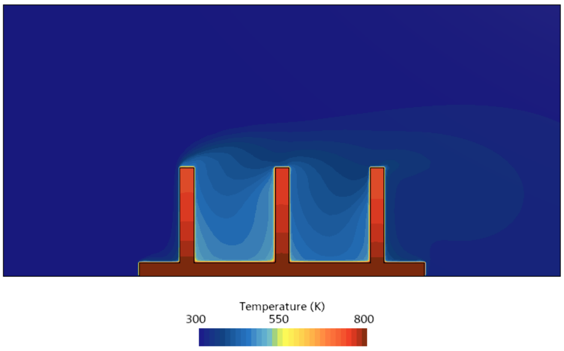
Submission via Blackboard. Deadline: Tuesday 16 April, 6 pm

Overview



This coursework aims to simulate conjugate heat transfer with both natural and forced convection.

The main new elements covered are:

- 2-dimensional flow;
- multiple continua (solid/fluid);
- heat transfer.

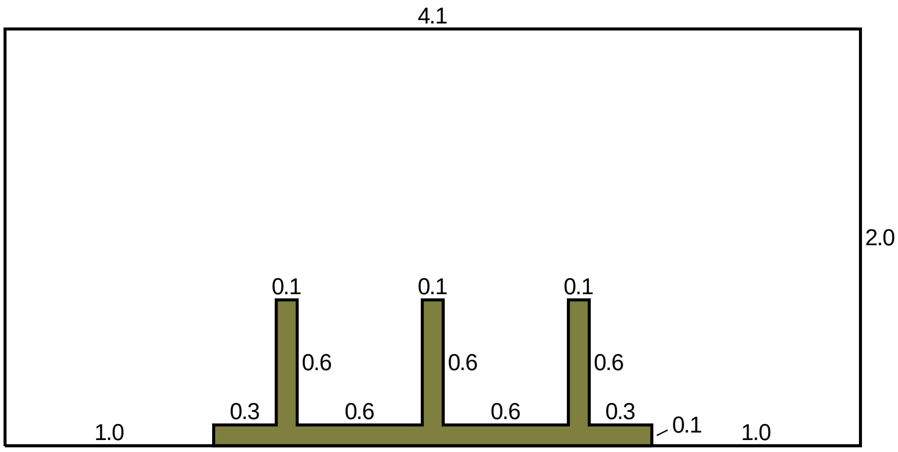
The geometry is two-dimensional and consists of two parts:

- aluminium cooling fins, heated at their base;
- a surrounding air cavity.

In Part A – natural convection – the left and right-boundaries are fixed-temperature walls.
In Part B – forced convection – they are inflow and outflow, respectively.

Geometry

Using the inbuilt CAD module, create a geometry incorporating two distinct 3-d bodies (fins and air) with the dimensions below (in metres):



Although the ultimate simulation is two-dimensional, use the CAD system to extrude your sketches a modest (i.e. clickable) distance to create three-dimensional bodies. Make sure that all boundaries (see below) are named.

Create `geometry parts` from CAD model. You should end up with two parts – fins and air – corresponding to the two original CAD bodies. Simultaneously assign these parts to `regions`, checking that an internal interface is created between them.

Boundary Types

The fin boundaries are the base (fixed-temperature wall) and the fin-air interface (wall).

The air boundaries are the lower boundary (adiabatic wall), top boundary (fixed-temperature wall), left and right boundaries (fixed-temperature walls in part (a); inflow and outflow respectively in part (b)) and the fin-air interface (wall).

Mesh

Initially, create separate 3-d meshes for each part, according to the following.
Air
- `Surface Remeshing` and `Trimmed Cell Mesher` models;
- `Base Size`: 0.05 m; `Maximum Cell Size`: 100% of `Base Size`;
- a `Surface Control` on the interface, with `Target Surface Size` 0.01 m.

Fins
- `Surface Remeshing` and `Trimmed Cell Mesher` models;
- `Base Size`: 0.01 m; `Maximum Cell Size`: 100% of `Base Size`.

After creating 3-d meshes, use the `Menu` pull-down menu and choose `Convert to 2D`. Create a mesh plot to check.[1]

Continua Models

Create additional `continua` and rename them "Fluid" and "Solid". Assign these to the appropriate 2-d `regions`. Once this is done you should delete any unwanted 3-d `regions` and `continua` and rename objects to avoid confusion. Use the following models:

Fluid Continuum
- `Two Dimensional`
- `Steady`
- `Gas`
- `Coupled Flow`
- `Ideal Gas`
- `Coupled Energy`
- `Turbulent`
- `Reynolds-Averaged Navier-Stokes`
- `K-Epsilon Turbulence`
- `Standard K-Epsilon`
- `High y+ Wall Treatment`
- `Gravity` (found in the "Optional" box)

[1] An alternative way to create 2-d meshes is to "badge" the original geometric parts for 2-d meshing first. However, sadly, this doesn't appear to work with the `Trimmed Cell Mesher` in the current version of StarCCM+.

Solid Continuum
- `Two Dimensional`
- `Steady`
- `Solid`
- `Coupled Solid Energy` (found in the "Optional" box)
- `Constant Density`

Boundary Conditions

Fluid
- Bottom: adiabatic wall
- Top: fixed-temperature (300 K) wall

In part A (natural convection):
- Left and right: fixed-temperature (300 K) wall

In part B (forced convection):
- Left: Velocity Inlet (turbulent intensity: 0.05; viscosity ratio: 100; velocity: 0.5 m s^{-1}; temperature: 300 K)
- Right: Outlet

Solid
- Base: fixed-temperature (800 K) wall

Stopping Criteria

- Disable "`Maximum Steps`".
- Set additional `monitor` criteria to stop when all of `Continuity`, `Energy`, `X-momentum` and `Y-momentum` normalised residuals are below 10^{-4}.

Post-Processing

You will have to create your own reports and plots as necessary.

Main Submission (One PDF file please)

Please submit the following.

(0) A single plot of the mesh.

Part A: natural convection (left and right boundaries are fixed-temperature walls)

(A1) The final residuals plot.

(A2) A scalar plot of temperature.

(A3) A velocity-vector plot, with vectors coloured by temperature.

(A4) A statement of the total heat transfer into the domain through the base of the fins.

Part B: forced convection (left boundary: inlet; right boundary: outlet)

(B1) The final residuals plot.

(B2) A scalar plot of temperature.

(B3) A velocity-vector plot, with vectors coloured by temperature.

(B4) A statement of the total heat transfer into the domain through the base of the fins.

(B5) The mass flux and the total heat transport inward through the left boundary.
Confirm that the latter is given by

$$(\text{mass flux}) \times (\text{specific heat capacity}) \times (\text{absolute temperature})$$

and use the mass flux to deduce the density at inlet.

(B6) The pressure drop (left to right), based on average pressures on these boundaries and the power required to drive this flow.

(B7) Comment (briefly!) on the advantages and disadvantages of natural and forced convection as a means of cooling in this instance.

Secondary Submission (Convert to a second PDF file please)

The StarCCM+ `Summary Report` at the end of your computation for Part B.