CAElinux

An introduction to Salome & Code_Aster:

a simple linear statics analysis of a piston

JCugnoni, CAElinux.com, 2005

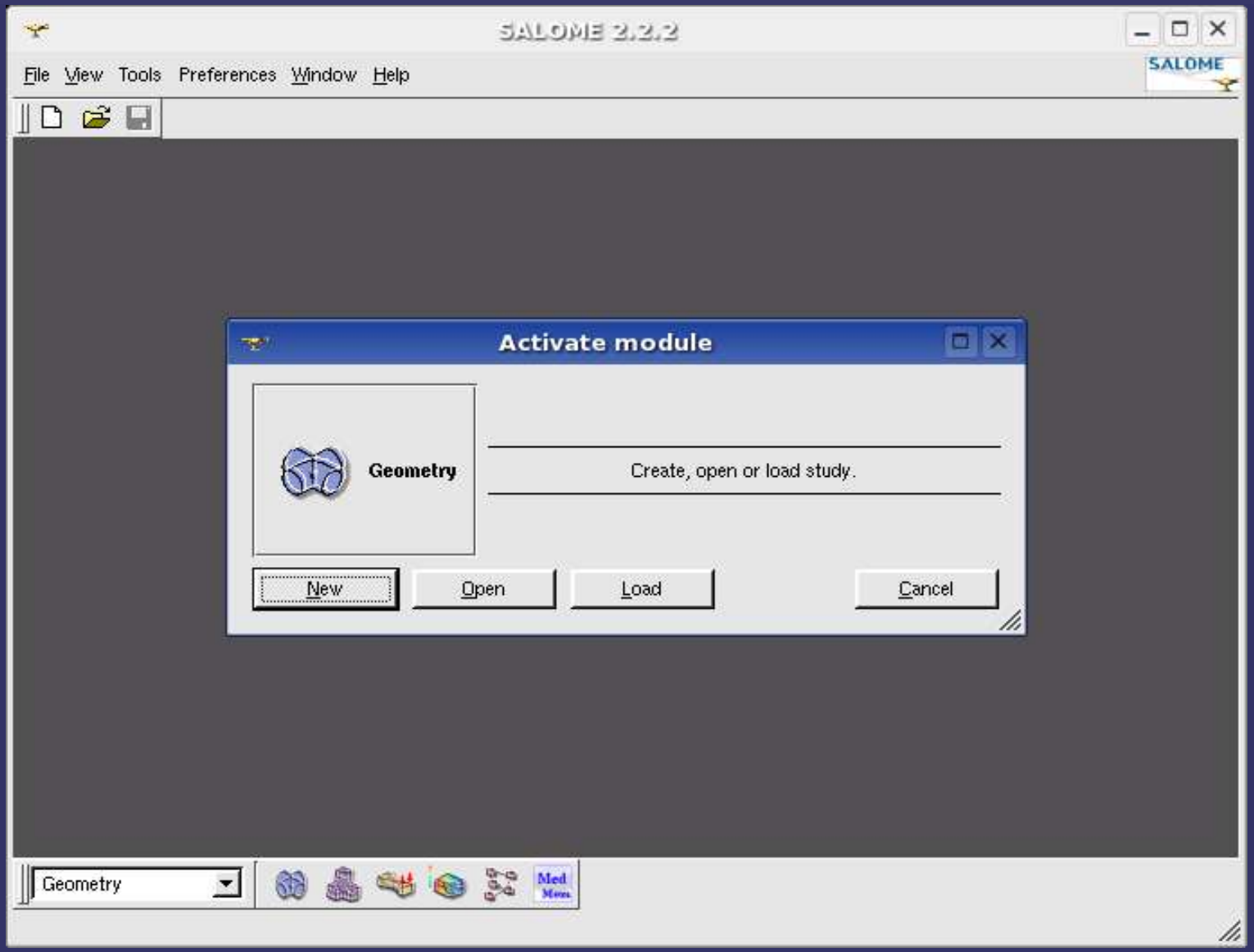
Overview

- SALOME
  - Import & prepare STEP geometry
  - Meshing & group creation
  - Exporting to MED file
- Code_Aster
  - Create a new analysis project
  - Edit command file
  - Run analysis
- SALOME
  - Import MED result file
  - Visualiye results

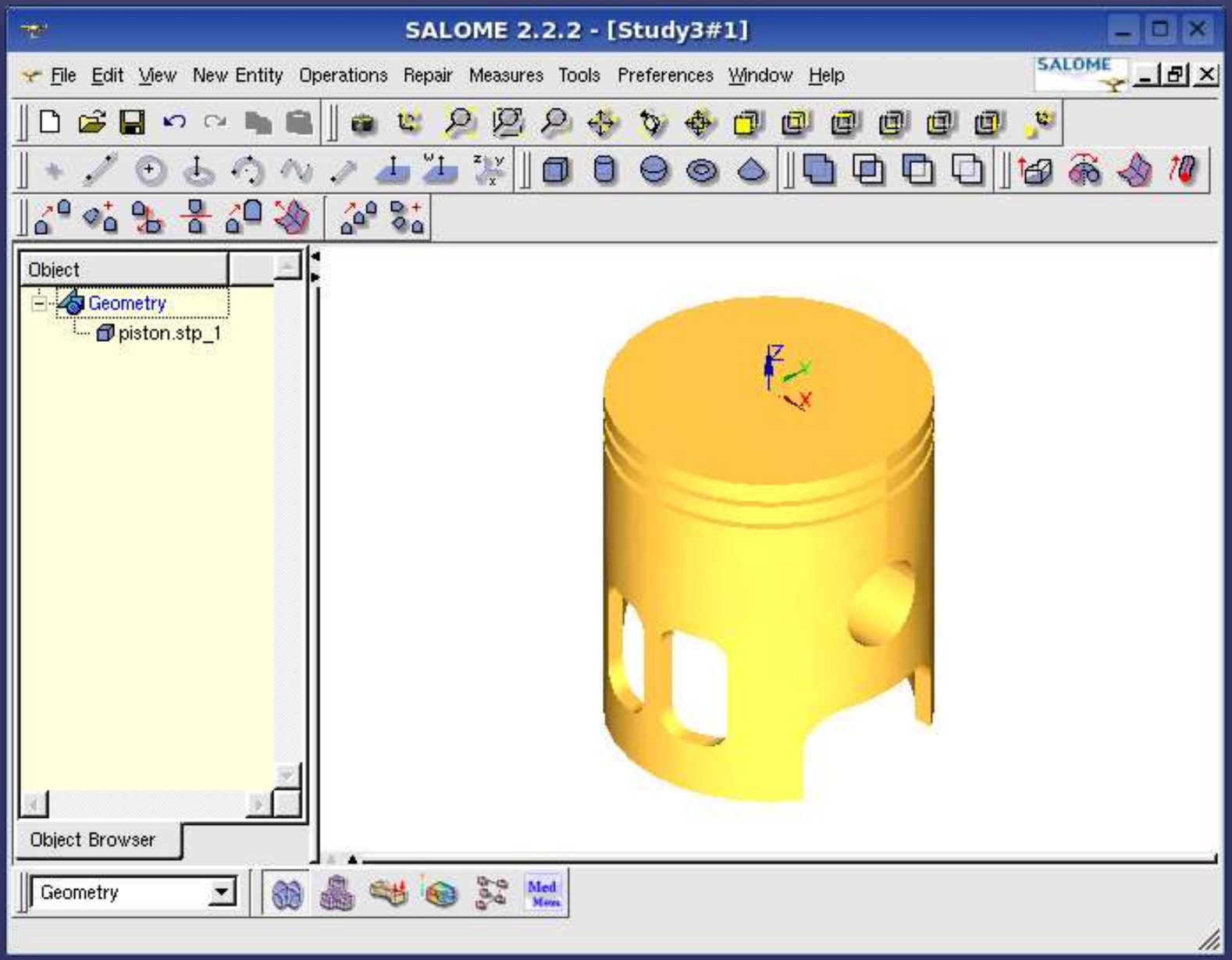
SALOME: import geometry

- Start Salome and select Geometry module
- Choose New in the dialog
- Select Menu File-> Import
  - Choose STEP file format
  - Select file “piston.stp”
- Click Menu View -> View Mode -> Shading
- Finally, click the Fit All button in the View toolbar

Import Geometry



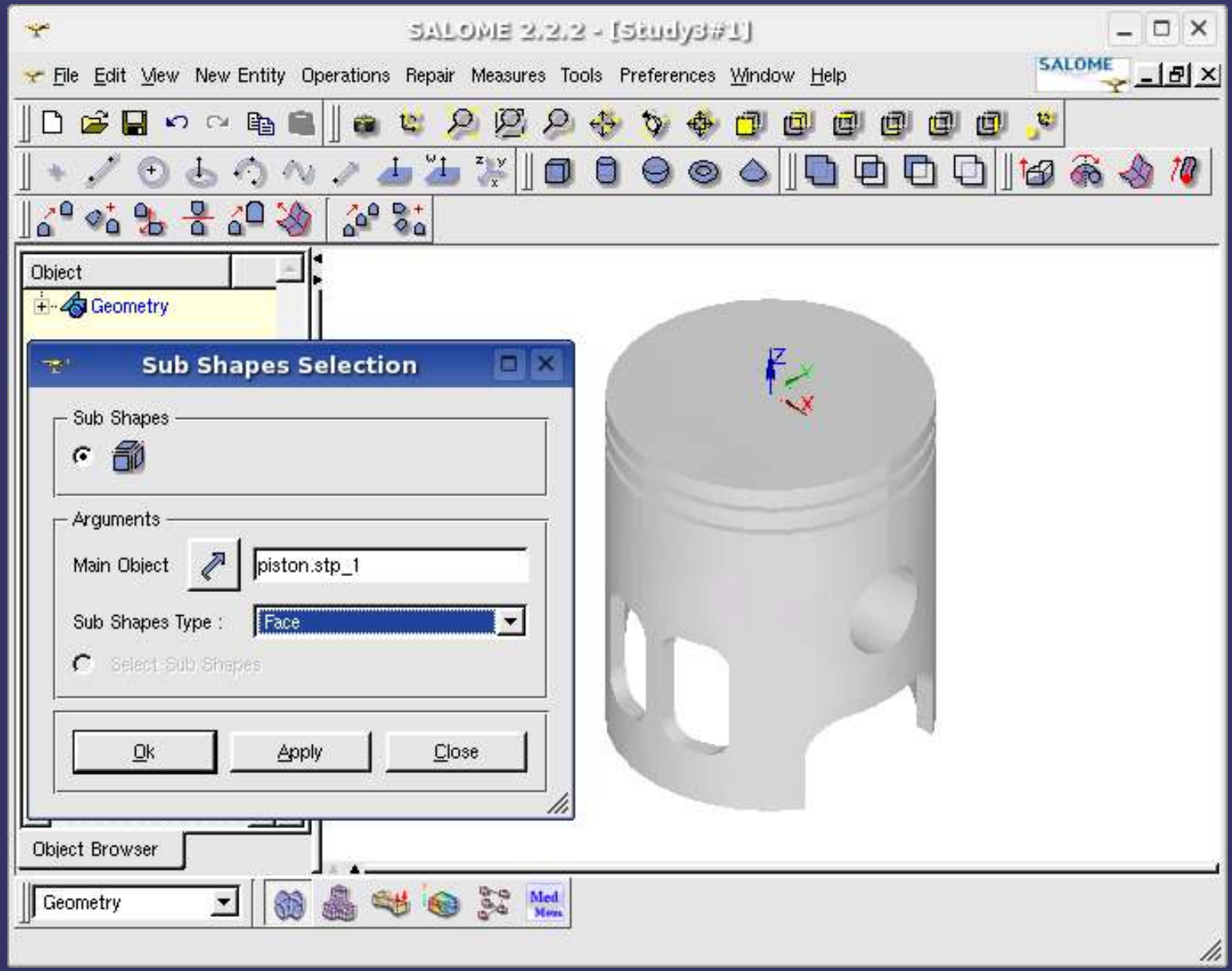
Import Geometry



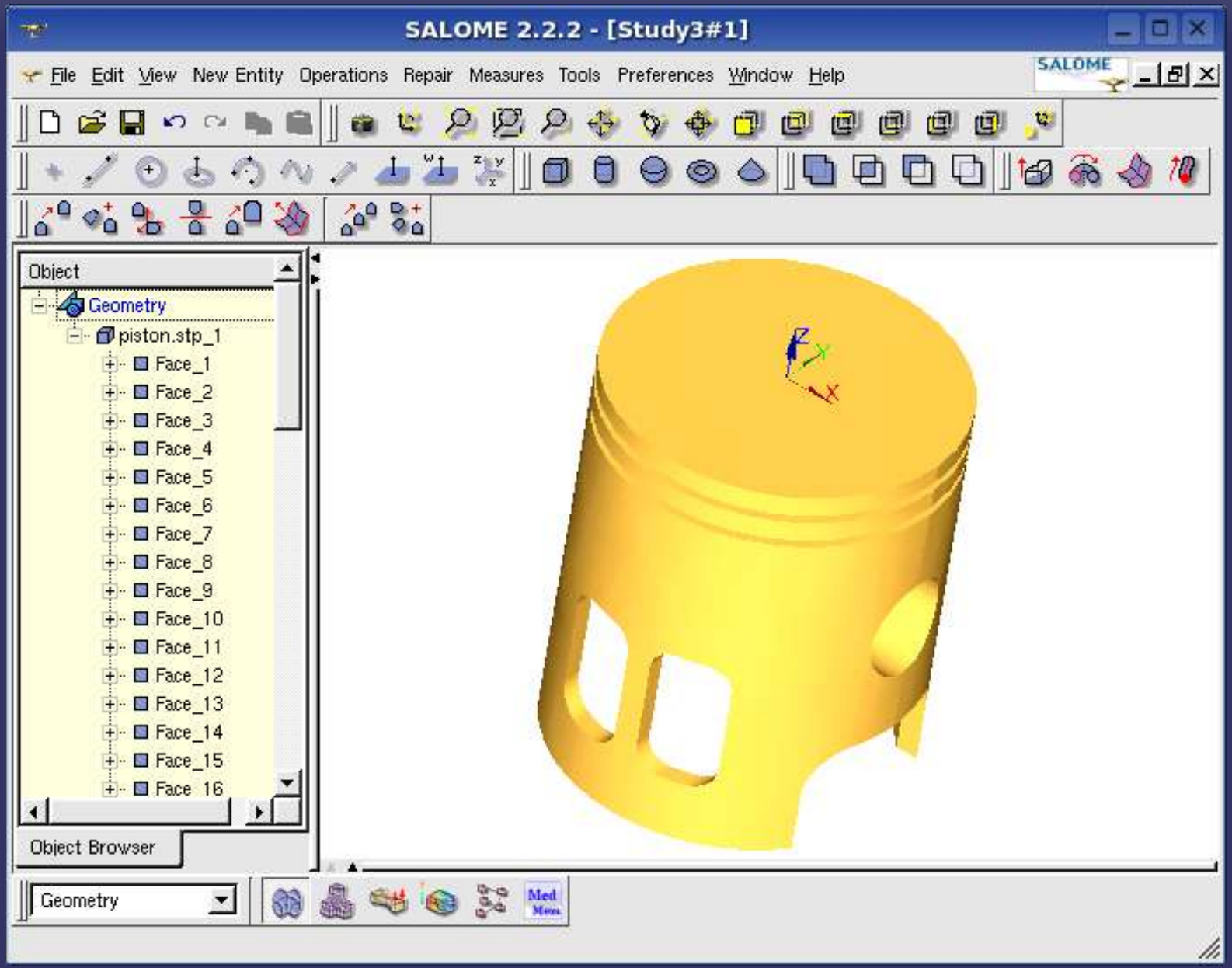
Prepare geometry

- Click Menu New Entity -> Explode
- Select the piston.stp_1 object in the 3D view or in the tree view
- Choose Sub Shape Type = Face
- Click OK & confirm your choice
- Click on + in the tree view to expand piston.stp_1 object

Prepare geometry



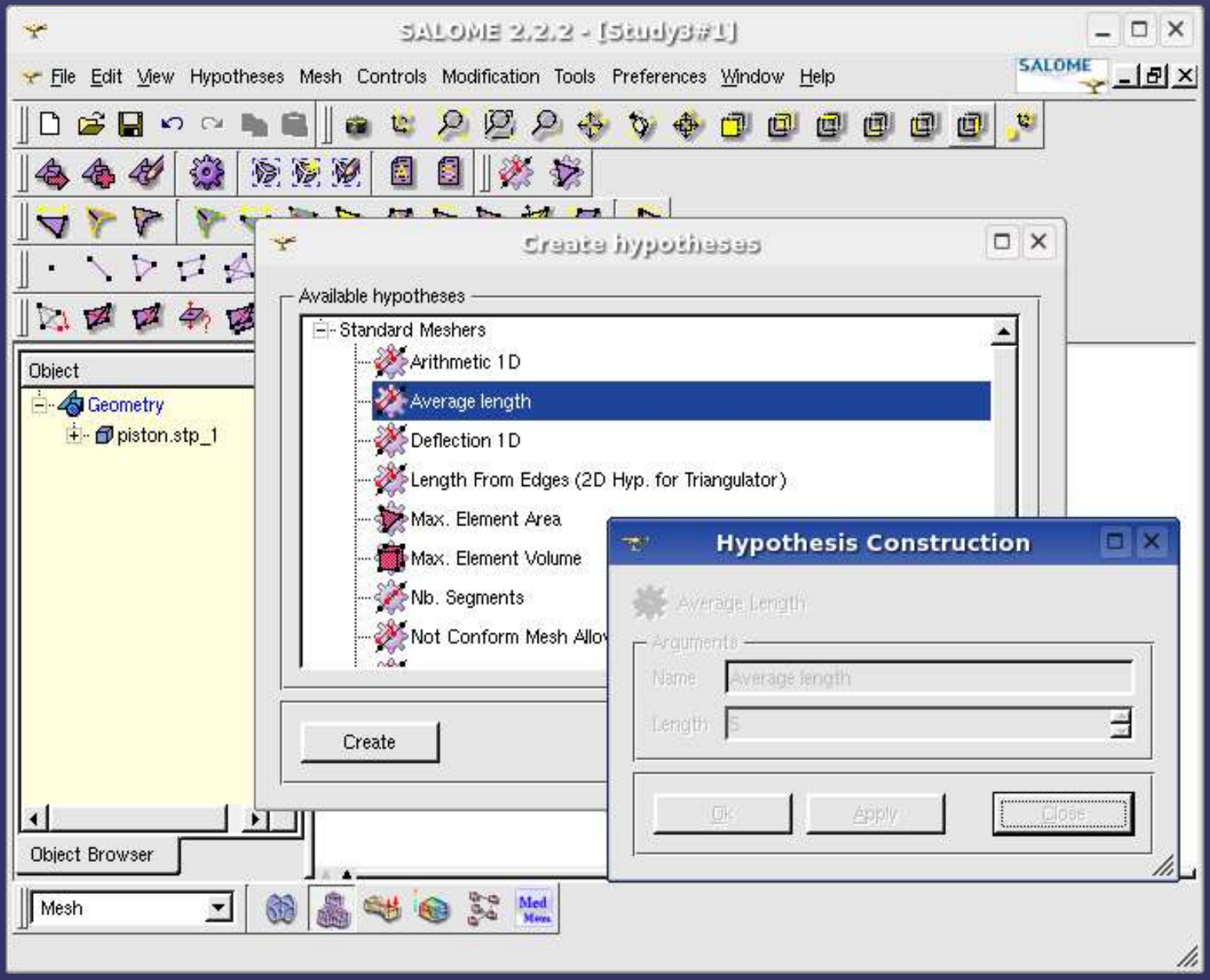
Prepare geometry



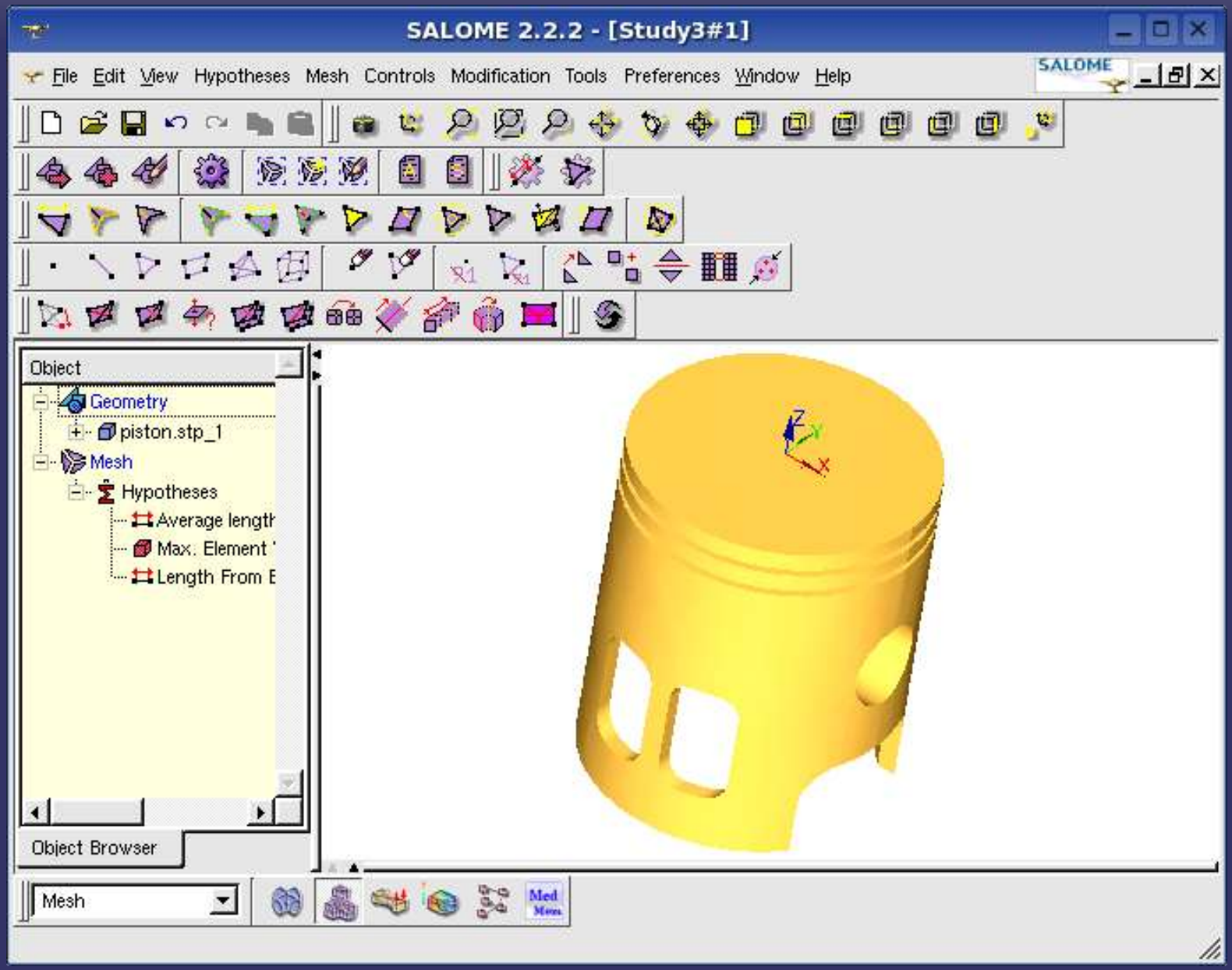
Meshing

- Change to MESH module
- Choose Menu Hypotheses -> Create hypotheses
  - Select “Average Length” & click Create
  - Enter Length = 5 & click OK
  - Select “Length from Edges” & click Create
  - Select “Max Element Volume” & Create
  - Enter Max Volume = 100 & click OK
  - Close New Hypotheses dialog

Meshing


SALOME 2.2.2 - [Study3#1]
File Edit View Hypotheses Mesh Controls Modification Tools Preferences Window Help
SALOME
Create hypotheses
Available hypotheses
Standard Meshers
Arithmetic 1D
Average length
Deflection 1D
Length From Edges (2D Hyp. for Triangulator)
Max. Element Area
Max. Element Volume
Nb. Segments
Not Conform Mesh Allo
Create
Hypothesis Construction
Average Length
Arguments
Name
Average length
Length
5
Ok
Apply
Close
Object
Geometry
piston.stp_1
Object Browser
Mesh
Med

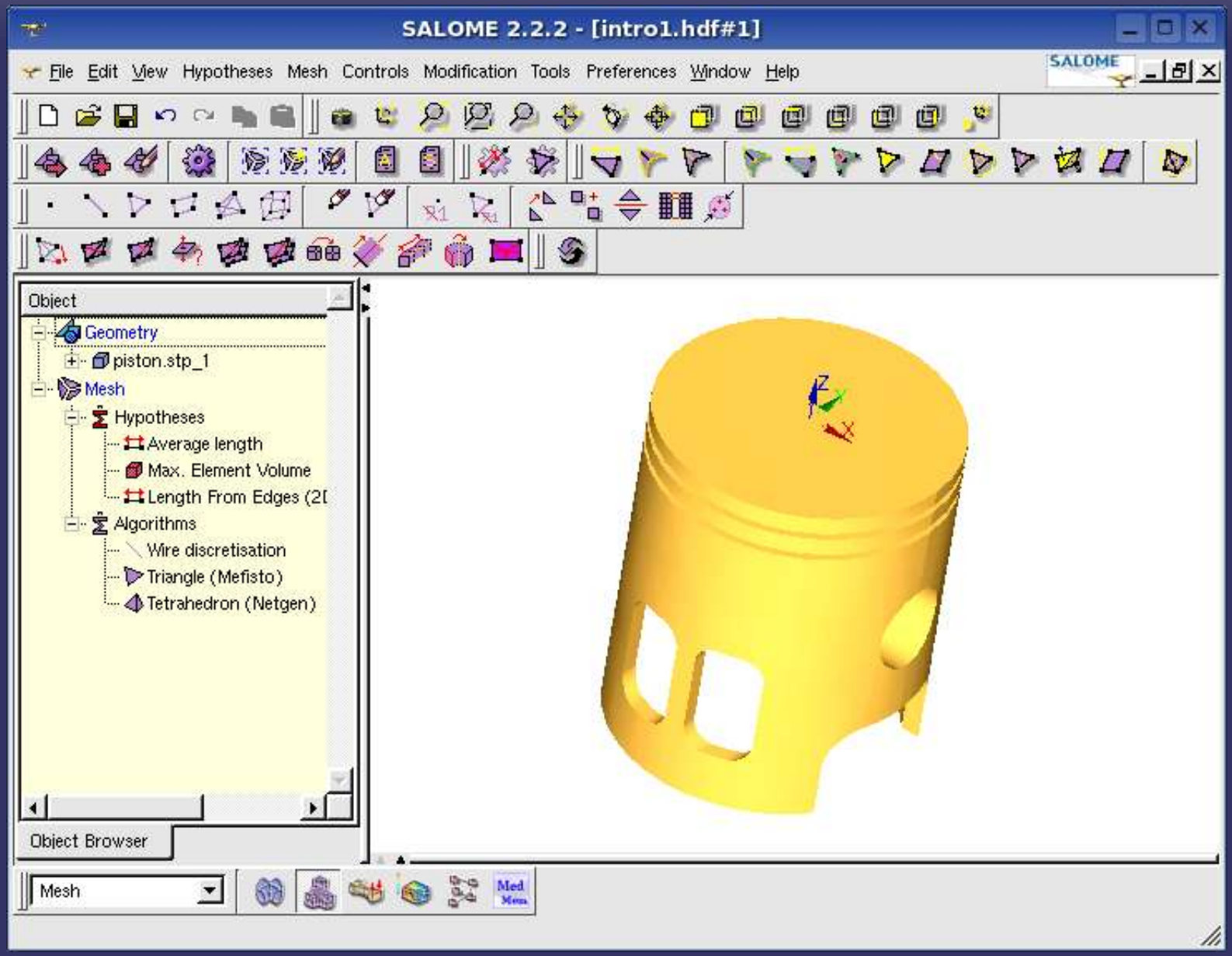
Meshing


SALOME 2.2.2 - [Study3#1]
File Edit View Hypotheses Mesh Controls Modification Tools Preferences Window Help
SALOME
Object
Geometry
piston.stp_1
Mesh
Hypotheses
Average length
Max. Element
Length From E
Object Browser
Mesh
Med

Meshing (2)

- Click Menu Hypotheses -> Create Al-gorithms
  - Select “Wire Discretisation” & click Create
  - Select “Triangle (Mefisto)” & click Create
  - Select “Tetrahedron (Netgen)” & click Create

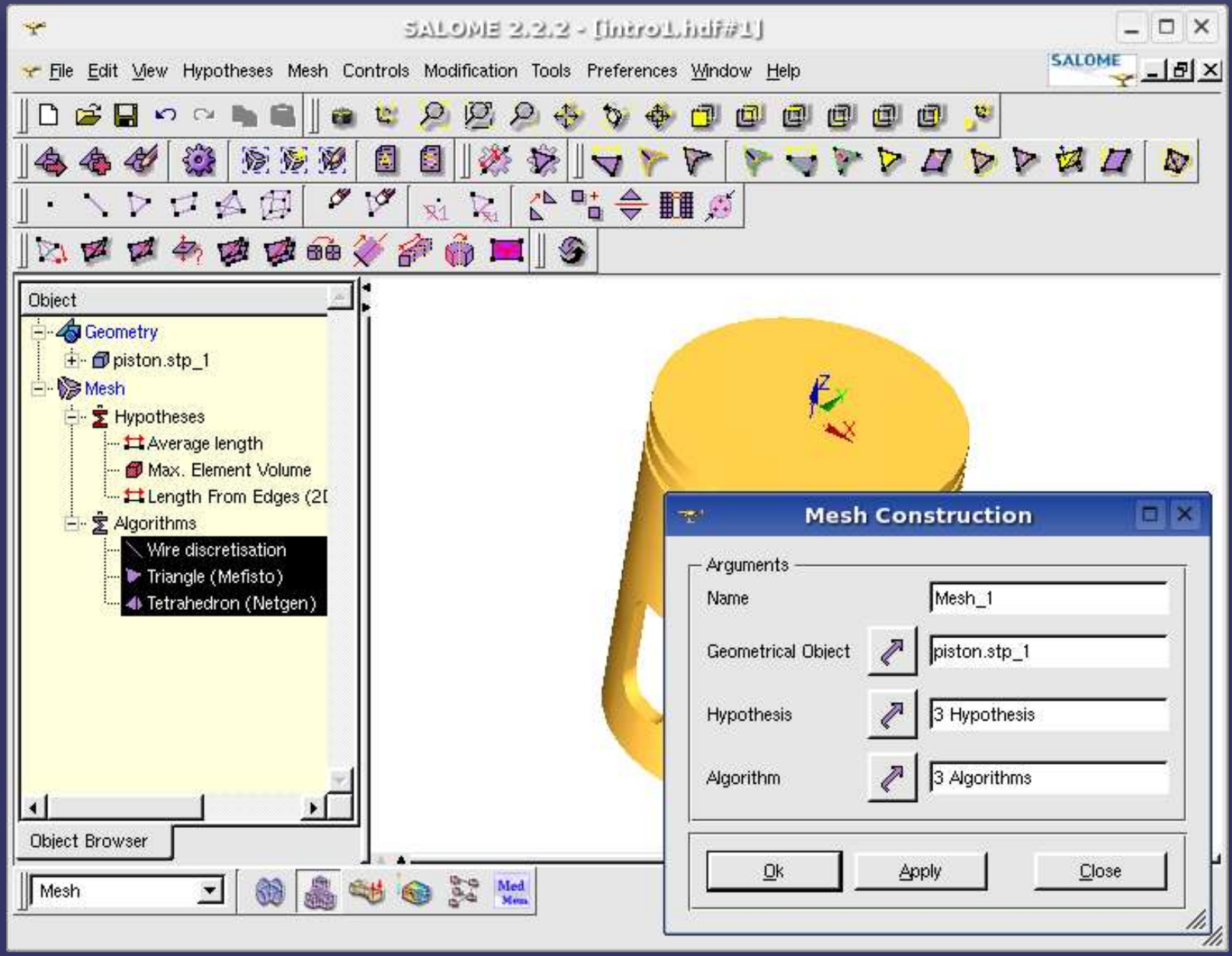
Meshing (2)


SALOME 2.2.2 - [intro1.hdf#1]
File Edit View Hypotheses Mesh Controls Modification Tools Preferences Window Help
SALOME
Object
Geometry
piston.stp_1
Mesh
Hypotheses
Average length
Max. Element Volume
Length From Edges (2[
Algorithms
Wire discretisation
Triangle (Mefisto)
Tetrahedron (Netgen)
Object Browser
Mesh

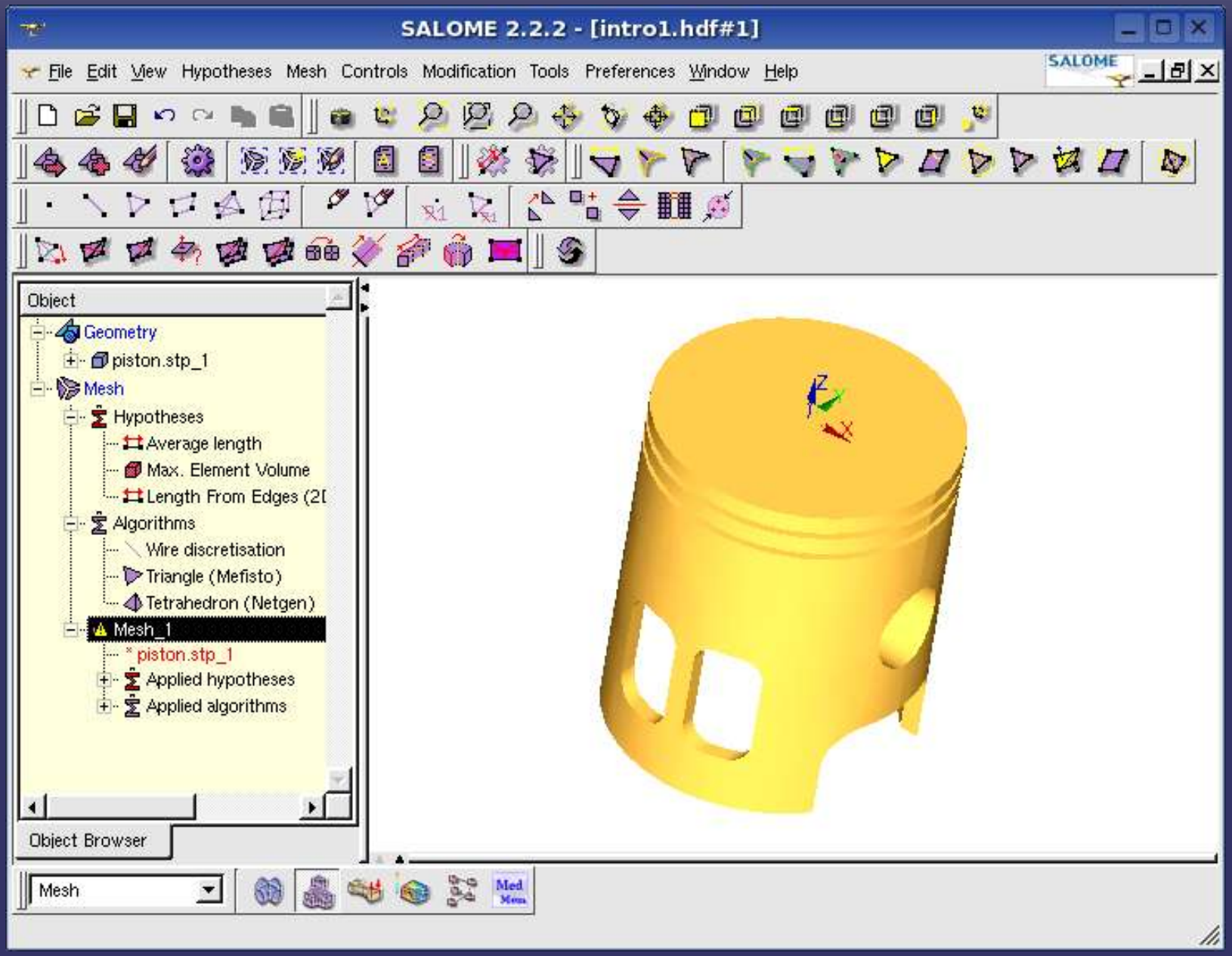
Meshing (3)

- Select menu Mesh-> Global Hypothese
  - Click on Arrow button near Geometrical object
  - Select “piston.stp_1” object
  - Click on Arrow button near Hypotheses
  - By holding the Shift key, select the 3 hypotheses in the tree view
  - Click on Arrow button near Algorithms
  - By holding the Shift key, select the 3 algorithms in the tree view
  - Click OK to create the mesh

Meshing (3)


SALOME 2.2.2 - [intro1.hdf#1]
File Edit View Hypotheses Mesh Controls Modification Tools Preferences Window Help
SALOME
Object
Geometry
piston.stp_1
Mesh
Hypotheses
Average length
Max. Element Volume
Length From Edges (2[
Algorithms
Wire discretisation
Triangle (Mefisto)
Tetrahedron (Netgen)
Object Browser
Mesh
Mesh Construction
Arguments
Name
Mesh_1
Geometrical Object
piston.stp_1
Hypothesis
3 Hypothesis
Algorithm
3 Algorithms
Ok
Apply
Close

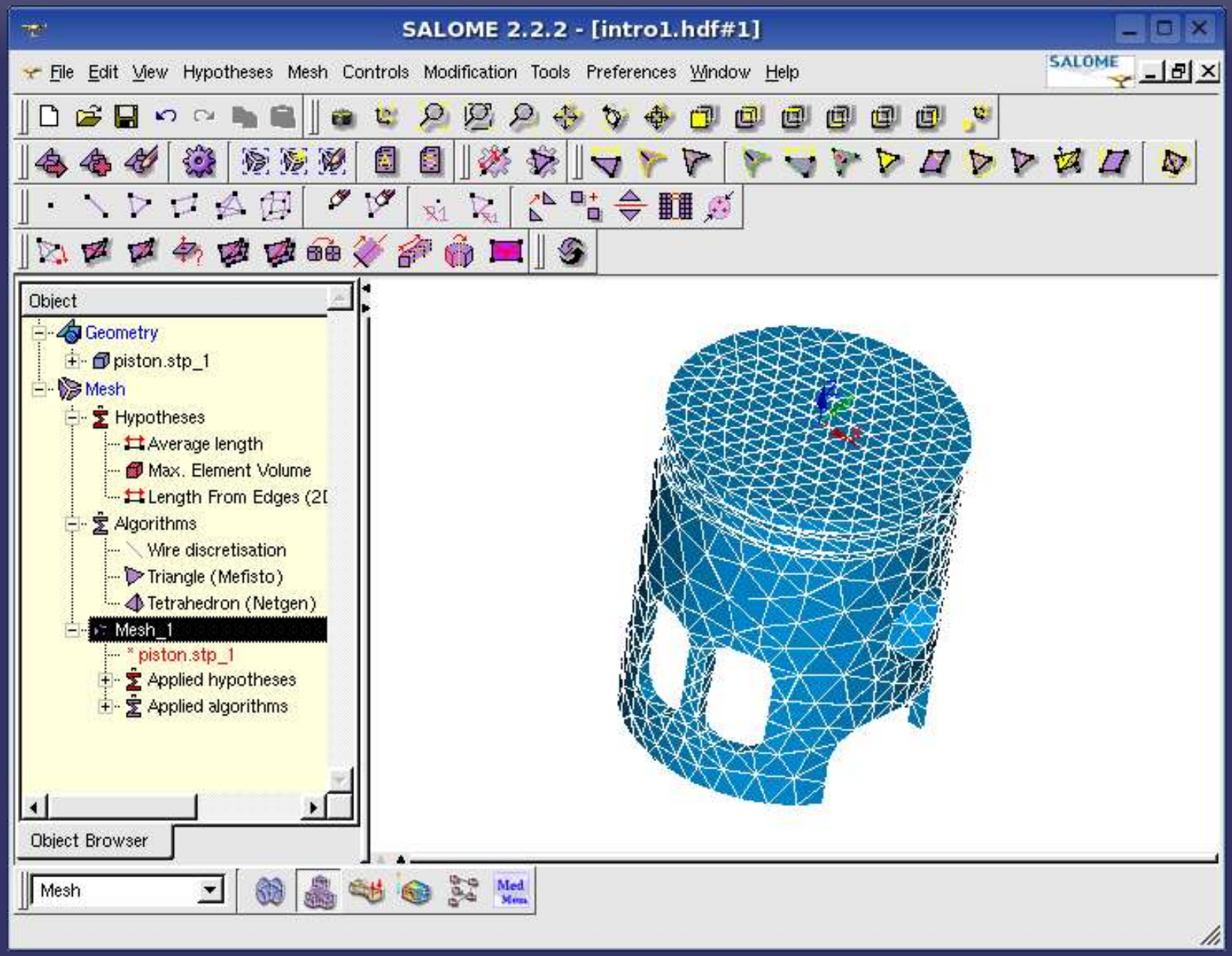
Meshing (3)


SALOME 2.2.2 - [intro1.hdf#1]
File Edit View Hypotheses Mesh Controls Modification Tools Preferences Window Help
SALOME
Object
Geometry
piston.stp_1
Mesh
Hypotheses
Average length
Max. Element Volume
Length From Edges (2[
Algorithms
Wire discretisation
Triangle (Mefisto)
Tetrahedron (Netgen)
Mesh_1
* piston.stp_1
Applied hypotheses
Applied algorithms
Object Browser
Mesh
Med
Mem

Meshing (4)

- Right click the Mesh_1 object in the tree view & select Compute
- When finished, right click the Mesh_1 object & select Update
- Finally, right click the Mesh_1 object & select Display Only

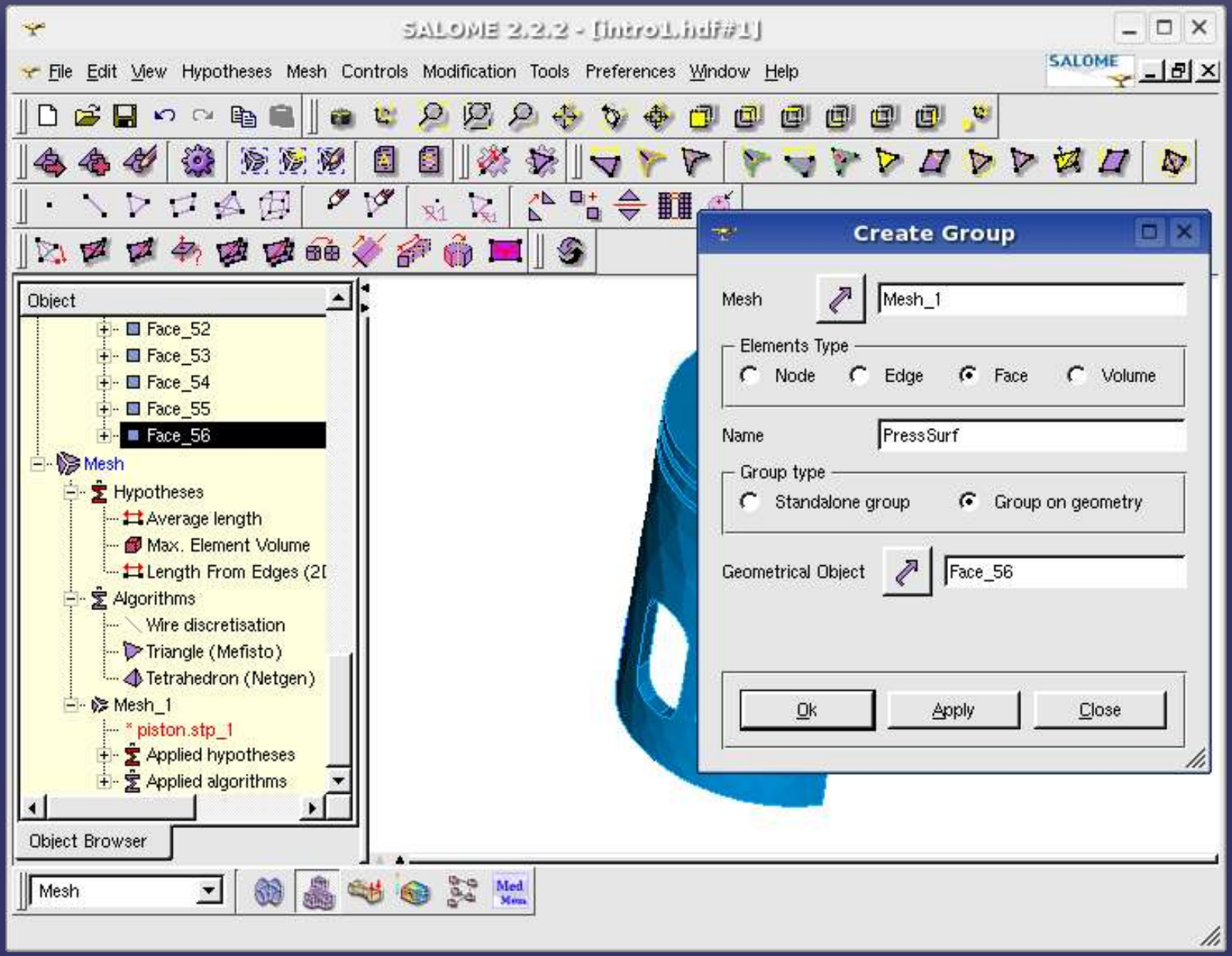
Meshing (4)


SALOME 2.2.2 - [intro1.hdf#1]
File Edit View Hypotheses Mesh Controls Modification Tools Preferences Window Help
SALOME
Object
Geometry
piston.stp_1
Mesh
Hypotheses
Average length
Max. Element Volume
Length From Edges (2[
Algorithms
Wire discretisation
Triangle (Mefisto)
Tetrahedron (Netgen)
Mesh_1
* piston.stp_1
Applied hypotheses
Applied algorithms
Object Browser
Mesh
Med

Create Groups

- Select Menu Mesh-> Create Group
- Select Mesh = Mesh_1
- Choose Elements type = Face
- Choose Group type = Grp on Geometry
- Select "Face_56" from tree view
- Enter name = PressSurf
- Click OK

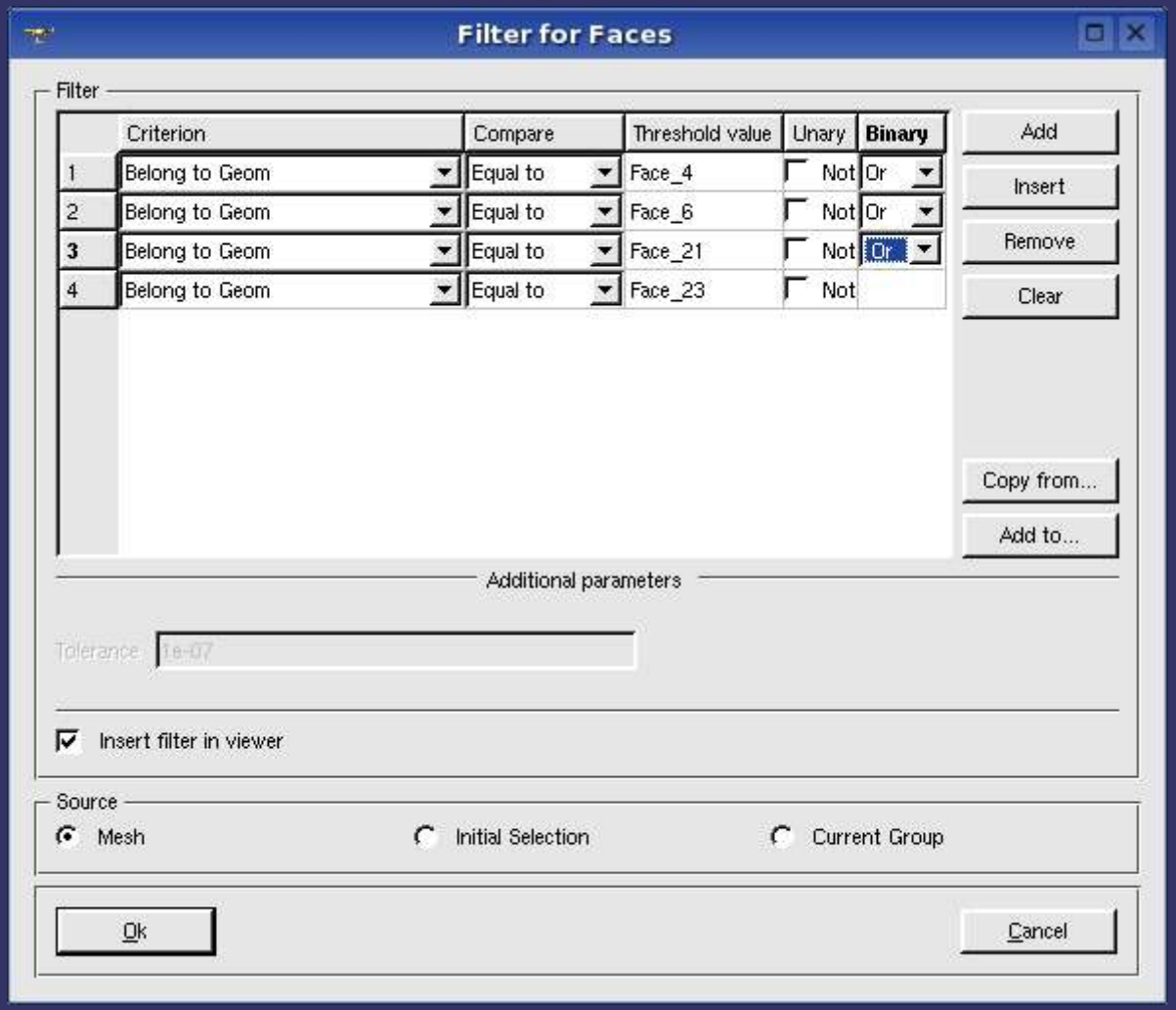
Create Groups

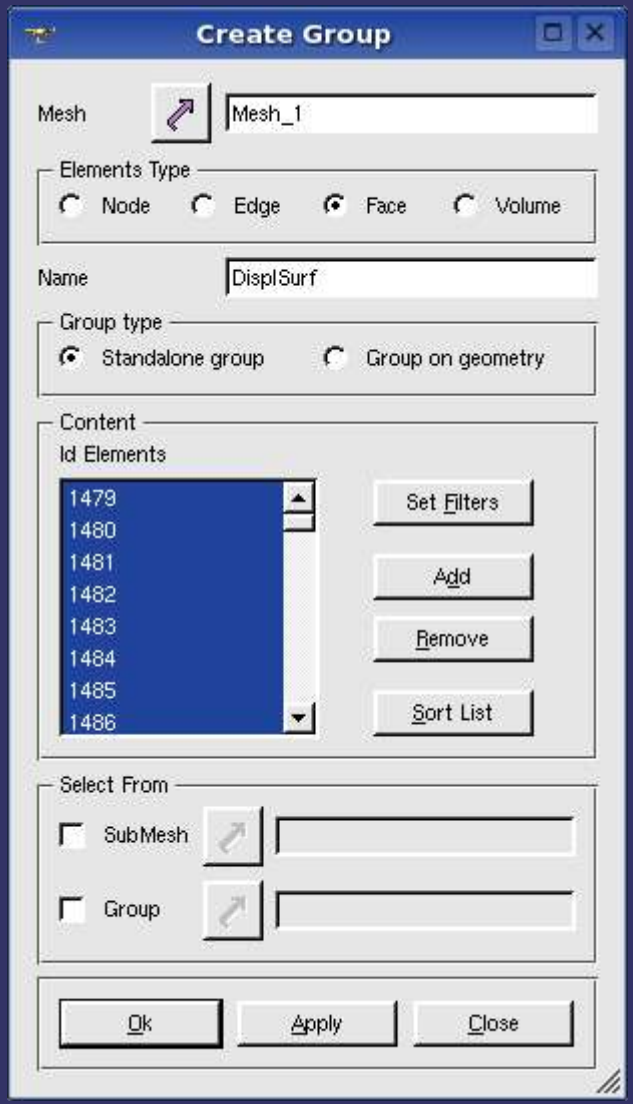


Create Groups (2)

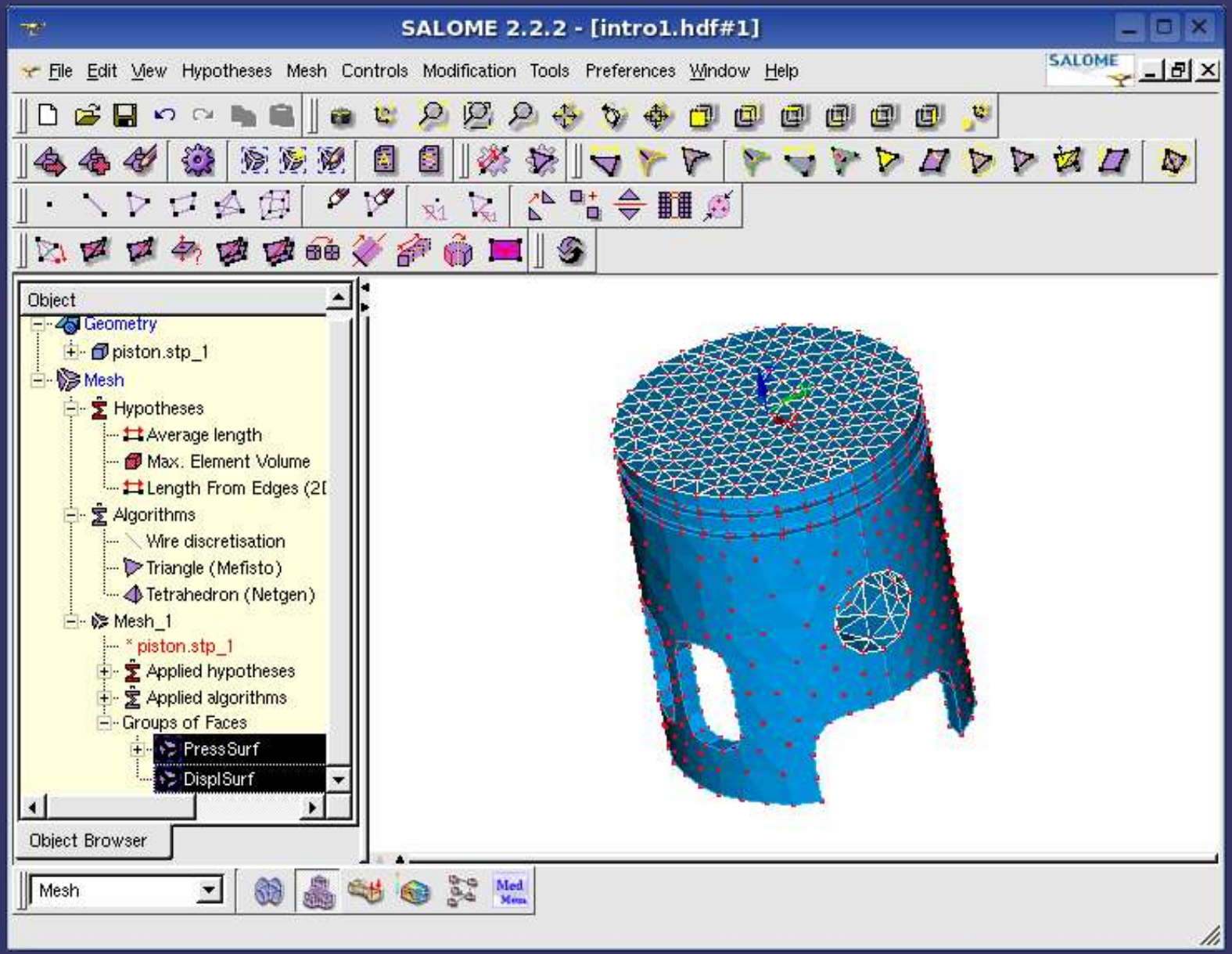
- Select Menu Mesh-> Create Group
- Select Mesh = Mesh_1
- Choose type = Face & Standalone
- Click Set Filters
  - Click Add, choose Criterion= Belong to Geom & enter Face_4 in Value, choose Binary= Or
  - Redo the same operation for Face_6, Face_21 & Face_23
  - Select Source= Mesh & click OK
- Click Add & enter name = DisplSurf
- Click OK

Create groups (2)





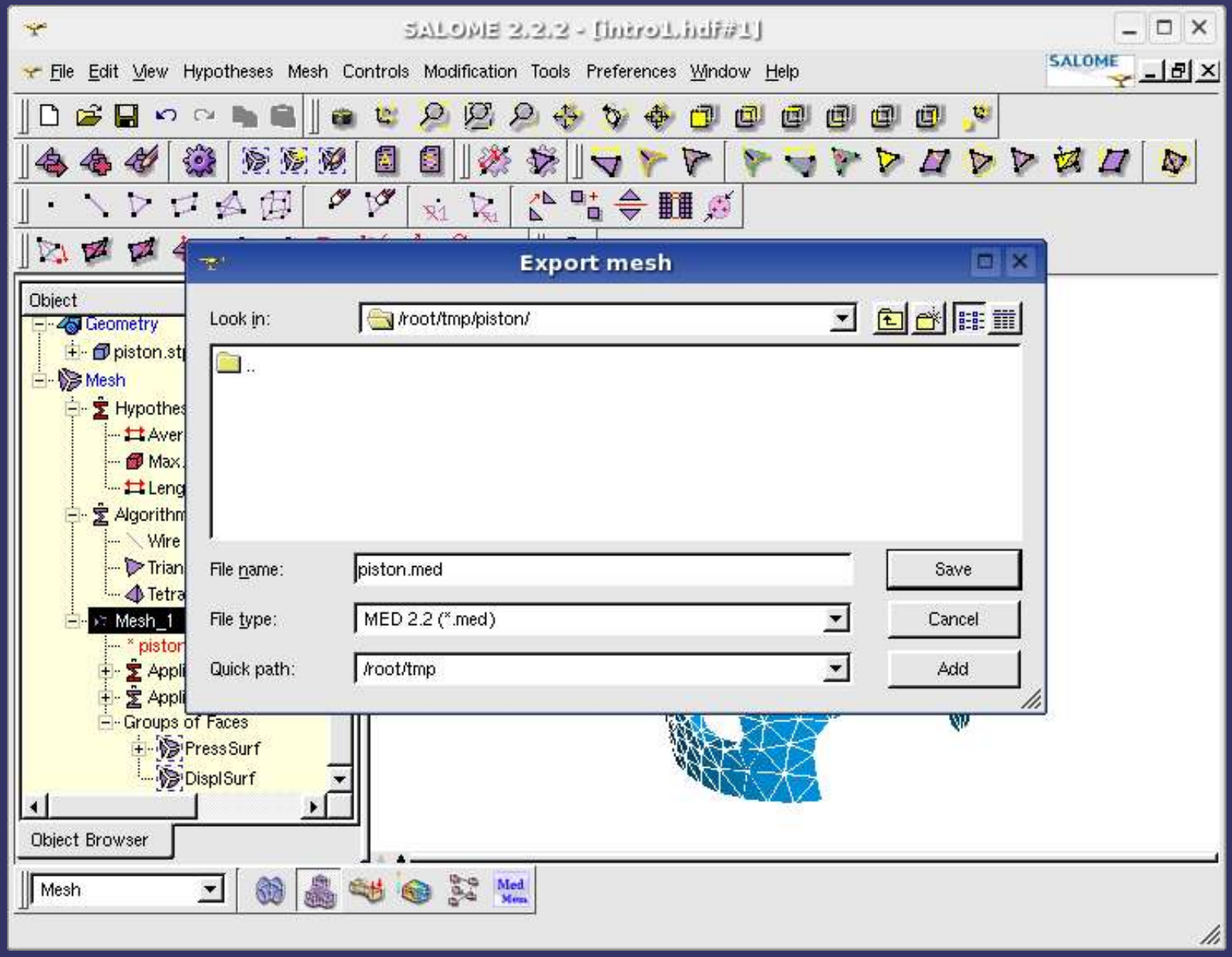
Create groups (2)



Export Mesh

- Right click Mesh_1 in tree view
- Choose Export to MED
- Select format MED 2.2
- Choose directory /root/tmp/piston (create it if necessary)
- Enter filename = piston.med
- Click Save

Export Mesh


SALOME 2.2.2 - [intro1.hdf#1]
File Edit View Hypotheses Mesh Controls Modification Tools Preferences Window Help
SALOME
Export mesh
Look in:
/root/tmp/piston/
..
File name:
piston.med
Save
File type:
MED 2.2 (*.med)
Cancel
Quick path:
/root/tmp
Add
Object
Geometry
piston.st
Mesh
Hypothes
Aver
Max.
Leng
Algorithm
Wire
Trian
Tetra
Mesh_1
Appli
Appli
Groups of Faces
PressSurf
DisplSurf
Object Browser
Mesh
Med

New FE Analysis

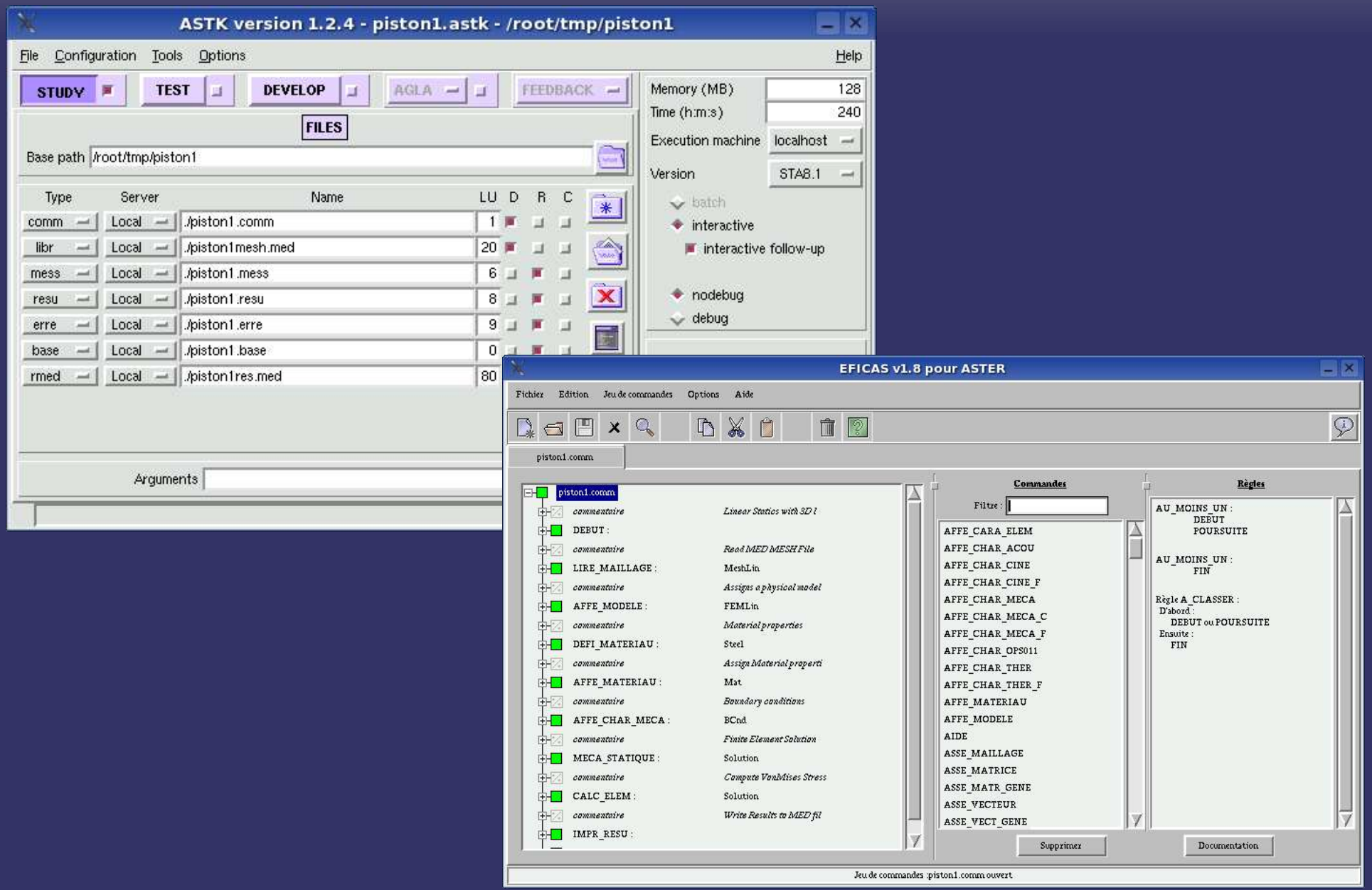
- Reduce Salome window & go back to Desktop
- Start "New FE Analysis"
  - Enter project name= piston1
  - Select base directory = /root/tmp
  - Select piston.med file in /root/tmp/piston
  - Select the template LinearStatics3D.comm in /opt/helpers/Templates
  - Click GO to create project

New FE Analysis

Edit FE model

- Now start the program ASTK
- Click menu File-> Open
- Open /root/tmp/piston1/piston1.astk
- Now double click on piston1.comm in the file list to open the FE command editor EFICAS

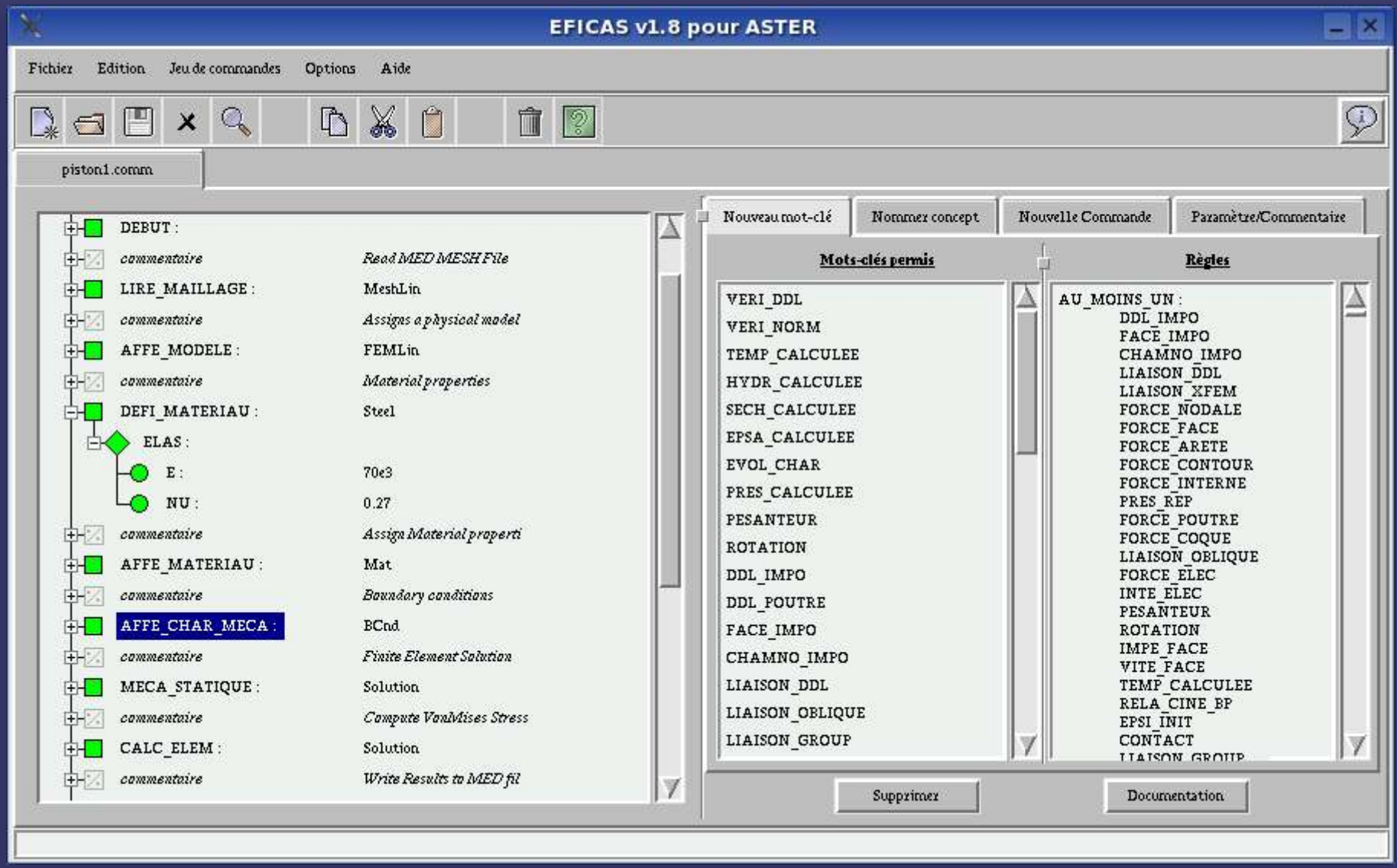
Edit FE model



Edit FE model (2)

- Read the comments in the comm file, explore the details of the commands and try to understand the structure of the proposed FE modelling
- Expand the Defi_Materiau command
  - Select E & enter 70e3 (validate with enter)
  - Select Rho & click 'Supprimer' to delete

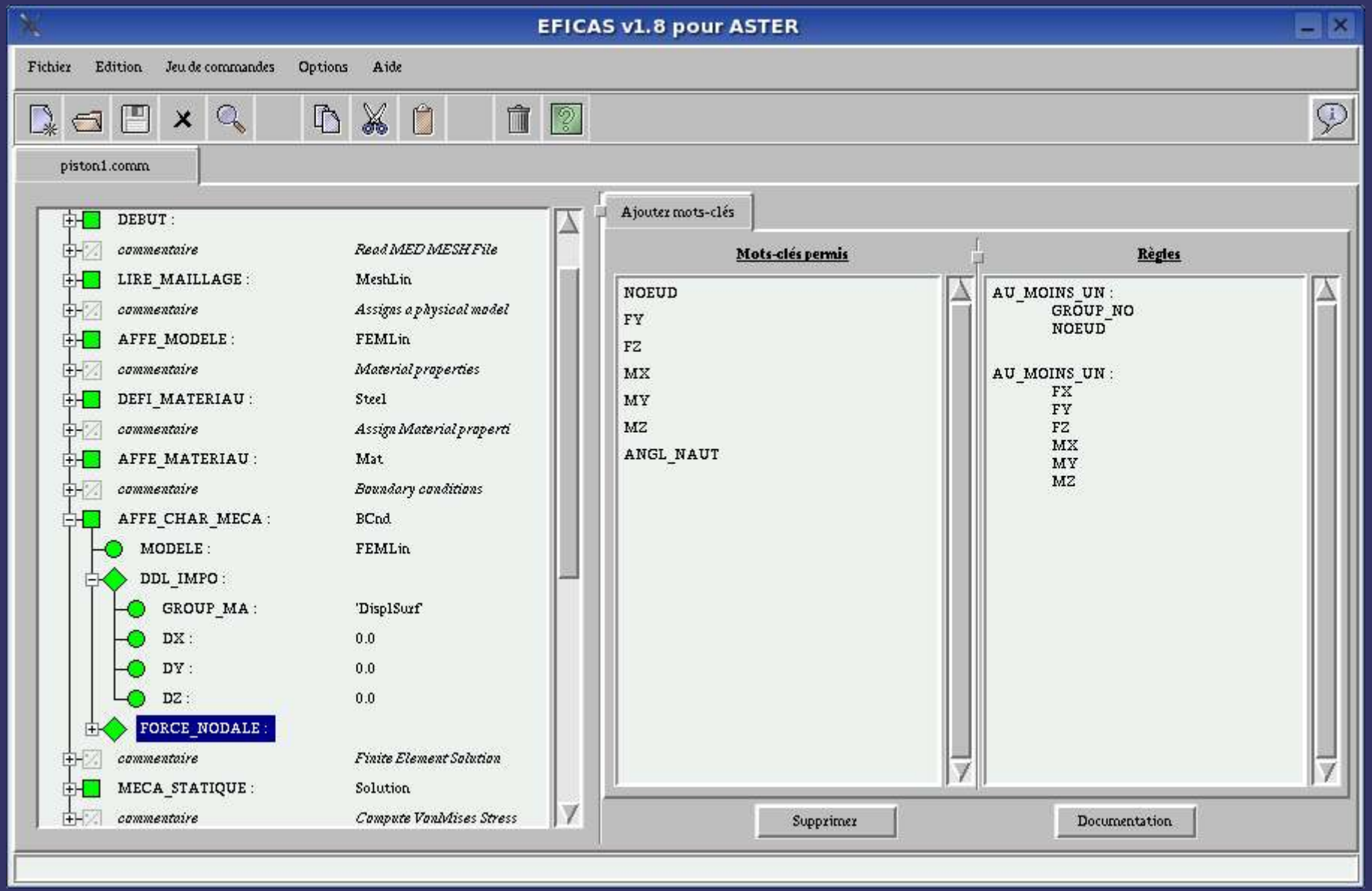
Edit FE model (2)



Edit FE model (3)

- Expand command Affe_Char_Meca that assigns the boundary conditions
- In Ddl_Impo, select Group_No & click 'Supprimer' to delete this option.
- Select Ddl_Impo & double click Group_MA in the list on the right.
- In Valeur enter 'DisplSurf' & press the top 'hand' button
- Validate by a click on 'Valider'

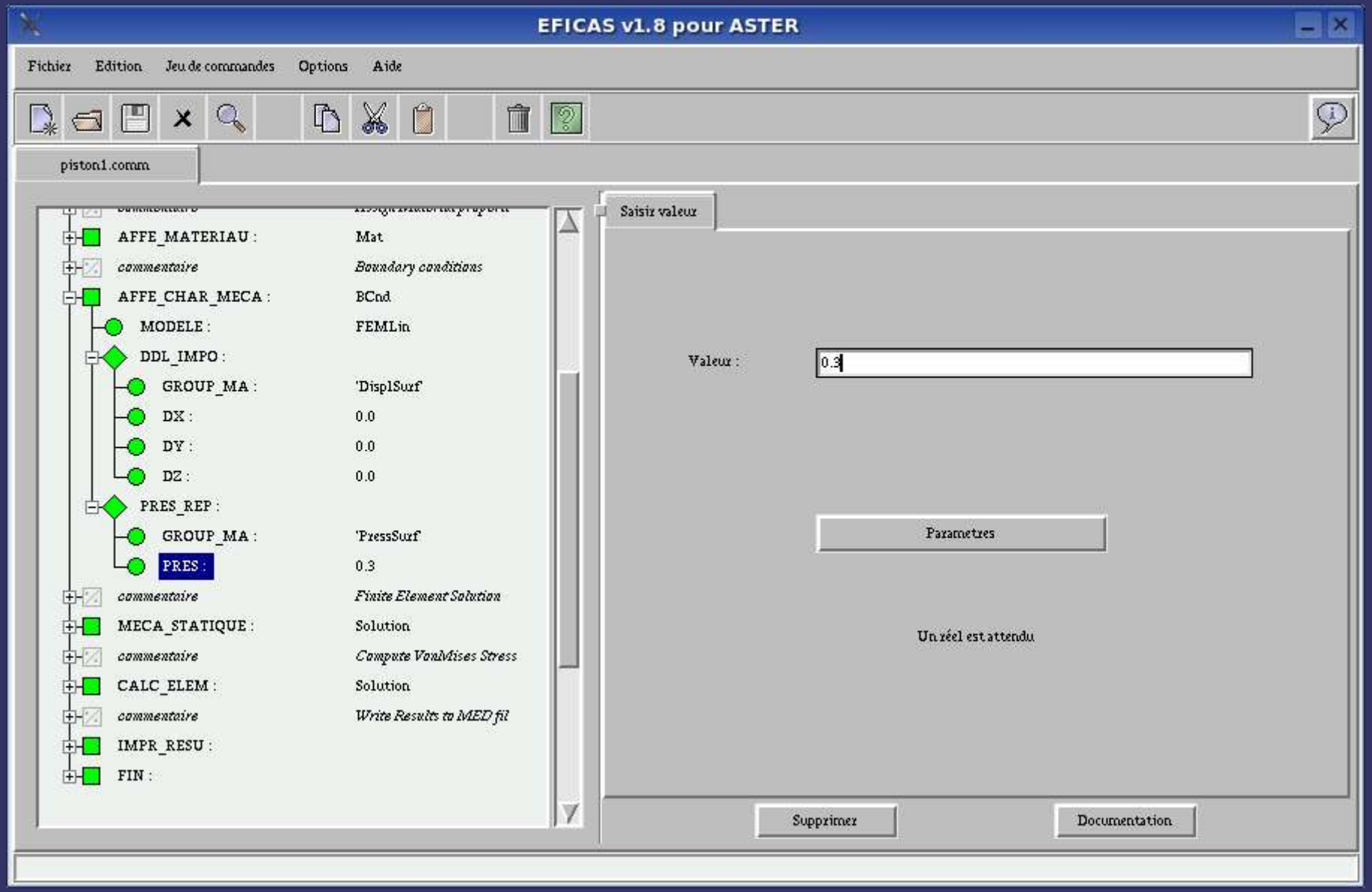
Edit FE model (3)



Edit FE model (4)

- Select Force_Nodale & click 'Supprimer'
- Select Affe_Char_Meca & double click on option Pres_Rep in the list
  - Double click on the Group_MA option & enter group name = 'PressSurf', validate by 'Valider'
  - Select Pres_Rep & double click the Pres option, enter value = 0.3, press enter.
- Click menu Fichier-> Enregistrer to save the file

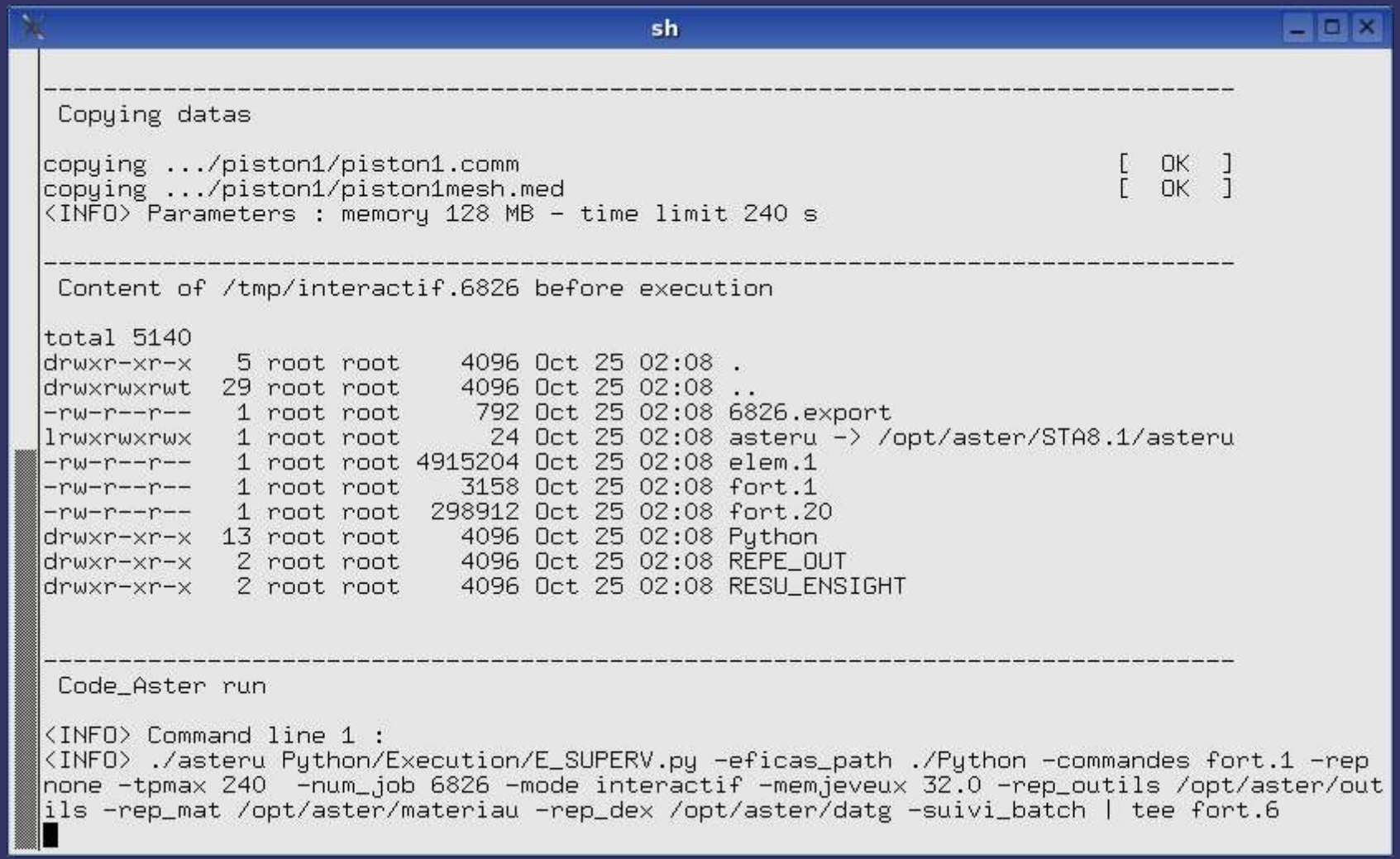
Edit FE model (4)



Solve the FE problem

- Go back to ASTK
- Click Run and try to follow the resolution process.
- When finished, double click piston1.erre file, you should see no errors (marked by < F> flag)
- Double click piston1.resu to view performance data

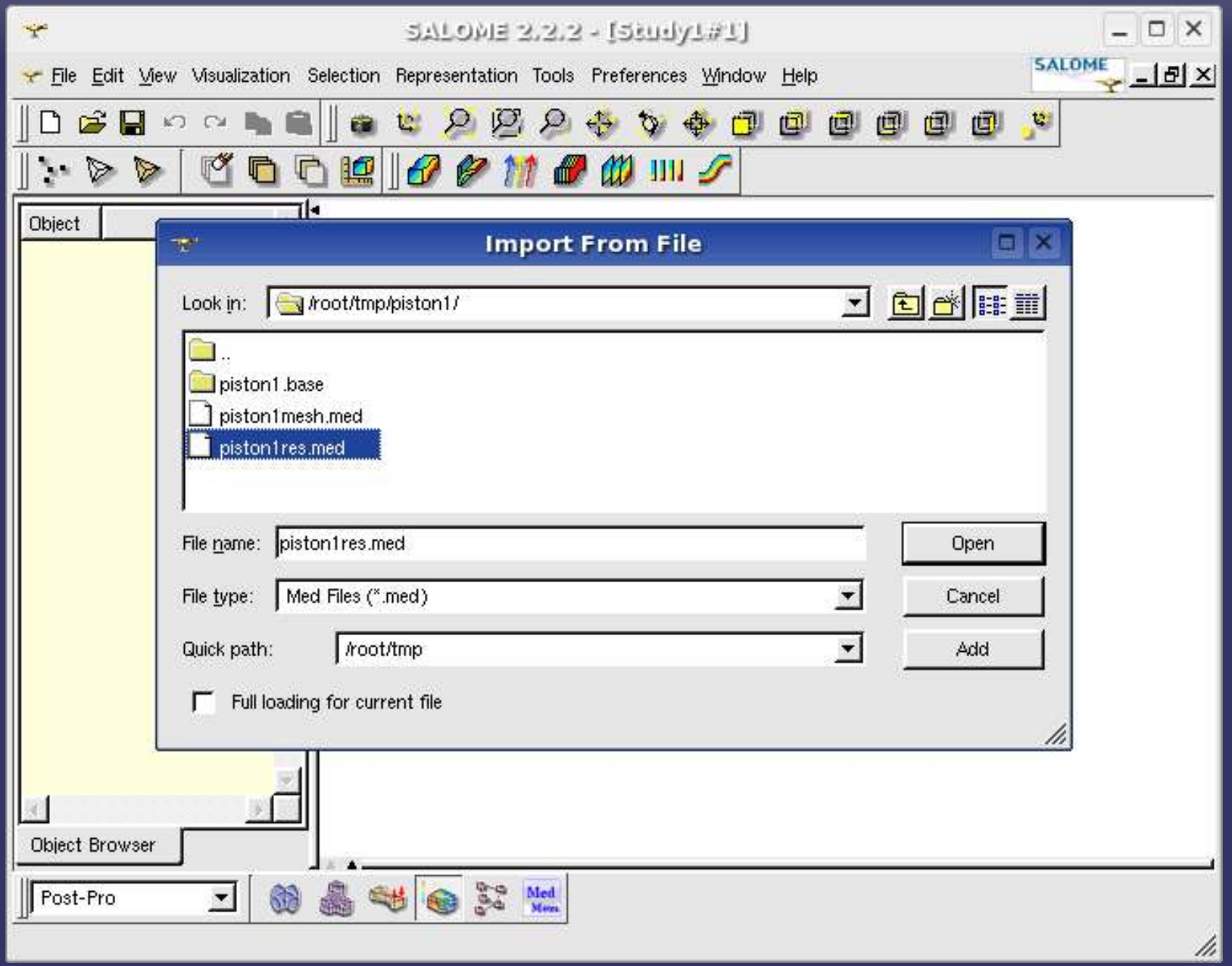
Solve FE problem



Post processing

- Start a new Salome session
- Choose PostPro module
- Validate by a click on New
- Choose menu File-> Import from File
- Select file piston1res.med in /root/tmp/piston1
- Expand the piston1res object to view the content of the file

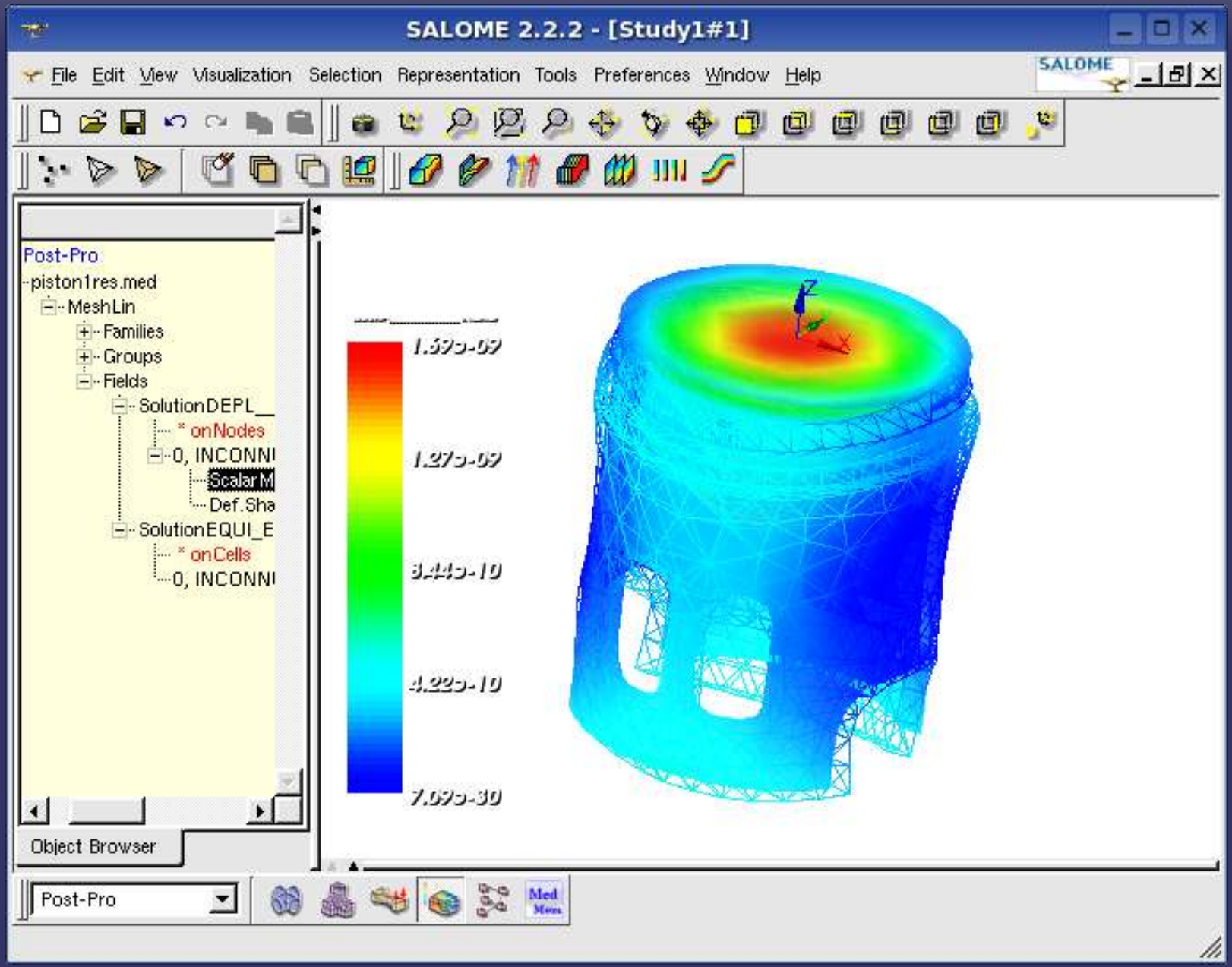
Post processing



Post processing (2)

- Right click SolutionDepl... -> 0,Inconnue... and choose Scalar Map
- Validate the default choices.
- Right click scalar map, choose properties -> opacity & enter 75.
- Right click SolutionDepl... -> 0,Inconnue... and choose Deformed Shape
- Enter scale= 1.7e10 and check magnitude coloring. Validate

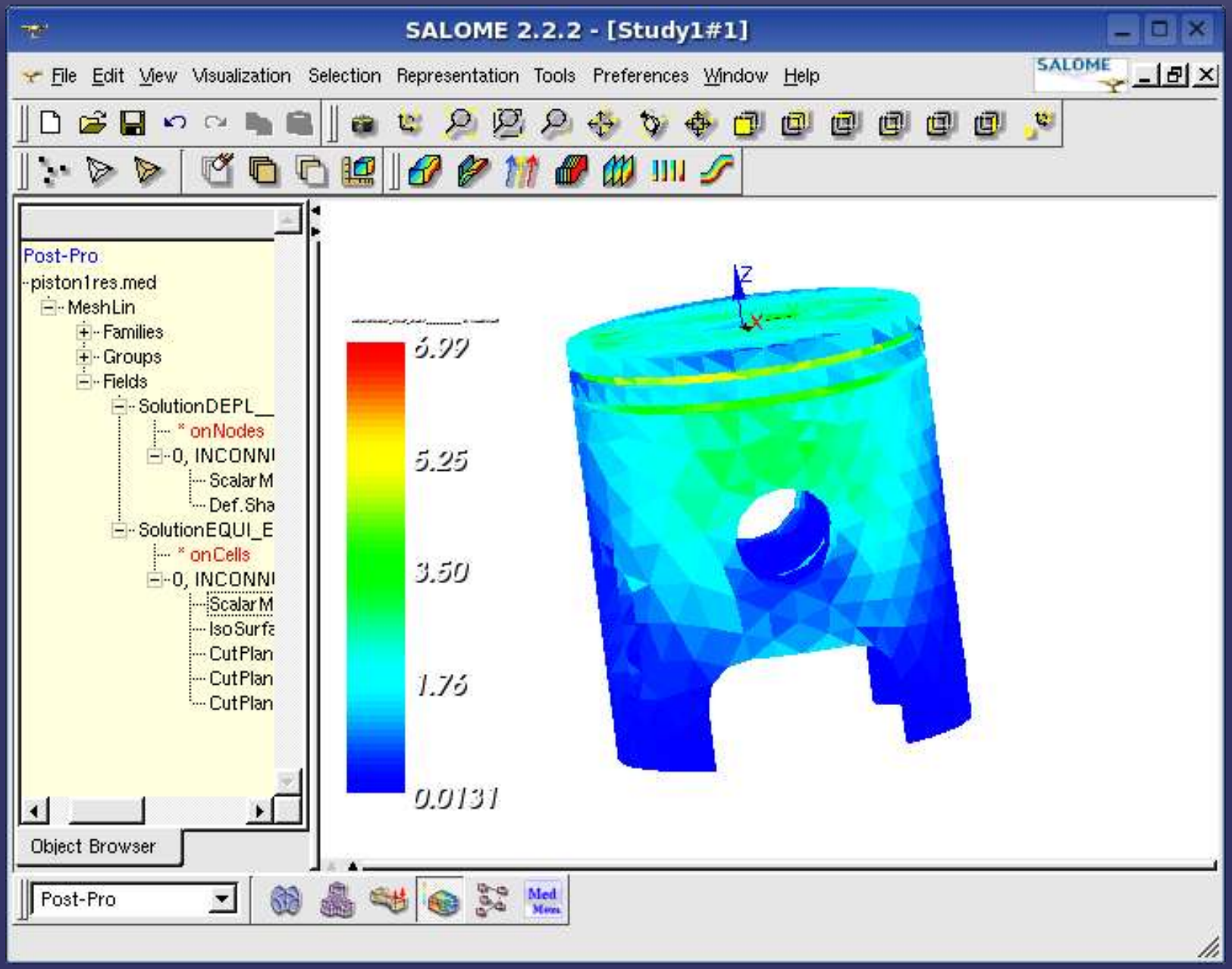
Post Processing (2)



Post Processing (3)

- Right click SolutionEqui_Elno_Sigm -> 0,Inconnu and choose scalar map
- Right click the new Scalar Map object and choose display Only.
- Now you can study the VonMises stresses in the center of each element.
- Try to create other representations of the stress field like IsoSurface or Cut Planes.

Post Processing (3)



CAELinux

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

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

www.caelinux.com

J.Cugnoni, CAElinux.com , 2005