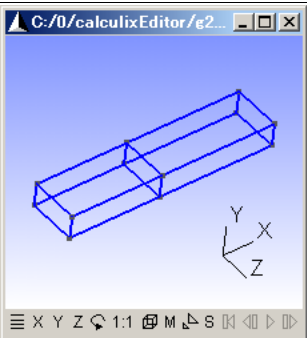
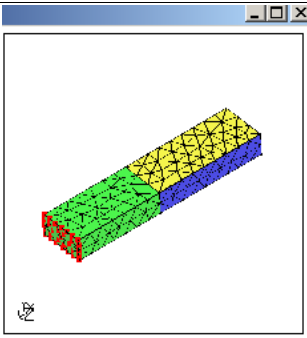
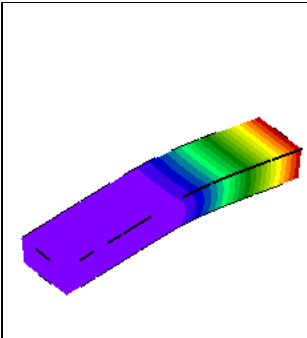
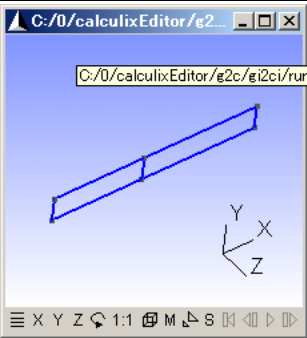
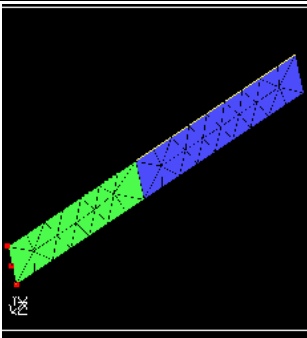

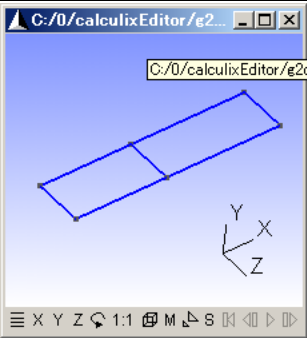
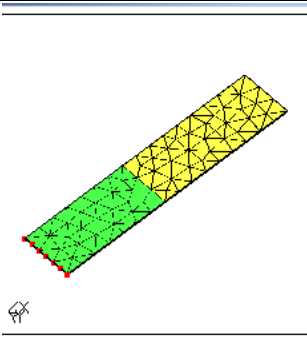
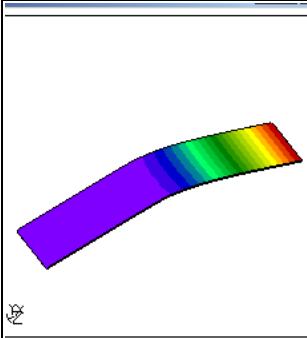


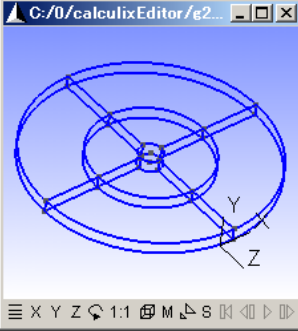
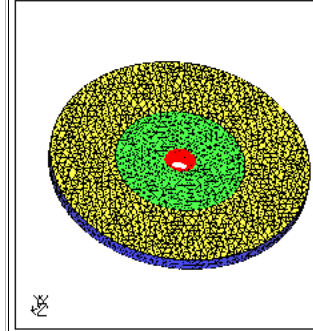
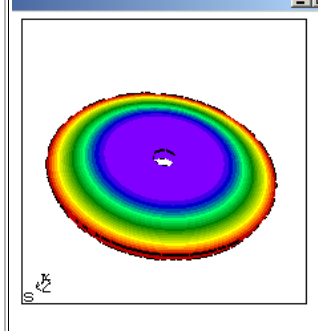
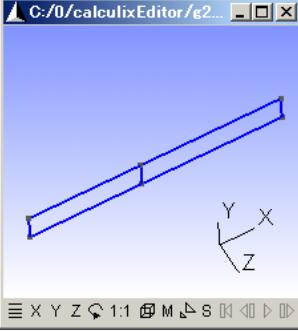
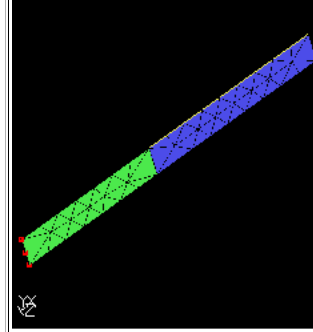
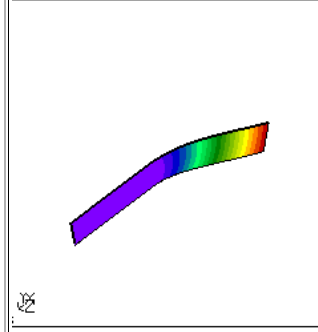
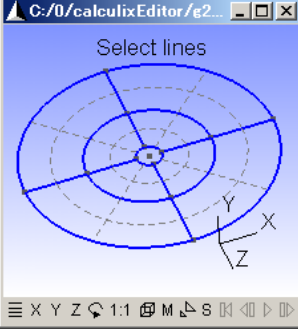
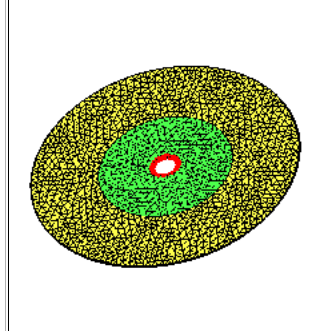
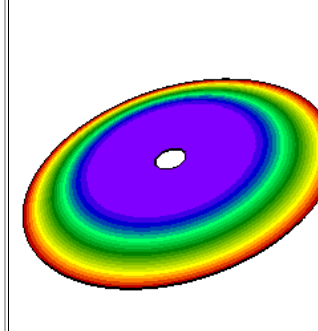
Gmsh2.5.1(later?) exported **Abaqus inp** -----> **Calculix inp**[Tweet](#)[gi2ci.py](#) is [here](#) INCOMPLETION Version 0.021 2011/Aug/26**Gmsh** (on 2011/Aug/25 : 2.5.1 is **not released** version but **Development version**.)

\*(cannot use before Gmsh 2.5.0)

\* **2D-Tri**(C2D3 , C2D6) or **3D-Tetra**(C3D4 , C3D10) only(cannot treat **Recombined mesh** , it will make 2D-Rect , 3D-prism , Hexa)

	Gmsh	Calculix(Pre)	Calculix(Post)	MAX Displacement
<a href="#">3D Solid (C3D10)</a>				9.90[mm]
<a href="#">2D Plain Stress (CPS6)</a>				9.52[mm]
<a href="#">2D Shell (S6)</a>				9.72[mm]

	Gmsh	Calculix(Pre)	Calculix(Post)	Max Displacement
--	------	---------------	----------------	------------------

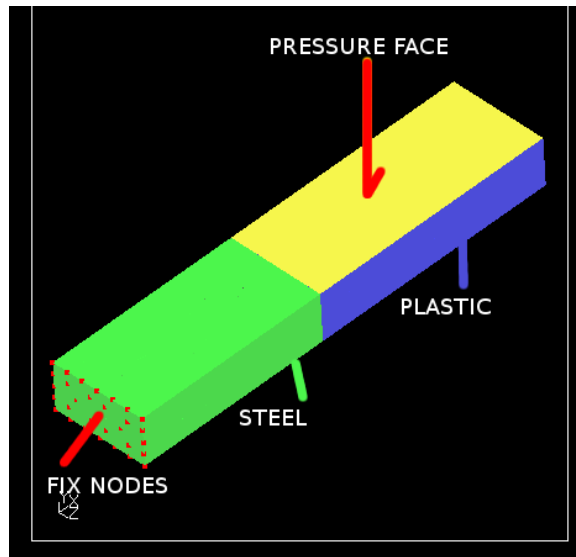
<p><a href="#">3D</a> <a href="#">Axisymmetry</a> <a href="#">(C3D10)</a></p>				<p>10.3[mm]</p>
<p><a href="#">2D</a> <a href="#">Axisymmetry</a> <a href="#">(CAX6)</a></p>				<p>10.2[mm]</p>
<p><a href="#">2DSHELL</a> <a href="#">(S6)</a></p>				<p>10.1[mm]</p>

[Return](#)

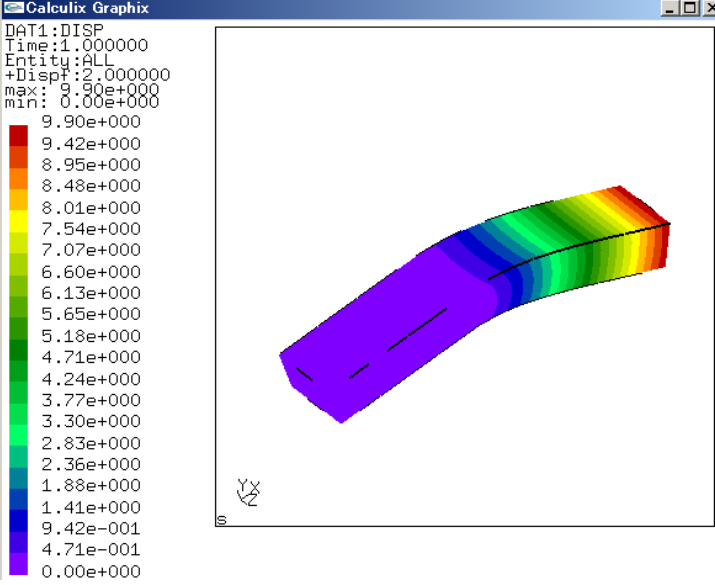
Powered by [FC2.com](#)

Gmsh2.5.1(lator?) (it will not work 2.5.0)

Gmsh 3DMesh ----&gt; Export Abaqus inp ----&gt; Calculix inp



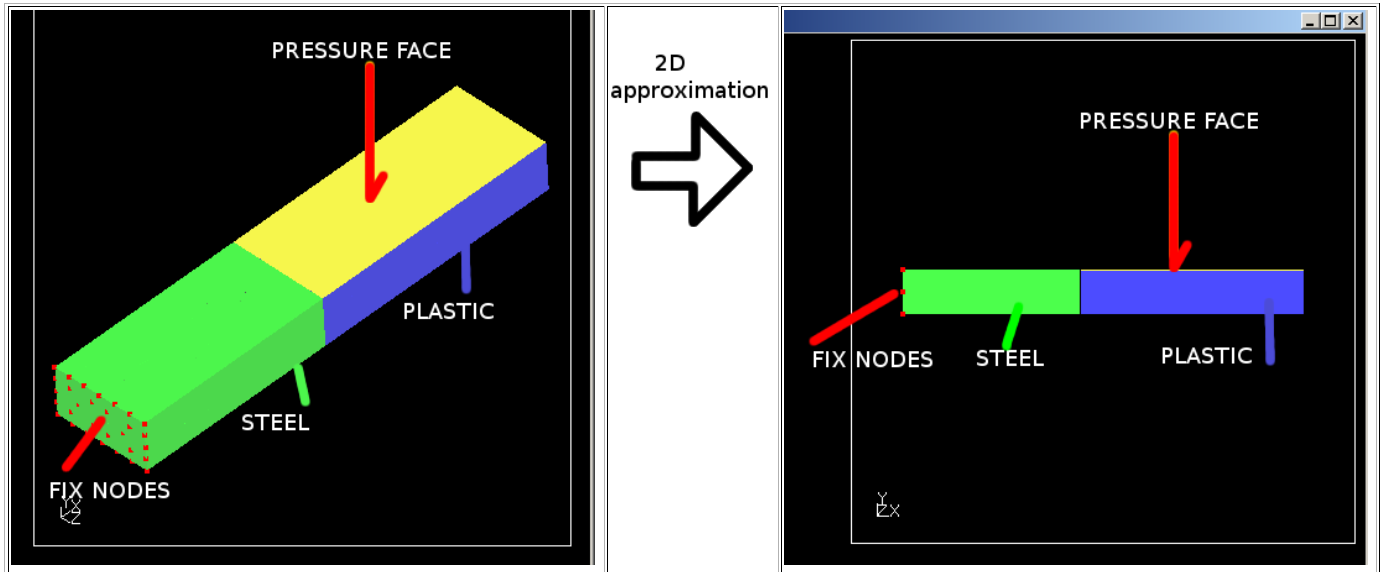
Make Geometry with Gmsh  and define Physical groups	<pre> cl1 = 5; Point(1) = {10, 0, 0, cl1}; Point(2) = {50, 0, 0, cl1}; Point(3) = {100, 0, 0, cl1}; Line(1) = {1, 2}; Line(2) = {2, 3}; Extrude {0, 10, 0} {   Line(1, 2); } Extrude {0, 0, 20} {   Surface(6, 10); } Physical Surface(183) = {31}; Physical Surface(184) = {49}; Physical Volume(181) = {1}; Physical Volume(182) = {2}; </pre>	
Mesh with 3D-2order	<pre> Mesh 3D set Order 2 </pre>	
Mesh save	Save as gmsh.inp (Mesh -- Abaqus INP)	
Mesh Convert to Calculix	<pre> &gt;python gi2ci2.py gmsh.inp calc.inp 3 C3 </pre>	<pre> 1) gmsh.inp ---- input file (gmsh's abaqus format ) 2) calc.inp ---- output file (calculix format , surface elements are converted to node group ) 3) 3 ---- 3D 4) C3 --- PreWord 'C3' will make Element Type 'C3D10' , if 'F3' --&gt; 'F3D10' </pre>
Check and make BC with calculix	<pre> &gt;cgx -c calc.inp </pre>	
	<pre> prnt set comp SURFACE184 down plot e VOLUME181 g view elem on view edge off frame plus e VOLUME182 b plus n SURFACE183 plus f SURFACE184 send all abq names send SURFACE184 abq pres 1.0 </pre>	
make solver file (sol.inp)	<pre> *INCLUDE, INPUT=calc.inp *INCLUDE, INPUT=all.nam  *MATERIAL, NAME=STEEL *ELASTIC 200000 , 0.3 </pre>	

	<pre> *MATERIAL, NAME=PLA *ELASTIC 1000 , 0.3  *SOLID SECTION , Elset=VOLUME181 , Material=STEEL *SOLID SECTION , Elset=VOLUME182 , Material=PLA *STEP *STATIC *BOUNDARY SURFACE183,1,3,0 *DLOAD *include,input=SURFACE184.dlo *NODE PRINT,NSET=Nall U *EL PRINT,ELSET=Eall S *NODE FILE U *EL FILE S *END STEP </pre>	
solver start	>ccx sol	
view result	>cgx -v sol.frd	 <p>Calculix Graphix</p> <p>DAT1: DISP Time: 1.000000 Entity: ALL +Dispr: 2.000000 max: 9.90e+000 min: 0.00e+000</p> <p>9.90e+000 9.42e+000 8.95e+000 8.48e+000 8.01e+000 7.54e+000 7.07e+000 6.60e+000 6.13e+000 5.65e+000 5.18e+000 4.71e+000 4.24e+000 3.77e+000 3.30e+000 2.83e+000 2.36e+000 1.88e+000 1.41e+000 9.42e-001 4.71e-001 0.00e+000</p> <p>Max Displacement = 9.90[mm]</p>

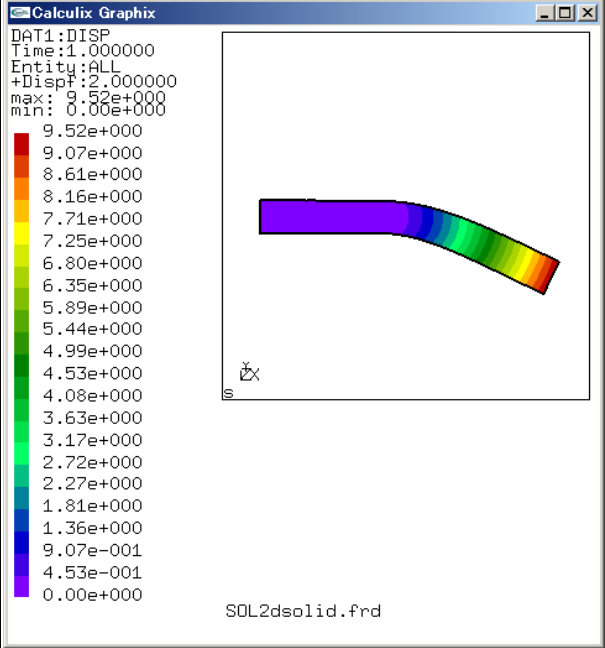
[Return](#)

Gmsh2.5.1(lator?) (it will not work 2.5.0)

Gmsh 2DMesh ---&gt; Export Abaqus inp ---&gt; Calculix inp



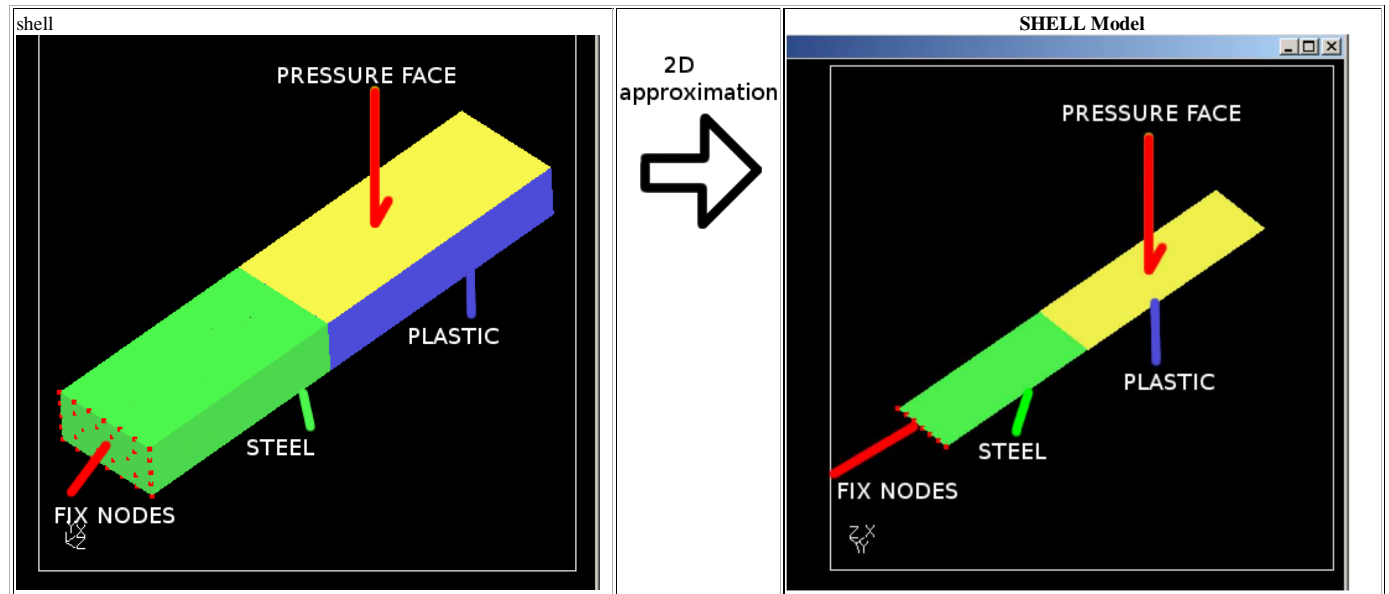
Make Geometry with Gmsh and define Physical groups	<pre> c11 = 5; Point(1) = { 10, 0, 0, c11}; Point(2) = { 50, 0, 0, c11}; Point(3) = { 100, 0, 0, c11}; Line(1) = { 1, 2}; Line(2) = { 2, 3}; Extrude {0, 10, 0} {   Line{ 1, 2}; } Physical Surface(181) = { 6}; Physical Surface(182) = { 10}; Physical Line(183) = { 4}; Physical Line(184) = { 7}; </pre>	
Mesh with 2D-2order	Mesh 2D set Order 2	
Mesh save	Save as gmsh.inp (Mesh -- Abaqus INP)	
Mesh Convert to Calculix	>python gi2ci2.py gmsh.inp calc.inp 2 CPS	1) gmsh.inp ---- input file (gmsh's abaqus format ) 2) calc.inp ---- output file (calculix format , line elements are converted to node group ) 3) 2 ---- 2D 4) CPS --- PreWord 'CPS' will make Element Type 'CPS6'
Check and make BC with calculix	>cgx -c calc.inp	
	<pre> prnt set comp CURVE184 down plot e SURFACE181 g view elem on view edge off frame plus e SURFACE182 b plus n CURVE183 plus f CURVE184 send all abq names send CURVE184 abq pres 1.0 </pre>	
make solver file (sol.inp)	<pre> *INCLUDE, INPUT=calc.inp *INCLUDE, INPUT=all.nam  *MATERIAL, NAME=STEEL *ELASTIC </pre>	

	<pre>200000 , 0.3  *MATERIAL, NAME=PLA *ELASTIC 1000 , 0.3  *SOLID SECTION , Elset=SURFACE181 , Material=STEEL 20 *SOLID SECTION , Elset=SURFACE182 , Material=PLA 20 *STEP *STATIC *BOUNDARY CURVE183,1,3,0 *DLOAD *include,input=CURVE184.dlo *NODE PRINT,NSET=Nall U *EL PRINT,ELSET=Eall S *NODE FILE U *EL FILE S *END STEP</pre>	
solver start	>ccx sol	
view result	>cgx -v sol.frd	<div><p>Calculix Graphix</p><p>DAT1:DISP Time:1.000000 Entity:ALL +Disp:2.000000 max: 9.52e+000 min: 0.00e+000</p><p>9.52e+000 9.07e+000 8.61e+000 8.16e+000 7.71e+000 7.25e+000 6.80e+000 6.35e+000 5.89e+000 5.44e+000 4.99e+000 4.53e+000 4.08e+000 3.63e+000 3.17e+000 2.72e+000 2.27e+000 1.81e+000 1.36e+000 9.07e-001 4.53e-001 0.00e+000</p><p>SOL2dsolid.frd</p></div> <p><b>Max Displacement = 9.52[mm]</b> (3D Result was 9.90[mm] )</p>

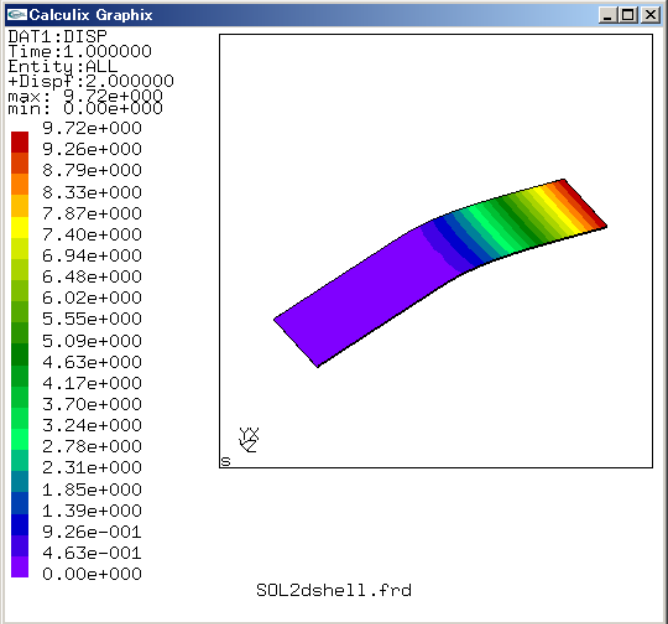
[Return](#)

Gmsh2.5.1(lator?) (it will not work 2.5.0)

Gmsh 2DMesh (SHELL MODEL) ----&gt; Export Abaqus inp ----&gt; Calculix inp



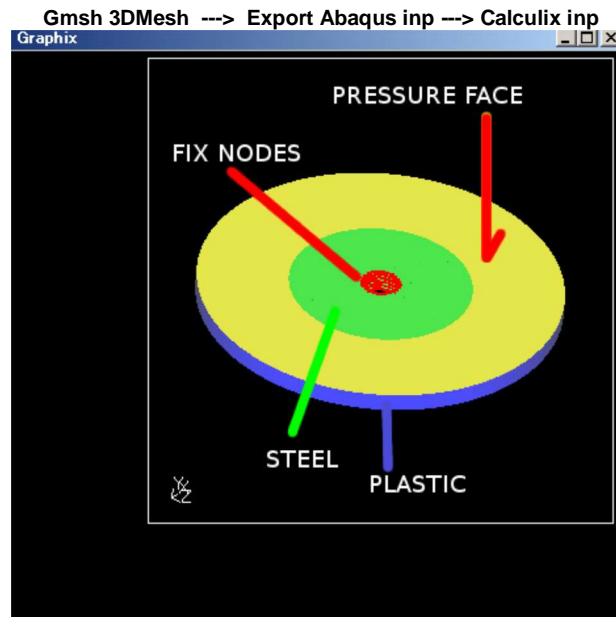
Make Geometry with Gmsh and define Physical groups	<pre> c11 = 5; Point(1) = { 10, 0, 0, c11 }; Point(2) = { 50, 0, 0, c11 }; Point(3) = { 100, 0, 0, c11 }; Line(1) = { 1, 2 }; Line(2) = { 2, 3 }; Extrude { 0, 10, 0 } {   Line { 1, 2 }; } Physical Surface(181) = { 6 }; Physical Surface(182) = { 10 }; Physical Line(183) = { 4 }; Physical Line(184) = { 7 }; </pre>	
Mesh with 2D-2order	Mesh 2D set Order 2	
Mesh save	Save as gmsh.inp (Mesh -- Abaqus INP)	
Mesh Convert to Calculix	>python gi2ci2.py gmsh.inp calc.inp 2 S	1) gmsh.inp ---- input file (gmsh's abaqus format ) 2) calc.inp ---- output file (calculix format , line elements are converted to node group ) 3) 2 ---- 2D 4) S ---- PreWord 'S' will make Element Type 'S6'
Check and make BC with calculix	>cgx -c calc.inp	
	<pre> prnt set comp SURFACE182 down plot e SURFACE181 plus e SURFACE182 b plus n CURVE183 plus f SURFACE182 send all abq names send SURFACE182 abq pres 1.0 </pre>	
make solver file (sol.inp)	<pre> *INCLUDE, INPUT=calc.inp *INCLUDE, INPUT=all.nam  *MATERIAL, NAME=STEEL </pre>	

	<pre> *ELASTIC 200000 , 0.3  *MATERIAL, NAME=PLA *ELASTIC 1000 , 0.3  *SHELL SECTION , Elset=SURFACE181 , Material=STEEL 10 *SHELL SECTION , Elset=SURFACE182 , Material=PLA 10 *STEP *STATIC *BOUNDARY CURVE183,1,6,0 *DLOAD *include,input=SURFACE182.dlo *NODE PRINT,NSET=Nall U *EL PRINT,ELSET=Eall S *NODE FILE U *EL FILE S *END STEP </pre>	
solver start	>ccx sol	
view result	>cgx -v sol.frd	 <p>Calculix Graphix</p> <p>     DAT1:DISP      Time:1.000000      Entity:ALL      +Disp:2.000000      max: 9.72e+000      min: 0.00e+000   </p> <p>     9.72e+000      9.26e+000      8.79e+000      8.33e+000      7.87e+000      7.40e+000      6.94e+000      6.48e+000      6.02e+000      5.55e+000      5.09e+000      4.63e+000      4.17e+000      3.70e+000      3.24e+000      2.78e+000      2.31e+000      1.85e+000      1.39e+000      9.26e-001      4.63e-001      0.00e+000   </p> <p>SOL2dshe11.frd</p> <p><b>Max Displacement = 9.72[mm]</b> (3D Result was 9.90[mm] )</p>

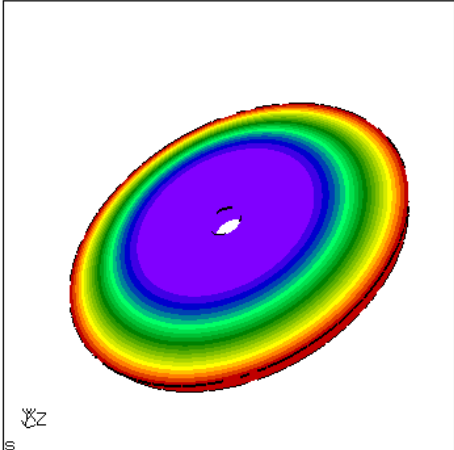
[Return](#)



Gmsh2.5.1(lator?) (it will not work 2.5.0 )



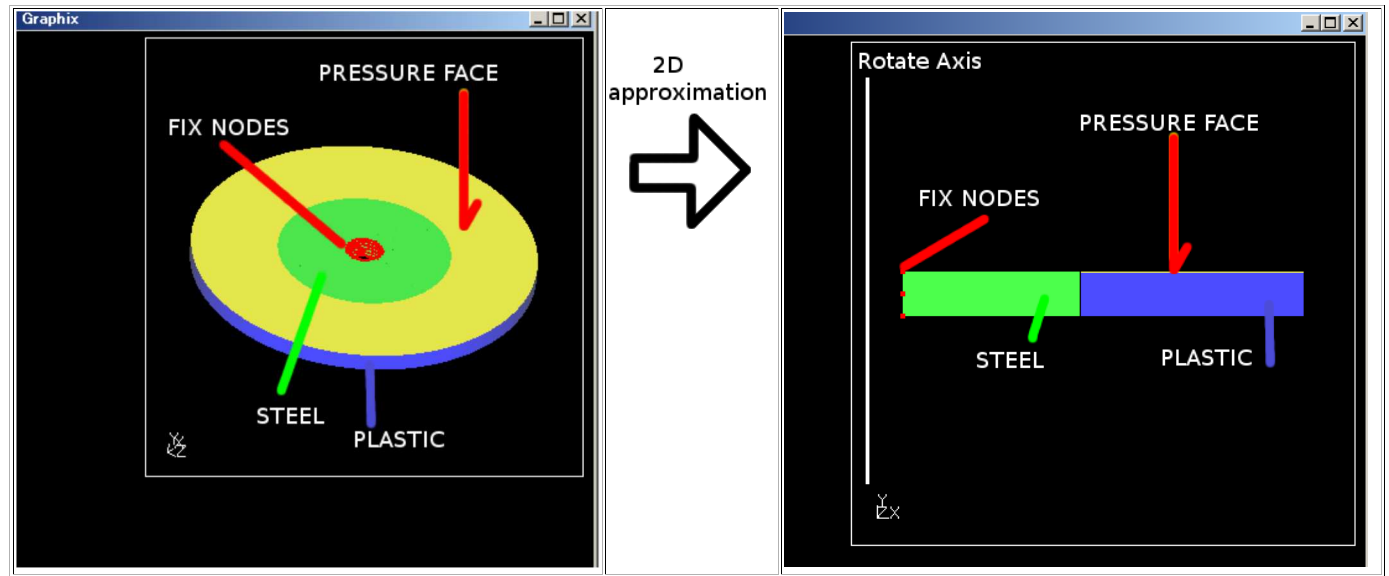
Make Geometry with Gmsh and define Physical groups	<pre> cl1 = 5; Point(1) = {10, 0, 0, cl1}; Point(2) = {50, 0, 0, cl1}; Point(3) = {100, 0, 0, cl1}; Line(1) = {1, 2}; Line(2) = {2, 3}; Extrude {0, 10, 0} {   Line{1, 2}; } Extrude {{0, 1, 0}, {0, 0, 0}, Pi/2} {   Surface{6, 10}; } Extrude {{0, 1, 0}, {0, 0, 0}, Pi/2} {   Surface{54, 32}; } Extrude {{0, 1, 0}, {0, 0, 0}, Pi/2} {   Surface{76, 98}; } Extrude {{0, 1, 0}, {0, 0, 0}, Pi/2} {   Surface{142, 120}; } Physical Volume(181) = {4, 6, 7, 1}; Physical Volume(182) = {3, 5, 8, 2}; Physical Surface(183) = {97, 141, 163, 31}; Physical Surface(184) = {49, 71, 180, 115}; </pre>	
Mesh with 3D-2order	Mesh 3D <b>set Order 2</b>	
Mesh save	Save as gmsh.inp (Mesh -- Abaqus INP)	
Mesh Convert to Calculix	> <b>python gi2ci2.py gmsh.inp calc.inp 3 C3</b>	1) gmsh.inp ---- input file (gmsh's abaqus format ) 2) calc.inp ---- output file (calculix format , surface elements are converted to node group ) 3) 3 ---- 3D 4) C3 --- PreWord 'C3' will make Element Type 'C3D10'
Check and make BC with calculix	> <b>cgx -c calc.inp</b>	
	<pre> prnt set comp SURFACE184 down plot e VOLUME181 g view elem on view edge off frame plus e VOLUME182 b plus n SURFACE183 plus f SURFACE184 send all abq names send SURFACE184 abq pres 1.0 </pre>	

make solver file (sol.inp)	<pre>*INCLUDE, INPUT=calc.inp *INCLUDE, INPUT=all.nam  *MATERIAL, NAME=STEEL *ELASTIC 200000 , 0.3  *MATERIAL, NAME=PLA *ELASTIC 1000 , 0.3  *SOLID SECTION , Elset=VOLUME181 , Material=STEEL *SOLID SECTION , Elset=VOLUME182 , Material=PLA *STEP *STATIC *BOUNDARY SURFACE183,1,3,0 *DLOAD *include,input=SURFACE184.dlo *NODE PRINT,NSET=Nall U *EL PRINT,ELSET=Eall S *NODE FILE U *EL FILE S *END STEP</pre>	
solver start	> <b>ccx sol</b>	
view result	> <b>cgx -v sol.frd</b>	<div><div>Calculix Graphix</div><div><div> DAT1:DISP Time:1.000000 Entity:ALL +Disp:2.000000 max: 1.03e+001 min: 0.00e+000 </div><div> 1.03e+001 9.78e+000 9.29e+000 8.80e+000 8.31e+000 7.82e+000 7.34e+000 6.85e+000 6.36e+000 5.87e+000 5.38e+000 4.89e+000 4.40e+000 3.91e+000 3.42e+000 2.93e+000 2.45e+000 1.96e+000 1.47e+000 9.78e-001 4.89e-001 0.00e+000 </div></div><div></div><div>Max Displacement = 10.30[mm]</div></div>

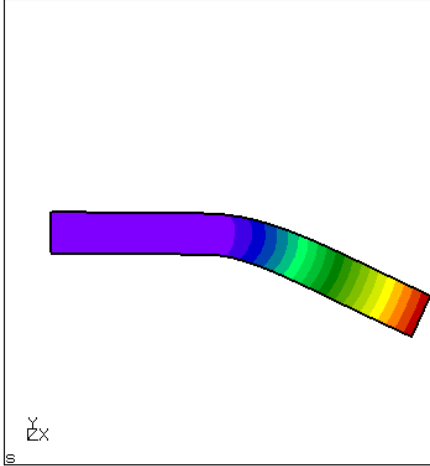
[Return](#)

Gmsh2.5.1(lator?) (it will not work 2.5.0)

Gmsh 2DMesh ---&gt; Export Abaqus inp ---&gt; Calculix inp



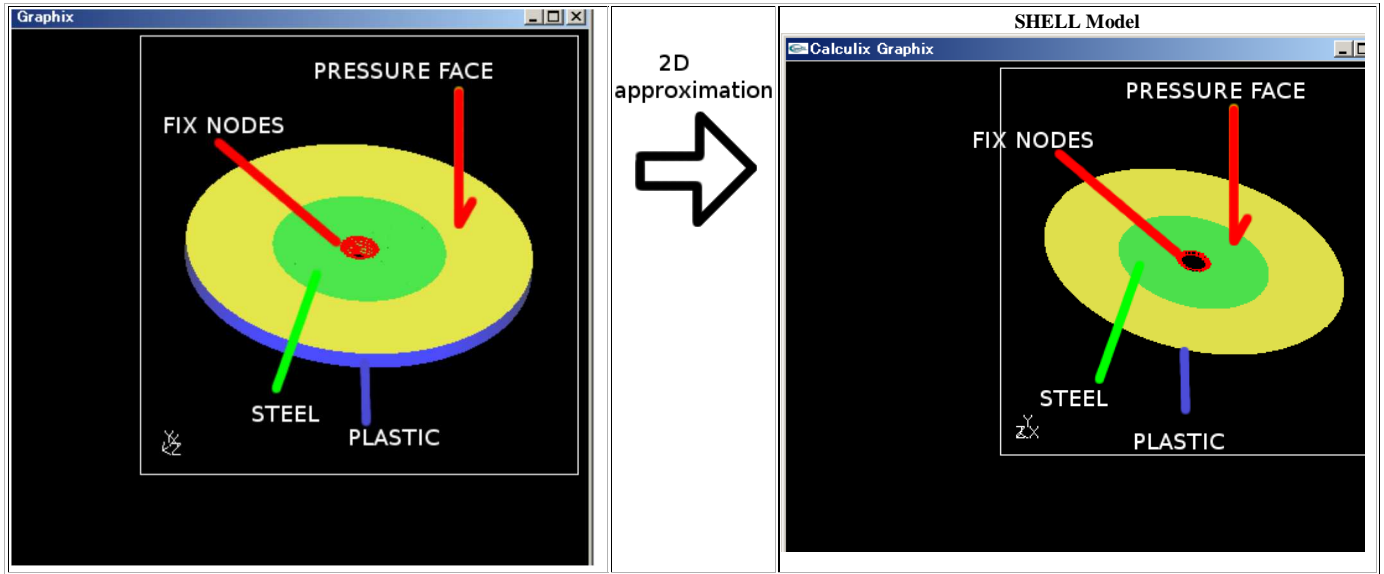
Make Geometry with Gmsh and define Physical groups	<pre> c11 = 5; Point(1) = {10, 0, 0, c11}; Point(2) = {50, 0, 0, c11}; Point(3) = {100, 0, 0, c11}; Line(1) = {1, 2}; Line(2) = {2, 3}; Extrude {0, 10, 0} {   Line{1, 2}; } Physical Surface(181) = {6}; Physical Surface(182) = {10}; Physical Line(183) = {4}; Physical Line(184) = {7}; </pre>	
Mesh with 2D-2order	Mesh 2D set Order 2	
Mesh save	Save as gmsh.inp (Mesh -- Abaqus INP)	
Mesh Convert to Calculix	>python gi2ci2.py gmsh.inp calc.inp 2 CAX	1) gmsh.inp ---- input file (gmsh's abaqus format ) 2) calc.inp ---- output file (calculix format , line elements are converted to node group ) 3) 2 ---- 2D 4) CAX --- PreWord 'CAX' will make Element Type 'CAX6'
Check and make BC with calculix	>cgx -c calc.inp	
	<pre> prnt set comp CURVE184 down plot e SURFACE181 g view elem on view edge off frame plus e SURFACE182 b plus n CURVE183 plus f CURVE184 send all abq names send CURVE184 abq pres 1.0 </pre>	
make solver file (sol.inp)	<pre> *INCLUDE, INPUT=calc.inp *INCLUDE, INPUT=all.nam  *MATERIAL, NAME=STEEL *ELASTIC </pre>	

	<pre>200000 , 0.3  *MATERIAL, NAME=PLA *ELASTIC 1000 , 0.3  *SOLID SECTION , Elset=SURFACE181 , Material=STEEL *SOLID SECTION , Elset=SURFACE182 , Material=PLA *STEP *STATIC *BOUNDARY CURVE183,1,3,0 *DLOAD *include,input=CURVE184.dlo *NODE PRINT,NSET=Nall U *EL PRINT,ELSET=Eall S *NODE FILE U *EL FILE S *END STEP</pre>	
solver start	>ccx sol	
view result	>cgx -v sol.frd	<div><div>Calculix Graphix</div><div><div>DATA: DISP Time: 1.000000 Entity: ALL +Disp: 2.000000 max: 1.02e+001 min: 0.00e+000</div><div><div>1.02e+001 9.74e+000 9.25e+000 8.76e+000 8.28e+000 7.79e+000 7.30e+000 6.82e+000 6.33e+000 5.84e+000 5.36e+000 4.87e+000 4.38e+000 3.90e+000 3.41e+000 2.92e+000 2.43e+000 1.95e+000 1.46e+000 9.74e-001 4.87e-001 0.00e+000</div><div></div></div><div>Max Displacement = 10.2[mm] (3D Result was 10.3[mm] )</div></div></div>

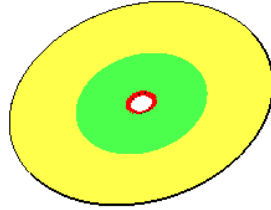
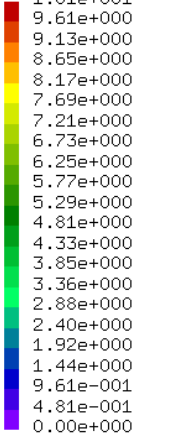
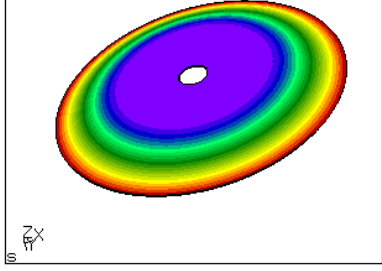
[Return](#)

Gmsh2.5.1(lator?) (it will not work 2.5.0)

Gmsh 2DMesh (SHELL MODEL) ----&gt; Export Abaqus inp ----&gt; Calculix inp



Make Geometry with Gmsh and define Physical groups	<pre> c11 = 5; Point(1) = { 10, 0, 0, c11 }; Point(2) = { 50, 0, 0, c11 }; Point(3) = { 100, 0, 0, c11 }; Line(1) = { 1, 2 }; Line(2) = { 2, 3 };  Extrude { { 0, 1, 0 }, { 0, 0, 0 }, Pi/2 } {   Line { 1, 2 }; } Extrude { { 0, 1, 0 }, { 0, 0, 0 }, Pi/2 } {   Line { 3, 7 }; } Extrude { { 0, 1, 0 }, { 0, 0, 0 }, Pi/2 } {   Line { 11, 15 }; } Extrude { { 0, 1, 0 }, { 0, 0, 0 }, Pi/2 } {   Line { 19, 23 }; } Physical Surface(181) = { 6, 14, 22, 30 }; Physical Surface(182) = { 10, 18, 26, 34 }; Physical Line(183) = { 4, 12, 20, 28 }; </pre>	
Mesh with 2D-2order	Mesh 2D set Order 2	
Mesh save	Save as gmsh.inp (Mesh -- Abaqus INP)	
Mesh Convert to Calculix	>python gi2ci2.py gmsh.inp calc.inp 2 S	1) gmsh.inp ---- input file (gmsh's abaqus format ) 2) calc.inp ---- output file (calculix format , line elements are converted to node group ) 3) 2 ---- 2D 4) S --- PreWord 'S' will make Element Type 'S6'
Check and make BC with calculix	>cgx -c calc.inp	

	<pre> prnt set comp SURFACE182 down plot e SURFACE181 plus e SURFACE182 b plus n CURVE183 plus f SURFACE182 send all abq names send SURFACE182 abq pres 1.0 </pre>	
make solver file (sol.inp)	<pre> *INCLUDE, INPUT=calc.inp *INCLUDE, INPUT=all.nam  *MATERIAL, NAME=STEEL *ELASTIC 200000 , 0.3  *MATERIAL, NAME=PLA *ELASTIC 1000 , 0.3  *SHELL SECTION , Elset=SURFACE181 , Material=STEEL 10 *SHELL SECTION , Elset=SURFACE182 , Material=PLA 10  *STEP *STATIC *BOUNDARY CURVE183,1,6,0 *DLOAD *include,input=SURFACE182.dlo *NODE PRINT,NSET=Nall U *EL PRINT,ELSET=Eall S *NODE FILE U *EL FILE S *END STEP </pre>	
solver start	>ccx sol	
view result	>cgx -v sol.frd	<div> <div> <p>Calculix Graphix</p> <p>DAT1:DISP Time:1.000000 Entity:ALL +Disp1:2.000000 max: 1.01e+001 min: 0.00e+000</p>  </div> <div>  <p>SOL2dshellaxis.frd</p> </div> </div> <p><b>Max Displacement = 10.1[mm]</b> (3D Result was 10.3[mm] )</p>

Powered by [FC2.com](http://fc2.com)