Sorbonne Université

Estimation of the displacement at the top of a gravity dam thanks to various modeling techniques.

Authors :
Alexandre RICHARD
Valentin Duvivier

Supervisors :
Sylvain Patinet

December 14, 2020

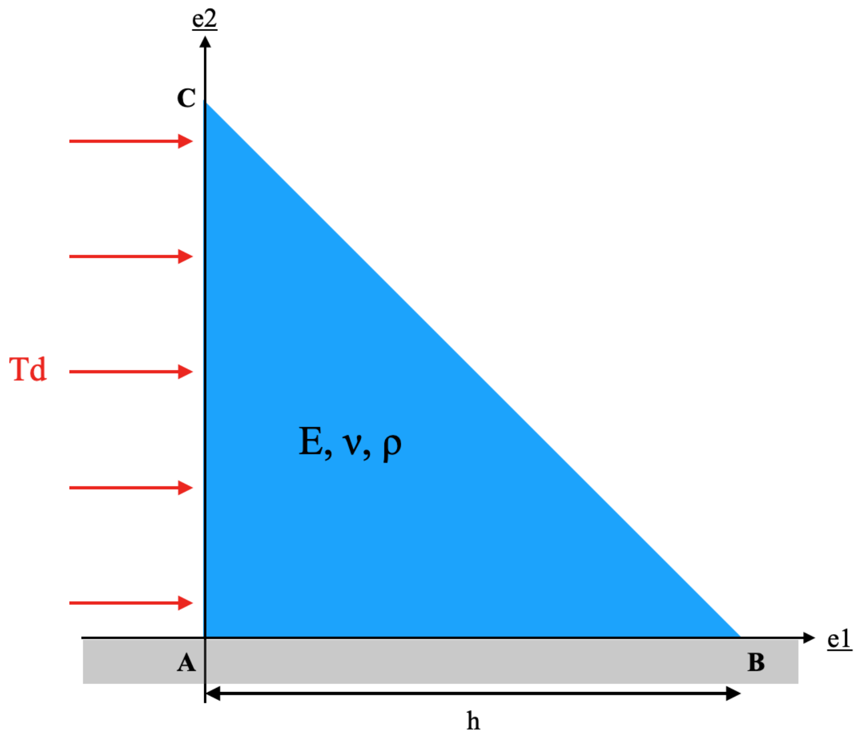
Contents

1 Introduction

We would like to study a prism-shaped gravity dam with a triangular section. This section is more precisely an isosceles right-angled triangle. The identical sides of the triangle are AB = AC = h = 15 m.

The third dimension of the prism is the longest dimension of the dam and is parallel to the third vector of the Cartesian spatial basis ($\underline{e_1}$, $\underline{e_2}$, $\underline{e_3}$). Moreover, this length is considered very high compared to the section dimensions and thereby the plane strains hypothesis can be assumed as true.

The section ABC defined in the plane orthogonal to $\underline{e_3}$ and represented on the figure below is consequently now considered :



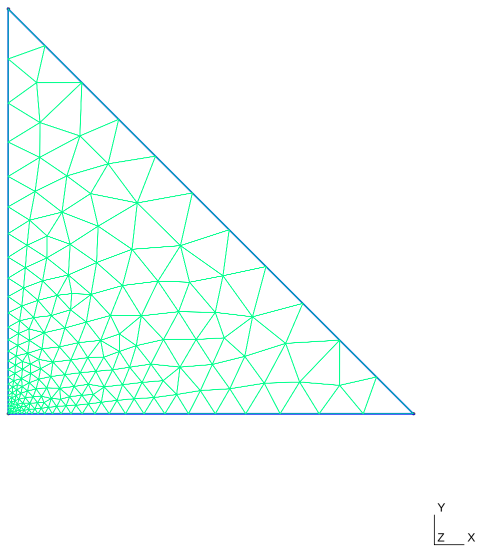
During this work we will consider a material with the following caracteristics :

$$\begin{cases} \text{Young modulus} & : & E = & 35 \;\; GPa \\ \text{Poisson ratio} & : & \nu = & 0.2 \\ \text{Material density} & : & \rho_c = & 2300 \;\; kg.m^{-3} \end{cases}$$

2 Study with T3 elements

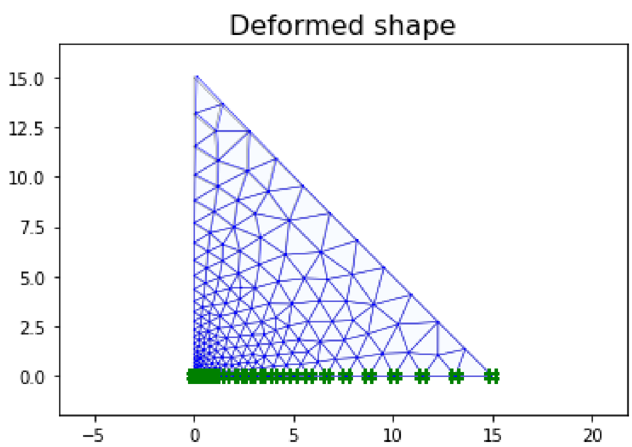
2.1 Pre-processing

We build a geometric mesh to represent the dam such that we have more nods where the stresses are expected to be maximum (i.e. in A). This position will be later verified during post-process.

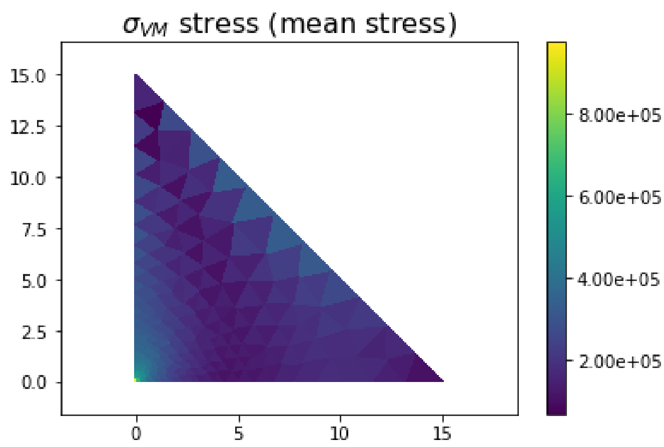


2.2 Result code

Here below, we display the resulting VonMises stress field as well as the displacement field. The purpose will be to do a first raw check of the behavior of the code :



(a) displacement



(b) Von Mises stress field

One can see that as expected the stress is the highest in A, even though it's high as well along the hypotenuse.

Moreover, as one could have expected, and as we fix bottom line, the max displacement is occurring in C. Nevertheless, as we consider a dam, the displacement isn't to high considering the pressure applied to left hand side line which of the following form :

$$\underline{T}^d = p\underline{e_1} = \frac{1}{2}\rho_\omega g h$$

Note that we normalized the pressure with respect to P_{atm} such that as P_{atm} is applied everywhere on the mesh's edges, we can cancel it in every terms. Moreover a big assumption done here is that the pressure doesn't depend on y, which is physically false.

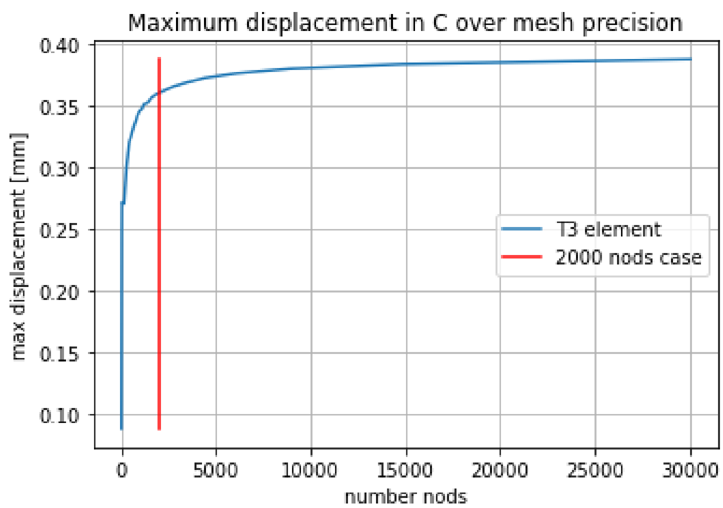
We are now to measure the stress in A and the displacement in C to confirm our visual interpretation and measure influence of the mesh on the precision of the result.

2.3 Post-processing influence of mesh size

2.3.1 Max displacement in C

The displacement as seen during post-process is maximum in C. Besides, we know from $GeoT3.msh$ that C point is defined by nod 3. We are then to check for the displacement in C looking directly to the displacement of nod 3 and by looking at where max displacement occurs.

Once we get the value at point C, we aim to study how the displacement behaves depending on the number of nods. To see the influence of the mesh size, we change the mesh density and we look at the evolution of the displacement of the top of the structure in function of the number of nods in the mesh.



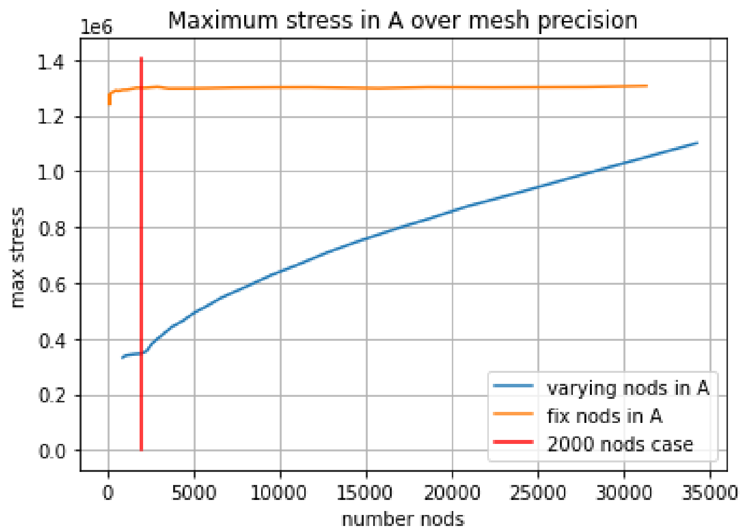
As we see on the graphic, we obtain a stable value of the displacement at the top when the number of nods in the mesh is near 5000. This information provides basics but yet very important details on the mesh's precision.

An important point is as well to see that the displacement is quiet low, which is expected given the force applied to it and its resistance to bending by construction.

Let's run the same type of analysis on A before going any further on the data analysis.

2.3.2 Von Mises stress at point A

Here below is the same type of analysis on point A concerning maximum stress.



This time, one can see that there are two possible behavior for the stress field. We indeed have a different meshing in point A and in point B-C (i.e. same in B and C but different from nods in A).

Therefore, as the maximum stress is being calculated in A, the result depends on either we increase the nods in A or in B-C (with respect to each other).
Thus :

- we either increase the number of nods close to A (varying nods in A case) and so we fix the one in B and C ;
- or we fix the one in A and make vary the nods in B-C (fix nods in A case).

As one could have expected, the finer the mesh the more precise the solution will get. Thus, the finer the mesh close to point A (orange line), the more precise will be the stress calculated in that point. This result shows that more than the number of nods in the mesh, one must pay attention to where to place them. It may end up in an economy of calculation time as a few hundreds of nods would then provide a better result than with thousands of nods.

2.4 Data analysis

To conclude, the precision over the stress field looks much better than the one over displacement field (at least in case of fix A). Then only hundreds of nods are necessary to get a good result over the stress field while thousands are needed in the case of displacement. However the maximum stress in A has been approached here by considering the max stress in the element containing A and directly in contact with the pressure to the left. Therefore, we have here the precision of stress on one element.

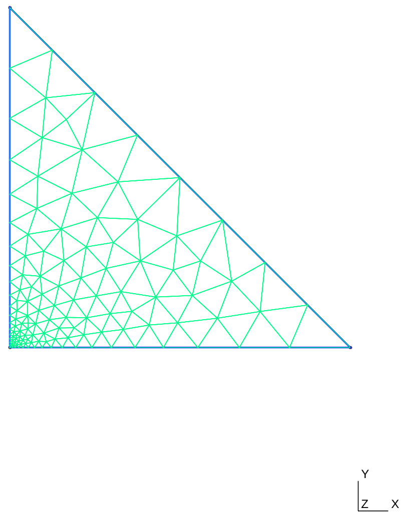
Besides, what one must take into account is that we have a much bigger number of nods in A than in C. Therefore, for instance, when we have a thousand nods, it's more likely to have half of it to the left down part, where A is, than in C, where max displacement is. Thus, the above graphs as well show that to increase the number of nods in a point makes the result at it more precise, as one could have expected.

Now we have more hindsight on the precision from the numerical calculations, we will in next part compare them to T6 elements.

3 Study with T6 elements

3.1 Pre-processing

As we change the number of point defining our initial triangle, we are to redefine the geometrical mesh, here below is the result :



What changes from previous mesh is that we have here 6 points defining each element. In fact, we will only consider homothety and dilatations for the form, namely we have linear variations, not quadratic ones.

3.2 Code

Now that the reference shape as been redefined, we are to modify some elements of the code related to the initial shape such as $B_e[\underline{a}]$, K_e, etc. Here below are some of these elements :

```
def compute_Be_matrix(self,xi,eta):
    N, DN = self.shape_functions(xi,eta)
    Be = np.zeros((3,self.el_dof))
    GN = np.dot(self.invJac,DN)
    Be[0,0::2] = GN[0,:]
    Be[1,1::2] = GN[1,:]
    Be[2,0::2] = GN[1,:]
    Be[2,1::2] = GN[0,:]
    return Be
```

Listing 1: Be matrix

```python
def elementary_stiffness(self, mat, sect=None):

    """ Elementary stiffness matrix :math:`[K_e]` shape=(12,12)"""

    Ke = np.zeros((self.el_dof,self.el_dof)) #self.el_dof=12 here
    for i in range(self.ngauss):
        Ke += self.wg[i] * self.detJ*np.dot(self.Be[i].T,np.dot(mat.C,self.Be[i]))

    return Ke
```

Listing 2: Ke matrix

```python
def stresses(self, Ue, mat, E, nu, sect=None):
    C     = mat.C_matrix()
    sig_g = np.zeros((4*self.ngauss,))

    for i in range(self.ngauss):
        sig_g[[4*i, 4*i+1, 4*i+3]] = np.dot(C, np.dot(self.Be[i], Ue))
        sig_g[4*i+2] = lamb*np.trace(self.Be[i])

    return sig_g
```

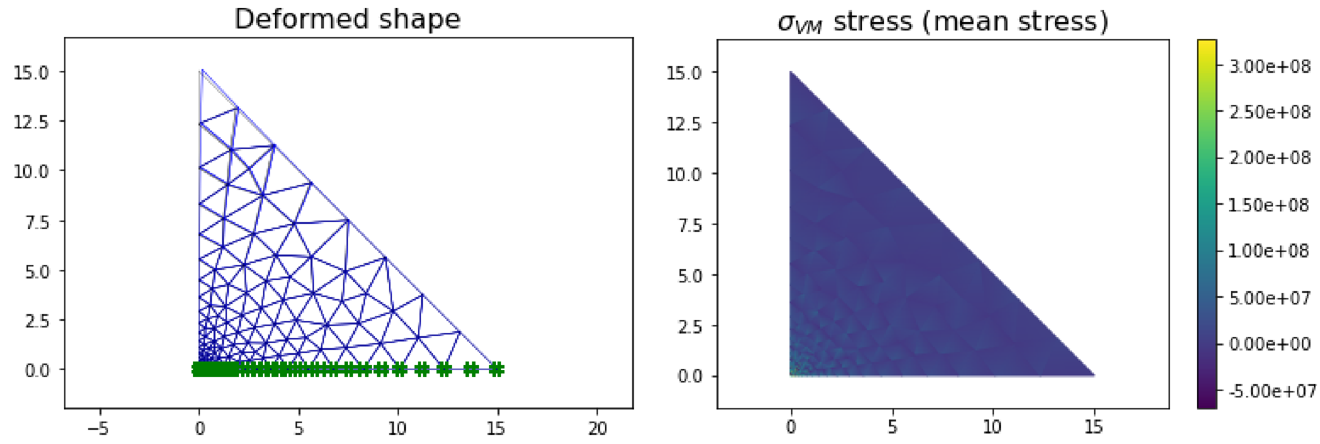
Listing 3: Stress tensor σ

These two last codes more particularly present a fundamental feature of the T6 case. Indeed, more than adding points to define the elements, we use the Gauss interpolation using 3 Gauss points. Thereby, what is $wg[i]$ is in fact the "weight" of a Gauss point (i.e. how much the values calculated in this point are to be taken into account).

We then combine 3 Gauss points for each element and deduce matrices, tensors, etc. In the following parts we will see what this new approach brings out.

3.3 Pressure action

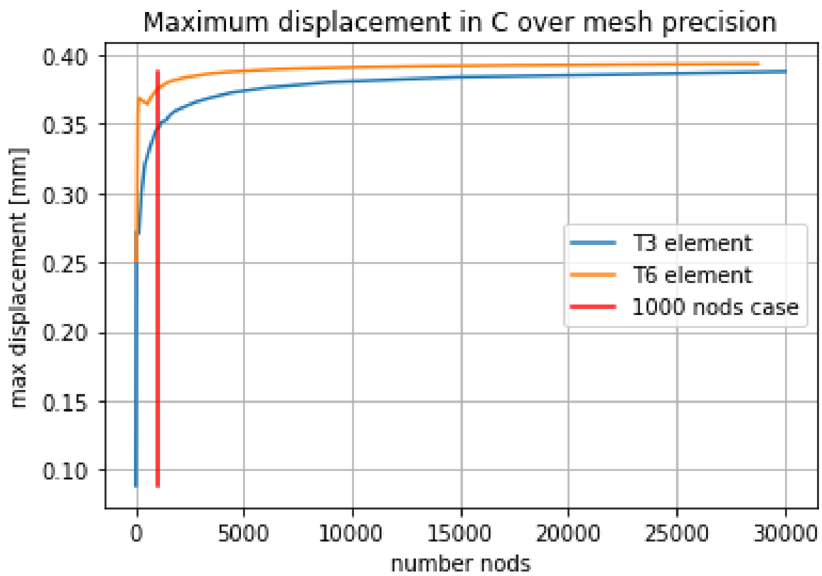
We perform the same analysis than before, except that we now work on T6 elements.



(a) displacement **(b)** Von Mises stress field

We will more specifically look at C point where the deformation is still the maximum. The idea will be to quantify how different T3 and T6 results are with respect to the mesh's precision.

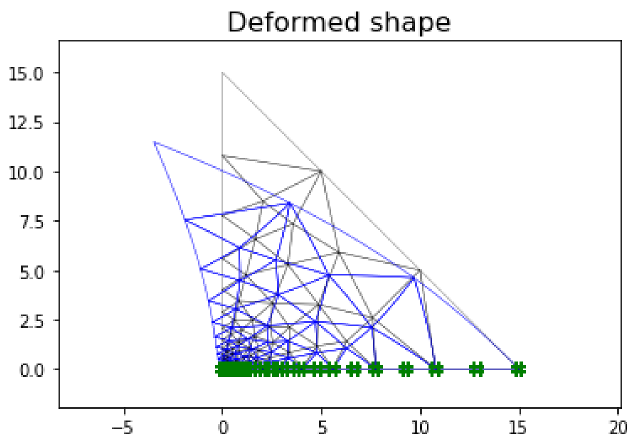
Looking to the displacement field from T6 element and by comparison to previous results, we obtain the following graph :



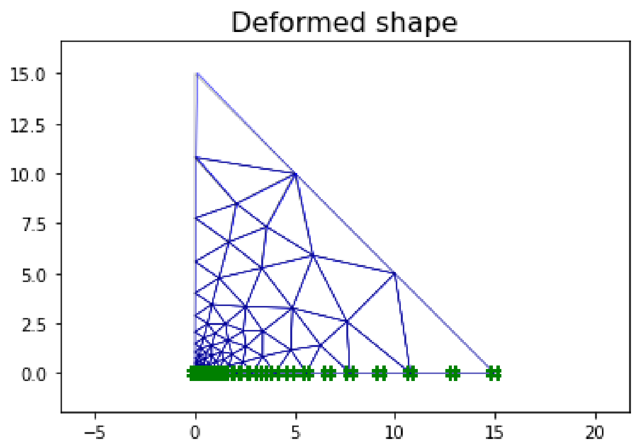
One can see that the graph provides a major result bounded to the reference form : the T6 element gets much quicker to a value close to its final one. Moreover, as one can note, the value to the right (i.e. nods $\to \infty$) looks as expected to be the same for both elements, which tells us that they both behave the same, except that T6 element asks less calculation time for a similar or even better precision on the result.

Now we have seen what the T6 reference element is capable of with respect to T3 element, we will use its capacities in order to model gravity over the mesh before concluding on the TP.

4 Gravity action



(a) Gravity - F$\approx 10^8$ N



(b) Gravity - F$\approx 10^2$ N

These graphs have for purpose to highlight the influence and the presence of the gravity force. We then see that this force, together with our boundary conditions, implies that the dam breaks leftwards for high values. This result can be deduced as well looking at stresses where we see that the geometry implies a break towards bottom left. In fact, the stress all over the dam are leftward-wise for σ_{xx} components and as well the stress at the hypotenuse is of equivalent order such that this movement is to be expected.

Now that we've talked physically about the form of the displaced field, for what is up to the comparison between the two graphs, one sees that with realistic values ($Weight \approx 10^2$ N), we have a dam holding, which is what we expect from it with considered medium characteristics (e.g. high Young modulus). Besides, one can see that high values are needed to bend the dam, confirming our model respecting true dam's resistance.

5 Energy estimate

5.1 Post-process

During post process, a feature that one can calculate is the potential energy. This value provides indication on how much is the structure loaded. As the forces we apply to it tend to compress the dam (given the values taken by stresses at edges), we then expect a negative potential energy. Overall, as we don't consider any speed, we are even to estimate this potential energy as the one gathered by the dam from applied stresses.

However, here below is a part of the code used to derive Ep's value for a given meshing :

```python
def treat_potential_energy(self, F, K, U):
    """ Potential energy calculation """

    Ep = .5*np.dot(U,K.dot(U)) - np.dot(F, U)

    print("\nEp - potential energy")
    print(f"{Ep:.2f} J")

    return Ep
```

Listing 4: Ep value

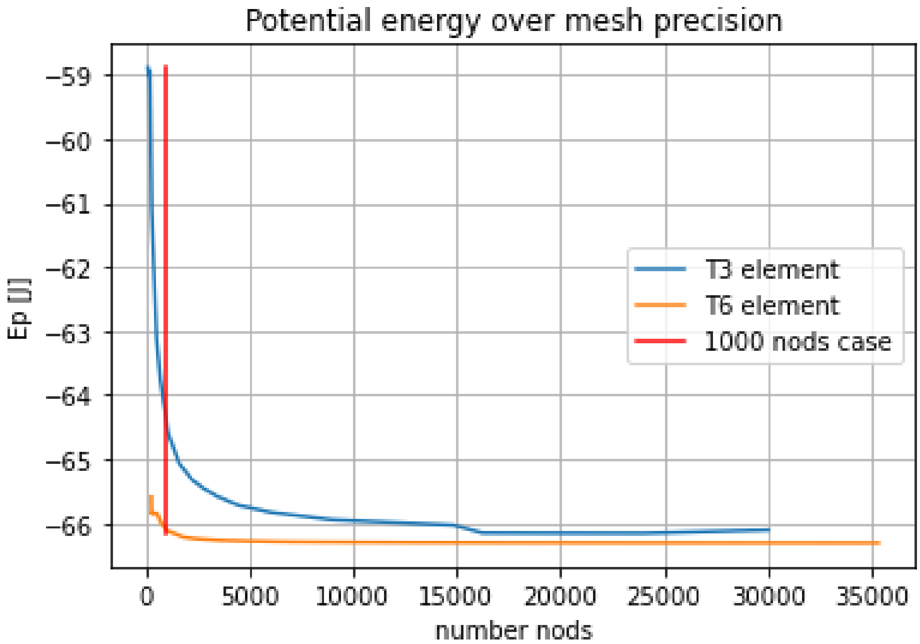
One can see that this formula rewrites the one provided in the subject by recalling that :

$$
\begin{aligned}
E_{ph} = \quad & \frac{1}{2}\int_{\Omega^h} \underline{\underline{\epsilon}}(\underline{u_h}) : A : \underline{\underline{\epsilon}}((\underline{u}_h)dV - \int_{\Omega^h} \underline{f_v}.\underline{u_h}dV - \int_{\partial\Omega_F} \underline{T^d}.\underline{u_h}dS \\
K = \quad & \int_{E_e} [B_e(\underline{a})]^t [A][B_e(\underline{a})]dV_{E_e} \\
[\epsilon_e] = \quad & \{B_e\}[U_e]
\end{aligned}
$$

By identification from local to global notation, one see that we can rewrite epsilon terms from B_e and U_e terms and then rewrite this as a function of K. In the end, one end up with the expression detailed here above [4].

5.2 Convergence study

As we did for displacement precision for T3 and T6 elements, we will run an analysis on the potential energy's precision. From previous result and by intuition, we are expecting the result to show a better precision for T6 element for a given mesh density. Here is the graph confirming this point :



6 Conclusion

We can conclude on the fact that T6 element, even if not used at a hundred percent of its capacities (i.e. we block curving of element), presents more precision than T3 element. This implies that with only a few nods we can reach the same precision.

No exact values are provided in this conclusion but one can visually estimate to 1000 the number of nods needed to reach an equivalent precision in T6 than in T3 element with 30000 nods.

This observation implies that we can reduce drastically the time of calculation while keeping a good precision.

With more hindsight, we could develop finite element methods using higher order element while ensuring a soft simplicity.