Gmsh2.5.**1**(later?) exported **Abaqus inp** -----> **Calculix inp**

Tweet

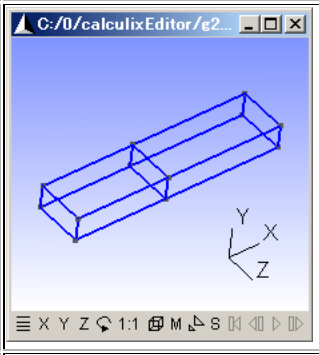
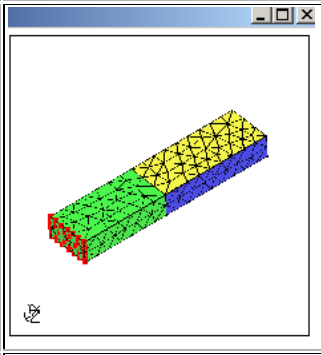
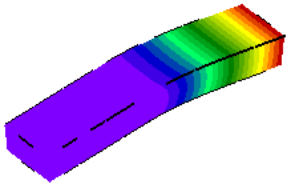
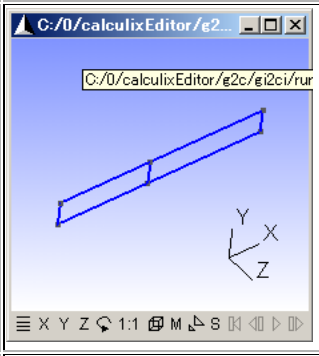
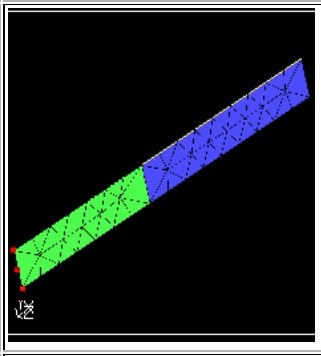
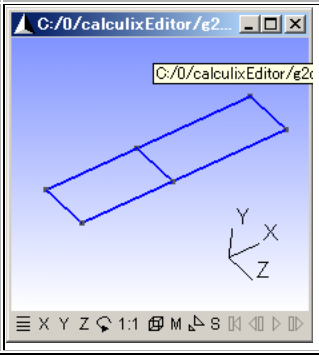
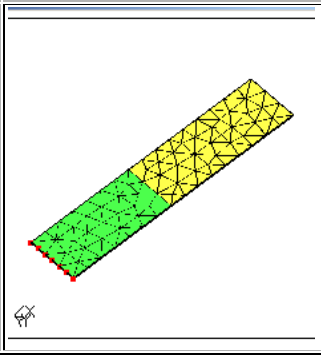
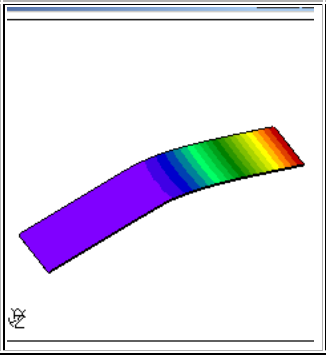
---

gi2ci.py is here *INCOMPLETION Version 0.021 2011/Aug/26*

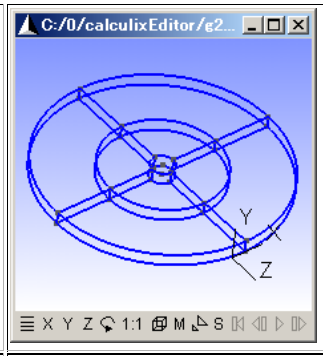
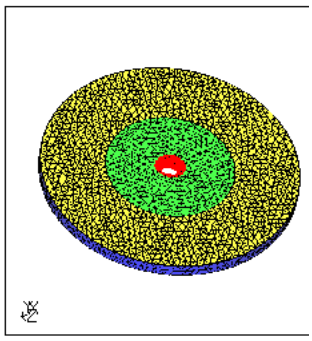
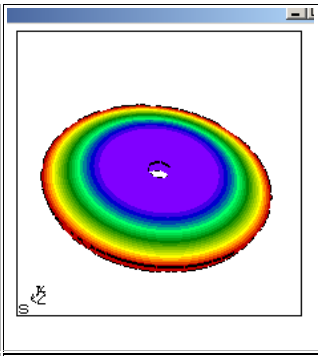
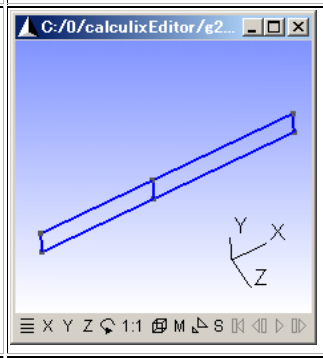
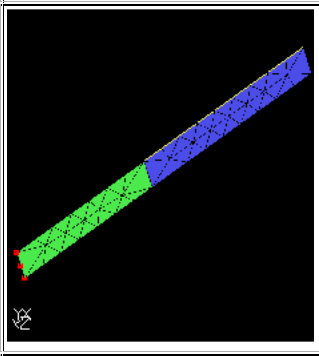
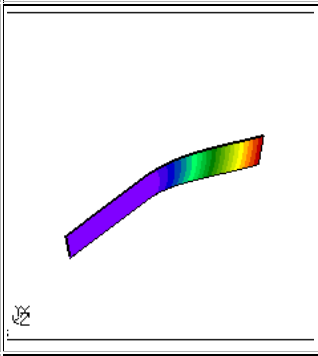
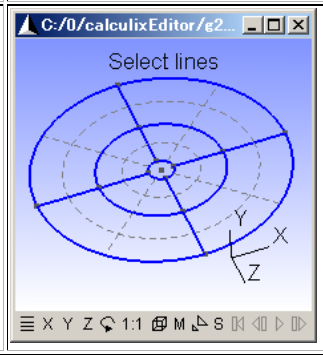
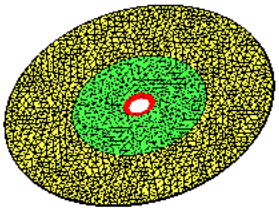
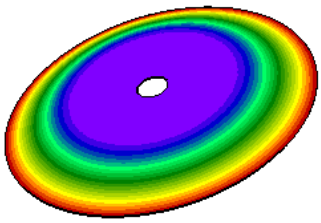
Gmsh (on 2011/Aug/25 : 2.5.1 is **not released** version but **Development version**:)

---

*(can**not** use before Gmsh 2.5.**0**)
* **2D-Tri**(C2D3 , C2D6) or **3D-Tetra**(C3D4 , C3D10) only
((can**not** treat ***Recombined mesh*** , it will make 2D-Rect , 3D-prism , Hexa)

	Gmsh	Calculix(Pre)	Calculix(Post)	MAX Displacement
3D Solid **(C3D10)**				**9.90[mm]**
2D Plain Stress **(CPS6)**				**9.52[mm]**
2D Shell **(S6)**				**9.72[mm]**

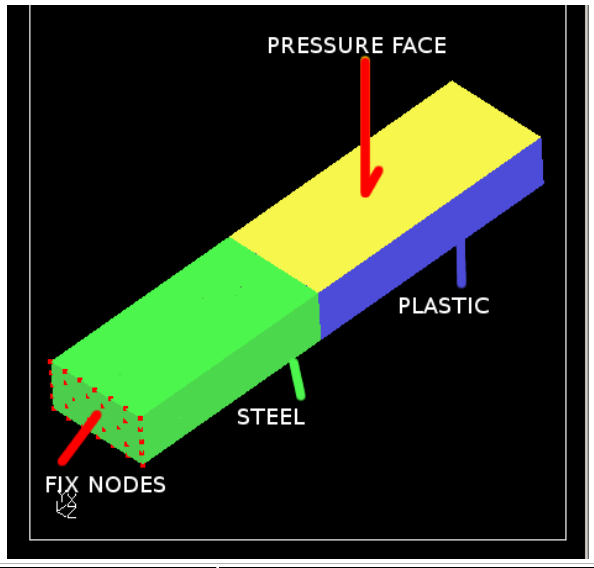
	Gmsh	Calculix(Pre)	Calculix(Post)	Max Displacement

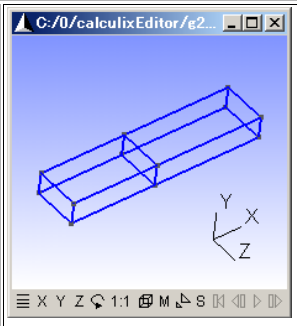
3D Axissymetry **(C3D10)**				**10.3[mm]**
2D Axisymmetry **(CAX6)**				**10.2[mm]**
2DSHELL (S6)				**10.1[mm]**

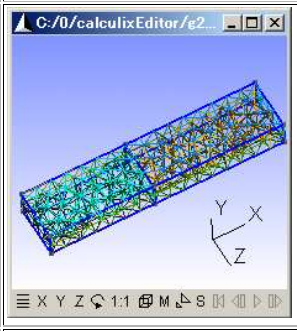
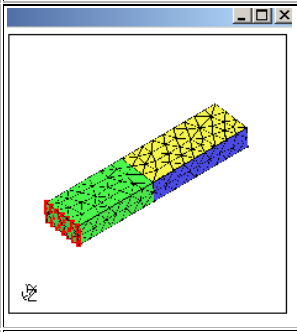
Return

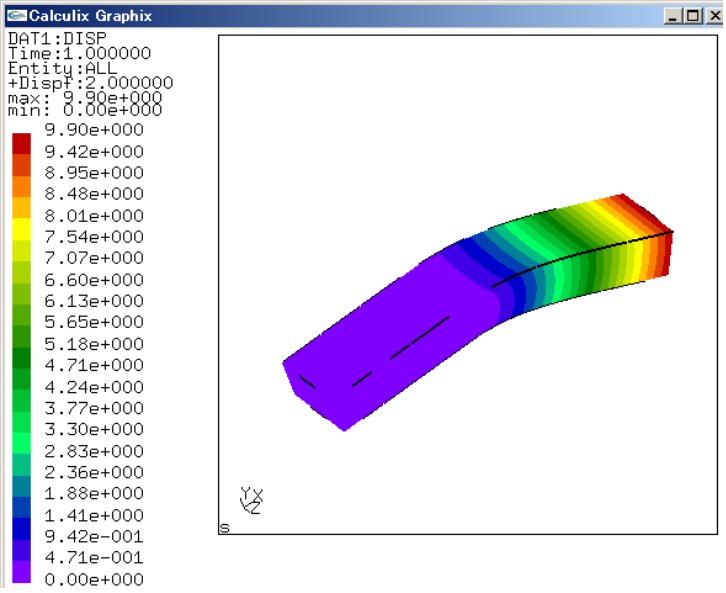
Gmsh2.5.1(lator?) (it will not work **2.5.0**)

Gmsh 3DMesh ---> Export Abaqus inp ---> Calculix inp



Make Geometry with Gmsh
 	
and define Physical groups
	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};
		

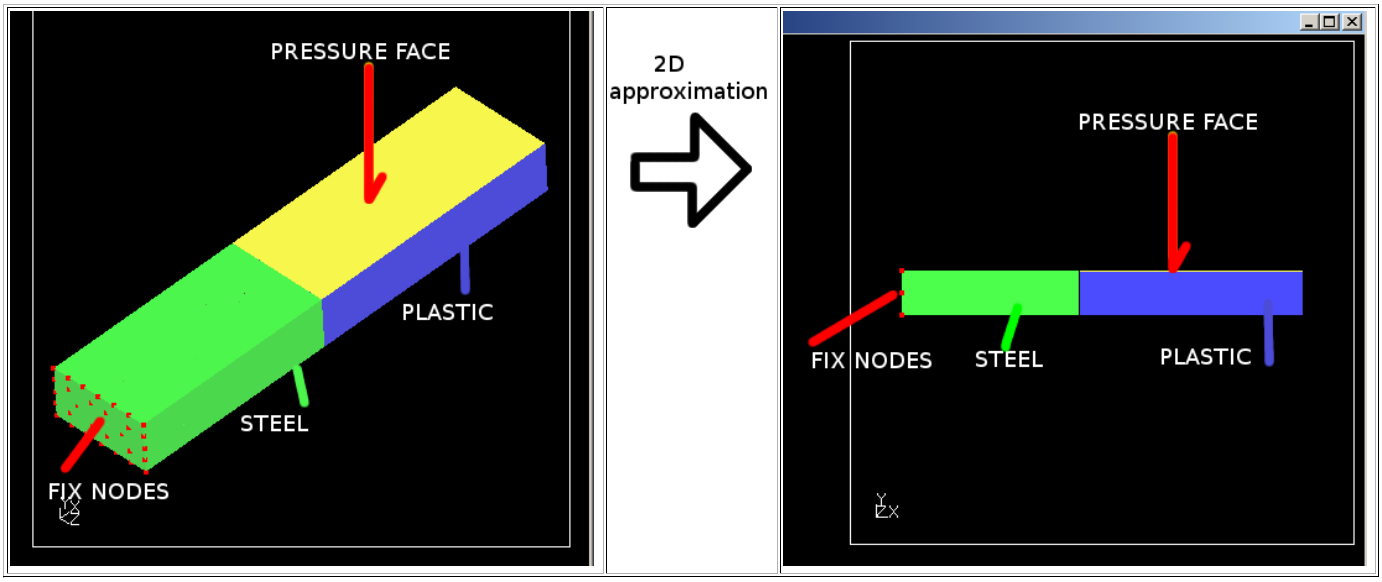
Mesh with 3D-**2order**	Mesh 3D **set Order 2**	
Mesh save	Save as gmsh.inp (Mesh -- Abaqus INP)	
Mesh Convert to Calculix	>**python gi2ci2.py gmsh.inp calc.inp 3 C3**	1) gmsh.inp ---- input file (gmsh's abaqus format) 2) calc.inp ---- output file (calculix format , surface elements are converted to node group) 3) 3 ---- 3D 4) C3 --- PreWord 'C3' will make Element Type 'C3D10' , if 'F3' --> 'F3D10'
Check and make BC with calculix	>**cgx -c calc.inp**	
	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	
make solver file (sol.inp)	*INCLUDE, INPUT=calc.inp *INCLUDE, INPUT=all.nam *MATERIAL, NAME=STEEL *ELASTIC 200000 , 0.3	

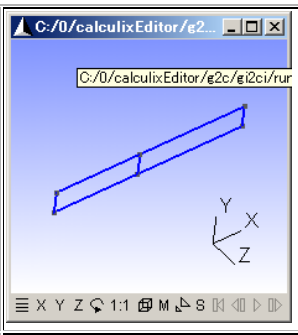
	*MATERIAL, NAME=PLA *ELASTIC 1000 , 0.3 *SOLID SECTION , Elset=VOLUME181 , Material=STEEL *SOLID SECTION , Elset=VOLUME182 , Material=PLA *STEP *STATIC *BOUNDARY SURFACE183,1,3,0 *DLOAD *include,input=SURFACE184.dlo *NODE PRINT,NSET=Nall U *EL PRINT,ELSET=Eall S *NODE FILE U *EL FILE S *END STEP	
solver start	**>ccx sol**	
view result	**>cgx -v sol.frd**	 **Max Displacement = 9.90[mm]**

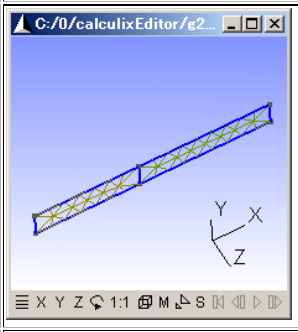
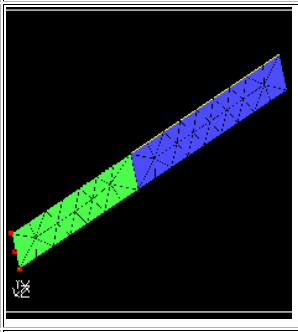
Return

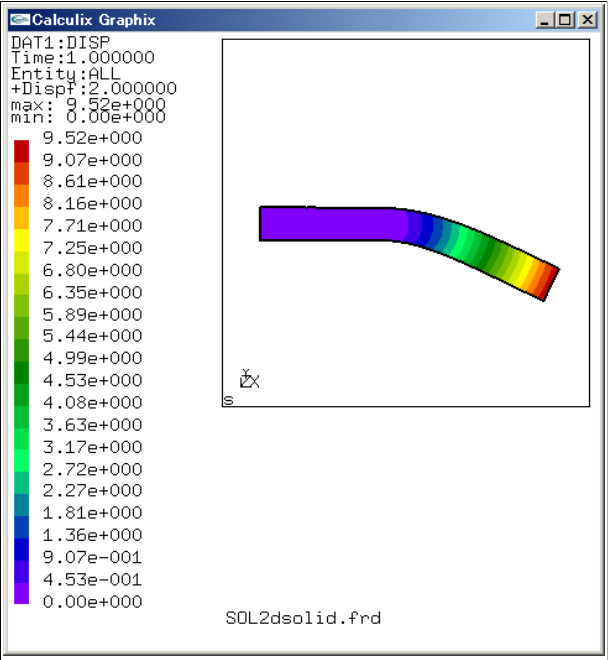
Gmsh2.5.1(lator?) (it will not work **2.5.0**)

Gmsh 2DMesh ---> Export Abaqus inp ---> Calculix inp



Make Geometry with Gmsh
 	
and define Physical groups
	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};
	}
	Physical Surface(181) = {6};
	Physical Surface(182) = {10};
	Physical Line(183) = {4};
	Physical Line(184) = {7};
		

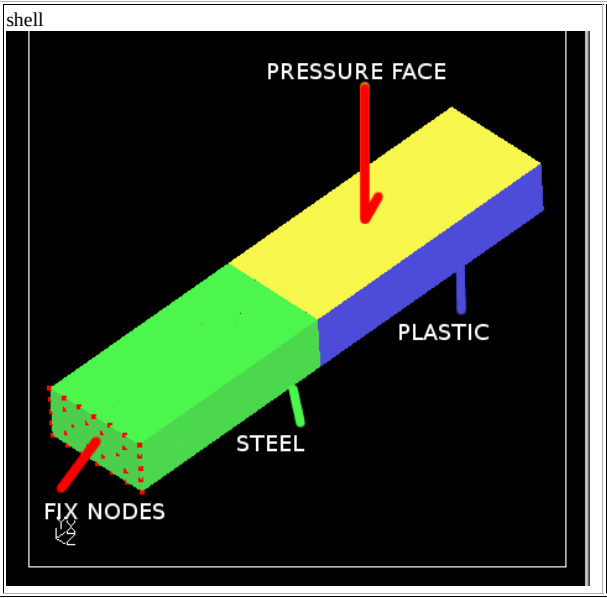
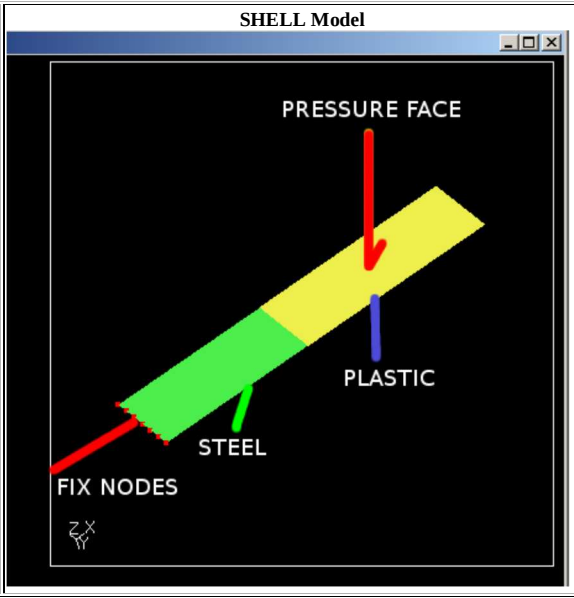
Mesh with 2D-**2order**	Mesh 2D **set Order 2**	
Mesh save	Save as gmsh.inp (Mesh -- Abaqus INP)	
Mesh Convert to Calculix	>**python gi2ci2.py gmsh.inp calc.inp 2 CPS**	1) gmsh.inp ---- input file (gmsh's abaqus format) 2) calc.inp ---- output file (calculix format , line elements are converted to node group) 3) 2 ---- 2D 4) CPS --- PreWord 'CPS' will make Element Type 'CPS6'
Check and make BC with calculix	>**cgx -c calc.inp**	
	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	
make solver file (sol.inp)	*INCLUDE, INPUT=calc.inp *INCLUDE, INPUT=all.nam *MATERIAL, NAME=STEEL *ELASTIC	

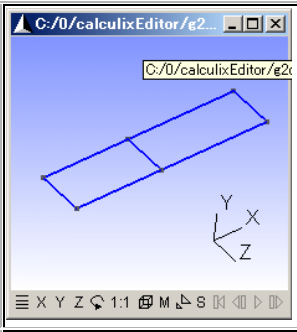
	200000 , 0.3 *MATERIAL, NAME=PLA *ELASTIC 1000 , 0.3 *SOLID SECTION , Elset=SURFACE181 , Material=STEEL 20 *SOLID SECTION , Elset=SURFACE182 , Material=PLA 20 *STEP *STATIC *BOUNDARY CURVE183,1,3,0 *DLOAD *include,input=CURVE184.dlo *NODE PRINT,NSET=Nall U *EL PRINT,ELSET=Eall S *NODE FILE U *EL FILE S *END STEP	
solver start	>**ccx sol**	
view result	>**cgx -v sol.frd**	 **Max Displacement = 9.52[mm]** (3D Result was 9.90[mm])

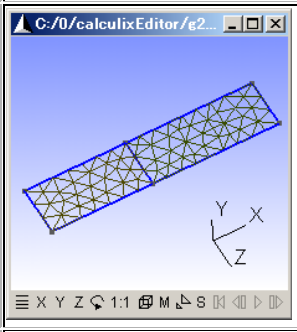
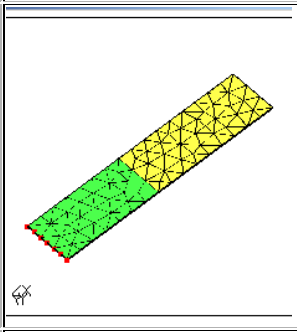
Return

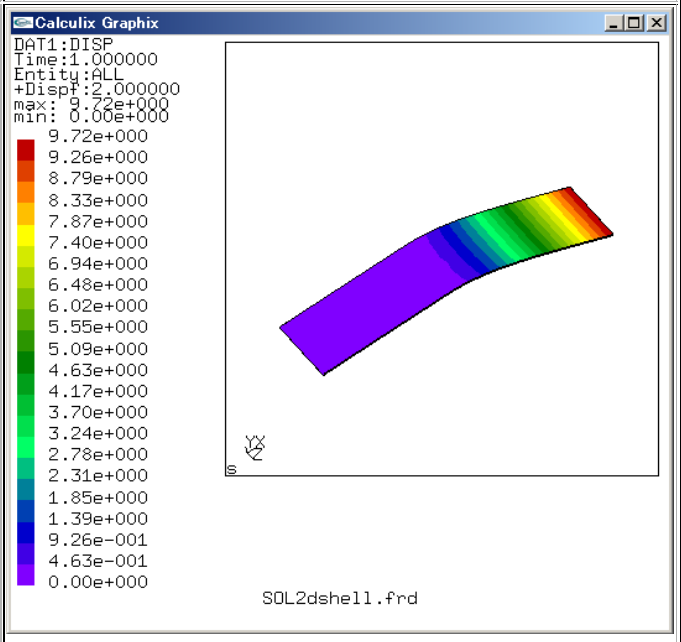
Gmsh2.5.1(lator?) (it will not work **2.5.0**)

Gmsh 2DMesh (SHELL MODEL) ---> **Export Abaqus inp ---> Calculix inp**

shell		**SHELL Model**
	2D approximation	

Make Geometry with Gmsh
 	
and define Physical groups
	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};
	}
	Physical Surface(181) = {6};
	Physical Surface(182) = {10};
	Physical Line(183) = {4};
	Physical Line(184) = {7};
		

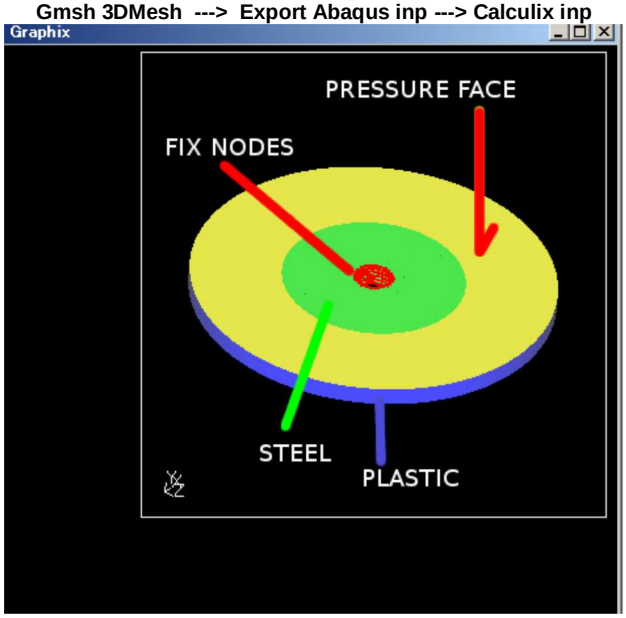
Mesh with 2D-**2order**	Mesh 2D **set Order 2**	
Mesh save	Save as gmsh.inp (Mesh -- Abaqus INP)	
Mesh Convert to Calculix	>**python gi2ci2.py gmsh.inp calc.inp 2 S**	1) gmsh.inp ---- input file (gmsh's abaqus format) 2) calc.inp ---- output file (calculix format , line elements are converted to node group) 3) 2 ---- 2D 4) S --- PreWord 'S' will make Element Type 'S6'
Check and make BC with calculix	>**cgx -c calc.inp**	
	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	
make solver file (sol.inp)	*INCLUDE, INPUT=calc.inp *INCLUDE, INPUT=all.nam *MATERIAL, NAME=STEEL	

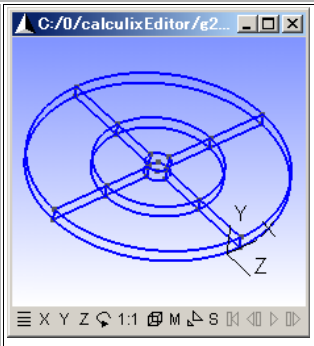
	*ELASTIC 200000 , 0.3 *MATERIAL, NAME=PLA *ELASTIC 1000 , 0.3 *SHELL SECTION , Elset=SURFACE181 , Material=STEEL 10 *SHELL SECTION , Elset=SURFACE182 , Material=PLA 10 *STEP *STATIC *BOUNDARY CURVE183,1,6,0 *DLOAD *include,input=SURFACE182.dlo *NODE PRINT,NSET=Nall U *EL PRINT,ELSET=Eall S *NODE FILE U *EL FILE S *END STEP	
solver start	>**ccx sol**	
view result	>**cgx -v sol.frd**	 **Max Displacement = 9.72[mm]** (3D Result was 9.90[mm])

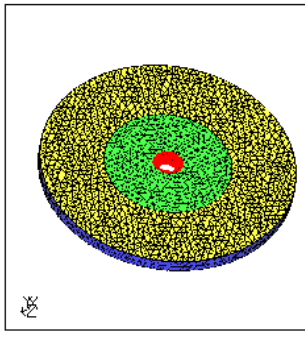
Return

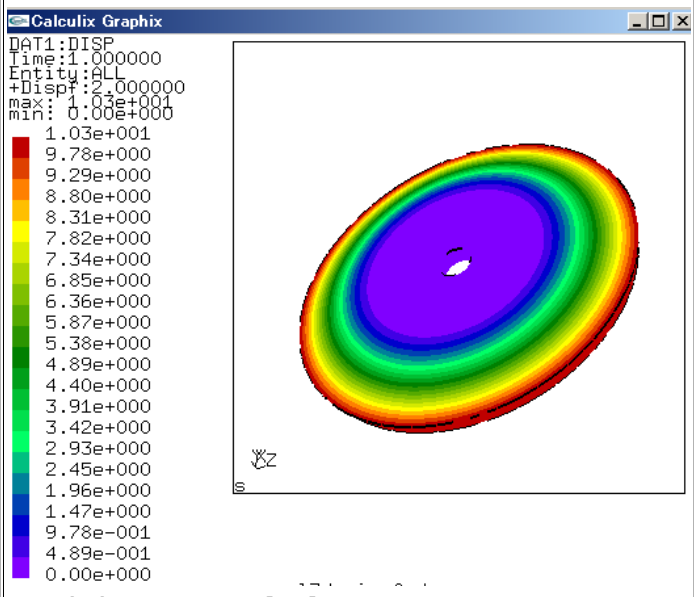
Gmsh2.5.1(lator?) (it will not work **2.5.0**)

Gmsh 3DMesh ---> Export Abaqus inp ---> Calculix inp



Make Geometry with Gmsh
 	
and define Physical groups
	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};
		

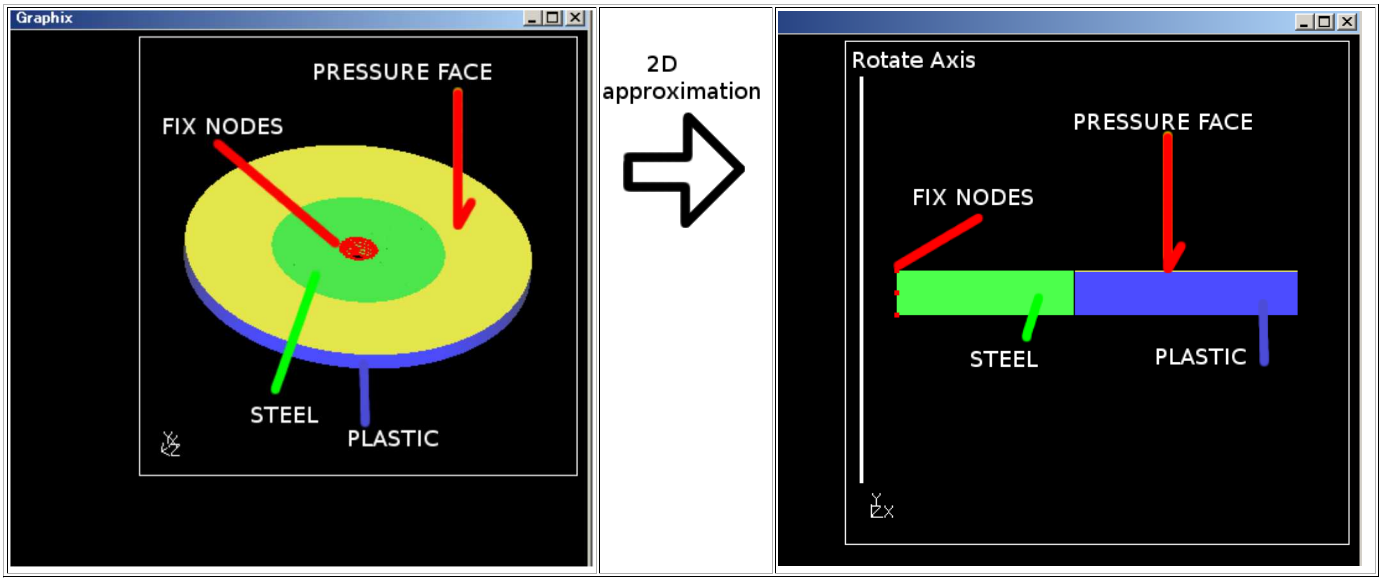
Mesh with 3D-**2order**	Mesh 3D **set Order 2**	
Mesh save	Save as gmsh.inp (Mesh -- Abaqus INP)	
Mesh Convert to Calculix	>**python gi2ci2.py gmsh.inp calc.inp 3 C3**	1) gmsh.inp ---- input file (gmsh's abaqus format) 2) calc.inp ---- output file (calculix format , surface elements are converted to node group) 3) 3 ---- 3D 4) C3 --- PreWord 'C3' will make Element Type 'C3D10'
Check and make BC with calculix	>**cgx -c calc.inp**	
	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	

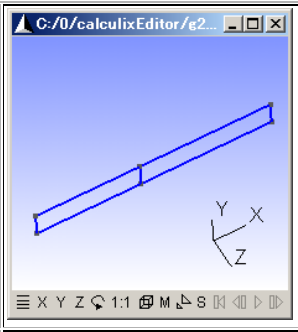
make solver file (sol.inp)	*INCLUDE, INPUT=calc.inp *INCLUDE, INPUT=all.nam *MATERIAL, NAME=STEEL *ELASTIC 200000 , 0.3 *MATERIAL, NAME=PLA *ELASTIC 1000 , 0.3 *SOLID SECTION , Elset=VOLUME181 , Material=STEEL *SOLID SECTION , Elset=VOLUME182 , Material=PLA *STEP *STATIC *BOUNDARY SURFACE183,1,3,0 *DLOAD *include,input=SURFACE184.dlo *NODE PRINT,NSET=Nall U *EL PRINT,ELSET=Eall S *NODE FILE U *EL FILE S *END STEP	
solver start	>**ccx sol**	
view result	>**cgx -v sol.frd**	 **Max Displacement = 10.30[mm]**

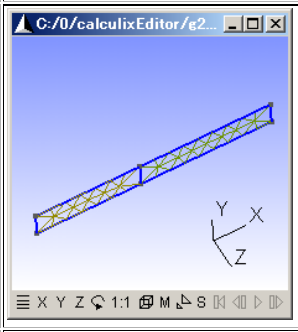
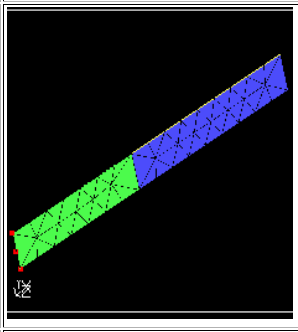
Return

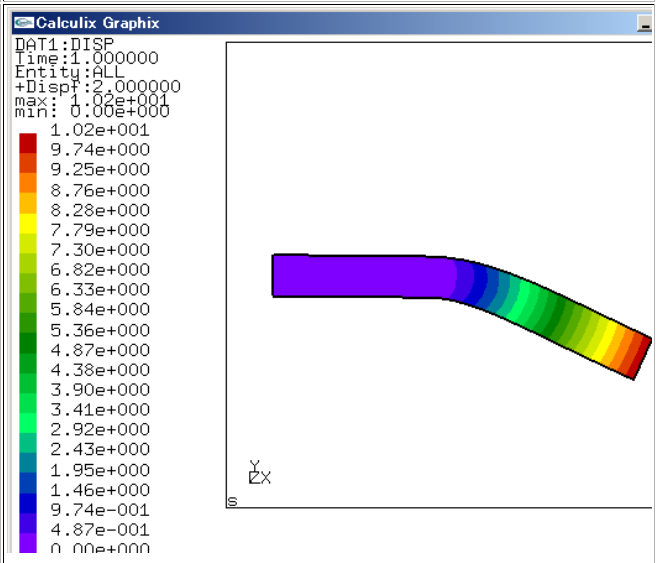
Gmsh2.5.1(lator?) (it will not work **2.5.0**)

Gmsh 2DMesh ---> Export Abaqus inp ---> Calculix inp



Make Geometry with Gmsh
 	
and define Physical groups
	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};
	}
	Physical Surface(181) = {6};
	Physical Surface(182) = {10};
	Physical Line(183) = {4};
	Physical Line(184) = {7};
		

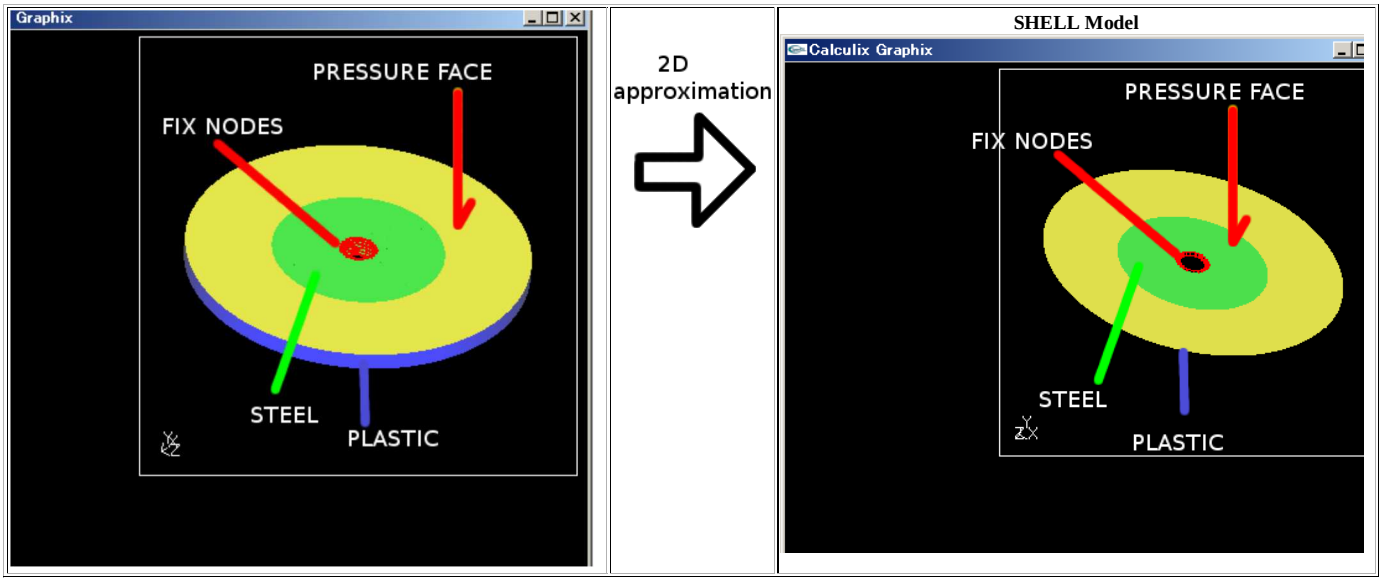
Mesh with 2D-**2order**	Mesh 2D **set Order 2**	
Mesh save	Save as gmsh.inp (Mesh -- Abaqus INP)	
Mesh Convert to Calculix	>**python gi2ci2.py gmsh.inp calc.inp 2 CAX**	1) gmsh.inp ---- input file (gmsh's abaqus format) 2) calc.inp ---- output file (calculix format , line elements are converted to node group) 3) 2 ---- 2D 4) CAX --- PreWord 'CAX' will make Element Type 'CAX6'
Check and make BC with calculix	>**cgx -c calc.inp**	
	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	
make solver file (sol.inp)	*INCLUDE, INPUT=calc.inp *INCLUDE, INPUT=all.nam *MATERIAL, NAME=STEEL *ELASTIC	

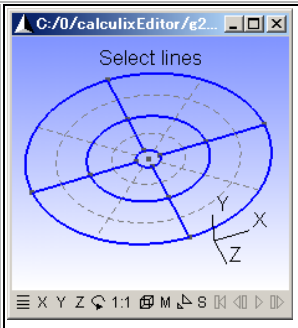
	200000 , 0.3 *MATERIAL, NAME=PLA *ELASTIC 1000 , 0.3 *SOLID SECTION , Elset=SURFACE181 , Material=STEEL *SOLID SECTION , Elset=SURFACE182 , Material=PLA *STEP *STATIC *BOUNDARY CURVE183,1,3,0 *DLOAD *include,input=CURVE184.dlo *NODE PRINT,NSET=Nall U *EL PRINT,ELSET=Eall S *NODE FILE U *EL FILE S *END STEP	
solver start	>**ccx sol**	
view result	>**cgx -v sol.frd**	 **Max Displacement = 10.2[mm]** (3D Result was 10.3[mm])

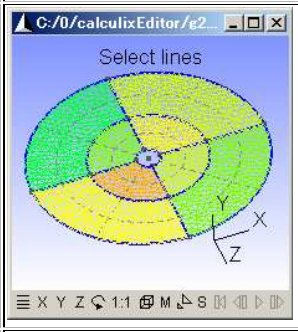
Return

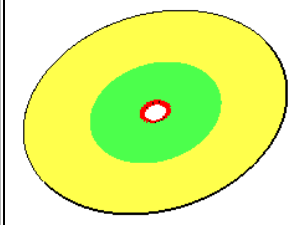
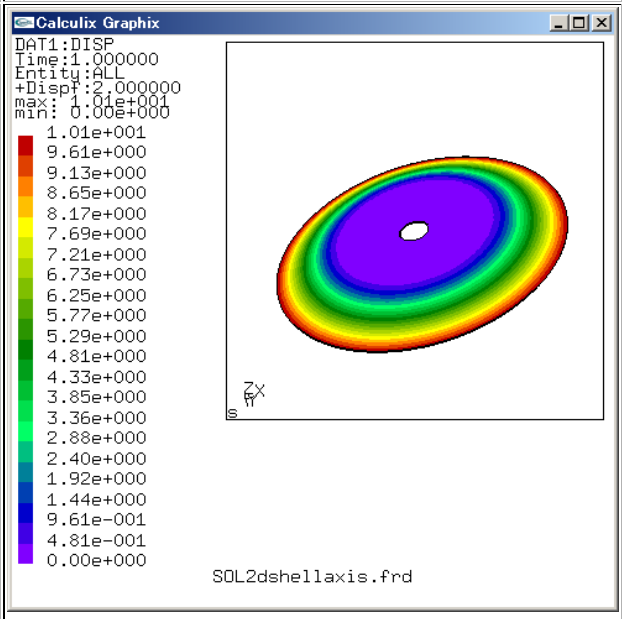
Gmsh2.5.1(lator?) (it will not work **2.5.0**)

Gmsh 2DMesh (SHELL MODEL) ---> **Export Abaqus inp ---> Calculix inp**



Make Geometry with Gmsh
 	
and define Physical groups
	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, 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};
		

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**	

	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	
make solver file (sol.inp)	*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	
solver start	>**ccx sol**	
view result	>**cgx -v sol.frd**	 **Max Displacement = 10.1[mm]** (3D Result was 10.3[mm])

Return