1. Notes on the software
2. Assigned exercise (submission via Blackboard; deadline: Monday 10th February, 6 pm)

---

1. NOTES ON THE SOFTWARE

`STAR-CCM+` generates a single simulation file (type `.sim`). You are advised to save this initially on the local disk, not your P-drive or USB drive, as these are slow to access. It can be copied elsewhere afterwards.

1.1 Typical Steps in a Simulation

0. **Start server process** (`[File]` ↓ `New Simulation`)

1. **Create a CAD model** (`Geometry > 3D-CAD Models` → `New`)

2. **Create computational geometry**
   2.1 Create a geometry part (*CAD model* → `New Geometry Part`)
   2.2 Create a region (`Geometry > Parts >` *part* → `Assign Parts to Regions`)
   2.3 Define boundary <u>types</u> (`Regions >` *region* `> Boundaries`)

3. **Generate a mesh (automated, parts-based meshing)**
   3.1 Set up (`Geometry > Operations` → `New > Mesh` → `Automated Mesh)`
   3.2 Choose geometric part and mesh models
   3.3 Set default mesh parameters (set a sensible value for `base size`)
   3.4 Set any mesh custom controls (`Automated Mesh > Custom Controls`)
   3.5 Generate volume mesh (button on toolbar)

4. **Define fluid continuum and model equations**
   4.1 <u>If</u> it doesn't exist, create it (`Continua` → `New Physics Continuum`)
   4.2 Choose model equations (`Continua >` *physics* → `Select Models...`)

5. **Define boundary conditions** (in `Regions`)
   5.1 Set any inflow boundary properties
   5.2 Set any other non-default boundary properties

6. **Set up any reports, monitors, plots or scenes used to check progress of simulation**

7. **Set stopping criteria** (`Stopping Criteria` → `New Monitor Criterion...`)

8. **Run** (button on toolbar)

9. **Analysis and plotting** (reports/derived parts/scenes/plots …)

1.2 Choice of Mesh Models

For this coursework use the following mesh models:
- Surface Remesher
- Polyhedral Mesher
- Prism Layer Mesher

`Base size` and `prism layer` parameters take global values except where overridden using `custom controls`. The prism-layer option puts thin prismatic cells near `wall` boundaries, so boundary types must be correctly identified – at least as wall or non-wall – before meshing.

1.3 Choice of Model Equations

For this coursework uncheck the "`Auto-select`" box and use the following model physics:
- Three Dimensional
- Gas
- Segregated Flow
- Constant Density
- Steady
- Turbulent
  – Reynolds-Averaged Navier-Stokes (RANS)
  – K-Epsilon Turbulence / Standard K-Epsilon / High y^+ Wall Treatment

1.4 Convergence Criteria

For this coursework set stopping criteria to end the calculation automatically when normalised residuals for mean-flow variables (continuity and x, y, z momentum) are all below 10^{-4}. If necessary, either increase the default maximum number of iterations or disable that criterion.

1.5 Reports

These include forces, moments, coefficients, max and min values, fluxes, areas etc.
- **Create:** (`Reports` → `New >` *various*)
- **Set the parts** to which the report is to be applied;
- **Set any other report properties**
- **Run the report:** right-click and select `Run Report`. Result in the output window.
- Optionally, set monitors and plots from a right-click option on your report.

1.6 Export

Graphical Output
Export field plots ("`Scenes`") and line graphs ("`Plots`") to file by right-clicking on an unfilled area of the plot window (or on the name in the object tree) and choosing `Hardcopy` (or `Hardcopy to Clipboard`).

Tables
Tables (`Tools > Tables`) can be extracted and exported as comma-separated-variable (CSV) files, then imported into other software for plotting / analysis.

1.7 Field Functions

Field functions (`Automation > Field Functions`) use intrinsic or user-defined variables designated by one dollar sign for scalars (e.g. `${Pressure}`) or two dollar signs for vectors (e.g. `$${Velocity}` for the whole vector, or `$${Velocity}[2]` for the 2 component).

Field-function scripting is similar to C and related programming languages. Note in particular:

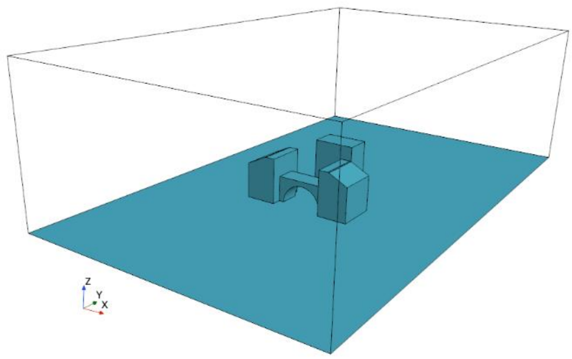
- arrays are **zero-indexed**; e.g. vector components are numbered 0, 1, 2, not 1, 2, 3;
- names are **case-sensitive**; e.g. `${Time}` and `${time}` are not the same;
- exponents are obtained by a `pow()` function and not by an operator.

Important variables which may enable you to code field functions for inflow profiles, etc., are:

- `$${Position}[0]`, `$${Position}[1]`, `$${Position}[2]` for x, y, z respectively.
- `${Time}` for time, t.

2. ASSIGNED EXERCISE: Wind Loading

2.1 Set-Up



Use the `STAR-CCM+` CAD module to create the building complex shown. (See the Appendix for dimensions and buildings A1, A2, B, C).

The fluid domain is generated by subtracting the buildings and tower from an outer cuboid of dimensions

$$-60 < x < 60\,, \quad -80 < y < 120\,, \quad 0 < z < 60$$

Dimensions are in metres. The origin and cardinal directions are defined in the Appendix. Positive x and positive y should correspond to east and north, respectively; **please adhere to this coordinate system.**

You will consider two velocity directions: wind from the south or north. Assign boundary types:

`Velocity Inlet`	on south or north as required;
`Pressure Outlet`	on the opposite side to the inlet;
`Symmetry plane`	on west, east and top;
`Wall`	on buildings and ground.

Create separate boundaries for each building A1, A2, B, C (see Appendix) and bottom boundary.

Generate a polyhedral mesh (base size 5 m) with prism sublayer (4 layers; total thickness 1 m) and surface growth rate 1.05. Apply custom controls to all buildings: surface size (target 10% of base, minimum 1% of base) and prism layer (total thickness 0.2 m).

Use the model equations described in Section 1.3 above and the stopping criteria described in Section 1.4.

If it continues to occur after initial transients, a run-time warning that turbulent eddy viscosity has been limited can be prevented by increasing the allowed ratio (by, say, a couple of orders of magnitude) at
`Solvers > K-Epsilon Turbulent Viscosity`

You will consider two velocity-inlet conditions – uniform velocity or log-law profile.

- For uniform velocity at inlet use a constant velocity

$$U = 20 \text{ m s}^{-1}.$$

- For a log-law velocity profile (defined using a `Field Function`) at inlet use:

$$U = \frac{u_\tau}{\kappa} \ln\left(\frac{z + z_0}{z_0}\right)$$

  where $\kappa = 0.41$, $z_0 = 0.3$ m and the friction velocity u_τ is chosen to make the wind speed equal to the reference (20 m s^{-1}) at the reference height of 10 m.

In both cases set turbulent intensity 0.05 and turbulent viscosity ratio 100 on the inlet boundary.

The pressure coefficient is the local non-dimensional pressure, defined as

$$c_p = \frac{p - p_{\text{ref}}}{\frac{1}{2}\rho U_0^2}$$

and is a built-in field function (`Automation > Field Functions`). Set reference density (as defined in the `Continua`) and reference velocity (as the inflow velocity) in its properties window. Leave p_{ref} as 0.

2.2 Items to Submit

Upload only the following two items (in PDF format) to Blackboard. You will be penalised if:

- you include padding, such as unnecessary plots, explanations or repetition of instructions;
- you use a screenshot where a simple text statement would do;
- you use non-PDF format or more separate files;
- your main submission exceeds 7 pages.

Item 1: Main Submission

(1) A clipped region from a screen shot showing the feature tree for the CAD modeller.

For UNIFORM velocity at inflow and flow FROM THE SOUTH:

(2) A graph of residuals history.

(3) One appropriate plot (use your judgement) for each of the following:

- geometry;
- mesh;
- velocity vectors on the horizontal plane $z = 2$ m;
- velocity vectors on the streamwise vertical plane $x = 0$ m;
- shaded plot of the pressure coefficient on the buildings and ground;
- streamlines (starting from an upstream horizontal line 2 m above ground).

(4) The streamwise forces on each of the individual building components A1, A2, B, C.

(5) The maximum pressure coefficient on building B (the bridge).

For UNIFORM velocity at inflow and flow FROM THE NORTH:

(6) Velocity vectors on the horizontal plane $z = 2$ m;

(7) The streamwise forces on each of the individual building components A1, A2, B, C.

For the LOG-LAW MEAN-VELOCITY PROFILE at inflow and flow FROM THE SOUTH:

(8) The field-function script (or scripts) that you used to define the magnitude of velocity.

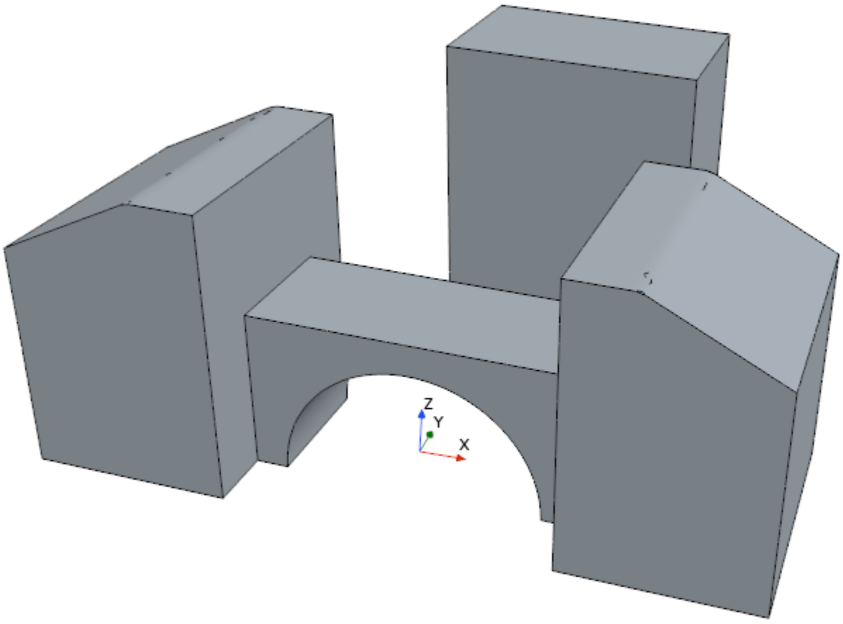
(9) Velocity vectors on the streamwise vertical plane $x = 0$ m.

(10) The streamwise forces on each of the individual building components A1, A2, B, C.

Item 2: StarCCM+ Summary Report

The summary report for the calculation (`[File]` ↓ `Summary Report > To File…`) should be created *after* doing all of items (1-10), so that all your working – and, in particular, the non-uniform inflow profile – is included. Make sure that the file submitted to Blackboard is in PDF format.

APPENDIX: GEOMETRY



All dimensions are in <u>metres</u>. Buildings A1 and A2 are mirror images in the plane $x = 0$. *The complex A1-B-A2 is symmetric about the plane* $y = 0$.