional Buoyant Cavity Flows' in ASME Winter Annual Meeting, Anaheim, HTD-60, December 1986.

The experimental cavity had a 5:1 aspect ratio with a temperature difference between the longer vertical sides of 45.8 degrees. The following diagram shows the 0.5m by 2.5m rectangular cavity with the surface temperatures used in this simulation. The top and bottom of the cavity were assumed to be adiabatic allowing no heat to be transferred, but they did allow for viscous forces to act on the flow.

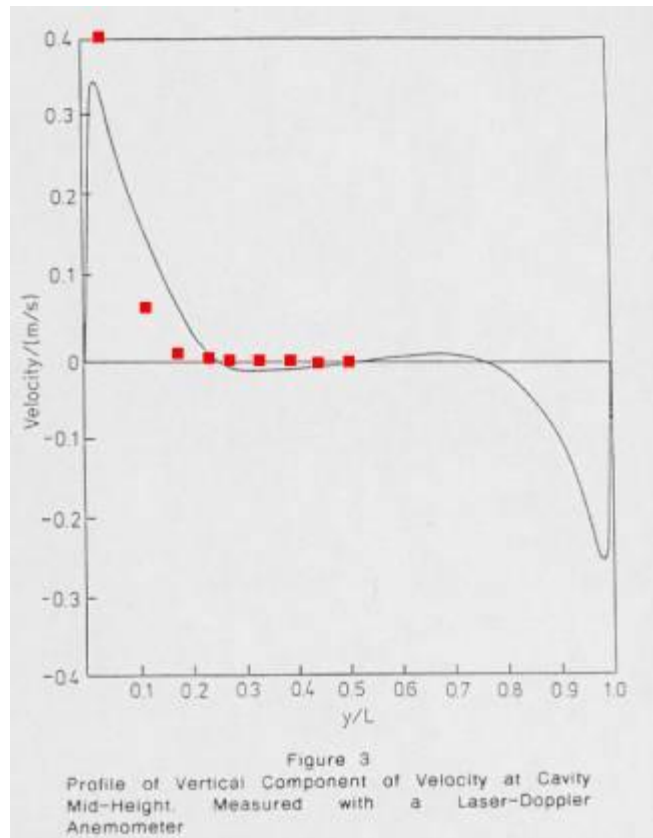


To obtain an accurate solution to this kind of natural convection problem using CFD would normally require a high mesh density close to each heated surface and the introduction of a low Reynolds number turbulence model. The new surface heat transfer algorithms used in Ambiens obviate the need for any of this special meshing or problem specific algorithmic changes and also give physically realistic results across varying mesh densities (see the section on Scaling and the larger cell simulation later in this section).



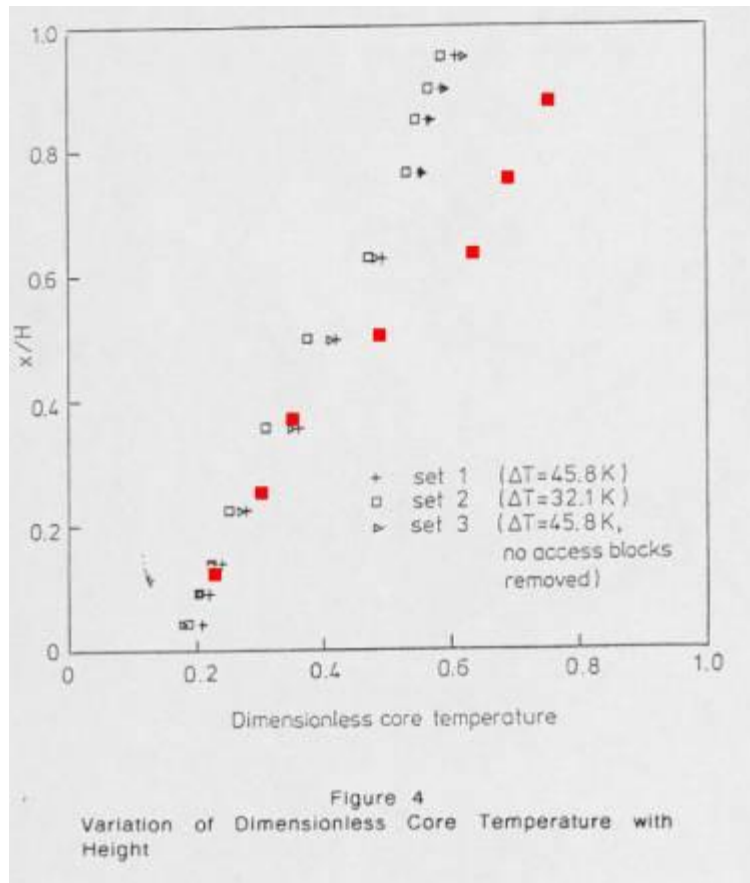
The pictures on the left show the results of the Ambiens simulation. The picture on the left shows the velocity field, with a peak velocity of 0.43 m/s slightly above the centre on the left wall and slightly below the centre on the right wall. The temperature plot has the corresponding scale just below. The simulation took approximately 7 seconds on an 800MHz Pentium 3 Xeon processor.

The following image has been scanned from the original paper and shows the vertical velocity profile half way up the cavity. The line clearly shows the large upward (positive) velocity close to the heated surface and the large downward velocity (negative) near the colder surface. A largely stagnant core is also clearly shown in the centre of the cavity. Additional red marks indicate the Ambiens results for the velocity at each of the 9 cells across the centre of the cavity in the simulation.



For such a relatively coarse mesh the velocity profile has been predicted very well. The peak simulated speed is slightly higher than the measurement and the simulated profile decays a little too fast, but taken together this becomes an almost exact match for the measured convective flux close to the wall. It should be noted that the peak simulated velocity of 0.43m/s is also very close to the measured peak of 0.41m/s, only the simulation reaches its peak just a little too early as it travels up the cavity. Some part of the discrepancies may be attributable to the three dimensional nature of the experiment and end wall effects. Note the lack of symmetry in the measured profile where the upward velocity is in fact greater than the downward.

The next image, again taken from the original paper, shows the variation in dimensionless core temperature. This is effectively a plot indicating the vertical stratification within the cavity through its centre. Once again red marks show the Ambiens results for seven locations through the vertical centre of the cavity.



The results agree very well at the colder/lower end of the cavity and only start to diverge after the centre. Fairly obviously the average value of the non-dimensional core temperature should be 0.5, i.e. half way between the two surface temperatures, but it can be seen from the diagram that the measured values average somewhere around 0.4 so the overall temperature in the cavity was too low. Other CFD products have found a similar trend with their runs and agree that heat must have escaped through the insulation (see diagram below). The greatest heat will escape where the temperature gradients are highest, in this case at the top of the cavity, and explain the relatively colder measurements from the top of the cavity.

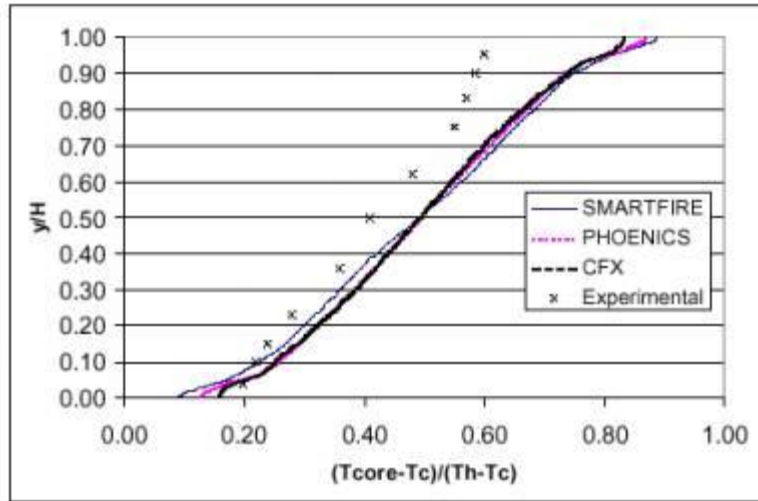
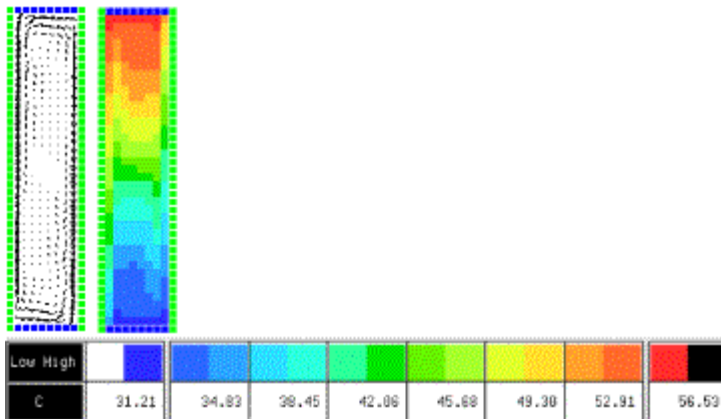


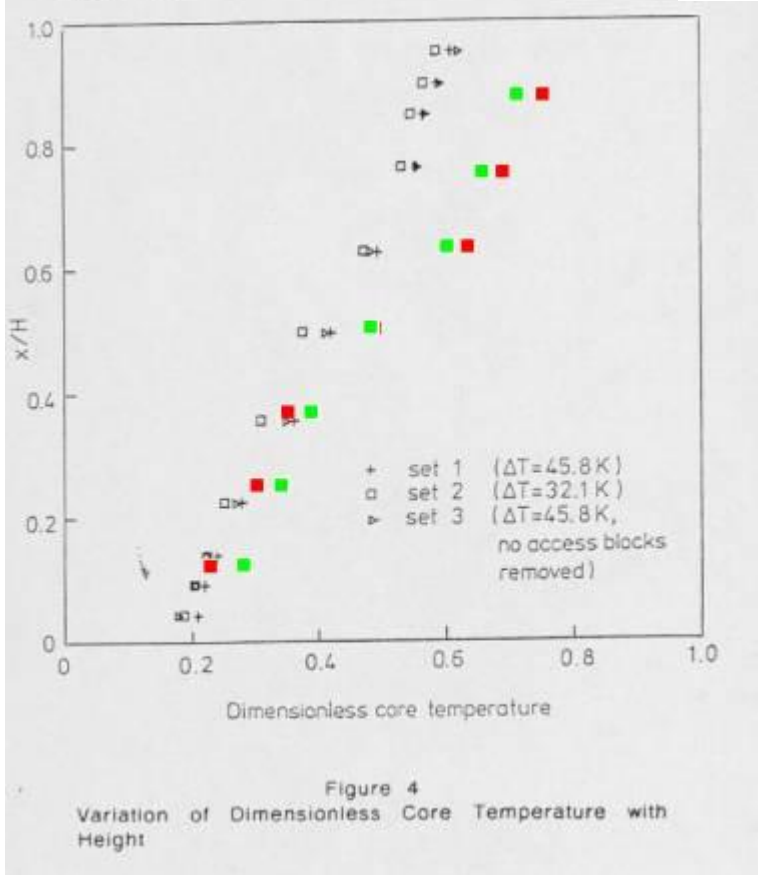
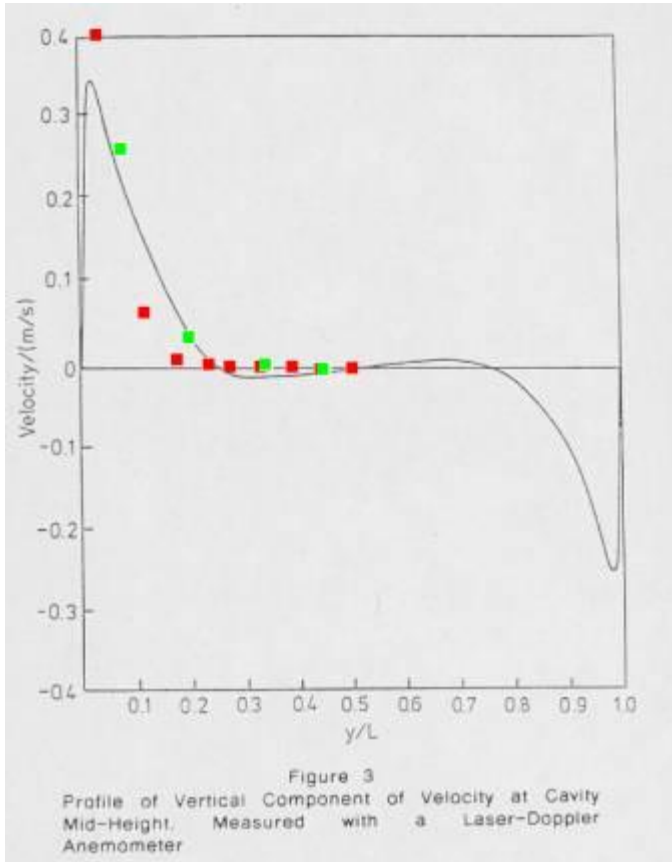
Figure 13 Temperature variation along the $x/L = 0.5$ axis

As another test a second model of the cavity was built with even fewer cells. This simulation demonstrates the flexibility of the new boundary heat transfer algorithms and the very useful data that can still be obtained with even a very rough model of a problem.

The following pictures show the Ambiens results from the new model. The peak velocity was 0.26m/s and the temperature scale is below. The simulation took approximately one second on an 800MHz Pentium 3 Xeon processor.



The temperature variation is not captured quite as well as before, but the convective and thermal fluxes have scaled almost perfectly which explains the overall closeness of the results. The following diagrams are copies of the earlier velocity and temperature plots but this time with the addition of green marks for the new results.



These results clearly demonstrate that the Ambiens boundary heat transfer algorithms don't need a very high mesh density close to a heated surface in order to obtain

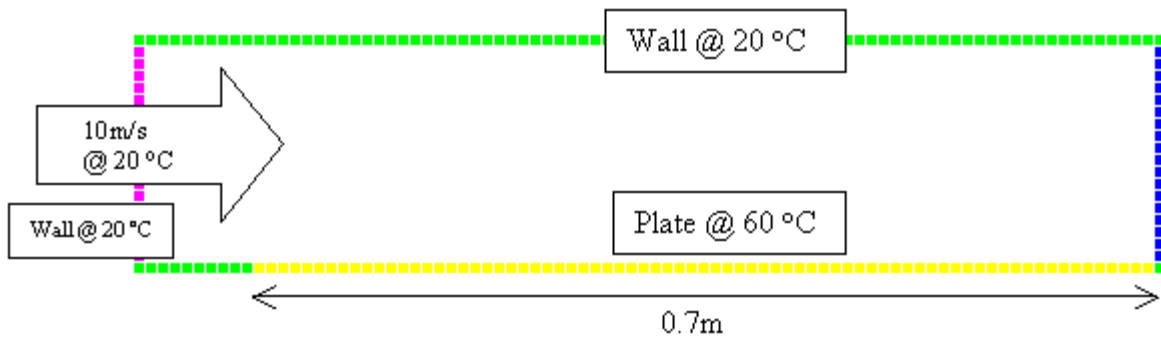
accurate results for a natural convection system. In fact, the results from either of the simulations are accurate enough for day-to-day design or engineering use.

Hot Plate Examples

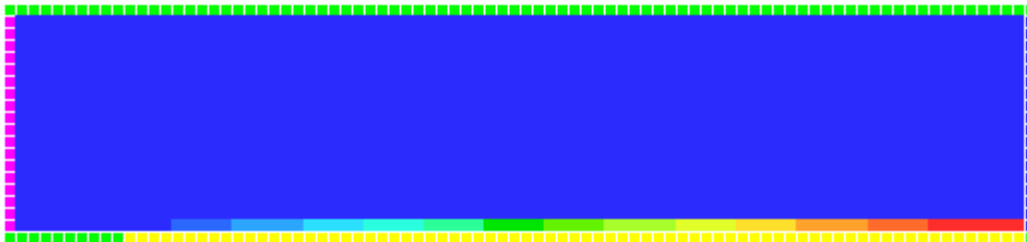
Forced convection heat transfer is an important feature of the air movement inside a building, which is generally comprised of both natural and forced convection in varying degrees. Two examples are presented here. The first is fully turbulent forced convection, while the second is laminar. The simulations for the turbulent case are compared to semi-empirical formulae that have been developed for the engineering sciences. The laminar case admits an almost complete analytic solution based on boundary layer theory, which is used for the comparison.

Turbulent Case

The first fully turbulent configuration is as follows.



Each cell is 1cm by 1cm. The result for air temperature is shown below. The simulation took approximately 9 seconds on an 800MHz Pentium 3 Xeon processor.



Only the three cells nearest the hot plate at the outlet have become hotter and are above 20 °C. They are 22.80 °C, 20.10 °C and 20.01 °C. The vast majority of the heat absorbed from the plate remains in the first cell adjacent to the plate. This is very much in keeping with the experimental findings for turbulent boundary layer thickness for these conditions.

The semi empirical formula for calculating the average turbulent heat transfer from a plate is taken from Holman[1] p.181 and is as follows.

$$\bar{h} = \frac{k}{L} Pr^{\frac{1}{3}} (0.037 Re^{0.8} - 850)$$

Where k is the conductivity of air, L the length of the plate, Pr the Prandtl number and Re the Reynolds number. The thermal properties are taken at the film temperature which is the average of the surface and air temperatures, 40°C or 313K . This gives the following values.

$$\mu = 2.007 \times 10^{-5}$$

$$Pr = 0.7$$

$$k = 0.02723$$

$$c = 1007$$

$$Re = \frac{u_\infty L \rho}{\mu} = \frac{(10)(0.75)(1.128)}{2.007 \times 10^{-5}} = 4.215 \times 10^5$$

Where μ is the viscosity, ρ is the density, c is the specific heat capacity in $\text{J/kg}^\circ\text{C}$ and u_∞ is the stream velocity. Substituting these values into the formula gives.

$$\bar{h} = \frac{0.02723}{0.75} 0.7^{\frac{1}{3}} (0.037(4.215 \times 10^5)^{0.8} - 850) = 10.3$$

The total rate of heat transfer from the plate q , is defined as.

$$q = \bar{h}A(T_w - T_\infty) = (10.3)(0.75)(60 - 20) = 309$$

Where A is the area of the plate (the plate has a depth of 1 metre), T_w is the plate temperature and T_∞ is the air stream temperature. So according to this formula the rate of heat output from the plate is 309 Watts.

To compare the simulation results the rate of increase in thermal energy of the air leaving the plate can be calculated as follows. The volume of air leaving the cell adjacent to the plate is V the velocity, 10m/s , multiplied by the cell height, 1cm , and then the unit depth of 1m . Multiplying this by the density of air will give the mass flux m' or kg/s as follows.

$$V \times 0.01 \times 1.0 \times \rho = 10 \times 0.01 \times 1.0 \times 1.128 = 0.1128$$

The energy flow is then calculated as

$$q = m'c\Delta T = 0.1128 \times 1007 \times (22.80 - 20.0) = 318.05$$

ΔT is the temperature rise from the incoming air stream temperature to the temperature of the air leaving the end of the plate. So the rate of heat gain from only the first cell adjacent to the plate is 318.05 Watts. The total simulated energy flow is then calculated from the sum of all the cells at the outlet. Obviously if ΔT is zero there is no energy increase and flux, so we only need to sum the next two cells where ΔT is non-zero. The energy output for the next two cells based on the outgoing simulation temperatures are as follows.

$$0.1128 \times 1007 \times (20.10 - 20.0) = 11.36$$

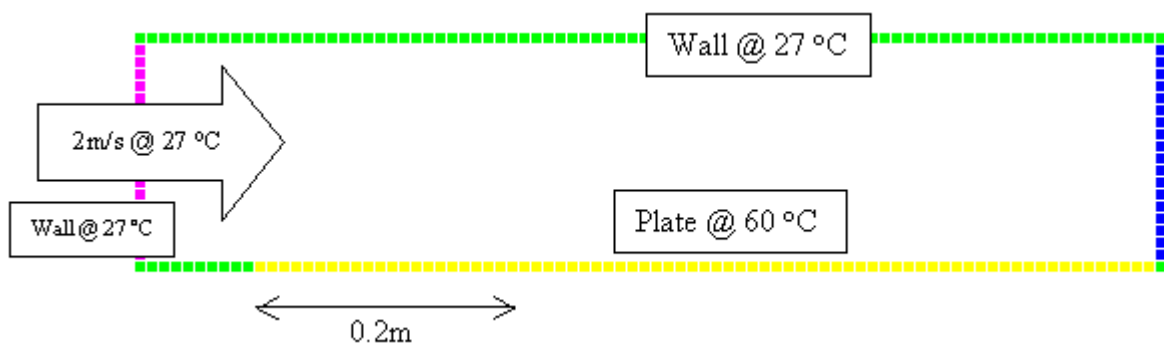
And

$$0.1128 \times 1007 \times (20.01 - 20.0) = 1.136$$

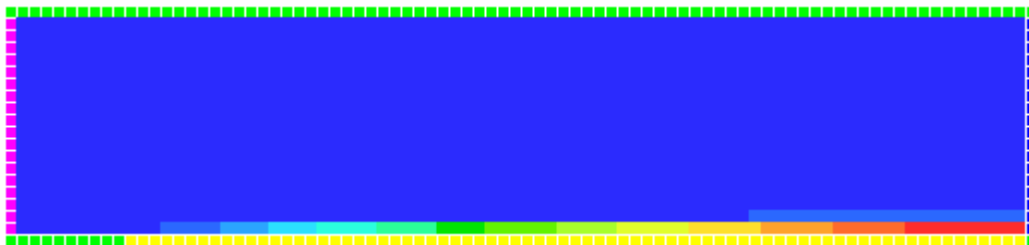
This gives a total output of 330.5 Watts which differs by only 6.7% from the result given by the semi-empirical formula.

Laminar Case

The laminar case is almost the same configuration but running with a reduced velocity of 2m/s. The configuration is shown in the following diagram.



In this case the heat transferred over the first 20cm of the plate is to be calculated. The next image shows the result of the simulation which took approximately 2 seconds on an 800MHz Pentium 3 Xeon processor.



Although we are not directly interested in anything beyond the 20cm position it is worth noting that at the end of the 75cm plate the heat is just starting to enter the second cell away from the plate in a discernable way. This is in line with the theoretical result that the boundary layer thickness at 75cm is just over 1cm.

The analytical solution to this problem relies on the following formula from Holman[1] p.171.

$$\bar{h} = 0.664 \frac{k}{L} \text{Pr}^{\frac{1}{2}} \text{Re}^{\frac{1}{2}}$$

The symbols are as defined in the previous section. This formula is derived from an analytical treatment of both the momentum and energy conservation within the

boundary layer. The following web page has a good description of the theory with only a slightly different treatment to Holman's
<http://www.owl.net.rice.edu/~azul01/momentum.htm>.

The thermal properties are taken at the film temperature which is the average of the surface and air temperatures, 43.5 °C or 316.5K. This gives the following values.

$$\nu = 1.736 \times 10^{-5}$$

$$Pr = 0.7$$

$$k = 0.02749$$

$$c = 1006$$

$$Re = \frac{u_w L}{\nu} = \frac{(2)(0.2)}{1.736 \times 10^{-5}} = 23,041$$

Where ν is the kinematic viscosity, and the other symbols are as described in the previous section. Substituting these values into the formula above gives.

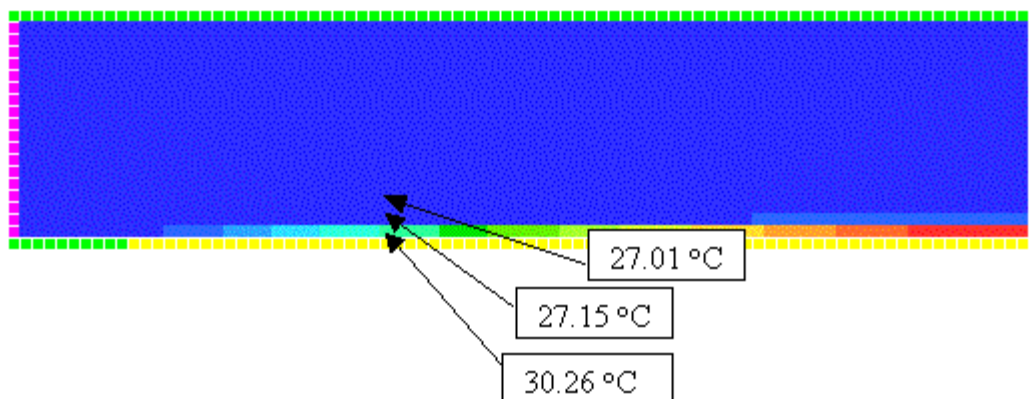
$$\bar{h} = (0.664) \frac{0.02749}{0.2} (23,401)^{\frac{1}{2}} = 12.39$$

So the average heat transfer coefficient over the first 20cm of the plate is 12.39W/m² °C. Substituting this into the following heat flow equation we obtain.

$$q = \bar{h}A(T_w - T_\infty) = (12.39)(0.2)(60 - 27) = 81.774$$

The total heat flow based on the analytical treatment is 81.774 Watts.

We can compute the heat flow from our CFD simulation in a similar way to the method described in the previous section. Firstly we find the temperatures of the cells just past the 20cm mark whose temperature is not equal to 27°C, see the following diagram.



It is now possible to compute the total heat flow as follows. The mass flux \dot{m} through each cell is.

$$V \times 0.01 \times 1.0 \times \rho = 2 \times 0.01 \times 1.0 \times 1.2 = 0.024$$

The energy flow for the cell adjacent to the plate is then calculated as

$$q = m'c\Delta T = 0.024 \times 1006 \times (30.26 - 27.0) = 78.71$$

and the next two cells are as follows.

$$0.024 \times 1006 \times (27.15 - 27.0) = 3.62$$

And

$$0.024 \times 1006 \times (27.01 - 27.0) = 0.24$$

This gives a total output of 82.57 Watts which differs by only 1% from the result given by the formula based on boundary layer theory.

[1] Holman, J. P. "Heat Transfer", Fourth Edition, McGraw Hill, 1976.