INFRANOR

OPERATING MANUAL

SERIES SMTBD1

OPTIONAL FUNCTIONS

(Version 1.0)

European version 1.0

BPCW OPTIONS

This manual describes various options of the BPCW software for the control of the SMTBD1 amplifier: **Motor cogging torque compensation, digital oscilloscope, dynamic drive simulation**. The general information about the digital amplifier commissioning are described in the standard SMTBD1 manual.

Maintenance procedures should be attempted only by highly skilled technicians using proper test equipment. Read your warranty provision carefully before attempting to adjust or service the unit.

RECEIVING AND HANDLING

Upon delivery of the equipment, inspect the shipping containers and contents for indications of damages incurred in transit. If any of the items specified in the bill of lading are damaged, or the quantity is incorrect, do not accept them until the freight or express agent makes an appropriate notation on your freight bill or express receipt.

Claims for loss or damage in shipment must not be deducted from your invoice, nor should payment be withheld pending adjustment of any such claims.

Store the equipment in a clean, dry area. It is advisable to leave the equipment in its shipping container until ready for use. Each amplifier is checked carefully before shipment. However, upon receipt, the user should make sure that the amplifier received corresponds to or is properly rated in terms of rated voltage and current for the type of motor which is to be driven. The descriptive label affixed to the amplifier specifies electrical ratings.

OPERATING MANUAL
INFRANOR
SERIES SMTBD1
CT-EMF OPTIONS
(APRIL 1997)

1.0.0 GENERAL

1.1.0 Introduction

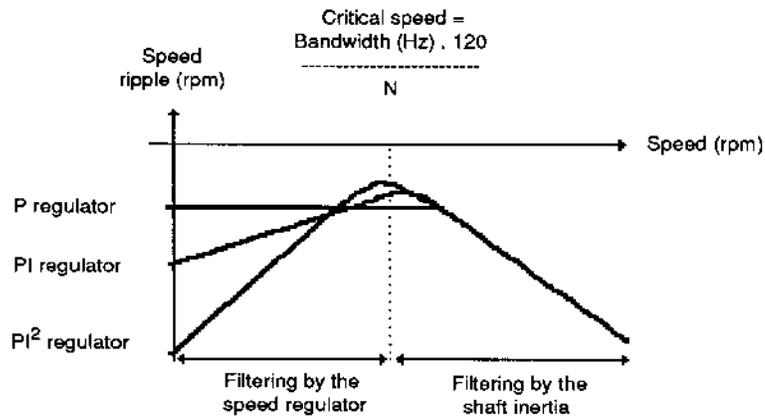The cogging torque of brushless permanent magnet motors results from the interaction between the rotor magnets and the stator slots. This torque is due to the difference of reluctance between the copper of the windings and the iron for the stator teeth as shown below. This disturbance is called "Cogging torque".
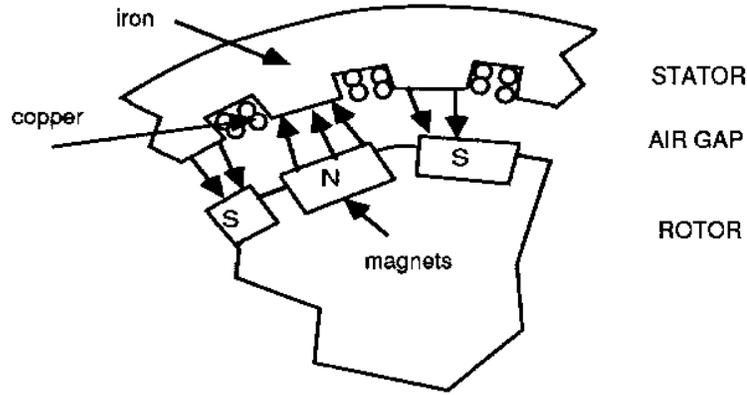
The cogging torque of a brushless permanent magnet motor is measured at motor standstill and when power off. It can easily be empirically evaluated by simply moving the stator manually; the number of resistant points over one shaft revolution (N) depends on the number of slots and magnets per motor pole. The amplitude of the cogging torque disturbance is a multiple of the motor rotation speed.

The cogging torque influence on the motor speed ripple is shown below. At high rotation speeds, the cogging torque is naturally filtered by the motor inertia and has little influence on the motor speed ripple. At low speeds, the cogging torque is filtered by the PI or PI^2 regulator and has also little influence on the speed ripple. But the cogging torque generates a high speed ripple when reaching the critical speed level which is depending on the regulator bandwidth (**Bandwidth**) and on the motor specifications (**N**).

The **Cogging torque compensation** option, available in the SMTBD1 amplifier allows cogging torque compensation on any brushless permanent magnet motor with "speed 1" resolver, after having made a previous identification on the motor and saved the disturbance shape. This compensation permits increased the torque control accuracy in special applications (force control on the motor shaft in torque mode) and to reduce the speed ripple around the critical speed value.

2.0.0 <u>DESCRIPTION</u>

The cogging torque compensation on a brushless permanent magnet motor requires the **CT-EMF** option on the SMT-BD1 amplifier. The hardware key allowing the access to the option menus of the **BPCW** parameter setting software is also necessary and must be plugged into PC parallel port.

The functions concerning the cogging torque compensation are in the **Advanced functions** menu of the **BPCW** software shown below.

The **Cogging torque acquisition procedure** function allows the automatic identification of the motor cogging torque. The correct identification of the cogging torque is described in Chapter 3, section 3 (Commissioning).

The **Cogging torque compensation** function allows for enabling or disabling the cogging torque compensation after having made an identification by means of the function described above. The function enabling is displayed by a screen marker in the **Advanced functions** menu.

3.0.0 COMMISSIONING

3.1.0 Warning

This option is only to be used in applications where the motor cogging torque influence inhibits the required performance: accurate force control on the motor shaft in torque mode or very low speed ripple at the critical regulation speed level (see Chapter 1, Introduction). It is possible to check, by means of the following formula, the influence of the motor rotation speed on the speed and torque ripple frequency:

Ripple frequency (Hz) ~ motor speed (rpm) x number of stator slots x number of rotor magnets per pole / 60.

CAUTION

The cogging torque compensation can only be used on motors with a one pole pair resolver "speed 1".

3.2.0 Checking the configuration

Check the standard amplifier configuration as described in Chapter 6 of the standard manual "SERIES SMTBD1".

Check the presence of the **CT- EMF** option on the logic board (see Appendix "Location diagram of hardware options").

Check that the **Cogging torque acquisition procedure** and **Cogging torque compensation** functions are accessible in the **Advanced functions** menu of the **BPCW** software. If not, check that the hardware key is correctly plugged in the parallel port (the PC must be off).

3.3.0 Powering

Turn amplifier on as described in Chapter 6 of the standard SMTBD1 manual.

3.4.0 Starting and adjustment

Disable the **Cogging torque compensation** function in the **Advanced functions** menu.

Start and adjust the amplifier as described in Chapter 6 of the standard SMTBD1 manual, by means of the **BPCW** software.

3.5.0 Automatic cogging torque identification

The motor must be uncoupled from its load in order to avoid any disturbance on the shaft during the identification procedure (load torque, viscous friction, etc..).

Select the **Software control** function in the **Setup** menu (only in the BPCW parameter setting software version 2.0 and next) and switch on **STOP** position in the **RUN** module.

Select the PI^2 speed regulator and run the **AUTOTUNING** function of the **CONTROLLER** module by choosing the maximum bandwidth (**Bandwidth = High**).

Check that the motor shaft is free and that its rotation will not endangedred the operator.
Check further that the limit switches (FC+ and FC-) and the CI input are disabled.

Execute the **Cogging torque acquisition procedure** from the **Advanced functions** menu. The acquisition procedure includes a motor shaft rotation of 1 to 2 revolutions in both directions. The motions are made at low speed and the whole procedure only takes a few minutes. The motor shaft must not be disturbed during the identification procedure (load torque or shaft handling).

After the identification procedure, the motor cogging torque is automatically stored in a memory of the amplifier. When changing the motor or the amplifier, the automatic identification procedure of the motor cogging torque must be redone according to the description above.

3.6.0   Compensation utilization

Select the **Cogging torque compensation** function of the **Advanced functions** menu. Function enabling is displayed by a screen marker in the menu.

To check the compensation effects on the motor's cogging torque, uncouple the motor from its mechanical load in order to release the shaft and proceed as follows:

- Select the amplifier to torque mode by using the current control input CI (pin 4 of X4) and short circuit the analog command input CV (pins 15, 16 and 17 of X4).

- Execute the **OFFSET COMPENSATION** function accessible in the **ANALOG INPUT** module or by means of the **OFFSET** push button on the amplifier's front panel, in order to get an actual zero torque command.

- Put the motor under control by means of the **AUTOMATIC** or **MANUAL** functions of the **RUN** module.

- Check for the reduction of the resistant points due to the cogging torque on the shaft by turning the shaft manually. Compare the behavior **with** compensation (Cogging torque compensation selected in the Advanced functions menu) and **without** compensation (Cogging torque compensation not selected in the Advanced functions menu).

- Check that no resistant point remains over one complete motor shaft revolution when the compensation is enabled. If some resistant points do remain, redo the acquisition procedure described in section 3.5.0 and check for no disturbance of the motor shaft.

4.0.0 <u>APPENDIX</u>

Location diagram of the harware option:
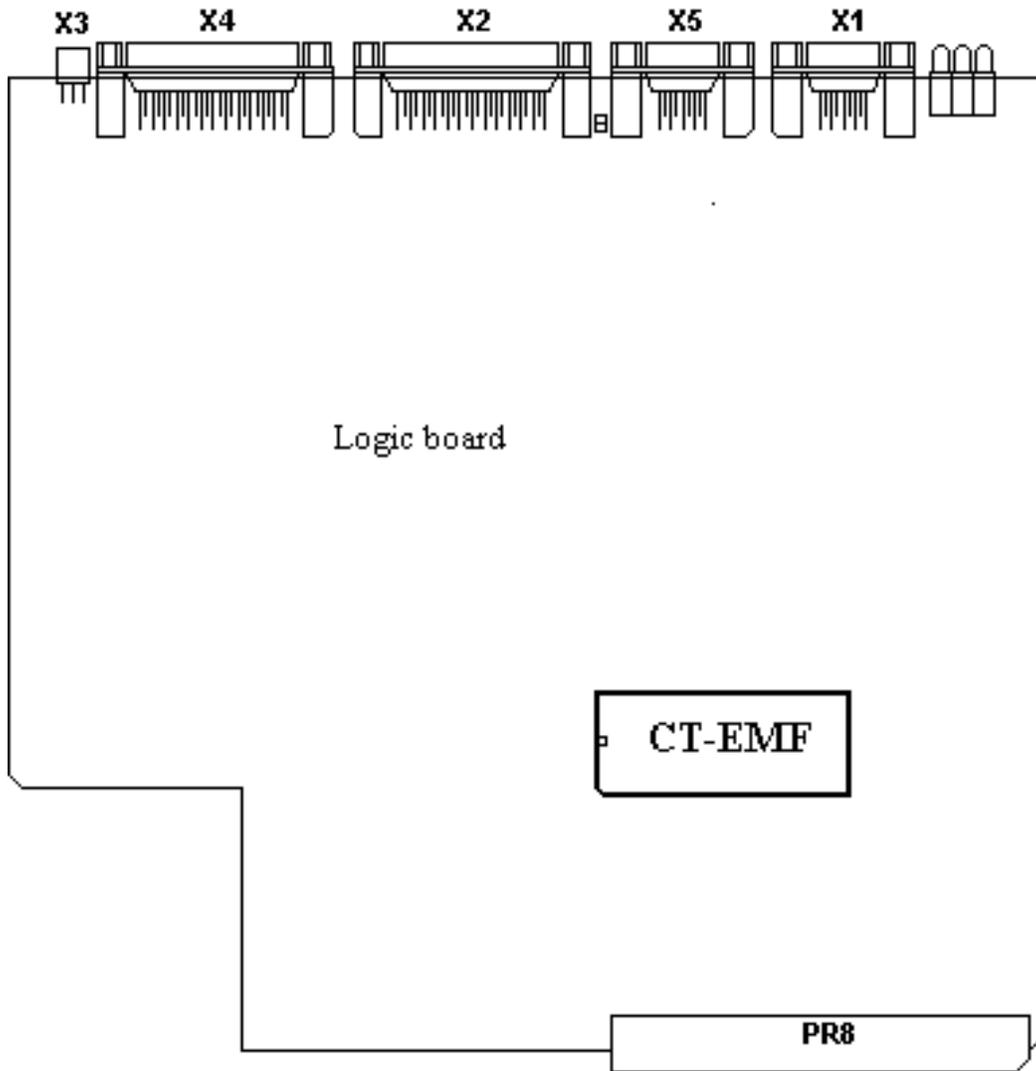
OPERATING MANUAL
INFRANOR
# SERIES SMTBD1
DIGITAL OSCILLOSCOPE OPTION
(APRIL 1997)

1.0.0 GENERAL

1.1.0 Introduction

The use of the **Digital Oscilloscope** function included in the BPCW software requires the hardware key allowing access to the Option Menus. This key must be plugged into the parallel port of the PC. This option simulates the operation of a memory oscilloscope in order to make acquisition of the SMT-BD1 control signals and to display them on the PC screen. The acquisition of the signals to be displayed is not a continuous but a sequential one. At first, the triggering conditions must be defined: during the triggering, the signals are stored in the amplifier and transmitted to the PC by the operator for display.

The graphic window of the digital oscilloscope is made of five different areas as shown below: the parameter setting menus, the screen, the display information, the triggering control and the cursors control.
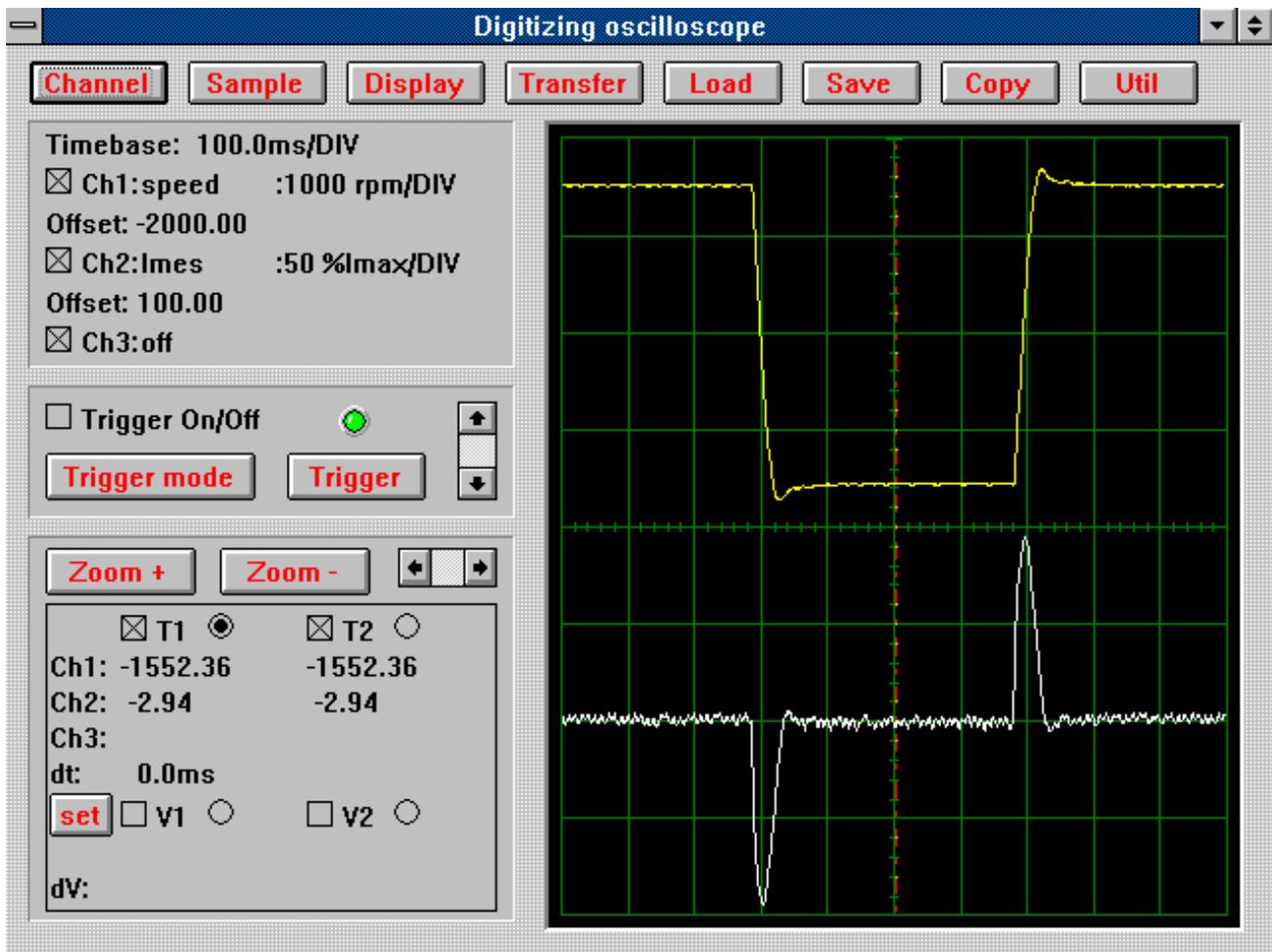
2.0.0 FUNCTIONS DESCRIPTION

2.1.0 Menu description

The menu contains 8 buttons which are directly accessible on the front panel:

| Channel | Sample | Display | Transfer | Load | Save | Copy | Util |

2.2.0 Oscilloscope channel selection (channel)

This dialog box allows the oscilloscope channel selection. It is possible to simultaneously select three channels, but the data transfer time from the amplifier to the PC increases according the number of selected channels. Only the signal of channel 1 can be selected to trigger the oscilloscope (Trigger).

The signals available on these channels are:

Signal	Unit	Signification
Speed	rpm	Speed monitor
Speed ref	rpm	Speed input command
Imes	%Imax	Current monitor
Idc	%Imax	Current control
Iq	%Imax	Current monitor (induction motor)
Id	%Imax	Direct current monitor
I²t	%Imax²t	Calculation of the amplifier I²t
Resolver	resolver resolution	Position given by the resolver
Pos error	encoder edge	Position error (optional:Indexer/Electronic gearing)
Iflux ref	%Imax	Magnetization current inputcommand (induction motor)
Iflux mes	%Imax	Magnetization current monitor (induction motor)

Notes

- The unit of **speed** and **speed ref** in rpm is only correct if it is a two pole, one speed resolver.
- Resolver signal unit: 65536 ppr.
- Imax is the maximum current delivered by the amplifier for a given current rating.
- A change in the use of a channel cancels the curves stored in the oscilloscope (and displayed on the screen).

**Range**: from 0.05 unit/DIV to 10000 units/DIV.

Offset: S ignal level in the middle of the screen. It allows a shift of the signal on the screen. The offset unit is the same as the signal unit.

The signals selected on channels 1 and 2 are also analog signals available on DAC1 and DAC2 outputs of the amplifier test connector X3.

Saving of the parameters in the EEPROM saves the signal DAC1 and DAC2 outputs.

2.3.0   Sample period

This dialog box allows the definition of the sample period. It is between 0,5 ms and 2 s. The signal acquisition is made on this time basis. A change of the sample period value cancels the curves that are presently stored in the oscilloscope (and displayed on the screen).
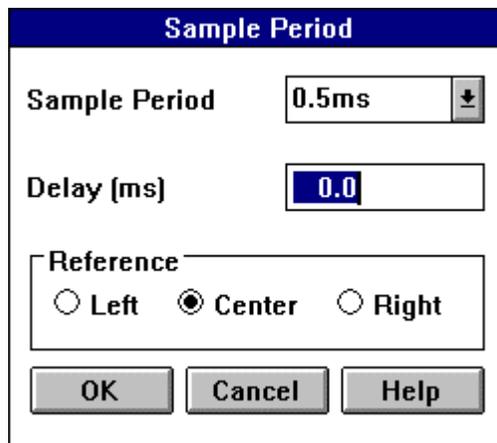
It is also possible to define the triggering level by means of the buttons of the **reference** group and of **delay**:
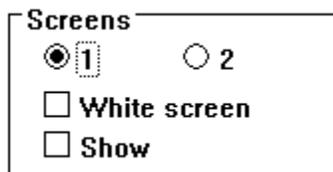Triggering = **reference - delay** .
A **delay** of -2 ms invokes a triggering 2 ms after the reference point.
Reference indicates the place of triggering. It can be located on the left, on the right or in the middle of the screen according to the selected option.

2.4.0   Display Screen

This dialog box define the various controls of the oscilloscope screen display.

1:	The display area is on a single screen and all curves are superimposed on this screen.
2:	The display is on two screens: Channels 1 and 3 on the first screen and Channel 2 on the second one.
White screen:	In standard, the screen background is black. When selecting this option, the background becomes white. This allows to copy the present display and use it in another Windows application with a white background.
Show:	The signal information is displayed at the bottom of the screen. It transferred to the clip board by clicking on the **Copy** button.

○ Off   ○ Frame
○ Axes   ◉ Grid

This option group allows the frame display on the screen:

Off: no graduation on the frame.
Frame: displays the outer edge with graduations.
Axes: displays the horizontal and vertical axes in the middle of the screen, with graduation.
Grid: displays the screen with a grid graduation of 10 horizontal and 8 vertical divisions.

┌ Connect dots ──────
  ○ Off   ◉ On

Connect dots: allows a better signal display (display without point discontinuity).
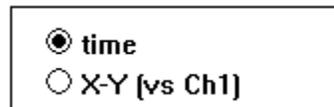
◉ time
○ X-Y [vs Ch1]

Time: Channel signal display according to the time.
X-Y: Channel display according to the signal of Channel 1. Channel 1 is on the X-axis and Channels 2 and/or 3 on the Y-axis.

2.5.0   Signal acquisition (transfer)

The **transfer** function allows for the transfer of the signals acquired in the amplifier or the result signals of a simulation to the oscilloscope.

The transfer from the amplifier (**Transfer from drive**) is only possible when the amplifier is in use.
The transfer time is depending on the communication speed and on the number of selected curves.
Transfer time at 4800 baud: about 12 s per curve.

The transfer is only possible if:
- Triggering has occured (triggering requirements are met),
- Signal acquisition is done. If the sample period is large, there is time delay after the triggering and before the transfer. The resolution is 1024. The acquisition time is: 024 x **Sample Period**.

The transfer from simulation (**Transfer from simulation**) is only possible if the simulation module is open and if a simulation is finished.

2.6.0   Loading a file into the oscilloscope (load)

This dialog box allows to reload into the oscilloscope a file which has previously been saved in the PC. If there is no modification, these file have an **.OSC** extension.
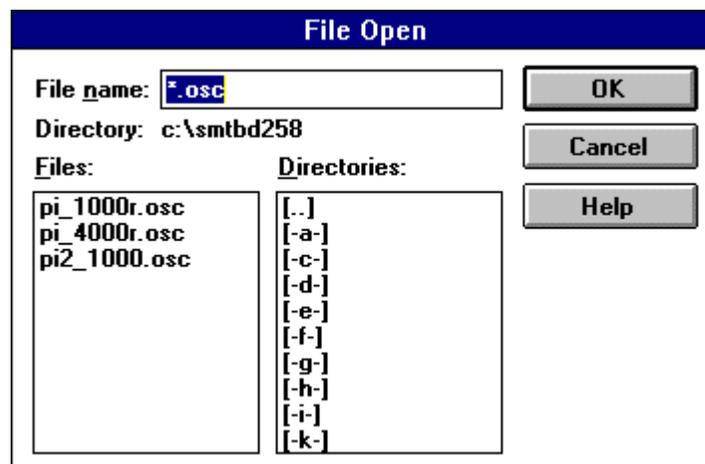
File name: complete file name or filtering mask (***.OSC** for the oscilloscope files).
Files: all file names in the current directory which meet the requirements defined by means of the mask name in the edition box **File name**.
Directory: this box allows to move inside the directories and from one disk to another.

2.7.0 Saving of the oscilloscope curves (save)

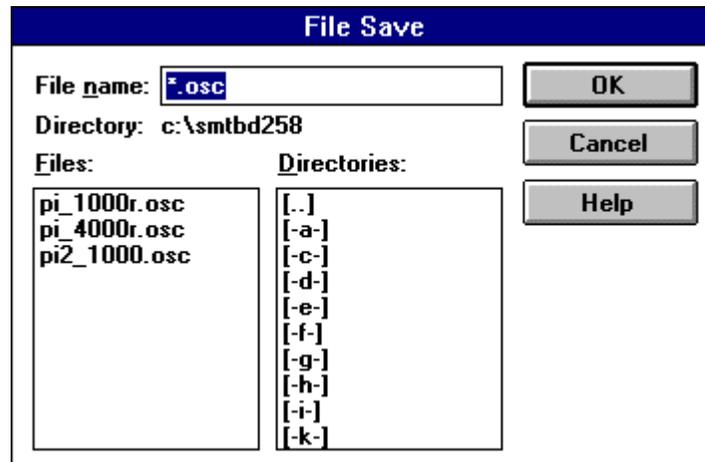This dialog box allows for saving the curves that are presently in the oscilloscope (in the PC memory) in a disk file. This makes it possible to display them later by means of the **Load** function. It is recommended to give these files a name with the **.OSC** extension.

Directory: This box allows to move inside the directories and from one disk to another.
File name: Complete name of the file to be saved.

2.8.0 Oscilloscope screen copy (copy)

Copy the oscilloscope screen to the clip board; it can then be transfered in another application. Before clicking on this button, it is possible to choose the options **White screen** and **Show** in the dialog box **Display**.
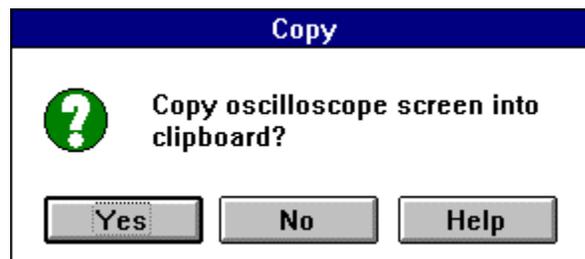
2.9.0 Useful tip for low speed measurement (util)

This function allows the measurement of very low speeds. The measurement resolution of the **speed** signal is 1,8 rpm. Measurements at lower speeds require position derivation.

Procedure:
- Select the resolver signal on channel **1** or **2**. The signal of channel is used for the triggering. Channel 3 must be **off**
- Select the **sample period** parameter according to the required resolution, by means of the following formula:
  speed monitor resolution (rpm) = 0,9 / **Sample period** (ms),
- Define the triggering threshold and mode,
- Click on the **Trigger** and make the transfer as soon as the acquisition is done
- Open the dialog box **Util** and select the appropriate option, The calculated speed signal (**Speed**) will be allocated on the oscilloscope channel 3.

3.0.0 <u>WINDOW DISPLAY DESCRIPTION</u>

3.1.0   <u>Display information</u>

The information concerning the time basis and the oscilloscope channels are displayed in this area according to the options selected by the operator:

```
Timebase:  50.0ms/DIV
☒ Ch1:speed      :1000 rpm/DIV
Offset: 0.00
☒ Ch2:Imes       :10 %Imax/DIV
Offset: 0.00
☐ Ch3:off
```
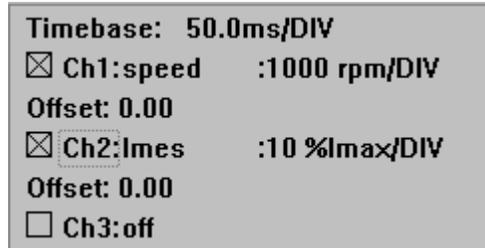
The 3 boxes in the line **Display** allow to temporarily cancel or display the curves on the screen (but they remain in the PC memory). On the black background, curve **1** is yellow, curve **2** is white and curve **3** is clear magenta. On the white background, curve **1** is black, curve **2** is blue and curve **3** is green.

3.2.0   <u>Triggering control (trigger)</u>

Only the channel 1 signal can be used for triggering (trigger) the oscilloscope.

```
☐ Trigger On/Off        ◉        ▲
[ Trigger mode ]    [ Trigger ]   ▼
```

Trigger On/Off allows to cancel or display the triggering level cursor on the oscilloscope screen (in red).

The triggering level is adjusted by means of the "arrow up" and "arrow down".

The **Trigger** button enables the oscilloscope.
After after trigger enabling, the data will be transferred in order to be displayed.

NOTE
The transfer is only possible if:
- Triggering has been made (triggering requirements are met),
- Signal acquisition is over. If the sample period is large, there is a certain delay time after the triggering before making the transfer. The processed resolution is 1024. The acquisition time is: 1024 x **Sample Period**.

The **Trigger mode** button allows to define the triggering mode:
trigger mode

```
          Trigger mode
◉ Positive level        [  OK  ]
○ Negative level        [ Cancel ]
○ Positive edge         [  Help  ]
○ Negative edge
○ Error
```
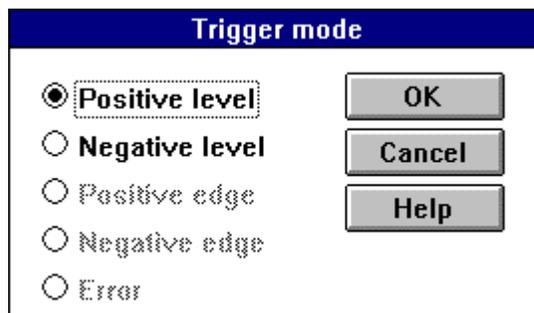
There are 5 ways to control signal acquisition triggering:

- Positive level : signal of channel 1 ~ triggering level,
- Negative level : signal of channel 1 < triggering level,
- Positive edge :on the signal "up" edge,
- Negative edge :on the signal "down" edge,
- Error :when an amplifier default occurs.

The 3 last possibilities are only available on amplifier version 2.0 or higher
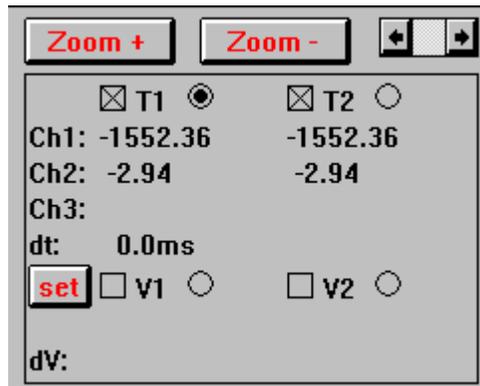
4.3.3 Use of the cursors

The use of the cursors allows accurate measurement of the signal values on the screen.

There are two vertical cursors for time measurement and two horizontal cursors for signals level measurement. Both vertical cursors (T1 and T2) are red and signal values measured at the cursors position correspond to **Ch1**, **Ch2** and **Ch3**. The signal units are indicated in part 2.1.1.

Both horizontal cursors (V1 and V2) have the same color as the signals to which they correspond (defined by the button **set**).

The cursors display is activated or disabled by means of the appropriate boxes. The button allows the selection of the cursor to be adjusted (by clicking directly on the oscilloscope screen).

Both buttons **Zoom +** and **Zoom-** allows to increase or reduce the curves.

4.0.0 <u>COMMISSIONING</u>

Procedure for signal acquisition in the oscilloscope:

- Select the signals by means of the **Channel** button. The oscilloscope triggering signal must be on **channel 1**

- Select the sample period by means of the **Sample** button

- Click, if necessary, on **Trigger On/Off** on the oscilloscope front panel in order to display the triggering cursor on the oscilloscope screen

- Click on the **Trigger mode** button to open the dialog box which allows triggering mode selection

- Adjust the triggering level by means of the "up" and "down" arrows

- Click on the **Trigger** button to activate triggering

- Click on the **Transfer** button and on **Confirmation to transfer data from drive**

- If **Waiting for trigger** message appears, check that the triggering requirement is met or re-adjust, if necessary, the triggering level

- The red LED is on: the signal is triggered (information is not instantaneous !)

OPERATING MANUAL
INFRANOR
SERIES SMTBD1
DYNAMIC SIMULATION OPTION
# (APRIL 1997)

1.0.0 GENERAL

1.1.0 Introduction

The use of the dynamic drive simulation module of the BPCW software requires the hardware key allowing access to the option menus; this key must be plugged into the parallel port of a PC. This simulation module allows to quickly evaluate, with theoretical load data, the dynamic performances of the various drives (amplifier + motor). This method guides the operator in the choice of the most appropriate SMTBD1 amplifier and motor for the application, before making concrete tests on the machine.

The axis drive and load parameters are entered into the simulation panel which is similar to the amplifier adjustment panel. The display (block diagram) allows for quick modification of the load, motor, amplifier, controls and duty cycle parameters. The simulated duty cycle can include up to 10 motions with trapezoidal speed profiles. The software then calculates all amplifier control signals (current, speed and position input commands and monitors during the duty cycle simulation. The " digital oscilloscope " function of the BPCW software makes it possible to display the signals calculated in the simulation for analysis and storage in a PC file.

The analysis of the various simulation results allows for comparison of the current consumption for different motors, to detect current saturation in the amplifier or to test the various drive adjustments in order to guide the operator in the choice of the appropriate drive. Correct simulation results depend on accurate knowledge of the load parameters.

2.0.0 <u>ADJUSTABLE PARAMETERS</u>

The drive and load parameters are entered in the simulation panel accessible in the **Simulation** sub menu of the **Utilities** menu (**BPCW** software). The display as a block diagram allows quick modifications of load, motor, amplifier, drives and duty cycle parameters.
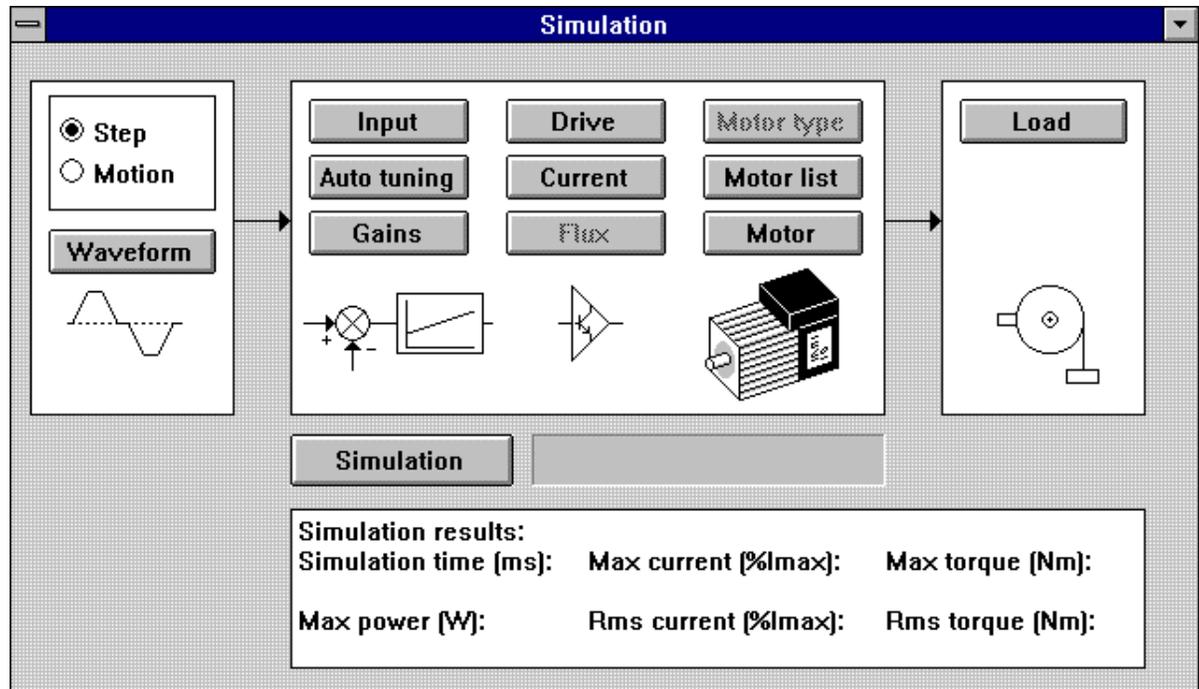
2.1.0 <u>Load model</u>

The **Load** module of the simulation screen modifies all load parameters for simulation calculations.

Shaft reflected inertia (g. m²) parameter defines the value of the load inertia reflected to the motor shaft.

Viscous friction (Nm/rd/s) parameter defines the viscous friction coefficient of the load reflected to the motor shaft: Viscous friction torque (Nm) = Viscous friction * Motor speed (rpm) * 2 * π / 60.

Dry friction (Nm) parameter defines the dry friction torque value of the load reflected to the motor shaft. The sign of this torque is opposite of the motor rotation.

Constant load torque (Nm) parameter defines the constant torque value of the load reflected to the motor shaft (i.e. the torque resulting from gravity, in the case of a vertical load).

2.2.0 <u>Motor model</u>

The **Motor list** module allows for automatic motor parameter initialization by selecting, the model used.

The **Motor** module of the simulation panel modifies the motor parameters for simulation calculation.

The **Torque constant (Nm/A)** parameter defines the coefficient between torque and current in the motor: Motor torque (Nm) = **Torque constant** * Motor supply current (Arms).

The **Inertia (g. m²)** parameter defines motor inertia value.

2.3.0   <u>**Amplifier model**</u>

The **Drive** module of the simulation panel allows the choice of the **SMTBD1** amplifier type used for the simulation calculations.

The **Current size** parameter defines the amplifier current size.

The **Current** module modifies the parameters of the amplifier current limit.

The **Maximum current (%)** parameter defines the maximum current value delivered by the amplifier. It can vary from 20 % to 100 % of the amplifier current size.

The **Rated current (%)** parameter defines the threshold (1²t) of the RMS current delivered by the amplifier. It can vary from 0% to 50% of the amplifier current size.

The **Input** module modifies the parameters of the amplifier speed input command.

The **Maximum speed (rpm)** parameter defines the maximum motor rotation speed. The range is between 100 and 14000 rpm.

The **Acceleration time (ms)** parameter defines the acceleration or deceleration time of the motor from zero to maximum speed. It can vary from 0 to 32 seconds.

The **Autotuning** module makes it possible to choose the regulator type and the bandwidth adjustment used for the simulation calculations.

The regulator type (**P, PI** or **PI²**) is selected in the upper part of the module:
In position **P**, the regulator is only proportional.
In position **PI**, the speed regulator is proportional plus one traditional integral gain.
In position **PI²**, the speed regulator is proportional plus two integral terms. The second term increases the shaft stiffness and the regulation accuracy at very low speeds.

The **Bandwidth** is selected in the lower part of the module: **Low** = 50Hz, **Medium** = 75 Hz and **High** =100 Hz these values correspond to the cut-off frequency for a phase shift of 45$^\text{o}$ of the speed regulation loop. The regulator gains are automatically calculated according to the motor and load specifications when leaving the module by means of the **OK** push button.

The **Gains** module modifies the gain values of the regulator used for the simulation calculations.

The **Speed error low pass filter** parameter defines the cut-off frequency a - 3 db of the first order fitter which acts on the speed error. This parameter value is depending on the selected bandwidth.

The **Proportional speed gain** parameter defines the regulator proportional gain which acts on the speed error. The adjustment range is between 0 and 4 095.

The **Integral 1 speed gain** parameter defines the first integral gain of the regulator, which acts on the speed error. The adjustment range is between 0 and 255.

The **Integral 2 speed gain** parameter defines the second integral gain of the regulator, which acts on the speed error. The adjustment range is between 0 and 1.

Both **Stability Gain** push buttons allow to increase (**-->**) or decrease (**<--**) the stability gain of the speed regulation loop.

2.4.0 Duty cycle model

The **Waveform** module modifies the speed input command used in the simulation calculations. The speed input command type (**Step** or **Motion**) is selected in the upper part of the module.

In **Step** position, the **Waveform** push button defines the amplitude of the speed input command step entered into the amplifier, in the **Speed reference** parameter.

In **Motion** position, the amplifier speed input command corresponds to a duty cycle with a sequence of 10 parameter set motions. The **Waveform** push button defines the parameters of each motion according to a trapezoidal speed path, as shown below.

The **Acceleration time (ms)** parameter defines the duration (in ms) of the motor acceleration.

The **Running speed (rpm)** parameter defines the constant motor rotation speed in rpm.

The **Running time (ms)** parameter defines the duration (in ms) of the motor constant speed.

The **Braking time (ms)** parameter defines the duration (in ms) of the motor braking.

The **Dwell time (ms)** parameter defines the motor standstill duration (in ms) between two rotations.

The thumb-indexes numbered from 1 to 10 allow to select each new motion in order to modify its parameters.

The **View** push button allows to graphically display, in the window at the bottom of the module, the complete duty cycle including all ten motion sequences.

The **Total time** parameter gives the complete cycle duration (in ms) which is displayed in the window at the bottom of the module, via the **View** function.

The **Repeat cycle** function makes it possible to indefinitely repeat the whole programmed duty cycle. This function is very useful for fast programming of the periodical input command signals.

3.0.0 <u>SIMULATION</u>

3.1.0   <u>Simulation calculations</u>

The **Simulation** module makes it possible to calculate all amplifier signals by means of the parameter values defined for the drive model.

The **Sample period (ms)** parameter defines the sample period for the storage of the signals calculated during the simulation. The variation range is between 0.5 and 2000. The storage capacity for the signal display on the digital oscilloscope is 1000 points per signal. The time interval corresponding to the simulation signal display on the digital oscilloscope screen is then equal to: 1000 x **Sample period** seconds.

Example:

Sample period = 1 ms, the simulation signals will be displayed on the screen within an interval of 1s.

The **Delay (ms)** defines the delay value for the storage of the signals calculated during the simulation. The variation range is between 0.5 and 2000. This parameter makes it possible to display the simulation signals with a better resolution on a time interval far from the beginning of the duty cycle.

The **Background computation** function allows the **BPCW** software to make the simulation calculation in the background. It is then possible to go on working on an other windows of the **Infranor digital drive** or **Digitizing oscilloscope** and to communicate with the **SMTBD1** amplifier during the simulation calculations. It is also possible to stop the simulation (**Pause**) in order to modify a parameter and to go on with the simulation (**Resume**).

The **Start** push button starts the simulation calculations.

The **Pause** push button temporarily stops the execution of the simulation calculations in order to modify a parameter, if necessary. The **Resume** push button resums the simulation calculations.

The **Abort** push button stops the simulation and cancells the **Simulation** module.

The **Close** push button allows for leaving the **Simulation** module without modification.

3.2.0 <u>Result processing</u>

3.2.1   <u>Characteristic simulation values</u>

The **Simulation results** module allows for the following evolution of the critical system parameters during simulation. This makes it possible to quickly check for correct motor and amplifier choices in the simulated application.

The **Max current (%Imax)** parameter gives the value of the maximum current delivered by the amplifier during simulation. This value is used to check the choice of the amplifier size.

The **Rms current (%Imax)** parameter gives the value of the RMS current delivered by the amplifier during simulation. This value is used to check the choice of the amplifier rated current (rated current).

The **Max torque (Nm)** parameter gives the value of the motor maximum torque during simulation. This value is used to check the choice of the motor size.

The **Rms torque (Nm)** parameter gives the value of the motor RMS torque during simulation. This value is used to check the choice of the motor rated torque (rated torque).

The **Max power (W)** parameter gives the maximum value of the power consumed by the load during simulation. This value is used to check the motor and amplifier choices

3.2.2 <u>**Simulation signal display**</u>

The signals calculated during simulation can be displayed in the digital oscilloscope module of the **BPCW** software. This module is accessible via the **Digitizing oscilloscope** sub menu of the **Utilities** menu.

The simulation signals to be displayed are selected in the **Channel** menu. The signal choice is the same as one for the SMT-BD1 amplifier signals.

The signal acquisition is made in the **Transfer** menu with the **Transfer from simulation** choice.

After simulation, all signals can be displayed in the digital oscilloscope by selecting them one after another in the **Channel** menu and by making a new acquisition in the **Transfer** menu with the **Transfer from simulation** choice.

The use of the digital oscilloscope is described in the appropriate options manual.