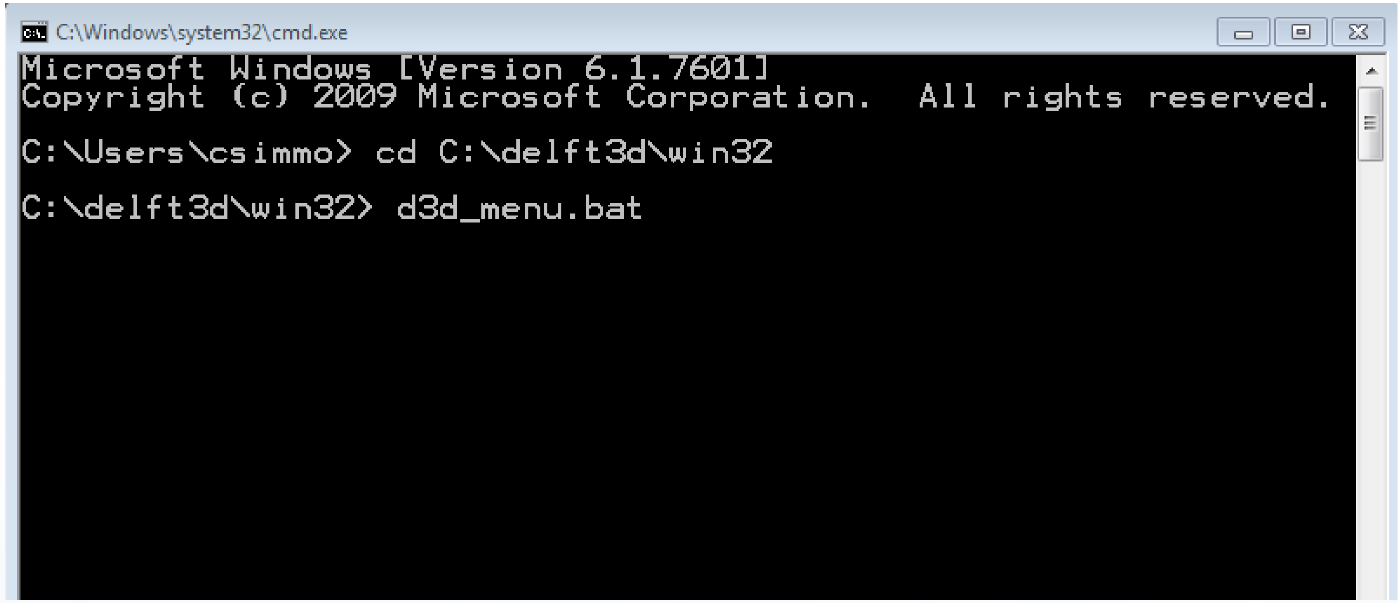
SAND2016-9627 TR

SNL-Delft3D-CEC

3D/2D modeling suite for integral water solutions

Building a Basic Model

Delft3D initialization



- Using a Windows operation system, start the Delft3D program.
- From the Windows command line, run d3d_menu.bat

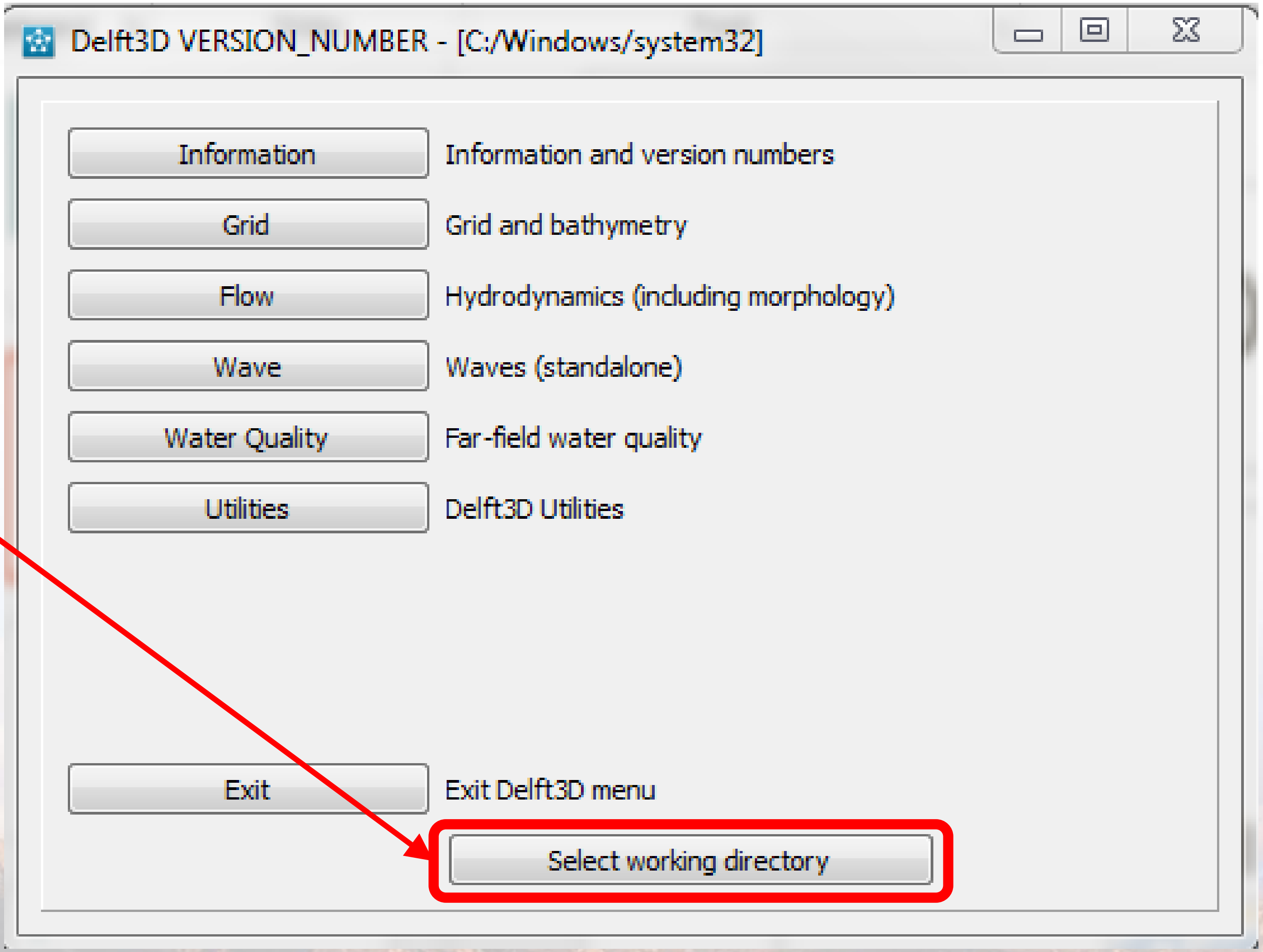
Delft3D Documentation

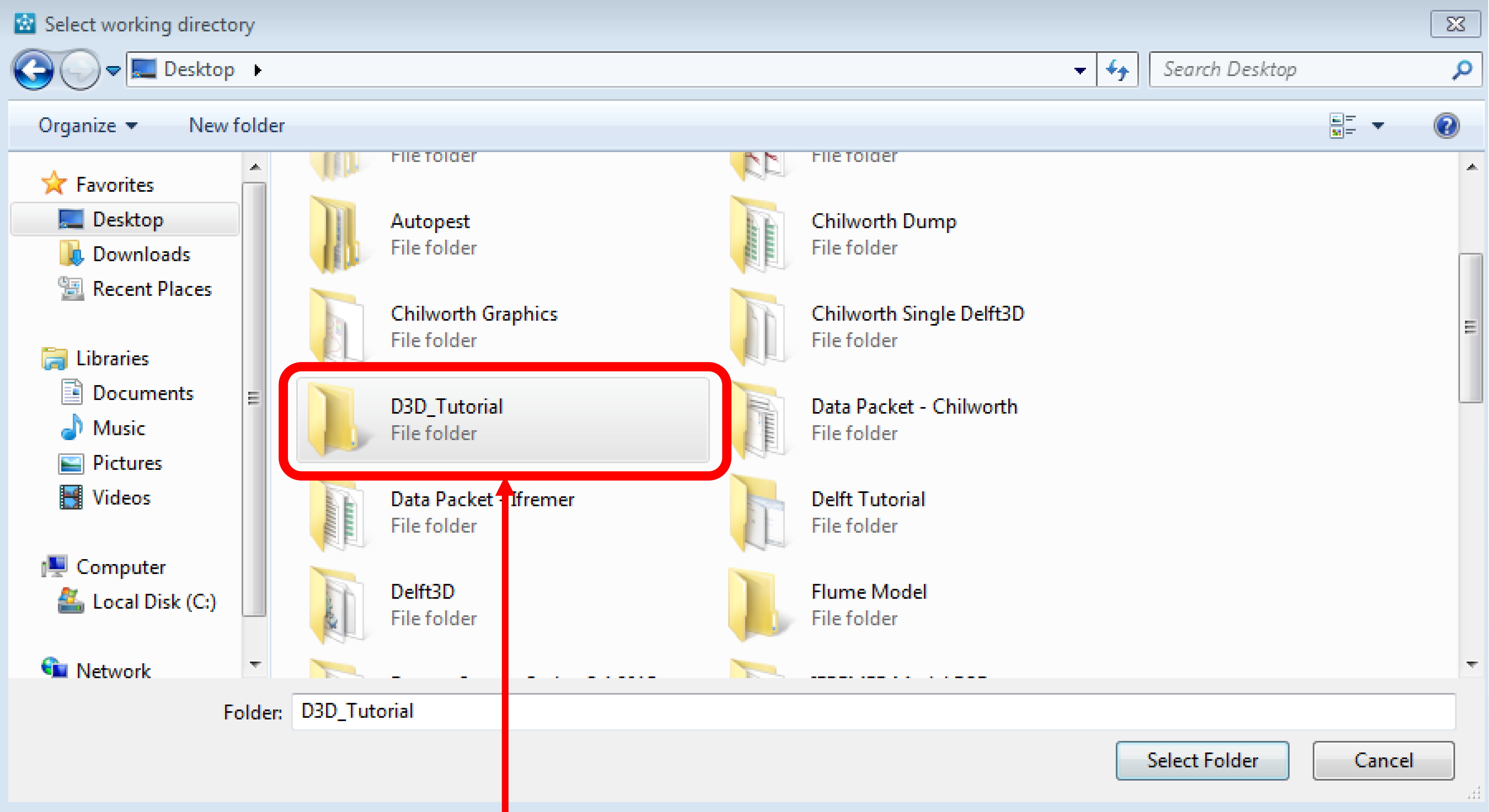
- SNL-Delft3D-CEC is derived from the original Delft3D FLOW module, thus most of D3D usage can be applied.
    - D3D FLOW Manual : http://content.oss.deltares.nl/delft3d/manuals/Delft3D-FLOW_User_Manual.pdf

- This tutorial will go into the usage of the turbine module implemented specifically in SNL-Delft3D-CEC.

Setting Things Up

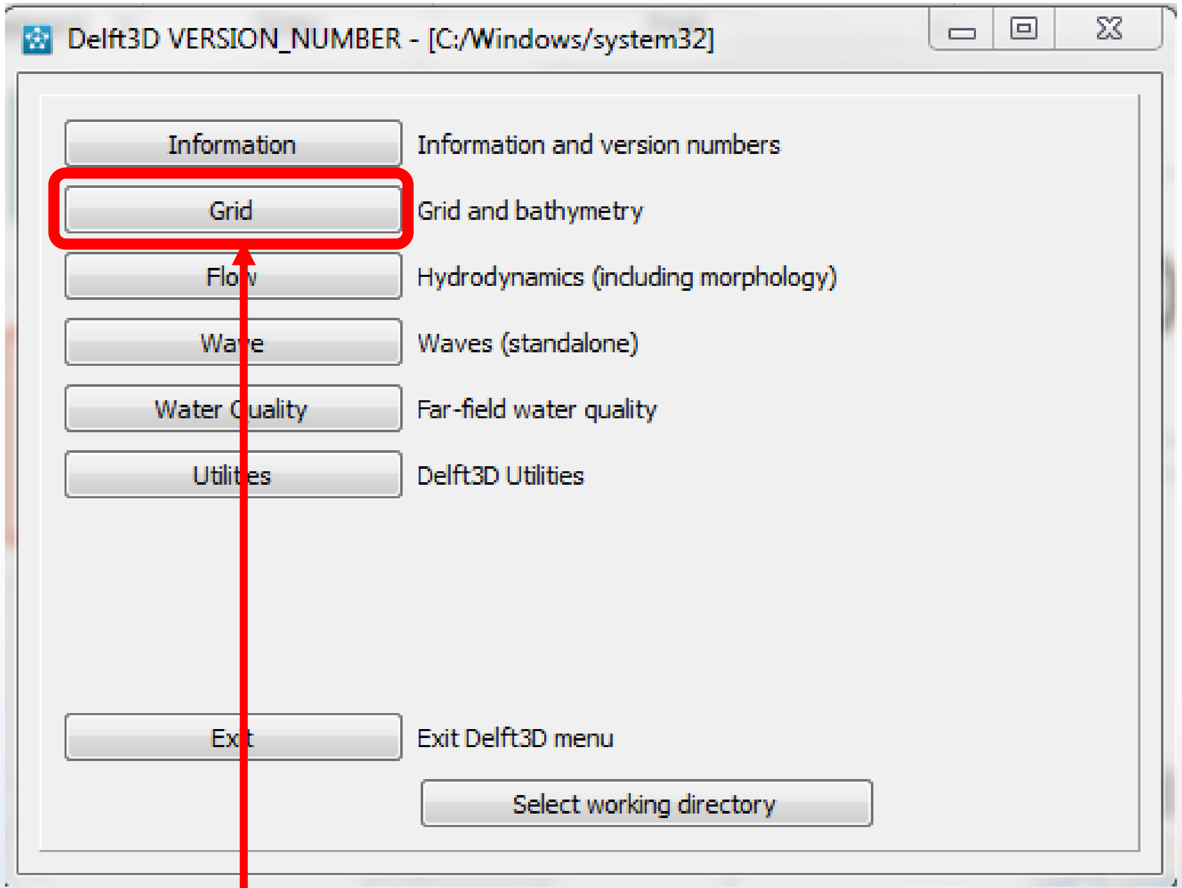
- Select a working directory were all files from this tutorial will be saved

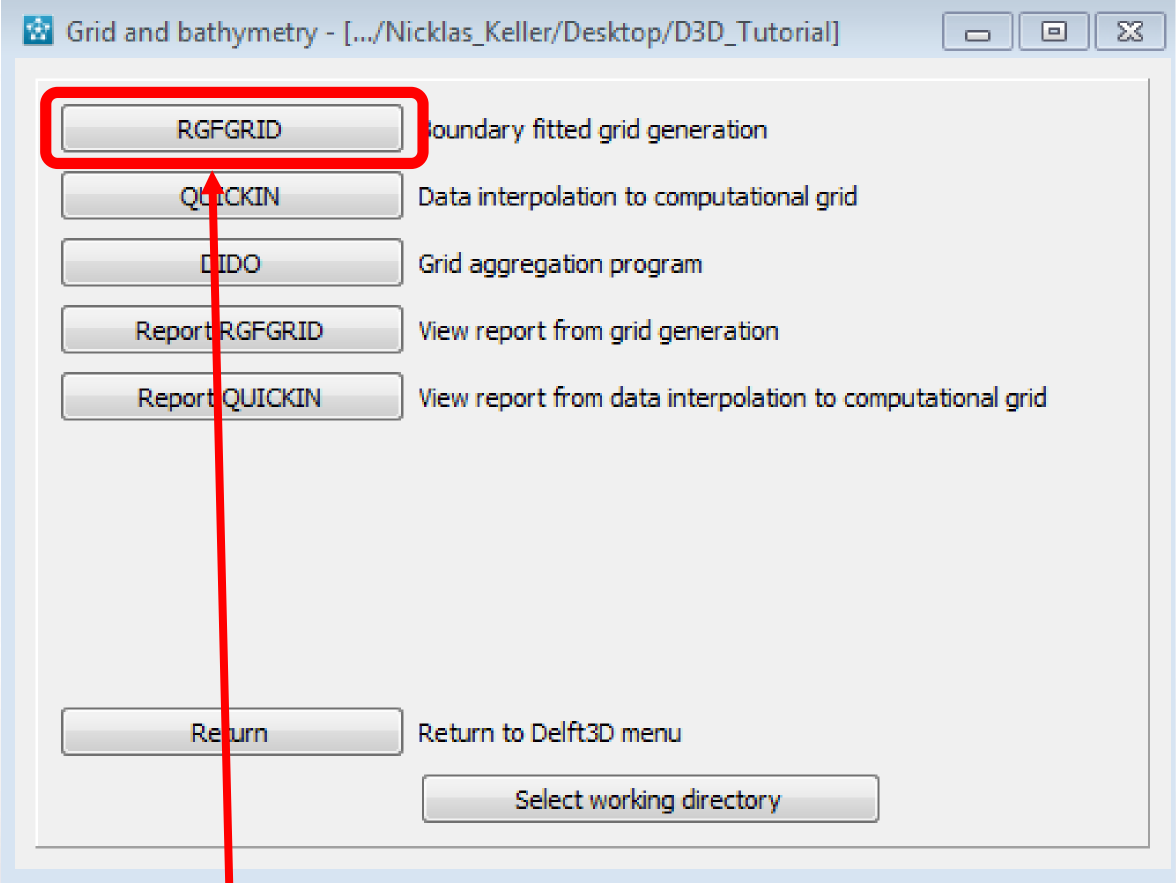




- Create and select a new folder

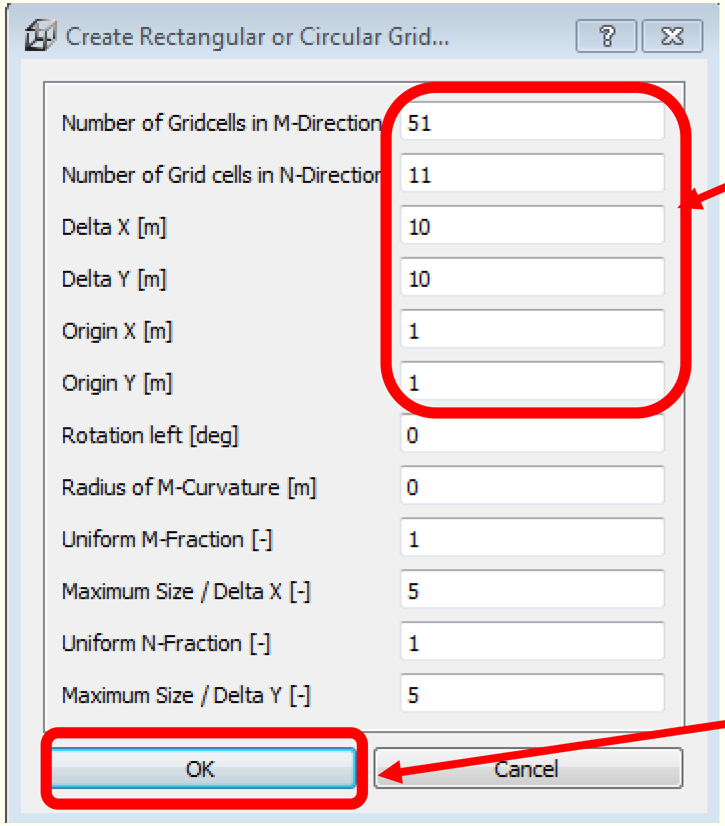
Creating a Grid





- Select Grid
- Select RGFGRID

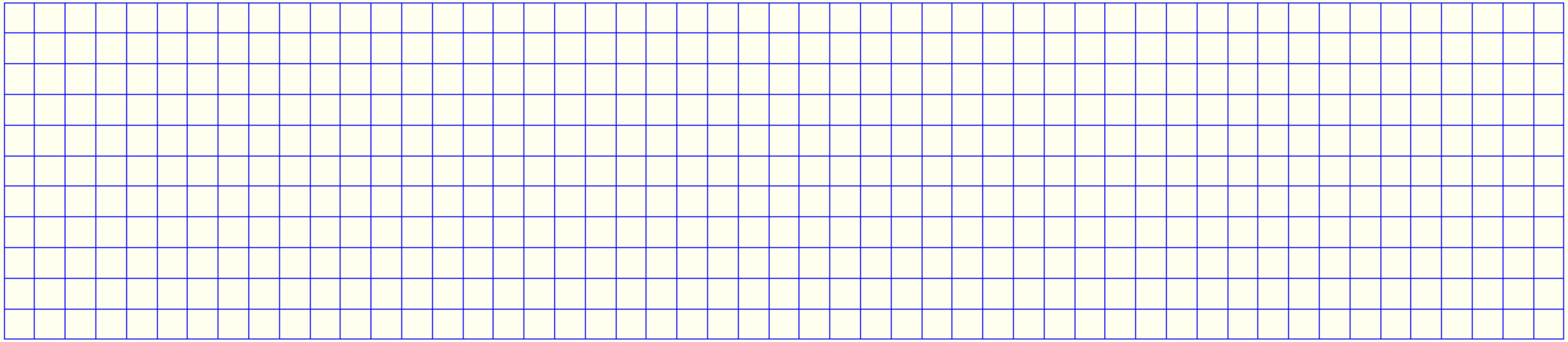
Sandia National Laboratories



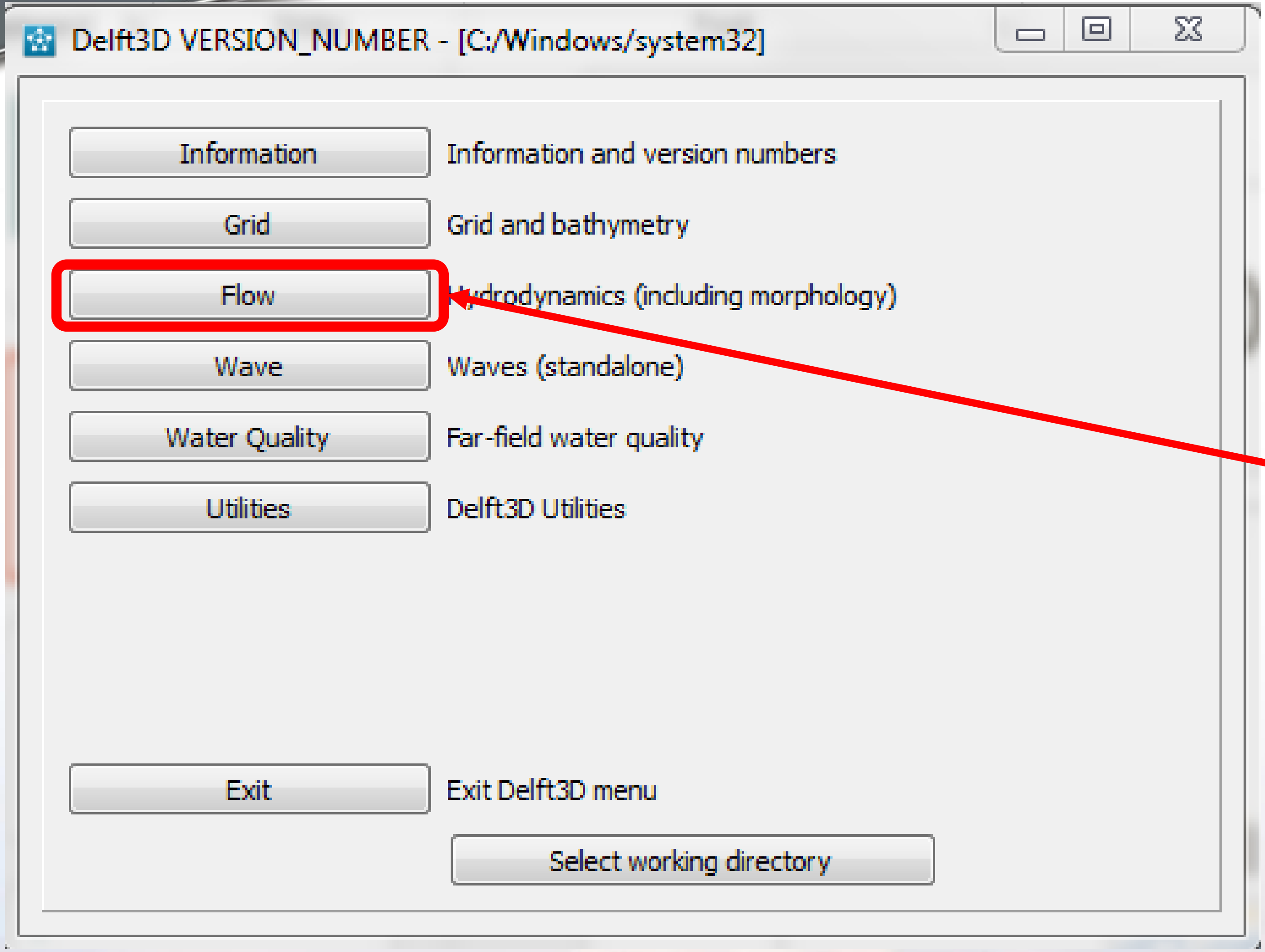
- Select Operations>Create Rectangular or Circular Grid
- We will build a straight flow channel with flow from east to west. It will be 110 m across and 510 m long. We will use an odd number of cells in each direction so that MHK devices can be centered.
- *M*-direction = 51, *N*-direction = 11,
  Delta *X*, *Y* = 10, 10, (grid spacing)
  and the Origin *X*, *Y* = 1,1
- Click OK

- Save Project as D3D

- The grid file types created by RGFGRID are .d3d, .grd, and .enc

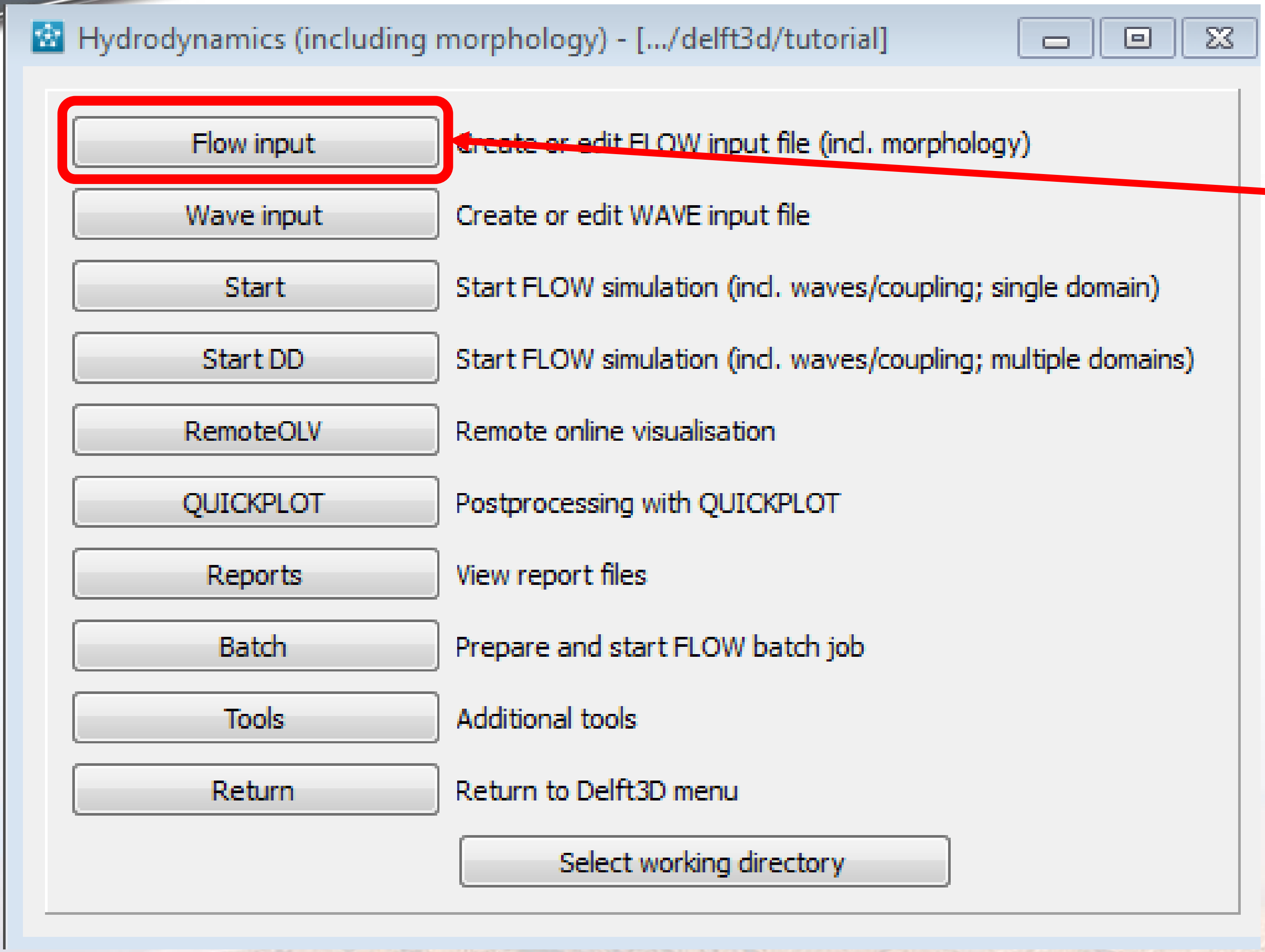


Flow

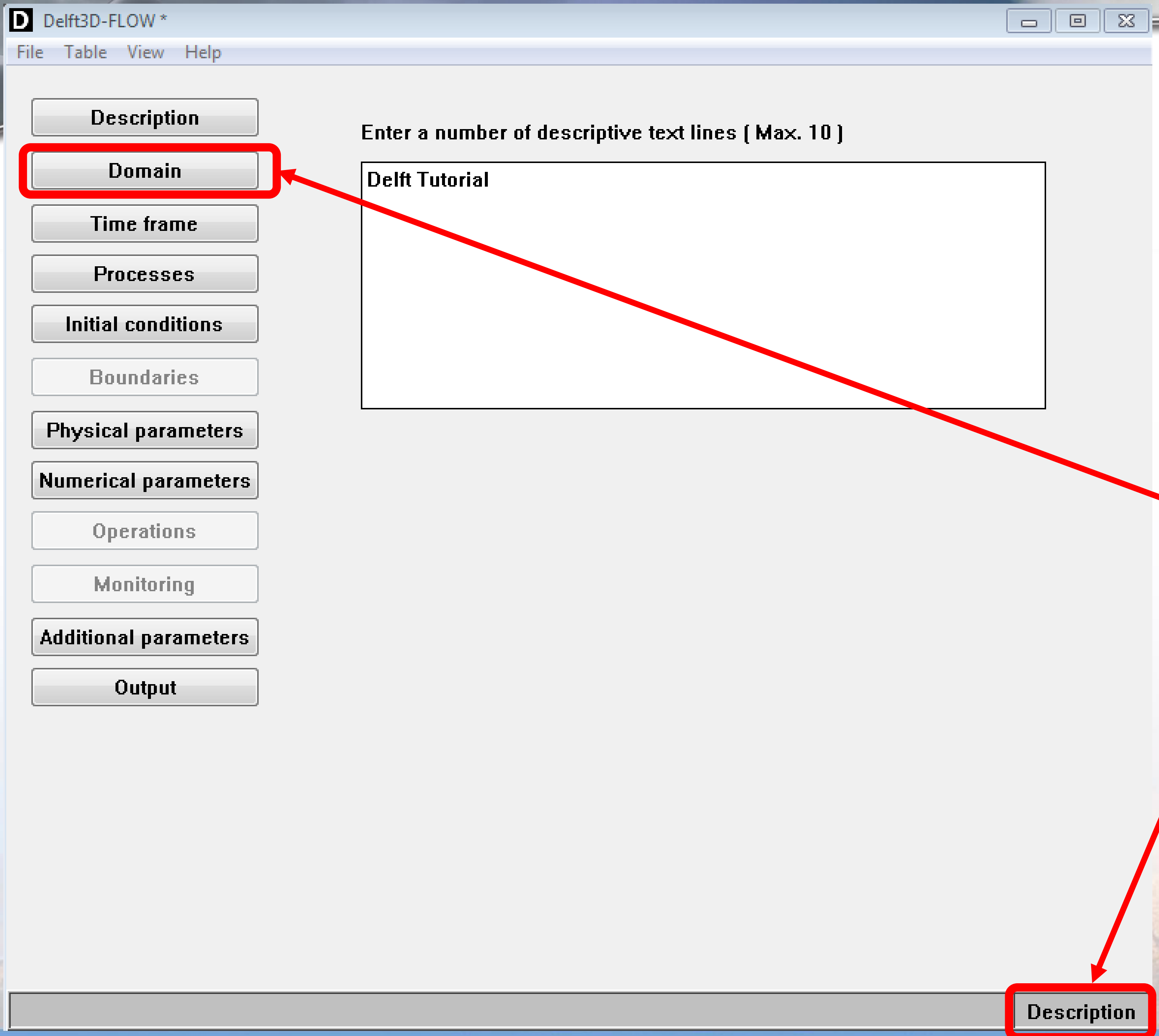


- Close RGFGRID and select Return
- This will put you back in the main Delft3D menu
- Select Flow

Flow



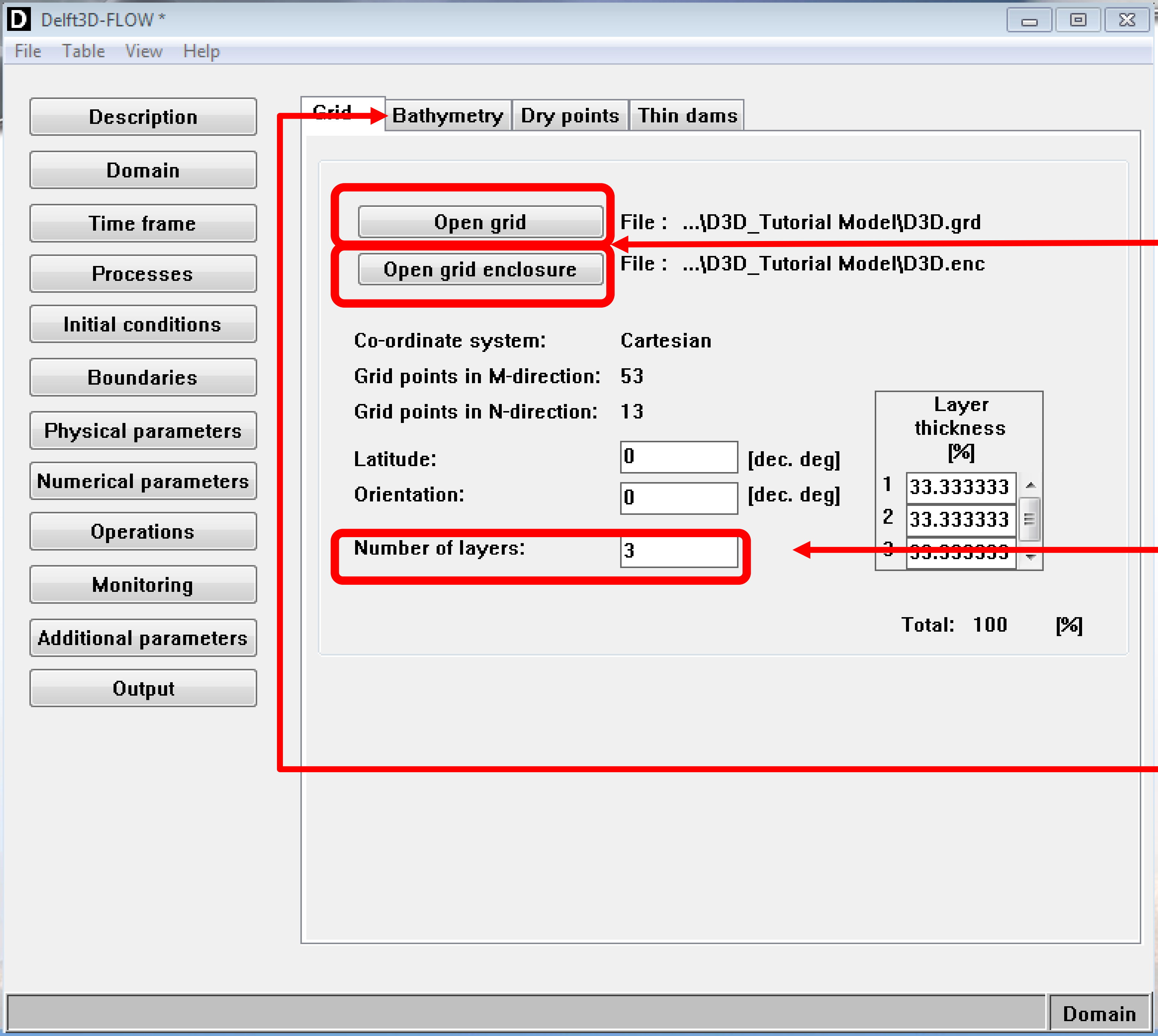
- Select Flow input



Description

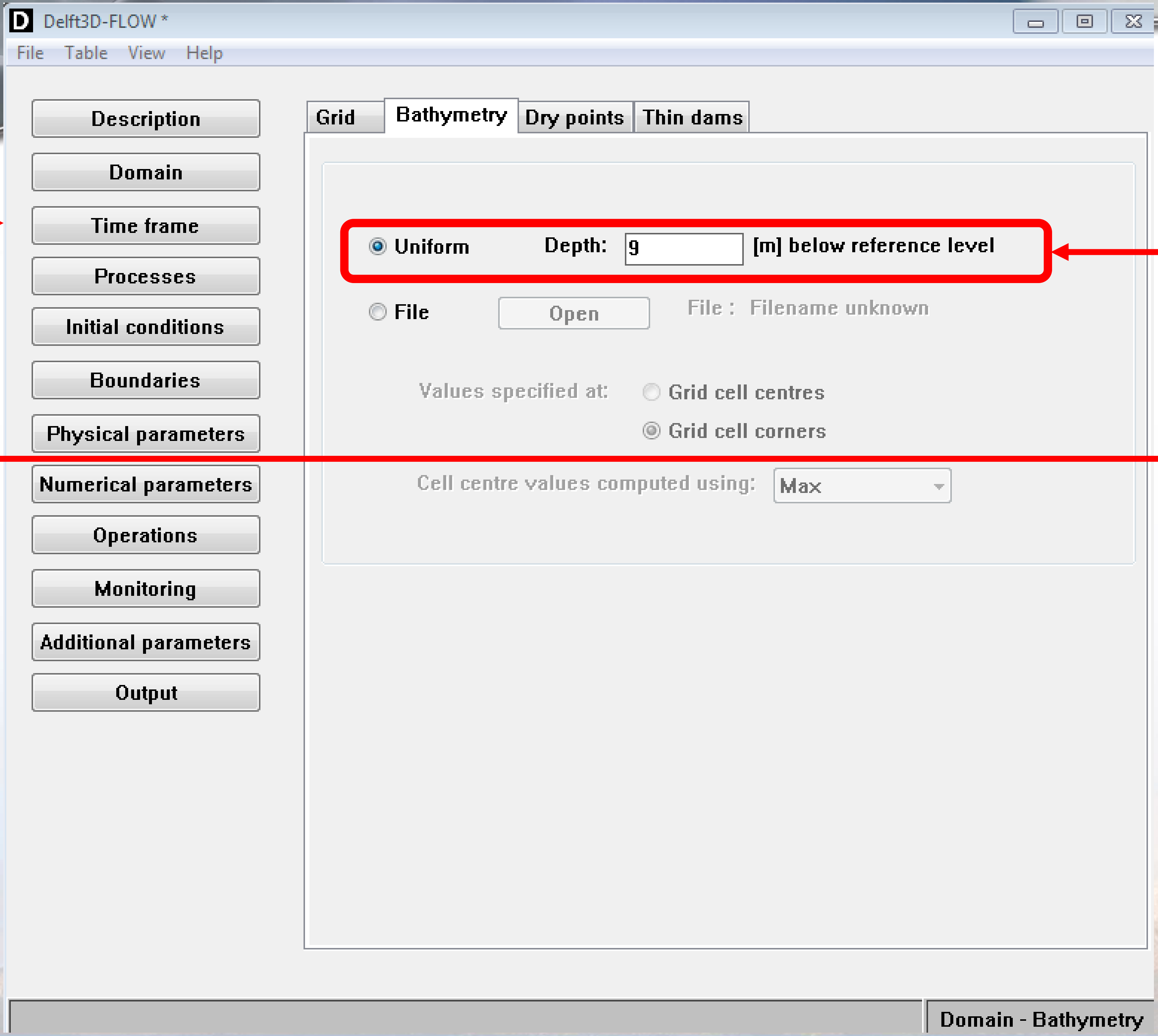
- The default tab in Flow is a text box where a description of the experiment can be written.
- Enter “Delft Tutorial”
- Select Domain
- Bottom right corner indicates which tab Flow is currently on.

Setting a Domain



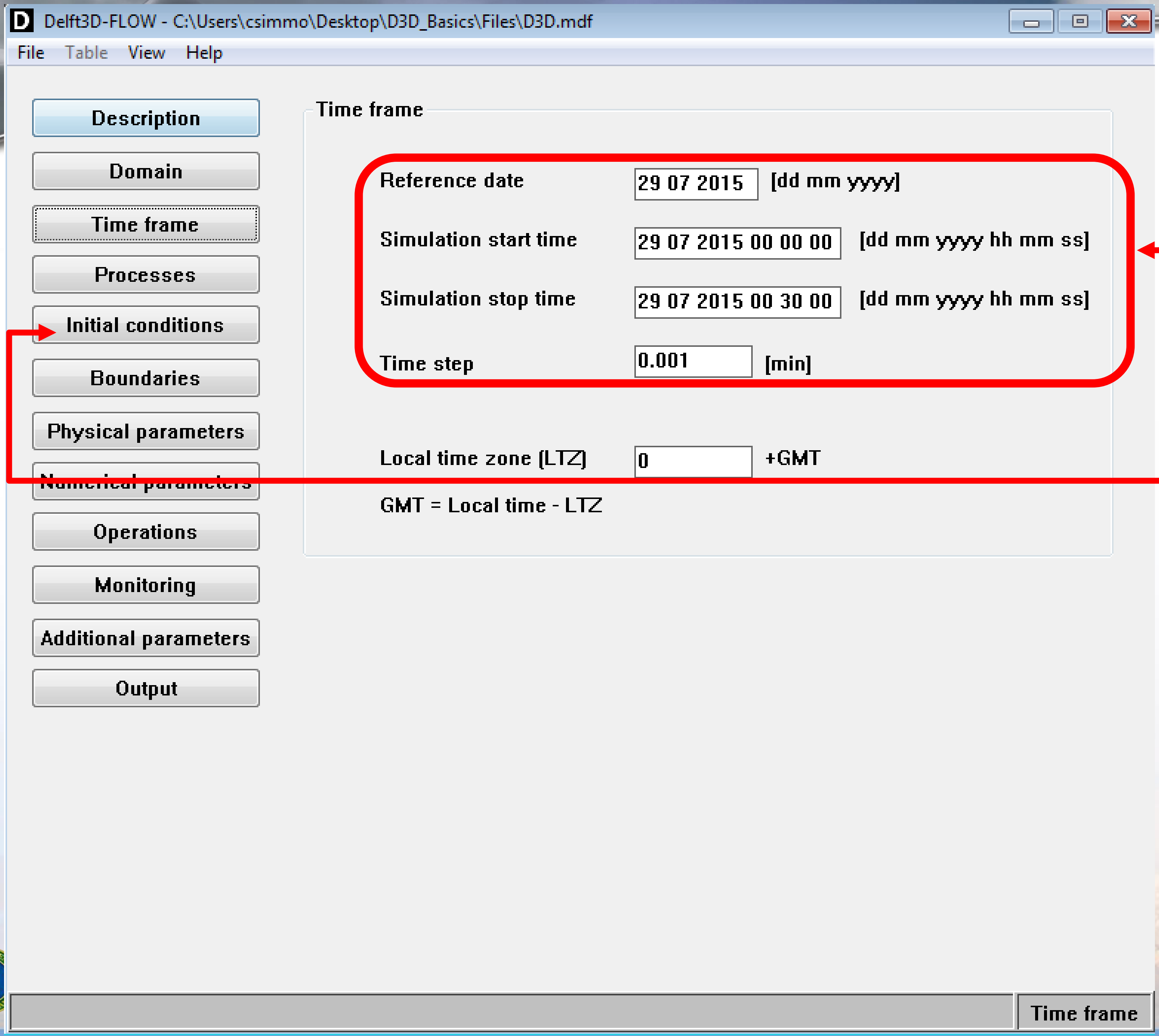
- Open the grid and grid enclosure created by RGFGRID:
  - D3D.grd
  - D3D.enc
- Set Number of vertical layers to 3
- Select Bathymetry

Setting a Domain



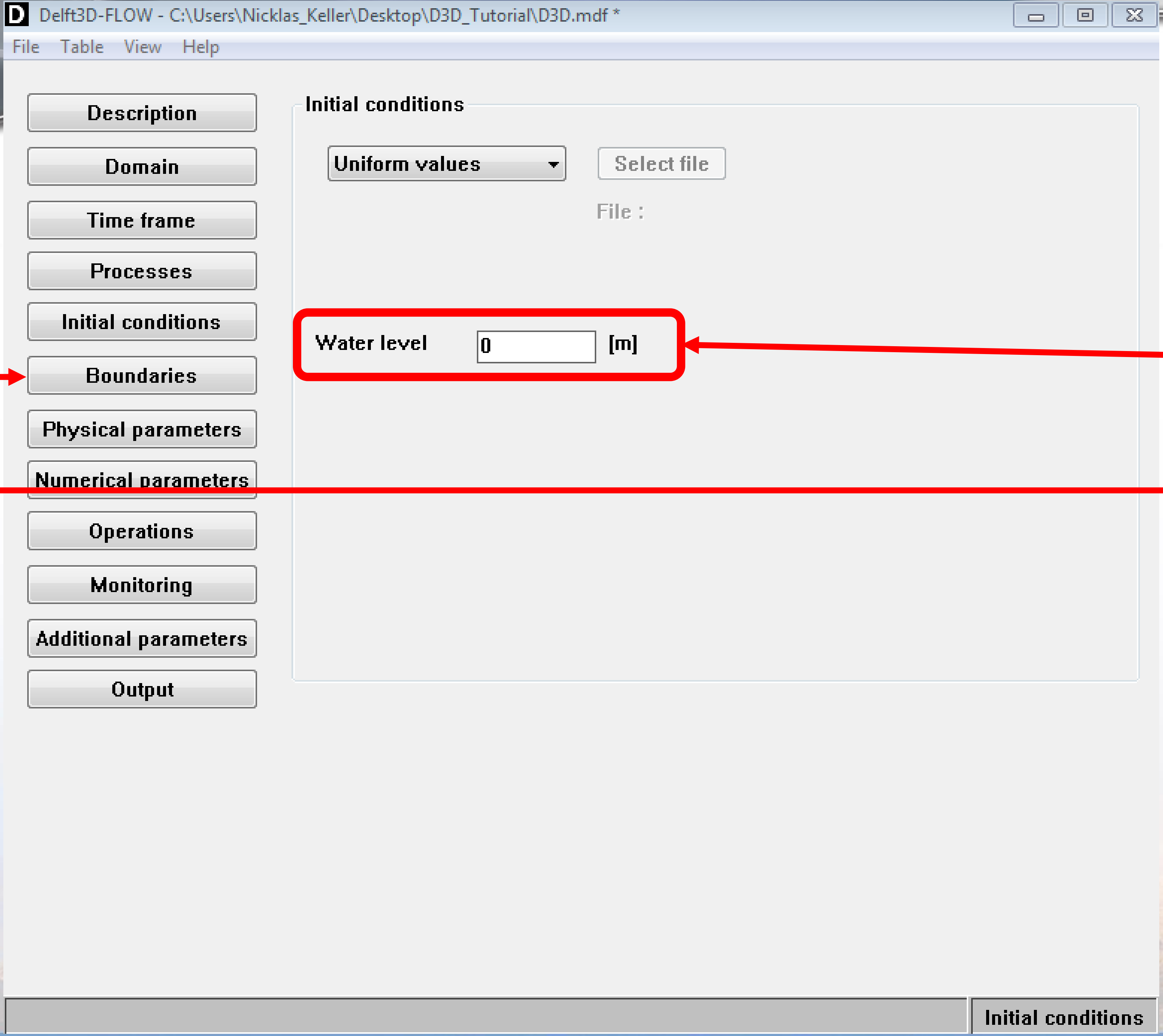
- Set Bathymetry to a Uniform Depth of 9 meters.
- Click on Time frame

Simulation Time Frame

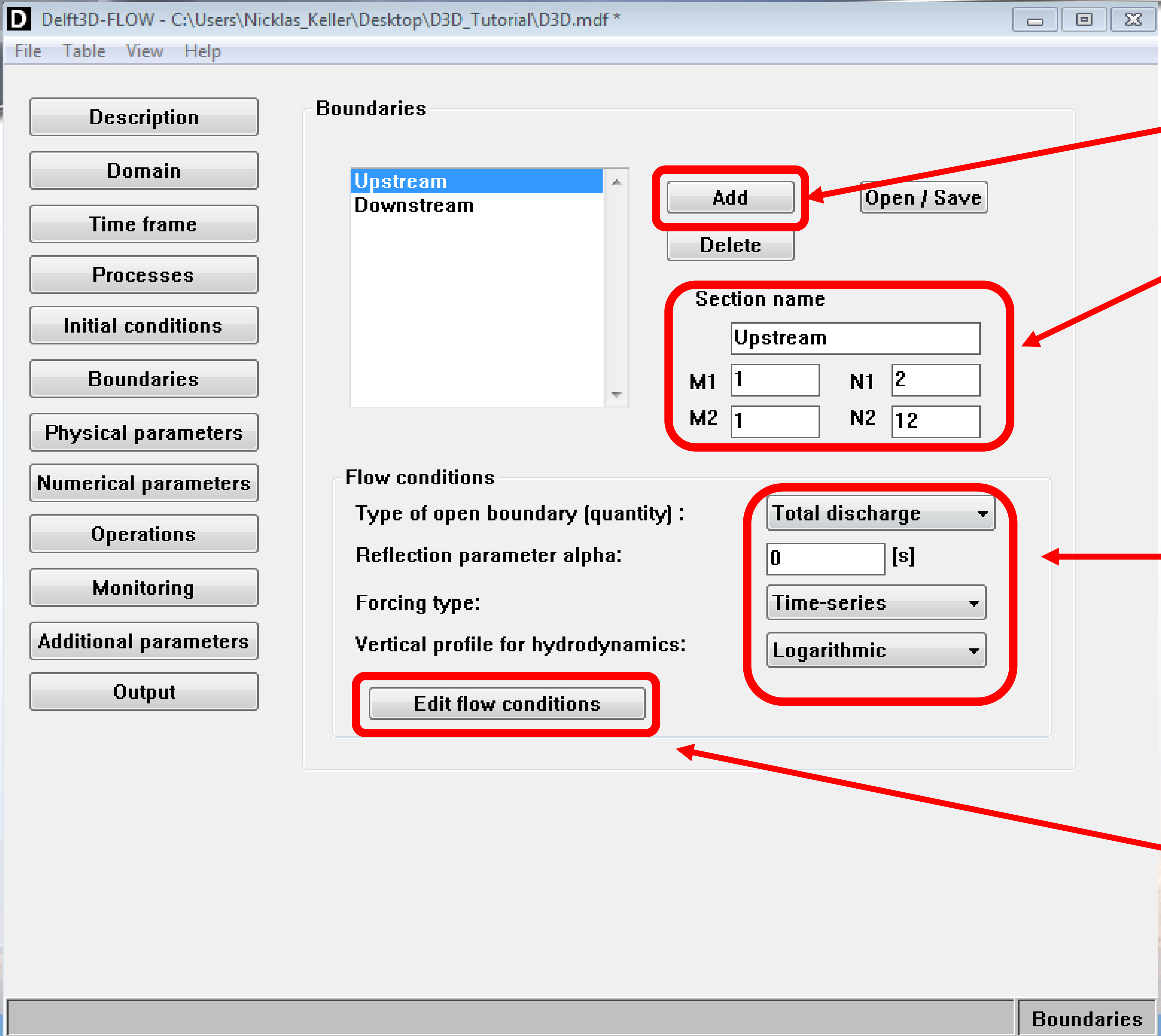


- Let's simulate 30 minutes with a 0.001 second time step
- Click on Initial Conditions

Setting Initial Conditions



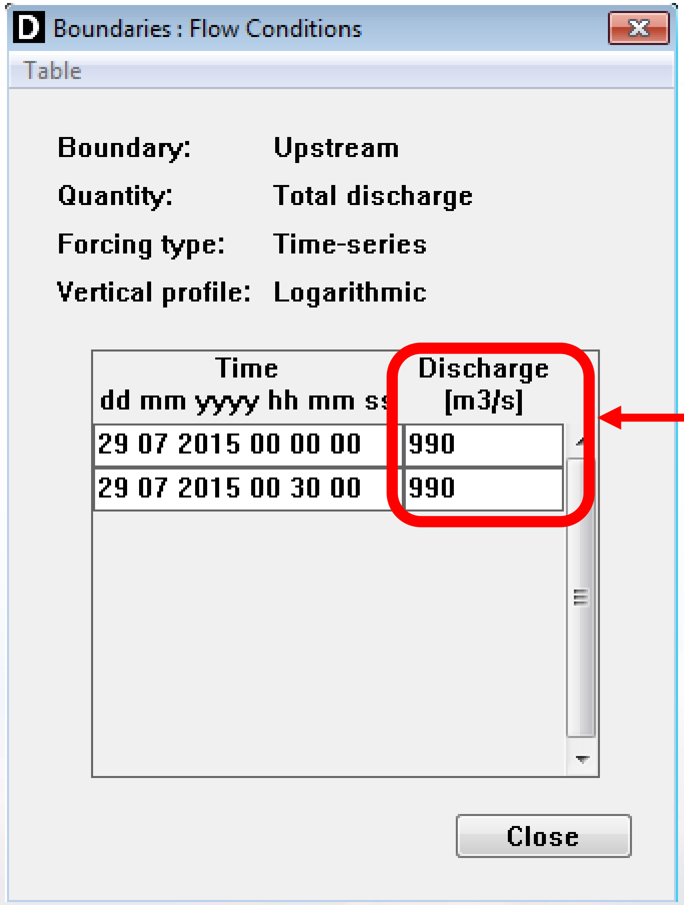
- Set initial water level equal to 0 meters
- Select Boundaries



Flow

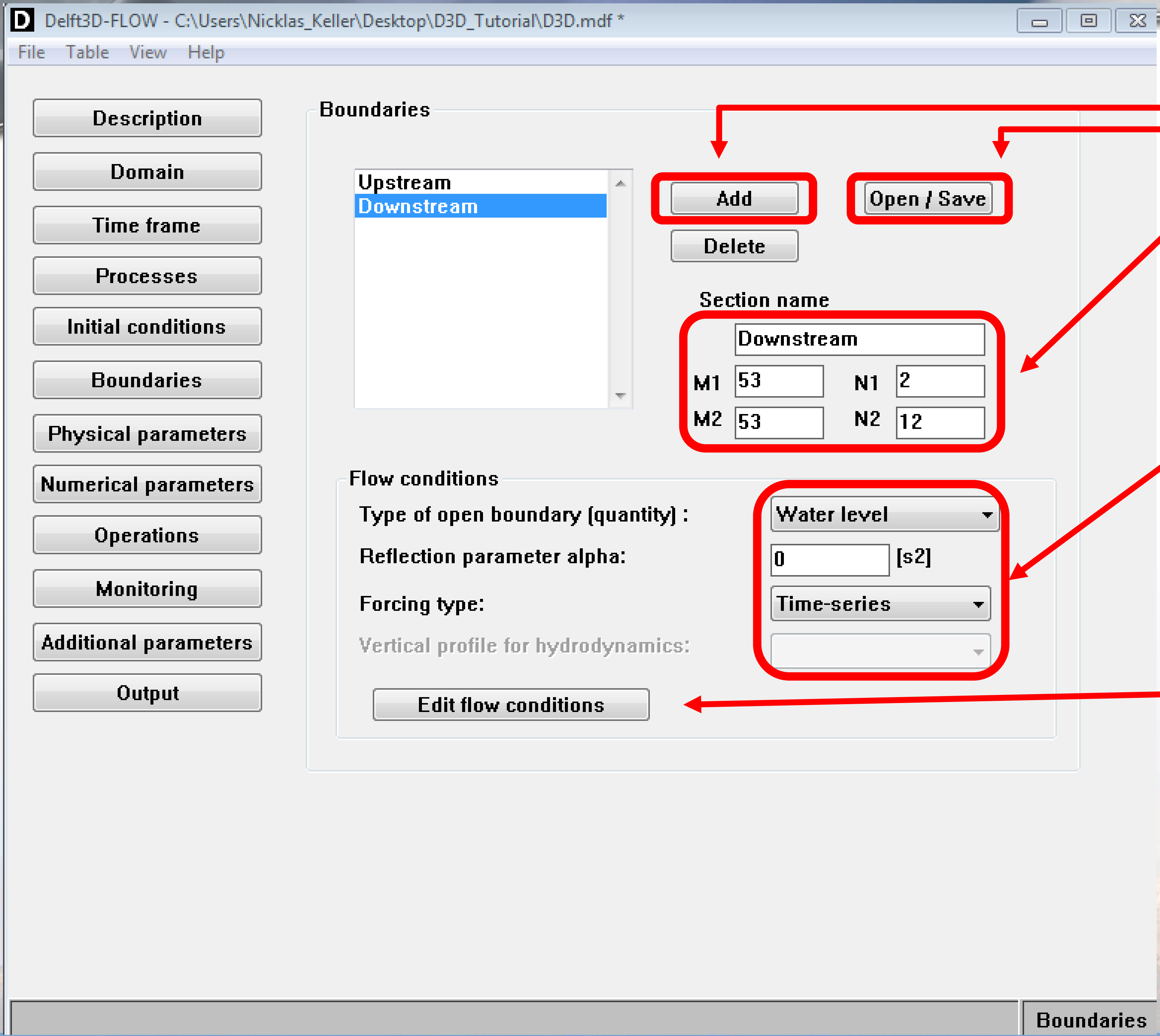
- In Boundaries, select add
- Title the section name upstream, and assign the following: $M1=1$, $N1=2$,$M2=1$, $N2=12$
- Under flow conditions, change the type of open boundary to Total discharge and the forcing type to Time-series
- Select Edit flow conditions

Flow



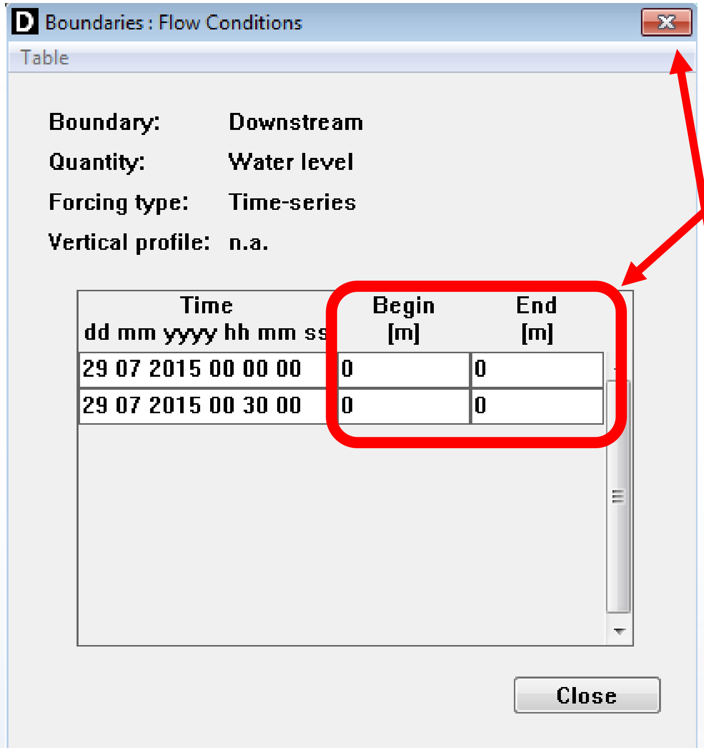
- Set discharge volumetric flow rate to 990 m^3/s
- Close this box

Flow

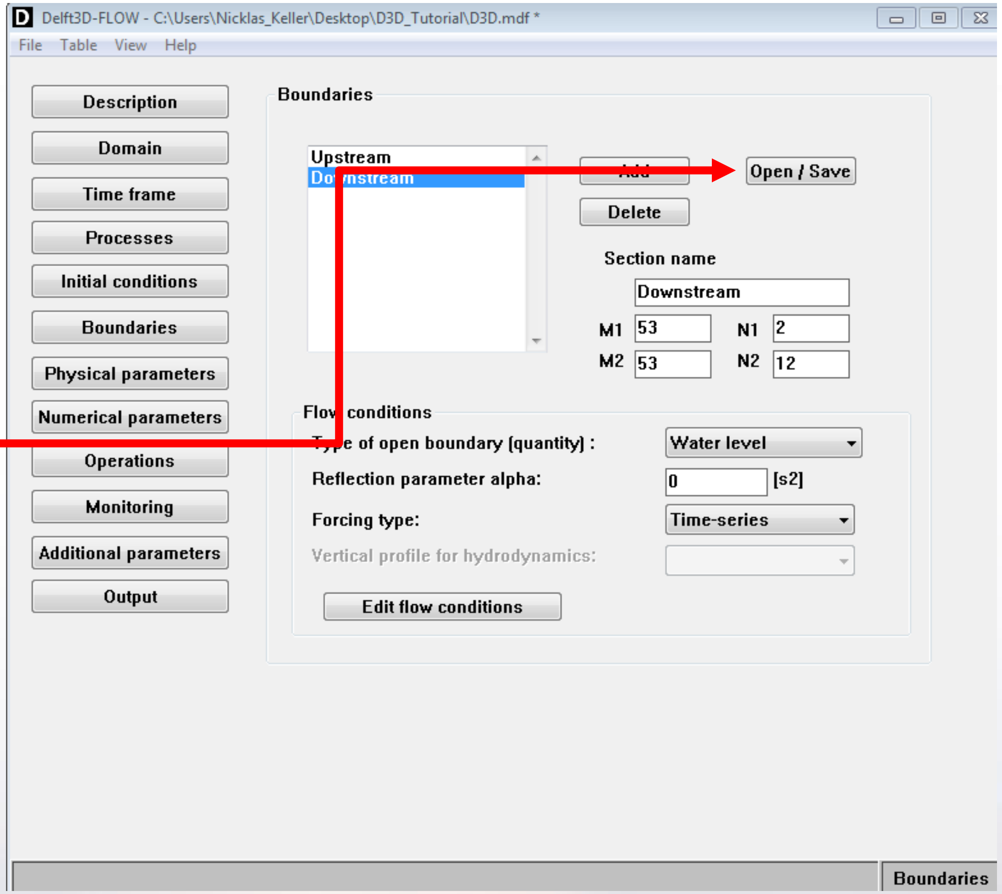


- Select Add
- Title the section name downstream, and assign the following: *M*1=53, *N*1=2, *M*2=53, and *N*2=12
- Make sure the type of open boundary is "Water level", and change the forcing type to "Time-series"
- Select "Edit flow conditions" for Downstream boundary. Set "Begin" to 0.
- Select Open/Save

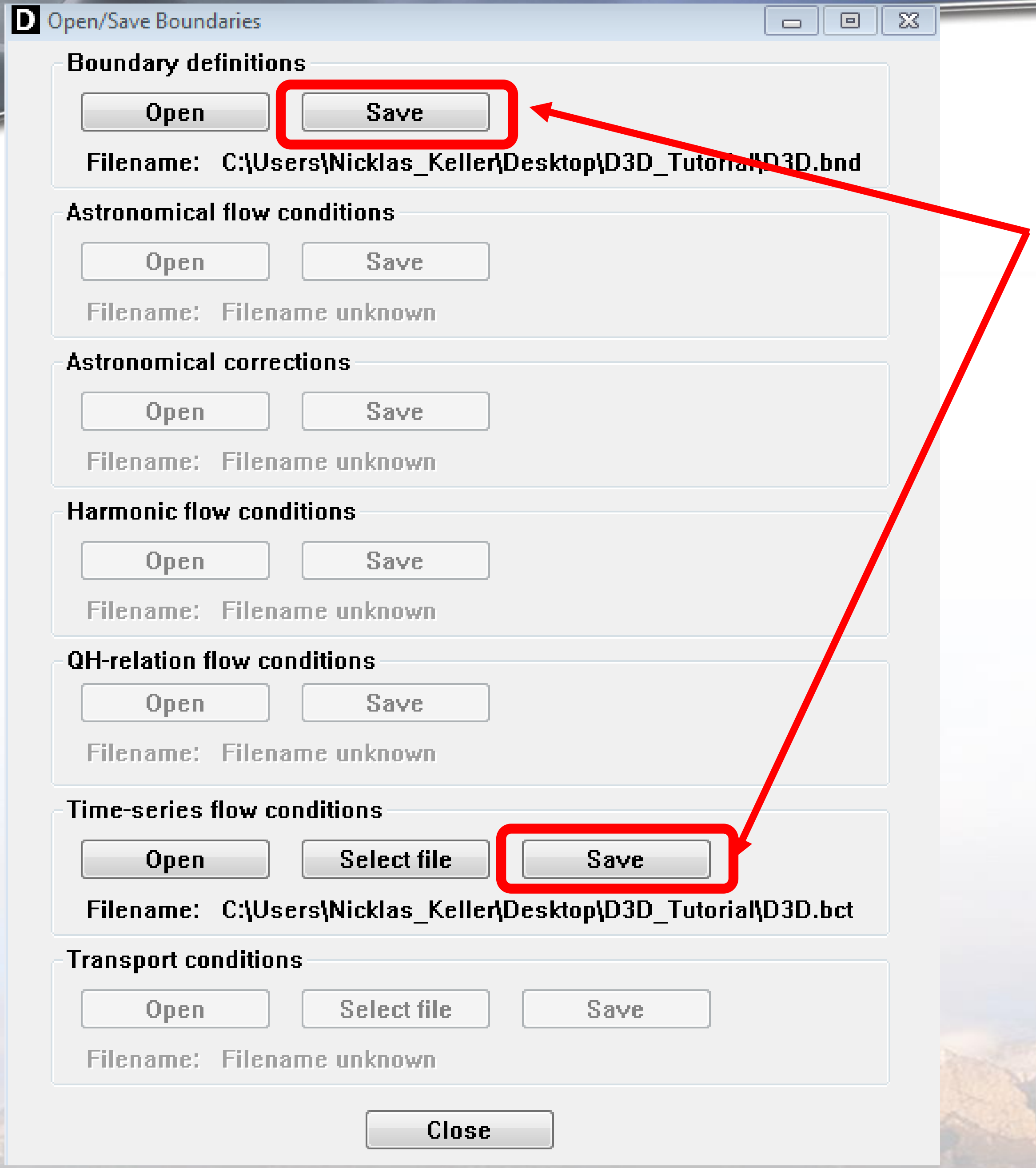
Sandia National Laboratories



- Set “Begin” and “End” to 0.
- Close this box
- Select “Open/Save” to save these files in order to implement them in your simulation.



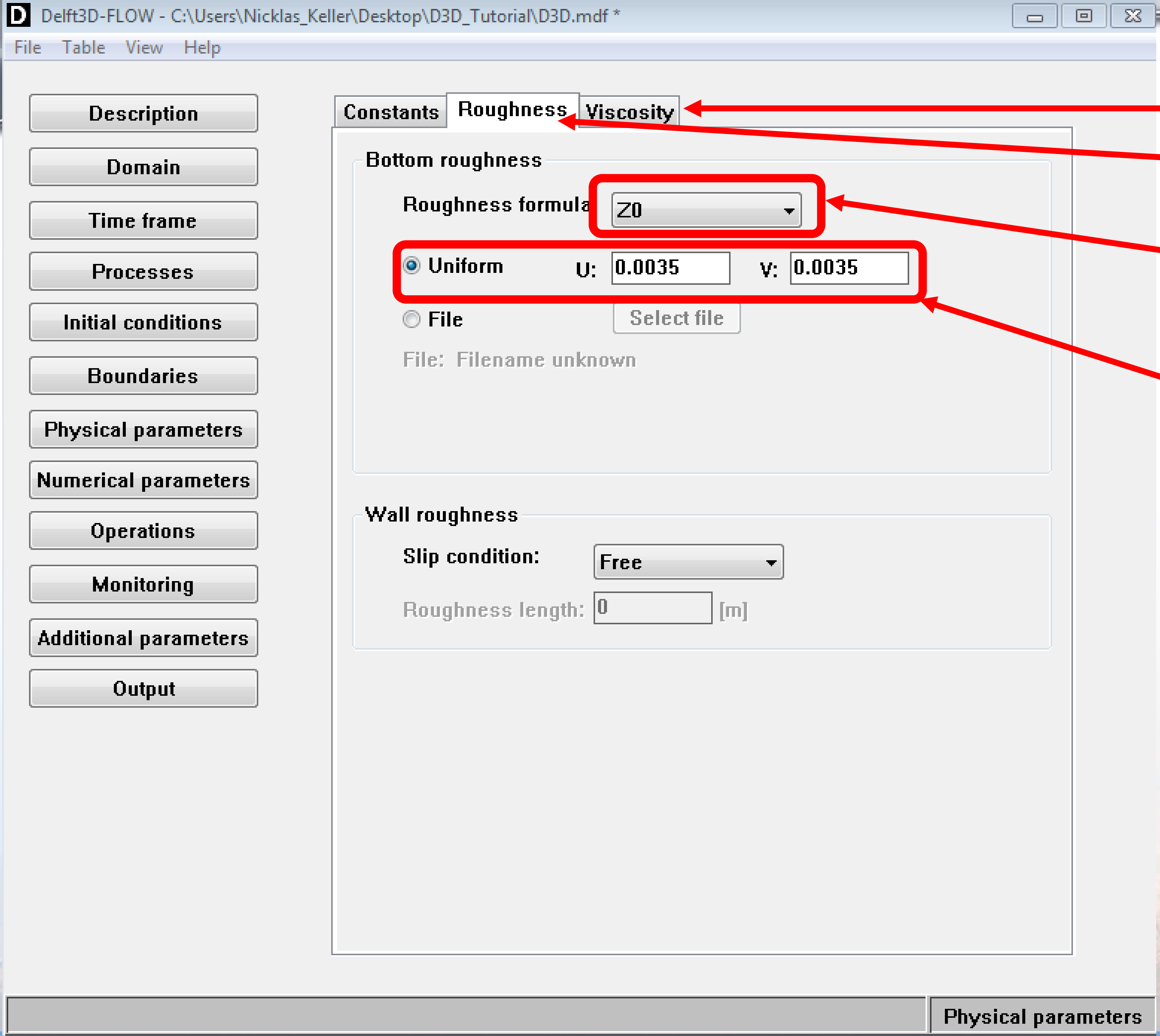
Flow



- Save Boundary definitions and Time-series flow conditions as D3D.bnd and D3D.bct, respectively

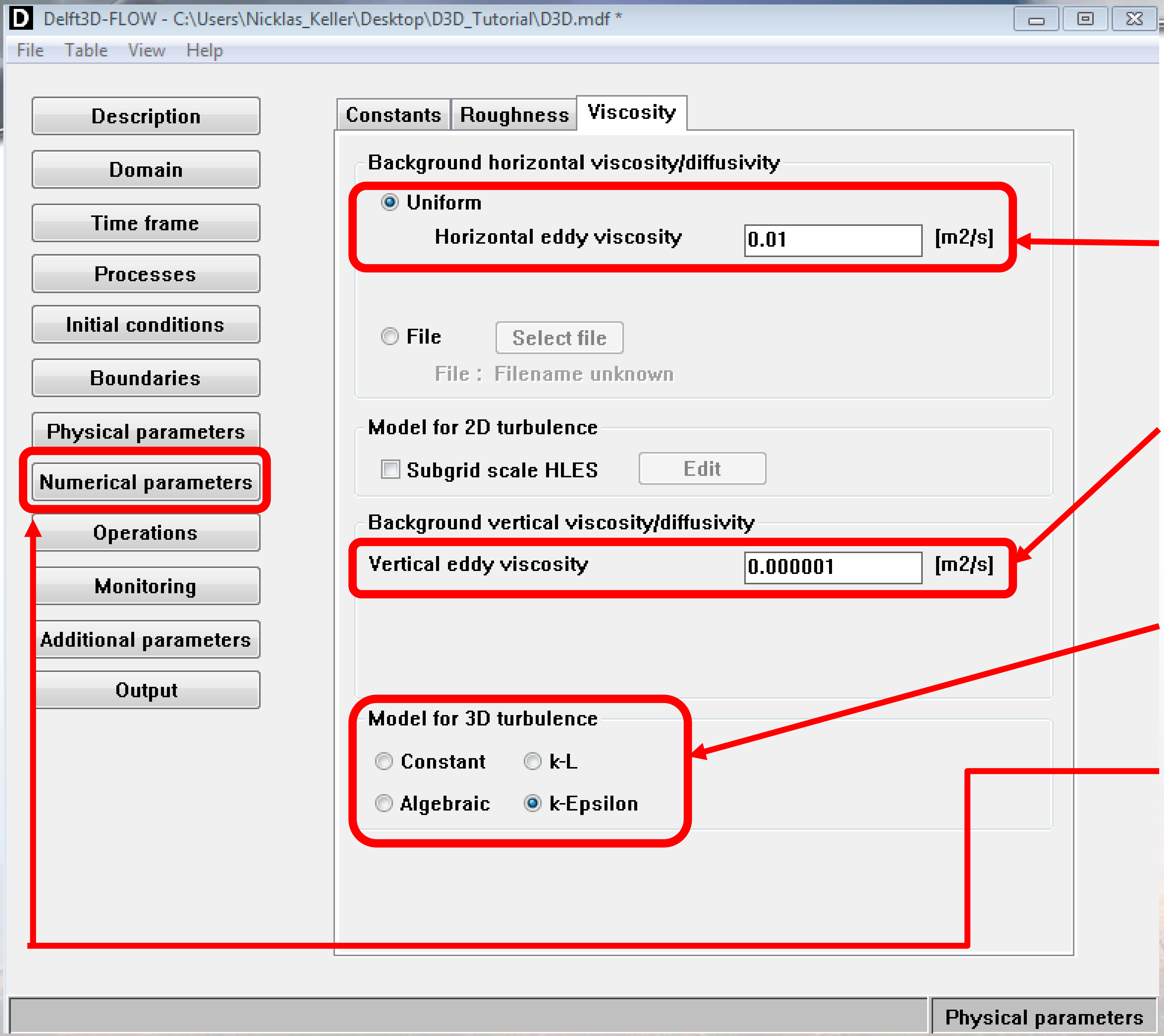
- Close and select Physical Parameters

Flow



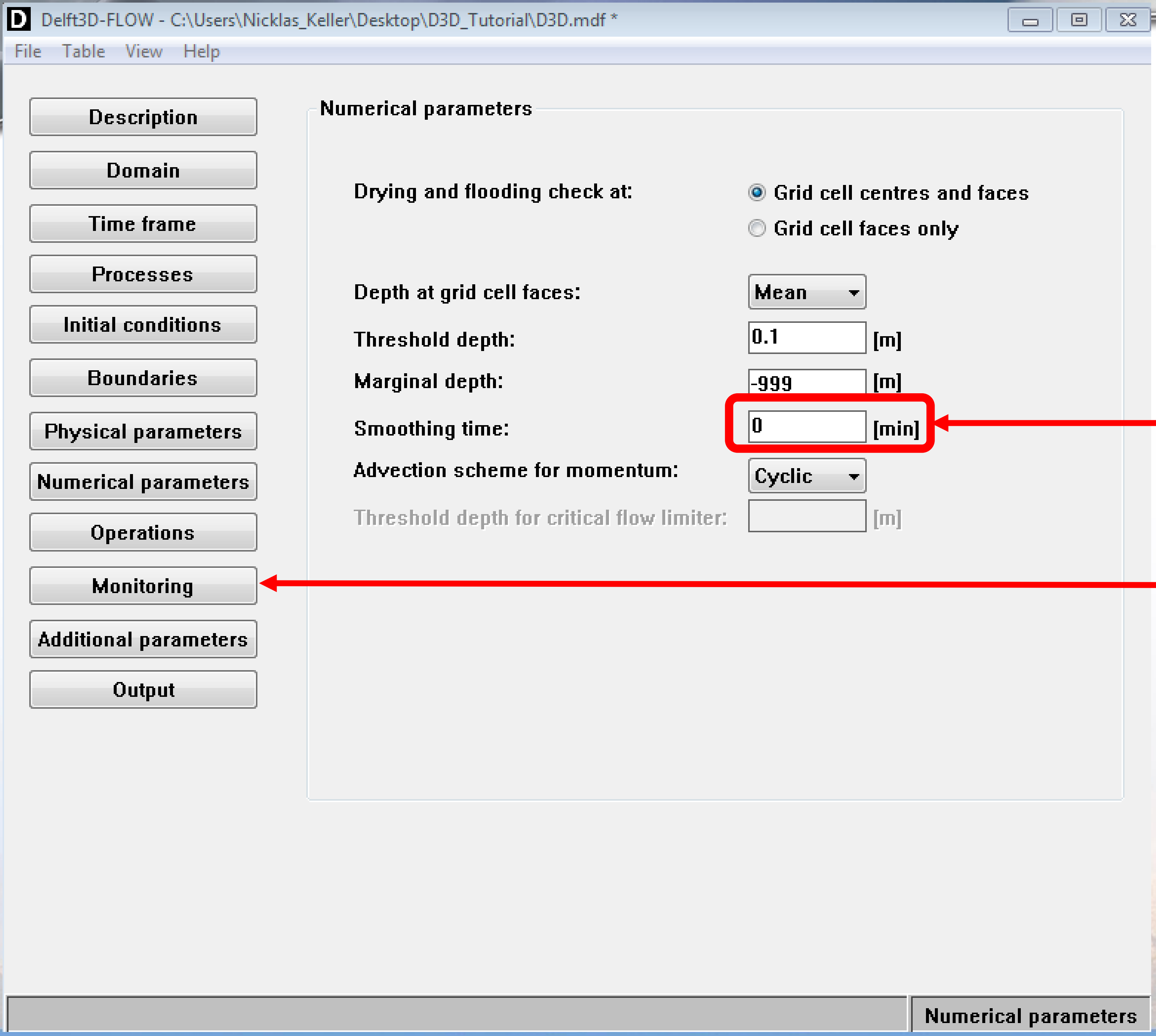
- Select Roughness
- Set Roughness formula to Z0
- Select Uniform, setting: U=0.0035 and V=0.0035
- Select Viscosity

Flow



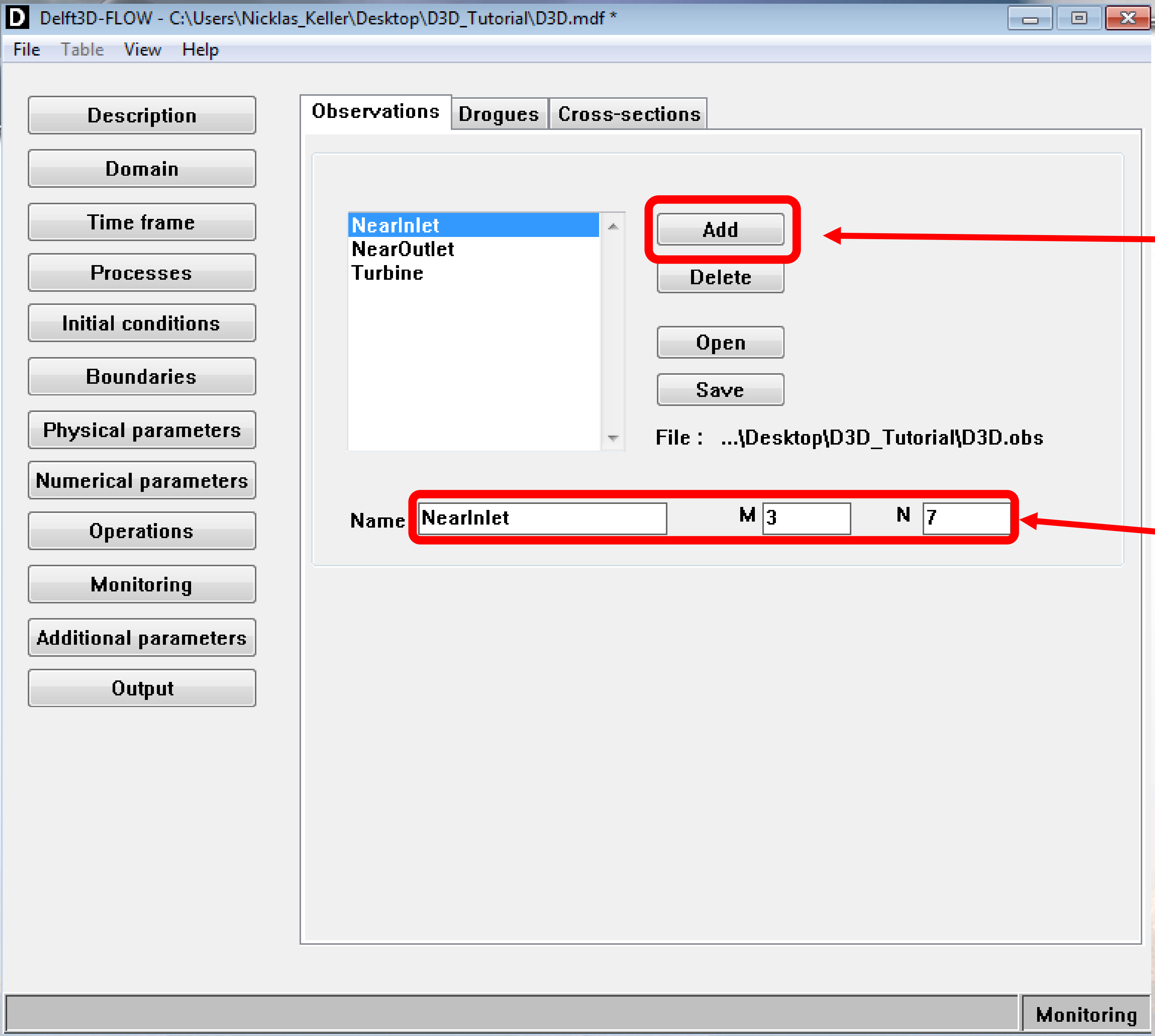
- Set Horizontal eddy viscosity to 0.01 m^2/s
- Set Vertical eddy viscosity to 0.000001 m^2/s
- Under Model for 3D turbulence, Select k-Epsilon
- Select Numerical parameters

Flow



- Make sure “Smoothing time” is 0 minutes
- Select Monitoring

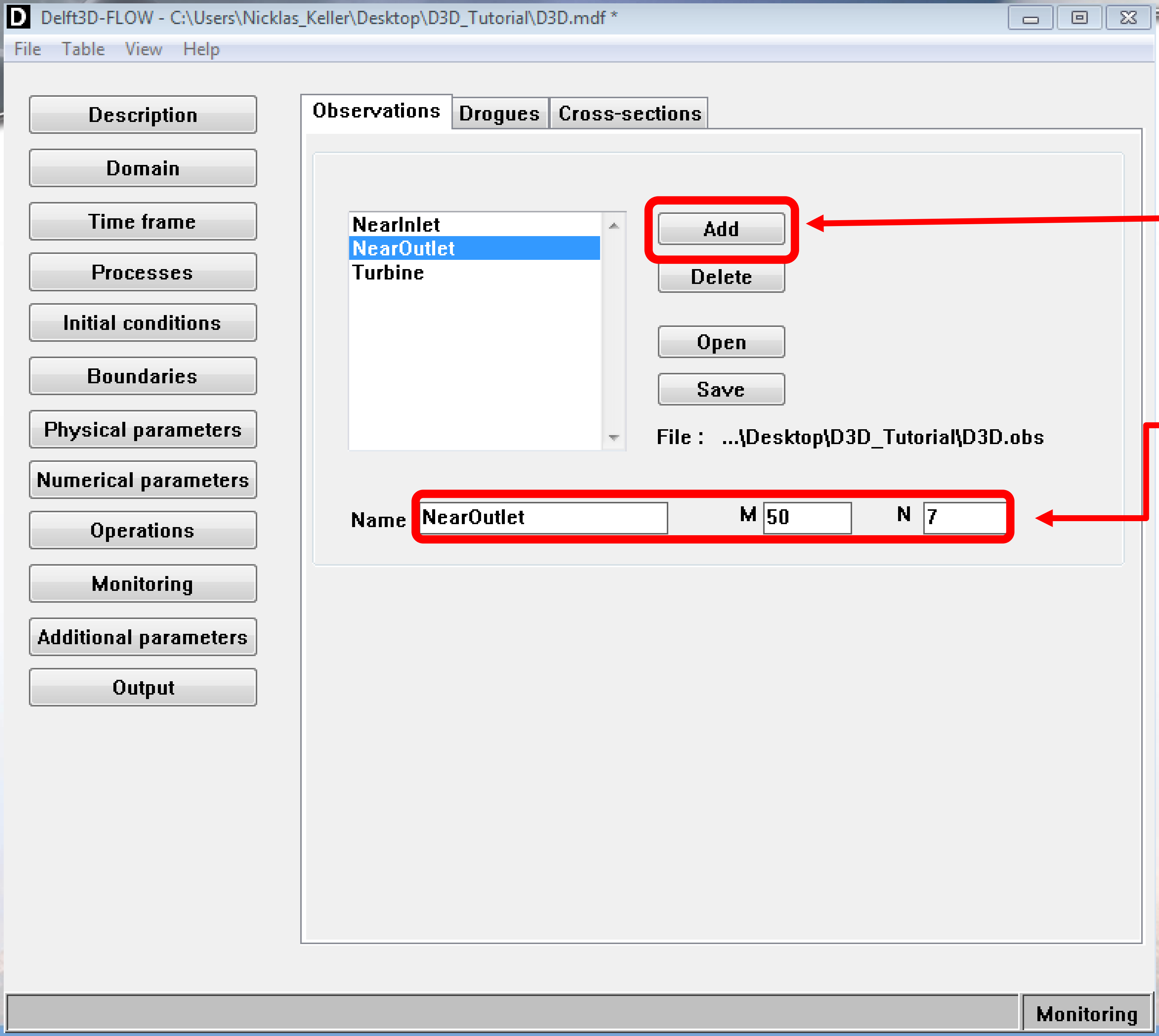
Flow



- Select Add to add monitoring point within grid

- Name this monitoring cell NearInlet with M=3, N=7

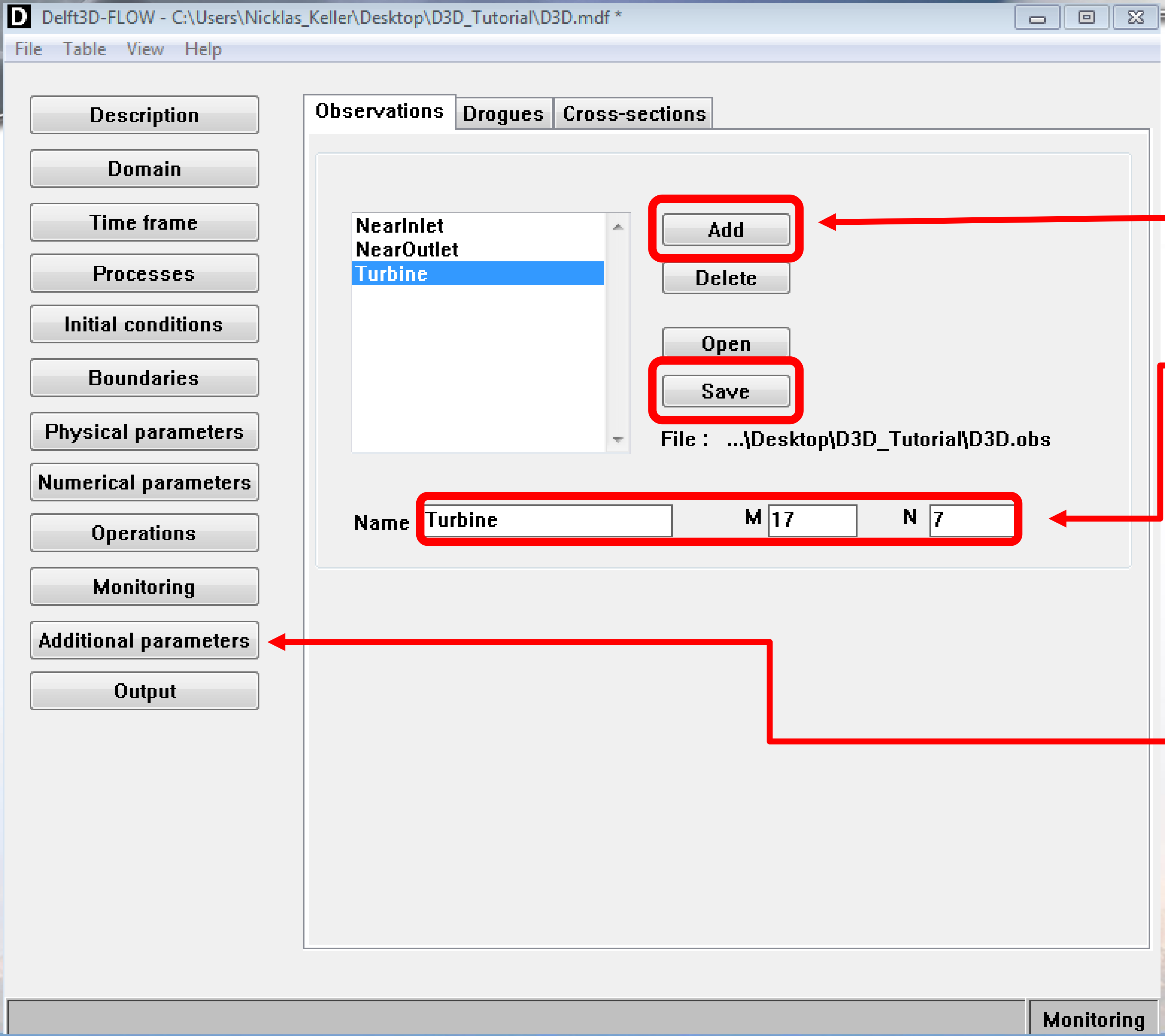
Flow



- Select Add
- Name this monitoring cell NearOutlet with M=50, N=7

Sandia National Laboratories

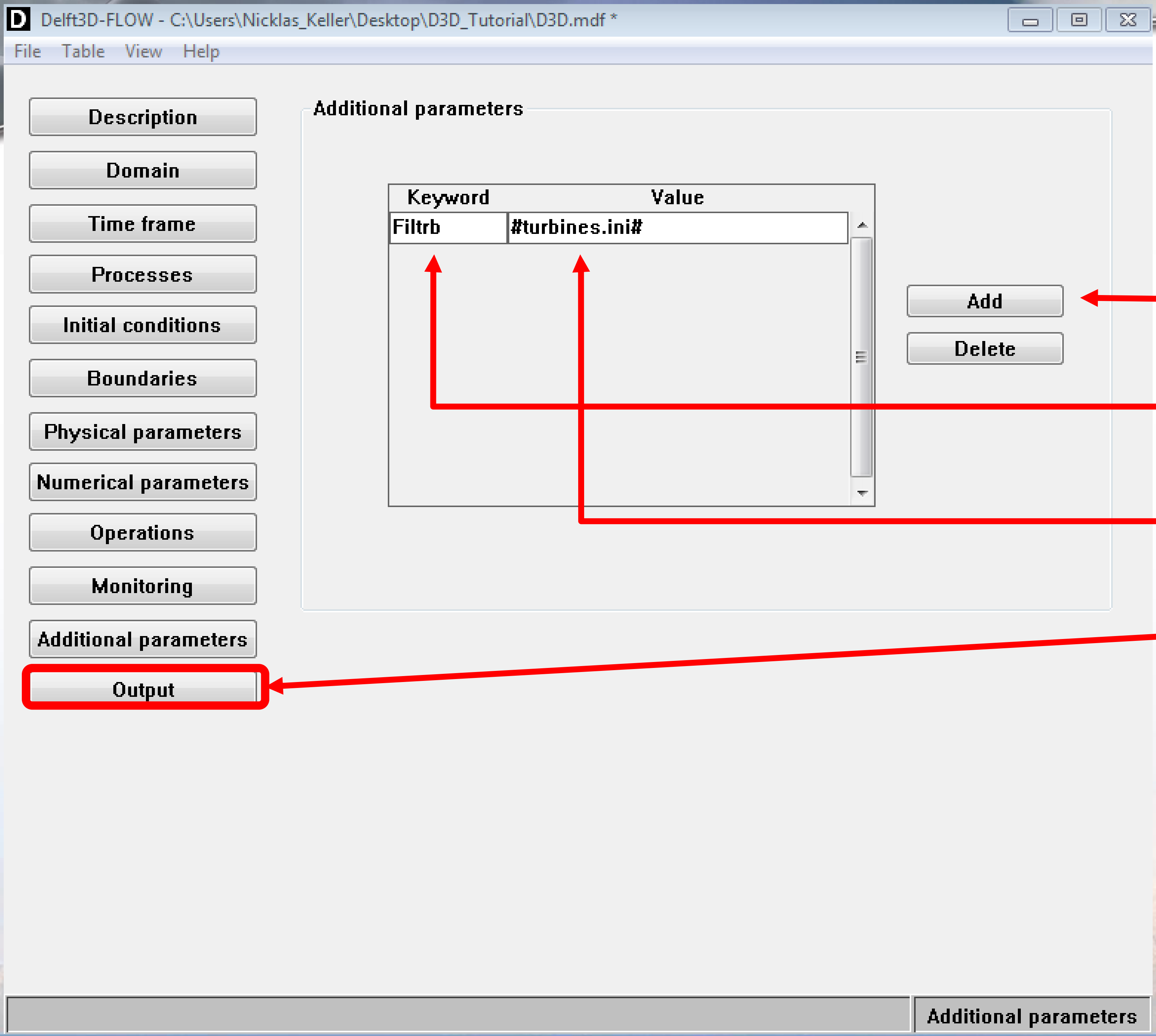
Flow



- Select Add
- Name this monitoring cell Turbine with *M*=17, *N*=7
- Select Save and save as D3D.obs
- Select Additional parameters

Sandia National Laboratories

Additional Parameters (Adding a Turbine)



- Select add
- Set keyword = Filtrb
- Set value = #turbines.ini#
- Select Output
- Next, we will look at the turbines.ini file within a text editor.

Turbines.ini file

turbines.ini - Notepad

File Edit Format View Help

```
[TurbineFileInformation]
    FileVersion   = 01.00
[General]
    CurvesFil     = #curves.trb#
[Turbine]
    Name          = #Tutorial Turbine#
    Diameter      = 5
    XYLoc         = 176,56
    Orientation   = 180
    VertPos       = #fixed#
    AxisLevel     = -4.5
    ThrustCurve   = #Turbine Type 1#
    PowerCurve    = #Turbine Type 1#
    NDiaDist4Vel  = 1.0
    Beta_p        = .05
    Beta_d        = 1.2
    Cep4          = 4
    Cep5          = 5.2
    TurbineModel     = 1
    TurbulenceModel = 1
```

- Copy the “turbines.ini” and “curves.trb” files located in the sample file directory into the D3D_Tutorial directory you’ve created.
- Although no modifications are necessary for this case, we will now walk through the components of the turbines.ini file.
- The turbines.ini file positions the turbine, sets the turbulence model, and allows for additional turbulence control.
- Turbines.ini is an ASCII file which can be edited in a text editor.
- The curves.trb file contains the power (C_P) and thrust (C_T) coefficients for the turbine.

Turbines.ini file

turbines.ini - Notepad

File Edit Format View Help

```
[TurbineFileInformation]
    FileVersion   = 01.00
[General]
    CurvesFil     = #curves.trb#
[Turbine]
    Name          = #Tutorial Turbine#
    Diameter      = 5
    XYLoc         = 176,56
    Orientation   = 180
    VertPos       = #fixed#
    AxisLevel     = -4.5
    ThrustCurve   = #Turbine Type 1#
    PowerCurve    = #Turbine Type 1#
    NDiaDist4Vel  = 1.0
    Beta_p        = .05
    Beta_d        = 1.2
    Cep4          = 4
    Cep5          = 5.2
    TurbineModel     = 1
    TurbulenceModel = 1
```

- “CurvesFil” links to curves.trb file specified.
- “Name” is any name you choose.
- Sets the location of the turbine
  - “fixed” is a constant (x,y,z) location
  - “floating” has a fixed x and y but z moves with sigma layer (see Delft 3D manual for sigma layer)
- “AxisLevel” sets hub height of turbine.
  - Water level is z = 0 m, positive above water surface.

Turbines.ini file

turbines.ini - Notepad

File Edit Format View Help

```
[TurbineFileInformation]
    FileVersion    = 01.00
[General]
    CurvesFil      = #curves.trb#
[Turbine]
    Name           = #Tutorial Turbine#
    Diameter       = 5
    XYLoc          = 176,56
    Orientation    = 180
    VertPos        = #fixed#
    AxisLevel      = -4.5
    ThrustCurve    = #Turbine Type 1#
    PowerCurve     = #Turbine Type 1#
    NDiaDist4Vel   = 1.0
    Beta_p         = .05
    Beta_d         = 1.2
    Cep4           = 4
    Cep5           = 5.2
    TurbineModel      = 1
    TurbulenceModel = 1
```

- ThrustCurve and PowerCurve reference “table-name” in curves.trb file.

Turbines.ini file

turbines.ini - Notepad

File Edit Format View Help

```
[TurbineFileInformation]
    FileVersion   = 01.00
[General]
    CurvesFil     = #curves.trb#
[Turbine]
    Name          = #Tutorial Turbine#
    Diameter      = 5
    XYLoc         = 176,56
    Orientation   = 180
    VertPos       = #fixed#
    AxisLevel     = -4.5
    ThrustCurve   = #Turbine Type 1#
    PowerCurve    = #Turbine Type 1#
    NDiaDist4Vel  = 1.0
    Beta_p        = .05
    Beta_d        = 1.2
    Cep4          = 4
    Cep5          = 5.2
    TurbineModel     = 1
    TurbulenceModel = 1
```

- “NDiaDist4Vel” is Number of Diameters Distance for velocity
- If TurbineModel = 0, the momentum extraction of the turbine is calculated using an upstream reference velocity determined by NDiaDist4Vel
- If TurbineModel = 1, the momentum extraction of the turbine is calculated using velocity at turbine and NDiaDist4Vel is ignored.
  - This option is useful for one turbine and when using turbine arrays where upstream velocity is not well-defined

Turbines.ini file

turbines.ini - Notepad

File Edit Format View Help

```
[TurbineFileInformation]
    FileVersion   = 01.00
[General]
    CurvesFil     = #curves.trb#
[Turbine]
    Name          = #Tutorial Turbine#
    Diameter      = 5
    XYLoc         = 176,56
    Orientation   = 180
    VertPos       = #fixed#
    AxisLevel     = -4.5
    ThrustCurve   = #Turbine Type 1#
    PowerCurve    = #Turbine Type 1#
    NDiaDist4Vel  = 1.0
    Beta_p        = .05
    Beta_d        = 1.2
    Cep4          = 4
    Cep5          = 5.2
    TurbineModel     = 1
    TurbulenceModel = 1
```

- TurbineModel = 0, Upstream velocity turbine model.
  - $F_t = \frac{1}{2}\rho A C_T U_\infty^2$
  - Where the force of the turbine is a function of density, turbine area, coefficient of thrust, and ambient upstream velocity.
- TurbineModel = 1, disc velocity turbine model.
  - $F_t = \frac{1}{2}\rho A C_T' U_d^2$
  - $C_T' = 4\frac{1-\sqrt{1-C_T}}{1+\sqrt{1-C_T}}$
  - Where the force of the turbine is a function of density, turbine area, modified coefficient of thrust, and velocity at the turbine (disk).

Roc, T., Conley, D. C., & Greaves, D. (2013). Methodology for tidal turbine representation in ocean circulation model. Renewable Energy, 51, 448-464.)

Turbines.ini file

turbines.ini - Notepad

File Edit Format View Help

```
[TurbineFileInformation]
    FileVersion    = 01.00
[General]
    CurvesFil      = #curves.trb#
[Turbine]
    Name           = #Tutorial Turbine#
    Diameter       = 5
    XYLoc          = 176,56
    Orientation    = 180
    VertPos        = #fixed#
    AxisLevel      = -4.5
    ThrustCurve    = #Turbine Type 1#
    PowerCurve     = #Turbine Type 1#
    NDiaDist4Vel   = 1.0
    Beta_p         = .05
    Beta_d         = 1.2
    Cep4           = 4
    Cep5           = 5.2
    TurbineModel      = 1
    TurbulenceModel  = 1
```

- TurbulenceModel = 0 no turbulence source or sink is applied at the turbine.
  - β_P, β_D, $C_{\varepsilon 4}$, $C_{\varepsilon 5}$ are ignored

- TurbulenceModel = 1 uses Rethore turbulence model
  - β_P, β_D, $C_{\varepsilon 4}$, $C_{\varepsilon 5}$ adjust source terms added to k-ε model at turbine location as follows:
  - $S_k = \frac{1}{2} C_T' (\beta_P U^3 - \beta_D\, U k)$
  - $S_\varepsilon = \frac{1}{2} C_T' \left(C_{\varepsilon 4} \beta_P \frac{\varepsilon}{k} U^3 - C_{\varepsilon 5} \beta_D\, U \varepsilon \right)$
  - $C_T' = 4 \frac{1-\sqrt{1-C_T}}{1+\sqrt{1-C_T}}$
  - Where U is velocity, k is turbulent kinetic energy (tke), ε is dissipation of tke, C_T' is the modified coefficient of thrust.

Rethore, P. E. M., Sørensen, N. N., Bechmann, A., & Zahle, F. (2009, March). Study of the atmospheric wake turbulence of a CFD actuator disc model. In 2009 European Wind Energy Conference and Exhibition.

Curves.trb file

curves.trb - Notepad

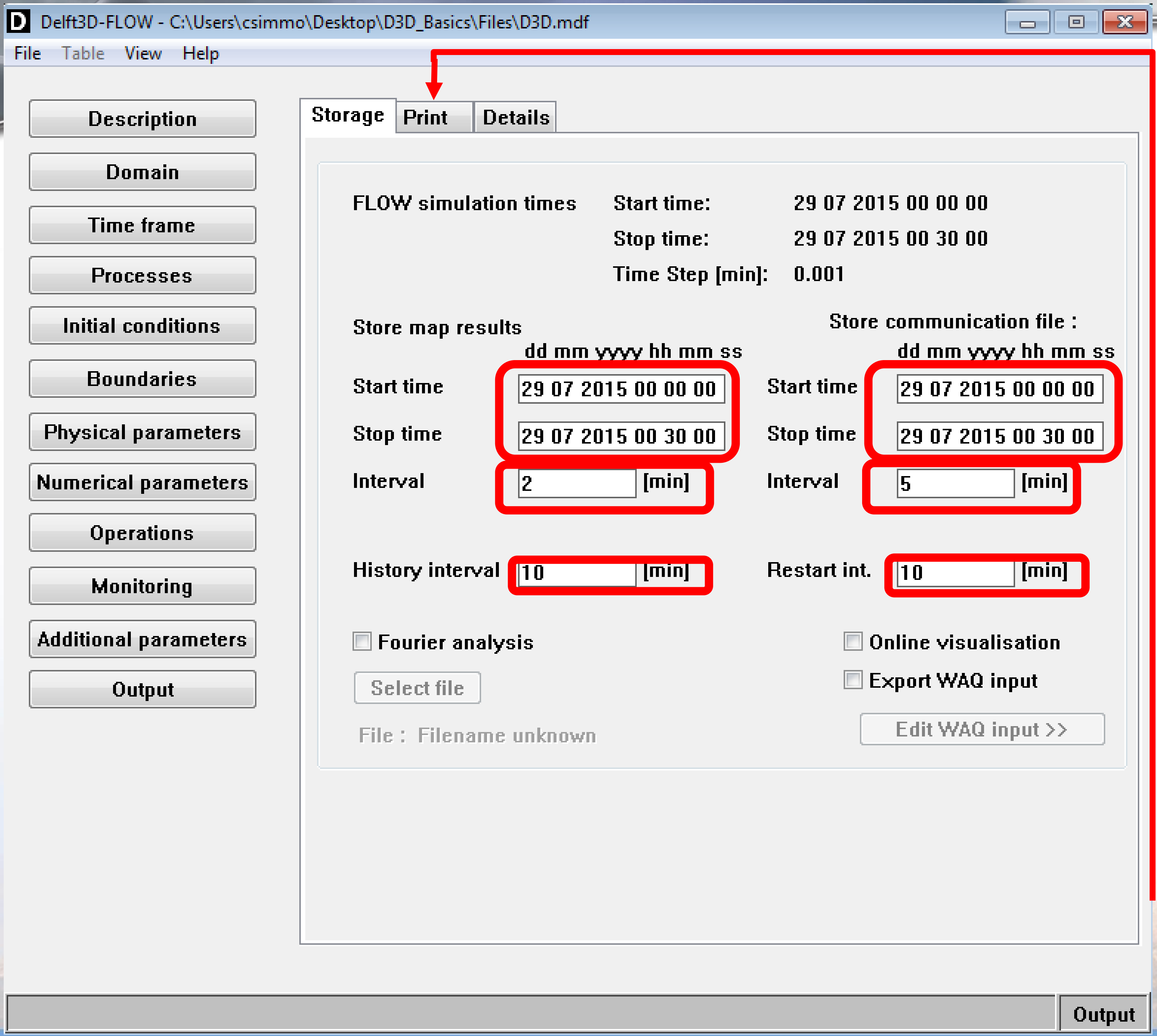
File Edit Format View Help

```
table-name                'Turbine Type 1     '
parameter                 'velocity           '                    unit '[m/s]'
parameter                 'thrust coefficient '                    unit '[ - ]'
parameter                 'power coefficient  '                    unit '[ - ]'
-99.0 .90264481   .9
-3.0  .90264481   .9
-2.0  .90264481   .7
-1.0  .90264481   .5
 0.0  .90264481   .1
 1.0  .90264481   .5
 2.0  .90264481   .7
 3.0  .90264481   .9
99.0  .90264481   .9
```

u(m/s) C_T C_P

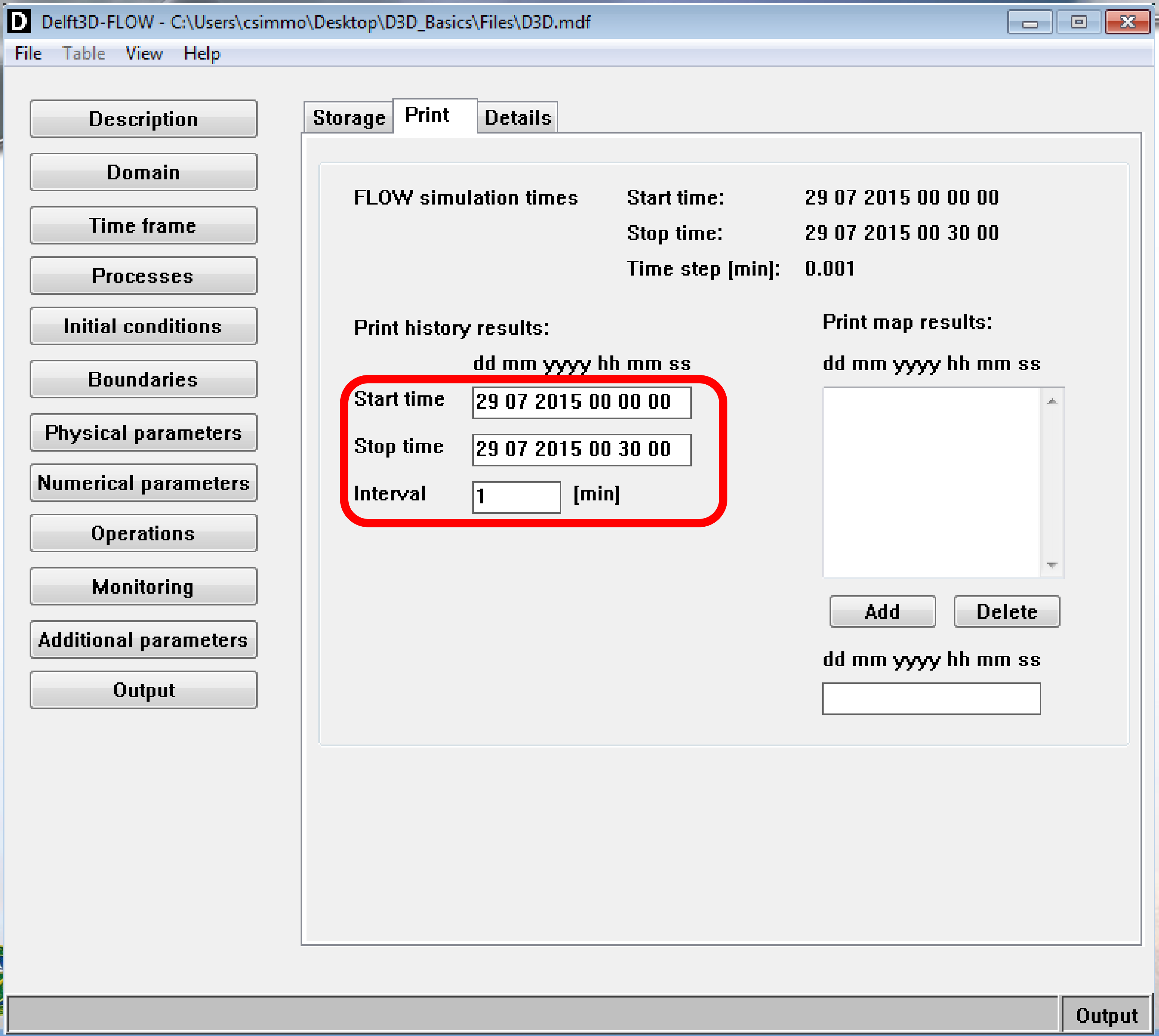
- After opening curves.trb, make sure “table-name” matches the values specified for ThrustCurve and PowerCurve found in turbines.ini.
- First column matches with first parameter ‘velocity’, second column with second parameter ‘thrust coefficient’ and third column with third parameter ‘power coefficient’.
- Each row of data represents velocity with corresponding thrust and power coefficients at that velocity.
- In this example, C_T is constant while C_P varies.

Data Output

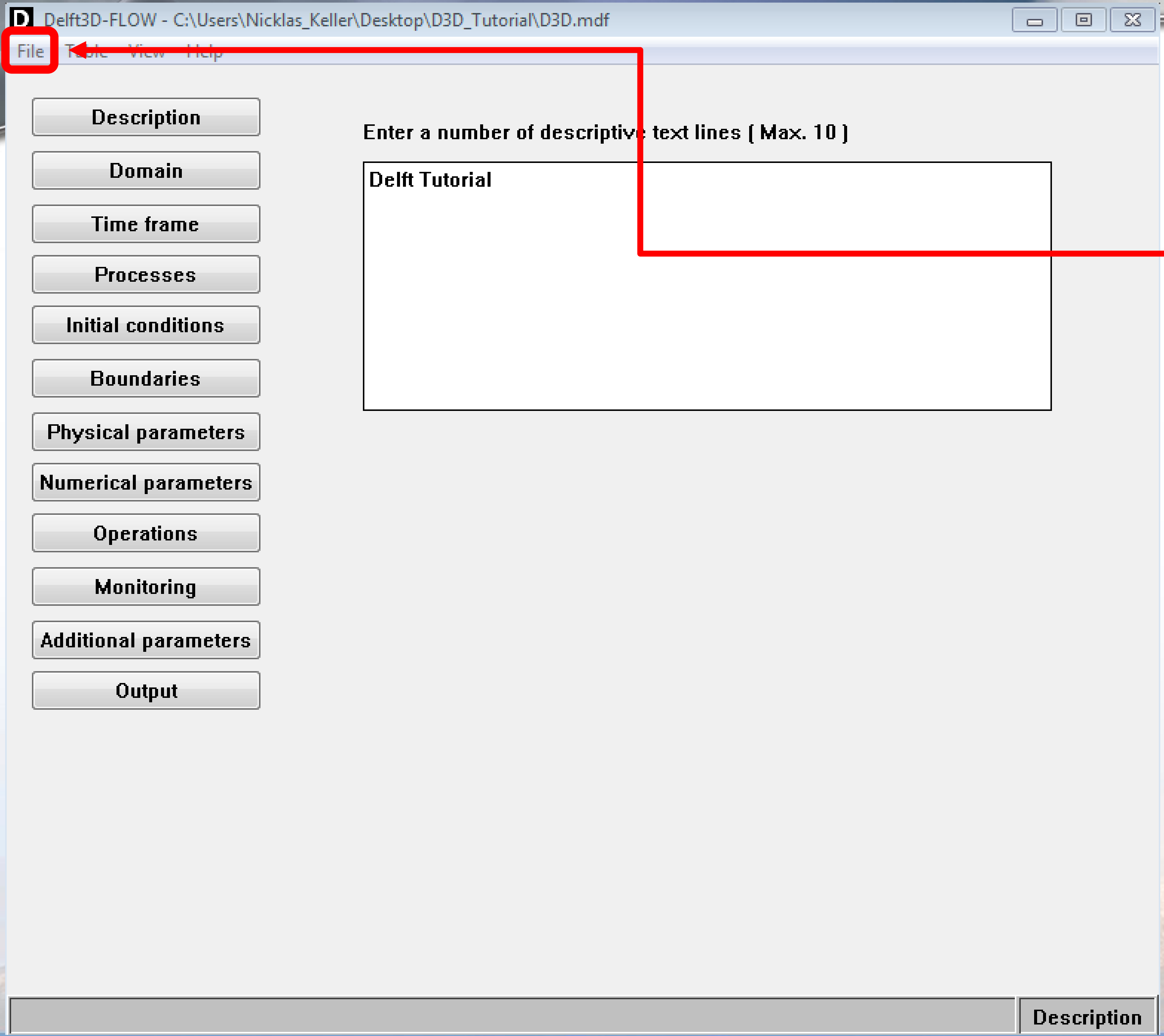


- Return to the D3D Flow GUI and select the Output box.
- Make sure the start and stop times are indicated to run for 30 minutes
- Assign these values.
- Store map: interval = 2 min, history interval=10 min
- Store communication file: interval = 5 min, restart int.= 10 min.
- Select Print tab

Data Output



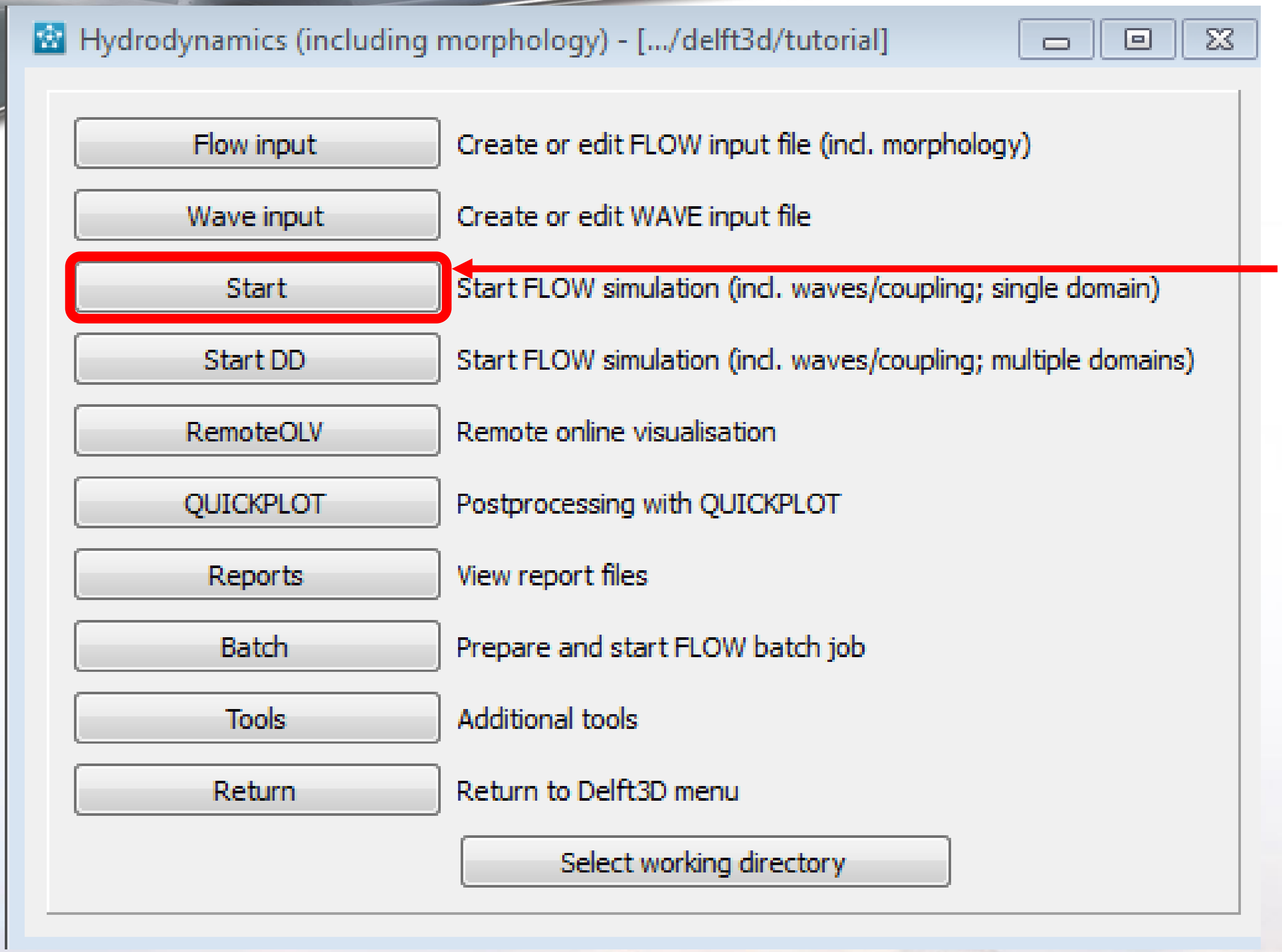
- Make sure start and stop time are set for 30 minutes
- Set Print History interval to 1 minute



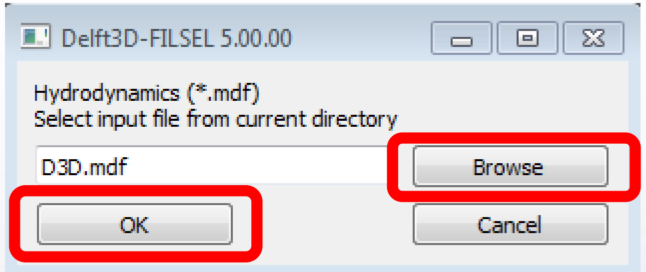
Saving Your File

- Select File>Save MDF As
- Save as “D3D.mdf”

Running a Flow Model

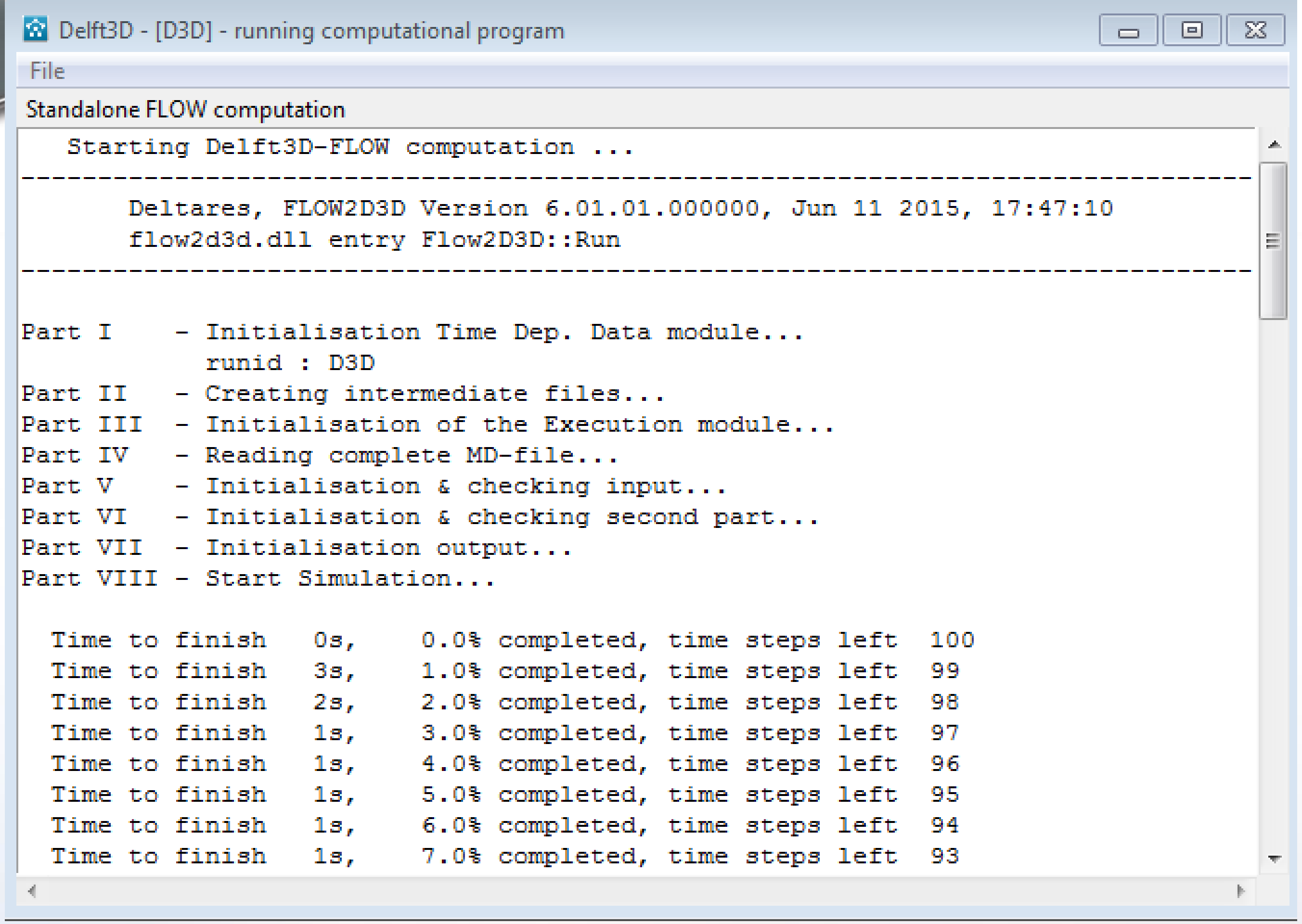


- After saving the MDF, return to the Flow menu
- Select Start



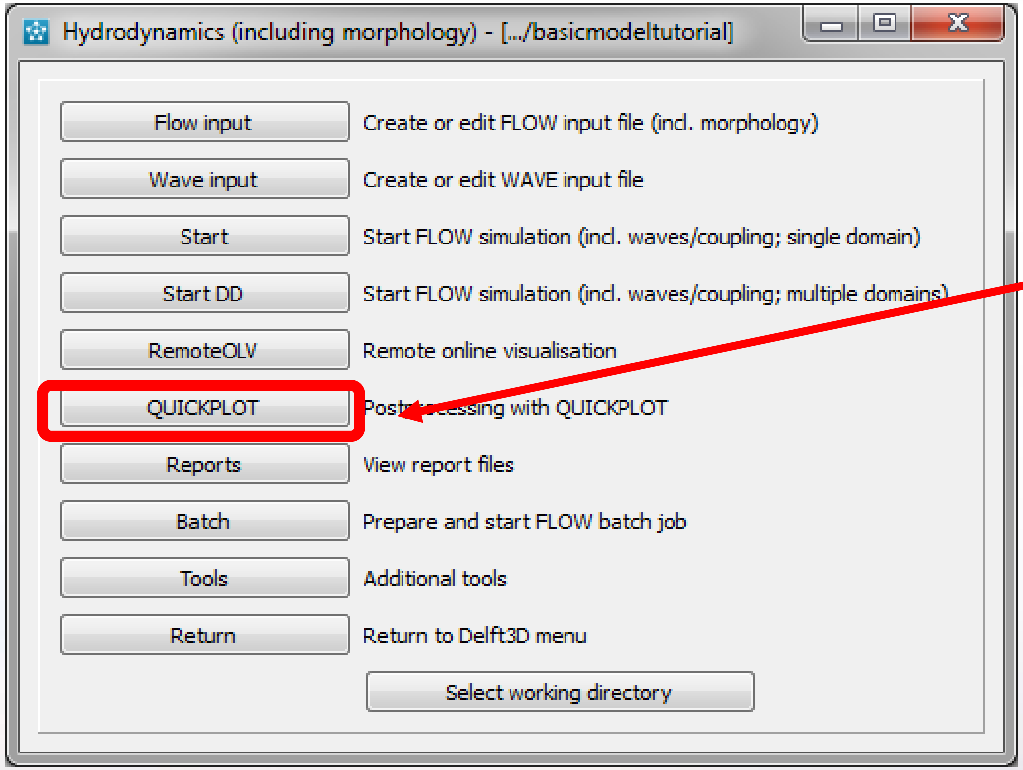
- Browse to find the correct .mdf file
- Select OK

Running a Flow Model



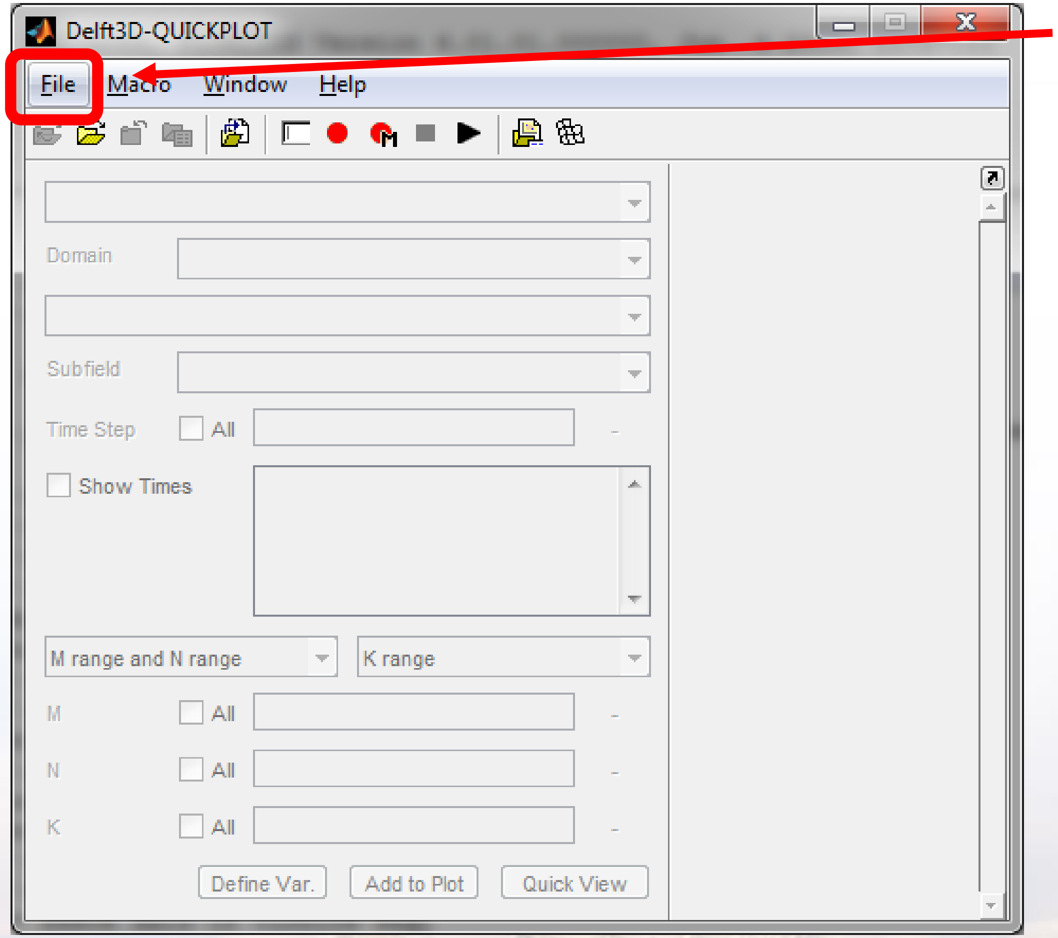
- At this point, the model should be running. Run time: ~3 min.
- Upon successful completion of run, we can now post process this information.
- If file fails to get through Parts I-VIII, check directory for td-diag-casename and tri-diag-casename, where casename is the name of your .mdf file
- tri-diag-casename is always available and displays simulation summaries.

Post Processing with QUICKPLOT



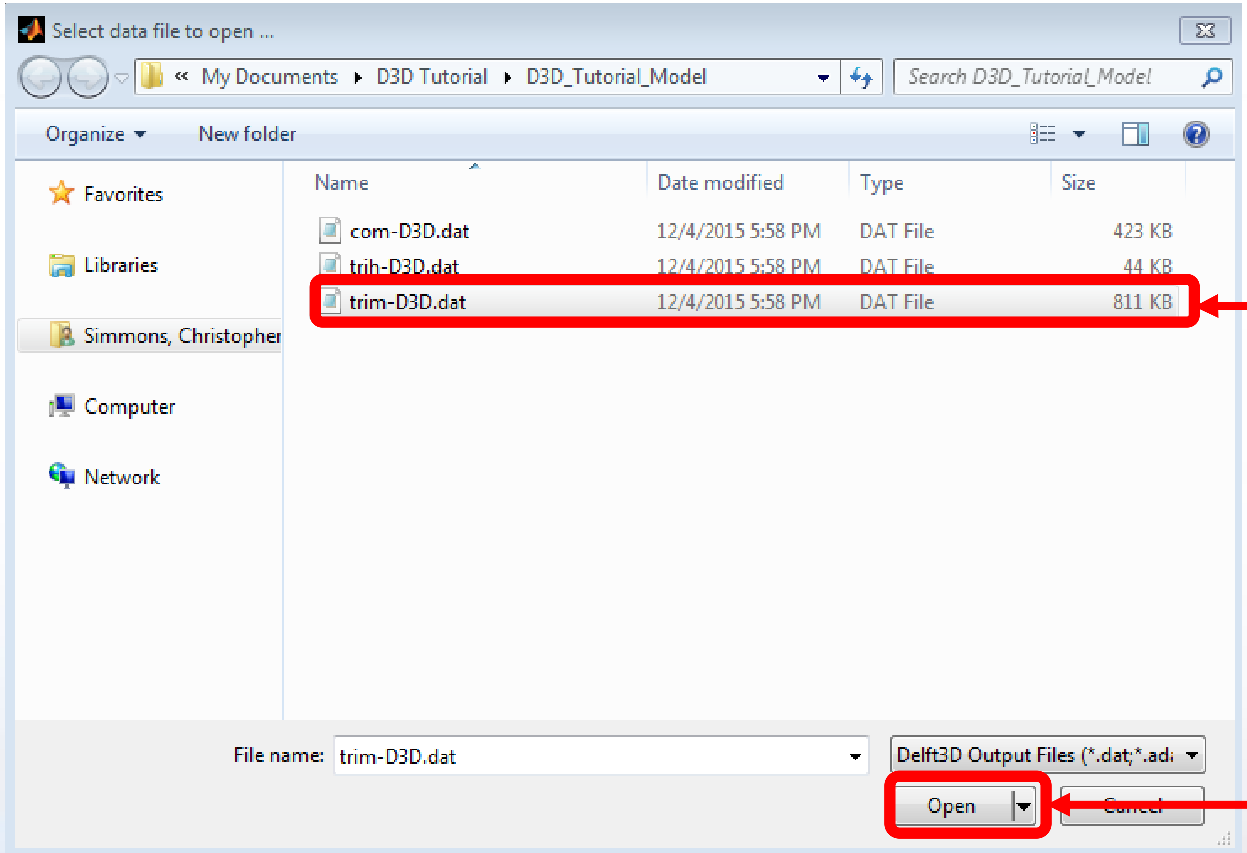
- After the model has completed running, close the window and return to the main menu and click “Utilities”.
- Select QUICKPLOT.
- QUICKPLOT is a post-processing tool that utilizes MATLAB plotting to display various plots.
- MATLAB does not need to be installed on your computer in order to run QUICKPLOT.

QUICKPLOT



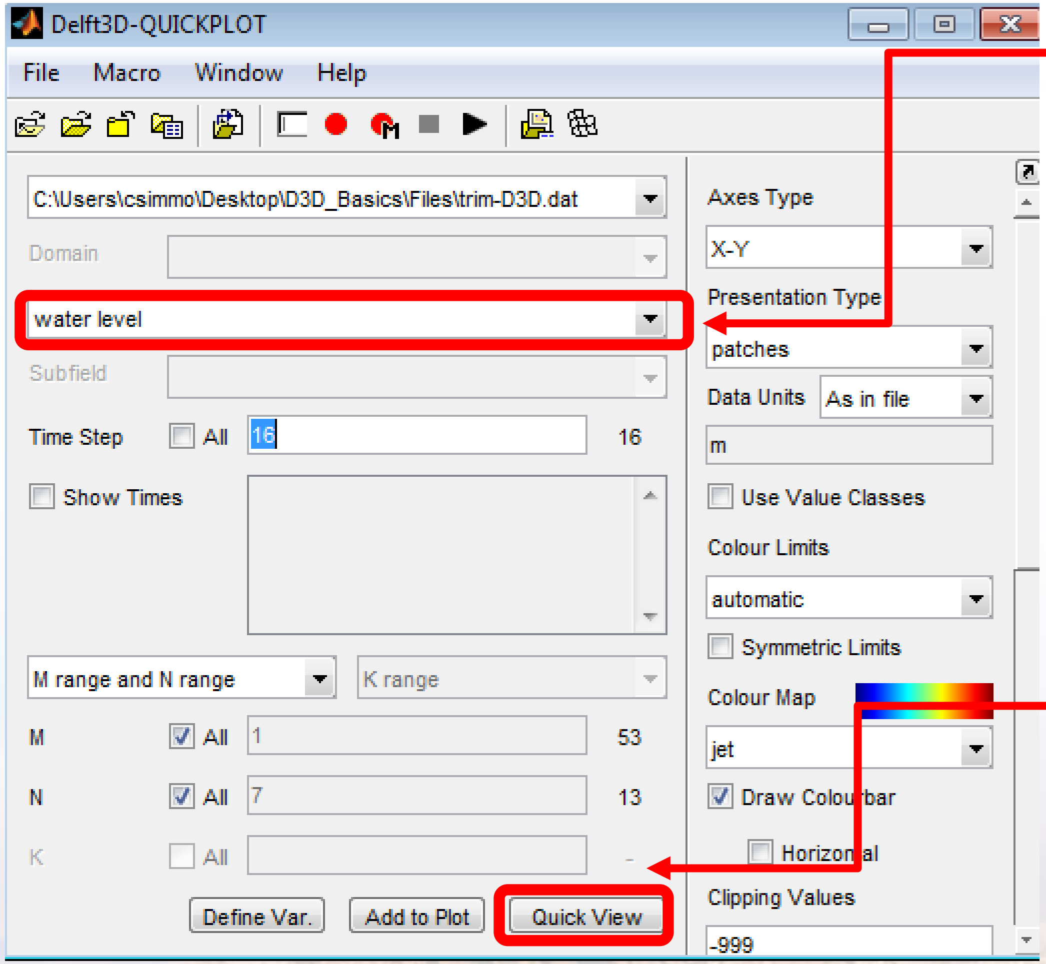
- Select File
- Select Open File

QUICKPLOT



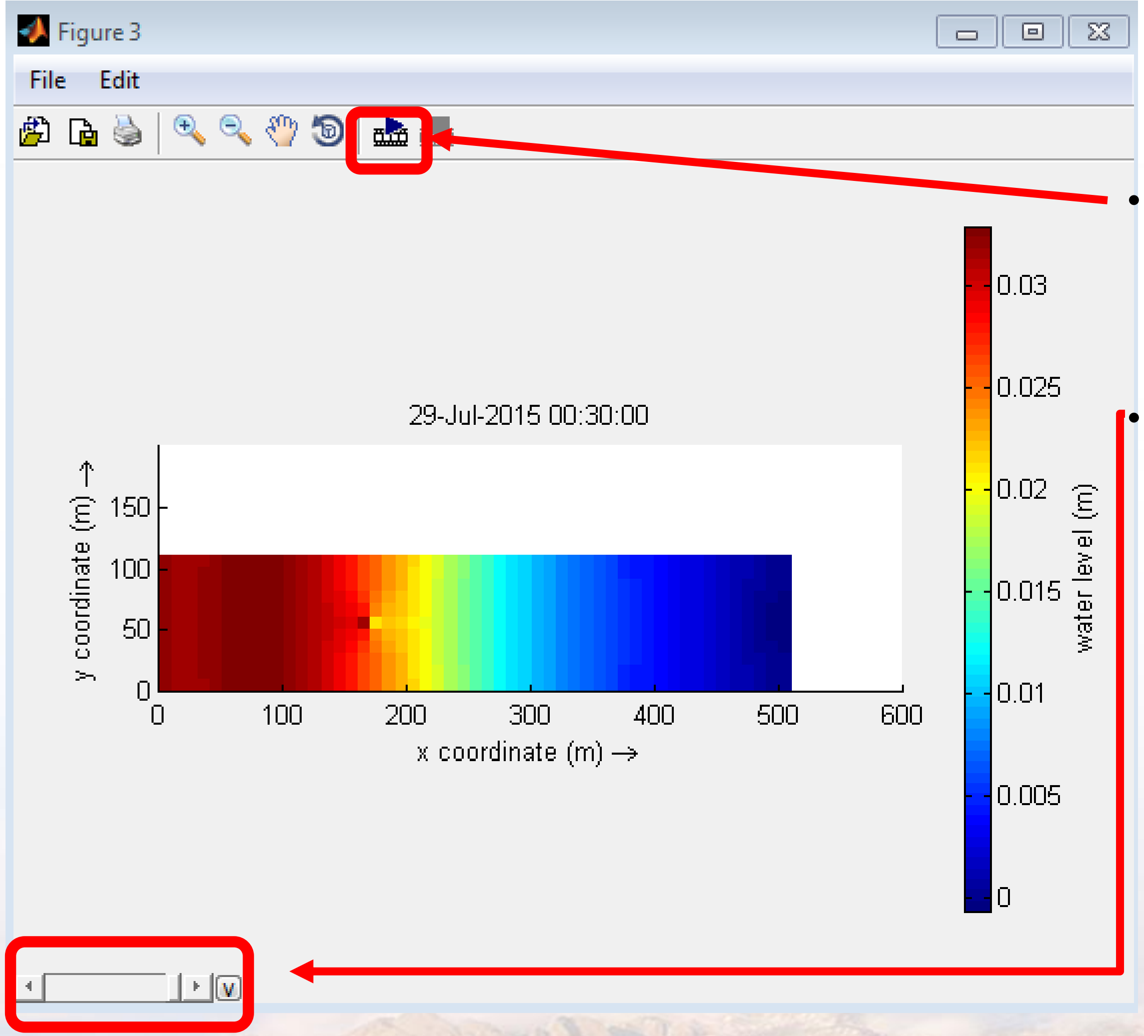
- Open “trim-D3D.dat”
- Be sure to open the trim-D3D.dat rather than the trih-D3D.dat.
- The “trim-D3D.dat” file contains output data at all cell nodes and faces written out at each time step specified in the “Store map results” section of the Data Output.
- Since we specified a two minute map storage interval over a 30 minute simulation, there will be 15 (30/2) time steps plus initial time step.

QUICKPLOT – Water Level Contour



- To view the water level at the final time step, select water level from the drop down box.
  - The highest time step was the last recorded solution.
- Select Quick View

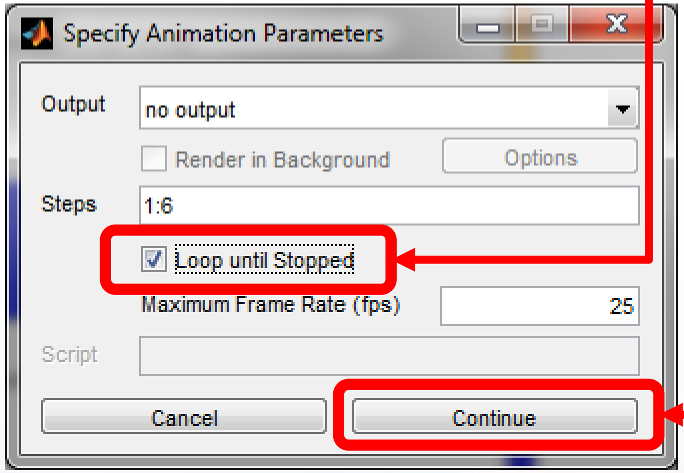
QUICKPLOT – Water Level Contour



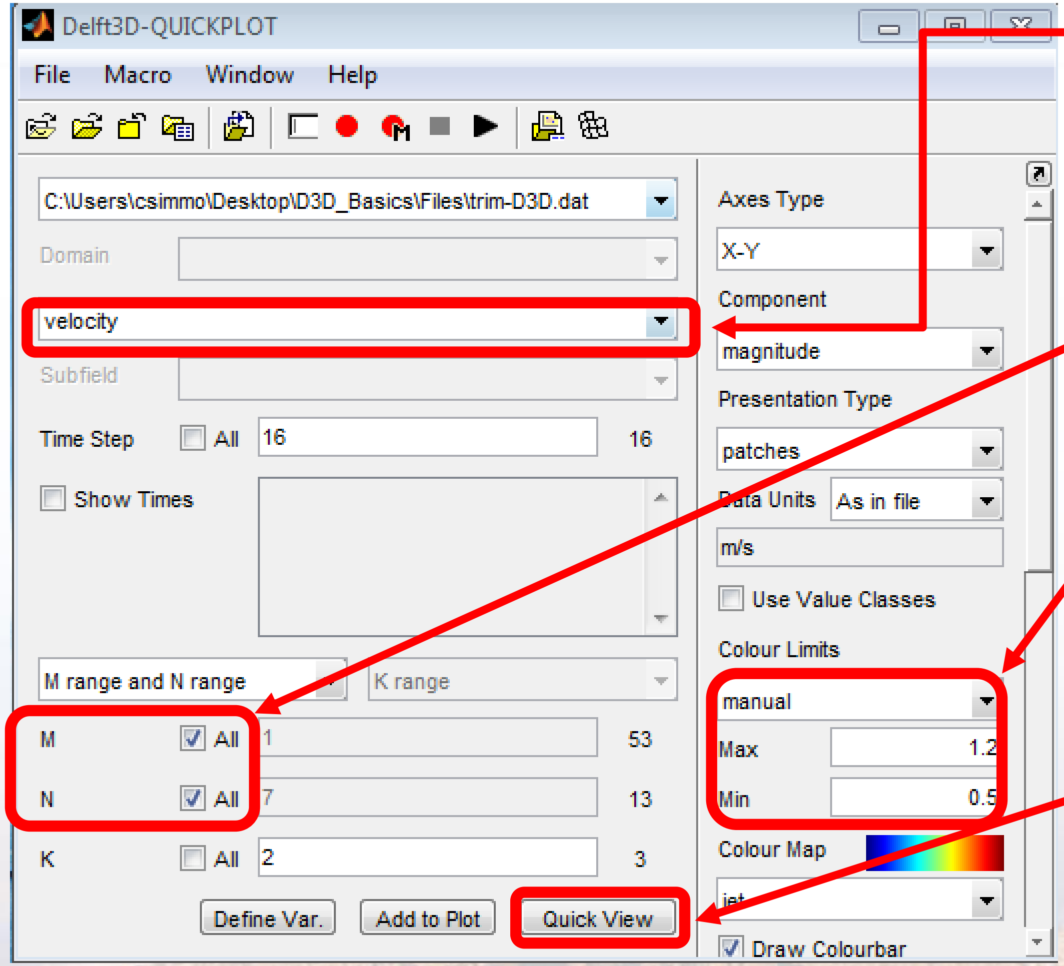
- Select Specify Animation Parameters to animate.
- This bar allows viewing at different times.

QUICKPLOT – Water Level Contour

- Check Loop until Stopped
- Select Continue

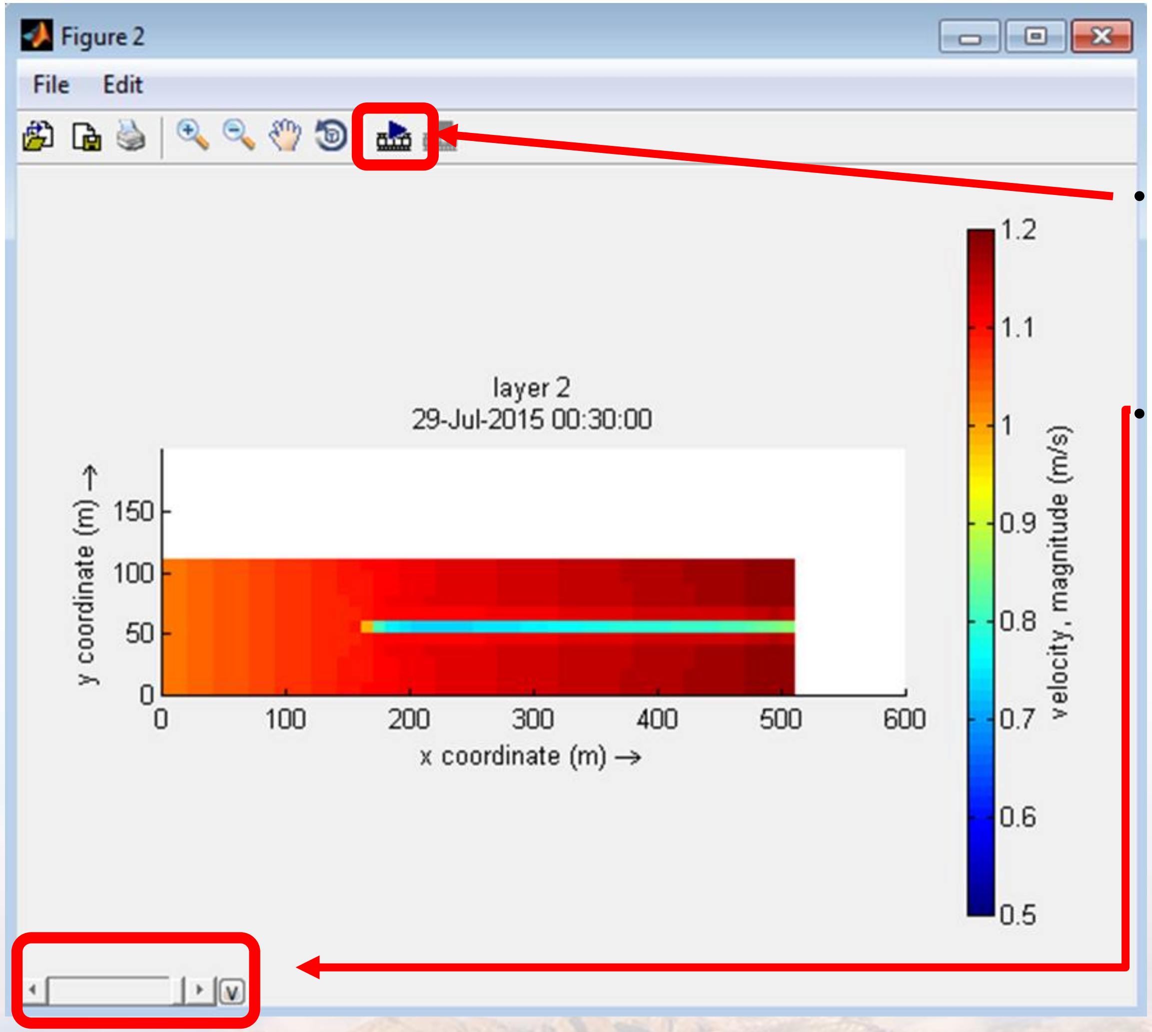


QUICKPLOT – Velocity Contour



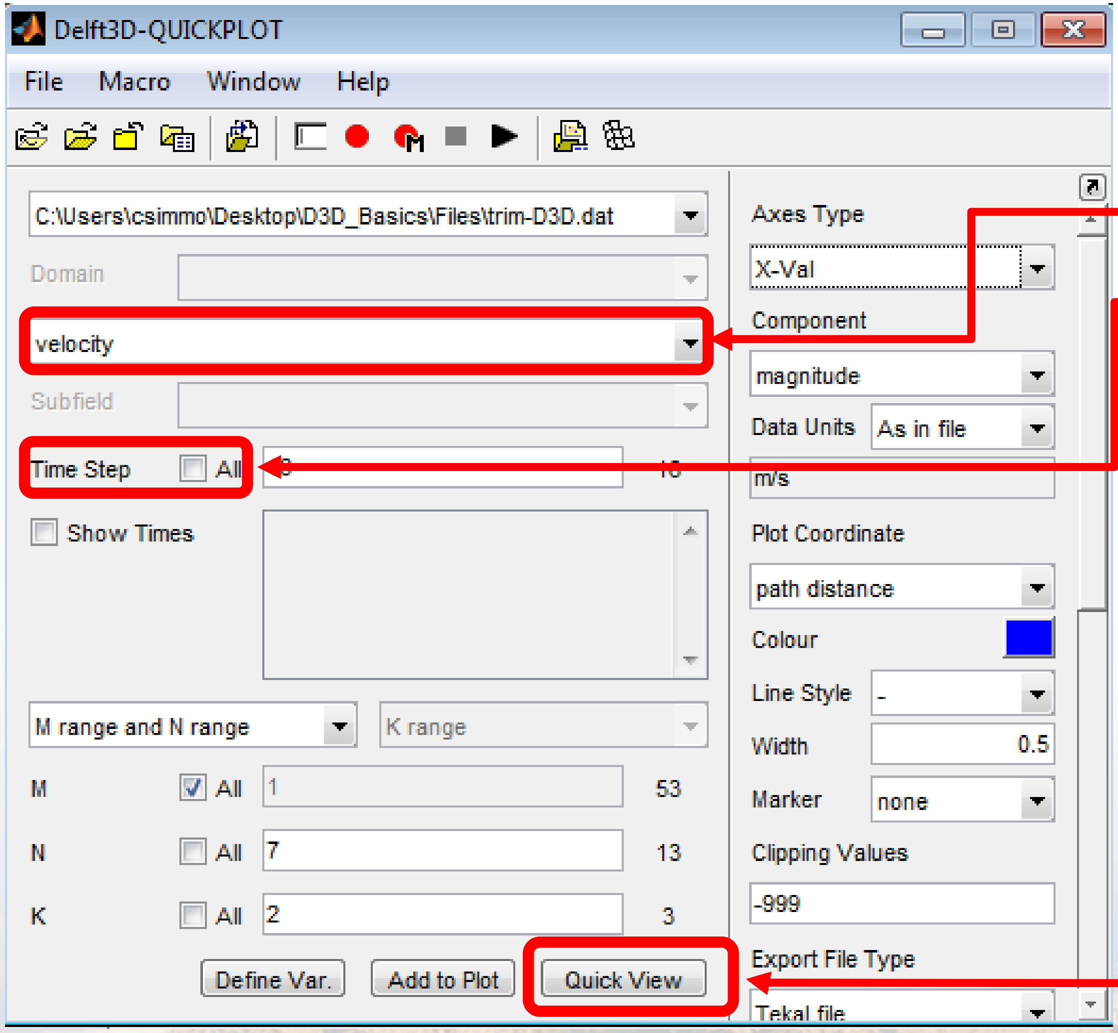
- To view the velocity, select velocity from the drop down box, and uncheck Time Step.
- Select all M and all N.
- Manually enter max and min color limits.
- Select Quick View

QUICKPLOT – Velocity Contour



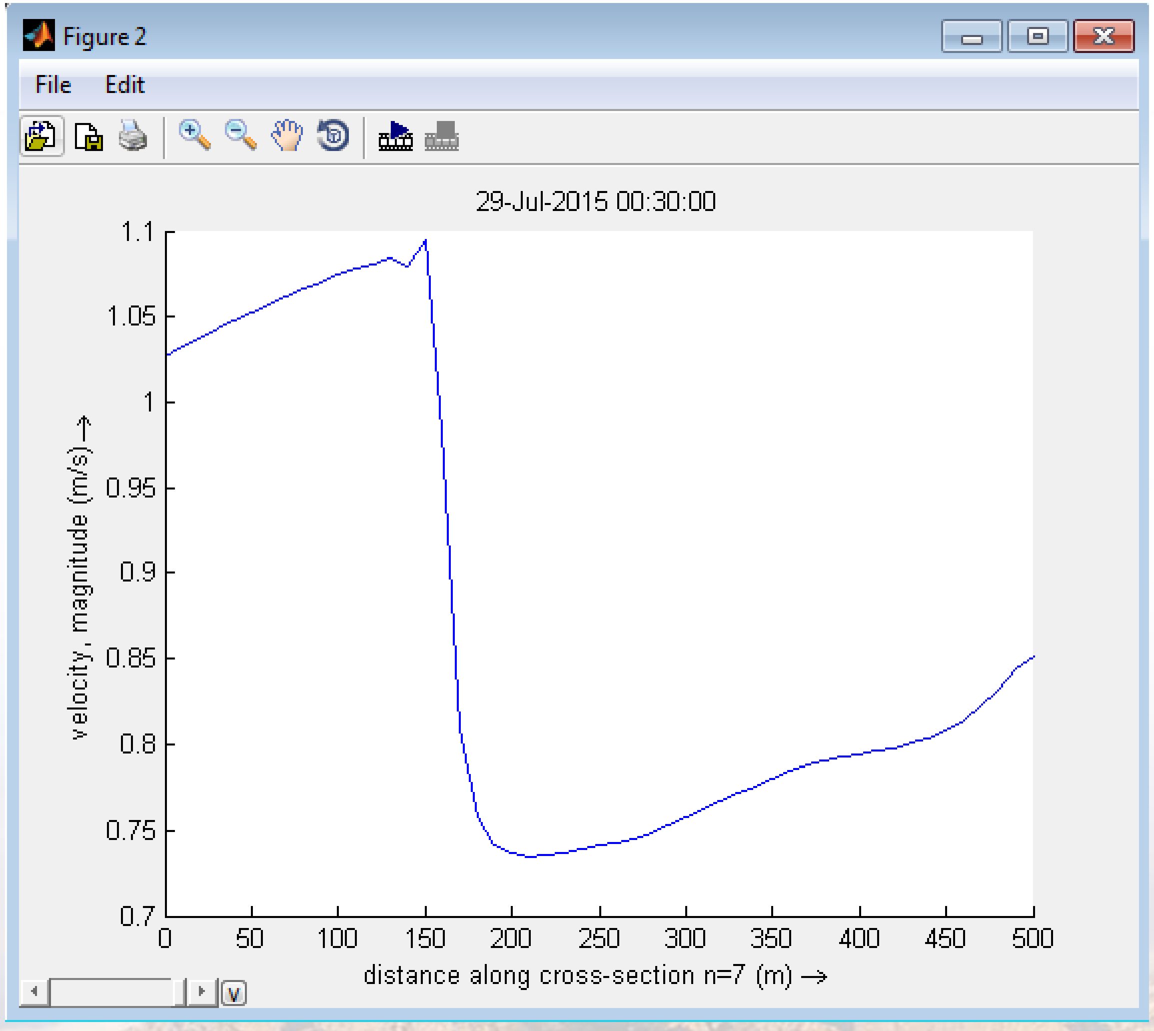
- Select Specify Animation Parameters to animate.
- This bar allows viewing at different times.

QUICKPLOT – Centerline Velocity



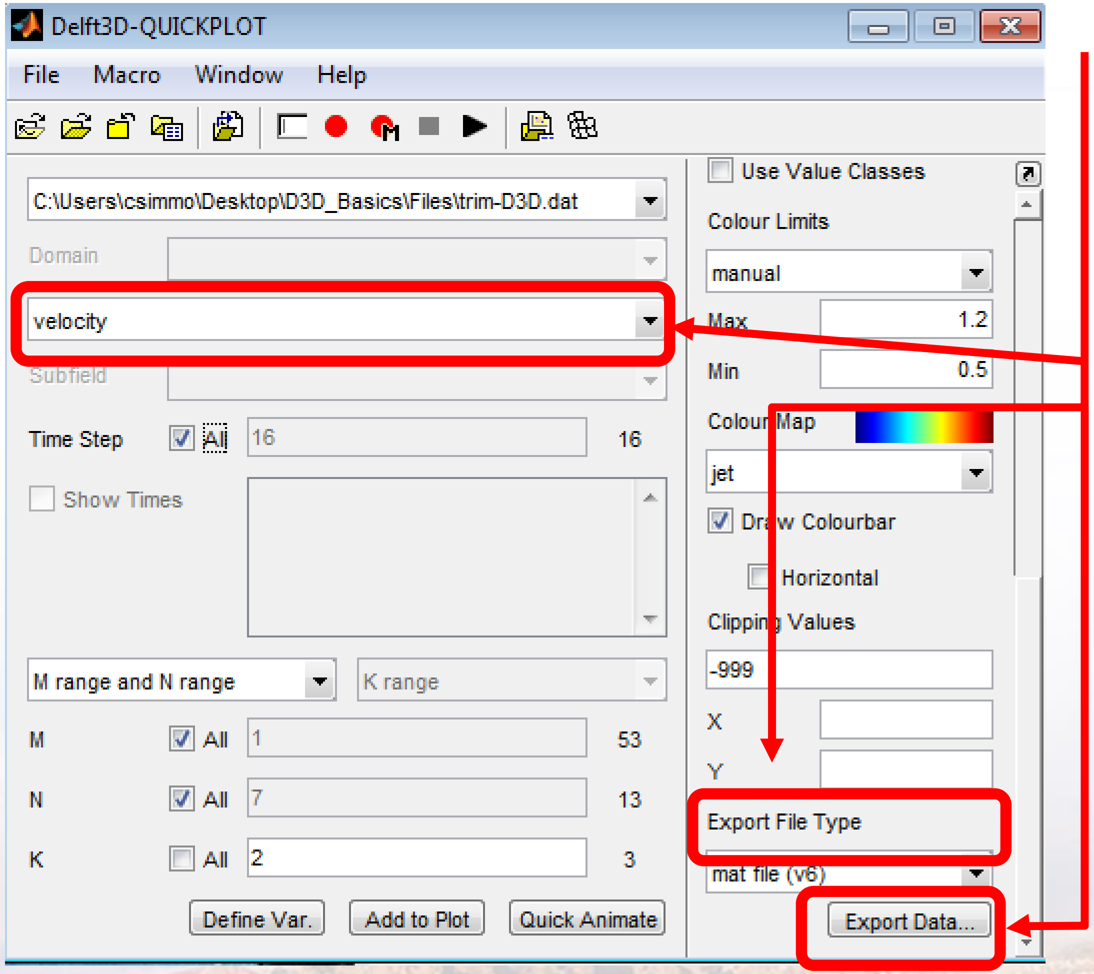
- To view a different plot, such as centerline velocity, return to the QUICKPLOT menu.
- Select velocity
- Uncheck Time Step.
- Choose centerline values for N and K.
- Select Quick View

QUICKPLOT – Centerline Velocity



- Using this button, you can choose to view by time or by layers.

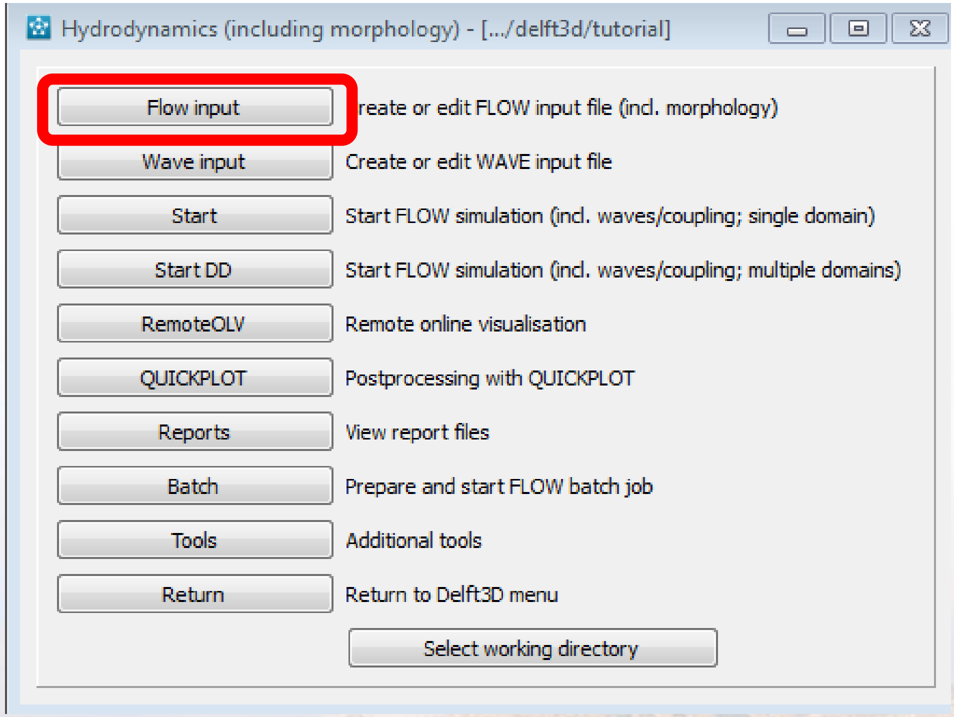
QUICKPLOT

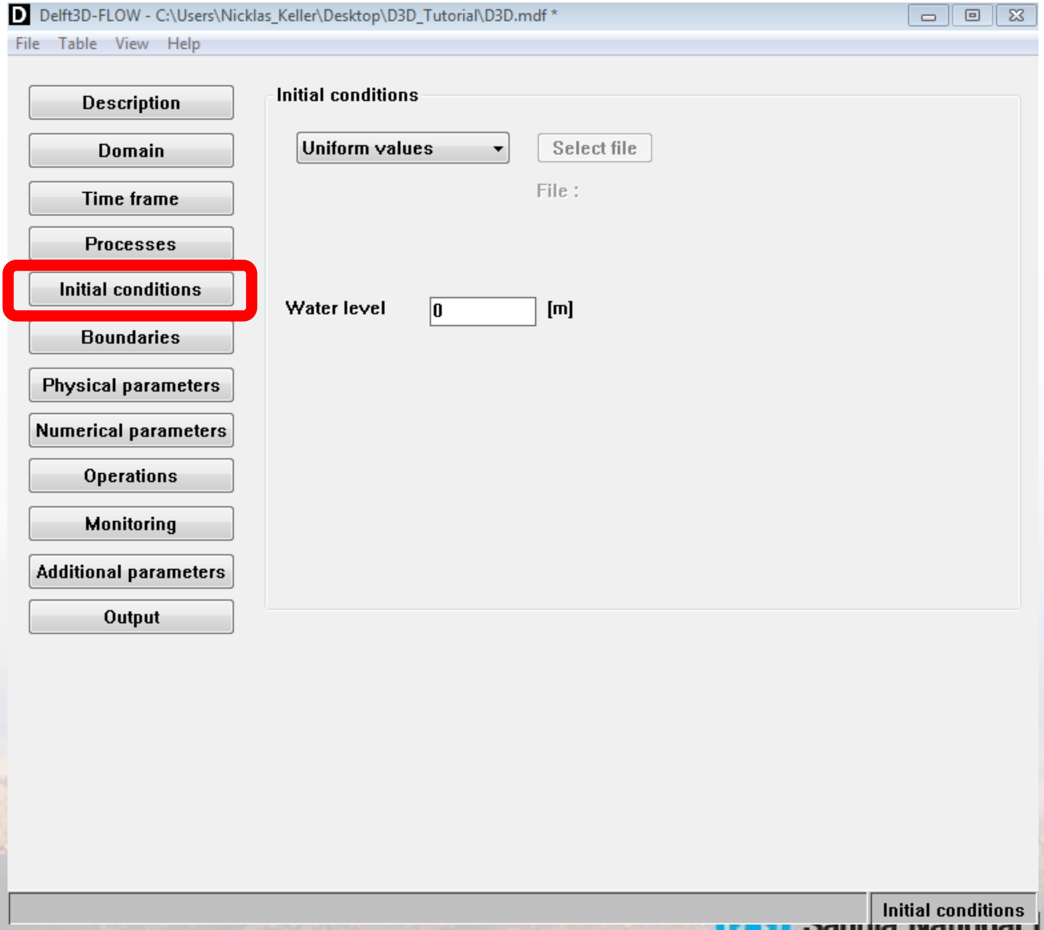


- To export data, choose Export File Type
- Select Export Data…
- QUICKPLOT will export data according to selection in the drop down window.

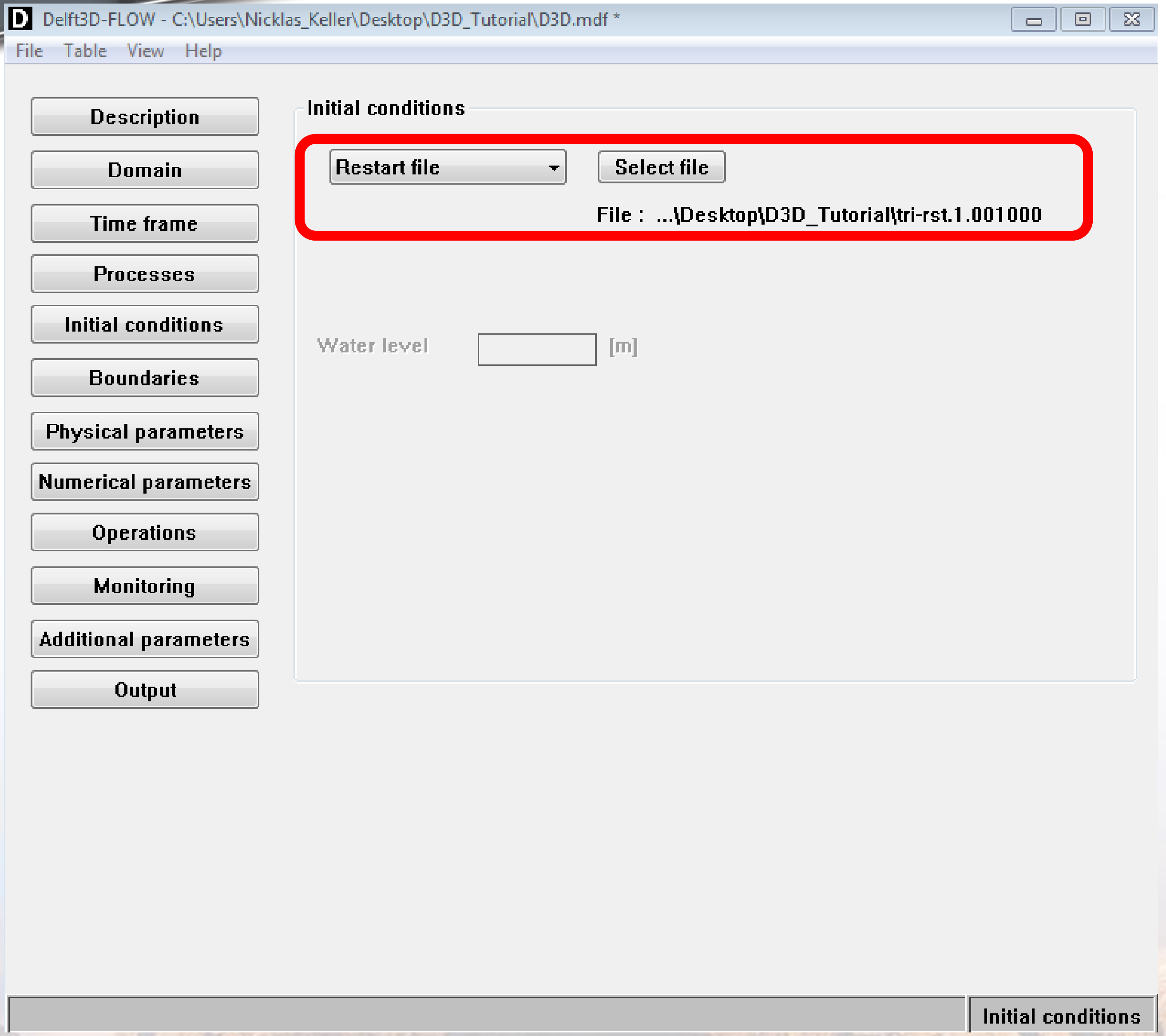
Starting Simulation from Restart File

In order to get the most accurate results, we will first need to establish a restart file that we produced from the first run we made. Return to the flow input menu and select Initial conditions.





Restarting a Simulation



- Change the dropdown menu to "Restart file"
- In the directory where the tutorial is located, there should now be a file that is titled, "tri-rst.D3D…..001000" (This is the restart file for Delft3D)
- Rename this file: tri-rst.1.001000
- Select file tri.rst.1.001000