1 Detailed instruction to operate SICT

This is a step-by-step instruction to the user with limited MATLAB and turbulence experience on how to operate SICT; A Simple User Interface of a Complex Turbulence Model. As you continue to learn about SICT the less you will depend on the instructions. The first time you use SICT, you should read all instructions carefully. The next time, you might only need to read the most important bullet points (written in bold letters).

2 Download and open SICT

- Go to https://github.com/josefinaalgotsson/SICT/releases and download the SICT package by clicking "Source code (zip)".

- Save the zipped folder on your computer and unzip the folder.

- Open the file runSICT.m in MATLAB and run the script by clicking "Run". This will open the SICT user interface figure. If you are prompted with the message "File .../runSICT.m is not found in the current folder or on the MATLAB search path.", choose to change the current folder by clicking "Change folder". The SICT user interface figure will now open.

- Take your time to look at figure 1 and get familiar with the different components of SICT. If you don't understand them completely, don't worry. Focus on getting familiar with the text for now.

- SICT opens with a default case ready to run. If you want, all you have to do is press the yellow button "Set" to implement the default settings (Described in the steps 1, 2 and 3 below) and then press "Run". Then, you've already been able to simulate a turbulence case with a k model. How cool!

- If you have tried out the default case and/or want to set up your own case, follow the steps below to set up a new case to run.

3 Set up and run a simulation

1. Set up domain, boundary conditions and initial conditions.

    - **Set up geometry of your domain** (red box, fig. 1) by changing the inputted values in boxes "Domain height [m]" and "Cell size [m]". When pressing the "Set" button in stage 2, this will affect the scale of the z-axis in your domain (dashed red ellipse, fig 1). If the domain size is not evenly divisible with the cell size, an error message will appear in the command window when pressing "Set".

- **Set boundary conditions for mean velocity, U** (blue box, fig. 1) by choosing either "Fixed flux" or "Fixed value" in both the "Surface:" and "Bottom:" popup menue. This will affect the near surface and near bottom value of the velocity (dashed blue circles, fig 1). As well as update the units of the boundary conditions (to the right in the solid blue box). For example, a fixed value in the surface represents a case where we know the velocity of the water in the surface, and that this velocity is constant. On the other hand if we for example want to simulate a certain stress in the surface, we choose the constant flux option and input u_*^2 which represent the momentum flux into the water at the surface. Note that the cases validated for SICT always have a fixed value of 0 m/s in the bottom.
- Make settings for stratification (orange boxes, fig. 1). These will have effect on the form of the stratification (dashed orange ellipse, fig 1).
  - **Choose whether a linear equation of state or the MATLAB-function sw_dens is used to calculate density** (top right orange box, fig. 1) by choosing one of two radiobuttons.
  - **Choose stratification form** (bottom orange box, fig. 1) by choosing either "Linear stratification" or "2-layer stratification" in the pop-up menue. Linear stratification will set up linear profiles of temperature and salinity between T1 and T2 and S1 and S2 respectively. 2-layer stratification creates two layers where the top and bottom layer have temperature and salinity T1 and S1 and T2 and S2 respectively.
  - **Make changes to the absolute stratification** (bottom orange box, fig. 1) by changing the values of the top and bottom temperature and salinity; T1, T2, S1 and S2. T1 and S1 represent the temperature and salinity in the surface and T2 and S2 are temperature and salinity in the bottom. **If the 2-layer stratification is chosen, change the top layer thickness** by editing the "Top layer thickness [m]:" box. A bug in SICT updates the calculated buoyancy frequency squared improperly immediately after switching back to linear stratification in the popup menue. It is updated correctly when pressing the "Set" button.
- **If desirable, make changes to constants** of the equation of state, change the roughness lengths at the surface or the bottom, or change the value of gravitational acceleration (green box, fig. 1) by clicking the "Advanced..." button to open an external window. Then click the "Save & Close" button. The button doesn't actually close the new figure, it only makes it invisible. Closing the figure by actually pressing the top corner will close it completely and make it impossible to open the Advanced figure again. Use the "Reboot" button and make your settings again.

2. **Apply your settings** by clicking the "Set"-button. This will plot terms in the

plotting windows (large dashed black box in fig 1). From left to right the terms plotted are; The mean velocity, U, the turbulent kinetic energy, k, the production, P_b of T.K.E through buoyancy, production P_s of turbulent kinetic energy through shear, dissipation $\mathcal{E}$ of turbulent kinetic energy , turbulent viscosity ν_t, shear stress τ, and density ρ of the fluid. SICT also calculates and displays the mean buoyancy frequency squared if the linear stratification is chosen (dashed black box inside bottom orange box). The Clock and Simulation time is also reset by pressing the "Set" button (dashed black box to the left). Some common errors at this stage are setting a cell size not equally divisible with the domain height. This makes it impossible for SICT to divide the domain into a set of equal size nodes. Another common error is forgetting to set the Top layer thickness correctly when switching between linear stratification and 2-layer stratification.

3. Make time stepping settings

   - **Set simulation time** (cyan box, fig. 2) by editing the "t_max [s]:" box.
   - **Set time step** (magenta box, fig. 2) by editing the "Delta t [s]:" box.
   - **Set save interval** (violet box, fig. 2) by editing the "Save output every " box. This setting applies from the so far simulated time to the end of the forthcoming simulation. That is, the save interval applies for the additional t_max time. It is possible to run the simulation a certain amount of time (t_max) further, then changing the time step and save interval to run the model an additional t_max time further and so on. It is also possible, but not recommended to change other settings of SICT during simulation.

4. **Run your simulation** by clicking the "Run" button. SICT will update the plotting windows when the inputted t_max time has been simulated. This time will be added to the clock. It is possible to continue the simulation by simply clicking the "Run" button again. SICT will continue the simulation with the additional inputted t_max time. This simulated time will also be added to the clock (Bottom right Tracker panel). In between clicking "Run" it is possible to change both t_max, delta t and save interval, enabling the user to adjust the temporal resolution in certain stages of the simulation. You have now simulated your own custom made turbulence case. Good job!

5. Set up output file and folder and save simulation output

   - **Choose filename** (teal box, fig. 2) by either editing the "Filename:" box or pressing the "Browse..." button to choose a pre-existing filename
   - **Choose output folder** (lime green box, fig. 2) by either editing the "Folder:" box or pressing the "Browse..." button to choose a folder. You must enter a path to an existing folder.
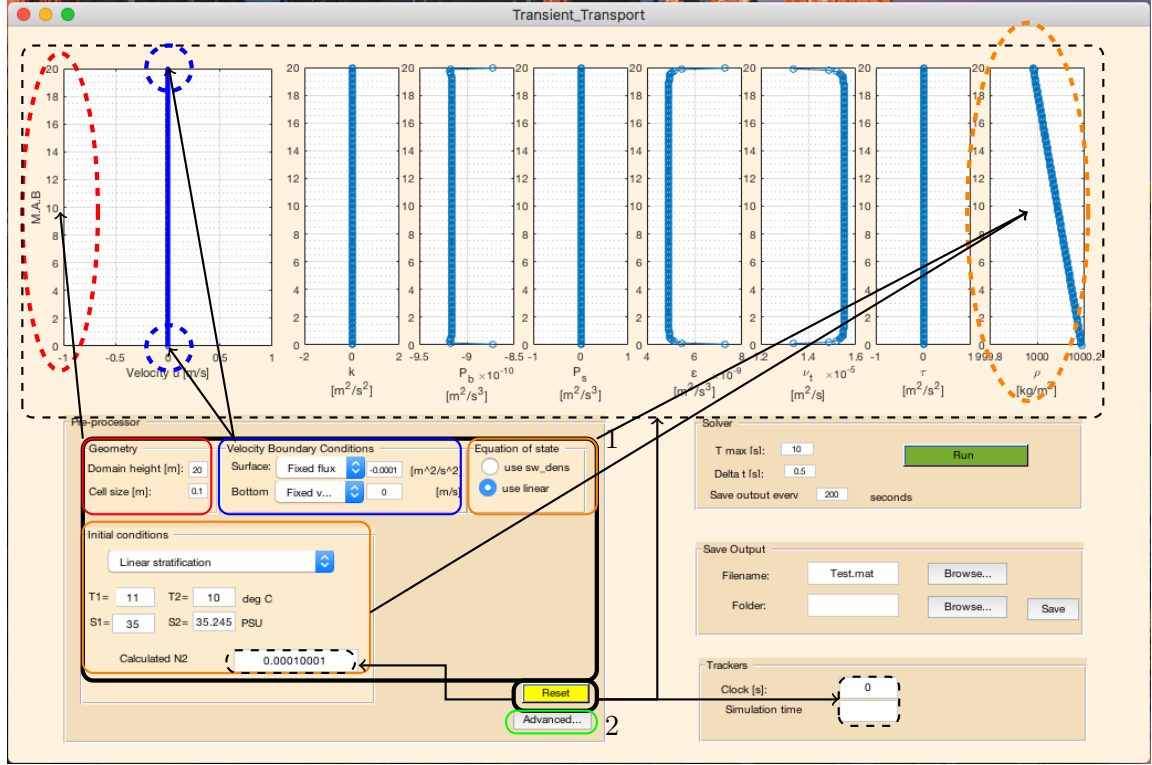
Figure 1: SICT in a stage where geometry of the domain and boundary conditions have been set. Solid black boxes show stages which the user should use in numerical order to manage SICT. Solid coloured boxes show settings modules which the user can modify to produce the requested scenario. Dashed boxes show affected areas changing in accordance with the boundary conditions and the geometry of the domain.

6. **Save output** by pressing the "Save" button. When the output is saved, a pop-up window appears confirming the file name and output folder.

3.1   SICT output

The outputted variables include time series for temperature, T, salinity, S and velocity, U as well as other time series, model constants and settings for the last simulation. The output also contains the boundary conditions and domain grid.The outputted variables saved when using the Save button are listed below. Matrices contain some variables at saved time steps (usave, Tsave, Ssave, tsave, ksave for example), where the values at one time step are inputted in a column, where the first row corresponds to the surface and the last row corresponds to the bottom of the domain. These columns are stacked side by side
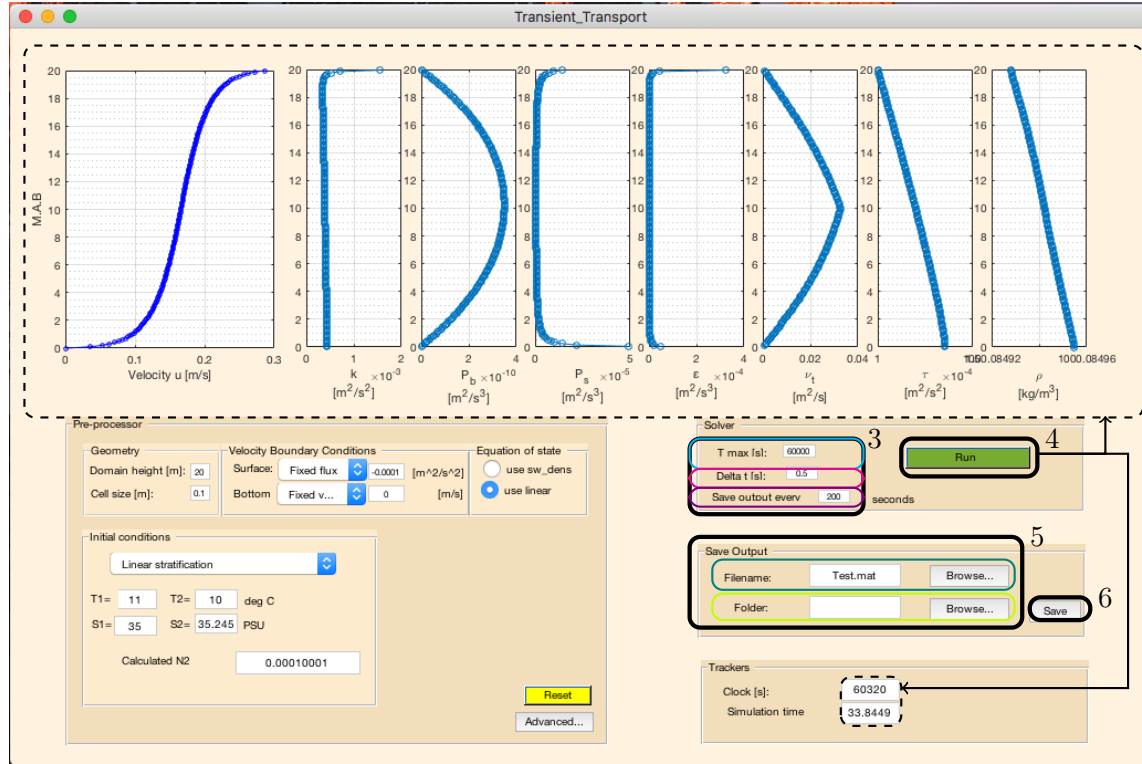
Figure 2: SICT in a stage where the model has been run for a certain amount of time. Solid coloured boxes show stages which the user should go through in chronological order to manage SICT. Dashed boxes show affected areas changing in accordance with pressing the Run button. Box 3 shows the time stepping settings module handling simulation time, size of the time step and the saving interval. Box 4 shows the button for running the simulation. Box 5 shows the settings module for outputting model data. Box 6 shows the button for saving output to the selected file and folder.

where every next column represents the next saved time step.

Table 1: Outputted MATLAB variables from SICT. Boundary conditions (bcNk, bcNS, bcNT...) are defined as a 1 if the boundary condition is Dirichlet and a 2 if the boundary condition is Neuman.

Variable name	
alfa	Thermal expansion coefficient
bcNk	Boundary condition for k at surface boundary
bcNS	Boundary condition for S at surface boundary
bcNT	Boundary condition for T at surface boundary
bcNu	Boundary condition for U at surface boundary
bcSk	Boundary condition for k at bottom boundary
bcSS	Boundary condition for S at bottom boundary
bcST	Boundary condition for T at bottom boundary
bcSu	Boundary condition for u at bottom boundary
beta	Saline contraction coefficient
boundValNk	Boundary condition value for k at surface boundary
boundValNS	Boundary condition value for S at bottom boundary
boundValNT	Boundary condition value for T at surface boundary
boundValNu	Boundary condition value for U at surface boundary
boundValSk	Boundary condition value for at bottom boundary
boundValSS	Boundary condition value for S at bottom boundary
boundValST	Boundary condition value for T at bottom boundary
boundValSu	Boundary condition value for U at bottom boundary
breakDep	depth at which density jump is situated in case of 2-layer stratification
cb	Model constant
Cmu0	Stability function
deltat	Size of time step
deltaz	Size of grid cells
eosChoice	Choice for equation of state
Epssave	Matrix containing dissipation at saved time steps, with depth in rows and time in columns.
faceZ	Column vector containing positions of faces between nodes
filename	Filename of file
g	Gravitational acceleration
H	Depth of domain
kar	von Karmans constant

Table 2: Outputted MATLAB variables from SICT.

Variable name	
ksave	Matrix containing k at saved time steps, with depth in rows and time in columns.
momFluxsave	Matrix containing momentum flux at saved time steps with depth in rows and time in columns
nodeZ	Column vector containing positions of nodes
nutPsave	Matrix containing total turbulent viscosity at saved time steps with depth in rows and time in columns
Pbsave	Matrix containing production of k from buoyancy at saved time steps, with depth in rows and time in columns.
Pssave	Matrix containing production of k from shear at saved time steps, with depth in rows and time in columns.
rho0	Reference density
S0	Reference salinity
Ssave	Matrix containing S at saved time steps, with depth in rows and time in columns.
stratChoice	Choice for stratification
T0	Reference temperature
tsave	Row vector containing t at saved time steps
Tsave	Matrix containing T at saved time steps, with depth in rows and time in columns.
usave	Matrix containing U at saved time steps, with depth in rows and time in columns.
z0b	Roughness length at the bottom
z0s	Roughness length at the surface