

## THE PROSPECT OF USING THE FINITE ELEMENT METHOD AND ANSYS FOR MATHEMATICAL MODELING OF THERMO-MECHANICAL CALIBRATION LARGE AXISYMMETRIC STRUCTURES WITH HIGH REQUIREMENTS FOR THE GEOMETRIC ACCURACY

© Sherstiuk D.G., Kulik A.V., Gladkiy M.A., 2013

За допомогою математичного моделювання визначено напружено-деформований стан осесиметричної конструкції та технологічної оправки під час термомеханічного калібрування. Термомеханічне калібрування визначається багатьма взаємозалежними факторами, що впливають на остаточний результат, і його відповідність строгим вимогам точності. Методом кінцевих елементів за допомогою числового моделювання у програмному середовищі ANSYS встановлена відповідність розрахункових параметрів відносно реального процесу термомеханічного калібрування.

The article carries out using mathematical modeling of the definition of the shells stress-strain state and technological mandrel in the process of thermo-mechanical calibration. Thermo-mechanical calibration determined by many interrelated factors that affect the final result, and its compliance with the stringent requirements of accuracy. The finite element method for using the numerical simulation in ANSYS corresponded between the calculated parameters with respect to the real thermo-calibration process.

**Problem.** Determination possibilities, advantages and disadvantages of mathematical modeling using the finite element method for thermo-mechanical calibration process of large axisymmetrical structures (shells).

**Analysis of recent research and publications.** Thermo-mechanical calibration (TMC) of welded large axisymmetrical structures as part of cylindrical or conical shells produced by the hard external devices as a final operation to make the product of required geometric accuracy and reduces stress. The technology is based on the difference thermal expansion coefficients of materials used for titanium billets in the form of rings and shells with aluminum alloys. This method of correction used in cases where other methods cannot achieve the required effect, because of significant dimensions and design characteristics of the shells.

Thermo-mechanical calibration is determined by many interrelated factors that affect the final result – matching stringent requirements of precision product – 1.5–2 mm to 4000 mm diameter.

Solving this problem requires the consideration of all possible external and internal factors, such as geometry design shell and furnace, irregularity heating zones with using convection heating, the rate of temperature and exposure time in the oven.

The maximum number of factors must be taken into account for the construction of an integrated and adequate thermo-mechanical calibration model.

Using mathematical modeling allows the replace mend of natural experiments, as conducting a full-scale production in diameter 4000 mm is not desirable from an economic point of view, and large-scale modeling does not provide the necessary understanding of the complexity of thermo-mechanical calibration.

As the basis of the mathematical model used various methods of structural analysis can be. These methods include: finite element method (FEM), finite differences (FD) and boundary element method (BEM).

The method uses finite differences replacing differential equations finite-difference. The main drawback of this method is the difficulty in describing the precise boundaries and optimal overlay the area calculation of finite-difference grid.

Essence of FD is to move from the initial differential equations in partial derivatives describing processes in continuous media with given boundary conditions to equivalent integral equations. If such a transition is possible, the solution is obtained with minimal computational cost and with higher accuracy than FEM. Field application of FD limited linear problem. Therefore, it is especially widely used in linear elasticity. [1]

Equations and functions of all these three methods can be solved by analytical or numerical methods. For analytical methods solution is an algebraic function, which substituted the parameters that define the field. For numerical solution methods there is a set of numerical values that describe the field for a private communication given parameters.

Numerical methods for solving the form-results give numerical values of unknown quantities.

Describing complex phenomena using FEM for the thermo-mechanical calibration under typical for the characteristic deformation of metals and alloys, creating a detailed picture of the studied processes is necessary to create a complex mathematical model, which, in turn, requires the use of effective modern software's environment not only for the determination of deformation and temperature parameters, but also to describe the structural changes and properties of metal processing sidewalls.

Finite element method as a numerical solution methods implemented in the the world famous software complexes: COSMOS, Abaqus, ANSYS, etc.

COSMOS software family does not create contact elements automatically. For complex computational models, this leads to unnecessary time spent in the preparation of design models calculation results.

Abaqus is in the top three after «ANSYS» and «MSC/NASTRAN», widely used in the oil and gas industry U.S., Canada, as one of the most important sectors of its sales program [2], but the use of software for payments ANSYS is more appropriate due to its more widespread and the corresponding volume of the solutions and their approximation to the real processes in solving scientific and engineering problems such organizations as NASA [3-4], which involved in the development of rocket and space technology.

Using simulation module Mechanical module system – Workbench, ANSYS provides the user with all the necessary tools modeling.

The software solves ANSYS finite element stationary and non-stationary, linear and nonlinear problems of the mechanics of atomic energy, chemical and automotive industry. [5-7]

These application examples, comparisons with other software complexes and opportunities for the creation of mathematical models are sufficient basis for the application of numerical solution of the finite element method programmed in ANSYS.

**The formulation of the objectives of article.** Expediency definitions of mathematical modeling for thermo-mechanical calibration using the finite element method and its numerical solution in the ANSYS, as a research method for finding the optimal process parameters including the maximum number the factors of influence.

### **The main material.**

#### **Description of the mathematical model of the stress-strain state of shell**

Theoretical description of the stress-strain state of the shell during thermo-mechanical calibration can be obtained on the basis of "displacement method" – one of the most well-known FEM options, using the principle of possible displacements.

In matrix form three-dimensional body can be represented as [8]:

$$\iiint \{\delta \varepsilon\}^T \{\sigma\} dx dy dz = \iiint \{\delta u\}^T \{q\} dx dy dz + \iint \{\delta u\}^T \{p\} dS, \quad (1)$$

where  $\{\varepsilon\}$  – vector strain;  $\{u\}$  – vector of displacement;  $\{\sigma\}$  – vector of stresses;  $\{p\}$  – vector of the surface load;  $\{q\}$  – vector bulk load.

The equilibrium condition is written in cylindrical coordinates, for a shell of revolution based on (1) takes the form:

$$\iiint \{\delta \varepsilon_z\}^T \{\sigma_z\} r d\varphi ds dz = \iiint \{\delta u\}^T \{p\} r d\varphi ds \quad (2)$$

Index  $z$  vectors in strains and stresses shows that they correspond layer shell is distance  $z$  from the middle surface.

According to the hypothesis of direct normal:

$$\{\varepsilon_z\} = \{\varepsilon\} + z\{\mathcal{N}\}, \quad (3)$$

where  $\{\mathcal{N}\}$  – Vector changes curvature.

For layer at a distance  $z$  from the middle layer of stress vector  $\{\sigma_z\}$  is defined by:

$$\{\sigma_z\} = [D]\{\varepsilon\} + z[D]\{\mathcal{N}\}, \quad (4)$$

where  $[D]$  – matrix of elastic constants for anisotropic shell.

After substituting (3) and (4) in (2) and integration over  $z$  left side of the equation we obtain the equilibrium condition shell rotation:

$$h \iiint \{\delta\varepsilon\}^T [D]\{\varepsilon\} r d\varphi ds + \frac{h^3}{12} \iiint \{\delta\mathcal{N}\}^T [D]\{\mathcal{N}\} r d\varphi ds = \iiint \{\delta u\}^T \{p\} r d\varphi ds \quad (5)$$

where  $h$  is the thickness of the shell.

Later in solving the problem of finite element method to ask the displacement field in the element and express movement in any place through the vector of nodal displacements:

$$\{u\} = [\Phi]\{u_n\} \quad (6)$$

where  $\{u_n\}$  – Vector of nodal displacements,  $[F]$  – matrix that describes the displacement at any point through the vector of nodal displacements.

Vector strain  $\{\varepsilon\}$  and change the curvature vector  $\{\mathcal{N}\}$  Can be written as:

$$\{\varepsilon\} = [B_1]\{\varepsilon_n\}; \quad \{\mathcal{N}\} = [B_1]\{\mathcal{N}_n\} \quad (7)$$

Substituting (7) in equation (5) we obtain the element stiffness matrix shell:

$$[K] = [K_1] + [K_2] = h \iint [B_1]^T [D] [B_2] r d\varphi ds + \frac{h^3}{12} \iint [B_2]^T [D] [B_2] r d\varphi ds, \quad (8)$$

Plugins  $[K_1]$  i  $[K_2]$  meet the work force membrane bending moments.

Vector load element is:

$$\{F\} = \iint \{\Phi\}^T \{p\} r d\varphi ds \quad (9)$$

The final equilibrium equations shell will look like:

$$[K]\{u_n\} = \{F\} \quad (10)$$

Components of stiffness matrix are the ratio (8), where the integrals are taken on a cylindrical surface element length  $l$  and radius  $r$ .

### **Search the numerical solution of the mathematical model in ANSYS of the thermo-mechanical calibration**

The software environment ANSYS, finite element analysis is usually unknown is moving, and the result of the decision in this case is the vector displacement of nodes  $\{u_n\}$ . Moving in the other points is calculated by interpolation element.

After displacement field approximation (within item) corresponding polynomial is called the "form function" and depends on the number of nodes for a single element can be calculated strain and tension. This scheme calculation shows that the greatest accuracy achieved by determining the displacements at the nodes [9]:

$$\{u_n\} = [K]^{-1}\{F\} \quad (11)$$

Thus, theoretical and numerical solutions coincide.

For the numerical solution of the problem under consideration, using ANSYS to undergo several successive stages of preparation and calculation of the mathematical model thermo-mechanical calibration (TMC). These stages are [9-10]:

- 1) identification of the problem;
- 2) creating geometry models and task materials (Engineering Data);
- 3) breakdown model in the shell and external technological mandrel on the finite element mesh;

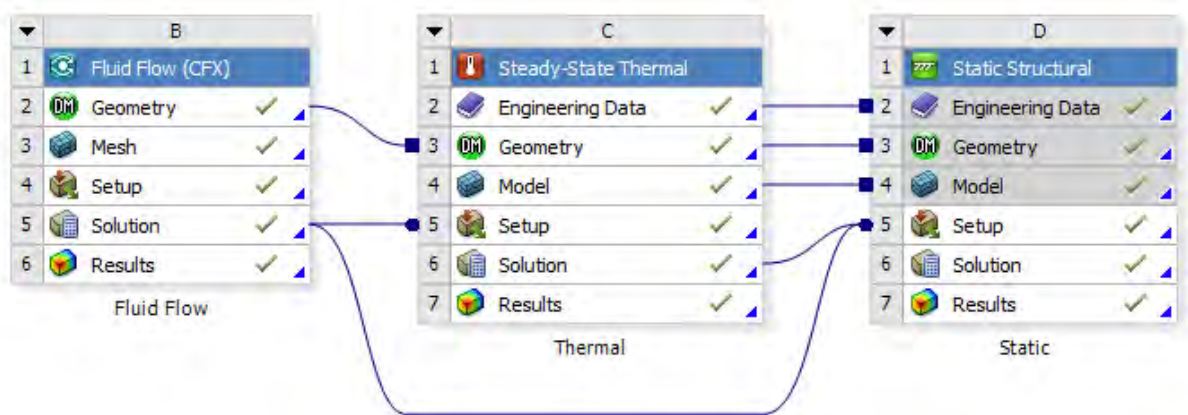
- 4) Addition to the model boundary conditions;
- 5) Numerical solution of equations;
- 6) Analysis results.

The process of thermo-mechanical calibration of large shells belonging to the class settlement cases Static structural (ANSYS), because all the elements involved structures or fixed permanently as technological equipment, or have small displacement of linear expansion of materials due to heating.

At the same time stressed-deformed state is not limited to installation stresses and strains during the actual calibration, but also extends to the foregoing, to the static analysis, the distribution of temperature fields during convection heating in a furnace, and determines residual stress after deformation.

Thus the method of finding the numerical solution of the mathematical model TMC has three successive stages (Fig.1)

- 1) investigation of convective heat and conduct heat carrier (air);
- 2) static analysis (determination of stresses and strains) based on the distribution of temperature fields;
- 3) modeling of residual stresses on the basis of the stress and strain.
- 4) evaluation of the adequacy of the model.

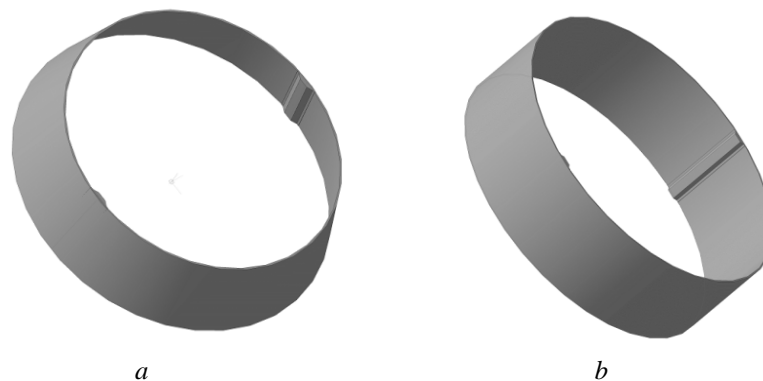


*Fig. 1. Scheme interconnection modules ANSYS for sequential determination of thermal fields and parameters of the stress-strain state of shell*

### **Modeling convection heat shells in thermo-mechanical calibration programmed in ANSYS and results analysis**

Simulation begins with the construction of three-dimensional drawings, ie the task of geometric parameters and technological shell mandrels.

Model conical and cylindrical shell, created in SAD-complex COMPASS 12V, in accordance with the design documentation, and shown in Figure 2:



*Fig. 2. Conical (a) and cylindrical (b) of the beam shell functional junction*

A geometric model of technological mandrel was created similarly.

The material for the product design process heating optimally take material product Amh6 and MF-15 as the mandrel material (Table 1)

Table1

**Mechanical and physical properties of the alloy AMr6 and cast iron GG-15 [12-11]**

Alloy	Limit of proportionality, MPa	Tensile strength, MPa	$E \times 10^4$ MPa	$\rho$ , kg / m <sup>3</sup>	$\alpha$ , 10 <sup>-6</sup> / C	$\mu$
AMr6	130-145	275-305	7.1.	2,640	24.7	0,32-0,36
GG-15	130	150	6.8	6800-7200	11.10	0,28-0,29

Module Engineering Data allows you to specify the change of mechanical parameters of materials on temperature in the tab Tabular Data, except as indicated in Table 1, fixed values. This feature is especially important for the task key properties of gray iron, during thermo-mechanical calibration – coefficient of linear expansion ( $\alpha$ ), which is non-linear increase of the temperature increase [13].

There are two methods of finite element analysis: h-method (h-length of the finite element) method and p (p – order polynomial approximation function). To improve the accuracy of the solution h-method requires increasing the number of elements. The type and amount of items determine affecting the accuracy of the calculations.

Obviously, the smaller the size of the linear finite element h the greater the number of elements in the model, and the computation time exponent but growing, and analysis errors is reduced.9.

However, errors are reduced not to zero because of the increasing number of elements accumulated rounding errors in the computer.

According to the p-method for improving the accuracy should improve order polynomial approximation function increases the number of nodes in each element of the grid.

According to the recommendations for the use of FEM mathematical models [9, 14–15, 17]:

- ordered grid has the advantage against arbitrary meshes;
- rectangular (cubic) grid with 8 nodes has an advantage against the mesh of triangular elements with intermediate nodes;
- landslides cubic polynomial approximation requires smaller grid.

These recommendations are based on the basic principles of organic modeling and calculations ANSYS. Yes degenerate elements (wedge or in the form of tetrahedral), formed with quadrangles and hexahedron without intermediate nodes, provides a lower accuracy compared with the elements of the intermediate nodes (hexagonal grid), and should not be used in areas with high stress gradients [15].

Generate hexagonal grid, for a given algorithm in ANSYS, involves the creation of a surface predominantly quadrilateral elements, and then broach this grid inside. At least the elements are in the form of tetrahedral and pyramid [4].

Drawing a hexagonal grid on selected object is in ANSYS based on the calculation of the normalized volume (I):

$$I = \left(\frac{V}{3}\right) \frac{1}{I_c S^2}, \quad (12)$$

where V – volume of the body, and C – ratio to the unit sphere and  $I_c = \frac{4\pi}{3}$ , S – area of the body.

When  $I \geq 2$ , an object imposed mainly hexagonal grid, with  $I \leq 2$  hexahedron replaced by elements in the form of a tetrahedron or a prism with the greatest possible number of nodes.

For conical and cylindrical shells coefficient normalized volume of  $1.36 \times 10^{-4}$  and  $3.47 \times 10^{-4}$ , mesh overlay ago, will consist of the elements in the form of tetrahedrons. Finite element mesh binding process, based on  $I = 2.36$  – for conical mandrel (2.6 – for cylindrical), will consist of the elements in the form of parallelepipeds.

Changing the degree of the polynomial approximation function, according to method depends on selecting the type of finite elements and the corresponding number of nodes.

For hexagonal wire mesh is the most optimal SOLID 90 (cubic element) and SOLID 92 (tetrahedron), which has 20 nodes and 10, respectively [10]. These types of items are set automatically when selecting algorithm generating hexagonal grid and have the maximum possible number of nodes in the element.

A condition for creating hexagonal grid corresponds to the way to generate meshes – Hex Dominant. This method fills the volume mesh generation of these bodies parallelepiped, and where this is not possible, uses triangular prisms [14].

Hexagonal mesh of finite elements for cylindrical and conical shell after shell connection with technological mandrel shown in Fig. 3

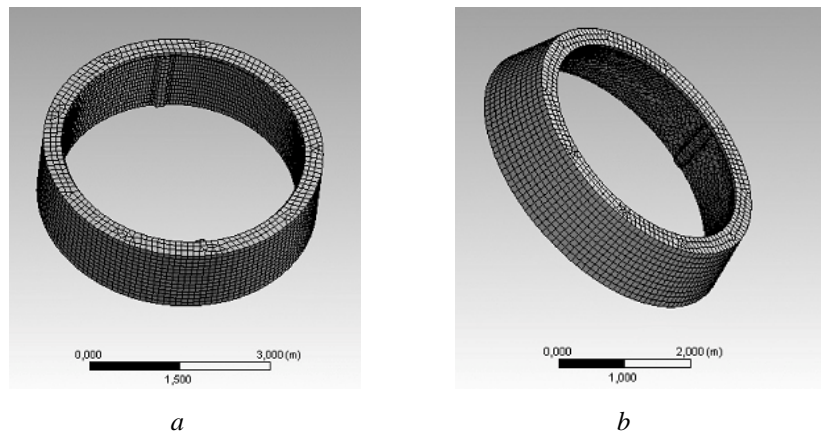


Fig. 3. Finite element mesh for the cylindrical (a) and conical (b) shell with technological mandrel

In Workbench is the ability of local changes in the grid. More frequent grid to where the gradient is high strain or stress in areas of special interest. In the case of thermo-mechanical predetermined calibration welded shells area of special interest is a functional beam intersection and adjoining technological mandrel surface technology, flashy in Fig. 4.

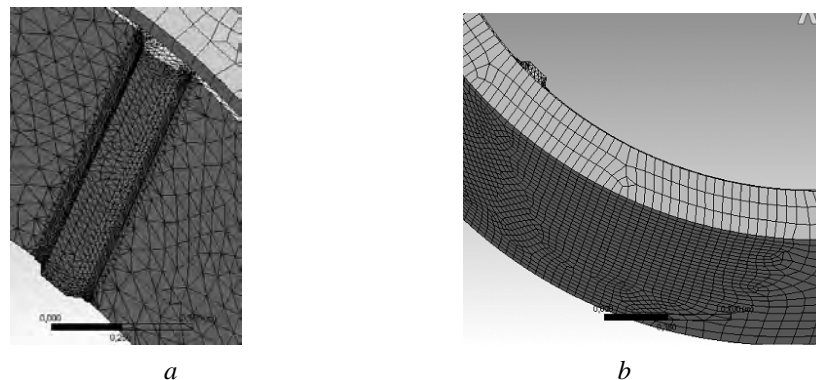


Fig. 4. Dense mesh beams functional junction (a) and surface binding process (b)

The function of local mesh changes – Sphere of Influence (zone changes as sphere) defines radius of the sphere within which the elements are generated with a given size and Refinement (seal mesh) – increases by 1-3 orders of magnitude the number of items selected surface or edge compared to the density total body mesh elements.

Shown in Fig. 3 and Fig. 4 finite element mesh has length side element – 10 cm in functional joint density grid increased by one order ( $h=1$  cm), the type of items – SOLID 90 and SOLID 92. The total number of nodes finite element mesh is 275 000.

The resulting finite element mesh imported module later in the formation of the physical conditions of the process Pre-ANSYS CFX.

The relevant tabs indicate that the problem is a dynamic Simulation Type-Transient (initial parameters change over time) and set the type of calculation of heat transfer Heat Transfer-Thermal Energy.

The next step is to determine the boundary conditions. In Fig.5 the mathematical model with input and output planes air.

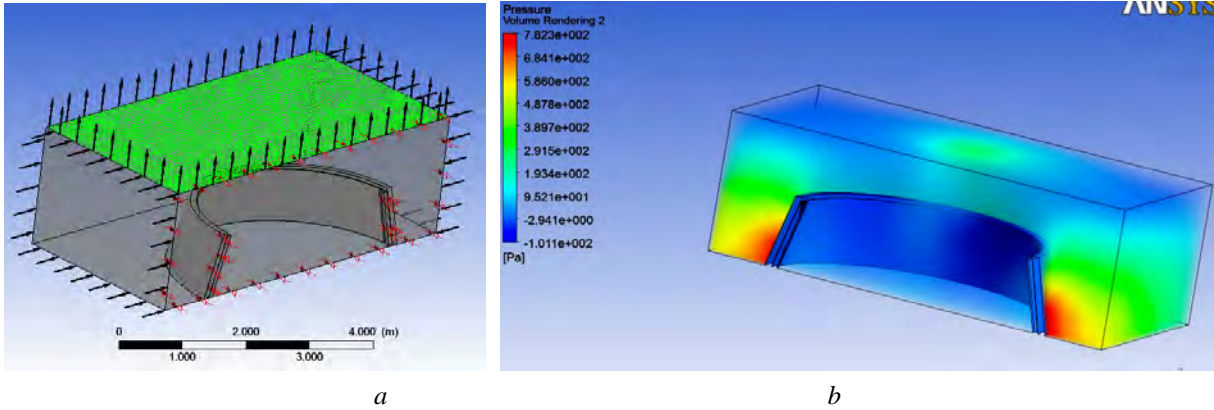


Fig. 5. Boundary conditions of the mathematical model of convection heating (a) and three-dimensional picture of the distribution of pressure coolant during thermo-mechanical calibration (b)

Fig. 5b gives an idea of the distribution of pressure during thermo-mechanical calibration with existing furnace design with bilateral lateral input of heated air.

The results of pressure distribution in the module are transferred to CFX model calculation Steady-State Thermal, which will set the parameters convection shell and mandrel materials and process temperature. In Fig. 6 shows the result of calculation of heat flow during thermo-mechanical calibration. Uneven distribution of heat zones at an important element in building as close to the real conditions of the process.

The final stage of the simulation is to import the data of pressure and heat flows in Static Structural module for the parameters stress-strain state of shell.

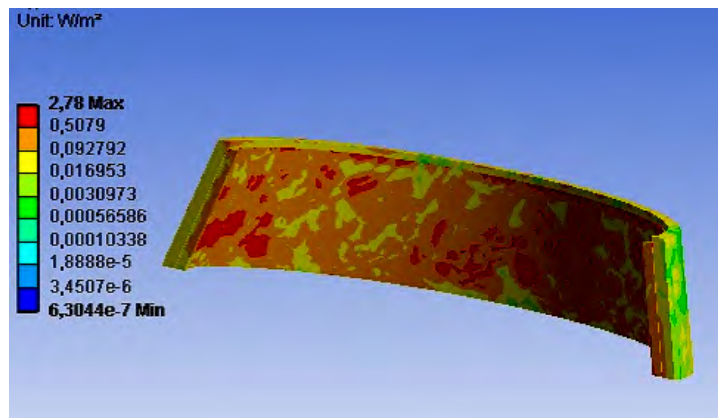


Fig. 6. Distribution total heat flow during the process temperature process

### Numerical solutions of the stress-strain state of shells during thermo-mechanical calibration programmed in ANSYS and results analysis

At this step, the initial conditions are given in the form of admissible displacements (displacement) and external surfaces shell binding on the vertical axis Z, left axis projected displacements and deformations of X and Y are free.

The next step is the task of the initial parameters of the heating process and calculation of parameters of the stress-strain state (relative strain and equivalent stress).

The most optimal relative validity of the results is to test the dependence of the final heating temperature, total strain and equivalent stress [13]. The final heating temperature should not exceed 350 °C. At temperatures above 350 °C alloys belonging to the system Al-Mg-Mn, characterized by relatively low yield strength and formation  $\beta$ -Phase alloy, which dramatically reduces the corrosion resistance of the alloy and can lead to the appearance on the surface of the metal oily plaque [16].

Checking relationship and the desired final temperature thermo-mechanical process parameters calibration is implemented in a series of 12 experiments with a given rate of heating 150 °C / hour, the initial temperature of 20 °C and a final temperature range of 100-350 °C. The technological gap between the equipment and the shell is 10 mm. Baseline data and simulation results are presented in Table 2.

Maximum equivalent stresses indicated in Table 2, observed on area, but at least – the technology mandrel. The upper limit of the range of equivalent stress points on the plastic deformation of shells.

The upper limit of the range of values of the total strain (Table 2), describes technological equipment, lower – the shell. Equal value of the lower boundary indicates complete overlap technological gap between the shell and equipment.

Table 2

Changing the stress-strain state of shells from the final heating temperature

Number	T, °C	Cylindrical shell			Conical shell		
		$\varepsilon_t, 10^{-3}$	$\sigma_f, \text{MPa}$	$\delta, \text{mm}$	$\varepsilon_t, 10^{-3}$	$\sigma_f, \text{MPa}$	$\delta, \text{mm}$
1	100	0,57–2,1	54,6–488	10,1–11	0,57–2,1	52,6–460	10,2–10,7
2	150	1,4–3,9	90,9–815	10,1–13,1	1,4–3,9	86,5–754	10,2–11,46
3	200	2,1–4,7	125–1024	10,1–16,1	2,1–4,7	120–1047	10,2–12,3
4	250	2,9–6,1	163–1454	10,1–19	2,9–6,1	156–1348	10,2–13,7
5	300	3,7–7,6	201–1789	10,1–24,2	3,7–7,6	187–1650	10,2–15,6
6	350	4,5–9	235–2100	10,1–28,6	4,5–9	221–1929	10,2–17,6

$\varepsilon_t$  – temperature deformation;  $\delta$  – total deformation;  $\sigma_e$  – equivalent stress.

Distribution of equivalent stress has the same character for cylindrical and conical shells and shown in Fig. 7.

Smaller range of desired values for the conical mandrel explained inclined plane design, which affects the

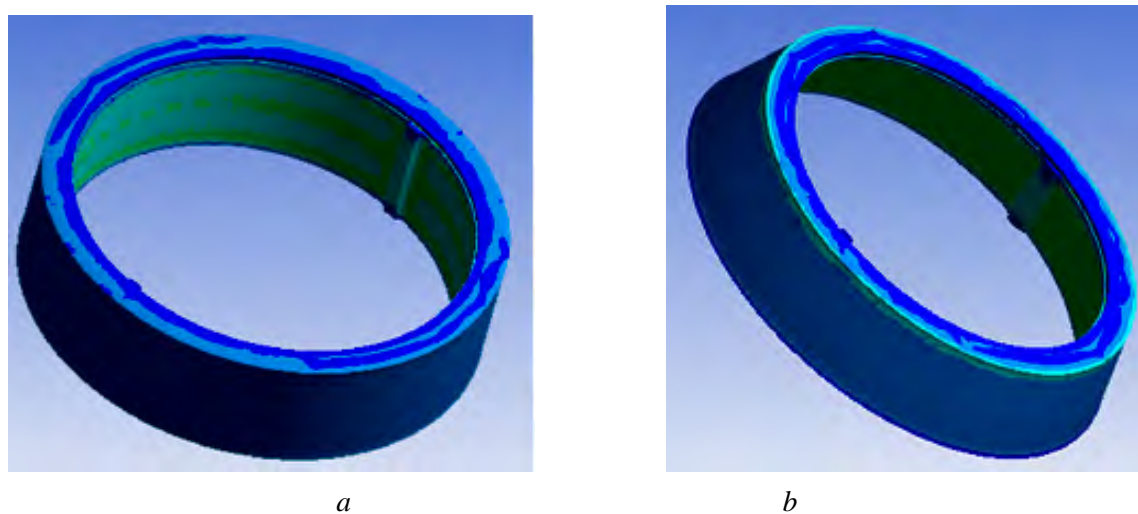


Fig. 7. Equivalent voltage cylindrical shell and mandrel at a temperature of 350 °C



General deformation of cylindrical and conical shells have the same character on the basis of its distribution, differing only in magnitude at critical points, for all combined to analyze the mode of heating (Fig. 8, 9).

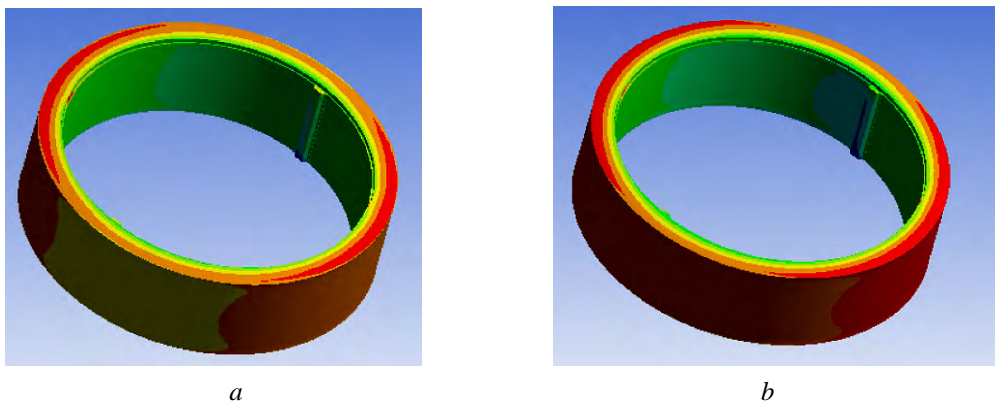


Fig. 8. Distribution of deformation of cylindrical shell and mandrel at 100 °C (a) and 350 °C (b)

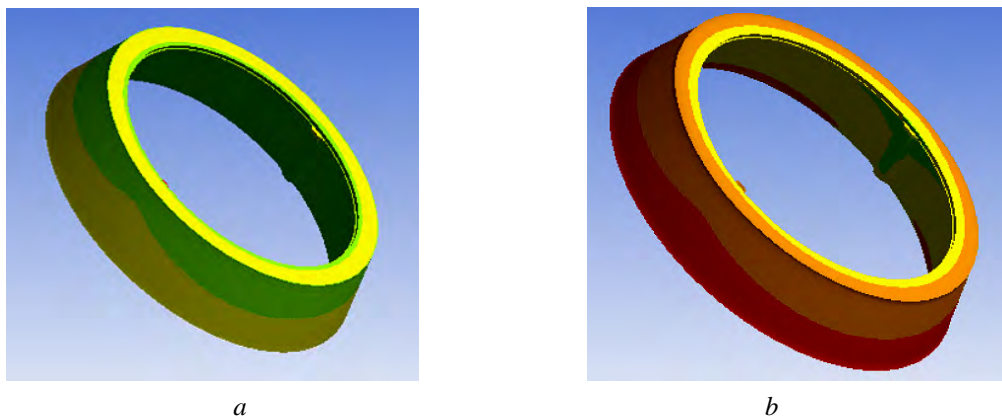


Fig. 9. Distribution of deformation conical shell and mandrel at 100 °C (a) and 350 °C (b)

The values in Table 2 and the nature of their distribution (Fig. 4-6), are especially important to determine the stress-strain state binding, defining opportunities thermo-mechanical process calibration of large shells.

Correlation final heating temperature, shown in Fig. 7, for the interrelation of the equivalent stress and total deformation, based on the simulation results.

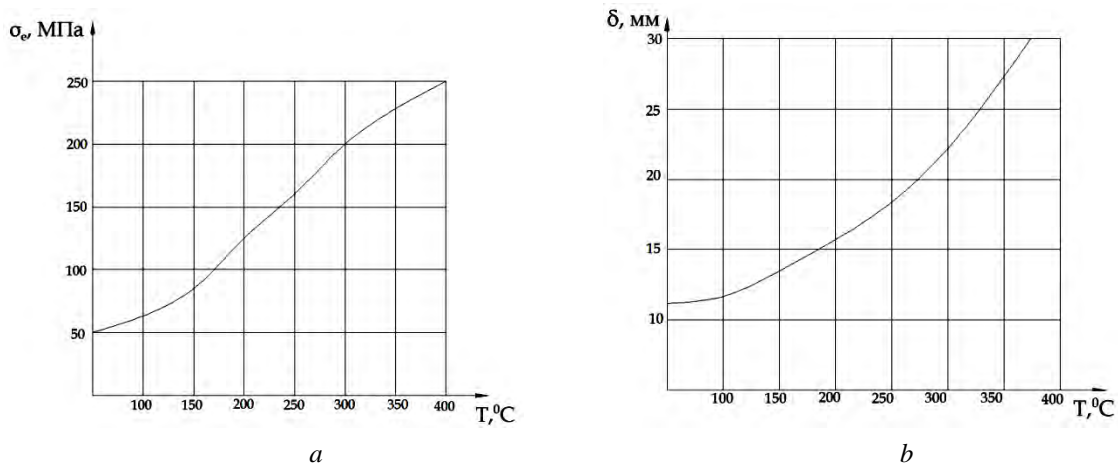


Fig. 10. Dependence of equivalent stresses (a) and total strain (b) for the binding process of the value of final heating temperature

Patterns are shown in Fig. 10 for the equivalent stress and total deformation coincide reference data. 12.

Mathematical model demonstrates compliance with the real process of thermo-mechanical calibration. Shown in Fig. 7 (a) equivalent stress in Table 2 with temperature 200 °C indicate the thermo-mechanical deformation as overlapping technological gap of 10 mm on each side, to a temperature of 200 °C equivalent temperature characterizing the thermal expansion of the material stress from heat. From Figure 7 also turns out that the temperature of the process should be in the range of 250-300 °C. This is also of junk  $\beta$ -phase alloy, also with the restriction that depends on the material processing mandrel – MF-15 can withstand voltages up to 200 MPa.

### Modeling of residual stresses

#### in the shell after thermo-mechanical calibration software environment ANSYS

The results of equivalent stress and relative strain shell and mandrel are the basis for further modeling.

So in the initial static analysis in section loads and boundary conditions introduced command object Commands (APDL), which includes the team recording the findings of stress in an external file. 18.

Next, create a duplicate of the original static analysis (Fig. 11a) with the same parameters of the model, but now part loads entered command object to read the file from an external stress (Fig. 11b).

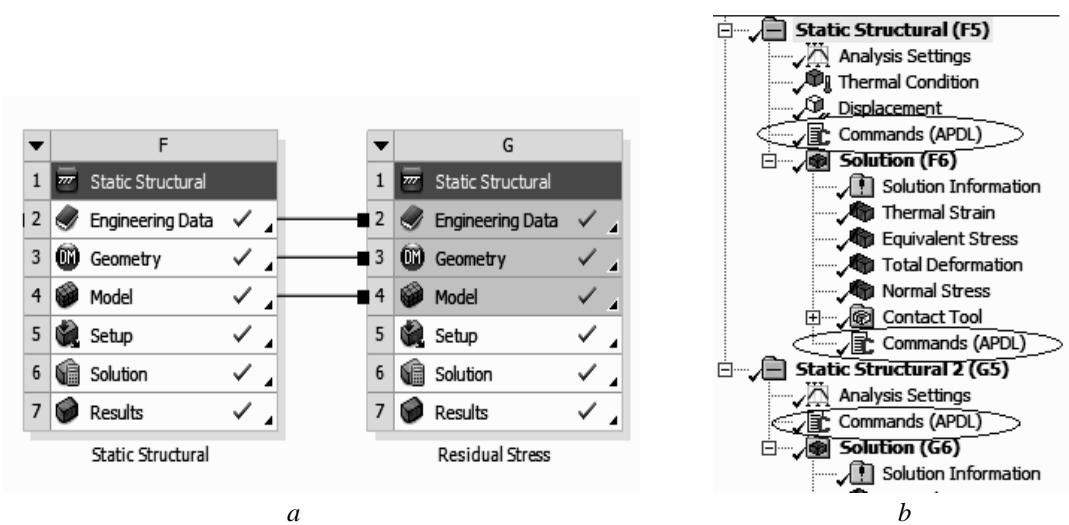


Fig. 11. Scheme interconnection modules ANSYS to determine the residual stress (a) and layout of command objects when creating models

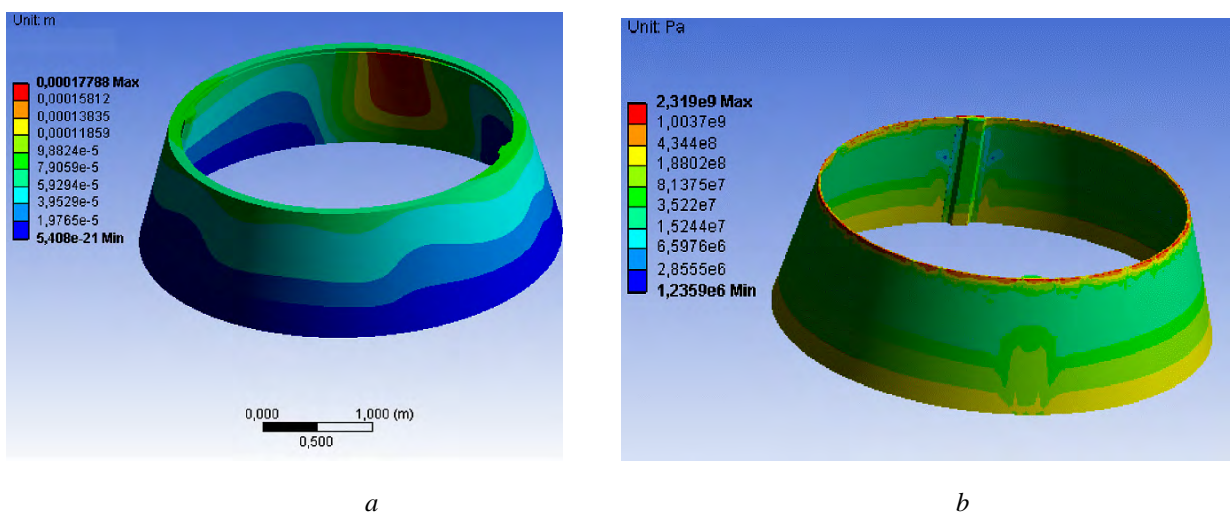


Fig. 12. Move (no relative deformation) shell and snap (a), residual stresses after cooling to shell craft temperature (b)

Since the stress field are read from an external file by itself balanced way, moving the shell and mandrel are close to zero (Fig. 12a), and efforts are shown in Fig. 12b, at the time of cooling shell to guild temperature, give an idea of the magnitude and nature of residual stresses.

**Summary.** ANSYS is the best choice for calculating the parameters of thermo-mechanical calibration, compared with other under consideration software complexes based on the most appropriate overall direction and wide use in engineering and scientific calculations of the stress-strain state of shells.

Task nonlinear behavior of mechanical parameters of materials, density and type of finite element mesh with the possibility of local compaction; provide, in addition to clarity of results, the main advantages of using ANSYS to simulate the process of ensuring the accuracy of thermo-mechanical calibration of large shells.

Until a few and not significant shortcomings in the modeling of TMC process in ANSYS, as in any software environment include:

- The need to set of many parameters on the preparation stages and initial conditions calculation process TMC;
- Correct interpretation of the results depends on the adequacy of the initial conditions for real process TMC for verification which may be necessary additional tests and combinations of initial parameters.

For considered in this work the initial parameters, were calculated the parameters of the stress-strain state design`s shell and technological mandrels that were match for previously defined in the literature parameters for the heat treatment process and corrections.

Defined deformation`s character, distribution of temperature and equivalent stress for cylindrical and conical shells, which is only possible when using this mathematical model and the proposed numerical solution of the equations that describe it.

Mathematical modeling of thermo-mechanical calibration using the finite element method and its numerical solution in ANSYS, appropriate research method for finding the optimal parameters based on the maximum number of impacts.

Using FEM and ANSYS for testing process TMC for responsible products can significantly reduce the cost and volume of development work in implementing in production.

It is planned the comprehensive study of thermo-mechanical calibration process to determine the optimal parameters of technology and consideration of all possible factors affecting the final accuracy shells geometry.

1. Фисенко В. Г. Численные расчеты электромагнитных полей в электрических машинах на основе метода конечных элементов: учеб. пособ. по курсу Электромагнитные расчеты. – М., 2002. – 46 с.  
2. Безопасность магистральный газопроводов с использованием анализа их несущей способности [Электронный ресурс]: метод построения прочностного симулятора для прогнозирования пожарной и промышленной безопасности трубопроводных сетей промышленных энергообъектов / Селеванов А.В. – Режим доступа: <http://www.all-referats.com/diploms/diplom-55384.html>.  
3. Application of ANSYS Workbench [Электронный ресурс] – Режим доступа: [http://ntrs.nasa.gov/archive/nasa/casi.ntrs.nasa.gov/20070014890\\_2007006932.pdf](http://ntrs.nasa.gov/archive/nasa/casi.ntrs.nasa.gov/20070014890_2007006932.pdf). ANSYS Cart3D Selected as NASA's `Best Software Of The Year [Электронный ресурс]: PR Newswire / Stevenson D. – Режим доступа: <http://www.prnewswire.com/news-releases/ansys-cart3d-selected-as-nasas-best-software-of-the-year-ansys-icem-cfd-cart3d-software-available-for-commercial-distribution-76649047.html>.  
4. Энергетическое машиностроение: инженерно-технический журнал; основан ANSYS, Inc., главн. ред. Д.А.Хитрых. – 2005. – ANSYS, Inc. 2009 – Выходит 4 раза в год. ISSN 1561-4738. – 2009. – № 11.  
4. Туйкин Н.М. Определение остаточного ресурса безопасной эксплуатации аппаратов химических производств с вмятинами в узлах пересечения патрубков и обечаек: автореф. дисс. на присвоен. степени канд. техн. наук: спец 05.26.03 "Пожарная безопасность" / Туйкин Наиль Мухамметович; Казанск. гос. технолог. ун-т – Казань, 2004. – 28 с.  
5. Нефтегазовое дело: научно-технический журнал; учредитель журнала Уфимский государственный нефтяной технический университет, главн. ред. Р.Н. Бахтизин. – 2002. – Уфа: Нефтегазовое дело, 2011 – Выходит 6 раз в год. ISSN 2073-0128. – 2011 – № 2.  
6. Балабух Леонид Иванович. Строительная механика ракет:

Учеб. для машиностроительных спец. вузов / Л.И. Балабух, Н.А. Алфутов – М.: Высш. шк., 1984. – 391 с. 6. Каплун Анатолий Евгеньевич. ANSYS в руках инженера: Практическое руководство / А.Е. Каплун, В.А. Морозов – М. Едиториал УРСС, 2003. – 272 с. 7. Ibid. ANSYS Operation Guide. Ibid. ANSYS Modeling and Meshing Guide. Ibid. ANSYS Structural Analysis Guide. Ibid. ANSYS Commands Reference. 8. Марочник стали и сплавов [Электронный ресурс]: характеристика материала АМзб / Бреславский Д.В. – Режим доступа: [http://www.splav.kharkov.com/mat\\_start.php?name\\_id=1422](http://www.splav.kharkov.com/mat_start.php?name_id=1422). 9. Справочник по чугуному литью / под ред. д-ра техн. наук Н.Г. Гиришовича. – 3-е изд. – Л.: Машиностроение, 1978. – 758 с. 10. Справочник в 4-х томах. Ковка и штамповка. / под ред. Е.И. Семенова. – М.: Машиностроение, 1984. – 568 с. 11. Бруйка В. А. Инженерный анализ в ANSYS Workbench: учеб. пособ / В.А. Бруйка, В.Г. Фокин, Е.А. Солдусова – Самара, 2010. – 271 с. 12. Басов Константин Александрович. ANSYS: справочник пользователя. – М.: ДМК Пресс, 2005. – 640 с. 13. Термическая обработка в машиностроении / под ред. Ю.М. Лахтина. – М.: Машиностроение, 1980. – 783 с. 13. Зінкевич О. І. Метод кінцевих елементів у техніці – М.: Мир, 1975 – 115 с. 14. Инициализация остаточных напряжений [Электронный ресурс]: Инициализация поля остаточных напряжений в конструкции / Лукашов Л.М. – Режим доступа: <http://www.cae-club.ru/content/initsializatsiya-ostatochnykh-napryazhenii>.