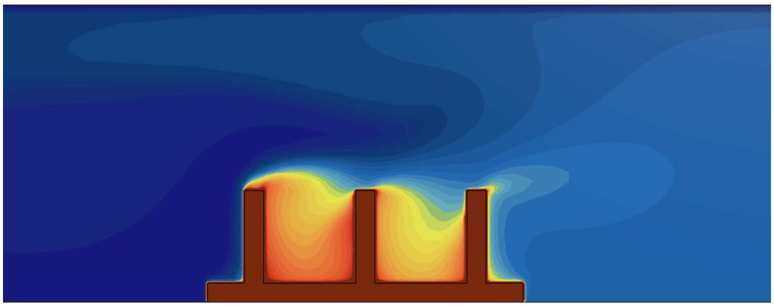
Submission via Canvas. Deadline: Wednesday 22 April (week 9), 2 pm.

Overview

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



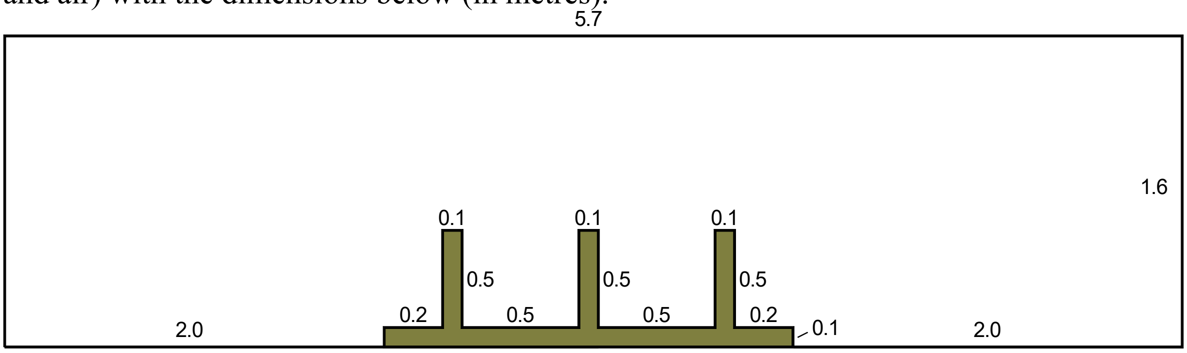
The main new elements covered are:
- 2-dimensional flow;
- multiple continua (solid/fluid);
- conjugate heat transfer.

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 adiabatic 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 simulation will be two-dimensional, extrude your CAD sketches a modest (i.e. clickable) distance to create 3D bodies. Make sure that all boundaries are named.

Create `geometry parts` from the CAD model. You should end up with two parts – fins and air – corresponding to the two original CAD bodies.

Execute the operation `Badge for 2D Meshing` for both parts (simultaneously). This will mark off the $z = 0$ face on each part.

Simultaneously assign these parts to `regions`, checking that an internal interface is created between them.

Boundary Types

Set boundary types for each region. For the fin region all boundaries are walls. For the air region: in Part A left and right boundaries are walls, whilst in Part B they are inflow and outflow, respectively.

Mesh

Set up separate `Automated Mesh` (*not* `Automated Mesh(2D)`) for each part as follows:

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`.

Run the mesh generator to produce a 2D mesh.

Continua Models

Create new and separate `continua` named "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)

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: adiabatic wall

In Part B (forced convection):
- Left: Velocity Inlet (turbulent intensity: 0.05; viscosity ratio: 100; velocity: 0.2 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

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. Choose a sensible temperature range: not just min-to-max.

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

(A4) The total heat transfer rate (per metre span) 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. Choose a sensible temperature range, not just min-to-max.

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

(B4) The heat transfer rate (per metre span) into the domain through the base of the fins.

(B5) The net heat flux (right minus left, per metre span) through inflow and outflow boundaries and the mass flow rate. Deduce from these the average exit temperature.

(B6) The pressure drop through the domain and the power required (per metre span) 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.