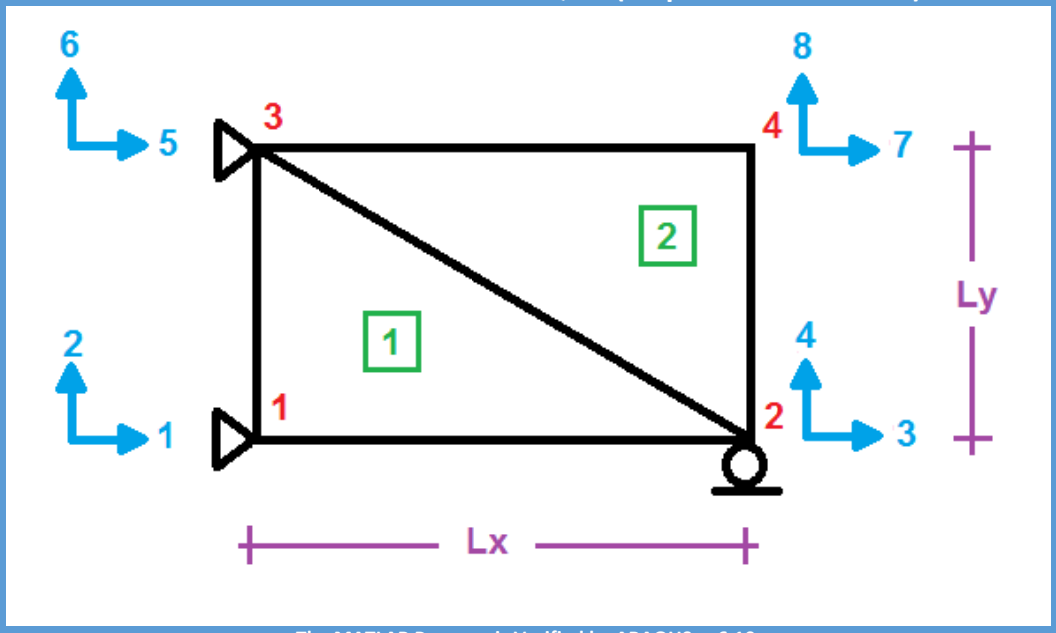
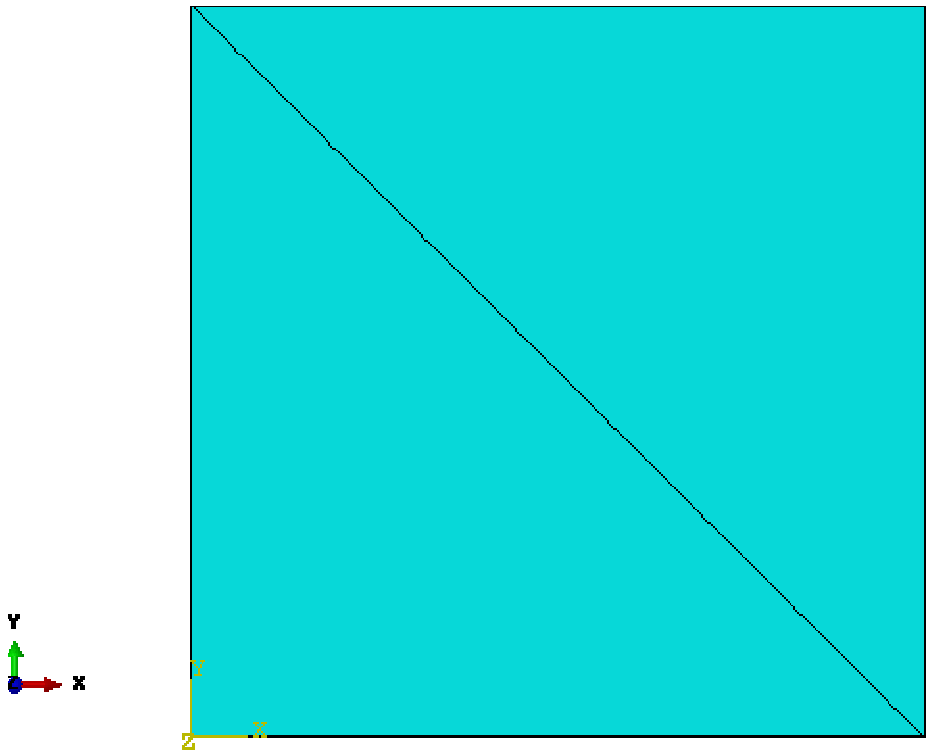
>> IN THE NAME OF GOD <<

Pushover Analysis of Triangular Steel Plate Element Subjected to Lateral Displacement with Effect of Geometric and Material Nonlinearity and Small strain in MATLAB and ABAQUS (Displacement Control)



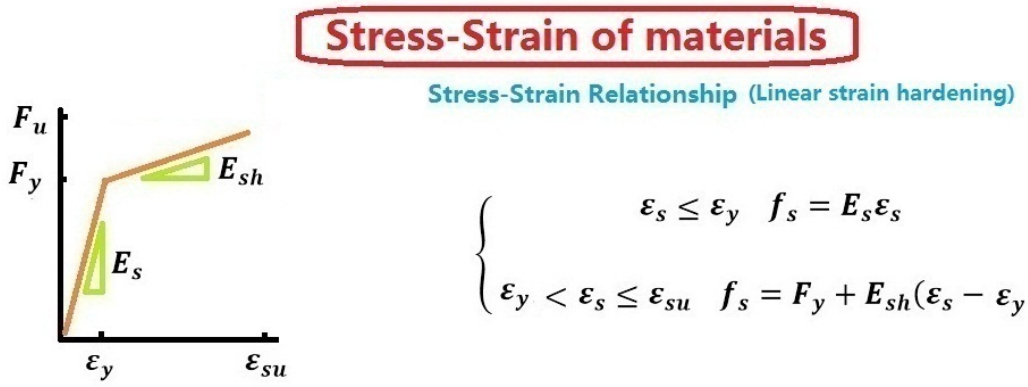
The MATLAB Program is Verified by ABAQUS v.6.10
This MATLAB program is written by Salar Delavar Ghashghaei - Date of Publication: March/23/2017
E-mail: salar.d.ghashghaei@gmail.com



Figure(1) Two elements triangular plate modelling in ABAQUS version 6.10

Define Parameters:

```
% Define Parameters in unit: [mm,kN]
P3=0; % [kN]
P7=0; % [kN]
P8=0; % [kN]
D8=.001;% [mm] Initial Displacement  [DOF (8)] Incremantal Displacement
D8max=0.8; % [mm] Maximum displacement [DOF (8)]
XY1i=[0 0]; % [x y] Point 1 Coordinate
XY2i=[200 0]; % [x y] Point 2 Coordinate
XY3i=[0 200]; % [x y] Point 3 Coordinate
XY4i=[200 200]; % [x y] Point 4 Coordinate
T=10;% [mm] triangular membrane element thickness
%% Steel Section Properties
fy =.24;% [kN/mm^2] Yield strength of steel section
Es =200;% [kN/mm^2] Modulus of elasticity of steel section
fu=1.5*fy;% Ultimate steel stress
ey=fy/Es;% Yield steel strain
esu=0.15;% Ultimate steel strain
Esh=(fu-fy)/(esu-ey);
V=.3;%  Poisson's  ratio
m = 1000; % number of calculation
itermax = 5000;% maximum number of iterations
tolerance = 1e-5; % specified tolerance for convergence
```



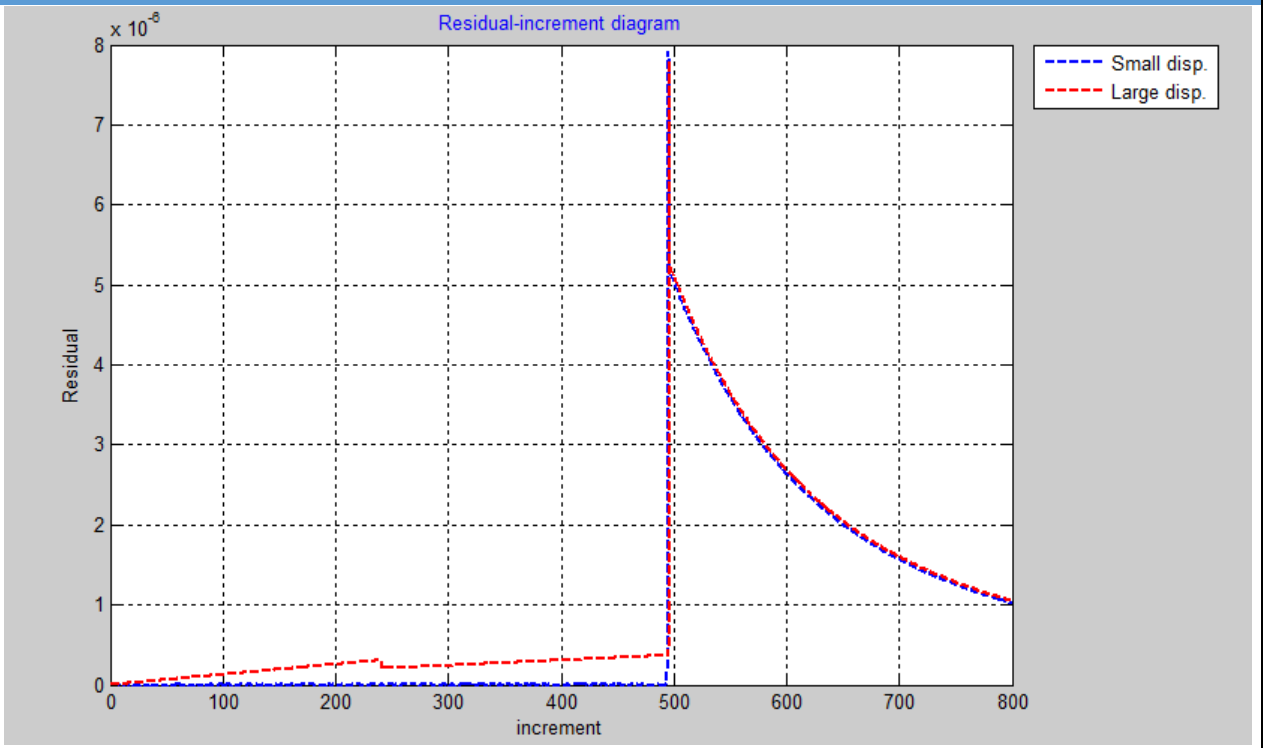
Figure(2) Bilinear stress-Strain Relation for steel modelling in MATLAB and ABAQUS version 6.10

Analysis Report:

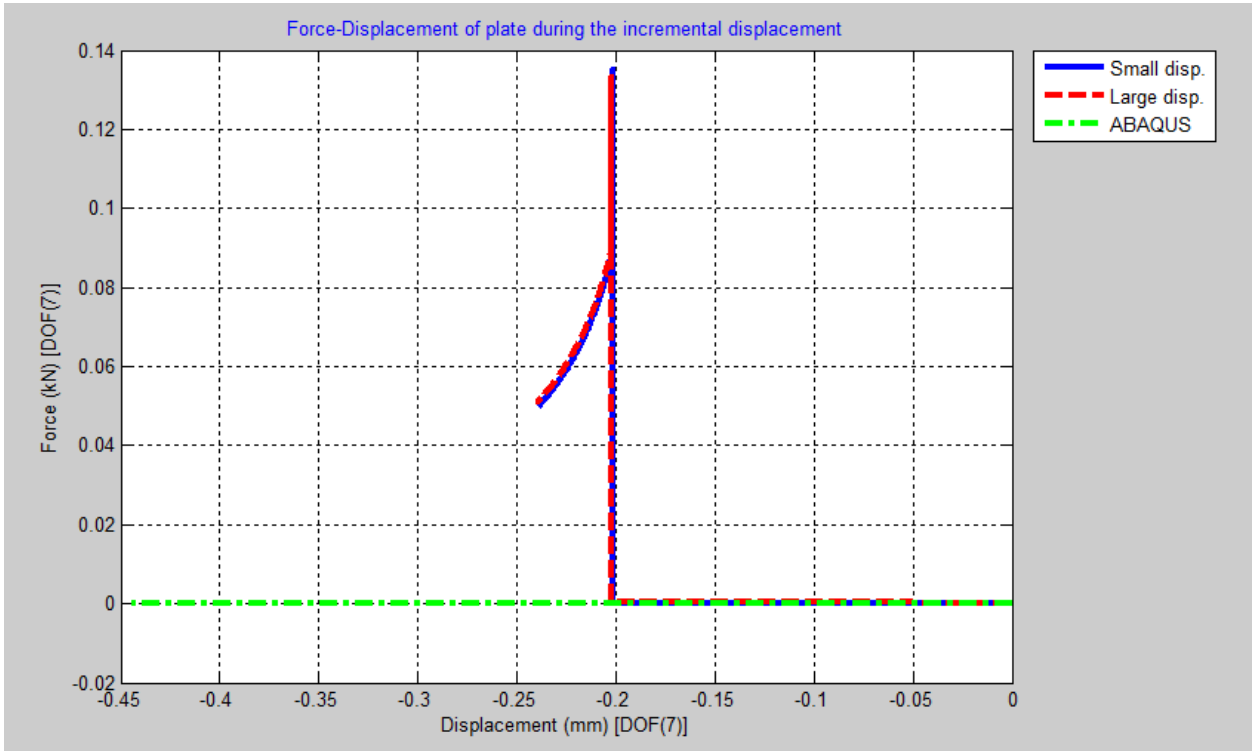
```
############################
# Small Displacement Analysis #
############################
(+)Increment 1 : It is converged in 2 iterations
(+)Increment 2 : It is converged in 2 iterations
(+)Increment 3 : It is converged in 2 iterations
(+)Increment 4 : It is converged in 2 iterations
(+)Increment 5 : It is converged in 2 iterations
(+)Increment 6 : It is converged in 2 iterations
(+)Increment 7 : It is converged in 2 iterations
(+)Increment 8 : It is converged in 2 iterations
(+)Increment 9 : It is converged in 2 iterations
(+)Increment 10 : It is converged in 2 iterations.
.
.
.
(+)Increment 791 : It is converged in 3 iterations
(+)Increment 792 : It is converged in 3 iterations
(+)Increment 793 : It is converged in 3 iterations
```

(+)Increment 794 : It is converged in 3 iterations
(+)Increment 795 : It is converged in 3 iterations
(+)Increment 796 : It is converged in 3 iterations
(+)Increment 797 : It is converged in 3 iterations
(+)Increment 798 : It is converged in 3 iterations
(+)Increment 799 : It is converged in 3 iterations
(+)Increment 800 : It is converged in 3 iterations
############################
 # Large Displacement Analysis #
############################
(+)Increment 1 : It is converged in 2 iterations
(+)Increment 2 : It is converged in 2 iterations
(+)Increment 3 : It is converged in 2 iterations
(+)Increment 4 : It is converged in 2 iterations
(+)Increment 5 : It is converged in 2 iterations
(+)Increment 6 : It is converged in 2 iterations
(+)Increment 7 : It is converged in 2 iterations
(+)Increment 8 : It is converged in 2 iterations
(+)Increment 9 : It is converged in 2 iterations
(+)Increment 10 : It is converged in 2 iterations
.
.
.
(+)Increment 791 : It is converged in 3 iterations
(+)Increment 792 : It is converged in 3 iterations
(+)Increment 793 : It is converged in 3 iterations
(+)Increment 794 : It is converged in 3 iterations
(+)Increment 795 : It is converged in 3 iterations
(+)Increment 796 : It is converged in 3 iterations
(+)Increment 797 : It is converged in 3 iterations
(+)Increment 798 : It is converged in 3 iterations
(+)Increment 799 : It is converged in 3 iterations
(+)Increment 800 : It is converged in 3 iterations

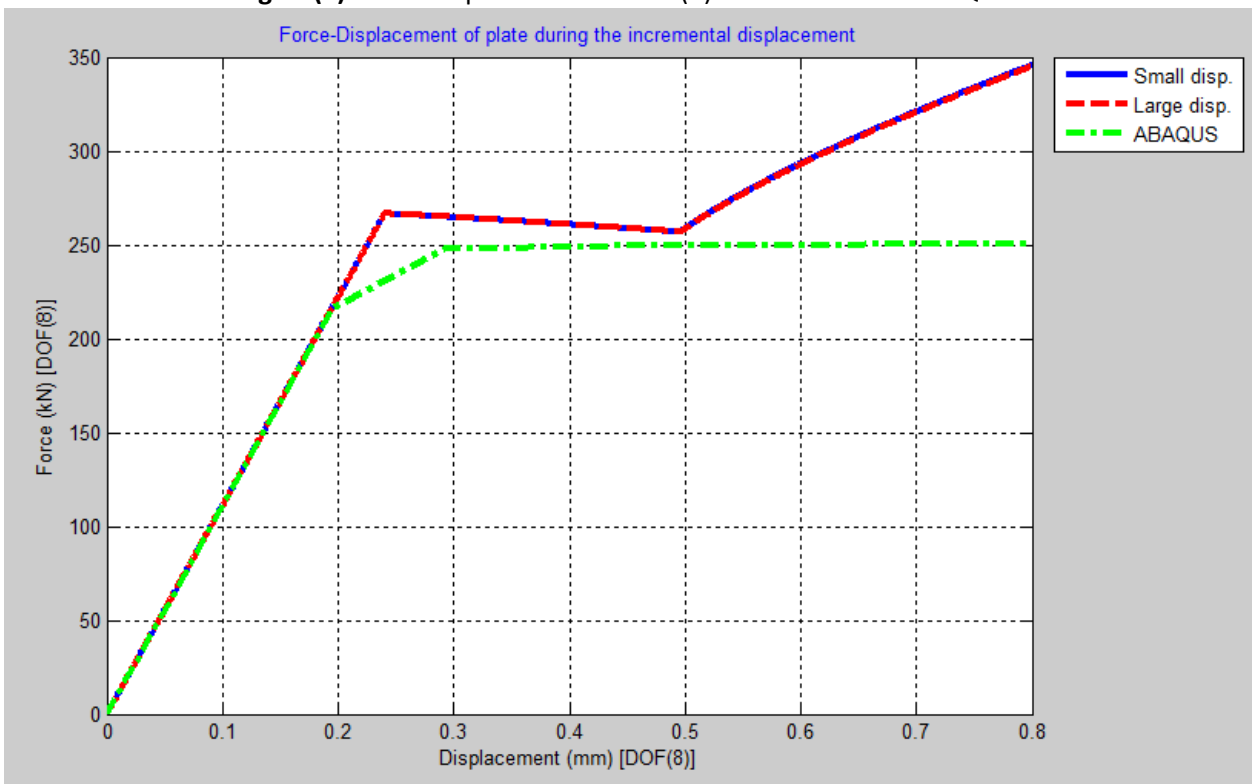
Plot :



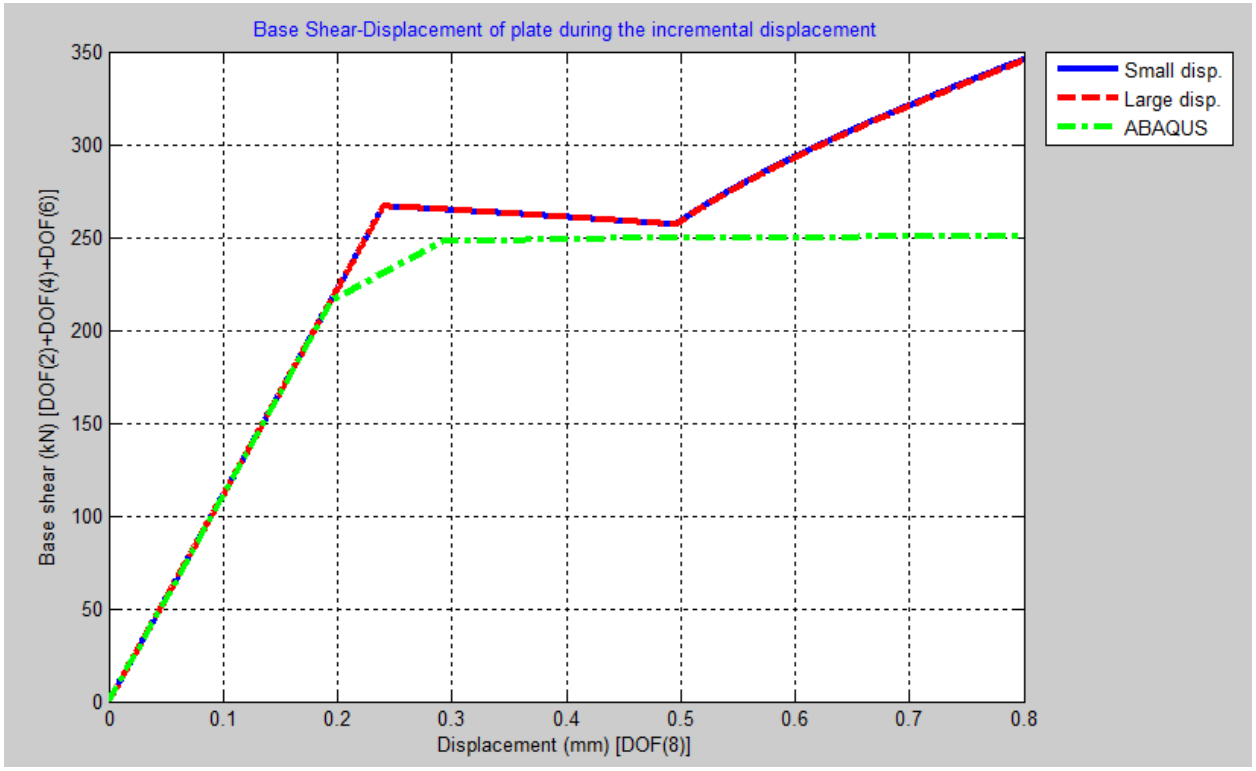
Figure(3) Residual-increment diagram



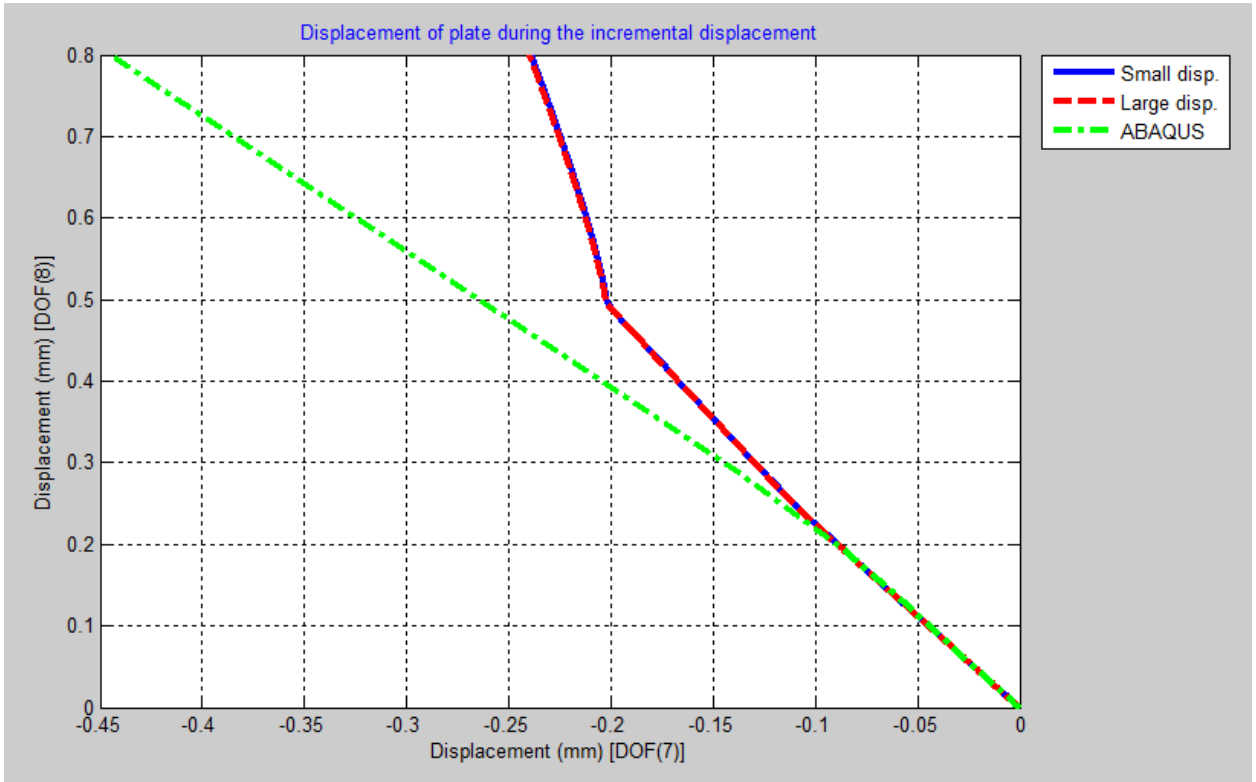
Figure(4) Force-Displacement of DOF(7) in MATLAB and ABAQUS



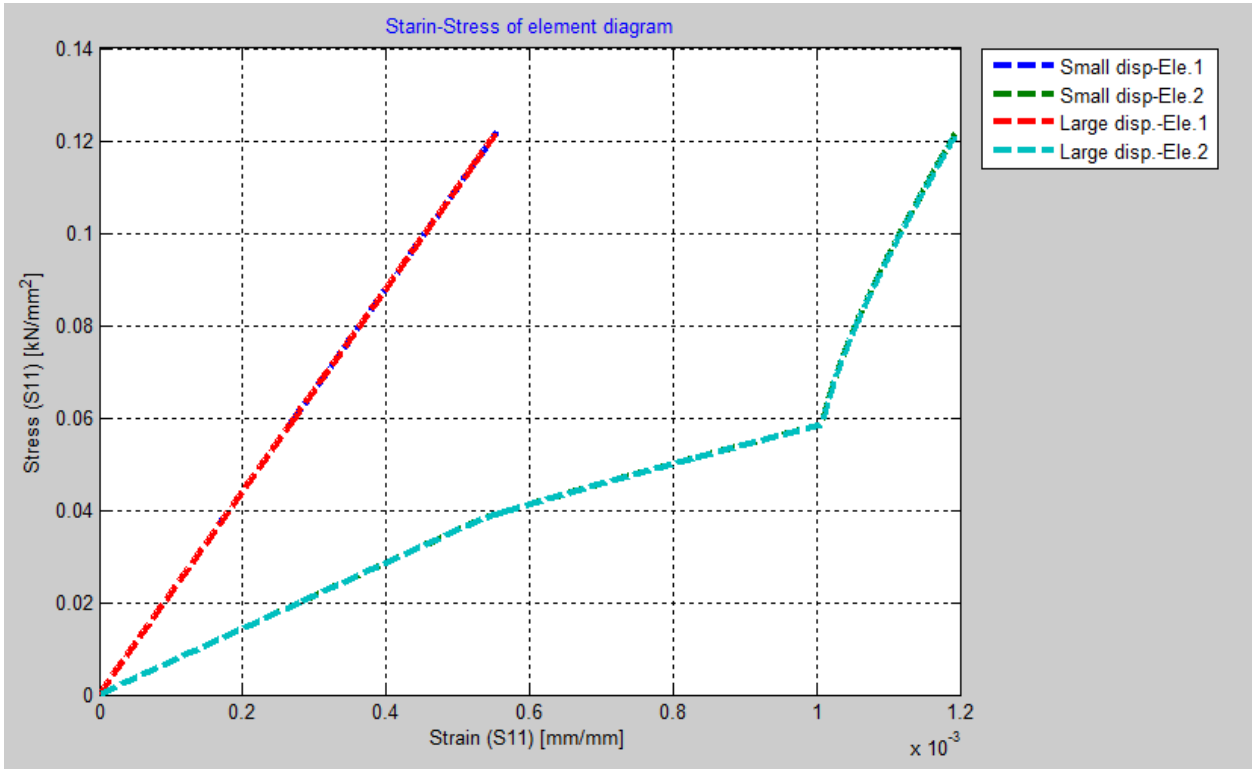
Figure(5) Force-Displacement of DOF(8) in MATLAB and ABAQUS



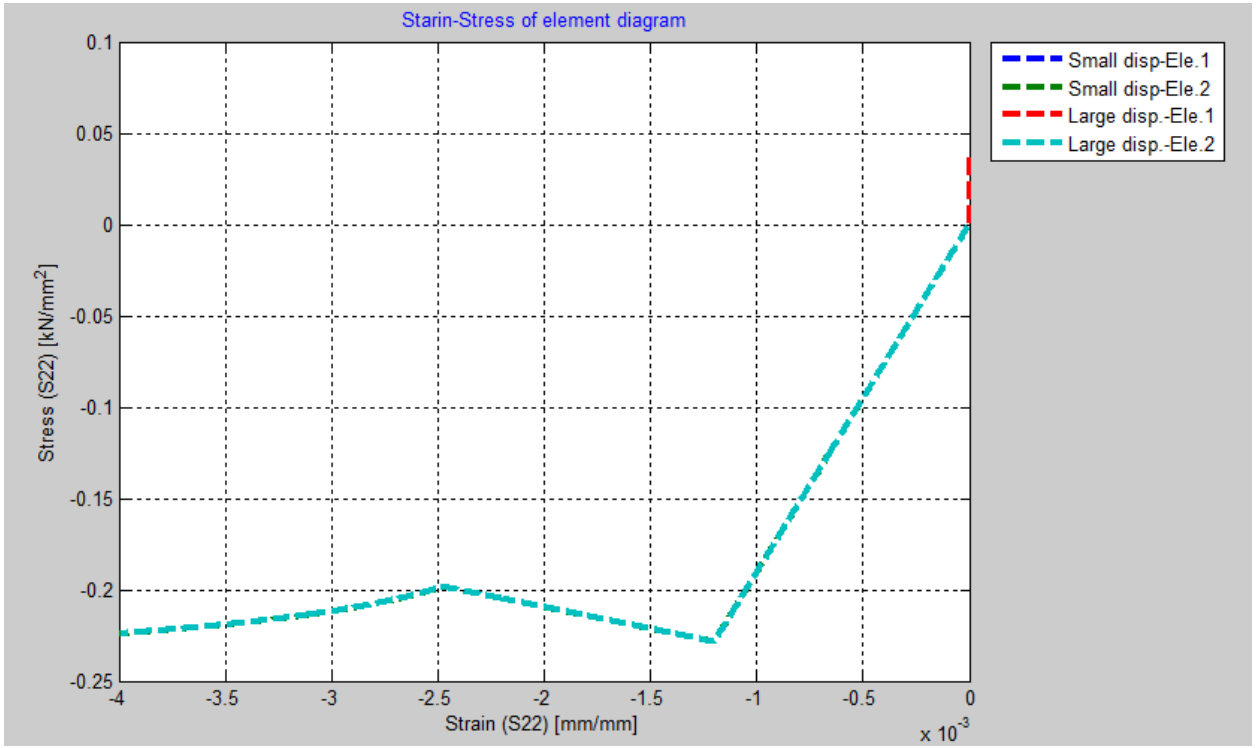
Figure(6) Base shear-Displacement of DOF(8) in MATLAB and ABAQUS



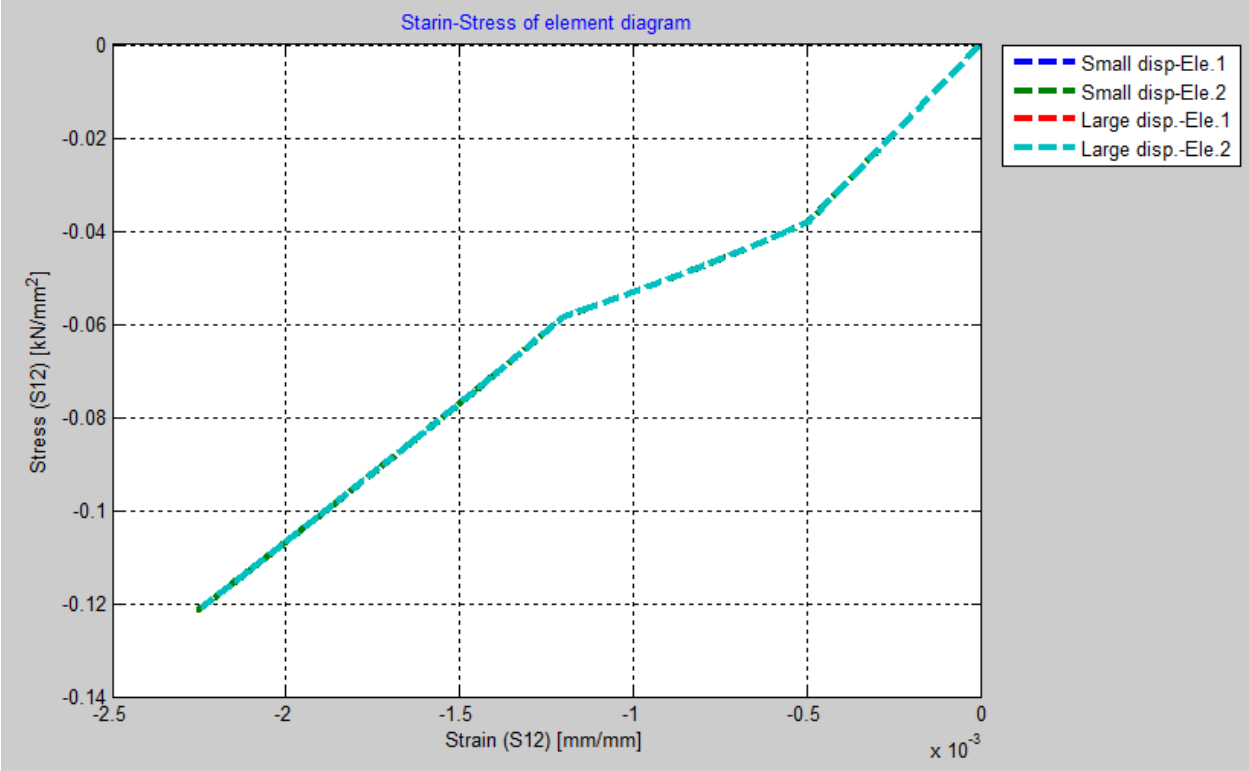
Figure(7) Displacement-X and Y during incremental displacement in MATLAB and ABAQUS



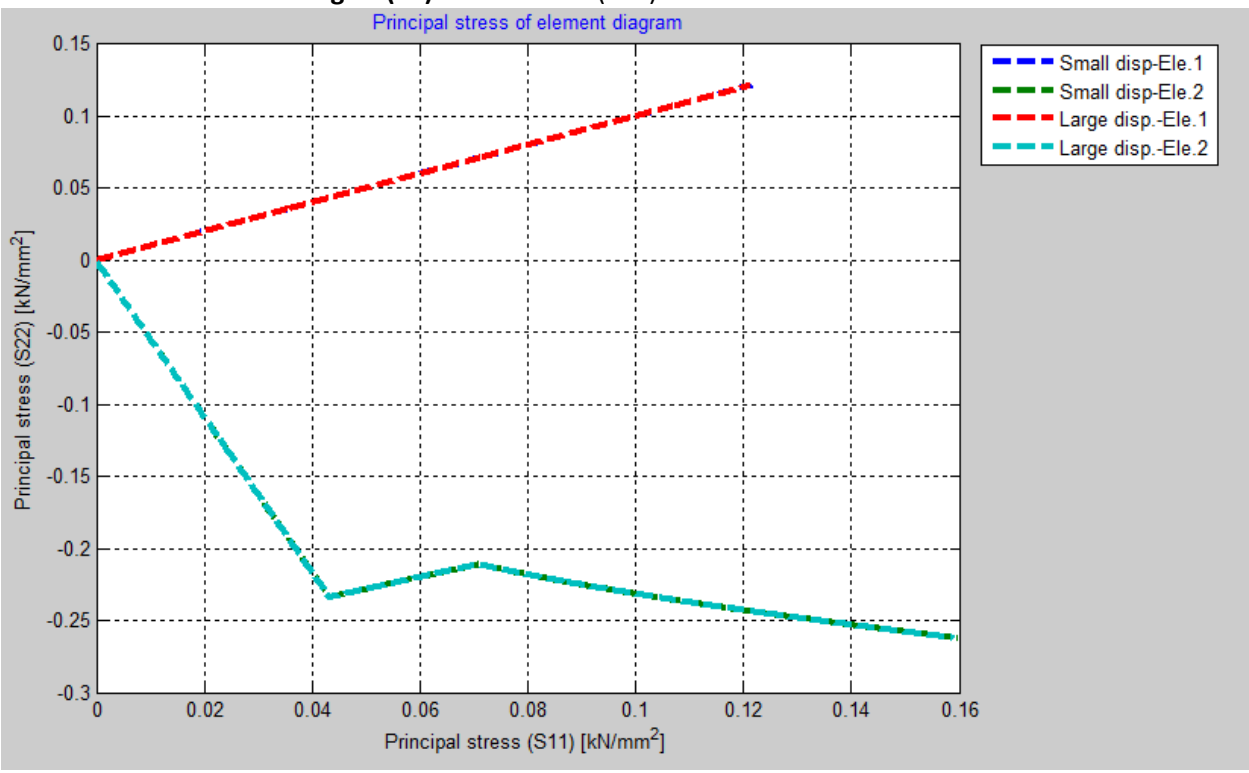
Figure(8) Strain-Stress (S11) of element in MATLAB



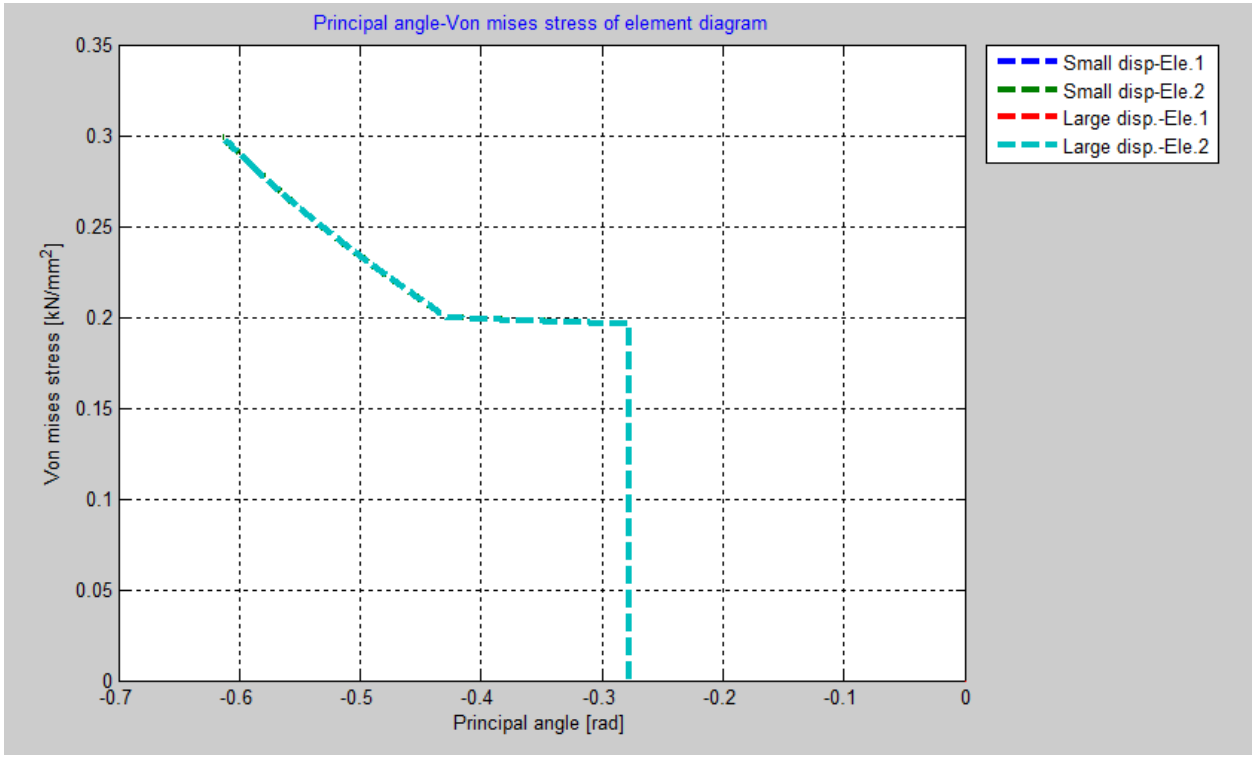
Figure(9) Strain-Stress (S22) of element in MATLAB



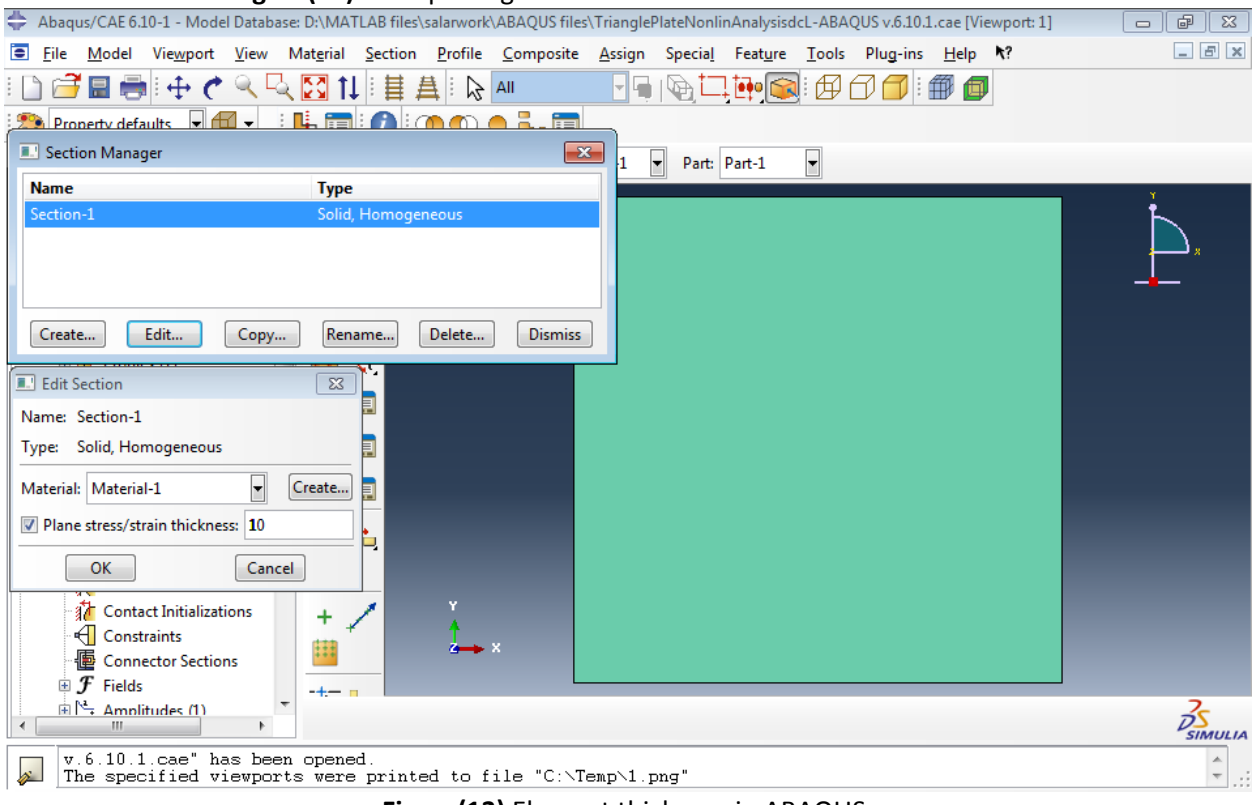
Figure(10) Strain-Stress (S12) of element in MATLAB



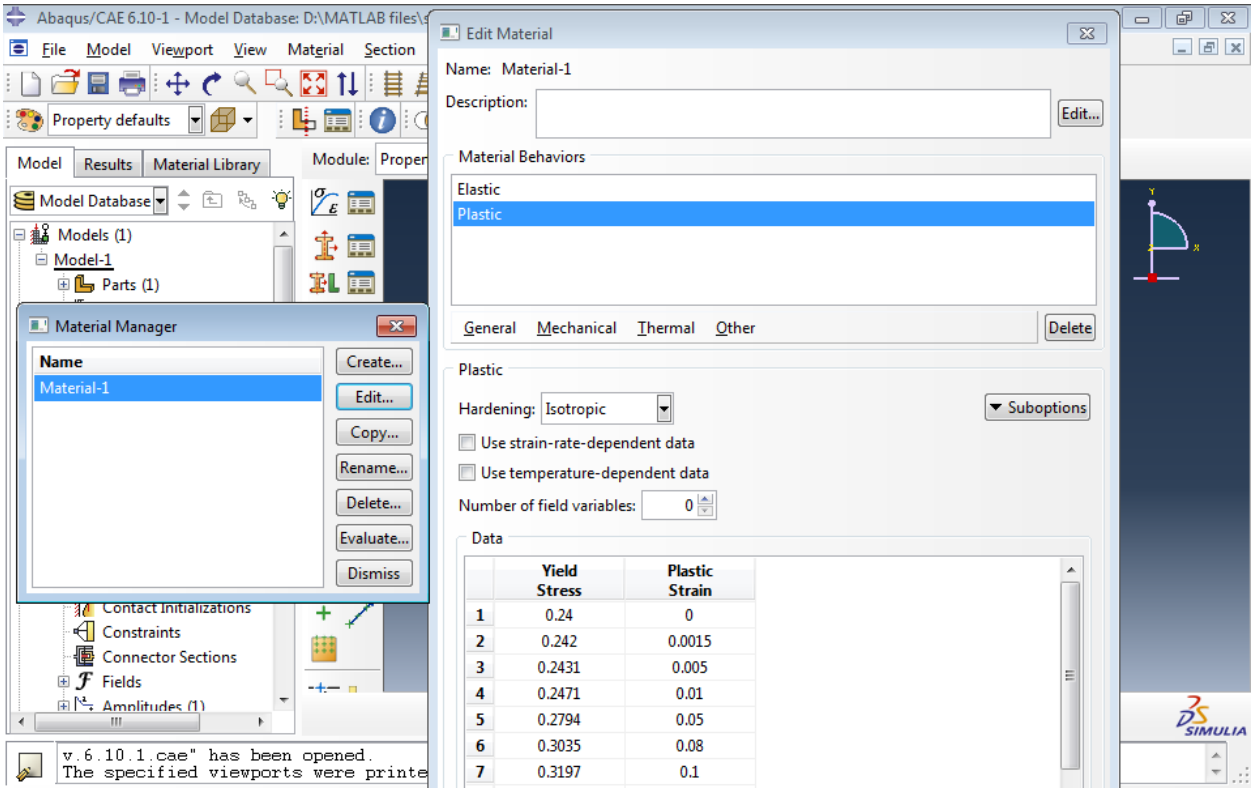
Figure(11) Principal stress of element in MATLAB



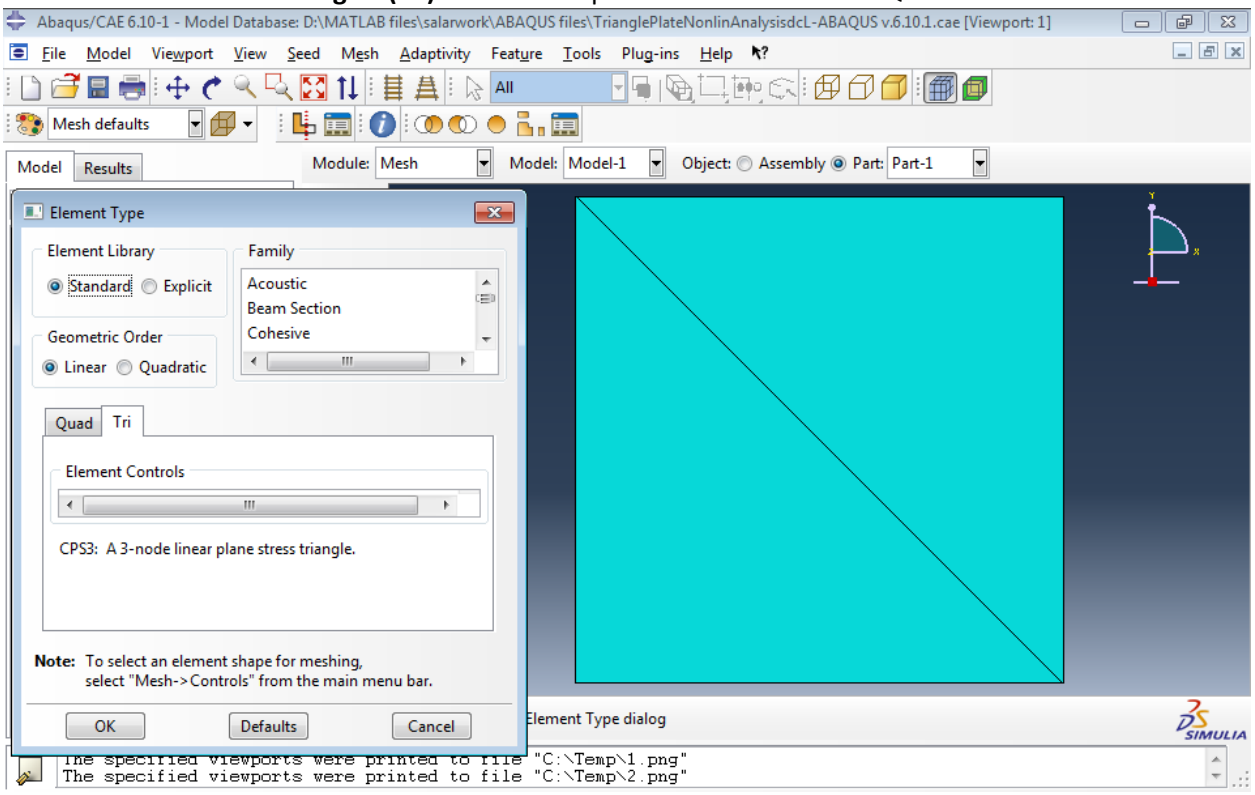
Figure(12) Principal angle-Von mises stress of element in MATLAB



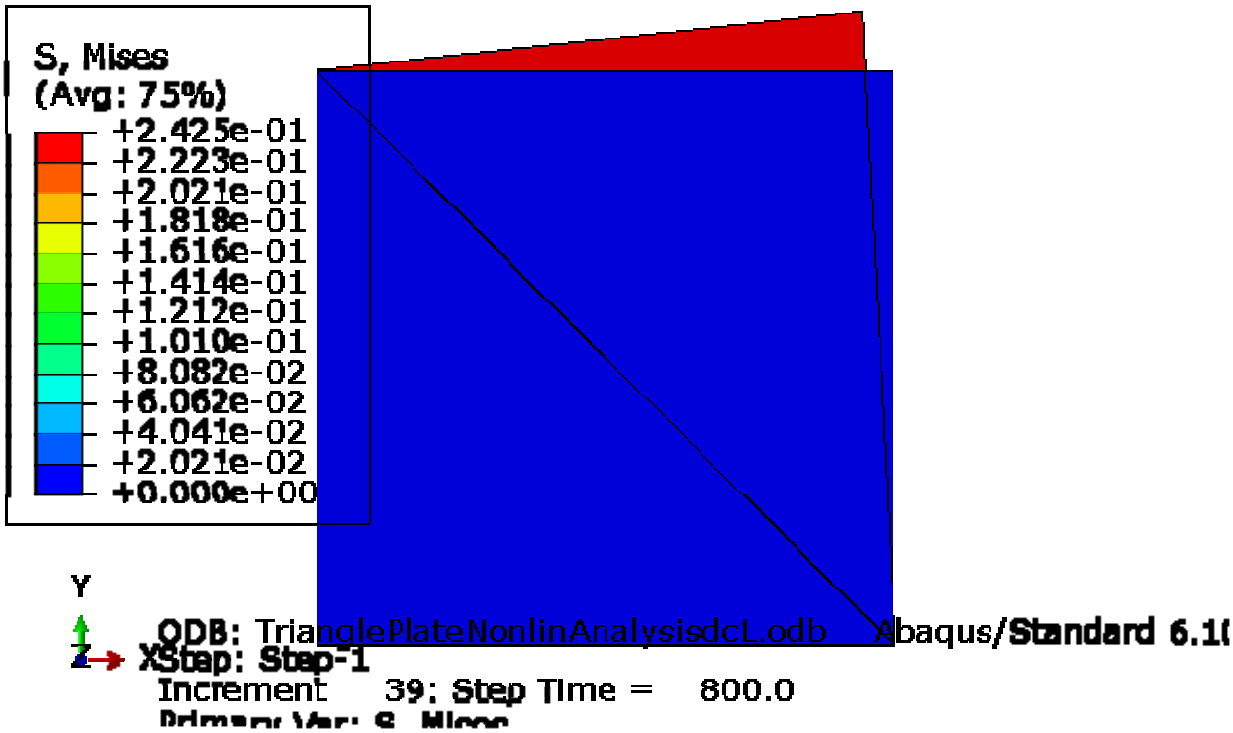
Figure(13) Element thickness in ABAQUS



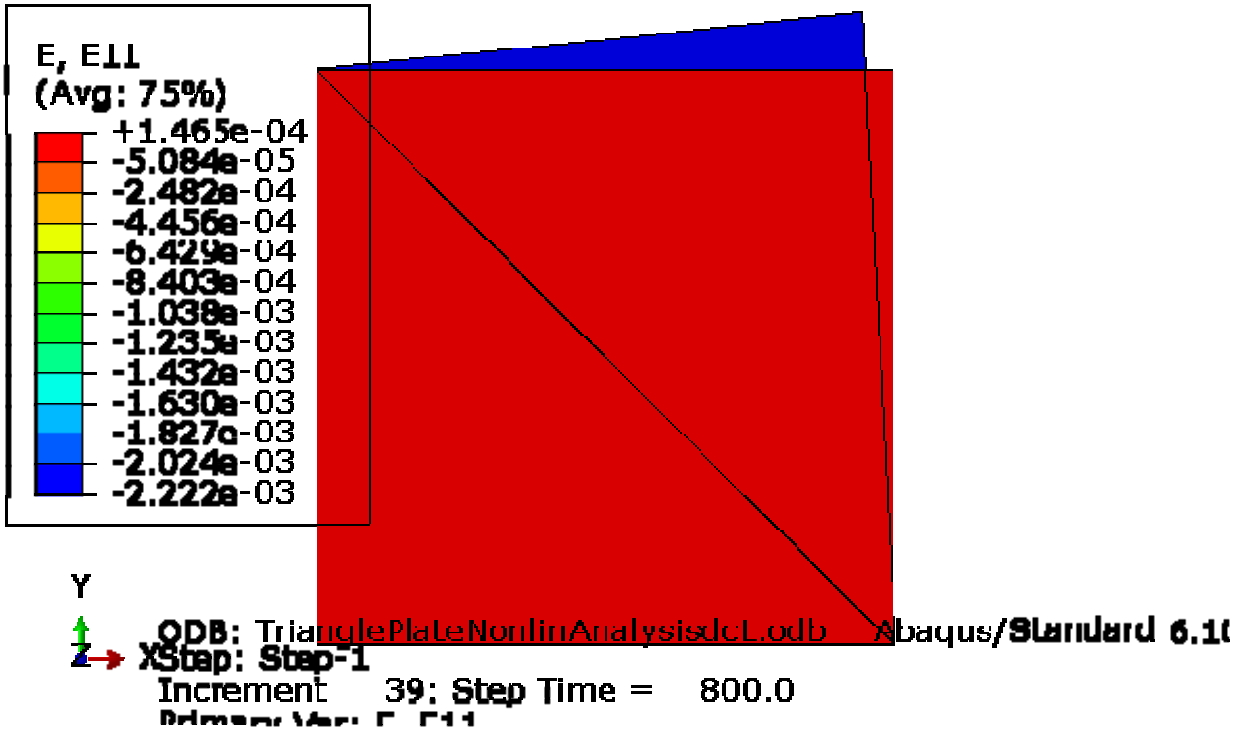
Figure(14) Material Properties definition in ABAQUS



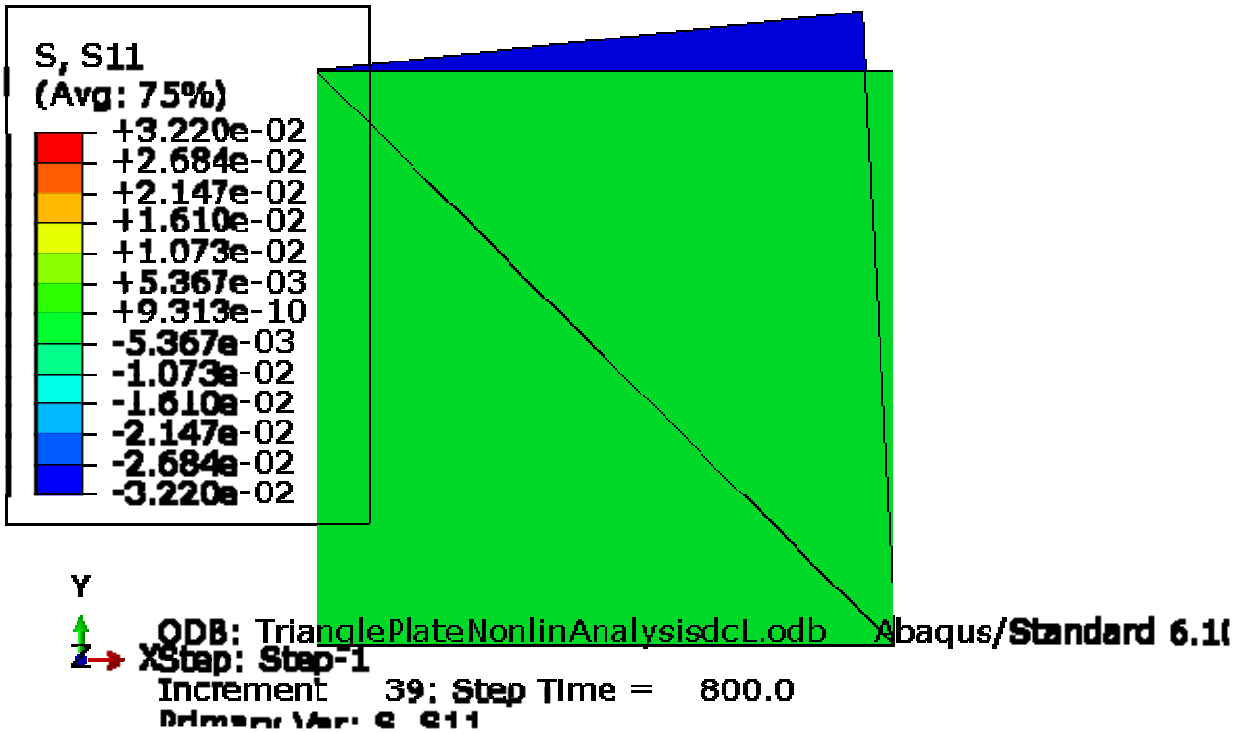
Figure(15) Mesh Properties definition in ABAQUS



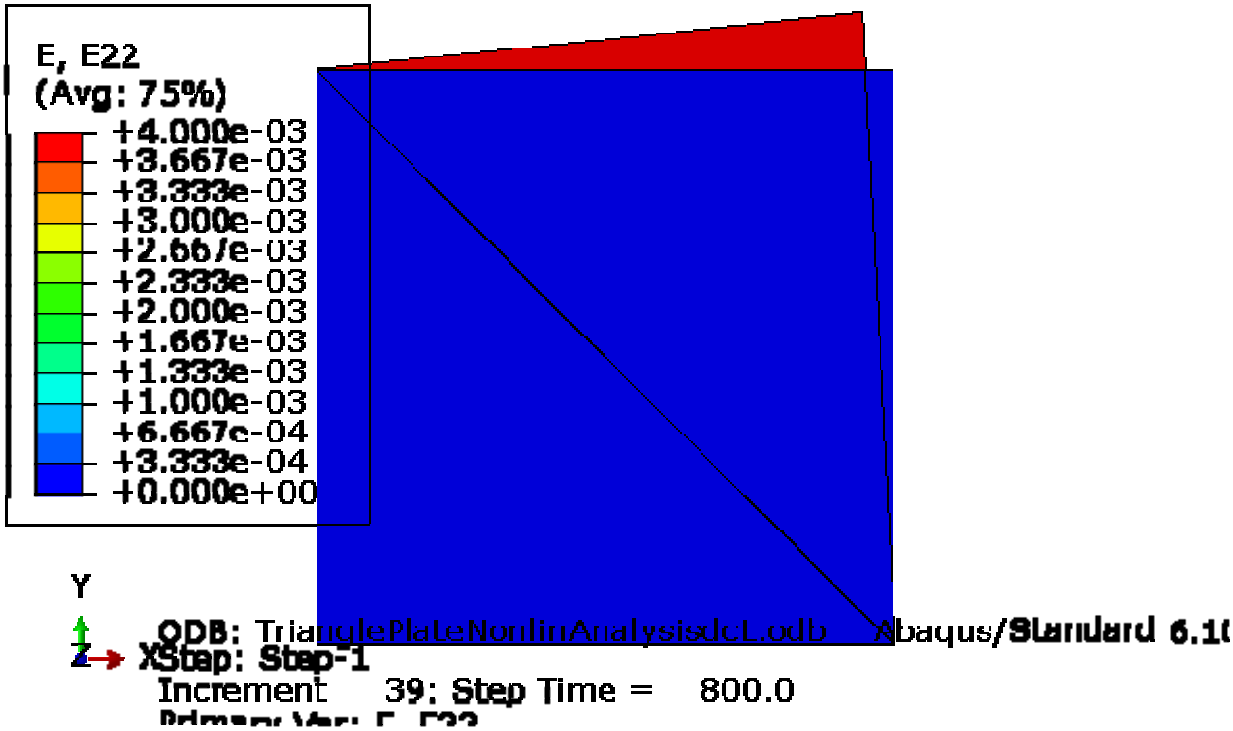
Figure(16) Von Mises stress of element in ABAQUS



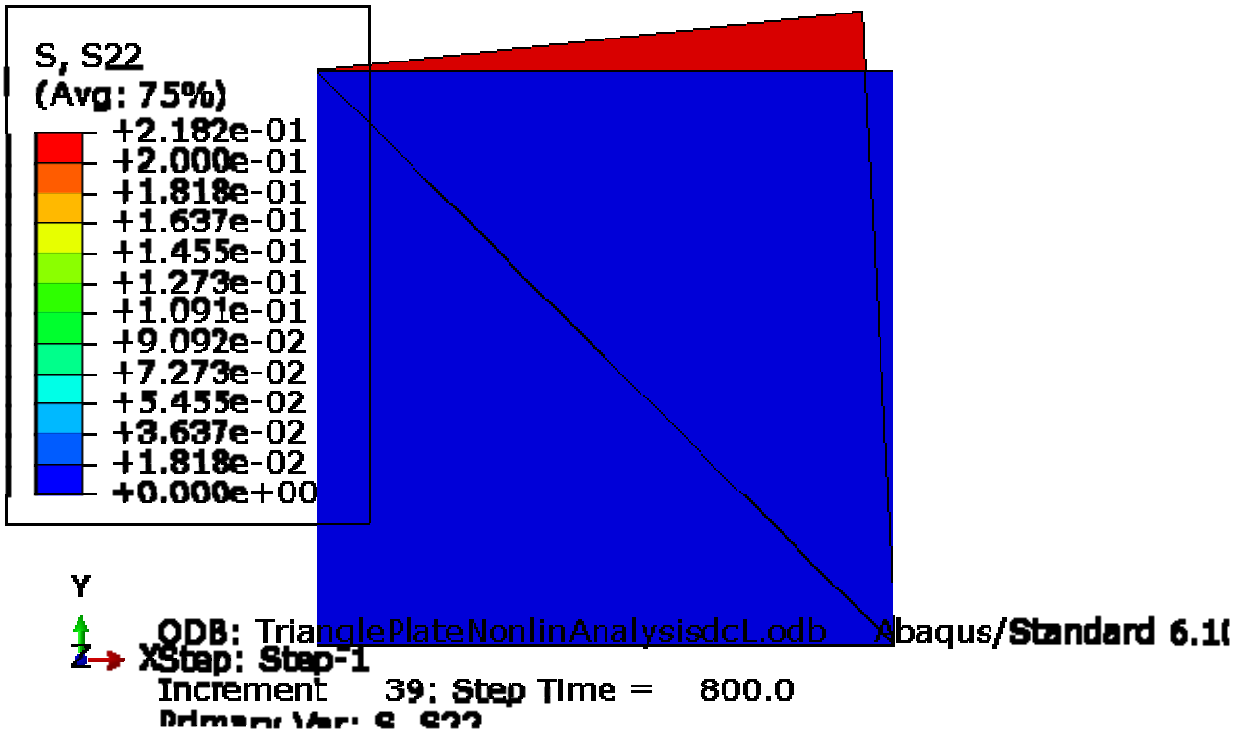
Figure(17) Strain (E11) of element in ABAQUS



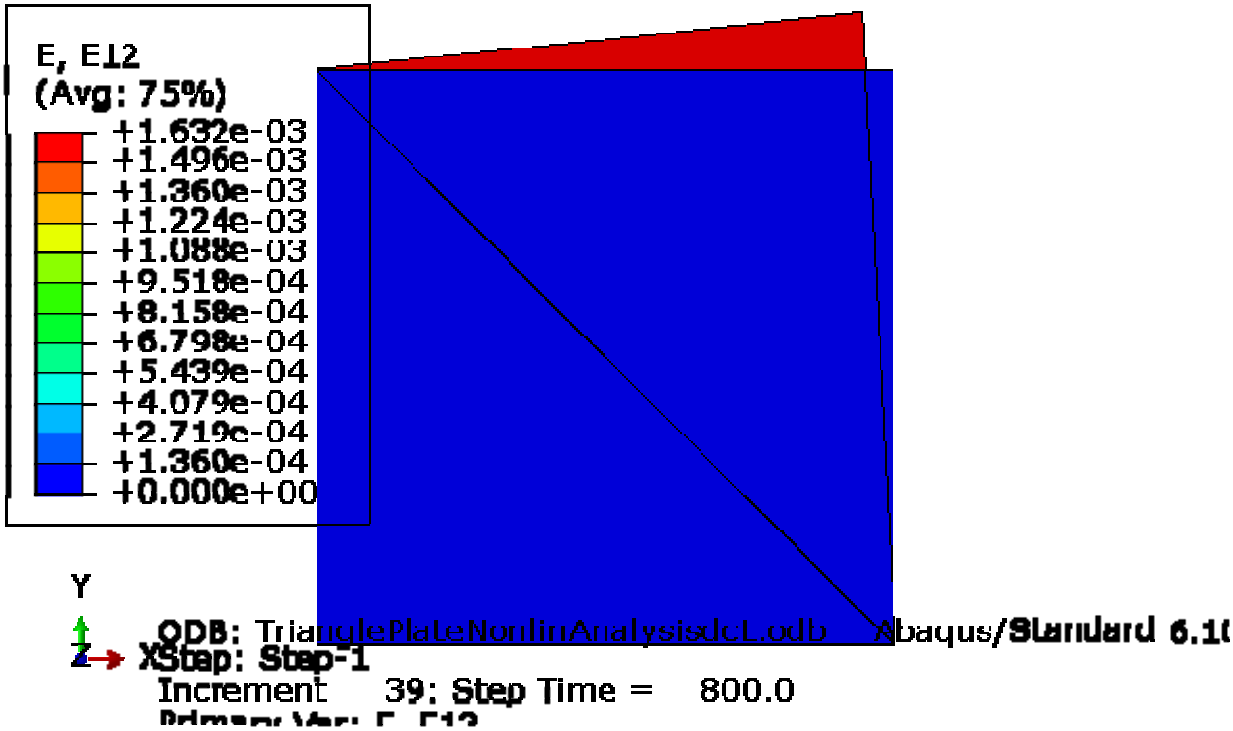
Figure(18) Stress (S11) of element in ABAQUS



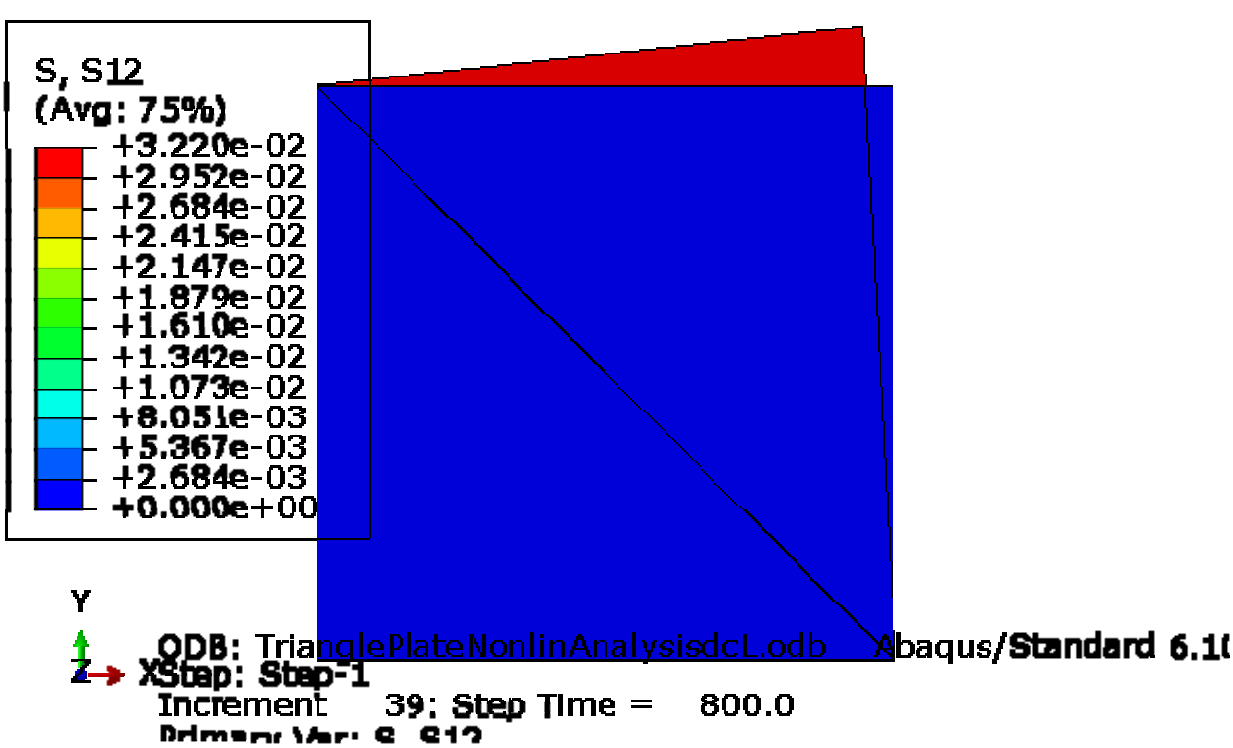
Figure(19) Strain (E22) of element in ABAQUS



Figure(20) Stress (S22) stress of element in ABAQUS



Figure(21) Strain (E12) of element in ABAQUS



Figure(22) Stress (S12) of element in ABAQUS