Application: AEF-A.16

Static analysis of bar mechanisms

KEY WORDS
Linear Static Analysis, Plane Geometric Model, Plane Voltage State, Linear Material, 1D Finite Element, Linear Finite Element, Machine Element, Mechanical Subassembly, Bar Mechanisms, Joints

CONTENT

A. PROBLEM DESCRIPTION

A.1 Introduction

In various finite element analysis (FEA) applications it is necessary to model not only a single part, but also a whole mechanism, including the joints between its elements.

Complex plane or spatial mechanisms can be reduced to bar mechanisms (levers) and toothed mechanisms (gears and racks). The methods of studying the complex mechanisms with bars and gears are very diverse, especially in the field of kinematics, starting from the hypothesis of the rigidity of the components. This paper aims to study the behavior of the elements of a bar mechanism, taking into account their elastic behavior.

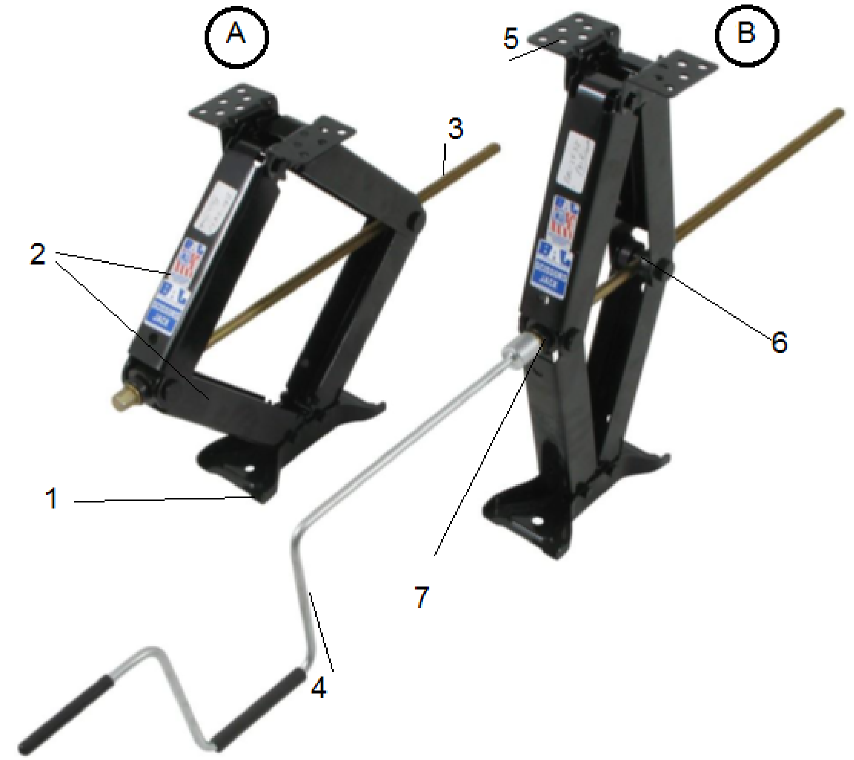
A.2 Application description

From a constructive point of view, the jack in the figure below is a flat mechanism consisting of four bars of various sections mounted on a support, a screw and a nut that make a helical coupling. The analyzed jack can operate within two positions: A - start lifting and B - maximum lifting.

The vertical movement of the upper plate 5 is determined by the change of the positions of the side segments 2, mounted by means of support joints 1, due to the axial movement of the nut 6. The screw 3 is actuated by means of the crank 4.

All the joints between the segments 2 and the support 1, between the segments 2 and the upper plate 5, as well as those between the lower and the upper segments are flat rotational couplings.

By rotating the crank 4, thanks to the nut 6, the side segments of the jack tend to approach each other, generating vertical movement, thus lifting the vehicle.



A.3 Application goal
The application aims to determine the maximum values of the fields of displacements, deformations, internal stresses produced in operation on the component elements. For this analysis, the use of one-dimensional elements was considered due to the simplicity of the geometric construction, the ease of modifying the profile of the studied elements but also the main objective - to use the joints in the study with finite elements.

B. PREPARATION OF THE MODEL FOR ANALYSIS

B.1 The model definition

In order to draw up the finite element analysis model associated with the present 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.

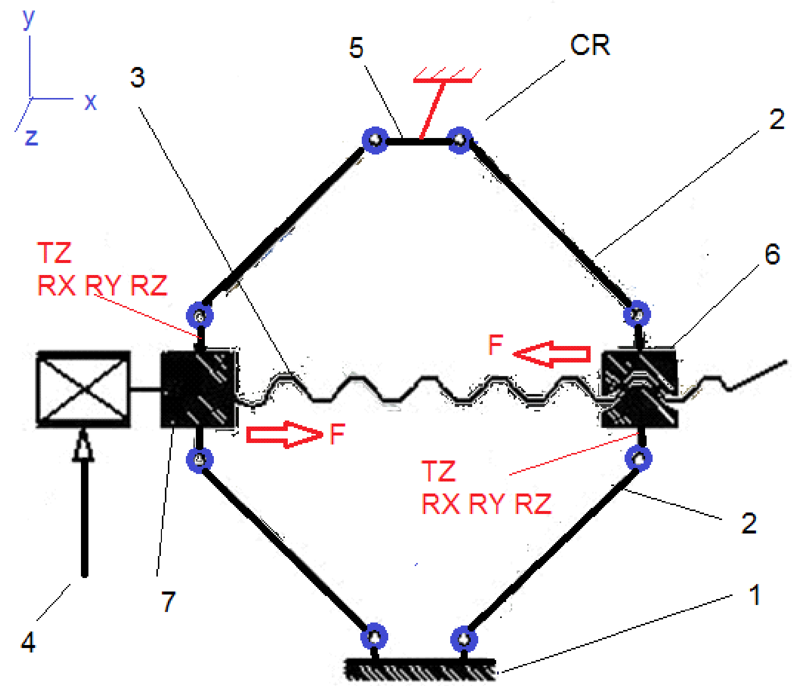
B.2 The analysis model description

The dimensions of the studied mechanism, respectively the lengths of the segments and their sections, are presented in chapter C.3.2, C.3.3 and C.3.4, these being, on the one hand, taken from the specialized literature and, on the other hand, imposed from constructively so that the problem is determined.

The construction of the segments 2, of the vertical zones afferent to the guide bush (7) and the nut 6, the upper plate 5 are constructed in the form of one-dimensional bars.

The connections between these bars are made with simple rotating joints (CR in the adjacent drawing). In addition, the vertical side segments will have translational movements only in the plane of the mechanism, without rotations.

The external loads, generated by the mass of the raised vehicle, are shaped by the rigid fixing of the support 1 and the upper plate 5, and the forces generated by the screw act on the horizontal axis, on the lateral vertical segments, in the direction of approaching the lateral segments.



B.3 Characteristics of the material

For finite element analysis the strength characteristics of the material, S235 steel (equivalent to OL 37) are:

- modulus of longitudinal elasticity, E = 204,000 N / mm^2;
- transverse contraction coefficient (Poisson), ν = 0.3.

C. PREPROCESSING OF FEA MODEL

C.1 Creating and saving the project

Create of the project

Toolbox: ↵ ⊟ Analysis Systems → ↵↵ Static Structural (the subproject window appears automatically); → [the name can be changed Static Structural în *Cric auto*].

Problem type setting (3D)

A : ↳ Geometry → ↵ Properties → Properties of Schematic A3: Geometry, ⊟ Advanced Geometry Options : ↵ Analysis Type , [select from list ↵, ↵ 3D] → [close window ↵ ✕].

Saving the project

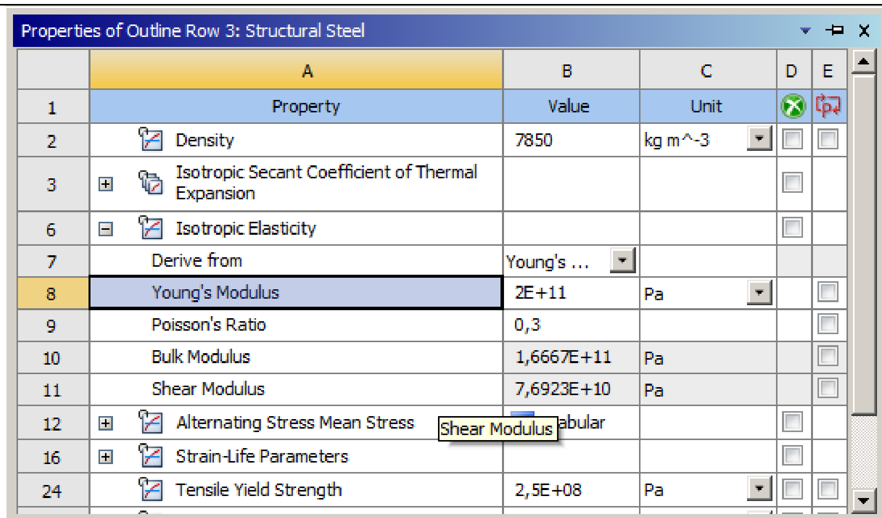
↵ Save As... → Save As, File name: [input name, *Cric*] → ↵ Save.

C.2 Modelling of material 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 column B (Unit) value, 204000] → ↵ Update Project → ↵ Return to Project (the other parameters remain the default).

If the window Properties of Outline Row 3: Structural Steel is not visible, the Outline and Properties or Reset Workspace options will be activated in the View menu, then the *Engineering Data Sources* command .


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	2E+11	Pa		
9	Poisson's Ratio	0,3			
10	Bulk Modulus	1,6667E+11	Pa		
11	Shear Modulus	7,6923E+10	Pa		
12	Alternating Stress Mean Stress	Shear Modulus abular			
16	Strain-Life Parameters				
24	Tensile Yield Strength	2,5E+08	Pa		

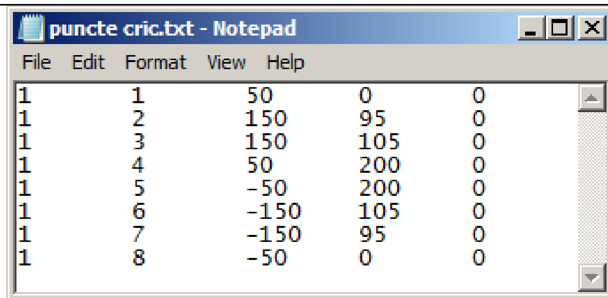
C.3 Geometric modelling

C.3.1 Loading DesignModeler (DM) module

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

C.3.2 Generating points

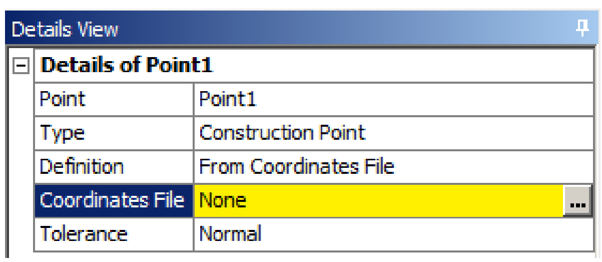
Using *Notepad* program generate a txt file with the form shown next to it, which will be saved as *puncte cric.txt*. It will contain the coordinates of the construction points of the jack.


puncte cric.txt - Notepad

File Edit Format View Help

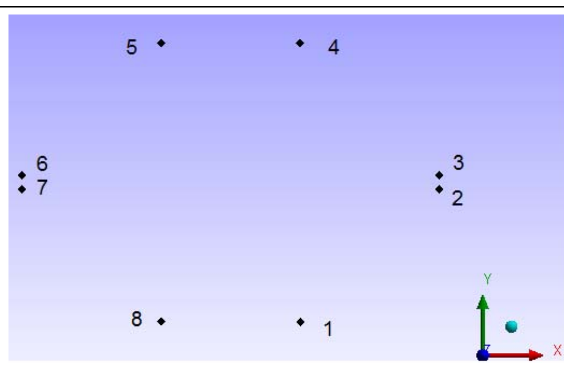
```
1   1   50    0    0
1   2   150   95   0
1   3   150   105  0
1   4   50    200  0
1   5   -50   200  0
1   6   -150  105  0
1   7   -150  95   0
1   8   -50   0    0
```

DM → Modeling → Create → Point → Details View → **Details of Point 1** → Definition: From Coordinates File; Coordinates File ...: select the saved file (*puncte cric.txt*) → Generate


Details View

Details of Point1	
Point	Point1
Type	Construction Point
Definition	From Coordinates File
Coordinates File	None
Tolerance	Normal

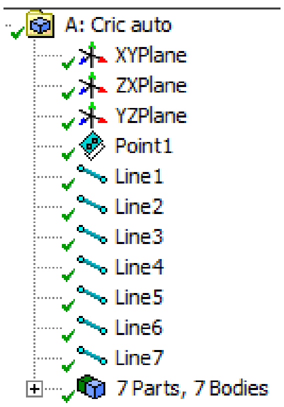
The 8 points in the xOy plane will be obtained automatically, as in the figure below:

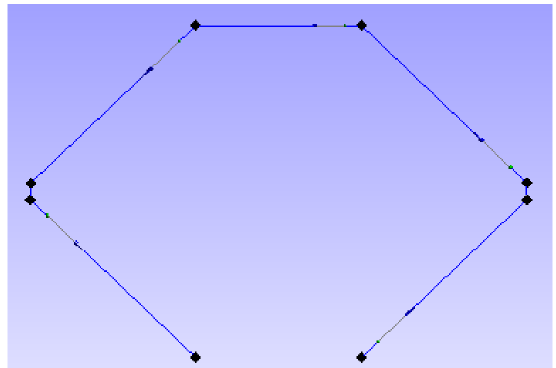


C.3.3 Segments generation

DM → Modeling → Concept → Lines From Points → Details View → **Details of line 1** → Point Segments [while holding down the *Ctrl* key, select points P1 and P2]: Apply → Generate.

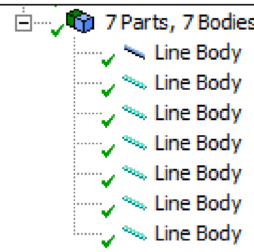
In the same way, using the option *Operation: Add Frozen* din Details View, the segments corresponding to the pairs of points are constructed:: (P2, P3), (P3, P4), (P4, P5), (P5, P6), (P6, P7), (P7, P8). The structure in the figure below will be obtained.





Rename segments

DM → Modeling → Line Body → Rename → the segments created according to the adjacent structure are renamed.

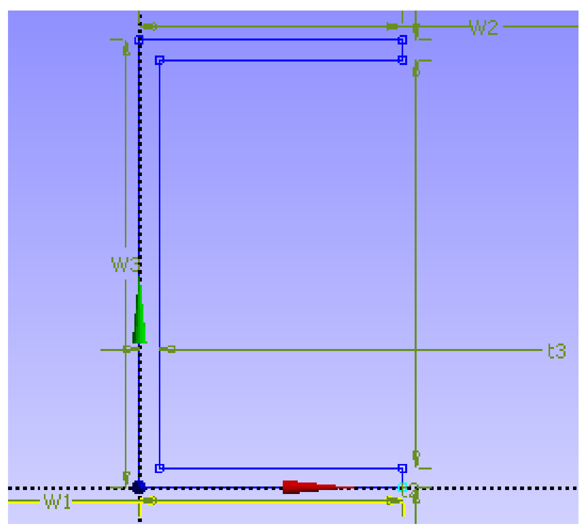


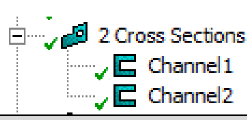
C.3.4 Generation of cross sections

DM → Modeling → Concept → Cross Section → Channel Section → Details View → **Details of Channel 1** → Sketch: *Channel1* (U34);

Dimensions: W1 = 20 mm, W2 = 20 mm, W2 = 34 mm, t1 = 1,5 mm, t2 = 1,5 mm, t3 = 1,5 mm → Generate.

The procedure will be the same for a profile *Channel2* (U30) with dimensions: W1 = 15 mm, W2 = 15 mm, W2 = 30 mm, t1 = 2 mm, t2 = 2 mm, t3 = 2 mm.

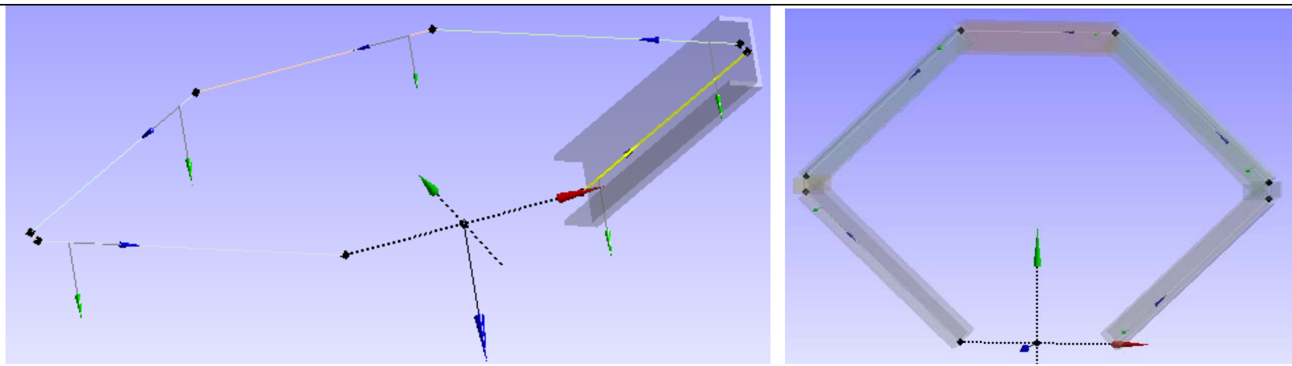




C.3.5 Assigning transverse profiles to the metal structure

In order to complete the procedure for generating a profile, proceed as follows:

DM → Dr_inf → Details View → **Details of Line Body** → Cross Section: Channel2; Offset Type: Centroid; Modeling → View → Cross Section Solids [the 3D section view option is activated] → [check that the profiles are oriented correctly; the profile orientation is visualized with the help of green arrows; if the profile is not symmetrical and is not project oriented, the orientation can be changed accordingly: Dr_inf → Details View → **Line Body-Edge** → Reverse Orientation?: Yes] → Offset Type: Centroid → Generate.



The last operation is repeated for all segments, taking care that the Channel1 (U34) profile is attached to the *Dr_med, Sup, Stg_med* segments and to the other segments - the Channel2 profile (U30).

C.3.6 Saving the geometric model

→ (Save Project) → File → Close Design Modeler.

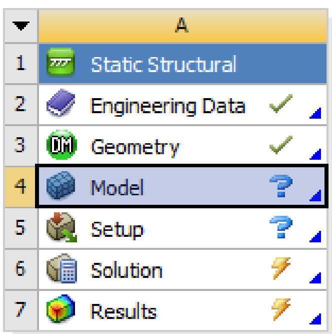
C.4 Finite element modelling

C.4.1 Activate the discretization module and set the finite element type

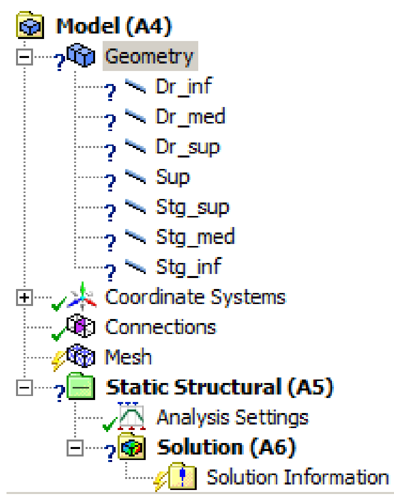
Launch of the finite element modeling module

, Project Schematic ↲↲ Model → launching module *Mechanical [ANSYS Multiphysics]*.

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



Cric auto



Setting the unit of measurement system

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

Setting the material characteristics

Outline: ↲ Geometry → Dr_inf → Details of "Dr_inf" → **Material** : ↲Assignment → [select from list ↲ , ↲ Structural Steel (setare implicită / default)]. The operation will be repeated for the other segments as well.

Setting the model type

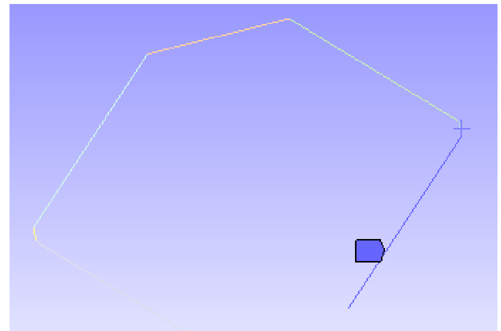
Outline: ↲ Geometry → Details of "Geometry", **Definition**: ↲3D Behavior.

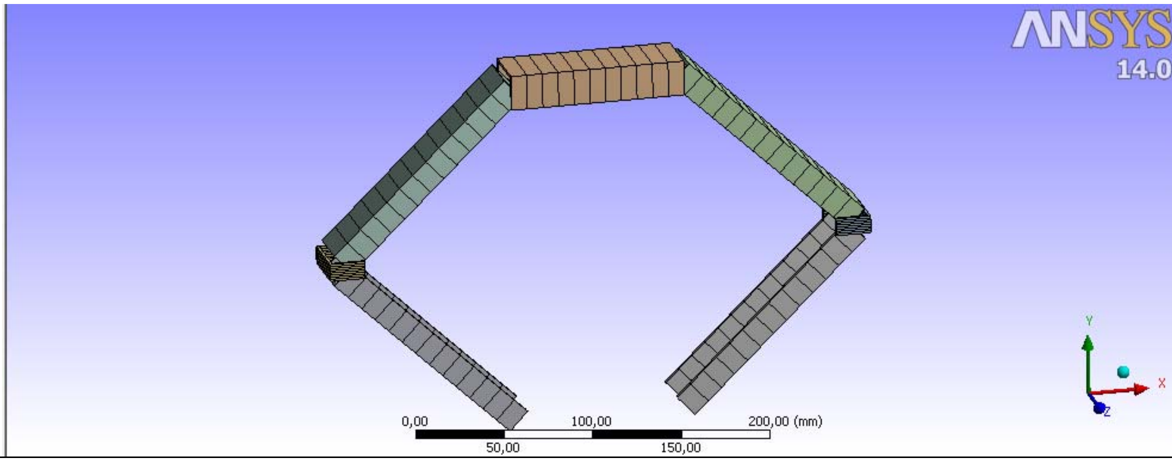
C.4.2 Model discretization and finite element size setting

, Outline: ↲ Mesh → Mesh Control → Sizing → Details of "Edge Sizing" - Sizing → **Scope** → Select Geometry: [a segment of the structure geometry will be selected with ↲ using the selection filter (Edge)] Apply; **Definition Element** → Size: Default → Update. The operation will be repeated for the other segments as well.

Details of "Edge Sizing" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge
Definition	
Suppressed	No
Type	Element Size
Element Size	Default
Behavior	Soft
Bias Type	No Bias





C.5 Modeling joints and constraints

Introduction of gravitational acceleration

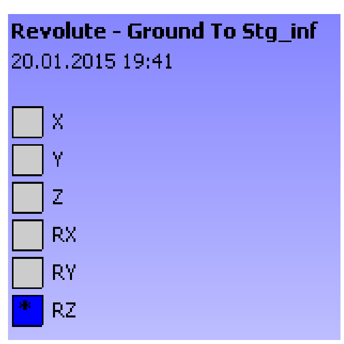
→ Static Structural (B5) → Inertial → Standard Earth Gravity (the selection of the gravitational acceleration implies the taking into account of the own weight of the metallic structure) → Details of "Standard Earth Gravity" → **Definition** → Direction: -Y Direction.

Insertion of the base connection joints

, Outline: Connections → Body-Ground → Revolute → Details of "Revolute - Ground To No Selection" → Mobile → Scope: [select Point 1 using the selection filter (Point)]. A rotating joint around the Oz axis is being considered

Details of "Revolute - Ground To No Selection"

Definition	
Connection Type	Body-Ground
Type	Revolute
Torsional Stiffness	0, N·mm/°
Torsional Damping	0, N·mm·s/°
Suppressed	No
Reference	
Coordinate System	Reference Coordinate System
Mobile	
Scoping Method	Geometry Selection
Scope	No Selection
Body	No Selection
Initial Position	Unchanged



The operation will also be repeated for Item 8, in connection with the base.

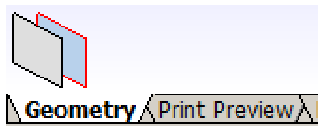
Insertion of joints between segments

Outline: ↵ Connections → Body-Body → Revolute → Details of "Revolute - No Selection To No Selection" →

Reference → Scope: [Point 2 on the Dr_inf segment is selected using the selection filter (Point)] → Apply

→ **Mobile** → Scope: [the same Point 2 is selected, but which is located on the Dr_med segment, using the selection filter (Point)] → Apply.

When selecting the same point, the symbol below will appear in the lower left corner of the graphic window, which is the command button to toggle the selection of the two entities (segments, in this case) by clicking with the mouse on the two planes.



Details of "Revolute - No Selection To No Selection"

Torsional Damping	0, N·mm·s/°
Suppressed	No
Reference	
Scoping Method	Geometry Selection
Scope	No Selection
Body	No Selection
Coordinate System	Reference Coordinate System
Behavior	Rigid
Pinball Region	All
Mobile	
Scoping Method	Geometry Selection
Scope	No Selection
Body	No Selection
Initial Position	Unchanged
Behavior	Rigid

Details of "Revolute - Dr_inf To Dr_med"

Torsional Damping	0, N·mm·s/°
Suppressed	No
Reference	
Scoping Method	Geometry Selection
Scope	1 Vertex
Body	Dr_inf
Coordinate System	Reference Coordinate System
Behavior	Rigid
Pinball Region	All
Mobile	
Scoping Method	Geometry Selection
Scope	1 Vertex
Body	Dr_med
Initial Position	Unchanged
Behavior	Rigid

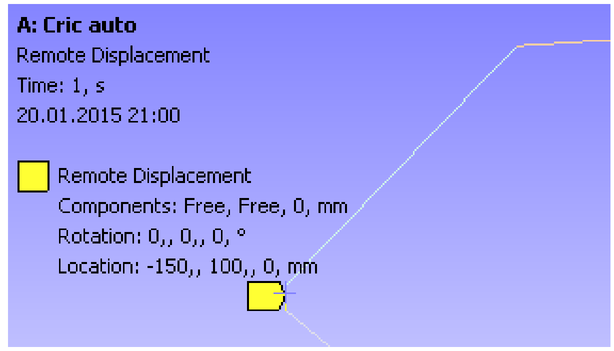
Repeat the operation for the other torques in points 3, 4, 5, 6, 7 and obtain the torques shown in the tree below.

- Connections
  - Joints
    - Revolute - Ground To Dr_inf
    - Revolute - Ground To Stg_inf
    - Revolute - Dr_inf To Dr_med
    - Revolute - Dr_med To Dr_sup
    - Revolute - Dr_sup To Sup
    - Revolute - Sup To Stg_sup
    - Revolute - Stg_sup To Stg_med
    - Revolute - Stg_med To Stg_inf

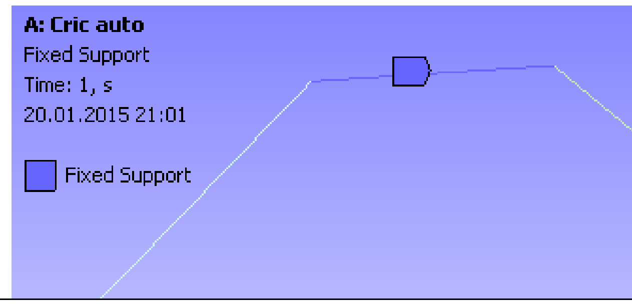
Introduction of operating constraints

The jack will work taking into account the hypothesis that the median lateral segments will be able to move only in the xOy plane, without the possibility to rotate.

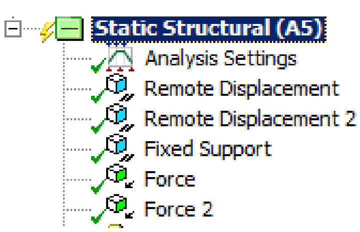
M → ↵ ? Static Structural (B5) → Supports → Remote Displacement → Details of "Remote Displacement" → **Scope** → Geometry: [select the lateral segment *Dr_med*] → Apply; **Definition** → X Component: free, Y Component: free, Z Component: 0, Rotation X: 0, Rotation Y: 0, Rotation Z: 0.
The procedure will be repeated for the segment as well *Stg_med*.

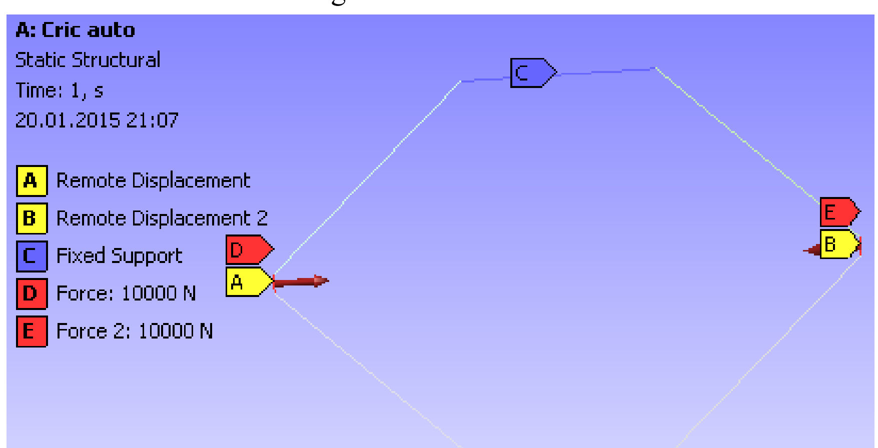


M → ↵ ? Static Structural (B5) → Supports → Fixed Support → Details of "Fixed Support 2" → **Scope** → Geometry: [will be selected with ↵ the upper segment *Sup*] → Apply.



C.6 Load modeling
<u>Input forces</u> M → ↵ ? Static Structural (B5) → Loads → Force → Details of "Force" → **Scope** → Geometry: [will be selected with ↵ lateral segment *Dr_med*] → Apply; **Definition** → Define by: Components → X Component = 10.000 N, Y Component = 0, Z Component = 0. The procedure will be repeated for the *Stg_med* segment, changing only the direction of the force, towards the inside of the mechanism..
The constraints and loads of the structure will look like in the figure below





D. SOLVING THE FEA MODEL

D.1 Launching the calculation module and select the types of results
In order to select the final data types to be analyzed after the launch of the calculation module, the series of commands presented below will be followed.

→ ↳ Solution (B6) → Insert → Deformation → Total [use the commands in the command box open with ↳].

The same result can be obtained by using commands:

↵ Solution (A6) → Deformation → Total [the buttons in the menu bars are used] precum şi / and

↵ Solution (A6) → Deformation → Directional.

For this type of structure, the *Beam* tool can be applied in order to visualize the linearized stresses on the component elements. It is customary in the design of bar structures to take into account the components of axial stresses arising from the effect of axial and bending loads in all directions. The following are the other types of results to be analyzed:

↵ Solution (A6) → Tools → Beam Tool.

↵ Solution (A6) → Beam Results → Axial Force.

↵ Solution (A6) → Beam Results → Bending Moment.

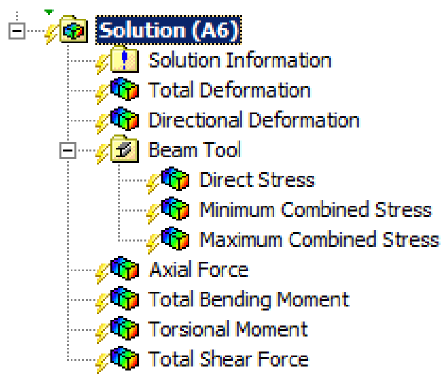
↵ Solution (A6) → Beam Results → Torsional Moment.

↵ Solution (A6) → Beam Results → Shear Force.

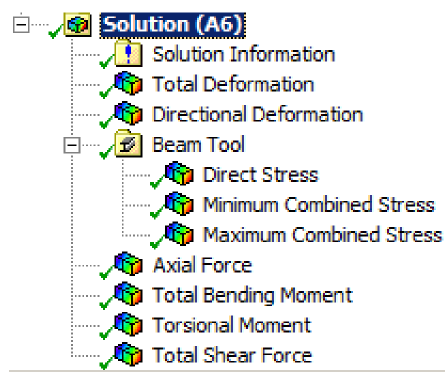
↵ Solution (A6) → Probe → Joint.

D.2. Launching the solving module

→ ↵ Solution (A6) → Solve.



→ Solve →



E. POST-PROCESSING OF RESULTS

E.1 Viewing the displacement fields

For suggestive results, set the view scale of the menu bars:

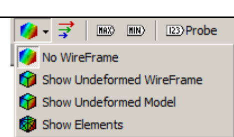
Result 8,6e+002 (Auto Scale) → Result 1.0 (True Scale).

Total deformation view

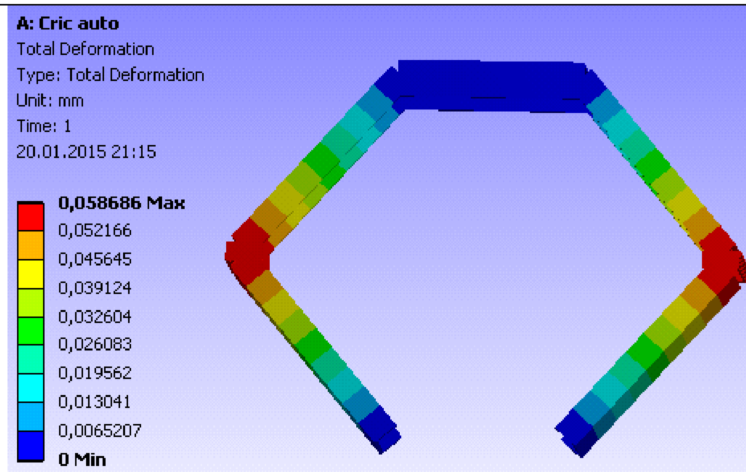
↵ Solution (A6) → Total Deformation → Graph → Animation.

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

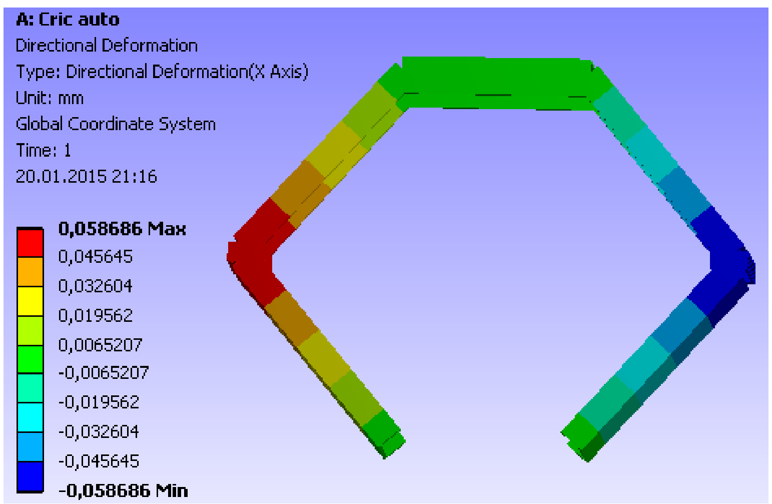


The display characteristics can be changed: the number of frames 10 Frames, as well as the running time of the simulation. At the same time, the result can be saved as a video file using the *Export Video File* command.



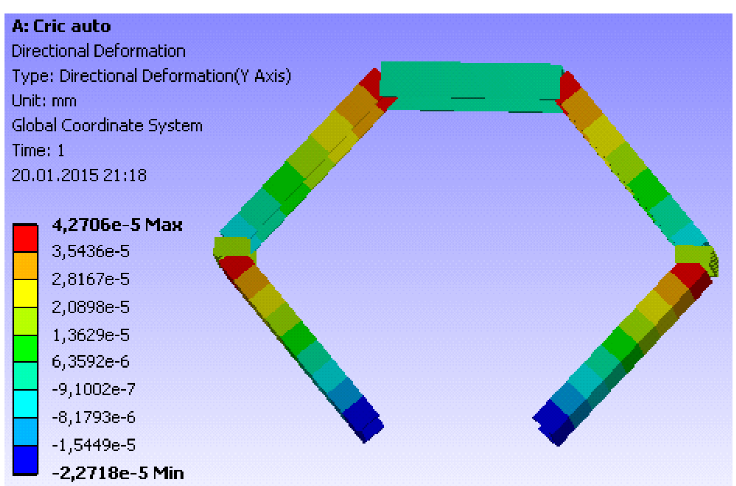
<u>One-way deformation view</u>

↵ Solution (A6) → Directional Deformation → Graph → Animation.



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

↵ Solution (A6) → Directional Deformation → Details of "Directional Deformation" → **Definition** → Orientation: Y Axis → Solve.

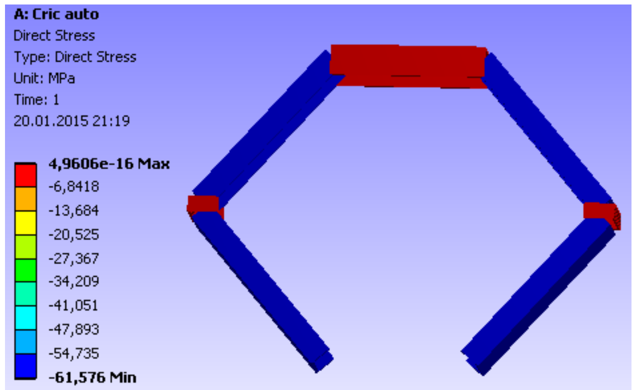


E.2. Visualize the fields of stress, forces and moments

Direct Stress

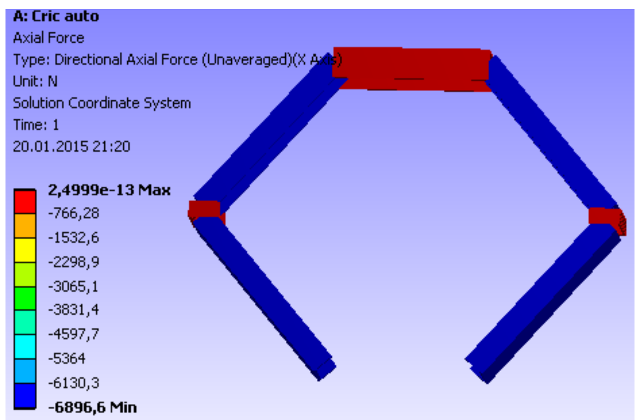
Direct Stress (σ_x) represents the component of the internal tension due to the axial force in an element of the mechanism.

↵ **Solution (A6)** → Direct Stress → Graph → Animation.



Directional Axial Force

↵ **Solution (A6)** → Axial Force → Graph → Animation.

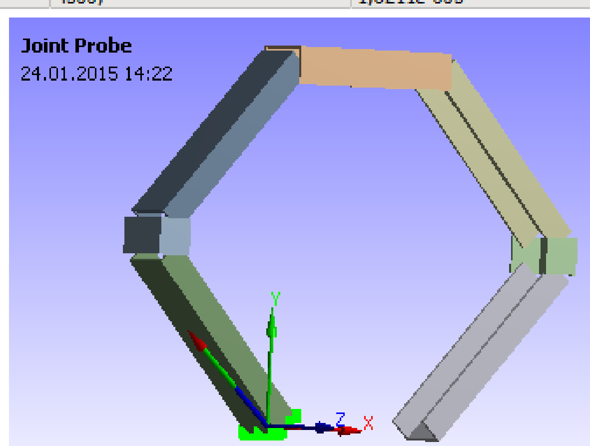


Joint Probe

↵ **Solution (A6)** → Joint Probe → Tabular Data.

Tabular Data

	Time [s]	Joint Probe (Total Force X) [N]	Joint Probe (Total Force Y) [N]	Joint Probe (Total Force Z) [N]	Joint Probe (Total Force Total) [N]
1	1,	-5000,	4500,	1,6211e-005	6726,8



F. RESULTS ANALYSIS

It is observed that, despite the fact that the modeling of the articulated bar mechanism was performed with the help of one-dimensional bodies, the results obtained are suggestive, being presented in a 3D environment, due to the ease of the program used to attach various profiles to the structure. executed by the user. Modifying the profiles of the articulated bars is very easy to do, this can be done even at the end of an analysis, following that after an update command, the results of the new analysis will change according to the new initial conditions.

The realization of the rotational torques is easy, it is not necessary their 3D construction and the precise modeling of their geometry. The definition of the rotational torques can take into account the elastic characteristics of the joint. The positioning of the torques according to the bar profile can be chosen from several variants, offered by ANSYS.

From the point of view of the total deformations, it is observed that the maximum value is 0.05 mm in the area of application of the stress, in the direction of the Ox axis.

Examining the graphical representation of the axial stresses, it is observed that the lateral segments (2) are subjected to the compression stress - represented in blue. The information regarding the deformations, corroborated with the information regarding the internal stresses, the combined maximum stresses lead to the conclusion that the structure withstands loads without problems, the values of the maximum stresses not exceeding the allowed material limit (compression σ_{ac}=80 ... 100 Mpa).

Definition	
Connection Type	Body-Ground
Type	Revolute
Torsional Stiffness	0, N·mm/°
Torsional Damping	0, N·mm·s/°
Suppressed	No
Reference	
Coordinate System	Reference Coordinate System
Mobile	
Scoping Method	Geometry Selection
Scope	1 Vertex
Body	Inf_stg
Initial Position	Unchanged
Behavior	Rigid
Pinball Region	Rigid Deformable Beam (Beta)
Stops	

G. CONCLUSIONS

From the point of view of the pre-processing phase, it can be seen that the use of 1D bodies involves minimal resources for both modeling and discretization. Another strong point is that the transverse profile of the sections can be modified / oriented very easily, without influencing the basic shape of the bar structure. Moreover, it is possible to use different profiles for each section. The sections can be connected in several ways, depending on the central axis of the profiles used.

The introduction of supports, constraints and demands is quick and easy. The declaration of the materials, as well as the discretization of the bar structure are controllable processes, which can be done automatically or manually.

Analyzing the results obtained by MEF, it can be seen that it provides much more data, at a time and with much lower resource consumption, than the analytical version. It can be seen that the structure of the beams is very little required, and smaller profiles can be used in order to achieve savings. Changing the profile of the beam sections and recalculating is done in a very short time, being an easy procedure.