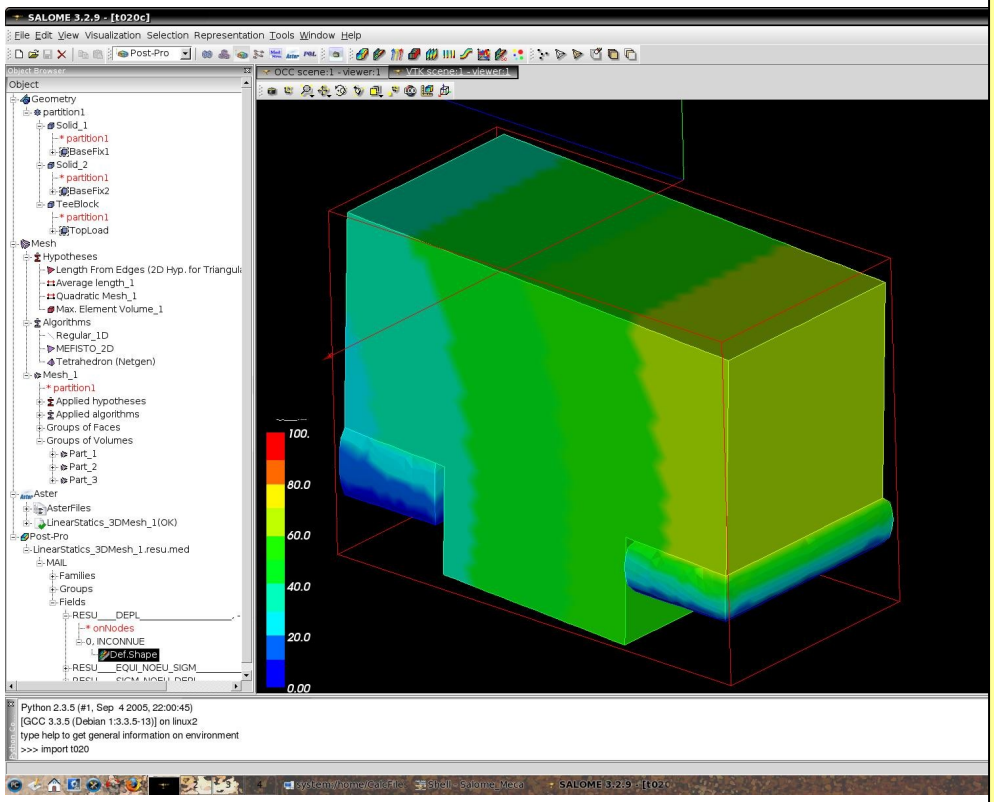


The object of this tutorial is to build a solid object using a script, then to mesh it and solve, and finally to display the solution.

A script has the advantage that, as in this example, a solution for a generic shape can be altered and applied to others of different sizes by changing the parameter values in the script.

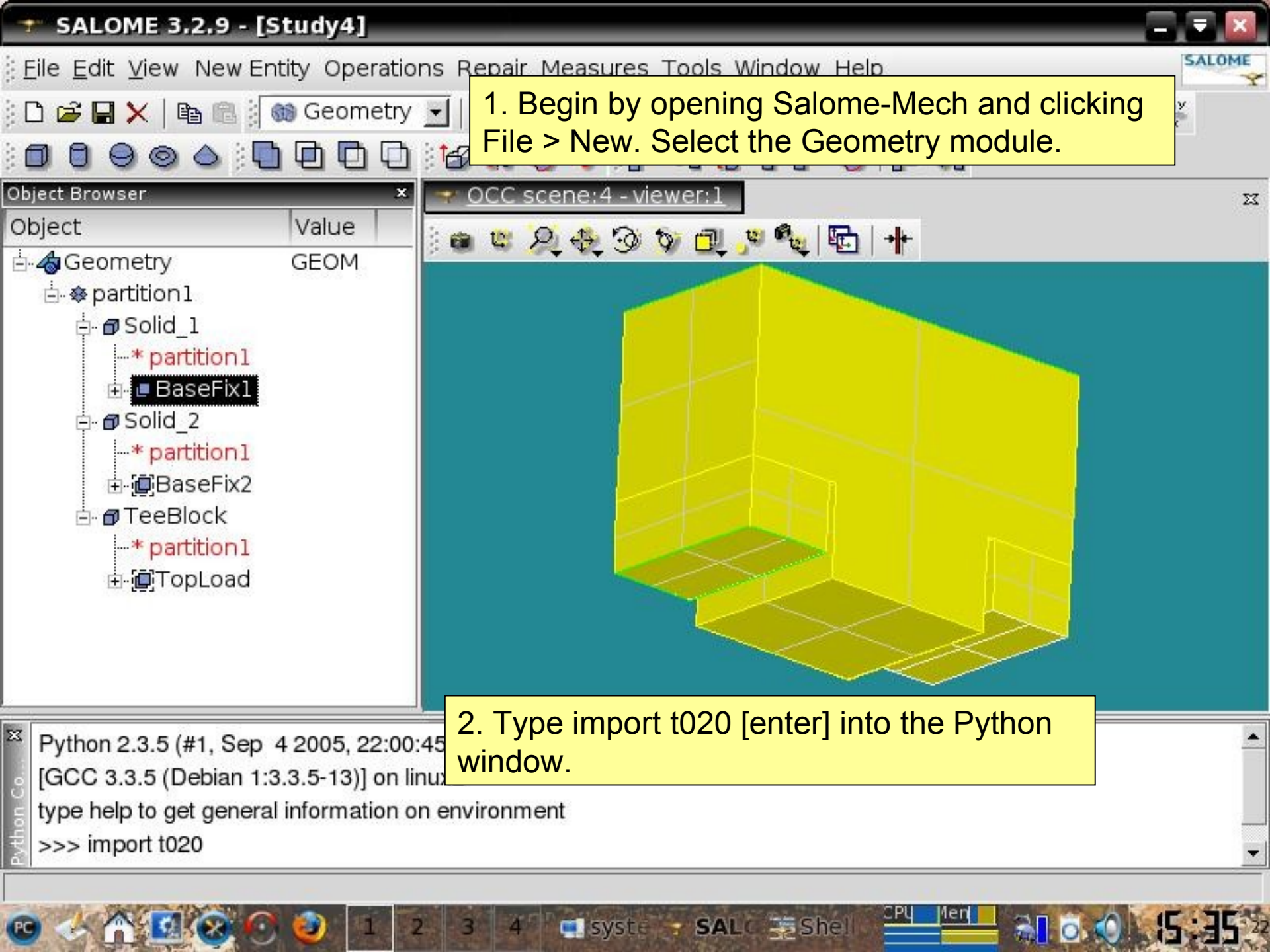
The object is a tee-shaped block on two soft supports. The tutorial demonstrates one method of partitioning the geometry and defining the three material properties.

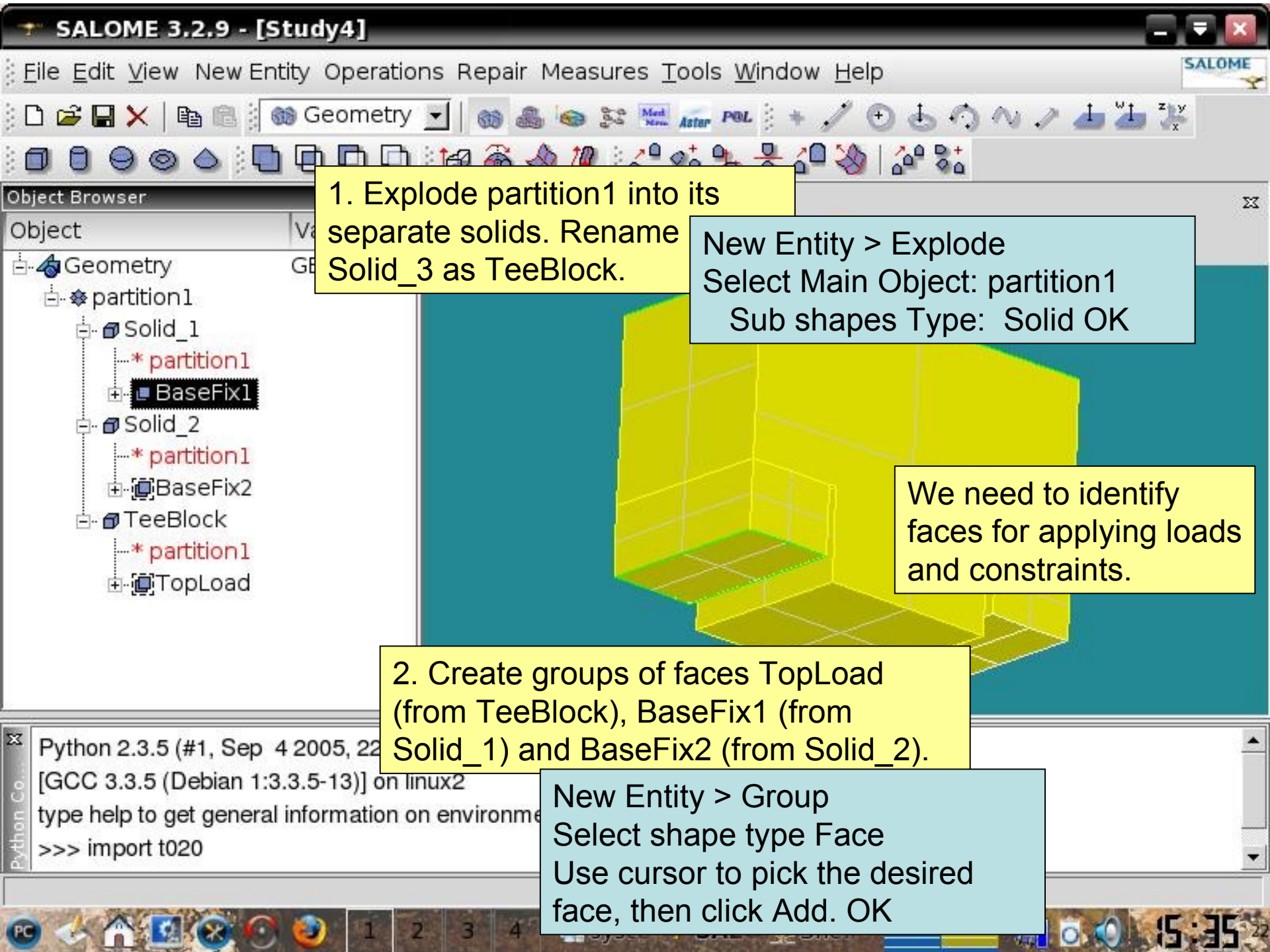
```
# test program t020
# more than 2 parts, nonplanar partition
import geompy
import salome
gg = salome.ImportComponentGUI("GEOM")

# create vertices and vectors
p0 = geompy.MakeVertex(0., 0., 0.)
vectorZ = geompy.MakeVectorDXDYDZ(0., 0., 100.)
planeZ = geompy.MakePlane(p0, vectorZ, 500.)

# create boxes
box1 = geompy.MakeBoxDXDYDZ(200., 300., 420.)
box1t = geompy.MakeTranslation(box1, 0., -300., -210.)
box2 = geompy.MakeBox(-10., -300., 100., 210., -200., 220.)
box2m = geompy.MakeMirrorByPlane(box2, planeZ)
box3 = geompy.MakeBox(0., -275., 110., 200., -200., 210.)
box3m = geompy.MakeMirrorByPlane(box3, planeZ)

# assemble boxes
block1 = geompy.MakeCut(box1t, box2)
block2 = geompy.MakeCut(block1, box2m)
block3 = geompy.MakeFuse(block2, box3)
block4 = geompy.MakeFuse(block3, box3m)

# create partitions
partition1 = geompy.MakePartition([block4], [box3, box3m])

# add objects in the study
id_solid1 = geompy.addToStudy(partition1,"partition1")
# display the boxes
gg.createAndDisplayGO(id_solid1)
gg.setDisplayMode(id_solid1,1)
```

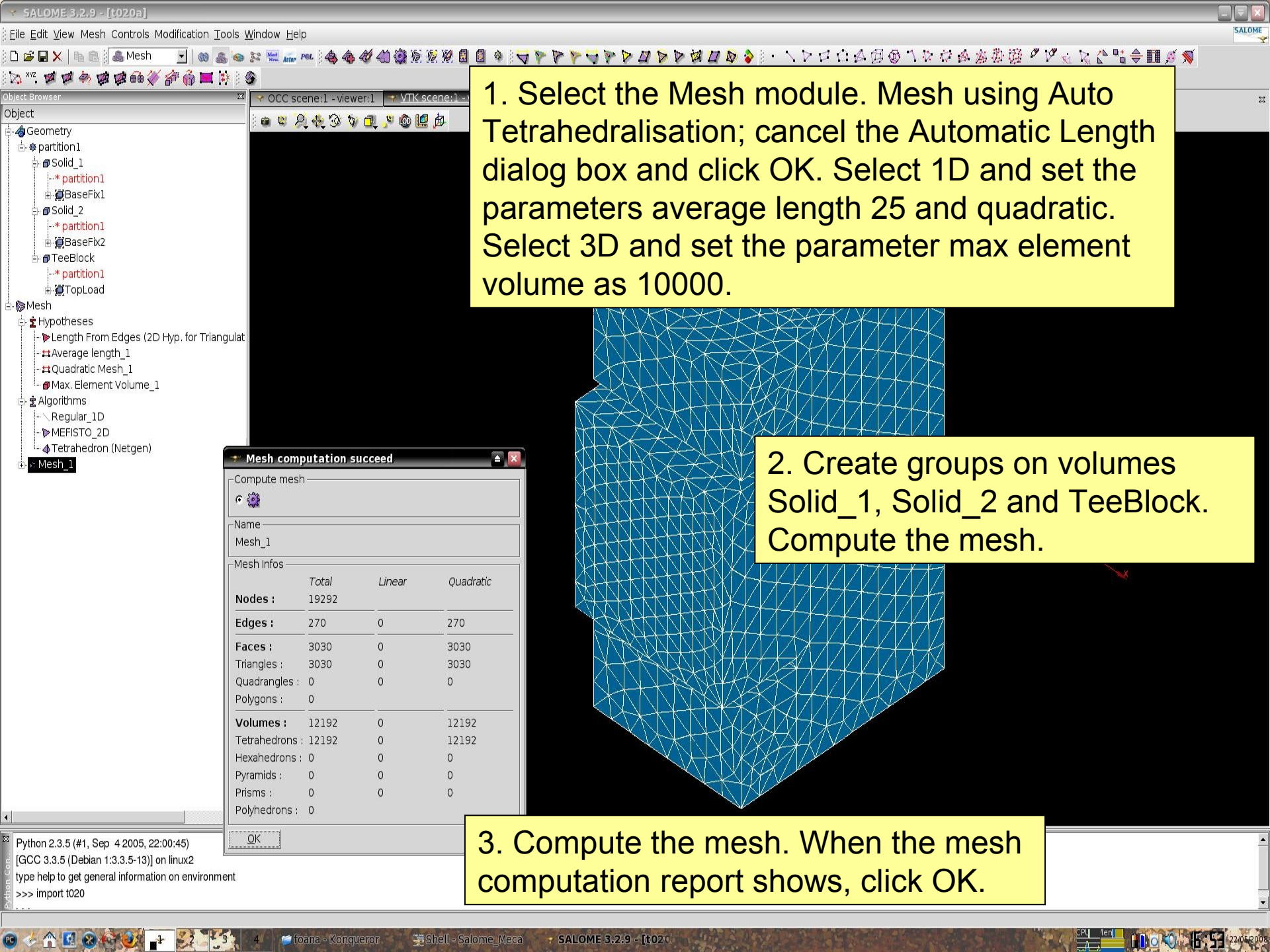
The shape is defined using a script called t020.py which creates a shape called partition1. The script file is saved to the /home folder.

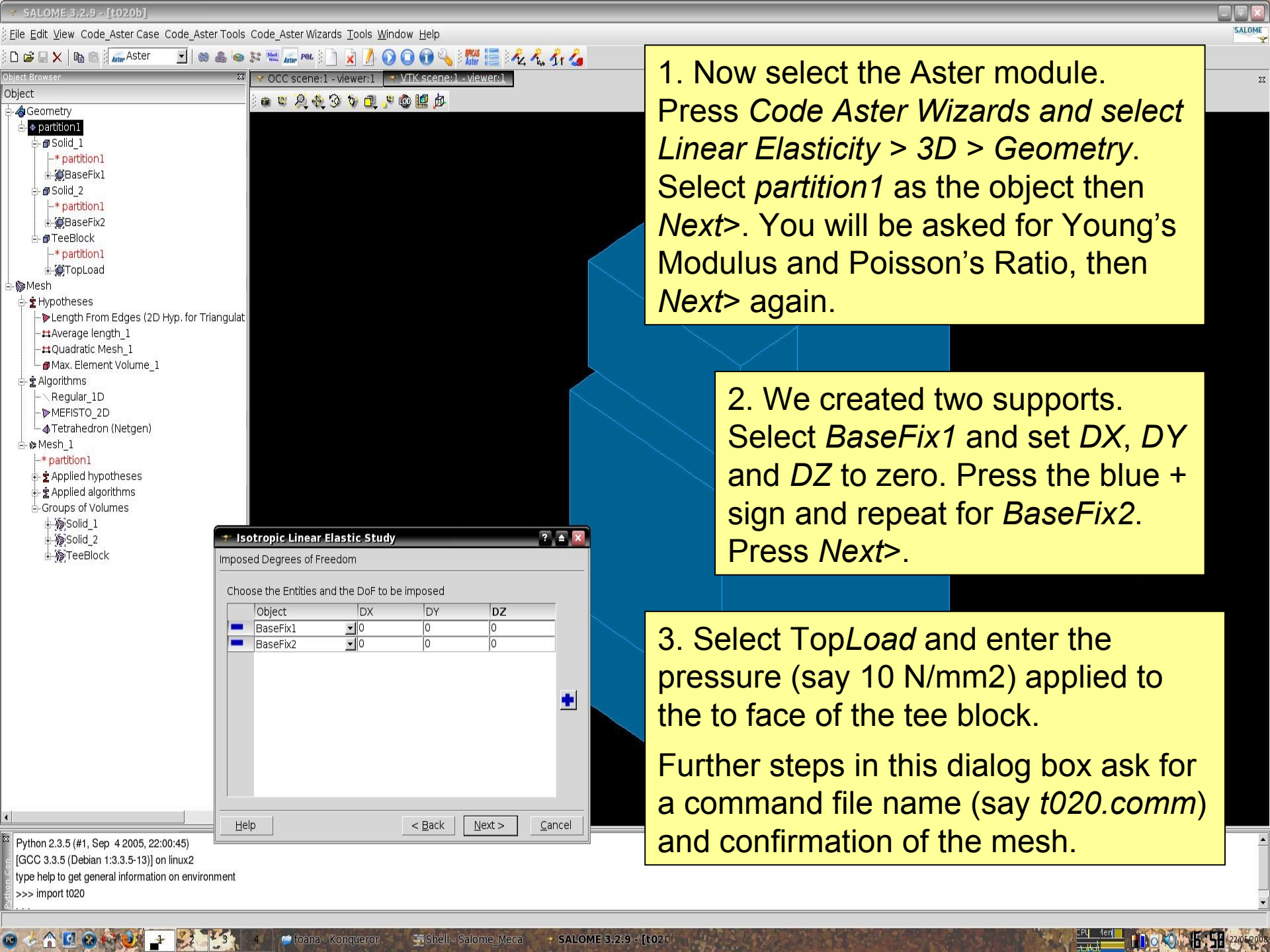
The script is written in Python, and in addition it uses special functions which are prefixed =*geompy*. These are 3D modelling tools which interact with Salome-Mech to create the geometry on which meshing and solution are based.

The functions are explained on the Salome website.


SALOME 3.2.9 - [Study4]
File Edit View New Entity Operations Repair Measures Tools Window Help
Geometry
1. Begin by opening Salome-Mech and clicking File > New. Select the Geometry module.
Object Browser
Object
Value
Geometry
GEOM
partition1
Solid_1
* partition1
BaseFix1
Solid_2
* partition1
BaseFix2
TeeBlock
* partition1
TopLoad
OCC scene:4 - viewer:1
2. Type import t020 [enter] into the Python window.
Python 2.3.5 (#1, Sep 4 2005, 22:00:45
[GCC 3.3.5 (Debian 1:3.3.5-13)] on linux
type help to get general information on environment
>>> import t020


SALOME 3.2.9 - [Study4]
File Edit View New Entity Operations Repair Measures Tools Window Help
SALOME
Geometry
Object Browser
Object
Geometry
partition1
Solid_1
* partition1
BaseFix1
Solid_2
* partition1
BaseFix2
TeeBlock
* partition1
TopLoad
1. Explode partition1 into its separate solids. Rename Solid_3 as TeeBlock.
New Entity > Explode
Select Main Object: partition1
Sub shapes Type: Solid OK
We need to identify faces for applying loads and constraints.
2. Create groups of faces TopLoad (from TeeBlock), BaseFix1 (from Solid_1) and BaseFix2 (from Solid_2).
New Entity > Group
Select shape type Face
Use cursor to pick the desired face, then click Add. OK
Python 2.3.5 (#1, Sep 4 2005, 22
[GCC 3.3.5 (Debian 1:3.3.5-13)] on linux2
type help to get general information on environme
>>> import t020
Python Co...
1
2
3
4
15:35

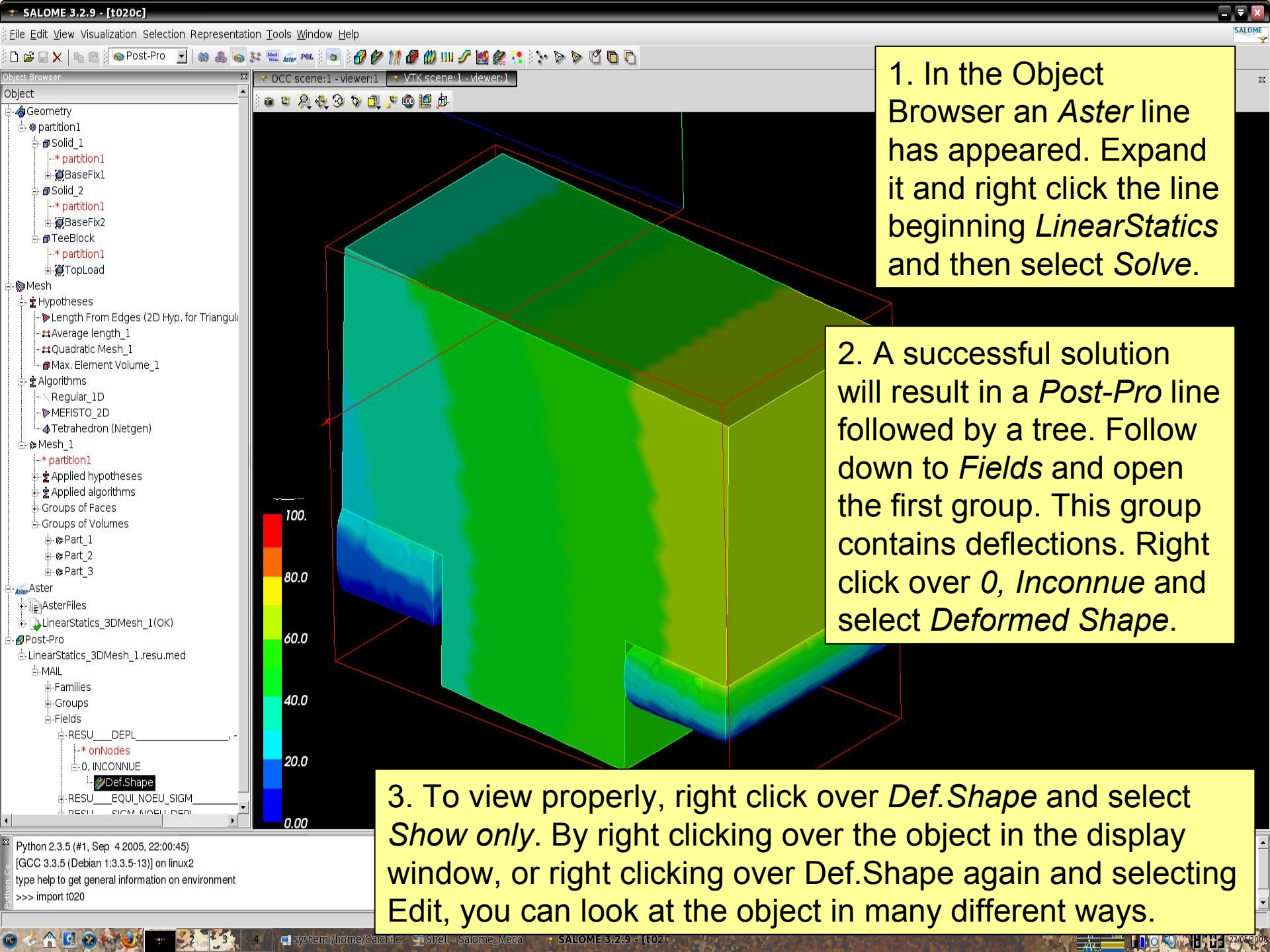

SALOME 3.2.9 - [t020a]
File Edit View Mesh Controls Modification Tools Window Help
Object Browser
Object
Geometry
partition1
Solid_1
* partition1
BaseFix1
Solid_2
* partition1
BaseFix2
TeeBlock
* partition1
TopLoad
Mesh
Hypotheses
Length From Edges (2D Hyp. for Triangulat
Average length_1
Quadratic Mesh_1
Max. Element Volume_1
Algorithms
Regular_1D
MEFISTO_2D
Tetrahedron (Netgen)
Mesh_1
OCC scene:1 - viewer:1
VTK scene:1
1. Select the Mesh module. Mesh using Auto Tetrahedralisation; cancel the Automatic Length dialog box and click OK. Select 1D and set the parameters average length 25 and quadratic. Select 3D and set the parameter max element volume as 10000.
Mesh computation succeed
Compute mesh
Name
Mesh_1
Mesh Infos
Total Linear Quadratic
Nodes : 19292
Edges : 270 0 270
Faces : 3030 0 3030
Triangles : 3030 0 3030
Quadrangles : 0 0 0
Polygons : 0
Volumes : 12192 0 12192
Tetrahedrons : 12192 0 12192
Hexahedrons : 0 0 0
Pyramids : 0 0 0
Prisms : 0 0 0
Polyhedrons : 0
OK
2. Create groups on volumes Solid_1, Solid_2 and TeeBlock. Compute the mesh.
3. Compute the mesh. When the mesh computation report shows, click OK.
Python 2.3.5 (#1, Sep 4 2005, 22:00:45)
[GCC 3.3.5 (Debian 1:3.3.5-13)] on linux2
type help to get general information on environment
>>> import t020
foana - Konqueror
Shell - Salome_Meca
SALOME 3.2.9 - [t020


SALOME 3.2.9 - [t020b]
File Edit View Code_Aster Case Code_Aster Tools Code_Aster Wizards Tools Window Help
Aster
Object Browser
Object
Geometry
partition1
Solid_1
* partition1
BaseFix1
Solid_2
* partition1
BaseFix2
TeeBlock
* partition1
TopLoad
Mesh
Hypotheses
Length From Edges (2D Hyp. for Triangulat
Average length_1
Quadratic Mesh_1
Max. Element Volume_1
Algorithms
Regular_1D
MEFISTO_2D
Tetrahedron (Netgen)
Mesh_1
* partition1
Applied hypotheses
Applied algorithms
Groups of Volumes
Solid_1
Solid_2
TeeBlock
OCC scene:1 - viewer:1
VTK scene:1 - viewer:1
Isotropic Linear Elastic Study
Imposed Degrees of Freedom
Choose the Entities and the DoF to be imposed
Object
DX
DY
DZ
BaseFix1
0
0
0
BaseFix2
0
0
0
Help
< Back
Next >
Cancel
Python 2.3.5 (#1, Sep 4 2005, 22:00:45)
[GCC 3.3.5 (Debian 1:3.3.5-13)] on linux2
type help to get general information on environment
>>> import t020
1. Now select the Aster module. Press Code Aster Wizards and select Linear Elasticity > 3D > Geometry. Select partition1 as the object then Next>. You will be asked for Young's Modulus and Poisson's Ratio, then Next> again.
2. We created two supports. Select BaseFix1 and set DX, DY and DZ to zero. Press the blue + sign and repeat for BaseFix2. Press Next>.
3. Select TopLoad and enter the pressure (say 10 N/mm2) applied to the to face of the tee block.
Further steps in this dialog box ask for a command file name (say t020.comm) and confirmation of the mesh.

```
DEBUT();

Steel=DEFI_MATERIAU(ELAS=_F(E=200000.,
                    NU=0.3,),);
Rubber1=DEFI_MATERIAU(ELAS=_F(E=22.5,
                    NU=0.3,),);
Rubber2=DEFI_MATERIAU(ELAS=_F(E=45.,
                    NU=0.3,),);

MAIL=LIRE_MAILLAGE(FORMAT='MED',);

MODE=AFFE_MODELE(MAILLAGE=MAIL,
                 AFFE=_F(TOUT='OUI',
                 PHENOMENE='MECANIQUE',
                 MODELISATION= '3D',),);

MAIL=MODI_MAILLAGE(reuse =MAIL,
                 MAILLAGE=MAIL,
                 ORIE_PEAU_3D=_F( GROUP_MA=('TopLoad',),),
                              );

MATE=AFFE_MATERIAU(MAILLAGE=MAIL,
              AFFE=(_F(GROUP_MA='Part_1',
                    MATER=Rubber1,),
                    _F(GROUP_MA='Part_2',
                    MATER=Rubber2,),
                    _F(GROUP_MA='Part_3',
                    MATER=Steel,),),);

CHAR=AFFE_CHAR_MECA(MODELE=MODE,
                 DDL_IMPO=(
                 _F(GROUP_MA='BaseFix1',
                              DX=0,
                              DY=0,
                              DZ=0,),
                 _F(GROUP_MA='BaseFix2',
                              DX=0,
                              DY=0,
                              DZ=0,),),
                 PRES_REP=(
                              _F(GROUP_MA='TopLoad',
                              PRES=10,),),);

RESU=MECA_STATIQUE(MODELE=MODE,
                 CHAM_MATER=MATE,
                 EXCIT=_F(CHARGE=CHAR,),);

RESU=CALC_ELEM(reuse =RESU,
                 MODELE=MODE,
                 CHAM_MATER=MATE,
                 RESULTAT=RESU,
                 OPTION=('SIGM_ELNO_DEPL','EQUI_ELNO_SIGM',),
                 EXCIT=_F(
                 CHARGE=CHAR,),);

RESU=CALC_NO(reuse =RESU,
                 RESULTAT=RESU,
                 OPTION=('SIGM_NOEU_DEPL', 'EQUI_NOEU_SIGM', ),);

IMPR_RESU(FORMAT='MED',
                 UNITE=80,
                 RESU=_F(MAILLAGE=MAIL,
                      RESULTAT=RESU,

NOM_CHAM=('SIGM_NOEU_DEPL','EQUI_NOEU_SIGM','DEPL',),),);

FIN();
```

The command file contains details of the loads, constraints and includes the material definition (Young's modulus and Poisson's Ratio).

Note that in the file created by Salome, the three parts are all of the same material. Edit the file as shown emboldened.


SALOME 3.2.9 - [t020c]
File Edit View Visualization Selection Representation Tools Window Help
Post-Pro
Object Browser
Object
Geometry
partition1
Solid_1
* partition1
BaseFix1
Solid_2
* partition1
BaseFix2
TeeBlock
* partition1
TopLoad
Mesh
Hypotheses
Length From Edges (2D Hyp. for Triangula
Average length_1
Quadratic Mesh_1
Max. Element Volume_1
Algorithms
Regular_1D
MEFISTO_2D
Tetrahedron (Netgen)
Mesh_1
* partition1
Applied hypotheses
Applied algorithms
Groups of Faces
Groups of Volumes
Part_1
Part_2
Part_3
Aster
AsterFiles
LinearStatics_3DMesh_1(OK)
Post-Pro
LinearStatics_3DMesh_1.resu.med
MAIL
Families
Groups
Fields
RESU____DEPL
* onNodes
0, INCONNUE
Def.Shape
RESU____EQUI_NOEU_SIGM
OCC scene:1 - viewer:1
VTK scene:1 - viewer:1
100.
80.0
60.0
40.0
20.0
0.00
Python 2.3.5 (#1, Sep 4 2005, 22:00:45)
[GCC 3.3.5 (Debian 1:3.3.5-13)] on linux2
type help to get general information on environment
>>> import t020
1. In the Object Browser an Aster line has appeared. Expand it and right click the line beginning LinearStatics and then select Solve.
2. A successful solution will result in a Post-Pro line followed by a tree. Follow down to Fields and open the first group. This group contains deflections. Right click over 0, Inconnue and select Deformed Shape.
3. To view properly, right click over Def.Shape and select Show only. By right clicking over the object in the display window, or right clicking over Def.Shape again and selecting Edit, you can look at the object in many different ways.

```
Steel=DEFI_MATERIAU(ELAS=_F(E=200000.,
                            NU=0.3,),);

Rubber1=DEFI_MATERIAU(ELAS=_F(E=200.,
                            NU=0.3,),);

Rubber2=DEFI_MATERIAU(ELAS=_F(E=400.,
                            NU=0.3,),);
```

The deflections are rather excessive with very soft rubber pads. This is a better choice of material properties.