An introduction to Salome & Code _Aster: A simple linear statics analysis of a piston

J.Cugnoni, CAElinux.com , 2005

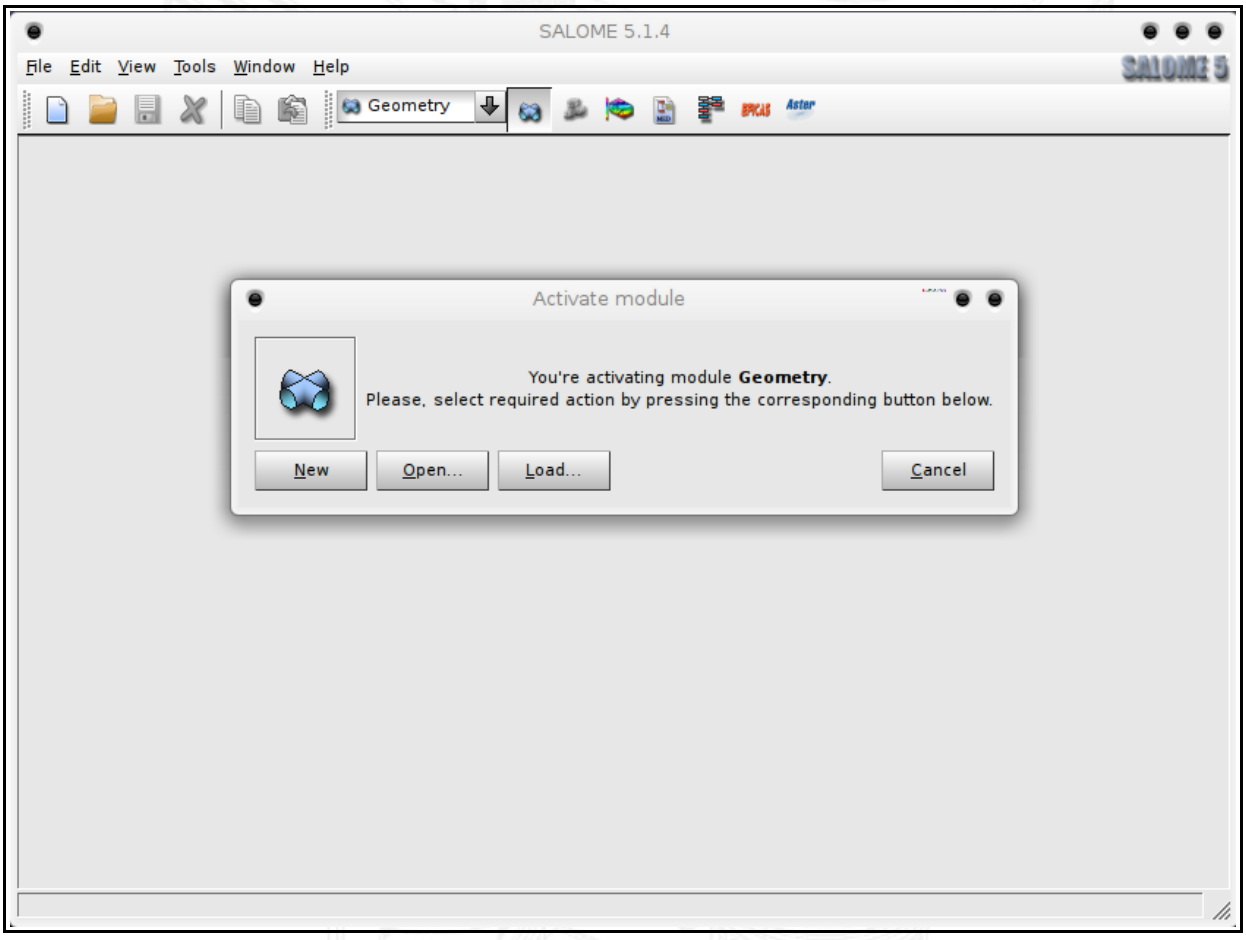
Updated to comply with SaloméMECA 2010.2
by
C.Andersen, for CAELinux.com, 2010

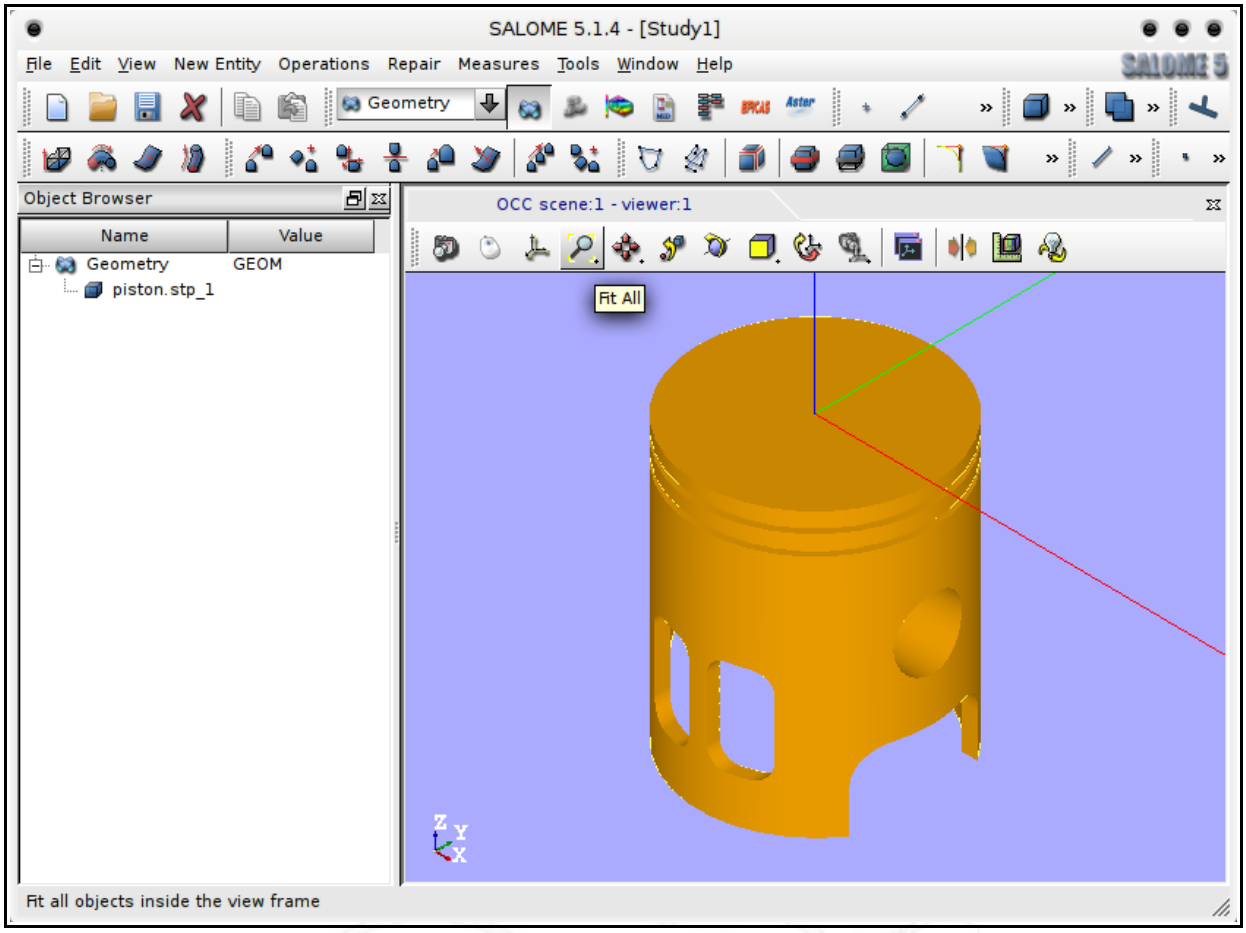
Overview:

- **Geometry module:**
  - Import and prepare STEP file format geometry, group creation
- **Mesh module:**
  - Meshing geometry in preparation for FE study
- **Aster module:**
  - Creating a linear statics study using the wizard
  - Running the calculation
  - Viewing the run-time performance results
  - Editing a command file with Eficas
- **Post-processing module (VISU)**
  - Post-processing the result of the study

Geometry module: Importing geometry

- Start SaloméMECA and select **Geometry** module, choose **New** in the dialog
- Select menu **File → Import**
  - Choose **STEP** file format in the pull-down menu
  - Select the file **Piston.stp**
- Click the **Fit All** icon in the View tool-bar
- If the geometry is presented as a wire-frame, click menu **View → View Mode → Shading**



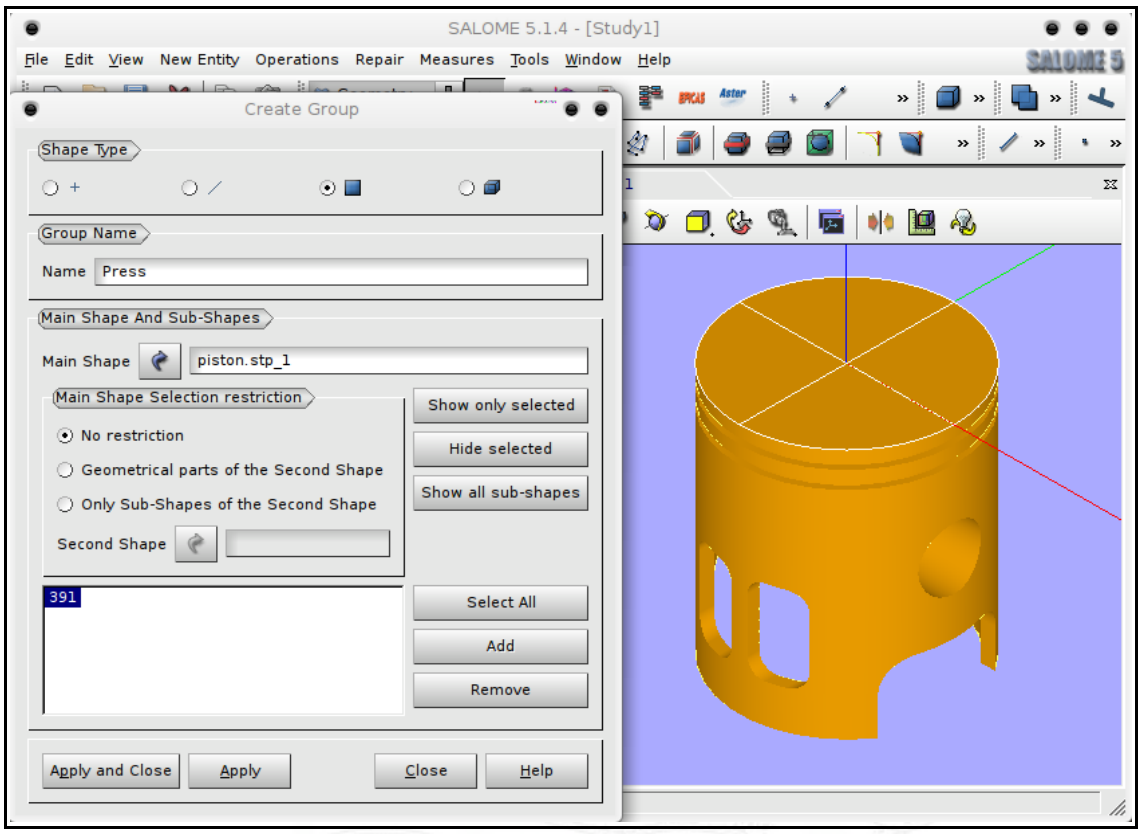


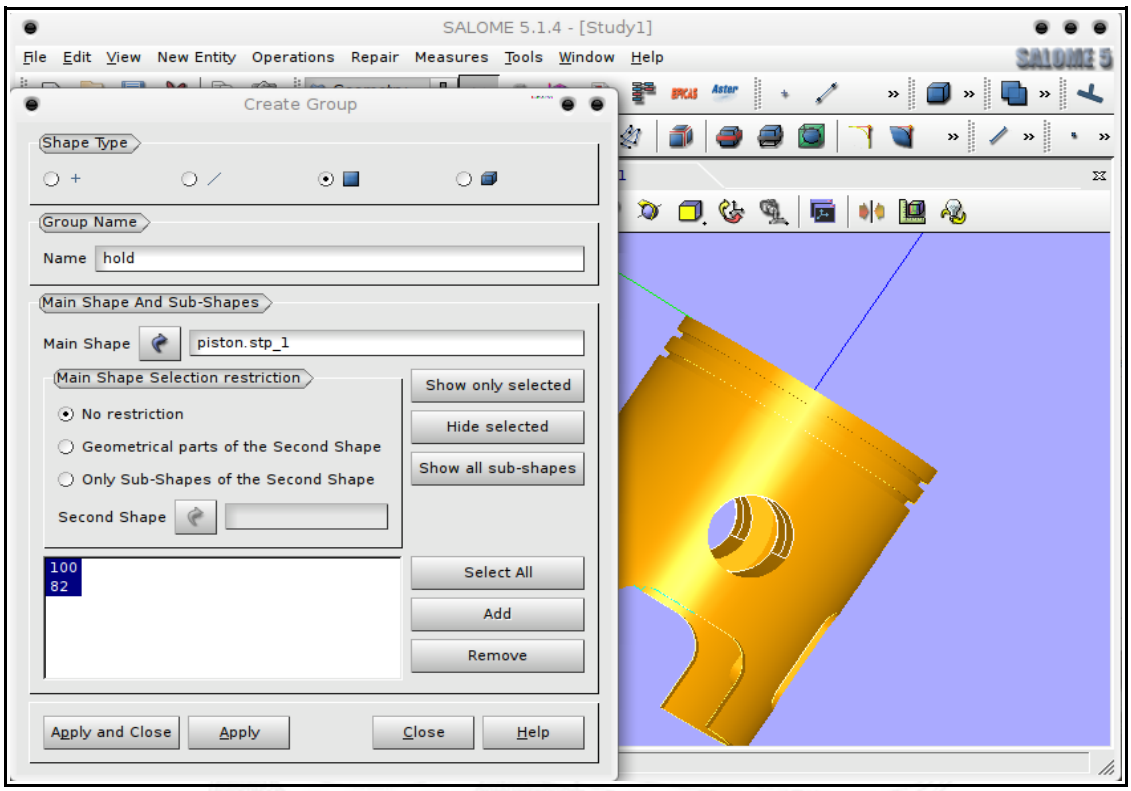
A note on importing geometry: When importing geometry from e.g. SolidWorks or other CAD packages, Salomé converts the units from**[mm]** to **[m]** to conform with the **mKs** SI-system – This means that if the piston was designed in SolidWorks to have a diameter of 60mm, the imported geometry will have a diameter of 0.06m – i.e. the coordinates along the e.g. x-axis will not be (0,0,0) → (60,0,0) in Salomé, but (0,0,0) → (0.06,0,0)

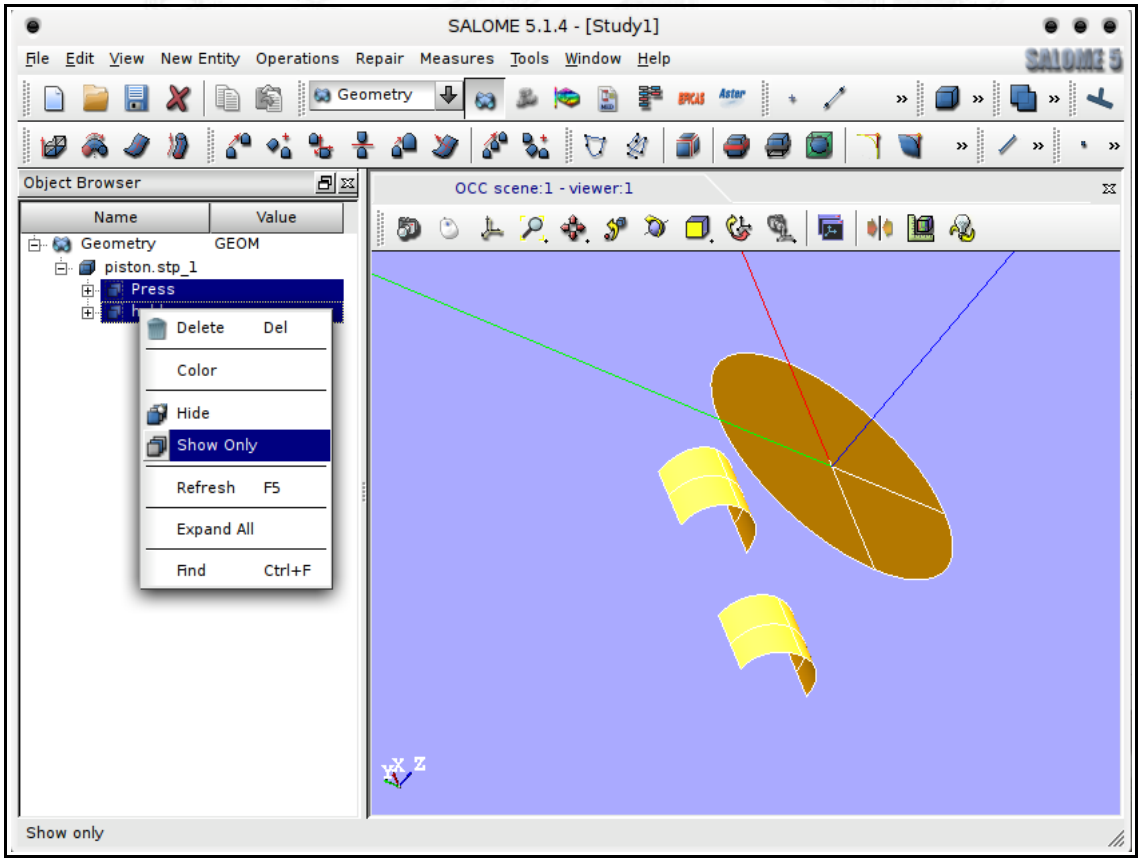
If in doubt, check the bounding box by going to menu **Measures → Dimensions → Bounding Box.**

Geometry module: Preparing geometry; creating groups

- Expand Geometry in the object browser and right-click **Piston.stp_1** and select **Create Group**
- Select Shape Type **Face**
  - Under **Group Name** enter '**Press**'
  - Move the mouse pointer over the piston geometry and press *LMB* to select the top, circular face.
  - Once the top, circular face is selected, press **Add**, followed by **Apply**.
  - The geometry group '**Press**' is now created
- Creating the '**hold**' group
  - Enter the name '**hold**' for this new group
  - Pressing and holding *CTRL* while using *RMB* to rotate the piston geometry, select the two upper halves of the holes going through the piston – hold *SHIFT* to select two individual faces
  - Click **Add**, followed by **Apply and Close**
- To make sure the correct face groups have been created, expand **piston.stp_1** in the object browser, select both groups (pressing and holding *CTRL* or *SHIFT*) in the object browser, right-click and select **Show Only**
- To return to a normal view, right-click **piston.stp_1** and select **Show Only**

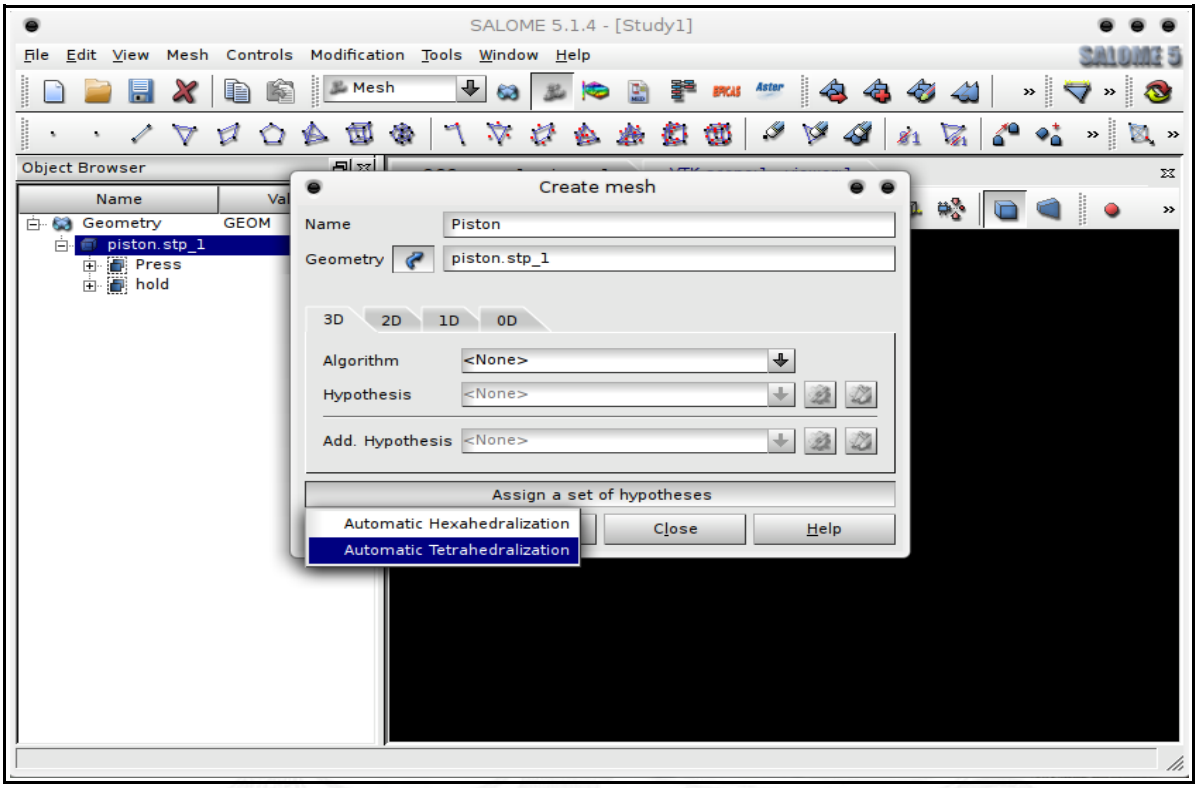

SALOME 5.1.4 - [Study1]
File Edit View New Entity Operations Repair Measures Tools Window Help
SALOME 5
Create Group
Shape Type
Group Name
Name Press
Main Shape And Sub-Shapes
Main Shape
piston.stp_1
Main Shape Selection restriction
No restriction
Geometrical parts of the Second Shape
Only Sub-Shapes of the Second Shape
Second Shape
Show only selected
Hide selected
Show all sub-shapes
391
Select All
Add
Remove
Apply and Close
Apply
Close
Help

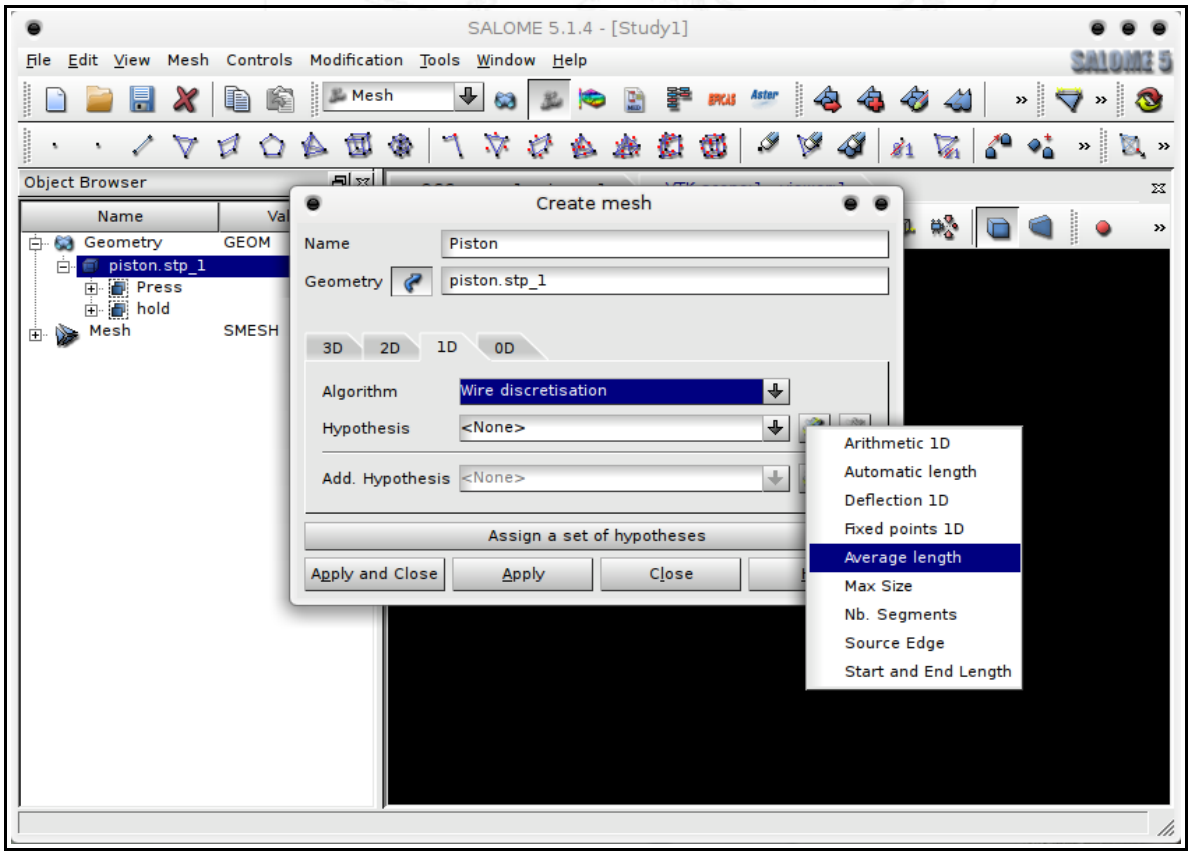

SALOME 5.1.4 - [Study1]
File Edit View New Entity Operations Repair Measures Tools Window Help
Create Group
Shape Type
Group Name
Name hold
Main Shape And Sub-Shapes
Main Shape piston.stp_1
Main Shape Selection restriction
No restriction
Geometrical parts of the Second Shape
Only Sub-Shapes of the Second Shape
Second Shape
Show only selected
Hide selected
Show all sub-shapes
100
82
Select All
Add
Remove
Apply and Close
Apply
Close
Help

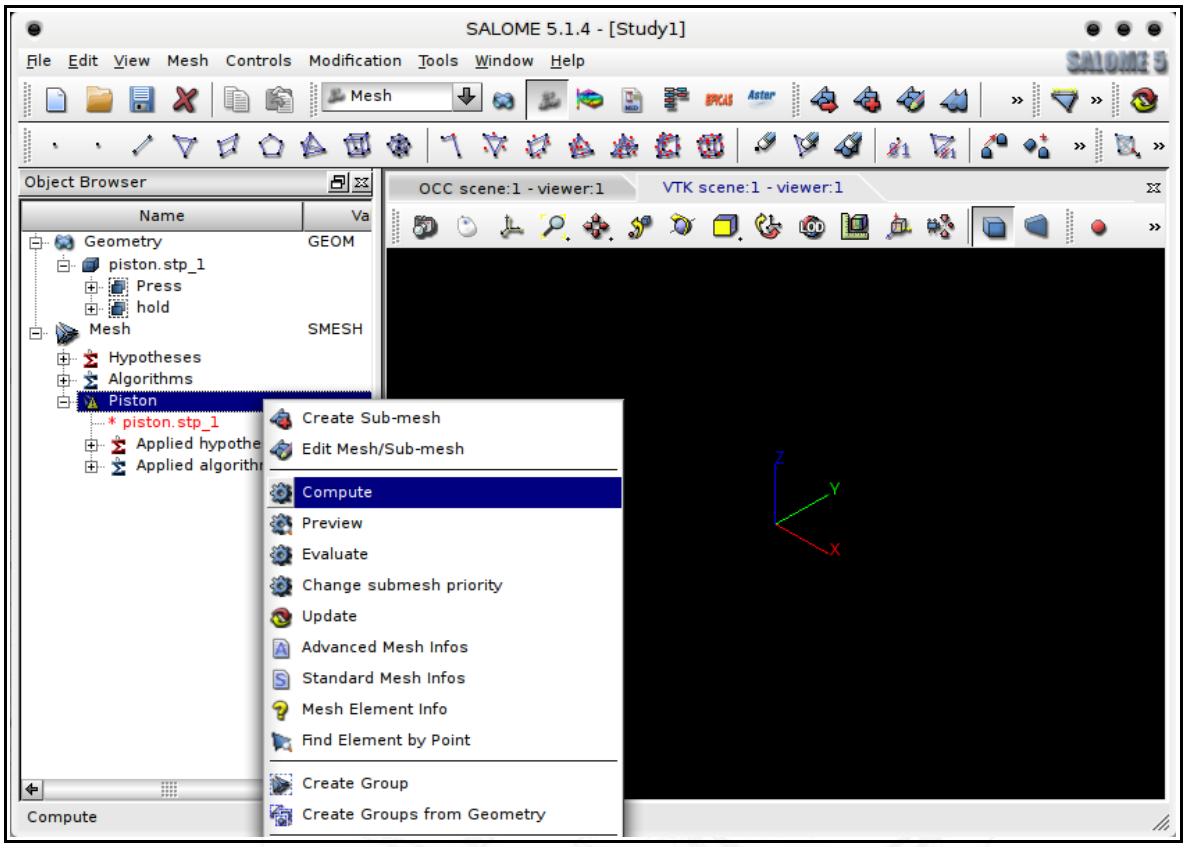

SALOME 5.1.4 - [Study1]
File Edit View New Entity Operations Repair Measures Tools Window Help
Geometry
Object Browser
Name Value
Geometry GEOM
piston.stp_1
Press
Delete Del
Color
Hide
Show Only
Refresh F5
Expand All
Find Ctrl+F
OCC scene:1 - viewer:1
Show only

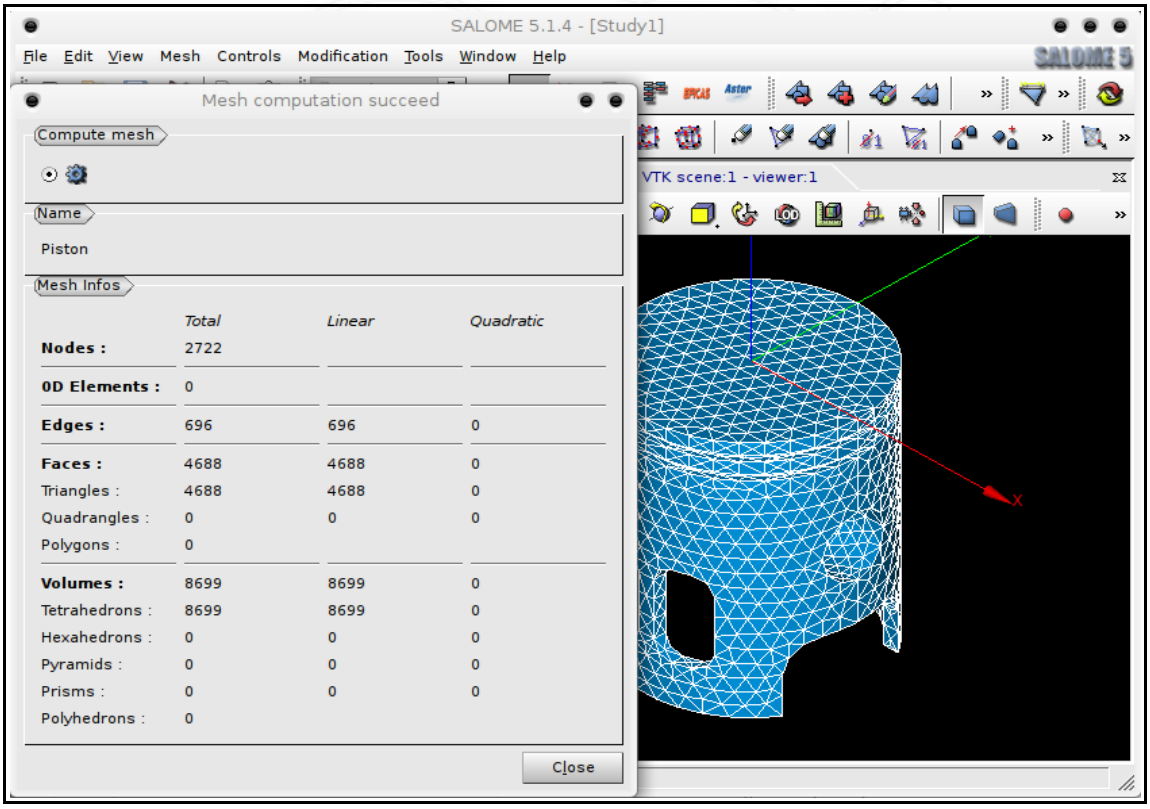
Meshing the geometry

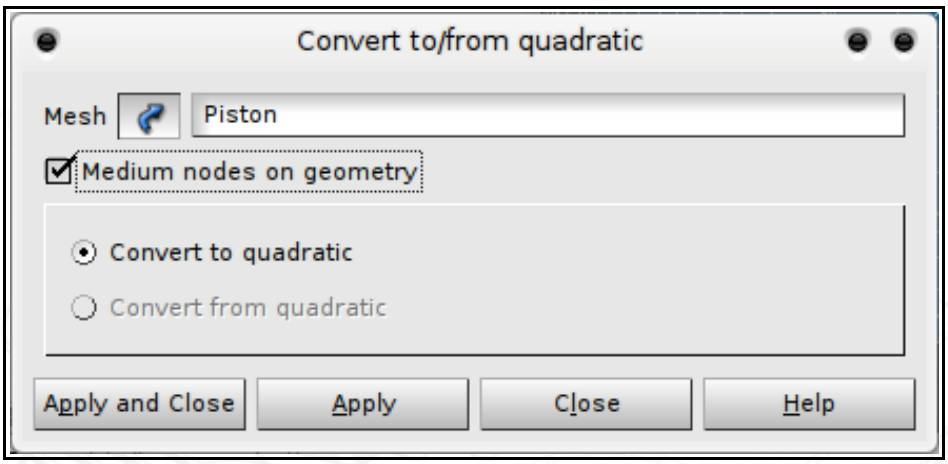
- Switch to the **MESH** module
- Click **Create Mesh**
    - Change the name from **Mesh_1** to **Piston**
    - Under **Geometry**, make sure **piston.stp_1** is selected, otherwise select **piston.stp_1** in the object browser and click the blue arrow.
- Click **Assign a set of hypotheses –** select **Automatic Tetrahedralization**
    - Click **Cancel** in the dialog **Hypothesis Construction – Max Length**
    - Click the **1D** tab, under Hypothesis**,** click the gear icon and select **Average Length**
    - Under **Length**, change the value to **0.005** – click **OK**
    - Click **Apply and Close**
- Expand **Mesh**, right-click the **Piston** mesh and select **Compute**
- When the mesh is computed, close the dialog
- Right-click on the **Piston** mesh (notice the yellow **!**-sign is gone) and select **Convert to/from Quadratic**
    - **Convert to quadratic** will automatically be selected
    - Tick **Medium Nodes on geometry**
    - Click **Apply and Close**


SALOME 5.1.4 - [Study1]
File Edit View Mesh Controls Modification Tools Window Help
Mesh
Object Browser
Name
Geometry GEOM
piston.stp_1
Press
hold
Create mesh
Name Piston
Geometry piston.stp_1
3D 2D 1D 0D
Algorithm <None>
Hypothesis <None>
Add. Hypothesis <None>
Assign a set of hypotheses
Automatic Hexahedralization
Automatic Tetrahedralization
Close
Help


SALOME 5.1.4 - [Study1]
File Edit View Mesh Controls Modification Tools Window Help
Mesh
Object Browser
Name
Geometry GEOM
piston.stp_1
Press
hold
Mesh SMESH
Create mesh
Name Piston
Geometry piston.stp_1
3D 2D 1D 0D
Algorithm Wire discretisation
Hypothesis <None>
Add. Hypothesis <None>
Assign a set of hypotheses
Apply and Close
Apply
Close
Arithmetic 1D
Automatic length
Deflection 1D
Fixed points 1D
Average length
Max Size
Nb. Segments
Source Edge
Start and End Length

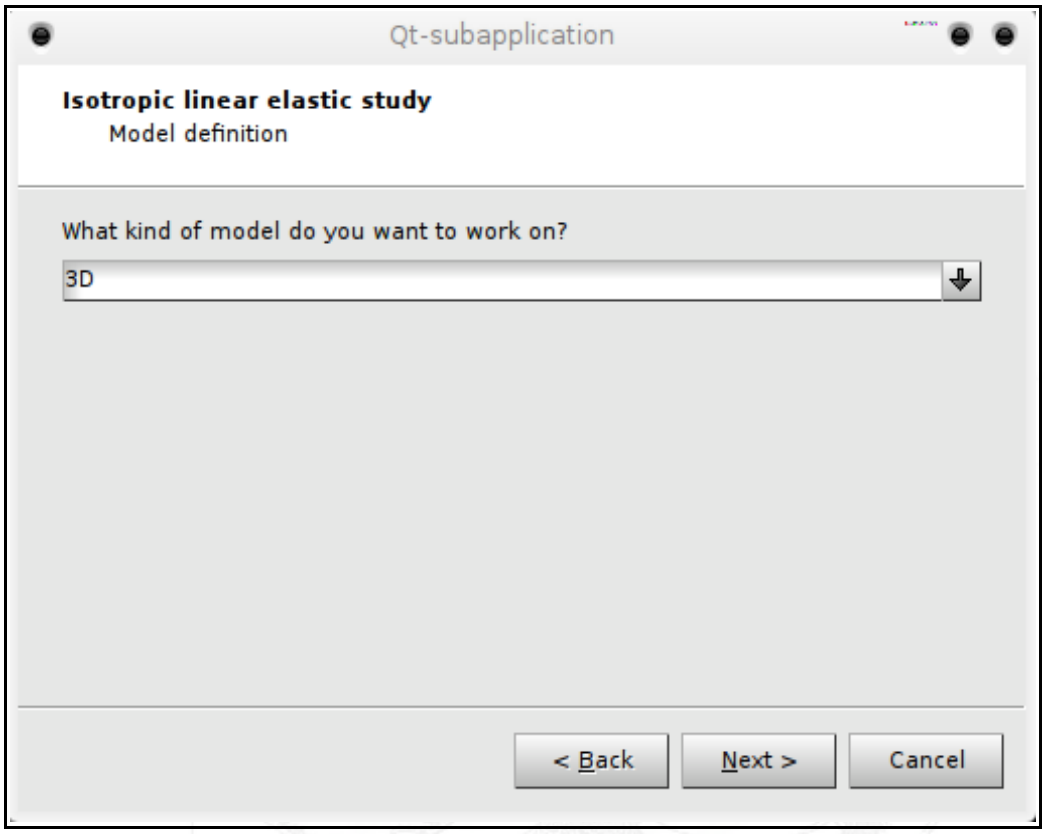

SALOME 5.1.4 - [Study1]
File Edit View Mesh Controls Modification Tools Window Help
Object Browser
Name
Geometry
piston.stp_1
Press
hold
Mesh
Hypotheses
Algorithms
Piston
piston.stp_1
Create Sub-mesh
Edit Mesh/Sub-mesh
Compute
Preview
Evaluate
Change submesh priority
Update
Advanced Mesh Infos
Standard Mesh Infos
Mesh Element Info
Find Element by Point
Create Group
Create Groups from Geometry
OCC scene:1 - viewer:1
VTK scene:1 - viewer:1
Compute

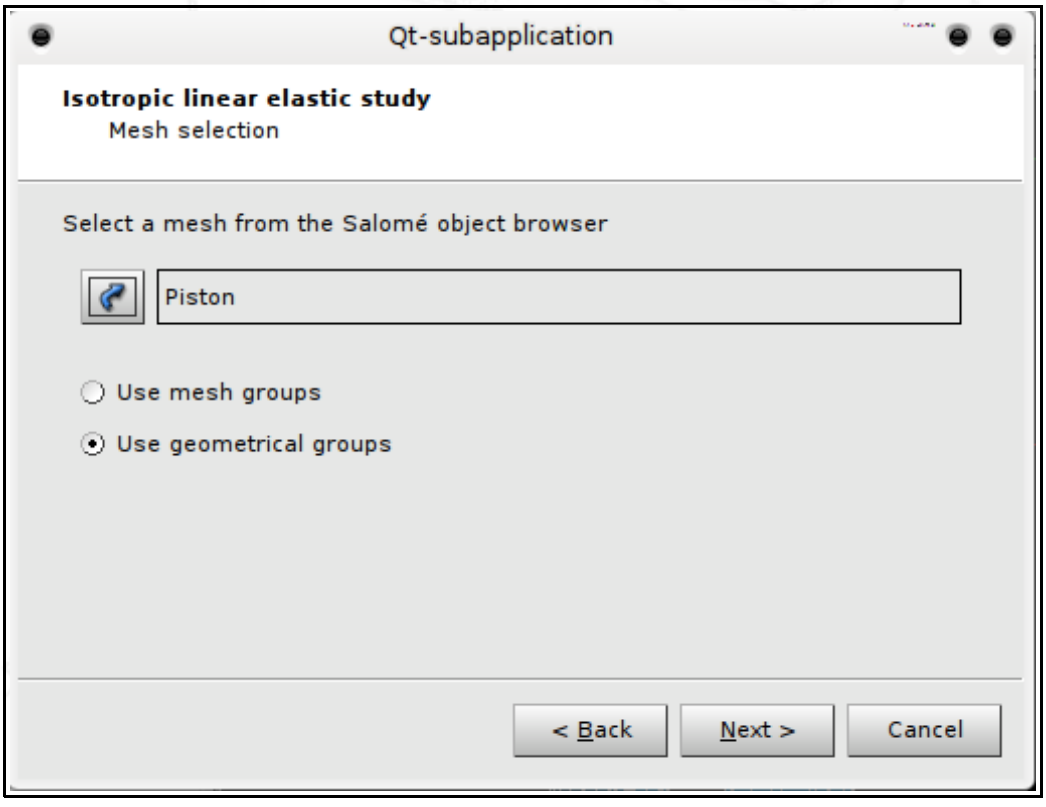

SALOME 5.1.4 - [Study1]
File Edit View Mesh Controls Modification Tools Window Help
Mesh computation succeed
Compute mesh
Name
Piston
Mesh Infos
Total Linear Quadratic
Nodes : 2722
0D Elements : 0
Edges : 696 696 0
Faces : 4688 4688 0
Triangles : 4688 4688 0
Quadrangles : 0 0 0
Polygons : 0
Volumes : 8699 8699 0
Tetrahedrons : 8699 8699 0
Hexahedrons : 0 0 0
Pyramids : 0 0 0
Prisms : 0 0 0
Polyhedrons : 0
Close
VTK scene:1 - viewer:1

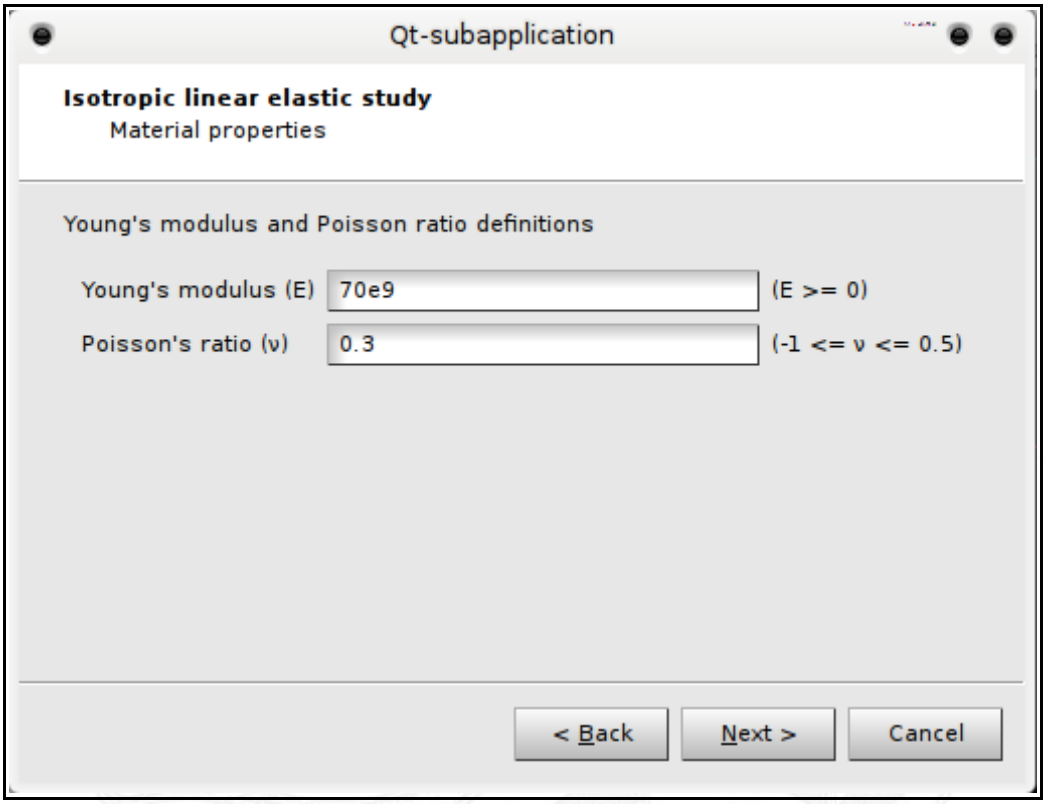

Convert to/from quadratic
Mesh
Piston
Medium nodes on geometry
Convert to quadratic
Convert from quadratic
Apply and Close
Apply
Close
Help

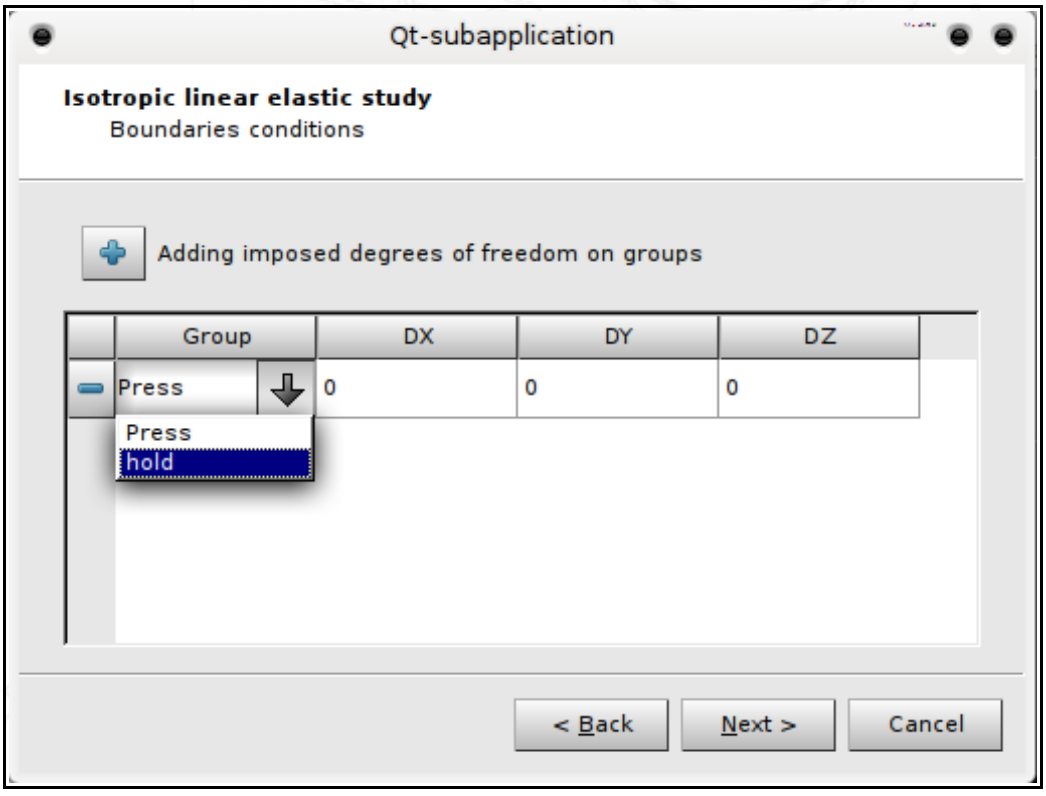
Aster module – Creating a study using a wizard

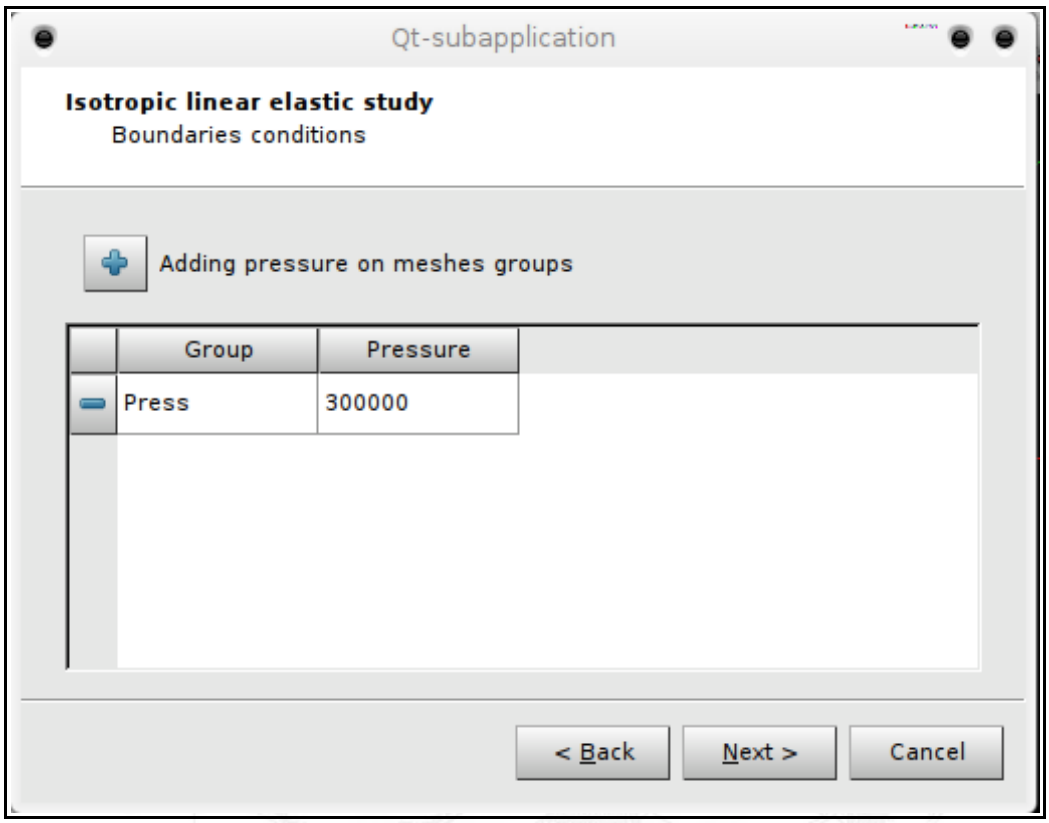
- Switch to the **Aster module**
- Click the **Linear Statics Wizard** icon
- In the following dialog **3D** is automatically selected – click **Next**
- In the next dialog select the **Piston mesh** in the object browser under **Mesh** and click the blue arrow
    - **Use Geometrical Groups** should be selected – this will import and create mesh groups created in the **Geometry** module
    - Click **Next**
- In the next dialog enter a **Young's Modulus (E)** value of **70e9** (or 70000000000) and a **Poisson's ration** of **0.3**
    - MKS SI-system is used here and thus all dimensions are in meters, and pressure will be in Pascals – 70e9Pa corresponds to a E-module of 70GPa
    - Click **Next**
- In the next dialog **Boundary Conditions** are imposed on the model
    - Double-click the the first row in the **Group** column, press the down-arrow and select the '**hold**' group
    - Leave values for **DX,DY** and **DZ** at **0** and click **Next**
- In the next dialog a group for applying pressure must be selected
    - Again, double-click the first row in the first column, pull down the menu and select the **Press** group
    - Double-click the first row in the second column and enter a pressure value of **300e3** – or 300000
    - Click **Next**
- In the next dialog a name and a destination for the created command file must be selected
    - Click the file dialog button
    - Navigate to the folder that should contain the study files and type in a name for the command file – click **Finish**

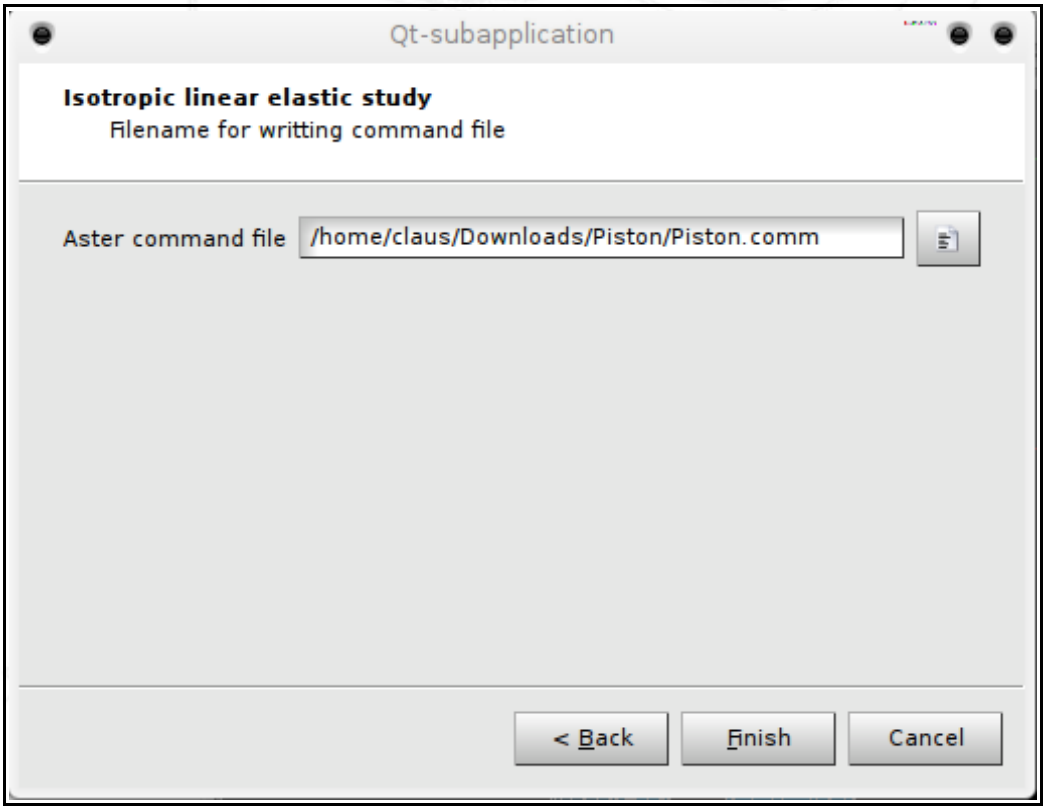

Qt-subapplication
Isotropic linear elastic study
Model definition
What kind of model do you want to work on?
3D
< Back
Next >
Cancel


Qt-subapplication
Isotropic linear elastic study
Mesh selection
Select a mesh from the Salomé object browser
Piston
Use mesh groups
Use geometrical groups
< Back
Next >
Cancel

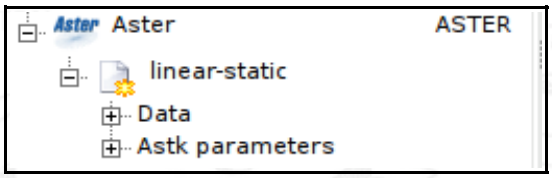

Qt-subapplication
Isotropic linear elastic study
Material properties
Young's modulus and Poisson ratio definitions
Young's modulus (E)
70e9
(E >= 0)
Poisson's ratio (ν)
0.3
(-1 <= ν <= 0.5)
< Back
Next >
Cancel


Qt-subapplication
Isotropic linear elastic study
Boundaries conditions
Adding imposed degrees of freedom on groups
Group
DX
DY
DZ
Press
0
0
0
Press
hold
< Back
Next >
Cancel

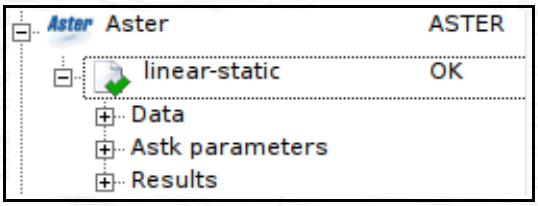

Qt-subapplication
Isotropic linear elastic study
Boundaries conditions
Adding pressure on meshes groups
Group
Pressure
Press
300000
< Back
Next >
Cancel


Qt-subapplication
Isotropic linear elastic study
Filename for writting command file
Aster command file
/home/claus/Downloads/Piston/Piston.comm
< Back
Finish
Cancel

- Expanding the **Aster** tree in the object browser a new study should now be present:
  - .

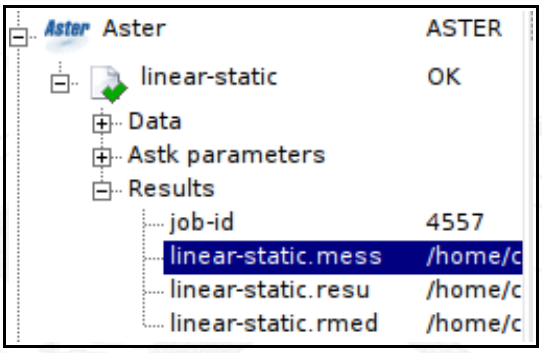


  - Right-click **linear-static** and select **Run**
  - A terminal window should now appear and a lot of text will scroll by
  - After the calculation has finished, the terminal will disappear and you will be presented with a green tick on the linear-static study:



  - Take a moment to thoroughly enjoy this seemingly innocently looking icon, because you WILL learn to love it once you start playing around and your studies fail :)

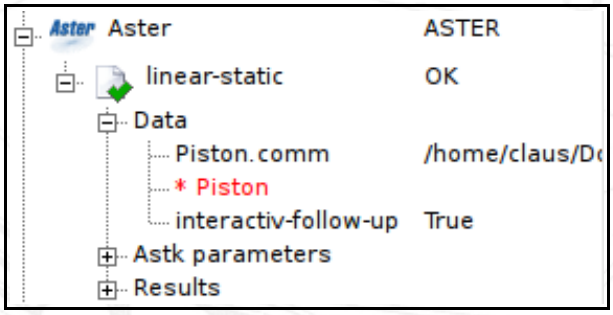
- Expand the Results tree in the Object Browser, there will be 4 entries:



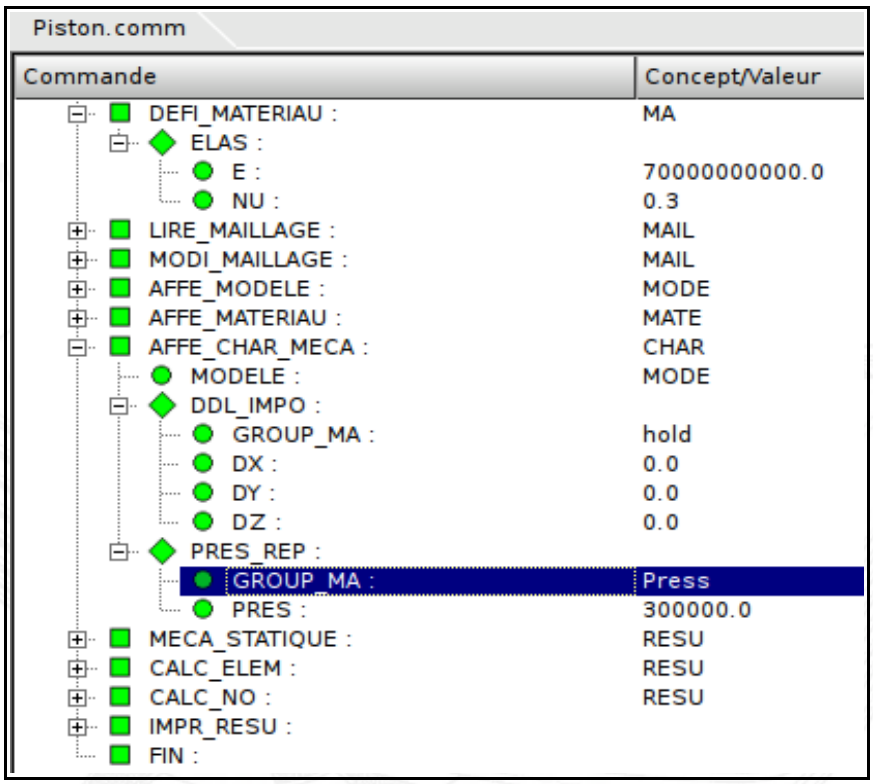
-
- Right-clicking each entry will give you options to **Read as Text** or **Edit as Text** – or other
- **job-id**: Self-explanatory
- **linear-static.mess**: This file contains messages produced by the calculation and should be the first place to search for errors.
- **linear-static.resu**: This file contains among other things, short description of errors encountered and a detailed summation of run-time performance, i.e. how long time it took for each command to run and overall time consumed, in this case:
  TOTAL_JOB: ELAPSED: 9.42 seconds
- **linear-static.rmed**: This file containing the results to be used in the Post-processing module

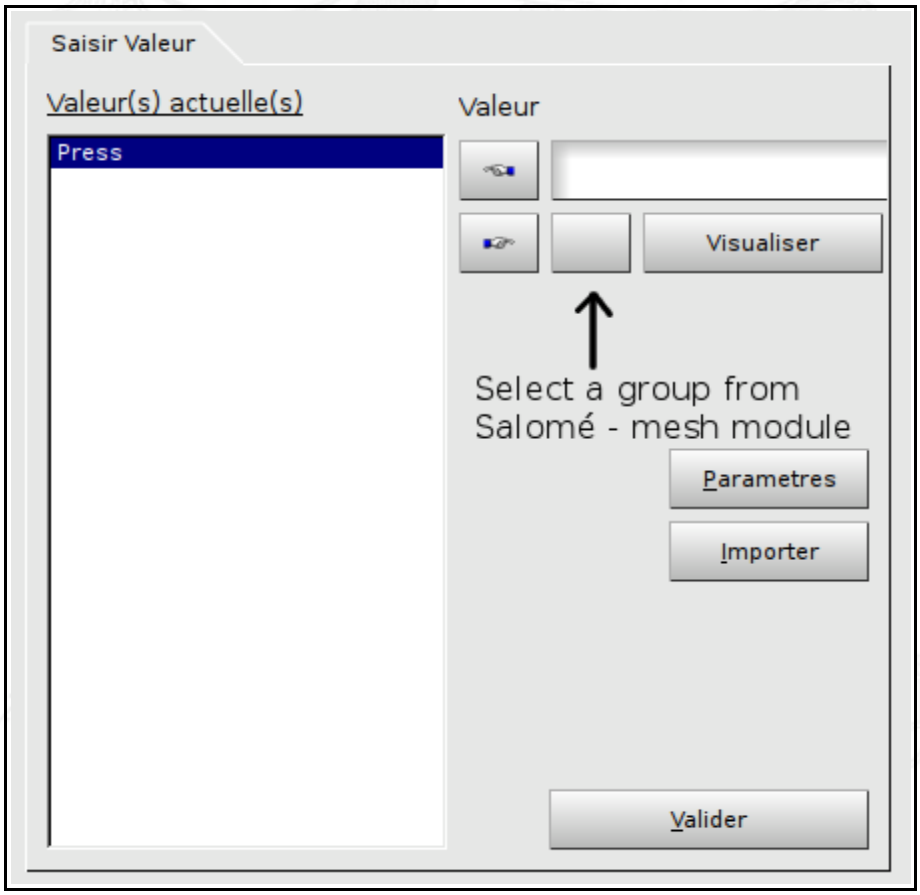
Aster module – Editing a command file

- Should a mistake have been made during the wizard, such as selecting a wrong group, entering a wrong value for pressure etc., the command file can easily be edited from within the Aster module:
  - Expand the **linear-static** study tree and expand the **Data** branch like so:
  - .



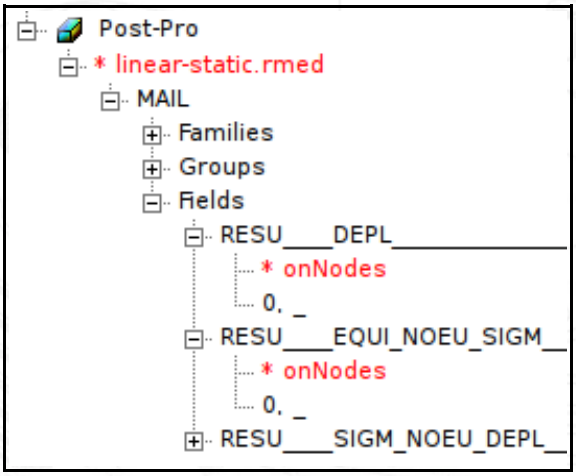
  - When right-clicking the **Piston.comm** command file, 3 choices are presented:
  - **Read as Text** : Will display the command file in a window in read-only mode
  - **Edit as Text** : Will open the command file in an external editor (of the users choice)
  - **Run Eficas**: Will open the command file in Eficas, an editor specially designed for creating and editing Code_Aster command files. This editor will take care of the very strict syntax Code_Aster requires, and is highly recommended for beginners for anything other than changing simple values in a command file.
- Right-click the **Piston.comm** file in the object browser and select **Run Eficas**
- In this introduction only 2 entries will be mentioned:
  - **DEFI_MATERIAU** – Define material:
    - **ELAS** – Elastic material
    - The values entered earlier are present here: **E-module**, **NU** (Poissons ratio)
  - **AFFE_CHAR_MECA** : Assign boundary conditions and loads:
    - **DDL_IMPO**: The face group '**hold**' is restricted in all directions, i.e. **DX,DY,DZ = 0** (DDL: Degrés De Liberties = degrees of freedom, IMPOsed)
    - **PRESS_REP** : A pressure of 300 x 10^3 Pascals is imposed on the **'Press'** face group
- To change a group under e.g. **PRES_REP**, simply click **GROUP_MA**, use the right hand arrow to move the name into the Value (Valeur) field and clear it. Either type the name of the group, *or* select the face group in the **Piston mesh** in the **MESH** module and press the button I've marked in the following picture – curiously the icon image is gone in my version. Use the left hand arrow to enter the new name, confirm with '**Valider**' – a new value/name should now be present under **PRESS_REP → GROUP_MA**


Piston.comm
Commande
Concept/Valeur
DEFI_MATERIAU : MA
ELAS :
E : 70000000000.0
NU : 0.3
LIRE_MAILLAGE : MAIL
MODI_MAILLAGE : MAIL
AFFE_MODELE : MODE
AFFE_MATERIAU : MATE
AFFE_CHAR_MECA : CHAR
MODELE : MODE
DDL_IMPO :
GROUP_MA : hold
DX : 0.0
DY : 0.0
DZ : 0.0
PRES_REP :
GROUP_MA : Press
PRES : 300000.0
MECA_STATIQUE : RESU
CALC_ELEM : RESU
CALC_NO : RESU
IMPR_RESU :
FIN :

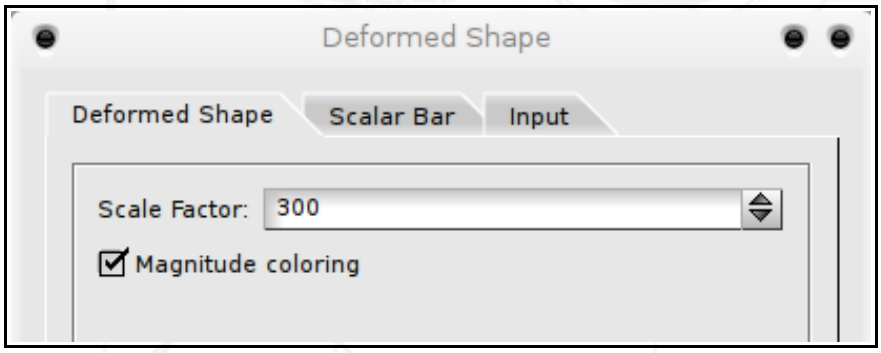

Saisir Valeur
Valeur(s) actuelle(s)
Press
Valeur
Visualiser
Select a group from
Salomé - mesh module
Parametres
Importer
Valider

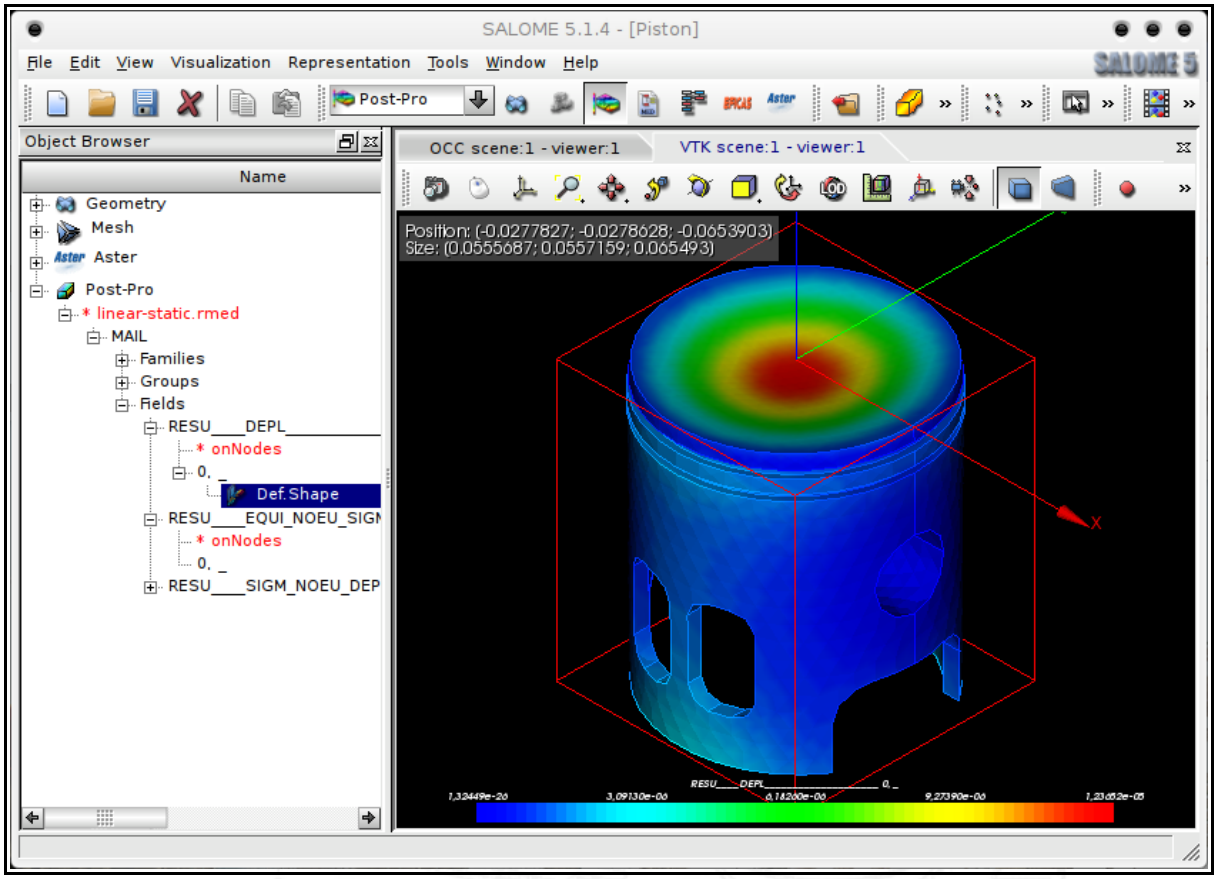
Post processing

- Click the **Post-Pro** icon to switch to the **Post-Pro module**
- Expand the Post-Pro tree in the object browser until it looks like this:
- .

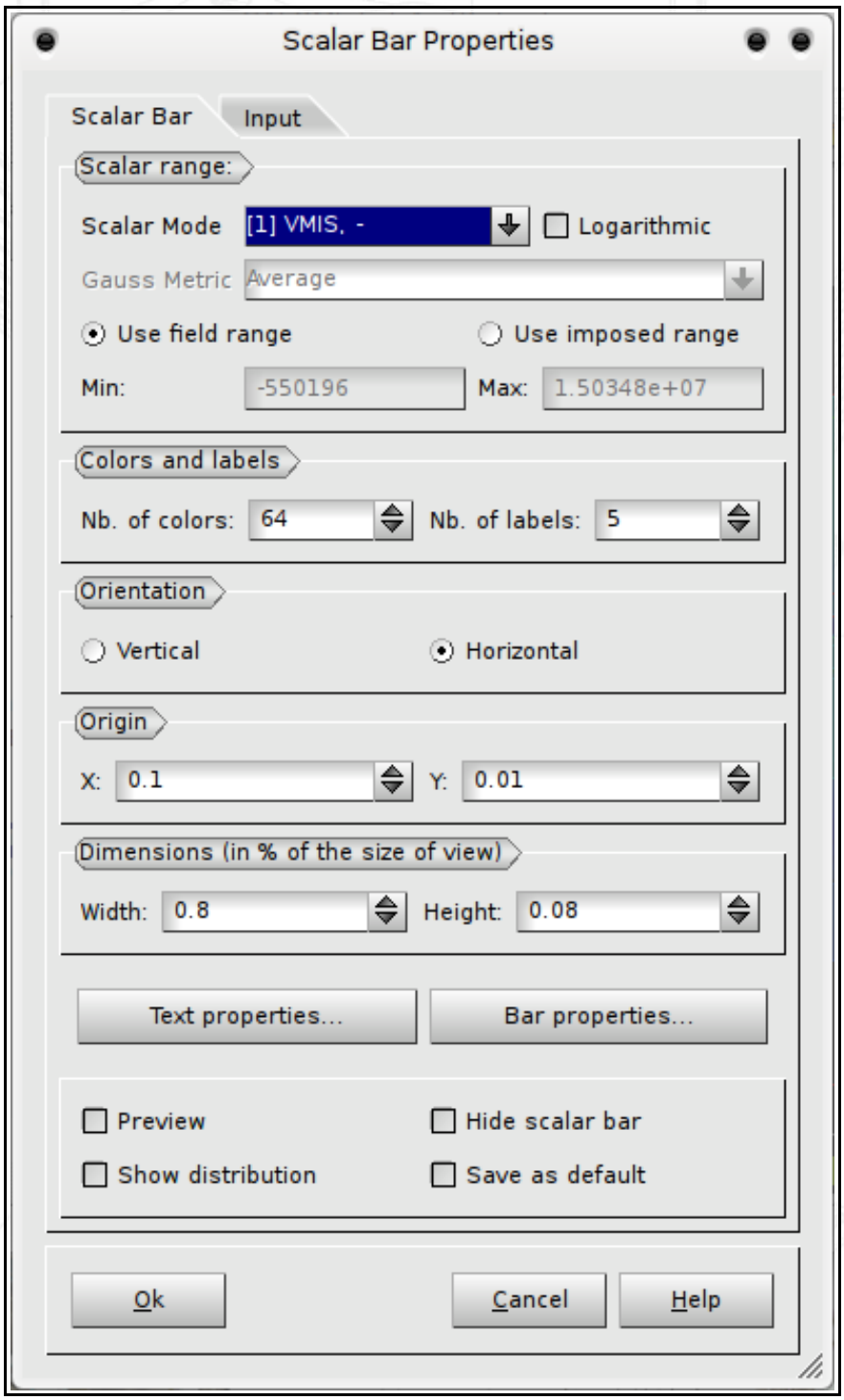


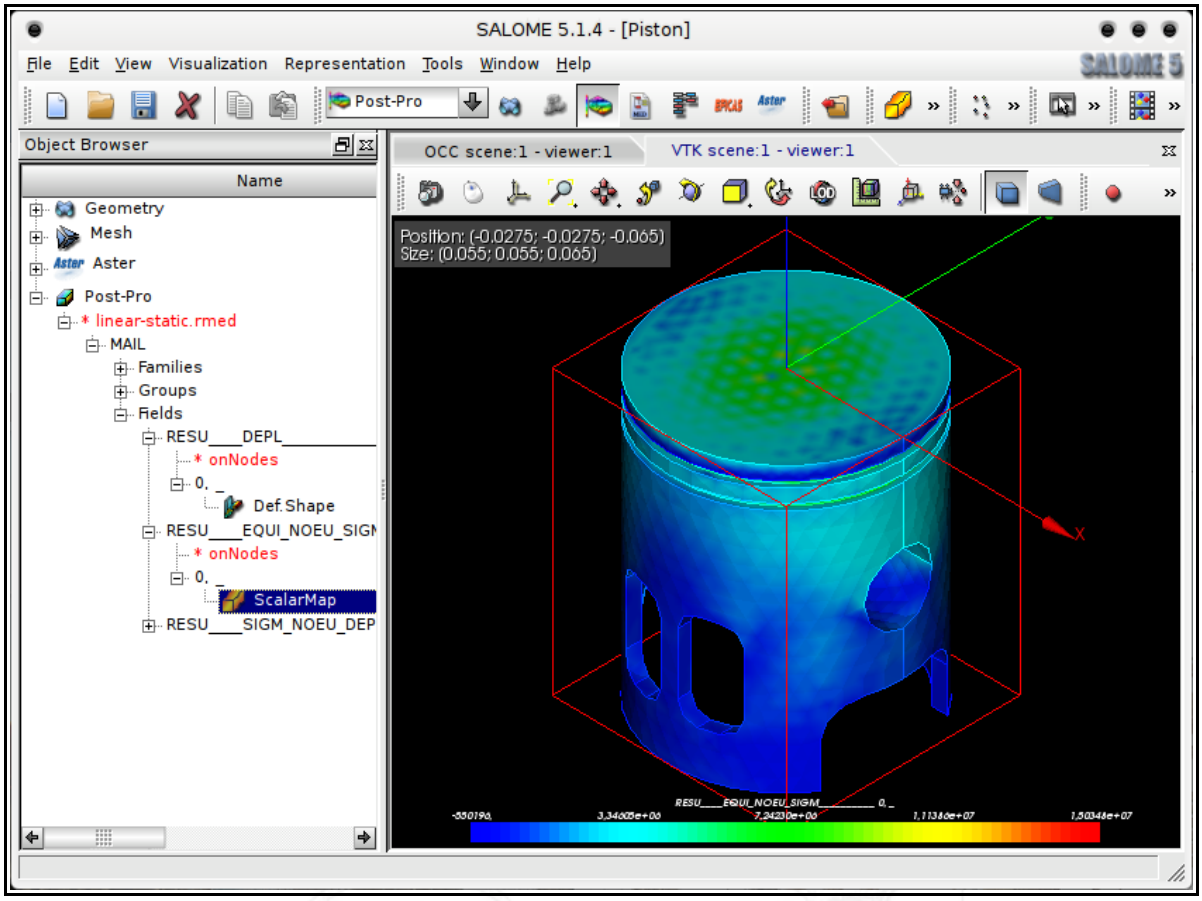
- Displacement / Deformation:
  - Under **RESU_DEPL_**, right-click **0,_** and select **Deformed Shape**
  - Enter a **Scale factor** of **300,** tick **Magnitude Coloring** and press **Apply and Close**
  - Now expand **0,_** , right-click **Def.Shape** and select **Show Only**




SALOME 5.1.4 - [Piston]
File Edit View Visualization Representation Tools Window Help
Post-Pro
Object Browser
Name
Geometry
Mesh
Aster
Post-Pro
* linear-static.rmed
MAIL
Families
Groups
Fields
RESU____DEPL_____
* onNodes
0, _
Def.Shape
RESU____EQUI_NOEU_SIGM
* onNodes
0, _
RESU____SIGM_NOEU_DEP
OCC scene:1 - viewer:1
VTK scene:1 - viewer:1
Position: (-0.0277827; -0.0278628; -0.0653903)
Size: (0.0555687; 0.0557159; 0.065493)

- Von Mises stresses:
  - Under **RESU__EQUI_NOUE_SIGM**, right-click **0,_** and select **Scalar Map**
  - In the pull-down menu **Scalar Mode**, select **VMIS.-** and click **OK**
  - Again, expand **0,_** , right-click **ScalarMap** and select **Show Only**

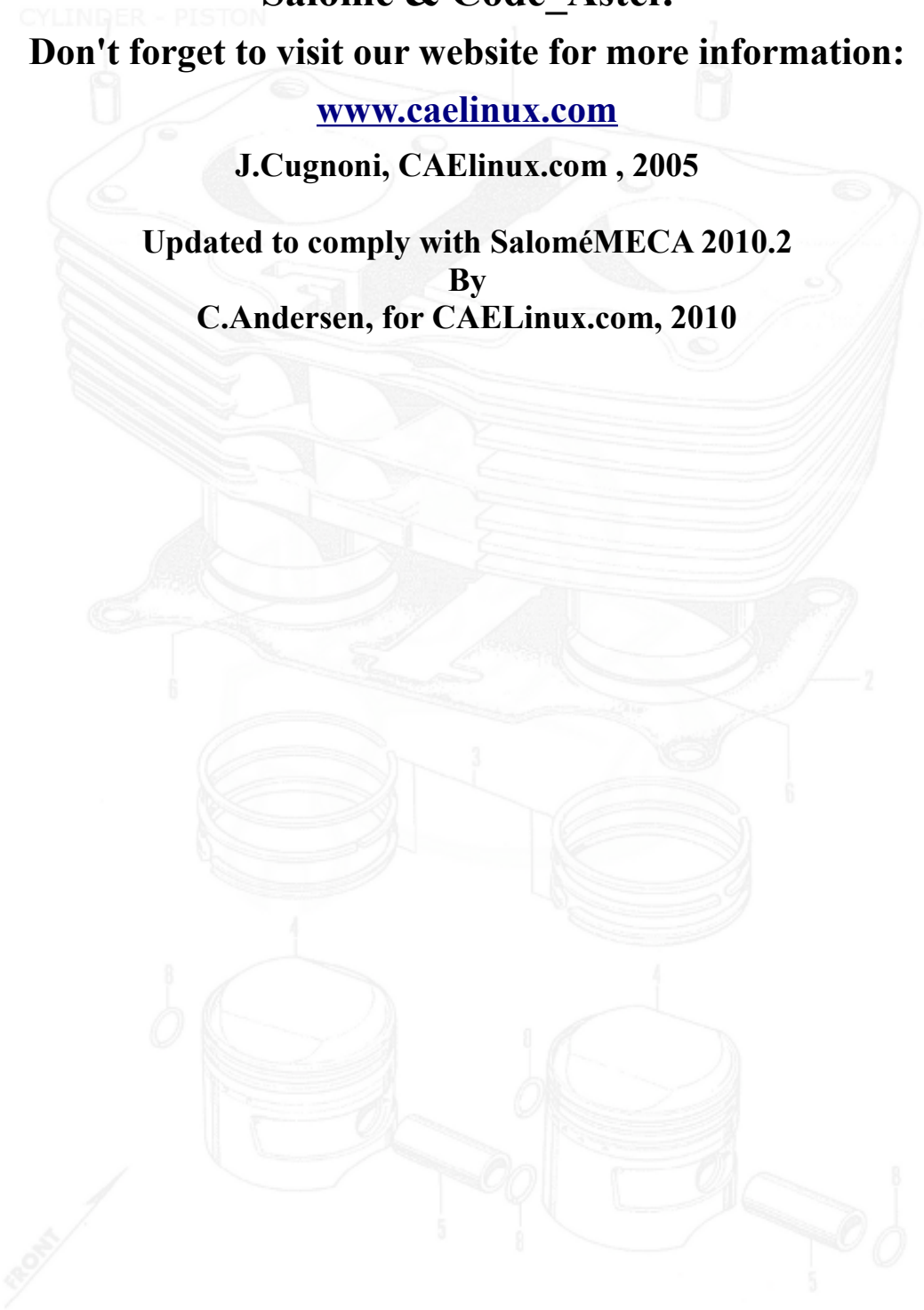



SALOME 5.1.4 - [Piston]
File Edit View Visualization Representation Tools Window Help
Post-Pro
Object Browser
Name
Geometry
Mesh
Aster
Post-Pro
* linear-static.rmed
MAIL
Families
Groups
Fields
RESU____DEPL
* onNodes
0, _
Def.Shape
RESU____EQUI_NOEU_SIGM
* onNodes
0, _
ScalarMap
RESU____SIGM_NOEU_DEP
OCC scene:1 - viewer:1
VTK scene:1 - viewer:1
Position: (-0.0275; -0.0275; -0.065)
Size: (0.055; 0.055; 0.065)

Congratulation , you have finished this first introduction to Salomé & Code_Aster.

Don't forget to visit our website for more information:

www.caelinux.com

J.Cugnoni, CAElinux.com , 2005

Updated to comply with SaloméMECA 2010.2
By
C.Andersen, for CAELinux.com, 2010