

# SNL-Delft3D-CEC

3D/2D modeling suite for integral water solutions



## Building a Basic Model

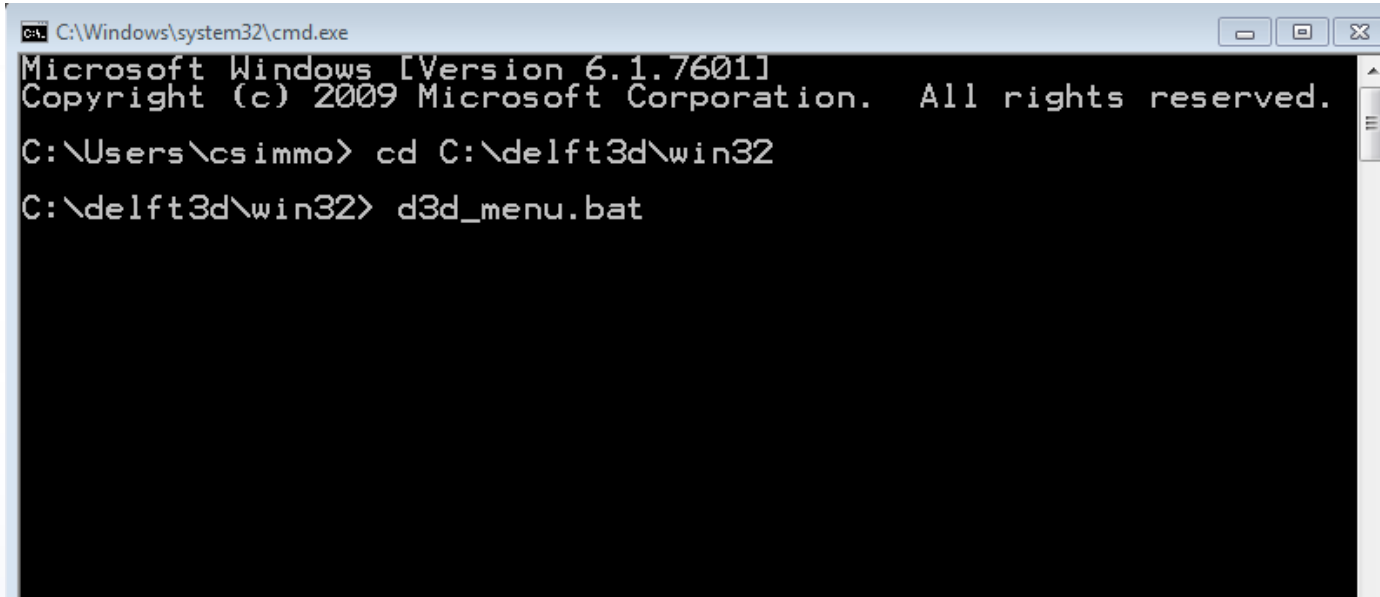


Sandia National Laboratories is a multi-mission laboratory managed and operated by Sandia Corporation, a wholly owned subsidiary of Lockheed Martin Corporation, for the U.S. Department of Energy's National Nuclear Security Administration under contract DE-AC04-94AL85000.



**Sandia National Laboratories**

# Delft3D initialization



```
C:\Windows\system32\cmd.exe
Microsoft Windows [Version 6.1.7601]
Copyright (c) 2009 Microsoft Corporation. All rights reserved.

C:\Users\csimmo> cd C:\delft3d\win32
C:\delft3d\win32> d3d_menu.bat
```

- Using a Windows operation system, start the Delft3D program.
- From the Windows command line, run d3d\_menu.bat

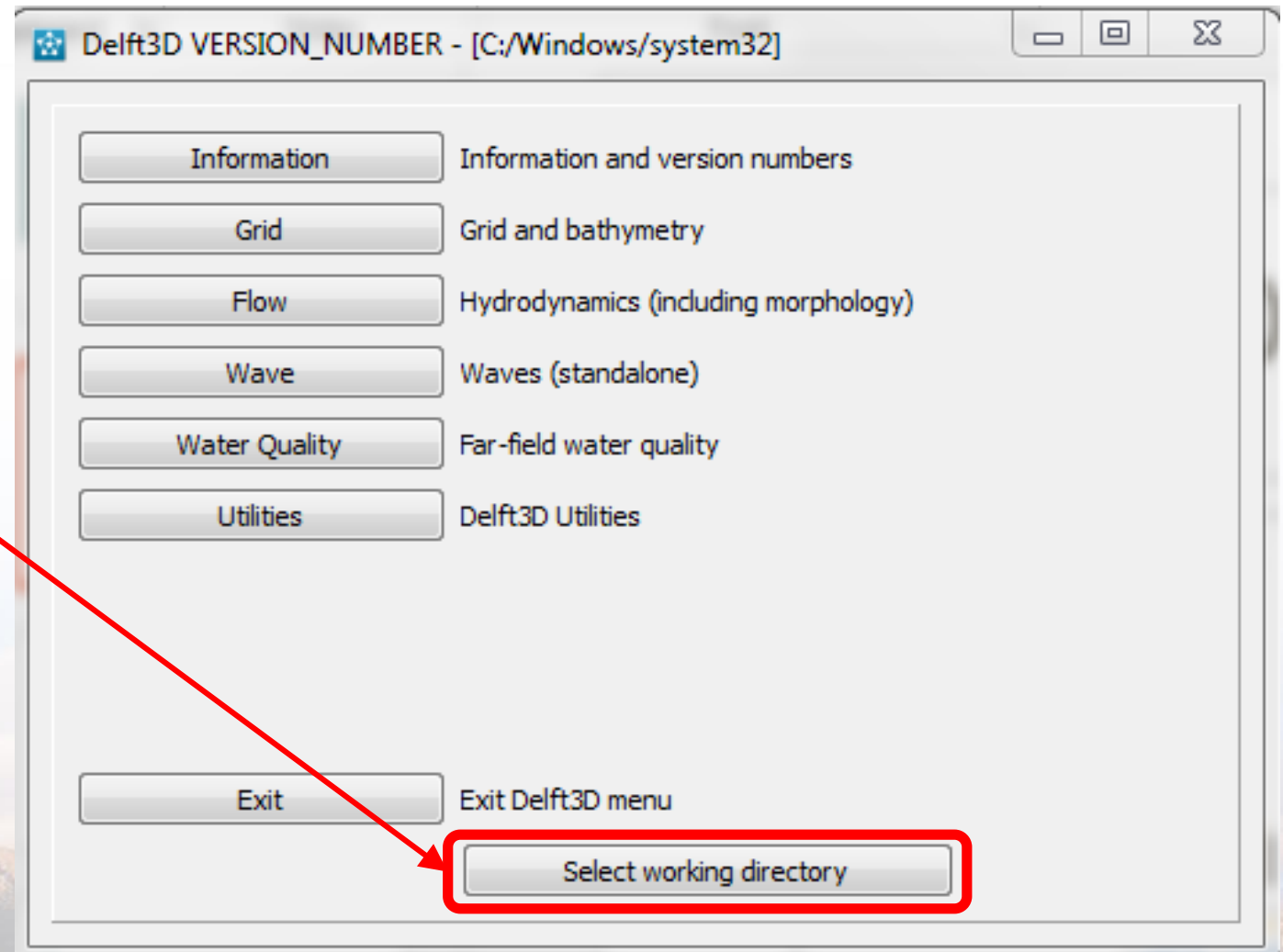
# Delft3D Documentation

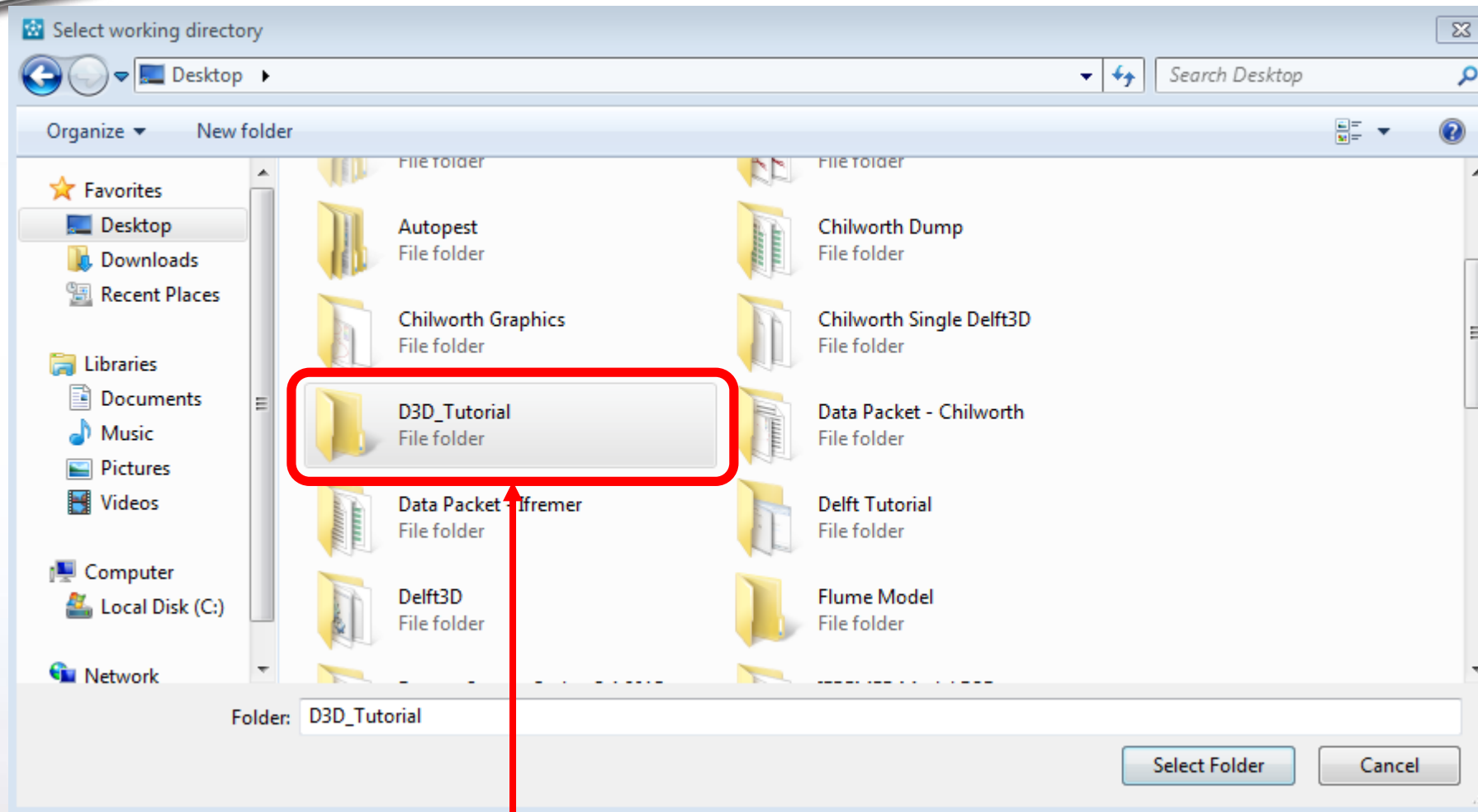
- SNL-Delft3D-CEC is derived from the original Delft3D FLOW module, thus most of D3D usage can be applied.
  - D3D FLOW Manual :  
[http://content.oss.deltares.nl/delft3d/manuals/Delft3D-FLOW\\_User\\_Manual.pdf](http://content.oss.deltares.nl/delft3d/manuals/Delft3D-FLOW_User_Manual.pdf)
- This tutorial will go into the usage of the turbine module implemented specifically in SNL-Delft3D-CEC.



# Setting Things Up

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



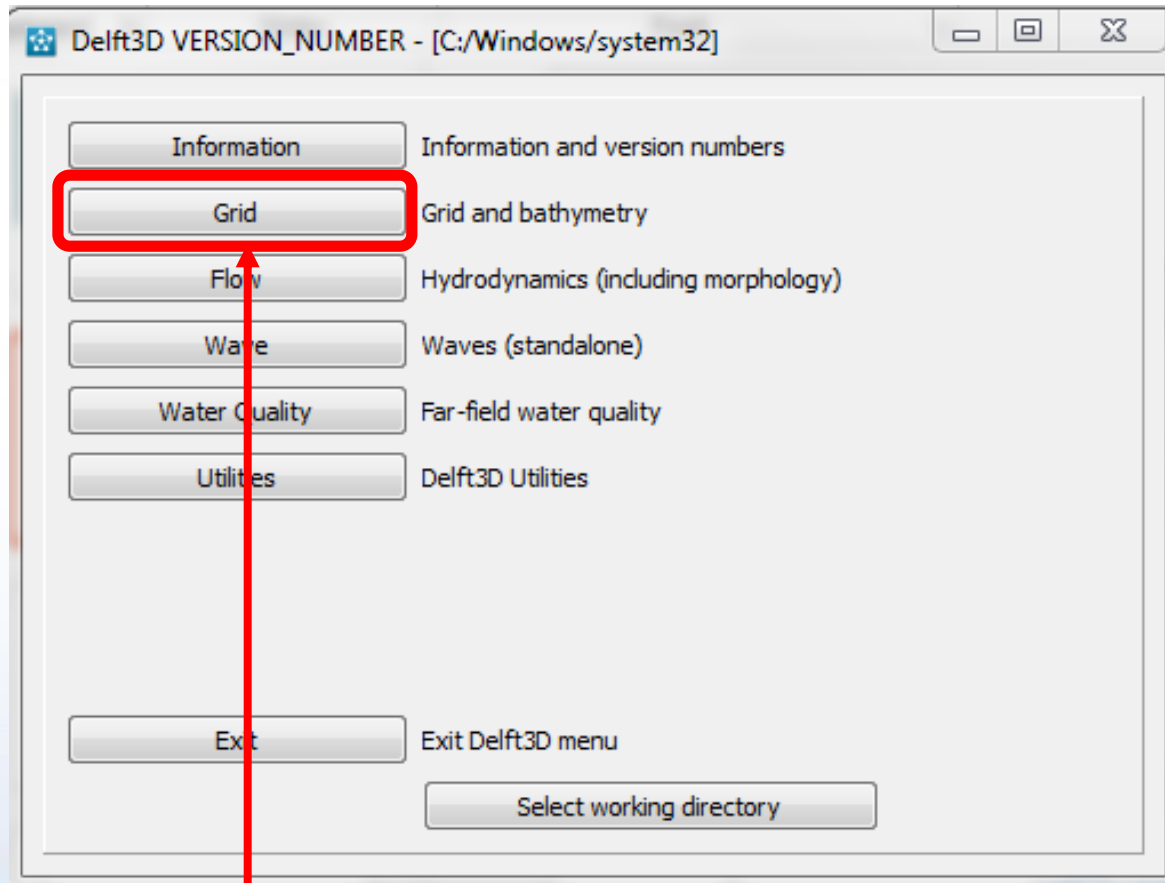


- Create and select a new folder

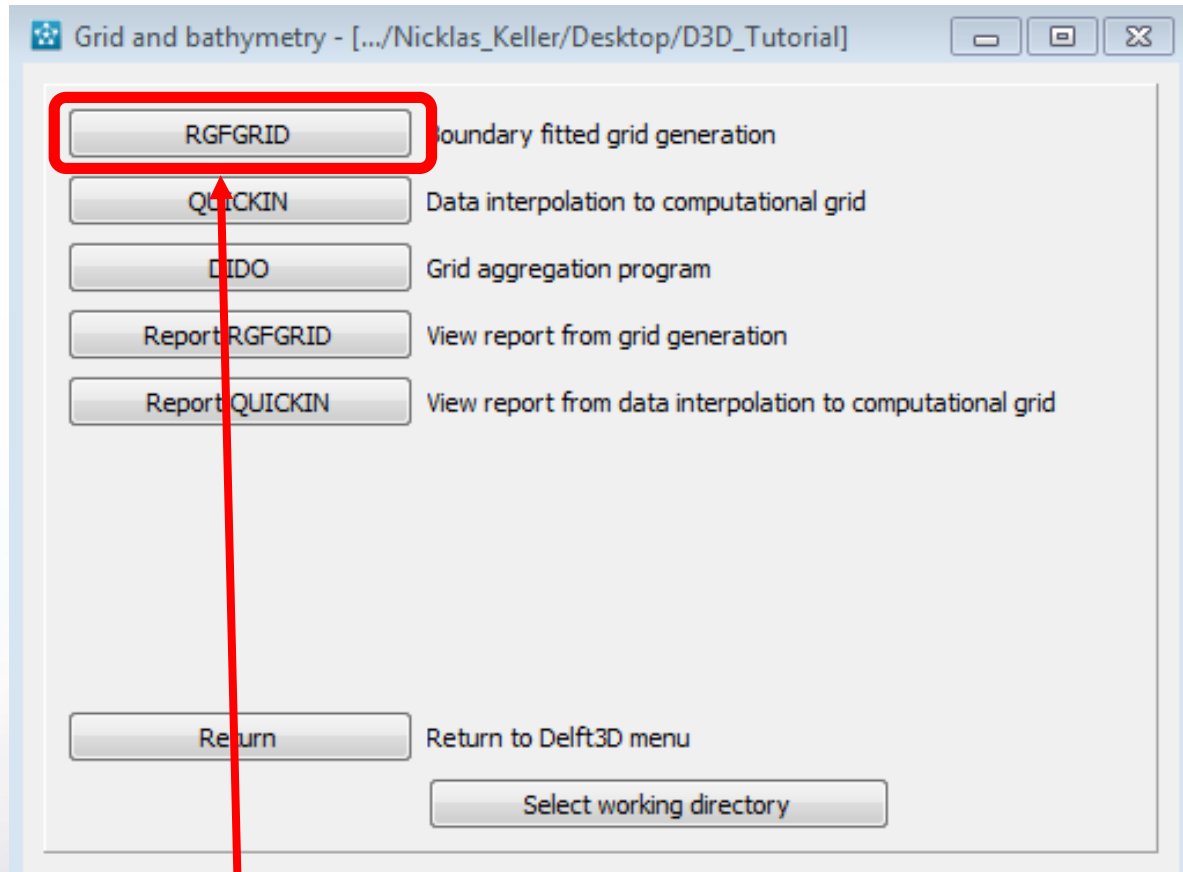




# Creating a Grid

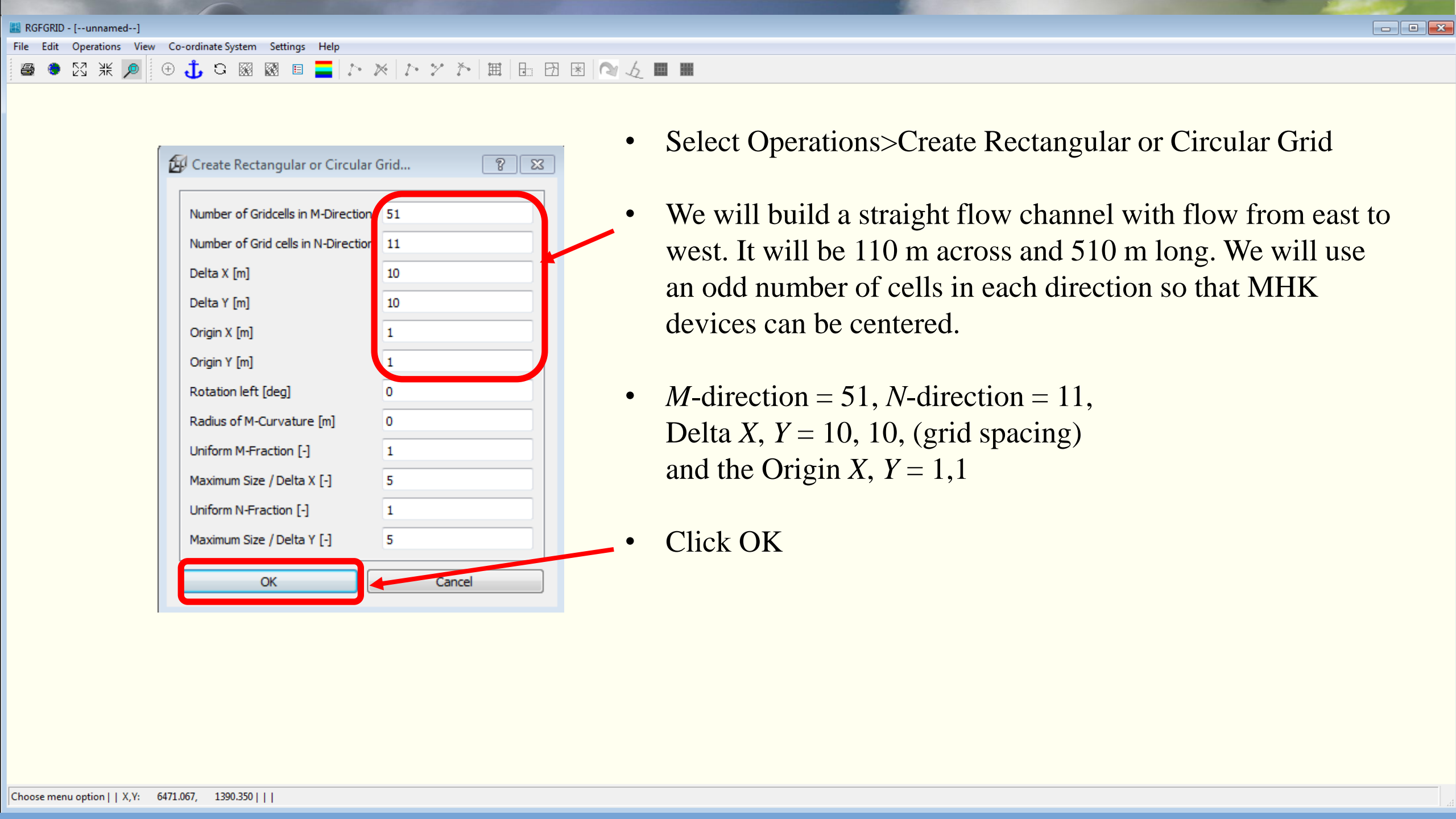


- Select Grid



- Select RGFGRID



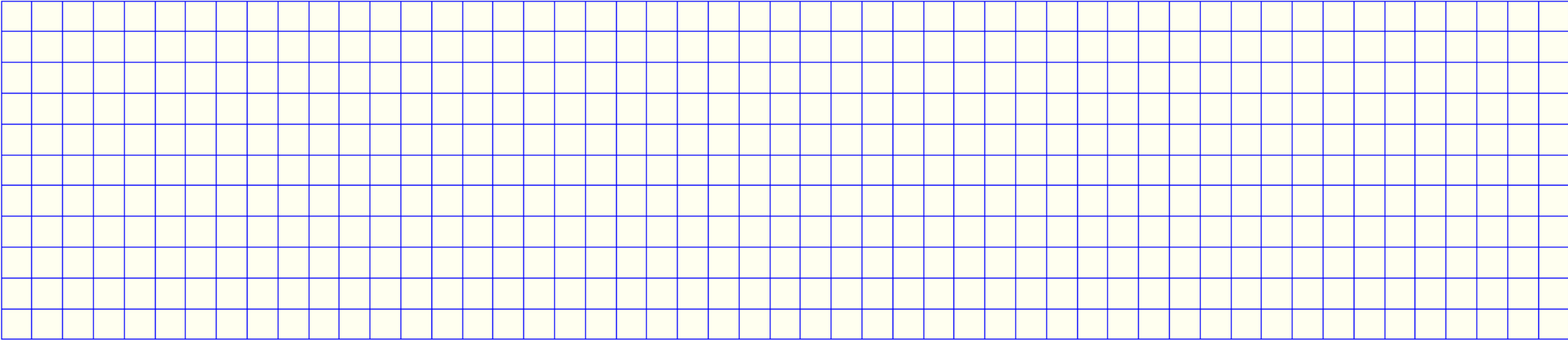


- Select Operations>Create Rectangular or Circular Grid
- We will build a straight flow channel with flow from east to west. It will be 110 m across and 510 m long. We will use an odd number of cells in each direction so that MHK devices can be centered.
- $M$ -direction = 51,  $N$ -direction = 11, Delta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


RGFGRID - [51x11]

File Edit Operations View Co-ordinate System Settings Help

- Save Project as D3D
- The grid file types created by RGFGRID are .d3d, .grd, and .enc

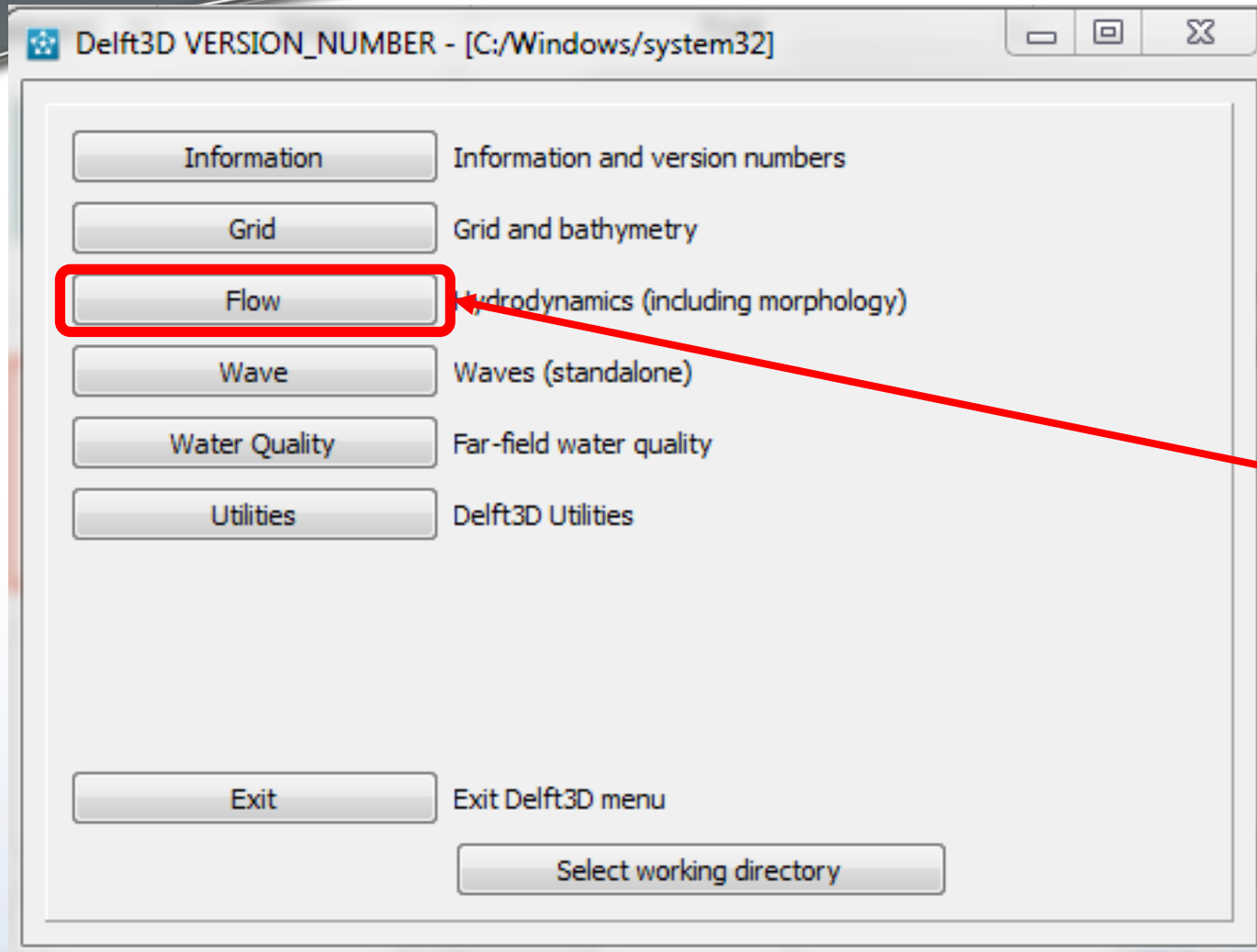


|| X,Y: 494.944, 24.996 | Cartesian |

 Sandia National Laboratories

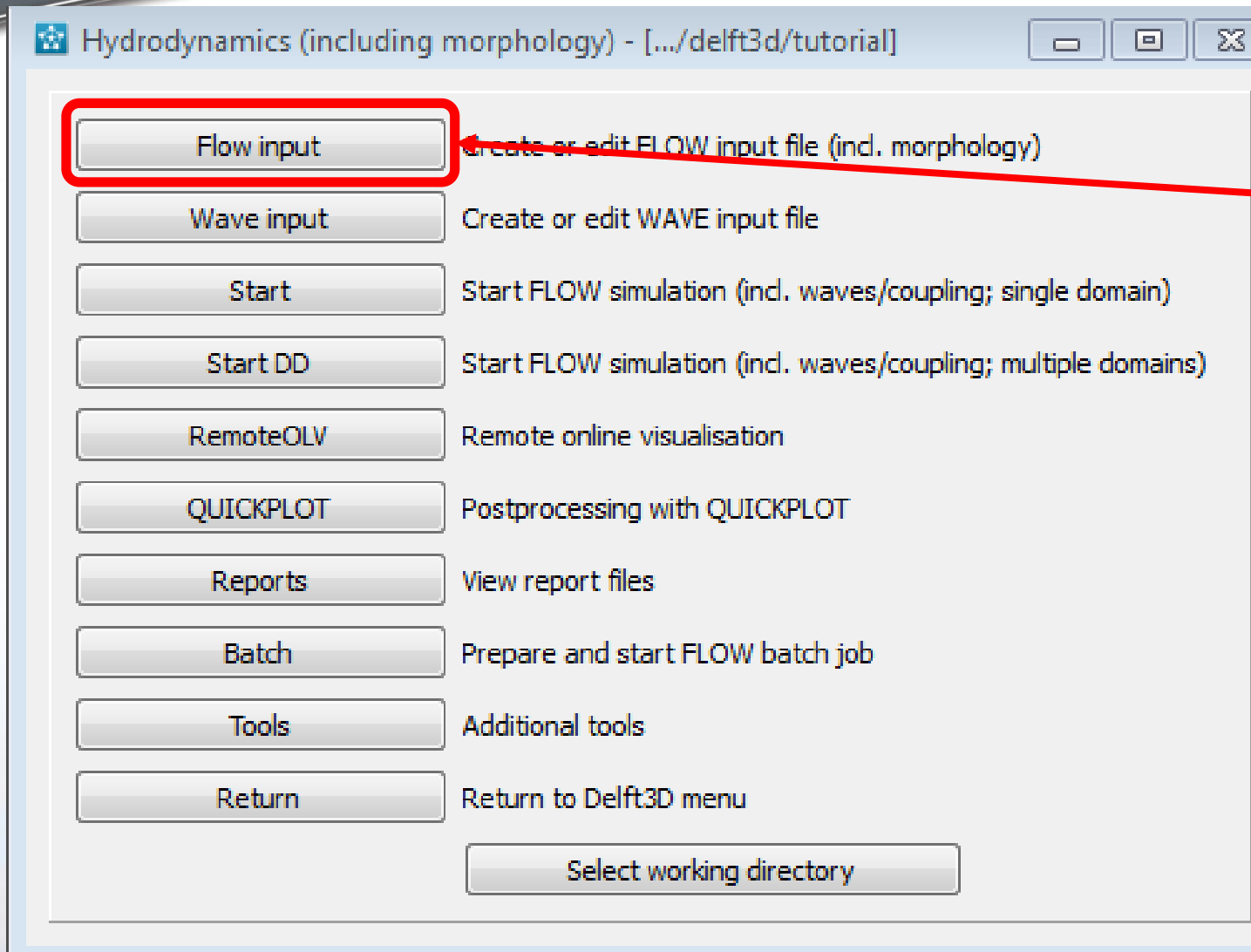


# Flow

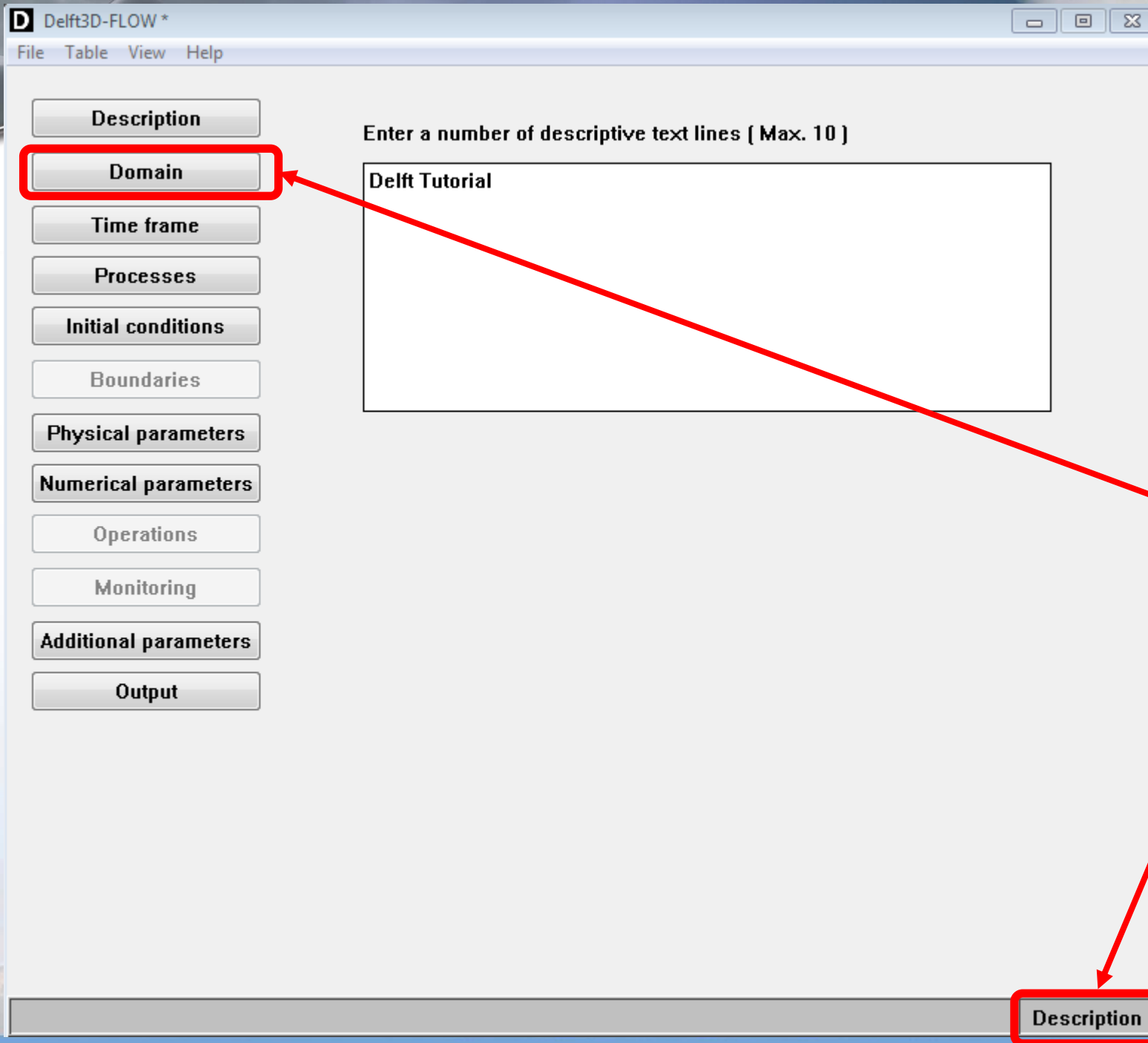


- Close RGFGRID and select Return
- This will put you back in the main Delft3D menu
- Select Flow

# 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

Delft3D-FLOW \*

File Table View Help

Description

Domain

Time frame

Processes

Initial conditions

Boundaries

Physical parameters

Numerical parameters

Operations

Monitoring

Additional parameters

Output

Grid Bathymetry Dry points Thin dams

Open grid File : ...\\D3D\_Tutorial Model\\D3D.grd

Open grid enclosure File : ...\\D3D\_Tutorial Model\\D3D.enc

Co-ordinate system: Cartesian

Grid points in M-direction: 53

Grid points in N-direction: 13

Latitude: 0 [dec. deg]

Orientation: 0 [dec. deg]

Number of layers: 3

Layer thickness [%]	
1	33.333333
2	33.333333
3	33.333333

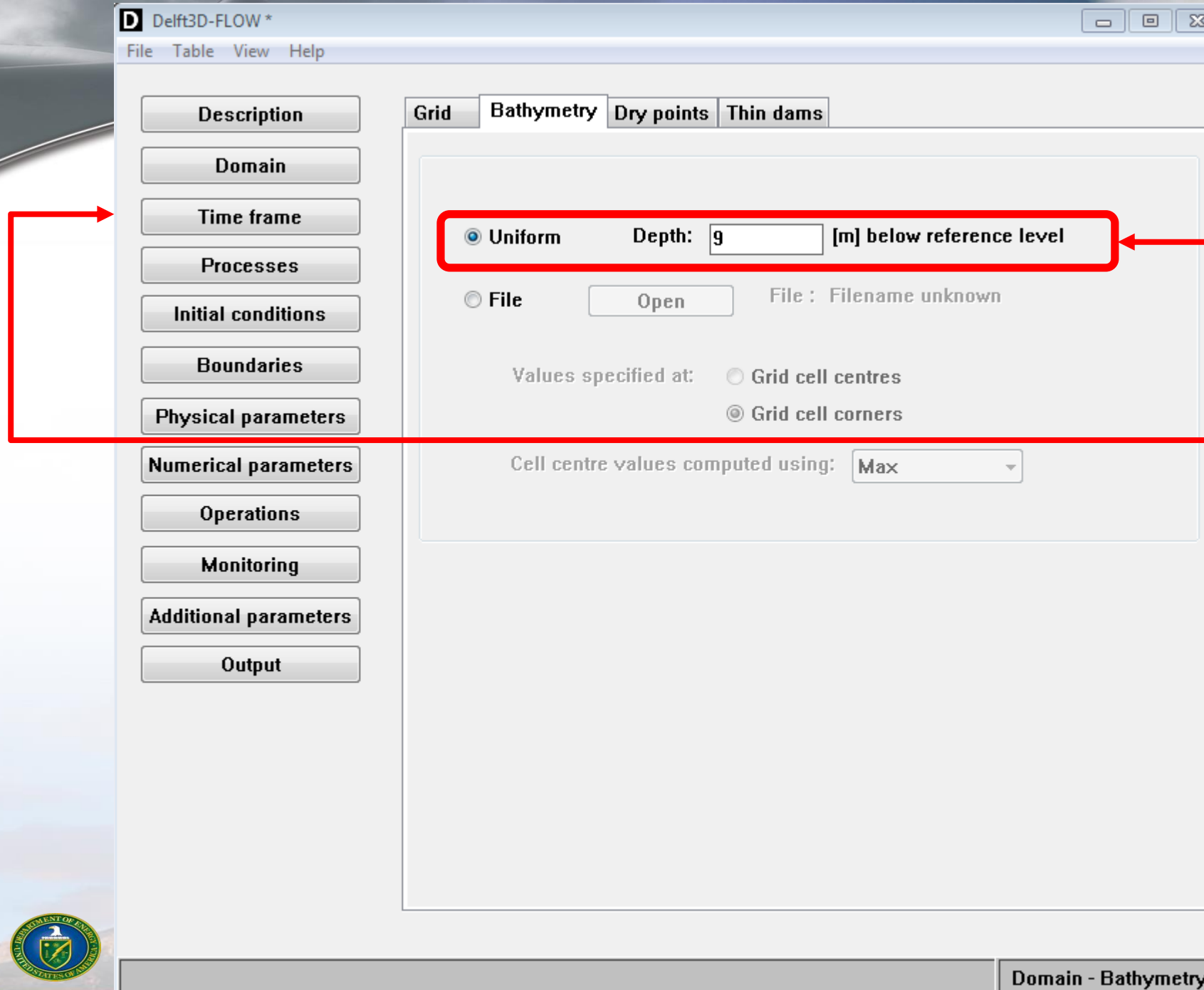
Total: 100 [%]

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

Delft3D-FLOW - C:\Users\csimmo\Desktop\D3D\_Basics\Files\D3D.mdf

File Table View Help

**Description**

Domain

**Time frame**

Processes

Initial conditions

Boundaries

Physical parameters

Numerical parameters

Operations

Monitoring

Additional parameters

Output

**Time frame**

Reference date  [dd mm yyyy]

Simulation start time  [dd mm yyyy hh mm ss]

Simulation stop time  [dd mm yyyy hh mm ss]

Time step  [min]

Local time zone (LTZ)  +GMT

GMT = Local time - LTZ

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



Delft3D-FLOW - C:\Users\Nicklas\_Keller\Desktop\D3D\_Tutorial\D3D.mdf \*

File Table View Help

Description

Domain

Time frame

Processes

Initial conditions

Boundaries

Physical parameters

Numerical parameters

Operations

Monitoring

Additional parameters

Output

Initial conditions

Uniform values

File :

Water level  [m]

Initial conditions

# Setting Initial Conditions

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



# Flow

D Delft3D-FLOW - C:\Users\Nicklas\_Keller\Desktop\D3D\_Tutorial\D3D.mdf \*

File Table View Help

Description

Domain

Time frame

Processes

Initial conditions

Boundaries

Physical parameters

Numerical parameters

Operations

Monitoring

Additional parameters

Output

Boundaries

Upstream  
Downstream

Add

Delete

Open / Save

Section name

Upstream

M1 1 N1 2

M2 1 N2 12

Flow conditions

Type of open boundary (quantity) :

Reflection parameter alpha:

Forcing type:

Vertical profile for hydrodynamics:

Total discharge

0 [s]

Time-series

Logarithmic

Edit flow conditions

Boundaries

- 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



**D** Boundaries : Flow Conditions

Table

Boundary: Upstream  
Quantity: Total discharge  
Forcing type: Time-series  
Vertical profile: Logarithmic

Time dd mm yyyy hh mm ss	Discharge [m <sup>3</sup> /s]
29 07 2015 00 00 00	990
29 07 2015 00 30 00	990

Close

- Set discharge volumetric flow rate to 990 m<sup>3</sup>/s
- Close this box

# Flow

Delft3D-FLOW - C:\Users\Nicklas\_Keller\Desktop\D3D\_Tutorial\D3D.mdf \*

File Table View Help

Description

Domain

Time frame

Processes

Initial conditions

Boundaries

Physical parameters

Numerical parameters

Operations

Monitoring

Additional parameters

Output

Boundaries

Upstream

Downstream

Add

Delete

Open / Save

Section name

Downstream

M1 53 N1 2

M2 53 N2 12

Flow conditions

Type of open boundary (quantity) :

Reflection parameter alpha:

Forcing type:

Water level

0 [s2]

Time-series

Vertical profile for hydrodynamics:

Edit flow conditions

Boundaries

- Select Add
- Title the section name downstream, and assign the following:  $M1=53$ ,  $N1=2$ ,  $M2=53$ , and  $N2=12$
- 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





**D** Boundaries : Flow Conditions

Table

Boundary: Downstream  
Quantity: Water level  
Forcing type: Time-series  
Vertical profile: n.a.

Time dd mm yyyy hh mm ss	Begin [m]	End [m]
29 07 2015 00 00 00	0	0
29 07 2015 00 30 00	0	0

Close

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

**D** Delft3D-FLOW - C:\Users\Nicklas\_Keller\Desktop\D3D\_Tutorial\D3D.mdf \*

File Table View Help

Boundaries

Upstream  
Downstream

Add Open / Save  
Delete

Section name  
Downstream

M1 53 N1 2  
M2 53 N2 12

Flow conditions  
Type of open boundary (quantity): Water level  
Reflection parameter alpha: 0 [s2]  
Forcing type: Time-series  
Vertical profile for hydrodynamics:

Edit flow conditions

Boundaries

**D Open/Save Boundaries**

**Boundary definitions**

Filename: C:\Users\Nicklas\_Keller\Desktop\D3D\_Tutorial\D3D.bnd

**Astronomical flow conditions**

Filename: Filename unknown

**Astronomical corrections**

Filename: Filename unknown

**Harmonic flow conditions**

Filename: Filename unknown

**QH-relation flow conditions**

Filename: Filename unknown

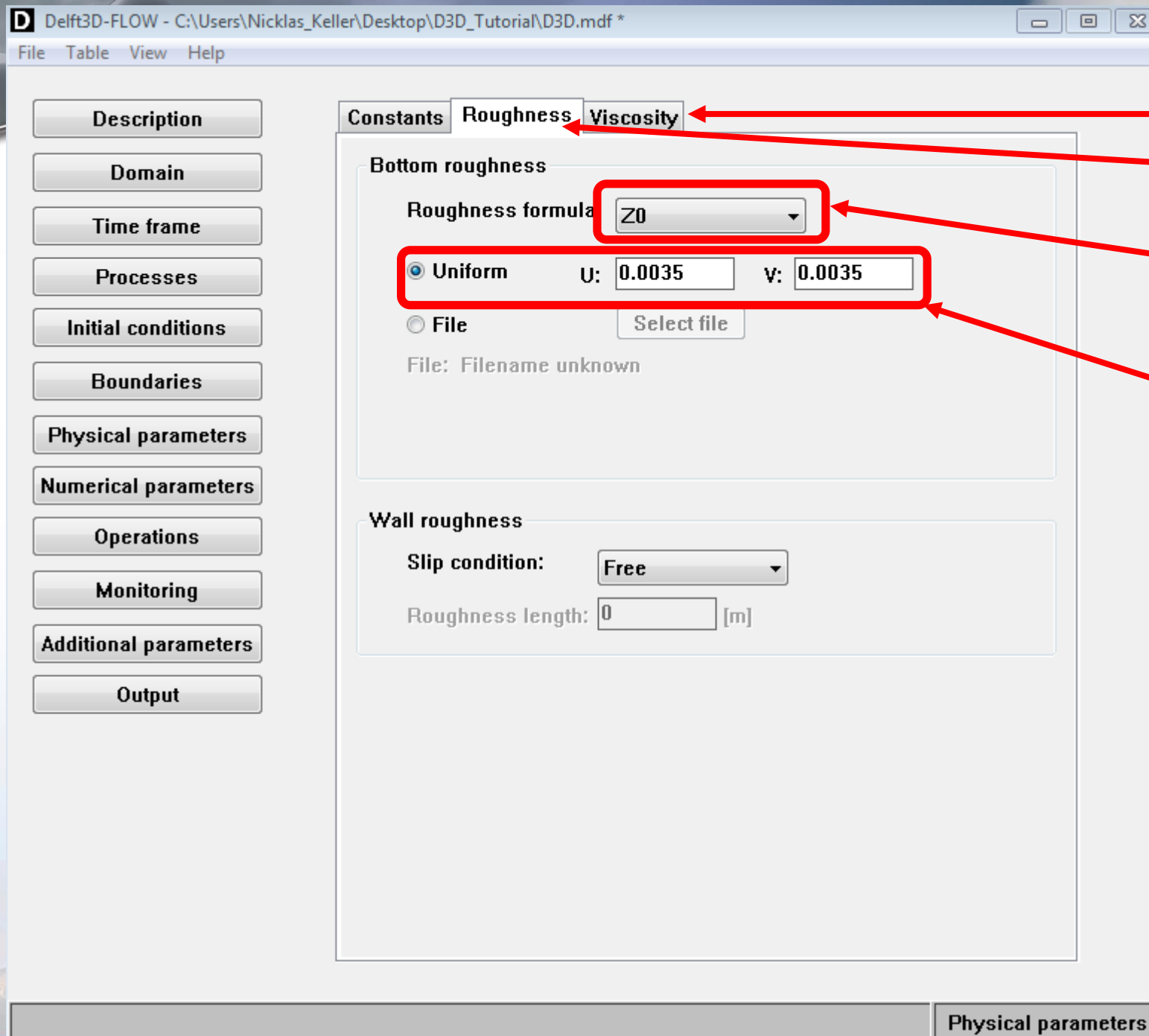
**Time-series flow conditions**

Filename: C:\Users\Nicklas\_Keller\Desktop\D3D\_Tutorial\D3D.bct

**Transport conditions**

Filename: Filename unknown

- 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

Delft3D-FLOW - C:\Users\Nicklas\_Keller\Desktop\D3D\_Tutorial\D3D.mdf \*

File Table View Help

Description

Domain

Time frame

Processes

Initial conditions

Boundaries

Physical parameters

**Numerical parameters**

Operations

Monitoring

Additional parameters

Output

Constants Roughness Viscosity

Background horizontal viscosity/diffusivity

☒ Uniform

Horizontal eddy viscosity  [m2/s]

☐ File

File : Filename unknown

Model for 2D turbulence

☐ Subgrid scale HLES

Background vertical viscosity/diffusivity

Vertical eddy viscosity  [m2/s]

Model for 3D turbulence

☐ Constant ☐ k-L

☐ Algebraic ☒ k-Epsilon

Physical parameters

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



# Flow

Delft3D-FLOW - C:\Users\Nicklas\_Keller\Desktop\D3D\_Tutorial\D3D.mdf \*

File Table View Help

Description

Domain

Time frame

Processes

Initial conditions

Boundaries

Physical parameters

Numerical parameters

Operations

Monitoring

Additional parameters

Output

Numerical parameters

Drying and flooding check at: ☒ Grid cell centres and faces  
☐ Grid cell faces only

Depth at grid cell faces: Mean

Threshold depth: 0.1 [m]

Marginal depth: -999 [m]

Smoothing time: 0 [min]

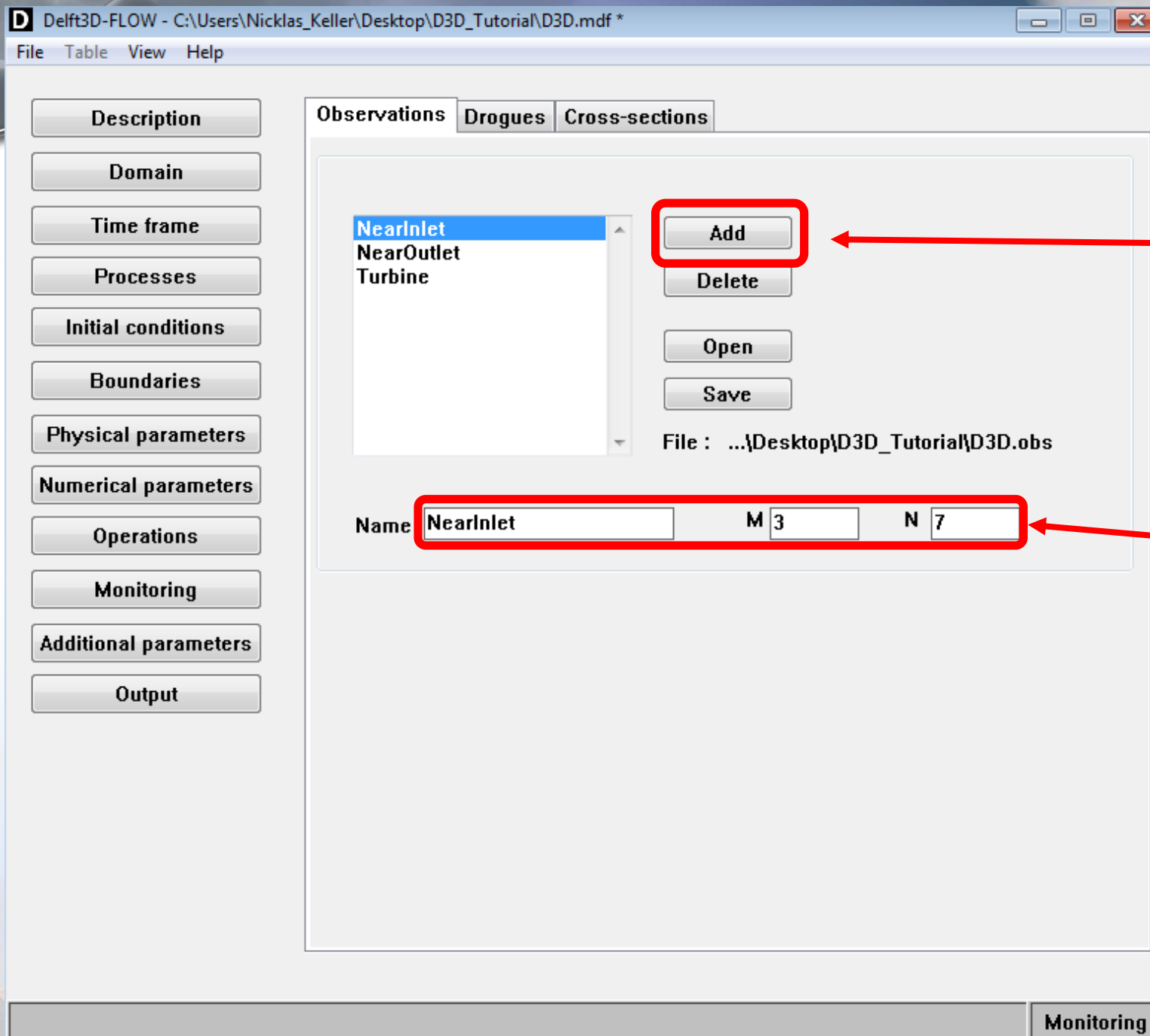
Advection scheme for momentum: Cyclic

Threshold depth for critical flow limiter: [m]

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





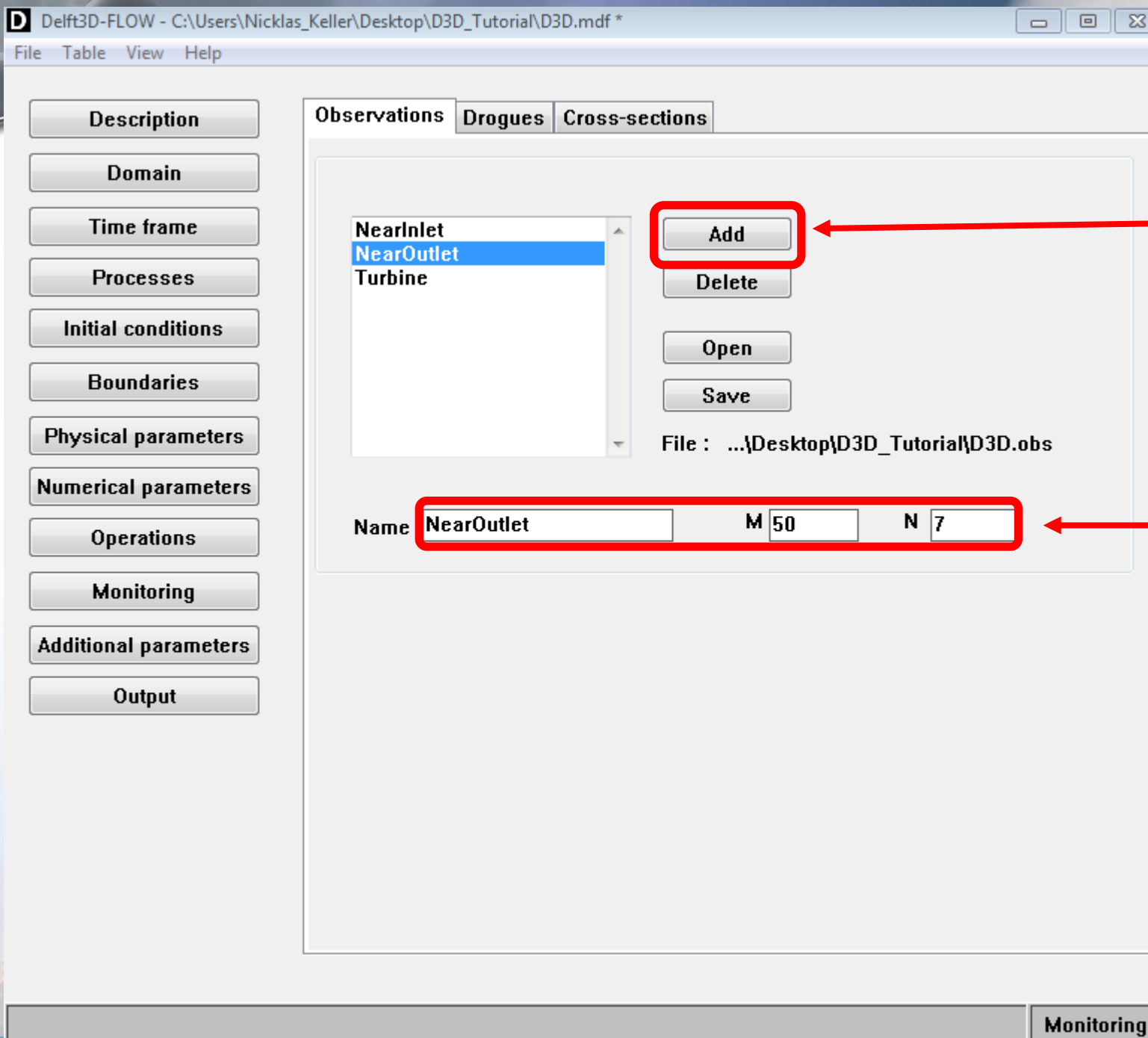


# Flow

- Select Add to add monitoring point within grid
- Name this monitoring cell NearInlet with  $M=3$ ,  $N=7$



# Flow

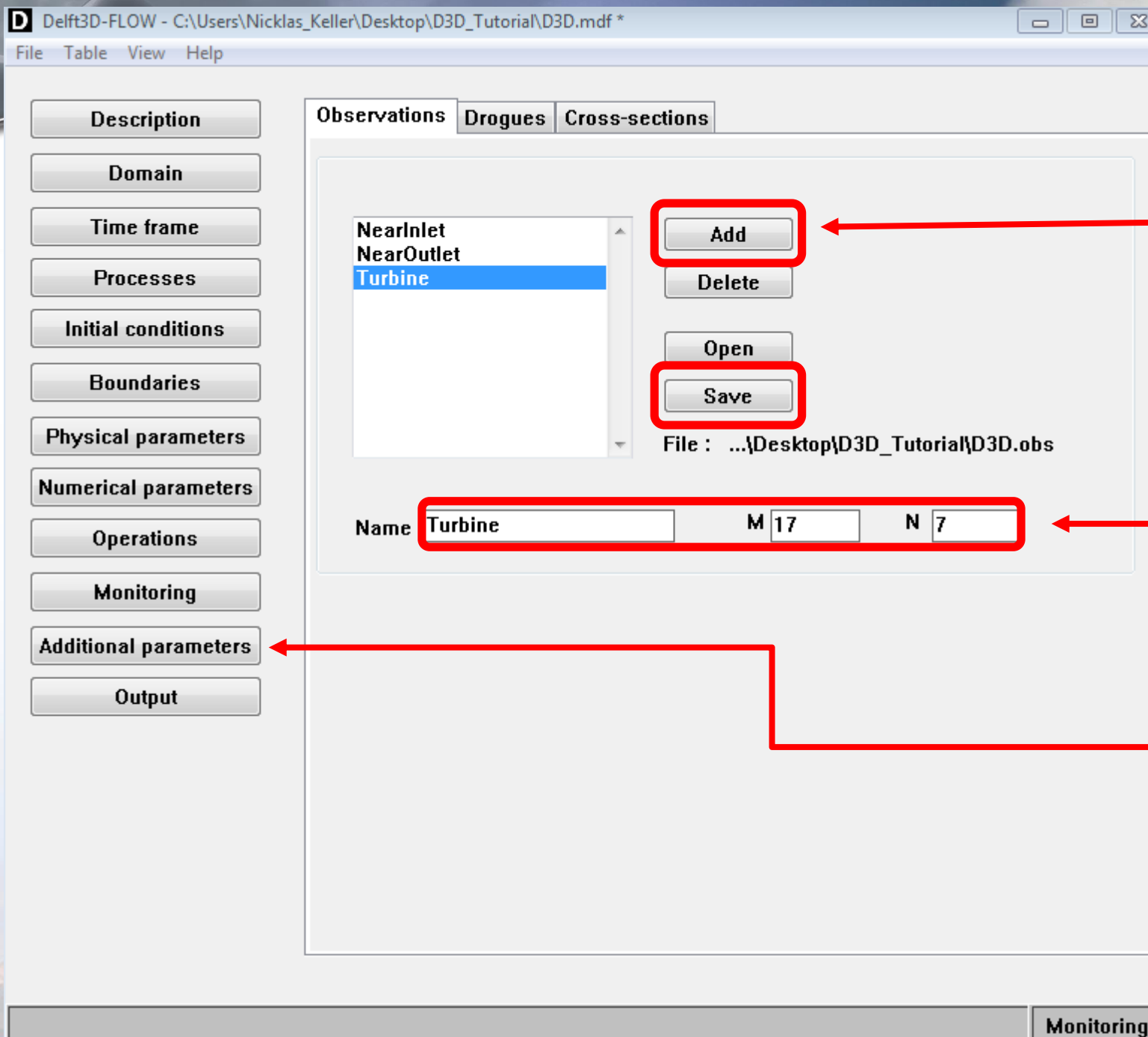


- Select Add

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



# Flow



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



D Delft3D-FLOW - C:\Users\Nicklas\_Keller\Desktop\D3D\_Tutorial\D3D.mdf \*

File Table View Help

Description

Domain

Time frame

Processes

Initial conditions

Boundaries

Physical parameters

Numerical parameters

Operations

Monitoring

Additional parameters

Output

Additional parameters

Keyword	Value
Filtrb	#turbines.ini#

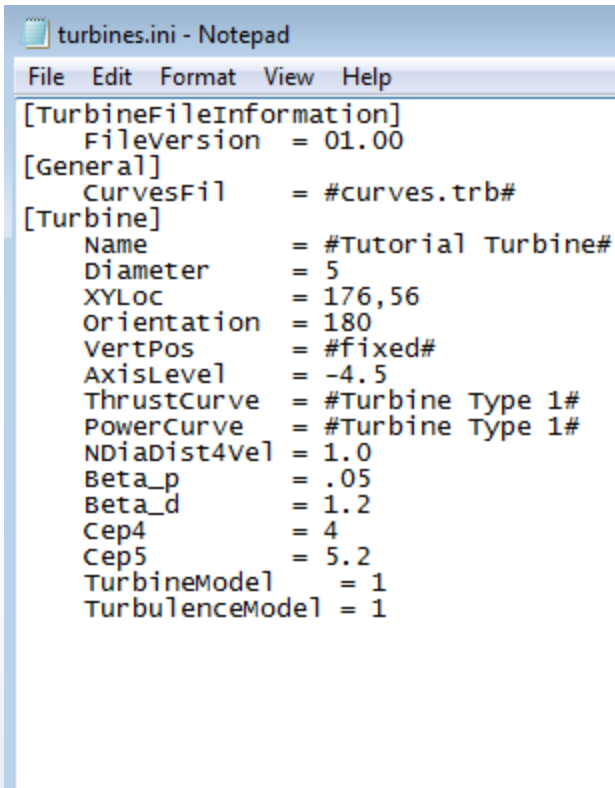
Add

Delete

## 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

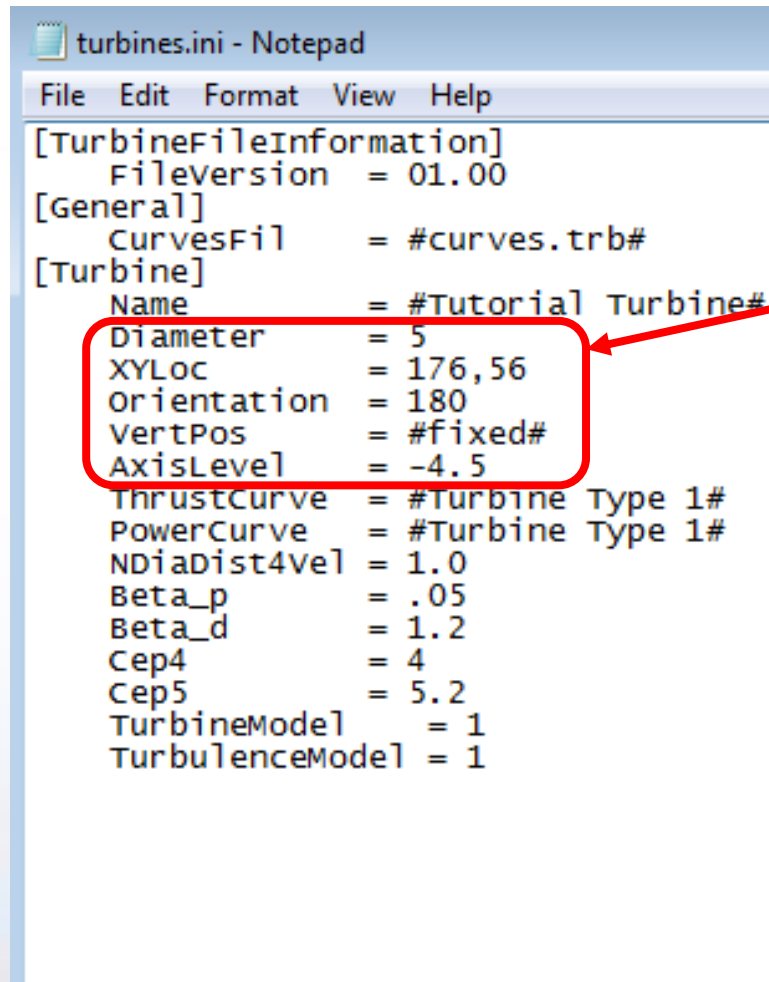


```
turbines.ini - Notepad
File Edit Format View Help
[TurbineFileInformation]
  FileVersion = 01.00
[General]
  CurvesFil = #curves.trb#
[Turbine]
  Name = #Tutorial Turbine#
  Diameter = 5
  XYLoc = 176,56
  Orientation = 180
  VertPos = #fixed#
  AxisLevel = -4.5
  ThrustCurve = #Turbine Type 1#
  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
```

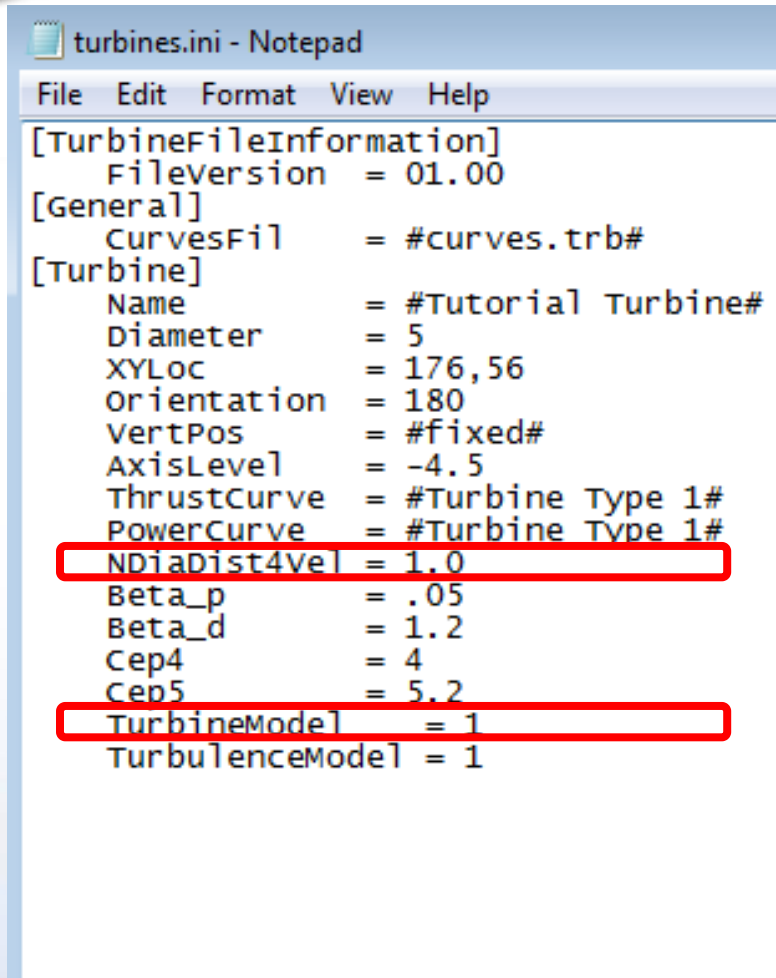
- “CurvesFil” links to curves.trb file specified.
- “Name” is any name you choose.
- Sets the location of the turbine
  - “fixed” is a constant (x,y,z) location
  - “floating” has a fixed x and y but z moves with sigma layer (see Delft 3D manual for sigma layer)
- “AxisLevel” sets hub height of turbine.
  - 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#
  ND1aD1st4vel = 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
```

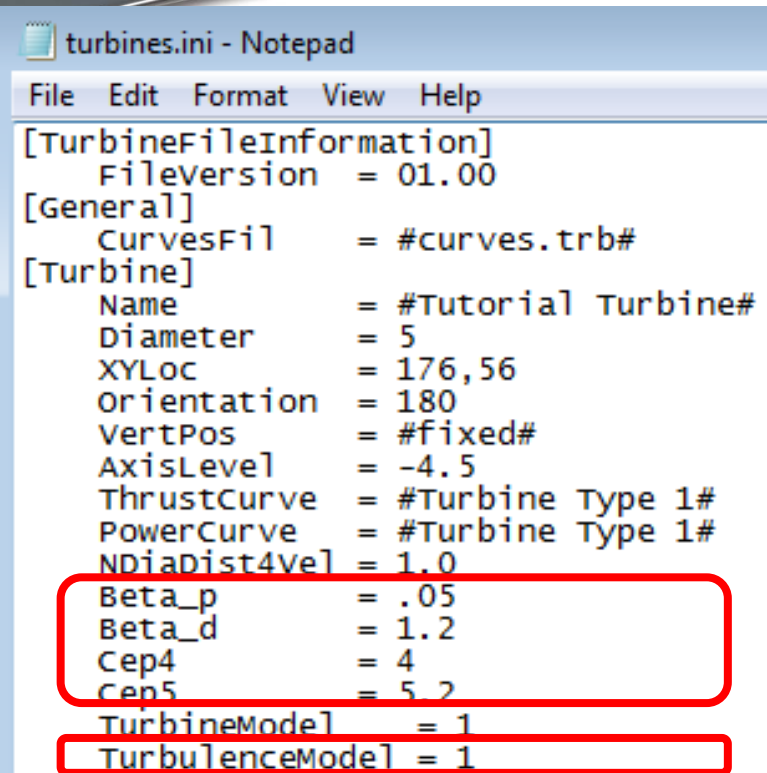
- TurbineModel = 0, Upstream velocity turbine model.
  - $F_t = \frac{1}{2} \rho A C_T U_\infty^2$
  - Where the force of the turbine is a function of density, turbine area, coefficient of thrust, and ambient upstream velocity.
- TurbineModel = 1, disc velocity turbine model.
  - $F_t = \frac{1}{2} \rho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



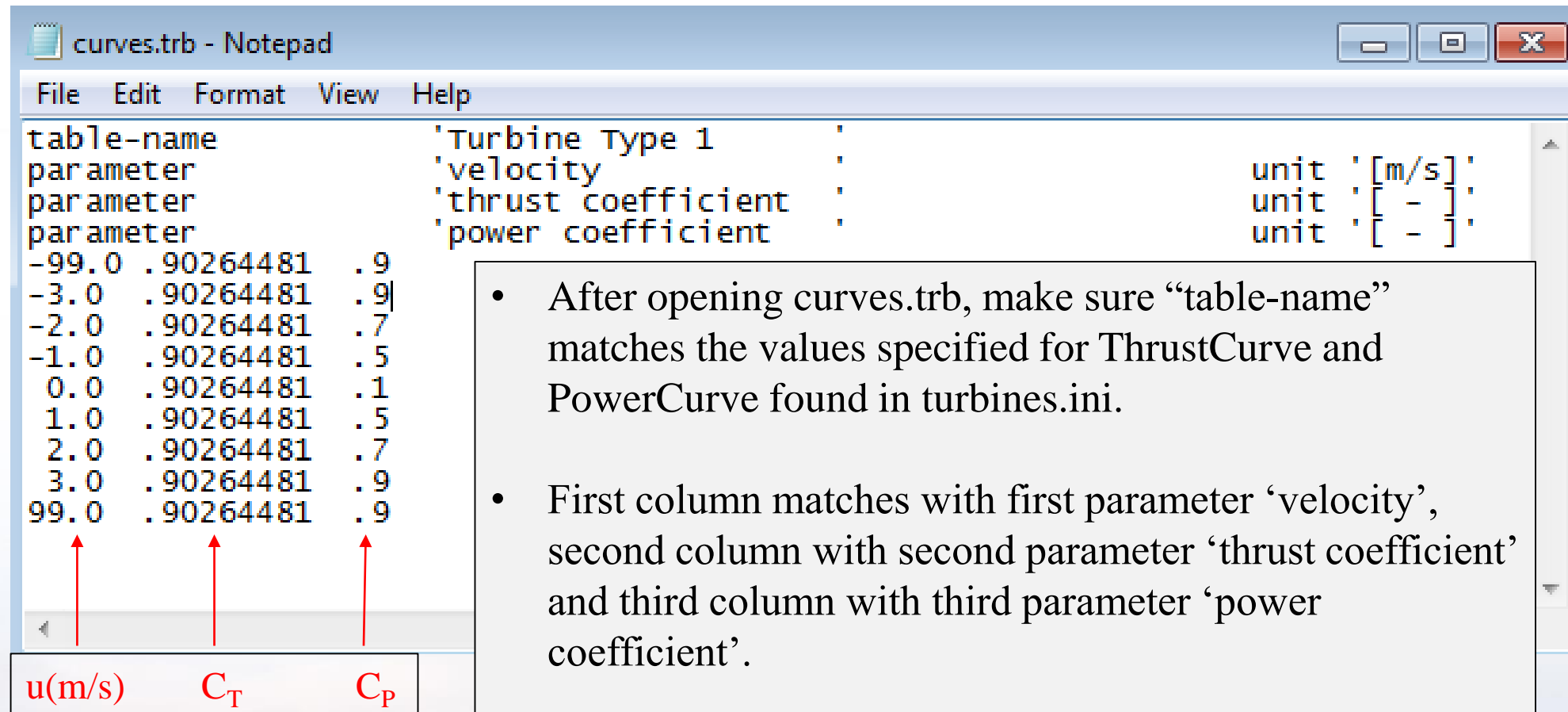
```
turbines.ini - Notepad
File Edit Format View Help
[TurbineFileInformation]
  FileVersion = 01.00
[General]
  CurvesFil = #curves.trb#
[Turbine]
  Name = #Tutorial Turbine#
  Diameter = 5
  XYLoc = 176,56
  Orientation = 180
  VertPos = #fixed#
  AxisLevel = -4.5
  ThrustCurve = #Turbine Type 1#
  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.
  - $\beta_P$ ,  $\beta_D$ ,  $C_{\varepsilon 4}$ ,  $C_{\varepsilon 5}$  are ignored
- TurbulenceModel = 1 uses Rethore turbulence model
  - $\beta_P$ ,  $\beta_D$ ,  $C_{\varepsilon 4}$ ,  $C_{\varepsilon 5}$  adjust source terms added to k- $\varepsilon$  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),  $\varepsilon$  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.





Delft3D-FLOW - C:\Users\csimmo\Desktop\D3D\_Basics\Files\D3D.mdf

File Table View Help

Storage **Print** Details

FLOW simulation times    Start time:    29 07 2015 00 00 00  
Stop time:    29 07 2015 00 30 00  
Time Step [min]:    0.001

Store map results    Store communication file :

dd mm yyyy hh mm ss    dd mm yyyy hh mm ss

Start time    29 07 2015 00 00 00    Start time    29 07 2015 00 00 00  
Stop time    29 07 2015 00 30 00    Stop time    29 07 2015 00 30 00  
Interval    2 [min]    Interval    5 [min]  
History interval    10 [min]    Restart int.    10 [min]

☐ Fourier analysis    ☐ Online visualisation  
    ☐ Export WAQ input  
File : Filename unknown   

Output

# 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



D Delft3D-FLOW - C:\Users\csimmo\Desktop\D3D\_Basics\Files\D3D.mdf

File Table View Help

Description  
Domain  
Time frame  
Processes  
Initial conditions  
Boundaries  
Physical parameters  
Numerical parameters  
Operations  
Monitoring  
Additional parameters  
Output

Storage Print Details

FLOW simulation times  
Start time: 29 07 2015 00 00 00  
Stop time: 29 07 2015 00 30 00  
Time step [min]: 0.001

Print history results:  
dd mm yyyy hh mm ss  
Start time 29 07 2015 00 00 00  
Stop time 29 07 2015 00 30 00  
Interval 1 [min]

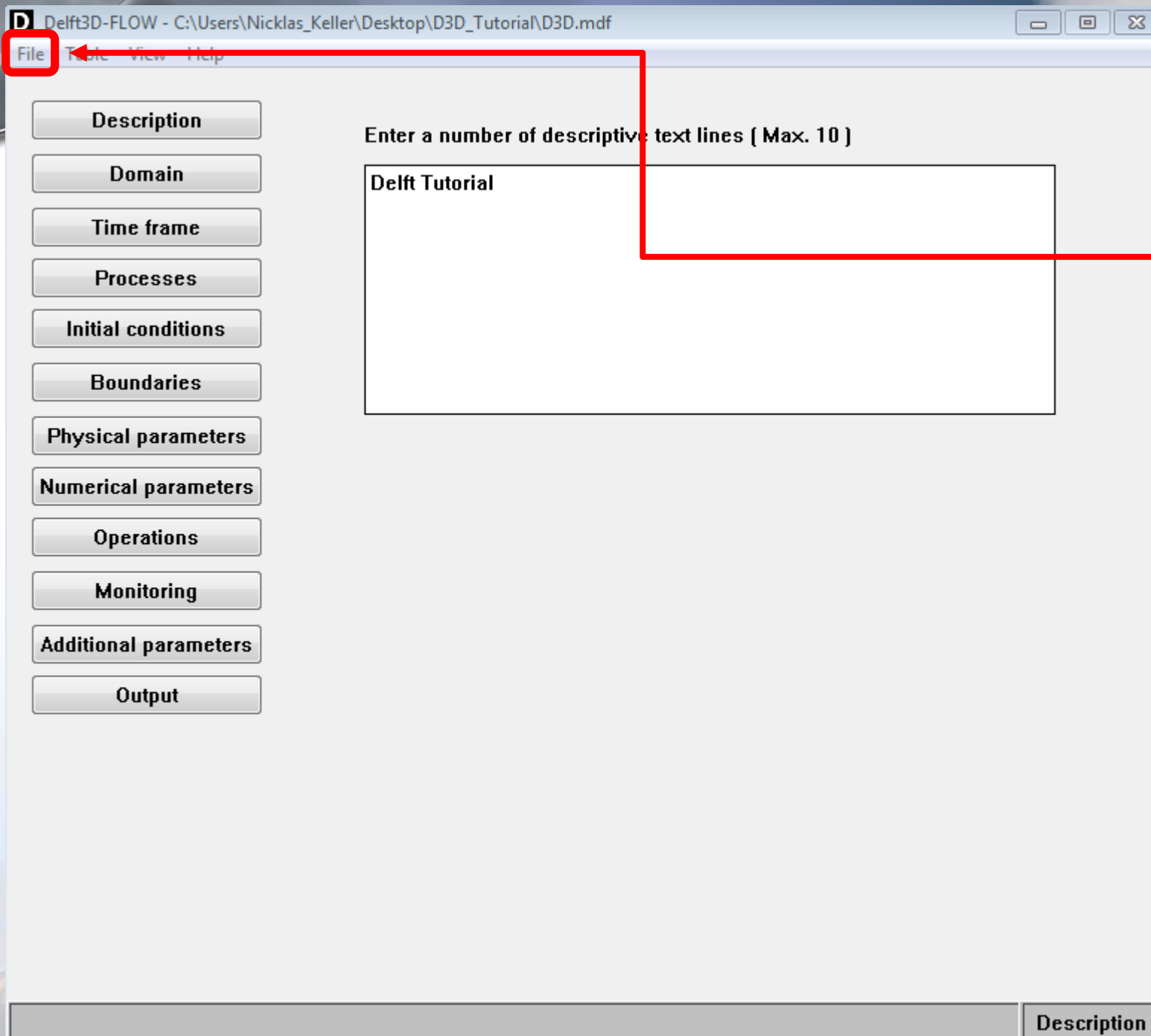
Print map results:  
dd mm yyyy hh mm ss  
  
Add Delete  
dd mm yyyy hh mm ss

Output

# Data Output

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

# Saving Your File

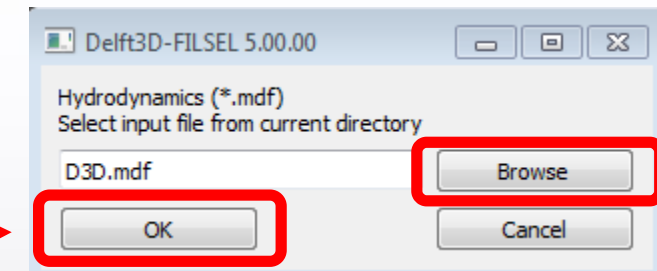
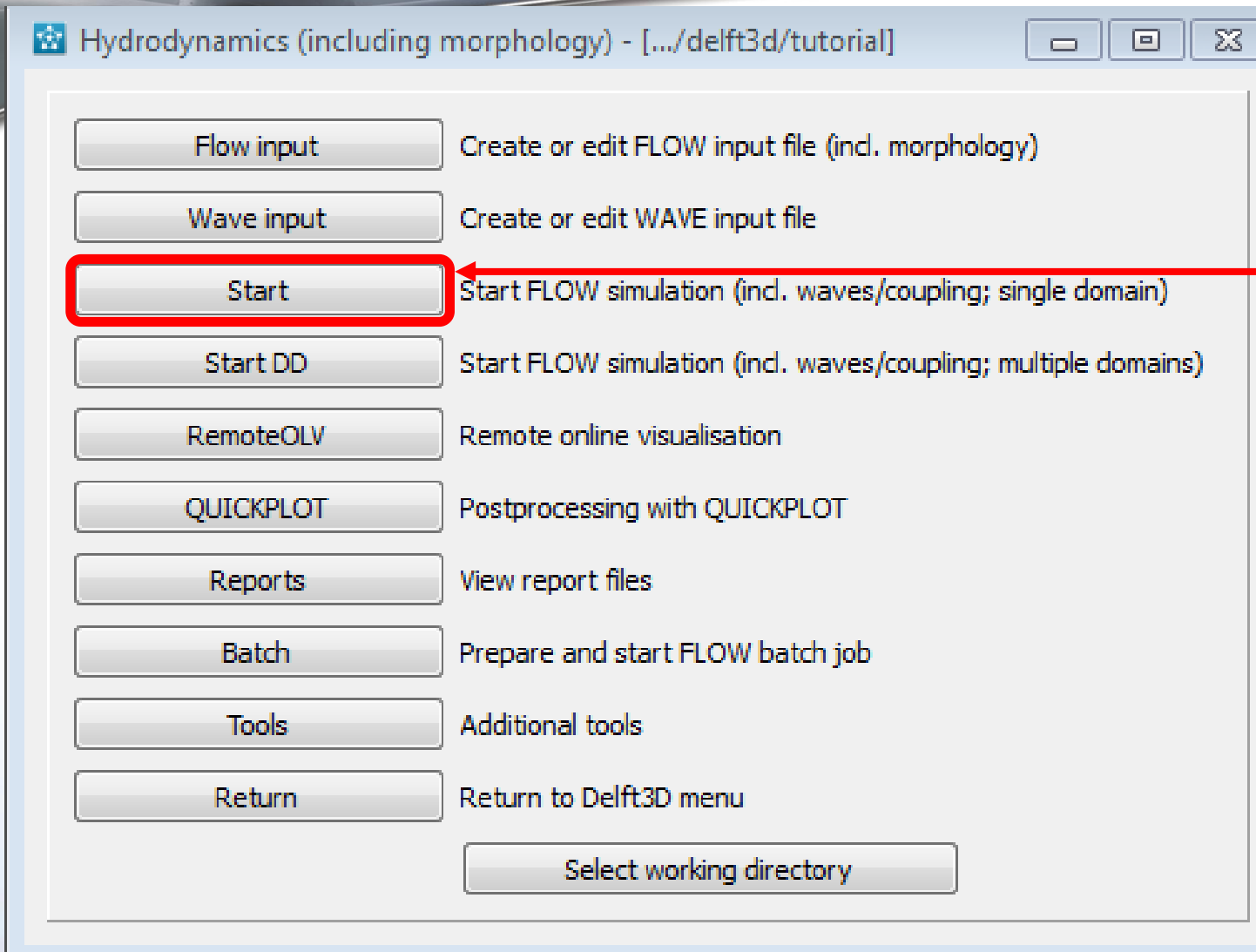


- Select File>Save MDF As
- Save as “D3D.mdf”



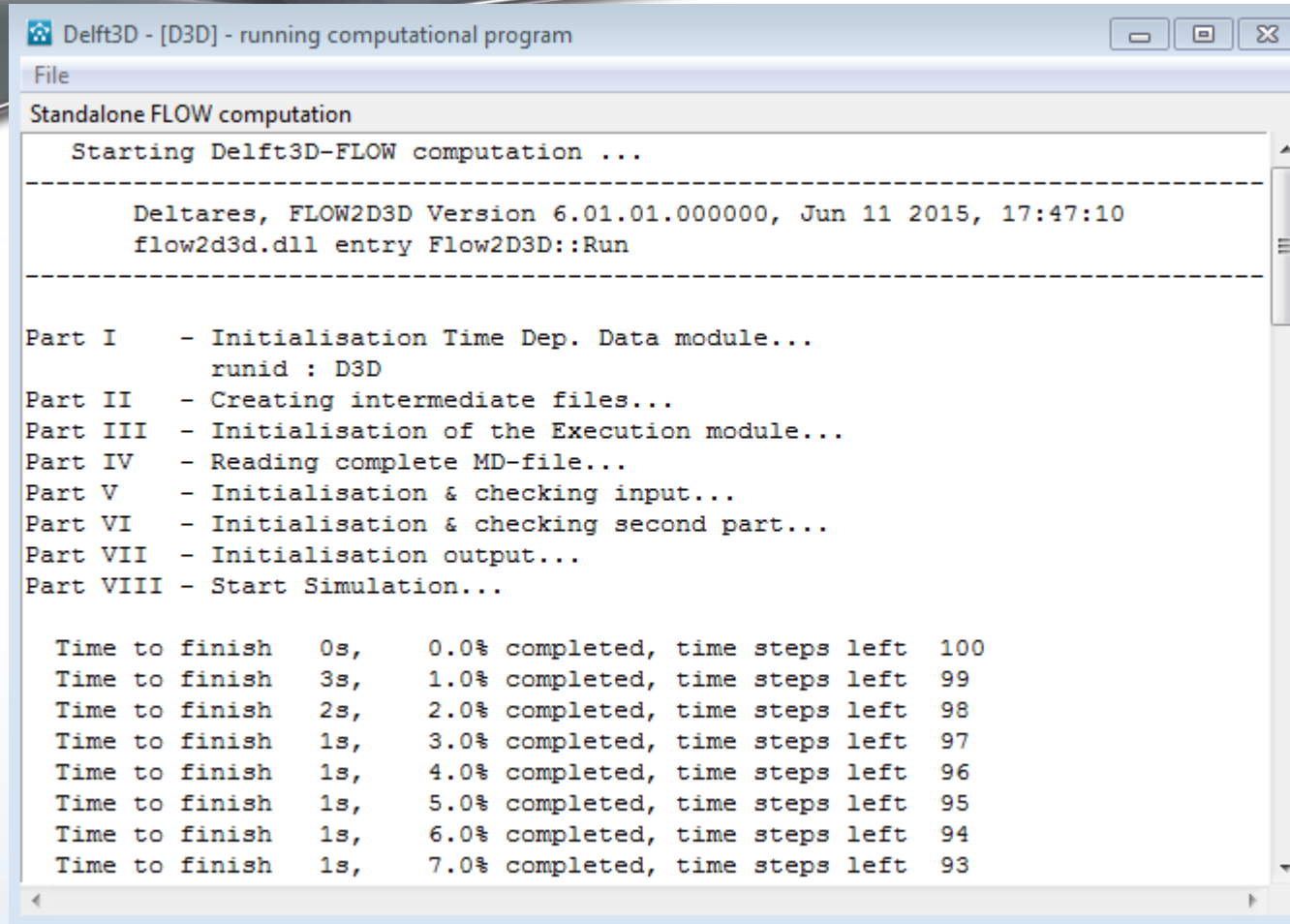
# Running a Flow Model

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



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

# Running a Flow Model



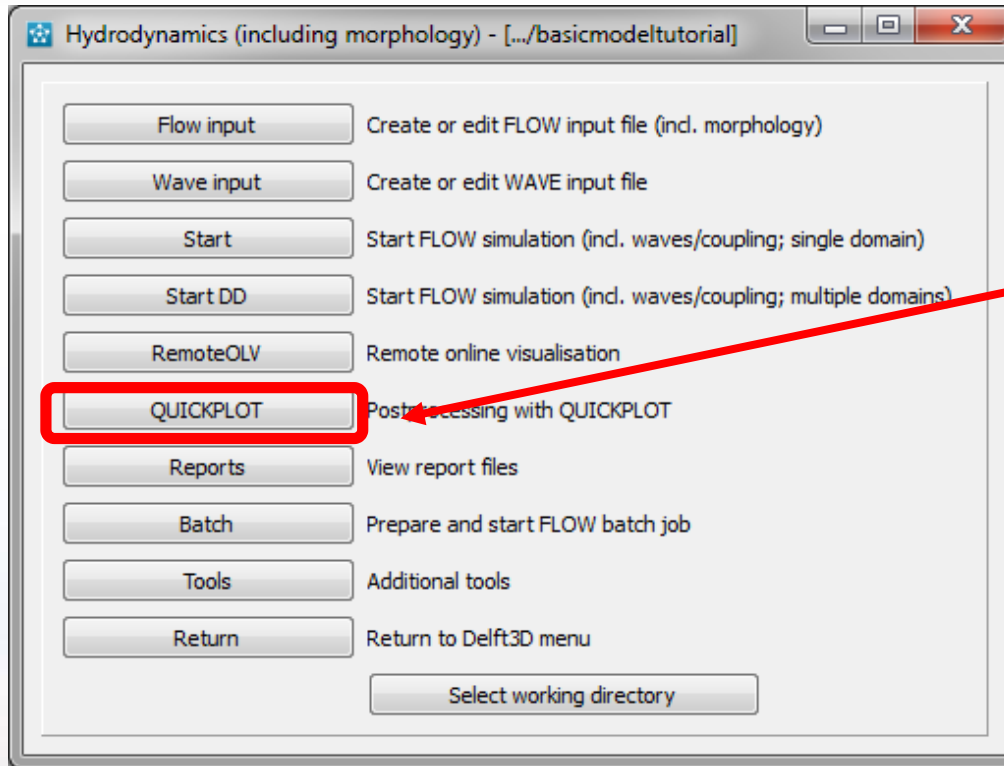
```
Delft3D - [D3D] - running computational program
File
Standalone FLOW computation
Starting Delft3D-FLOW computation ...
-----
Deltares, FLOW2D3D Version 6.01.01.0000000, Jun 11 2015, 17:47:10
flow2d3d.dll entry Flow2D3D::Run
-----
Part I    - Initialisation Time Dep. Data module...
            runid : D3D
Part II   - Creating intermediate files...
Part III  - Initialisation of the Execution module...
Part IV   - Reading complete MD-file...
Part V    - Initialisation & checking input...
Part VI   - Initialisation & checking second part...
Part VII  - Initialisation output...
Part VIII - Start Simulation...

Time to finish  0s,      0.0% completed, time steps left  100
Time to finish  3s,      1.0% completed, time steps left   99
Time to finish  2s,      2.0% completed, time steps left   98
Time to finish  1s,      3.0% completed, time steps left   97
Time to finish  1s,      4.0% completed, time steps left   96
Time to finish  1s,      5.0% completed, time steps left   95
Time to finish  1s,      6.0% completed, time steps left   94
Time to finish  1s,      7.0% completed, time steps left   93
```

- 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.



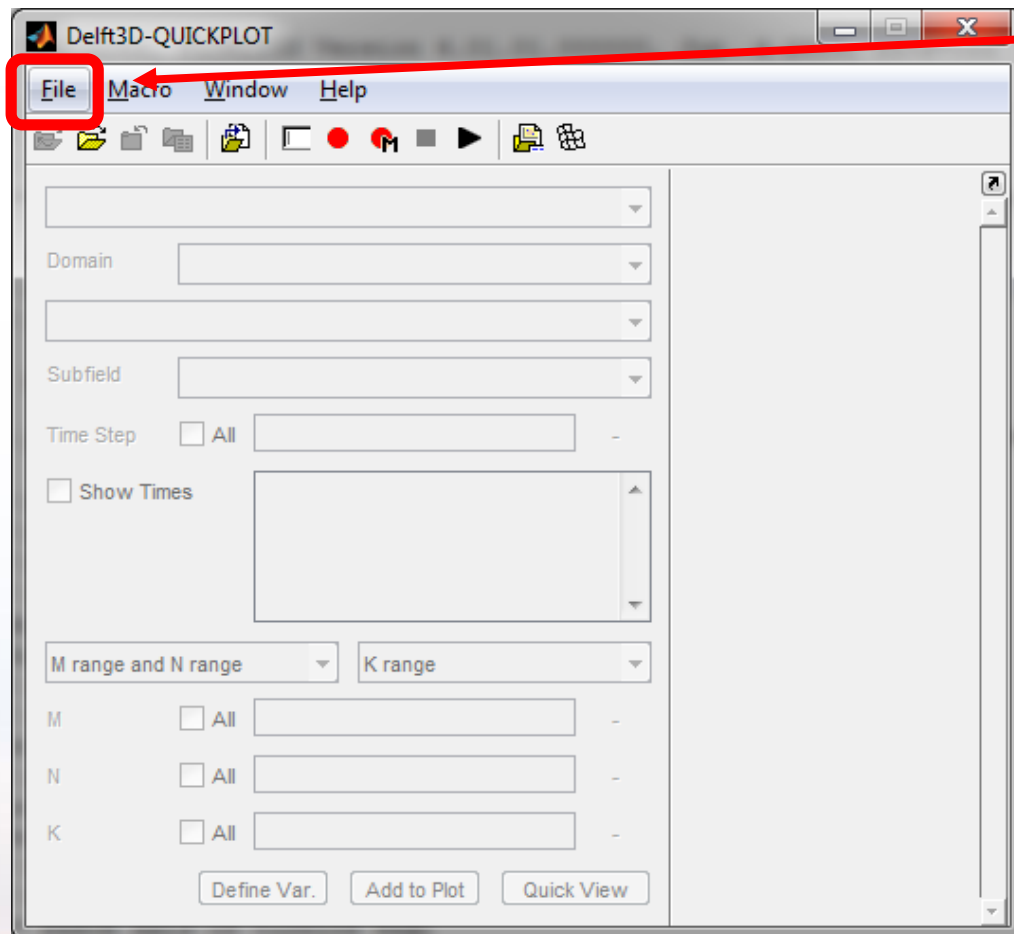
# Post Processing with QUICKPLOT



- 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.

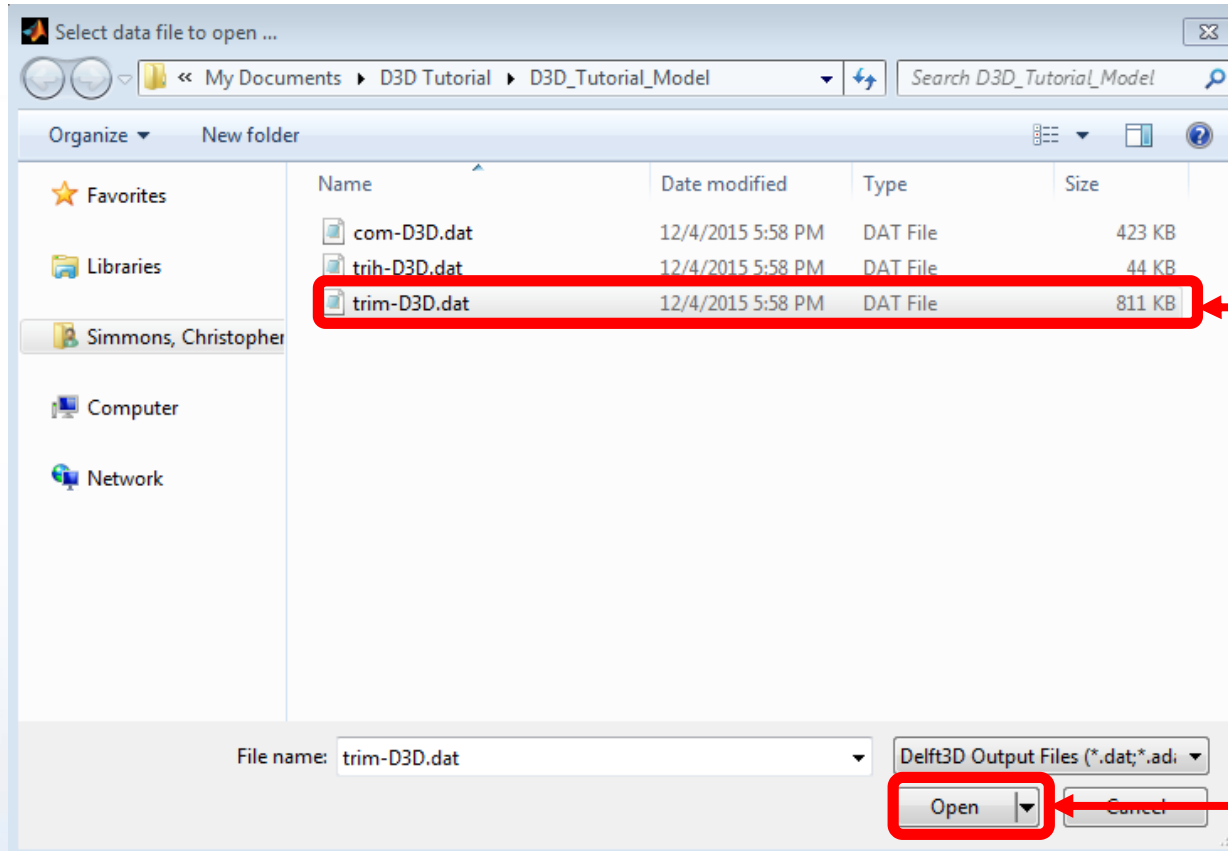


# QUICKPLOT



- Select File
- Select Open File

# QUICKPLOT



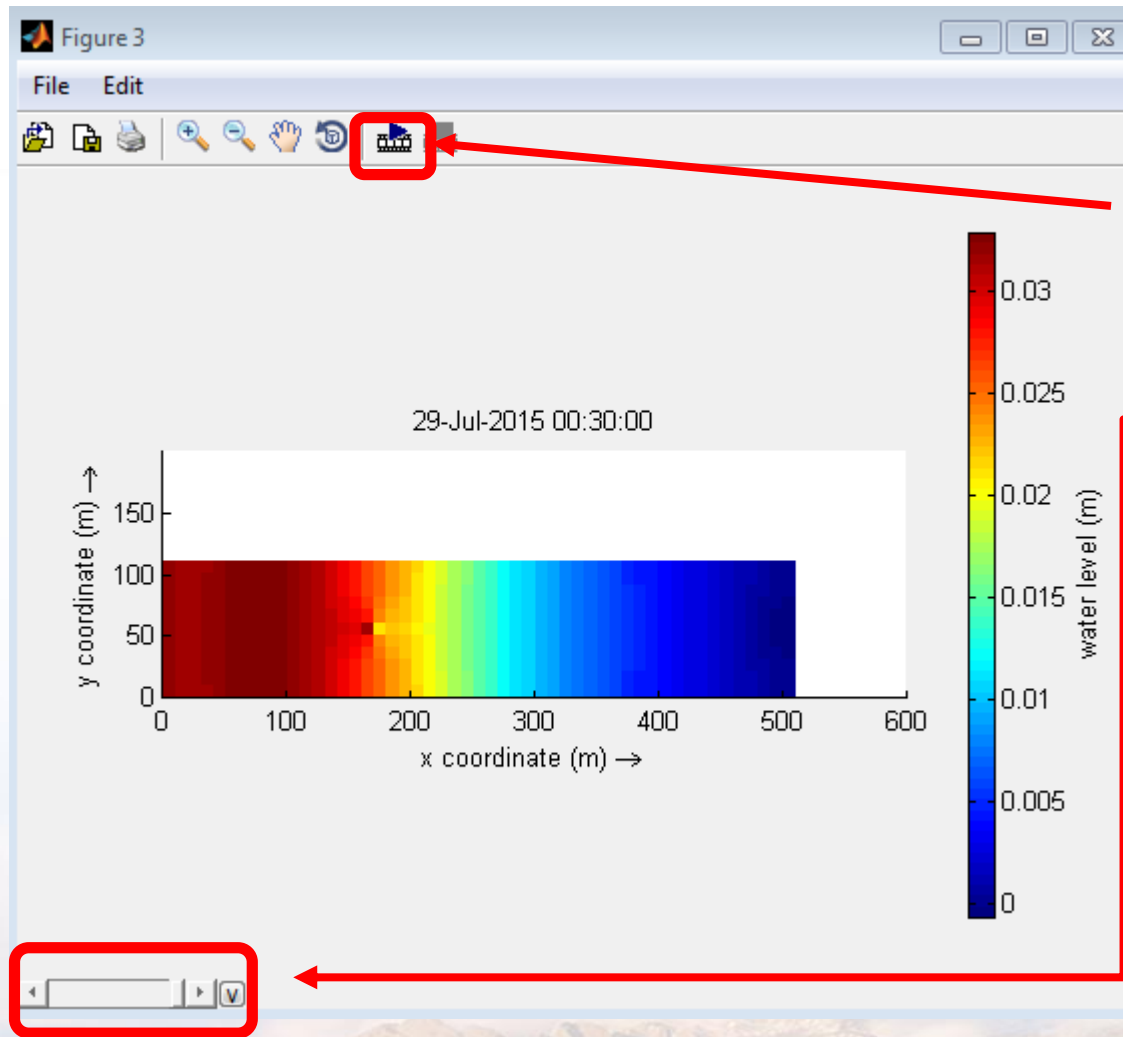
- Open “trim-D3D.dat”
- Be sure to open the trim-D3D.dat rather than the trih-D3D.dat.
- The “trim-D3D.dat” file contains output data at all cell nodes and faces written out at each time step specified in the “Store map results” section of the Data Output.
- Since we specified a two minute map storage 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



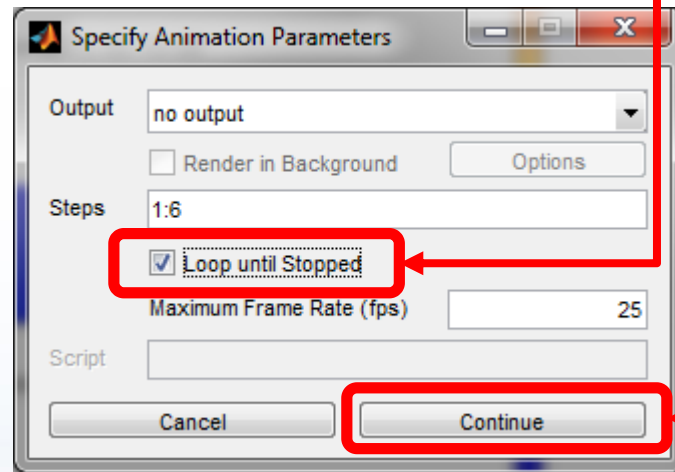
- Select Specify Animation Parameters to animate.

- This bar allows viewing at different times.

# QUICKPLOT – Water Level Contour

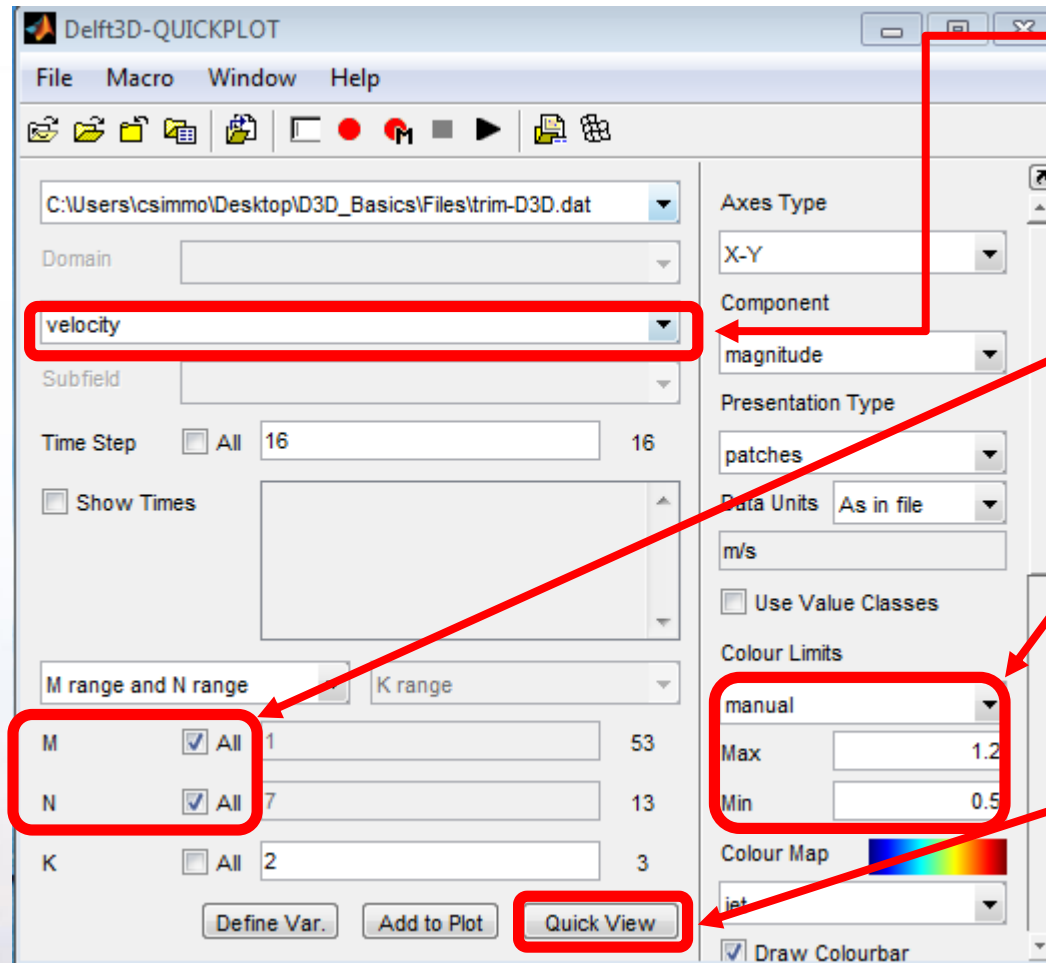
- Check Loop until Stopped

- Select Continue





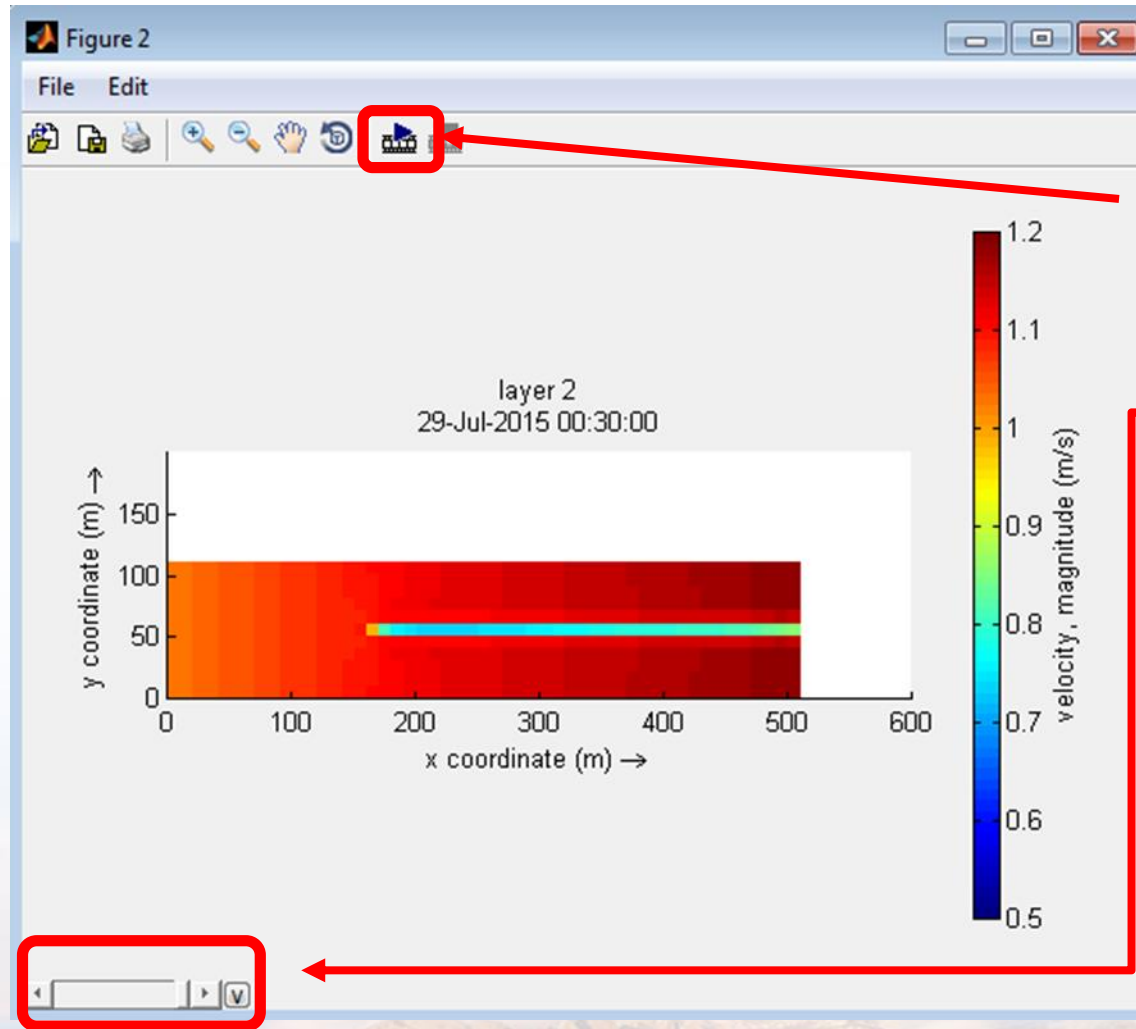
# QUICKPLOT – Velocity Contour



- 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



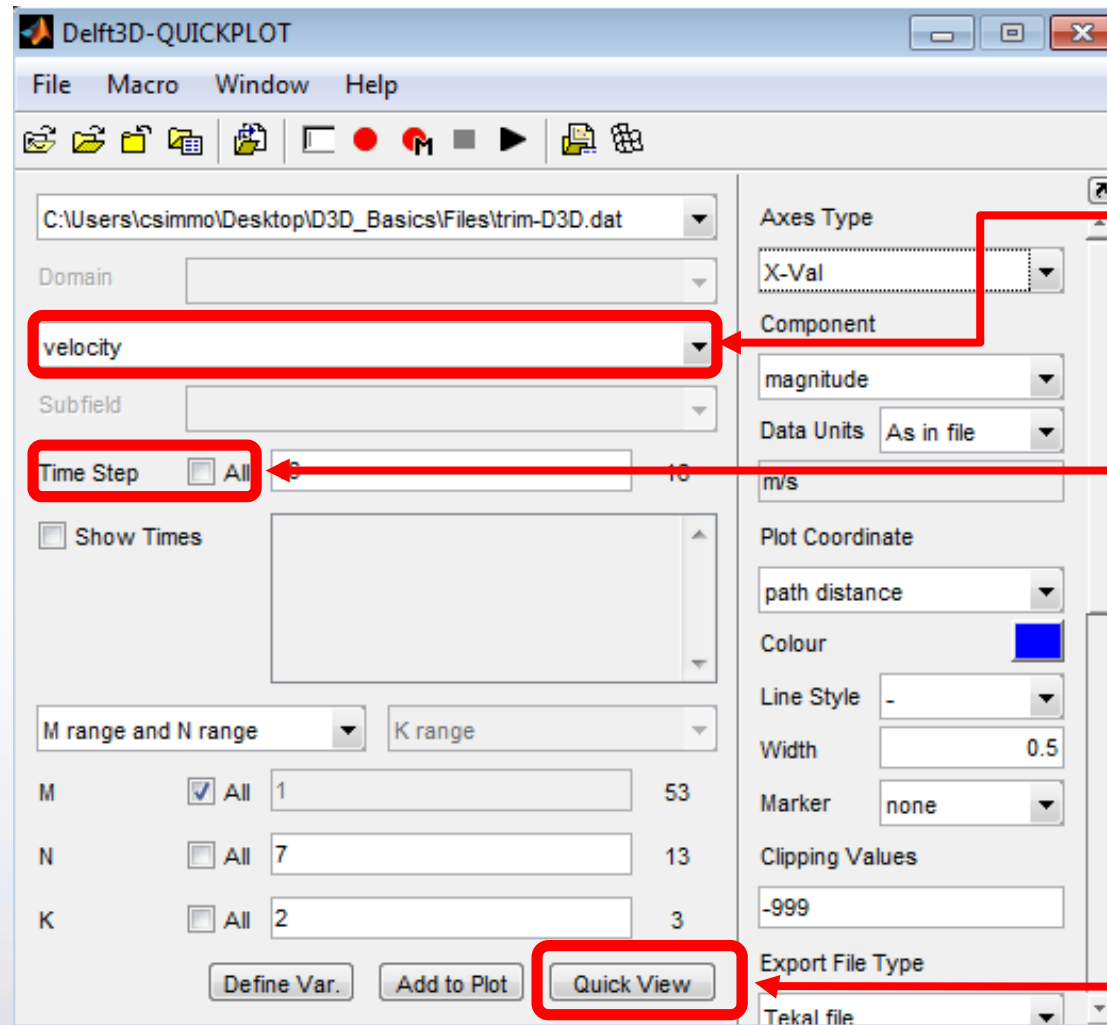
# QUICKPLOT – Velocity Contour



• Select Specify Animation Parameters to animate.

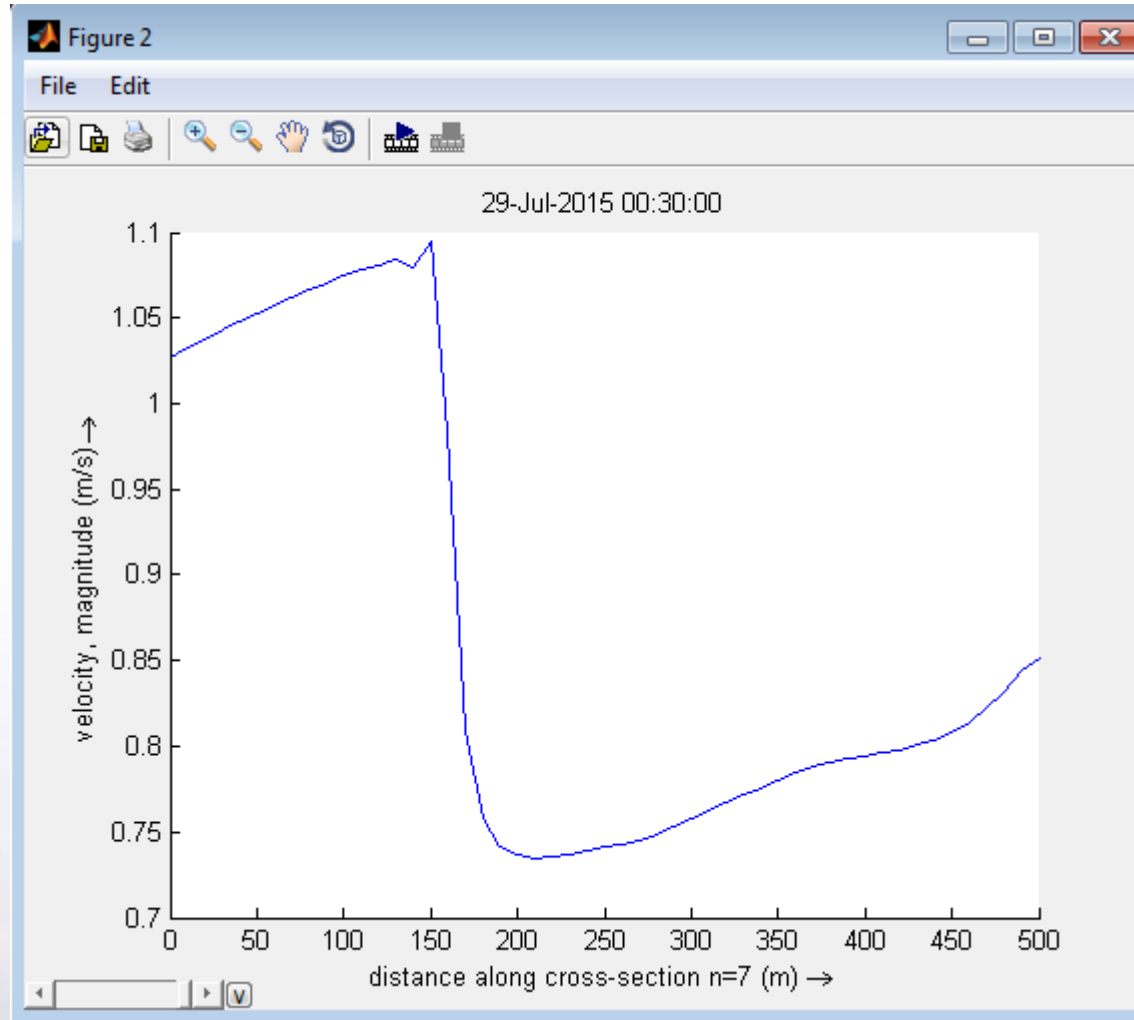
• This bar allows viewing at different times.

# QUICKPLOT – Centerline Velocity



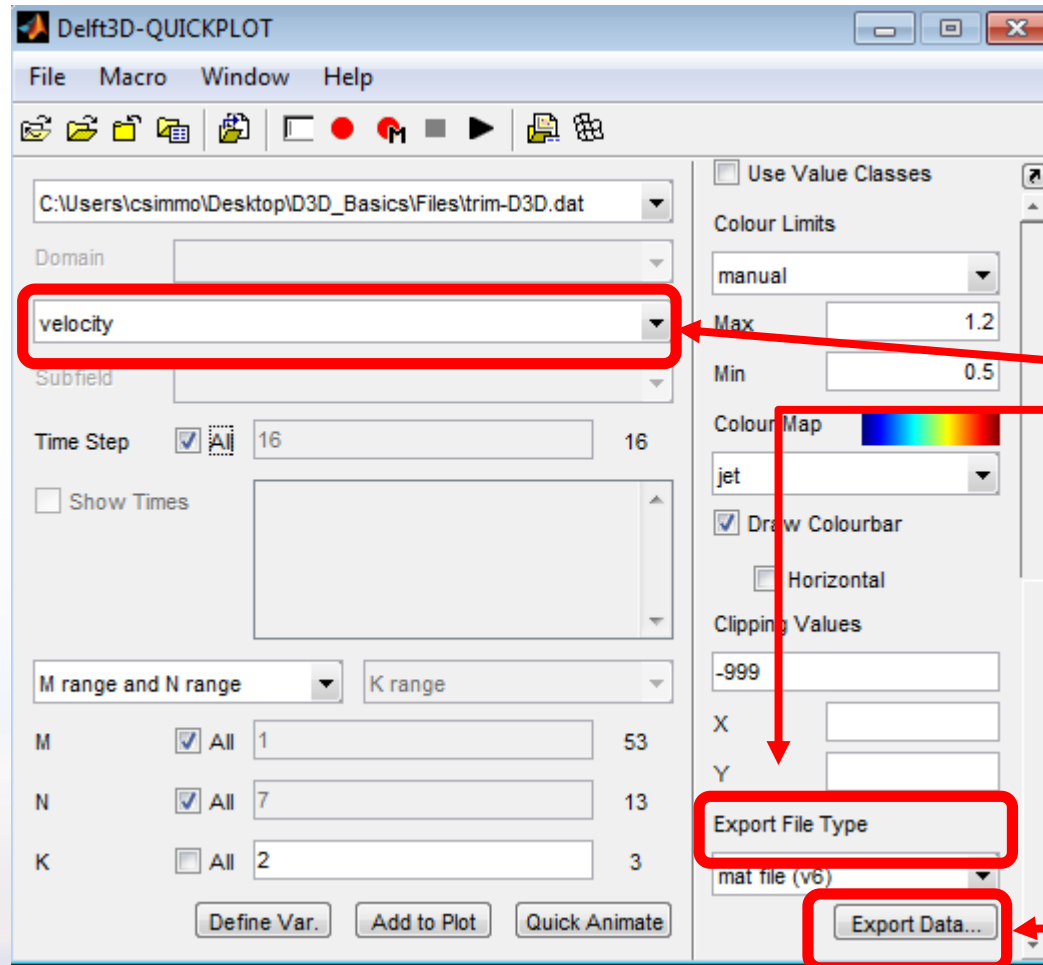
- To view a different plot, such as centerline velocity, return to the QUICKPLOT menu.
- Select velocity
- Uncheck Time Step.
- Choose centerline values for N and K.
- Select Quick View

# QUICKPLOT – Centerline Velocity



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

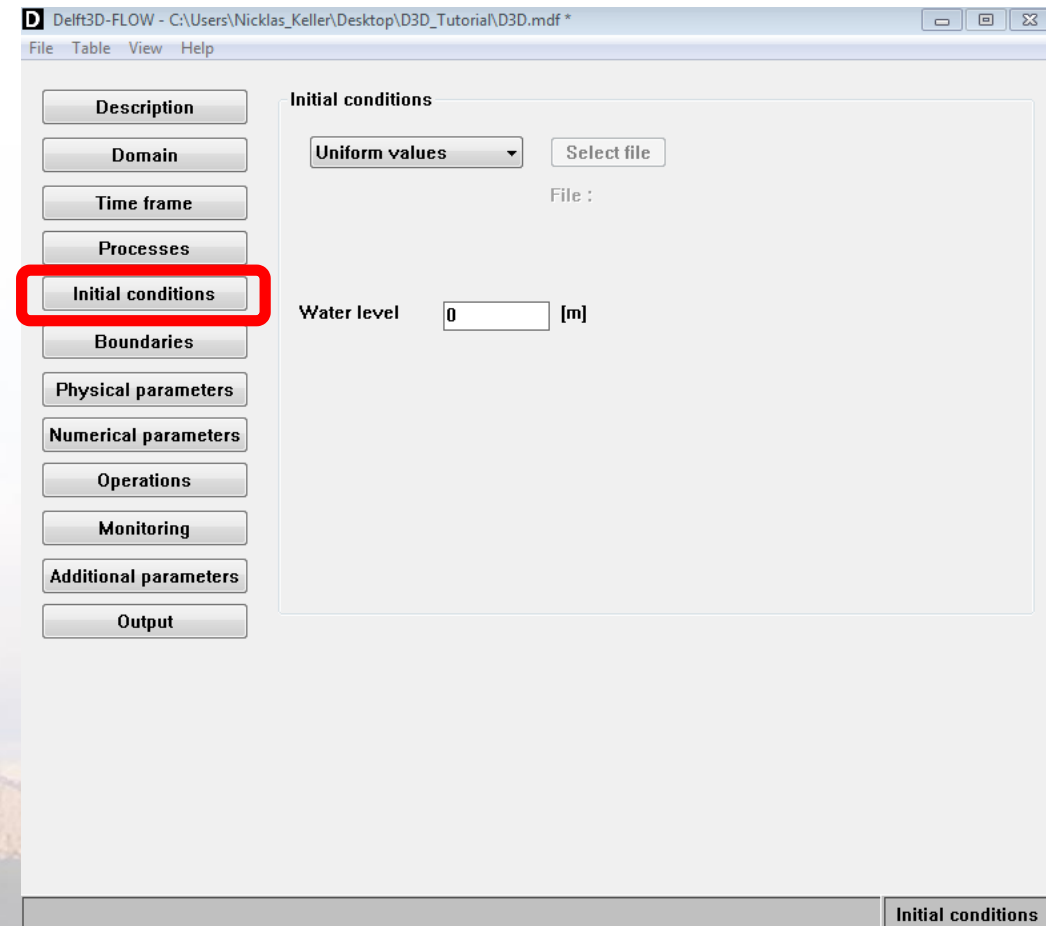
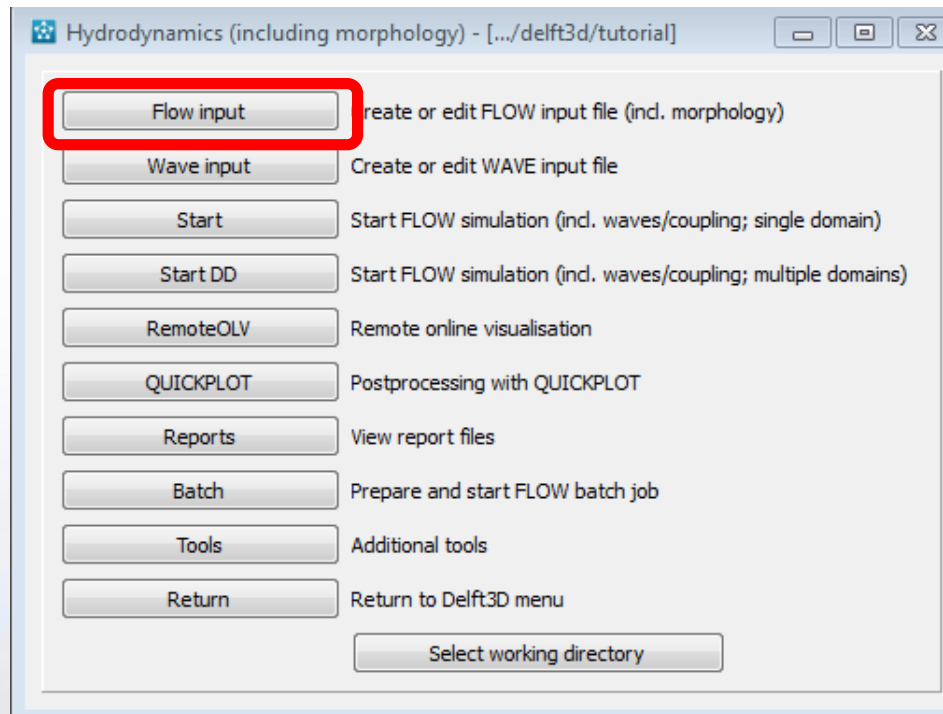
# QUICKPLOT



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

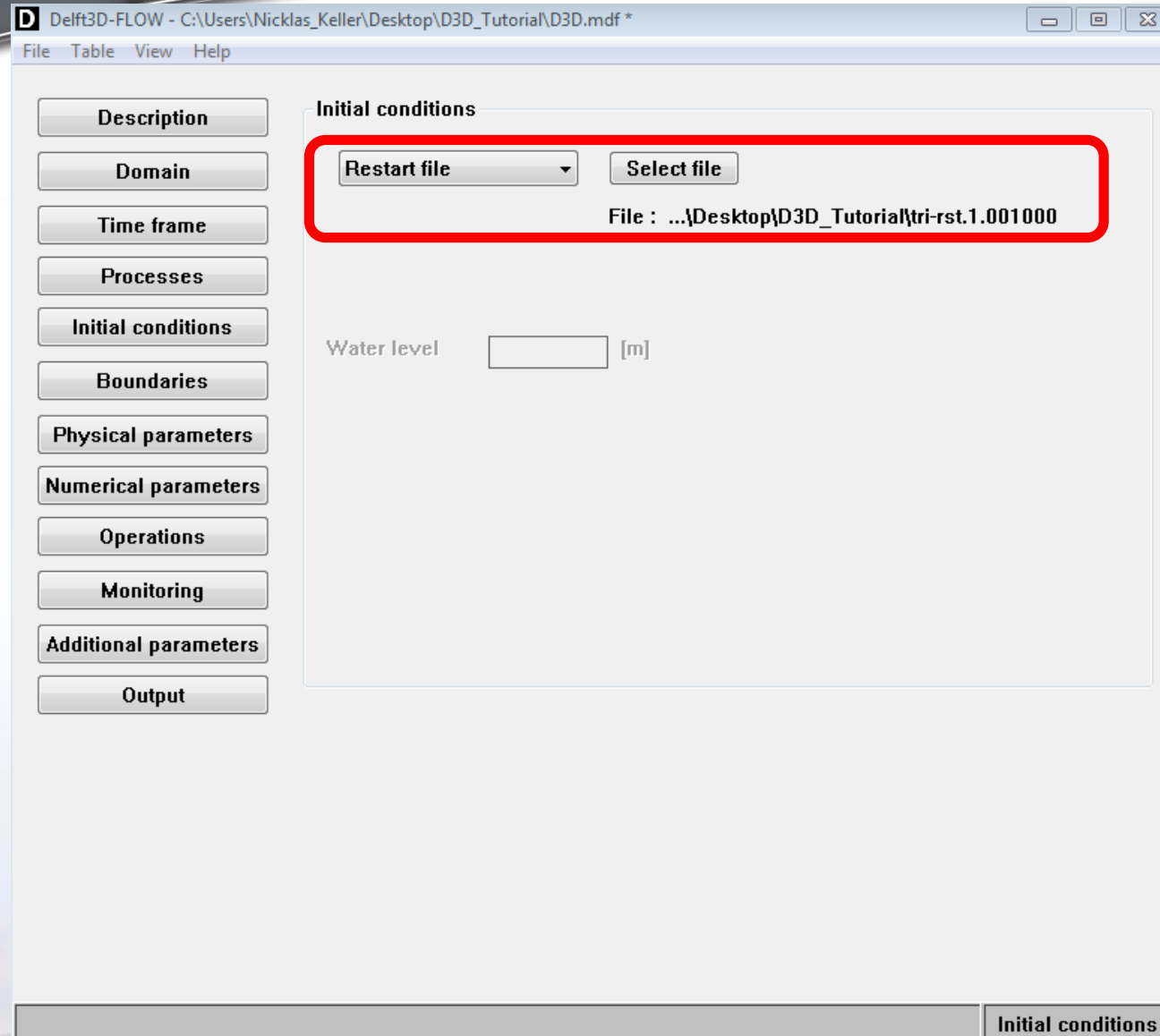
# Starting Simulation from Restart File

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

