

Package ‘telemac’

October 14, 2022

Type Package

Title R Interface to the TELEMAC Model Suite

Description An R interface to the TELEMAC suite for modelling of free surface flow. This includes methods for model initialisation, simulation, and visualisation. So far only the TELEMAC-2D module for 2-dimensional hydrodynamic modelling is implemented.

Version 0.1.1

URL <https://github.com/tpilz/telemac>

BugReports <https://github.com/tpilz/telemac/issues>

License GPL (>= 3)

Encoding UTF-8

Imports data.table, dplyr (>= 1.0.0), fs, gstat, magrittr, purrr, raster, Rcpp, rlang, sf, sp, stringr, sys, tidyr (>= 1.0.0), tidyselect

Suggests knitr, rgdal, rmarkdown, RTriangle, stars, terrainmeshr, tidyverse, testthat (>= 3.0.0), covr, spelling

RoxygenNote 7.1.2

VignetteBuilder knitr

LinkingTo Rcpp, RcppArmadillo

Config/testthat/edition 3

Language en-US

NeedsCompilation yes

Author Tobias Pilz [aut, cre] (<<https://orcid.org/0000-0002-5641-3918>>), Potsdam Institute for Climate Impact Research (PIK) [cph, fnd]

Maintainer Tobias Pilz <tpilz@pik-potsdam.de>

Repository CRAN

Date/Publication 2022-02-07 15:50:02 UTC

R topics documented:

cas	2
cli	4
geo	5
interpol	7
lines.t2d_tin	9
line_spacing	9
optionals	12
plot.t2d_geo	13
plot.t2d_res	14
plot.t2d_tin	14
read_cas	15
read_cli	15
read_geo	16
read_msh	17
read_results	17
read_slf_header	18
read_slf_variable	19
results	20
simulate_t2d	21
t2d	22
tin	23
tin2grid	26
write_cas	28
write_cli	29
write_geo	30
write_opt	30
write_results	31
write_slf_header	32
write_slf_variable	32
write_t2d	33
Index	34

cas

Steering file (.cas)*

Description

Initialise a steering file for use within TELEMAC.

Usage

```
cas(x, fname, ...)

## Default S3 method:
cas(x = NULL, fname = NULL, ...)

## S3 method for class 'list'
cas(x, fname = NULL, ...)

## S3 method for class 't2d_cas'
cas(x, fname = NULL, data = NULL, ...)

## S3 method for class 't2d_cas'
print(x, ..., n = 10)
```

Arguments

x	Either: NULL (default), in which case a simple template will be generated; a character string providing the name of an existing steering file; a list with named elements being the steering parameters and their values; an object of class t2d_cas.
fname	character string providing the name of the steering file that is to be generated (can also be used to replace an existing entry).
...	Arguments passed to or from other methods.
data	list that can be given to update an existing object x.
n	Maximum number of steering parameters to print.

Value

An object of class t2d_cas consisting of a list with steering parameters, and an attribute file pointing to a specific steering file.

Examples

```
# template steering parameters
cas_tpl <- cas()

# investigate object
cas_tpl
str(cas_tpl)
class(cas_tpl) # inherits from list

# e.g. subsetting works as with regular lists
cas_subset <- cas_tpl[1:5]
cas_subset

# update cas object (e.g. assign new file name)
cas_updated <- cas(cas_tpl, fname = "test.cas")
cas_updated
```

cli *Boundary (*.cli) object*

Description

Initialise a boundary (*.cli) object for use within TELEMAC.

Usage

```
cli(x, fname, ...)

## S3 method for class 'character'
cli(x, fname = NULL, ...)

## S3 method for class 'numeric'
cli(x, fname = NULL, ...)

## S3 method for class 't2d_geo'
cli(x, fname = NULL, ...)

## S3 method for class 'data.frame'
cli(x, fname = NULL, ...)

## S3 method for class 't2d_cli'
cli(x, fname = NULL, df = NULL, ...)

## S3 method for class 't2d_cli'
print(x, ..., n = 10)

## S3 method for class 't2d_cli'
x[...]
```

Arguments

x	Either: A character string providing the name of an existing boundary file; a numeric vector of ipobo values (see geo) resulting in template boundary values (closed boundary with zero prescribed depths / velocities); an object of class t2d_geo providing ipobo values converted into template boundary values; a data.frame with required columns (see 'Value') giving the boundary parameters; an object of class t2d_cli.
fname	character string providing the name of the boundary file that is to be generated (can also be used to replace an existing entry).
...	Arguments passed to or from other methods.
df	data.frame that can be given to update an existing object x.
n	Maximum number of boundary points to print.

Value

An object of class `t2d_cli` consisting of an attribute `file` pointing to a specific boundary file and a `data.frame` where each row represents a point along the mesh boundary. Columns refer to:

lihbor Depth conditions

liubor Velocity conditions in u direction

livbor Velocity conditions in v direction

hbor Prescribed depth if `lihbor = 5`

ubor Prescribed velocity if `liubor = 6`

vbor Prescribed velocity if `livbor = 6`

aubor Friction coefficient at boundary if `liubor` or `livbor = 2`

litbor Tracer conditions

tbor Prescribed value of tracer if `litbor = 5`

atbor Coefficient of flow relation

btbor Coefficient of flow relation

n Global number of boundary point

k Point number of boundary point numbering; can also represent a node colour

See TELEMAC-2D user manual for more information.

 geo

Mesh / geometry object

Description

Initialise a mesh (geometry) object for use within TELEMAC.

Usage

```
geo(x, fname, ...)
```

```
## S3 method for class 'character'
geo(x, fname = NULL, ...)
```

```
## S3 method for class 'list'
geo(x, fname = NULL, ...)
```

```
## S3 method for class 't2d_geo'
geo(x, fname = NULL, data = NULL, ...)
```

```
## S3 method for class 't2d_tin'
geo(x, fname = NULL, ..., dem, title = "")
```

```
## S3 method for class 't2d_geo'
print(x, ...)
```

Arguments

<code>x</code>	Either: A character string providing the name of an existing geometry file; a list with all required elements (see 'Value'); an object of class <code>t2d_geo</code> ; an object of class <code>t2d_tin</code> (requires input <code>dem</code>).
<code>fname</code>	character, name for the associated geometry file (can also be used to replace an existing entry).
<code>...</code>	Arguments passed to or from other methods: <code>read_geo</code> if <code>x</code> is a file name or <code>interpol</code> if <code>x</code> is a <code>t2d_tin</code> object.
<code>data</code>	list with all required elements (see Value) that can be given to update an existing object <code>x</code> .
<code>dem</code>	If <code>x</code> is of type <code>t2d_tin</code> : a single DEM or a named list with element elevation and further private variables from which the values will be interpolated to the TIN mesh points. Each element will be forwarded to <code>interpol</code> , argument <code>src</code> (see descriptions therein for supported types). Private variables must be a list with elements values (the object passed to <code>interpol</code>), <code>pars_interp</code> (variable-specific parameter list passed to <code>interpol</code> with priority over <code>...</code>), and <code>unit</code> (the unit of the variable). Parameters for interpolation include, e.g., the number of neighbours <code>n</code> , etc.
<code>title</code>	character, mesh title (not necessary, max. 72 characters).

Value

An object of class `t2d_geo` consisting of an attribute file pointing to a specific geometry file and a list with the following elements:

header General mesh information including title, precision (of numbers in the slf file in bytes), the numbers of mesh elements and points, and the number of points per mesh (3 in case of triangles which is the only supported value for now).

tin A mesh object of class `t2d_tin`, see `tin`.

elevation Values of mesh point elevations.

privars A named list of additional, in TELEMAC termed 'private' variables with elements values and `unit`.

Note

Also note the associated `plot.t2d_geo` method.

Examples

```
## Not run:
library(raster)
library(sf)
library(tidyverse)

# get a tin
bnd <- st_read(system.file("dem/boundary_lagos.gpkg", package = "telemac"))
tin_obj <- tin(list(boundary = bnd), s = 90, a = 100^2, q = 30)
```

```

# load raster
dem_rast <- raster(system.file("dem/dem_merit_lagos.tif", package = "telemac"))

# create a geo object (interpolates raster to mesh points)
geo_obj <- geo(tin_obj, dem = dem_rast)
geo_obj
str(geo_obj)

# adjust file name
geo_obj <- geo(geo_obj, fname = "geo.slf")
geo_obj

# plot: mesh elevations interpolated to grid with resolution s
plot(geo_obj, s = 30)

# add additional private variable (in this case Curve Numbers)
dem_priv <- list(
  # mandatory 'elevation' as raster object
  elevation = dem_rast,
  # additional variable 'cn' as (in this case) data.frame
  cn = list(values = as.data.frame(dem_rast, xy = TRUE, na.rm = TRUE) %>%
    select(x, y) %>%
    mutate(z = case_when(
      y > 740000 ~ 95,
      x > 534000 ~ 90,
      (x <= 534000) & (y <= 740000) ~ 80
    )),
  unit = "-",
  # nearest-neighbour interpolation of CN values
  pars_interp = list(n = 1))
)

geo_priv <- geo(tin_obj, dem = dem_priv)

geo_priv
str(geo_priv)
plot(geo_priv, s = 30, v = "cn")

## End(Not run)

```

interpol

Interpolate from source to target locations

Description

Interpolates values from source to target location using inverse distance weighting (IDW) of nearest neighbours.

Usage

```

interpol(trg, src, ...)

## S3 method for class 'SpatialPoints'
interpol(trg, src, ..., z, n = 5, output = "sp")

## S3 method for class 'matrix'
interpol(trg, src, ..., n = 5, output = "numeric")

## S3 method for class 'data.frame'
interpol(trg, src, ..., n = 5, output = "data.frame")

## S3 method for class 't2d_tin'
interpol(trg, src, ..., n = 5, output = "numeric")

```

Arguments

trg	The target locations. Either: a matrix or data.frame with x and y coordinates; an object of class SpatialPoints ; an object of class <code>t2d_tin</code> .
src	The source locations. Either: a matrix with x and y coordinates (argument <code>z</code> as to be provided); a data.frame with x and y coordinates and the variable of interest (" <code>z</code> "); an object of class SpatialPointsDataFrame , <code>raster</code> , or <code>stars</code> (only the first attribute is taken).
...	Further arguments passed to idw .
z	If <code>src</code> is a matrix: numeric vector of values at <code>src</code> locations to be interpolated to <code>trg</code> locations.
n	The number of nearest neighbours used for interpolation (default is 5).
output	The type of output: <code>numeric</code> , <code>sp</code> , or <code>data.frame</code> (see below).

Details

Function calls [idw](#). You can pass further arguments to that function, e.g. `idp` to influence the distance-based weighting of neighbours (default is 2).

Value

`output = "numeric"`: a vector of values interpolated to `trg` locations.

`output = "data.frame"`: a data.frame with x and y coordinates of `trg` location and interpolated values ("`z`").

`output = "sp"`: an object of class [SpatialPointsDataFrame](#) with the interpolated values at `trg` locations.

lines.t2d_tin	<i>Add TELEMAC mesh as lines to a plot</i>
---------------	--

Description

Adds the mesh (triangles) of an object of class t2d_tin as lines to an existing plot.

Usage

```
## S3 method for class 't2d_tin'
lines(x, ...)
```

Arguments

x An object of class t2d_tin.
 ... Arguments passed to [plot](#).

Value

No return value, called for side effects (plot).

line_spacing	<i>Adjust line vertex spacing</i>
--------------	-----------------------------------

Description

Function harmonises the lengths of the segments of lines, i.e. the spacing of vertices.

Usage

```
line_spacing(x, s, output = c("df", "sp"), ...)

## S3 method for class 'data.frame'
line_spacing(
  x,
  s,
  output = c("df", "sp"),
  ...,
  col_x = "x",
  col_y = "y",
  col_line = "line"
)

## S3 method for class 'numeric'
line_spacing(x, s, output = c("df", "sp"), ..., y, line = NULL)
```

```
## S3 method for class 'matrix'
line_spacing(x, s, output = c("df", "sp"), ..., line = NULL)

## S3 method for class 'SpatialLines'
line_spacing(x, s, output = "sp", ...)

## S3 method for class 'SpatialLinesDataFrame'
line_spacing(x, s, output = "sp", ...)
```

Arguments

x	Either: a data.frame with vertex coordinates and line identifier; a numeric vector with the x coordinates of line vertices; a matrix with two columns, the x and y coordinates of line vertices; an object of class SpatialLines*.
s	numeric value giving the target spacing of line vertices (i.e. line segment lengths) in units of the input coordinates.
output	Return either: "df", a data.frame (default) or "sp", an object of class SpatialLines.
...	Arguments passed to or from other methods.
col_x	If x is a data.frame: column with the x coordinates.
col_y	If x is a data.frame: column with the y coordinates.
col_line	If x is a data.frame: column with the line identifier.
y	The y coordinates of line vertices (if x is a numeric vector).
line	numeric vector of identifiers to distinguish individual lines via index (or row) in x (only needed if x is a vector or matrix and more than one line is given).

Value

If output == "df": a data.frame (or [tibble](#)) with elements x, y, and line (or col_x, col_y, col_line) defining the harmonised line(s).

If output == "sp": a [SpatialLines](#) object of the harmonised line(s).

Note

If x is a data.frame, all input arguments referring to columns of x support [quasiquote](#), i.e. they can be specified by their names, quoted or unquoted, or as column position.

Examples

```
# one line given as numeric vectors
x = c(1,1.5,2.5,3,3.2,5, 5.8, 6.5, 7, 6.7, 6, 5.5, 4.8, 4.3, 4, 4, 4.3)
y = c(1,1.8,2,2.8,3.3,4.5, 4.2, 3.8, 3, 2.7, 2.5, 2.5, 2.2, 2.5, 2.8, 3.2, 3.4)
plot(x, y, pch = 16, asp = 1)
lines(x, y)
ldf_harm <- line_spacing(x = x, y = y, s = 1)
points(ldf_harm$x, ldf_harm$y, pch = 16, col = "red")
lines(ldf_harm$x, ldf_harm$y, col = "red")
```

```

# two lines in a matrix
x2 = c(10, 12, 12.2, 11.5, 11.4, 12.5, 13.5, 14)
y2 = c(4, 4.2, 3.5, 3.3, 2.5, 2.2, 2.5, 3)
lmat <- rbind(cbind(x, y), cbind(x2, y2))
line <- c(rep(1, length(x)), rep(2, length(x2)))
plot(x, y, pch = 16, xlim = c(min(lmat[,1]), max(lmat[,1])),
     ylim = c(min(lmat[,2]), max(lmat[,2])), asp = 1)
points(x2, y2, pch = 16)
lines(x, y)
lines(x2, y2)
ldf_harm <- line_spacing(x = lmat, line = line, s = 1)
lh1 <- ldf_harm[ldf_harm$line == 1,]
lh2 <- ldf_harm[ldf_harm$line == 2,]
points(lh1$x, lh1$y, pch = 16, col = "red")
lines(lh1$x, lh1$y, col = "red")
points(lh2$x, lh2$y, pch = 16, col = "red")
lines(lh2$x, lh2$y, col = "red")

# data.frame
library(dplyr)
line1 <- data.frame(
  xcoord = x,
  ycoord = y
)
line2 <- data.frame(
  xcoord = x2,
  ycoord = y2
)
ldf <- bind_rows(line1, line2, .id = "id")
plot(line1, pch = 16, xlim = c(min(ldf$x), max(ldf$x)),
     ylim = c(min(ldf$y), max(ldf$y)), asp = 1)
points(line2, pch = 16)
lines(line1)
lines(line2)
ldf_harm <- line_spacing(ldf, s = 1, col_x = xcoord, col_y = ycoord, col_line = id)
line1_harm <- ldf_harm %>%
  filter(id == 1) %>%
  dplyr::select(xcoord, ycoord)
line2_harm <- ldf_harm %>%
  filter(id == 2) %>%
  dplyr::select(xcoord, ycoord)
points(line1_harm, pch = 16, col = "red")
points(line2_harm, pch = 16, col = "red")
lines(line1_harm, col = "red")
lines(line2_harm, col = "red")

# SpatialLines object
library(sp)
sl <- SpatialLines(list(Lines(Line(line1), 1), Lines(Line(line2), 2)))
plot(sl, asp = 1)
sl_harm <- line_spacing(sl, s = 1, output = "sp")
lines(sl_harm, col = "red")

```

 optionals

Optional input files

Description

Initialise optional input files for TELEMAC-2D.

Usage

```
optionals(x, fname, ...)

## S3 method for class 'character'
optionals(x, fname, ...)

## S3 method for class 'list'
optionals(x, fname, ...)

## S3 method for class 't2d_opt'
optionals(x, fname, ..., vals)

## S3 method for class 't2d_opt'
print(x, ..., n = 10)

## S3 method for class 't2d_opt_LINES'
print(x, ..., n = 10)
```

Arguments

x	Either: a character vector where each element represents a line of content for the optional input file; a list with multiple character vectors to provide the contents for multiple optional files; an object of type <code>t2d_opt</code> to add further optional file(s).
fname	character, file name(s).
...	Arguments passed to or from other methods.
vals	If x is a <code>t2d_opt</code> object: a character vector or a list with the values for the additional optional input file(s).
n	Maximum number of file lines to print.

Value

An object of type `t2d_opt` consisting of a `data.frame` with elements `file`, `file name(s)` of the optional input file(s), and `value`, an object of type `t2d_opt_LINES` that is essentially a list of character vectors where each element represents the contents of an optional input file.

Note

When providing optional input files to a t2d setup the user still needs to add the respective keywords to the steering (cas) file, otherwise the optional input will be ignored! The reason is that many optional input files require additional settings (keywords in the cas file) that cannot be foreseen.

So far only text-based optional input files are supported (e.g. SECTIONS INPUT FILE or FORMATTED DATA FILE) but no binary files.

Examples

```
# t2d_opt object
opt_obj <- optionals(c("# test file", "col1 col2", "1 2", "3 4"), fname = "optional.txt")
opt_obj
str(opt_obj)

# values (lines of the file) as t2d_opt_LINES object
opt_obj$value
str(opt_obj$value)

# multiple optional files
opt_obj <- optionals(list(c("# test file", "col1 col2", "1 2", "3 4"),
                        c("# test file 2", "col1 col2", "5 6", "7 8")),
                    fname = c("optional.txt", "optional2.txt"))
print(opt_obj, n = 4)

# change individual values via list methods
opt_obj$value[[1]][3] <- c("10 20")
opt_obj
```

plot.t2d_geo

Plot TELEMAC geometry

Description

Plots selected objects of class t2d_geo by interpolating the underlying mesh to a regular grid and using `plot` of package `raster`.

Usage

```
## S3 method for class 't2d_geo'
plot(x, s, v = "elevation", ...)
```

Arguments

x	An object of class t2d_geo.
s	numeric value defining the resolution for plotting (the mesh has to be interpolated to a grid for that).
v	character, name of the variable that is to be plotted (only one at a time; default is elevation).
...	Arguments passed to <code>plot</code> .

Value

No return value, called for side effects (plot).

plot.t2d_res	<i>Plot TELEMAC results</i>
--------------	-----------------------------

Description

Plots selected variable and timestep of objects of class t2d_res by interpolating the underlying mesh to a regular grid and using [plot](#) of package [raster](#).

Usage

```
## S3 method for class 't2d_res'
plot(x, s, v = NULL, t = 0, ...)
```

Arguments

x	An object of class t2d_res.
s	numeric value defining the resolution for plotting (the mesh has to be interpolated to a grid for that).
v	character, name of the variable that is to be plotted (only one at a time; default is to take the first variable that can be found).
t	integer, timestep that is to be plotted (only one at a time; default is to plot the first timestep).
...	Arguments passed to plot .

Value

No return value, called for side effects (plot).

plot.t2d_tin	<i>Plot TELEMAC mesh</i>
--------------	--------------------------

Description

Plots the mesh (triangles) of an object of class t2d_tin.

Usage

```
## S3 method for class 't2d_tin'
plot(x, ...)
```

Arguments

x An object of class `t2d_tin`.
 ... Arguments passed to `plot`.

Value

No return value, called for side effects (`plot`).

read_cas	<i>Read steering file (*.cas)</i>
----------	-----------------------------------

Description

Reads the steering file of a TELEMAC project.

Usage

```
read_cas(fname)
```

Arguments

fname character File name of the steering file to be read.

Value

A list with steering parameters and their values.

See Also

To obtain a `t2d_cas` object use function `cas`.

read_cli	<i>Read boundary (*.cli) file</i>
----------	-----------------------------------

Description

Reads the boundary file of a TELEMAC project.

Usage

```
read_cli(fname)
```

Arguments

fname character File name of the boundary file to be read.

Value

A data frame where each row represents a point along the mesh boundary. Columns refer to:

lihbor Depth conditions

liubor Velocity conditions in u direction

livbor Velocity conditions in v direction

hbor Prescribed depth if lihbor = 5

ubor Prescribed velocity if liubor = 6

vbor Prescribed velocity if livbor = 6

aubor Friction coefficient at boundary if liubor or livbor = 2

litbor Tracer conditions

tbor Prescribed value of tracer if litbor = 5

atbor Coefficient of flow relation

btbor Coefficient of flow relation

n Global number of boundary point

k Point number of boundary point numbering; can also represent a node colour

See TELEMAC-2D user manual for more information.

read_geo

Read geometry file (.slf)*

Description

Reads the geometry / mesh information of a TELEMAC project from a SELAFIN file

Usage

```
read_geo(fname, privar = TRUE)
```

Arguments

fname character File name of the geometry file to be read.

privar logical, shall variables in addition to bottom elevation (in TELEMAC denoted as private variables) be imported if available? Default: TRUE.

Details

At least variable BOTTOM (bottom elevations of the mesh) is expected in the geometry file.

Value

A list with header information (see output of [read_slf_header](#)); elevation, the mesh point elevations; privars, named list of additional 'private' variables.

read_msh	<i>Read Gmsh file</i>
----------	-----------------------

Description

Reads the mesh file generated by Gmsh (only format version 2 supported).

Usage

```
read_msh(fname)
```

Arguments

fname character, name of a Gmsh file.

Value

A list with the following elements:

nelem Number of mesh elements (triangles)

npoin Number of mesh points

ikle A nelem x 3 integer matrix of point indices (referring to x and y) defining the mesh elements

ipobo An integer vector of length npoin defining the mesh boundaries (inner boundaries are zero, outer boundaries numbered)

x A vector of length npoin giving the x coordinates of the mesh points

y A vector of length npoin giving the y coordinates of the mesh points

Note

So far only mesh file format version 2 is supported. Besides, only elements 'line' (considered as boundary) and 'triangle' (treated as actual mesh elements) can be handled.

read_results	<i>Read results from *.slf</i>
--------------	--------------------------------

Description

Reads the results of a TELEMAC simulation from a SELAFIN (*.slf) file.

Usage

```
read_results(fname, vars = "all", times = NULL, return_datetime = FALSE)
```

Arguments

fname	character	File name of the results file to be read.
vars		Selection of variables to be read from the file: either "all" (default) reading all variables, "none" giving only the header, a character vector of variable names, or a numeric vector of positions.
times	integer vector,	the timesteps to be read. Passed to read_slf_header .
return_datetime	logical,	return timesteps as datetime (POSIXct) object?

Value

A list with header (see output of [read_slf_header](#)) and values, which is a tidy data.frame where each line represents the value for a certain mesh point (with coordinates x and y) at a certain simulation timestep (note that this might be difficult to interpret if you used variable timestep lengths) for a specific variable.

read_slf_header	<i>Read slf header</i>
-----------------	------------------------

Description

Reads the header of a SELAFIN file (*.slf).

Usage

```
read_slf_header(fname)
```

Arguments

fname	character,	name of a SELAFIN file.
-------	------------	-------------------------

Value

A list with the following elements:

- title** Title of the file
- precision** Precision of numbers (bytes)
- nbv1** Number of variables
- varnames** Names of variables
- varunits** Units of variables
- date** Starting date-time ([POSIXct](#) object)
- ntimes** Number of timesteps
- nelem** Number of mesh elements (triangles)
- npoin** Number of mesh points

- ndp** Number of points per element (3 in case of triangles)
- ikle** A $n_{elem} \times ndp$ integer matrix of point indices (referring to x and y) defining the mesh elements
- ipobo** An integer vector of length $npoin$ defining the mesh boundaries (inner boundaries are zero, outer boundaries numbered)
- x** A vector of length $npoin$ giving the x coordinates of the mesh points
- y** A vector of length $npoin$ giving the y coordinates of the mesh points
- seek_head** Position in the SELAFIN file where the header ends (required by function [read_slf_variable](#))

read_slf_variable *Read slf variables*

Description

Reads the variables stored in a SELAFIN file (*.slf).

Usage

```
read_slf_variable(fname, seek_start, vars, nv, fsize, npoin, times = NULL)
```

Arguments

- | | |
|------------|---|
| fname | character, name of a SELAFIN file. |
| seek_start | integer, starting position to read data from the binary file; can be inferred with function read_slf_header . |
| vars | numeric, vector giving the indices of the variables that shall be read (can be inferred from header information, read_slf_header). |
| nv | integer giving the total number of variables in the file. |
| fsize | integer, precision of the values (number of bytes). |
| npoin | integer, number of mesh points. |
| times | integer vector, the timesteps to be read. If any specified timestep cannot be found in the file an error is raised. Default (NULL): all. |

Value

A list with the following elements:

time Timesteps in seconds relative to date (element in output of [read_slf_header](#))

values An array of dimensions ($npoin \times nv \times \text{length}(\text{time})$) with the values of the variables for each meshpoint and timestep

 results

Results object

Description

Initialise a results object to handle the results of TELEMAC-2D runs.

Usage

```

results(x, ...)

## Default S3 method:
results(x = NULL, fname = "results.slf", ...)

## S3 method for class 'character'
results(x, fname = NULL, log = NULL, ...)

## S3 method for class 't2d_res'
results(x, fname = NULL, log = NULL, ...)

## S3 method for class 't2d'
results(x, ...)

## S3 method for class 't2d_res'
print(x, ..., n = 10)

```

Arguments

x	Either: NULL (default), in which case a simple template without data will be generated; a character string providing the name of an existing results file; an object of class t2d_res (modify attributes); an object of class t2d (read data after model run).
...	Arguments passed to or from other methods, e.g. to read_results such as arguments vars, times, or return_datetime.
fname	character, name for the associated results file (can also be used to replace an existing entry).
log	File name of a log file from a simulation run, if it exists.
n	Maximum number of steering parameters to print.

Value

An object of class t2d_res consisting of an attribute file pointing to a specific results file, an attribute log pointing to the log of a simulation run, and a list with elements

header General information including title, precision (of numbers in the slf file in bytes), the numbers of mesh elements and points, the number of points per mesh (3 in case of triangles which

is the only supported value for now), the variable names and units, and the simulation start date.

tin An object of class `t2d_tin` representing the underlying mesh.

log The log messages (a character vector).

values A data.frame where each line represents the value for a certain mesh point (with coordinates `x` and `y`) at a certain simulation timestep (note that this might be difficult to interpret if you used variable timestep lengths) for a specific variable.

Note

Also note the associated `plot.t2d_res` method.

simulate_t2d	<i>TELEMAC-2D model run</i>
--------------	-----------------------------

Description

Conduct a TELEMAC-2D model run using the model's system command.

Usage

```
simulate_t2d(x, log = "run.log", res = NULL, vars = "all", exec)
```

Arguments

<code>x</code>	An object of class <code>t2d</code> .
<code>log</code>	character, a filename for logging of runtime messages to be created in the project directory (<code>x\$wdir</code>). Default: <code>run.log</code> . Set to <code>NULL</code> if no logfile shall be created (in that case the runlog will not be checked for errors).
<code>res</code>	character, the results file (must be of type <code>*.slf</code>). If <code>NULL</code> (default), the information from the existing steering file will be used or (if the file does not yet exist) will be extracted from <code>x\$res</code> .
<code>vars</code>	Selection of variables to be imported after the simulation: either <code>"all"</code> (default) reading all variables, <code>"none"</code> giving only the header, a character vector of variable names, or a numeric vector of positions. Importing all variables may require large storage capacities (depending on the number of variables, mesh points, and specified output intervals).
<code>exec</code>	character, the TELEMAC-2D executable if not already specified in <code>x</code> .

Value

An object of class `t2d` with new or updated element `res` (an object of class `t2d_res`).

Note

In case the project directory or any of the mandatory input files do not yet exist, this function will first call `write_t2d` and then run the model.

TELEMAC-2D runs are often rather long. During simulation time this command will, by default, write nothing to console but store the output in the specified log file.

In case of large projects with expected long model runtimes it might make more sense to run TELEMAC-2D directly instead of using this function.

t2d

Initialise a TELEMAC-2D project

Description

Initialises a project for 2-dimensional hydrodynamic modelling with TELEMAC-2D.

Usage

```
t2d(title = "", wdir = ".", cas, geo, cli, res = NULL, opt = NULL, exec = NULL)
```

```
## S3 method for class 't2d'
print(x, ...)
```

Arguments

title	character string giving the title of the project (argument TITLE in the steering file).
wdir	character string, the project directory where inputs and outputs will be written to. TELEMAC-2D's input filenames must be relative to this directory! Default: current working directory.
cas	Passed to <code>cas</code> (preferably a <code>t2d_cas</code> object).
geo	Passed to <code>geo</code> (preferably a <code>t2d_geo</code> object).
cli	Passed to <code>cli</code> (preferably a <code>t2d_cli</code> object).
res	Passed to <code>results</code> . Can be the name of an existing results file or left empty at the beginning in which case an empty template object will be generated.
opt	Object of class <code>t2d_opt</code> (if any; default: NULL).
exec	character string specifying the TELEMAC-2D executable (system command). Default: NULL. Only required for <code>simulate_t2d</code> to work.
x	An object of class <code>t2d</code> .
...	Optional arguments passed to <code>print</code> methods.

Details

First, make sure TELEMAC-2D is installed and works!

Value

An object of class t2d, that is a list with elements title, wdir, exec, cas, geo, cli, res, and opt. The latter five are objects of type t2d_*.

Examples

```
library(sf)
library(raster)

# template setup with example data
bnd <- st_read(system.file("dem/boundary_lagos.gpkg", package = "telemac"))
dem_rast <- raster(system.file("dem/dem_merit_lagos.tif", package = "telemac"))
tin_obj <- tin(list(boundary = bnd), s = 90, a = 100^2, q = 30)
geo_obj <- geo(tin_obj, dem = dem_rast)
cli_obj <- cli(geo_obj)
cas_obj <- cas()

# TELEMAC-2D setup
t2d_obj <- t2d("Test setup", "path/to/wdir",
             cas = cas_obj, geo = geo_obj, cli = cli_obj)
t2d_obj
```

tin

TIN object

Description

Initialise a TIN mesh object for use within TELEMAC.

Usage

```
tin(x, ...)

## S3 method for class 'character'
tin(x, ...)

## S3 method for class 'matrix'
tin(x, ..., ikle, ipobo)

## S3 method for class 'list'
tin(x, ..., s, s_brk, a, q = 30)

## S3 method for class 't2d_tin'
print(x, ...)
```

Arguments

x	Either: a character string providing the name of a SELAFIN (*.slf) or a Gmsh mesh (*.msh, format version 2) file of which the mesh will be extracted; a matrix of mesh points with 2 columns containing the x and y coordinates (arguments <code>ikle</code> and <code>ipobo</code> are required); a list with boundary and breakline definitions (see Details).
...	Arguments passed to or from other methods. If x is a list, further arguments passed to <code>triangulate</code> .
ikle	If x is a matrix of points: A matrix with 3 columns of indices referring to rows in x; each row represents a mesh element (triangle). In TELEMAC termed IKLE.
ipobo	If x is a matrix of points: A vector of indices referring to rows in x, each marking a boundary point. In TELEMAC termed IPOBO.
s	numeric, if x is a list: OPTIONAL value giving the resolution of vertices along the boundary line for triangulation. If not given, the points are used as they are supplied, otherwise <code>line_spacing</code> is called to ensure equal spacing of vertices with the given segment lengths.
s_brk	As s but for breaklines.
a	numeric, maximum triangle area; passed to <code>triangulate</code> . Default: squared spacing of points (either given as s or inferred from <code>x\$boundary</code>).
q	numeric, minimum triangle angle; passed to <code>triangulate</code> . Default: 30 degrees.

Details

If x is a list this function creates a Triangulated Irregular Network (TIN) using function `triangulate`. The following list elements are required to perform the triangulation:

boundary A matrix, data.frame, SpatialLines* or sf object with two columns, each row defining a point along the outer catchment boundary. Points are connected one-by-one to a line starting with the first point, i.e. make sure points are in the right order! The first and last point will be connected to close the boundary.

breaklines OPTIONAL, a matrix, data.frame, SpatialLines* or sf object with three columns x and y, the x and y coordinates of vertices along the breaklines, and line, an identifier to identify individual breaklines.

Value

An object of class `t2d_tin`, which is a list with the following elements:

points A matrix with the x and y coordinates (as columns) of mesh points.

triangles A matrix with 3 columns of indices referring to rows in points; each row represents a mesh element (triangle).

edges A matrix with 2 columns of indices referring to rows in points, the node points; each row represents an edge / segment of a triangle.

boundaries A vector of indices referring to rows in points, each marking a point of the outer catchment boundary.

breaklines A matrix with 2 columns of indices referring to rows in points, the vertices of the breaklines (used for mesh refinement during triangulation).

Note

Duplicated mesh points are silently removed.

Make sure breaklines do not intersect as this is not supported by the Triangle algorithm. A possible workaround to split intersecting breaklines in R using `sf` is shown in the examples.

If you want to construct a `t2d_tin` object and get the error Boundary points do not comply with requirements: [...] the reason might be that breaklines are too close to the boundary causing that points of the breaklines are used as boundary points which eventually results in a discontinuous outer boundary. Try to increase the distance of breaklines to the catchment boundary.

Examples

```
### BASIC FUNCTIONALITY ###
library(sf)

# load boundary as sf linestring
bnd <- st_read(system.file("dem/boundary_lagos.gpkg", package = "telemac"))

# create t2d_tin object
tin_obj <- tin(list(boundary = bnd), s = 90, a = 100^2, q = 30)

# inspection
tin_obj
str(tin_obj)
plot(tin_obj, pch = ".")

### DEALING WITH INTERSECTING BREAKLINES ###
library(sf)
library(tidyverse)

# example boundary and with intersecting breaklines
test_bnd <- st_linestring(
  matrix(c(seq(0,100,5), rep(100,21), seq(100,0,-5), rep(0,21),
           rep(0,21), seq(0,100,5), rep(100,21), seq(100,0,-5)),
         ncol = 2)
) %>% st_sfc()
test_brk <- list(
  st_linestring(matrix(c(seq(0,100,5), rep(50,21)), ncol = 2)),
  st_linestring(matrix(c(rep(50,21), seq(0,100,5)), ncol = 2)),
  st_linestring(matrix(c(seq(30,60,5), rep(60,11),
           rep(20,7), seq(20,70,5)), ncol = 2))) %>% st_sfc()

# get intersection points and define buffer of 2 around these points
pt_inters <- c(test_bnd, test_brk) %>%
  st_intersection() %>%
  st_collection_extract(type = "POINT") %>%
  st_buffer(2)
```

```

plot(test_bnd)
plot(test_brk, add = TRUE)
plot(pt_inters, add = TRUE)

# split breaklines
test_brk_unique <- st_difference(st_union(test_brk), st_union(pt_inters))
plot(test_bnd)
plot(test_brk_unique, add = TRUE)

# create mesh
tin_obj <- tin(list(boundary = test_bnd, breaklines = test_brk_unique),
              s = 2, s_brk = 2, a = 4, q = 30)
plot(tin_obj, pch = ".")

```

tin2grid

Interpolate TIN-based mesh values to grid

Description

Linearly interpolates the values of a mesh based on irregular triangles to a regular grid.

Usage

```

tin2grid(x, s, output, ...)

## S3 method for class 'data.frame'
tin2grid(
  x,
  s,
  output = c("data.frame", "list", "raster"),
  ...,
  col_x = "x",
  col_y = "y",
  col_z = "z",
  tinmat
)

## S3 method for class 't2d_geo'
tin2grid(
  x,
  s,
  output = c("data.frame", "list", "raster"),
  ...,
  v = "elevation"
)

## S3 method for class 't2d_res'

```

```

tin2grid(
  x,
  s,
  output = c("data.frame", "list", "raster"),
  ...,
  v = NULL,
  t = NULL
)

```

Arguments

<code>x</code>	Either: a <code>data.frame</code> with mesh coordinates and elevations; an object of class <code>t2d_geo</code> or <code>t2d_res</code> .
<code>s</code>	numeric value defining the resolution of the output grid.
<code>output</code>	character, either: <code>"data.frame"</code> (default), <code>"list"</code> , or <code>"raster"</code> to return a raster object. If multiple variables <code>v</code> or timesteps <code>t</code> are given, a <code>data.frame</code> will be returned in any case.
<code>...</code>	Arguments passed to or from other methods.
<code>col_x</code>	If <code>x</code> is a <code>data.frame</code> : column with the <code>x</code> coordinates.
<code>col_y</code>	If <code>x</code> is a <code>data.frame</code> : column with the <code>y</code> coordinates.
<code>col_z</code>	If <code>x</code> is a <code>data.frame</code> : column with the mesh values.
<code>tinmat</code>	integer matrix of point indices (referring to <code>x</code> and <code>y</code> coordinates) defining the mesh elements (such as element triangles of a <code>t2d_tin</code> object).
<code>v</code>	character, name of the variable(s) that is/are to be extracted and interpolated (default all variables that can be found). If <code>x</code> is of class <code>t2d_geo</code> elevation will be taken by default.
<code>t</code>	integer, timestep(s) that is/are to be extracted and interpolated (default: all timesteps).

Value

If `output == "data.frame"`: A `data.frame` with:

x or `col_x` X coordinates of the output grid

y or `col_y` Y coordinates of the output grid

z or `col_z` Interpolated values

variable OPTIONAL (more than one given): imported variable the current values refer to

timestep OPTIONAL (more than one given): simulation timestep the current values refer to

If `output == "list"`: A list with:

x X coordinates of the output grid

y Y coordinates of the output grid

z Matrix with interpolated values, where `z[i, j]` points to coordinates `x[i]` and `y[j]`

If `output == "raster"`: An object of class [raster](#).

Note

If you import many variables or timesteps or the mesh is huge or s very small the resulting dataset might become excessively large (especially if output is a `data.frame`)!

If x is a `data.frame`, all input arguments referring to columns of x support [quasiquote](#), i.e. they can be specified by their names, quoted or unquoted, or as column position.

Examples

```
# plot model bathymetry with ggplot
library(raster)
library(sf)
library(tidyverse)
# t2d_geo object
bnd <- st_read(system.file("dem/boundary_lagos.gpkg", package = "telemac"))
tin_obj <- tin(list(boundary = bnd), s = 90, a = 100^2, q = 30)
dem_rast <- raster(system.file("dem/dem_merit_lagos.tif", package = "telemac"))
geo_obj <- geo(tin_obj, dem = dem_rast)
# interpolate to regular grid as df and plot
geo_df <- tin2grid(geo_obj, s = 90, output = "data.frame")
ggplot(geo_df, aes(x = x, y = y, fill = value)) +
  geom_raster() +
  coord_equal() +
  scale_fill_viridis_c()
```

write_cas

Write steering file (.cas)*

Description

Writes the steering file for a TELEMAC project.

Usage

```
write_cas(x, ...)
```

S3 method for class 'data.frame'

```
write_cas(x, fname, ...)
```

S3 method for class 'list'

```
write_cas(x, fname, ...)
```

S3 method for class 't2d_cas'

```
write_cas(x, ...)
```

S3 method for class 't2d'

```
write_cas(x, ...)
```

Arguments

x	Either: An object of class <code>t2d_cas</code> ; a list with elements as in a <code>t2d_cas</code> object; a <code>data.frame</code> with columns <code>key</code> and <code>value</code> giving the steering parameters; an object of class <code>t2d</code> with element <code>cas</code> .
...	Arguments passed to or from other methods.
fname	character, a file name (extension <code>.cas</code>) where the steering parameters should be written to.

Value

Returns input `x` invisibly.

Note

An existing steering file will be silently overwritten.

Examples

```
## Not run:
# creates test.cas in current working directory
cas_tpl <- cas()
cas_tpl <- cas(cas_tpl, fname = "test.cas")
write_cas(cas_tpl)

## End(Not run)
```

write_cli	<i>Write boundary file (*.cli)</i>
-----------	------------------------------------

Description

Writes boundary conditions for a TELEMAC simulation.

Usage

```
write_cli(x)

## S3 method for class 't2d_cli'
write_cli(x)

## S3 method for class 't2d'
write_cli(x)
```

Arguments

x	An object of class <code>t2d_cli</code> or <code>t2d</code> .
---	---

Value

Returns input x invisibly.

write_geo	<i>Write geometry file (*.slf)</i>
-----------	------------------------------------

Description

Writes geometry / mesh information into a SELAFIN file.

Usage

```
write_geo(x)

## S3 method for class 't2d_geo'
write_geo(x)

## S3 method for class 't2d'
write_geo(x)
```

Arguments

x An object of class t2d_geo or t2d.

Value

Returns input x invisibly.

write_opt	<i>Write optional file(s)</i>
-----------	-------------------------------

Description

Writes optional TELEMAC-2D input file(s).

Usage

```
write_opt(x)

## S3 method for class 't2d_opt'
write_opt(x)

## S3 method for class 't2d'
write_opt(x)
```

Arguments

x An object of class t2d_opt or t2d.

Value

Returns input x invisibly.

write_results	<i>Write results to *.slf</i>
---------------	-------------------------------

Description

Writes data into a SELAFIN (*.slf) file.

Usage

```
write_results(x)

## S3 method for class 't2d_res'
write_results(x)

## S3 method for class 't2d'
write_results(x)
```

Arguments

x An object of class t2d_res or t2d.

Value

An updated version of input x.

Note

Writes only the available values (x\$values) into the file and adapts the header accordingly in case the header is referring to more variables that are not imported.

write_slf_header	<i>Write slf header</i>
------------------	-------------------------

Description

Writes the header of a SELAFIN file (*.slf).

Usage

```
write_slf_header(fname, header)
```

Arguments

fname	character, name of the SELAFIN file to be created.
header	A list with all header information (see output of read_slf_header).

Value

No return value, called for side effects.

Note

If a file fname already exists, it will be silently overwritten.

Make sure the datatypes in header (integer vs. double) match (compare with output of [read_slf_header](#)).

write_slf_variable	<i>Write slf variable</i>
--------------------	---------------------------

Description

Writes variable values into a SELAFIN file (*.slf).

Usage

```
write_slf_variable(fname, data)
```

Arguments

fname	character, name of the SELAFIN file (has to exist already and contain header information).
data	A list with variable values; see output of read_slf_variable for the required elements; in addition an element 'precision' (see read_slf_header) is needed.

Value

No return value, called for side effects.

`write_t2d`*Write TELEMAC-2D files*

Description

Writes all input files of a TELEMAC-2D project setup.

Usage

```
write_t2d(x)
```

Arguments

`x` An object of class `t2d`.

Value

Returns input `x` invisibly.

Note

This function is basically a wrapper around other `write_*` functions of the `telemac` package.

The associated file of `x$res` will replace entry `RESULTS FILE` in the steering parameters.

If `x$wdir` does not exist, it will be created.

If `x$wdir` is a relative path it will be considered relative to the current working directory.

All `t2d_*` objects will be written to the associated filenames into directory `wdir` (element of `x`). In case the associated filenames contain a relative path the files will be written relative to `wdir`.

Existing files will be silently overwritten!

Parameters `BOUNDARY CONDITIONS FILE`, `GEOMETRY FILE`, `RESULTS FILE`, and `TITLE` will be adapted to the current setup.

Index

[.t2d_cli (cli), 4

cas, 2, 15, 22
cli, 4, 22

geo, 4, 5, 22

idw, 8
interpol, 6, 7

line_spacing, 9
lines.t2d_tin, 9

optionals, 12

plot, 9, 13–15
plot.t2d_geo, 6, 13
plot.t2d_res, 14, 21
plot.t2d_tin, 14
POSIXct, 18
print.t2d (t2d), 22
print.t2d_cas (cas), 2
print.t2d_cli (cli), 4
print.t2d_geo (geo), 5
print.t2d_opt (optionals), 12
print.t2d_opt_LINES (optionals), 12
print.t2d_res (results), 20
print.t2d_tin (tin), 23

quasiquoteation, 10, 28

raster, 8, 13, 14, 27
read_cas, 15
read_cli, 15
read_geo, 6, 16
read_msh, 17
read_results, 17, 20
read_slf_header, 16, 18, 18, 19, 32
read_slf_variable, 19, 19, 32
results, 20, 22

simulate_t2d, 21, 22

SpatialLines, 10
SpatialPoints, 8
SpatialPointsDataFrame, 8

t2d, 22
tibble, 10
tin, 6, 23
tin2grid, 26
triangulate, 24

write_cas, 28
write_cli, 29
write_geo, 30
write_opt, 30
write_results, 31
write_slf_header, 32
write_slf_variable, 32
write_t2d, 22, 33