Design and analysis procedure for centrifuge devices with a Product Lifecycle Management (PLM) system

W.F. Morales, J. Laue, A. Zweidler, S.M. Springman
Institute for Geotechnical Engineering, ETH Zurich

Abstract: Structural analysis of new devices to be used in centrifuge facilities can be a challenging task. Initial calculations, based on simple basic static principles, are usually used. However, as devices placed in a geotechnical centrifuge are often complex structures, advanced analysis methods to assess their performance under enhanced acceleration fields are desirable. This paper presents the analysis of the performance of a new strongbox designed for a centrifuge facility at ETH Zurich by using a commercial Product Lifecycle Management (PLM) software.

Keywords: PLM, Finite element, enhanced acceleration, centrifuge facility, device design

1 INTRODUCTION

Every new structural element to be used in a centrifuge must be designed with a sufficient factor of safety, and should be verified with an initial proof-test at 125% of the design working stresses applied to the device inflight (Schofield, 1980). This is essential for the safety of operators and researchers, as well as for the physical stability of the rotating devices in the centrifuge. Even small loose pieces may cause considerable damage due to the high momentum reached by elements during spinning in a centrifuge.

The devices used for modelling of increasingly complex prototype scenarios are becoming more and more sophisticated. Analysis of the performance of such devices is required when they are subjected to enhanced acceleration fields. Initial analysis, based on simple statics principles and simplified structural elements, should be performed once the preliminary concepts have been developed. The results of the initial analyses might not be accurate enough to decide whether or not the device is safe to be used in a centrifuge facility due to the simplification of the static system. Finite element analyses of a complete system can then provide a more reliable option to verify the performance of these new devices and to offer opportunities for revision of relevant design details. New features or improvements can be evaluated numerically before the structures are built. However, designing, drafting and analysing the structures in different computer software packages might be problematic so that it is of special interest to be able to perform all of these actions with a single software program.

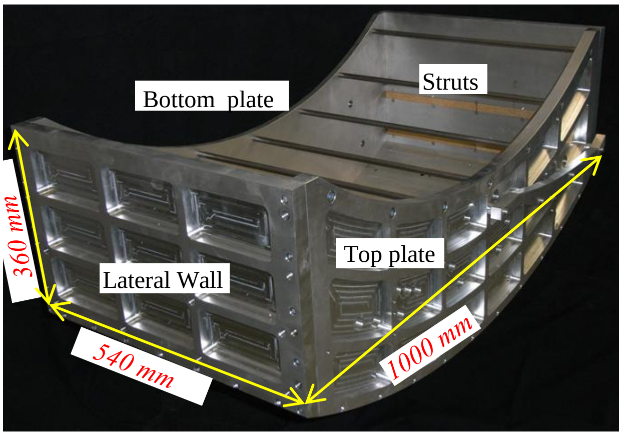
Product lifecycle management (PLM) is the activity of managing, in the most effective way, a product all the way across its lifecycles; from the very first idea, design, construction, use and maintenance, until it is retired or disposed (Stark, 2011). PLM can become an important tool when there are several steps and stakeholders between the conceptualisation and construction of the devices to be used for centrifuge modelling. This paper presents the analysis of the performance of a set of newly designed devices, including a strongbox, for a centrifuge facility at ETH Zurich, using commercial PLM software CATIA (Computer Aided Three-dimensional Interactive Application) [1].

2 NEW STRONGBOX AT ETH ZURICH

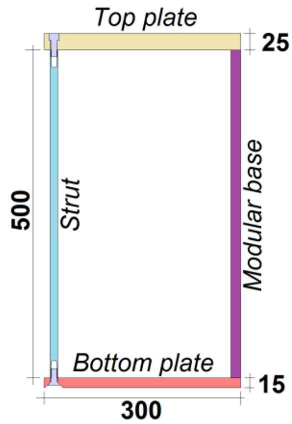
A new strongbox was designed and built at ETH Zurich for centrifuge modelling of river levees. The form of the box is an annular sector of dimensions 1000x500x300 mm, as shown in Figure 1.

[1] http://www.3ds.com/products/catia/welcome/

The length (1000 mm) is determined by the top and bottom plates. The width of the box (500 mm) is given by the design requirements and therefore controls the length the connection struts (the box is designed to stand without the need of both lateral walls). The height (300 mm) is chosen to fit the depth of the channel of the drum centrifuge. A full description of its features is given by Morales et al. (2012).



a) Isometric view



b) Cross section (dimensions in mm)

Figure 1: New semi-circular strongbox at ETH Zurich (after Morales et al., 2012).

All pieces of the structure, except for the connection struts, are made of anticorodal-110, an aluminium alloy of Swiss origin. The struts are made of standard steel St37-2. Bolts are all class 8.8. The material properties are listed in Table 1.

Table 1: Material properties used in the FEM analyses (adapted from Morales et al., 2012).

	Anticorodal-110	**Steel (St37-2)**	**Steel for bolts**
Unit weight (kN/m^3)	27	78.5	78.5
Yield strength (MPa)	240	235	640
Ultimate strength (MPa)	295	360	800
Young's Modulus (GPa)	69	210	210
Poisson's ratio (-)	0.325	0.28	0.28

The design phase included an assessment of the performance of the structure under an increased gravity condition. The analysis has been conducted with both hand calculation and numerical modelling. The latter was carried out via Finite Element Method (FEM) within the *Generative Assembly Structural Analysis* (GAS) workbench of CATIA.

3 ANALYSIS REQUIREMENTS

Schofield (1980) presents a code of practice for safe operation of the Cambridge Geotechnical Centrifuge, which also includes the factors of safety and conditions to be taken into account when designing new devices. These safety regulations are followed when operating the geotechnical centrifuge at ETH Zurich. Following manufacture and preliminary testing at 1g, any new device must be proof tested at 1.25 times the design g-level, but not lower than the design level when incorporating extraordinary loadings, before allowing it to be used in further studies. Table 2 summarises the factors of safety suggested by Schofield (1980). The factor of safety can be calculated as the failure load divided by the allowable load (Beer et al., 2002).

Table 2: Design factors of safety for new devices (adapted from Schofield, 1980).

	Aluminium	**Steel**	**Steel bolts**	**Perspex**
Factor of safety [-]	2.5	2.5	3.25	2.5

4 ANALYSIS BASED ON BASIC STATICS

As an initial assessment of the behaviour of the structure, a hand calculation was performed. In this calculation each component was analysed as a completely isolated structure and using basic statics. The bottom plate and the modular base were assumed to be completely fixed to the channel of the drum centrifuge. Hence, they were not analysed.

The box is assumed to be completely filled with a heavy fluid (γ=20 kN/m^3), which at 100-g represents a maximum hydrostatic pressure of 600 kPa, not recognising the change in g-level with radius. In the following, a description of the analysis is presented. The results are shown in Table 4 (compared with the numerical analysis results).

4.1 Top plate

The top plate was analysed as a 3-spans beam. The ribs of the piece were assumed as roller supports for the plate. The width is 165 mm and the thickness is 12.5 mm. The width corresponds to the distance between struts and the thickness is the smallest of the plate (Figure 2). The vertical reaction at *R1* is assumed to be transmitted as a pull-out force to the circular base. The reaction at *R4* is transferred as a pull-out force to the strut.

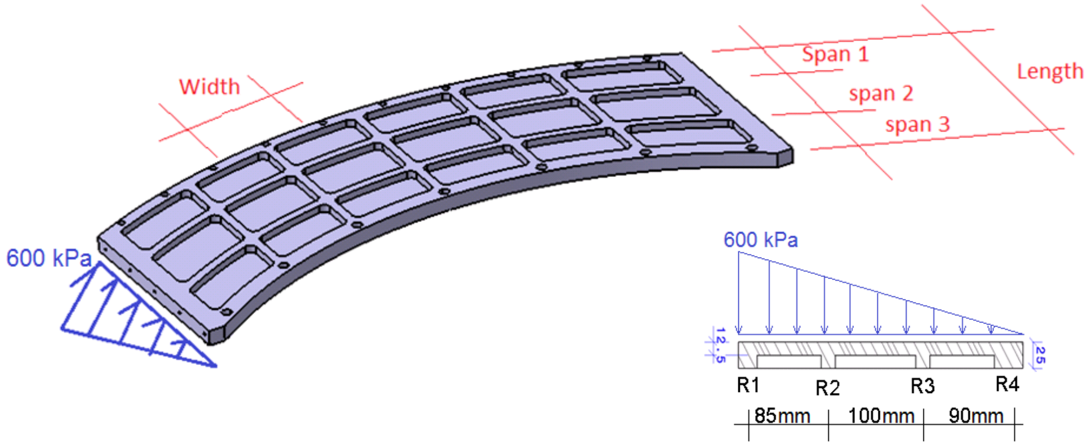
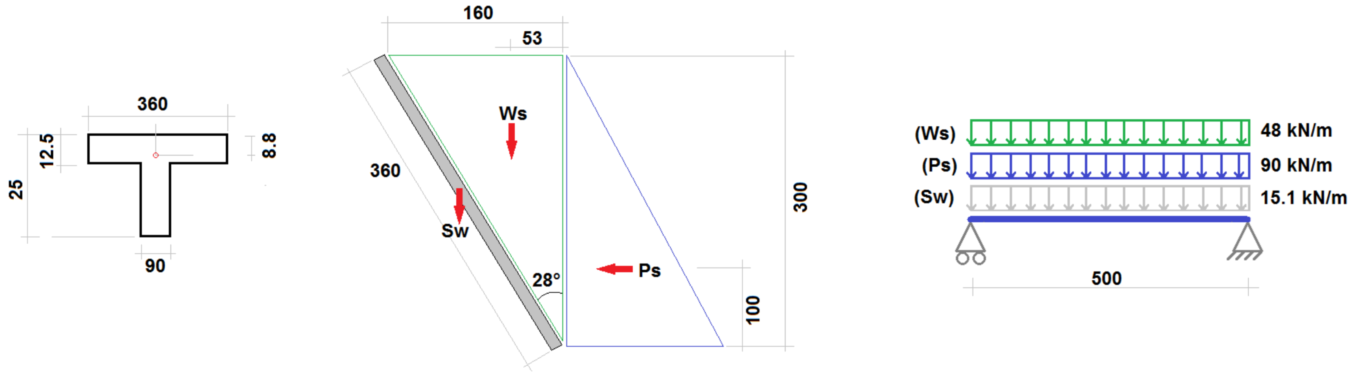


Figure 2: Sketch of analysis of the top plate.

4.2 Lateral Wall

The lateral wall was analysed as a single span beam of 0.50 m length and 0.36 m width. The connection with the bottom plate is neglected, therefore, it is assumed to be supported in two points only. The analysis takes into account the variable thickness of the plate by calculating an equivalent centroid as a 'T-beam' (Figure 3a).

The wall is assumed to be uniformly loaded by its self-weight (Sw), the wedge of soil (Ws) and the earth pressure of the surrounding soil (Ps) (Figure 3b). The inclination of the wall arises from the angle of 28° formed between the wall and the line joining the bottom of the wall and the centre of the drum centrifuge.



a) Equivalent 'T-beam'. b) Loads acting on the wall. c) Forces used for the analysis.

Figure 3: Sketch of analysis of the lateral walls (dimensions in mm).

4.3 Struts and screws

The struts were analysed as simple supported beams subjected to their own weight (increased 100 times). Pull-out force and shear forces acting on the screws were assumed to be equal to the sum of the support reactions of the top plate and lateral walls at the point of analysis.

5 ANALYSIS VIA FEM WITH A PLM SOFTWARE

The finite element method is one of the most widely used techniques in computational mechanics (Brenner & Carstensen, 2004). The methods are used extensively because engineers and scientists can model structures mathematically and solve very complex problems numerically (Bathe, 2007).

CATIA is a commercial PLM software. It is composed of different workbenches with specific tasks such drafting, assembly design, Computer-aided manufacturing Computer Aided Modelling (CAM) processes, among others. Its *Generative Assembly Structural Analysis* (GAS) workbench creates a FEM analysis directly from the assembly design. This is very convenient, as all the assembly constraints (surface contacts) can be used directly for the contact definitions of the FEM model.

5.1 Bolt modelling

Modelling the solid bolt is the closest simulation of the actual bolted connection (Montgomery, 2002). However, the high computational cost involved led to the development of diverse methodologies. These are defined as virtual connections, as they analyse the connection as a set of contact connections among the connected surfaces. A further discussion of these methodologies is given in Montgomery (2002) and (Kim et al., 2007).

The GAS workbench can represent bolted connections as a *bolt tightening connection property*, *virtual bolt tightening connection property* or as a *user-defined connection property*. The first is used when the whole bolt geometry is being simulated, whereas the other two create a link between pieces when the geometry of the bolt is not taken into account in the model.

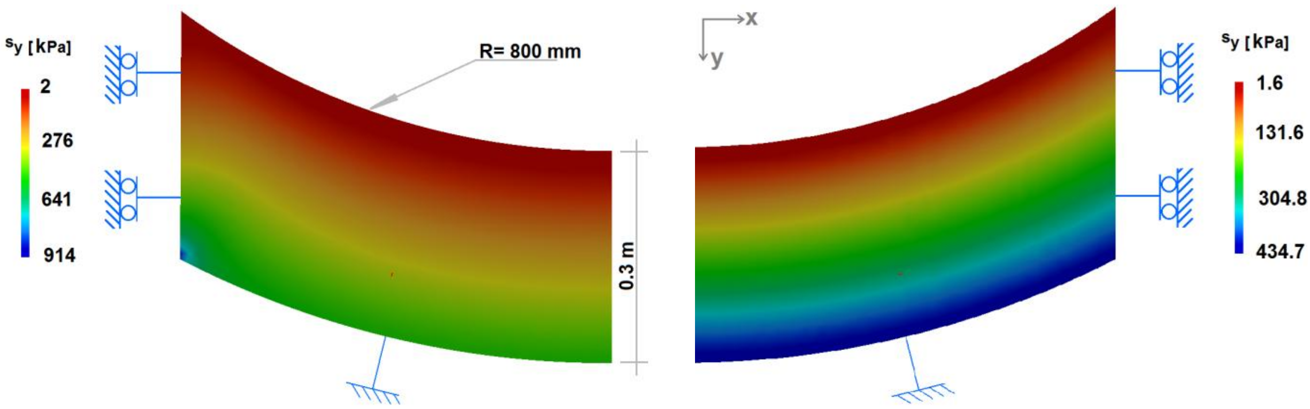
A virtual connection does not calculate the stresses on the connection. Instead, it applies an equal stress to the link and to the supporting pieces. Therefore, if internal stresses in the connections are of interest, modelling the complete bolt geometry is required (as it was done for the new strongbox). Virtual connections are useful if a simulation a bond between pieces is sought regardless of the real stresses inside the connection. Each of the representations have its own characteristics, which are discussed in the user's documentation (Dassault Systèmes, 2000).

5.2 Displacements and loads

Prescribed displacement and forces are used as boundary conditions of the boundary value problem. CATIA offers a full range of options for these conditions. The *user-defined constraint* option allows each degree of freedom of the selected element to be fixed. This is used to define fixity of displacements in the radial direction for a centrifuge device, as shown in Figure 4.

Acceleration and *rotation* are the two most important loading options for design of a centrifuge device. *Acceleration* specifies a value for the acceleration on each Cartesian direction, whereas *rotation* simulates rotation of the structure around a specific axis, which is defined by the user. Most analyses may be carried out by increasing the acceleration on one axis only, but in cases in which the height of the device is large compared to the radius of the centrifuge, it might not be representative anymore.

Figure 4 illustrates the difference in stresses obtained for a 0.3 m thick soil mass (γ=15 kN/m^3) filling the new strongbox at ETH Zurich (cf. Section 2 and Morales et al., 2012). The left-side figure is the result of defining a fixed value of 100 times gravity (y_{accel}=980m/s^2), whereas the right-side figure corresponds to defining a rotation around the centre axis of the drum centrifuge at a constant angular speed of 285r.p.m. (equivalent to y_{accel} = 980m/s^2 at R=1.10m). Although the stresses distribution, and the maximum stress in the middle part are equal (450 kPa), a stress concentration is observed at the lower corner with the *acceleration* option. This stress concentration leads the maximum stress with the *acceleration* option (914 kPa) to be 2 times larger than with the *rotation* option.



a) Applied acceleration on y axis = 980 m/s^2. b) Rotation around the centre at ω=285 r.p.m.

Figure 4: Stresses in Y direction obtained with different procedures in CATIA.

5.3 Procedure for Finite Element Analysis with CATIA

In particular, the following steps have been applied using the PLM system CATIA:

a) All components of the new device are drawn with the *part design* workbench.
b) A new ‘assembly’ is created from the *assembly design* workbench, including a *contact constraint* at each contact interface between pieces.
c) The bolts are included with a *contact constraint* between the bolt’s head and its connecting piece.
d) A *coincidence constraint* is defined between the bolt’s shaft and the hole in the connecting piece.
e) A piece is added with an edge coinciding with the rotation centre.
f) The *GAS* workbench is started to create a default FEM analysis.
g) All pieces are assigned a mesh and property.
h) The mesh and properties related to the piece added in step c) are deleted, as it will not be modelled.
i) The displacement constraints of the device are defined.
j) The *rotation* load condition is applied and all the elements that will be loaded by the increased gravity are selected. The angular speed is calculated as:

$$\omega\,[r.p.m.] = \sqrt{\frac{ng}{r}} \cdot \frac{60}{2 \cdot \pi} \qquad (1)$$

k) The analysis is executed.

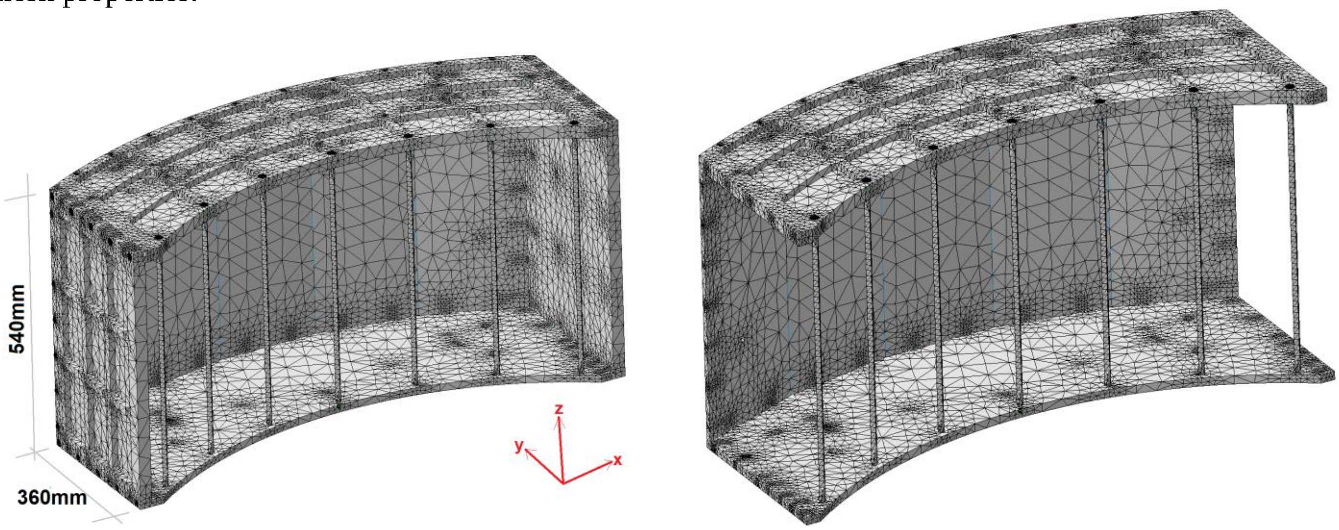
5.4 Analysis of the new strongbox

5.4.1 Analyses performed

Two analyses were performed. The first analysis evaluated the performance of the complete strongbox. The second analysis calculated the scenario in which both lateral walls are left out. All materials are assumed to be homogeneous, isotropic with linear-elastic behaviour. An increased gravity equivalent to 100-g was applied by rotating the structure around the centre axis of the drum centrifuge at a constant angular velocity of 285 rpm.

5.4.2 Mesh

Figure 5 shows the mesh developed for the analyses of the strongbox. The geometry is meshed with low-order tetrahedral elements (4 nodes per element). Contact elements are automatically created at the interfaces of each piece. These are elements representing the link between two bodies which are prevented from inter-penetrating at their common boundary, and will behave as if they were allowed to move arbitrarily relative to each other as long as they do not come into contact (Dassault Systèmes, 2000). A denser mesh is defined in the vicinity of holes for the bolts. Table 3 presents a summary of the mesh properties.



a) Analysis including the lateral walls. b) Analysis excluding the lateral walls.

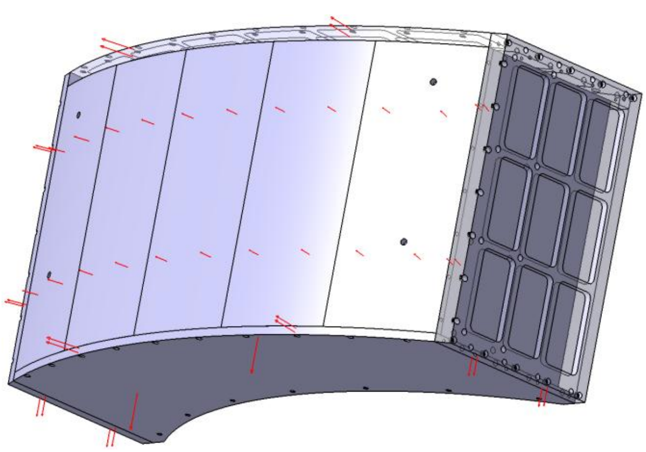
Figure 5: Meshes for FEM analyses of the new strongbox.

Table 3: Mesh properties.

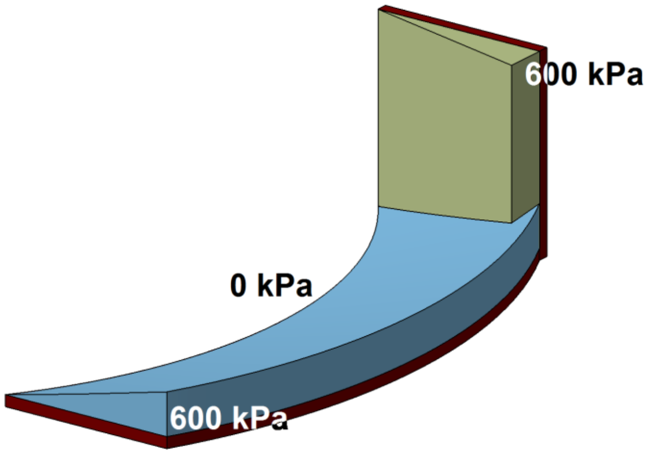
	Nodes	Tetrahedral elements	Contact elements
With lateral walls	82351	314530	12295
Without lateral walls	48130	181561	6450

5.4.3 Displacement constraints and loads

The red arrows in Figure 6a indicate the direction in which the displacements are constrained on the curved face and the base. The box is allowed to deform in all directions except in the normal direction of the contacts between the box and the channel of the drum centrifuge. The holes on the modular base were modelled as clamped elements, as they are used to fix the box to the wall of the drum.



a) Displacement restrictions.



b) Hydrostatic pressure applied to the plates and walls.

Figure 6: Boundary conditions for the analyses.

The systems have been analysed with two consecutive loading scenarios. The influence of the enhanced gravity field on the structure is studied using the constant angular velocity of rotation as described before (i.e. equivalent to 100-g). The hydrostatic pressure is added as an external force using a pressure increasing linearly from 0 kPa at a radius of 800 mm (from the drum axis) to 600 kPa at a radius of 1100 mm. This simulates the strongbox filled with a heavy fluid (γ=20 kN/m^3). The These pressure values take the influence on the fluid of an increased gravity of 100-g. Figure 6b shows the pressure applied to the plates and lateral walls.

6 RESULTS

Table 4 contains a summary of the results obtained from the analyses. The values of stresses acting under these conditions are evaluated as a von Mises stress (Equation 2) and compared to the ultimate stress of the material to ensure this is smaller and the item can be considered safe (Beer et al., 2002).

The factors of safety achieved for the pieces composing the strongbox are greater than the minimum values specified by Schofield (1980) (Table 2). The minimum factor of safety of 3.25 for bolts is not fulfilled if the lateral walls are omitted. However, this arrangement is unlikely to be used in practice and the factor of safety is still 2.2 under these extreme conditions.

$$\sigma_v = \sqrt{\frac{(\sigma_1-\sigma_2)^2+(\sigma_2-\sigma_3)^2+(\sigma_1-\sigma_3)^2}{2}} \quad (2)$$

The total displacements of the box for both scenarios with combined loading are shown in Figures 7 and 8. The largest deformations are located in the centre of the connection struts, which are not influenced much by the absence of the side walls (Table 4). The maximum deformation of the top plate is in the centre. This increases when the lateral side walls are excluded and deformation of the top plate becomes more uniform (Figure 8). Relatively small deformations (<0.1 mm) are visible close to the connections with the lateral walls and struts.

Figures 9 and 10 represent the von Mises stress distributions for all of the structural elements, with the exception of the bolts and limited to 50 MPa (for visualisation purposes). Maximum stresses, as expected, are localized around the bolted connections and any ribs. The largest stresses are concentrated in the struts (without showing the bolts). Little influence of removing the side walls is apparent in the von Mises stresses for the struts, strut bolts and the top plate.

On the one hand, overestimated stresses (negative values of the true percentage relative error) in the struts, top plate and lateral wall are obtained from the hand calculations. This implies smaller factors of safety for these pieces (conservative design). On the other hand the stresses in the bolted connections are underestimated (positive values of the true percent relative error). This leads to larger factors of safety, therefore, a non-conservative analysis. Displacement obtained from the hand calculations are completely offset the range of the values obtained from the numerical analysis.

These results (from the hand calculation) might not be accurate enough to decide whether or not the device is safe to be used in a centrifuge facility due to the nature of the assumptions made, or even an oversimplification of the statics system.

Table 4: Results from the FEM analyses and hand calculation.

Item	**Units**	**With lateral walls**	**Without Lateral walls**	**Hand calculation**	**% Difference***
Maximum von Mises stress in struts	MPa	54.7	52.1	168.0	-207
Factor of safety in rods	-	*6.6*	*6.9*	*2.14*	*68*
Maximum von Mises stress in top plate	MPa	61.5	66.1	89.9	-46
Factor of safety in top plate	-	*4.8*	*4.8*	*3.29*	*31*
Maximum von Mises stress in lateral wall	MPa	51.8	N/A	198.5	-283
Factor of safety in lateral wall	-	*5.7*	*N/A*	*1.49*	*74*
Maximum von Mises stress in strut bolts	MPa	105.4	108.7	40.1	62
Factor of safety in strut bolts	-	*7.6*	*7.4*	*19.9*	*-162*
Maximum von Mises stress in connection bolts	MPa	253.2	361.2	128.7	49
Factor of safety in bolts	-	*3.2*	*2.2*	*6.2*	*-94*
Maximum total displacement of top plate	mm	0.179	0.212	0.095	47
Maximum total displacement of lateral wall	mm	0.293	N/A	8.52	*-2808*
Maximum total displacement of struts	mm	0.362	0.366	3.5	-867

* True percent relative error of the hand calculation compared to the FEM analysis with the lateral walls.

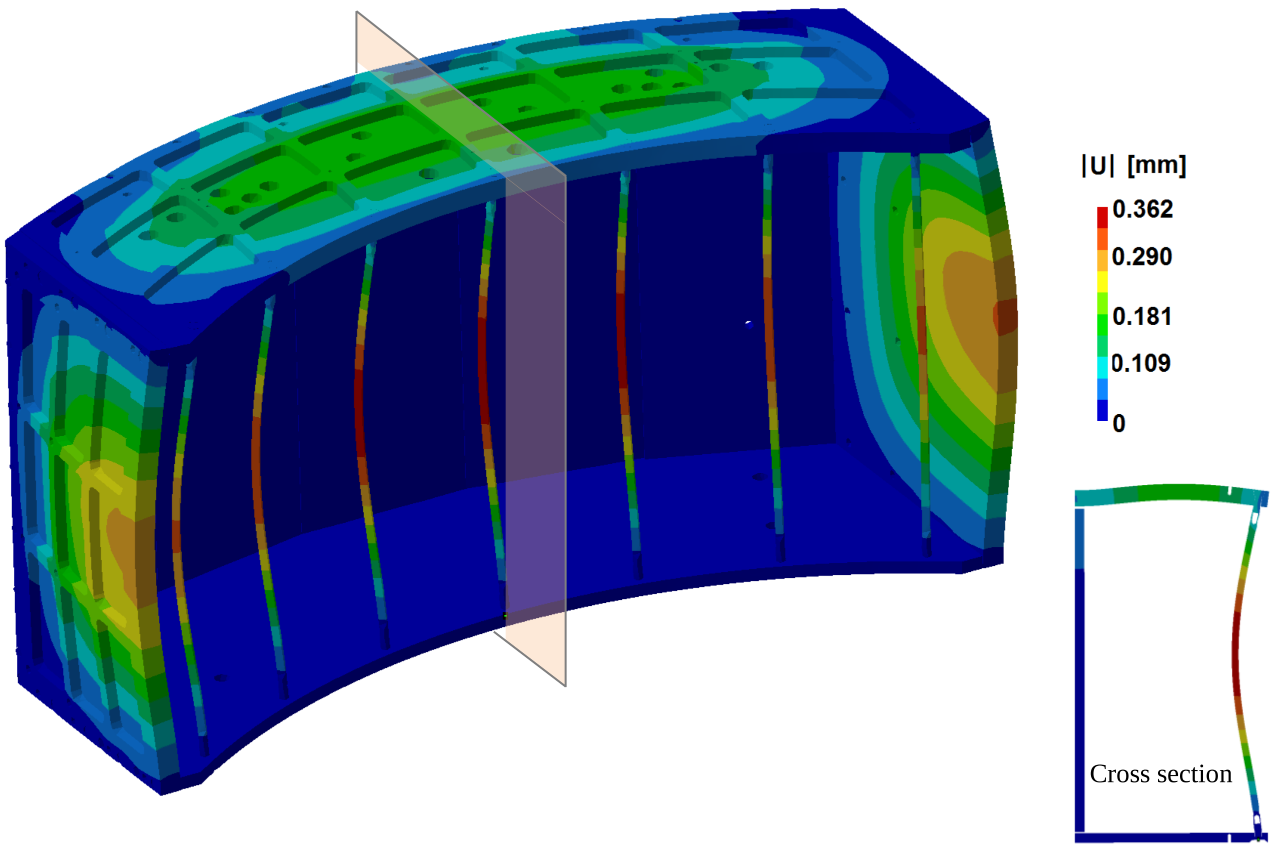


Figure 7: Total displacements for the analysis with the lateral walls (deformations are scaled up 100 times).

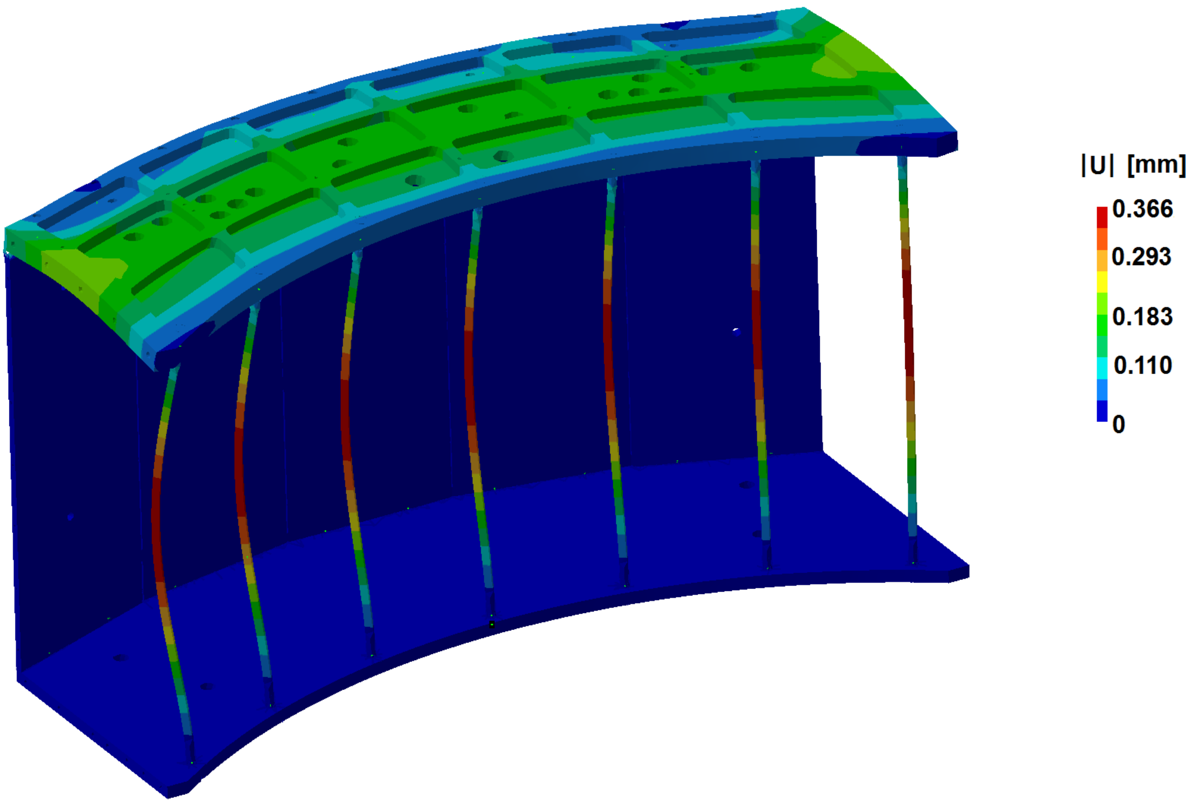


Figure 8: Total displacements for the analysis without the lateral walls (deformations are scaled up 100 times).

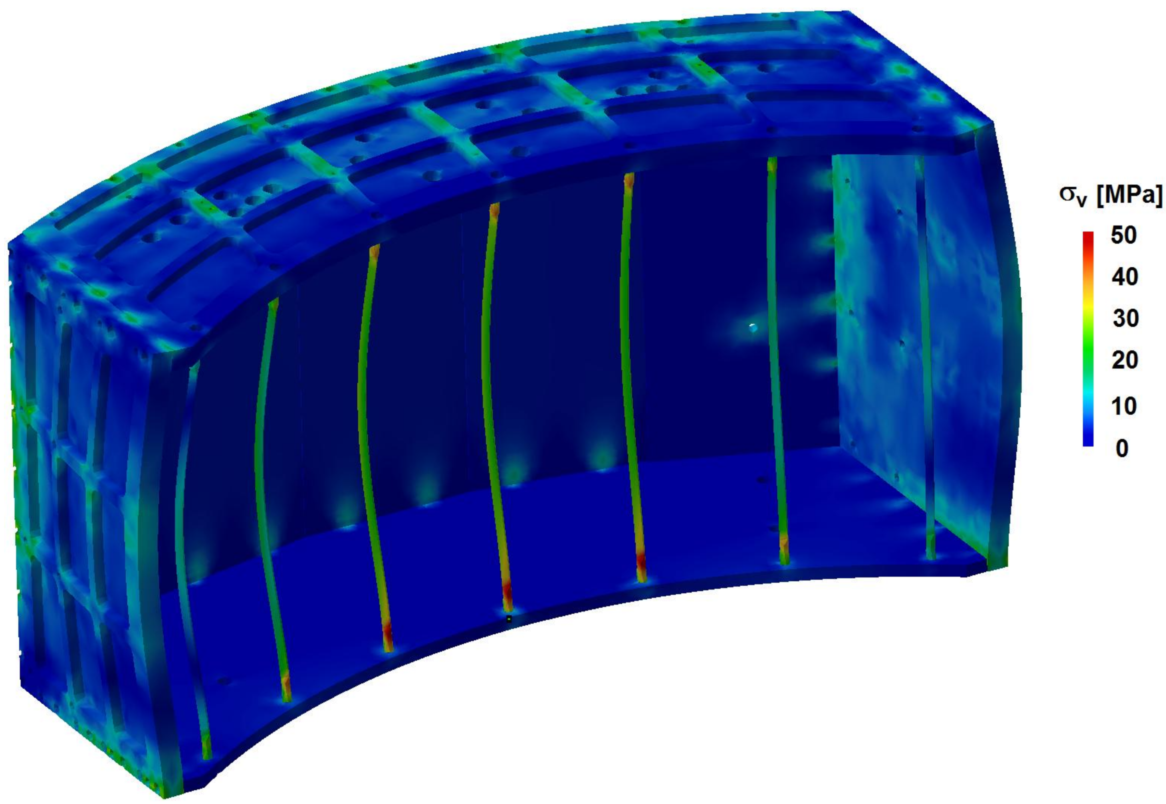


Figure 9: von Mises stresses for the analysis using the lateral walls (deformations are scaled up 100 times).

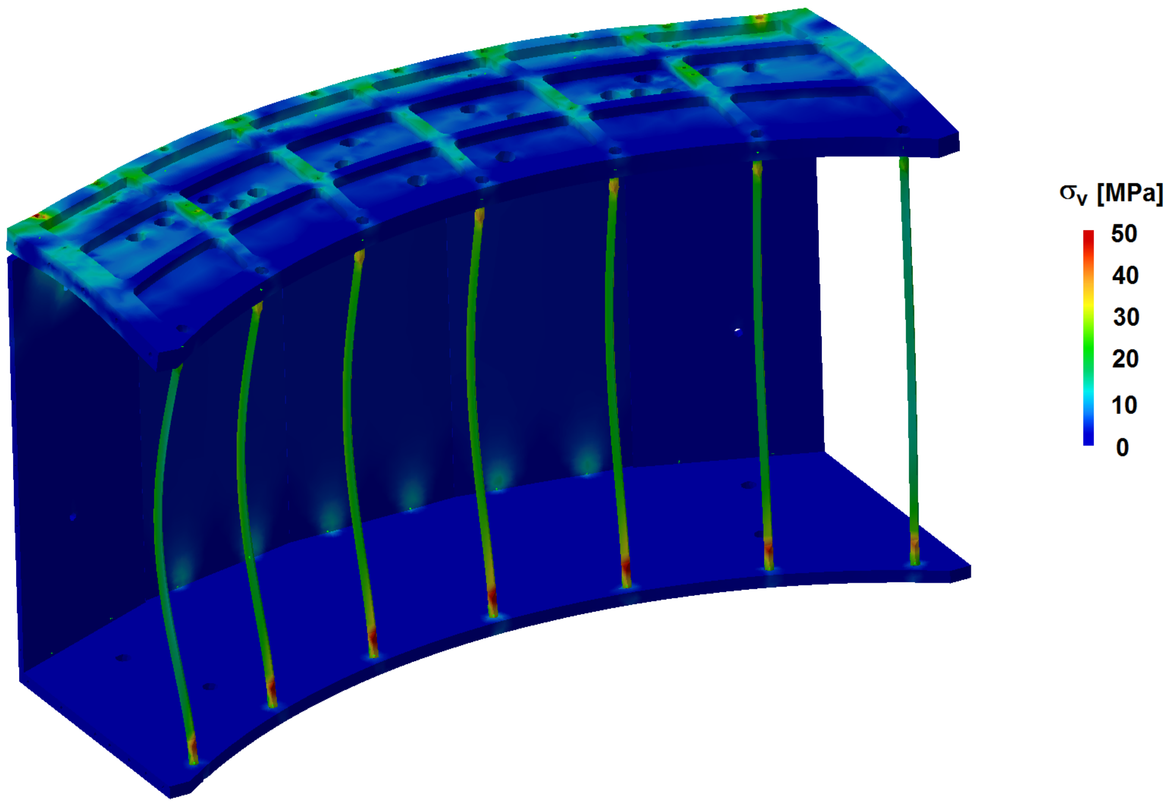


Figure 10: von Mises stresses for the analysis without the lateral walls (deformations are scaled up 100 times).

7 CLOSING REMARKS

This contribution has presented how PLM software can be used effectively during the full design and building procedure of new devices to be used within an enhanced gravity field. This process, once implemented for a new device, simplifies the analysis steps at the same time that it increases the confidence in the factors of safety calculated. The procedure should lead to the improvement of the designs through optimization phases, in order to achieve light and stable devices to be used in centrifuge facilities. Hand calculations are a useful tool for initial design and verification purposes. However, do to the assumptions involved, the values obtained with them might not be accurate enough.

REFERENCES

Bathe, K.-J. (2007). *Finite Element Method*. In B. Wah (Ed.), *Wiley Encyclopedia of Computer Science and Engineering* (pp. 1-12), John Wiley & Sons, Inc. (Pub).

Beer, F., Russell, J. and Dewolf, J. (2002). *Mechanics of Materials*. McGraw Hill (Pub).

Brenner, S. C. and Carstensen, C. (2004). *Finite Element Methods*. In E. Stein, R. de Borst & T. J. R. Hughes (Eds.), *Encyclopedia of Computational Mechanics* (Vol. 1: Fundamentals, pp. 73-118), John Wiley & Sons, Ltd (Pub).

Dassault Systèmes. (2000). CATIA V5 R19 User's documentation.

Kim, J., Yoon, J. C. and Kang, B. S. (2007). Finite element analysis and modeling of structure with bolted joints. *Applied Mathematical Modelling, 31* (*5*), 895-911.

Montgomery, J. (2002). Methods for modeling bolts in the bolted joint. *in ANSYS User's conference*, ANSYS (eds.).

Morales, W. F., Laue, J. and Springman, S. M. (2012). Controlled water-level system for breach analysis of levees within an enhanced acceleration field *in 2nd Eurofuge conference*, A. Bezuijen (eds.).

Schofield, A. N. (1980). Cambridge geotechnical centrifuge operations. *Géotechnique*, *30* (*3*), 227-268.

Stark, J. (2011). *Product Lifecycle Management 21st century paradigm for product realisation* (Vol. 1). Springer (Pub), London.