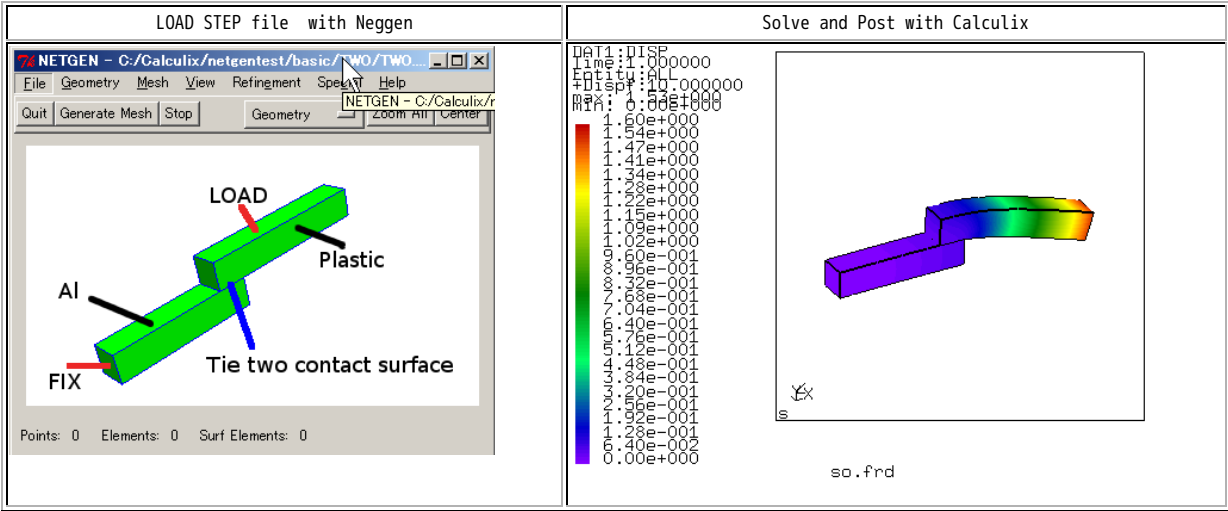
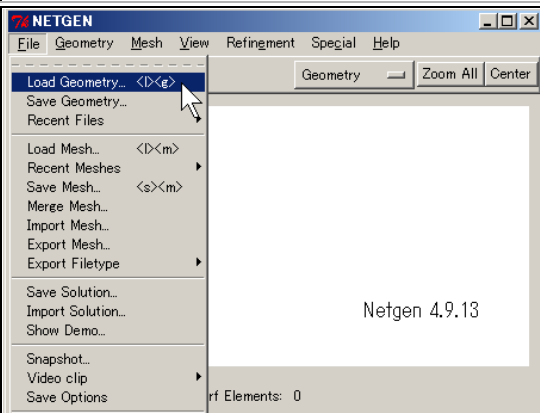
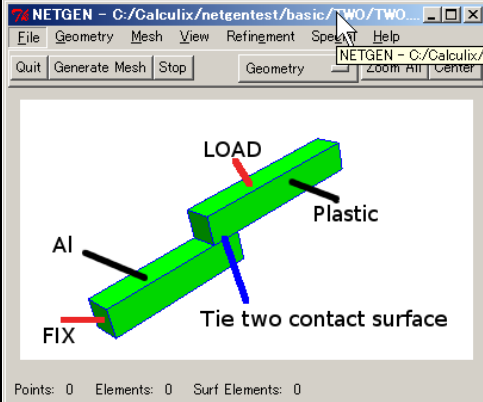


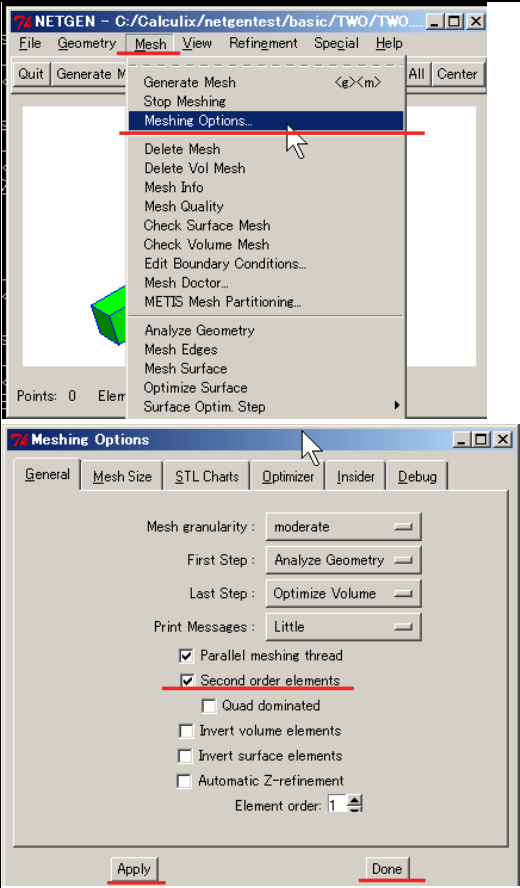
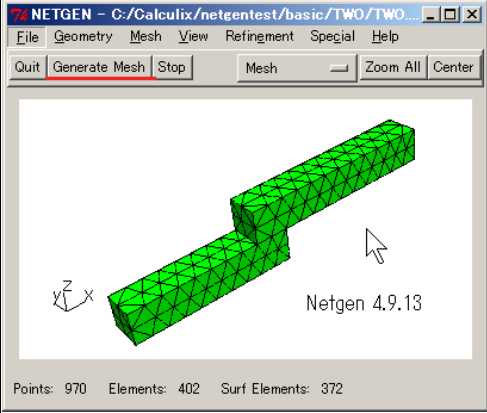
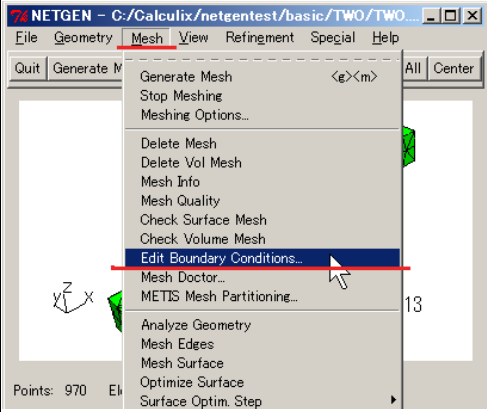
CAD Ass'y Model(.STEP) --> Netgen(.VOL) --> Calculix(.INP)

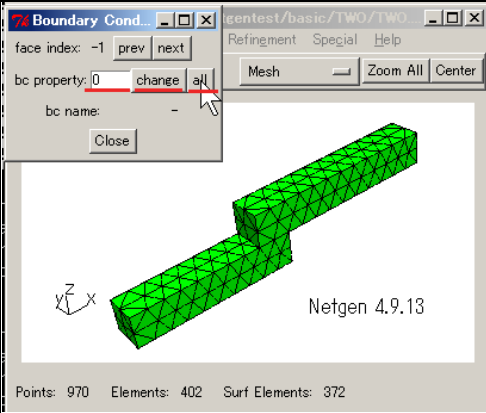
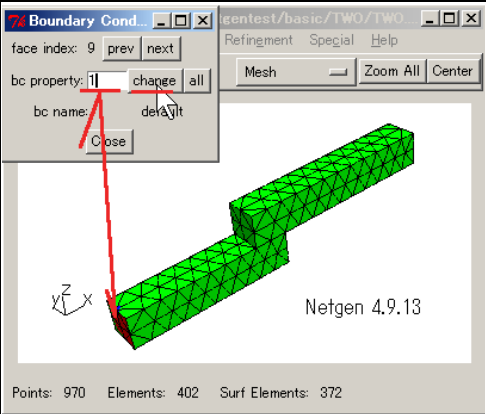
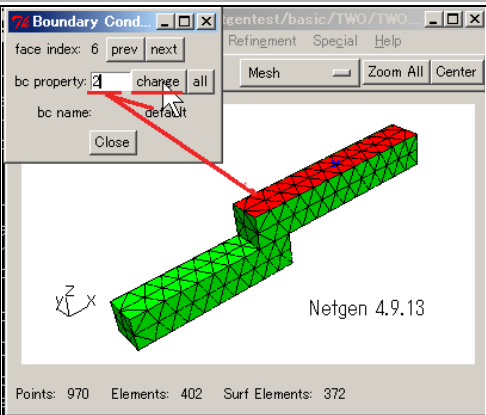
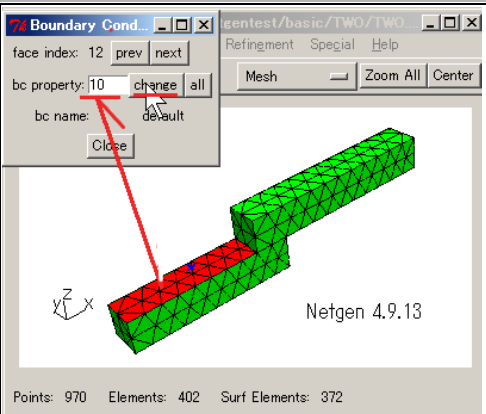
with [vol2ci.py](#) (Version 0.023 2011/Nov/28 : bug fix for NetgenSTL-'surfaceelements'g ')

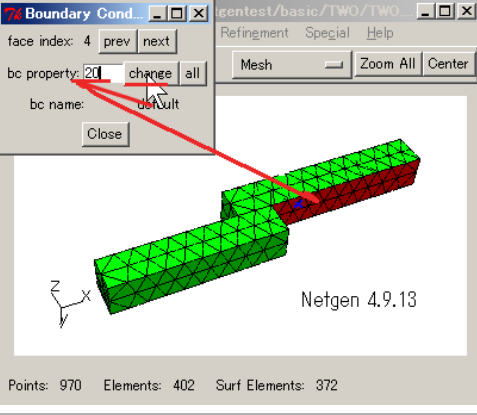
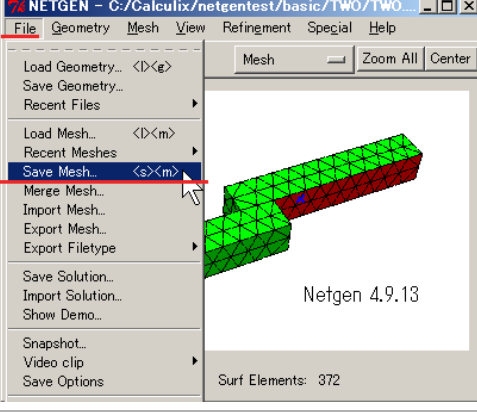
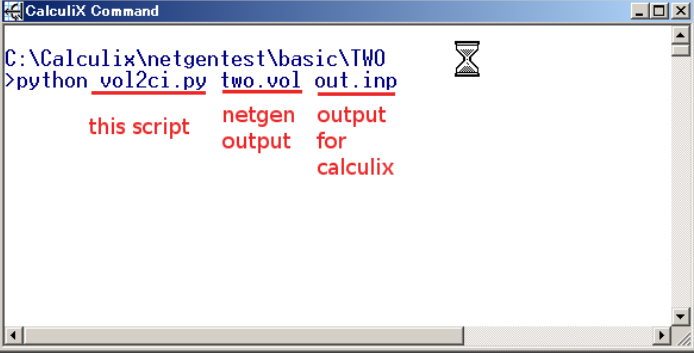
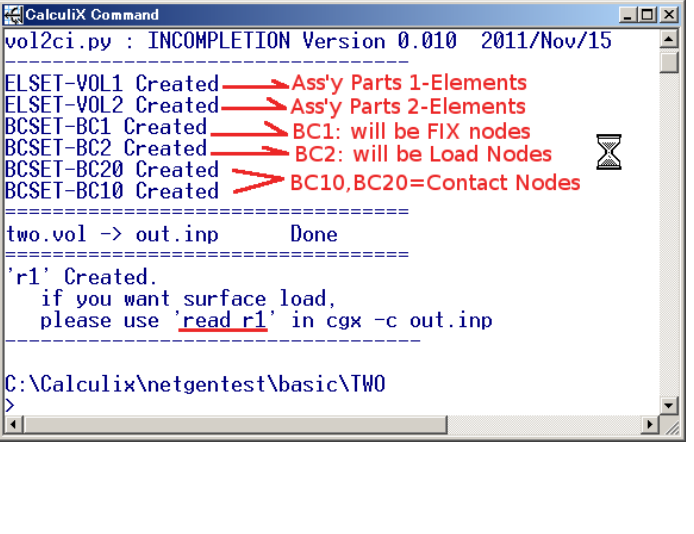
Tweet

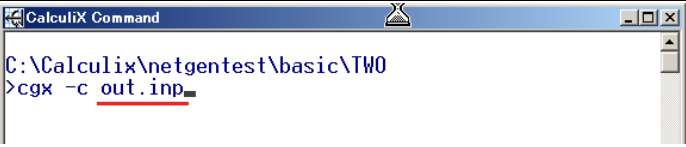
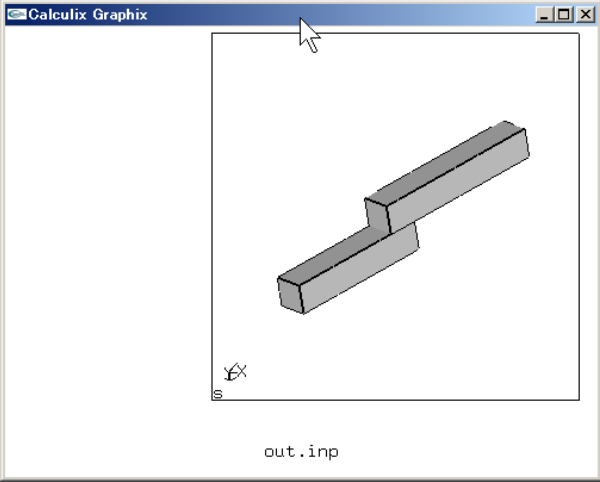
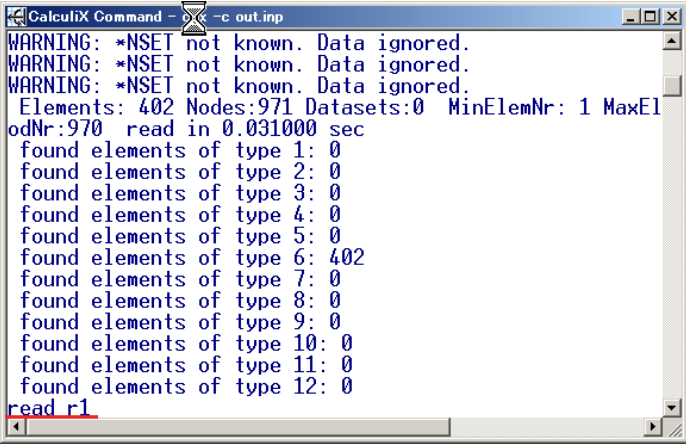
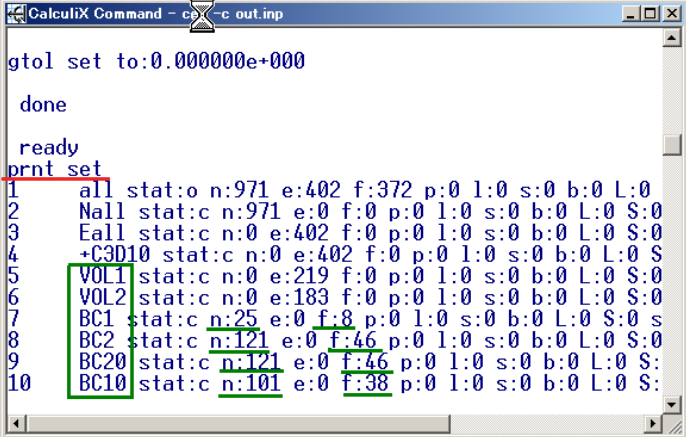


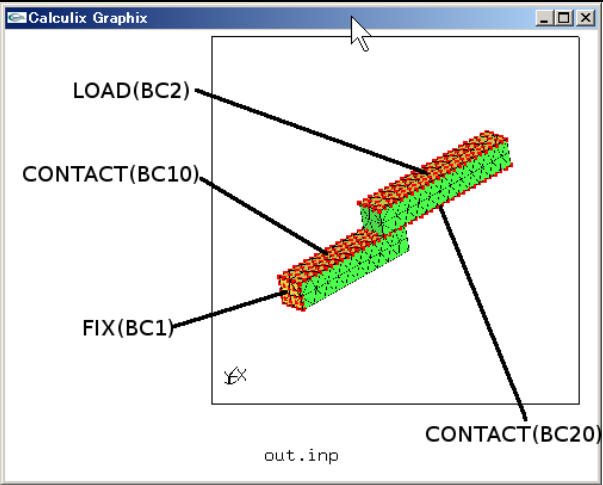
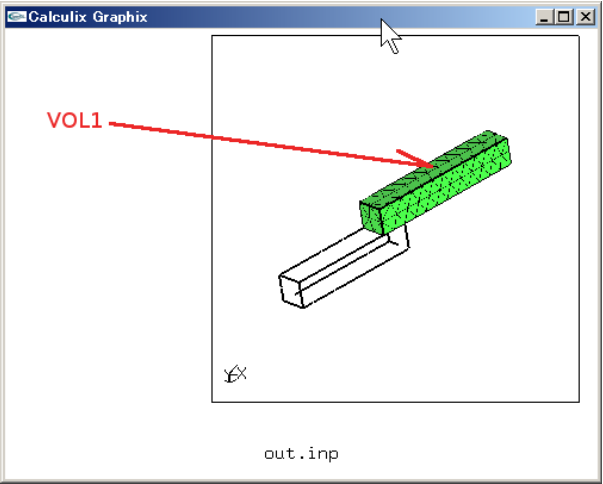
| command | Screen | Comment |
|-----------------------------------|---|--|
| Run Netgen File / LoadGeometry |  | Load CAD file(STEP, IGES etc) |
| |  | Make a plan to decide Boundary Condtion. |

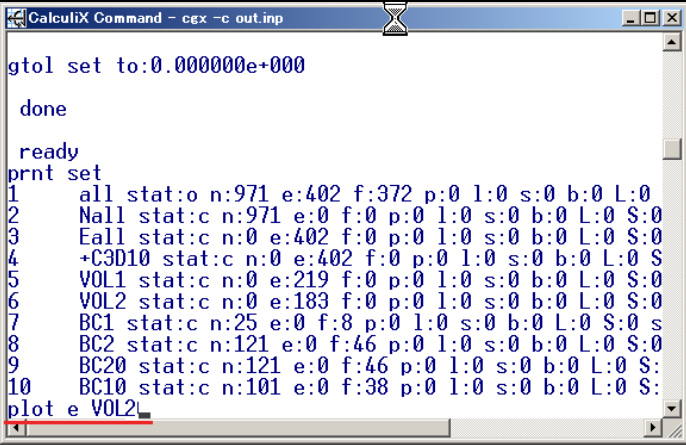
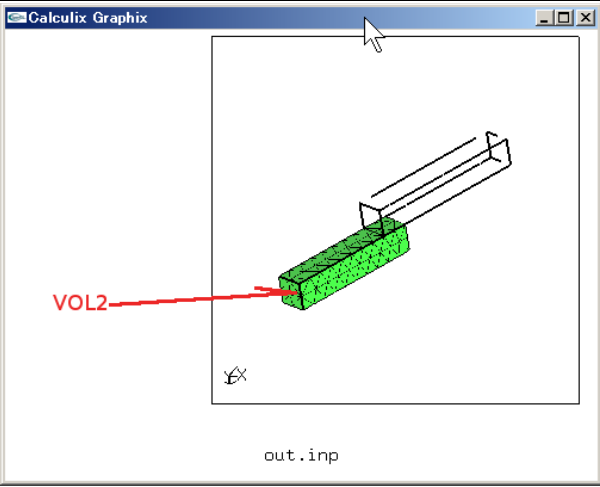
| | | |
|--|---|---------------------|
| Mesh/MeshOptions /secondOrderElements |  <p>The screenshot shows the NETGEN application window. The 'Mesh' menu is open, and 'Meshing Options...' is highlighted. Below it, the 'Meshing Options' dialog box is displayed. In the 'General' tab, 'Second order elements' is checked, and 'Element order' is set to 1. Other options like 'Parallel meshing thread' and 'Automatic Z-refinement' are also visible.</p> | I want to use C3D10 |
| Generate Mesh |  <p>The screenshot shows the NETGEN application window with a 3D model of a mechanical part. The model is rendered with a green mesh. The status bar at the bottom indicates 'Points: 970 Elements: 402 Surf Elements: 372'.</p> | |
| Mesh/Edit Boundary Conditions |  <p>The screenshot shows the NETGEN application window. The 'Mesh' menu is open, and 'Edit Boundary Conditions...' is highlighted. The status bar at the bottom shows 'Points: 970'.</p> | |

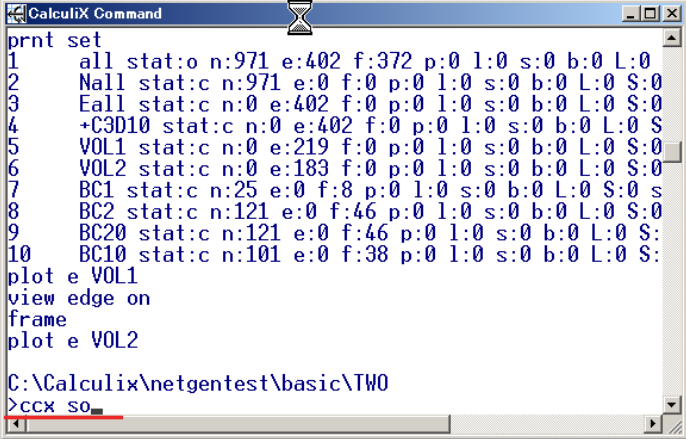
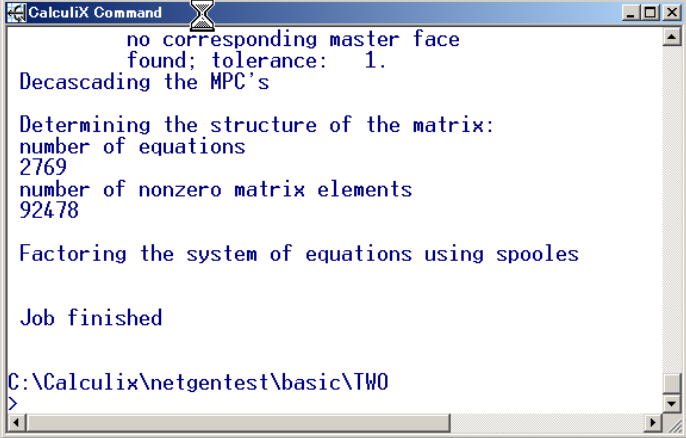
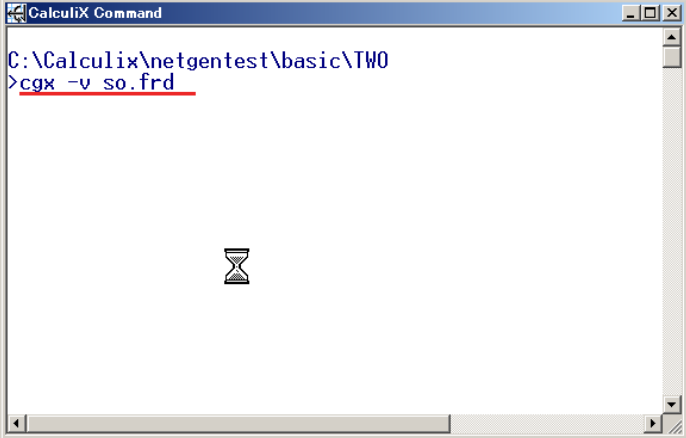
| | | |
|---|---|--|
| 0/all/change |  | (reset all surface's BC) |
| (Select FIX surface) 1/Change |  | (set 'FIX' surface's BC to '1') (of course , you can use other number) |
| (Select LOAD surface) 2/Change |  | (set 'LOAD' surface's BC to '2') |
| (Select one Contact Surface) 10/Change |  | |

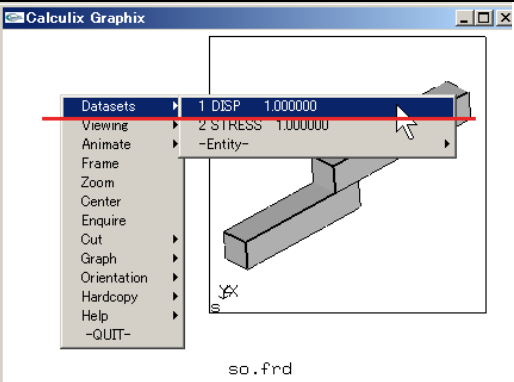
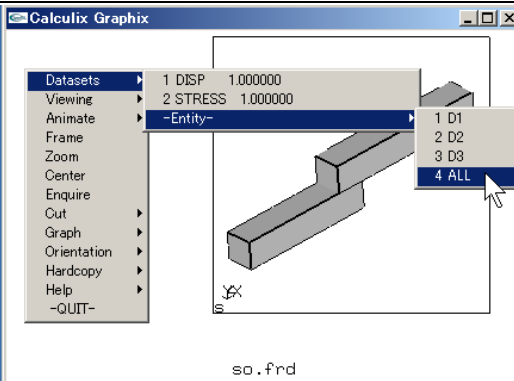
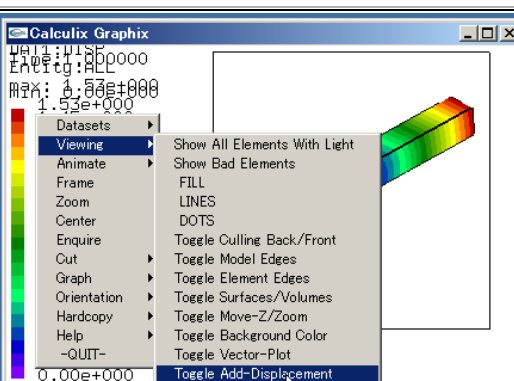
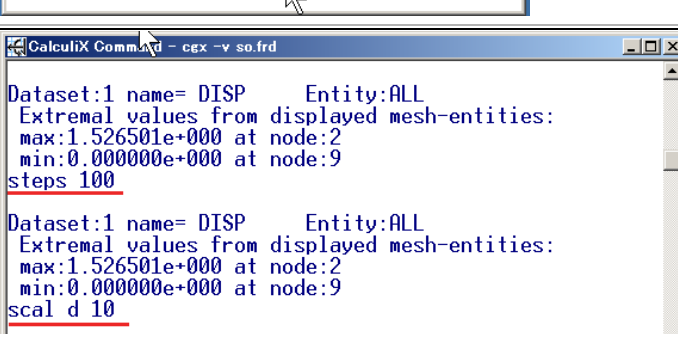
| | | |
|---|--|--|
| <p>(Select another Contact Surface) 20/Change</p> |  | |
| <p>File / SaveMesh</p> |  | <p>for example , two.vol</p> |
| <pre>>python vol2ci.py two.vol out.inp</pre> |  | <p>run Python script <u>vol2ci.py</u></p> <p><u>two.vol</u> is netgen mesh file made by above.</p> <p><u>out.inp</u> is filename made by script. (any name you like)</p> |
| |  | <p>vol2ci.py will make out.inp that include :</p> <ol style="list-style-type: none"> 1) Node info such as *NODE,Nset=Na11 1, 40, 0, 20 2, 50, 0, 20 2) ELEMENT info such as *ELEMENT, TYPE=C3D10, ELSET=Eall 1,34,137,143,117,439,484,473,358,752,751 3) ELSET set such as *ELSET,ELSET=VOL1 1, 2, 4) NODE set (for Boundary Condition) *NSET,NSET=BC1 9, |

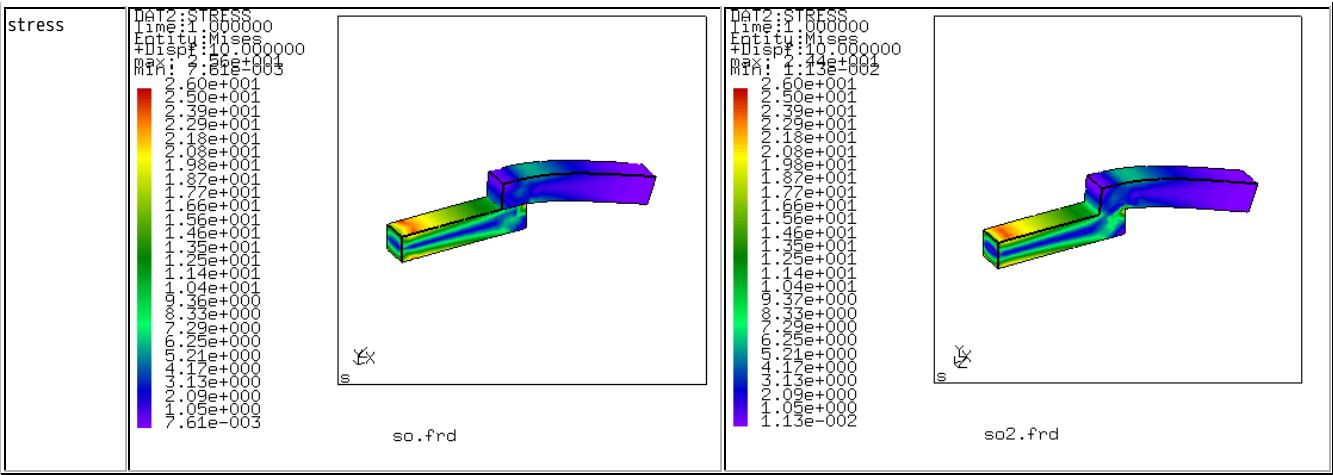
| | | |
|-----------------|--|--|
| | | 12, |
| >cgx -c out.inp |  | 5) r1 for using in cgx |
| |  | |
| read r1 |  | <p>r1 is ..</p> <p>plus n BC1 comp BC1 down comp BC1 down plus f BC1 send BC1 abq sur send BC1 abq pres 1.0</p> <p>Role of r1 is</p> <p>1) disp node BC (plus n BC1) 2) make surface set (comp BC1 down , comp BC1 down) 3) disp surface BC (plus f BC1) 4) make surface set for contact (send BC1 abq sur) 5) make surface set for press (send BC1 abq pres 1.0)</p> |
| prnt set |  | <p>VOL# is Parts set BC# is BoundaryCondition's No</p> |

| | | |
|-------------|--|--|
| | <div><div>Calculix Graphix</div><div></div></div> | confirm BC's |
| plot e VOL1 | <div><div>Calculix Command - cgx -c out.inp</div><div><pre>gtol set to:0.000000e+000 done ready prnt set 1 all stat:o n:971 e:402 f:372 p:0 l:0 s:0 b:0 L:0 2 Nall stat:c n:971 e:0 f:0 p:0 l:0 s:0 b:0 L:0 S:0 3 Eall stat:c n:0 e:402 f:0 p:0 l:0 s:0 b:0 L:0 S:0 4 +C3D10 stat:c n:0 e:402 f:0 p:0 l:0 s:0 b:0 L:0 S 5 VOL1 stat:c n:0 e:219 f:0 p:0 l:0 s:0 b:0 L:0 S:0 6 VOL2 stat:c n:0 e:183 f:0 p:0 l:0 s:0 b:0 L:0 S:0 7 BC1 stat:c n:25 e:0 f:8 p:0 l:0 s:0 b:0 L:0 S:0 s 8 BC2 stat:c n:121 e:0 f:46 p:0 l:0 s:0 b:0 L:0 S:0 9 BC20 stat:c n:121 e:0 f:46 p:0 l:0 s:0 b:0 L:0 S:0 10 BC10 stat:c n:101 e:0 f:38 p:0 l:0 s:0 b:0 L:0 S:0 plot e VOL1</pre></div></div> | netgen add Number to each parts automatically , So it is need to confirm each part's set. |
| | <div><div>Calculix Graphix</div><div></div></div> | found 'VOL1 is upeer parts.' |

| | | |
|--------------------------------------|--|--|
| plot e VOL2 |  | |
| |  | found 'VOL2 is lower parts.' |
| quit cgx | confirm output file BC1.dlo --- press surface set for *DLOAD . BC1.sur --- surface set BC2.dlo BC2.sur BC10.dlo BC10.sur BC20.dlo BC20.sur | |
| prepare solver file(named so.inp) | <pre> *INCLUDE, INPUT=out.inp *include,INPUT=BC10.sur *include,INPUT=BC20.sur *MATERIAL, Name=AL *ELASTIC 71000 , 0.35 *MATERIAL, Name=PLA *ELASTIC 1000 , 0.30 *SOLID SECTION,MATERIAL=PLA , ELSET=VOL1 *SOLID SECTION,MATERIAL=AL , ELSET=VOL2 *TIE,POSITION TOLERANCE=1 SBC10,SBC20 *AMPLITUDE,NAME=A1 </pre> | with *TIE command , fix two surface (I don't know TOLERANCE=1 is adequate or |

| | | |
|----------------|--|---|
| | <pre>0,0.1 *STEP *STATIC *BOUNDARY BC1,1,3,0 *DLOAD,AMPLITUDE=A1 *INCLUDE, INPUT=BC2.dlo *NODE FILE U *EL FILE S *END STEP</pre> | <p>not)</p> <p>SBC10 is described in BC10.sur as *SURFACE, NAME=SBC10</p> |
| >ccx so |  | |
| |  | |
| >cgx -v so.frd |  | |

| | | |
|--------------------------------------|---|--|
| (datasets/Disp) | <div></div> | |
| (datasets/Entity/ALL) | <div></div> | |
| (Viewing Toggle Add-Displacement) | <div></div> | |
| steps 100 scal d 10 | <div></div> | |



[return](#)