Application: AEF-A.11
Compression strained springs

KEY WORDS
Linear Static Analysis, Linear Material, 3D Geometric Model, 3D Finite Element, Linear Finite Element, Classical Verification, Machine Element

CONTENT

A. PROBLEM DESCRIPTION

A.1 Introduction

Many technical products contain mechanical elements 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.

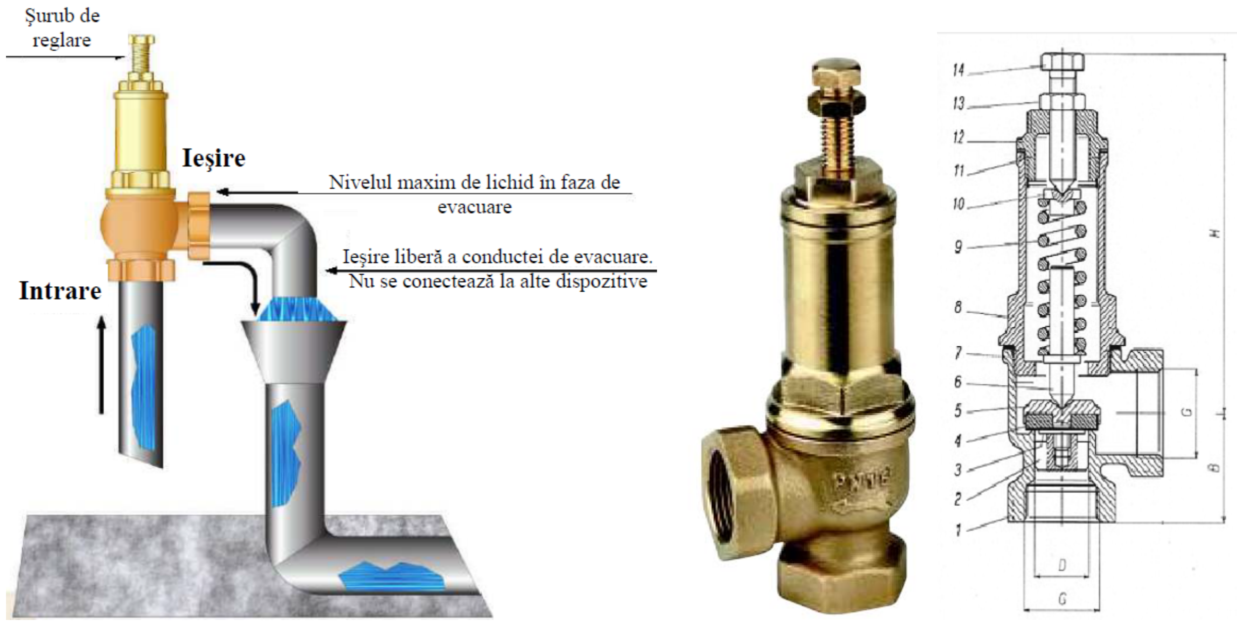
A.2 Application description

Safety valves are designed to protect tanks, pipes, boilers, boilers or other equipment containing pressurized fluids. These prevent pressure limits from being exceeded when all automatic control and monitoring equipment no longer operates.

Many safety valves (see the figures below, Spring safety valve, Fi-Fi brass body, PN 16, DN ½ ”… 3”, http://www.prestcom-instal.ro, accessed Apr. 2014) have in composition of active elastic elements used to obtain elastic characteristics imposed by the functional requirements. In this case, by changing the coil spring inside the valve, valves with different operating characteristics can be made.

The helical spring has the role of generating an axial force that compensates the force generated by the fluid pressure inside the installation and when the latter increases, the spring will compress by opening the exhaust circuit.

The coil spring used must comply with certain geometrical constraints (to fit within the available space) and to operate (to ensure the force necessary for the operation of the installation, to compress when an overpressure occurs, to generate a sufficiently large stroke so that the section of the circuit is suitable for emergency evacuation and, last but not least, return to working order after restoration of working pressure).



a. ***b.*** ***c.***

A.3 Application goal
In this application it is presented the analysis of the fields of displacements, deformations and tensions in the structure of the elastic element of helical spring type in the valve composition presented above (PN 16, DN 3/4 ”) as well as the values of forces generated by compressing the spring with a certain displacement. which oppose the opening of the valve at nominal working pressures. The values of the geometric and mounting parameters of the helical spring are: d = 2 mm, D1 = 17 mm, the number of turns n = 5 and the pitch t = 5.75 mm. The coil spring is made of spring steel, 50VCr11A, treated at 50-55 HRC. Axial compression of the spring (3) with the screw (6) in the drawing above will generate a force that compensates for the pressure inside the container on the front surface of the valve piston (according to the product data sheet, the valve piston surface is 283 mm2). This application monitors the value of the dependence between the value of the compression of the spring and the force generated on the valve piston, in order to design the valve as well as the study of internal stresses in the spring to check if the material meets the operating requirements.

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,

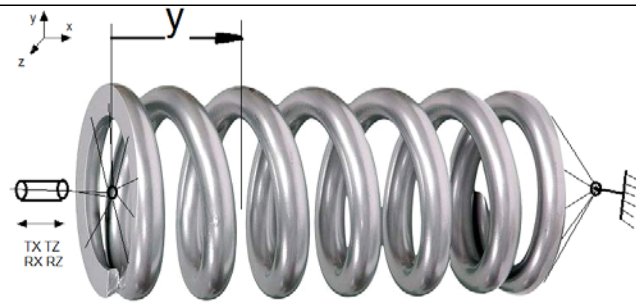
- restrictions induced by links with adjacent elements,
- external and internal loads (own weight),
- material characteristics.

B.2 The analysis model description

The geometric shape and dimensions of the helical spring are shown in the adjacent figure. For the analysis, the structure of the helical spring is modeled with 3D finite elements.

In order to simulate the behavior of the helical spring as close as possible to reality, taking into account the increased rigidity of the surfaces on which the spring is placed, two associated rigid elements are introduced.

In order for the analysis model to have the same behavior as the real model, it is necessary to associate boundary conditions that involve translation constraints according to the X and Z directions of the XYZ coordinate system, respectively only motion will be allowed on OY, simulating the placement of the helical arc in the valve seat. In order to generate the translational movement along the OY axis, a rotational translation coupling is introduced associated with the master point of the rigid element at the bottom, corresponding to the point of application of the force.



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 in *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

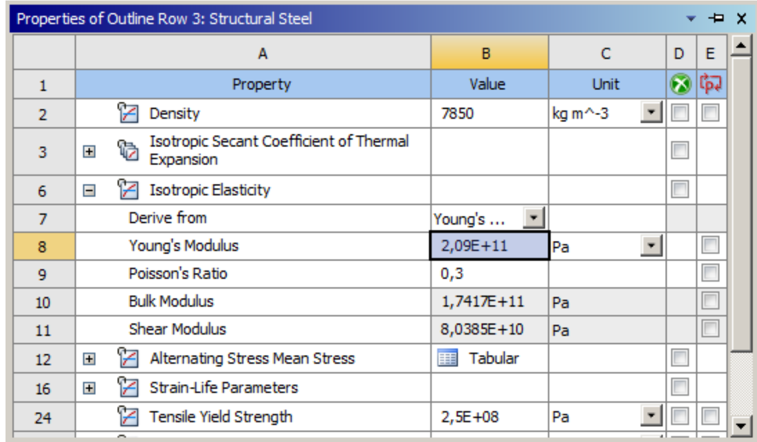
↵ Save As... → Save As , File name: [input name, Spring] → ↵ Save

C.2 Modelling of material and environment characteristics

, 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 box from 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.3 Geometric modelling

C.3.1 Model loading, DesignModeler (DM)

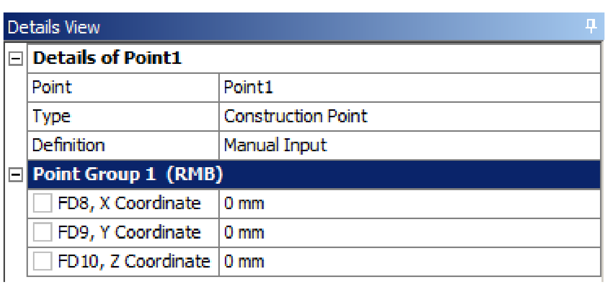
, Project Schematic: ↳ Geometry → ↵ New Geometry... → ANSYS Workbench: ↵ Millimeter, ↵OK.

C3.2 Spring helix generating

, Tree Outline: ↵Sketching → (Look at face/Plane/Sketching) [automatically view of default plane XY Plane];

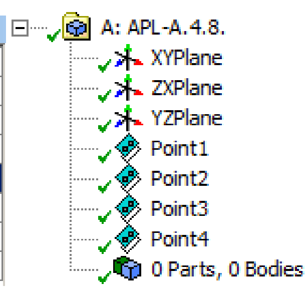
, Modeling → Create → Point [in the 3D modeling area the point P1 is created based on the Cartesian coordinates] → Details View → **Details of Point 1** → Definition: Manual Input; **Point Group 1 (RMB)** → x = 0; y = 0; z = 0 → Generate.

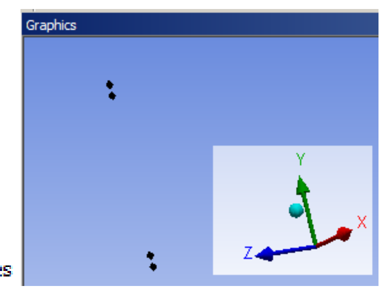
The points P2… P4 are constructed in the same way, using the resulting Cartesian coordinates based on the dimensions given in the model for analysis: P2 (0; 2; 0); P3 (0; 30.75; 0); P4 (0; 32.75; 0).



Details View

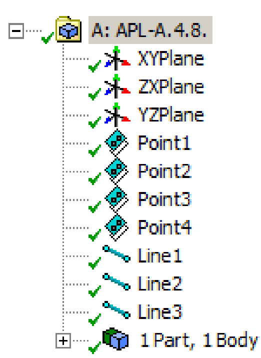
Details of Point1	
Point	Point1
Type	Construction Point
Definition	Manual Input
Point Group 1 (RMB)	
FD8, X Coordinate	0 mm
FD9, Y Coordinate	0 mm
FD10, Z Coordinate	0 mm

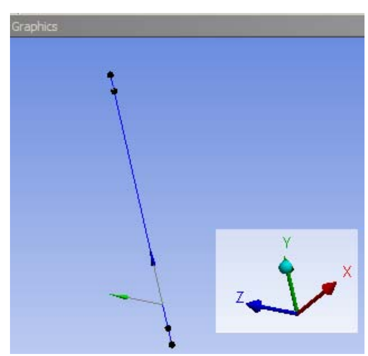




→ Modeling → Concept → Lines From Points → Details View → **Details of line 1** → Point segments → (points P1 and P2 are selected by holding down the *Ctrl* key) → Apply → Generate.

The segments P2P3 and P3P4 are built in the same way.

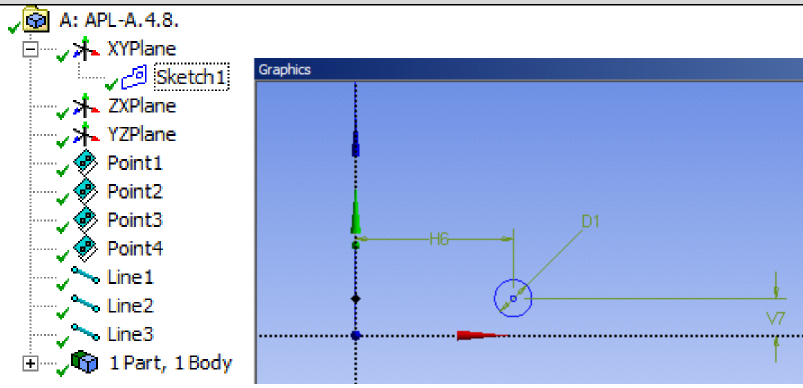




C3.3 Solid generating

Generating the spring section

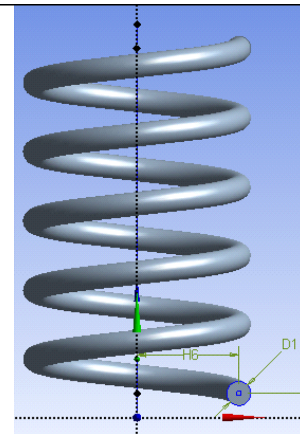
→ Sketching → Draw → Circle → Dimensions → Diameter → Details View → **Dimensions:1** → D1 = 2 mm → Dimensions → Horizontal → **Dimensions:2** → H6 = 8,5 mm → Vertical → **Dimensions:3** → V7 = 2 mm → Generate.



Generating the central area of the resort

→ Sweep → Details View → **Details of Sweep1** → Profile: Sketch1 (the previously drawn circle is selected) → Apply → Path: Line2 (select the P2P3 segment from the graphic area) → Apply → Twist Specification: Pitch → Pitch = 5,75 mm → Generate.

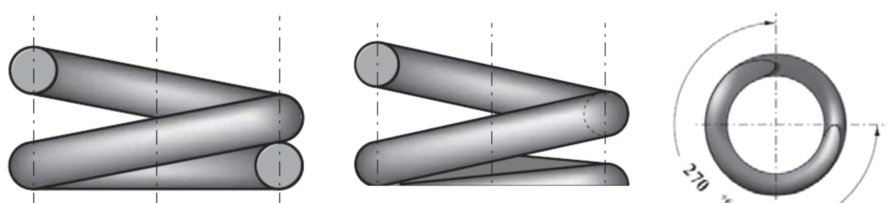
Obs. The geometric characteristics of the spring can also be entered using the number of steps: Twist Specification: Turns → Turns=5 → Generate.



Generating the end areas of the spring

Because in this geometric shape of the spring the contact with other fixing elements will be made in the form of very small seating surfaces (geometrically, the contact is only a point), it is necessary to create appropriate seating areas.

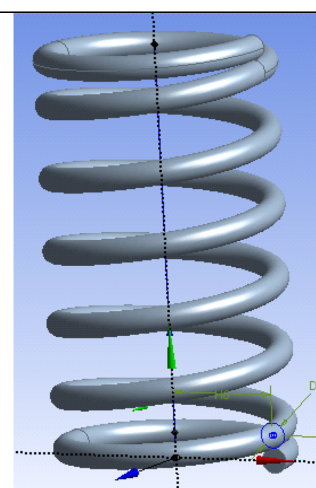
There are two constructive forms, shown in the figures below. For this application, form 2 will be used. To make the polished ends, the spring must be extended on one side and on the other with a coil with a smaller step, so that a contact spot of at least 270° can be made when milling.



capăt așezat   capăt așezat și șlefuit

→ Sweep → Details View → **Details of Sweep2** → Profile: Sketch1 (select the circle previously drawn, located at the axis on the axis Oy = 2 mm) → Apply → Path: Line1 (the P1P2 segment is selected from the graphics area) → Apply → Twist Specification: Turns → Turns = 1 → Generate.

→ Sweep → Details View → **Details of Sweep3** → Profile: (select the circular section of the spring located at 32.75 mm on the Oy axis) → Apply → Path: Line3 (select the P3P4 segment from the graphics area) → Apply → Twist Specification: Turns → Turns = 1 → Generate.

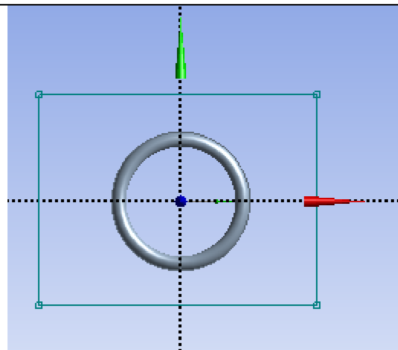


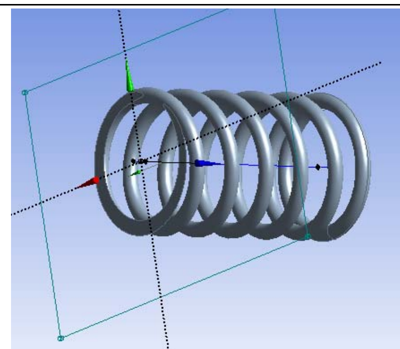
The milling surfaces of the spring are milled by extruding command (using the option to cut from the material) some rectangular surfaces drawn in planes parallel to the xOz plane at elevations of 1.25 mm and 31.5 mm, respectively.

→ New Plane → Details View → **Details of Plane4** → Type: From Plane → Base Plane: ZXPlane → Transform 1 (RMB): Offset Z → Value 1: 1,25 mm → Generate. Repeat the sequence of commands to generate the plan at 31.5 mm.

→ Plane4 → → Sketching → <u>Draw</u> → Rectangle (a rectangle is constructed so that the spring fits into it) → Modeling → Extrude → → Details View → **Details of Extrude3** → Geometry: Sketch3 (select the rectangle from the Plane4 plane) → Operation : Cut Material → Direction: Reversed → Depth: 10 mm → Generate.

Follow the same steps for the other end of the spring.





C.3.4 Save of geometric model

, ↵ (Save Project) → <u>File</u> → Close Design Modeler.

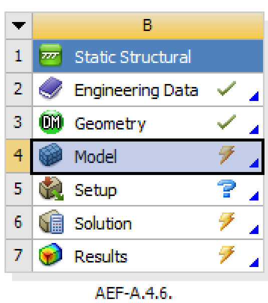
C.4 Finite element modelling

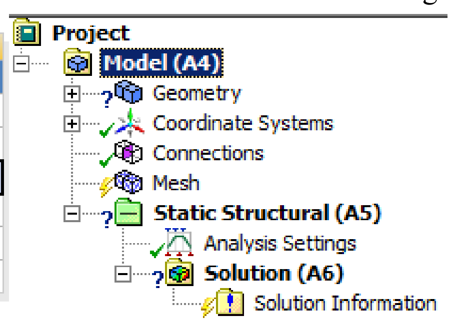
C.4.1 Launching the finite element modeling module and set the material characteristics and problem type

<u>Launching of the finite element modeling module</u>

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

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





<u>Setting the unit of measure system</u>

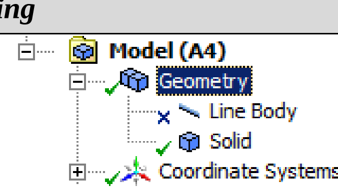
: ↵ Units → ↵ Metric (mm, kg, N, s, mV, mA) (the system of units of measurement is usually set by default).

C.4.2 Model discretization and analysis type setting

<u>Disable help items</u>

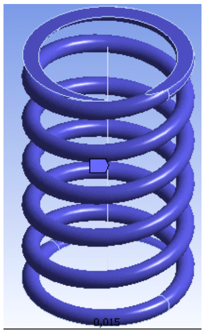
P1P2, P2P3 and P3P4 segments will be disabled so as not to affect discretization and other subsequent operations.

, ↵ Geometry → ↳ Line Body → Suppress Body.



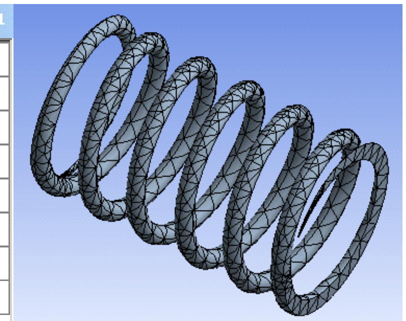
, ↵ Mesh → ↵ Mesh Control → ↵ Sizing → Details of "Sizing" - Sizing → **Scope** → Select Geometry: [will be selected with ↵ spring geometry, using the selection filter (Body)] Apply; **Definition Element** → Size: 0,002 m → Update.

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



Details of "Body Sizing" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Type	Element Size
Element Size	2,e-003 m
Behavior	Soft



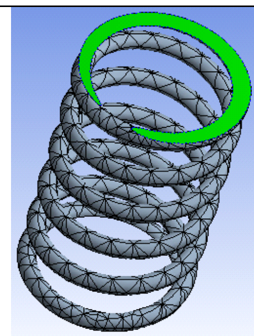
C.5 *Supports and restraints modelling*

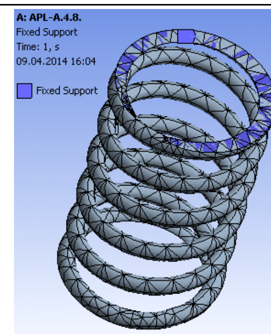
Input the gravitational acceleration

Because the weight of the spring is very small (approximately 75 g), the influence of the weight force (0.73 N) on the results of the analysis is very small, taking into account the value of the other stresses.

Input restraint

, ↵ Static Structural (A5) → Supports → Fixed Support → Details of „Fixed Suports" → **Scope** → Geometry: [select with ↵ the milled surface at the end of the spring at a height of 1.25 mm, using the (Face)] → Apply.



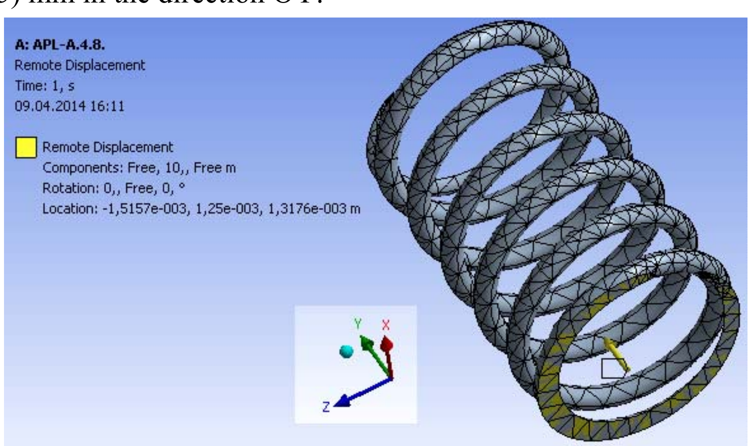


C.6 Load modeling

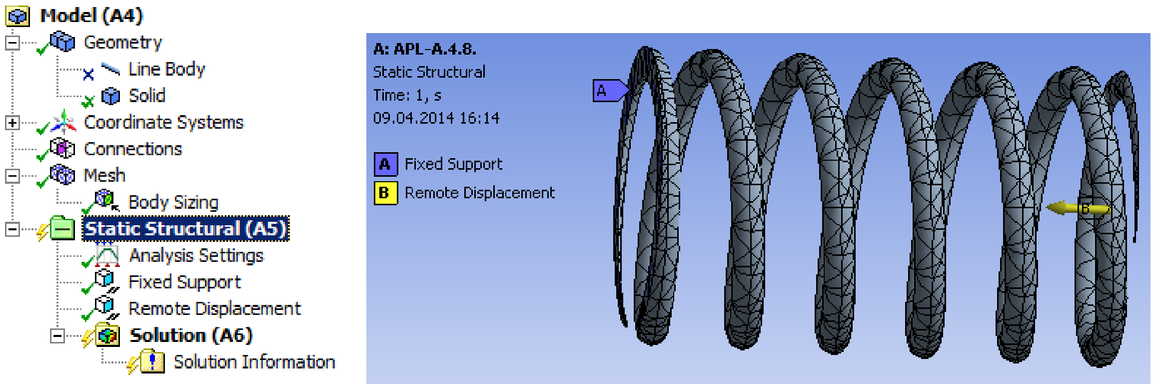
Input Remote Displacement

, ↵ Static Structural (A5) → Supports → Remote Displacement → → Details of „Remote Displacement" → **Scope** → Geometry: [select with ↵ the milled surface at the end of the spring at a height of 31.5 mm, using the (Face)] → Apply; **Definition** → X Component: Free, Y Component: 10 mm, Z Component: Free; Rotation X: 0, Rotation Y: Free, Rotation Z: 0, Behavior: Rigid.

A virtual rigid element was formed which consists of the front milled surface of the spring, located at an elevation of 31.5 mm, which is connected to a rotary translation coupling on the OY axis and which is printed a displacement of 10 (15) mm in the direction OY.



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



D. SOLVING THE FEA MODEL

D.1 Setting the convergence criterion for solving the model

Outline: → ↵ Solution (A6) → ↵ Solution Information, Details of "Solution Information", ↵ Solution Information: ↵Solution Output → [selecting from the list with ↵, ↵Force Convergence] (the force convergence criterion is adopted). These steps will be repeated and chosen „Displacements Convergence".

D.2. Setting results

Selecting the types of 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.

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

The same result can be obtained by using commands:

↳ Solution (A6) → ↵ Insert → Deformation → Total [the buttons in the menu bars are used] and:

↳ Solution (A6) → ↵ Insert → Deformation → Directional.

↳ Solution (A6) → ↵ Insert → Stress → Equivalent (von-Mises).

↳ Solution (A6) → ↵ Insert → Stress → Error.

↳ Solution (A6) → ↵ Insert → Stress → Maximum Principal.

↳ Solution (A6) → ↵ Insert → Strain → Equivalent (von-Mises).

↳ Solution (A6) → ↵ Insert → Strain → Vector Principal.

↳ Solution (A6) → ↵ Insert → Tools → Stress Tool → Safety Factor.

↳ Solution (A6) → ↵ Insert → Energy → Strain Energy.

Next, the other types of results to be analyzed are set, respectively the reactions in the supports:

↳ Solution (A6) → ↵ Insert → Probe → Force Reaction;

↳ Solution (A6) → ↵ Insert → Probe → Moment Reaction.

D.3 Launching the solving module

Outline: Analysis Settings → Details of "Analysis Settings" → **Solver Controls** → Large Deflection: On → ↵ Solution (A6) → Solve.

E. POST-PROCESSING OF RESULTS

E.1 Viewing the displacement field

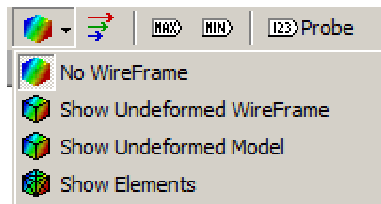
For suggestive results, set the view scale of the menu bars: Result 8,6e+002 (Auto Scale) → Result 1.0 (True Scale).

Total deformation viewing

M, 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 Underformed WireFrame* will be selected, an option that displays the undeformed and warped models in the same representation.



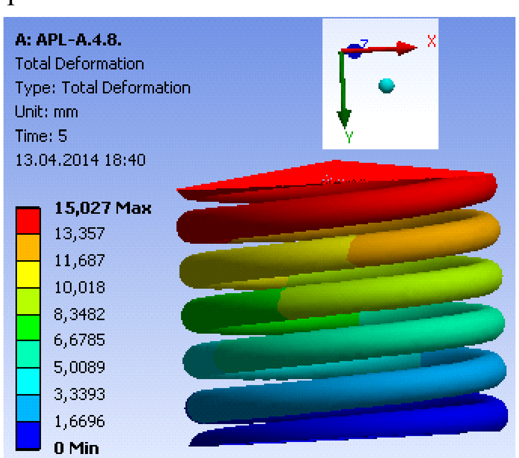
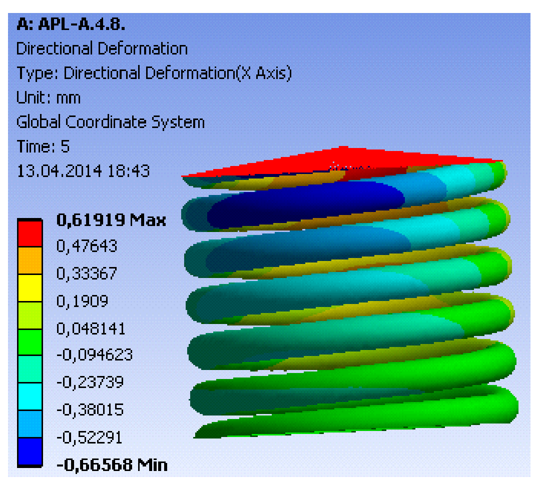
The display characteristics can be changed: the number of frames 10 Frames, as well as the running time of the simulation 2 Sec (Auto). At the same time, the result can be saved as a video file using the *Export Video File* command.

Visualization of the deformation in one direction

M, Outline: ↵ Solution (A6) → Directional Deformation → Graph → Animation ▶ ■.

If you want to view it in another direction, follow the steps below:

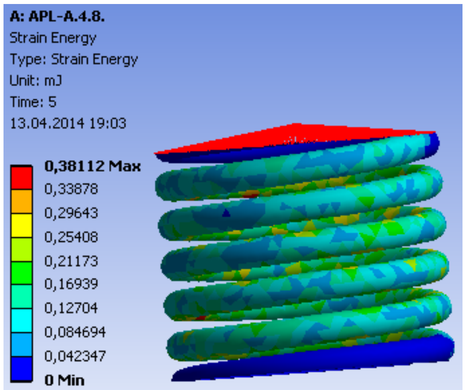
M, Outline: ↵ Solution (A6) → Directional Deformation → Details of „Directional Deformation" → **Definition** → Orientation : X Axis → Solve.

Total Deformations	*Directional Deformation*
↵ Solution (A6) → Total Deformation → Graph → Animation ▶ ■.	↵ Solution (A6) → Directional Deformation → Graph → Animation ▶ ■.
	

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

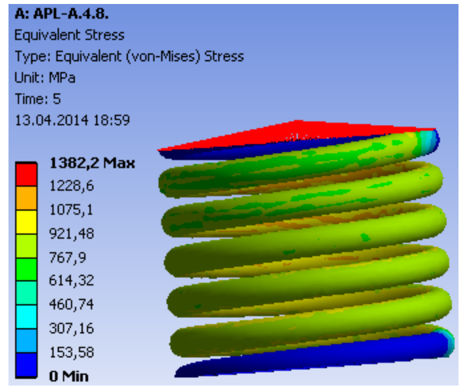
Strain Energy

↵ Solution (A6) → Strain Energy → Graph → Animation.



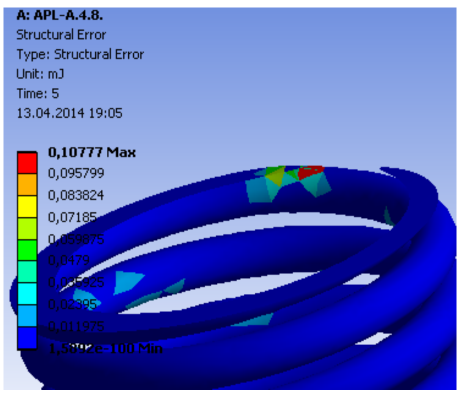
Equivelent Stress

↵ Solution (A6) → Equivalent Stress → Graph → Animation.



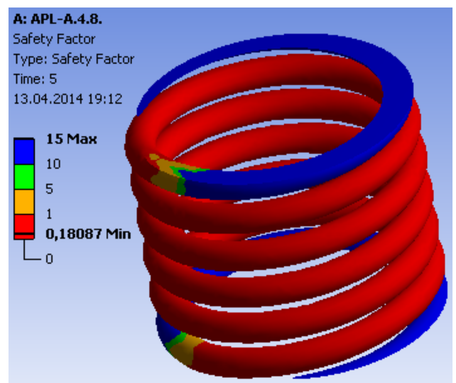
Structural Error

↵ Solution (A6) → Structural Error → Graph → Animation (sau Tabular Data).



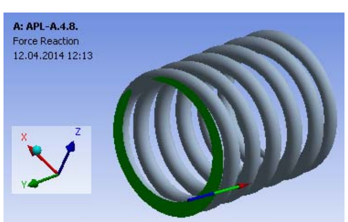
Safety Factor

↵ Solution (A6) → Safety Factor → Graph → Animation (sau Tabular Data).



Force reaction in support

↵ Solution (A6) → Force Reaction → Graph → Tabular Data.

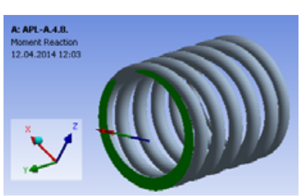


Tabular Data

	Time [s]	Force Reaction (X) [N]	Force Reaction (Y) [N]	Force Reaction (Z) [N]	Force Reaction (Total) [N]
1	1,75	-2,2517e-002	-34,33	-3,3442	34,493
2	2,25	-4,279e-002	-44,059	-4,3879	44,277
3	2,75	-7,1138e-002	-53,753	-5,4735	54,031
4	3,5	-0,13174	-68,226	-7,1793	68,603
5	4,	-0,18594	-77,828	-8,3664	78,277
6	4,5	-0,25271	-87,392	-9,5883	87,916
7	5,	-0,33295	-96,916	-10,839	97,521

Moment reaction in support

↵ Solution (A6) → Moment Reaction → Graph → Tabular Data.



Tabular Data

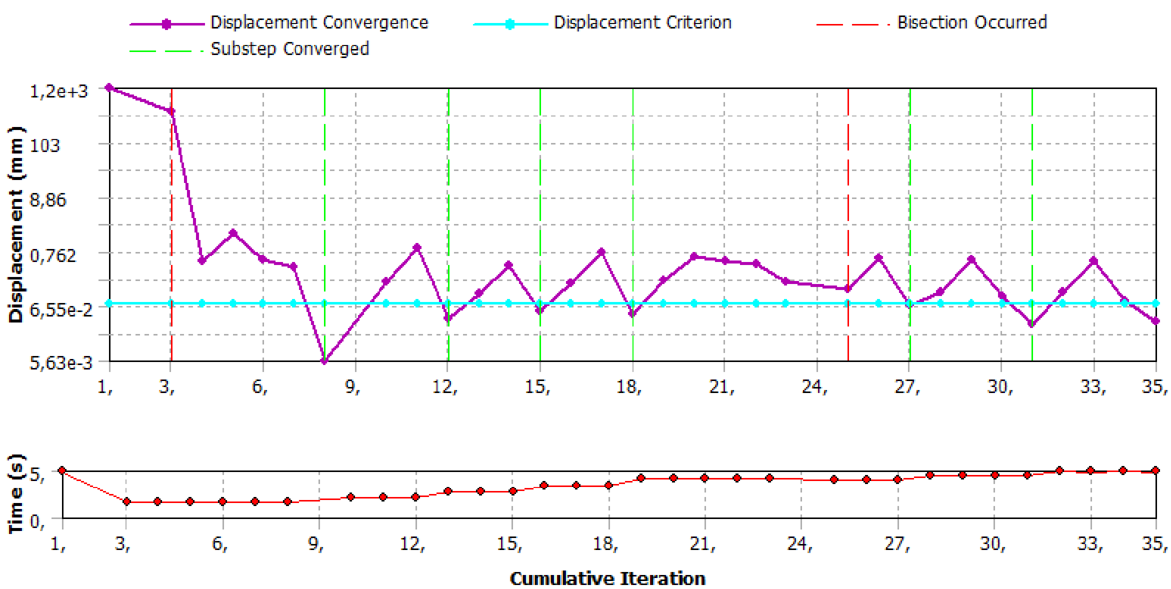
	Time [s]	Moment Reaction (X) [N·mm]	Moment Reaction (Y) [N·mm]	Moment Reaction (Z) [N·mm]	Moment Reaction (Total) [N·mm]
1	1,75	87,054	5,0396	-49,646	100,34
2	2,25	109,64	6,5898	-63,599	126,93
3	2,75	131,09	8,2072	-77,444	152,48
4	3,5	160,9	10,713	-97,993	188,7
5	4,	179,04	12,445	-111,52	211,3
6	4,5	195,67	14,194	-124,9	232,57
7	5,	210,64	15,98	-138,09	252,38

E.3 Visualizing the convergence of solutions

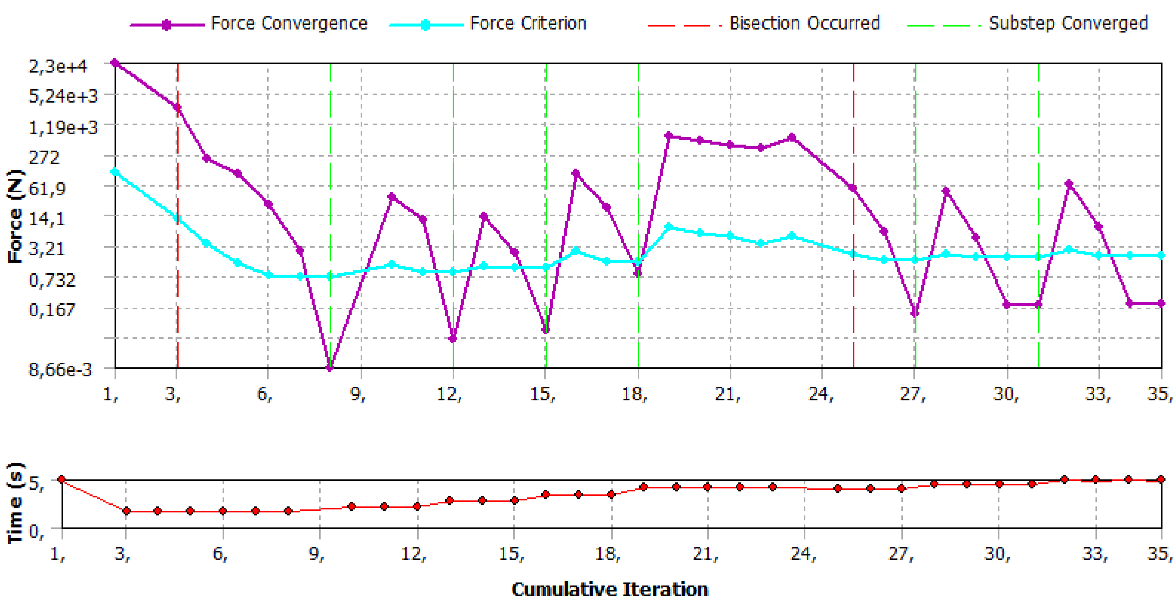
, Outline: ↵ **Solution (A6)** → Solution Information → Details of "Solution Information" → **Solution Informations** → Solution Output : Displacements Convergence.

Repeat these steps and select "*Force Convergence*".

Displacement Convergence

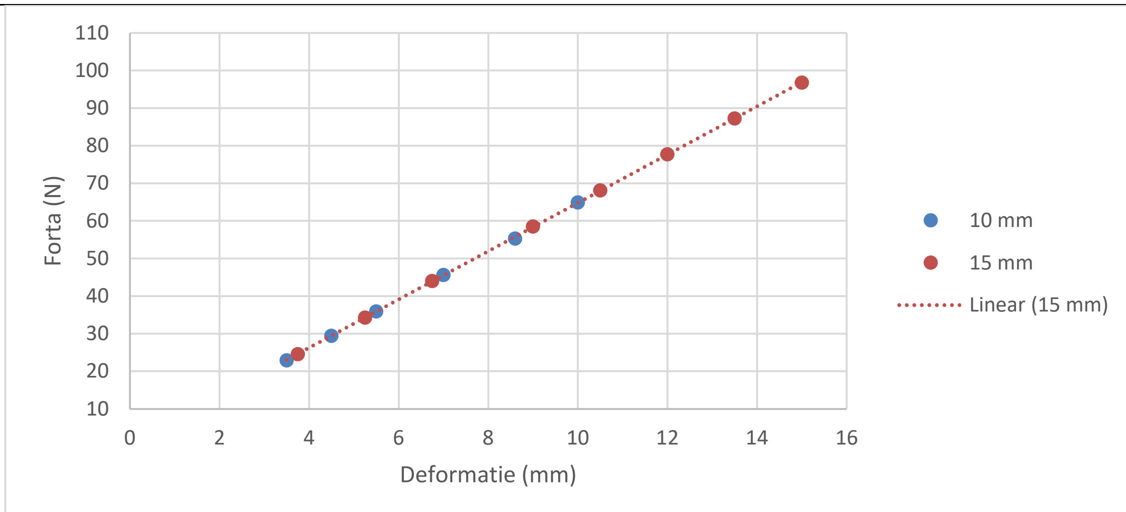


Force Convergence



F. ANALYSIS OF RESULTS

F.1 Interpretation of results
The characteristic of the spring was drawn after performing two simulations with deformations of 10 mm (blue dots), respectively 15 mm (red dots). A perfect overlap of the results is observed, and it can be concluded that this graph is correct.



From the point of view of the deformations in the area 0… 15 mm, it is observed that the graph is a straight line segment, so in this interval the spring works in the elastic zone of the deformations. The value of the force generated by the spring can be extracted from the graph, depending on the value of its deformation.

For example, at a spring compression of 10 mm, the force generated on the valve piston is about 65 N. According to the technical data of the safety valve analyzed, the valve seat has a front surface of 283 mm^2. According to the relation p = F / S, the value of the nominal pressure obtained by the valve is obtained: p = 2.3 bar. For a spring compression of 15 mm, the operating pressure becomes p = 3.4 bar.

Because the maximum compression of this spring depends on the pitch, the number of turns, and the diameter of the turn:

$x = (p - d) \cdot n = (5{,}75 - 2) \cdot 5 = 18{,}75\ mm$,

it can be concluded that this valve will operate in a range of working pressures between 0.7 barr (corresponding to a 2 mm spring compression) and 3.75 barr (for 16 mm compression).

F.2 Accuracy and convergence analysis
The information regarding the deformations, corroborated with the information regarding the equivalent stresses, the structural error, the convergence of the solutions lead to the conclusion that the spring withstands the loads without problems, the values of the maximum stresses not exceeding the allowed limit value of the material. Increased attention must be paid to the connections at the exit of the spring propeller, at both ends, these two areas being important concentrators of stresses and a discretization finish is required, here appearing the maximum structural errors. The much lower values of the structural error field (max 0.107 mJ, subchapter E.2) indicate that the stress values are close to the exact ones. In addition, from subchapter. E.3 highlights the fast convergence (25 steps) of the model solving algorithm and the calculation time is reduced.

G. CONCLUZII / CONCLUSIONS

In order to use the valve for higher ranges of working pressures (its body withstanding pressures of 16 barr), it is necessary to change the spring with some with different characteristics: either with a larger coil diameter or better materials.

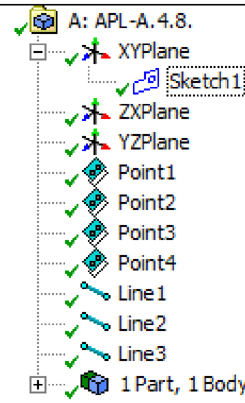
To demonstrate the concept, the diameter of the coil will be changed from 2 mm to 2.5 mm, as follows:

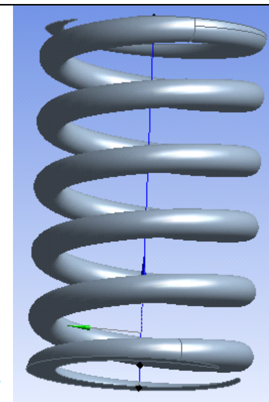
DM → Sketching → ↵ Sketch1 → Details View → **Dimensions:3** → D1 = 2,5 mm → Generate.

DM → Sketching → ↵ Sweep2 → Details View →

Details of Sweep2 → Twist Specifications: Pitch

→ Pitch = 2,5 mm Generate.

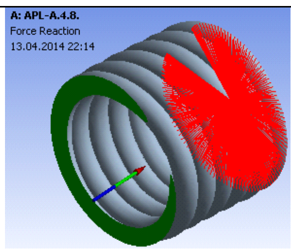




Repeat the steps for Sweep3.The spring in the figure from above will be obtained. Next, the analysis operations will be performed according to the steps presented above.

M → File → Refresh All Data → Solve.

↵ **Solution (A6)** → Force Reaction → Graph → Tabular Data.



Tabular Data

	Time [s]	Force Reaction (X) [N]	Force Reaction (Y) [N]	Force Reaction (Z) [N]	Force Reaction (Total) [N]
1	0,5	-4,4687e-003	-24,4	-2,2287	24,501
2	1,	-1,9466e-002	-48,712	-4,5406	48,923
3	1,75	-6,717e-002	-85,019	-8,1745	85,411
4	2,25	-0,11957	-109,11	-10,713	109,64
5	2,75	-0,19182	-133,12	-13,346	133,79
6	3,5	-0,3436	-168,97	-17,471	169,87
7	4,	-0,47781	-192,76	-20,33	193,83
8	4,5	-0,6425	-216,45	-23,265	217,7
9	5,	-0,8382	-240,05	-26,259	241,49

Based on the data obtained by simulating a 15 mm compression, the characteristic of the 2.5 mm diameter coil spring is obtained. The maximum deformation of this spring is 16.25 mm. Depending on the characteristics of this spring, the nominal working pressures will be in the range (1,2; 8,4) barr. It can be seen that for any spring, following a fairly simple analysis, it can be checked whether it will work properly but the values of the nominal working pressures of the safety valves can also be estimated.