



FINITE ELEMENT METHOD

AEF-A.2.9

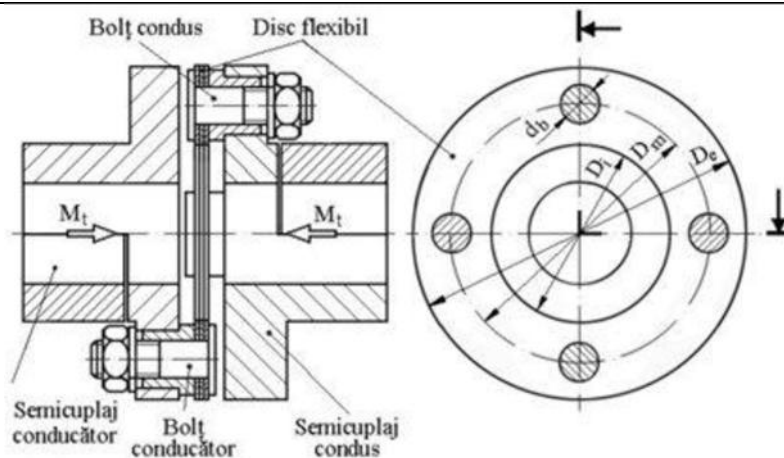
LINEAR STATIC ANALYSIS OF FLEXIBLE METALLIC ELEMENTS

INTRODUCTION

There are many technical products in mechanical components that have distinct compact structures required by the main function to perform. Representatives 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 connection 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 last category. The two half-couplings are rigidly attached to the shaft ends by tightening assemblies. The torque is transmitted from the steering half-coupling via two bolts through a pack of metal discs to the other two bolts attached to the driven half-coupling.



PURPOSE OF THE APPLICATION

In the present application, it is intended to determine the strength and rigidity characteristics of the flexible metallic discs mounted in the package between the two half-couplings in order to evaluate the behavior of this type of coupling in operation in various applications.

The particular case study is represented by coupling with $z = 3$ flexible elements in the form of annular discs with the following dimensions: $D_m = 95$ mm, $D_i = 80$ mm, $D_e = 110$ mm, $d_b = 10$ mm and thickness $g = 0,5$ mm. The annular discs are made of spring steel, 50VCr11A, treated at 50-55 HRC. The torque transmitted is $M_t = 40.000$ Nmm.

Application FEA-A.2.9

LINEAR STATIC ANALYSIS OF FLEXIBLE SHEET METAL ELEMENTS

A. DEFINITION OF THE APPLICATION

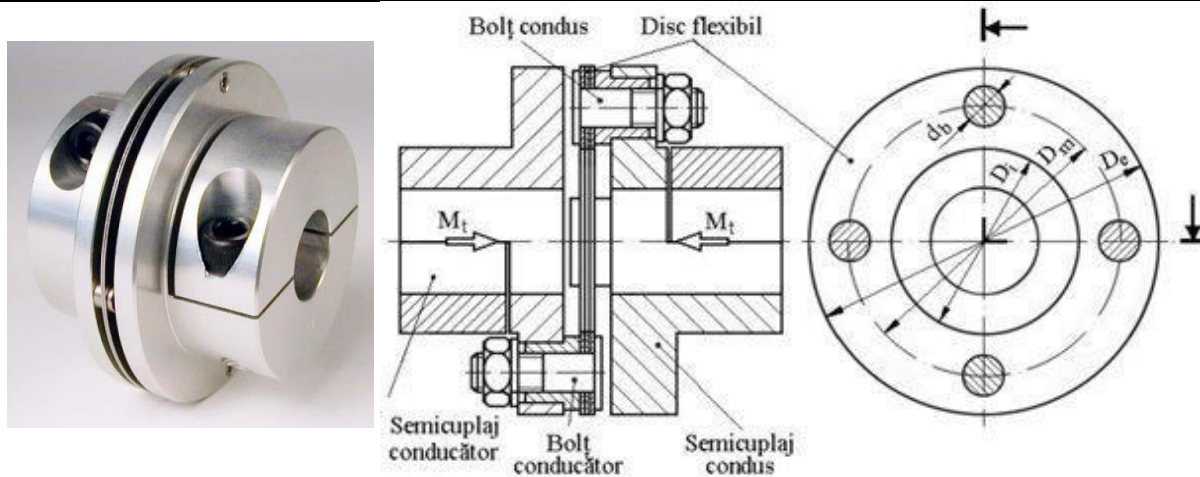
A.1. Introduction

Many technical products contain mechanical components that have distinct compact structures required by the main function they perform. Representatives of this group of components are elastic elements (springs), damping elements, supporting elements (housings), etc. The specificity of these elements is usually determined by their fixed or quasi-fixed links with adjacent parts. To obtain precise results, the finite element analysis of these components requires an accurate definition of the solid model, the constraints imposed by links with adjacent elements, and the loads.

A.2. Description of the application

Mechanical couplings are important connection subsystems used for transmitting power (rotation and torque), usually between 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 latter category. The two half-couplings are rigidly attached to the shaft ends by tightening assemblies. The torque is transmitted from the driving half-coupling via two bolts through a pack of metal discs to the other two bolts attached to the driven half-coupling.



A.3. Purpose of the application

The aim of this application is to determine the strength and rigidity characteristics of the flexible metallic discs mounted in the package between the two half-couplings, in order to evaluate the behavior of this type of coupling during operation in various applications. The specific case study involves a coupling with $z = 3$ flexible elements in the form of annular discs with the following dimensions: $D_m = 95$ mm, $D_i = 80$ mm, $D_e = 110$ mm, $d_b = 10$ mm and thickness $g = 0,5$ mm. The annular discs are made of spring steel, 50VCr11A, treated to 50-55 HRC. The transmitted torque is $M_t = 40.000$ Nmm.

B. PREPARATION OF THE ANALYSIS MODEL

B.1. Defining the analysis model

To prepare the finite element analysis model associated with the application above, the following must be identified:

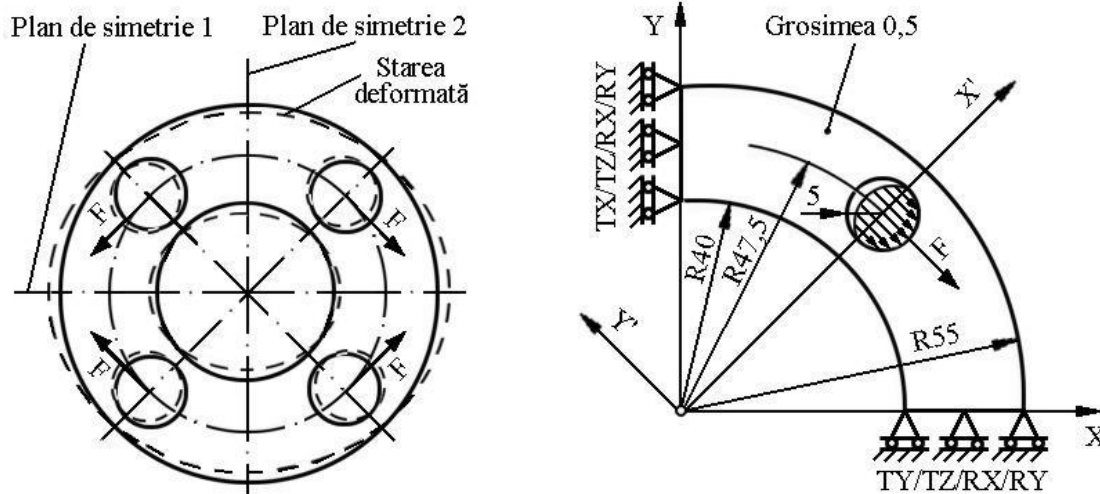
- Geometric shape and dimensions.
- Restrictions induced by connections with adjacent elements.
- External loads.
- Material characteristics.

B.2. Description of the analysis model

Defining the analysis model involves identifying the values of geometric and physical parameters relative to the XY coordinate system. Considering the geometric symmetry relative to two perpendicular planes of the undeformed and deformed states of the structure, the symmetry of the loading, and the isotropy of the mechanical characteristics, only a quarter of the annular disc structure is considered for analysis.

To ensure the analysis model behaves like the real case, boundary conditions must be associated with points on the X and Y axes that allow displacement *only* in the directions of these axes.

The loading of the analysis model, a consequence of the action of a bolt with increased stiffness, is done with a force $F = Mt/Dm = 421.05 \text{ N}$, distributed with a parabolic variation on the cylindrical contact semi-surface, equivalent to the torque action. The load acts circumferentially relative to a predefined local coordinate system.



B.3. Establishing material characteristics (50VCr11A)

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

- Longitudinal modulus of elasticity (Young's Modulus): $E = 209000 \text{ N/mm}^2$.
- Transverse contraction coefficient (Poisson's ratio): $\nu = 0,3$.

C. PRE-PROCESSING OF THE ANALYSIS MODEL

C.1. Geometric modeling

Activating the solid generation module and setting the unit of measurement for lengths

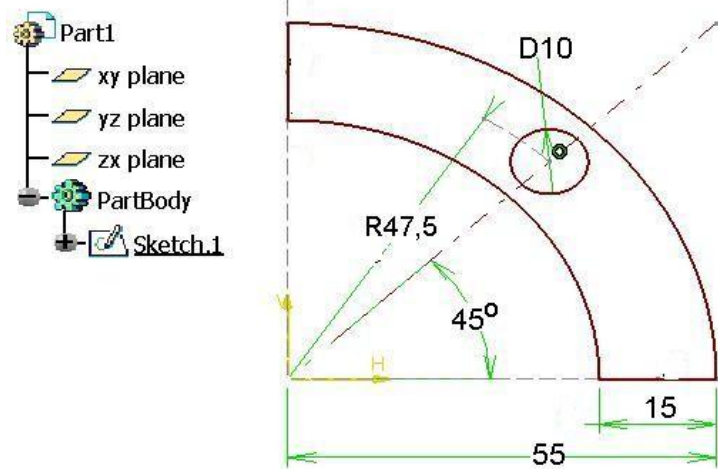
Start → Mechanical Design → Part Design → **New part**: New part name: Disc cuplaj.

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

↵ OK.

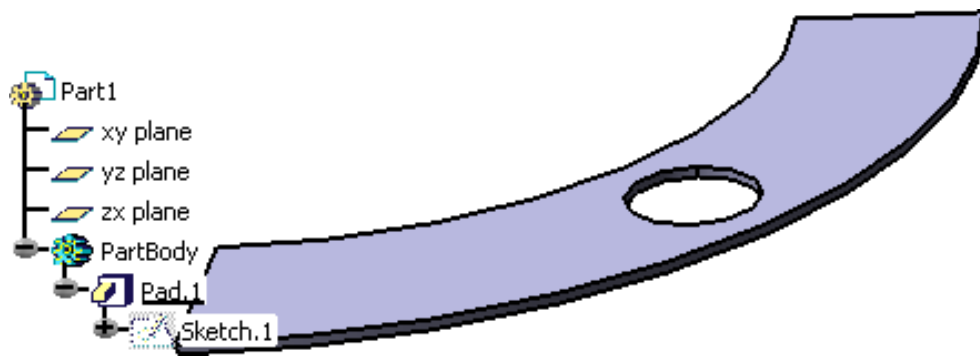
Generating the reference sketch

(Sketcher) → xy plane → (Arc) → [draw the two circular arcs] → (Line) → [draw the two circular arcs] → (Circle) → [draw the circle] → (Constraint) [enter dimensions successively by selecting entities] → (Exit Workbench).



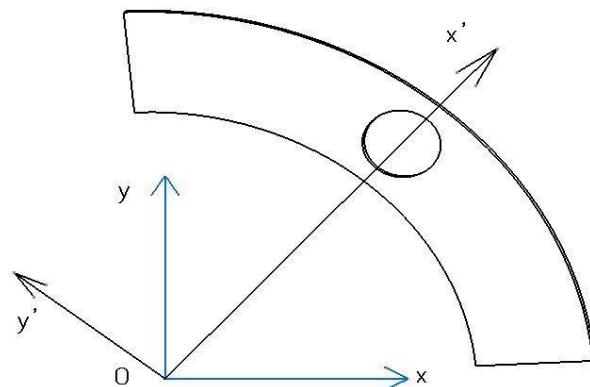
Generating the solid model

(Pad) → **Pad Definition:** Length: 0,5; Selection: [select Sketch.1]; ↵OK.



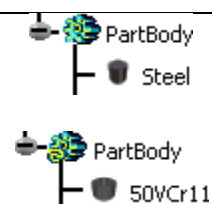
Creating the local coordinate system

Insert → (Axis System) → **Axis System Definition:** Axis system type: Euler angles; Origin: 0,0,0; Angle 1 =45 deg; Angle 2 = 45 deg; Angle 3 = 0 deg; ↵OK.



C.2. Modeling material characteristics

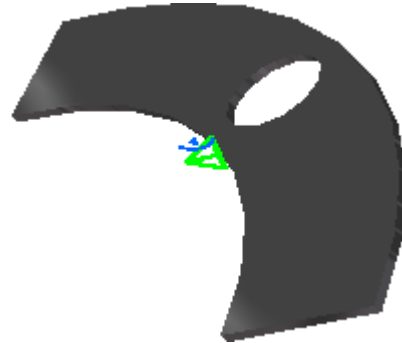
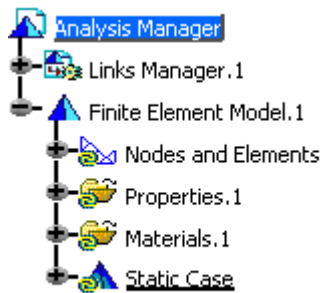
[Select body (PartBody) in the tree structure] → (Apply Material) → **Library (ReadOnly):** Metal, Steel; Apply; ↵OK. [Select Steel from the tree structure] → **Properties:** Feature Properties, Feature Name 50VCr11; Analysis: Young Modulus 209000 N/mm², Poisson Ratio 0,3; Apply; ↵OK.



C.3. Finite element modeling

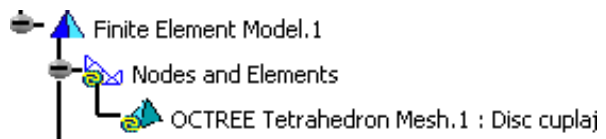
Activating the finite element analysis module

Start → Analysis & Simulation → Generative Structural Analysis → **New Analysis Case: Static Analysis**, ↵OK.



Modifying finite element dimensions

[Select the **OCTREE Tetrahedron Mesh: Part1** specification in the tree structure, or the symbol on the solid model structure] → **OCTREE Tetrahedron Mesh: Size: 1 mm; Absolute Sag 0,5 mm**; ↵OK.

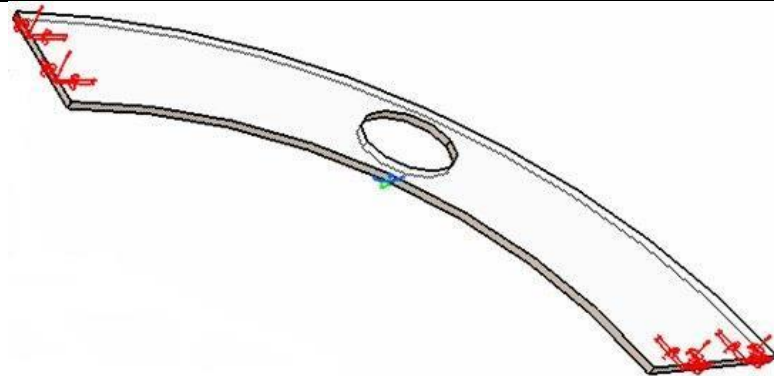
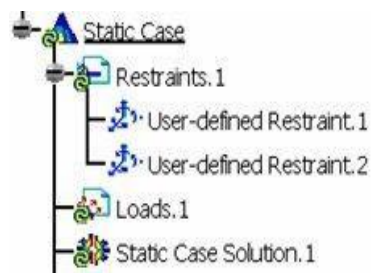


C.4. Modeling geometric constraints

Introducing displacement restrictions

 → **User-defined Restraint: Supports** [select the end face in the XOZ plane]; [deactivate: ☐ Restrain Translation 1, ☐ Restrain Rotation 3 (allows translation along the OX axis)], ↵OK.

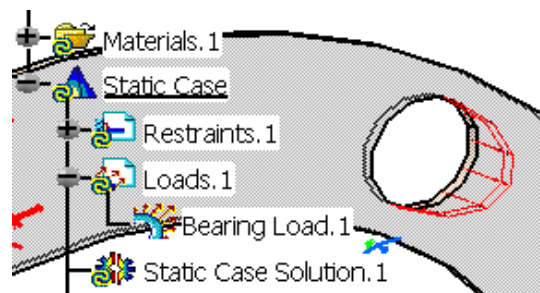
 → **User-defined Restraint: Supports** [select the end face in the YOZ plane]; [deactivate: ☐ Restrain Translation 2, ☐ Restrain Rotation 3 (allows translation along the OY axis)], ↵OK.



C.5. Modeling loads

Introducing the force

 (Bearing Load) → **Bearing Load** → Supports [select the cylindrical surface of the bolt hole]; Axis System $\perp$ User, Current Axis [select the local coordinate system, **Axis System.1**] Force vector X 0N, Y -421,05 N, Z 0N; Angle 180deg; Orientation Parallel; Type Parabolic; ↵OK.



D. SOLVING THE FINITE ELEMENT MODEL

D.1. Launching the calculation module

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

E. POST-PROCESSING OF RESULTS

E.1. Visualizing the deformed state / Animation

 (Deformation).

 (Animate) → **Animate**: Steps number: 5 (other parameters can be modified); ↵Close.

Observation

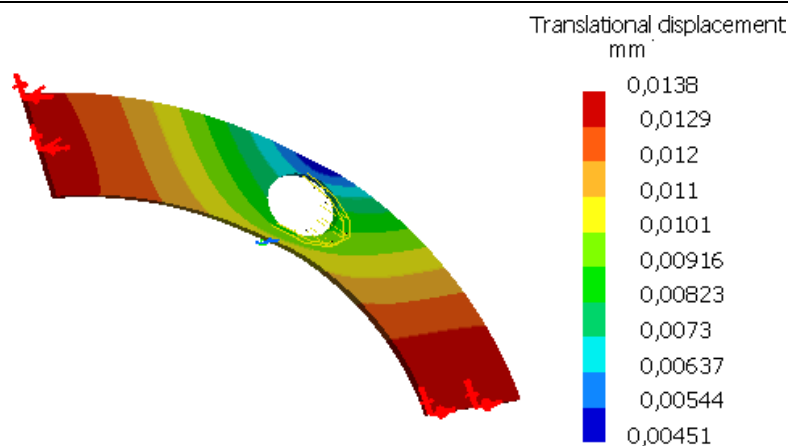
To modify the amplitude of displacements during animation for better visualization, follow the sequence:

 → **Amplification magnitude**: Factor: 1 (other parameters can be set); ↵OK.



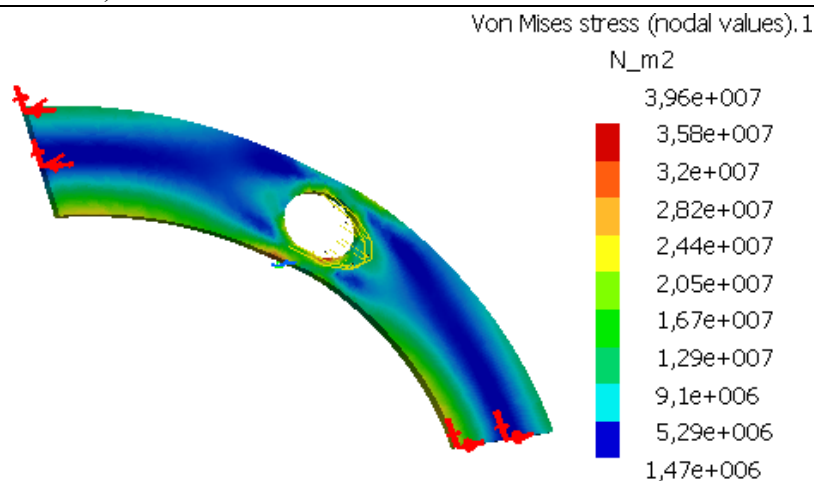
E.2. Visualizing the displacement field

 (Displacement).



E.3. Visualizing the stress field

 (Von Mises Stress).



F. CONCLUSIONS

The data obtained in this application can be used in design for strength and rigidity verification. The torsional rigidity of the coupling is calculated with the relation $k_t = 2zM_t/\delta D_m = 252630$ Nmm/rad, considering $\delta = 0,01$ mm (Fig. E.2) as the circumferential displacement of the bolt. The high value of torsional rigidity recommends this coupling for use in mechatronic systems. The torsional rigidity value can be used for comparison with the admissible value in verification calculations and in the calculation relations for the command-and-control modules of the system containing the coupling.

On the other hand, Fig. E.3 highlights the maximum Von Mises stress of 39.6 MPa, which is below the yield strength of the material used.

G. EXERCISE

Perform the finite element analysis of the elastic element, made of **50VCr11A** treated to 50-55 HRC, from the coupling component shown below [Patent RO 114670] with the following dimensions: $D = 141,4$ mm, $l = 100$ mm, $R1 = 25$ mm, $R2 = 7,5$ mm, $d = 8$ mm, $b = 20$ mm. The torque transmitted by the coupling is $M_t = 40.000$ Nmm.

