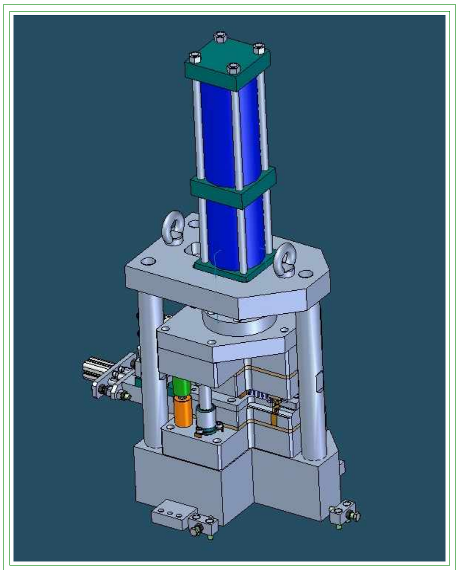
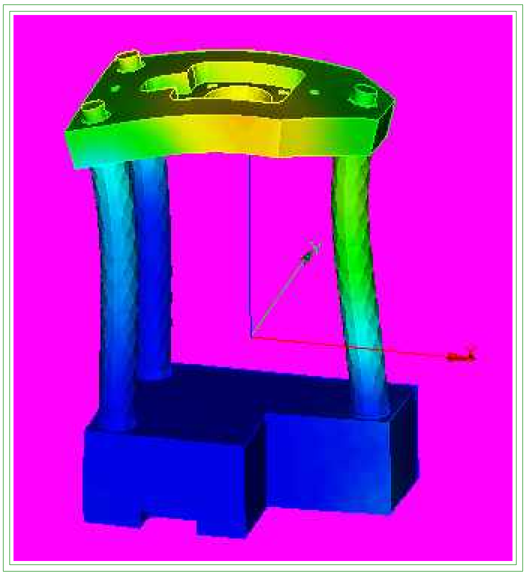
TUTORIAL

FEM Analysis on the frame of a little press actuated by a pneumo-hydraulic actuator.





A.Marchetti - www.stimarchetti.it

January 2009

Preliminary note

In this document, the procedure to execute the FEM analysis on the frame of a little press is described.

Software involved in the analysis are:

1. Solid Edge on Windows XP for 3D modeling
2. Salome-Meca 3.2.9 on Caelinux 2008 for pre and post-process
3. Code-Aster 9.2 on Caelinux 2008 as FEM solver

This document is focused on to provide a user guide for the beginners of Salome and Code-Aster, more than to be the report of the FEM analysis itself. Consequently, more attention is given to the procedure than to the analysis of the numerical results.

Author: A.Marchetti cacciatorino@libero.it www.stimarchetti.it

Index

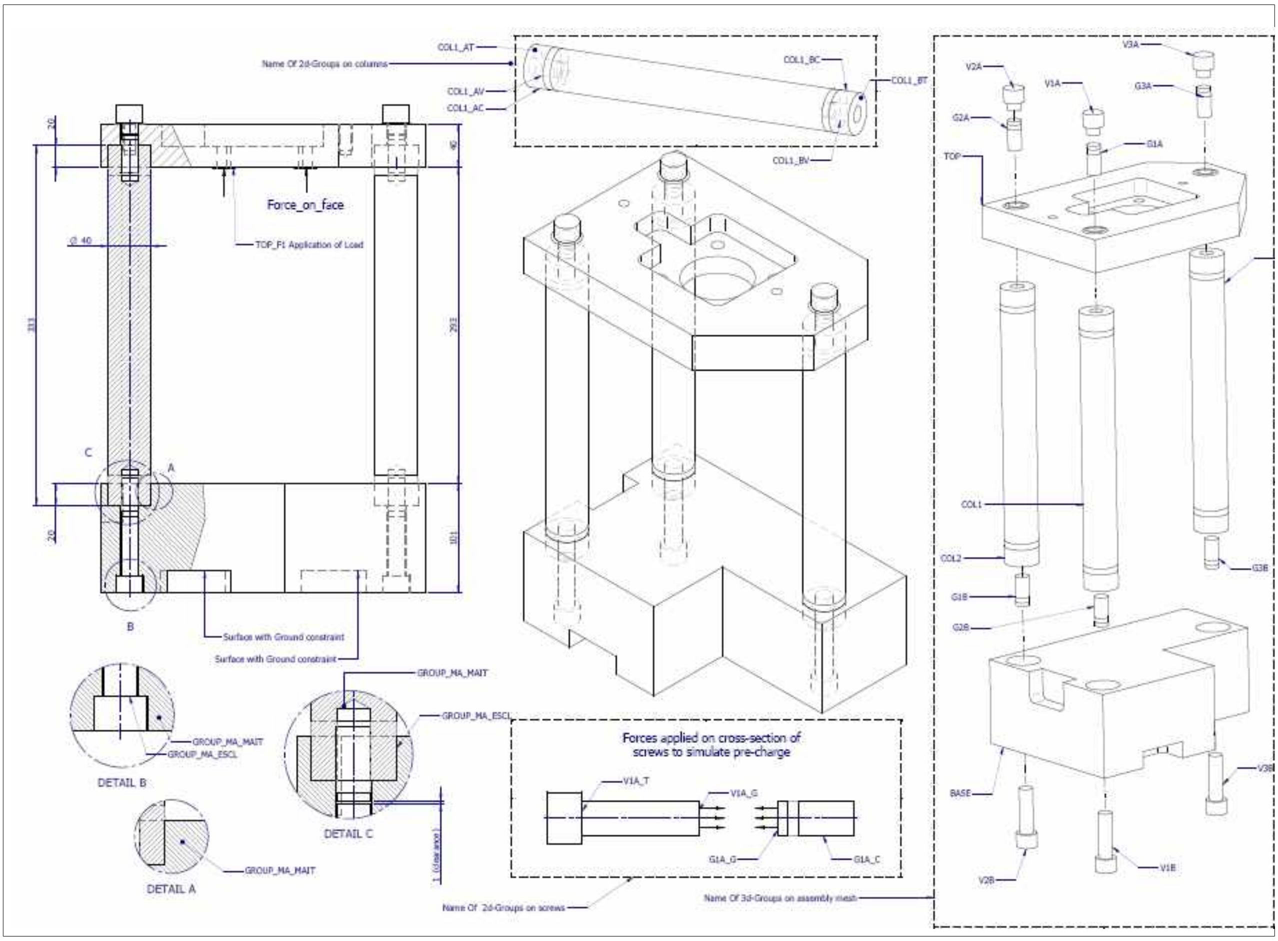
- Attached file:
  - ➔ 3D model step file
  - ➔ Analysis hdf file
  - ➔ Code-Aster command files (n.2)

1. CASE DESCRIPTION

1.1 Introduction

The examined structure is the frame of a little press designed to bend and punch an electric terminal contact. The die is actuated by a pneumo-hydraulic actuator, which is connected to the top of the structure by means of 4 hexagonal nuts screwed on 4 studs that are part of the actuator.

The force is transmitted to the basement of the structure trough three columns. The columns are connected to the top and the base by means of screws.



1.2 Simplifications and modifications on the model

The model that was designed for building was simplified and modified to make it more suitable for a FEM analyisis:

- Simplifications:

- Elimination of little holes
- Elimination of hand-tools socket on screws and components
- Elimination of unnecessary components

- Modifications:

- Creation of round bosses where screws and nuts transfer load, to limit load application areas.
- Making of restrains on columns and screws: Code-Aster requires that contacting surfaces on two adjacent parts have almost the same area, so both columns and screws were subjected to a lathing cut to obtain surfaces similar to the cylindrical ones of the holes where they were inserted.
- Cut of screws to apply pre-load: each screw was cut along its cross-section to obtain two half parts about 5 mm above the point where screws are inserted in the column. The obtained surfaces (with a clearance of 1 mm between them) were used to apply traction force to simulate pre-load.

1.3 Material Properties

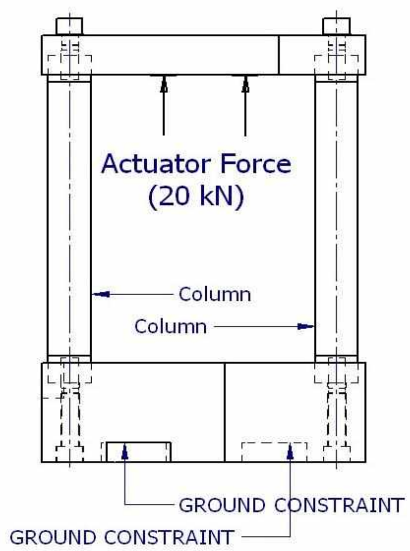
The same material was used for all the components, with the following properties:

- Young Module E=210.000 MPA
- Poisson coefficient ν = 0.3
- Behavior: linear elasticity
- Break point 430 Mpa, Yeld point 275 MPa

1.4 Applied loads, constraints, screws pre-load.

1.4.1. Applied loads:

Actuator force is about 20 kN. This leads to have a pressure of 26 Mpa where actuator is connected to the structure (Code-Aster requires that loads are expressed per surface unit, and not simply as forces)

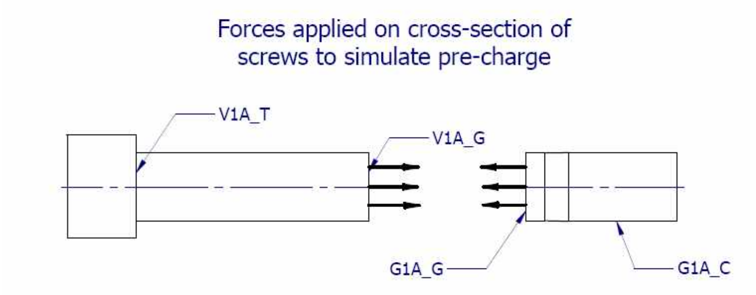


1.4.2. Applied constraints are:

- Ground constraints on sockets in the base.
- Mate constraint between the head screw surfaces and their housing surfaces on components
- Mate constraint between the head column surfaces and the bottom surface of the holes where they are fitted.
- Mate constraint with axial sliding capability between cylindrical column surfaces and related cylindrical surfaces of the holes where they are fitted.
- Mate constraint between bodyscrews and related threaded holes on columns

1.4.3. Screw preload

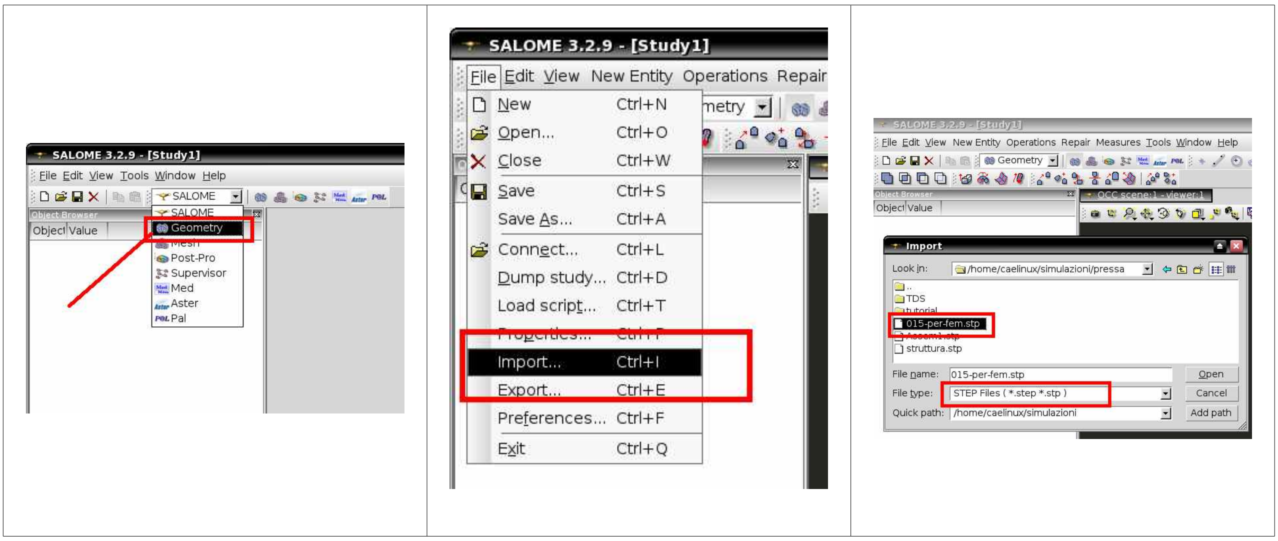
A tensile stress of 250 MPa was applied to the body of each half-screw, in order to simulate preload, with direction perpendicular to the cross section and going out from the surface (i.e. is a tension). If a screw type 8.8 is used, the load at the 70% of yeld point is about 450 MPa, so we are safe using 250 MPa.



2 GEOMETRY CREATION

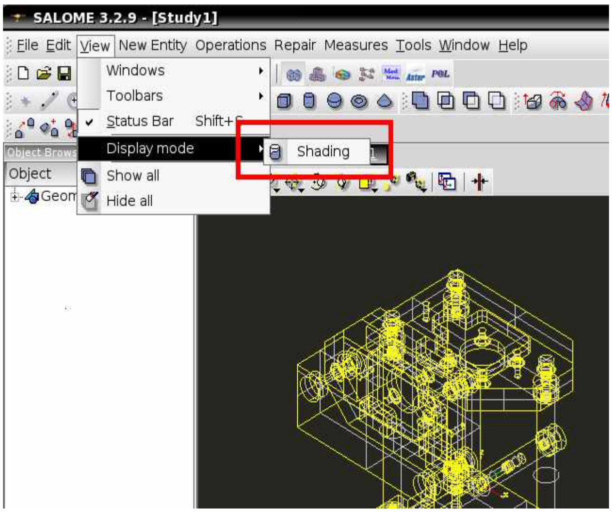
2.1 Exporting from Solid Edge and importing into Salome

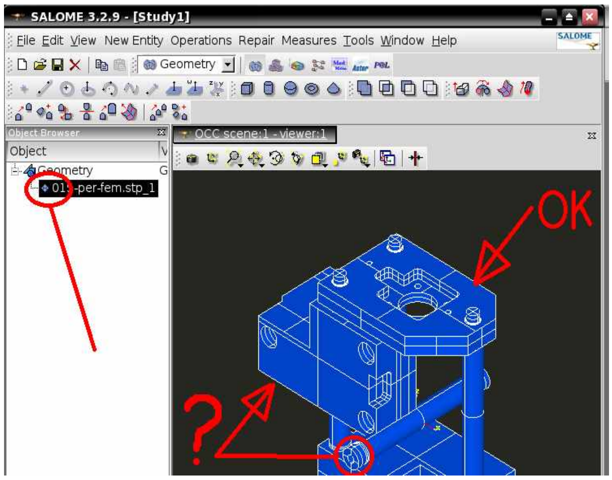
After the model is completely defined using Solid Edge (the modeling module included in Salome is too much difficult to use), we can export it in a step format, STEP203 or STEP214; iges export is not a good option when assemblies are involved because Salome imports them as single bodies and not assemblies. After the step file has been created, we turn-off windows and reboot the pc in the Caelinux environment. Next steps are: run Salome, activate geometry module, import the step file.



Now we can save the analysis file, calling it: STRUTTURA.hdf.

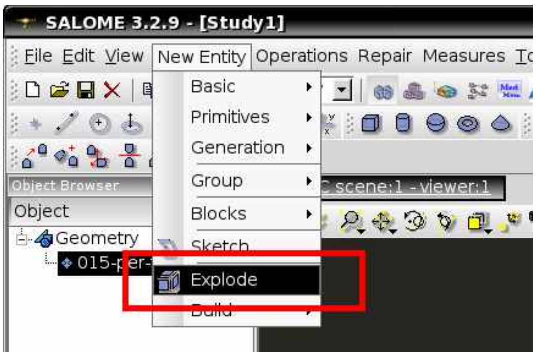
By deafult, when Salome starts, the view mode is wireframe, so we switch in shading mode as first operation. In the left pane, we find the features tree: the first icon is a “compound” icon, which is the name used by Salome to call the assemblies of components. We immediately note that the step file was imported incorrectly: there is the assembly with all its components in the correct location, but every component is imported one more time in the 0,0,0 point.

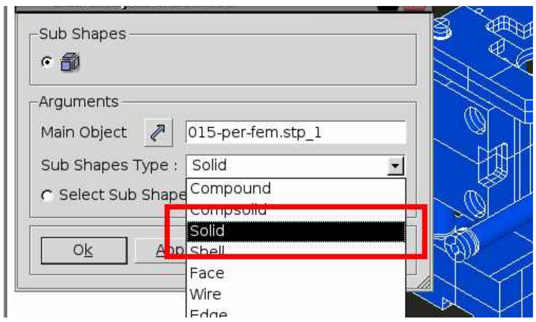


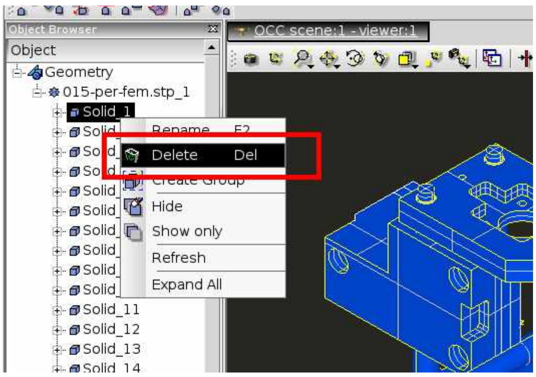


The way to solve this problem is to explode the compound extracting all its components, to be able to delete all the components that were imported at the 0,0,0 point.

Note: this problem is related to the Salome-SolidEdge interaction: it is reasonable to think that others cad packages have different behaviors.

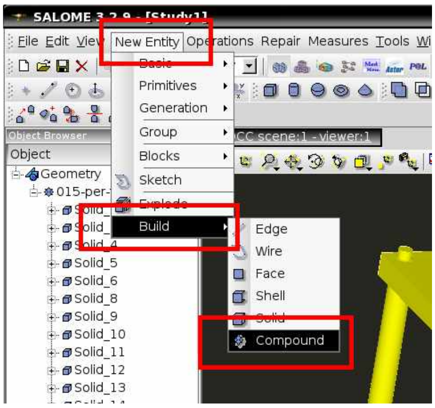


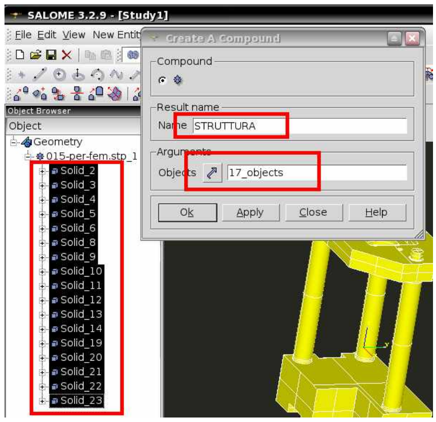


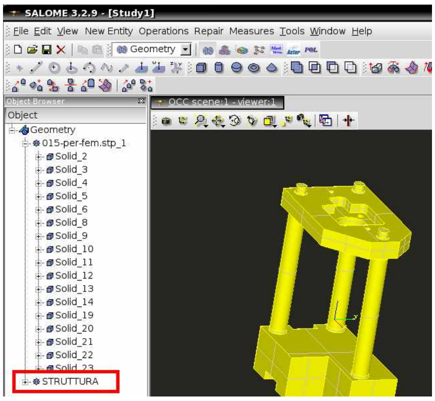


2.2 Compound Creation

Let's create a new compound with all and only remaining components and call it “STRUTTURA”. Now we have the correct compound to start to work on.







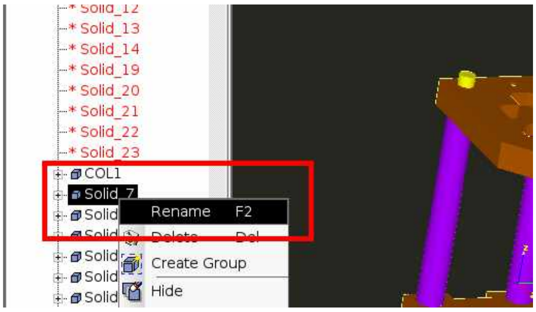
2.3 Extracting components and surfaces

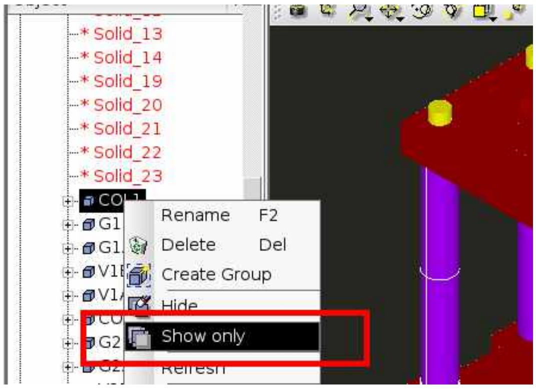
We explode the STRUTTURA compound as we seen before, and rename every components as we like (we can follow the schema provided in the LOAD_SCHEMA.pdf document). When this operation is finished, we go to the next step, that is the operation to extract surfaces from every components. Not all surfaces need to be extracted, but only:

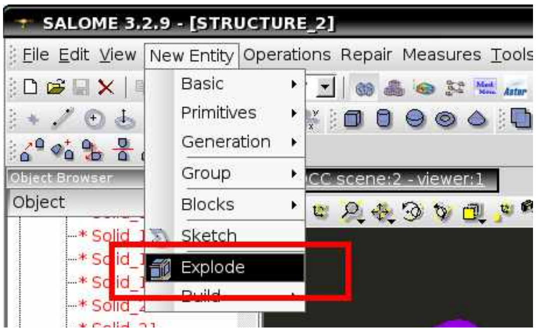
- Surfaces to be used as ground constraints or as mate constraints to other components.
- Surfaces to be used as load application areas
- Surfaces to be used to create submeshes

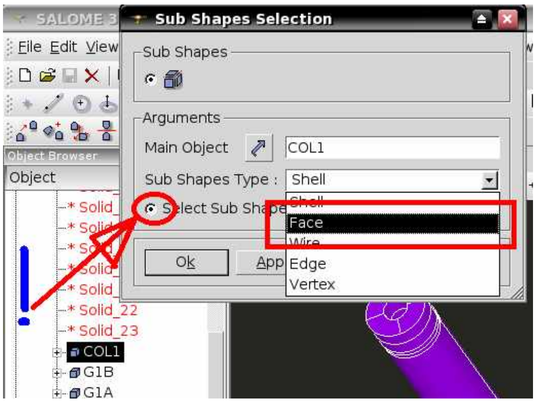
IMPORTANT NOTICE: Name to be used as name for components and surfaces are allowed to be long no more as 8 characters, and the character "-" is a source of troubles when running the code-aster analysis, use the "_" character instead.

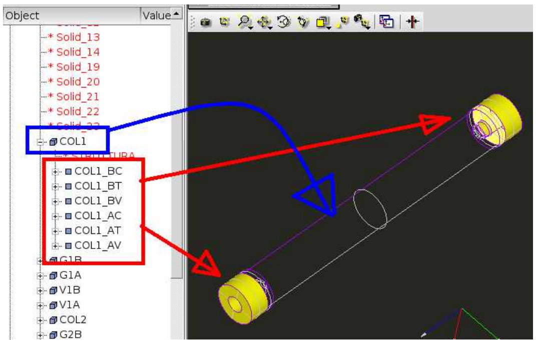
To make it easier the operation of surfaces extraction, it is better to enable the visibility of the only interested component, switching off the others. To select more surfaces, hold shift-click when clicking each surface. Look at the LOAD-SCHEMA.pdf file to understand what surfaces are to be extracted. By default, extracted surfaces are named as Face_1, Face_2, Face_3 etc by Salome, so they need to be renamed as shown in the provided schema. This work-flow is to be repeated for each component, so it could be necessary a long time to complete the task.











When these operations are completed, the work on the geometry is terminated. We save the file STRUTTURA.hdf and switch to the mesh module.

3 Mesh Creation

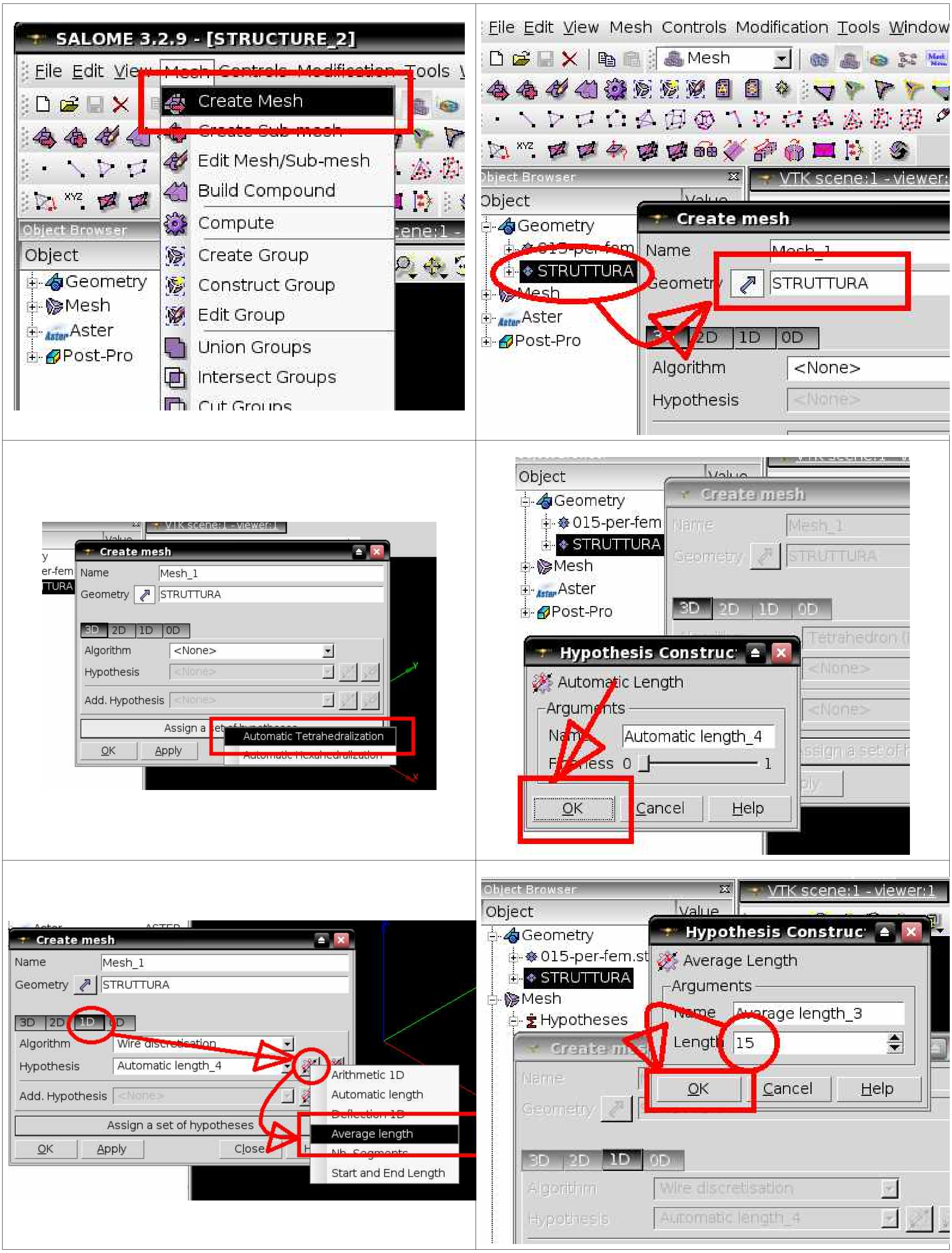
3.1 Preliminary note

Only tetrahedric elements are present in the mesh. Hesahedric elements and beam elements were not used.

3.2 Primary mesh creation

After the mesh module was activated, it's the time to create the primary mesh: we need to indicate to the software which geometry the mesh is to be calculated on, and the hypothesis to be used for the meshing process. It is better to avoid to indicate a too much fine mesh, to limit the time and the memory that Code_Aster will need to terminate the analysis.

As a calculation hypothesis, we choose “Automatic Tetrahedralization”, then switch to the tab “1D”, where we can impose a value of 15 for the “Average length” parameter: this means that the mesh elements will have a side of about 15 mm as an average value. It is a big length, so the mesh will be very coarse, but we will refine it later in some particular areas, because if a fine mesh was imposed on all the geometry, the number of elements could be too much high and the time needed to execute the analysis could increase too much.


SALOME 3.2.9 - [STRUCTURE_2]
File Edit View Mesh Controls Modification Tools
Create Mesh
Create Sub-mesh
Edit Mesh/Sub-mesh
Build Compound
Compute
Create Group
Construct Group
Edit Group
Union Groups
Intersect Groups
Cut Groups
Object Browser
Object
Geometry
Mesh
Aster
Post-Pro
File Edit View Mesh Controls Modification Tools Window
Create mesh
Name
Mesh_1
Geometry
STRUTTURA
3D 2D 1D 0D
Algorithm
<None>
Hypothesis
Add. Hypothesis
Assign a set of hypotheses
Automatic Tetrahedralization
Automatic Hexahedralization
OK
Apply
015-per-fem
Hypothesis Construction
Automatic Length
Arguments
Name
Automatic length_4
Fineness 0 1
OK Cancel Help
Tetrahedron
Wire discretisation
Automatic length_4
Arithmetic 1D
Automatic length
Deflection 1D
Average length
Nb. Segments
Start and End Length
Close
VTK scene:1 - viewer:1
015-per-fem.st
Hypotheses
Average Length
Average length_3
Length
15

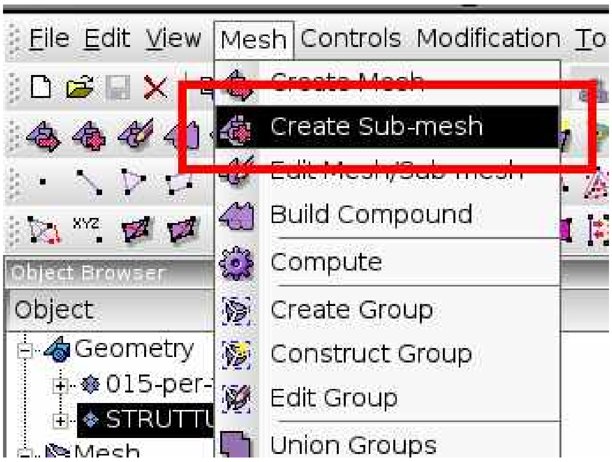
3.3 Submesh Creation

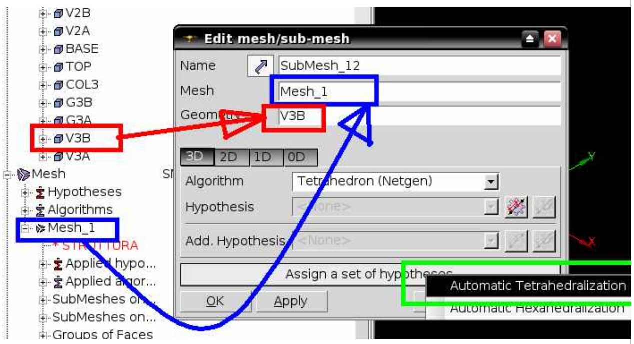
The process to create submeshes is quite the same compared to the one to create meshes: the only accessory information that the software asks to the operator is to indicate areas or volumes where submesh are to be applied. We will use solids and surfaces that were previously defined in the geometry module.

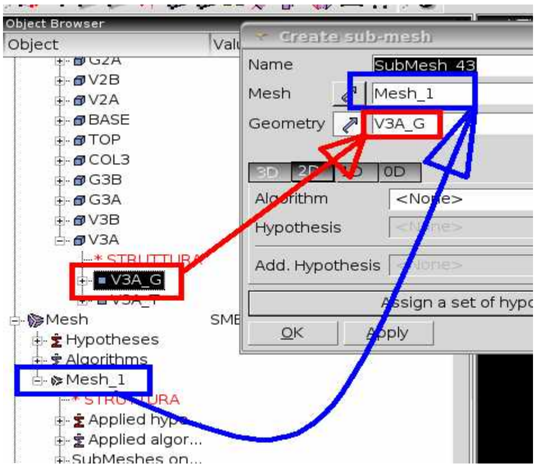
In this work, submeshes on solids were applied on each half-screw, then it means on the following 3d-groups: V1A, V2A, V3A, V1B, V2B, V3B, G1A, G2A, G3A, G1B, G2B, G3B, On the other side, submeshes on 2d groups were imposed on some relevant surfaces: basically surfaces by means of a component is in contact to the near one and areas where external load are applied. For a complete list please refer to the provided hdf file.

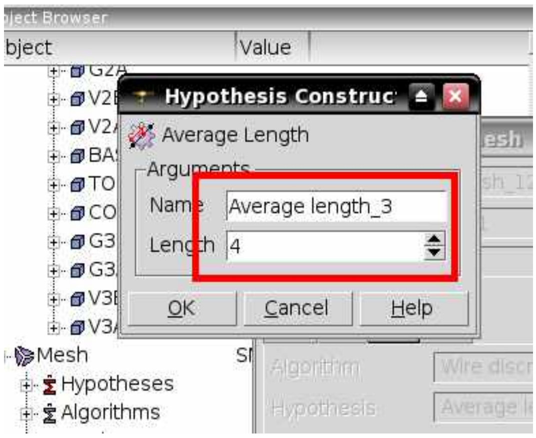
The same hypothesis was imposed to all submeshes: the average length has a value of 4 mm.

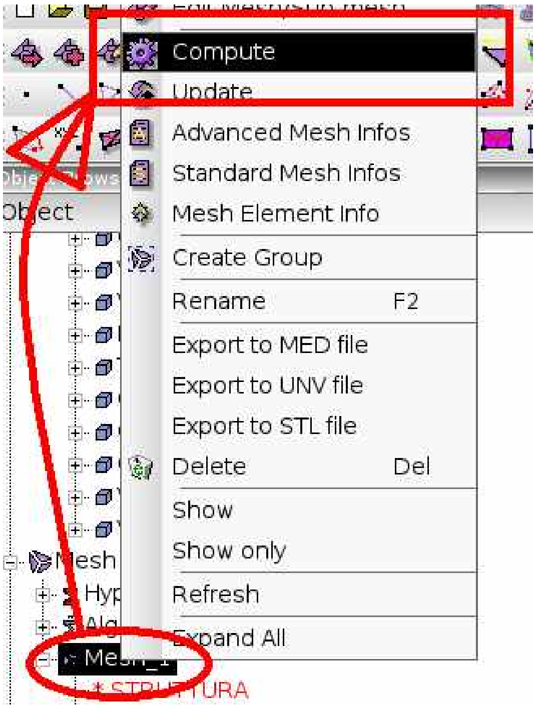
When the definition of mesh and submeshs is complete, we go to compute the mesh. It will need a few seconds (less then a minute).

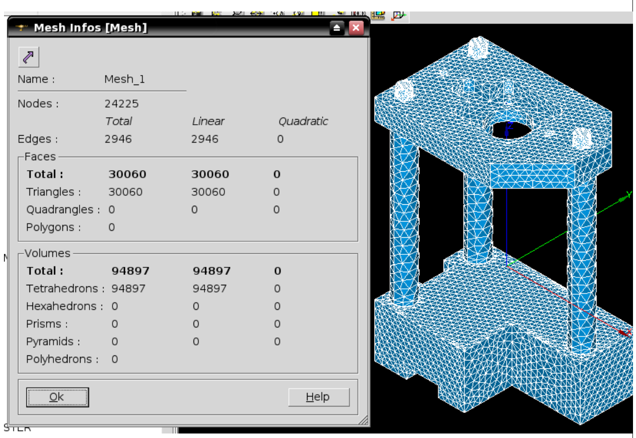

File Edit View Mesh Controls Modification To
Create Mesh
Create Sub-mesh
Edit Mesh/Sub-mesh
Build Compound
Compute
Create Group
Construct Group
Edit Group
Union Groups
Object Browser
Object
Geometry
015-per-
STRUTTU
Mesh


Edit mesh/sub-mesh
Name
SubMesh_12
Mesh
Mesh_1
V3B
3D 2D 1D 0D
Algorithm
Tetrahedron (Netgen)
Hypothesis
Add. Hypothesis
Assign a set of hypotheses
OK
Apply
Automatic Tetrahedralization
Automatic Hexahedralization
V2B
V2A
BASE
TOP
COL3
G3B
G3A
V3B
V3A
Mesh
Hypotheses
Algorithms
Mesh_1
STRUTTURA
SubMeshes on...
Groups of Faces


Object Browser
Object
Create sub-mesh
Name
SubMesh_43
Mesh
Mesh_1
Geometry
V3A_G
3D 2D 1D 0D
Algorithm
<None>
Hypothesis
<None>
Add. Hypothesis
<None>
Assign a set of hyp
OK
Apply
V2B
V2A
BASE
TOP
COL3
G3B
G3A
V3B
V3A
V3A_G
Mesh
Hypotheses
Algorithms
Mesh_1
STRUTTURA
Applied algor...
SubMeshes on...


Value
Hypothesis Construc
Average Length
Arguments
Name
Average length_3
Length
4
OK
Cancel
Help
Mesh
Hypotheses
Algorithms

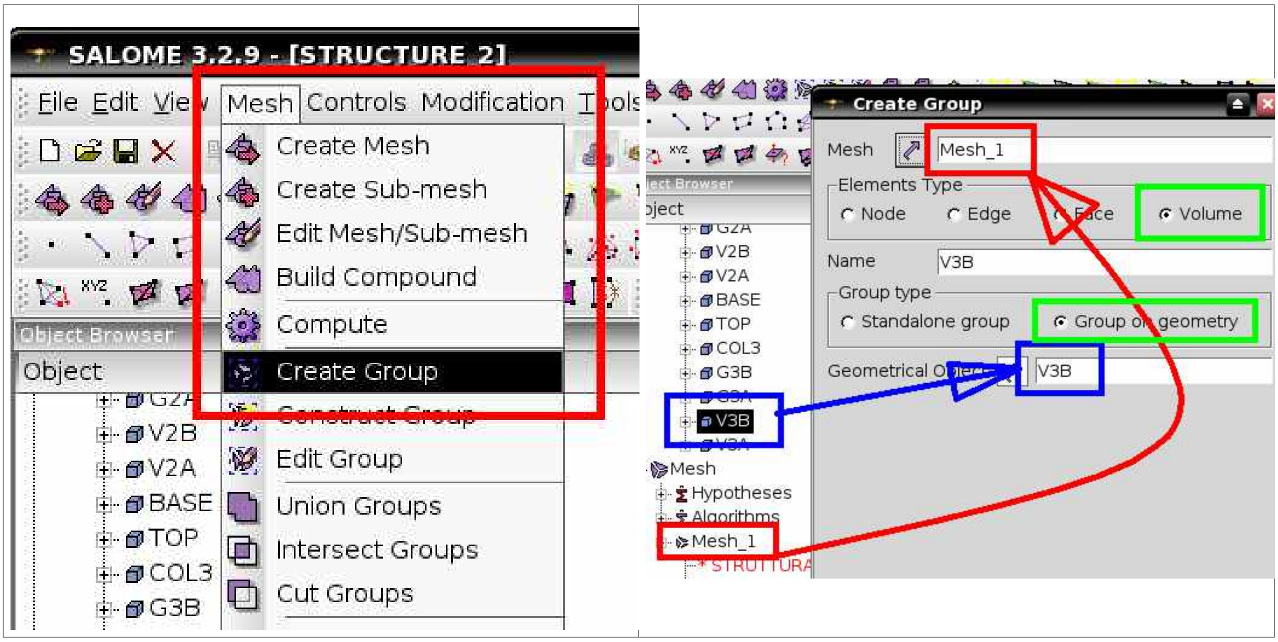

Compute
Update
Advanced Mesh Infos
Standard Mesh Infos
Mesh Element Info
Create Group
Rename F2
Export to MED file
Export to UNV file
Export to STL file
Delete Del
Show
Show only
Refresh
Expand All
Object
Mesh
Mesh_1
STRUTTURA


Mesh Infos [Mesh]
Name : Mesh_1
Nodes : 24225
Total Linear Quadratic
Edges : 2946 2946 0
Faces
Total : 30060 30060 0
Triangles : 30060 30060 0
Quadrangles : 0 0 0
Polygons : 0
Volumes
Total : 94897 94897 0
Tetrahedrons : 94897 94897 0
Hexahedrons : 0 0 0
Prisms : 0 0 0
Pyramids : 0 0 0
Polyhedrons : 0
Ok
Help

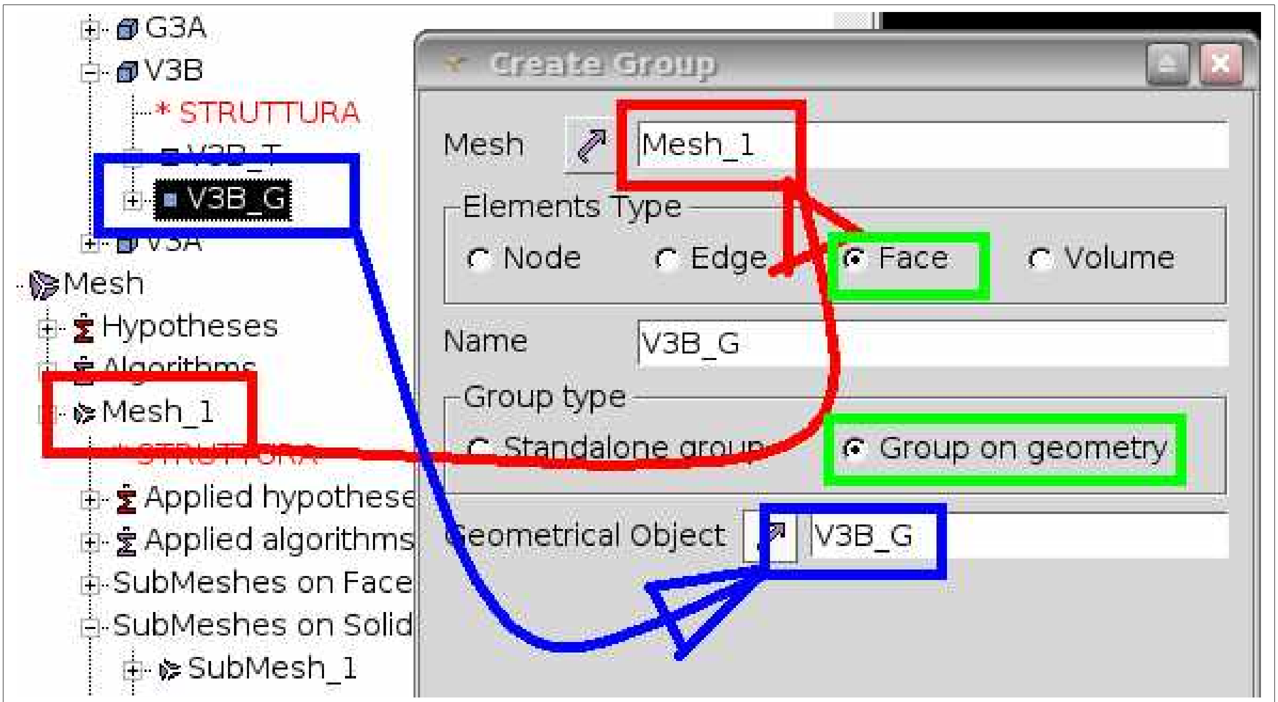
3.4 Creation of 3D groups of meshes

The need to create 2d and 3d groups of meshes is due to the need to have geometric entity where the fem solver will apply loads and constraints. The work-flow to build groups of meshes is the same for both 2d and 3d entities. By default, the software assigns to each group the same name of the geometrical entity they are based on. These entities are the ones we defined in the geometry module.

Please refer to the hdf example file for a list of the groups to create.



3.5 Creation of 2D groups of meshes



Now the work on mesh is complete. We save the hdf file and switch to the Aster module.

4 WRITING THE CODE-ASTER COMMAND FILE

4.1 What is the command file

The command file is a text file containing instruction and definitions used by code-aster to solve the problem starting to the mesh provided to it. Then it will contain the definition of the mesh, definition of various groups, definition of materials, definition of external loads and internal and external constraints, and, last, definition of the physical entities to be calculated and written as results of the calculation. The command file has "comm" extension.

NOTE: the text file is to be edited in windows environment if code-aster runs on windows, and on linux if code-aster runs on linux. This is due to the fact that the character of "end of line" is different in the two environments.

4.2 The EFICAS editor

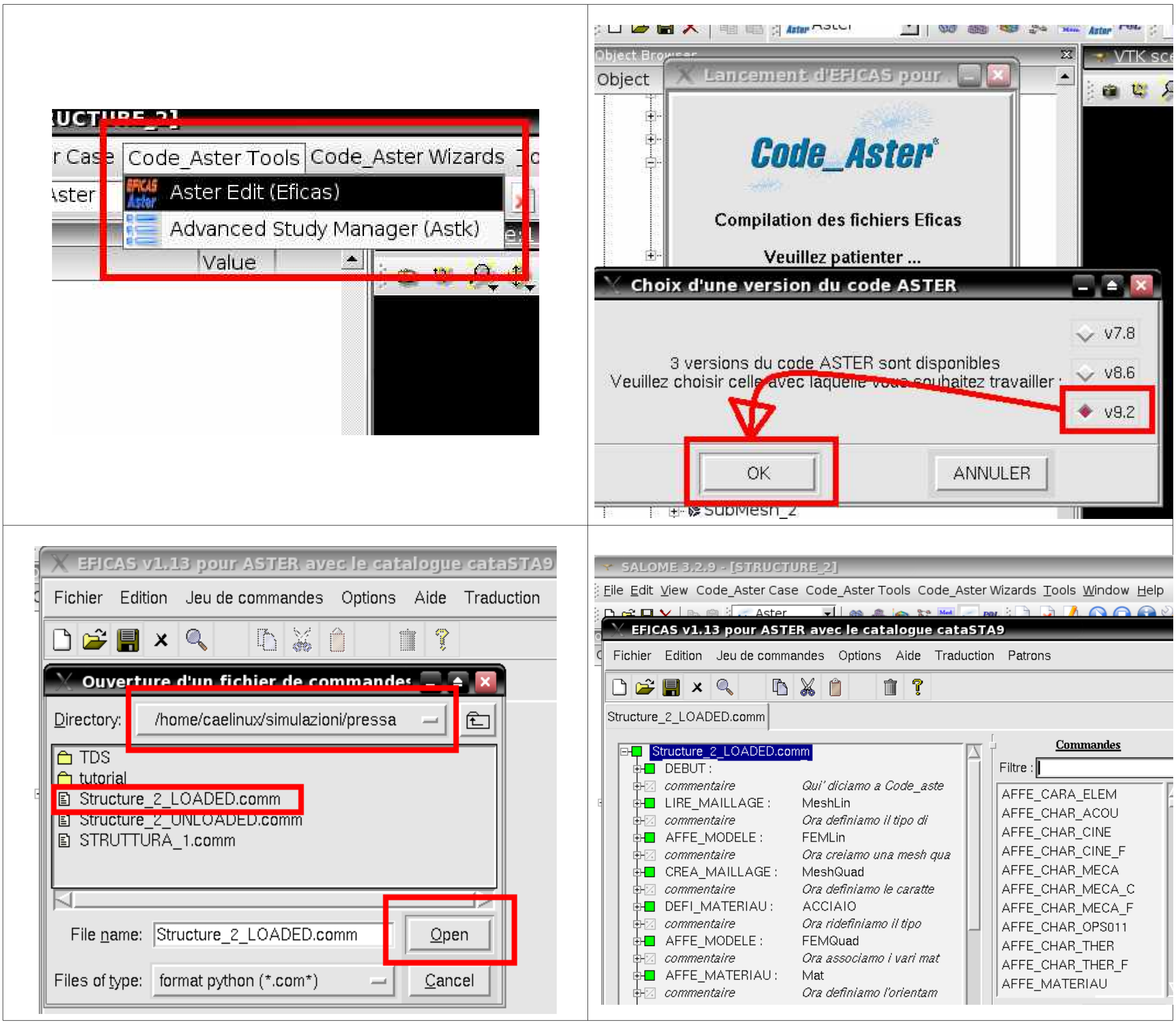
The Eficas editor is a tool that help us to write the command file in a various ways, giving us some advantages:

- Clear division between each section of the file
- Suggestion for the command to be used for each section
- Correct syntax when the comm file is written to the disk

It could be a good idea to start from an existing command file and to modify it to fit it to our need, we can find a lot of examples in the caelinux site or in the code-aster forum.

For this exercise two command files are given:

- Structure_2_UNLOADED.comm where only the screw-preloads are applied to the structure
- Structure_2_LOADED.comm where the screw-preloads and the actuator force (20 kN) are applied to the structure.



For this case, a simple linear analysis is performed.

4.3 Description of the sections of the command file

The command file is shared into sections:

- ➔ DEBUT: is the starting instruction
- ➔ LIRE_MAILLAGE: In this section we tell to code aster what is the mesh to be read. The standard mesh format for salome is MED.
- ➔ AFFE_MODELE: In this section, we define which mesh groups are to be subjected to a 3D modelization. So we will add all 3d groups, all 2d groups where loads are applied and, last, all the 2d groups to be used as “ESCLAVE” (see AFFE_CHAR_MECA section) which are the surfaces that moves under the action of an other 3d group.
- ➔ CREA_MAILLAGE: Here we give to code-aster the instruction to transform the original mesh in a quadratic one: it means we create a MeshQuad from a MeshLin.
- ➔ DEFI_MATERIAU: In this section we define the different materials to be used for each 3d group of the mesh. In this case, the same material was used for all components, so it is necessary to define just one material. The only information needed to solve this problem are the Young Module and the Poisson coefficient.
- ➔ AFFE_MODELE: It is the same of the previous operation, but applied on the MeshQuad instead of the MeshLin.
- ➔ AFFE_MATERIAU: In this section, we define the association between each material and each 3d-group. In this case the same

material is assigned to all parts, so we use the parameter:

TOUT: OUI (All: Yes) .

➔ MODI_MAILLAGE: We have to define the parameter ORIE_PEAU_3D: it means we need to list some 2d-groups to which the correct normal-orientation will be imposed by the software. In the list there will be all surfaces where external loads are applied and all the surfaces used as ESCLAVE in the constraints.

➔ AFFE_CHAR_MECA: It is the more important section in the file, where loads and constraints are defined. In this example we find:

- DDL_IMPO: It means to impose a certain DOF to a certain group. To impose DX=0, DY=0, DZ=0 means to give the ground constraint to a 2d group of the mesh.
- LIAISON_MAIL: It is the place where we define the mating conditions between group of meshes:
  - GROUP_MA_MAIT: It is the moving body in the mating (MAITRE = Mother) and it is always a 3d group.
  - GROUP_MA_ESCL: It is the leaving surface (ESCLAVE = Slave) and it is always a 2d group.
  - DDL_MAIT, DDL_ESCL, DNOR: If this parameter is given, it means that contacting surfaces of MAIT and ESCLAVE can slide one onto the other, but without loosing the contact. If the parameter is absent, it means that surfaces can't move, so they behave as they were welded together.
- FORCE_FACE: Here the external load applied on every surfaces

(2d-group of mesh) is described. Force is defined by its direction (Z Axis in this case) and its module. **NOTE**: Module is to be intended as force per unit of surface, so MPa in this case.

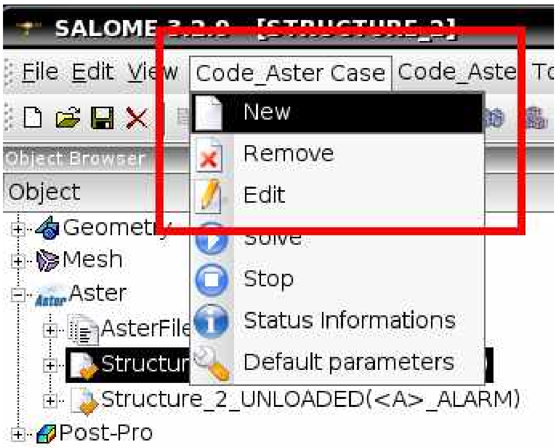
- ➔ MECA_STATIQUE: Here we tell to code-aster how is the case, declaring boundary conditions (vincoli) and materials to be used (Mat).
- ➔ CALC_ELEM: Here we ask to code-aster what results we aspect from the calculation.
- ➔ PROJ_CHAMP: It means that we want that results calculated on the MeshQuad are to be projected on the MeshLin.
- ➔ IMPR_RESU: Here we ask to code aster to project on the MeshLin only some results. In this case for example:
  - DEPL Displacement
  - SIGM_NOEU_DEPL Stresses calculated according to nodes displacement
  - EQUI_NOEU_SIGM Principal and equivalent stresses in the nodes.
- ➔ FIN Command to terminate the case (Fin = end in french)

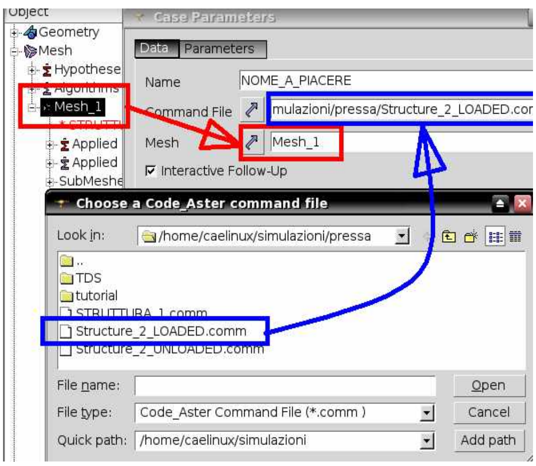
5. Analysis of execution

5.1 New case creation

In order to create a new case, we need to indicate:

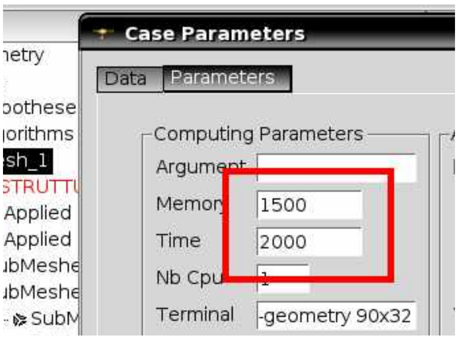
- Case Name
- Command file
- Mesh to be used



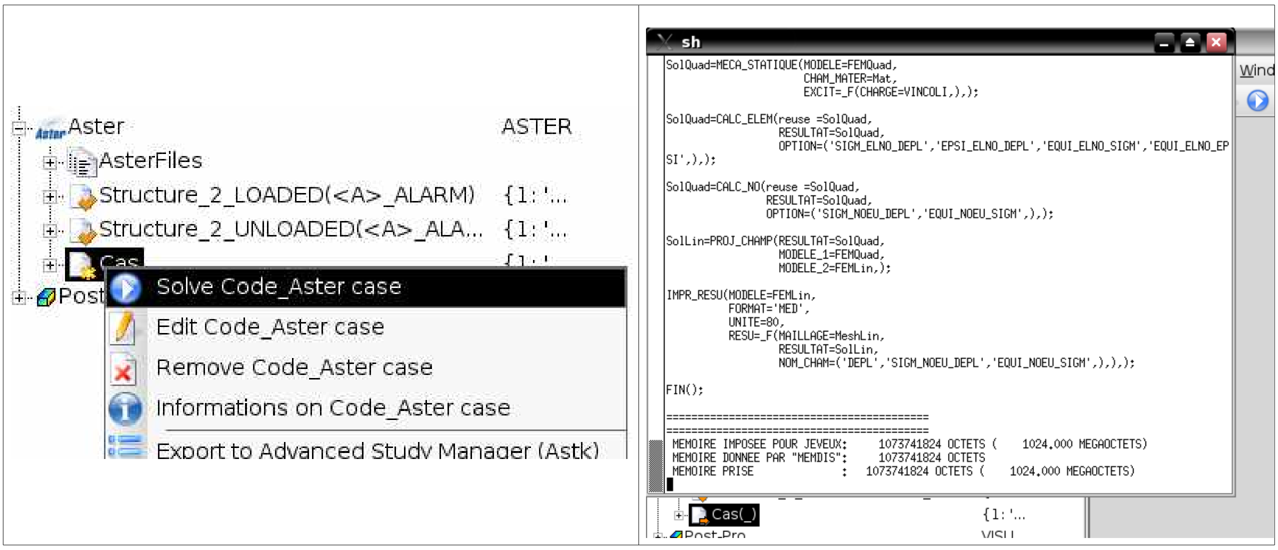


5.2 Standard parameters modifcation

It is necessary to modify the code-aster parameter (time limit and memory limit) given by default on CaeLinux, otherwise the solver is not able to complete the analysis. We put 1500 MB as memory limit a 2000 seconds as time limit.

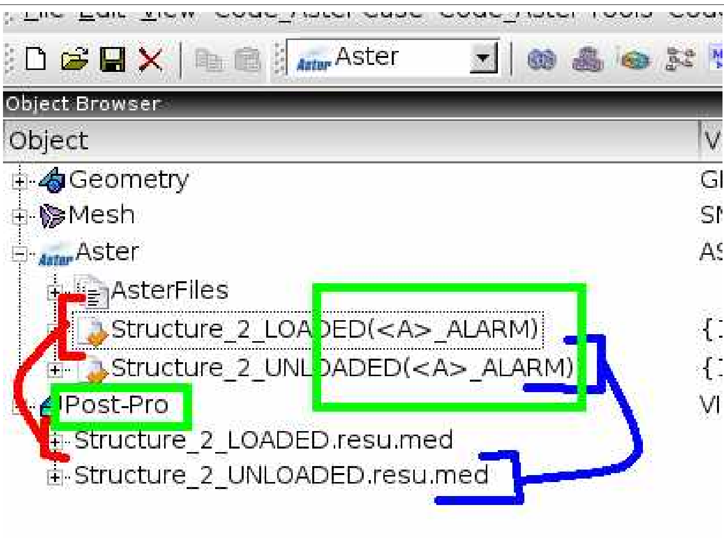


Now we can run Code-Aster and wait until it complete calculation.



5.3 Calculation End

If all went gone, we will find a sign “OK” near the name of the case in the left pane of Salome, otherwise a tag “ALARM” if the system found troubles for which it gives us a warning.



On a laptop with a Intel Centrino processor running at 1.5 GHz and 2 GB of RAM, the execution needs about 20 minutes to be completed.

Now it's time to switch to Post-Pro module.

6 SHOWING RESULTS IN POST-PRO

6.1 What are available fields

In the post-pro we have all the results that we asked to code-aster to calculate for us:

a) DEPL: Desplacements, splitted into:

- Modulus
- DX
- DY
- DZ

b) EQUI_NOEU_SIGM: Stress in the elements, available as:

- Modulus: Module
- VMIS: Von Mises Equiv. Stress
- TRESCA: Tresca Equiv. Stress
- PRIN_1, PRIN_2, PRIN_3: Principal Stresses
- VMIS_SG: Product of VMIS for the sign of the princ. stress

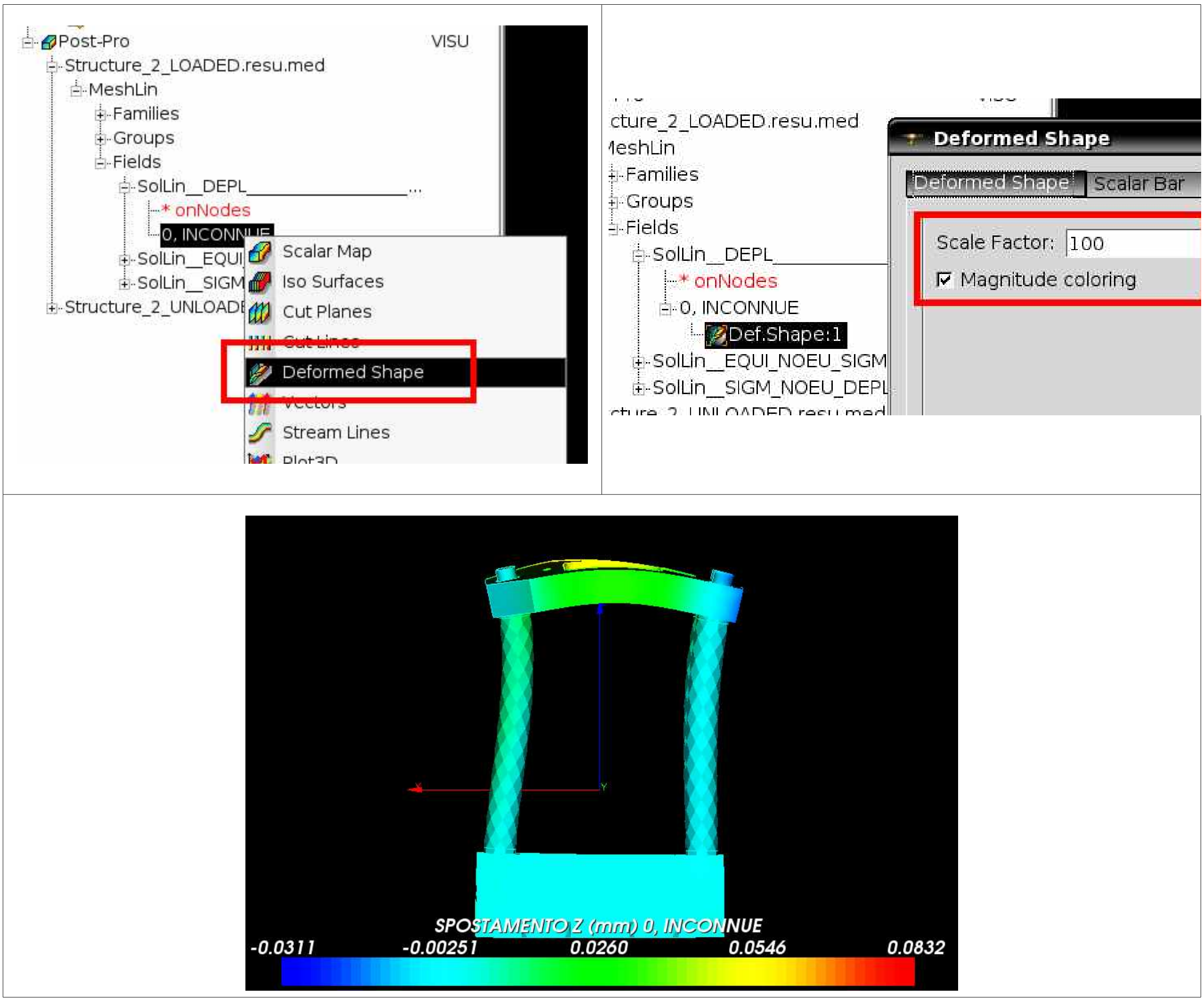
c) SIGM_NOEU_DEPL: Stresses calculated based on nodes displacements

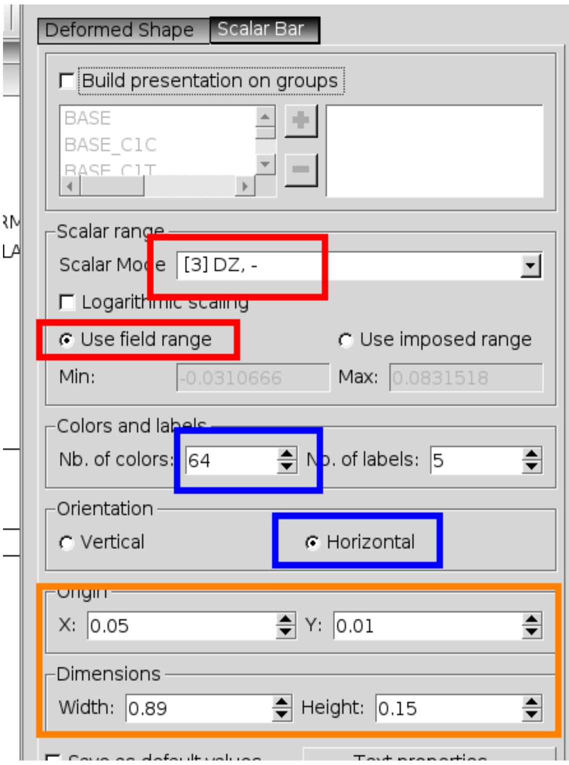
- SIXX, SIYY, SIZZ
- SIXY, SIXZ, SIYZ

6.2. How to create and manage visualization

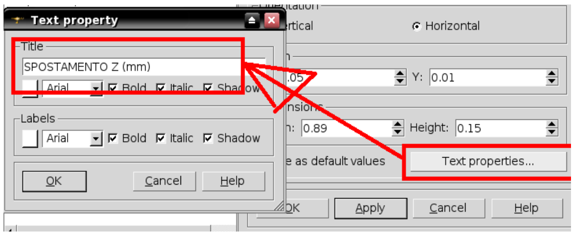
When we expand the post-pro tree, we find the word “INCONNUE” under each field.

For example, let's see how it is possible to show the displacement of the structure under the external load. We have various different kinds of representation, let's choose the “Defomed shape”: we can decide the amplification factor, the name and position for the range bar, if we want to view all the assembly or just only one or more components.

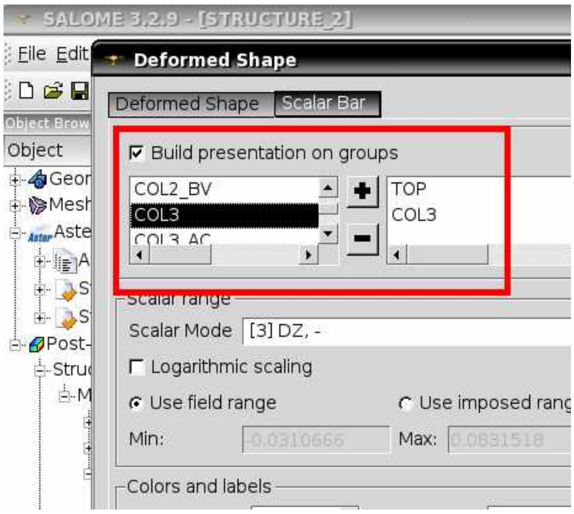




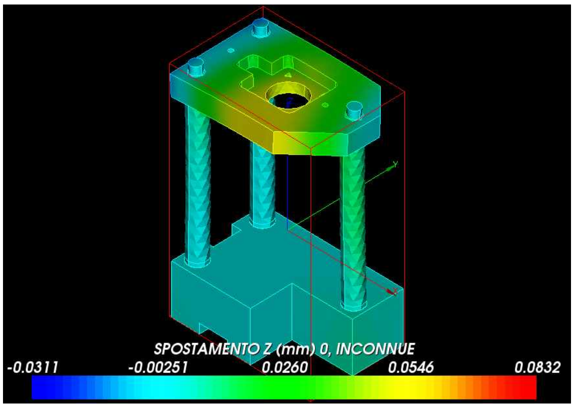
Deformed Shape
Scalar Bar
Build presentation on groups
BASE
BASE_C1C
BASE_C1T
Scalar range
Scalar Mode
[3] DZ, -
Logarithmic scaling
Use field range
Use imposed range
Min:
-0.0310666
Max:
0.0831518
Colors and labels
Nb. of colors:
64
Nb. of labels:
5
Orientation
Vertical
Horizontal
Origin
X:
0.05
Y:
0.01
Dimensions
Width:
0.89
Height:
0.15



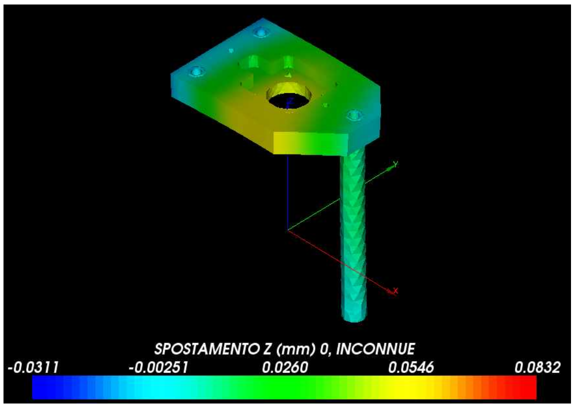
Text property
Title
SPOSTAMENTO Z (mm)
Arial
Bold
Italic
Shadow
Labels
OK
Cancel
Help
Horizontal
Y:
0.01
0.89
Height:
0.15
Text properties...
Apply



SALOME 3.2.9 - [STRUCTURE_2]
File
Edit
Deformed Shape
Scalar Bar
Object
Build presentation on groups
COL2_BV
COL3
COL3_AC
TOP
COL3
Scalar range
Scalar Mode
[3] DZ, -
Logarithmic scaling
Use field range
Min:
-0.0310666
Max:
0.0831518
Colors and labels



SPOSTAMENTO Z (mm) 0, INCONNUE
-0.0311
-0.00251
0.0260
0.0546
0.0832



SPOSTAMENTO Z (mm) 0, INCONNUE
-0.0311
-0.00251
0.0260
0.0546
0.0832

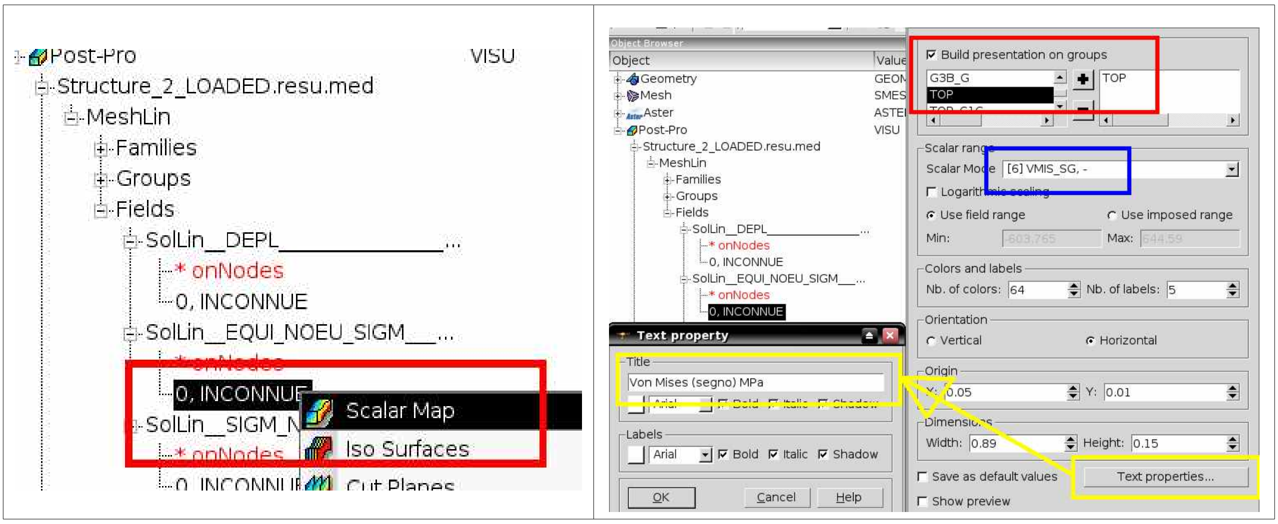
7 NUMERICAL RESULTS ANALYSIS

7.1 Contact verification

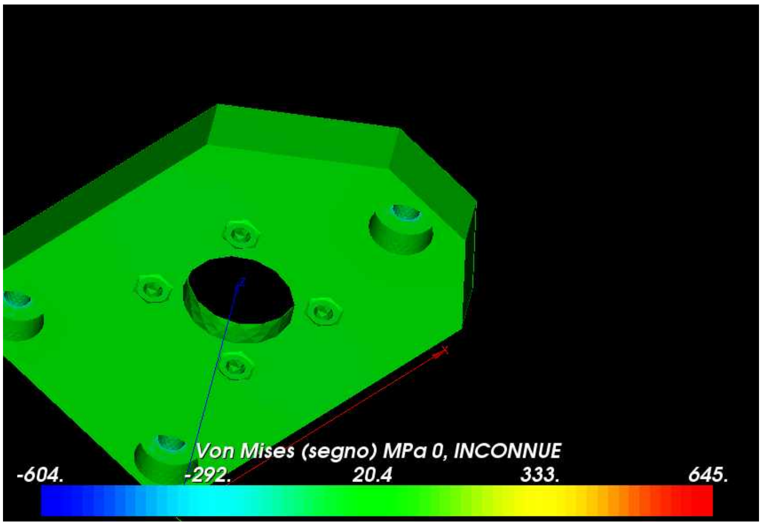
The constraints acting between parts connected by means of screw react in both senses: tension and compression. This is important to understand, mainly where the flat surfaces of the columns are contacting the flat bottom surface of the hole where they are fitted in. It is clear that in the real world the behavior of a such sort of coupling is totally different: the constraint react only against a compression, but it is free under a traction. In the real machines, it is mandatory that pre-load on screws must be always bigger than the tension due to external load, in order to avoid separation of components. So, the first step of our analysis could be this one: to investigate if the preload on the screws is enough to maintain columns and flange in contact even if all the external load is applied. If this check is ok, we can assume that the type of constraint, even if it is "philosophically" wrong, does not affect too much the correctness of the simulation.

A parameter that we can use for this could be VMIS_SG showed at the bottom surface on the column housings or at the top surfaces of the columns. If we find a negative value (i.e. a compression) we are ok.

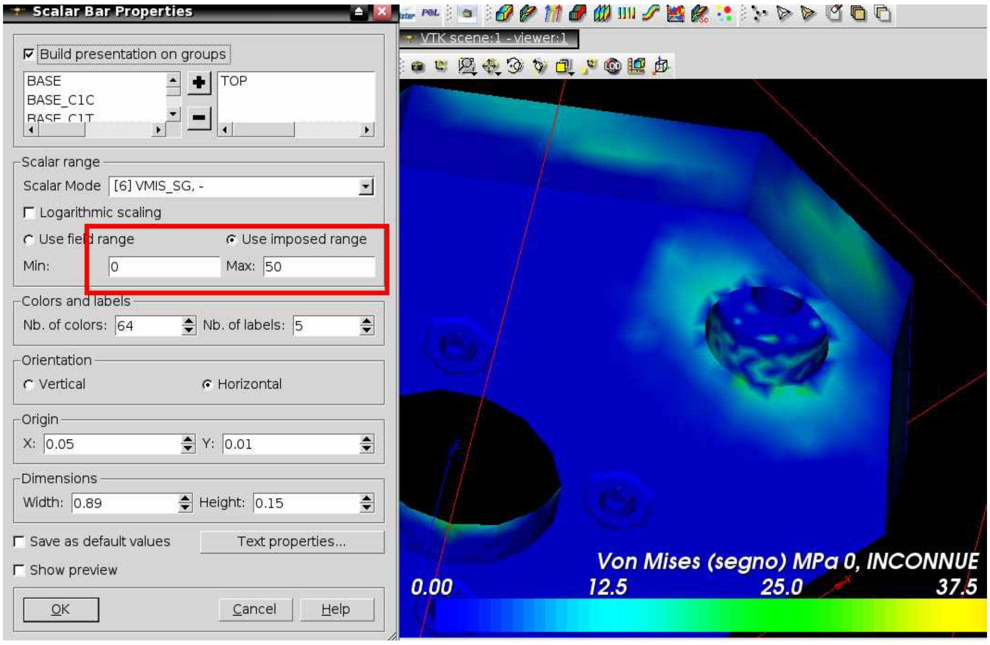
So, let's show the VMIS_SG value, related to the TOP component only.



We obtain the following image:

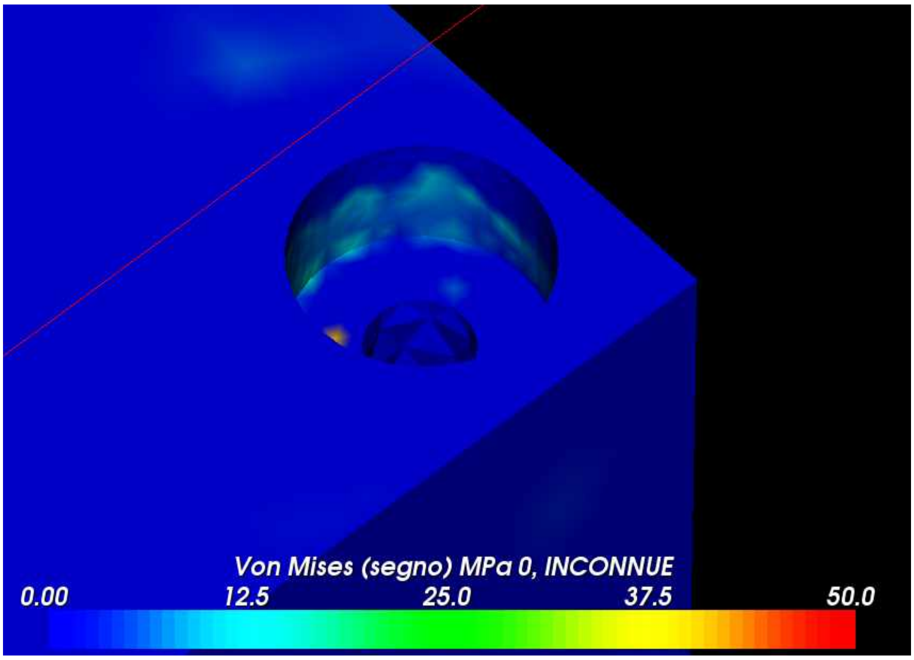


We note that the range proposed by the software is too much wide for our needs. We have to modify the visualization imposing a range enter 0 and 50 MPa:



Here we see that at the bottom of the hole, the value of the tension is 0 or less.

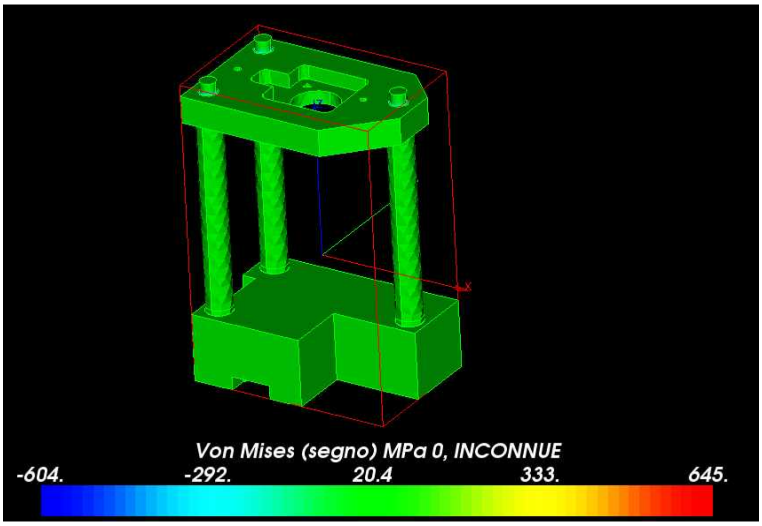
Let's make the same verification on the BASE component:



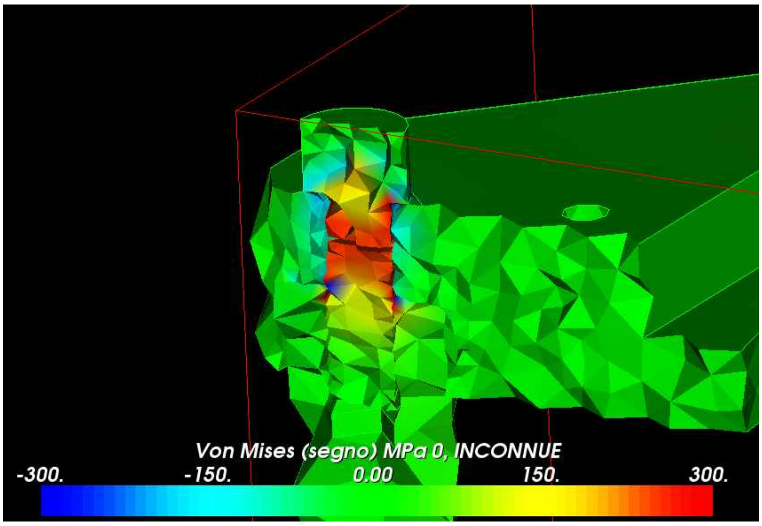
Here the stress value is 0 or less than 0 too. We can be reasonably sure that the behavior is correct, and the initial assumption was ok.

7.2 Tensions verification

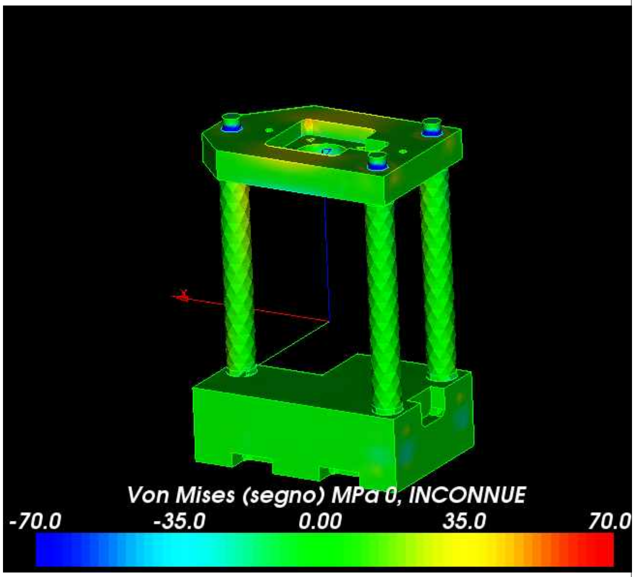
Let's show the VMIS_SG value on the whole structure:

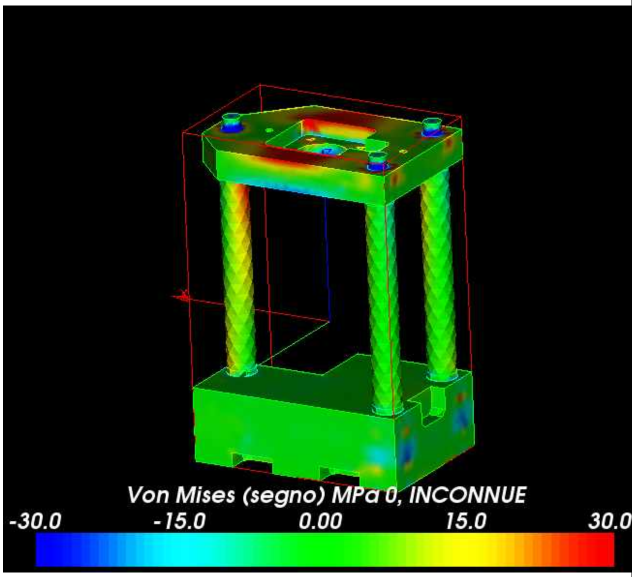


The entire structure seems to be not charged. Actually is a problem in the too much wide range of the visualization. Let's impose a range -300/+300 and let's activate a clipping plane along the column COL3 axis:



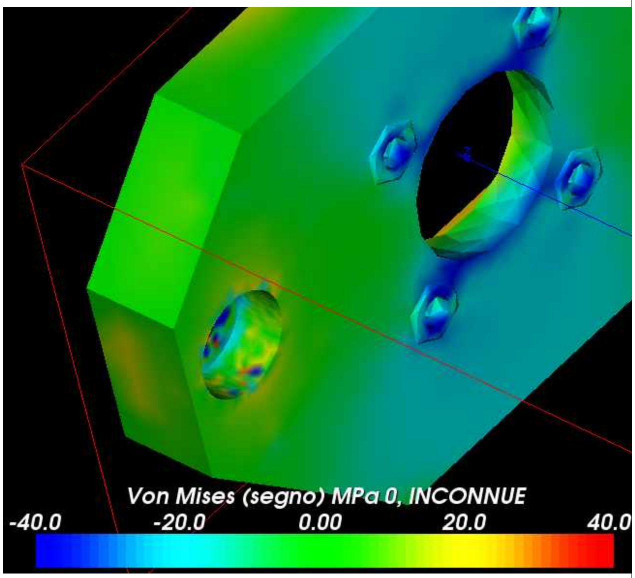
We understand that the stress in the body screw is around 200-250 MPa, as we imposed as boundary condition. The TOP component seems to be not stressed, but for the action of the preload of the screws. Let's make two visualizations with different range: -70/+70 and -30/+30:

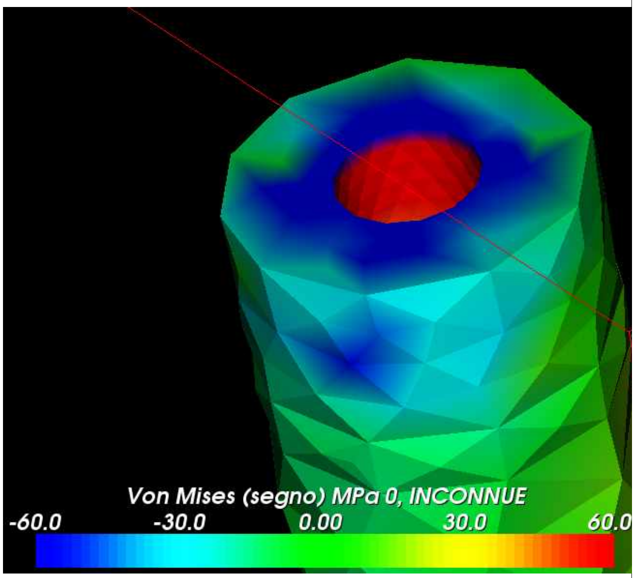




Here it is clear that the maximum stress level in the TOP component is around 30 MPa, while in the front column it is around 15 MPa.

Now let's try to see what's happen to the components that are most likely more stressed: TOP and COL3.

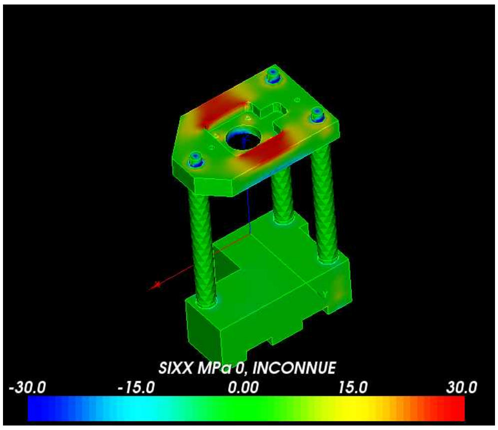


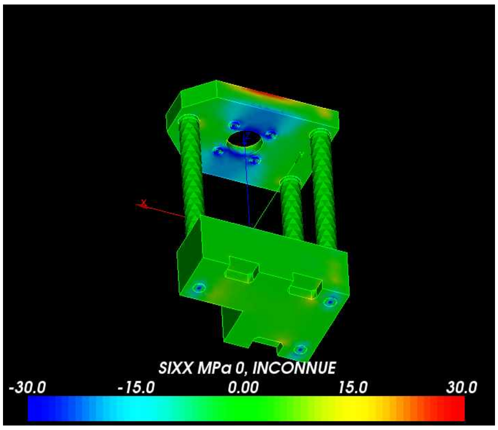


It is notable that in the TOP component the maximum stress area is located near the points of the connection of the cylinder. In the component COL3 the maximum tensile stress is located in the threaded hole but we find a little compression at the contact with the cylindrical surface of its housing. This is due to the restrain formed by the column and its housing and the bending of the column consequent to the bending of the TOP component.

Anyway the stress values are quite low if compared to the strength of the material.

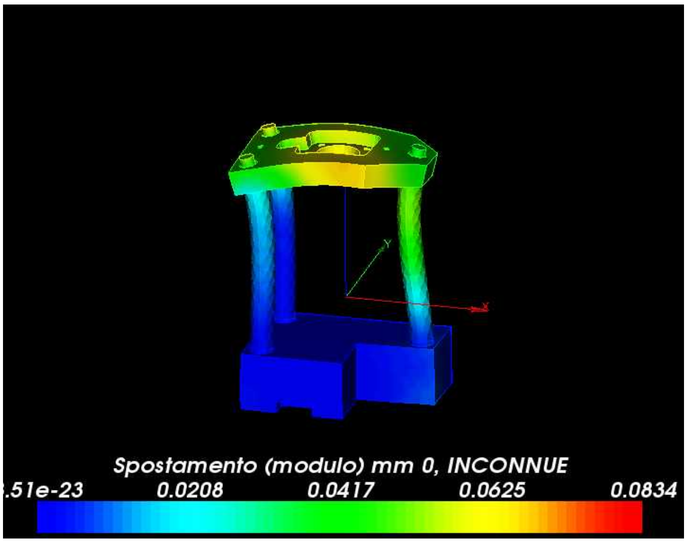
As a last step, we can create a visualization where the SIXX is showed: this give us a direct idea of the bending stress on the TOP component:



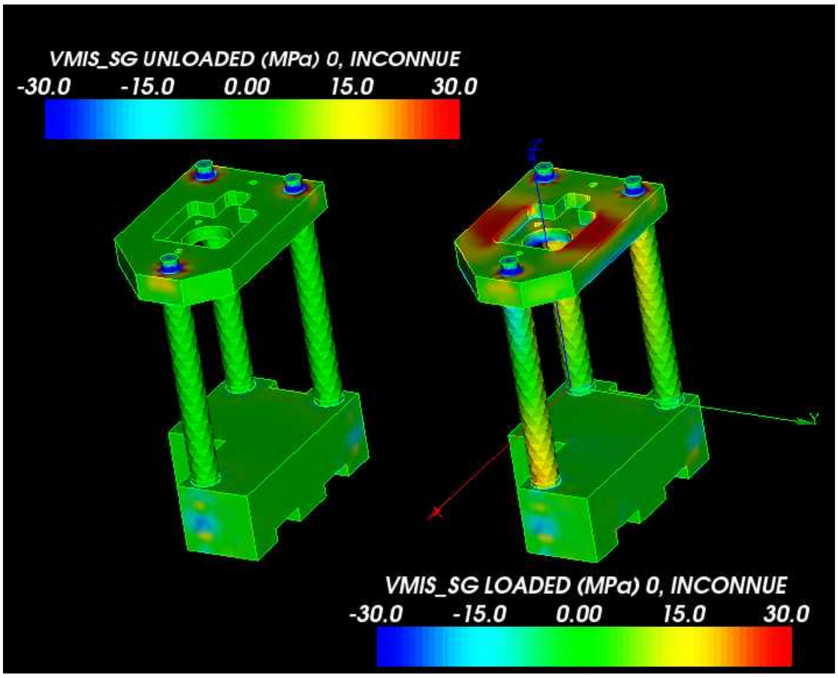


A further assurance that the stress values are moderate.

Here, a representation of the deformed shape with an amplification factor of 500:



7.3 Differences between unloaded and loaded structure



Here the unloaded structure is compared to the same but with the load applied. We note the bending of the TOP component.

8. ACKNOWLEDGMENTS

I would like to thank:

Matteo, member of www.cad3d.it, for his explication about how to apply pre-load on screws.

Gerod, member of www.cad3.it for reviewing this document.

Thomas De Soza member of the forum www.code-aster.org for help me to understand the meaning and the syntax of the comm file.