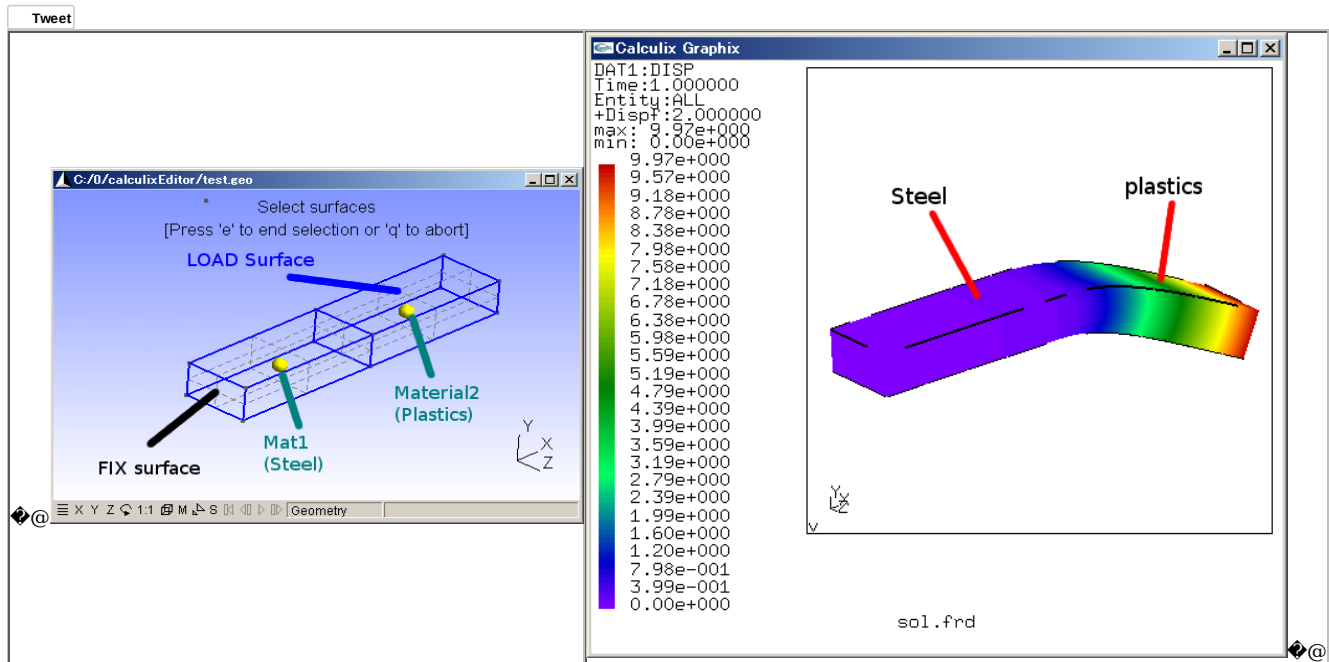


Gmsh Assembly -> Calculix (via unv)



gmsh -- [gmsh](#)

Calculix -- [Calculix](#) [WindowsVersion](#)

You need Python2.* . if you don't have it , please get and install it.
(I don't know the script will run with Python3)

- get **Media:unv2xUpd2.zip** from <http://www.caelinux.org/wiki/index.php/Proj:UNVConvert>
- extract zip file and
copy **unv2abaqus.py** to **unv2calculix.py**
copy **unv2x.py** to **unv2xc.py**
- open **unv2calculix.py** with TextEditor
change line 1
from **unv2x** import *
to
from **unv2xc** import *

change line 112
fil.write('%8d' % (lst.pop(0)))
to
fil.write((' %8d' +ls) % (lst.pop(0)))
- save and close TextEditor
- open **unv2xc.py** with TextEditor
change line 179-180

self.datasetsIds=[2411,2412,2467]
self.datasetsHandlers=[UNV2411Reader,UNV2412Reader,UNV2467Reader]

to
self.datasetsIds=[2411,2412,2467,**2477**]
self.datasetsHandlers=[UNV2411Reader,UNV2412Reader,UNV2467Reader,**UNV2467Reader**]
- save and close TextEditor

Make Assy with Gmsh

// Gmsh project created on Wed Aug 17 15:55:39 2011

lc = 5;

Point(1) = {0, 0, 0, lc};

Point(2) = {50, 0, 0, lc};

Point(3) = {50, 10, 0, lc};

Point(4) = {0, 10, 0, lc};

Point(5) = {100, 0, 0, lc};

Point(6) = {100, 10, 0, lc};

Line(1) = {1, 2};

Line(2) = {2, 3};

Line(3) = {3, 4};

Line(4) = {4, 1};

Line(5) = {2, 5};

Line(6) = {5, 6};

Line(7) = {6, 3};

Line Loop(8) = {4, 1, 2, 3};

Plane Surface(9) = {8};

Line Loop(10) = {2, -7, -6, -5};

Plane Surface(11) = {10};

Extrude {0, 0, 20} {

 Surface{9, 11};

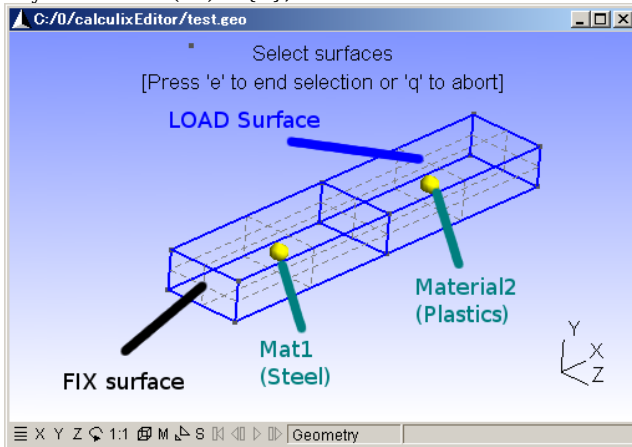
}

Physical Surface(56) = {20};

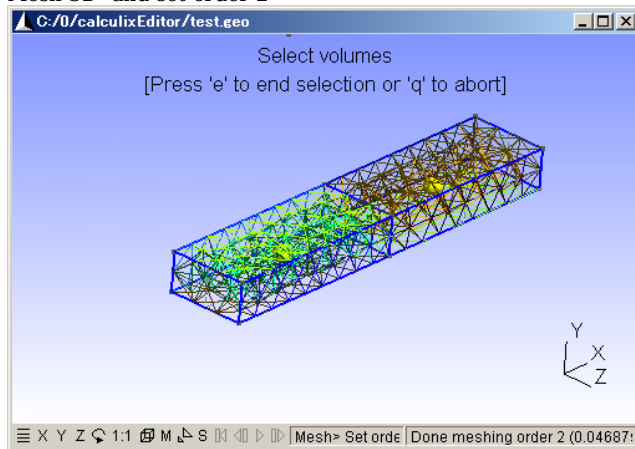
Physical Surface(57) = {46};

Physical Volume(58) = {1};

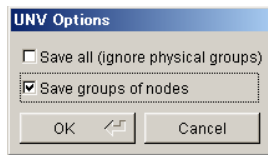
Physical Volume(59) = {2};



- Mesh 3D and set order 2



- Save as **gmsb.unv** (check **Save Group of nodes**)



-
- Run convert script

```
C:\0\calculixEditor
>c:\Python26\python.exe unv2calculix.py gmsh.unv gmsh
[1, 3, 5, 10, 2, 4, 6, 7, 8, 9]
UNV file converted successfully to Abaqus INP format
```

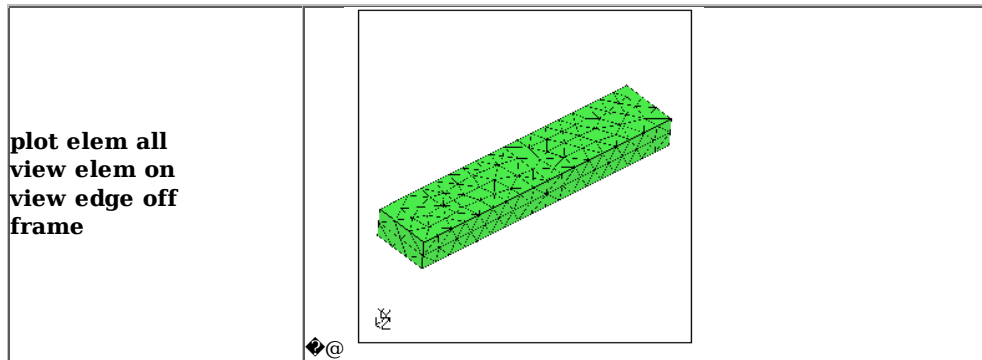
gmsh.unv is input file name , **gmsh** is output inp file name (it will make **gmsh.inp**)

- Run calculix

```
@cgx -c gmsh.inp
CalculiX Command - cgx -c gmsh.inp

C:\0\calculixEditor
>cgx -c gmsh.inp
on a WinXP machine, nodename MTRH0238, release 5,
```

- display mesh



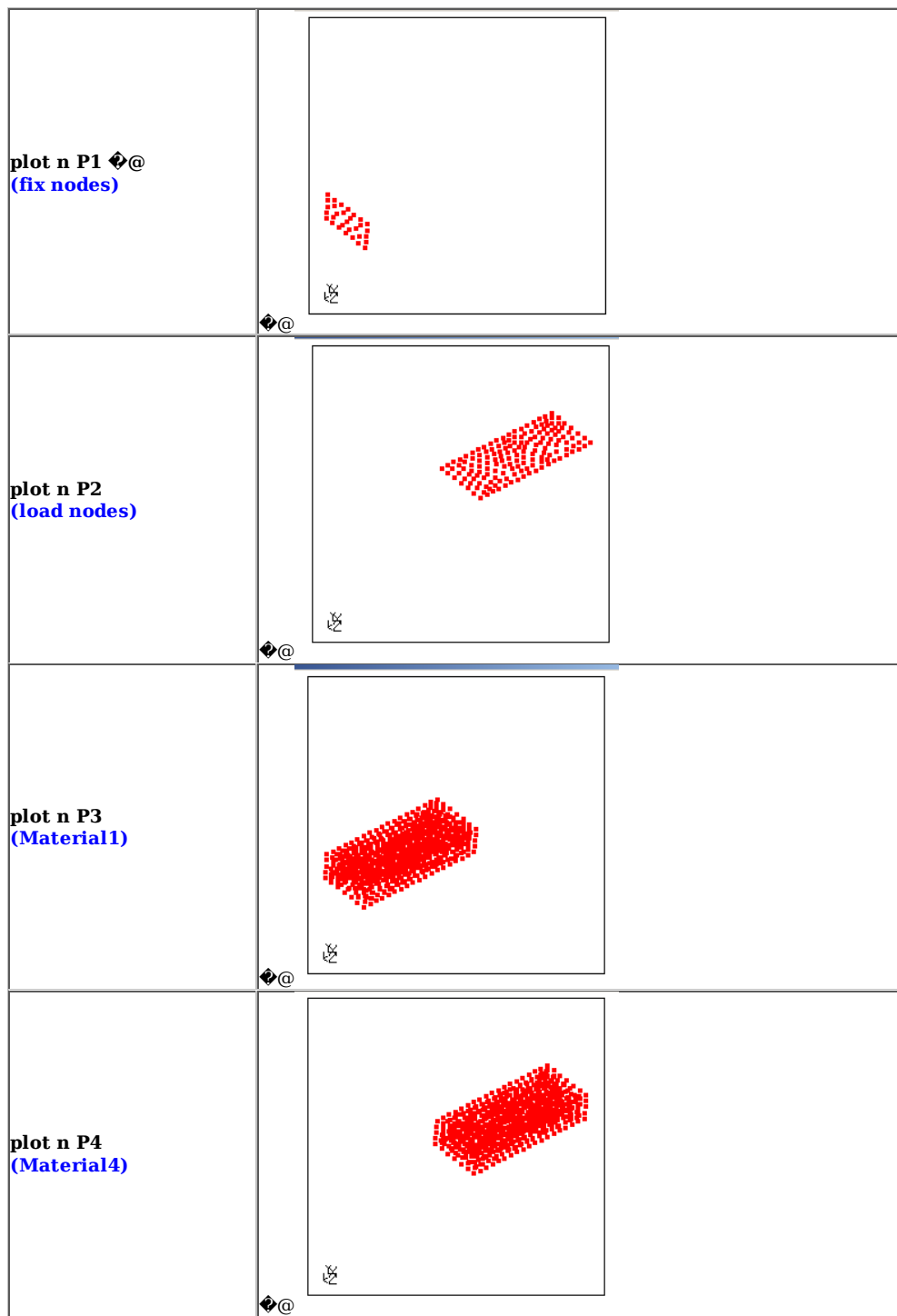
-
- check set

```
@prnt set
1 all stat:o n:1562 e:813 f:436 p:0 l:0 s:0 b:0 L:0 S:0 se:0 sh:0
2 NALL stat:c n:1562 e:0 f:0 p:0 l:0 s:0 b:0 L:0 S:0 se:0 sh:0
3 C3D10 stat:c n:0 e:813 f:0 p:0 l:0 s:0 b:0 L:0 S:0 se:0 sh:0
4 +C3D10 stat:c n:0 e:813 f:0 p:0 l:0 s:0 b:0 L:0 S:0 se:0 sh:0
5 PERMANENTGROUP1 stat:c n:31 e:0 f:0 p:0 l:0 s:0 b:0 L:0 S:0 se:0 sh:0
6 PERMANENTGROUP2 stat:c n:161 e:0 f:0 p:0 l:0 s:0 b:0 L:0 S:0 se:0 sh:0
7 PERMANENTGROUP3 stat:c n:819 e:0 f:0 p:0 l:0 s:0 b:0 L:0 S:0 se:0 sh:0
8 PERMANENTGROUP4 stat:c n:774 e:0 f:0 p:0 l:0 s:0 b:0 L:0 S:0 se:0 sh:0
```

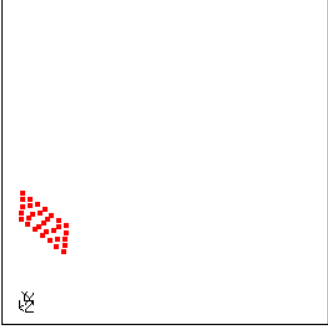
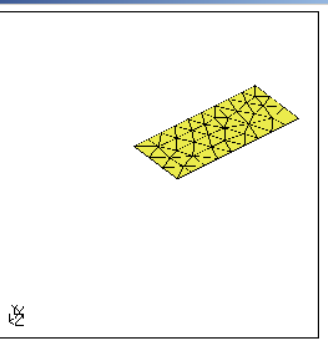
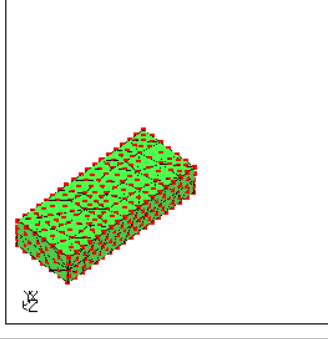
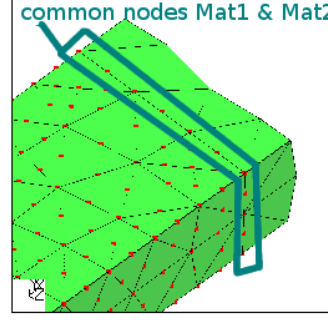
- So set names are too long , copy sets for convenience.

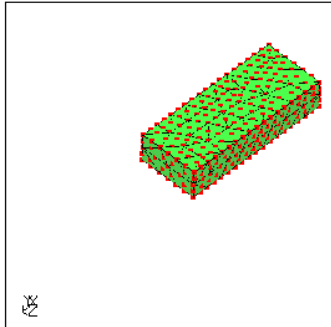
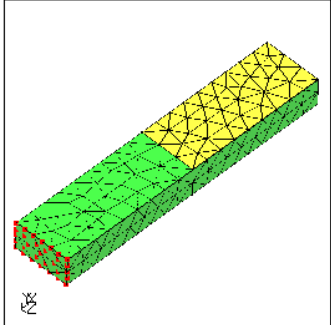

```
seta P1 set PERMANENTGROUP1
seta P2 set PERMANENTGROUP2
seta P3 set PERMANENTGROUP3
seta P4 set PERMANENTGROUP4
```

display and check each sets



- then make surface set and element sets form these node sets

seta FIX set P1 plot n FIX (FIX is fix node sets)	 ⚡@
comp P2 down seta LOAD set P2 plot f LOAD (LOAD is face sets)	 ⚡@
seta M1 set P3 setr M1 set P4 comp M1 up plot e M1 plus n P3 (M1 is material 1 sets) (because there are common nodes between P3 and P4 , 'setr M1 set P4' is needed)	 ⚡@
⚡@(If not using 'setr M1 set P4' elements sets is not correct) plot e M1 plus n P3	 ⚡@

<pre>seta M2 set P4 setr M2 set P3 comp M2 up plot e M2 plus n P4</pre>	
<pre>plot e M1 plus e M2 plus n FIX plus f LOAD</pre> <p>(confirm each sets)</p>	
<pre>send all abq send M1 abq names send M2 abq names send LOAD abq pres 1 send FIX abq names</pre>	

- sol.inp is like this

```
*INCLUDE, INPUT=all.msh
*INCLUDE, INPUT=FIX.nam
*INCLUDE, INPUT=M1.nam
*INCLUDE, INPUT=M2.nam

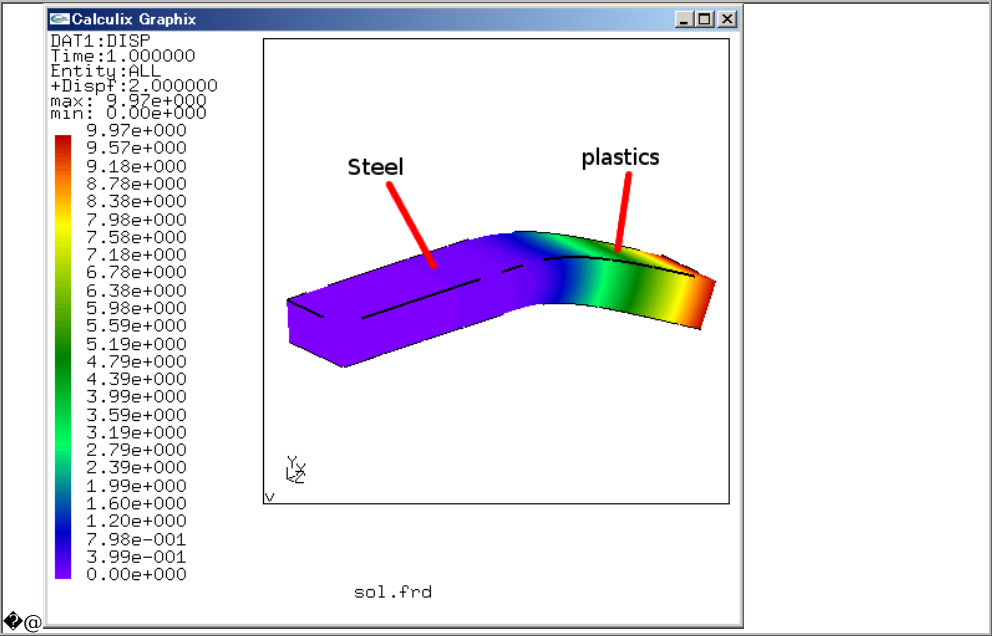
*MATERIAL, NAME=STEEL
*ELASTIC
200000 , 0.3

*MATERIAL, NAME=PLA
*ELASTIC
1000 , 0.35

*SOLID SECTION , Elset=EM1 , Material=STEEL
*SOLID SECTION , Elset=EM2 , Material=PLA
*STEP
*STATIC
*BOUNDARY
NFIx,1,3,0
*DLOAD
*include,input=LOAD.dlo
*NODE PRINT,NSET=Nall
U
*EL PRINT,ELSET=Eall
S
*NODE FILE
U
*EL FILE
S
*END STEP
```

ccx sol

cgx sol.frd



[return](#)

002071 now: 000001

///