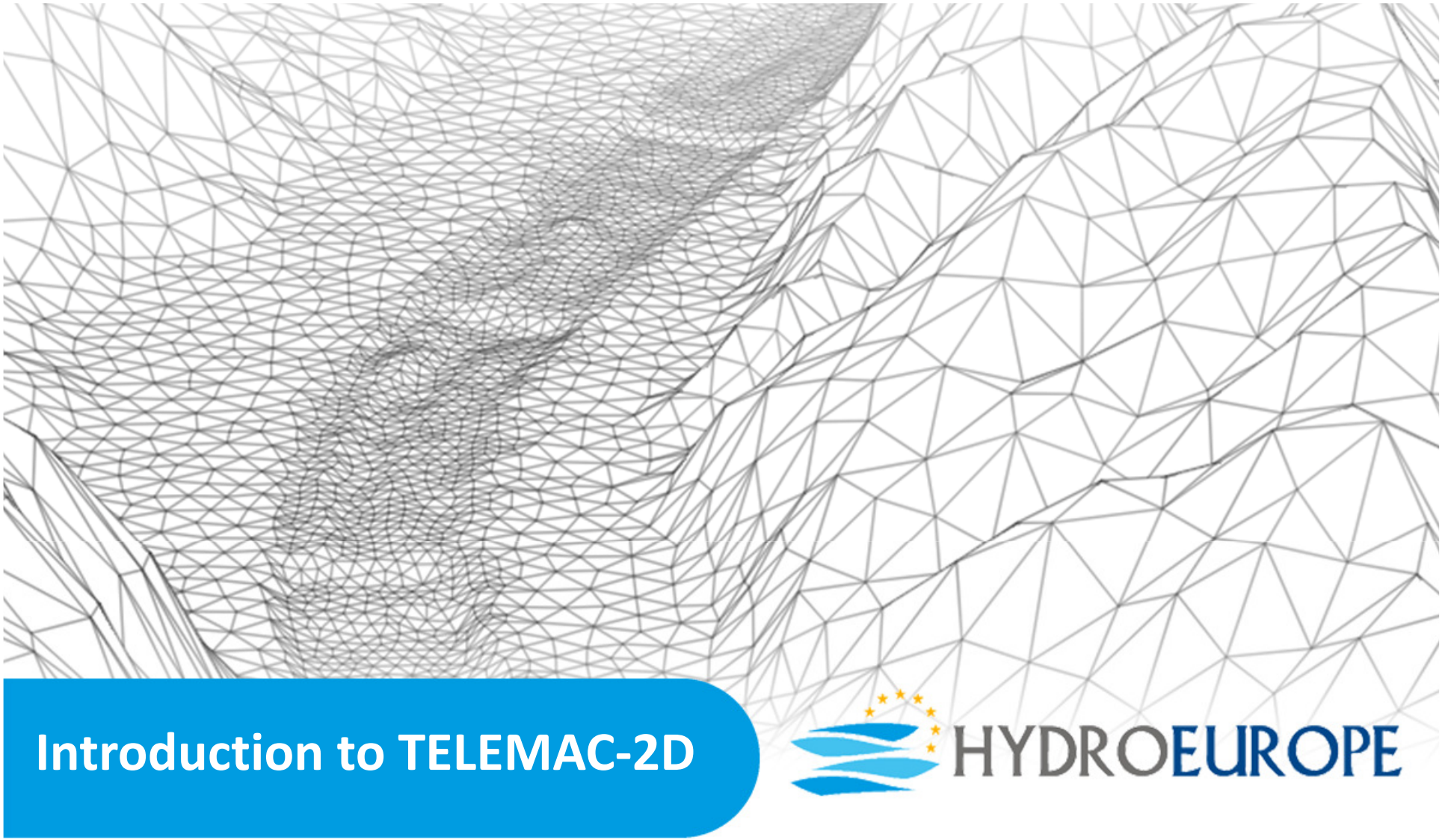


Introduction to TELEMAC-2D

HYDROEUROPE

Erasmus+ UNIVERSITAT POLITÈCNICA DE CATALUNYA Vrije Universiteit Brussel b-tu Brandenburgische Technische Universität Cottbus - Senftenberg Newcastle University POLITECHNIKA WARSZAWSKA Université Nice Sophia Antipolis POLYTECH NICE-SOPHIA

TELEMAC-2D at glance

TELEMAC 2D

- **Industrial code developed by EDF consortium (free and open source)**
- **Based on approximation of 2D (de Saint-Venant Equations) SWEs solutions**
- **Numerical scheme**:

Finite elements (SUPG) or Finite volumes (Roe ou cinétique)

- **Spatial /temporal discretization:**

Non-structured /CFL criterion dependent

- **Wetting and drying treatment:**

Uses threshold value to ensure positivity preservation of computed water depth (mass creation occurs)in finite elements and finite volumes (Roe). The finite volumes cinétique ➔ positivity of solution is ensured

TELEMAC-2D at glance

TELEMAC 2D

- **Flow regime changes treatment:**

Finite elements ➔ upwinding treatment to handle flow regime changes

Finite volumes (Roe and cinétique) ➔ handle numerical discontinuities

- **Operational aspects**

  OS: Windows and Linux

  GUI: No official GUI, third-party tools avaliable

  parallel computing: Yes

- **Scalability:**

The same simulation can be run on your laptop as on university's supercomputer

Open source, means no license limitations (in contrary to DHI Mike21)

Theoretical aspects

TELEMAC2D solves Saint-Venant equations

Additionally user may define a tracer which will be transported by advection-diffusion equation.

A tracer can be a passive pollutant but also heat (temperature).

$$\frac{\partial h}{\partial t} + u\vec{\nabla}(h) + h\,div(\vec{u}) = S_h \qquad \text{continuity}$$

$$\frac{\partial u}{\partial t} + \vec{u}\vec{\nabla}(u) = -g\frac{\partial Z}{\partial x} + S_x + \frac{1}{h}div(h\nu_t\vec{\nabla}u) \qquad \text{momentum along x}$$

$$\frac{\partial v}{\partial t} + \vec{u}\vec{\nabla}(v) = -g\frac{\partial Z}{\partial y} + S_y + \frac{1}{h}div(h\nu_t\vec{\nabla}v) \qquad \text{momentum along y}$$

$$\frac{\partial T}{\partial t} + \vec{u}\vec{\nabla}(T) = S_T + \frac{1}{h}div(h\nu_T\vec{\nabla}T) \qquad \text{tracer conservation}$$

How to use TELEMAC2D?

Telemac computational engine is separated from mesh generator, pre/post-processors

Pros:

• You can prepare simulation with your laptop and run it on a supercomputer

• You can split the job between several people

Cons:

•That’s a bit non-intuitive concept, which comes form ’80s, however it’s still popular between open source models

Mesh generation

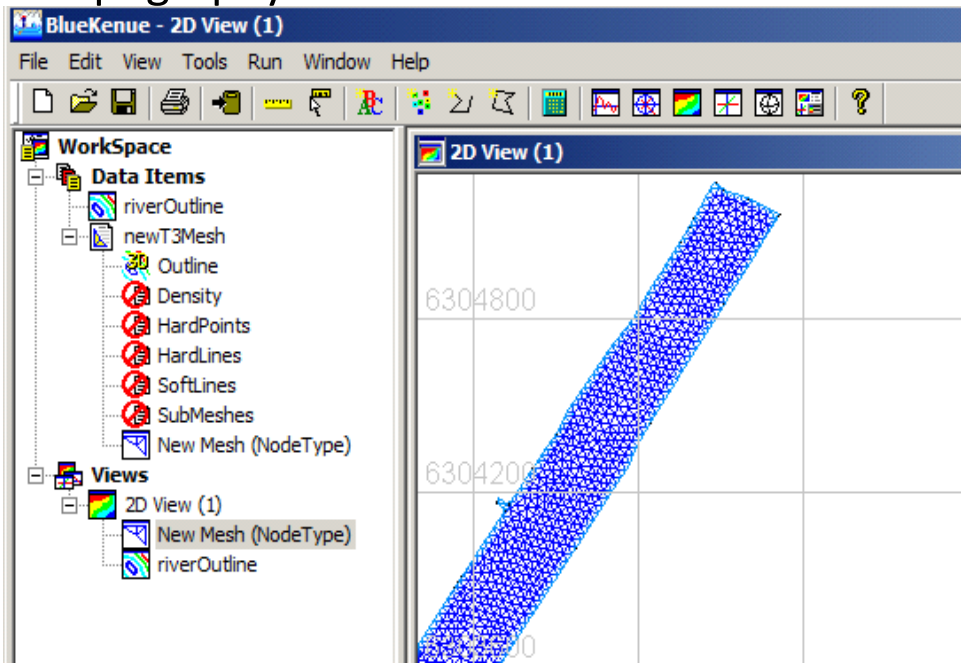
There is no official mesh generator for TELEMAC

Blue Kenue (BK) from Canadian Hydraulic Center (CHC) is a free software with mesh generating capabilities, compatible with TELEMAC

BK accepts the following file formats with input topography:

- *.asc* ARC Info ASCII raster files
- *.shp* ESRI shape files with lines or points
- *.xyz* text files with topography points

All input data should be prepared in the same coordinate system



Case files

Each simulation in TELEMAC is driven by a *case file*

Case files are text files (yet another legacy from ’80s)

Text files are vey convenient if you want to quickly make some tiny changes (like value of one parameter, timestep, etc.)

However maintaining a strict format

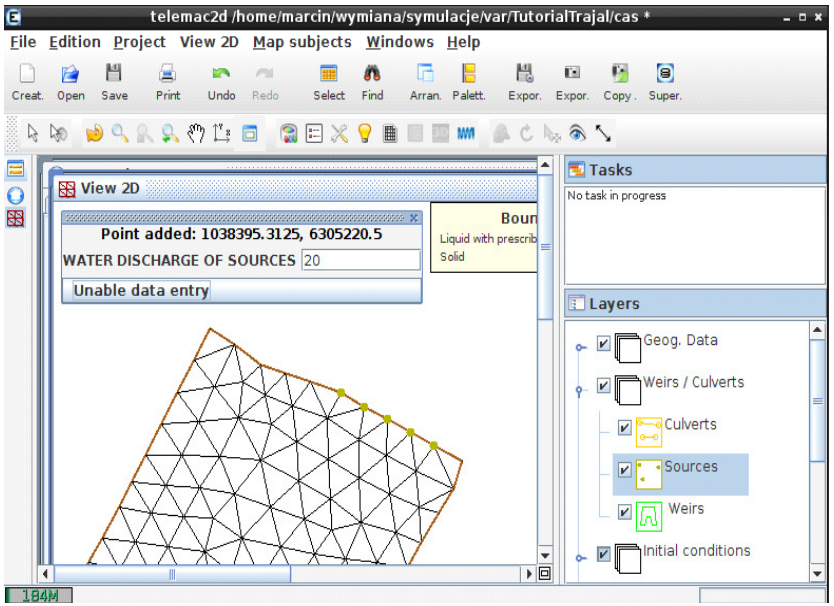
KEY = VALUE is hard for humans

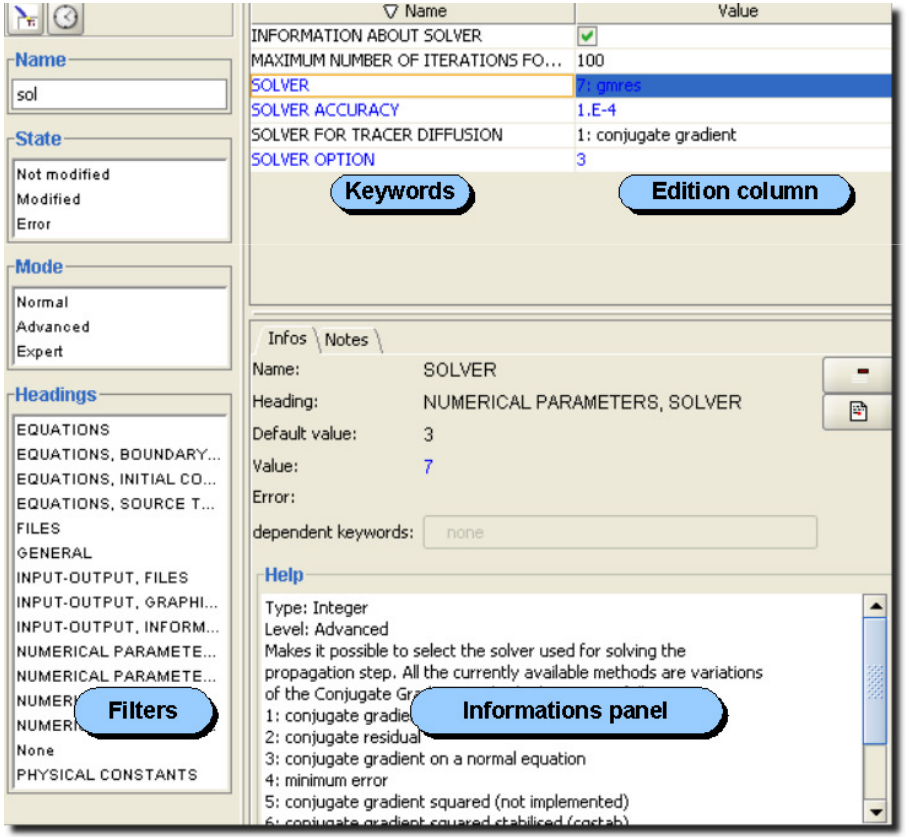
Small typos may lead to annoying errors

```
/-------------------------------------------------
/ INPUT-OUTPUT, FILES
/-------------------------------------------------
STEERING FILE            ='cas1'
GEOMETRY FILE            ='cas1-geo1'
RESULTS FILE             ='res2d-dryDomain'
FRICTION DATA FILE       ='geo1'
BOUNDARY CONDITIONS FILE ='cas1-cas2.conlim'

/-------------------------------------------------
/ INPUT-OUTPUT, GRAPHICS AND LISTING
/-------------------------------------------------
VARIABLES FOR GRAPHIC PRINTOUTS =U,V,US,H,F,L,S
LISTING PRINTOUT PERIOD         =50
GRAPHIC PRINTOUT PERIOD         =500

/-------------------------------------------------
/ NUMERICAL PARAMETERS
/-------------------------------------------------
INITIAL TIME SET TO ZERO =true
```

Pre-processor

For ease of use of case files, a graphical pre-processor Fudaa-prepro can be used to generate or modify case files





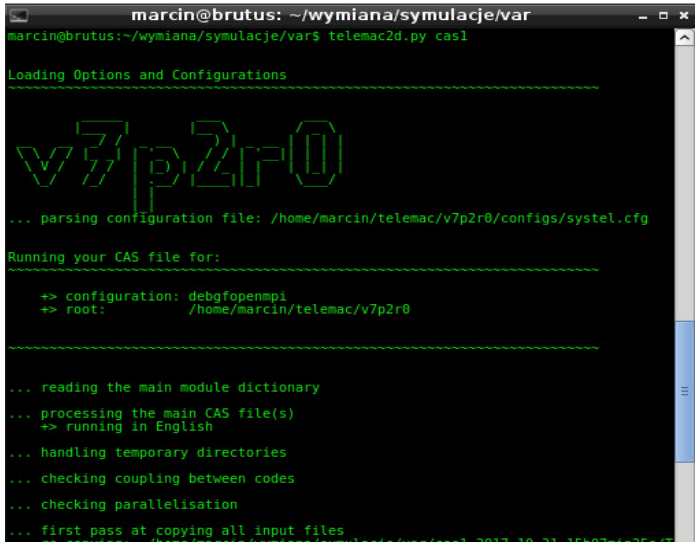
Running a simulation scenario

•Telemac installation requires FORTRAN compiler and several auxiliary tools (python, perl etc.).

•In order to save time on installation and configuration a virtual installation (through VirtualBox) can be used.

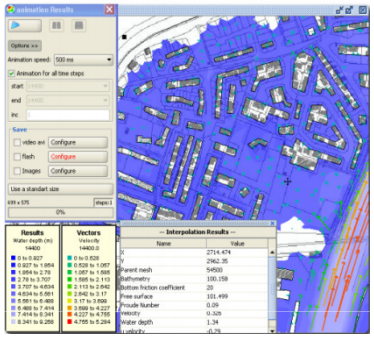
•Telemac is launched from a command prompt (bash shell) by typing

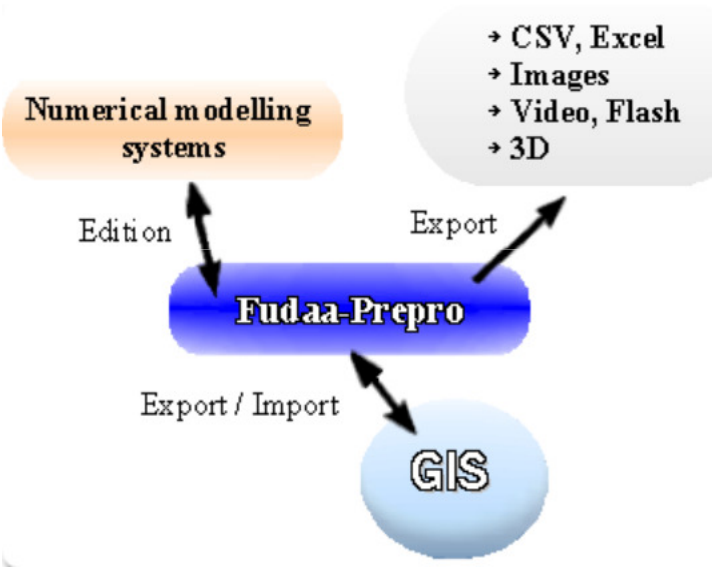
```
telemac2d.py caseFile
```



Results visualisation

For post-processing of Telemac results you can use:
Fudaa Pre-pro, it provides nice animation and reporting capabilities





Blue Kenue – a windows freeware software,
provided by CHC, the same which can be used
for mesh generation

OpenEarthTools – a set of MATLAB, R , Python tools, which are capable of processing Telemac results.

TELEMAC-3D

One of the biggest advantages of Telemac is the possibility to perform 3D simulations with the same mesh

The only required thing is enter the number of layers

3D simulations can be used for water quality simulation, sediment transport

Variable density in 3D simulation also provides capabilities of simulating hot water discharges and fresh-salty water mixing

References

Telemac homepage with comunity forum:

http://opentelemac.org

Fudaa-prepro

http://prepro.fudaa.fr/

Blue Kenue

https://www.nrc-cnrc.gc.ca/eng/solutions/advisory/blue_kenue_index.html

Virtual Box for virtual installation of TELEMAC:

https://www.virtualbox.org/

Open Earth Tools

https://publicwiki.deltares.nl/display/OET/OpenEarth