eman ta zabal zazu

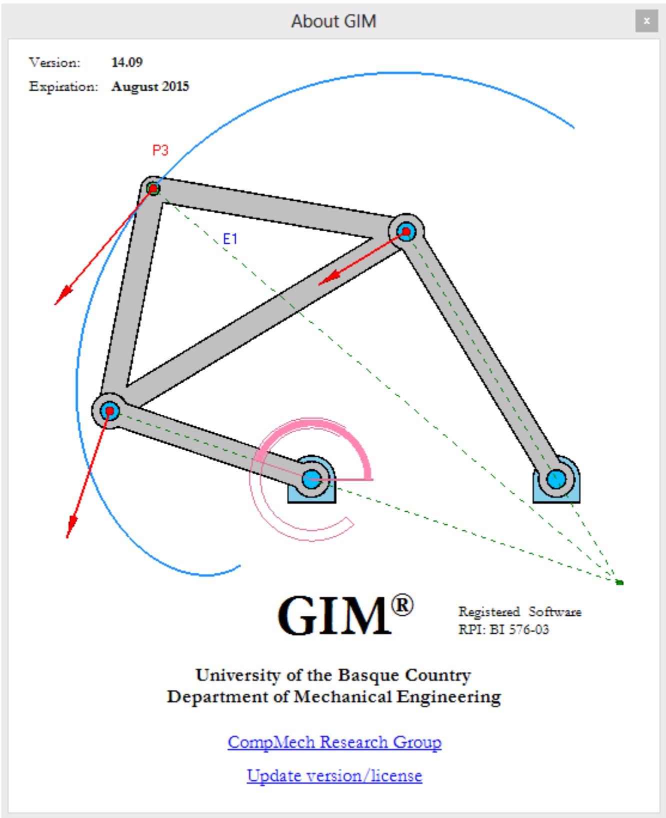
Universidad del País Vasco
Euskal Herriko Unibertsitatea

Ingeniaritza Goi Eskola Teknikoa
Escuela Técnica Superior de Ingeniería
Bilbao

Tutorial of GIM Software

COMPMECH Research Group

www.ehu.es/compmech



Faculty of Engineering in Bilbao
Department of Mechanical Engineering
University of the Basque Country (UPV/EHU)

Contents

1 What is GIM?

GIM is a registered software created by the COMPMECH Research Group, belonging to the Department of Mechanical Engineering of the University of the Basque Country (UPV/EHU). The software is intended for educational purposes, in particular destined to the field of kinematic analysis, motion simulation and synthesis of planar mechanisms, and also static analysis of mechanical structures.

This tutorial refers to GIM Software, Part I, which focuses on the kinematic analysis and motion simulation of mechanisms. Planar mechanisms with n-ary elements joined by revolute and prismatic pairs can be introduced. The position problem is solved iteratively using a numerical method, several of its conditions can be controlled and visualized. Inputs can be defined with a polynomial up to the quintic, so position, velocity and acceleration can be specified at motion extremes.

The program allows plotting the paths of points of interest, as well as the curvature center and curvature center locus of the trajectory, and also the area swept by specific elements of the mechanism can be depicted. Velocities and accelerations can be obtained and the corresponding vectors visualized as the motion is performed, also graphs and tables associated with this data can be plotted. Every kinematic construction can be represented: Instantaneous Center of Rotation (ICR), fixed and moving polodes, acceleration pole, main circles (inflection circle, return circle and Bresse circle), and so on (see illustrative example in Fig. 1).

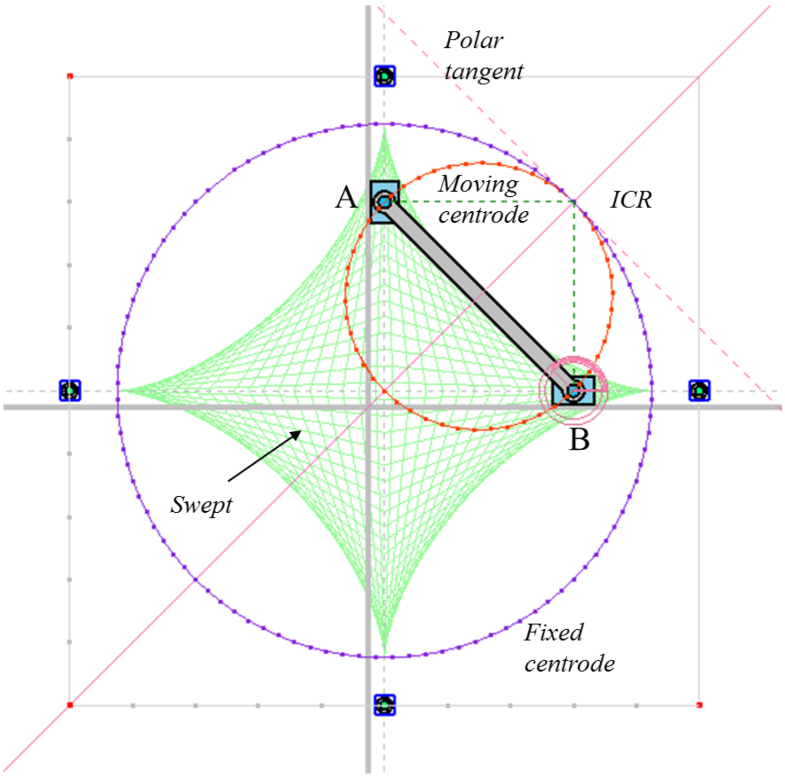


Figure 1: Illustrative example: PRRP mechanism

1.1 Application of GIM software

This software has been developed by COMPMECH Research Group with the aim of approaching the difficulties students usually encounter when facing up to kinematic analysis of mechanisms. A deep understanding of the kinematic analysis is necessary to go a step further into design and synthesis of mechanisms. In order to support and complement the theoretical lectures, GIM software is used during the practical exercises, serving as an educational complementary tool reinforcing the knowledge acquired by the students.

1.2 Access to GIM software

GIM software can be freely downloaded from the COMPMECH web site in the following link: http://www.ehu.es/compmech/software/

If you have used this software, please cite our ownership in any of your publications as follows:

1. In the bibliography cite:

   Petuya, V., Macho, E., Altuzarra, O., Pinto, Ch. and Hernández, A. *Educational Software Tools for the Kinematic Analysis of Mechanisms*. Comp. Appl. Eng. Education. First published online: February 24, 2011. DOI: 10.1002 cae.20532. ISSN: 1061-3773.

2. In the acknowledgments cite:

   The authors wish to acknowledge Alfonso Hernández, CompMech, Department of Mechanical Engineering, UPV/EHU, for the permission to use the GIM software. (www.ehu.es/compmech).

2 Kinematic Analysis

Regarding the kinematic analysis of planar mechanisms, GIM software has two main modules: **Geometry** and **Motion**.

- Geometry module is the one in charge of defining a specific design of the mechanism object of study.
- Motion module performs the kinematic analysis and motion simulation of the mechanism.

2.1 Geometry module

In Fig. 2, the starting main window associated with Geometry module is shown. In the following sections, the different options included in this module are explained in detail.

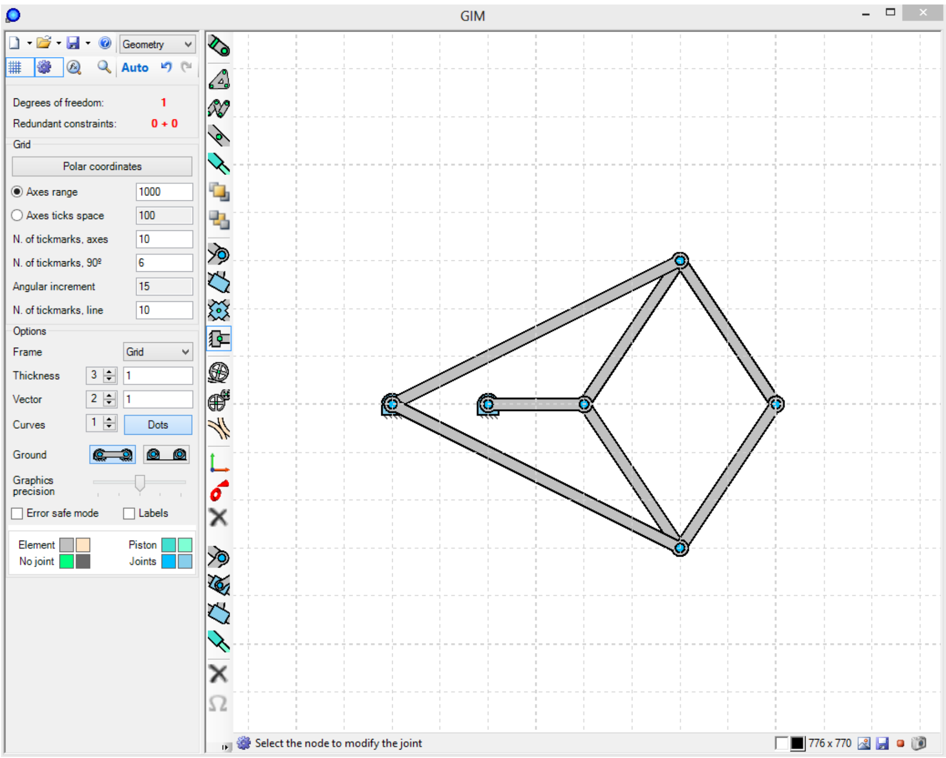


Figure 2: Main window of Geometry module

2.1.1 Starting steps

On the left-hand side of the main window several options appear. The icons located at the upper left-hand corner serve to perform common actions, such as creating a new document, open an existing document or saving the work. Just below, a second bar contains some other icons which display two specific menus, as shown in Fig. 3.

The first one, *Grid* menu (see Fig. 3 left), has the following options:

- *Polar coordinates*: serves to define the design of the mechanism using polar coordinates.
- *Axes range*: permits defining a specific range for the grid axes.
- *Axes ticks space*: serves to define the number of tickmarks appearing in the grid.

It also allows to vary the number of tickmarks appearing in the axes. If the Grid menu is selected the points that will serve to define the geometry of the mechanism will be adjusted to the axes of the grid.

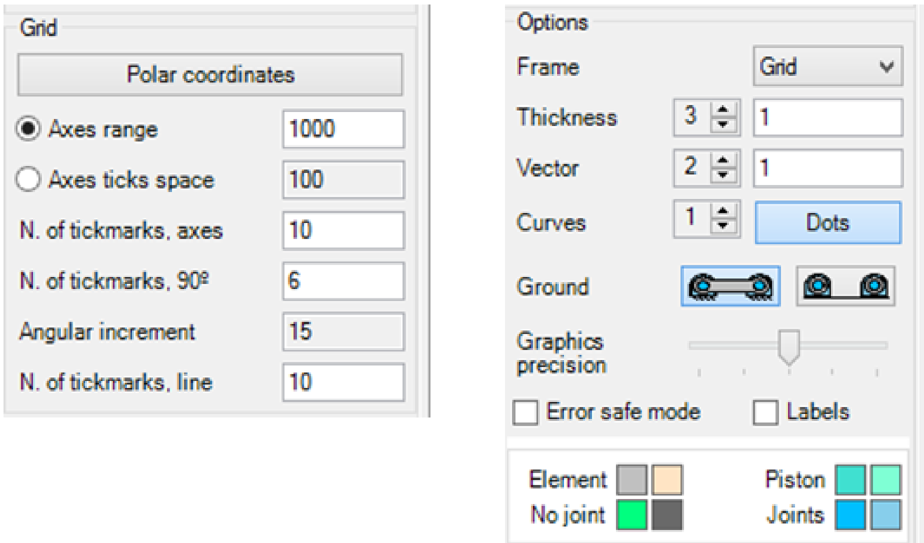


Figure 3: *Grid* and *Options* menus of Geometry module

The second menu, *Options* menu, (see Fig. 3 right), adds the following possibilities:

- *Frame*: it includes three possibilities. The default one, *Grid*, displays a full grid all over the drawing window. Another option is *Axes*, which displays the two principal axes XY of the grid. Finally, is *None* is selected no grid is shown.
- *Thickness factor*: it permits modifying the thickness of the elements.
- *Graphics precision*: it permits modifying the precision of the graphics displayed.
- *Error safe mode*: it avoids closing the program if some error occurs.
- *Ground*: it offers two possibilities to visualize the ground.
- *Labels*: selecting this option enables displaying the labels of each node, element and joint of the mechanism.
- *Colors*: it shows the color legend of each elements and joint type.

2.1.2 Defining the geometry of the mechanism

In this section, it is explained how to define a specific design of a mechanism under study. To do so, the designer will use the icons included in the vertical bar on the left-hand side of the drawing screen. As shown in Fig. 4, these icons serve for: creating the nodes of the mechanism, creating the elements, defining the type of joints (revolute or prismatic joints can be established), establish the fixed joints, modifying or deleting elements, etc.

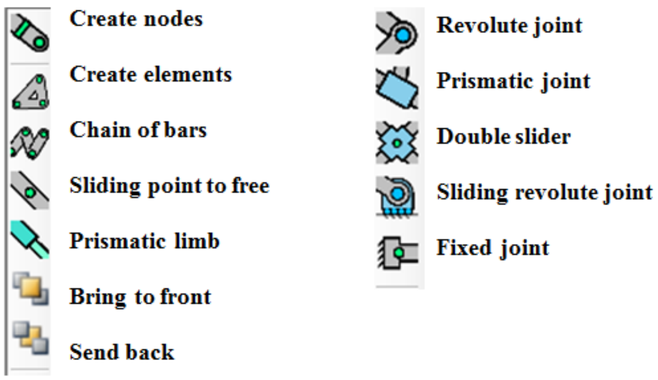


Figure 4: Icons included in Geometry module to design a mechanism

The process to depict the mechanism is very simple: select the desired icon and locate the cursor mouse on the drawing screen. In general, the basic design process consists in: *first*, create the nodes of the mechanism; *second*, define the elements between the established nodes and *finally*, define all the joint types.

- **Nodes**: To create the nodes just select the corresponding icon named *Create nodes*, and then, having the mouse cursor on the drawing screen, just click on the desired location. Note that it is also possible to define specific (*x,y*) coordinates of the node by writing the numerical values on the boxes that appear at lower left-hand corner once the *nodes* icon has been selected.

- **Elements**: Once the nodes have been depicted, the elements can be created. To do so, select the *Create elements* icon, and again move the mouse cursor to the drawing screen. Click once on the first node forming the element, and click **twice** on the last node of the element.

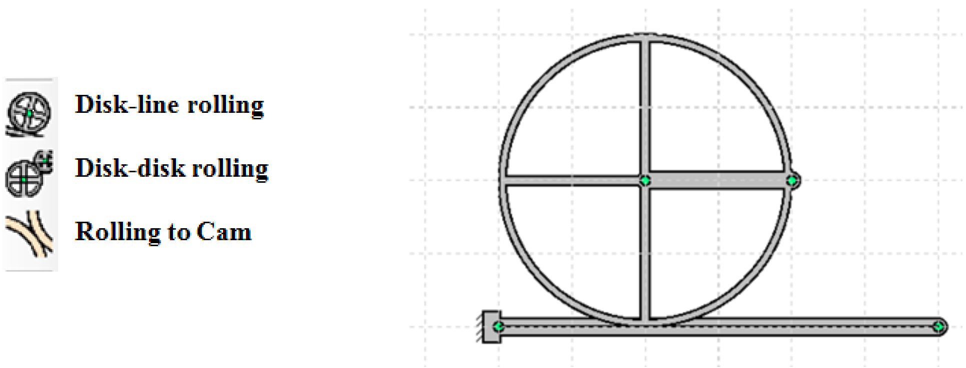
- **Kinematic joints**: To define the kinematic joints of the mechanism just select the corresponding icon (revolute joint, prismatic joint, double slider or fixed joint) and, having the mouse located on the drawing screen, click on the corresponding node. It must be taken into account that to establish a double slider joint it is necessary to previously define two bars. Then, once the double slider icon is selected, click on the bars and the slider will be automatically created in the intersection point between the bars.

As it can be seen in Fig. 4 there exist other options such as bring to front or back any element, establish a prismatic limb, create a chain of bars, etc.

2.1.3 Establishing rolling contact

The software permits the user to analyze a special case consisting in the rolling contact between two elements, these being a disk and a line or between two disks. Besides, it is also possible to transform the rolling contact into a cam joint. For that, the three icons appearing in Fig. 5a must be used.

One important thing is that the first action is to define two elements, in particular two bars. Then, once the bars have been created, the user has to select the icon associated with the case he/she is interested in. For example, if we are interested in studying the rolling contact between a disk and a line, the first icon of Fig. 5a has to be selected. By following the instructions indicated in the lower part of the window, the user first has to click on one of the bars (that will be transformed into a disk) and then clicks on the other bar (which will constitute the line). The radius of the disk will be automatically established by the program in order to achieve the contact among the two elements. An example of this type of contact is shown in Fig. 5b.



(a) (b)

Figure 5: (a) Options of rolling contact; (b) Example of disk-line rolling

As said, the software offers two additional possibilities. One of them is to establish a disk-disk rolling contact. The steps for doing that are the same ones that have been previously explained but selecting the corresponding icon (the second icon inf Fig. 5a). The second option is to transform the rolling contact into a cam contact. First, follow the steps to achieve a disk-line or a disk-disk contact, and second, select the last icon of Fig. 5a. Notice that the color of the disk will change into orange, which indicates that the cam contact has been established. An example of a cam contact is shown in Fig. 6.

2.1.4 Additional options

Once a specific mechanism has been defined by the user, which implies that all the elements have been created and all the kinematic joints have been properly defined, the program offers additional options that are displayed in Fig. 7.

On the one hand, the icons at the left enable to set the origin of the fixed frame at any point of interest, to modify the location of any node and to delete any node or element.

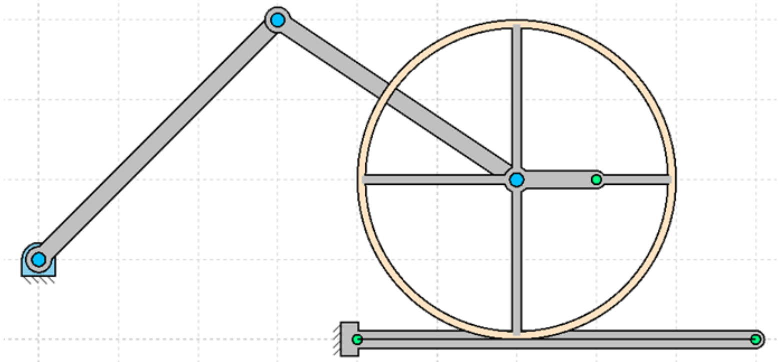


Figure 6: Example of cam contact between a disk and a line

On the other hand, the icons at the right can be used to vary geometric dimensions of the mechanism. First, we have to select the constraint that we want to keep fixed (the upper icons) and then, selecting the last icon, we can edit any of the dimensions and parameters that are not fixed.

Figure 7: Additional options in *Geometry* module

An example regarding the modifying of the dimensions of a mechanism is shown in Fig. 8. The *Absolute angle constraint* icon has been chosen in order to fixed the angle colored in red in the figure. Then, selecting the *Edit* icon, all the possible dimensional parameters that can be modified appear in green color. The values of the parameters can be inserted in the left-hand side box of the edit constraint.

2.1.5 Library of mechanisms

Many types of planar mechanisms are included in a library that can be accessed with the Open icon at the top. One, two or three degree-of-freedom mechanisms are available, their geometry being completely defined so that the kinematic analysis of the mechanism can be directly done. Besides, some types of structures are included in the library that are related to the *Statics* module.

A preview of some types of planar mechanisms that can be found in the library is shown in Fig. 9. Once a specific mechanism of the library has been selected, it is automatically opened in *Geometry* module. If the user wants to vary any of its dimensions,

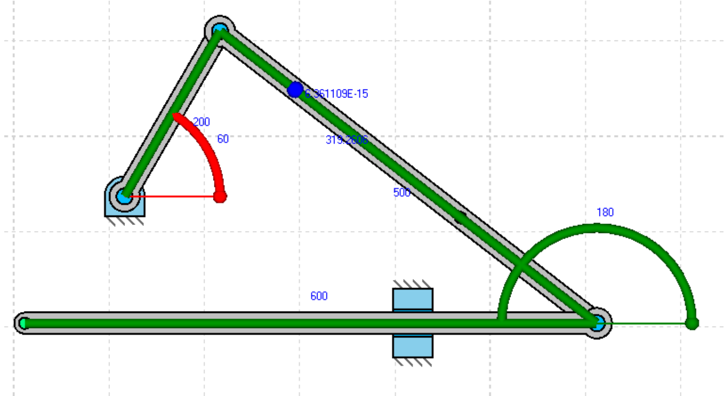


Figure 8: Modifying the geometric dimensions

this can be done by following the instructions already explained in Section 2.1.4.

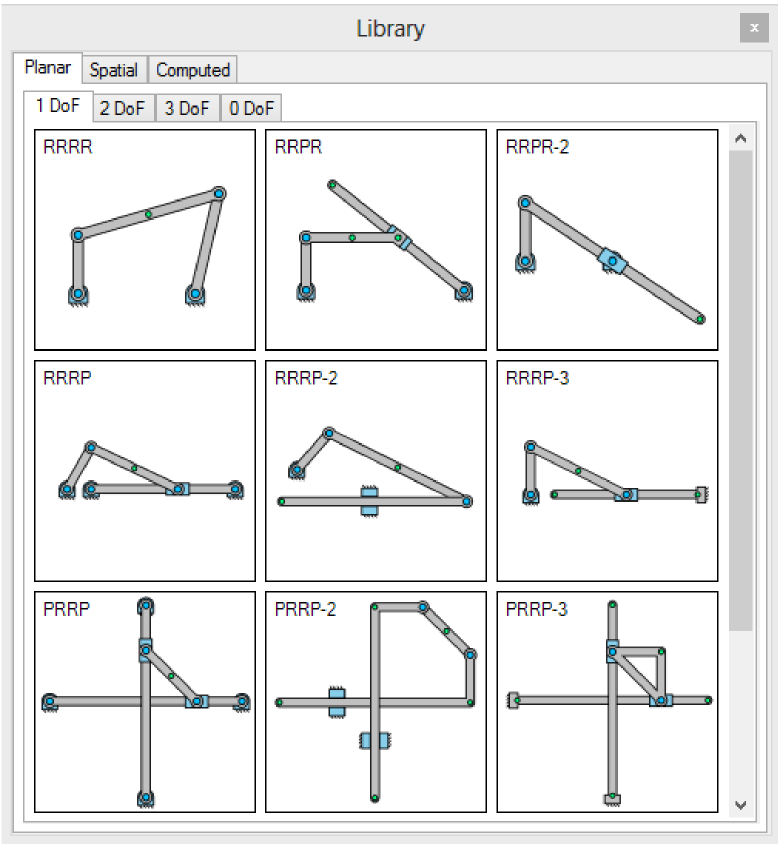


Figure 9: Library of mechanisms

2.2 Motion module

Motion module is the module in charge of carrying out the motion simulation and kinematic analysis of a planar mechanism under study. The user can access this module only once the geometry of the mechanism has been completely defined in Geometry module. To get into Motion module, just select the option *Motion* from the display menu on the upper bar of the main window.

In Fig. 10, the starting main window associated with Motion module is shown, using the four-bar mechanism as an illustrative example.

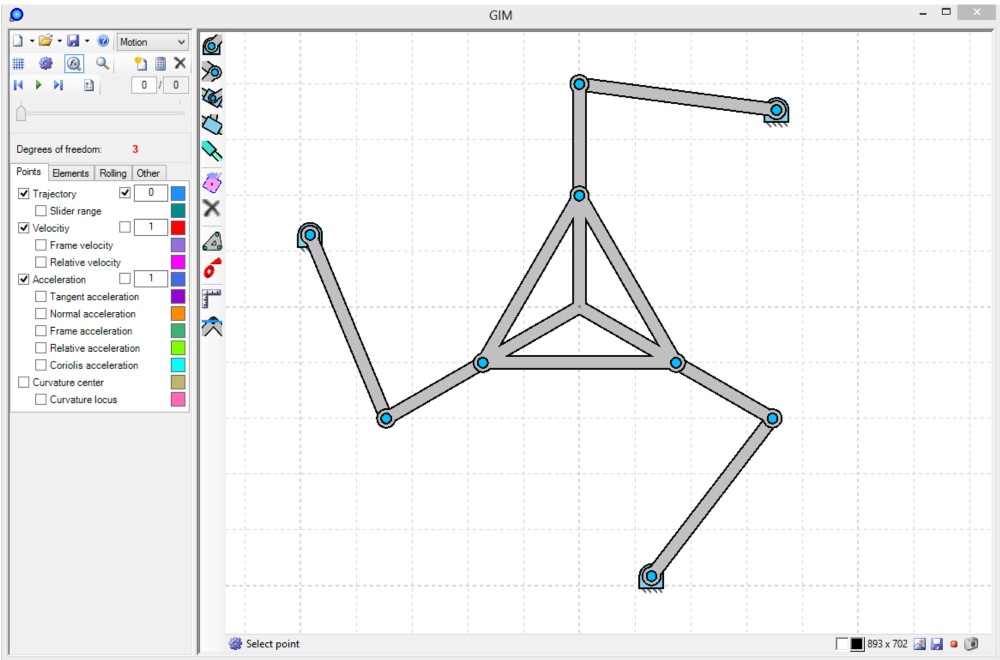


Figure 10: Main window of Motion module

This screen contains three main parts:

1. The vertical bar close to the drawing screen in which several icons appear so as to define the inputs' motion. The first five icons serve to define the inputs of the mechanism. The inputs are: fixed rotation actuator, floating rotation actuator, slider R actuator, slider actuator and piston actuator. Besides, the next icons ar use to modify or delete any actuator, define the reference element and modify the coupler position. The two last icons permit selecting specific elements or nodes, and showing graphs respectively.

   The way the input conditions has to be established is detailed in Section 2.2.1.

2. The top horizontal bar containing the controls of the motion simulation (play/pause, next step, previous step). Besides, just below the motion controls, it is displayed the number of degrees of freedom of the mechanism.

3. Four menus named **Points**, **Elements**, **Rolling** and **Other**. This is the main part of the Motion module, as it contains all the options that can be visualized in relation

to important kinematic features of the mechanism, as for example, velocities and accelerations, points' trajectory, Instantaneous Center of Rotation (ICR), fixed and moving polodes, main circles, rolling contact properties, and so on (see Fig. 11).

Detailed explanation of the operation of this part is given in Section 2.2.2.

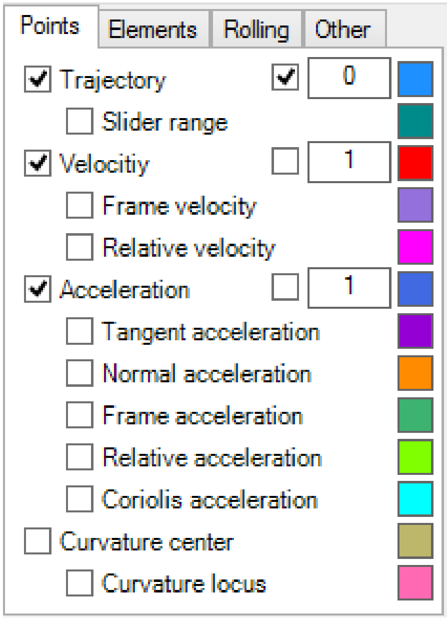


Figure 11: Menus of Motion module

2.2.1 Defining the input conditions

The first step to proceed with the kinematic analysis of a mechanism under study is to define the input(s) of the mechanism. For a n-DOF mechanism, n number of inputs must be established. As highlighted in Fig. 12, the icons in the vertical bar serve to define the following type of inputs: fixed rotation actuator, floating rotation actuator, slider rotation actuator, slider actuator and piston actuator.

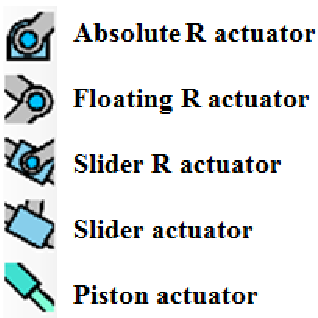


Figure 12: Vertical bar of Motion module

By using the four-bar mechanism as an example, let us chose one of the fixed revolute joints as the actuated one, as shown in Fig. 13a. The process is very simple, first select any of the icons of the vertical bar associated with the input you want to specify, and then click on the corresponding input joint of the mechanism. Next, define the characteristics of the input (position, velocity, acceleration) by establishing the desired parameters on the right-hand side box menu, as shown in Fig. 13b.

The default input option consists in a linear increment of position, that is, constant velocity. However, the characteristics of the input actuation can be varied by the user, by modifying the parameters of position, velocity and acceleration displayed in the menu of Fig. 13b. A polynomial up to the quintic can be specified. The graphs on the right-hand side are displayed according to the established input conditions.

2.2.2 Getting into the kinematics

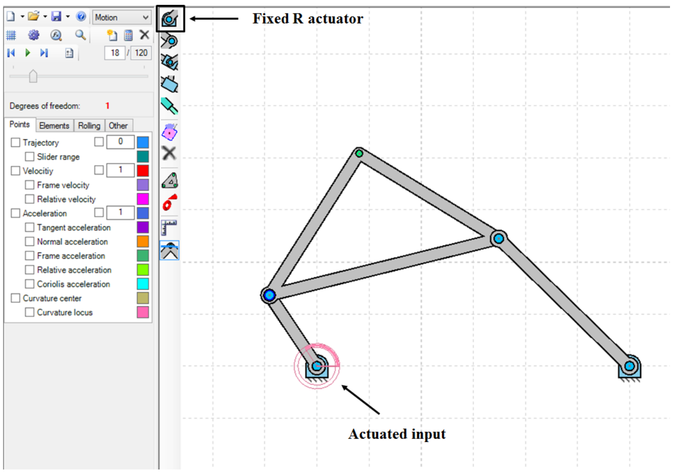
Once the input (or inputs in the case of more than one DoF) of the mechanism has been defined and the motion has been computed, the user is ready to analyze several kinematic properties of the mechanism.

As stated in Section 2.2, in particular focusing on the options of Fig. **??**, there exist two menus regarding the kinematic properties that can be obtained. On the one hand, in relation to **points** of interest of the mechanism (**Points** menu), the following data can be displayed:

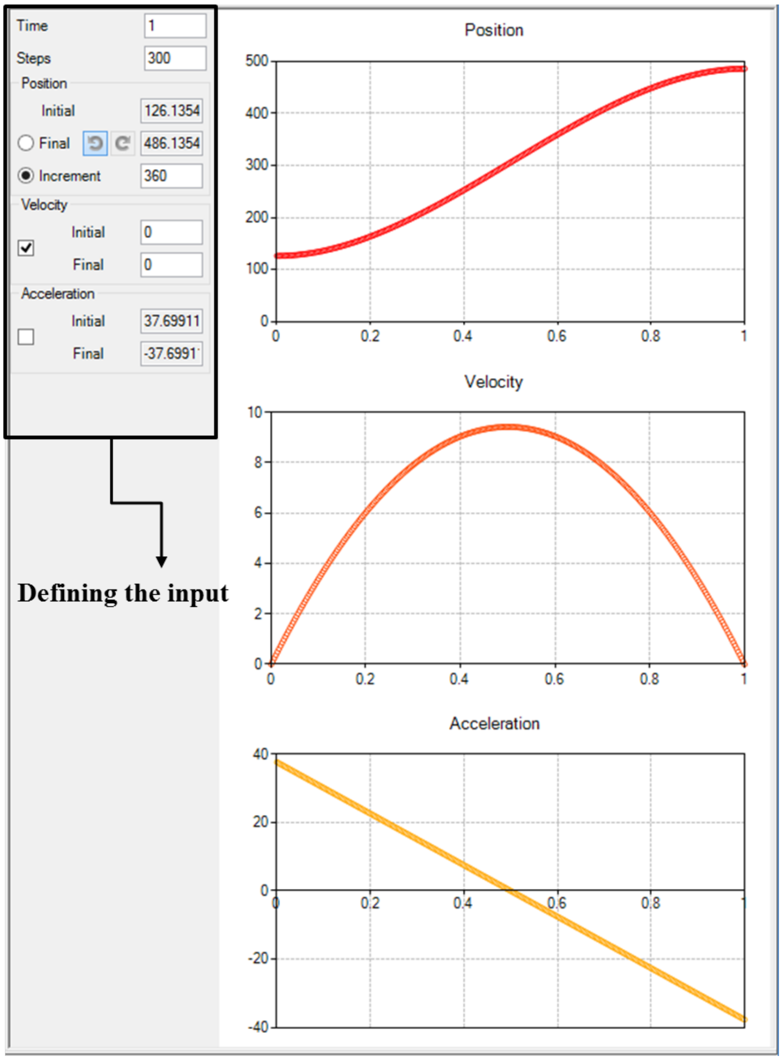
- Trajectory
- Velocity
  - Frame velocity
  - Absolute velocity
- Acceleration
  - Tangent acceleration
  - Normal acceleration
  - Frame acceleration
  - Absolute acceleration
  - Coriolis acceleration
- Curvature center
  - Curvature locus

On the other hand, regarding the **elements** of the mechanism (**Elements** menu), the user can obtain the nest features:

- Swept



(a)



(b)

Figure 13: Defining the input actuation: (a) selecting the input; (b) Establishing the input conditions

- Angular velocity
- Angular acceleration
- ICR (Instantaneous Center of Rotation)
  - Pole velocity
  - Fixed centrode
  - Moving centrode
  - Centrode tangent
- ICA (Instantaneous Center of Acceleration)
  - Components
  - Inflection circle
  - Inversion circle
- Line envelope
  - Return circle

In addition to this, if any mechanism including a **rolling contact** has been established, there exists another menu named **Rolling** which allows the user to analyze the following options:

- Contact motion
- Contact tangent
- Contact velocities
- Contact accelerations
- Contact point trajectory

In order to visualize any of the aforementioned options (or various options simultaneously) it is just needed to click the box (or boxes) associated with the desired option included in *Point* or *Element* menu, and then select the point or element of interest.

An illustrative example is shown in Figs. 14 and 15. In Fig. 14, some properties of a point of interest, in this case the coupler point **P**, are displayed: the velocity and acceleration, its trajectory and the curvature center of the trajectory. Figure 15 displays significant kinematic features associated with a specific element, in this case the moving platform: the ICR, and the fixed and moving centrodes.

It is essential to *remark* that all of the kinematic features can be visualized not only for a certain position but all along the motion of the mechanism. In order to get this, just press the play button included in the motion controls.

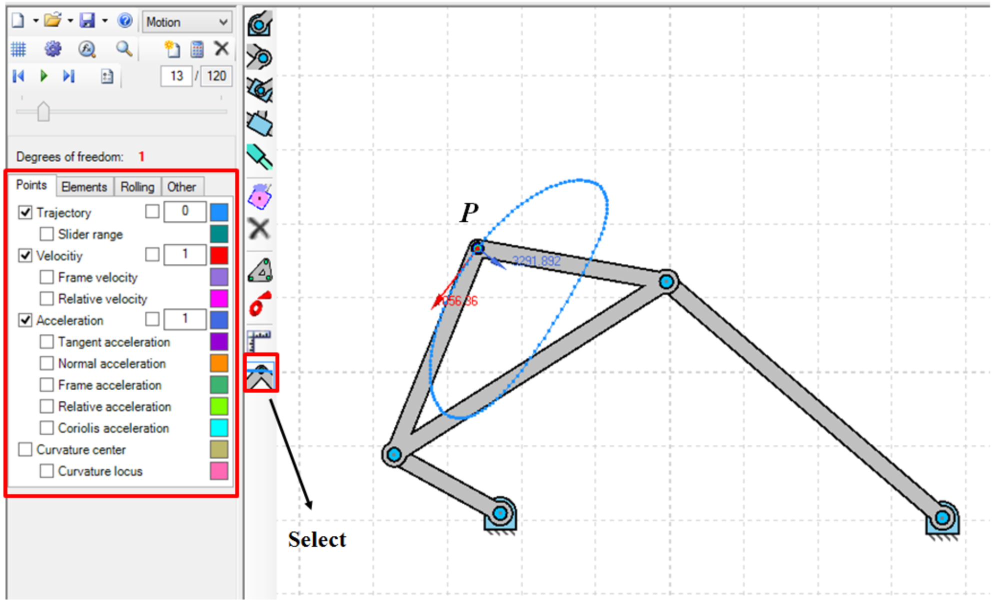


Figure 14: Showing kinematic properties of the coupler point

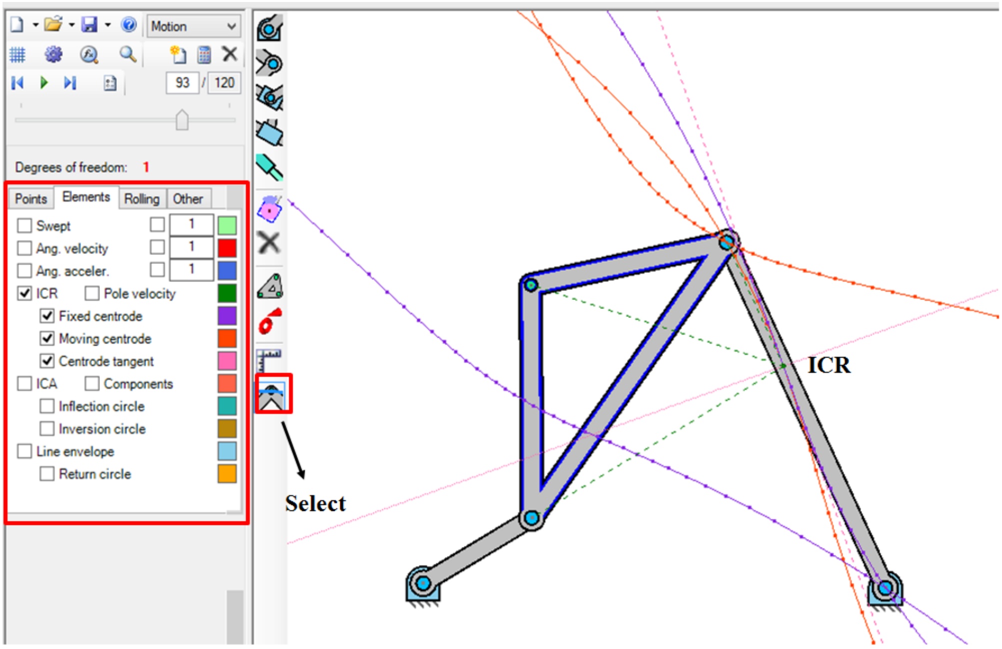


Figure 15: Showing kinematic properties of the moving platform

2.2.3 Graphical plots

Motion module allows to represent the graphs of features associated with points or elements. These graphics are obtained by selecting the icon indicated in Fig. 16, which displays the menu also shown in the figure. This menu is divided into the characteristics of points and the ones associated with elements.

With respect to points, the following graphs can be obtained:

- Coordinates

- Velocity
- Acceleration
- Tangential acceleration
- Normal acceleration
- Curvature Center
- Curvature radius

And regarding the elements:

- Angular velocity
- Angular acceleration
- ICR
- Pole velocity
- IC of acceleration

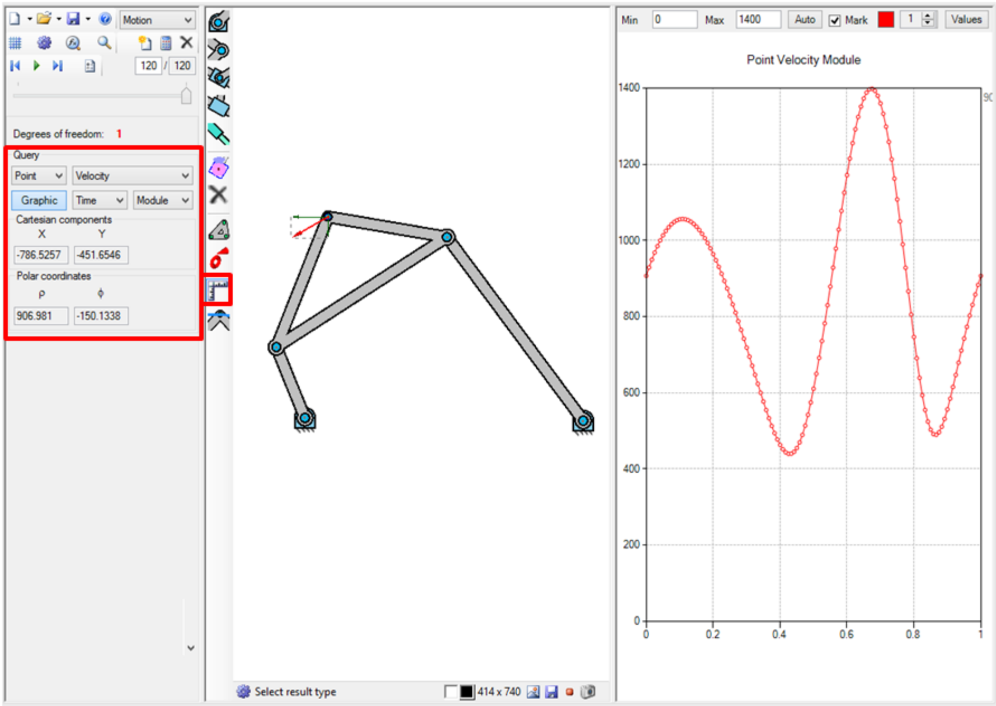


Figure 16: Obtaining graphs of features associated with points or elements

3 Dimensional Synthesis

Until now the two modules named Geometry and Motion have been deeply explained. GIM software also incorporates a module intended for dimensional synthesis of planar mechanisms.

This module, named Synthesis, can be directly accessed from the top menu. In particular, this module analyzes the four-bar mechanism, which is displayed in the drawing screen when the module is opened (see Fig. 17).

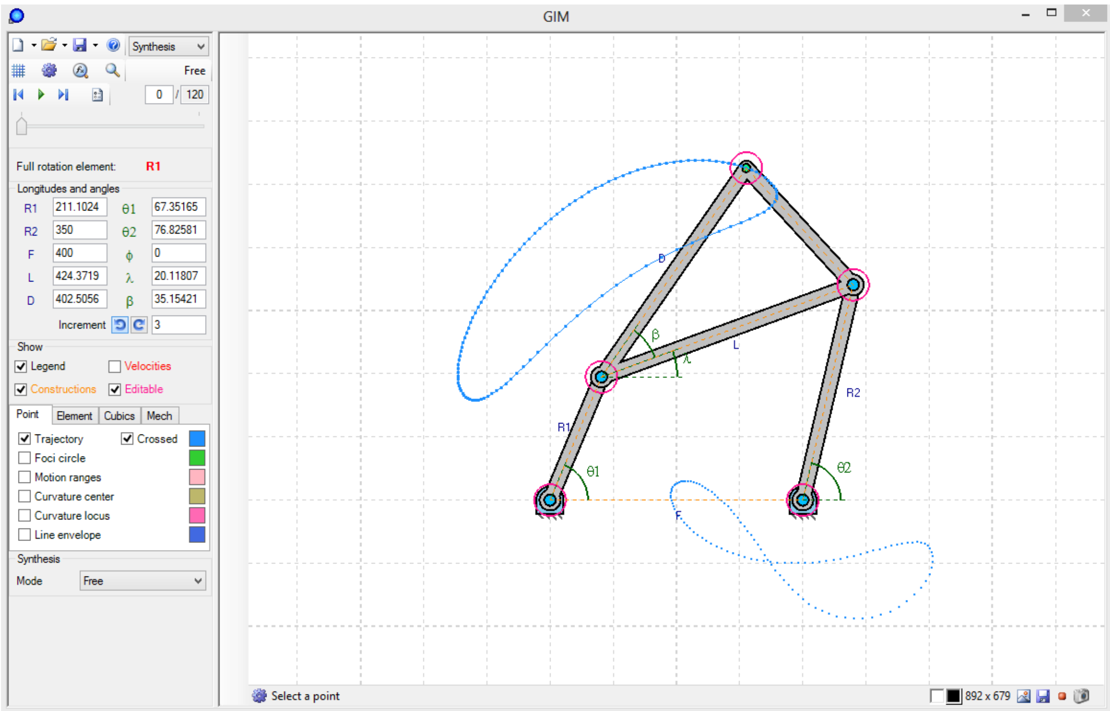
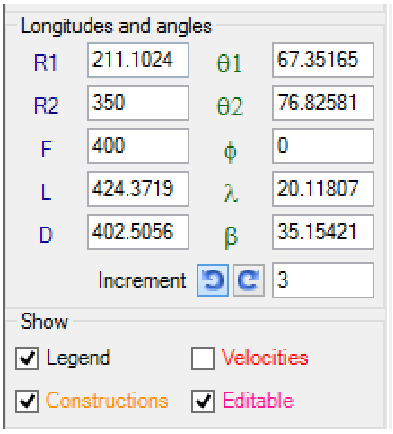


Figure 17: Main window of Synthesis module

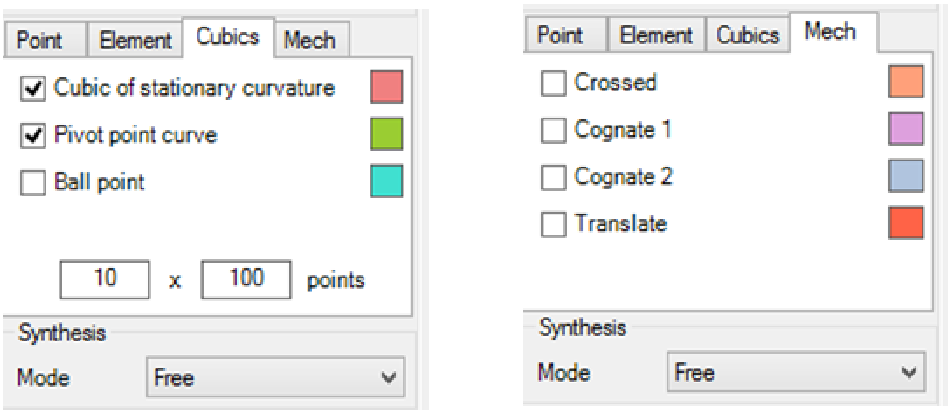
All the dimensional parameters and the actuators that define the inputs of the mechanism can be visualized in the screen. The numeric values of these parameters appear on the left-hand side column, as shown in Fig. 18a. To vary the dimensions the user can directly type the values in the corresponding box, or make use of the *Editable* option. The latter option is shown in Fig. 18a. If the icon is selected, some pink circles appear on the mechanism. By clicking in any of these circles the corresponding location of the node can by modified, and so the associated longitudes and angles.

3.1 First steps into Synthesis module

On the left column, at the bottom, four menus are displayed. The first two of them are named *Point* and *Element* respectively. They show many kinematic properties of points of interest and of different elements. They have similar options to the ones included in Motion module and that have been already analyzed in Section 2.2. Additionally, they can show the trajectory of the end-effector (two trajectories if the crossed solution of



(a)



(b)

Figure 18: (a) Longitudes and angles; (b) *Cubics* and *Mech* menus

the four-bar establishes a different path) and the motion range of each bar.

The remaining two menus are called *Cubics* and *Mech*. They are displayed in Fig. 18b. The first one allows the user to depict the cubic of stationary curve and the pivot point curve, and also the location of the Ball point. Bear in mind that by using the controls at the top, the motion of the mechanism can be visualized and simultaneously the evolution of the aforementioned curves along the motion.

In *Mech* menu many interesting options are available. Firstly, the crossed architecture of the four-bar can be displayed and also the two cognates. When any of these options is selected, the icon named *Change* appears which allows the user to change into the crossed solution or into any of the two cognates. Secondly, selecting the *Translate* icon, a translation bar connected to the end-effector of the original mechanism and to the end-effector of any of the cognates is achieved. By clicking twice on this icon, the redundant elements of the whole generated mechanism are eliminated.

3.2 Dimensional synthesis methods

3.2.1 Precision points

In GIM software there are also implemented various graphical constructions to get the design of the four-bar that meets three, four or five precision points. To do that, in the left column, select the mode called *Precision points* and using the submode icon select the number of points of interest: 3, 4 or 5. In Fig. 19, an example with three precision points is shown. The three precision points are plotted in red, and the trajectory of the coupler point is depicted in blue. Besides, the graphical constructions that are inherent to the graphical method are shown. If the user wants to see only the trajectory, the constructions can be eliminated just by inactivating the corresponding icon in Fig. 18a.

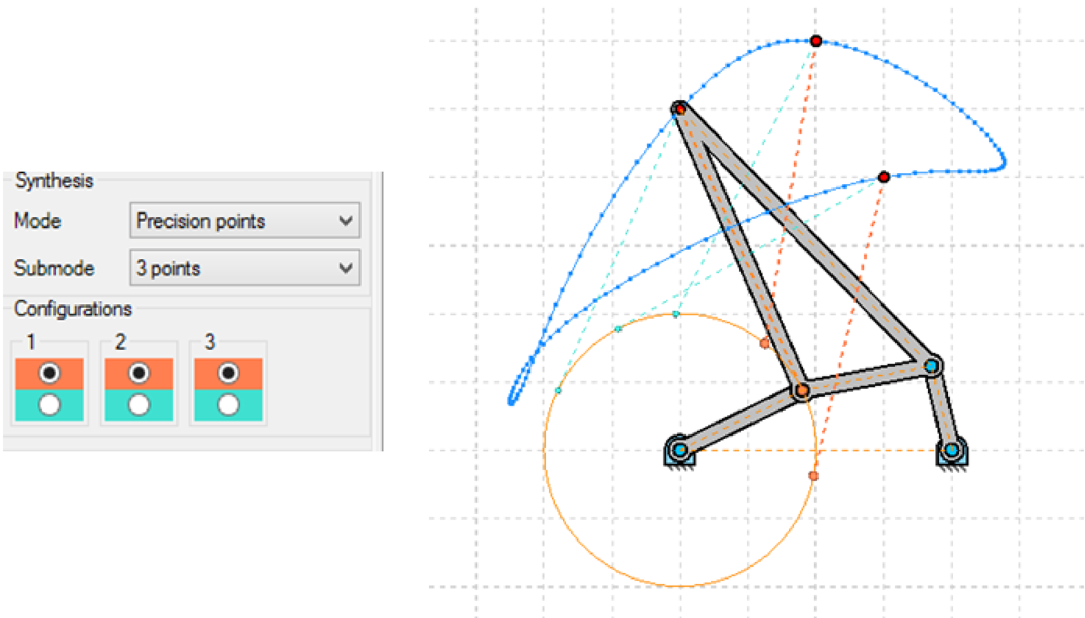


Figure 19: Cubic of stationary curve and the pivot point curve

Note that the graphical constructions achieve more than one solution, depending on the intersection points that are selected (depicted in blue or orange in Fig. 19). To visualize all the possible combinations the user can activate any of the configurations that appear below the submode icon. At the time of changing the combination the resulting mechanism is automatically displayed on the screen. Of course, changing the configurations implies obtaining a different geometry of the four-bar and thus, a different trajectory of the coupler point. As an example, by choosing two different combinations, the resultant mechanism and the corresponding trajectories are shown in Fig. 20. Note that the location of three precision points has not been altered. The user can visualize all the possibilities and chose the optimum geometry that best fits the desired trajectory.

Regarding the number of precision points, not only three but also four or five precision points can be established. The process is similar to the one explained. In the submode icon the user has to select 4 or 5 precision points, and the four-bar mechanism appears automatically in the drawing screen. Again, the different configurations are

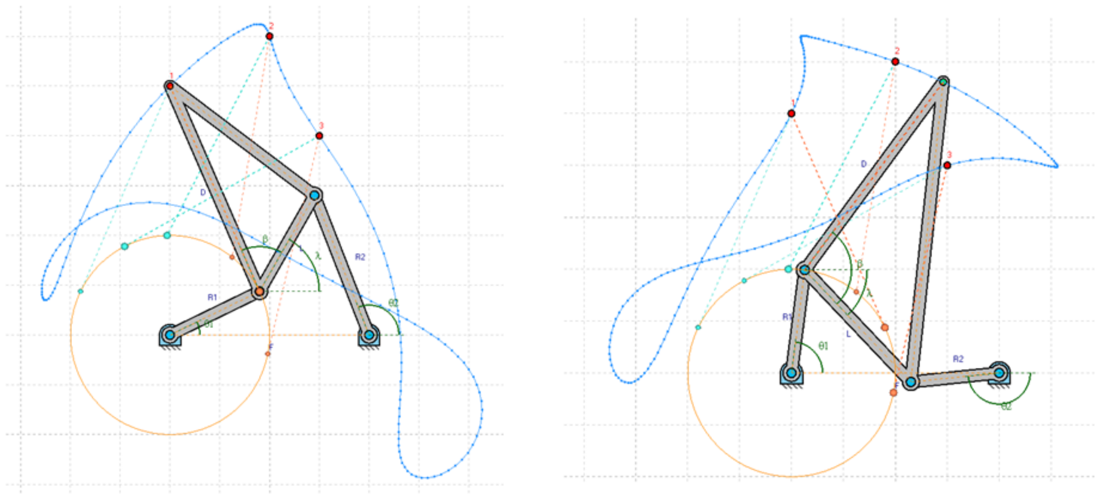


Figure 20: Different trajectories for three precision points

available, and the resulting mechanism can be visualized.

In relation to the location of the precision points, for the three cases (3,4 or 5 points), the points' location is initially established by the program as a default position. In the three cases their location can be changed just by clicking in any of the points and then, moving the mouse to wherever we want or typing the exact location in the boxes located at the left column.

3.2.2 Solid element guiding

The graphical constructions to perform the dimensional synthesis for solid element guiding are also implemented in the software. Three options are available: 3 postures and moving, 3 postures and fixed, and 4 postures. The difference between the first two cases lies in the possibility of varying the location of the fixed joints (the first case) or maintaining them at the same position (the second case).

In Fig. 21, an example of *3 postures and fixed* can be visualized. The three locations of the solid are plotted in pink. The end-point of the solid is colored in red, and by clicking in it, the position and orientation of the solid can be varied. In this way, we can modify their location and the program displays automatically the resultant mechanism. Another example with four postures is shown in Fig. 22.

3.2.3 Function generation

The method known as Function generation is used whenever we want the end-effector to follow a trajectory as a function of the input of the mechanism. In the case of the four-bar, this relation is found between the angular values of the input and output actuators. To set these values, the user has to select Function generation in the Mode icon. Then, a menu is displayed in which the angular values of the input angle θ_1 and the output angle θ_2 can be typed. Besides, the possibility of establishing reference values exists.

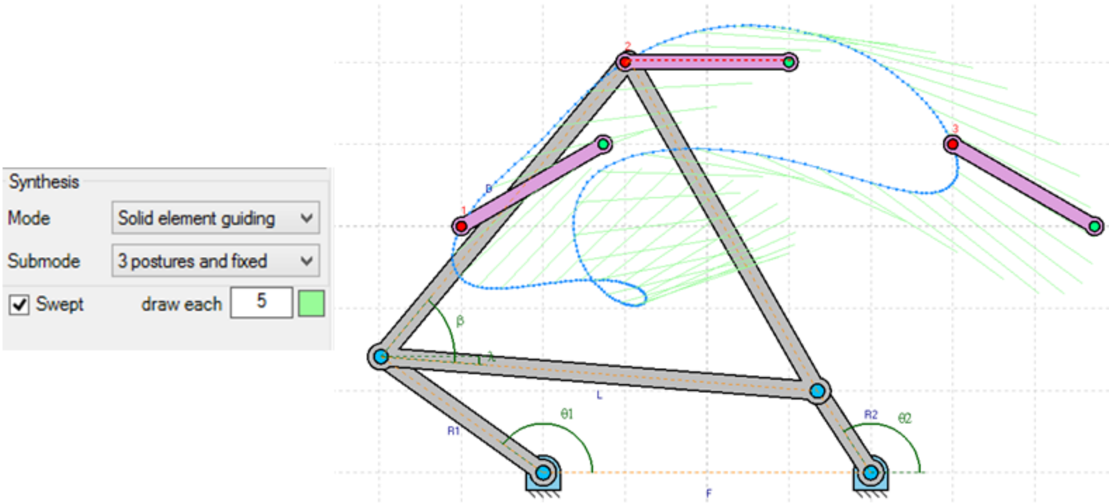


Figure 21: Solid element guiding with 3 postures

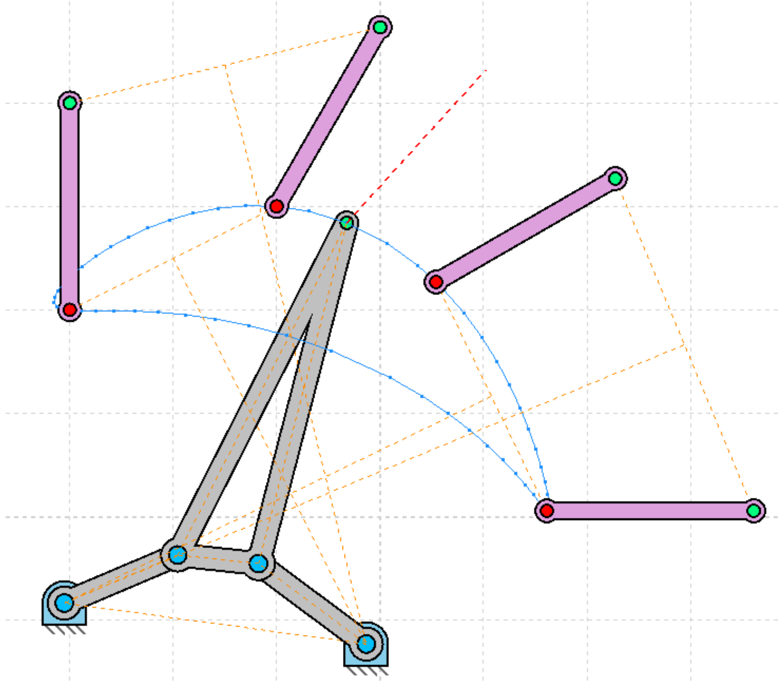


Figure 22: Solid element guiding with 4 postures

4 Statics

4.1 Main window of Statics module

GIM software incorporates an additional module intended for static analysis of several planar structures, specially planar trusses.

As in previous cases, the first step is to define the geometry of the truss, which is done using the Geometry module and following the instructions given in Section 2.1.2. As an illustrative example, the planar truss shown in Fig. 24 will be analyzed. Before performing the static analysis of the truss it is essential to verify that the structure has zero

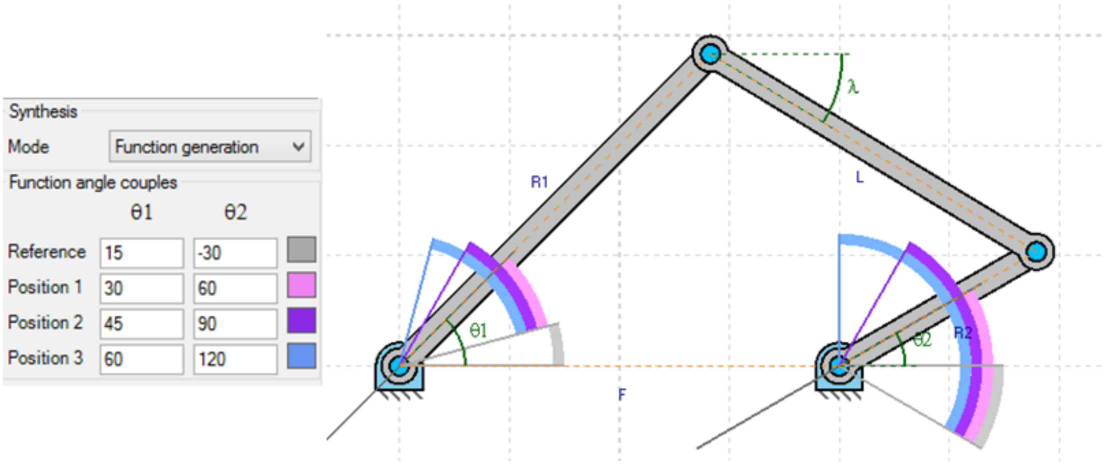


Figure 23: Example of Function generation

degrees of freedom. Otherwise, something has not been correctly defined and needs to be changed accordingly.

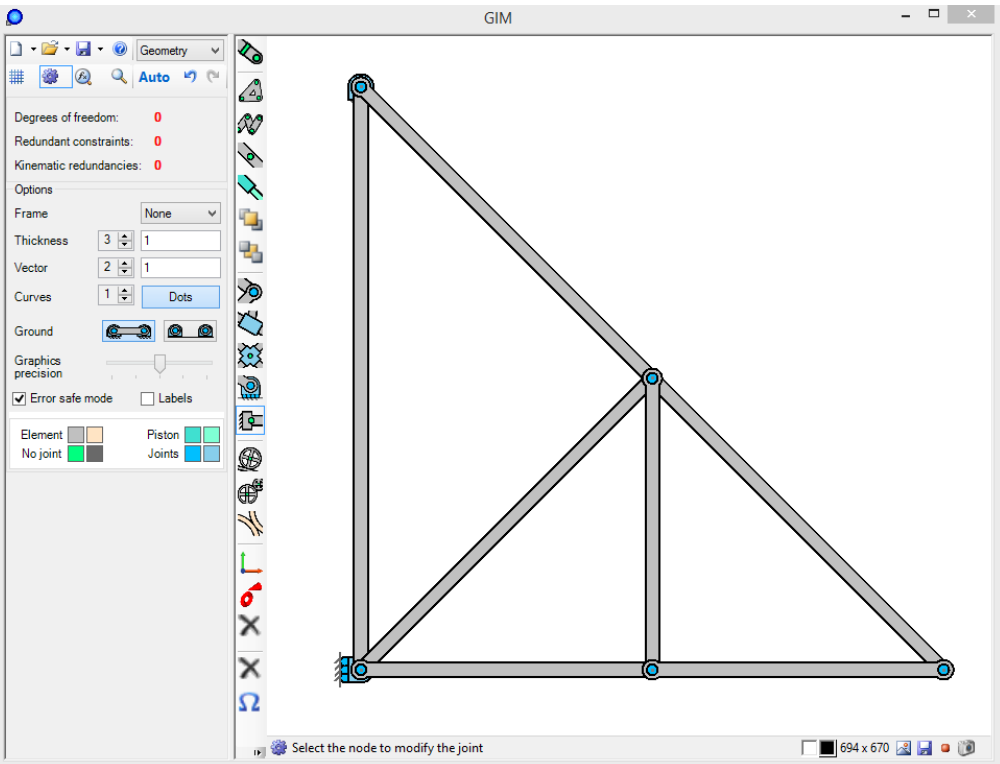


Figure 24: Illustrative example of a planar truss

The next step is to get inside Statics module, located at the displayable top menu in which all the main modules appear. By selecting Statics module a new screen in which the associated options appear will be created, as it can be observed in Fig. 25. The vertical bar closed to the drawing screen contains several options of Statics module which are the following:

- Creating new nodes
- Adding forces
- Adding moments
- Adding distributed forces
- Incorporating parabolic wires

4.2 Capacities of Statics module

An example of how to add a specific force on any of the nodes is shown in Fig. 25. The value of the force can be modified by specifying the corresponding values in the menu which appears at the bottom on the left-hand side. In this case, we have specified a unitary vertical force, from top to bottom, so the value is -1 in the vertical direction.

Similarly, moments can be applied to a node defined in any bar of the truss. For that, once the geometry of the truss has been created the user can define a point by selecting the *New node* option, and then clicking on the bar. Its location can be varied by activating the *Modify* icon.

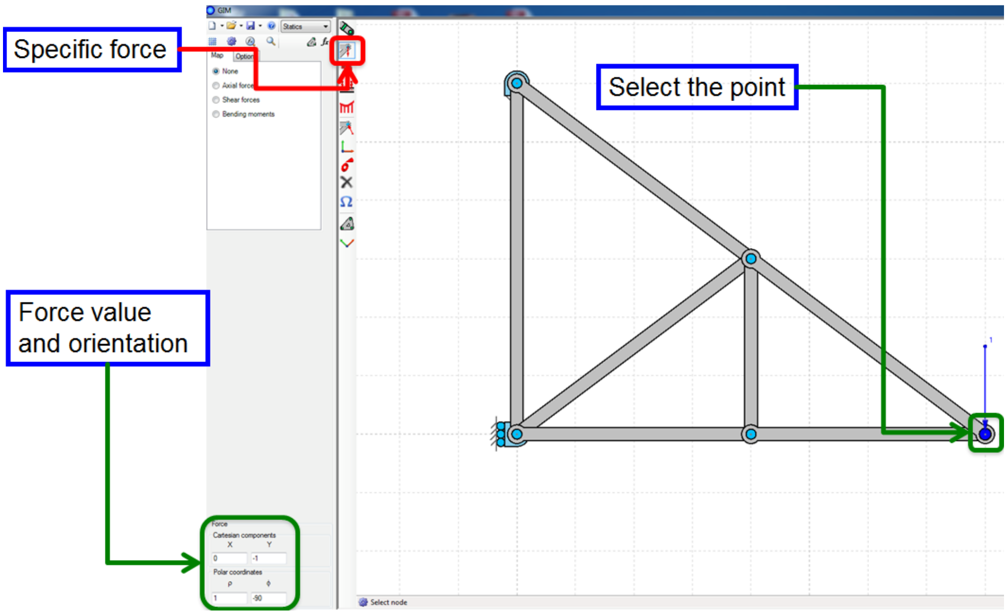


Figure 25: Adding specific forces

Besides, parabolic wires can be added to any bar of the structure. For that, the corresponding icon named *Parabolic wire* has to be selected, and then click on the nodes where the wire will have its starting and final point respectively. When activating the *Parabolic wire* icon, a new menu will be displayed at the bottom. This menu contains many related parameters that can be varied by the user, and they are automatically changed in the drawing screen.

The program permits also obtaining the *Free solid diagrams* for any of the bars and solids integrating the structure under analysis. The instructions to visualize these diagrams are shown in Fig. 26. Once the icon is activated, the user just clicks on the corresponding solid and the diagram will appear at the right-hand side.

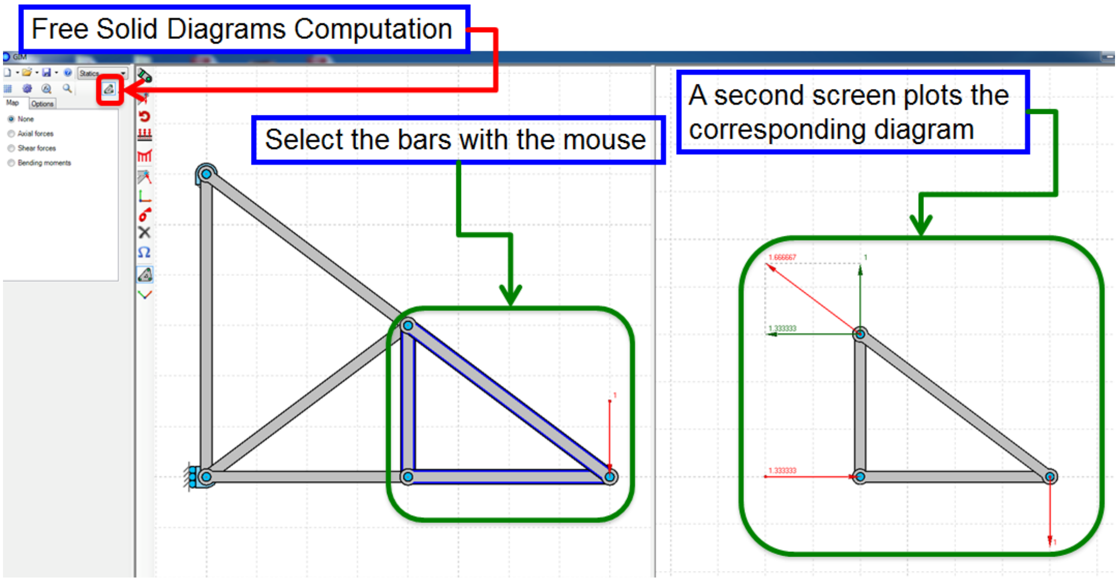


Figure 26: Obtaining free solid diagrams

On the other hand, the *Map* menu presents the following options:

- Axial forces
- Shear forces
- Bending moments

Selecting any of these options the values of the axial forces, shear forces or bending moments can be obtained and visualized on the drawing screen. Besides, an specific color is given for each value so that it is easily detected which bars have the same force or, for example, which ones suffer the maximum or minimum forces or moments.

Following with our example, if we select the option of *Axial forces*, their corresponding values will appear as shown in Fig. 27.

Many examples of planar trusses can be found in the library of the software. Select the option *Open Library*, at the top, and choose the option $0DoF$, where the geometry of many well-known planar trusses is already defined. Then, all the options of Statics module can be used to analyze the selected truss.

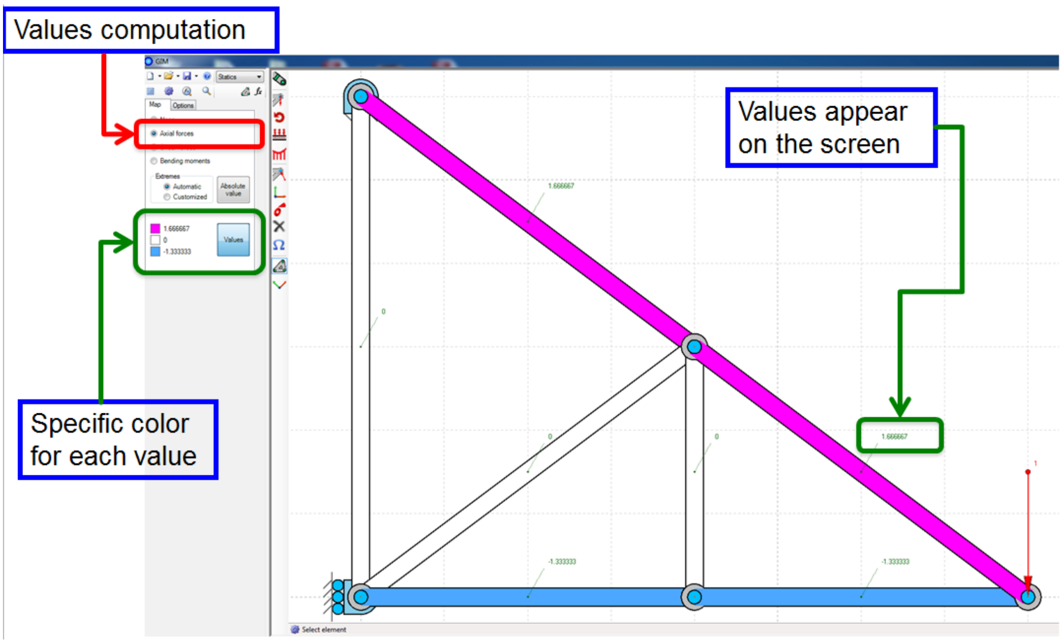


Figure 27: Values of axial forces

5 Examples

In the following link the user can find several videos illustrating various mechanisms that can be analyzed using GIM software.

http://www.ehu.es/compmech/software/

These videos, shown at the right-hand side of the web page, contain many examples explaining step-by-step how to create the geometry of the mechanisms, how to perform the kinematic analysis and visualize the motion of the mechanism, getting into the dimensional synthesis of the four-bar mechanism and perfomr static analysis of planar trusses.