



TOPSPIN

Users Guide

Copyright (C) 2005 by Bruker BioSpin GmbH

All rights reserved. No part of this publication may be reproduced, stored in a retrieval system, or transmitted, in any form, or by any means without the prior consent of the publisher.

Part Number H9469SA1 V2/January 31st 2005

Product names used are trademarks or registered trademarks of their respective holders.

Bruker software support is available via phone, fax, e-mail, Internet, or ISDN.
Please contact your local office, or directly:

Address: Bruker BioSpin GmbH
Software Department
Silberstreifen
D-76287 Rheinstetten
Germany

Phone: +49 (721) 5161 455
Fax: +49 (721) 5161 943
E-mail: nmr-software-support@bruker-biospin.de
FTP: <ftp.bruker.de> / <ftp.bruker.com>
WWW: www.bruker-biospin.de / www.bruker-biospin.com

Chapter 1	Getting Started	15
1.1	Document Conventions	15
	Font Conventions	15
	File/directory Conventions	15
	User Action Conventions	15
1.2	TOPSPIN Overview	16
	Functionality	16
	Available Documentation	17
1.3	TOPSPIN license	17
1.4	Startup TOPSPIN	18
1.5	Configuration	18
1.6	How to Display Spectra	19
	How to Open Data from the Menu	19
	How to Open Data from the Browser	20
	How to Define Alias Names for Data	20
	How to Open Data in Other Ways	20
1.7	How to Display Peaks, Integrals, ... together with the Spectrum	20
1.8	How to Display Projections/1D Spectra with 2D Spectra	21
1.9	How to Superimpose Spectra in Multiple Display	21
1.10	How to Print or Export the Contents of a Data Window	22
	How to Print Data	22
	How to Copy a Data Window to Clipboard	22
	How to Store (Export) a Data Window as Graphics File	22
1.11	How to Process Data	22
1.12	How to Archive Data	23
1.13	How to Import NMR Data Stored in Special Formats	24
1.14	How to Fit Peaks and Deconvolve Overlapping Peaks	24
1.15	How to Compute Fids by Simulating Experiments	24
1.16	How to Add Your Own Functionalities	25
	How to Create Macros	25
	How to Create AU (automation) Programs	25
	How to Create Python Programs	26
1.17	How to Automate Data Acquisition	26
Chapter 2	The TOPSPIN Interface	27
2.1	The Topspin Window	27
	How to Use Multiple Data Windows	28
	How to Use the Title bar	29
	How to Use the Menu bar	29
	How to Use the Upper Toolbar	30
	How to Use the Lower Toolbar	31
2.2	Command Line Usage	33

	How to Put the Focus in the Command Line	33
	How to Retrieve Previously Entered Commands	33
	How to Change Previously Entered Commands	33
	How to Enter a Series of Commands	34
2.3	Command Line History	34
2.4	Starting TOPSPIN commands from a Command Prompt	35
2.5	Function Keys and Control Keys	36
2.6	Help in Topspin	39
	How to Open Online Help documents	39
	How to Get Tooltips	39
	How to Get Help on Individual Commands	40
	How to Use the Command Index	40
2.7	User Defined Functions Keys	41
2.8	How to Open Multiple TOPSPIN Interfaces	41
Chapter 3	Trouble Shooting	43
3.1	General Tips and Tricks	43
3.2	History, Log Files, Stack Trace	43
3.3	How to Show or Kill TOPSPIN processes	47
3.4	What to do if TOPSPIN hangs	47
3.5	How to Restart User Interface during Acquisition	48
Chapter 4	Dataset Handling	49
4.1	The Topspin Browser and Portfolio	49
	How to Open the Browser/Portfolio	51
	How to Open the Browser/Portfolio in a separate window	51
	How to Put the Focus in the Browser/Portfolio	51
	How to Select Folders in the Browser	51
	How to Expand/Collapse a Folder in the Browser	52
	How to Expand a Folder showing Pulse program and Title	52
	How to Add/Remove a Top Level Data Directory	53
	How to Open a New Portfolio	53
	How to Save the current Portfolio	54
	How to Remove Datasets from the Portfolio	54
	How to Find Data and Add them to the Portfolio	54
	How to Sort Data in the Portfolio	54
	How to Add, Remove or Interpret Alias Names	55
4.2	Creating Data	56
	How to Create a New Dataset	56
4.3	Opening Data	57
	How to Open Data Windows Cascaded	58
	How to Open Data from the Browser	59
	How to Open Data from the Portfolio	60

	How to Automatically Select the first <i>expno/procno</i> of a dataset	60
	How to Open Data from the Topspin menu	61
	How to Open Data from the Explorer, Konqueror or Nautilus	63
	How to Open Data from the Command Line	64
	How to Open Special Format Data	65
	How to Open a ZIP or JCAMP-DX file from the Windows Explorer	66
4.4	Saving/Copying Data	66
	How to Save or Copy Data	66
	How to Save an Entire Dataset	67
	How to Save Processed Data	68
	How to Save Acquisition Data	68
	How to Save Processed Data as Pseudo Raw Data	68
4.5	Deleting Data	68
	How to Delete a Specific Dataset	68
	How to Delete Types of Datasets	69
4.6	Searching/Finding Data	71
	How to Find Data	71
	How to Display one of the Found Datasets	73
	How to Select Data from the Found Datasets	73
	How to Add Selected Datasets to the Portfolio	74
	How to Save Selected Datasets to a List	74
4.7	Handling Data Files	74
	How to List/Open the Current Dataset Files	74
	How to List/Open the current Dataset Files in the Windows Explorer	75
4.8	Data Consistency Check	76
Chapter 5	Parameter Handling	79
5.1	Processing Parameters	79
	How to Set a Processing Parameter from the Command Line	79
	How to Set Processing Parameters from the Parameter Editor	80
	How to Undo the Last Processing Parameter Change	81
	How to Display Processing Status Parameters	81
	How to Change Processed Data Dimensionality	81
5.2	Acquisition Parameters	82
	How to Set Acquisition Parameters	82
	How to Set an Acquisition Parameter from the Command Line	82
	How to Set Acquisition Parameters from the Parameter Editor	83
	How to Undo the Last Acquisition Parameter Change	84
	How to Set Pulse Program Parameters	84
	How to Display Acquisition Status Parameters	84
	How to Get Probehead/Solvent dependent Parameters	84
	How to Change Acquisition Data Dimensionality	84

	How to Set Lock Parameters	85
	How to Set Routing Parameters	85
Chapter 6	Data Processing	87
6.1	Interactive Processing	87
	How to Process Data with Single Commands	87
	How to Process data with Composite Commands	88
6.2	Semi-automatic Processing	88
	How to Use the Processing Guide in <i>Automatic</i> mode	88
	How to Use the Processing Guide in <i>Interactive</i> mode	90
6.3	Processing Data with AU programs.	90
6.4	Serial Processing using Python programs	91
Chapter 7	Printing/Exporting Data	95
7.1	Printing/plotting Data.	95
	How to Print/Plot from the Menu	95
	How to Plot Data from the Processing guide	97
	How to Plot Data with the Plot Editor	97
	How to Print the Integral list	97
	How to Print the Peak list	98
7.2	Exporting Data	99
	How to Copy data to Other Applications	99
	How to Store (Export) a Data Window as Graphics File	99
Chapter 8	1D Display	101
8.1	The 1D Data Window	101
8.2	Displaying one Dataset in Multiple windows	102
	How to Reopen a Dataset in a Second/Third etc. Window	102
	How to Rescale or Shift one Dataset in Multiple windows	103
8.3	Changing the Display of a 1D Spectrum or FID	104
	How to Change the Vertical Scaling of the FID or Spectrum	104
	How to Change the Horizontal Scaling of the FID or Spectrum	104
	How to Shift a Spectral Region to the Left or to the Right	105
	How to Shift the Spectrum Up or Down	106
8.4	Using the Tab bar	106
	How to Display the Spectrum	107
	How to Set Processing Parameters	107
	How to Set Acquisition Parameters	108
	How to Edit the Title	109
	How to Edit the Pulse Program	109
	How to Display the Peak list	110
	How to Display the Integral list	115
	How to view Sample Information	121
	How to Open the Jmol Molecule Structure Viewer	122

	How to Display the FID	124
8.5	1D Display Options	124
	How to Toggle between Hertz and ppm Axis Units	124
	How to Switch on/off the Spectrum Overview display	124
	How to Switch Y-axis Display	125
8.6	Show Display Properties/Regions/Files.....	126
	How to Superimpose the Cursor Information	127
	How to Superimpose the Title on the Spectrum	127
	How to Superimpose the main Status Parameters on the Spectrum ..	127
	How to Superimpose the Integral Trails/Labels on the Spectrum	127
	How to Superimpose Peak Labels on the Spectrum	127
	How to Show Individual Data Points of the Spectrum	127
	How to Display the Main Dataset Properties	128
	How to Display a List of Files of a Dataset	128
8.7	Saving Display Region	130
	How to Save the Display Region for Re-display	131
	How to Save the Display Region for Baseline or Phase Correction ...	131
	How to Save the Display Region for Strip FT	131
Chapter 9	2D Display	133
9.1	The 2D Data Window	133
9.2	Changing the Display of a 2D spectrum	134
	How to Change the Intensity Scaling	134
	How to Switch on/off Square 2D layout	135
	How to Zoom a 2D spectrum in/out	136
	How to Shift a Spectral Region in the F2 direction (left/right)	137
	How to Shift a Spectral Region in the F1 direction (up/down)	137
9.3	Show Display Properties/Regions/Files.....	138
9.4	Using the Tab bar.....	139
	How to Set Processing Parameters	140
	How to Set Acquisition Parameters	140
	How to Display the Peak list	141
	How to Display the Integral list	142
	How to Display the FID	143
9.5	2D Display Options	144
	How to Switch between Hertz and ppm Axis Units in F2 and F1	144
	How to Switch on/off the Spectrum Overview display	144
	How to Switch on/off the Projection display	145
	How to Switch on/off the Grid display	147
	How to Display a 2D Spectrum in Contour Mode	148
	How to Set the 2D Contour Levels	149
	How to Store interactively set Contour Levels	150

	How to Display a 2D spectrum in Image Mode	151
	How to Display a 2D Spectrum in Oblique Mode	151
	How to Rotate a 2D Spectrum in Oblique Mode	153
	How to Switch between Displaying Positive and Negative levels	153
Chapter 10	3D Display	155
10.1	Plane Display Mode	155
	How to Switch to 2D Plane Display	156
	How to Display various Plane Orientations	156
	How to Display various Plane Positions (numbers)	157
10.2	Cube Display Mode	157
	How to Display the 3D Cube	157
	How to Rotate the 3D Cube	158
	How to Scale Up/Down the 3D Cube	158
	How to Reset the Cube Size and Orientation	158
	How to Switch Depth Cueing on/off	159
	How to Display a Cube Front or Side view	159
10.3	Using the Tab bar	159
Chapter 11	1D Interactive Manipulation	161
11.1	1D Interactive Phase Correction	161
	How to Switch to Phase Correction Mode	161
	How to Perform a Typical 1D Interactive Phase Correction	162
	How to Set the Phase Pivot Point	162
	How to Perform Default Zero Order Phase Correction	163
	How to Perform Interactive Zero Order Phase Correction	163
	How to Perform Interactive First Order Phase Correction	163
	How to Perform 90, -90 or 180° Zero Order Phase Correction	163
	How to Reset the Phase to the Original Values	163
	How to Change the Mouse Sensitivity	163
	How to Return from Phase Correction Mode with/without Save	164
11.2	1D Interactive Integration	164
	How to Switch to Integration Mode	165
	How to Define Integral Regions	166
	How to Select/Deselect Integral Regions	166
	How to Read Integral Regions from Disk	167
	How to Perform Interactive Bias and Slope Correction	168
	How to Set the Limit for Bias Determination	169
	How to Change the Mouse Sensitivity	169
	How to Calibrate/Normalize Integrals	169
	How to Scale Integrals with respect to Different Spectra	170
	How to Delete Integral Regions from the Display	170
	How to Scale Selected Integrals	171

	How to Move the Integral Trails Up/Down	172
	How to Cut Integral Regions	172
	How to Save Integral Regions	172
	How to Undo the Last Region Operation	173
	How to Return from the Integration Mode with/without Save	173
11.3	1D Interactive Calibration	173
	How to Switch to Calibration Mode	173
	How to Calibrate a Spectrum Interactively	174
11.4	1D Multiple Display	175
	How Switch to Multiple Display Mode and Read Multiple Spectra	175
	How to Select/Deselect Datasets	178
	How to Remove a Dataset from Multiple Display	178
	How to Display the Sum or Difference Spectra	179
	How to Save the Sum or Difference Spectra	179
	How to Display the Next/Previous Name/Expno	179
	How to Toggle between Superimposed and Stacked Display	180
	How to Shift and Scale Individual Spectra	180
	How to Switch on/off the Display of Datapaths and Scaling Factors	181
	How to Return from Multiple Display mode	182
	How to Set the Colors of the 1 st , 2 nd , .. Dataset	182
11.5	1D Interactive Baseline Correction	182
	How to Switch to Baseline Correction Mode	182
	How to Perform Polynomial Baseline Correction	183
	How to Perform Sine Baseline Correction	183
	How to Perform Exponential Baseline Correction	184
	How to Preview the Baseline Corrected Spectrum	184
	How to Reset the Baseline Correction Line	185
	How to Change the Mouse Sensitivity	185
	How to Save the Baseline Correction and/or Return	185
	How to Perform Cubic Spline Baseline correction	185
	How to Delete Spline Baseline Points from the screen	186
	How to Return from Cubic Spline Baseline mode with/without Save	187
11.6	1D Interactive Peak Picking	187
	How to Switch to Peak Picking Mode	188
	How to Define New Peak Picking Ranges	188
	How to Change Peak Picking Ranges	189
	How to Pick Peaks in Peak Picking Ranges only	189
	How to Delete all Peak Picking Ranges	189
	How to Define Peaks Manually	190
	How to Pick Peaks Semi-Automatically	190
	How to Delete Peaks from the Peak List	190

	How to Return from Peak Picking Mode with/without Save	191
Chapter 12	2D Interactive Manipulation	193
12.1	2D Interactive Phase Correction	193
	How to Switch to 2D Interactive Phase Correction	193
	How to Perform a Typical 2D Interactive Phase Correction	194
	How to Scale or Shift Individual Rows/Columns	197
	How to Perform Smooth Phase Correction	198
	How to Perform 90, -90 or 180° Zero Order Phase Correction	199
	How to Reset the Phase to the Original Values	199
	How to Change the Mouse Sensitivity	199
	How to Show the Next/Previous Row or Column	199
	How to Arrange Rows or Columns	200
	How to Return from Multi-1D Phase to 2D Phase Display	200
	How to Return from 2D Phase Mode	200
12.2	2D Multiple Display and Row/Column Handling	200
	How Switch to Multiple Display mode and Read Multiple Spectra	201
	How to Align Multiple 2D Spectra	203
	How to Scan Rows/Columns	203
	How to Grab a Row/Column	204
	How to Extract a Row/Column	205
	How to Copy Contour Levels from First to Other Spectra	205
	How to Switch on/off 2D contour display	205
	How to Position the Baseline of the Row/Column	206
12.3	2D Interactive Calibration	206
	How to Switch to 2D Calibration mode	206
	How to Perform 2D Calibration	207
12.4	2D Chemical Shift Distance Measurement	208
	How to Measure a 2D Chemical Shift Distance	208
12.5	2D Integration	208
Chapter 13	Data Window Handling	211
13.1	Data Windows	211
	How to Move a Data Window	212
	How to Resize a Data Window	212
	How to Select (activate) a Data Window	213
	How to Open a New empty Data Window	214
	How to Arrange Data Windows	214
	How to Iconify (minimize) a Data Window	216
	How to De-iconify a Data Window	217
	How to Maximize a Data Window	217
	How to Restore the Size and Position of a Data Window	217
	How to Close a Data Window	217

	How to Iconify all Data Windows	218
	How to Maximize all Data Windows	218
	How to Activate the Next Data Window	218
13.2	Window Layouts	218
	How to Save the Current Window Layout	218
	How to Read a Window Layout	218
	How to Swap Data Windows	219
Chapter 14	Analysis	221
14.1	1D Chemical Shift Distance Measurement	221
	How to Measure a Chemical Shift Distance	221
14.2	1D Signal to Noise Calculation	222
	How to Perform Interactive S/N Calculation	222
	How to Delete the Signal Region or Noise Region	223
	How to Edit the Limits of the Signal Region or Noise Region	223
	How to Change the Width of the Signal Region or Noise Region	224
14.3	Solids Line Shape Analysis	225
	Spectrum Preparation	225
	Switch to Line Shape Analysis Mode	225
	The simulation procedure	227
	Simulation details	232
14.4	Relaxation Analysis	235
14.5	Multiplet Analysis	241
	Selecting a Multiplet	247
	Selecting a Level in a Multi-Level Multiplet	247
	Designating a Level in Multi-Level Multiplets	248
	Save and close	253
Chapter 15	Acquisition	255
15.1	Acquisition Guide	255
15.2	Acquisition Toolbar	257
15.3	Acquisition Status Bar	259
15.4	Tuning and Matching the Probehead	261
15.5	Locking	263
15.6	BSMS Control Panel	264
15.7	Interactive Parameter Adjustment (GS)	266
15.8	Running an Acquisition	268
15.9	Shape tool	272
Chapter 16	Configuration/Automation	299
16.1	NMR Superuser and NMR Administration password	299
	How to Change the NMR Administration Password	300
16.2	Configuration	300
	How to Perform a Default Configuration on a Datastation	301

	How to Perform a Customized Configuration on a Datastation	301
16.3	Parameter set conversion	302
16.4	Automation	303
	How to Install AU Programs	303
	How to Open the AU Program Dialog Box	303
	How to Switch to the List of User defined AU Programs	303
	How to Switch to the List of Bruker defined AU Programs	304
	How to Create an AU Program	304
	How to Edit an Existing AU Program	304
	How to Execute an AU Program	304
	How to Delete an AU Program	304
	How to Show Comments (short descriptions) in the AU Program List	305
Chapter 17	Remote Control	307
17.1	Remote control	307
17.2	How to Establish a Remote Connection from your PC	307
17.3	How to Configure TCP/IP Communication Ports	309
17.4	How to Make a Remote Connection without a Local License	310
17.5	Security of Remote Connections	310
17.6	How to Allow Remote Connections	311
17.7	Note to Windows XP Service pack 2	312
17.8	How to Access ICON-NMR from a Remote Web Browser	312
Chapter 18	User Preferences	313
18.1	User Preferences	313
	How to Define the Startup Dataset	314
	How to Set Automatic Startup Actions	315
	How to Change the Preferred Editor	315
	How to Configure the Tab Bar	316
	How to Configure the Right-click Menu Function	316
18.2	Changing Colors	317
	How to Change Colors of Data Objects on the Screen	317
	How to Change Colors of Data Objects on the Printer	317
	How to Change Colors of the Lock Display	317
	How to Create a New Data Window Color Scheme	318
	How to Read a Different Data Window Color Scheme	318
18.3	Changing Fonts	319
	How to Change All Fonts of the Topspin Interface	319
	How to Change the Font of the TOPSPIN menu	320
	How to Change the Font of the Tab bar	320
	How to Change the Font of Dialog Boxes	321
	How to Change the Font of the Browser	321
	How to Change the Font of the Command Line	322

	How to Change the Font of the Status Line	322
18.4	Command Line Preferences	322
	How to Resize the Command Line	322
	How to Set the Minimum and Maximum Command Line Size	323
18.5	Disabling/Enabling Toolbar Buttons, Menus and Commands	323
	How to Hide the Upper and Lower Toolbars	323
	How to Hide the Menubar	324
	How to Disable/Remove Toolbar Buttons	324
	How to Disable/Remove Menus or Commands	324
	How to (Re)enable a disabled Command/Menu	326
	How to (Re)enable All Commands/Menus	327
18.6	Resizing/Shifting Toolbar Icons	327
	How to Change the Toolbar Icon Size	327
	How to Shift Toolbar Icons to the Right	327
Chapter 19	User Extensions	329
19.1	User Notebook	329
19.2	Macros	330
19.3	AU Programs	330
19.4	Python Programs	331
19.5	Button Panels	332
19.6	Adding User Defined Buttons to the Toolbars	334
19.7	Adding User Defined Menus to the Menubar	337
19.8	Adding User Defined Guides	339

Chapter 1

Getting Started

1.1 Document Conventions

Font Conventions

abs - commands to be entered on the command line are in courier bold italic

ProcPars - menus, buttons, icons to be clicked are in times bold italic

fid - filenames are in courier

name - any name which is not a filename is in times italic

File/directory Conventions

<tshome> - the TOPSPIN installation (home) directory

User Action Conventions

☞ a single user action

1. the first action of a sequence
2. the second action of a sequence
3. etc.
 - a) the first action of a sub-sequence

- b) the second action of a sub-sequence
- c) etc.

1.2 TOPSPIN Overview

Functionality

TOPSPIN is an integrated software package for:

- Displaying NMR spectra
- Printing and plotting spectra
- Exporting displays and plots in various graphics and metafile formats
- Importing NMR data from files of various formats
- Archiving spectra in various formats such as JCAMP-DX and ZIP
- E-mailing data
- Processing 1D-3D fids and spectra: window multiplication, various transforms (Fourier, Hilbert, DOSY), phase correction, baseline correction, integration, peak picking, linear prediction, smoothing, adding spectra etc.
- Displaying multiple superimposed spectra (1D and 2D).
- Simulating 1D and multi-dimensional fids, given a pulse program and a spin system (“virtual spectrometer nmr-sim”)
- Calculating T1/T2 relaxation times
- Fitting peaks with Lorentzian and Gaussian line shape models, deconvolve overlapping peaks
- Multiplet analysis
- Automatic 1D, 2D and 3D peak picking
- Automatic 1D, 2D and 3D integration
- Line shape analysis of solids spectra
- Data acquisition with Bruker Avance type spectrometers
- Supporting automated and walk-up spectrometers (ICON-NMR)
- Remote spectrometer control including web-enabled ICON-NMR
- Adding user defined functionalities to TOPSPIN (AU programs, Macros and Python programs)

Available Documentation

Under the TOPSPIN *Help* menu, you will find the following documentation:

- Avance Beginners Guide: Step-by-step 1D HR experiments
- Users guide: User Interface, Display, Interactive functions
- Control and Function keys: Control and Function keys in TOPSPIN and Plot Editor
- Pulse Program Catalog: graphics representation of Bruker pulse programs
- Command Index: alphabetical list of commands with execute, search, help and macro options
- Parameter Index: alphabetical list of parameters with help option
- Acquisition Reference Guide: acquisition commands, parameters and files.
- Processing Reference Guide: processing commands, parameters and files, including Linear Prediction, Relaxation, T1/T2 and Deconvolution.
- Installation Guides: TOPSPIN, Windows, Linux, Release letter
- Programming manuals: AU, Macro and Python Programming
- Application manuals: High-resolution, Solids, LC, Diffusion
- Accessory manuals: operation of VTU, MAS, ACB, Cortab, ATM
- NMR Guide: NMR Guide knowledge base

1.3 TOPSPIN license

TOPSPIN requires a license for startup. A license can be ordered online from:

www.bruker-biospin.de/NMR/nmrsoftw/licenses/index.html

If your PC controls a spectrometer, TOPSPIN will start up without a license. Furthermore, you can use TOPSPIN for developer purposes with restricted functionality. In this case you have to start it from a Windows Command prompt or Linux shell as follows;

```
topspin -developer
```

1.4 Startup TOPSPIN

Under Windows

☞ Click the TOPSPIN icon on the desktop

or

☞ Start TOPSPIN from a Command Prompt as follows:

1. Click **Start** → **Run**, enter *cmd* and click **OK**
2. In the Command Prompt:
 - a) Enter `cd <tshome>`
 - b) Enter `topspin`

where <tshome> is the directory where TOPSPIN is installed.

Under Linux

1. Open a Linux Shell or Terminal Window
2. In the Shell/Terminal:
 - a) Enter `cd <tshome>`
 - b) Enter `./topspin`

where <tshome> is the directory where TOPSPIN is installed.

1.5 Configuration

After the installation of TOPSPIN, it must be configured once. TOPSPIN may be used in two different ways:

on a computer which controls a spectrometer

The command *cf* must be executed once, to configure the spectrometer hardware. Just type this command and follow the instructions on the screen. At the end of the dialog, further configuration commands, like *expinstall*, are offered and can be started from there.

on a computer which is used as datastation

The only configuration command to be executed is *expinstall*. This allows

you to install pulse programs, AU programs, lists etc. Just type this command and follow the instructions on the screen, selecting *Installation for Datastation (default)*.

Note that the commands *cf* and *expinstall* can be started from the command line or from the *Options* or *Spectrometer* menu. However, the latter menu is only available after *cf* has been performed once, choosing *Installation for spectrometer*.

After the configuration has finished, TOPSPIN is ready to be used. The configuration only needs to be repeated when you have installed a new version of TOPSPIN or if your spectrometer hardware has changed.

More details on configuration can be found in chapter 16.2 and the descriptions of *cf* and *expinstall* in the Acquisition Reference Manual.

1.6 How to Display Spectra

In this chapter, opening data in standard Bruker format is described. Opening other data formats is described in chapter 4.

Please note that a standard Bruker dataset is a directory tree rather than a single file:

```
<dir>\data\<user>\nmr\<dataset name>\<expno>\pdata\<procno>
```

e.g.

```
c:\bruker\topspin\data\guest\nmr\exam1d_13C\1\pdata\1
```

How to Open Data from the Menu

Open the *File* menu and click *Open...* A dialog box appears. Select the first option, the Browser type *File Chooser* and click *OK*. A file browser appears. Navigate to your data directory and expand it to the level of *names*, *expnos*, or *procnos* (double-click a directory to expand it). Select the desired item and click *Display*.

The selected dataset replaces the contents of the currently selected (active) window. If no data window was displayed, a new one will be created. Alternatively, you can first create a new window by clicking *Window* → *New Window [Alt+w]* and then open a dataset from the file browser in that window.

The file browser can also be opened by entering *reb* on the command line.

How to Open Data from the Browser

TOPSPIN has data browser which, by default, displays the top level data directory (<dir>) with Bruker example data. You can add your own data directories, local or remote, as follows:

1. Move the cursor into the browser area
2. Right-click and choose *Add New Data Dir...* in the popup menu
3. Enter the desired data directory (<dir>) and click *OK*

Your data directory will now appear in the browser

In order to display data from the Browser, proceed as follows:

1. Expand your top level directory (<dir>) in the browser to the level of the data *name*, *expno* or *procno*
2. Select the desired item and drag it into the data area

How to Define Alias Names for Data

1. Open the dataset for which you want to define an alias name
2. Click the *Alias* tab at the top of the data browser.
3. Right-click in the browser and choose

Define alias for data in selected window

Alternatively, you can enter the command *dalias* on the command line.

How to Open Data in Other Ways

TOPSPIN provides various other ways of displaying data. You can, for example, use the portfolio instead of the browser. Furthermore, you can use command line commands like *re*, *rew*, *rep* and *dir*. Details on these features can be found in chapter 4.3 and in the Processing Reference Manual.

1.7 How to Display Peaks, Integrals, ... together with the Spectrum

When a spectrum is displayed, you can superimpose its title, parameters, integrals, and peaks as follows:


1. Move the cursor into the data window that contains the spectrum

2. Right-click and choose **Display Properties...** in the popup menu
3. Check the desired items and click **OK**

Please note that the selected items are only shown if they are available. For example, peaks and integrals are only shown if peak picking and integration have been performed, respectively (see also chapter 11).

1.8 How to Display Projections/1D Spectra with 2D Spectra


To display projections or 1D spectra in tandem with a 2D spectrum:

1. Open a 2D spectrum
2. If no projections are shown, click the  button in the upper toolbar or enter `.pr` on the command line.
3. Move the cursor into the F1 or F2 projection area.
4. Right-click and choose one of the options. With **External Projection...** an existing 1D spectrum can be read. This can be a regular 1D spectrum or a 2D projection that was stored as a 1D spectrum. With **Internal Projection** the positive projection can be calculated and displayed.

An alternate way to calculate projections is the following:

 Click **Processing** → **Display Projections...**[`projd`]

1.9 How to Superimpose Spectra in Multiple Display

TOPSPIN allows you to compare multiple spectra in Multiple Display mode. To enter this mode, click the  button in the upper toolbar or enter `.md` on the command line. When you open a dataset now, for example drag one from the browser, it will be superimposed on the current spectrum rather than replacing it. Several multiple display functions are available now in the data window toolbar. Most importantly, you can *scale* and *shift* each spectrum individually. This allows exact alignment of corresponding peaks of different spectra.

Multiple display mode is supported for 1D and 2D spectra. In 2D, you can superimpose an arbitrary number of 1D or 2D spectra.

1.10 How to Print or Export the Contents of a Data Window

How to Print Data

A TOPSPIN data window may contain various objects like an fid, a spectrum, expansions of a spectrum, superimposed spectra, spectrum components such as parameters, peaks, integrals, cross sections etc. Whatever the content of the data window is, it can be printed as follows: type **Ctrl+p** or click **File** → **Print...**, select **Print active window** in the appearing dialog box and click **OK**.

The other options in this dialog box enable you to use or create plot layouts. Details on this can be found in the Plot Editor manual. Just type **plot** to start the Plot Editor and open the manual from its **Help** menu.

The colors of the printed data can be chosen in the User Preferences dialog box. Just enter the command **set** or click **Options** → **Preferences...** and click **Printer** in the left part of the dialog box.

How to Copy a Data Window to Clipboard

Under MS Windows, you can easily copy the data window contents to other applications. To do that, type **copy** or click **Edit** → **Copy**. This will copy the data window contents to the clipboard. After that you can paste the clipboard contents to any Windows application.


How to Store (Export) a Data Window as Graphics File

The clipboard and metafile formats are resizable vector formats. In addition to this, TOPSPIN allows you to save the contents of a data window in a graphics file of selectable type, e.g. **.png**, **.jpg**, **.jpeg**, **.bmp**, **.emf** and **.wmf**. To do that, click **File** → **Export...** The resolution of such a *screen dump* equals the resolution of your screen. Note that when you import a graphics file into another program, resize, it you loose information. Therefore we recommend to resample rather than resize graphics.

1.11 How to Process Data

Since this manual is not a general NMR text book, we assume here that you are familiar with terms like window multiplication, Fourier Transform, phase correction, etc.

Any FID or a spectrum displayed in a TOPSPIN window can be processed by:

- typing a command on the command line, e.g. **ft**
- invoking a command from the **Processing** or **Analysis** menu, e.g. **Processing** → **Fourier Transform...**
- entering an interactive data manipulation mode by clicking a tool button, e.g. 
- entering a user defined command (usually an AU or a Python program, see **Help** → **Programming**)

Processing and analysis commands require certain parameters to be set correctly. Most commands in the **Processing** or **Analysis** menu, like **wm** and **ftf** open a dialog box showing the available options and required parameters for that command. Other commands such as **em**, **ft**, ... start processing immediately. Before you use them, you must set their parameters from the parameter editor. To do that, enter **edp** or click the **ProcPars** Tab of the data window.

If you are a new or occasional user we recommend you to process your data with the TOPSPIN Processing Guide. This will guide you through the typical sequence of processing steps. To start the Processing Guide, click **Processing** → **Processing Guide**. In **Automatic mode**, the Processing Guide will simply execute each processing command when you click the corresponding button. This requires the processing parameters to be set correctly. In interactive mode (**Automatic mode** unchecked), the Processing Guide will, at each step, open a dialog box offering you the available options and required parameters. For example, the phase correction button offers various automatic algorithms as well as an option to switch to interactive phasing mode.

1.12 How to Archive Data

TOPSPIN 1.3 provides the following methods for data archiving:

- Copying a dataset to a desired destination directory which could for instance be located on a server. Type **wrpa**, click **File** → **Save...** or type **Ctrl+s**.
- Saving a dataset in a ZIP file. A standard Bruker dataset is a directory tree which contains several files. “Zipping” a dataset stores the entire data directory tree into a single file with the extension **.bnmr.zip**. To zip a dataset, type **tozip**, click **File** → **Save...** or type **Ctrl+s**. To unpack and display a zipped dataset, enter **fromzip**. Note that **.bnmr.zip** files are fully com-

patible with the well known PC zip format and can be unpacked with any common unzip utility. “Zipping” can be applied to 1D, 2D, 3D and 4D data.

- Saving a dataset in JCAMP-DX format. This format is a IUPAC standard, and is available for 1D and 2D datasets. Data and parameters are stored in readable text (ASCII) format. To store data in JCAMP, type *tojdx*, click *File* → *Save...* or type *Ctrl+s*. To convert and display a JCAMP-DX file, type *fromjdx*.
- E-mailing data to a desired destination. Type *smail* or click *File* → *Send To...* The mailing format is either zip or JCAMP-DX, both of which allow for data compression in order to keep the transferred data size as small as possible.

1.13 How to Import NMR Data Stored in Special Formats

TOPSPIN allows you to convert various data formats to standard Bruker format for display and processing. Click *File* → *Open...* and select *Open NMR data stored in special formats*. Then follow the instruction on the screen.

1.14 How to Fit Peaks and Deconvolve Overlapping Peaks

Peaks of a 1D and 2D NMR spectrum can be approximated by a Lorentzian, Gaussian, or a mixture of these line shapes. Overlapping peaks may be deconvolved into a sum of such line shapes.

TOPSPIN shows the deconvolution result, i.e. peak positions, line widths and integrals on the screen and stores it in the file *dconpeaks.txt*. Furthermore, it switches to multiple display mode to show the original spectrum and the sum of the computed line shapes, superimposed.

To start deconvolution, expand the spectrum on the display to show the peak or peak group of interest. Then type *dcon* or click *Analysis* → *Deconvolution...*

1.15 How to Compute Fids by Simulating Experiments

TOPSPIN includes a “virtual spectrometer” that computes fids of any dimension. A “real” spectrometer excites a sample with high frequency pulses defined by a pulse program and measures the resulting fid. The virtual spectrometer performs this

task mathematically by solving the quantum mechanical Liouville equation. The “sample” must be entered in form of a spin system description. For the computation, the same pulse program and acquisition parameters are taken as for the real experiment. The result is a time domain signal which can be processed with TOPSPIN in the same way a measured fid is processed. Techniques such a selective excitation, gradient enhanced spectroscopy, and the handling of mixtures are supported.

To start the virtual spectrometer:

☞ Enter *nmr sim* on the command line or click *Analysis* → *Simulate Fid...*

1.16 How to Add Your Own Functionalities

The TOPSPIN functionality can be extended with various user defined commands, programs etc.

How to Create Macros

Writing a macro is the simplest way to create a user defined command. Just enter the command *edmac*, select a file, and enter a sequence of regular TOPSPIN commands and/or Python commands. Save the file under a name of your preference. You have created a new TOPSPIN command. Just enter its name on the command line to execute it.

The command *xmac* shows a list of all available macros and allows you to execute one.

How to Create AU (automation) Programs

Writing an AU program is another way of creating a new TOPSPIN command. AU programs are more complex and more powerful than macros. They are C-language programs, which may contain C-statements, regular TOPSPIN commands, and various predefined AU macros and functions. AU programs can perform various tasks such as dataset handling, parameter handling, acquisition, processing, analysis, and printing. Note that AU programs do not support graphics related tasks.

TOPSPIN is delivered with a large set of Bruker AU programs for data processing and acquisition. Just enter *edau* to see them listed in a dialog box. The easiest way to create a new AU program is to select a Bruker AU program, save it under

a new name and modify it to your needs. The chosen name is now available as a new TOPSPIN command. Alternatively, you can open a new file from the AU dialog box and write your AU program from scratch.

For details on Bruker AU programs and writing your own AU programs, click **Help** → **Programming** → **AU Programming**.

How to Create Python Programs

Writing a Python program is yet another way of creating a new TOPSPIN command. Python is a new generation scripting and object oriented programming language. Python programs are even more powerful than AU programs. They are easy to use and allow you to execute TOPSPIN commands, handle NMR data and parameters, generate graphics, and interact with the TOPSPIN user interface via dialogs, windows etc. To create a Python program, enter the command **edpy**, select a file and insert your Python statements. Graphics and interface features programmed in Python look and work the same as regular TOPSPIN features.

For details on Python programming, click **Help** → **Programming** → **Python Programming**. The examples mentioned there, like *pycmd1*, are delivered with TOPSPIN. Just enter their names on the command line to execute them.

The Python dialog window is also available from the TOPSPIN menu:

☞ Click **File** → **Open...** and select **Open other file** → **Python program**.

1.17 How to Automate Data Acquisition

TOPSPIN provides special user interfaces for automation, walk-up, bio-molecular experiments, etc. To open these interfaces:

☞ Type **iconnmr** or click **Spectrometer** → **ICONNMR**.

Chapter 2

The TOPSPIN Interface

2.1 The Topspin Window

The TOPSPIN window consist of several areas, bars, fields and buttons. The main part is a split pane which consists of the data area and the browser. Note that the browser can be inactive [hit **Ctrl+d**] or displayed as a separate window.

Fig. 2.1 shows the Topspin window with two data windows in the data area and the browser as an integral part.

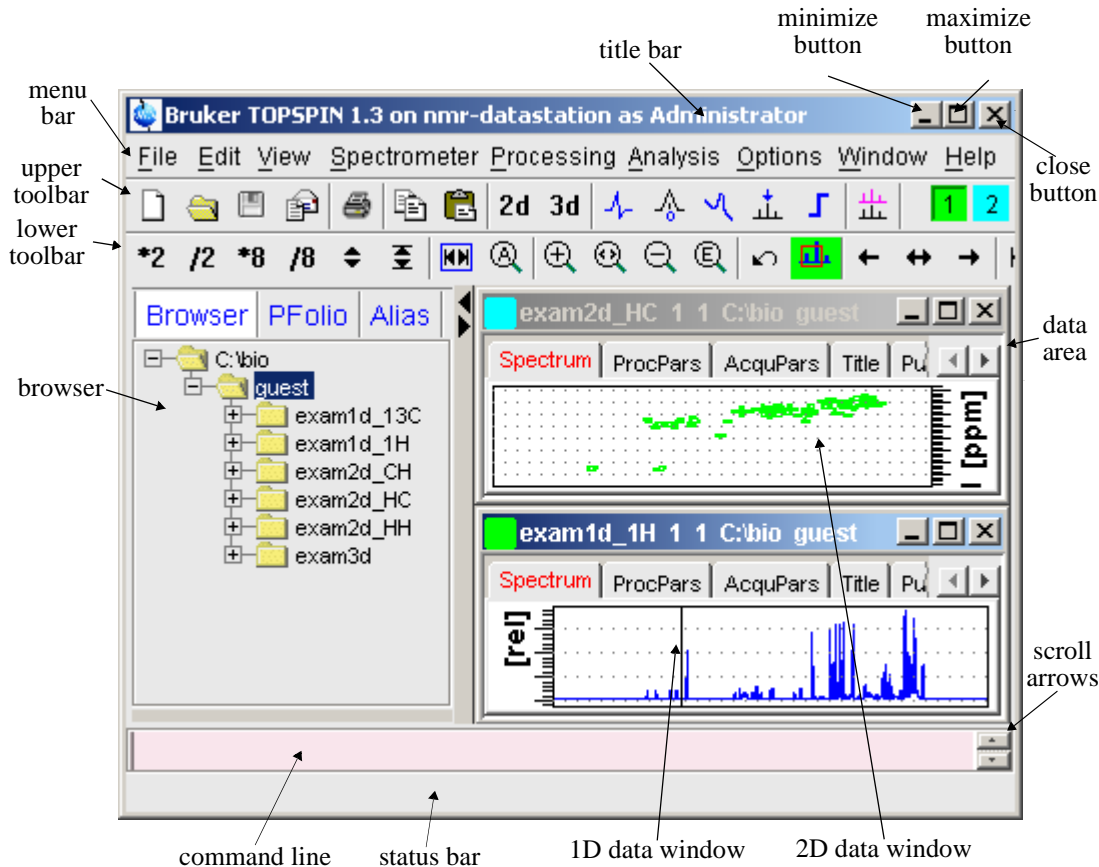


Figure 2.1

Note that the menus and toolbars depend on the data dimensionality. The descriptions below hold for 1D data. For 2D and 3D data, the menus and toolbars are similar and will be discussed in the chapters 9, 10 and 12, respectively.

How to Use Multiple Data Windows

TOPSPIN allows you to use multiple data windows. Data windows can be opened from the browser or from the **Window** menu. They can contain the same of different datasets. Data windows can be arranged from the **Window** menu. One of them is the active (current) data window. The active window:

- can be selected by clicking inside the window or hitting F6 repeatedly.
- has a highlighted title bar
- has the mouse focus
- is affected by menu, toolbar and command line commands

A cursor line (1D) or crosshair (2D) is displayed in all data windows at the same position. Moving the mouse affects the cursor in all data windows.

How to Use the Title bar

In the title bar you can:

- Left-click-hold & drag to move the window
- Double-click to maximize the window
- Right-click to open the title bar menu.
- Access the minimize, maximize and close buttons at the right
- Access the title bar menu button at the left

How to Use the Menu bar

The menu bar contains the following menus:

- ***File*** : performing data/file handling tasks
- ***Edit*** : copy & paste data and finding data
- ***View*** : display properties, browser on/off, notebook
- ***Spectrometer*** : data acquisition and acquisition related tasks
- ***Processing*** : data processing
- ***Analysis*** : data analysis
- ***Options*** : setting various options, preferences and configurations
- ***Window*** : data window handling/arrangement
- ***Help*** : access various manuals.

Experienced users will usually work with keyboard commands rather than menu commands. Note that the main keyboard commands are displayed in square brackets [] behind the corresponding menu entries. Furthermore, right-clicking any menu entry will show the corresponding command.








How to Use the Upper Toolbar

The upper toolbar contains buttons for data handling, switching to interactive modes, display settings, and starting acquisition.

Buttons for data handling:



The functions of the individual buttons are:




-  Create a new dataset [*Ctrl+n, new*]
-  Open a dataset [*Ctrl+o, open*]
-  Save the current dataset [*Ctrl+s, sav*]
-  Email the current dataset [*smail*]
-  Print the current dataset [*Ctrl+p, print*]
-  Copy the data path of the active data window to the clipboard [*copy*]
-  Paste the data path on the clipboard to the active data window [*paste*]
- 2d** Switch to the last 2D dataset [*. 2d*]
- 3d** Switch to the last 3D dataset [*. 3d*]





For more information on dataset handling, please refer to chapter 4.3.

Buttons for interactive manipulation



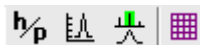
The functions of the individual buttons are:

-  Switch to phase correction mode
-  Switch to calibration mode
-  Switch to baseline correction mode





-  Switch to peak picking mode
-  Switch to integration mode
-  Switch to multiple display mode
-  Switch to distance measurement mode

For more information on interactive manipulation, refer to chapter 11 and 12.

Buttons for display options



The functions of the individual buttons are:

-  Toggle between Hz and ppm axis units
-  Switch the y-axis display between abs/rel/off
-  Switch the overview spectrum on/off
-  Toggle grid between fixed/axis/off





For more information on display options, please refer to chapter 8.5 and 9.5.


How to Use the Lower Toolbar


The lower toolbar contains buttons for display manipulations.

Buttons for vertical scaling (intensity manipulation)




-  Increase the intensity by a factor of 2 [***2**]
-  Decrease the intensity by a factor of 2 [**/2**]
-  Increase the intensity by a factor of 8 [***8**]
-  Decrease the intensity by a factor of 8 [**/8**]


 Increase/decrease the intensity smoothly


 Reset the intensity [*.vr*]


Buttons for horizontal scaling (zooming):





 Reset zooming (horizontal scaling) to full spectrum [*.hr*]


 Reset zooming (horizontal scaling) and intensity (vert. scaling) [*.all*]


 Zoom in (increase horizontal scaling) [*.zi*]

 Zoom in/out smoothly

 Zoom out (decrease horizontal scaling) [*.zo*]


 Exact zoom via dialog box [*.zx*]


 Retrieve previous zoom [*.z1*]


 Retain horizontal and vertical scaling when modifying dataset or changing to different dataset. Global button for all data windows [*.keep*]


Buttons for horizontal shifting



 Shift to the left, half of the displayed region [*.s1*]

 Smoothly shift to the left or to the right




 Shift to the right, half of the displayed region [*.sr*]

 Shift to the extreme left, showing the last data point [*.s10*]

 Shift to the extreme right, showing the first data point [*.sr0*]

Buttons for vertical shifting




-  Shift the spectrum baseline to the middle of the data field [*.su*]
-  Smoothly shift the spectrum baseline up or down.
-  Shift the spectrum baseline to the bottom of the data field [*.sd*]


2.2 Command Line Usage

How to Put the Focus in the Command Line

In order to enter a command on the command line, the focus must be there. Note that, for example, selecting a dataset from the browser, puts the focus in the browser. To put the focus on the command line:

 Hit the **Esc** key

or


 Click inside the command line

How to Retrieve Previously Entered Commands

All commands that have been entered on the command line since TOPSPIN was started are stored and can be retrieved. To do that:

 Hit the \uparrow (**Up-Arrow**) key on the keyboard

By hitting this key repeatedly, you can go back as far as you want in retrieving previously entered commands. After that you can go forward to more recently entered commands as follows:

 Hit the \downarrow (**Down-Arrow**) key on the keyboard

How to Change Previously Entered Commands

1. Hit the \leftarrow (**Left-Arrow**) or \rightarrow (**Right-Arrow**) key to move the cursor
2. Add characters or hit the **Backspace** key to remove characters
3. Mark characters and use **Backspace** or **Delete** to delete them, **Ctrl+c**

to copy them, or **Ctrl+v** to paste them.

In combination with the arrow-up/down keys, you can edit previously entered commands.

How to Enter a Series of Commands

If you want to execute a series of commands on a dataset, you can enter the commands on the command line separated by semicolons, e.g.:

em;ft;apk

If you intend to use the series regularly, you can store it in a macro as follows:

☞ right-click in the command line and choose ***Save as macro***.

2.3 Command Line History

TOPSPIN allows you to easily view and reuse all commands, which were previously entered on the command line. To open a command history control window; click **View** → **Command Line History**, or right-click in the command line and choose **Command Line History**, or enter the command ***cmdhist*** (see Fig. 2.2).

It shows all commands that have been entered on the command line since TOPSPIN was started. You can select one or more commands and apply one of the following functions:

Execute

Execute the selected command(s).

Append

Append the (first) selected command to the command line. The appended command can be edited and executed. Useful for commands with many arguments such as ***re***.

Save as..

The selected command(s) are stored as a macro. You will be prompted for the macro name. To edit this macro, enter ***edmac <macro-name>***. To execute it, just enter its name on the command line.

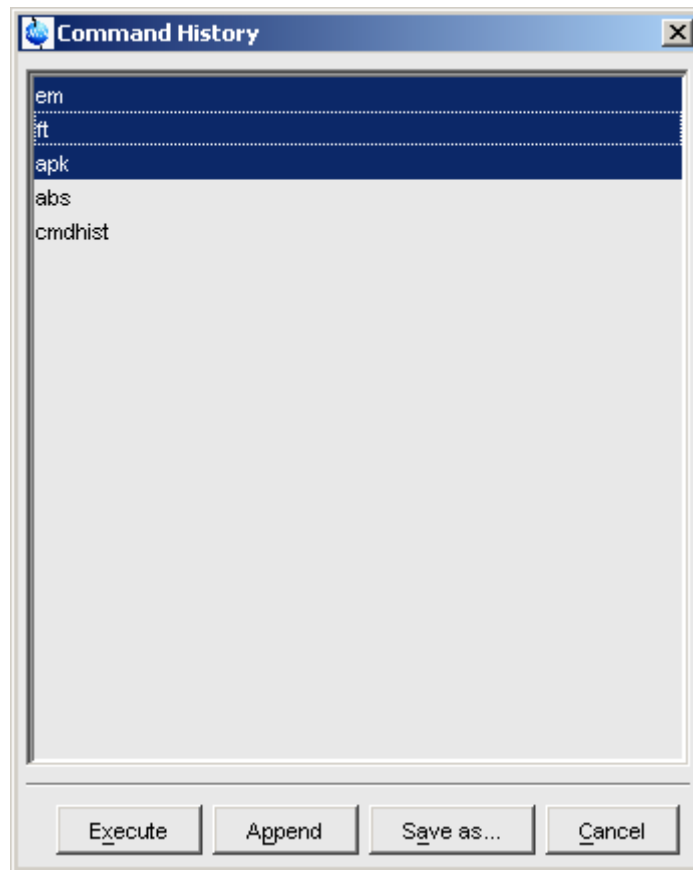


Figure 2.2

2.4 Starting TOPSPIN commands from a Command Prompt

TOPSPIN commands can be executed from a Windows Command Prompt or Linux Shell. To do that:

1. Open a Windows Command Prompt or Linux Shell
2. Enter a Topspin command in the following format:

```
<tshome>\prog\bin\sendgui <topspincommand>
```

where <tshome> is the TOPSPIN installation directory.

Examples:

```
C:\ts1.3\prog\bin\sendgui re exam1d_1H 1 1 C:/bio joe
reads the dataset C:/bio/joe/nmr/exam1d_13C/1/pdata/1.
```

```
C:\ts1.3\prog\bin\sendgui ft
executes a 1D Fourier transform.
```

Commands are executed on the currently active data window.

2.5 Function Keys and Control Keys

For several TOPSPIN commands or tasks, you can use a control-key or function-key short cut.

Focus anywhere in TOPSPIN

<i>Esc</i>	Put the focus in the command line
<i>Shift+Esc</i>	Display menu bar and toolbars (if hidden)
<i>F2</i>	Put the focus in the browser / portfolio
<i>F1</i>	Search for string in command help or NMR Guide [<i>help</i>]
<i>F6</i>	Select the next window in the data area
<i>Alt+F4</i>	Terminate TOPSPIN [<i>exit</i>]
<i>Ctrl+d</i>	Switch the browser/portfolio on/off
<i>Ctrl+o</i>	Open data [<i>open</i>]
<i>Ctrl+f</i>	Find data [<i>find</i>]
<i>Ctrl+n</i>	New data [<i>new</i>]
<i>Ctrl+p</i>	Print current data [<i>print</i>]
<i>Ctrl+s</i>	Save current data [<i>sav</i>]
<i>Ctrl+w</i>	Close active window [<i>close</i>]
<i>Ctrl+c</i>	Copy a text that you selected/highlighted in an error box, dialog box, pulse program, title etc., to the clipboard
<i>Ctrl+v</i>	Paste text from the clipboard into any editable field.

Focus in the Command Line

Ctrl+Backspace	Kill current input
Ctrl+Delete	Kill current input
UpArrow	Select previous command (if available).
DownArrow	Select next command (if available).

Focus in the Browser/Portfolio

UpArrow	Select previous dataset
DownArrow	Select next dataset
Enter	In the Browser: expand selected node
Enter	In the Portfolio: display selected data

Focus anywhere in TOPSPIN

<i>Scaling Data</i>	
Alt+PageUp	Scale up the data by a factor of 2 [*2]
Alt+PageDown	Scale down the data by a factor 2 [/2]
Ctrl+Alt+PageUp	Scale up by a factor of 2, in all data windows
Ctrl+Alt+PageDown	Scale down by a factor of 2, in all data windows
Alt+Enter	Perform a vertical reset
Ctrl+Alt+Enter	Perform a vertical reset in all data windows
<i>Zooming data</i>	
Alt+Plus	Zoom in [.zi]
Alt+Minus	Zoom out [.zo]
Ctrl+Alt+Plus	Zoom in, in all data windows
Ctrl+Alt+Minus	Zoom out, in all data windows
<i>Shifting Data</i>	
Alt+UpArrow	Shift spectrum up [.su]
Alt+DownArrow	Shift spectrum down [.sd]
Alt+LeftArrow	Shift spectrum to the left [.sl]
Alt+RightArrow	Shift spectrum to the right [.sr]
Ctrl+Alt+UpArrow	Shift spectrum up, in all data windows
Ctrl+Alt+DownArrow	Shift spectrum down, in all data windows
Ctrl+Alt+LeftArrow	Shift spectrum to the left, in all data windows
Ctrl+Alt+RightArrow	Shift spectrum to the right, in all data windows

Focus in a Table (e.g. peaks, integrals, nuclei, solvents)

<i>delete</i>	Delete the selected entries
<i>home</i>	Select the first entry
<i>end</i>	Select the last entry
<i>Shift+Home</i>	Select the current and first entry and all in between
<i>Shift+End</i>	Select the current and last entry and all in between
<i>DownArrow</i>	Select next entry
<i>UpArrow</i>	Select previous entry
<i>Ctrl+a</i>	Select all entries
<i>Ctrl+c</i>	Copy the selected entries to the clipboard
<i>Ctrl+z</i>	Undo last action
<i>Ctrl+y</i>	Redo last undo action

Focus in a Plot Editor

<i>F1</i>	Open the Plot Editor Manual
<i>F5</i>	Refresh
<i>ctrl+F6</i>	Display next layout
<i>ctrl+Shift+F6</i>	Display previous layout
<i>Ctrl+tab</i>	Display next layout
<i>delete</i>	Delete the selected objects
<i>Ctrl+a</i>	Select all objects
<i>Ctrl+i</i>	Open TOPSPIN Interface
<i>Ctrl+c</i>	Copy the selected object from the Clipboard
<i>Ctrl+l</i>	Lower the selected object
<i>Ctrl+s</i>	Save the current layout
<i>Ctrl+m</i>	Unselect all objects
<i>Ctrl+n</i>	Open a new layout
<i>Ctrl+o</i>	Open an existing layout
<i>Ctrl+p</i>	Print the current layout
<i>Ctrl+r</i>	Raise the selected object
<i>Ctrl+t</i>	Reset X and Y scaling of all marked objects
<i>Ctrl+v</i>	Paste the object from the Clipboard
<i>Ctrl+w</i>	Open the attributes dialog window.
<i>Ctrl+x</i>	Cut the selected object and place it on the Clipboard
<i>Ctrl+z</i>	Undo the last action

Note that the function of function keys can be changed as described in chapter 2.7.

2.6 Help in Topspin

TOPSPIN offers help in various ways like online manuals, command help and tooltips.

How to Open Online Help documents

The online help manuals can be opened from the **Help** menu. For example, to open the manual that you are reading now:

☞ Click **Help** → *User's Guide*

To open the Avance Beginners Guide guide:

☞ Click **Help** → *Avance Beginners Guide*

To open the Processing Reference guide:

☞ Click **Help** → *Processing Reference Manual*

Note that most manuals are stored in the directory:

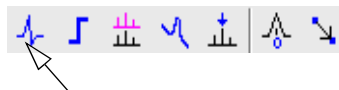
```
<tshome>/prog/docu/english/xwinproc/pdf
```

The most recent versions can be downloaded from:

```
www.bruker-biospin.de
```

How to Get Tooltips

If you hold the cursor over a button of the toolbar, a tooltip will pop up. This is a short explanation of the buttons function. For example, if you hold the cursor over the interactive phase correction button, you will see the following:



Interactive phase correction [.ph]

The corresponding command line command, in this case **.ph**, is indicated between square brackets.

Note that the tooltip also appears in the status bar at the bottom of the TOPSPIN window.

How to Get Help on Individual Commands

To get help on an individual command, for example **ft**:

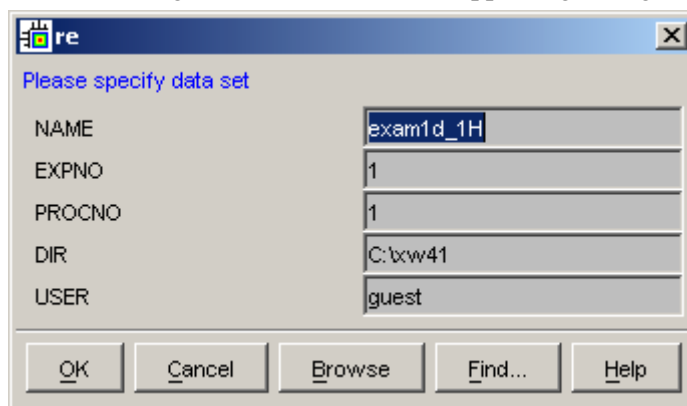
☞ Enter **ft?**

or

☞ Enter **help ft**

In both cases, the HTML page with a description of the command will be opened.

Note that some commands open a dialog box with a **Help** button. Clicking this button will show the same description as using the **help** command. For example, entering **re** and clicking the **Help** button in the appearing dialog box



opens the same HTML file as entering **help re** or **re?**.

How to Use the Command Index

To open the TOPSPIN command index:

☞ Enter **cmdindex**

or

☞ Click **Help** → **Command Index**

From there you can click any command and jump to the corresponding help page.

2.7 User Defined Functions Keys

The default assignment of functions keys is described in chapter 2.5 and in the document:

 *Help* → *Control and function keys*

You may assign your own commands to functions keys. Here is an example of how to do that:

1. Open the file `cmdtab_user.prop`, located in the subdirectory `userdefined` of the user properties directory (to locate this directory, enter *hist* and look for the entry "User properties directory="). The file `cmdtab_user.prop` is initially empty and can be filled with your own command definitions.
2. Insert e.g. the following lines into the file:

```
_f3=$em  
_f3ctrl=$ft  
_f3alt=$pk  
_f5=$halt  
_f5ctrl=$reb  
_f5alt=$popt
```

3. Restart TOPSPIN

Now, when you hit the **F3** key, the command *em* will be executed. In the same way, **Ctrl+F3**, **Alt+F3**, **F5**, **Ctrl+F5** and **Alt+F5** will execute the commands *ft*, *pk*, *halt*, *reb* and *popt*, respectively. You can assign any command, macro, AU program or Python program to any function keys. Only the keys **Alt+F4**, **F6**, **Ctrl+F6**, and **Alt+F6** are currently fixed. Their function cannot be changed.

2.8 How to Open Multiple TOPSPIN Interfaces

TOPSPIN allows you to open multiple User Interfaces. This is, for example, useful to run an acquisition in one interface and process data in another. To open an addition interface, enter the command *newtop* on the command line or click *Window* → *New Topspin*. To open yet another interface, enter *newtop* in the first or in the second interface. The display in each interface is completely independent from the others. As such, you can display different datasets or different aspects of the same

dataset, e.g. raw/processed, regions, scalings etc. When the dataset is (re)processed in one interface, its display is automatically updated in all TOPSPIN interfaces.

The command ***exit*** closes the current Topspin interface. Interfaces that were opened from this interface remain open. Entering ***exit*** in the last open TOPSPIN interface, finishes the entire TOPSPIN session. The position and geometry of each TOPSPIN interface is saved and restored after restart.

Chapter 3

Trouble Shooting

3.1 General Tips and Tricks

On a spectrometer, make sure the commands *cf* and *expinstall* have been executed once after installing TOPSPIN. *cf* must be executed again if your hardware configuration has changed. Sometimes, executing *cf* is useful in case of acquisition problems.

On a datastation, a default configuration is automatically done during the installation. No configuration commands are required. Only if you want to use AU programs, you must run *expinstall* once.

3.2 History, Log Files, Stack Trace

If you have a problem with TOPSPIN and want to contact Bruker, it is useful to have as much information as possible available. If TOPSPIN is still running, you can view log files with the commands *hist* and *ptrace*. If TOPSPIN hangs, you can create a stack trace by hitting **Ctrl+** (Linux) or **Ctrl+Break** (Windows) in the TOPSPIN startup window.

3.2.1 History log file

By default, the history (protocol) feature is switched on. This means all TOPSPIN commands will be protocolled and can be examined by entering *hist* on the command line.

If, for some reason, history is switched off, you can switch it on as follows:

1. Click *Options* → *Preferences*, click *Miscellaneous*
2. Check the entry "*Record commands in protocol file*"
3. Click *OK*.

3.2.2 All log files

A full protocol including not only TOPSPIN commands but also dataserver and network traffic is shown by the command *ptrace*. This opens the following dialog

window:

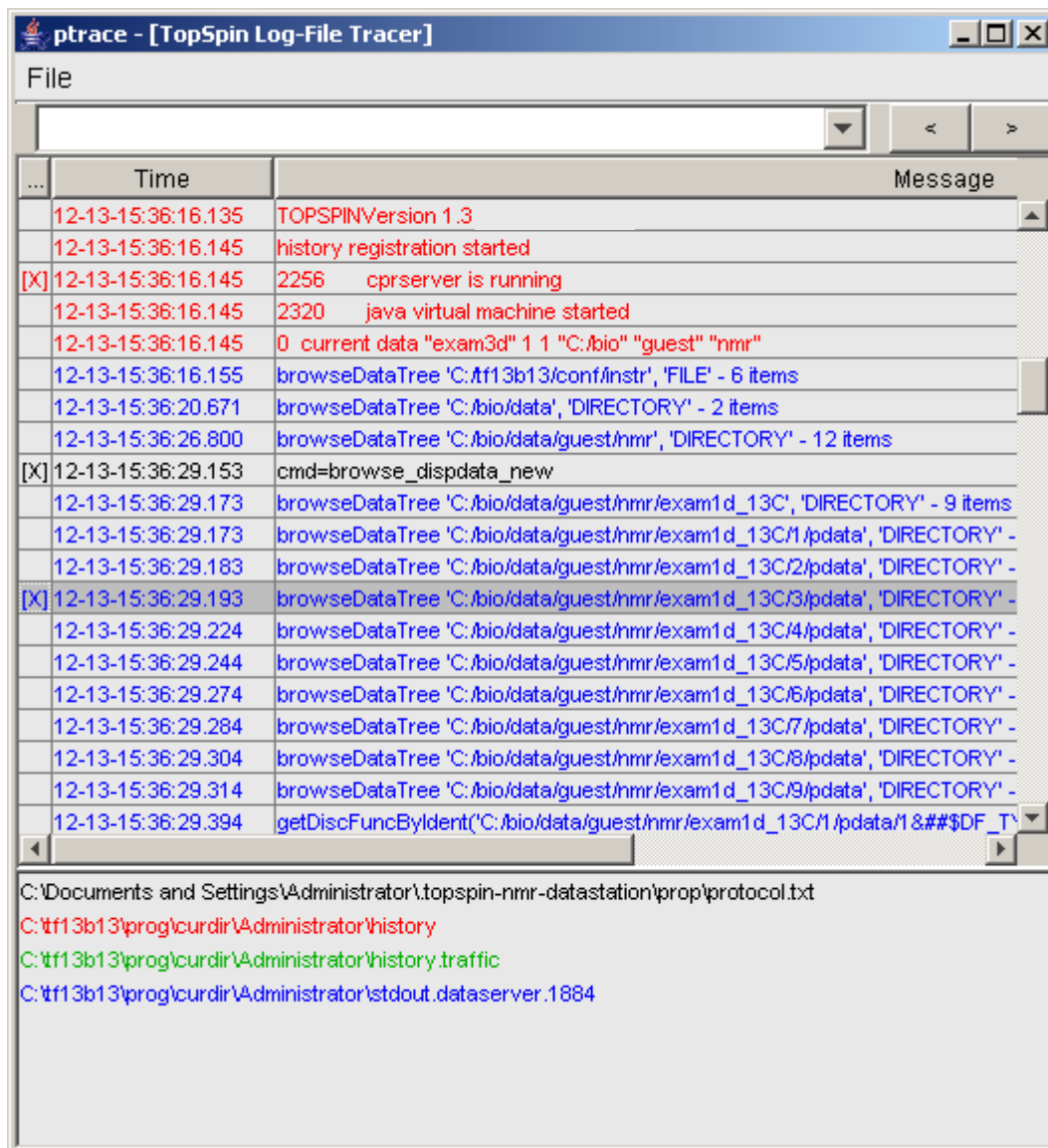


Figure 3.1

Here TOPSPIN log messages from various log files are displayed time sorted.

Messages from different log files are shown in different colours. The color assignment and location of the files is shown in the lower part of the dialog window.

ptrace supports the following functions:

Search keywords

To search the displayed log files for a certain keyword, just enter it in the text bar at the top of the window and hit Enter.

Anchor specific entries

To anchor an entry:

double-click in the first column of the entry

The entry is marked with an **X**. Now you can use the < > buttons at the upper right of the window to go to the previous or next entry, respectively.

Add log files

By default, the `procol.txt`, `history` and `dataserver` log files are shown. To include additional log files to the ***ptrace*** list:

click ***File*** → ***Add***, specify the file and click ***Open***.

Note that most log files are stored in the directory:

<tsHOME>/prog/curdir/<user>

Save the log messages

To save log messages in a text file;

click ***File*** → ***Save*** and specify the output text file

3.2.3 Create a Stack Trace

If TOPSPIN hangs it can be useful to send Bruker a stack trace about a possible cause. You can create a stack trace as follows.

1. Move the cursor into the TOPSPIN startup window.
2. Under Windows: hit ***Ctrl+Break*** (= ***Ctrl+Pause***)
Under Linux: hit ***Ctrl+*** (Control backslash)
3. Copy the appearing text into a text file.

3.3 How to Show or Kill TOPSPIN processes

To show the currently running TOPSPIN processes, enter the command *show* or *kill* on the command line. A list of processes will appear showing the process command, dataset etc.

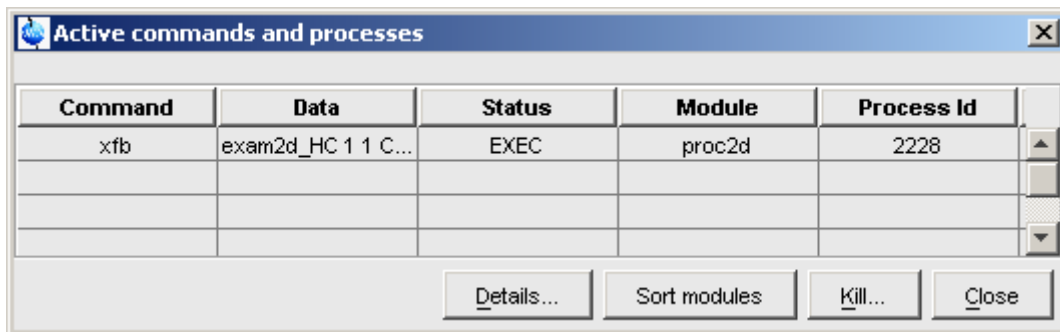


Figure 3.2

Fig. 3.2 shows a list with one process (command *xfb*). To kill a process, select it in the list and click the button *Kill...*

The command *show all* or *kill all* work like *show* and *kill*, except that they also show TOPSPIN system processes. Note that killing such processes may kill TOPSPIN.

3.4 What to do if TOPSPIN hangs

If, for some reason, TOPSPIN hangs, please do the following.

Under Linux:

1. Open a Shell
2. Enter `<tshome>/prog/bin/script/killtopspin`
where `<tshome>` is the TOPSPIN installation directory.

Under Windows:

1. Click **Start** → **Programs** → **Bruker TOPSPIN** → **TOPSPIN 1.3** → **Bruker Utilities 1.3**
2. In the appearing window:

Click **killtopspin**.

Normally, this kill all TOPSPIN processes including *cpr*, *cprserver*, *dataserver* and *java*.

3.5 How to Restart User Interface during Acquisition

If Topspin hangs up during a data acquisition, you can restart the user interface without disturbing the acquisition. To do that:

1. Open the file:

```
<tshome>/prog/curdir/<user>/history
```

where <tshome> is the TOPSPIN home directory and <user> is the user who started TOPSPIN. Look for the term 'Java Virtual Machine' and check its PID.

2. Open the Task Manager (Windows) or System Monitor (Linux)
3. Stop the Java(w).exe process with the PID found in the history file.
4. Open a Windows Command Prompt or Linux Shell
5. Go to the TOPSPIN Installation directory
6. Enter **topspin -client**

Chapter 4

Dataset Handling

4.1 The Topspin Browser and Portfolio

TOPSPIN offers a data browser/portfolio from which you can browse, select, and open data. Furthermore it allows you to define and list alias names for data. The browser appears at the left of the TOPSPIN window and can be controlled from the *View* menu.

The browser is similar to the Windows Explorer. It shows data directory trees and allows you to expand/collapse their elements. Figure 4.1 shows a TOPSPIN browser with one top level data directory and one dataset fully expanded.

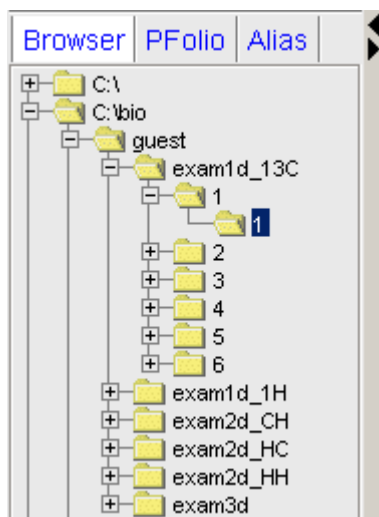


Figure 4.1

The TOPSPIN portfolio shows the list of datasets that has been opened so far by the current user. Each dataset that you open, is automatically added to the current portfolio. A portfolio with five datasets look like this:



Figure 4.2

Each line displays one dataset showing its *name*, *expno*, *procno*, *top level directory* and *user*.

The TOPSPIN Alias list show a list of alias names for datasets. Just right-click any

entry to define, remove or interpret alias names.

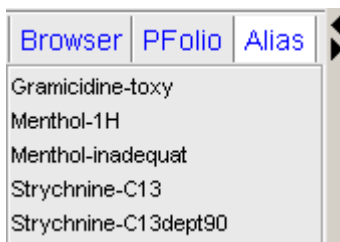


Figure 4.3

How to Open the Browser/Portfolio

☞ Click *View* → *Browser Panel On/Off* [*Ctrl+d*]

The browser or portfolio will appear depending on which one was used last.

How to Open the Browser/Portfolio in a separate window

The browser or portfolio can be opened in a separate window as follows:

☞ Click *Options* → *Preferences* [*set*], click *Window settings* and check *Display dataset browser in a separate window*.

You must restart TOPSPIN for the change to take effect.

How to Put the Focus in the Browser/Portfolio

☞ Hit the *F2* key

or

☞ Click inside the browser or portfolio

How to Select Folders in the Browser

To select a particular folder:

☞ Left-click the folder button

or

☞ Hit the arrow-up/down keys while the focus is in the browser

To select multiple folders:

☞ Hold the **Ctrl** key and left-click multiple folders to select them

or

☞ Hold the **Shift** key and left-click two folders to select these two and all in between.

How to Expand/Collapse a Folder in the Browser

To expand a collapsed folder:

☞ Click the + button to the left of the folder button

or Double-click the folder button

or Hit the **Right-Arrow** key while the folder is highlighted

or Right-click the folder button and choose **Expand fully** from the popup menu to fully expand the folder

To collapse an expanded folder:

☞ Click the - button to the left of the folder button

or Double-click the folder button

or Hit the **Left-Arrow** key while the folder is highlighted

How to Expand a Folder showing Pulse program and Title

☞ Right-click the data name folder button and choose

Expand fully & show PULPROG /Title from the popup menu

Fig. 4.4 shows an example of an expanded dataset showing the pulse program and title.

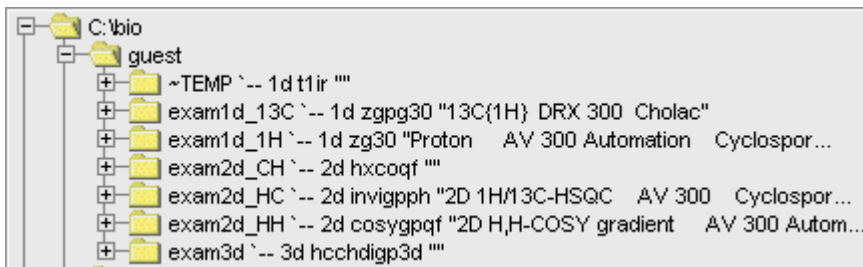


Figure 4.4

Note that collapsing the data name folder will deselect the display of the pulse program and title.

How to Add/Remove a Top Level Data Directory

To add a new top level data directory, local or remote, to the browser:

1. Right-click any position in the browser
2. Choose *Add new data dir...* from the popup menu (see Fig. 4.5)
3. Enter the top level directory in the dialog box and click **OK**

To remove an existing top level directory from the browser:

1. Right-click the folder button of the top level directory
2. Choose *Remove selected data dir...* from the popup menu (see Fig. 4.5)

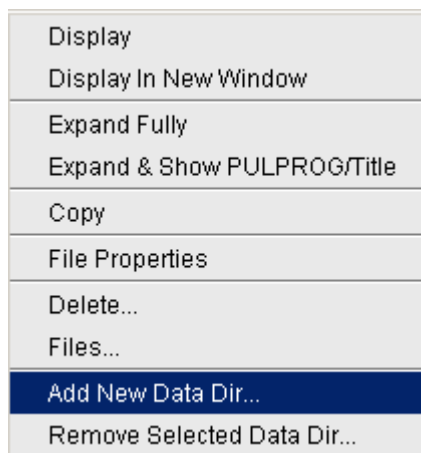


Figure 4.5

3. Click **OK** to confirm the appearing message

By default, the browser shows the TOPSPIN installation directory with the Bruker example datasets. To suppress this feature click *Options* → *Preferences [set]*, click *Program startup actions* and uncheck *Show TOPSPIN default data directory in data browser*.

How to Open a New Portfolio

1. Right-click inside the portfolio

2. Choose *Open portfolio...* from the appearing popup menu (see Fig. 4.6)
3. Navigate to the folder that contains the portfolio files
4. Select the desired portfolio file (extension `.prop`)
5. Click *Open*

How to Save the current Portfolio

1. Right-click inside the portfolio.
2. Choose *Save portfolio...* from the appearing popup menu (see Fig. 4.6).
3. Specify a folder and filename in the appearing dialog box. The filename must have the extension `.prop`.
4. Click *Save*.

How to Remove Datasets from the Portfolio

To remove a single dataset from the portfolio:

1. Right-click the dataset.
2. Choose *Remove from portfolio...* from the popup menu (see Fig. 4.6).
3. Click *OK* in the appearing alert box.

To remove multiple datasets from the portfolio:

1. Hold the *Ctrl* key and left-click several datasets to select them or hold the *Shift* key and left-click two datasets to select these two and all in between.
2. Right-click any of the selected datasets.
3. Choose *Remove from portfolio...* from the popup menu (see Fig. 4.6).
4. Click *OK* in the appearing alert box.

How to Find Data and Add them to the Portfolio

1. Click *Edit* → *Find data* [*Ctrl+F* | *Find*].
2. Specify the search criteria and click *OK*.
3. Select dataset(s) and click *Add to portfolio*.

How to Sort Data in the Portfolio

1. Right-click inside the portfolio

2. Choose **Sort mode** from the popup menu (see Fig. 4.6).
3. Click **Alphabetical** to sort data in alphabetical order
or
Click **Recent first** to sort data by date of last open.

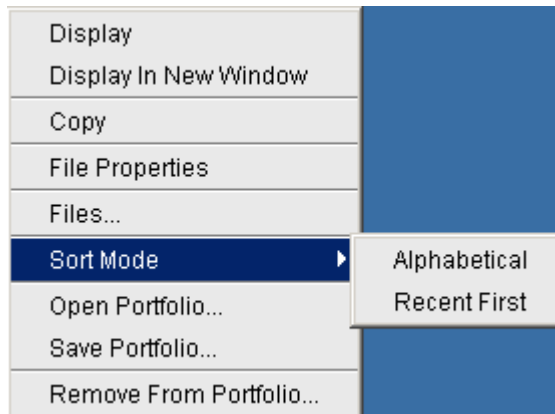


Figure 4.6

How to Add, Remove or Interpret Alias Names

To add an alias name:

1. Click the **Alias** tab in the browser.

2. Right-click in the Alias table to open the popup menu (see Fig. 4.7).

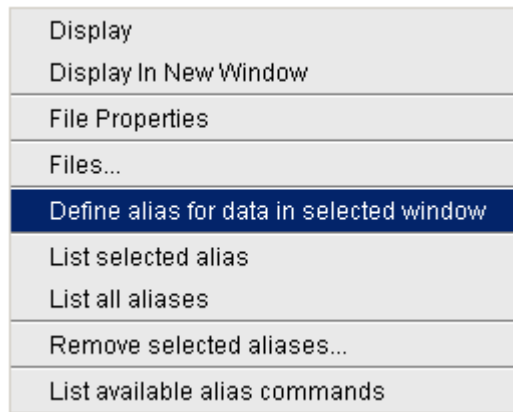


Figure 4.7

3. Click *Define alias names for data in selected window*.
4. Enter an alias name in the appearing dialog box and click **OK**. Note that alias names must begin with a letter.

To remove an alias name:

1. Right-click the *alias name*
2. Click *Remove selected aliases...* from the popup menu (see Fig. 4.7)

Furthermore, the popup menu offers entries to display the dataset, list its properties and print the full dataset specification.

4.2 Creating Data

How to Create a New Dataset

1. Click *File* → *New* [*new*, *Ctrl+n*]

or

Click the button  in the upper toolbar.

2. Specify the dataset *name*, *expno*, *procno*, *dir*, and *user* in the appearing dialog box. If one or more datasets are open, the fields are initialized with the current dataset (see Fig. 4.8).

3. Click the down-arrow of the **Solvent** box and choose a solvent from the list, or type a solvent name.
4. Click the down-arrow of the **Experiment** box and choose a parameter set from the list, or type a parameter set name.
5. Type the dataset title in the **TITLE** box.
6. Click **OK**.

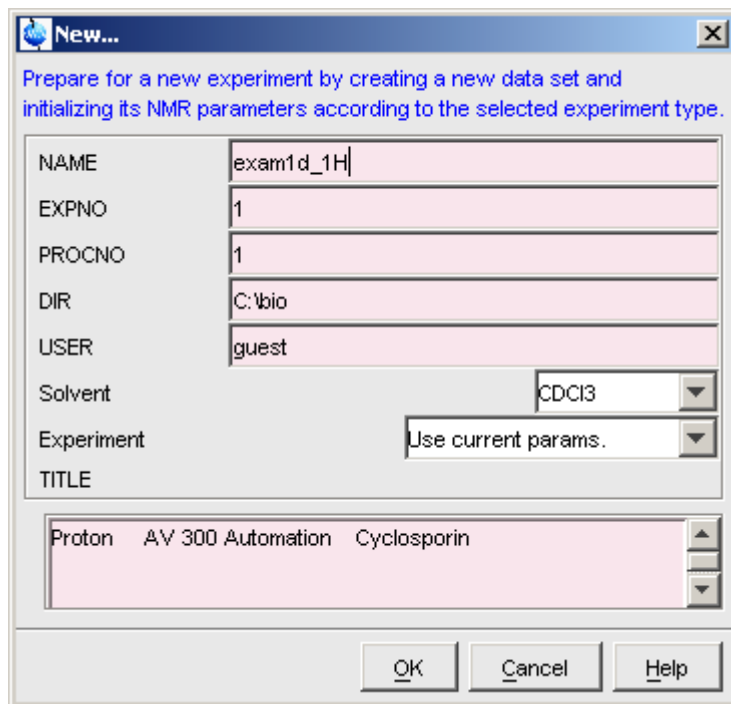


Figure 4.8

A dataset will be created and initialized with the parameters of the chosen experiment. No fid or spectrum are available yet. They can be created by data acquisition and data processing, respectively.

4.3 Opening Data

TOPSPIN allows you to open data in several ways, from the browser, the menu, the

Explorer or the command line. Furthermore, data can be opened:

- in an existing data window replacing the current dataset.
- in a data window which is in multiple display mode, being superimposed on the current spectra.
- in a new data window which becomes the active window.

Note that if a dataset is already displayed in one window and it is opened in a second existing window, it still replaces the dataset in the latter one. As a result, the same dataset will be displayed in two windows (see also command **reopen**).

How to Open Data Windows Cascaded

By default, a new data window appears maximized, filling the entire data field and covering possibly existing window. You can, however, configure TOPSPIN to open new windows cascaded. This is convenient if you want to open several data windows and then select one.

To open new windows cascaded:

1. Click **Options** → **Preferences** [**set**]
2. Click **Window Setting** in the left part of the dialog box.

The right part of the dialog box shows the window settings (see Fig. 4.9).

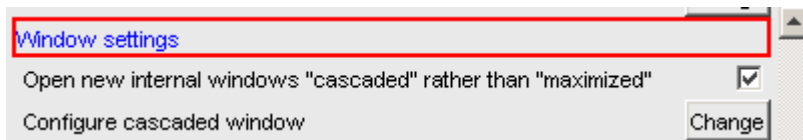


Figure 4.9

3. Check *Open new internal windows 'cascaded' rather than 'max'*.
4. Optionally you can configure the cascaded windows by clicking the respective **Change** button. This will open the dialog box shown in Fig. 4.10.

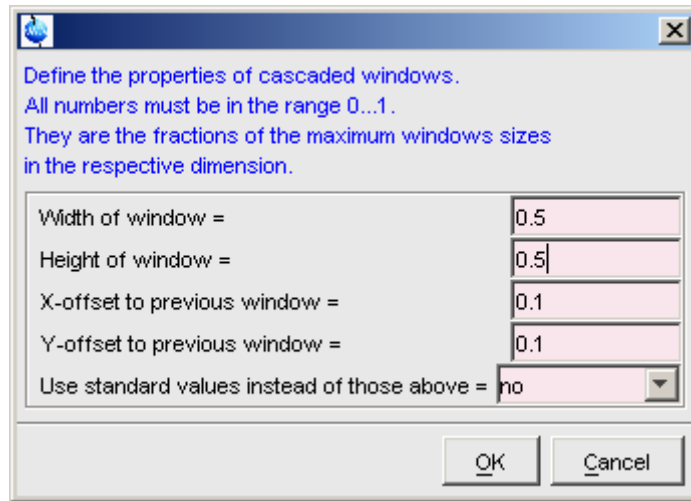


Figure 4.10

5. Here you can specify the data window sizes and offsets as fractions of the maximum window sizes.
6. Click **OK** to close the dialog box.

How to Open Data from the Browser

In the browser:

- ☞ Left-click-hold a data *name*, *expno* or *procno* and drag it into the data area. The data will be displayed in a new data window.
- or Left-click-hold a data *name*, *expno* or *procno* and drag it into an open data window. The data will replace the currently displayed data.
- or Left-click-hold a data *name*, *expno* or *procno* and drag it into an empty data window created with **Alt+w n**.
- or Left-click-hold a data *name*, *expno* or *procno* and drag it into a multiple display data window. The data will be superimposed on the currently displayed data.
- or Right-click a data *name*, *expno* or *procno* and choose **Display** from the pop-up menu; the data will be displayed in the current data window.
- or Right-click a data *name*, *expno* or *procno* and choose **Display in new win-**

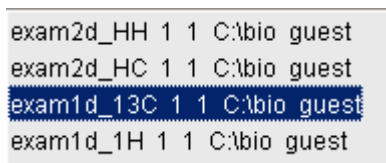
dow from the popup menu; the dataset will be displayed in a new data window.

or Hold the **Ctrl** key and left-click several datasets to select them or hold the **Shift** key and left-click two datasets to select these two and all in between. Then right-click one of the selected datasets and choose **Display** from the popup menu. A new window will be opened showing the selected datasets in multiple display mode. However, if the current window was already in multiple display mode, the selected spectra will be superimposed on the currently displayed spectra.

How to Open Data from the Portfolio

The portfolio offers the same possibilities to open a dataset as the browser. Additional options are:

☞ Hit the **Enter** key to display the highlighted dataset in the current window.



```
exam2d_HH 1 1 C:\bio guest
exam2d_HC 1 1 C:\bio guest
exam1d_13C 1 1 C:\bio guest
exam1d_1H 1 1 C:\bio guest
```

Figure 4.11

☞ Double-click a dataset to display it in the current window.

How to Automatically Select the first *expno/procno* of a dataset

If you open a dataset from the Browser by clicking a data *name*, there might be more than one *expno* and/or *procno* available. By default, TOPSPIN then opens a dialog box from which you can select the desired *expno/procno* combination (see Fig. 4.12). You can, however configure TOPSPIN to automatically open the first available *expno/procno* combination. To do that:

1. Click **Options** → **Preferences** [**set**].
2. Click **Miscellaneous** in the left part of the dialog box.
3. Uncheck the item *Display expno/procno list when opening data*.
4. Click **OK** to close the dialog box.

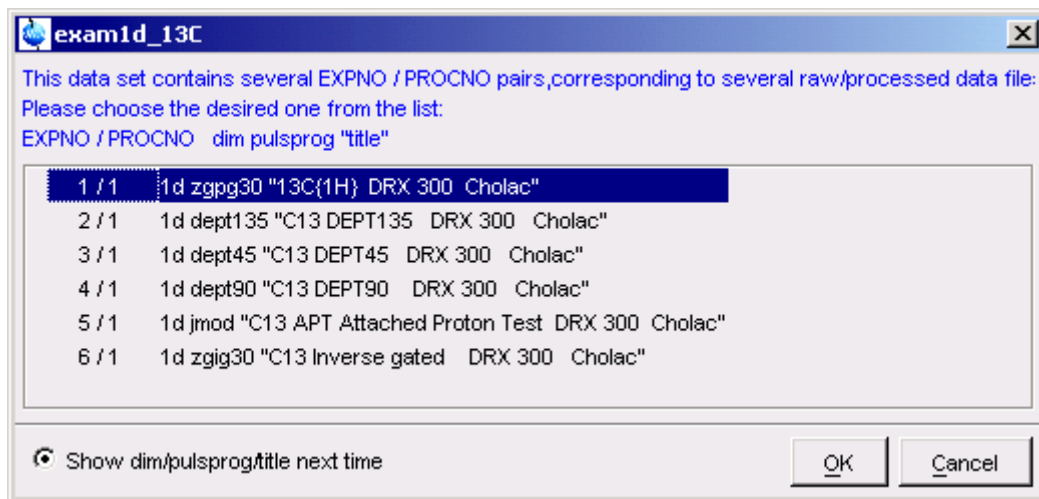


Figure 4.12

How to Open Data from the Topspin menu

1. To open a dataset:

☞ Click the  button in the upper toolbar.

or

☞ Click **File** → **Open** [*open, Ctrl+o*] (see Fig. 4.13).

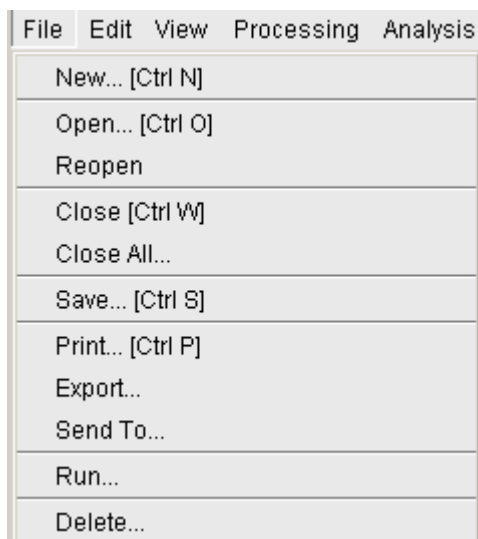


Figure 4.13

2. In the appearing dialog box (see Fig. 4.14)

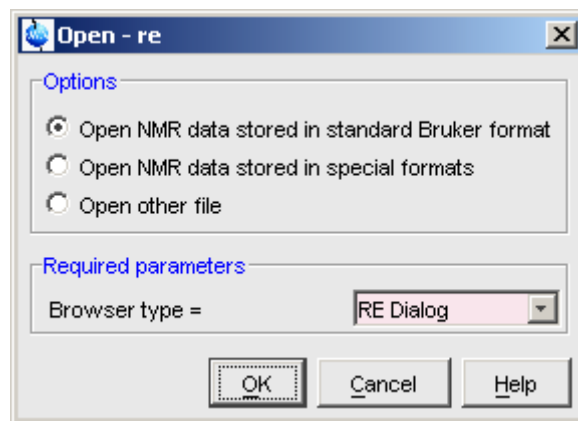


Figure 4.14

- a) Select the option *Open NMR data stored in standard Bruker format*.
- b) Select the browser type *RE Dialog*.
- c) Click *OK*.

3. In the appearing dialog box (see Fig. 4.15).

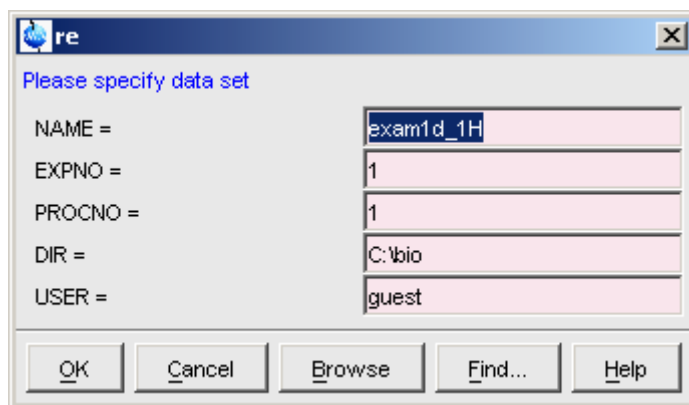


Figure 4.15

- a) Specify the dataset *name*, *expno* etc.
- b) Click **OK**.

Note that the dataset specification consists of the five variable parts of the data directory tree, in this case:

C:\bio\data\guest\nmr\exam1d_1H\1pdata\1

The text boxes are initialized with the dataset in the current data window.

How to Open Data from the Explorer, Konqueror or Nautilus

You can open a dataset from the Windows Explorer as follows:

1. Open the Windows Explorer. You can do that in two different ways:

☞ from the Windows **Start** button. Navigate to the data *name*, *expno* or *procno*.

or

☞ by entering the command **expl** in TOPSPIN. The Explorer shows the contents of the current dataset *procno* directory. Navigate to the desired data *name*, *expno* or *procno*. **expl** can also be used with the argument **top** to open the TOPSPIN installation directory, **home** to open user home directory or with an absolute pathname to open that directory.

2. Now you can open a dataset with:

☞ **drag & drop:** click-hold a dataset *name* or any of its sub-folders or files and drag it into the TOPSPIN data area or data window.

or

☞ **copy & paste:** right-click a dataset and choose *copy* from the popup menu. In TOPSPIN, click *Edit* → *Paste* [*paste*] (see Fig. 4.16).

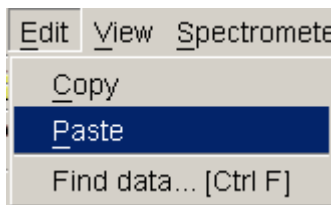


Figure 4.16

Likewise, a dataset can be opened from the Windows Search window or Internet Browser.

How to Open Data from the Command Line

To open a dataset from the command line:

1. Enter **re**
2. Specify a dataset in the appearing dialog box (see Fig. 4.15).
3. Click **OK**

To open a new *procno* of the current dataset:

1. Enter **rep**
2. Specify a *procno* in the appearing dialog box.
3. Click **OK**

To open a dataset in a new window:

1. Enter **rew**
2. Specify a dataset in the appearing dialog box.
3. Click **OK**

To open a new *procno* of the current dataset in a new window:

1. Enter **repw**

2. Specify a *procno* in the appearing dialog box.
3. Click **OK**

To open a data browser and read a dataset from there:

1. Enter **reb**
2. Select a dataset from the appearing dialog box.
3. Click **Display**

Note that **re**, **rep** and **reb**:

- Replace the data in the currently selected data window.
- Open the data in a new window when they are used after typing **Alt+w n**
- Add the data in the currently selected window if this is in multiple display mode.

whereas **rew** and **repw** :

- Always open the dataset in a new window.

How to Open Special Format Data

Apart from the standard Bruker data format, TOPSPIN is able to read and display various other formats. To do this:

☞ Click **File** → **Open** [**open**, **Ctrl+o**]

select the option **Open NMR data stored in special formats**, select the desired file type (see Fig. 4.17) and click **OK**..

A dialog will appear which depends on the chosen file type. Just follow the instructions on the screen.

The following file types are supported:

- JCAMP-DX - Bruker TOPSPIN¹ data stored in JCAMP-DX format
- Zipped TOPSPIN - Bruker TOPSPIN data stored in ZIP format
- WINNMR - Bruker WINNMR data
- A3000 - Bruker Aspect 3000 data

1. Note that the TOPSPIN data format is identical to the XWIN-NMR data format.

- VNMR - data acquired on a Varian spectrometer
- JNMR - data acquired on a Jeol spectrometer
- Felix - 1D data, FID or spectrum, which are stored in FELIX format.

Note that in all cases, the data are stored in a single data file which is unpacked/converted to standard Bruker format, i.e. to a data directory tree.

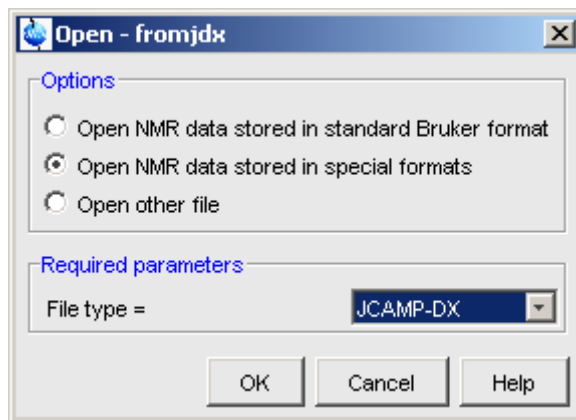


Figure 4.17

How to Open a ZIP or JCAMP-DX file from the Windows Explorer

Data stored in ZIP or JCAMP-DX format can also be opened directly from the Windows Explorer. You can do that as follows:

- ☞ **drag & drop**: click-hold a file with the extension `.dx` or `.zip` and drag it into the TOPSPIN data area or data window.
- ☞ **copy & paste**: right-click a file with the extension `.dx` or `.zip` and choose *copy* from the popup menu. In TOPSPIN, click *Edit* → *Paste* [*paste*].

4.4 Saving/Copying Data

How to Save or Copy Data

You can save the current dataset as follows:

1. Click **File** → **Save** [**Ctrl+S**].

This will open a dialog box (see Fig. 4.18).

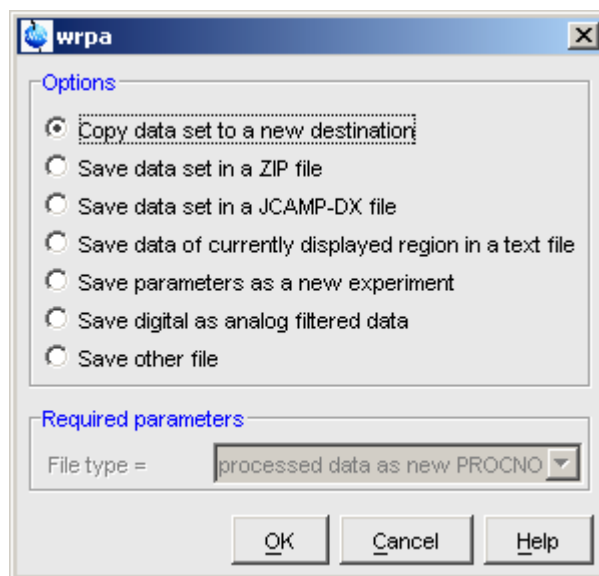


Figure 4.18

2. Select an option and, if applicable, a file type.
3. Click **OK** to execute the option.

The six options correspond to the following command line commands:

- **wrpa** - copies the current data to a new data *name* or *expno*
- **tozip** - convert a dataset of any dimension to ZIP format
- **tojdx** - convert a 1D or 2D dataset to JCAMP-DX format
- **totxt** - convert a 1D or 2D dataset text format
- **wpar** - write parameter set
- **convdta** - save digitally filtered data as analog filtered data
- **wrp, wra, genfid, wmisc** - write various files

How to Save an Entire Dataset

1. Click **File** → **Save** [**Ctrl+S**].

2. Select the option *Copy dataset to a new destination* [*wrpa*] and click *OK*
3. Specify the dataset variables and click *OK*

How to Save Processed Data

1. Click *File* → *Save* [*Ctrl+s*].
2. Select the option *Save other file*
3. Select File type *Processed data as new procno* [*wrp*] and click *OK*
4. Enter a processing number (*procno*) and click *OK*

How to Save Acquisition Data

1. Click *File* → *Save* [*Ctrl+s*].
2. Select the option *Save other file*
3. Select File type *Acqu. data as new expno* [*wra*] and click *OK*
4. Enter a experiment number (*expno*) and click *OK*

How to Save Processed Data as Pseudo Raw Data

1. Click *File* → *Save* [*Ctrl+s*]
2. Select the option *Save other file*
3. Select File type *1r/li as fid* [*genfid*] or *2rr/2ii as ser* [*genser*]
4. Click *OK*
5. Enter a destination *expno*.
(optionally, you can specify further data path specifications)
6. Click *OK*

4.5 Deleting Data

How to Delete a Specific Dataset

☞ Right-click the data *name*, *expno* or *procno* in the browser, then click *Delete...*

In each case, a delete dialog will appear. The dialog box for a data *expno*, for an is shown in Fig. 4.19.

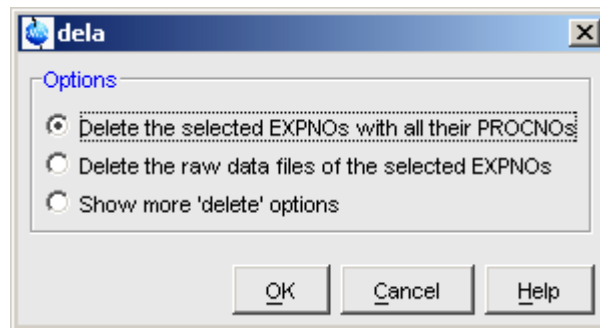


Figure 4.19

You can choose to delete just the raw data, delete the entire expno with all procnos or open further delete options. In the later case, the dialog box shown in Fig. 4.20 will appear.

How to Delete Types of Datasets

To delete certain types of data like 1D raw data, 2D processed data etc.:

☞ Click **File** → **Delete...**

or

☞ Enter **delete** on the command line.

The dialog window shown in Fig. 4.20 will appear. Here you can select the data type and selection criteria.

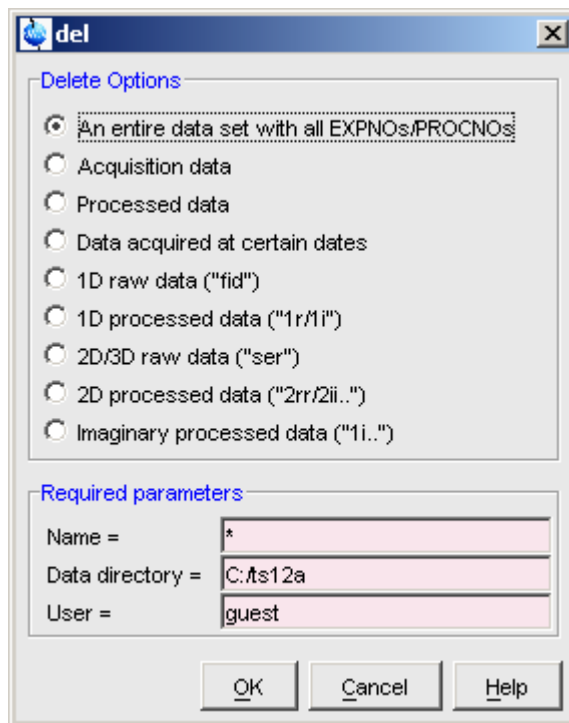


Figure 4.20

1. Select a data type option

For each option, the corresponding command appears in the title of the dialog box. These commands can also be used to delete data from the command line.

2. Specify the *Required parameters*

Note that you can use the wildcards:

- Asterix (*) for any character and any number of characters.
- Question mark (?) for any single character.

3. Click *OK*

A dialog box will appear showing the matching datasets. For example, if you select the option *An entire dataset ...* :

- 1. Select dataset entries for deletion (selected entries are highlighted).**

To select multiple entries: click them holding the *Shift* or *Ctrl* key.

2. Click *Delete* to delete the entire data directory.

If you select the option *Acquisition data* or *Processed data*, you can choose between deleting the data files only and deleting the entire *expno* or *procno* directory, respectively (see Fig. 4.21).

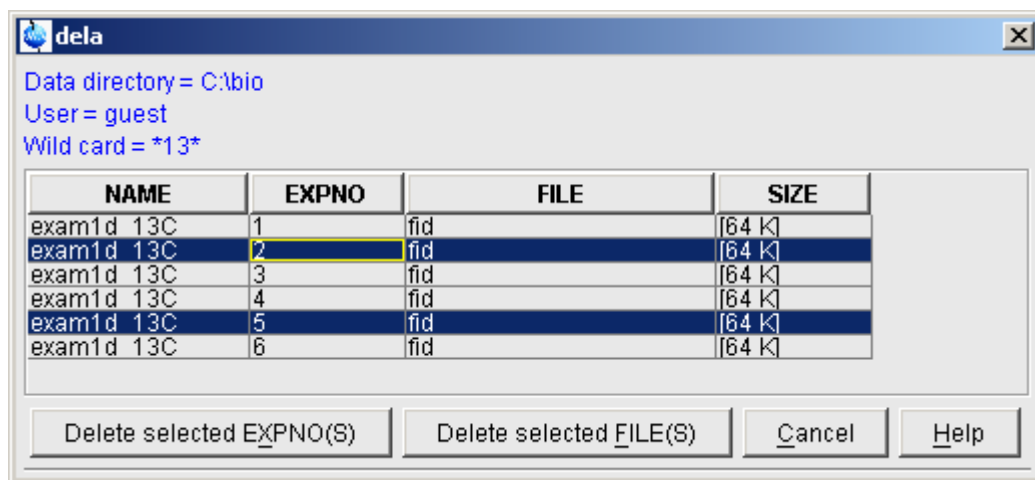


Figure 4.21

4.6 Searching/Finding Data

How to Find Data

You can find TOPSPIN data according to various criteria. To start searching do the following:

1. Click *Edit* → *Find data* [*Ctrl+f* | *find*] to open the *Find data* window (see Fig. 4.22).

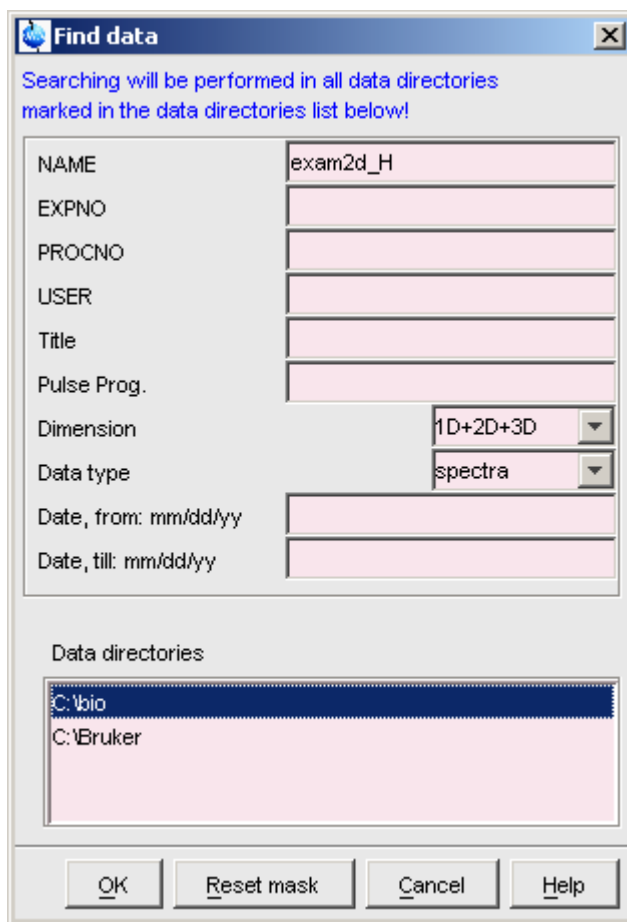


Figure 4.22

2. Specify the search criteria. Note that:
 - Dataset variables are searched that contain the specified string.
 - Search is restricted to data created between the specified dates. Note that the acquisition date is evaluated.

The **Reset mask** button resets the default criteria.

3. Click **OK**
to get a list of data that fulfil these criteria (see Fig. 4.23).

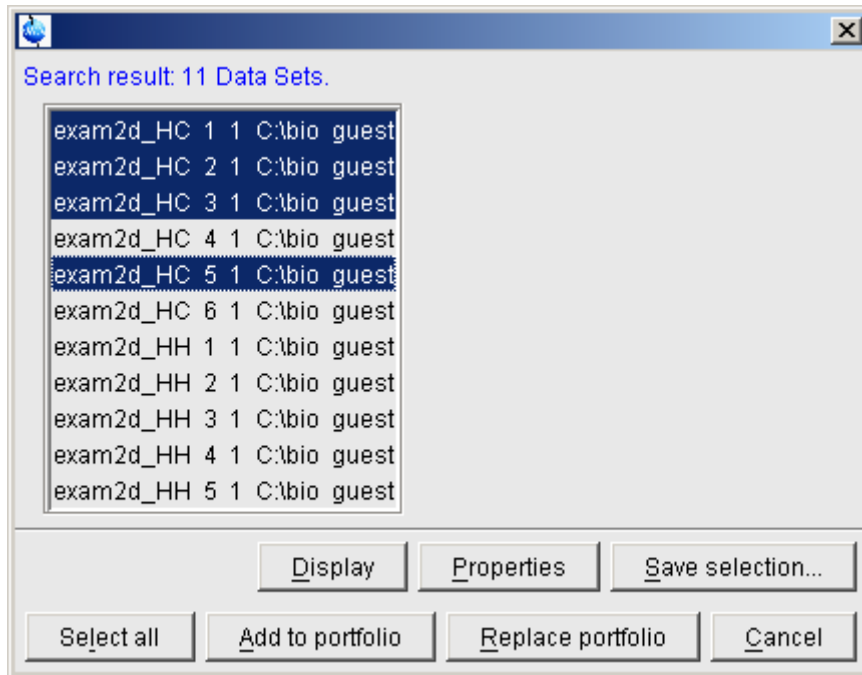


Figure 4.23

Note that the current search criteria are preserved until you exit TOPSPIN.

How to Display one of the Found Datasets

In the search result window (see Fig. 4.23):

1. Click one dataset to select it.

Optionally: click *Properties* to view the datasets properties.

2. Click *Display*
to display the selected dataset in the current data window.

Note that if the search result consist of only one dataset, this is automatically selected and you can skip step 1.

How to Select Data from the Found Datasets

In the search result window (see Fig. 4.23):

☞ Hold the **Ctrl** key and left-click several datasets to select these datasets.

or

☞ Hold the **Shift** key and left-click two datasets to select these datasets and all datasets in between.

or

☞ Click **Select all** to select all datasets in the search result.

How to Add Selected Datasets to the Portfolio

1. Select the desired dataset(s) as described above.
2. To add them to the portfolio:

☞ Click **Add to portfolio** to extend the current portfolio.

or

☞ Click **Replace portfolio** to replace the current portfolio.

How to Save Selected Datasets to a List

1. Select the desired dataset(s) as described above.
2. Click **Save selection...**
3. In the appearing browser:
 - a) Navigate to the desired list directory.
 - b) Enter or select the list filename.
 - c) Click **OK**

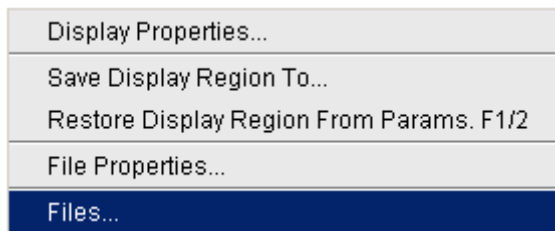
Dataset lists can be used by the acquisition or by serial processing (command **serial**).

4.7 Handling Data Files

How to List/Open the Current Dataset Files

A Bruker dataset is represented by a directory tree which contains files in the *ex-pno* and *procno* subdirectories. These files contain the actual data, parameters, lists etc.

- ☞ Right-click inside the data window and choose **Files** from the popup menu.



If the spectrum is displayed, the files in the *procno* subdirectory are shown. If the **Fid** is displayed, the files in the *expno* subdirectory are shown.

- ☞ Select a file and click **Open** to view its contents.

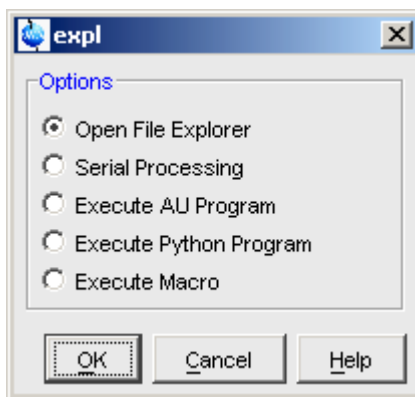
Note that this only makes sense for ascii files.

How to List/Open the current Dataset Files in the Windows Explorer

To list the current dataset files in the Windows Explorer:

1. Click **Files** → **Run...**
2. Select **Open file explorer [exp1]** in the appearing dialog box
3. Click **OK**

Alternatively, you can enter the command **exp1** on the command line. The Win-



dows Explorer will be opened showing the processed data files (the files in the *procno* directory) of the current dataset. Under Linux a Web browser like KDE

Konqueror or Gnome Mozilla will be opened.

To open a file:

☞ Double-click the file or right-click the folder icon and choose *Open*

If TOPSPIN data area contains no datasets, the **expl** command opens the Explorer showing the users home directory. When entered on the command line, **expl** can also be used with the argument **top** to open the TOPSPIN installation directory, **home** to open user home directory or with an absolute pathname to open that directory.

4.8 Data Consistency Check

TOPSPIN maintains audit trail files for compliance with GLP, GMP and FDA requirements.

The processing command that creates processed data from the raw data, e.g. **em**, creates the processing audit trail file `auditp.txt` and inserts the first entry. Any processing command that modifies/updates the processed data, e.g. **ft**, makes an additional entry. Furthermore, any command that changes one or more processing status parameters makes an additional entry. The audit trail can be checked with the command **audit**. This command opens a dialog box (see Fig. 4.24).

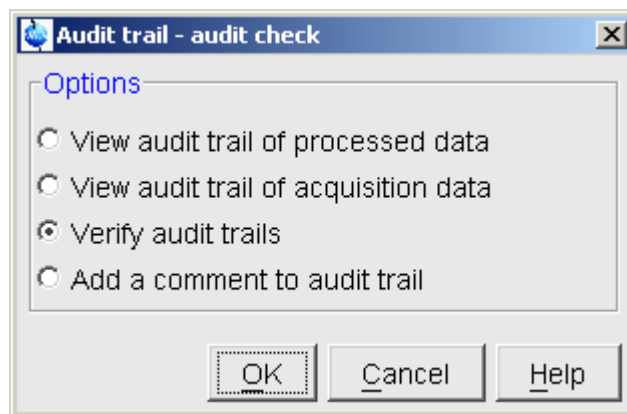


Figure 4.24

The first two entries allow you to view the audit trail files. The third entry performs an audit trail check, i.e. a data consistency check. If both raw and processed

data are consistent, you will get the message shown in Fig. 4.25).

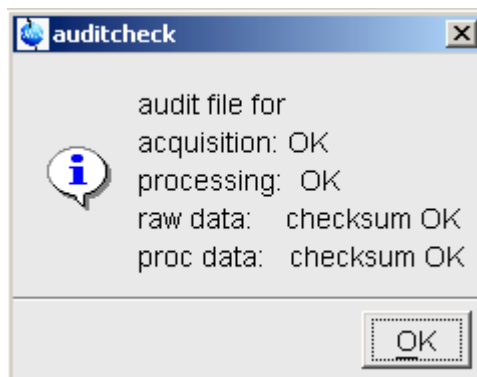


Figure 4.25

If the data have been manipulated outside of TOPSPIN, e.g. with third party software, the checksum will be inconsistent. Fig. 4.26 shows the message for inconsistent processed data.

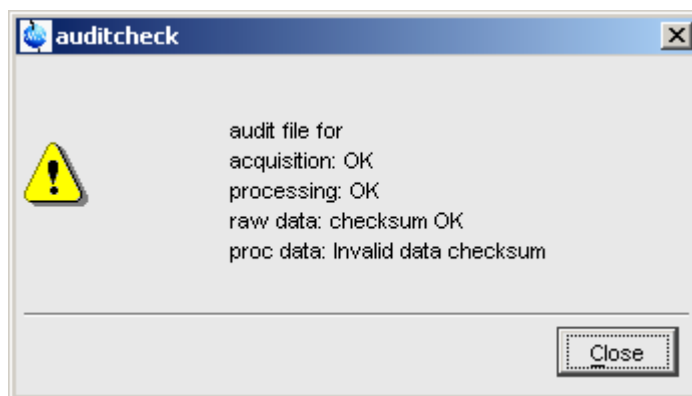


Figure 4.26

The fourth entry in Fig. 4.24 allows you to add a comment to one of the audit trail files (raw or processed).

Chapter 5

Parameter Handling

5.1 Processing Parameters

Processing parameters can be set/changed in three different ways:

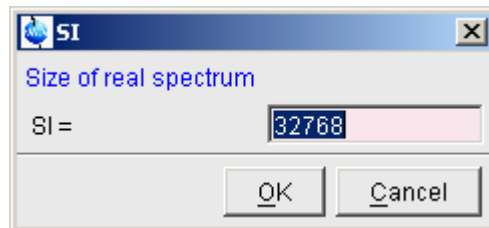
- from the parameter editor: click the *ProcPars* tab or enter *edp*
- from the command line: e.g. enter *si*
- from a command dialog box: e.g. *wm*

How to Set a Processing Parameter from the Command Line

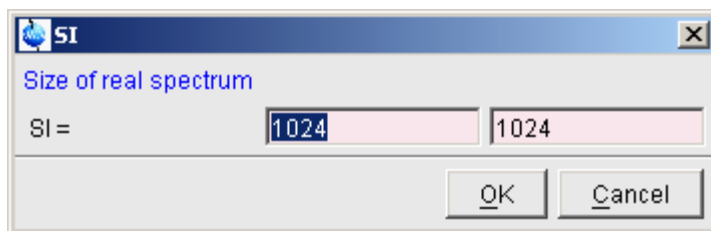
Enter the parameter name on the command line. For example to set the size:

1. Enter *si*

for 1D data, the following dialog box will appear:



for 2D data, the following dialog box will appear:



2. Specify the desired value(s), e.g. 32768 or 32k
3. Click **OK**

How to Set Processing Parameters from the Parameter Editor

To open the processing parameter editor:

- Click the **ProcPars** tab in the Tab bar of the data window.
- or*
- Enter **edp** on the command line.

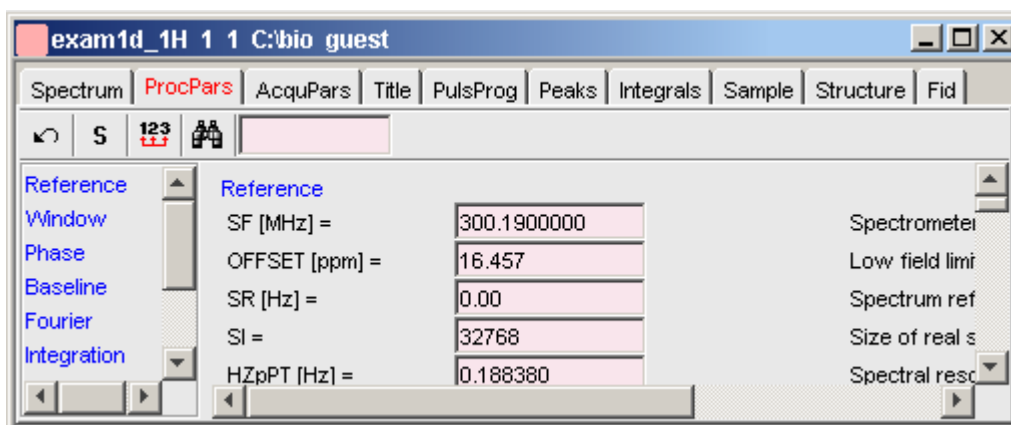




Figure 5.1

At the left of the parameter editor window you will see a list of parameter sections.


The processing parameter editor supports the following functions:

- ☞ Enter (part of) a parameter name in the search field and click  .
- ☞ Click a parameter section, e.g. **Phase** at the left of the dialog box. The section becomes highlighted and the corresponding parameters will appear in the right part of the dialog box.
- ☞ Click in a parameter field, e.g. PHC0, to set the parameter value.
- ☞ Hit the **Tab** key to jump to the next parameter field.
- ☞ Hit **Shift+Tab** to jump to the previous parameter field.
- ☞ Use the scroll bar at the right of the dialog box to move to parameters further up or down in the dialog box.

How to Undo the Last Processing Parameter Change


- ☞ Click the following button:
 -  Undo last parameter change.

How to Display Processing Status Parameters

- ☞ Click the following button:
 -  Show processing status parameters.

Note that the command **dpp** opens the parameter editor and automatically shows the status parameters.

How to Change Processed Data Dimensionality

- ☞ Click the following button:
 -  Change data dimensionality.

This changes the number of parameter columns and value of the processing parameter PPARMOD.

The parameter editor does not allow you to modify status parameters. Processing status parameters reflect the status of the processed data and are used for further processing, display or plotting. Changing them can make the dataset inconsistent. In rare cases, however, it can be useful to change a status parameter and TOPSPIN allows you to do that from the command line. If, for instance, you want to change the F1 status parameter MC2 of a 2D dataset, you have to enter:

s mc2

Note that the command **s** is used for 1D, 2D and 3D dataset. TOPSPIN automatically recognizes the dimensionality of the data and displays the parameter in all relevant dimensions. Note that, for example, the parameter MC2 only exists in F1.

5.2 Acquisition Parameters

How to Set Acquisition Parameters

Acquisition parameters can be set/changed as follows:

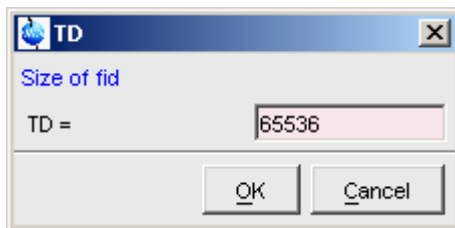
- from the parameter editor: click the **AcquPars** tab or enter **eda**
- from the command line: e.g. enter **td**
- from the interactive parameter adjustment window (enter **gs**)

How to Set an Acquisition Parameter from the Command Line

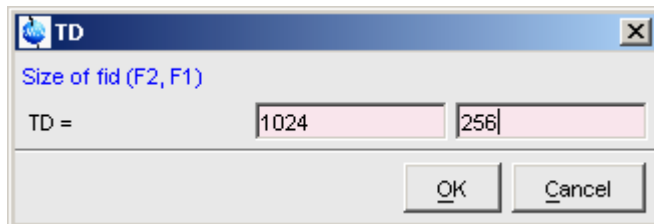
Enter the parameter name on the command line. For example to set the time domain size:

1. Enter **td**

for 1D data, the following dialog box will appear:



for 2D data, the following dialog box will appear:



2. Specify the desired value(s), e.g. 65536 or 64k
3. Click **OK**

How to Set Acquisition Parameters from the Parameter Editor

To open the acquisition parameter editor:

☞ Click the *AcquPars* tab in the Tab bar of the data window.

or

☞ Enter *eda* on the command line.

Fig. 5.2 shows an example of the acquisition parameter editor with the Experiment parameters displayed.

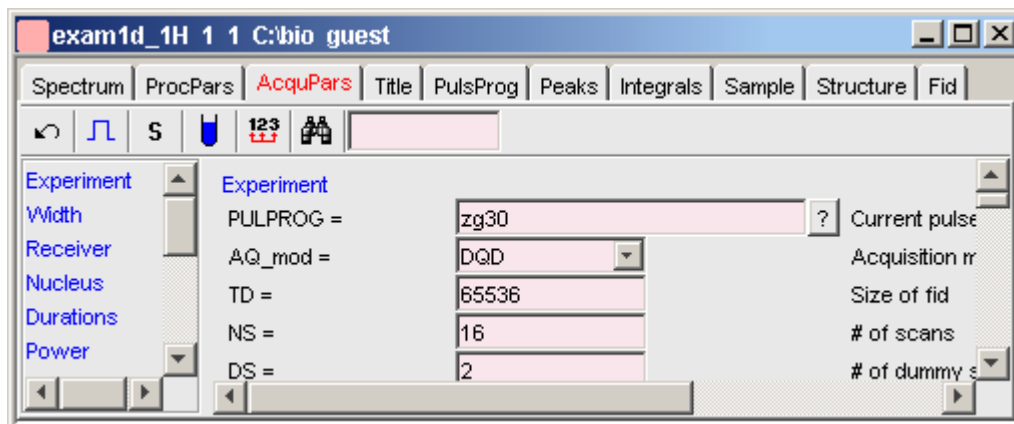



Figure 5.2


The processing parameter editor supports the following functions:

- ☞ Enter (part of) a parameter name in the search field and click .
- ☞ Click a parameter section, e.g. **Experiment** at the left of the dialog box. The section becomes highlighted and the corresponding parameters will appear in the right part of the dialog box.
- ☞ Click in a parameter field, e.g. TD, to set the parameter value.
- ☞ Hit the **Tab** key to jump to the next parameter field.
- ☞ Hit **Shift+Tab** to jump to the previous parameter field.

- ☞ Use the scroll bar at the right of the dialog box to move to parameters further up or down in the dialog box.


How to Undo the Last Acquisition Parameter Change


- ☞ Click the following button:

 Undo last acquisition parameter change.

How to Set Pulse Program Parameters

- ☞ Click the following button:


 Show pulse program parameters [*ased*]

The button will change to . To make this the default setting:

Click *Options* → *Preferences*, click *Miscellaneous*, check the entry "Show reduced parameter set (*ased*)" and click *OK*.

How to Display Acquisition Status Parameters

- ☞ Click the following button:

 Show acquisition status parameters.

Note that the command *dpa* opens the acquisition parameter editor and automatically shows the status parameters.

How to Get Probehead/Solvent dependent Parameters


- ☞ Click the following button:

 Set probehead/solvent dependant parameters [*getprosol*].

Probehead and solvent dependant parameters can be set up with the command *edprosol*.

How to Change Acquisition Data Dimensionality

- ☞ Click the following button:

 Change data dimensionality.

This changes the number of parameter columns and value of the acquisition parameter *PARMODE*.

How to Set Lock Parameters

Enter the command ***edlock*** and set the lock parameters in the appearing dialog box. For a detailed description of ***edlock***, please refer to the Acquisition Reference manual or enter ***edlock?*** on the command line.

How to Set Routing Parameters

Enter the command ***edasp*** and set the routing parameters in the appearing dialog box. For a detailed description of ***edasp***, please refer to the Acquisition Reference manual or enter ***edasp?*** on the command line.

Chapter 6

Data Processing

6.1 Interactive Processing

Interactive processing allows full control over the processing sequence. However, it requires detailed knowledge about the required parameters (see chapter 5.1) and commands. Therefore, it is only suitable for the advanced user. New or intermediate users are recommended to use the Processing Guide for semi-automatic processing (see chapter 6.2).

How to Process Data with Single Commands

Data can be processed by entering single commands on the command line. A typical 1D processing sequence would be:

em : exponential window multiplication

ft : Fourier transform

apk : automatic phase correction

sref : automatic calibration (referencing)

abs : automatic baseline correction

This allows you full control over each individual processing step.

How to Process data with Composite Commands

Data can also be processed with so called composite commands. These are combinations of single processing commands. The following composite commands are available.

- ***ef*** : Exponential multiplication + Fourier transform
- ***efp*** : Exponential multiplication + Fourier transform + phase correction
- ***fmc*** : Fourier transform + magnitude calculation
- ***fp*** : Fourier transform + phase correction
- ***gf*** : Gaussian multiplication + Fourier transform
- ***gfp*** : Gaussian multiplication + Fourier transform + phase correction

They can be entered on the command line or clicked from the menu. For the latter option:

☞ Click *Processing* → *More transforms* → *Shortcuts*

Just like single commands, composite commands can be used in Macros, AU programs and Python programs.

6.2 Semi-automatic Processing

How to Use the Processing Guide in *Automatic* mode

The Processing Guide in automatic mode guides you through the entire processing sequence of data selection, processing, printing and archiving with minimum user interaction.

1. Click *Processing* → *Processing Guide*

The Processing Guide window will appear as an integral part of the current data window (see Fig. 6.1).

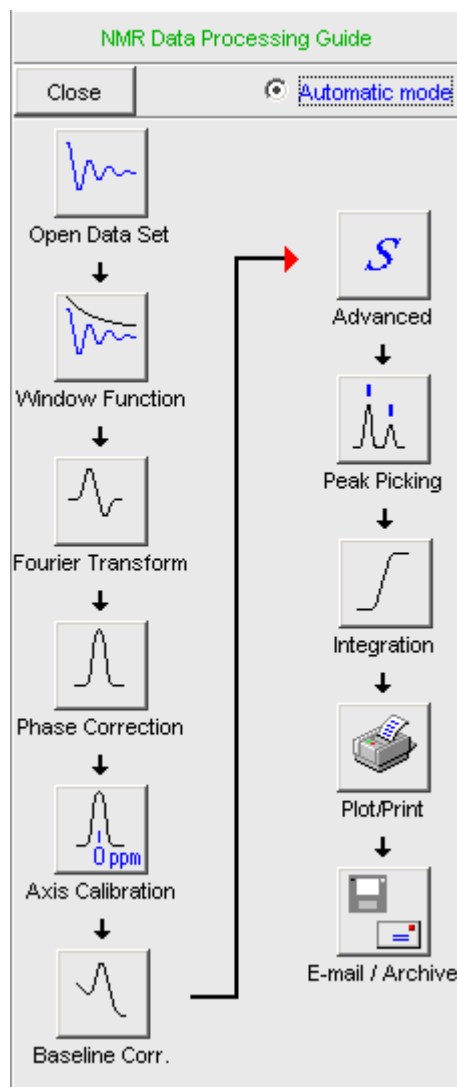


Figure 6.1

2. In the Processing Guide window:
 - a) Check *Automatic mode*
 - b) Click *Open data set* and click **OK** to open a dataset manually, e.g. from the browser or click **Browse** to open the File Chooser.

c) Click *Window function* → *Fourier Transform* → *etc.*

Each processing step will be executed without user interaction.

How to Use the Processing Guide in *Interactive mode*

The Processing Guide in interactive mode guides you through the entire processing sequence of data selection, processing, printing and archiving requiring some user interaction.

1. Click *Processing* → *Processing Guide*

The Processing Guide window will appear as an integral part of the current data window.

2. In the Processing Guide window:

a) Uncheck *Automatic mode*

b) Click *Open data set* and click *OK* to open a dataset manually, e.g. from the browser or click *Browse* to open the File Chooser.

c) Click *Window function* → *Fourier Transform* → *etc.*

For each step a dialog box will appear where you can enter options, parameters etc. For details on these items, please refer to the corresponding commands in the Processing Reference Guide.

6.3 Processing Data with AU programs

Data processing can be performed by using AU programs. An AU program is actually a C-program which contains TOPSPIN commands (macros) and/or C-language statements. Various standard AU programs are delivered with TOPSPIN. A typical 1D processing AU program is *proc_1d*. A simplified version of this AU program is:

```
EF
APK
SREF
ABS
AUTO PLOT
QUIT
```

It executes the commands *ef*, *apk*, *sref*, *abs* and *autoplot*. To run this AU

program, just enter **proc_1d** on the command line ¹. You can create your own AU programs with the command **edau**. Note that an AU program must end with **QUIT** or **QUITMSG**("your message"), and that all statements must be specified in capital letters. For more information on AU programs, please refer to the AU program reference manual:

Click **Help** → **Programming** → **AU programming**

As an alternative to AU programs, you can also write Python programs, which allow you to use TOPSPIN commands, User Interface functions and Graphic functions. For more information:

☞ Click **Help** → **Programming** → **Python Programming**

6.4 Serial Processing using Python programs

TOPSPIN allows you to process a series of datasets using serial scripts. The dataset list and command(s) to be used can be easily setup from the TOPSPIN interface as follows. Enter the command **serial** on the command line. This will open the

1. Before you can use any Bruker AU program, **expinstall** must have been executed once.

dialog window shown in Fig. 6.2.

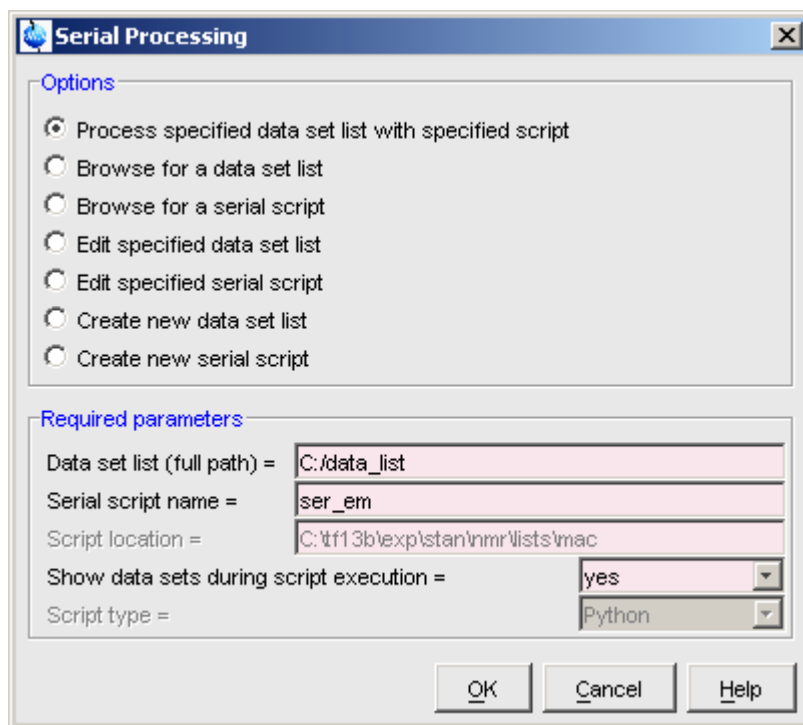


Figure 6.2

Here you can set up and start data processing of a series of datasets using scripts, which can be TOPSPIN macros or Python programs.

The dialog offers you the following options:

Process specified data set list with specified script

Process the data in the specified dataset list using the specified serial script. The flag ***Show data sets during script execution*** allows you to either display the currently processed dataset or remain on the current dataset.

Browse for a dataset list

Browse for an existing dataset list, starting in the users home directory.

Browse for a serial script

Browse for existing serial script. Depending on the parameter *Script type*, the browser opens the `../exp/stan/nmr/py` for *Script type* Python or `../exp/stan/nmr/lists/mac` *Script type* macro. TOPSPIN, by default, searches for files named `ser_*`.

Edit specified data set list

View or edit the list specified in the field *Data Set List*.

Edit specified serial script

View or edit the script specified in the field *Serial script name*

Create new data set list

Opens a dialog box for finding datasets. Proceed as follows:

1. Specify all search criteria and click **OK**
2. In the appearing Search result box:
 - a) Select the desired datasets.
 - b) Click **Save selection..**
3. In the appearing browser:
 - a) Navigate to the desired list directory.
 - b) Enter or select the list filename.
 - c) Click **OK**

Alternatively, you can create a dataset list manually. The format of a list entry is:

```
<name> <expno> <procno> <dir> <user>
```

An example of a dataset list is:

```
exam1d_13C 1 1 C:/bio guest  
exam1d_13C 2 1 C:/bio guest  
exam1d_13C 3 1 C:/bio guest
```

Create new serial script

Opens the appropriate editor to create a new script. For *Script type* Python, **edpy** is executed, for *Script type* macro, **edmac** is executed. The name of a serial script (macro of Python) must start with `ser_`. Python scripts must have the extension `.py`. A standard example is the python script `ser_efp.py`.

An example of a simple processing sequence is exponential window multiplication, Fourier transform and automatic phase correction of a 1D dataset. A TOPSPIN macro performing this task would be:

```
ef  
apk
```

A Python programs performing the same task would be:

```
EF()  
APK()
```

Note that Python programs are much more versatile than macros. Details on Python programming can be found under:

Help → Programming → Python programming

Note that serial processing can also be started as follows:

- Click ***Processing → Serial Processing...***

or

- Click ***File → Run***, select ***Execute Serial script on Data set list*** and click ***OK***

Chapter 7

Printing/Exporting Data

7.1 Printing/plotting Data

How to Print/Plot from the Menu

The current data window can be printed as follows:

1. From the TOPSPIN menu:

☞ Click the button  in the upper toolbar

or Click *File* → *Print*

or Enter *print* or *Ctrl+p*

All these actions are equivalent; they open the Print dialog box (see Fig. 7.1).

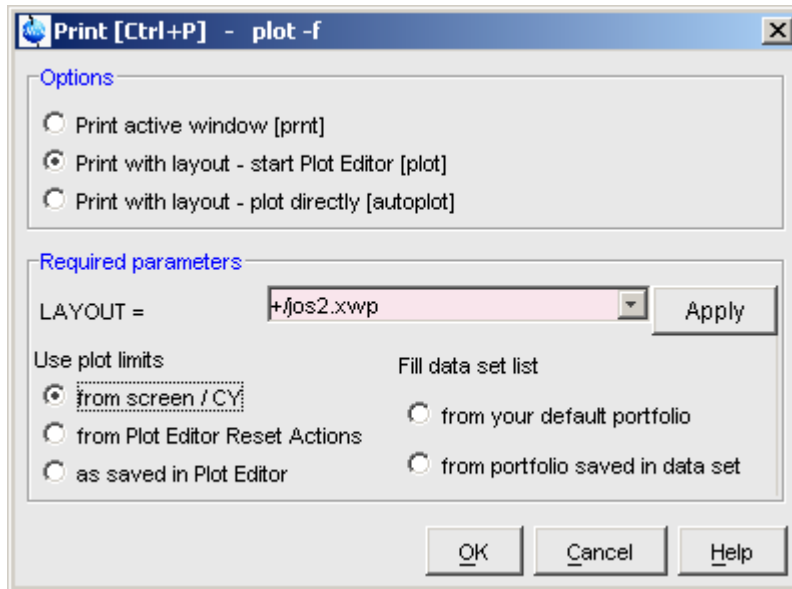


Figure 7.1

2. In the Print dialog box:
 - a) Select *Print active window [prnt]*
 - b) Click *OK*

Before printing starts, the operating system print dialog box will appear. Here you can, for example, select the printer name and the printer properties.

The Print dialog box (see Fig. 7.1) contains two further options:

- *Print with layout - start Plot Editor [plot]*
If you select this option and click *OK*, the Plot Editor will be started. This option is equivalent to entering *plot* on the TOPSPIN command line.
- *Print with layout - plot directly [autoplot]*
Selecting this option activates the Plot Editor layout list box. Select the desired layout and click *OK* to print. Standard layouts are delivered with TOPSPIN. They use the Windows default printer. User defined layouts use the printer defined in the Plot Editor. On a 1D dataset, only 1D layouts are listed, on a 2D dataset only 2D layouts are listed etc.

For the last two options, the following [Required Parameters](#) are available:

Use plot limits:

- ***from screen/ CY***
The plot limits and maximum intensity are used as they are on the screen (processing parameter F1P, F2P and CY, respectively).
- ***from Plot Editor Reset Actions***
The plot limits and maximum intensity are set according to the Plot Editor Reset Actions (right-click inside the Plot Editor data field and choose ***Automation*** to set the Reset Actions).
- ***as saved in Plot Editor***
The plot limits and maximum intensity are set in the specified layout

Fill dataset list:

- ***from your default portfolio***
The portfolio contains the current TOPSPIN dataset plus the data from the default Plot Editor portfolio.
- ***from port folio saved in dataset***
The portfolio contains the current TOPSPIN dataset plus the data from the portfolio stored in this dataset.

How to Plot Data from the Processing guide

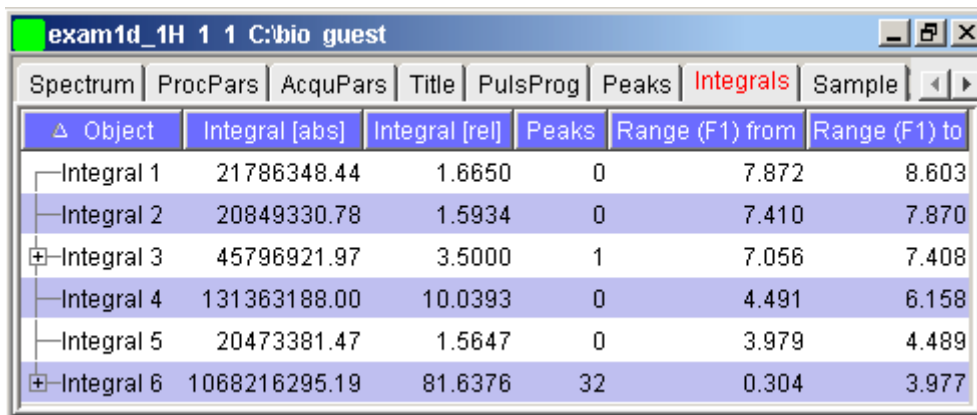
Printing/plotting data can be done from the Processing guide by clicking the ***Plot/Print*** button. If ***Automatic mode*** is checked, the active data window will be printed as it appears in the screen. If ***Automatic mode*** is unchecked, you will get the dialog box as displayed in Fig. 7.1.

How to Plot Data with the Plot Editor

The Plot Editor can be started from the Plot Editor or from the command line (command ***plot***). The Plot Editor allows you to create layouts and plot data. The complete functionality is described in the online manual, which can be opened from the Plot Editor ***Help*** menu.

How to Print the Integral list

1. Click the ***Integrals*** tab of the data window (see Fig. 7.2).
2. Enter ***print*** or ***Ctrl+p*** to print it.

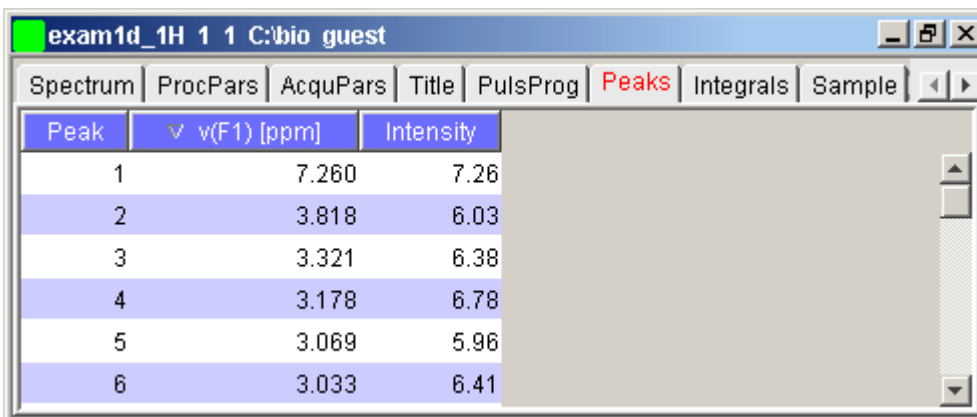


Object	Integral [abs]	Integral [rel]	Peaks	Range (F1) from	Range (F1) to
Integral 1	21786348.44	1.6650	0	7.872	8.603
Integral 2	20849330.78	1.5934	0	7.410	7.870
Integral 3	45796921.97	3.5000	1	7.056	7.408
Integral 4	131363188.00	10.0393	0	4.491	6.158
Integral 5	20473381.47	1.5647	0	3.979	4.489
Integral 6	1068216295.19	81.6376	32	0.304	3.977

Figure 7.2

How to Print the Peak list

1. Click the *Peaks* tab of the data window (see Fig. 7.3).
2. Enter *print* or *Ctrl+p*



Peak	v(F1) [ppm]	Intensity
1	7.260	7.26
2	3.818	6.03
3	3.321	6.38
4	3.178	6.78
5	3.069	5.96
6	3.033	6.41

Figure 7.3

7.2 Exporting Data

How to Copy data to Other Applications

Under MS Windows, you can easily copy the data window contents to other applications. To do that:

☞ Click *Edit* → *Copy* [*copy*].

This will copy the data window contents to:

- the clipboard. After that you can paste the clipboard contents to any Windows application.
- the Windows Metafile file `screenDump.wmf` in the user properties directory (enter *hist* to locate this directory). The Windows Metafile can be imported by other applications or send to a different computer.

Please note:

Some programs, when importing spectra from the clipboard or metafile, do not display the contained information correctly. Particularly when you resize the imported graphics, sections of the text, the spectrum, or the axis sometimes have disappeared. Usually this is only a display problem. When you print the respective page, the representation is correct.

How to Store (Export) a Data Window as Graphics File

The clipboard and metafile formats are resizable vector formats. In addition to this, TOPSPIN allows you to save the contents of a data window in a graphics file of selectable type, e.g. `.png`, `.tif`, `.wmf` etc. To do that:

1. Click *File* → *Export...* [*exportfile*].
2. Navigate to the storage folder.
3. Enter the destination filename and extension.
4. Click *Export*

The resolution of such a *screen dump* equals the resolution of your screen. When you import a graphics file into an other program, you may lose information when resizing the graphics.

Chapter 8

1D Display

8.1 The 1D Data Window

The 1D data window consists of a data field, a title bar, a Tab bar and buttons. Fig. 8.1 shows a data window with a 1D spectrum.

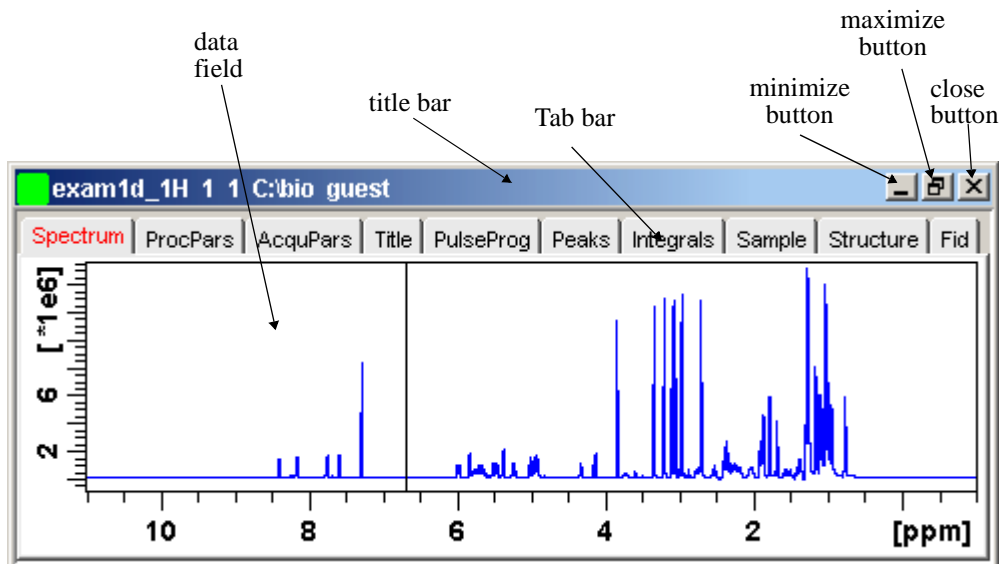


Figure 8.1

8.2 Displaying one Dataset in Multiple windows

TOPSPIN allows you to display one dataset in multiple data windows. This is, for example, convenient to view various regions or various objects (spectrum, fid, parameters etc.) of the same dataset.

How to Reopen a Dataset in a Second/Third etc. Window

1. Select (activate) the desired dataset.
2. Click *File* → *Reopen* [*reopen*].

Multiple data windows with the same dataset are indicated with a number in square brackets, e.g. [1], in the title bar (see Fig. 8.2).

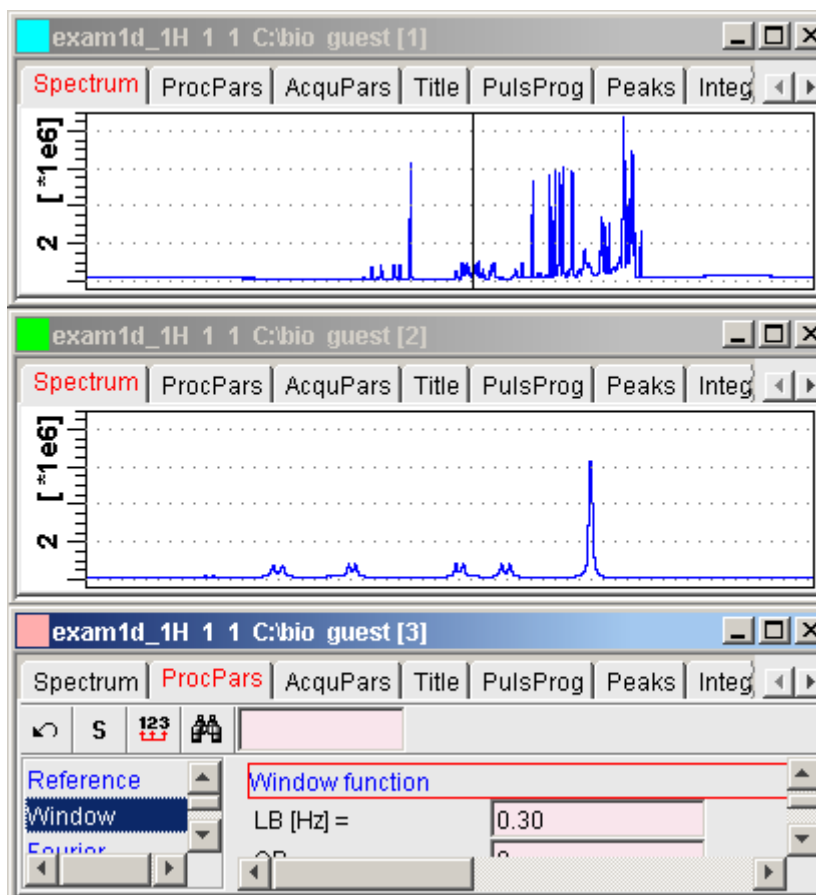


Figure 8.2

How to Rescale or Shift one Dataset in Multiple windows

Display manipulation buttons like ***2** and **↑** only work on the active data window. The same counts for the keys **Alt+PageUP** and **Alt+PageDown**. However, when used with the control key, they work on all windows, for example:

☞ Hit **Ctrl+*2**, **Ctrl+Alt+PageUp** or **Ctrl+Alt+PageDown**

8.3 Changing the Display of a 1D Spectrum or FID

TOPSPIN offers buttons to scale or shift the spectrum vertically and horizontally.


How to Change the Vertical Scaling of the FID or Spectrum

☞ Hit one of the following the keys:

- **Alt+PageUp** : Increase the intensity by a factor of 2.
- **Alt+PageDown** : Decrease the intensity by a factor of 2.
- **Alt+Enter** : Reset the intensity.






or

☞ Click-hold the button and move the mouse:

 Change the intensity smoothly.

or

☞ Click one of the following buttons:


-  Increase the intensity by a factor of 2 [***2**].
-  Increase the intensity by a factor of 8 [***8**].
-  Decrease the intensity by a factor of 2 [**/2**].
-  Decrease the intensity by a factor of 8 [**/8**].
-  Reset the intensity [**.vr**].

Alternatively, you can enter the corresponding commands as specified between square brackets [].

To manipulate all data windows, press the **Ctrl** key while clicking one of the above buttons.

How to Change the Horizontal Scaling of the FID or Spectrum


☞ Click-hold the button and move the mouse:

 Zoom in/out smoothly.

or

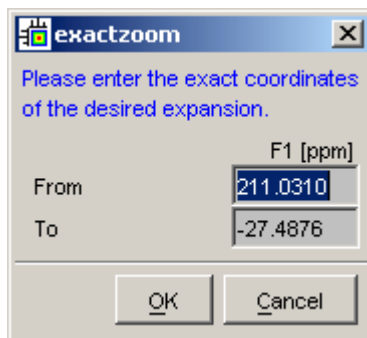
☞ Click one of the following buttons:

 Zoom in (increase horizontal scaling) [**.zi**].

 Zoom out (decrease horizontal scaling) [**.zo**].


 Perform an exact zoom via a dialog box [**.zx**].


a) Enter the coordinates of the desired region in the dialog box:




b) Click **OK**

 Retrieve previous zoom [**.z1**].

 Reset zooming (horizontal scaling) to full spectrum [**.hr**].


 Reset horizontal (zooming) and vertical (intensity) scaling [**.all**].

 Retain horizontal and vertical scaling when modifying dataset or changing to different dataset [**.keep**]. Effects all data windows.

Alternatively, you can enter the corresponding commands as specified between square brackets [].





How to Shift a Spectral Region to the Left or to the Right

☞ Click-hold the following button and move the mouse:

 Smoothly shift to left or right.

or

☞ Click one of the following buttons:

-  Shift to the left, half of the displayed region [`.sl`].
-  Shift to the right, half of the displayed region [`.sr`].
-  Shift to the extreme left, showing the last data point [`.sl0`].
-  Shift to the extreme right, showing the first data point [`.sr0`].

Alternatively, you can enter the corresponding commands as specified between square brackets [].

How to Shift the Spectrum Up or Down



To shift the FID or spectrum display up or down:

☞ Click-hold the button and move the mouse:

-  Smoothly shift the spectrum baseline up/down.

or

☞ Click one of the following buttons:

-  Shift the spectrum baseline to the middle of the data field [`.su`].
-  Shift the spectrum baseline to the bottom of the data field [`.sd`].

Alternatively, you can enter the corresponding commands as specified between square brackets [].

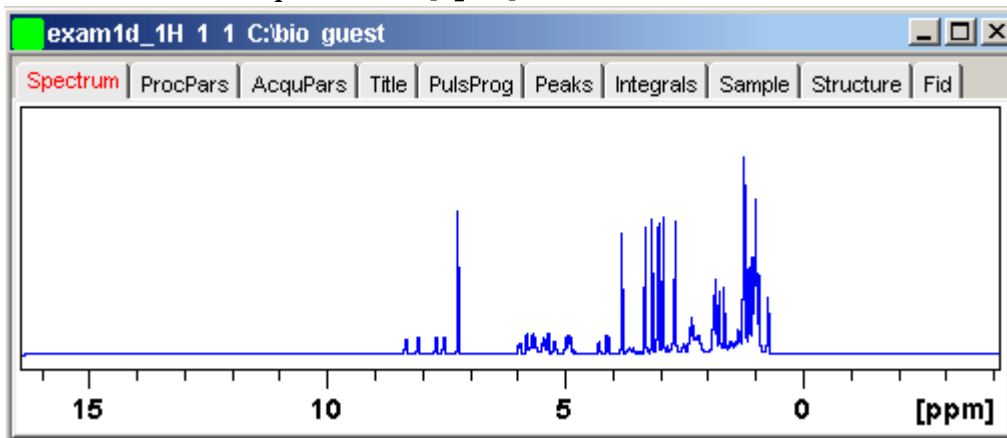
8.4 Using the Tab bar

Tabs of the data window can be activated by clicking them or by entering the corresponding commands, as specified between square brackets, on the command line. Note that command line commands always work on the currently selected (active) data window.

The Tab bar can be configured from the User Preference box (command `set`).

How to Display the Spectrum

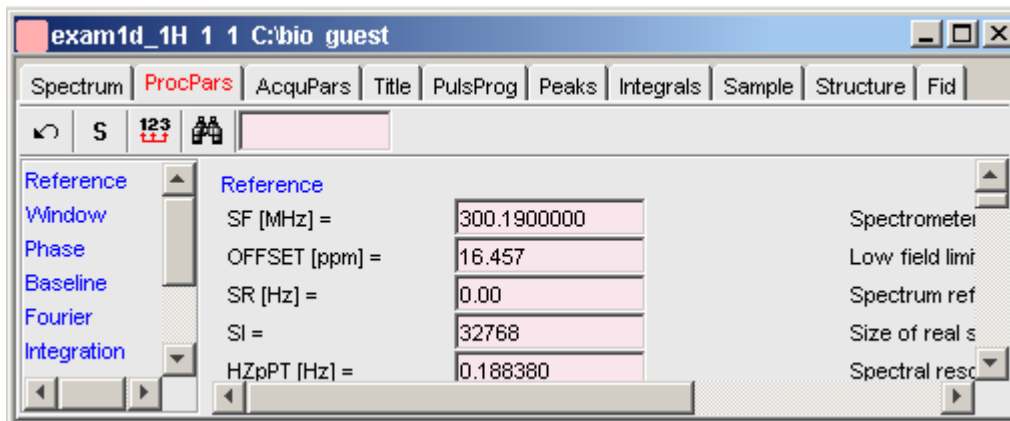
☞ Click the *Spectrum* tab [*spec*]



This displays the processed data. If these do not exist, the text 'No processed data available' appears.




How to Set Processing Parameters

☞ Click the *ProcPars* tab [*edp*]



This opens the processing parameter editor (see also chapter 5.1). The following extra buttons are available:

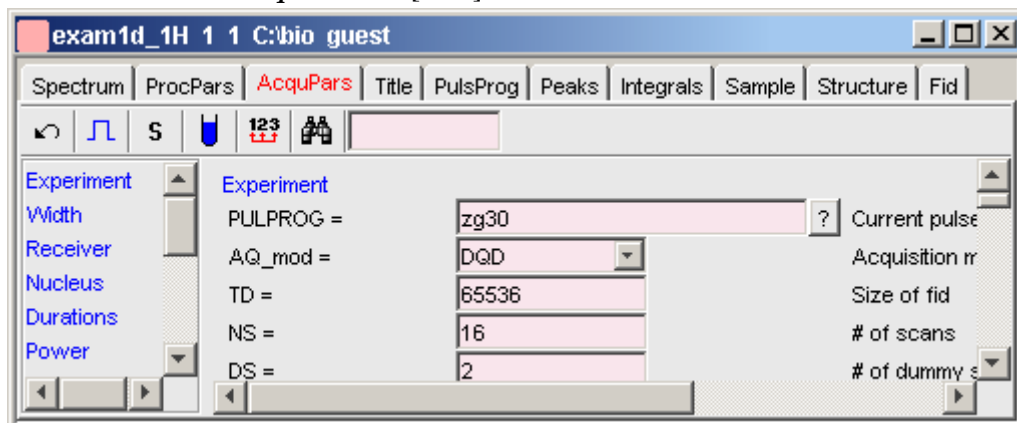
☞ Undo last value change. Can be used to undo multiple changes.

-  Status parameter display. The button turns green when activated [*dpp*].
-  Change processed dataset dimensionality (parameter PPARMOD).
-  Search for specified parameter.







Changed parameters are automatically saved.

How to Set Acquisition Parameters

☞ Click the *AcquPars* tab [*eda*]



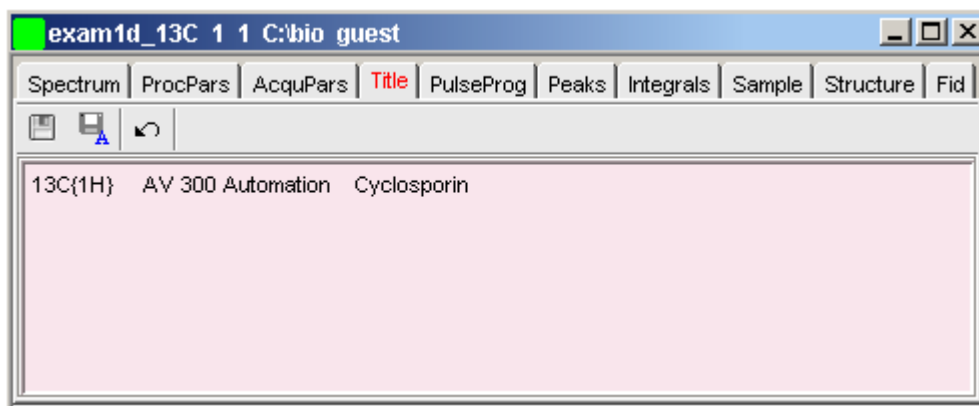
This opens the acquisition parameter editor (see also chapter 5.1)). The following extra buttons are available:

-  Undo last value change. Can be used to undo multiple changes.
-  Show pulse program parameters [*ased*].
-  Status parameter display. The button turns green when activated [*dpa*].
-  Set probehead/solvent dependant parameters [*getprosol*].
-  Change raw dataset dimensionality (parameter PARMODE).
-  Search for specified parameter.




Changed parameters are automatically saved.

How to Edit the Title

Click the *Title* tab [*edt i*]

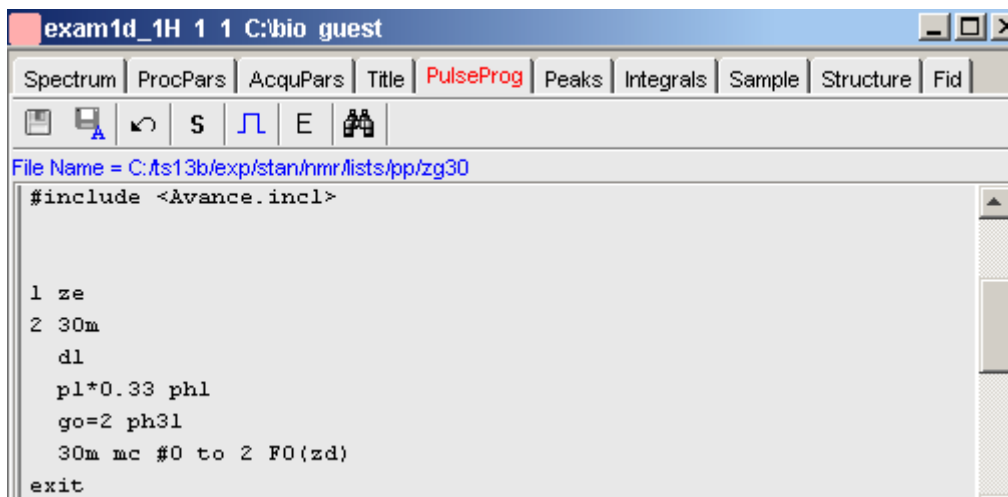


This allows you to edit the title that appears in the data window and on the plot.







-  Save the title file under its current name.
-  Save the title file under a new name.
-  Reload the title file. Undo modifications since the last save.

How to Edit the Pulse Program

Click the *PulsProg* tab [*edcpu1*]

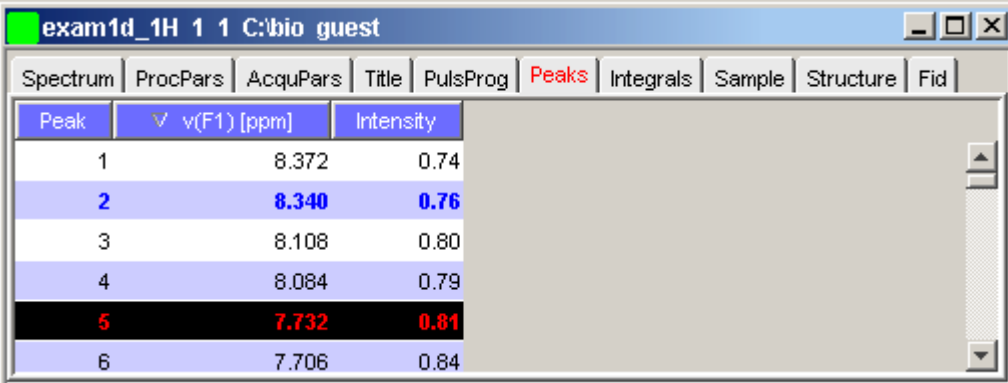


This allows you to edit the current pulse program. The following extra buttons are available here:

-  Save the pulse program under its current name.
-  Save the pulse program under a new name.
-  Reload the pulse program. Undo modifications since the last save.
-  Switch to status pulse program.
-  Show the pulse program in an external editor.
-  Start the graphical pulse program display [*nmrsim*].

How to Display the Peak list

Click the *Peaks* tab




Peak	$\nu(F1)$ [ppm]	Intensity
1	8.372	0.74
2	8.340	0.76
3	8.108	0.80
4	8.084	0.79
5	7.732	0.81
6	7.706	0.84

Figure 8.3

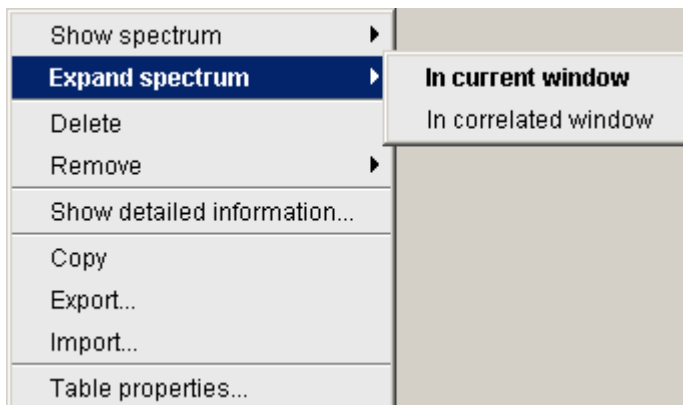
This displays the peak list. By default, the peak list shows the following entries:

- Peak* : the peak number
- $\nu(F1)$ [ppm] : the chemical shift
- Intensity*: the peak intensity

Display the spectral region around a peak

-  Right-click the desired peak

this will open the popup menu shown in Fig. 8.4.

**Figure 8.4**

Here you can choose from the following options:

- ☞ *Show spectrum* → *In correlated window*
to open a new data window showing the full correlated spectrum
- ☞ *Expand spectrum* → *In current window*
to change the current data window to spectrum display, showing the region around the selected peak
- ☞ *Expand spectrum* → *In correlated window*
to open a new data window showing the region around the selected peak

Export entries of the peak list

Entries of the peak list can easily be exported to Excel or any other program as follows:

1. For multiple peaks:
Select the desired entries while pressing the **Ctrl** or **Shift** key
2. Right-click an entry to open the popup menu (see Fig. 8.4)
3. Click one of the following menu items:
 - **Copy**
Copy the selected peak(s) entry to the Clipboard. Equivalent to clicking **Edit** → **Copy** or hitting **Ctrl+c**. Copied peaks can easily be pasted in any other application such as Excel.

- **Export...**

Export selected peaks. Opens a dialog box where you can specify the export file (.cvs, .txt or .xml). Check the box in the lower-left corner to export the selected peaks only or uncheck it to export the entire list. Then click *Export*.

Import a peak List

A peak list from a different dataset can be imported as follows:

1. Right-click an entry to open the popup menu (see Fig. 8.4)
2. Click **Import...**
3. In the appearing dialog box, navigate to the directory where the list resides and select the peak list (.xml or .txt).

As such you can import a peak list from a different dataset or a previously exported list from the current dataset. Note that peak picking commands store the peak list in the processed data directory under the name `peak.txt`.

Delete/remove peaks from the peak list

To delete one peak:

- ☞ Right-click the peak and choose *Delete* from the popup menu

To delete multiple peaks:

1. Select the peaks while pressing the *Ctrl* or *Shift* key
2. Right-click one of the peaks and choose *Delete* from the popup menu

To remove possible duplicate peaks:

- ☞ Right-click any entry and choose *Remove* → *Duplicate peaks*

To remove possible peaks outside of the spectrum:

- ☞ Right-click any entry and choose *Remove* → *Peaks positioned outside of the spectrum*

Shortcuts

Double-click a peak : zoom into spectrum, i.e. show region around that peak

Enter key : zoom into spectrum, i.e. show region around selected peak(s))

Delete key : delete the selected peak(s) from the peak list

Ctrl+c : copy selected peaks to the Clipboard.

Ctrl+a : select all peaks.

Home : select the first peak

End : select the last peak.

Shift+Home : select current and first peak and all in between.

Shift+End : select current and last peak and all in between.

Note that these keys only work when the cursor focus is in the data window.

More features

The right-click popup menu has the following entries (see Fig. 8.4):

Table properties

Set the shown columns and specify their decimal places.

Show detailed information

Show peak information, dataset information and peak picking parameters.

When you move the cursor over the peak list, the active peak will be highlighted in blue (see peak 3 in Fig. 8.5). If the correlated spectrum is also displayed, a vertical line moves along, showing corresponding position in the spectrum (see Fig. 8.5)

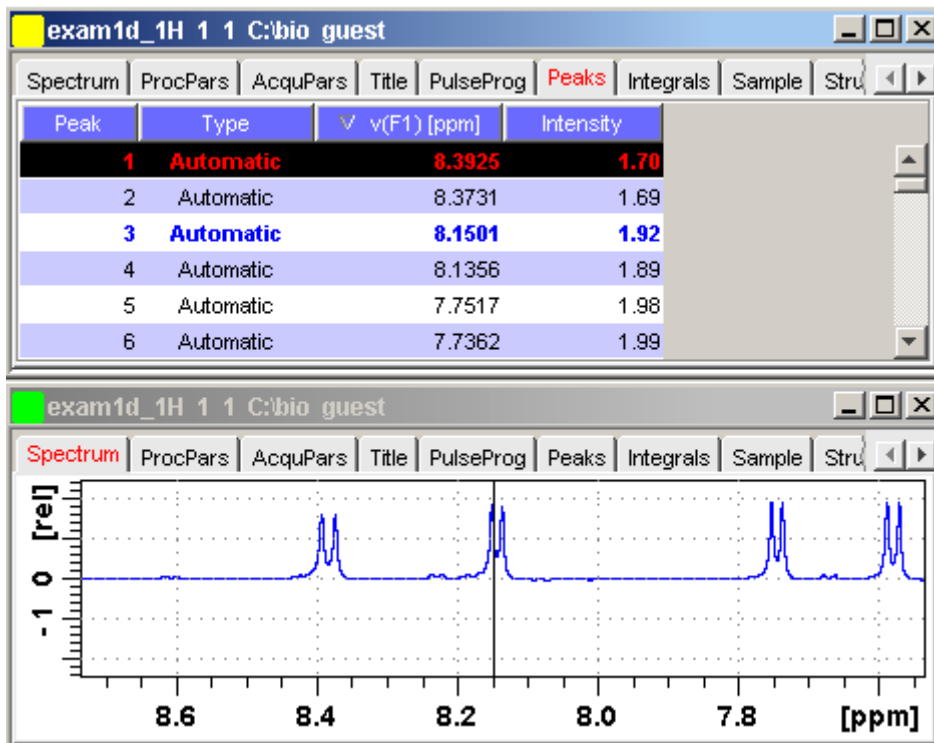


Figure 8.5

As soon as you click a peak, it is selected and displayed in **red** (see peak 1 in Fig. 8.5). Note that this peak remains selected, i.e. is used by **Enter** and **Delete**, until a different peak is selected.

To extend the peak list with *Regions*, *Type* and *Index* entries, right-click any part of the header bar.

To sort the peaks according to peak number, ppm value or intensity, click the header of the respective entry.

Note that when you delete peaks from the peak list, they are automatically removed from the corresponding file. You cannot undo a delete action.

Peaks are only available if peak picking has been done (command **pp**). The peak list can be printed with **print [Ctrl+p]**. List items can be selected with the mouse, copied with **Ctrl+c** and pasted to other applications, e.g. a text editor.

How to Display the Integral list

☞ Click the *Integrals* tab [*li*, *lipp*, *lippf*]

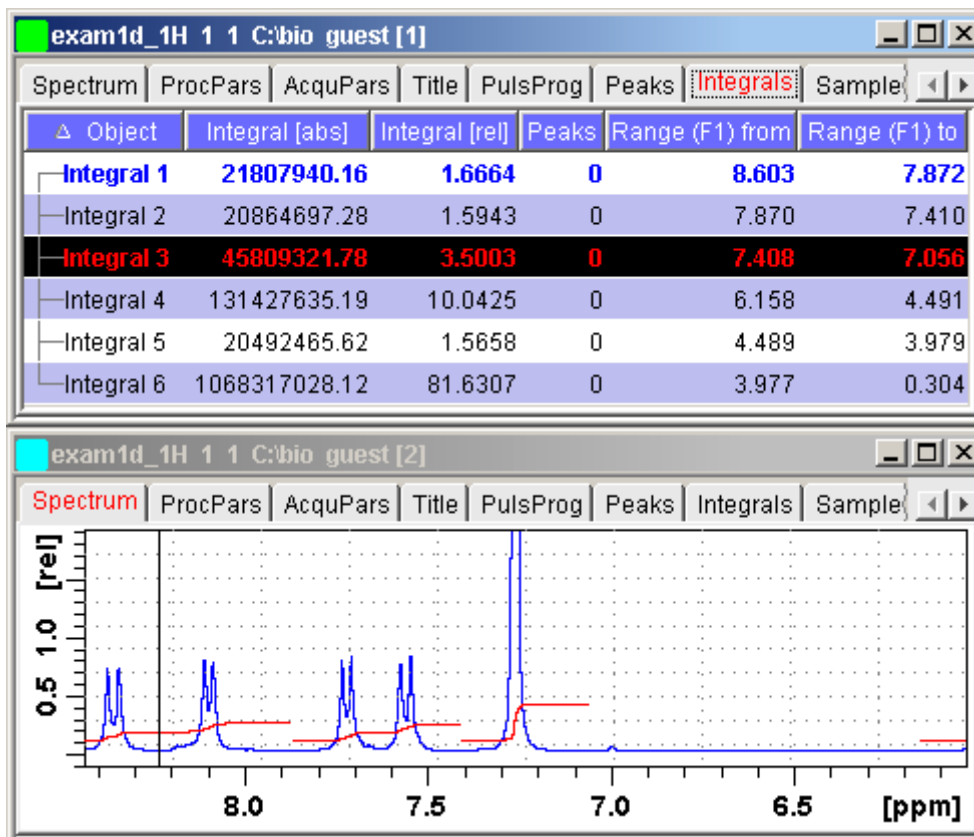


Figure 8.6

This displays the integral list (upper part of Fig. 8.6). By default, this shows the following items:

Object : the integral number

Integral [abs] : the absolute integral value

Integral [rel] : the relative integral value

Peaks : the number of peaks within the integral range

Range (F1) from : the left edge of the integral range

Range (F1) to : the right edge of the integral range

Please note the difference between the following items:

- **selected integral**: the entry that has been clicked last (Integral 3 in Fig. 8.6). If you right-click an entry, it is selected and you can execute one of the commands from the popup menu (see Fig. 8.7) The keys **Enter** and **Delete** work on the selected entry.
- **active integral**: the entry on which cursor resides (Integral 1 in Fig. 8.6). The active integral is also marked in the correlated spectrum by a black vertical line (see lower part of Fig. 8.6 and description below). When you move the cursor over the integral list, the vertical line in the correlated spectrum moves along with it and vice versa.

Display the spectral region around an integral

To display the spectral region around a particular integral:

- ☞ Right-click the desired integral

This will open the popup menu shown in Fig. 8.7.

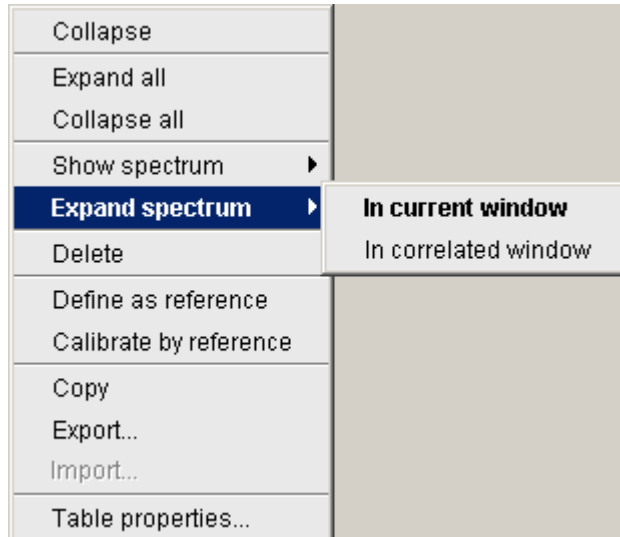


Figure 8.7

Here you can choose from the following options:

- ☞ *Show spectrum* → *In correlated spectrum*

to open a new data window showing the full correlated spectrum

☞ **Expand spectrum → In current window**

to change the current data window to spectrum display, showing the region around the selected integral

☞ **Expand spectrum → In correlated window**

to open a new data window showing the region around the selected integral (lower part of Fig. 8.6)

Note that clicking the marked entry in the right-click popup menu is equivalent to pressing of the **Enter** key.

Export/Import Entries of the Integral List

Entries of the integral list can easily be exported to Excel or any other program as follows:

1. For multiple integrals:

Select the desired entries while pressing the **Ctrl** or **Shift** key

2. Right-click an entry to open the popup menu (see Fig. 8.7)

3. Click one of the following menu items:

- **Copy**

Copy the selected integral(s) entry to the Clipboard. Equivalent to clicking **Edit** → **Copy** or hitting **Ctrl+c**. Copied integrals can easily be pasted in any other application such as Excel.

- **Export...**

Export selected integrals. Check the box in the lower-left corner to export the selected integrals only or uncheck it to export the entire list. Then click **Export**.

TOPSPIN 1.3 supports exporting 1D integrals in `.csv` format. Future version will also 1D exporting integrals in `.txt` format and importing 1D integrals. These features are already supported for 2D and 3D integrals.

Calibrate Integrals to Compare Spectra

Integrals from the current and other spectra can be calibrated with respect to a reference integral. To do that:

1. Right-click the reference integral and choose **Define as reference** from the popup menu. This will determine the calibration constant.

2. Right-click any integral and choose *Calibrate by reference*
This will divide all integrals by the calibration constant, setting the reference integral to 1.0.

Now you can read any other spectrum, and calibrate its integrals with respect to the reference integral defined above. To do that:

1. Read the spectrum
2. Enter *int* to define the integral ranges (if this has not been done yet)
3. Click the *Integrals* tab
4. Right-click any integral in the list and choose *Calibrate by reference* from the popup menu.

Note that the calibration constant is lost when TOPSPIN is restarted.

Display the integral list with peaks

The integral list in Fig. 8.6 shows only integrals. However, if peak picking has been done, the integral list also shows the peaks within each integral range (see Fig. 8.8).

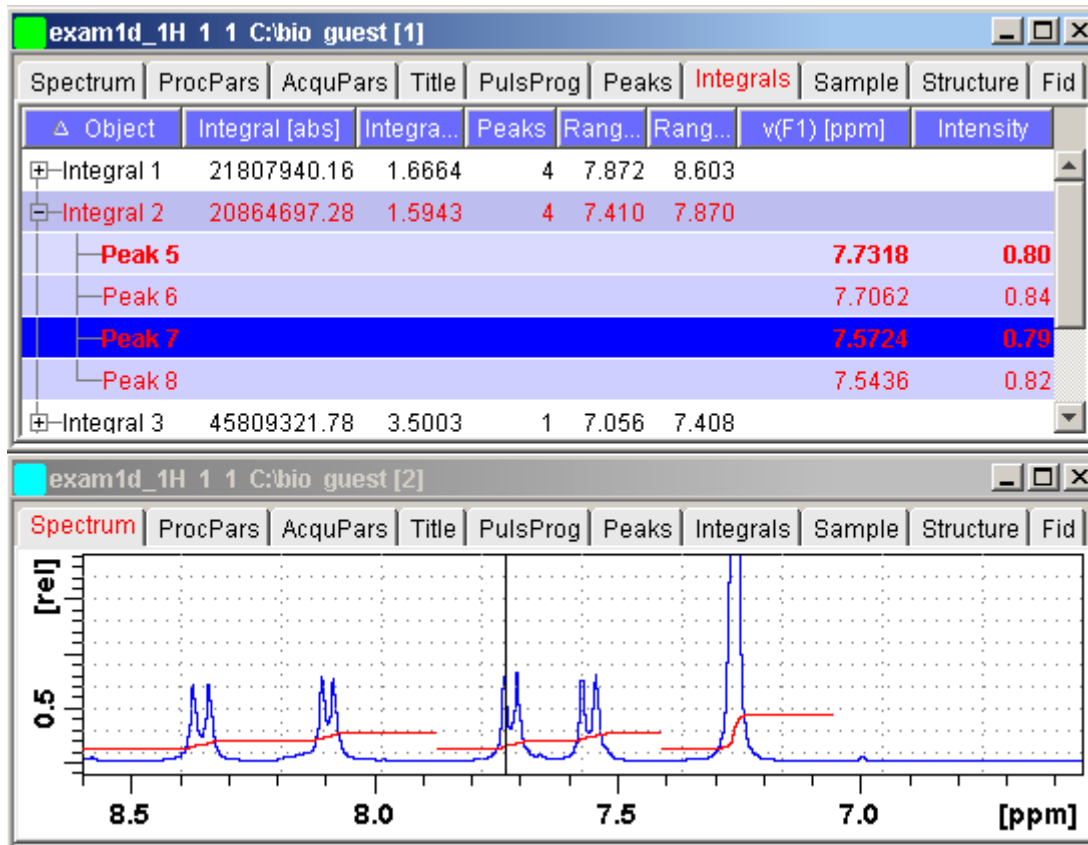


Figure 8.8

Note that the integral entries can be collapsed, (hiding the peaks) or expanded (showing the peaks). As soon as one or more integrals entries are expanded, two extra columns appear showing:

$\nu(F1)$ [ppm] : the chemical shift of the peak

Intensity: the peak intensity

Depending on whether or not integrals are expanded, the right-click popup menu contains the following extra items:

- **Expand**
Expand the current integral showing all peaks within it.

- **Expand all**
Expand all integrals showing all peaks within them.
- **Collapse all**
Collapse all integrals hiding all peaks within them.

In addition to the integral entry, an individual peak within an integral can be activated (by placing the cursor on it) or selected (by clicking it). In Fig. 8.8, peak 7 is selected and the correlated spectrum is displayed. Peak 5 is active which is also shown by the vertical line in the correlated spectrum.

Delete an Integral from the Integral List

To delete one integral:

- ☞ Right-click the integral and choose **Delete** from the popup menu

To delete multiple integrals:

1. Select the integrals while pressing the **Ctrl** or **Shift** key
2. Right-click one of the integrals and choose **Delete** from the popup menu

Shortcuts

Enter key : zoom into spectrum, i.e. show region around selected integral(s))

Delete key : delete the selected integral(s) from the integral list

Ctrl+c : copy selected integrals to the Clipboard.

Ctrl+a : select all peaks.

Home : select the first peak

End : select the last peak

Shift+Home : select current and first peak and all in between

Shift+End : select current and last peak and all in between

Double-clicking an integral will show the peaks within the integral region if they exist. If they do not exist, it will zoom into spectrum showing the integral region.

Note that these keys only work when the cursor focus is in the data window.

More features

The right-click popup menu has the following extra entry:

Table properties

to set the shown columns and specify their decimal places

How to view Sample Information

☞ Click the *Sample* tab

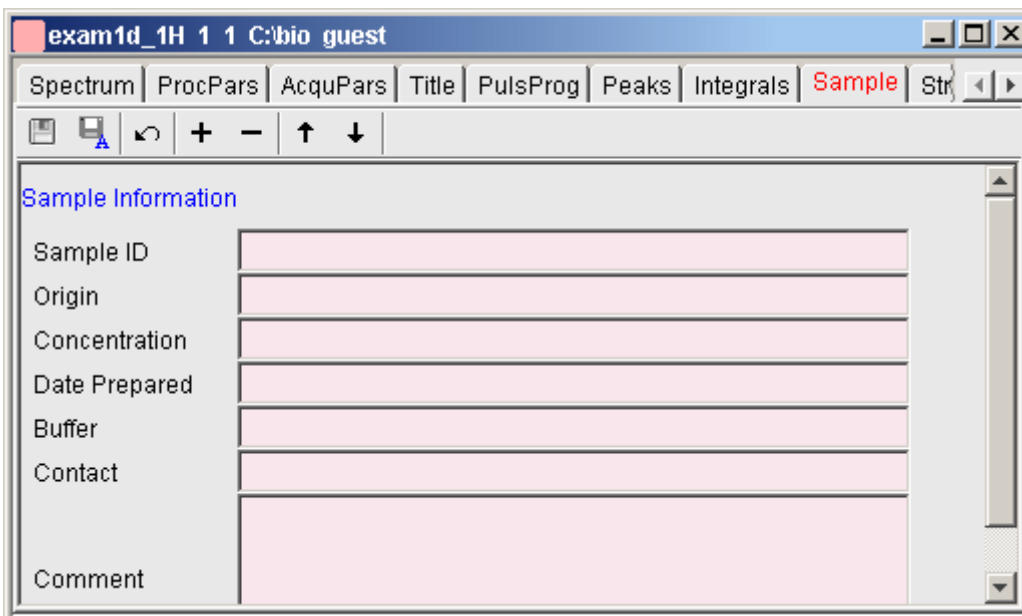






Figure 8.9

This table can be used to fill out any sample information you want to store with the dataset. The table can easily be modified or extended with the following functions:

To select an item: double-click it!

-  Save the sample information table with the dataset.
-  Save the sample information table as default.
-  Reload the original table discarding any changes
-  Add a new item to the table. You will be prompted for an identification name and the desired number of lines

- Remove the **selected item** from the table
- ↑ Move the **selected item** one place up in the table
- ↓ Move the **selected item** one place down in the table

How to Open the Jmol Molecule Structure Viewer

☞ Click the *Structure* tab [*jmo1*]:

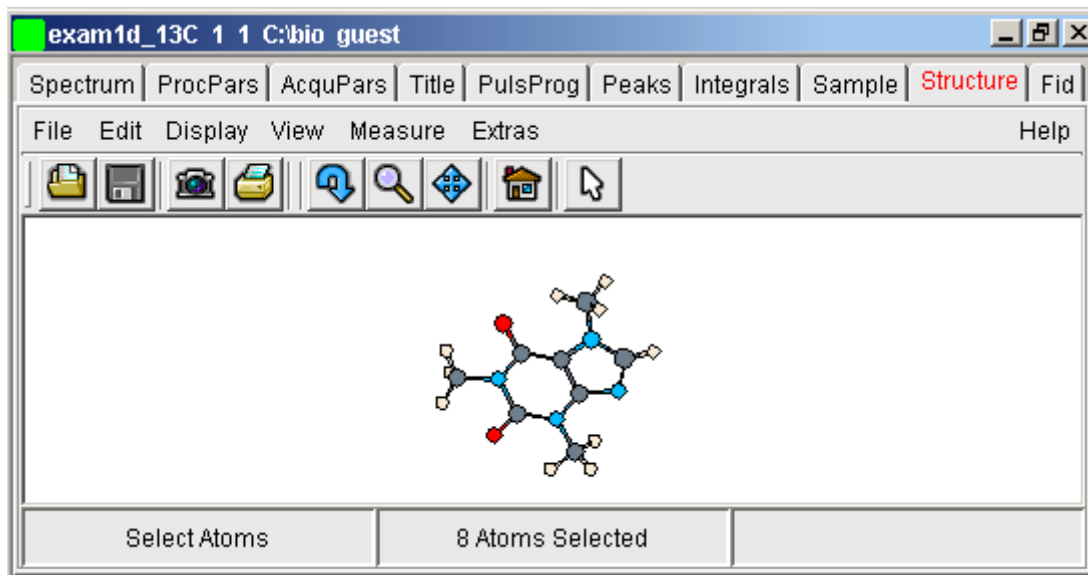


Figure 8.10

opens the *Jmol* molecule structure viewer. TOPSPIN 1.3 contains Jmol version 10. This has the following features:

- The viewer displays the structure file that resides in the *expno* of the current dataset. If this does not exist, the structure file defined by the acquisition parameter CHEMSTR is displayed. CHEMSTR can define a full pathname or a filename. In the latter case, the file is searched for in the directory defined in the User Preferences. To set this directory, click *Options* → *Preferences*, select *Directory pathnames*, enter a directory and click *OK*. If no structure file is found, you can open one by clicking *File* → *Open* in the Molecule Viewer

- The following structure file types are supported: .xyz, .mol, .pdb, .cml, .out, .mmlgp, .res, .cif, .gpr, .hin, .nwo.
- Secondary structure elements of proteins (backbone, cartoons, ribbons, ...) can be displayed in selectable sizes and colors.

- Mouse button effects:

Rotate a molecule around the x- and y-axis by pressing the left mouse button, and moving the mouse left/right or up/down, respectively.

Rotate a molecule around the z-axis by pressing the middle mouse button and moving the mouse left/right .

Zoom in or out a molecule by pressing the middle mouse button, and moving the mouse up or down.

- RASmol command scripts are supported. To send a RASmol command to the currently displayed molecule enter:

```
⌘ jmol <RASmol command>
```

Here are some example:

```
⌘ jmol zoom 400
```

```
⌘ jmol ribbon 200
```

```
⌘ jmol color ribbon yellow
```

You may create Topspin macros containing RASmol commands. Just enter **edmac** on the TOPSPIN command line and insert the RASmol commands in the appearing editor. Here is an example:

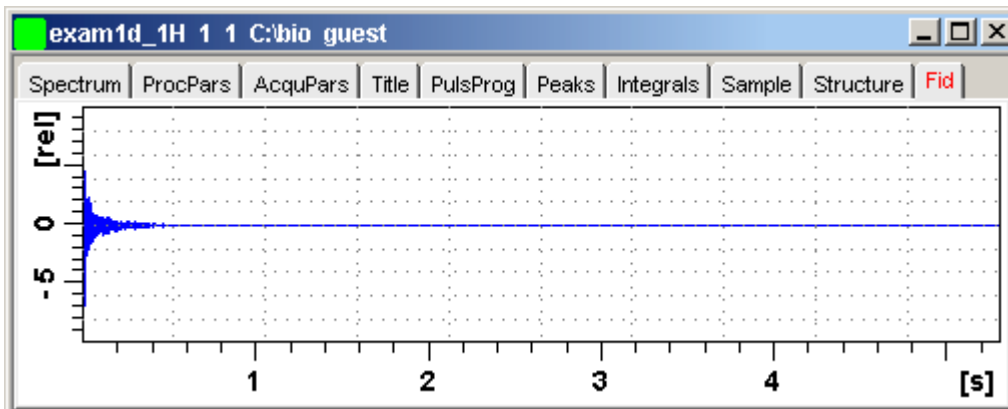
```
jmol load /mystructures/alphahelix.pdb # load a structure  
jmol backbone 0.7 # display its backbone with 0.7 Angstrom size  
jmol color backbone yellow # change backbone color  
jmol background green # change background color  
jmol zoom 200 # zoom structure
```

The available RASmol commands are described in the **Jmol** Help menu.

- Multiple molecules (or multiple aspects of one molecule) can be displayed simultaneously. To do that just open multiple data sets or open the same dataset in multiple data windows and click on the **Structure** Tab in each window.

How to Display the FID

Click the *Fid* tab [*fid*]



displays the raw data. If these do not exist, the text 'No raw data available' appears. The following additional buttons appear at the right of the lower toolbar:



Show FID in shuffled mode



Show FID in unshuffled mode

If you open a new dataset, the *Spectrum* tab is activated, no matter which tab was selected before. If you switch to any interactive manipulation mode, for example to phase correction mode, the Tab bar is replaced by a toolbar for that mode.

8.5 1D Display Options

How to Toggle between Hertz and ppm Axis Units


Click the following toggle button in the upper toolbar:



Toggle between Hz and ppm axis units [*. hz*]

How to Switch on/off the Spectrum Overview display

The spectrum overview shows the entire spectrum at the top of the data window. It is useful when only a certain region of the spectrum is displayed. In the overview, the displayed region is marked as a green area. To switch on the spectrum overview, click the following toggle button in the upper toolbar:

 Switch the spectrum overview display on/off [.ov]

To shift the displayed region, simply click-hold the green area in the overview spectrum and move the mouse (see Fig. 8.11).

How to Switch Y-axis Display

Click the following toggle button in the upper toolbar:

 Switch the y-axis display between abs/rel/off [.y]

Fig. 8.11 shows a data window with the spectrum overview on, ppm axis units, and absolute y-axis display.

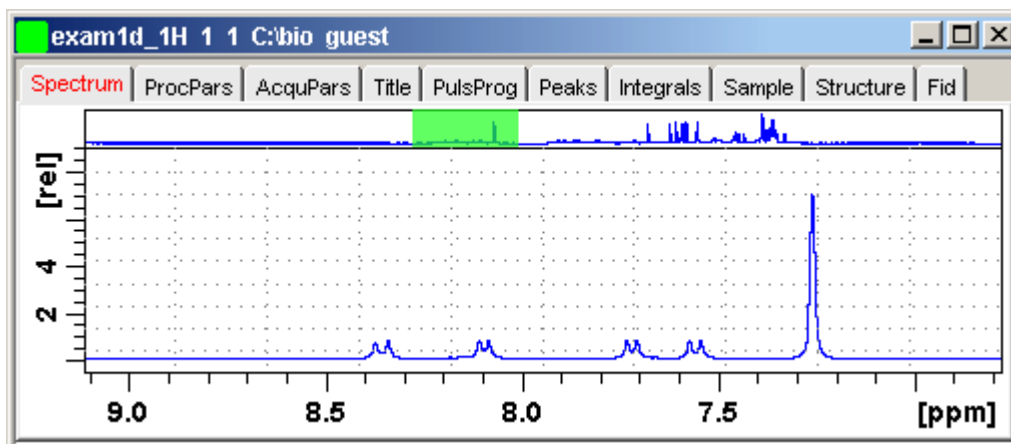


Figure 8.11

8.6 Show Display Properties/Regions/Files

If you right-click inside the data window, the following popup menu will appear:



If you choose *Display Properties...*, a dialog box (see Fig. 8.12) will appear.

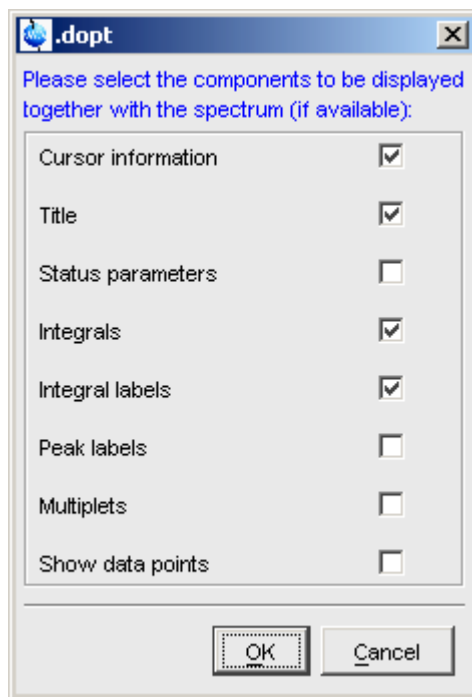


Figure 8.12

Here you can check or uncheck the spectrum components that you want to be displayed in the data window.

you the *Display Properties...* dialog box can also be opened from the *View* menu.

How to Superimpose the Cursor Information

To superimpose the cursor information on the spectrum:

1. Right-click in the data window and choose *Display Properties* [*. dopt*]
2. Check *Cursor information* in the appearing dialog box and click *OK*

How to Superimpose the Title on the Spectrum

1. Right-click in the data window and choose *Display Properties...* [*. dopt*]
2. Check *Title* in the appearing dialog box and click *OK*

How to Superimpose the main Status Parameters on the Spectrum ¹

1. Right-click in the data window and choose *Display Properties...* [*. dopt*]
2. Check *Status parameters* in the appearing dialog box and click *OK*

How to Superimpose the Integral Trails/Labels on the Spectrum

1. Right-click in the data window and choose *Display Properties...* [*. dopt*]
2. Check *Integrals* and, if desired, *Integral labels* in the appearing dialog box
3. Click *OK*

If no integrals appear, the integral regions have not been determined yet. This can be done with the *int* command.

How to Superimpose Peak Labels on the Spectrum

1. Right-click in the data window and choose *Display Properties...* [*. dopt*]
2. Check *Peak list* in the appearing dialog box and click *OK*

If no peak labels appear, peak picking has not been done yet. This can be done with the *pp* command.

How to Show Individual Data Points of the Spectrum

1. Right-click in the data window and choose *Display Properties...* [*. dopt*]
2. Check *Show data points* in the appearing dialog box and click *OK*
3. Expand the spectral region where you want to see individual points.

1. These are the status parameters that also appear on the plot.

How to Display the Main Dataset Properties

☞ Right-click inside the data window and choose **File Properties**

An information box as displayed in Fig. 8.13 will appear.

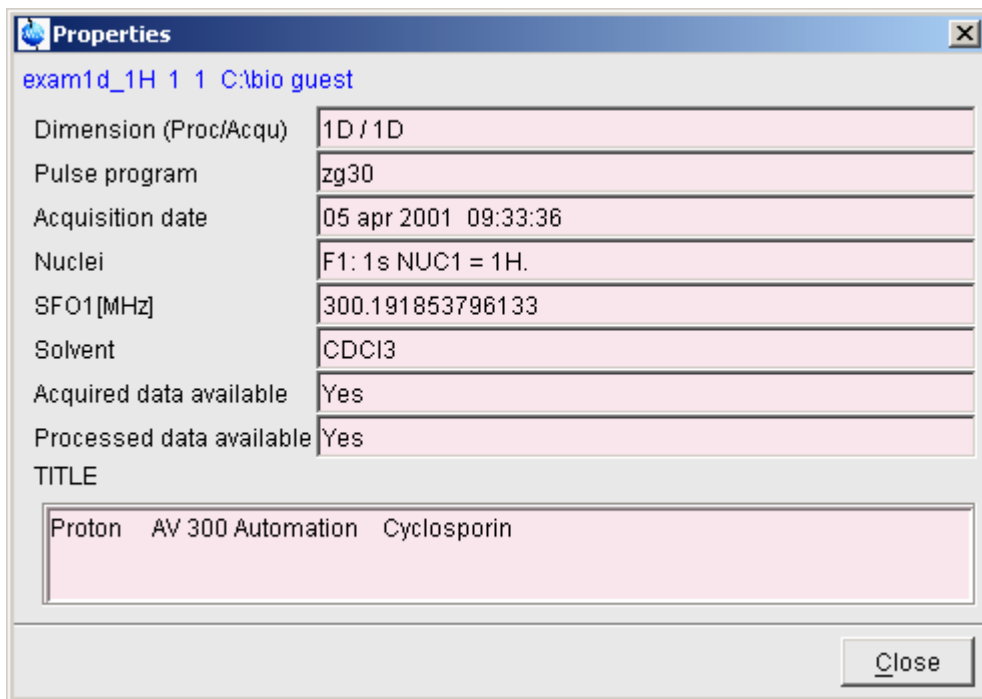


Figure 8.13

Note that this is status information which cannot be changed.

How to Display a List of Files of a Dataset

☞ Right-click inside the data window choose **Files**

Fig. 8.14 shows the file list when the **Fid** tab is active, i.e. when the raw data are displayed. It is the contents of the *expno* directory.

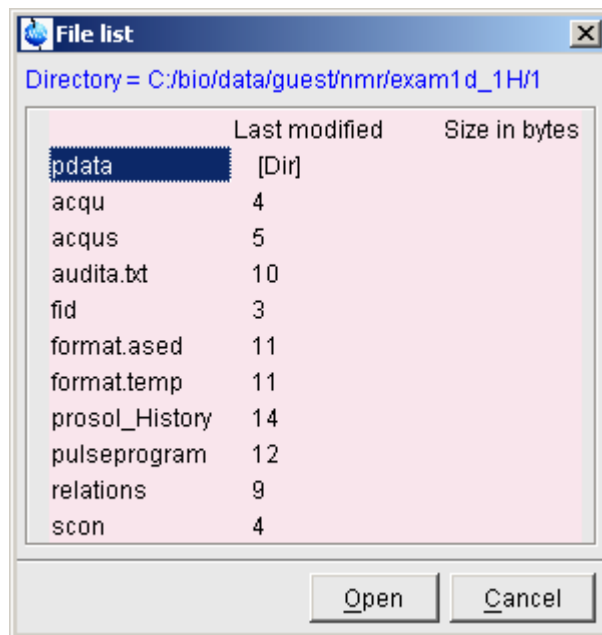


Figure 8.14

Fig. 8.15 shows the file list that appears when the *Spectrum* tab is active, i.e. when the processed data are displayed. It is the contents of the *procno* directory.

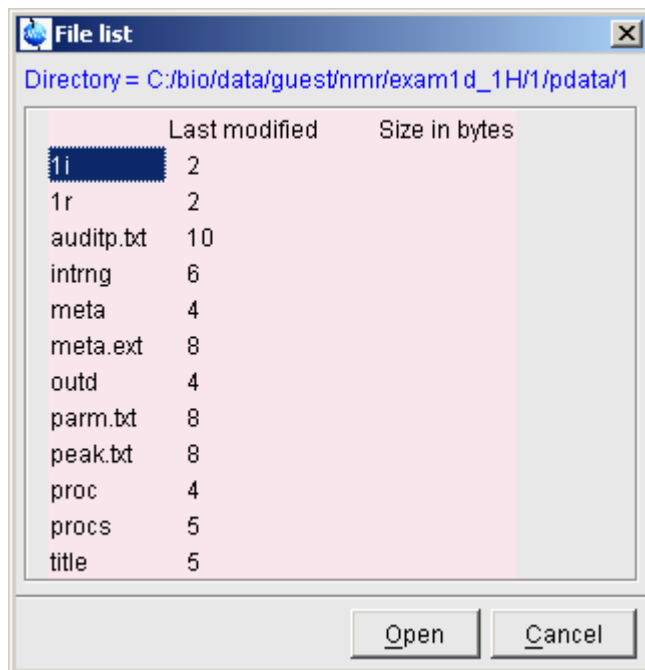


Figure 8.15

The contents of any file in the list can be displayed as follows:

1. Select a filename (it will be highlighted)
2. Click *Open*

Note that this only makes sense for ascii files, e.g. `acqu*`, `proc*` or files with the extension `.txt`.

Dataset files can also be displayed/opened with the command `expl`. This opens the Windows Explorer, or under Linux, the Konqueror or Mozilla, showing the contents of the `procno` directory.

8.7 Saving Display Region

The currently displayed spectral region can be stored as follows:

- ☞ Right-click in the data window and choose *Save Display Region To...*

This will open the dialog box shown in Fig. 8.16.

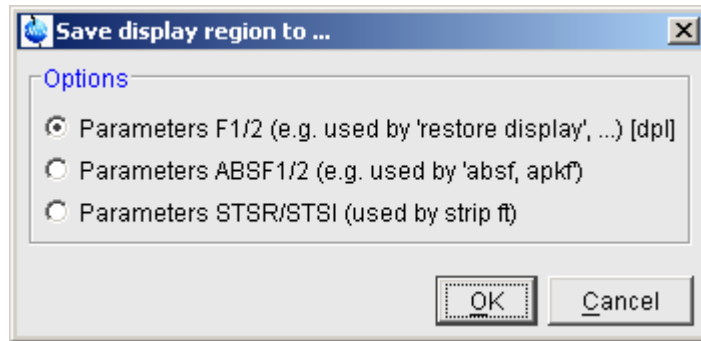


Figure 8.16

How to Save the Display Region for Re-display

- ☞ Click **Parameters F1/2** [*dpl*]

The saved region can be restored as follows:

- ☞ Right-click in the data window and choose

Restore Display Region from Params F1/2

How to Save the Display Region for Baseline or Phase Correction

- ☞ Click **Parameters ABSF1/2**

The region will be stored in the processing parameters ABSF1 and ABSF2. These are used by the commands ***absf*** and ***apkf***.

How to Save the Display Region for Strip FT

- ☞ Click **Parameters STSR/STSI**

The region will be stored in the processing parameters STSR and STSI. These are used for Fourier Transform commands like ***ft*** and ***trf*** to perform strip FT.

Chapter 9

2D Display

9.1 The 2D Data Window

The 2D data window consists of a data field, a title bar, a Tab bar and buttons.

Fig. 9.1 shows a data window with a 2D spectrum.

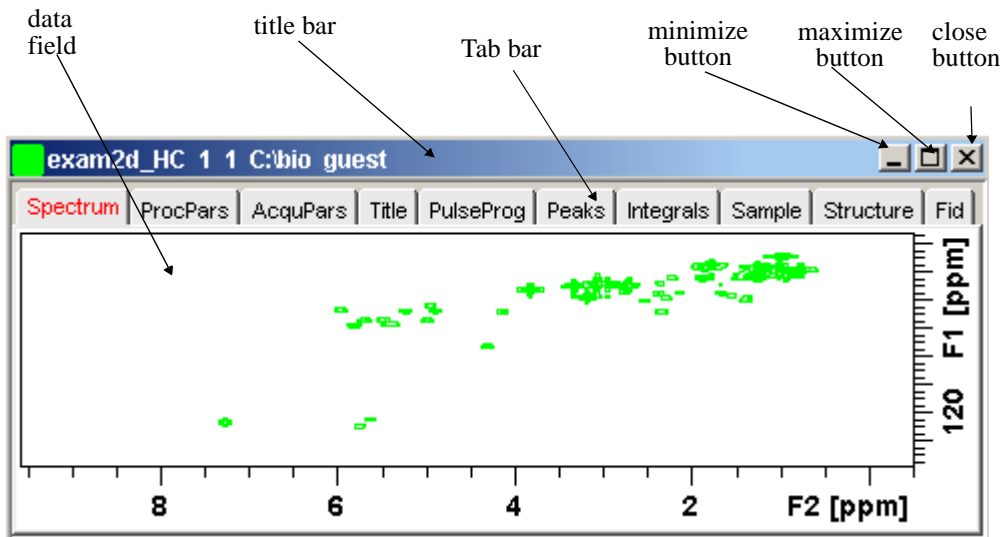


Figure 9.1

9.2 Changing the Display of a 2D spectrum

TOPSPIN offers various buttons to scale or shift a 2D spectrum both vertically and horizontally.

How to Change the Intensity Scaling

☞ Click the button:

Change the intensity scaling (contour levels) [*edlev*]


or

☞ Hit one of the keys:


- **Alt+PageUp** : Increase the intensity by a factor of 2.
- **Alt+PageDown** : Decrease the intensity by a factor of 2.
- **Alt+Enter** : Reset the intensity.

or

☞ Click-hold the button and move the mouse:


 Increase/decrease the intensity smoothly.


or


 Click one of the following buttons:

 Increase the intensity (decrease the levels) by 2 [***2**].

 Increase the intensity (decrease the levels) by 8 [***8**].

 Decrease the intensity (increase the levels) by 2 [**/2**].

 Decrease the intensity (increase the levels) by 8 [**/8**].

 Reset the intensity to the last saved intensity (contour levels) [**.vr**].

Alternatively, you can enter the corresponding commands as specified between square brackets [].

To manipulate all data windows, press the **Ctrl** key while clicking one of the above buttons.

How to Switch on/off Square 2D layout

Right-click inside the data field and click *Square Layout On/Off*

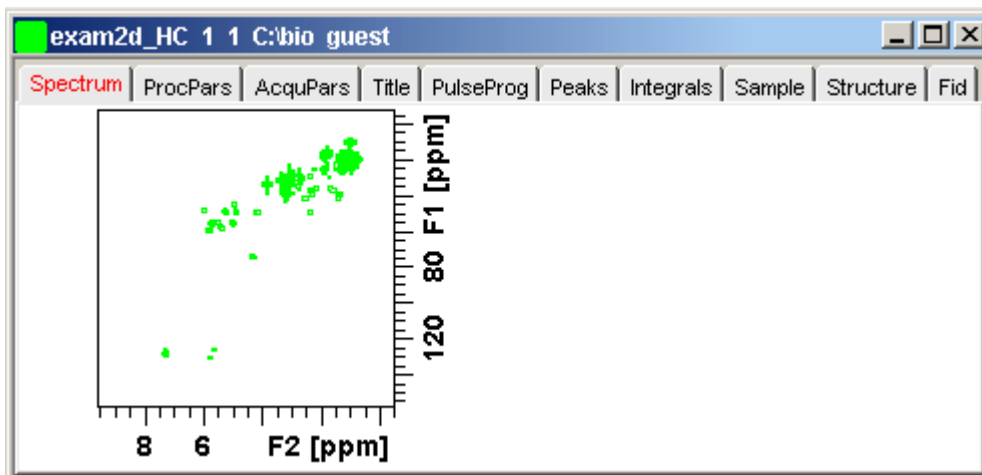





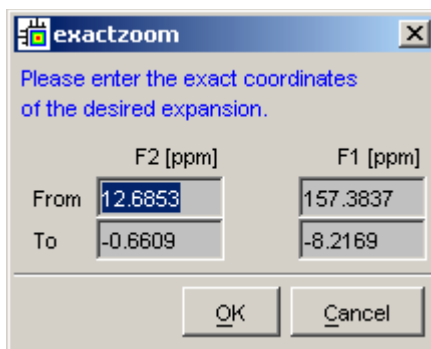
Figure 9.2

The F2 scaling will be adjusted to reach a square display.

How to Zoom a 2D spectrum in/out






☞ Click one of the following buttons:

-  Zoom in [. *zi*].
-  Zoom out [. *zo*].
-  Perform an exact zoom via a dialog box [. *zx*].




a) Enter the coordinates of the desired region in the dialog box.



b) Click **OK**

-  Retrieve previous zoom [**.z1**].
-  Reset F2 direction zooming to full spectrum [**.f2r**].
-  Reset F1 direction zooming to full spectrum [**.f1r**].
-  Reset F2- and F1-zooming to full spectrum [**.all**].
-  Retain horizontal and vertical scaling when modifying dataset or changing to different dataset. Effects all data windows [**.keep**].

Alternatively, you can enter the corresponding commands as specified between square brackets [].


How to Shift a Spectral Region in the F2 direction (left/right)

 Click one of the following buttons:

-  Shift to the left, half of the displayed region [**.s1**].
-  Shift to the right, half of the displayed region [**.sr**].


or



 Click-hold the button and move the mouse:

-  Smoothly shift up/down and left/right.

Alternatively, you can enter the corresponding commands as specified between square brackets [].


How to Shift a Spectral Region in the F1 direction (up/down)

 Click one of the following buttons:

-  Shift the spectrum up, half of the displayed region [**.su**].
-  Shift the spectrum down, half of the displayed region [**.sd**].

or

 Click-hold the button and move the mouse:

 Smoothly shift up/down and left/right.

Alternatively, you can enter the corresponding commands as specified between square brackets [].

9.3 Show Display Properties/Regions/Files

If you right-click inside the data window, the popup menu shown in Fig. 9.3 will appear.



Figure 9.3

Here you can select various display properties, region setting and file properties. If you choose *Display Properties...*, a dialog box (see Fig. 9.4) will appear.

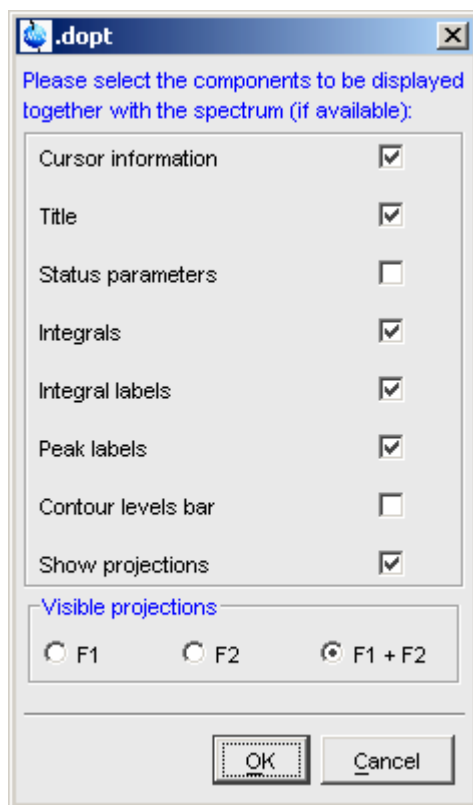


Figure 9.4

Here you can set various display options including parameter, integrals, peaks, contours and projections.

9.4 Using the Tab bar

The 2D data window is a tabbed pane. This means its contents depends on the currently **active tab** in the Tab bar. The individual tabs are basically the same as for 1D display (see chapter 8.4). There are, however, some differences, which are discussed below.

How to Set Processing Parameters

☞ Click the *ProcPars* tab [*edp*]

The 2D processing parameter editor contains a column for each of the two dimensions F2 and F1. Note that not all parameters exist in both dimensions (see Fig. 9.5).

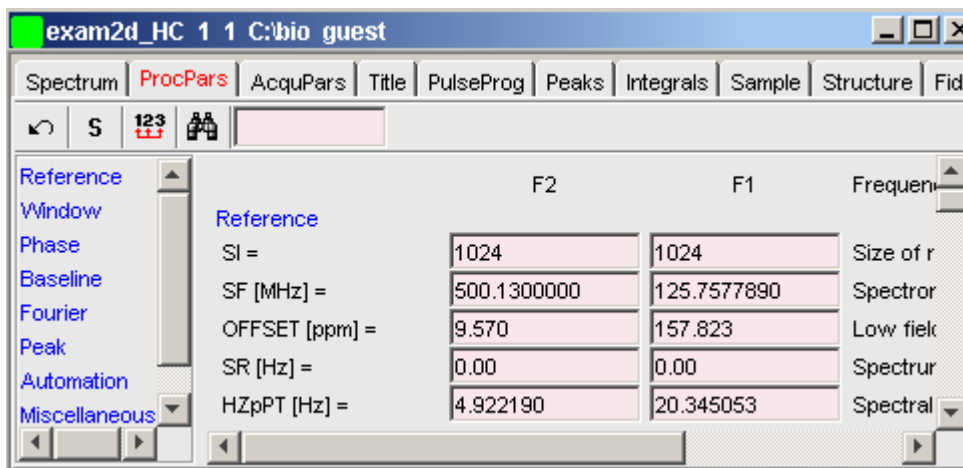


Figure 9.5

How to Set Acquisition Parameters

☞ Click the *AcquPars* tab [*eda*]

The 2D acquisition parameter editor contains a column for each of the two dimensions F2 and F1. Note that not all parameters exist in both dimensions (see Fig. 9.6).

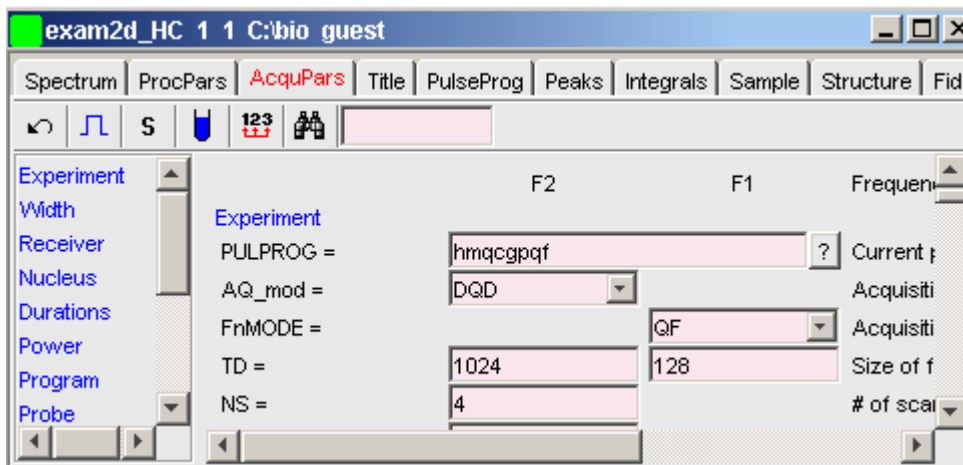


Figure 9.6

How to Display the Peak list

Click the **Peaks** tab

The screenshot shows the 'Peaks' tab in the software interface. The window title is 'exam2d_HC 1 1 C:\bio guest'. The 'Peaks' tab is selected, and the peak list is displayed as follows:

Peak	$\nu(F2)$ [ppm]	$\nu(F1)$ [ppm]	Intensity	Annotation
1	7.2485	128.2981	-32061761.86	pk1
2	4.2892	74.4625	-9039344.24	pk2
3	5.7988	59.4354	-9582402.76	pk3
4	5.3677	58.3070	-8589930.29	pk4

Figure 9.7

This displays list of peaks if these have been calculated (command **pp**). The list is basically the same as for 1D spectra. The only difference is that there are two columns for the two dimensions now and an extra column for peak annotations:

$\nu(F2)$ [ppm] : the chemical shift in the F2 direction

$\nu(F1)$ [ppm] : the chemical shift in the F1 direction

Annotation: the peak annotation

To specify or edit an annotation, click inside the *Annotation* field and enter a character string. The peak annotations are shown in the correlated spectrum (see Fig. 9.8)

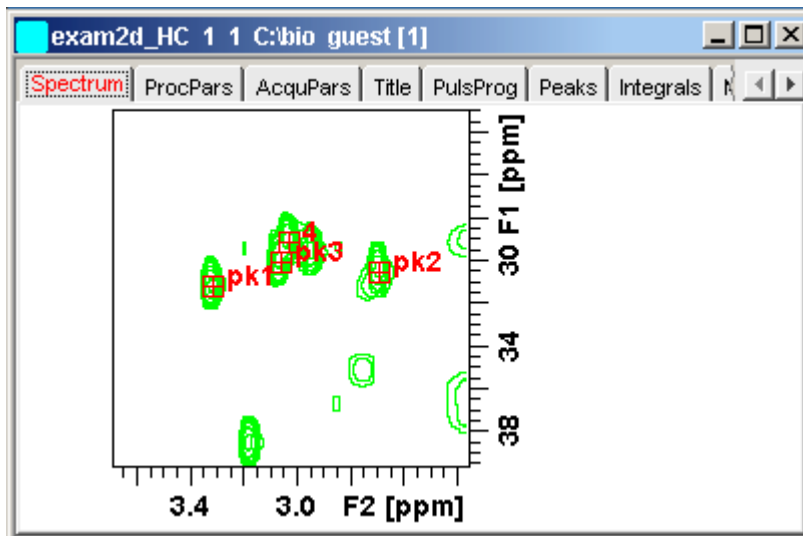


Figure 9.8

How to Display the Integral list

☞ Click the *Integrals* tab

Object	Integral [abs]	Integral [rel]	Peaks	$\nu(F2)$ [ppm]	$\nu(F1)$ [ppm]	Intensity
Integral 1	0.00	0.0000	1			
Integral 2	0.00	0.0000	1			
Integral 3	0.00	0.0000	1			
Integral 4	19711450.50	1.0000	1			
Integral 5	0.00	0.0000	1			
Integral 6	0.00	0.0000	2			
Peak 6				5.5751	126.2371	-3446703.37
Peak 7				5.6386	126.2227	-4532891.75
Integral 7	0.00	0.0000	1			

Figure 9.9

This displays list of integrals if these have been calculated (command `int`). The list is basically the same as for 1D spectra. The only difference is that, when peaks are shown, there are two columns for the chemical shift:

$\nu(F2)$ [ppm] : the chemical shift in the F2 direction

$\nu(F1)$ [ppm] : the chemical shift in the F1 direction


Furthermore, a stored or exported 2D integral list can be imported as follows:

1. Right-click an entry to open the popup menu
2. Click **Import...**
3. In the appearing dialog box, navigate to the directory where the list resides and select the integral list.

As such you can import an integral list from a different dataset or a previously exported list from the current dataset. Note that integration commands store the integral list in the processed data directory under the name `integrals.txt`. Exported integrals are stored in the files `<name>.txt` and `<name>.reg`, where `<name>` is the name specified by the user.

How to Display the FID

- Click the **Fid** tab [`fid`]

2D raw data consist of a series of FIDs which are displayed in a row. Individual FIDs can be displayed by zooming out. To do that, click  repeatedly. Now you can shift and zoom in/out the data to display different FIDs (see Fig. 9.10)

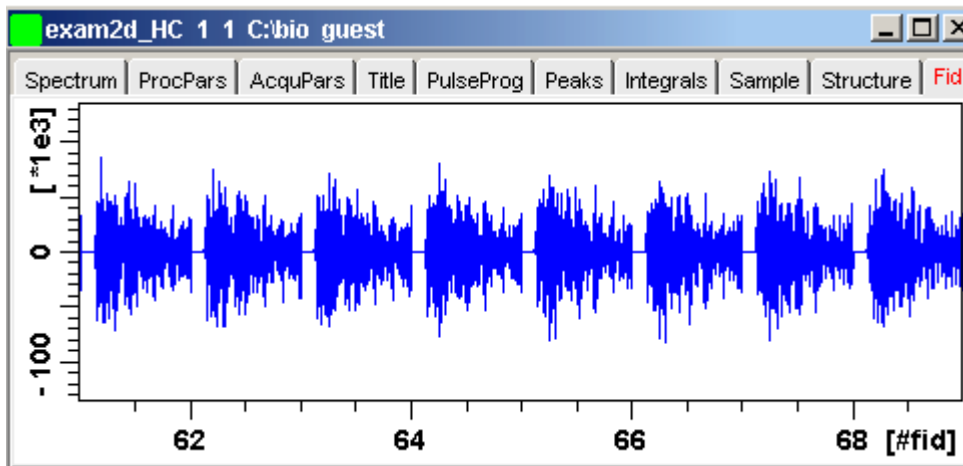



Figure 9.10

9.5 2D Display Options

How to Switch between Hertz and ppm Axis Units in F2 and F1

Click the following multi-state button in the upper toolbar:

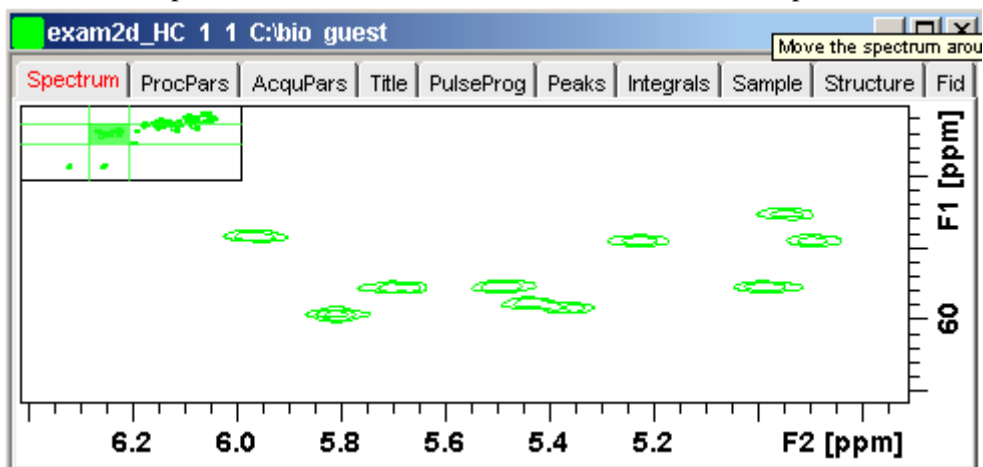
 Switch between Hz and ppm axis units in F2 and F1 [.hz]

How to Switch on/off the Spectrum Overview display

Click the following toggle button in the upper toolbar:


 Switch the spectrum overview display on/off [.ov]

With the spectrum overview on, the data window will, for example, look like this:

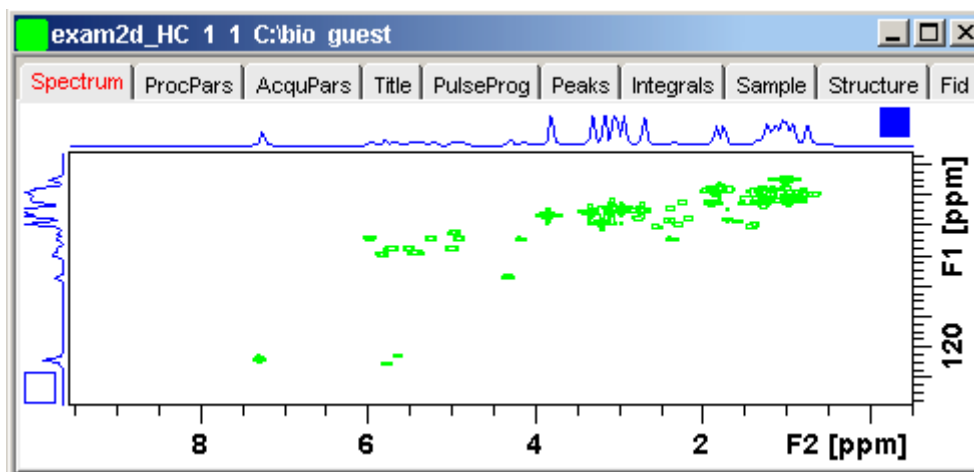


How to Switch on/off the Projection display

☞ Click the following toggle button in the upper toolbar:

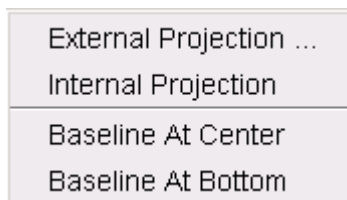
 Switch the projection display on/off [.pr]

With the projections displayed, a 2D dataset looks like this:



In this example, the F1 projection is selected as indicated by the filled blue square

whereas the F2 projection is not selected. A selected projection can be rescaled using the toolbar rescale buttons of function keys. If you right-click inside the projection area of the data window, the following popup menu appears:



Clicking ***External Projection*** opens the a dialog box where you can specify or search for a 1D dataset and display this as a projection of the current 2D dataset.

Clicking ***Internal Projection*** calculates and displays the positive projection and displays it along with the 2D spectrum.

Clicking ***Baseline at Bottom*** or ***Baseline at Center*** allows you to put the projection baseline at the respective positions.

Various other projection features are available. To access them:

click ***Processing*** → ***Calculate projections*** [*proj*]

This will open the dialog box shown in Fig. 9.11.

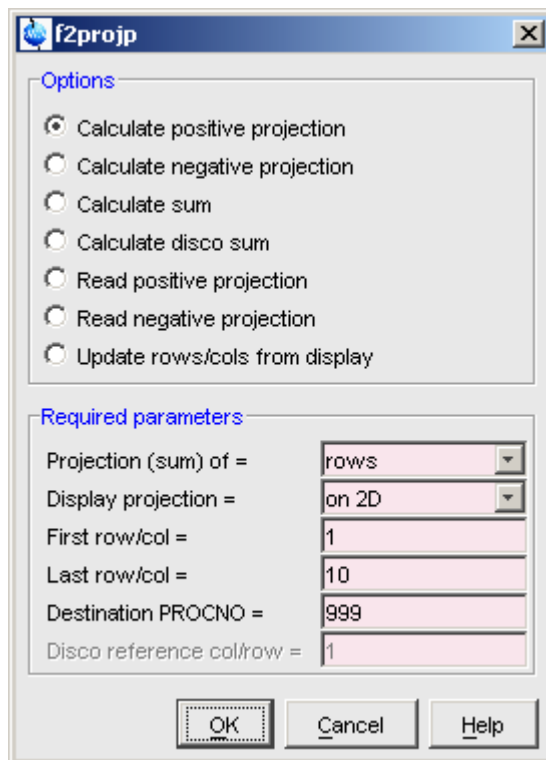


Figure 9.11

From here, you can calculate positive, negative, sum and disco projections and either show them with the 2D spectrum or display them in separate data window as a 1D data. For more details on the corresponding commands (as shown in the header of the dialog box), please refer to the Processing Reference Manual.

How to Switch on/off the Grid display

☞ Click the following multi-state button in the upper toolbar:


 Switch between 'no grid', 'axis aligned grid' and 'fixed grid' [**.gr**]

Fig. 9.12 shows an example of axis aligned grid display.

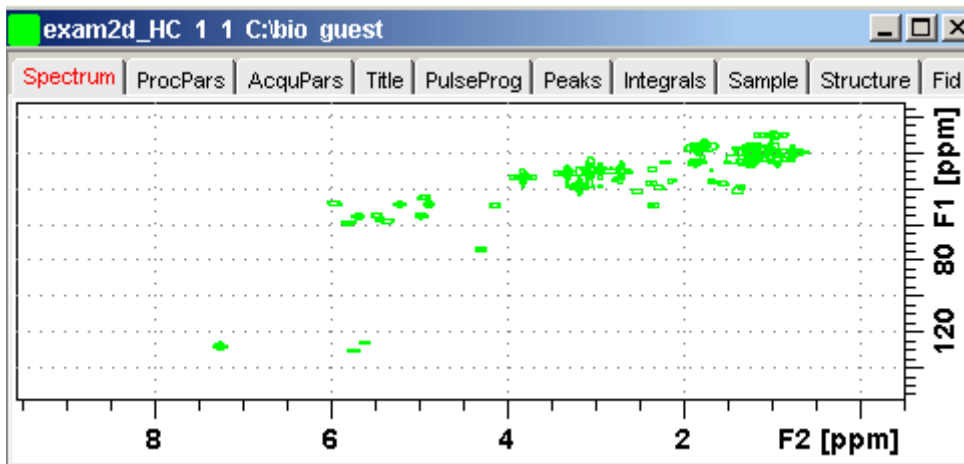



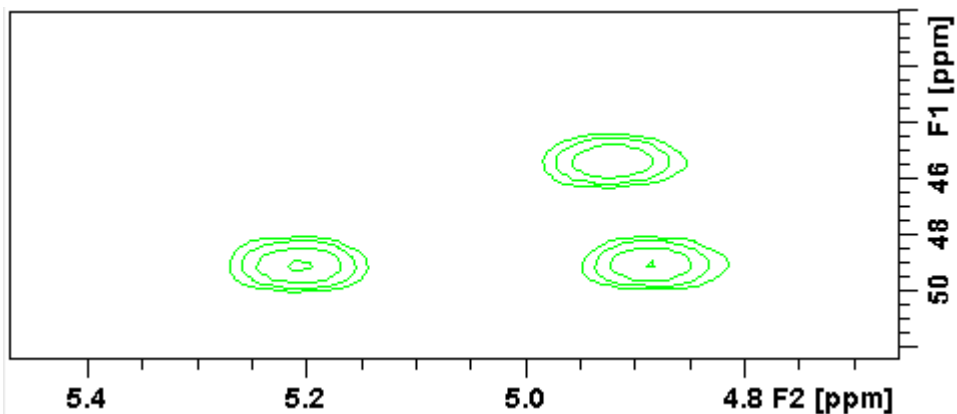
Figure 9.12 Axis aligned grid display

How to Display a 2D Spectrum in Contour Mode

☞ Click the following button in the upper toolbar:

 Switch to contour display mode [.co]

In contour mode, a spectral region looks like this:



How to Set the 2D Contour Levels

☞ Click the following button in the lower toolbar

 Edit contour levels [*edlev*, *.lv*]

This will open the following dialog box shown in Fig. 9.13. Contour levels can be entered manually or they can be calculated.

Manual setup

This allows you to create an arbitrary sequence of levels

1. Enter the level values in the fields 1, 2, ... at the top of the dialog box.
2. Click **Apply** to update the display or **OK** to store the levels, update the display and close the dialog box.

Calculation

This allows you to easily create a geometric or equidistant sequence of levels.

1. Click one of the following items:
 - **Multiply with increment**
to create a geometric sequence of levels.
 - **Add increment**
to create an equidistant sequence of levels.
2. Enter the desired *Base level*, *Level increment* and *Number of levels*.
3. Click **Fill** to display and activate the sequence.
4. Click **Apply** to update the display or **OK** to store the levels, update the display and close the dialog box.

The [Contour level sign](#) allows you to select positive levels, negative levels or both.

exam2d_HC 1 1 C:\bio guest

1	2253560.0	-2253560.0
2	4056408.0	-4056408.0
3	7301534.4	-7301534.4
4	13142761.9	-13142761.9
5	23656971.5	-23656971.5
6	42582548.6	-42582548.6
7	76648587.5	-76648587.5
8	137967457.5	-137967457.5
9	248341423.6	-248341423.6
10	447014562.4	-447014562.4

Required parameters

Calculation method

Multiply with increment
 Add increment

Contour level sign

Positive & Negative
 Positive
 Negative


	Positive	Negative
Base level	2253560.0	-2253560.0
Level increment	1.800	1.800
Number of levels		16

Fill Clear Apply

OK Cancel

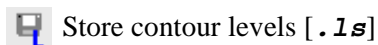
Figure 9.13

How to Store interactively set Contour Levels

To store contour levels that were set interactively, for example by clicking  or

pressing **Alt+PageUp**:

☞ Click the following button in the lower toolbar:

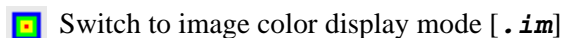


The levels are stored in the file:

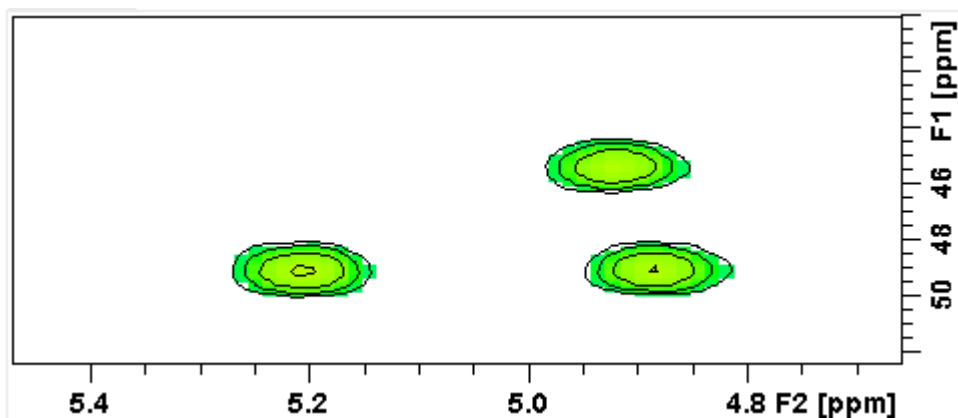
`/<dir>/data/<user>/nmr/<name>/<expno>/pdata/<procno>/clevels`

How to Display a 2D spectrum in Image Mode

☞ Click the following button in the upper toolbar:



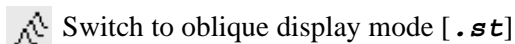
In image mode, a spectral region looks like this:



Note that in image mode, the contours are superimposed, in black, on the image.

How to Display a 2D Spectrum in Oblique Mode

☞ Click the following button in the upper toolbar



In oblique mode, a spectral region looks like in Fig. 9.14.

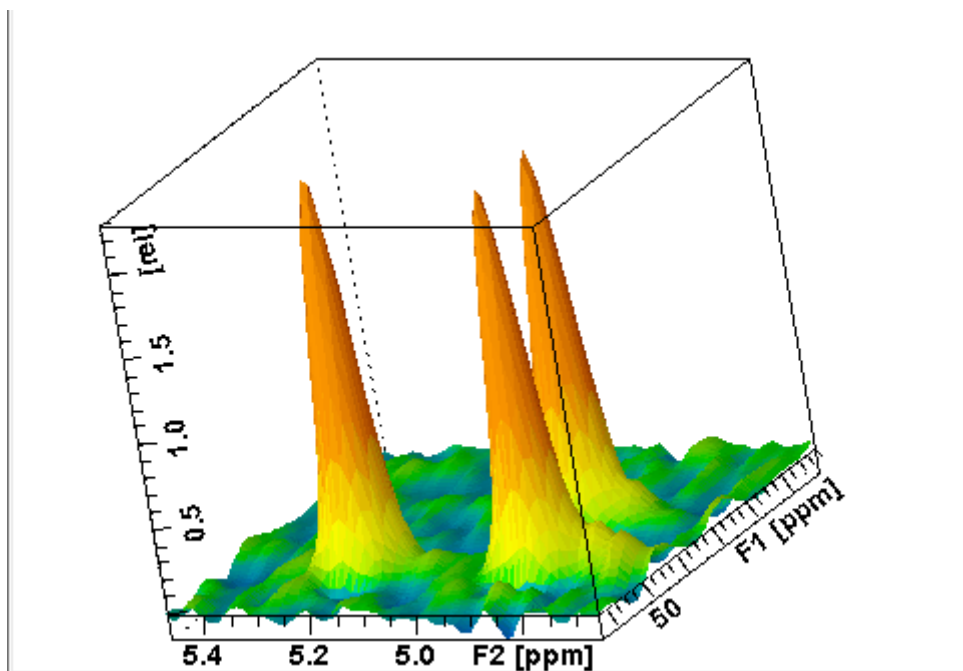


Figure 9.14

In this mode you can manipulate the display in various ways. Just right-click inside the data window and choose one of the options from the appearing popup menu (see Fig. 9.15)

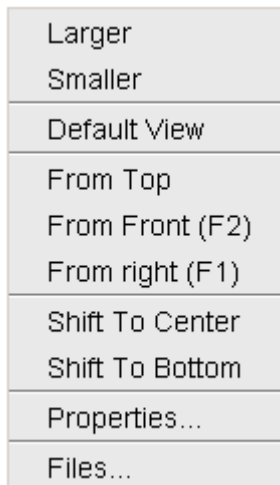




Figure 9.15

How to Rotate a 2D Spectrum in Oblique Mode


Click-hold one of the following button and move the mouse up/down:

 Rotate around x-axis.

 Rotate around y-axis.

How to Switch between Displaying Positive and Negative levels

Click the following multi-state button in the lower toolbar:

 Switch between *positive*, *negative* and *both* contours [**. 1 t**].

Chapter 10

3D Display

10.1 Plane Display Mode

3D data can be displayed as 2D planes or as a 3D cube. By default, the first F3-F1 plane is displayed (see Fig. 10.1) The plane orientation and number is shown. The cube in the lower left corner graphically indicates which plane is displayed. The full 2D display functionality is available (see chapter 9).

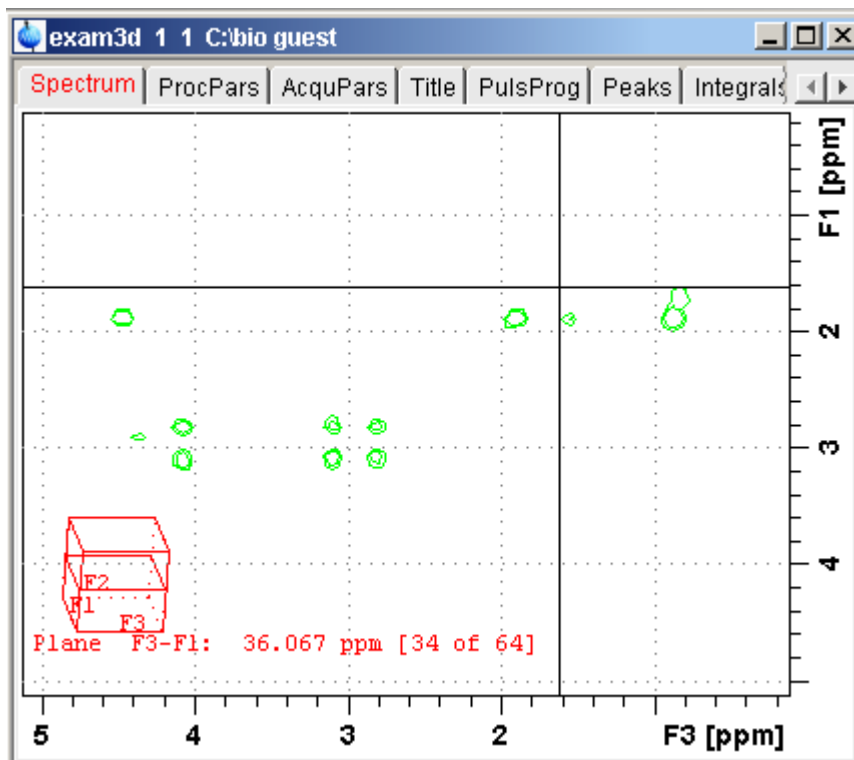





Figure 10.1


How to Switch to 2D Plane Display


If the 3D cube is displayed you can switch to 2D plane display by clicking one of the following buttons:


-  Switch to 2D contour display.
-  Switch to 2D image display.
-  Switch to 2D oblique display.

How to Display various Plane Orientations

Click one of the following buttons:


-  Show F1-F2 planes.


 Show F2-F3 planes.

 Show F3-F1 planes.


How to Display various Plane Positions (numbers)

Click one of the following buttons:

 Show the next plane.

 Show the previous plane.


 Scan planes smoothly.

 Enter the exact plane number.

10.2 Cube Display Mode

How to Display the 3D Cube

Click the following button:

 Show 3d cube (see Fig. 10.2).

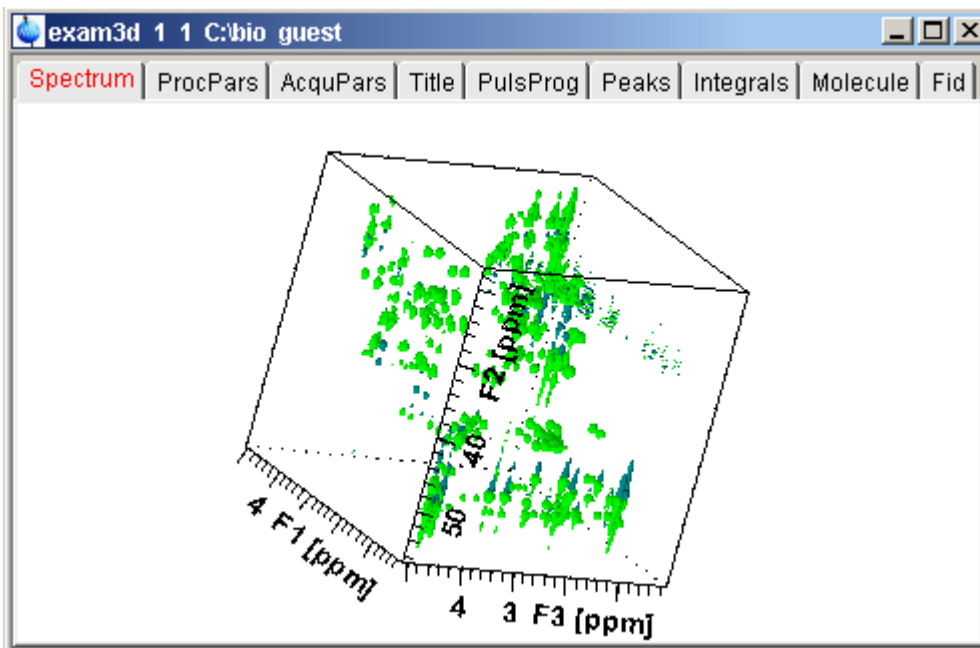





Figure 10.2

How to Rotate the 3D Cube

Click-hold one the following buttons and move the mouse up/down:


-  Rotate cube around x-axis.
-  Rotate cube around y-axis.
-  Rotate cube around z-axis.

How to Scale Up/Down the 3D Cube

1. Right-click inside the data window.
2. Choose *Larger* or *Smaller* from the popup menu (see Fig. 10.3).

How to Reset the Cube Size and Orientation

Click the following button:

-  Reset to default size and orientation.

How to Switch Depth Cueing on/off

1. Right-click inside the data window
2. Choose *Depth Cueing On/off* (see Fig. 10.3)

Depth cueing makes data points which are closer to the viewer appear brighter and those that are further away appear dimmer. This increases the depth effect of the 3D image.

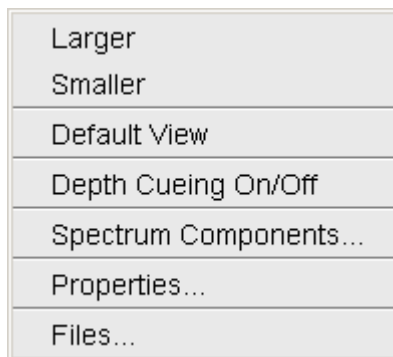


Figure 10.3

How to Display a Cube Front or Side view

Click one the following buttons:

- 12 Show F1-F2 plane.
- 23 Show F2-F3 plane.
- 31 Show F3-F1 plane.

10.3 Using the Tab bar

The 3D data window is a tabbed pane. This means its contents depends on the currently **active tab** in the Tab bar. The individual tabs are basically the same as for 1D and 2D display (see chapter 8.4). When you click the *ProcPars* or *AcquPars* tab, you will find a parameters column for each of the three dimensions F3, F2 and F1. The *Fid* tab allows you to display the 3D raw data as a series of 1D FIDs. The *Peaks* tab will display the 3D peak list with a column for the chemical shift in each of the three dimensions. Similarly, the *Integrals*

tab will display the 3D integral list. Peaks and integrals only appear if they have been calculated (commands **pp** and **int**, respectively). When the integral list shows peaks, with a column for the chemical shift in each of the three dimensions appears. Like in 2D, a stored or exported 3D integral list can be imported (see Fig. 9.4).

Chapter 11

1D Interactive Manipulation

The upper toolbar of the 1D menu offers various buttons for interactive manipulation. If you click such a button, the active data window will switch to the corresponding mode. An interactive manipulation mode is data window specific, i.e. it only applies to the active window.

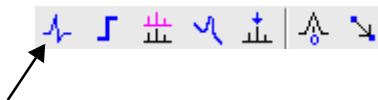
11.1 1D Interactive Phase Correction

Manually acquired spectra can be phased corrected automatically, with commands like `apk` or `apks` or, interactively, in phase correction mode.

11.1.1 1D Interactive Phase Correction Procedure

How to Switch to Phase Correction Mode

Click the indicated button in the upper toolbar:



or enter `.ph` on the command line.

The Tab bar of the active data window will be replaced by a toolbar (see Fig.

11.1:)

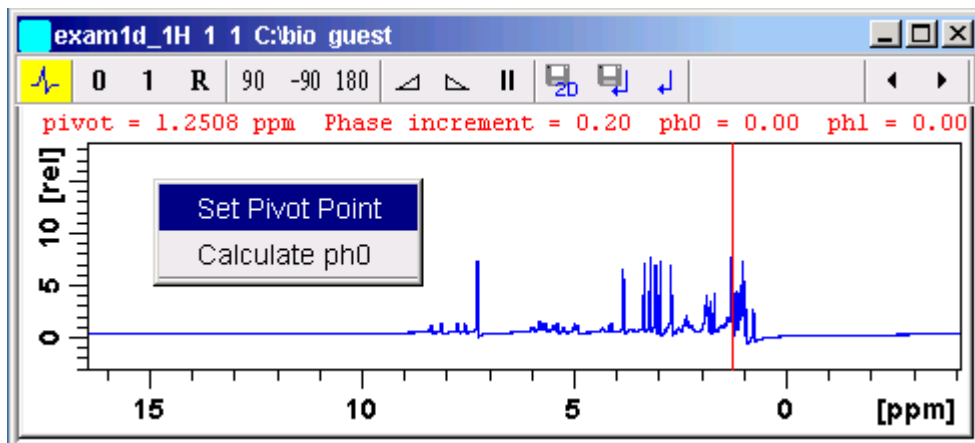





Figure 11.1 Data window in phase correction mode

-  The yellow button indicates that you are in phase correction mode.
-  Some buttons will turn green when they are clicked. As long as a button is green, it is active.

How to Perform a Typical 1D Interactive Phase Correction

For a typical 1D phase correction, take the following steps:

1. Click-hold the button **0** and move the mouse until the reference peak is exactly in absorption mode.
2. Click-hold the button **1** and move the mouse until the entire spectrum is exactly in absorption mode.
3. Click the button  to save and execute the phase correction and return.

11.1.2 1D Interactive Phase Correction Details

How to Set the Phase Pivot Point

By default, the phase pivot point is set to the biggest magnitude intensity of the displayed region of the spectrum. To change the pivot point:


1. Right-click on the desired pivot point position
2. Choose *Set pivot point* from the popup menu (see Fig. 11.1)

How to Perform Default Zero Order Phase Correction


1. Right-click in the data window
2. Choose *Calculate $ph0$* in the popup menu (see Fig. 11.1)

The spectrum will automatically be corrected according to the calculated value.

How to Perform Interactive Zero Order Phase Correction




1. Click-hold the following button (button turns green):
 Zero order phase correction (parameter PHC0).
2. Move the mouse until the reference peak is exactly in absorption mode.
3. Release the mouse (button turns grey).

How to Perform Interactive First Order Phase Correction

1. Click-hold the following button (button turns green):
 First order phase correction (parameter PHC1).
2. Move the mouse until the entire spectrum is exactly in absorption mode.
3. Release the mouse (button turns grey).

How to Perform 90, -90 or 180° Zero Order Phase Correction

☞ Click one of the following buttons:

-  Perform 90 zero order phase correction [*.ph90*].
-  Perform -90° zero order phase correction [*.phm90*].
-  Perform 180° zero order phase correction [*.ph180*].




How to Reset the Phase to the Original Values

☞ Click the following button:

-  Reset zero and first order phase values [*.phr*].

How to Change the Mouse Sensitivity


☞ Click one of the following buttons:

-  Increase (double) the mouse sensitivity [`.inc`].
-  Decrease (halve) the mouse sensitivity [`.dec`].
-  Reset the mouse sensitivity.

How to Return from Phase Correction Mode with/without Save

To return while saving the phase correction to the current dataset:

 Click the following button:


 Save, execute and return [`.sret`].

This will perform the following tasks:

- Execute phase correction (command `pk`).
- Save the current phase correction values.
- Leave the phase correction mode.


To return without save:

 Click the following button:

 Return, discarding any changes [`.ret`].

To return while saving the phase correction to the source 2D dataset:

 Click the following button:

 Save to 2D [`.s2d`].

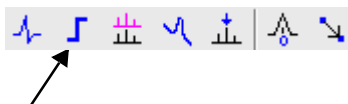
This is only applicable on rows or columns extracted from 2D data. The phase values will be saved to the 2D dataset from which the current 1D dataset was extracted.

11.2 1D Interactive Integration

Integration of 1D data can be done automatically, with the commands `abs` and `li` or, interactively, as described in this paragraph.

How to Switch to Integration Mode

☞ Click the indicated button in the upper toolbar:



or enter `.int` on the command line.

The Tab bar of the active data window will be replaced by a toolbar (see Fig. 11.2).

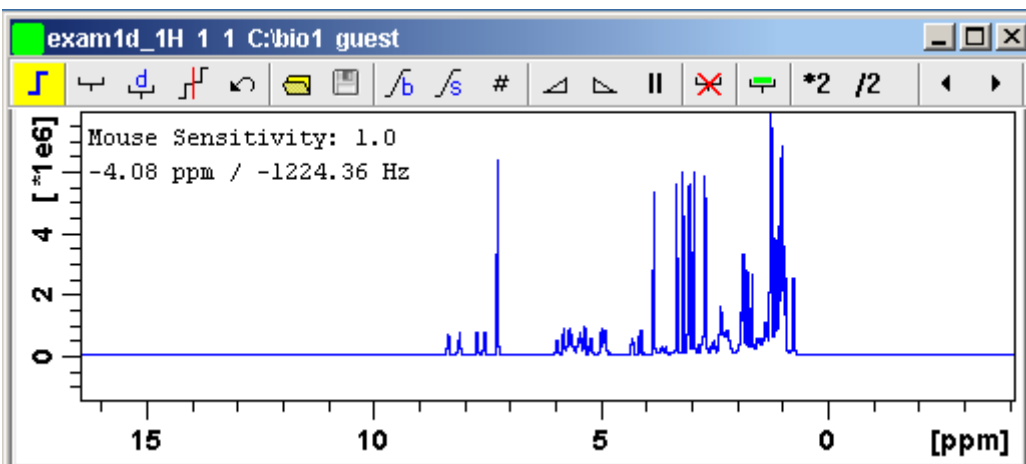




Figure 11.2 Data window in integration mode


-  The yellow button indicates that the data window is in integration mode.
-  Some buttons will turn green when they are clicked. As long as a button is green, it is active.

If integral regions have already been determined, for example with `abs` or with a previous interactive integration, these regions are displayed in the data window, along with the integral values. You can remove them, change them or add to them, as described below.

How to Define Integral Regions

To define integral regions interactively:


1. Click the following button (button turns green):

 Define integral region interactively.

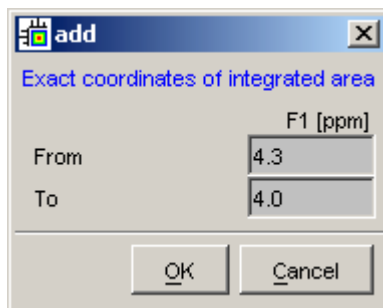
2. Put the red cursor line at one edge of a peak or multiplet.
3. Left-click-hold and drag the cursor line to the other edge of the peak or multiplet.
4. Do step 2 and 3 for all regions to be defined.
5. Click the green button to leave the "define region" mode (button turns grey).

To define integral regions via a dialog box:

1. Click the following button:

 Define region via dialog.

2. In the appearing dialog box:




Enter the exact values for the region limits.

3. Click **OK** to define the selected region.

How to Select/Deselect Integral Regions

To select/deselect all displayed integral regions:

1. Click the following button:

 Select/Deselect all regions.

To select a single integral region:

1. Right-click in the integral region.
2. Choose *Select/Deselect* from the popup menu.

Selected integral regions are indicated by a color filled integral label. In the Fig. 11.3, the two left most regions have already been selected, the right most region is currently being selected.

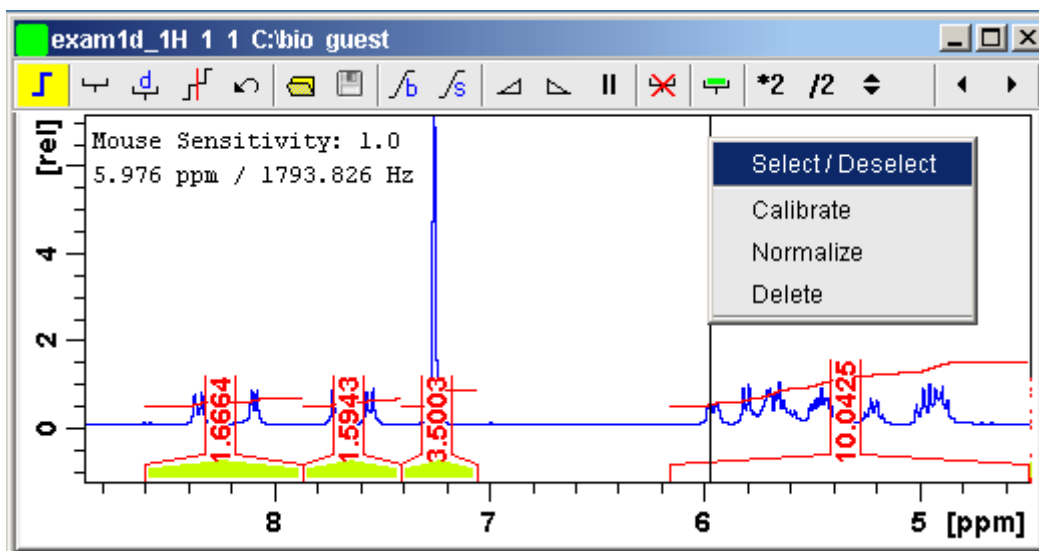



Figure 11.3

How to Read Integral Regions from Disk

You can read integral regions from disk which have been stored by automatic integration (command *abs*) or by a previous interactive integration.

To read integrals:

1. Click the following button:

 Read integral regions.

The following popup menu will appear:

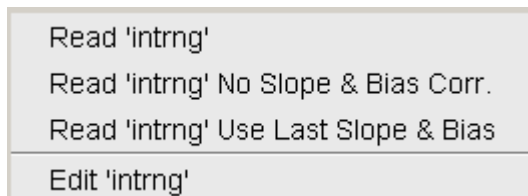



Figure 11.4

2. From the popup menu, choose one of the following entries:
 - ***Read 'intrng'***
to read the last saved integral regions and apply the saved slope and bias correction values.
 - ***Read 'intrng' no slope & bias corr.***
to read the last saved integral regions but do not apply the saved slope and bias correction values.
 - ***Read 'intrng' use last slope & bias***
to read the last saved integral regions applying the last slope and bias correction values.
 - ***Edit 'intrng'***
to edit the file (`intrng`) that contains the integral regions and slope and bias correction values. Changes in this file are automatically shown on the screen.

How to Perform Interactive Bias and Slope Correction

To perform interactive bias correction:

1. Select the integral(s) that you want to correct (right-click in the region).
If no integral is selected, bias correction will work on all integrals.
2. Click-hold the following button (it turns green) and move the mouse,



 Integral bias correction.
until the integral bias is correct.
3. Release the mouse (button turns grey).

To perform interactive slope correction:

1. Select the integral(s) that you want to correct (right-click in the region).

If no integral is selected, slope correction will work on all integrals.

2. Click-hold the following button (it turns green) and move the mouse,

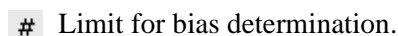


until the integral slope is correct.

3. Release the mouse (button turns grey).

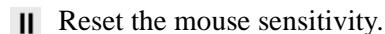
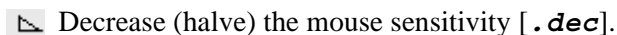
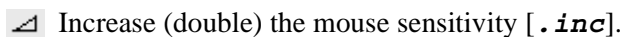
How to Set the Limit for Bias Determination

☞ Click the following button:



How to Change the Mouse Sensitivity

☞ Click one of the following buttons:



How to Calibrate/Normalize Integrals

Calibrating integrals means setting the value of a reference integral and adjusting all other integrals accordingly. To do that:

1. Right-click in the reference integral region.
2. Choose **Calibrate** from the popup menu (see Fig. 11.5).
3. Enter the desired value for the reference integral and click **OK**

Normalizing integrals means setting the sum of all integrals and adjusting individual integral values accordingly. To do that:

1. Right-click in the reference integral region.
2. Choose **Normalize** from the popup menu (see Fig. 11.5).

3. Enter the desired sum of all integrals and click **OK**

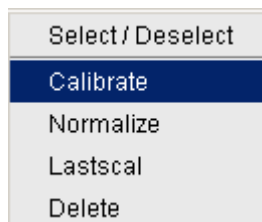


Figure 11.5

Calibrating and normalizing only effects the current dataset. To scale integrals with respect to a reference dataset, choose *lastscal* from the right/click popup menu (see below).

How to Scale Integrals with respect to Different Spectra

Integrals can be scaled with respect to the last spectrum that was integrated interactively. To do that:

1. Right-click in the reference integral region.
2. Choose *Lastscal* from the popup menu (see Fig. 11.5).

As such, you can compare integrals of different spectra. Note that this only make sense for spectra which have been acquired under the same experimental conditions. The scaling factor is stored in the file:

```
<tshome>/prog/curdir/<user>/intscale
```

How to Delete Integral Regions from the Display

To delete the selected integral regions from the display:

☞ Click the following button:


 Delete selected integral regions from the display.

To delete a single integral region from the display:

1. Right-click in the integral region.
2. Choose **Delete** from the popup menu (see Fig. 11.5)

To delete all integral regions from the display:

☞ Click the following buttons:


 Select all integral regions.


 Delete selected integral regions from the display.


Note that regions are only deleted from the screen. Regions which are saved on disk (in the `int.rng` file) are not affected.


How to Scale Selected Integrals

Integral scaling only manipulates selected integrals. However, if no integrals are selected, it works on all integrals.

 Click one of the following buttons:


 Scale up selected integrals by a factor of 2.

 Scale down selected integrals by a factor of 2.

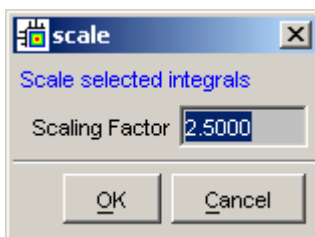
 Scale selected integrals up/down smoothly.

To scale up/down integrals by a factor entered via a dialog:

1. Click the following button:

 Scale integrals via a dialog.


2. Enter a scaling factor, e.g. 2.5. in the appearing dialog.



3. Click **OK** to apply this factor.

To scale all integrals to the same height:

 Click the following button:




 Scale/unscale all integrals to the same height.

The individual scaling factor for each region is displayed above the integral. Clicking this button again rescales all integrals to their original height.

How to Move the Integral Trails Up/Down


To move the integrals (selected and unselected) up or down:

☞ Click one of the following buttons:

-  The left edge of the lowest integral is put just above the baseline.
-  The right edge of the highest integral is put at 3/4 of the window height.
-  Shift all integral trails up/down smoothly.

How to Cut Integral Regions

1. Click the following button (button turns green):

 Cut integral region.

2. Move the red cursor line into an integral region to the position where it must be cut and click the left mouse button.
3. Do step 2 for each integral region that must be cut.
4. Click the (green) button to leave the *cut integral* mode (button turns grey).

How to Save Integral Regions

1. Click the following button:

 Save integral regions.

The following popup menu will appear:




Figure 11.6

2. Choose one of the following entries:
 - **Save regions to 'intrng'**
Save the currently displayed integral regions including the slope and bias correction values.

- **Save Regions to 'reg'**
Save the integral regions to the file `reg`.
- **Save & show list**
Save the currently displayed integral regions including the slope and bias correction values and show the integrals on the screen.

How to Undo the Last Region Operation


☞ Click the following button:

 Undo the last region operation.

How to Return from the Integration Mode with/without Save

To return and save the integrals to the current dataset:

☞ Click the following button:


 Save integrals and return [`.sret`].

As such, you will:

- save the integral regions and corresponding slope and bias corrections to the file `intrng`.
- save the integral regions, slope and bias corrections and integral values to the file `integrals.txt`. This file is displayed when you click the **Integrals Tab**.
- leave the integration mode.

To return without save:

☞ Click the following button:

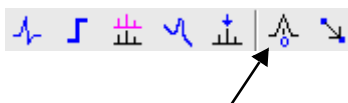
 Return, discarding any changes [`.ret`].

11.3 1D Interactive Calibration

A 1D spectrum can be calibrated (referenced), automatically, with the command **sref** or, interactively, as described below.

How to Switch to Calibration Mode

☞ Click the indicated button in the upper toolbar



or enter `.cal` on the command line. The Tab bar of the active data window will be replaced by a toolbar.

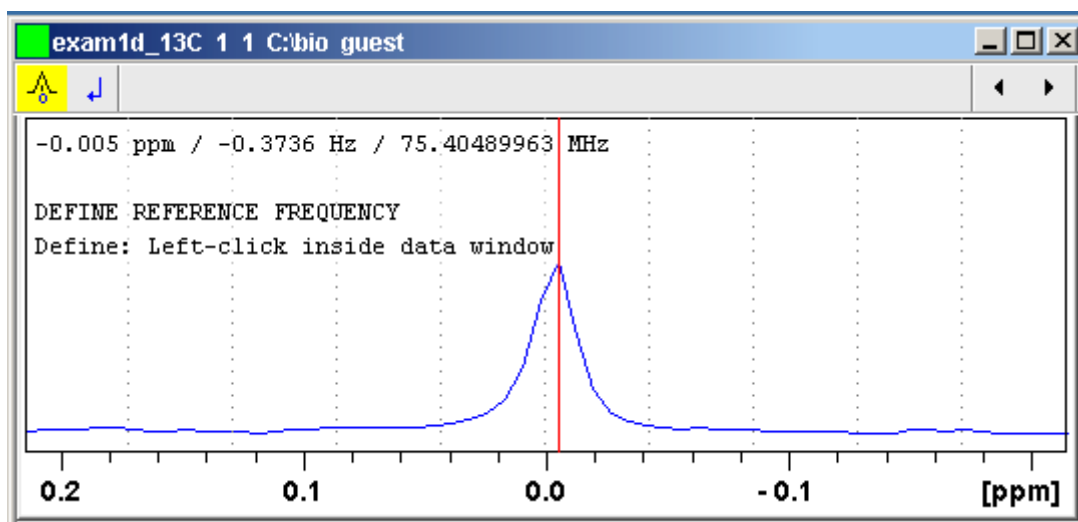


Figure 11.7 Data window in calibration mode



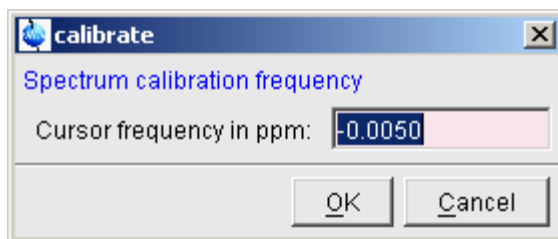
The yellow button indicates that the data window is in calibration mode.

How to Calibrate a Spectrum Interactively

In calibration mode:

1. Position the red cursor line at the reference peak.
2. Left-click at that position.

The following dialog box will appear:



Note that the units (Hz or ppm) correspond to the axis units of the display.

3. Enter the frequency you want to assign to the reference peak.
4. Click **OK**

The spectrum will be calibrated and re-displayed. TOPSPIN will automatically leave calibration mode.


11.4 1D Multiple Display

The multiple display mode allows you to display multiple superimposed spectra. The spectra will be ppm aligned or Hz aligned, according to the selected axis unit. Each spectrum can be individually shifted and scaled allowing exact alignment of corresponding peaks in different spectra. The number of superimposed spectra is unlimited.

Although multiple display is normally used for spectra with matching nuclei, it allows you to superimposed spectra with non-matching nuclei. You will get a warning that the nuclei do not match. Just just click **OK** to continue.

How Switch to Multiple Display Mode and Read Multiple Spectra

One way to superimpose data in multiple display is to read one dataset, switch to multiple display mode and add other datasets:

1. Read a 1D dataset.
2. Click the  button in the upper toolbar or type `.md` on the command line.

The data window will switch to multiple display mode.

3. Add a dataset as follows:

☞ Left-click-hold the dataset in the browser and drag it into the data win-

dow.

or

- ☞ Right-click the dataset in the browser and choose **Display** from the popup menu.

or

- ☞ Enter **re** and specify the additional dataset in the appearing dialog box.

Another way to superimpose data in multiple display is to read multiple datasets simultaneously:

1. In the browser or portfolio:

- ☞ Hold down the **Ctrl** key and click multiple datasets to select them.

or

- ☞ Hold down the **Shift** key and click two datasets to select these two and all datasets in between.

2. Right-click any of the selected data:

- ☞ Choose **Display** from the popup menu.

This will show the data in the active data window if that is in multiple display mode or, otherwise, show the data in a new window.

or

- ☞ Choose **Display in new window** from the popup menu.

This will show the data in a new window.

In multiple display mode, the Tab bar of the active data window is replaced by a toolbar. Fig. 11.8 shows three comparable 1D spectra and the sum of all three.

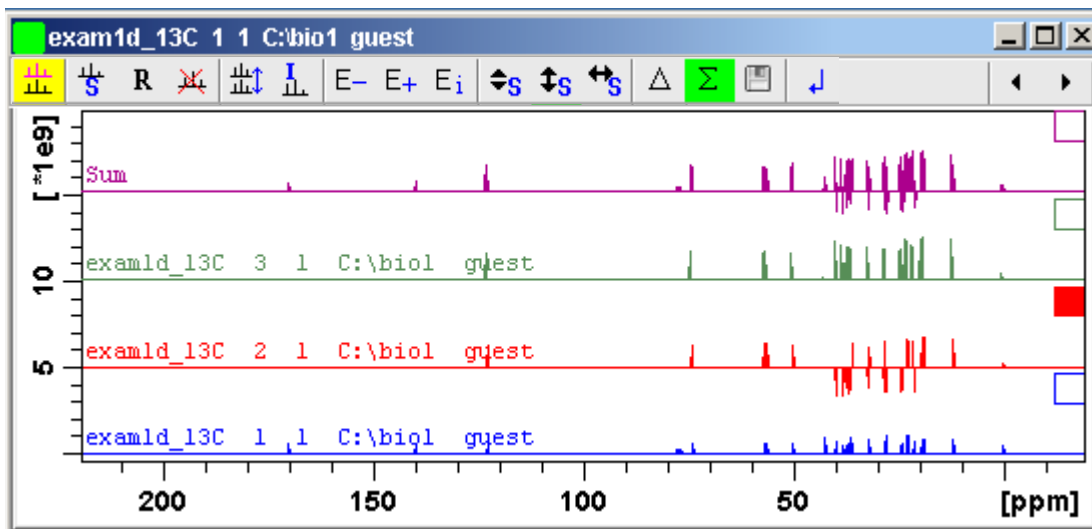




Figure 11.8 Data window in multiple display mode

-  The yellow button indicates that the data window is in multiple display mode.
-  Some buttons will turn green when they are clicked. As long as a button is green, it is active.

Furthermore, the browser/portfolio is split in two parts as shown in Fig. 11.9.

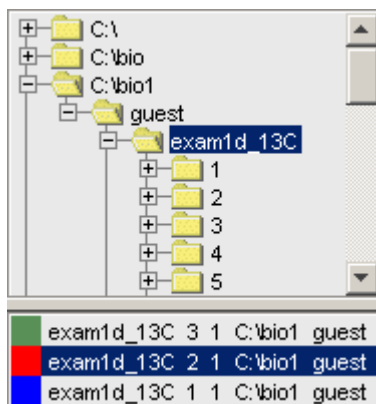


Figure 11.9

The additional lower part shows:

- which datasets are displayed in the active data window.
- which datasets are selected (these are highlighted).

How to Select/Deselect Datasets

To select a dataset:

- ☞ Click in the corresponding area in the data window.
- or* Click the small square at the upper right of the spectrum.
- or* Click the corresponding entry in the lower part of the browser/portfolio.

In the lower part of the browser/portfolio, you can:

- ☞ Click one dataset to select it.
- or* Hold down the **Ctrl** key and click multiple datasets to select them.
- or* Hold down the **Shift** key and click two datasets to select these two and all datasets in between.

When you select a dataset, the corresponding small square is filled (see Fig. 11.8) and its entry in the lower part of the browser is highlighted (see Fig. 11.9).

Note that:


- no spectrum selected = all spectra selected
- scale/shift buttons of the data window toolbar only work on selected spectra

To deselect a dataset:

- ☞ Select a different dataset.


To deselect all datasets:

- ☞ Click the following button:

 Deselect all datasets.


How to Remove a Dataset from Multiple Display


1. Select the dataset(s) you want to remove.
2. Click the following button:


 Remove selected data from the screen.

Note that the data on disk are not affected. Furthermore, the first spectrum cannot be removed from the screen.

How to Display the Sum or Difference Spectra


 Click one of the following button (button turns green):

 Show the difference between the first and the sum of the other datasets.

 Show the sum of all datasets in the multiple display window.

How to Save the Sum or Difference Spectra


1. Click the following button:

 Save the displayed sum or difference spectrum.

2. In the appearing dialog box, specify the destination *procno*.


How to Display the Next/Previous Name/Expno

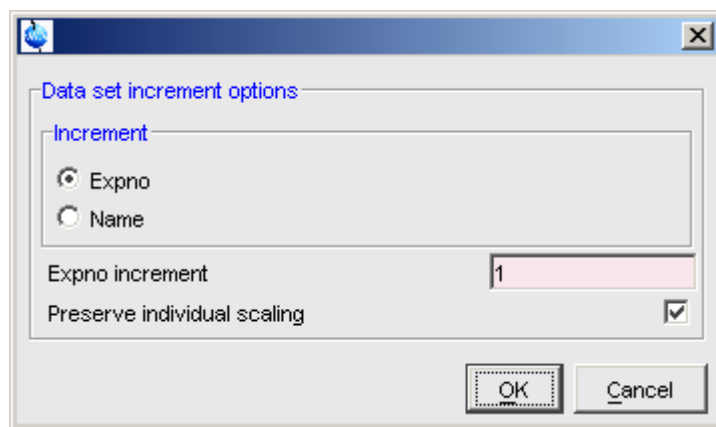
To compare a series of spectra you can interactively increment or decrement the dataset name or expno. A dataset name is incremented according to the ICON-NMR naming convention of increasing extensions, e.g. name.001, name.002 etc.

 Click one of the following button (button turns green):

 Show the previous name/expno.

 Show the previous name/expno.

 Set the increment options. Clicking this button will open the following dialog:



Here you can choose to increment the expno or name, set the expno increment and switch individual scaling on/off.

How to Toggle between Superimposed and Stacked Display



☞ Click the following button:




Toggle between superimposed and stacked display.

How to Shift and Scale Individual Spectra

To compare the intensity and chemical shift of corresponding peaks, you can shift and scale individual spectra. To do this:

1. Display the spectra in multiple display mode as described above.
2. Expand the spectra to display the desired region or peak.
3. Select one of the spectra (e.g. by clicking it in the lower part of the browser).
4. Click-hold the  button and move the mouse to align the intensities.
5. Click-hold the  button and move the mouse to align the peak positions.

The alignment can be facilitated by showing the difference spectrum ( button) and minimize that. Clicking the **R** button resets individual scaling and shifting.

The performed scaling and shifting are displayed in the data window (see Fig. 11.10 and 11.11).

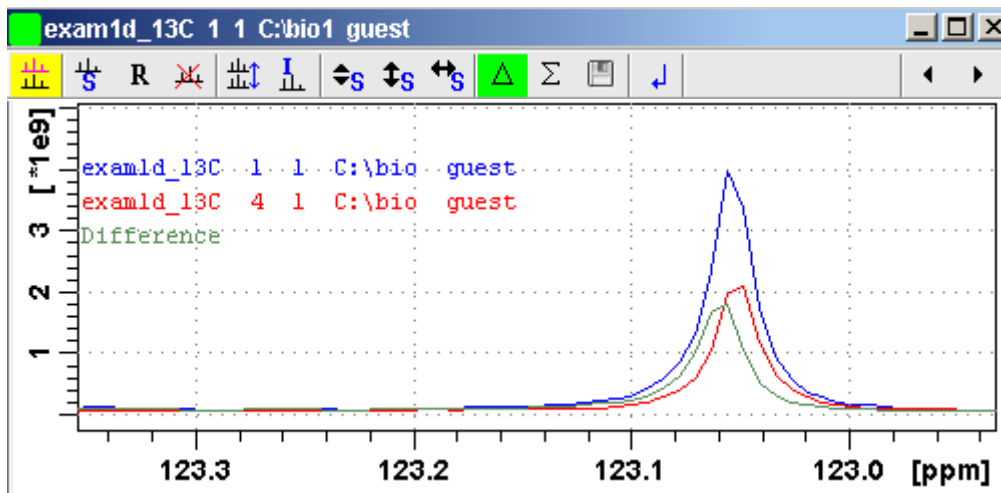


Figure 11.10

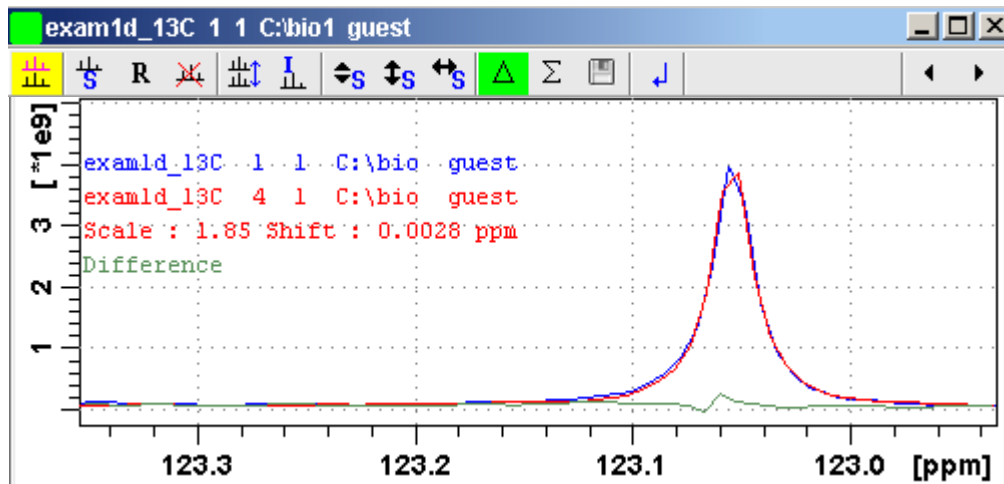



Figure 11.11


How to Switch on/off the Display of Datapaths and Scaling Factors

☞ Click the following button:

 Switch on/off display of datapaths and scaling factors.

How to Return from Multiple Display mode

☞ Click the following button:

 Return from multiple display mode [`.ret`].

How to Set the Colors of the 1st, 2nd, .. Dataset

The colors of the different datasets in the multiple display mode can be set in the *User preferences* dialog box. To set, for example, the color of the second spectrum:

☞ Click *Options* → *Preferences* and click the *Change* button for the item *Color of 2nd 1D spectrum*.

11.5 1D Interactive Baseline Correction

Baseline correction can be performed with commands like *abs* or *absd* or, interactively, as described below.

How to Switch to Baseline Correction Mode

☞ Click the indicated button in the upper toolbar:



or enter `.bas1` on the command line.

The Tab bar of the active data window will be replaced by a toolbar (see Fig. 11.12).

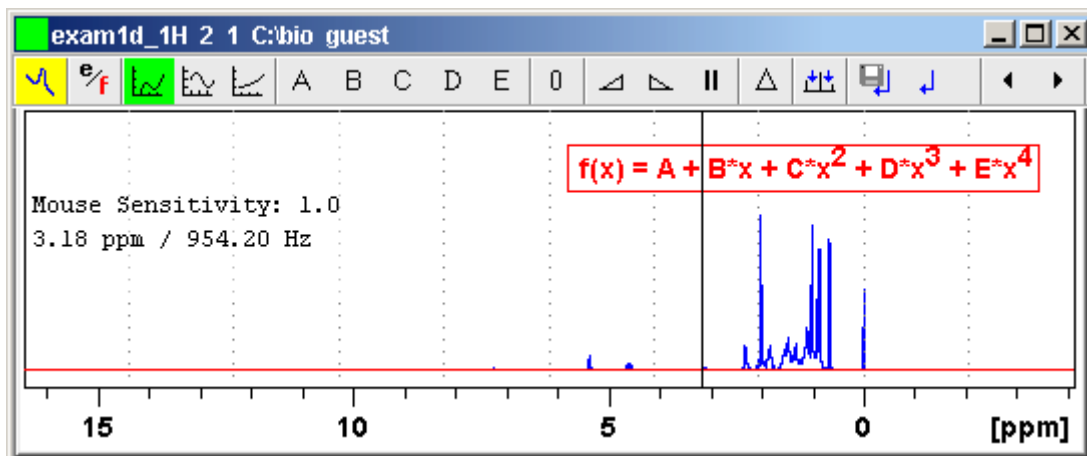





Figure 11.12 Data window in baseline correction mode

-  The yellow button indicates that the data window is in baseline correction mode.
-  Some buttons will turn green when they are clicked. As long as a button is green, it is active.

How to Perform Polynomial Baseline Correction

1. Click the following button (button turns green):

 Perform polynomial baseline correction.


In the data window, a red horizontal line will appear as well as the equation that describes the polynomial function:

$$f(x) = A + B*x + C*x^2 + D*x^3 + E*x^4$$

2. Click-hold button **A** and move the mouse until the red line coincides with the first point of the spectrum.
3. Repeat step 2 with the buttons **B**, **C**, **D** and **E** until the red line coincides with the entire baseline of the spectrum.

How to Perform Sine Baseline Correction

1. Click the following button (button turns green):

 Perform sine baseline correction.


A red horizontal line will appear as well as the equation describing the sine function:

$$f(x) = A + B \cdot \sin(C \cdot x + D)$$

2. Click-hold button **A** and move the mouse until the red line coincides with the first point of the spectrum.
3. Repeat step 2 with the buttons **B**, **C** and **D** until the red line coincides with the entire baseline of the spectrum.

How to Perform Exponential Baseline Correction

1. Click the following button (button turns green):

 Perform exponential baseline correction.

A red horizontal line will appear as well as the equation describing the exponential function:

$$f(x) = A + B \cdot \exp(C \cdot x)$$

2. Click-hold button **A** and move the mouse until the red line coincides with the first point of the spectrum.
3. Repeat step 2 with the buttons **B** and **C** until the red line coincides with the entire baseline of the spectrum.

How to Preview the Baseline Corrected Spectrum



Before actually performing the baseline correction, you can preview the result by displaying the difference between the uncorrected spectrum and the red correction line.

To do that:

1. Click the following button (button turns green):

 Preview corrected spectrum (show difference).


The corrected spectrum will be displayed in red.

2. If the baseline is correct, click the  button to save the correction. If further correction is needed, click the  button to show the original spectrum and the red correction line.

How to Reset the Baseline Correction Line

1. Click the following button:

 Reset the red correction line to zero.

If the difference spectrum is displayed (the  button is active), clicking the reset button will restore the original spectrum.

How to Change the Mouse Sensitivity

☞ Click one of the following buttons:

 Increase (double) the Mouse Sensitivity [*.inc*].


 Decrease (halve) the Mouse Sensitivity [*.dec*].

 Reset the Mouse Sensitivity.

How to Save the Baseline Correction and/or Return

To return while saving the baseline correction:

☞ Click the following button:


 Save baseline correction and Return [*.sret*]

This will perform the following tasks:

- Execute the baseline correction [*bcm*].
- Save the baseline correction values **A**, **B**, **C**, **D** and **E**.
- Leave the baseline correction mode.


To return while discarding any changes:

Click the following button:

 Return, Discarding any changes [*.ret*].

How to Perform Cubic Spline Baseline correction

☞ Click the following button:

 Define points for cubic spline baseline correction.

The toolbar of the data window will change as shown in Fig. 11.13.

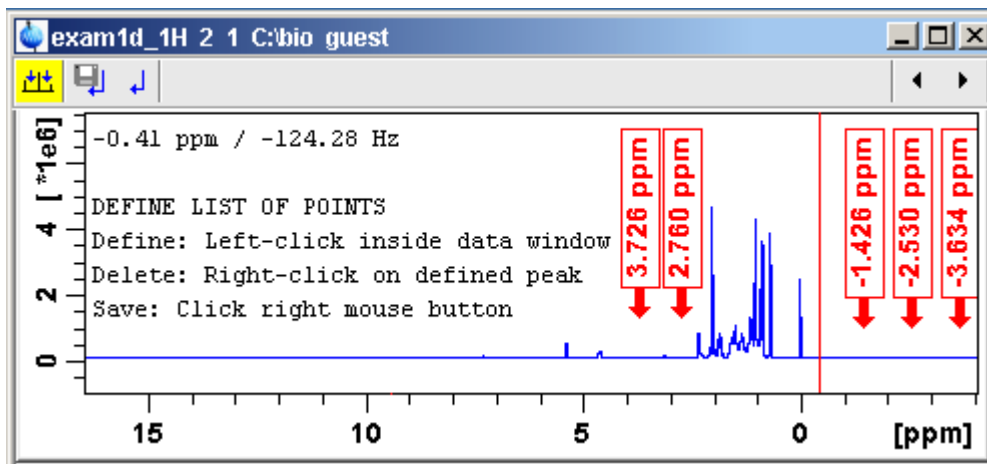


Figure 11.13 Data window in spline baseline correction mode

The cursor line in the data window turns red. If a list of baseline points already exists, you are prompted to overwrite or append to these points. If you choose *Append*, the labels of the existing points are displayed on the screen. If you choose *Overwrite*, no labels are displayed. Nevertheless, the existing points are only overwritten when you define and save new points.

To define new baseline points:

1. Move the cursor line to a baseline point and left-click at that position.
2. Do this for at least five baseline points.

Fig. 11.13 shows a spectrum with five defined baseline points. Note that here the points have been chosen at the right part of the spectrum for display reasons only.

How to Delete Spline Baseline Points from the screen

To delete one baseline point:

1. Right-click the baseline point position in the data window.
2. Choose *Delete Current* from the popup menu (see Fig. 11.14).

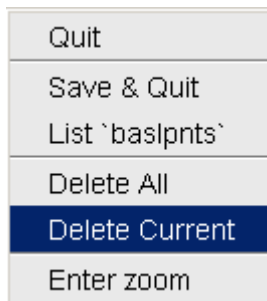


Figure 11.14


To delete all baseline points:

1. Right-click any position in the data window.
2. Choose *Delete All* from the popup menu (see Fig. 11.14).

How to Return from Cubic Spline Baseline mode with/without Save


To return while saving the baseline points:

☞ Click the following button:

 Save baseline points and Return [*.retsab*].

To return while discarding any changes:

☞ Click the following button:

 Return, Discarding any changes [*.ret*].

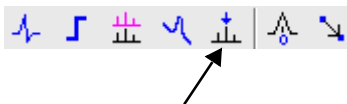
Alternatively, you can right-click in the data window and choose *Save & Quit* or *Quit*, respectively.

11.6 1D Interactive Peak Picking

Peak picking can be performed, automatically, with the commands *pps* or, interactively, in the peak picking mode.

How to Switch to Peak Picking Mode

☞ Click the indicated button in the upper toolbar:



or enter `.pp` on the command line.

The Tab bar of the active data window will be replaced by a toolbar (see Fig. 11.15).

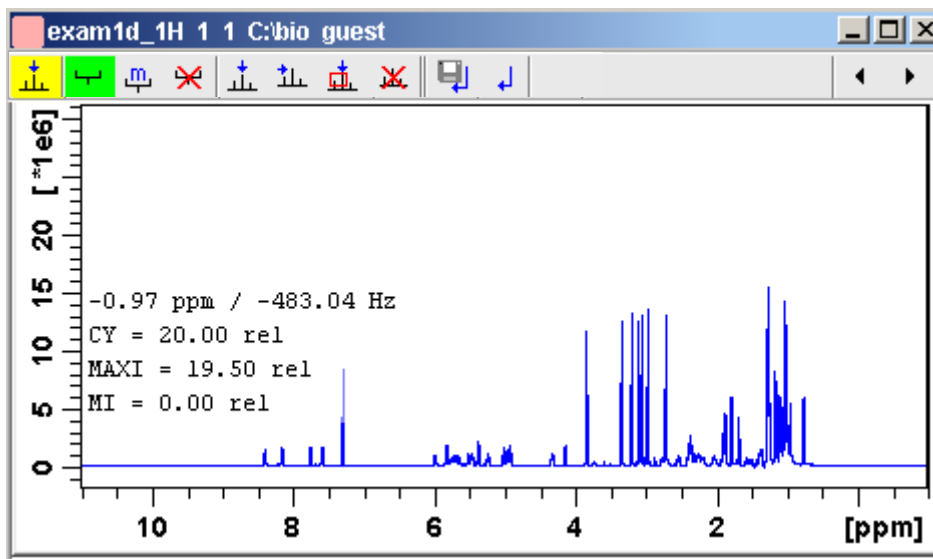





Figure 11.15 Data window in peak picking mode

-  The yellow button indicates that you are in peak picking mode.
-  Some buttons will turn green when they are clicked. As long as a button is green, it is active.

Note that the  button is automatically activated, i.e. you are in *Define peak picking range* mode


How to Define New Peak Picking Ranges

1. Put the cursor at the upper-left corner of a peak picking range.

2. Left-click-hold and drag the mouse to the lower-right corner of the range.
The peak picking range will be marked green. The minimum and maximum intensity are set and the peaks in the range are picked and displayed.
3. Repeat step 1 and 2 for each peak picking range to be defined.
4. Click the green button to leave the "Define peak picking range" mode.

Note that the parameters MI and MAXI are set to the lowest minimum and the highest maximum intensity, respectively, of all ranges.

How to Change Peak Picking Ranges

1. Click the following button (button turns green):
 Change peak picking ranges.
2. Put the cursor on one of the edges of the peak picking range.
The cursor turns into a double-headed arrow.
3. Left-click-hold and drag the peak range edge to its new position.
4. Optionally: repeat step 2 and 3 for the other edge and for other peak ranges.
5. Click the green button to leave the "Change peak picking range" mode.



How to Pick Peaks in Peak Picking Ranges only

Peaks in a peak range are automatically picked when the range is defined. If peaks have been deleted from a range, they can be picked again as follows:


1. Right-click in the data field.
2. Choose *Pick Peaks On Ranges* from the popup menu.

Alternatively, you can enter *pp1* on the command line. This command can be entered in Interactive peak picking mode or in the normal display mode.

How to Delete all Peak Picking Ranges

 Click the  button in the data window toolbar.

or

 Right-click in the data field and click *Delete All Ranges* in the popup menu.

How to Define Peaks Manually

1. Click the following button (button turns green):



Define peaks manually.

A red vertical line will appear in the data window.

2. Put the red cursor line at the desired peak and click the left mouse button. The peak label will appear at the top of the data window.
3. Repeat step 2 for each peak to be defined.
4. Click the green button to leave the "Define peaks" mode.

How to Pick Peaks Semi-Automatically

1. Click the following button (button turns green):



Define peaks semi-automatically.

2. Move the cursor into the data window.
3. Put the cursor line near the desired peak.
4. Left-click to pick forward

or

Right-click to pick backward (see Fig. 11.16).

A red cursor line will appear at the nearest peak whose intensity is between MI and MAXI.

5. Right-click to add the selected peak to the peak list (see Fig. 11.16).

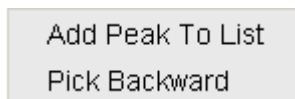


Figure 11.16

The peak label will appear at the top of the data window.

6. Click the green button to leave the "define peaks semi-automatically" mode.

How to Delete Peaks from the Peak List

To delete a specific peak:

1. Right-click on a defined peak.

2. Choose *Delete peak under cursor* from the popup menu (see Fig. 11.17).

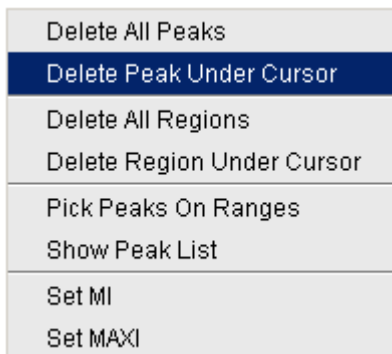



Figure 11.17

To delete all peaks:

☞ Click the  button in the data window toolbar.

or

☞ Right-click in the data field and click *Delete All Peaks* in the popup menu.

How to Return from Peak Picking Mode with/without Save

To return while saving the peak list and peak ranges:

☞ Click the following button:


 Save the Peak Region and Peak List and Return [*.sret*].

This will:

- Save the peak list to the file `peak.txt` and the peak ranges to the file `peakrng`.
- Leave the peak picking mode.

To return while discarding any changes:

☞ Click the following button:

 Return, discarding any changes [*.ret*].

Chapter 12

2D Interactive Manipulation

The upper toolbar of the 2D menu offers various buttons for interactive manipulation. If you click such a button, the active data window will switch to the corresponding mode. An interactive manipulation mode is data window specific, i.e. it only applies to the active window.

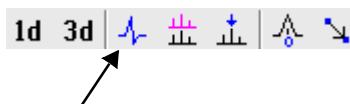
12.1 2D Interactive Phase Correction

2D spectra can be phase corrected interactively in both the F2 and F1 direction by selecting certain rows and/or columns and phase correct them.

12.1.1 2D Interactive Phase Correction Procedure

How to Switch to 2D Interactive Phase Correction

Click the corresponding button in the upper toolbar as indicated below:



or enter `.ph` on the command line.

The Tab bar of the active data window will be replaced by a toolbar. Fig. 12.1

shows an example of an unphased 2D inverse spectrum.

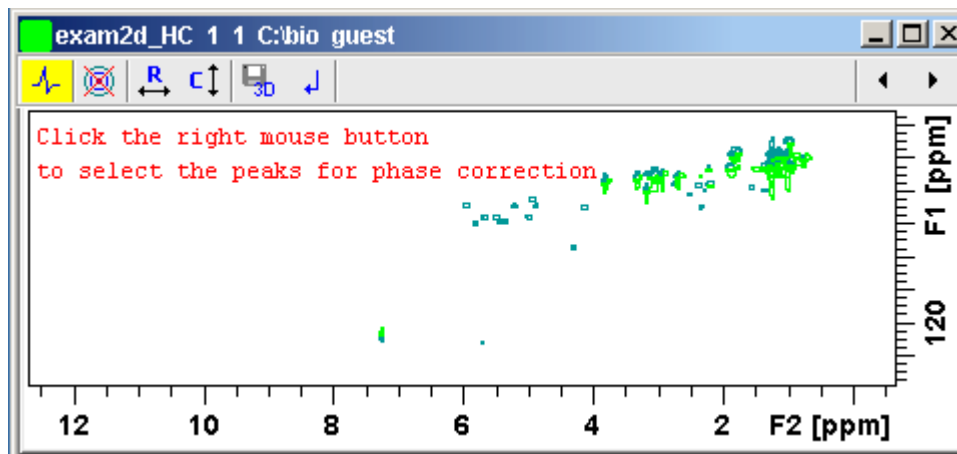








Figure 12.1 Data window in phase correction mode

-  The yellow button indicates that you are in phase correction mode.
-  Toggle the contour display on/off.
-  Switch to *phase row* mode to display rows of selected peaks.
-  Switch to *phase columns* mode to display columns of selected peaks.
-  Save the phase values to the 3D data from which this 2D was extracted.
-  Return.

How to Perform a Typical 2D Interactive Phase Correction

In this example we will perform F1 phase correction (columns) only. Take the following steps:

1. Select two or more peaks in different parts of the spectrum. To do that:
 - a) Zoom in on a peak by drawing a box around it. To do that, click-hold the left mouse button and move the mouse (see Fig. 12.2).
 - b) Right-click at the peak position and choose **Add** from the popup menu.

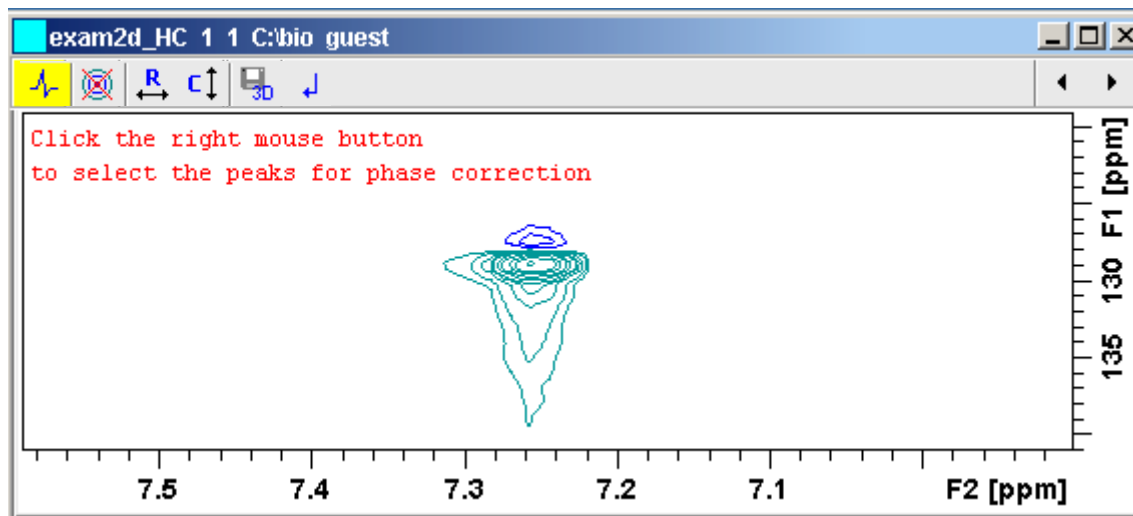


Figure 12.2


- c) Click the  button to display the full spectrum.
- d) Zoom in on the next peak and add in the same way as the first one.
- e) Zoom in on the next peak etc.

Fig. 12.3 shows an example of three selected peaks.

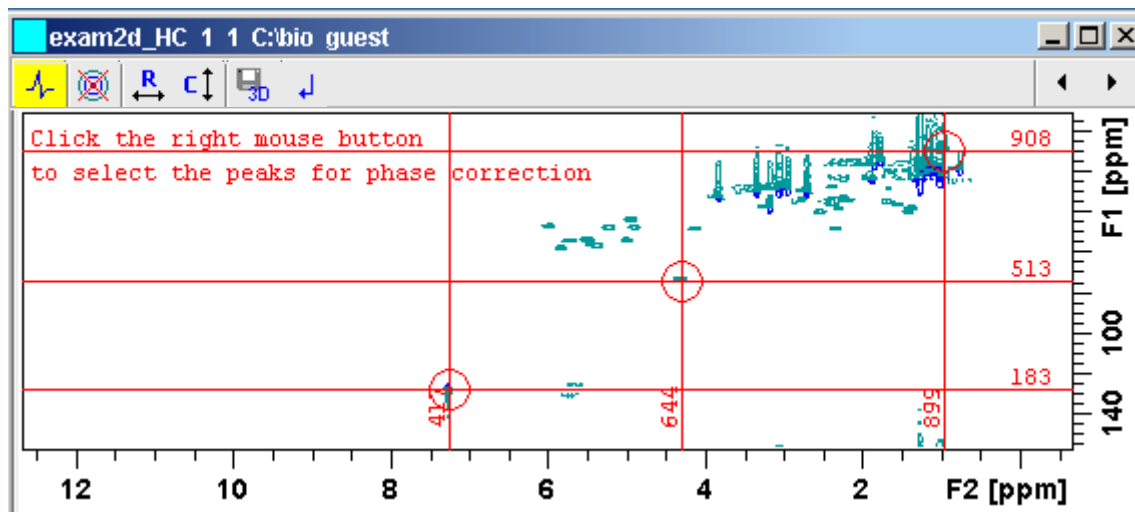



Figure 12.3

2. Click the button  to phase correct the columns (F1).

A new data window called *Phase 2D* will appear showing the columns of the selected peaks (see Fig. 12.4).

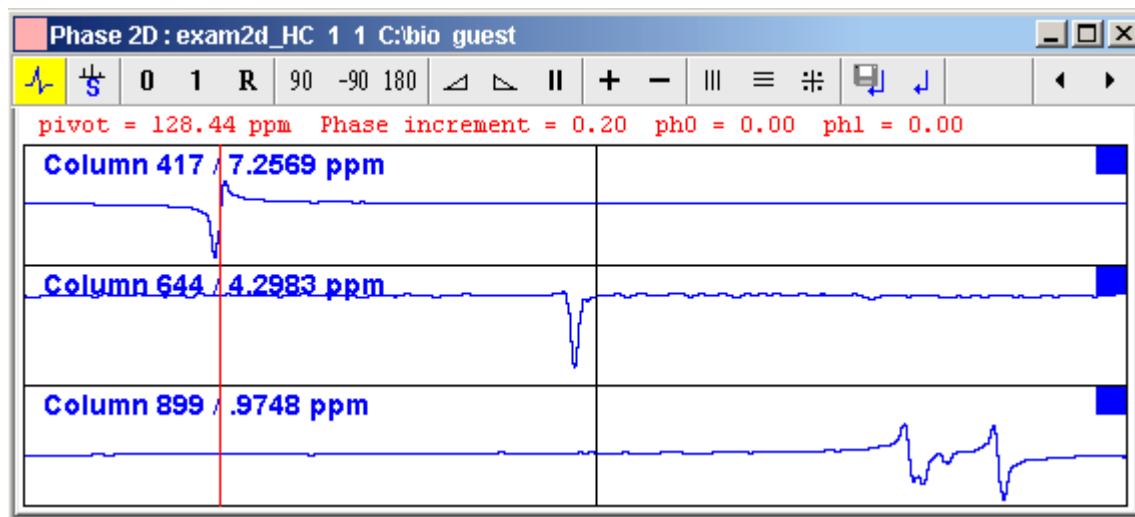



Figure 12.4

Note that the toolbar and the right-click popup menu offer the full 1D phase correction functions.

By default, all columns are selected as indicated by the filled blue squares ■. The red vertical line indicates the default pivot point in the upper column.

3. A typical way to perform phase correction is:

- Click-hold the **0** button for zero order correction and move the mouse until the reference peak of the first column is exactly in absorption mode.
- Click-hold the **1** button for first order correction and move the mouse until the reference peak in other column is exactly in absorption mode.
- Click the  button to execute, save and return (see Fig. 12.5).

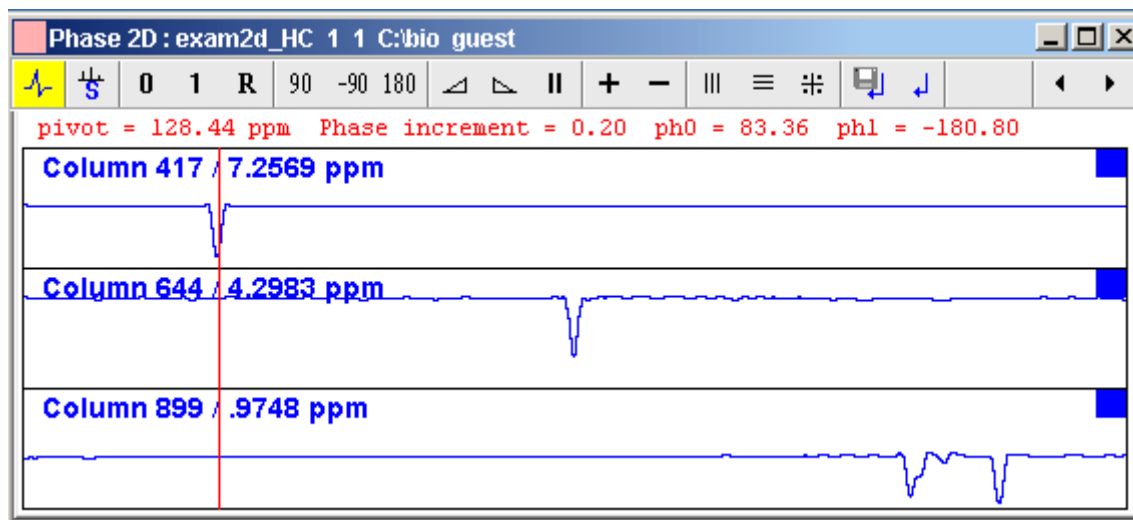


Figure 12.5

12.1.2 2D Interactive Phase Correction Details

How to Scale or Shift Individual Rows/Columns

To select one row or column:

- Click in the corresponding part of the data window.

The selected row/column will be marked with a filled blue square ■ whereas unselected rows/columns will be marked with an unfilled blue square □. Selecting

a single row /column allows you to shift and scale it separately from the other rows/columns as shown in Fig. 12.6.

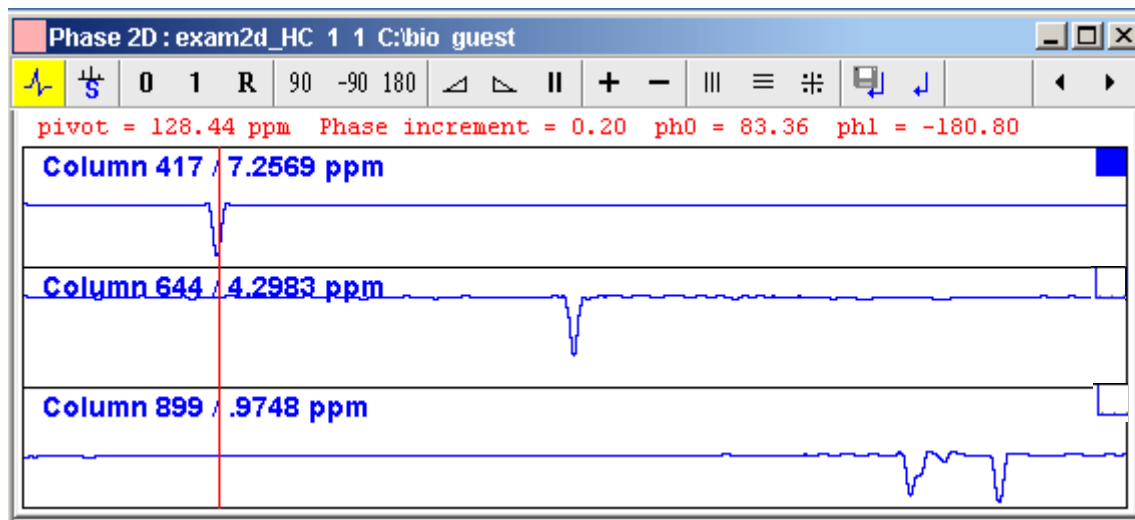



Figure 12.6

To select all rows or columns,


☞ Click the following button:

 Select all rows or columns.

How to Perform Smooth Phase Correction

To perform zero order phase correction:

1. Click-hold the following button (it turns green) and move the mouse:

 Zero order phase correction.


until the reference peak of the first row/column is exactly in absorption mode.

2. Release the mouse (button turns grey).

The parameter PHC0 will be set accordingly.

To perform first order phase correction:

1. Click-hold the following button (it turns green) and move the mouse:

 First order phase correction.

until the reference peak of the second and further rows/columns is exactly in absorption mode.

2. Release the mouse (button turns grey).

The parameter PHC1 will be set accordingly.

How to Perform 90, -90 or 180° Zero Order Phase Correction

Click one of the following buttons:


 90° zero order phase correction.

 -90° zero order phase correction.

 180° zero order phase correction.


How to Reset the Phase to the Original Values


Click the following button:


 Reset zero and first order phase.

How to Change the Mouse Sensitivity

Click one of the following buttons:


 Increase (double) the mouse sensitivity [*.inc*].

 Decrease (halve) the mouse sensitivity [*.dec*].

 Reset the mouse sensitivity to 1.0.


How to Show the Next/Previous Row or Column

To show the next row/column, click the following button:

 Show next row/column.

Note that only the selected row/column is increased. If all rows/columns are selected, only the first one is increased.




To show the previous row/column, click the following button:

 Show previous row/column.

Note that only the selected row/column is decreased. If all rows/columns are selected, only the first one is decreased.


How to Arrange Rows or Columns

Click one of the following buttons:

-  Arrange rows/columns horizontally.
-  Arrange rows/columns vertically (see Fig. 12.6).
-  Arrange rows/columns vertically in a split window.

How to Return from Multi-1D Phase to 2D Phase Display


Click the following button:

 to save, execute and return.

This will perform the following tasks:

- Execute phase correction.
- Save the current phase correction values.
- Leave the multi-1D phase mode.

Click the following button:

 to return to the 2D phase display without save.

How to Return from 2D Phase Mode

Click the following button:

 Return.

12.2 2D Multiple Display and Row/Column Handling

2D multiple display shows a 2D spectrum with an arbitrary number of 1D and/or 2D spectra superimposed.

Spectra are ppm aligned or Hz aligned, according to the selected axis unit.


A superimposed 1D spectrum is automatically displayed in the direction of the matching nucleus (for a hetero-nuclear 2D) or in the F2 direction (for a homo-

nuclear 2D).

Although multiple display is normally used for spectra with matching nuclei, it allows you to superimposed spectra with non-matching nuclei. You will get a warning that the nuclei do not match. Just click **OK** to continue.

How Switch to Multiple Display mode and Read Multiple Spectra

Switching to multiple display and reading multiple spectra can be done in two different ways:

- Read a 2D dataset and click  to switch to multiple display mode. Then add 1D and/or 2D spectra, e.g. from the browser or with **re**.

or

- Select multiple spectra in the browser, right-click one of them and click **Display**.

For a more detailed description of reading multiple data in multiple display mode, see chapter 11.4.

In multiple display mode, the Tab bar of the active data window is replaced by a toolbar (see Fig. 12.7).

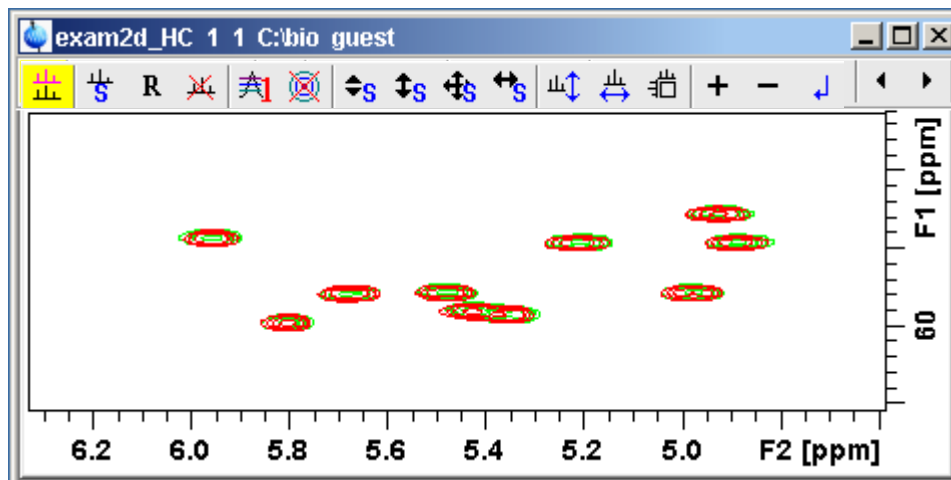


Figure 12.7 Multiple display with two 2D spectra superimposed

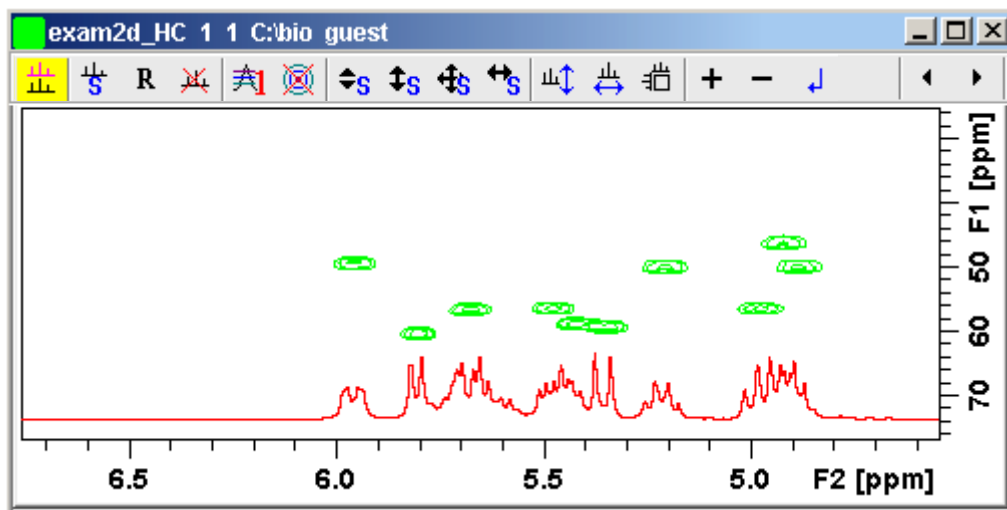




Figure 12.8 Multiple display with a 1D spectrum superimposed on a 2D spectrum

-  The yellow button indicates that the data window is in multiple display mode.
-  Some buttons will turn green when they are clicked. As long as a button is green, it is active.

The browser/portfolio in multiple display is split in two parts (see Fig. 12.9). The additional lower part shows:

- which datasets are displayed in the active data window
- which datasets are selected (they are highlighted)

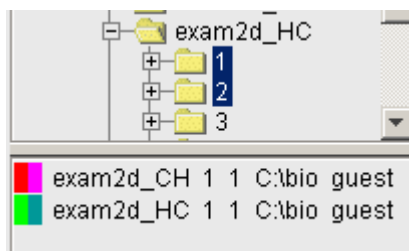


Figure 12.9

How to Align Multiple 2D Spectra

2D spectra in multiple display can be individually shifted. To do that:


1. Select one of the spectra in the lower part of the browser.
2. Click-hold the  button and move the mouse.

Fig. 12.10 shows a region of two comparable $^1\text{H}/^{13}\text{C}$ inverse 2D datasets which are shifted relative to each other.

Clicking the **R** button resets individual scaling and shifting.

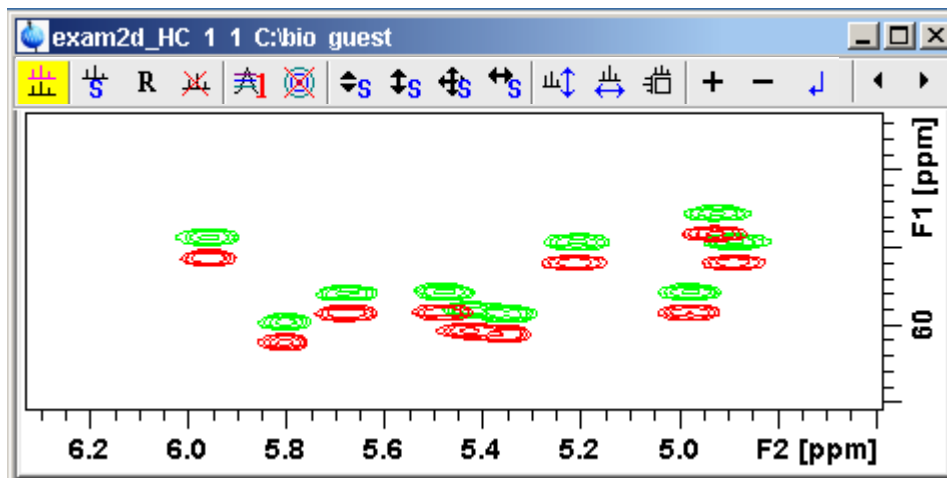




Figure 12.10

How to Scan Rows/Columns

Click the following button (it turns green) and move the mouse in the data field:

 to scan rows in the 2D spectrum.

Click the following button (it turns green) and move the mouse in the data field:

 to scan columns in the 2D spectrum.

To scale up the displayed row/column:

-  Click the left mouse button or turn the mouse wheel up.

To scale down the displayed row/column:

☞ Click the middle mouse button or turn the mouse wheel down.

How to Grab a Row/Column

You can grab a row or column, i.e. keep it displayed in the data window as follows:

1. Scan rows or columns as described above and hold at the desired position.
2. Right-click in the data window.
3. Choose **Grab Row/Column** from the popup menu (see Fig. 12.11).

Note that a grabbed row/column appears in the lower part of the browser. It can be selected there and individually scaled or shifted.

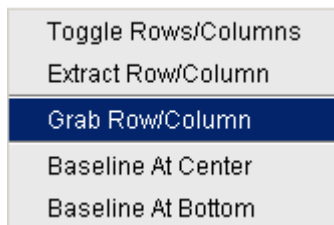


Figure 12.11

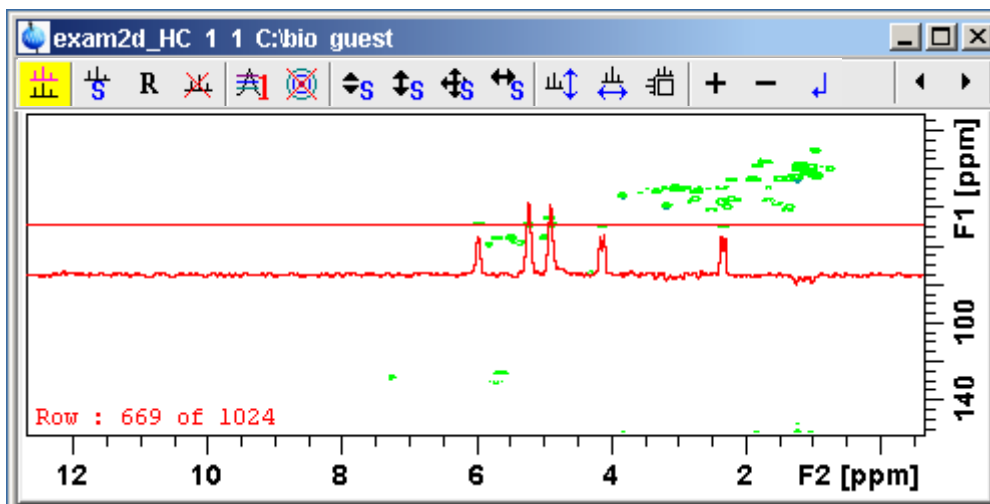


Figure 12.12

Fig. 12.12 shows row 669 with the 1D baseline at the center of the data window.

How to Extract a Row/Column

1. Scan rows or columns as described above and hold at the desired position.
2. Right-click in the data window and choose **Extract Row/Column** from the popup menu (see Fig. 12.11).
3. Specify the row/column number and output *procno* in the dialog box. Note that the ROW/COLUMN field is initialized with the grabbed row/column or, if no grabbing was done, with the current row/column.
4. Click **OK**

The extracted row or column is stored as a 1D dataset under the specified PROC-NO and displayed in a new data window. In the upper left part is this, the row number and source 2D dataset is specified (see Fig. 12.13).

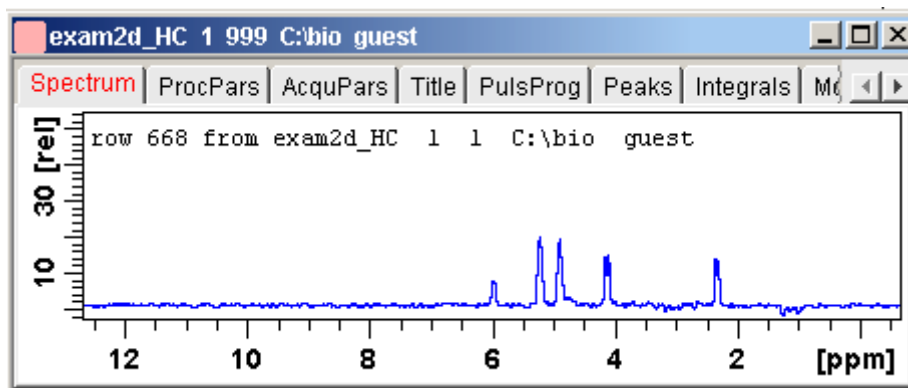


Figure 12.13

How to Copy Contour Levels from First to Other Spectra

Click the following button:



Copy contour levels from the first to the other spectra.

Note that the contour levels are only changed on screen, not on disk.

How to Switch on/off 2D contour display

Click the following button:

 Switch on/off 2D contour display.

How to Position the Baseline of the Row/Column

To put the baseline at the center of the data window:

1. Right-click in the data window.
2. Choose in *Baseline At Center* from the popup menu (see Fig. 12.11).

To put the baseline at the bottom of the data window:


1. Right-click in the data window.
2. Choose in *Baseline At Bottom* from the popup menu (see Fig. 12.11).

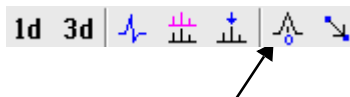
This works both in the scan submode or on a grabbed row/column.

12.3 2D Interactive Calibration

A 2D spectrum can be calibrated, automatically with the command *sref* or, interactively as described below.

How to Switch to 2D Calibration mode

 Click the corresponding button in the upper toolbar:.



The Tab bar of the active data window will be replaced by a toolbar (see Fig. 12.14).

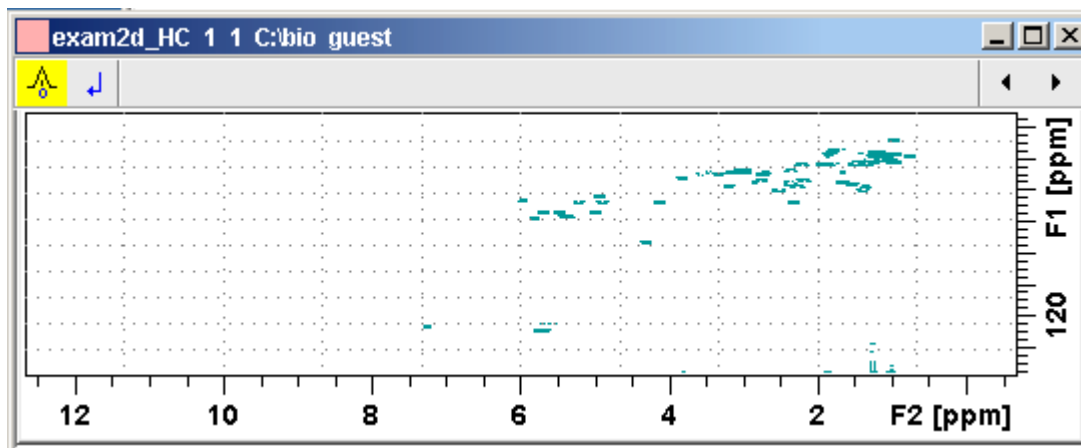


Figure 12.14 Data window in calibration mode



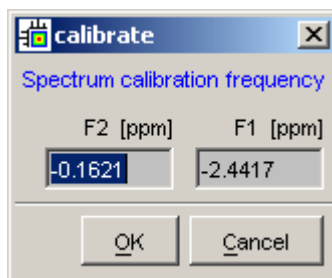
The yellow button indicates that the data window is in calibration mode.

How to Perform 2D Calibration

In calibration mode:

1. Left-click in the data window at the reference peak.

The following dialog box will appear:



Note that the units for F2 and F1 (Hz or ppm) correspond to the axis units of the display.


2. Enter the F2 and F1 frequency you want to assign to the reference peak.
3. Click **OK**.

The spectrum will be calibrated and re-displayed. The calibration button will turn grey again.

12.4 2D Chemical Shift Distance Measurement

How to Measure a 2D Chemical Shift Distance

1. Click the following button (button turns green):

 Chemical shift distance measurement.

2. Click-hold the left mouse button at one peak position and drag the mouse to another peak position.

The distance in ppm, will be displayed.

3. Right-click in the data window to quit distance mode (button turns grey).

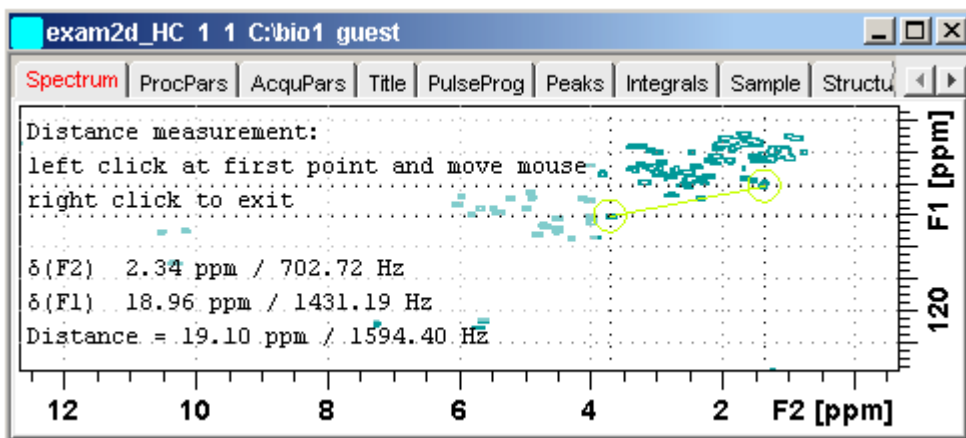


Figure 12.15 Data window in distance measurement mode

12.5 2D Integration

TOPSPIN provides automatic 2D integration. Before you do this, you must first perform peak picking using the **pp** command.

Automatic 2D integration can be started as follows:

☞ click *Analysis* → *Integration* [*int*]

This opens the following dialog box.

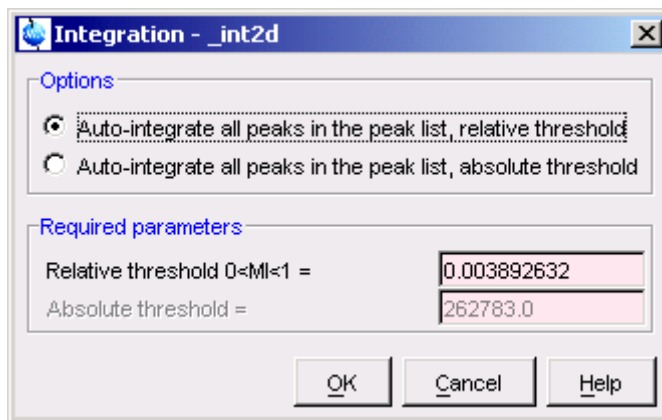


Figure 12.16

Here you can choose between integration using a relative or absolute threshold and set the required parameters. Integral regions are only determined for picked peaks. The calculated integrals will be marked in the data field with the letter **I** and can be listed by clicking the *Integrals* tab.

Fig. 12.17 shows a region of peaks after peak picking. Fig. 12.18 shows the same region after 2D integration. Here you can see the integral labels and areas (in this case yellow). The area color can be set in the user preferences (command *set*) as *Color of 3rd spectrum*.

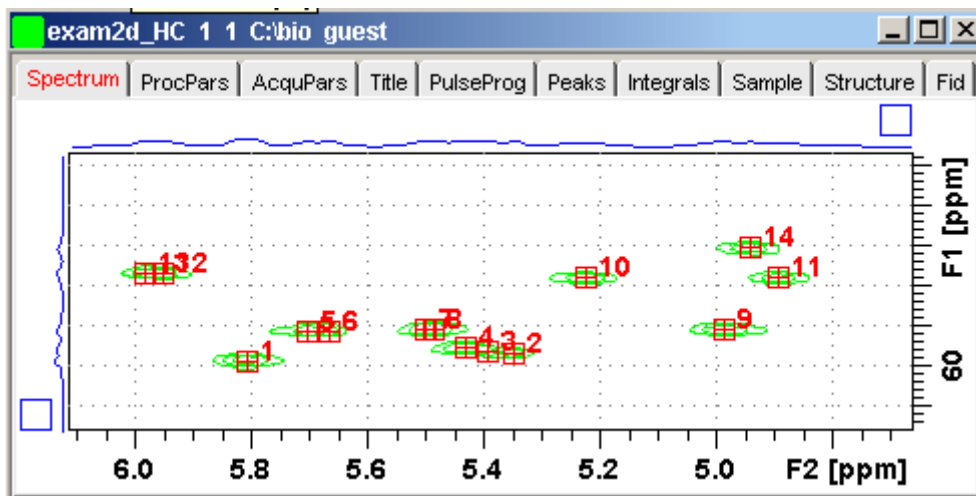


Figure 12.17

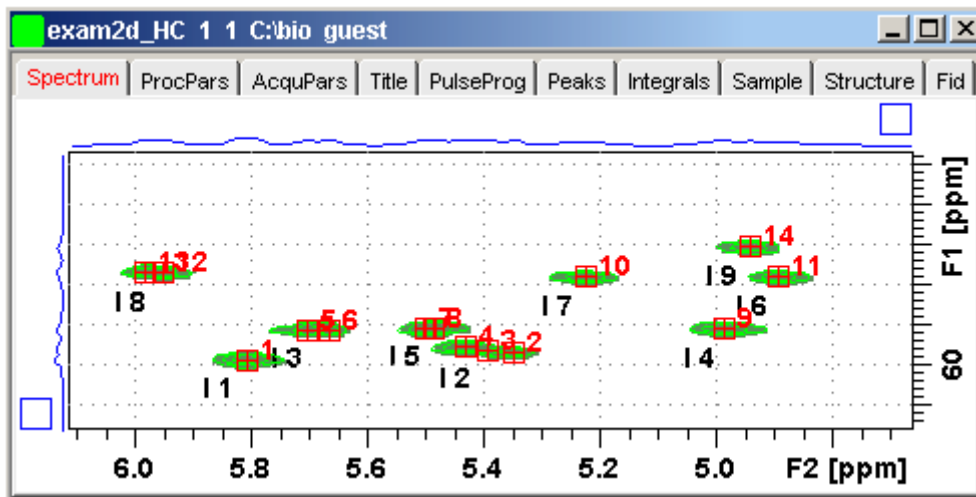


Figure 12.18

Chapter 13

Data Window Handling

13.1 Data Windows

The TOPSPIN window has a data area that may contain multiple data windows. The size of the data area depends on the overall size of the TOPSPIN window and on presence of the Browser and/or Processing Guide. Fig. 13.1 shows the TOPSPIN window with the Browser and three data windows.

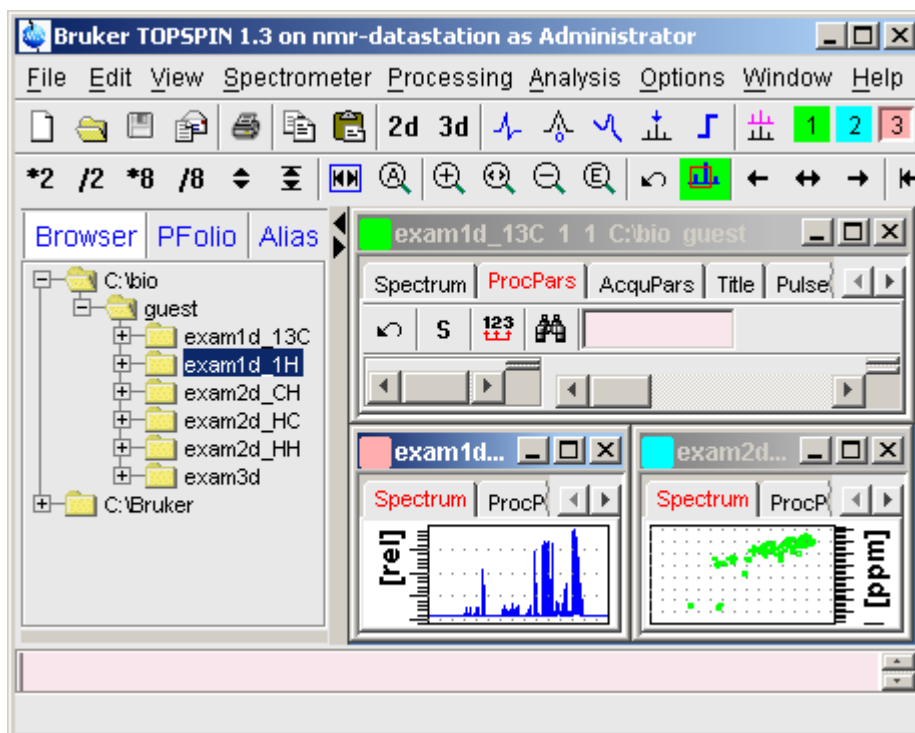


Figure 13.1

Note that the three data windows show different data objects: 1D processing parameters, a 1D spectrum and a 2D spectrum.

How to Move a Data Window

☞ Click-hold the title bar and move the mouse.

How to Resize a Data Window

1. Move the cursor to the window edge until it becomes a double-headed arrow.
2. Left-click-hold that position and move the mouse.

Depending on the position of the double-headed arrow, you can change the window height, width or both (see Fig. 13.2)

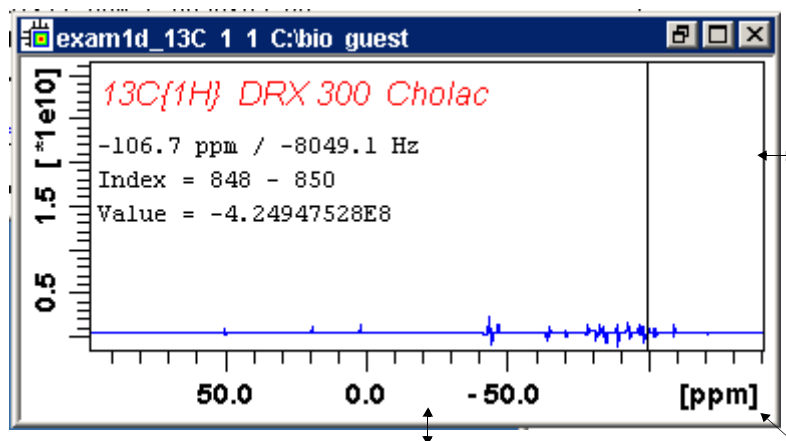


Figure 13.2

How to Select (activate) a Data Window

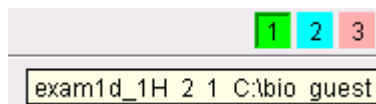
The active data window is the window of which the title bar is highlighted. The TOPSPIN menu, tool bars and command line commands correspond to and act on that window. Only one data window is active at a time.

To activate a different data window:

☞ Click in the desired data window or click its title bar.

or

☞ Click one of the colored radio buttons above the data area. The pressed radio button (the green one in the example below) corresponds to the current dataset.



If you hold the cursor over one of the buttons without clicking it and wait a few seconds, the corresponding dataset specification will be shown.

or

☞ Click **Window** → x *dataname expno procno dir user*

where x is the number of the desired window and *dataname*, *expno*, *procno*, *dir* and *user* refer to the dataset displayed in that window.

or

- ☞ Hit the **F6** key to activate the next window. Repeat that until the desired window is the active window.

How to Open a New empty Data Window

- ☞ Click **Window** → **New window** [**Alt+w-n**]

The new data window will become the active window and will, by default, cover the entire data area, hiding possible existing data windows. To open a dataset in the new window, drag a dataset from the browser or from the Windows Explorer into the new window or click **File** → **Open** (see also chapter 4.3).

How to Arrange Data Windows

If the data area contains multiple data windows, you can arrange them in various ways. All the arrange commands arrange the windows left to right and/or top to bottom in the order in which the windows have been active. The currently active data window will therefore be positioned at the top and/or left of the data area.

To arrange the data windows as a grid:

- ☞ Click **Window** → **Arrange as a Grid**

Depending on the number of windows, they will be arranged vertically and/or horizontally (see Fig. 13.3).

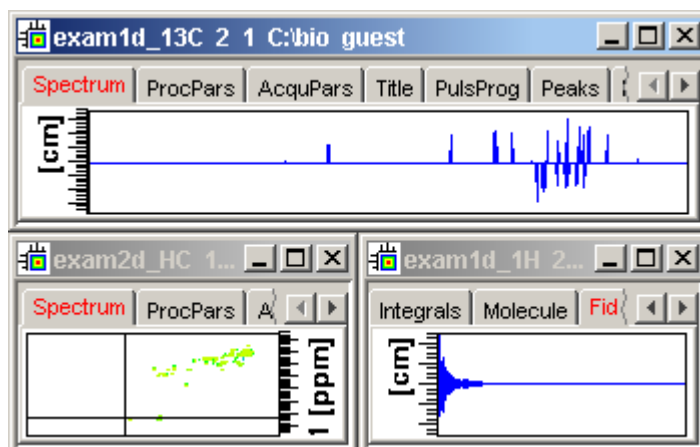


Figure 13.3

To arrange data windows in stack (see Fig. 13.4):

☞ Click **Window** → **Arrange in Stack**

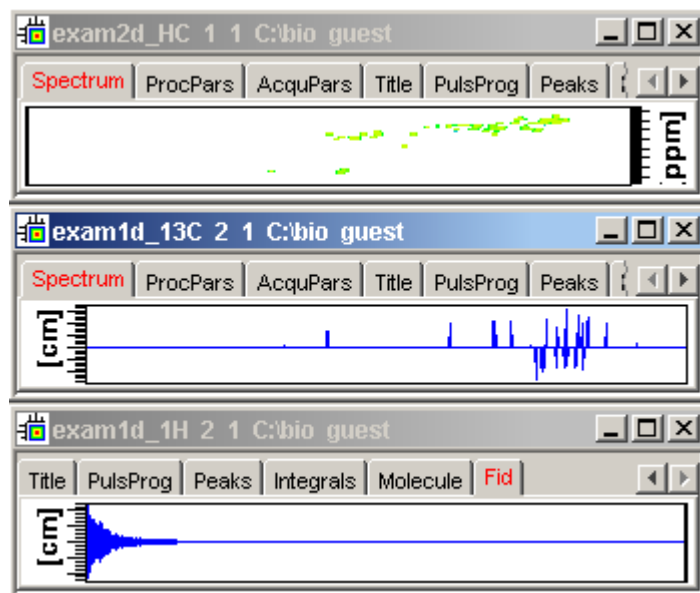


Figure 13.4

To arrange data windows side by side (see Fig. 13.5):

☞ Click **Window** → **Arrange Side-by-Side**

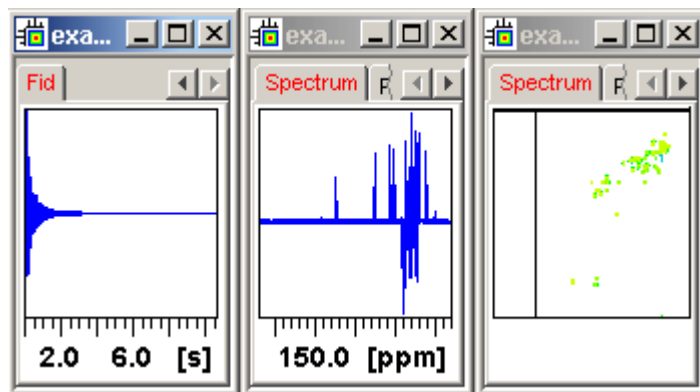


Figure 13.5

To cascade data windows (see Fig. 13.6):

☞ Click **Window** → **Cascade**

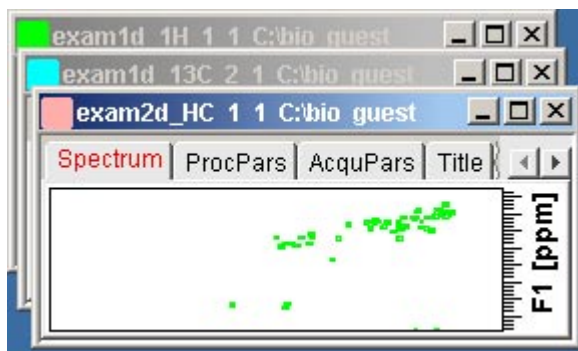



Figure 13.6

Note that you can instruct TOPSPIN to open new data windows cascaded rather than maximized as well configure cascaded windows (command **set** → **Window settings**, see also chapter 4.3)


How to Iconify (minimize) a Data Window

☞ Click the  button in the windows title bar


or

☞ Click **Window** → **Iconify all** to iconify all windows.

How to De-iconify a Data Window


☞ Click the  button or double-click the title bar.


How to Maximize a Data Window

☞ Click the  button or double-click the title bar.

The window will cover the entire data area. Note that a maximized window cannot be moved or resized but can be restored (in size and position), iconified or closed.

How to Restore the Size and Position of a Data Window

☞ Click the  button or double-click the title bar.


Note that this is only possible if the title bar contains the  button. This is only the case after the window has been maximized or iconified.

How to Close a Data Window


To close the active data window:

☞ Click **File** → **Close** [**Ctrl-w**]

or

☞ Click the  button in the windows title bar.

To close any data window:

☞ Click the  button in the data windows title bar

or

☞ Click the title bar and then click **File** → **Close** [**Ctrl-w**].

To close all data windows:

☞ Click **File** → **Closeall** [**closeall**]

How to Iconify all Data Windows

☞ Click *Window* → *Iconify all*

How to Maximize all Data Windows

☞ Click *Window* → *Maximize all*

The active window will be displayed on top, all other windows are hidden.

How to Activate the Next Data Window

☞ Click *Window* → *Next window [F6]*.

The windows title bar will become highlighted.

13.2 Window Layouts

A data window layout defines the position, geometry and window type of one or more TOPSPIN windows. The following windows types are available:

- data windows
- lock display window
- acquisition display window
- BSMS display window
- temperature unit window

How to Save the Current Window Layout

1. Click *Window* → *Save layout*
2. In the appearing dialog box:
Specify the layout File name (extension `.prop`) and click *Save Layout*

How to Read a Window Layout

1. Click *Window* → *Read layout*
2. In the appearing dialog box:
Specify or click the layout File name and click *Read Layout*

Windows are arranged according to the following rules:

- Each currently displayed window type gets the position and geometry to the corresponding definition in the layout.
- If a window type is displayed but not defined in the layout, it keeps its current position and geometry.
- If a window type is defined in the layout but not displayed, the layout definition is ignored.
- Multiple *data* windows are, arbitrarily, assigned to the available data window definitions.

How to Swap Data Windows

Within a certain layout, you can easily swap two TOPSPIN windows with the command ***swin***. If the data area contains exactly two windows, ***swin*** simple swaps their position and geometry. If it contains more than two data windows, ***swin*** opens a list from which you can select any window to be swapped with the currently selected (active) window. Swapping windows can also be executed from the ***Window*** menu.

Chapter 14

Analysis

This chapter describes various TOPSPIN analysis methods including chemical shift measurement, signal to noise calculation, solids line shape analysis, T1/T2 relaxation analysis and multiplet analysis.

14.1 1D Chemical Shift Distance Measurement

How to Measure a Chemical Shift Distance

1. Click the following button (button turns green):



Chemical shift distance measurement.

2. Left-click-hold at one peak position and drag the mouse to another peak position.

The distance in ppm, will be displayed.

3. Right-click in the data window or move the cursor out of the data window to leave distance measurement mode (button turns grey).

14.2 1D Signal to Noise Calculation

How to Perform Interactive S/N Calculation

1. Click *Analysis* → *Signal/Noise Calculation* [*.sino*].

The current signal region (parameters SIGF1-SIGF2) and noise region (parameters NOISF1-NOISF2) are displayed.

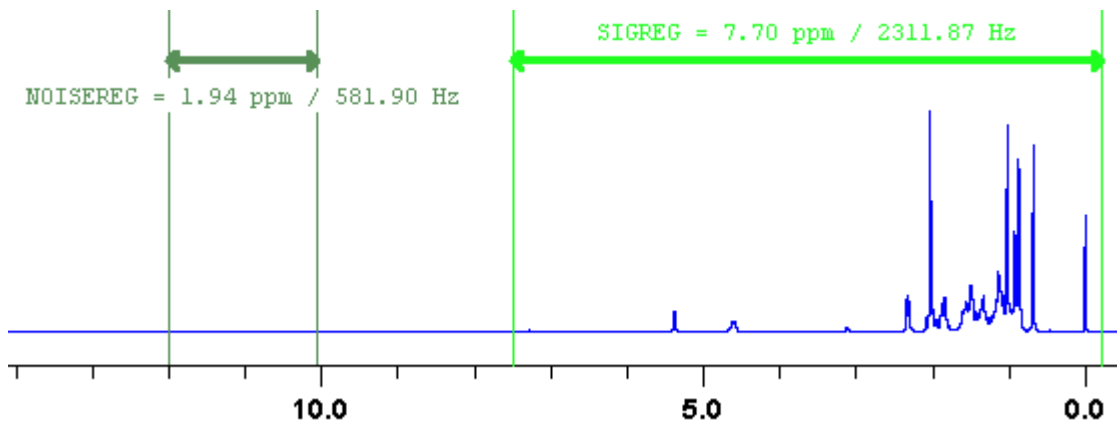


Figure 14.1 Data window in S/N measurement mode

2. Move the mouse into the data window.
3. Left-click-hold and drag the mouse from one edge of the *signal* region to the other edge.
A horizontal double-headed arrow will indicate the signal region.
4. Left-click-hold and drag the mouse from one edge of the *noise* region to the other edge.
A horizontal double-headed arrow will indicate the noise region.
5. Right-click any position in the data window. The popup menu as shown in Fig. 14.2 will appear.

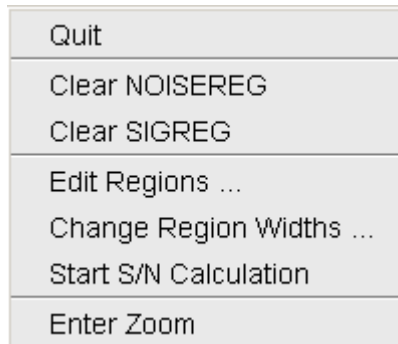


Figure 14.2

Choose *Start S/N calculation*

The other entries allow you to redefine or clear the regions. After the noise calculation has finished, the result will appear on the screen.

How to Delete the Signal Region or Noise Region

To delete the current signal region:

1. Right-click in the data window.
2. Choose *Clear SIGREG* from the popup menu (see Fig. 14.2).

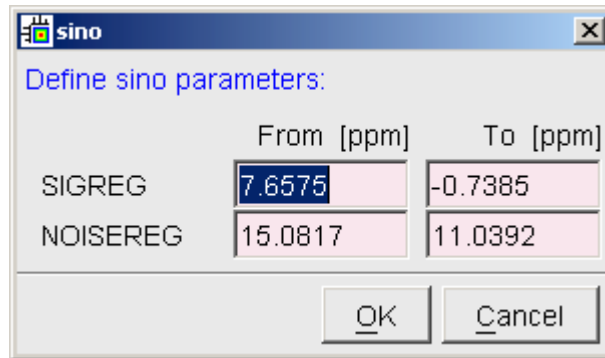
To delete the current noise region:

1. Right-click in the data window.
2. Choose *Clear NOISEREG* from the popup menu (see Fig. 14.2).

How to Edit the Limits of the Signal Region or Noise Region

1. Right-click in the data window.
2. Choose *Edit regions...* from the popup menu (see Fig. 14.2).

3. Enter new limit values in the appearing dialog box.

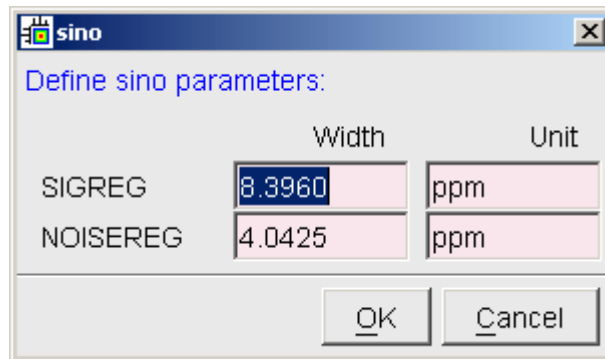


4. Click **OK**

The S/N value is automatically recalculated and displayed.

How to Change the Width of the Signal Region or Noise Region

1. Right-click in the data window.
2. Choose *Change region width...* from the popup menu (see Fig. 14.2).
3. Enter new width values in the appearing dialog box.



4. Click **OK**

Note that as you change the width, the right limit is modified correspondingly. The left limit is kept. The S/N value is automatically recalculated and displayed.

14.3 Solids Line Shape Analysis

Solids Line Shape Analysis allows you to simulate and fit calculated spectra to various experimental 1D solid NMR spectra. The following fitting models are available:

- Gauss/Lorentz
- Chemical Shift Anisotropy
- Quadrupolar Central Peak of the +/- 1/2 Transition of a Quadrupolar Nucleus
- All Quadrupolar Transitions of a Quadrupolar Nucleus
- The Combination of the Chemical Shift Anisotropy and Quadrupolar Interaction

You can simulate powder spectra of static or rotating samples at single or double axis conditions. Both rotation angles can be set. The inner and outer rotating speeds are freely adjustable. For rotating samples, a maximum of ten rotation side bands and five DOR bands can be set. You can simulate one 1D spectrum with a maximum of 25 observable nuclei, i.e. 25 sites of a nucleus. Ten other nuclei can be defined as dipolar coupling partners of the observed nucleus (Topspin 1.3 allows only one observed nucleus (site) if you define heteronuclear dipolar couplings).

Spectrum Preparation

Before starting Solids Line Shape Analysis, the 1D spectrum must be properly phase corrected and baseline corrected.

Switch to Line Shape Analysis Mode

To switch to Solids Line Shape Analysis mode:

☞ click *Analysis* → *Solids Lineshape Analysis* [*solaguide*]

This opens the workflow as shown in Fig. 14.3.

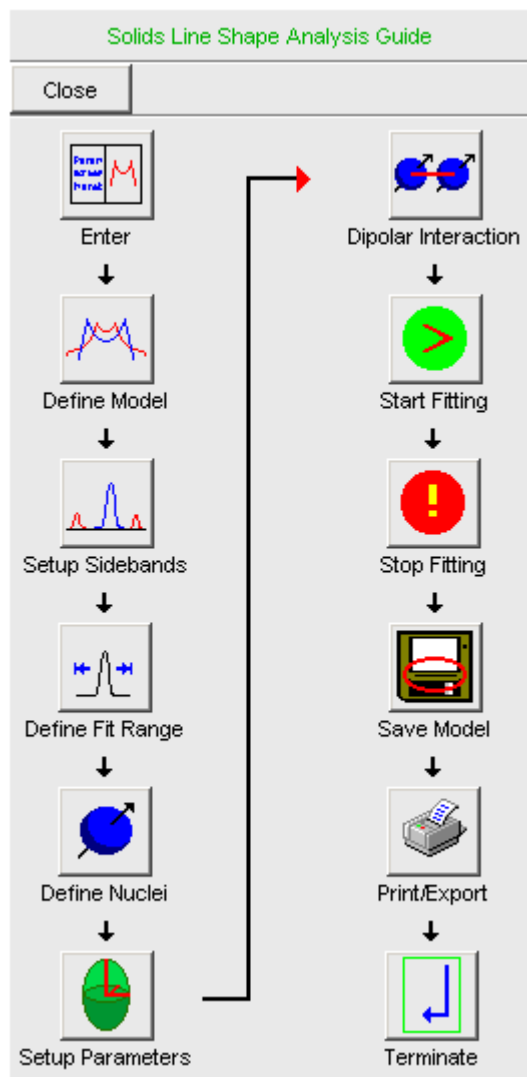


Figure 14.3

Clicking *Enter* here will split the data window in two. Experienced users can enter this mode directly and skip the workflow with the command *sol1a*.

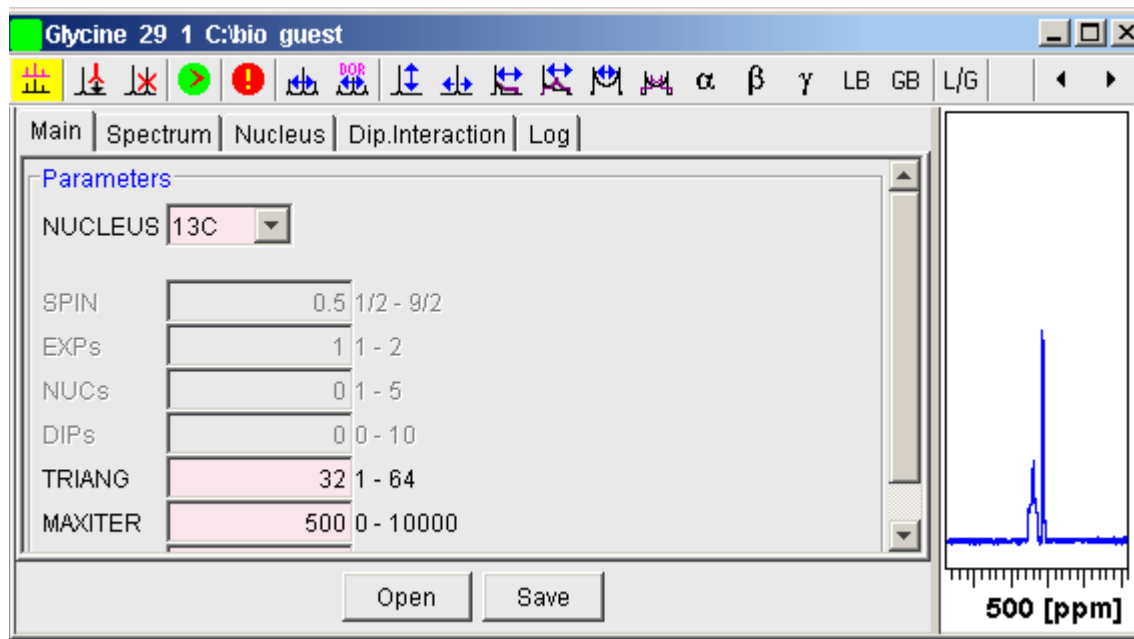


Figure 14.4

The right part of this window is the data window showing the 1D experimental and calculated spectrum. The left part is the parameter window with five panels, where the second one, the *Main* panel is selected by default.

The simulation procedure

The simulation procedure consists of the following steps:

Set Optional parameters

In the *Main* panel, you find some parameters which you can normally leave unchanged. See the section *Simulation Details* below for more information. If a simulation on the current dataset has already been done and stored, you can read this by clicking *Open*. If not, you will start from scratch and setup the simulation as described below.

Define the Model

Click the *Spectrum* panel and select the *Model* according to your experiment. You can refine the experimental conditions by checking one of the following

boxes:

- *All* for all quadrupolar transitions.
- *DOR* for performed double rotation experiments.
- *Sync* for rotor synchronized experiments.

When a parameter is greyed, this means it cannot be changed for the selected model or it is related to one of the other (checked) parameters.

Define Rotation Parameters

From the *Spectrum* panel, set the values for:

- MASR - rotation speed of the single axis MAS (VAS) experiments or DOR outer axis speed.
- DORR - inner rotation speed in DOR experiment.

and set their checkmark if they must be optimized during the simulation.

Set the value for:

- SBands: the number of side bands on one side of the central transition.

Define the Spectral Region


Typically, non-overlapping experimental peaks are fitted in separated simulations. Before each simulation, the region to which it is applied must be defined.

1. Zoom in on the region to be simulated.
2. Right-click in the data window choosing *Define Fitting Region Using Display Region* or click *DefReg* in the panel window.

This will set the parameters F1P and F2P in the Spectrum panel. Alternatively, you can enter the values in the respective parameter fields.

Define Nuclei parameters

For each observable nucleus (site), a set of parameters (see below) must be set. To do that:

1. Click the *Nucleus* panel or the  button.
2. Click *Add* if the nucleus is not shown yet.
3. Adjust the nuclei parameters until the calculated spectrum approximately fits the experimental spectrum. You can do that as follows:

☞ Enter the values in the parameter field

or

☞ Click the radio button to the right of the parameter, click-hold the corresponding toolbar button (colored green) and move the mouse horizontally.

Note that the calculated spectrum in the data field is automatically updated as you adjust a parameter (see Fig. 14.5).

4. Check the parameters which must be optimized during the simulation.

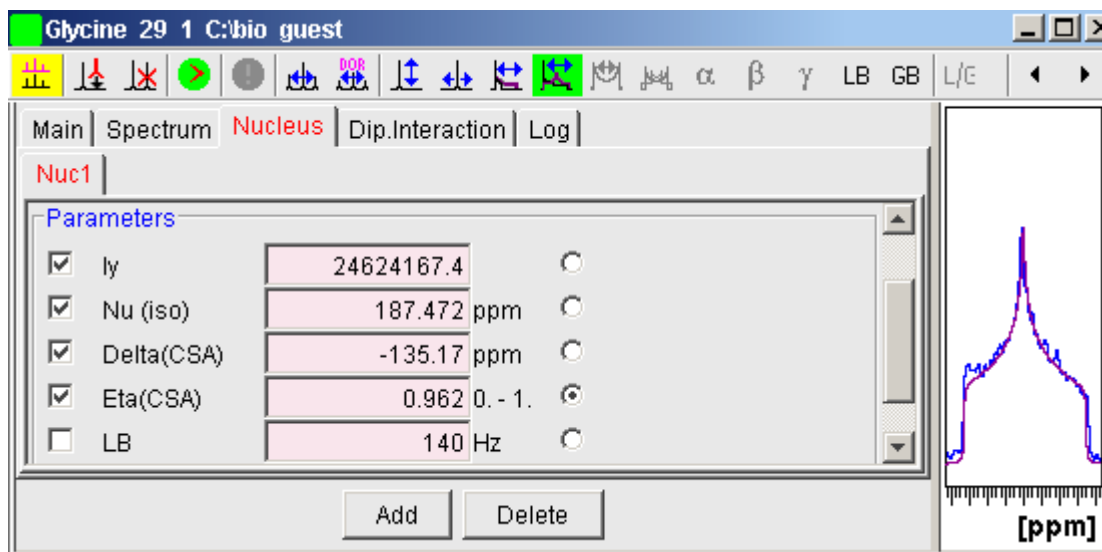


Figure 14.5

Define Dipolar coupling nuclei (if they exist)

Dipolar coupling nuclei can be defined if only one observe nucleus is defined. To set dipolar coupling parameters:

1. Click the **Dip. Interaction** panel.
2. Click **Add** if the nucleus is not shown yet.
3. Set the nuclei parameters as follows:

☞ Enter the values in the parameter field

or

- Click the radio button to the right of the parameter, click-hold the corresponding toolbar button (colored green) and move the mouse horizontally.

4. Check the parameters to be optimized during the simulation.

Start the Simulation

Now you can start the iterative simulation. To do that:

- Click the  toolbar button.

The simulated spectrum is displayed in the data window and continuously updated (see Fig. 14.6).

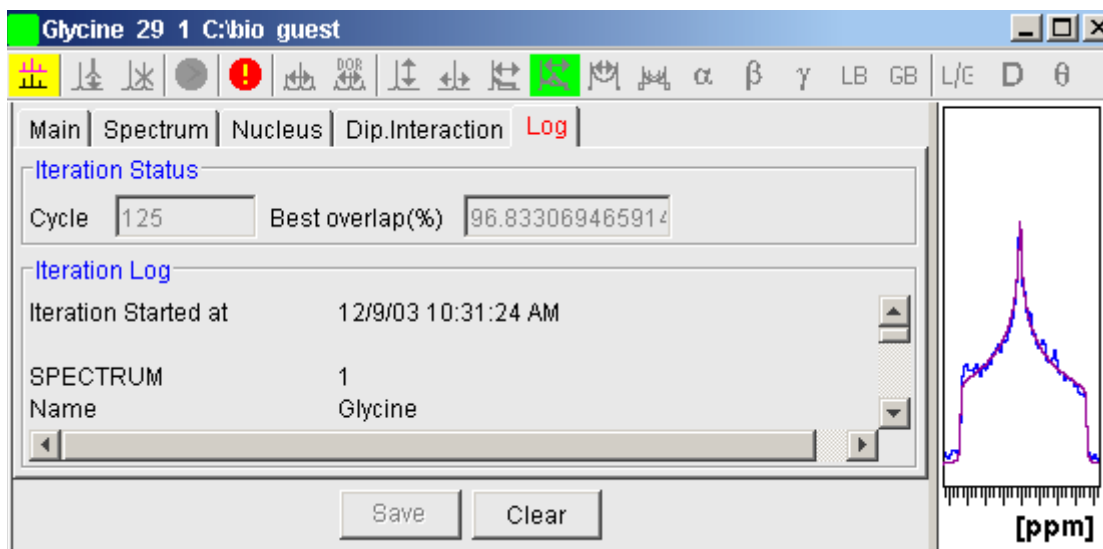


Figure 14.6

The parameter window will switch to the **Log** panel showing:

- *Iteration Status*, including the iteration *Cycle* and the *Best Overlap* percentage so far.
- *Iteration Log* with the starting parameters and the results of the fit. Parameters which are marked with an asterisk have been optimized dur-

ing the simulation. They are automatically updated in the respective panels.

To save the Log panel information, click the *Save* button.

To clear the Log panel, click the *Clear* button.

During the simulation process, you can freely switch to other panels to view the parameter being optimized. After the simulation has finished, the *Nucleus* panel will show the optimized values (see Fig. 14.7).

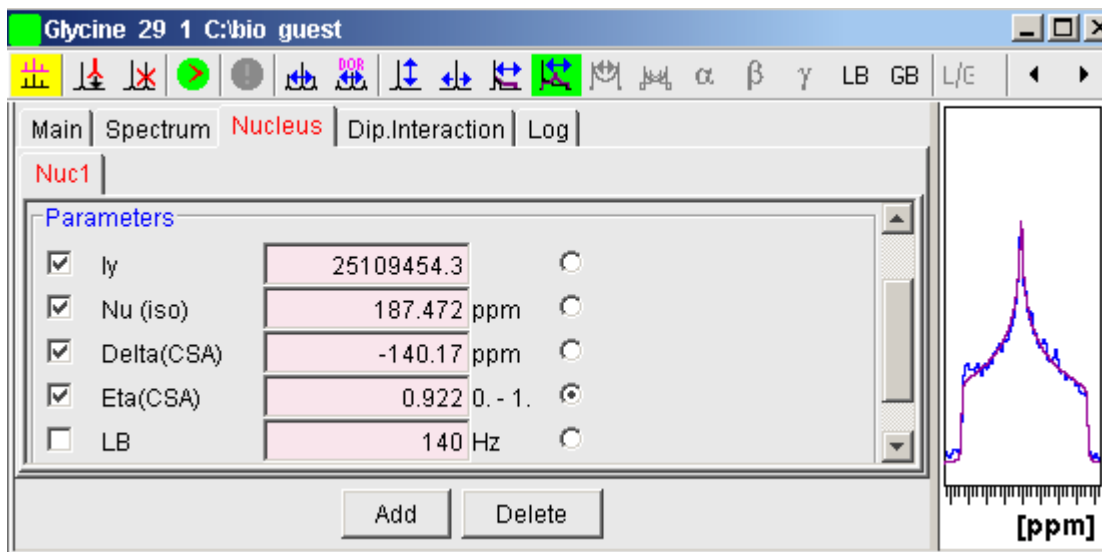


Figure 14.7

Abort the Simulation

To abort a running simulation:

Click the  toolbar button.

After a few seconds the iteration stops and the best spectrum will be shown in the data window.

Save the Simulation

After the simulation process is finished for all spectral regions of interest:

1. Switch to the *Main* panel.

2. Click *Save*

to save all parameters.

Exit Solids Lineshape Analysis

To leave the solids analysis mode:

☞ Click the  toolbar button.

Simulation details

Basic parameters

The Main panel shows you a list of basic parameters:

- NUCLEUS: the observe nucleus. By default, this parameter is set to the value of the acquisition parameter NUC1.
- SPIN: spin of the observe nucleus. It is automatically set according to the selected NUCLEUS.
- EXPs: number of experimental spectra.
- NUCs: number of nuclei (sites of the observe nucleus).
- DIPs: number of dipolar coupling nuclei.
- TRIANG: number of triangles involved in powder spectrum simulation with several random oriented crystallites. The default value of 32 generally results in a good quality spectrum.
- MAXITER: maximum number of iterations.
- SSIZE: initial step size for the iterated parameters. The value represents the fraction of the initial parameter value. It ranges from 0.0 to 1.0 with a default of 0.1.

The values of NUCs and DIPs will automatically be updated when you add or delete nuclei from the *Nucleus* and *Dip.Interaction* panels.

Spectrum parameters

Spc1

TOPSPIN 1.3 supports only one experimental spectrum to be fitted.

Models

Available fitting models are: *Gauss/Lorentz*, *CSA*, *QUAD central*, *QUAD all*

and *QUAD & CSA*.

Experimental Spectrum

Shows the datapath variables of the experimental spectrum.

Parameters

The following parameters are available.

- MASR - rotation speed of the single axis MAS (VAS) experiments or DOR outer axis speed.
- DORR - inner rotation speed in DOR experiment.
- Angle and AngleInt - the outer and inner rotation angles.
- SBands and DORBands - number of calculated side bands.
- F1 and F2 - the left and right edge of the experimental spectrum.
- F1P and F2P - the limits of the region to be fitted. These must be within the F1-F2 range. To define F1P and F2P interactively, expand the spectrum and right-click in the data window choosing ***Define Fitting Region Using Display Region*** or click ***DefReg*** in the panel window.

The displayed parameters SI, O1P, SF, SW, SWH, LB, GB and SR are used by the fitting calculations. They are defined by the corresponding processing parameters (command ***edp***).

Nucleus Parameters

The section *Model* shows the spectrum model type, which was selected in the *Spectrum* panel. The section *Parameters* contains the nucleus dependent model parameters. The available parameters and the corresponding toolbar buttons are:



Iy - Signal intensity.














Nu(iso) - isotope chemical shift given in ppm.



Delta(CSA) - Chemical shift anisotropy parameter in ppm (can be positive, negative or zero).



Eta(CSA) - Asymmetry parameter ($0 \leq \text{Eta} \leq 1$).

-  CQ(Quad) - Quadrupolar coupling constant in kHz.
-  Eta(Quad) - Asymmetry parameter of the quadrupolar coupling tensor.
($0 \leq \text{Eta} \leq 1$)
-  α Alpha Euler angle of the 'CSA & Quad' tensor.
-  β Beta Euler angle of the 'CSA & Quad' tensor.
-  γ Gamma Euler angle of the 'CSA & Quad' tensor.
-  LB Line broadening parameter (half width if GB=0).
-  GB Gauss broadening parameter. If GB>0 then LB must be negative.
-  L/G Gauss component of the Gauss/Lorentz ratio. $0.0 \leq GL \leq 1.0$
Lorentz curve: GL=0, Gauss curve: GL=1. Used only by the *Gauss/Lorentz* model
-  D Dipolar coupling.
-  θ Theta (dipolar).
-  ϕ Phi (dipolar).

Dipolar nucleus parameters

TOPSPIN 1.3 supports a maximum of ten dipolar coupling partners. Note, however, that you can only simulate one observable nucleus (site) at a time if you define dipolar coupling nuclei.

- NUCLEUS - Coupling nucleus partner.
- Spin - Nucleus dependent Spin (Read only).
- D(dip) - Dipolar coupling constant.
- Angle2 - Euler angle of dipolar coupling vector.
- Angle3 - Euler angle of dipolar coupling vector.

Note that Dipolar couplings are invariant to the first Euler angle (rotation around the Z-axis), so this angle value cannot be set.

The Simplex algorithm

The Simplex iteration minimizes the least square difference of the experimental spectrum and the superimposed simulated spectra between F1P and F2P. The 'Best overlap %' value is determined as described below.

Calculate the area between the curves of the experimental and the calculated spectra.

$$A(\text{dif}) = \text{Sum}(\text{Abs}(\text{Yexp}(i) - \text{Ycalc}(i)))$$

Calculate the area of the experimental spectrum.

$$A(\text{exp}) = \text{Sum}(\text{Abs}(\text{Yexp}(i)))$$

Compare these to area values.

$$\text{Overlap}(\%) = 100 * (1 - A(\text{dif}) / A(\text{exp}))$$

Overlap(%) = 100, if A(dif)=0. This is the theoretical maximum of the signal overlap.

Overlap(%) = 0, if the calculated spectrum is similar to the experimental one, with same area, but they do not overlap.

Overlap(%) > 0, but <100. Partial overlap.

Overlap(%) > 70-90, good agreement.

Overlap < 0. No agreement. Change the initial parameters and start a new simulation.

14.4 Relaxation Analysis

Typically, relaxation data consist of a series of 1D FIDs measured with varying delays and stored as pseudo 2D data. To analyze these data, Topspin offers an easy to use T1/T2 Relaxation Guide. Relaxation curves of various experiment types with up to six components can be fitted.

To start the Relaxation Guide:

☞ Click **Analysis** → **T1/T2 Relaxation** [**t1guide**].

This will open the dialog box as shown in Fig. 14.8.

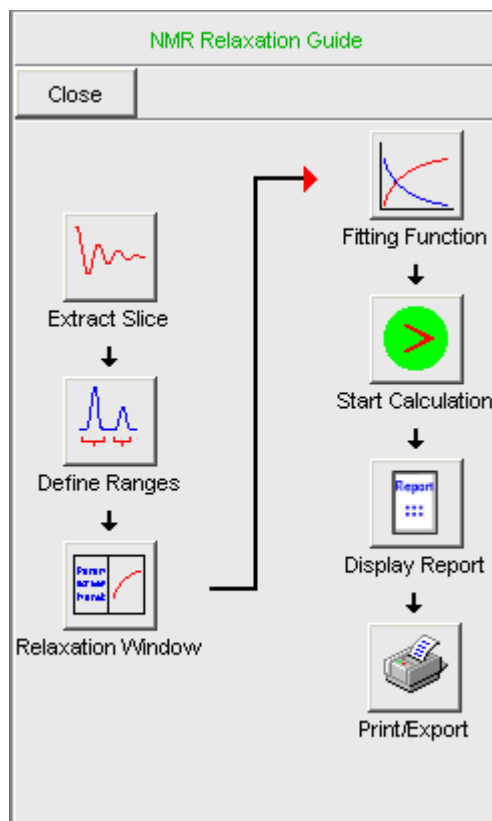


Figure 14.8

Just click the successive icons and follow the instructions on the screen. Note that holding the cursor over an icon shows the command line command that is executed when the icon is clicked. If you prefer to execute these commands from the command line, just click the *Close* button to close the Relaxation Guide.

Extract Slice

Prompts you for the FID or spectrum to be extracted for peak determination (see Fig. 14.9). Click *FID* to extract an FID or *Spectrum* to extract a spectrum. Note that the latter only works if the pseudo 2D data have been processed. If you click

FID, the extracted FID is automatically processed. We recommend to enter the

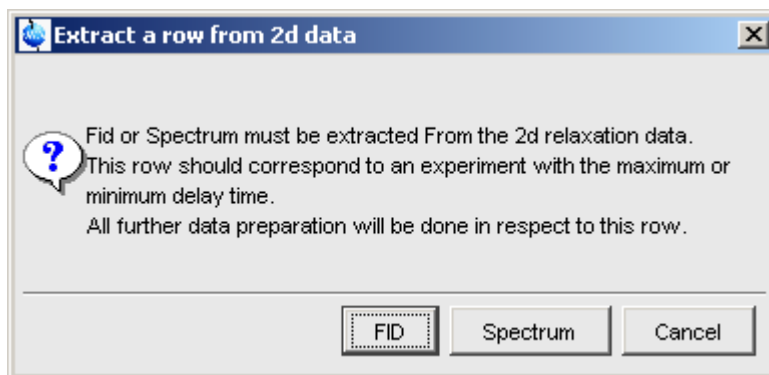


Figure 14.9




FID or spectrum number which was measured with the longest delay. It can be found in the *vdlist* file in the EXPNO data directory. A new data window will appear, showing the extracted 1D-FID.

Transform/Phase

Processes the extracted 1D FID performing exponential multiplication, Fourier transform and automatic phase correction.

Define Ranges

Switches to interactive integration mode. Here you can define the ranges for the peaks to be included in the relaxation analysis.

1. Click the  button in the data window toolbar.
2. Put the red cursor line at one edge of a peak or multiplet.
3. Left-click-hold and drag the cursor line to the other edge of the peak or multiplet.
4. Do step 2 and 3 for all regions to be defined.
5. Click the  button and choose ***Export Regions to Relaxation Module***
6. Click  to return [*.ret*].

Relaxation Window

Switches the 1D data window to relaxation analysis mode (see Fig. 14.10)

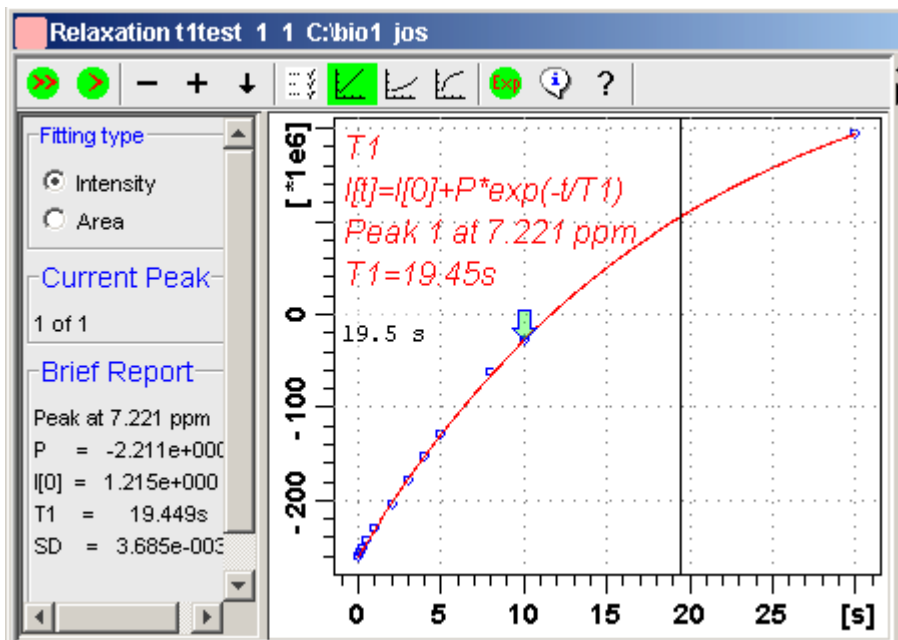



Figure 14.10

and performs a default fitting. By default, this is one-component, T1-intensity fitting (Function type *uxnmrt1*) for peak 1. If the dataset was already fitted, the previous type of fitting is performed. The fitting curve is displayed in the data section and a *Brief Report* is shown in the parameter section. If this default fitting is appropriate, you can view, interpret and print the results as described below. If not, you can perform the desired fitting as described below.

Perform Fitting and Calculate the Relaxation Time

Depending on the experiment, you can perform the appropriate fitting as follows:

1. Select a **Fitting type**: *Intensity* or *Area*. Either every point reflects the intensity of the biggest peak in the defined integral range or the integral itself. Both of them can be used but, depending on the experiment, one of them usually give a better fitting curve.
2. Click the  button to open the parameter dialog box. Select a **Function Type** and set the required parameters (see below). Click **OK**.

3. Perform fitting and calculate the relaxation time:



Fit the relaxation curve for the current peak.



Fit the relaxation curve for all peaks.



View and interpret the results as described below.

Function Types and Parameters

The TOPSPIN relaxation routine offers functions for various relaxation experiments with up to 6 components:



uxnmrt1 for one-component T1 experiments. Set the parameter *List file name* to the list type used during the acquisition. The T1 fitting function is defined by the function:

$$I(t) = I(0) + P \times \exp\left(\frac{t}{T1}\right)$$

where *I* is Intensity or Area according to the Fitting Type. The best fit is calculated by varying *I*(0), *P* and *T1* in an iterative process according to the Levenberg-Marquardt algorithm. Clicking  and  executes the commands **ct1** (current peak) and **dat1** (all peaks), respectively.

uxnmrt2 for one-component T2 experiments. Set the parameter *List file name* to the list type used during the acquisition. A T2 fitting function is defined by the function:



$$I(t) = P \times \exp\left(\frac{t}{T2}\right)$$

where *I* is Intensity or Area according to the Fitting Type. The best fit is calculated by varying *P* and *T2* in an iterative process according to the Levenberg-Marquardt algorithm. Clicking  and  executes the commands **ct2** (current peak) and **dat2** (all peaks), respectively.

invrec, satrec, cpt1rho, expdec, gaussdec, lorgauss linear, varbigdel, varlidel, vargrad, vardamp: these functions can be used for various experiments with up to 6 components, except for **cpt1rho** and **lorgauss** which allow only 4 and 3 components, respectively. They all use the simplex algorithm and require some parameters to be set:

- Enter the **Number of components**

- Click the **Setup** button to set the *Iteration Control parameters*. For each component, the initial guess (G) and step rate (S) can be set. The initial guess for I[0] must be selected such that the sum of all components does not exceed 1. If there is only one component, I[0] is usually set to 1. The step rate is usually set to about one tenth of the initial guess. If the step rate of a variable is set to zero, then this variable is not changed during the iterations.








Clicking  and  executes the commands *simfit* (current peak) and *simfit all* (all peaks), respectively.

View the Fitting Results

When the fitting procedure has finished, the fitting curve is displayed in the data section and a *Brief Report* appears in the parameter section (see Fig. 14.10). The latter consists of:

- the calculated relaxation value
- the fitted parameters
- the standard deviation SD

For further examination of the result, click one of the following buttons:

-  Show the fitting result of the previous peak/area.
-  Show the fitting result of the next peak/area.
-  Switch x-axis to linear scaling.
-  Switch x-axis to logarithmic scaling.
-  Switch x-axis to quadratic scaling.
-  Export integrals to dataset ~TEMP and exit.
-  Show an extended report, including the fitted intensity or area distribution. This consists of the same information as the brief report plus a table with the intensity or area distribution. Example:

```
Dataset : C:/bio/data/guest/nmr/t1test/1/pdata/1
INTENSITY fit :
```



```

I [t]=I [0]+P*exp (-t/T1)
12 points for Peak 1,  Cursor Point =  7.221 ppm
Results      Comp. 1

I [0]  =   1.215e+000
P      =   -2.211e+000
T1     =    19.449s
SD     =    3.685e-003

      tau      ppm      integral      intensity
30.000s    7.221  2.5811e+009  1.9737e+008
10.000s    7.221 -3.2898e+008 -2.9056e+007
8.000s     7.221 -7.8525e+008 -6.4616e+007
5.000s     7.221 -1.6289e+009 -1.3101e+008
...

```

Print, Export or Copy the Fitting Results

To print the fitting curve:

☞ Click **File** → **Print**

To export the fitting curve as a graphics file:

☞ Click **File** → **Export**

To copy the fitting curve to the Windows Clipboard:

☞ Click **Edit** → **Copy**

14.5 Multiplet Analysis

TOPSPIN offers a multiplet analysis package. To start this:

☞ Click **Analysis** → **Multiplet Analysis** [*managuide*].

This will open the Multiplet Analysis Guide. This will guide you step by step through the multiplet analysis process.

Alternatively, you can enter the command **mana** which directly switches to Multiplet Analysis mode. The data window Tab bar will change to a multiplet analysis

toolbar. Fig. 14.11 shows a region of an ethanol spectrum.

Multiplet analysis can be performed in three different ways:

- define multiplet by region
- define multiplet manually
- define multiplet by free grid

For multi-level multiplets, the following methods are available:

- couple existing multiplets into a multi-level multiplet
- define multi-level multiplet by coupled grid

Multiplet analysis uses a maximum intensity search within a user-defined capture range (see Fig. 14.18). If peak picking has been performed (required for defining a multiplet by region), the found peaks within the capture range are used. However, only peaks with the defined minimum intensity are used (see Fig. 14.18).

The three methods are described here for the ethanol CH₃, OH and CH₂ group, respectively. You can use one of these methods, depending on the type of spectrum and your own preference. In this example, the chosen methods for each group is arbitrary.

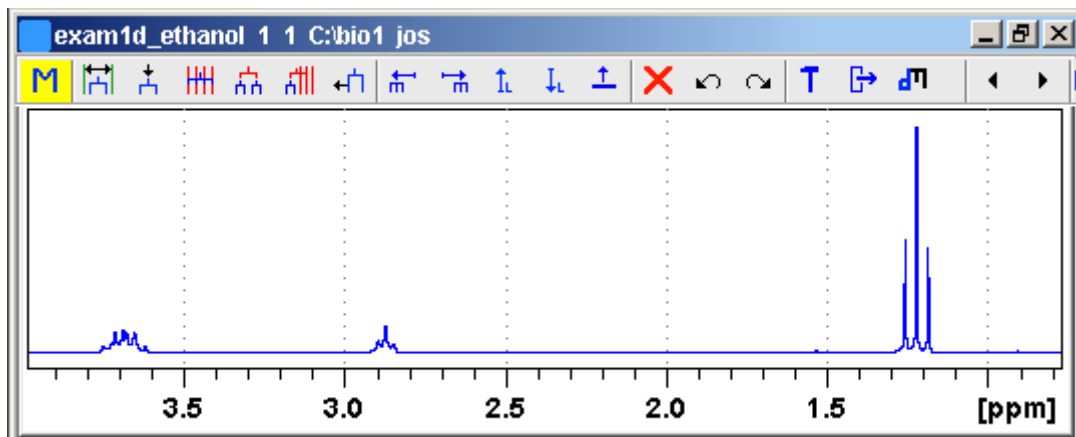



Figure 14.11

14.5.1 Define multiplet by region

In this mode, you simply define the region around the multiplet. The multiplet is

automatically defined from the peaks within the region. At least two peaks must exist in the region to be defined. To define the multiplet:

1. Click the  button (it turns green).
2. Click-hold the left mouse button on one side of the multiplet region, drag the mouse and release it at the other side of the region.

The multiplet will be displayed.

3. Click the  button to leave region mode.

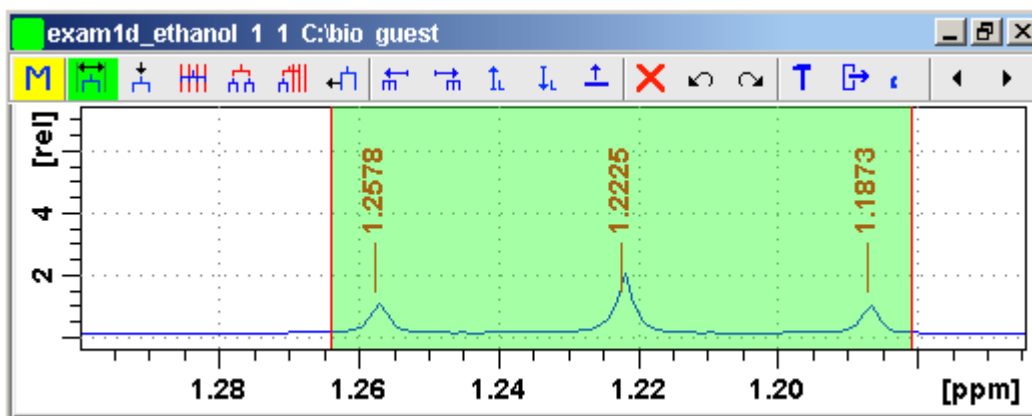


Figure 14.12

In Fig. 14.13, the triplet of the ethanol CH₃-group is being defined. The marked area shows the defined region.

14.5.2 Define multiplet manually.

In this mode, you can define a multiplet by assigning individual peaks. No prior

peak picking is required. If, however, peaks have been picked, they are used.

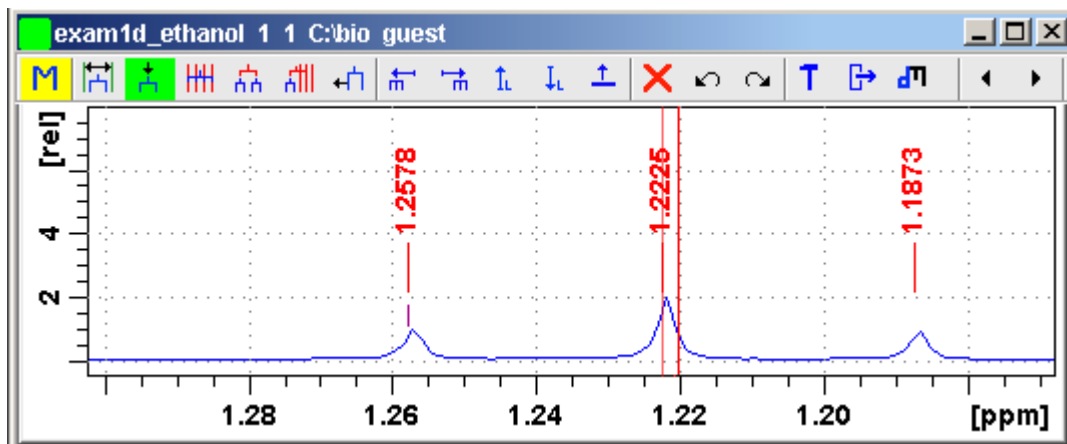



Figure 14.13

To define the multiplet:

1. Zoom in on the region around the desired multiplet.
2. Click the  button (it turns green).
3. Left-click the peaks to be defined.
(Note that left-clicking a peak again will undefine it).
4. Right-click in the data window and select *Define Multiplet* in the popup menu.

The multiplet will be displayed.

5. Click the  button to leave this mode.



In Fig. 14.13, the left peak is already assigned and the cursor line is within the capture range of the central peak as indicated by the faint line on the peak maximum.

Manual analysis can also be started by right-clicking in the data field and choosing *Define Multiplet Manually*.

14.5.3 Free grid analysis

In this mode, you can define a multiplet by assigning one peak manually and all other peaks by free-grid analysis. The grid consists of a predefined number of distance lines.

To define the multiplet:

1. Zoom in on the region around the desired multiplet.
2. Click the  button (it turns green).
3. Right-click in the data window and, if necessary, set the number of distance lines.
4. Left-click the central peak to be defined. For a multiplet with an even number of peaks, left-click one of the central peaks. The central peak will be marked. Note that clicking a marked peak again will unmark it.
5. Move the mouse to the left or right until the grid lines coincide with the remaining multiplet peaks and click the left mouse button. The multiplet will be displayed.
6. Click the  button to leave this mode.

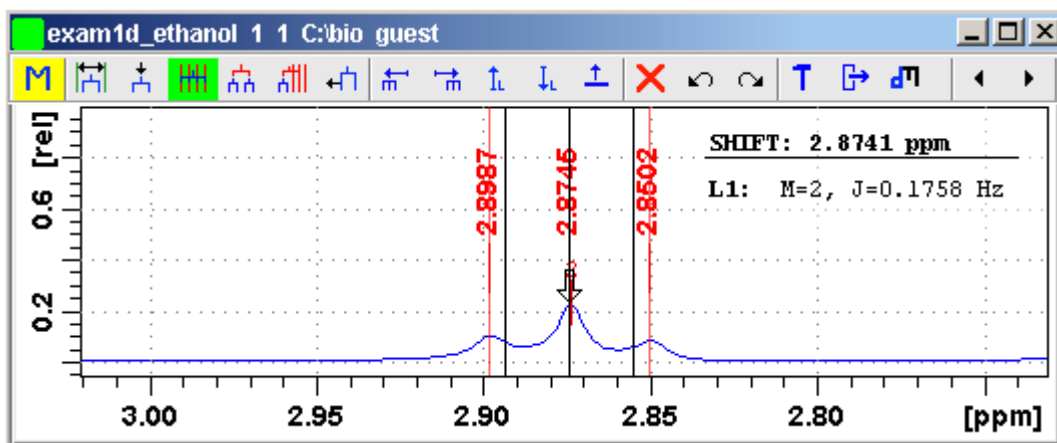




Figure 14.14

In Fig. 14.14, the triplet of the ethanol OH-group is being defined. The central peak is defined and the black grid lines are in the capture range of the other peaks as shown by the faint lines on the peak maxima.

14.5.4 Couple existing Multiplets into a Multi-level Multiplet


In this mode, you can defined a multi-level multiplet by coupling already defined multiplets. To do that:

1. Click the  button (it turns green).
2. Select all multiplets to be coupled by left-clicking them.
3. Move the mouse to the left or right until the grid lines coincide with the multiplet peaks and click the left mouse button.
4. Right-click in the data window and select *Define Multiplet* in the popup menu.
The multi-level multiplet will be displayed.
5. Click the  button to leave this mode.

14.5.5 Define Multi-level Multiplet by Coupled Grid

In this mode, you can define a multi-level multiplet by defining part of the multiplet with one of the methods above and using coupled grid analysis to define the complete multi-level multiplet.

To define a multi-level multiplet:

1. Zoom in on the region around the desired multi-level multiplet.
2. Define the leftmost of rightmost multiplet using one of the methods described above.
3. Click the  button (button turns green).
4. Right-click in the data window and, if necessary, set the multiplicity to the number of multiplets in the multi-level multiplet.
5. Move the mouse to the left or right until the grid lines coincide with the multiplet peaks and click the left mouse button.
The multiplet is defined now.

6. Click the  button to leave this mode.

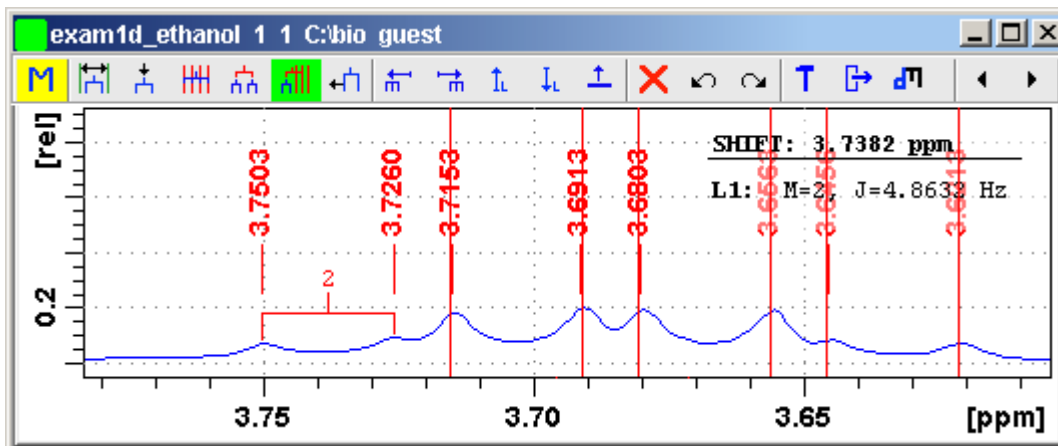



Figure 14.15


In Fig. 14.15, the multi-level multiplet of the ethanol CH₂-group is being defined. The left-most doublet is already defined. The (red) lines of the coupled grid are positioned on the remaining six peaks of the multiplet.

14.5.6 Handling Defined Multiplets

Selecting a Multiplet


You can select a particular multiplet simply by clicking it in the data field. The currently selected multiplet is displayed in the color of the second spectrum (default red). Alternatively, you can select a multiplet from the toolbar as follows:


 Select the previous multiplet.

 Select the next multiplet.

Selecting a Level in a Multi-Level Multiplet


Multi-level multiplets can be assigned for a group which couples with multiple other groups, for example the ethanol CH₂ group. Each level can be selected and designated.

 Select the next level (up) of a multi-level multiplet.

 Select the previous level (down) of a multi-level multiplet.

Designating a Level in Multi-Level Multiplets

A level of a multi-level multiplet can be designated to connect with another multiplet. To do that:


 Right-click in the multiplet and choose *Designate Multiplet*

The designated level is displayed in the color of the fourth spectrum (default purple). Note that the selected level can be different from the designated level.

14.5.7 Define Multiplet Identifiers

To define multiplet identifiers:

1. Left-click the multiplet.
2. Right-click the multiple and select *Define Multiplet Identifier* in the popup menu.
3. Specify the multiplet identifier in the appearing dialog and click **OK**

Alternatively, multiplet identifiers can be defined by clicking the  button and double-clicking the respective multiplet lines in the appearing Report dialog. This will open the Identifier dialog (see Fig. 14.16). Just fill out the field **Identifier** and click **OK**.

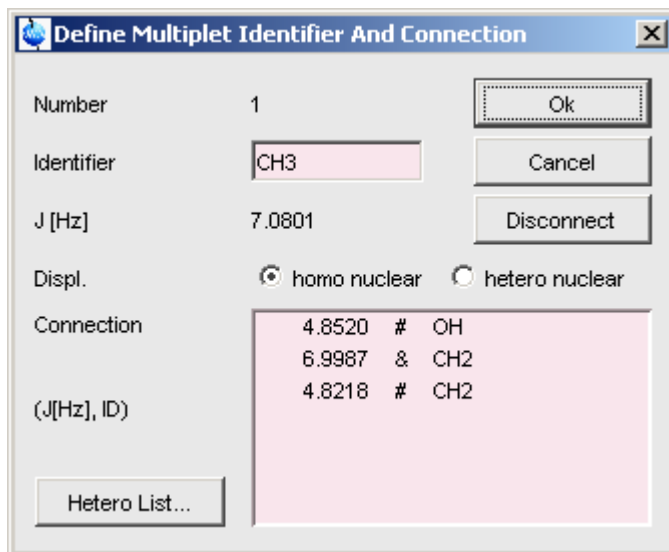


Figure 14.16


Note that the dialog also offers a button to **Disconnect** the current multiplet and a field *Connection* that shows all possible connections using the following flags:

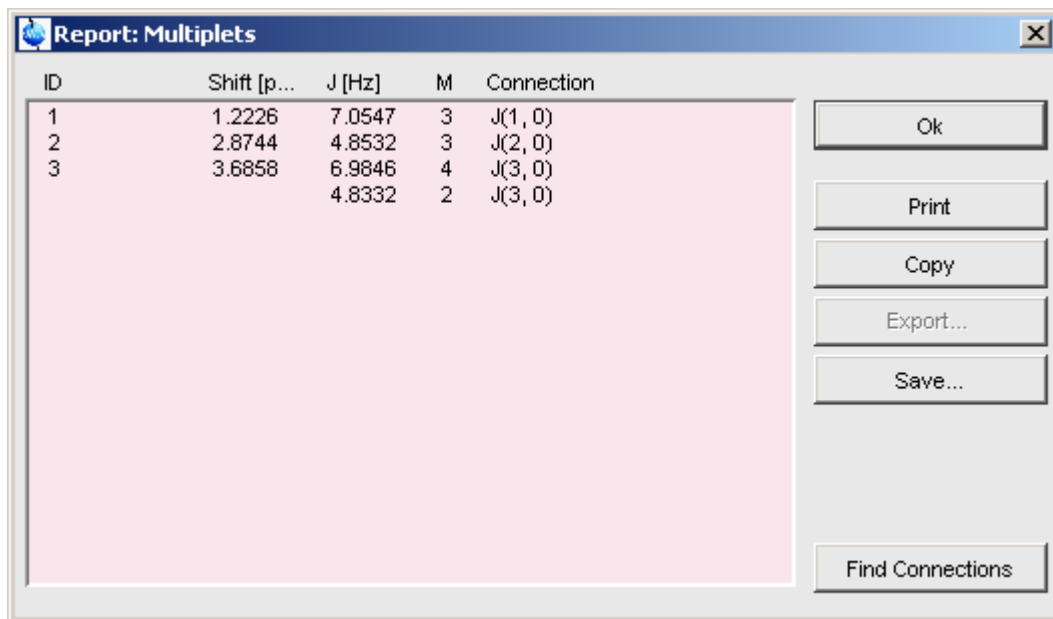
- & : the current connection
- # : a non-existing connection
- ! : a different existing connection

14.5.8 Define Multiplet Connections

Once the multiplets of a spectrum are defined, you can define the connections between them.

To define all connections:

1. Click the button  to see the Report dialog (see Fig. 14.17).



ID	Shift [p...]	J [Hz]	M	Connection
1	1.2226	7.0547	3	J(1, 0)
2	2.8744	4.8532	3	J(2, 0)
3	3.6858	6.9846	4	J(3, 0)
		4.8332	2	J(3, 0)

Figure 14.17



- The multiplets appear in the order in which they have been defined.



Newly defined multiplets appear with numbers as ID's and no multiplet connections (x,0) defined.

- Click the button **Find Connections** to make the multiplet connections:
 - Set the maximum difference between related couplings or accept the default.
 - Set the lower limit for couplings or accept the default.
 - Check the box **Change already defined Connections** if applicable.
 - Click **OK** to define the connections.

Note that the Report Dialog also offer buttons for Printing the multiplet information, copying it to the clipboard and saving it to a text file.

To define individual connections:

- Select the first multiplet. In case of a multi-level multiplet, click the  or  button to select the required level.

2. Right-click and select **Designate Multiplet** from popup menu.
3. Select the second multiplet. In case of a multi-level multiplet, click the  or  button to select the required level.
4. Right-click and choose the **Connect Multiplets** from popup menu.

To define individual connections:







To disconnect multiplets:

1. Select on of the multiplets to be disconnected.
2. Right-click and select **Disconnect Multiplets** from popup menu. Shifting a Multiplet Line


Alternatively, you can open disconnect a multiplet from the Identifier dialog (see Fig. 14.16)

14.5.9 Further Multiplet Functions

The Multiplet Analysis window offers several other function such as:

-  Shift multiplet line. Click this button and move the vertical cursor line into the capture area of the desired line which is then marked by a faint line. The left-click-hold and move the mouse to the desired position.
-  Shift multiplet vertically. Click this button and move the mouse to put the horizontal line cursor above or below the multiplet tree. Then left-click to shift the multiplet to that position.
-  Remove the currently selected multiplet. Clicking this button several times allows you to remove all multiplets.
-  Show Daisy multiplets.
-  Undo the last multiplet action.
-  Redo the last multiplet action.

14.5.10 How to Set Multiplet Options

The  button opens a dialog box where you can set various multiplet options as

shown in Fig. 14.18.

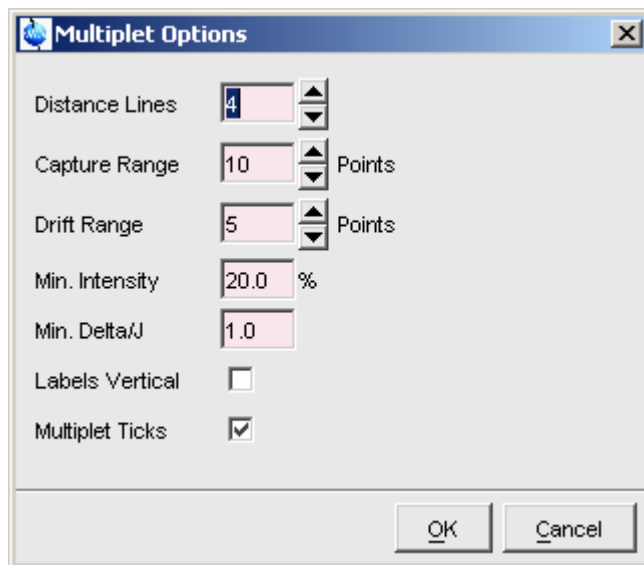


Figure 14.18

- Distance lines: the default number of distance lines in the multiplet dialogs.
- Capture range: the search range for maximum intensity of peak position in manual mode.
- Drift range: the maximum difference in data points between line distances within one multiplet
- Min. Intensity: The minimum intensity of a peak compared to the reference peak to be accepted as a multiple line.
- Min. Delta/J: the minimum ratio of the difference in chemical shift of the coupling groups and the coupling constant. Below this value, the coupling constant in the Report box is indicated with a question mark to suggest possible second order effect.
- Labels Vertical: displays multiplet labels 90° rotated.

All toolbar functions are also available from a popup menu which appears when you right-click in the data window (see Fig. 14.19).

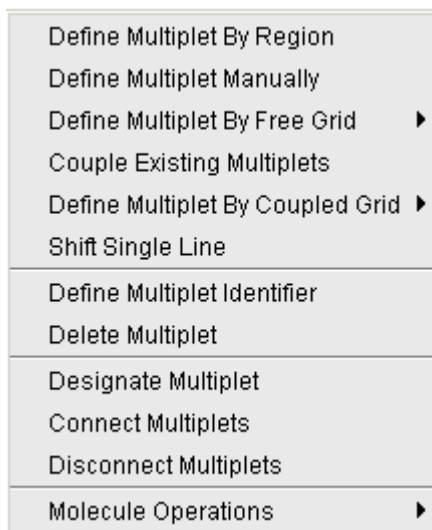





Figure 14.19

Save and close

When you have finished multiplet analysis you can save your work and exit from multiple mode as follows:

-  Save multiplet analysis.
-  Save multiplet analysis and quit.
-  Quit multiple analysis mode.

The multiplet analysis result is saved in the file `mult.txt` in the *procno* data directory.

Chapter 15

Acquisition

This chapter describes TOPSPIN acquisition as far as the interface is concerned. Individual acquisition commands are described in the Acquisition Reference manual.

15.1 Acquisition Guide

If you are a new or occasional user, we recommend you to acquire your data with the TOPSPIN Acquisition Guide. This will guide you through the typical sequence of acquisition steps. To start the Acquisition Guide, click *Spectrometer* → *Acquisition Guide*.

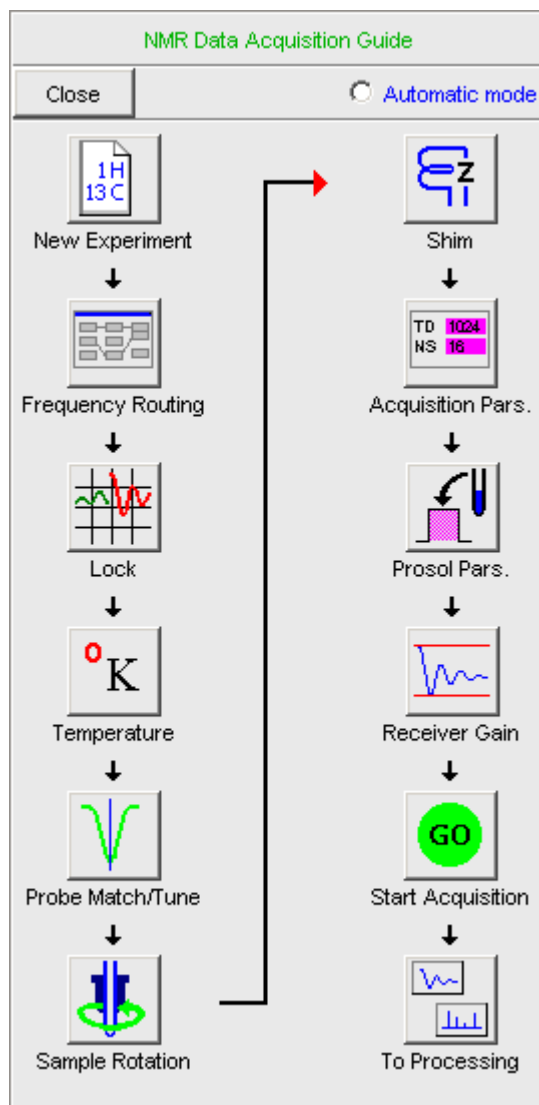


Figure 15.1

In *Automatic mode*, the Acquisition Guide will simply execute each command when you click the respective button. This requires the acquisition parameters to be set correctly. In interactive mode (*Automatic mode* unchecked), the Acquisition Guide will, at each step, open a dialog box offering you the available options and

required parameters. Note that the last button *To processing* will open the processing equivalent of the Acquisition Guide, the Processing Guide.

15.2 Acquisition Toolbar

Acquisition can be prepared, started and controlled from *Spectrometer* menu. Clicking this menu opens the pulldown menu shown in Fig. 15.2 :

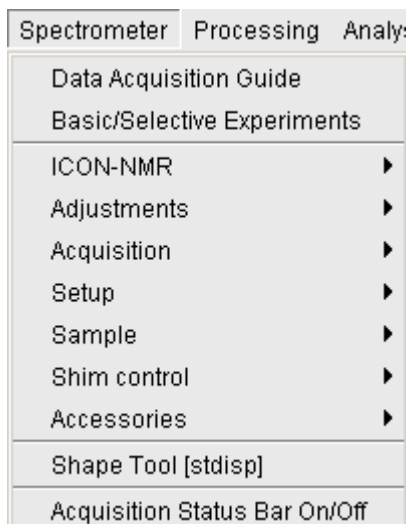


Figure 15.2

Here you find several categories of acquisition related commands. Each entry give access to a submenu with various commands. Fig. 15.3, for example, shows the *Setup* submenu.

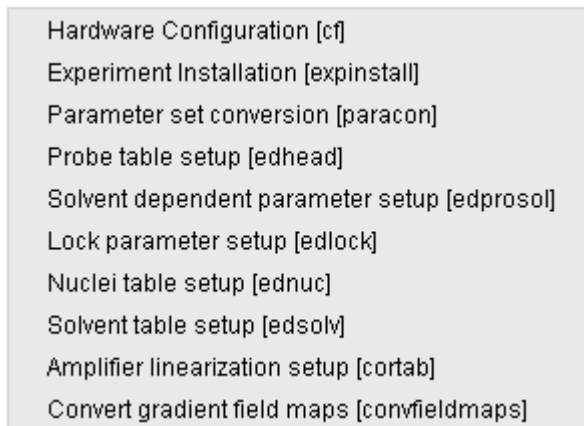





Figure 15.3

For most entries, the command line command, for example *expinstall*, is specified in square brackets. Furthermore, Topspin can be configured (command *set*) such that right-clicking any menu entry will display the corresponding command line command.

For convenience, common acquisition commands can also be started from the TOP-SPIN toolbar. The right part of the upper toolbar shows the following buttons:



- ▶ Start the acquisition. Acquires NS scans in the current dataset, overwriting possibly existing data. Equivalent to the command *zg*.
- Halt the acquisition. This stops the acquisition after the current scan has finished. All data acquired so far are saved. Equivalent to the command *halt*.
- STOP Stop the acquisition. This stops the acquisition immediately. Data acquired after the last disk write (if any) are lost. Equivalent to the command *stop*.
- 📡 Open the online FID display window. Shows the currently acquired FID. Only works during acquisition. Equivalent to the command *acqu*.
- 🔒 Open the Lock display window. Shows the lock signal. Equivalent to the command *lockdisp*.
- 🕒 Calculate the experiment time. Shows the total experiment time and file size of the raw data. Equivalent to the command *expt*.

-  Set SFO1, O1, O2 and O3 interactively. Puts a red cursor line in the data window. A left-click at the desired frequency opens a dialog box where you can set O1, O2 and/or O3.
-  Set the sweep width to the current region and the spectrometer frequency to the center of the current region. Update the parameters SW and SFO1, respectively.
-  Setup a frequency list interactively.

15.3 Acquisition Status Bar

The acquisition can be followed and controlled from the acquisition status bar. Before you use the acquisition status bar, it must be configured from the User Preferences window. To do that:

1. Click *Options* → *Preferences* [*set*].
2. Click *Acquisition status bar* in the left part of the *User preferences* box. This will show the status bar items (see Fig. 15.4).

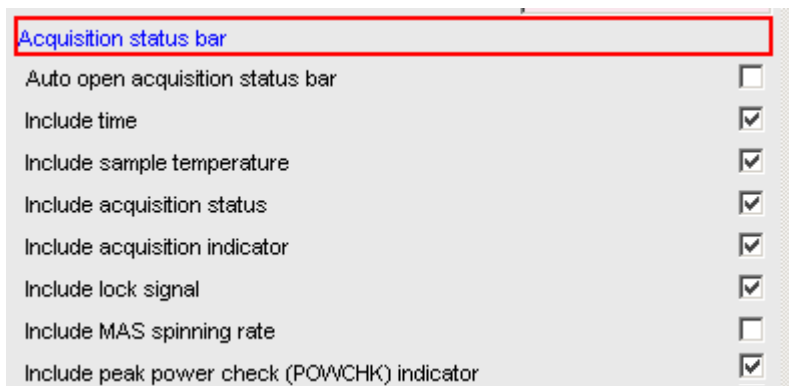


Figure 15.4

3. Check the desired entries.

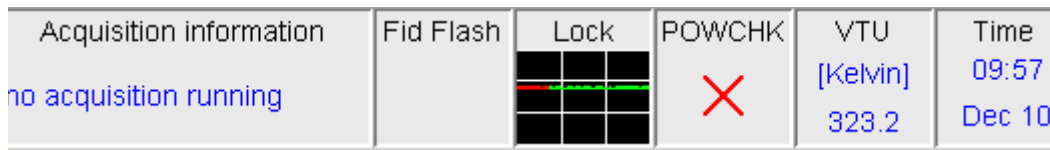
To switch the acquisition status bar on:

-  Click *Spectrometer* → *Acquisition Status Bar On/Off*

or

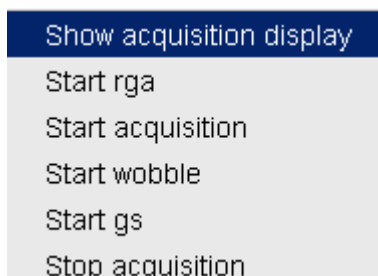
- Right-click in the status line at the bottom of the TOPSPIN window and choose the popup menu *Acquisition Status Bar On/Off*

With the entries selected above, the right part of the status bar will look like this.



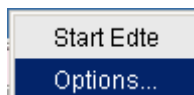
The acquisition status bar not only displays information, it also allows you to perform various actions, e.g.:

- Double-click the **Time** field to view detailed time and date information.
- Double-click the **Lock** field to open the lock display.
- Double-click the **Fid Flash** field to open the wobble display.
- Right-click the **Fid Flash** field to switch on/off FID flashing.
- Right-click the **Acquisition information** field to open the following popup menu:



from this menu, you can start various acquisition commands.

- Right-click the **VTU** field to open the following popup menu:



Clicking *Options* will open the following dialog box:

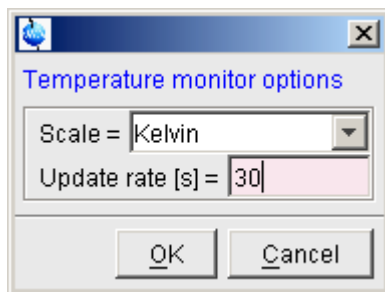


Figure 15.5

15.4 Tuning and Matching the Probehead

Tuning and matching of conventional probeheads, (non ATM), is performed with the wobble procedure. To start this:

1. Enter **wobb** on the command line.

This opens the wobble window (see Fig. 15.6).

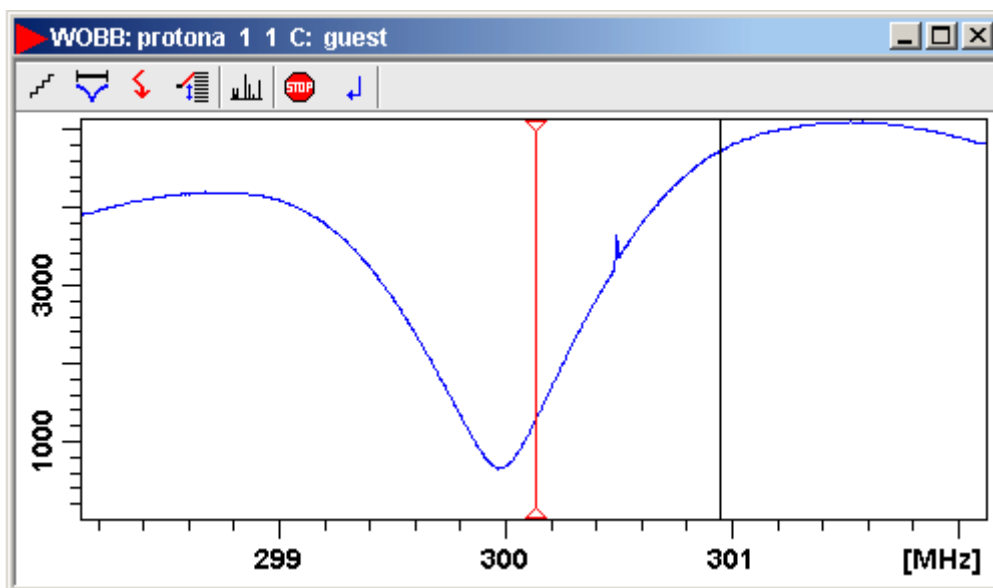








Figure 15.6


The buttons of the wobble toolbar have the following functions:

-  Change the number of wobble steps [*wbst*].
-  Change the wobble sweep width [*wbsw*].
-  Change the wobble frequency.
-  Switch to the next channel/nucleus (if available).
-  Stop the wobble procedure.
-  Stop the wobble procedure and close the wobble window.

2. Turn the tuning and matching knobs on the probehead until the wobble curve is exactly in the middle and its minimum reaches the zero line.

Automatic tuning and matching of ATM probeheads can be performed with the command *atma*.

15.5 Locking

The lock display can be opened by clicking the  button in the toolbar or entering `lockdisp` on the command line. The lock display window will appear (see Fig. 15.7).

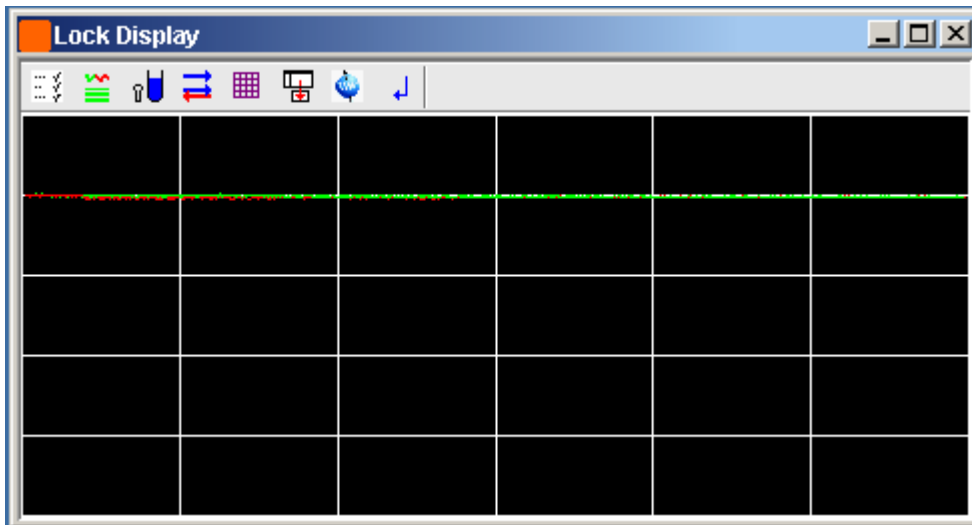










Figure 15.7

Here, you can view the lock signal, either during the lock-in procedure or, as shown above, after lock-in has been successful. At the top of the lock window the following buttons are available:

-  Open the user preferences window [`set`].
-  Toggle lock monitor mode.
-  Lock the magnetic field [`lock`].
-  Toggle lock display mode between *single* and *dual* color. Colors can be set in the User preferences (command `set`).
-  Switch grid mode between *both*, *vertical*, *horizontal* and *off*.
-  Make the lock display external.
-  Put focus into TOPSPIN window.

 Close the lock display window.

Note that an external lock display window is independent from the TOPSPIN window. You can for example, minimize TOPSPIN while keeping the lock display open.

The lock display can also be opened by double-clicking the lock field in the acquisition status bar (see par. 15.3)

The lock process can be started by entering **lock** on the command line. This command is described in the Acquisition Reference manual.

15.6 BSMS Control Panel

The BSMS control panel allows shimming, locking, sample handling and helium level measurement. To open this panel:

 Enter ***bsmsdisp*** on the command line

The BSMS Control Suite window will appear (see Fig. 15.8)

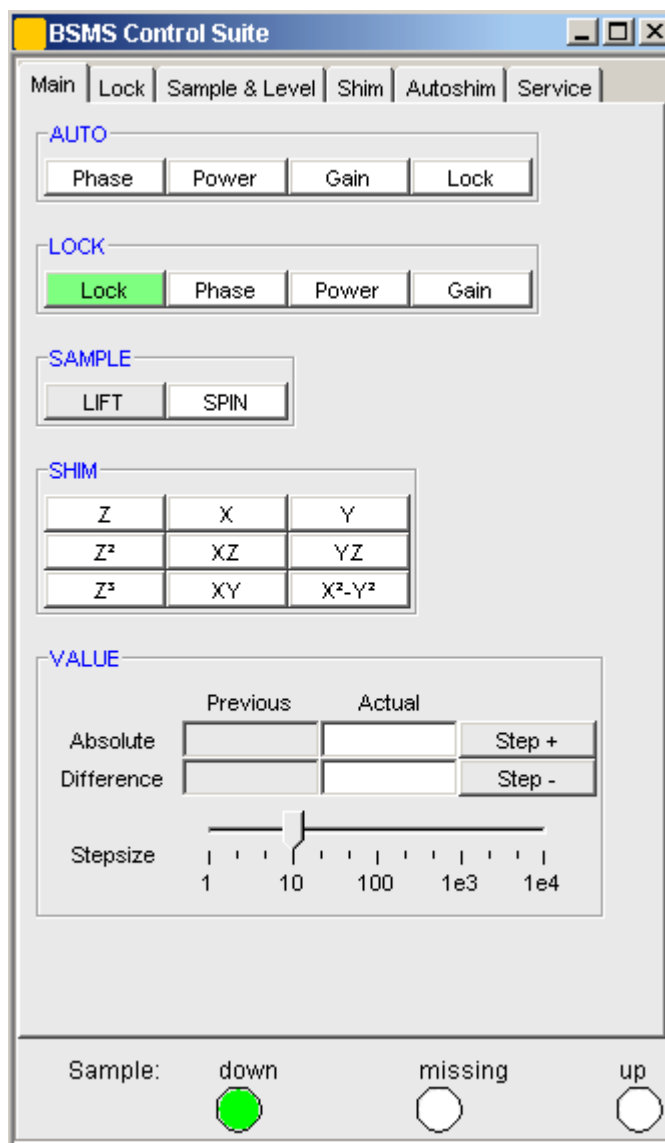


Figure 15.8

The individual functions BSMS control panel are described in detail in the Acquisition Reference manual.

15.7 Interactive Parameter Adjustment (GS)

Several parameters can be adjusted interactively, while observing the acquired FID. To start this:

Enter *gs* on the command line.

A split window will appear showing:

- the FID display (see Fig. 15.9)
- the GS parameter adjustment dialog (see Fig. 15.10)

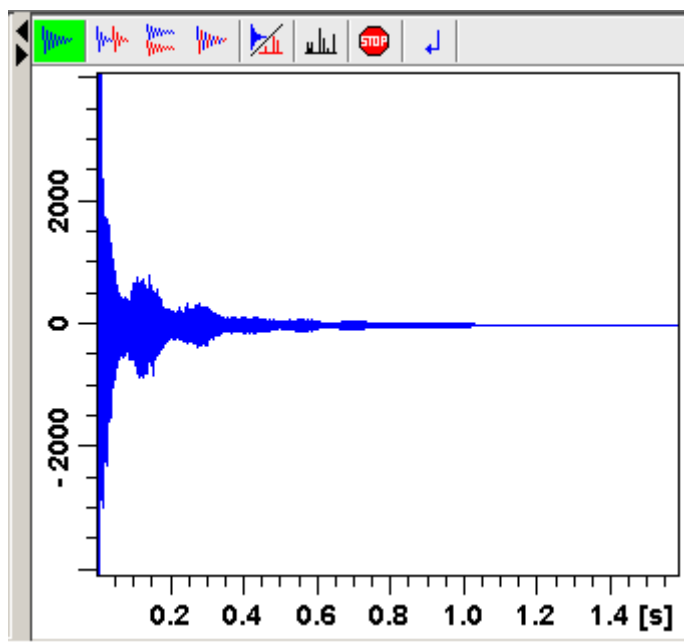


Figure 15.9

The buttons of the FID display are the same as for the acquisition command *zg* (see paragraph Fig. 15.8).

The GS parameter adjustment dialog offers tabs at the top of the window to select power, frequency, delay etc. The selected parameter is shown in the middle of the window. The slider at the right of the window allows you to change the selected parameter. The current value can be viewed and modified in the field below the slider. The sensitivity of the slider can be set in the field *Sensitivity* above the

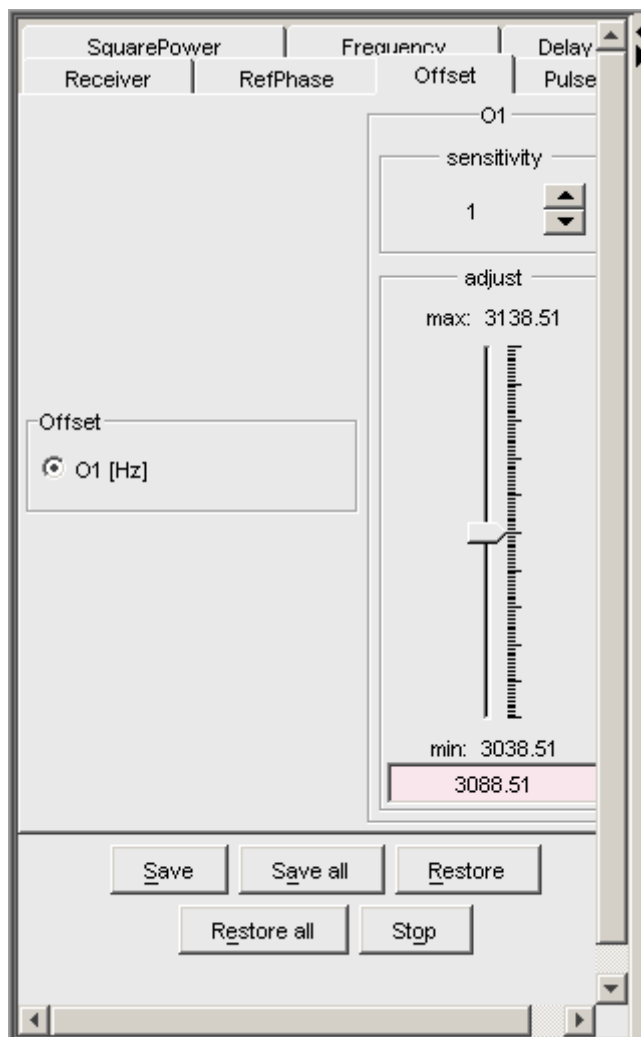


Figure 15.10

slider. The effect of the change can be viewed in the FID display, the right part of the window. This can be manipulated with the FID display buttons as described in chapter 15.8.

At the bottom of the window you find the following buttons:

- **Save** : Save the value of the current parameter.

- **Save all** : Save the values of all changed parameters.
- **Restore** : Restore the value of the current parameter.
- **Restore all** : Restore the value of all changed parameters.
- **Stop** : Stop the acquisition and quit the GS window.

15.8 Running an Acquisition

A typical acquisition is performed as follows:

1. Create a new dataset.
 - a) Click **File** → **New** [**new, Ctrl+n**].

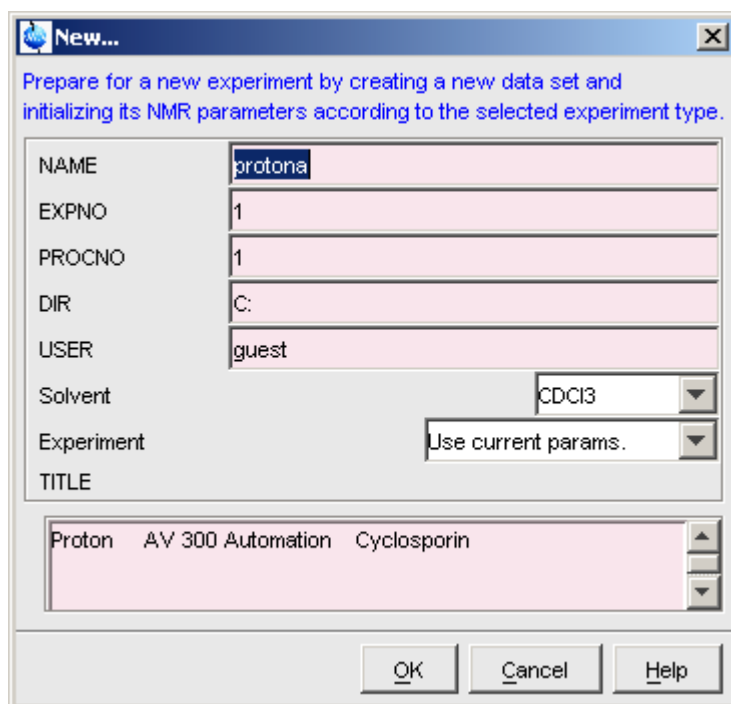


Figure 15.11

- b) Specify the datapath variables *name*, *expno*, *procno*, *dir* and *user*, select the desired *Solvent* and *Experiment*, enter the *Title* and click **OK**.

The dataset will appear in the data field with no raw and no processed data available.

2. Click the *AcquPars* tab to display the acquisition parameters.

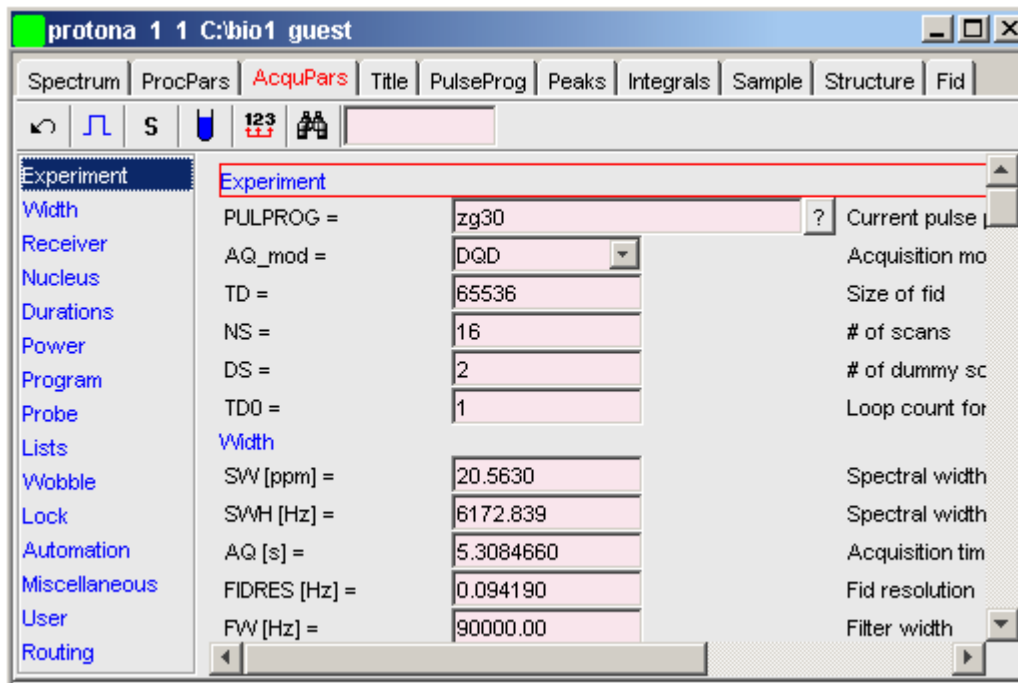




Figure 15.12

- a) Optionally: click  to show the pulse program parameters only.
- b) Click the  button to read the prosol parameters

or

Set the relevant parameters manually.

As an alternative to step 2, you can set the acquisition parameters interactively in the GS window (see par. 15.7).

3. To start the acquisition:
 - Click  in the upper toolbar or enter **zg** on the command line.

The FID display window will appear:

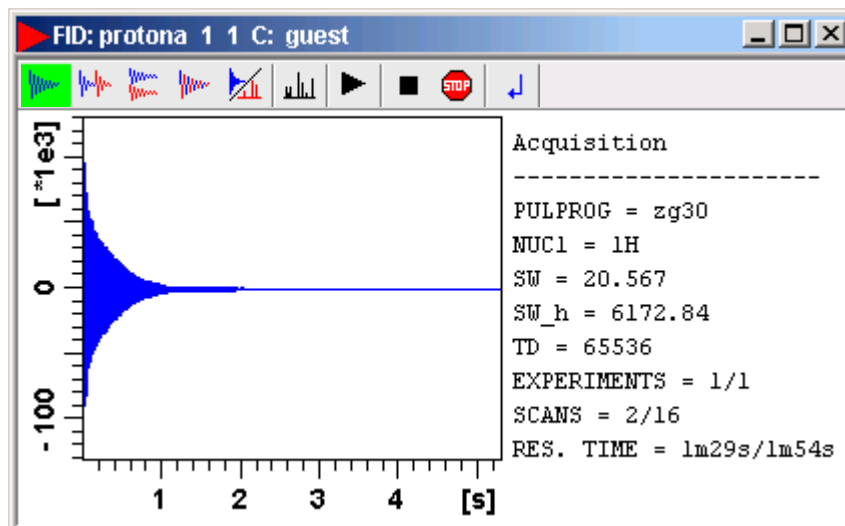















Figure 15.13


The buttons in the toolbar have the following functions:

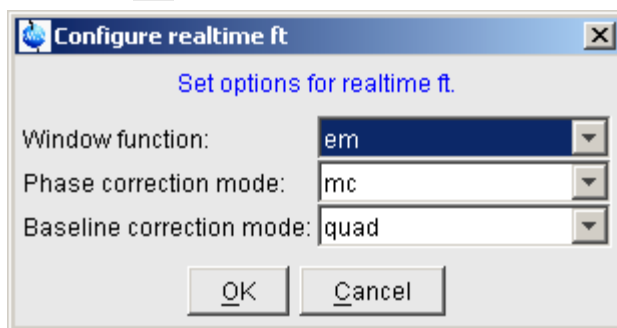
-  Show FID in shuffled mode.
-  Show FID in unshuffled mode, horizontally arranged.
-  Show FID in unshuffled mode, vertically arranged.
-  Show FID in unshuffled mode, interleaved.
-  Switch between FID and spectrum.
-  Switch to spectrum data window.
-  Stop the acquisition [*stop*].
-  Halt the acquisition [*halt*].
-  Close the FID window.

Clicking the  button to switch to real time FT, turns the button green and

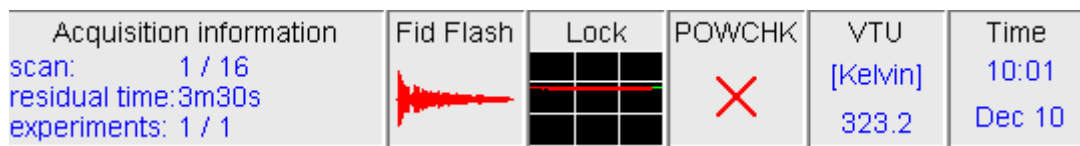
opens two extra buttons:

-  Switch between FID and spectrum.
-  Real time FT settings.
-  Toggle calculation of peak with at 50%, 5.5% and 1.1% height
(Shown as status parameters).


Clicking the  button opens the following dialog window:



The acquisition information will appear in the acquisition status bar at the bottom of the TOPSPIN window, for example:



4. When the acquisition has finished:

-  Enter **efp** to process the FID.

The processed spectrum will be displayed in the data window that was opened upon creating the dataset (see Fig. 15.14).

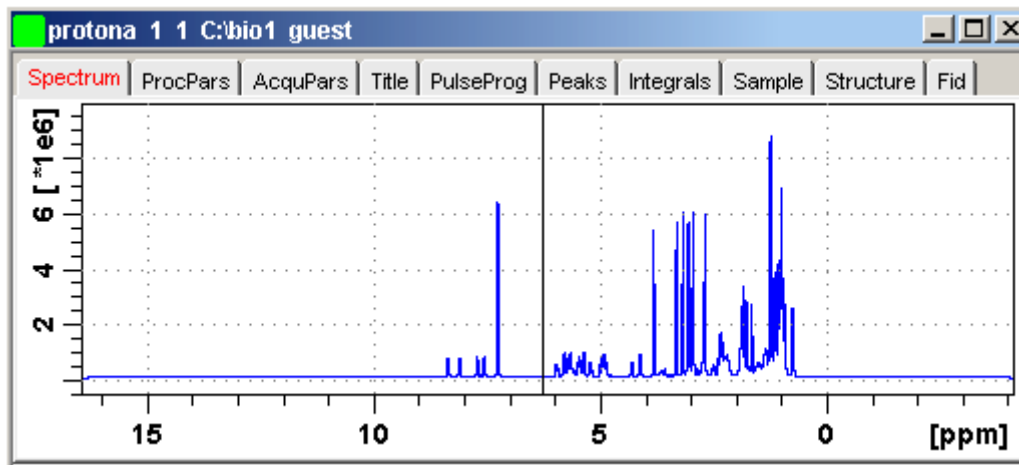


Figure 15.14

From here, the processed data can be analysed, printed and/or archived.

15.9 Shape tool

The Shape tool interface allows you to create/manipulate RF shapes and gradients.

To start the Shape Tool interface:

☞ Click *Spectrometer* → *Shape Tool*

or enter *stdisp* on the command line.

The Shape Tool window will appear (see Fig. 15.15.). This consists of a toolbar, a command line and a split pane with a data section at the right and a parameter section at the left.

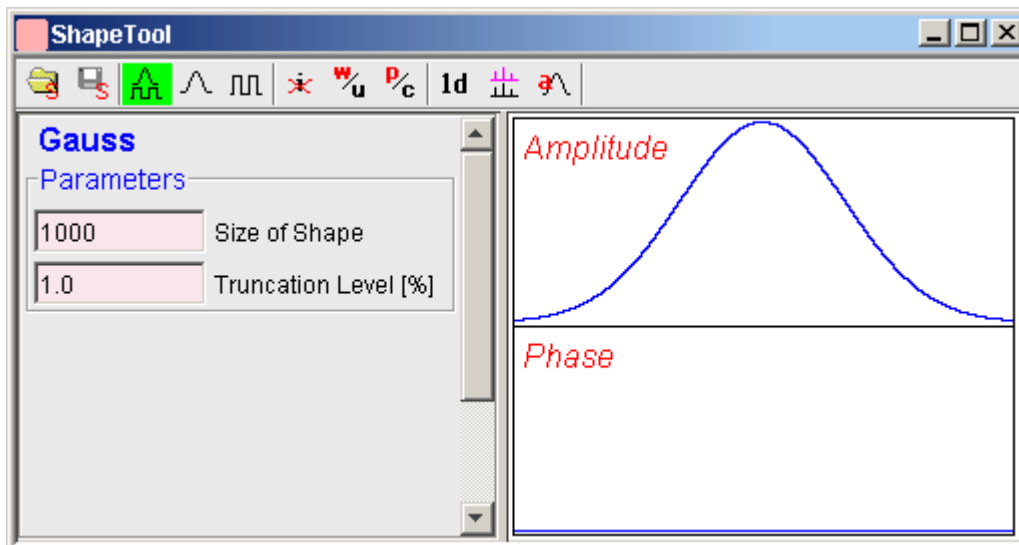


Figure 15.15

By default, a 1000 point Gauss shape is displayed with Truncation level 1.0.

The TOPSPIN menu is changed showing the additional *Shapes* and *Manipulate* menus and the adjusted *File*, *Analysis* and *Options* menus.


File Edit View Shapes Analysis Manipulate Options Window Help

Note that all functions of the interactive Shape Tool can also be performed non-interactively with the TOPSPIN command *st*. This command must be entered with the appropriate arguments on the command line while the associated dataset is displayed and selected. For a description of the *st* command see:

☞ *Help* → *Application manuals* → *Shape Tool*

15.9.1 How to Open a Shape/Gradient







To open an existing shape/gradient, click the button:

 Open a shape or gradient.

select *Shape* or *Gradient*, select a file from the appearing file list and click *OK*. The selected shape/gradient will appear in the data section.


15.9.2 How to Use Shape Tool Display Options

The Shape Tool toolbar offers the following display options:

-  Display amplitude and phase.
-  Display amplitude only.
-  Display phase only.
-  Toggle cursor information on/off. Cursor information consists of the point number and amplitude/phase value.
-  Wrap/unwrap phase (actual phase/phase modulo 360°).
-  Toggle between polar and cartesian coordinates.



15.9.3 How to Save a Shape/Gradient as a 1D dataset

To save the current shape/gradient stored under a procno of the associated dataset, click:

-  Save shape/gradient as a 1D dataset.


15.9.4 How to Superimpose Multiple Shapes/Gradients

Several shapes/gradient can be displayed superimposed using the following buttons:

-  Add the currently displayed shape to multiple display.
-  Switch to multiple display mode. This shows all 'added' shapes/gradients superimposed.

15.9.5 How to Save a Shape/Gradient

To save a shape/gradient, click the button:

-  Save a Shape or Gradient or a fraction of it.

and select *Save Shape*, *Save Gradient* or *Save Fraction*. A dialog window appears

where you can enter a (File)Name, Title, Flip Angle and the Type of Rotation.

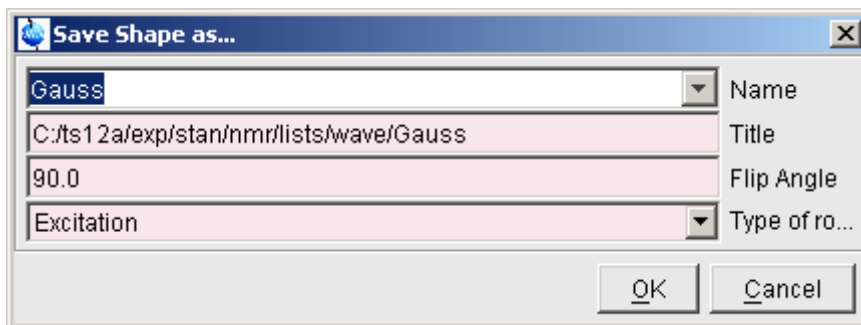
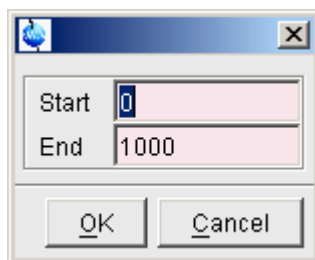


Figure 15.16

If you save a fraction of a shape or gradient, the above dialog is preceded by the following dialog:



Enter the *Start* and *End* point and click **OK**.

15.9.6 How to Generate Shapes/Gradients

The Shape Tool allows you to generate RF and gradients shapes. Various *Basic*, *Classical* or *Adiabatic* shapes can be created from the **Shapes** menu. Additionally *Solids*, *Imaging* and *Decoupling* shapes are available.

How to Generate a Gauss shape


To generate a Gauss shape:

- ☞ Click **Shapes** → **Classical Shapes** → **Gauss**

A Gaussian shaped curve will appear in the data section. The parameter section shows the two parameters that can be set for a Gauss:

Size of Shape: the number of shape data points

Truncation level: the minimum amplitude at the edge of the shape

If you change these parameters, the displayed shape will automatically be updated. To save the shape, click , enter a *Name*, e.g. *mygauss*, *Title*, *Flip Angle* and the *Type of Rotation* (see chapter 15.9.5) and click **OK**.

How to Generate a ShapFour shape

To generate an RF shape that is defined by Fourier coefficients:

☞ Click *Shapes* → *Classical Shapes* → *ShapFour*

The following dialog box will appear:

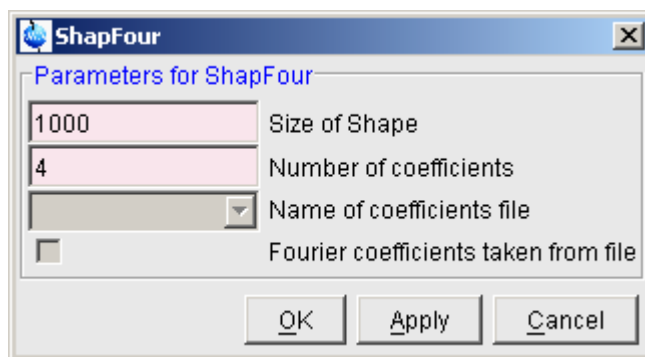


Figure 15.17

Here you can enter the *Size of Shape* and the *Number of coefficients*. Clicking **OK** or **Apply** will open a further dialog box (see Fig. 15.18), where you can enter the required coefficients. A ShapFour shape is defined by two coefficients arrays $a[0,1,\dots]$ and $b[1,\dots]$. Note that the element $a[0]$ is listed separately.

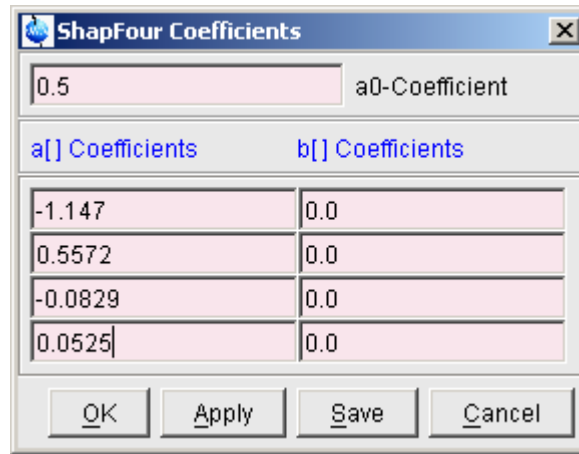


Figure 15.18

Here, you can enter the desired coefficients and then click:

Save to store the coefficients for later usage. You will be prompted for a filename which will be stored as:


`<user-home>/topspin-hostname/shapetool`

Apply to display the shape while keeping the dialog box open for possible further changes.

OK to display the shape and close the dialog box.

With the coefficients above you have created a *Rsnob* shape.

To save it:

Click , enter a *Name*, e.g. *myrsnob*, *Title*, *Flip Angle* and the *Type of Rotation* (see chapter 15.9.5) and click **OK**.

15.9.7 How to Analyse Shapes

The Shape Tool interface offers several functions to analyse shapes. Most of these functions are only meaningful for RF shapes.

To access these functions:

 Click **Analysis**

in the TOPSPIN menu bar. This will open the pull-down menu shown in Fig. 15.19.

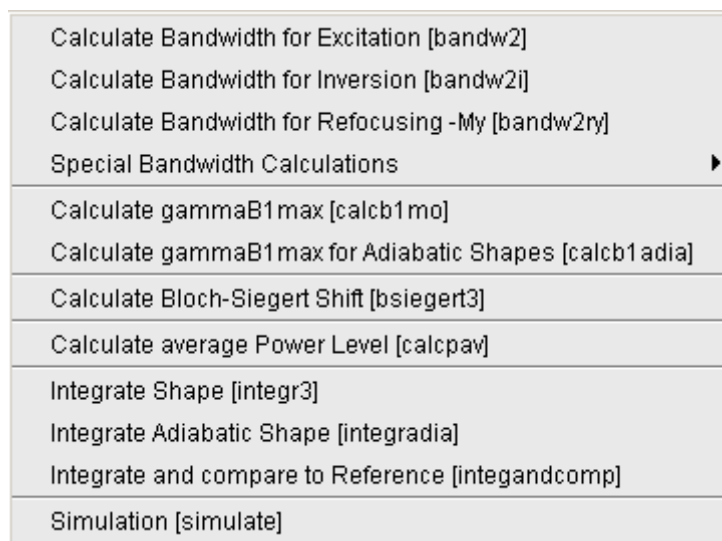


Figure 15.19

From here, you can start the analysis functions discussed below.

How to Calculate the Bandwidth for Excitation

To calculate the excitation band width factor $\Delta\omega*\Delta T$:

☞ click *Analysis* → *Calculate Bandwidth for Excitation* [*bandw2*]

For the excitation band width, the $\sqrt{(M_x^2 + M_y^2)}$ magnetization is used.

Fig. 15.20 shows the resulting parameter section for a Gauss shape.

Gauss

bandw2

90.0 Total rotation [degree]

Results

2.1220 DeltaOmega*DeltaT factor using transv. magn.

Calculator

100.0 DeltaOmega [Hz]

21220.0 DeltaT [usec]

update parameters

Figure 15.20

As you can see, for a 90° Gaussian shape the bandwidth factor is 2.122.

The bandwidth factor is the product of the width of excitation (DeltaOmega) and the pulse length (DeltaT).

This means a pulse length of 21220 μsec gives a band width ($\Delta\omega$) of 100.0 Hz. The bandwidth is the excitation width at the 3 dB point, i.e. the point where the magnetization has dropped to 70.8 %.

When you change one of the parameters in the parameter section, the others are automatically adjusted. Clicking the **Update Parameters** button will store the length of the shaped pulse (DeltaT) to the associated dataset parameter (default P11).

How to Calculate the Bandwidth for Inversion

To calculate the inversion band width factor $\Delta\omega*\Delta T$:

☞ Click *Analysis* → *Calculate Bandwidth for Inversion* [*bandw2i*]

For the inversion band width, the Mz magnetization is used.

How to Calculate Bandwidth for Refocusing

To calculate the refocusing band width factor $\Delta\omega*\Delta T$:

☞ Click *Analysis* → *Calculate Bandwidth for Refocusing* [*bandw2ry*]

For the refocusing band width, the -My magnetization is used.

How to Calculate Special Bandwidths

Further routines for the evaluation of various magnetization components are available under the menu item *Special Bandwidth Calculations*.

How to Calculate the Maximum RF Field Strength for Classical Pulses

To calculate the RF field strength for classical pulses:

☞ Click *Analysis* → *Calculate gammaB1max* [*calcb1mo*]

Fig. 15.21 shows the resulting parameter section for a Gauss shape.

Gauss	
calcb1mo	
13000.0	Length of pulse [usec]
180.0	Total rotation [degree]
0.0	Offset [Hz]
Results	
93.45	GammaB1(max)/2pi at 0 Hz (on res.) [Hz]
2675.2543	Corresp. 90 deg square pulse [usec]

Figure 15.21

The result consists the field strength ($\gamma B_{1\max}/2\pi$) and the corresponding 90° square pulse length. To obtain this result, the integral of the shaped pulse is compared to that of a square pulse of same length, with the latter being normalized to 1.

The maximum power is:

$$\gamma B_{1\max}/2\pi = \gamma B_1/2\pi \text{ (square pulse of same length) / integral ratio}$$

where

$$\gamma B_1/2\pi \text{ (square pulse of same length) } = (1 / \text{pulse length} * (360^\circ / \text{flip angle}))$$

How to Calculate the Maximum RF Field Strength for Adiabatic Pulses

To calculate the RF field strength for adiabatic pulses:

☞ Click *Analysis* → *Calculate gammaB1max for Ad. shapes*
[*calcb1adia*]

The screenshot shows the HypSec software interface with the following sections:

- HypSec** (Title)
- calcb1adia** (Section Header)
- Input:** A text box containing "10610.0" with the label "Length of pulse [usec]" to its right.
- Results** (Section Header)

9.4305E05	Sweep rate on resonance [Hz/sec]
387.42	GammaB1(max)/2pi/sqrt(Q) [Hz]
- Calculator** (Section Header)

5.0	Q
866.29	GammaB1/2pi

Figure 15.22

Fig. 15.22 shows the resulting parameter section for a *HypSec* shape.

The result $\gamma B1(\max)/2\pi / \sqrt{(Q)}$ is calculated from the on-resonance sweep rate.

Here, Q is the quality or adiabaticity factor. After entering the appropriate value for Q the value for $\gamma B1(\max)/2\pi$ is obtained. As a rule of thumb Q is set to 5 for inversion pulses. However, for decoupling pulses, Q should be set between 2 and 3 to allow a lower decoupling power.

How to Calculate the Bloch-Siegert Shift

To calculate the phase difference due to the Bloch-Siegert shift:

☞ Click *Analysis* → *Calculate Block-Siegert Shift* [*bsiebert3*]

Input parameters are the length of the shaped pulse in μs , $\gamma B1(\max)$ in Hz and the offset of a theoretical signal relative to the frequency of the RF-pulse in Hz. The result consists of the phase difference due to Bloch-Siegert shift in Degree and the corresponding frequency shift in Hz.

How to Calculate the Average Power Level

To calculate the average power:

☞ Click *Analysis* → *Calculate average Power Level* [*calcpav*]

The result is the percentage of the average power of a square pulse of the same length.

How to Integrate Classical Shapes

To calculate the power level required for a classical shaped pulse:

☞ Click *Analysis* → *Integrate Shape* [*integr3*]

Gauss	
integr3	
10610.0	Length of pulse [usec]
180.0	Total rotation [degree]
7.5	90 deg hard pulse [usec]
Results	
0.4116	Integ Ratio comp. to square on res.
-7.7110	Corresponding difference [dB]
49.2815	Change of power level [dB]
update parameters	

Figure 15.23

Fig. 15.23 shows the parameter section where you can enter the length of the soft pulse, the flip angle and the length of the 90° hard pulse. Soft pulse and hard pulse length are initialized with the values of the corresponding parameters of the associated dataset. If these values are zero or if no dataset is associated, default values are taken, a 10000 μ s soft pulse and 50 μ s hard pulse.

The result for a Gaussian shape with 1000 points and 1% truncation is shown. The *Integral ratio* and the *Corresponding difference* in dB are calculated from shape amplitude and phase, assuming equal pulse length and flip angle for soft

and hard pulse. The *Change in power level*, however, is calculated from the specified pulse lengths and rotation.

Clicking the **Update Parameters** button saves the length of the shaped pulse and the change in required power level to the corresponding parameters of the associated dataset. Note that the latter is the sum of the shown *Change of power level* and the current hard pulse power level (e.g. PL1).

How to Integrate Adiabatic Shapes

To calculate the power level required for an adiabatic shaped pulse:

☞ Click *Analysis* → **Integrate Adiabatic Shape** [*integradia*]

The change of power level is calculated from the length of the corresponding 90° pulse and the length of the hard pulse.

Clicking the **Update Parameters** button saves the length of the shaped pulse and the required power level to the corresponding parameters of the associated dataset.

How to Start Simulation

To evaluate the shape behaviour, you can start the TOPSPIN NMRSIM routine:

☞ Click *Analysis* → **Simulation** [*simulate*]

This allows you for example, to view the excitation profile, or, for adiabatic pulses, check the adiabaticity.

15.9.8 How to Manipulate Shapes

Shapes can be manipulated in various ways. Most of these functions are only meaningful for RF shapes.

Frequency Encoding

To perform a phase (and amplitude) modulation encoding one or more offset frequencies:

☞ Click **Manipulate** → **Phase modulation acc. to Offset Freq.** [*offs*]

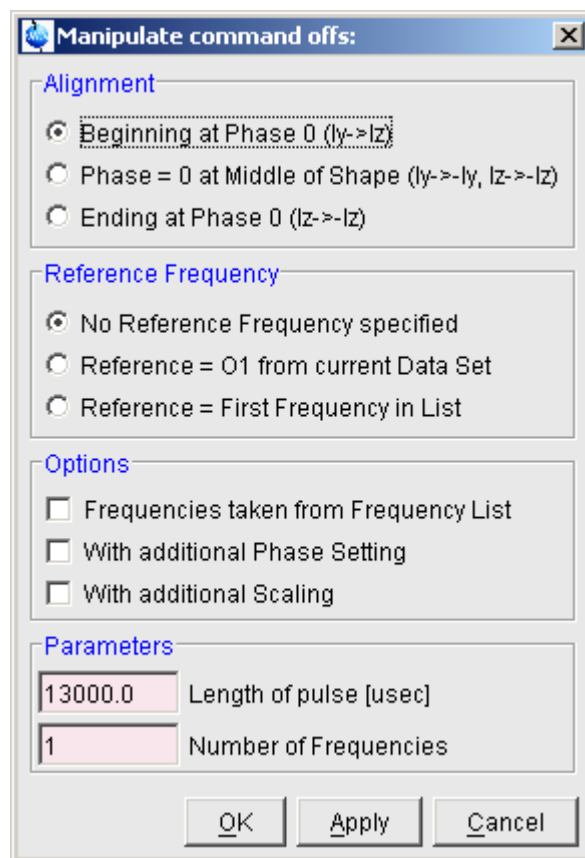


Figure 15.24

A dialog box will appear (see Fig. 15.24) where you can set the required parameters.


The **Alignment** radio buttons allow you to define the position of phase 0; the beginning, middle or end of the shape.

The **Reference Frequency** radio buttons allow you to choose whether the reference frequency:

- is not used
- is set to O1 of the associated dataset

- is set to the first entry of the frequency list. The **Option** 'Frequencies taken from Frequency' List will automatically be checked.

The **Options** check boxes allow you to set:

- *Frequencies taken from a Frequency list.* The **Parameter** Name of Frequency List will automatically appear. Frequency lists can be set up from the TOPSPIN interface with the command *edlist* or, interactively, by clicking the  button.
- *With additional Phase Setting*
- *With additional Scaling (0-100%):* the relative amplitude for each excitation region. The overall amplitude will be divided by the number of frequencies. For example, for 3 frequencies with equal scaling factors, each frequency will contribute 33%.

The **Parameters** fields:

- The *Length of Pulse*
- The *Number of Frequencies* or, when the **Option** *Frequencies taken from a Frequency list* is checked, *Name of the Frequency List*.

Clicking **Apply** or **OK** will open a new dialog box where the selected parameters are shown.

If no reference frequency is used, the values entered as frequencies have to be difference frequencies.

How to Calculate a Shape from an Excitation Region

A shape can be modulated according to specific regions in a spectrum. To do that, take the following steps:

1. Switch to the associated dataset.
2. Determine the integral regions interactively.
3. Switch back to the Shape Tool interface.
4. Click *Manipulate* → *Calc. Shape from excitation Region* [**region**].

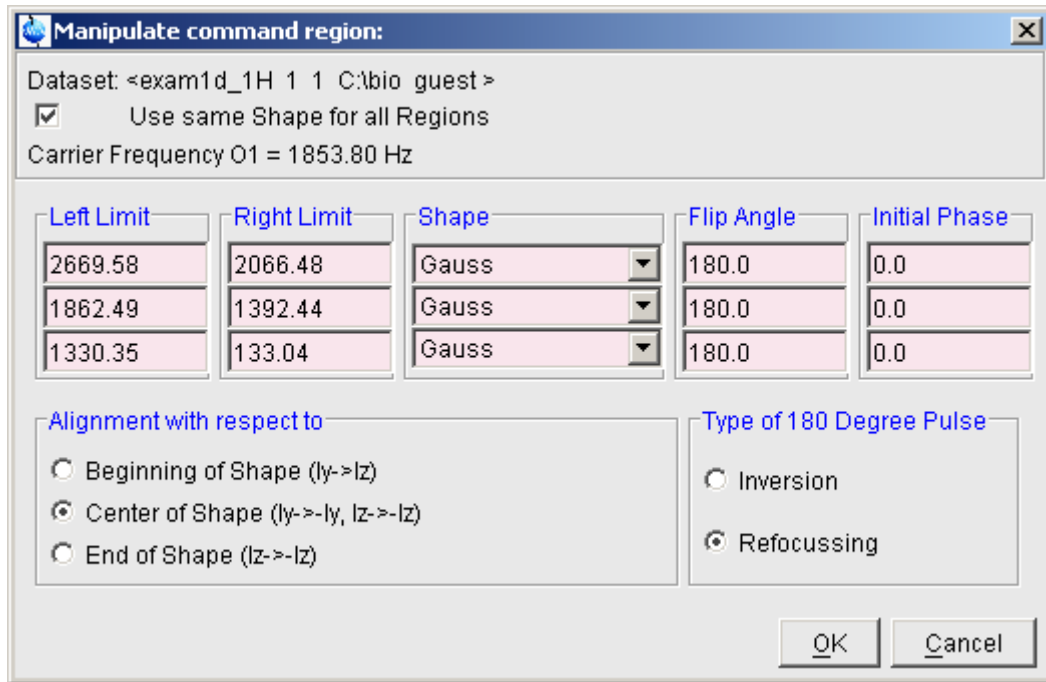


Figure 15.25

If the `intrng` file does not exist, you will be prompted for the number of regions and the carrier frequency O1. In that case, the fields **Left Limit** and **Right Limit** are empty. Note that overlapping regions are not allowed.

In Fig. 15.25 the default values for Flip Angle, Initial Phase, Alignment and Type of 180 Degree Pulse are shown. Depending on the application these settings have to be modified accordingly.

You can use the same shape for each region of all different shapes. In the latter case, the check box *Use same Shape for all Regions* must be unchecked.

How to Add Shapes

To add multiple shapes:

☞ Click *Manipulate* → *Add Shapes* [`addshapes`]

A dialog box will appear where you can select up to 10 shapes from the available shape files.

A minimum of two shapes must be selected. Note that the selected shapes replace the current shape.

The file size of each shape is shown in the *Size* column. Note that if the sizes differ, the desired alignment must be selected.

For each added shape, you can specify a scaling factor between 0 and 100% in the *Scaling* column.

The *Scale resulting shape* radio button allows you to rescale the resulting shape:

- to 100%
- to the average of the individual scaling factors

The alignment of the added shapes may be with respect to the *Beginning of Shape*, the *Center of Shape* or the *End of Shape*. If *Arbitrary Alignment* is selected, the start point must be entered for each shape in the *Start* column. In other cases the *Start* entry is deactivated.

How to Perform Single Sine Modulation

To perform an amplitude modulation such that the pulse excites at two symmetric sidebands (+/- offset) with opposite phase:

☞ Click *Single Sine Modulation* [*sinm2*]

and set *Offset frequency* to be relative to the carrier frequency.

How to Perform Single Cosine Modulation

To perform an amplitude modulation such that the pulse excites at two symmetric sidebands (+/- offset) with the same phase:

☞ Click *Manipulate* → *Single Cosine Modulation* [*cosm2*]

and set *Offset frequency* to be relative to the carrier frequency.

How to Create a Shape with a Linear Sweep

To create a shape with a phase modulation according to a linear frequency sweep:

☞ Click *Manipulate* → *Linear Sweep* [*sweep*]

and set the pulse length and sweep width. Then specify the Q factor to obtain the value for $\gamma B_1(\max)/2\pi$. Q is the quality or adiabaticity factor which, as a rule of thumb Q is set to 5 for inversion pulses and between 2 and 3 for decoupling puls-

es.

How to Create a Shape with Const. Adiabaticity Sweep

To create a shape with a phase modulation according to a constant adiabaticity sweep:

☞ *Manipulate* → *Const. Adiabaticity Sweep* → [*caSweep*]

and set the pulse length and sweep width. Then specify the Q factor to obtain the value for $\gamma B_1(\max)/2\pi$. Q is the quality or adiabaticity factor which, as a rule of thumb Q is set to 5 for inversion pulses and between 2 and 3 for decoupling pulses.

How to Calculate a Shape with Amplitude to the Power x

To create a shape with an amplitude to the power of a specified exponent:

☞ Click *Manipulate* → *Power of Amplitude* [*power*]

and enter the desired exponent.

How to Scale the Amplitude

To scale the amplitude of a shape to a specified percentage:

☞ Click *Manipulate* → *Scale Amplitude* [*scale*]

and enter the desired scaling factor (percentage).

How to Add Constant Phase

To add a constant phase to the shape:

☞ Click *Manipulate* → *Add constant Phase* [*addphase*]

and enter the phase to be added.

How to Perform Time Reversal

To time reverse a shape:

☞ Click *Manipulate* → *Time Reversal* [*trev*]

How to Expand Shape

To expand a shape according to a specified phase list (supercycle):

☞ Click *Manipulate* → *Expand Shape* [*expand*]

and selected a phase cycle. The following standard phase cycles are offered:

mlev4, mlev16, p5(150 deg), p5(330 deg), p5m4, p5m16, p9, p9m16, p5p9

For example, **mlev4** has the following supercycle: 0 0 180 180

In addition, the entry **YourOwn** appears, which allows you to define your own phase cycles. These can be saved for later usage. The specified file must have the extension `.expand`. It is stored in the directory:

```
<user-home>/ .topspin-hostname/shapetool
```

15.9.9 How to Set Shape Tool Options

When the Shape Tool window is selected, the upper three entries in the TOPSPIN *Options* menu concern the Shape Tool.

How to Change the Relation between Shape Tool and TOPSPIN parameters

To define the relation between Shape Tool parameters and TOPSPIN acquisition parameters:

☞ Click *Options* → *Define Parameter Table*

A dialog box will appear with a list box for each parameter (see Fig. 15.26).

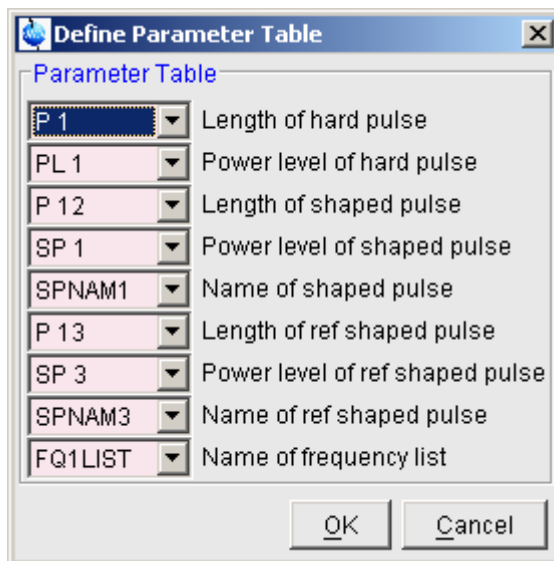


Figure 15.26

Make any changes you want and click **OK**. The acquisition parameters of the associated dataset can be set/viewed from the TOPSPIN interface by clicking **AcquPars** or entering **eda**.

How to Change the Shape Storage Directory

To change the shape storage directory:

- ☞ Click **Options** → **Set path to Shape Directory**

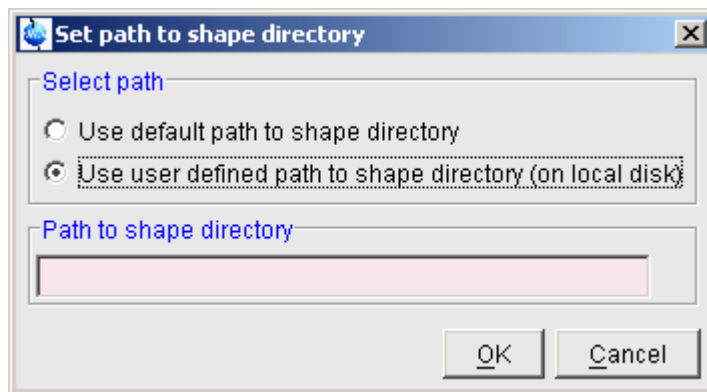


Figure 15.27

In the appearing dialog box, you can select the default path or select and specify a different path. Note that a shape base directory must be `shape` and a gradient base directory must be `gp`. Opening and saving shapes/gradients will access the specified directory.

How to select the Associated Dataset

To set the dataset associated to the Shape Tool interface:


☞ Click *Options* → *Select associated dataset*


A list of open datasets will appear. Select the desired dataset and click *OK*.

15.9.10 Examples of using the Shape Tool



Example 1: Create a Shaped Pulse for Multiple Solvent Suppr. using WET

To create a shaped pulse:

1. Acquire a reference 1D spectrum.
2. Click  to switch to frequency list mode.
 - Select the frequency list type and enter the list name *freqlist*. If you get a message that this list already exists, click *Overwrite* to overwrite the existing list or click *Cancel* and specify a new name.
 - Move the vertical cursor line to the desired frequencies.
 - Left-click the desired frequencies.

- Click  to save the frequency list and return.
3. To view the frequency list:
 - ☞ Enter ***edlist f1 freqlist*** and click **OK**.
 It should look, for example, like this:


```
0 300.19
3759.74
8775.99
8712.96
```

 Click **Cancel** to close the list.
 4. Enter ***stdisp*** to start the Shape Tool interface.
 5. Proceed as follows:
 - ☞ Click , choose **Open Shape**, select ***Sinc1.1000*** and click **OK**
 - or*
 - ☞ Click **Shapes** → **Classical Shapes** → **Sinc** and enter:
 - Size of shape: 1000**
 - Number of cycles: 1**
 in the parameter section.
 6. Click **Manipulate** → **Phase modulation acc. to Offset Freq. [offs]**
 In the appearing dialog box (see Fig. 15.28):
 - Select **Ending at Phase 0**.
 - Select **Reference = O1 from current Data Set**.
 - Check the box **Frequencies taken from Frequency List**.
 - Set **Length of Pulse** to **10000**.
 - Set **Name of Frequency List** to **freqlist** or the name you specified in step 2.
 - Click **OK**.
 7. Check the frequency values in the appearing dialog box (see Fig. 15.29) and click **OK**.
 8. Click  and click **Save Shape**, enter a **Name** and **Title**, set the **Flip Angle**

to 90° and the *Type of Rotation to excitation*

Now you have created a shape that can be used in a WET experiment.

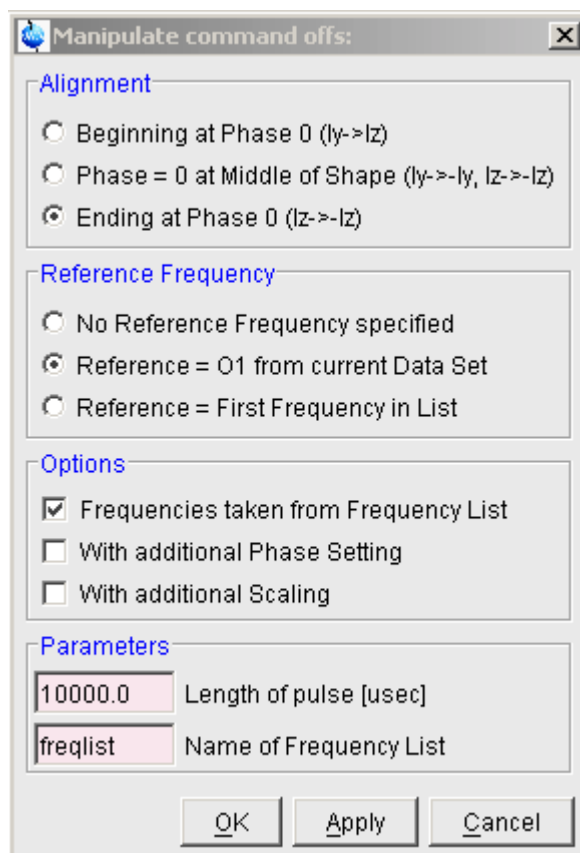


Figure 15.28

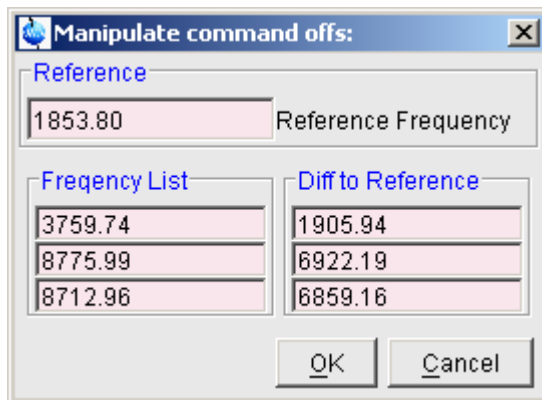


Figure 15.29


Example 2: Two-step Procedure to a Selective Experiment

1. Read/acquire a 1D reference spectrum.
2. Determine the width of the region to be excited. Let's assume this is 40 Hz.
3. Create a new dataset.
4. Read the pulse program *selgpse*.
5. Enter **ased** and set all acquisition parameters for *selgpse*, except for the shaped pulse parameters.
6. Enter **stdisp** to start the Shape Tool interface.
7. Click **Shapes** → **Classical Shapes** → **Gauss**
8. In the parameter section, enter:




Size of shape: 1000

Truncation level : 1
9. Click **Options** → **Define Parameter Table**
 - Set *Length of shaped pulse* to P12
 - Set *Power level of shaped pulse* to SP2
 - Set *Name of shaped pulse* to SPNAM2
 and click **OK**.
10. Click **Analysis** → **Calculate Bandwidth for Refocusing -My**

- Enter **Total Rotation**: 180
this calculates the bandwidth factor, which is the product of the width of excitation ($\Delta\Omega$) and the pulse length (ΔT).
 - Enter **Delta Omega**: 40 (the bandwidth determined in step 2)
this calculates the shaped pulse length ΔT !!!
 - Click **update parameters** to store this pulse length as P12.
11. Click **Analyses** → **Integrate Shape**
 - Click **update parameters** to store the name of the shaped pulse to SPNAM2
 12. On the associated dataset: enter the command **ased** and check the values of **P12** and **SP2**. If the steps above were performed correctly, they should have the values of *DeltaT* and *Change of power level* (+ hard power level), respectively.

Note that if the peak is off-resonance, you need to determine the chemical shift difference between the peak and O1 and set the parameter SPOFFS2 accordingly. To determine the proper value, click , put the cursor on the desired peak and subtract the value of SFO1 from the MHz value displayed in the upper left of the data field.

Example 3: Create a Shaped Pulse for Sel. Excitation from Integral Regions

1. Read/acquire a 1D reference spectrum.
2. Click  to switch to integration mode.
3. Click  to define regions around the peaks to be excited.
4. Click  to save these regions and return.
5. Create a new dataset.
6. Read the pulse program *selgpse*.
7. Enter **ased** and set all acquisition parameters for *selgpse*, except for the shaped pulse parameters.
8. Switch back the reference spectrum.
9. Enter **stdisp** to start the Shape Tool interface.
10. Click **Shapes** → **Classical Shapes** → **Gauss**
11. In the parameter section, enter:

Size of shape: 1000

Truncation level : 1

12. Click *Options* → *Define Parameter Table*

- Set *Length of shaped pulse* to P12
- Set *Power level of shaped pulse* to SP2
- Set *Name of shaped pulse* to SPNAM2

13. Click *Manipulate* → *Calc. Shape from excitation Region* [*region*]

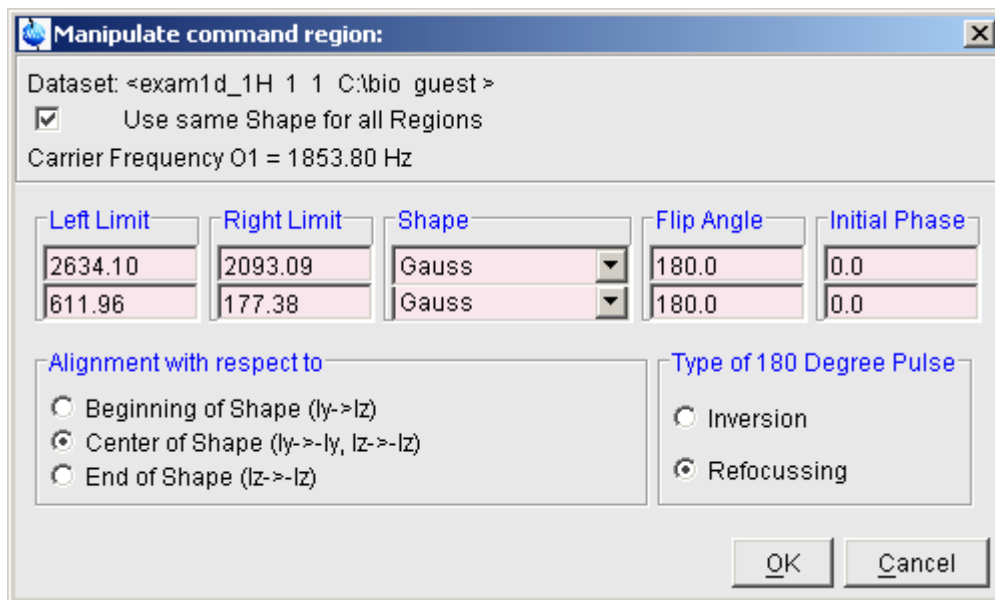
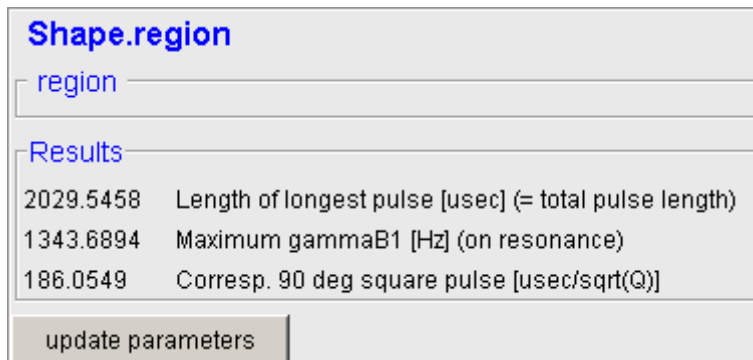


Figure 15.30

14. Click **OK**



Shape.region

region

Results

2029.5458	Length of longest pulse [usec] (= total pulse length)
1343.6894	Maximum gammaB1 [Hz] (on resonance)
186.0549	Corresp. 90 deg square pulse [usec/sqrt(Q)]

update parameters

Figure 15.31

15. Click **Options** → **Select associated dataset** and select the *selgpse* dataset.

16. Click **update parameters**, specify a name for the shaped pulse and click **OK**.

17. Now switch to the *selgpse* dataset and start the experiment.

Chapter 16

Configuration/Automation

16.1 NMR Superuser and NMR Administration password

During TOPSPIN installation, you are prompted to define:

- the username for the so called NMR Superuser. Under Windows this must be the name of an existing user. Under Linux it can also be a non-existing user, which is then automatically created by the installation program. After the installation, the NMR Superuser is the owner of all TOPSPIN program files. Logging in as this user allows you to remove these files, change file permissions etc. The name of the NMRSUPERUSER will be stored in the file:

```
<tshome>/conf/nmrsuperuser
```

- the NMR Administration password to be used for TOPSPIN configuration commands. This password can be freely chosen and is not connected to any user. It is asked for by TOPSPIN commands like *cf*, *expinstall* etc. The encrypted NMR Administration password is stored in the file:

```
<tshome>/conf/nmradminpassword
```

Note that the NMR Superuser login password and the NMR Administration password have different purposes and are totally independent. Changing one of them does not affect the other.

How to Change the NMR Administration Password

The NMR Administration password can be changed as follows

Under Windows

1. Login as NMR Superuser or Administrator.
2. Open a Command Prompt.
3. Enter:

```
<x>\perl\bin\perl <x>\prog\bin\installnmr <x> <NMRSUPERUSER>
```

where <x> in the TOPSPIN installation directory.

4. Enter the old password and new password as requested.

Under Linux

1. Login as NMR Superuser or root.
2. Open a Shell.
3. Enter:

```
<x>/prog/bin/installnmr <x> <NMRSUPERUSER>
```

where <x> in the TOPSPIN installation directory.

4. Enter the old password and new password as requested.

If you don't know the old NMR Administration password, you can still define a new one. In that case, you have to delete the file:

```
<x>\conf\nmradminpassword
```

before you run the *installnmr* script.

16.2 Configuration

The main configuration steps are performed by the commands **cf** and **expinstall**. They can be started from the:

- Command line
- **Options** → **Spectrometer tools** menu
- **Spectrometer** → **Setup** menu

However, the **Spectrometer** menu is only available after **cf** has been performed

once, choosing *Installation for spectrometer*.

How to Perform a Default Configuration on a Datastation

A default configuration can be used on a datastation. It is automatically performed (no **cf** required) during the installation of TOPSPIN on a new computer, a new disk or in a new TOPSPIN installation directory. The default configuration name is *Bruker_default_av500* and corresponds to a Avance 500 MHz spectrometer.

For manual or interactive data processing, the automatic default configuration is sufficient. If, however, you want to use AU programs, you must execute **expinstall** once, selecting *Installation for Datastation (Default)*.

How to Perform a Customized Configuration on a Datastation

If you want to configure your datastation according to a spectrometer other than default, you must first copy the configuration directory:

```
<tshome>/conf/instr/<instrum>
```

from that spectrometer to the datastation. Here:

<tshome> is TOPSPIN home, the directory where TOPSPIN is installed. Note that this can be different on the spectrometer than on the datastation.

<instrum> is the configuration name.

After copying the configuration directory, you have to perform **expinstall** as follows:

- Click *Spectrometer* → *Setup* → *Experiment installation* or enter **expinstall** on the command line

Follow the instructions on the screen. In successive dialog boxes check/select the options below and click *Next* to continue:

- *Installation for Datastation (Customize)*
- *High Resolution Systems*
- The configuration name as it was copied from your spectrometer
- The items you want to install
- Select the desired printer and paper format for the parameter sets
- The spectrometer frequency, acquisition mode and pre-scan-delay

In the last dialog box, click *Finish*. The installation of the selected items, will start

now. Wait until this process has finished.

For more details on *expinstall*, please refer to the description of this command in the Acquisition Reference manual.

16.3 Parameter set conversion

The command *paracon* changes the basic frequency in parameter sets. This allows you to use parameter sets which were created on a spectrometer with a different frequency. It opens dialog box shown in Fig. 16.1

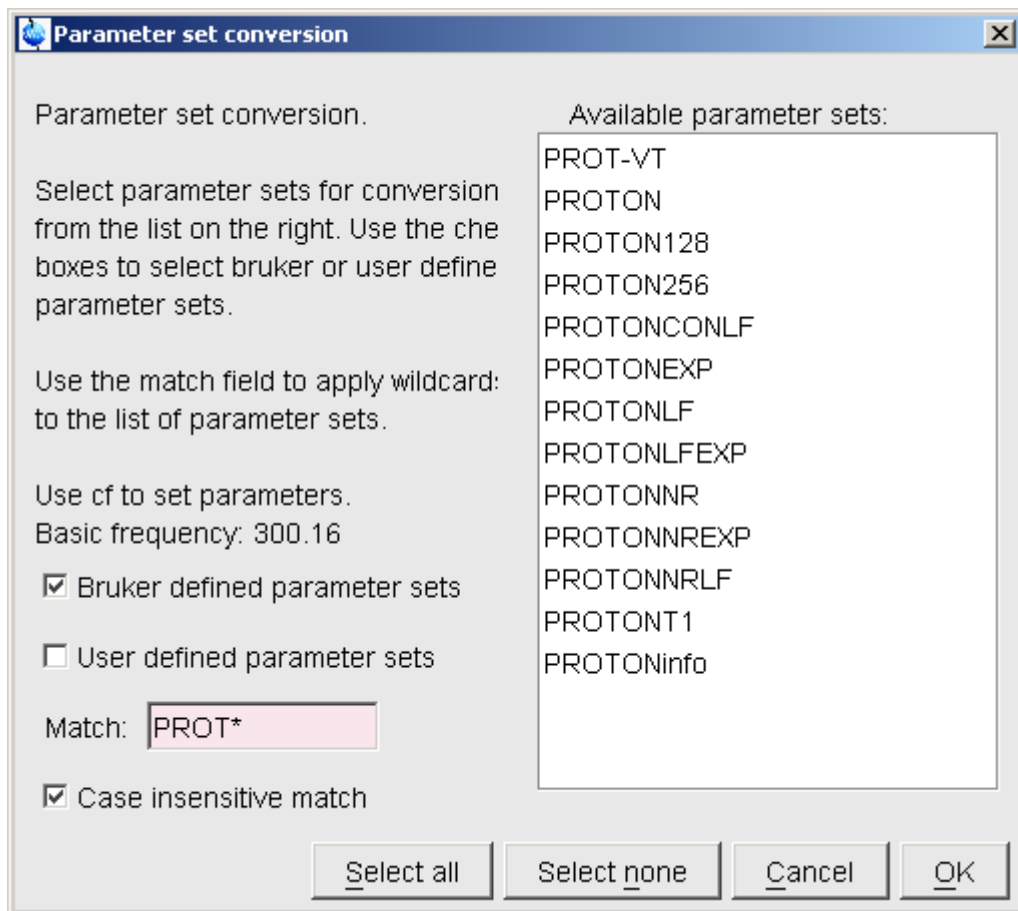


Figure 16.1

Here you can setup a list of available parameter sets. You can select Bruker and/or User defined parameter sets and use a match string. The matching parameter sets appear in the right part of the dialog box. To start the conversion, select one or more parameter sets and click **OK**.

16.4 Automation

How to Install AU Programs

To install AU programs, you have to run the command ***expinstall*** (see chapter 16.2).

How to Open the AU Program Dialog Box

To get a list of all AU programs, enter ***edau*** or:

1. Click ***File*** → ***Run...***
2. Click ***Execute an AU program*** in the appearing dialog box.
3. Click ***OK***

A dialog box showing either the Bruker defined or User defined AU programs.

Fig. 16.2 shows a dialog box with two User defined AU programs.

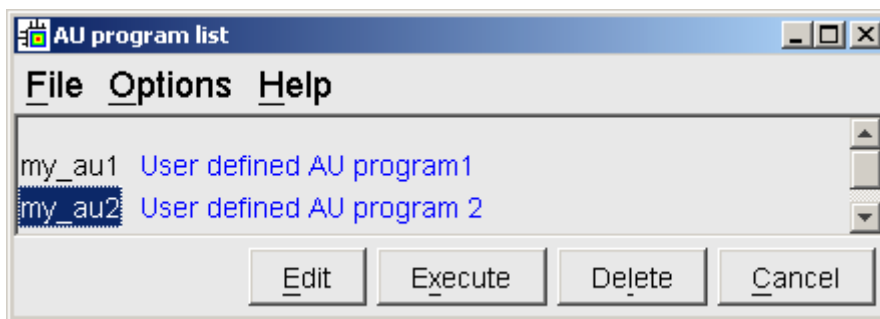


Figure 16.2 List with two User defined AU programs

Note that Bruker AU programs are only shown if the command ***expinstall*** has been executed once, after the installation of TOPSPIN.

How to Switch to the List of User defined AU Programs

- ☞ Click ***Options*** → ***User defined*** in the AU program dialog box

How to Switch to the List of Bruker defined AU Programs

☞ Click *Options* → *Bruker defined* in the AU program dialog box

How to Create an AU Program

1. Click *File* → *New* in the AU program dialog box.
2. Enter the AU program lines in the edit field of the appearing dialog box.
3. Click *Save as...* to store the AU program under a new name.
4. You will be prompted to compile the AU program: click *OK*.

Alternatively, you can enter **edau <name>** on the command line to create the AU program <name>.

How to Edit an Existing AU Program

1. Double-click the AU program name in the AU program dialog box

or

Click the *Edit* button to edit the highlighted AU program.

2. Modify the AU program according to you wishes.
3. Click *Save* to store the AU program under the name shown in the title bar.
4. You will be prompted to compile the AU program: click *OK*.

Alternatively, you can enter **edau <name>** on the command line to edit the AU program <name>.

How to Execute an AU Program

1. Select the AU program in the AU program dialog box.
2. Click the *Execute* button.

Alternatively, you can enter **<name>** or **xau <name>** on the command line to execute the AU program <name>.

If the AU program has not been compiled, compilation is automatically performed before the execution starts.

How to Delete an AU Program

1. Select the AU program in the AU program dialog box.
2. Click *File* → *Delete* or click the *Delete* button.

How to Show Comments (short descriptions) in the AU Program List

To switch on/off the comments in the AU program list:

☞ Click *Options* → *Comment on/off* in the AU program dialog box.

A comment is a short description of the AU program which is also part of the AU program header.

Chapter 17

Remote Control

17.1 Remote control

TOPSPIN supports remote control of your spectrometer over the local network or over the internet. This means you can control your spectrometer from your office desk or from your PC at home. Using your local TOPSPIN interface, you have access to the remote data directories and remotely running TOPSPIN commands. Furthermore, in TOPSPIN 1.3 and newer, ICON-NMR is web-enabled which means it can be controlled from any web browser which is networked to the spectrometer.

Remote access is operating system independent.

17.2 How to Establish a Remote Connection from your PC

In order to establish a remote connection, please do the following on the local PC:

1. Click *Options* → *Preferences* [*set*].
2. Click *Miscellaneous* in the left part of the dialog box.
3. Click the *Change* button at the entry *Setup remote systems*.
 - Server ID: an arbitrary name which is just a mnemonic to easily identify the remote system

- Hostname or address: the hostname or IP address of the remote host
- TCP/IP port number: the port number of the remote TOPSPIN

The Server ID can be freely chosen, whereas the hostname and TCP/IP port number can be identified using the *hist* command. Run *hist* on the remote system and look for the line:

To connect to this Topspin use host=, port=

Before you can establish a remote connection, you have to make sure the remote PC is open for remote access (see chapter 17.6). Then do the following on the local PC:

1. Click *Options* → *Remote connection...*
2. Select a *Remote system* from the appearing dialog box and click **OK**.

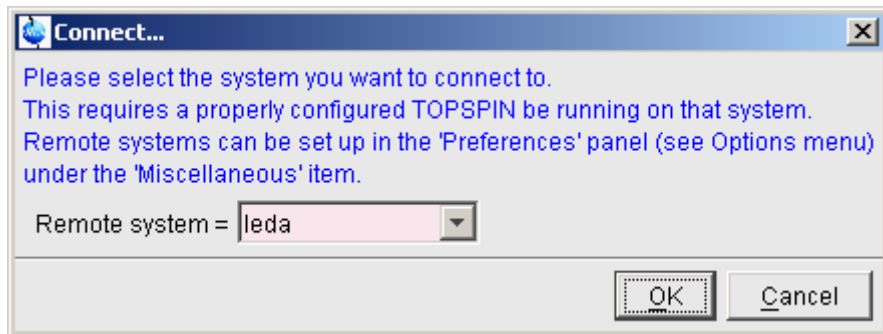


Figure 17.1

Now you are prompted for a user and password on the remote computer:

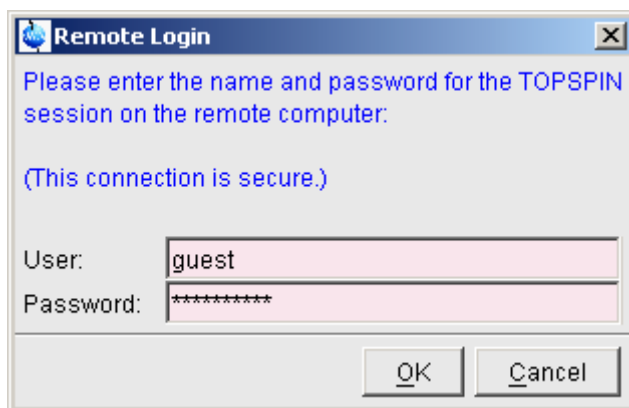


Figure 17.2

The TOPSPIN browser will show the data directory tree of the remote system. You are now ready to acquire, process or analyse remote data.

17.3 How to Configure TCP/IP Communication Ports

The communication between various parts of the TOPSPIN program is based on TCP/IP technology and uses a high-level protocol known as CORBA (an acronym for 'Common Object Request Broker Architecture'). This concerns communication between the TOPSPIN interface on the one hand and various other objects like e.g. the command interpreter and servers for datasets and hardware control on the other hand. CORBA allows these objects to run on the local host or, alternatively, on a remote host in the network. Each component automatically allocates two TCP/IP ports for its services. One for local network traffic and another one for SSL secured remote connections. To give system administrators control over this traffic (e.g. for firewall configuration), default port numbers can be configured by entries in the file:

```
<tshome>/prog/server/corba.conf
```

This file contains one line for each object and looks like this:

```
ConfigurationServer: 5500  
CPR: 5500  
CPRCallback: 5501  
NMRDataServer: 5505
```

FidDataServer:	5510
GSServer:	5511
BsmsServer:	5512
EdacbServer:	5513
EdteServer:	5514
LockDataServer:	5515

For the component NMRDataServer several instances could be active which then automatically use increasing port numbers (5505, 5506, 5507...). The ports used for remote connection are derived by adding 100 to these numbers. That means to enable remote control over a firewall secured connection the ports 5600-5615 must be opened on the firewall for the local system.

17.4 How to Make a Remote Connection without a Local License

TOPSPIN requires a license to operate on your local computer. However, you can make a remote connection without a local license.

To do this, you have to start TOPSPIN as follows:

1. Open a Windows Command Prompt or Linux Shell.
2. Go to the TOPSPIN installation directory.
3. Enter ***topspin -client***
4. TOPSPIN will start up and show an empty data field but no browser. Before startup you will get the error message: "*The program failed to communicate with local*". Just click **OK** to continue.
5. Click **Options** → **Remote connection...** and establish a remote connection as described in chapter 17.2.

Note that without a local license:

- TOPSPIN on the remote system must run with a licence.
- Local data cannot be accessed.

17.5 Security of Remote Connections

All data transferred during a remote control session is, by default, encrypted. This ensures that nobody can see data or commands by observing and recording your

network traffic. TOPSPIN uses the Secure Socket Layer (SSL) technology for encryption, which is also used for secure web sites. SSL needs digital certificates on both sides of a connection to achieve a valid authentication. After a TOPSPIN installation, default certificate files are provided to secured connections. There is a chance, however small, such connections are attacked by a malicious person who also has the same default certificate files. If, for this reason, you want to create your own set of certificate files, you can do this by executing a script that is installed in <tshome>/prog/server/make_new_certificates and follow the instructions given there. To start a remote connection the new certificate files must be installed on BOTH machines.

17.6 How to Allow Remote Connections

For security reasons, remote access to a PC is, by default, prohibited for all systems. To open a PC for remote access:

1. Click *Options* → *Preferences* [*set*]
2. Click *Miscellaneous* in the left part of the dialog box.
3. Click the *Change* button to the right of the object *Enable/Disable remote access*.
4. Enter the Administration password as requested
5. In the appearing dialog box, click *Enable* (see Fig. 17.3)

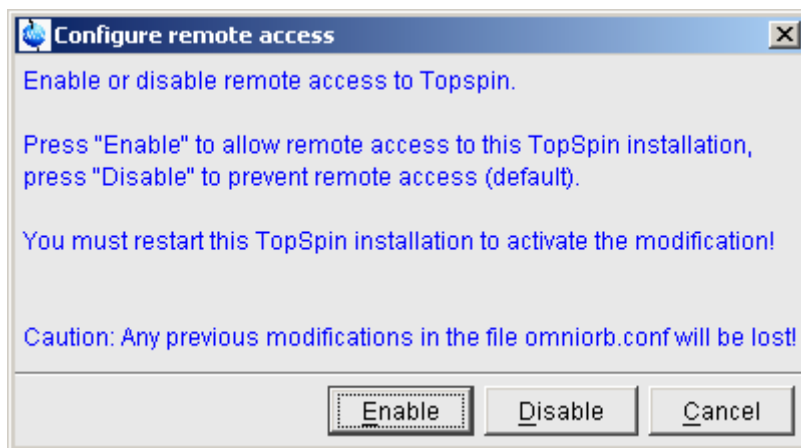


Figure 17.3

6. Click *OK*

Now a running TOPSPIN on this PC can be accessed from any computer in the network.

17.7 Note to Windows XP Service pack 2

If the remote PC is running under Windows XP with Service Pack 2, the Windows Firewall is enabled by default. In this case, the Firewall must be configured for TOPSPIN operations. This can be done either during TOPSPIN installation (TOPSPIN ≥ 1.3 only) or with the script `setfirewall.cmd`.

Note that the firewall must to be configured for TOPSPIN :

- spectrometer control
- remote access

TOPSPIN can be used for processing/analysis on a local datastation, even if it is not registered on the firewall. However, you will get a few Security Alert messages during startup.

17.8 How to Access ICON-NMR from a Remote Web Browser

In TOPSPIN 1.3 and newer, ICON-NMR is web-enabled. The standard Bruker Automation Software now provides a built-in website which can be activated to allow remote access to ICON-NMR from any web browser which is networked to the spectrometer workstation. Experiments may be cancelled/submitted, the run may be paused or halted and spectra in PDF format are available from the browser window. No added software apart from ICON-NMR is required and configuration is performed via the standard ICON-NMR Configuration window. For security a SSL/HTTPS connection is supported. Pocket PC Internet Explorer is also supported for full spectrometer control via Windows Mobile 2003 (TM) or equivalent. PDF files of spectra may also be viewed on this platform making the mobile pocket spectrometer a reality.

Chapter 18

User Preferences

18.1 User Preferences

TOPSPIN can be tailored to your preference in many respects. This ranges from startup options to spectrum objects, menu settings, remote connections etc. Every standard user can create his/her own set of preferences.

To set user preferences:

- Click *Options* → *Preferences* [*set*]

A dialog box will appear with, at the left side, the categories that can be tailored

(see Fig. 18.1)

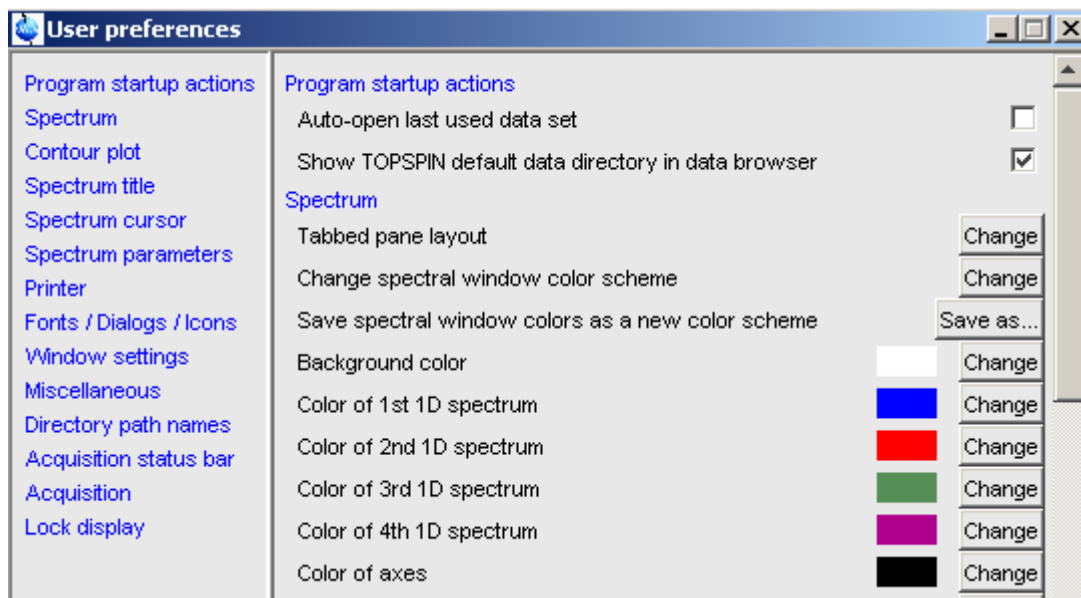


Figure 18.1

Click the category of which you want to view/change certain objects. It will become highlighted and the corresponding objects will be displayed at the right part of the dialog box. For example, if you click *Spectrum*, the spectrum objects will appear at the top of the dialog box. The rest of this paragraph will describe some examples of setting various user preferences.

How to Define the Startup Dataset

TOPSPIN allows you to define the dataset to be displayed after startup. To do that:

1. Open the Windows Explorer and navigate to the user properties directory (enter *hist* in TOPSPIN to identify this directory).
2. Open the file `autostart.prop` and enter the desired startup command with the following format:

```
_VERSION=1.0
cmd_<arbitrary_name>=<topspin_command>
```

Note that the file normally exists and the first line is already there.

Example:

```
_VERSION=1.0  
cmd_read_spectrum=re exam1d_13C 1 1 c:/bruk/ts guest
```

The above lines would cause TOPSPIN to display the dataset:

```
C:/bruker/ts/data/guest/nmr/exam1d_13C/1/pdata/1
```

after startup.

Note that you can use a single forward slash (/) or a double backslash (\\) as path separator. An example of the latter would be:

```
_VERSION=1.0  
cmd_read_spectrum=re exam1d_13C 1 1 c:\\bruk\\ts guest
```

How to Set Automatic Startup Actions

1. Click *Options* → *Preferences* [**set**].
2. Check or uncheck the options under *Program startup actions*, for example:

Auto-open last used data set

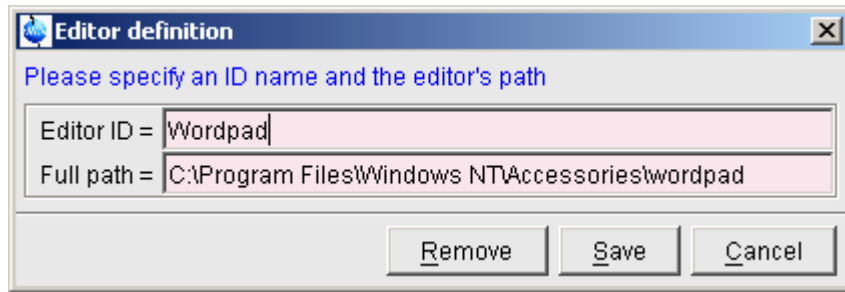
Show NMR Data Processing Guide

How to Change the Preferred Editor

You can choose your preferred editor as it is used by commands like **edau**, **edpul**, **edcpd** etc. To do that:

1. Click *Options* → *Preferences* [**set**].
2. Click *Miscellaneous* in the left part of the dialog box.
3. Click the **Change** button to the right of the object *Preferred text editor* enter the desired Editor and its path. For example for Wordpad under

Windows 2000, this would look like:



4. Click *Save* to save the changes.

How to Configure the Tab Bar

The default Tab bar at the top of the data window consist of Tabs to switch between various dataset objects like *Spectrum*, *Parameters*, *Title*, etc. You can, however, configure the Tab bar to contain Tabs for interactive data manipulation like *phase correction*, *integration* etc. These Tabs have the same function as the corresponding buttons in the upper tool bar (see chapter 11 and 12) but are easier to access. You can configure the Tab bar as follows:

1. Click *Options* → *Preferences*
2. Click *Spectrum* in the left part of the *User preferences* box.
3. Click the *Change* button to the right of the object *Tabbed pane layout*.
4. Check the desired Tabs, uncheck the others.
5. Click *OK*

How to Configure the Right-click Menu Function

By default, right-clicking a pull-down or popup menu, either displays or executes the corresponding command line command. This can be configured as follows:

1. Click *Options* → *Preferences*
2. Click *Miscellaneous* in the left part of the *User preferences* box.
3. Check/uncheck the item:

Right click on a menu item shows associated command.
4. Click *OK*

18.2 Changing Colors

How to Change Colors of Data Objects on the Screen

The color of various objects in a data window on the screen, like 1st, 2nd and 3rd spectrum, axis, parameters etc. can be changed. To set these colors:

1. Click *Options* → *Preferences* [**set**].
2. Click *Spectrum* in the left part of the *User preferences* box.
3. Click the *Change* button to the right of the object you want to change e.g.:



4. Select the desired color in the appearing dialog box and click **OK**
5. Click *Apply*

How to Change Colors of Data Objects on the Printer

The color of data objects on the printer is independent from the color of the corresponding object on the screen. To set print colors:

1. Click *Options* → *Preferences* [**set**].
2. Click *Printer* in the left part of the *User Preferences* box.
3. Click the *Change* button to the right of the object you want to change, e.g.:



4. Select the desired color in the appearing dialog box and click **OK**
5. Click *Apply*

How to Change Colors of the Lock Display

The colors of lock display objects can be changed as follows:

1. Click *Options* → *Preferences* [**set**].
2. Click *Lock display* in the left part of the *User preferences* box.

3. Click the **Change** button to the right of the object you want to change, e.g.:



4. Select the desired color in the appearing dialog box and click **OK**
5. Click **Apply**

How to Create a New Data Window Color Scheme

To create a new data window color scheme:

1. Click **Options** → **Preferences** [**set**].
2. Change any color of the objects *Spectrum*, *Spectrum title*, *Spectrum cursor* or *Spectrum parameters*.
3. Click **Apply**
4. Click **Spectrum** in the left part of the *User preferences* box.
5. Click the button **Save as...** to the right of the object:

Save spectral window colors as a new color scheme

and enter a new name in the appearing dialog box.

6. Click **OK**

How to Read a Different Data Window Color Scheme

1. Click **Options** → **Preferences** [**set**].
2. Click **Spectrum** in the left part of the *User preferences* box.
3. Click in the list box to the right of the object:

Change spectral window color scheme

and select an element from the appearing list.

4. Click **OK**

TOPSPIN is delivered with two colors schemes:

- *light* (default): a white background with black axes
- *dark*: a dark blue background with a white axes

18.3 Changing Fonts

How to Change All Fonts of the Topspin Interface

1. Click *Options* → *Preferences* [*set*].
2. Click *Fonts/Dialogs/Icons* in the left part of the dialog box.
3. Set the entry *Change size fonts listed above by points*
You can enter a positive or negative number.

Fig. 18.2 shows an example of increasing the font sizes by 4 points.



Figure 18.2

4. Click **OK** to store the new value.



Figure 18.3

Fig. 18.3 shows the same part of the Preferences dialog box dialog box after the change of fonts. Note that:

- The value all four font entries has been increased by 4.
- The font of the dialog box itself is larger.
- The change size has been reset to 0.

How to Change the Font of the TOPSPIN menu

To change the font of the TOPSPIN menu:

1. Click *Options* → *Preferences* [**set**].
2. Click *Fonts/Dialogs/Icons* in the left part of the dialog box.
3. Click the *Change* button to the right of the *Menu font* object:



Menu font dialog / Plain / 18 Change

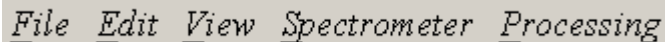
4. Select the desired *name, style* and/or *size* in the appearing dialog box.
5. Click **OK** to store the new font.
6. Click *Apply*

The default menu font is *Dialog/Plain/18* and looks like this:



File Edit View Spectrometer Processing

After changing the font to, for instance, to *Serif/Italic/18*, the menu looks like this:



*F*ile *E*dit *V*iew *S*pectrometer *P*rocessing

A change in menu font also affects all sub-menus and popup menus.

How to Change the Font of the Tab bar

1. Click *Options* → *Preferences* [**set**].
2. Click *Fonts/Dialogs/Icons* in the left part of the dialog box.
3. Click the *Change* button to the right of the *Dialog window font* object.
4. Select the desired *name, style* and/or *size* in the appearing dialog box.
5. Click **OK** to store the new font.
6. Click *Apply*

Fig. 18.4 and Fig. 18.5 shows a Tab bar with font size 10 and 14 respectively.

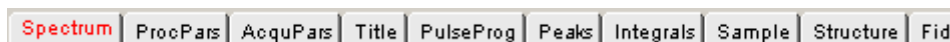


Figure 18.4 Tab bar with font size 10

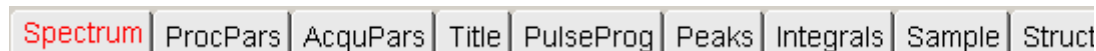


Figure 18.5 Tab bar with font size 14

How to Change the Font of Dialog Boxes

1. Click *Options* → *Preferences* [*set*].
2. Click *Fonts/Dialogs/Icons* in the left part of the dialog box.
3. Click the *Change* button to the right of the *Dialog window font* object.
4. Select the desired *name, style* and/or *size* in the appearing dialog box.
5. Click *OK* to store the new font.
6. Click *Apply*

Fig. 18.6 shows an example of a dialog box with the font *Time New Roman Italic*

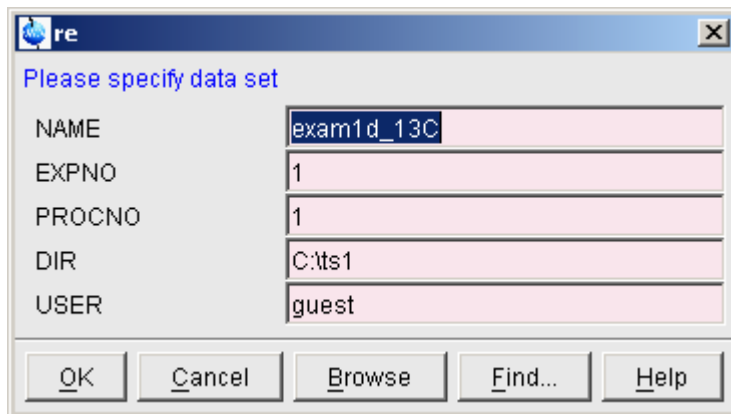


Figure 18.6

How to Change the Font of the Browser

To change the font of the browser:

1. Click *Options* → *Preferences* [**set**].
2. Click *Fonts/Dialogs/Icons* in the left part of the dialog box.
3. Click the *Change* button to the right of the *Dialog window font* object.
4. Select the desired *name, style* and/or *size* in the appearing dialog box.
5. Click **OK** to store the new font.
6. Click *Apply*

TOPSPIN must be restarted for this change to become effective. Note that this change will affect all dialog boxes.

How to Change the Font of the Command Line

1. Click *Options* → *Preferences* [**set**].
2. Click *Fonts/Dialogs/Icons* in the left part of the dialog box.
3. Click the *Change* button to the right of the *Command line font* object.
4. Select the desired *name, style* and/or *size* in the appearing dialog box.
5. Click **OK** to store the new font.
6. Click *Apply*

How to Change the Font of the Status Line

1. Click *Options* → *Preferences* [**set**].
2. Click *Fonts/Dialogs/Icons* in the left part of the dialog box.
3. Click the *Change* button to the right of the *Status line font* object.
4. Select the desired *name, style* and/or *size* in the appearing dialog box.
5. Click **OK** to store the new font.
6. Click *Apply*

18.4 Command Line Preferences

How to Resize the Command Line


By default, the TOPSPIN command line shows one command, the command that is

currently entered, e.g.:



```
apk
```

However, you can resize the command line to show the currently entered command plus the last and second last command, e.g.:



```
em  
ft  
apk
```

You can toggle between the two different command line sizes as follows:

- Click **View** → **Resize command line**

or

- Right-click in the command line and click **Resize command line**

How to Set the Minimum and Maximum Command Line Size

By default, the size of the command line can be toggled between 1 and 3. You can, however, change this minimum and maximum value, respectively. To do that:

1. Click **Options** → **Preferences** [**set**].
2. Click **Fonts/Dialogs/Icons** in the left part of the dialog box.
3. Specify the *Minimum visible command lines* (> 0).
4. Specify the *Maximum visible command lines* (\geq *Minimum visible*).
5. Click **Apply**

18.5 Disabling/Enabling Toolbar Buttons, Menus and Commands

How to Hide the Upper and Lower Toolbars

Right-click in an empty area of one of the toolbars and choose:

Hide the toolbars

from the appearing popup menu (see Fig. 18.7).

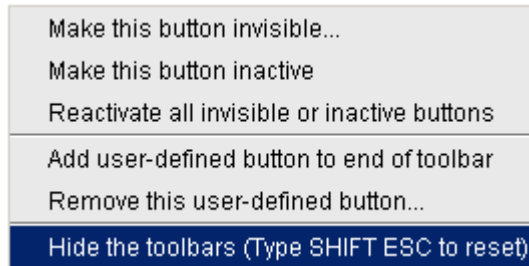


Figure 18.7

How to Hide the Menubar

Right-click in an empty area of the Menubar and choose:

Hide the menubar

from the appearing popup menu.

To restore the toolbars:

☞ hit the keys SHIFT+ESC

How to Disable/Remove Toolbar Buttons

Buttons of the upper or lower toolbar can be disabled or removed as follows:

1. Right-click a toolbar button.
2. Choose one of the entries:
 - ☞ *Make this button invisible* to remove the button from the toolbar
 - ☞ *Make this button inactive* to disable the button (it will appear greyed)

To restore the complete toolbars, click:

☞ *Reactivate all invisible or inactive buttons* to restore the default toolbar

How to Disable/Remove Menus or Commands

By default, all existing TOPSPIN commands can be entered from the menu and/or from the command line. You can, however, selectively disable/remove commands or menus. This is typically done by system administrators who want to disable certain functions for standard users.

Open the Menu Configuration table as follows:

1. Click *Options* → *Preferences* [**set**].
2. Click *Fonts/Dialogs/Icons* in the left part of the dialog box.
3. Click the *Change* button of the entry *Disable/Enable menus and commands*.

The table consists of the following columns:

Command	Description	Menu	Status
---------	-------------	------	--------

You can disable/remove:

- *Menus*, e.g. *Options*
to be found in the column **Menu**
- *Sub-menus*, e.g. *Options* → *Administration tools*
to be found in the column **Menu**
- *Menu entries*, e.g. *Options* → *Preferences*
to be found in the column **Description**
- *Commands*, e.g. *set*
to be found in the column **Command**

Note that in this configuration table, each *menu entry* corresponds to a certain *command*. In the TOPSPIN menu, most commands are indicated in square brackets behind the corresponding menu entries, e.g.:

☞ *Processing* → *Fourier transform* [**ftf**]

Menu entries which are not used very often appear without the corresponding command indicated, e.g.:

☞ *Options* → *Preferences*

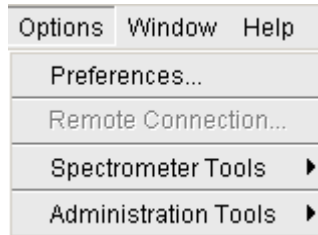
However, if you want, you can look up the corresponding command, in this case **set**, in the configuration table and use it.

As an example, we will describe how you can disable/remove the *menu entry* **Remote connection..** in the *Options* menu:

1. Open the configuration table as described above.
2. Scroll to the *Description* **Remote connection**

3. Set the *Status* to [disabled](#).
4. Click *Apply*

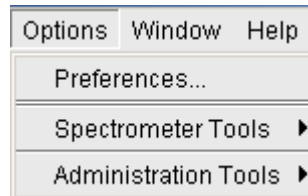
Clicking the TOPSPIN *Options* menu, will now show the following popup menu:



The entry *Remote connection..* is greyed and can no longer be used.

Go back to the Configuration table, change the *Status* of *Remote connection...* to [invisible](#) and click *Apply*.

Clicking the TOPSPIN *Options* menu, will now show following popup menu:



The entry *Remote connection..* has disappeared.

Note that if you disable or remove a *menu entry*, the corresponding *command* is automatically disabled. For example, if you disable:

☞ *Analysis* → *Deconvolution* [*dcon*]

entering *dcon* on the command line will lead to an error message.

How to (Re)enable a disabled Command/Menu

1. Open the Menu Configuration table as described above.
2. Set the Status of a disabled or removed (invisible) entry to [enabled](#).
3. Click *OK* to close the Configuration table.

How to (Re)enable All Commands/Menus

1. Open the Menu Configuration table as described above.
2. Click the **Reset** button to enable all menus and commands.
3. Click **OK** to confirm the appearing message.
4. Click the **OK** button to close the Configuration table.

18.6 Resizing/Shifting Toolbar Icons

How to Change the Toolbar Icon Size

1. Right-click of the toolbar,
2. In the appearing popup menu (see Fig. 18.8), click:
☞ *Change Icon Size...*

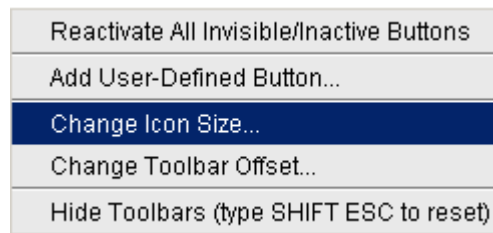


Figure 18.8

3. Enter the icon size in the appearing dialog and click **OK**.

How to Shift Toolbar Icons to the Right

1. Right-click at an empty area of the toolbar.
2. In the appearing popup menu (see Fig. 18.8), click:
☞ *Change Toolbar Offset...*
3. Enter the toolbar offset in the appearing dialog and click **OK**.


Chapter 19

User Extensions

TOPSPIN offers various ways to extend the standard commands, buttons, programs etc.

19.1 User Notebook

You can create your own user specific notebook with the command:

 *View* → *Notebook* [*nbook*]

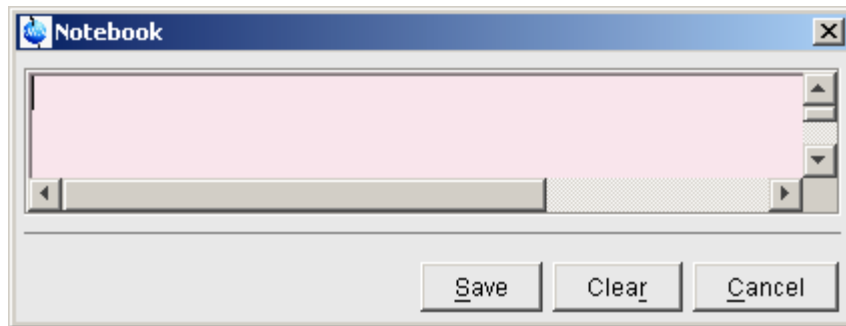


Figure 19.1

This can be used to store and retrieve any personal notes, information etc.

19.2 Macros

A macro contains a sequence of TOPSPIN commands. It can be created with the command **edmac**. A simple macro for processing and plotting the current dataset is:

```
em
ft
apk
sref
autoplot
```

All entries in a macro file must be written in lower case letters.

In TOPSPIN 1.3 and newer, a macro may contain Python commands. Any line in a macro that starts with:

```
py>
```

executes a Python command. An example of such a macro is:

```
re exam1d_13C 1 1 C:\bio guest
efp
py>NEWWIN() # open new window
re exam1d_1H 1 1 C:\bio guest
efp
py>x = INPUT_DIALOG()
py>if x == None: EXIT()
py>y = 2* int(x[0])
py>MSG("done: y=" + str(y))
```

Note that commands like `NEWWIN()`, `INPUT_DIALOG()`, `MSG()` and `EXIT()` are Bruker defined whereas "x=" and "if" are original Python commands.

Once created, a macro can be executed by entering its name on the command line.

19.3 AU Programs

An AU program may contain TOPSPIN commands, AU macros and C-language statements. It can be created with the command **edau**. A simple AU program which performs the **efp** command on a series of dataset *expno*'s is:

```
#include <lib/util.h>
```

```

int first, max;
char string[80];

first = expno;
GETINT ("Enter first expno to process : ",first)
max = 10;
GETINT ("Enter number of expnos : ",max)
WPAR("tmpmefp", "proc")
expno = first;
TIMES(max)
RPAR("tmpmefp", "proc")
EFP
IEXPNO
END
DEXPNO
DELPAR("tmpmefp")
QUITMSG("--- multiefp finished ---")

```

Note that TOPSPIN commands like EFP and RPAR and AU macros like IEXPNO are written in upper case letters whereas C-language statements like are written in lowercase letters. Once created, an AU program can be executed by entering its name on the command line.

For more information on writing AU programs:

☞ Click **Help** → **Programming** → **AU programming**

19.4 Python Programs

A Python program may contain TOPSPIN commands, User Interface functions and Graphic functions. It is created with **edpy**.

The Python program below reads a region of the real part of a spectrum and the corresponding region of the imaginary part and displays both. The simplest form of DISPLAY_DATA_LIST is used.

```

from TopCmds import *
import math
region = [80, 72] # define region in ppm
# open testdata, don't display
testdata = ["exam1d_13C", "1", "1", "c:/Bruker/topspin", "guest"]

```

```

RE(testdata, "n")
# read real and imaginary points of the region
reals = GETPROCdata(region[0], region[1])
imags = GETPROCdata(region[0], region[1], data-
const.PROCdata_IMAG)
if reals == None or imags == None: EXIT()
# set up list of data to be displayed and respective axis info list
dataList = [reals, imags]
# display the data in the list
DISPLAY_DATALIST(dataList)

```

For more information on writing Python program:

☞ Click *Help* → *Programming* → *Python programming*

19.5 Button Panels

A button panel is a window with user-defined buttons for executing TOPSPIN commands, AU programs, Python programs or macros. It appears as an integral part of the active data window and acts on that. Bruker delivers a few standard button panels like *bnmr* and *bnmrse1*. To create your own button panels, you can modify one of these or write them from scratch.

In this description we will create a very simple button panel with some 1D processing commands and print/export buttons (see Fig. 19.2)

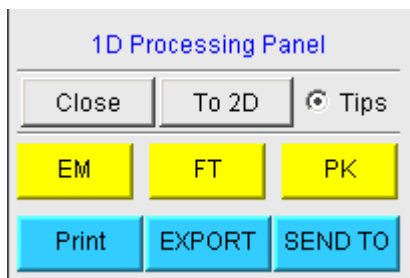


Figure 19.2

To write this button panel, take the following steps:

1. Open the Windows Explorer and navigate to the subdirectory `userdefined`

of the users properties directory ¹.

2. Create a text file with the name

```
cmdpanel_<name>.prop
```

where <name> is the name of the button panel.

3. Enter the button definitions including *Panel title*, *Colors*, *Toggle buttons*, *Top buttons*, *Panel layout*, *Panel buttons* and *Tooltips*.

4. Save the file under a name `cmdpanel_<xxx>.prop`

where <xxx> is the actual name of your command panel.

Make sure the extension of the file is `.prop` and not `.txt`, `.prop.txt` or anything else.

5. Enter `bpan <xxx>` on the command line to open the button panel.

Here is an example for a small button panel for 1D processing:

```
# Color definitions used in this file (RGB)

BLUE1=51$ 204$ 255
YELLOW1=255$ 255$ 0
GREEN1=84$ 196$ 20

# Title definition

TITLE=1D Processing Panel
TITLE_COLOR=0$ 0$ 255

# Toggle button definition

TOGGLE_BUTTON=To 2D
TOGGLE_CMD=bp
```

```
an bproc2d
TOGGLE_TIP=Switch to 2D processing

# Top row button definition

TOP_BUTTONS=EM$ $FT$ $PK$ $
TOP_COLORS=YELLOW1$ YELLOW1$ YELLOW1
TOP_CMDS=em$ ft$ pk
TOP_TIPS=Exponential multiplication $\
Fourier transform$\
Phase correction
```

1. To locate this, enter `hist` and look for the entry "User properties directory=".

```
# Panel button definitions
# LAYOUT format: rows columns hgap vgap
PAN_LAYOUT=1$ 3$ 8$ 8
PAN_BUTTONS=Print$ $ EXPORT$ $SEND TO$ $
PAN_COLORS=BLUE1$ BLUE1$ BLUE1
PAN_CMDS=prnt$ exportfile$ smail
PAN_TIPS=Print the spectrum<br>\
as it appears on the screen$\
Export the dataset<br>\
to png, jpg, bmp etc.$\
Send the dataset by email
```

Note that:

- The *Close* button and *Tips* switch are automatically created. You don't need to specify them.
- The TOGGLE button is typically, but not necessarily, used to call another button panel. In this example it calls the panel *bproc2d*.
- Items must be separated with the "\$" character, button items with "\$ \$"
- A "\" followed by "end of line" continues an item on the next line.
- Tooltips may use html tags for text formatting.
- Commands may be specified as single commands like "em" or as composite commands like "em\nft\npk". Note that in the latter case, the commands must be separated by "\n".

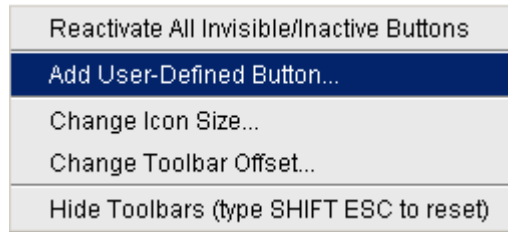
19.6 Adding User Defined Buttons to the Toolbars

The upper and lower toolbar at the top of the TOPSPIN window can be extended with user defined buttons. They can be assigned to any TOPSPIN command, macro, AU program or Python program.

To create a user defined button, take the following steps:

1. Right-click at an empty area of the toolbar.
2. In the appearing popup menu, click:

Add User-Defined Button



3. In the appearing dialog box (see Fig. 19.3)
 - a) Choose between the options *text label* or *icon*.
The corresponding parameters are enable/disabled.
 - b) Enter the *command name*, the *tool tip* and the *label text* or *icon file* pathname.
 - c) For a text label: set its *font*, *text color* and *background color*.

- a) Set the *separator* flag to *yes* or *no* and select the data *dimension(s)* for which the button must appear.

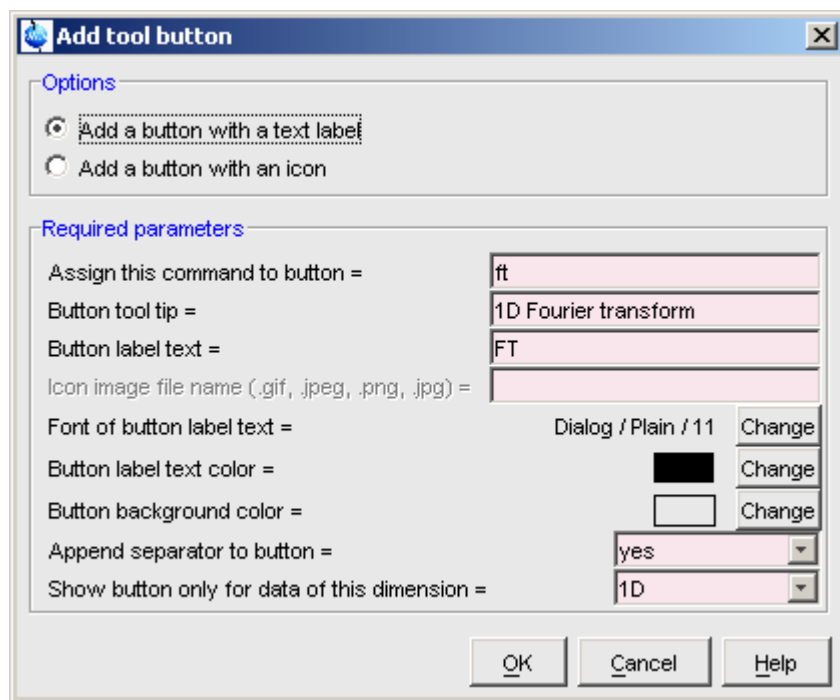


Figure 19.3

In the example above, a button **FT** is created with a separator, which only appears for 1D datasets, and executes the command **ft**.

How do I Remove a User Defined Toolbar Button

Right-click the toolbar button and click:

Remove this user-defined button...

in the appearing popup menu (see Fig. 9.5).

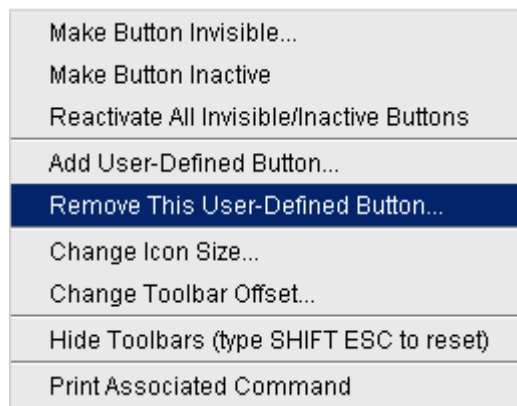


Figure 19.4

The button definitions are stored in the file `toolbar_user.prop` which resides in the subdirectory `userdefined` of the user properties directory. To locate this directory enter the TOPSPIN command **hist**. A dialog box will show the contents of the `history` file. Near the top of this file, you will see an entry "User properties directory=". For icon image buttons, the formats `.gif`, `.jpg`, `.jpeg` and `.png`, are supported. Standard TOPSPIN toolbar icons have a size of 16 * 16 pixels. If your own icons have a different size, they are automatically rescaled and displayed at the standard size.

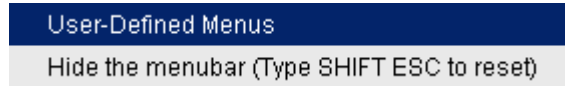
19.7 Adding User Defined Menus to the Menubar

The menubar at the top of the TOPSPIN window can be extended with user defined menus. They can be assigned to any TOPSPIN command, macro, AU program or Python program. They are specific for the dimensionality of the active dataset.

To create a user defined menu, take the following steps:

1. Open a dataset of the desired dimensionality.
2. Right-click at an empty area of the menubar.
3. In the appearing popup menu, click:

User_defined Menus



4. In the appearing dialog box (see Fig. 19.5).
 - a) Click **Add Menu** and specify the **Menu Name** in the appearing dialog.
 - b) Click **Add Menu Item** and specify the **Menu Item** name and the corresponding **Command** in the appearing dialog.
 - c) Click **Apply**.

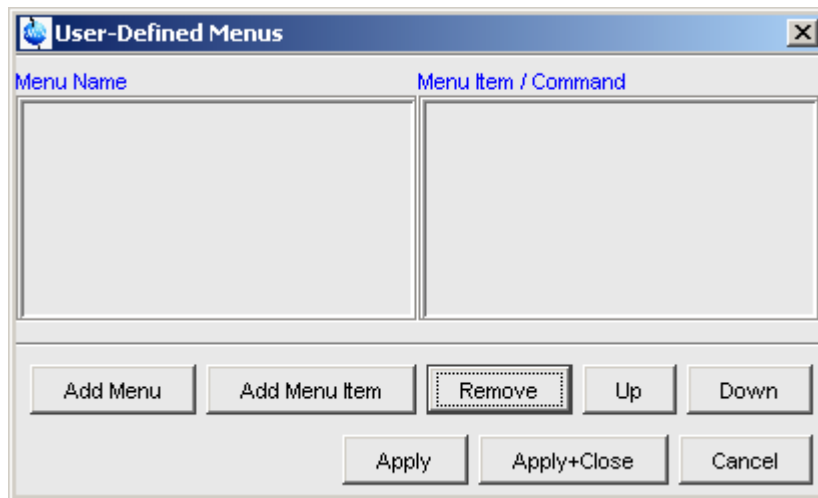


Figure 19.5

The new menu will appear in the menubar.

Furthermore the dialog box in Fig. 19.5 contains the following buttons:

- **Remove** : Remove menu names and/or menu items
- **Apply+Close** : Apply any changes and close the dialog box
- **Up/Down** : Move up/down the menu name list
- **Cancel** : Close the dialog box discarding any changes.

User defined menu definitions are stored in the file `umbar_menubar1d.prop`, `umbar_menubar2d.prop` or `umbar_menubar3d.prop` depending on the data dimensionality. These files resides in the subdirectory `userdefined` of the

user properties directory. To locate this directory enter the TOPSPIN command *hist*.

19.8 Adding User Defined Guides

TOPSPIN offers several guides like the Acquisition Guide (command *agguid*), the Processing Guide (command *prguid*) and the T1/T2 Guide (command *tlguid*). You can set up your own guides, which can be adaptations of Bruker guides or new written ones. In order to do that, you must edit the file `toolbar_user.prop` and `cmdtab_user.prop` in the directory:

```
<user properties directory>/userdefined/
```

To identify this directory, enter the command *hist* in TOPSPIN.

The file `toolbar_user.prop` contains the guide definition. If the file does not exist, it must be created. Here is an example of a user-modified T1/T2 guide.

```
MyT1T2Toolbar=\
NM=t1fid_40.gif, NM2=$Extract fid, TIP=$Do rser,
CMD=_t1_fid, END=,\
NM=t1spec_40.gif, NM2=$Transform, TIP=$Do ef+apk,
CMD=_t1_spec, END=,\
NM=t1ranges_40.gif, NM2=$Ranges, TIP=$Enter integ. mode,
CMD=_t1_ranges, END=,\
NM=t1screen_40.gif, NM2=$Relax. Window, TIP=$Enter Relax
mode, CMD=_t1_relax, END=,\
NM=-, END=,\
NM=myicon.gif, NM2=$Fit Methods, TIP=$Select fit funcs.,
CMD=_t1_func, END=,\
NM=c:/myicons/myicon1.gif, NM2=$Start, TIP=$ct1,ct2, sim-
fit, CMD=_t1_start, END=,\
NM=c:/myicons/myicon2.gif, NM2=$Show Report, TIP=$report,
CMD=_t1_report, END=,\
NM=t1print_40.gif, NM2=$Print it, TIP=$print,
CMD=_t1_export, END=
```

Notes to this guide definition:

- The original Bruker guides are defined in the file `toolbar.prop` in the directory `<tshome>/classes/prop`. The corresponding commands are defined in `cmdtab_main.prop` in the same directory.
- The bold lines in the example above are user-modified lines.
- `NM=t1fid_40.gif`: a Bruker defined icon with a size of 40x40 pixels
- `NM=myicon.gif`: a user-defined icon, which must be located in the user properties directory (Caution: not in its subdirectory `userdefined`)
- `NM=C:/myicons/myicon1.gif`: a user-defined icon located `C:/myicons`. Using the absolute pathname allows you to store icons in an arbitrary directory.
- `NM=-` : indicates the start of the second icon column
- `NM2=` : the text to appear underneath of the icon. The \$ sign is mandatory.
- `CMD=` : the command to be executed when the icon is clicked. This can be a regular TOPSPIN command, a macro or an AU or Python program.
- `TIP=` : the tooltip to be displayed when the cursor is held over the icon. Note that the \$ sign is mandatory.

Before you can start a user defined guide, you must edit the file `cmdtab_user.prop` and define the corresponding command in the file, for example:

```
myt1guide=EM=J, MC=N, CL=tutor.TutStarter, ME=startTutorial,AR=MyT1T2Toolbar;My T1 T2 Tutorial=
```

Here:

- "MyT1T2Toolbar" is the toolbar identifier as it is used in the file `toolbar_user.prop`.
- "My T1 T2 Tutorial" is the title as it appears at the top of the guide.

If your guide is a Bruker modified guide, you can also redefine the original Bruker command, i.e. specify `t1guide` instead of `myt1guide`. Note that the original Bruker guide is then no longer accessible.

If you want to access a user defined guide from the TOPSPIN toolbar, you have to create a new toolbar button. To do that, right-click in an empty area of the toolbar and define a button in the appearing dialog.

Index

Symbols

- *2 command 31, 37, 104, 135
- *8 command 31, 104, 135
- .2d command 30
- .3d command 30
- .all command 32, 105, 137
- .basl command 182
- .bmp files 22
- .bzip files 23, 66
- .cal command 174
- .co command 148
- .dec command 164, 169, 185, 199
- .dopt command 127
- .dx files 66
- .emf files 22
- .f1r command 137
- .f2r command 137
- .gif files 337
- .gr command 147
- .hr command 32, 105
- .hz command 124, 144
- .im command 151
- .inc command 164, 169, 185, 199
- .int command 165
- .jpeg files 22, 337
- .jpg files 22, 337
- .keep command 32, 105, 137
- .ls command 151
- .lt command 153
- .lv command 149
- .md command 21, 175
- .ov command 125, 144
- .ph command 39, 161, 193
- .ph180 command 163
- .ph90 command 163
- .phr command 163
- .png files 22, 99, 337
- .pp command 188

- .pr command 21, 145
- .prop files 54, 218
- .ret command 164, 173, 182, 185, 187, 191
- .retsab command 187
- .s2d command 164
- .sd command 33, 106, 137
- .sino command 222
- .sl command 32, 106, 137
- .sl0 command 32, 106
- .sr command 32, 106, 137
- .sr0 command 32, 106
- .sret command 164, 173, 185, 191
- .st command 151
- .su command 33, 106, 137
- .tif files 99
- .txt files 130
- .vr command 32, 104, 135
- .wmf files 22, 99
- .y command 125
- .zi command 32, 105, 136
- .zl command 32, 105, 137
- .zo command 32, 105, 136
- .zx command 32, 105, 136
- /2 command 31, 37, 104, 135
- /8 command 31, 104, 135
- ? command help 40

Numerics

- 180 degree phase correction
 - 1D 163
 - 2D 199
- 90 degree phase correction
 - 1D 163
 - 2D 199
- 90 degree phase correction
 - 1D 163
 - 2D 199

A

abs command 87, 90, 164, 165, 167, 182
absd command 182
absorption mode 162, 163, 197, 198, 199
acqu command 258
acquisition
 commands 258, 260
 mode 301
 parameters 82, 83, 269
 status bar 259, 271
 toolbar 257
Acquisition Reference Manual 19, 85, 264, 265, 302
AcquPars tab 82, 108, 140, 269
activate
 a data window 213
 the next data window 218
add
 comment to audit file 77
 data to portfolio 74
 directory to the browser 20, 53
 found data to portfolio 54, 74
 functionalities to Topspin 25
 peak in 2D phase correction 194
 peak to peak list 190
add increment in 2D levels 149
align
 intensities in multiple display 180
 peak positions in multiple display 180
Alt-F11 key 104, 134
Alt-F4 key 36, 41
Alt-F6 key 41
Analysis menu 23, 29
anisotropy 225
apk command 87, 90, 161
apks command 161
arrange
 data windows 214
 data windows horizontally 216
 data windows vertically 215
 rows/columns in 2D phase correction 200
ascii files 75, 130
ased command 84, 108
Aspect 3000 data 65
ATM probehead 261, 262
atma command 262

AU macros 25, 330, 331
AU program reference manual 91
AU programs 19, 25, 90, 303, 330, 334, 337
audit
 trail check 76
 trail files 76
audit command 76
auditp.txt file 76
automate data acquisition 26
automatic
 1D baseline correction 87
 1D calibration 87, 173
 1D integration 164
 1D peak picking 187
 1D phase correction 87, 161
 2D calibration 206
 allocation of TCP/IP ports 309
 compilation of an AU program 304
 configuration of a datastation 301
 selection of the first expno/procno 80
 startup actions 315
 tuning and matching 262
automatic mode
 of the Acquisition Guide 256
 of the Processing Guide 23, 88
automation 25, 26, 303
autoplot command 90, 96
autostart.prop file 314
Avance spectrometers 16, 301
axis units 31, 124, 144, 175, 207

B

background of a data window 318
Backspace key 33
backward peak picking 190
baseline correction
 1D 87, 182
 mode 30, 182
bcm command 185
bias correction 168
bio-molecular experiments 26
bnmr command 332
bnmrsl command 332
bpan command 333
browser 27, 28, 36, 49, 59

- font 321
- in multiple display mode 177, 202
- Bruker
 - AU programs 25, 303
 - data format 19, 23, 24, 62, 65, 66, 74
 - example datasets 20, 53
- BSMS control panel 264
- BSMS display window 218
- bsmsdisp command 264
- C**
- calibration 87
 - 1D interactive 173
 - 2D interactive 206
 - mode 30, 173, 206
- cf command 18, 43, 299, 300, 301
- chemical shift 180
- chemical shift distance
 - 1D 221
 - 2D 208
- CHEMSTR parameter 122
- C-language 25, 90, 330, 331
- clevels file 151
- clipboard 22, 36
- close
 - active data window 217
 - lock display window 263, 264
 - the active data window 36
 - wobble window 262
- close button 29
- close command 36
- cmdhist command 34
- cmdindex command 40
- cmdtab_user.prop file 41
- collapse a data directory 49, 52
- color scheme 318
- colors
 - of data objects 317
 - of the lock display 317
 - on the printer 317
- command
 - audit 76
 - definition 41
 - dialog box 79
 - help 39
 - index 40
 - interpreter 309
 - line 33, 34, 322
- command line
 - commands 258
 - focus 33, 36
 - font 322
 - history 33, 34
 - preferences 322
 - resize 323
 - usage 33
- Command Prompt 18, 300
- compile
 - AU program 304
- components
 - of a spectrum 22
- composite commands 88
- configuration
 - commands 18
 - customized 301
 - default 301
 - directory 301
 - name 301
 - of a datastation 18
 - of a firewall 309
 - of a spectrometer 18
 - of TCP/IP communication ports 309
 - of the acquisition status bar 259
 - of the tab bar 316
 - of the Topspin menu 325
 - password 299
- consistency check 76
- contour display 194
 - of 2D spectra 148
 - of 3D planes 156
- contour levels 134, 149
- control keys 36
- convdta command 67
- conventions in this document 15
- convert
 - data to JCAMP-DX 67
 - data to text 67
 - data to ZIP 67
- copy & paste data 29, 64, 66
- copy command 22, 30, 99

- copy data 66
 - CORBA 309
 - corba.conf file 309
 - C-program 90
 - create
 - an AU program 25, 91, 304, 330
 - an empty data window 59
 - data window color scheme 318
 - dataset 30
 - macro 25, 330
 - new data window 19
 - new dataset 56, 268
 - plot layouts 22, 97
 - Python program 26, 331
 - set of user preferences 313
 - user defined command 25
 - user notebook 329
 - C-statements 25
 - Ctrl key 52, 54, 60, 71, 74, 104, 135, 176, 178
 - Ctrl-c key 33, 36, 114
 - Ctrl-d key 27, 36, 51
 - Ctrl-f key 36
 - Ctrl-F3 key 41
 - Ctrl-F5 key 41
 - Ctrl-F6 key 41
 - Ctrl-n key 30, 36, 56, 268
 - Ctrl-o key 30, 36, 61, 65
 - Ctrl-p key 22, 30, 36, 97, 114
 - Ctrl-s key 23, 30, 36, 67, 68
 - Ctrl-v key 34, 36
 - Ctrl-w key 36
 - cube display 155
 - cubic spline baseline correction 185
 - cursor information 127
 - customized configuration 301
 - cut integrals 172
- D**
- data
 - area 20, 27, 59, 64, 66, 211, 214, 217
 - compression 24
 - dimensionality 28, 81, 84, 108
 - directory 49, 53, 71
 - field 33, 101, 106, 133
 - object colors 317
 - data window 19, 27, 29, 59, 64, 66
 - 2D 133
 - 3D 155
 - color scheme 318
 - contents 22, 99
 - creation 59
 - handling 29, 211
 - in 1D peak picking mode 188
 - in 1D phase correction 162
 - in 2D calibration mode 207
 - in 2D multiple display mode 201
 - in 2D phase mode 194
 - in baseline correction mode 183
 - in calibration mode 174
 - in integration mode 165
 - in multiple display mode 177
 - in S/N measurement mode 222
 - in spline baseline correction mode 186
 - objects 22
 - popup menu 126, 138
 - printing 95
 - reopen 102
 - tab bar 1D 106
 - tab bar 2D 139
 - tab bar 3D 159
 - toolbar 21
- dataset
 - active 213
 - colors 182
 - dir 19
 - directory tree 19, 63
 - expno 19, 50, 56
 - files 75, 130
 - handling 25, 49
 - last 2D 30
 - last 3D 30
 - last used 315
 - name 19, 50, 56, 64
 - procno 19, 50, 56
 - procno directory 63
 - properties 73, 128
 - selection in multiple display 178
 - specification 63, 213
 - title 57
 - top level directory 50

- user 19, 50, 56
- variables 68
- datastation
 - configuration 301
- dcon command 24, 326
- dconpeaks.txt file 24
- deconvolution 24
- default
 - color scheme 318
 - command line size 322
 - configuration 301
 - find criteria 72
 - menu font 320
 - pivot point in 1D phase correction 162
 - pivot point in 2D phase 197
 - plane in 3D display 155
 - port numbers 309
 - printer 96
 - Tab bar 316
 - zero order phase correction 163
- define
 - cubic spline baseline points 185
 - integral regions 166
 - NMR ADMINISTRATOR password 299
 - NMR SUPERUSER 299
 - peak ranges 188
 - peaks 190
- de-iconify
 - data window 217
- delete
 - AU program 304
 - data 68
 - integrals from display 170
 - noise region S/N 223
 - peaks from a peak list 190
 - signal region S/N 223
 - spline baseline points 186
- Delete key 33
- depth cueing in 3D 159
- deselect
 - data in multiple display 178
 - integral regions 166
- dialog box font 321
- difference spectrum
 - in baseline correction 184
 - in multiple display 179
- dimensionality of a dataset 28, 81, 82, 84, 108
- dipolar coupling 229
- dir command 20
- disable
 - commands 324
 - menus 324
 - toolbar buttons 324
- Display
 - button 19, 65, 73
 - menu item 59, 176
- display
 - 1D acquisition status parameters 108
 - 1D data 101
 - 1D FID 124
 - 1D integral list 115
 - 1D peak list 110
 - 1D processed data 107
 - 1D raw data 124
 - 1D spectrum 107
 - 1D spectrum overview 124
 - 2D contours 148
 - 2D data 133
 - 2D FID 143
 - 2D grid 147
 - 2D integral list 142
 - 2D peak list 141
 - 2D projections 145
 - 2D spectrum in contour mode 148
 - 2D spectrum in image mode 151
 - 2D spectrum in oblique mode 151
 - 3D cube 157
 - 3D data 155
 - columns in 2D phase correction 194
 - contours in 2D phase 194
 - data from the browser 20, 60
 - data from the portfolio 60
 - data in multiple windows 102
 - dataset file list 128
 - dataset properties 128
 - expno/procno list 60
 - found dataset 73
 - full spectrum in 2D phase correction 195
 - integrals 20
 - JCAMP data 24

- manipulation 103
- manipulations 31
- mode of the lock window 263
- molecule structure 122
- options 31
- options 1D 124
- peaks 20
- planes of 3D data 155, 156
- positive/negative 2D levels 153
- processing status parameters 1D 108
- projections 21
- rows in 2D phase correction 194
- settings 30
- special data formats 65
- spectra 19
- spectrum overview in 2D 144
- status parameters 81, 84
- sum/difference spectrum 179
- superimposed 1D spectra 175
- superimposed 2D spectra 200
- y-axis 31, 125
- zipped data 23
- display properties 21
- distance measurement
 - 1D 221
 - 2D 208
 - mode 31
- DORR parameter 228
- double rotation experiments 228
- double-headed arrow 212, 222
- Down-Arrow key 33, 37
- dpa command 108
- dpp command 81, 84, 108
- drag & drop data 64, 66

E

- eda command 82, 83, 108, 140
- edasp command 85
- edau command 25, 91, 303, 304, 330
- edcpul command 109
- edit
 - AU program 304
 - commands 34
 - contour levels 149
 - integral ranges 168

- pulse program 109
- signal/noise regions 223
- title of a dataset 109
- Edit menu 29
- edlev command 134, 149
- edlock command 85
- edmac command 25, 330
- edp command 23, 79, 80, 107, 140
- edprosol command 84
- edpy command 26, 331
- edti command 109
- ef command 88, 90
- efp command 88, 271
- em command 23, 87
- email data 24
- empty data window 59
- enable
 - disabled commands 326
 - menus/commands 323
- Enter key 37, 60
- equidistant sequence of levels 149
- Esc key 33
- Euler angle 234
- execute
 - AU program 303, 304
 - macro 25, 330
 - Python program 26
- exit command 36, 42
- expand
 - a data directory 37
 - data directory 19, 20, 49
 - data directory fully 52
 - data directory in the browser 52
 - individual spectra in multiple display 180
 - spectrum 24
- experiment time 258
- expinstall command 18, 43, 258, 299, 300, 301, 303
- expl command 63, 75, 76, 130
- Explorer 49, 63, 75, 130
- expno 56, 60, 63, 68, 128
- exponential
 - baseline correction 184
 - window multiplication 87, 88
- export data 22, 99
- expt command 258

extend the Topspin functionality 329
external projection 21
extract a row/column in 2D 205

F

F1 dimension 137, 144, 193
F1 key 15, 36
F1-F2 plane 156, 159
F2 dimension 137, 144, 193
F2 key 36, 51
F2-F3 plane 157, 159
F3 key 41
F3-F1 plane 155, 157, 159
F5 key 41
F6 key 36, 41, 214
FDA 76
fid command 124, 143
FID display 258, 260, 266, 269
Fid tab 124, 128, 143
File Chooser 19
File menu 19, 29
file size of raw data 258
files of a dataset 64, 67, 75, 128, 130
find command 36, 54, 71
find data 29, 54, 71
firewall 309
first order phase correction
 1D 163
 2D 197, 198
fitting peaks 24
fmc command 88
focus 33, 36, 51
font
 of dialog windows 321
 of the browser 321
 of the command line 322
 of the interface 319
 of the menu 320
 of the status line 322
 of the tab bar 320
forward peak picking 190
Fourier transform 87, 88
fp command 88
fromjdx command 24
fromzip command 23

ft command 23, 87
ftf command 23, 325
function keys 36

G

Gaussian deconvolution 24
Gaussian window multiplication 88
genfid command 67, 68
genser command 68
geometric sequence of levels 149
geometry of a Topspin window 218, 219
getprosol command 84, 108
gf command 88
gfp command 88
GLP 76
GMP 76
Gnome Mozilla 76
grab rows/columns in 2D 204
gradient enhanced spectroscopy 25
graphic functions 331
graphics files 22, 99
grid
 display 31, 147, 263
 window arrangement 214
gs command 82, 266
GS parameter adjustment window 266

H

halt an acquisition 258, 259
halt command 258, 270
helium level 264
Help
 button 40
 menu 29, 39
 menu of the Plot Editor 22, 97
help
 in Topspin 39
 on commands 40
help command 40
Hertz axis units 124, 144
hist command 41, 44, 99, 308, 314, 333, 337, 339
history file 337
hostname of a remote system 308
HTML page 40

I

iconify
 all data windows 218
 data window 216
iconnmr command 26
image display
 of 2D data 151
 of 3D planes 156
import data 24
inconsistent dataset 81
initial guess 240
install
 AU programs 19, 303
 pulse programs 19
 Topspin 299
installation directory 53, 300, 301
installnmr script 300
int command 127
Integral
 list 2D 142
integral
 bias 168
 display 20
 label 167
 list 1D 115
 regions 165, 166, 167, 172
 scaling factor 171
 slope 169
 trails 127, 172
 values 165
Integrals tab 115, 142, 173
integrals.txt file 173
integration
 1D automatic 164, 167
 1D interactive 164, 167
 mode 31, 165, 173, 295
intensity
 alignment in multiple display 180
 decrease 1D 104, 134
 increase 1D 104, 134
 manipulation 31
 reset 1D 32, 104, 105, 134
 scaling 2D 135
interactive
 1D baseline correction 182

 1D calibration 173
 1D data manipulation 161
 1D integration 164
 1D peak picking 187
 1D phase correction 161
 1D signal to noise calculation 222
 2D calibration 206
 2D data manipulation 193
 2D phase correction 193
 data manipulation 23, 30, 316
 modes 30
 parameter adjustment 82, 266
 processing 87
 Processing Guide mode 90
interface fonts 319
internal projection 21
Internet Browser 64
intrng file 168, 171, 173
IP address of a remote system 308
IUPAC standard 24

J

JCAMP-DX format 24, 65, 66, 67
Jeol spectrometer 66
Jmol molecule structure viewer 122
JNMR data 66

K

KDE konqueror 75
keyboard commands 29
kill command 47
Konqueror 76, 130

L

Left-Arrow key 33, 37, 52
Levenberg-Marquardt algorithm 239
li command 115, 164
Liouville equation 25
lipp command 115
lippf command 115
list 142
 1D integrals 115
 2D integrals 142
 2D peaks 141
 AU programs 303

- baseline points 186
- Bruker AU programs 304
- color schemes 318
- data files 74, 128
- EXPNO/PROCNO 60
- found data 72
- integrals 97, 173
- macros 25
- opened datasets 50
- parameter sets 57
- peaks 98, 127, 190
- peaks 1D 110
- plot layouts 96
- solvents 57
- user defined AU programs 303
- local
 - host 309
 - network 307
 - Topspin interface 307
- lock
 - parameters 85
 - signal 263
- lock command 264
- lock display
 - colors 317
 - mode 263
 - window 218, 258, 260, 263
- lockdisp command 258, 263
- lock-in procedure 263
- Lorentzian deconvolution 24
- lower toolbar 31
- M**
- macros
 - in AU programs 90, 331
 - in Topspin 25, 41, 88, 330, 334, 337
- magnitude calculation 88
- managuide command 241
- MASR parameter 228
- matching the probehead 261
- MAXI parameter 190
- maximize
 - all data windows 218
 - data window 217
 - the Topspin window 29
- maximize button 29
- maximum
 - command line size 323
- MC2 parameter 81
- menu
 - bar 29
 - commands 29
 - configuration table 325, 326
 - entries 29, 325
 - font 320
 - settings 313
- MI parameter 190
- minimize
 - data window 216
- minimize button 29
- minimum
 - command line size 323
- Minus key 37
- mixtures 25
- molecule structure viewer 122
- mouse sensitivity
 - in 1D baseline correction 185
 - in 1D integration 169
 - in 1D phase correction 163
 - in 2D phase correction 199
- move
 - data window 212
 - Topspin window 29
- Mozilla 76, 130
- multiple display
 - 1D 31, 175, 201
 - 1D/2D 21, 60, 65
 - 2D 21, 200
 - in deconvolution 24
- multiple display mode 60, 175
- multiple window display 102
- multiplet 166, 237
- multiplet analysis 241
- multiply with increment in 2D levels 149
- N**
- nbook command 329
- negative
 - 2D levels 153
- new command 30, 36, 56, 268

- newtop command 41
- next
 - channel for wobbling 262
 - command 37
 - dataset in Browser 37
 - parameter field 81, 83
 - plane in 3D 157
 - row/column in 2D phase correction 199
 - window in data area 36, 214, 218
- NMR ADMINISTRATION password 299
- NMR SUPERUSER 299
- nmradminpassword file 299
- nmrsim command 25, 110
- nmrsuperuser file 299
- noise region 222
- NOISF1 parameter 222
- NOISF2 parameter 222
- notebook 329
- NS parameter 258
- O**
- O1 parameter 259
- O2 parameter 259
- O3 parameter 259
- objects
 - of a dataset 22, 102, 212, 313, 316, 317, 318
 - of Topspin 309
- oblique display
 - of 2D spectra 151, 153
 - of 3D planes 156
- online help 39
- online manual
 - plot editor 97
 - Topspin 39
- open
 - browser/portfolio 51
 - data 30, 36, 49, 59, 60, 61
 - data from the browser 19
 - data from the command line 64, 66
 - data from the Explorer 63
 - data from the menu 19
 - IconNmr interfaces 26
 - Linux Shell 18
 - new data window 214
 - new portfolio 53

- new procno 64
- online help documents 39
- special format data 65
- title bar menu 29
- Topspin command index 40
- open command 30, 36, 61, 65
- Options menu 29
- overview spectrum
 - 1D 124
 - 2D 144

P

- paper format 301
- paracon command 302
- parameter
 - adjustment window 82, 266
 - change 81, 84
 - display 20
 - editor 79, 80, 82, 83
 - field 81, 83
 - files 74
 - handling 79
 - name 79, 82
 - search 108
 - value 81, 83
- parameter set 57, 67
- PARMODE parameter 108
- paste command 30, 64, 66
- peak
 - alignment 21, 180
 - display 20
 - fitting 24
 - group 24
 - labels 127, 190
 - list 190, 191
 - list 1D 110, 190
 - list 2D 141
 - picking mode 187
 - position 221
 - position alignment 180
 - ranges 191
- peak picking
 - 1D automatic 187
 - 1D interactive 187
 - mode 31

- mode 1D 188
 - ranges in 1D 189
- peak.txt file 191
- peakrng file 191
- Peaks tab 98, 110, 141
- phase correction
 - 1D automatic 87, 88
 - 1D interactive 161
 - 2D interactive 193
 - first order 1D 163
 - mode 30
 - mode 1D 162
 - mode 2D 194
 - pivot point in 1D 162
 - values 1D 164
 - zero order 1D 163
- PHC0 parameter 81, 163, 198
- PHC1 parameter 163, 199
- pivot point
 - in 1D phase correction 162
 - in 2D phase correction 197
- pk command 164
- plane display in 3D 156
- plot
 - data from the menu 95
 - data from the Plot Editor 97
 - data from the Processing Guide 97
 - layouts 22
- plot command 22, 96, 97
- Plot Editor 22, 96
- Plus key 37
- polynomial
 - baseline correction 183
- port numbers 309
- portfolio 20, 36, 49, 54, 60, 74, 202
 - file 54
 - in multiple display mode 177
- position
 - of a Topspin window 219
- position of a Topspin window 218
- position the baseline of a row/column in 2D 206
- positive
 - 2D levels 153
- pp command 114, 127, 208
- PPARMOD parameter 81, 84, 108
- ppl command 189
- ppm axis units 124, 144
- pps command 187
- preferences 29, 313
- pre-scan-delay 301
- preview
 - the baseline corrected spectrum 184
- previous
 - commands 33
 - dataset in browser 37
 - parameter field 81, 83
 - plane in 3D 157
 - row/column in 2D phase correction 199
 - zoom 32, 105, 137
- print
 - 1D peak list 114
 - active window 22, 96
 - colors 317
 - data from the menu 95
 - data from the Processing Guide 97
 - integral list 97
 - metafiles 99
 - peak list 98
 - the current dataset 36
- print command 30, 36, 95, 97, 114
- printer colors 317
- prnt command 96
- probehead/solvent dependent parameters 84
- proc_1d AU program 90
- process data 22, 29
 - from the command line 87
 - from the Processing Guide 88
 - with AU programs 90
 - with composite commands 88
- processing
 - parameters 79, 80
 - parameters 1D 107
 - parameters 2D 140
- Processing Guide 23, 88
- Processing menu 23, 29
- Processing Reference Manual 20, 39
- procno 60, 64, 68, 75, 129
- ProcPars tab 23, 79, 80, 107, 140
- projections of a 2D spectrum 21, 145
- properties

- of a dataset 73, 128
- of a printer 96
- prosol parameters 269
- pseudo raw data 68
- pulse program 19, 24, 52, 109
 - display 110
 - parameters 84, 108, 269
- PulsProg tab 109
- Python programs 26, 331, 334, 337

Q

- QUIT AU macro 91
- QUITMSG AU macro 91

R

- re command 20, 64, 176
- read
 - color scheme 318
 - data formats 65
 - integrals from disk 167
 - the prosol parameters 269
 - window layout 218
- reb command 19, 65
- re-enable disabled commands 326
- reference peak
 - in 1D calibration 174
 - in 1D phase correction 162, 163
 - in 2D calibration 207
 - in 2D phase correction 197
- referencing 87
- reg file 173
- relaxation
 - curve 235
- remote connections 313
- remote control 307
- remove
 - commands 324
 - data from multiple display 178
 - datasets from the portfolio 54
 - menus 324
 - toolbar buttons 324
 - top level directory from the browser 53
- reopen a dataset 102
- reopen command 58, 102
- rep command 20, 64

- replace the current portfolio 74
- repw command 64
- rescale
 - 2D projection 146
 - data in multiple windows 103
- reset
 - 1D baseline correction 185
 - 1D phase values 163
 - 3D cube size and orientation 158
 - default find criteria 72
 - F1 zoom factor in 2D 137
 - F2 and F1 zoom factor 137
 - F2 zoom factor in 2D 137
 - intensity 1D 104
 - intensity in 2D 135
 - search mask 72
 - zoom factor 32, 105
- resize
 - command line 323
 - data window 212
 - graphics 99
- resolution of a screen dump 22, 99
- restore
 - adjusted acquisition parameters (GS) 268
 - size and position of a data window 217
 - toolbar 324
 - uncorrected baseline 185
- retrieve
 - previously entered commands 33
- rew command 20, 64
- Right-Arrow key 33, 37, 52
- rotate 3D cube 153, 158
- routing parameters 85

S

- S/N value 224
- Sample tab 121
- sav command 30, 36
- save
 - 1D baseline correction 184, 185
 - 1D integrals 173
 - 1D phase correction 164
 - acquisition data 68
 - adjusted acquisition parameters (GS) 267
 - AU program 25, 304

- color scheme 318
- cubic spline baseline points 187
- current window layout 218
- data 23, 30, 36, 66
- data in analog filtered format 67
- data to a JCAMP-DX file 24
- data to a ZIP file 23
- data window to a graphics file 22, 99
- entire dataset 66, 67
- integral regions 172
- macro 25
- parameters 108
- peak list and peak ranges 191
- phase correction 2D 197, 200
- portfolio 54
- processed data 68
- processed data as pseudo raw data 68
- pulse program 110, 121
- sum or difference spectrum 179
- title of a dataset 109
- scale
 - 1D integrals 171
 - 1D spectrum 104
 - 2D spectrum 134
 - 3D cube 158
 - individual spectra in multiple display 21, 175, 180
 - row/column in 2D phase 198
- scaling factor
 - of integrals 171
- scan
 - planes of 3D data 157
 - rows/columns in 2D 203
- screen dump 22, 99
- screenDump.wmf file 99
- scroll bar 81, 84
- search
 - criteria 54, 71, 73
 - data 71
 - field in the parameter editor 81, 83
 - result window 73
- select
 - AU program 304
 - color scheme 318
 - data 49
 - data in multiple display 176, 178
 - data window 213
 - default printer 301
 - expno/procno combination 60
 - first expno/procno 60
 - folders in the browser 51
 - font 320, 321, 322
 - integral regions 166, 168
 - lock signal color 318
 - multiple datasets 60, 74
 - multiple folders 51
 - peaks in 2D phase correction 194
 - plot layout 96
 - printer 96
 - remote system 308
 - row/column in 2D phase 197
 - spectra in 2D multiple display 203
 - spectrum color 317
 - spectrum print color 317
 - Topspin window 219
- selective excitation 25
- semi-automatic
 - peak picking 190
 - processing 88
- sensitivity
 - of the GS slider 266
 - of the mouse 163, 169, 185, 199
- serial command 91
- serial processing 91
- Server ID 307
- set
 - 1D acquisition parameters 108
 - 2D acquisition parameters 140
 - 2D processing parameters 140
 - colors for multiple display 182
 - contour levels 149
 - lock parameters 85
 - phase pivot point 162
 - processing parameters 79
 - processing parameters 1D 107
 - routing parameters 85
 - user preferences 314
- set command 106, 313
- SFO1 parameter 259
- Shell 18

- shift
 - 1D data down 106
 - 1D data smoothly 106
 - 1D data to the extreme left 106
 - 1D data to the extreme right 106
 - 1D data to the left 106
 - 1D data to the right 106
 - 1D data up 106
 - 1D/2D data 32, 37
 - 2D data down 137
 - 2D data smoothly 137
 - 2D data to the left 137
 - 2D data to the right 137
 - 2D data up 137
 - data in multiple windows 103
 - individual spectra in multiple display 21, 175, 180
 - row/column in 2D phase 198
- Shift key 52, 60, 71, 74, 176, 178
- Shift-Tab key 81, 83
- short description
 - of an AU program 305
- shortcuts for processing 88
- show command 47
- SIGF1 parameter 222
- SIGF2 parameter 222
- signal region 222
- signal to noise calculation 222
- Simplex algorithm 235, 239
- simulation of powder spectra 225
- simulating experiments 24
- sine
 - baseline correction 183
- single commands 87
- slider sensitivity 266
- slope correction 168
- smail command 24, 30
- smooth
 - 1D phase correction 198
 - 1D scaling 32, 104
 - 1D shifting 105
 - 1D zooming 32, 104
 - 1D/2D shifting 32
 - 2D scaling 135
 - 2D shifting 137
 - scaling of 1D integrals 171
 - scanning of planes in 3D 157
 - shifting of 1D integrals 172
- solaguide command 225
- Solids Line Shape Analysis 225
- solvent 57
- sort data in portfolio 54
- sort mode 55
- spec command 107
- special format data 24
- spectrometer
 - frequency 259
- spectrometer hardware 18, 19
- Spectrometer menu 29, 257
- spectrum
 - display 1D 107
 - objects 313, 314
 - overview 1D 124
 - overview 2D 144
- Spectrum tab 107, 124, 129
- split pane 27
- square 2D layout 135
- square brackets 29, 39, 102, 106
- sref command 87, 90, 173, 206
- stacked multiple display 180
- Start button 63
- startup actions 313, 315
- status line font 322
- status parameter
 - change 81
 - display 84, 127
 - display 1D 108
- stop an acquisition 258
- stop command 258, 270
- store
 - 2D contour levels 150
- structure file 122
- Structure tab 122
- sum spectrum
 - in multiple display 179
- superimpose
 - 1D spectra 175
 - cursor information 127
 - integral trails/labels 127
 - main status parameters 127

- peak labels 127
- spectra 21
- title of a dataset 127

SW parameter 259

swap data windows 219

sweep width 259

swin command 219

switch

- to the last 2D data 30
- to the last 3D data 30

synchronized experiments 228

T

T1 calculation 16

T1/T2 Relaxation 235

t1guide command 235

T2 calculation 16

Tab bar

- 1D 102, 106
- 2D 134, 139
- 3D 159
- configuration 106, 316
- font 320
- usage 106

Tab key 81, 83

tabbed pane 139, 159, 316

TCP/IP port 308

temperature unit window 218

Terminal Window 18

time domain signal 25

title bar 29, 101, 102, 133, 212, 213, 216, 217, 218, 304

title file 109

title of a dataset 20, 52, 57, 109, 127, 268, 318

Title tab 109

tojdx command 24, 67

toolbar

- 1D 161
- 2D 193
- configuration 324
- extension 334
- for acquisition 257
- icons 337
- of the data window 21, 124
- of the FID display window 270

- tips 39
- usage 30

tooltips 39

top level directory 20, 49, 53

Topspin

- color schemes 318
- communication 309
- data area 64, 66
- fonts 320
- help 39
- installation 301
- installation directory 301
- portfolio 50
- startup 17, 18
- tailoring 313
- window 27

topspin command 18

Topspin icon 18

totxt command 67

tozip command 23, 67

tshome directory 18, 39, 301, 309

tuning the probehead 261

U

undo

- last parameter change 81, 84
- last region operation 173

unzip utility 24

Up-Arrow key 33, 37

upper toolbar

- 1D 30, 161
- 2D 193

user defined

- AU programs 303
- buttons 334, 337, 339
- commands 23, 25
- functions keys 41
- plot layouts 96

User Interface functions 331

user preferences 313

user properties directory 41, 99, 314

User's guide 39

V

Varian spectrometer 66

vertical scaling 104
View menu 29, 49, 126
virtual spectrometer 24
VNMR data 66

W

walk-up 26
wbst command 262
wbsw command 262
wildcards 70
window layout 218
Window menu 29, 219
WINNMR data 65
wm command 23, 79
wmisc command 67
wobb command 261
wobble

- display 260
- frequency 262
- procedure 261
- steps 262
- sweep width 262
- window 261

wpar command 67
wra command 67, 68
wrp command 67, 68
wrpa command 23, 67, 68

X

xau command 304
x-axis rotation in 3D 153, 158
xmac command 25

Y

y-axis display 31, 125
y-axis rotation in 3D 153, 158

Z

z-axis rotation in 3D 158
zero order phase correction

- 1D 163
- 2D 197, 198

zg command 258, 269
ZIP data format 23, 65
zoom

1D data 32, 104
2D data 136
in 2D phase correction 194
reset in 1D 105
reset in 2D 137
zoom 1D/2D data 37