

PVMOS manual - Version 0.83

B.E. Pieters*

*Institut für Energie und Klimaforschung - IEK5 Photovoltaik,
Forschungszentrum Jülich, 52425 Jülich, Germany*

(Dated: February 1, 2016)

I. INTRODUCTION

This is the manual for the Photo-Voltaic MOdule Simulator (PVMOS). PVMOS is an ordinary differential equation solver using finite-differences specifically designed to electrically model solar modules. For more information on how that works I refer the reader to [1]. The purpose of this document is to document how PVMOS is operated and installed. In the following sections I discuss in order, the installation, basic usage and operating principles and finally a detailed discussion of all available functions.

II. DISCLAIMER

If this software ever does anything you think I claim it does or should do, consider yourself lucky.

PVMOS Copyright (C) 2014 B. E. Pieters

This program comes with ABSOLUTELY NO WARRANTY. This is free software, and you are welcome to redistribute under certain conditions. You should have received a copy of the GNU General Public License along with this program. If not, see <http://www.gnu.org/licenses/>.

III. WHAT IS PVMOS

In its essence PVMOS is a Poisson solver. The Poisson equation is solved for several stacked, 2D “electrodes”, where each electrode is coupled to the electrodes above and below (note that a stack of 2D electrodes makes a 3D structure). The electrodes itself are simply conductors and behave linearly (Ohmic). However, the connection between the electrodes can be non-linear (e.g. a diode). Now, there are several limitations of the structures that PVMOS handles. The main limitation is that the meshes are 2D as in PVMOS all electrodes share the same 2D mesh. This means that PVMOS is specifically designed for flat layered structures. PVMOS uses straight forward finite-differences to solve the coupled Poisson equations. As such the 2D meshes in PVMOS divide the 2D surface in rectangular elements. Now every element in the mesh must be associated with certain properties. Storing properties on a per element basis would put a heavy burden on the memory resources. For that reason elements are grouped into “areas” where each area constitutes a certain combination of properties. A mesh therefore also contains a list of all areas and every element is a member of one of the areas in the mesh (note that an element is always assigned to one area, i.e. you need at least one area for every unique combination of local properties). Also note that as all electrodes share the same 2D mesh, one area contains the properties for all electrodes and connections between them.

PVMOS is a commandline program which must be given an file. The PVMOS input file is processed sequentially and contains commands that create or modify meshes, perform a simulation, or export data to a file. A complete list of commands is presented in Section VII A. In PVMOS we can define more than one mesh at the same time. This is useful as it allows you to create new meshes by joining two meshes, e.g., to create several series connected cells in a thin-film module. In order to reference one particular mesh, each mesh has a name. This name must thus be provided to any command that operates on a mesh. Likewise, some commands operate on an area within a mesh and thus areas also have a name. To reference an area within a mesh we use the following syntax `<mesh-name>.<area-name>`, i.e. the name of the mesh followed by a dot and the name of the area within that particular mesh. Sometimes we

*Electronic address: b.pieters@fz-juelich.de

need to select elements in a mesh (for example when we want to assign a set of elements to a certain area). To this end each mesh has a list of selected elements. If you select elements in a mesh the list is occupied by the element ID's, after which you can do operations on the selected elements. Note that elements are selected on a per mesh basis.

It may be useful to store intermediate results, especially as sometimes creating the mesh may take some time and you want to do many different calculations based on the same mesh. For such cases PVMOS has a binary file format to store complete meshes with all elements, areas, properties and simulation results. This mesh may be loaded later to continue where you left of. The primary focus is to store intermediate results and as such not much value is put on compatibility between different PVMOS versions, i.e. compatibility between the file format of different PVMOS versions is not guaranteed (in general compatibility will break if a new version makes any change in the mesh data structures). The binary format is also used for a dynamic library for GNU Octave (an open source tool largely compatible with Matlab), which allows you to generate a mesh for PVMOS within octave and store it in binary form for use in PVMOS.

IV. BASIC USAGE

The input for PVMOS is a plain text file with commands. To solve a problem PVMOS is typically called from the command line with as an argument the file name describing the problem. With these input files you can specify the geometry of your solar cell/module including the local properties such as electrode sheet resistances and solar cell properties. You can also specify which calculations you want PVMOS should perform and what data to save. A call to PVMOS from the command line looks like this

```
pvmos [options/variables] <input-file>
```

where the optional argument [options/variables] is used to set options and define variables. Valid options and variable definitions are:

- “-q” Set verbosity level to quiet (i.e. suppress all non essential output)
- “-n” Set verbosity level to normal
- “-v” Set verbosity level to verbose
- “-db” Set verbosity level to debug
- “-vf” Indicates next argument is a variable file
- “<name>=<value>” Define variable <name>. Note there should be no white spaces within the definition

A variable file is a file with one variable per line of the form “<name>=<value>”. In variable files all spaces are ignored. Lines that do not follow this format are ignored. Note that not all variable names are allowed, see Section VII B. The last option must be the PVMOS input file, a plain text input file with commands for PVMOS.

V. INSTALLATION INSTRUCTIONS

PVMOS is a command-line application written entirely in C. For the operation PVMOS depends on several libraries, most notably `cholmod`, for the solving of sparse linear systems. PVMOS has been tested on Linux and Windows systems. To install PVMOS you need to either compile the source, for which a Makefile is provided, or you need to download pre-compiled binaries for your system (if available). Apart from the PVMOS program itself the package also includes several optional components that may be useful when working with PVMOS. The optional packages are:

- The `pvmos-mesh` package for GNU Octave
- The `extract-poly` extension for Inkscape
- The `polytransform` program.

The optional components may aid the creation of specific device geometries in PVMOS. The `pvmos-mesh` package for GNU Octave allows the creation of rectilinear meshes for PVMOS from within GNU Octave (GNU Octave is a mathematical program similar to Matlab). PVMOS allows the use of polygons to define areas in a device. The `extract-poly` extension for Inkscape aid the creation of complex polygons. Inkscape is a vector graphics program and as such

suitable to create and manipulate polygons graphically. The extension allows the user to extract polygon data from an Inkscape graphics. The polytransform program is a commandline utility implementing some basic transformations to manipulate polygons for use in PVMOS. The polytransform program implements some basic operations on polygons like scaling rotation, etc. Note that newer versions of PVMOS also implements such transformations internally so the polytransform tool is now redundant.

A. Compiling from source

Here are the instructions to compile the PVMOS executable. The instructions describe compilation on UNIX/Linux. On Windows you'll probably want to use something like Msys/Mingw or Cygwin to compile PVMOS.

1. Install PVMOS's dependencies (**cholmod** and **libmatheval**). These libraries should be readily available in most linux distributions (the **cholmod** library is part of **libsuitesparse**).
2. Download the pvmos package (**pvmos-0.83.tar.gz**)
3. Unpack the archive and do `./configure` [optional arguments] followed by `make`, and as root `make install`

At the end of the configure command the script will present a summary of the configuration. Please check the configuration and proceed if all is correct. To get a full list of configure options run `./configure --help`. Here are some options you may need:

- `--with-blas=<BLAS link flag >` Link to your preferred BLAS library (see below)
- `--enable/disable-openblas-multithreaded` Enable/disable multithreading for OpenBLAS, per default is it disabled (and that is good).
- `--enable/disable-generate-octave-pkg` Generate a package for GNU Octave (default is enabled).
- `--enable/disable-dmalloc` Link PVMOS to the `dmalloc` memory allocator (see below). The default is disabled.
- `--enable/disable-inkscape-extension` Enable/disable the installation of the extract-poly extension for inkscape (if inkscape is installed). Per default this option is enabled.

The performance of PVMOS is strongly dependent on the performance of the sparse linear solver (i.e. **cholmod**). In turn the performance of the **cholmod** library depends strongly on the BLAS library it is linked to. If your **cholmod** library is linked to a less than optimal BLAS you may still obtain a decent performance by linking PVMOS to another BLAS library. For this reason PVMOS will search for an optimized BLAS on your system. You may also provide a BLAS library explicitly by specifying the appropriate linker flags using `--with-blas=<lib >`. You may also specify `--with-blas=no`, in which case the BLAS library is used to which **cholmod** on your system happens to be linked to. If OpenBLAS is available on your system PVMOS will, per default, try to link to it. This is generally a good choice but often it is found that multi-threading in OpenBLAS leads to a poor performance in **cholmod**. For this reason PVMOS will turn off multi-threading in OpenBLAS. You can enable multithreading in OpenBLAS explicitly using the `--enable-openblas-multithreaded` option.

If you compile PVMOS on Windows using mingw chances are you are linking to the "mscrt.dll" library. It appears the memory allocator therein performs rather poorly with PVMOS. In this case it is recommended you specify `--enable-dmalloc` to the configure script. This will link PVMOS to another memory allocator which should result in a better performance.

B. Pre-compiled binaries (Windows)

There are two pre-compiled versions for windows, 32 and 64 bit. If your system supports it I suggest to use 64 bit as 32 bit applications are rather limited in the maximum allocatable memory space (i 2Gb), you may run into this limit for larger problems. The binary distribution consists of an executable with several dll's. The installation procedure for PVMOS manual as no installer is provided (you can help PVMOS by making one). So here is the procedure for a binary install:

1. Download the binary
2. Unpack the files in a directory of your choice
3. Optional: Add the path to your PVMOS executable to the PATH variable

1. Setting the “PATH” variable

A properly set PATH variable will allow you to execute PVMOS from the command line without typing the full path to the PVMOS executable. Here I describe how to edit your path variable in Windows 7. First another word of warning. Adding things to the path variable is mostly harmless, however, if you break your path string (e.g. delete a part) you may seriously impair the functioning of programs that need the path variable. Follow these instructions on your own risk!

1. Click on the start menu
2. type “PATH” in the “search programs and files” field, you should now find options to edit the path variable for the system or for the current user, select to edit it for the current user
3. Select the path variable
4. Click on Edit
5. Scroll to the end of the string and add `;<installation directory>`, where `<installation directory>` points to the location of your PVMOS executable
6. Click on OK
7. Click on OK

C. Installing the GNU-Octave PVMOS-mesh package

The PVMOS package for GNU Octave is a set of tools to create and save binary PVMOS meshes from within GNU Octave. A more detailed description of the package can be found in Section VIII. Installing the PVMOS package in Octave is through the “pkg” command, i.e. the usual procedure to install packages in GNU Octave. Prepared packages are available for download and are named `PVMOS-mesh-|version|.tar.gz`. The package can also be created from source using “make PVMOS-mesh.pkg”. After installing the package you can use “pkg load PVMOS-mesh” to load the functions for use.

VI. RECOMMENDED SOFTWARE

Obviously you are free to use whatever software you want. However, to process data from PVMOS or to get data into PVMOS I often use file formats which will not be supported by all software packages. I thus here detail my software setup in the hope it helps you. All software here is free and open source software.

A. Get data into PVMOS: GNU Octave & Inkscape

These programs are useful when defining complex geometries. If you want to define complex geometries in PVMOS there are basically three ways:

1. Use the PVMOS scripting language. The advantages of this method is that it can be very accurate and fast and is independent of any other software, the downside is that it quickly becomes tedious
2. `mkpvmosmesh` with GNU Octave. This allows you to use all the data and image processing power of Octave to generate meshes. However, this library only allows you to create mesh topologies consisting of rows and columns (i.e. meshes easily defined with a matrix)
3. Define geometries using polygons. PVMOS has good support for polygons to select and specify various areas in your device. Complicated shapes may be created in a graphics program such as inkscape and exported to PVMOS. For this I have an extension to export data from inkscape for use in PVMOS. In combination with the trace bitmap feature of inkscape you can quickly define appropriate polygons for complicated geometries if you provide a bitmap image, e.g. you can create a polygon for the metalization pattern of a device by tracing a picture. This method potentially has a lower memory footprint compared to the GNU Octave option.

Inkscape is a powerful vector graphics tool. It is convenient to use such a tool to graphically create and manipulate polygons for PVMOS. To this end I created an extension for Inkscape that takes a “path” object and approximates the path with a polygon. In general a path in inkscape consists of a set of nodes with cubic Bezier curves inbetween. To export this data as a polygon we need to first “flatten” the curve, which replaces the Bezier curves with plain straight lines, and then export this data in a PVMOS compatible form. This we do with the export-poly extension (included with PVMOS).

To use extract-poly extension select one or more paths in Inkscape and go to Extensions-¿Export-¿Extract Poly. This will present the user with a dialog where you can set the desired “flatness”. The flatness parameter controls the accuracy with which the polygon approximates the original Bezier curve, where a smaller flatness value makes the approximation more accurate but also results in more points in the polygon. If you press apply a new dialog pops up with the extracted coordinates, which you can copy and paste to a text file. You probably have to transform the polygon coordinates to the desired units, which can be done within PVMOS or using the polytransform utility.

B. Analyse simulated data: Gnuplot

The data-format that PVMOS uses for element-wise data export is laid out such that Gnuplot can make sense out of the data. Other plotting tools may not support this format. The element-wise data export allows you to export data from the mesh on a per-element basis and is particularly useful for inspecting meshes to ensure a correct mesh definition. For this reason I recommend the use of Gnuplot. Other data formats that PVMOS should not be a problem for other plotting tools.

VII. PVMOS INPUT FILE FORMAT

A. Command Reference

CREATING MESHES

Keyword	Description
newmesh	Create a new, rectangular mesh. The command takes seven arguments: <ol style="list-style-type: none"> 1. x1, x-coordinate of the lower left corner 2. y1, y-coordinate of the lower left corner 3. x2, x-coordinate of the upper right corner 4. y2, y-coordinate of the upper right corner 5. Nx, Number of elements in x direction 6. Ny, Number of elements in y direction 7. mesh-name, Name of the new mesh
addcol_left	Add a column of elements to the left side of a mesh. This command requires the mesh has no holes and the left side of the mesh is flat, i.e. the left edge is vertical. This command will add one element to the left of each element at the left edge of the mesh. The newly added elements inherit all properties from the element on their right. The command takes two arguments: <ol style="list-style-type: none"> 1. mesh-name, Name of the new mesh 2. dx, width of the column to be added.
addcol_right	Add a column of elements to the right side of a mesh. This command requires the mesh has no holes and the right side of the mesh is flat, i.e. the right edge is vertical. This command will add one element to the right of each element at the right edge of the mesh. The newly added elements inherit all properties from the element on their left. The command takes two arguments: <ol style="list-style-type: none"> 1. mesh-name, Name of the new mesh 2. dx, width of the column to be added.

<code>addrow_top</code>	<p>Add a row of elements to the top side of a mesh. This command requires the mesh has no holes and the top side of the mesh is flat, i.e. the top edge is horizontal. This command will add one element on top of each element at the top edge of the mesh. The newly added elements inherit all properties from the element below. The command takes two arguments:</p> <ol style="list-style-type: none"> 1. <code>mesh-name</code>, Name of the new mesh 2. <code>dy</code>, width of the row to be added.
<code>addrow_bottom</code>	<p>Add a row of elements below a mesh. This command requires the mesh has no holes and the bottom side of the mesh is flat, i.e. the bottom edge is horizontal. This command will add one element below each element at the bottom edge of the mesh. The newly added elements inherit all properties from the element above it. The command takes two arguments:</p> <ol style="list-style-type: none"> 1. <code>mesh-name</code>, Name of the new mesh 2. <code>dy</code>, width of the row to be added.
<code>joinmesh</code>	<p>Create new mesh by joining two meshes. Make sure the meshes touch but do not overlap. The function takes offset values as input which allow you to "shift" the second mesh to align it to the first. The command takes 5 arguments:</p> <ol style="list-style-type: none"> 1. <code>x_off</code>, x-offset in coordinate system of the second mesh 2. <code>y_off</code>, y-offset in coordinate system of the second mesh 3. <code>mesh1-name</code>, Name of the first mesh 4. <code>mesh2-name</code>, Name of the second mesh 5. <code>mesh3-name</code>, Name of the resulting mesh
<code>joinmesh_h</code>	<p>Create new mesh by joining two meshes. Make sure the meshes touch but do not overlap. The function takes a y-offset value as input which allow you to "shift" the second mesh in the y-direction to align it to the first. The x-offset is computed such that the meshes touch but do not overlap (if possible with the given y-offset). The command takes 4 arguments:</p> <ol style="list-style-type: none"> 1. <code>y_off</code>, y-offset in coordinate system of the second mesh 2. <code>mesh1-name</code>, Name of the first mesh 3. <code>mesh2-name</code>, Name of the second mesh 4. <code>mesh3-name</code>, Name of the resulting mesh
<code>joinmesh_v</code>	<p>Create new mesh by joining two meshes. Make sure the meshes touch but do not overlap. The function takes an x-offset value as input which allow you to "shift" the second mesh in the x-direction to align it to the first. The y-offset is computed such that the meshes touch but do not overlap (if possible with the given x-offset). The command takes 4 arguments:</p> <ol style="list-style-type: none"> 1. <code>x_off</code>, x-offset in coordinate system of the second mesh 2. <code>mesh1-name</code>, Name of the first mesh 3. <code>mesh2-name</code>, Name of the second mesh 4. <code>mesh3-name</code>, Name of the resulting mesh
<code>dupmesh</code>	<p>Duplicate a mesh. The command takes 2 arguments:</p> <ol style="list-style-type: none"> 1. <code>mesh1-name</code>, Name of the mesh to be duplicated 2. <code>mesh2-name</code>, Name of the resulting copy
<code>cleanupmesh</code>	<p>Remove unused areas. The command takes 1 argument:</p> <ol style="list-style-type: none"> 1. <code>mesh-name</code>, Name of the mesh
<code>add.electrode</code>	<p>Adds an electrode to a certain mesh. The command takes one argument:</p> <ol style="list-style-type: none"> 1. <code>mesh-name</code>, Name of the mesh

TRANSFORMING MESHES

Keyword	Description
---------	-------------

<code>mesh.scalemove</code>	<p>Scale and translate the x- and y- coordinates in a mesh. The coordinates are scaled before translating. The command takes five arguments:</p> <ol style="list-style-type: none"> 1. <code>mesh-name</code>, Name of the mesh 2. <code>fx</code>, scaling in x-direction 3. <code>fy</code>, scaling in y-direction 4. <code>dx</code>, move in x-direction 5. <code>dy</code>, move in y-direction
<code>mesh.rotate</code>	<p>Rotate mesh around a center coordinate. Rotation can only be performed over multiples of 90 degrees. The command takes four arguments:</p> <ol style="list-style-type: none"> 1. <code>mesh-name</code>, Name of the mesh 2. <code>x</code>, x-coordinate of rotation center 3. <code>y</code>, y-coordinate of rotation center 4. <code>d</code>, degrees to rotate over (must be a multiple of 90)
<code>mesh.flipx</code>	<p>Flip mesh in x direction, i.e. mirror in the central x-coordinate. The command takes one argument:</p> <ol style="list-style-type: none"> 1. <code>mesh-name</code>, Name of the mesh
<code>mesh.flipy</code>	<p>Flip mesh in y direction, i.e. mirror in the central y-coordinate. The command takes one argument:</p> <ol style="list-style-type: none"> 1. <code>mesh-name</code>, Name of the mesh
<code>mesh.boundingBox</code>	<p>Change the mesh bounding box. This argument scales and moves a mesh such that it fits in a specified rectangle (the bounding box). The mesh is either scaled preserving the aspect ratio or without preserving it. The command will set the variables “Fx”, “Fy”, “Dx”, and “Dy” which allow you to perform the exact same transformation on other objects using the <code>mesh.scalemove</code> and the <code>poly.scalemove</code> commands. The command takes six arguments:</p> <ol style="list-style-type: none"> 1. <code>mesh-name</code>, Name of the mesh 2. <code>x1</code>, x-coordinate of the lower left corner of the bounding box 3. <code>y1</code>, y-coordinate of the lower left corner of the bounding box 4. <code>x2</code>, x-coordinate of the upper right corner of the bounding box 5. <code>y2</code>, y-coordinate of the upper right corner of the bounding box 6. <code>R</code>, Argument can be either 0, do not preserve mesh aspect ratio, or 1, preserve mesh aspect ratio.
<code>poly.scalemove</code>	<p>Scale and translate the x- and y- coordinates in the currently loaded polygon. The coordinates are scaled before translating. The command takes four arguments:</p> <ol style="list-style-type: none"> 1. <code>fx</code>, scaling in x-direction 2. <code>fy</code>, scaling in y-direction 3. <code>dx</code>, move in x-direction 4. <code>dy</code>, move in y-direction
<code>poly.rotate</code>	<p>Rotate the currently loaded polygon around a center coordinate. Rotation can only be performed over multiples of 90 degrees. The command takes three arguments:</p> <ol style="list-style-type: none"> 1. <code>x</code>, x-coordinate of rotation center 2. <code>y</code>, y-coordinate of rotation center 3. <code>d</code>, degrees to rotate over (must be a multiple of 90)
<code>poly.flipx</code>	<p>Flip the currently loaded polygon in x direction, i.e. mirror in the central x-coordinate. The command takes no arguments.</p>
<code>poly.flipy</code>	<p>Flip the currently loaded polygon in y direction, i.e. mirror in the central y-coordinate. The command takes no arguments.</p>

poly_boundingbox Change the bounding box of the currently loaded polygon. This argument scales and moves a polygon such that it fits in a specified rectangle (the bounding box). The polygon is either scaled preserving the aspect ratio or without preserving it. The command will set the variables “Fx”, “Fy”, “Dx”, and “Dy” which allow you to perform the exact same transformation on other objects using the **mesh_scalemove** and the **poly_scalemove** commands. The command takes five arguments:

1. **x1**, x-coordinate of the lower left corner of the bounding box
2. **y1**, y-coordinate of the lower left corner of the bounding box
3. **x2**, x-coordinate of the upper right corner of the bounding box
4. **y2**, y-coordinate of the upper right corner of the bounding box
5. **R**, Argument can be either 0, do not preserve mesh aspect ratio, or 1, preserve mesh aspect ratio.

SELECTING ELEMENTS

Keyword	Description
select_rect	<p>Select a rectangular area in a mesh. If currently nodes are already selected in the mesh, the command selects the subset of selected nodes within the rectangle. If the latter is not desired use deselect first. All select commands set the variable “Nsel” to the number of selected elements. The command takes five arguments:</p> <ol style="list-style-type: none"> 1. x1, x-coordinate of the lower left corner of the selected rectangle 2. y1, y-coordinate of the lower left corner of the selected rectangle 3. x2, x-coordinate of the upper right corner of the selected rectangle 4. y2, y-coordinate of the upper right corner of the selected rectangle 5. mesh-name, Name of the mesh
select_rect_contour	<p>Select the contour of a rectangle in a mesh. If currently nodes are already selected in the mesh, the command selects the subset of selected nodes within the rectangle. If the latter is not desired use deselect first. All select commands set the variable “Nsel” to the number of selected elements. The command takes six arguments:</p> <ol style="list-style-type: none"> 1. x1, x-coordinate of the lower left corner of the selected rectangle 2. y1, y-coordinate of the lower left corner of the selected rectangle 3. x2, x-coordinate of the upper right corner of the selected rectangle 4. y2, y-coordinate of the upper right corner of the selected rectangle 5. d, distance from the contour within which elements are selected 6. mesh-name, Name of the mesh
select_circ	<p>Select a circular area in a mesh. If currently nodes are already selected in the mesh, the command selects the subset of selected nodes within the circle. If the latter is not desired use deselect first. All select commands set the variable “Nsel” to the number of selected elements. The command takes four arguments:</p> <ol style="list-style-type: none"> 1. x_c, center x-coordinate of the selected circle 2. y_c, center y-coordinate of the selected circle 3. r, radius of the selected circle 4. mesh-name, Name of the mesh
select_circ_contour	<p>Select the contour of a circle in a mesh. If currently nodes are already selected in the mesh, the command selects the subset of selected nodes within the circle. If the latter is not desired use deselect first. All select commands set the variable “Nsel” to the number of selected elements. The command takes five arguments:</p> <ol style="list-style-type: none"> 1. x_c, center x-coordinate of the selected circle 2. y_c, center y-coordinate of the selected circle 3. r, radius of the selected circle 4. d, distance from the contour within which elements are selected 5. mesh-name, Name of the mesh

select_poly	<p>Select an area within a polygon-contour. In order to use this command you must first load or define a polygon from file with the load_poly or define_poly commands. If currently elements are already selected in the mesh, the command selects the subset of selected elements within the polygon. If the latter is not desired use deselect first. All select commands set the variable "Nsel" to the number of selected elements. If a polygon circles an element more than once the node is selected if the node is circled an uneven number of times. This allows one to define polygon areas with holes. To make a hole, define one outer part of the polygon and one inner part and have a break between these two parts, i.e. the two parts should not be connected by a line segment, (see load_poly and loaddefine_poly). The command takes one argument.</p> <ol style="list-style-type: none"> 1. mesh-name, Name of the mesh
select_poly_contour	<p>Select an area around a polygon-contour. In order to use this command you must first load or define a polygon from file with the load_poly or define_poly commands. If currently elements are already selected in the mesh, the command selects the subset of selected elements near to the polygon. If the latter is not desired use deselect first. All select commands set the variable "Nsel" to the number of selected elements. This command is often useful to refine the mesh along the polygon before using select_poly to assign elements to a new area. It may also be useful to define things such as cracks. If no line segment is present between two subsequent coordinates in the polygon (i.e. a break in the polygon), no nodes are selected between these coordinates. The command takes three arguments.</p> <ol style="list-style-type: none"> 1. distance, Distance from the polygon 2. loop, Argument takes either 0 (not looped) or 1 (looped). In looped modus the last point in the polygon is connected to the first point. 3. mesh-name, Name of the mesh
load_poly	<p>Load a polygon from file. This command is used in conjunction with the select_poly and the select_poly_contour commands. Once a polygon is loaded you can use it to select until a new load_poly command is given. Polygons may contain breaks (i.e. absence of a line segment between two coordinates). The command takes one argument.</p> <ol style="list-style-type: none"> 1. file-name, Name of the file describing the polygon (one 2D coordinate per line, i.e., two columns, 1: x-coordinate, 2: y-coordinate, empty lines indicate a break)
define_poly	<p>Define polygon within the input file. See also the load_poly command for an alternative method. The difference between this command and the load_poly command is that with this command you can define the polygon within the PVMOS input file. As such it is best suited for simple polygons. The define_poly command marks the start and the end of a table defining the polygon. Between the two define_poly commands, one coordinate (x- and y-value) per line is expected and an empty line indicates a break, i.e.,</p> <pre> define_poly x1 y1 x2 y2 x3 y3 xN yN xN+1 yN+1 xM yM define_poly </pre> <p>where (x_i, y_i) is the i-th coordinate of the polygon and between coordinates N and $N + 1$ there is a break.</p>
select_area	<p>Select all nodes assigned to a given area. If currently nodes are already selected in the mesh, the command selects the subset of selected nodes which are assigned to the given area. If the latter is not desired use deselect first. The command takes one argument.</p> <ol style="list-style-type: none"> 1. area-name, Name of the mesh and area (<mesh-name>.<area-name>)
invertselect	<p>Inverts a selection within a mesh, i.e. select all unselected elements. The command takes one argument.</p> <ol style="list-style-type: none"> 1. mesh-name, Name of the mesh

deselect deselects a selection within a mesh. The command takes one argument.

1. **mesh-name**, Name of the mesh

delete_elements deletes all selected elements from a mesh. The command takes one argument.

1. **mesh-name**, Name of the mesh

MANUALLY CHANGING THE MESH TOPOLOGY

Keyword	Description
split_x	Split selected elements in x-direction. If no elements are selected, all elements are split. As the topology of the mesh changed all selected nodes in the mesh are un-selected after this command. The command takes one argument <ol style="list-style-type: none"> 1. mesh-name, Name of the mesh
split_y	Split selected elements in y-direction. If no elements are selected, all elements are split. As the topology of the mesh changed all selected nodes in the mesh are un-selected after this command. The command takes one argument <ol style="list-style-type: none"> 1. mesh-name, Name of the mesh
split_xy	Split selected elements in both x- and y-direction. If no elements are selected, all elements are split. As the topology of the mesh changed all selected nodes in the mesh are un-selected after this command. The command takes one argument <ol style="list-style-type: none"> 1. mesh-name, Name of the mesh
split_long	Split selected elements in their longest direction. If no elements are selected, all elements are split. As the topology of the mesh changed all selected nodes in the mesh are un-selected after this command. The command takes one argument <ol style="list-style-type: none"> 1. mesh-name, Name of the mesh
split_coarse	Split selected elements until the node-edges are all smaller than a given length. If no elements are selected, all elements are split. As the topology of the mesh changed all selected nodes in the mesh are un-selected after this command. The command takes three arguments <ol style="list-style-type: none"> 1. mesh-name, Name of the mesh 2. lx, Maximum edge length in x-direction 3. ly, Maximum edge length in y-direction
resolve_rect	Split elements along the edges of a rectangle until all element edges are smaller than a given length. This command can be used to ensure a particular rectangle fits accurately in the mesh. The command takes six arguments <ol style="list-style-type: none"> 1. mesh-name, Name of the mesh 2. l, Maximum edge length 3. x1, x-coordinate of the lower left corner of the selected rectangle 4. y1, y-coordinate of the lower left corner of the selected rectangle 5. x2, x-coordinate of the upper right corner of the selected rectangle 6. y2, y-coordinate of the upper right corner of the selected rectangle
resolve_circ	Split elements along the circumference of a circle until all element edges are smaller than a given length. This command can be used to ensure a particular circle fits accurately in the mesh. The command takes five arguments <ol style="list-style-type: none"> 1. mesh-name, Name of the mesh 2. l, Maximum edge length 3. x_c, center x-coordinate of the selected circle 4. y_c, center y-coordinate of the selected circle 5. r, radius of the selected circle

resolve_poly Split elements along the contour of a polygon until all element edges are smaller than a given length. This command can be used to ensure a particular polygon fits accurately in the mesh. The command requires a polygon to be defined with either **define_poly** or **load_poly**. Breaks in the polygon are not resolved, i.e. the routine only resolves the line segments within the polygon. The command takes three arguments

1. **mesh-name**, Name of the mesh
2. 1, Maximum edge length
3. **c**, Loop the polygon or not (1, loop, i.e. resolve the line segment between the start and end coordinates, 0 do not loop)

simplify Attempt to simplify a mesh. If elements are selected they are un-selected as the topology of the mesh changed. The command takes one argument

1. **mesh-name**, Name of the mesh

SAVING AND LOADING MESHES

Keyword	Description
savemesh	Save a mesh to file in the PVMOS binary format, so it can be loaded again at a later time (see the loadmesh command). Compatibility of the saved meshes with past or future versions of PVMOS is <i>not</i> guaranteed. The command takes two arguments. <ol style="list-style-type: none"> 1. mesh-name, Name of the mesh to be saved. 2. file-name, file name to save the mesh to.
loadmesh	Load a mesh saved to file in the PVMOS binary format (see the savemesh command). The command takes two arguments. <ol style="list-style-type: none"> 1. file-name, file name of the file containing the mesh data. 2. mesh-name, Name to assign to the loaded mesh

EXPORTING DATA

Keyword	Description
save_vars	Saves all user defined variables to a file. The command takes one argument: <ol style="list-style-type: none"> 1. file-name, file-name to save the data in.
printmesh	Export the mesh in a manner that is compatible with (probably only) Gnuplot (www.gnuplot.info/). The resulting plot draws the contour of each element in the mesh. If a selection of elements is made for the mesh, only the selected nodes are plotted. The command takes two arguments: <ol style="list-style-type: none"> 1. mesh-name, Name of the mesh 2. file-name, file-name to save the data in. <p>The output file will contain coordinates in columns. For each element the file contains the coordinates of the lower left- and the upper right corners empty line:</p> <pre> x1 y1 x2 y1 x2 y2 x1 y2 <empty line></pre>
printconn	Print lateral connections in the electrodes in a format compatible with Gnuplot (www.gnuplot.info/). When plotting the file (with vectors) a vector is drawn between the center of each element to the center of the adjacent elements to which it is connected. If a selection of elements is made for the mesh, only the selected nodes are plotted. This routine may be useful when inspecting generated meshes. The command takes two arguments: <ol style="list-style-type: none"> 1. mesh-name, Name of the mesh 2. file-name, filename to save the data in. <p>The output is coordinates in columns. For each element the file contains the following data where xc, yc is the center of the current element and xca_i, yca_i is the center coordinate of the i-th adjacent</p> <pre> xc yc xca_1 yca_1 element: xc yc xca_2 yca_2 ...</pre>

printarea

Print the geometry of the mesh which identifies each element and the area it belongs to. The file-format is laid out such that it is compatible with a surface plot in Gnuplot (www.gnuplot.info/). If a selection of elements is made for the mesh, only the selected nodes are plotted. The command takes two arguments:

1. **mesh-name**, Name of the mesh
2. **file-name**, file-name to save the data in.

The output file will contain data in columns. The file contains coordinates, the element ID and the corresponding area ID. Note that the parameters for each area can be exported with the **printpars** command. For each element it plots 2 times 2 data lines with an empty line in-between. Between the data of two elements are two empty lines. This file is formatted such that when plotted with "splot" in gnuplot you can plot a surface for each element in the mesh, which allows you to see the areas in the defined geometry. For each element the following data is printed to the file:

```
x1    y1    element-ID area-ID
x1    y2    element-ID area-ID
<empty line>
x2    y2    element-ID area-ID
x2    y1    element-ID area-ID
<empty line>
<empty line>
```

printV

Print the electrode potentials per element for each stored solution. The output is formatted for gnuplot's splot command, such that a surface plot plots each element individually. If a selection of elements is made for the mesh, only the selected nodes are plotted. The command takes two arguments:

1. **mesh-name**, Name of the mesh
2. **file-name**, file-name to save the data in.

The output file will contain data in columns. The file contains coordinates followed by the potential in each electrode for each solution. For each element it plots 2 times 2 data lines with an empty line in-between. Between the data of two elements are two empty lines. This file is formatted such that when plotted with "splot" in gnuplot you can plot a surface for each electrode in each element in the mesh. For each element the following data is printed to the file, where the subscripts indicate the electrode index and the superscript the solution index:

```
x1    y1    V01 V11 V...1 VN1 V02 V12 V...2 VN2 ...
x1    y2    V01 V11 V...1 VN1 V02 V12 V...2 VN2 ...
<empty line>
x2    y2    V01 V11 V...1 VN1 V02 V12 V...2 VN2 ...
x2    y1    V01 V11 V...1 VN1 V02 V12 V...2 VN2 ...
<empty line>
<empty line>
```

printpar

Print a summary of the parameters per area, including both area-name and area-ID. The command takes two arguments:

1. **mesh-name**, Name of the mesh
2. **file-name**, file-name to save the data in.

print_solpar

Export the solar cell parameters. Exports a file with I_{sc} , V_{oc} , I_{mpp} , V_{mpp} , FF and P_{max} . The command takes two arguments:

1. **mesh-name**, Name of the mesh
2. **file-name**, file-name to save the data in.

printIV

Export the IV characteristics of the device. Exports a file with two columns, the first contains the applied voltage and the second the corresponding simulated total current. The command takes two arguments:

1. **mesh-name**, Name of the mesh
2. **file-name**, file-name to save the data in.

printInIp	<p>Integrate external currents over selected elements. Exports a file with four columns, the applied voltage, the integrated current injected from the positive node, the integrated current emitted to the negative node, and the total current over the whole device. This may be used for sanity checking of the solution or, if your device has more than one connection to the applied bias, to distinguish between the connections (e.g. extract the current injected through one particular bus-bar). The command takes two arguments:</p> <ol style="list-style-type: none"> 1. mesh-name, Name of the mesh 2. file-name, file-name to save the data in.
printlocalJV	<p>Print a JV characteristics for one particular location and one inter-electrode index. This is useful to check the PVMOS diode model. The command takes 8 parameters:</p> <ol style="list-style-type: none"> 1. mesh-name, Name of the mesh 2. x, x-coordinate 3. y, y-coordinate 4. i, inter-electrode index 5. Vstart, Start voltage for the JV characteristics 6. Vend, End voltage for the JV characteristics 7. Nstep, Number of voltage steps in the JV characteristics 8. file-name, file-name to save the data in.
print_probe	<p>Print local electrode voltages. This command prints for one specific location the local voltage for every available simulation in every electrode. The command takes 4 parameters:</p> <ol style="list-style-type: none"> 1. mesh-name, Name of the mesh 2. x, x-coordinate 3. y, y-coordinate 4. file-name, file-name to save the data in.
surfVplot	<p>Export the electrode voltages for a specific solution. Unlike the print-commands like printV the data is interpolated and mapped on a regular mesh. The command takes eight arguments:</p> <ol style="list-style-type: none"> 1. mesh-name, Name of the mesh 2. x1, x-coordinate of the lower left corner of the selected rectangle 3. y1, y-coordinate of the lower left corner of the selected rectangle 4. x2, x-coordinate of the upper right corner of the selected rectangle 5. y2, y-coordinate of the upper right corner of the selected rectangle 6. Nx, Number of points in the regular mesh along the x-direction 7. Ny, Number of points in the regular mesh along the y-direction 8. Va, Applied voltage (if the specified voltage is not available the closest value will be taken) 9. file-name, file-name to save the data in.

surfVjplot

Export the junction voltages for a specific solution. Just like in the **surfVplot** command the data is interpolated and mapped on a regular mesh. The junction voltage is defined for the one and two diode models. For other models the junction voltage may not be available, in which case it is set to 0. The command takes eight arguments:

1. **mesh-name**, Name of the mesh
2. **x1**, x-coordinate of the lower left corner of the selected rectangle
3. **y1**, y-coordinate of the lower left corner of the selected rectangle
4. **x2**, x-coordinate of the upper right corner of the selected rectangle
5. **y2**, y-coordinate of the upper right corner of the selected rectangle
6. **Nx**, Number of points in the regular mesh along the x-direction
7. **Ny**, Number of points in the regular mesh along the y-direction
8. **Va**, Applied voltage (if the specified voltage is not available the closest value will be taken
9. **file-name**, file-name to save the data in.

surfPplot

Export the local power density for a specific solution. Just like in the **surfVplot** command the data is interpolated and mapped on a regular mesh. The command takes eight arguments:

1. **mesh-name**, Name of the mesh
2. **x1**, x-coordinate of the lower left corner of the selected rectangle
3. **y1**, y-coordinate of the lower left corner of the selected rectangle
4. **x2**, x-coordinate of the upper right corner of the selected rectangle
5. **y2**, y-coordinate of the upper right corner of the selected rectangle
6. **Nx**, Number of points in the regular mesh along the x-direction
7. **Ny**, Number of points in the regular mesh along the y-direction
8. **Va**, Applied voltage (if the specified voltage is not available the closest value will be taken
9. **file-name**, file-name to save the data in.

surfJplot

Export the current densities in the electrodes and through the solar cells. Unlike the **print**-commands like **printV** the data is mapped on a regular mesh. The command takes eight arguments:

1. **mesh-name**, Name of the mesh
2. **x1**, x-coordinate of the lower left corner of the selected rectangle
3. **y1**, y-coordinate of the lower left corner of the selected rectangle
4. **x2**, x-coordinate of the upper right corner of the selected rectangle
5. **y2**, y-coordinate of the upper right corner of the selected rectangle
6. **Nx**, Number of points in the regular mesh along the x-direction
7. **Ny**, Number of points in the regular mesh along the y-direction
8. **Va**, Applied voltage (if the specified voltage is not available the closest value will be taken
9. **file-name**, file-name to save the data in.

surfEplot

Export the local electric field in x- and y- direction in the electrodes. Just like in the **surfVplot** command the data is mapped on a regular mesh. The command takes eight arguments:

1. **mesh-name**, Name of the mesh
2. **x1**, x-coordinate of the lower left corner of the selected rectangle
3. **y1**, y-coordinate of the lower left corner of the selected rectangle
4. **x2**, x-coordinate of the upper right corner of the selected rectangle
5. **y2**, y-coordinate of the upper right corner of the selected rectangle
6. **Nx**, Number of points in the regular mesh along the x-direction
7. **Ny**, Number of points in the regular mesh along the y-direction
8. **Va**, Applied voltage (if the specified voltage is not available the closest value will be taken
9. **file-name**, file-name to save the data in.

surfFc

Export the locally collected photo-current. The locally collected current is the portion of the local photo-current which is collected at the terminals. Just like in the **surfVplot** command the data is mapped on a regular mesh. The command sets the short circuit current density to 0 for specific points and computes the difference in terminal current. Note that the full non-linear response of the system is considered (you ask why?, why not!). The command takes 11 arguments:

1. **mesh-name**, Name of the mesh
2. **file-name**, file-name to save the data in.
3. **connection-index**, Index of the connection for which the photo-current collection is computed. The first connection, between electrode 0 and 1, has index 0
4. **Va**, applied voltage
5. **x1**, x-coordinate of the lower left corner of the selected rectangle
6. **y1**, y-coordinate of the lower left corner of the selected rectangle
7. **x2**, x-coordinate of the upper right corner of the selected rectangle
8. **y2**, y-coordinate of the upper right corner of the selected rectangle
9. **Nx**, Number of points in the regular mesh along the x-direction
10. **Ny**, Number of points in the regular mesh along the y-direction
11. **RI**, Boolean, if true the internal series resistance is considered (i.e., consider bias dependent collection of the diode due to an internal series resistance)

surffc

Export the differential local photo-current collection efficiency. The differential photo-current collection efficiency is the portion of a change in photo-current that is collected at the terminals. This is a differential value, i.e. it computes the change in terminal current for an infinitesimal small change in photo-current. Just like in the **surfVplot** command the data is mapped on a regular mesh. The command takes 11 arguments:

1. **mesh-name**, Name of the mesh
2. **file-name**, file-name to save the data in.
3. **connection-index**, Index of the connection for which the photo-current collection is computed. The first connection, between electrode 0 and 1, has index 0
4. **Va**, applied voltage
5. **x1**, x-coordinate of the lower left corner of the selected rectangle
6. **y1**, y-coordinate of the lower left corner of the selected rectangle
7. **x2**, x-coordinate of the upper right corner of the selected rectangle
8. **y2**, y-coordinate of the upper right corner of the selected rectangle
9. **Nx**, Number of points in the regular mesh along the x-direction
10. **Ny**, Number of points in the regular mesh along the y-direction
11. **RI**, Boolean, if true the internal series resistance is considered (i.e., consider bias dependent collection of the diode due to an internal series resistance)

surfAreaplot

Export the area definition. Unlike the **print**-commands like **printdef** the data is mapped to a regular mesh. The command takes seven arguments:

1. **mesh-name**, Name of the mesh
2. **x1**, x-coordinate of the lower left corner of the selected rectangle
3. **y1**, y-coordinate of the lower left corner of the selected rectangle
4. **x2**, x-coordinate of the upper right corner of the selected rectangle
5. **y2**, y-coordinate of the upper right corner of the selected rectangle
6. **Nx**, Number of points in the regular mesh along the x-direction
7. **Ny**, Number of points in the regular mesh along the y-direction
8. **file-name**, file-name to save the data in.

MANAGING SIMULATION RESULTS

purge_results	Delete all simulation results in a mesh. The command takes one argument: <ol style="list-style-type: none"> 1. mesh-name, Name of the mesh
purge_result.i	Delete the simulation results at a specified simulation index. The command takes two arguments: <ol style="list-style-type: none"> 1. mesh-name, Name of the mesh 2. simulation_index, Index of the simulation

MANIPULATING LOCAL PROPERTIES

Keyword	Description
assign_properties	Assign nodes to a defined area. If no nodes are selected all nodes in the mesh are assigned to the specified area. The command takes one arguments: <ol style="list-style-type: none"> 1. area-name, Name of the area (<mesh-name>.<area-name>)
set_Rel	Set an electrode resistance per area. If the specified area does not exist it will be newly created. The command takes three arguments: <ol style="list-style-type: none"> 1. area-name, Name of the area (<mesh-name>.<area-name>) 2. electrode-index, Index of the electrode. The first electrode has index 0 3. value, Sheet resistance value (Ω)
set_Rvp	Set the contact resistance between the positive node and an electrode per area. If the specified area does not exist it will be newly created. Together with set_Rvn this command allows the application of an extranal voltage. The command takes three arguments: <ol style="list-style-type: none"> 1. area-name, Name of the area (<mesh-name>.<area-name>) 2. electrode-index, Index of the electrode. The first electrode has index 0 3. value, Contact resistance (Ωcm^2)
set_Rvn	Set the contact resistance between the negative node and an electrode per area. If the specified area does not exist it will be newly created. Together with set_Rvp this command allows the application of an extranal voltage. The command takes three arguments: <ol style="list-style-type: none"> 1. area-name, Name of the area (<mesh-name>.<area-name>) 2. electrode-index, Index of the electrode. The first electrode has index 0 3. value, Contact resistance (Ωcm^2)
set_Rvg	Set the resistance between electrode and the local ground level. If the specified area does not exist it will be newly created. This command is used in conjunction with the set_Vg and set_sel_Vg commands to simulate leakage currents to ground Note that the ground level is fixed, i.e. independent of the applied bias voltage. The command takes three arguments: <ol style="list-style-type: none"> 1. area-name, Name of the area (<mesh-name>.<area-name>) 2. electrode-index, Index of the electrode. The first electrode has index 0 3. value, Resistance to ground (Ωcm^2)
set_JV	Specify a tabular data set to use as a JV characteristics per area. If the specified area does not exist it will be newly created. The command takes three arguments: <ol style="list-style-type: none"> 1. area-name, Name of the area (<mesh-name>.<area-name>) 2. connection-index, Index of the connection. The first connection, between electrode 0 and 1, has index 0 3. file-name, Name of a file containing two columns, voltage and current density (V, Acm^{-2})

set_2DJV

Specify a two-diode model for the JV characteristics per area. If the specified area does not exist it will be newly created. The command takes eight arguments:

1. **area-name**, Name of the area (<mesh-name>.<area-name>)
2. **connection-index**, Index of the connection. The first connection, between electrode 0 and 1, has index 0
3. **J01**, Saturation current density for the first diode (ideality factor one) (Acm^{-2})
4. **J02**, Saturation current density for the second diode (ideality factor two) (Acm^{-2})
5. **Jph**, Photo current density (Acm^{-2})
6. **Rs**, Series resistance (Ωcm^2)
7. **Rsh**, Shunt resistance (Ωcm^2)
8. **Eg**, Band gap (eV)

set_1DJV

Specify a one-diode model for the JV characteristics per area. If the specified area does not exist it will be newly created. The command takes eight arguments:

1. **area-name**, Name of the area (<mesh-name>.<area-name>)
2. **connection-index**, Index of the connection. The first connection, between electrode 0 and 1, has index 0
3. **J0**, Saturation current density (Acm^{-2})
4. **nid**, Ideality factor
5. **Jph**, Photo current density (Acm^{-2})
6. **Rs**, Series resistance (Ωcm^2)
7. **Rsh**, Shunt resistance (Ωcm^2)
8. **Eg**, Band gap (eV)

set_PTJV

Specify the photo-transistor model for the JV characteristics per area. See reference [3] for details on the photo-transistor model for solar cells. If the specified area does not exist it will be newly created. See the command **set_PTJV_extra** to set extra parameters for the model. The command takes eleven arguments:

1. **area-name**, Name of the area (<mesh-name>.<area-name>)
2. **connection-index**, Index of the connection. The first connection, between electrode 0 and 1, has index 0
3. **Jsbc**, Dark saturation current density of the main BC diode (Acm^{-2})
4. **Jsbe**, Dark saturation current density of the main BE diode (Acm^{-2})
5. **Rs**, Series resistance (Ωcm^2)
6. **Rsh**, Shunt resistance (Ωcm^2)
7. **EgBE**, Band gap of the BE diodes (eV)
8. **PhiBC**, Barrier of the BC junction (eV)
9. **Bf**, Forward operation gain
10. **Vaf**, Forward operation Early voltage (V)

set_PTJV_extra	<p>Specify extra parameters for the photo-transistor model for the JV characteristics per area. See reference [3] for details on the photo-transistor model for solar cells. To use this command you must first set the photo-transistor model with the command set_PTJV. The command takes ten arguments:</p> <ol style="list-style-type: none"> 1. Jse, Dark saturation current of the leakage BE diode (Acm^{-2}) 2. Jsc, Dark saturation current of the leakage BC diode (Acm^{-2}) 3. Nf, Main BE diode ideality factor 4. Nr, Main BC diode ideality factor 5. Ne, Leakage BE diode ideality factor 6. Nc, Leakage BC diode ideality factor 7. Var, Reverse operation Early voltage (V) 8. XTIBE, Temperature dependency of the BE diodes 9. XTIBC, Temperature dependency of the BC diodes 10. XTIB, Temperature dependency of the forward gain
set_R	<p>Specify a resistance for the JV characteristics per area. If the specified area does not exist it will be newly created. The command takes three arguments:</p> <ol style="list-style-type: none"> 1. area-name, Name of the area (<mesh-name>.<area-name>) 2. connection-index, Index of the connection. The first connection, between electrode 0 and 1, has index 0 3. R, Resistance (Ωcm^2)
set_T	<p>Specify a local temperature. The command takes two arguments:</p> <ol style="list-style-type: none"> 1. area-name, Name of the area (<mesh-name>.<area-name>) 2. T, Temperature (K)
set_Vg	<p>Specify a local ground level. In conjunction with the set_Rvg commands this command allows the simulation of leakage currents to ground. The ground level is set per area, allowing spatial variations in ground level. Note that the ground level is fixed, i.e. independent of the applied bias voltage. The command takes two arguments:</p> <ol style="list-style-type: none"> 1. area-name, Name of the area (<mesh-name>.<area-name>) 2. Vg, Ground level potential (V)
set_sel_Rel	<p>Set an electrode resistance per selection, i.e. change a parameter for all elements within a selection. If no elements are selected this command sets a parameter for all areas. If elements are selected this command may create new areas to make sure the change is confined to the selected elements. To do this it appends an area modifier name to existing area names. If the modified area name already exists the modifications are applied to this existing area. The command takes three arguments:</p> <ol style="list-style-type: none"> 1. area-name, Area name modifier (<mesh-name>.<modifier>) 2. electrode-index, Index of the electrode. The first electrode has index 0 3. value, Sheet resistance value (Ω)
set_sel_Rvp	<p>Set the contact resistance between the positive node and an electrode per selection, i.e. change a parameter for all elements within a selection. If no elements are selected this command sets a parameter for all areas. If elements are selected this command may create new areas to make sure the change is confined to the selected elements. To do this it appends an area modifier name to existing area names. If the modified area name already exists the modifications are applied to this existing area. Together with set_Rvn this command allows the application of an extrinsic voltage. The command takes three arguments:</p> <ol style="list-style-type: none"> 1. area-name, Area name modifier (<mesh-name>.<modifier>) 2. electrode-index, Index of the electrode. The first electrode has index 0 3. value, Contact resistance (Ωcm^2)

set_sel_Rvn

Set the contact resistance between the negative node and an electrode per selection, i.e. change a parameter for all elements within a selection. If no elements are selected this command sets a parameter for all areas. If elements are selected this command may create new areas to make sure the change is confined to the selected elements. To do this it appends an area modifier name to existing area names. If the modified area name already exists the modifications are applied to this existing area. Together with **set_Rvp** this command allows the application of an extrinsic voltage. The command takes three arguments:

1. **area-name**, Area name modifier (<mesh-name>.<modifier>)
2. **electrode-index**, Index of the electrode. The first electrode has index 0
3. **value**, Contact resistance (Ωcm^2)

set_sel_JV

Specify a tabular data set to use as a JV characteristics per selection, i.e. change a parameter for all elements within a selection. If no elements are selected this command sets a parameter for all areas. If elements are selected this command may create new areas to make sure the change is confined to the selected elements. To do this it appends an area modifier name to existing area names. If the modified area name already exists the modifications are applied to this existing area. The command takes three arguments:

1. **area-name**, Area name modifier (<mesh-name>.<modifier>)
2. **connection-index**, Index of the connection. The first connection, between electrode 0 and 1, has index 0
3. **file-name**, Name of a file containing two columns, voltage and current density (V , Acm^{-2})

set_sel_2DJV

Specify a two-diode model for the JV characteristics per selection, i.e. change a parameter for all elements within a selection. If no elements are selected this command sets a parameter for all areas. If elements are selected this command may create new areas to make sure the change is confined to the selected elements. To do this it appends an area modifier name to existing area names. If the modified area name already exists the modifications are applied to this existing area. The command takes eight arguments:

1. **area-name**, Area name modifier (<mesh-name>.<modifier>)
2. **connection-index**, Index of the connection. The first connection, between electrode 0 and 1, has index 0
3. **J01**, Saturation current density for the first diode (ideality factor one) (Acm^{-2})
4. **J02**, Saturation current density for the second diode (ideality factor two) (Acm^{-2})
5. **Jph**, Photo current density (Acm^{-2})
6. **Rs**, Series resistance (Ωcm^2)
7. **Rsh**, Shunt resistance (Ωcm^2)
8. **Eg**, Band gap (eV)

set_sel_1DJV

Specify a one-diode model for the JV characteristics per selection, i.e. change a parameter for all elements within a selection. If no elements are selected this command sets a parameter for all areas. If elements are selected this command may create new areas to make sure the change is confined to the selected elements. To do this it appends an area modifier name to existing area names. If the modified area name already exists the modifications are applied to this existing area. The command takes eight arguments:

1. **area-name**, Area name modifier (<mesh-name>.<modifier>)
2. **connection-index**, Index of the connection. The first connection, between electrode 0 and 1, has index 0
3. **J0**, Saturation current density (Acm^{-2})
4. **nid**, Ideality factor
5. **Jph**, Photo current density (Acm^{-2})
6. **Rs**, Series resistance (Ωcm^2)
7. **Rsh**, Shunt resistance (Ωcm^2)
8. **Eg**, Band gap (eV)

set_sel_PTJV

Specify the photo-transistor model (see reference [3] for details on the photo-transistor model for solar cells) for the JV characteristics per selection, i.e. change a parameter for all elements within a selection. If no elements are selected this command sets a parameter for all areas. If elements are selected this command may create new areas to make sure the change is confined to the selected elements. To do this it appends an area modifier name to existing area names. If the modified area name already exists the modifications are applied to this existing area. See the command **set_sel_PTJV_extra** to set extra parameters for the model. The command takes eleven arguments:

1. **area-name**, Area name modifier (<mesh-name>.<modifier>)
2. **connection-index**, Index of the connection. The first connection, between electrode 0 and 1, has index 0
3. **J_{sb}**, Dark saturation current density of the main BC diode (Acm^{-2})
4. **J_{sbe}**, Dark saturation current density of the main BE diode (Acm^{-2})
5. **R_s**, Series resistance (Ωcm^2)
6. **R_{sh}**, Shunt resistance (Ωcm^2)
7. **E_{gBE}**, Band gap of the BE diodes (eV)
8. **Phi_{BC}**, Barrier of the BC junction (eV)
9. **B_f**, Forward operation gain
10. **V_{af}**, Forward operation Early voltage (V)

set_sel_PTJV_extra

Specify extra parameters for the photo-transistor model (see reference [3] for details on the photo-transistor model for solar cells) for the JV characteristics per selection. To use this command you must first set the photo-transistor model with the command **set_sel_PTJV**. The command takes ten arguments:

1. **J_{se}**, Dark saturation current of the leakage BE diode (Acm^{-2})
2. **J_{sc}**, Dark saturation current of the leakage BC diode (Acm^{-2})
3. **N_f**, Main BE diode ideality factor
4. **N_r**, Main BC diode ideality factor
5. **N_e**, Leakage BE diode ideality factor
6. **N_c**, Leakage BC diode ideality factor
7. **V_{ar}**, Reverse operation Early voltage (V)
8. **XTIBE**, Temperature dependency of the BE diodes
9. **XTIBC**, Temperature dependency of the BC diodes
10. **XTIB**, Temperature dependency of the forward gain

set_sel_R

Specify a resistance for the JV characteristics per selection, i.e. change a parameter for all elements within a selection. If no elements are selected this command sets a parameter for all areas. If elements are selected this command may create new areas to make sure the change is confined to the selected elements. To do this it appends an area modifier name to existing area names. If the modified area name already exists the modifications are applied to this existing area. The command takes three arguments:

1. **area-name**, Area name modifier (<mesh-name>.<modifier>)
2. **connection-index**, Index of the connection. The first connection, between electrode 0 and 1, has index 0
3. **R**, Resistance (Ωcm^2)

set_sel_T

Specify a local temperature per selection, i.e. change a parameter for all elements within a selection. If no elements are selected this command sets a parameter for all areas. If elements are selected this command may create new areas to make sure the change is confined to the selected elements. To do this it appends an area modifier name to existing area names. If the modified area name already exists the modifications are applied to this existing area. The command takes two arguments:

1. **area-name**, Area name modifier (<mesh-name>.<modifier>)
2. **T**, Temperature (K)

set_sel.Vg	<p>Specify a local ground level per selection, i.e. change a parameter for all elements within a selection. If no elements are selected this command sets a parameter for all areas. If elements are selected this command may create new areas to make sure the change is confined to the selected elements. To do this it appends an area modifier name to existing area names. If the modified area name already exists the modifications are applied to this existing area. The command takes two arguments:</p> <ol style="list-style-type: none"> 1. area-name, Area name modifier (<mesh-name>.<modifier>) 2. Vg, Ground level potential (V)
-------------------	--

NUMERICAL SETTINGS

Keyword	Description
set.SplitX	<p>It is sometimes useful to prevent the adaptive meshing algorithms from splitting certain nodes in x- or y-direction. This commands toggles the splitting of nodes in x-direction for a specified area (per default all nodes can be split in all directions). The command takes one argument:</p> <ol style="list-style-type: none"> 1. area-name, Name of the area (<mesh-name>.<area-name>)
set.SplitY	<p>It is sometimes useful to prevent the adaptive meshing algorithms from splitting certain nodes in x- or y-direction. This commands toggles the splitting of nodes in y-direction for a specified area (per default all nodes can be split in all directions). The command takes one argument:</p> <ol style="list-style-type: none"> 1. area-name, Name of the area (<mesh-name>.<area-name>)
set_sel.SplitX	<p>It is sometimes useful to prevent the adaptive meshing algorithms from splitting certain nodes in x- or y-direction. This commands toggles the splitting of nodes in x-direction for a selection (per default all nodes can be split in all directions). If no elements are selected this command sets a parameter for all areas. If elements are selected this command may create new areas to make sure the change is confined to the selected elements. To do this it appends an area modifier name to existing area names. If the modified area name already exists the modifications are applied to this existing area. The command takes one argument:</p> <ol style="list-style-type: none"> 1. area-name, Area name modifier (<mesh-name>.<modifier>)
set_sel.SplitY	<p>It is sometimes useful to prevent the adaptive meshing algorithms from splitting certain nodes in x- or y-direction. This commands toggles the splitting of nodes in y-direction for a selection (per default all nodes can be split in all directions). If no elements are selected this command sets a parameter for all areas. If elements are selected this command may create new areas to make sure the change is confined to the selected elements. To do this it appends an area modifier name to existing area names. If the modified area name already exists the modifications are applied to this existing area. The command takes one argument:</p> <ol style="list-style-type: none"> 1. area-name, Area name modifier (<mesh-name>.<modifier>)
maxiter	<p>Set the maximum number of iterations for solving the non-linear system. The command takes one argument:</p> <ol style="list-style-type: none"> 1. maxiter, Maximum number of iterations (default: 25)
tol.V	<p>Absolute voltage tolerance for the break-off criterion. The command takes one argument:</p> <ol style="list-style-type: none"> 1. tol.V, Absolute voltage tolerance V (default: $10^{-5}V$)
rel.tol.V	<p>Relative voltage tolerance for the break-off criterion. The command takes one argument:</p> <ol style="list-style-type: none"> 1. tol.V, Relative voltage tolerance – (default: 10^{-5})
tol.kcl	<p>Absolute current tolerance for the break-off criterion (KCL stands for Kirchhoff's Current Law). The command takes one argument:</p> <ol style="list-style-type: none"> 1. tol.kcl, KCL tolerance A (default: $10^{-5}A$)
rel.tol.kcl	<p>Relative current tolerance for the break-off criterion (KCL stands for Kirchhoff's Current Law). The command takes one argument:</p> <ol style="list-style-type: none"> 1. tol.kcl, Relative KCL tolerance – (default: 10^{-5})

N_Lin_Search	<p>Number of steps for a linear search in the Newton-Raphson direction. If an iterative adaption of the potentials would lead to increase of the error in the Kirchhoff Current Laws, we perform a linear search in the Newton-Raphson direction. If a better solution is not found within the number of steps specified, Gmin stepping is initiated (see the settings GminSteps, GminMax, and GminFac. The command takes one argument:</p> <ol style="list-style-type: none"> 1. N, Number of steps – (default: 10)
GminStep	<p>Number of Gmin stepping steps. In case of convergence problems Gmin stepping is triggered. In Gmin stepping a minimum conductance is placed in parallel with all non-linear devices. The value of the minimum conductance is then step wise reduced to 0. This parameter is used in conjunction with GminMax and GminFac. The command takes one argument:</p> <ol style="list-style-type: none"> 1. N, Number of steps – (default: 6)
GminMax	<p>Maximum value of the minimum conductance. In case of convergence problems Gmin stepping is triggered. In Gmin stepping a minimum conductance is placed in parallel with all non-linear devices. The value of the minimum conductance is then step wise reduced to 0. This value sets the initial minimum conductance after Gmin stepping is triggered. This parameter is used in conjunction with GminSteps and GminFac. The command takes one argument:</p> <ol style="list-style-type: none"> 1. Gmin, Minimum conductance $\Omega^{-1}\text{cm}^{-2}$ (default: 10^{-2})
GminFac	<p>Factor to step-wise reduce Gmin during Gmin stepping. In case of convergence problems Gmin stepping is triggered. In Gmin stepping a minimum conductance is placed in parallel with all non-linear devices. The value of the minimum conductance is then step wise reduced to 0. This value controls the rate with which the Gmin value is reduced (higher is faster). This parameter is used in conjunction with GminSteps and GminMax. The command takes one argument:</p> <ol style="list-style-type: none"> 1. f, Factor to reduce Gmin – (default: 10)
GminStart	<p>In poorly converging cases it may be beneficial to start with one or more iterations with a minimum conductance. This parameter allows you to specify a number of iterations run with a Gmin value set by GminMax. Note that this procedure is different from Gmin stepping as the Gmin value is not changed during these iterations and abruptly changes from the GminMax value to Gmin after the set number of iterations. A typical setting would be 1.</p> <ol style="list-style-type: none"> 1. GminStart, Number of GminStart iterations – (default: 0)
Gmin	<p>Set the overall Gmin value. This value is <i>always</i> used and should therefore be small as otherwise it affects the results.</p> <ol style="list-style-type: none"> 1. Gmin, Minimum conductance $\Omega^{-1}\text{cm}^{-2}$ (default: 0.0)
SOLVING	
Keyword	Description
solve	<p>Solve the system (do a voltage sweep). The command takes four arguments:</p> <ol style="list-style-type: none"> 1. mesh-name, Name of the mesh 2. V_start, Start voltage 3. V_end, End voltage 4. N_step, Number of voltage steps
refine_oc	<p>Given a voltage sweep is present which includes voltages above and below V_{oc}, this command tries to refine the IV characteristics to include the open circuit point. The command takes four arguments:</p> <ol style="list-style-type: none"> 1. mesh-name, Name of the mesh 2. tol.i, current tolerance 3. tol.v, voltage tolerance 4. Nmax, Maximum number of iterations

refine_mpp	<p>Given a voltage sweep is present of at least three points, which includes voltages above and below V_{mpp}, this command tries to refine the IV characteristics to include the maximum power point. The command takes four arguments:</p> <ol style="list-style-type: none"> 1. mesh-name, Name of the mesh 2. tol_i, current tolerance 3. tol_v, voltage tolerance 4. Nmax, Maximum number of iterations
adaptive_solve	<p>Solve the system and adapt the mesh at one specified voltage. The command takes four arguments:</p> <ol style="list-style-type: none"> 1. mesh-name, Name of the mesh 2. V_a, Applied voltage 3. threshold, Relative threshold for node splitting, a parameter between 0 and 1 that controls how aggressive the mesh is adapted, where lower values lead to a more aggressive mesh adaption (typical values between 0.3 and 0.5). 4. N_step, Number adaptive meshing iterations
CONDITIONALS	
if	<p>Start an if conditional block. The if command takes three arguments</p> <ol style="list-style-type: none"> 1. value 1, a (floating point) number 2. Comparison operator, Comparison operator, one of “<”, “>”, “==”, “!=”, “<=”, or “>= 3. value 2, a (floating point) number
else	Start an else block as part of an if conditional block. The else command takes no arguments
endif	End a if conditional block. The endif command takes no arguments
while	<p>Start a while loop block. The while command takes three arguments</p> <ol style="list-style-type: none"> 1. value 1, a (floating point) number 2. Comparison operator, Comparison operator, one of “<”, “>”, “==”, “!=”, “<=”, or “>= 3. value 2, a (floating point) number
endwhile	End a while loop block. The endwhile command takes no arguments
VERBOSITY LEVELS	
Keyword	Description
out_quiet	Set verbosity to the minimum (only says something when it crashes). The command takes no arguments.
out_normal	Set verbosity to the normal level. The command takes no arguments.
out_verbose	Output additional data that may be interesting. The command takes no arguments.
out_debug	Output additional data that is only interesting for someone who is chasing bugs in the code. The command takes no arguments.
MISC.	
Keyword	Description
tic	Start the timer.
toc	Print seconds since last tic command (or start of execution if no tic command was issued)
define	<p>Define a variable. PVMOS variables only support numerical values. These variables can be used wherever a numerical value is required (see Section VIIB). The command takes two arguments:</p> <ol style="list-style-type: none"> 1. name, name of the variable 2. value, value of the variable
ifndef	<p>Conditionally define a variable, defines a variable only if it is not already defined. PVMOS variables only support numerical values. These variables can be used wherever a numerical value is required (see Section VIIB). As it is possible to pass variables to PVMOS at runtime, this way of defining variables allows you to change the value of a variable from its default value to a value defined at runtime. The command takes two arguments:</p> <ol style="list-style-type: none"> 1. name, name of the variable 2. value, value of the variable

B. Variables, Expressions, and Mesh Data Fields

PVMOS also features “expressions”, which are placed between square brackets, i.e., [$\langle \text{expression} \rangle$]. You can refer to variables, as defined with the **define** command (see the previous section), in expressions. Furthermore, PVMOS can evaluate mathematical expressions, provided PVMOS is compiled with libmatheval support. Expressions can be placed anywhere in the PVMOS input file, also as part of a word. Nesting of expressions is also allowed. Perhaps the best way to explain the use of expressions is with a few examples.

Define $N = 10$:

```
define N 10
```

Initialize mesh10:

```
newmesh 0 0 1 1 [N] [N] mesh[N]
```

Define some variables and calculate with them:

```
define Jph 0.03
```

```
define kT 0.0259
```

```
define J0 1e-12
```

```
define Voc [kT*log(Jph/J0+1)]
```

PVMOS with libmatheval supports the following functions:

- **exp** - exponential
- **log** - logarithmic
- **sqrt** - square root
- **sin** - sine
- **cos** - cosine
- **tan** - tangent
- **cot** - cotangent
- **sec** - secant
- **csc** - cosecant
- **asin** - inverse sine
- **acos** - inverse cosine
- **atan** - inverse tangent
- **acot** - inverse cotangent
- **asec** - inverse secant
- **acsc** - inverse cosecant
- **sinh** - hyperbolic sine
- **cosh** - cosine
- **tanh** - hyperbolic tangent
- **coth** - hyperbolic cotangent
- **sech** - hyperbolic secant
- **csch** - hyperbolic cosecant
- **asinh** - hyperbolic inverse sine
- **acosh** - hyperbolic inverse cosine
- **atanh** - hyperbolic inverse tangent

- `acoth` - hyperbolic inverse cotangent
- `asech` - hyperbolic inverse secant
- `acsch` - hyperbolic inverse cosecant
- `abs` - absolute value
- `step` - with value 1 defined for $x = 0$ Heaviside step function
- `nandelta` - Dirac delta function with not-a-number at $x = 0$
- `delta` - Dirac delta function with infinity at $x = 0$
- `erf` - error function

In addition libmatheval defines the following constants:

- `e` - e
- `log2e` - $\log 2(e)$
- `log10e` - $\log 10(e)$
- `ln2` - $\ln(2)$
- `ln10` - $\ln(10)$
- `pi` - π
- `pi_2` - $\pi/2$
- `pi_4` - $\pi/4$
- `1_pi` - $1/\pi$
- `2_pi` - $2/\pi$
- `2_sqrtpi` - $2/\sqrt{\pi}$
- `sqrt` - $\sqrt{2}$
- `sqrt1.2` - $\sqrt{1/2}$

Note that function and constant names cannot be redefined as a variable.

Sometimes it is useful to access data related to a mesh. To this end a mechanism is implemented to access data in a mesh. The syntax for this mechanism is to place a mesh-object name and a request for a certain data field between curly brackets: `{<mesh-name>.<data-field<(arguments-if-any)>>}`. Some data-fields take arguments (comma separated, between brackets, and without spaces). The following data-fields are available:

MESH-OBJECT DATA FIELDS

Data-Field	Description
<code>Nel</code>	Return the number of elements in the mesh <ul style="list-style-type: none"> • Input: None • Return Value: Number of elements (integer)
<code>Nelec</code>	Return the number of electrodes in the mesh <ul style="list-style-type: none"> • Input: None • Return Value: Number of electrodes (integer)
<code>Narea</code>	Return the number of areas in the mesh <ul style="list-style-type: none"> • Input: None • Return Value: Number of areas in the mesh (integer)

<code>Nva</code>	<p>Return the number of simulation results</p> <ul style="list-style-type: none"> • Input: None • Return Value: Number of simulated results (integer)
<code>Nsel</code>	<p>Return the number of selected elements in the mesh</p> <ul style="list-style-type: none"> • Input: None • Return Value: Number of selected elements in the mesh (integer)
<code>area(i_area)</code>	<p>Return the name of the area with index number <code>i_area</code></p> <ul style="list-style-type: none"> • Input: index number <code>i_area</code>, where $0 \leq i_area < Narea$ • Return Value: Name of the area (string), if no area exists with the specified index number the return value is “_AREA_NOT_DEFINED_” (I trust this to be an sufficiently unlikely name of an area).
<code>el(x,y)</code>	<p>Return the element index number at the specified coordinate (x,y)</p> <ul style="list-style-type: none"> • Input: Coordinate (x,y) • Return Value: The element index <code>i_el</code> (integer). Normally the following holds $0 \leq i_el < Nel$. If the coordinate is not in the mesh the return value is -1
<code>X(i_el)</code>	<p>Return the x-coordinate of element <code>i_el</code></p> <ul style="list-style-type: none"> • Input: Element index number <code>i_el</code>, where $0 \leq i_el < Nel$ • Return Value: The x-coordinate of the specified element, or “NaN” if the specified element does not exist
<code>Y(i_el)</code>	<p>Return the y-coordinate of element <code>i_el</code></p> <ul style="list-style-type: none"> • Input: Element index number <code>i_el</code>, where $0 \leq i_el < Nel$ • Return Value: The y-coordinate of the specified element, or “NaN” if the specified element does not exist
<code>V(i_sim)</code>	<p>Return the applied voltage of the simulation with index <code>i_sim</code></p> <ul style="list-style-type: none"> • Input: Simulation index number <code>i_sim</code>, where $0 \leq i_sim < Nva$ • Return Value: The applied voltage of the specified simulation, or “NaN” if the specified simulation does not exist
<code>I(i_sim)</code>	<p>Return the total injected current of the simulation with index <code>i_sim</code></p> <ul style="list-style-type: none"> • Input: Simulation index number <code>i_sim</code>, where $0 \leq i_sim < Nva$ • Return Value: The injected current of the specified simulation, or “NaN” if the specified simulation does not exist
<code>Vi(V)</code>	<p>Return the simulation index with an applied voltage closest to voltage <code>V</code></p> <ul style="list-style-type: none"> • Input: Voltage <code>V</code> • Return Value: The simulation index <code>i_sim</code> (integer). For the simulation index the following holds $0 \leq i_el < Nva$. If no simulation is available the return value is “-1”
<code>VI(I)</code>	<p>Estimate the applied voltage such that the total injected current equals <code>I</code> A.</p> <ul style="list-style-type: none"> • Input: Current <code>I</code> (Ampere) • Return Value: Estimated voltage for which the total current is <code>I</code> or “NaN” if the value cannot be determined.
<code>IV(V)</code>	<p>Estimate the total current at an applied voltage <code>V</code> V.</p> <ul style="list-style-type: none"> • Input: Current <code>V</code> (Volt) • Return Value: Estimated current for an applied volatage <code>V</code> or “NaN” if the value cannot be determined.

<code>Vel(i_el,i_elec,i_sim)</code>	<p>Return the potential in element <code>i_el</code>, electrode <code>i_elec</code>, and for simulation with index <code>i_sim</code>.</p> <ul style="list-style-type: none"> • Input: Element index number <code>i_el</code>, where $0 \leq i_el < N_{el}$, electrode index number <code>i_elec</code>, where $0 \leq i_elec < N_{elec}$, and simulation index number <code>i_sim</code>, where $0 \leq i_sim < N_{va}$ • Return Value: Voltage for the specified element index, electrode index, and simulation index, or NaN if the requested data is not available
<code>VVel(i_el,i_elec,Va)</code>	<p>Estimate the applied voltage such that the potential in element <code>i_el</code> in electrode <code>i_elec</code> equals <code>Va</code> V</p> <ul style="list-style-type: none"> • Input: Element index number <code>i_el</code>, where $0 \leq i_el < N_{el}$, electrode index number <code>i_elec</code>, where $0 \leq i_elec < N_{elec}$, and the potential <code>Va</code> • Return Value: Estimated voltage for which the specified potential is equal to <code>V</code> or “NaN” if the value cannot be determined.
<code>V4Pnt(i_el,i_elec, j_el,j_elec,Va)</code>	<p>Estimate the applied voltage such that the potential difference between element <code>i_el</code>, electrode <code>i_elec</code> and <code>j_el</code>, electrode <code>j_elec</code> equals <code>Va</code> V. Where the potential at point <code>i</code> is positive w.r.t point <code>j</code>, i.e. $V(i) - V(j) = V_a$</p> <ul style="list-style-type: none"> • Input: Element index number <code>i_el</code>, where $0 \leq i_el < N_{el}$, electrode index number <code>i_elec</code>, where $0 \leq i_elec < N_{elec}$, Element index number <code>j_el</code>, where $0 \leq j_el < N_{el}$, electrode index number <code>j_elec</code>, where $0 \leq j_elec < N_{elec}$, and the potential <code>Va</code> • Return Value: Estimated voltage for which the specified potential is equal to <code>V</code> or “NaN” if the value cannot be determined.
<code>BBx1</code>	Return the lower x-coordinate of the mesh bounding box
<code>BBx2</code>	Return the upper x-coordinate of the mesh bounding box
<code>BBy1</code>	Return the lower y-coordinate of the mesh bounding box
<code>BBy2</code>	Return the upper y-coordinate of the mesh bounding box
<code>Impp</code>	<p>Return the current at the maximum power point.</p> <ul style="list-style-type: none"> • Input: None • Return Value: Best estimate of the maximum power point current (Ampere) or 0.0 if the maximum power point cannot be determined.
<code>Vmpp</code>	<p>Return the voltage at the maximum power point.</p> <ul style="list-style-type: none"> • Input: None • Return Value: Best estimate of the maximum power point voltage (Volt) or 0.0 if the maximum power point cannot be determined.
<code>Isc</code>	<p>Return the short circuit current.</p> <ul style="list-style-type: none"> • Input: None • Return Value: Best estimate of the short circuit current (Ampere) or 0.0 if the short circuit point cannot be determined.
<code>Voc</code>	<p>Return the open circuit voltage</p> <ul style="list-style-type: none"> • Input: None • Return Value: Best estimate of the open circuit voltage (Volt) or 0.0 if the open circuit point cannot be determined.
<code>SurfArea(area)</code>	<p>Compute the surface area of the area with name <code>area</code>.</p> <ul style="list-style-type: none"> • Input: Area name <code>area</code> • Return Value: Total surface area of area <code>area</code>. A non-existing area will lead to a surface area of 0.

Note that you can use mesh data fields in expressions and vise versa. In addition it is allowed to nest expressions and mesh data fields.

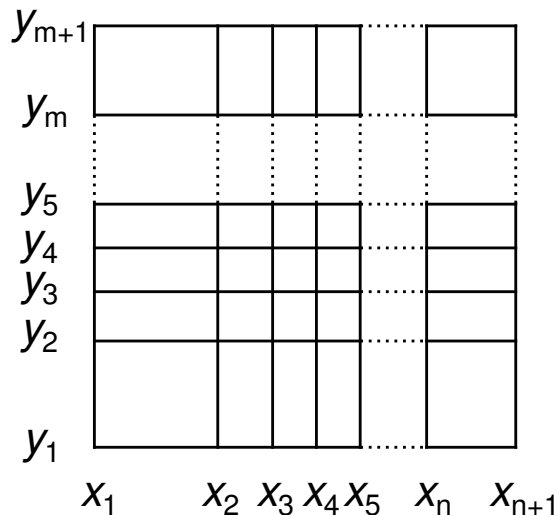


FIG. 1: The mesh topology of meshes created by mkpvmosmesh.

VIII. THE PVMOS-MESH PACKAGE FOR GNU OCTAVE

The GNU Octave package PVMOS-mesh allows you to create and save meshes in binary format for use in PVMOS. GNU Octave is a versatile Matlab-like program. Linking to Octave allows you to, for example, use GNU Octave’s ability to work with bitmap images when creating a mesh for a given problem. In general GNU Octave is very good for creating and manipulating matrices. The PVMOS library allows you to create rectilinear meshes for PVMOS. In rectilinear the element properties can be represented by matrices as the elements are organized in rows and columns. This disallows sub-grids, however, it does allow you to vary the width and height of the rows and columns, i.e. a rudimentary variable mesh can be created.

The core of the library is the “mkpvmosmesh” function, which creates and saves a mesh for PVMOS. There are several utility functions to ease the creation of the data structures used by mkpvmosmesh from, for example, images. We start by describing the input required by the mkpvmosmesh command.

`mkpvmosmesh(Na, Nel, Area_Index, Area_Def, x, y, filename)`

- **Na**, The number of area’s in the mesh
- **Nel**, The number of electrodes
- **Area_Index**, A matrix with index numbers, each number referring to one area
- **Area_Def**, The area definition struct array describing the properties of each area
- **x**, A vector with x coordinates of element boundaries
- **y**, A vector with y coordinates of element boundaries
- **filename**, A string containing the filename for the resulting PVMOS mesh

In Fig. 1 illustrates a possible topology of a mesh created by mkpvmosmesh. The depicted mesh would be $m \times n$ -elements. To specify the widths and heights of the columns and rows, respectively, $n + 1$ x - and $m + 1$ y -coordinates are specified (i.e., the **x** and **y** vectors for the mkpvmosmesh command). For this mesh the **Area_Index** would be an $m \times n$ matrix with integers assigning each element to one area. The **Area_Def** struct-array described the properties of each area. Note that a struct is a data structure containing various fields. In this case the various field describe various properties of an area, e.g. electrode sheet resistances and diode properties. The numbers in the **Area_Index** matrix should correspond to index numbers of the **Area_Def** struct-array.

The fields in the area definition struct are presented in Table III. Within the area definition struct there is a struct array of the connection struct, describing the inter electrode connections. The connection struct fields are listed and described in V

TABLE III: The fields in the area definition struct

Field	Default	Value	Unit	Description
name	"default"	-		String with the area name. Take care that the names should be unique!
Rel	1	$[\Omega]$		Array of length Ne1 with the sheet resistance of each electrode.
Rvp	-1	$[\Omega\text{cm}^2]$		Array of length Ne1 with the resistance to the positive node (external voltage source). If the value is negative the elements will have no direct connection to the external source.
Rvn	-1	$[\Omega\text{cm}^2]$		Array of length Ne1 with the resistance to the ground node (external voltage source). If the value is negative the elements will have no direct connection to the external source.
T	300	$[\text{K}]$		Scalar value for the temperature in the area
SplitX	1	-		Boolean value (0 is false, 1 is true) indicating whether an element can be split in the x -direction during adaptive meshing. If set to 0 the element will not be split.
SplitY	1	-		Boolean value (0 is false, 1 is true) indicating whether an element can be split in the y -direction during adaptive meshing. If set to 0 the element will not be split.
conn	-	-		Structure array of length Ne1-1 with the properties of the connection between the electrodes

TABLE V: The fields in the connection struct

Field	Default	Value	Unit	Description
model	"ONED"	-		String with the model name. Possible models are: ONED One diode model TWOD Two diode model PHOTOT Photo-transistor model JVD Tabular JV data
Jph	0	$[\text{Acm}^{-2}]$		Photo-current density for the models ONED, TWOD, and PHOTOT
Rs	1e-5	$[\Omega\text{cm}^2]$		Series resistance for the models ONED, TWOD, and PHOTOT
Rsh	1e3	$[\Omega\text{cm}^2]$		Shunt resistance for the models ONED, TWOD, and PHOTOT
J01	1e-12	$[\text{Acm}^{-2}]$		Dark saturation current for the first diode, used in models ONED and TWOD
J02	1e-8	$[\text{Acm}^{-2}]$		Dark saturation current for the second diode, used in model TWOD
nid1	1	$[-]$		Ideality factor of the first diode, used in models ONED and TWOD
nid2	2	$[-]$		Ideality factor of the second diode, used in model TWOD
Eg	1.2	$[\text{eV}]$		Band gap of the diode, used in models ONED and TWOD
Jsbc	5e-4	$[\text{Acm}^{-2}]$		Dark saturation current density of the main BC diode
Jsbe	5e-15	$[\text{Acm}^{-2}]$		Dark saturation current density of the main BE diode
Jse	0	$[\text{Acm}^{-2}]$		Dark saturation current of the leakage BE diode
Jsc	0	$[\text{Acm}^{-2}]$		Dark saturation current of the leakage BC diode
Nf	1	$[-]$		Main BE diode ideality factor
Nr	1	$[-]$		Main BC diode ideality factor
Ne	1.5	$[-]$		Leakage BE diode ideality factor
Nc	2	$[-]$		Leakage BC diode ideality factor
Vaf	∞	$[\text{V}]$		Forward operation Early voltage
Var	∞	$[\text{V}]$		Reverse operation Early voltage
Bf	3	$[-]$		Forward operation gain
EgBE	1.2	$[\text{eV}]$		Band gap of the BE diodes
PhiBC	0.3	$[\text{eV}]$		Barrier of the BC junction
XTIBE	3	$[-]$		Temperature dependency of the BE diodes
XTIBC	3	$[-]$		Temperature dependency of the BC diodes
XTIB	3	$[-]$		Temperature dependency of the forward gain

Several commands are provided to ease the use of the `mkpvmosmesh` command. To initialize the area definition struct array (`Area_Def`), the package provides the function:

```
Area_Def=PVMOS_AreaProperties(Nel)
```

It initializes the struct array with one area, using the default values listed in Tables III and V.

I feel it is convenient to work with “selection masks” when defining new areas. A selection mask is essentially a matrix, with the same size as the `Area_Index` matrix, i.e. one matrix element per mesh element, where a 1 selects the element and a 0 not. There are two functions that allow you to use masks to build up the various areas in the `Area_Index` matrix and the `Area_Def` area definition struct array. The functions are:

```
[Area_Index, Area_Def]=PVMOS_SetMask(Area_Index,Area_Def, mask, Mask_Area)
```

- `Area_Index`, A matrix with index numbers, each number referring to one area
- `Area_Def`, The area definition struct array describing the properties of each area
- `mask`, Mesh element selection mask
- `Mask_Area`, Area definition struct to apply to the selected elements (i.e. this area definition will be added to the `Area_Def` area definition struct array)

and

```
[Area_Index, Area_Def]=PVMOS_ModifyMask(Area_Index,Area_Def, mask, modlist, name)
```

- `Area_Index`, A matrix with index numbers, each number referring to one area
- `Area_Def`, The area definition struct array describing the properties of each area
- `mask`, Mesh element selection mask
- `modlist`, Cell array specifying the properties to be changed for the given elements
- `name`, String with the addition to the area names for the selected elements

The difference between the functions is that `PVMOS_SetMask` sets the area for the selected elements and adds one area to the mesh, whereas the `PVMOS_ModifyMask` changes only selected properties for the selected elements, creating as many areas as required. The newly created areas keep the name of the area's they were created from but with the addition of the `name` string at the end of the area name.

IX. EXAMPLES

In this section I discuss several examples which can be found in the example directory.

A. Monolithically series connected mini-module and a defect

In the folder `Examples/ThinFilm` you can find a PVMOS input file `thin_film.mos`. This file creates a thin-film *a*-Si:H mini-module of 8x8 cm² with 8 cells of 1 cm wide. In addition the third cell has a defect. This example demonstrates many basic PVMOS operations such as creating new meshes, joining meshes, selecting elements, locally refine the mesh, create new area's, setting parameters for area's, and assigning elements to an area.

B. Metal wrap-through module

This example is located in `Examples/Pinup`. The example entails the files several files:

- `generate_mesh.m` Generates the mesh from the images listed below using GNU octave
- `pinup.frontmetal.png` Image used by `generate_mesh.m` to define the areas where there is metal at the front
- `pinup.vias.png` Image used by `generate_mesh.m` to define the areas where the front contact is connected to the contact foil through vias
- `pinup.backisolation.png` Image used by `generate_mesh.m` to define the areas where the back contact is isolating (around the vias)
- `pinup.back2contactfoil.png` Image used by `generate_mesh.m` to define the areas where the back contact is connected to the contact foil
- `pinup.foilisolation.png` Image used by `generate_mesh.m` to define the areas where the contact foil is isolating
- `pinup.contactfoil_vn.png` Image used by `generate_mesh.m` to define the areas the contact foil is connected to ground
- `pinup.contactfoil_vp.png` Image used by `generate_mesh.m` to define the areas where the contact foil is connected to the positive node.
- `pinup.mos` The PVMOS input file

This example demonstrates how a PVMOS mesh can be generated from images using GNU octave where the images are used as selection masks to select elements and change some parameters for these elements. To run this example you need to install the `pvmos-mesh` package in GNU Octave. The example simulates a metal wrap-through solar cell with three electrodes, the front electrode (emitter and front metal), the back electrode (metal contact to the base) and a contact foil at the back which contacts the front metal through vias and the back contact. After running the `generate_mesh.m` script in GNU Octave a mesh is written to `pinup.bin`. This mesh is loaded in the PVMOS script `pinup.mos`.

C. Crystalline silicon module

In `Examples/CSiSTD` we do a simulation of a crystalline solar cell with three bus-bars. This example shows how one can create a rectilinear mesh for a typical crystalline silicon solar cell grid (without using the GNU Octave). Optionally we also demonstrate the inclusion of a complicated shading pattern.

-
- [1] B. E. Pieters, "PVMOS: A Free and Open Source Simulation Tool for Solar Modules," *submitted to: Solar Energy Materials and Solar Cells*, 2015.
 - [2] T. A. Davis. (retrieved 22 May 2014) BLAS performance bug and its effect on sparse "bench" in MATLAB 7.4. [Online]. Available: <http://www.cise.ufl.edu/research/sparse/cholmod/blasbug.html>
 - [3] T. Ott, Thomas Walter, and Thomas Unold. "Phototransistor effects in Cu(In,Ga)Se₂ solar cells," *Thin Solid Films* 535, 275-278, 2013.