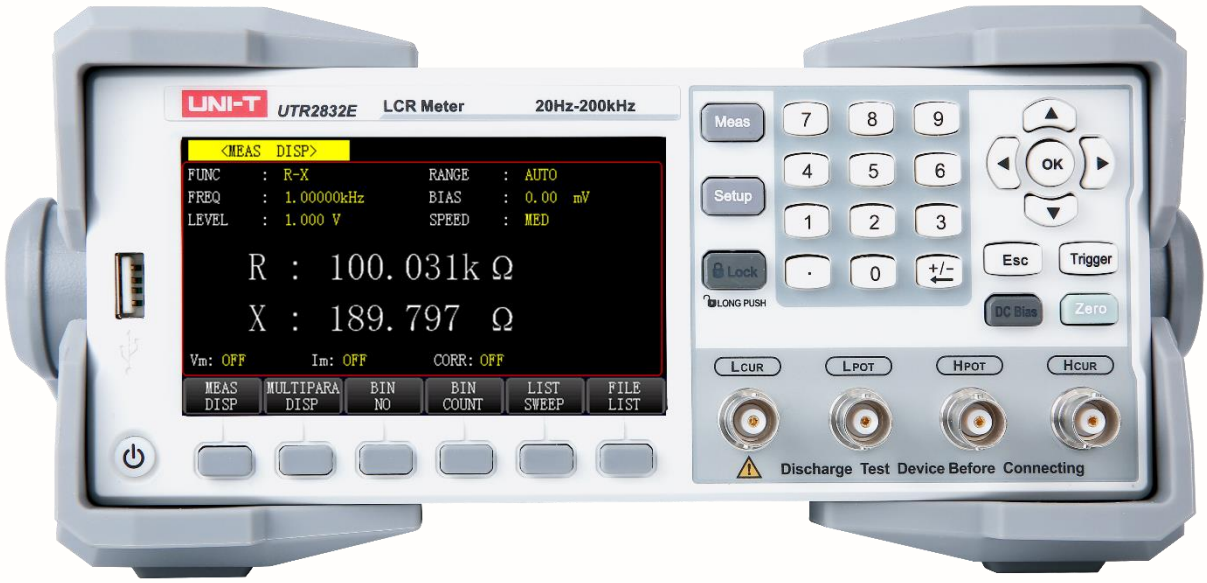
UNI-T®

Instruments.uni-trend.com



Programming Manual

UTR2830 Series LCR Meter

V1.0

2024.5.30

Warranty and Statement

Copyright

Brandmark Information

UNI-T is the registered trademark of Uni-Trend Technology (China) Co., Ltd.

Statement

- UNI-T products are protected by patents (including obtained and pending) in China and other countries and regions.
- UNI-T reserves the right to change specifications and prices.
- The information provided in this manual supersedes all previous publications.
- The information provided in this manual is subject to change without notice.
- UNI-T shall not be liable for any errors that may be contained in this manual. For any incidental or consequential damages arising out of the use or the information and deductive functions provided in this manual.
- No part of this manual shall be photocopied, reproduced or adapted without the prior written permission of UNI-T.

Product Certification

UNI-T has certified that the product conforms to China national product standard and industry product standard as well as ISO9001:2008 standard and ISO14001:2004 standard. UNI-T will go further to certificate product to meet the standard of other member of the international standards organization.

1. SCPI

SCPI (Standard Commands for Programmable Instruments) is a standardized instrument programming language that builds on existing standards IEEE 488.1 and IEEE 488.2 and follows the floating point rules of IEEE 754 standard, ISO 646 message exchange 7-bit encoding notation (equivalent to ASCII programming) and many other standards.

This section introduces the format, symbols, parameters, and abbreviations of the SCPI command.

1.1 Command String Parse

The host computer can send a string of commands to the instrument and the command parser of the instrument starts to parsing after catching the terminator (\n) or an input buffer overflow.
For example
Valid command string:
AAA:BBB CCC;DDD EEE;:FFF
The instrument command parser is responsible for all command parsing and execution, and you must understand its parsing rules before writing a program.

1.2 Command Parse Rule

Command parser only parses and responds to ASCII data.
The command parser starts to executing when receive the end mark. The instrument receives the following contents as end mark.
CR+LF
The command parser will terminate the parsing immediately after parsing an error, and the current command will be invalidated.
The command parser is case-insensitive for parsing command strings.
The command parser supports abbreviated form of command and the detailed see the following section.
Add ADDR☐Local address::☐ in front of RS485 mode, SCPI protocol, the local address can set to 1-32.
It is convenient for communicating with multiple machines via SCPI protocol.
For Example: ADDR☐1::☐IDN? ☐ represents a space
The instrument sends
The end mark of data send by the instrument
The default data end mark sent by the instrument is 0x0D 0x0A (CRLF).
Use semicolon ";" can send multiple commands.

1.3 Symbol Stipulation and Definition

This chapter uses some symbols that are not part of the command tree, but only for a better understanding of the command string.
<> The text in angle brackets indicates the parameter of the command.
[ ] The text in square brackets indicates the optional command.
{} When the curly brackets contain several parameter items, it means that only one item can be selected from them.
( ) The abbreviated form of the parameter is enclosed in parentheses.
Capital letter Abbreviated form of the command.
☐ Space character It represents a space and only for reading.

1.4 Command Tree Structure

SCPI commands have a tree-like structure with three level (Note: the command parser of this instrument can parse any level), where the highest level is called the subsystem command. SCPI uses a colon (:) to separate high level commands from low level commands.

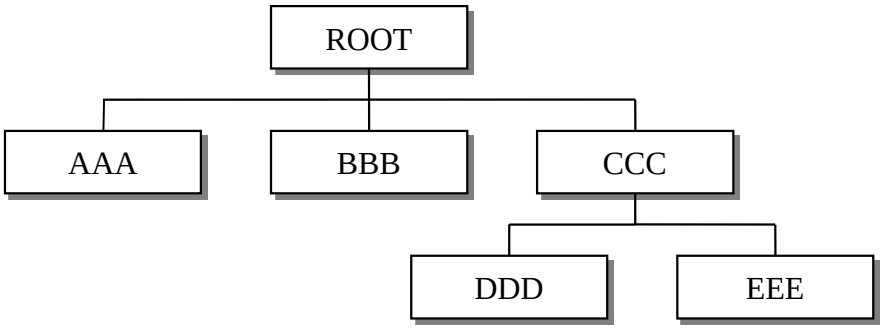


Figure 1-1 Command Tree Structure

For Example

ROOT:CCC:DDD ppp

ROOT Subsystem command

CCC Second level

DDD Third level

ppp Parameter

1.5 Command and Parameter

A command tree is consist of command and [parameter], use a blank to separate (ASCII: 20H).

For example AAA:BBB 1.234

Command [parameter]

1.6 Command

Command words can be in long command format or in abbreviated form. Long format facilitates engineers to better understand the meaning of the command string; abbreviated form is suitable for writing.

1.7 Parameter

Single character command, no parameter

For Example AAA:BBB

Parameter can be string format and its abbreviated form is also follow the last section " command abbreviated rule"

For example AAA:BBB 1.23

Parameter can be numerical value format

<integer> integer 123, +123, -123

<float> floating point number

1. <fixfloat>: fixed floating point number: 1.23, -1.23
2. <Sciloat>: floating point number represented by scientific notation: 1.23E+4, +1.23e-4
3. <mpfloat>: floating point number represented by multiplying power: 1.23k, 1.23M, 1.23G, 1.23u

Table 0-1 Abbreviation of Multiplying Power

Numerical Value	Multiplying Power
1E18(EXA)	EX
1E15(PETA)	PE
1E12(TERA)	T
1E9(GIGA)	G
1E6(MEGA)	MA
1E3(KILO)	K
1E-3(MILLI)	M
1E-6(MICRO)	U
1E-9(NANO)	N
1E-12(PICO)	P
1E-15(PEMTO)	F
1E-18(ATTO)	A

Note: The multiplying power is case-insensitive, and the writing is differently from the standard name.

1.8 Separator

The instrument command parser can only receive allowable separator. Other separator will cause error "Invalid separator".

The allowable separator is as follows.

; Semicolon is for separating two commands.

For Example: AAA:BBB 100.0 ; CCC:DDD

: Colon is for separating command tree or restart the command tree.

For Example: AAA : BBB : CCC 123.4; : DDD : EEE 567.8

? Question mark is for querying.

For Example: AAA ?

☐ **Blank** is for separating the parameter.

For Example: AAA:BBB☐1.234

2. SCPI Command Reference

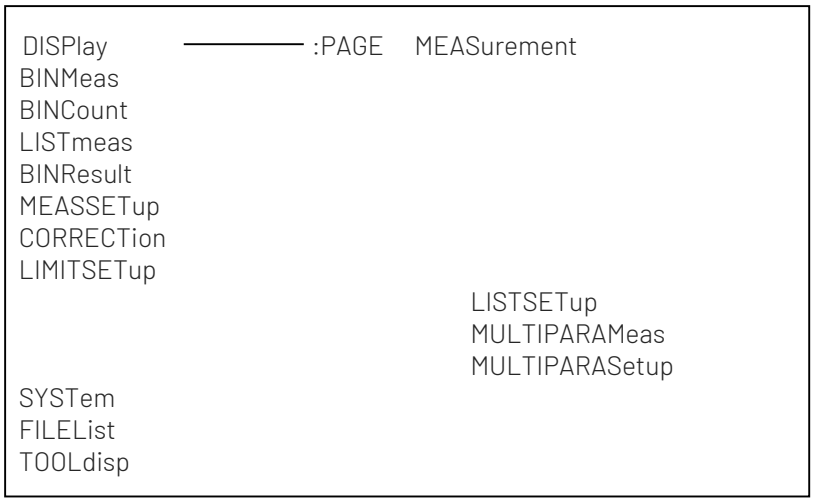
The data convention in this manual is as follows.
NR1: integer, for example: 123
NR2: fixed-point number, for example: 12.3
NR3: floating-point number, for example: 12.3E+5
NL: carriage return, integer 10

2.1 Subsystem Command

2.1.1 DISPlay Subsystem Command Set

DISPlay subsystem command set is used to set the display page of the instrument, and the character ? is used to query the current page.

Command tree



:PAGE command is used to set the display page of the instrument, and the character ? is used to query the current page.

Command syntax: DISPlay:PAGE <page name>

<page name> is as follows.

MEASurement	Set the display page to: Measurement
BINMeas	Set the display page to: Bin
BINCount	Set the display page to: BIN count
LISTmeas	Set the display page to: List sweep
BINResult	Set the display page to: BIN result
MEASSETup	Set the display page to: Measurement setup
CORRECTion	Set the display page to: Correction
LIMITSETup	Set the display page to: Limit list setup
LISTSETup	Set the display page to: List sweep setup
MULTIPARAMeas	Set the display page to: Multi-parameter
MULTIPARASetup	Set the display page to: Multi-parameter setup
SYSTem	Set the display page to: System
FILEList	Set the display page to: File list
TOOLdisp	Set the display page to: Tool

For example: WrtCmd ("DISP:PAGE MEAS"); Set the display page to: Measurement

Query syntax: DISPlay:PAGE?
Query return: <page name><NL>

<page name> is as follows.

<MEASurement>	indicates that the current page is: Measurement
<BINMeas>	indicates that the current page is: BIN
<BINCount>	indicates that the current page is: BIN count
<LISTmeas>	indicates that the current page is: List sweep
<BINResult>	indicates that the current page is: BIN result
<MEASSETup>	indicates that the current page is: Measurement setup
<CORRECTion>	indicates that the current page is: Correction
<LIMITSETup>	indicates that the current page is: Limit list setup
<LISTSETup>	indicates that the current page is: List sweep setup
<MULTIPARAMeas>	indicates that the current page is: Multi-parameter
<MULTIPARASetup>	indicates that the current page is: Multi-parameter setup
<SYSTem>	indicates that the current page is: System
<FILEList>	indicates that the current page is: File list
< TOOLdisp >	indicates that the current page is: Tool

2.1.2 FREQuency Subsystem Command Set

FREQuency subsystem command set is used to set the measuring frequency of the instrument, and the character ? is used to query the current measuring frequency.

Command syntax:

FREQuency { <value> | MIN | MAX }

The details are as follows.

<value>	The parameter can be the data format of NR1, NR2 or NR3 with the suffix of Hz, kHz, MHz.
MIN	Set the measuring frequency to 20 Hz
MAX	Set the measuring frequency to 100 kHz (the maximum of UTR2830 is 100 kHz, the maximum of UTR2832 is 200 kHz)

For example: WrtCmd ("FREQ 1KHZ"); Set the measuring frequency to 1000 Hz.

Query syntax: FREQuency?

Query return: <NR3><NL>

2.1.3 VOLTage Subsystem Command Set

VOLTage subsystem command set is used to set the measuring voltage of the instrument, and the character ? is used to query the current measuring voltage.

Command syntax:

VOLTage { <value> | MIN | MAX }

The details are as follows.

<value>	The parameter can be the data format of NR1, NR2 or NR3 with the suffix of V.
MIN	Set the measuring voltage to 10 mV
MAX	Set the measuring voltage to 2 V

For example: WrtCmd ("VOLT 1V"); Set the measuring voltage to 1V.

Query syntax: VOLTage?

Query return: <NR3><NL>

2.1.4 CURRent Subsystem Command Set

CURRent subsystem command set is used to set the measuring current of the instrument, and the character ? is used to query the current measuring current.

Command syntax:
CURRent { <value> | MIN | MAX }

The details are as follows.

<value>	The parameter can be the data format of NR1, NR2 or NR3 with the suffix of mA.
MIN	Set the measuring current to 100μA
MAX	Set the measuring current to 20 mA

For example: WrtCmd ("CURR 10mA"); Set the measuring current to 10 mA.

Query syntax: CURRent ?
Query return: <NR3><NL>

2.1.5 AMPLitude Subsystem Command Set

AMPLitude subsystem command set is used to set the automatic level control (ALC) switch of the instrument, and the character ? is used to query the current state of the automatic level control (ALC) switch.

Command syntax:
AMPLitude:ALC { ON | OFF | 1 | 0 }

In here,
Character 1 (integer 49) is equal to ON.
Character 0 (integer 48) is equal to OFF.
For example: WrtCmd ("AMPL:ALC 0"); Set the automatic level control (ALC) switch to OFF.

Query syntax: AMPLitude:ALC?
Query return: <NR1><NL>
(**Note: UTR2832E has this command function, and it is not available for UTR2830E.**)

2.1.6 Output RESister Subsystem Command Set

Output RESister subsystem command set is used to set the output resistance mode of the instrument, and the character ? is used to query the current output resistance.

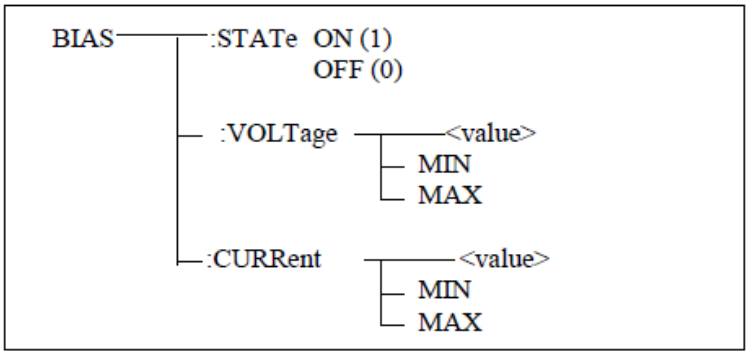
Command syntax:
ORESister { 30 | 50 | 100 }

For example: WrtCmd ("ORES 30"); Set the output resistance to 30 OHM

Query syntax: ORESister?
Query return: <NR1><NL>

2.1.7 BIAS Subsystem Command Set

BIAS subsystem command set is used to set the bias voltage switch of the instrument (**only for UTR2832E**).
Command tree



:STATe command is used to set the bias voltage switch of the instrument, and the character ? is used to query the current state of bias voltage switch.

Command syntax:

BIAS:STATe { ON | OFF | 1 | 0 }

In here,
Character 1 (integer 49) is equal to ON.
Character 0 (integer 48) is equal to OFF.
For example: WrtCmd ("BIAS:STATe 0"); Set the DC bias to OFF.

Query syntax: BIAS:STATe?
Query return: <NR1><NL>

:VOLTage command is used to set the bias voltage of the instrument, and the character ? is used to query the current bias voltage.

Command syntax:

BIAS:VOLTage { <value> | MIN | MAX }

The details are as follows.

<value>	The parameter can be the data format of NR1, NR2,NR3.
MIN	Set the measuring bias voltage to 0 V.
MAX	Set the measuring bias voltage to 5 V.

For example: WrtCmd ("BIAS:VOLT MIN"); Set the DC bias voltage to 0 V.

Query syntax: BIAS:VOLTage?
Query return: <NR3><NL>

:CURRent command is used to set the bias current of the instrument, and the character ? is used to query the current bias current.
Command syntax:

BIAS: CURRent { <value> | MIN | MAX }

The details are as follows.

<value> The parameter can be the data format of NR1, NR2,NR3.

MIN Set the measuring bias current to 0 A.

MAX Set the measuring bias current to 50 mA.

For example: WrtCmd ("BIAS:CURR MIN"); Set the DC bias current to 0 A.

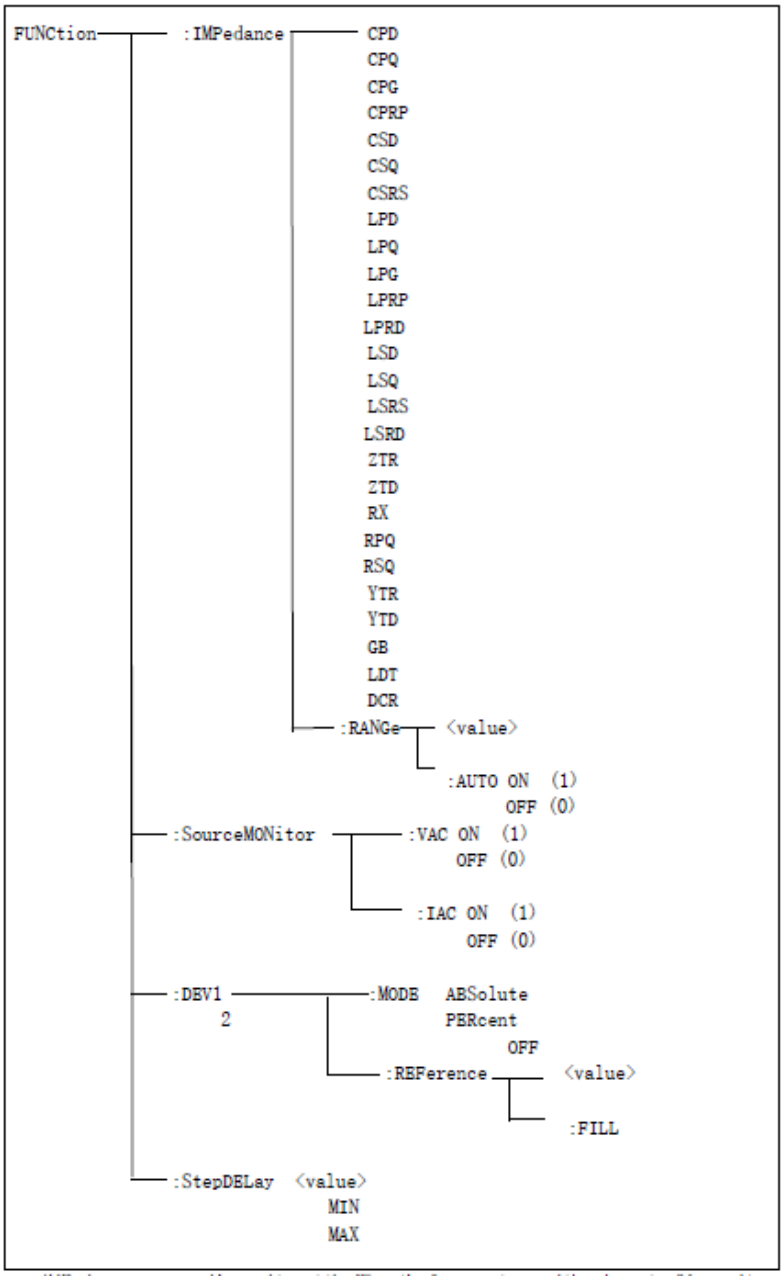
Query syntax: BIAS: CURRent?

Query return: <NR3><NL>

Note: When the internal resistance is 30 Ω, the bias voltage range is ±1.5 V， and the bias current is ±50 mA; when the internal resistance is 50 Ω, the bias voltage range is ±2.5 V， and the bias current is ±50 mA; when the internal resistance is 100 Ω, the bias voltage range is ±5 V， and the bias current±50 mA.

2.1.8 FUNCtion Subsystem Command Set

FUNCtion subsystem command set is used to set the measuring function of range, current/voltage monitor, deviation measurement and nominal value.



:IMPedance command is used to set the "Function" parameter, and the character ? is used to query the current "Function" parameter.

Command syntax: FUNCtion:IMPedance <function>

The details are as follows.

CPD	Set "Function" to Cp-D	LPRP	Set "Function" to Lp-Rp
CPQ	Set "Function" to Cp-Q	LSD	Set "Function" to Ls-D
CPG	Set "Function" to Cp-G	LSQ	Set "Function" to Ls-Q
CPRP	Set "Function" to Cp-Rp	LSRS	Set "Function" to Ls-Rs
CSD	Set "Function" to Cs-D	RX	Set "Function" to R-X
CSQ	Set "Function" to Cs-Q	ZTD	Set "Function" to Z-θ°
CSRS	Set "Function" to Cs-Rs	ZTR	Set "Function" to Z-θr
LPD	Set "Function" to Lp-D	GB	Set "Function" to G-B
LPQ	Set "Function" to Lp-Q	YTD	Set "Function" to Y-θ°
LPG	Set "Function" to Lp-G	YTR	Set "Function" to Y-θr
LPRD	Set "Function" to Lp-Rd	RPQ	Set "Function" to Rp-Q
LSRD	Set "Function" to Ls-Rd	RSQ	Set "Function" to Rs-Q
DCR	Set "Function" to DCR	LDT	Set "Function" to LDT

For example: WrtCmd ("FUNC:IMP RX"); set the "Function" parameter to R-X.

Query syntax: FUNCtion:IMPedance?

Query return: <function><NL>

:IMPedance:RANGe command is used to set the range of LCR measurement, and the character ? is used to query the current range.

Command syntax: FUNCtion:IMPedance:RANGe <value>

In here,

<value> The parameter can be impedance range of DUT, and the data format can be NR1, NR2,NR3.

For example: WrtCmd ("FUNC:IMP:RANG 1000"); Set the range of the instrument to 1 kOHM.

Query syntax: FUNCtion:IMPedance:RANGe?

Query return: <value><NL>

In here,

<value> can be set to:

3	10	30
100	300	1000 3000
10000	30000	100000

:IMPedance:RANGe:AUTO command is used to set auto range of the instrument, and the character ? is used to query the current auto range.

Command syntax:

FUNCtion:IMPedance:RANGe:AUTO { ON (1) / OFF (0) }

In here,

Character 1 (integer 49) is equal to ON.

Character 0 (integer 48) is equal to OFF.

For example: WrtCmd ("FUNC:IMP:RANG:AUTO ON"); Set the auto range for the instrument.

Query syntax: FUNCtion:IMPedance:RANGe:AUTO?

Query return: <NR1><NL>

:Source MONitor:VIAC command is used to set the current/voltage monitor of the instrument, and the character ? is used to query the current state of current/voltage monitor.

Command syntax:

FUNCtion:SMONitor:VIAC { ON (1) / OFF (0) }

In here,

Character 1 (integer 49) is equal to ON.

Character 0 (integer 48) is equal to OFF.

For example: WrtCmd ("FUNC:SMON:VIAC ON"); Set the current/voltage monitor of the instrument to "ON".

Query syntax: FUNCtion:SMONitor:VIAC?
Query return: <NR1><NL>

: DEV<n>:MODE command is used to set the deviation measurement of the instrument, and the character ? is used to query the current state of deviation measurement.

Command syntax:

FUNCtion:DEV<n>:MODE { ABSolute | PERCent | OFF }

In here,
ABSolute : absolute deviation
PERCent : percentage deviation
OFF: measured value
In here, <n>:
Character 1 (integer 49) Set the deviation mode for the primary parameter
Character 2 (integer 50) Set the deviation mode for the secondary parameter
For example: WrtCmd ("FUNC:DEV1:MODE ABS");

Query syntax: FUNCtion:DEV<n>:MODE?

Query return: { ABS | PERC | OFF } <NL>

:DEV<n>:REFerence command is used to set the deviation nominal value of the instrument, and the character ? is used to query the current deviation nominal value.
Command syntax: FUNCtion:DEV<n>:REFerence<value>
In here,
<value> The parameter can be the data format of NR1, NR2,NR3.
<n>:
Character 1 (integer 49) Set the deviation nominal value for the primary parameter
Character 2 (integer 50) Set the deviation nominal value for the secondary parameter
For example: WrtCmd ("FUNC:DEV1:REF 10");

Query syntax: FUNCtion:DEV<n>:REFerence?
Query return: <NR3><NL>

:DEV<n>:REFerence:FILL command is used to set the deviation nominal value of the instrument, and control the instrument to measure once, and then replicate the primary and secondary parameter results into a deviation nominal value.
Command syntax: FUNCtion:DEV<n>:REFerence:FILL
In here,
<n>:
Both Character 1 (integer 49) and haracter 2 (integer 50) can be used to set the deviation nominal value for the primary and secondary parameter.
For example: WrtCmd ("FUNC:DEV1:REF:FILL");

:StepDELay command is used to set the step delay of the instrument, and the character ? is used to query the current step delay.

Command syntax:

StepDELay { <value> | MIN | MAX }

The details are as follows.

<value>	The parameter can be the data format of NR1, NR2 or NR3, and 0—60 seconds with 1 ms resolution.
MIN	Set the delay to 0 second
MAX	Set the delay to 60 seconds

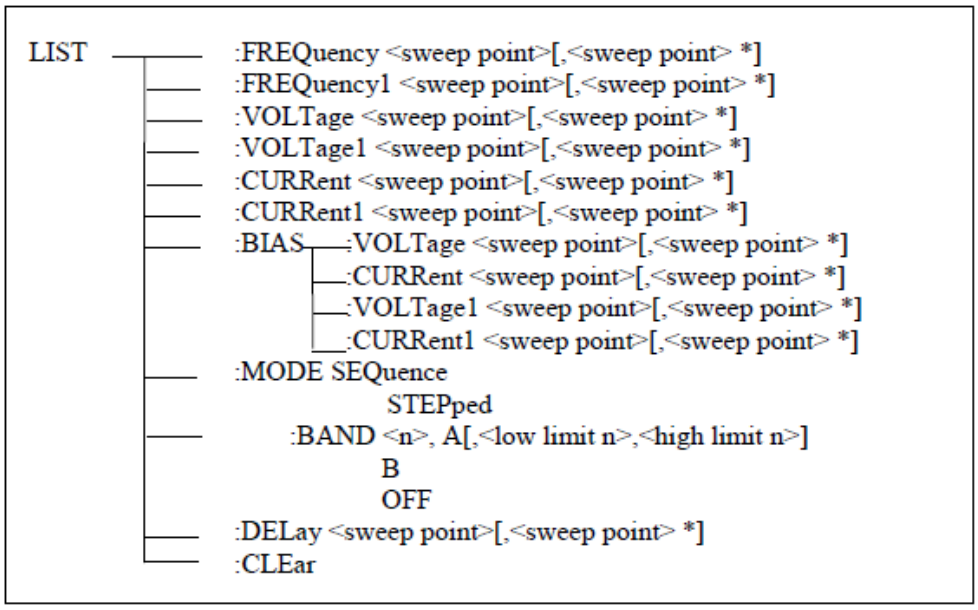
For example: WrtCmd ("FUNC:SDEL 5000"); Set the step delay to 5 seconds.

Query syntax: FUNC:SDEL?
Query return: <NR3><NL>

2.1.9 LIST Subsystem Command Set

LIST subsystem command set is used to set the list sweep function of sweep point, sweep mode and comparator.

Command tree



:FREQuency command is used to clear the sweep point in List A and set the frequency of the sweep point, and can query the current frequency of each sweep points.

Command syntax: LIST:FREQuency <value>,<value>*

Note: * indicates that a maximum of 201 sweep points.

In here,

<value>	The parameter can be the data format of NR1, NR2 or NR3.
<value>	**The parameter value should be within the range of 20-200000 (UTR2832), otherwise, the query returns an error.**

For example: WrtCmd ("LIST:FREQ 2E2,1E3,5E3,1E4")

Set the sweep point 1 in List A to 200 HZ;

Set the sweep point 2 in List A to 1 KHZ;

Set the sweep point 3in List A to 5 KHZ;

Set the sweep point 4 in List A to 10 KHZ.

Query syntax: LIST:FREQuency?
Query return: <NR3>,<NR1>*<NL>

:FREQuency1 command is used to clear the sweep point in List B and set the frequency of the sweep point, and can query the current frequency of each sweep points.

Command syntax: LIST:FREQuency1 <value>,<value>*

Note: * indicates that a maximum of 201 sweep points.

In here,
<value> The parameter can be the data format of NR1, NR2 or NR3.
<value> The parameter value should be within the range of 20-200000 (UTR2832), otherwise, the query returns an error.

For example: WrtCmd ("LIST:FREQ1 2E2,1E3,5E3,1E4")
Set the sweep point 1 in List B to 200 HZ.
Set the sweep point 2 in List B to 1 kHZ.
Set the sweep point 3in List B to 5 kHZ.
Set the sweep point 4 in List B to 10 kHZ.
Query syntax: LIST:FREQuency1?
Query return: <NR3>,<NR1>*<NL>
Note: List A and List B cannot be the same sweep parameter.

: VOLTage command is used to clear the sweep point in List A and reset the measuring voltage of the sweep point, and can query the current voltage of each sweep points.
Command syntax: LIST:VOLTage <value>,<value>*
Note: * indicates that a maximum of 201 sweep points.
In here,
<value> The parameter can be the data format of NR1, NR2 or NR3.
For example: WrtCmd ("LIST:VOLT 0.5,1.0,2.0")
Set the sweep point 1 in List A to 0.5 V.
Set the sweep point 2 in List A to 1.0 V.
Set the sweep point 3 in List A to 2.0 V.
Query syntax: LIST:VOLTage?
Query return: <NR3>,<NR1>*<NL>
Note: The voltage of each sweep points should be within the range of 10 mV-2 V, otherwise, the query returns an error.

: VOLTage1 command is used to clear the sweep point in List B and reset the measuring voltage of the sweep point, and can query the current voltage of each sweep points.
Command syntax: LIST:VOLTage1 <value>,<value>*
Note: * indicates that a maximum of 201 sweep points.
In here,
<value> The parameter can be the data format of NR1, NR2 or NR3.
For example: WrtCmd ("LIST:VOLT1 0.5,1.0,2.0")
Set the sweep point 1 in List B to 0.5 V.
Set the sweep point 2 in List B to 1.0 V.
Set the sweep point 3 in List B to 2.0 V.
Query syntax: LIST:VOLTage1?
Query return: <NR3>,<NR1>*<NL>
Note: The voltage of each sweep points should be within the range of 10 mV-2 V, otherwise, the query returns an error.
Note: List A and List B cannot be the same sweep parameter.

:CURRent command is used to clear the sweep point in List A and reset the measuring current of the sweep point, and can query the current of each sweep points.
Command syntax: LIST:CURRent <value>,<value>*
Note: * indicates that a maximum of 201 sweep points.
In here,
<value> The parameter can be the data format of NR1, NR2 or NR3.
For example: WrtCmd ("LIST:CURR 1E-3,1E-2,2E-2")
Set the sweep point 1 in List A to 1 mA.
Set the sweep point 2 in List A to 10 mA.
Set the sweep point 3 in List A to 20 mA.
Query syntax: LIST:CURRent?
Query return: <NR3>,<NR1>*<NL>
Note: When the internal resistance is 100 Ω, the measuring current of sweep point should be within the range of 100 uA-20 mA; when the internal resistance is 50 Ω, the measuring current of sweep point should be within the range of 200 uA-40 mA; when the internal resistance is 30 Ω, the measuring current of sweep point should be within the range of 333 uA-66.7 mA. Otherwise, the query returns an error.

:CURRent1 command is used to clear the sweep point in List B and reset the measuring current of the sweep point, and can query the current of each sweep points.
Command syntax: LIST:CURRent1 <value>,<value>*
Note: * indicates that a maximum of 201 sweep points.
In here,
<value> The parameter can be the data format of NR1, NR2 or NR3.
For example: WrtCmd ("LIST:CURR1 1E-3,1E-2,2E-2")
Set the sweep point 1 in List B to 1 mA.
Set the sweep point 2 in List B to 10 mA.
Set the sweep point 3 in List B to 20 mA.
Query syntax: LIST:CURRent1?
Query return: <NR3>,<NR1>*<NL>
Note: When the internal resistance is 100 Ω, the measuring current of sweep point should be within the range of 100 uA-20 mA; when the internal resistance is 50 Ω, the measuring current of sweep point should be within the range of 200 uA-40 mA; when the internal resistance is 30 Ω, the measuring current of sweep point should be within the range of 333 uA-66.7 mA. Otherwise, the query returns an error.
Note: List A and List B cannot be the same sweep parameter.

: BIAS:VOLTage command is used to clear the sweep point in List A and reset the bias voltage of the sweep point, and can query the bias voltage of each sweep points.
Command syntax: LIST:BIAS:VOLTage <value>,<value>*
Note: * indicates that a maximum of 201 sweep points.
In here,
<value> The parameter can be the data format of NR1, NR2 or NR3.
For example: WrtCmd ("LIST:BIAS:VOLT -1.0,1.0,2.0")
Set the DC bias voltage of sweep point 1 in List A to -1.0 V.
Set the DC bias voltage of sweep point 2 in List A to 1.0 V.
Set the DC bias voltage of sweep point 3 in List A to 2.0 V.
Query syntax: LIST:BIAS:VOLTage?
Query return: <NR3>,<NR1>*<NL>
Note: Only UTR2832E has a bias voltage, and it is not available for UTR2830E.

: BIAS:VOLTage1 command is used to clear the sweep point in List B and reset the bias voltage of the sweep point, and can query the bias voltage of each sweep points.
Command syntax: LIST:BIAS:VOLTage1 <value>,<value>*
Note: * indicates that a maximum of 201 sweep points.
In here,
<value> The parameter can be the data format of NR1, NR2 or NR3.
For example: WrtCmd ("LIST:BIAS:VOLT1 -1.0,1.0,2.0")
Set the DC bias voltage of sweep point 1 in List B to -1.0 V.
Set the DC bias voltage of sweep point 2 in List B to 1.0 V.
Set the DC bias voltage of sweep point 3 in List B to 2.0 V.
Query syntax: LIST:BIAS:VOLTage1?
Query return: <NR3>,<NR1>*<NL>
Note: Only UTR2832E has a bias voltage, and it is not available for UTR2830E.
Note: List A and List B cannot be the same sweep parameter.

:BIAS:CURRent command is used to clear the sweep point in List A and reset the bias current of the sweep point, and can query the bias current of each sweep points.
Command syntax: LIST:BIAS:CURRent <value>,<value>*
Note: * indicates that a maximum of 201 sweep points.
In here,
<value> The parameter can be the data format of NR1, NR2 or NR3.
For example: WrtCmd ("LIST:BIAS:CURR 1E-4,1E-3,1E-2")
Set the DC bias current of sweep point 1 in List A to 100 uA.
Set the DC bias current of sweep point 2 in List A to 1 mA.
Set the DC bias current of sweep point 3 in List A to 10 mA.
Query syntax: LIST:BIAS:CURRent?

Query return: <NR3>,<NR1>*<NL>
Note: Only UTR2832E has a bias voltage, and it is not available for UTR2830E.

:BIAS:CURRent1 command is used to clear the sweep point in List B and reset the bias current of the sweep point, and can query the bias current of each sweep points.
Command syntax: LIST:BIAS:CURRent1 <value>,<value>*
Note: * indicates that a maximum of 201 sweep points.
In here,
<value> The parameter can be the data format of NR1, NR2 or NR3.
For example: WrtCmd ("LIST:BIAS:CURR1 1E-4,1E-3,1E-2")
Set the DC bias current of sweep point 1 in List B to 100 uA.
Set the DC bias current of sweep point 2 in List B to 1 mA.
Set the DC bias current of sweep point 3 in List B to 10 mA.
Query syntax: LIST:BIAS:CURRent1?
Query return: <NR3>,<NR1>*<NL>
Note: Only UTR2832E has a bias voltage, and it is not available for UTR2830E.
Note: List A and List B cannot be the same sweep parameter.

:MODE command is used to set the sweep mode of the instrument, and can query the current sweep mode .

Command syntax: LIST:MODE { SEQuence | STEPped }

In here,
SEQuence: continuous mode
STEPped: single step mode
For example: WrtCmd ("LIST:MODE SEQ")

Query syntax: LIST: MODE?

Query return: { SEQ | STEP } < NL>

:BAND<n> command is used to set the limit data in sweep list, and can query the current limit data.
Command syntax: LIST:BAND <n>,<parameter>,<low limit n>,<high limit n>
In here,
<n>: 1~201(NR1) : the sweep point in Line n
<parameter>:
A: The measured result is compared with the upper/lower limit of primary parameter.
B: The measured result is compared with the upper/lower limit of secondary parameter.
OFF: not compare
<low limit n> : Data format of NR1, NR2 or NR3, the lower limit data of the sweep point in Line n.
<high limit n> : Data format of NR1, NR2 or NR3, the upper limit data of the sweep point in Line n.
For example:
WrtCmd ("LIST:BAND 1,A,10,20") Set the sweep point 1 to perform the primary parameter comparison, the lower limit is 10, the upper limit is 20.
WrtCmd ("LIST:BAND 2,B,30,40") Set the sweep point 2 to perform the secondary parameter comparison, the lower limit is 30, the upper limit is 40.
WrtCmd ("LIST:BAND 3,OFF,0,0") Turn of the comparison function of sweep point 3.

Query syntax: LIST:BAND? <value>
In here,
<value>: the sweep point in which line.
Query return: <n>,<parameter>,<low limit n>,<high limit n>

:DELay command is used to set the delay of sweep list, and can query the current delay.
Command syntax: LIST:DELay <value>[,< value > *]
Note: * indicates that a maximum of 201 sweep points.

In here,
<value> The parameter can be the data format of NR1, NR2 or NR3.
For example: WrtCmd("LIST:DEL 0.001,0.1,1")
Set the delay of sweep point 1 to 1 ms.
Set the delay of sweep point 2 to 100 ms.
Set the delay of sweep point 3 to 1 s.
Query syntax: LIST:DELay?
Query return: <NR3>,<NR1>*<NL>
Note: * indicates that a maximum of 201 sweep points.

:CLEar command is used to clear all the data under the sweep list mode.
Command syntax:
LIST:CLEAR
For example: WrtCmd("LIST:CLEAR")

2.1.10 APERture Subsystem Command Set

APERture subsystem command set is used to set the test speed and average time, and the character ? is used to query the current test speed and average time.

Command syntax: APERture { FAST | MEDium | SLOW } [,<value>]

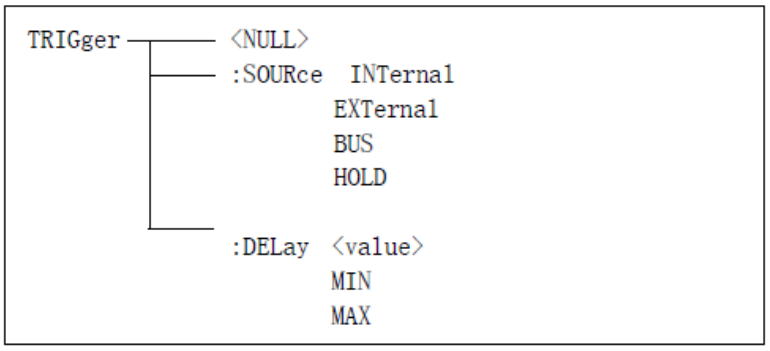
In here,
FAST: fast speed 75 time/s
MEDium: medium speed 11 time/s
SLOW: slow speed 2.7 time/s
<value>: 1 ~ 255 (NR1) average number
For example: WrtCmd("APER MED,55"); Set the test speed to medium, and set the average number to 55.

Query syntax: APERture?

Query return: { FAST | MED | SLOW },<NR1><NL>

2.1.11 TRIGger Subsystem Command Set

TRIGger subsystem command set is used to set the trigger source and trigger delay of the instrument.
Command tree



TRIGger command is used to trigger the instrument once.
Command syntax: TRIGger

For example: WrtCmd ("TRIG"); Start the instrument test once.

:SOURce command is used to set the trigger source mode of the instrument, and the character ? is used to query the current trigger source mode.

Command syntax:

TRIGger:SOURce { INTernal | EXTernal | BUS | HOLD }

In here,

INTernal : automatically trigger by the instrument, and it is the default setting.

EXTernal : triggered by HANDLER port

BUS: triggered by RS232 port

HOLD: triggered by pressing the TRIGGER key on the front panel

For example: WrtCmd ("TRIG:SOUR BUS"); Set the trigger source to BUS.

Query syntax: TRIGger:SOURce?

Query return: { INT | EXT | BUS | HOLD } <NL>

:DELay command is used to set the trigger delay, and the character ? is used to query the current the trigger delay.

Command syntax:

TRIGger:DELay { <value> | MIN | MAX }

The details are as follows.

<value> The parameter can be the data format of NR1, NR2 or NR3, and 0—60 seconds with 1 ms resolution.

MIN Set the delay to 0 second

MAX Set the delay to 60 seconds

For example: WrtCmd ("TRIG:DEL 5"); Set the delay to 5 ms.

WrtCmd ("TRIG:DEL 5000"); Set the delay to 5 s.

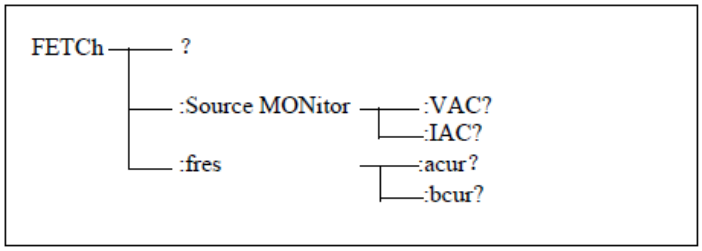
Query syntax: TRIGger:DELay?

Query return: <NR3><NL>

2.1.12 FETCh? Subsystem Command Set

FETCh? subsystem command set is used to set UTR2830 to output a measured result.

Command tree



FETCh? command is used to send the last measured result of UTR2830 to the output buffer area.

Query syntax: FETCh?

For example: WrtCmd ("FETCh?");

UTR2830E uses ASCII for result data transmission, the details are as follows.

ASCII data output format on the measurement, BIN and BIN No. display page is shown below.

SN.NNNNNESNN , SN.NNNNNESNN , SN or SNN , SN or SNN NL,
<DATA A> <DATA B> <STATE> <BIN NO.>

In here,

Format of <DATA A>,<DATA B>: <DATA A>(measured data of the primary parameter), <DATA B>(measured data of the secondary parameter)uses 12 bits

ASCII format: SN.NNNNNESNN (S: +/-, N: 0~9, E: Exponent Sign)

Format of <BIN No.>: BIN result is displayed in the following table.

Data	BIN result
0	BIN OUT
+1	BIN 1
+2	BIN 2
+3	BIN 3
+4	BIN 4
+5	BIN 5
+6	BIN 6
+7	BIN 7
+8	BIN 8
+9	BIN 9
+10	BIN AUX

<BIN No.> can only be displayed when the comparator is enabled (ON).
The data format of <BIN No.> uses ASCII 2 or 3 bits,
for example, SN or SNN (S: +/-, N: 0~9)

ASCII data output format on the list sweep page is displayed in Figure 6, data returned instead of the sweep point number.

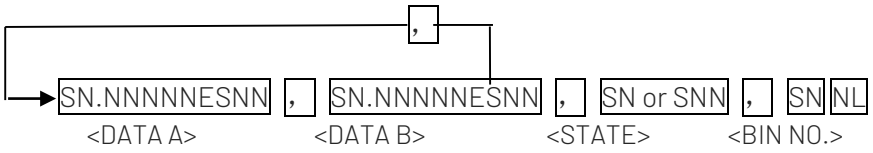


Figure 6 ASCII data format 2 (List sweep)

In here,
The data format of <DATA A>,<DATA B> and <STATE> is the same as described above.
The data format of <BIN NO.>:
The data format of <Input/Output>: the comparator result in the list sweep

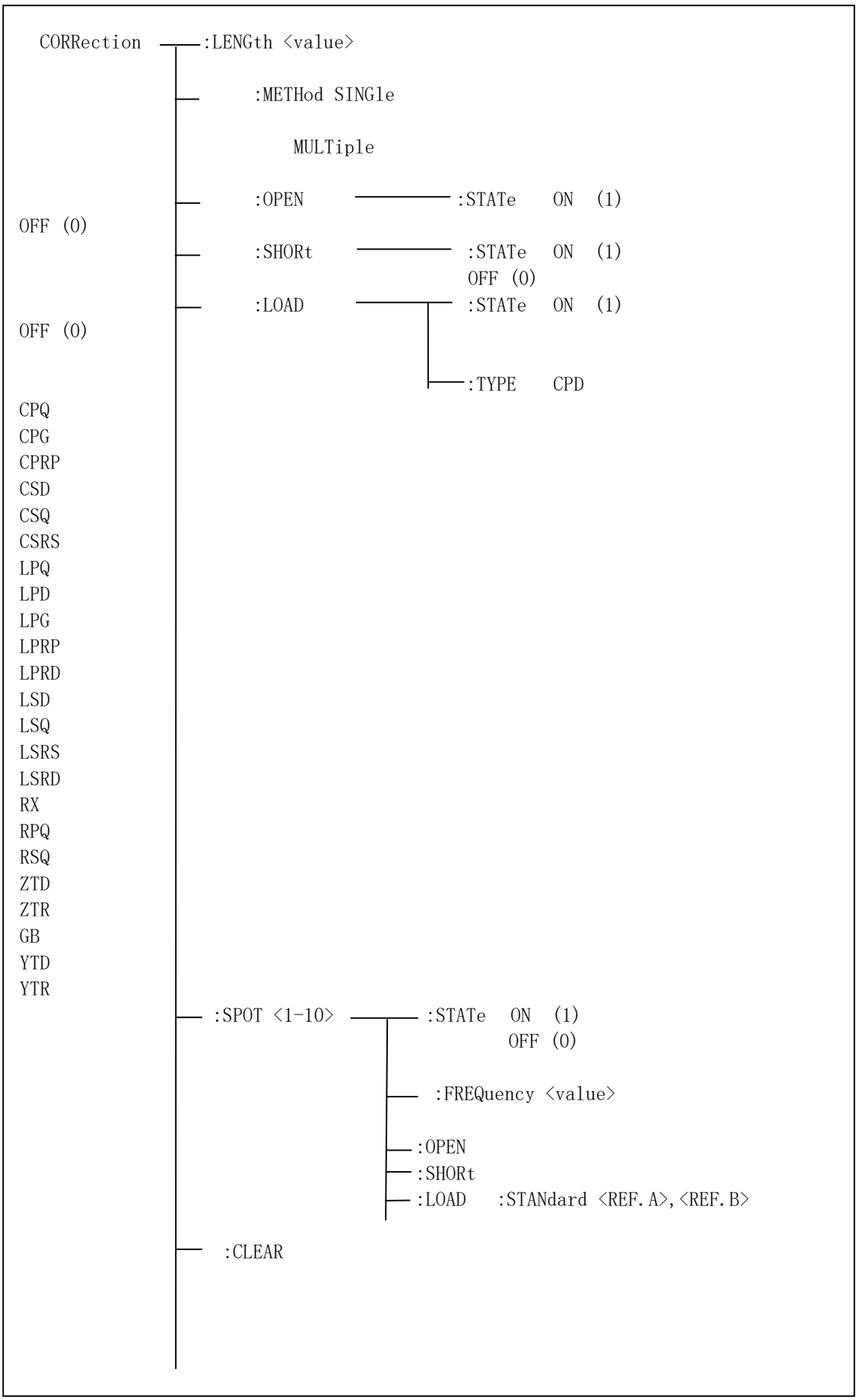
Data	Result
-1	Low
0	Pass
+1	High

When the comparator function is disabled (OFF), <Input/Output> data output result is 0.
<Input/Output> data output format uses ASCII 2 bits fixed length,
for example, SN (S: +/-, N: 0~1)

2.1.13 CORRection Subsystem Command Set

CORRection subsystem command set is used to set the correction, open-circuit correction, short-circuit correction and load correction.

Command tree



:LENGth command is used to set the cable length for instrument's correction, and the character ? is used to

query the current cable length.
Command syntax: CORRection:LENGth <value>
In here,
<value>: 0, 1, 2 or 4.
For example: WrtCmd("CORR:LENG 1") is used to set the cable length to 1 meter.

Query syntax: CORRection:LENGth?
Query return: <NR1><NL>

:METHod command is used to set the correction mode, and the character ? is used to query the current correction mode.
Command syntax: CORRection:METHod { SINGle | MULTi }

In here,
SINGle Set or return to the single channel mode
MULTi Set or return to multi-channels mode
For example: WrtCmd("CORR:METH MULT") is used to set the instrument to multi-channels mode (not supported for now).

Query syntax: CORRection:METHod?

Query return: { SINGle | <NL> | MULTi }

:OPEN command is used to execute the open-circuit correction for 41 preset test points (UTR2832).
Command syntax: CORRection:OPEN
For example: WrtCmd("CORR:OPEN")

:OPEN:STATe command is used to set the open-circuit correction, and the character ? is used to query the current state of the open-circuit correction.
Command syntax:

CORRection:OPEN:STATe { ON | OFF | 1 | 0 }

In here,
Character 1 (integer 49) indicates that the open-circuit correction is enabled, and it is equal to ON.
Character 0 (integer 48) indicates that the open-circuit correction is forbidden, and it is equal to OFF.
For example: WrtCmd("CORR:OPEN:STAT ON")

Query syntax: CORRection:OPEN:STATe?
Query return: <NR1><NL>

:SHORt command is used to set the short-circuit correction for 41 preset test points (UTR2832).
Command syntax: CORRection:SHORt
For example: WrtCmd("CORR:SHOR")

:SHORt:STATe command is used to set the short-circuit correction, and the character ? is used to query the current state of the short-circuit correction.

Command syntax:
CORRection:SHORt:STATe { ON | OFF | 1 | 0 }

In here,
Character 1 (integer 49) indicates that the short-circuit correction is enabled, and it is equal to ON.
Character 0 (integer 48) indicates that the short-circuit correction is forbidden, and it is equal to OFF.

For example: WrtCmd ("CORR:SHOR:STAT ON")

Query syntax: CORRection:SHORt:STATe?
Query return: <NR1><NL>

:LOAD:STATe command is used to set the load correction, and the character ? is used to query the current state of the load correction.

Command syntax:
CORRection:LOAD:STATe { ON | OFF | 1 | 0 }

In here,
Character 1 (integer 49) indicates that the load correction is enabled, and it is equal to ON.
Character 0 (integer 48) indicates that the load correction is forbidden, and it is equal to OFF.
For example: WrtCmd ("CORR: LOAD: STAT ON")

Query syntax: CORRection:LOAD:STATe?
Query return: <NR1><NL>

:LOAD:TYPE command is used to set the combined parameter for the load correction, and the character ? is used to query the type of combined parameter.

FuctionThe details are as follows.

CPD	Set "Function" to Cp-D	LPRP	Set "Function" to Lp-Rp
CPQ	Set "Function" to Cp-Q	LSD	Set "Function" to Ls-D
CPG	Set "Function" to Cp-G	LSQ	Set "Function" to Ls-Q
CPRP	Set "Function" to Cp-Rp	LSRS	Set "Function" to Ls-Rs
CSD	Set "Function" to Cs-D	RX	Set "Function" to R-X
CSQ	Set "Function" to Cs-Q	ZTD	Set "Function" to Z-θ°
CSRS	Set "Function" to Cs-Rs	ZTR	Set "Function" to Z-θr
LPQ	Set "Function" to Lp-Q	GB	Set "Function" to G-B
LPD	Set "Function" to Lp-D	YTD	Set "Function" to Y-θ°
LPG	Set "Function" to Lp-G	YTR	Set "Function" to Y-θr
LPRD	Set "Function" to Lp-Rd	RPQ	Set "Function" to Rp-Q
LSRD	Set "Function" to Ls-Rd	RSQ	Set "Function" to Rs-Q

For example: WrtCmd ("CORR:LOAD:TYPE CPD")

Query syntax: CORRection:LOAD:TYPE?
Query return: <function><NL>

:SPOT<n>:STATe command is used to set the specified frequency point, and the character ? is used to query thecurrent state of each frequency point (frequency point 1, frequency point 2).

Command syntax:
CORRection:SPOT:STATe <n>, { ON | OFF | 1 | 0 }

In here,
Character 1 (integer 49) is equal to ON.
Character 0 (integer 48) is equal to OFF.

<n>: 1-10
1 frequency point 1
2 frequency point 2
For example: WrtCmd ("CORR:SPOT:STAT 1,ON")

Query syntax: CORRection:SPOT:STATe? <n>
<n>: 1-10
Query return: <NR1><NL>

:SPOT:FREQuency<n> command is used to set the frequency of specified correction point, and the character ? is used to query the current frequency of specified correction point.
Command syntax: CORRection:SPOT:FREQuency <n>,<value>
In here,
<value> : data format of NR1，NR2 or NR3
<n>:1-10

For example: WrtCmd ("CORR:SPOT:FREQ 1,2E3") is used to set the frequency 1 to 2 kHZ.
Note: <value> should be within the range of 20~200000 (UTR2832), otherwise, the query returns an error.

Query syntax: CORRection:SPOT:FREQuency? <n>
<n>: 1-10
Query return: <NR3><NL>

:SPOT:OPEN<n> command is used to set the open-circuit correction for the specified correction point.
Command syntax: CORRection:SPOT:OPEN <n>
In here,
<n>:1-10
For example: WrtCmd ("CORR:SPOT:OPEN 1"); perform the open-circuit correction for correction point 1.

:SPOT:SHORt<n> command is used to set the short-circuit correction for the specified correction point.
Command syntax: CORRection:SPOT:SHORt <n>
In here,
<n>:1-10
For example: WrtCmd ("CORR:SPOT:SHOR 1") ; perform the short-circuit correction for correction point 1.

:SPOT:LOAD:STANdard<n> command is used to set the load correction under the standard reference of the specified correction point, and can query the current the load correction under the standard reference of the specified correction point.
Command syntax: CORRection:SPOT:LOAD:STANdard <n>,<REF.A>,<REF.B>
In here,
<n>:1-10
<REF.A>: data format of NR1，NR2 or NR3, it is the standard reference for the primary parameter.
<REF.B>: data format of NR1，NR2 or NR3, it is the standard reference for the secondary parameter.
For example: WrtCmd ("CORR:SPOT:LOAD:STAN 1,100.7,0.0002")
Query syntax: CORRection:SPOT:LOAD:STANdard? <n>
<n>:1-10
Query return: <NR3>,<NR3><NL>

:CLEAR command is used to clear all the correction data.
Command syntax: CORRection:CLEAR

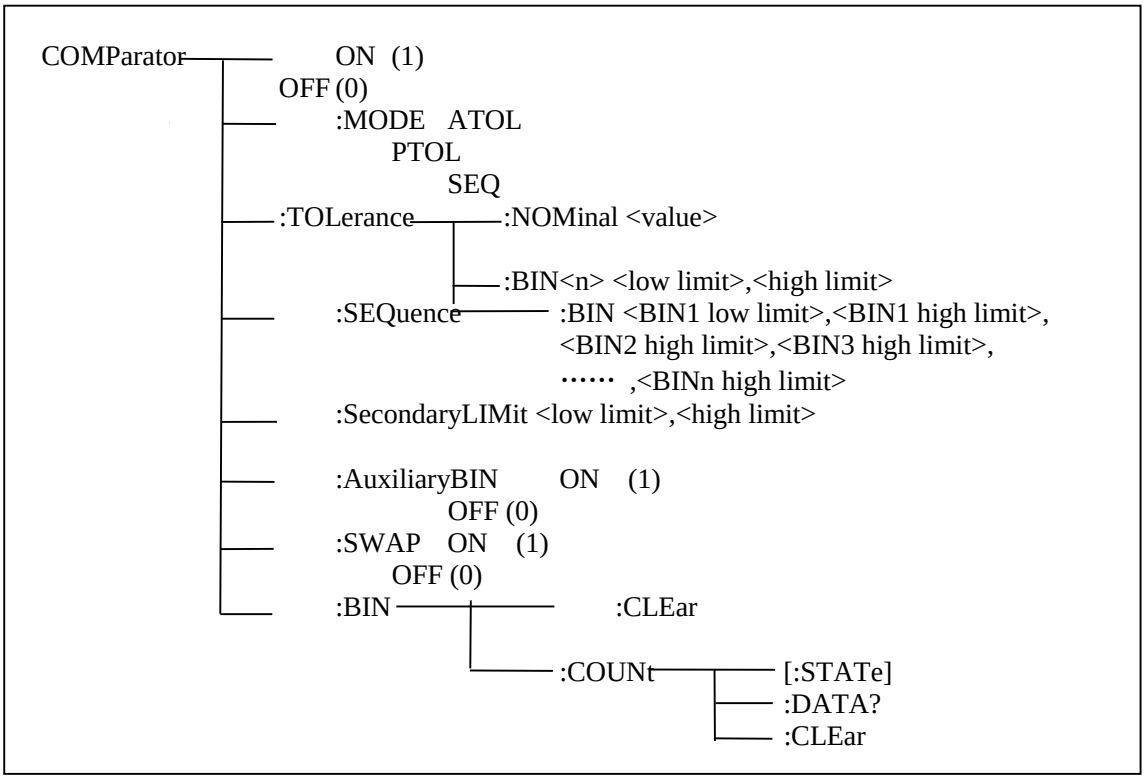
2.1.14 COMParator Subsystem Command Set

COMParator subsystem command set is used to set the comparator function, including the comparator switch and limit list setup.

Command tree

COMParator 子系统命令集用于设定档比较器功能，包括比较开关的设定，极限列表的设定。

命令树见下一页：



COMParator command is used to switch on/off the comparator function, and can query the current state of the comparator function.

Command syntax: COMParator { ON | OFF | 1 | 0 }

In here,

Character 1 (integer 49) is equal to ON.

Character 0 (integer 48) is equal to OFF.

For example: WrtCmd ("COMP ON"); switch on the comparator function of the primary parameter.

Query syntax: COMParator?

Query return: <NR1><NL>

:MODE command is used to set the comparator mode, and the character ? is used to query the current comparator mode.

Command syntax: COMParator:MODE { ATOLerance | PTOLerance | SEQuence }

In here,

ATOLerance: absolute error mode
PTOLerance: relative error mode
SEQuence: continuous mode
For example: WrtCmd("COMP:MODE ATOL")

Query syntax: COMParator:MODE?

Query return: { ATOL | PTOL | SEQ } <NL>

:TOLerance:NOMinal command is used to set the nominal value for the tolerance mode of comparator (this function is only available when the tolerance is enabled), and can query the nominal value of the tolerance mode.
Command syntax: COMParator:TOLerance:NOMinal <value>
In here,
<value> The nominal value can be the data format of NR1, NR2 or NR3.
For example: WrtCmd("COMP:TOL:NOM 100E-12") ; Set the nominal value to 100p.

Query syntax: COMParator:TOLerance:NOMinal?
Query return: <NR3><NL>

:TOLerance:BIN<n> command is used to set the upper/lower limit for each BIN under the tolerance mode of comparator (this function is only available when the tolerance mode is enabled), and can query the current upper/lower limit of each Bin.
Command syntax: COMParator:TOLerance:BIN <n>,<low limit>,<high limit>
In here,
<n>: 1~9 (NR1): BIN No.
<low limit>: data format of NR1, NR2 or NR3: lower limit data
<high limit> : data format of NR1, NR2 or NR3: upper limit data
Note: The lower limit data should be less than the upper limit data, otherwise, an error message will be prompted.
For example: WrtCmd("COMP:TOL:BIN 1,-5,5"); Set Limit 1: the lower limit is -5, and the upper limit is 5.
WrtCmd("COMP:TOL:BIN 2,-10,10") ; Set Limit 2: the lower limit is -10, and the upper limit is 10.

Query syntax: COMParator:TOLerance:BIN? <n>
Query return: <low limit>,<high limit><NL>

:SEQuence:BIN command is used to set the upper/lower limit for the continuous mode of comparator, (this function is only available when the continuous mode is enabled), and can query the current upper/lower limit of each Bin.
Command syntax: COMParator:SEQuence:BIN <BIN1 low limit>,<BIN1 high limit>,
<BIN2 high limit>,..., <BINn high limit>
In here,
<BIN1 low limit> data format of NR1, NR2 or NR3, it is the lower limit of BIN 1
<BIN1 high limit> data format of NR1, NR2 or NR3, it is the upper limit of BIN 1
<BINn high limit> data format of NR1, NR2 or NR3, it is the upper limit of BIN n (the maximum of n is 9)
Note: The lower limit should be less than the upper limit.
For example: WrtCmd("COMP:SEQ:BIN 10,20,30,40,50")
Set the lower limit of BIN 1 to 10, the upper limit of BIN 1 to 20
Set the upper limit of BIN 2 to 30
Set the upper limit of BIN 3 to 40
Set the upper limit of BIN 4 to 50

Query syntax: COMParator:SEQuence:BIN?
Query return: <BIN1 low limit>,<BIN1 high limit>,<BIN2 high limit>,...,
<BINn high limit><NL>

:Secondary LIMit command is used to set the upper/ lower limit for secondary parameter, and can query the

current upper/ lower limit of secondary parameter.
Command syntax: COMParator:SLIMit <low limit>,<high limit>
In here,
<low limit> data format of NR1, NR2 or NR3, it is the lower limit
<high limit> data format of NR1, NR2 or NR3, it is the upper limit
Note: The lower limit data should be less than the upper limit data, otherwise, an error message will be prompted.
For example: WrtCmd ("COMP:SLIM 0.001,0.002"); Set the lower limit of the secondary parameter to 0.001, and the upper limit of the secondary parameter to 0.002.

Query syntax: COMParator:SLIMit?
Query return: <NR3>,<NR3><NL>

:Auxiliary BIN command is used to set the AUX switch, and can query the current state of the AUX switch.

Command syntax:
COMParator:AuxiliaryBIN { ON | OFF | 1 | 0 }

In here,
Character 0 (integer 48) is equal to OFF.
Character 1 (integer 49) is equal to ON.
For example: WrtCmd ("COMP:ABIN ON"); Switch on the secondary parameter comparison.

Query syntax: COMParator:AuxiliaryBIN?
Query return: <NR1><NL>

:SWAP command is used to set the swap of primary/secondary parameter comparison, and query the current state of swap function.
For example:
Function parameter: when the swap mode is switched on, the function parameter Cp-D changes to D-Cp; at this point, BIN 1~9 limit setting turns to the D upper/lower limit setting and BIN 2 sets Cp limit. If the swap mode is switched off, the parameter sequence is not changed.

Command syntax:
COMParator:SWAP { ON | OFF | 1 | 0 }

In here,
Character 0 (integer 48) is equal to ON.
Character 1 (integer 49) is equal to OFF.
For example: WrtCmd ("COMP:SWAP ON")

Query syntax: COMParator:SWAP?
Query return: <NR1><NL>

:BIN:CLEar command is used to clear the limit data of each BIN in the limit list.
Command syntax: COMParator:BIN:CLEar
For example: WrtCmd ("COMP:BIN:CLE")

:BIN:COUNT command is used to set the BIN NO. Switch (ON/OFF), and can query the current state of the BIN NO. switch.
Command syntax:

COMParator:BIN:COUNt { ON | OFF | 1 | 0 }

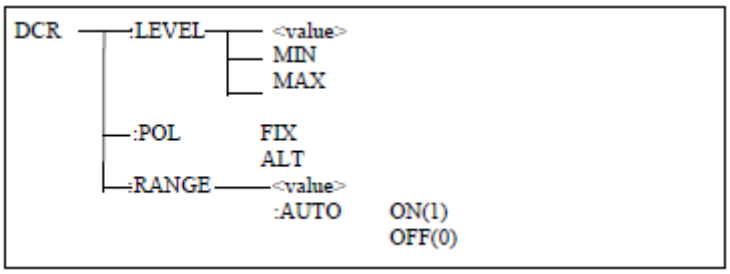
In here,
Character 0 (integer 48) is equal to OFF.
Character 1 (integer 49) is equal to ON.
For example: WrtCmd ("COMP:BIN:COUN ON")
Query syntax: COMParator:BIN:COUNt?
Query return: <NR1><NL>

:BIN:COUNt:DATA? command is used to set the BIN NO. result, and can query the compare result of BIN NO.
Query syntax: COMParator:BIN:COUNt:DATA?
Query return: <BIN1 count>,<BIN2 count>,...,<BIN9 count>,<OUT OF BIN count>,<AUX BIN count><NL>
In here,
<BIN1-9 count> data format of NR1, BIN NO. result is BIN 1-9
<OUT OF BIN count> data format of NR1,BIN NO. result is BIN OUT
<AUX BIN count> data format of NR1, BIN NO. result is BIN AUX

:BIN COUNT:CLEar command is used to clear all the BIN NO. results.
Command syntax: COMParator:BIN:COUNt:CLEar
For example: WrtCmd ("COMP:BIN:COUN:CLE")

2.1.15 DCR Subsystem Command Set

DCR subsystem command set is used to set and query the range, polarity, level of DCR measurement.
Command tree



:LEVEL is used to set the measuring level of DCR measurement, and the character ? can query the current level.
Note: Only UTR2832E can set different level for DCR measurement, the level of UTR2830E is fixed at 1 V.
Command syntax:

DCR:LEVEL { <value> | MIN | MAX }

The details are as follows.
<value> data format of NR1, NR2 or NR3
MIN Set the measuring level to 50 mV
MAX Set the measuring level to 2 V
For example: WrtCmd ("DCR:LEVEL MIN"); Set the measuring level to 50 mV.

Query syntax: DCR:LEVEL?
Query return: <NR3><NL>

:RANGE command is used to set the range of DCR measurement, and the character ? can query the current range.
Command syntax: DCR:RANGE <value>
In here,

<value> the impedance range of DUT or data format of NR1, NR2 or NR3
For example: WrtCmd ("DCR:RANG 1000") ; Set the range 1 kOHM.

Query syntax: DCR:RANG?
Query return: <value><NL>
In here,
<value>
1 3 10 30
100 300 1000 3000
10000 30000 100000

DCR:RANGe:AUTO command is used to set the automatic range, and the character ? is used to query the current state of the automatic range.

Command syntax:

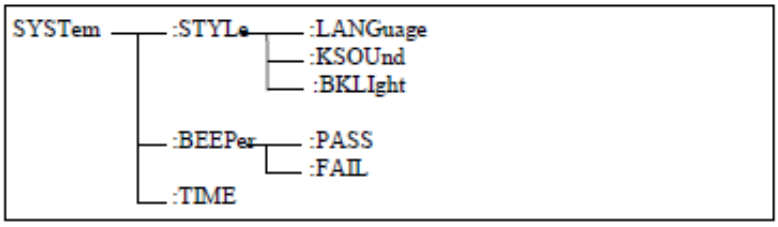
DCR:RANG:AUTO { ON (1) | OFF (0) }

In here,
Character 1 (integer 49) is equal to ON.
Character 0 (integer 48) is equal to OFF.
For example: WrtCmd ("DCR:RANG:AUTO ON"); Switch on the automatic range.

Query syntax: DCR:RANG:AUTO?
Query return: <NR1><NL>

2.1.16 SYSTEM Subsystem Command Set

SYSTEM subsystem command set is used to set and query the system function of language, key sound, backlight brightness and time.
Command tree



:STYLe:LANGuage command is used to set the language, and the character ? is used to query the current language.

Command syntax:

SYSTem:STYLe:LANGuage { ENGlish | CHInese }

The details are as follows.
For example: WrtCmd ("SYST:STYL:LANG ENG"); Set the language to English.

Query syntax: SYST:STYL:LANG?

Query return: { ENGlish | CHInese } <NL>

:STYLe:KSOUnd command is used to set the key sound, and the character ? is used to query the current state of key sound.

Command syntax:

SYSTem:STYLe: KSOUnd { ON | OFF | 1 | 0 }

For example: WrtCmd("SYST:STYL:KSOU ON"); Switch on the key sound.

Query syntax: SYST:STYL:KSOU?
Query return: <NR1><NL>

:STYLe:BKLIght command is used to set the backlight brightness, and the character ? is used to query the current backlight brightness.
Command syntax:
SYSTem:STYLe:BKLIght <value>
In here,
<value>: 0,1,2,3,4,5
0: The brightness is 10%.
1: The brightness is 30%.
2: The brightness is 50%.
3: The brightness is 70%.
4: The brightness is 90%.
5: The brightness is 100%.
For example: WrtCmd("SYST:STYL:BKLI 2"); Set the backlight brightness to 50%.

Query syntax: SYST:STYL:BKLI?
Query return: <NR1><NL>(<NR1 takes value as 0-5>)

:BEEPer:PASS command is used to set the PASS beeper, and the character ? is used to query the current beeper.
Command syntax:

SYSTem:BEEPer:PASS { OFF | SHORt | LONG }

For example: WrtCmd("SYST:BEEP:PASS SHOR"); Set the PASS beeper to short sound.

Query syntax: SYST:BEEP:PASS?

Query return: { OFF | SHORt | LONG } <NL>

:BEEPer:FAIL command is used to set the FAIL beeper, and the character ? is used to query the current beeper.
Command syntax:

SYSTem:BEEPer:FAIL { OFF | SHORt | LONG }

For example: WrtCmd("SYST:BEEP:FAIL LONG") ; Set the PASS beeper to long sound.

Query syntax: SYST:BEEP:FAIL?

Query return: { OFF | SHORt | LONG } <NL>

:TIME command is used to set the time, and the character ? is used to query the current time.
Command syntax:
SYSTem:TIME <value1>,<value2>*

Note: * indicates a maximum of 6.
<value1>: year
<value2>: month
<value3>: day
<value4>: hour
<value5>: minute
<value6>: second
For example: WrtCmd ("SYST:TIME 2021,4,20,10,18,18"); Set the time to be 2021-4-20 10:18:18.

Query syntax: SYST:TIME?
Query return: XXXX-XX-XX XX:XX:XX<NL> (Year-Month-Day Hour:Minute:Second)

2.1.17 *IDN? Subsystem

*IDN? subsystem is used to return the instrument version number.
Query syntax: *IDN?
Query response: <Manufacturer>,<MODEL>,<Sn>,<Revision>
For example: send *IDN?
Query eturn: UNIT,UTR2830E,CDB3223300005,REV1