



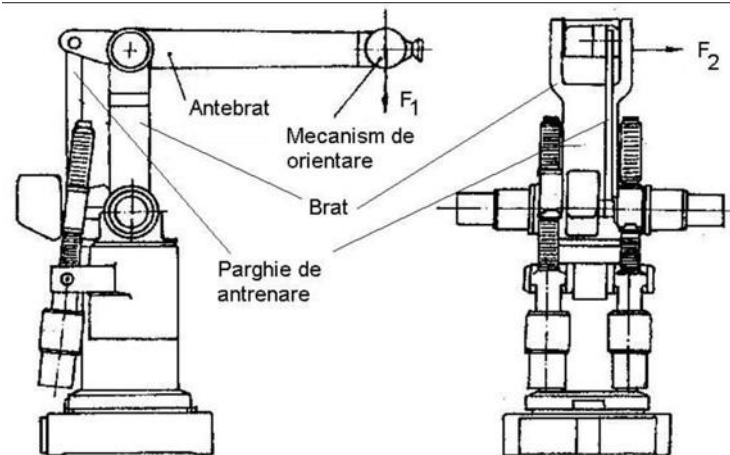
LINEAR STATIC ANALYSIS OF THE INDUSTRIAL ROBOT ARM

INTRODUCTION

The structures of the guiding mechanisms of the industrial robots, in view of their good behavior, are constructively reduced in size and increased stiffness. In addition, their geometric configurations, usually spatial, are complex, dictated by the construction and mounting conditions and the need for reduced momentum of inertia. Thus, following static analyzes of such structures, which involve the identification of displacement and tension fields, it is mainly intended to determine the values of displacements, rigidities and equivalent stresses for subsequent comparison of their values with the permissible limits, but and for their use in subsequent dynamics calculations. The finite element analyzes of these structures are carried out successively, starting from structures obtained by classical calculations, and ending with step-by-step changes with the optimum solution that assumes minimum mass and maximum stiffness under acceptable adhesion conditions.

DESCRIPTION OF THE APPLICATION

The diagram below shows the constructional diagram of an articulated industrial robot, in which the main structural parts (motor couplers, drive elements and support elements) of the guiding mechanism can be identified. Of particular importance in terms of construction, with major influences on the dynamic and precision performance of the industrial robot, are the two supporting elements, the forearm and the arm. Unlike the arm which is driven directly from the motor coupler via a reducer servo system, the arm is trained by means of a parallelogram mechanism of a drive train with a reducer which is fixed to the supporting structure of the guiding structure.



PURPOSE OF THE APPLICATION

In this application we present the static analysis of the mechanical fields in the forearm structure of the abovementioned robot guiding mechanism. This element, with a single-block structure, made of aluminum alloy has the following dimensions: $L_1 = 250$ mm, $L_2 = 800$ mm, $R = 80$ mm, $R_1 = 70$ mm, $R_2 = 15$ mm, $R_4 = 25$ mm, $R_5 = 60$ mm, $a = 5$ mm, $b = 120$ mm, $c = 80$ mm, $e = 60$ mm. The forearm structure for finite element analysis is loaded with forces $F_1 = 1000$ N and $F_2 = 80$ N, representing the equivalent dynamic loads corresponding to the manipulated object and the orientation mechanism, reduced to the axis of the bore of the orientation mechanism.

FEA Application A.2.3

STATIC ANALYSIS OF AN INDUSTRIAL ROBOT ARM STRUCTURE

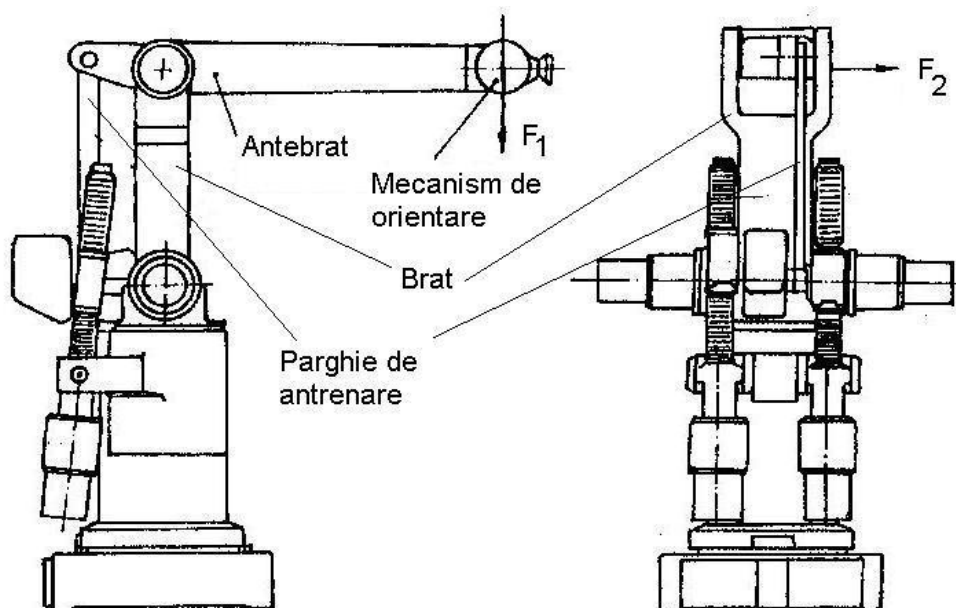
A. DEFINING THE APPLICATION

A.1. Introduction

The structures of industrial robot guidance mechanisms, for good operational behavior, are constructively designed with low masses and high stiffness. In addition, their geometric configurations, which are usually spatial, are complex, dictated by construction and assembly conditions and the need for reduced moments of inertia. Therefore, following static finite element analyses of such structures, which involve identifying displacement and stress fields, the main goal is to determine the values of displacements, stiffness, and equivalent stresses for subsequent comparison of these values with admissible limits, but also for their use in subsequent dynamic calculations. The finite element analyses of these structures are performed successively, starting from structures obtained through classical calculations, and ending, through step-by-step modifications, with the optimal solution that involves minimum mass and maximum stiffness under admissible strength conditions.

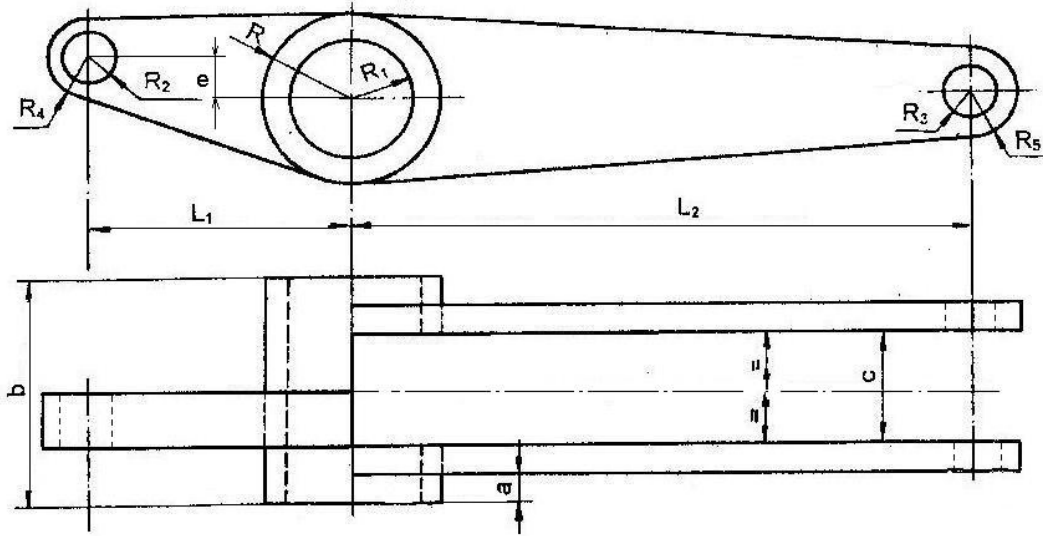
A.2. Description of the Application

The figure below presents the structural-constructive scheme of an articulated industrial robot, in which the main constructive subassemblies (motor joints, driving elements, and support elements) of the guidance mechanism can be identified. Of particular importance from a constructive point of view, with major influences on the dynamic and precision performance of the industrial robot, are the two support elements: the forearm and the arm. Unlike the arm, which is driven directly from the motor joint through a servo-system with a reducer, the forearm is driven via a parallelogram mechanism by a drive servo-system with a reducer that is fixed on the support housing of the guidance structure.



A.3. Purpose of the Application

This application presents the static analysis of the mechanical fields in the structure of the forearm from the guidance mechanism of the robot above. This element, with a monobloc structure, made of aluminum alloy (duralumin), has dimensions: $L_1 = 250$ mm, $L_2 = 800$ mm, $R = 80$ mm, $R_1 = 70$ mm, $R_2 = 15$ mm, $R_3 = 45$ mm, $R_4 = 25$ mm, $R_5 = 60$ mm, $a = 5$ mm, $b = 120$ mm, $c = 80$ mm, $e = 60$ mm. The forearm structure, for the finite element analysis, is loaded with the forces $F_1 = 1000N$ and $F_2 = 80N$, which represent the equivalent dynamic loads corresponding to the manipulated object and the orientation mechanism, reduced to the axis of the orientation mechanism's bore.



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:

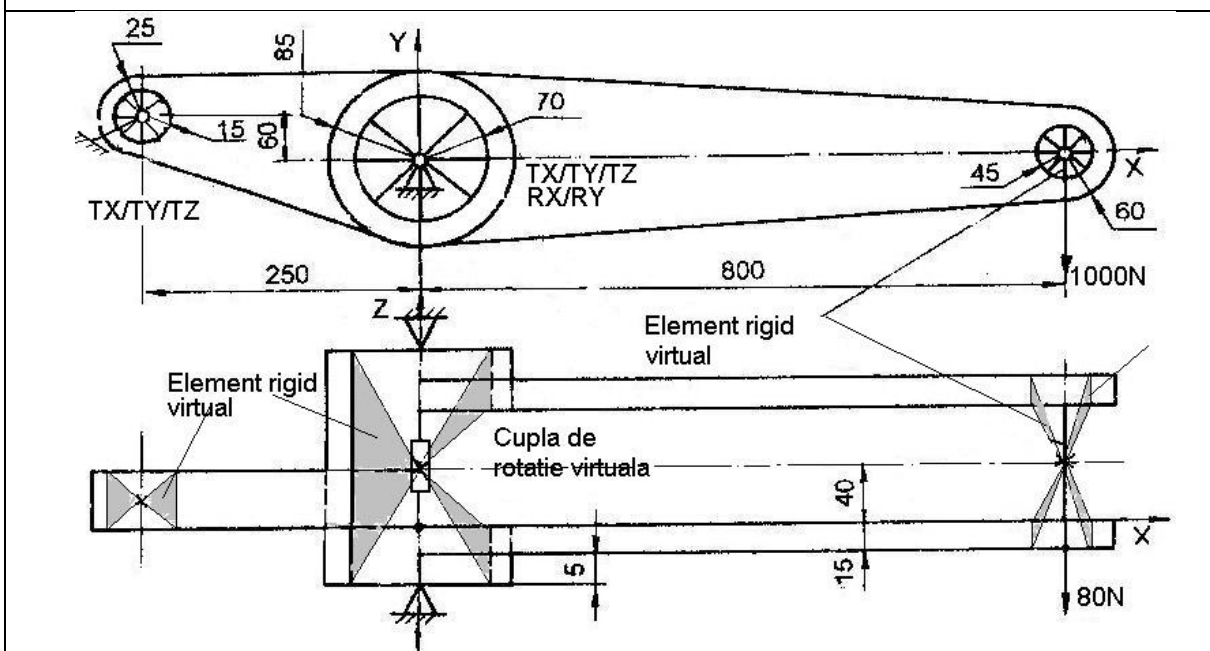
- The geometric shape and dimensions,
- Adjacent virtual rigid elements,
- Constraints induced by connections with external elements,
- Necessary virtual joints,
- External loads,
- Material characteristics.

B.2. Description of the Analysis Model

The figure below shows the detailed geometric shape and dimensions of the forearm to be analyzed.

The simulation of the forearm's connection with the driving lever is done via a virtual rigid element whose master node is fixed by canceling translations relative to the global coordinate system. The connection with the arm is also modeled by a virtual rigid element, but one whose master node is joined with a virtual rotation joint with an axis common to that of the bore in the median zone.

The loading in the frontal plane with the external force F_1 is achieved, considering that bearings are mounted in the connection bores with the orientation mechanism, by equating it with forces distributed on the semi-circumference of the bore according to a parabolic law (the bearings have flexible rings). In the axial plane, loading with the force F_2 is done via a rigid element that joins the two sides of the forearm, acting on the master node positioned in the center of symmetry of the arms' bores.



B.3. Establishing Material Characteristics

For the finite element analysis, the strength characteristics of the material, aluminum alloy (duralumin), are:

- Longitudinal modulus of elasticity, $E = 73000 \text{ N/mm}^2$
- Transverse contraction coefficient (Poisson's ratio), $\nu = 0.18$

C. PREPROCESSING THE ANALYSIS MODEL

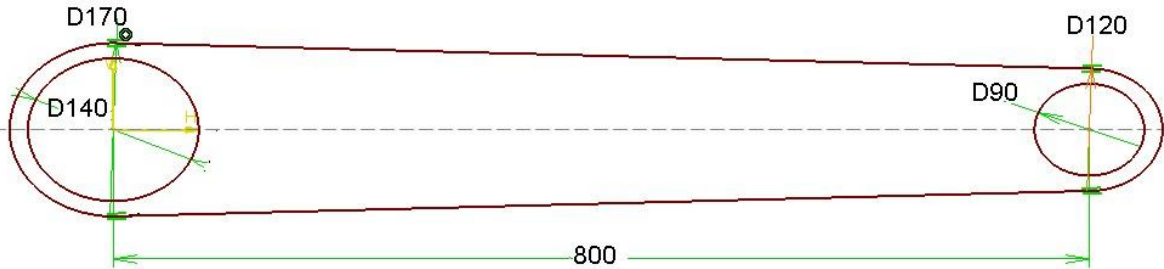
C.1. Geometric Modeling

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

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

Generate the reference sketch of the arm

 (Sketcher) → XY plane → (Circle)  → [draw the four circles that describe the fork arms] →  (Constraint) [successively introduce the dimensions by selecting the entities, using the Ctrl key for multiple selection] →  (Bi-Tangent Line) → [close the outer contour with tangent lines] →  (Quick Trim) → [delete the inner parts of the outer circles] →  (Exit Workbench).



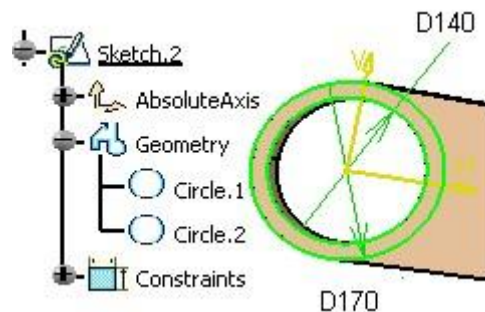
Generate the arm

 (Pad) → Pad Definition: Length: 15 mm, Selection: [select, Sketch.1], .



Generate the reference sketch for the outer reinforcement of the central bearing

 (Sketcher) → [select the outer face of the arm] → (Circle) → [draw two circles in the bearing area] →  (Constraint) → [successively introduce the diameter values] →  (Exit workbench).

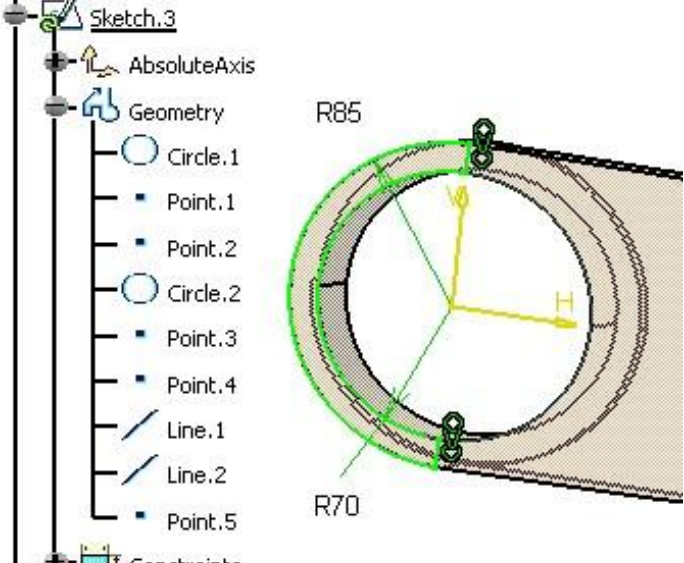


Generate the outer reinforcement of the bearing

 (Pad) → Pad Definition: Length: 5 mm, Selection: [select, Sketch.2], .



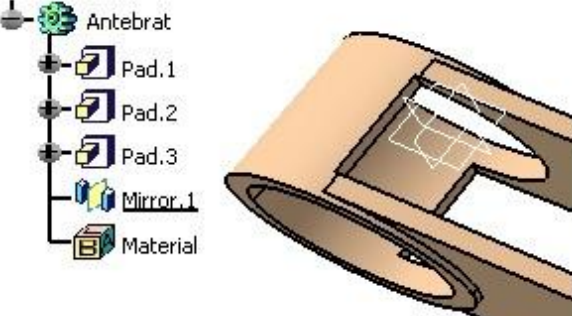
Generate the reference sketch for the inner-zone of the central bearing

 (Sketcher) → [select the inner face of the arm] →  (Arc) → [draw two semicircles covering quadrants II and III] →  (Constraint) → [successively introduce the radius values] →  (Line) → [draw two-line segments connecting the ends of the circular arcs] →  (Exit Workbench).	
--	--

Generate the inner zone of the central bearing

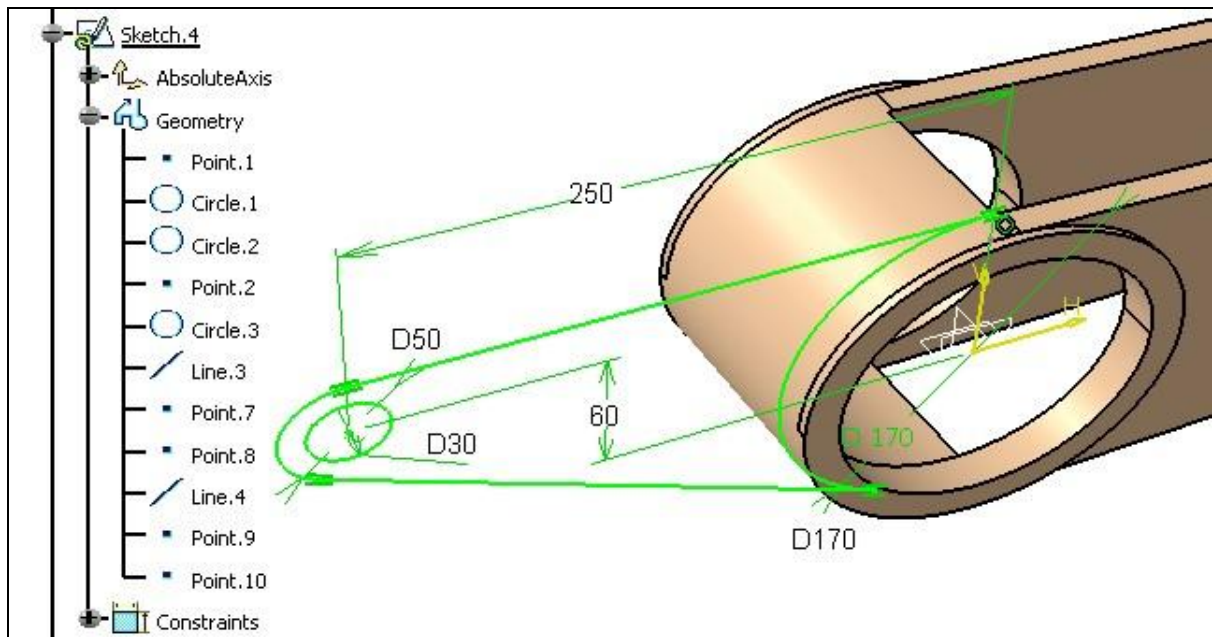
 (Pad) → Pad Definition: Length: 40 mm, Selection: [select, Sketch.3], ↵OK.	
---	---

Generate the main body

[Select the Antebrat half-body with the mouse] →  (Mirror) → Mirror Definition: Mirroring element: [select semicircular bearing surface, parallel to the XY plane], ↵OK.	
---	--

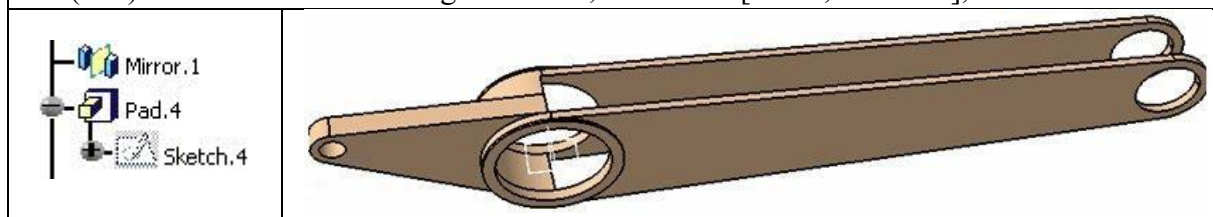
Generate the reference sketch of the console

 (Sketcher) → [select the inner face of the front arm] →  (Circle) → [draw three circles] →  (Constraint) → [successively introduce the dimensions by selecting the entities, using the Ctrl key for multiple selections] →  (Bi-Tangent Line) → [close the contour with two lines tangent to the circles with diameters of 100 and 170] →  (Quick Trim) → [delete the circular arc with a diameter of 170] →  (Exit Workbench).



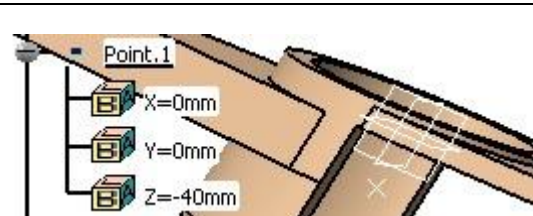
Generate the console body

(Pad) → Pad Definition: Length: 40 mm, Selection: [select, Sketch.4], OK.



Generate the points associated with the centers of the rigid elements

(Point) → Point Definition: Point type:
↓Coordinates X 0, Y 0, Z -30; Reference
Point: Default (Origin) OK. [Repeat this
sequence for the points (-250,60, -20),
(800,0,-40)].



C.2. Modeling the Material Characteristics

[Select the **Antebrat** specification] → (Apply Material)
→ Library (Read-only): Metal, Aluminum, Apply Material;
 OK.

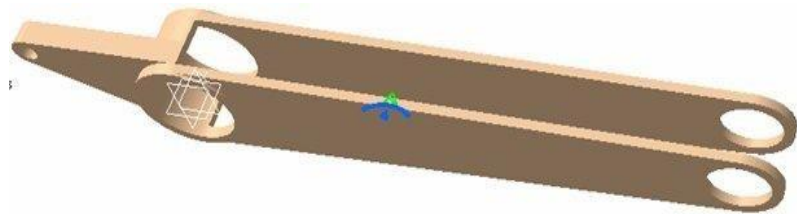
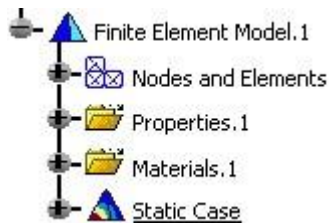
[Select the Aluminum specification from the tree structure] →
Properties: Feature Properties, Feature Name: DURAL;
Analysis, Young Modulus 73000 N/mm², Poisson Ratio 0.18;
Apply; OK.



C.3. Finite Element Modeling

Activate the finite element analysis module and set discretization parameters

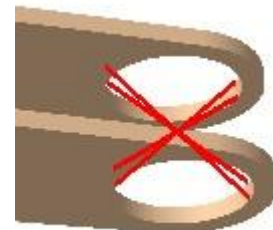
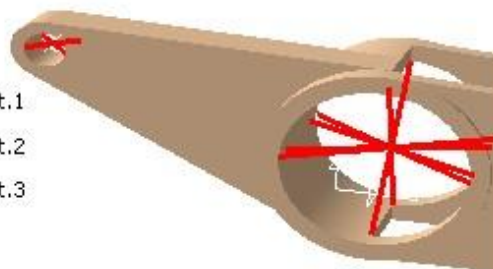
Start → Analysis & Simulation → Generative Structural Analysis → New Analysis Case Static Analysis, ↵ OK. [Select the  OCTREE Tetrahedron Mesh specification with the mouse in the tree structure, or the symbol on the solid model structure] → OCTREE Tetrahedron Mesh: Size 20,  Absolute Sag: 10, Element type:  Parabolic, ↵ OK.



C.4. Modeling Geometric Constraints

Generate the virtual rigid type elements

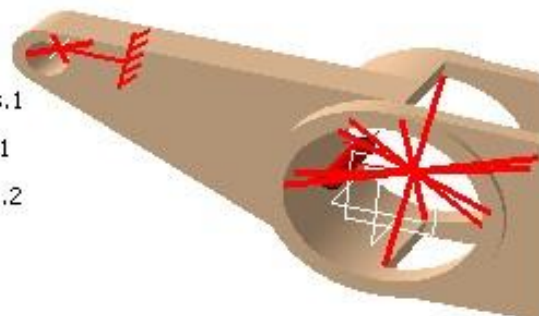
 (Rigid Virtual Part)  → Rigid Virtual Part: Handler: [select the first point on the axis, Point.1], Supports: [select the inner cylindrical surface with the mouse], ↵ OK. [Repeat this sequence for the other two rigid elements].

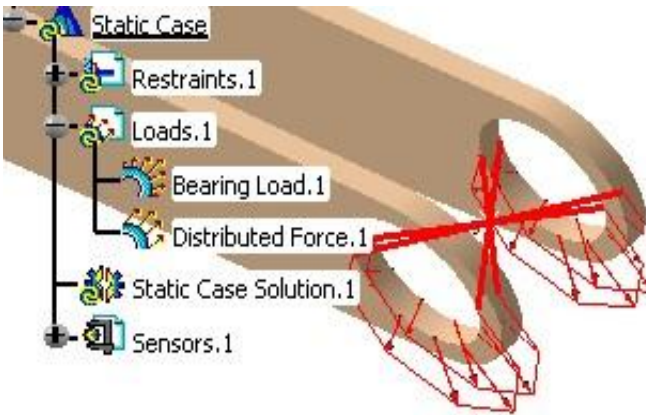


Introduce displacement restrictions and generate the rotation joint

 (Pivot) → Pivot: Supports [select the virtual rigid element from the bearing area], Axis System Type: Global, Released Direction X 0, Y 0, Z 1; ↵ OK.

 (Clamp) → Clamp: Supports [select the virtual rigid element associated with the console bore], ↵ OK.

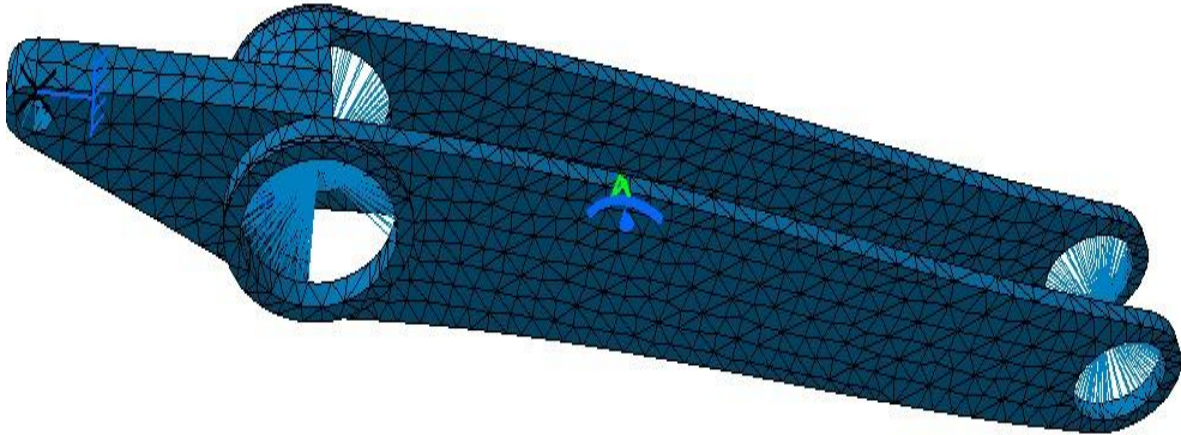


C.5. Modeling Loads	
<i>Introducing the forces</i>	
 (Bearing Load) → Bearing Load: Supports [select the surfaces where force F1 acts, using the Ctrl key]; Orientation: Radial; Variation: Sinusoidal; Force vector X 0N, Y – 1000N, Z 0N; ↵ OK.	
 (Distributed Force) → Distributed Force: Supports [select the virtual rigid element associated with the orientation mechanism]; Force vector: X 0 N, Y 0 N, Z 80 N; ↵ OK.	

D. SOLVING THE FINITE ELEMENT MODEL

D.1. Launching the Calculation Module
 (Compute) → Compute: ↓ All, ↵ OK → Computation Resources Estimation, ↵ Yes → Computation Status.

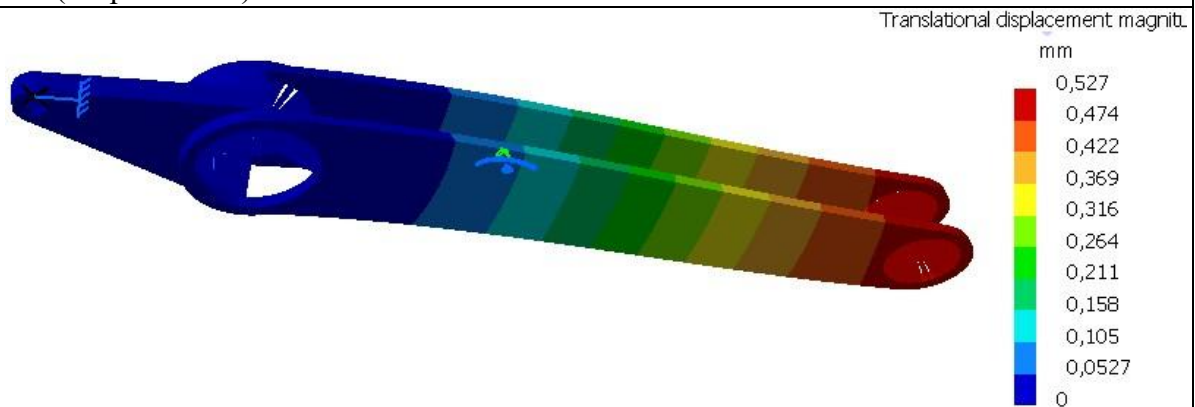
E. POSTPROCESSING THE RESULTS

E.1. Visualizing the Deformed State/Animation
 (Deformation).
 (Animate) → Animate: Steps number: 5 (other parameters can be modified to improve visualization), ↵ Close.

Observation To modify the displacement amplitude 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).

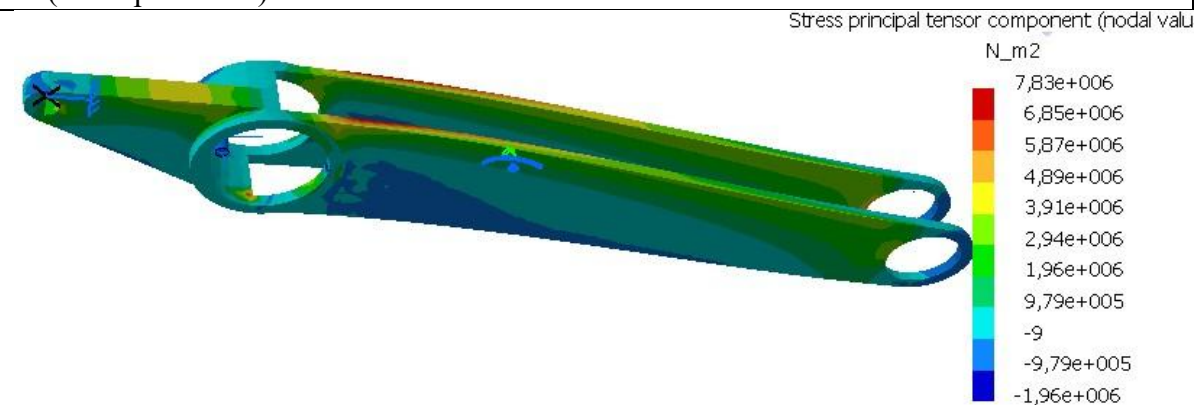


E.3 Visualizing the Stress Field

Visualizing the Principal Stress Field



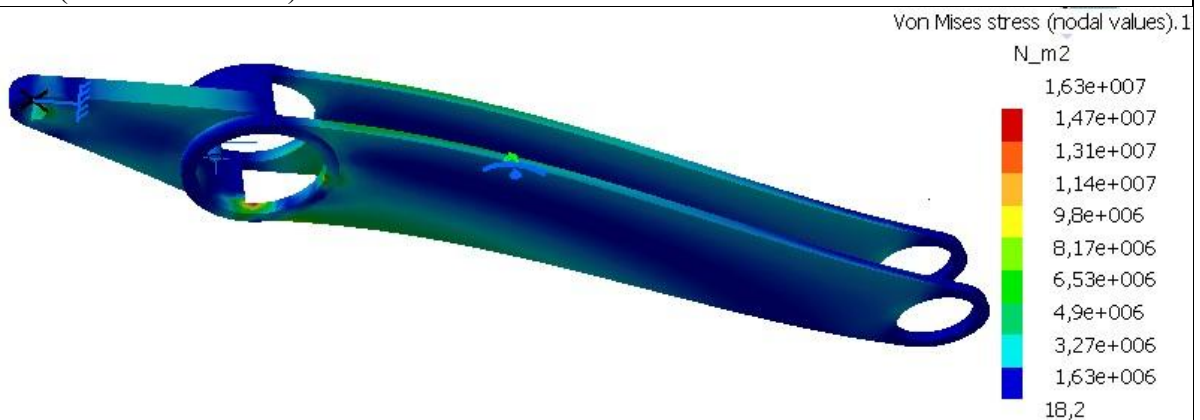
(Principal Stress).



Visualizing the Von Mises Stress Field



(Von Mises Stress).



F. CONCLUSIONS

For the design of the main elements of industrial robot guidance systems, the study of mechanical fields in their structures using high-performance programs based on FEA is irreplaceable.

The elastic displacement values associated with the points in the connection-zone of the forearm with the orientation mechanism (for example, in this application, the maximum displacement is 0.527 mm) are of particular importance for the robot's precision, which in specific applications are limited to admissible values.

If this condition is not met, the shape and dimensions of the forearm structure and/or the material used are modified in order to stiffen it while keeping its mass at the minimum possible values. From the analysis of the stress fields (7.83 MPa, principal stress; 16.3 MPa, maximum Von Mises stress), it is evident that the forearm structure is oversized.

G. EXERCISE

Determine the displacement and stress fields in the structure of the robot's guidance mechanism arm (fig. A.2), shown in the figure below. The dimensions of this element, expressed in mm, are determined based on the parameter, $a=60\text{mm}$. The arm's loads will be determined based on the forces loading the guidance mechanism presented in fig. A.2. The arm to be analyzed has a structure obtained by casting from steel with a longitudinal modulus of elasticity, $E=203950\text{ MPa}$, and Poisson's ratio, $\nu = 0.35$.

