12th Scientific/Research Symposium with International Participation
„METALLIC AND NONMETALLIC MATERIALS“ B&H, 19th-20th April 2018

NUMERICAL ANALYSIS OF THE STRESS CONDITION OF THE VESSEL UNDER PRESSURE

Nermin Bišćević, Husmira Bišćević
JU Mechanical-Traffic Mixed High School
Bihać
Bosnia and Herzegovina

Keywords: KOMIPS, shells, reactor, circular stress, meridional stress, stress and deformation state, equivalent stress, working load, suffix, cold water pressure.

SUMMARY
In this paper, a pressure vessel test is described which is viewed as an axis-symmetric structure in order to perform an initial check of the adopted model defined by the plate-like finite elements. This approach gives the possibility of comparing the results obtained for the stress and deformation field with the patterns of classical elasticity theory in the case of axis-symmetrical hand-wheel shells. Thermo-mechanical calculation of vessel pressure was performed on a computer using the finite element method (MKE) with the KOMIPS software system. The experiment was carried out on a vessel (reactor) under pressure in working conditions and at the cold water pressure test.

1. INTRODUCTION
The qualitative parameters derived from the condition analysis and the strength diagnostics are efficiently used in the following activities:

- designing,
- manufacture or purchase of construction,
- reconstruction or renovation of the structure,
- extension of the remaining lifetime,
- change of working regime and
- revitalization of the construction.

In order to make a correct and precise decision about these activities, it is necessary to have a good quality analysis and diagnostic of the strength of the construction.
The basis of the diagnostic strength of the supporting structure of the pressure vessel (reactor) represents the computer modeling and calculation of supporting structure (KOMIPS) using the numerical method of finite elements through the static and dynamic calculation of its bearing elements.

KOMIPS allows [1]:

- modeling and calculation of pressure vessel construction,
- determining the actual displacement and stress image,

- ⇨ findings of the actual behavior of the pressure vessel construction and its elements,
- ⇨ reliable prognosis of reaction of vessel construction under pressure in exploitation,
- ⇨ obtaining elements for decision making (work regime, remediation, reconstruction, revitalization, optimization, confirmation of choice of variant solution),
- ⇨ determining the causes of misbehaviours or reduction of the construction,
- ⇨ estimation of exploitation life and time of reliable construction work.

Any improvement of the behavior of the structure, which can be achieved by this approach, allows for the extension of the exploitation life of the structure and the increase of its reliability. The choice of the best design solution for pressure vessels required a detailed calculation of the stress and deformation condition. Thermomechanical calculation of pressure vessel stress was performed on a computer using the finite element method (MKE) with the KOMIPS software system.

In the first stage of the calculation the pressure vessel was observed as axis-symmetric construction, in order to carry out the initial check of the adopted model defined by the plate-like finite elements. This approach gives the possibility of comparing the obtained results for the stress and deformation field with the classical elasticity theory patterns in the case of axis-symmetric handwheel shells [1].

2. NUMERICAL ANALYSIS OF STRESS STATE OF THE VESSEL UNDER PRESSURE

2.1. Model testing of vessel behaviour (reactor) under pressure

For the initial model, an axis-simetric construction was constructed of 1044 panel elements (coarse model) exposed to an internal pressure of 1 MPa. The adopted unit load allows the results to be used for any size of the given pressure using scaling and superimposition within the linear theory.
Thanks to the axial symmetry, models of the half, quarter, and eighth pressure vessels were used beside the entire model. The results obtained in this case show that they are identical with the classical patterns of thin-walled cylindrical pressure vessels, with the expressions used to calculate the normal stress on the cylindrical part to calculate [1]:

- ➢ circular: $\sigma_c = \frac{p \cdot R}{t}$ ...(1)
- ➢ meridional: $\sigma_m = \frac{p \cdot R}{2 \cdot t}$ ...(2)

where is: p - the value of inner pressure, bar,
R - middle radius of the cylinder, mm and
t - the thickness of the involucre, mm.

For the case of the vessel under pressure (reactor):

$$\sigma_c = \frac{p \cdot R}{t} = \frac{1MPa \cdot 1352mm}{96\,mm} = 14.08\ MPa$$

$$\sigma_m = \frac{p \cdot R}{2 \cdot t} = \frac{1MPa \cdot 1352mm}{2 \cdot 96\,mm} = 7,04\,MPa$$

The stress results shown relate to the equivalent stress calculated on the form:

$$\sigma_{ekv} = \sqrt{\sigma_c{}^2 + \sigma_m{}^2 - \sigma_c \cdot \sigma_m} \qquad ...(3)$$

On the cylindrical part, far enough from the upper and lower coverlid of the reactor, thefollowing results were obtained:

- f_{max} = 0.206 mm,
- σ_{ekv}^{numer} = 12.0 MPa, what is in accordance with σ_{ekv}^{teor} = 12.16 MPa.

Figures 1 to 3 show the initial rough model, deformation field and stress [1].

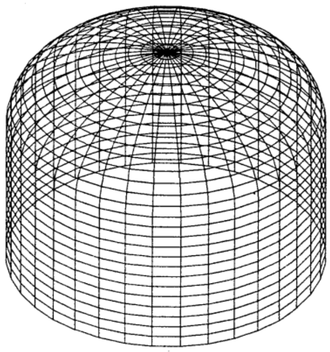
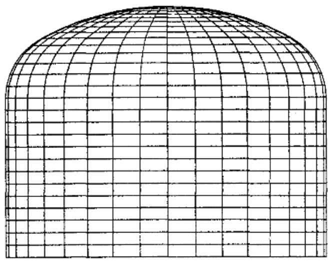
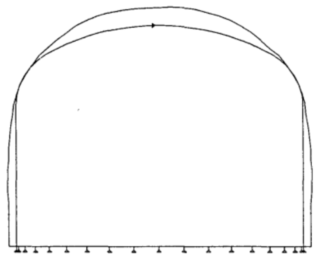


Figure 1. Model of the half of the vessel (reactor) under pressure of 1 MPa [1]

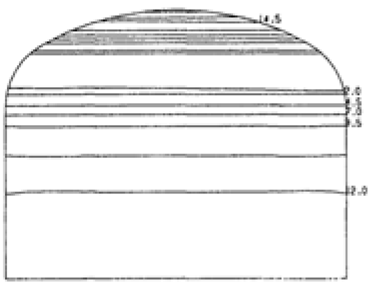


Projection of the model

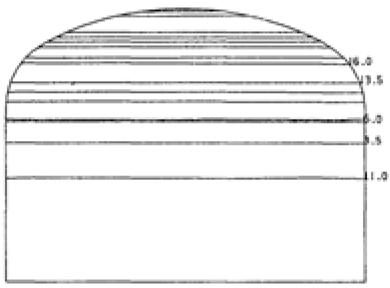


The field of the deformation - f_{max} = 0.206 mm

Figure 2. The half of the reactor – rough model A [1]



Equivalent stress 2 to 14.7 MPa with step 2.5 on the inside of the reactor

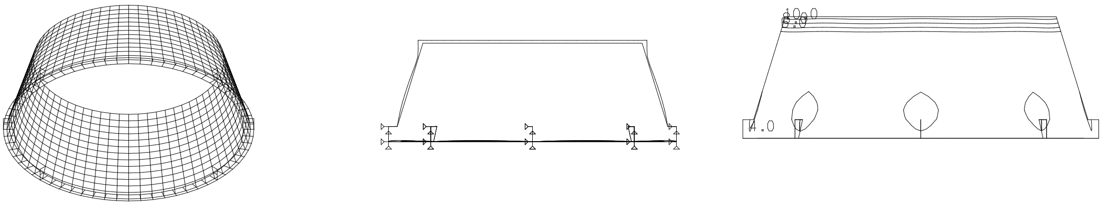


Equivalent stress 6 to 16.2 MPa with step 2.5 on the outside of the reactor

Figure 3. The half of the reactor - rough model B [1]

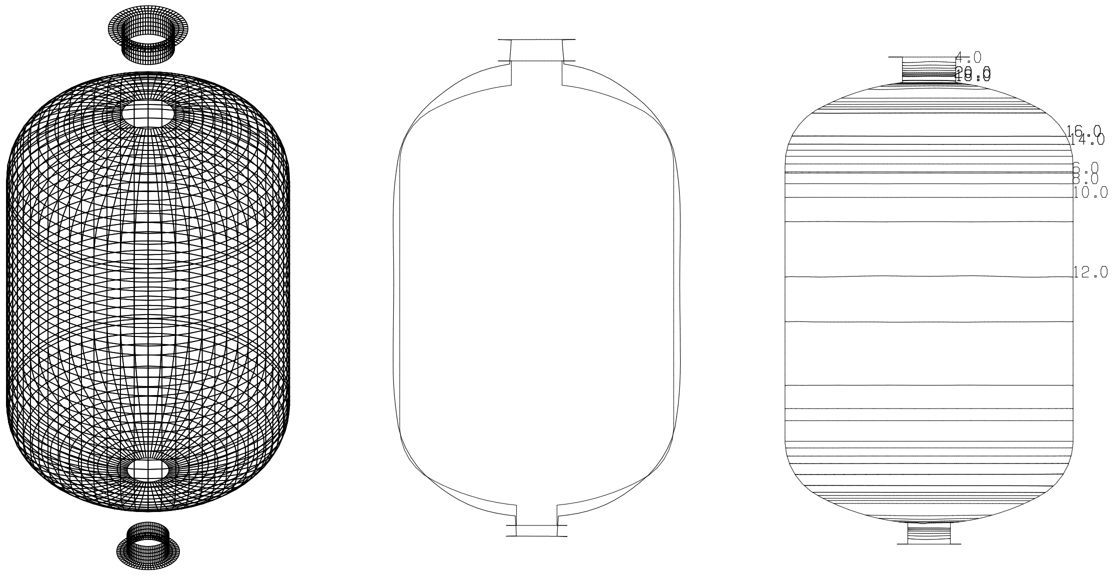
The emergence of this evenly distributed stress field enables the conclusion of the best possible construction. If the change in the geometry of the real structure (flanges, stretching, reinforcement) does not disturb the resulting field image of the stress field to a significant extent, the previous conclusion can be transferred to the real construction [1].

The next step is the introduction of a real reactor geometry, which implies the introduction of openings with flanges, the variable wall thickness around the flanges, the reinforcement and the supporting valve with the support, Fig. 4 and 5. Here, the geometric modeling of the substructure and their integration into the whole, as well as its deformed contour (pressure 1 MPa), is presented. The CIO reactor model has 5712 dots and a 5584 plate elements that are exposed to a pressure of 1 MPa. As in previous cases, the introduction of real reactor geometry into the calculation does not lead to significant changes in the stress value on the cylindrical part of the reactor



Carrier valve and suspension *Deformation of the valve* *Equivalent tension 4-10 MPa with the step 2 on the valve*

Figure 4. Modeling of understructure of the skirt of the reactor – a fine model

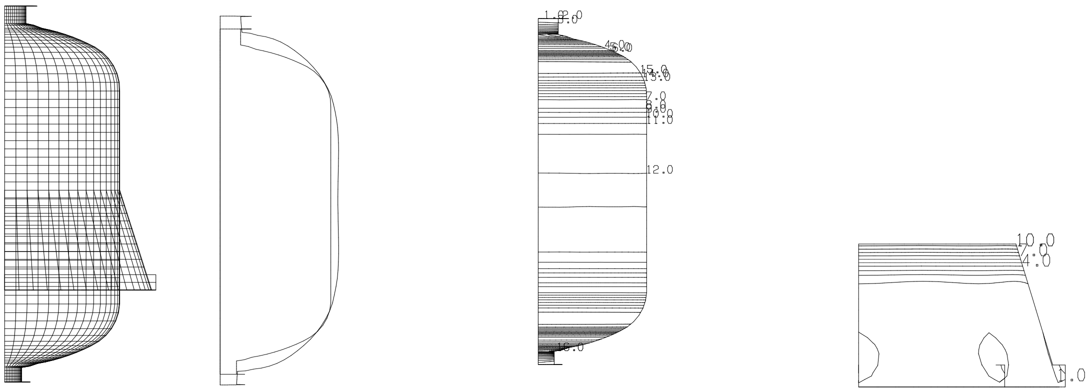


Reactor with flanges $f_{max} = 0.3$ *mm* *Equivalent stress 2-20 MPa with step 2 on the outside of the reactor*

Figure 5. Modeling of the substructure of the reactor shell - a fine model

2.2. Plate model calculation under the influence of all relevant loads
2.2.1. Stress and deformation condition under pressure of 1 MPa

The adopted panel model for the further budget has 1485 points and 1382 elements (Figure 6). Only a quarter of the model is observed . The current and deformation state is analyzed under a pressure of 1 MPa. In the case of a real model, cylinder tension retains their theoretical value and shape [1].



f_{max} = 0.21 mm

Equivalent stress of 1-16 MPa, step 1 on the outside of the reactor

Equivalent stress 1-11 MPa, step 1 on the valve

Figure 6. Plate model for a quarter of a structure [1]

The highest equivalent value of 16 MPa is located in the vicinity of the reactor flange. The symmetry of the deformation field and the stress versus the straight support is obvious.

Deformation and stress values for the work load (p = 3.44 MPa) are:

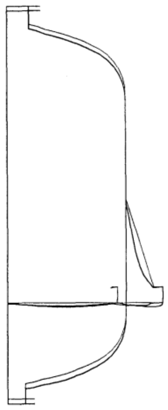
- maximum deformation $f_{max} = 0.21 \cdot 3.44 = 0.72$ mm,
- highest stress on the cylinder $\sigma_{ekv}^{max} = 13 \cdot 3.44 = 44.72$ MPa,
- the highest stress on the reactor $\sigma_{ekv}^{max} = 16 \cdot 3.44 = 55.04$ MPa and
- the highest stress on the valve $\sigma_{ekv}^{max} = 11 \cdot 3.44 = 37.80$ MPa [2] .

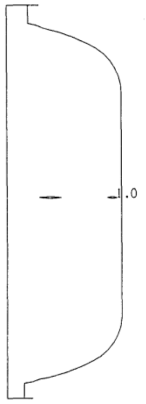
The values of deformation and stress in case of workload in the hydro-test (p = 10.94 MPa) are:

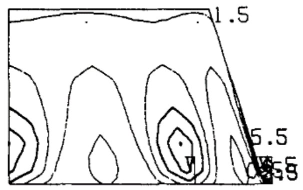
- maximum deformation $f_{max} = 0.21 \cdot 10.94 = 2.29$ mm,
- highest stress on the cylinder $\sigma_{ekv}^{max} =13 \cdot 10.94 = 142.22$ MPa,
- the highest stress on the reactor $\sigma_{ekv}^{max} = 16 \cdot 10.94 = 175.04$ MPa and
- the highest stress on the valve $\sigma_{ekv}^{max} = 11 \cdot 10.94 = 120.34$ MPa [2] .

2.2.2. Stress and deformation condition under its own weight and hydrostatic pressure

The influence of the reactor's own weight and hydrostatic fluid pressure on the hydro-test,on the overall deformation and stress is shown on the Figure 7 and 8 [1].







f_{max} = 0.09 mm

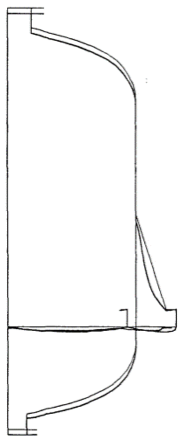
The maximum equivalent stress on the reactor - 1MPa

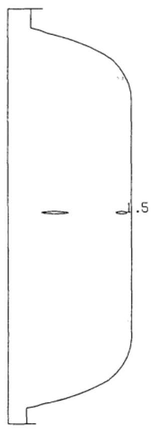
Equivalent stress 0.5 - 5.5 MPa step 1 on the valve

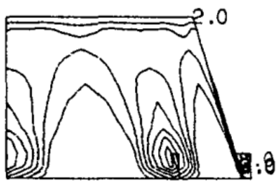
Figure 7. Demonstration of the reactor's weight on the overall picture of the deformation and the stress [1]

The obtained results of the calculation of the impact of the reactor's own weight and the hydrostatic pressure show that these effects can be neglected in relation to the influence of the reactor pressure effect.
The present analysis gives the possibility to conclude the following:

- the primary load is the pressure, while the influence of its own weight, hydrostatic pressure on the hydro-test and the temperature can be ignored except in case of significant difference in the temperature of the valve,
- there is a uniform distribution of the stress at the cylindrical part of the reactor,
- the stress distribution is uniformal in all horizontal planes (circular direction) and
- the stress distribution is approximately uniformal in all vertical planes (meridional direction).







f_{max} = 0.07 mm

The maximum equivalent stress on the reactor- 1.5MPa

Equivalent stress 2 - 8 MPa, step 2 on the valve

Figure 8. The impact of hydrostatic pressure of the reactor onto the complete figure of deformation and stress [1]

In this way, the validity of the model for the calculation is verified, which in the further course of the calculation will be the basic model exposed to the action of all defined types of loads.

2.3. Thermomechanical calculations of the stress for operating conditions and hydraulic test conditions

As in the previous examples, the validity of the model was verified and validated, the thermo-mechanical stress calculation of the reactor was performed on a computer using the finite element method with the KOMIPS software system. As an optimal solution, modeling of the reactor design was performed using the finite element of the thin plate [8].

Considered load cases are:

- *working pressure and own weight, and*
- *pressure at the hydro-test and its own weight*

The adopted operating pressure was 25 bars, and the pressure of hydra test was 109.39 bars. Also, dynamic calculations have been made to calculate the oscillations. The calculation model of the reactor element is given in Figures 9 to 11 .

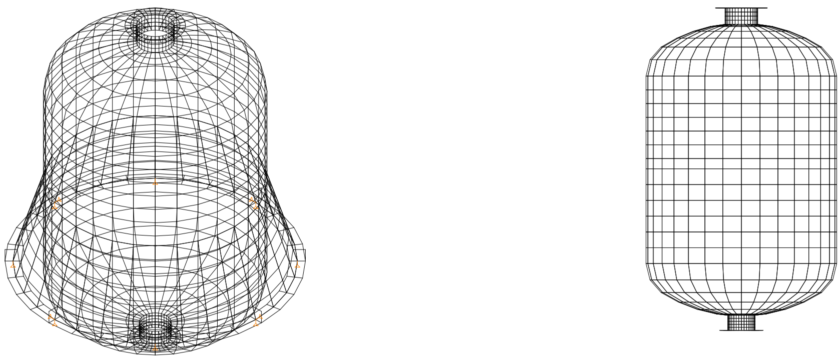
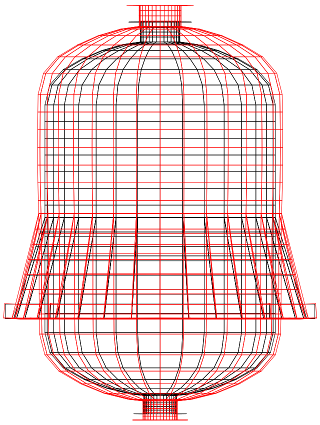


Figure 9. The view of the reactor shell calculation model [2]

Figure 10. The view of the calculated model opening on the reactor [2]

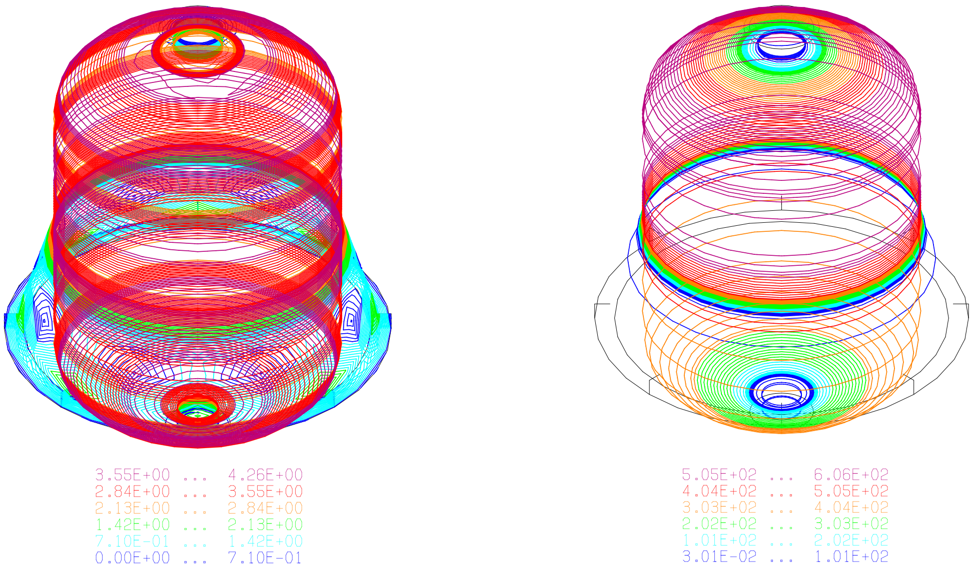
Figure 11. Calculated model of the reactor support – skirt [2]

Figures 12 and 13 show the distribution of the stress and energy deformation of the reactor under operating load (working pressure, own weight and temperature).



Maximum deformation 0.75 cm

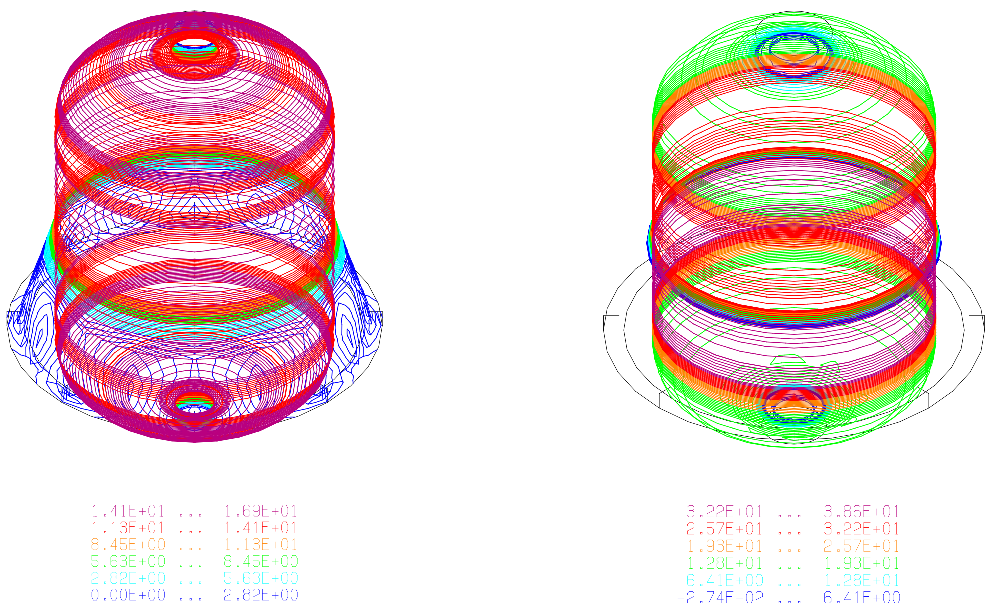
Figure 12. The view of the deformation distribution of the reactor at work load [2]



Equivalent stress, kN/cm^2 *The energy of the deformation, kNcm*

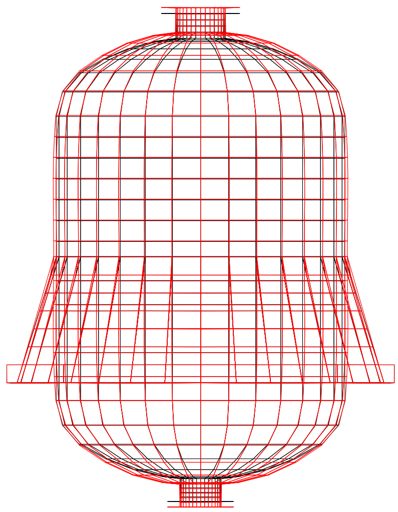
Figure 13. Display of the stress distribution and deformation energy on the reactor at work load [2]

The reactor behavior analysis was also performed under the cold water pressure test (CWP). Figure 14 shows the stress distribution and the deformation energy , and Figure 15 shows the deformation of the reactor under CWP conditions.



Equivalent stress, kN/cm²　　*The deformation energy, kNcm*

Figure 14. The stress distribution and the deformation energy of the reactor in conditions of cold water test [2]



Maximum deformation 0.25 cm

Figure 15. The reactor deformation in cold water pressure testing conditions [2]

The obtained stress values which take into account the effect of their own weight should be taken into consideration in the amount shown. Pressure stress values are obtained by multiplying the calculated stress gains (p = 1 MPa) with a value of 3.44. The total value of the stress is obtained by the addition of the above mentioned values, with the notion that the first two impacts may be neglected. Obtained stress values that take into account the impact of their own weight:

- cylinder σ_{eq}^{max} = 1.5 MPa and
- valve σ_{equiv}^{max} = 8.0 MPa.

and hydrostatic pressure in the hydrostatic test:

- cylindar σ_{ekv}^{max} = 1.0 MPa and
- valveσ_{ekv}^{max} = 5.5 MPa

should be taken in the amount shown.
Hydraulic pressure stress values are obtained by multiplying the calculated obtained stresses (p = 1 MPa) with a value of 10.94

- $\sigma_{ekv}^{max} = 16 \cdot 10.94 = 175.04$ MPa.

Under the operating load conditions (working pressure of 25 bar), the deformation of the reactor is relatively small and extremely favorable, ie. the reactor does not spread due to working pressure (the support of the reactor via the so-called "skirting" is an extremely favorable constructional solution). The stress field due to the working pressure of the reactor is also extremely convenient, evenly distributed and very homogeneous. There are no apparent stress concentrators. The maximum stress values are below the permissible value. The calculated values of the stress at the load of CWP are quite high and exceed the recommended permissible values (the ratio of the flow rate and the maximum stress obtained must be greater than or equal to 1.5) but it is still lower than the value of the flow rate obtained by examining the delivered samples of the new and exploited base material as well as components of welded compound [2].

Table 1 gives a summary section of the estimated reactor parameters in working conditions and CWP probes:

Table 1. Reactor calculation results [2]

		Reactor
Working load	Maximum deformation, f_{max}, cm	0.75
	Maximum stress,σ_{max}, MPa	42.6
	Deformation energy, E_d, kNcm	606
CWP (cold water pressure)	Maximum deformation, f_{max}, cm	0.25
	Maximum stress, σ_{max}, MPa	169
	Deformation energy, E_d, kNcm	28.6

3. CONCLUSION

The general conclusion of the numerical analysis is the necessity of examination of the chosen modelby an adequate experimental method, which should be carried out from the outside of the pressurized vessel (reactor) and which will not jeopardize the structural integrity of the reactor itself. Measurement of the deformation and stress state of the reactor in exploitation implies the application of the tenzometric method (metering strips) [2].

4. REFERENCES

[1] Maneski T.: *Princip projektovanja – program KOMPIS, Mašinski fakultet Beograd, 2004.,*

[2] Bišćević N.: *Mogućnosti primjene koncepta integriteta konstrukcije na spremnost za upotrebu posuda pod pritiskom*, Magistarski rad, Tehnički fakultet, Bihać, 2012.,

[3] ASTM E399 - 02, *Standard Test Method for Plane-Strain Fracture Toughness of Metallic Materials,* Annual Book of ASTM Standards, Vol. 03.01., 2002.,

[4] BS 7448. *Fracture mechanics toughness tests. Part 1. Method for determination of K_{Ic} critical CTOD and critical J values of metallic materials*, BSI, 2006.,

[5] ASTM E813 - 89, *Standard Test Method for JIC, A Measure of Fracture Toughness, Annual Book of ASTM Standards*, Vol. 03.01. p. 651, 1993.,

[6] ASTM E 1820 - 08, *Standard Test Method for Measurement of Fracture Toughness, Annual Book of ASTM Standards*, Vol. 03. 01., 2008.,

[7] BS 7448 - Part 2, *Fracture mechanics toughness tests - Methods for determination of K_{Ic},critical CTOD and critical J values of welds in metallic materials*, BSI, 2010.,

[8] Burzić Z., Sedmak S., Manjgo M.: Eksperimentalno određivanje parametara mehanike loma zavarenih spojeva, Integritet i vek konstrukcija, No. 2, str. 97, 2001.