FCVM MANUAL AND TUTORIAL

VERSION 1.0, 27 JANUARY 2025

CONTENTS

1. Introduction

fcVM is a finite element workbench and solver for performing collapse analyses of structures and soil bodies. It is based on the theory of elasto plasticity and gives insight in ductility and reserve strength beyond first yield. It also has the option to perform geometric nonlinear analyses to evaluate the effect of large displacements and structural instabilities. This document summarises the theory and application of fcVM.

2. Installation

Although the aim is to automate installation of fcVM and its dependencies through the FreeCAD add-on manager, for now the process is manual and can best be done using a package manager like Mamba.

2.1. **fcVM installation.** The workbench itself can be installed with the following steps:

- Create a fcVM-workbench directory in the FreeCAD Mod / directory, the location of which can be found by typing App.getUserAppDataDir() in the FreeCAD python console.
- Copy the entire fcVM-workbench repository into this new fcVM-workbench directory.
- Restart FreeCAD

If all went well, the fcVM workbench should now be an option in the FreeCAD workbench dropdown menu.

2.2. **Dependencies.** fcVM imports (from) the following packages (dependencies):

- numpy
- scipy (version 1.11.3)
- numba
- matplotlib
- scikit-sparse
- pyvista
- meshio

Some of these packages come shipped with FreeCAD, but others require manual installation. This can best be done with a package manager.

2.3. **Dependency installation with Mamba.** Mamba is a nimble and fast package manager that can be used to installed the required dependencies as follow:

- Download Miniforge3
    miniforge
- Run the installer:
    Miniforge3-Windows-x86_64.exe
- Find and run Miniforge on your system - this opens a Miniforge Prompt:
    (base) C:"path">
- Create a new virtual environment:
    (base) C:"path"> mamba create --name fcVM (or any other name of your choice)
- Change into the new environment:
    (base) C:"path"> mamba activate fcVM (or the another name of your choice)
- Install FreeCAD and dependencies:
    (fcVM) C:"path"> mamba install freecad scipy=1.11.3 numba matplotlib scikit-sparse pyvista meshio (with spaces and no commas)
- Check with python if the dependencies can be imported
    (fcVM) C:"path"> python
    >>> import scipy.sparse
    >>> import sksparse.cholmod
- If no errors, quit python and start freecad:
    (fcVM) C:"path"> freecad

- If you encounter a "black screen" then rename all opengl32sw.dll files on your system to opengl32.dll.

3. License Information

Copyright (c) 2024 - Harry van Langen hvlanalysis@gmail.com

This program is free software; you can redistribute it and/or modify it under the terms of the GNU Lesser General Public License (LGPL) as published by the Free Software Foundation; either version 2 of the License, or (at your option) any later version. for detail see the LICENCE text file.

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 Library General Public License for more details.

You should have received a copy of the GNU Library General Public License along with this program; if not, write to the Free Software Foundation, Inc., 59 Temple Place, Suite 330, Boston, MA 02111-1307 USA

4. Theory

fcVM is based on the finite element method. The element used is the 10-node tetrahedron (Fig. 1). This element models displacement fields up to second order, meaning that the element (unlike the 4-node tetrahedron) can curve upon deformation and that elastic stresses vary linearly in each element. In the evaluation of stresses , strains and element reactions 4 integration points are used, which allow for exact integration of elastic response.
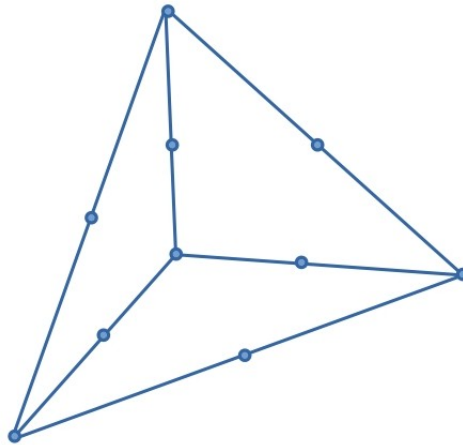
INCOMPLETE

Figure 1. 10-noded tetrahedral (TET10) element

5. Physical Input

In this section the user interface and the meaning of various settings is explained. When selecting the fcVM workbench from the workbench drop-down menu, the user interface (UI) of Figure 2. appears.
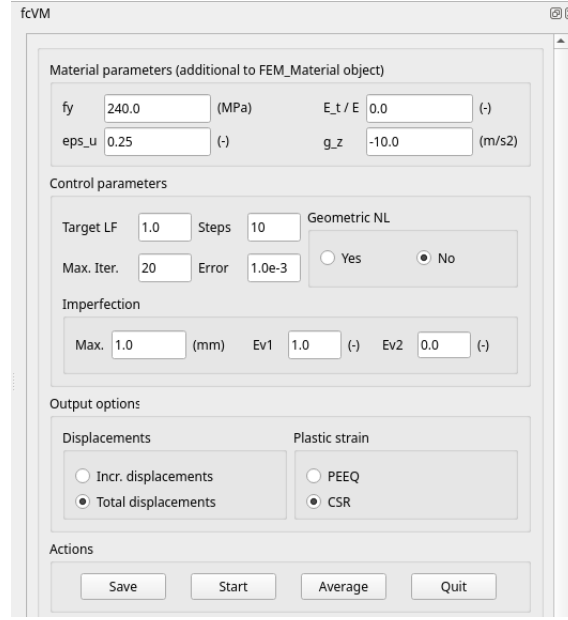
Figure 2. fcVM User Interface (UI)

It allows specification of material and analysis control parameters that are not available through the normal FreeCAD UI.

5.1. **fcVM units.** Before giving a setailed description of the input parameters, first a few words about the units used in fcVM.

Internally, fcVM works with the the units $[mm]$ for length, $[N]$ for force and $[M/mm^2] = [MPa]$ for stress, and the results are therefore also reported in those units. Input parameters from the FreeCAD material object (i.e. Density and Young's Modulus) are automatically converted in fcVM. Values entered in the fcVM UI need to follow the units shown in that UI.

5.2. **Material parameters in the FreeCAD material object.** Young's modulus and material density are entered in the FreeCAD material object. The user is free to choose the units for those parameters.

5.3. **Material parameters in the fcVM user interface.** The following material parameters are entered in the fcVM user interface.

5.3.1. *Yield strength f_y.* The yield strength gives the value of stress at which the material yields in a uniaxial tensile test. This is used in the von Mises plasticity model to calculate yield in general states of stress (ref.1). The stress combination at which the material yields is then expressed as

$$(\sigma_{xx} - \sigma_{yy})^2 + (\sigma_{yy} - \sigma_{zz})^2 + (\sigma_{zz} - \sigma_{xx})^2 + 6(\sigma_{xy}^2 + \sigma_{yz}^2 + \sigma_{zx}^2) = 2f_y^2$$

where $\sigma_{xx}, \sigma_{yy}, \sigma_{zz}$ are the normal stresses and $\sigma_{xy}, \sigma_{yz}, \sigma_{zx}$ are the shear stresses. For example, in uniaxial tension all stresses are zero except $\sigma_{xx} = \sigma$, so $2\sigma^2 = 2f_y$ and the material yields at $\sigma = f_y$. In pure shear, only $\sigma_{xy} = \tau$, so the material yields when $6\tau^2 = 2f_y^2$, or $\tau = \frac{f_y}{\sqrt{3}}$.

The yield strenth is entered in $[MPa]$.

5.3.2. *Hardening modulus E_t/E.* The hardening modulus is the ratio of the slopes of the stress-strain curve before and after first yielding, respectively. This allows the description of a bi-linear stress-strain curve. The default value is $E_t/E = 0$, implying an elastic-perfectly-plastic material model. Practical values for metals are in the range $E_t/E = 0.001 - 0.01$.
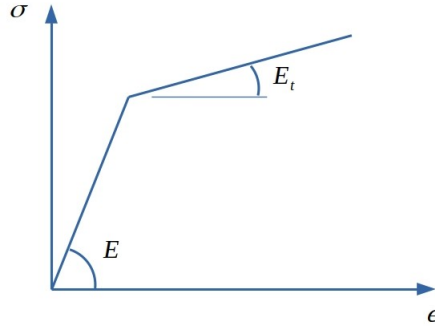

Figure 3. Hardening modulus

5.3.3. *Ultimate strain at rupture eps_u.* The ultimate strain at rupture (eps_u or ϵ_u) is the strain at which a specimen breaks in a uniaxial tensile test. To generalise this to general states of stress and strain, an equivalent strain ($PEEQ$ or ϵ_{eq}) is introduced using a formula similar to that of the von Mises stress

$$PEEQ = \epsilon_{eq} = \tfrac{\sqrt{2}}{3}\left[\left(\varepsilon_x^p - \varepsilon_y^p\right)^2 + \left(\varepsilon_y^p - \varepsilon_z^p\right)^2 + \left(\varepsilon_z^p - \varepsilon_x^p\right)^2 + 6\left(\varepsilon_{xy}^p\right)^2 + 6\left(\varepsilon_{xz}^p\right)^2 + 6\left(\varepsilon_{yz}^p\right)^2\right]^{1/2}$$

However, this formula does not distinguish between tension and compression. This is not in line with experiments where the metal is more brittle in tension than compression. To correct for this, a critical plastic strain (ε_{cr}) is introduced that distinguishes between tension and compression through the triaxiliality ($T = \frac{p}{f_y}$) of the stress state, i.e.

$$\varepsilon_{cr} = \exp\left(\tfrac{1}{2} - \tfrac{3}{2}T\right)\epsilon_u$$

Here $p = \tfrac{1}{3}(\sigma_{xx} + \sigma_{yy} + \sigma_{zz})$ is the hydrostatic tension (+ve) or compression (-ve) in the material. For uniaxial tension $T = \tfrac{1}{3}$ at failure and the crtitcal strain is equal to the critical strain in the unixaxial tensile test, as expected. For compression, however, $T = -\tfrac{1}{3}$ and $\varepsilon_{cr} = exp(1)\epsilon_u \approx 2.7\epsilon_u$.

Like in Miner's rule for fatigue, the damage towards rupture is assumed to accumulate in every plastic strain increment. This accumulated damage is called the Critical Strain Ration (CSR).

$$CSR = \sum_{steps} \tfrac{\Delta\epsilon_{eq}}{\varepsilon_{cr}(T)}$$

where $\Delta\epsilon_{eq}$ is the increment of equivalent plastic strain in a load step and $\varepsilon_{cr}(T)$ is the the critical plastic strain at the triaxiality T in the load step. The material is predicted to rupture when the CSR exceeds 1.

5.3.4. *Gravitational accelleration g_z.* Gravity is specified through the gravitational accelleration, which is assumed to work in z-direction. The unit of gravitational accelleration is $[m/s^2]$

6. ANALYSIS CONTROL INPUT

The analysis control parameters specify how the analysis will be executed. The equilibrium path is traced in an incremental iterative way (Figure 4). The load is applied in increments and equilibrium is achieved by performing iterative corrections.
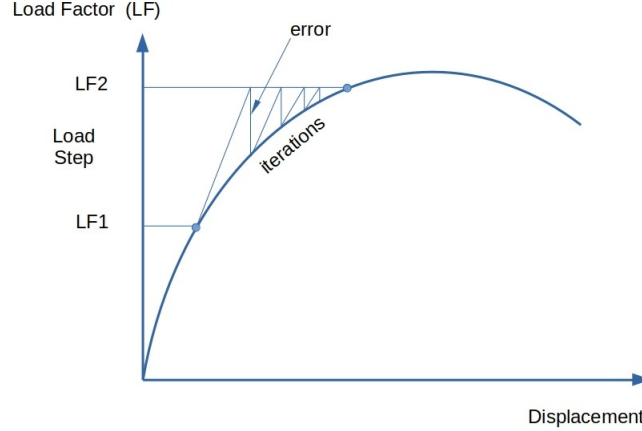
Figure 4. Incremental-iterative solution process

6.1. **Load control parameters.** The following input parameters control the incremental-iterative process.

6.1.1. *Target load factor TargetLF.* This is a factor that, when multiplied with the input loads, represents the actual load level. The target load factor may not be achieved if the structure collapses or if the specified number of steps is insufficient. In the latter case, additional load steps can be entered after the intermediate load displacement curve is produced by fcVM.

6.1.2. *Number of load Steps.* This is the number of steps in which the target load factor is applied. It should be noted that load steps are scaled to manage the speed of convergence. This may mean that fewer or more steps are required than expected from a simple division of the load factor increment by the number of steps.

6.1.3. *Maximum number of iterations Max.Iter.* In case the number of iterations in a load step exceeds the *Max.Iter.* input value, fcVM restarts the load step with a smaller load increment. The default value is 20. Reducing this value forces smaller load steps.

6.1.4. *Allowable equilibrium Error.* The equilibrium error is defined as $\frac{\|\underline{Q}_{ex} - \underline{Q}_{in}\|}{\|\underline{Q}_{ex}\|}$, where $\underline{Q}_{ex}$ is the external load vector and $\underline{Q}_{in}$ is the material reaction vector. When the equilibrium error is larger than the allowable value, the iteration process continues. The default value is 0.001 (0.1%), but values between 0.01 (1%) and 0.1 (10%) may be more appropriate for initial runs. A final analysis may then be conducted with a value of 0.001 to confirm the accuracy of the load-deflection curve.

6.2. **Geometric nonlinear analysis.** In a geometric nonlinear analysis the effects of deformation on the state of equilibrium is taken into account. This only makes sense when the deformations are large compared to the minimum dimensions of the structure (e.g. plate thickness) or when the stresses are large compared to the (lateral) stiffness of the structure (e.g. buckling). By choosing different settings for a geometric nonlinear analysis, several types of analysis can be performed, as follows

			Geometric NL	
			Y	N
Steps	1	type	Linear buckling analysis	Elastic analysis
		output	Buckling factors and shapes + Linear elastic deformation	Linear elastic deformation
	>1	type	Elastic-plastic, geometric non-linear analysis	Elastic-plastic analysis
		output	Elastic deformation + Geometric non-linear, elastic-plastic deformation + Buckling factors and shapes	Elastic deformation + Elastic-plastic defomation

Figure 5. Analysis options

6.2.1. *Geometric NL Yes/No.* This option switches between a geometric linear or nonlinear analysis. It is recommended to always start with a geometric linear analysis to get a feel for the physical characteristics of the model. A geometric linear analysis also executes much faster than a geometric nonlinear analysis. As Figure 5 shows, when selecting $Steps = 1$, a linear elastic (Geometric NL $= Yes$) or linear buckling analysis (Geometric NL $= No$) is performed.

6.2.2. *Imperfection Max..* In a geometric nonlinear analysis the initial imperfection of the structure becomes important. A small initial imperfection can lead to a violent loss of strength at high compressive load (buckling), wheras a realistic imperfection leads to a more gradual loss of stiffness and a lower limit load. Structural design codes typically specify an imperfection of a few percent of a characteristic structural length (span for global buckling or plate width for local buckling). The value of the maximum imprefection is entered in $[mm]$ and applied to the elastic buckling shape (see next section).

6.2.3. *Imperfection $Ev1, Ev2$.* The shape of the actual imperfection of a structure is unknown, so some assumption needs to be made. A conservative choice is the elastic buckling shape corresponding to the lowest buckling load, because it reduces the chance of following an instable equilibrium path before buckling occurs. This shape can be selected by chosing $Ev1 = 1.0$ and $Ev2 = 0.0$ as input parameters. In exceptional cases other bucling shapes may dominate. This may occur for example when local (transverse) loading triggers a local buckling pattern. If this is suspected, then it may be good to investigate the load deformation curve for an imperfection shape corresponding to the second buckling load. This can be done by setting $Ev1 = 0.0$ and $Ev2 = 1.0$. A linear combination of the two shapes is obtained when selecting both parameters non-zero.

6.3. **Output options.** The following options determine what output will be generated at the end of a series of load steps.

6.3.1. *Displacements Incr. displacements/Total displacements.* Selecting "Incremental Displacements" will display the displacement increments in the last load step. This option is useful when reviewing the collapse meachnism of a structure or soil body. Selecting "Total Displacements" will show the total accumulated deformation during the analaysis, which is required when reviewing the deformed geometry of a structure in a geometric nonlinear analysis.

6.3.2. *Plastic strain $PEEQ/CSR$.* As explained in section 5.3.3., both $PEEQ$ and CSR can be used to judge rupture of the material. The load-displacement curve can show one or the other by making the associated choice in the UI.

6.4. **Actions.** Several actions can be taken after completing the input

6.4.1. *Save.* This saves the information entered in the UI to a $*.inp$ file in the *control files* directory.

6.4.2. *Start.* This starts the fcVM analysis

6.4.3. *Average.* After completing the analysis it is possible to average results over faces and/or edges. Select the faces and/or edges and press $<$ *Average* $>$. The results will be displayed in the report view and in the *.avr file in the *output files* directory.

6.4.4. *Quit.* To exit the workbench press *Quit* or simply select another workbench.

7. OUTPUT

fcVM has several ways to inspect results during and after the analysis.

7.1. **Progress information in the UI and report view.** During execution of fcVM, the status of the analysis gets updated in the UI and the Report View.

7.1.1. *Progress information in the UI.* A progress slider (Figure 6) shows the percentage completion as the ratio of the current step divided by the number of steps specified in the input (section 6.1.1). This number may not reach 100% if the target load is achieved with fewer steps. The window also shows the number of the current step, the current load factor and the maximum values of PEEQ and CSR.
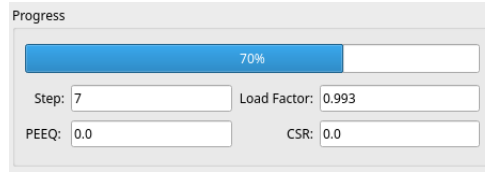

Figure 6. UI progress information

7.1.2. *Progress information in the Report View.* The report view gets updated continuously during the analysis and contains useful information about values and status. The first information that is worth noting is the output of number of elements (ne) and the number of nonzero elements in the global stiffness matrix (ns). After that, the mesh volume [mm] and total loads in x, y and z directions [N] are displayed. The latter serve as a check on the applied loads.

```
11:15:57  ne = 472
11:15:57  ns = 219480
11:15:57
11:15:57  mesh volume 180366.01427060258
11:15:57  loadsumx 49999.99999999978
11:15:57  loadsumy 7.270142102516845e-27
11:15:57  loadsumz -2.4233807008389483e-27
```

Figure 7. Mesh dimensions and applied loads

The report view also contains information about the iteration process. The equilibrium error is printed at the end of every iteration (Figure 8). This allows the user to see the convergence rate of the analysis.

```
12:03:40  Step: 1
12:03:40
12:03:40  Iteration: 0, Error: 4.29e-14
12:03:40
12:03:40  Step: 2
12:03:40
12:03:40  Iteration: 0, Error: 7.15e-14
12:03:40
12:03:40  Step: 3
12:03:40
12:03:40  Iteration: 0, Error: 1.80e-02
12:03:40
12:03:40  Iteration: 1, Error: 2.95e-03
12:03:40
12:03:40  Iteration: 2, Error: 4.58e-04
```

Figure 8. Load steps and iterations information

7.2. **Intermediate results.** After the analysis reaches the target load factor or the specified number of steps, fcVM displays a load displacement curve and a load strain curve.

7.2.1. *Load displacement and strain curves.* At this point it is possible to inspect intermediate results, stop the analysis or continue the analysis to the previously specified or a new target load level. The number of additional steps is at most equal to that entered in the UI previously (see section 6.1.2.), but this addition of load steps can be repeated indefinitely.
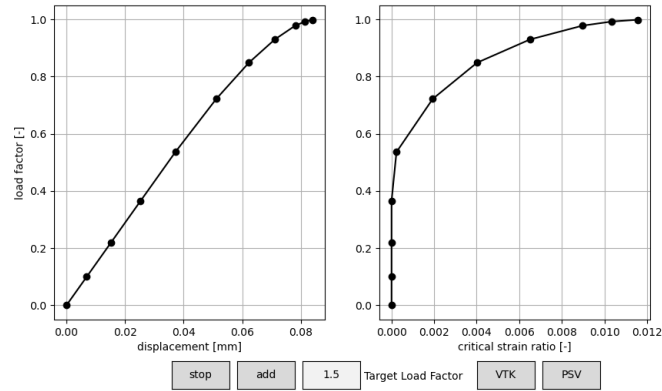

Figure 9. Load displacement and load CSR curves

The window also contains two buttons (labelled VTK and PSV, respectively) that give access to information about the stress and strain state in the material (Figures 10 and 11). Once inside the VTK or PSV display windows, the cutting tools can be toggled by pressing T and a screenshot can be made by pressing S.
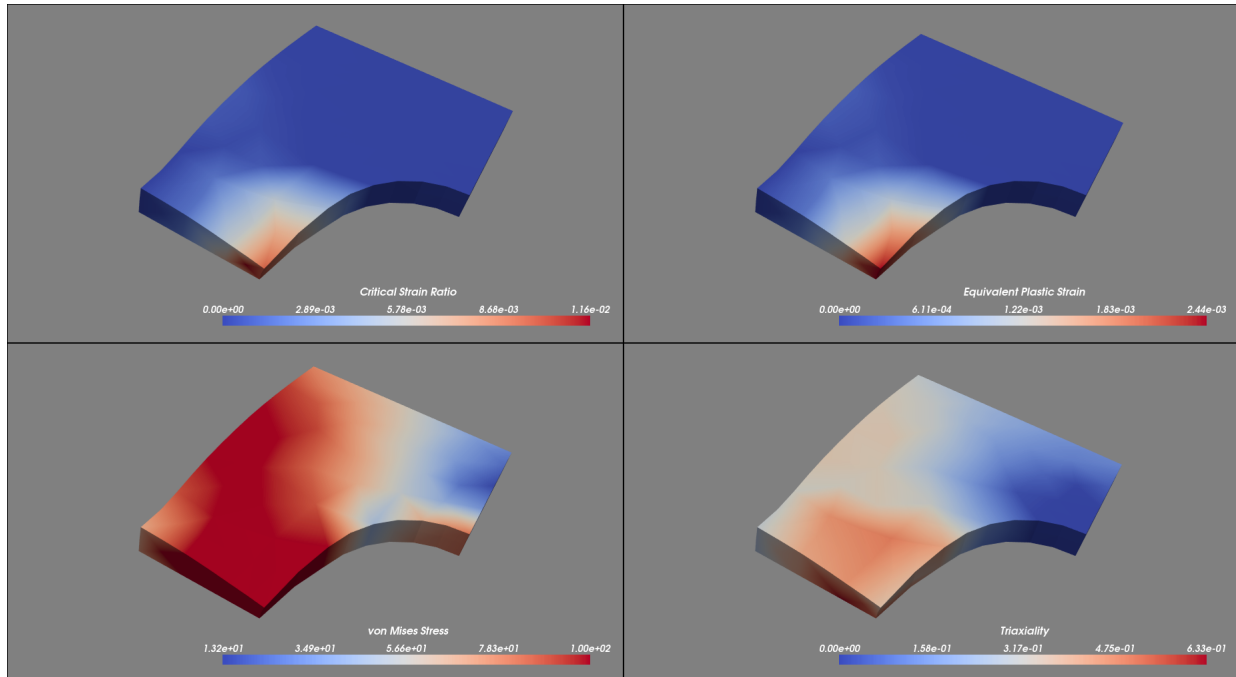
Figure 10. VTK output

Figure 11. PSV output

The load-displacement, -stress and -strain curves for the integration point having the maximum CSR are also printed in the Report View (Figure 12).

```
11:15:57  REACHED TARGET LOAD
11:15:57   ip_max    x-coor    y-coor    z-coor      load        un      peeq pressure   svmises     triax    eps_cr   csr_max
11:15:57        0  1.96e+02  9.15e+01  3.62e+00  0.00e+00  0.00e+00  0.00e+00  0.00e+00  0.00e+00  0.00e+00  0.00e+00  0.00e+00
11:15:57        0  1.96e+02  9.15e+01  3.62e+00  1.00e-01  6.93e-03  0.00e+00  1.68e+00  5.01e+00  1.68e-06  4.12e-01  0.00e+00
11:15:57        0  1.96e+02  9.15e+01  3.62e+00  2.20e-01  1.52e-02  0.00e+00  3.70e+00  1.10e+01  3.70e-06  4.12e-01  0.00e+00
11:15:57        0  1.96e+02  9.15e+01  3.62e+00  3.64e-01  2.52e-02  0.00e+00  6.12e+00  1.82e+01  6.12e-06  4.12e-01  0.00e+00
11:15:57        0  1.96e+02  9.15e+01  3.62e+00  5.37e-01  3.72e-02  0.00e+00  9.03e+00  2.69e+01  9.03e-06  4.12e-01  0.00e+00
11:15:57        0  1.96e+02  9.15e+01  3.62e+00  7.44e-01  5.16e-02  0.00e+00  1.25e+01  3.73e+01  1.25e-05  4.12e-01  0.00e+00
11:15:57        0  1.96e+02  9.15e+01  3.62e+00  9.93e-01  6.88e-02  0.00e+00  1.67e+01  4.97e+01  1.67e-05  4.12e-01  0.00e+00
11:15:57        0  1.96e+02  9.15e+01  3.62e+00  1.20e+00  8.32e-02  0.00e+00  2.17e+01  6.47e+01  2.17e-05  4.12e-01  0.00e+00
```

Figure 12. Load step information for the integration point with maximum CSR

7.3. **Final information in the report view and output files.** After stopping the analysis, the statistics of the analysis are printed in the report view (Figure 13) and the load-displacement, -stress, and -strain curves are printed in the $*.out$ file, located in the *output files* directory.

```
11:16:14  ----------------------------- SUMMARY -----------------------------
11:16:14
11:16:14  extract information from FreeCAD objects......................  0.003 seconds
11:16:14
11:16:14  prepare finite element input.................................  0.064 seconds
11:16:14
11:16:14  calculate the global stiffness matrix and global load vector...  0.023 seconds
11:16:14
11:16:14  solve the global stiffness matrix equation....................  17.127 seconds
11:16:14
11:16:14  map stresses to nodal points.................................  0.012 seconds
11:16:14
11:16:14  paste results in the FEM result object.......................  0.023 seconds
11:16:14
11:16:14  export results to VTK........................................  0.032 seconds
```

Figure 13. Analysis statistics

7.4. **Final results in the model view.** Depending on the analysis type, several result objects are added to the FreeCAD model view. With reference to the terminology of Figure 5, the following objects will appear.

7.4.1. *Elastic analysis.*

- ElasticPlasticDisplacementResults: the elastic deformation at load factor 1.0
- ElasticDisplacementResults: as above

7.4.2. *Elastic-plastic analysis.*

- ElasticPlasticDisplacementResults: the elastic-plastic deformation reached at the end of the final load step
- ElasticDisplacementResults: as per Elastic analysis

7.4.3. *Linear buckling analysis.*

- ElasticPlasticDisplacementResults: as per Elastic analysis
- ElasticDisplacementResults: as per Elastic analysis
- ElasticBucklingShape_lambda=xxx: the buckling shape corresponding to the first buckling load factor xxx
- ElasticBucklingShape_lambda=yyy: the buckling shape corresponding to the second buckling load factor yyy

7.4.4. *Elastic-plastic, geometric nonlinear analysis.*

- ElasticPlasticDisplacementResults: the elastic-plastic geometric nonlinear deformation reached at the end of the final load step
- ElasticDisplacementResults: as per Elastic analysis
- ElasticBucklingShape_lambda1=xxx: as per Linear buckling analysis
- ElasticBucklingShape_lambda2=yyy: as per Linear buckling analysis

7.5. **Export to Paraview.** fcVM automatically exports all rsults to a **.vtk* file, located in the *output files* directory. This file can be imported into Paraview for display of the following results:

- Critical Strain Ratio: scalar
- Equivalent Plastic Strain: scalar
- von Mises Stress: scalar
- Triaxiality: scalar
- Displacement: x, y, z components
- Elastic Displacement: x, y, z components
- Buckling shape for lambda1 = xxx: x, y, z components
- Buckling shape for lambda2 = yyy: x, y, z components
- Stress Tensor: xx, yy, zz, xy, yz, zx components
- Major Principal Stress: scalar
- Intermediate Principal Stress: scalar
- Minor Principal Stress: scalar
- Major Principal Stress Vector: x, y, z components (direction and magnitude of the major principal stress)
- Intermediate Principal Stress Vector: x, y, z components (direction and magnitude of the intermediate principal stress)
- Minor Principal Stress Vector: x, y, z components (direction and magnitude of the minor principal stress)

8. Limitations and Pitfalls

fcVM has some known limitations that should be taken into account when assessing the validity of results

8.1. **Material model.** fcVM uses a bi-linear elastic-plastic material model based on von Mises plasticity. Although widely used in engineering practice it is still a simplification of real material behaviour, especially with complex loading scenarios.

8.1.1. *Hardening Behavior.* The bi-linear model assumes a constant tangent modulus in the plastic region, meaning that after yielding, the material follows a linear strain-hardening path. Real materials exhibit nonlinear strain hardening. This can lead to overestimation or underestimation of stress levels in plastic zones.

8.1.2. *Strain Rate Dependency.* The model used in fcVM does not account for rate-dependent plasticity, making it inaccurate for High-strain-rate problems or creep.

8.1.3. *Bauschinger Effect.* The model only considers isotropic hardening, meaning that plastic deformation increases the yield surface uniformly in all directions. It can therefore not model the Bauschinger effect, where materials exhibit yield stress reduction in reverse loading. This is important in fatigue and forming processes.

8.2. **Tet10 finite element.**

8.2.1. *Locking Issues.* TET10 elements suffer from volumetric locking (overly stiff response) when dealing with nearly incompressible materials (such as plasticity with high Poisson's ratio, e.g., $\nu \approx 0.49$). This should not be an issue for metal plasticity, but may become limiting for incompressible clay.

8.2.2. *Element Distortion.* TET10 elements perform poorly when elements are highly distorted or have poor aspect ratios. GMSH warns against poor aspect ratios.

8.3. **Convergence and unloading.** Convergence of the incremental iterative solution process is not guaranteed, especially for geometric nonlinear problems. This may lead to small load steps and long solution times.

To overcome limit points, fcVM makes use of an arc-length or Riks method. This means that the sometimes the load increments reverse and the structure unloads. In those cases try to run the analysis with a more relaxed error bound.

9. Examples

In this section several examples show the application of fcVM and discuss the significance of the results.

9.1. **Tensile test on plate with hole.** In the first example a steel plate with a central hole is pulled to failure.
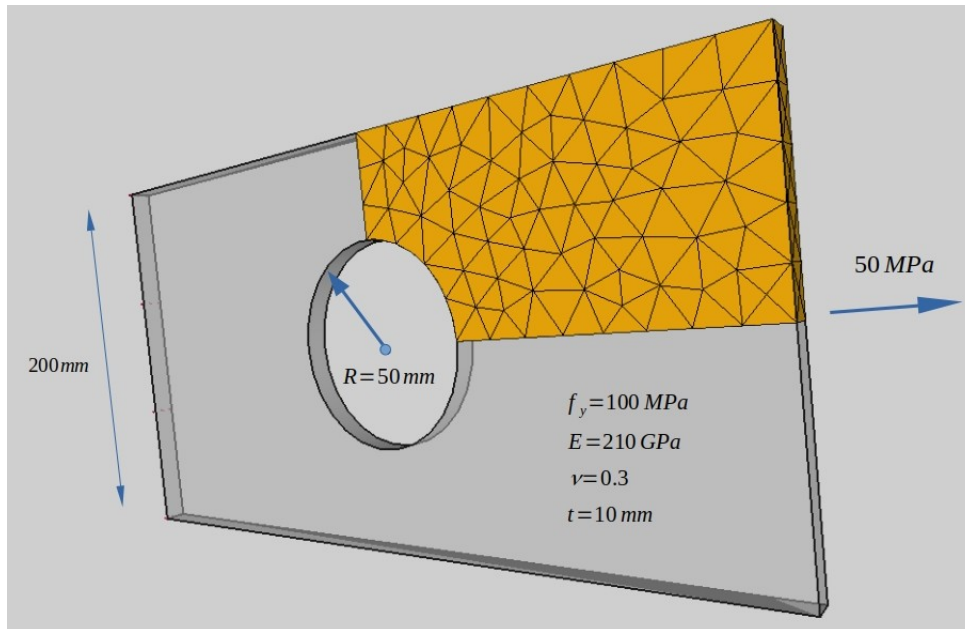
Figure 14. Tensile test on a steel plate with a hole

The settings of fcVM for this geometric linear analysis are shown in Figure 15. The target load factor is set at 1.5, implying that the applied stress is targeted to be $1.5 * 50 = 75 MPa$. The maximum number of steps is 10.

Figure 15. Parameter settings for the tensile test

The load defromation plot shows the result for the first 10 steps. It levels off at a load factor 1.0, indicating impending failure at an applied stress of $1.0 * 50 = 50 MPa$. This is in line with expectation, as the cross section at the hole is half of the full cross section.
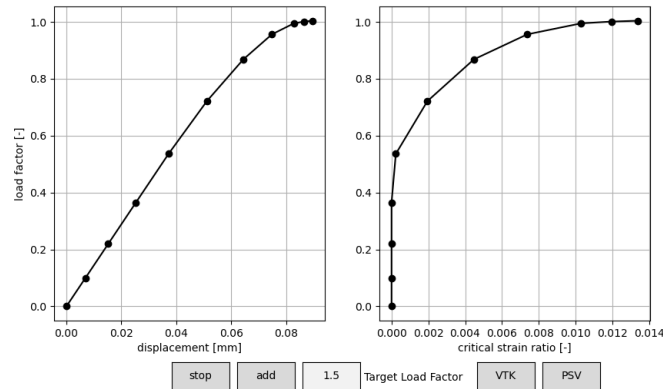
Figure 16. Load deformation curve for the first 10 load steps.

To investigate ductility of the test piece, the $< ADD >$ button is pressed to try and force the load to increase. After repeating this a few times, the load deformation curve of Figure 17 is produced. The blue dotted line shows the last elastic load step and the red dotted line is the point where $CSR = 1.0$, indicating rupture of the test piece.
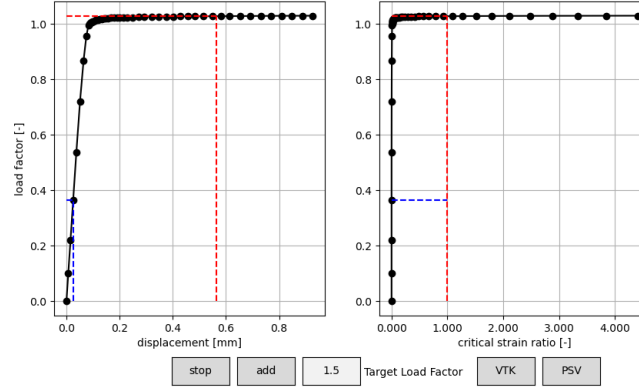
Figure 17. Load deformation curve showing the rupture point

Pressing the $< VTK >$ button produces the plots of Figure 18. The handles that allow cutting of the model have been suppressed by pressing the $< T >$ key. Pressing it again will bring them back. By pressing $< S >$ the plot will be saved as *.png in the *output files* directory. The CSR plot shows that the rupture will initiate on the inside of the hole, where the triaxiality is highest.
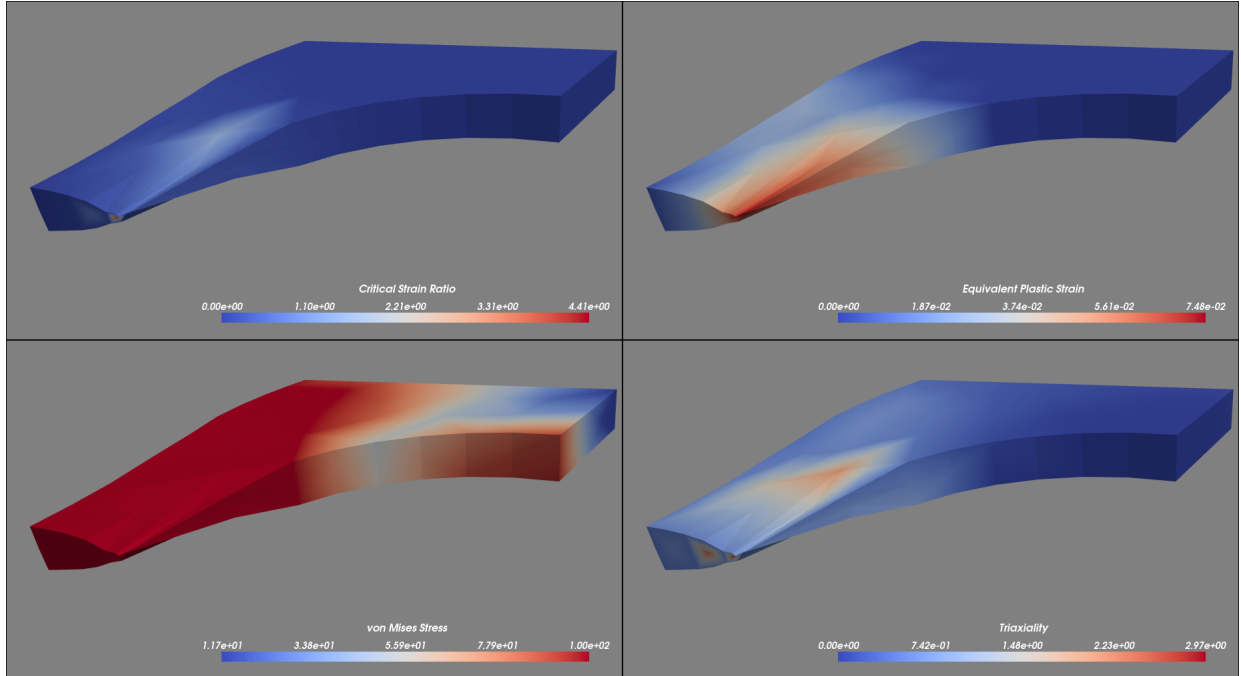
Figure 18. VTK plots

Pressing the $< PSV >$ button will display the principal stress vectors (Figure 19), with tension in red, compression in blue and the intermediate principal stresses in green. This figure also shows the handles for cutting the model.
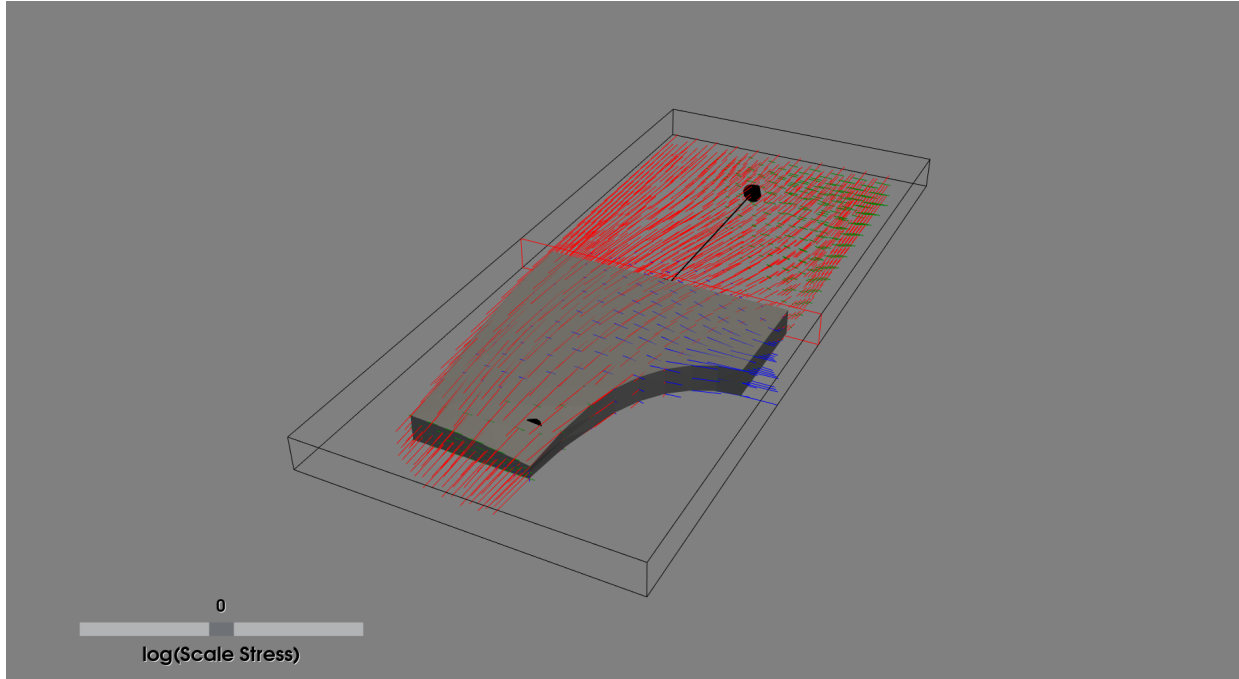
Figure 19. Principal stress vectors.

Finally, Figure 20 shows the deformed mesh with von Mises stress peaking at $f_y = 100MPa$.
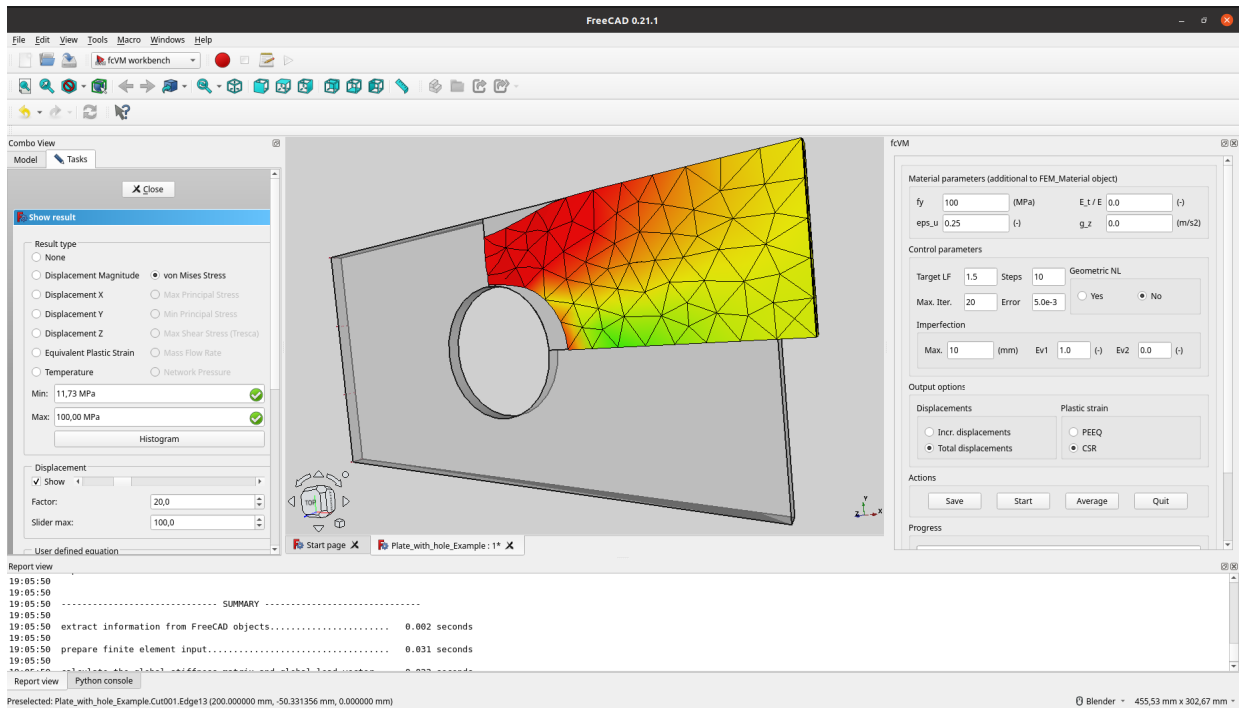
Figure 20. Deformed mesh with von Mises stress

9.2. **Embankment in soft clay.** Figure 21 shows the mesh for an embankment in a soft unconsolidated clay, with an undrained shear strength of $s_u = 5KPa$. To achieve this, the yield strength should be set at $f_y = 0.01MPa$.
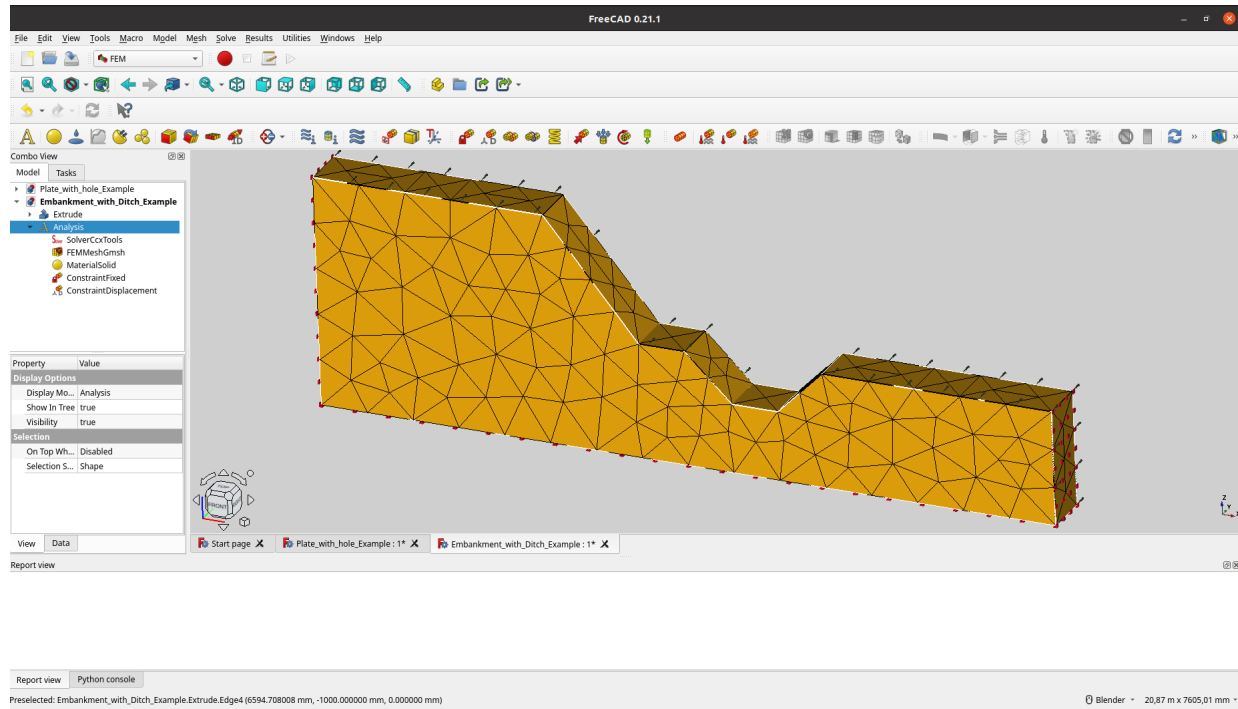
Figure 21. Embankment in soft clay

The self weight of the clay ($20kN/m^3$) is gradually applied to see if the slope is stable (i.e. reaches a load factor of 1.0 or higher). Figure 22 shows that this is not the case. The slope collapses at a load factor of 0.5.
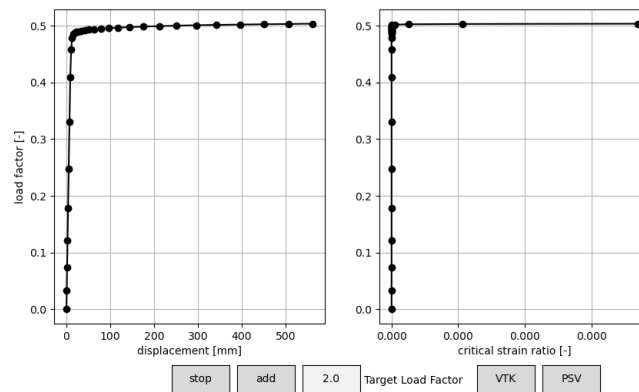
Figure 22. Load deflection curve for an embankment in clay

As the output option for displacement was set to Incremental, the deformed mesh (Figure 23) shows the classical slip circle collapse mechanism.
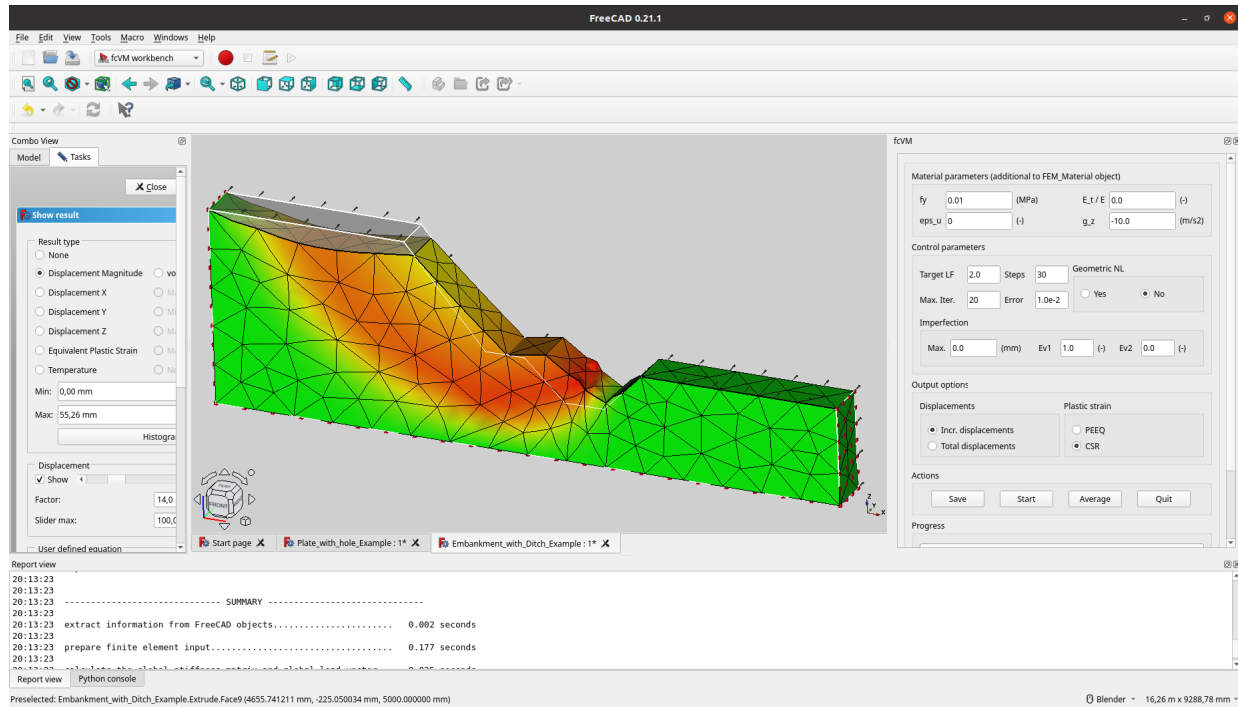
Figure 23. Failure mechanism for an embankment in clay

9.3. **Bar buckling.** A 10x10x1000mm steel bar is loaded at one end and clamped at the other. The theoretic Euler buckling load is $F_{Euler} = \frac{\pi^2 EI}{4L^2} = 432N$. The aspect ratio of the elements is limited to $1:5$.

Figure 24. Cantilever steel bar

To estimate the appropriate target load level for the analayis, first a linear buckling analysis is performed. The $ConstraintForce$ value is set to $1.0N$, the $GeometricNL$ switch to Y and the number of $Steps$ to 1 (Figure 25). The result shows that the calculated Euler buckling load is $434.8N$, which is within 0.7% of the theoretical value.
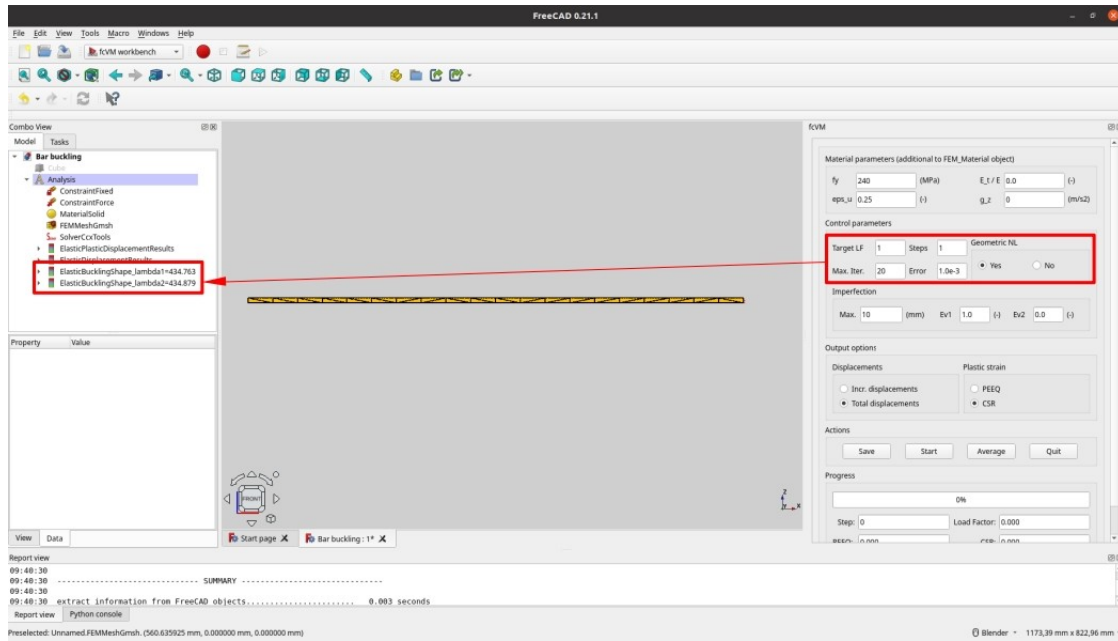
Figure 25. Euler buckling analysis

Next the $ConstraintForce$ value is set to $500.0N$, the $TargetLF$ to 1.0 and the number of steps to 10 (Figure 26). In general, target load factors between 1 and 10 result in good step sizes. The initial imperfection is set to the bar width, i.e. $10mm$.
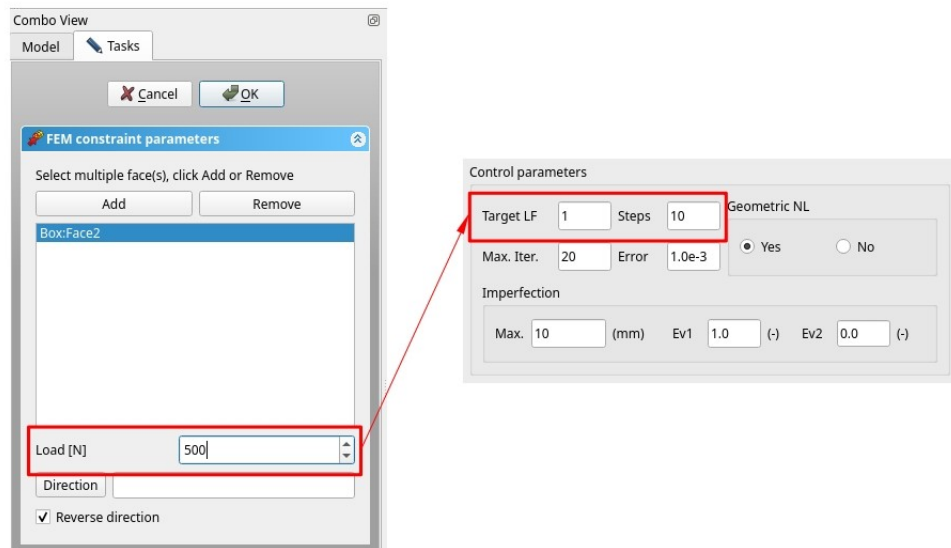

Figure 26. Adjusted Load and Target LF

The load deformation plot (Figure 27) shows a gradual growth of the imperfection, until the onset of pasticity (see CSR plot). After reaching the peak, the strength of the bar gradually reduces.
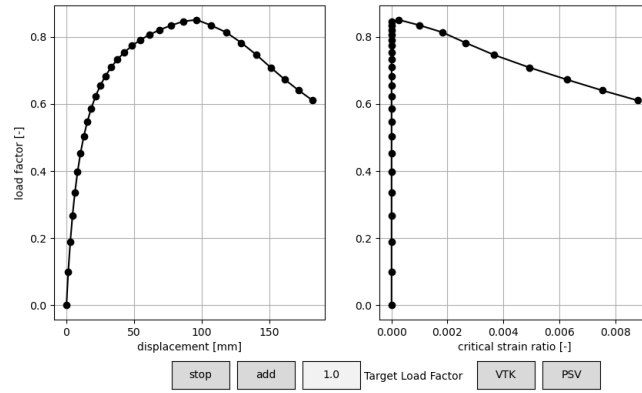
Figure 27. Load deformation plot of a buckling bar

9.4. **Cruciform column.** A steel cruciform column with dimensions as per Figure 28, is clamped at both ends.
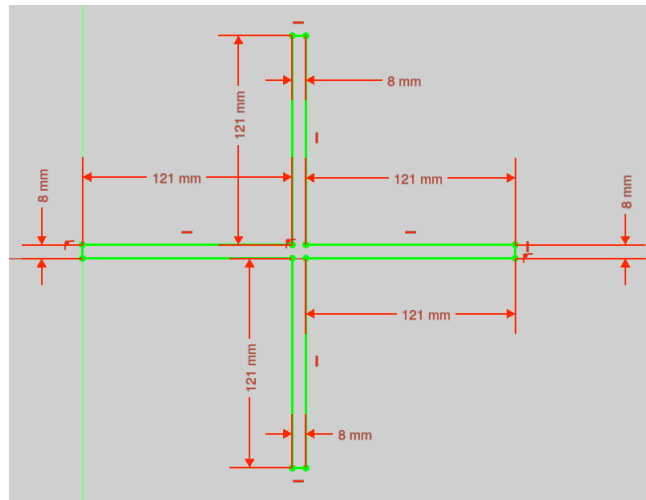

Figure 28. Cross section of a cruciform column

The free end of the column is loaded by a $1MN$ load and supported as per Figure 29. An Euler buckling analysis ($GeometricNL = Y$ and $Steps = 1$) shows that the buckling load is $1.4MN$ (Figure 29).
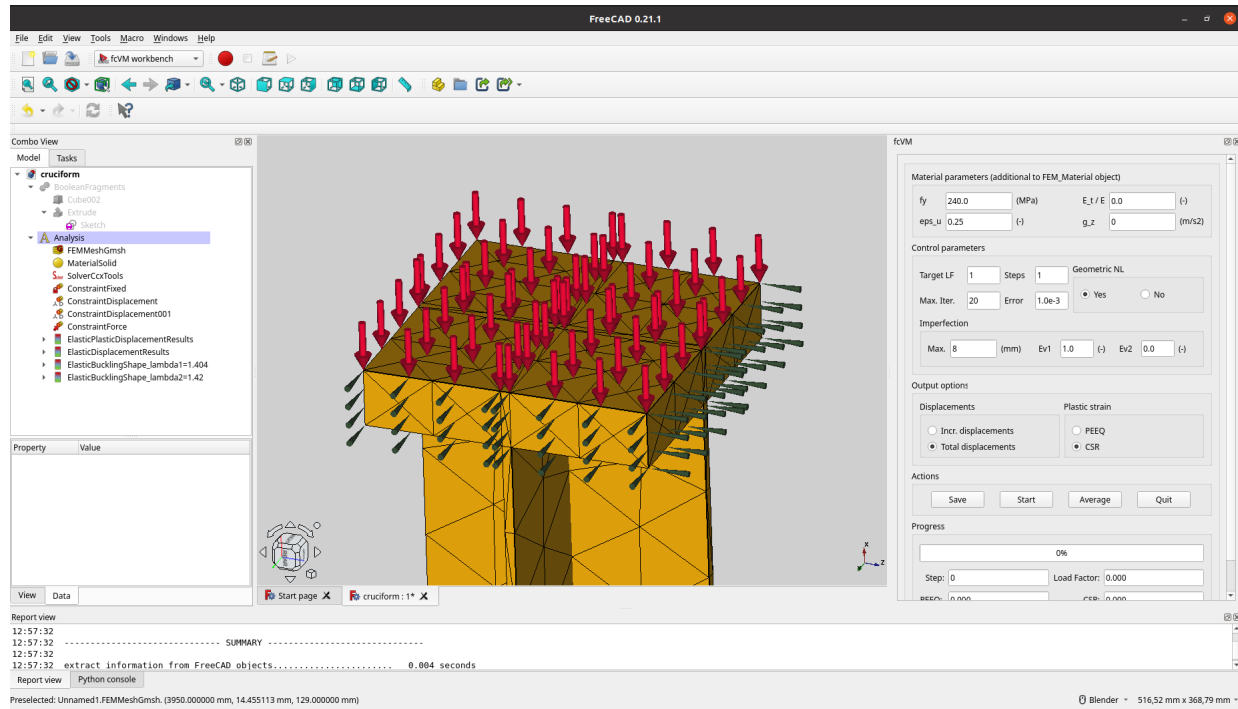
Figure 29. Detail of the column support and loading condition

After setting the initial $Imperfection\ Max. = 8mm$ (the web thickness) and increasing the *Steps* parameter to 10, the analysis is incremented 3 times to produce the load deflection curve of Figure 30. A slight drop in load is caused by the switch from an unstable load path to a stable load path.
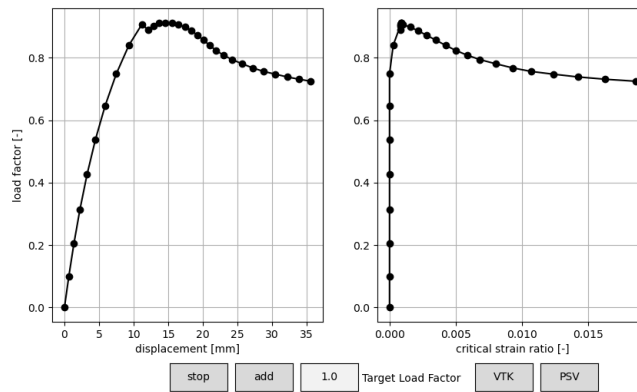

Figure 30. Load deflection curve for a cruciform column

Figure 31 clearly shows the post-peak torsional buckling of the column. The ultimate load is slightly below the pastic limit load of the column.
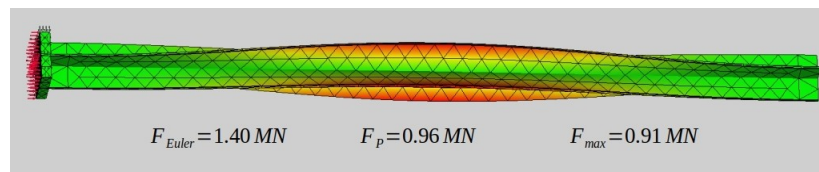

$$F_{Euler} = 1.40\,MN \qquad F_P = 0.96\,MN \qquad F_{max} = 0.91\,MN$$

Figure 31. Post-buckling deformed mesh