

NaysEddy ver 1.0

Example MANUAL

By: Mohamed Nabi, Ph.D.

Copyright 2014 iRIC Project. All Rights Reserved.

Contents

Introduct	tion	3
Getting st	started	4
Simulatio	on of flow over dunes	6
1. Pu	urpose of this example	6
2. Ge	enerate a computational grid	6
3. So	olve the flow	8
3.1.	Grid	9
3.2.	Flow conditions	9
3.3.	Time conditions	10
3.4.	Initial and boundary conditions	10
3.5.	Bed conditions	10
3.6.	Solution conditions	11
3.7.	Hot Start conditions	11
3.8.	Output	12
4. Ru	un the simulation	12
5. Vi	isualize the results	
Simulatio	on of flow over arbitrary topography	19
1. Pu	urpose of this example	19
2. Ge	enerate a computational grid	19
3. So	olve the flow	21
3.1.	Grid	21
3.2.	Flow conditions	22
3.3.	Time conditions	22
3.4.	Initial and boundary conditions	23
3.5.	Bed conditions	23
3.6.	Solution conditions	24
3.7.	Hot Start conditions	24
3.8.	Output	24
4. Ru	un the simulation	25
5. Vi	isualize the results	25

Introduction

This manual is to describe the work with NaysEddy by giving two examples, and explains how the user can solve the flow accurately using iRIC. You can understand how to apply NaysEddy sufficiently if you follow the procedures shown in this manual step by step. The user manual of iRIC and the solver manual for NaysEddy will help you to obtain a better understandings on the contents of this present examples manual.

NaysEddy is compiled under iRIC interface. Therefore, to use NaysEddy, basic knowledge about iRIC is necessary. This manual is based on the assumption that you have installed the iRIC software on your computer. If you have not installed the iRIC software, please download it from the following URL and install it on your computer.

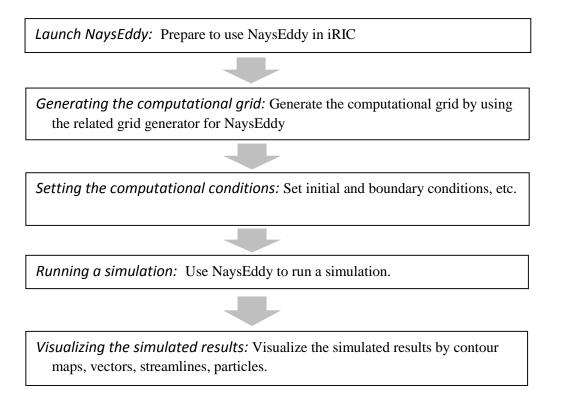
http://i-ric.org/downloads Software: iRIC version2.0

Main contents of this manual are,

Chapter 2. Example of three-dimensional flow computations on cosine-form dunes utilizing NaysEddy Chapter 3. Example of computations on a three-dimensional arbitrary bed topography using NaysEddy

Note : The main purpose of this manual is just to show the practical and concrete examples of computational procedure using NaysEddy. Some detailed physical explanations are thus omitted for simplicity, such as, model descriptions and the physical background of the each phenomenon, etc. The adequate setting of the computational grid is also very important to replicate detailed three-dimensional flow structures.

The following figure shows the steps necessary in simulation of flow using NaysLES.



Getting started

The starting step of the simulation in iRIC is the selection of a suitable solver. In order to select the desired solver, you need prior information about the functions of each solver. It can be obtained from iRIC user manual.

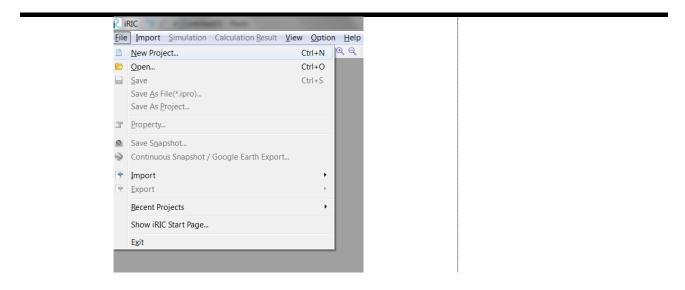
Here, we explain an application to NaysEddy. This solver can be run by opening iRIC, and selecting [Create New Project] from "iRIC Start Page".

Dialog of "iRIC Start Page" \rightarrow [Create New Project] \rightarrow [NaysEddy v.1.0 xxx]

iRIC Welcome to IRICI IRIC can simulate rivers	; from Colorado River to the Nile.
Start Simulation Project	Support
Create <u>New Project</u> Recent Solvers:	© <u>Open Project File</u> Recent Projects:

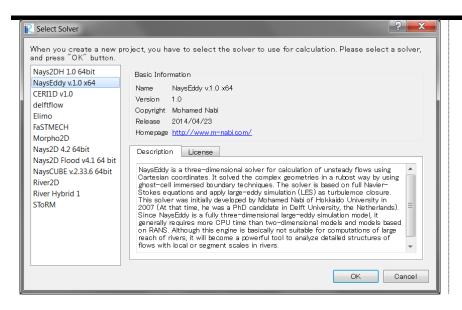
It is also possible to select your solver via menu bar.

Select Menu bar \rightarrow [New Project] \rightarrow [NaysEddy v.1.0 xxx]



A dialog will appear.

Select NaysEddy v.1.0 xxx \rightarrow [OK]



Simulation of flow over dunes

1. Purpose of this example

This is an example for applying NaysEddy in iRIC to calculate the flow on a bed, covered with dunes, using large-eddy simulation. Reading this section enables you to understand the basic operations of NaysEddy in simulating the flow.

2. Generate a computational grid

Generate a computational grid with a rectangular cross section to be applied for NaysEddy solver. The grid for NaysEddy is different than the common grids used in iRIC interface. For NaysEddy, you need to choose the related option carefully. A wrong selection may lead to unpredictable computational errors or instabilities. In order to generate a grid for NaysEddy, you can follow the next steps: Select Menu bar \rightarrow [Grid] \rightarrow [Select Algorithm to Create Grid]. Then from the dialog,

2 X C Select Grid Creating Algorithm Algorithm: Description: "Cartesian Grid for NaysEddy" is a Cartesian grid generator for the solver "NaysEddy". This engine Create grid from polygonal line and widt Create grid from from river survey data generates Cartesian grids on arbitrary topography. The bed topography can be selected from the Create grid by dividing rectangular region library, or can be defined by the user via an input file. The input file includes a uniform Cartesian Create grid by dividing rectangular region Create compound channel grid grid with nodes coordinates and bed elevation (on nodes), but the resolution of the grid is not Cartesian Grid for NaysEddy x64 necessary to be equal to the final grid. Multifunction Grid Genarator Simple Grid Genarator ٠ OK Cancel

[Select grid creating algorithm] \rightarrow [Cartesian Grid for NaysEddy xxx]

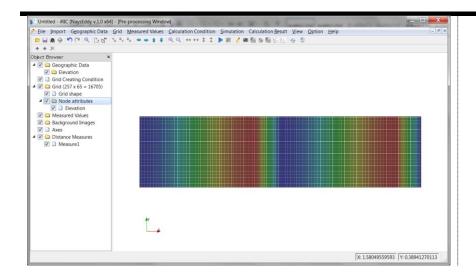
Press on OK. The grid dialog appears. In this example, we solve the flow on cosine form crested bed. The stoss side of the dunes are cosine form and they are followed by lee side with a defined angle.

Grid Creation		Bed topography \rightarrow [2D-Dune
Groups Grid	Geometry Bed topography 2D-Dune cosine V Bedform length [m] 1 Bedform width [m] 0.5 Bedform height [m] 0.1 Angle of lee-side [deg] 30 Number of bedforms 2 2 Bed topography file id the file here	cosine] Bedform length [m] \rightarrow 1 Bedform width [m] \rightarrow 0.5 Bedform height [m] \rightarrow 0.1
	Bed topography file id the file here Grid points Minimum x [m] 0 Minimum y [m] 0 Zero level [m] 0	Angle pf lee-side [deg] \rightarrow 30 Number of bedforms \rightarrow 2
	nx (imax=2°nx) 8 🖗 ny (jmax=2°ny) 6 🖗	Minimum x [m] $\rightarrow 0$
Reset	<u>C</u> reate Grid Cancel	Minimum y [m] $\rightarrow 0$
		Zero level [m] $\rightarrow 0$
		nx (imax=2^nx) \rightarrow 8
		ny (jmax=2^ny) → 6

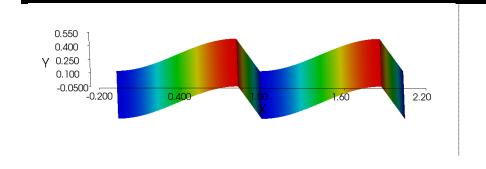
Click on [Create Grid]. You will be asked either you want to map geographic data to the grid attributes. Click on [Yes]. The computational grid will be generated as follows;

C Untitled - iRIC [NaysEddy v.1.0 x64]	- [Pre-processing Window]	_ 🗆 💌
	Grid Measured Values: Calculation Condition: Simulation Calculation Besult View Option Help 옷주 것 = ● ● I ● 의 역 + + + + 후 후 ▶ ■ I ≥ ■ 题 출 题 날 돈 ⊕ = 0	_ # ×
Cheict Bower × ✓ Geographic Data ✓ Geographic Data ✓ Grid Creating Condition 4 ✓ Grid Creating Condition 4 ✓ Grid Cate Condition 4 ✓ Grid Cate Attributes ✓ Grid Cate Attributes ✓ Grid Cate Attributes ✓ Beckground Images ✓ Aves ✓ Measure1		
	X: 1.43653273582 Y	: 0.65894395113

In order to see the counter map of the elevation, check in [Node attributes] and [Elevation] in the "Object browser".

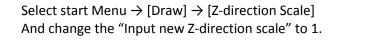


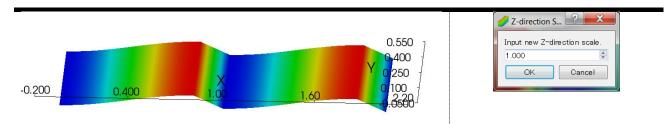
The bed topography can be also seen in a 3D view. In order to see the bed topography in 3D, select



Start menu \rightarrow [Grid] \rightarrow [Open Bird's-Eye View Window]

However, the ratio of bed height to bed with is automatically adjusted. You can change the aspect ratio by





You can rotate the geometry by keeping [Ctrl] key and the right mouse bottom.

3. Solve the flow

As the grid is now ready, you can start the solution of the flow. The first step in the solution of the flow is setting the conditions. The conditions can be set by

Start Menu \rightarrow [Calculation Condition] \rightarrow [Setting] The "Calculation Condition" dialog appear as it is shown in the following Figure.

3.1.Grid

Although, the flow model is three-dimensional, while the generated grid is two-dimensional. In order to change the grid to a three-dimensional form, we add horizontal layers. They are parallel layers starting from a minimum z-level. The value for [Minimum z] has to be smaller than the minimum z-value of the bed geometry. As in the grid generation, we have chosen a zero level of 0, with bed height of 0.1, it makes a minimum value of -0.05 for the bed.

Calculation Condition	? <mark>x</mark>	Minimum z [m] \rightarrow -0.06
Groups Grid Flow conditions Time conditions Initial and B.C. Bed conditions Solution conditi Hot Start conditi Output	Domain Minimum z [m] -0.06 Maximum z [m] 0.2 Grid points nz (kmax=2^nz) 6	Maximum z [m] → 0.2 nz (kmax=2^nz) → 6
Reset	Save and Close Cancel	

3.2. Flow conditions

In this part, you need to enter the physical properties related to the water.

Calculation Condition		? ×	Kinematic viscosity [m2/s] → 1e-6
Groups Grid Flow conditions Time conditions Initial and B.C. Bed conditions Solution conditi Hot Start conditi Output	Kinematic viscosity [m2/s] Density [kg/m3] Gravity [m/s2] Discharge [m3/s]	1e-06 1000 9.81 0.1	Density [kg/m3] \rightarrow 1000 Gravity [m/s2] \rightarrow 9.81 Discharge [m3/s] \rightarrow 0.1
Reset	Save and Clos	Cancel	

3.3. Time conditions

This part sets the starting time and ending time and it determines the time step.

Calculation Condition		? ×	Start time [s] $\rightarrow 0$
Groups Grid Flow conditions Time conditions Initial and B.C. Bed conditions Solution conditions Hot Start conditions Output	Start time [s] End time [s] Time step type CFL number Time step [s]	0 1000 Automatic V 0.35 0.002	End time [s] \rightarrow 1000 Time step type \rightarrow [Automatic] CFL number \rightarrow 0.35
Reset	Save and C	lose Cancel	

3.4. Initial and boundary conditions

The boundary conditions as well as the initial conditions can be set by this option.

Calculation Condition		? ×	B.C. x-direction \rightarrow [Periodic]
Groups Grid Flow conditions Time conditions Initial and B.C. Bed conditions	Boundary conditions B.C. x-direction B.C. y-direction B.C. z-direction	Periodic ▼ Side=wall ▼ Free Surface ▼	B.C. y-direction \rightarrow [Side wall] B.C. z-direction \rightarrow [Free surface]
Solution conditi Hot Start conditi Output	Initial conditions Initial conditions x-velocity y-velocity	Uniform discharge based	Initial conditions \rightarrow [Uniform discharge based]
	z-velocity Include initial perturbations Perturbations amplitude	0 Yes • 0.001	Include initial perturbations → [Yes]
Reset		Save and Close Cancel	Perturbations amplitude \rightarrow 0.001

3.5. Bed conditions

The bed topography, which was generated by the grid generator, can be modified by adding roughness and perturbations to mimic a realistic river bed.

Calculation Condition Groups Grid Flow conditions Time conditions Initial and B.C. Bed conditions Solution conditions Hot Start conditions Output	Bed type Sediment bed Sediment size [mm] 1 Roughness height [mm] 1 Add bed perturbations Yes Amplitude [mm] 0.001	Bed type \rightarrow [Sediment bed] Sediment size [mm] \rightarrow 1 Add bed perturbations \rightarrow [Yes] Amplitude [mm] \rightarrow 0.001
Reset	Save and Close Cancel	

3.6. Solution conditions

The solution conditions, such as the constant for turbulence model, the interpolation method for the immersed boundaries, and coefficients for the multigrid solver, can be set here.

Calculation Condition	<u>ି</u> କ୍ଲିକ୍ଲ୍ର	Smagorinsky constant \rightarrow 0.16
Groups Grid Flow conditions Time conditions Initial and B.C. Bed conditions Solution conditions Hot Start conditions Output	Turbulence Smagorinsky constant 0.16 IBM interpolation Tri−linear Multigrid Multigrid type Cycle index 2 Pre-smoothing 1 Post-smoothing 1 Maximum cycles per time iteration 50 Stop criterion 1e-08	IBM interpolation \rightarrow [Tri-linear] Multigrid type \rightarrow [W-Cycle] Cycle index \rightarrow 2 Pre-smoothing \rightarrow 1 Post-smoothing \rightarrow 1
Reset	Save and Close Cancel	Maximum cycles per time iteration \rightarrow 50

3.7. Hot Start conditions

In the case, you need to start the simulation from a prior generated hotstart file, this option can help you to read the related file.

Groups Grid Flow conditions Time conditions Initial and B.C. Bed conditions Solution conditions Hot Start conditions Output	Hot start No Starting file saved.dat Initialize time No	
---	--	--

3.8. Output

_

Groups Grid Flow conditions Time conditions Initial and B.C. Bed conditions Solution conditions Hot Start conditions Output	Time interval for iRic output [s] 10 Output for hot start Yes Time interval for hot start [s] 10 Tecplot output No Time interval for Tecplot [s] 1 Output folder c:¥temp Output file name output	Time interval for iRIC output \rightarrow 10 Output for hot start [s] \rightarrow [Yes Time interval for hot start [s] 10
Reset	Save and Close Cancel	Tecplot output \rightarrow [No] Output folder \rightarrow c:\temp Output file name \rightarrow output

4. Run the simulation

Now, you are in the state, able to run this simulation. To run the simulation choose,

Calculation Condition	Sin	nulation	Calculation <u>R</u> esult	View	Option
⊕, ⊖, +> >+ ‡ ‡		<u>R</u> un		Ctrl+	R
		<u>S</u> top			
	>-	<u>S</u> olver	Information		
		<u>E</u> xport	solver console log		

Before the simulation starts, iRIC interface ask you either you need to save the project. Click on [Yes] and select [Save as Project]. Make a new directory and save the project in that directory. Be aware that NaysLES requires relatively large space on the harddisk.

🛅 In	formation	Select How to Save Project
i	We recommend that you save the project before starting the solver. Do you want to save?	Please select how to save project from the followings: Save as File (*.ipro) Save as Project
	Yes No Cancel	OK Cancel

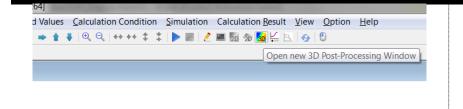
When the simulation starts, a new dialog appear as it is shown in the following figure.

<u>File</u> <u>Import</u> Simulatio	n Calculation <u>R</u> esult <u>V</u> iew <u>O</u> ption <u>H</u> elp	_ 8 ×
🖻 🖬 🙆 🥯 🗳 🍳	🖄 🖄 ½ ⅔ ⅔ 🖛 🔶 🛊 🧠 🧠 ++ ++ ‡ ‡ ▶ 💼 🧷 📾 🚳 🚳 🔛 🚱 🚱	
Step Number	= 5	A
	= 1.030137305765153E-003	
	= 7.620117977336137E-003	
Number of Poisson :	= 7	
Maximum divergence :	= 1.457229122675541E-006	
Total divergence	= -6.649122183099389E-008	
Step Number :	= 6	
Time Step :	= 9.646756459894061E-004 = 8.584793623325543E-003	
Number of Poisson :		
	= 1.223660271693072E-006	
Total divergence	= -3.852807927715811E-007	
Step Number :	= 7	
	= 9.100492734851636E-004	
	= 9.494842896810705E-003	
Number of Poisson		_
	= 1.072498124948806E-006	=
Total divergence	= -4.302554578731318E-007	
Step Number		
	= 8.669353861100247E-004	
	= 1.036177828292073E-002	
Number of Poisson		
	= 8.760383557182561E-007 = -4.800886896342428E-007	
aivergence		
Step Number :	= 9	
Time Step :	= 9 = 8.321196002303659E-004 = 1 119389788315110F-002	
Time :	= 1.119389788315110E-002	

5. Visualize the results

After simulation, you may need to visualize the results. iRIC uses a built-in post-processor which enables the iRIC users to visualize the results without using a third party software.

Select [calculation result] \rightarrow [Open new 3D post-processing window] in menu bar, or click on the button shown in the following figure



A 3D post-processing window will open. You can slide or rotate the object by keeping [Ctrl] key and using they mouse bottoms.

C:\temp\New folder (10) - iRIC [NaysEddy v.1.0 x64] - [P				Mo 🖓 💌 🗙
Eile Import Draw Simulation Animation Calcul			_ # ×	
	। 🔍 🔍 । ++ ++ ‡ ‡ । 🕨 📰 । 🧷 🛤 🗟 😓 😓 🔛	⊕ U		_
🗢 🝽 🗭 🕶 🌆 👘 👘 👘 👘 🖬				Ctrl + 💾 Pan
Vibrot Diverser × Jest Rutzone Jest Rutzone Jest Rutzone Sontrace Diversaria Ocherent Velocity magni Arrow Velocity regional Velocity Velocity Axes				C대 + Coom C대 + Rotate
Ľ,	Time: 80 sec			
Ready		X:	Y:	

In the left side, namely in the "object browser", options as "Contour", "Isosurface", "Arrow", "Streamlines" and "Particles" are available. They are different modules for visualizing the results.

For example, the velocity field can be easily understood by plotting the colors contours of the velocity.

Right click on [Contour] \rightarrow [Properties]

Contour Setting			? ×	Physical Value	\rightarrow	[Velocity
Physical Value: Velocity (magnitude) 🔻	Faces		Color Bar Setting	(magnitude)]		
Value range	Face001	Direction				
✓ Automatic Max: 1.87816 ✓ Fill upper area		⊚ I 🖲 Ј 🔘 К		Click on [Add]		
Min: 0 V Fill lower area		Range				
Division Number: 10		I Min 🗍 📄		Discottica N I		
Colormap		I Max 257 🖨		Direction \rightarrow J		
		J Min 32 荣 J Max 32 荣				
		K Min 1		Range →		
O Custom Setting		K Max 65 🌻		I Min \rightarrow 1		
		💌 Enabled		I Max \rightarrow 257		
Contour Setting Color Fringe Contour Figure Isolines	Add	Remove		J Min \rightarrow 32		
Color Fringe Contour Figure Cisolines	<u></u> uu	Teunose		K Min $\rightarrow 1$		
			OK Cancel			
<u>[</u>				K Max \rightarrow 65		

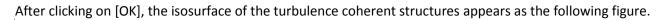
After clicking on [OK], the velocity contour appears as the following figure.

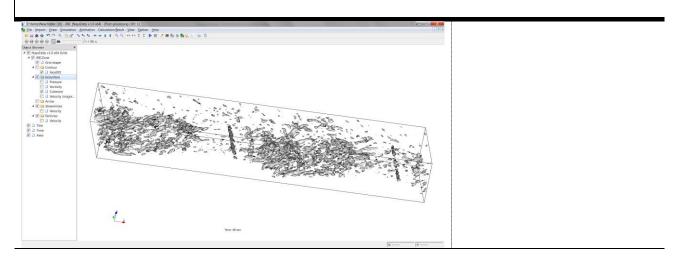
ChtempiNew folder (10) - iRIC (NaysEdd			Annual Property of the	The second se				10	. (e) 🗪
Ele Import Draw Simulation Anir									
- L & - ¹			EC 0 0						
a Drawer * a Drawer * Winder J. Monton * Winder J. Monton									λ
	V	0.000	0.268 0.53	Velocity (magnit 7 0.805 1.0	ude) 7 1.34	1.61	1.88		Ą
	4		1	ne: 80 sec				× W	

The isosurface is suitable to visualizing the flow structures.

Right click on [Isosurface] \rightarrow [Properties]

sosurface Setting	Physical Value \rightarrow [Coherent]
Physical Value: Coherent	
Region	Iso Value \rightarrow 250
✓ Full region	
I Min	
I Max 257 💭	
J Min J	
J Max 65 💭	
K Min 🖓 🔢	
К Мах	
Value Setting	
Iso Value 250	
Min Value -5832.37	
Max Value 5305.09	
OK Cancel	

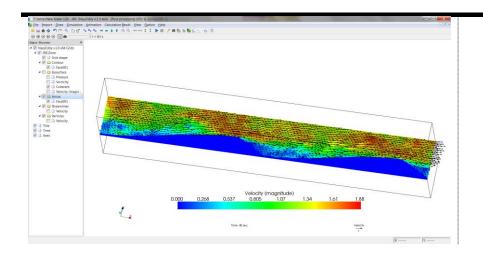




Right click on [Arrow] \rightarrow [Properties]

Arrow Setting	Uncheck [Auto]	
Length Auto Standard value: 1.000	Face001 Direction I I J K Range I Min 1 257 F J Min 32 F	Standard value $\rightarrow 1$ Length on screen $\rightarrow 30$ Minimum value to draw $\rightarrow 0.01$ Sampling rate $\rightarrow 7$
 Sampling rate: 7 Color Custom color By scalar value Pressure 	J Max 32 0 K Min 1 0 K Max 65 0 Enabled	Click on [Add] Direction \rightarrow J
	Add Remove	Range I Min \rightarrow 1 I Max \rightarrow 257 J Min \rightarrow 32 K Min \rightarrow 1 K Max \rightarrow 65

After clicking on [OK], the velocity vectors appear as the following figure. Here, the velocity vectors are drawn together with the velocity contours to make the visualization more clear.

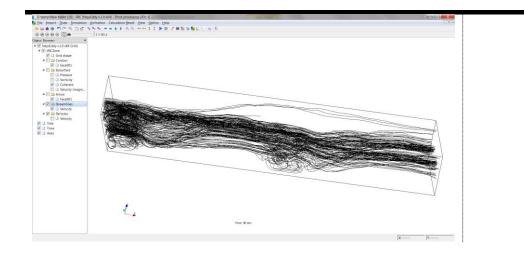


You can also visualize the streamlines.

Right click on [Streamlines] \rightarrow [Properties]

Streamline Setting	Range
1 Start Position	I Min \rightarrow 1
Range Specify the area to start streamlines. 🔞	I Max \rightarrow 1
I Min 1 🕏	J Min \rightarrow 1
I Max 1	J Max → 65
J Min J Max 65 🔄	K Min \rightarrow 1
K Min 🗍 主	K Max \rightarrow 32
К Мах 32 🐑	
Generation space interval	Generation space interval $\rightarrow 1/5$
 	
Color: Width: 😢 1 💿	
Add Remove	
OK Cancel	

After clicking on [OK], the streamlines appear as the following figure.



It is also possible to visualize the one-dimensional profiles (in intersections). For doing this, you need to open a new window by clicking on the diagram icon on the toolbar (the following figure).

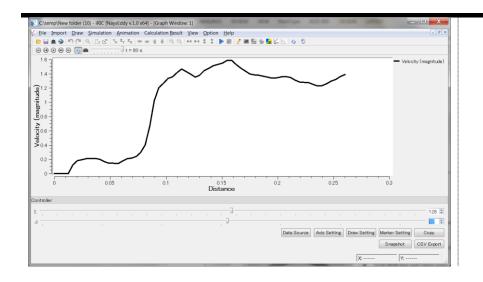
Select on toolbar [Diagram icon] \rightarrow [Click]



After clicking this icon, a dialog window will be open. In this dialog, you can define the desired variable and the intersection. For example, for the velocity, you can do as following figure

Data Source Setting	2	X Axis \rightarrow K
X Avis: Leternal Calculation Result External Three dimensional Data Pressure Vorticity Coherent	Add >> Selected Data Velocity (magnitude) (Add >> Setting OK	Select [Velocity (magnitude)] from left panel and transfer it to the right panel by [Add >>] Click on [Setting] Width → 4
	OK Cancel	

After clicking [OK], a new window appears in which *x*-axis is the disdance in *K*-direction and *y*-axis is the magnitude of the velocity. *I* and *J* coordinated can be determined by the slide bars under the diagram.



Simulation of flow over arbitrary topography

1. Purpose of this example

The purpose of this example to practice the way in which you can import a bed topography which is not included in the library of the solver. Such geometry need to be imported from an external data file.

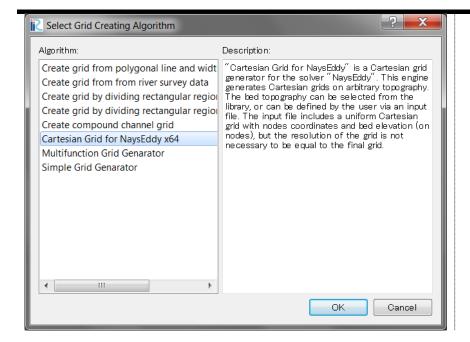
2. Generate a computational grid

Similar to the first example, you need to start NaysEddy. Because of the similarity to the first example, explanation about starting NaysEddy is omitted in this section. After opening NaysEddy, you need to generate a grid. Similar to last example, the grid generation process can be obtained by

Select Menu bar \rightarrow [Grid] \rightarrow [Select Algorithm to Create Grid].

Then from the dialog,

[Select grid creating algorithm] \rightarrow [Cartesian Grid for NaysEddy xxx]

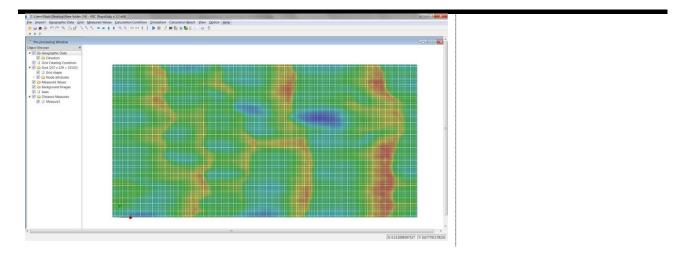


Note that selection of a false grid can lead to instability or immediate determination of solver. After the grid is selected, the "Grid Creation" dialog will be open as shown in the following figure. Here, we import the bed topography from an external file. This file is independent of iRIC and has to be generated manually, or by third party applications. For example, you can use Fortran or C to generate your desired topography. This topography can be the results of calculations or can be imported from experimental data. Here, we included a file for this purpose. This file was generated by calculating the morphodynamical change into a flume experiment. This file can be found attached in the directory.

Crid Creation		Bed topography \rightarrow [Custom]
Groups Grid	Geometry Bed topography Custom Bedform length [m] 1 Bedform width [m] 1 Bedform height [m] 0.1 Angle of lee-side [deg] 30 Number of bedforms 2 € Bed topography file ifform-ex02.dat Grid points 0 Minimum x [m] 0 Zero level [m] 0 nx (imax=2^nx) 8 € ny (jmax=2^ny) 7 €	Bed topography file → Read from harddisk (bedform-ex02.dat) nx (imax=2^nx) → 8 ny (jmax=2^ny) → 7
Reset	<u>C</u> reate Grid Cancel	

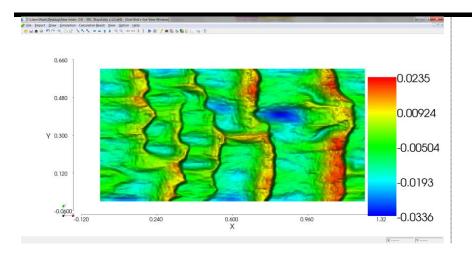
Click on [Create Grid]. You will be asked either you want to map geographic data to the grid attributes. Click on [Yes]. The computational grid will be generated as follows.

 $[\mathsf{Object Browser}] \rightarrow [\mathsf{Node attributes}] \rightarrow [\mathsf{Elevation}]$

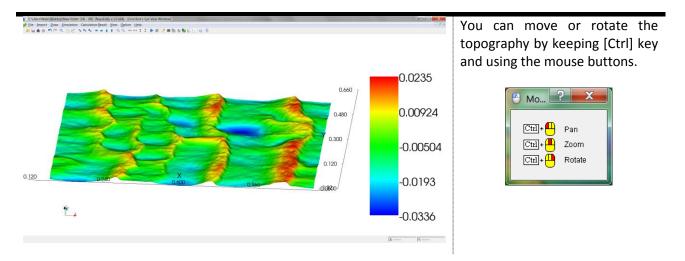


Using bird's-eye option, you can see the bed topography three-dimensionally. Similar to the former example, the bird's-eye view can be found under [Grid] in [Menu bar]

Start menu \rightarrow [Grid] \rightarrow [Open Bird's-Eye View Window]



Start menu \rightarrow [Grid] \rightarrow [Open Bird's-Eye View Window] \rightarrow Mouse Buttons



3. Solve the flow

After grid is generated successfully, you can start to solve the flow on the current grid. The flow conditions can be set, similar to the former example, by selecting [Calculation Condition] from the menu bar.

```
Select Menu bar \rightarrow [Calculation Condition] \rightarrow [Setting]
```

The dialog window for "Calculation Condition" appear as it is shown in the following figures. In the left panel of this dialog (Groups panel), you need to set the flow conditions.

3.1. Grid

The first step is initiating the grid in the vertical direction in order to convert it to a three-dimensional grid. The minimum value has to be smaller than the minimum value of the topography, otherwise the solver gives an error message, and requires you to adjust this value.

Groups Grid Flow conditions Time conditions Initial and B.C. Bed conditions Solution conditi	Domain Minimum z [m] -0.04 Maximum z [m] 0.045 Grid points nz (kmax=2^nz) 6	Maximum z [m] → 0.04 nz (kmax=2^nz) → 6
Hot Start conditi Output	Save and Close Cancel	

3.2. Flow conditions

You can set the physical properties of the flow.

R Calculation Condition	tion	? ×	Kinematic viscosity [m2/s]
Groups Grid Flow conditions Time conditions Initial and B.C. Bed conditions Solution conditi Hot Start conditi Output	Gravity [m/s2] Discharge [m3/s]	1e-06 1000 9.81 0.0068	1e-6 Density [kg/m3] → 1000 Gravity [m/s2] → 9.81 Discharge [m3/s] → 0.0068

3.3. Time conditions

The starting time, calculation time and the time step can be set here. Choosing Automatic time step allows the solver to calculate the suitable time step based of the CFL number. It leads to a safely calculated time step. If you set the time step manually, there is a risk that the time step exceeds the allowed time step locally or globally, which leads to instability. Hence we here prefer to use automatic calculation of the time step to avoid possible instabilities. Selecting smaller CFL number (< 0.35) does not decrease the instability, but increases the computational time. Large values of CFL number (>0.35) may lead to an instable solution.

	? ×	Start time [s] $\rightarrow 0$
Start time [s] End time [s] Time step type CFL number Time step [s]	0 300 Automatic • 0.35 0.002	End time [s] \rightarrow 300 Time step type \rightarrow [Automatic] CFL number \rightarrow 0.35
	End time [s] Time step type CFL number	Start time [s] 0 End time [s] 300 Time step type Automatic • CFL number 0.35

3.4. Initial and boundary conditions

Groups			
Grid	Boundary conditions		B.C. y-direction \rightarrow [Side wall]
Flow conditions Time conditions Initial and B.C. Bed conditions	B.C. x-direction B.C. y-direction B.C. z-direction	Periodic Side-wall Free Surface	B.C. z-direction \rightarrow [Free surface]
Solution conditi Hot Start conditi Output	x-velocity	iform discharge based ▼	Initial conditions → [Uniforn discharge based]
	y-velocity z-velocity Include initial perturbations Perturbations amplitude	0 0 Yes • 0.001	Include initial perturbations - [Yes]
Reset		ave and Close Cancel	Perturbations amplitude \rightarrow 0.00

3.5. Bed conditions

Groups Grid Flow conditions Time conditions Initial and B.C. Bed conditions Solution conditions Hot Start conditions Output	Bed type Sediment size [mm] Roughness height [mm] Add bed perturbations Amplitude [mm]	Sediment bed
---	--	--------------

Bed type \rightarrow [Sediment bed]
Sediment size [mm] \rightarrow 0.25
Add bed perturbations \rightarrow [No]

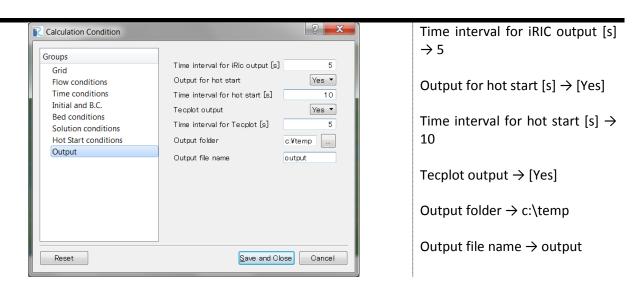
3.6. Solution conditions

Groups	Turbulence	IBM interpolation \rightarrow [Tri-lin
Flow conditions	Smagorinsky constant 0.16	
Time conditions	IBM interpolation Inverse-distance square 💌	Multigrid type \rightarrow [W-Cycle]
Initial and B.C. Bed conditions	Multigrid	
Solution conditi	Multigrid type	Cycle index \rightarrow 2
Hot Start conditi Output	Cycle index 2 💭	Cycle mack 7 2
Output	Pre-smoothing 1 💭	Dro smoothing > 1
	Post-smoothing	Pre-smoothing \rightarrow 1
	Maximum cycles per time iteration 50 🚔	
	Stop criterion 1e-08	Post-smoothing \rightarrow 1

3.7. Hot Start conditions

Calculation Condition		Hot start \rightarrow [No]
Groups Grid Flow conditions Time conditions Initial and B.C. Bed conditions Solution conditions Hot Start conditions Output	Hot start No Starting file saved.dat Initialize time No	
Reset	Save and Close Cancel	

3.8. Output



4. Run the simulation

Now, you are in the state, able to run this simulation. To run the simulation choose,

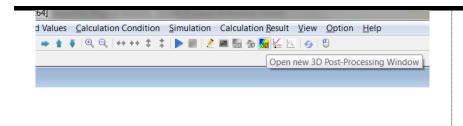
Start menu \rightarrow [Simulation] \rightarrow [Run]

Before the simulation starts, iRIC interface ask you either you need to save the project. Click on [Yes] and select [Save as Project]. Make a new directory and save the project in that directory.

5. Visualize the results

After simulation, you may need to visualize the results. iRIC uses a built-in post-processor which enables the iRIC users to visualize the results without using a third party software.

Select [calculation result] \rightarrow [Open new 3D post-processing window] in menu bar, or click on the button shown in the following figure



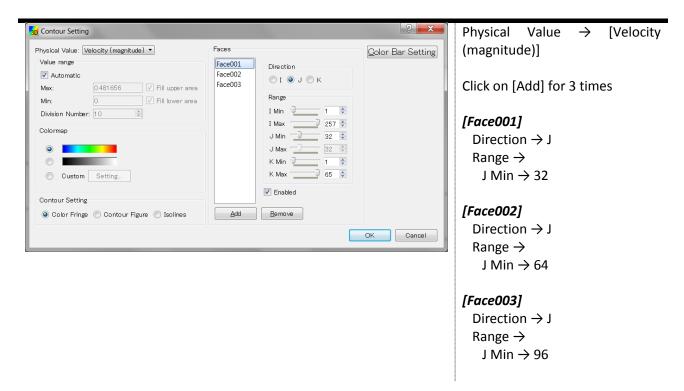
A 3D post-processing window will open. You can slide or rotate the object by keeping [Ctrl] key and using they mouse bottoms.

2 CUMPRIANDRAND/Ana blok (0) - KC (Navid Ar U.B. dt) - Pert growing (0): 1)	🕘 Mo 🖓 💌 🗙
■ De 目 Web U (1 - Web Control Control Decoret Control Decor	
Chief Brown X	
Det Element * * Statution	Ctil + Pan Ctil + Zoom Ctil + Rotate
L.	
True 15 soc	
X	

In the left side, namely in the "object browser", options as "Contour", "Isosurface", "Arrow", "Streamlines" and "Particles" are available. They are different modules for visualizing the results.

For example, the velocity field can be easily understood by plotting the colors contours of the velocity.

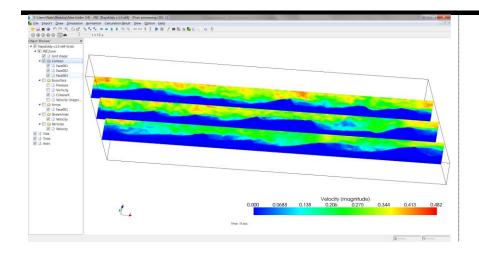
Right click on [Contour] \rightarrow [Properties]



The color bar can be adjusted by clicking on [Color Bar Setting] button on the former dialog.

Color Legend Setting	? <mark>×</mark>	Orientation \rightarrow [Horizontal]
Visible		Number of labels \rightarrow 8
Title : Velo	city (magnitude)	
Orientation : 💿 V	ertical 🧿 Horizontal	Width $\rightarrow 0.5$
Number of labels : 8		
Size	Position	Height $\rightarrow 0.1$
Specify Color bar size as rate against window size.	Specify Color bar left bottom position as rate against window size.	For X and Y, you can drag it by mouse to change the location
Width : 🔞 0.500 🚔	X : 😢 0.462 🚔	
Height : 🕑 0.100 🊔	Y: 🕖 0.099 荣	
0	OK Cancel	

The velocity contour appears as the following figure.

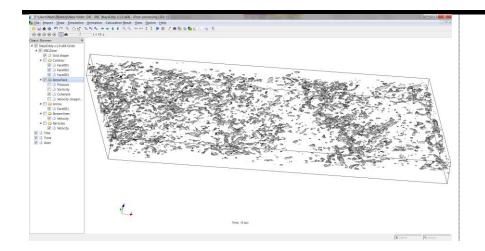


The isosurface is suitable to visualizing the flow structures.

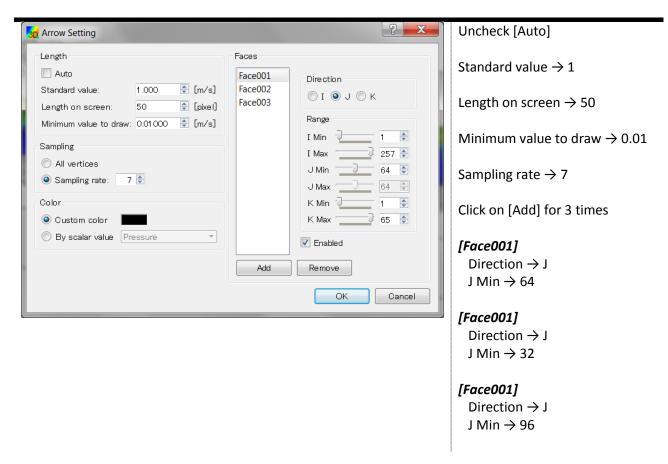
Right click on [Isosurface] \rightarrow [Properties]

Isosurface Setting	Physical Value \rightarrow [Coherent]
Physical Value: Coherent	
Region	Iso Value → 50
V Full region	
I Min 📜 🗍 🗐	
I Max 257 💭	
J Min 🥠 🕺 🗍	
J Max 129 🖤	
K Min .	
K Max 65 荣	
Value Setting	
Iso Value 50	
Min Value -2467.78	
Max Value 900.133	
OK Cancel	

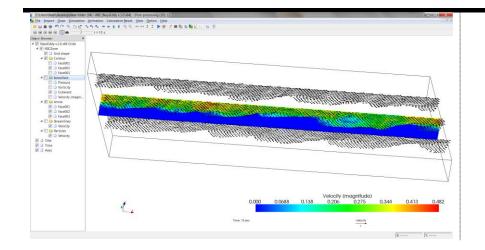
After clicking on [OK], the isosurface of the turbulence coherent structures appears as the following figure.



Right click on [Arrow] \rightarrow [Properties]



After clicking on [OK], the velocity vectors appear as the following figure. Here, the velocity vectors are drawn together with [Face002] of the velocity contours to make the visualization more clear. However, a small overlapping happens between the velocity vectors and the velocity contour section. To avoid this, we draw here the velocity contour in level J = 65 replacing of level 64.

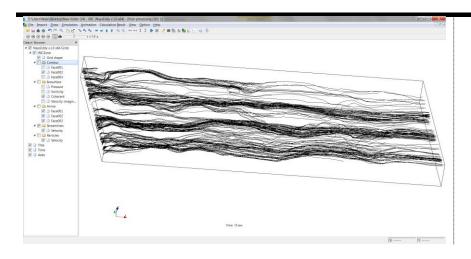


You can also visualize the streamlines.

Right click on [Streamlines] \rightarrow [Properties]

Streamline Setting	Range I Min → 1
Specify the area to start streamlines.	$I Max \rightarrow 1$
I Min 1 1	J Min \rightarrow 1
J Min 1	J Max → 129
J Max 129 🖨	K Min → 1
K Min '	K Max \rightarrow 36
K Max 36 🖨	
Generation space interval	Generation space interval \rightarrow 1/5
Color: 📕 Width: 🔞 1 荣	
Add Remove	
OK Cancel	

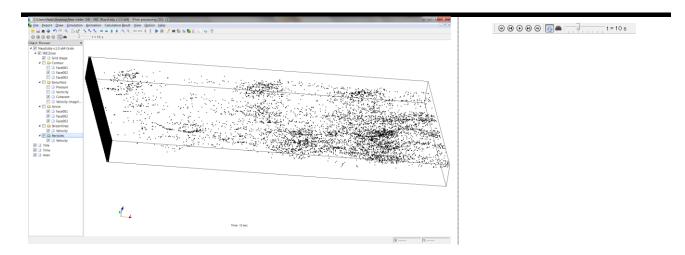
After clicking on [OK], the streamlines appear as the following figure.



It is also possible to visualize the motion of the flow by particles.

5	Particle Setting	Generation time interval $ ightarrow$ 1
	Generation time interval	$I Min \rightarrow 1$
	1 Start Position Range Specify the area to generate particles. 😖	$I \text{ Max} \rightarrow 2$ J Min $\rightarrow 1$
	I Min 1 2 2	J Max \rightarrow 129 K Min \rightarrow 1
	J Max K Min	K Max \rightarrow 65
	K Max 65 9 Generation space interval	Generation space interval $ ightarrow$ 1
	1/5 1 5 Cobr. Size: 3 2	Size \rightarrow 3
	Add Remove OK Cancel	

After clicking on [OK] and change the time step from the side bar, the following figure appear.



You can also make movie from your results. As an example, particles tracing is a suitable choice to observe the trajectory of the flow. In order to generate a movie from your results,

Select menu bar -> [File] -> [Continuous snapshot/Google Earth Export]

		? X	
9	Continuous Snapshot Wizard	Reen	
I	troduction		
	his wizard will generate continuous snapshots.You need to specify som formations for that.	e	
	Next	Cancel	

The dialog for setting the snapshots and the animation will open.

	napshot Wizard	2 - 70	- 15 S
Window Selecti	on		
Please select target	windows and specify	how you want to out	put files.
Target windows			
Post-processing	(3D): 1		
Output file	Layout	Background	
In one file	As is	white	
Respectively	Horizontally	🔘 transparent	
	Vertically		
	vertically		

Specify the place for saving the files and the format of snapshot files.

	2 X
Continue	bus Snapshot Wizard
ile Propert	ies
lease specify	the properties of output files.
Directory	
C:¥Users¥Na	bi¥Desktop
File name	
Prefix :	
	Prefix
Output file	img_
	Suffix length : 4 💭 Format : PNG (*.png) 💌
	Next Cancel

Specify whether you output the animation file of your result.

	vie Prop	the properties of output files.		
	Output mov			
	e name —			_
		File nar	me	
C	utout file			

Specify the start and end time for output of the snapshot files and the animation file.

Continuous Snapshot Wizard		
Continuous Snapshot Wizard Timestep Setting		
Please specify the start time step, stop time step, and the skip rate. start : 0 stop : 0 35.0011		
Skip rate : 1 🐑		
Next Cancel		

After setting all the properties, the dialog shown in left figure will appear. Click on [Finish], then the files will save the place you specified. The right figure shows the dialog which will appear during the output the files.

C\Users\Nabi\Decktop\img_0007.png	Continuous Snapshot
C:\Users\Nabi\Desktop\img_0007.png	16%
C:\Users\Nabi\Desktop\img_0008.png	Cancel