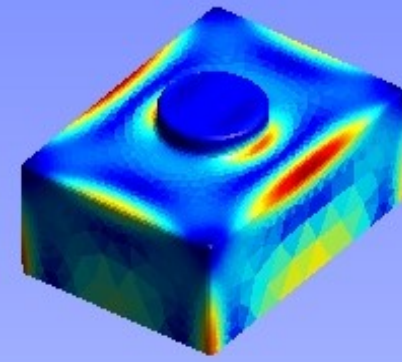
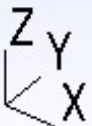


Simulazione numerica con strumenti open

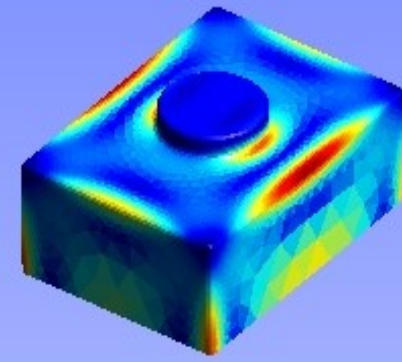


Roberto Lugli

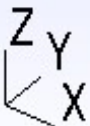
Code-Aster



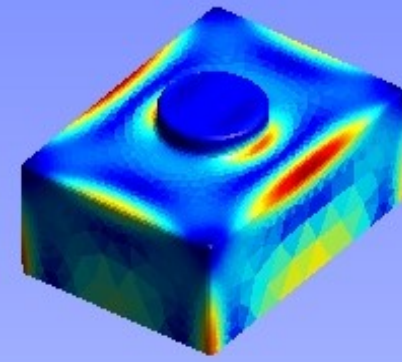
CAE linux



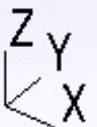
- E' una distribuzione linux dedicata alla simulazione numerica in ambito ingegneristico
- Basata su ubuntu 10.04.3 LTS 64 bit
- Liberamente scaricabile e installabile su hard disk, disponibile anche in live DVD e su chiavetta USB
- Gira su architetture a 64 bit per avere disponibili più di 2 GB di ram
- Disponibili i driver 3D open per la maggior parte delle schede video, scaricabili quelli proprietari



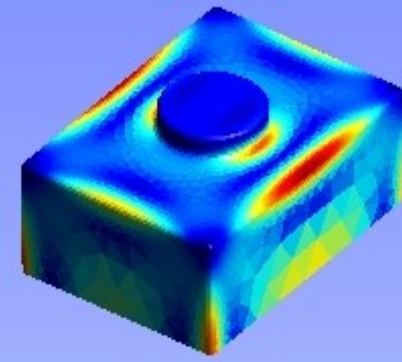
CAE linux



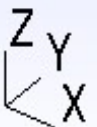
- Basato sui pacchetti open source
- Salome (EDF)
- Code aster (EDF)
- Code saturne CFD (EDF)
- Open foam CFD (ESI group)
- Elmer multiphysics (CSC)



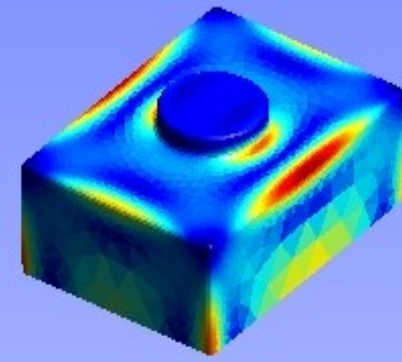
CAE linux



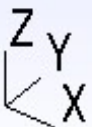
- Strumenti cad base: qcad, freeCAD
- Strumenti matematici: wxmaxima, octave
- Strumenti per scrittura: open office, latex
- Fornito con video tutorials ed esempi



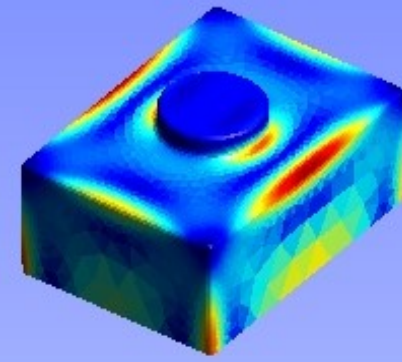
Code Aster la storia



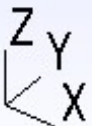
- Sviluppato dal 1989 all'interno di EDF per rispondere ai bisogni interni
- Un unico strumento permette di conservare e scambiare i lavori di ricerca e sviluppo e trasferirli all'ingegneria
- Garantire la validità e la qualità dei risultati indipendentemente dagli editor utilizzati
- Garantire nel tempo la vita del programma vista la durata delle opere realizzate



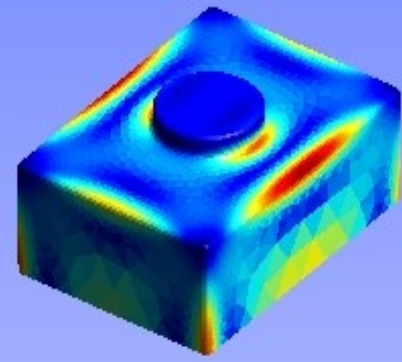
Code Aster la storia



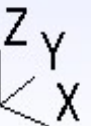
- Aster: Analisi delle Strutture e Termomeccanica per Studi e Ricerche
- Il cuore del programma è scritto in fortran ed i moduli in python
- E' uno strumento per esperti in continua evoluzione
- E' un solutore e necessita di un ambiente pre e post processing (salome)



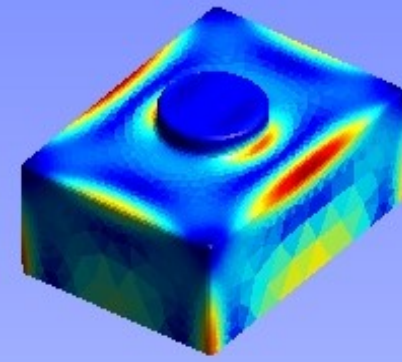
Code Aster open source



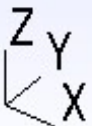
- Disponibile dal 2001 con licenza GNU GPL
- Garantire il futuro del suo strumento
- Oggi sviluppato da una rete di programmatori simile al mondo open source
- Da 8 anni gode dell'apporto della comunità
- Partenariati accademici e con istituti privati
- Una nuova versione ogni due anni
- Di recente integrato nel modulo salome-meca



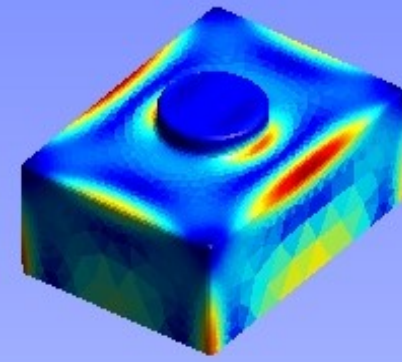
Code Aster caratteristiche



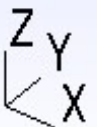
- Oltre alla termo-meccanica classica Aster offre una gamma di modelli ed analisi multifisiche
- Operatori non lineari consentono studi statici e dinamici, impliciti o espliciti
- Non linearità geometriche (contatto) e comportamento non lineare dei materiali (elastoplasticità)



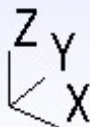
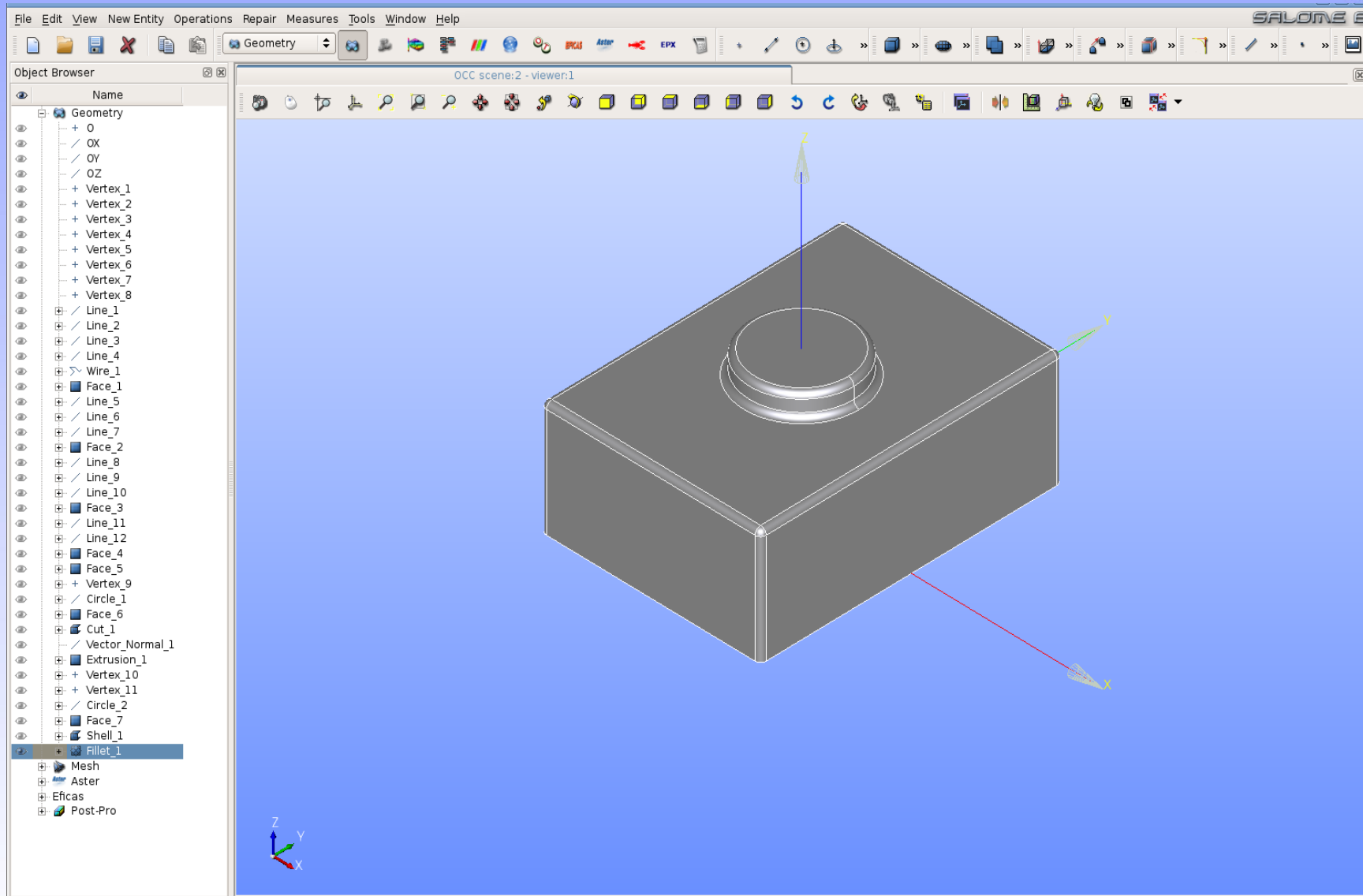
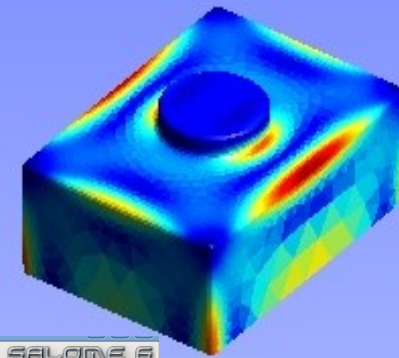
Code aster come funziona



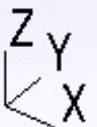
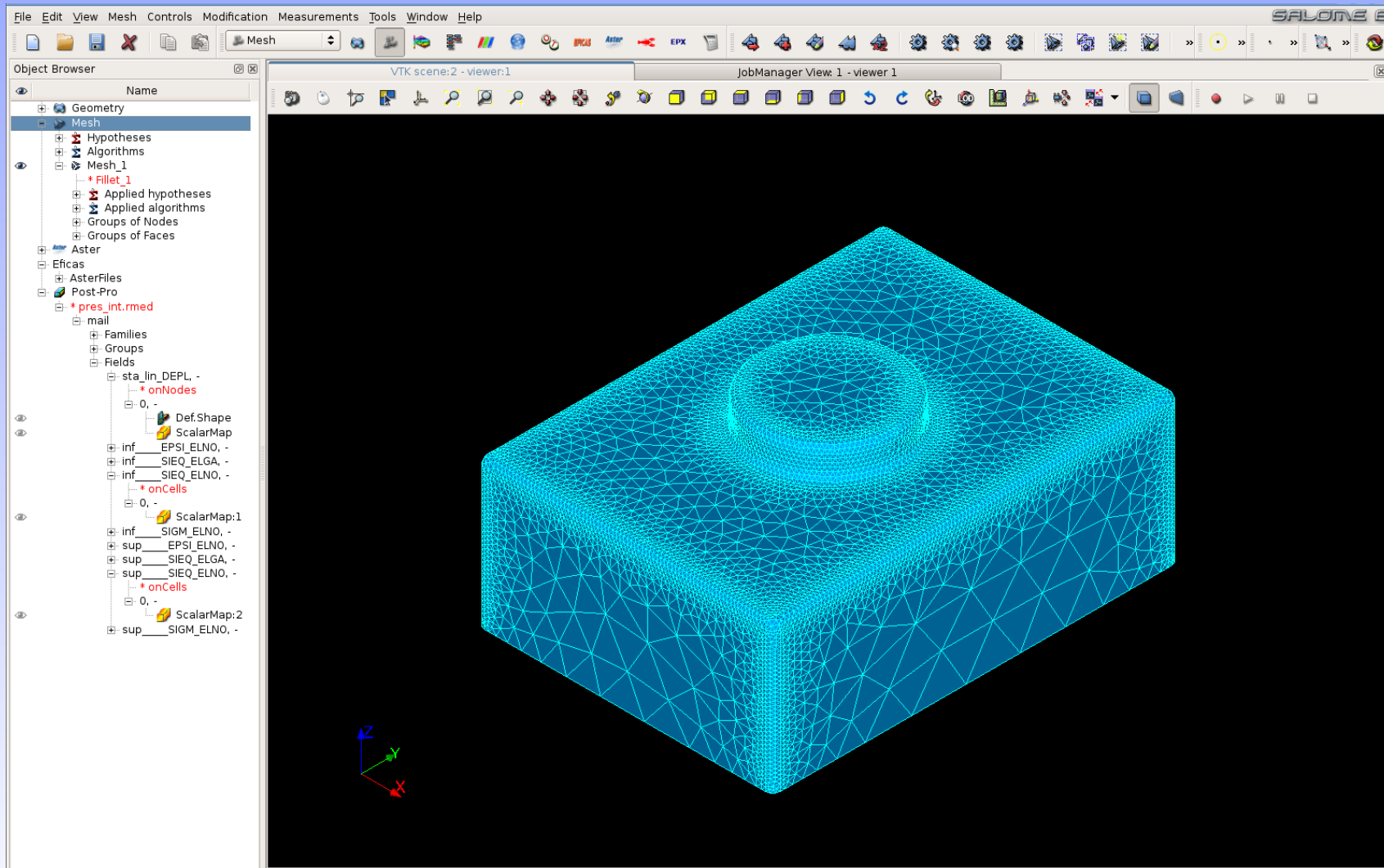
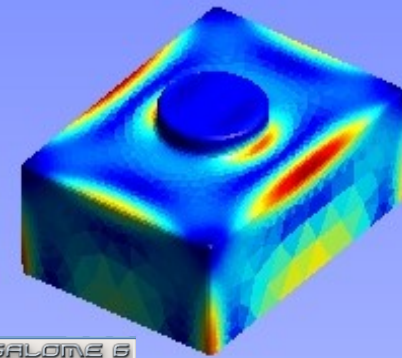
- Dispone di vari moduli
- Geometria
- Mesh
- Aster con gli strumenti ASTK per gestire gli studi ed EFICAS per scrivere i files di comando
- Post processor per visualizzare i risultati



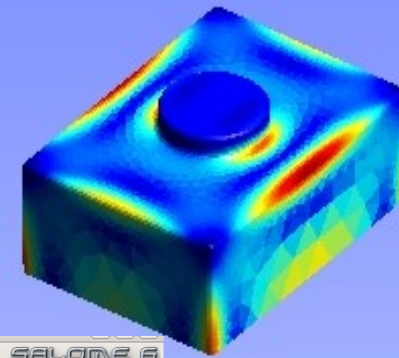
Modulo geometria



Modulo mesh



Modulo aster-eficas

The screenshot displays the Eficac software interface, which is used for managing Aster simulation commands. The main window is titled 'Eficac QT4 V6.6' and contains a command editor for a file named 'pres_int_unif.comm'.

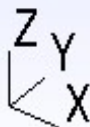
The interface includes a menu bar (Fichier, Edition, JdC, Aide, Traduction, Options) and a toolbar with various icons. On the left, there is an 'Object Browser' showing a tree structure of the simulation setup, including 'Geometry', 'Mesh', 'Hypotheses', 'Algorithms', and 'Aster'.

The central command editor is divided into two main sections:

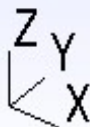
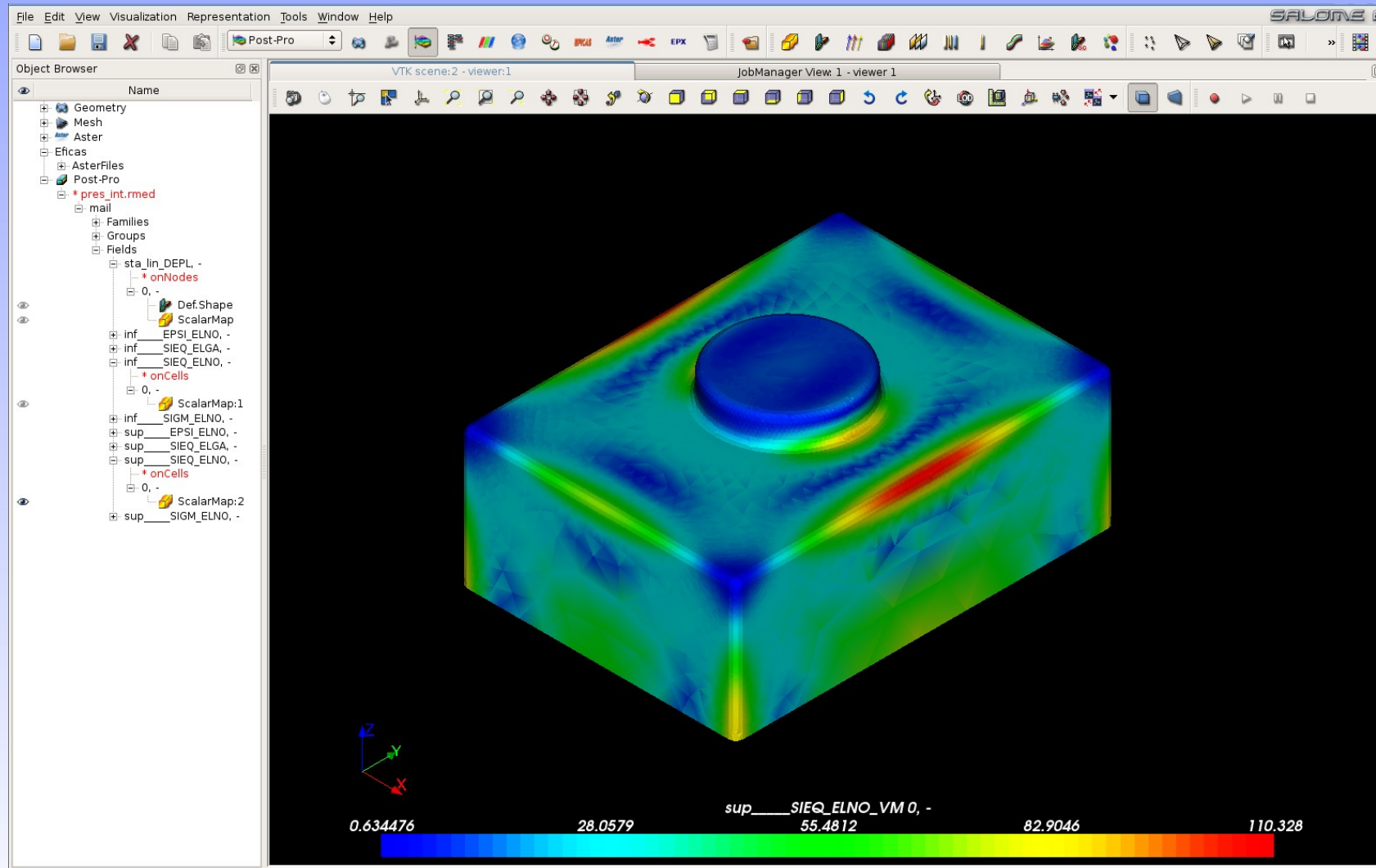
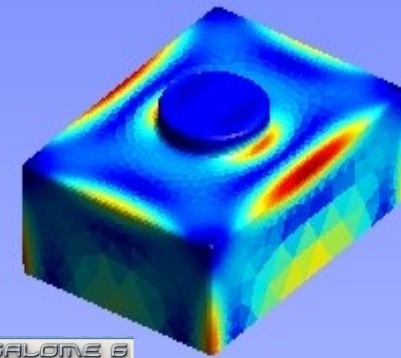
- Commande:** A table listing simulation commands and their associated concepts or values.
- Commandes :** A list of available commands, with a filter input and a 'Suivant' button.
- Règles :** A section for defining rules, such as 'AU_MOINS_UN : DEBUT POURSUITE' and 'Règle A_CLASSER : D'abord : DEBUT ou POURSUITE, Ensuite : FIN'.

A 'Valider' button is located at the bottom right of the command editor.

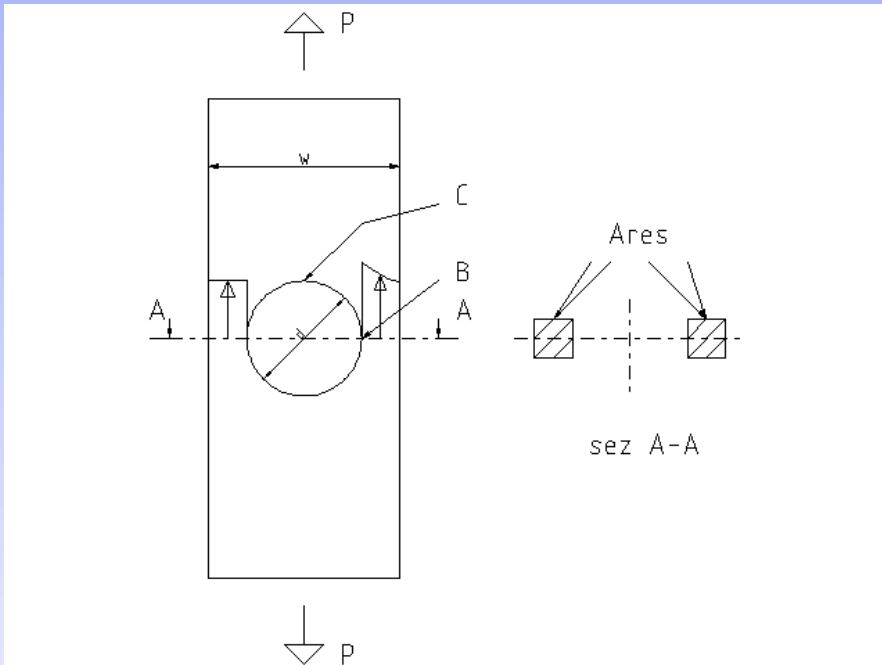
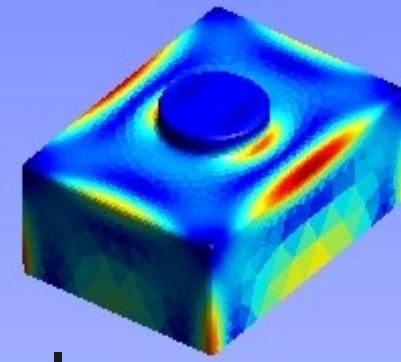
Commande	Concept/Valeur
DEBUT :	mail
LIRE_MALLAGE :	mail
MODI_MALLAGE :	PA66_120
DEFI_MATERIAU :	mode1
AFFE_MODELE :	mater
AFFE_MATERIAU :	shell
AFFE_CARA_ELEM :	vinco
AFFE_CHAR_MECA :	pres22
MECA_STATIQUE :	sta_lin
CALC_CHAMP :	csta_lin
POST_CHAMP :	sup
POST_CHAMP :	inf
IMPR_RESU :	
IMPR_RESU :	
FIN :	



Modulo post-processing



Esempio: lastra forata



- Tensione nominale

$$\sigma_n = \frac{P}{A_{res}} = \frac{P}{(W-d)} = \frac{P/W}{(W-d)/W} = \frac{P/W}{1-d/W}$$

$$\alpha_k = 2 + (1 - d/W)^3$$

- Tensione teorica

$$\sigma_t = \alpha_k \cdot \sigma_n$$

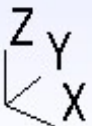
- Lastra $W=100$, foro $d=60$, $P=10$

$$d/W = \frac{60}{100} = 0,6$$

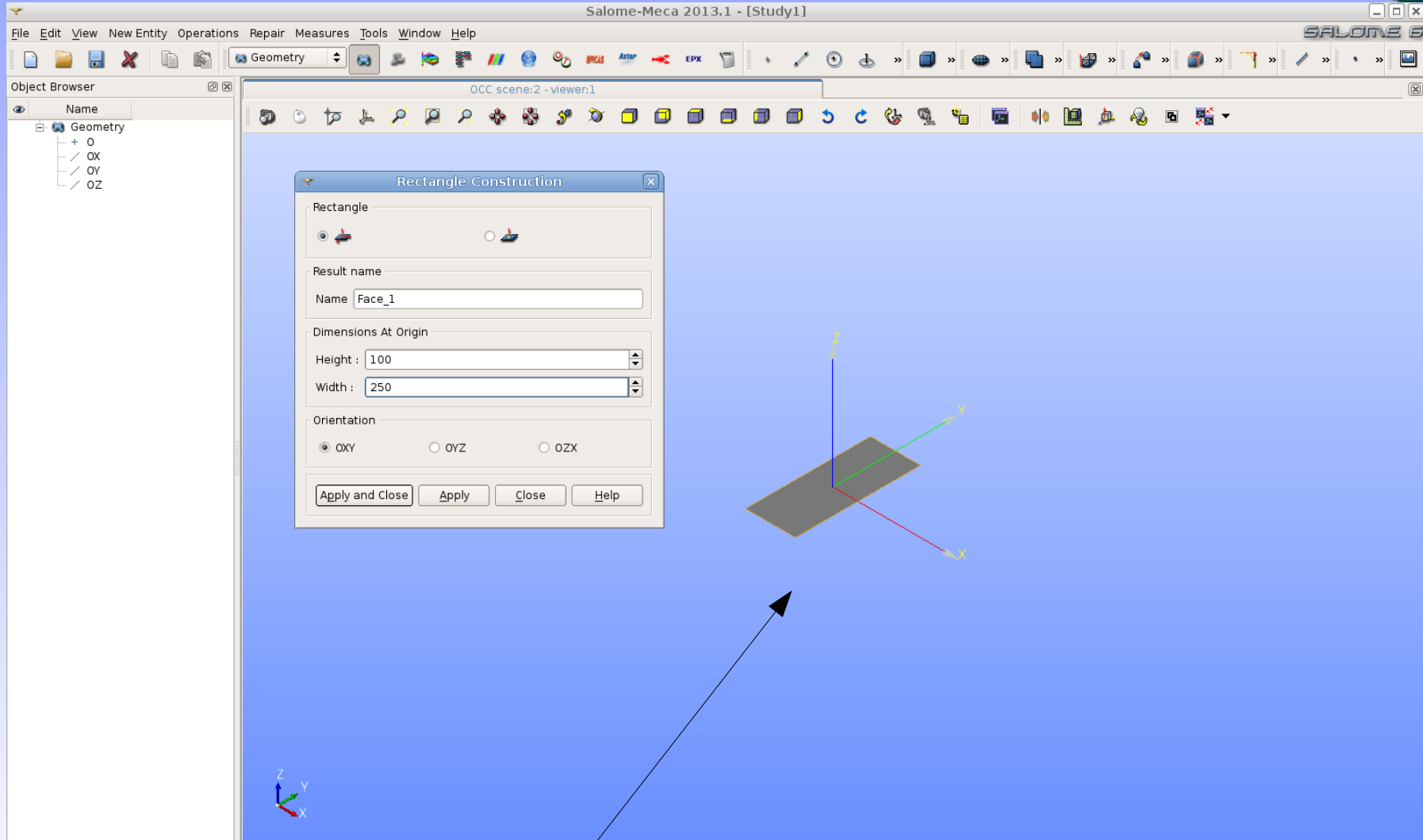
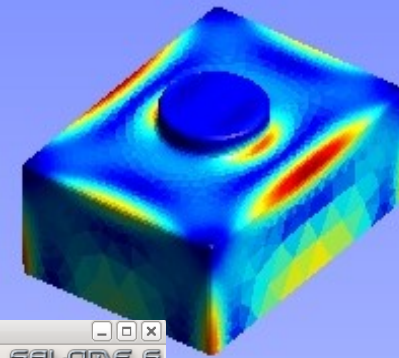
$$\sigma_n = \frac{10}{1-0,6} = \frac{10}{0,4} = \frac{100}{4} = 25 [MPa]$$

$$\alpha_k = 2 + (1 - 0,6)^3 = 2,064 \approx 2,1$$

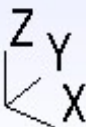
$$\sigma_t = 2,1 \cdot 25 = 52,5 [MPa]$$



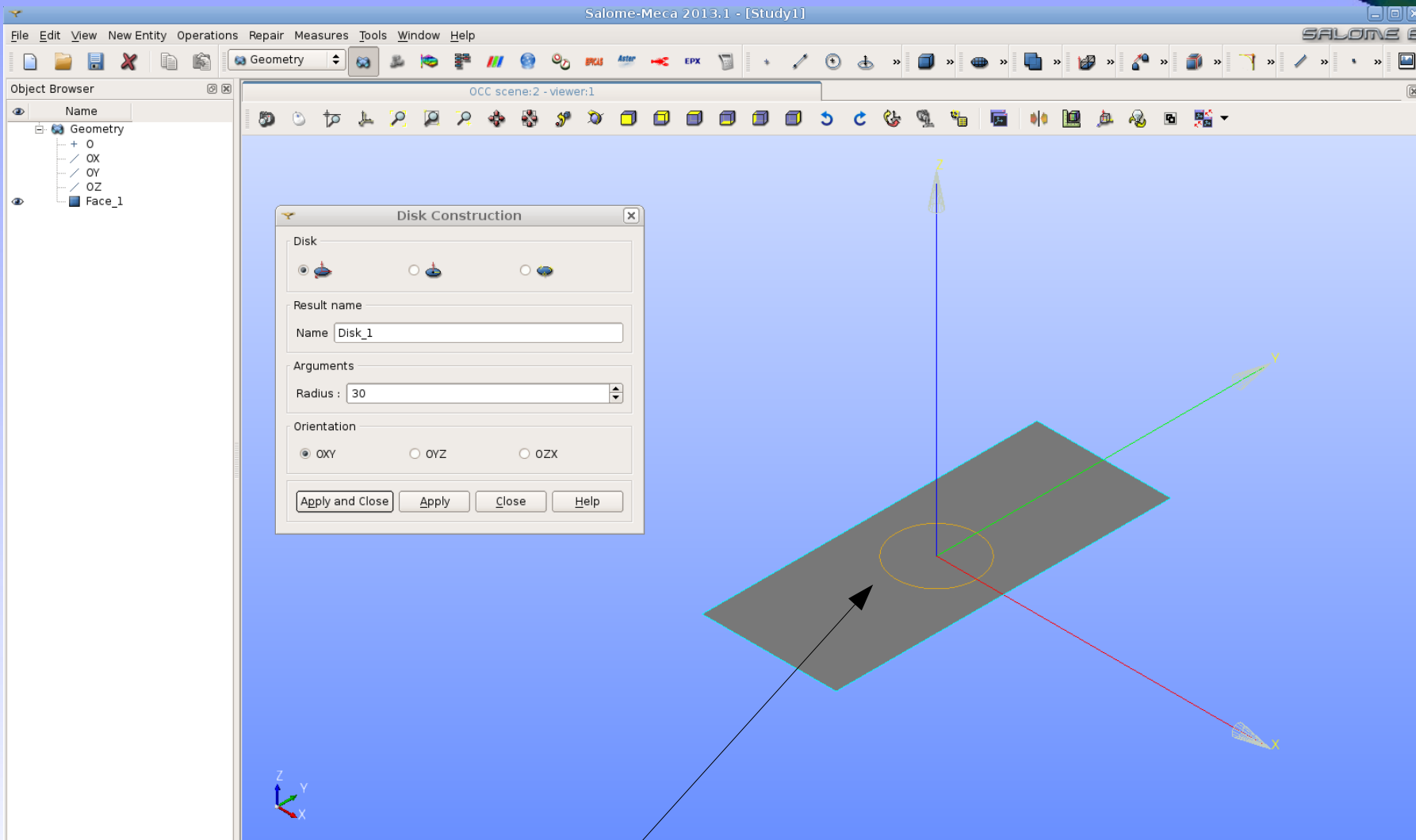
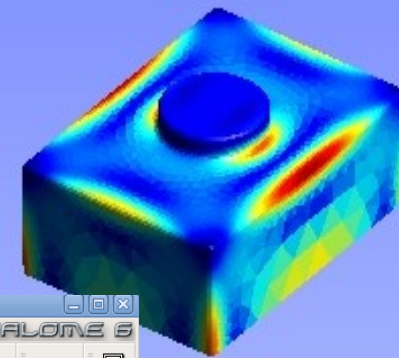
lastra forata: geometria



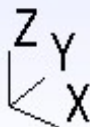
Si costruisce un rettangolo centrato nell'origine



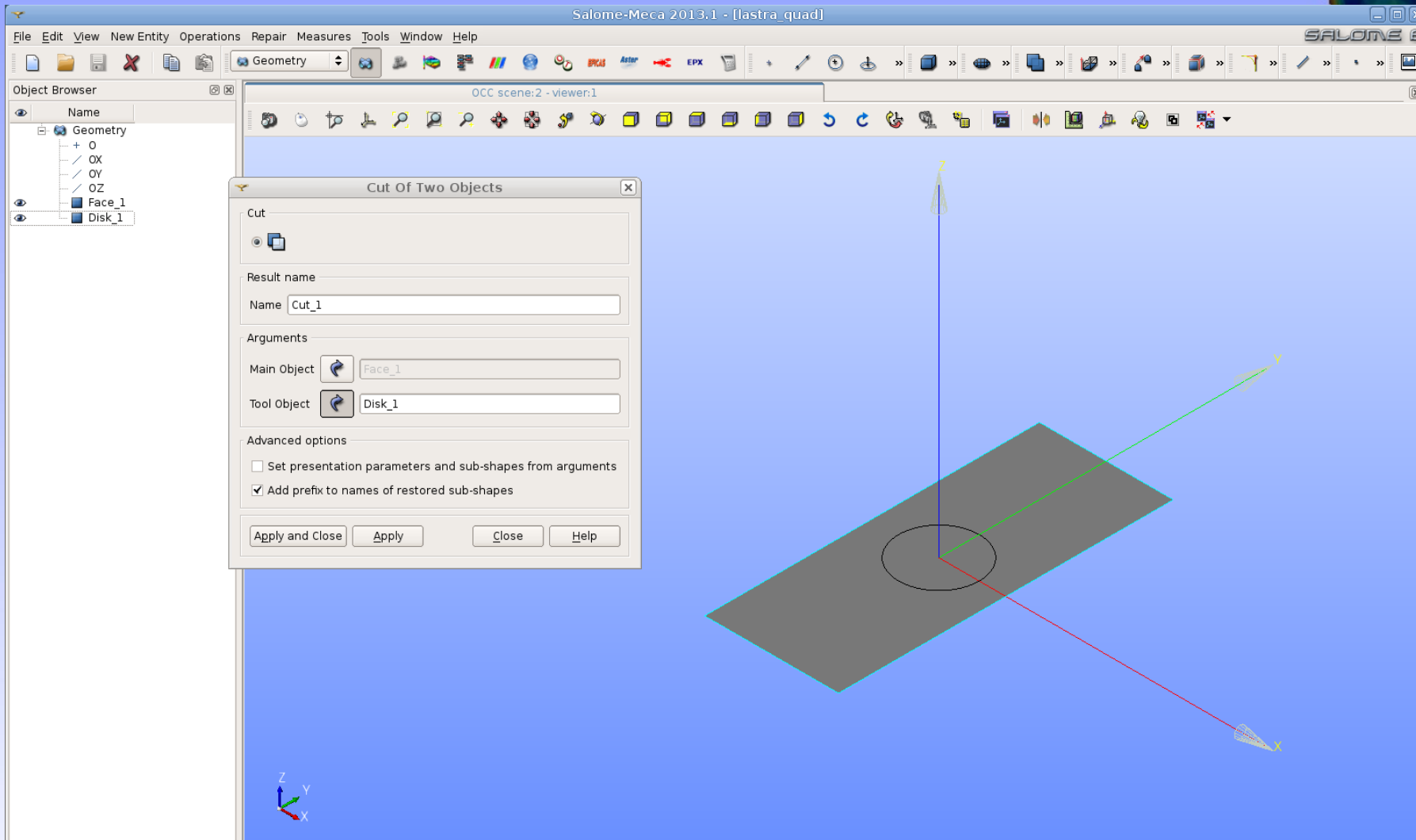
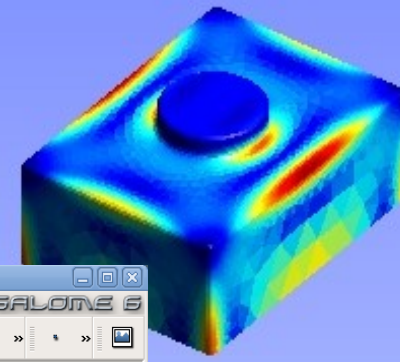
lastra forata: geometria



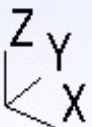
Si costruisce un disco centrato nell'origine



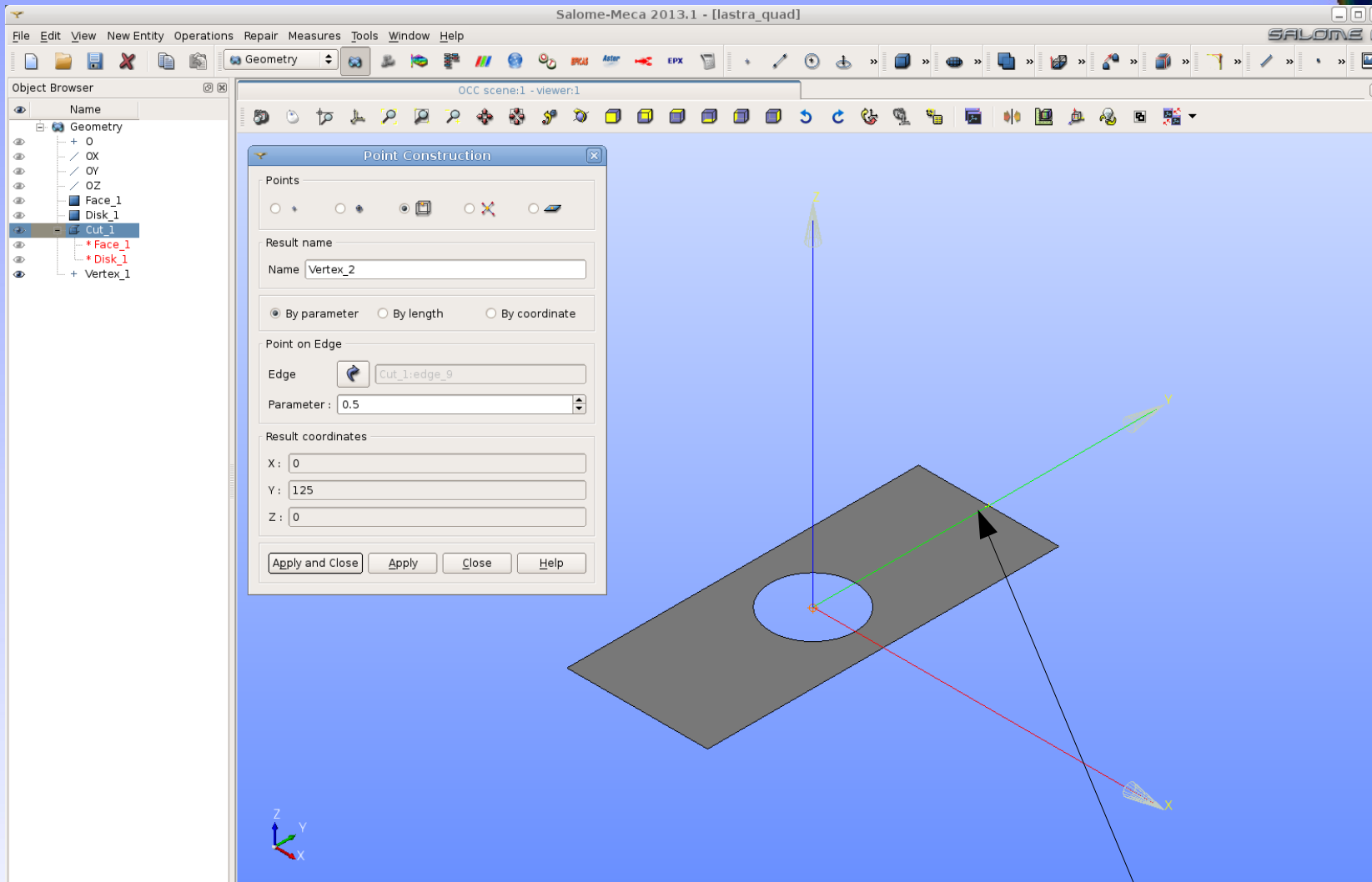
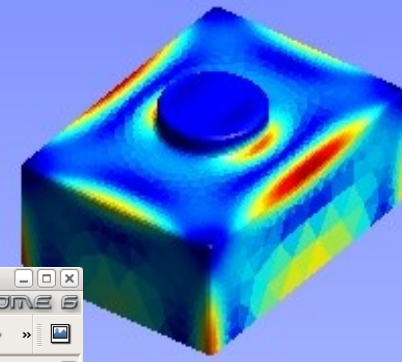
lastra forata: geometria



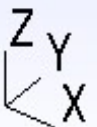
Si taglia il disco dal rettangolo con una sottrazione booleana



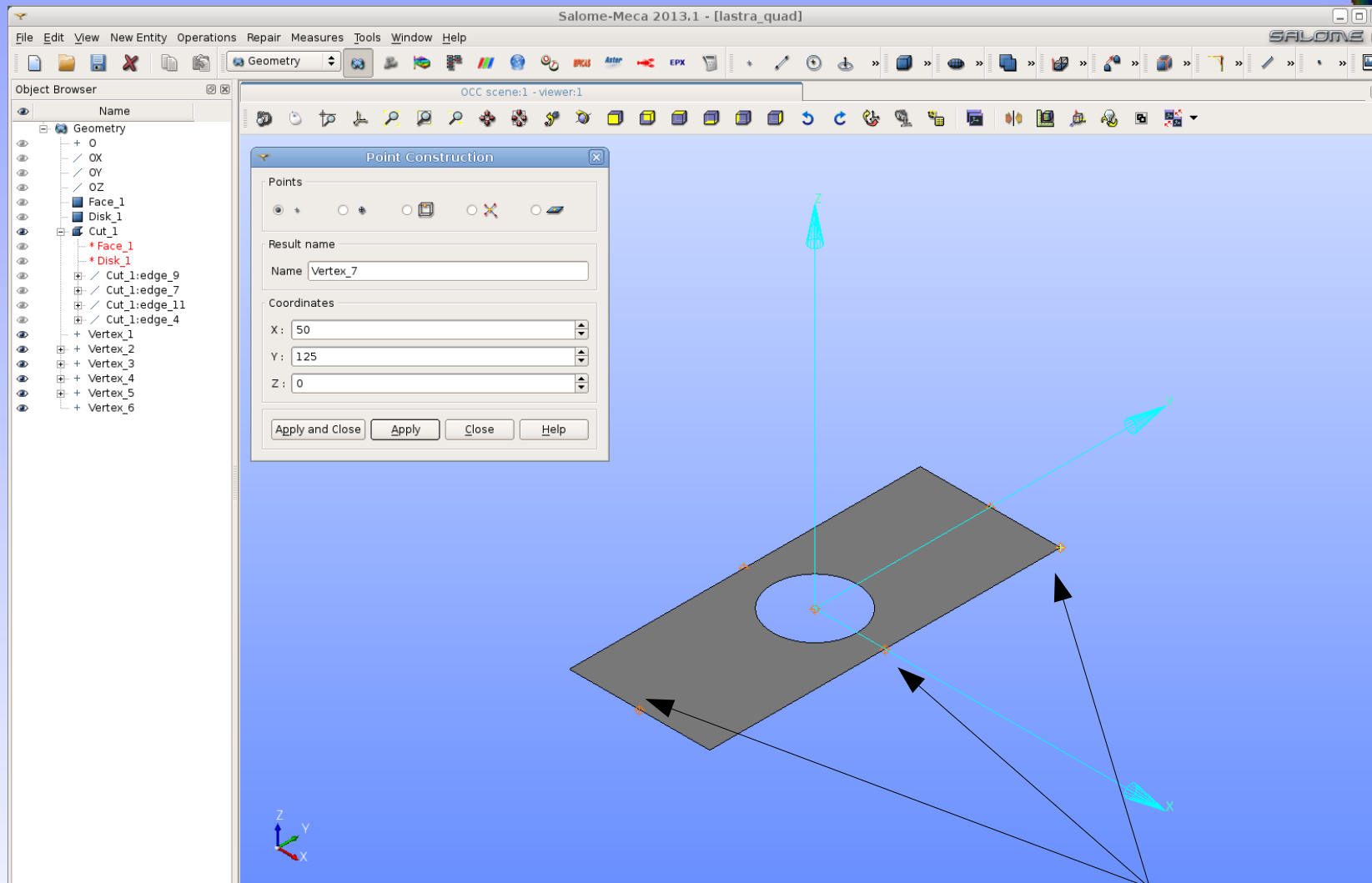
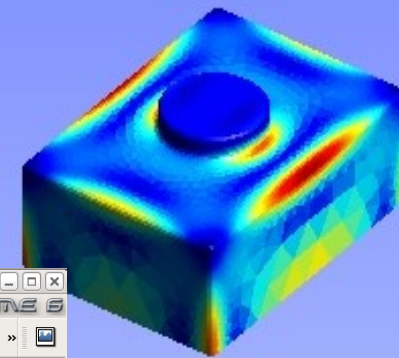
lastra forata: geometria



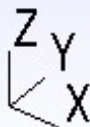
Si crea un vertice che sarà l'estremità di una linea



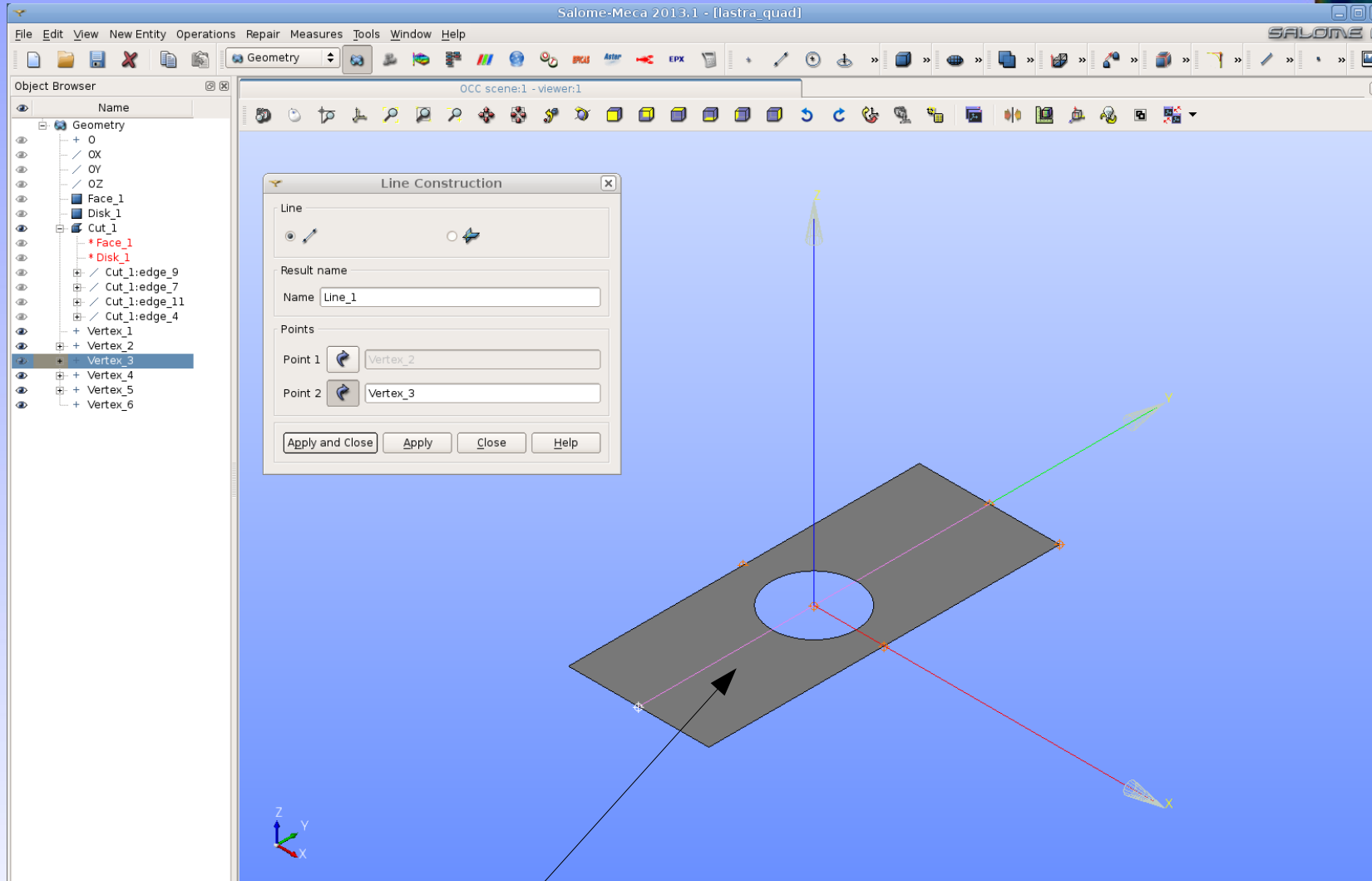
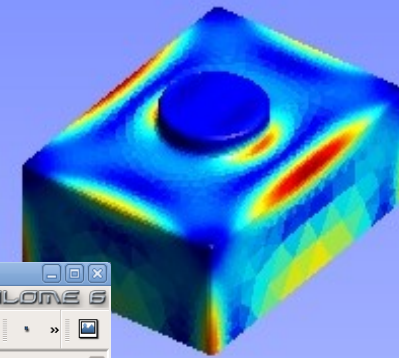
lastra forata: geometria



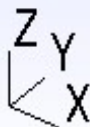
Si creano i vertici a metà di ogni bordo e agli angoli



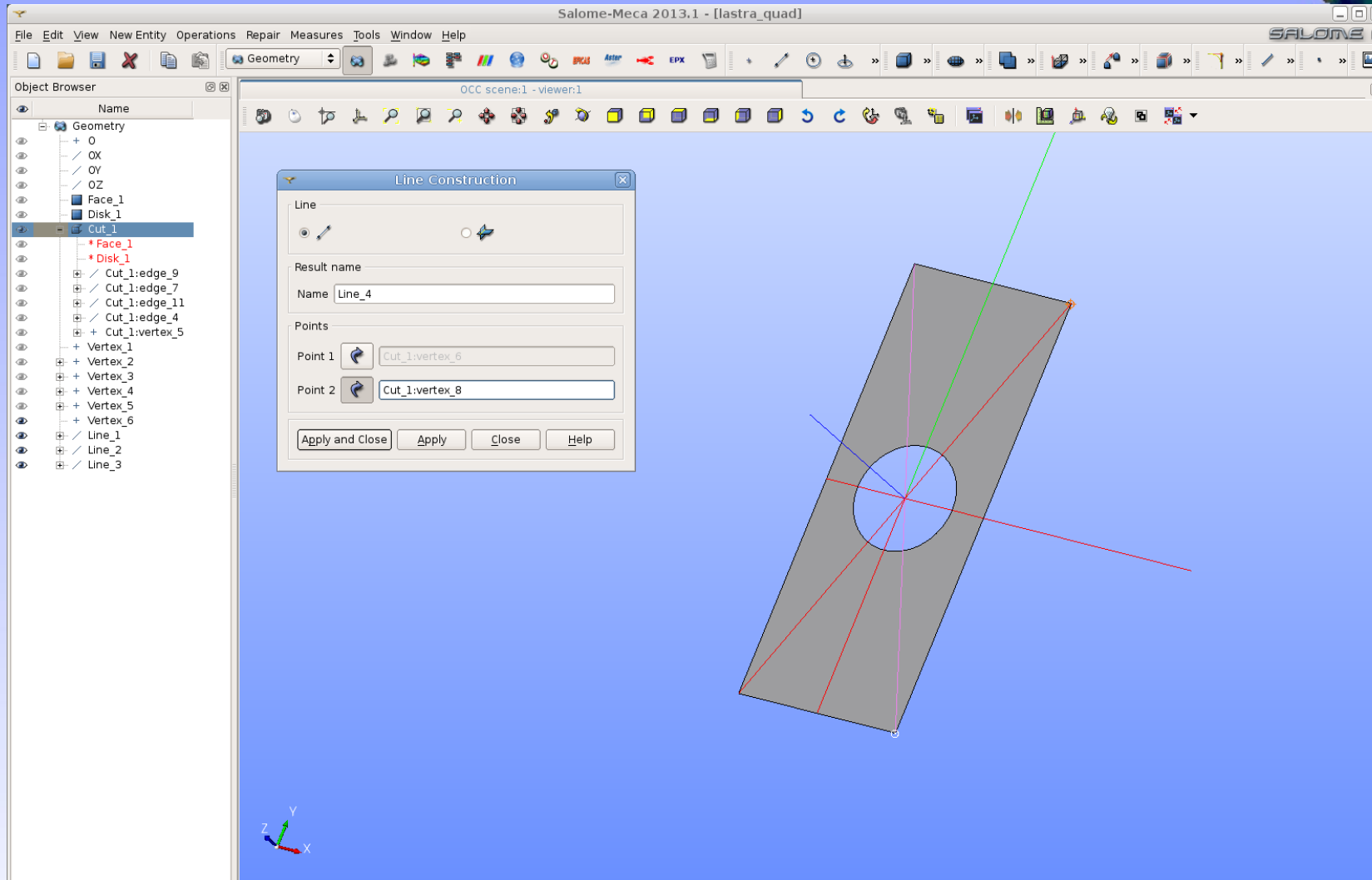
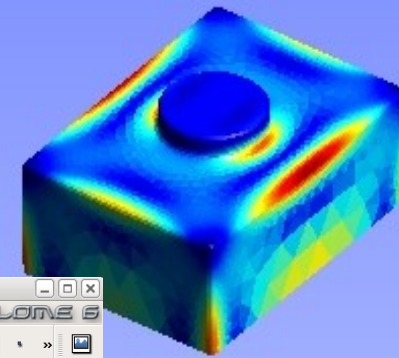
lastra forata: geometria



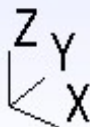
Si crea una linea



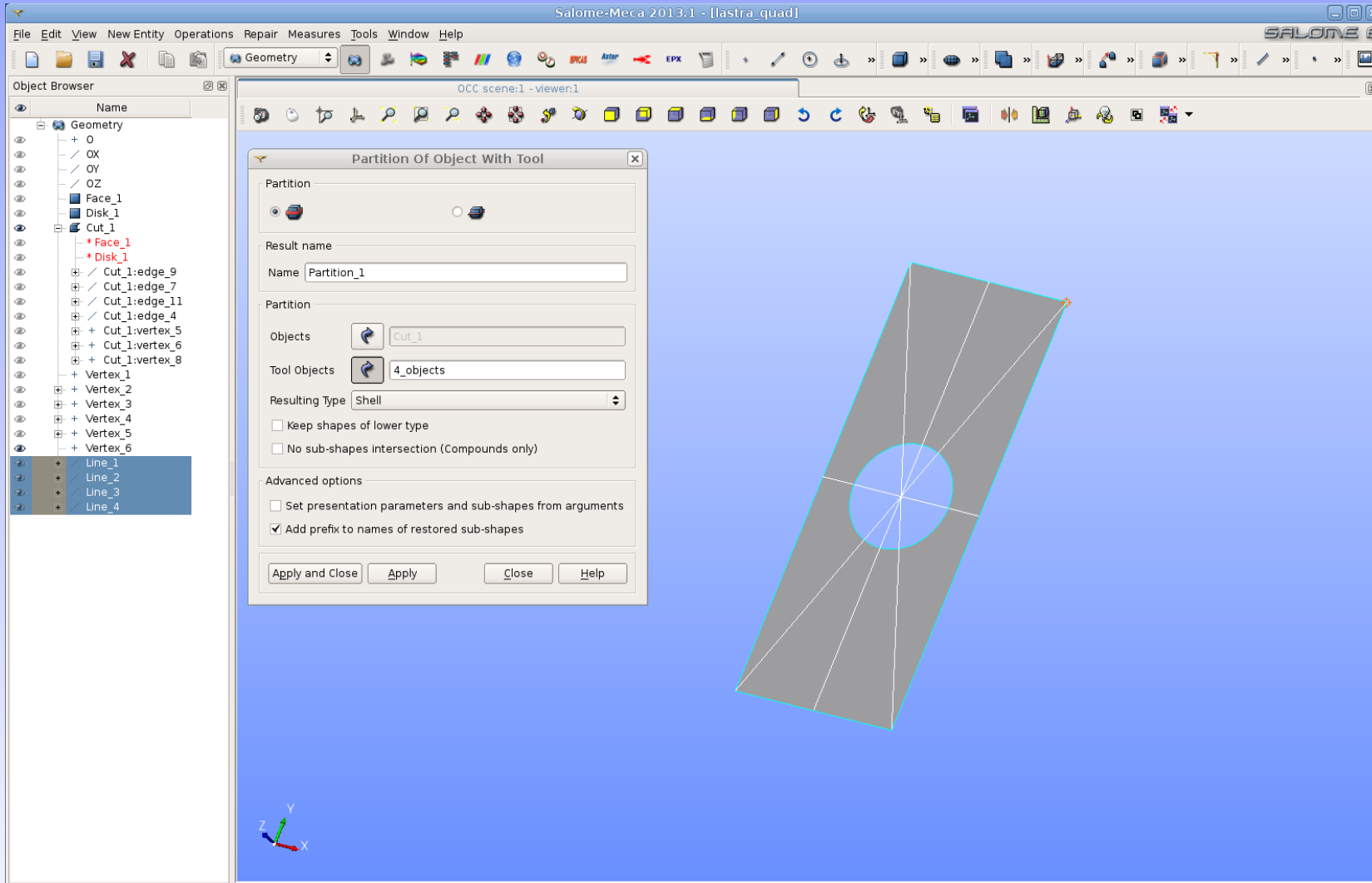
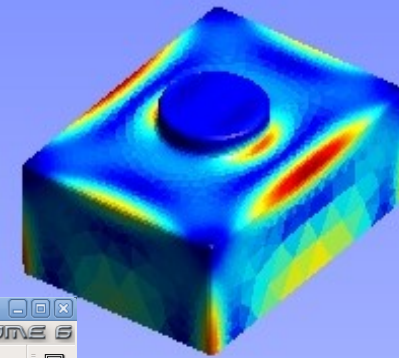
lastra forata: geometria



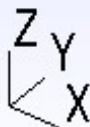
Si creano le mediane della lastra per dividerla in aree a 4 lati.
Esse permettono di ottenere elementi planari a 4 nodi.



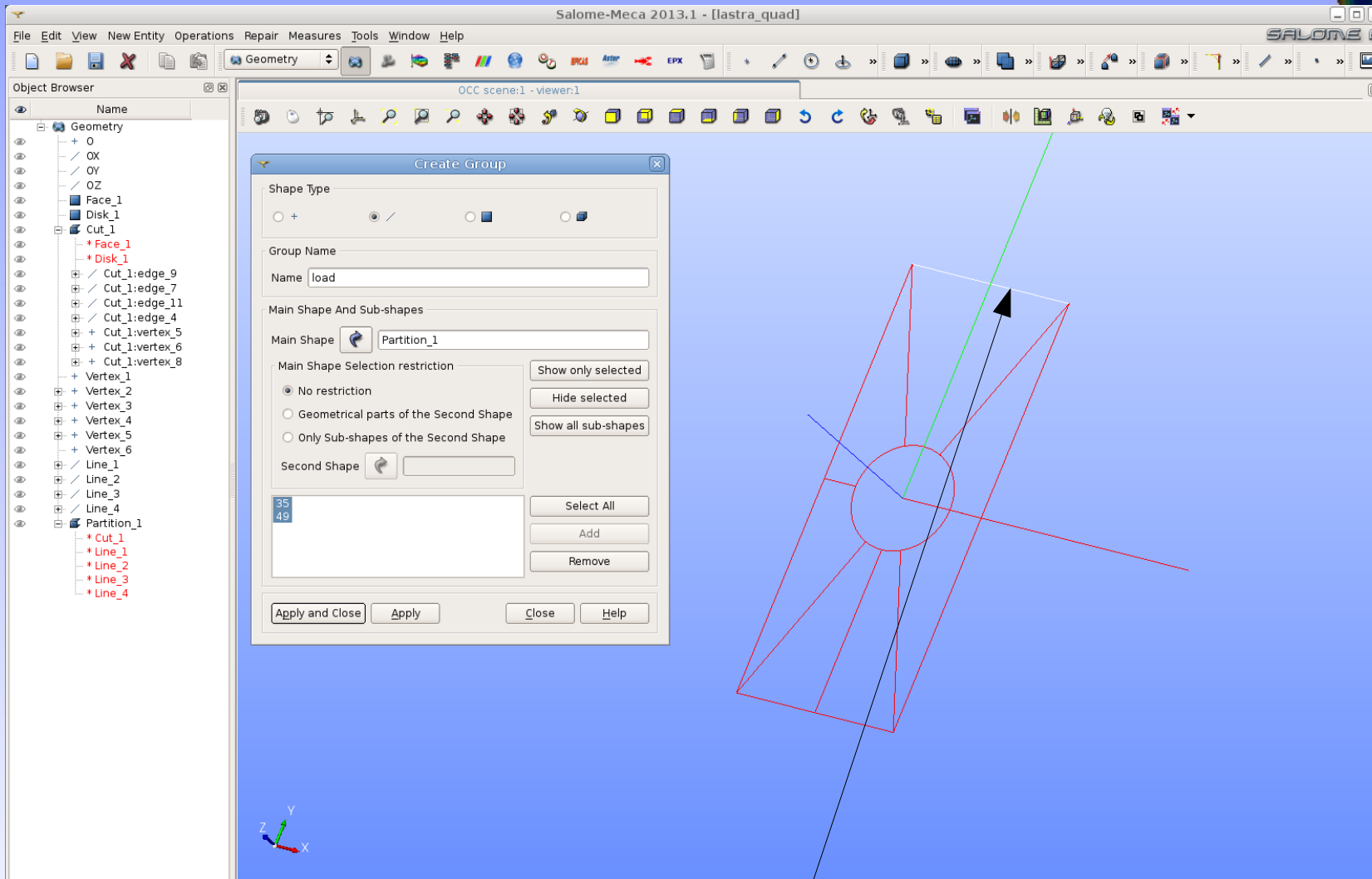
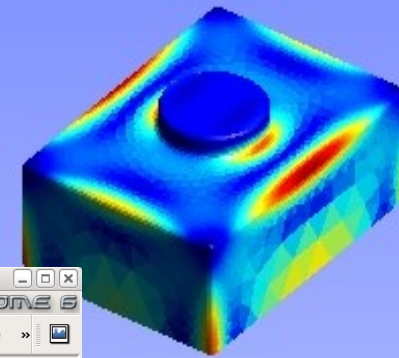
lastra forata: geometria



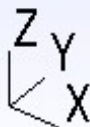
Si divide la lastra lungo le mediane con lo strumento di partizione



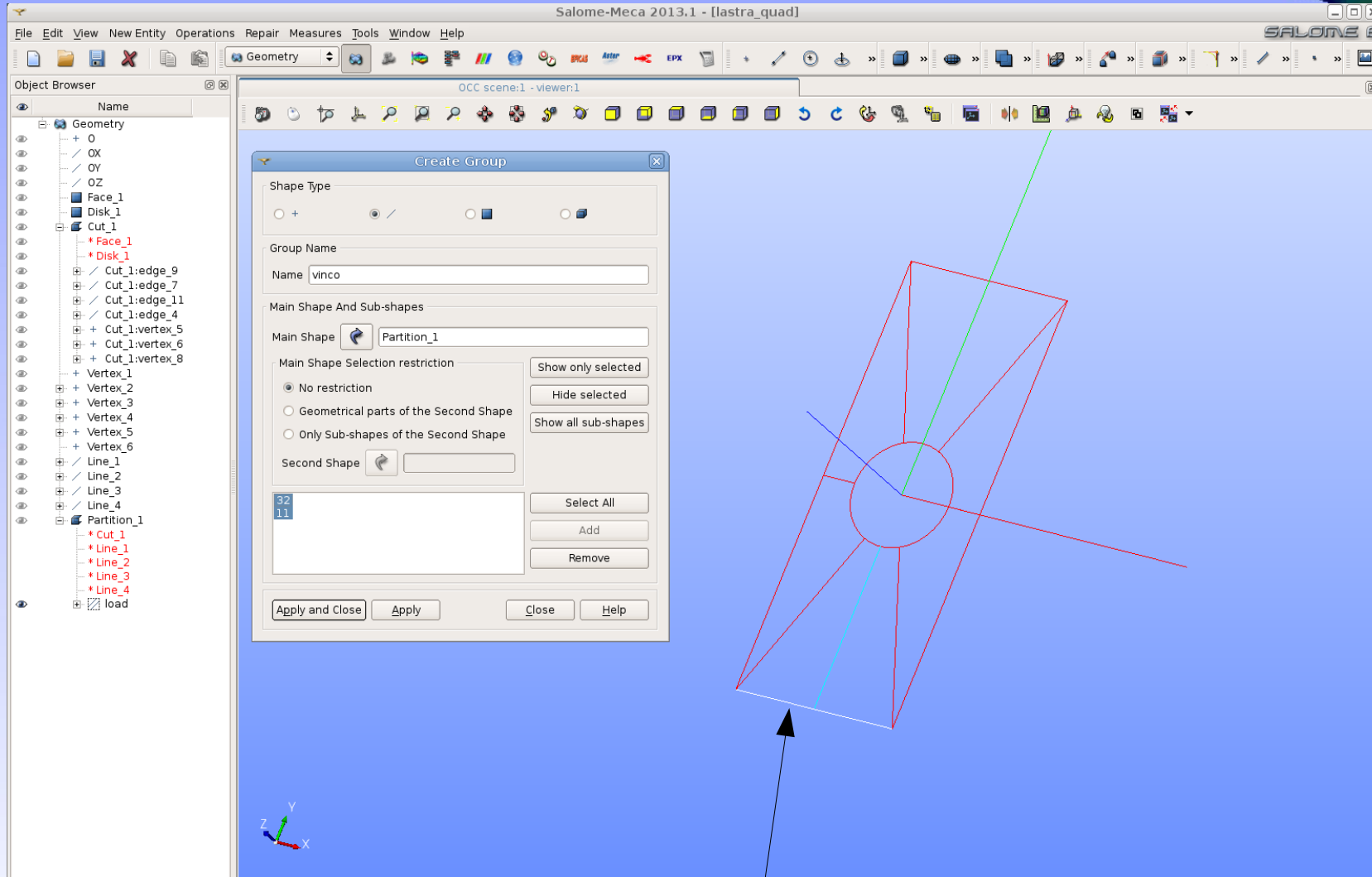
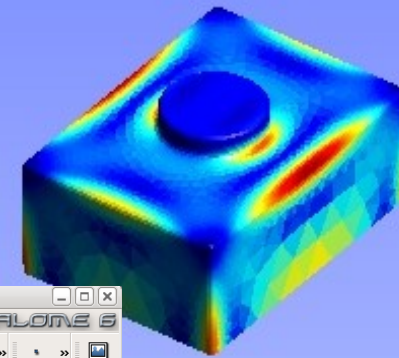
lastra forata: geometria



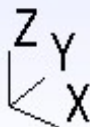
Si crea il gruppo di linee su cui applicheremo il carico



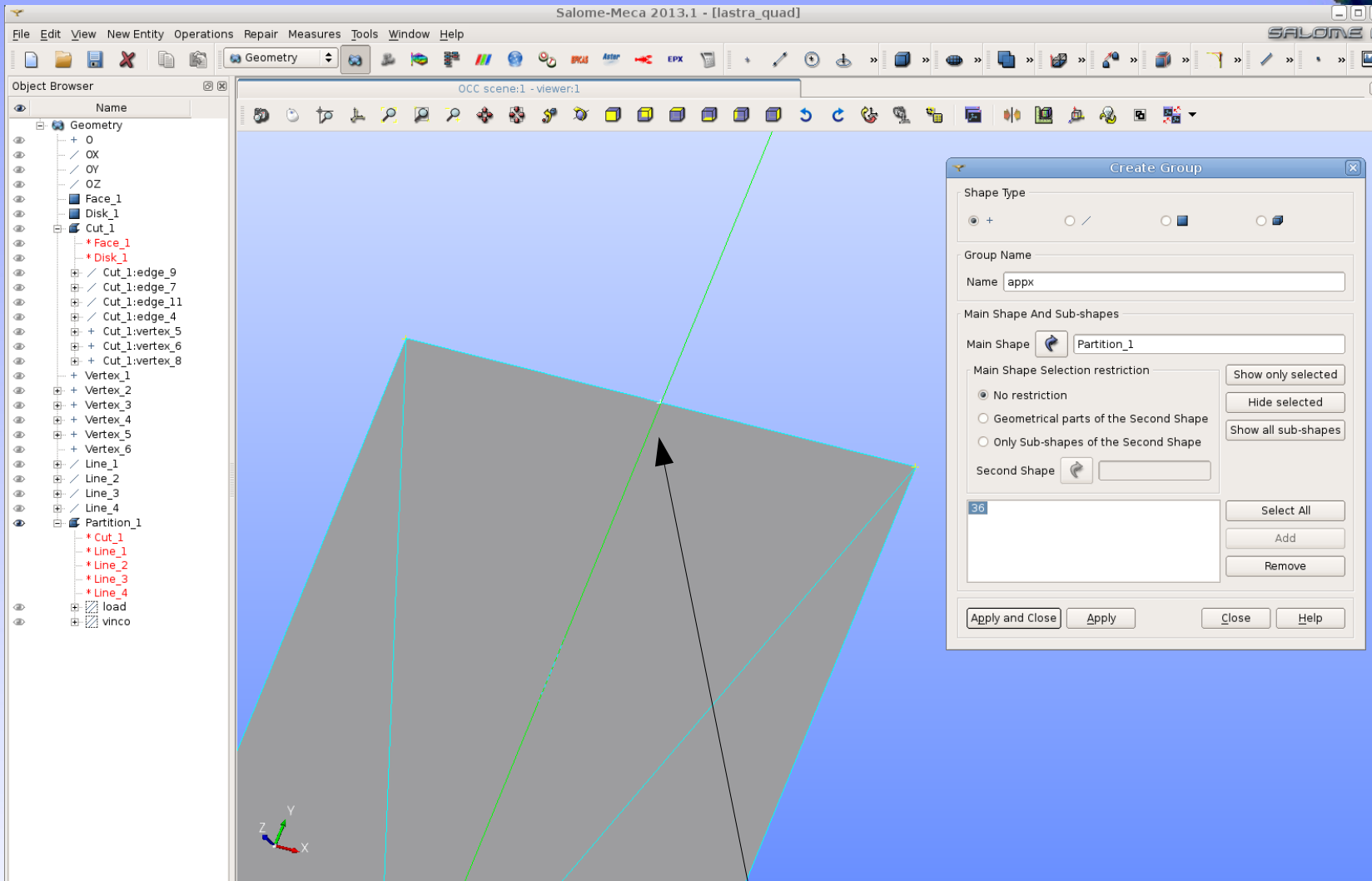
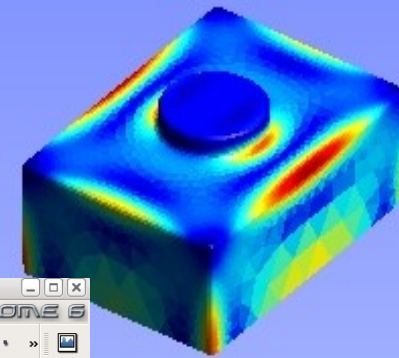
lastra forata: geometria



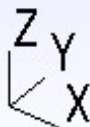
Si crea il gruppo di linee su cui applicheremo il vincolo in y



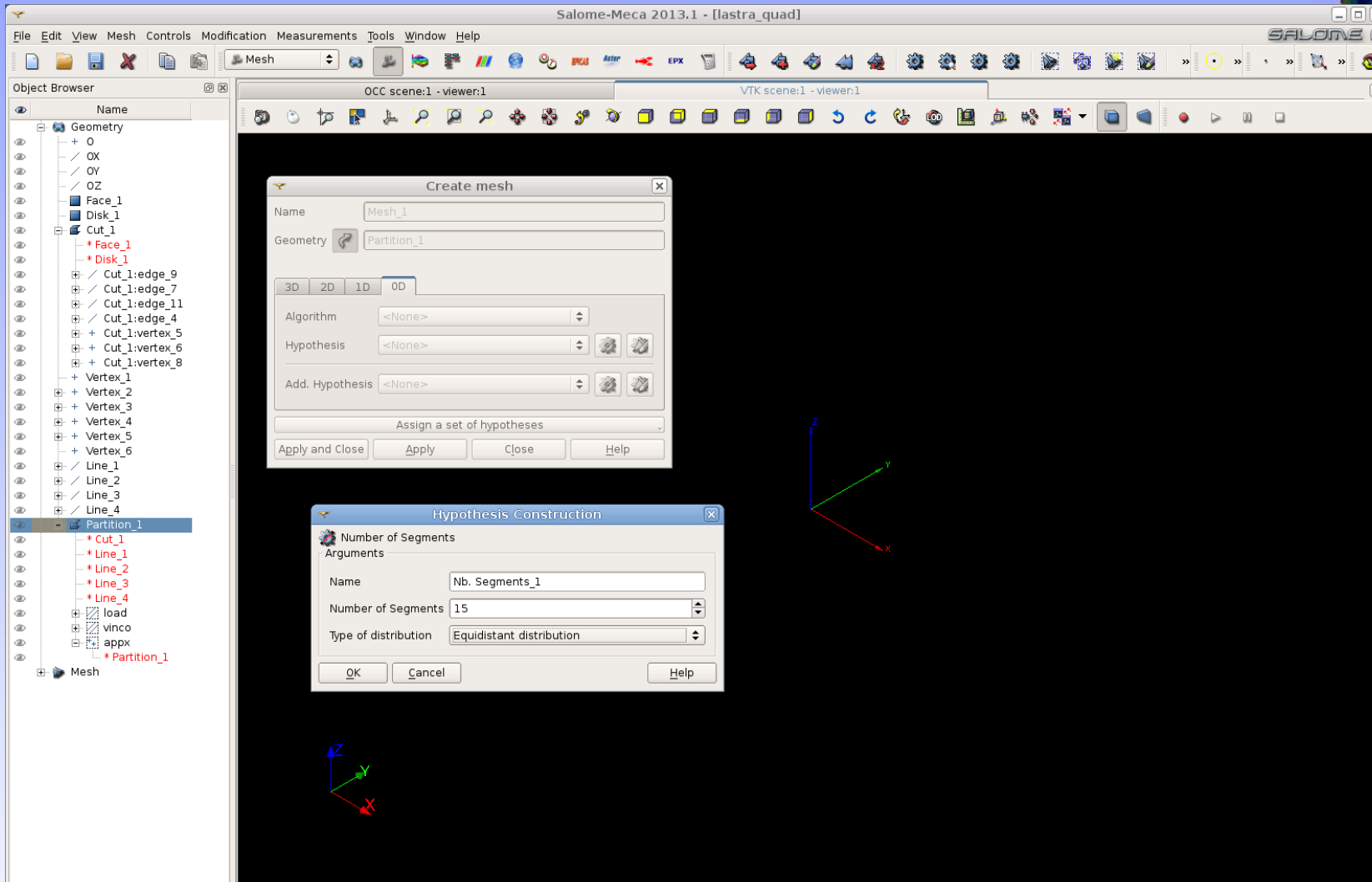
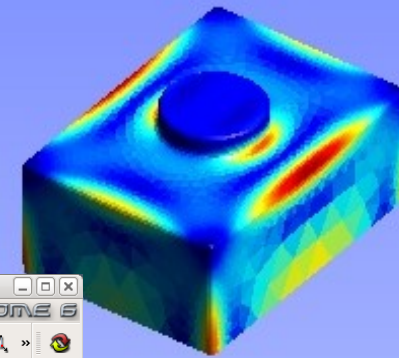
lastra forata: geometria



Si crea il gruppo costituito da un punto sull'asse di mezzeria per impedire spostamenti in x

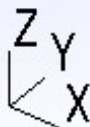


lastra forata: mesh

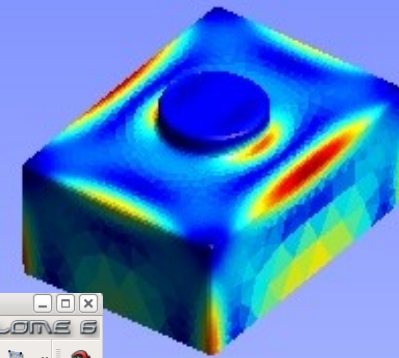


Impostiamo i parametri di mesh

- 1D dividiamo i bordi delle aree in 15 segmenti di lunghezza uguale
- 2D scegliamo una mesh a quadrilateri (mappata)



lastra forata: mesh



Salome-Meca 2013.1 - [lastra_quad]

Object Browser

- Geometry
 - 0
 - OX
 - OY
 - OZ
 - Face_1
 - Disk_1
 - Cut_1
 - *Face_1
 - *Disk_1
 - Cut_1:edge_9
 - Cut_1:edge_7
 - Cut_1:edge_11
 - Cut_1:edge_4
 - Cut_1:vertex_5
 - Cut_1:vertex_6
 - Cut_1:vertex_8
 - Vertex_1
 - Vertex_2
 - Vertex_3
 - Vertex_4
 - Vertex_5
 - Vertex_6
 - Line_1
 - Line_2
 - Line_3
 - Line_4
 - Partition_1
 - *Cut_1
 - *Line_1
 - *Line_2
 - *Line_3
 - *Line_4
 - load
 - vinco
 - appx
 - *Partition_1
- Mesh
 - Hypotheses
 - Algorithms
 - Mesh_1
 - *Partition_1
 - Applied hypotheses
 - Applied algorithms

Mesh computation succeed

Compute mesh

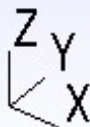
Mesh_1

Mesh Infos

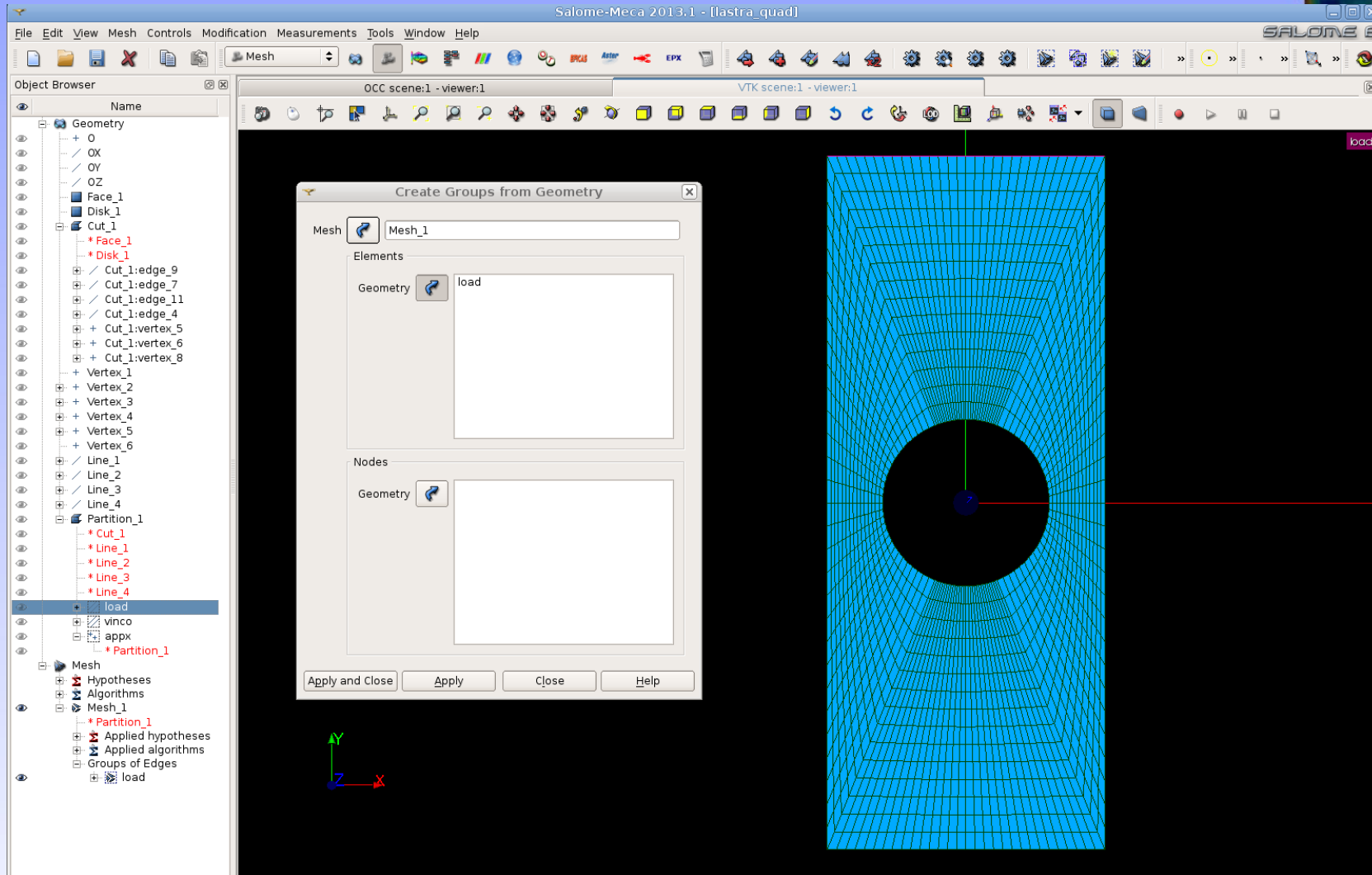
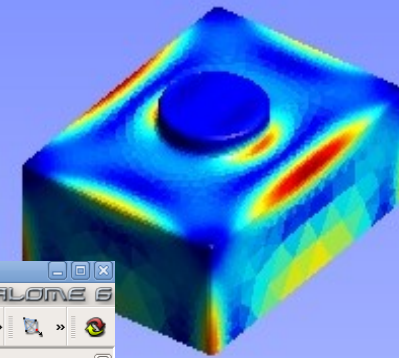
	Total	Linear	Quadratic
Nodes :	1920		
OD Elements :	0		
Balls :	0		
Edges :	360	360	0
Faces :	1800	1800	0
Triangles :	0	0	0
Quadrangles :	1800	1800	0
Polygons :	0		
Volumes :	0	0	0
Tetrahedrons :	0	0	0
Hexahedrons :	0	0	0
Pyramids :	0	0	0
Prisms :	0	0	0
Hexagonal prisms :	0		
Polyhedrons :	0		

Close

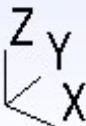
il comando "compute" calcola la mesh e si ottiene il riepilogo degli elementi creati



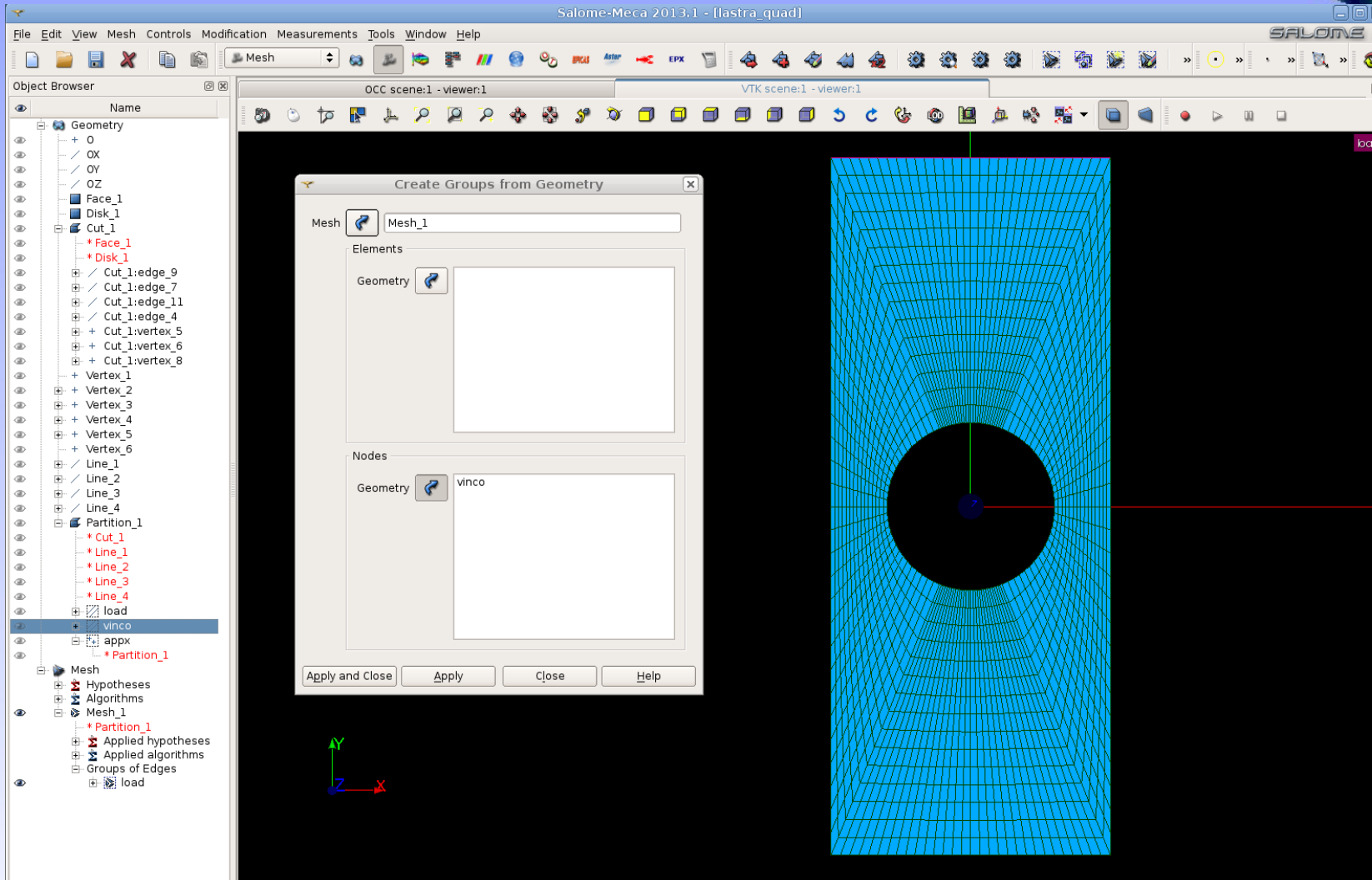
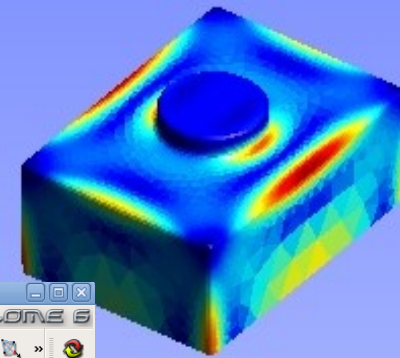
lastra forata: mesh



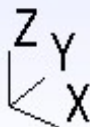
Dai gruppi della geometria si creano direttamente i gruppi di elementi della mesh.
Si può scegliere se creare gruppi di elementi o gruppi di nodi.
Il carico distribuito si applica ad elementi monodimensionali senza rigidezza.



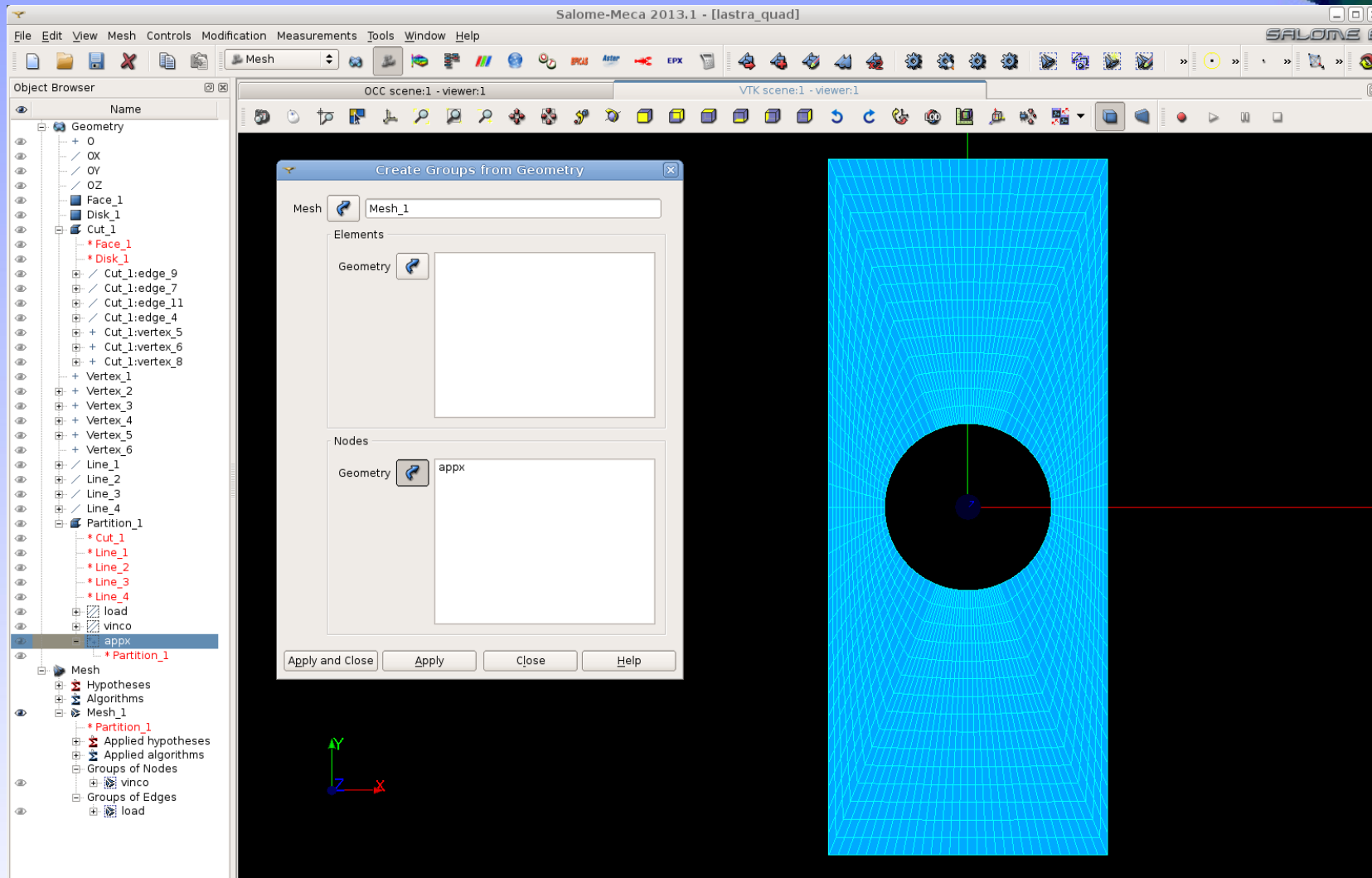
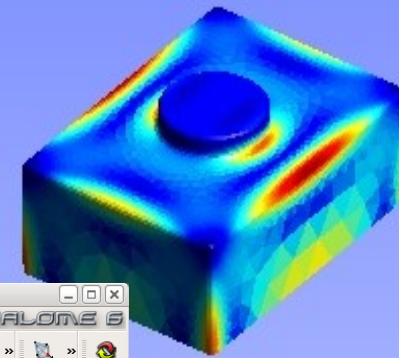
lastra forata: mesh



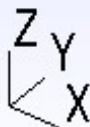
Si crea il gruppo dei nodi sul bordo inferiore della lastra: saranno vincolati in direzione y



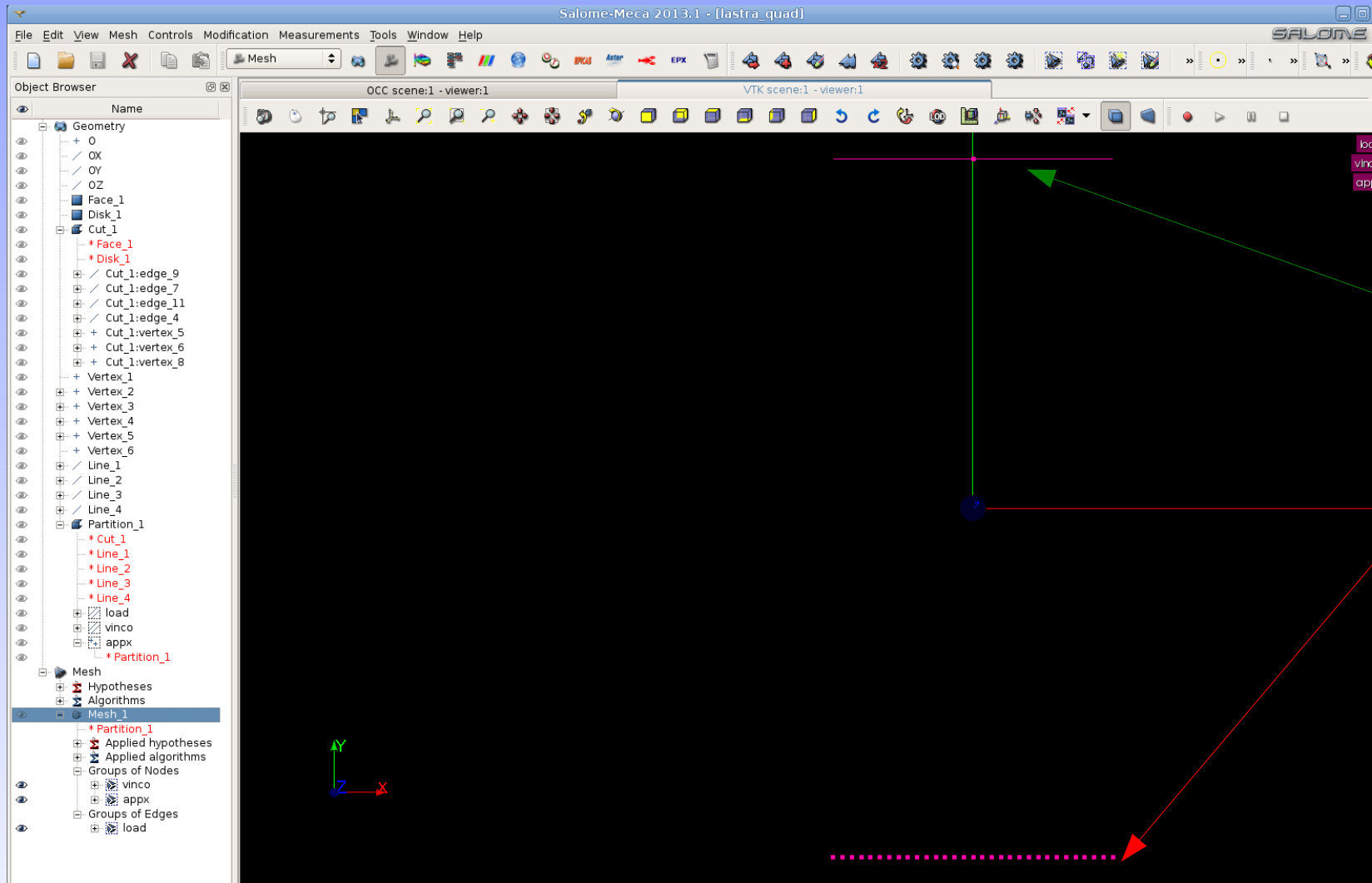
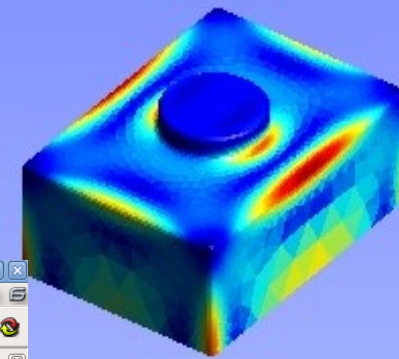
lastra forata: mesh



Si crea il gruppo della mesh costituito da un solo nodo, partendo dal gruppo della geometria contenente un solo punto: sarà vincolato in direzione x



lastra forata: mesh

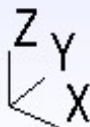


Elementi
monodim.

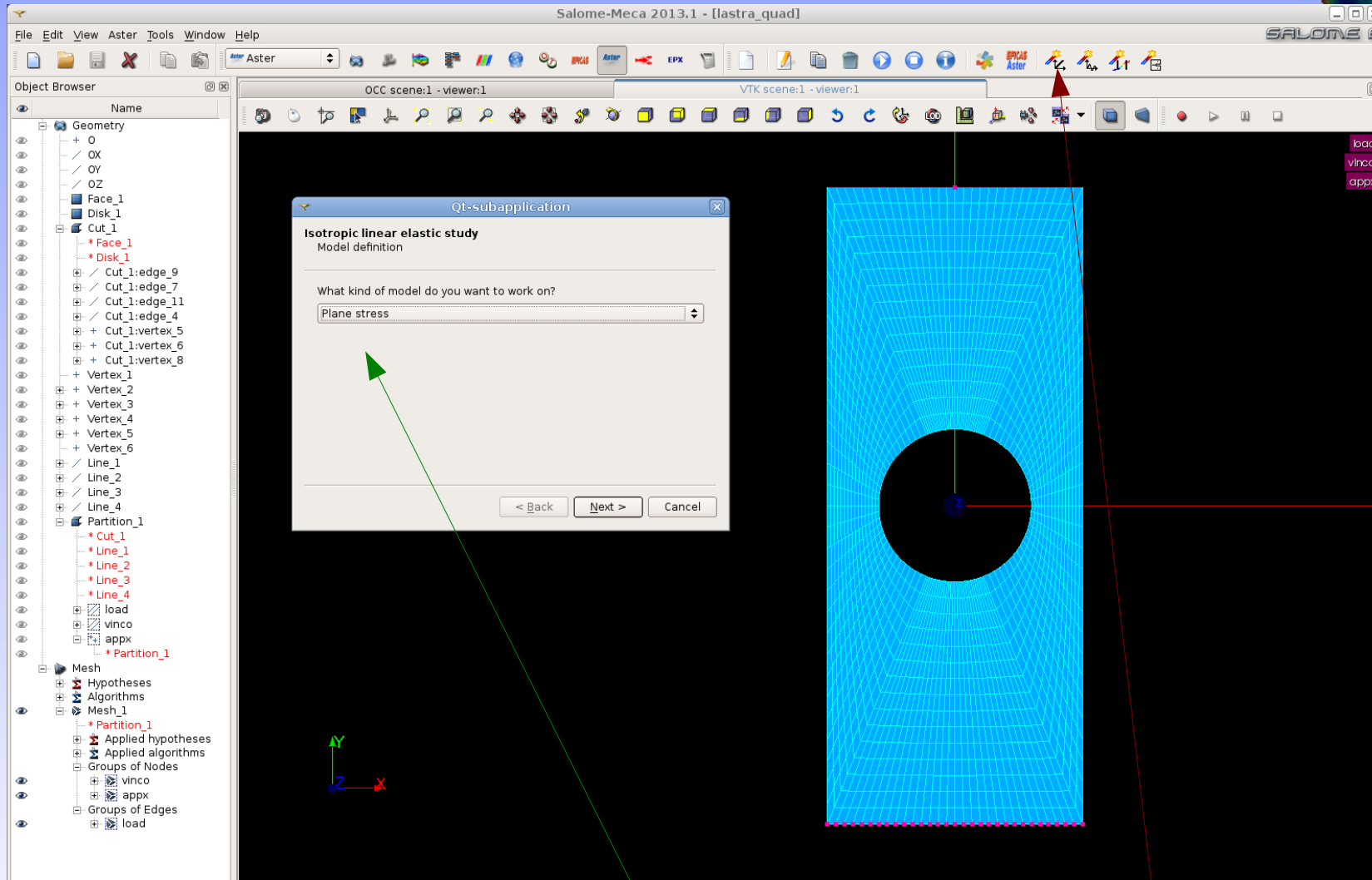
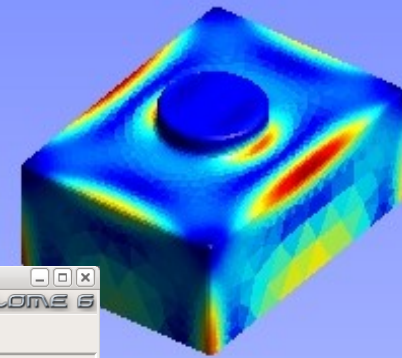
Nodi

La grafica permette di visualizzare gli elementi appartenenti ai gruppi creati:

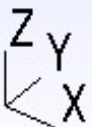
- i punti sono nodi
- le linee sono elementi monodimensionali



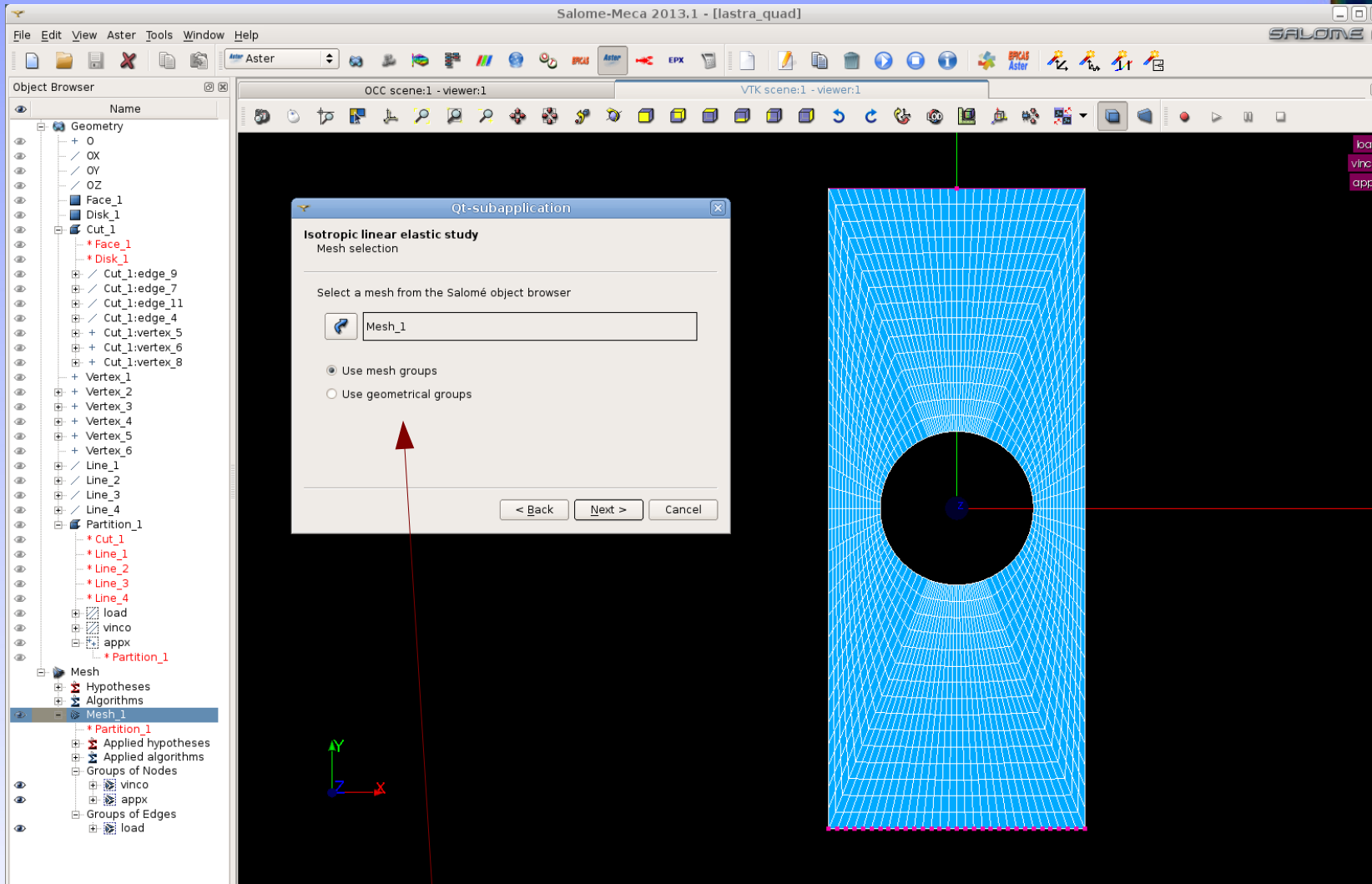
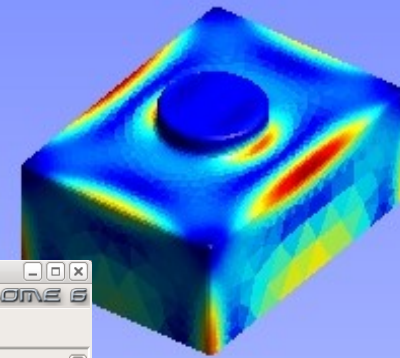
lastra forata: aster



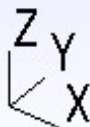
Nell'ambiente ASTER attiviamo l'assistente (wizard): "linear elastic". Scegliamo un'analisi in tensione piana (plane stress).



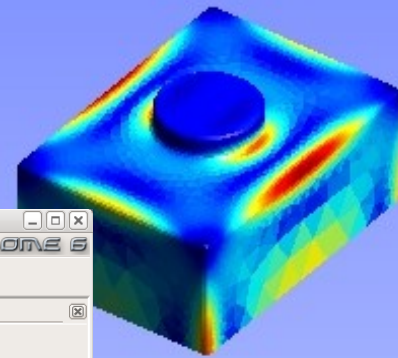
lastra forata: aster



Si sceglie la mesh dall'albero degli oggetti e se usare I gruppi della mesh o della geometria.



lastra forata: aster



Salome-Meca 2013.1 - [lastra_quad]

File Edit View Aster Tools Window Help

Object Browser

Geometry

- + 0
- / OX
- / OY
- / OZ
- Face_1
- Disk_1
- Cut_1
 - * Face_1
 - * Disk_1
 - / Cut_1.edge_9
 - / Cut_1.edge_7
 - / Cut_1.edge_11
 - / Cut_1.edge_4
 - + Cut_1.vertex_5
 - + Cut_1.vertex_6
 - + Cut_1.vertex_8
- + Vertex_1
- + Vertex_2
- + Vertex_3
- + Vertex_4
- + Vertex_5
- + Vertex_6
- / Line_1
- / Line_2
- / Line_3
- / Line_4
- Partition_1
 - * Cut_1
 - * Line_1
 - * Line_2
 - * Line_3
 - * Line_4
 - load
 - vinco
 - appx

Mesh

- Hypotheses
- Algorithms
- Mesh_1
 - * Partition_1
 - Applied hypotheses
 - Applied algorithms
 - Groups of Nodes
 - vinco
 - appx
 - Groups of Edges
 - load

Qt-subapplication

Isotropic linear elastic study

Material properties

Young's modulus and Poisson ratio definitions

Young's modulus (E) (E >= 0)

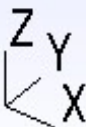
Poisson's ratio (nu) (-1 <= nu <= 0.5)

< Back Next > Cancel

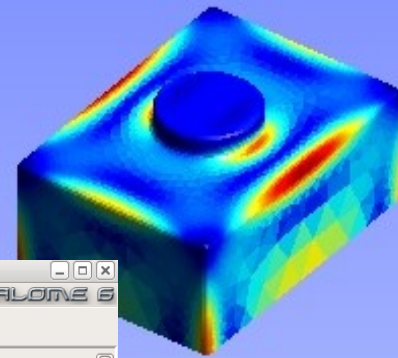
Si attribuiscono le proprietà del materiale della lastra:

- modulo di elasticità: 200 000 MPa
- coefficiente di Poisson: 0,3

Poiché il solutore elabora numeri puri, le unità di misura delle proprietà devono essere coerenti



lastra forata: aster



Qt-subapplication

Isotropic linear elastic study
Boundaries conditions

Adding imposed degrees of freedom on groups

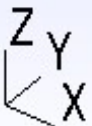
Group	DX	DY
vinco	0	0
appx	0	0

< Back Next > Cancel

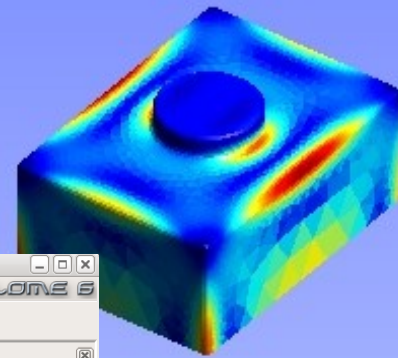
Object Browser

- Geometry
 - + 0
 - / OX
 - / OY
 - / OZ
 - Face_1
 - Disk_1
 - Cut_1
 - + * Face_1
 - + * Disk_1
 - / Cut_1.edge_9
 - / Cut_1.edge_7
 - / Cut_1.edge_11
 - / Cut_1.edge_4
 - + Cut_1.vertex_5
 - + Cut_1.vertex_6
 - + Cut_1.vertex_8
 - + Vertex_1
 - + Vertex_2
 - + Vertex_3
 - + Vertex_4
 - + Vertex_5
 - + Vertex_6
 - / Line_1
 - / Line_2
 - / Line_3
 - / Line_4
 - Partition_1
 - + Cut_1
 - + Line_1
 - + Line_2
 - + Line_3
 - + Line_4
 - load
 - vinco
 - appx
 - * Partition_1
- Mesh
 - + Hypotheses
 - + Algorithms
 - Mesh_1
 - * Partition_1
 - + Applied hypotheses
 - + Applied algorithms
 - Groups of Nodes
 - vinco
 - appx
 - Groups of Edges
 - load

Si impongono i vincoli allo spostamento di tutti i nodi dei gruppi creati



lastra forata: aster



Salome-Meca 2013.1 - [lastra_quad]

Object Browser

- Geometry
 - O
 - OX
 - OY
 - OZ
 - Face_1
 - Disk_1
 - Cut_1
 - Face_1
 - Disk_1
 - Cut_1:edge_9
 - Cut_1:edge_7
 - Cut_1:edge_11
 - Cut_1:edge_4
 - Cut_1:vertex_5
 - Cut_1:vertex_6
 - Cut_1:vertex_8
 - Vertex_1
 - Vertex_2
 - Vertex_3
 - Vertex_4
 - Vertex_5
 - Vertex_6
 - Line_1
 - Line_2
 - Line_3
 - Line_4
 - Partition_1
 - Cut_1
 - Line_1
 - Line_2
 - Line_3
 - Line_4
 - load
 - vinco
 - appx
- Mesh
 - Hypotheses
 - Algorithms
 - Mesh_1
 - Partition_1
 - Applied hypotheses
 - Applied algorithms
 - Groups of Nodes
 - vinco
 - appx
 - Groups of Edges
 - load

Qt-subapplication

Isotropic linear elastic study

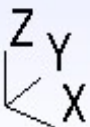
Boundaries conditions

Adding pressure on meshes groups

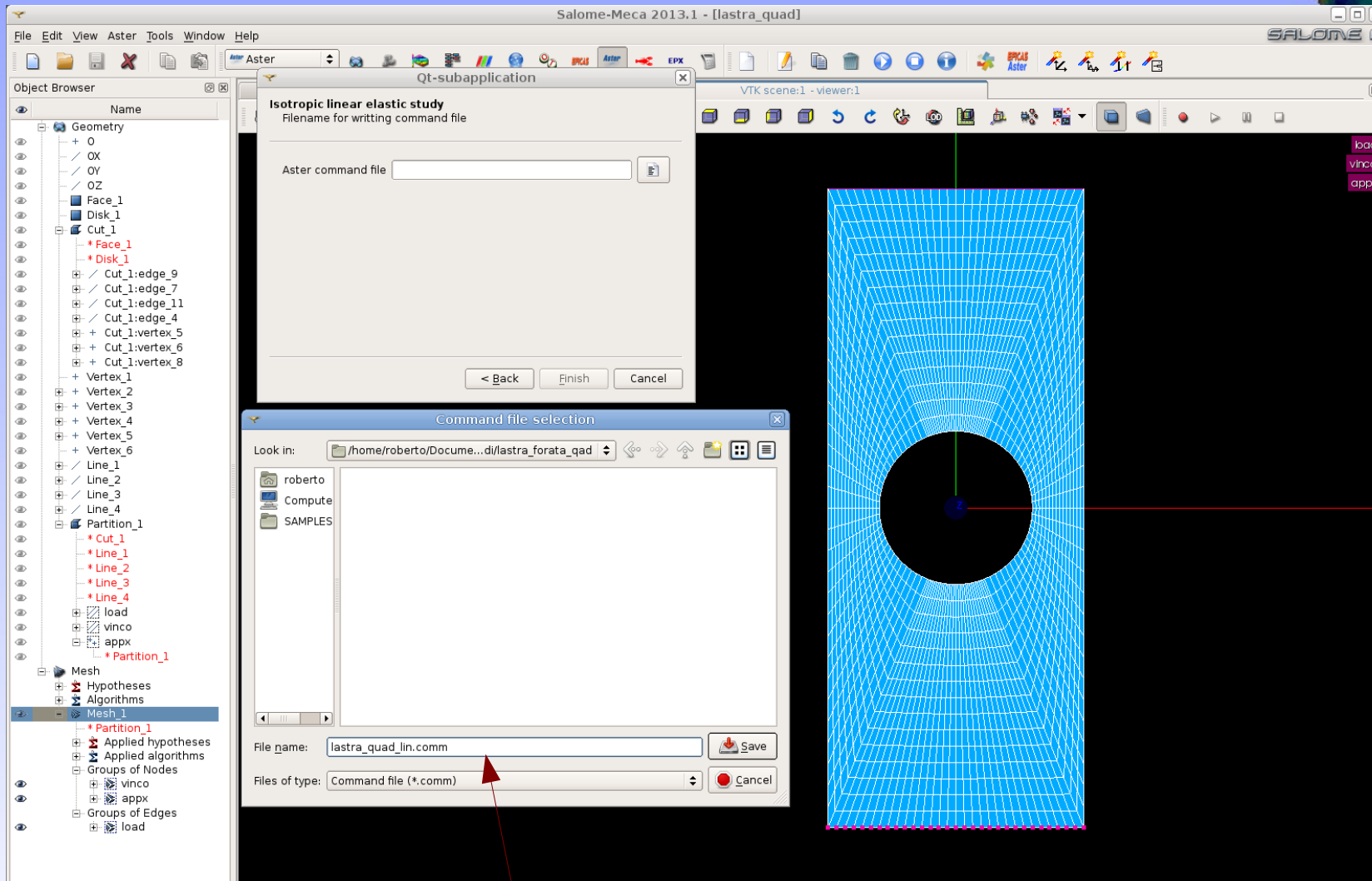
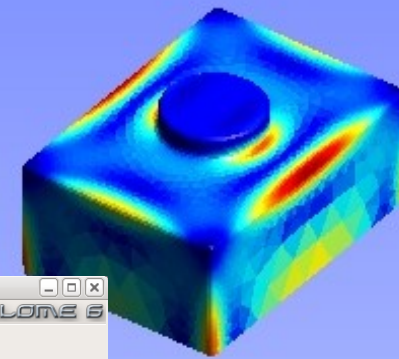
Group	Pressure
load	-10

< Back Next > Cancel

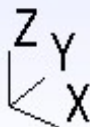
Si applica la pressione al gruppo di elementi monodimensionali.
Il segno negativo dirige la pressione all'esterno dell'area.
Applichiamo -10 MPa come nell'esempio teorico precedente.



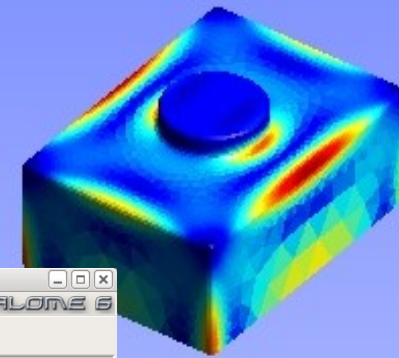
lastra forata: aster



Si specifica il percorso del file di comando contenente le istruzioni per il solutore



Esempio: aster



Salome-Meca 2013.1 - [lastra_quad]

Object Browser

lastra_quad_lin.comm

Commande	Concept/Valeur
lastra_quad_lin.comm	
DEBUT :	MA
DEFI_MATERIAU :	
ELAS :	MAIL
LIRE_MALLAGE :	MED
FORMAT :	
b_format_med :	MAIL
MODI_MALLAGE :	MAIL
MAILLAGE :	MAIL
ORIE_PEAU_2D :	
AFFE_MODELE :	MODE
MAILLAGE :	MAIL
AFFE :	
AFFE_MATERIAU :	MATE
MAILLAGE :	MAIL
AFFE :	
AFFE_CHAR_MECA :	CHAR
MODELE :	MODE
DDL_IMPO_1 :	
GROUP_NO :	vinco
DX :	0.0
DY :	0.0
DDL_IMPO_2 :	
GROUP_NO :	appx
DX :	0.0
DY :	0.0
PRES_REP :	
MECA_STATIQUE :	RESU
CALC_ELEM :	RESU
CALC_NO :	RESU
IMPR_RESU :	
FIN :	

Saisir Valeur

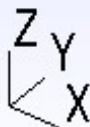
Valeur: 0

Paramètres

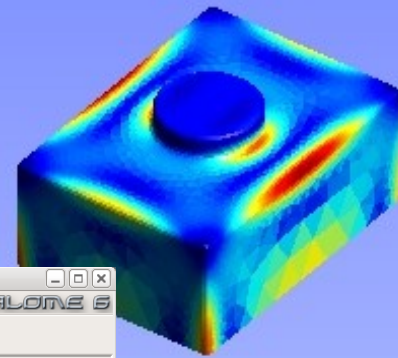
Un réel est attend.

Valider

Entriamo nell'editor dei file di comando (EFICAS) per cambiare i vincoli dei gruppi di nodi: togliamo dalla base della lastra il vincolo in direzione x

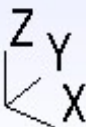


lastra forata: aster

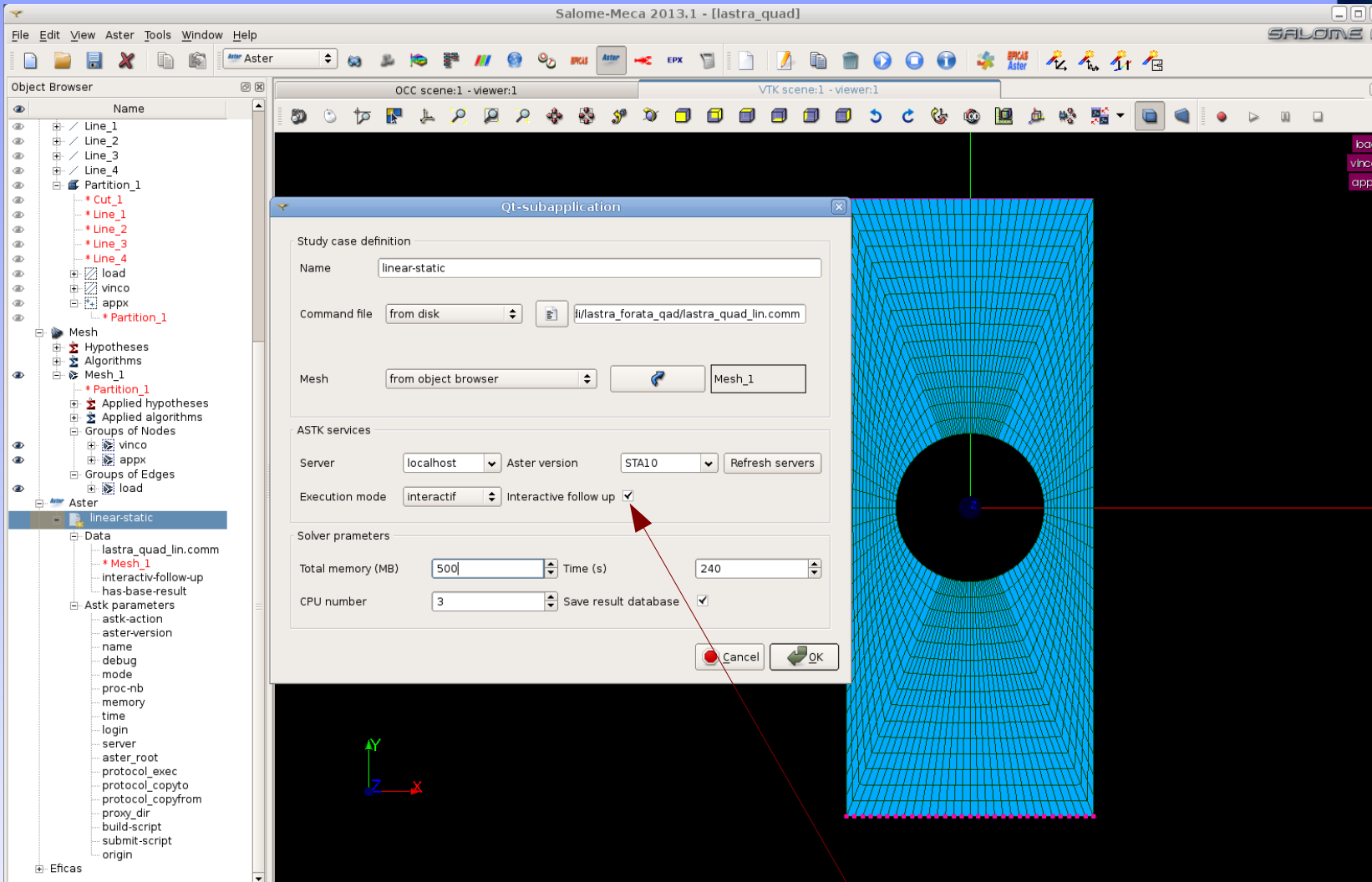
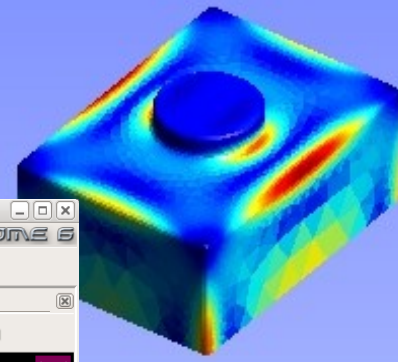


Commande	Concept/Valeur
lastra_quad_lin.comm	
DEBUT :	MA
DEFI_MATERIAU :	
ELAS :	
LIRE_MALLAGE :	MAIL MED
FORMAT :	
b_format_med :	
MODI_MALLAGE :	MAIL
MAILLAGE :	MAIL
ORIE_PEAU_2D :	
AFFE_MODELE :	MODE MAIL
MAILLAGE :	
AFFE :	
AFFE_MATERIAU :	MATE MAIL
MAILLAGE :	
AFFE :	
AFFE_CHAR_MECA :	CHAR MODE
MODELE :	
DDL_IMPO :	
DDL_IMPO 1 :	
GROUP_NO :	vinco 0.0
DY :	
DDL_IMPO 2 :	
GROUP_NO :	appx 0.0
DX :	
DY :	0.0
PRES_REP :	
MECA_STATIQUE :	RESU
CALC_ELEM :	RESU
CALC_NO :	RESU
IMPR_RESU :	
FIN :	

Dal gruppo contenente un solo nodo eliminiamo il vincolo in direzione y

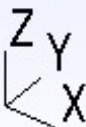


lastra forata: aster

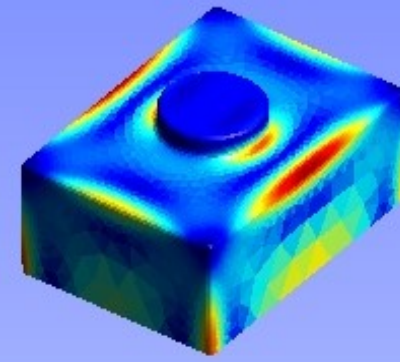


Si impostano i parametri dello studio:

- Percorso file di comando
- Mesh
- Macchina su cui eseguire il calcolo
- La versione del solutore
- Il modo interattivo mostra I messaggi del solutore
- RAM da destinare al calcolo
- Tempo di calcolo massimo
- Numero delle CPU da impiegare nel calcolo



lastra forata: aster



```
bash
Memory (MB) : 500,0
Number of processors : 5 (OpenMP)
Number of nodes : (MPI)
Number of processors : (MPI)
Node : interactif

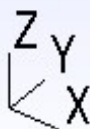
Debug mode : nodebug
BTC script : generated

-----
Version ASTK Server : 1.11.0
Version ASTK Client : 1.10.0
-----
#####
[ OK ]
adding a symbolic link /tmp/roberto-amd64-interactif-0001-3516-amd64/Python/Cats/cata.pg to
/opt/salome13/SALOME-NECR-2013.1-LGPL/tools/Code_aster_standalone_20131_public/STAR10/com
mande/cata.pg...
[ OK ]
copying .../STAR10/elements...
[ OK ]
-----
Copying datas
copying .../lastra_forata_qad/lastra_q
copying .../lastra_forata_qad/linear=SERV.pg -eficas_path ./Python -commandes fort.1 -rnm
(LINFO) Parameters : memory 500 MB - tirrep_utils /opt/salome13/SALOME-NECR-2013.1-LGPL/As
tutils -rep_ast /opt/salome13/SALOME-NECR-2013.1-LG
#####public/STAR10/materiau -rep_dex /opt/salome13/SALOME-NE
NECR-2013.1-LGPL/tools/Code_aster_standalone_20131_public/STAR10/datg -suivi_batch -menjeveu
x 52,5 -tmax 240
```

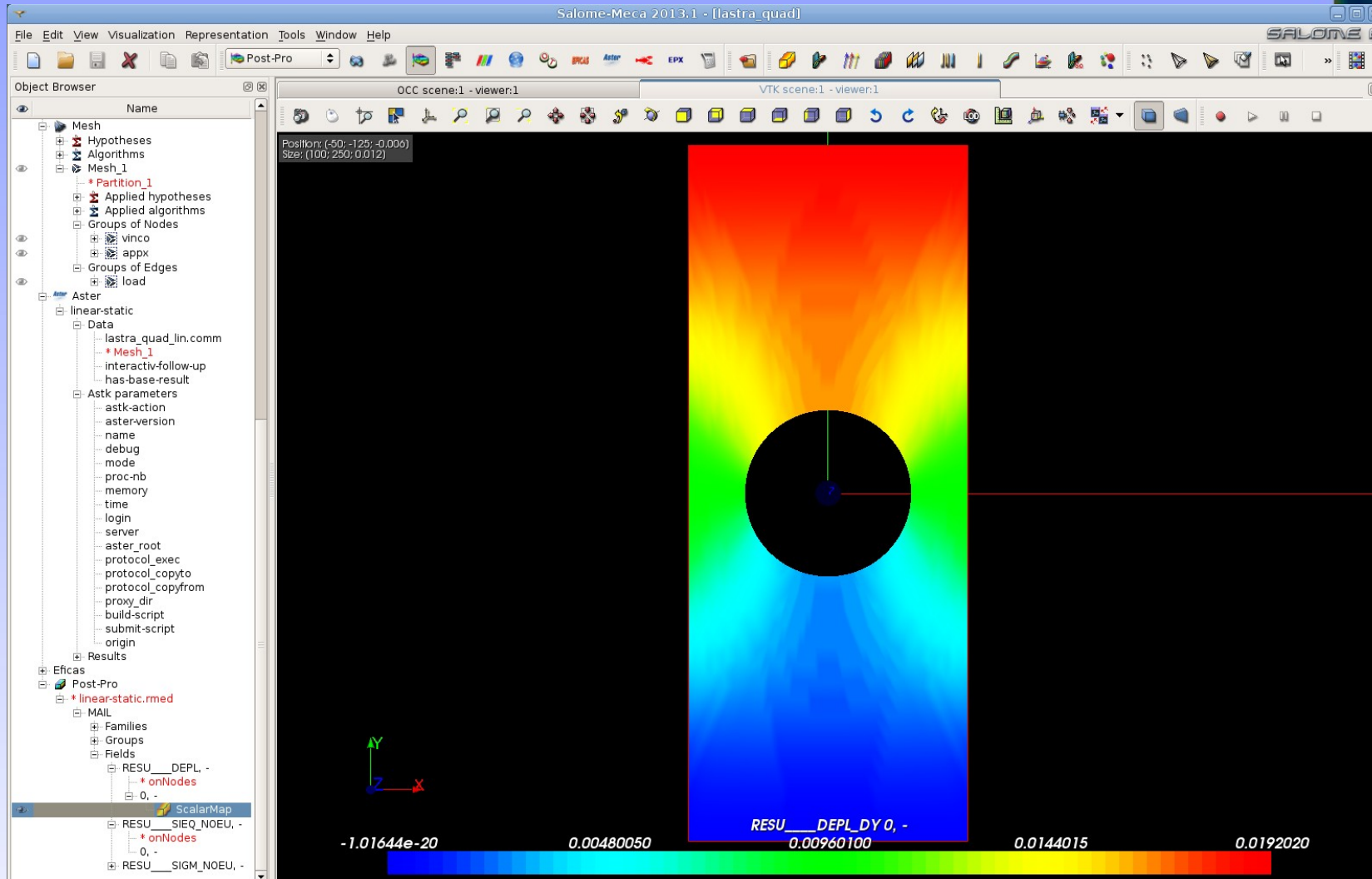
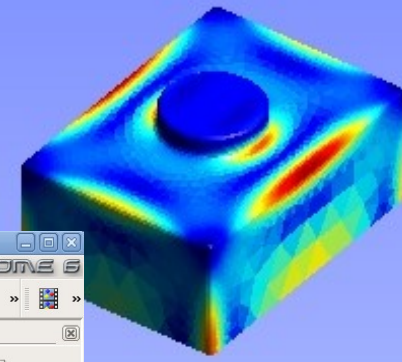
Python Console
>>>

Message Window
[12 Oct 2013] Job
'linear-static'
submitted

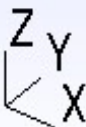
Premendo sul bottone “run” si lancia il calcolo e compare la finestra “bash” con i messaggi del solutore. Il calcolo progredisce nel tempo e si arresta con un messaggio di conclusione corretta.



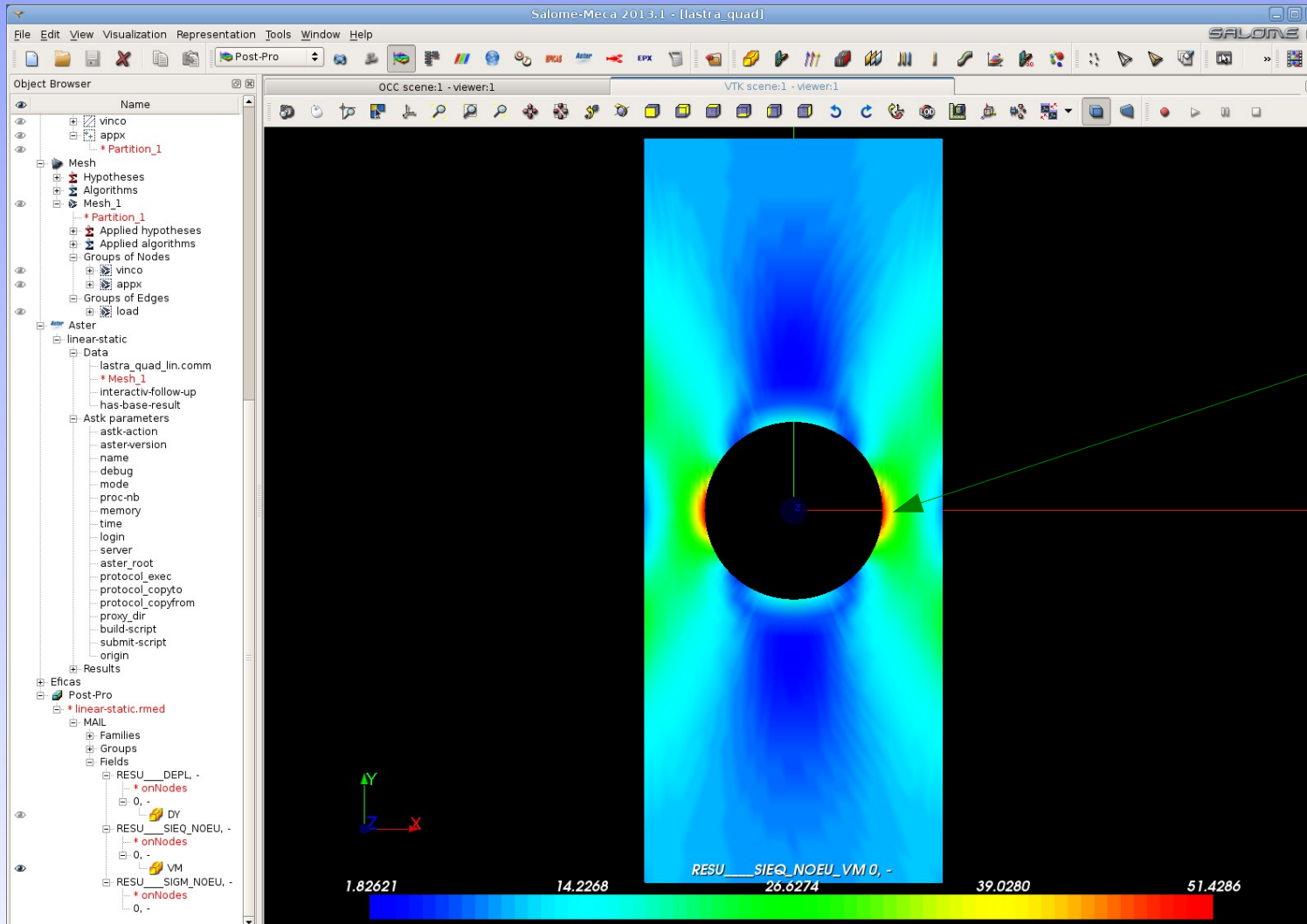
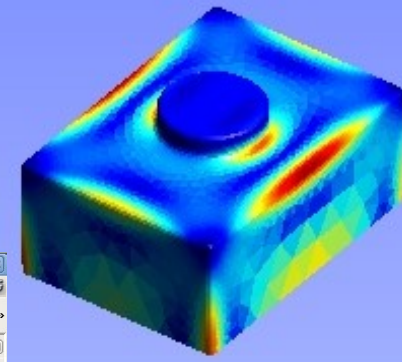
lastra forata: post-pro



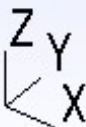
Spostamenti in direzione y rappresentati per frange colorate



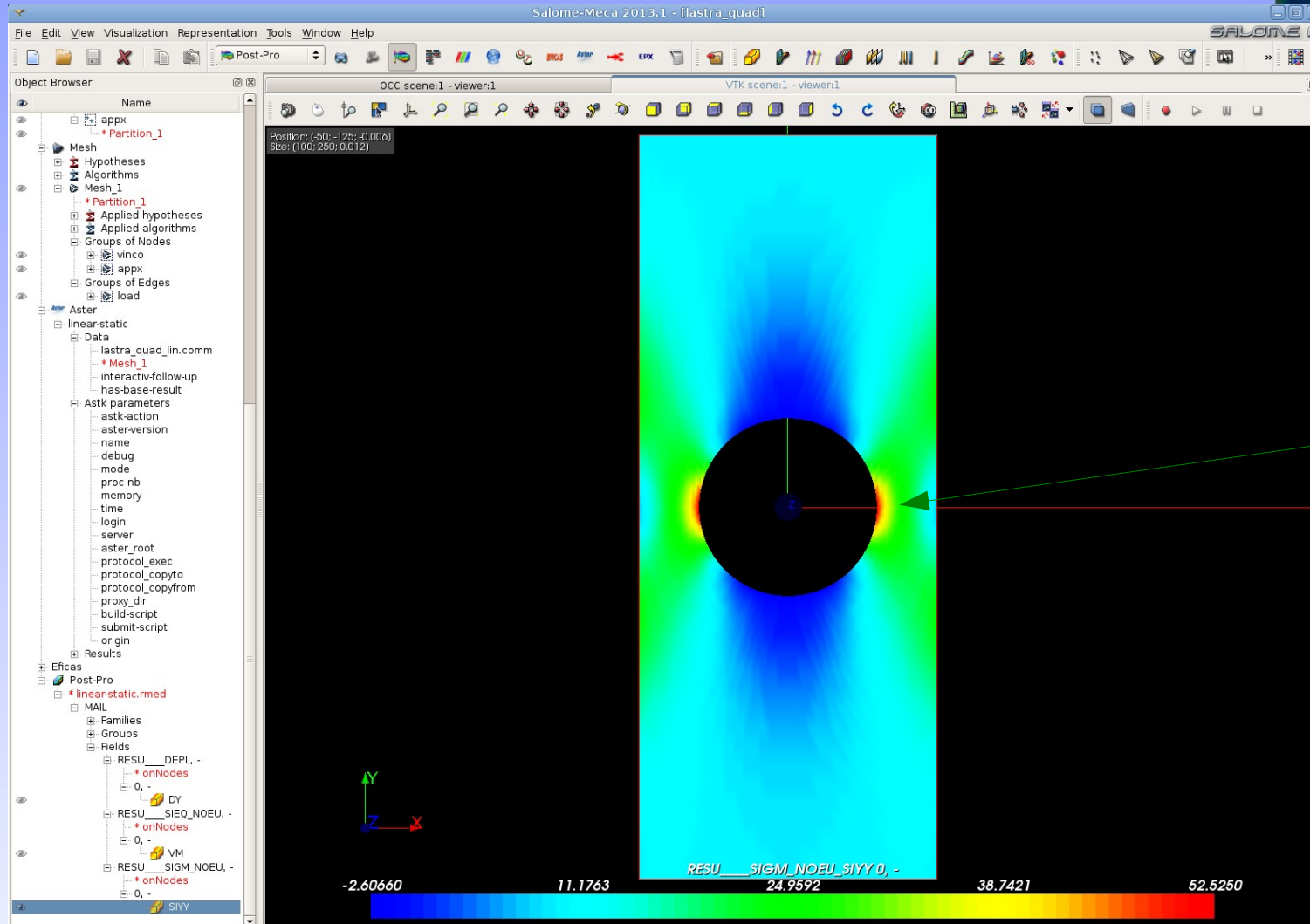
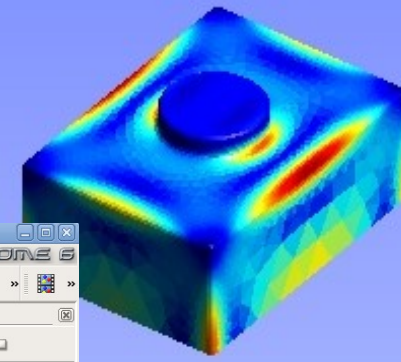
lastra forata: post-pro



Tensione ideale secondo Von Mises rappresentata per frange colorate

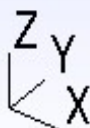


lastra forata: post-pro

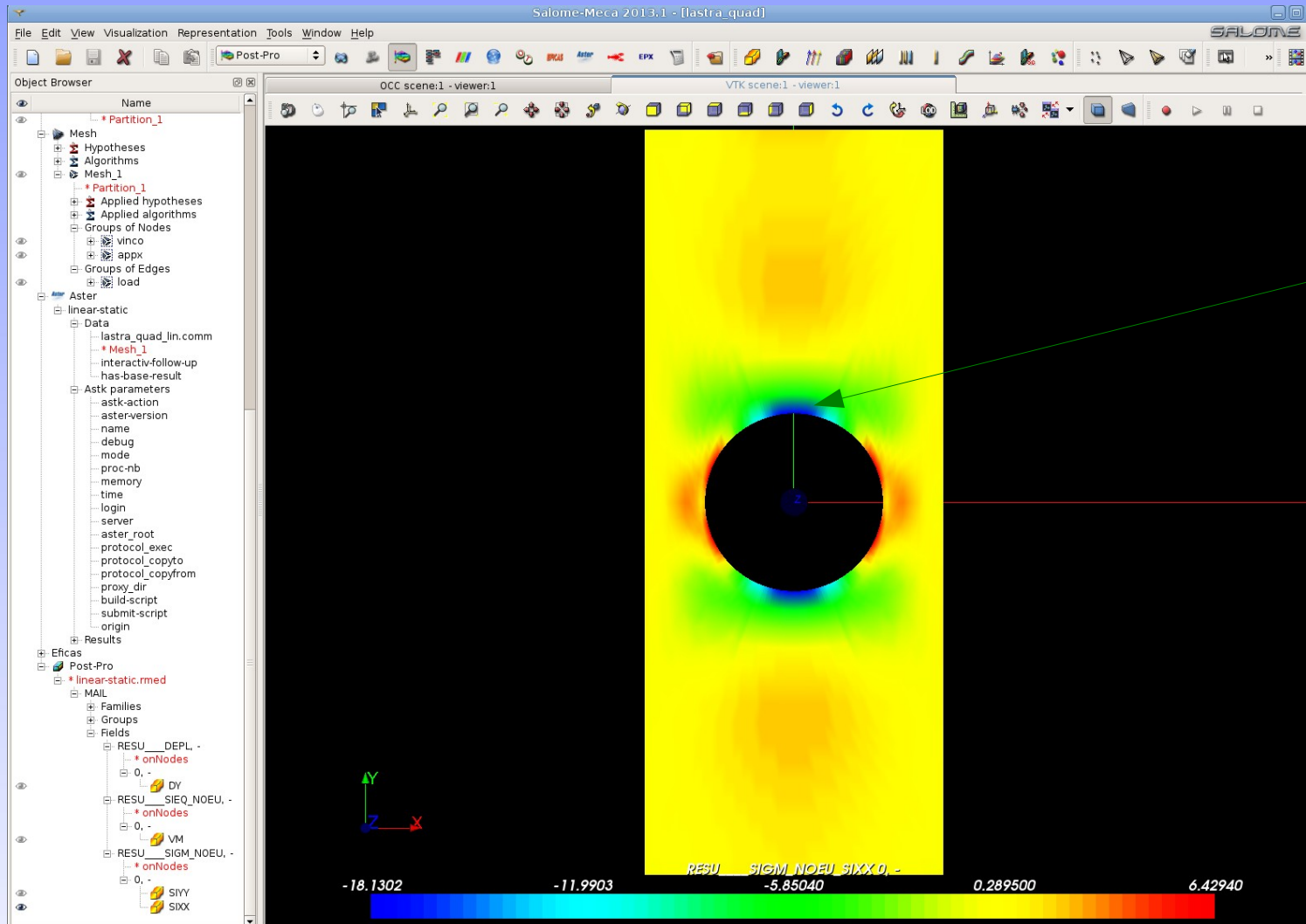
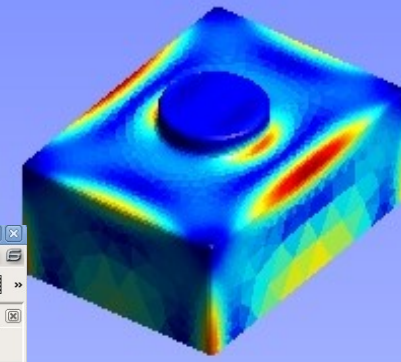


52,5 MPa

Tensioni in direzione y (σ_y) rappresentate per frange colorate.
Si noti la corrispondenza con la tensione teorica calcolata in precedenza
pari a 52,5 MPa.

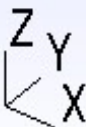


lastra forata: post-pro

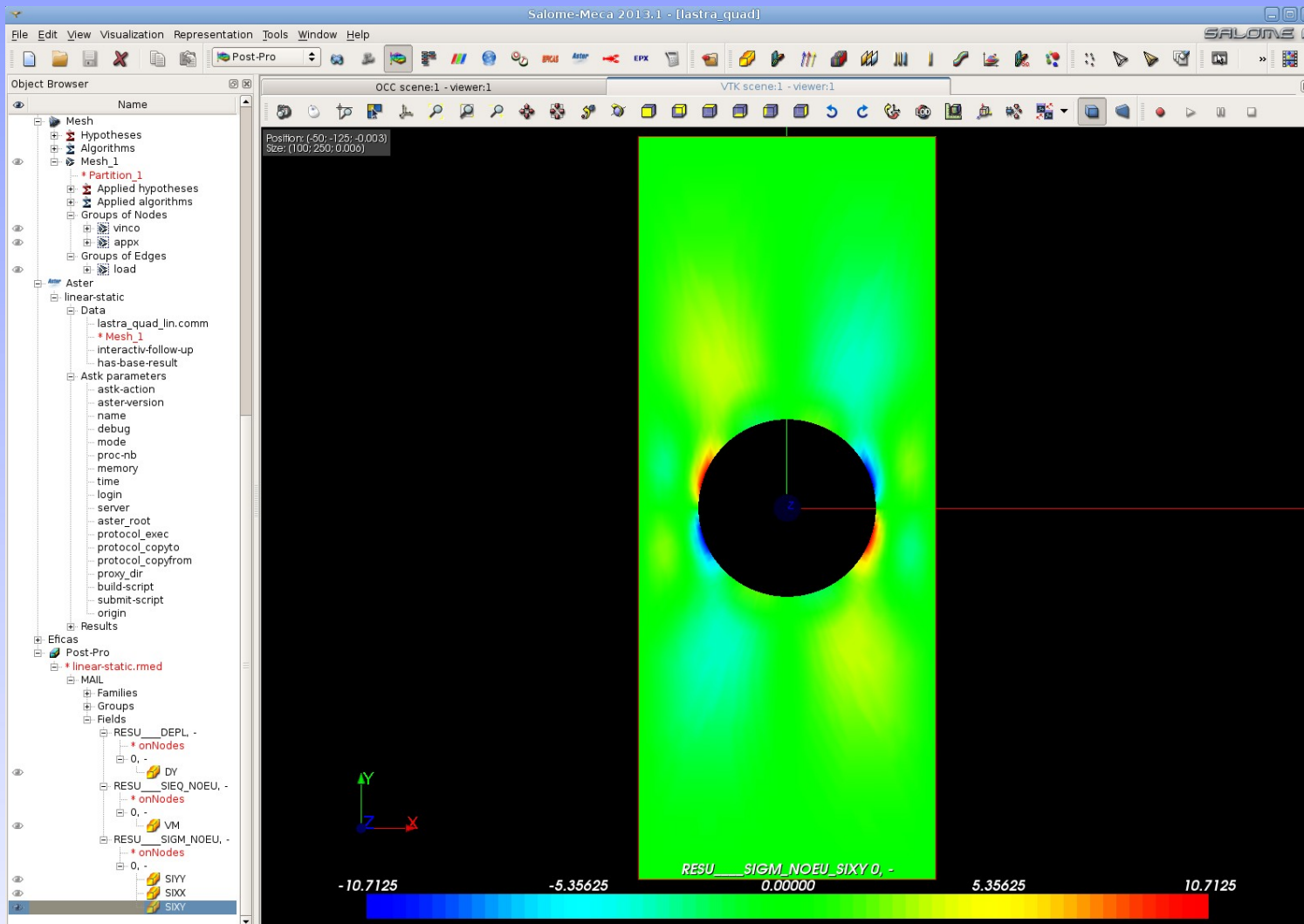
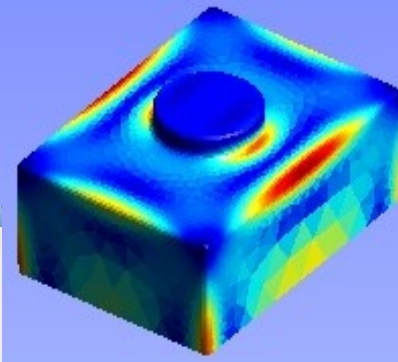


Tensione nel punto C:
-18,1 MPa

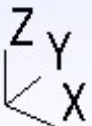
Tensioni in direzione x (σ_x) rappresentate per frange colorate. La teoria dice che per foro piccolo la tensione nel punto C è compressiva e paragonabile alla tensione nominale. Nel nostro esempio la tensione nominale è pari a 25 MPa.



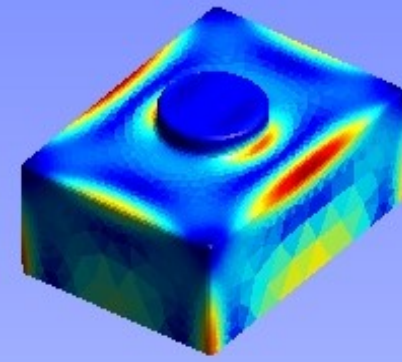
lastra forata: post-pro



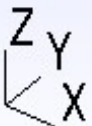
Tensioni taglienti τ_{xy} rappresentate per frange colorate. Si noti l'antisimmetria rispetto agli assi cartesiani dell'andamento dei colori, quindi delle tensioni. In un punto dato, ho la stessa tensione del suo simmetrico ma di segno opposto.



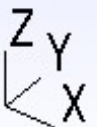
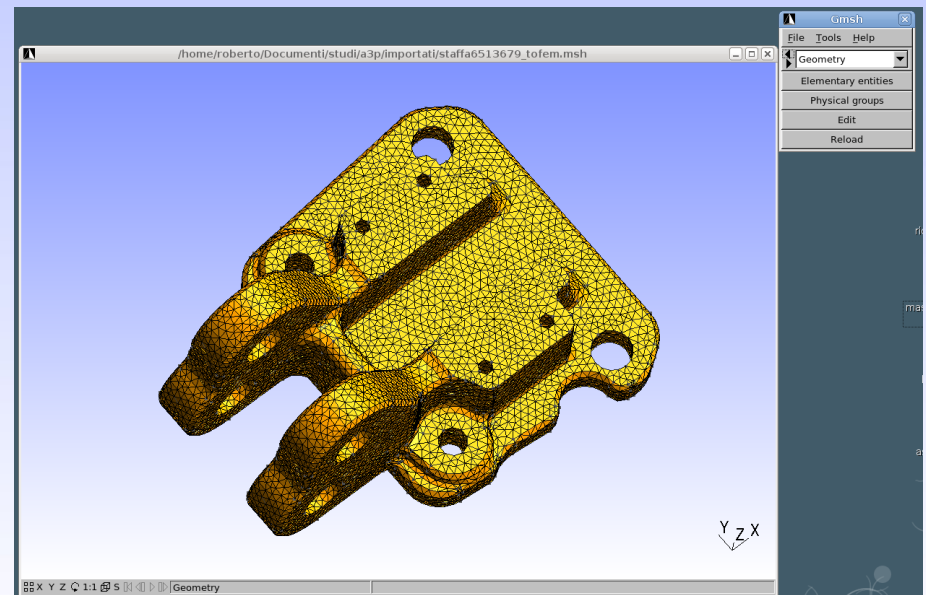
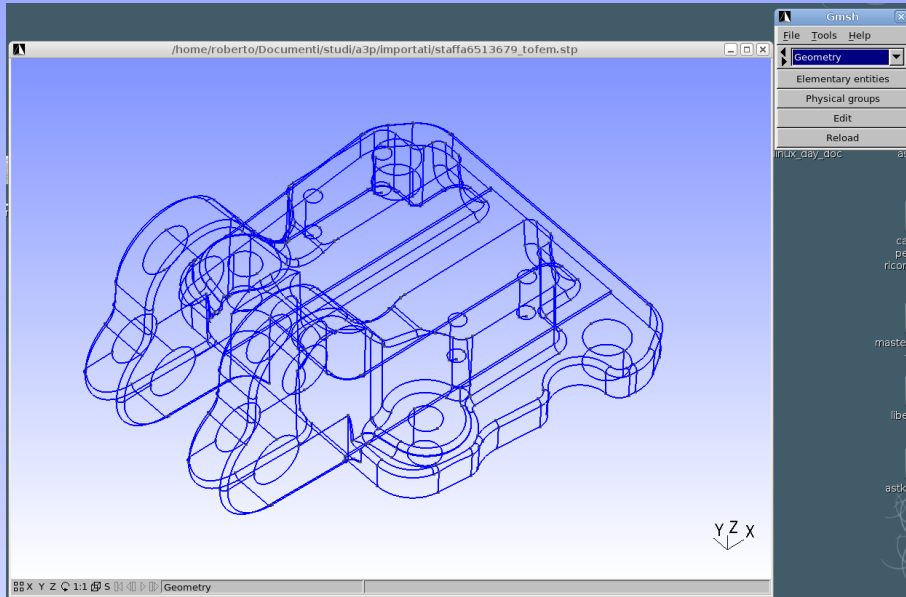
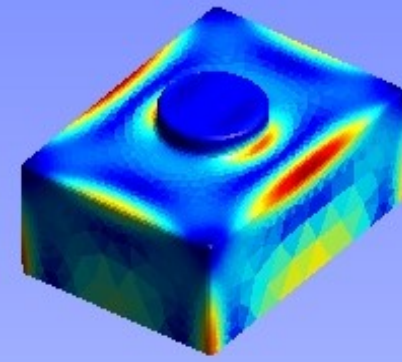
Moduli esterni



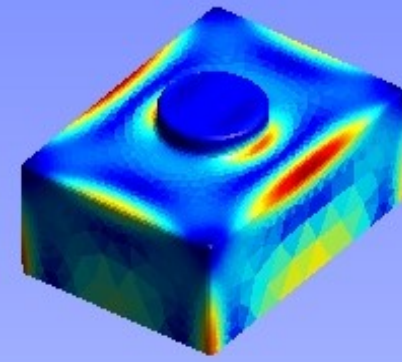
- GMSH strumento pre-post processor
- Serve a creare geometrie, costruirvi sopra la mesh di elementi e a visualizzare i risultati
- Librerie MED per lo scambio di dati all'interno del salome-meca



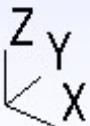
GMSH



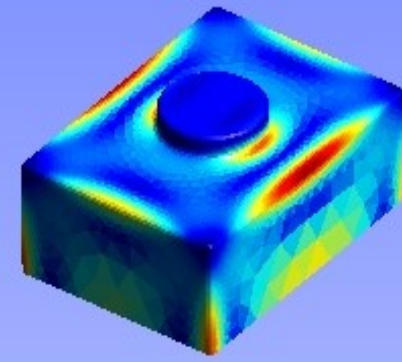
Code aster gli elementi



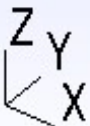
- 0D masse concentrate, molle e smorzatori
- 1D travi e travi curve, barre, cavi: usati per strutture snelle con asse neutro dove una dimensione prevale sulle altre due
- 1D tubi: usati per simulare tubi dritti o curvi
- 2D elementi shell e plate: usati per strutture snelle con superficie neutra dove una dimensione è piccola rispetto le altre due
- 3D elementi di volume: usati per strutture dove tutte e tre le dimensioni sono confrontabili



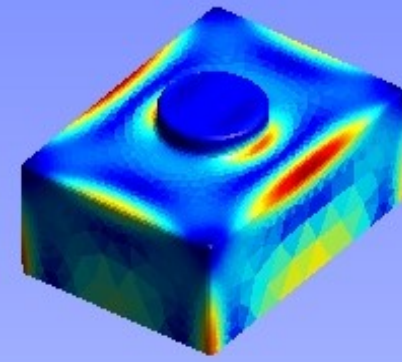
Elementi in dettaglio



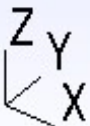
- Travi di eulero e di timoshenko e travi multifibra
- Strutture planari e curve, strutture spesse con rapporto spessore/lunghezza circa 1/10
- Strutture planari in grandi spostamenti e con materiale non lineare
- Volumi 3D dove simulare la meccanica del continuo



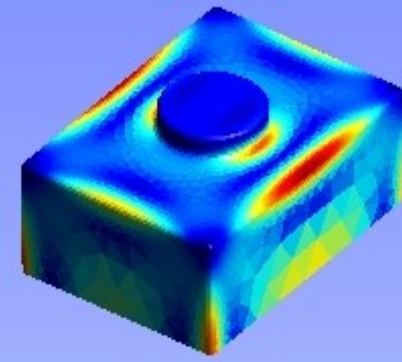
I carichi



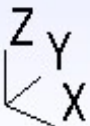
- Forze di volume: gravità, forza centrifuga, ecc...
- Forze di superficie: pressioni, forze su bordi, ecc..
- Forze concentrate: forze su singoli nodi
- Forze su elementi specifici: forze distribuite sulle travi, sui gusci o pressioni all'interno degli elementi tubo
- Carichi termici: flusso di calore, temperatura iniziale, ecc...
- Deformazioni iniziali



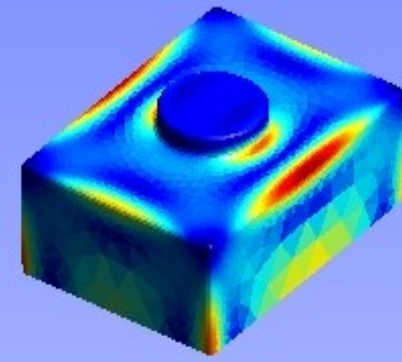
Solutori



- Solutori meccanici
 - Lineare e non lineare: MECA_STATIQUE, STAT_NON_LINE
 - Dinamici: DYNA_LINE_TRAN, DYNA_NON_LINE
 - Analisi modale: CALC_MODAL,
 - Dinamici su base modale: DYNA_TRAN_MODAL, DYNA_VIBRA
- Altri solutori
 - Termici: THER_LINEAIRE, THER_NON_LINE
 - Acustici: PHENOMENE_ACOUSTIQUE



Materiali non lineari

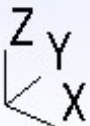


- Curva bilineare:

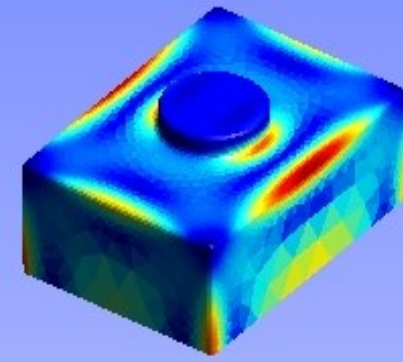
```
MA=DEFI_MATERIAU(ELAS=_F(E=210000.0,  
                          NU=0.3, ),  
                ECRO_LINE=_F(D_SIGM_EPSI=394.0,  
                              SY=235.0, ), );
```

- Curva fornita dall'utente:

```
TRACTION = _F (  
    ◆ SIGM = sigm_f,  
    )
```



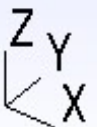
Materiali non lineari



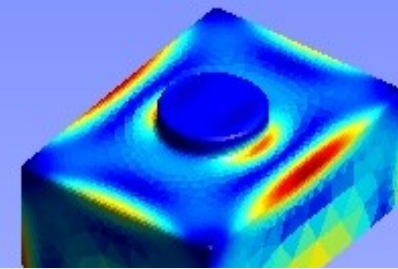
- Curva esponenziale di incrudimento:

$$\begin{aligned} \epsilon_{CRO_PUI} &= \frac{\sigma_y}{E} \left(\frac{\sigma}{\sigma_y} \right)^n \\ \sigma_y &= \text{sig}_y, \\ A_PUI &= a, \\ N_PUI &= n, \end{aligned}$$

$$\epsilon = \frac{\sigma}{E} + a \frac{\sigma_y}{E} \left(\frac{\sigma}{\sigma_y} \right)^n$$



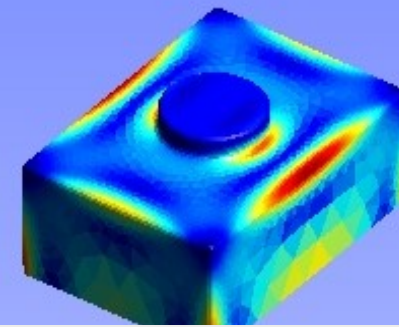
File di comando



```
DEBUT();
MA=DEFI_MATERIAU(ELAS=_F(E=210000.0,
                        NU=0.3),);
MAIL1=LIRE_MAILLAGE(FORMAT='MED',);
MODE1=AFFE_MODELE(MAILLAGE=MAIL1,
                  AFPE=_F(TOUT='OUI',
                          PHENOMENE='MECANIQUE',
                          MODELISATION='3D',),);
MAIL1=MODI_MAILLAGE(reuse =MAIL1,
                   MAILLAGE=MAIL1,
                   ORIE_PEAU_3D=_F(GROUP_MA=('bor','mar',),),);
MATE1=AFPE_MATERIAU(MAILLAGE=MAIL1,
                    AFPE=_F(TOUT='OUI',
                            MATER=MA,)),);
CHAR1=AFPE_CHAR_MECA(MODELE=MODE1,
                     DDL_IMPO=( _F(GROUP_MA='app',
                                   DX=0.0,),
                               _F(GROUP_MA='cer',
                                   DX=0.0,
                                   DY=0.0,
                                   DZ=0.0,),
                               _F(GROUP_MA='sim',
                                   DZ=0.0,)),
                     PRES_REP=( _F(GROUP_MA='bor',
                                   PRES=244.86,),
                                _F(GROUP_MA='mar',
                                   PRES=20.26,)),);
RESU1=MECA_STATIQUE(MODELE=MODE1,
                    CHAM_MATER=MATE1,
                    EXCIT=_F(CHARGE=CHAR1,)),);
RESU1=CALC_ELEM(reuse =RESU1,
                RESULTAT=RESU1,
                OPTION=('SIGM_ELNO_DEPL','EQUI_ELNO_SIGM','ERRE_ELEM_SIGM',),
                EXCIT=_F(CHARGE=CHAR1,)),);
```

```
semi_sottile_le.comm x
INDI1=CREA_CHAMP(TYPE_CHAM='ELEM_ERRE_R',
                 OPERATION='EXTR',
                 RESULTAT=RESU1,
                 NOM_CHAM='ERRE_ELEM_SIGM',
                 NUME_ORDRE=1,);
MACR_ADAP_MAIL(MAILLAGE_N=MAIL1,
               MAILLAGE_NP1=CO('MAIL2'),
               ADAPTATION='RAFF_DERA',
               CHAM_GD=INDI1,
               NOM_CMP_INDICA='ERREST',
               CRIT_RAFF_PE=0.2,
               CRIT_DERA_PE=0.25,
               NIVE_MAX=8,
               NIVE_MIN=0,
               QUALITE='OUI',);
#CALCOLO SULLA SECONDA MAGLIA
MODE2=AFPE_MODELE(MAILLAGE=MAIL2,
                  AFPE=_F(TOUT='OUI',
                          PHENOMENE='MECANIQUE',
                          MODELISATION='3D',),);
MATE2=AFPE_MATERIAU(MAILLAGE=MAIL2,
                    AFPE=_F(TOUT='OUI',
                            MATER=MA,)),);
CHAR2=AFPE_CHAR_MECA(MODELE=MODE2,
                     DDL_IMPO=( _F(GROUP_MA='app',
                                   DX=0.0,),
                               _F(GROUP_MA='cer',
                                   DX=0.0,
                                   DY=0.0,
                                   DZ=0.0,),
                               _F(GROUP_MA='sim',
                                   DZ=0.0,)),
                     PRES_REP=( _F(GROUP_MA='bor',
                                   PRES=244.86,),
                                _F(GROUP_MA='mar',
                                   PRES=20.26,)),);
```

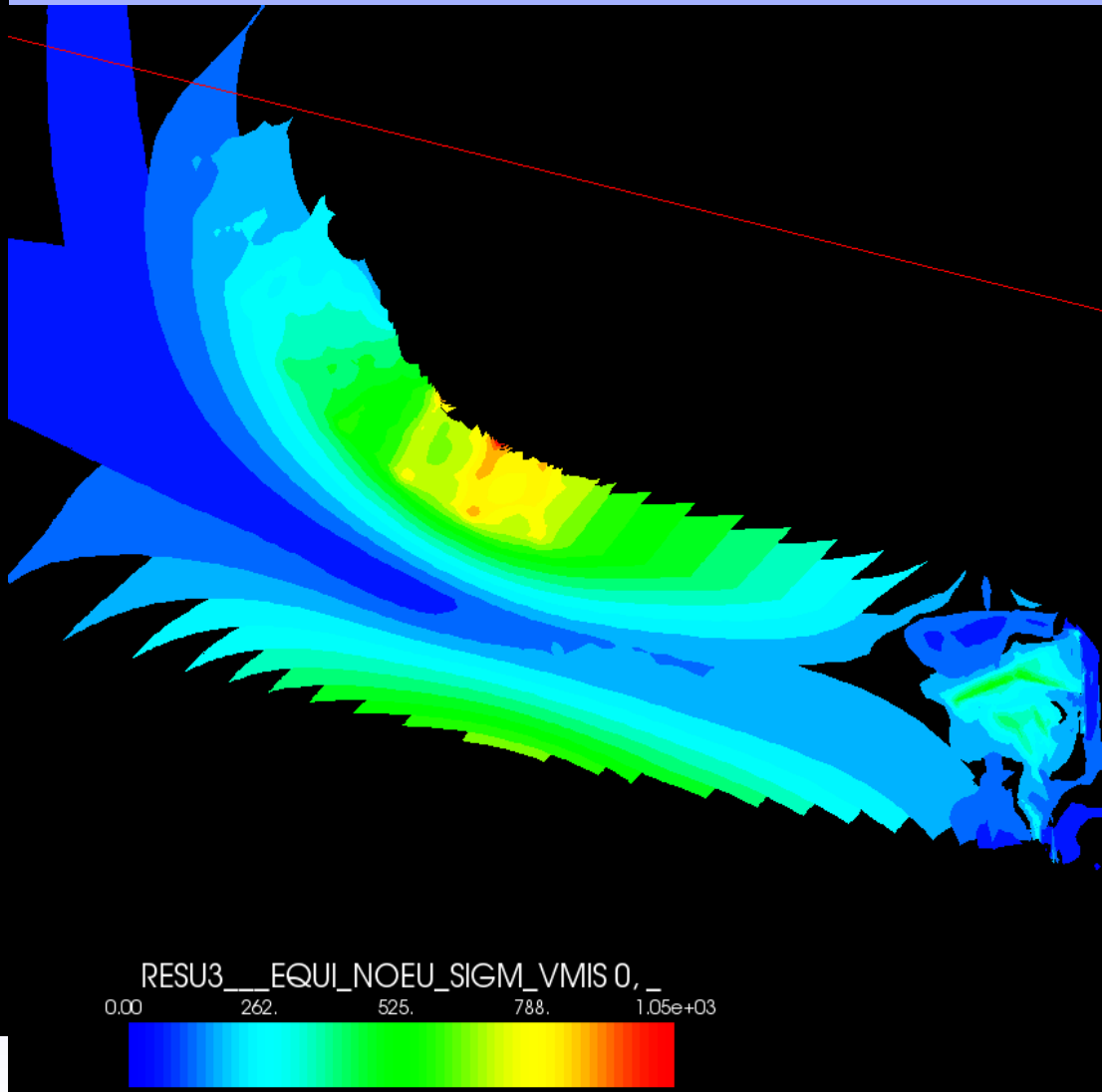
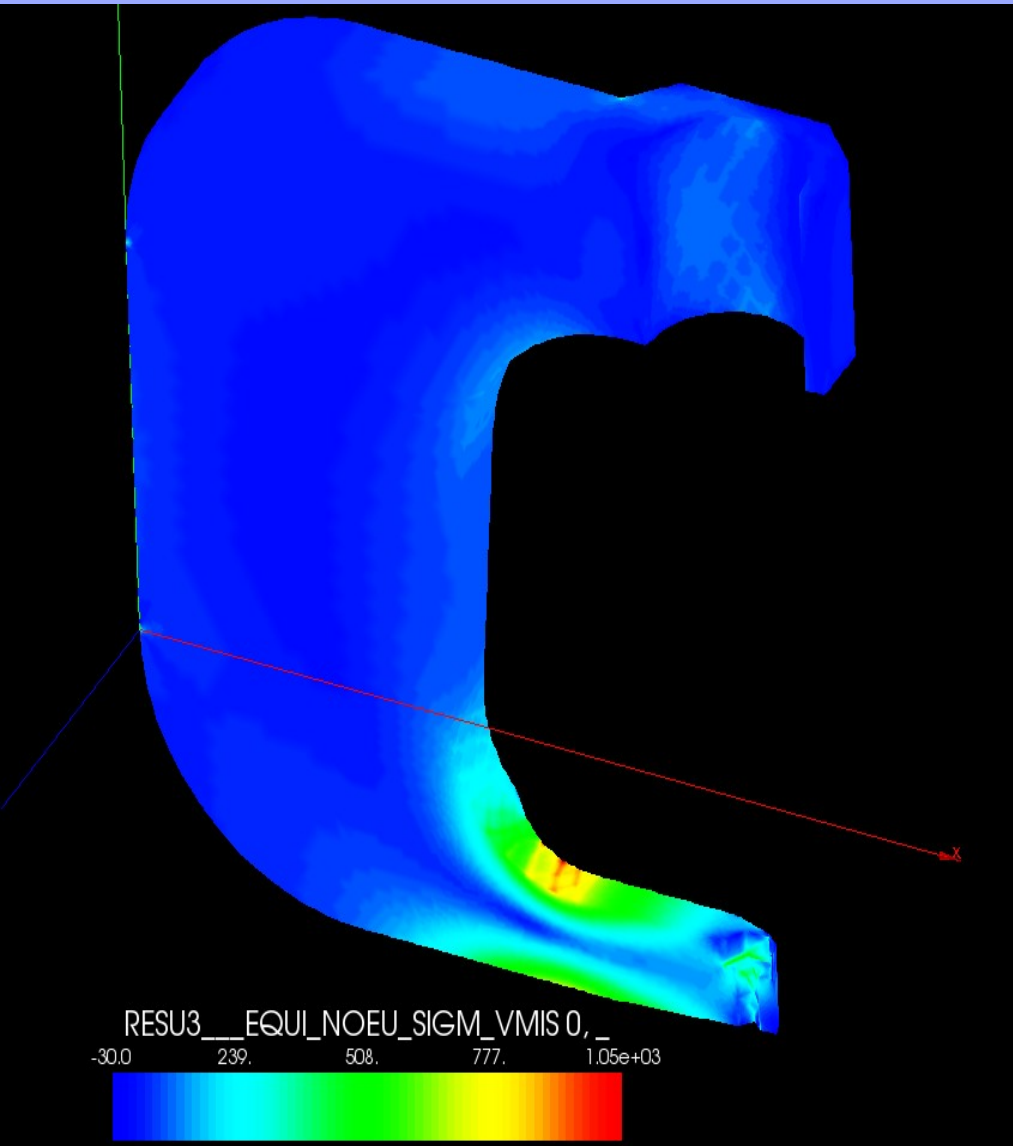
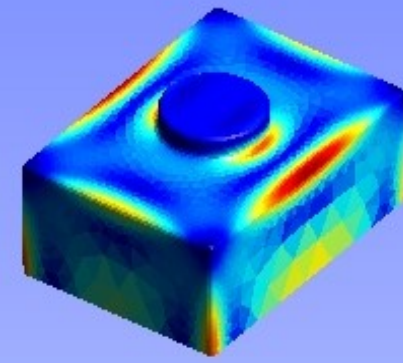
File di comando



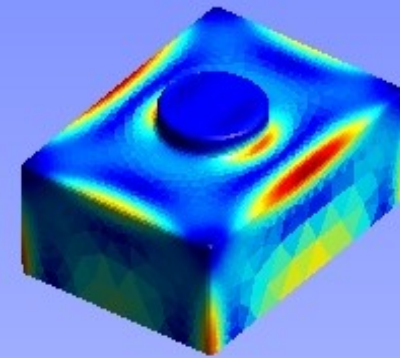
```
semi_sottile_le.comm x
RESU2=MECA_STATIQUE(MODELE=MODE2,
  CHAM_MATER=MATE2,
  EXCIT=_F(CHARGE=CHAR2,,));
RESU2=CALC_ELEM(reuse =RESU2,
  RESULTAT=RESU2,
  OPTION=('SIGM_ELN0_DEPL', 'EQUI_ELN0_SIGM', 'ERRE_ELEM_SIGM',),
  EXCIT=_F(CHARGE=CHAR2,,));
INDI2=CREA_CHAMP(TYPE_CHAM='ELEM_ERRE_R',
  OPERATION='EXTR',
  RESULTAT=RESU2,
  NOM_CHAM='ERRE_ELEM_SIGM',);
MACR_ADAP_MAIL(MAILLAGE_N=MAIL2,
  MAILLAGE_NP1=CO('MAIL3'),
  ADAPTATION='RAFF_DERA',
  CHAM_GD=INDI2,
  NOM_CMP_INDICA='ERREST',
  CRIT_RAFF_PE=0.2,
  CRIT_DERA_PE=0.25,);
#calcolo sulla terza maglia
MODE3=AFFE_MODELE(MAILLAGE=MAIL3,
  AFFE=_F(TOUT='OUI',
  PHENOMENE='MECANIQUE',
  MODELISATION='3D',),);
MATE3=AFFE_MATERIAU(MAILLAGE=MAIL3,
  AFFE=_F(TOUT='OUI',
  MATER=MA,));
CHAR3=AFFE_CHAR_MECA(MODELE=MODE3,
  DDL_IMPO=(_F(GROUP_MA='app',
  DX=0.0,),
  _F(GROUP_MA='cer',
  DX=0.0,
  DY=0.0,
  DZ=0.0,),
  _F(GROUP_MA='sim',
  DZ=0.0,)),
  PRES_REP=(_F(GROUP_MA='bor',
  PRES=244.86),
  _F(GROUP_MA='mar',
  PRES=20.26,)),);
```

```
semi_sottile_le.comm x
CHAR3=AFFE_CHAR_MECA(MODELE=MODE3,
  DDL_IMPO=(_F(GROUP_MA='app',
  DX=0.0,),
  _F(GROUP_MA='cer',
  DX=0.0,
  DY=0.0,
  DZ=0.0,),
  _F(GROUP_MA='sim',
  DZ=0.0,)),
  PRES_REP=(_F(GROUP_MA='bor',
  PRES=244.86,),
  _F(GROUP_MA='mar',
  PRES=20.26,)),);
RESU3=MECA_STATIQUE(MODELE=MODE3,
  CHAM_MATER=MATE3,
  EXCIT=_F(CHARGE=CHAR3,,));
RESU3=CALC_ELEM(reuse =RESU3,
  RESULTAT=RESU3,
  OPTION=('SIGM_ELN0_DEPL', 'EQUI_ELN0_SIGM', 'ERRE_ELEM_SIGM',),
  EXCIT=_F(CHARGE=CHAR3,,));
INDI3=CREA_CHAMP(TYPE_CHAM='ELEM_ERRE_R',
  OPERATION='EXTR',
  RESULTAT=RESU3,
  NOM_CHAM='ERRE_ELEM_SIGM',);
RESU3=CALC_NO(reuse =RESU3,
  RESULTAT=RESU3,
  OPTION=('SIGM_NOEU_DEPL', 'EQUI_NOEU_SIGM',),);
IMPR_RESU(FORMAT='MED',
  UNITE=80,
  RESU=_F(MAILLAGE=MAIL3,
  RESULTAT=RESU3,
  NOM_CHAM=('SIGM_NOEU_DEPL', 'EQUI_NOEU_SIGM', 'DEPL',),),);
IMPR_RESU(FORMAT='MED',
  RESU=_F(CHAM_GD=INDI3,));
FIN(FORMAT_HDF='NON',
  PERFORMANCE='OUI',
  INFO_RESU='OUI',);
```

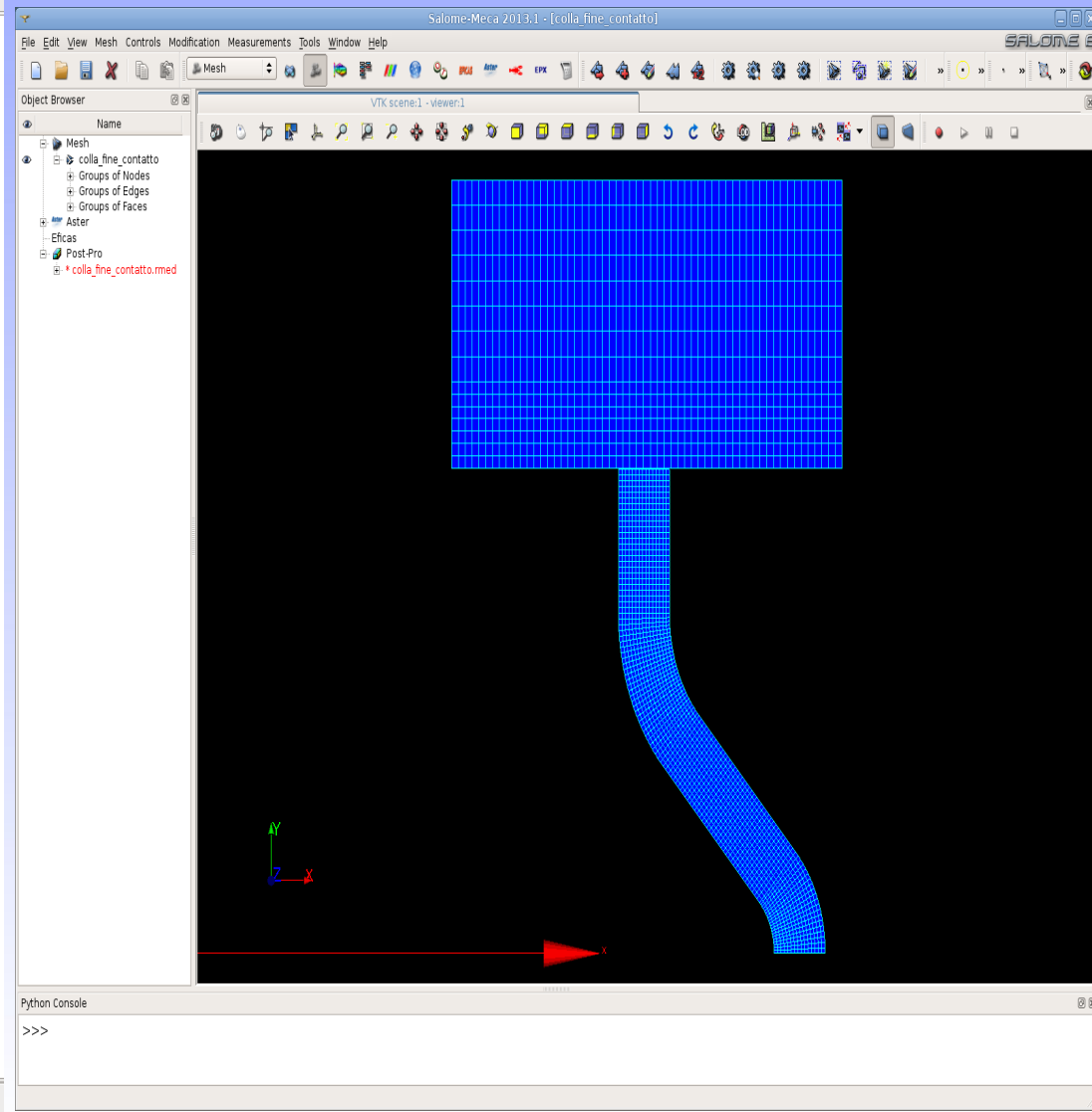

File di comando: risultati



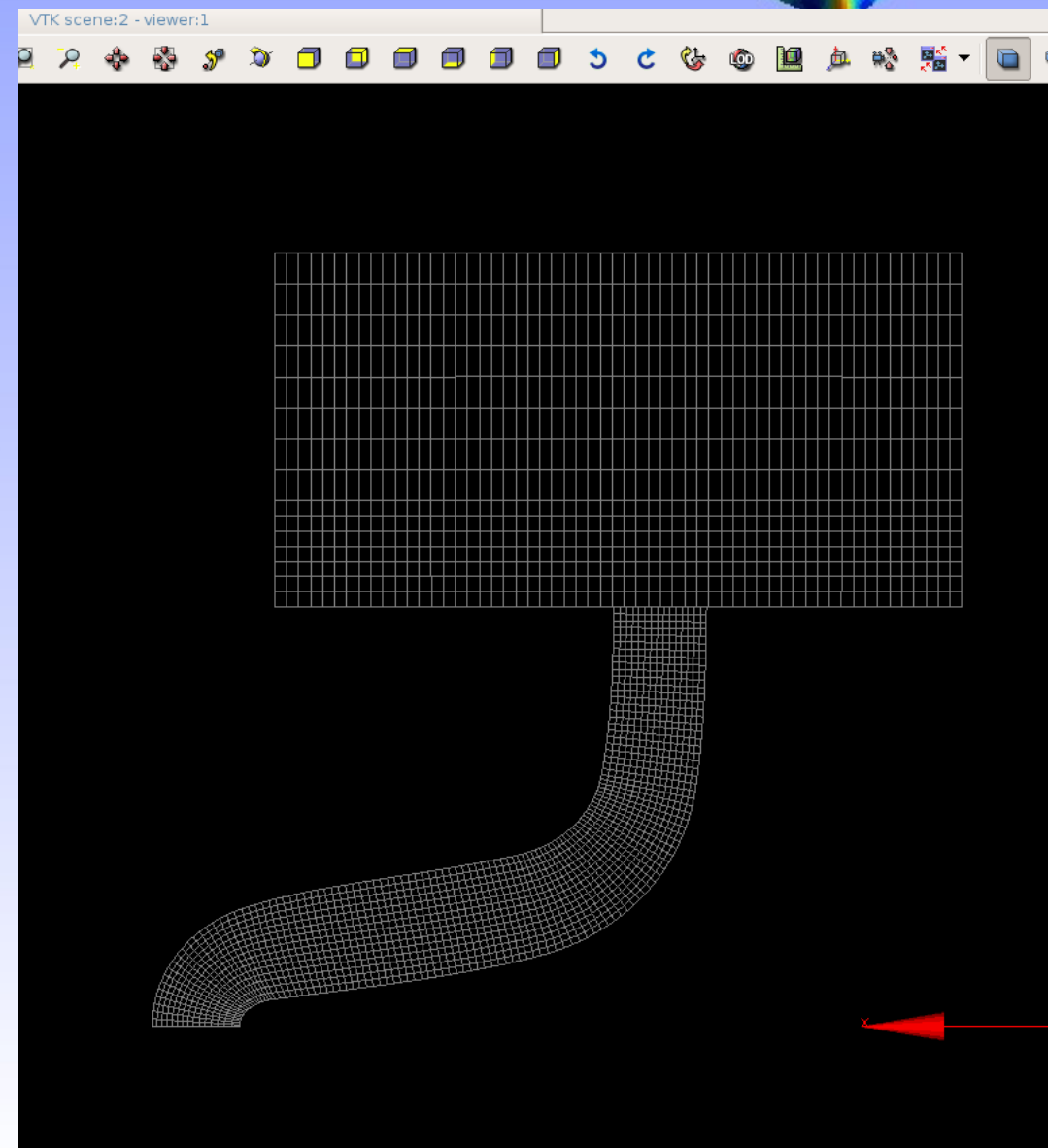
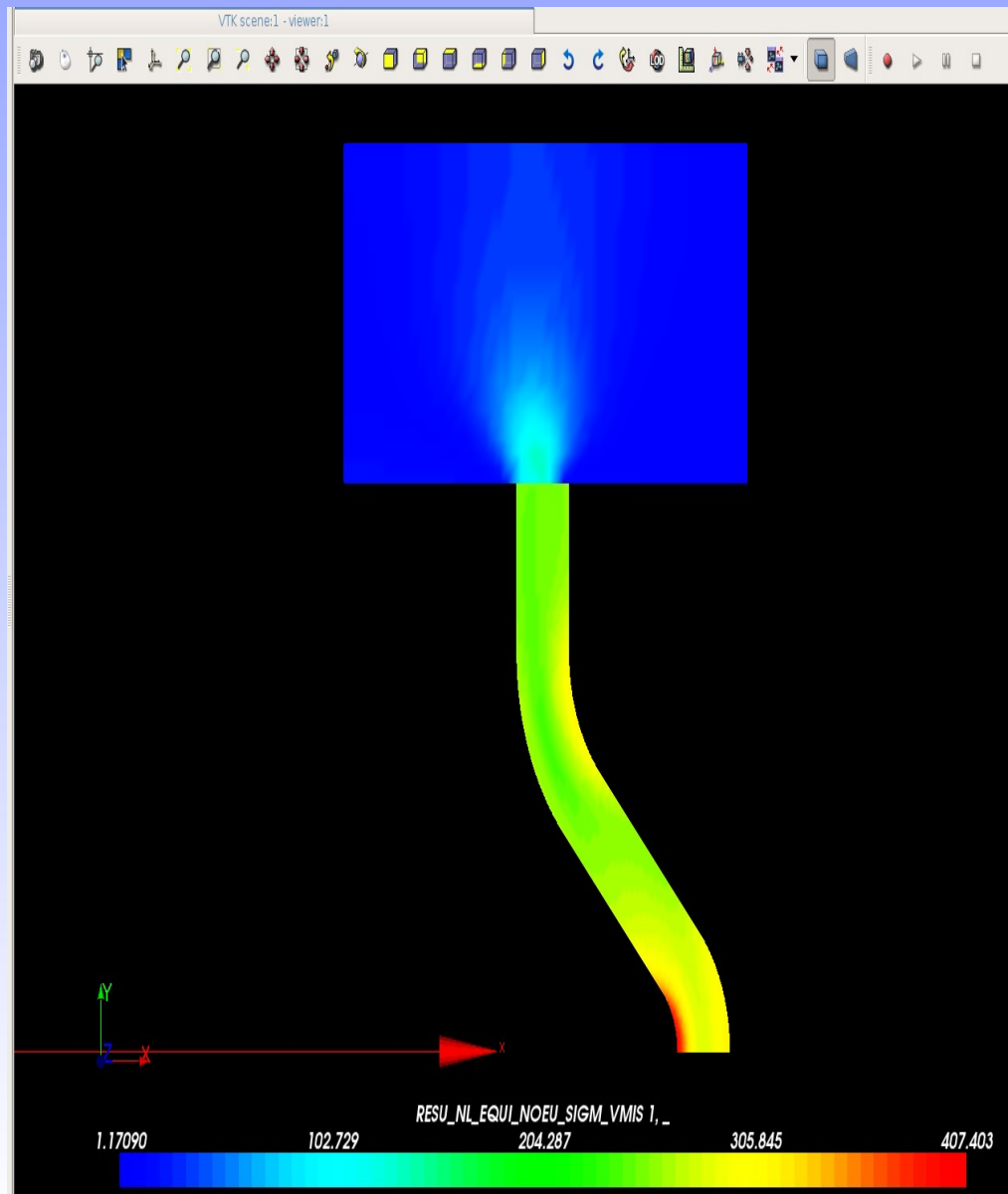
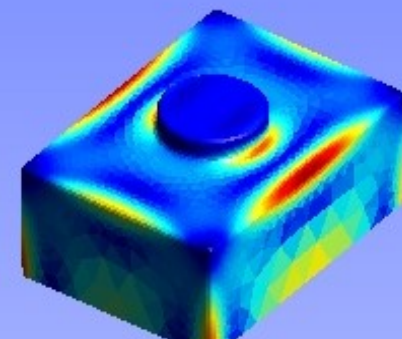
Esempi



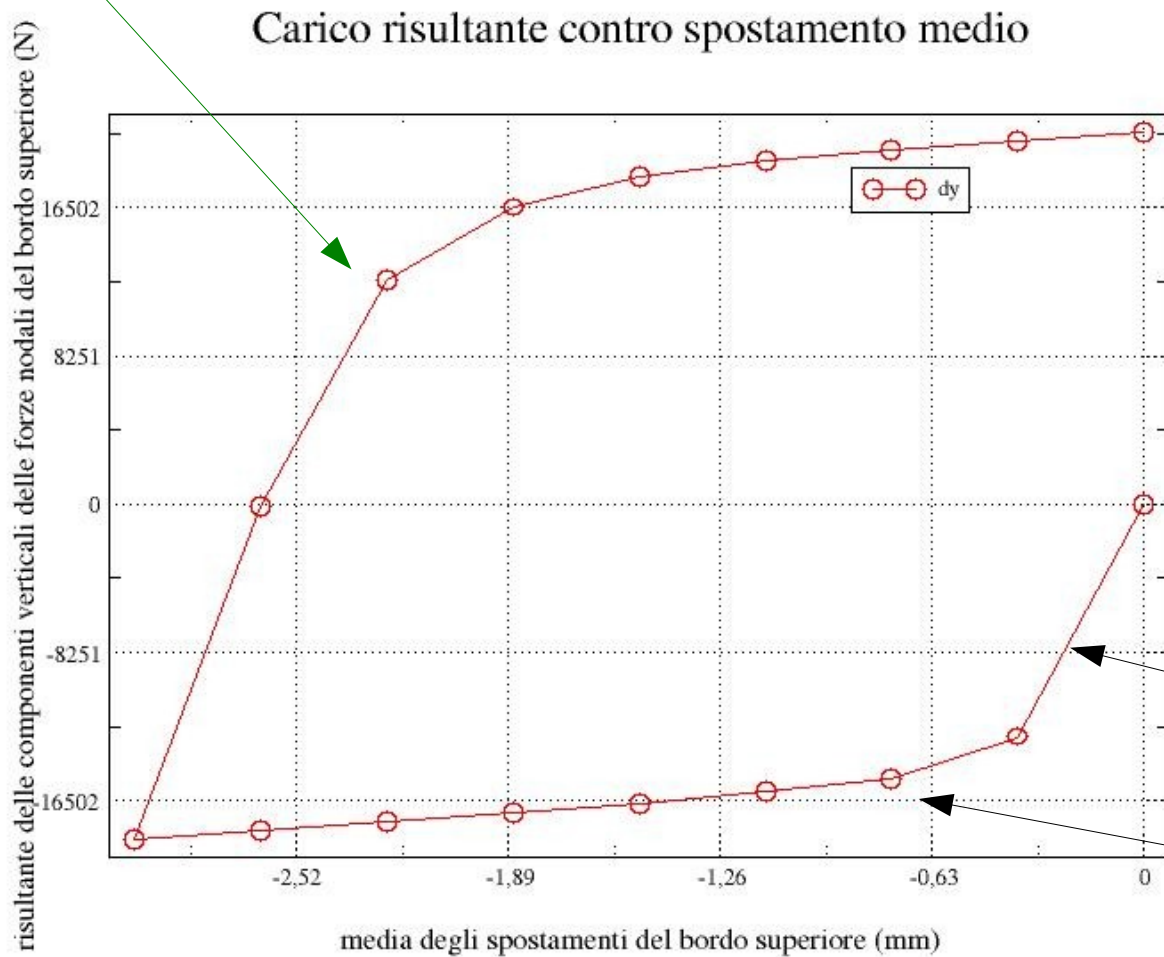
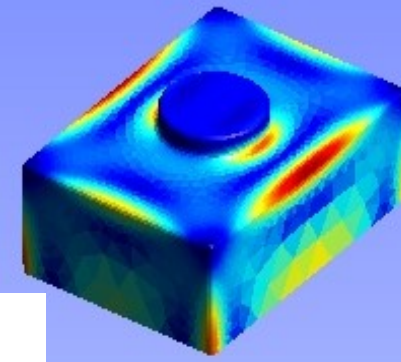
```
colla_fine_contatto.geo (~/.Documenti/studi/collassabil
File Modifica Visualizza Cerca Strumenti Documenti Ajuto
colla_fine_contatto.geo x
// definizione costanti
// lunghezza caratteristica 1
lc1= 0.1;
// raggio interno minore
r1= 17.5;
// raggio interno maggiore
r2= 22;
// spessore tubo
sp= 1.5;
// raggio interno prima curva
rc1= 3.0;
// raggio interno seconda curva
rc2= 1.5;
// semialtezza totale
h= 10;
// angolo prima rotazione
alfa= Pi/4;
// lunghezza primo spostamento
ls1= 4;
// definizione punti
Point(9) = {r2, 0, 0, lc1};
Point(10) = {r2+sp, 0, 0, lc1};
Line(100) = {9, 10};
Extrude {{0, 0, 1}, {r2-rc2, 0, 0}, Pi/4} {Line{100}; Layers{20, 1}; Recombine;}
Extrude {-Sin(alfa)*ls1, Cos(alfa)*ls1, 0} {Line{101}; Layers{40, 1}; Recombine;}
Extrude {{0, 0, 1}, {r2-rc2+(rc2+sp)*Cos(alfa)-ls1*Sin(alfa)+rc1*Cos(alfa),
(rc2+sp)*Sin(alfa)+ls1*Cos(alfa)+rc1*Sin(alfa), 0}, -Pi/2+alfa}
{Line{105}; Layers{30, 1}; Recombine;}
Extrude {0, 3, 0} {Line{109}; Layers{25, 1}; Recombine;}
// definizione gruppi
Physical Line(117) = {100};
Physical Line(118) = {113};
Physical Surface(119) = {104, 108, 112, 116};
// definizione pistone
// lunghezza caratteristica pistone
```



Esempi



Esempi

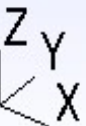


Curva di compressione

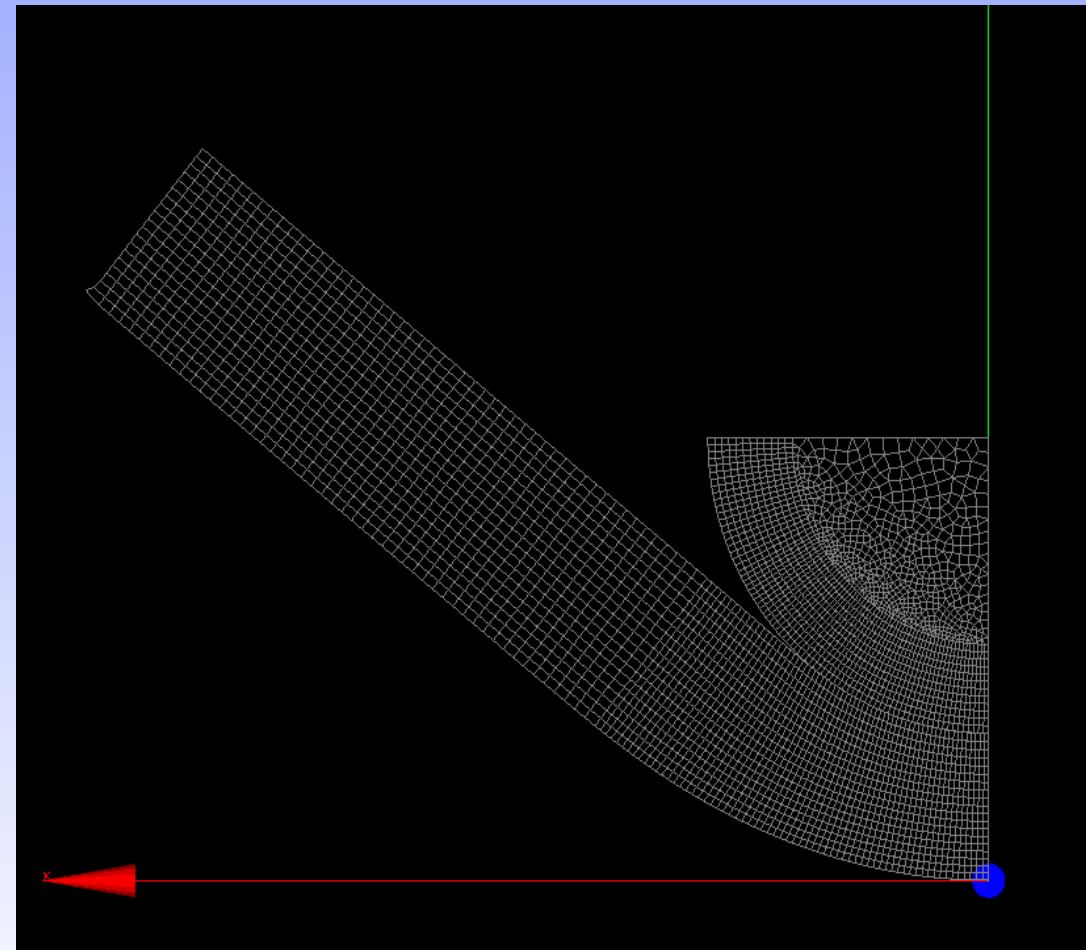
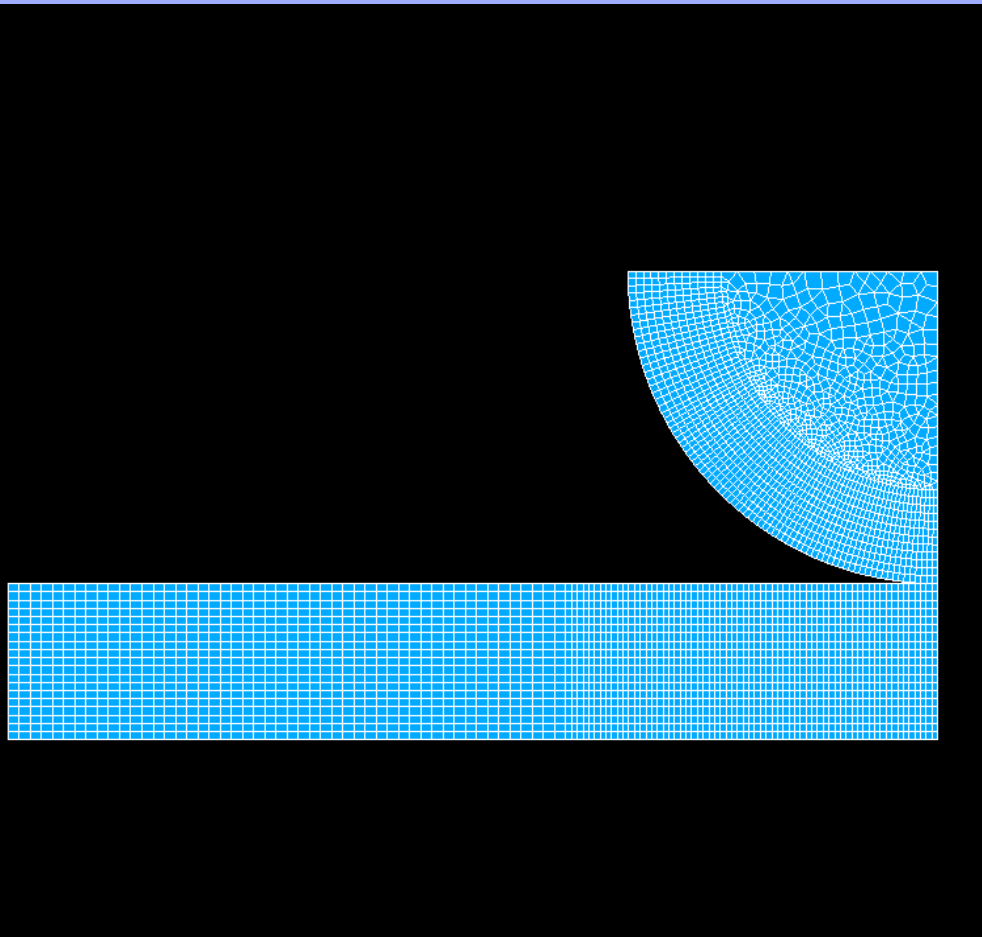
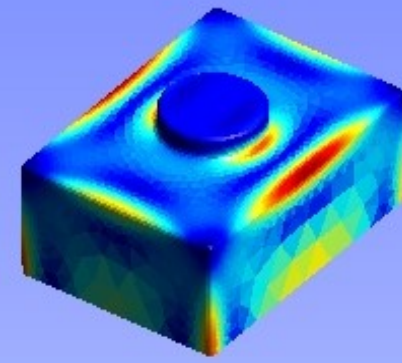
Tratto lineare

Tratto plastico

Durante lo schiacciamento si ha un tratto lineare iniziale, poi l'anello collassa e si deforma indefinitamente a carico praticamente costante.

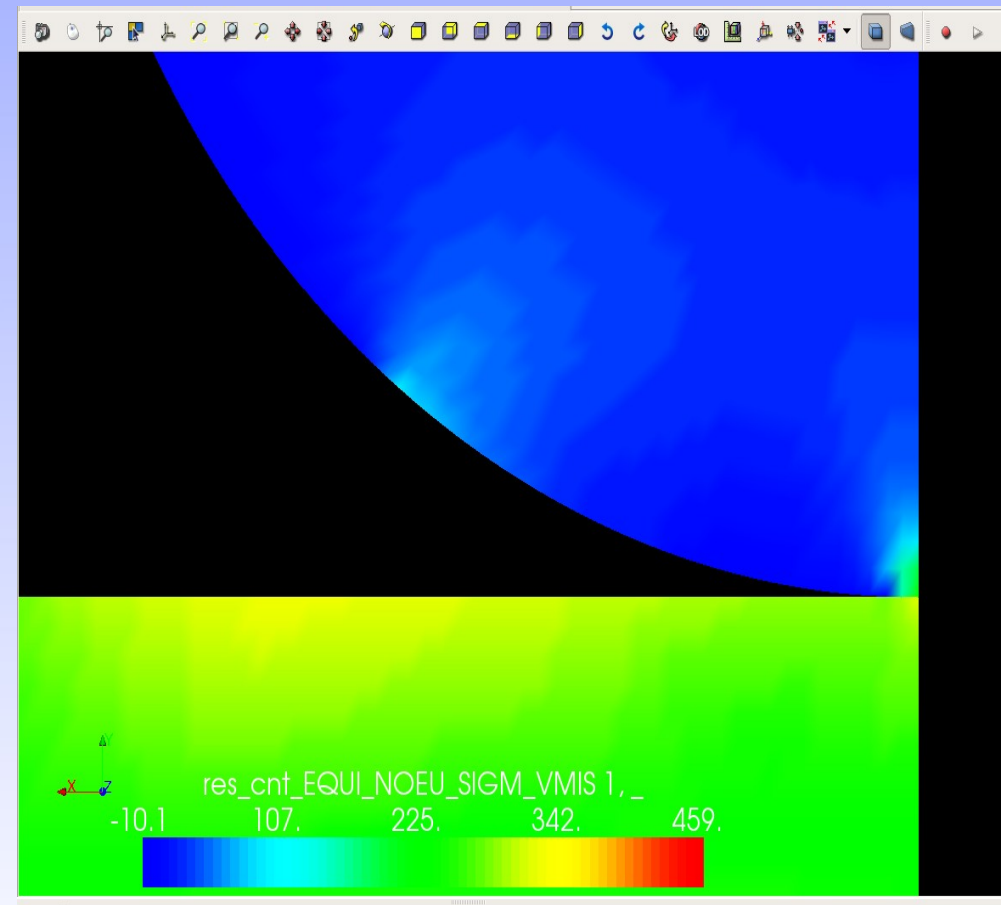
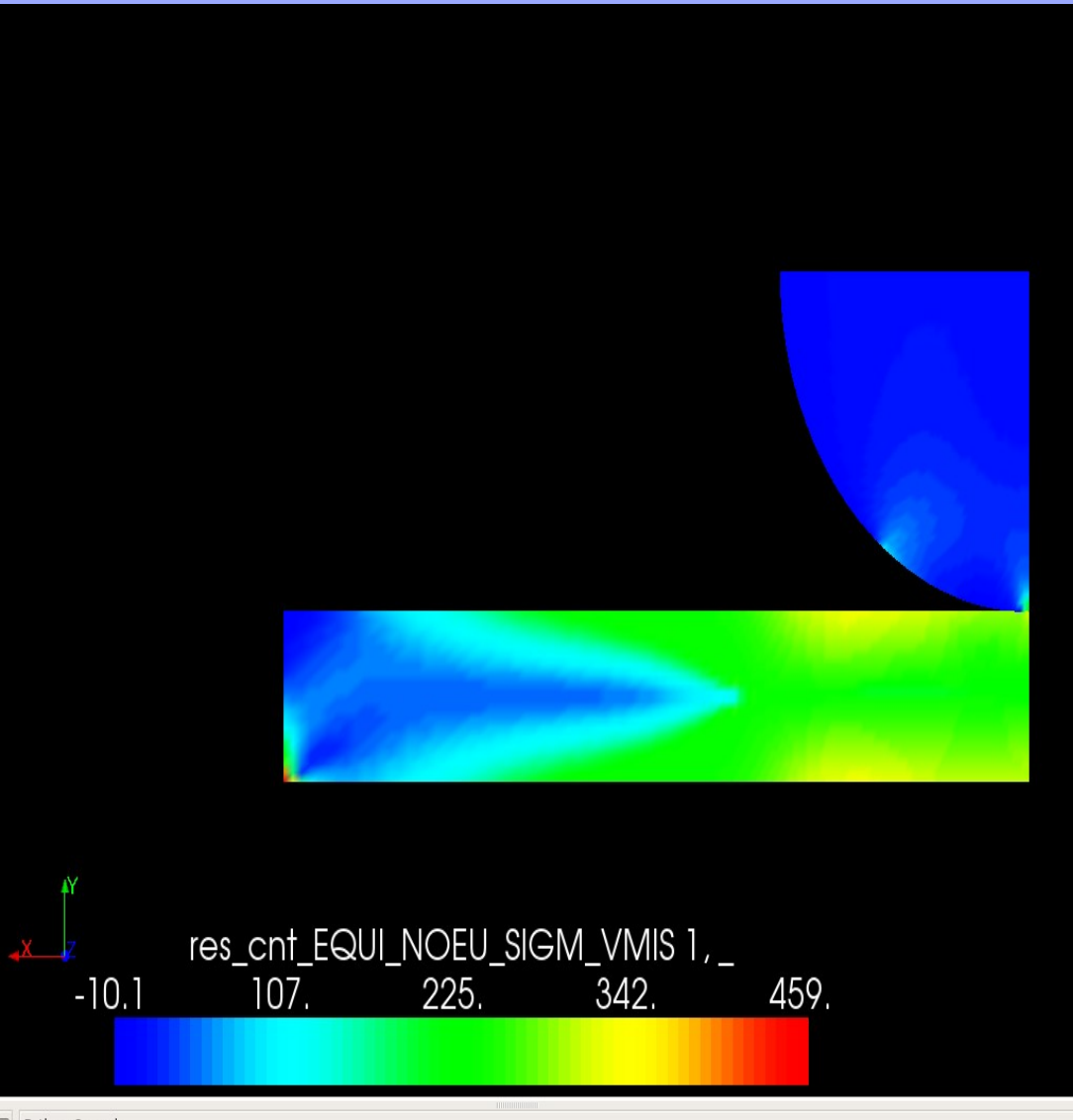
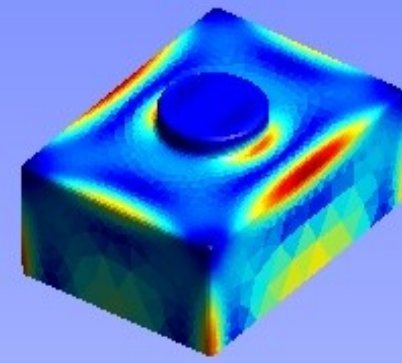


Esempi

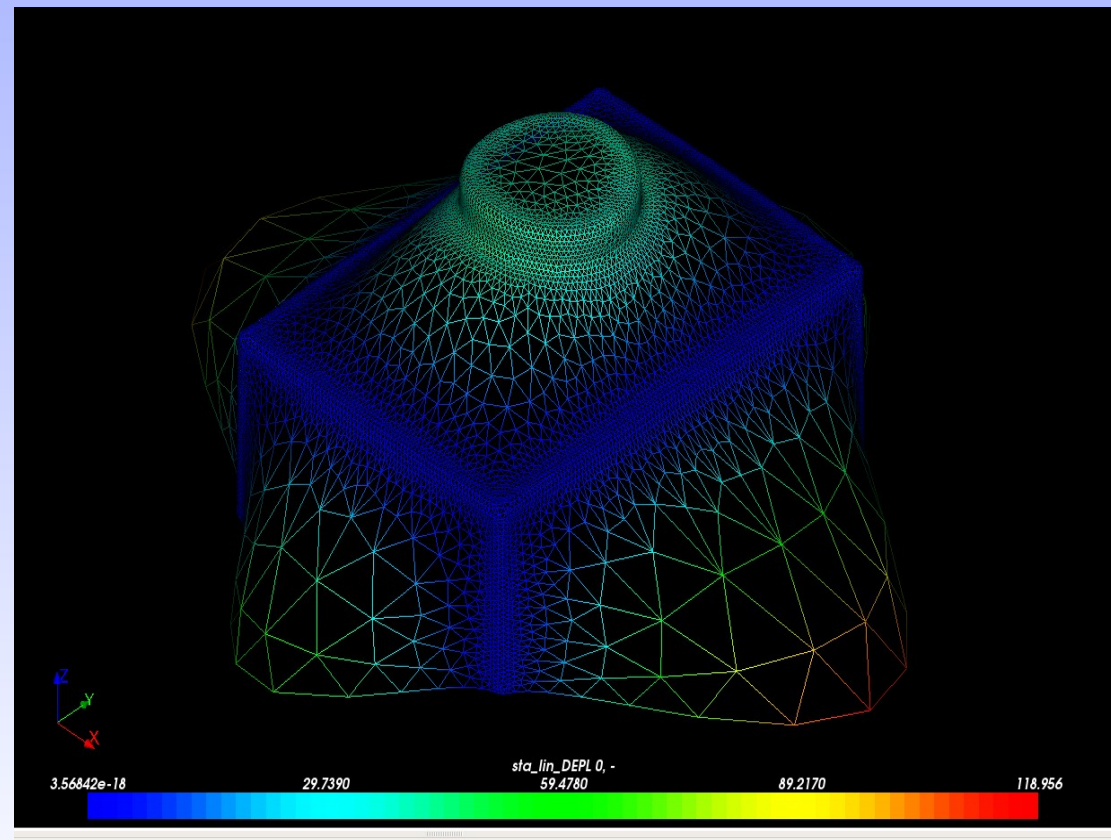
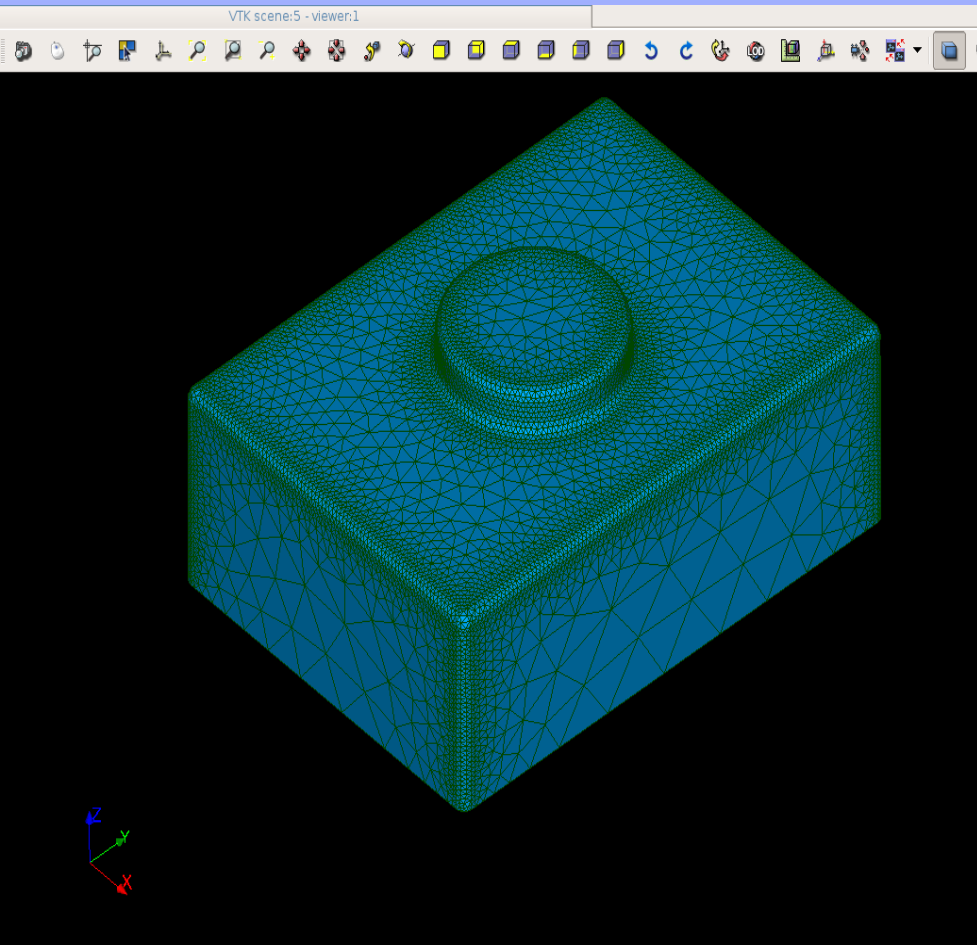
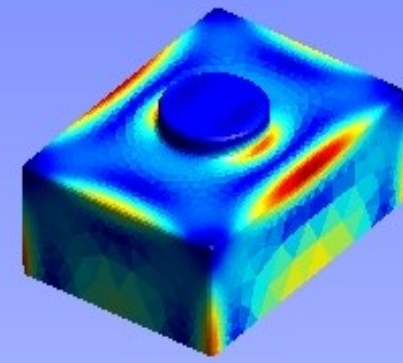


Z
Y
X

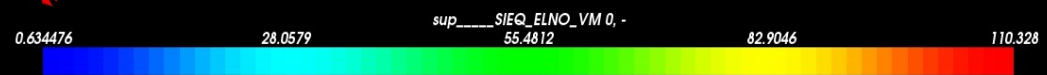
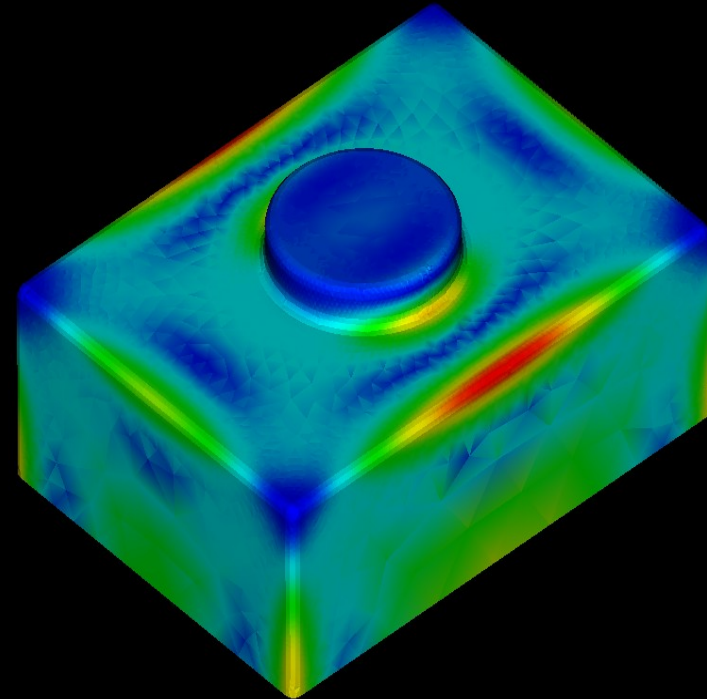
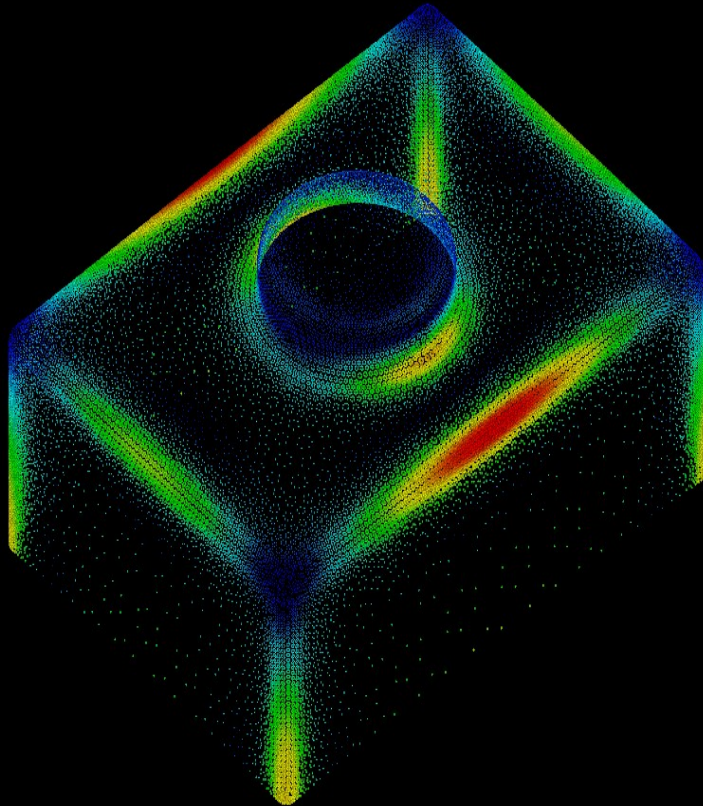
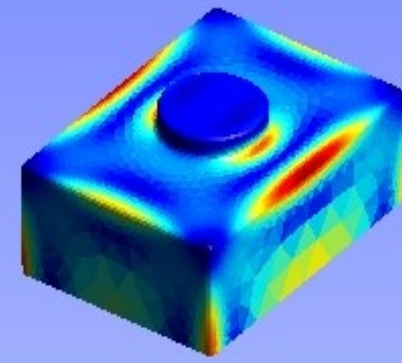
Esempi



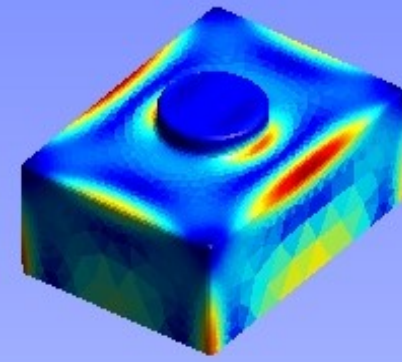
Esempi



Esempi



Siti di riferimento



- <http://www.caelinux.com/>
- <http://www.caelinux.org/wiki/>
- <http://www.code-aster.org/>
- <http://www.salome-platform.org/>
- <http://geuz.org/gmsh/>

