

# **VEMLab**

A MATLAB library  
for the virtual element method

## **VEMLab Primer**

Version 2.2

October 2018

## Copyright and License

VEMLab, Copyright © 2018  
by Alejandro Ortiz-Bernardin  
<http://camlab.cl/research/software/vemlab/>

CAMLab - Computational and Applied Mechanics Laboratory  
CEMCEN - Center for Modern Computational Engineering  
Department of Computer Science  
Department of Mechanical Engineering  
Facultad de Ciencias Físicas y Matemáticas  
Universidad de Chile  
Av. Beauchef 851, Santiago 8370456, Chile

-----



Your use or distribution of VEMLab or any derivative code implies that you agree to this License.

This program is free software: you can redistribute it and/or modify it under the terms of the GNU General Public License as published by the Free Software Foundation, either version 3 of the License, or (at your option) any later version.

This program is distributed in the hope that it will be useful, but WITHOUT ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE. See the GNU General Public License for more details.

You should have received a copy of the GNU General Public License along with this program. If not, see <<http://www.gnu.org/licenses/>>.

## **TABLE OF CONTENTS**

1	Summary of updates.....	3
2	Features of VEMLab.....	4
3	Source code.....	4
4	Up and running with VEMLab.....	5
5	Examples.....	6
5.1	Displacement patch test.....	6
5.2	Cantilever beam subjected to a parabolic end load.....	6
5.3	Creating and using a custom meshfile.....	8
6	Plotting in GiD.....	10
7	Setting plot and output options.....	13
8	VEMLab's website.....	14

## 1 Summary of updates

From VEMLab v2.1 to VEMLab v2.2:

- Fix `disp()` in `plot_and_output_options.m`: `disp("Hello")` seems to work only in newer versions of MATLAB. So, it is changed to the standard MATLAB format `disp('Hello')`.
- Results that are postprocessed in the graphical user interface of GiD are now ordered in folders.
- Add option to plot deformed domain in MATLAB figures when using the LinearElastostatics module (see function "plot\_and\_output\_options.m" located in the folder "config".)
- Add a function to read a meshfile having the domain type declared as "Custom", which is useful for defining the meshfile manually or using an external mesh generator or using a customized version of the mesh generators available in VEMLab. (See example "Creating and using a custom meshfile" in the VEMLab Primer for details.)

From VEMLab v2.0.2 to VEMLab v2.1:

- Add customized wrench domain (for PolyMesher mesh generator only).
- Add customized plate with a hole domain (for PolyMesher mesh generator only).
- Add the following test: "square\_plate\_with\_source2\_poisson2d.m" in test folder.
- Add the following test: "plate\_with\_hole\_linelast2d.m" in test folder.
- Add the following test: "wrench\_linelast2d.m" in test folder.
- Fix iteration counter in PolyMesher function.

From VEMLab v2.0.1 to VEMLab v2.0.2:

- Fix several bugs when using `vemlab_method='FEM2DQ4'` and `vemlab_method = 'FEM2DT3'`.
- Add a control variable in `config.m` that permits to explicitly set the number of Gauss points to integrate the FEM2DQ4 stiffness matrix and body force vector.

From VEMLab v2.0 to VEMLab v2.0.1: the following features have been added

- More detailed manual in folder "doc."
- Improvement to the plotting of axis and fonts in MATLAB figures.

From VEMLab v1.0 to VEMLab v2.0: the following features have been added

- Two-dimensional Poisson problem
- Setup of plot and output options in function "plot\_and\_output\_options" located in folder "config."
- Additional plotting options (stresses, strains, fluxes and gradients) to MATLAB figures, text files and GiD files.
- Option to plot solutions to VTK files.

VEMLab 1.0: (Initial release of code)

- Two-dimensional linear elastostatics (plane strain and plane stress)
- Solution methods: VEM (polygonal elements), FEM (3-node triangles, 4-node quadrilateral)
- Boundary conditions: Dirichlet, Neumann on boundary edges; can be a constant or a function.
- Meshers: PolyMesher, `distmesh2d`, `quad4mesh`; customized for rectangular domains only (requires adjustments for other domain types)
- Meshes need to be generated separately and stored inside folder "mesh\_files" located in the folder "test."

- Meshes must be generated with the functions “create\_” located in the folder “mesher.”
- Solutions can be plotted to MATLAB figures, text files and GiD files.

## 2 Features of VEMLab

VEMLab is a free and open source MATLAB library for the virtual element method.

Features:

- Two-dimensional linear elastostatics (plane strain and plane stress) and two-dimensional Poisson problem.
- Solution methods: linear VEM (polygonal elements), FEM (3-node triangles, 4-node quadrilaterals).
- Boundary conditions: Dirichlet, Neumann on boundary edges; can be a constant or a function.
- Meshers: PolyMesher, distmesh2d, quad4mesh; PolyMesher is customized for rectangular domain, wrench domain and plate with a hole domain; distmesh2d and quad4mesh are customized for rectangular domain only. Domains can be extended for any of the meshers, but it requires adjustments to some interface functions (see the instructions that are available in functions [create\\_polygonal\\_mesh.m](#), [create\\_quadrilateral\\_mesh.m](#) and [create\\_triangular\\_mesh.m](#) in folder “mesher”).
- Meshes need to be generated separately and saved to folder “test/mesh\_files.”
- Meshes must be generated with the functions “create\_” located in the folder “mesher.” Then, the files containing the generated meshes will be automatically saved to folder “test/mesh\_files” for their use.
- Solutions can be plotted to MATLAB figures, text files, GiD files and VTK files.

**About plotting capabilities:** Currently, the MATLAB plotting of stresses and strains (linear elastostatic problem) or fluxes and gradients (Poisson problem) is very limited and won't recognize holes that might come with the domain geometries. It will work ok with rectangular geometries without holes, but the mesh will need to be very refined to obtain a colormap on the whole geometry ... just try a coarse mesh and then a refined mesh to see what is being said.

If quality colormap plots are required for stresses, strains, fluxes and gradients, it is highly recommended using the GiD output files to visualize them in "GiD the pre and postprocessor" ([www.gidhome.com](http://www.gidhome.com)).

On the other hand, the MATLAB plotting of primary field variables (like displacements) presents no limitations and produces quality colormaps.

## 3 Source code

All the information related to VEMLab and its source code is available on the web:

<http://camlab.cl/research/software/vemlab/>

Download the code before proceeding with the rest of this primer.

## 4 Up and running with VEMLab

VEMLab is a library. You need to create a main .m file and place it inside the folder “test.” The main file has the typical structure of a FEM simulation. Simply follow the test problems (they are given with detailed comments) that are provided inside the folder “test” to write your own .m files or modify the ones provided. More details are given below.

In order to run a simulation a main .m function must be prepared. This main function, when executed, will start the simulation and drive it until its end. Some examples of main functions are provided in the folder “test.”

The folder “test/mesh\_files” has various text files that contain the information of predefined meshes that are ready to be used and read from the example main files that are available in the folder “test.” In order to generate new meshes, the user must use the following functions that are available with instructions in the folder “mesher”:

- `create_polygonal_mesh.m` for polygonal elements.
- `create_quadrilateral_mesh.m` for four-node quadrilateral elements.
- `create_triangular_mesh.m` for three-node triangular elements.

The previous functions create text files that contain the information of the generated meshes and save them to the folder “test/mesh\_files” so that they are available to be read from the main functions.

**Important note about mesh generation:** `create_polygonal_mesh.m` is a wrapper for PolyMesher, `create_triangular_mesh.m` is a wrapper for distmesh2d, and `create_quadrilateral_mesh.m` is a wrapper for quad4mesh; PolyMesher is customized for rectangular domain, wrench domain and plate with a hole domain; distmesh2d and quad4mesh are customized for rectangular domain only. Domains can be extended for any of the meshers, but it requires some adjustments to some interface functions (see the instructions that are available in functions `create_polygonal_mesh.m`, `create_quadrilateral_mesh.m` and `create_triangular_mesh.m` in folder “mesher”).

The following must be taken into account when preparing the main file:

The method to be used in the simulation is specified by the variable `vemlab_method`. For instance, to perform the simulation with

- VEM, declare this variable as `vemlab_method='VEM2D'`.
- FEM, declare this variable as `vemlab_method='FEM2DQ4'` for four-node quadrilateral elements, and `vemlab_method='FEM2DT3'` for three-node triangular elements.

The four-node quadrilateral and three-node triangular elements are particular instance of polygonal elements, and as such they can be used to simulate with the VEM specifying `vemlab_method='VEM2D'`. However, polygonal meshes with elements of more than three edges cannot be used when specifying `vemlab_method='FEM2DT3'`, and polygonal meshes with elements of three or more than four edges cannot be used when specifying `vemlab_method='FEM2DQ4'`.

Before starting the simulation it is important to setup the options for the plots and output files. These options can be activated or deactivated in the function `plot_and_output_options.m` that is located in the folder “config.”

The output files created with the simulation are saved to folder “test/output\_files.” Inside this folder there are three subfolders that contain specific output files, as

follows: the folder “GiD” contains output files that are readable in the postprocessor of GiD (<https://www.gidhome.com/>), the folder “VTK” contains output files that are readable in the Visualization Toolkit VTK (<https://www.vtk.org/>) or in the visualization application ParaView (<https://www.paraview.org/>), and the folder “txt” contains the output files in text format.

## 5 Examples

### 5.1 Displacement patch test

This test consists in the solution of the linear elastostatic problem with  $\mathbf{b} = \mathbf{0}$  and essential (Dirichlet) boundary conditions  $\mathbf{g} = [x_1 \ x_1 \ +x_2]^T$  imposed along the entire boundary of a unit square domain. Plane strain condition is assumed with the following material parameters:  $E_V = 1 \times 10^7$  psi and  $\nu = 0.3$ . The main file for this problem is provided as the file `linear_patch_test_linelast2d.m` that is located in the folder “test.” The polygonal mesh and the VEM results are shown in Fig. 5.1. The relative L2-norm of the error and the relative H1-seminorm of the error obtained for the mesh shown in Fig. 5.1(a) are  $2.5493 \times 10^{-16}$  and  $1.3766 \times 10^{-15}$ , respectively. Therefore, as predicted by the theory, the VEM solution coincides with the exact solution given by  $\mathbf{g}$  within machine precision.

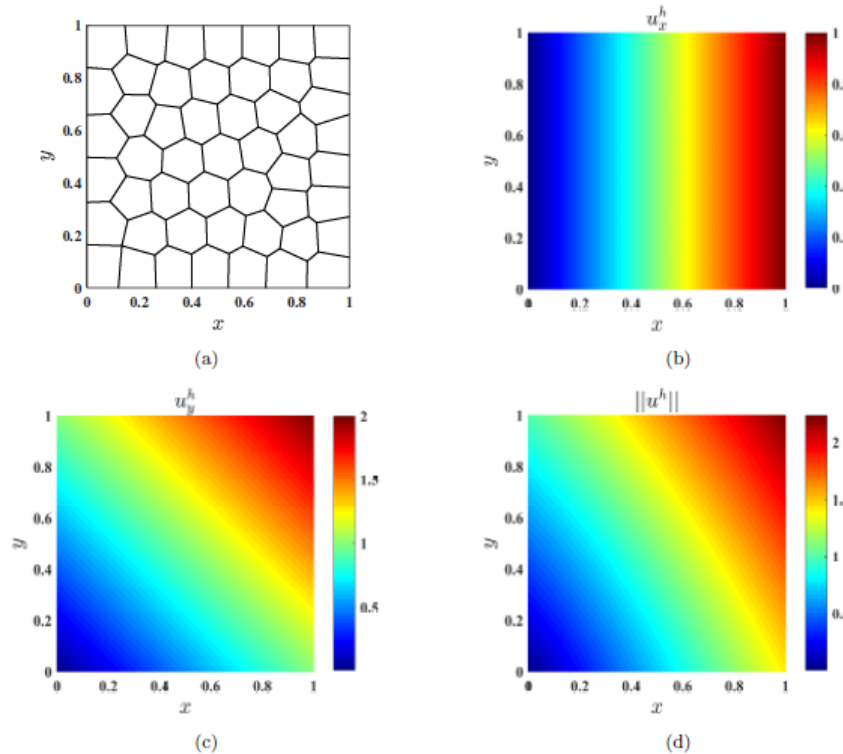


Fig. 5.1: Solution for the displacement patch test using VEMLab. (a) Polygonal mesh, (b) VEM horizontal displacement, (c) VEM vertical displacement, and (d) norm of the VEM displacement. The relative L2-norm of the error is  $2.5493 \times 10^{-16}$  and the relative H1-seminorm of the error is  $1.3766 \times 10^{-15}$ .

### 5.2 Cantilever beam subjected to a parabolic end load

The VEM solution for the displacement field on a cantilever beam of unit thickness subjected to a parabolic end load  $P$  is computed using VEMLab. Fig. 5.2 illustrates the geometry and boundary conditions. Plane strain state is assumed. The essential boundary conditions on the clamped edge are applied according to the analytical solution given by Timoshenko and Goodier:

$$u_x = -\frac{Py}{6\bar{E}_Y I} \left( (6L - 3x)x + (2 + \bar{\nu})y^2 - \frac{3D^2}{2}(1 + \bar{\nu}) \right),$$

$$u_y = \frac{P}{6\bar{E}_Y I} (3\bar{\nu}y^2(L - x) + (3L - x)x^2),$$

where  $\bar{E}_Y = E_Y / (1 - \nu^2)$  with the Young's modulus set to  $E_Y = 1 \times 10^7$  psi, and  $\bar{\nu} = \nu / (1 - \nu)$  with the Poisson's ratio set to  $\nu = 0.3$ ;  $L = 8$  in. is the length of the beam,  $D = 4$  in. is the height of the beam, and  $I$  is the second-area moment of the beam section. The total load on the traction boundary is  $P = -1000$  lbf.

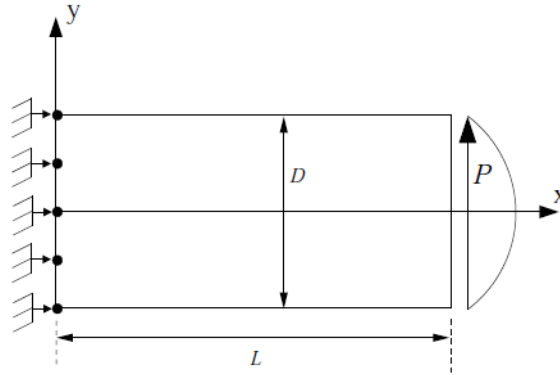


Fig. 5.2: Model geometry and boundary conditions for the cantilever beam problem.

In order to solve this problem in VEMLab, the function `cantilever_beam_linelast2d.m` is used. This function is located in the folder "test." The polygonal mesh and the VEM displacements results are shown in Fig. 5.3.

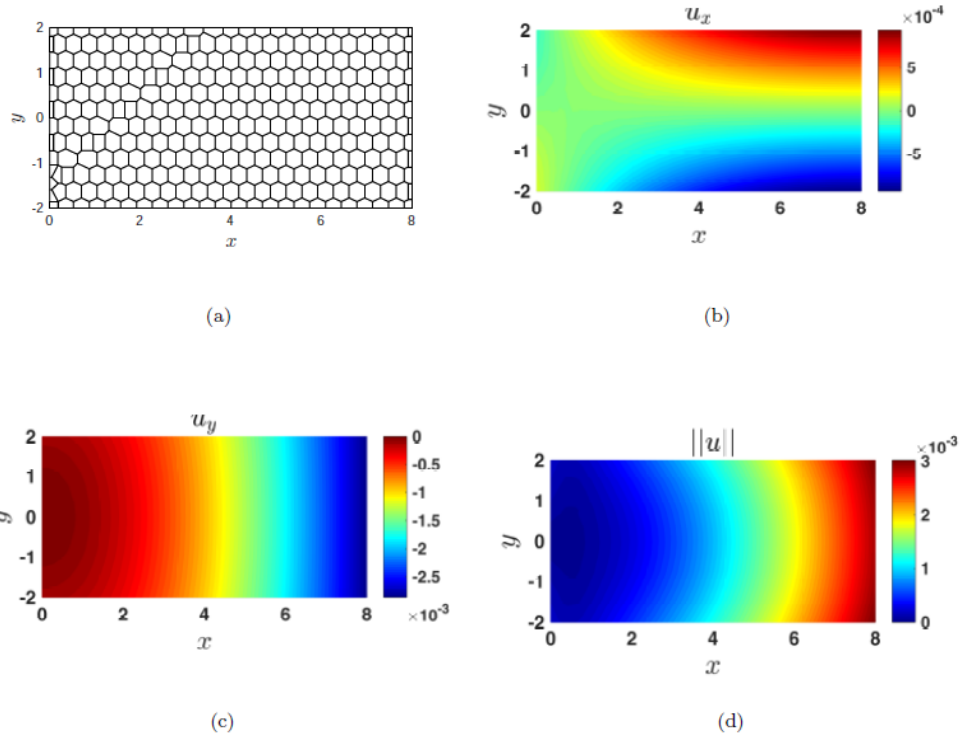


Fig. 5.3: Solution for the cantilever beam subjected to a parabolic end load using VEMLab. (a) Polygonal mesh, (b) VEM horizontal displacement, (c) VEM vertical displacement, (d) norm of the VEM displacement.



A performance comparison between VEM and FEM is conducted. For the FEM simulations, three-node triangles (T3) are used. The performance of the two methods are compared in Fig. 5.4, where the relative  $H^1$ -seminorm of the error and the normalized CPU time are each plotted as a function of the number of degrees of freedom (DOF). The normalized CPU time is defined as the ratio of the CPU time of a particular model analyzed to the maximum CPU time found for any of the models analyzed. From Fig. 5.4 it is observed that for equal number of degrees of freedom both methods deliver similar accuracy and the computational costs are about the same as the mesh is refined.

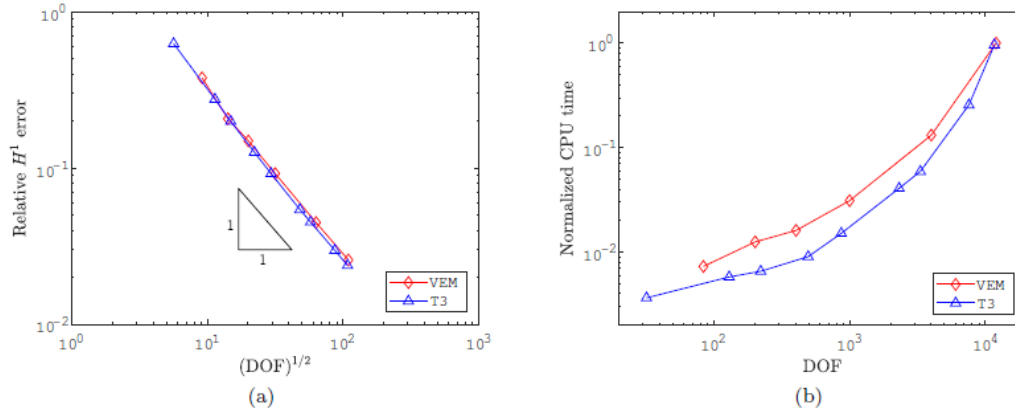


Fig. 5.4: Cantilever beam subjected to a parabolic end load. Performance comparison between the VEM and the FEM (three-node triangles (T3)). (a) Relative  $H^1$ -seminorm of the error as a function of the number of degrees of freedom and (b) normalized CPU time as a function of the number of degrees of freedom.

### 5.3 Creating and using a custom meshfile

This example illustrates how to define a customized mesh when the mesh is not generated with the mesh generation functions provided in the source code of VEMLab.

Consider the cantilever beam subjected to an end load that is shown in Fig. 5.5. The domain is a rectangle of dimensions  $27.2 \times 17$ . The material parameters are  $E_y = 1e7$  and  $\nu = 0.3$ , and plane strain condition is assumed. The end load is set to  $-10000$ . The mesh has been defined manually. The clamped side is defined as the Dirichlet boundary and the loaded end is defined as the Neumann boundary. The custom mesh file starts with the keyword "Custom" and is followed, as usual, by the quantity of nodes (in this case, 44) and their coordinates; then, the number of polygonal elements (in this case 7) and their connectivity. The three final lines of the mesh file correspond, respectively, to the list of the nodes located on the Dirichlet boundary (in this case 2 nodes), the list of nodes located on the Neumann boundary (in this case 2 nodes) and the bounding box  $xmin, xmax, ymin, ymax$ , where the domain is contained. The bounding box area can be larger than the domain area. The bounding box is only used for plotting purposes using MATLAB figures --- it is not used for any other type of plot (e.g., GiD and VTK). The complete meshfile is provided in the folder "test/mesh\_files" as the file "vem\_letters\_7poly\_elems.txt" and is reproduced below. The main function used to run this problem is the function `vem_letters_linelast2d.m` that is in the folder "test".

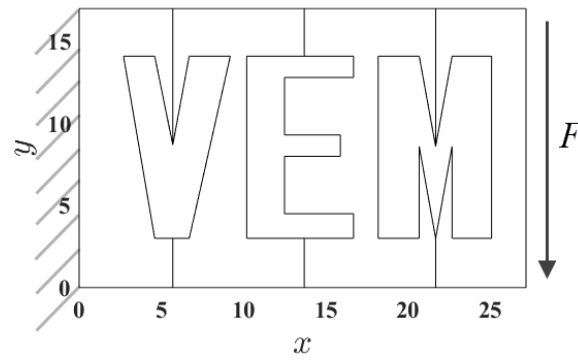


Fig. 5.5: A cantilever beam subjected to an end load and discretized with a customized mesh formed by 7 polygonal elements.

```
# domain type
Custom
# nodal coordinates: number of nodes followed by the coordinates
44
0.00  0.00
5.70  0.00
5.70  3.00
4.60  3.00
2.70  14.10
4.60  14.10
5.70  8.70
5.70  17.00
0.00  17.00
13.70  0.00
13.70  3.00
10.20  3.00
10.20  14.10
13.70  14.10
13.70  17.00
16.70  3.00
16.70  4.50
12.50  4.50
12.50  8.00
15.90  8.00
15.90  9.30
12.50  9.30
12.50  12.80
16.70  12.80
16.70  14.10
21.70  0.00
21.70  3.00
20.70  8.60
20.70  3.00
18.20  3.00
18.20  14.10
20.70  14.10
21.70  8.60
21.70  17.00
22.70  8.60
22.70  3.00
25.10  3.00
25.10  14.10
22.70  14.10
27.20  0.00
27.20  17.00
6.70  3.00
9.20  14.10
6.70  14.10
```

```
# element connectivity: number of elements followed by the connectivity (each line:
nodes_per_element(nel) node1 node 2 ... node_nel)
7
9 1 2 3 4 5 6 7 8 9
8 4 3 42 43 44 7 6 5
13 2 10 11 12 13 14 15 8 7 44 43 42 3
14 12 11 16 17 18 19 20 21 22 23 24 25 14 13
23 10 26 27 28 29 30 31 32 33 34 15 14 25 24 23 22 21 20 19 18 17 16 11
12 30 29 28 27 35 36 37 38 39 33 32 31
11 26 40 41 34 33 39 38 37 36 35 27
# indices of nodes located on the Dirichlet boundary
1 9
# indices of nodes located on the Neumann boundary
40 41
# xmin, xmax, ymin, ymax of the bounding box
0 27.2 0 17
```

## 6 Plotting in GiD

MATLAB plots available in VEMLab provide limited options as discussed in Section 2. GiD postprocessing is the most complete option to visualize all the variables that are computed in VEMLab. When activated in `plot_and_output_options.m`, results are written to GiD files and saved to folder “`test/output_files/GiD`”. GiD can be downloaded from its webpage: <https://www.gidhome.com/>.

The following figures summarize the procedure to visualize the stresses in the `postprocess` module of GiD. The same procedure applies for any of the other variables computed by VEMLab (e.g., displacements, strains, gradient, fluxes, etc.)

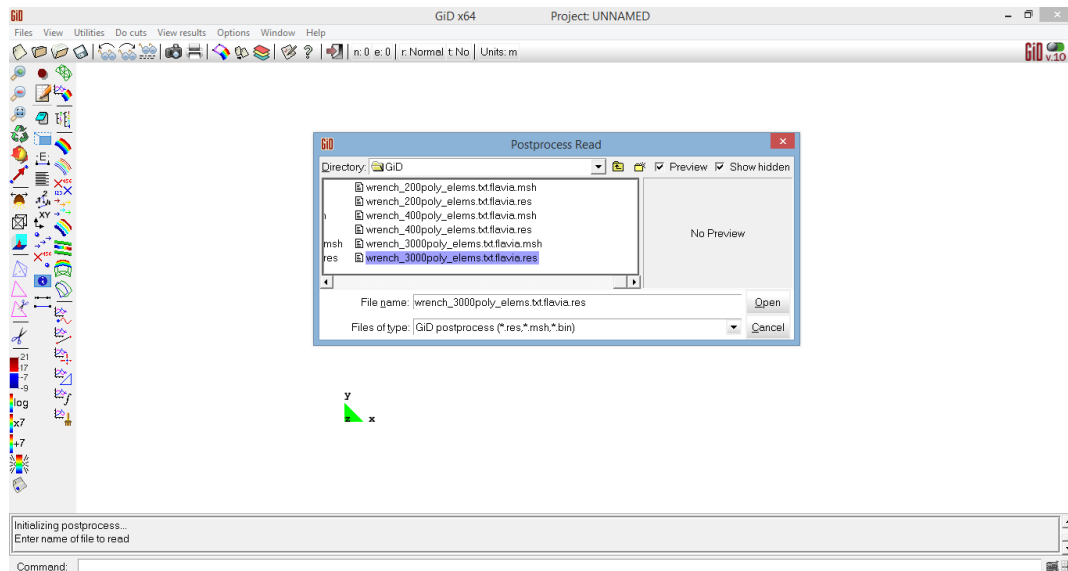


Fig. 6.1: In the `postprocess` module, open the `.res` file that was saved to folder “`test/output_files/GiD`.”

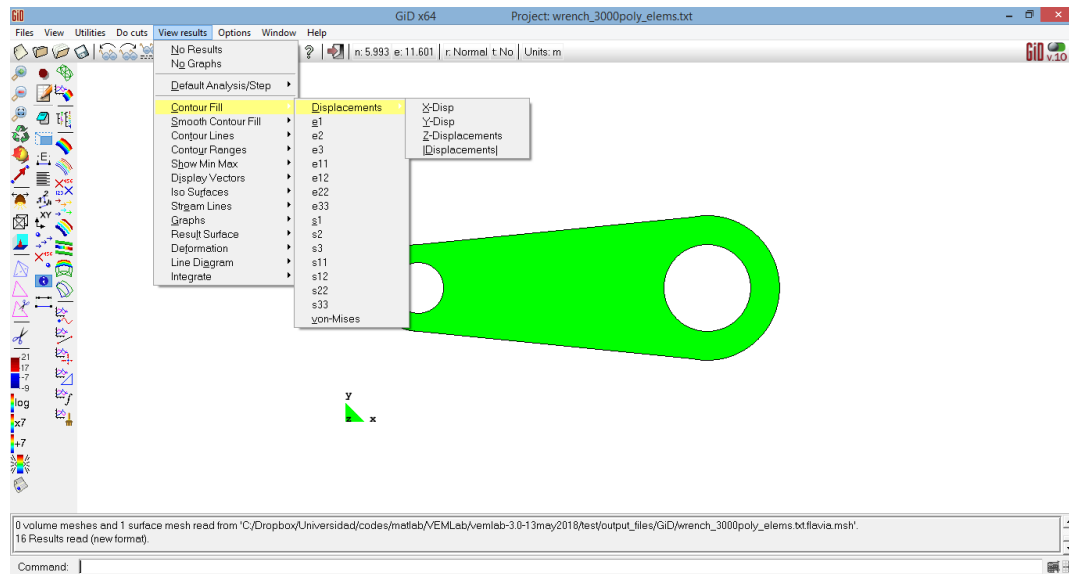


Fig. 6.2: In the `postprocess` module, click on “View results” to display a menu with various options for plotting results. See for instance, “Contour Fill.”

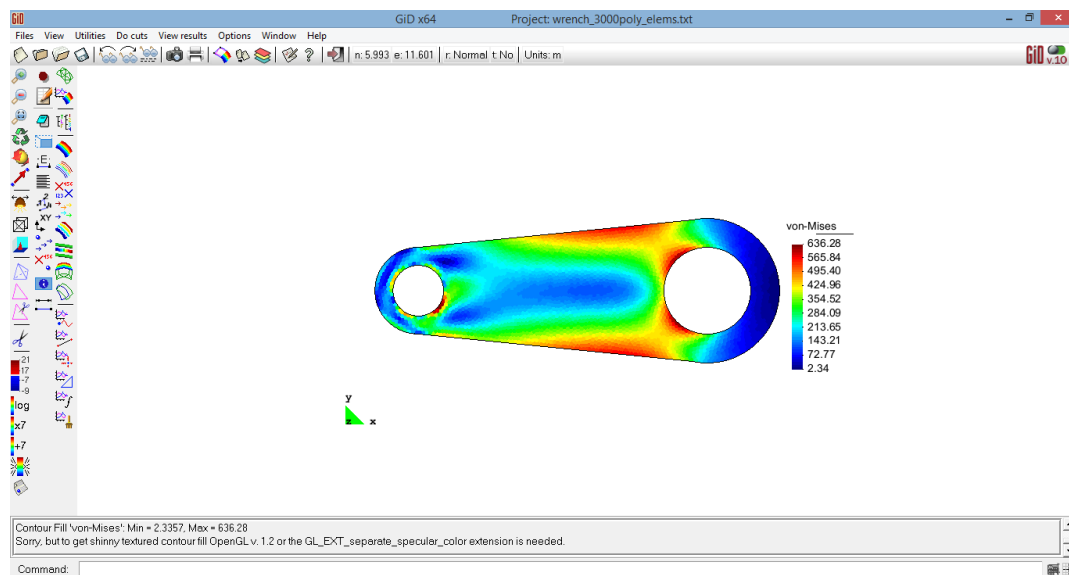


Fig. 6.3: In `View results->Contour Fill`, click on `von-Mises` to plot the von Mises stress.

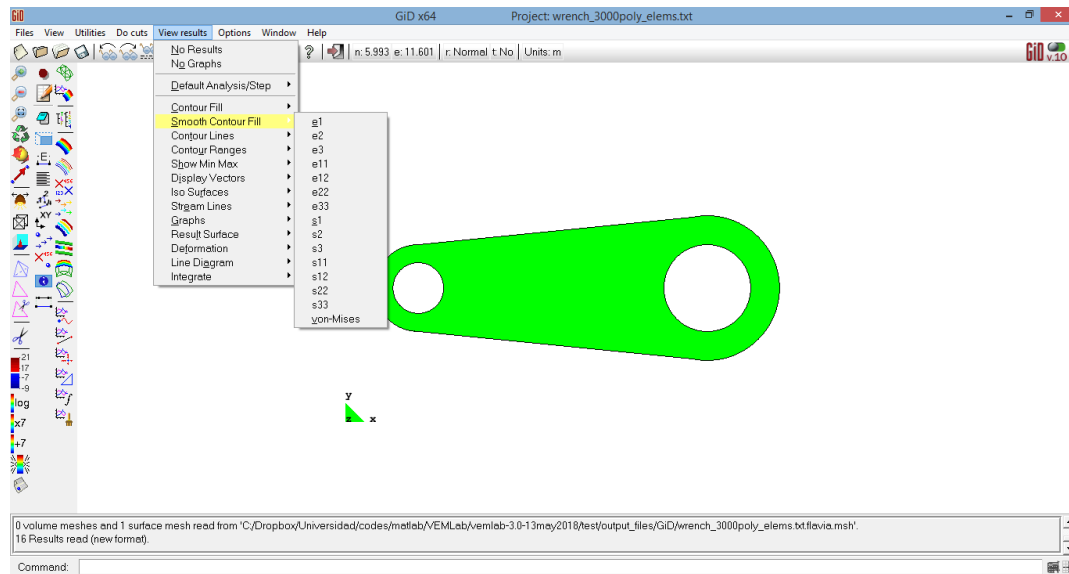


Fig. 6.4: If desired, stresses can be smoothed using the menu **View results->Smooth Contour Fill**.

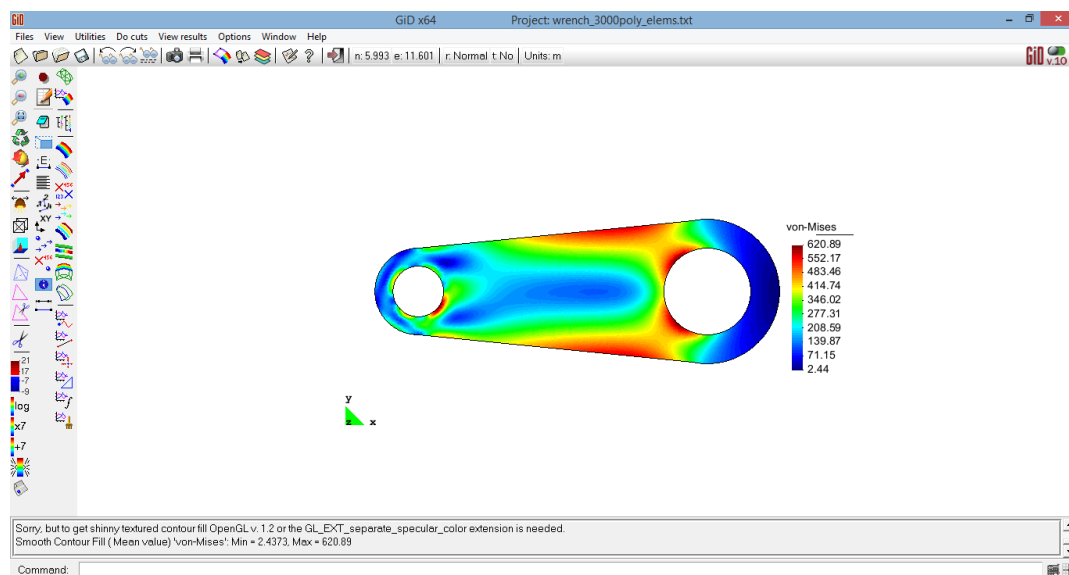


Fig. 6.5: In **View results->Smooth Contour Fill**, click on **von-Mises** to plot the smoothed von Mises stress.

Also, GiD can be used to plot deformed shapes as summarized in the following figures.

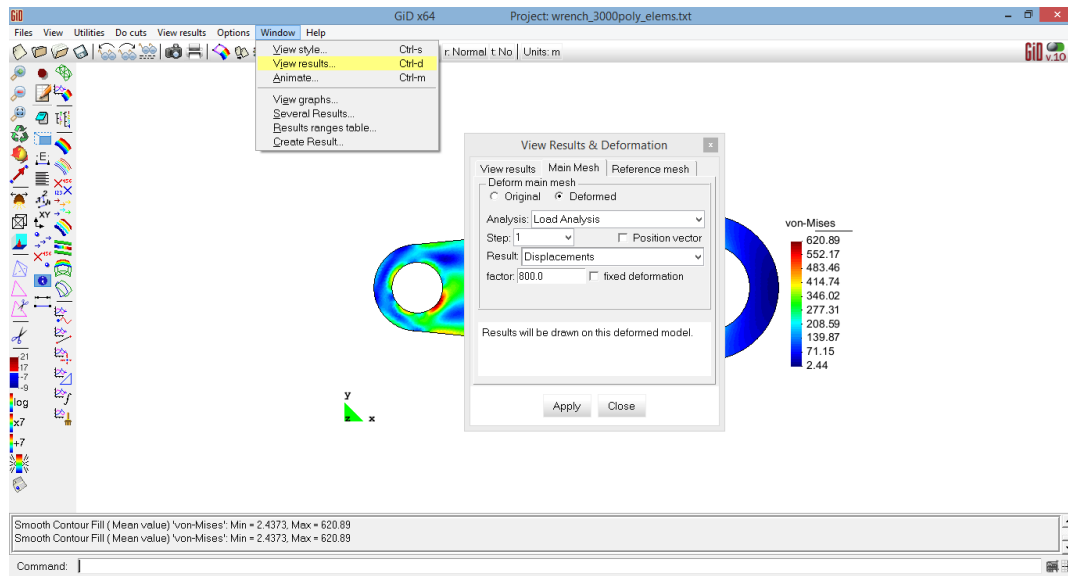


Fig. 6.6: In the postprocess module, click on **Window->View results** and on the displayed box click on **Main Mesh**, select **Deformed** and enter the desired magnification factor in the **factor** text box.

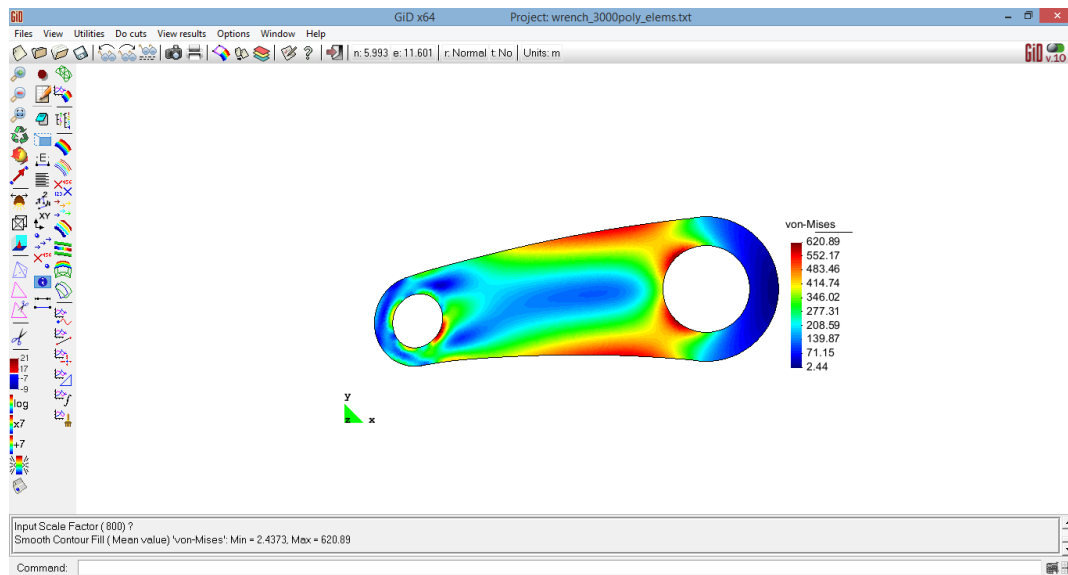


Fig. 6.7: Once the magnification factor is set, click on **Apply** to display the magnified deformed shape.

## 7 Setting plot and output options

The plot and output options must be setup by the user. If these are not setup, the program will use the default options. To setup these options go to the file `plot_and_output_options.m` that is in the folder "config" and activate by setting "yes" or deactivate by setting "no" the available options. Some options include the following (see function `plot_and_output_options.m` for the complete set of options available):

```

%% GENERAL

plot_mesh='yes';
plot_mesh_over_results='yes';
write_solutions_to_text_file='yes';
write_solutions_to_GiD_file='yes';
write_solutions_to_VTK_file='yes';

%% POISSON MODULE

% plotting of main variables to MATLAB figures
poisson2d_plot_scalar_field.u='yes';

% plotting of fluxes to MATLAB figures
poisson2d_plot_flux.qx='no';
poisson2d_plot_flux.qy='no';
poisson2d_plot_flux.qnorm='yes'; % norm of the flux

% plotting of gradients to MATLAB figures
poisson2d_plot_grad.dx='no';
poisson2d_plot_grad.dy='no';
poisson2d_plot_grad.dnorm='yes'; % norm of the gradient

%% LINELAST MODULE

% options for plotting deformed domain to MATLAB figures
linelast2d_plot_deformed_domain='yes';
linelast2d_scale_for_plotting_deformed_domain=1; % a number > 1 will scale the
                                                % deformed domain when plotting to
                                                % MATLAB figures

% plotting of main variables to MATLAB figures
linelast2d_plot_displacement.ux='yes';
linelast2d_plot_displacement.uy='yes';
linelast2d_plot_displacement.unorm='yes'; % norm of the displacement

% plotting of stresses to MATLAB figures
linelast2d_plot_stress.s11='no';
linelast2d_plot_stress.s12='no';
linelast2d_plot_stress.s22='no';
linelast2d_plot_stress.s33='no';
linelast2d_plot_stress.s1='no';
linelast2d_plot_stress.s2='no';
linelast2d_plot_stress.s3='no';
linelast2d_plot_stress.vm='no';

% plotting of strains to MATLAB figures
linelast2d_plot_strain.e11='no';
linelast2d_plot_strain.e12='no';
linelast2d_plot_strain.e22='no';
linelast2d_plot_strain.e33='no';
linelast2d_plot_strain.e1='no';
linelast2d_plot_strain.e2='no';
linelast2d_plot_strain.e3='no';

```

## 8 VEMLab's website

Check VEMLab's website for newer versions:

<http://camlab.cl/research/software/vemlab/>

--- THE END ---