

13 CAT

13.1. Introduction to the CAT-menu

The CAT (Computer Aided Teleoperation) menu covers

- the **Monitor-** and **Master-Mode** of mechanisms,
- the **Simulation** and **Control** of real and simulated **cameras** and
- the **Framegrabber control**.

13.2. The CAT Panel



Figure 35: The CAT-Panel

13.2.1 The Monitor/Master-Mode Buttons

- **Activate Mechanism:**

Selection of the active mechanism which can be a robot or any other kinematic structure. The selected mechanism is displayed in the *active-field*.

- **Monitor Mode ON/OFF:**

Toggle function for activation/deactivation of the joint-position sensor interface between KISMET and the real robot controller for the active robot.

This function switches to the **monitoring-mode**. In monitoring mode, KISMET is using the joint-position sensor data as provided by the real robot controller for synthetic image generation. The workcell display will show the real robots position in the simulated workcell environment. In this mode, the KISMET display provides the visual interface for man-in-the-loop control of telemanipulation operations, or for surveillance of of automatic (pre-programmed) task sequences. The default state is "OFF".

- **Master Mode ON/OFF**

In this functional mode, KISMET can be used as a **master device** (position controller, open loop) for a real robot controller (graphical master concept). When active, a set of positional

data is sent to the controller of the active robot via UNIX-message communications, after each rendering cycle . An interface program (user generated) is necessary to pick up the position data from the message queue and to perform the data-distribution to the robot control-hardware. The default state is "OFF".

13.2.2 Camera Simulation & Control

This subpanel is used for control of simulated CAMERA-views and for ONLINE-Control of "real" workcell cameras during teleoperation.

- **Select**

This function is used to select the active CAMERA from the CAMERA-menu, which is automatically generated during model startup. It includes the logical names of all CAMERAs in the application model-database. The active CAMERA has the input focus for all other commands in the "Camera Simulation & Control" submenu.

- **Pointing**

Can be used to specify a new 3D-lookpoint (in "top + side view" display mode) for the "active camera". A single CAMERA motion is carried out.

- **View PYRAMID ON/OFF**

Enable/disable the overlay of the CAMERA viewing pyramid. The tip of the pyramid is located at the CAMERA view-location. The base of the pyramid shows the current focal plane of the CAMERA. The default state is "OFF".

- **Tracking ON/OFF**

Enable/disable CAMERA-Tracking mode for the active CAMERA (use the "Select CAMERA"-function to activate a camera). KISMET allows automatic tracking for simulated cameras. The CAMERA must be part of a ROBOT to let this functionality work. It is, not the CAMERA is moved, but the ROBOT, i.e. the CAMERA motion mechanism. FRAMES are used as a tracking target. The FRAME to be tracked is PICKed after selection of this function. Additionally, KISMET calculates the distance (Dist) to the tracking target. The distance value can be used to implement a model-based autofocus feature (see the command "**Send DATA**"). The default state is "OFF".

This option can be enabled simultaneously for all CAMERAs in the simulation model. This is, each camera can be put in TRACKING-mode independant from the other cameras.

- **Send DATA ON/OFF**

Enable/disable the CAMERA command interface to the camera controller. For monitoring applications, KISMET can be used to provide high-level kinematic control for the camera actuators in the real workcell.

A special UNIX-process (communications server) must exist on the workstation to handle the communications between KISMET and the real controller(s). The default state is "OFF".

- **Dist+Fovy**

This valuator function is used for the CAMERA-overlay view to change the two parameters DIST (focal distance of the real camera) and FOVY (field-of-view angle). When the CAMERA is in ONLINE-mode, the current parameters are sent to the camera controller.

- **LM** Change DIST attribute
- **MM** Change FOVY attribute
- **BM** No function

- **MAIN view**

The main simulation display (large viewport) is connected with the current position of the active camera. This function is particularly useful for the generation of animation sequences (here the camera is defined as a robot and can be controlled by a robot program) or for simulation of camera views in training or monitoring applications. The 'byrdseye view' parameter dataset will be modified automatically when this command is active.

13.2.3 Framegrabber Control

This subpanel is used for video-framegrabber control. This mode requires the realtime framegrabber (Live Video Digitizer, Indy Camera, Galileo Board etc.) option in your graphics hardware and the videolab software subsystem installed on your workstation. The submenu allows optionally the display of a video window or an overlay of a KISMET wireframe-model display on top of the scanned video camera image. This latter functionality can be useful for model verification against the real workcell and to some extent geometry editing.

- **Video Overlay ON/OFF**

Enable/disable the realtime video-display. A realtime video channel is opened to the current system video device (as selected in the SGI system program 'videopanel'). The video image is rendered on top of the synthetic workcell view. The video input signal is rendered once per frame cycle, i.e. the framerate of the video overlay is the same as the workcell rendering framerate. The default state of this function is "OFF".

- **GRAPHICS Overlay**

Note: You have to define a CAMERA to use this command. The CAMERA-overlay viewport is used to display the FRAMEGRABBER images with the simulated CAMERA wireframe overlay. The active CAMERA view is used for the wireframe model overlay.

Toggles the overlay of the wireframe model drawing on top of the video image. The video-image is displayed in the CAMERA-overlay viewport. The simulated, virtual CAMERA view is drawn in wireframe mode as an overlay on the scanned video-image. This is, the function is possible only if at least one CAMERA is defined in the workcell model. The simulated wireframe and the real image correspond, if the view parameters of the simulated CAMERA are similar to the view attributes of the real camera. These values have to be set once for the simulated CAMERA, using the CAMERA submenu (to set the FOVY parameter) and the motion commands (to adjust the CAMERAs pitch/roll/yaw parameters). If the virtual CAMERA is not actuated, i.e. the CAMERA itself is not actuated or its actuators are not defined as a ROBOT in the KISMET-model, you can use the "Edit FRAME" commands to adjust the viewing position and -direction of its reference FRAME.

- **RESET**

Resets the FRAMEGRABBER hardware to its default parameters.

- **Param. SET**

This command displays a form-window, which allows to set all hardware parameters of the

video framegrabber during runtime. See the **videopanel(1)** manual page of your SGI workstation and the documentation of the Live Video Digitizer about further details about these parameters.

14 DETAIL

14.1. Introduction to the KISMET DETAIL-LEVEL Concept

This chapter describes the DETAIL submenu, visible for the user while executing KISMET in the non-full-screen-mode. In full-screen-mode a different user interface using pop-up menus will appear. The corresponding menu-names are surrounded by {}.

In KISMET a hierarchical, recursive, tree like data model is used. The operator can specify interactively and on-line the level of detail to be displayed. Components or model parts can be activated or deactivated for display at any time. This allows to render higher levels of detail without loss of realtime performance. There is no logical limitation on the number of detail-levels or model size.

Figure 36 shows the rendering of a flange assembly (from the CATROB model) in two different detail levels.

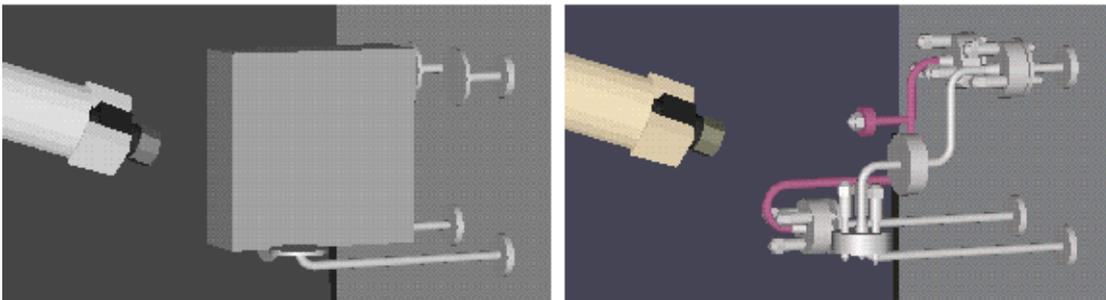


Figure 36: Workcell display using DETAIL-LEVELS

A bounding box which is used to mark a non-accessible area is rendered in the left picture, whereas the right image shows the detailed rendering of the flange assembly after switching the detail level.

To achieve the detail-leveling functionality in KISMET, we use a hierarchical, tree-like structure of ABSTRACT files. Each ABSTRACT file defines an arbitrary number of FRAME records. Now, every FRAME can reference another ABSTRACT-file, called the "son"-file. This is, you can define any numbers of "sons" from one '.mpc' file. Figure 37 shows the tree-like structure of ABSTRACT files ('.mpc'), as used in KISMET.

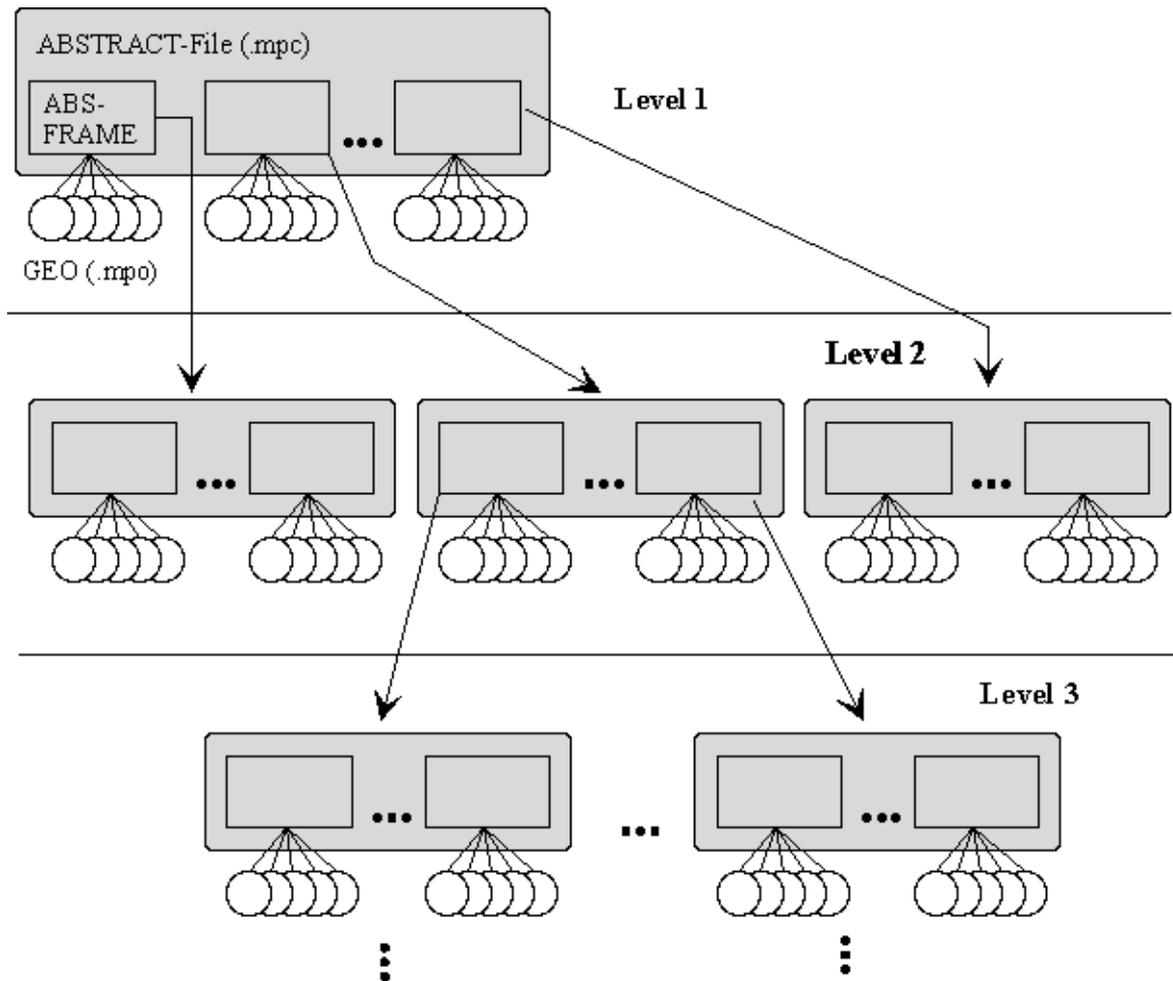


Figure 37: Hierarchical Structure of ABSTARCT-Files

14.2. DETAIL Submenu {Edit ABSTRACT}

The DETAIL submenu is used

- to load, activate and deactivate model-parts and details and
- to create an new (robot) model.

In KISMET, an ABSTRACT is a kind of special FRAME, which acts as a reference for an assembly, or to another, more detailed representation of the part. The physical representation of the ABSTRACT is a file (‘.mpc-file’) which defines a part with its FRAMES and file-references to the GEO- element data (‘.mpo-files’). An ABSTRACT-file is also used to define exactly one

ROBOT.



Figure 38: The Detail Submenu

As a special feature in KISMET, each frame defined in an ABSTRACT file can store the file reference to another ABSTRACT file. This feature is used to define multiple levels of detail in KISMET. It is possible to control the simulation performance (frame rate) over a wide range in KISMET, using this feature. To get the optimum performance, you should only activate those parts and details of the application model which are necessary for operation in the current stage of the simulation. The following section explains the different commands in the „DETAIL“ Submenu.

- **Pick Frame {Pick ABS_Node}**

This command is used to start a PICK operation to identify the ABSTRACT node for successive operations. An ABS-node is picked at the origin of its associated coordinate system as drawn in the workcell display. When an ABSTRACT is picked, all GEO-parts connected to this ABSTRACT are highlighted

- **Pick GEO{Pick ABS_NODE via GEO}**

This is another command to PICK an ABSTRACT. It is computationally more demanding as the previous command. Here you can pick an ABSTRACT by picking any GEO-part connected to this ABSTRACT, i.e. to all FRAMES in the ABSTRACT file.

- **Activate{Activate ABS-Branch}**

A previously deactivated ABSTRACT is activated for display. All GEO-parts connected to the reference FRAME of the ABSTRACT remain active for display (see 'Swap Level' command).

- **Deactivate{Deactivate ABS-Branch}**

An active ABSTRACT branch is deactivated for display. This command is useful to

increase drawing speed (frame rate) drastically in the simulation. You should deactivate all model parts which are currently not essential for the operation. The ABSTRACT branch is not removed from the runtime database (main memory). Therefore it can be reactivated immediately for display by an 'Activate' operation.

- **Load{Load ABS-Branch}**

Another ABSTRACT branch is loaded from the model database and activated for display. The memory required for model storage is allocated from the operating system. The function is used to load a model part or detail level which is marked in the database ('.mpc-file') with the 'W' attribute (wait), or to reload parts which were deleted previously in the same simulation session.

- **Swap LEVEL{Swap DETAIL Level}**

This command loads/activates the next detail level (more detailed) for the picked ABS-Branch. At the same time, all GEO-parts connected to the reference frame of the ABSTRACT are deactivated for display. This is, the current GEOMETRY of the ABSTRACT-frame is replaced by a more detailed alternative geometry, which is defined in the loaded ABSTRACT-file ('.mpc file').

- **More GLOBAL{More Details}**

This command is used to increase the detail level for the whole application model by one. The command works similar like a set of 'Activate' or 'Load'/'Swap LEVEL' commands. Only, it works with the whole model. It is mainly for „lazy“ operators. Using this function, there is no fine control concerning which parts to load/activate/swap or not. You may find for some models, that the performance is reduced drastically after using this command. This can happen when too many parts (more than expected) are activated.

- **Save as{Write ABS_FILE}**

This function is used to save changes in the model to the file system. After picking an ABSTRACT (and the corresponding '.mpc file') using the 'Pick' Frame' or 'Pick GEO' command, the identified ABSTRACT is written to the database. Since an 'mpc file' can have file references to other mpc-files and one '.dof'-file, the user has different options:

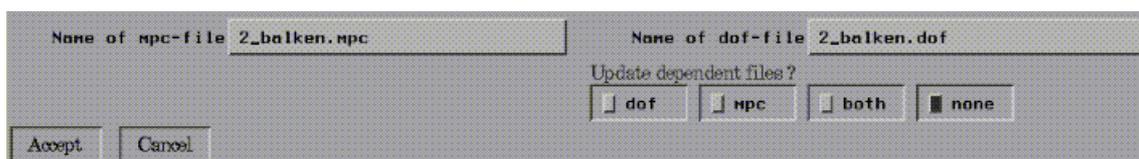


Figure 39: The Save-as Panel

- **Update dof-file:**

The referenced dof file is checked and updated if necessary. KISMET checks, whether each moveable frame in the picked mpc-file is referenced in the dof-file or not. If a moveable frame is not referenced, the dof-file will be updated using the following default values (for frame No. i):

```
<menu_text> ::= „name_of_frame“_DOF
<out_text> ::= Z_i
<n_entries> ::= 1
<type> ::= 0
```

```
n_const ::= 1
<const> ::= 0.1
```

(see User Data Specification).

- **Update mpc-file**
The referenced mpc-file will be updated.
- **Update both**
The referenced 'dof-' and 'mpc'- file will be updated.
- **Update none**
Only the picked mpc-file will be updated, none of the referenced mpc- or dof-file will be updated.
- **Save ROOT{Write ABS-Root File}**
The ABSTRACT-root, i.e. the uppermost '.mpc' file in the application model, is written to the file system.
- **Unload{Delete ABS-Branch}**
Deletes an ABSTRACT-branch from the runtime database (main memory): The reference in the current model is not changed or deleted. This is, the abstract can be reloaded later in the simulation session. The function is useful to make space in the main model for other parts, for example to avoid memory swapping. It is also used to increase drawing performance.
- **Delete REFERENCE{Delete ABS-Reference}**
Removes a REFERENCE to an ABSTRACT-file from the specified (picked) node. From this moment, the ABSTRACT file (detail or assembly) cannot be loaded or activated in the current simulation session. The abstract file in the database is not affected by this operation, because it may be referred to in other application models or parts of the current model (instancing).
- **Print NODENAME{Print Nodename}**
the fully qualified nodename of the ABSTRACT is printed in the KISMET-textport. The nodename is a list of all FRAME-names from the model-root (ABS-ROOT) to the selected ABSTRACT. The abstract names are separated by a '.' (dot). The full qualified nodename is used in SCRIPT-files to uniquely identify a FRAME or ABSTRACT node. This naming convention is necessary, because KISMET allows instancing of GEO-parts, FRAMEs and ABSTRACT-files.
- **Create Robot**
A new model is created: Starting with the '.sim-File' and leading the user through the complete data structure a new model will be created. This command can be used for example to give an existing modell a new name after changes have been made.If no 'dof-'file is available in the current model, a dof-file will be generated automatically.

Usage:

Create robot opens the **Enter sim file** name panel:

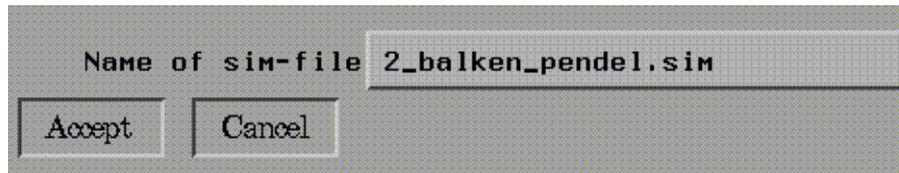


Figure 40: The Enter-Sim-file Panel

The next step will lead you to the **Enter mpc-base** name Panel:

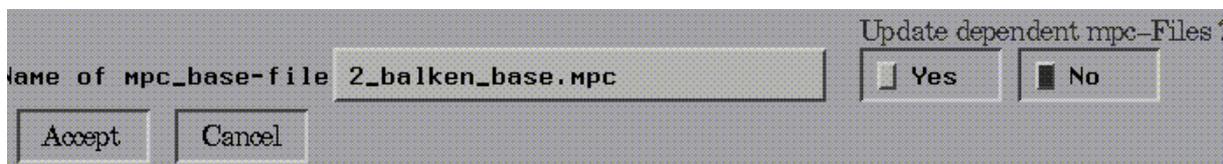


Figure 41: The Enter base_mpc file name panel

Finally the menu gives you the possibility to update and save all mpc-files of the current model. The Save as Panel will appear (see Figure 39: The Save-as Panel).

15 MODEL

15.1. Introduction to the MODEL-submenu

This menu contains submenus responsible for

- editing and changing of geometric parts (Edit GEO)
- creating and moving of frames (Edit FRAME)
- editing colour tables (Edit COLOUR)
- editing tables defining surface properties (Edit MATERIAL)



Figure 42: The Cell Modeling Panel

15.2. Edit GEO

The "Edit GEO" submenu is used for local creation, positioning and editing of geometry parts (GEO-elements). Together with the "Edit FRAME" menu it provides the basic interface for model creation, or local model correction during on-line remote handling sessions. The basic concept behind KISMET is to import geometry parts from CAD-systems. Although, in some situation the build-in geometry editor can be useful. Geometry placement operations (translate, rotate) are performed with a local copy of the GEO-element. This is, you have to CONFIRM the modification of the PLACEMENT when you are done with a GEO-part. The current GEO-object is highlighted after identification (PICK).



Figure 43: Edit GEO

- **Pick**

This command is used to start a PICK operation to identify the GEO-part to be edited. You

may PICK the part at its surface area. However, picking is faster if you use the picking-REGION, i.e. the little white square which is drawn approximately in the centre of the GEO-part.

- **CONFIRM**

This command is used to confirm any changes during GEO-placement editing ("Rotate" and "Translate"). These operations are carried out with local copies of the operand.

- **Rotate**

The GEO-part is rotated in 3 dimensions relative to its reference FRAME. A general 6 parameter transformation $(dx, dy, dz, \alpha, \beta, \gamma)$ is used by KISMET for the transformation.

- **LM Change α_G , i.e. rotate GEO around X-axis**
- **MM Change β_G , i.e. rotate GEO around Y'-axis**
- **BM Change γ_G , i.e. rotate GEO around Z"-axis**

- **Translate**

The GEO-part is translated relative to its reference FRAME in 3 dimensions.

- **LM Translate GEO about d_x along FRAME-X-axis**
- **MM Translate GEO about d_y along FRAME-Y-axis**
- **BM Translate GEO about d_z along FRAME-Z-axis**

- **SCALE xyz**

Scales the GEO-part along the

- **X-Axis, if LM is used,**
- **Y-Axis, if MM is used**
- **Z-Axis, if BM is used.**

- **SCALE all**

Scales the GEO-part along the X-, Y- and Z-axis.

- **Morphing**

Activates the interactive morphing function, i.e. a linear interpolation of a geometric part into a similar part with different shape is performed.

- **Edit Flex**

Leads to the Edit Flex subpanel. See chapter 15.2.1 Edit Flex.

- **READ**

This command inserts a GEO-element into the model. KISMET requests you to pick the reference FRAME for the new part. The new element is connected to this FRAME. To select a certain GEO the **File selection panel** will pop up to offer you a selection of 'mpo-files'.

- **DUPLICATE**

The function is used to copy the previously picked GEO-element. KISMET requests you to pick the reference FRAME for the copied part. The instanced element is connected to this FRAME. An identical copy (instance) of the part is used.

- **CREATE**

This command is used to create (model) a new 3D GEO-part, which is added to the model. The selection of the GEO-primitive is carried out in an upcoming submenu. The parameters are entered in a form window. A special form is available for each primitive. The dimension for modelling is ,mm'. The new GEO-part is connected to a reference FRAME. KISMET will ask you to PICK the FRAME, so make sure, that the FRAME is visible in the current viewport when the CREATE command is used. Each GEO-part is stored as an ASCII-file in the model database in the directory \$kis_home/mpolib. The standard suffix for GEO-files is ,.mpo'.

The following list of basic primitives is available in the "Primitive"-submenu:



Figure 44: The Primitive Submenu

The 2D-geometry editor 2Ded is called for the interactive definition of the sweep contour when the SWEEP-primitives are selected for modelling. The contour is defined from segments, using POLYLINE, ARC, and B-SPLINE 2D-modelling elements. The 3D-surface is formed by a translation (linear sweep), or rotation (rotational sweep), respectively. See chapter 21 The 2Ded-Editor for SWEEP Operations for details.

The newly created parts can be used with the BOOLEAN-operations to form more complicated GEO-parts. The results are saved as POLYHEDRON models.

- **DELETE**

The selected (picked) GEO is deleted from the datatree. The operation is carried out in the runtime model. To delete a part from the file system, use the UNIX command 'rm'.

- **Edit NURB**

Leads to the "Edit Nurb" submenu. See chapter 15.2.2 Edit NURB.

- **DRAWMODE**

Determines the drawmode for the picked GEO-part.



Figure 45: GEO Drawmode

- **Transparent**

The picked GEO-part is drawn in TRANSPARENT-mode. All other parts keep the previous drawing mode. This function sets an attribute in the model datastructure. This attribute is reset when a "Drawmode" command is executed. Note: The transparent drawing mode in KISMET is using the α -bitplanes. On workstations with limited graphics hardware, using this command can give unpredictable results.
- **Wireframe**

The picked GEO-part is drawn in WIREFRAME-mode. All other parts keep the previous drawing mode. This function sets an attribute in the model datastructure. This attribute is reset when a "Drawmode" command is executed.
- **Flat Shaded**

See chapter 9.2. The Display Panel for further information.
- **GOURAUD Shaded**

See chapter 9.2. The Display Panel for further information.
- **INVERT Faces**

The orientation of all polygonal facets of the selected GEO-part is reversed.
- **Interpolate Normals**

Interpolates smooth normal vectors on the polygon edges of the selected GEO-part. KISMET will ask for the maximum angle between the adjacent facets to be used for smoothing.
- **CSG Operation**

This command is used for interactive execution of Boolean Modelling Operations. After the command is selected, you have to PICK the two parts (OPERANDS) for the following boolean operation. The type of boolean operation (Union, Difference, Intersection) will be requested by KISMET. The operands will not be effected by the calculation. The calculation is carried out with local copies of the operand parts.

The computation is carried out on the basis of the KISMET-internal, faceted polyhedron-model. The operation is not restricted to a subset of the available GEO-primitives. For successful computation the operands should have a closed surface shell (solid-parts). The operands should intersect eachother to achieve satisfying results.
- **Write POLYHEDRON**

The picked GEO-part is saved in the file `,$kis_home/mpolib/<filename>` as a POLYHE-

DRON model. The parameter <filename> is entered by the user. Note: The default extension of GEO-partfiles is ,mpo'. This extension is used by KISMET in the file selection panel.

- **Write EXT_POLYHEDRON**

The picked GEO-part is saved in the file , $\$kis_home/mpolib/<filename>$ ' as an EXTENDED_POLYHEDRON model. The difference to an ordinary polyhedron is, that for each surface vertex a surface normal is stored in the model. This form of storage should be used for parts with smoothly curved surfaces, to allow for GOURAUD-shading when you recall the model. The parameter <filename> is entered by the user. Note: The default extension of GEO-partfiles is ,mpo'. This extension is used by KISMET in the file selection panel.

- **Save all GEOs**

Saves every GEO in the current model.

15.2.1 Edit Flex

This submenu is used to manipulate geometric primitives of the id '90' (KISMET Geometry-Specification: ELADYN_NURBS) as well as id '91' (KISMET Geometry-Specification: ELADYN_POLY). The elastodynamic behaviour can be controlled and influenced, parameters can be adjusted. Some items in this submenu can also be used for manipulating other flexible parts (sub-id '-10' in the KISMET Geometry-Specification).

The 'Edit FLEX' - Submenu is located under the main menu 'MODELS' - 'Edit GEO'. For some operations, the specific geo part must be identified first ('PICK Geo', marked with (P_GEO) in following description). The submenu consists of 11 buttons:



Figure 46: Edit FLEX

- **Pick CP (P_GEO)**

A specific mass knot of the ElaDyn-Object can be selected. After the identification of the GEO, all mass knots are highlighted (white tetraeder). The desired mass knot can now be picked with the mouse (left mouse-button). The coordinates of the knot are printed out in the textport. For some operations, a specific knot must be identified first with this feature (marked with (P_CP) in following descriptions).

- **Move CP (P_GEO) (P_CP)**

The selected mass point can be moved by the mouse. A horizontal movement shifts the knot:

- in x-direction, if the left mouse button is pressed
- in y-direction, if the middle mouse button is pressed
- in z-direction, if both mouse buttons are pressed

The coordinates of the knot are printed in the lower left edge of the viewport.

- **Fix CP (P_GEO) (P_CP)**

The selected mass knot is fixed. All bounding conditions are activated, the knot can't move any more until the knot is released.

- **Release CP (P_GEO) (P_CP)**

The selected mass knot is released. All bounding conditions are deactivated, the knot can move in any direction.

- **Freeze**

All elastodynamical calculations are stopped. All flexible parts are 'frozen' until the button is pressed again (Toggle-Button).

- **Reinit Flex (P_GEO)**

The elastodynamical properties of the geo are reinitialized. This feature is useful for interactive geometric changes. First, the geo is frozen, then the knots can be moved to change the geometric shape. Last, the elastodynamic object is reinitialized.

- **Add Force (P_GEO) (P_CP)**

An external force can put on the selected mass knot.

A horizontal movement of the mouse changes the force in a specific direction:

- in x-direction, if the left mouse button is pressed
- in y-direction, if the middle mouse button is pressed
- in z-direction, if both mouse buttons are pressed

The current force-values in each direction (unit: Newton) are printed out in the lower left edge of the viewport.

- **No Forces (P_GEO)**

All external forces of the selected object are set to zero.

- **Calculation**

Opens the calculation-window, where all the parameters of the elastodynamic calculation

can be set and changed:



Figure 47: The Calculation-Panel

- **calc. repeat fact (number, unsigned)**
number of repetitions of the elastodynamical calculation before new drawing
- **time step (number, float)**
Integration time for the elastodynamical calculation (only required if time step definition = Const.)
- **time step def.(Bool, Auto/Const)**
Switch Real Time/Constant Time
- **Motion Calc.(Bool, Real/Slow)**
Switch Real Time/Slowmotion (10*, used to prevent numerical instabilities)
- **Dynamic Calc.(Bool, Real/Quasi)**
Switch complete dynamic calculation/simplified dynamic calculation
- **ElaDyn Type (Bool, I/II)**
Switch elastodynamical options
- **Object move (Bool, On/Off)**
Switch global object move
- **Gravity (Bool, On/Off)**
Switch gravity. The direction of the gravity vector points along the negativ y-axis in world coordinates (9,81 m/s²). Only geo-objects with the Kismet-id 90 are influenced by the gravity.
- **Ground (Bool, On/Off)**
- **Switch ground (y=0 plane)**
- **Draw Poly (Bool, Yes/No)**

- **Switch drawmode: polygonal drawing**
- **Draw NURBS (Bool, Yes/No)**
- **Switch drawmode: NURBS drawing**
- **Parameter (P_GEO)**

Opens the parameter-window, where all the specific parameters of an elastodynamic object can be set and changed (all parameters are floats):

The screenshot shows a parameter panel with the following fields and values:

geo-name	galle_flex1.mpo		
geo-knot mass	0.002000	object mass	0.220000
geo-springs stiff.	1.200000	basic-springs stiff.	0.2000
knot damp	0.200000	object damp	2.1000
object-spring stiff.	1.200000	torque-spring stiff.	0.02000

At the bottom left, there are two buttons: "Accept" and "Cancel".

Figure 48: P_GEO Parameter Panel

- **geo-knot mass**
mass of knots with geometrical link (outer knots), defined in [kg]
- **object mass**
mass of the whole elastodynamic object [kg] (mass of all inner knots)
- **knot damp**
damp factor of the elastodynamical mass knot, defined in [Ns/m]. The damp force is proportional to the damp factor and the velocity of the referred knot.
- **object damp**
damp factor of the elastodynamical system, defined in [Ns/m]. Used for global movements
- **geo-springs stiff**
spring constant of the springs which links the geometrical knots
Two kinds of spring characteristics are implemented:
sconst > 0 : spring force is proportional to the spring length [N/m]
sconst < 0 : spring force is proportional to the quadratic spring length [N/m²]
- **basic-springs stiff**
spring constant of the springs which force the knots to the original positions (return springs), maintain the original shape of object
Two kinds of spring characteristics are implemented:
sconst > 0 : spring force is proportional to the spring length [N/m]
sconst < 0 : spring force is proportional to the quadratic spring length [N/m²]

- **object-spring stiff**
spring constant of the springs with global links (inner knots, connections)
Two kinds of spring characteristics are implemented:
sconst > 0 : spring force is proportional to the spring length [N/m]
sconst < 0 : spring force is proportional to the quadratic spring length [N/m²]
- **torque-spring stiff.**
spring constant of the torque-springs of inner knots, defined in [Nm]
(maintains the global orientation of the knot)

- **Show Parts**

The components of all elastodynamic objects are shown. Mass knots are drawn as coloured tetraeders with their specific number (the colour is dependend from their interaction status), springs are drawn as green lines. Additionally, the sum of the inner forces of each mass knot are shown as red vectors, and the shift of the mass knots relative to there equilibrium positions ist displayed in orange. The direction of each inner knot is visualized as a blue arrow.

- **Save ElaDyn (P_GEO)**

The selected elastodynamic part is saved in a Kismet geo-file (*.mpo'). The name of the file must be specified in the textport.

- **Quit**

The submenu is been closed.

15.2.2 Edit NURB

This submenu is used to manipulate geometric primitives of the id '80' (KISMET Geometry-Specification: NURBS) as well as id '90' (KISMET Geometry-Specification: ELADYN_NURBS). NURBS (Non-Uniform Rational B-Spline Surface) are free-form surface represented by a matrix of characteristic points, the controlpoints. The shape of the geometry can be influenced by shifting the positions of the controlpoints. The 'Edit NURBS' - Submenu is located under the main menu 'MODELS' - 'Edit GEO'.



Figure 49: Edit NURB

For some operations, the specific geo part must be identified first ('PICK Geo', marked with (P_GEO) in following description), other operations need a specified knot ('Pick CP', marked with (P_CP)), too.

The submenu consists of 7 buttons:

- **Pick CP (P_GEO)**

A specific controlpoint of the NURBS-Object can be selected. After the identification of the GEO, all controlpoints are highlighted (white tetraeder). The desired controlpoint can now be picked with the mouse (left mouse-button). The coordinates of the point are printed out in the textport.

- **Move CP (P_GEO) (P_CP)**

The selected controlpoint can be moved by the mouse.
A horizontal movement shifts the controlpoint:

- in x-direction, if the left mouse button is pressed
- in y-direction, if the middle mouse button is pressed
- in z-direction, if both mouse buttons are pressed

The coordinates of the point are printed in the lower left edge of the viewport.

- **Move ROW (P_GEO) (P_CP)**

The whole row of the selected controlpoint can be moved by the mouse.
A horizontal movement shifts the row:

- in x-direction, if the left mouse button is pressed
- in y-direction, if the middle mouse button is pressed
- in z-direction, if both mouse buttons are pressed .

- **Move COL (P_GEO) (P_CP)**

The whole column of the selected controlpoint can be moved by the mouse.
A horizontal movement shifts the column:

- in x-direction, if the left mouse button is pressed
- in y-direction, if the middle mouse button is pressed
- in z-direction, if both mouse buttons are pressed

- **Save ElaDyn (P_GEO)**

The selected NURBS object is saved in a Kismet geo-file ('*.mpo', KISMET-ID '80'). The name of the file must be specified in the textport.

- **Drawmode (P_GEO)**

The drawmode of the selected NURBS can be defined (Filled, Outline Poly, Outline Patch, Isolines)

- **Done**

The submenu is been closed.

15.3. Edit FRAME

15.3.1 Introduction to "Edit Frame"

The kinematic structure in this part of KISMET is regarded as being composed of rigid links connected by joints with contact along a surface, a curve or in a point. Robot kinematics deals with the geometry of robot link motion with respect to a fixed-reference coordinate system as a

function of time without regard to the forces and moments that cause this motion. It deals with the spatial configuration of mechanisms, in particular with the relations between the joint variables and the position and orientation of a robot end-effector. The kinematics problem consists of the direct and the inverse kinematics subproblems.

Further parts of this document extend this basic schema with an extended model, to allow for simulation of

- elastostatic effects, this is torsion and bending of flexible links
- robot arm dynamics
- control algorithms

For the purpose of clarification of the following sections, a list of definitions is given in this section. The definitions are given in alphabetical, not in logical or topological order.

ABSTRACT	represents all data associated with a FRAME. This is, it defines also the topological relations used for the purpose of detail levels. It includes the definition of a FRAME itself, the references to the SHAPE (GEO-element) data together with the relative PLACEMENT of the SHAPE elements with respect to the ABSTRACT-FRAME.
AXIS_PLACEMENT	defines a fixed relative position and orientation of one coordinate system with respect to another coordinate system. For the purpose of term simplification, this term is mostly called PLACEMENT.
BASE	represents a link, which is fixed to the WORLD system or whose motion is prescribed by the motion of another ROBOT.
DOF	defines the relative motion between two FRAMEs of a JOINT by locking some spatial degrees of freedom (DOF) and establishing parameters for the unconstrained degrees of freedom.
FRAME	is a specific kind of coordinate system attached to a kinematic such as WORLD, BASE, JOINT or LINK. Different ways of specifying the placement of a FRAME relative to another are possible.
JOINT	connects two LINKS and constraints the relative motion of one LINK to another via a DOF. A JOINT is kinematically defined by a FRAME which defines the axis of motion, and another moving FRAME which is connected to the LINK.
LINK	rigid body whose motion is constrained by one or more JOINTs.
WORLD	defines the reference coordinate FRAME of the workcell simulation. All relative motions of FRAMEs can be expressed by a chain of PLACEMENT transformations in WORLD coordinates.

15.3.2 The "Edit Frame" Panel

The "Edit FRAME" submenu is used for local creation, positioning, and editing of sub-assembly references (FRAMEs) and actuated JOINTS. It provides the basic functions for kinematic modelling.

In KISMET, a FRAME can be a robot BASE, JOINT or LINK reference coordinate system, or simply a reference system for part assemblies. The FRAMEs are grouped in a hierarchical model tree, which builds the kinematical skeleton of the workcell. All other objects and primitives, such as GEO-elements (shape geometry), robot TCP and ZP, CAMERAs, sensors, workframes (WFRM: the equivalent to the robot TCP on passive workcell parts) etc., are kinematically connected to the FRAMEs. In the first place, a FRAME is the property of a 3D coordinate system. Additionally, each FRAME has a name (identifier), a reference to other FRAMEs for the definition of kinematics and assembly-level topology, and other parameters, depending on the nature of the FRAME.



Figure 50: The Edit Parts (FRAME) Panel

- **Pick FRAME**

This command is used to start a PICK operation to identify the FRAME to be edited. A FRAME is picked at the origin of its coordinate system as drawn in the workcell display. When a FRAME is picked, this is activated for editing, all GEO-parts connected to this FRAME are highlighted.

- **Pick GEO**

This is another command to PICK a FRAME. It is computationally more demanding as the previous command. Here you can pick a FRAME by picking any GEO-part connected to this FRAME.

- **Rotate**

The FRAME is rotated relative to its kinematic predecessor FRAME. For modelling FRAMES (type ≥ 50) with a general 6 parameter transformation $(dx, dy, dz, \alpha, \beta, \gamma)$, you may rotate the FRAME in 3 dimensions, whereas for JOINT-FRAMES (type ≤ 3) the Denavit-Hartenberg (DH) notation $(a_i, \alpha_i, \theta_i, d_i)$ is used. This is, you can rotate the JOINT-FRAME in 2 dimensions (a_i, θ_i) only.

For modelling-FRAMES (type ≥ 50):

- **LM** Change a_i , i.e. rotate FRAME around X-axis

- **MM** Change b_i , i.e. rotate FRAME around Y'-axis
- **BM** Change g_i , i.e. rotate FRAME around Z'-axis

For JOINT-FRAMES (type <= 3):

- **LM** Change θ_i . This is the joint angle from the x_{i-1} axis to the x_i axis about the z_{i-1} axis.
- **MM** Change α_i . This is the offset angle from the z_{i-1} axis to the z_i axis about the x_i axis.
- **BM** No function

- **Translate FRAME**

The FRAME is translated relative to its kinematic predecessor FRAME. For modelling FRAMES (type >= 50) with a general 6 parameter transformation ($dx, dy, dz, \alpha, \beta, \gamma$), you may translate the FRAME in 3 dimensions, whereas for JOINT-FRAMES (type <= 3) the Denavit-Hartenberg (DH) notation (a_i, α_i, q_i, d_i) is used. This is, you may shift the JOINT-FRAME in 2 dimensions only.

For modelling-FRAMES (type >=50):

- **LM** Change d_x , i.e. translate FRAME along X-axis
- **MM** Change d_y , i.e. translate FRAME along Y-axis
- **BM** Change d_z , i.e. translate FRAME along Z-axis

For JOINT-FRAMES (type <= 3):

- **LM** Change d_i . This is the distance from the origin of the "S"_i coordinate frame to the intersection of the z_{i-1} axis with the x_i axis along the z_{i-1} axis.
- **MM** Change a_i . This is the shortest distance between the z_{i-1} system (i.e. the rotation axis) and z_i axis.
- **BM** No function

- **Show FRAMES**

This function is used to activate the display of all frames in the current workcell. The FRAMES are displayed as a coordinate frame and with the logical name.

- **Drawmode:**

Determines the drawmode of the GEO-parts belonging to the picked FRAME.

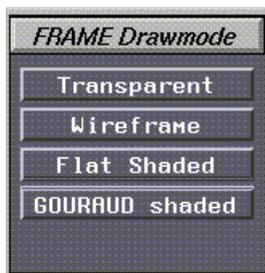


Figure 51: FRAME Drawmode

- **Transparent**

All GEO-parts of the picked FRAME are rendered in TRANSPARENT-mode. All

other parts keep the previous drawing mode. This function sets an attribute in the model datastructure of the GEO-parts. The attribute is reset when the next global "Drawmode" command is executed. Note: The transparent drawing mode in KISMET is using the α -bitplanes. On workstations with limited graphics hardware, using this command can give unpredictable results.

- **Wireframe**

All GEO-parts of the picked FRAME are rendered in WIREFRAME-mode. All other parts keep the previous drawing mode. This function sets an attribute in the model datastructure of the GEO-parts. This attribute is reset when the next global "Drawmode" command is executed.

- **FLAT Shaded**

All GEO-parts of the picked FRAME are rendered in FLAT- shaded mode.

- **GOURAUD Shaded**

All GEO-parts of the picked FRAME are rendered in GOURAUD- shaded mode.

- **Create FRAME**

A new FRAME is added to the workcell. All necessary parameters are requested by KISMET in an interactive dialogue. You have to PICK two FRAMEs after the command is executed. The first FRAME defines the ABSTRACT-file, i.e. the branch in the model tree where the new tree will be stored when you save the workcell. The second FRAME to be picked defines the kinematic predecessor. The new FRAME will move together with its predecessor.

For all FRAMEs (JOINTS and modelling), you have to specify a name. This is a textstring, which is used for identification by some textual commands (SCRIPT-commands). For JOINTs (FRAME types: 0,1,2,3), you have to enter additionally the basic kinematic parameters min-, max joint angle (motion limits), max. velocity and max. acceleration.

- **Connect**

The kinematic predecessor of a FRAME is set to a new frame. From this moment the reconnected FRAME and all its kinematic successors are kinematically connected to the new location. This is, the command is used to change the kinematic topology. The placement of the reconnected frame in absolute workcell coordinates (world) is not modified through this function.

For example, it is used when a tool-exchange is to be performed. Move the robot to the toolbox and the location of the tool. Then reconnect the reference FRAME of the tool with the toolbox reference system. The tool will stay in the toolbox when you move the robot. Next move the robot hand to the new tool at its location in the toolbox. Reconnect the tool reference FRAME with the robot hand FRAME. When the robot is moving, the new tool will be connected with the robot hand.

- **connect GEO**

A GEO-element is reconnected to a new reference FRAME. The placement of the reconnected GEO in absolute workcell coordinates (world) is not modified through this function.

- **to WFRM**

This command is used to reconnect a FRAME with all its associated parts and objects to a

new reference system. The target location is defined in this case by a workframe (WFRM). See also the "Reconnect FRAME" command.

You can use this function to perform a tool exchange without moving to the exact locations of the new reference positions of the FRAME-assembly. The reconnected FRAME will "jump" to the new reference location. You have to define a WFRM at the target positions.

- **Successor**

A new kinematic successor FRAME is assigned to the picked FRAME. This command is useful for joint types 2 and 3. These types are used within closed kinematic loops.

- **Add ABSTRACT**

A reference to another ABSTRACT-file (i.e. a subassembly, a ROBOT- or DEVICE-mechanism, or another DETAIL-level) is added to the FRAME node. In any case, the ABSTRACT reference is in fact an `.mpc` file. The ABSTRACT-file must already exist in the model database, i.e. in `,$kis_home/mpclib`. The file is identified by picking in the file panel.

The user must specify the initial read-status of the new ABSTRACT-reference. There are two choices, either initial "READ" or "WAIT". This is:

- **READ** the file is opened immediately, all substructures are loaded and activated for display.
- **WAIT** only the reference to the file (name) is added to the model-tree. The new file will be loaded on user request. For this, refer to the commands:
 - "Activate"
 - "Load"
 - "Swap LEVEL"
 - "More GLOBAL"

in the "DETAIL" submenu. Similar commands are available in the SCRIPT interface.

- **Print Nodename**

The fully qualified nodename of the FRAME is printed in the KISMET-textport. The nodename is a list of all FRAME-names from the model-root (ROOT-FRAME) to the selected FRAME. The frame names are separated by a `.` (dot). The full qualified nodename is used in SCRIPT-files to uniquely identify a FRAME or ABSTRACT node. This naming convention is necessary, because KISMET allows instancing of GEO-parts, FRAMEs and ABSTRACT-files.

15.4. Edit COLOUR

KISMET is using two tables for lighting and/or shading. An RGB colour table which is used for wireframe drawing (no lighting), the background and everything around the workcell display. Another table is used for MATERIAL definitions. A MATERIAL defines the basic colour and the reflection parameters of GEO-surfaces. The two tables are, with some limitations, user definable. They are set in the colour (extension `.col`) and material (extension `.mat`) files, re-

spectively. See also chapter 15.5. Edit Material.

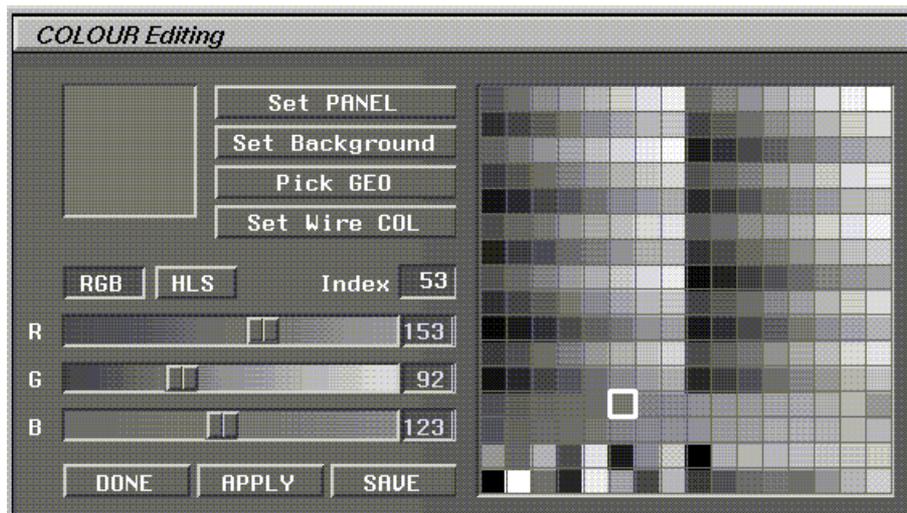


Figure 52: The Colour Editing Panel

Set PANEL

The settings of the current colour are used for the panel background (i.e. for the user interface).

Set Background

The selected colour is used as background colour for all views.

Pick GEO

Selects the GEO part, the colour of which should be changed.

Set Wire COL

Sets the wireframe colour of a selected GEO-part.

RGB/HLS

Allows the modification of the

- colour parameters R,G and B (Red, Green and Blue- according to the RGB-colour model)
- or the colour parameters H, L and S (Hue, Lightness, Saturation - according to the HLS-color model).

The current MATERIAL-index (see "Set MATERIAL Index" command) is used.

Index

Shows the index of the currently selected colour for editing. This number corresponds to the index in the KISMET colour file (.col)

DONE

The colour editing window will be closed

APPLY

The whole colour table as edited is transferred to the scene (until now all colour modifications are local to the colour editing window).

SAVE

The whole colour table as edited previously is transferred to the scene (until now all colour modifications are local to the colour editing window). Additionally a colour file (.col) is saved to the harddisk, i.e. this function is used for permanent changes to the models colour table.

15.5. Edit Material

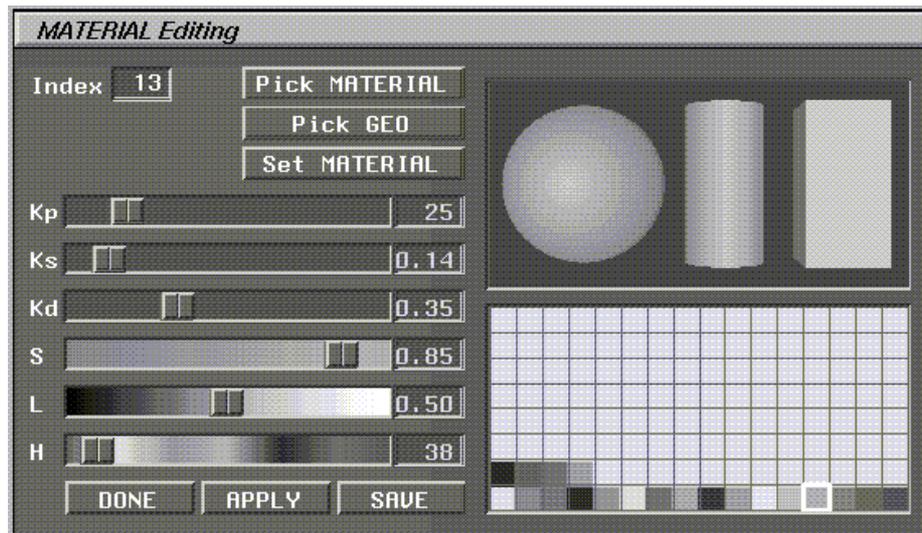


Figure 53: MATERIAL Editing

- **Index**
The index corresponds to the material definition in the `mat` file.
- **Pick MATERIAL**
Selects the MATERIAL index to be edited from the scene window. You have to pick a GEO-part from the scene window. The MATERIAL parameters associated with this GEO-part (and possibly some other parts) will be subsequently modified using the sliders.
- **PICK GEO**
This command is used to identify a GEO-part in the main scene window. The MATERIAL index as selected in the MATERIAL will be attached to the GEO-part as picked, when the **Set MATERIAL** button will be pressed subsequently..
- **Set MATERIAL**
The currently selected MATERIAL is attached to the selected GEO-part. Use the **PICK GEO** button to select the part
- **Kp, Ks, Kd**
Allows modification of the MATERIAL parameters Kd, Ks and Kp. (coefficients for Diffuse Reflection, Specular Reflection, Specular Power).
- **S, L, H**
Allows modification of the MATERIAL parameters S, H and L. (coefficients for Saturation,

Hue and Lightness) according to the HLS colour model..

- **DONE**

This button is used to terminate MATERIAL editing mode. The MATERIAL window will be closed. No changes will take effect in the scene window unless you have pressed the APPLY button before.

- **APPLY**

The whole MATERIAL table as edited is transferred to the scene (until now all modifications are local to the editing window).

- **SAVE**

The whole MATERIAL and COLOUR tables as edited are transferred to the KISMET scene (until now all modifications are local to the colour editing window). Additionally the MATERIAL and COLOUR files (.mat and .col) are saved to the harddisk, i.e. this function is used for permanent changes to the models MATERIAL table.

16 DYNAMICS

16.1. Introduction to "Elastomechanics"

16.1.1 Elastomechanics Basics

KISMET provides the possibility to calculate the static deflections of a robot in real time. Therefore it is required to expand the robot model, i.e. to provide the necessary data-sets. In order to extend the model, there are 3 frametypes, which have to be integrated into the kinematical model and enable KISMET to fulfill the task.

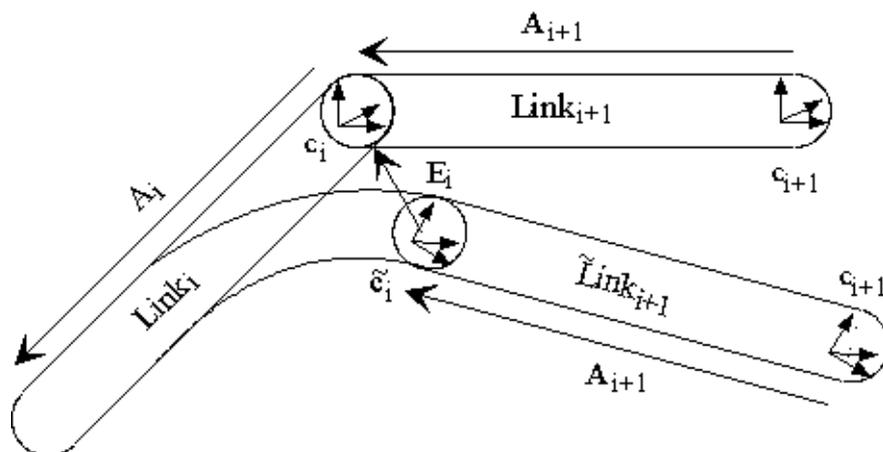


Figure 54: Rigid and non-rigid kinematical chains

Figure 54 shows two links of a kinematical chain, connected by a revolute joint in the rigid and non rigid case. The matrices A_i are the relative transformation matrices between two joints in the stiff case, while the E_i denote the relative transformation matrix between a joint in the "flexible" case and the same joint in the rigid case.

To enable KISMET to show both cases alternatively, the rigid model of a robot or a kinematical chain has to be extended as shown in figure 54. Two main steps have to be done by the user:

1. A frame \tilde{c}_i must be created and connected with the frame c_i . Frame c_i indicates the position and orientation of joint i in the rigid case, while \tilde{c}_i is responsible for the nonrigid case.
2. The order of succession of the frames has to be changed for the nonrigid case. While frame c_i is the predecessor of frame c_{i+1} in the rigid case, frame \tilde{c}_i is the predecessor in the nonrigid case. KISMET provides two data sets, to calculate and show the robot in the rigid and non rigid case simultaneously.

In the next chapter the necessary steps for extending an existing robot model is shown in detail. For further information about the kinematics of flexible links and joints see [39],[40],[41], or [42].

16.1.2 Modeling

3 frame types to handle the modeling of a flexible robot have been implemented in KISMET:

- "Deflected Joint Frame": Showing the new, deflected position and orientation with respect to the undeflected position and orientation. This frame must be created using the "EDIT FRAME", "CREATE FRAME" submenu, using the joint frame as kinematical predecessor,

located at the same position as the joint frame. In the example shown in the previous chapter, c_i denotes the joint frame and \tilde{c}_i denotes the "Deflected Joint Frame".

- "Centre of Mass Frame": Located at the centre of mass of a link, using a frame of frame type 10 as predecessor, this frame type is used to define certain material datas, e.g. stiffness matrix, modulus of elasticity, mass, etc.

Note: the orientation of this frame has to be the same as that of the predecessor frame.

- "Elastic Joint Frame": Located at the position of the joint frames, using a frame of type 10 as predecessor, this frame is used to define the elasticity of the joints.

Note: the orientation of this frame has to be the same as that of the predecessor frame.

The main steps of modeling are:

1. Create a "Deflected Joint Frame" (frametype 10) and connect it with the first joint-frame to simulate deflections arising from joint elasticities. Use the next joint frame as successor frame.
2. Repeat step 1 for all joints.
3. Create a "Centre of Mass Frame" (frametype 60) and connect it at the "Deflected Joint Frame" created in step 1.
4. Move "Centre of Mass Frame" to the center of gravity of the first link. Take care that the orientation of the frame is the same as that of the predecessor "Deflected Joint Frame".
5. Create a "Elastic Joint Frame" (frametype 61) and connect it to the "Deflected Joint Frame" created at step 2. Take care that there is no difference between the orientation of this frame and the predecessor frame.
6. Repeat step 4-6 for all joints.
7. Define the external forces acting on the tip of the robot.
8. Switch into the "Deflection Mode".

Step 1-3 can be done under the "Edit Frame/Create Frame" Submenu.

Notice: If you create two "Deflected Joints" for one joint, both demanding a predecessor and successor, the second one is of course the successor of the first one. You can solve the problem of connecting the first one with a successor, which is not yet existing (the second frame) by using an arbitrary joint as successor and changing that after the creation of the second frame by using the "Set Frame Successor" submenu in the "Edit Frame" menu.

The creation of the frames in step 4 and step 5 can also be done in the "Edit Frame"/"Create Frame" submenu. For the input of the material data sets there are two possibilities. KISMET needs two data set files for each joint, which will be saved in the directory "mpdlib". On the one hand a file with the suffix ".stf" containing the stiffness matrix of the link, on the other hand a file with the suffix ".def", containing additional material data. If the links of the robot can be modeled as bars, the input of the data set for the ".def" file is sufficient. The stiffness matrix for the ".stf" file will be generated automatically. In any other case the stiffness matrix has to be defined by the user. Both cases will be handled in the "Create Frame" submenu.

Notice that for the modeling of the flexibility of the joints a file with the suffix ".stf" will be generated as well. Due to the modeling of joints as springs, this matrix has a diagonal form with the spring constants for the x,y,z- directions as elements.

The generated files will be saved if you use the "Edit Abstract"/ "Write ABS.FILE" submenu in the directory "mpdlib". The filenames consist of the name of the links of type 60 or 61 with the

suffixes ".stf" and ".def"

The units used in the present version are: [N], [mm] , [kg] , [rad] . For the output on the screen [grad] is used.

For step 8, the starting of the real time calculating of the deflections press the "L" key. Returning to the "stiff" case is enabled by pressing the "L" key again.

16.2. "Dynamics" and "Control" Basics

Kismet simulates the so called „Inverse-“ or „Indirect-Dynamic“ behaviour of a kinematic structure as well as the so called „Direct Dynamics“. Inverse Dynamics means to calculate the necessary torques at the joints in order to get the desired behaviour of the joint variables . Direct Dynamics means to calculate the behaviour of the joint variables due to given torques . The Inverse Dynamics feature enables the user to calculate and display the torques and force at joint level during execution of a robot teachfile or robot program. Pendulums are simple examples for the Direct Dynamics feature of KISMET. If you want to simulate the „direct dynamic“ behaviour of robots, use the „Control“ menu, which is responsible for the simulation of the dynamic behaviour of a mechanism combined with control algorithms enabling the user to specify .

	Direct Dynamics	Inverse Dynamics
Kinematic structures with no active elements (e.g. pendulums)	Dynamics Sub-menu	Dynamics Sub-menu
Kinematic structures with active elements (e.g. robots)	Control Sub-menu	Dynamics Sub-menu

Table 9: Sub-menu selection

16.3. Dynamics Main Menu

The following sub-menus are provided by the MBS-Dynamics panel:



Figure 55: Dynamics Main Menu

Explanation:

- **Elastomechanics**

Leads to the Elastomechanics Sub-menu. This menu is used to enter required properties (gravity, forces) for the simulation of elastomechanic effects like torsion and bending of robot links. An elastomechanic approach is used in KISMET to allow realtime calculation of deflections. The stiffness data required for this option (reduced stiffness matrices) are provided by FEM-programms. The FastKey "L" is used to toggle the elastomechanics option. The default value is "OFF". See chapter 16.3.1 The Elastomechanics Panel.

- **Dynamics**

Calls the DYNAMICS submenu for realtime simulation of the dynamic behaviour of mechanisms. Direct Dynamics as well as Invers Dynamics will be calculated in real time. See chapter 16.3.2 Dynamics Sub-menu.

- **Control**

Leads to the Control Sub-menu for realtime simulation of the dynamic behaviour of robots resulting from the dynamic properties of the robot itself and its control system. See chapter 16.3.4 The Control Panel.

16.3.1 The Elastomechanics Panel

The following menus and functions are provided by the Elastomechanics panel:

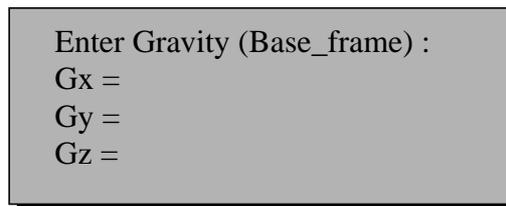


Figure 56: Elastomechanics Panel

- **Edit Gravity:**

Input of gravity direction and acceleration with reference to the base frame. Default values:

$$G_x = 0 \text{ m/s}^2, G_y = 9.810621 \text{ m/s}^2, G_z = 0 \text{ m/s}^2$$



- **Edit external Forces:**

Input of external forces and torques acting on the last created „Deflected Joint Frame“.

Enter external Forces (Base Frame):
 fx =
 fy =
 fz =
 mx =
 my =
 mz =

- **Scale Deflections:**

Input of the deflection scaling factor. Enables the magnifying of the deflections.

Rotational Scale:
 Old Value: 1.0 New Value:
 Translational Scale:
 Old Value: 1.0 New Value:

- **Enter Deflection Values:**

Input of Deflection Values

Change at Joint 1 (y/n) ?
 Actual Value: 0.4567
 New Value:

- **Enter Stiffness**

Creation of a new stiffness matrix file by either typing in the values or by modification of an existing ".stf"-file.

Do you want to use an existing data set (y/n)?
 comment:
 type:
 area:
 .
 .
 .

- **Enter JOINT Data:**

Input of the spring constants describing the flexible behaviour of the joints.

Enter Joint Data: Kx = Ky = Kz =

- **Set Limits:**

Determination of the greatest absolute value of a stiffness matrix file. Can be useful to determine a scaling factor.

- **Show Deflections:**

The elastostatic behaviour of a mechanism will be calculated and visualised on the screen. Fastkey: „L“

- **Diagnostic FILE:**

Saves a diagnostic file containing all stiffness Matrices, forces and torques of the current model. Filename: „defl_diag_out“.

16.3.2 Dynamics Sub-menu

The following functions and menus are provided by the Dynamics sub-menu:

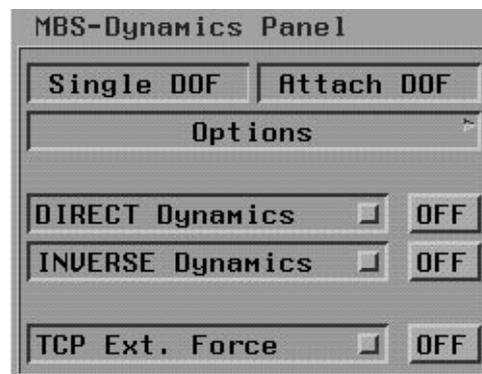


Figure 57: The Dynamics Panel

- **Options:**

Leads to the Options sub-menu (see Chapter 16.3.3 Dynamics-Options sub-menu).

- **Direct Dynamics [On/Off]:**

Start (or Stop) of the real-time simulation of the dynamic behaviour of all mechanisms in the current model that have been enabled for this purpose. Default parameters such as simulation time, step size, etc. are defined in the „kismetrc“-file and can be changed using the Options sub-menu „Initial Conditions“ (see Chapter 16.3.3 Dynamics-Options sub-menu).

- **Inverse Dynamics [On/Off]:**

Starts the Inverse Dynamics feature. According to the options defined in the Options sub-menu torques and/or forces will be displayed. There are basically 3 main applications for

this menu:

1. Using Inverse Dynamics together with one of the „Motion“ sub-menu: according to the configuration of the kinematic structure the static forces and torques will be displayed.
2. Using Inverse Dynamics together with the „Direct Dynamics On“ sub-menu: the forces and torques in the non-static dynamic case will be displayed.
3. Using Inverse Dynamics together with the „Run teachfile“ sub-menu: during the execution of a robot teachfile the forces and torques necessary to perform the demanded motion will be displayed. Usage: 1.) „Inverse Dynamics Start“ 2.) „Run teachfile“

- **TCP Ext. Force:**

Visualisation of the forces resulting from the contact of a mechanism with an elastodynamic object. Furthermore the resulting forces and torques in all joints of the colliding mechanism will be calculated and visualised if demanded.

16.3.3 Dynamics-Options sub-menu

The following commands and sub-menus are provided :

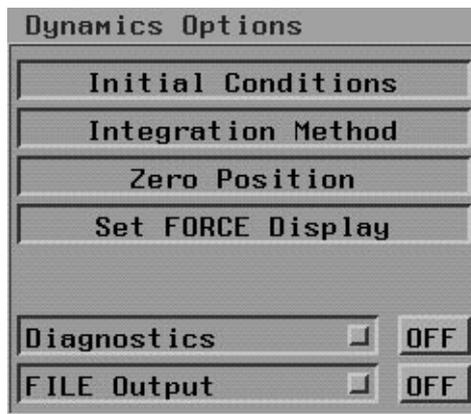


Figure 58: Dynamics Options

- **Initial Conditions:**

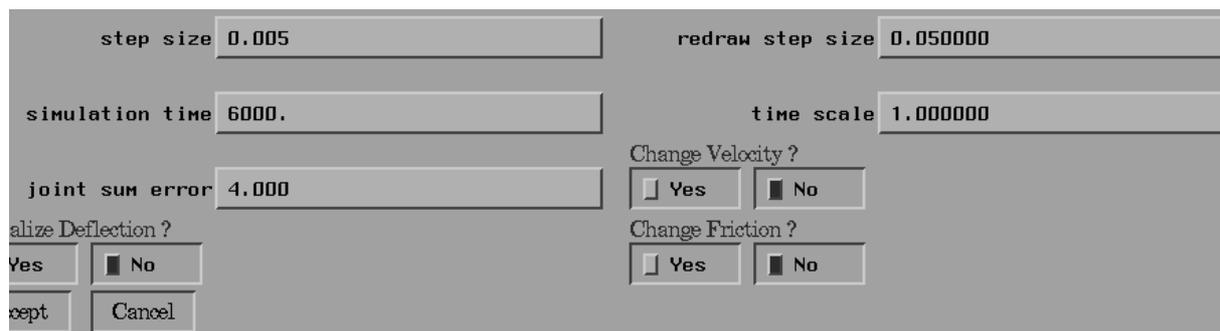


Figure 59: Initial Conditions

- **step size:**

Definition of integration step size

- **redraw step size:**

influences the relationship of CPU-time provided for drawing routines and CPU-time spent for calculation of the dynamic behaviour.

- **simulation time:**

self explainable, isn't it?

- **time scale:**

Defines a „slow motion“ factor. `time_scale = 10` means that for a simulation time of 6 sec the simulation will be 10 times slower than real-time and therefore will last 60 seconds.

- **joint sum error:**

Normally the moment of inertia matrix will be calculated and inverted each time step, which is very cpu-time consuming. For slow moving objects it might not be necessary to calculate this matrix every time step. For a `joint_sum_error > 0` the calculation of the inertia matrix will take place if:

$$\sum_{i=0}^n (j_{i, \text{new}} - j_{i, \text{old}}) > \text{joint_sum_error}$$

with: $j_{i, \text{new}}$ = current joint value of joint i

and: $j_{i, \text{old}}$ = value of joint i during at matrix update

- **Change velocity ?:**

The default velocity of all joints is set to 0 at the beginning of the dynamic simulation. Use `Change_Velocity` to change the initial velocity of all joints.

- **Initialise Deflection?**

If your dynamic model contains flexible joints you can change the initial deflection of the joint and it's corresponding link. The default value is set to 0.

- **Change friction:**

Changes the friction parameters.

- **Integration Method:**

Defines the integration method: Runge-Kutta, Polygon, Heun, Craig or Adams-Bashford. Default: Polygon

- **Zero Position:**

If your model contains flexible joints you can change the initial deflection of the joint and it's corresponding link by defining a new zero position of your joint.

- **Set Force Display:**

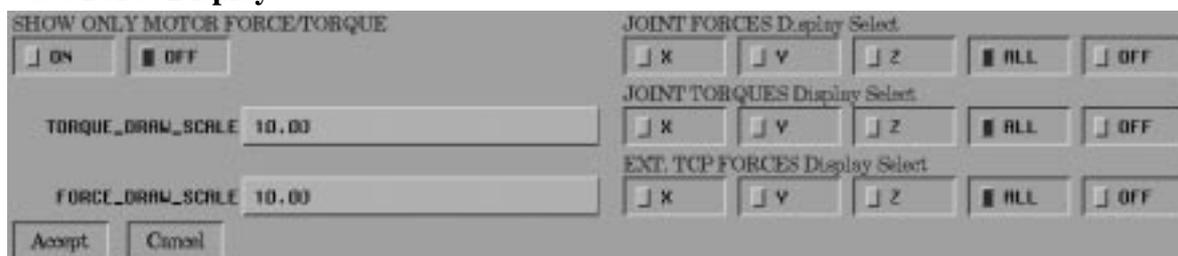


Figure 60: Set Force Display

Visualisation of torques and forces as arrows. You can display all components of the force/torque vectors or just the x-, z, or z- component. „Show Only Motor Force/Torque“ is very

useful in combination with the „TCP Ext. Force“ feature of the Dynamics-Sub-menu (see Chapter 16.3.2 Dynamics Sub-menu). Only the components that correspond to the motor torques/forces are displayed (z-component of the torque for rotational DOF, z-component of the force for translational DOF). This is for example very useful for force feedback applications where these torque components can be used to control motors. The default values can be found in the „kismetrc“-file in the current kis_home directory.

- **Diagnostics:**

Only for development purpose.

- **FILE Output:**

Output of different user definable values into a file during a dynamic simulation. This function contains 2 defined configurations with default values specially chosen for Direct- and Inverse- Dynamics and one function enabling the user to define his own configuration with variables of interest.



Figure 61: File Output Panel

- **Status [On/Off]:**

Defines whether the chosen variables will be written out into a file during the dynamic simulation or not.

- **Dir. Dyn.:**

$s(t)$ will be saved in a file.

- **Ind. Dyn.:**

$s(t)$, $v(t)$, $a(t)$ and $t(t)$ will be saved in a file.

- **User Def.:**

A user defined set of variables will be saved in a file. The user can select a set of variables created and saved earlier or create a new file („diag“-file).

Usage:

After selecting "User Def" a file selection menu displaying already existing „diag“ files will be shown. The user can

a) select a file and leave the selection menu with "Accept" or

b) the user can leave the selection menu with "Cancel" in order to create a new

„diag“- file. The following variable select menu will appear:

Position	Yes	Position_X	No
Position_y	No	Position_Z	No
Velocity	Yes	Acceleration	Yes
Force_X	No	Force_Y	No
Force_Z	No	Force	No
Torque_X	Yes	Torque_y	Yes
Torque_Z	Yes	Torque	No
Link_Defl_Tx	No	Link_Defl_Ty	No
Link_Defl_Tz	No	Link_Defl	No
Link_Defl_Rx	No	Link_Defl_Ry	No
Link_Defl_Rz	No	Controller_In	No
Controller_Out	No	dummy	No
Actuator_Angle	No	Demanded_Pos	No
Control_Block_Nr	No	Control_Block_Nr	No
Filename	testfile.diag		
Accept	Cancel		

Figure 62: Variable select menu

After leaving the variable select menu the file select menu will appear again.

16.3.4 The Control Panel

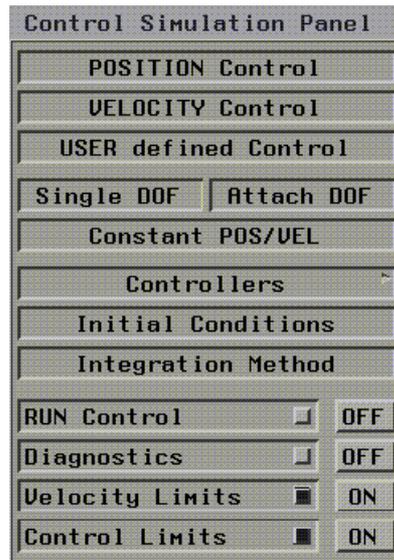


Figure 63: The Control Panel

- **POSITION Control, VELOCITY Control, USER defined Control:**

Selection of the desired typ of control system. Position- and Velocity Control uses a default PID- control system which can be configured as P-, PI, or of course PID- control system by using the Controllers submenu. See chapter 16.3.4.1 The Controller Panel for details.

- **User Defined Control:**

Uses the control system defined by the „.csf“ files. See the *KISEMT Data Specification* for details.

- **Single DOF/Attach DOF, Constant POS/VEL:**

Input of nominal values. The user can either specify a constant nominal value by using Constant POS/VEL or non-constant values .

Non constant nominal values can be changed by using the Single DOF/Attach DOF command. The user can associate a specific joint to one of the mouse buttons (left- or middle-mouse-button). The velocity- or position- nominal-value of the specified joint will be changed by moving the mouse. Both, the nominal values and the current joint values will be displayed on the screen in the upper right corner. To select this feature press Single DOF first followed by Attach DOF. Use the right mouse button to get a list of all available joints.

- **Controllers:**

Leads to the Controllers sub-menu. See chapter 16.3.4.1 The Controller Panel for details.

- **Initial Conditions:**

Leads to the Initial Conditions sub-menu. See chapter 16.3.3 Dynamics-Options sub-menu, for details.

- **Integration method:**

Selection of the integration method. See chapter 16.3.3 Dynamics-Options sub-menu, for

details.

- **Run Control:**

Starts the simulation of the dynamic behaviour of the mechanism and its control system.

- **Diagnostics:**
- **Velocity Limits:**
- **Control Limits:**

16.3.4.1 The Controller Panel

The "Position Control/Velocity Control"- option uses a default PID-control system. With this sub- menu the user can

- select a P-, PI- or PID-control system and
- change the default control parameters.

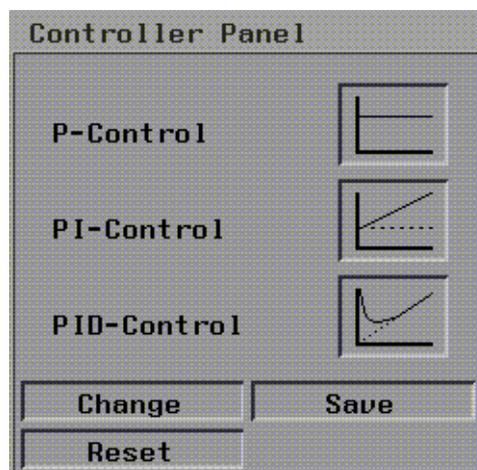


Figure 64: The Controller Panel

After selection of either "Position Control" or "Velocity Control" the default values are loaded by reading the „.con“ files from the directory „kis_home/mpdlib“. The filenames consist of the name of the Centre-of Gravity frame (type 60) and the suffix „.con“. If no files are available the user has to define them via textport.

- **P-Control:**

Automatic loading of the P-Control Parameter K_r or input via textport for each Joint:

```
Joint(Example_1) Actual Value Kr = 1 New Value:
```

- **PI-Control:**

Automatic loading of the PI-Control parameters K_r and T_i for each joint or input via textport:

```
Joint(Example_1) Actual Value Kr = 1 New Value:
Joint(Example_1) Actual Value Ti = 1 New Value:
```

- **PID-Control:**

Automatic loading of the PI-Control parameters , , and for each joint or input via textport:

```
Joint(Example_1)Actual Value Kr = 1 New Value:  
Joint(Example_1)Actual Value Ti = 1 New Value:  
Joint(Example_1)Actual Value Td = 1 New Value:  
Joint(Example_1)Actual Value Tv = 1 New Value:
```

- **Change:**

Select the type of controller you want to change and enter the parameters as described above.

- **Save:**

Saves the control parameters for your selected type of controller. Reset: Resets the parameters of the selected type of controller to default values (if available).

- **Reset:**

Resets the parameters of the selected type of controller to default values (if available).