



LINEAR STATIC ANALYSIS OF METALIC WIRE ELEMENTS

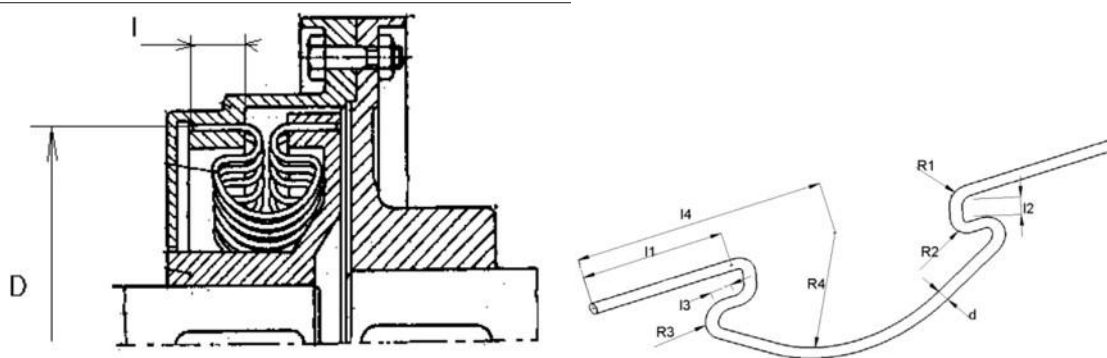
2

INTRODUCTION

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

DESCRIPTION OF THE APPLICATION

Mechanical couplings are important link subsystems used for transmitting power (rotation and torque) between usually two shafts belonging to different subsystems. Many mechanical couplings consist of active elastic elements used to obtain torsional elastic characteristics imposed by functional requirements or, in the case of controlled precision transmissions, to obtain increased torsional rigidity and transverse flexibility (to compensate for mounting deviations). The figure below shows a coupling belonging to the first category. The two semi-couples are rigidly attached to the ends of the shafts by means of feather assemblies. The torque is transmitted from the semi-conductor to the semi-coupling driven by a set of curved bows whose ends are freely positioned in boreholes on the two half-shells. Thus, during operation, there is relative movement between the curved bows and the semi-couples in which they are mounted.



PURPOSE OF THE APPLICATION

In this case, the analysis of the field of displacements, deformations and stresses in the structure of the curved beam elastic element of the coupling component shown above is presented. The values of the geometric and mounting parameters of those, $z = 60$, bar springs are: $d = 2$ mm, $l_1 = 30$ mm, $R_1 = 2,5$ mm, $l_2 = 7,5$ mm, $R_2 = 2,5$ mm, $l_3 = 15$ mm, $R_3 = 3$ mm, $l_4 = 50$ mm, $R_4 = 30$ mm, $l = 10$ mm, $D = 200$ mm. The bar springs are made of spring steel, 50VCr11A, treated at 50-55 HRC. The torque transmitted is $M_t = 400,000$ Nmm.

Application AEF-A.2.2

LINEAR STATIC ANALYSIS OF ELASTIC METAL WIRE ELEMENTS

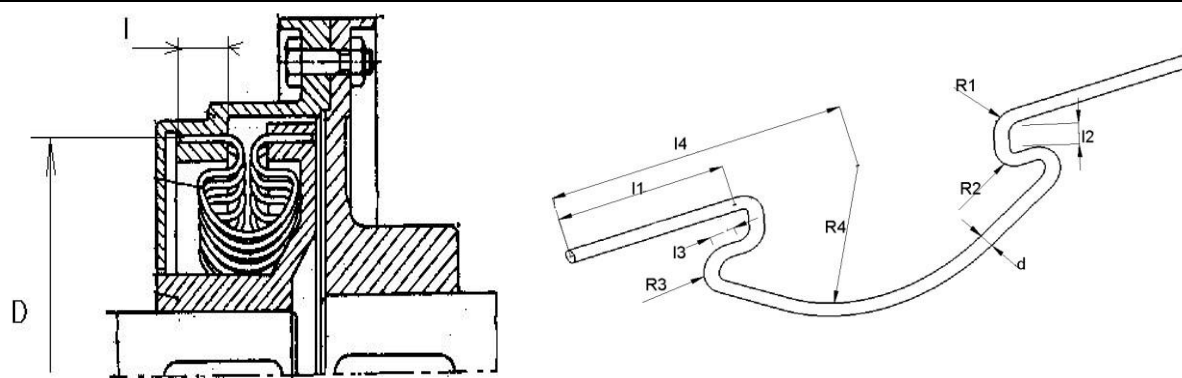
A. APPLICATION DEFINITION

A.1. Introduction

Many technical products contain mechanical components with distinct, compact structures required by their main function. Representative of this group of components are elastic elements (springs), damping elements, support elements (housings), etc. The specificity of these elements is usually given by their fixed or quasi-fixed connections with neighboring parts. The finite element analysis of these components, to obtain precise results, requires the accurate definition of the solid model, the constraints imposed by the connections with neighboring elements, as well as the loads.

A.2. Description of the Application

Mechanical couplings are important connecting subsystems used to transmit power (rotational motion and torque) between, usually, two shafts belonging to different subsystems. Many mechanical couplings include active elastic elements used to obtain torsional elastic characteristics imposed by functional requirements or, in the case of controlled precision transmissions, to obtain increased torsional stiffness and transverse flexibility (to compensate for assembly deviations). The figure below shows a coupling belonging to the first category. The two semi-couplings are rigidly fixed on the shaft ends by keyway assemblies. The torque is transmitted from the driving semi-coupling to the driven semi-coupling through a set of curved bar springs whose ends are freely positioned in bores on the two semi-couplings. Thus, during operation, there is relative motion between the curved bar springs and the semi-couplings in which they are mounted.



A.3. Purpose of the Application

This application presents the analysis of the displacement, strain, and stress fields in the structure of the curved bar elastic element from the coupling presented above. The values of the geometric and assembly parameters for the, $z = 60$, bar springs are: $d = 2$ mm, $l_1 = 30$ mm, $R_1 = 2.5$ mm, $l_2 = 7.5$ mm, $R_2 = 2.5$ mm, $l_3 = 15$ mm, $R_3 = 3$ mm, $l_4 = 50$ mm, $R_4 = 30$ mm, $l = 10$ mm, $D = 200$ mm. The bar springs are made of spring steel, 50VCr11A, treated to 50-55 HRC. The transmitted torque is $M_t = 400,000$ Nmm.

B. PREPARING THE ANALYSIS MODEL

B.1. Defining the Analysis Model

To prepare the finite element analysis model associated with the application above, it is necessary to identify

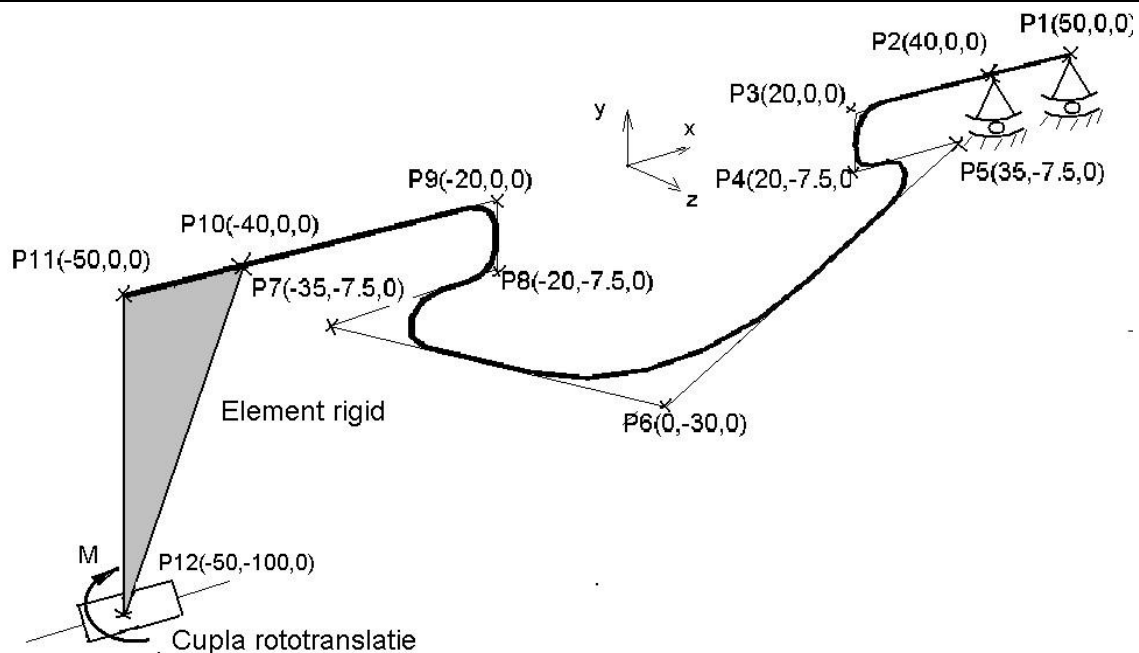
- The shape and geometric dimensions of the curved bar spring,
- Adjacent elements with increased rigidity,
- Constraints induced by the connections with adjacent elements,
- External loads,
- Material characteristics.

B.2. Description of the Analysis Model

The geometric shape and dimensions of the curved bar spring are presented in the figure below. For the analysis, the structure of the curved bar spring is modeled with 1D finite elements, and thus, the geometric model is reduced to a curved line which is modeled starting from a broken line to which the required fillets are applied.

In order to simulate the behavior of the curved bar spring as close to reality as possible, considering the increased rigidity of the semi-couplings, a rigid element is introduced associated with the driving semi-coupling. For the analysis model to have the same behavior as the real model, it is necessary to associate boundary conditions that involve translation constraints along the three directions of the XYZ coordinate system and rotation constraints about the Y and Z axes, simulating the connections of the curved springs with the semi-couplings.

In addition, to generate the rotational motion of the spring ends around the coupling axis, a sliding pivot joint is introduced, associated with the master point of the rigid element. Also, for a behavior as close as possible to the real model, the loading is done through the rigid element that simulates the driving semi-coupling, upon which the moment $M = M_t/z = 6666.66 \text{ Nmm}$ acts.



B.3. Establishing Material Characteristics

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

- Longitudinal modulus of elasticity, $E = 209,000 \text{ N/mm}^2$;
- Transverse contraction coefficient (Poisson's ratio), $\nu = 0.3$.

C. PREPROCESSING THE ANALYSIS MODEL

C.1. Geometric Modeling

Activate the solid generation module and set the unit of measurement for lengths.

Start → Mechanical Design → Part Design → New part: New part name: Arc. Tools → Options... → Options: Parameters and Measure; Units; Length, Millimeter (mm); ↵ OK

Generate the reference points.

 (Point) → Point Definition: X 50mm, Y 0mm, Z 0mm; ↵OK. [Similarly, enter the coordinates of the other points (fig. B.2): P2(40,0,0), P3(20,0,0), P4(20,-7.5,0), P5(35,7.5,0), P6(0,-30,0), P7(-35,-7.5,0), P8(-20, -7.5,0), P9(-20,0,0), P10(-40,0,0), P11(-50,0,0), P12(-50, -100,0)].

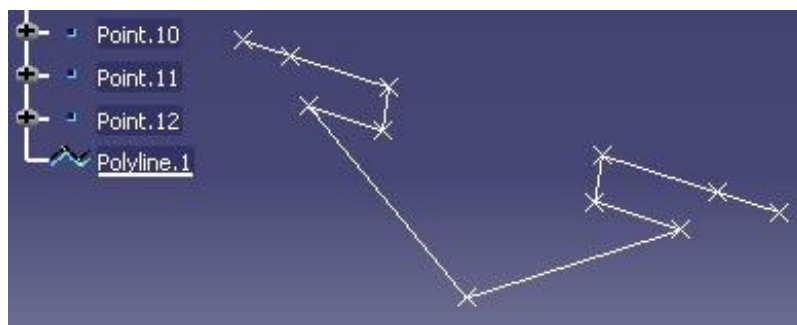


Generate the polyline through points.

 (Polyline) **Polyline Definition** [select with the mouse from the graphical area or from the tree structure the points Point.1 to Point.11, simultaneously generating their list in the Polyline Definition window], ↵ OK.

Observation

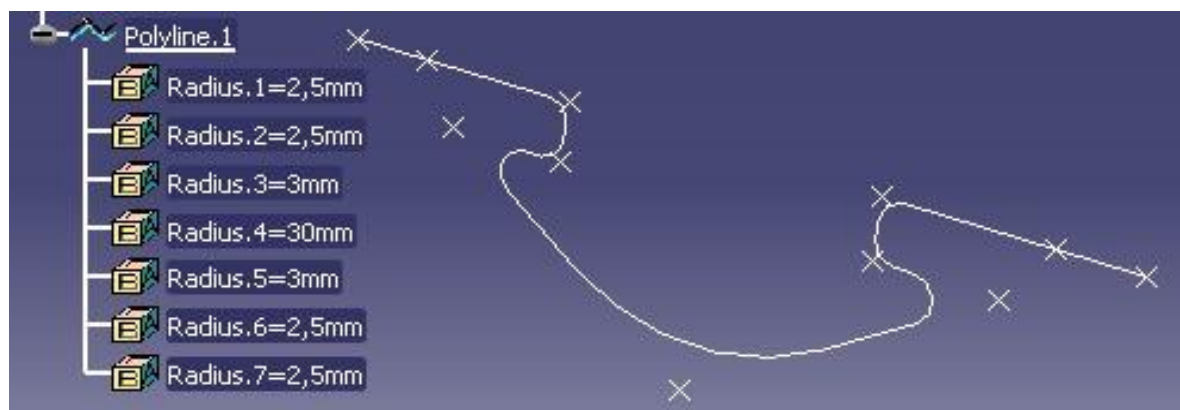
If the (Polyline) button is not found in the command area (Toolbar), the  following sequence will be executed: View → Toolbars → Customize... **Customize:** Commands, Categories ↓ All Commands, Commands ↓ Polyline... → place the command symbol with the mouse (Drag Drop) in the command bar area (Toolbar), ↵ Close.



Generate the polyline with fillets.

[Select the Polyline.1 specification in the tree structure] → **Polyline Definition:**

Point.3, Radius: 2.5 mm, add; Point.4, Radius: 2.5 mm, add; Point.5, Radius: 3 mm, add; Point.6, Radius: 30 mm, add; Point.7, Radius: 3 mm, add; Point.8, Radius: 2.5 mm, add; Point.9, Radius: 2.5 mm, add; ↵ OK.



C.2. Modeling the Material Characteristics

[Select the PartBody specification] → (Apply Material) → **Library (Read-only):** Metal, Steel, Apply Material; ↵OK. [Select the Steel specification from the tree structure] → **Properties:** Feature Properties, Feature Name: **50VCr11A**; Analysis, Young Modulus 204000 N/mm², Poisson Ratio 0.3; Apply; ↵OK.



C.3. Finite Element Modeling

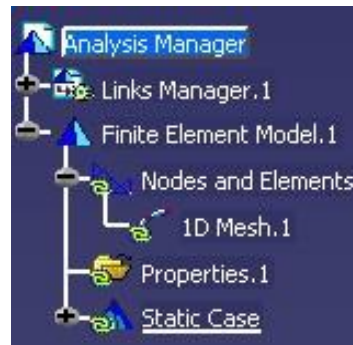
Activate the advanced finite element discretization module.

Start → Analysis & Simulation → Advanced Meshing Tools → New Analysis Case: Static Analysis, ↵OK.



Enter the finite element type and discretization characteristics.

 (Beam Mesher) → Beam Meshing: [select the polyline Polyline.1], Element size: 2 mm, ☒ Sag control, Sag: 0.5 mm, Min size: 0.1 mm, ↵OK.

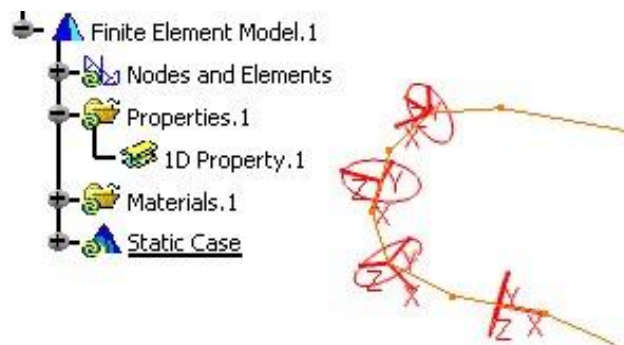


Activate the finite element analysis module.

Start → Analysis & Simulation → Generative Structural Analysis.

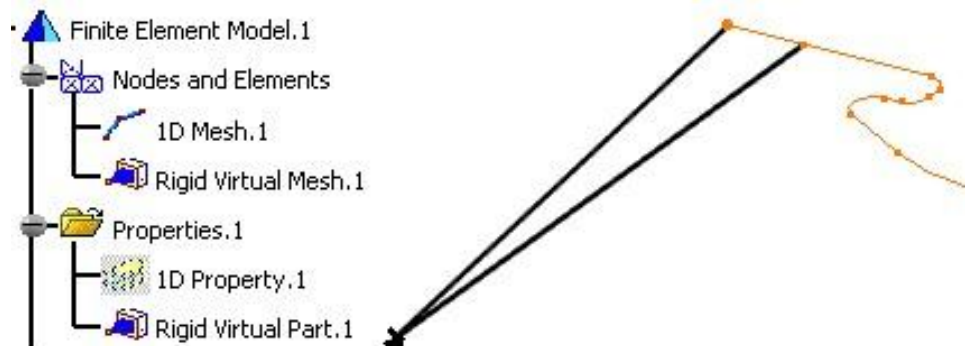
Enter the finite element properties.

 (1D Property) → **1D Property**: Supports: [select 1DMesh.1 from the tree structure]; **Type** ↓ Cylindrical beam , → Beam Definition: Radius: 1 mm, ↵OK → ↵OK.



Generate the virtual rigid element.

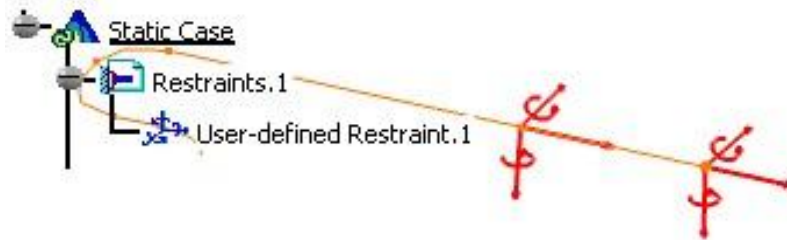
 (Rigid Virtual Part) → **Rigid Virtual Part** → Supports: [select the segment P10, P11]; Handler: P12; ↵OK.



C.4. Modeling Geometric Constraints

Introduce displacement-type restrictions.

 (User-defined Restraint) → User-defined Restraint: Supports [select with the  mouse the segment  P1P2]; Restrain  Translation 2, Restrain Translation 3, Restrain Rotation 2, Restrain Rotation 3; ↵OK.



Introduce the sliding pivot joint restriction.

 (Sliding Pivot) → Sliding pivot: Supports [select with the mouse the rigid element Rigid Virtual Mesh.1],  Display locally, Released Direction: X 1, Y 0, Z 0; ↵OK.



C.5. Modeling Loads

Introduce the torque.

 (Moment) → Moment: Supports [select with the mouse the rigid element, Rigid Virtual Mesh.1];  Display locally; Moment Vector: X 0.67Nm, Y 0 Nm, Z 0 Nm; ↵OK.



D. SOLVING THE FINITE ELEMENT MODEL

D.1. Launching the Calculation Module

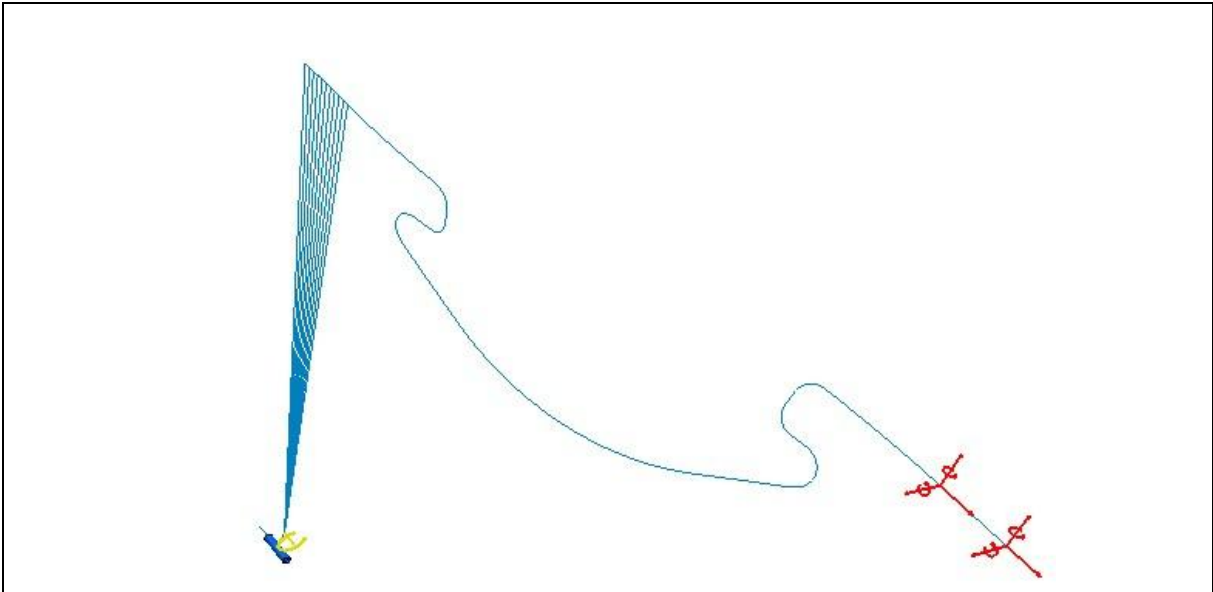
 (Compute) → Compute: ↓ All, ↵OK → Computation Resources Estimation, ↵Yes → Computation Status.

E. POSTPROCESSING THE RESULTS

E.1. Visualizing the Deformed State/Animation

 (Deformation).

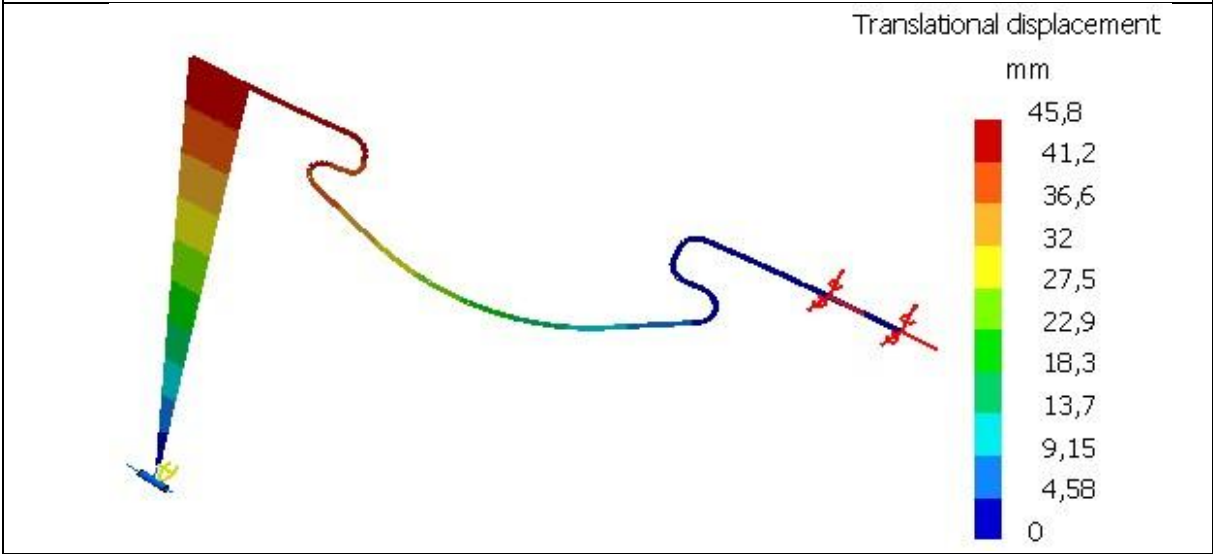
 (Animate) → Animate: Steps number: 20 (other parameters can also be modified), ↵Close.



Observation
To modify the amplitude of the displacements in the animation for improved suggestiveness, the following sequence can be followed: →  **Amplification magnitude:** Factor: 1 (other parameters can also be set), ↵OK.

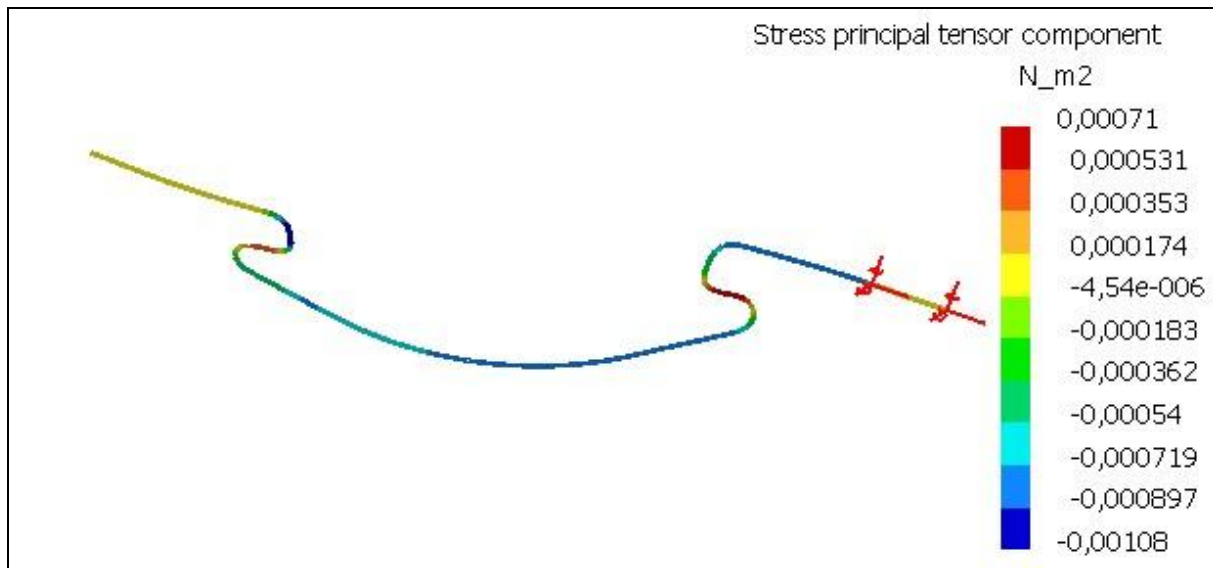
E.2. Visualizing the Displacement Field

 (Displacement).



E.3. Visualizing the Internal Stress Field

 (Principal Stress).



F. CONCLUSIONS

The data obtained in this application can be used in design for strength and stiffness checks. The torsional stiffness of the coupling is calculated with the relation $k_t = z\delta D m^2 M_t = 5240 \text{ Nmm/rad}$, considering $\delta = 45.8 \text{ mm}$ (fig. E.2) as the relative circumferential displacement of the semi-couplings due to the elastic deformation of the bar springs under the action of the torque M_t . The value of torsional stiffness can be used for comparison with the admissible one in verification calculations and in the calculation formulas for dynamic systems where the coupling plays an important role in attenuating torsional vibrations. On the other hand, fig. E.3 shows the maximum stress in the curved bar is much reduced (0.00071 N/m^2), which indicates that it has large deformation reserves.

G. EXERCISE

Perform a finite element analysis of the helical compression spring from the Cardeflex coupling, presented below, with the following dimensions: $D=200 \text{ mm}$, $D_1=16 \text{ mm}$, $d=3 \text{ mm}$, $z=4 \text{ coils}$, $L=50 \text{ mm}$. The helical springs are made of spring steel 50VCr11A treated to 50-55 HRC. The torque transmitted by the coupling is $M_t = 80 \text{ Nm}$.