

CAELinux and finite element analysis.

Tutorial 1, perforated cube

Jean-Marc LICHTLE *

May 28, 2006

Contents

1 Introduction	1
2 Important note	1
2.1 CAELinux Installation	1
2.2 Add functionalities to CAELinux	2
3 Create the part with SALOME	2
3.1 Create the cube	2
3.2 Create the cylinder	3
3.3 Perforate the cube with the cylinder	4
3.4 Explode the volume into different faces	4
3.5 Remember:	4
4 Mesh with SALOME	4
4.1 Hypotheses and basic algorithms	5
4.2 Apply the various hypothesis to the elements	5
4.3 Create mesh groups for the analysis	6
4.4 Export to Code-Aster	6
5 Create the ASTER folder with New FE Analysis	7
6 Set-up the command file with EFICAS	7
6.1 Material properties	9
6.2 Apply the mechanical loads	9
7 Analysis with CODE-ASTER	11
8 Post-processing with SALOME	11
8.1 Visualize the displacements	11
8.2 Visualize the stress	11
8.3 Understand the results	12
9 Relationships between the various softwares	12
10 Conclusion, author, translation	13

Summary

The aim of this document is to provide an introduction to CAELinux as well as the softwares included in this Linux distribution specialized in finite element analysis. Next, we will learn how to use:

- SALOME, both a 3D editor, mesh generator and post-processor software

* Ingénieur Arts et Métiers promotion CH73

- EFICAS, the Code-Aster command-file editor
- CODE-ASTER, the finite element method software launched via the ASTK graphical user interface.

1 Introduction

This tutorial aims at simulating the stress and displacements of a perforated cube, one face being fixed and the opposite face submitted to a distributed pressure. The hole's axis is parallel to the faces on which the loads are applied. For this tutorial, files and folders are named after cube_perce.xxx, the xxx extension referring to the file format.

2 Important note

The following tutorial is based on the Beta 1 version of CAELinux, october 2005. With a more up-to-date version (Beta2?), various informations may be inaccurate.

2.1 CAELinux Installation

The CAELinux DVD is both an install DVD and a bootable DVD. As a result, you could either install CAELinux on your hard-drive or simply use it as a Live DVD such as KNOPPIX, UBUNTU Live or FRENZY in the BSD world. From a personal standpoint, i was not successful in launching SALOME as metionned, and this on several configurations: IBM laptop, SIEMENS laptop, SIEMENS desktop and so on. I ended up installing it on my personal computer (a white box PC) and on an IBM at my workplace.

CAELinux is based on PCLinuxOS, a fork from Mandriva which in turn used to be Mandrake Linux. Given this, users accustomed with Mandriva will find themself at home. The installation procedure is a breeze, just select your keyboard configuration accurately. If not, you might have to tweak some parameters afterwards.

2.2 Add functionalities to CAELinux

I was rather surprised to find a working LaTeX environment in CAELinux. This detail is a great help as I am currently typing this tutorial. However, bad luck as it goes, the VI version, my favorite text editor, is a old bloated piece of software which would only finds its way in Rescue Distributions like "Everything you need on a single floppy". The Gimp is there too but not Xfig, too bad. Being a FreeBSD user, i struggled for some time to find the useful trick: the 10.0 revision of Mandrake looks like the accurate version to add packages to CAELinux. One only has to know basic commands like find and rpm and that's about it. Adding the "enhanced" version of VI with syntax highlighthings and others much appreciated add-ons was easy. Installing Xfig only required inserting a CD. To go straight to the point: don't throw your old Mandrake / Mandriva CDs, keep them close to your machine, they might be useful when adding functionalities to CAELinux.

3 Create the part with SALOME

Not being an advanced 3D product creation software or CAD tool, SALOME will still create advanced geometric shapes and corresponding volumes. We will use this functionality to build a volume and create a mesh. I have to mention here that SALOME is able to import CAD files like iges and step, as well as brep, a format i didn't knew before.

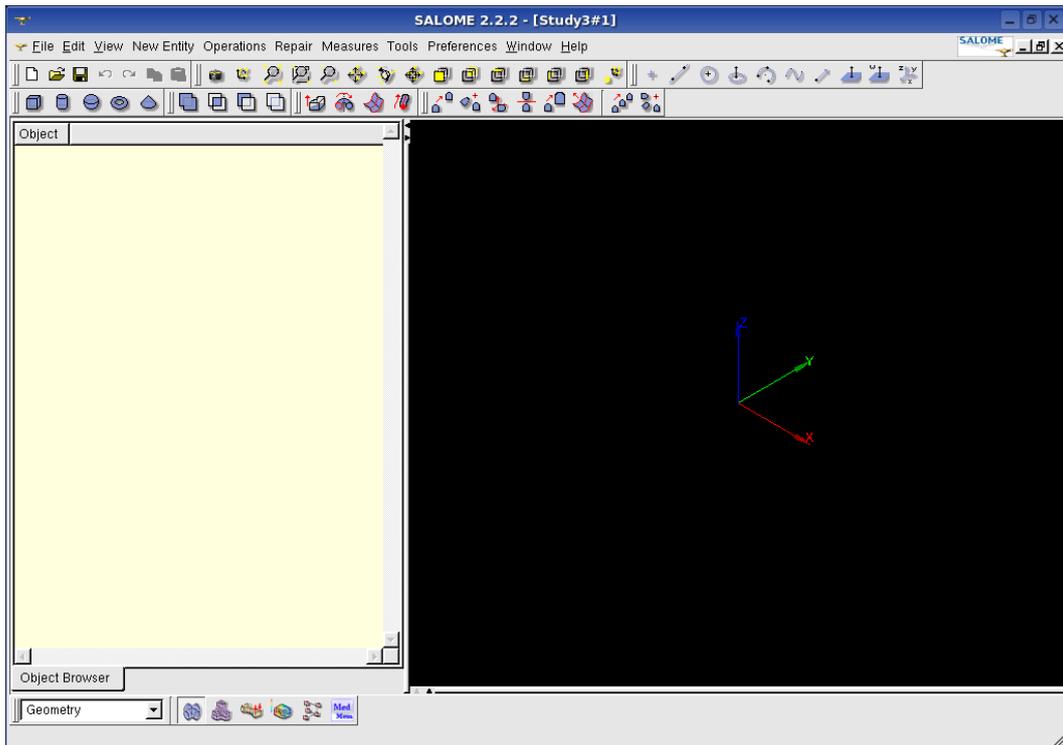


Figure 1: Starting window

3.1 Create the cube

Launch SALOME, open File/New, select Geometry then close the various Python interpreter windows by adjusting the display window, using the small dedicated arrows on the bottom left. Please bear in mind that SALOME can be used in Geometry mode, but also in Mesh mode, Post-Pro mode (as a post-processor for Code-Aster), and so on. Be aware of this: if you don't find a specific item in the menu, or a right-click doesn't do what it is supposed to do, ask yourself if you are really in the mode you are supposed to be!

A cube is, well, a square vector-extruded face. This face is the result of a transformed "sketch" figure. The first step is then to create this part from the various mentioned elements. To do so:

Menu New entity / Sketch, enter successively the following coordinates (0,0), (2,0), (2,2), (0,2), and validate each set of coordinates using the "Apply" icon. The "Sketch closure" allows to validate the drawing and close to window. Use the magnifying glass icon on the left, "Fit all", to display the new drawing using the available space.

Bear in mind that the units are not indicated. Enter the values using the system you want, just be accurate: pressure in Pascal if the dimensions are expressed in meters and force in Newton. It is important to understand that we currently created 4 lines, connected to make a square. So this is still a line and not a surface. In a following step, we will transform those 4 lines into a plane object, also called "face" in SALOME.

Let's stop for a while to explain how does the mouse work. Using CTRL, the mouse buttons allow to change the view as follow:

- CTRL + right click: turn in the view
- CTRL + left click: keep the orientation, zoom in or out
- CTRL + middle: move the view, size and orientation remaining the same.

All this can also be done using the various magnifying glasses located on the right in the menu toolbar. Before starting to modify the square into face, let's have a look at the left corner of the screen. A right-click on "Geometry" and a "Expand all" selection displays a line Wire_1. A click on Wire_1 modifies the square's color and a click on the square brings the selection back to Wire_1. The menu

on the left will change according to the status of the project, to describe how the different elements were created.

To modify the square Wire_1 into a face (or plane surface), click in the "New Entity" menu, "Build", "Face". A click on the "Wire(s)" arrow followed by "Wire_1" in the left menu displays "Wire_1" in the dialog box. Please bear in mind that, if the selection was on Wire_1 (left panel) before starting to create the face, a click on the arrow would have been enough to select Wire_1. Validate with OK to close the dialog box. You now see a new element "Face_1" in the menu, click on the + sign to "open" the line, you will then see the Wire_1 in red. The face Face_1 is a product of the Wire_1 transformation. Up to now, this detail seems rather pointless, but soon, it will be nice to quickly find this f*****! primitive we forgot everything about!

Once the basic face has been created, one needs to create a vector to construct the different extrusions. To do so, Menu, New Entity, Basic, Vector. You then have the choice to choose between two kind of generation method: either a vector described by two points (the default choice), or a vector defined by its coordinates. We are gonna use the later, so click on the right icon then input the coordinates 0,0,1 and validate with OK. Vector_1 is then displayed in the left panel.

We now have everything we need to extrude the cube, that is: one face and one non parallel vector. To extrude Menu, New Entity, Generation, Extrusion, Base = Base_1, Vector = Vector_1, enter 2 in Height and validate with OK. I'm gonna go straight to the point now, manipulating the dialog box not being rocket science. To quickly get a nice cube, go to menu View, Display Mode, Shading and you can even play with your new creation. Come back to the wire view (View, Display Mode, Wireframe) to go further in this tutorial.

3.2 Create the cylinder

Now it is time to draw the circle which we will use to define the hole in the cube. To do so, one needs a center, a vector and a radius. We previously created this vector. Only a center point is currently missing. Menu New, Basic, Point, then enter the coordinates 1,1,0 and validate. A point named Vertex_1 appears in the left menu and on the drawing, in the middle of a cube's face. Drawing the circle isn't much complicated, menu New Entity, Basic, Circle, Center Point = Vertex_1, Vector = Vector_1, Radius = 0.5 and then OK. One may be tempted to change the circle into a face. However, this shortcut doesn't work. One needs to first transform the circle into a "Wire" then the "Wire" element into a face.

Let's go to menu New Entity, Build, Wire, Objects = Circle_1 then OK. A figure "Wire_2" is added to the parts list in the left menu. To transform it into a face, menu New Entity, Build, Face, Wire(s) = Wire_2 and finally OK.

The cylinder's extrusion associated to the circle is similar to the cube's extrusion: menu New Entity, Generation, Extrusion, Base = Face_2, Vector = Vector_1, Height = 2 and OK. You just created Prism_2, a cylinder with the following properties: radius=0.5, height=2 inside the cube and located on a face.

3.3 Perforate the cube with the cylinder

In order to get the final volume, a perforated cube, one just needs now to subtract the cylinder to the cube. Menu Operations, Boolean, Cut, Main Object = Prism_1, Tool Object = Prism_2 then OK. If everything went smoothly, you should now have a perforated cube. Check by using a shaded view, menu View, Display Mode, Shading. Use Display Only (after a right click) on Cut_1 to cancel the other parts already on the screen.

3.4 Explode the volume into different faces

But we are not done yet. In the introduction, I explained that the load condition on the part would be a fixed boundary condition and a distributed pressure load on the opposite face. The geometry we already defined here doesn't allow such a thing. The next step is to explode the cube into faces. Let's begin with menu New Entity, Explode, Main Object = Cut_1, Sub Shapes = Face then OK. This operation creates faces numbered from 3 to 9; face 3 being the one constructed on the X and

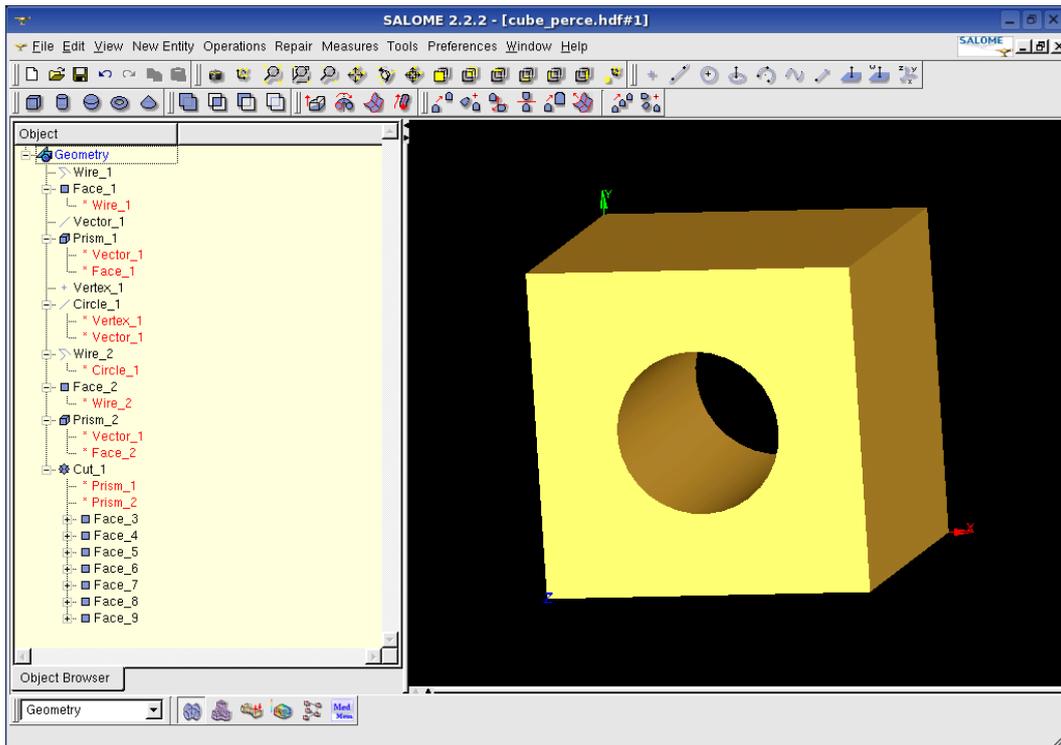


Figure 2: Final Geometry display

Z axis, face 8 being the opposite one. To identify the face, use the right click - Display Only on each face's description or simply click on the crosses displayed on the faces. You could also use a right click on Geometry, the first line on the left menu then Display to..erm..display the different parts of the geometry.

3.5 Remember:

We are now at the end of the first part and we created a perforated cube. It is important to keep the following in mind:

- Holding CTRL and moving the mouse allow one to turn / resize / translate the view.
- The left menu displays the procedures we went through to get our part. Be smart and use a right-click - Expand All to get all the needed details of the part's tree.
- Right-click, Display or Display Only are used to complete or limit a display and identify a component.

Now, it is time to save our progress, menu File, Save then enter the file name. By default, SALOME proposes Study1. In this tutorial, we will just call the file cube_perce. This creates, in the working folder, a file named cube_perce.hdf, hdf being the SALOME format.

4 Mesh with SALOME

The goal is now to divide the previous volume into a mesh, a bunch of smaller volumes on which we will apply the various material properties and calculate the stresses resulting from the load. It is out of question to discuss the theory behind the meshing, a whole manual wouldn't be enough. So, we will only apply a basic recipe.

Leave the Geometry mode and enter the Mesh mode (on the bottom left of the screen). There are three main steps in order to produce a mesh: the basic hypothesis's definition, its application to the desired volume and the designation of the faces on which we will later apply the loads.

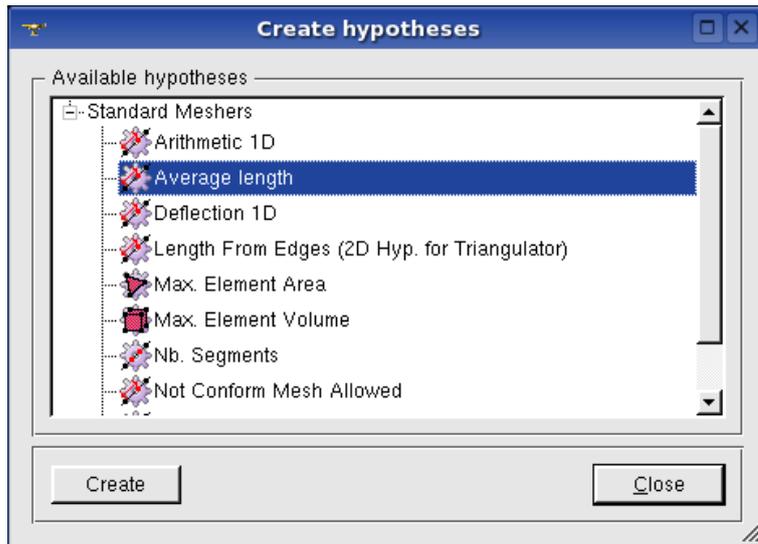


Figure 3: Window Create hypotheses

4.1 Hypotheses and basic algorithms

The Hypotheses menu Create Hypotheses displays (could you guess?) a window "Create hypotheses". We will create 3 hypotheses:

- Average length, click on Create. A new dialog box appears, enter 0.2 in length and choose OK.
- Length From Edges Then Create.
- Max. Element Volume Then Create, keep 1 as Max Volume value and validate with OK.

What did we just do? I feel quite embarrassed to give you a satisfying explanation up to this point. One thing is for sure, the Average Length value we just enter has a direct effect on the mesh's refinement. If you got a standard vintage PC like mine, RAM 256Mo with a 1.6Ghz processor, a good start would be to just choose a value roughly 1/10th of the cube's diagonale. A finer mesh could lead to very intensive calculations.

The Hypothes menu, Create Algorithms leads to a dialog box similar to the previous one. Select here Wire discretization, Triangle (Mefisto) then Tetrahedron (Netgen) by validating Create between two selections. Close...closes the dialog box after the third selection. So, the game is over: hypothesis and algorithms are now ready for action. An important matter here: right-click on the new Mesh entity at the end of the left menu, then Expand All so you see 6 lines corresponding to the 3 hypotheses and 3 algorithms we just setted up.

4.2 Apply the various hypothesis to the elements

The menu Mesh, Global Hypotheses opens a dialog box called Mesh Construction. This one belongs to the same group as the one we worked on previously. Name = Mesh_1 is straight forward. Enter the Cut_1 value to Geometrical Object and then you will need to fill Hypothesis and Algorithm. Here we are gonna use a multiple selection for both of the fields. So, click on the Hypothesis arrow then click on the first line of the left menu. The dialog box displays Average length. Hold on the Upper case and select the last line of hypotheses, the display changes, you now see "3 Hypotheses". Follow the very same procedure for Algorithm by selecting (a multiple selection) the three algorithms Wire discretization affected to (??) Tetrahedron (Netgen) and press OK. A new element Mesh_1 has been created in the left menu. Now it is finally time to calculate the meshes, so right click on Mesh_1 and then on "Compute". The screen freezes for a few seconds (roughly a dozen on my machine). Do it again by selecting "Update" this time. After a few seconds, you should see that the perforated mesh changes. You guess at this very moment that the mesh is about to be drawn. A new click on Mesh_1 followed by a snappy "Display Only" and the screen is now displaying the mesh, which was the aim of this chapter.

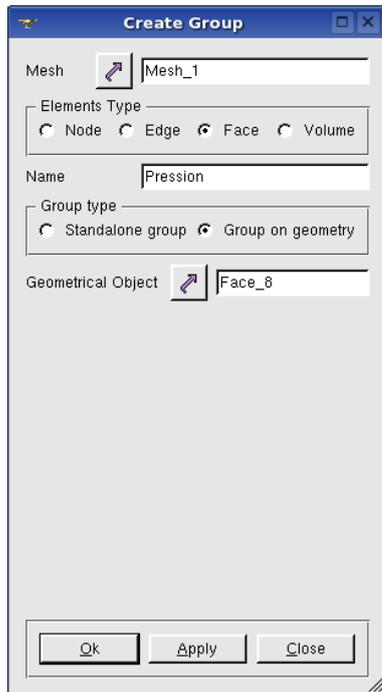


Figure 4: Dialogue "Create Group"

4.3 Create mesh groups for the analysis

The part is now meshed. Before going to the next step, the one defining the loads applied to the structure, one task still remains ahead of us. We need to name the faces (and so the elementary meshes that are parts of the faces) so we will be able to called them accurately in the next step. Menu Mesh, Create Group to display the "Create Group" dialog box. Click on Face and Group on Geometry to mimic the 4 figure on page 7. Use the previous procedures (click on the arrow then click on the object in the left menu) to fill up the fields Mesh and Geometrical Object. The result of the first dialog window is the one shown on figure 4, page 7. Name the face 3 "Base" and the face 8 "Pression" (those names won't be translated since they will be used later in the command file). Those names are free, you really can do whatever you feel like, i tend to prefer Base to designate a fixed face which isn't gonna move under the load and Pression the face on which we will apply the pressure load. One still needs to:

- Save the file using the Save menu. You can save it under the previous name and scratch the previously saved file or give it a new name.
- Export the mesh (with the designated faces) so that Code-Aster can read the data.

4.4 Export to Code-Aster

The export is launched by a right click on the Mesh_1 element of the tree then Export to MED. In the dialog window, please make sure you choose the File Type MED 2.2 and give a name to the file. It doesn't matter if it is the same as the one you gave to the previous file since it has been saved in the SALOME format (extensions are different). Now, you should have two files, cube_perce.hdf (SALOME format) and cube_perce.med (CODE-ASTER format). You may reduce the SALOME window as we won't need it in the next chapter.

5 Create the ASTER folder with New FE Analysis

Click on the "New FE Analysis" icon. "Create New Aster Job" welcomes you. As it is stated, this is your entry point for CODE-ASTER. In fact, this tiny program is about to create a new folder which you

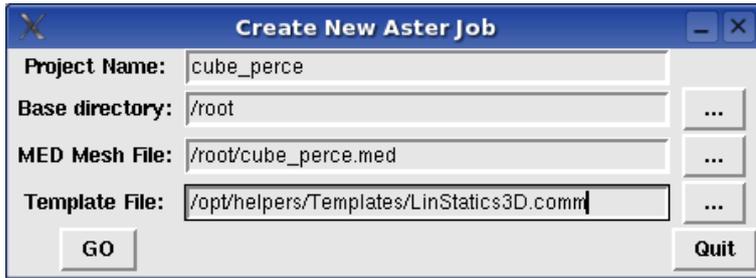


Figure 5: Create a new ASTER study

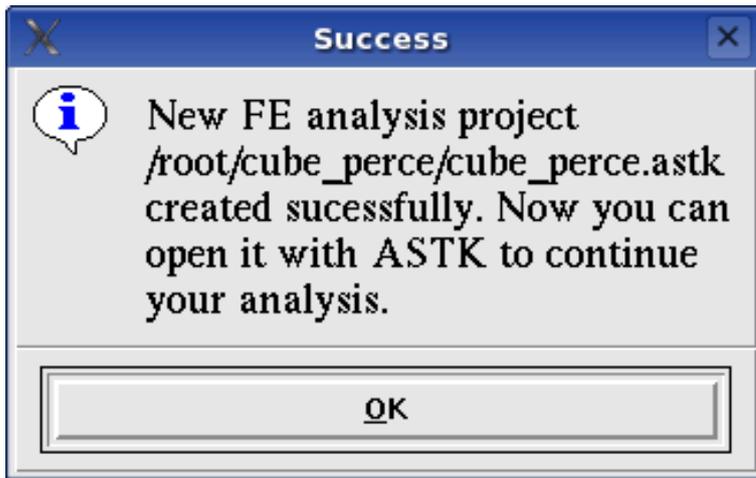


Figure 6: Creating the ASTER folder

will define on the first line 1, in a folder defined line 2, using the MED file designated line 3 and using the model on line 4. Easy isn't it? To summarize:

- Name the folder line 1. I do use the previous name cube_perce.
- Choose the folder in which the previous folder will be created, /root by default.
- Choose the MED file you just created for this new analysis.
- Select LinStatics3D.comm among the 3 available models.

Validate, you should get a Confirmation box similar to the ?? figure on page 8.

6 Set-up the command file with EFICAS

Now it is time to click on the ASTK icon on the desktop. This opens an ASTK window "ASTK Version ... New". The menu allows, thanks to File, Open, to load the file cube_perce.astk already located in cube_perce. The window is similar to the 7 figure displayed on page 8. A double-click on ./cube_perce.comm opens the CODE-ASTER command line editor. A new window like 8 page 9 appears on the screen.

In this window, we will define the boundary conditions, the loads, the material properties and so on. The content of the various tabs corresponds to groups of command lines in the file cube_perce.comm. We could edit this file by hand, but EFICAS (which would mean EFFICIENT in english) will provide us some great help and will thankfully advise us in order to create the file and also to check the syntax.

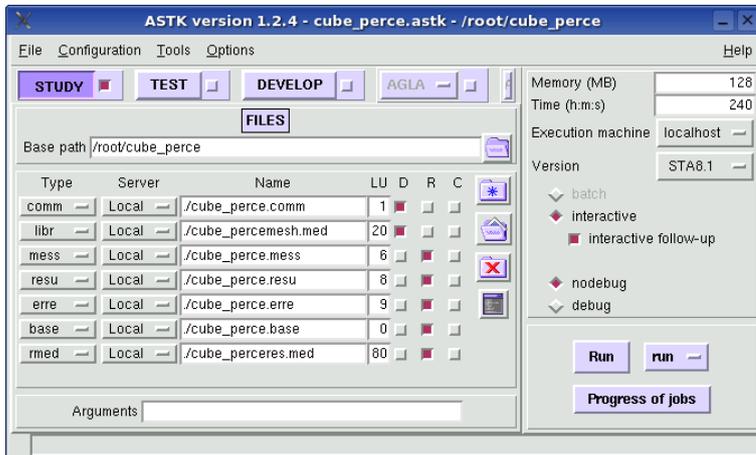


Figure 7: ASTK dialog box one the command file (*.comm) is loaded

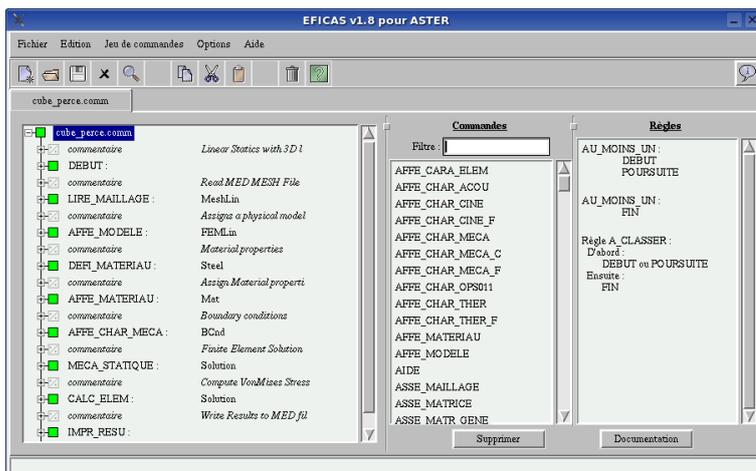


Figure 8: The EFICAS window

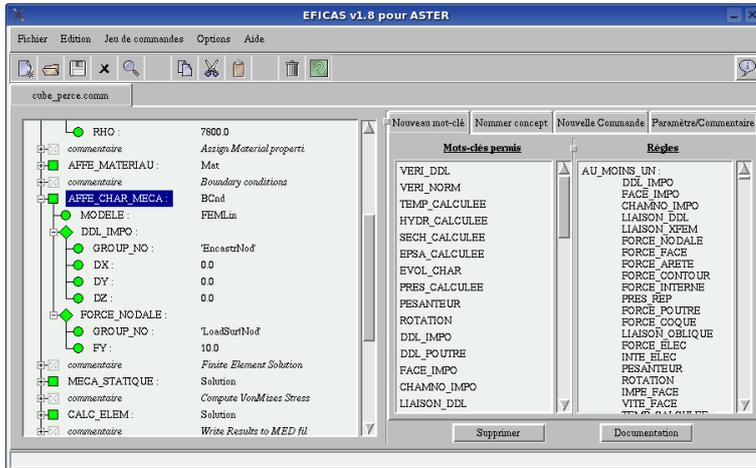


Figure 9: Original AFFE_CHAR_MECA in

6.1 Material properties

Open the DEF1_MATERIAU tab by clicking on the + sign at the start of the line. Open also the new tab ELAS. A material's name and three numerical values are mentioned there, Steel to designate ...steel, then the Young modulus E, the POISSON coefficient NU and the volumetric mass RHO. No units are specified, we just use the basic ones, Newton, meter and so on. You have the ability to rename the material by clicking on DEF1_MATERIAU then "Nommer Concept" in the right panel. Additionally, you can change the values E, NU or RHO by clicking on the appropriate lines and entering the new value. Hit Enter to validate your value. As for the first test, you can let the default values. You can also add a material by clicking on "Nouvelle commande" in the right panel then DEF1_MATERIAU in the next list. This creates a new line DEF1_MATERIAU in the left panel, highlighted in red, meaning that the definition is not complete yet. To delete a line you might have wrongly created (or just for the shake of trying), select the offending line and click on the Supprimer icon in the right part of the window.

6.2 Apply the mechanical loads

The line "AFFE_CHAR_MECA", which I recommend you to open by clicking on "+", is by far the most important of this whole tutorial. That's where we will do the most important modifications. See the figure 9 on page 10. Start by deleting the group of lines FORCE_NODALE, which is not needed in this tutorial. Click on FORCE_NODALE and then on the supprimer icon. We will first modify DDL_IMPO to apply a boundary condition to the face "Base" of our cube. Select GROU_NO and delete it. Change it to GROU_MA by clicking on DDL_IMPO then GROU_MA in the right list (double click). Enter "Base" in the dialog box on the right then on the hand whose fingers are pointed on the left to transfer this value in the "Valeur(s) actuelle(s)" list. The screen 10 page 10 shows the result of those manipulations. By our last inputs, we have been telling the mechanical code that the face "BASE" is a fixed boundary because DX, DY and DZ are zeros. Now, one still has to apply the homogeneous pressure on the face PRESSION. Click on AFFE_CHAR_MECA then choose PRES_REP in the list. This is the homogeneous pressure. With a double-click, a new line PRES_REP is created under DDL_IMPO. We just need to indicate on which face to apply this load:

- Click on GROU_MA (double click) to add a line and open a new input window. Enter "Presion" in the input field like we did previously for "Base".
- Click again on PRES_REP (on the left) then double-click on PRES in the list of available keywords and enter 1e6. Validate with Enter. The value 1e6 is in Pascal so it means 10bars.

Once the pressure has been validated, all the icons should be green. Save this new version of the cube_perce.comm file with Fichier, Enregistrer. You may reduce the EFICAS window, we shouldn't need it anymore.

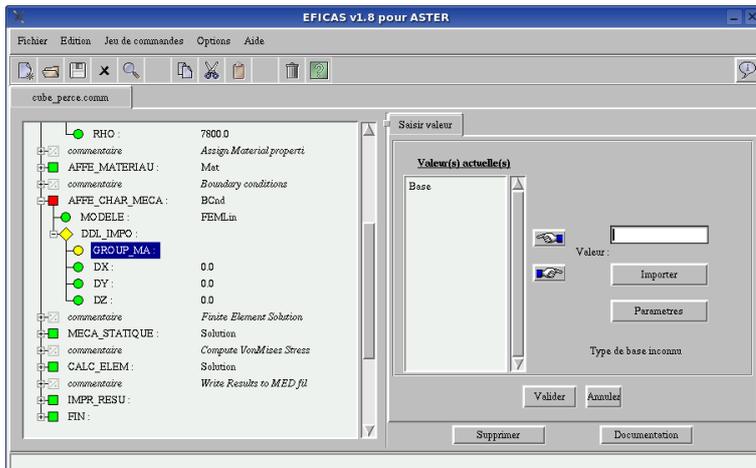


Figure 10: DDL_IMPO for the base

7 Analysis with CODE-ASTER

That's it, the ASTER configuration file has been created, we now launch the analysis. This is the least interactive step because one only needs to click on RUN in the ASTK window to see the machine crunching numbers. You can follow the analysis's progress in the text window if you are snappy enough to get a glimpse. It's particularly interesting to have a look at the start of the lines. You would see <I> for some steps but a <F> would mean that there is a mistake and that the calculation will not be successful. If you have got doubts, click on `/cube_perce.erre` once the calculation has ended to see the messages / errors summary.

8 Post-processing with SALOME

Let's say everything went to a happy end. You can now wake up SALOME, open a new window, this time in the Post-Pro mode and not Geometry or Mesh. Load the result of the calculation with File, Import from File then select the file `/cube_perce/cube_perceres.med`. A line + Post-Pro should appear on the left. A right click followed by Expand All and the result's tree is here! The interesting lines are SolutionDEPL as well as 0,INCONNUE and SolutionEQUI_ELNO_SIGM, followed by 0,INCONNUE. Those two groups are respectively the displacements and stresses calculated in the part.

8.1 Visualize the displacements

To visualize the displacements, proceed as explained:

- Click on 0,INCONNUE corresponding to the group DEPL then right click and select the option Scalar Map. Validate the new dialog box, there are no mandatory change for the time being, the default values are ok. A colored cube is displayed and a graduated bar assigns numerical values to the colors. One recognizes the Base in blue (displacement value = 0), and the face Pression is fully colored.
- Again, click on 0, INCONNUE and validate Deformed Shape. Validate Magnitude coloring. The mesh will be displayed along with the previous shape. Select now the new line Def.Shape:1 and then, using a right click, select Display Only. The view, similar to **11** on page **12** only shows the mesh corresponding to the deformed part. Turn the figure to really see the shape under load. Clearly, the displacements look bigger than usual: the amplification factor has been briefly displayed in the screen in which we selected Magnitude Coloring.
- Click on Def.Shape:1 then right-click on sweep. Your rolling eyes discover the part being crushed under the stress, starting from the initial free case to the nominal load.

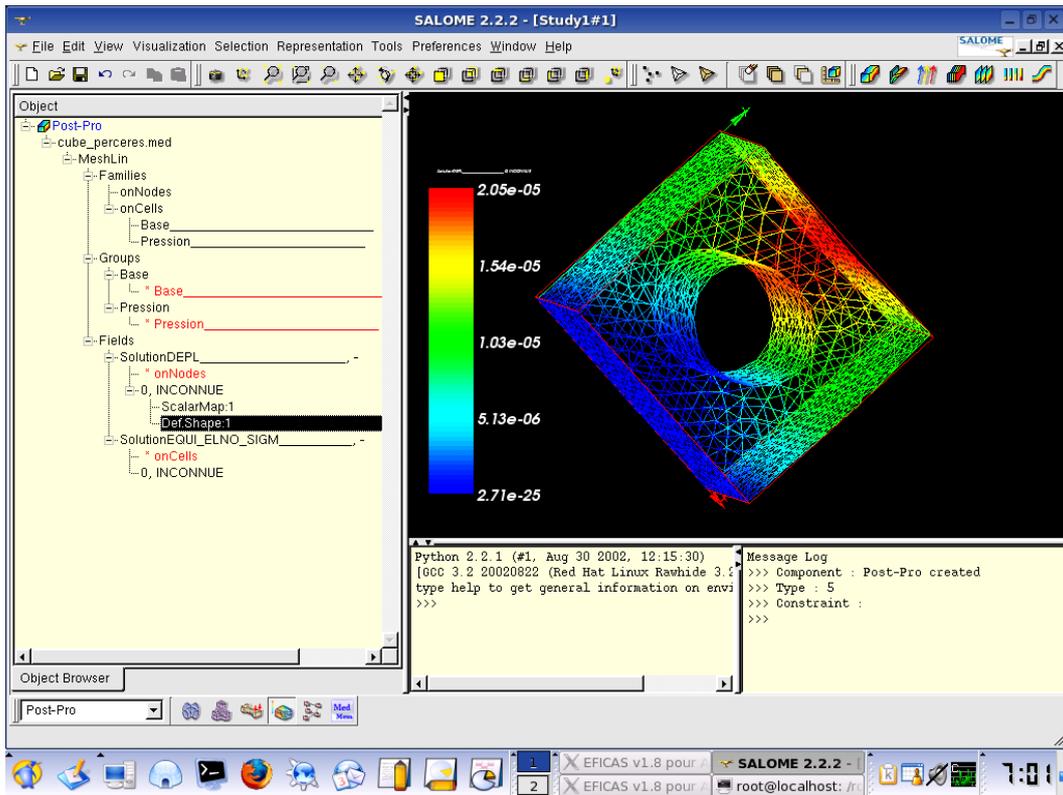


Figure 11: Deformed view

8.2 Visualize the stress

The idea behind it is similar, you should easily find how to display the colors corresponding to the stress. Keep in mind that, this time, there is no deformation view but only a stress field. You are free to change the display mode, going to wire frame for instance. If you are eager to try a few things, you will discover how to increase or reduce the amplification factor.

8.3 Understand the results

The part we created is 2m big. In fact, it is rather closer to a Charles de Gaulle aircraft carrier's propeller bearing than a motorcycle's part. I decided to go with these dimensions since it would be easier to type. The interested reader should feel free to do it again using more common dimensions. The 1MPas distributed pressure on a $2*2$ is equivalent to a 400tons force. The figure 11 clearly shows that the maximum displacement is roughly, given the material, $2e-5$ m at the center, half of that on the extremities.

9 Relationships between the various softwares

One of the main difficulty i encountered when i discovered CAELinux and the softwares allowing the finite element analysis was the understanding of the various links between the softwares. I especially had a hard time getting the fact that Code-Aster wasn't an interactive software but it was running as a standalone solver, using one click on the run icon within ASTK. We browsed together the tutorial, getting from one software to the other to finally go back to the first one. To really get to undersand our path, a small scheme is worth thousands words. Here are some explanations regarding the figure 12 on page 13.

- The SALOME mesher is more than a simple mesh generator. SALOME can also import a CAO file like step or iges or it can be used to generate parts, being surface or volume. The later

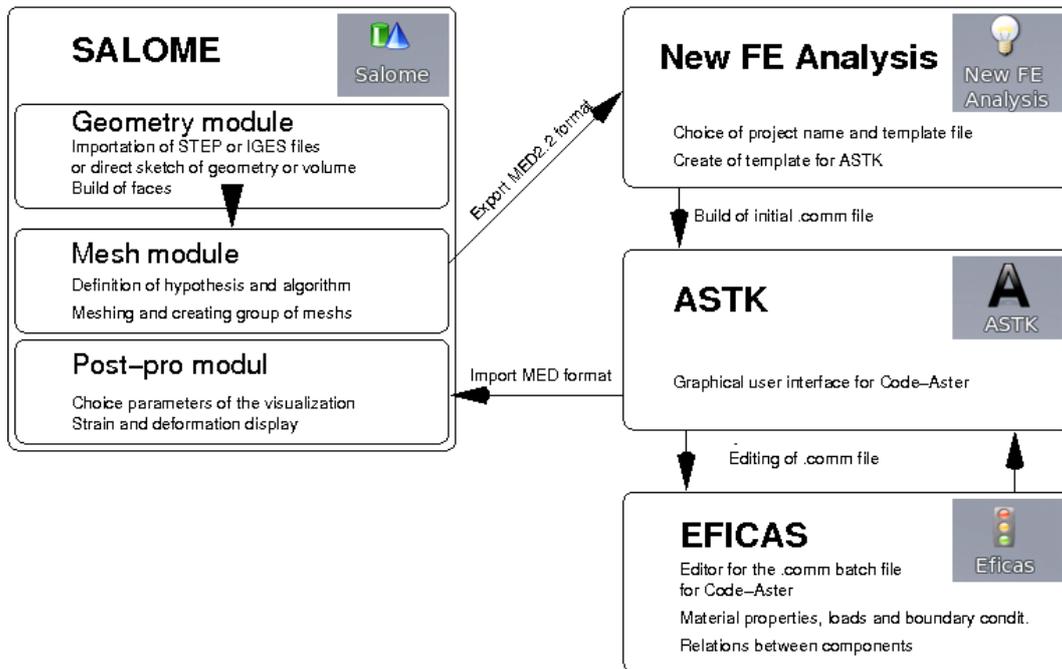


Figure 12: Overview of the various softwares

option has been used for this tutorial.

- Salome has many modules, a geometry module in which we defined the perforated mesh but in which we also exploded the faces. The mesh module gathers the geometrical datas to create meshed structures. The previously designated faces are used to create the groups of mesh linked to the faces. We only created meaningful groups regarding the load.
- Salome works with its own file format, corresponding to the hdf extension. To work together with Code-Aster the datas are exported in med (version 2.2 and no 2.1 like the one by default) which is specific to Code-Aster. Following the same philosophy, the datas are imported into SALOME using this very same format.
- New FE Analysis is a small script with only one purpose: to gather informations like the name of the working folder, the name of the folder to be created for the study, the name of the med file and so on. Once done, the script gathers the informations in a raw Code-Aster command file, save this file in the new folder, and that's about it.
- ASTK is a Graphical User Interface for Code-Aster. We used it to launch EFICAS, the command line editor for the file created by New FE Analysis. Once we are done with it and the file has been saved, ASTK launches Code-Aster thanks to the RUN icon. Code-Aster runs and displays its output in a non-interactive shell window.
- EFICAS, shortcut for Editeur de Fichier Code-Aster (it also means EFFICIENT), is the most important software to be used. The groups of meshes are given properties (material, properties,...) and, in this tutorial, the mechanical loads and boundary conditions. EFICAS only works on the command file, then, once it has been saved, one should come back to ASTK to launch the Code-Aster calculation.
- Once done, you can come back to SALOME and, this time using the Post-Processor module, you can graphically display your results. Bear in mind that the datas are imported from Code-Aster.

10 Conclusion, author, translation

This tutorial is the first one of a serie of others in which i wish to write down my various experiences with CAELinux. I feel free to get some help from an intern who should spend a few weeks on this subject in my company. Its contributions will be clearly mentionned if anything should be published.

Author's right: the present document is GPLed. If extracts are used in some articles or documents, please acknowledge the author of the initial document.

The author: Jean-Marc LICHTLE Engineer "Arts et Métiers", Chalons 1973-1977. At this time, we used to study complex structures using a slide rule. The luckiest ones got an electronic calculator which painfully handle trigonometry. It has changed!

Translation: This quick and dirty translation has been carried out by Laurent Malod-Panisset. In order to improve it, feel free to send an email.

You can send your comments in french and german to:

jean-marc.lichtle@gadz.org

In english: laluciol@club-internet.fr