1 Aim

This test case is for checking the capability of the written Isogeometric analysis code of a 2D Piezoelectric plate under mechanical loading

2 Problem description

Section 7.3.1 in Documentation
A 2D piezoelectric plate subjected to mechanical displacements is considered, as shown in Fig. (2) . The material used is PZT-PIC151 ceramics.

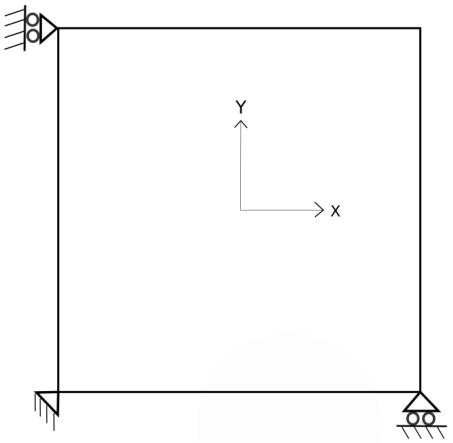

Figure 1: 2D Piezoelectric Plate

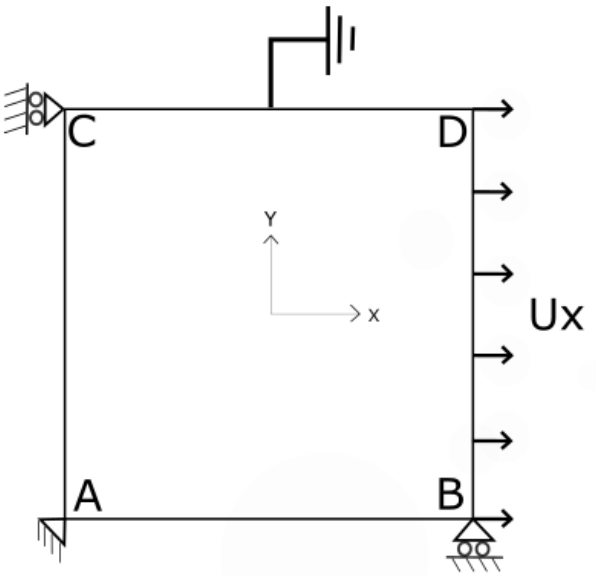

Figure 2: 2D Piezoelectric Plate with loading

The movement of the bottom edge AB and left edge AC of 2D piezoelectric plate is fixed in y-direction and x-direction respectively, as shown in figure(2). The top edge CD is grounded (Electric potential $\Phi = 0$), and a displacement load of 100 nm (1e-4 mm) is applied on the right edge BD. The results for a single element case are discussed in the below sections.
The results generated by IGA code are compared with inbuilt Abaqus piezoelectric element **CPE4E**.

3 How to run the Program

1. The code is written in python and external libraries numpy, matplotlib.pyplot, sys, path from pathlib and math are used.

2. Please use any environment which will compile python programs

3. Place all the files in a single folder.

4. A file named Input.py can be edited to change the dimensions of the plate. User can change Length, Height and Thickness of the plate.
   (The results discussed below are for Length = 10 mm, Height = 10 mm and Thickness = 1 mm)

```
#----------------------------------Dimensions of 2D Plate-------------------------------------#
Thick   = 1.0  #Thickness of the plate in mm
Length  = 10.0 #Length of the plate    in mm
Height  = 10.0 #Height of the plate    in mm
```

5. Before you run the file, please make sure that the working directory is same as the folder which Consists the Program.

6. Use command >>> python Main_Program.py to run the program.

7. The contour plots will be saved in the folder **Results.**

8. A "log.txt" file is created in the same folder which contains the values of the results plotted.

4 Results and discussions

Section 7.3.3 in Documentation
Abaqus plane strain full integration piezoelectric element (**CPE4E**) is used for analysis. The below figures show the values of displacements (U), electrical potentials (EPOT) and the reaction forces (RF) for both Abaqus and IGA element.

Figure(3) and Figure(4) show the displacement (U1) values of the single CPE4E element and single IGA element at 100 % loading in x-direction respectively.

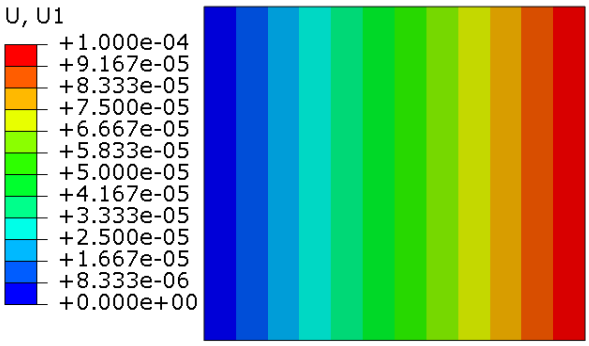

Figure 3: CPE4E Element:U1
Abaqus generated result

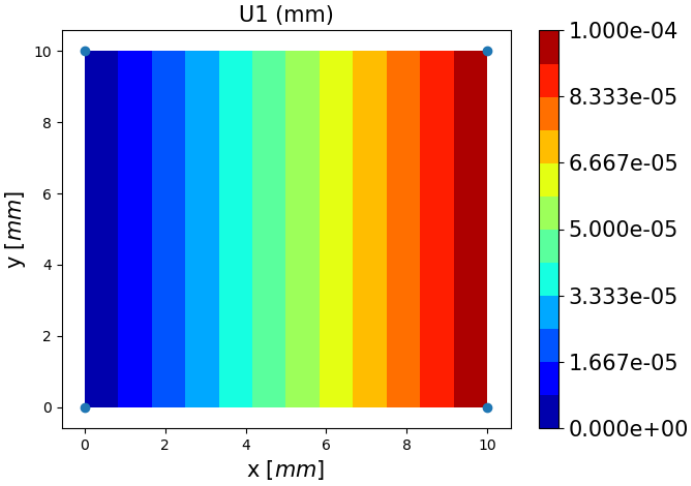

Figure 4: IGA Piezoelectric Element:U1
Program generated result

Figure(5) and Figure(6) show the displacement (U2) values of the single CPE4E element and single IGA element at 100 % loading in y-direction respectively.

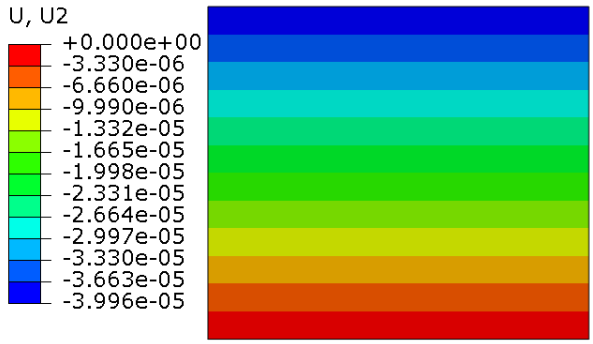

Figure 5: CPE4E Element:U2
Abaqus generated result

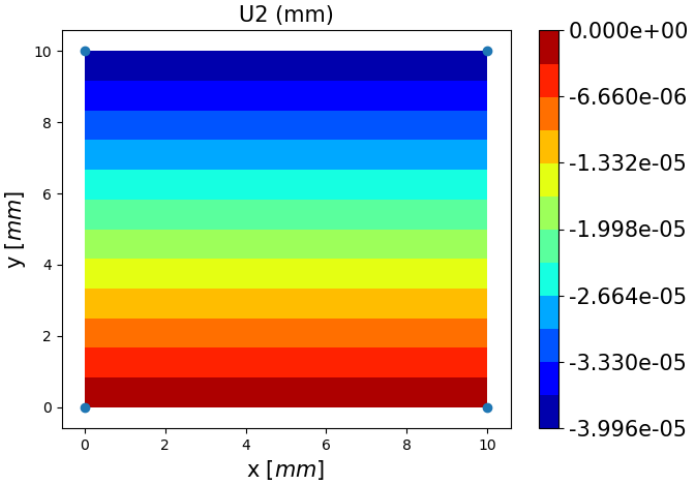

Figure 6: IGA Piezoelectric Element:U2
Program generated result

Figure(7) and Figure(8) show the Reaction force values (RF1) of the single CPE4E element and single IGA element at 100 % loading in x-direction respectively.

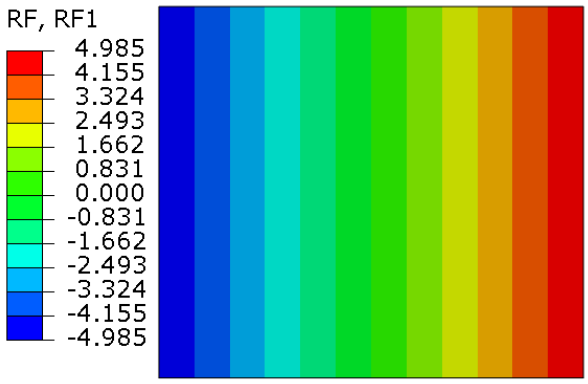

Figure 7: CPE4E Element:RF1
Abaqus generated result

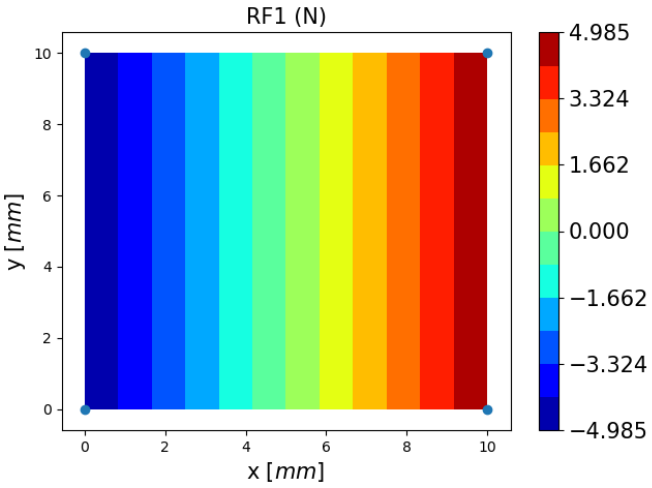

Figure 8: IGA Piezoelectric Element:RF1
Program generated result

Figure(9) and Figure(10) show the Electrical potential values (EPOT) of the single CPE4E element and single IGA element at 100 % loading respectively.

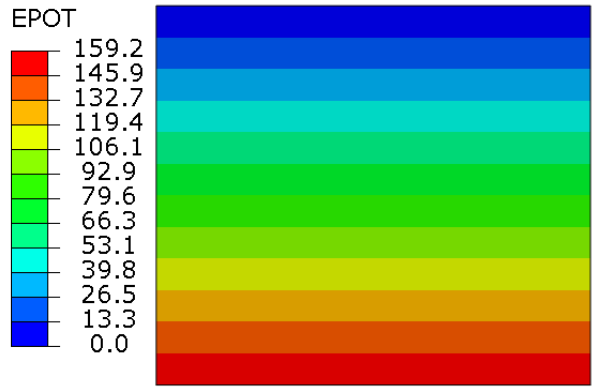

Figure 9: CPE4E Element:EPOT
Abaqus generated result

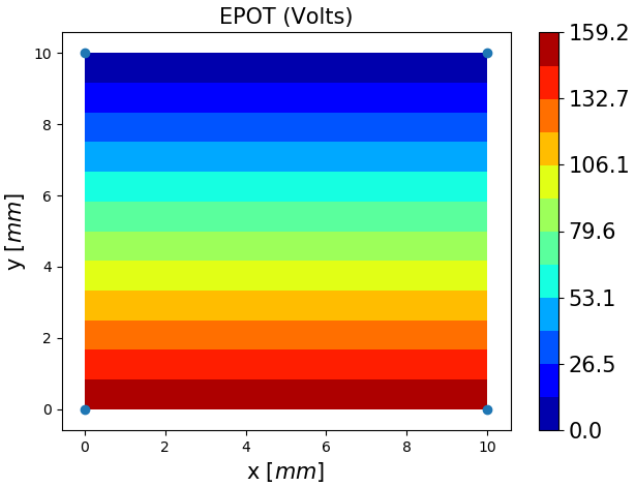

Figure 10: IGA Piezoelectric Element:EPOT
Program generated result