Exercise 1: Convection Between Plates

PALM group

Institute of Meteorology and Climatology, Leibniz Universität Hannover

last update: 21st September 2015

Exercise 1: Convection Between Plates

Please try to carry out a run with following initial and boundary conditions and create the required output.

Exercise 1: Convection Between Plates

Please try to carry out a run with following initial and boundary conditions and create the required output.

- The simulation should represent a stationary convective boundary layer between two uniformly heated/cooled plates with zero mean flow.

Exercise 1: Convection Between Plates

Please try to carry out a run with following initial and boundary conditions and create the required output.

- The simulation should represent a stationary convective boundary layer between two uniformly heated/cooled plates with zero mean flow.
- A free-slip condition for velocity shall be used at the bottom and top boundary.

Exercise 1: Convection Between Plates

Please try to carry out a run with following initial and boundary conditions and create the required output.

- The simulation should represent a stationary convective boundary layer between two uniformly heated/cooled plates with zero mean flow.
- A free-slip condition for velocity shall be used at the bottom and top boundary.
- The sensible heat flux at the bottom and top boundary shall be constant throughout the simulation.

Exercise 1: Convection Between Plates

Please try to carry out a run with following initial and boundary conditions and create the required output.

- The simulation should represent a stationary convective boundary layer between two uniformly heated/cooled plates with zero mean flow.
- A free-slip condition for velocity shall be used at the bottom and top boundary.
- The sensible heat flux at the bottom and top boundary shall be constant throughout the simulation.

Simulation features:

Exercise 1: Convection Between Plates

Please try to carry out a run with following initial and boundary conditions and create the required output.

- The simulation should represent a stationary convective boundary layer between two uniformly heated/cooled plates with zero mean flow.
- A free-slip condition for velocity shall be used at the bottom and top boundary.
- The sensible heat flux at the bottom and top boundary shall be constant throughout the simulation.

Simulation features:

- domain size: about $2000 \times 2000 \times 1000\,\mathrm{m}^3$ ($x/y/z$)

Exercise 1: Convection Between Plates

Please try to carry out a run with following initial and boundary conditions and create the required output.

- The simulation should represent a stationary convective boundary layer between two uniformly heated/cooled plates with zero mean flow.
- A free-slip condition for velocity shall be used at the bottom and top boundary.
- The sensible heat flux at the bottom and top boundary shall be constant throughout the simulation.

Simulation features:

- domain size: about $2000 \times 2000 \times 1000\,\mathrm{m}^3$ $(x/y/z)$
- grid size: $50\,\mathrm{m}$ equidistant

Exercise 1: Convection Between Plates

Please try to carry out a run with following initial and boundary conditions and create the required output.

- The simulation should represent a stationary convective boundary layer between two uniformly heated/cooled plates with zero mean flow.
- A free-slip condition for velocity shall be used at the bottom and top boundary.
- The sensible heat flux at the bottom and top boundary shall be constant throughout the simulation.

Simulation features:

- domain size: about $2000 \times 2000 \times 1000\,\mathrm{m}^3$ $(x/y/z)$
- grid size: $50\,\mathrm{m}$ equidistant
- simulated time: $3600\,\mathrm{s}$

Exercise 1: Convection Between Plates

Please try to carry out a run with following initial and boundary conditions and create the required output.

- The simulation should represent a stationary convective boundary layer between two uniformly heated/cooled plates with zero mean flow.
- A free-slip condition for velocity shall be used at the bottom and top boundary.
- The sensible heat flux at the bottom and top boundary shall be constant throughout the simulation.

Simulation features:

- domain size: about $2000 \times 2000 \times 1000\,\mathrm{m}^3$ $(x/y/z)$
- grid size: $50\,\mathrm{m}$ equidistant
- simulated time: $3600\,\mathrm{s}$
- surface heatflux: $0.1\,\mathrm{K\ m\ s}^{-1}$

Exercise 1: Convection Between Plates

Please try to carry out a run with following initial and boundary conditions and create the required output.

- The simulation should represent a stationary convective boundary layer between two uniformly heated/cooled plates with zero mean flow.
- A free-slip condition for velocity shall be used at the bottom and top boundary.
- The sensible heat flux at the bottom and top boundary shall be constant throughout the simulation.

Simulation features:

- domain size: about $2000 \times 2000 \times 1000\,\mathrm{m}^3$ $(x/y/z)$
- grid size: $50\,\mathrm{m}$ equidistant
- simulated time: $3600\,\mathrm{s}$
- surface heatflux: $0.1\,\mathrm{K\ m\ s}^{-1}$
- heatflux at top: $0.1\,\mathrm{K\ m\ s}^{-1}$

Exercise 1: Convection Between Plates

Please try to carry out a run with following initial and boundary conditions and create the required output.

- The simulation should represent a stationary convective boundary layer between two uniformly heated/cooled plates with zero mean flow.
- A free-slip condition for velocity shall be used at the bottom and top boundary.
- The sensible heat flux at the bottom and top boundary shall be constant throughout the simulation.

Simulation features:

- domain size: about $2000 \times 2000 \times 1000\,\mathrm{m}^3$ $(x/y/z)$
- grid size: $50\,\mathrm{m}$ equidistant
- simulated time: $3600\,\mathrm{s}$
- surface heatflux: $0.1\,\mathrm{K\ m\ s}^{-1}$
- heatflux at top: $0.1\,\mathrm{K\ m\ s}^{-1}$
- initial temperature: $300\,\mathrm{K}$ everywhere

Exercise 1: Convection Between Plates

Please try to carry out a run with following initial and boundary conditions and create the required output.

- The simulation should represent a stationary convective boundary layer between two uniformly heated/cooled plates with zero mean flow.
- A free-slip condition for velocity shall be used at the bottom and top boundary.
- The sensible heat flux at the bottom and top boundary shall be constant throughout the simulation.

Simulation features:

- domain size: about $2000 \times 2000 \times 1000\,\mathrm{m}^3$ $(x/y/z)$
- grid size: $50\,\mathrm{m}$ equidistant
- simulated time: $3600\,\mathrm{s}$
- surface heatflux: $0.1\,\mathrm{K\ m\ s}^{-1}$
- heatflux at top: $0.1\,\mathrm{K\ m\ s}^{-1}$
- initial temperature: $300\,\mathrm{K}$ everywhere
- initial velocity: zero everywhere

Questions to be Answered:

- How does the flow field look like after 60 minutes of simulated time? (What kind of output do you need to answer this?)

Questions to be Answered:

- How does the flow field look like after 60 minutes of simulated time? (What kind of output do you need to answer this?)
- How do the horizontally and temporally averaged vertical temperature and heat flux profiles look like?

Questions to be Answered:

- How does the flow field look like after 60 minutes of simulated time? (What kind of output do you need to answer this?)
- How do the horizontally and temporally averaged vertical temperature and heat flux profiles look like?
- Is it really a large-eddy simulation, i.e., are the subgrid-scale fluxes much smaller than the resolved-scale fluxes? (How long should the averaging time interval be?)

Questions to be Answered:

- How does the flow field look like after 60 minutes of simulated time? (What kind of output do you need to answer this?)
- How do the horizontally and temporally averaged vertical temperature and heat flux profiles look like?
- Is it really a large-eddy simulation, i.e., are the subgrid-scale fluxes much smaller than the resolved-scale fluxes? (How long should the averaging time interval be?)
- How do the total kinetic energy and the maximum velocity components change in time? Has the flow become stationary?

Questions to be Answered:

- How does the flow field look like after 60 minutes of simulated time? (What kind of output do you need to answer this?)
- How do the horizontally and temporally averaged vertical temperature and heat flux profiles look like?
- Is it really a large-eddy simulation, i.e., are the subgrid-scale fluxes much smaller than the resolved-scale fluxes? (How long should the averaging time interval be?)
- How do the total kinetic energy and the maximum velocity components change in time? Has the flow become stationary?
- Has the domain size and grid size been chosen appropriately?

Hints (I)

PALM parameter names are displayed by courier style, e.g. `end_time`.

Hints (I)

PALM parameter names are displayed by courier style, e.g. `end_time`.

- Domain size
  - Is controlled by grid size (`dx`, `dy`, `dz`) and number of grid points (`nx`, `ny`, `nz`). Since the first grid point along each of the directions has index 0, the total number of grid points used are `nx`+1, `ny`+1, `nz`+1. The total domain size in case of cyclic horizontal boundary conditions is (`nx`+1)*`dx`, (`ny`+1)*`dy`.

Hints (I)

PALM parameter names are displayed by courier style, e.g. `end_time`.

- Domain size
  - Is controlled by grid size (`dx`, `dy`, `dz`) and number of grid points (`nx`, `ny`, `nz`). Since the first grid point along each of the directions has index 0, the total number of grid points used are `nx`+1, `ny`+1, `nz`+1. The total domain size in case of cyclic horizontal boundary conditions is (`nx`+1)*`dx`, (`ny`+1)*`dy`.
- Initial profiles
  - Constant with height. See parameter `initializing_actions` for available initialization methods. See `ug_surface`, `vg_surface` and `pt_surface` for initial values of velocity and potential temperature.

Hints (I)

PALM parameter names are displayed by courier style, e.g. `end_time`.

- Domain size
  - Is controlled by grid size (`dx`, `dy`, `dz`) and number of grid points (`nx`, `ny`, `nz`). Since the first grid point along each of the directions has index 0, the total number of grid points used are `nx`+1, `ny`+1, `nz`+1. The total domain size in case of cyclic horizontal boundary conditions is (`nx`+1)*`dx`, (`ny`+1)*`dy`.
- Initial profiles
  - Constant with height. See parameter `initializing_actions` for available initialization methods. See `ug_surface`, `vg_surface` and `pt_surface` for initial values of velocity and potential temperature.
- Boundary conditions
  - For velocity, see `bc_uv_b` and `bc_uv_t`. See also `prandtl_layer`, because Neumann conditions don't allow to use a Prandtl-layer.

Hints (I)

PALM parameter names are displayed by courier style, e.g. `end_time`.

- Domain size
  - Is controlled by grid size (`dx`, `dy`, `dz`) and number of grid points (`nx`, `ny`, `nz`). Since the first grid point along each of the directions has index 0, the total number of grid points used are `nx`+1, `ny`+1, `nz`+1. The total domain size in case of cyclic horizontal boundary conditions is (`nx`+1)*`dx`, (`ny`+1)*`dy`.
- Initial profiles
  - Constant with height. See parameter `initializing_actions` for available initialization methods. See `ug_surface`, `vg_surface` and `pt_surface` for initial values of velocity and potential temperature.
- Boundary conditions
  - For velocity, see `bc_uv_b` and `bc_uv_t`. See also `prandtl_layer`, because Neumann conditions don't allow to use a Prandtl-layer.
  - For temperature / heat flux, see `surface_heatflux` and `top_heatflux`. Prescribing of heat flux at the boundary requires a Neumann boundary condition for temperature, see `bc_pt_b` and `bc_pt_t`.

Hints (I)

PALM parameter names are displayed by courier style, e.g. `end_time`.

- Domain size
  - Is controlled by grid size (`dx`, `dy`, `dz`) and number of grid points (`nx`, `ny`, `nz`). Since the first grid point along each of the directions has index 0, the total number of grid points used are `nx`+1, `ny`+1, `nz`+1. The total domain size in case of cyclic horizontal boundary conditions is (`nx`+1)*`dx`, (`ny`+1)*`dy`.
- Initial profiles
  - Constant with height. See parameter `initializing_actions` for available initialization methods. See `ug_surface`, `vg_surface` and `pt_surface` for initial values of velocity and potential temperature.
- Boundary conditions
  - For velocity, see `bc_uv_b` and `bc_uv_t`. See also `prandtl_layer`, because Neumann conditions don't allow to use a Prandtl-layer.
  - For temperature / heat flux, see `surface_heatflux` and `top_heatflux`. Prescribing of heat flux at the boundary requires a Neumann boundary condition for temperature, see `bc_pt_b` and `bc_pt_t`.
  - Use a Neumann condition also for the perturbation pressure both at the bottom and the top (`bc_p_b`, `bc_p_t`).

Hints (I)

PALM parameter names are displayed by courier style, e.g. `end_time`.

- Domain size
  - Is controlled by grid size (`dx`, `dy`, `dz`) and number of grid points (`nx`, `ny`, `nz`). Since the first grid point along each of the directions has index 0, the total number of grid points used are `nx`+1, `ny`+1, `nz`+1. The total domain size in case of cyclic horizontal boundary conditions is (`nx`+1)*`dx`, (`ny`+1)*`dy`.
- Initial profiles
  - Constant with height. See parameter `initializing_actions` for available initialization methods. See `ug_surface`, `vg_surface` and `pt_surface` for initial values of velocity and potential temperature.
- Boundary conditions
  - For velocity, see `bc_uv_b` and `bc_uv_t`. See also `prandtl_layer`, because Neumann conditions don't allow to use a Prandtl-layer.
  - For temperature / heat flux, see `surface_heatflux` and `top_heatflux`. Prescribing of heat flux at the boundary requires a Neumann boundary condition for temperature, see `bc_pt_b` and `bc_pt_t`.
  - Use a Neumann condition also for the perturbation pressure both at the bottom and the top (`bc_p_b`, `bc_p_t`).
- Simulation time: See parameter `end_time`

Hints (II)

Hints for data output.

Hints (II)

Hints for data output.

- Variables
  - Output variables are chosen with parameters `data_output` (3d-data or 2d-cross-sections) and `data_output_pr` (profiles).

Hints (II)

Hints for data output.

- Variables
  - Output variables are chosen with parameters `data_output` (3d-data or 2d-cross-sections) and `data_output_pr` (profiles).
- Output intervals
  - Output intervals are set with parameter `dt_data_output`. This parameter affects all output (cross-sections, profiles, etc.). Individual temporal intervals for the different output quantities can be assigned using parameters `dt_do3d`, `dt_do2d_xy`, `dt_do2d_xz`, `dt_do2d_yz`, `dt_dopr`, etc.

Hints (II)

Hints for data output.

- Variables
  - Output variables are chosen with parameters data_output (3d-data or 2d-cross-sections) and data_output_pr (profiles).
- Output intervals
  - Output intervals are set with parameter dt_data_output. This parameter affects all output (cross-sections, profiles, etc.). Individual temporal intervals for the different output quantities can be assigned using parameters dt_do3d, dt_do2d_xy, dt_do2d_xz, dt_do2d_yz, dt_dopr, etc.
- Time averaging
  - Time averaging is controlled with parameters averaging_interval, averaging_interval_pr, dt_averaging_input, dt_averaging_input_pr.

Further Hints

Further Hints

You will find some more detailed information to solve this exercise in the PALM-online-documentation under:

`http://palm.muk.uni-hannover.de/trac/wiki/doc/app/examples/cbl`

(Attention: This documentation is for atmospheric convection with free upper lid.)

Further Hints

You will find some more detailed information to solve this exercise in the PALM-online-documentation under:

```
http:
//palm.muk.uni-hannover.de/trac/wiki/doc/app/examples/cbl
```

(Attention: This documentation is for atmospheric convection with free upper lid.)

Please also visit

```
http://palm.muk.uni-hannover.de/trac/wiki/doc/app/netcdf
```

where the complete PALM netCDF-data-output and the respective steering parameters are described.

How to Start?

How to Start?

- Create a data directory for a new run:

```
cd ~/palm/current_version
mkdir -p JOBS/uniform_plates/INPUT
```

How to Start?

- Create a data directory for a new run:

```
cd ~/palm/current_version
mkdir -p JOBS/uniform_plates/INPUT
```

- Create the parameter file and set the required parameters in

```
JOBS/uniform_plates/INPUT/uniform_plates_p3d
```

How to Start?

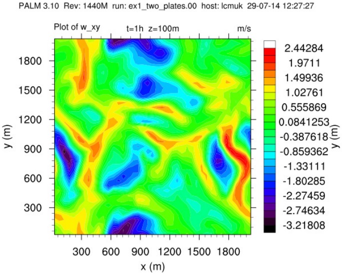
- Create a data directory for a new run:
```
cd ~/palm/current_version
mkdir -p JOBS/uniform_plates/INPUT
```
- Create the parameter file and set the required parameters in
```
JOBS/uniform_plates/INPUT/uniform_plates_p3d
```
- Start the run with `mrun`-command
```
mrun -d uniform_plates -h <hi> -K parallel ...
```
and analyze the output files.

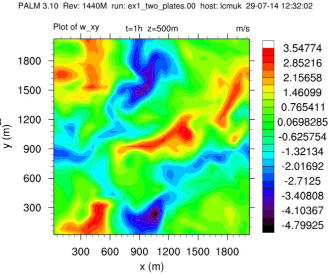
How to Start?

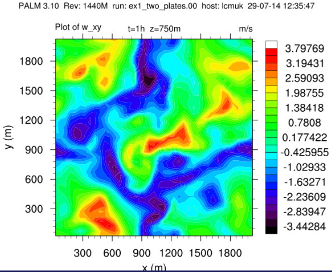
- Create a data directory for a new run:
  ```
  cd ~/palm/current_version
  mkdir -p JOBS/uniform_plates/INPUT
  ```
- Create the parameter file and set the required parameters in
  `JOBS/uniform_plates/INPUT/uniform_plates_p3d`
- Start the run with `mrun`-command
  ```
  mrun -d uniform_plates -h <hi> -K parallel ...
  ```
  and analyze the output files.

Good Luck!

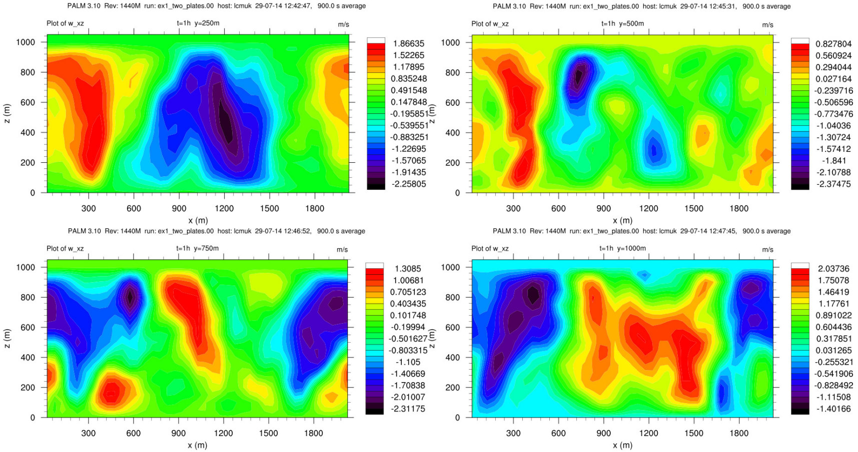
xy-cross sections (instantaneous at $t = 3600\,\mathrm{s}$)





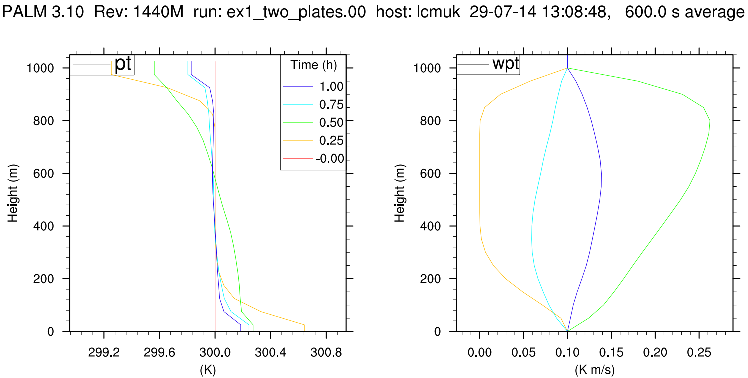


xz-cross sections (900 s average)



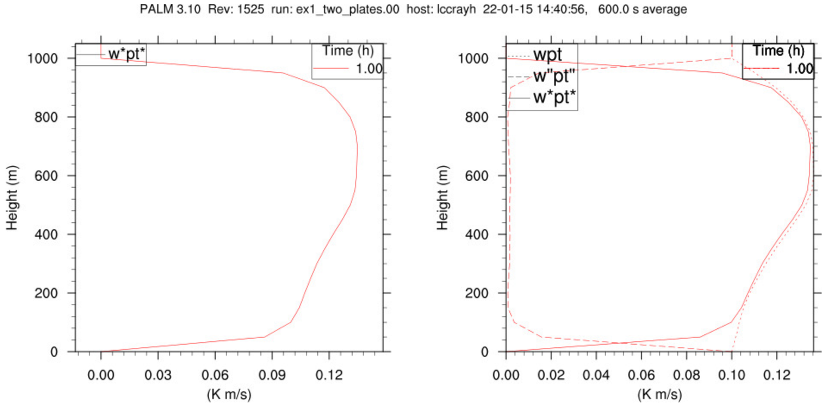
Leibniz Universität Hannover

Vertical profiles

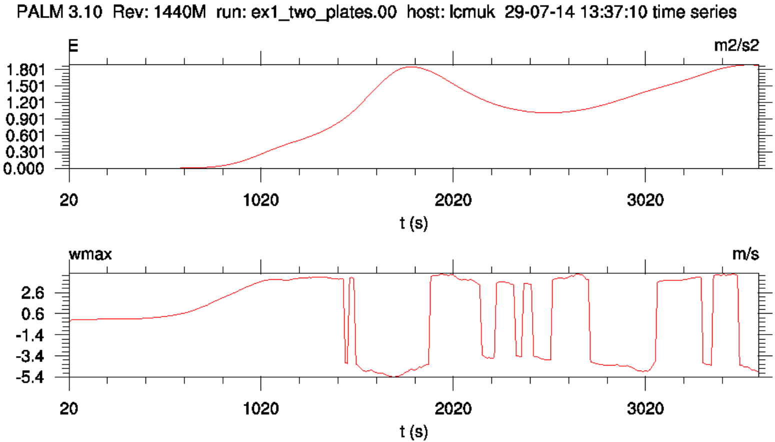


Results

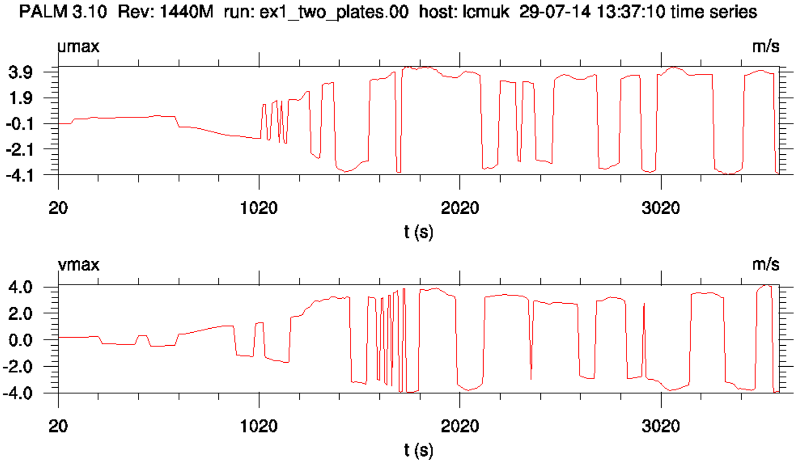
LES?


PALM 3.10 Rev: 1525 run: ex1_two_plates.00 host: lccrayh 22-01-15 14:40:56, 600.0 s average
w*pt*
Time (h)
1.00
Height (m)
1000
800
600
400
200
0
0.00
0.03
0.06
0.09
0.12
(K m/s)
wpt
w"pt"
w*pt*
Time (h)
1.00
Height (m)
0.00
0.02
0.04
0.06
0.08
0.10
0.12
(K m/s)

Time series (I)



Time series (II)



Answers to question I

How does the flow field look like after 60 minutes of simulated time?

- Useful output: for example instantaneous or time-averaged cross-sections of vertical velocity (frames 8–9).
- Flow field shows narrower updrafts and broader downdrafts, cellular pattern close to the heated/cooled plates in xy-sections of vertical velocity.
- The temporal mean of vertical velocity exhibits a circulation spanning the whole depth of the model domain.

Answers to question II

How do the horizontally and temporally averaged vertical temperature and heat flux profiles look like?

- PALM standard profile output contains potential temperature and its vertical flux (shown in frame 10).
- Heating the lower plate and cooling the upper plate induces convection resulting in a well-mixed boundary layer where the potential temperature profile is constant with height. Temperature gradients remain at the domain boundaries since convective turbulence cannot remove them in the vicinity of the walls.
- In case of horizontal homogeneity, the temperature equation reduces to $\frac{\partial \theta}{\partial t} = -\frac{\partial \overline{w'\theta'}}{\partial z}$ in the present case. In a stationary state, it follows that $\frac{\partial \theta}{\partial t} = 0$. Thus, the flux profile $\overline{w'\theta'}$ has to be constant with height – as can be seen in frame 10.
- The total vertical heat flux is positive in the whole modeling domain indicating upward transport of warmer air parcels and downward transport of colder air parcels.

Answers to question III

Is it really a large-eddy simulation? Duration of averaging time?

- It is a large-eddy simulation because the sub-grid fluxes are negligibly small throughout the bulk of the mixed layer. There, the resolved flux is dominating the total flux indicating a well-resolved turbulent flow (frame 11). Sub-grid fluxes dominate close to the surface where the turbulent-eddies cannot be resolved.
- Typically, the averaging time should contain several large-eddy turnover times. The large-eddy turnover time can be defined as $\tau_\mathrm{l} = L/u$ where L is the length-scale of the largest eddies in the flow and u is their typical velocity scale. τ_l can be interpreted as a typical time a turbulent eddy needs to traverse the modeling domain. In our case, L is proportional to the domain height ($L \approx 1000\,\mathrm{m}$) and u is about $5\,\mathrm{ms}^{-1}$ (see time series of wmax on frame 12). Thus, $\tau_\mathrm{l} \approx 200\,\mathrm{s}$. An averaging time of 600 s chosen here is, thus, appropriate.

Answers to question IV

Has the flow become stationary?

- The time series of total kinetic energy E and the maximum velocities wmax, umax and vmax shown in frames 12-13 exhibit a spin-up phase of the model up to $t \approx 2000\,\mathrm{s}$. During this initialization time, turbulence is triggered by random perturbations until turbulence starts to develop.
- A stationary state can be seen by means of an (almost) non-changing E with time. Constant maxima of the velocity components also indicate a stationary flow.

Answers to question V

Has the domain size and grid size been chosen appropriately?

- A domain size is generally appropriately chosen in case that several of the dominating flow structures fit into the modeling domain. From the xy-cross sections in frame 8 it becomes apparent that the typical hexagonal flow structures close to the surface can hardly be seen. The xz-cross sections in frame 9 also contain only one circulation. Thus, the domain size in our example seems to be too small to capture several energy-containing flow structures.
- The grid size should be chosen in the way that the dominating flow structures can be represented by at least several grid points (4-5). A grid spacing of 50 m as chosen in this exercise is appropriate since the flow structures exhibit horizontal length scales of about 1 km (see frame 8).