Command Reference

1. The signs in this manual are as follows:

NR1 :integer,e.g:123.

NR2 :fix-point number,e.g:12.3.

NR3 :floating-point number,e.g:12.3E+5.
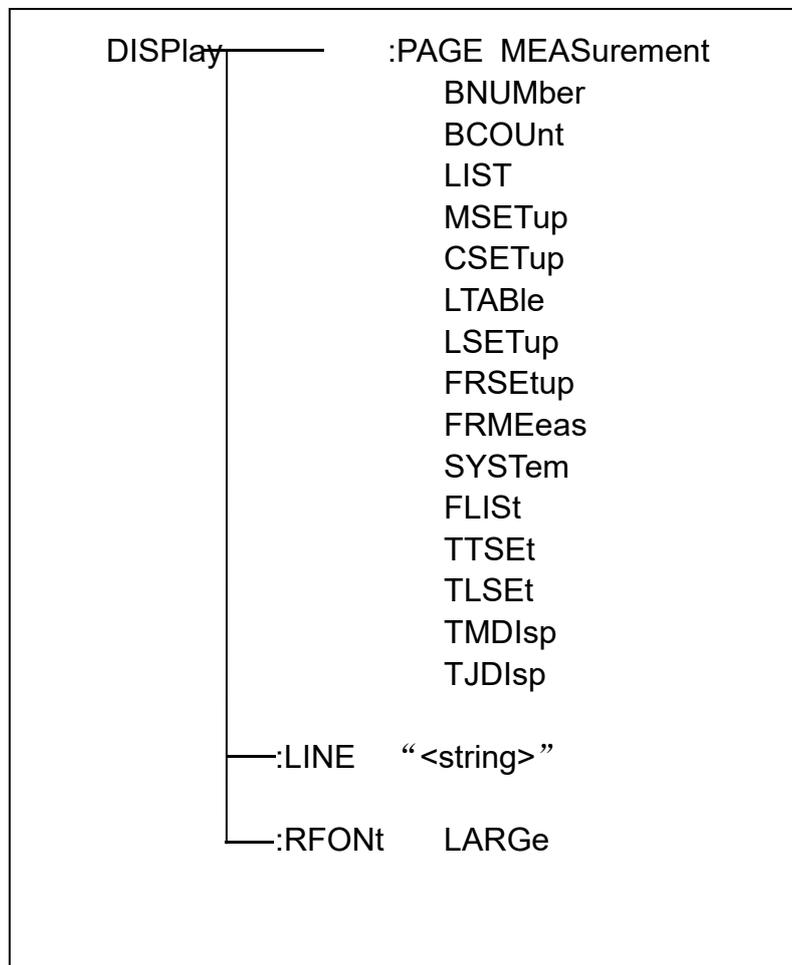
NL :carriage key, integer:10.

^END：EOI signal in IEEE-48.

Subsystem commands

●DISPlay   ●ORESister ●TRIGger   ●CORRection

●FREQuency ●BIAS   ●INITiate   ●COMParator

●VOLTage   ●FUNCtion   ●FETCh?   ●Mass MEMory

●CURRent   ●LIST   ●ABORT   ●TRAN

●AMPLitude ●APERture   ●STATus   ●FRES

DISPlay subsystem command:

DISPlay subsystem command are mianly used to set the display page for the instrument. The:DISPlay? Query returns the current page.

Command tree:

```
DISPlay ──────────    :PAGE  MEASurement
                              BNUMber
                              BCOUnt
                              LIST
                              MSETup
                              CSETup
                              LTABle
                              LSETup
                              FRSEtup
                              FRMEeas
                              SYSTem
                              FLISt
                              TTSEt
                              TLSEt
                              TMDIsp
                              TJDIsp

        ──:LINE    "<string>"

        ──:RFONt    LARGe
```

TINY
OFF

The :PAGE command sets the display page.
The :DISPlay :PAGE? Query returns the current page.
Command syntax: DISPlay:PAGE<page name>
<Page name> can be set as the following items:

MEASurement   Set the display page as the LCR measurement display.
BNUMber   Set the display page as the bin number display.
BCOUnt   Set the display page as the bin count display.
LIST   Set the display page as the list sweep display.
MSETup   Set the display page as the measurement display.
CSETup   Set the display page as the correction setup.
LTABle   Set the display page as the limit table setup.
LSETup   Set the display page as the list sweep setup.
FRSEtup   Set the display page as the frequency response setup.（Only
                 part of the instruments）
FRMEeas   Set the display page as the frequency response
                 measurement display.
SYSTem   Set the display page as the system setup page.
FLISt   Set the display page as the file list page.
TTSEt   Set the display page as the transformer test setup.（Only part
                 of the instruments）
TLSEt   Set the display page as the transformer limit setup.（Only part
                 of the instruments）
TMDIsp   Set the display page as the transformer measurement display.
                 （Only part of the instruments）
TJDIsp   Set the display page as the transformer judge display. （Only
                 part of   the instruments）

For example: WrtCmd（"DISP:PAGE MEAS"),Set the display page as
                 theLCR measurement display.

Query syntax: DISPlay:PAGE?
Return format:<page name><NL^END>

<page name>can be set as the following items:
<LCR MEAS DISP>   The current page is the LCR measurement display
                    page.
<BIN No. DISP>   The current page is the bin number display page.
<BIN COUNT DISP>   The current page is the bin count display page.
<LIST SWEEP DISP>   The current page is the list sweep display page.

\<MEAS SETUP\>	The current page is the measurement setup page.
\<CORRECTION\>	The current page is the correction page.
\<LIMIT TABLE SETUP\>	The current page is the limit table setup page.
\<LIST SWEEP SETUP\>	The current page is the list sweep setup page.
\<FREQ RESP SETUP\>	The current page is thefrequency response setup page.（Only part of the instruments）
\<FREQ RESP MEAS\>	The current page is the frequency response measurement .（Only part of the instruments）
\<SYSTEM SETUP\>	The current page is the system setup page.
\<FILE LIST\>	The current page is the file list page.
\<TRANS TEST SET\>	The current page is the transformer test setup page.（Only part of the instruments）
\<TRANS LIMIT SET\>	The current page is the transformer limit setup page.（Only part of the instruments）
\<TRANS MEAS DISP\>	The current page is thetransformer measurement display page.（Only part of the instruments）
\<TRANS JUDGE DISP\>	The current page is thetransformer judge display page.（Only part of the instruments）

The :LINE command is used to set the current measurement item which can be a substring with up to 20 characters. The :LINE? Query returns the current measurement item. The character string of the measurement item can be used as the file name when saving a file.

Command syntax：DISPlay:LINE"\<string\>"
Where,
\<string\>can be an ASCⅡ character string(maximun number is 20).
For example：WrtCmd（"DISP:LINE " Resistor meas""）;
Command syntax： DISPlay:LINE?
Return format: \<string\>\<NL^END\>

The :ResultFONt command is used to set the current font of the measurement result. The :ResultFONt? Query returns the current font of the measurement result.
Command syntax：DISPlay:RFONt \<font\>
  \<font\> can be the following information:
LARGe : Use large character to display the measurement result,12ms/meas
TINY : Use tiny character to display the measurement result,5ms/meas
OFF: Measurement result will not be displayed but can be read from the bus.

Query syntax: DISPlay:RFONt?
Return format: <font><NL^END>
<font> can be the following information:
LARGE
TINY
OFF

FREQuency Subsystem
The :FREQuency command sets the oscillator frequency. The FREQuency?
query returns the current test frequency setting.

Command Syntax:

FREQuency $\left\{ \begin{array}{l} \text{<value>} \\ \text{MIN} \\ \text{MAX} \end{array} \right\}$

Where,
<value>   NR1,NR2 or NR3 data format followed by Hz,kHz,MHz.
MIN   Sets to the minimum value (20 Hz)
MAX   Sets to the maximum value (200 kHz)

Example
WrtCmd（"FREQ 1KHZ"）Set test frequency to 1000 Hz.

Query Syntax: FREQuency？
Return Format :   <NR3><NL^END>

VOLTage Subsystem

The :VOLTage subsystem commands are mainly used to set the measurement
voltage. The VOLTage? query returns the current measurement voltage .

Command Syntax:

VOLTage $\left\{ \begin{array}{l} \text{<value>} \\ \text{MIN} \\ \text{MAX} \end{array} \right.$

Where,
<value>   NR1, NR2, or NR3 format followed by V .
MIN   Set the measurement voltage as 5mV.
MAX   Set the measurement voltage as 2V.

Example: WrtCmd("VOLT 1V")   Set the measurement voltage level as1 V.
Query Syntax :VOLTage?
Return format : <NR3><NL^END>

CURRent Subsystem

The :CURRent subsystem commands are mainly used to set the measurement current. The CURRent? query returns the current measurement voltage .

Command Syntax:

CURRent ⎧ <value>
⎨ MIN
⎩ MAX

Where,
<value>   NR1, NR2, or NR3 format followed by mA .
MIN   Set the measurement voltage as 50uA.
MAX   Set the measurement voltage as 20mA.

Example: WrtCmd("CURR 10mA")   Set the measurement current as10mA..
Query Syntax :CURRent?
Return format : <NR3><NL^END>

AMPLitude subsystem commands

The :AMPLitude subsystem commands are mainly used to set the auto level Control（ALC）function as ON or OFF. The :AMPLitude:ALC? Query returns the current status of the ALC function.

Command Syntax:

AMPLitude:ALC ⎧ ON
⎨ OFF
⎨ 1
⎩ 0

Where,
Character 1(49) is equal to ON.
Character 1(48) is equal to OFF.
Example: WrtCmd("AMPL:ALC 0") Set the ALC function as OFF.

Query Syntax : AMPLitude:ALC？
Query Syntax :AMPLitude:ALC？
Return format :<NR1><NL^END>

Output RESister subsystem commands

The :Output RESistor subsystem commands are mainly used to set the output Internal resisitor mode. The Output RESistor？ Query returns the current output internal resistance status.

Command Syntax:

$$\text{ORESister} \begin{cases} 30 \\ 100 \end{cases}$$

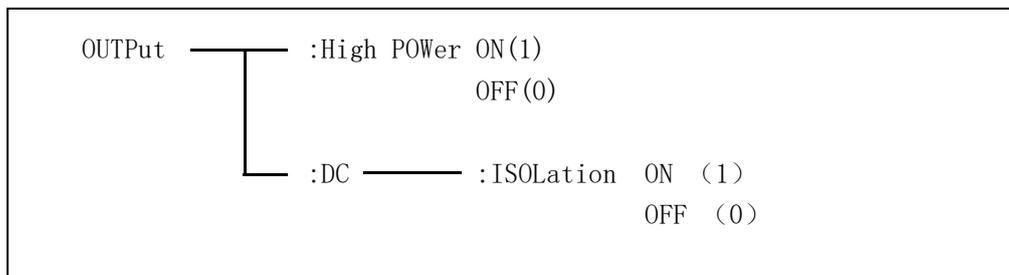Example:WrtCmd("ORES 30")；Set the output internal resistance status.
Query Syntax :ORESister?
Return format :<NR1><NL^END>

Output subsystem commands

The :OUTPout subsystem commands are mainly used to set the status of the DC 100mA/10V bias source and DC isolation function.

Command tree:

```
OUTPut ──────┬── :High POWer ON(1)
             │                OFF(0)
             │
             └── :DC ──────── :ISOLation  ON （1）
                                          OFF （0）
```

THe OUTPut: HPOWer command is used to set the 100mA/10V DC bias source as ON or OFF. THe OUTPut: HPOWer? Query the current status of the 00mA/10V DC bias source. Before using this command, please ensure the option component 100mA/10V DC bias source has been installed.

Command Syntax:

$$\text{OUTPut:HPOWer} \begin{cases} \text{ON} \\ \text{OFF} \\ 1 \\ 0 \end{cases}$$

Where,
Character 1(49) is equal to ON.
Character 0(48) is equal to OFF.

Command Syntax: OUTPut:HPOWer?
Returned format :   <INT> <NL^END> or <OPT1><NL^END>, OPT means the 100mA/10V DC bias source is used, or unused.

THe OUTPut: DC:ISOLation command is used to set the DC isolation function Of the 100mA/10V bias source as ON or OFF.
THe OUTPut: DC:ISOLation? Query the current status of the DC isolation function. Before using this command, please ensure the option component 100mA/10V DC bias source has been installed.

Command Syntax:

```
                              ┌  ON
                              │  OFF
   OUTPut:DC:ISOLation  ┤
                              │  1
                              └  0
```
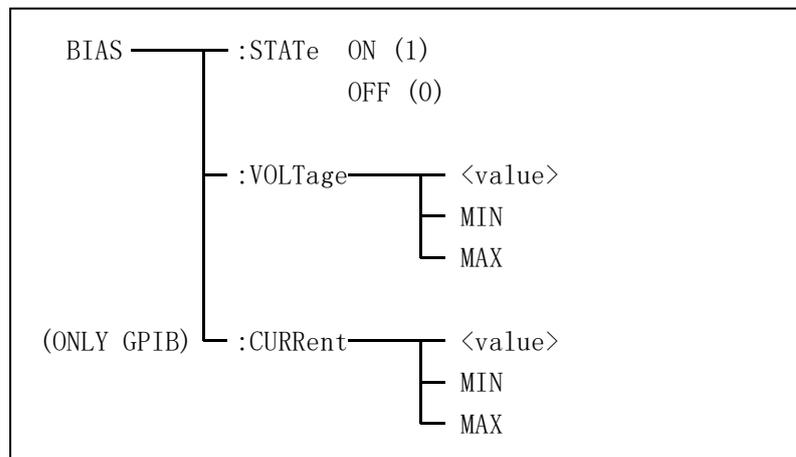
Where,
Character 1(49) is equal to ON.
Character 0(48) is equal to OFF.

Command Syntax: OUTPut:DC:ISOLation?
Return format :   <NR1> <NL^END>。

BIAS subsystem commands

The :BIAS subsystem commands are mainly used to set the internal bias Voltage and the bias status.
Command tree

```
   BIAS ──────── :STATe   ON  (1)
                              OFF (0)

                 :VOLTage──────  <value>
                             ├  MIN
                             └  MAX

   (ONLY GPIB) └ :CURRent──────  <value>
                             ├  MIN
                             └  MAX
```

The BIAS:STATe command is used to the bias status. The BIAS:STATe ？ Query returns the current bias status.
Command Syntax:

BIAS:STATe
$$\begin{cases} \text{ON} \\ \text{OFF} \\ 1 \\ 0 \end{cases}$$

Where,
Character 1(49) is equal to ON.
Character 0(48) is equal to OFF.
Example: WrtCmd（"BIAS:STATe 0"） Set the DC bias function as OFF.

Query Syntax :BIAS:STATe？
Return format :<NR1><NL^END>

The BIAS:： VOLTage command is used to the bias voltage. The BIAS: VOLTage ？ Query returns the current bias status.

Command Syntax:

BIAS:VOLTage
$$\begin{cases} \text{<value>} \\ \text{MIN} \\ \text{MAX} \end{cases}$$

Where,
<value>   NR1, NR2, or NR3   data format.
MIN   Set the bias voltage as 0V.
MAX   Set the bias voltage as 2V.
Example: WrtCmd（"BIAS:VOLT MIN"） Set the DC bias voltage as 0V.

Query Syntax :BIAS:VOLTage?
Return format : <NR3><NL^END>

The BIAS:： CURRent command is used to set the external bias current. The BIAS: CURRent ？ Query returns the bias current . The external bias current is controlled by serial interface, so only GPIB interface supports this command.

Command Syntax:

BIAS: CURRent
$$\begin{cases} \text{<value>} \\ \text{MIN} \\ \text{MAX} \end{cases}$$

Where,
<value>
MIN   Set the bias current as 0A.

MAX   Set the bias current as 100mA.
Example: WrtCmd("BIAS:CURR MIN") Set the DC bias curren as 0A.
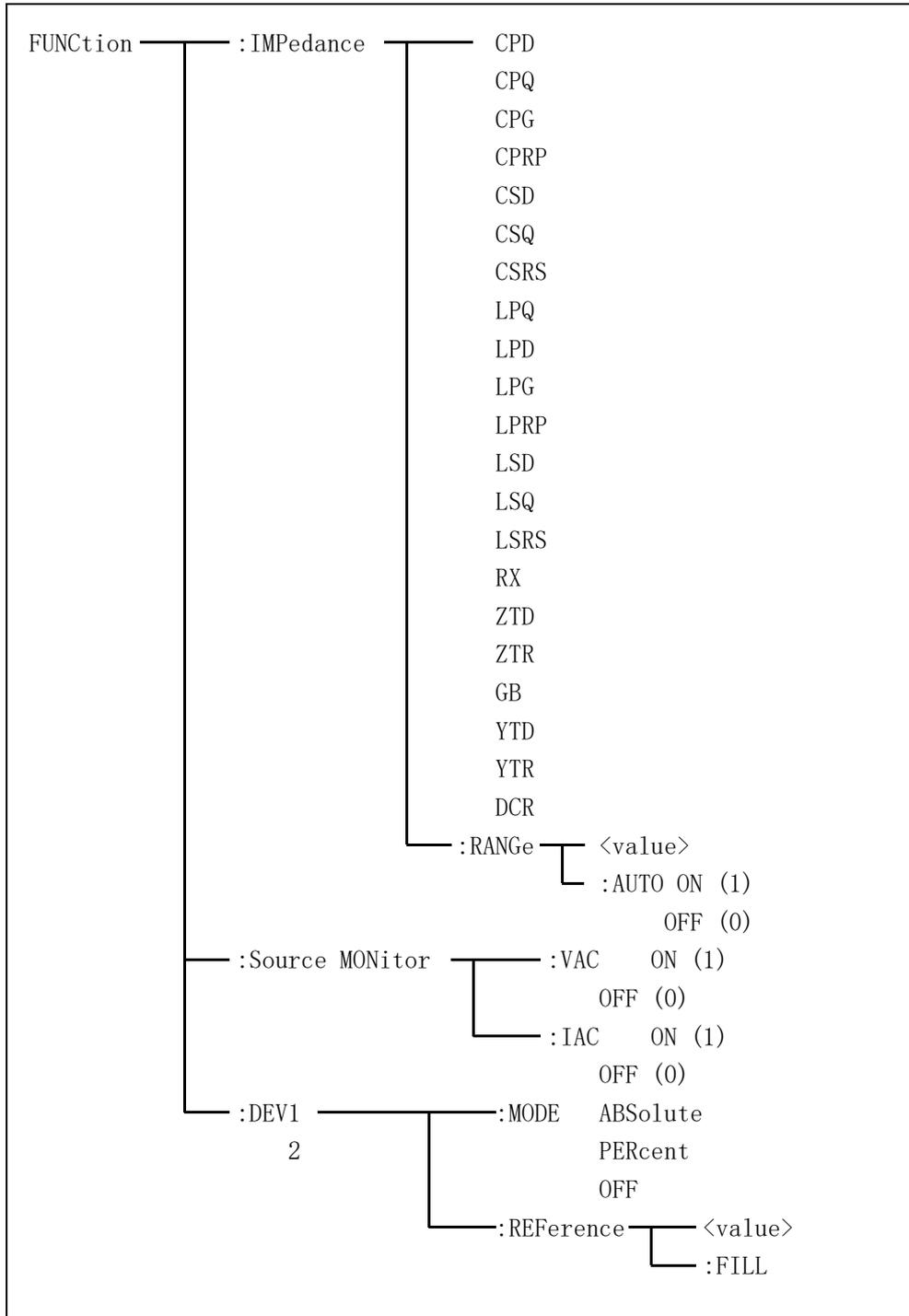
Query Syntax :BIAS:CURRent?
Return format : <NR3><NL^END>

FUNCtion Subsystem

The FUNCtion subsystem commands are mainly used to set measurement functions, range, current/voltage monitor ON/OFF control, deviation display mode,nominal setting.

Command tree:

```
FUNCtion ──────── :IMPedance ──────── CPD
                  │                   CPQ
                  │                   CPG
                  │                   CPRP
                  │                   CSD
                  │                   CSQ
                  │                   CSRS
                  │                   LPQ
                  │                   LPD
                  │                   LPG
                  │                   LPRP
                  │                   LSD
                  │                   LSQ
                  │                   LSRS
                  │                   RX
                  │                   ZTD
                  │                   ZTR
                  │                   GB
                  │                   YTD
                  │                   YTR
                  │                   DCR
                  │            :RANGe ──────── <value>
                  │                       └─── :AUTO ON (1)
                  │                                  OFF (0)
                  │ :Source MONitor ──────── :VAC   ON (1)
                  │                       │        OFF (0)
                  │                       └─── :IAC   ON (1)
                  │                                  OFF (0)
                  └─ :DEV1 ──────────── :MODE  ABSolute
                       2                       PERcent
                                               OFF
                                        :REFerence ──────── <value>
                                                        └─── :FILL
```

The FUNCtion :IMPedance command is used to set instrument functions.
The FUNCtion:IMPedance? query returns the current function parameters..

Command Syntax: FUNCtion:IMPedance <function>
<function> can be one of the following items.

CPD	Set the function as Cp-D	LPRP	Set the function as Lp-Rp
CPQ	Set the function as Cp-Q	LSD	Set the function as Ls-D
CPG	Set the function as Cp-G	LSQ	Set the function as Ls-Q
CPRP	Set the function as Cp-Rp	LSRS	Set the function as Ls-Rs
CSD	Set the function as Cs-D	RX	Set the function as R-X
CSQ	Set the function as Cs-Q	ZTD	Set the function as Z-θ°
CSRS	Set the function as Cs-Rs	ZTR	Set the function as Z-θr
LPQ	Set the function as Lp-Q	GB	Set the function as G-B
LPD	Set the function as Lp-D	YTD	Set the function as Y-θ°
LPG	Set the function as Lp-G	YTR	Set the function as Y-θr

Example:WrtCmd("FUNC:IMP RX")   Set the function as R-X.

Query Syntax :FUNCtion :IMPedance?
Return format :<function><NL^END>

The FUNCtion :IMPedance:RANGe is used to set the range. The FUNCtion: IMPedance:RANGe? Query returns the current range.
Command Syntax: FUNCtion:IMPedance : RANGe <value>
Where,<value> can be the impedance of the DUT or NR1,NR2 or NR3 data format followed by OHM or KOHM.

Example: WrtCmd("FUNC:IMP:RANG 1KOHM")   Set the range as 1KOHM.

Query Syntax :FUNCtion :IMPedance :RANGe?
Return format :<value><NL^END>
Where,<value>can be

10	30	100
300	1000	3000
10000	30000	100000

The FUNCtion :IMPedance:RANGe :AUTO command is used to set the automatic range selection status. The FUNCtion:IMPedance:RANGe:AUTO? query returns the current range status.
Command Syntax:

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

Where,
Character 1(49) is equal to ON.
Character 0(48) is equal to OFF.

Query Syntax : FUNCtion:IMPedance:RANGe:AUTO?
Return format :<NR1><NL^END>

The FUNCtion :Source MONitor: VAC command is used to set the voltage monitor as ON or OFF. The FUNCtion :Source MONitor: VAC ? query returns the voltage monitor status.

Command Syntax:

FUNCtion:SMONitor:VAC   { ON (1)
                           OFF (0)

Where,
Character 1(49) is equal to ON.
Character 0(48) is equal to OFF.

Example: WrtCmd("FUNC:SMON:VAC ON")   Set the voltage monitor as on.

Query Syntax :FUNCtion:SMONitor:VAC？
Return format :<NR1><NL^END>

The FUNCtion :Source MONitor: IAC command is used to set the current monitor as ON or OFF. The FUNCtion :Source MONitor: IAC ? query returns the current monitor status.

Command Syntax:

FUNCtion:SMONitor:IAC   { ON (1)
                           OFF (0)

Where,
Character 1(49) is equal to ON.
Character 0(48) is equal to OFF.

Example: WrtCmd("FUNC:SMON IAC ON")   Set the current monitor as on.

Query Syntax :FUNCtion:SMONitor:IAC？
Return format :<NR1><NL^END>

The FUNCtion :DEV<n>:MODE command is used to set the deviation measurement mode. The FUNCtion :DEV<n>:MODE? query returns the current deviation measurement mode.

Command Syntax:

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

Where,
ABSolute   Absolute value deviation display
PERCent   Percent deviation display
OFF   Real value display
Where,<n> is:
Character 1 (49) is equal to the nominal value of the primary parameter.
Character 2 (50) is equal to the nominal value of the secondary parameter.

Example: WrtCmd("FUNC:DEV1:MODE ABS");

Query Syntax : FUNCtion:DEV<n>:MODE?
Return format :

ABS
PERC } <NL^END>
OFF

The FUNCtion :DEV<n>:REFerence <value> command is used to set the nominal value of the deviation. The FUNCtion :DEV<n>:REFerence <value>? query returns the current nominal value of the deviation.

Command Syntax: FUNCtion:DEV<n>:REFerence<value>
Where,
<value> is NR1, NR2, or NR3 data format.
Where,<n> is:
Character 1 (49) is equal to the nominal value of the primary parameter.
Character 2 (50) is equal to the nominal value of the secondary parameter.

Example: WrtCmd("FUNC:DEV1:REF 10");

Query Syntax :FUNCtion:DEV<n>:REFerence?
Return format :<NR3><NL^END>

The FUNCtion :DEV<n>:REFerence:FILL command is used to set the nominal value of the deviation. This command directs the instrument to make a test and then copies the results of the primary and the secondary parameters as the nominal values of the deviation.

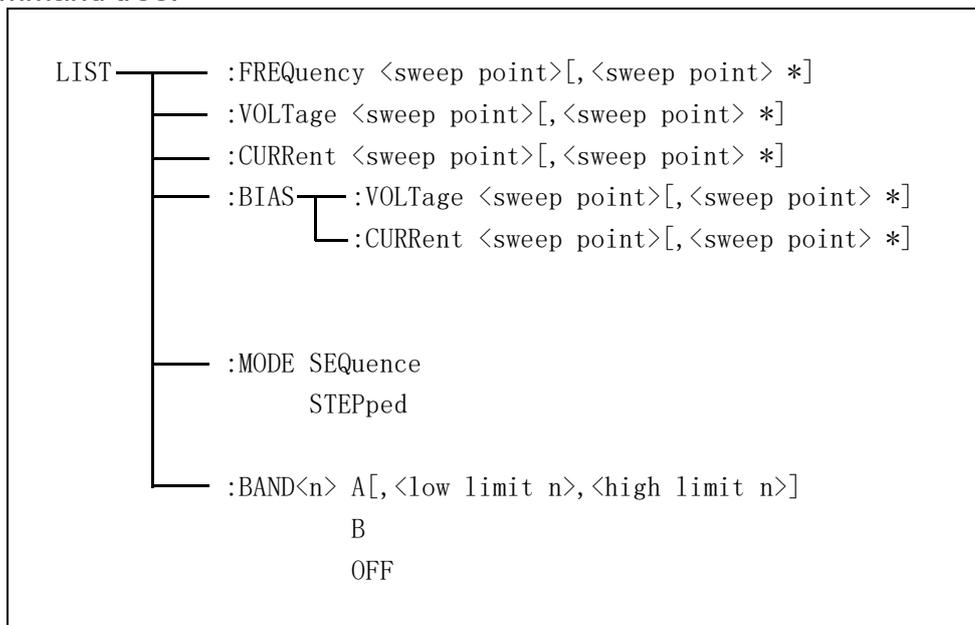Command Syntax: FUNCtion:DEV<n>:REFerence:FILL
Where,<n> is:

Character 1(49) or Character 2(50) is equal to the nominal value of the primary and secondary parameter.
Example: WrtCmd("FUNC:DEV1:REF:FILL");

LIST Subsystem

The LIST subsystem commands are mainly used to set the list sweep function, Sweep points,sweep mode,sweep limits.
Command tree:

```
LIST ─────────  :FREQuency <sweep point>[,<sweep point> *]
          ├──   :VOLTage <sweep point>[,<sweep point> *]
          ├──   :CURRent <sweep point>[,<sweep point> *]
          ├──   :BIAS ──┬── :VOLTage <sweep point>[,<sweep point> *]
          │             └── :CURRent <sweep point>[,<sweep point> *]
          │
          │
          ├──   :MODE  SEQuence
          │            STEPped
          │
          └──   :BAND<n> A[,<low limit n>,<high limit n>]
                        B
                        OFF
```

The LIST :FREQuency command is used to clear the original sweep point and set the frequencies of the sweep points. The LIST :FREQuency? Query returns the current frequency of each sweep point.

Command Syntax: LIST:FREQuency <value> [,<value> *]
NOTE: * part means 10 sweep points at most can be set.

Where,
<value> is NR1, NR2, or NR3 data format.
<value> should be set from 20Hz~200KHz or return format will report errors.
Example: WrtCmd("LIST:FREQ 1E3,2E3,3E3,4E3")
Set the frequency of the sweep point 1 as 1KHz;
Set the frequency of the sweep point 2 as 2KHz;
Set the frequency of the sweep point 3 as 3KHz;
Set the frequency of the sweep point 4 as 4KHz;
NOTE: HZ(hertz）is the suffix unit,MAHZ 和 MHZ 即为 MHz(1E6 Hz).

Query Syntax : LIST:FREQuency?
Return format :<NR3>[,<NR3> *]<NL^END>

The LIST :VOLTage command is used to clear the original voltage of the each sweep point and reset the voltage. The :VOLTage? query returns the current voltage of each sweep point.

Command Syntax: LIST:VOLTage <value>[,<value> *]
NOTE: * part means 10 sweep points at most can be set.
Where,
<value> is NR1, NR2, or NR3 data format.

Example:
WrtCmd("LIST:VOLT 1.5")
Set the frequency of the sweep point 1 as 1.5V.
WrtCmd("LIST:VOLT 1E-2,2E-2,3E-2,4E-2")
Set the frequencies of the sweep point 1, 2, 3 and 4 respectively as 10mV, 20mV, 30mV and 40mV.
NOTE: This command can be followed bu suffix unit V.

Query Syntax : LIST:VOLTage?
Return format :<NR3>[,<NR3> *]<NL^END>
<value> should be set from 5mV to 2V, or return format will report errors.

The LIST :CUURent command is used to clear the original current of the each sweep point and reset the current. The LIST :CUURent? query returns the current current of each sweep point.

Command Syntax: LIST:CURRent <value>[,<value> *]
NOTE: * part means 10 sweep points at most can be set.
Where,
<value> is NR1, NR2, or NR3 data format.

Example:
WrtCmd("LIST:CURR 100MA")
Set the measurement current of the sweep point 1 as 100mA.
WrtCmd("LIST:CURR 1E-2,2E-2,3E-3,4E-3")
Set the measurement current of the sweep point 1, 2, 3 and 4 respectively as 10mA, 20mA, 30mA and 40mA.
NOTE: This command can be followed bu suffix unit A.

Query Syntax : LIST:CURRent?
Return format :<NR3>[,<NR3> *]<NL^END>
NOTE: The current of each sweep point should be set from 50uA to 20mA,or return format will report errors.

The LIST:BIAS:VOLTage command is used to clear original DC bias voltage of each sweep point and reset the voltage. The LIST:BIAS:VOLTage? query returns the current DC bias voltage of each sweep point.

Command Syntax: LIST:BIAS:VOLTage <value>[,<value> *]
NOTE: * part means 10 sweep points at most can be set.
Where,
<value> is NR1, NR2, or NR3　data format.

Example:
WrtCmd("LIST:BIAS:VOLT 1.5V")
Set the DC bias voltage of the sweep point 1 as 1.5V.

Query Syntax : LIST:BIAS:VOLTage?
Return format :<NR3>[,<NR3> *]<NL^END>
NOTE: The DC bias voltage of　0V,1.5V, 2V not within,or return format will report errors.

The LIST:BIAS:CURRent command is used to clear original DC bias current of each sweep point and reset the current. The LIST:BIAS:CURRent? query returns the current DC bias current of each sweep point.

Command Syntax: LIST:BIAS:CURRent <value>[,<value> *]
NOTE: * part means 10 sweep points at most can be set.
Where,
<value> is NR1, NR2, or NR3　data format.

Example:
WrtCmd("LIST:BIAS:CURR 100MA")
Set the DC bias current of the sweep point 1 as 100mA.
WrtCmd（"LIST:BIAS:CURR 1E-2,2E-2,3E-2,4E-2"）
Set the currents of the sweep points 1, 2, 3 and 4 respectively as 10mA, 20mA, 30mA and 40mA.

Query Syntax : LIST:BIAS:CURRent?
Return format :<NR3>[,<NR3> *]<NL^END>

The LIST:MODE command is used to set the list sweep mode. The LIST:MODE? query returns the current list sweep mode.

Command Syntax:

LIST:MODE $\left\{ \begin{array}{c} \text{SEQuence} \\ \text{STEPped} \end{array} \right\}$

Where,
SEQuence means sequential mode.
STEPped means single step mode.

Example: WrtCmd("LIST:MODE SEQ")

Query Syntax : LIST:MODE?
Return format :

$\left\{ \begin{array}{c} \text{SEQ} \\ \text{STEP} \end{array} \right\}$ <NL^END>

The LIST:BAND<n> command is used to set the limits of list sweep table.The LIST:BAND<n>? query returns the current limits .

Command Syntax: LIST:BAND<n><parameter>[,<low limit n>,<high limit n>]
Where,

<n>		1 to 10（NR1 format）: sweep points on the n_{th} line.
<parameter>:	A	Compare the primary parameter of the test results with the high and low limits.
	B	Compare the secondary parameter of the test results with the high and low limits.
	OFF	No comparison
<low limit n>		NR1, NR2, or NR3　data format, low limit of the sweep point on the n_{th} line.
<high limit n>		NR1, NR2, or NR3　data format, high limit of the sweep point on the n_{th} line.

Example: WrtCmd("LIST:BAND1 A,10,20")
           WrtCmd("LIST:BAND3 OFF")

Query Syntax : LIST:BAND<n>?
Return format :<parameter>,<low limit n>,<high limit n>

APERture Subsystem:

The APERture Subsystem command are mainly used to set the measurement speed,average times used in measurement. The APERture? query returns the

current measurement speed ,average times.
Command Syntax:

```
          ┌  FAST   ┐
APERture ─┤  MEDium ├─  [,<value>]
          └  SLOW   ┘
```

Where,
FAST：   30 times/sec
MEDium：   30 times/sec
SLOW：   30 times/sec
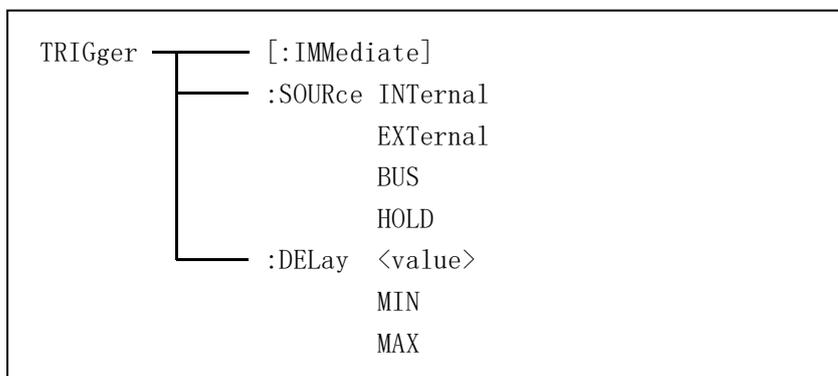<value>   1 to 255 in NR1
Example: WrtCmd("APER MED,55");

Query Syntax :APERture?
Return format :
```
          ┌  FAST ┐
          ┤  MED  ├─  ,<NR1><NL^END>
          └  SLOW ┘
```

TRIGger Subsystem commands

The TRIGger subsystem commands are mainly used to set the instrument trigger source,trigger delay and trigger measurement.
Command tree:

```
TRIGger ───┬─── [:IMMediate]
           │
           ├─── :SOURce INTernal
           │            EXTernal
           │            BUS
           │            HOLD
           └─── :DELay  <value>
                        MIN
                        MAX
```

The TRIGger[:IMMediate] command is used to trigger a test.
Command Syntax: TRIGger[:IMMediate]
Example: WrtCmd("TRIG");

The TRIGger :SOURce command is used to trigger source mode. The TRIGger :SOURce ? query returns the current trigger source mode.

Command Syntax:

$$\text{TRIGger:SOURce} \begin{cases} \text{INTernal} \\ \text{EXTernal} \\ \text{BUS} \\ \text{HOLD} \end{cases}$$

Where,

INTernal   The default trigger mode.

EXTernal   Triggered by HANDLER interface.

BUS   Triggered by RS232C interface or GPIB interface

HOLD   Triggered by pressing "TRIGGER"

Example: WrtCmd("TRIG:SOUR BUS");

Query Syntax : TRIGger:SOURce?

Return format :

$$\begin{cases} \text{INT} \\ \text{EXT} \\ \text{BUS} \\ \text{HOLD} \end{cases} \text{<NL^END>}$$

The TRIGger :DELay command is used to set the delay time after triggering.

The TRIGger :DELay ? query returns the current delay time.

Command Syntax:

$$\text{TRIGger:DELay} \begin{cases} \text{<value>} \\ \text{MIN} \\ \text{MAX} \end{cases}$$

Where,

<value>   In NR1, NR2, or NR3   data format,from 0 to 60s with 1ms as the resolution.

MIN   set the delay time as 0s.

MAX   set the delay time as 60s.

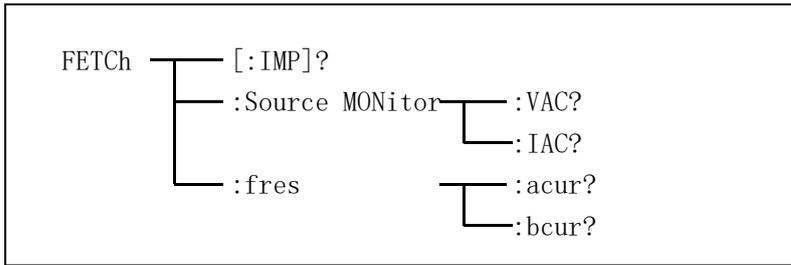Example: WrtCmd("TRIG:DEL 5S"); set the delay time as 5s.

Query Syntax :TRIGger:DELay?

Return format :<NR3><NL^END>

FETCh? Subsystem command

The FETCh? Subsystem commands are mainly used to direct the instrument to input a measurement result.

Command tree:

```
FETCh ─────── [:IMP]?
         │
         ├─── :Source MONitor ─┬─ :VAC?
         │                     └─ :IAC?
         │
         └─── :fres         ─┬─ :acur?
                             └─ :bcur?
```

The FETCh [:IMP]? query directs the instrument to input the last measurement result to the output buffer zone.

Query Syntax   :FETCh[:IMP?
Example: WrtCmd("TRIG:SOUR BUS");
            WrtCmd("TRIG");
            WrtCmd("FETC?");

The instrument applies ASCⅡto delivery result,details are follows.
On measurement display page,bin NO.display page,bin count display page,ASCⅡdata output format are described as below:

| SN.NNNNNESNN | , | SN.NNNNNESNN | , | SN | , | SN 或 SNN | NL^END |

     \<DATA A>   \<DATA B>   \<状态>   \<档号>

Where,
<DATA A>,<DATA B> format: <DATA A>(primary measurement data),<DATA B>(secondary measurement data)
12_digits ASCⅡ format are as below:

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

Status	Description
-1	(In data buffer memory) no data
0	Common measurement data
+1	Analog LCR unbalance
+2	A/D converter is not working
+3	Signal source is over loading
+4	Constant voltage cannot be adjusted

<status> format: When above measurement data is used,<status> data will display measurement status.

The output format of the <status> display data uses 2-digits ASCⅡ: SN(S:+/-,N:from 0 to 4)

NOTE: When <status> is -1, +1 or +2, the measurement data is 9.9E37.When <status> is 0, +3 or +4, the real measurement data is beyond the limits.
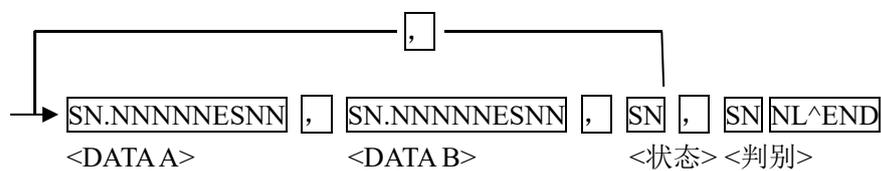
<Bin NO.>format: The data displays the sorting results of the displayed bin, shown as below.

Data	Sort result
0	Out of tolerance
+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	Auxiliary Bin

Only when the instrument compare function is set as ON,<bin No.> data can be displayed .
The output format of the <bin No.> data applies 2 to 3 digits ASCⅡ: SN or SNN(S:+/-,N:from 0 to 9)

On list sweep display page,the ASCⅡdata format is shown as below,that is, the return-circuit replaces sweep point number.

```
                          ┌─── , ───┐
        ┌─────────────────┴─────────┴─────────────┐
        │                                          │
        ▼
    SN.NNNNNESNN  ,   SN.NNNNNESNN  ,   SN  ,   SN NL^END
    <DATA A>          <DATA B>           <状态> <判别>
```

Where,
Descriptions for <DATA A>,<DATA B>,<Status> are the same described before.

<Judge> format is as below；
<Input/Output>format: The data displayed the compare result of the list sweep.

Data	Result
−1	low
0	pass
+1	high

When the compare function of the list sweep measurement is turned off, the output result of <Input/Output> is 0.

<Input/Output> data format applies 2-digits ASCⅡ format: SN (S:+/-,N:from 0 to1)

On transformer single-machine measurement display page, ASCⅡ data output format is as below:

SN	,	SN.NNNNNESNN	,	SN.NNNNNESNN	,	SN	NL^END

<Serial number>　　<DATA A>　　　　<DATA B>　　　　<Status>

<SN> is described as below：

SN	Description
1	TURN
2	Lx
3	LK
4	DCR

Description of <DATA A> is the same as that described above.Secondary parameter of <DATA B> has two formats:

When the measurement parameter is the turn ratio(TURN), secondary parameter <DATA B> is the phase and input format is + （in-phase), -(anti-phase);

When the measurement parameter is inductance Lx, the secondary parameter <DATA B> is the quality factor Q, its description is the same as that of <DATA A>.

Meanwhile when the measurement serial number is equal to or larger than 3(that is to say testing leakage inductance LK and DC resistance DCR), no secondary parameter will be displayed but directly display the status. The status format is: SN: +/-,N is 0/1.

Status	Description
−1	Low

0	Pass
1	High

In the frequency response test display page, ASCⅡ data output format as follows:
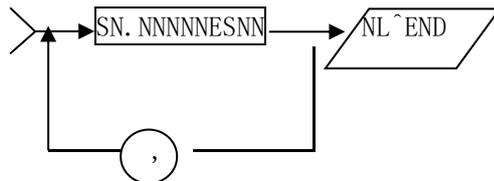
Where,
floating-point number SN.NNNNNESNN column shows eight (twelve) data,as following:

（Ct　　Pt　　Kp　　Qm）
Amax　FAmax　Amin　FAmin
Bmax　FBmax　Bmin　FBmin

Description:

（Ct、Pt、Kp、Qm）　　Optional special parameters, you can obtain details from our company.

Amax　　Maximum primary parameter measured in frequency band of frequency response measurement.

FAmax　　The frequency corresponding to the maximum primary parameter measured in the frequency band.

Amin　　Minimum primary parameter measured in frequency band of frequency response measurement.

FAmin　　Minimum primary parameter measured in frequency band of frequency response measurement.

Bmax　　Maximum secondary parameter measured in frequency band of frequency response measurement.

FBmax　　The frequency corresponding to the maximum secondary parameter measured in the frequency band.

Bmin　　Minimum primary parameter measured in frequency band of frequency response measurement.

FBmin　　The frequency corresponding to the minimum secondary parameter measured in the frequency band.

Above:
NR1 :integer,e.g:123.
NR2 :fix-point number,e.g:12.3.
NR3 :floating-point number,e.g:12.3E+5.

NL :carriage key, integer:10.
^END：EOI signal in IEEE-48.

The FETCh :Source MONitor:VAC? query directs the instrument to send the last measured voltage monitoring results to the output buffer.

Query syntax:FETCh:SMONitor:VAC?
Return format:<NR3><NL^END>
**NOTE: When the voltage monitor as OFF, query returns 9.9E37.**

The FETCh:Source MONitor:IAC? query directs the instrument to send the last measured current monitoring results to the output buffer.

Query syntax:FETCh:SMONitor:IAC?
Return format:<NR3><NL^END>
NOTE: When the current monitor as OFF, query returns 9.9E37.

The FETCh:acur? Query when on the frequency response measurement page, the instrument sends out the curve of the primary parameter in the following format.
The FETCh:acur? Query when on the frequency response measurement page, the instrument sends out the curve of the secondary parameter in the following format.
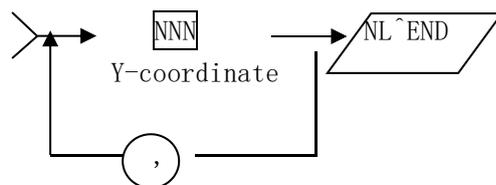
Where,
<Y-coordinate> the total number of decimal 0 to 150 is 240, the X-coordinate is the order number of each point.
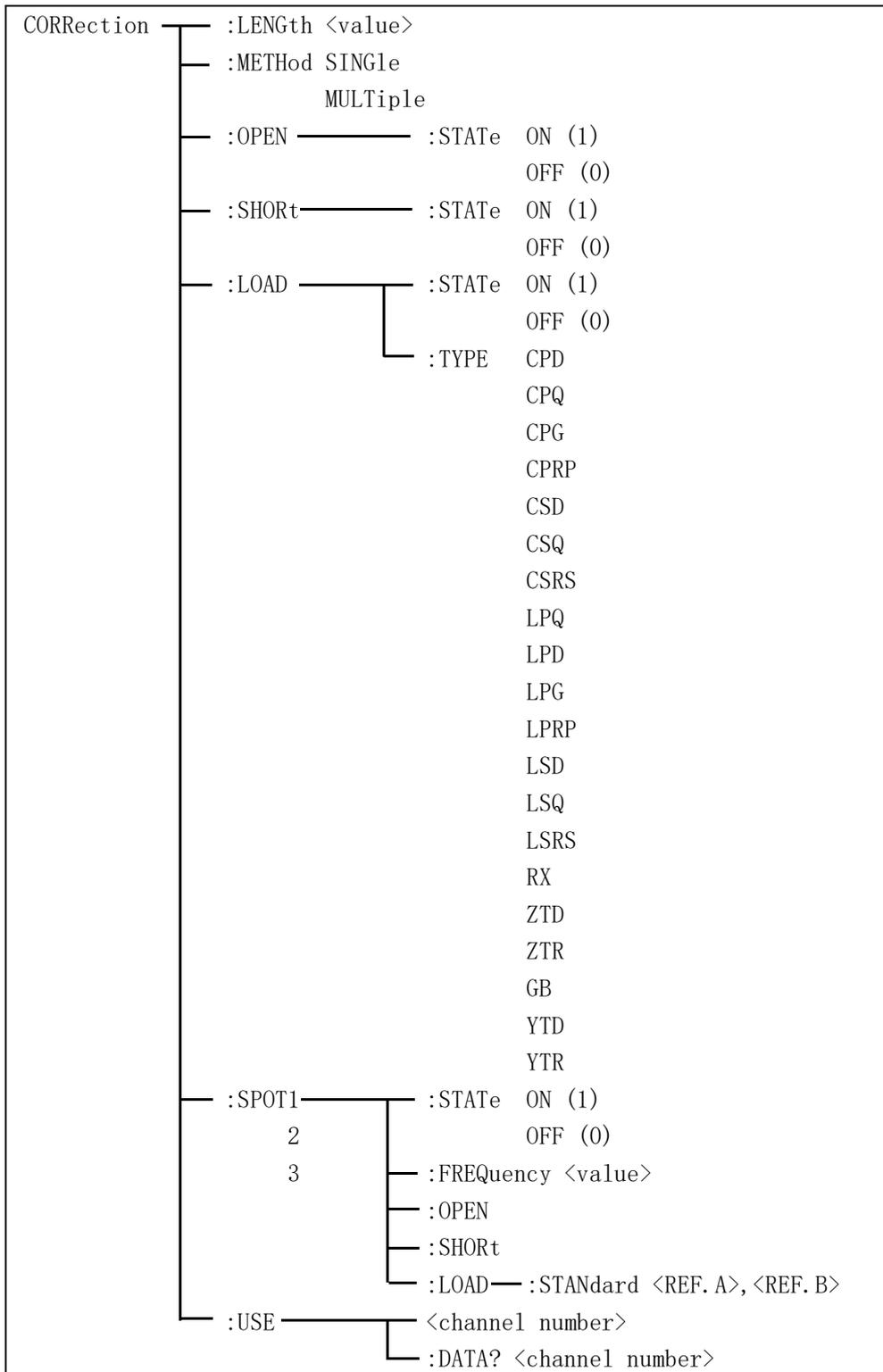Example: " 12,123, 33, 44,　5,120, 64..."
User can use high-level language such as VB drawings on computer based on this data.

CORRection subsystem commands

The CORRection subsystem commands are mainly used to set the correction function, OPEN, SHORT,LOAD.
Command tree:

```
CORRection ──┬── :LENGth <value>
             ├── :METHod SINGle
             │             MULTiple
             ├── :OPEN ──────── :STATe   ON (1)
             │                           OFF (0)
             ├── :SHORt ─────── :STATe   ON (1)
             │                           OFF (0)
             ├── :LOAD ──────┬── :STATe   ON (1)
             │               │            OFF (0)
             │               └── :TYPE    CPD
             │                            CPQ
             │                            CPG
             │                            CPRP
             │                            CSD
             │                            CSQ
             │                            CSRS
             │                            LPQ
             │                            LPD
             │                            LPG
             │                            LPRP
             │                            LSD
             │                            LSQ
             │                            LSRS
             │                            RX
             │                            ZTD
             │                            ZTR
             │                            GB
             │                            YTD
             │                            YTR
             ├── :SPOT1 ──────┬── :STATe   ON (1)
             │        2       │            OFF (0)
             │        3       ├── :FREQuency <value>
             │                ├── :OPEN
             │                ├── :SHORt
             │                └── :LOAD ── :STANdard <REF.A>,<REF.B>
             └── :USE ───────┬── <channel number>
                             └── :DATA? <channel number>
```

The CORRection :LENGth is used to set the correction cable length. The CORRection :LENGth? Query returns the current set cable length.
Command Syntax:CORRection:LENGth <value>
Where,
<value> is 0,1,2 or 4 followed by M

Example: WrtCmd("CORR:LENG 1M")
        Set the cable length as 1 meter.

Query syntax:CORRection:LENGth?
Return format:<NR1><NL^END>

The CORRection: METHod command is used to set the correction mode. The CORRection :METHod? Query returns the current correction mode.

Query syntax:CORRection:METHod $\left\{ \begin{array}{l} \text{SINGle} \\ \text{MULTi} \end{array} \right\}$

Where,
SINGle   Set or return single channel mode.
MULTi   Set or return multi channel mode.
Example: WrtCmd("CORR:METH MULT")
                Set the instrument as multi channel mode.
Query syntax: CORRection;METHod?
Return format:

$\left\{ \begin{array}{l} \text{SINGle} \\ \text{MULTi} \end{array} \right\}$ <NL^END>

The CORRection: OPEN command is used to execute open correction for 41 preset test points.
Command Syntax:CORRection:OPEN
Example: WrtCmd("CORR:OPEN")

The CORRection: OPEN: STATe command is used to set the open correction ON or OFF. The CORRection :STATe ? Query returns the current open correction status.

Command Syntax:

CORRection:OPEN:STATe $\left\{ \begin{array}{l} \text{ON} \\ \text{OFF} \\ 1 \\ 0 \end{array} \right\}$

Where,
1(decimal 49) is equal to ON.
0(decimal 48) is equal to OFF.

Example: WrtCmd("CORR:OPEN:STAT ON")

Query syntax: CORRection:OPEN:STATe?
Return format:<NR1><NL^END>

The CORRection: SHORt command is used to execute short correction for 41 preset test points.
Command Syntax: CORRection:SHORt
Example: WrtCmd("CORR:SHOR")

The CORRection: SHORt: STATe command is used to set the short correction status .The CORRection :SHORt: STATe ? Query returns the current short correction status.
Command Syntax:

CORRection:SHORt:STATe $\left\{ \begin{array}{c} \text{ON} \\ \text{OFF} \\ 1 \\ 0 \end{array} \right\}$

Where,
1(decimal 49) is equal to ON.
0(decimal 48) is equal to OFF.
Example:WrtCmd("CORR:SHOR:STAT ON")

Query syntax: CORRection:SHORt:STATe?
Return format:<NR1><NL^END>

The CORRection: LOAD: STATe command is used to set load correction .The CORRection : LOAD: STATe ? Query returns the current load correction status.
Command Syntax:

CORRection:LOAD:STATe $\left\{ \begin{array}{c} \text{ON} \\ \text{OFF} \\ 1 \\ 0 \end{array} \right\}$

Where,
1(decimal 49) is equal to ON.
0(decimal 48) is equal to OFF.
Example: WrtCmd("CORR：LOAD:STAT ON")

Query syntax: CORRection:LOAD:STATe?
Return format: <NR1><NL^END>

The CORRection: LOAD: STATe command is used to set the tested parameter type.The CORRection :LOAD: STATe ? Query returns the current parameter type.
Command Syntax:
Details of function are as follows: Set the function as

CPD	Set the function asCp-D	LPRP	Set the function as Lp-Rp
CPQ	Set the function asCp-Q	LSD	Set the function as Ls-D
CPG	Set the function asCp-G	LSQ	Set the function as Ls-Q
CPRP	Set the function asCp-Rp	LSRS	Set the function as Ls-Rs
CSD	Set the function asCs-D	RX	Set the function as R-X
CSQ	Set the function asCs-Q	ZTD	Set the function as Z-θ°
CSRS	Set the function asCs-Rs	ZTR	Set the function as Z-θr
LPQ	Set the function asLp-Q	GB	Set the function as G-B
LPD	Set the function asLp-D	YTD	Set the function as Y-θ°
LPG	Set the function asLp-G	YTR	Set the function as Y-θr

Example: WrtCmd("CORR:LOAD:TYPE CPD")
Query syntax: ORRection:LOAD:TYPE?
Return format:<function><NL^END>

The CORRection: SPOT<n>: STATe command is used to set the state of some specific frequency spots.The CORRection :SPOT<n>: STATe ? Query returns the current state of each frequency spot(FREQ 1,FREQ 2 or FREQ 3).
Command Syntax:

$$\text{CORRection:SPOT<n>:STATe} \left\{ \begin{array}{c} \text{ON} \\ \text{OFF} \\ 1 \\ 0 \end{array} \right\}$$

Where,
1(decimal 49) is equal to ON.
0(decimal 48) is equal to OFF.
 <n>:
Frequency   1
Frequency   2
Frequency   3
Example:WrtCmd("CORR:SPOT1:STAT ON")

Query syntax: CORRection:SPOT<n>:STATe?
Return format:<NR1><NL^END>

The CORRection: SPOT<n>: FREQuency command is used to set the frequencies of specific frequency spots1,2,3. The CORRection :SPOT<n>: FREQuency ? Query returns the current frequency of specific frequency spot.

Command Syntax:

CORRection:SPOT<n>:FREQuency <value>

Where,

<value> can be NR1,NR2 or NR3 data format followed by HZ,KHZ and MHZ.

<n>:
  Frequency 1
  Frequency 2
  Frequency 3

Example: WrtCmd("CORR:SPOT1:FREQ 2KHZ")
          Set the frequency of requency spot 1 as 2KHZ

NOTE: <value> should be ranged from 20HZ to 200KHZ,or return format will report errors

Query syntax: CORRection:SPOT<n>:FREQuency?
Return format: <NR3><NL^END>

The CORRection :SPOT<n> :OPEN command is used to execute open correction for specific frequency spots (frequency1, frequency2 ,frequency3).

Query syntax: CORRection:SPOT<n>:OPEN
Where,
<n>:
Frequency 1
Frequency 2
Frequency 3

Example: WrtCmd("CORR:SPOT1:OPEN")
          Execute open correction for frequency spot 1.

The CORRection :SPOT<n> :SHORt command is used to execute short correction for specific frequency spots (frequency1, frequency2 ,frequency3).

Query syntax: CORRection:SPOT<n>:SHORt
 Where,
<n>:
Frequency 1
Frequency 2

Frequency 3

Example: WrtCmd("CORR:SPOT1:SHOR")
        Execute short correction for frequency spot 1.

The CORRection :SPOT<n> :LOAD :STANdard command is used to set the standard reference of specific frequency spots (frequency1, frequency2 ,frequency3)which are taking load correction. The CORRection :SPOT<n> :LOAD :STANdard? Query returns the current standard reference of specific frequency spot.
Query syntax:CORRection:SPOT<n>:LOAD:STANdard <REF.A>,<REF.B>
<n>:
Frequency 1
Frequency 2
Frequency 3

<REF.A>   can be NR1,NR2 or NR3 data format and taken as the standard
           reference of the primary parameter.
<REF.B>   can be NR1,NR2 or NR3 data format and taken as the standard
           reference of the secondary parameter.

Example: WrtCmd("CORR:SPOT1:LAOD:STAN 100.7,0.0002")

Query syntax: CORRection:SPOT<n>:LOAD:STANdard?
Return format: <NR3>,<NR3><NL^END>

The CORRection :USE command is used to set the number of channels for instrument multi-channel correction. The CORRection :USE? query returns the current number of channels in use.
Query syntax: CORRection:USE <channel number>
Where,
<channel number> is 1 to 127 （NR1,NR2 or NR3 data format).
Example: WrtCmd("CORR:USE 10")
          Set the number of channels to 10

Query syntax: CORRection:USE?
Return format:<channel number><NL^END>

The CORRection :USE:DATA? Query returns the OPEN/SHORT/LOAD correction measurement data of specific frequency spot 1,2 or 3.
Query syntax:CORRection:USE:DATA? <channel number>

Where,

<channel number> is 0 to 127(NR1)

Return format:

<open1 A>,<open1 B>,<short1 A>,<short1 B>,<load1 A>,<load1 B>,

<open2 A>,<open2 B>,<short2 A>,<short2 B>,<load2 A>,<load2 B>,

<open3 A>,<open3 B>,<short3 A>,<short3 B>,<load3 A>,<load3 B>,
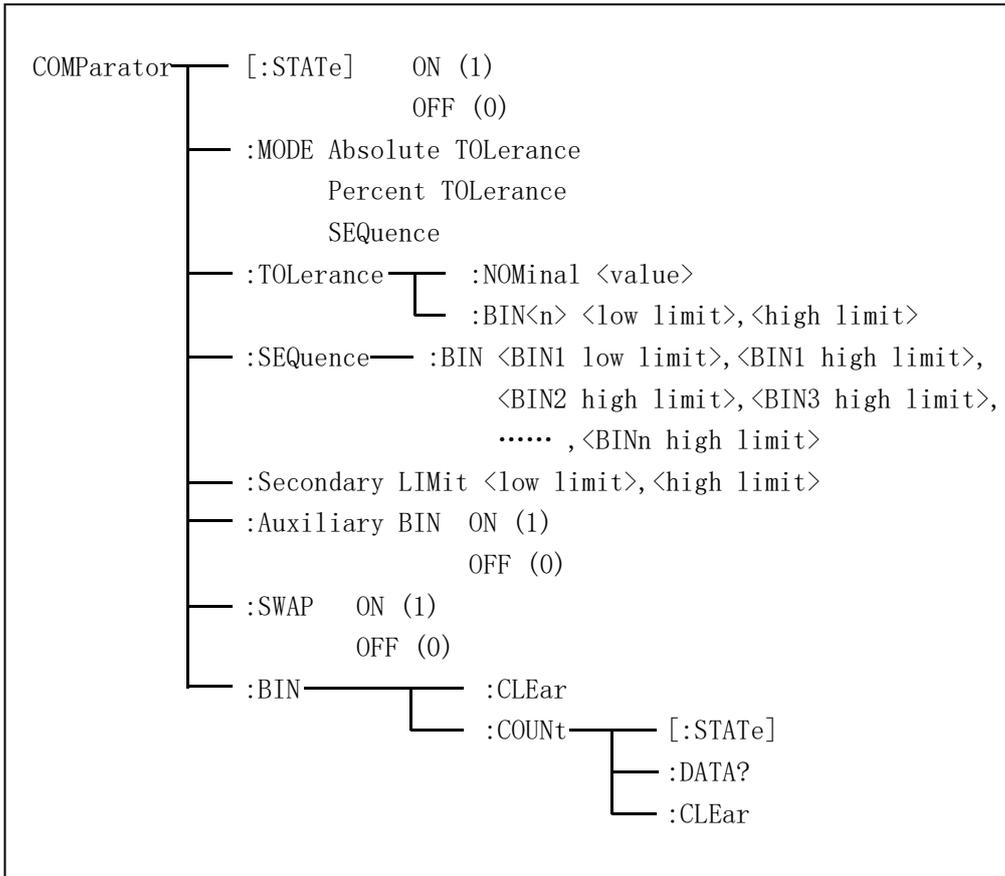
Where,

<open1/2/3 A>   is NR3 data format and the primary open correction data at frequency spot 1/2/3.

<open1/2/3 B>   is NR3 data format and the secondary open correction data at frequency spot 1/2/3.

<short1/2/3 A>   is NR3 data format and the primary short correction data at frequency spot 1/2/3.

<short1/2/3 B>   is NR3 data format and the secondary short correction data at frequency spot 1/2/3.

<load1/2/3 A>   is NR3 data format and the primary load correction data at frequency spot 1/2/3.

<load1/2/3 B>   is NR3 data format and the secondary load correction data at frequency spot 1/2/3.

COMParator subsystem commands

The COMParataor subsystem commands are used to set the bin comparator function including ON/OFF setting, limit table setting.

Command tree:

```
COMParator ──┬── [:STATe]    ON  (1)
             │                OFF (0)
             ├── :MODE Absolute TOLerance
             │          Percent TOLerance
             │          SEQuence
             ├── :TOLerance ──┬── :NOMinal <value>
             │                └── :BIN<n> <low limit>,<high limit>
             ├── :SEQuence ── :BIN <BIN1 low limit>,<BIN1 high limit>,
             │                     <BIN2 high limit>,<BIN3 high limit>,
             │                     ······ ,<BINn high limit>
             ├── :Secondary LIMit <low limit>,<high limit>
             ├── :Auxiliary BIN   ON  (1)
             │                    OFF (0)
             ├── :SWAP    ON  (1)
             │            OFF (0)
             └── :BIN ──┬── :CLEar
                        └── :COUNt ──┬── [:STATe]
                                     ├── :DATA?
                                     └── :CLEar
```

The COMParataor [STATe] commands is used to set the comparator function as ON or OFF. The COMParataor [STATe]? query returns the current comparator state.
Command Syntax:

$$
\text{COMParator[:STATe]} \left\{ \begin{array}{c} \text{ON} \\ \text{OFF} \\ 1 \\ 0 \end{array} \right\}
$$

Where,
1(decimal 49) is equal to ON.
0(decimal 48) is equal to OFF.
Example: WrtCmd("COMP ON")

Query syntax:COMParator[:STATe]?
Return format:<NR1><NL^END>

The COMParataor:MODE command is used to set the comparator mode
The COMParataor:MODE? query returns the current mode.

Command Syntax:

COMParator:MODE { ATOLerance / PTOLerance / SEQuence }

Where,
ATOLerance means absolute tolerance mode.
PTOLerance means proportional tolerance mode.
SEQuence   means sequential tolerance mode.
Example:WrtCmd("COMP:MODE ATOL")

Query syntax:COMParator:MODE?
Return format:

{ ATOL / PTOL / SEQ } <NL^END>

The COMParataor:Tolerance:NOMinal command is used to set the nominal Vale(this function is valid only when the limit mode is set as deviation mode). The COMParataor:Tolerance:NOMinal? query returns the current nominal mode.

Command Syntax:
COMParator:TOLerance:NOMinal <value>
Where,
<value> is a nominal value in NR1,NR2 or NR3 data format.
Example: WrtCmd("COMP:TOL:NOM 100E-12")

Query syntax:COMParator:TOLerance:NOMinal?
Return format:<NR3><NL^END>

The COMParataor:Tolerance:BIN<n> command is used to set the high and the Low Limits of each bin (this function is valid only when the limit mode is set as deviation mode).The COMParataor:Tolerance:BIN<n>? query returns the current nominal high and the low Limits of each bin.

Command Syntax:
COMParator:TOLerance:BIN<n><low limit>,<high limit>
Where,
<n> is the bin number from 1 to9
<low limit> is the low limit in NR1,NR2 or NR3 data format.
<high limit> is the high limit in NR1,NR2 or NR3 data format.

NOTE: The low limit should be smaller than the high limit,or error information will be reported.

Example: WrtCmd("COMP:TOL:BIN1 -5,5")
            WrtCmd("COMP:TOL:BIN2 -10,10")

Query syntax:COMParator:TOLerance:BIN<n>?
Return format:<low limit>,<high limit><NL^END>

The COMParator:SEQuence:BIN command is used to set the high and the Low Limits of sequential mode (this function is valid only when the limit mode is set as the sequential mode).The COMParataor:SEQuence:BIN? Query returns the current high and the low Limits of each bin.

Command Syntax:
COMParator:SEQuence:BIN <BIN1 low limit>,<BIN1 high limit>,<BIN2 high limit>,···, <BINn high limit>
Where,
<BIN1 low limit> is the low limit of BIN1 in NR1,NR2 or NR3 data format.
<BIN1 high limit>is the high limit of BIN1 in NR1,NR2 or NR3 data format.
<BINn high limit>is the high limit of BINn (the maximum of n is 9) in NR1,NR2 or NR3 data format.
NOTE: The low limit should be smaller than the high limit,or error information will be reported.

Example: WrtCmd("COMP:SEQ:BIN 10,20,30,40,50")

Query syntax:COMParator:SEQuence:BIN?
Return format:<BIN1 low limit>,<BIN1 high limit>,<BIN2 high limit>,···, <BINn high limit><NL^END>

The COMParator:Secondary LIMIT command is used to set the high and the Low Limits of the secondary parameter. The COMParator:Secondary LIMIT? Query returns the current high and the low Limits of the secondary parameter.
Command Syntax:
COMParator:SLIMit <low limit>,<high limit>
Where,
<low limit> is the low limit in NR1,NR2 or NR3 data format.
<high limit>is the high limit in NR1,NR2 or NR3 data format.
NOTE: The low limit should be smaller than the high limit,or error information will be reported.

Example: WrtCmd("COMP:SLIM 0.001,0.002")

Query syntax:COMParator:SLIMit?
Return format:<NR3>,<NR3><NL^END>

The COMParator:Auxiliary BIN command is used to set the Auxiliary bin as ON or OFF. The COMParator:Auxiliary BIN? query returns the current Auxiliary bin State.
Command Syntax:

COMParator:Auxiliary BIN $\left\{ \begin{array}{c} \text{ON} \\ \text{OFF} \\ 1 \\ 0 \end{array} \right\}$

Where,
1(decimal 49) is equal to ON.
0(decimal 48) is equal to OFF.
Example:WrtCmd("COMP:ABIN ON")

Query syntax:COMParator:Auxiliary BIN？
Return format:<NR1><NL^END>

The COMParator:SWAP command is used to set the swap mode ON or OFF. For example: the original function parameter is Cp-D,after the SWAP mode is set as ON, the function parameter will be changed as D-Cp. In this case, the Limits from BIN1 to BIN9 become the high and the low limits of D, the original Secondary limits become that of Cp. That is to say, this function is to make swap comparison between the primary and the secondary parameters. On the Contrary, If OFF is selected, the comparison will be made according to the original sequence. The COMParator:SWAP? Query returns the current state of the swap function.

Command Syntax:

COMParator:SWAP $\left\{ \begin{array}{c} \text{ON} \\ \text{OFF} \\ 1 \\ 0 \end{array} \right\}$

Where,
1(decimal 49) is equal to ON.
0(decimal 48) is equal to OFF.
Example:
WrtCmd("COMP:SWAP ON")

Query syntax:COMParator:SWAP?
Return format:<NR1><NL^END>

The COMParator:BIN: CLEar command is used to clear all limits on limit table setup page.

Command Syntax: COMParator:BIN:CLEar
Example:WrtCmd("COMP:BIN:CLE")

The COMParator: BIN: COUNt [:SYATe] command is used to set the bin count function as ON or OFF.The COMParator: BIN: COUNt [:SYATe] ? query returns the current state of the bin count function.
Command Syntax:

COMParator:BIN:COUNt[:STATe] $\left\{ \begin{array}{c} \text{ON} \\ \text{OFF} \\ 1 \\ 0 \end{array} \right\}$

Where,
1(decimal 49) is equal to ON.
0(decimal 48) is equal to OFF.
Example: WrtCmd("COMP:BIN:COUN ON")

Query syntax:COMParator:BIN:COUNt[STATe]?
Return format:<NR1><NL^END>

The COMParator: BIN: COUNt :DATA ? query returns the current comparison result of the bin count.
Query syntax: COMParator:BIN:COUNt:DATA?
Return format: <BIN1 count>,<BIN2 count>,···,<BIN9 count>,<OUT OF BIN count>,<AUX BIN count><NL^END>
Where,
<BIN1-9 count> is the count result of BIN1-9, in NR1 data format.
<OUT OF BIN count> is the count result of the OUT OF BIN,in NR1 data for format.
<AUX BIN count> is the count result of the auxiliary bin, in NR1 data format.

The COMParator: BIN: COUNt [:SYATe] command is used to clear all bin count results.
Query syntax: COMParator:BIN:COUNt:CLEar

Example: WrtCmd("COMP:BIN:COUN:CLE")

Mass MEMory subsystem commands

The Mass MEMory subsystem commands are used for file storing and load.
Command tree:

```
Mass MEMory ──┬── :LOAD ──── :STATe <record number>
              └── :STORe ─── :STATe <record number>
```
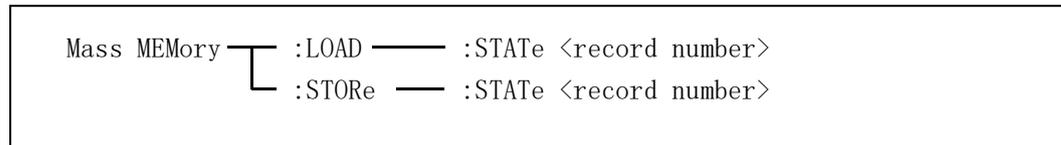
The MMEMory:LOAD:STATe commands is used to load the existed file .
Query syntax: MMEMory:LOAD:STATe <value>
Where,
<value>is the file number ranging from 0 to 9 (NR1).
Example: WrtCmd("MMEM:LOAD:STAT 1");

The MMEMory:STORe:STATe commands is used to storing the current setting to a file.
Query syntax: MMEMory:STOR:STATe <value>
Where,
<value>is the file number ranging from 0 to 9 (NR1).
Example: WrtCmd("MMEM:STOR:STAT 1");

TRAN subsystem commands

The TRAN subsystem commands are mainly used to set transformer parameters,such as turn-ratio, primary inductance, leakage inductance, DC resistance,working mode and limits of each parameter.(These commands are valid for some instruments)

Command tree:

```
TRAN ──┬── :TURN ──┬── :STAT ──┬── ON (1)
       │           │           └── OFF(0)
       │           ├── :FREQ <value>
       │           ├── :LEVEl <value>
       │           ├── :Np LIMit <value>,<low limit>,<high limit>
       │           └── :Ns LIMit <value>,<low limit>,<high limit>
       ├── :Lk ────┬── :STAT ──┬── ON (1)
       │           │           └── OFF(0)
       │           ├── :FREQ <value>
       │           ├── :LEVEl <value>
       │           └── :LIMIt <value>,<low limit>,<high limit>
       │
       ├── :Lx ────┬── :STAT ──┬── ON (1)
       │           │           └── OFF(0)
       │           ├── :FREQ <value>
       │           ├── :LEVEl <value>
       │           └── :LIMIt <value>,<low limit>,<high limit>
       │
       ├── :DCR ───┬── :STAT ──┬── ON (1)
       │           │           └── OFF(0)
       │           └── :LIMIt <value>,<low limit>,<high limit>
       └── :MODE ──┬── SEQ
                   └── STEP
```
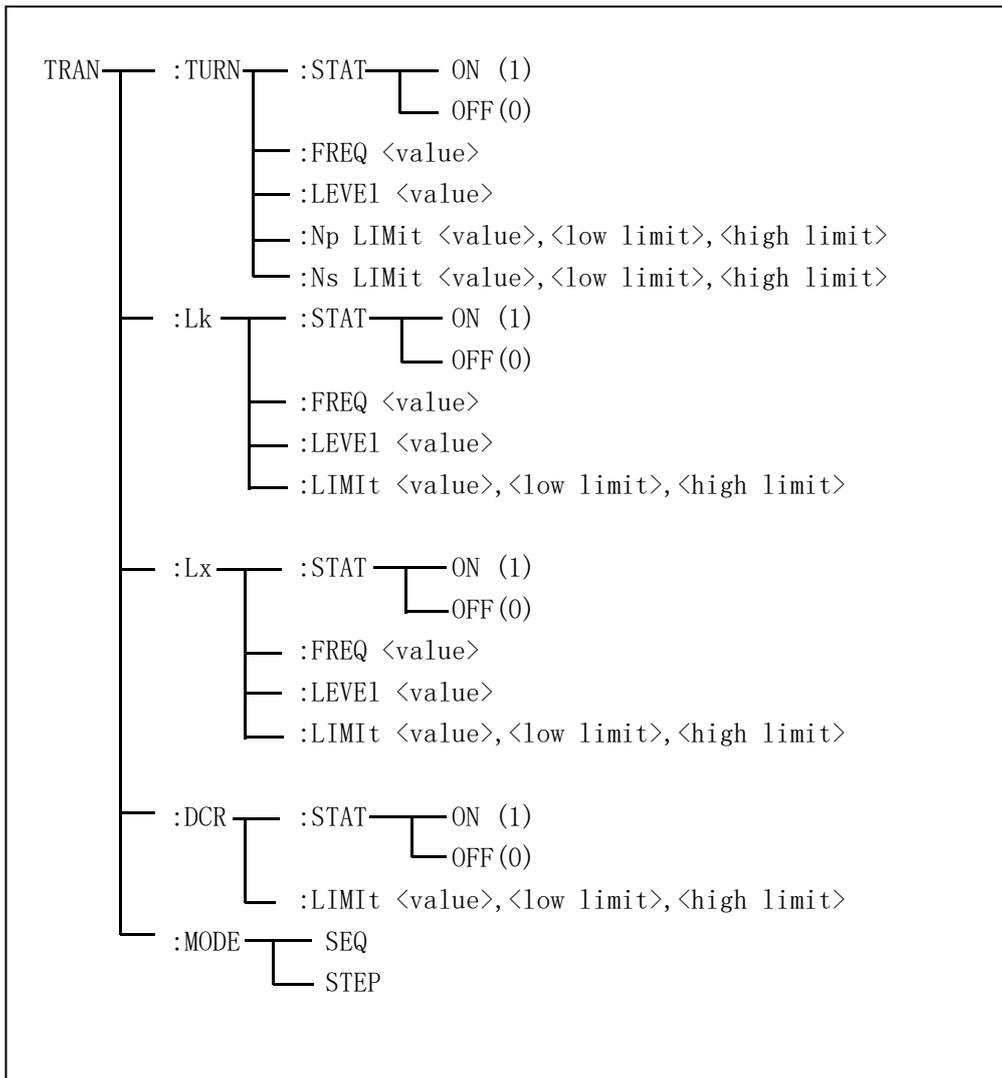
The TRAN:TURN:STAT command is used to set the turn-ratio ON or OFF.The TRAN:TURN:STAT? Query returns the current state of the turn-ratio.

Query syntax:

TRAN:TURN:STAT $\left\{ \begin{array}{c} \text{ON} \\ \text{OFF} \\ 1 \\ 0 \end{array} \right\}$

Where,
1(decimal 49) is equal to ON.
0(decimal 48) is equal to OFF.

Example: WrtCmd("TRAN:TURN:STAT:ON")
ALL test parameter setting of turn-ratio is valid.

Query syntax:TRAN:TURN:STAT?
Return format:<NR1><NL^END>

The TRAN:TURN:FREQ command is used to set the test frequency of turn-radio.The TRAN:TURN:FREQ? Query returns the current test frequency of turn-radio.

Query syntax:TRAN: TURN: FREQ <value>

Where,

<value> can be NR1,NR2 or NR3 data format followed by HZ,KHZ,and MHZ.

Example: WrtCmd("TRAN:TURN:FREQ 1KHZ")

    Set the test frequency of turn-radio as 1KHZ.

Query syntax:TRAN:TURN:FREQ?

Return format:<NR3><NL^END>

The TRAN:TURN:LEVEL command is used to set the test level of turn-radio.The TRAN:TURN:LEVEL? Query returns the current test level of turn-radio.

Command Syntax: TRAN:TURN:LEVEl <value>

Where,

<value> can be NR1,NR2 or NR3 data format followed by mV or V.

NOTE:<value> ranges from 5mV to 4V. Beyond this range, error information will be reported.

Example: WrtCmd("TRAN:TURN:LEVE 1V")

Query syntax:TRAN:TURN:LEVE?

Return format:<NR3><NL^END>

The TRAN:TURN:NP LImit command is used to set the nominal value,high and low limits of primary turn.The TRAN:TURN:NP? Query returns the current nominal value,high and low limits of primary turn.

Command Syntax: TRAN:TURN:NP LImit <value>,<low limit>,<high limit>

Where,

<value> is the primary nominal value in NR1,NR2 or NR3 data format.

<low limit>is the primary low limit in NR1,NR2 or NR3 data format.

<high limit>is the primary high limit in NR1,NR2 or NR3 data format.

NOTE:The low limit should be smaller than the high limit,or error information will be reported.

Example: WrtCmd("TRAN:TURN:NPLI 100,-0.01,0.01")

Query syntax:TRAN:TURN:NPLI?

Return format:<NR3>,<NR3>,<NR3><NL^END>

The TRAN:TURN:Ns Limit command is used to set the nominal value,high and low limits of primary turn.The TRAN:TURN:NS? Query returns the current nominal value,high and low limits of primary turn.

Command Syntax: TRAN:TURN:NS LImit <value>,<low limit>,<high limit>

Where,

<value> is the secondary nominal value in NR1,NR2 or NR3 data format.

<low limit>is the secondary low limit in NR1,NR2 or NR3 data format.

<high limit>is the secondary high limit in NR1,NR2 or NR3 data format.

NOTE:The low limit should be smaller than the high limit, or error information will be reported.

Example: WrtCmd("TRAN:TURN:NSLI 100,-0.01,0.01")

Query syntax:TRAN:TURN:NSLI?

Return format:<NR3>,<NR3>,<NR3><NL^END>

The TRAN:LK:STAT command is used to set the leakage inductance test parameter ON or OFF.The TRAN:LK:STAT? Query returns the current leakage inductance state.

Command Syntax:

$$\text{TRAN:Lk:STAT} \left\{ \begin{array}{c} \text{ON} \\ \text{OFF} \\ 1 \\ 0 \end{array} \right\}$$

Where,

1(decimal 49) is equal to ON.

0(decimal 48) is equal to OFF.

Example: WrtCmd("TRAN:Lk:STAT:ON")

   Set all leakage inductance test paramerers to be valid.

Query syntax:TRAN:Lk:STAT?

Return format:<NR1><NL^END>

The TRAN:LK:FREQ command is used to set the test frequency of the leakage inductance.The TRAN:LK:FREQ? Query returns the current test frequency of the leakage inductance.

Command Syntax:TRAN:Lk:FREQ <value>

Where,

<value> is NR1,NR2 or NR3 data format followed by HZ,KHZ, and MHZ.

Example:WrtCmd("TRAN:Lk:FREQ 1KHZ")

   Set the test frequency of the leakage inductance is 1KHZ.

Query syntax:TRAN:Lk:FREQ?
Return format:<NR3><NL^END>

The TRAN:LK:LEVEl command is used to set the test level of the leakage inductance.The TRAN:LK:LEVEl? Query returns the current test level of the leakage inductance.

Command Syntax:TRAN:Lk:LEVEl <value>
Where,
<value> is NR1,NR2 or NR3 data format followed by mV or V.
NOTE: <value> ranges from 5mv to 2V. beyond this range,error information will be reported.
Example:WrtCmd("TRAN:Lk:LEVE 1V")

Query syntax:TRAN:Lk:LEVE?
Return format:<NR3><NL^END>

The TRAN:LK:LIMIt command is used to set the LK nominal value, high and low limits. The TRAN:LK:LEVEl? Query returns the current LK nominal value, high and low limits.
Command Syntax:TRAN:Lk:LIMIt <value>,<low limit>,<high limit>
Where,
<value> is the LK nominal value in NR1,NR2 or NR3 data format followed by H.
<low limit> is the LK low limit in NR1,NR2 or NR3 data format.
<high limit> is the LK high limit in NR1,NR2 or NR3 data format.
NOTE:The low limit should be smaller than the high limit, or error information will be reported.
Example:WrtCmd("TRAN:Lk:LIMIt 0.01H,-0.01,0.01")

Query syntax:TRAN:Lk:LIMIt?
Return format:<NR3>,<NR3><NR3><NL^END>

The TRAN:Lx:STAT command is used to set the primary Lx ON or OFF.The TRAN:Lx:STAT? Query returns the current state of the primary Lx.
Command Syntax:

$$
\text{TRAN:Lx:STAT} \left\{ \begin{array}{c} \text{ON} \\ \text{OFF} \\ 1 \\ 0 \end{array} \right\}
$$

Where,

1(decimal 49) is equal to ON.

0(decimal 48) is equal to OFF.

Example:WrtCmd("TRAN:Lx:STAT:ON")   Set all Lx parameters to be valid.

Query syntax:TRAN:Lx:STAT?

Return format:<NR1><NL^END>

The TRAN:Lx:FREQ command is used to set the Lx test frequency.The TRAN:Lx:FREQ? Query returns the current Lx test frequency.

Command Syntax:TRAN:Lx:FREQ <value>

Where,

<value> is NR1,NR2 or NR3 data format followed by HZ,KHZ and MHZ.

Example:WrtCmd("TRAN:Lx:FREQ 1KHZ")   Set the Lx test frequency as 1KHZ

Query syntax:TRAN:Lx:FREQ?

Return format:<NR3><NL^END>

The TRAN:Lx:LEVEl command is used to set the Lx test level.

The TRAN:Lx:LEVEl? Query returns the current Lx test level.

Command Syntax:TRAN:Lx:LEVEl <value>

Where,

<value> can be NR1,NR2 or NR3 data format followed by mV or V.

NOTE: <value> ranges from 5mv to 2V. beyond this range, error information will be reported.

Example:WrtCmd("TRAN:Lx:LEVE 1V")

Query syntax:TRAN:Lx:LEVE?

Return format:<NR3><NL^END>

The TRAN:Lx:LIMIt command is used to set the Lx nominal value, high and low limits. The TRAN:LK:LEVEl? Query returns the current nominal value, high and low limits.

Command Syntax:TRAN:Lx:LIMIt <value>,<low limit>,<high limit>

Where,

<value> is the Lx nominal value in NR1,NR2 or NR3 data format followed by H.

<low limit> is the Lx low limit in NR1,NR2 or NR3 data format.

<high limit> is the Lx high limit in NR1,NR2 or NR3 data format.

NOTE:The low limit should be smaller than the high limit, or error

information will be reported.

Example:WrtCmd("TRAN:Lx:LIMI 0.01H,-0.01,0.01")

Query syntax:TRAN:Lx:LIMI?
Return format:<NR3>,<NR3><NR3><NL^END>

The TRAN:Lx:STAT command is used to set the Zx test parameter to be ON or OFF.The TRAN:Lx:STAT? Query returns the current state of the Zx test parameter.
Command Syntax:

$$
\text{TRAN：DCR：STAT} \left\{ \begin{array}{c} \text{ON} \\ \text{OFF} \\ 1 \\ 0 \end{array} \right\}
$$

Where,
1(decimal 49) is equal to ON.
0(decimal 48) is equal to OFF.
Example:WrtCmd("TRAN:DCR:STAT:ON")

Query syntax:TRANDCR:STAT?
Return format:<NR1><NL^END>

The TRAN:DCR:LIMIt command is used to set the DCR nominal value, high and low limits. The TRAN:DCR:LEVEl? Query returns the current DCR nominal value, high and low limits.
Command Syntax:
TRAN:DCR:LIMIt <value>,<low limit>,<high limit>
Where,
<value> is the DCR nominal value in NR1,NR2 or NR3 data format followed by Ω.
<low limit> is the DCR low limit in NR1,NR2 or NR3 data format.
<high limit> is the DCR high limit in NR1,NR2 or NR3 data format.
NOTE:The low limit should be smaller than the high limit, or error information will be reported.
Example:WrtCmd("TRAN:DCR:LIMI 50Ω,-0.01,0.01")

Query syntax:TRAN:DCR:LIMI?
Return format:<NR3>,<NR3><NR3><NL^END>

The TRAN:MODE is used to set the transformer working mode.The TRAN:MODE? Query returns the current working mode of the transformer.
Command Syntax:

TRAN:MODE $\left\{\begin{array}{l} \text{SEQuence} \\ \text{STEPped} \end{array}\right\}$

Where,
SEQuence refers to sequence mode.
STEPped refers to single step mode.
Example:WrtCmd("TRAN:MODE SEQ")

Query syntax:TRAN:MODE?
Return format:

$\left\{\begin{array}{c} \text{SEQ} \\ \text{<NL^END>} \\ \text{STEP} \end{array}\right\}$

FRSEtup subsystem commands

The FRSEtup subsystem commands are used to set the parameters of frequency response curve analysis, set the starting frequency, ending frequency and the display range of the primary and secondary parameters of the measured parts(To enable the frequency response curve of the tested parts to be fully and moderately displayed in the display screen) .
Command tree:

```
FRSEtup ──────── :FREQuency <start>,<stop>

            ├──── :A <min>,<max>

            ├──── :B <min>,<max>

            ├──── :COORdinate   linear
            │                   log
            ├──── :SCALe   AUTO
            │              hold
            └──── :POINt   120
                           240
                           480
                           960
```
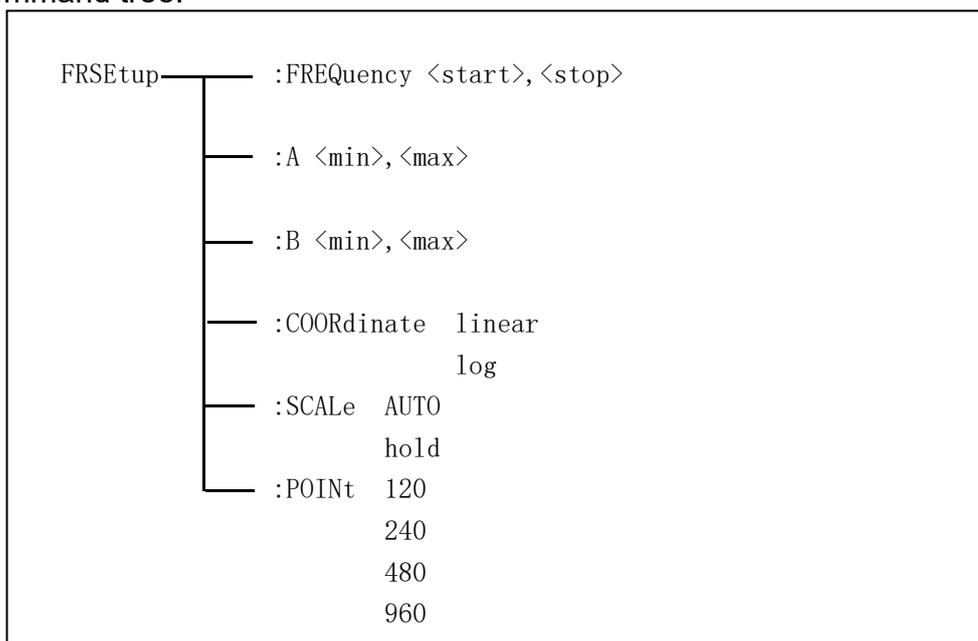
The FRSEtup:FREQuency is used to set the starting frequency, ending frequency of the frequency response test.The FRSEtup:FREQuency? Query returns the current starting frequency, ending frequency of the instrument.
Command Syntax:FRSEtup:FREQuency <start>,<stop>
Where,
<start> is NR1,NR2 or NR3 data format followed by HZ,KHZ and MHZ,sweeping start frequency.
<stop> is NR1,NR2 or NR3 data format followed by HZ,KHZ and MHZ,sweeping end frequency.
Example:WrtCmd("FRSE:FREQ 20HZ,300KHZ")

Query syntax:FRSEtup:FREQuency?
Return format:<NR3>,<NR3><NL^END>
NOTE: Sweep frequency start and end frequency should be between 20HZ~300KHZ,or error information will be reported.

The FRSEtup:A is used to set the high and low limits allowed for the primary parameter drawing. The FRSEtup:A? Query returns the current high and low limits allowed for the primary parameter drawing.
Command Syntax:FRSEtup:A <min>,<max>
Where,
<min> is NR1,NR2 or NR3 data format ,the low limits allowed to be displayed when primary parameter drawing.
<max> is NR1,NR2 or NR3 data format ,the high limits allowed to be displayed when primary parameter drawing.
Example:WrtCmd("FRSE:A 0.00000,10.0000")

Query syntax:FRSEtup:A?
Return format:<NR3>,<NR3><NL^END>

The FRSEtup:B is used to set the high and low limits allowed for the secondary parameter drawing. The FRSEtup:B? Query returns the current high and low limits allowed for the secondary parameter drawing.
Command Syntax:FRSEtup:B <min>,<max>
Where,
<min> is NR1,NR2 or NR3 data format ,the low limits allowed to be displayed when secondary parameter drawing.
<max> is NR1,NR2 or NR3 data format ,the high limits allowed to be displayed when secondary parameter drawing.
Example:WrtCmd("FRSE:B 0.00000,10.0000")

Query syntax:FRSEtup:B?

Return format:<NR3>,<NR3><NL^END>

The FRSEtup:COORdinate is used to set the scanning mode of the frequency response curve. The FRSEtup:COORdinate? Query returns the current scanning mode of the frequency response curve.
Command Syntax:FRSEtup:COORdinate <mode name>
Where,
<mode name>   The scan mode consists of linear and log
linear   Linear scan mode
log   Logarithmic   scan mode
Example:WrtCmd("FRSE:COOR linear")

Query syntax:FRSEtup:COORdinate?
Return format:<mode name><NL^END>

The FRSEtup:SCALe is used to set the scope of the display drawing.The FRSEtup:SCALe? Query returns the current mode of the display drawing.
Command Syntax:FRSEtup:SCALe <mode name>
Where,
<mode name>   The mode of the display drawing scope consists of linear
                      and AUTO and HOLD
AUTO   Automatically changes the display category mode, after a measurement, the system automatically corrects the display range of the graphics to be able to display the full range of graphics.
HOLD   Lock the way to change the display category mode, and when the display high and low limits of the primary and secondary parameters are set,the drawing category is the display high and low limit, the drawing may be incomplete.
Example:WrtCmd("FRSE:SCAL AUTO")

Query syntax:FRSEtup:SCALe?
Return format:<mode name><NL^END>

The FRSEtup:POINt is used to set the sweep points.The FRSEtup:POINt? Query returns the current sweep points.
Command Syntax:FRSEtup:POINt <number>
Where,
<number>   The sweep points consists of 120,240,480,960

Example:WrtCmd("FRSE:POIN 240")

Query syntax:FRSEtup:POINt?
Return format:<number><NL^END>

GPIB common command

● *RST   ● *TRG   ● *IDN   ● *TST
● *ESE   ● *SRE   ● *ESR   ● *STB
● *OPC   ● *CLS

- The *RST command resets the instrument.
  Command Syntax:*RST
  Example:WrtCmd("*RST");

- The *TRG command triggers the measurement and sends the result to the output buffer.
  Command Syntax:*TRG
  Example:WrtCmd("*TRG");

- The *CLS command clears the standard event status register and the service request status register.
  Command Syntax:*CLS
  Example:WrtCmd("*CLS");

- The *IDN? Query returns the instrument.
  Query syntax:*IDN?
  Return format:
  <manufacturer>,<model>,<firmware><NL^END>
  Where,
  <manufacturer>   Name of Manufacturer
  <model>   Instrument Model
  <firmware>   Firmware Version
  Example:WrtCmd("*IDN?");

- The *TST? Query executes an internal self test and returns the test result as the sum of all existing errors codes. If there are no error the instrument returns 0.
  Query syntax:*TST?
  Return format:0<NL^END>
  Where,
  0（NR1 format）
  Example:WrtCmd（"*TST?"）;

- The *ESE (standard Event Status Enable command) command sets each open bit of the standard event status register. This command returns setups of each open bit for the standard event status permission register.
  Query syntax:*ESE<value>
  Where,
  <value>   NR1 format：decimal expression for each bit of operation status register.
  Descriptions for each byte of the standard event status register are shown as follows:

Bit number	Description
7	Power On(PON) Bit
6	User Request(URQ) Bit
5	Command Error(EME) Bit
4	Execution Error(EXE) Bit
3	Device Dependent Error(DDE) Bit
2	Query Error(QYE) Bit
1	Request Control(RQC) Bit
0	Operation Complete(OPC) Bit

Query syntax:*ESE?
Return format:<value><NL^END>
Example:WrtCmd("*ESE?");

- The *SRE （Service Request Enable command）command sets each open bit of the service status byte register. This command returns the current setups for each open bit of the status byte permission register.

  Query syntax:*SRE<value>

  Where,

  <value>　NR1 format：decimal expression for each permission bit of the status byte register.

  Descriptions for each byte of the status byte register are shown as follows:

Bit number	Description
7	Operation Status Register Summary Bit
6	RQS(Request Service) Bit
5	Standard Event Status Register Summary
4	BitMAV(Message Available) Bit
3-0	Always 0(zero)

  Query syntax:*SRE?

  Return format:<value><NL^END>

  Example:WrtCmd("*SRE?");

- The *ESR? Query returns the contents of the standard event status register.

  Query syntax:*ESR?

  Return format:<value><NL^END>

  Where,

  <value>　NR1 format：decimal expression for each bit of operation status register.

  Descriptions for each byte of the standard event status register are shown as follows:

Bit number	Description
7	Power On(PON) Bit
6	User Request(URQ) Bit
5	Command Error(EME) Bit
4	Execution Error(EXE) Bit
3	Device Dependent Error(DDE) Bit
2	Query Error(QYE) Bit
1	Request Control(RQC) Bit
0	Operation Complete(OPC) Bit

  Example:WrtCmd("*ESR?");

- The *STB? Query returns contents of the standard service status byte register. The execution of this command will not affect contents of the standard status byte register.

  Query syntax:*STB?

  Return format:<value><NL^END>

  Where,

  <value>　NR1 format：decimal expression for contents of the standard status byte register.

  Descriptions for each bit of the standard status byte register

Bit number	Description
7	Operation Status Register Summary Bit
6	RQS(Request Service) Bit
5	Standard Event Status Register Summary
4	BitMAV(Message Available) Bit
3-0	Always 0(zero)

　Example:WrtCmd("*STB?");

- The *OPC command equals to set the OPC bit of the standard event status register when the instrument finishes all parameter measurements. Ever since all pending operations has been completed, this command will inform the instrument to add a ASCⅡ number "1"(decimal number: 49) into the output buffer.

  Command Syntax:*OPC

  Example:OUTPUT 717;"*OPC"!　Set the OPC bit of the instrument when the last command is done.

  Query syntax:*OPC?

  Return format:1<NL^END>

  Where,

  1　ASCⅡ number 1（decimal number:49）

  Example:WrtCmd("*OPC?")