



## FINITE ELEMENT METHOD

AEF-A.2.5

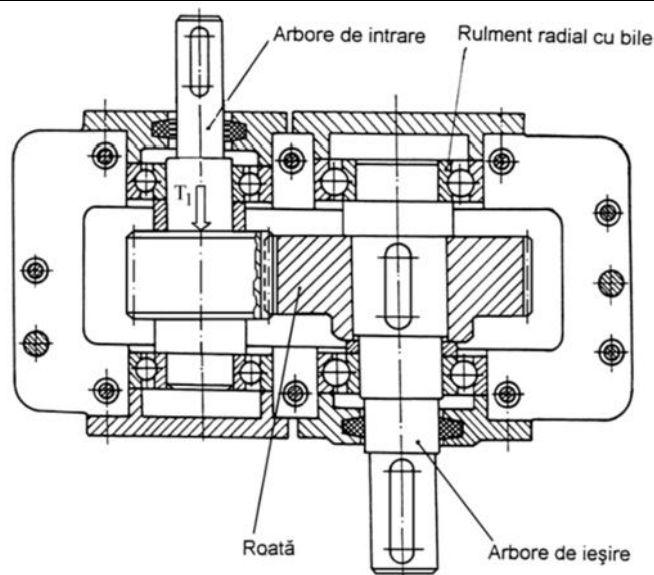
# LINEAR STATIC ANALYSIS OF THE INPUT SHAFT OF GEAR REDUCER

## INTRODUCTION

The functioning of the gear reducers is also influenced by the stiffness of the shafts on which the gears are mounted, as well as the stiffness of the wheel teeth. The identification of stress and strain fields in these parts, using finite element analysis, leads to the detailing of some useful conclusions about the values and distributions of these fields for the design of the gears.

## DESCRIPTION OF THE APPLICATION

The following figure shows the overall drawing of a cylindrical one step gear reducer which transmits and modifies the rotational movement and the torque with the transmission ratio  $i = 2$ . On the ends of the input and output shafts are mounted elastic couplings. The input shaft, the common body with the sprocket, as well as the output shaft are supported on radial ball bearings which are radially and axially fixed to the housing subassembly.



## PURPOSE OF THE APPLICATION

In this case, the analysis of the strain and stress fields in the input pinion shaft structure is presented. The analysis is performed in order to determine the areas heavily requested and, on the other hand, to show that the material flow limit is not specified under the maximum load conditions. The modeling and analysis is performed for the common body input shaft with the five-section cylindrical sprocket having the following sets of geometric dimensions (diameter, length), expressed in mm: (36,60), (40,40), (51,50), (48, 15), (40, 15). A wedge channel with the dimensions  $b = 8$  mm,  $h = 3.5$  mm,  $l = 50$  mm is practiced on the outer section. On the fourth section is practiced straight teeth with parameters: module,  $m = 3$  mm; number of teeth  $z_1 = 20$ ; head coefficient  $h_{0a}^* = 1$ ; game coefficient,  $c_0^* = 0.25$ ; connection coefficient,  $r_f^* = 0.38$ ; displacement coefficients  $x_1 = -0.25$ ,  $x_2 = 0.25$ ; rack angle,  $\alpha_0 = -0.25$ ; the distance between axes  $a_w = 90$  mm. The torque transmitted by the input shaft is  $T_1 = 22$  Nm. The gear shafts are made of thermally treated alloy 40Cr10 by improving to 250-270 HB.

# FEA Application A.2.5

## LINEAR STATIC ANALYSIS OF THE INPUT SHAFT OF A CYLINDRICAL REDUCER

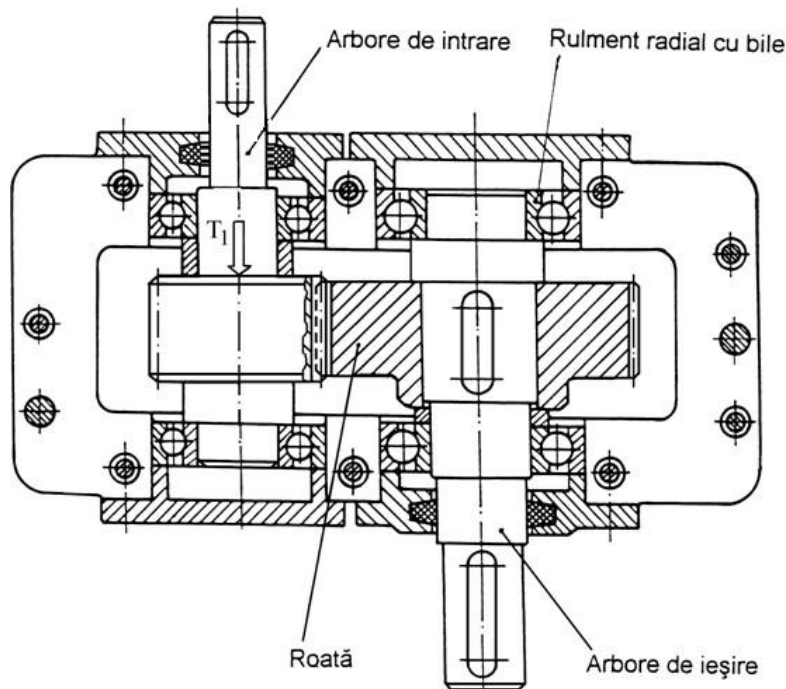
### A. DEFINING THE APPLICATION

#### A.1. Introduction

The operation of gears within speed reducers is also influenced by the stiffness of the shafts on which the gears are mounted, as well as the stiffness of the gear teeth. Identifying the stress and strain fields in these parts, using finite element analysis, leads to drawing useful conclusions regarding the values and distributions of these fields for gear design practice.

#### A.2. Description of the Application

The figure below shows the assembly drawing of a single-stage cylindrical reducer that transmits and modifies the rotational motion and torque with a transmission ratio of  $i = 2$ . Elastic couplings are mounted on the ends of the input and output shafts. The input shaft, which is a common body with the pinion, as well as the output shaft, are supported on radial ball bearings that are radially and axially fixed in the housing subassembly.



#### A.3. Purpose of the Application

This application presents the analysis of the strain and stress fields in the structure of the input pinion shaft. The analysis is performed to establish the highly-stressed zones and, on the other hand, to show that the material's yield limit is not reached under maximum load conditions. The modeling and analysis are performed for the input shaft-pinion common body, which has five cylindrical sections with the following sets of geometric dimensions (diameter, length), expressed in mm: (36, 60), (40, 40), (51, 50), (48, 15), (40, 15).

A keyway with dimensions  $b = 8 \text{ mm}$ ,  $h = 3,5 \text{ mm}$ ,  $l = 50 \text{ mm}$  is machined on the outer section. On the fourth section, straight teeth are machined with the parameters: module,  $m = 3 \text{ mm}$ ; number of teeth  $z_1 = 20$ ; addendum coefficient  $h_a^* = 1$ ; clearance coefficient,  $c^* = 0.25$ ; root fillet coefficient,  $r_f^* = 0.38$ ; profile shift coefficients  $x_1 = -0,25$ ,  $x_2 = 0,25$ ; rack angle,  $\alpha_0 = 20$ ; center distance  $a_w = 90 \text{ mm}$ . The torque transmitted by the input shaft is  $T_1 = 22 \text{ Nm}$ . The reducer shafts are made of 40Cr10 alloy steel, heat-treated by hardening and tempering to 250-270 HB.

## 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:

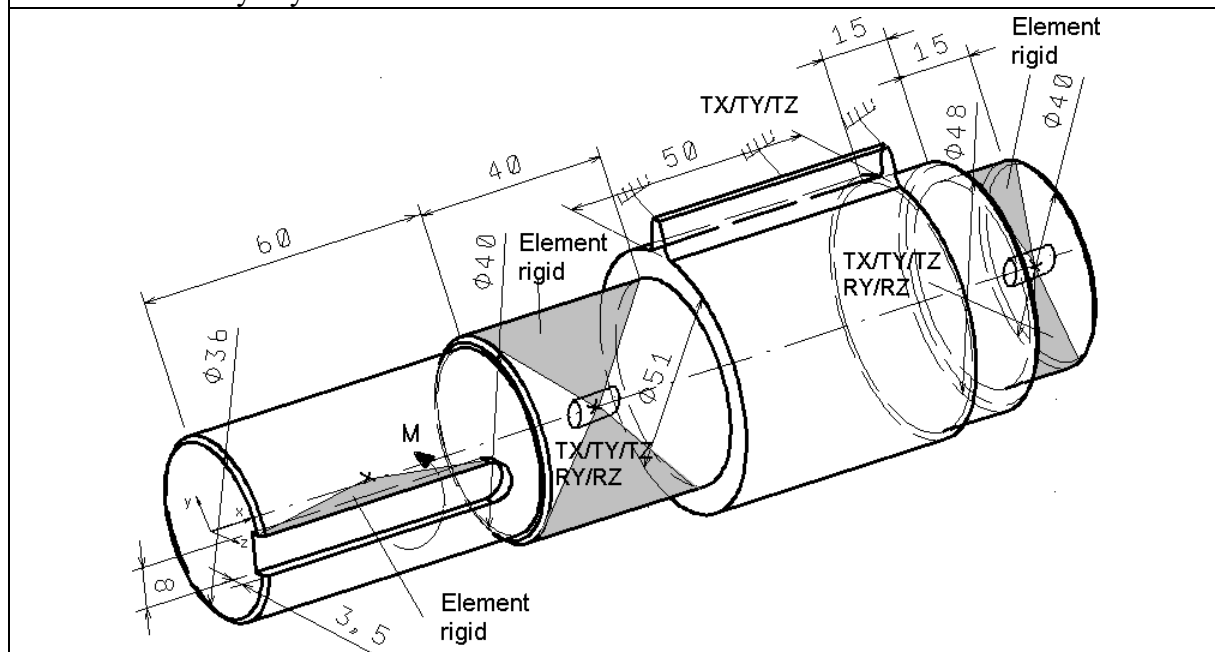
- The geometric shape and dimensions,
- Adjacent elements with high stiffness,
- Constraints induced by connections with adjacent elements,
- External loads,
- Material characteristics.

### B.2. Description of the Analysis Model

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

For the analysis model to behave as in reality in the zones supported by radial bearings, these are replaced with a rigid element each, which connects the axis with the cylindrical surface of the bearing seat. In addition, associated with the master nodes on the axis, a pivot-type constraint element is introduced for each to allow torsional deformation of the shaft. These constraint elements are automatically associated with boundary conditions that involve translation restrictions along the three XYZ coordinate system directions and rotation restrictions about the OY and OZ axes.

Also, for a behavior as close as possible to the real model, considering the high stiffness of the drive coupling and the parallel key, the loading with the moment  $M = T_1$  is done through a rigid element associated with them, which connects the axis with the lateral surface of the keyway.



### B.3. Establishing Material Characteristics

For the finite element analysis, the strength characteristics of the material, 40Cr10 steel, hardened and tempered to 250-270 HB, are:

- Longitudinal modulus of elasticity,  $E = 204,000 \text{ N/mm}$ .
- Transverse contraction coefficient (Poisson's ratio),  $\nu = 0.3$ .

## C. PREPROCESSING THE ANALYSIS MODEL

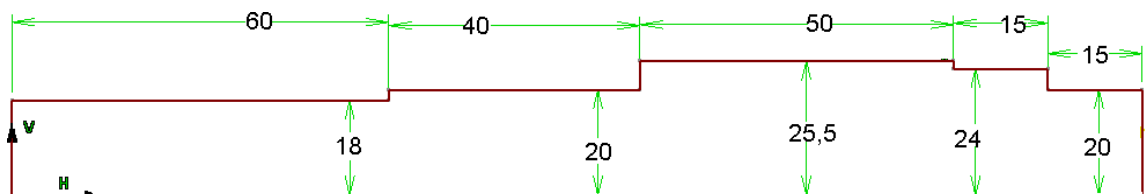
### C.1. Geometric Modeling

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

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

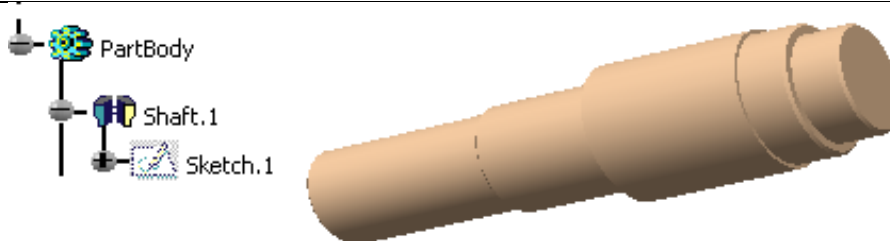
*Generate the reference sketch*

 (Sketcher) → xy plane →  → [draw the shaft axis] →  (Profile) → [draw the shaft contour] →  (Constraint) [successively introduce the dimensions by selecting the lines, using the Ctrl key for multiple selections] →  (Exit Workbench).



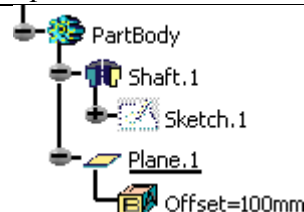
*Generate the solid model*

 (Shaft) → **Shaft Definition**: Profile Selection [select Sketch.1], First angle 360deg,  
 ↵ OK.



*Create the tooth profile plane*

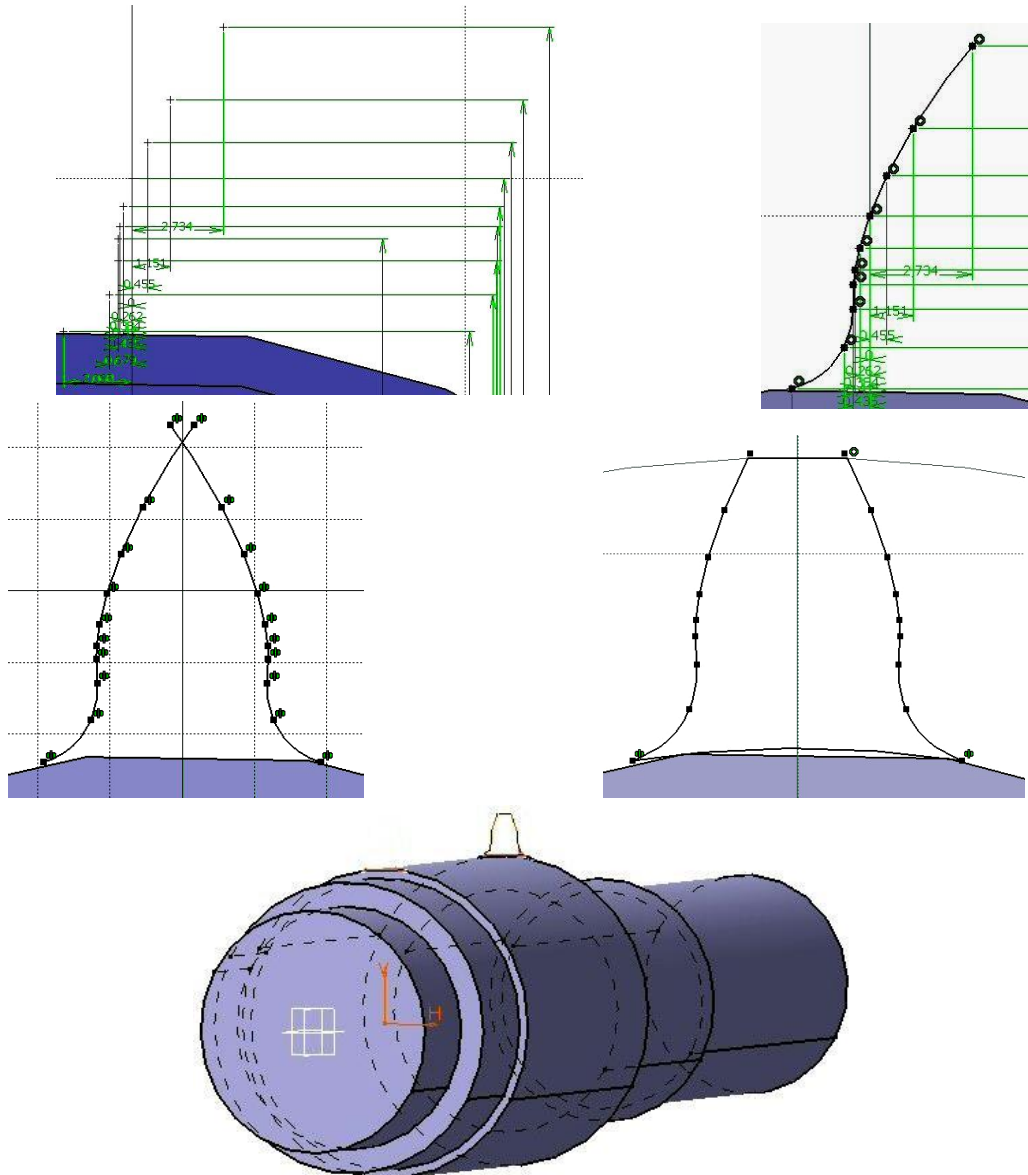
 (Plane) → **Plane Definition**:  
 → Plane Type: Offset from plane  
 [select YZ plane] → Offset: 100 mm  
 ↵ OK.

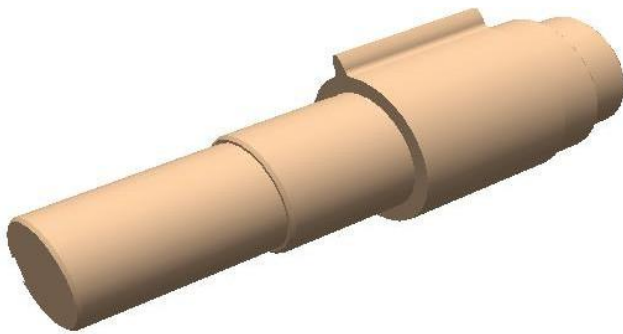
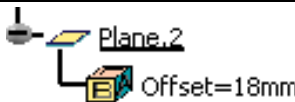
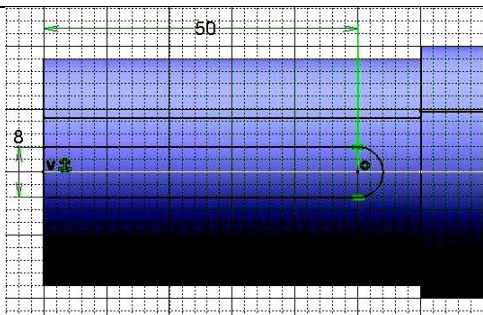
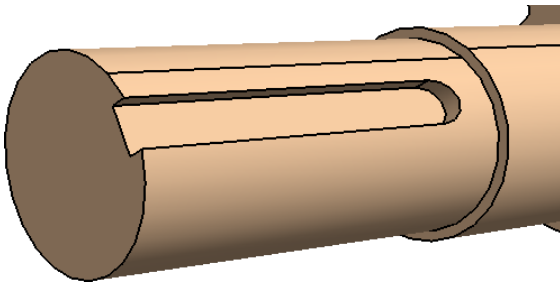
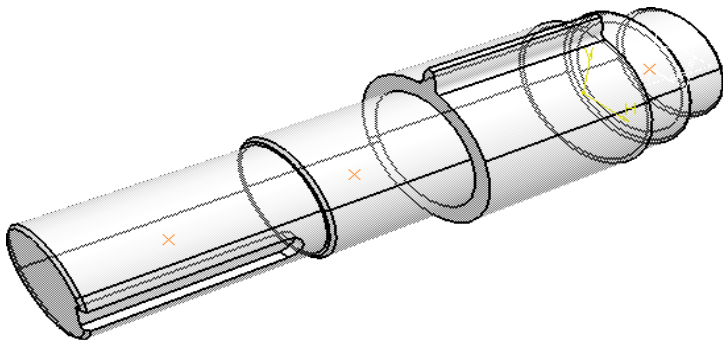
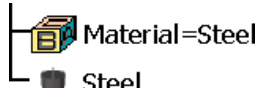


*Generate the tooth profile*

 (Sketcher) → [select Plane.1] →  
 (Point by Using Coordinates) → **Point Definition**: Cartesian H: 2.734476 V: 34.53048,  
 ↵ OK, [similarly, the points (1.1507395; 32.344), (0.455428; 31.0395), (0; 30),  
 (-0.26291; 29.15523), (-0.38445; 28.56465), (-0.42017; 28.1904), (-0.43505; 27.52858),  
 (-0.67886; 26.50773), (-2.06827; 25.41598)] are also generated] (the algorithm for obtaining  
 the coordinates of these points as well as other data necessary for obtaining the tooth profile is  
 presented in Annex 2)

→  (Spline) → [the flank and root fillet line is generated by selecting the previously generated points (the final point is generated by double-selection)] →  (Rotate) → **Rotation Definition** [select the previously generated spline line], Duplicate: Instance(s): 1, ☐ Duplicate mode, [select the center of the HV system], Value: 3.97637 (see Annex 2),  OK →  (Symmetry) → [select the spline line obtained after rotation] → [select the V-axis symmetry line] →  (Project 3D Elements) → [select the root circle, the base circle of the cylinder on which the teeth are generated] →  (Circle) → [generate the tip circle with a diameter of 65 mm (see Annex 2)] →  (Quick Trim),  (Trim),  (Break) → [generate the tooth contour] → Tools → Sketch Analysis → Corrective Actions [corrective actions are activated to obtain a closed profile], Geometry (Profile), Status (Closed), Comment (4 Curves),  (Construction/Standard Elements) → [points P1 ... P10 are transformed into construction points],  Close →  (Exit Workbench).



|                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                 |                                                                                      |                                                                                       |
|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|--------------------------------------------------------------------------------------|---------------------------------------------------------------------------------------|
| Generate the tooth                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                              |                                                                                      |                                                                                       |
|  (Pad) → <b>Pad Definition:</b> First Limit Type: ↓ Dimension, Length: 50; Profile Selection: [select the sketch associated with the tooth profile, Sketch.2]; Reverse Direction, ↓OK.                                                                                                                                                                                                                                                                                                                                                                                                         |    |                                                                                       |
| Generate the keyway plane                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                       |                                                                                      |                                                                                       |
|  (Plane) → <b>Plane Definition:</b> Plane Type: Offset from plane [select zx plane], Offset: 18 mm ↓OK.                                                                                                                                                                                                                                                                                                                                                                                                                                                                                        |    |                                                                                       |
| Generate the keyway                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                             |                                                                                      |                                                                                       |
|  (Sketcher) → [select the Plane.2 specification from the specification tree] →  (Constraint) [successively introduce the dimensions by selecting the lines, using the Ctrl key for multiple selections],  (Exit Workbench).<br> (Pocket) → <b>Pocket Definition:</b> Selection [select the keyway contour, Sketch.3], Depth: 3,5 mm, ↓OK. |                                                                                      |                                                                                       |
|                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                              |  |                                                                                       |
| Generate the chamfers                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                           |                                                                                      |                                                                                       |
|  (Chamfer) → <b>Chamfer Definition:</b> Length.1: 1 mm, Angle: 45 deg, Object(s) to chamfer: 4 elements [select the 4 edges of the end sections], ↓OK.                                                                                                                                                                                                                                                                                                                                                                                                                                       |                                                                                      |                                                                                       |
| Generate the points associated with the master nodes of the rigid elements                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                      |                                                                                      |                                                                                       |
|  (Point) → <b>Point Definition:</b> Point type: ↓ Coordinates X 30, Y 0, Z 0; Reference Point: Default (Origin) ↓OK. [repeat this sequence for points (80,0,0) (172.5,0,0) ].                                                                                                                                                                                                                                                                                                                                                                                                                |  |                                                                                       |
| C.2. Modeling the Material Characteristics                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                      |                                                                                      |                                                                                       |
| [Select the PartBody specification in the tree structure] →  (Apply Material) → <b>Library (ReadOnly):</b> Metal, Steel → Apply Material, ↓OK.                                                                                                                                                                                                                                                                                                                                                                                                                                               |                                                                                      |  |

[Select Steel from the tree structure] → **Properties:** Feature Properties, Feature Name: OLC45; Analysis, Young Modulus 204000 N/mm<sup>2</sup>, Poisson Ratio 0,3 → Apply, ↵OK.



### C.3. Finite Element Modeling

*Activate the finite element analysis module*

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

#### Observation

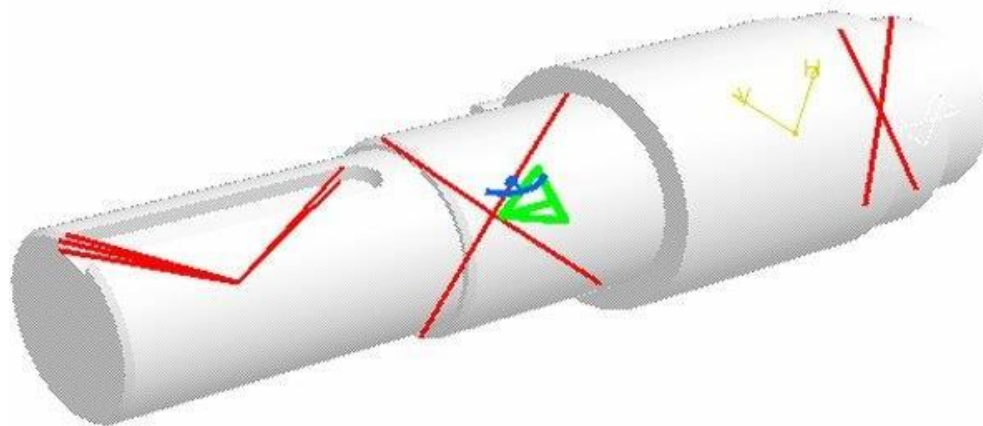
By activating the Generative Structural Analysis module, the solid model is implicitly set for discretization into linear tetrahedral finite elements (OCTREE Tetrahedron). Optionally, to modify the discretization characteristics (finite element type and its dimensions, globally or locally), the following sequence can be used. [Select with the mouse the specification OCTREE Tetrahedron Mesh: Part1, in the tree structure, or the symbol on the solid model structure ] → OCTREE Tetrahedron Mesh: Size: 5 mm (modifications can also be made for other parameters, if necessary), ↵OK.

### C.4. Modeling Geometric Constraints

*Generate the virtual rigid elements*



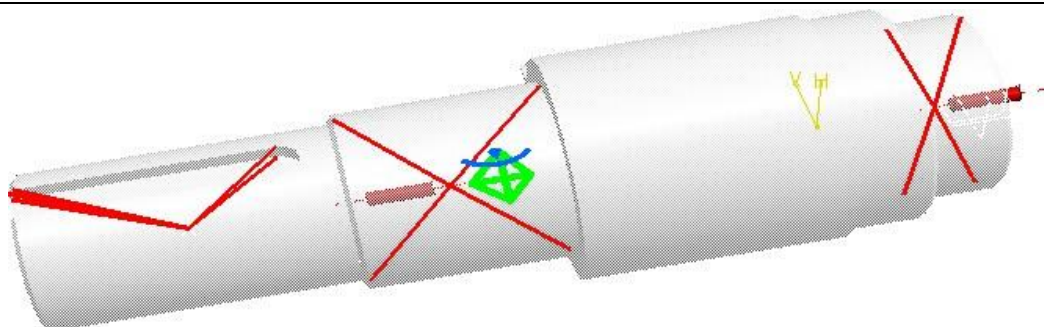
(Rigid Virtual Part) → **Rigid Virtual Part:** Handler: [select the first point on the axis], Supports: [select lateral surface of keyway], ↵OK. [Repeat this sequence for the two rigid elements between the points on the axis and the bearing mounting surfaces].



*Generate the pivot-type constraints*

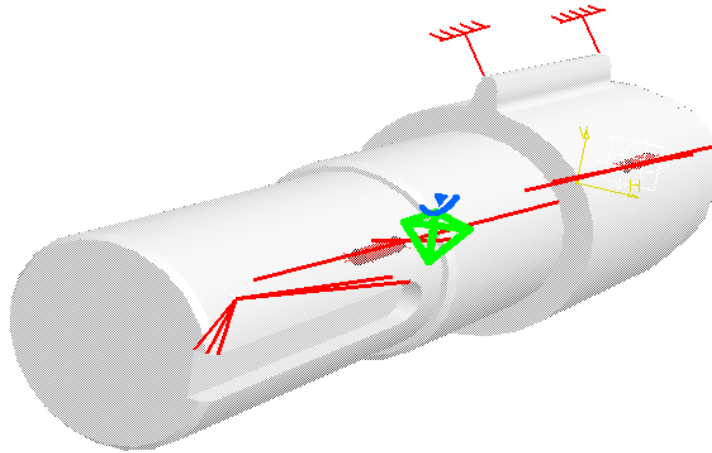


(Pivot) → **Pivot:** Supports [select rigid element, Rigid Virtual Part.2], Axis System Type: Global, Released Direction X 1, Y 0, Z 0; ↵OK. [Repeat this sequence to generate the pivot associated with the rigid element, Rigid Virtual Part.3].



### Introducing displacement restrictions

 (Clamp) → **Clamp**: Supports [select the tip edge of the tooth], ↵OK.

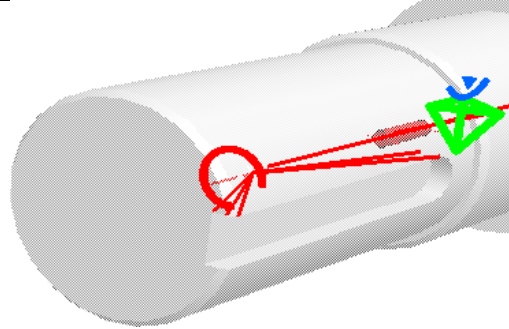


## C.5. Modeling Loads

### Introducing the torque



(Moment) → **Moment**: Supports [select the virtual rigid element associated with the key, Rigid Virtual Part.1]; Moment vector X 22 Nm, 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

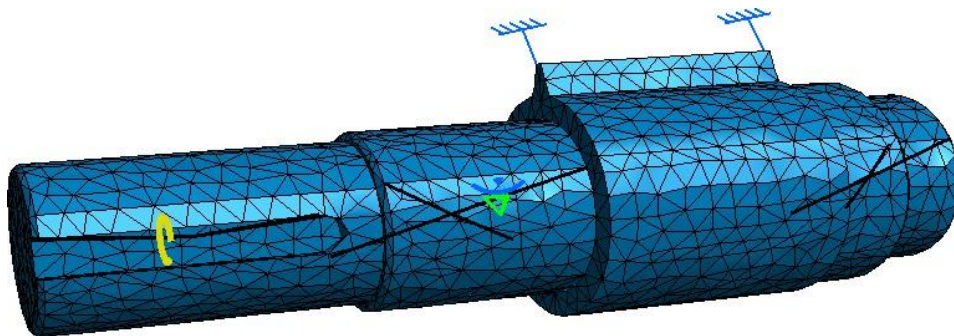
### E.1. Visualizing the Deformed State/Animation



(Deformation).



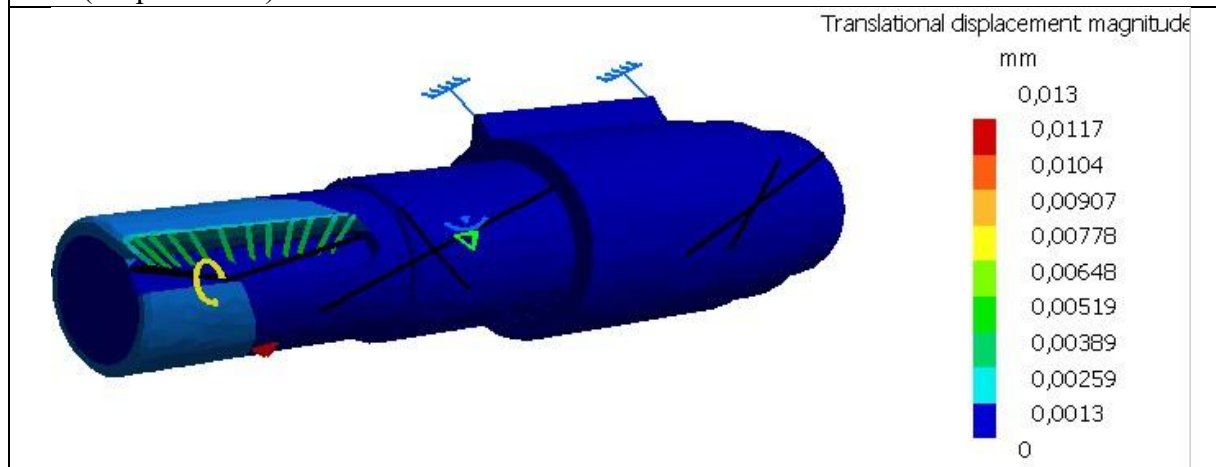
(Animate) → **Animate**: Steps number: 5 (other parameters can be modified), ↵Close.



### E.2. Visualizing the Displacement Field



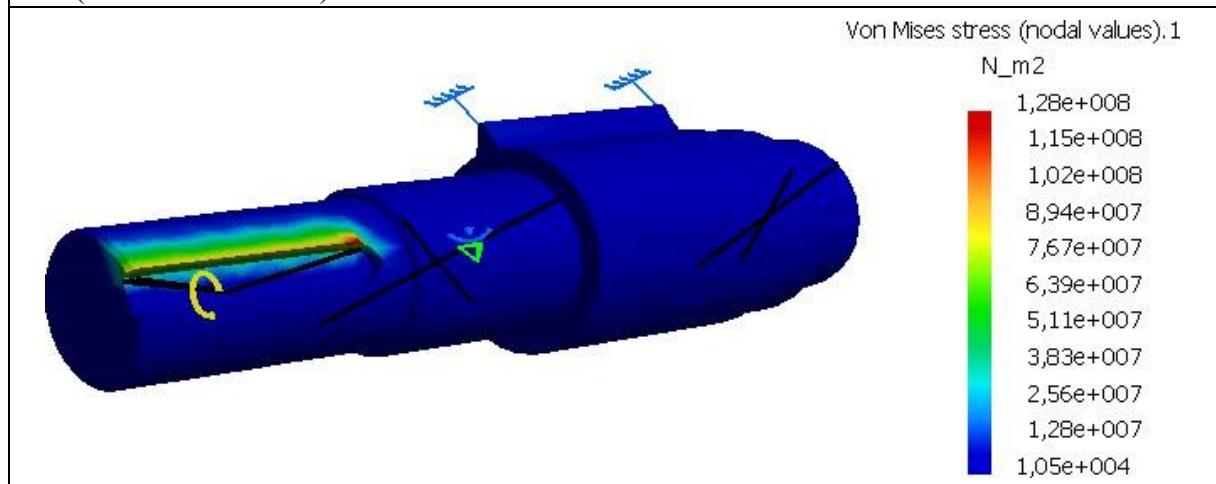
(Displacement).



### E.3. Visualizing the Stress Field



(Von Misses Stress).



## F. CONCLUSIONS

From the analysis of the displacement fields, it is evident that the analyzed element is a rigid structure (maximum displacements are small, 0.013 mm, in the outer area of the keyway). Due to this characteristic (high torsional stiffness), the influence of the shaft's deformations on the gear's operation is reduced. The maximum Von Mises stresses, approx. 128 MPa, in the keyway area, indicate the shaft's strength is at the admissible limit.

## G. EXERCISE

Perform a finite element analysis for the output shaft of the reducer from figure A.2. The geometric dimensions (diameter, length) of the six sections, expressed in mm, are: (50, 60), (60, 20), (70, 25), (85, 60), (95, 15), (70, 15). The keyways have dimensions:  $b = 12$  mm,  $h = 5$  mm,  $l = 50$  mm. The torque transmitted by the output shaft is  $M = 44$  Nm. The output shaft is made of OLC 45 steel, heat-treated by hardening and tempering to 250-270 HB.