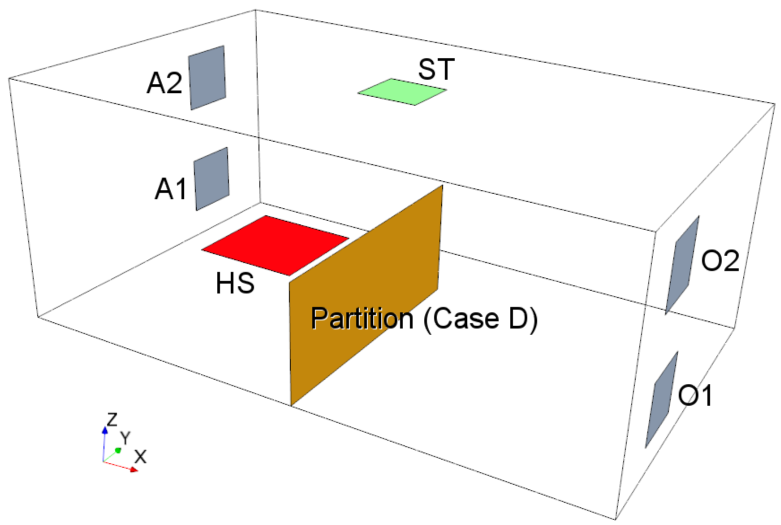
1. Notes on the software
2. Assigned exercise (submission via Blackboard; deadline: Thursday Week 9, 11 pm)

1.1 Features of the Exercise

- *Natural ventilation* – driven by localised heat sources;
- *Forced ventilation* – driven by external pressure differences;
- *Buoyancy forces* – due to temperature-dependent density changes;
- *Reynolds-stress turbulence modelling*
- *Zero-thickness interior walls*
- *Reports* – post-processed data: volume/surface averages, mass flow rate etc.;
- *Extraction and export of data* – export data to process with other programs;
- *Derived parts* – plane sections, streamlines, line probes.

1.2 Overview of the Exercise

The object is to compute the steady temperature distribution in a model room with various ventilation configurations. A sketch of the room is given below; detailed dimensions are given in Appendix 1. Primary outputs will be the temperature distribution in the room, volume-average temperature and ventilation flow rates. The effects of stack-induced ventilation and the inclusion of an internal partition will be considered.



1.3 Geometry

The detailed room geometry is given in Appendix 1. The built-in 3D-CAD system can be used to create the room volume. There are face-imprinted sketches for an area heat source HS, possible window openings on walls opposite and adjacent to the heat source (O1, O2, A1, A2) and a roof-centre stack (or chimney) ST. Various combinations of these will be examined. Finally, an internal partition will be added as a zero-thickness internal wall; the creation and meshing of this is described in Appendix 2.

1.4 Boundary Types

When vents are closed leave them as smooth walls. When open, they will be treated (see the assigned exercise in Section 2) as either:

- stagnation inlets (fixed external *total* pressure); or
- pressure outlets (fixed external *static* pressure).

In the case of a stack, a negative static pressure is set to simulate wind effects.

1.5 Mesh

Use `Surface Remesher` and `Trimmer` models with a base size of 0.15 m.

1.6 Model Physics

Uncheck the “Auto-select recommended models” box.

Use the following model physics:

- `3-Dimensional`
- `Steady`
- `Gas`
- `Coupled Flow` solver (raise the default Courant number in `Solver` parameters)
- `Constant Density` (but see below)
- `Turbulent`
- `Reynolds-Averaged Navier-Stokes`
- `Reynolds Stress Turbulence`
- `Linear Pressure Strain`
- `High` y^+ `Wall Treatment`

To make density vary with temperature (but Boussinesq and incompressible) choose:

- `Gravity`
- `Boussinesq Model`
- `Coupled Energy`

To match other temperature defaults, set

- `Physics > Reference Values > Reference Temperature` to 300K.

After model physics has been defined:

- The heat source boundary HS should have its `Thermal Specification` changed to `Heat Source`, and a `Heat Source` value of 15000 W (15 kW) set.

1.7 Convergence Criteria

Without a forced inflow a residuals-based criterion for convergence is unsuitable. Uncheck the `Maximum Steps` criterion and use your own judgement as to when to *manually* stop iterating. This should be based on suitable metrics becoming steady (e.g. room-average temperature, or average temperature on the relevant outlet surface).

1.8 Derived Parts

Derived parts are additional geometric entities used mainly for plotting or extracting data or averages on. The most important ones here are

- plane sections;
- line probes;
- streamlines.

1.9 Reports

Reports are used to analyse data on one or more volumes or surfaces. Examples include forces and force coefficients, areas and volumes, averages, maximum and minimum values.

You may have to define various parameters in the corresponding properties window:

- the volume or surface on which the report is to be applied;
- directions (for vector quantities such as forces, projected areas, etc);
- normalising parameters (e.g. reference density, velocity, area).

Run a report by right clicking and choosing `Run Report`. You can also create monitors and plots showing evolution over iterations or time from the right-click options.

1.10 Export

Plots
The default format for plot files is `.png`. Plots and scenes can be exported to file by right-clicking on an unfilled part of them (or on the scene name in the object tree if white space is hard to find) and choosing `Hardcopy`.

Tables
Here, use an `XYZ Internal Table`. Set parts and variables to extract. `Extract`, then `Tabulate`. Data from tables can be exported as `.csv` files, which contain comma-delimited columnar data and are easily imported into other software packages such as `Excel`, `gnuplot` or `Matlab` for plotting or further analysis.

2. ASSIGNED EXERCISE

Apart from the CAD feature tree, all plots should use plot files, not screen dumps. *Please do not submit any other explanations or plots. If you cannot obtain one particular item do not spend excessive time on it.*

Upload only the following two files to Blackboard (**in PDF format**). Note, in particular, that you will be penalised if:

- you include unnecessary padding, such as additional plots or explanations;
- you use non-PDF format or more separate files;
- the page count for items 1-10 exceeds 10 pages.

Item 1: Main Submission

Geometry and Mesh

Submit:

1. A limited screen dump showing the feature tree for the 3D-CAD.

2. A single representative plot of the overall geometry.

3. One or two representative plots of the mesh.

Cases A, B, C: Different Ventilation Strategies

Make sure that the heat source is operating. The following table defines one lower inflow (`stagnation inflow`) and one higher outflow (`pressure outlet`) for each case.

Case	Stagnation inlet	Pressure outlet	Non-standard
A	A1	O2	–
B	O1	A2	–
C	O1	ST	reduce outlet gauge pressure to –50 Pa

Compare and contrast these cases (side-by-side) via:

4. Plots (one for each case) of final temperature on the horizontal plane $z = 1$ m. Use a range 300 – 325 K.

5. Composite plots (similar for each case) of temperature and overlaid velocity vectors on one or more vertical planes to illustrate flow and temperature distributions. Use a common temperature scale of 300 – 325 K and contrasting constant-colour vectors.

6. Plots (one for each case) of streamlines initiated at the inflow. Colour by temperature.

7. Reported values for each case of:
   (i) room average temperature;
   (ii) average temperature at outlet;
   (iii) mass flow rate at outlet.

8. For all cases, determine whether the average temperature and mass flow rate at outlet is consistent with the applied heat flux on surface HS. Show all your logic and working. Suggest reasons for any discrepancy.

9. A graph, created *outside* of StarCCM+, of temperature along a vertical line at the centre of the room. All cases A-C should be included in the same graph.

Case D: Partitioned Room

10. Include the partition that spans part of the centre of the room (see Appendices 1 and 2 for where and how to include this). Use the O1–ST inlet / stack outlet configuration (as in Case C) and compute the flow.

    Discuss the effects on room temperature distribution and flow behaviour of including this partition.

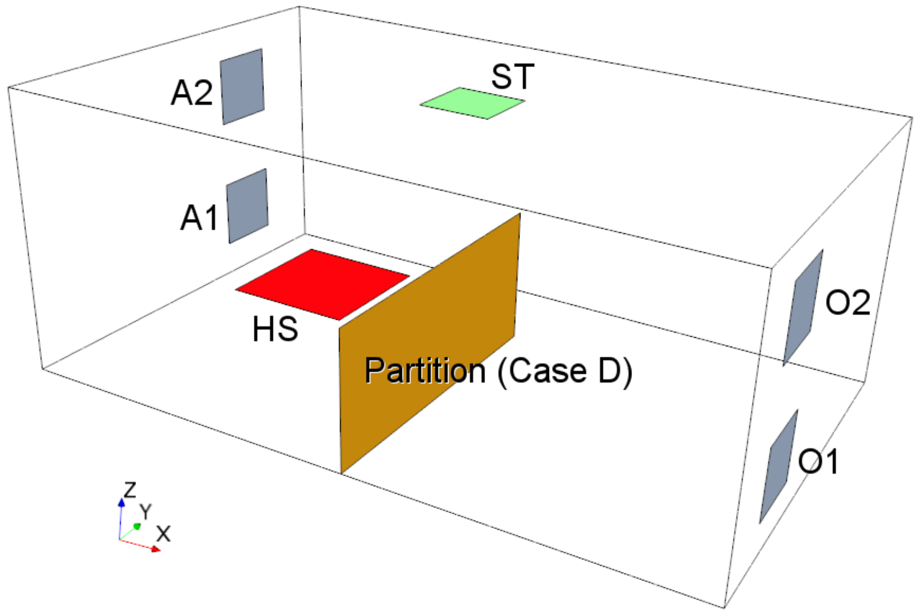
    This part is deliberately left open-ended. You may use whatever plots, metrics or discussion you choose to support your argument. Marks will be given for showing initiative.

<u>Item 2: StarCCM+ Summary Report</u>

The summary report for the calculation (`File > Summary Report`) should be created after doing all of Case D (if possible), so that all your working is included.

Make sure that the file submitted to Blackboard is in PDF format. (If you wish you can reopen the HTML file with Microsoft Word and save as PDF. You should reduce the font size at the same time.)

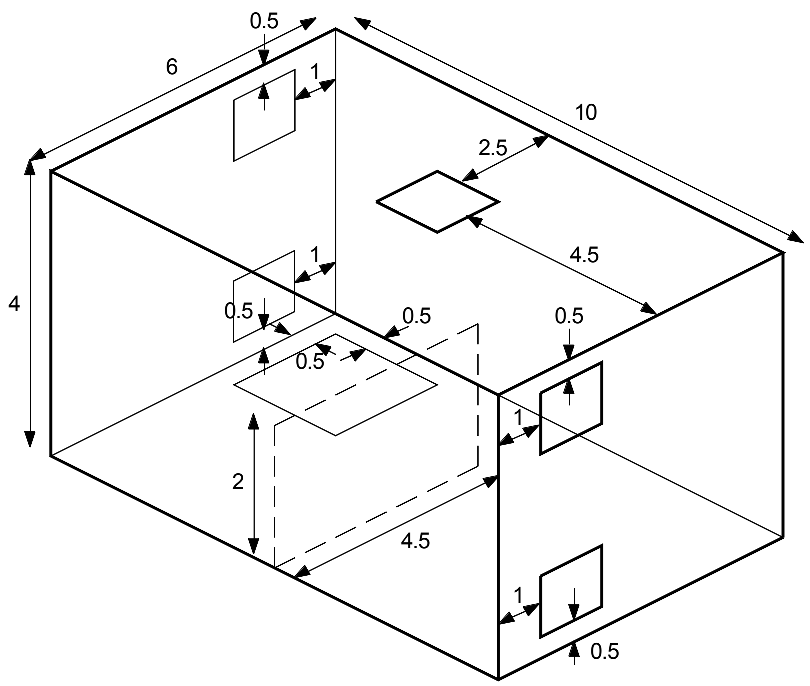
Appendix 1: Geometry



The room is 10 m long × 6 m wide × 4 m high. The face vents (O1, O2, A1, A2) and ceiling stack (ST) are **1 m × 1 m squares**. The heat source (HS) is a **2 m × 2 m square**. Vents, stack and heat source are flush to room faces and may be formed by imprinting sketches to isolate them as separate surfaces.

The separate internal partition (dashed in the diagram below) of **4.5 m × 2 m** is only used in Case D. Instructions for constructing this as a zero-thickness *sheet* are given in Appendix 2.

Other dimensions (in metres) are given below. The origin of coordinates is at floor centre of the room and you should adhere to the orientation of *xyz* coordinates given above.



Appendix 2: Creating Zero-Thickness Internal Walls

These avoid having to fit a fine mesh round something of non-zero, but small thickness.

- In `3D-CAD` create any necessary transform planes and sketches for the internal walls. Right-click the sketches and create `sheets`.

- Select the main body and all sheets simultaneously, right-click and choose
  `Boolean Imprint`
  (otherwise subsequent surface wrapping may modify edges).

- Exit `3D-CAD`. Select the whole CAD object and right click; choose
  `New Geometry Part`
  `I`t will convert it to multiple parts.

- Select all relevant geometric parts simultaneously, right-click and choose
  `Create Mesh Operation > Surface Preparation > Extract Volume.`
  Execute this operation (either by checking the box in the original creation, or right-clicking and executing once created).

- Right-click the new "`Extract Volume`" and assign to a `Region`, remembering to change the second option to
  `Create a Boundary for Each Part Surface`
  and the last option box to
  `Do Not Create Interfaces From Contacts`

- Set up and execute an automated-mesh operation using the "`Extract Volume`" geometric part only.