1 Aim

This test case is for checking the capability of the written Isogeometric analysis code with a linear elastic loading.

2 Problem description

Section 7.1.1 in Documentation
A 2D plate is subjected to mechanical loading as shown in Figure(2). The material used is PZT-PIC151 ceramics. The movement of bottom edge AB is fixed in y-direction and left edge AC in x-direction. A displacement of 100 nm (1e-4 mm) is given on the right edge BD.

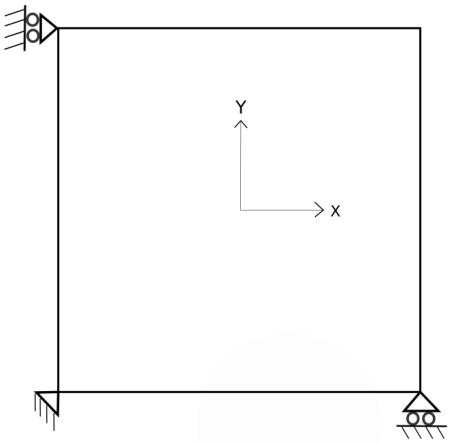

Figure 1: 2D Plate

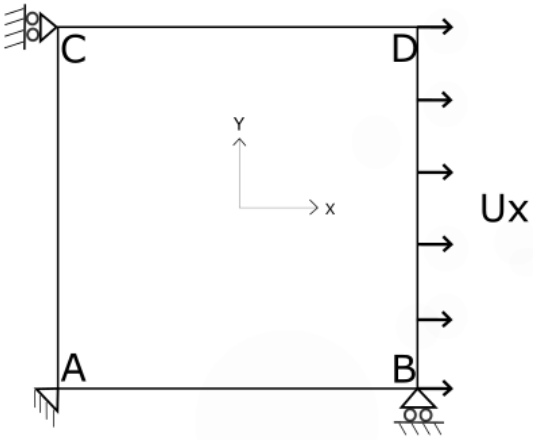

Figure 2: 2D Plate with loading

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

Considering the accuracy of the IGA simulation results over FEM results, IGA code generated results can be compared with the Abaqus inbuilt element generated results. An Abaqus plane strain full integration element (**CPE4**) is used for this purpose.
The below figures show the values of displacements (U) and reaction forces (RF) for both Abaqus and IGA element.

A similar contour is used for the program generated results and Abaqus results for easy comparison.
Figure(3) and Figure(4) show the displacement (U1) values of the single CPE4 element and single IGA element at 100 % loading in x-direction respectively.

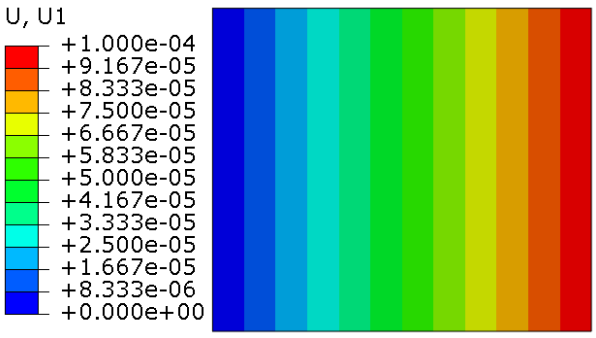

Figure 3: CPE4 Element:U1
Abaqus generated result

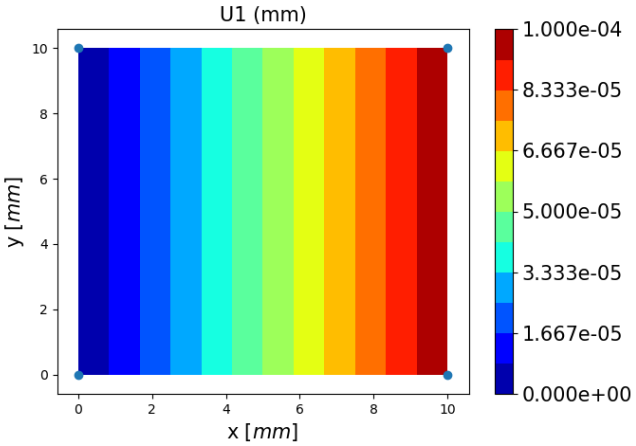

Figure 4: IGA Element:U1
Program generated result

Figure(5) and Figure(6) show the displacement (U2) values of the single CPE4 element and single IGA element at 100 % loading in y-direction respectively.

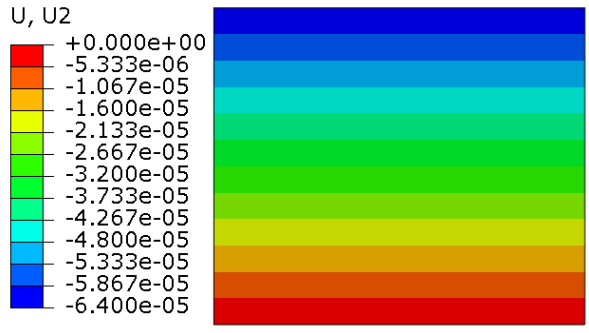

Figure 5: CPE4 Element:U2
Abaqus generated result

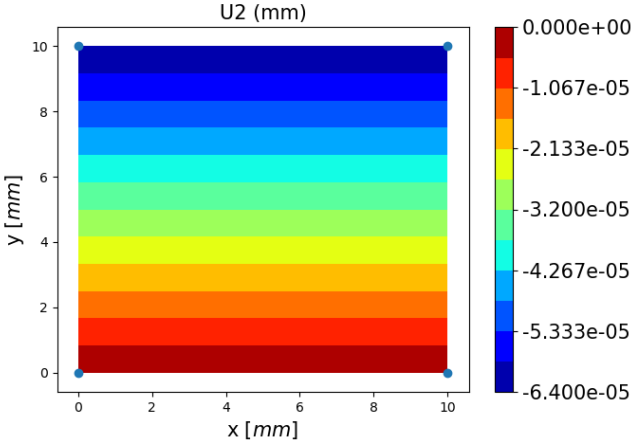

Figure 6: IGA Element:U2
Program generated result

Figure(7) and Figure(8) show the Reaction force values (RF1) of the single CPE4 element and single IGA element at 100 % loading in x-direction respectively.

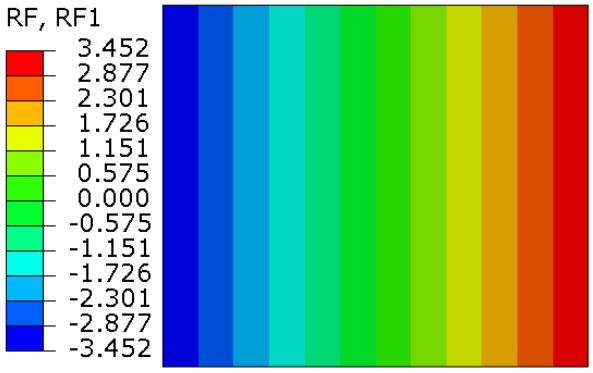

Figure 7: CPE4 Element:RF1
Abaqus generated result

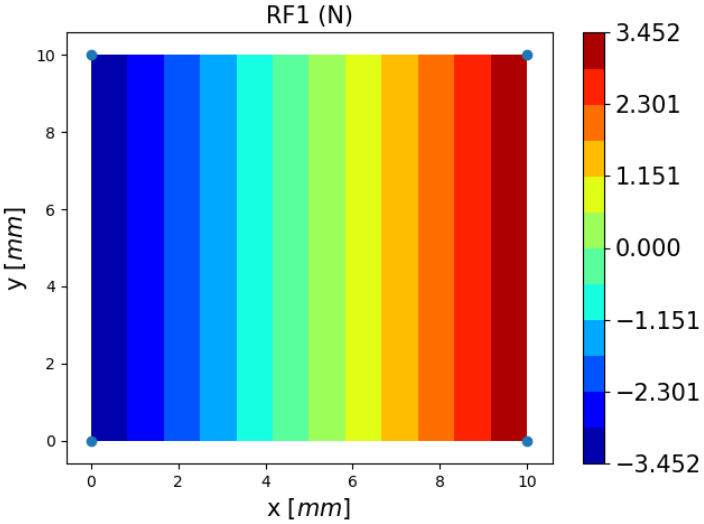

Figure 8: IGA Element:RF1
Program generated result

Reaction force in the y-direction is not reported since the loading is given only in the x-direction, and for the given fixed BCS, the RF2 is negligible.