E3: Flow around a cubical building

Institute of Meteorology and Climatology, Leibniz Universität Hannover

Description and requirements

Description:
The aim of this exercise is to simulate the flow around a single cube using

1. cyclic,
2. non-cyclic boundary conditions.

You will learn how to set up topography and non-cyclic boundary conditions using a turbulent inflow condition.

Required lectures:

- Topography
- Non-cyclic boundary conditions
- Restart runs

Required files:

- Parameter file
- Topography file

Input files can be downloaded from the supplementary material.
Within the parameter files, some options are left blank (marked as “???”) and need to be filled in before starting the simulation.

Part 1 – Flow around a cube with cyclic boundary conditions

Simulate the flow around a single cube using the standard cyclic boundary conditions and the following setup:

- Domain size: $(80m)^3$
- Grid size: 2m, equidistant
- Single cube (edge length = 40m) located in the domain center
- Neutral boundary layer in open channel
- Initial velocity: $u = 1\ m\ s^{-1}$, $v = 0\ m\ s^{-1}$
- External pressure gradient of $-2*10^{-4}$ Pa m^{-1} in x direction
- No Coriolis force
- Simulation time: 1h

Part 1 – Questions to be answered

Try to answer the following questions by using appropriate model output

- Which significant flow patterns occur in the simulation?
  - Have a look at averaged horizontal and vertical slices of u, v, and w
- How do you explain the shape of the horizontally and time-averaged vertical profiles of velocity and momentum flux?
- Is the simulation a fully developed large-eddy simulation?
  - Have a look at the ratio of resolved and subgrid-scale fluxes
  - How do the time series change over time?

Part 1 – Proceeding

Please proceed as follows:

- Create a new job directory containing the given
  - namelist file: `flow_around_cube_cyclic_p3d`
  - topography file: `flow_around_cube_cyclic_topo` (alternatively use provided python script to create static input file `flow_around_cube_cyclic_static`)
- Start the simulation using command

```
palmrun -r flow_around_cube_cyclic -a "d3#" -X4
```

- Check the output and try to answer the above-mentioned questions
- Proceed with part 2

Part 2 – Noncyclic boundary conditions

Simulate the flow around a single cube using non-cyclic boundary conditions and the following setup:

- Domain size: 160m * 80m * 80m (x * y * z)
- Grid size: 2m, equidistant
- Single cube (edge length = 40m) centered at (x, y) = (120m, 40m)
- Neutral boundary layer in open channel
- Non-cyclic boundary condition along streamwise direction (turbulence-recycling method)
- Simulation time: 1h
- Initialize with precursor run (cyclic-fill method)
- Setup of precursor run:
  - Domain size: $(80m)^3$
  - Initial velocity: u = 1 m s^{-1}, v = 0 m s^{-1}
  - External pressure gradient of $-2*10^{-4}$ Pa m^{-1} in x direction
  - Simulation time: 1h

Part 2 – Questions to be answered

Try to answer the following questions by using appropriate model output

- Does the precursor run yield fully developed turbulence for the main run?
  - Have a look at the time series data and vertical profiles.
- Can you identify any differences in the flow field between the simulations of part 1 and part 2?
    - Compare the output of part 2 with those of part 1
- Is the simulation size large enough to simulate the flow around the building?
  - Does the influence of the building reach the recycling area?
  - Are all requirements met for the outflow boundary condition?

Part 2 – Proceeding

Please proceed as follows:

- Prepare the precursor run by creating a new job directory with the given namelist file: `flow_around_cube_noncyclic`

Start the precursor run using command
```
palmrun -r flow_around_cube_noncyclic -a "pcr restart" -X4
```

- Check the output and try to answer the first question.
- Copy the given
  - namelist file: `flow_around_cube_noncyclic_p3d`
  - topography file: `flow_around_cube_noncyclic_topo` (alternatively create static input file `flow_around_cube_noncyclic_static`)

  into the job directory.
- Start the main run using command
  ```
  palmrun -r flow_around_cube_noncyclic -a "d3# cyclic" -X4
  ```
- Check the output and try to answer the above-mentioned questions

Plot PALM output via palmplot

The following commands will help you to plot the simulation output using `palmplot`:

- Time series:

```
palmplot ts file_1=flow_around_cube_cyclic_ts.000.nc format_out=pdf \
  file_out=flow_around_cube_cyclic_plot_ts \
  var='E E* umax vmax wmax' no_rows=5
```

- Profiles:

```
palmplot pr file_1=flow_around_cube_cyclic_pr.000.nc format_out=pdf \
  file_out=flow_around_cube_cyclic_plot_pr
```

- xy-cross sections:

```
palmplot xy file_1=flow_around_cube_cyclic_av_3d.000.nc format_out=pdf \
  file_out=flow_around_cube_cyclic_plot_xy var='u v w' \
  no_rows=3 no_columns=2 sort=time mode='Both' zs=10 ze=10
```

- xz-cross sections:

```
palmplot xz file_1=flow_around_cube_cyclic_av_3d.000.nc format_out=pdf \
  file_out=flow_around_cube_cyclic_plot_xz var='u v w' \
  no_rows=3 no_columns=2 sort=time mode='Both' ys=40 ye=40
```

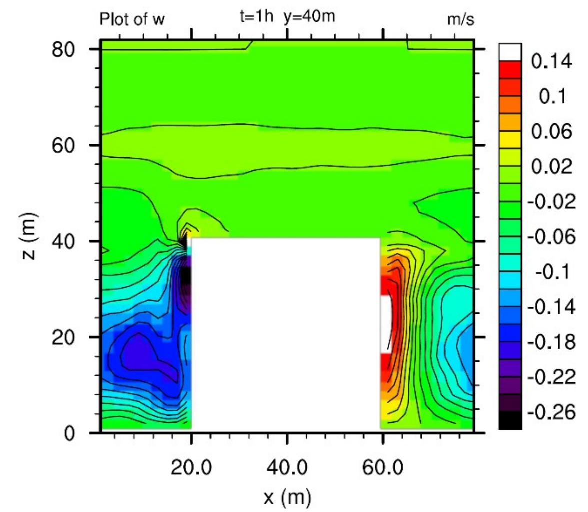
For more information about plotting with palmplot, please visit:
https://palm.muk.uni-hannover.de/trac/wiki/doc/app/ncl

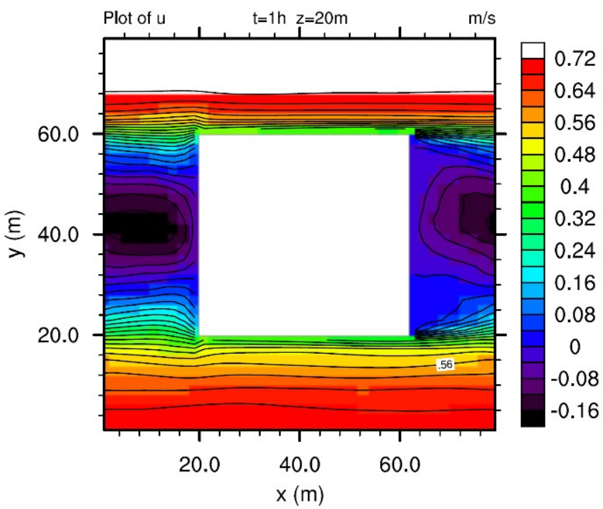
E3: CUBE/NONCYC

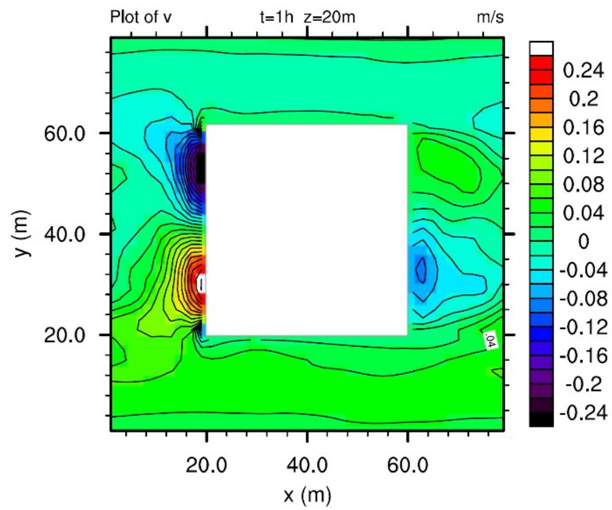
Answers to the questions

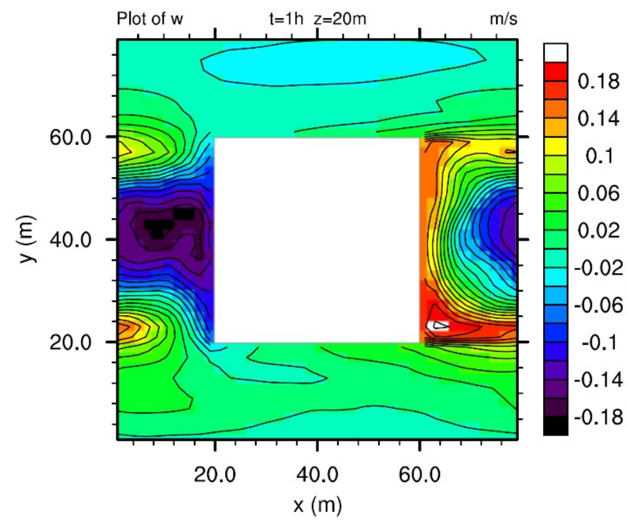
Part 1: Which significant flow patterns occur in the simulation?

- Reversed streamwise flow in the gap between leeward and windward cube wall,
- Diverging spanwise flow in the gap with nearly same magnitude as reversed spanwise flow.
- Descending mean flow near the windward cube wall,
- Ascending mean flow near the leeward cube wall.





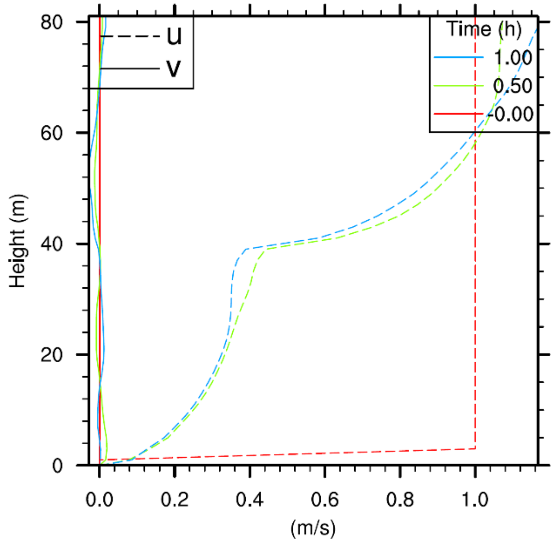


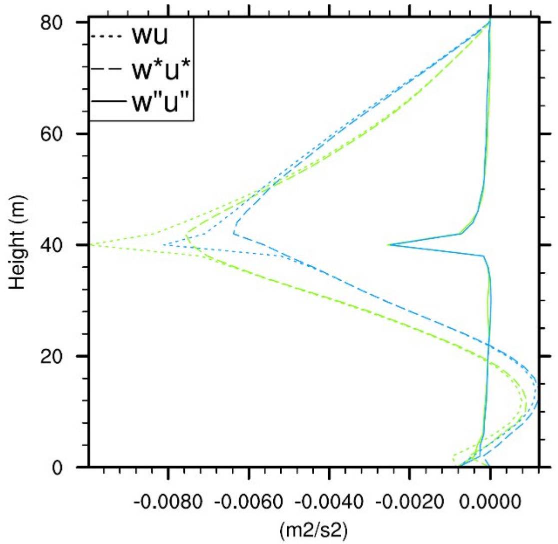


Answers to the questions

Part 1: How do you explain the shape of the horizontally and time-averaged vertical profiles of velocity and momentum flux?

- u increases according to logarithmic law; within canopy layer, higher roughness (due to cube) limits increase of u
- v is much smaller than u and fluctuates around zero. Time average should be increased to further eliminate these fluctuations.
- Flow is forced by u-component, and cube does not alter the mean wind direction
- wu profile can be split into three regions:
  - z=40m to 80m: linear increase with height, i.e., flow is decelerated; wu is negative, i.e., roughness of cube top causes deceleration.
  - z=15m to 40m: decreasing with height, i.e., flow is accelerated! **But no evidence for this in u-profile! caution when interpret domain-averaged profiles within the canopy layer!**
  - z=0 to 15m: increasing with height, i.e., flow decelerates due to surface friction.

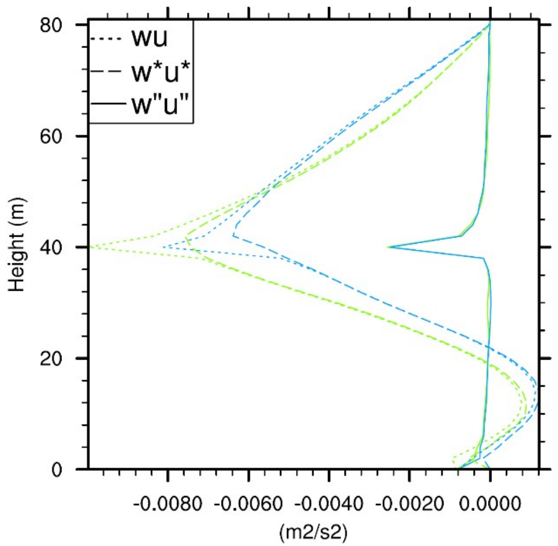




Answers to the questions

Part 1: Is the simulation a fully developed large-eddy simulation?

- Except near the surface and at the domain top, subgrid-scale momentum flux w"u" is one order of magnitude smaller than the resolved-scale counterpart w*u*, hence we can conclude, that the grid spacing is sufficiently small in order to resolve the energy-containing eddies within the simulation.

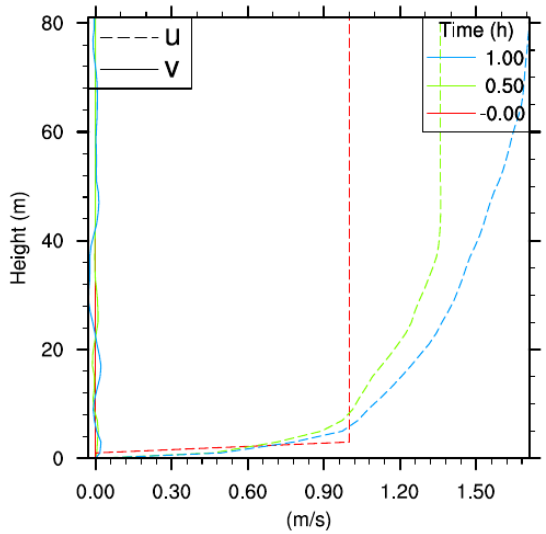


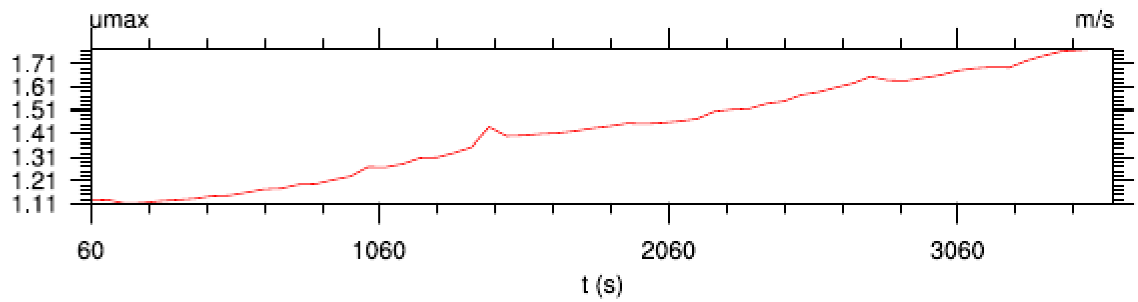
- Profiles of u and momentum flux still change in time. Hence, one hour might not be sufficient for the spin up of the model.

Answers to the questions

Part 2: Does the precursor run yield fully developed turbulence for the main run?

- Horizontally averaged profiles of u still change between t=0.5h and t=1h.
- Time series of umax increases during the whole simulation.
- The mean flow as well as the turbulent fluctuations seem to still develop as indicated by the u and umax, respectively. For better results, the simulation should be simulated for some additional time.

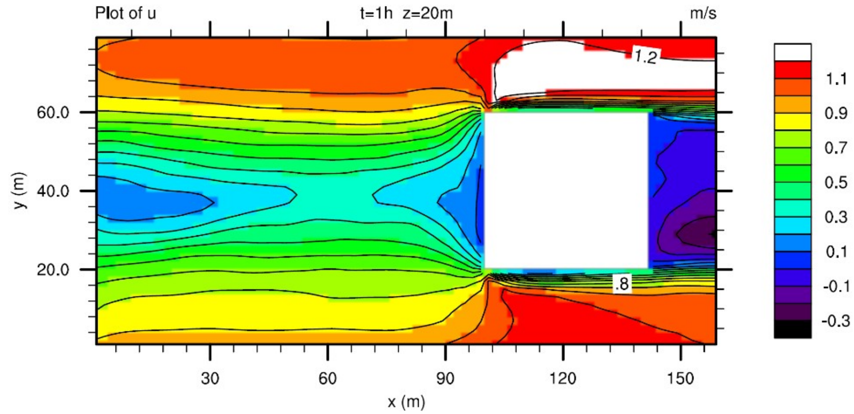


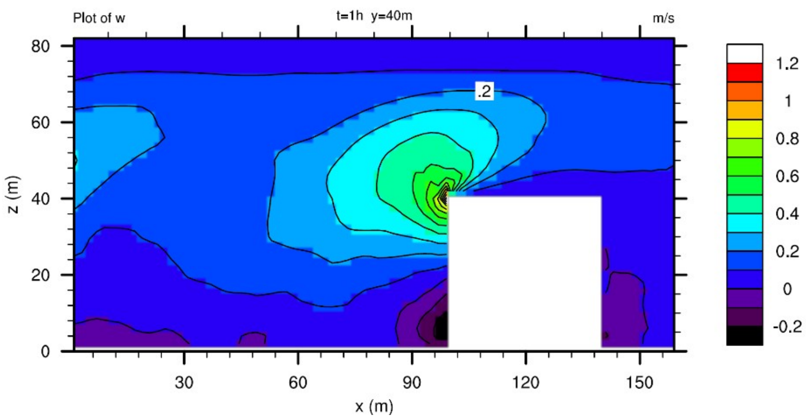


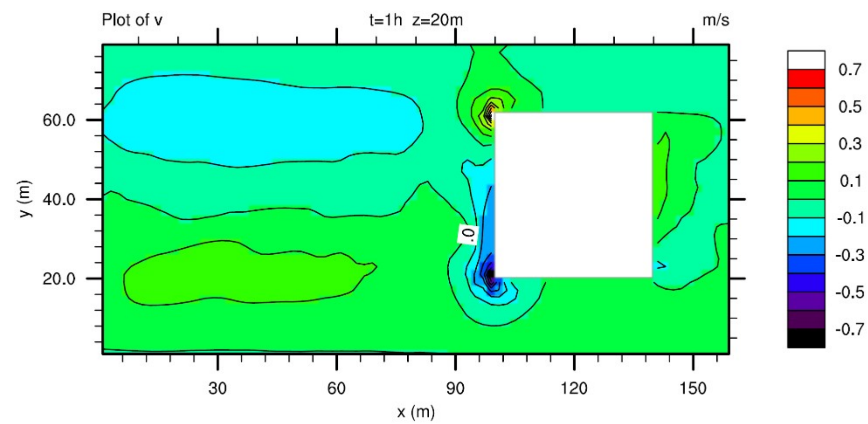
Answers to the questions

Part 2: Can you identify any differences in the flow field between the simulations of both parts?

- Accelerated flow to the northern and southern walls, strongly decelerated flow at the leeward wall
- Diverging spanwise flow with maxima and minima of v attached to the windward building edges
- Upward flow at upper windward cube edge, downward flow close to the ground at windward wall
- Maximum/minimum values of v are further moved to building edges than in the cyclic simulation
- No upward movement at the leeward building wall



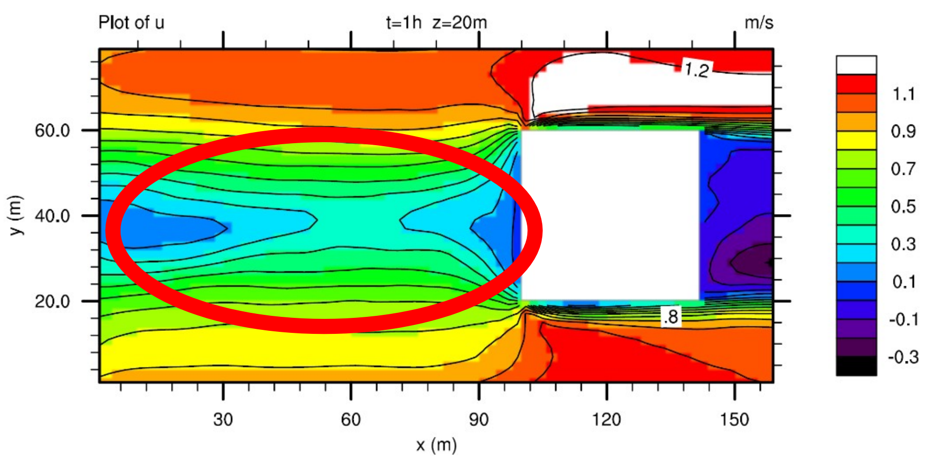


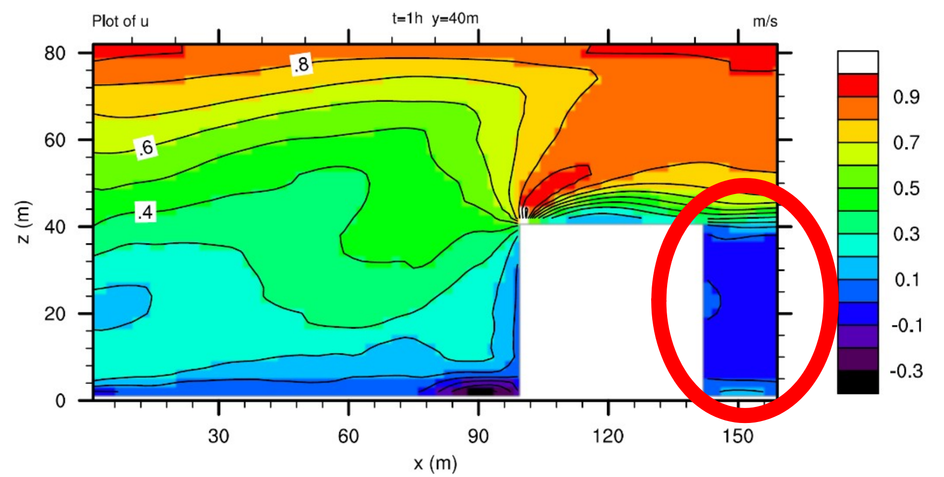


Answers to the questions

Part 2: Is the simulation size large enough to simulate the flow around the building?

- The xy-cross section of u shows reduced values along y=40m in the whole domain in front of the building. This is an indication that the influence of the blocking effect of the building reaches up to the recycling plane and is then recycled to the inflow boundary. The recycling area should therefore be further away from the building.
- The radiation boundary condition requires u>0 at the outflow boundary at every grid point at all time. This requirement is not met and can cause numerical errors influencing the results of the simulation. Therefore, the simulation requires a larger buffer zone behind the building.





The end

PALM online:

https://palm.muk.uni-hannover.de

Our YouTube channel:

youtube.com/user/palmhannover