Package 'Rvcg'

October 31, 2022

Type Package

Title Manipulations of Triangular Meshes Based on the 'VCGLIB' API

Version 0.22

Date 2022-10-31

Description Operations on triangular meshes based on 'VCGLIB'. This package integrates nicely with the R-package 'rgl' to render the meshes processed by 'Rvcg'. The Visualization and Computer Graphics Library (VCG for short) is an open source portable C++ templated library for manipulation, processing and displaying with OpenGL of triangle and tetrahedral meshes. The library, composed by more than 100k lines of code, is released under the GPL license, and it is the base of most of the software tools of the Visual Computing Lab of the Italian National Research Council Institute ISTI <http://vcg.isti.cnr.it>, like 'metro' and 'MeshLab'. The 'VCGLIB' source is pulled from trunk <https://github.com/cnr-isti-vclab/vcglib> and patched to work with options determined by the configure script as well as to work with the header files included by 'RcppEigen'.

Depends R (>= 3.1.0)

Imports Rcpp, grDevices, stats, utils

Suggests Morpho, rgl

LinkingTo Rcpp, RcppEigen, RcppArmadillo

License GPL (>= 2) | file LICENSE

BugReports https://github.com/zarquon42b/Rvcg/issues

Copyright see files COPYRIGHTS for detailed information

LazyLoad yes

Biarch yes

URL https://github.com/zarquon42b/Rvcg,
https://github.com/cnr-isti-vclab/vcglib

Encoding UTF-8

RoxygenNote 7.2.1

NeedsCompilation yes

Author Stefan Schlager [aut, cre, cph],
Girinon Francois [ctb],
Tim Schaefer [ctb]

Maintainer Stefan Schlager <zarquon42@gmail.com>

Repository CRAN

Date/Publication 2022-10-31 13:02:47 UTC

R topics documented:

`Rvcg-package` *Interface between R and vcglib libraries for mesh operations*

Description

Provides meshing functionality from vcglib (meshlab) for R. E.g. mesh smoothing, mesh decimation, closest point search.

Details

Package:	Rvcg
Type:	Package
Version:	0.22
Date:	2022-10-31
License:	GPL
LazyLoad:	yes

Author(s)

Stefan Schlager

Maintainer: Stefan Schlager <zarquon42@gmail.com>

References

To be announced

checkFaceOrientation *check the orientation of a mesh*

Description

check the orientation of a mesh assuming that expansion along normals increases centroid size

Usage

```
checkFaceOrientation(x, offset = NULL)
```

Arguments

x	mesh of class mesh3d
offset	numeric: amount to offset the mesh along the vertex normals. If NULL a reasonable value will be estimated.

Details

assuming that a correctly (i.e outward) oriented mesh increases its centroid size when 'growing' outwards, this function tests whether this is the case.

Value

returns TRUE if mesh is oriented correctly and FALSE otherwise

Examples

```
data(dummyhead)
## now we invert faces inwards
checkFaceOrientation(dummyhead.mesh)

if (requireNamespace("Morpho", quietly = TRUE)) {
dummyinward <- Morpho::invertFaces(dummyhead.mesh)
checkFaceOrientation(dummyinward)
}
```

dummyhead *dummyhead - dummy head and landmarks*

Description

A triangular mesh representing a dummyhead - called by data(dummyhead)

Format

`dummyhead.mesh`: triangular mesh representing a dummyhead.

`dummyhead.lm`: landmarks on mesh 'dummyhead'

humface *Example mesh and landmarks*

Description

A triangular mesh representing a human face - called by data(humface)

Format

humface: triangular mesh representing a human face. humfaceClean: triangular mesh representing a human face but without errors or isolated pieces. humface.lm: landmarks on mesh 'humface'- called by data(humface)

meshInfo *print number of vertices and triangular faces of a mesh*

Description

print number of vertices and triangular faces of a mesh

Usage

```
meshInfo(x)
```

Arguments

x	triangular mesh

meshintegrity *check if an object of class mesh3d contains valid data*

Description

checks for existance and validity of vertices, faces and vertex normals of an object of class "mesh3d"

Usage

```
meshintegrity(mesh, facecheck = FALSE, normcheck = FALSE)
```

Arguments

mesh	object of class mesh3d
facecheck	logical: check the existence of valid triangular faces
normcheck	logical: check the existence of valid normals

Value

if mesh data are valid, the mesh is returned, otherwise it stops with an error message.

---

nfaces *get number of vertices from a mesh*

Description

get number of vertices from a mesh

Usage

```
nfaces(x)
```

Arguments

x	triangular mesh

Value

integer: number of triangular faces

---

nverts *get number of vertices from a mesh*

Description

get number of vertices from a mesh

Usage

```
nverts(x)
```

Arguments

x	triangular mesh

Value

integer: number of vertices

setRays *helper function to create an object to be processed by vcgRaySearch*

Description

create a search structure from a matrix of coordinates and one of directional vectors to be processed by vcgRaySearch

Usage

```
setRays(coords, dirs)
```

Arguments

coords	k x 3 matrix (or a vector of length 3) containing the starting points of the rays
dirs	k x 3 matrix (or a vector of length 3) containing the directons of the rays. The i-th row of dirs corresponds to the coordinate stored in the i-th row of coords

Value

an object of class "mesh3d" (without faces) and the vertices representing the starting points of the rays and the normals storing the directions.

vcgArea *compute surface area of a triangular mesh*

Description

compute surface area of a triangular mesh

Usage

```
vcgArea(mesh, perface = FALSE)
```

Arguments

mesh	triangular mesh of class mesh3d
perface	logical: if TRUE, a list containing the overall area, as well as the individual per-face area are reported.

Value

surface area of mesh

Examples

```
data(humface)
vcgArea(humface)
```

vcgBallPivoting *Ball pivoting surface reconstruction*

Description

Ball pivoting surface reconstruction

Usage

```
vcgBallPivoting(
  x,
  radius = 0,
  clustering = 0.2,
  angle = pi/2,
  deleteFaces = FALSE
)
```

Arguments

x	k x 3 matrix or object of class mesh3d
radius	The radius of the ball pivoting (rolling) over the set of points. Gaps that are larger than the ball radius will not be filled; similarly the small pits that are smaller than the ball radius will be filled. 0 = autoguess.
clustering	Clustering radius (fraction of ball radius). To avoid the creation of too small triangles, if a vertex is found too close to a previous one, it is clustered/merged with it.
angle	Angle threshold (radians). If we encounter a crease angle that is too large we should stop the ball rolling.
deleteFaces	in case x is a mesh and deleteFaces=TRUE, existing faces will be deleted beforehand.

Value

triangular face of class mesh3d

Examples

```
if (requireNamespace("Morpho", quietly = TRUE)) {
require(Morpho)
data(nose)
nosereko <- vcgBallPivoting(shortnose.lm)
}
```

vcgBary *get barycenters of all faces of a triangular mesh*

Description

get barycenters of all faces of a triangular mesh

Usage

```
vcgBary(mesh)
```

Arguments

mesh	triangular mesh of class "mesh3d"

Value

n x 3 matrix containing 3D-coordinates of the barycenters (where n is the number of faces in mesh.

Examples

```
data(humface)
bary <- vcgBary(humface)
## Not run:
require(rgl)
points3d(bary,col=2)
wire3d(humface)

## End(Not run)
```

vcgBorder *find all border vertices and faces of a triangular mesh*

Description

Detect faces and vertices at the borders of a mesh and mark them.

Usage

```
vcgBorder(mesh)
```

Arguments

mesh	triangular mesh of class "mesh3d"

Value

bordervb	logical: vector containing boolean value for each vertex, if it is a border vertex.
borderit	logical: vector containing boolean value for each face, if it is a border vertex.

Author(s)

Stefan Schlager

See Also

vcgPlyRead

Examples

```
data(humface)
borders <- vcgBorder(humface)
## view border vertices
## Not run:
require(rgl)
points3d(t(humface$vb[1:3,])[which(borders$bordervb == 1),],col=2)
wire3d(humface)
require(rgl)

## End(Not run)
```

---

vcgClean *Clean triangular surface meshes*

Description

Apply several cleaning algorithms to surface meshes

Usage

```
vcgClean(mesh, sel = 0, tol = 0, silent = FALSE, iterate = FALSE)
```

Arguments

mesh	triangular mesh of class 'mesh3d'
sel	integer vector selecting cleaning type (see "details"),
tol	numeric value determining Vertex Displacement Ratio used for splitting non-manifold vertices.
silent	logical, if TRUE no console output is issued.
iterate	logical: if TRUE, vcgClean is repeatedly run until nothing more is to be cleaned (see details).

Details

the vector sel determines which operations are performed in which order. E.g. removing degenerate faces may generate unreferenced vertices, thus the ordering of cleaning operations is important, multiple calls are possible (sel=c(1,3,1) will remove unreferenced vertices twice). available options are:

- 0 = only duplicated vertices and faces are removed
- 1 = remove unreferenced vertices
- 2 = Remove non-manifold Faces
- 3 = Remove degenerate faces
- 4 = Remove non-manifold vertices
- 5 = Split non-manifold vertices by threshold
- 6 = merge close vertices (radius=`tol`)
- 7 = coherently orient faces
  CAVEAT: sel=6 will not work keep vertex colors

Value

cleaned mesh with an additional entry

`remvert` vector of length = number of vertices before cleaning. Entries = 1 indicate that this vertex was removed; 0 otherwise.

Examples

```
data(humface)
cleanface <- humface
##add duplicated faces
cleanface$it <- cbind(cleanface$it, cleanface$it[,1:100])
## add duplicated vertices
cleanface$vb <- cbind(cleanface$vb,cleanface$vb[,1:100])
## ad unreferenced vertices
cleanface$vb <- cbind(cleanface$vb,rbind(matrix(rnorm(18),3,6),1))
cleanface <- vcgClean(cleanface, sel=1)
```

---

vcgClost *Project coordinates onto a target triangular surface mesh.*

Description

For a set of 3D-coordinates/triangular mesh, the closest matches on a target surface are determined and normals at as well as distances to that point are calculated.

Usage

```
vcgClost(
  x,
  mesh,
  sign = TRUE,
  barycentric = FALSE,
  smoothNormals = FALSE,
  borderchk = FALSE,
  tol = 0,
  facenormals = FALSE,
  ...
)
```

Arguments

x	k x 3 matrix containing 3D-coordinates or object of class "mesh3d".
mesh	triangular surface mesh stored as object of class "mesh3d".
sign	logical: if TRUE, signed distances are returned.
barycentric	logical: if TRUE, barycentric coordinates of the hit points are returned.
smoothNormals	logical: if TRUE, laplacian smoothed normals are used.
borderchk	logical: request checking if the hit face is at the border of the mesh.
tol	maximum distance to search. If distance is beyond that, the original point will be kept and the distance set to NaN. If tol = 0, tol is set to 2*diagonal of the bounding box of mesh.
facenormals	logical: if TRUE only the facenormal of the face the closest point has hit is returned, the weighted average of the surrounding vertex normals otherwise.
...	additional parameters, currently unused.

Value

returns an object of class "mesh3d" with:

vb	4 x n matrix containing n vertices as homolougous coordinates.
normals	4 x n matrix containing vertex normals.
quality	numeric vector containing distances to target.
it	3 x m integer matrix containing vertex indices forming triangular faces.Only available, when x is a mesh.
border	integer vector of length n: if borderchk = TRUE, for each clostest point the value will be 1 if the hit face is at the border of the target mesh and 0 otherwise.
barycoords	3 x m Matrix containing barycentric coordinates of closest points; only available if barycentric=TRUE.
faceptr	vector of face indeces on which the closest points are located

Note

If large part of the reference mesh are far away from the target surface, calculation can become very slow. In that case, the function `vcgClostKD` will be significantly faster.

Author(s)

Stefan Schlager

References

Baerentzen, Jakob Andreas. & Aanaes, H., 2002. Generating Signed Distance Fields From Triangle Meshes. Informatics and Mathematical Modelling.

See Also

`vcgPlyRead`

Examples

```
data(humface)
clost <- vcgClost(humface.lm, humface)
```

---

vcgClostKD *Project coordinates onto a target triangular surface mesh using KD-tree search*

Description

For a set of 3D-coordinates/triangular mesh, the closest matches on a target surface are determined (by using KD-tree search) and normals at as well as distances to that point are calculated.

Usage

```
vcgClostKD(
  x,
  mesh,
  sign = TRUE,
  barycentric = FALSE,
  smoothNormals = FALSE,
  borderchk = FALSE,
  k = 50,
  nofPoints = 16,
  maxDepth = 64,
  angdev = NULL,
```

```
  weightnorm = FALSE,
  facenormals = FALSE,
  threads = 1,
  ...
)
```

Arguments

x	k x 3 matrix containing 3D-coordinates or object of class "mesh3d".
mesh	triangular surface mesh stored as object of class "mesh3d".
sign	logical: if TRUE, signed distances are returned.
barycentric	logical: if TRUE, barycentric coordinates of the hit points are returned.
smoothNormals	logical: if TRUE, laplacian smoothed normals are used.
borderchk	logical: request checking if the hit face is at the border of the mesh.
k	integer: check the kdtree for thek closest faces (using faces' barycenters.
nofPoints	integer: number of points per cell in the kd-tree (don't change unless you know what you are doing!)
maxDepth	integer: depth of the kd-tree (don't change unless you know what you are doing!)
angdev	maximum deviation between reference and target normals. If the none of the k closest triangles match this criterion, the closest point on the closest triangle is returned but the corresponding distance in $quality is set to 1e5.
weightnorm	logical if angdev is set, this requests the normal of the closest points to be estimated by weighting the surrounding vertex normals. Otherwise, simply the hit face's normal is used (faster but slightly less accurate)
facenormals	logical: if TRUE only the facenormal of the face the closest point has hit is returned, the weighted average of the surrounding vertex normals otherwise.
threads	integer: threads to use in closest point search.
...	additional parameters, currently unused.

Value

returns an object of class "mesh3d" with:

vb	4 x n matrix containing n vertices as homolougous coordinates.
normals	4 x n matrix containing vertex normals.
quality	numeric vector containing distances to target.
it	3 x m integer matrix containing vertex indices forming triangular faces.Only available, when x is a mesh.
border	integer vector of length n: if borderchk = TRUE, for each clostest point the value will be 1 if the hit face is at the border of the target mesh and 0 otherwise.
barycoords	3 x m Matrix containing barycentric coordinates of closest points; only available if barycentric=TRUE.

Note

Other than `vcgClost` this does not search a grid, but first uses a KD-tree search to find the `k` closest barycenters for each point and then searches these faces for the closest match.

Author(s)

Stefan Schlager

References

Baerentzen, Jakob Andreas. & Aanaes, H., 2002. Generating Signed Distance Fields From Triangle Meshes. Informatics and Mathematical Modelling.

See Also

`vcgPlyRead`

---

vcgClostOnKDtreeFromBarycenters

search a KD-tree from Barycenters for multiple closest point searches on a mesh

---

Description

search a KD-tree from Barycenters for multiple closest point searches on a mesh

Usage

```
vcgClostOnKDtreeFromBarycenters(
  x,
  query,
  k = 50,
  sign = TRUE,
  barycentric = FALSE,
  borderchk = FALSE,
  angdev = NULL,
  weightnorm = FALSE,
  facenormals = FALSE,
  threads = 1
)
```

Arguments

`x`	object of class "vcgKDtreeWithBarycenters"
`query`	matrix or triangular mesh containing coordinates
`k`	integer: check the kdtree for thek closest faces (using faces' barycenters).

sign	logical: if TRUE, signed distances are returned.
barycentric	logical: if TRUE, barycentric coordinates of the hit points are returned.
borderchk	logical: request checking if the hit face is at the border of the mesh.
angdev	maximum deviation between reference and target normals. If the none of the k closest triangles match this criterion, the closest point on the closest triangle is returned but the corresponding distance in $quality is set to 1e5.
weightnorm	logical if angdev is set, this requests the normal of the closest points to be estimated by weighting the surrounding vertex normals. Otherwise, simply the hit face's normal is used (faster but slightly less accurate)
facenormals	logical: if TRUE only the facenormal of the face the closest point has hit is returned, the weighted average of the surrounding vertex normals otherwise.
threads	integer: threads to use in closest point search.

Value

returns an object of class "mesh3d" with:

vb	4 x n matrix containing n vertices as homolougous coordinates.
normals	4 x n matrix containing vertex normals.
quality	numeric vector containing distances to target.
it	3 x m integer matrix containing vertex indices forming triangular faces.Only available, when x is a mesh.
border	integer vector of length n: if borderchk = TRUE, for each clostest point the value will be 1 if the hit face is at the border of the target mesh and 0 otherwise.
barycoords	3 x m Matrix containing barycentric coordinates of closest points; only available if barycentric=TRUE.

Author(s)

Stefan Schlager

See Also

vcgCreateKDtreeFromBarycenters, vcgSearchKDtree, vcgCreateKDtree

---

vcgCreateKDtree *create a KD-tree*

Description

create a KD-tree

Usage

```
vcgCreateKDtree(mesh, nofPointsPerCell = 16, maxDepth = 64)
```

Arguments

mesh	matrix or triangular mesh containing coordinates
nofPointsPerCell	number of points per kd-cell
maxDepth	maximum tree depth

Value

returns an object of class vcgKDtree containing external pointers to the tree and the target points

See Also

vcgSearchKDtree

Examples

```
data(humface)
mytree <- vcgCreateKDtree(humface)
```

---

vcgCreateKDtreeFromBarycenters

create a KD-tree from Barycenters for multiple closest point searches on a mesh

---

Description

create a KD-tree from Barycenters for multiple closest point searches on a mesh

Usage

```
vcgCreateKDtreeFromBarycenters(mesh, nofPointsPerCell = 16, maxDepth = 64)
```

Arguments

mesh	matrix or triangular mesh containing coordinates
nofPointsPerCell	number of points per kd-cell
maxDepth	maximum tree depth

Value

returns an object of class vcgKDtreeWithBarycenters containing external pointers to the tree, the barycenters and the target mesh

See Also

vcgClostOnKDtreeFromBarycenters, vcgSearchKDtree, vcgCreateKDtree

Examples

```
## Not run:
data(humface);data(dummyhead)
barytree <- vcgCreateKDtreeFromBarycenters(humface)
closest <- vcgClostOnKDtreeFromBarycenters(barytree,dummyhead.mesh,k=50,threads=1)

## End(Not run)
```

---

vcgCurve *calculate curvature of a triangular mesh*

Description

calculate curvature of faces/vertices of a triangular mesh using various methods.

Usage

```
vcgCurve(mesh)
```

Arguments

mesh	triangular mesh (object of class 'mesh3d')

Value

gaussvb	per vertex gaussian curvature
meanvb	per vertex mean curvature
RMSvb	per vertex RMS curvature
gaussitmax	per face maximum gaussian curvature of adjacent vertices
borderit	per face information if it is on the mesh's border (0=FALSE, 1=TRUE)
bordervb	per vertex information if it is on the mesh's border (0=FALSE, 1=TRUE)
meanitmax	per face maximum mean curvature of adjacent vertices
K1	Principal Curvature 1
K2	Principal Curvature 2

Examples

```
data(humface)
curv <- vcgCurve(humface)
##visualise per vertex mean curvature
## Not run:
require(Morpho)
meshDist(humface,distvec=curv$meanvb,from=-0.2,to=0.2,tol=0.01)

## End(Not run)
```

vcgDijkstra *Compute pseudo-geodesic distances on a triangular mesh*

Description

Compute pseudo-geodesic distances on a triangular mesh

Usage

```
vcgDijkstra(x, vertpointer, maxdist = NULL)
```

Arguments

x	triangular mesh of class mesh3d
vertpointer	integer: references indices of vertices on the mesh, typically only a single query vertex.
maxdist	positive scalar double, the maximal distance to travel along the mesh when computing distances. Leave at NULL to traverse the full mesh. This can be used to speed up the computation if you are only interested in geodesic distances to neighbors within a limited distance around the query vertices.

Value

returns a vector of shortest distances for each of the vertices to one of the vertices referenced in vertpointer. If maxdist is in use (not NULL), the distance values for vertices outside the requested maxdist are not computed and appear as 0.

Note

Make sure to have a clean manifold mesh. Note that this computes the length of the pseudo-geodesic path (following the edges) between the two vertices.

Examples

```
## Compute geodesic distance between all mesh vertices and the first vertex of a mesh
data(humface)
geo <- vcgDijkstra(humface,1)
if (interactive()) {
require(Morpho);require(rgl)
meshDist(humface,distvec = geo)
spheres3d(vert2points(humface)[1,],col=2)
}
```

vcgFaceNormals *Compute normalized face normals for a mesh.*

Description

Compute normalized face normals for a mesh.

Usage

```
vcgFaceNormals(mesh)
```

Arguments

mesh	triangular mesh of class 'mesh3d', from rgl

Value

3xn numeric matrix of face normals for the mesh, where n is the number of faces.

Examples

```
data(humface);
hf_facenormals <- vcgFaceNormals(humface);
```

vcgGeodesicPath *Compute geodesic path and path length between vertices on a mesh*

Description

Compute geodesic path and path length between vertices on a mesh

Usage

```
vcgGeodesicPath(x, source, targets, maxdist = 1e+06)
```

Arguments

x	triangular mesh of class mesh3d from the rgl package.
source	scalar positive integer, the source vertex index.
targets	positive integer vector, the target vertex indices.
maxdist	numeric, the maximal distance to travel along the mesh edges during geodesic distance computation.

Value

named list with two entries as follows. 'paths': list of integer vectors, representing the paths. 'geodist': double vector, the geodesic distances from the source vertex to all vertices in the graph.

Note

Currently no reachability checks are performed, so you have to be sure that the mesh is connected, or at least that the source and target vertices are reachable from one another.

Examples

```
data(humface)
p = vcgGeodesicPath(humface,50,c(500,5000))
p$paths[[1]];   # The path 50..500
p$geodist[500]; # Its path length.
```

---

vcgGeodist *Compute pseudo-geodesic distance between two points on a mesh*

Description

Compute pseudo-geodesic distance between two points on a mesh

Usage

```
vcgGeodist(x, pt1, pt2)
```

Arguments

x	triangular mesh of class mesh3d
pt1	3D coordinate on mesh or index of vertex
pt2	3D coordinate on mesh or index of vertex

Value

returns the geodesic distance between pt1 and pt2.

Note

Make sure to have a clean manifold mesh. Note that this computes the length of the pseudo-geodesic path (following the edges) between the two vertices closest to these points.

Examples

```
data(humface)
pt1 <- humface.lm[1,]
pt2 <- humface.lm[5,]
vcgGeodist(humface,pt1,pt2)
```

vcgGetEdge *Get all edges of a triangular mesh*

Description

Extract all edges from a mesh and retrieve adjacent faces and vertices

Usage

```
vcgGetEdge(mesh, unique = TRUE)
```

Arguments

mesh	triangular mesh of class 'mesh3d'
unique	logical: if TRUE each edge is only reported once, if FALSE, all occurences are reported.

Value

returns a dataframe containing:

vert1	integer indicating the position of the first vertex belonging to this edge
vert2	integer indicating the position of the second vertex belonging to this edge
facept	integer pointing to the (or a, if unique = TRUE) face adjacent to the edge
border	integer indicating if the edge is at the border of the mesh. 0 = no border, 1 = border

Examples

```
require(rgl)
data(humface)
edges <-vcgGetEdge(humface)
## Not run:
## show first edge
lines3d(t(humface$vb[1:3,])[c(edges$vert1[1],edges$vert2[2]),],col=2,lwd=3)
shade3d(humface, col=3)
## now find the edge - hint: it is at the neck.

## End(Not run)
```

vcgImport *Import common mesh file formats.*

Description

Import common mesh file formats and store the results in an object of class "mesh3d" - momentarily only triangular meshes are supported.

Usage

```
vcgImport(
  file,
  updateNormals = TRUE,
  readcolor = FALSE,
  clean = TRUE,
  silent = FALSE
)
```

Arguments

file	character: file to be read.
updateNormals	logical: if TRUE and the imported file contais faces, vertex normals will be (re)calculated. Otherwise, normals will be a matrix containing zeros.
readcolor	if TRUE, vertex colors and texture (face and vertex) coordinates will be processed - if available, otherwise all vertices will be colored white.
clean	if TRUE, duplicated and unreferenced vertices as well as duplicate faces are removed (be careful when importing point clouds).
silent	logical, if TRUE no console output is issued.

Value

Object of class "mesh3d"

with:

vb	4 x n matrix containing n vertices as homolougous coordinates
it	3 x m matrix containing vertex indices forming triangular faces
normals	4 x n matrix containing vertex normals (homologous coordinates)

in case the imported files contains face or vertex quality, these will be stored as vectors named $quality (for vertex quality) and $facequality

if the imported file contains vertex colors and readcolor = TRUE, these will be saved in $material$color according to "mesh3d" specifications.

Note

currently only meshes with either color or texture can be processed. If both are present, the function will mark the mesh as non-readable.

Author(s)

Stefan Schlager

See Also

vcgSmooth

Examples

```
data(humface)
vcgPlyWrite(humface)
readit <- vcgImport("humface.ply")
```

---

vcgIsolated *Remove isolated pieces from a surface mesh or split into connected components*

Description

Remove isolated pieces from a surface mesh, selected by a minimum amount of faces or of a diameter below a given threshold. Also the option only to keep the largest piece can be selected or to split a mesh into connected components.

Usage

```
vcgIsolated(
  mesh,
  facenum = NULL,
  diameter = NULL,
  split = FALSE,
  keep = 0,
  silent = FALSE
)
```

Arguments

mesh	triangular mesh of class "mesh3d".
facenum	integer: all connected pieces with less components are removed. If not specified or 0 and diameter is NULL, then only the component with the most faces is kept.
diameter	numeric: all connected pieces smaller diameter are removed removed. diameter = 0 removes all component but the largest ones. This option overrides the option facenum.
split	logical: if TRUE, a list with all connected components (optionally matching requirements facenum/diameter) of the mesh will be returned.
keep	integer: if split=T, keep specifies the number of largest chunks (number of faces) to keep.
silent	logical, if TRUE no console output is issued.

Value

returns the reduced mesh.

Author(s)

Stefan Schlager

See Also

vcgPlyRead

Examples

```
## Not run:
data(humface)
cleanface <- vcgIsolated(humface)

## End(Not run)
```

---

vcgIsosurface *Create Isosurface from 3D-array*

Description

Create Isosurface from 3D-array using Marching Cubes algorithm

Usage

```
vcgIsosurface(
  vol,
  threshold,
  from = NULL,
  to = NULL,
  spacing = NULL,
  origin = NULL,
  direction = NULL,
  IJK2RAS = diag(c(-1, -1, 1, 1)),
  as.int = FALSE
)
```

Arguments

vol	an integer valued 3D-array
threshold	threshold for creating the surface
from	numeric: the lower threshold of a range (overrides threshold)
to	numeric: the upper threshold of a range (overrides threshold)

spacing	numeric 3D-vector: specifies the voxel dimensons in x,y,z direction.
origin	numeric 3D-vector: origin of the original data set, will transpose the mesh onto that origin.
direction	a 3x3 direction matrix
IJK2RAS	4x4 IJK2RAS transformation matrix
as.int	logical: if TRUE, the array will be stored as integer (might decrease RAM usage)

Value

returns a triangular mesh of class "mesh3d"

Examples

```
#this is the example from the package "misc3d"
x <- seq(-2,2,len=50)
g <- expand.grid(x = x, y = x, z = x)
v <- array(g$x^4 + g$y^4 + g$z^4, rep(length(x),3))
storage.mode(v) <- "integer"
## Not run:
mesh <- vcgIsosurface(v,threshold=10)
require(rgl)
wire3d(mesh)
##now smooth it a little bit
wire3d(vcgSmooth(mesh,"HC",iteration=3),col=3)

## End(Not run)
```

---

vcgIsotropicRemeshing *Isotropically remesh a triangular surface mesh*

Description

Isotropically remesh a triangular surface mesh

Usage

```
vcgIsotropicRemeshing(
  x,
  TargetLen = 1,
  FeatureAngleDeg = 10,
  MaxSurfDist = 1,
  iterations = 3,
  Adaptive = FALSE,
  split = TRUE,
  collapse = TRUE,
  swap = TRUE,
```

```
  smooth = TRUE,
  project = TRUE,
  surfDistCheck = TRUE
)
```

Arguments

x	mesh of class mesh3d
TargetLen	numeric: edge length of the target surface
FeatureAngleDeg	define Crease angle (in degree).
MaxSurfDist	Max. surface distance
iterations	ToDo
Adaptive	enable adaptive remeshing
split	enable refine step
collapse	enable collapse step
swap	enable dge swap
smooth	enable smoothing
project	enable reprojection step
surfDistCheck	check distance to surface

Value

returns the remeshed surface mesh

Examples

```
## Not run:
data(humface)
resampledMesh <- vcgIsotropicRemeshing(humface,TargetLen=2.5)

## End(Not run)
```

---

vcgKDtree *perform kdtree search for 3D-coordinates.*

Description

perform kdtree search for 3D-coordinates.

Usage

```
vcgKDtree(target, query, k, nofPoints = 16, maxDepth = 64, threads = 1)
```

Arguments

target	n x 3 matrix with 3D coordinates or mesh of class "mesh3d". These coordinates are to be searched.
query	m x 3 matrix with 3D coordinates or mesh of class "mesh3d". We seach the closest coordinates in target for each of these.
k	number of neighbours to find
nofPoints	integer: number of points per cell in the kd-tree (don't change unless you know what you are doing!)
maxDepth	integer: depth of the kd-tree (don't change unless you know what you are doing!)
threads	integer: threads to use in closest point search.

Value

a list with

index	integer matrices with indeces of closest points
distances	corresponding distances

---

vcgKmeans *fast Kmean clustering for 1D, 2D and 3D data*

Description

fast Kmean clustering for 1D, 2D and 3D data

Usage

```
vcgKmeans(x, k = 10, iter.max = 10, getClosest = FALSE, threads = 0)
```

Arguments

x	matrix containing coordinates or mesh3d
k	number of clusters
iter.max	maximum number of iterations
getClosest	logical: if TRUE the indices of the points closest to the k-centers are sought.
threads	integer: number of threads to use

Value

returns a list containing

centers	cluster center
class	vector with cluster association for each coordinate

If getClosest=TRUE

selected	vector with indices of points closest to the centers

See Also

vcgSample

Examples

```
require(Rvcg);require(rgl)
data(humface)
set.seed(42)
clust <- vcgKmeans(humface,k=1000,threads=1)
```

---

vcgMeshres *calculates the average edge length of a triangular mesh*

Description

calculates the average edge length of a triangular mesh, iterating over all faces.

Usage

```
vcgMeshres(mesh)
```

Arguments

mesh	triangular mesh stored as object of class "mesh3d"

Value

res	average edge length (a.k.a. mesh resolution)
edgelength	vector containing lengths for each edge

Author(s)

Stefan Schlager

Examples

```
data(humface)
mres <- vcgMeshres(humface)
#histogram of edgelength distribution
hist(mres$edgelength)
#visualise average edgelength
points( mres$res, 1000, pch=20, col=2, cex=2)
```

vcgMetro *evaluate the difference between two triangular meshes.*

Description

Implementation of the command line tool "metro" to evaluate the difference between two triangular meshes.

Usage

```
vcgMetro(
  mesh1,
  mesh2,
  nSamples = 0,
  nSamplesArea = 0,
  vertSamp = TRUE,
  edgeSamp = TRUE,
  faceSamp = TRUE,
  unrefVert = FALSE,
  samplingType = c("SS", "MC", "SD"),
  searchStruct = c("SGRID", "AABB", "OCTREE", "HGRID"),
  from = 0,
  to = 0,
  colormeshes = FALSE,
  silent = FALSE
)
```

Arguments

mesh1	triangular mesh (object of class 'mesh3d').
mesh2	triangular mesh (object of class 'mesh3d').
nSamples	set the required number of samples if 0, this will be set to approx. 10x the face number.
nSamplesArea	set the required number of samples per area unit, override nSamples.
vertSamp	logical: if FALSE, disable vertex sampling.
edgeSamp	logical: if FALSE, disable edge sampling.
faceSamp	logical: if FALSE, disable face sampling.
unrefVert	logical: if FALSE, ignore unreferred vertices.
samplingType	set the face sampling mode. options are: SS (similar triangles sampling), SD (subdivision sampling), MC (montecarlo sampling).
searchStruct	set search structures to use. options are: SGIRD (static Uniform Grid), OCTREE, AABB (AxisAligned Bounding Box Tree), HGRID (Hashed Uniform Grid).
from	numeric: minimum value for color mapping.

to	numeric: maximum value for color mapping.
colormeshes	if TRUE, meshes with vertices colored according to distance are returned
silent	logical: if TRUE, output to console is suppressed.

Value

ForwardSampling, BackwardSampling
: lists containing information about forward (mesh1 to mesh2) and backward (mesh2 to mesh1) sampling with the following entries

- maxdist maximal Hausdorff distance
- meandist mean Hausdorff distance
- RMSdist RMS of the Hausdorff distances
- area mesh area (of mesh1 in ForwardSampling and mesh2 in BackwardSampling)
- RMSdist RMS of the Hausdorff distances
- nvbsamples number of vertices sampled
- nsamples number of samples

distances1, distances2
: vectors containing vertex distances from mesh1 to mesh2 and mesh2 to mesh1.

forward_hist, backward_hist
: Matrices tracking the sampling results

if colormeshes == TRUE

mesh1, mesh2
: meshes with color coded distances and an additional entry called quality containing the sampled per-vertex distances

Note

this is a straightforward implementation of the command line tool metro http://vcg.isti.cnr.it/vcglib/metro.html

References

P. Cignoni, C. Rocchini and R. Scopigno. Metro: measuring error on simplified surfaces. Computer Graphics Forum, Blackwell Publishers, vol. 17(2), June 1998, pp 167-174

Examples

```
if (requireNamespace("Morpho", quietly = TRUE)) {
require(Morpho)
data(humface)
data(dummyhead)
## align humface to dummyhead.mesh
humfalign <- rotmesh.onto(humface,humface.lm,dummyhead.lm)
samp <- vcgMetro(humfalign$mesh,dummyhead.mesh,faceSamp=FALSE,edgeSamp=FALSE)
## create heatmap using Morpho's meshDist function
```

```
}

## Not run:
## create custom heatmaps based on distances
mD <- meshDist(humfalign$mesh,distvec=samp$distances1)

## End(Not run)
```

---

vcgNonBorderEdge *Get all non-border edges*

Description

Get all non-border edges and both faces adjacent to them.

Usage

```
vcgNonBorderEdge(mesh, silent = FALSE)
```

Arguments

mesh	triangular mesh of class 'mesh3d
silent	logical: suppress output of information about number of border edges

Value

returns a dataframe containing:

vert1	integer indicating the position of the first vertex belonging to this edge
vert2	integer indicating the position of the second vertex belonging to this edge
border	integer indicating if the edge is at the border of the mesh. 0 = no border, 1 = border
face1	integer pointing to the first face adjacent to the edge
face2	integer pointing to the first face adjacent to the edge

See Also

vcgGetEdge

Examples

```
data(humface)
edges <-vcgNonBorderEdge(humface)
## show first edge (not at the border)
## Not run:
require(Morpho)
require(rgl)
```

```
lines3d(t(humface$vb[1:3,])[c(edges$vert1[1],edges$vert2[2]),],col=2,lwd=3)

## plot barycenters of adjacent faces
bary <- barycenter(humface)
points3d(bary[c(edges$face1[1],edges$face2[1]),])
shade3d(humface, col=3)
## now find the edge - hint: it is at the neck.

## End(Not run)
```

---

vcgObjWrite *Export meshes to OBJ-files*

Description

Export meshes to OBJ-files

Usage

```
vcgObjWrite(mesh, filename = dataname, writeNormals = TRUE)
```

Arguments

mesh	triangular mesh of class 'mesh3d' or a numeric matrix with 3-columns
filename	character: filename (file extension '.obj' will be added automatically.
writeNormals	write existing normals to file

Examples

```
data(humface)
vcgObjWrite(humface,filename = "humface")
unlink("humface.obj")
```

---

vcgOffWrite *Export meshes to OFF-files*

Description

Export meshes to OFF-files

Usage

```
vcgOffWrite(mesh, filename = dataname)
```

Arguments

mesh	triangular mesh of class 'mesh3d' or a numeric matrix with 3-columns
filename	character: filename (file extension '.off' will be added automatically.

Examples

```
data(humface)
vcgOffWrite(humface,filename = "humface")
unlink("humface.off")
```

---

vcgPlyRead *Import ascii or binary PLY files.*

Description

Reads Polygon File Format (PLY) files and stores the results in an object of class "mesh3d" - momentarily only triangular meshes are supported.

Usage

```
vcgPlyRead(file, updateNormals = TRUE, clean = TRUE)
```

Arguments

file	character: file to be read.
updateNormals	logical: if TRUE and the imported file contais faces, vertex normals will be (re)calculated.
clean	logical: if TRUE, duplicated and unreference vertices will be removed.

Value

Object of class "mesh3d"

with:

vb	3 x n matrix containing n vertices as homolougous coordinates
normals	3 x n matrix containing vertex normals
it	3 x m integer matrix containing vertex indices forming triangular faces
material$color	Per vertex colors if specified in the imported file

Note

from version 0.8 on this is only a wrapper for vcgImport (to avoid API breaking).

Author(s)

Stefan Schlager

See Also

vcgSmooth,

---

vcgPlyWrite *Export meshes to PLY-files*

Description

Export meshes to PLY-files (binary or ascii)

Usage

```
vcgPlyWrite(mesh, filename, binary = TRUE, ...)

## S3 method for class 'mesh3d'
vcgPlyWrite(
  mesh,
  filename = dataname,
  binary = TRUE,
  addNormals = FALSE,
  writeCol = TRUE,
  writeNormals = TRUE,
  ...
)

## S3 method for class 'matrix'
vcgPlyWrite(mesh, filename = dataname, binary = TRUE, addNormals = FALSE, ...)
```

Arguments

mesh	triangular mesh of class 'mesh3d' or a numeric matrix with 3-columns
filename	character: filename (file extension '.ply' will be added automatically, if missing.
binary	logical: write binary file
...	additional arguments, currently not used.
addNormals	logical: compute per-vertex normals and add to file
writeCol	logical: export existing per-vertex color stored in mesh$material$color
writeNormals	write existing normals to file

Examples

```
data(humface)
vcgPlyWrite(humface,filename = "humface")
## remove it
unlink("humface.ply")
```

vcgQEdecim *Performs Quadric Edge Decimation on triangular meshes.*

Description

Decimates a mesh by adapting the faces of a mesh either to a target face number, a percentage or an approximate mesh resolution (a.k.a. mean edge length

Usage

```
vcgQEdecim(
  mesh,
  tarface = NULL,
  percent = NULL,
  edgeLength = NULL,
  topo = FALSE,
  quality = TRUE,
  bound = FALSE,
  optiplace = FALSE,
  scaleindi = TRUE,
  normcheck = FALSE,
  qweightFactor = 100,
  qthresh = 0.3,
  boundweight = 1,
  normalthr = pi/2,
  silent = FALSE
)
```

Arguments

mesh	Triangular mesh of class "mesh3d"
tarface	Integer: set number of target faces.
percent	Numeric: between 0 and 1. Set amount of reduction relative to existing face number. Overrides tarface argument.
edgeLength	Numeric: tries to decimate according to a target mean edge length. Under the assumption of regular triangles, the edges are half as long by dividing the triangle into 4 regular smaller triangles.
topo	logical: if TRUE, mesh topology is preserved.
quality	logical: if TRUE, vertex quality is considered.
bound	logical: if TRUE, mesh boundary is preserved.
optiplace	logical: if TRUE, mesh boundary is preserved (may lead to unwanted distortions in some cases).
scaleindi	logical: if TRUE, decimatiion is scale independent.
normcheck	logical: if TRUE, normal directions are considered.

qweightFactor	numeric: >= 1. Quality range is mapped into a squared 01 and than into the 1 - QualityWeightFactor range.
qthresh	numeric: Quality threshold for decimation process.
boundweight	numeric: Weight assigned to mesh boundaries.
normalthr	numeric: threshold for normal check in radians.
silent	logical, if TRUE no console output is issued.

Details

This is basically an adaption of the cli tridecimator from vcglib

Value

Returns a reduced mesh of class mesh3d.

Author(s)

Stefan Schlager

See Also

vcgSmooth

Examples

```
data(humface)
##reduce faces to 50%
decimface <- vcgQEdecim(humface, percent=0.5)
## view
## Not run:
require(rgl)
shade3d(decimface, col=3)

## some light smoothing
decimface <- vcgSmooth(decimface,iteration = 1)

## End(Not run)
```

---

vcgRaySearch *check if a mesh is intersected by a set of rays*

Description

check if a mesh is intersected by a set of rays (stored as normals)

Usage

```
vcgRaySearch(x, mesh, mintol = 0, maxtol = 1e+15, mindist = FALSE, threads = 1)
```

Arguments

x	a triangular mesh of class 'mesh3d' or a list containing vertices and vertex normals (fitting the naming conventions of 'mesh3d'). In the second case x must contain x$vb = 3 x n matrix containing 3D-coordinates and x$normals = 3 x n matrix containing normals associated with x$vb.
mesh	triangular mesh to be intersected.
mintol	minimum distance to target mesh
maxtol	maximum distance to search along ray
mindist	search both ways (ray and -ray) and select closest point.
threads	number of threads used during search.

Details

vcgRaySearch projects a mesh (or set of 3D-coordinates) along a set of given rays (stored as normals) onto a target and return the hit points as well as information if the target mesh was hit at all. If nothing is hit along the ray(within the given thresholds), the ordinary closest point's value will be returned and the corresponding entry in quality will be zero.

Value

list with following items:

vb	4 x n matrix containing intersection points
normals	4 x n matrix containing homogenous coordinates of normals at intersection points
quality	integer vector containing a value for each vertex of x: 1 indicates that a ray has intersected 'mesh' , while 0 means not
distance	numeric vector: distances to intersection

Examples

```
data(humface)
#get normals of landmarks
lms <- vcgClost(humface.lm, humface)
# offset landmarks along their normals for a negative amount of -5mm
lms$vb[1:3,] <- lms$vb[1:3,]+lms$normals[1:3,]*-5
intersect <- vcgRaySearch(lms, humface)
## Not run:
require(Morpho)
require(rgl)
spheres3d(vert2points(lms),radius=0.5,col=3)
plotNormals(lms,long=5)
spheres3d(vert2points(intersect),col=2) #plot intersections
wire3d(humface,col="white")#'
```

```
## End(Not run)
```

---

vcgSample *Subsamples points on a mesh surface*

Description

Subsamples surface of a triangular mesh and returns a set of points located on that mesh

Usage

```
vcgSample(
  mesh,
  SampleNum = 100,
  type = c("km", "pd", "mc"),
  MCsamp = 20,
  geodes = TRUE,
  strict = FALSE,
  iter.max = 100,
  threads = 0
)
```

Arguments

mesh	triangular mesh of class 'mesh3d'
SampleNum	integer: number of sampled points (see details below)
type	character: seclect sampling type ("mc"=MonteCarlo Sampling, "pd"=PoissonDisk Sampling,"km"=kmean clustering)
MCsamp	integer: MonteCarlo sample iterations used in PoissonDisk sampling.
geodes	logical: maximise geodesic distance between sample points (only for Poisson Disk sampling)
strict	logical: if type="pd" and the amount of coordinates exceeds SampleNum, the resulting coordinates will be subsampled again by kmean clustering to reach the requested number.
iter.max	integer: maximum iterations to use in k-means clustering.
threads	integer number of threads to use for k-means clustering

Details

Poisson disk subsampling will not generate the exact amount of coordinates specified in SampleNum, depending on MCsamp the result will contain more or less coordinates.

Value

sampled points

Examples

```
data(humface)
ss <- vcgSample(humface,SampleNum = 500, type="km",threads=1)
## Not run:
require(rgl)
points3d(ss)

## End(Not run)
```

---

vcgSearchKDtree *search an existing KD-tree*

Description

search an existing KD-tree

Usage

```
vcgSearchKDtree(kdtree, query, k, threads = 0)
```

Arguments

kdtree	object of class vcgKDtree
query	atrix or triangular mesh containing coordinates
k	number of k-closest neighbours to query
threads	integer: number of threads to use

Value

a list with

index	integer matrices with indeces of closest points
distances	corresponding distances

See Also

vcgCreateKDtree

Examples

```
## Not run:
data(humface);data(dummyhead)
mytree <- vcgCreateKDtree(humface)
## get indices and distances for 10 closest points.
closest <- vcgSearchKDtree(mytree,dummyhead.mesh,k=10,threads=1)

## End(Not run)
```

---

vcgSmooth *Smoothes a triangular mesh*

Description

Applies different smoothing algorithms on a triangular mesh.

Usage

```
vcgSmooth(
  mesh,
  type = c("taubin", "laplace", "HClaplace", "fujiLaplace", "angWeight",
    "surfPreserveLaplace"),
  iteration = 10,
  lambda = 0.5,
  mu = -0.53,
  delta = 0.1
)
```

Arguments

mesh	triangular mesh stored as object of class "mesh3d".
type	character: select smoothing algorithm. Available are "taubin", "laplace", "HClaplace", "fujiLaplace", "angWeight" (and any sensible abbreviations).
iteration	integer: number of iterations to run.
lambda	numeric: parameter for Taubin smooth (see reference below).
mu	numeric:parameter for Taubin smooth (see reference below).
delta	numeric: parameter for Scale dependent laplacian smoothing (see reference below).and maximum allowed angle (in radians) for deviation between normals Laplacian (surface preserving).

Details

The algorithms available are Taubin smoothing, Laplacian smoothing and an improved version of Laplacian smoothing ("HClaplace"). Also available are Scale dependent laplacian smoothing ("fujiLaplace") and Laplacian angle weighted smoothing ("angWeight")

Value

returns an object of class "mesh3d" with:

vb	4xn matrix containing n vertices as homolougous coordinates.
normals	4xn matrix containing vertex normals.
quality	vector: containing distances to target.
it	4xm matrix containing vertex indices forming triangular faces.

Note

The additional parameters for taubin smooth are hardcoded to the default values of meshlab, as they appear to be the least distorting

Author(s)

Stefan Schlager

References

Taubin G. 1995. Curve and surface smoothing without shrinkage. In Computer Vision, 1995. Proceedings., Fifth International Conference on, pages 852 - 857.

Vollmer J., Mencl R. and Mueller H. 1999. Improved Laplacian Smoothing of Noisy Surface Meshes. Computer Graphics Forum, 18(3):131 - 138.

Schroeder, P. and Barr, A. H. (1999). Implicit fairing of irregular meshes using diffusion and curvature flow: 317-324.

See Also

vcgPlyRead, vcgClean

Examples

```
data(humface)
smoothface <- vcgSmooth(humface)
## view
## Not run:
require(rgl)
shade3d(smoothface, col=3)

## End(Not run)
```

vcgSphere *create platonic objects as triangular meshes*

Description

create platonic objects as triangular meshes

Usage

```
vcgSphere(subdivision = 3, normals = TRUE)

vcgSphericalCap(angleRad = pi/2, subdivision = 3, normals = TRUE)

vcgTetrahedron(normals = TRUE)

vcgDodecahedron(normals = TRUE)

vcgOctahedron(normals = TRUE)

vcgIcosahedron(normals = TRUE)

vcgHexahedron(normals = TRUE)

vcgSquare(normals = TRUE)

vcgBox(mesh = vcgSphere(), normals = TRUE)

vcgCone(r1, r2, h, normals = TRUE)
```

Arguments

subdivision	subdivision level for sphere (the larger the denser the mesh will be)
normals	if TRUE vertex normals are calculated
angleRad	angle of the spherical cap
mesh	mesh to take the bounding box from
r1	radius1 of the cone
r2	radius2 of the cone
h	height of the cone

vcgStlWrite *Export meshes to STL-files*

Description

Export meshes to STL-files (binary or ascii)

Usage

```
vcgStlWrite(mesh, filename = dataname, binary = FALSE)
```

Arguments

mesh	triangular mesh of class 'mesh3d' or a numeric matrix with 3-columns
filename	character: filename (file extension '.stl' will be added automatically.
binary	logical: write binary file

Examples

```
data(humface)
vcgStlWrite(humface,filename = "humface")
unlink("humface.stl")
```

vcgSubdivide *subdivide the triangles of a mesh*

Description

subdivide the triangles of a mesh

Usage

```
vcgSubdivide(
  x,
  threshold = NULL,
  type = c("Butterfly", "Loop"),
  looptype = c("loop", "regularity", "continuity"),
  iterations = 3,
  silent = FALSE
)
```

Arguments

x	triangular mesh of class "mesh3d"
threshold	minimum edge length to subdivide
type	character: algorithm used. Options are Butterfly and Loop (see notes)
looptype	character: method for type = loop options are "loop","regularity","continuity" (see notes)
iterations	integer: number of iterations
silent	logical: suppress output.

Value

returns subdivided mesh

Note

The different algorithms are (from meshlab description):

- **Butterfly Subdivision:** Apply Butterfly Subdivision Surface algorithm. It is an interpolated method, defined on arbitrary triangular meshes. The scheme is known to be C1 but not C2 on regular meshes
- **Loop Subdivision:** Apply Loop's Subdivision Surface algorithm. It is an approximant subdivision method and it works for every triangle and has rules for extraordinary vertices. Options are "loop" a simple subdivision, "regularity" to enhance the meshe's regularity and "continuity" to enhance the mesh's continuity.

Examples

```
data(humface)
subdivide <- vcgSubdivide(humface,type="Loop",looptype="regularity")
```

---

vcgUniformRemesh *Resample a mesh uniformly*

Description

Resample a mesh uniformly

Usage

```
vcgUniformRemesh(
  x,
  voxelSize = NULL,
  offset = 0,
  discretize = FALSE,
  multiSample = FALSE,
```

```
  absDist = FALSE,
  mergeClost = FALSE,
  silent = FALSE
)
```

Arguments

x	triangular mesh
voxelSize	voxel size for space discretization
offset	Offset of the created surface (i.e. distance of the created surface from the original one).
discretize	If TRUE, the position of the intersected edge of the marching cube grid is not computed by linear interpolation, but it is placed in fixed middle position. As a consequence the resampled object will look severely aliased by a stairstep appearance.
multiSample	If TRUE, the distance field is more accurately compute by multisampling the volume (7 sample for each voxel). Much slower but less artifacts.
absDist	If TRUE, an unsigned distance field is computed. In this case you have to choose a not zero Offset and a double surface is built around the original surface, inside and outside.
mergeClost	logical: merge close vertices
silent	logical: suppress messages

Value

resampled mesh

Examples

```
## Not run:
data(humface)
humresample <- vcgUniformRemesh(humface,voxelSize=1,multiSample = TRUE)
require(rgl)
shade3d(humresample,col=3)

## End(Not run)
```

---

vcgUpdateNormals *updates vertex normals of a triangular meshes or point clouds*

Description

update vertex normals of a triangular meshes or point clouds

Usage

```
vcgUpdateNormals(mesh, type = 0, pointcloud = c(10, 0), silent = FALSE)
```

Arguments

mesh	triangular mesh of class 'mesh3d' or a n x 3 matrix containing 3D-coordinates.
type	select the method to compute per-vertex normals: 0=area weighted average of surrounding face normals; 1 = angle weighted vertex normals.
pointcloud	integer vector of length 2: containing optional parameters for normal calculation of point clouds. The first enty specifies the number of neighbouring points to consider. The second entry specifies the amount of smoothing iterations to be performed.
silent	logical, if TRUE no console output is issued.

Value

mesh with updated/created normals, or in case mesh is a matrix, a list of class "mesh3d" with

vb	4 x n matrix containing coordinates (as homologous coordinates
normals	4 x n matrix containing normals (as homologous coordinates

Examples

```
data(humface)
humface$normals <- NULL # remove normals
humface <- vcgUpdateNormals(humface)
## Not run:
pointcloud <- t(humface$vb[1:3,]) #get vertex coordinates
pointcloud <- vcgUpdateNormals(pointcloud)

require(Morpho)
plotNormals(pointcloud)#plot normals

## End(Not run)
```

---

vcgVertexNeighbors *Compute mesh adjacency list representation or the vertex neighborhoods of specific mesh vertices.*

Description

Compute the k-ring vertex neighborhood for all query vertex indices vi. If only a mesh is passed (parameter x) and the other parameters are left at their default values, this compute the adjacency list representation of the mesh.

Usage

```
vcgVertexNeighbors(x, vi = NULL, numstep = 1L, include_self = FALSE)
```

Arguments

x	tmesh3d instance from the rgl package
vi	optional, vector of positive vertex indices for which to compute the neighborhoods. All vertices are used if left at the default value NULL.
numstep	positive integer, the number of times to extend the neighborhood from the source vertices (the k for computing the k-ring neighborhood). Setting this to high values significantly increases the computational cost.
include_self	logical, whether the returned neighborhood for a vertex i should include i itself.

Value

list of positive integer vectors, the neighborhoods.

Examples

```
data(humface)
adjacency_list <- vcgVertexNeighbors(humface)
v500_5ring = vcgVertexNeighbors(humface, vi=c(500), numstep = 5)
```

---

vcgVFadj *find all faces belonging to each vertex in a mesh*

Description

find all faces belonging to each vertex in a mesh and report their indices

Usage

```
vcgVFadj(mesh)
```

Arguments

mesh	triangular mesh of class "mesh3d"

Value

list containing one vector per vertex containgin the indices of the adjacent faces

vcgVolume *Compute volume for manifold meshes*

Description

Compute volume for manifold meshes

Usage

```
vcgVolume(x)
```

Arguments

x	triangular mesh of class mesh3d

Value

returns volume

Note

Please note, that this function only works reliably on watertight, coherently oriented meshes that constitute a manifold. In case your mesh has some issues regarding non-manifoldness or there are isolated pieces flying around, you can use vcgIsolated and vcgClean to remove those.

Examples

```
mysphere <- vcgSphere()
vcgVolume(mysphere)
## Not run:
## here is an example where the mesh has some non-manifold vertices

mysphere <- vcgSphere(normals=FALSE)
## add a degenerate face
mysphere$it <- cbind(mysphere$it,c(1,2,1))
try(vcgVolume(mysphere))

## fix the error using vcgClean():
vcgVolume(vcgClean(mysphere,sel=0:6,iterate=TRUE))

## End(Not run)
```

vcgWrlWrite *Export meshes to WRL-files*

Description

Export meshes to WRL-files

Usage

```
vcgWrlWrite(mesh, filename = dataname, writeCol = TRUE, writeNormals = TRUE)
```

Arguments

mesh	triangular mesh of class 'mesh3d' or a numeric matrix with 3-columns
filename	character: filename (file extension '.wrl' will be added automatically.
writeCol	logical: export existing per-vertex color stored in mesh$material$color
writeNormals	write existing normals to file

Examples

```
data(humface)
vcgWrlWrite(humface,filename = "humface")
unlink("humface.wrl")
```

Index