2 User Interface

The presented program is a stand-alone application designed for the Macintosh$^{®}$ platform. The program and the corresponding files are graphically represented by distinctive icons (Fig. 2–1) appearing in the windows of the operating system.

Fig 2–1: Graphic representations of the program and the corresponding file.

Moreover, the program supports an user interface which enables via windows, menus and graphic elements an organized and simple formulation of an arbitrary kinematic system, an easy control of the computation and a clear representation of the computed data. It has been designed according to the *User interface guidelines* of a Macintosh$^{®}$ platform to a great extent.

It basically consists of the following elements:

- Project windows with various tools.

- Menus.

- Graphic elements used for the definition of the topology of kinematic systems.

- Diagram elements for graphic representation of computed data.

- Various input windows for the definition of the constant parameters and functions.
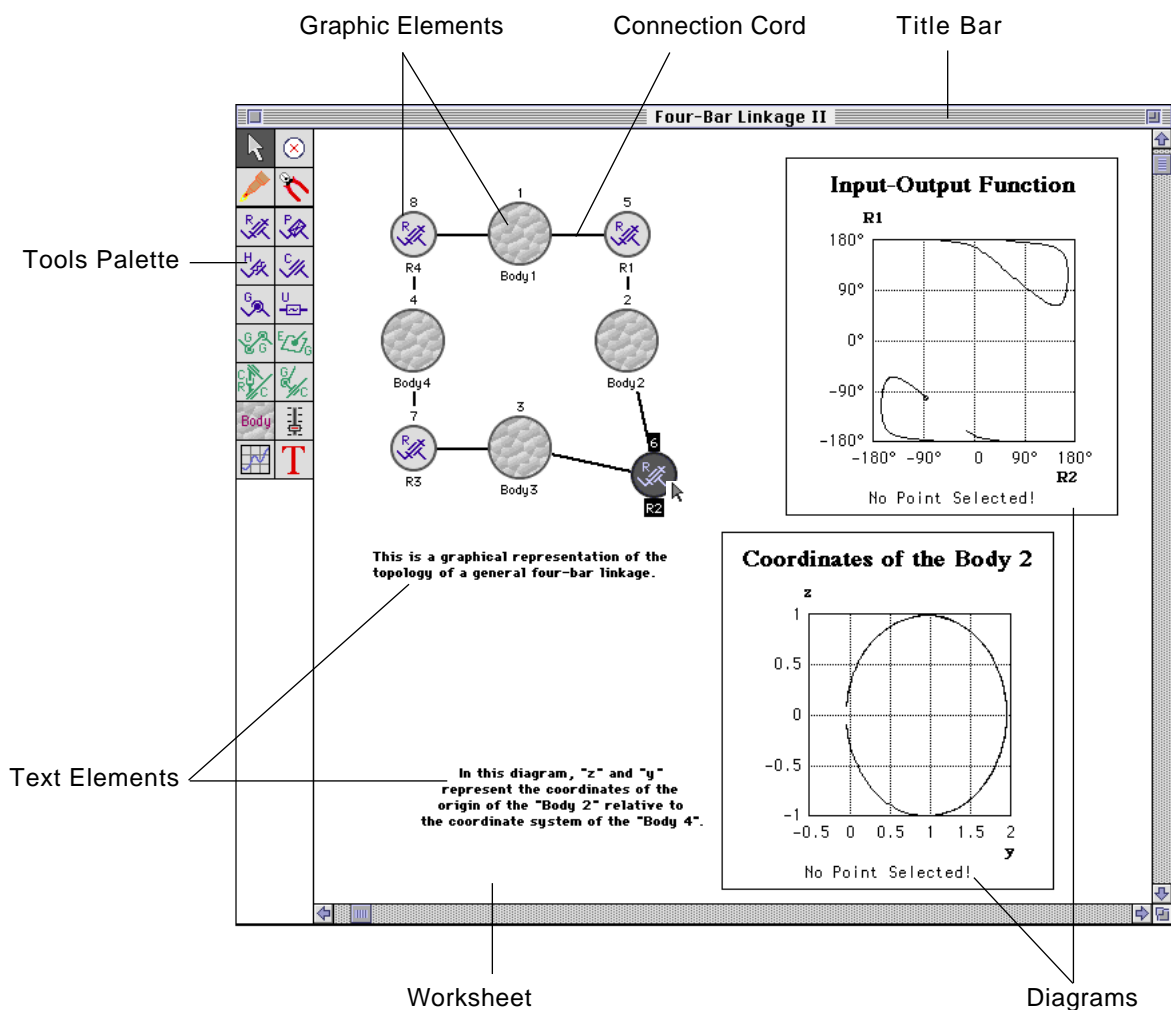
2.1 Project Window

Each examined kinematic system is associated with its *Project window* which represents the main component of the user interface (see Fig. 2–2). Moreover, it is possible to open more than just one *Project window* at the same time. However, various commands, tools and computation procedures can be used for the currently active window and the corresponding kinematic system only. The most important parts of a *Project window* are the title bar where the name of the project is shown, the tools palette and the worksheet.

2.1.1 Worksheet

On the worksheet graphic elements which represent various elements of kinematic systems, diagrams and text elements can be generated and displayed (see Fig. 2–2). These graphic elements can be, like in the common graphic editors, selected and dragged to any position on the worksheet. Moreover, the selected elements can be copied, pasted and duplicated by using the "duplicate" command or deleted by pressing the *Delete* key. The size of the worksheet can be specified by the user and any of its regions can be accessed by using the scroll-bars if it is larger than the *Topology window*.

2.1.2 Tools Palette

A tool is activated by clicking the field with its symbol in the tools palette (see Fig. 2–2). After a tool has been selected, the shape of the cursor in the worksheet is changed appropriately indicating the currently active tool.

Fig 2–2: *Project window* with graphic elements representing the topology of the linkage, text elements and diagrams.

The tools palette is divided into two main sections. Its first section contains tools for manipulation on various graphic objects and data displayed in the worksheet (See Fig. 2–3). It contains following tools:

- *Selector Tool*

It is used to select and drag graphic objects on the worksheet and to activate input dialog boxes of structural or control elements.

- *Point Picker Tool*

It is used to display the accurate coordinates values of any point on a curve in a diagram element. By clicking a curve in a diagram element, the closest point is selected and its coordinates are displayed on the bottom of the diragram element. In the case that the computed data of a kinematic system is shown in more than one diagram element, the point which corresponds to the same solution set of variables as the clicked point is selected in each diagram (See subsection 2.5.1).

- *Connecting Tool*

- With this tool a body or a control element can be connected to any of the created joints or constraint elements. The *Connecting Tool* doesn't connect two elements of the same type, and it allows to make only two connections to each joint or constraint element. Furthermore, it doesn't allow any connections between the control elements and bodies. After selecting the *Connecting Tool*, the actual connection is established in three steps as follows:

1.) Click with the *Connecting Tool* the symbol of one of the elements on the worksheet.

2.) Pull the connection cord from the initially selected element without releasing the mouse button till the second element is targeted.

3.) If the targeted element can be connected to the initially chosen element, it is highlighted and the connection is established as soon as the mouse button is released.

- *Nippers Tool*

It is used to disconnect the connection cords which were created with the *Connecting Tool*. After selecting the *Nippers Tool*, the nippers cursor opens as soon as it is close enough to any cord which can be cut by clicking the mouse button.
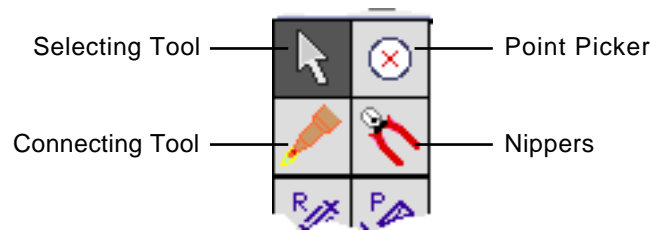
Fig 2–3: Fields of the tools palette where the manipulation tools are activated.

The second section of the tools palette contains the tools for the creation of elements of kinematic networks, diagram elements and text elements (See Fig. 2–4).
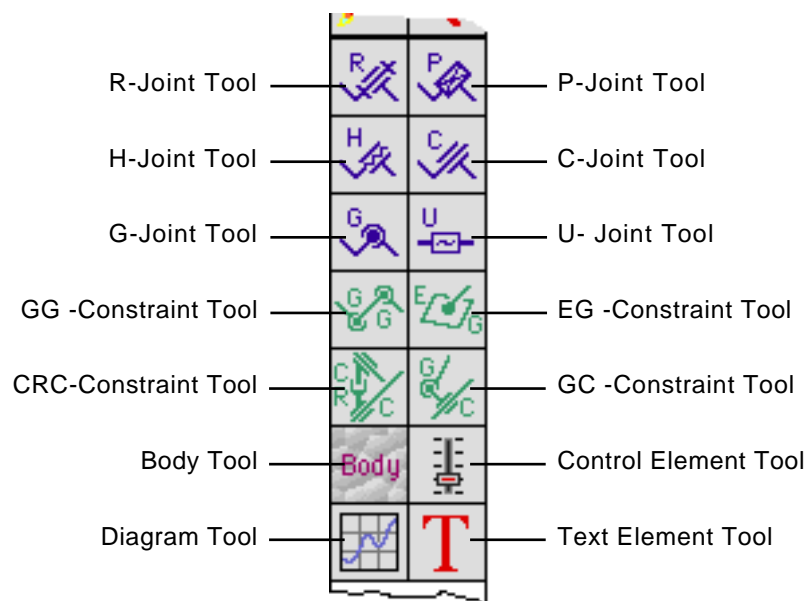
Fig 2–4: Fields of the tools palette where the creation tools are activated.

After selecting a creation tool, a new element is generated by clicking into the worksheet of the *Topology window.*

The creation tools are:

- *Joint Tools*

These tools are used for the creation of all types of joints described in the section. For each joint type a specific tool is provided and activated by selecting the appropriate tool field of the tools palette which is marked with the symbol of the joint.

- *Constraint Element Tools*

These tools are used for the creation of constraint elements discussed in the section.

***- Body, Control Element** and *Diagram tools***

They are used for the creation of body, control elements and diagram elements, respectively.

- Text Tool

It is used to insert text elements anywhere on the worksheet. By selecting the *Text Tool* and clicking the spot in the worksheet, thus defining the position of the desired text, the dialog box is shown in which the text can be written and various fonts, styles and sizes can be chosen.

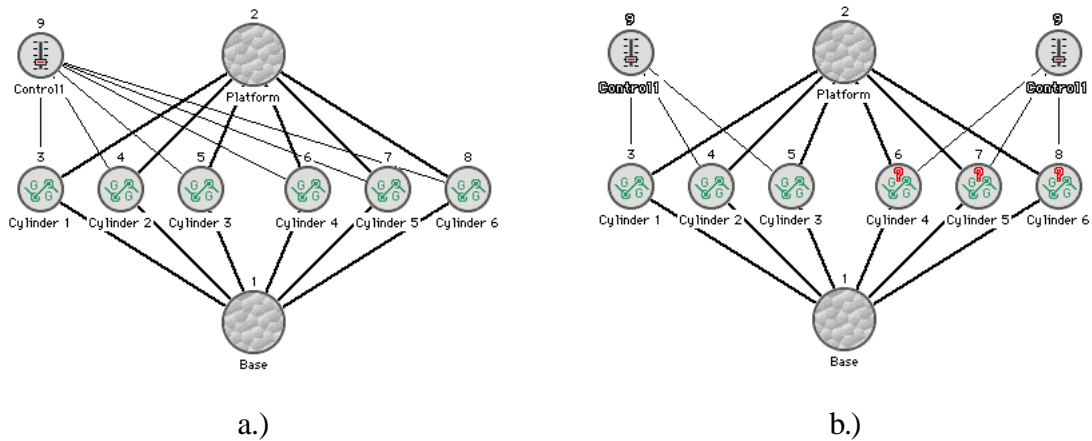## 2.2 Graphical Representation of Structural and Control Elen

Each element of a kinematic system is represented by an icon which corresponds to its type (See Fig. 2–5). At the bottom of each icon a label with the default element name is attached while on the top the current element number is added. Both, the name label and the current number can be changed by the user in the same way as the file names on the desktop of the Finder[®] on the Macintosh[®] computers.
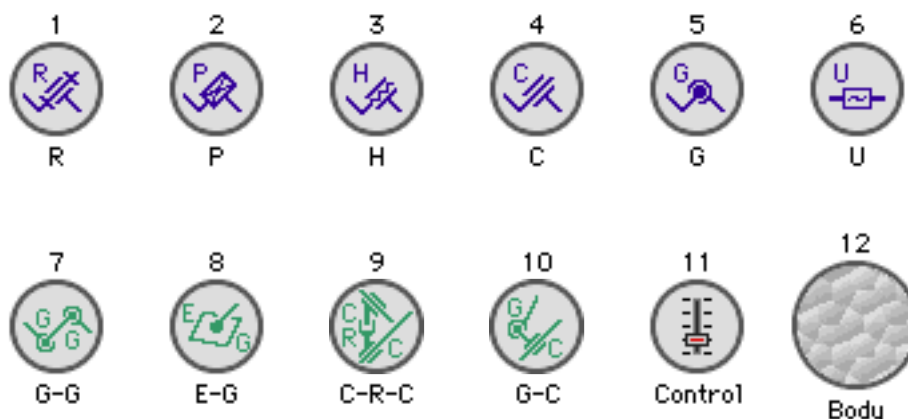
Fig 2–5: The symbols of provided joints, constraint elements, control element and body.

The symbols contain a question mark as long as their element parameters are not defined or necessary connections are missing.

Since a graphic representation of a kinematic system can be sometimes very complex, each element of a kinematic network can be represented by more than just one graphic element, the so called *Aliases*. Each of these *Aliases* can be positioned anywhere on the worksheet and connected to other elements (see Fig. 2–6). Their distinctive style of the name label and the object number indicates that a graphic element belongs to a group of graphic elements representing one structural or control element.

a.)   b.)

Fig 2–6: The equivalent topology representations a.) and b.) of a Stewart platform. In the case b.), the control element is represented by two graphic elements (*Aliases*).

2.3 Defining Parameters of Structural and Control Elements

The parameters of a structural or control element are determined in a special input box which is opened either by double clicking the symbol of an element on the worksheet or by choosing command *Object Info* from the *Object Menu*. An input box is divided into sections and its configuration depends on the type of a structural or a control element. In the title bar of each input dialog box, the name of the current element is shown while its type, corresponding object number and graphic symbol are displayed below the title bar.

2.3.1 Input Boxes of Revolute, Prismatic, Helical, Cylindrical and Global Joint

These types of dialog boxes are very similar to each other and they are divided into the following sections:

- The *Parameters* section

It allows either a definition of the general constant geometric parameters (*Full parameters*), or the *Denavit-Hartenberg (DH)parameters* as described in the subsections 3.2.2 and 3.2.5, respectively. The type of the geometric parameters is chosen by the pop-up menu which is positioned next to the title of the *Parameters* section.

In the case of *Full parameters*, (See Fig. 2–7), two separated subsections *Body 1* and *Body 2* are shown in the *Parameters* section. In each subsection a set of vectors $\mathbf{a}_i$, $\mathbf{m}_i$ and $\mathbf{n}_i$ describing the position of the joint and the orientation of its axis are determined relative to one of the connected bodies (See subsection 3.2.2.).

At the beginning of the *Body 1* subsection, a pop-up menu is displayed which is used to choose one of the connected bodies as the reference body of the vector coordinates defined in this subsection. The name of the remaining body is shown in the title of the *Body 2* subsection and automatically defined as the reference body of this subsection. To each body name appearing in the both subsection titles and pop-up menus, the corresponding object number is attached.

In order to simplify the definition of the vectors $\mathbf{m}_i$ and $\mathbf{n}_i$, it is sufficient to determine vectors $\mathbf{m}_i$ and $\mathbf{n}_i$ which are not parallel and only the direction and orientation of the vector $\mathbf{n}_i$ has to be defined correctly. Internally, the coordinates of the vector $\mathbf{n}_i$ and the component of the vector $\mathbf{m}_i$ which is normal to the vector $\mathbf{n}_i$ are normalized.
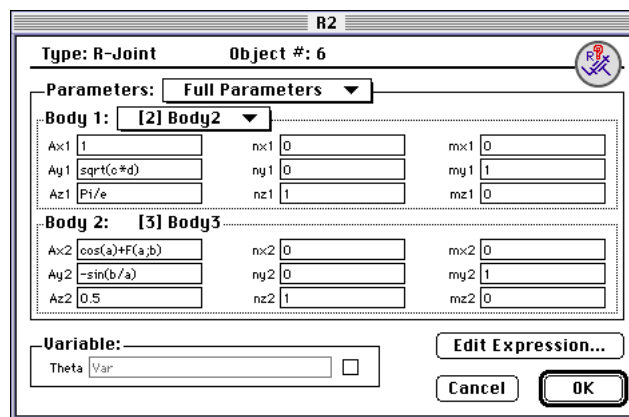

Fig 2–7: Input box -*Full parameters* mode

In the case of *DH parameters*, however, three editable input fields for the *Denavit-Hartenberg parameters* (See subsection 3.2.2)are shown and the pop-up menu in which the reference body has to be chosen is enabled (Fig. 2–8).
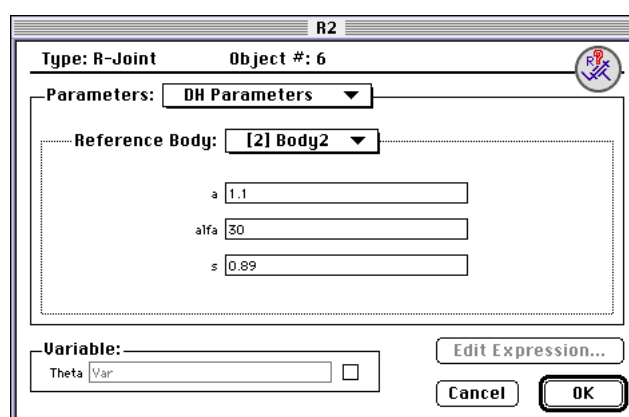

Fig 2–8: Input box-*Denavit Hartenberg parameters* mode.

The *DH parameters* of a global joint are determined with the same parameters as the revolute joint and the additional two axes of rotation are automatically determined in such a way that all axes are perpendicular to each other.

- The *Variables* section

In this section, the values of any variable parameter of the current joint can be defined either by a constant or a function which uses one or more variable parameters, supplied by control elements. In this section, an editable text field is assigned to each variable parameter and activated by clicking the check box, which is positioned to the right of the field (Fig. 2–9). If this text field is activated, the value of the variable can be appropriately determined.
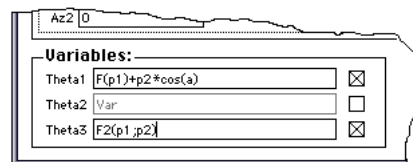
Fig 2–9: A *Variables* section of the input box of the global joint. The expressions for *Theta1* and *Theta3*, contain also variables *p1* and *p2* which are supplied by the control elements.

In general, parameters can be defined by various mathematical expressions, which can be typed directly into the corresponding editable text fields. These mathematical expressions are checked and evaluated one after another by the parser when the box is closed with the *OK* button. In the case that either the syntax of any expression is not understood by the parser or a runtime error has been detected, the box is not closed and the cursor is positioned to the editable field where the problems have been first encountered.

All changes made to the parameters are ignored if the dialog box is closed by clicking the *Cancel* button.

Additionally, the expressions of parameters can be defined by using the *Math editor* where all available functions and constants are listed and arbitrary expressions can be defined and analyzed. Moreover, also new functions and constants which are specific only to the current joint and used for the definitions of its parameters can be defined in the *Math editor*. In order to open the input box of the *Math editor*, one of the editable text fields has to be first activated and the button *Edit Expression...* has to be clicked.
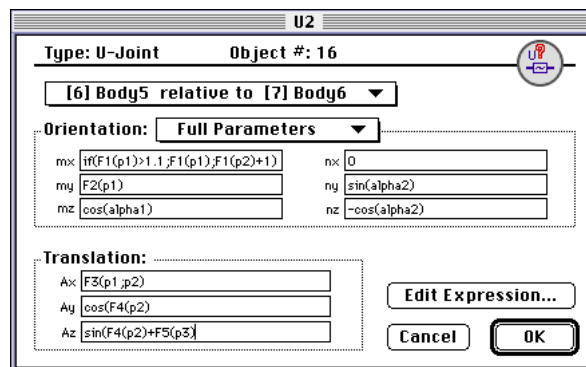
2.3.2 Input Box of User Defined Joints

In this input dialog box, an arbitrary relative transformation of the two connected bodies can be defined.

At the top of this input box a pop-up menu is displayed where one of the connected bodies is chosen as the reference body. The input dialog box is further divided into:

- The *Orientation* section

In this section the relative orientation of the two connected bodies is defined by the two vectors $\mathbf{m}_i$ and $\mathbf{n}_i$ (Full-parameters, See Fig. 2–10) which define the coordinate system of the rotated body relative to the reference body (See subsection 3.2.2).

Fig 2–10: Input box of a user defined joint (*Full parameters mode*).

- The *Translation* section

It is used to determine the coordinates of the vector $\mathbf{a}_i$ describing the relative translation between the origins of the body coordinate systems of the two connected bodies.

The vector coordinates can be determined by constant values or any function which is accepted by the *Parser*. In order to define a movable joint at least one vector coordinate has to be determined by an expression which uses one or more variable parameters. These variable parameters are supplied by control elements which are connected to the user defined joint.The buttons of this input box function in the same way as those in the previously discussed input box.

2.3.3 Input Box of Bodies

The input box of a body is used for the definition of point and direction vectors in a body coordinate system. The point and direction vectors are needed to compute the body position and

orientation relative to the coordinate system of an arbitrary reference body of the kinematic system. This box contains the following elements:
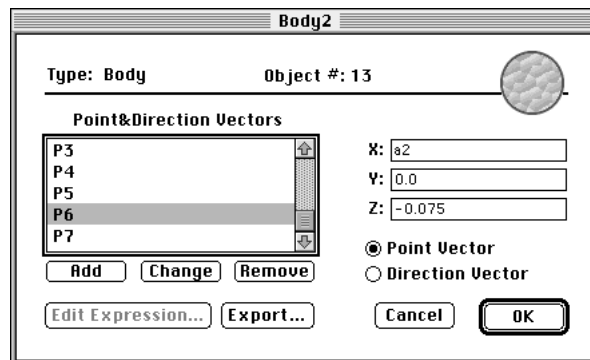

Fig 2–11: Input box of a body.

- The *Point&Direction Vectors* scroll-list

In this scroll-list all direction and point vectors of the currently edited body are listed. By selecting a vector in the scroll-list, its coordinates are immediately displayed in the three editable text fields which are positioned next to the scroll-list. Additionally, appropriate radio button positioned below the three editable text fields is activated indicating the type of the currently selected vector. The coordinates of the selected vector can be determined by using various constants and functions which are defined and analyzed by the parser.

- The *Change* button

- By clicking this button, positioned underneath the scroll-list, the changes made to the vector coordinates are accepted only if the parser doesn't detect any syntax or run-time errors while checking the expressions in the editable text fields. Otherwise, the cursor is positioned to the editable text field where the first error has been encountered.

- The *Add* button

By clicking this button, a new vector is created and added to the *Point&Direction Vectors* scroll-list. Before adding a vector to the *Point&Direction Vectors* scroll-list, its coordinates have to be defined and the type of the vector has to be determined by clicking either the *Point Vector* or *Direction Vector* radio button. A unique name among all vectors of the current body is automatically assigned to the new vector. The names of point vectors are composed of the letter *P* and its current number while the names of direction vectors begin with the letter *D* instead. Also in this case the expressions of the vector coordinates are controlled by the parser before the vector is added to the list.

- The *Remove* button

By clicking this button, the selected vector is removed from the *Point&Direction Vectors* scroll-list.

- The *Export...* button

By clicking this button, the *Export Data* dialog box of the body is opened (see Fig. 2–12).In the scroll-text field an arbitrary text can be written. This text is used for generating export text files (See "Exporting Data" on page 34.). It is automatically attached to the export text file when the file is generated. With the help of separators and vector notation we are able to specify strings which are replaced by the actual computed data.
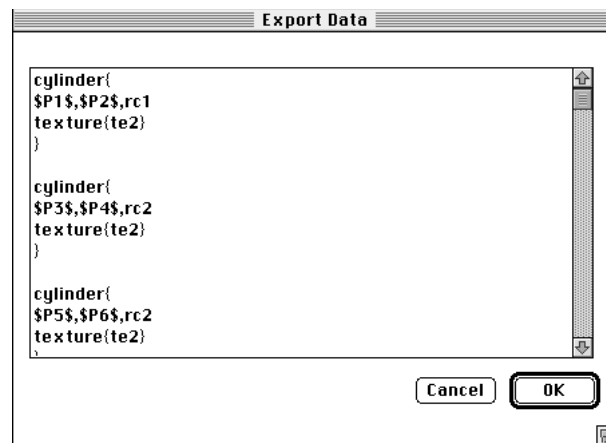

Fig 2–12: *Export Data* dialog box.

2.3.4 Input Box of Control Elements

This input box (Fig. 2–13) is used to define the properties and the name of the parameter which is supplied by this control element. In the main section, the following data can be defined:

- The *Variable Name* text field

In this editable text field the name of the parameter is shown and can be changed. However, it has to be unique among all constant and variable parameter names used for the definition of the entire kinematic system. To each variable parameter a default name is assigned when the corresponding control element is created.

- The *Test Value* text field

In this editable text field the test value of the parameter is set in order to test all functions which use the current parameter. In the case that the test value causes a run-time error in any function using this parameter, the corresponding elements of the kinematic system are set to *undefined* which is indicated by a question mark in their symbols.

- The *Max. Increment* text field

This text field is used to determine the maximal increment of the variable parameter. The maximal increment is used in the numerical iteration in order to find a controlled variable and check if the computed solution vector is unique.

- The *Range* subsection

In this subsection the range of the current parameter is determined in the *Max*. and *Min*. text fields. Moreover, by checking the *Periodic* checkbox the current parameter can be defined as a periodic variable parameter.
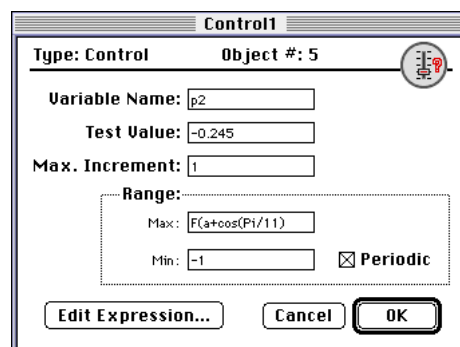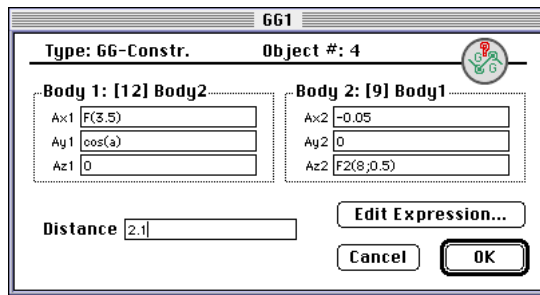

Fig 2–13: Input box of a control element.

Beside the usual syntax analysis of the mathematical expressions also the name of the parameter, the test value, maximal increment and the range are checked when the *OK* button is clicked. In the cases that the name of the parameter is not unique, the test value is out of range, the increment is greater than the range or the range limits are not correct, the box is not closed and the cursor is positioned to the editable field where the first error has been encountered.

2.3.5 Input Dialog Box of Global-Global Constraint Elements

This input dialog box is used to determine a point in each of the two connected bodies and the distance between these two points (See 3.2.3.).

GG1

Type: GG-Constr.   Object #: 4

Body 1: [12] Body2
Ax1 F(3.5)
Ay1 cos(a)
Az1 0

Body 2: [9] Body1
Ax2 -0.05
Ay2 0
Az2 F2(8;0.5)

Distance 2.1

Edit Expression...

Cancel   OK

Fig 2–14: Input box of a global-global constraint element.

The input dialog box of Global-Global constraint elements consists of the following elements:

- The *Body 1* and *Body 2* sections

In the *Body 1* section the coordinates of the point C_{i-1} (vector $\mathbf{a}_{i-1}$) are determined in the co-ordinate system of the body whose name is displayed in the title of this section.

The coordinates of the second point in the other body are determined in the same way in the *Body 2* section.

- The *Distance* text field

This edit text field is used to determine the distance between the two points which were de-fined in the sections *Body 1* and *Body 2*.

The point coordinates can be defined by expressions which contain only constants while the distance can be also determined by using variable parameters which are provided by one or more control elements.

2.3.6 Input Dialog Box of Eben-Global Constraint Elements

The Input dialog box of *Eben-Global* constraint elements,(Fig. 2–15), is used to determine a point in one and a plane in another of the two connected bodies (See subsection 3.2.3).

It consists of the following elements:

- The *Body 1* section

In this section the body in which the plane is fixed has to be chosen in the pop-up menu posi-tioned next to the section title. The coordinates of the direction vector $\mathbf{n}_{i-1}$ define the orienta-tion of the plane relative to the body coordinate system. Furthermore, the normal distance

between the plane and the origin of the corresponding body coordinate system is determined in the editable text field *Distance*.

- The *Body 2* section

This section is used to determine the coordinates of the point whose motion is restricted to the plane defined in the *Body 1* section. This point is fixed to the body whose name is displayed in the title of this section. Its coordinates (vector $\mathbf{a}_i$) are determined relative to the coordinate system of the body in which the point is fixed.

- The *Offset* text field

This text field is used to determine the offset of the plane defined in the section *Body 1* in which the point of the other body is positioned. The plane is moved parallel to its direction vector by adding the offset to the normal distance determined in the *Offset* editable text field. The offset can be defined by a mathematical expression containing one or more variable parameters supplied by control elements.
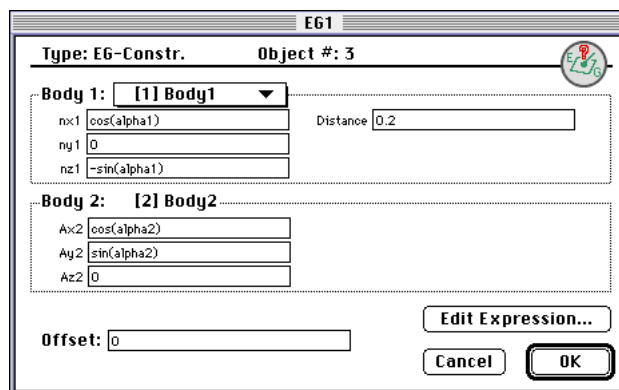

Fig 2–15: Input box of an eben-global constraint element.

2.3.7 Input Dialog Box of C-R-C Global Constraint Elements

This input box is used for the definition of the relative position and orientation of two skew lines which represent the axes of the relative rotation and translation of each connected body (See subsection 3.2.3.)
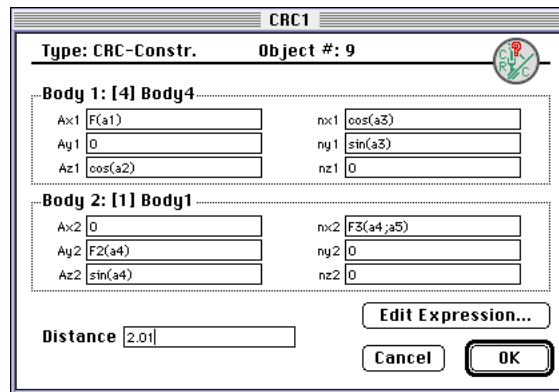
Fig 2–16: Input box of an C-R-C constraint element.

- The *Body 1* and *Body 2* sections

In each of the *Body 1* or *Body 2* section the coordinates of the reference point C_i on the axis (vector $\mathbf{a}_i$) are determined in the first three editable text fields while the direction of the corresponding axis (vector $\mathbf{n}_i$)is determined in the last three editable text fields. Note that all coordinates determined in either section are defined relative to the coordinate system of the body whose name is displayed in the title of the actual section.

- The *Distance* text field

This edit text field is used to determine the normal distance between the two skew lines which were defined in the sections *Body 1* and *Body 2*.
The point coordinates can be defined by expressions which contain only constants while the distance can be also determined by using variable parameters which are provided by one or more control elements.

2.3.8 Input Dialog Box of Global-Cylindrical Constraint Elements

This input dialog box is used for the determination of a cylindrical surface fixed to one body and a point C_i which is fixed to the other body. The motion of this point is restricted to the cylindrical surface (See subsection 3.2.3).
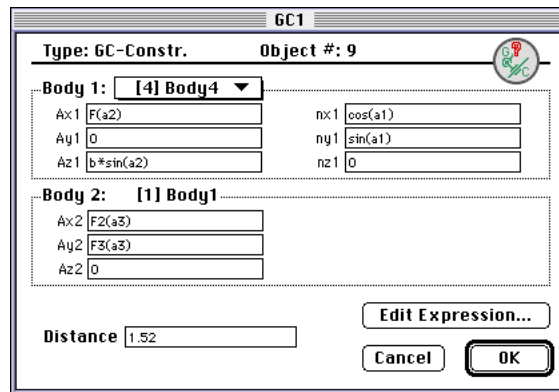
Fig 2–17: Input box of an global-cylindrical constraint element.

- The *Body 1* section

The body in which the line is fixed has to be chosen in the pop-up menu positioned next to the section title.In the section *Body 1* the coordinates of the reference point C_{i-1} on the axis (vector $\mathbf{a}_{i-1}$) are determined in the first three editable text fields while the direction of the corresponding axis (vector $\mathbf{n}_{i-1}$)is determined in the last three editable text fields. Note that all coordinates in this section are determined relative to the coordinate system of the body whose name is displayed in the pop-up menu next to the section title.

- The *Body 2* section

In the three editable text fields of the *Body 2* section the coordinates of the point C_i (vector $\mathbf{a}_i$) whose motion is restricted to the cylindrical surface, defined in the previous section, are defined relative to the coordinate system of the body whose name is displayed in the section title.

- The *Distance* text field

This edit text field is used to determine the shortest distance between the point and the line.

The point coordinates can be defined by expressions which contain only constants while the distance can be also determined by using variable parameters which are provided by one or more control elements.

2.4 Math Editor

In order to simplify the definition of constants and complicated mathematical expressions, a special math editor is provided.

The *Math editor* is a dialog box where the desired mathematical expression can be created, tested and the eventual syntax or run-time errors located. Furthermore, it allows creation of user defined constants and functions which can be used for further definition of more complex mathematical expressions.
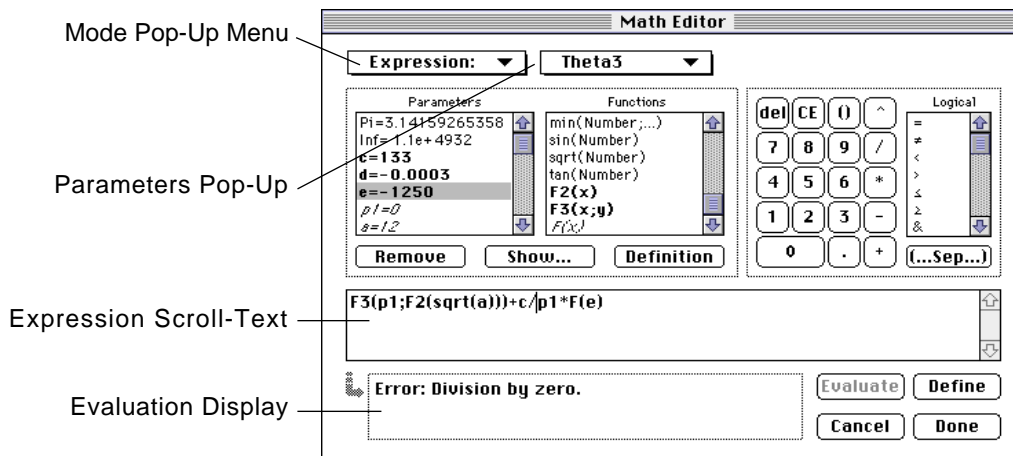
Fig 2–18: *Math editor.*

The math editor supports the following features:

- The *Mode* pop-up menu

In this pop-up menu various input modes can be chosen (Fig. 2–18). The *Expression* mode is used to define an expression for a parameter of a structural or control element. Moreover, the *Function* and *Parameter* modes are used to define new functions and parameters, respectively, which are needed for the further definitions of expressions in the *Expression* mode.

If the *Expression* mode is selected another pop-up menu is activated where the names of all currently editable parameters are listed. At the same time the definition of the parameter which is currently selected in this pop-up menu is edited in the Expression scroll-text.

Furthermore, in the case of the *Function* or *Parameter* mode an editable text field is activated where the name of the new parameter or function with its arguments can be defined.

- The *Parameters* scroll-list

In this scroll-list names and values of all currently available constant and variable parameters are displayed (Fig. 2–18). There are three types of parameters:

1.) Constant parameters which are always available, cannot be altered or removed from the list and their names in the scroll-list are written in the plain style.

2.) All constant and variable parameters (defined by control elements) which have been defined by the user, but they cannot be altered or removed from the list at the moment. This type of parameters is marked by the italic style.

3.) Constant parameters which can be added or removed from the *Parameters* scroll-list, their values can be altered and their names in the list are written with the bold face letters.

- The *Functions* scroll-list

In this scroll-list names and arguments of all currently available functions are displayed. Like in the case of the *Parameters* scroll-list, three types of functions can be shown in the *Functions* scroll-list.

- The *Definition* button

This button is used to insert the definition of a function or value of a parameter into the currently edited expression.

- The *Show* button

This button activates the *Show* dialog box in which the definition of a function or value of a parameter is displayed. The function or parameter has to be selected in the corresponding scroll-list.

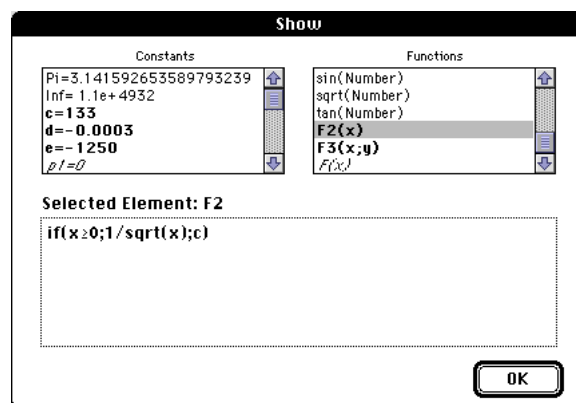**Fig 2–19:** *Show* dialog box.

- The *Remove* button

This button is used to remove parameters and functions from the *Parameters* and *Functions* scroll-list. However, only functions and parameters which are written in bold face can be removed.

- The *Keyboard*

Beside the digits, the keyboard contains also the basic mathematical operations and the *(...sep...)* button which is used to set separators in the argument lists of functions with more than one argument.

- The *Logical* scroll-list

In this scroll-list all standard symbols used in the boolean algebra are listed.

- The *Expression* scroll-text field

In this scroll-text field a mathematical expression of an arbitrary length can be created (Fig. 2–18). A mathematical expressions can use any of the functions, parameters or symbols which appear in the three scroll-lists. In order to insert a parameter or function name with its arguments into an expression, the insertion point has to be determined and the name of the parameter or function can be either typed directly or inserted by double clicking it in the scroll-list. Moreover, the whole definition of the function can be inserted at the insertion point by selecting the function name in the scroll-list and clicking the *Definition* button. When a function name along with its arguments is inserted, the first argument is automatically highlighted.

- The *Evaluation* display

In this display the values of the created expressions or error messages are shown (Fig. 2–18).

- The *Evaluate* button

This button is used to test the value of an expression displayed in the expression scroll-text. If the expression caused a run-time error or its syntax is not correct, an error message is shown in the evaluation display.

- The *Define* button

This button is used to confirm the new definition of a parameter expression and to add a new function or parameter to the corresponding list. The new expression is analyzed before the changes are accepted and an error message is shown in the evaluation display in the case of an error.

The names of new parameters and functions have to be unique, compatible with the syntax which is understood by the parser and the function arguments have to correspond to the defini-

tion of the function. If this is not the case, an alert box is shown where the encountered problem is explained.

- The *Done* and *Cancel* buttons

The *Done* button is used to close the *Math editor* and store all changes made to the parameters while the *Cancel* button is used to close the box without accepting the new changes.

- The syntax of mathematical expressions

The mathematical expressions defined in the math editor must comply with the syntax which is understood by the parser (the algorithm which evaluates these expressions). The mathematical expressions can contain the following symbols:

- characters: A,B,C,D,E,F,G,H,I,J,K,L,M,N,O,P,Q,R,S,T,U,V,W,Z,X,Y,a,b,c,d,e,f,g,h,i,j,k,l,

m,n,o,p,q,r,s,t,uv,w,z,x,y,1,2,3,4,5,6,7,8,9,0;

- comma: ".";

- mathematical operation symbols: $+,-,/,*,\wedge,=,\neq,<,>,\leq,\geq,\&,|,\sim$;

- the parentheses: ();

The names of parameters or functions must begin with a letter and have to be unique among all parameter or function names appearing in the math editor scroll-lists.

Each function has specific name which is defined in the editable text field next to the modes pop-up menu (Fig. 2–20). Each function name is accompanied with a list of its variables which are separated by the separator ";". Moreover, between the function name and the argument list should be no space.
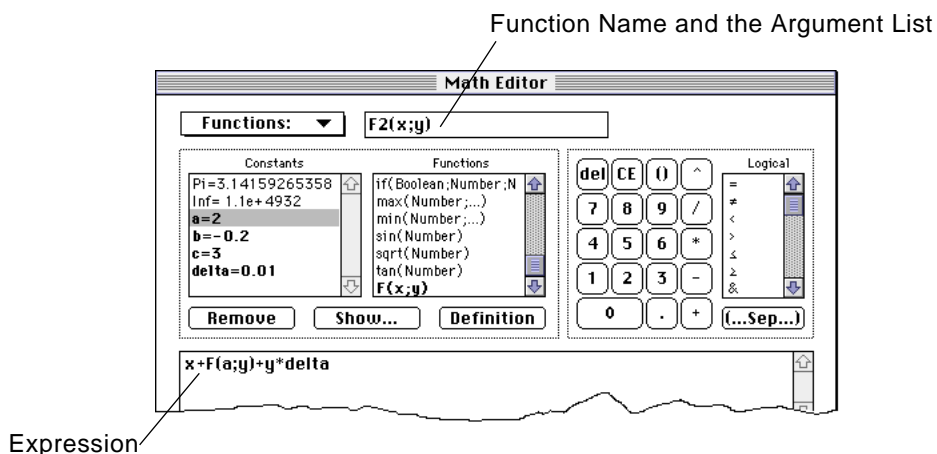
Function Name and the Argument List

Expression

Fig 2–20: Defining a function.

2.5 Diagram

Diagrams can be created on the worksheet of the *Project window* simultaneously by using the *Diagram Tool* in the same manner as in the case of the structural or control elements. More-over, they are graphic objects which can be selected, dragged, modified and deleted. Each diagram element consists of the following parts (See Fig. 2–21):

- The diagram title.
- The area where the curves are plotted.
- The labels of the vertical and horizontal axes which indicate the selected variable parameters.
- The vertical and horizontal scales.
- The display where the coordinates of the selected point are shown.
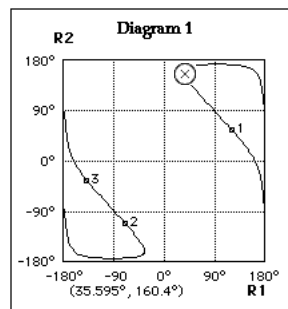

Fig 2–21: Diagram element.

2.5.1 Function of the *Point Picker* tool and Markers

Markers are set with the *Point Picker* tool. Whenever the *Point Picker* tool is moved close enough to a point belonging to any curve in a diagram element, the coordinates of that point are instantly displayed at the bottom of the diagram. Moreover, if there is more than one diagram element on the worksheet, in all diagrams the corresponding local coordinates are simultaneously displayed. If the *Point Picker* tool is close enough to a point (coordinates displayed) a new marker is set simply by clicking that spot. A marker is displayed as a tiny rectangle with specific number on the side in each diagram. Each marker can be selected by clicking it with the *Selector* tool and deleted by pressing the *Delete* key. Whenever a marker is selected the current coordinates of the marker are displayed in all existing diagrams simultaneously.

2.5.2 Configuring a Diagram

The variable parameters, title and scales are defined in a special *Diagram Info* dialog box (see Fig. 2–22). It is opened either by double clicking the diagram element or by choosing the *Object Info...* command from the *Object* menu.
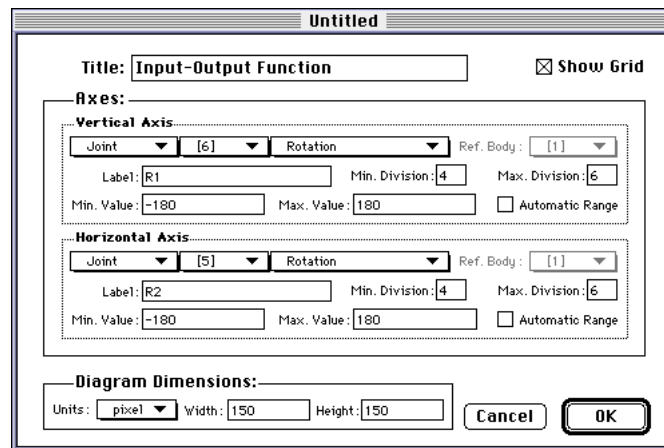
Fig 2–22: *Diagram info* dialog box.

The *Diagram Info* dialog box contains the following elements:

- The *Title* text field

The diagram title is determined in this text field.

- The *Show Grid* checkbox.

If this checkbox is checked, the grid in the diagram is shown.

- The *Axes* section

This section is divided into the subsections *Vertical Axis* and *Horizontal Axis*. In each subsection a variable parameter is chosen with the help of the three pop-up menus as follows:

1.)In the first pop-up menu the type of an element or a special variable is chosen. In this pop-up menu all types of elements which are used in the current kinematic system and special variables are listed.

2.)After choosing a type of an element, the object number of a particular element is selected in the second pop-up menu where all available object numbers belonging to the elements of the same type are collected.

3.) Finally, in the third pop-up menu the variable parameter of the chosen structural element is selected. If the chosen variable parameter needs a reference coordinate system, as for example a coordinate of a point vector defined in a body, the pop-up menu *Ref. Body*, where the desired reference body can be selected, is automatically activated.

- The *Label* text field

This text field is used to determine the label which represents the chosen variable.

- The *Min. Division* and *Max. Division* editable fields

In these two text fields the minimum and maximum number of the grid divisions are determined.

- The *Min. Value* and *Max. Value* editable fields

In the editable fields *Min. Value* and *Max. Value*, however, the range of the scale is defined. The limits of the number of divisions and the value range are used in the special algorithm which optimizes the scale of a coordinate axis.

- The *Automatic Range* checkbox

-If this check box is checked, a special algorithm determines a new scale of an axis automatically by using the extreme values of the chosen parameter. As soon as a new extreme value of the current parameter is found, the diagram is automatically updated with the new scale.

- The *Diagram Dimensions* section

This section is used to determine the dimensions of the plotting area of a particular diagram. In the *Units* pop-up menu, the unit can be defined as centimeter, inch or pixel while the width and the height of the plotting area are determined in the *Width* and *Height* editable fields.

Moreover, the scale unit is automatically defined in degrees if the variable parameter represents an angle of rotation. By clicking the *OK* button, the data in the editable text fields is checked and the *Diagram Info* dialog box is closed if no errors occurred. In the case that the limits of the number of divisions or value range is not defined correctly, the cursor is automatically positioned to the editable text field where the error was detected.

2.6 The Menus

The menus in the presented application supply commands for file handling, manipulation on graphic objects and parameters of kinematic systems as well as commands for the control of computation.

2.6.1 The *File* Menu

Fig 2–23: *File* Menu

The *File* menu contains standard commands which are used for the file handling, setting of preferences, page set-up, printing and quitting the application. The command *Preferences* opens the *Preferences* dialog box which contains a special scroll-list where the following preference cases can be activated:

- *Icons* preferences are used to set the font, the text style and the size of the name and the number labels attached to the symbol of a structural or control element. Additionally, after checking the *Snap Icons* check-box, the elements snap to an invisible grid on the worksheet. By checking the *Retain Tool* check-box, the tool is not automatically changed back to the *Selector tool*.
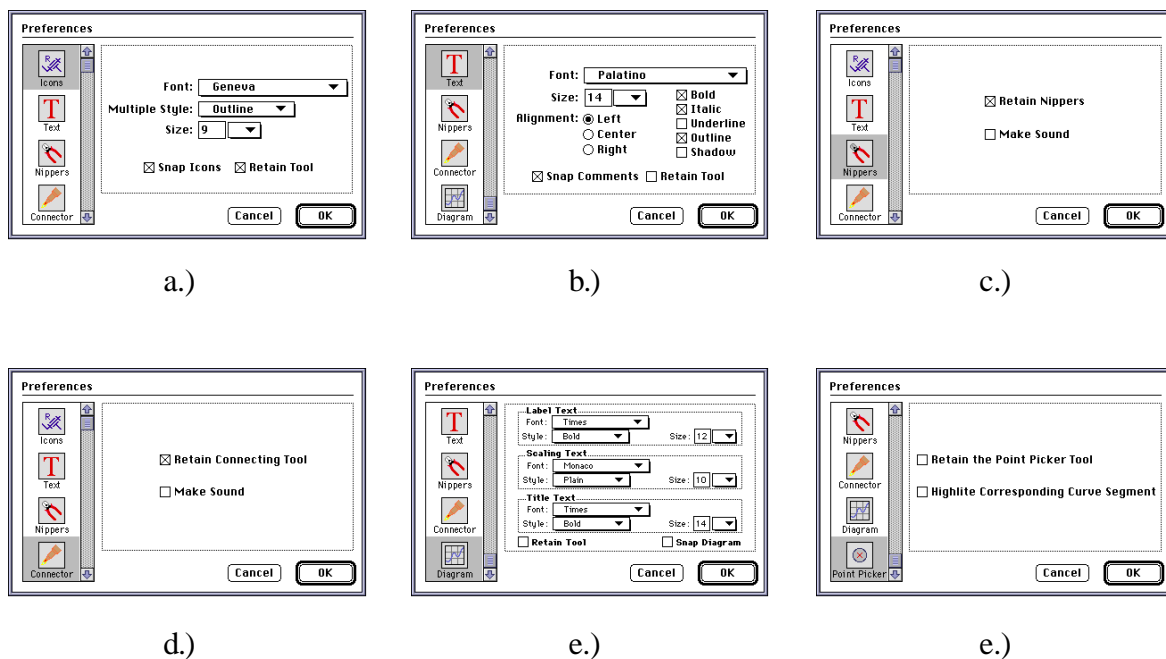

a.)   b.)   c.)

d.)   e.)   e.)

Fig 2–24: Cases of the *Preferences* window.

- In the *Text* preferences case, the default font as well as the style, size and justification of a text element can be determined. Similar to the *Icons* preferences case, there are also the *Snap Comments* and *Retain Tool* checkbox.

- The *Nippers* and the *Connector* cases are basically the same. They contain the *Retain Tool* and *Make Sound* checkboxes.

- The *Diagram* case is divided into the *Label Text*, the *Scaling Text* and the *Title Text* sections. In each of these sections, the font as well as the style and size of the corresponding text type are determined. It contains also the *Retain Tool* and *Snap Diagram* checkboxes.

- The *Point Picker* case is very similar to the *Nippers* and the *Connector* cases. A curve segment is emphasized in all diagrams by selecting a point on it, if the *Highlite Corresponding Curve Segment* check-box is checked.

The new settings are immediately applied to all existing elements on the current worksheet as soon as the *Preferences* dialog box is closed by clicking the *OK* button.

If the *Preferences* dialog box is closed by clicking the *Cancel* button, all changes made in the preference cases are ignored.

2.6.2 The *Edit* Menu

Beside the standard commands, this menu contains the special command *Alias* which is used to generate aliases representing one or more structural or control elements as follows:

1.) Select one or more graphic elements.

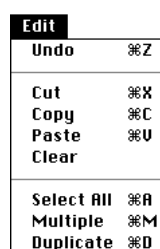2.) Choose the command *Alias* in the *Edit* menu or type the *Command* and *M* keys simultaneously.

```
┌─────────────────┐
│ Edit            │
├─────────────────┤
│ Undo       ⌘Z   │
│                 │
│ Cut        ⌘X   │
│ Copy       ⌘C   │
│ Paste      ⌘V   │
│ Clear           │
│                 │
│ Select All ⌘A   │
│ Multiple   ⌘M   │
│ Duplicate  ⌘D   │
└─────────────────┘
```

Fig 2–25: *Edit* Menu

2.6.3 The *Object* Menu

The *Object* menu is used for manipulation on graphic elements of structural elements, control elements, diagrams and text elements.

```
┌──────────────────┐
│ Object           │
├──────────────────┤
│ Object Info...   │
│ Define Selected  │
└──────────────────┘
```

Fig 2–26: *Object* Menu

The *Object* menu contains the following commands:

- Object Info...

This command opens info dialog boxes of all selected graphical elements.

- Define Selected

By choosing this command, the parameters of all selected structural or control elements are checked. In the case that there are selected elements whose data is correct and complete but it hasn't been confirmed yet (question mark in the graphic symbol), the command *Define Selected* confirms it and the element is considered as defined.
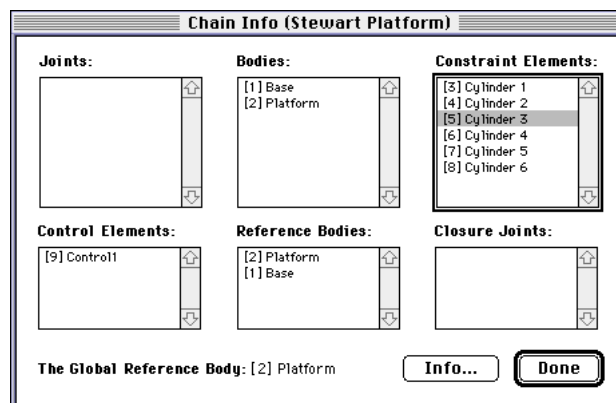
2.6.4 The *Chain* Menu

Fig 2–27: *Chain* Info.

This menu contains commands which are used for inspection and manipulation of the global data of a kinematic chain.

It supports the following commands:

- Chain Info...

This command is available as soon as a *Project window is* active and it opens the *Chain Info* dialog box.

Fig 2–28: *Chain* Info Box.

In the scroll-lists *Joints*, *Constraints*, *Controls* and *Bodies*, the names of all structural and control elements are listed. Furthermore, reference bodies of all clusters are listed in the *Reference Bodies* scroll-list while all closure joints are listed in the *Closure Joints* scroll-list. The global reference body of the current kinematic system is displayed at the bottom of the info box. By selecting a name of an element in any scroll-list and clicking the *Info...* button, the info box of the element is opened.

- Edit Globals...

This command is, like the *Chain Info*, available as soon as a *Project window* is active and it opens the *Math editor* which is used to define global constant parameters and functions. The global parameters and functions can be shared by all structural and control elements of a kinematic system. Furthermore, the names of global parameters and functions have to be unique among all existing parameters and functions defined in the current kinematic system. Therefore, a special algorithm checks each new name and returns a message if the name is not correct or unique. In the case that the definition of a global function or constant is altered, definitions of all elements of the kinematic system are checked on eventual run time errors. If an error is encountered in a mathematical expression, the corresponding structural or control element is considered as undefined and a question mark is displayed in its symbol.

- Show Tree

After choosing this command the currently used tree structure is shown by selecting the graphic objects of all structural elements except the base body and the closure joints. This command is enabled only after the topology analysis has been done.

2.6.5 The *Computation* Menu

This menu is used to start and control the computation process (see Fig. 2–29). It contains the following commands:

- Run

By choosing this command the computation procedure is started. The computation procedure consists of the following consecutive steps:

a.) The kinematic chain is checked in order to find out whether its elements have been defined and connected properly. If there is a problem the computation procedure is aborted and a dialog box with a warning is displayed.

b.) The topology of the examined kinematic chain is analyzed in order to find the optimal reference body (reference frame) and closure conditions in the case of loops.

c.) As next the program attempts to find any possible posture of the examined kinematic chain. (The initial values for the first assembly iteration are obtained randomly.). Note that this procedure is repeated automatically as long as no posture is found.

d.) After the initial posture was found the computation algorithm attempts to find further solutions (possible postures) belonging to the current closure mode. Note that after all possible postures of one mode were found the step "c.)" is automatically repeated until a new posture (a new closure mode) is found.
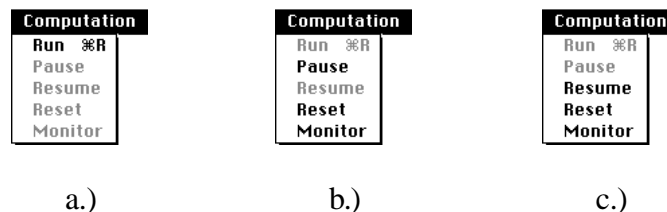
- *Pause*

This command is available only when the computation process is running and it is used to stop the computation temporarily.

- *Resume*

This command continues the computation process if it was interrupted by the *Pause* command.

- *Reset*

By choosing this command, the computation process is terminated and the entire data which has been computed is erased.

a.)   b.)   c.)

Fig 2–29: States of the *Computation* Menu: a.) before starting the computation, b.) the computation is running and c.) the computation is temporarily stopped.

- Monitor

By selecting *Monitor* command, the *Computation Monitor* floating window is opened displaying important current data such as the number of assembly attempts and the number of all currently known curve segments. Moreover, with the help of the *Emphasized Curve* Pop-Up menu we can select the curve segment which should be emphasized. If zero is chosen in this Pop-Up menu all curves are drawn in the normal drawing mode.
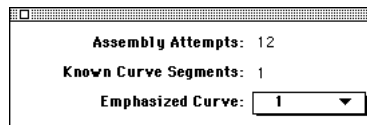

Fig 2–30: *Computation monitor* window.

- Comp. Options...

By selecting *Comp. Options...* command, the *Computation Options* dialog box is opened where the current iteration step and the precision threshold can be specified.
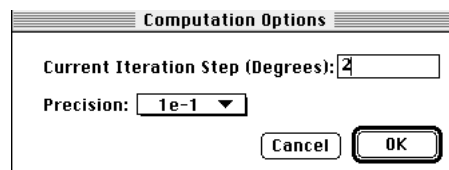

Fig 2–31: *Computation Options* dialog box.

2.7 Exporting Data

The program **Joint** supports also an interface which is used to export text files of the computed data. With various elements of the export interface it is possible to prescribe the form of the exported text files. This is very useful in the case that the computed data should be further processed by some other application which requires certain syntax (animation, spread sheet, etc.).

When exporting a text file, we have to specify which data should be written to the file. In order to do this, we have to write the name of the variable which must be separated from the remaining text by special separators. The separators have to be placed immediately before and immediately after the string which will be replaced by the actual value during the text file generation. For example "...$Rotation.1$..." will be replaced by the actual value of the variable named *Rotation.1*. The separator is in this case "$". In this way it is possible to specify any kind of data which should be saved in the text file.

2.7.1 Naming Conventions for the Exported Data

The export interface supports the following name conventions for various variables as well as point coordinates and direction vectors:

- Rotation variables (R-joint, C-joint, and H-joint):
The name is specified by "Rotation.n ". The "n" is the object number of the corresponding joint. For example, "Rotation.3" denotes the variable belonging to the joint with the object number 3 (shown at the top of the joint symbol in the *Project window*).

- Translation variables (P-joint, C-joint):
The name is specified by "Translation.n". ("n" like in the case of rotation.)

- Variables of a global joint (G-joint):
The names of the components of a rotation vector are specified by "G-variable1.n", "G-variable2.n" and "G-variable3.n" respectively. The "n" is the object number of the corresponding joint.

- Coordinates of a Point Vector:
For example the names of the point coordinates, belonging to the point named "P9", are specified by "P9x.n", "P9y.n" and "P9z.n" respectively. The "n" is the object number of the corresponding joint. NOTE! The coordinate notation x, y and z has to be lower case (Not "P9X.n").

If only the name of a point is given, for example "P9.n", the point will be interpreted as a vector and its coordinates in the current reference coordinate system will be written to the text file as a list of values. The characters which will in the text file define the beginning, the end and the separators of the vector coordinates can be defined by the user according to the desired vector notation (See subsection 2.7.2.).

- Coordinates of a Direction vector:
For example, the names of the vector coordinates, belonging to the vector named "D9", are specified by "D9x.n", "D9y.n" and "D9z.n" respectively. Other conventions are the same as in the case of a point vector.

2.7.2 The *Export* Interface

The export interface consists of the *Export* menu and various dialog boxes where the file format can be specified.
The *Export* menu contains the *Export Options...*, *Data Editor...* and the *Export...* commands.

Export
Export Options...
Export Editor...
Export Data...

Fig 2–32: *Export* menu.

- Export Options...

By choosing *Export Options...* the *Options* dialog box is opened.

Options

Substitution Syntax:

Substitution Marker: [$]   Vector Notation: [<][x][,][y] ... [>]

Transformation Matrix:

e1x: [1]   e2x: [0]   e3x: [0]   tx: [0]
e1y: [0]   e2y: [0]   e3y: [1]   ty: [0]
e1z: [0]   e2z: [1]   e3z: [0]   tz: [0]
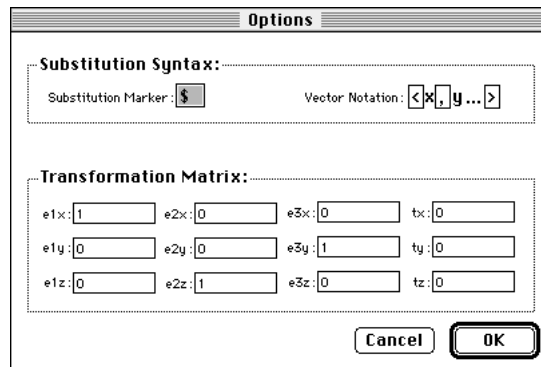
[Cancel]   [OK]

Fig 2–33: *Options* dialog box.

In this dialog box the following options can be set:

- The *Substitution Marker* (should be some special character).

- The *Vector Notation.* Each vector is in the export text specified by special characters at the beginning and at the end of a list of its coordinates. The user can specify also the separators between the coordinates of a vector. For example, the vector $\mathbf{a} = (2.5, 3, -1.2)$ would appear in the text file as "<2.5, 3, -1.2>". In this example the characters "<", ">" were chosen to mark the beginning and the end of this vector while "," represents a separator.

- The *Transformation Matrix* is used to transform the reference coordinate system which was used for the computation. This does not affect the computation and only the exported point coordinates and direction vectors are transformed.
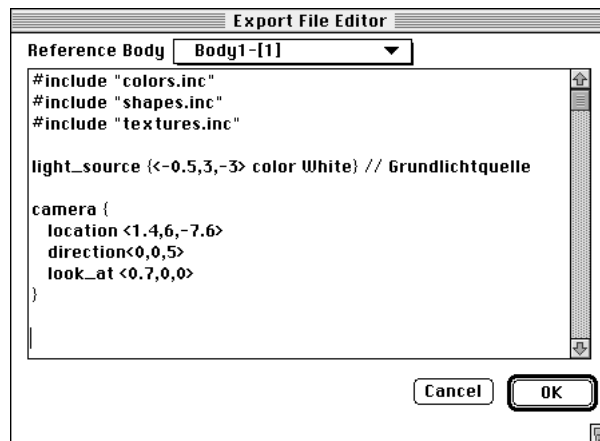

- Export Editor...

By choosing *Export Editor...* the *Export FileEditor* dialog box is opened.
- The *Reference Body* pop-up menu is used to choose the body in which the global reference frame for the exported data is fixed. The exported data (i.e. points, direction vectors) are always transformed relative to this coordinate system.

- The scroll-text field is used to define the header of a file-record of the export text file. A file-record corresponding to one point (one position of a mechanism) is defined as an arbitrary text including data computed by using this point only. The user can write an arbitrary text and specify which values of variables and where they will be inserted in the file-record. This is achieved by using the naming conventions discussed earlier (See subsection 2.7.1.).

Fig 2–34: *Export Editor* dialog box.

Moreover, any local export data specified at the bodies (See subsection 2.3.3) will be automatically attached to the end of the text defined in this scroll-text field. When working with a local *Export File Editor* belonging to one body, the object number following the point/direction vector name is not necessary if this vector belongs to the same body. A correct full name is in this case created automatically when the local text is attached to the main text of the data record.

- Export Data...

By choosing *Export Data...* the *Data Export* dialog box is opened (see Fig. 2–35). It consists of the following elements:
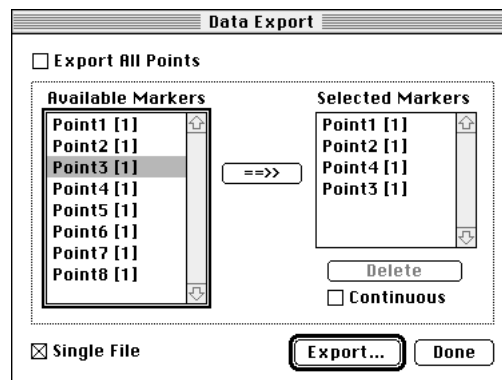
-*Export All Points* check-box. If this check-box is checked, for all points the corresponding file-records will be written to the text file in the same order as they were computed.

-The section where the segments or certain points of the solution paths can be chosen. This section is used in conjunction with the markers set in the diagrams (See subsection 2.5.1). In the scroll-list *Available Markers* one or more desired markers can be selected and transported to the *Selected Markers* scroll-list by clicking the button with the arrow (placed between these two scroll-lists). The points can be removed from the *Selected Markers* scroll-list by clicking the *Delete* button.

If the *Continuous* check-box is checked, a new file-record is added to the text file for each point of the entire segment between the points which are specified by the first and the last marker in *Selected Markers* scroll-list. If the last marker in the *Selected Markers* scroll-list equals the first marker in the same list, for each point of the whole input-output curve a new file-record will be added to the file. If the *Continuous* check-box is not checked new file-records are created only for the points specified by the markers in the *Selected Markers* scroll-list.

Fig 2–35: *DataExport* dialog box.

- *Single File* check-box. If this check-box is checked, all data records corresponding to the specified points (selected markers) will be written to one text file. Otherwise for each point a new file is generated. This check box would be normally checked if a table of desired values of variables corresponding to different points (selected markers) should be generated.

- The *Export...* button. By clicking this button, the *Save* dialog box is opened where the name of a new file (files) is specified and a new folder can be created.

In the case that more than just one marker (point) has been selected and the *Single File* check-box is not checked, we will get a file for each point. The name of each file will be composed of the name specified in the *Save* dialog box and the current point number, both separated by an underscore.