GigaMesh Short Manual

This is an application-based manual for the GigaMesh Software framework. Many possible applications are only mentioned here and it is also possible to combine some of the steps described here. Most bugs that might occur can be solved by selecting another option and then the buggy option again, by saving and reloading the model or saving the model and restarting GigaMesh.

First Steps

The GigaMesh User Interface

After starting GigaMesh, open a mesh via ***File*** → *Open* (`ctrl` + `o`) and the following dialogue. Various 3D-formats are supported. If an error occors, load the file in MeshLab (freeware), export it as .ply and enable *binary encoding*. GigaMesh only displays vertex colors.
On the top there is the menu bar where most commands are located.
In the bar below there are buttons for light settings, for reloading the current mesh (e. g. if something unwanted happens) and the inspection view which shows erroneous vertices. Presets for matted and metallic rendering are also found here, as well as shortcuts for the background grid and transparency.
In the left side bar some information is displayed, important for this manual are the *DPI, Label ID* and *SelMVerts*.
The right side bar offers some options for the surface representation. If

the mesh is not showing what it should, choose another option and then the first option again.
In the menu below are options to move the plane, lights and camera.
There are shortcuts for many steps described in this manual. They can be displyed via ***?*** → *Keyboard Shortcuts*.

Settings

Under ***Settings*** are some parameters of interest. The background can be modified here (grid or circles, to put it in front, choose __*Settings* → *Grid shift depth*), and it can be chosen between orthographic or perspective view. Furthermore, the *DPI-Settings* (`shift` + `F7`) (zoom, see chapter 'Export Screenshots') and lighting settings aswell as the specular- and shininess-settings are found here. Presets for these functions are found in the button-bar.
Under ***Colors*** several options are found, perhaps of interest is the *Backface Color* (color of the unscanned backside of a surface). There is also a *Sketch Rendering* option and its settings.

Navigating

Check if *Move Camera* is set in the right side bar. To move the camera around the mesh, click and hold the left mouse button to move the camera around the central point. Use the right mouse button to drag the mesh without turning it. To zoom, use the mouse wheel or ***Settings*** → *Ortho Scale (Set DPI)*. For a more precise orientation, keyboard shortcuts can be used for rotation steps of 1° or 90° in all

directions. Choose ***?*** → *Keyboard Layout* to display these shortcuts. For a even finer orientation, use ***View*** → *Rotate - e. g. Roll* and fill in e. g. (-)0.5 for a rotation of 0.5 degree.
If needed, the rotation center can be set to a selected point on the mesh (`ctrl` + left click) via ***View*** → *SelPrim to View Reference*. To save the current orientation of the mesh, press `F6` and save the file. A transformation matrix is created in a .tex file in the mesh directory. This matrix can be applied again via ***Edit*** → *Apply 4x4 Matrix - All Vertices*.

The Mesh Plane

A plane is always present in the workspace. It is needed for several operations and can be used e. g. as clipping plane (***View*** → *Mesh Plane as Clipping Plane*) or to calculate and visualize distances to the mesh (hight maps). It can be shown/hidden it via ***View*** → *Mesh Plane* (`shift` + `F1`).
There are several options to set this plane. Most intuitive is ***Select*** → *Plane - 3 Points* where the plane is defined by three points on the mesh. To select them, hold `ctrl` and click on the mesh. Another and probably the best option is to orientate the mesh precisely (has to be done anyway) and then choose ***Select*** → *Plane - Set HNF by View*. Here the plane is set orthogonal to the camera position. Turning the camera 90° allows to move the plane in position (holding shift and dragging the plane with the right mouse button).
The plane can be moved precisely with the same keyboard shortcuts as the camera while holding `shift`.

Scaling

To scale a mesh, choose ***Edit*** → *Apply Scale/Skew - All Vertices*. Enter the scale factor in the appearing box, e. g. 0.5 for half the size or 2 to double the size of the mesh.

For selective scaling along one or two axis, choose ***Edit*** → *Apply 4x4 Matrix - All Vertices* and enter a 4x4 matrix, e. g. for uniform scaling:

> *(scale factor) 0 0 0*
>
> *0 (scale factor) 0 0*
>
> *0 0 (scale factor) 0*
>
> *0 0 0 1*

When inserted, this matrix looks like this:

> *2 0 0 0 0 2 0 0 0 0 2 0 0 0 0 1*

Transparency

Transparency can be enabled via ***View*** → *Transparency*.or the button in the button bar. The settings can be adjusted via ***View*** → *Transparency Settings*. It can be set dependend of the function value and so gradients of transparency can be created, e. g. in combination to the *Distance to Point* or *Distance to Plane* functions.

Labeling and Function Values

Labeling is used to group parts of the mesh by selected properties. These can be unconnected parts of the mesh (see chapter Manual Mesh Cleaning) or the function value. This value is calculated for each vertex with the various functions GigaMesh offers such as distance functions and feature vectors. It is possible to select all vertices with the same function value or all vertices with a function value higher or lower than a set value. To view the function value of a vertex, hold `ctrl` and click on the mesh with the left mouse button. The FuncVal in the left side bar shows the function value of the set point. To copy it, press `i`. To select labeled areas, choose *Select → Vertices - SelMVerts - by Value/Type → Vertices - Label No. Insert the label number(s) to select (e. g. 1 5 9) or leave unselected (e. g. -1). To select several labels at once, leave a `space` between the numbers. Function values can be visualized with colorramps. Similar colors show a similar function value. See chapter *Colorramps* to manipulate them.

Examples

Cut the Mesh by Function Value

Labeling can be used to cut a mesh with a plane (in general this also works with points, spheres or cones). Set the mesh plane where the cut should be made and choose ***Functions*** *→ Distance to Plane*. All vertices on the plane have now the function value 0, this number increases as the distance to the plane becomes greater, on one side with positive, on the other side with negative numbers. Now click ***Select*** *→ Vertices - SelVerts - FuncVal </>* to select vertices with a

function value higher/lower than the value entered in the right box. In this example, enter 0.
The vertices on one side of the plane are now selected and can be removed via ***Edit*** → *Remove SelMVerts*. Note that entering another value expands the selection. To start again, choose ***Select*** → *Vertices - SelMVerts - Deselect All*.

Label Areas by Function Value (Example: Cuneiform Tablet)

To label several areas of interest, compute the feature vectors (see chapter Feature Vectors) and open the new .ply-file with these vectors included. Choose ***Functions*** → *Feature Vector Length (Man/1-Norm)* or another feature vector visualization. It is recommended to use isolines now by choosing ***Colorramp*** → *Isolines* To adjust them, use *Isolines - Distance* and *Isolines - Offset* until only one is left on the border of the area of interest. These lines run along the surface and stay on vertices with the same function value. Select a point near the border of an area of interest by holding `ctrl` and clicking on it with the left mouse button. Activating ***View*** → *Edges* mand zooming in makes them easier to find. The function value is displayed in the left side bar. Then choose ***Select*** → *Vertices - SelVerts - FuncVal </>*. The function value should be pasted automaticly in the right field. Click *OK* and all vertices in the areas of interest are selected. If it is the other way around, choose ***Select*** → *SelMVerts - Invert selection*. Remove the green crosses on each vertex by clicking ***View*** → *Vertices* → *Selected (SelMVerts)*.

To label the slected vertices, choose ***Labeling*** → *Label SelVerts*. In the right side bar, select *Vertex Labels* to display them. Each label number is displayed in one of ten colors.

Mesh Cleaning

Automatic Mesh Cleaning

Mesh cleaning consists of several steps that can be executed separately. The easiest and often satisfying approach is ***Edit*** → *Automatic Mesh Polishing*. It removes outliners, zero area vertices etc. and fills holes. The slider is the threshold of the size of unconnected parts of the mesh to be deleted. Default is 10%, everything that is smaller than 10% of the largest part of the mesh will be deleted. Preventing the longest polyline from getting filled makes sense with incomplete scanned objects, e. g. vessels with an unscanned interior or stationary structures where it is not possible to scan all sides. It is recommended but not necessary to store the result as a new file.

Automatic Mesh Cleaning for multiple Files

The script `automatic_cleaning.sh` in the folder ‘Scripts’ in the ‘GigaMesh’ installation folder applies the Automatic Mesh Cleaning Process to multiple files. Copy the file `automatic_cleaning.sh` to the directory with the 3D data, open a terminal and navigate to this folder (e. g. by typing `cd Documents/FolderWith3Ddata`). Then type

`./automatic_cleaning.sh` and hit `enter`. For each input file, the output consists of one or two files, one with the appendix _GMC after the original file name, meaning *GigaMeshCleaned* and if needed, one with the appendix _GMCF, meaning *GigaMeshCleanedFilled* with the holes filled.

Manual Mesh Cleaning

For a more selective cleaning of the mesh some steps have to be done manually.

To remove only selected parts of the mesh, choose ***Labeling*** → *Label - All Vertices / SelMVerts*. When asked, is not necessary to set the label step number as function value. Every part of the mesh that is not connected to others gets now a Label ID. Hold `ctrl` and click on the object. In the left side bar, the *Label ID* of the selected vertex and its connected components is shown, usually 1 for the largest. To keep this and delete and all parts that are unconnected to it, choose ***Select*** → *Vertices - SelMVerts - by Value/Type* → *Vertices - Label No*. Insert the Label ID(s) to select (e. g. 1) or leave unselected (e. g. -1). The selection may also be inverted via ***Select*** → *Vertices - SelMVerts - Invert selection*. If more than one element should be selected, leave a space between the label numbers (e. g. 1 2 3 5). When everything unwanted is selected, choose ***Edit*** → *Remove SelMVerts*.

The further manual cleaning operations are

Select → *Faces - Sticky* and ***Edit*** → *Remove - SelMVerts*.

Select → *Faces - Non-manifold* and ***Edit*** → *Remove - SelMVerts*.

Select → *Faces - Zero Area* and ***Edit*** → *Remove - SelMVerts*.

These steps are included in the *Automatic Mesh Polishing* application.

Filling Holes can be separately done by choosing ***Analyze*** → *Mesh Borders to Polylines* and ***Edit*** → *Fill Holes (PSALM)*.

Polygonal Selection

Another option to manually delete parts of the mesh (e. g. parts that belong to a label number that should be kept) is to choose ***Select*** → *Primitive - Polyline*. Hold `ctrl` and click in the workspace to create a polyline (shown in red). Click with the right mouse button to set the last point and close it. In the left side bar, the number of selected vertices (SelMVerts) is displayed. The selected vertices may be deleted by ***Edit*** → *Remove SelMVerts*.

Reopen filled holes

If there are some holes in the mesh that should not have been filled, choose ***Select*** → *Faces - with Synthetic Vertices*. Then click ***Labeling*** → *Label - SelMVerts* and do not set the function value. Choose ***Select*** → *Vertices - SelMVerts - Deselect All*. Then ***View*** → *Vertices* → *Selected (SelMVerts)*. The next step is ***Select*** → *Vertices - SelMVerts - GUI*. Then hold `ctrl` and click on the synthetic vertices marked in orange to select those to be reopened. A green star appears on them. Then use ***Edit*** → *Remove filled Holes (SelMVerts)* to delete the filling.

Bug: If it does not work, choose ***Select*** → *Prinitive - Vertex - SelVert/SelPrim* and then ***Select*** → *Vertices - SelMVerts - GUI* again or restart GigaMesh.
To get rid of the green and orange crosses/surfaces, uncheck ***View*** → *Selected (SelMFaces)*, ***View*** → *Vertices* → *Solo* and ***Select*** → *Vertices - SelMVerts - Deselect All*.

Cross Sections

To create a cross section of an object, navigate the camera to the desired view position of the profile (typically front view).
Save the view position by pressing `F6`. If the mesh is moved accidentally during the following steps, the cross section will be crooked. If that happens, press `shift` + `F12` to go back to the default view and start again.
Choose ***Select*** → *Plane - Set HNF by View* to create a cutting plane through the mesh. Rotate the camera by 90° (Keyboard Shortcut `X` or `Y`) and the plane is seen as a line. Drag it where the cross section should be made using the right mouse button while holding `shift`.
Choose ***Functions*** → *Distance to Plane*, in the pop-up window, click *OK* and check ***Colorramp*** → *Isolines.* If needed, select the desired surface representation in the right side bar. There will be a cross section made at every isoline. In order to reduce them or to change their distance, select ***Colorramp*** → *Isolines - Distance* and move the slider until the desired number of isolines is reached. Slide it to the far right to create only one cross section where the plane is and rotate the mesh back to the default view (`shift` + `F12`).

Optionally, the profile can be controlled before export by ***Colorramp*** *→ Isolines only*.

Click ***Analyze*** *→ Iso Line to Polylines* and ***File*** *→ Export Screenshots → Screenshots (SVG+PNG)*. When prompted, it is not necessary to save as PNG.

In Inkscape (freeware, recommended for post-processing) the result will load as a group to include the background grid, a ruler, the plane and the profile line(s). If ***Colorramp*** *→ Isolines only* is (left) unchecked, also the view of the mesh will also be included. To ungroup it, right click and select *ungroup*.

To create multiple cross sections of an approximately rotational symmetric object, perform the described steps after an unwrapping and keep more isolines.

Unwrappings

Meshes can be unwrapped using the surface of a cylinder, a cone or a sphere. Of course, this only makes sense with approximately rotational symmetric surfaces. The closer the surface of the object is to the surface of the cone or sphere the more precise will be the results.

Before starting, orientate the object and save the orientation hitting `F6`. The unwrapping process will use the vertical axis.

Cone Unwrapping

Orientate the object with the rotational axis in vertical position and hit `F6`. It is necessary to specify where exactly the rotational axis of the cone is.

To do so, choose from two different options:

1. Choose the button *Polar Grid*, move the camera to a view position along the rotational axis and align the outer contour that should be unwrapped to the circles by draging the mesh with the right mouse button. Then choose ***Edit*** → *Cone - Set Axis Central Pixel* to set the rotational axis. To view the central pixel, click on *Additional cross-hair* in the button bar.
2. Choose ***Select*** → *Positions - SelPrims*, hold `ctrl` and click on three or more points on more or less rotational symmetric parts at the same elevation around the object, the last one with the right mouse button. Blue dots appear at these points. Do this on two or three elevations to get a good result (e. g. 5 points each on 3 elevations). Then choose ***Analyze*** → *Positions - SelPrims - Compute Circle Centers*. Confirm the two boxes showing up. The calculated rotational axis is shown as a green line.

Choose ***Select*** → *Cone* and press and hold `ctrl`.

Bug: If nothing happens, choose ***Select*** → *Sphere* and ***Select*** → *Cone* again.

When holding `ctrl`, a sketch of a cone appears in the right side bar. Click on the two points on the mesh that represent the upper and lower end of the cone in the sketch. It does not matter if the larger/smaller or upper/lower diameter is set first. When finished, a reddish cone with horizontal (seen from the orientated position) isolines appears in the workspace. To unroll the complete mesh and not only the selected cone, choose ***Select*** → *Cone: Cover the whole mesh* to enlarge the cone until it covers the hight of the whole mesh. It is also possible to adjust the upper and lower radius manually via ***Edit*** → *Cone - Set Data*.

The next step may be easier if transparency is enabled (***View*** → *Transparency*).

In order to unwrap an object, it has to be cut somewhere. Choose ***Edit*** → *Set Prime Meridian for Rollouts*. Move the slider until the red line on the cone reaches the desired cutting position. Choose ***Edit*** → *Cone - Unroll Mesh* to perform the unwrapping. Select the desired surface representation in the right side bar.

The unwrapped mesh is still a 3D-Model and can be navigated and manipulated normally. Starting from here it is also possible to perform another unwrapping if necessary. Save and reload the model after each unwrapping.

Sphere Unwrapping

Orientate the object with the rotational axis in vertical position and hit `F6`. Choose ***Select*** → *Sphere*, then press and hold `ctrl`.

Bug: If nothing happens, choose ***Select*** → *Cone* and ***Select*** → *Sphere* again.

A sphere can be defined by four points, a sketch of such a sphere appears in the right side bar, showing how many points out of four are already set. While holding `ctrl`, click on various points at the surface of your model to set these points. When finished, a reddish sphere with horizontal isolines will appear in your workspace.

The next step may be easier if transparency is enabled (***View*** *→ Transparency*).

In order to unwrap an object, it has to be cut somewhere. Choose ***Edit*** *→ Set Prime Meridian for Rollouts*. Move the slider until the red line on the cone reaches the desired cutting position. Choose ***Edit*** *→ Sphere - Unroll Mesh* to perform the unwrapping. Select the desired surface representation in the right side bar.
The unwrapped mesh is still a 3D-Model and can be navigated and manipulated normally. Starting from here it is also possible to perform another unwrapping if necessary. Save and reload the model after each unwrapping.

Cylinder Unwrapping

To unwrap an object on a cylindrical surface, set the central pixel of the rotation axis (see Cone Unwrapping). Then choose ***Edit*** *→ Cylinder - Set Radius*. Enter the radius in mm, click *OK* and choose ***Edit*** *→ Cylinder - Unroll Mesh*. Other than a cone unwrapping, the mesh will be streched and/or compress to fill the cylinder’s unrolled surface.

Extrude Profile Lines

It is possible to create an extrusion of a profile line around the rotational axis of an object. First, this axis has to be set by one of the following options

1. Choose the button *Polar Grid*, move the camera to a view

position along the rotational axis and align the outer contour that should be unwrapped to the circles by draging the mesh with the right mouse button. Then choose ***Edit*** → *Cone - Set Axis Central Pixel* to set the rotational axis. To view the central pixel, click on *Additional cross-hair* in the button bar.

2. Choose ***Select*** → *Positions - SelPrims*, hold `ctrl` and click on three or more points on more or less rotational symmetric parts at about the same elevation around the object, the last one with the right mouse button. Blue dots appear at these points. Do this on two or three elevations to get a good result (e. g. 5 points each on 3 elevations). Then choose ***Analize*** → *Positions - SelPrims - Compute Circle Centers*. Confirm the two boxes showing up. The calculated rotational axis is shown as a green line.

Then choose ***Functions*** → *Angle bases on Cone Axis*. Also choose ***Select*** → *Primitive - Vertex - SelVert/SelPrim*. Then hold `ctrl` and click on a point on the mesh where, following the rotational axis, the extrusion should be made. An orange cross appears there. Press `I` (for ‘information’) and copy the number from the *Function value*. Then choose ***Analize*** → *Iso Value - Set*, enter the copied number and click *OK*. Then choose ***Analize*** → *Iso Line to Polylines*, followed by ***Edit*** → *Extrude Polylines*.
The extruded polyline lies now inside the mesh, interlacing with it. To display or save the extruded part only, choose ***Select*** → *Vertices - SelMVerts - by Value/Type* → *Vertices - Synthetic*. Then choose ***Select*** → *Vertices - SelMVerts - Invert Selection*. Then the original mesh can

be deleted by choosing ***Edit*** → *Remove - SelMVerts*.

Distance Visualization

There are several options to visualize distances using isolines, colorramps or gradients. To change their visualization settings, see chapter 'Colorramps'.

Distance to a Plane

For flat objects like stone inscriptions a visualization of the distance to a plane can be of interest. Orientate your object with the side of interest orthogonally to the camera. Choose ***Select*** → *Plane - Set HNF by View*. Then choose ***Functions*** → *Distance to Plane*. This can also be applied to a sphere/cone unrolling and equals a distance to sphere/cone calculation in 2.5D. This function is also needed for cross sections.

Distance to a Cone

To set a cone, see chapter Cone unwrapping.

When a cone is set in the mesh, choose ***Functions*** → *Distance to Cone*.

Distance to a Point

This function shows the distance from a selected point. Either the absolute distance is calculated or the geodesic distance along the surface, where also more than one point can be visualized. Press `ctrl` and click on the spot of interest. An orange cross will appear there.

Absolute Distance

To calculate the absolute distance to a point, choose ***Functions*** → *Distance to SelPrin (COG)* and press *OK*. Adjust colorramps and isolines as prefered.

Geodesic Distance

To calculate the geodesic distance to the set point, choose ***Analyze*** → *Geodesic Distance to SelVerts*. Press *NO* and then *YES* in the following dialogues. It is also possible to select several points at the same time. To do so, choose ***Select*** → *Vertices - SelMVerts - GUI*. To view them (optional), choose ***View*** → *Vertices* → *Selected (SelMVerts)*. Then choose ***Analyze*** → *Geodesic Distance to SelVerts*. Isolines or a repeated colorramp (see chapter "Colorramps") improve the visualization of the result.

Distance Measurement between Points

Absolute Distance

To measure the absolute distance from point to point, choose ***Select*** → *Positions - SelPrims*. Then hold `ctrl` and click with the left mouse button on the points to measure. Blue dots appear at each point. There may be set more than two points at once but the last point should be set with the right mouse button to end the polyline. It is also possible to set another polyline afterwards. To display the measuring results, choose ***Analyze*** → *Positions - SelPrims - Euclidean Distances*. A box appears with the lengths of the segments of the measured polyline(s), ordered as measured.

To deselect the points, choose ***Select*** → *Positions - SelPrims - Deselect All*.

Geodesic Distance

To measuer the geodesic distance between two points along the surface, press `ctrl` and click on the first point. Choose ***Analyze*** → *Geodesic Distance to SelVert*. Then press `ctrl` again and click on the second point. Press `i` to view the information about it. The function value is the distance between these points.

Volume

The volume of a mesh can be computed via ***Analyze*** → *Compute Volume (Plane)*. If the plane intersects the mesh, the two values represent the volume on both sides of the plane. To get the total volume, combine these values or move the plane aside and only one value >0 remains. Computing the volume works best with watertight meshes, if the mesh has holes, the volume is estimated.

Colorramps

Colorramps are used to visualize calculated function values like distances and curvatures. It can be chosen from several colorramps under the entry ***Colorramp*** in the menu bar. They also can be manipulated in many ways with the settings below. *Invert* inverts the Colorramp, *Logarithmic* increases the size of one end of the colorrramp on cost of the other end. The *Quantil Min/Max* settings shrink the colorramp to its center from both ends, the area covered by the two outer colors is enlarged. The colorramp can also be repeated vie __*Colorramp → Repeat Map (Waves)* and the intervall can be set via *Set Intervall (Wavelength)*.
Under ***Colorramp*** also the *Isolines* and their settings (e. g. *- distance, - offset, - only*) are found. To set isolines in a certain distance, choose ***Colorramp*** → *Isolines - Distance by Number*.

Feature Vectors

These vectors contain additional information per vertex concerning surface and volume of a set of spheres intersecting the mesh. This operation is time consuming (hours of computing time, depending on file size and computing power) and runs without graphical user interface. It is recommended to clean the mesh before calculating the feature vectors.
They are computed by opening a terminal and typing `cd GigaMesh/mesh`. GigaMesh stands here for the GigaMesh installation folder. Then start the program `nohup`

`./meshgeneratorfeaturevectors25d_threads -f <path-to-filename> [-r 1] &`.

The parameter given behind the option -r displays the radius of the maximal sphere im mm for the MSII (Multi Scale Integral Invariant) Filter. The default value has been set to the unit 1.0. It can be changed into some other appropriate number depending on the size of the features to be detected. An educated guess is the maximum size of the feature width. Smaller values will only detect noise and bigger values lose the fine tuning when operating with the feature vectors in GigaMesh.
Note again that depending on the file size of the object this step can take several hours. Every time when having processed 5000 vertices the program approximates the estimated time of finishing. This is written to the standard output device which is the terminal. Multithreading is active.

Finally, there should have appeared six additional files adjacent to the data file. The naming convention added the radius _r1.0, the number of scales in power of two _r1.0 means 2^4 = 16 equidistant radii in the interval [r, 0] , and a rasterization of e.g. the volume with e.g. _v256 means 256 voxels along the diameter of the biggest sphere. One of the files is a short text file containing information on the process itself (with the suffix info.txt), four of them store the feature vectors separated in only volume or only surface or both, or store the normals. They have the suffix *.mat,
where the wildcard * stands for surface, volume, vs or normal.
These files can be used to re-import feature vectors if they have been

discarded when saving other information important for a certain visualization. The sixth data file is a *.ply -file which also contains the feature vectors and can be processed by the graphical user interface of GigaMesh for visualization purposes.

Visualizing Feature Vectors

There are several ways to visualize feature vectors. Open the computed <filename>_r1.00_n4_v256.ply file. Select the deepest feature that should be visualized by holding `ctrl` and clicking on it. Then choose ***Functions*** → *Feature Correlation to SelVert*. Good results can also be achieved via ***Functions*** → *Feature Vector Length (Man/1-Norm)* and ***Functions*** → *Feature Vector Bounded Variation (BV)*. Use the ***Colorramp*** settings to adjust the visualization (especially the *Quantil* settings). Try also isolines for some visualization purposes.

Export Screenshots

There are several ways to export images from GigaMesh. Everything that is visible in the workspace will also be exported in the image, e. g. the plane (uncheck ***View*** → *Mesh Plane*) or solo vertices (uncheck ***View*** → *Vertices* → *Solo*). Adjust the lighting settings if needed (***Settings*** → *Lighting ff.*). Using ***Settings*** → *Ortho Scale (Set DPI)* allows to enter the DPI manually. Save the desired object orientation with `F6` before proceeding. In order to export true scale screenshots, see if ***Settings*** → *Orthographic View is enabled* (default). The exported images have their DPI-number included in their file names.

Single Screenshot

For a simple screenshot of the workspace, choose ***File*** → *Export Screenshots* → *Screenshot (TIFF/PNG)*. When asked for tiled rendering, choose *NO* for a screenshot of the current view with screen resolution and background grid. When tiled rendering is used, the screenshot will cover the whole mesh in a resolution depending on the DPI settings and not include the background grid.

Multi-View Images

It is possible to export multiple views of an object via ***File*** → *Export Screenshots* → *Screenshot Views*. The settings for DPI and tiled rendering are the same as for single screenshots.

For a classic orthogonal six-side view, check ***File*** → *Export Image Stack* → *Camera on Longitudinal Orbit (Vertical Axis)*.

For eight views (e. g. common for cuneiform tablets) check ***File*** → *Export Image Stack* → *Camera on Altitudinal Orbit (Horizontal Axis)*.

Ruler

A ruler can be exported via ***File*** → *Export Screenshots* → *Screenshot Ruler*. Do not change the DPI between exporting the ruler and the associated screenshots.

Image Stack for Videos

To create an image stack that can be used for videos, the camera is moved around the object creating several hundred images. Specify the parameters for the camera orbit via ***File*** *→ Export Image Stack*. To move the camera around the object horizontally, choose ***File*** *→ Export Image Stack → 360° Circular Orbit Horizontal Axis*. To move the camera around the object vertically, choose ***File*** *→ Export Image Stack → 360° Circular Orbit Vertical Axis*.

Bringing the DPI into the Images

The script `setdpi.sh` in the folder 'Scripts' in the 'GigaMesh' installation folder encodes the DPI values into the exported .PNG images. This has not to be applied to .SVG files. Copy the file `setdpi.sh` to the directory with the images, open a terminal and navigate to this folder (e. g. by typing `cd Documents/FolderWithImages`). Then type `./setdpi.sh` and hit `enter`. This will take the DPI value from the file name and encode it into the file. The input files are moved to a new sub-folder named 'converted'.