Application: AEF-A.12
Torsional springs

KEY WORDS
Linear Static Analysis, Linear Material, 1D Geometric Model, 1D Finite Element, Linear Finite Element, CATIA Geometric Modeling, Classical Method Verification, Machine Element

CONTENT

A. PROBLEM DESCRIPTION

A.1 Introduction

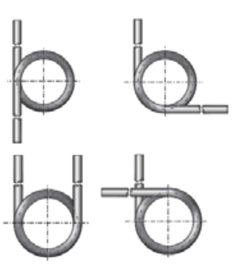
There are mechanical components in many technical products that have distinct compact structures required by the main function to be performed. Representative of this group of components are the elastic elements (springs), the damping elements, the supporting elements (housings), etc. The specificity of these elements, as a rule, is given by their fixed or quasi-fixed connections with the neighboring parts. The finite element analysis of these components, in order to obtain precise results, presupposes the accurate definition of the solid model, of the restrictions imposed by the connections with the neighboring elements, as well as of the loads.

A.2 Application description

Springs are machine elements that, due to the shape and elastic properties of the materials, store the mechanical work of external forces, at deformation, and return it, almost in whole or in part, in the period of return to the original shape.

These cylindrical coil springs are made of round diameter round wire. The introduction of force or torque occurs through the arm at the beginning and end of each spring.

These springs have a linear torque characteristic and can be made by cold or hot forming. There may be various constructive forms, shown in the figure below. They can be used in various applications, some of which are shown in the images below.




A.3 Application goal

In the case of this application, the analysis of the fields of displacements, deformations and tensions of an elastic element of curved bar type from the composition of the devices presented above is presented. The values of the geometric parameters of the spring were taken from the specialized literature, as follows: d = 2.5 mm - the diameter of the coil; D_m = 50 mm - average diameter; n = 8.5 turns; Δ = 0.5 mm distance between turns.

B. THE FEA MODEL

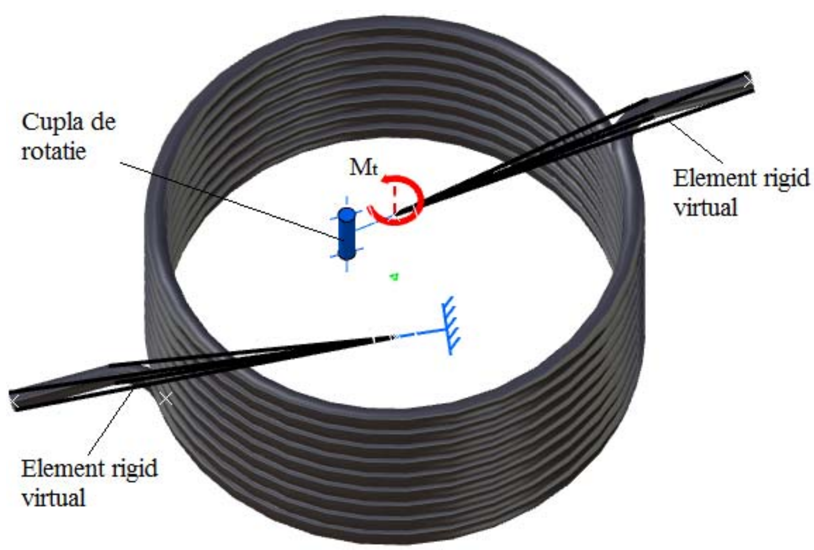
B.1 The model definition

In order to draw up the finite element analysis model associated with the above application, it is necessary to identify:

- geometric shape and dimensions,
- restrictions induced by links with adjacent elements,
- external and internal loads (own weight),
- material characteristics.

B.2 The analysis model description

The geometric shape and dimensions of the helical spring are shown in the figure below.



B.3 Characteristics of the material

For finite element analysis, the strength characteristics of the 50VCr11A spring steel material treated at 50-55 HRC are:

- modulus of longitudinal elasticity, E = 209,000 N / mm2;
- transverse contraction coefficient (Poisson), ν = 0.3.

C. PREPROCESSING OF FEA MODEL

C.1 Creating and saving the project

Creating of the project

, Toolbox: ↵⊟ Analysis Systems → ↵↵ Static Structural (the subproject window appears automatically); → [the name can be changed Static Structural în / in *Torsional spring*].

Problem type setting (3D)

A: ↳ Geometry → ↵ Properties → Properties of Schematic A3: Geometry, ⊟ Advanced Geometry Options: ↵ Analysis Type, [select from the list ↵, ↵3D] → [close the window, ↵ ✕].

Saving of the project

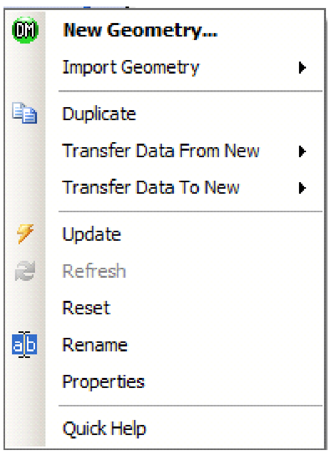
↵ Save As… → Save As, File name: [input name, *Torsional Spring*] → ↵ Save

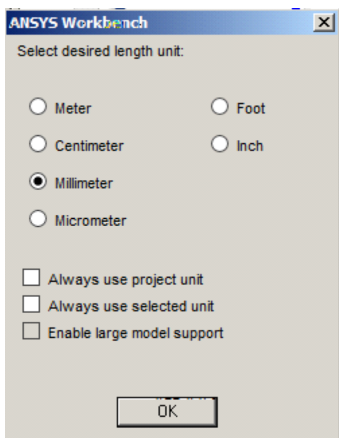
C.2 Geometric modelling
C.2.1 Importing the geometric model of the spring

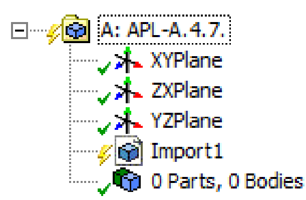
This application will aim to use a geometric model made in another drawing / design environment. The model is made in advance in CATIA v5R21 in the form of a 1D body, with the geometric construction data presented in the Model section for analysis. The file, originally saved in the specific format of the CATIA software (.catpart) will be saved under the extension of a universal transfer format (.igs or .stp).

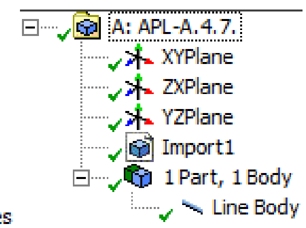
Toolbox ↳ Geometry → Import Geometry → Browse → (navigate to the directory structure of the HDD and identify the file T*orsiunal Spring-1D.igs*) → ↵ (OK);

Toolbox ↳ ↳ Geometry → **ANSYS Workbench**: Select desired length unit: Millimeter → ↵ (OK) → DM → Generate.









C.2.2 Creating the geometric model in CATIA

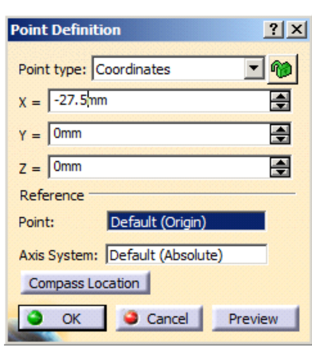
Activating the shape generation module and setting the unit of measure for lengths

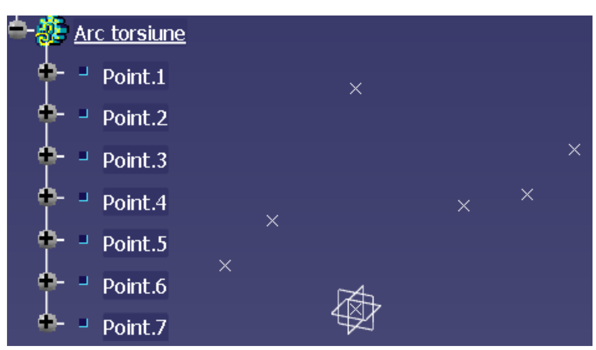
CATIA → Start → Shape → Generative shape design → **New part**: New part name: *Spring*.

Tools → Options…→ **Options**: Parameters and Measure; Units; Length, Milimeter (mm); ↵ OK.

Generating of reference points

(Point) → **Point Definition**: X -27.50mm, Y 0mm, Z 0mm; ↵OK. [similarly, the coordinates of some auxiliary points are introduced - which will help to achieve the geometry of the arc P2(-32.5,0,0), P3(-44.5,5,0), P4 (32.5, 5, 25.5), P5(44.5, 5, 25.5) and the points P6(0, 0, 0) şi P7(0, 0 , 25.5) used to create virtual rigid elements].



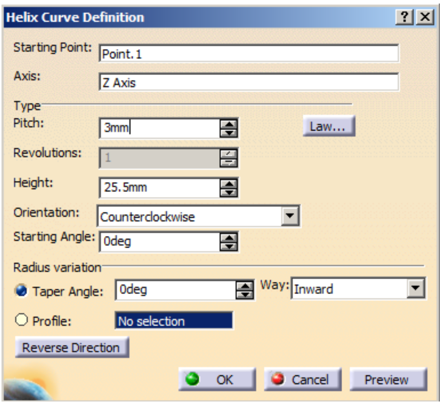


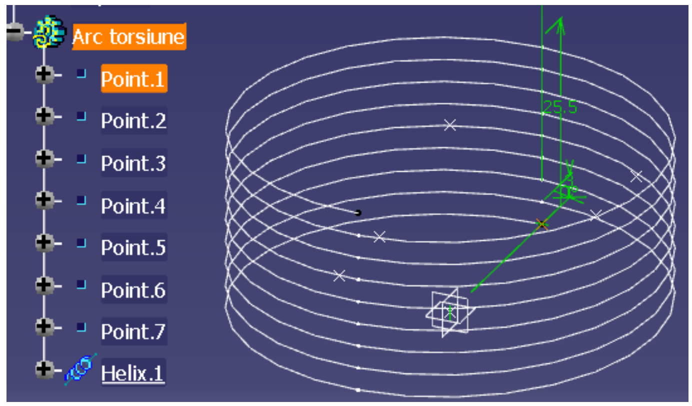
Helical spring generation


(Helix) → **Helix Curve Definition** select with the mouse in the graphic area or in the tree structure the point Point.1 for Starting Point and for Axis, with the help of a right click on the selection box choose the OZ axis, then fill in the step values: 3 mm and the height of the spring = 8.5 steps x 3 mm = 25.5 mm], ↵ OK.

Obs.

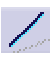
The Helix command can be found in the Wireframe menu.

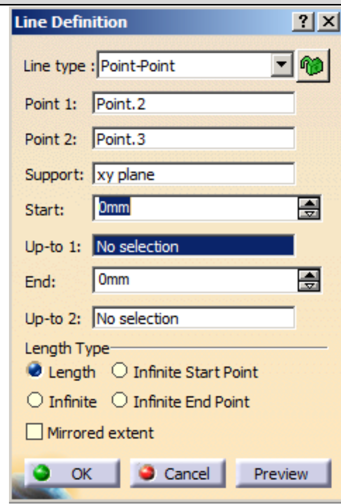


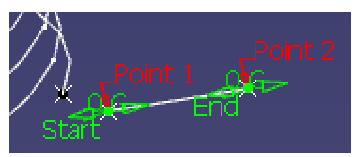


C.2.3 Generating of clamping areas at the ends of the spring

Generating of clamping areas at the ends of the spring

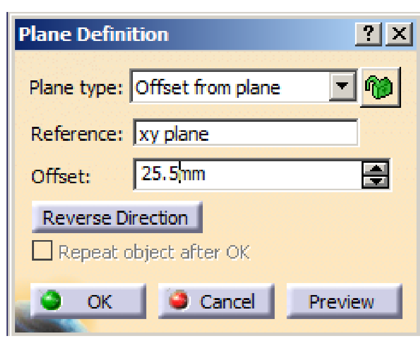

(Line) → [se construieşte un segment de dreaptă din extremitatea arcului de cerc realizat anterior în punctul P3 / a line segment is constructed from the end of the circle arc previously made at point P3] → **Line Definition** → Line type: Point to point, Point 1: Point.2, Point 2: Point.3, Support: xy plane ↵ OK





This command is repeated for the other end of the spring, using points P4 and P5, as follows::


(Plane) → **Plane Definition** → [A plane parallel to XOY is constructed, one at a distance (Offset) of 25.5 mm from XOY] ↵ OK.

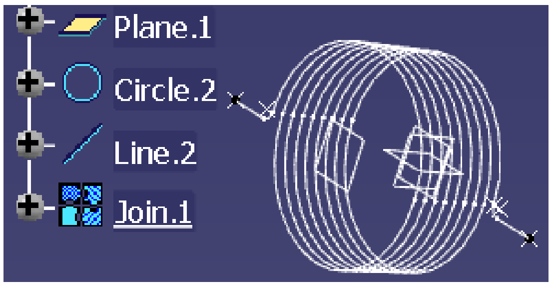


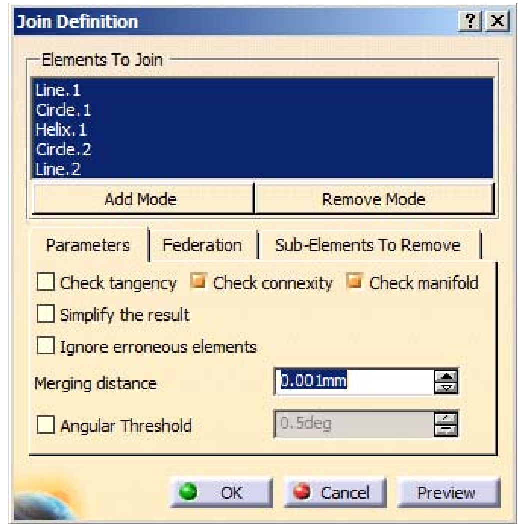
(Line) → [a right segment is constructed from the end of the circle arc previously made at point P5] → **Line Definition** → Line type: Point to point, Point 1: Point.4, Point 2: Point.5, Support: Plane.1 ↵ OK

(Corner) → [the 5 mm pitch helix is connected to the P1P2 and P4P5 segments with a radius of 5 mm] → **Corner Definition** → Circle type: Center and point, Center: Point.2, Point: Point.1, Support: xy plane, Start: 0 deg, End: 90deg ↵ OK

Joining the five segments of the arc

(Join) → **Join Definition** [select Helix.1, Circle.1, Line.1, Circle.2, Line.2] ↵ OK.





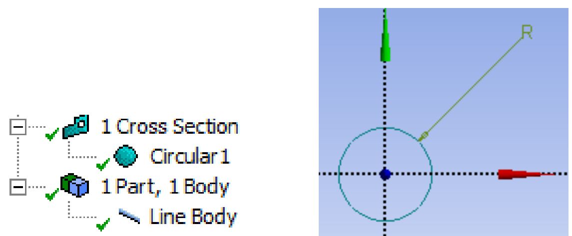
Generating the spring section

ANSYS Workbench → DM → Modeling →

Concept → Cross Section → Circular →

Details View → **Dimensions:1** → R1 = 1,25 mm →

Generate.

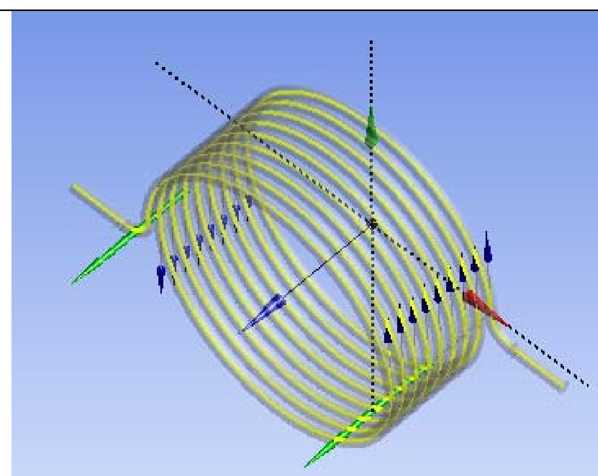


Assigning a cross-sectional profile to the spring

Generating a profile does not mean assigning it to a 1D part. To complete the procedure, proceed as follows

DM → ↵ Line Body → Details View → **Details of Line Body** → Cross Section: Circular 1;

Modeling → View → Cross Section Solids [the 3D viewing option is activated] → Generate .



Building a point in the 3D space of the spring

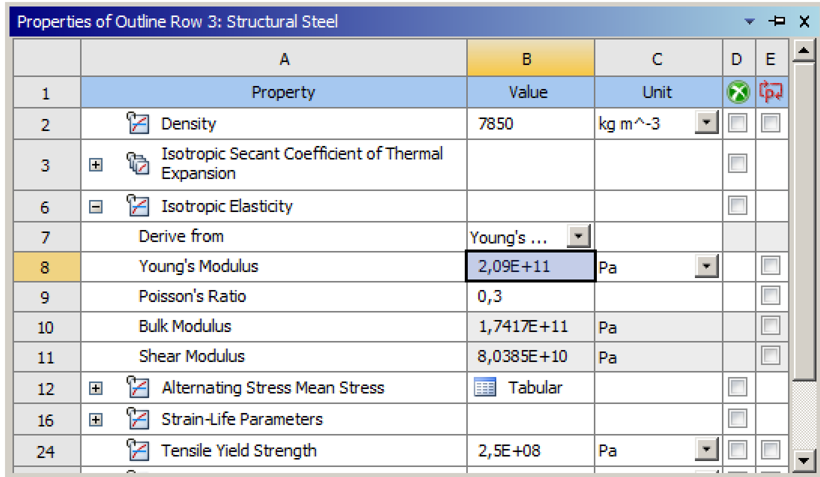
DM → Create → Point → Details View → **Details of Point 1** → Definition: Manual Input; **Point Group 1 (RMB)** → X Coordinate = 0, Y Coordinate = 0, Z Coordinate = 0 → Generate.

This point will be used to create a virtual rigid element for the introduction of forces.

C.3 Modelling of material characteristics

Changing the value of the modulus of elasticity

, Project Schematic: ↳ Engineering Data ✓ → ↵ Edit... → Outline of Schematic A2: Engineering Data: ↵ Structural Steel , Properties of Outline Row 3: Structural Steel: ⊟ Isotropic Elasticity → Young's Modulus , Young's Modulus , [select from column C (Unit) cu ↵▼, ↵MPa], [input in column B (Unit) valoarea / value, 209000] → ↵ Update Project → ↵ Return to Project (the other parameters remain the default).



Properties of Outline Row 3: Structural Steel

	A	B	C	D	E
1	Property	Value	Unit		
2	Density	7850	kg m^-3		
3	⊞ Isotropic Secant Coefficient of Thermal Expansion				
6	⊟ Isotropic Elasticity				
7	Derive from	Young's ...			
8	Young's Modulus	2,09E+11	Pa		
9	Poisson's Ratio	0,3			
10	Bulk Modulus	1,7417E+11	Pa		
11	Shear Modulus	8,0385E+10	Pa		
12	⊞ Alternating Stress Mean Stress	Tabular			
16	⊞ Strain-Life Parameters				
24	Tensile Yield Strength	2,5E+08	Pa		

C.4 Finite element modelling

C.4.1 Launching the finite element modeling module and set the material characteristics and problem type

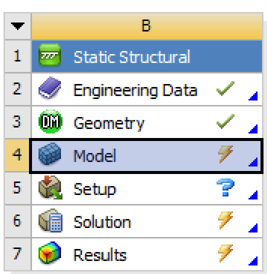
Launching of the finite element modeling module

, Project Schematic: ↳ Model → ↵ Edit... → [launching module *Mechanical [ANSYS Multiphysics*].

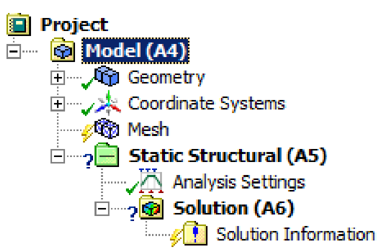
Setting the unit of measure system

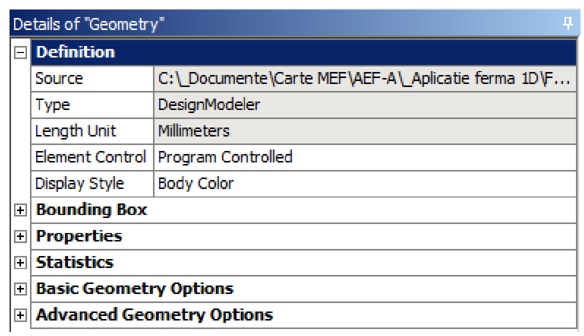
: ↵ Units → ↵ Metric (mm, kg, N, s, mV, mA) (the system of units of measurement is usually set by default).

↵ Geometry → Details of "Geometry" → **Definition** → Element Control ▼: Program Controlled .



AEF-A.4.6.





Details of "Geometry"

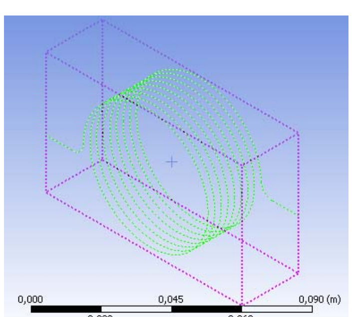
⊟ **Definition**	
Source	C:_Documente\Carte MEF\AEF-A_Aplicatie ferma 1D\F...
Type	DesignModeler
Length Unit	Millimeters
Element Control	Program Controlled
Display Style	Body Color
⊞ **Bounding Box**	
⊞ **Properties**	
⊞ **Statistics**	
⊞ **Basic Geometry Options**	
⊞ **Advanced Geometry Options**	

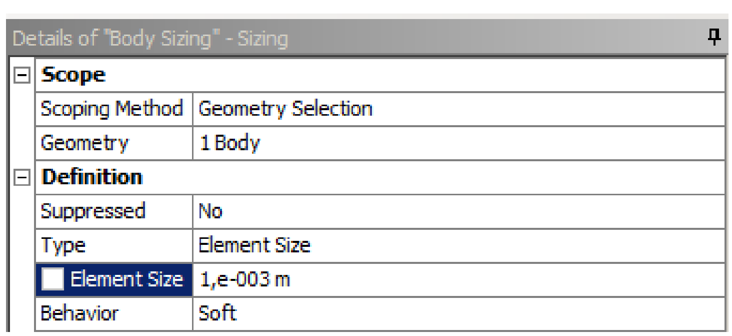
Setting the material characteristics

Outline: ↵⊞ Geometry → ↵Line Body → Details of "Line Body , **Material** : ↵Assignment → [select from list↵▸, ↵Structural Steel (set by default)].

C.4.2 Model discretization and analysis type setting

, Outline: ↵ Mesh → Mesh Control ▾ → Sizing → Details of "Sizing" - Sizing → **Scope** → Select Geometry: [select with ↵ the geometry of spring using (Body)] Apply; **Definition Element** → Size: 0,001 m → Update

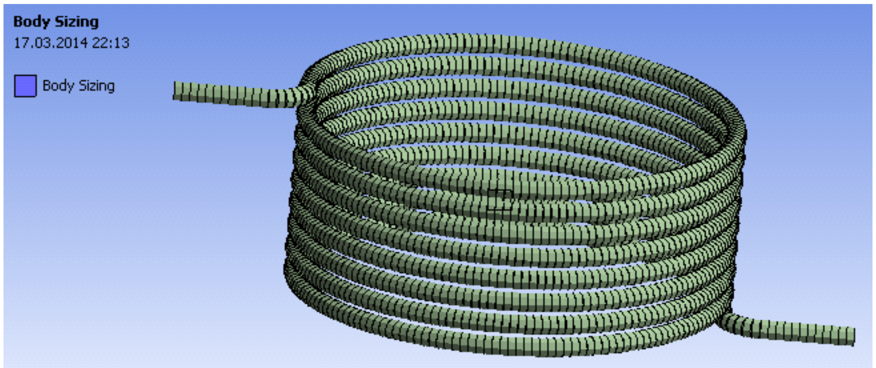




Details of "Body Sizing" - Sizing

⊟ **Scope**	
Scoping Method	Geometry Selection
Geometry	1 Body
⊟ **Definition**	
Suppressed	No
Type	Element Size
Element Size	1,e-003 m
Behavior	Soft

For a proper view of the discretization, this will be done: ↵ Show Mesh.



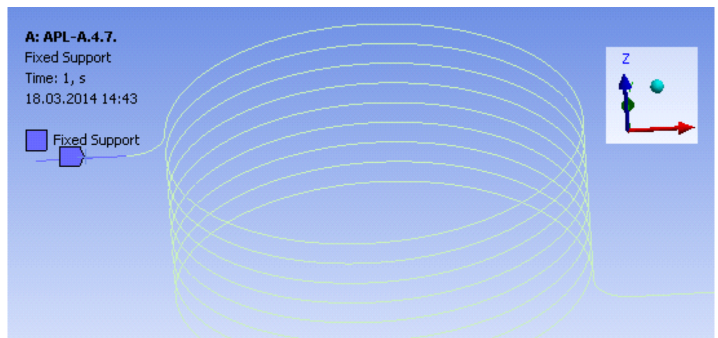
C.5 *Supports and restraints modelling*

Input the gravitational acceleration

Since the weight of the spring is very small (about 58 g), the influence of the weight force (0.56 N) on the analysis results is very small, taking into account the value of the applied torque (which is 1 Nm, which corresponds to a force of of 40 N acting at the helix end point).

Input restraint

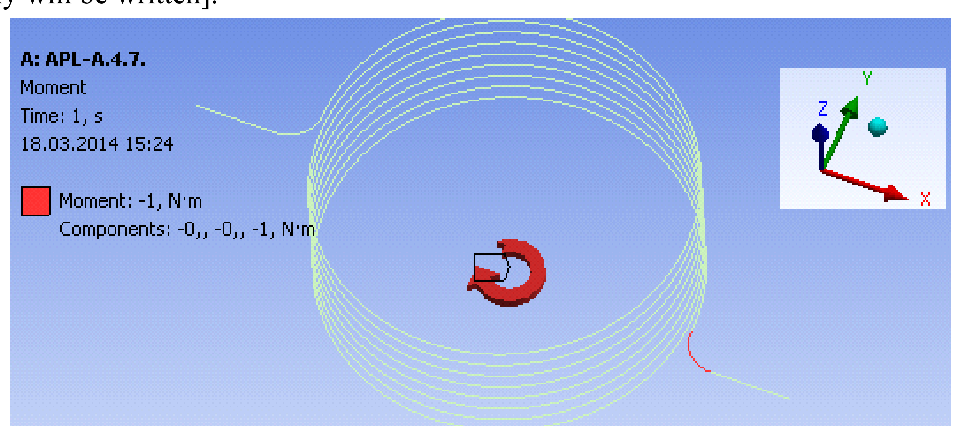
Outline: ↵ Static Structural (A5) → Supports → Fixed Support → Details of „Fixed Suports" → **Scope** → Geometry: [select with ↵ the end segment of the spring at a height of 25.5 mm, using the selection filter (Edge)] → Apply.



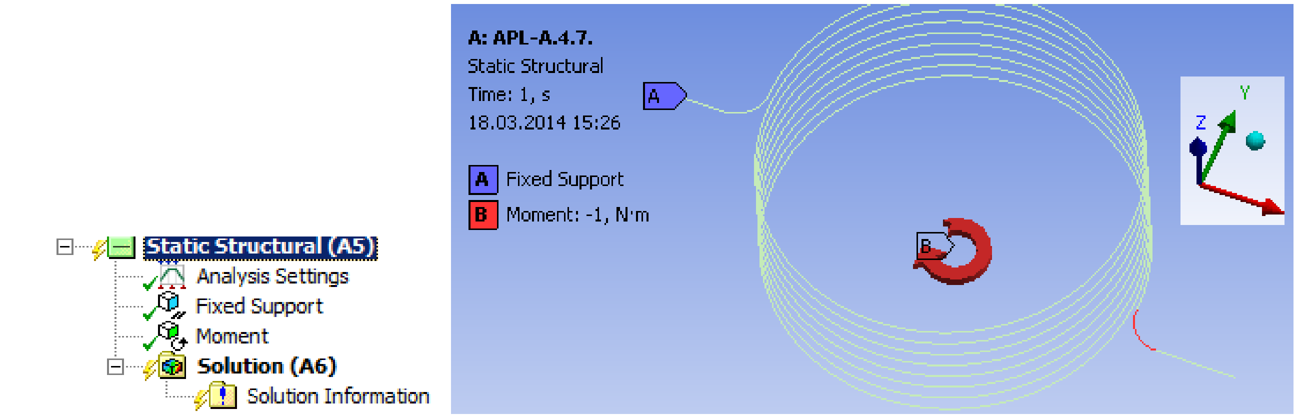
C.6 Load modeling

Input Remote Moment

Outline: ↵ Static Structural (A5) → Loads → Moment → → Details of „Moment" → **Scope** → Geometry: [will be selected with ↵ the arc of the connecting circle between the spring propeller and the terminal segment at dimension 0, using the option (Edge)] → Apply; **Definition** → Magnitude: -1 Nm; Moment → Promote Remote Point → ↵ Moment - Remote Point → Details of „Moment - Remote Point" → **Scope** → X Coordinate = 0, Y Coordinate = 0, Z Coordinate = 0 [the coordinates of the point P1 (0, 0, 0) made previously will be written].



The constraints and loads of the resort will look like the figure below



D. SOLVING THE FEA MODEL

D.1 Setting results

In order to select the final data types to be analyzed after the launch of the calculation module, follow the series of commands presented below.

→ Solution (A6) → Insert → Deformation → Total [use the commands in the open command box with].

The same result can be obtained by using the commands:

, Outline: Solution (A6) → Insert → Deformation → Total. [the buttons in the menu bars are used] and

Solution (A6) → Deformation → Directional.

For this type of structure, the Beam tool can be applied in order to visualize the linearized stresses on the component elements. It is customary, in the process of designing bar structures, to take into account the components of axial stresses that arise from the effect of axial and bending loads in all directions. The following are the other types of results to be analyzed:

Solution (A6) → Tools → Beam Tool.

Solution (A6) → Beam Results → Axial Force.

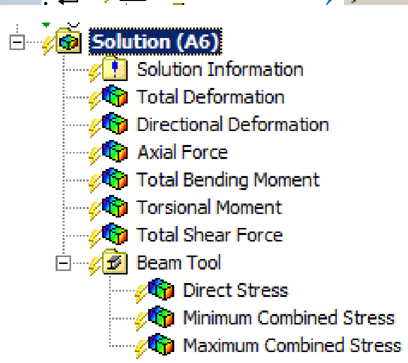
Solution (A6) → Beam Results → Bending Moment.

Solution (A6) → Beam Results → Torsional Moment.

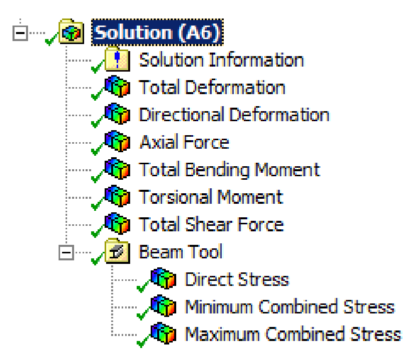
Solution (A6) → Beam Results → Shear Force.

D.2 Lansarea modulului de rezolvare a modelului / Launching the solving module

, Outline: Solution (A6) → Solve.



→ Solve →



E. POST-PROCESSING OF RESULTS

E.1 Viewing the displacement field

For suggestive results, set the view scale of the menu bars:

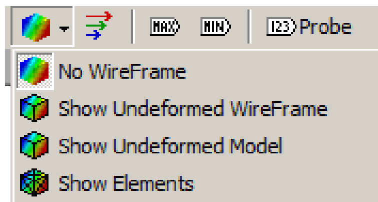
Result 8,6e+002 (Auto Scale) → Result 1.0 (True Scale).

Total deformation viewing

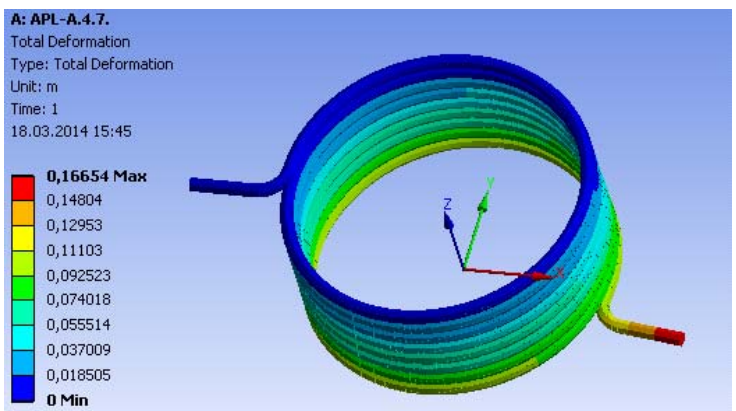
Outline: ↵ Solution (A6) → ↵ Total Deformation → Tab-ul Graph → Animation.

If the images are not suggestive enough, in terms of how the work is distorted, you can return to changing the display scale by selecting a higher value: Result 1,7e+003 (2x Auto).

Various forms of distorted state representation can be used by calling the (Edge) button. Show Showformed WireFrame will be selected, an option that displays the undeformed and warped models in the same representation

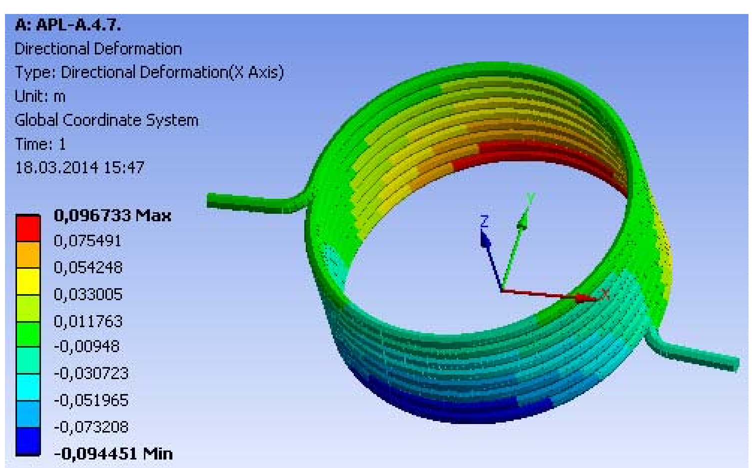


The display characteristics can be changed: the number of frames 10 Frames, as well as the running time of the simulation 2 Sec (Auto). At the same time, the result can be saved as a video file using the *Export Video File* command.



Visualization of the deformation in one direction

Outline: ↵ Solution (A6) → Directional Deformation → Graph → Animation.



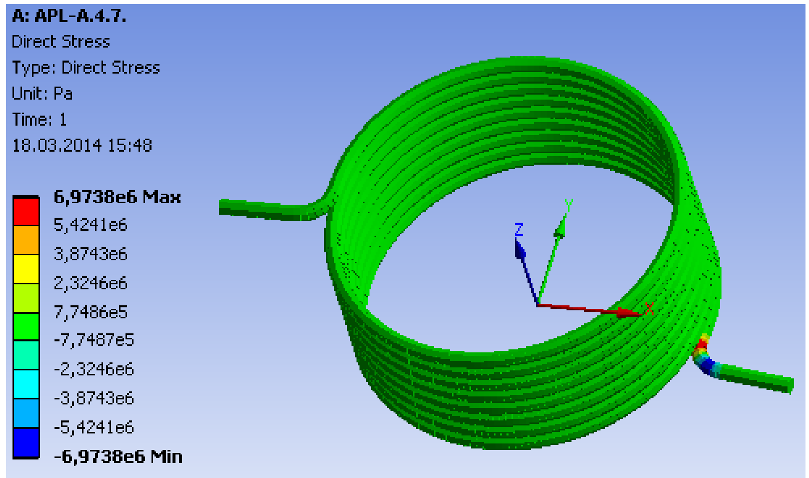
If you want to view it in another direction, follow the steps below:
, Outline: ↵ ? Solution (A6) → Directional Deformation → Details of „Directional Deformation" → **Definition** → Orientation : Y Axis → Solve.

E.2 Visualizing the fields of stresses, forces and moments

Direct stress

Direct Stress (σ_x) represents the component of the internal tension due to the axial force in the spring.

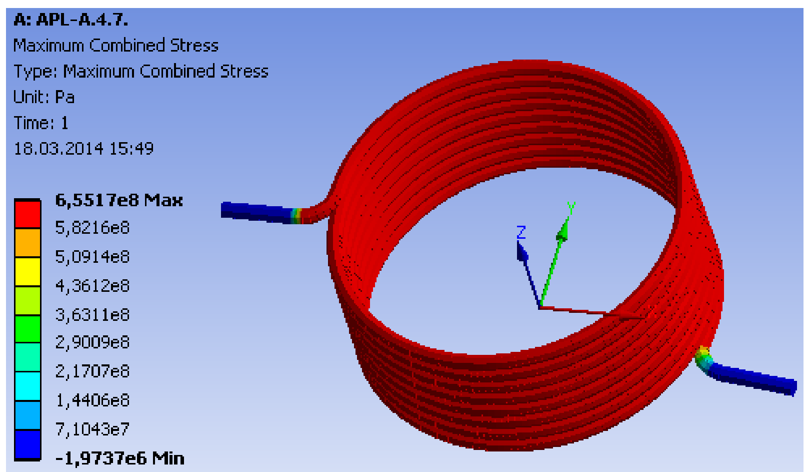
, Outline: ↵ Solution (A6) → Direct Stress → Graph → Animation.



Maximum Combined Stress

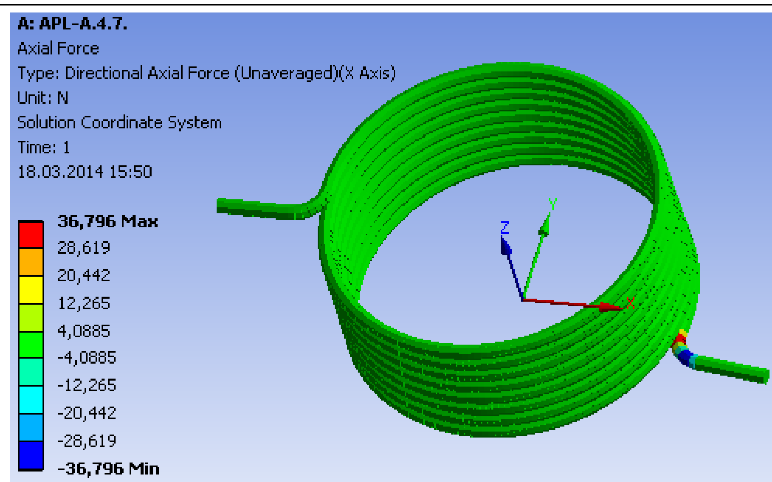
Maximum Combined Stress – represents a linear combination of *Direct Stress* and *Maximum Bending Stress*.

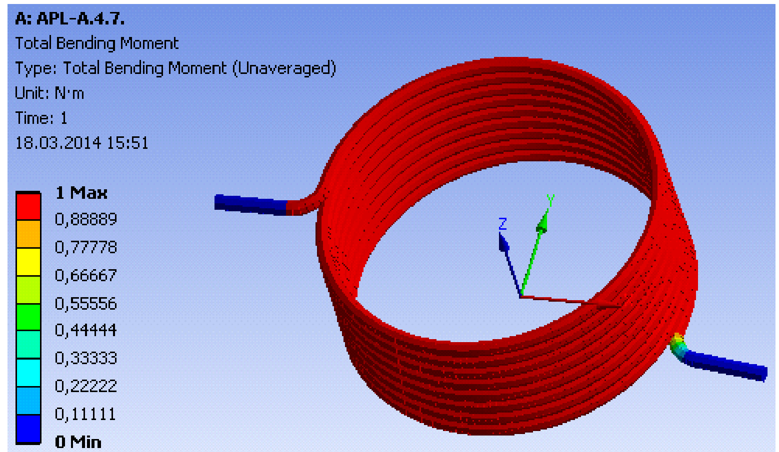
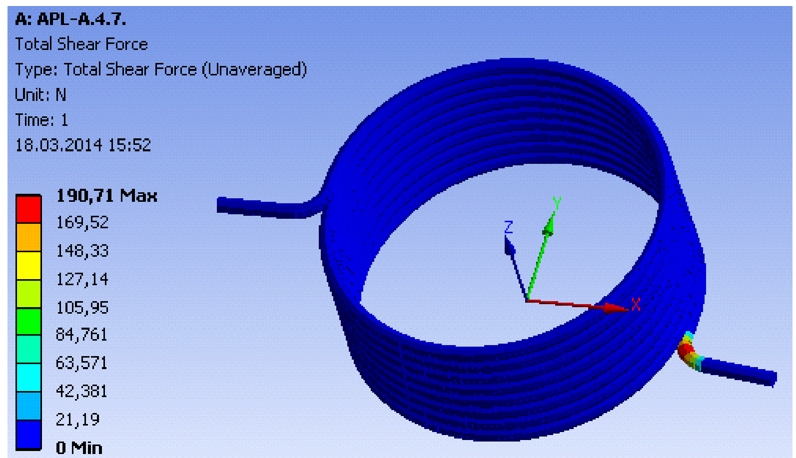
, Outline: ↵ Solution (A6) → Maximum Combined Stress → Graph → Animation.



Directional Axial Force

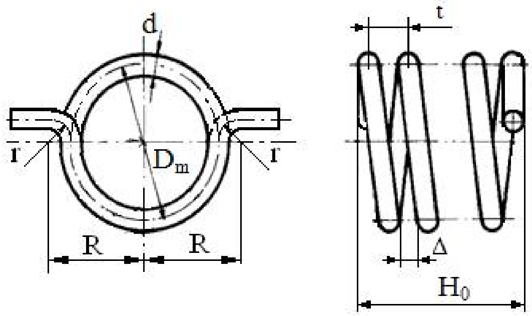
, Outline: ↵ Solution (A6) → Axial Force → Graph → Animation.



Bending Moment Outline: ↵ Solution (A6) → Total Bending Moment → Graph → Animation.	
Shear Force Outline: ↵ Solution (A6) → Total Shear Force → Graph → Animation.	

F. ANALYSIS OF RESULTS

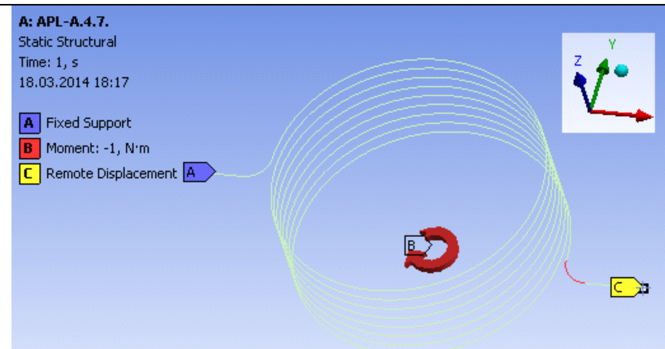
F.1 Interpretation of results
It is observed that, despite the fact that the spring modeling was performed using a 1D body, the results obtained are suggestive, being presented in a 3D environment. From the point of view of the total deformations, it is observed that the maximum value is 166 mm, corresponding to the extremity of the segment in the drive area. It is observed that the areas with high shear and bending efforts are those corresponding to the connection areas between the spring propeller and the right segment. The information regarding the deformations, corroborated with the information regarding the internal stresses, the combined maximum stresses lead to the conclusion that the spring withstands loads without problems, the values of the maximum stresses not exceeding 6.5 x 10^8 Pa, value below the allowed material limit. Particular attention must be paid to the connections at the outlet of the spring propeller at both ends, these two areas being important concentrators of stresses.

F.2 Prezentarea rezultatelor obţinute prin metoda clasică
Known geometric parameters: • d = 2.5 mm - the diameter of the coil; • Dm = 50 mm - average diameter; • n = 8.5 turns; • Δ = 0.5 mm, the clearance between turns. Type of support area (support) and number of turns in this area: symmetrical outer hooks; connection radius, r = 2d; radius of action of the loading force, R = D_m / 2 + r. Based on the constructive data of the spring in the figure above, the displacement and stiffness are calculated for a load M = 1,000 Nmm. The following values are obtained: $\theta_n = \frac{64\, M_{tn}\, D_m\, n}{E\, d^4}$ = 213,14 grd k = $\frac{E\, d^4}{64\, D_m\, n} \frac{180}{\pi}$ = 4,7 Nmm/grd 
F.3 Comparative analysis of results
Using classical methods of Strength of Materials, the results are obtained by relatively simple calculations and can be compared with those obtained with MEF. On the other hand, by classical methods, very few results are obtained: only the angular displacement and the rigidity of the spring.

G. CONCLUZII / CONCLUSIONS

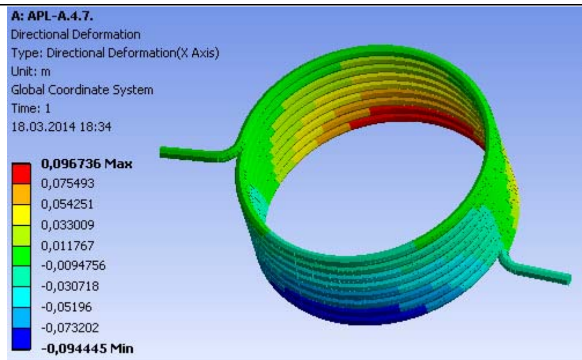
From the point of view of the pre-processing phase, it can be seen that the use of 1D bodies involves minimal resources for both modeling and discretization. Another strong point is that the profile of the spring can be modified / oriented very easily, without influencing the basic shape.
The introduction of supports, constraints and demands is quick and easy. The declaration of materials as well as discretization are controllable processes, which can be done automatically or manually.
Comparing the results obtained by the classical method and FEM, it can be seen that they are comparable, at least in the case of angular displacement, which was calculated classically, the finite element method providing much more data, over time and resource consumption much smaller.
It can be seen that the spring is very strongly stressed in the connection area, at the exit of the propeller towards the extremities. The modification of these areas and the recalculation by FEM is done in a very short time, being an easy procedure. On the other hand, the model for analysis can be changed very easily, and it can change the supports and the demands very easily. In the case of geometries imported from other modeling programs, the geometric model will have to be modified in the original software, which will lead to the resumption of the procedure from the beginning.
For example, the analysis model can be modified by introducing an additional constraint, represented by the obligation for the final segment of the spring in the moment (force) request area to move in a plane. This means that the spring will not be deformed on the Oz axis.

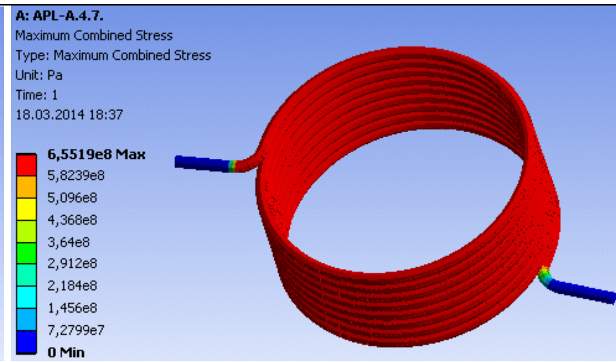
, Outline: ↲ Static Structural (A5) → Supports → Remote Displacement → Details of „Remote Displacements” → **Scope** → Geometry: [will be selected with ↲ the end of the end segment of the spring located at dimension 0 mm, using the option (Vertex)] → Apply → Definition → X Component: Free, Y Component: Free, Z Component: 0, Rotation X: 0, Rotation Y: 0, Rotation Z: Free.



The model for the analysis will look like this:

The results obtained for *Directional Deformation (X Axis)* as well as for *Maximum Combined Stress* are presented below.



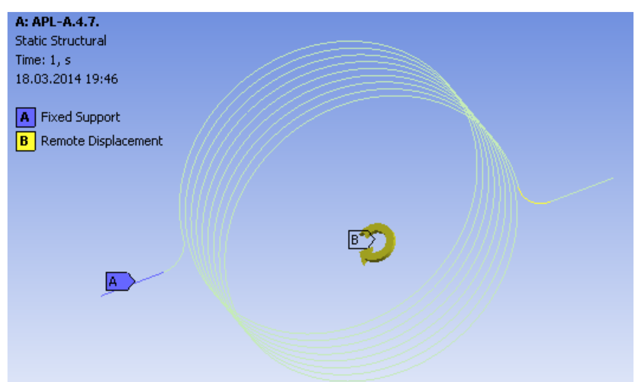


The results are almost identical to those obtained in the previous example. This is due to the fact that, by applying a moment via a Remote Point, the action of this request is required to take place only around the Oz axis, so it will only act in a plane parallel to xOy - equivalent to the newly imposed constraint in the second example.

Another model for analysis can be considered by replacing the moment applied to the spring with an imposed displacement of a certain angular value.

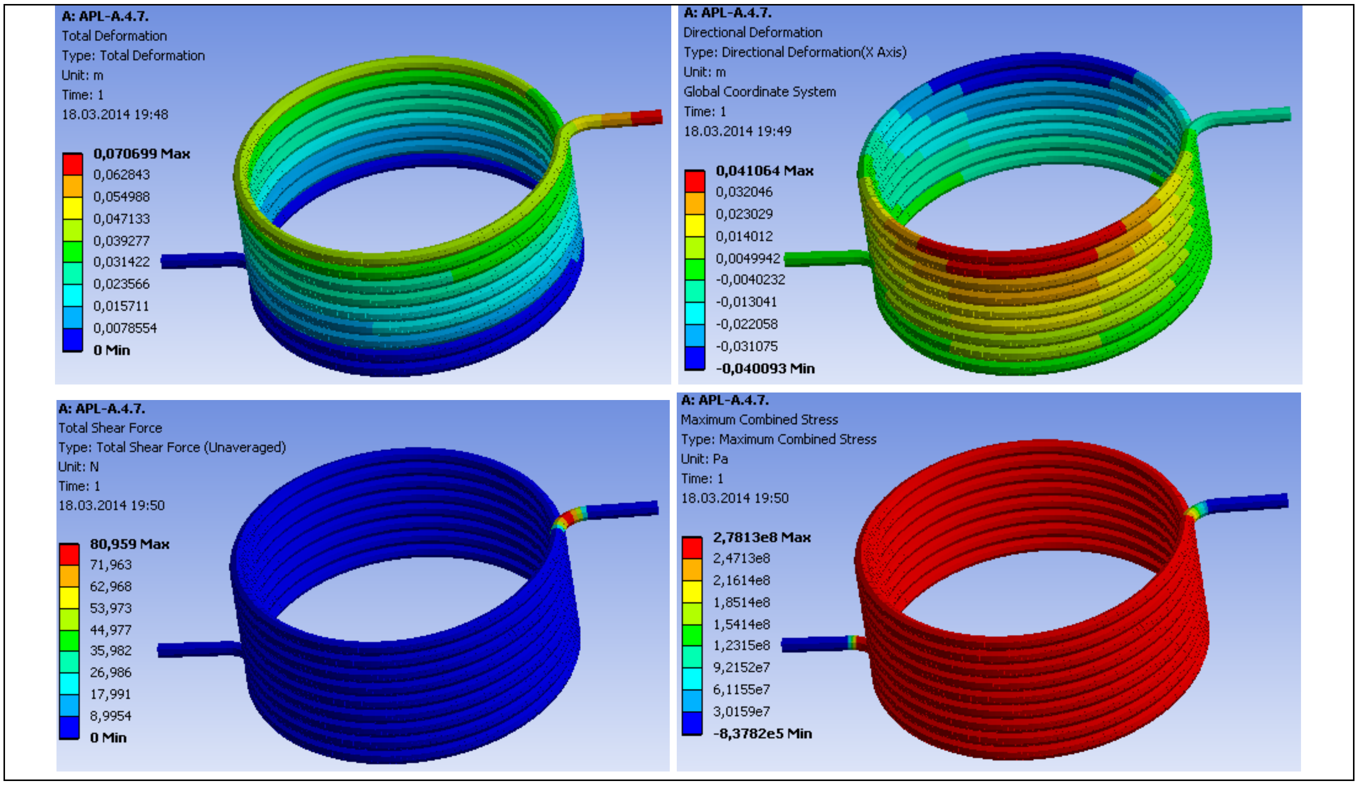
For this, the action of the moment will be suspended and an imposed angular displacement will be introduced.

, Outline: Moment → Suppress; Supports → Remote Displacement → Details of „Remote Displacements” → **Scope** → Geometry: [will be selected with ↲ the arc of the connecting circle between the spring propeller and the terminal segment at dimension 0, using the selection filter (Edge)] → Apply → Definition → X Component: Free, Y Component: Free, Z Component: 0, Rotation X: 0, Rotation Y: 0, Rotation Z: -90°;



Remote Displacement → Promote Remote Point → ↲ Remote Displacement - Remote Point → Details of „Remote Displacement - Remote Point” → **Scope** → X Coordinate = 0, Y Coordinate = 0, Z Coordinate = 0 [the coordinates of the point P1 (0, 0, 0) made previously will be written] → Solve.

The results obtained, for an imposed displacement of -90°, are presented below.


A: APL-A.4.7.
Total Deformation
Type: Total Deformation
Unit: m
Time: 1
18.03.2014 19:48
0,070699 Max
0,062843
0,054988
0,047133
0,039277
0,031422
0,023566
0,015711
0,0078554
0 Min
A: APL-A.4.7.
Directional Deformation
Type: Directional Deformation(X Axis)
Unit: m
Global Coordinate System
Time: 1
18.03.2014 19:49
0,041064 Max
0,032046
0,023029
0,014012
0,0049942
-0,0040232
-0,013041
-0,022058
-0,031075
-0,040093 Min
A: APL-A.4.7.
Total Shear Force
Type: Total Shear Force (Unaveraged)
Unit: N
Time: 1
18.03.2014 19:50
80,959 Max
71,963
62,968
53,973
44,977
35,982
26,986
17,991
8,9954
0 Min
A: APL-A.4.7.
Maximum Combined Stress
Type: Maximum Combined Stress
Unit: Pa
Time: 1
18.03.2014 19:50
2,7813e8 Max
2,4713e8
2,1614e8
1,8514e8
1,5414e8
1,2315e8
9,2152e7
6,1155e7
3,0159e7
-8,3782e5 Min