

CALCULIX LAUNCHER

VERSION 0.32

<http://calculix.de/>

<http://calculixforwin.blogspot.com/>

<http://www.calculixforwin.com/>

Table of Contents

Installation notes.....	2
General Notes About Meshers and Mesh Formats.....	4
Settings for GMSH.....	5
Units in CalculiX.....	6
How to get started. Learning Curve For CalculiX.....	8
How CalculiX Works.....	9
Main Principles. How To Create Model And Apply Boundary Conditions.....	13
Quality Of Mesh.....	14
Free Structural Software.....	19

Installation notes

CalculiX Launcher is the collection of portable software, that includes:

- CalculiX CCX (finite element solver);
- CalculiX CGX (pre-processor and post-processor for CCX);
- GMSH (simple parametric 3d program and mesher);
- Mesh converters from UNV, MSH, Abaqus INP, VOL formats to CCX INP format.

All programs (binaries) are placed under ../bin sub-folder. Some of them are statically linked or require some common libraries (OpenGL and GTK2 for linux), that makes launcher very portable.

There is no special installation procedure.

Xterm terminal emulator should be installed in the system (Launcher uses it by default). Some linux distributions don't have xterm by default, so install it, as usual (in ubuntu 'sudo apt-get install xterm')

Unzip archive to any place, you want.

There is one important note, that the full path to the launcher folder should not contain spaces (blanks). For example:

```
/home/JohnSmith/Desktop/CL32-linux  
and  
/home/John_Smith/Desktop/CL32-linux  
are OK,
```

```
but  
/home/John Smith/Desktop/CL32-linux  
is not.
```

(To see or copy the full path in Ubuntu, press "Ctrl+L" in the active folder)

VERY IMPORTANT NOTE:

DON'T USE BLANKS, CYRILLIC AND ETC SYMBOLS IN THE NAMES OF FILES, FOLDERS, WHEN YOU WORK WITH LAUNCHER. FILES WON'T BE RECOGNIZED PROPERLY.

Second step is making all binaries executable. This is the standard procedure for linux systems. At first – make executable the file, called "Launcher", then go to the ../bin folder and do the same with all files.

Select binary file with mouse, then make right click and select "Properties" and go to "Permissions" tab. Allow this file to run as program (see Fig. 1) below. Do it with all files!

Also you may cd.. to the /bin folder and type

```
chmod 777 *
```

It will mark all files as executable.

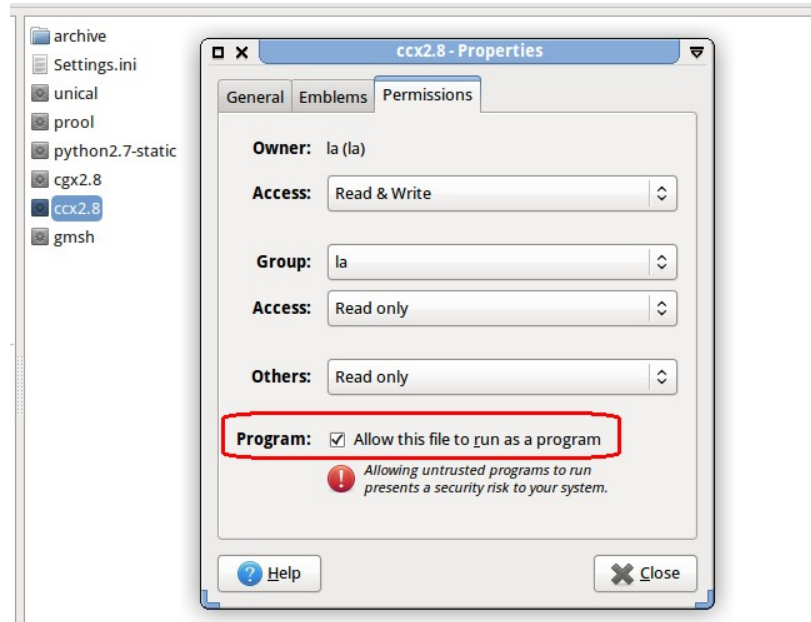


Fig. 1 – Making file executable.

Launcher comes with simple text editor for CalculiX (with code highlighting and snippets for boundary conditions), but you may use any other popular text and code editors, for example gedit, scite and etc. Run launcher, go to menu > Settings and fill in 4th line from above.

Default terminal emulator is x-term, but you can install and use xfce4-terminal. In ubuntu install it with:

```
sudo apt-get install xfce4-terminal
```

Run launcher, go to menu > Settings and type 'xfce4-terminal' in 3d line from above.

Most preferable configuration of hardware and operational system is 64 bit. It gives you ability to solve large problems without limitation by memory. If you want calculation to be performed in a parallel way (for multi-cores processors), create environment variables OMP_NUM_THREADS with value equal to number of cores. To setup environment variable permanently in ubuntu, open terminal emulator and type:

```
sudo gedit ~/.pam_environment
```

and then create two lines in the text file:

```
OMP_NUM_THREADS=2
export OMP_NUM_THREADS
```

You can create this file

After re-log-in the calculation will run a parallel way (you will be able to see it in output for CCX).

For more information read CCX help 'How to perform CalculiX calculations in parallel'.

You can find more information about the Calculix Launcher at this page:

<http://calculixforwin.blogspot.com/2015/05/calculix-launcher.html>

General Notes About Meshers and Mesh Formats

GMSH is open-source pre-postprocessor (finite-element generator) developed by Christophe Geuzaine and Jean-François Remacle.

<http://gmsht.info/>

GMSH can be used as GUI application or console application (you can run it from command line)

It has ability to export mesh with INP extension, but it produces different names for element types, than CCX and you need additional program (mesh converter) to make it valid for CCX input. There are 3 types of mesh formats can be transferred from GMSH to CalculiX:

- Abaqus INP can be converted to CalculiX INP with python scripts and with using free converter, coded by Sergey Prool (Kharkiv, Ukraine);
- UNV format is supported by dip28 converter (python scripts). It works for tetrahedral and triangular mesh with nodal groups;
- MSH format (with python script).

The most powerful CAD and Mesh package is Salome-Platform.

Then you may get it at the:

<http://www.code-aster.org/V2/spip.php?article303>

<http://www.alneos.com/downloads/salome-meca-linux-32-en>

<http://www.salome-platform.org/>

Read more about that and see video-lessons at here:

<http://calculixforwin.blogspot.com/2015/12/salome-platform-and-salome-meca.html>

Output mesh format from Salome is UNV format. It works with 'unical' converter for many types of mesh. Also you may try to use combination: MED format > GMSH > CalculiX.

Netgen mesh generator (for 3d models with merged bodies) usually can be installed from Synaptic Package Manager. It is also available in caelinux2013 distribution. Netgen have export to the abaqus INP and VOL format. VOL is supported natively in CalculiX CGX (read CGX help for that, how to deal with CAD models) and with python script, provided by dip28 (3d tetras).

In any case you may need to open converted file and correct some output manually. For example output element type for unical converter can be changed from S8R (multilayer shells for elastic and plastic analysis) to CPS8 (plane stress).

To play with mesh conversion you can use files in the '.../samples/Converters' folder.

Settings for GMSH

Run GMSH from Launcher menu, go to Tools>Options>General-Advanced (see Fig. 2).

Setup text editor for text file (if you want to use parametric features in future). On Fig. 2 gedit is selected as default Ubuntu editor. When you start GMSH, it creates file 'untilted.geo' for writing all your actions in the command mode. You can use this feature to make parametric models with variables. But sometimes, if you will make a mistake in the code, GMSH will crash at start. In this case you can remember the default place for 'untilted.geo' and delete (or rename it), to restore normal work.

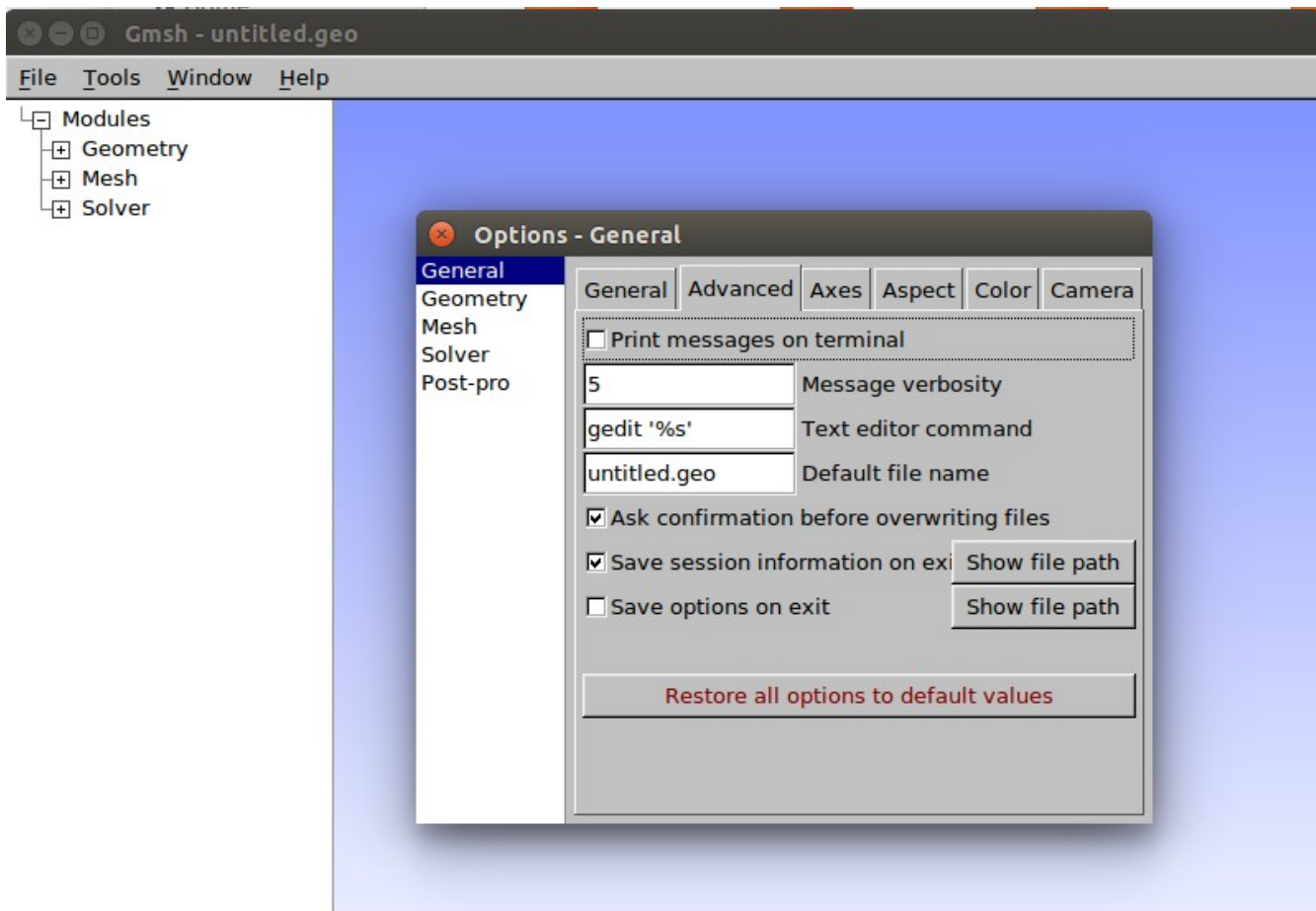


Fig. 2 – GMSH Options

Units in CalculiX.

CalculiX is old-school classical FEA code, and it doesn't have any certain «units» inside.

It means, that you can use any units, whatever you want, but the units should be compatible with each other.

The most simple way – to **use SI system**, just make sure that your model scaled properly and all linear dimensions of your mesh are in meters.

When you provide some input for the program, all values can be dimensionless or have some units. It is absolutely the same when you use any abstract formulas.

For example the deflection of a beam of span “L”, under the distributed load “q”, with cross section moment of inertia equal to “J” and Young's Modulus of material equal to “E” is given by the formula (bending part of full deflection).

$$y = \frac{qL^4}{8EJ}$$

There are no units that come with this kind of formulas, but you may vary with them, staying within one system for each type of value.

Let's calculate the rectangular steel beam 1x0.75 in (width=1'', height=3/4''), with span equal to 3 ft for distributed load equal to 18 lb/ft:

$$L = 3 \text{ ft} = 3 \times 12 = 36 \text{ in};$$

$$q = 18 \text{ lb/ft} = \frac{18 \text{ lb}}{12 \text{ in}} = 1.5 \text{ lb/in};$$

$$\text{Young's Modulus } E = 29,000,000 \text{ lb/in}^2;$$

$$J = \frac{bh^3}{12} = \frac{1 \times 0.75^3}{12} = 0.03516 \text{ in}^4;$$

$$y = \frac{1.5 \times 36^4}{8 \times 29,000,000 \times 0.03516} = 0.309 \text{ in (or 7.8mm, or 0.0078 m)}$$

Now let's do the same with SI system:

$$L = 0.0254 \times 36 = 0.914 \text{ m};$$

$$q = 1.5 \text{ lb/in} = 4.448 \text{ (N/lb)} \times 1.5 \text{ lb} / (0.0254 \text{ m}) = 262.7 \text{ N/m};$$

$$E = 29 \times 10^6 \text{ lb/in}^2 = 2 \times 10^{11} \text{ Pa};$$

$$y = \frac{262.7 \times 0.914^4}{8 \times 2 \times 10^{11} \times 1.463 \times 10^{-8}} = 0.0078 \text{ m}$$

in⁴ can be converted to m⁴ using simple proportion:

$$1 \text{ in} \rightarrow 0.0254 \text{ m};$$

$$1 \text{ in}^4 \rightarrow (0.0254)^4 \text{ m}^4 = 4.162 \times 10^{-7} \text{ m}^4;$$

$$0.03516 \text{ in}^4 \Rightarrow 4.162 \times 10^{-7} \times 0.03516 = 1.463 \times 10^{-8} \text{ m}^4.$$

When you will calculate this beam in CalculiX CCX, make sure that the mesh is properly scaled in «inches» and apply Young's modulus 29e6 psi at the *ELASTIC card.

It is little bit not so easy, when you are making modal analysis, because there is the difference in «pound-force» and «pound mass» in terms of Newton's law definitions (F=ma).

When using imperial units in CalculiX, the typical material card will be (for steel):

```
*MATERIAL,NAME=Steel
```

```
*ELASTIC
```

```
2.9000E+07,0.3
```

```
*DENSITY
```

```
7.3500E-04
```

(where the steel density 0.284 lb/in³ is corrected by gravity acceleration =386.4 in/s² ; 0.284/386.4= 0.000735)

Table 1 – Input data for mild steel, depend on unit system and units for

linear dimension (mesh)

Units for mesh, to be scaled	meters	inch	ft	mm
Young's Modulus (or stress units)	2e11 Pa	2.9e7 psi	4.18e9 psf	2e5 N/mm2= MPa
Poisson Ratio	0.3	0.3	0.3	0.3
Density for modal analysis	7850 kg/m3	7.35e-4 lbf/in3	15.25 lbf/ft3	7.85e-9 Ns ² /mm4
Density	7850 kg/m3	0.284 lbf/in3	490.5 lbf/ft3	7.85e-9 Ns ² /mm4
Gravity Acceleration	9.8 m/s2	386.4 in/s2	32.17 ft/s2	9800 mm/s2
Thermal Expansion	1.2e-5 1/degC	6.67e-6 1/degF	6.67e-6 1/degF	1.2e-5 1/degK
Thermal Conductivity	60.5 W/(m*degC)	8.09e-4 BTU/s in F	9.71e-3 BTU/s in F	60.5 N/(s*degK)
Specific Heat	434 J/(kg*degC)	0.1036 BTU/lbm*F	0.1036 BTU/lbm*F	434e6 mm ² /(s ² *degK)

How to get started. Learning Curve For CalculiX

Learning of CalculiX gives many potential benefits even if finally you won't use it or buy some commercial program (or apply for on-line service such us [simscale](#))

Finite element analysis can be applied in many practical applications, not only engineering ones but biotechnologies, medical problems, architectural innovations and etc.

The way of learning CalculiX depends on initial knowledge. Some people may get into troubles, because they don't really understand how computer program works, some people need better understanding of numerical methods and etc.

Here is approximate learning curve table, depend on what kind of experience you have and what kind of problems are interesting for you.

<p>I have good experience with FEA software of old style (before 1995-2000), understand console applications and main principles of text file interface.</p>	<p>In this case you can start directly with CalculiX CCX and CGX help system and verification samples. This collection of examples can be useful for you: https://github.com/mkraska/CalculiX-Examples</p>
<p>I have good experience in FEA software but with using modern interfaces and CAD. Text file interface in too hard for me. Nothing appears when I double click on program.</p>	<p>You need some initial understanding of how console applications work. Go to "How CalculiX Works" section. This is pretty much style of how people were dealing with old DOS and UNIX apps, even they were commercial and modern for their time.</p>
<p>I don't have experience with FEA software, but I know physics of my problem.</p>	<p>Go to "Main Principles. How To Create Model And Apply Boundary Conditions"</p>
<p>I am not familiar with FEA, but I want to calculate barn and check stress in the girder.</p>	<p>Probably CalculiX is not the program you really need. You can exploit your native redneck's intuition, what is needed even for professionals, when they are taking final decision, what is good and what is not applicable. Also, there are many free templates and web services for you. Go to section "Free Structural Software"</p>

The ways to learn CalculiX are:

- 1) Read help files and learn examples.
- 2) See the video lessons:
<https://www.youtube.com/user/calculix09/videos>
- 3) Solve your own problems and verification examples.
- 4) Learn GMSH and Salome-Platform.

How CalculiX Works

CalculiX includes two independent programs: CCX is solver - console application with text file interface, CGX - is pre-postprocessor with command and graphical user interface.

Visually CCX includes binary executable file and some libraries (dll files at Windows, stored at the same folder with ccx.exe). When running it reads input file (usually have .INP or .inp extension),

recognize data, starting from numbers and coordinates of nodes under *NODE card, elements, groups (sets) and finally - boundary conditions and specific instructions within *STEP card.
For example, you have text file test.inp, with following text inside.

```
**Node numbers and coordinates
*NODE,NSET=Nall
1,    0,    0,
2,    1.5,  0,
3,    0,    -1,
  **Truss elements, type T3D2 (valid for CCX >2.11)
*ELEMENT,TYPE=T3D2,ELSET=EAll
1,    1,    2
2,    2,    3

**Material properties, Young's Modulus E=2e11 Pa, Poisson ratio=.3
*MATERIAL,NAME=STEEL
*ELASTIC
2e11,.3

**Cross section of each truss element, Area=0.01
*SOLID SECTION,ELSET=EAll,MATERIAL=STEEL
1e-2

**Define nodal set for supports
*nset,NSET=support
1,
3

**Fix nodes of set in all three directions (X,Y,Z)
*BOUNDARY
support,1,3

*STEP
*STATIC

**Apply load 1e5 N on node #2 opposite to Y direction (2)
*CLOAD
2,2,-1e5

**Request for output
**Reactions in DAT text file
*NODE PRINT,nset=support
RF
**Deflection and stress in FRD file for CGX
*NODE FILE,OUTPUT=3D
U
*EL FILE
S
*END STEP
```

This is simple bracket of two elements, you can draw it (starting from nodes) on sheet of paper.

It has 3 nodes and two truss elements. In CGX preprocessor it looks like shown on picture below (after entering commands "plus ea all" and "plus na all").

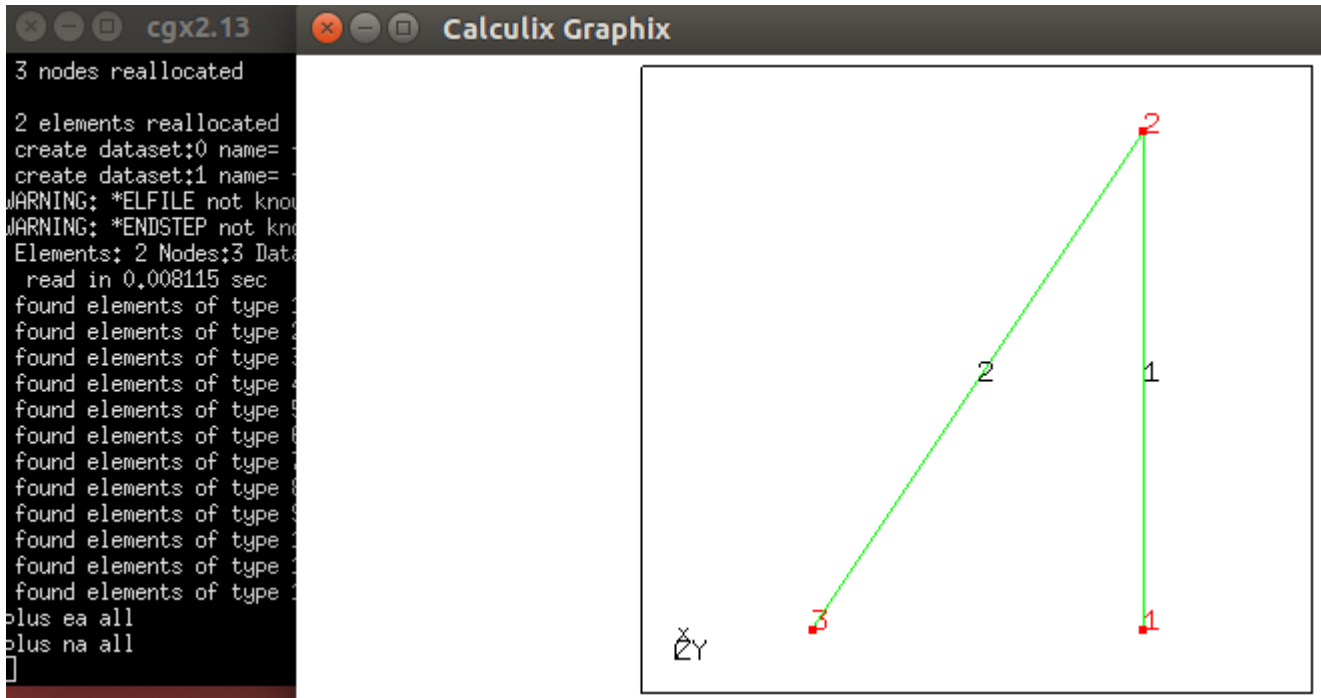


Fig.3

Let's save this file at some folder, also called working folder and run solver in any terminal emulator (in Windows it will be standard cmd application)

At the beginning type "cd path" to get into working folder with stored CCX executable (diamond icon on picture below) and then type

```
ccx test
```

If ccx executable is placed outside, you have to type path to ccx instead of ccx. Finally you will see some output and message "Job is finished"

In the beginning we had only 2 files: ccx and test.inp, but after calculation many other files has been created (by ccx). Two of them are most interesting: test.dat – is text file with reaction forces (requested by *NODE PRINT card) and test.frd – also text file, but not very readable. This file is used for post-processor and can be run by CGX

```
path_to_cgx_exe -v test.frd
```

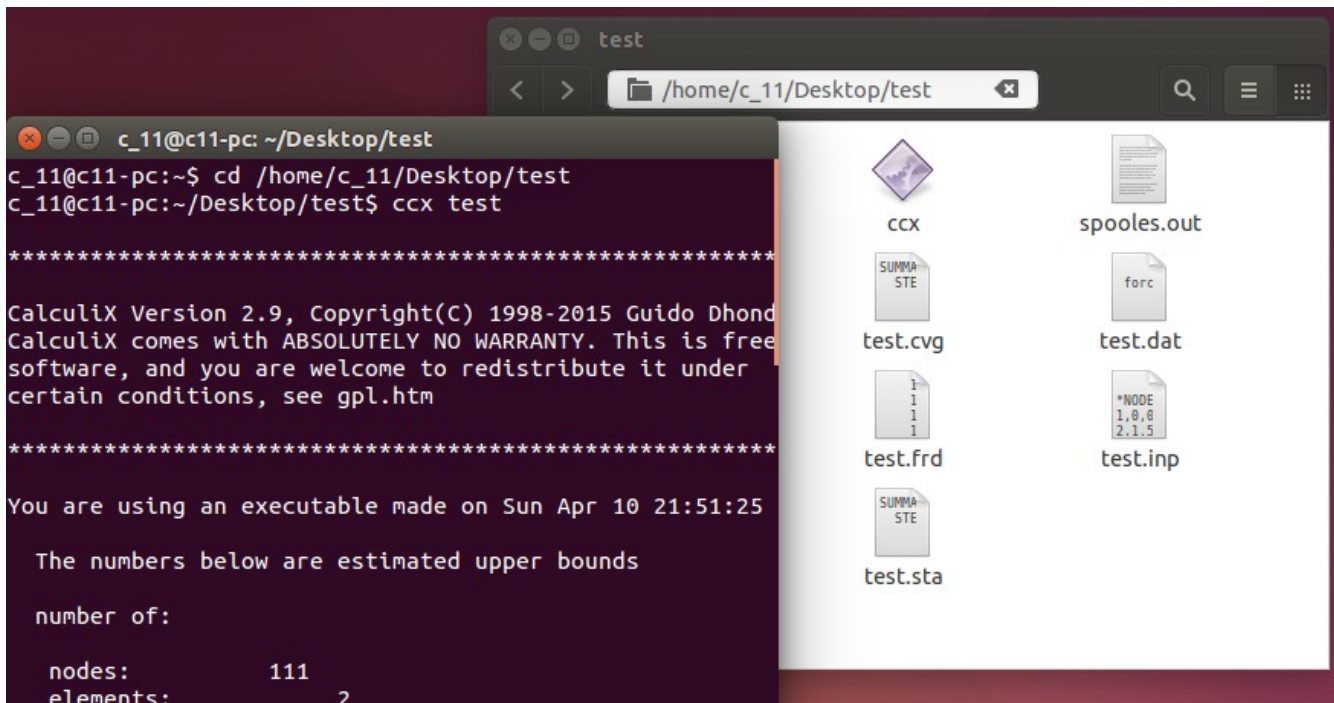


Fig. 4 – Files, created by CCX

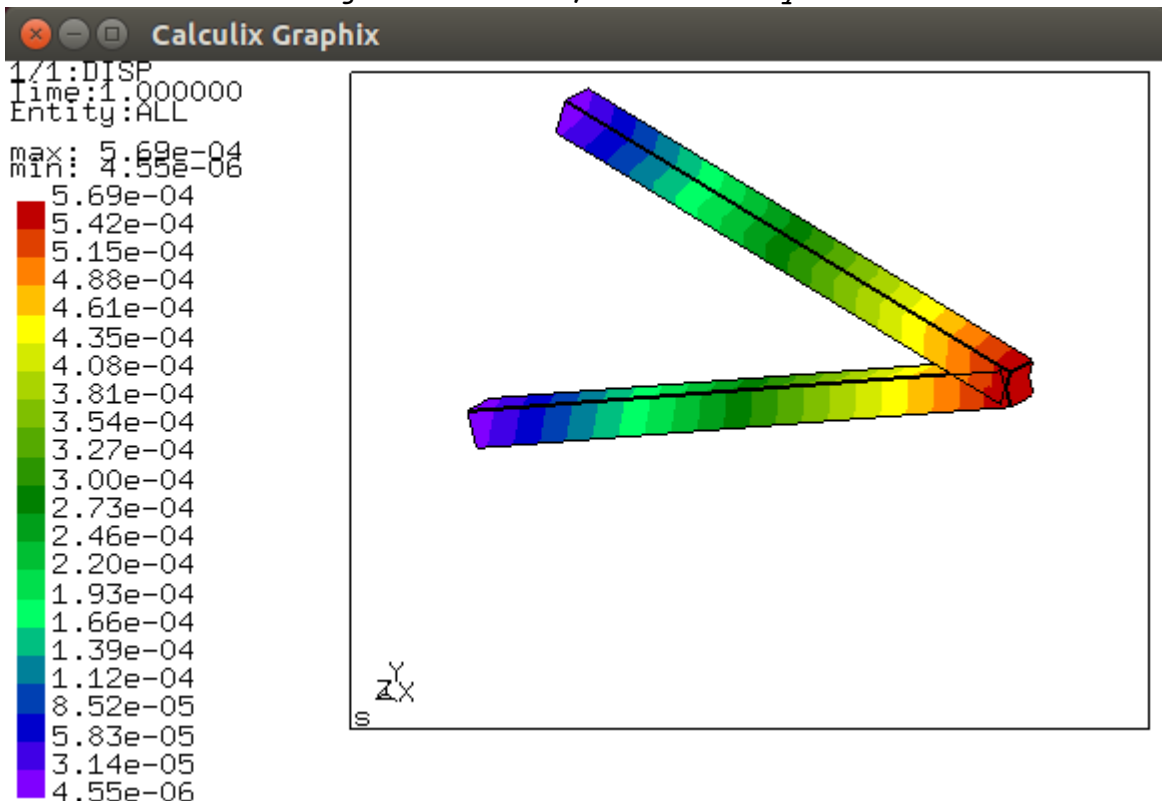


Fig. 5 – Deflection in CalculiX CGX (post-processor)

Main Principles. How To Create Model And Apply Boundary Conditions

In the example above, the model has been created manually in text editor. Practically it is not possible, when you have complex model with many elements. But before calculation you should obtain the same kind of text input file. You can place some statements and cards outside input file (in the other files). In this case you have to add reference to the additional files:

```
*INCLUDE, INPUT=Mesh_1_OUT.inp
```

where Mesh_1_OUT.inp is the file, placed at the same folder.

Usually most complex part of preprocessing is creating mesh and sets (groups) for boundary conditions. It can be done in CGX, GMSH. Salome-Platform, Netgen or any other (free or commercial) preprocessor.

Calculix CGX has capabilities of CAD program, what means, that you have ability to create CAD model, with points, lines, faces, volumes (bodies) and mesh it. It has very efficient mesher for creating structured mesh, such as in example on official web-site:

<http://calculix.de/fw05.html>

<http://calculix.de/fw05lav.html>

<http://calculix.de/fw05lat.html>

Very detailed examples provided by Prof. Martin Kraska:

<https://github.com/mkraska/CalculiX-Examples>

For people who completely spoiled by modern graphical interfaces you can learn very detailed guide, created by Jeff Baylor.

<http://www.bconverged.com/content/calculix/doc/GettingStarted.pdf>

Salome-Platform is most powerful open-source CAD and mesher.

You can learn it, using [youtube lessons](#).

Initially it is native linux software, but now has ports to Windows (not recommended due to many bugs). More advanced version of Salome-Platform, called Salome_MECA is for linux only, and it has direct connection with Code_Aster, industrial class of FEA program.

The main steps of creating the mesh in both programs are:

1. Create CAD model or import it from any free or commercial software

(using STEP or IGES formats).

2. Prepare CAD model for meshing (scale it, make some treatment to remove degenerated entities, apply partition operation (Boolean fragments in GMSH) to obtain monolithic mesh for coincident bodies).
3. Create groups by selection of entities (points, lines, faces, bodies).
4. Setup meshing options and parameters and mesh it.
5. Check mesh quality (aspect ratio, Jacobian for curved parts of model), refine it (re-mesh) if needed.
6. Export created mesh using UNV, MSH or INP type of mesh format. In Salome-Platform you have to pull groups from Geometry to Mesh module .
7. Use converter to obtain mesh format compatible with CalculiX. Finally you can open mesh in CalculiX CGX in pre-processor mode, check it and apply some boundary conditions using 'send' card:

```
path_to_cgx_exe -c mesh.INP
```

Quality Of Mesh

There is important practical question: how fine the mesh should be for certain task?

It depends on type of problem and what are you actually looking for. The goal of static calculation is investigation of stress state. Any practical model has stress concentrators with high stress values and singularity points where maximal stress is infinite value by theory (in numerical analysis it will grow along with mesh refinement). Also any weld seam has many welding defects (sharp stress concentrators) and these are also singularity points, which cannot be investigated based on stress values. This thing called local stress state and it is interesting only in terms of fatigue analysis for cyclic loading (usually $> 20,000$ cycles in the definition of AISC manual). Many technical codes provide stress limits for general stress state (it is also called "fiber stress"). Investigation of general stress state is subject of structural mechanics. For example beam under the bending has max. stress, that can be calculated by formula $\text{"Max. bending moment"} / \text{"Section Modulus"} = M/S$. It will give you fiber stress in outer fibers. This stress is important in terms of structural capacity. If you will drill little hole in the flange on the beam, maximal (local) stress will be about 3 times more, than fiber, but it won't affect capacity (only max number of cycles in terms of fatigue calculation, if the stress is positive in the tensioned side of beam). When you're looking for local stress state, you should make fine mesh to obtain correct results.

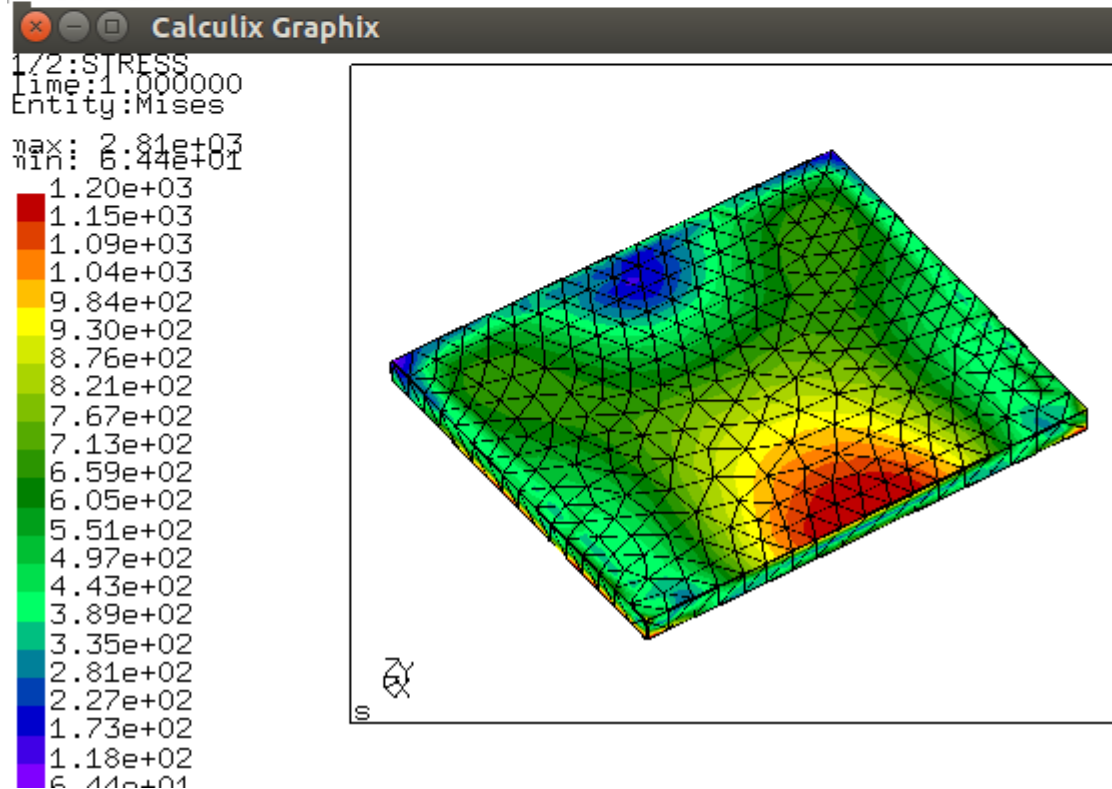
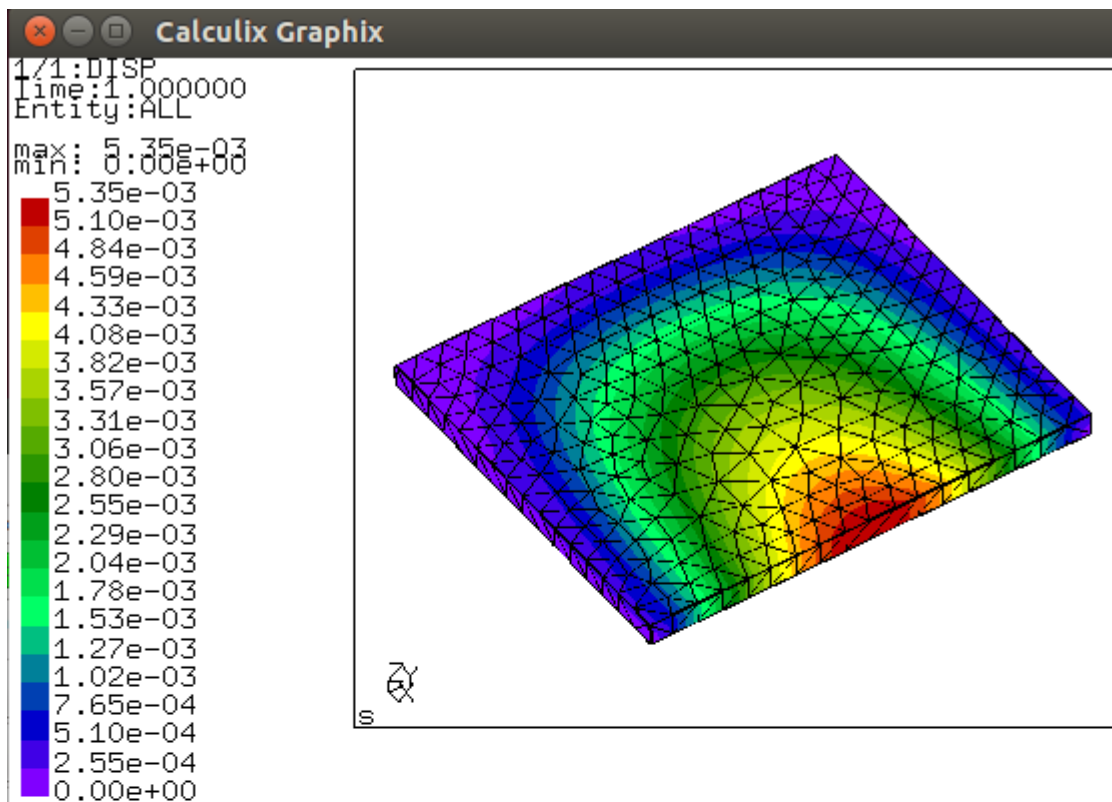


Fig. 6 Rough 3d tetrahedral mesh: max displacement $5.35e-3$ inch
Stress distribution shown on picture below.

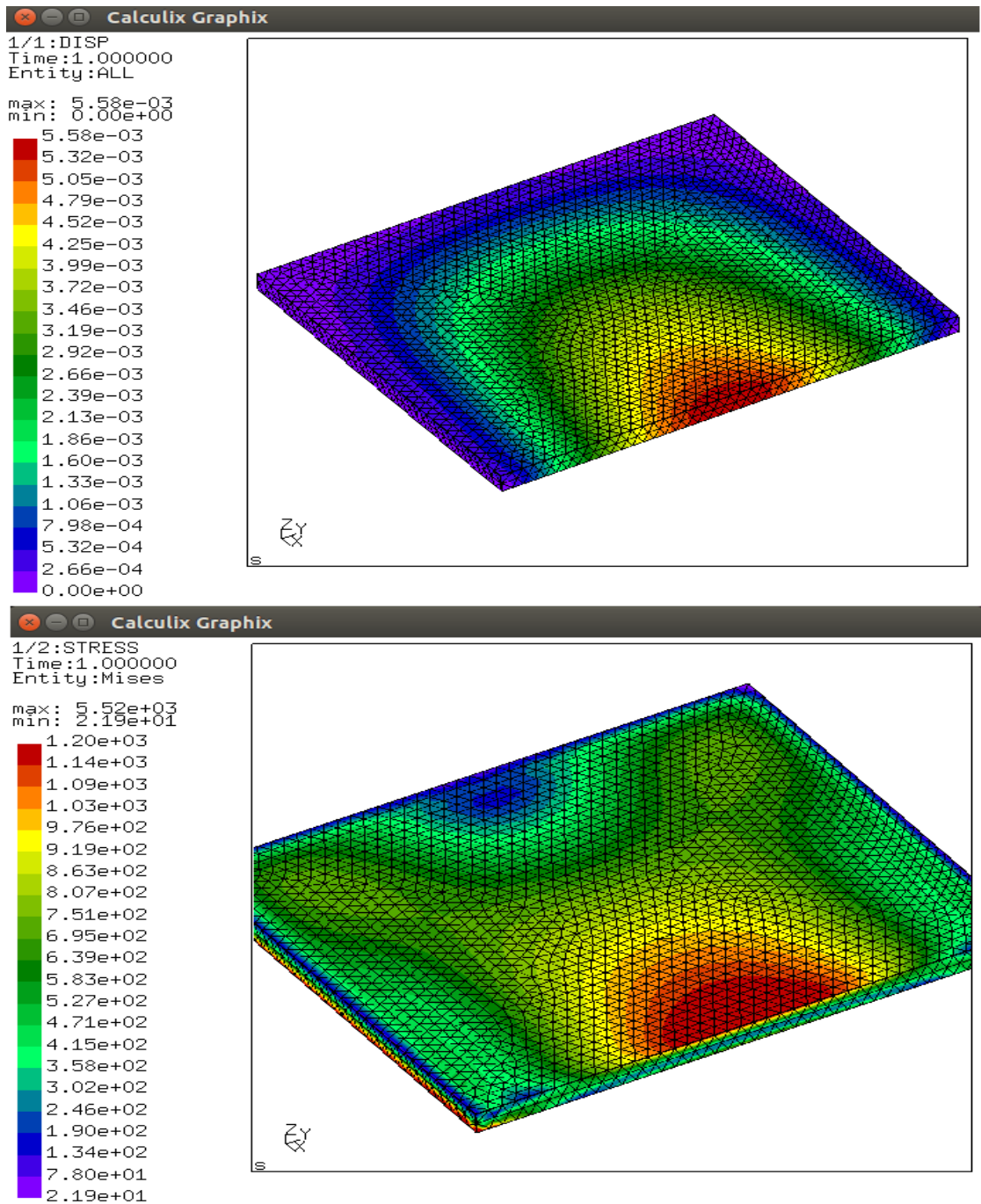


Fig. 7 - Fine 3d tetrahedral mesh (2 elements per thickness): max displacement $5.58e-3$ inch. Difference is about 4% with rough meshing (not critical in most applications)
Stress distribution is similar to rough mesh. Maximal stress is in singularity point for this model and vary with mesh quality

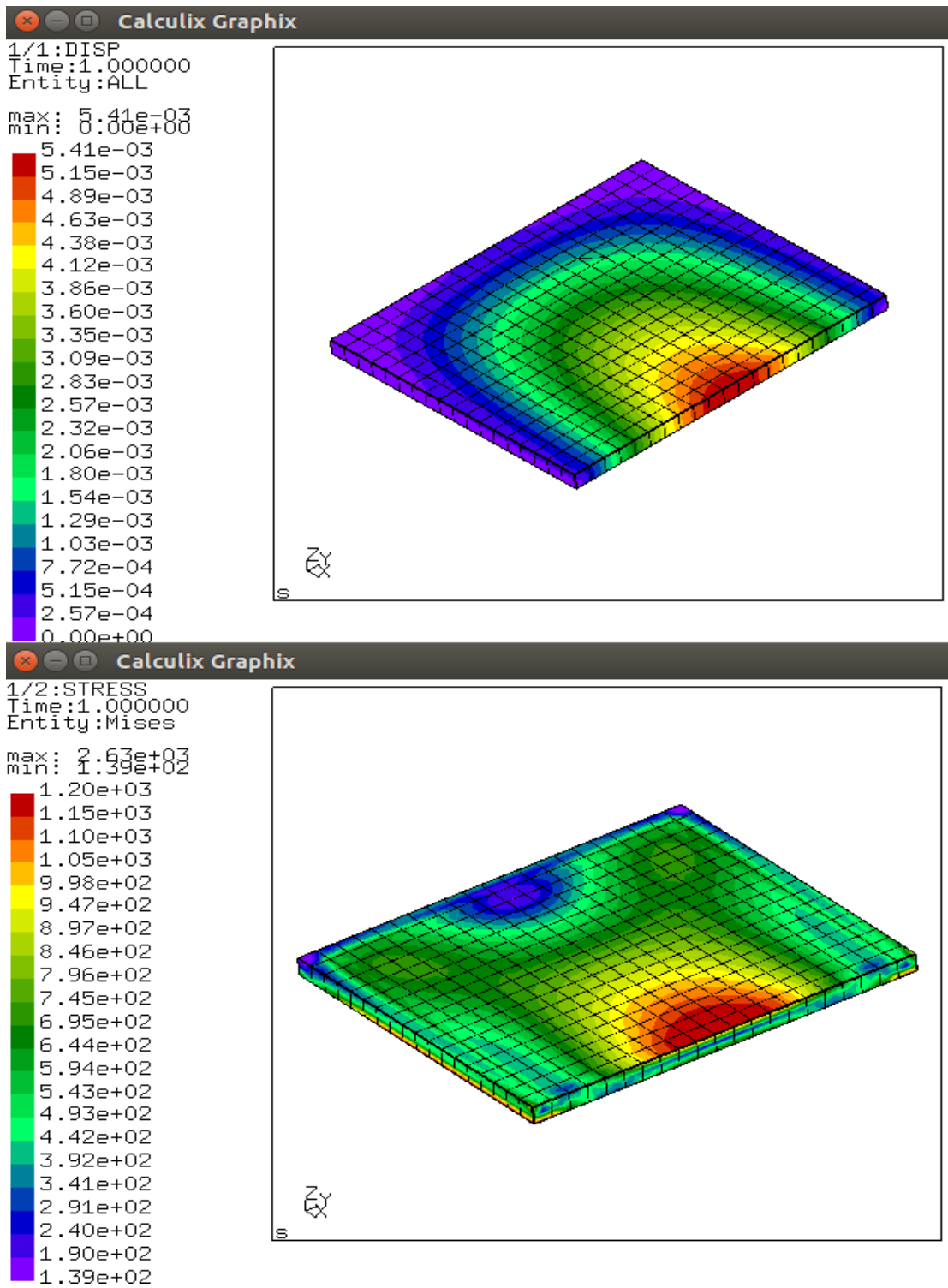


Fig. 8 - Structured C3D20 type mesh: max displacement 5.41e-3 inch.

Modal analysis usually make sense to determine natural frequencies and check if they are close to frequency of the load. In

this case local stress state and fine meshing is not needed (only general rigidity and masses make sense).

1. 3d *tetrahedral mesh* is very popular type of mesh, produced by open-source libraries such as Netgen.

There is prejudice about this type, that tetrahedral mesh causes wrong or inaccurate results. This is not true in general, but if you are using extended elements with high aspect ratio (http://docs.salome-platform.org/latest/gui/SMESH/aspect_ratio_3d_page.html) you really can obtain wrong results (that visually will look like correct). This issue is most critical for thin walled structures, because in this case you should create many elements to keep average aspect ratio within 1..3. In this case is much better to use shell elements instead of volumetric ones.

2. Linear or quadratic meshes? Quadratic mesh (with mid-nodes) is much more accurate (and it is able to guarantee well results in case of good aspect ratio), than linear. Try to avoid linear meshes, because you are at risk of getting wrong results even if they looks correctly. Very fine mesh in this case is not a protection, because of accumulation of rounding error. The structure will be more rigid than for real, stress results can be wrong.

3. Negative jacobian issue and gen3dim mistake in CalculiX. This problem may appear for rough quadratic mesh on curved geometry. In CalculiX CGX you can check mesh for bad elements in menu Viewing> Show bad elements. GMSH does this check automatically after meshing (click on strip under graphical window to see output), also it has plugin AnalyseCurvedMesh (Tools>Plugins). Latest versions of Salome also do this check and improve mesh automatically. Also when converting linear into second order, you may un-check option "keep mid-nodes on geometry".

Free Structural Software

Frame3dd

Free open-source software for static and dynamic structural analysis of 2D and 3D frames and trusses with elastic and geometric stiffness.

<http://frame3dd.sourceforge.net/>

calc4fem

Opensource LibreOffice template for static structural analysis of 2D frames and trusses with collection of custom basic functions for manipulation of results.

<https://sourceforge.net/projects/calc4fem/>

z88Aurora

General purpose FEA software with user interface (partly opensorce, freeware for commercial use)

<https://en.z88.de/>

Framework

Structural software for 2D and 3D frames (for Windows, but works with winehq)

<http://members.ziggo.nl/wolsink/>

On-line databases with collection of spreadsheets.

<https://www.engineersedge.com>

<http://www.steeltools.org/home>

On-line services (CAD+CAE)

<http://feacluster.com/calculix.php>

<http://www.alneos.com/en>

<https://www.simscale.com/>

<https://www.onshape.com/>