Advanced Computational Concepts about Projecting a Multiple Designs of Self-Supporting Metallic Structure using Finite Element Method in Determination the Buckling Factor and Running the Stress Analysis

MADALINA CALBUREANU
Faculty of Mechanics
University of Craiova
25 Nicolae Titulescu St., G2, 1fl, 5,
200219, Craiova
ROMANIA
madalina.calbureanu@gmail.com

EMIL ALBOTA
Faculty of Civil, Industrial and
Agricultural Buildings
Technical University of Civil Engineering
124 B-dul Lacul Tei,
020396, Bucharest, ROMANIA
emil_albota@yahoo.com

RALUCA MALCIU
Faculty of Mechanics
University of Craiova
Doljului, C8c, 19, Craiova
ROMANIA
rmalciu@yahoo.com

ROMULUS LUNGU
Faculty of Electrotechnics
University of Craiova
107, Decebal St.
ROMANIA
romulus_lungu@yahoo.com

DAN CALBUREANU
University of Craiova
25 Nicolae Titulescu St., G2,
1fl, 5, 200219, Craiova
ROMANIA
dan.calbureanu@gmail.com

Abstract: - The paper presents the fully and correctly computation of a self-supporting metallic structure by using the capabilities of NISA-program and Finite Element Method. The stress and buckling analysis using SHELL and TRUSS Elements are performed for optimizing the design of the structure. The system of loading forces includes all the types of loads specified by the international standards for the civil buildings. The values of loads depend on the structure location zone. Permanent, temporary (quasi-permanent and variable) loads and the extremity loads are considered. It is been also realized a designing of the structure based on the obtained results. The advanced computational concept used in designing multiple shapes for the metallic structures were introduced by MACRO program. This program can realize automatically modeling of the structure introducing in a parametrical way the dimensions of the metallic building.

Key-Words: - Self-supporting metallic structure, design, buckling factor, stress analysis

1 Introduction

As a result of the need for low cost buildings, with easy assembly and disassembly, for moving from a location to another and high resistance to corrosion it is more necessary projecting and designing the self-supported metallic structures.

The destination of this kind of structure can be:
- material deposit;
- garage;
- hangar for airplanes;
- shop;
- space for repairing activities.

Function of the destination, the factor of safety is rising with the importance of the stuff inside (materials, animals and people).

The modeling of the self-supporting metallic structure consists in sample assembly of prefabricated elements [2].

For various uses and improving the design, the calculation program can be adapted by using different solutions such as:
- semi cylindrical shape;
- an elevated semi cylinder;
- a quarter of a cylinder;
- an elevated quarter of cylinder;
- incomplete cylindrical shape with center angle (a) bigger than 90° and less than 180°;
- incomplete cylindrical shape but elevated.

A full and correct calculation of this structure can be made only by computing with Finite Element Method [4], because the structure was constrained by many complex loads and we need to compute more variants of shapes and loads.

This paper realizes a MACRO program using NISA capabilities [5] for designing multiple cylindrical shapes subjected to multiple load cases.

2 Problem Formulation

The capabilities of NISA-program, the MACRO language were used for introducing in parametrical way a number of structure's elements dimensions such as:
- the radius of the structure;
- the center angle of the structure;
- the angle corresponding for one module;
- the number of archways taken into account;
- the thickness of the iron sheet;
- the density of the material.

The mechanical characteristics which can be taken according with the beneficiary [2]:

a) The wind action: the product is designed to be used in regions at 800 m height above the sea level, where the wind speed is almost 120 km/h or at 1200m above the sea level, where the wind speed is 150 km/h maximum.

b) The snow action: the product is designed to support the snow action by static external forces, which act on vertical direction over the exposed construction elements.

Technical characteristics and performances consist in:
- the resistance to the aggression of the environment: the product is designed to be used in continental and temperate climate;
- the capability to install technological equipment.

The technological equipment is represented by:
- Technological force with uniform distribution of 0.75kN/m^2;
- Vertical concentrated technological force of 1 kN/m^2;
- Horizontal concentrated technological force of 1 kN/m^2, applied at 0.9 m height from the ground.

The basic shape is described in fig. 1 with the main dimensions.

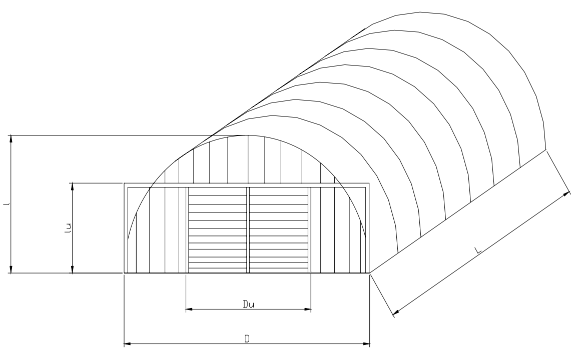

Fig. 1 The basic shape for the self-supported metallic structure

The calculation was made for such a number of archways as to obtain a significant length of the structure.

The number of archways, the module iron sheet's thickness and the dimensions of one element such as: the length of the module, the length of the wings, the side-walls inclination and the length of the middle-side were also introduced in parametrical way in MACRO program [5].

The variety of the metallic structures shapes can be visualized in figure 2.

In figure 2, the parameter n represents the number of elements on one archway of the structure, "D" is the diameter of the cylindrical shape, "H" is an over height for high type of structures and "a" is the angular opening of the piece of cylindrical shape.

The parameter n can take the values between 5 and 21 for different sizes of diameter of the building.

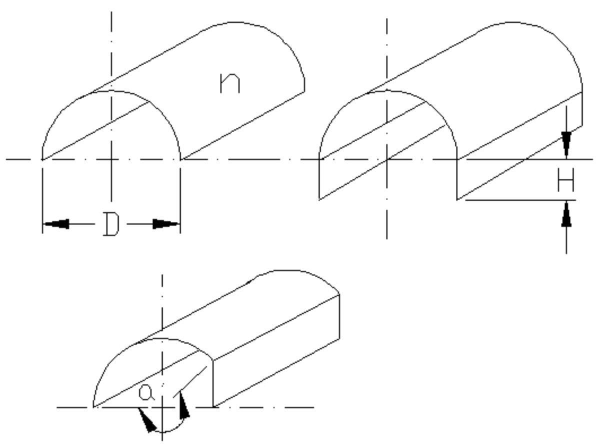

Fig. 2 Parameters for design of different metallic structures shapes

The material for the metallic structure is OL37K iron sheet, generally used for steel building, with the mechanical characteristics given in table 1.

Table 1 Mechanical characteristic for material

Steel	Stretching Strain limit R_{cH} [N/mm^2]	Stretch Resistance R_m [N/mm^2]	Breaking elongation A[%]
OL37k	240	360	26

There are three types of submitted profiles:

1. Open profile with the shape and dimensions as shown in fig. 3 form (1):

a) the length of medium fiber is 1000 mm;

b) the thickness of the sheet iron is 0.5÷3 mm;

c) stiffening elements have filleted corners radii: R5÷R15 mm;

d) module curved radii: R4000÷R14000 mm;

e) the dimensions can be modified but the shape of the profile must remain the same.
This open profile gives the advantages of low material consumption and feasibleness, which has no problem with the temperature variations (the dilatations are free), and the disadvantage of low stability of the element.
2. Closed profile with the shape and dimensions as shown in fig. 3 form (2):
a) the length of medium fiber is 1000 mm;
b) the thickness of the sheet iron is 0.5÷3 mm
c) stiffening elements have filleted corners radii: R5÷R15 mm.
This closed profile is made of two iron sheets, so it has a bigger material consumption and weight and the execution is more difficult than in the case of the first one. Also the dilatations aren't free, so problems can appear there because of the temperature variations. Instead it is more resistant at the applied forces.

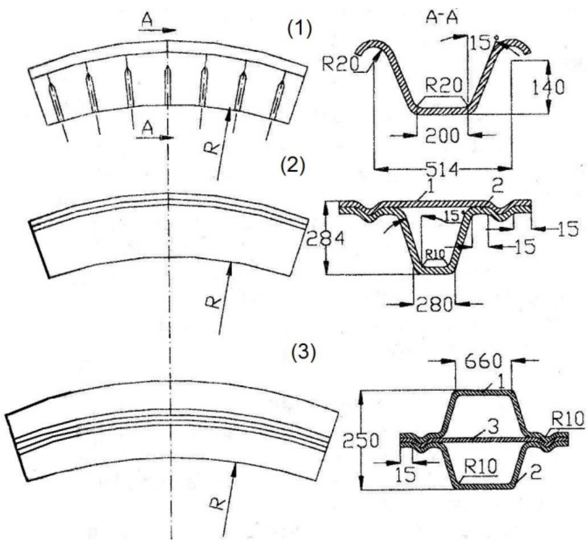

Fig. 3 Three types of module element profile

3. Closed profile with the shape and dimensions as shown in fig. 3 form (3):
a) the length of medium fiber is 1000 mm;
b) the thickness of the sheet iron is 0.5÷3 mm
c) stiffening elements have filleted corners radii: R5÷R15 mm.
This profile is the strongest, but it is made of three iron sheets, so the self weight is the biggest. The material consumption is the highest and the temperature variation causes large dilatations.

3 Steps in generation of the finite element model of the structure

The geometrical and finite model is generated following the next steps [4]:
- the nodes are defined in XOY plan by starting from one side of the archway;
- the nodes obtained are rotated with the generating angle of one element module;
- the SHELL elements (finite elements) are built with NKTP = 20, NORDR = 14, by connecting the proper nodes;
- the elements are rotated around OX axis of Global Coordinate System for building the semi cylindrical shape of one archway;
- more new archways are generated by translating along OX axis the old archway, till we obtain the final number of archways;
- the TRUSS elements are introduced at the first three archways from each extremity of the structure for equilibrate the stresses on entire building;
- the load system is added according to the international standards.

4 The system of loading forces for the model

The system of loading forces includes all the types of loads specified by the international standards for the civil buildings. The values of loads depend on the structure location zone.
The types of loads applied to the structure are:
- Permanent loads, "p";
- Temporary loads:
b1. - quasi-permanent loads "c";
b2. - Variable loads "v";
- Extremity loads "e".

4.1 Permanent loads

The permanent loads are applied continuously, with a practically constant intensity in proportion to the time, figure 4. They are given by:
- the body force of the building permanent elements (the structural elements, the shutting off elements) made by iron or duralumin;
- the thermal isolation;
- working accessories, such as illumination and ventilation.

4.1.1 The elements body force

The elements body force is taken into account according to the coefficient "n" - chosen as recommended by the standard STAS 10101/0A-77: n = max. 1.1; the result for this modeling is a uniform distributed load on the entire structure [9].

4.1.2 Loads given by the thermal isolation

The load given by the thermal isolation is determined considering 6 N/m^2 for a 10 mm thickness layer, having a uniform distributed load on the entire structure, too. The coefficient "n" was chosen n = 1.1.

4.1.3 Loads **given by the working accessories**
The loads given by the working accessories, such as illumination and ventilation, are taken into account as it follows:
(1) Vertical technological loads, for roofs: p_{31} = 500 N/m^2, according to STAS 10101/2A1-87; the loading was modeling as a uniform distributed loading on the entire structure.
(2) Vertical-concentrated load p_{32} = 1000 N/m^2 on each 4 m;
(3) Horizontal, linear and uniform distributed load p_{33} = 1000 N/m^2, applied at 0.9 m above the ground. The coefficient "n" is chosen according to the standard STAS 10101/0A-77: n=max.1.1.

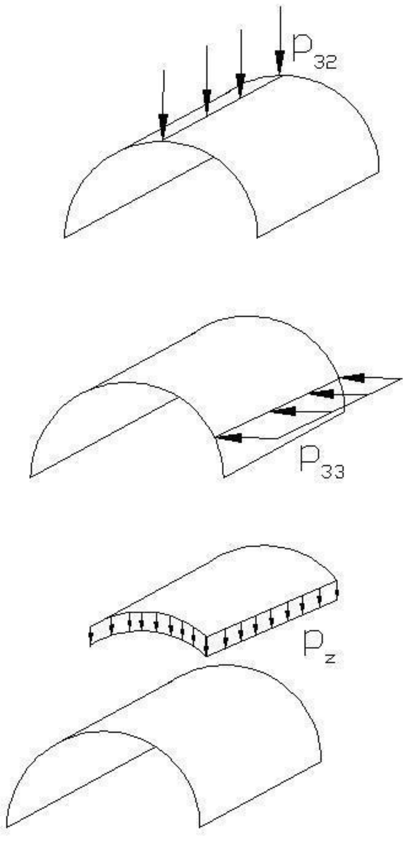

Fig. 4 The permanent loads and the quasi-permanent snow load (for the regions with cold climate)

4. 2 The temporary loads
The temporary system loads are represented by the quasi-permanent loads and variable loads.

4.2.1. Quasi-permanent loads
The quasi-permanent loads owing to the deposed dust are not taken into account in the verifying calculus because of their insignificant influence.

4.2.2. Variable loads
The variable loads - figure 5 -are given by the following actions:
- The load given by the snow uniform distributed weight: p_z = 1200 N/m^2;
- The load given by the wind action, with the following modeling components:
(a) the normal one: variable along the archway; it was determined function of the basic dynamic pressure at that height: g_v = 1100 N/m^2 (according to STAS 10101/20-78, for a location outside the city, zone "E")
(b) the internal one: uniform distributed on the entire structure; p_{v2}=-440 N/m^2;
(c) the frontal anterior one: uniform distributed; p_{v3f}=-960 N/m^2; F_{v3f}= 1508 R^2 N;
(d) the frontal posterior one: uniform distributed; p_{v3f}= - 720 N/m^2; F_{v3f}= -1131 R^2 N.

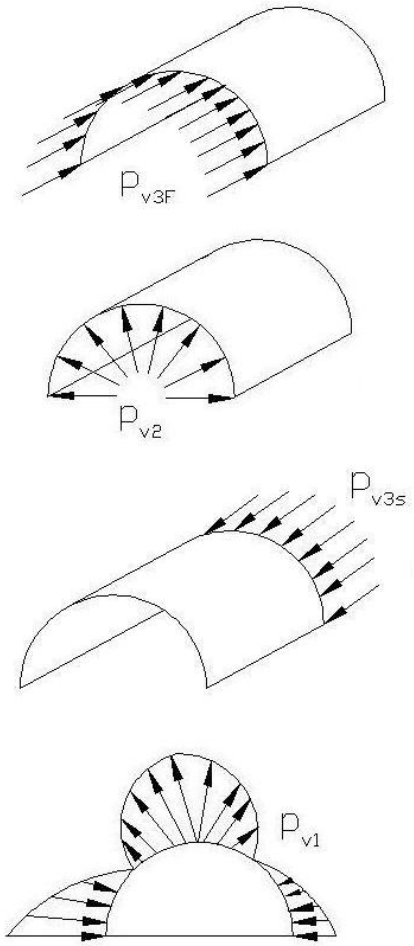

Fig. 5 The variable loads given by the wind

The coefficients "n" and "n^d" are considered according to STAS 10101/0A-77, zone E, as it follows:
- for loads given by the snow weight: n = 1.6; n^d = 0.6;
- for loads given by the wind action: n = 1.2; n^d = 0.4.

The normalized intensity of the loads due to the wind action (d) is expressed by:

$$P_{v1} = \beta \cdot c_{n1} \cdot g_v \quad (1)$$

where: - β - dynamic coefficient;
- c_{n1}- coefficient of the normal component of the side wind action, given in Table 1;
- g_v – basic dynamic pressure at that height: g_v = 1100N/m^2 (according to STAS 10101/20-78, for "E" zone and location outside the localities.

Table 1 The coefficient of the normal component of the side wind action

α	c_{n1}
0°	+1.0
15°	+0.9
30°	+0.5
45°	-0.1
60°	-0.7
75°	-1.1
90°	-1.2
105°	-1.0
120°	-0.6
135°	-0.2
150°	+0.1
165°	+0.3
180°	+0.4

The load due to the wind action expressed by relation (1) is represented in fig.5.

The internal load due to the wind action (b) is defined as:

$$P_{v2} = \beta \cdot c_{n3} \cdot g_v \quad (2)$$

where β and g_v have the same meaning and c_{n3} is referring to the rectangular closed buildings in-plane: c_{n3}= -0.4.
So, it is obtained:

$$P_{v2} = -\ 440 \text{ N/m}^2$$

The frontal load due to the in front wind action (extremity effect) (a) is expressed by:

$$P_{v3} = \beta \cdot c_n \cdot g_v \quad (3)$$

where: $\beta = 1.2$; $c_n = 0.8$; $g_v = 1000$ N/m^2.
So, it is obtained: $P_{v3\,f} = 960$ N/m^2.
The corresponding concentrated force will be:

$$F_{v3\,f} = P_{v3\,f} \cdot \pi \cdot R^2 / 2,$$

that is:

$$F_{v3\,f} = 1508\ R^2 \text{ [N]}.$$

The frontal load due to the "from behind" wind action (c) is expressed by:

$$P_{v3} = \beta \cdot c_n \cdot g_v \quad (4)$$

where:

$$\beta = 1.2;\ c_n = 0.6;\ g_v = 1000 \text{ N/m}^2$$

So, it is obtained:

$$P_{v3\,f} = -720 \text{ N/m}^2$$

The corresponding concentrated force will be:

$$F_{v3\,s} = P_{v3\,s} \cdot \pi \cdot R^2 / 2,$$

that is:

$$F_{v3\,s} = -1131\ R^2 \text{ [N]}$$

The internal load (b) and the frontal loads (c), (a) due to the wind action are shown respectively in figure 5.
The boundary conditions were introduced by fitting the basis elements in foundation [1].
The uniform distributed loads on the entire structure were introduced by modifying the material's specific weight and depending on the unit pressures.

4.3 The multiple load cases for preparing the stress and buckling analysis

The structure was calculated for three distinct groups of loads:
1) The load's hypothesis for normal exploitation:

$$\sum p_i + \sum c_i + ng \sum v_j \quad (5)$$

where:
ng –group coefficient with the following values:
ng = 1, if there is a single variable load;
ng = 0.9, if there are two or three variable loads;
ng = 0.8, if there four or more variable loads.

2) The load's hypothesis for limit exploit:
This kind of load case is made by permanent loads, quasi-permanent, variable loads and exceptional loads regarding the climate zone.

3) The load's hypothesis for fundamental group:

$$\sum n_i p_i + \sum n_i c_i + ng \sum n_j v_j \quad (6)$$

where:
ng –group coefficient with he same values as in case 1).
In this case the equation (2) becomes:
$np_1 + ng \cdot n \cdot v_v$ (wind action and body force)
The load's hypotheses considered in our case are:
I. $p_1 + p_2 + p_{31} + p_{32} + p_{33} + ng \cdot v_z + ng \cdot v_v$
(all the loads are present);
II. $p_1 + p_2 + p_{31} + p_{32} + p_{33} + ng \cdot v_z$ (without the wind action).

4.4 The MACRO program for multiple shape and designs in modeling the building

In order to realize the MACRO programs were followed the steps presented before in paragraph 3.

First part of the program defines all the parameters:

- the radius of the structure- r;
- the angle corresponding for one module - ALFAR;
- the number of archways taken into account-NA;
- the thickness of the iron sheet - g;
- the density of the material – DENSR;
- the gravity acceleration at the location where the structure will be used – GRAV.

```
MACRO, WALL1
let v=0.001
LET NA=8
let r=10000.*V
let g=2*V
Let ALFAR=15.
LET N=(180/(alfar))-1
LET NM=N+1
LET DENSR=7850
LET GRAV=9.81
LET DENSF=DENSR+1700./(G*GRAV)
```

- the nodes are defined in XOY plan by starting from one side of the archway;
- the nodes obtained are rotated with the generating angle of one element module;

```
NOD,ADD,1,0/(-240.*V)/0/5
NOD,ADD,2,(120.*V)/(-240.*V)/0/5
NOD,ADD,3,(152.25*V)/(-120.*V)/0/5
NOD,ADD,4,(184.5*V)/0/0/5
NOD,ADD,5,(329.5*V)/0/0/5
NOD,ADD,6,(-120.*V)/(-240.*V)/0/5
NOD,ADD,7,(-152.25*V)/(-120.*V)/0/5
NOD,ADD,8,(-184.5*V)/0/0/5
NOD/ROT/1T8,9T16,C0/0/0/1/0/0/(ALFAR)
```

- the SHELL elements (finite elements) are built with NKTP = 20, NORDR = 14, by connecting the proper nodes;
- the elements are rotated around OX axis of Global Coordinate System for building the semi cylindrical shape of one archway;

```
ELE,ADD,2,20/1/1/1,2/3/11/10
ELE,ADD,3,20/1/1/1,3/4/12/11
ELE,ADD,4,20/1/1/2,4/5/13/12
ELE,ADD,5,20/1/1/1,6/1/9/14
ELE,ADD,6,20/1/1/1,7/6/14/15
ELE,ADD,7,20/1/1/1,8/7/15/16
ELE,ROT,ALL, , ,X0./0./(ALFAR),(N)
ELE,TRS,ALL, , ,0.514/0./0.,(NA-1)
```

- more new archways are generated by translating along OX axis the old archway, till we obtain the final number of archways;
- the TRUSS elements are introduced at the first three archways from each extremity of the structure for equilibrate the stresses on entire building;

```
let xcmax=(184.5+145.+5.)*v+514.*v*(na-1)
let xcmin=184.5*v+(na-1)*514.*v
let ycmin=r+200.*v
let ycmax=- ycmin
let zcmin=-5.
let zcmax=r+1.
ele,del,cp/0/(xcmin)/(xcmax)/(ycmin)/(ycmax)/(zc
min)/(zcmax)
nod,del,cp/0/(xcmin)/(xcmax)/(ycmin)/(ycmax)/(zc
min)/(zcmax)
nod,trs,4/5/12/13, ,-0.514/0./0. ,1
let yrmin=r*cos(alfar)-0.004
let yrmax=r+0.004
let zrmin= -0.004
let zrmax=r*sin(alfar)+0.004
ele,add,2000,20/1/1/3.8/16/1539/1537
ele,rot,2000, , ,x0./0./ (alfar),(n)
let ztmax=r+1.
let trx=na*0.514
ele,trs,2000t2011, , ,(trx)/0./0.,1
NOD,MER,ALL,1.0E-06
ELE,ADD,3000,14/1/2/4,12/16
ELE,ROT,3000, , ,X0./0./(ALFAR),(N-1)
LET TX=0.514
ELE,TRS,3000T3010, , ,(TX)/0./0.,2
LET TX1=0.514*(NA-3)
ELE,TRS,3000T3032, , ,(TX1)/0./0.,1
ELE,COL,3000T3200,GREEN
LET N1=NA-1
LET XN=350.*V+(N1)*600*V
LET YN=R+500.*V
LET ZN=2.*V
DIS,ADD,CP/0/(-XN)/(XN) /(-YN)/(YN)/(-
ZN)/(ZN),1,0/0/0/0/0/0
```

- the properties of the material are introduced for the SHELL elements and for the TRUSS elements:

```
MAT,ADD,1,EX,2.1E11
MAT,ADD,1,NUXY,.3
MAT,ADD,1,ALPX,1.2E-05
MAT,ADD,1,DENS,(DENSF)
```

```
MAT,ADD,2,EX,2.1E11
MAT,ADD,2,NUXY,.3
MAT,ADD,2,ALPX,1.2E-05
MAT,ADD,2,DENS,(DENSR)
let g1=2*g
let g2=4*g
LET A=0.0002
PROP, ADD,1,(g)/(G)/(G)/(G)
PROP, ADD,2,(g1)/(G1)/(g1)/(g1)
PROP, ADD,3,(g2)/(g2)/(g2)/(g2)
PROP, ADD,4,(A)/(A)
NOD,MER,ALL,1.0E-06
```

- the wind loads are different along the one archway so it is necessary to introduce them as follows:

```
LET INA=0
REPEAT NA
LET INM=0
REPEAT NM
LET INM=INM+1
LET NES=(1+(INM-1)*7)+7*NM*INA
LET NEF=(7+(INM-1)*7)+7*NM*INA
IF (INM.EQ.1)THEN
LET RKINM1=1
LET RKINM2=0.9
ELSE
IF(INM.EQ.2)THEN
LET RKINM1=0.9
LET RKINM2=0.5
ELSE
IF(INM.EQ.3)THEN
LET RKINM1=0.5
LET RKINM2=-0.1
ELSE
IF(INM.EQ.4)THEN
LET RKINM1=-0.1
LET RKINM2=-0.7
ELSE
IF(INM.EQ.5)THEN
LET RKINM1=-0.7
LET RKINM2=-1.1
ELSE
IF(INM.EQ.6)THEN
LET RKINM1=-1.1
LET RKINM2=-1.2
ELSE
IF(INM.EQ.7)THEN
LET RKINM1=-1.2
LET RKINM2=-1.
ELSE
IF(INM.EQ.8)THEN
LET RKINM1=-1.
LET RKINM2=-0.6
ELSE
IF(INM.EQ.9)THEN
LET RKINM1=-0.6
LET RKINM2=-0.2
ELSE
IF(INM.EQ.10)THEN
LET RKINM1=-0.2
LET RKINM2=.1
ELSE
IF(INM.EQ.11)THEN
LET RKINM1=.1
LET RKINM2=.3
ELSE
IF(INM.EQ.12)THEN
LET RKINM1=.3
LET RKINM2=.4
ENDIF
ENDIF
ENDIF
ENDIF
ENDIF
ENDIF
ENDIF
ENDIF
ENDIF
ENDIF
ENDIF
ENDIF
```

- the load system is added according to the international standards;

```
LET P1=P*RKINM1*(-1)
LET P2=P*RKINM2*(-1)
IF(P1.LT.0)THEN
LET P3=P1+440.
ELSE
LET P3=P1-440.
ENDIF
IF(P2.LT.0)THEN
LET P4=P2+440.
ELSE
LET P4=P2-440.
ENDIF
PRE,ADD,(NES)T(NEF),1,1/(P3) /(P3) /(P4) /(P4)
CONTINUE
LET INA=INA+1
CONTINUE
LET PV2=1357.2*(R)*(R)/12
LET PV3=(-1017.9/12.)*(R)*(R)
LET YMIN=-R-0.002
LET YMAX=R+0.002
LET ZMIN= -1
LET ZMAX=R+1.
FOR,ADD,CP/0/-0.330/-
0.328/(YMIN)/(YMAX)/(ZMIN)/(ZMAX),3,(PV2)
```

```
/0.0/0.
LET XMIN=0.514*N1+0.329
LET XMAX=0.514*N1+0.33
FOR,ADD,CP/0/(XMIN)/(XMAX)/(YMIN)/(YMA
X)/(ZMIN)/(ZMAX),4,(PV3)/0./0.
LET X1=-0.350
LET X2=.514*N1+0.33
LET Y1=R*COS(ALFAR)-0.002
LET Y2=R*COS(ALFAR)+0.002
LET Z1=R*SIN(ALFAR)-0.002
LET Z2=R*SIN(ALFAR)+0.002
LET Y3=(-R)*COS(ALFAR)-0.002
LET Y4=(-R)*COS(ALFAR)+0.002
LET V1=(-1000.)*0.514/2.
LET V2=(-V1)
FOR,ADD,CP/0/(X1)/(X2)/(Y1)/(Y2)/(Z1)/(Z2),5,0
./(V1)/0.
FOR,ADD,CP/0/(X1)/(X2)/(Y3)/(Y4)/(Z1)/(Z2),6,0
./(V2)/0.
FOR,ADD,99/1594/650,5,0./0./-1000.
ENDMACRO
```

5 The results of static and buckling analysis

The purpose of this analysis is to estimate the maximum load that a structure can support before becoming unstable in terms of elastic behavior or before it crushes off [7].
The equation is given by:

$$Ku = -\lambda K_G u \quad (7)$$

$$(K + \lambda_i K_G)u_i = 0, \quad (8)$$

where:
K – matrix of linear loading;
K_G - matrix of initial loading;
λ_i – the "i"-th value of the multiplication coefficient for computed K_G .
u_i - the "i"-th value of the displacement.
This analysis is named bifurcate stability because its performance is in two steps.
The first step is the static analysis which computes the efforts for one set loading.
The second step is the valuable analysis which first studies the geometrical stiffness matrix (based upon the efforts founded in static situation).
The buckling factors obtained from the stability analysis can be negative, showing that the maximum stability loadings are obtained when the reference loadings are reversed in that direction and are rising up by the buckling factors [8].
The most of the practical stability problems can be analyzed like non-linear problems.

6 The results of static and buckling analysis

The static analysis for the model computes the equivalent stresses, the displacements and the reaction in the rigid nodes from the basis of the structure by using the NISA-program [3]. The output data of the equivalent stresses are relevant. There were compared their values with the admissible stress of our material. There were needed the displacements in the nodes of the frontal archways with the frontal sides and the other sides.
There were taken into account 8 archways, a convenient number for the verifying calculus of the building. Then, the eight archways structure was loaded by the loads system previously presented. The results are shown in the Table 2.
As it can be noticed, there was obtained a non-uniform and unsuitable distribution of the stresses: there were obtained big stresses in the frontal sides of the structure and the middle side remained unloaded. It can be also noticed that the structure was relieved of the wind action pressures, in order to see if the results were different in structure load [8].
This solution is inadequate from the economic point of view, because only 10% of material is fully loaded.
This is the reason why it was necessary to model the structure so the maximum stresses from the frontal sides were distributed on 3-4 archways inside the structure.
The adopted solution was the loading of the structure frontal 3 archways with stiffener bars type stretch-compression, having two nodes binding each two elements with maximum radius, 11 bars for each archway.

Table 2 Von Misses stress distribution with/without wind action

Module's width [mm]	Diameter [m]	Stress Distributions Von Misses [N/mm^2] (including wind action)	Stress distributions Von Misses [N/mm^2] (without wind action)
0.5	4	146	218
	8	539	342
	12	-	-
0.8	4	73.3	99
	8	282	320
	12	-	-
	16	-	-

	20	-	-
1	8	240	204
	12	-	-
	16	-	-
1.5	16	245	248
	20	-	-
	24	-	-
	28	-	-
2	12	102	162
	16	103	175
	20	237	302
	24	-	-
2.5	16	122	132
	20	152	163
	24	196	205
	28	228	240

The loading of the structure with bars resulted from the following reasons: there were obtained big stresses in the frontal sides of the structure and the middle side remained unloaded.
So, there was obtained a uniform distribution of stresses. The old solution became inadequate for economical reasons.
In consequence, the structure was loaded with bars and the stresses became distributed on 2-3 archways inside the structure.
For computing the output data of our analysis, there was also computed the buckling factor [4] in all the analyzed cases - table 3.

Table 3 The buckling factor

The module's width [mm]	Diameter [m]	Stress distributions Von Mises [N/mm^2]	Buckling factor
0.5	4	44.5	14.39
	8	148	3.99
	12	404	-
0.8	4	23.5	13.8
	8	102	7.85
	12	220	2.38
	16	-	-
	20	-	-
1	8	97	7.2
	12	162	2.82
	16	305	-
1.5	16	199	2.43
	20	314	-
	24	-	-
	28	-	-
2	12	108.5	5.75
	16	168	3.2
	20	202	2.59
	24	-	-
2.5	16	124	3.1
	20	167	2.4
	24	176	1.6
	28	195	-

7 Conclusion

The self-supported metallic structure is realized by combining one type module element technologically optimized. The building activity consists in a very easy assembly of the elements.
The vault is self-supporting and is no need to use sustained pillars supports or columns. The foundation is isolated to vibration [1].
Another big advantage of this kind of shape is given by using the interior space in 95-100% rate. The easy assembly way gives the possibility to disassembly and assembly again in another location. The extending in length is easy and economic. The fire protection is assured by the metallic material. This kind of building needs a low cost of manufacturing comparatively with a similar classical building. There is no necessary specially maintenance. This kind of building has a nice design and its life guarantee is 30 years.

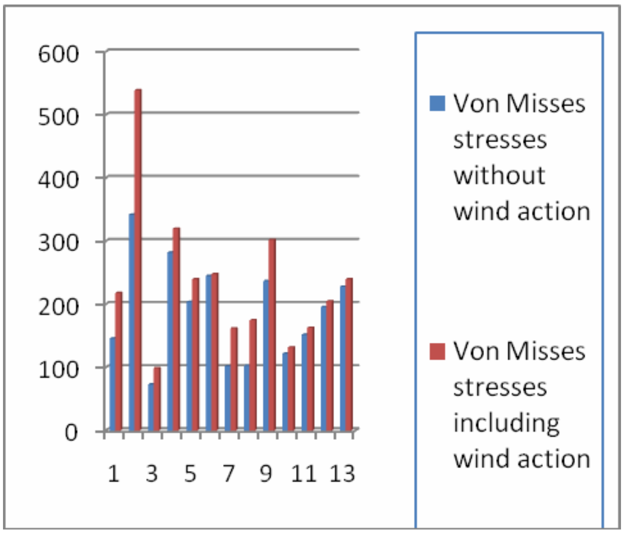

Fig. 6 The comparison between the values of the Von Misses stresses with or without wind action

The comparison of the values of the Von Misses stresses without wind action and including wind action reveals the fact that only for several cases of the pair iron sheet thickness – diameter of metallic structure the differences are significantly major – figure 6.

The MACRO program realizes the possibility of rapid calculus for stress analysis and for the determination of the buckling factor. All the dimensions and design parameters can be easy modified in the program.
There was noticed from the obtained results that the structure was modified in the way given by the equivalent stresses values.
The possibly pairs for the iron sheet thickness – diameter of metallic structure considered in the study are presented in table 4. This pairs are on Ox axis in the figure 6 and figure 7.
Figure 7 presents the buckling factor for every pair of the module's width (iron sheet thickness) – diameter of the metallic structure.

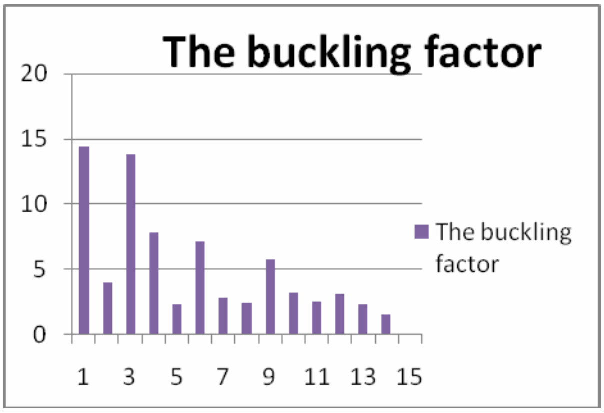

Fig. 7 The buckling factor values for different values of the pairs iron sheet thickness – diameter of the metallic structure

So, the best stability of the structure is achieved when the diameter of the building is small and the thickness is large.

Table 4 The possibilities of pairs iron sheet thickness – diameter of metallic structure

No	Module's width[mm]	Diameter [m]
1	0.5	4
2	0.5	8
3	0.8	4
4	0.8	8
5	0.8	12
6	1	8
7	1	12
8	1.5	16
9	2	12
10	2	16
11	2	20
12	2.5	16
13	2.5	20
14	2.5	24

There was adopted a solution that realized a uniform value of stress distribution for the whole structure.

References:
[1] Ashwani Jain, D. K. Soni - *Foundation Vibration Isolation Methods,* 3rd WSEAS Int. Conference on APPLIED and THEORETICAL MECHANICS, 2007, p.163
[2] Călbureanu, M., Malciu, R., - *The Stress And Buckling Analysis Of A Self-Supporting Metallic Structure Using Shell And Truss Elements* - The 10-th International Symposium on Experimental and stress analysis and Material Testing - ARTENS SIBIU, 2004, vol. II, p. 5-7 - 5-10
[3] Costa, V, Sousa, A, - *A Control Volume-Based FEM for the Solution of the Three-Dimensional, Elastic Stress-Strain Equations,* WSEAS Proceedings 2nd World MCME (Mathematics & Computers in Mechanical Engineering, *MCME 2000*) VOULIAGMENI, GREECE, July 9-16, 2000, pp. 5551-5556
[4] Engineering Mechanics Research Corporation *NISA Users Manual*, Troy, Michigan, 2000;
[5] Ian M. Smith, I.M., Griffiths D.V., *Programming the Finite Element Method,* 4 edition, Wiley, 2004
[6] Maksimovic, S., Zeljkovic, V., Ugricic, M., - *Multilevel Optimization Approach Applied to Structural Design Including Material Consolidation* - Proceedings of the 2nd WSEAS Int. Conference on Applied and Theoretical Mechanics, Venice, Italy, November 20-22, 2006, pp. 13
[7] Teng, J.G., Rotter, J.M., - *Buckling of Thin Metal Structures*. E & FN Spon, London (in Press), 2002.
[8] Xinwu Wang - *Experimental Research and Finite Element Analysis on Behavior of Steel Frame with Semi-Rigid Connections,* 5th WSEAS Int. Conf. on ENVIRONMENT, ECOSYSTEMS and DEVELOPMENT, 2007,p. 80
[9] Zienkiewicz, O.C., Taylor, R.L. - *The Finite Element Method for Solid and Structural Mechanics,* sixth edition, Jordan Hill, 2005