
Ground Loop Design™

ClimateMaster Geothermal Design Studio



User's Manual (English)

GLD™ 2010 Edition for Windows®



www.gaiageo.com



www.climatemaster.com

Copyright Notice

Ground Loop Design™ Premier 2010 User's Guide

© 2011 Celsia, LLC. All Rights Reserved.

This guide, as well as the software described in it, is furnished for information purposes only to licensed users of the GLD software product and is furnished on an "AS IS" basis without any warranties, whatsoever, express or implied. This may be used or copied only in accordance with the terms of the included End-User License Agreement. The information in this manual is subject to change without notice, and should not be construed as a commitment by Gaia Geothermal. Gaia Geothermal assumes no responsibility or liability for errors or inaccuracies that may occur in this book.

Except as permitted by such license, no part of this publication may be reproduced, stored in a retrieval system, or transmitted in any means, electronic, mechanical, recording, or otherwise, without the prior written consent of Gaia Geothermal.

Other brand and product names are trademarks or registered trademarks of the respective holders. Microsoft Excel, Windows, Windows 95, Windows 98, Windows NT, Windows Explorer, Windows ME, Windows XP, Windows 2000 and Windows Vista are registered trademarks of Microsoft Corporation. Netscape Navigator is a registered trademark of Netscape Corporation. GeoCube is a trademark of Precision Geothermal, LLC. Trane Trace is a trademark of the Trane Company. The <Virtual Environment> is a trademark of IES, Inc.

The Ground Loop Design™ Premier 2010 Edition User's Manual
Originally printed in October 2010
Printed in USA
Part No. GGENG-1107

Visit our Web site at <http://www.gaiageo.com>.

Software Versions Available

Three versions of GLD™ are available: the ClimateMaster Standard/Professional Edition, the ClimateMaster Premier Edition and the Residential Edition

The ClimateMaster Standard/Professional Edition is an advanced design suite that enables users to rapidly select heat pump equipment, design vertical, horizontal, pond and hybrid heat exchanger systems, simulate the performance of vertical and vertical/hybrid systems and perform a fast and accurate life cycle costing analysis.

The ClimateMaster Premier Edition includes everything in the Standard/Professional Edition and adds an enhanced and expanded user interface, the patent pending Computational Fluid Dynamics (CFD) Module for automated piping design, 8760 hourly simulations of vertical systems as well as export functions to AUTOCAD and leading building energy simulation tools.

The Residential Edition, a fully featured residential design suite is not described in this manual. More info can be found about the Residential Edition at www.gaiageo.com

END USER SOFTWARE LICENSE AGREEMENT

PLEASE READ THIS END USER SOFTWARE LICENSE AGREEMENT ("End User Agreement") CAREFULLY BEFORE USING THE SOFTWARE.

BY USING THIS SOFTWARE, YOU ARE AGREEING TO USE THE SOFTWARE SOLELY IN ACCORDANCE WITH ITS INTENDED USE AND YOU ARE CONSENTING TO BE BOUND BY THIS END USER AGREEMENT. IF YOU DO NOT AGREE TO ALL OF THE TERMS OF THIS END USER AGREEMENT, PROMPTLY RETURN AND DO NOT USE THE SOFTWARE.

Single User License. Subject to the terms and conditions of this End User Agreement, Celsia, LLC, doing business as Gaia Geothermal ("Gaia") and its suppliers grant to you ("Customer") a non-exclusive, non-transferable dongle-based license to use the GROUND LOOP DESIGN™ software program in object code form and all related materials included herewith, including written materials, binders and other containers (hereinafter the "Software") on supported operating systems.

Use. Upon: a) receipt of full payment by Gaia, or a Gaia authorized reseller of the applicable license fees, Customer will be able to use this Software, pursuant to the limitations set forth herein.

Limitations. Customer's full use of this Software is limited to the number of authorized licenses Customer has purchased. Customer agrees to use reasonable efforts to protect the Software from any unauthorized use, modification, reproduction, distribution and publication. Customer may not transfer any of the rights granted to Customer hereunder, unless Customer receives prior written authorization from Gaia and only if Customer transfers all of Customer's rights granted hereunder, without retaining any of the Software, or any copies thereof or any rights thereto. Except as otherwise expressly provided under this End User Agreement, Customer shall have no right, and Customer specifically agrees not to:

- (i) make error corrections to or otherwise modify or adapt the Software nor create derivative works based upon the Software, or to permit third parties to do the same; or
- (ii) copy, in whole or in part, decompile, translate, reverse engineer, disassemble or otherwise reduce the Software to human-readable form.

Upgrades and Additional Copies. For purposes of this End User Agreement, "Software" shall also include (and the terms and conditions of this End User Agreement shall apply to) any upgrades, updates, bug fixes or modified versions (collectively, "Upgrades") or backup copies of the Software licensed or provided to Customer by Gaia or an authorized distributor for which Customer has paid the applicable license fees and holds the corresponding software keys. Notwithstanding the foregoing, Customer acknowledges and agrees that Gaia shall have no obligation to provide any Upgrades under this End User Agreement. If Upgrades are provided, Customer acknowledges and agrees that: (i) Customer has no license or right to use any such additional copies or Upgrades unless Customer, at the time of acquiring such copy or Upgrade, already holds a valid license to the original Software.

Notices of Proprietary Rights. Customer agrees to maintain and reproduce all trademark, copyright, patent, and notices of other proprietary rights on all copies, in any form, of the Software in the same form and manner that such trademark, copyright, patent, and notices of other rights are included on the Software. Except as expressly authorized in this End User Agreement, Customer shall not make any copies or duplicates of any Software without the prior written permission of Gaia. Customer may make such backup copies of the Software as may be necessary for Customer's lawful use, provided Customer affixes to such copies all trademark, copyright, patent, and notices of other proprietary rights that appear on the original.

Proprietary Rights. Customer shall own the physical media on which the Software is recorded, but the Software is and will remain the sole and exclusive property of Gaia. Gaia's rights under this Section will include, but not be limited to: (i) all copies of the Software, in whole and in part; and (ii) all Intellectual Property Rights in the Software. For purposes herein, "Intellectual Property Rights" means patent rights (including patent applications and disclosures), copyrights (including, but not limited to, rights in audiovisual works and moral rights), trade secret rights, rights of priority and any other intellectual property right recognized in any country or jurisdiction in the world. "Moral Rights" means any right to claim authorship to or to object to any distortion, mutilation, or other modification or other derogatory action in relation to a work, whether or not such would be prejudicial to the author's reputation, and any

similar right, existing under common or statutory law or any country in the world or under any treaty, regardless of whether or not such right is denominated or generally referred to as a “moral right”.

Confidential Information. Customer agrees that Customer will not disclose or, except as expressly permitted in this End User Agreement, use any Software or other technical information disclosed to Customer by Gaia (“Confidential Information”), and that Customer will take all reasonable measures to maintain the confidentiality of all Confidential Information in Customer’s possession or control, which will in no event be less than the measures Customer uses to maintain the confidentiality of Customer’s own information of equal importance. Confidential Information will not include information that: (i) is in or enters the public domain without breach of this End User Agreement; (ii) Customer receives from a third party without restriction on disclosure and without breach of a nondisclosure obligation; or (iii) Customer develops independently, which Customer can prove with written evidence. Customer acknowledges that the Software is a trade secret of Gaia, the disclosure of which would cause substantial harm to Gaia that could not be remedied by the payment of damages alone. Accordingly, Gaia will be entitled to preliminary and permanent injunctive relief and other equitable relief for any breach of this Section.

Limited Warranty. Gaia warrants that the Software will substantially conform to its published specifications for a period of thirty (30) days from the later of receipt of the Software or receipt of access to the Software. Gaia further warrants that the media on which the Software is contained will be free from defects for a period of thirty (30) days from the later of receipt of the Software or receipt of access to the Software. This limited warranty extends only to Customer as the original licensee. Provided that (a) Customer has notified Gaia of such substantial non-conformance or defect during the applicable warranty period, and (b) Gaia has confirmed such Software or media to be substantially non-conforming or defective, as Customer’s sole and exclusive remedy and Gaia’s and its suppliers’ entire liability under this limited warranty, Gaia will, at its option, repair, replace, or refund the Software free of charge. Except as expressly provided in this End User Agreement, the Software is provided “AS IS” without warranty of any kind. Gaia does not warrant that the Software is error free or that Customer will be able to operate the Software without problems or interruptions. Gaia reserves the right to charge additional fees for repairs or replacements performed outside of the limited warranty period.

This warranty does not apply if the Software (i) is licensed for beta, evaluation, testing or demonstration purposes for which Gaia does not receive a license fee, (ii) has been altered, except by Gaia, (iii) has not been installed, operated, repaired, or maintained in accordance with instructions supplied by Gaia, (iv) has been subjected to abnormal physical or electrical stress, misuse, negligence, or accident, or (v) is used in ultrahazardous activities.

If the dongle license key becomes damaged, replacement keys can be obtained for a \$150 fee. To obtain a replacement key for a damaged key, Customer must send the damaged key to Gaia, or a Gaia authorized reseller. In the case of a lost dongle license key, Customer will be charged the full list price of the Software to replace the lost dongle license key.

The authorized distributors of the Software, who are appointed by Gaia, are not permitted to alter the terms of this End User Agreement in any manner.

Disclaimer. EXCEPT AS SPECIFIED IN THIS WARRANTY, ALL EXPRESS OR IMPLIED CONDITIONS, REPRESENTATIONS, AND WARRANTIES INCLUDING, WITHOUT LIMITATION, ANY IMPLIED WARRANTY OR CONDITION OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE, NONINFRINGEMENT, SATISFACTORY QUALITY OR ARISING FROM A COURSE OF DEALING, USAGE, OR TRADE PRACTICE, ARE HEREBY EXCLUDED TO THE EXTENT ALLOWED BY APPLICABLE LAW.

IN NO EVENT WILL GAIA OR ITS SUPPLIERS BE LIABLE FOR ANY LOST REVENUE, PROFIT, OR DATA, OR FOR SPECIAL, INDIRECT, CONSEQUENTIAL, INCIDENTAL, OR PUNITIVE DAMAGES HOWEVER CAUSED AND REGARDLESS OF THE THEORY OF LIABILITY ARISING OUT OF THE USE OF OR INABILITY TO USE THE SOFTWARE EVEN IF GAIA OR ITS SUPPLIERS HAVE BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES. IN NO EVENT SHALL GAIA’S OR ITS SUPPLIERS’ TOTAL LIABILITY TO CUSTOMER, WHETHER IN CONTRACT, TORT (INCLUDING NEGLIGENCE), OR OTHERWISE, EXCEED THE PRICE PAID BY CUSTOMER. THE FOREGOING LIMITATIONS SHALL APPLY EVEN IF THE ABOVE-STATED WARRANTY FAILS OF ITS ESSENTIAL PURPOSE. BECAUSE SOME STATES OR JURISDICTIONS DO NOT ALLOW LIMITATION OR EXCLUSION OF CONSEQUENTIAL OR INCIDENTAL DAMAGES, THE ABOVE LIMITATION MAY NOT APPLY TO CUSTOMER.

Term and Termination. This End User Agreement is effective until terminated. Customer’s license rights under this End User Agreement will terminate immediately without notice from Gaia if Customer fails to comply with any

provision of this End User Agreement. Upon termination, Customer must destroy all copies of Software and the corresponding keys in its possession or control.

Compliance With Law. Each party agrees to comply with all applicable laws, rules and regulations in connection with its activities under this End User Agreement. Without limiting the foregoing, Customer acknowledges and agrees that the Software, including technical data, is subject to United States export control laws, including the United States Export Administration Act and its associated regulations, and may be subject to export or import regulations in other countries. Customer agrees to comply strictly with all such regulations and acknowledges that Customer has the responsibility to obtain licenses to export, re-export, or import the Software.

Restricted Rights. The Software shall be classified as “commercial computer software” as defined in the applicable provisions of the Federal Acquisition Regulation (the “FAR”) and supplements thereto, including the Department of Defense (DoD) FAR Supplement (the “DFARS”). The parties acknowledge that the Software was developed entirely at private expense and that no part of the Software was first produced in the performance of a Government contract. If the Software is supplied for use by DoD, the Software is delivered subject to the terms of this End User Agreement and either (i) in accordance with DFARS 227.702-1(a) and 227.7202-3(a), or (ii) with restricted rights in accordance with DFARS 252.227-7013(c)(1)(ii) (OCT 1988), as applicable. If the Software is supplied for use by a Federal agency other than DoD, the Software is restricted computer software delivered subject to the terms of this End User Agreement and (i) FAR 12.212(a); (ii) FAR 52.227-19; or (iii) FAR 52.227-14 (ALT III), as applicable.

General. This End User Agreement will bind and inure to the benefit of each party’s successors and assigns, provided that Customer may not assign or transfer this End User Agreement, in whole or in part, without Gaia’s written consent. This End User Agreement shall be governed by and construed in accordance with the laws of the State of California, United States of America, as if performed wholly within the state and without giving effect to the principles of conflict of law. No failure of either party to exercise or enforce any of its rights under this End User Agreement will act as a waiver of such rights. If any portion hereof is found to be void or unenforceable, the remaining provisions of this End User Agreement shall remain in full force and effect. This End User Agreement is the complete and exclusive agreement between the parties with respect to the subject matter hereof, superseding and replacing any and all prior agreements, communications, and understandings (both written and oral) regarding such subject matter.

Conventions Used in This Document

The following symbols are used in this document to highlight certain information and features included in the **User’s Guide** and **GLD** software program.



This caution symbol notifies the user that care must be taken at the specified location.



This star-shaped symbol highlights new features in GLD Premier 2010



The round symbol highlights suggestions for using the program more effectively or for improving designs.

Contents

Preface: Before You Begin.....	1
Typical Uses and Users.....	1
System Requirements for Installing Ground Loop Design.....	2
Hardware requirements.....	2
Software requirements.....	2
Operating system requirements.....	3
Internet browser requirements.....	3
Installation procedure.....	3
Initial installation.....	3
Installation of updated versions or re-installation.....	3
Program Licensing.....	5
Software License Dongle.....	5
How to Transfer the Program Between Computers.....	6
Macintosh Computers.....	6
Chapter 1: Ground Loop Design Overview.....	7
General Program Features.....	7
New in Premier Financial 2010 Edition.....	7
The Design Studio.....	8
Customization.....	9
Custom Company Info.....	9
Metric/English Units.....	9
Internationalization.....	10
Heat Pump and Zone/Loads Models Introduction.....	10
Heat Pump Module.....	11
Zones/Loads Modules.....	11
Zone Manager Loads Module.....	12
Average Block Loads Module.....	12
Design Modules.....	13
Borehole Design Module.....	14
Description.....	14
Theoretical Basis.....	15
Horizontal Design Module.....	17
Description.....	17
Theoretical Basis.....	17
Surface Water Design Module.....	19
Description.....	19
Theoretical Basis.....	19
Lifecycle Costing and CO ₂	20
Thermal Conductivity Module.....	21
Additional Modules.....	21
Reports.....	21
Project Reports.....	21
Monthly and Hourly Inlet Temperature Reports.....	21
Zone/Loads Reports.....	22
Lifecycle Reports.....	22
Thermal Conductivity Report.....	22
CFD Module Reports.....	22
Data Reference Files.....	23
Program Help and Support.....	23
References.....	23

Chapter 2: Adding/Editing Heat Pumps..... 25

Heat Pump Model.....	25
Description.....	25
Theoretical Basis.....	26
Capacity and Power.....	26
Flow Rate.....	27
Load Side Corrections.....	27
Entering Data into the Add/Edit Heat Pumps Module.....	28
Creating a New Series and/or Manufacturer.....	28
Editing Pump Data.....	29
Pump Series Controls.....	29
Pump Edit Controls.....	30
Save Control.....	30
Edit Pump Information Control.....	30
Delete Series Control.....	30
General Information.....	31
Capacity, Power, and Flow Rates.....	31
Load Side Corrections.....	32
Load Temperatures Panel.....	32
Load Flows Panel.....	33
Testing Input Data.....	34
Exiting the Add/Edit Heat Pumps Module.....	35
Heat Pump File Descriptions.....	35
Adding Pump Sets Obtained from External Sources.....	36
Other Resources	37

Chapter 3: Loads and Zones38

The Ground Loop Design Loads Model.....	38
Zone Files.....	39
The Zone Manager Loads Module.....	40
Managing Zones in the Loads Tabbed Panel.....	41
New and Copy.....	41
Remove and Clear.....	41
ReNUMBER.....	41
Summary View Toggle Button.....	41
Entering Loads.....	42
Design Day Loads.....	43
Annual Equivalent Full-Load Hours.....	43
Equivalent Hours Calculator.....	44
Days per Week.....	45
Pump Matching and Selection	45
Auto-Select.....	46
Manual Select.....	46
Details.....	47
Clear.....	47
Custom Pump (Customization).....	47
Automatic Heat Pump Selection Options for the Entire Zone Set.....	48
Auto-Select All Pumps.....	48
Update/Reselect Current Pumps.....	48
Working Series Selection in the Heat Pumps Tabbed Panel.....	49
Choosing the Active Series.....	49
Inlet Load Temperatures.....	50
The Average Block Loads Module.....	50

Managing the Average Block Loads.....	52
New.....	52
Clear.....	52
Entering Loads.....	52
Monthly Loads.....	53
Hourly Loads.....	54
Graphical View of Loads.....	54
Pump Selection.....	54
Details and Clear.....	55
Custom Pump (Customization).....	55
Pump Continuous Update Feature.....	55
The Studio Link System.....	55
Making a Link.....	56
Unlinking.....	56
Studio Link Status Lights.....	57
Importing Loads Data From External Programs.....	57
Importing Loads into the Average Block Loads Module.....	57
Importing Loads From 3rd Party Programs.....	58
Importing Loads From Spreadsheets.....	60
Importing Loads into the Zone Manager Loads Module.....	62
Importing Loads From 3rd Party Programs.....	63
Importing Loads From Spreadsheets.....	63
When Imported Data is Not Detailed Enough	65
Review of Loads Entry.....	66
Design Day Loads.....	67
Annual Equivalent Full-Loads Hours.....	67
Surface Water Loads.....	68

Chapter 4: The Borehole Design Module..... 69

Overview.....	69
General Features.....	69
Opening Projects.....	71
New Projects.....	71
Existing Projects.....	71
Saving Projects.....	71
Typical Operation.....	71
Entering Data into the Tabbed Panels	72
<i>Information</i>	72
<i>Extra kW</i>	73
Pump Power Calculator.....	74
<i>Pattern</i>	75
Vertical Grid Arrangement.....	75
Separation between Vertical Bores.....	75
External Grid Files.....	77
G Function Calculator.....	80
Export to AutoCAD.....	80
Bores per Parallel Loop.....	80
Fixed Length Mode.....	80
<i>U-Tube</i>	81
Pipe Parameters.....	82
Borehole Diameter and Backfill (Grout) Information.....	83
<i>Soil</i>	83
Drilling Log Conductivity Calculator.....	85
Diffusivity Calculator.....	85
Modeling Time Period.....	86

<i>Fluid</i>	87
Design Heat Pump Inlet Fluid Temperatures.....	87
Design System Flow Rate	87
Solution Properties.....	88
<i>Results</i>	89
Design Day Results: Fixed Temperature.....	90
Design Day Results: Fixed Length.....	92
Optional Cooling Tower and Boiler.....	94
Monthly Simulation Results: Fixed Length.....	96
Graphing Module.....	98
Optional Cooling Tower and Boiler.....	100
Hourly Data Results: Fixed Length	102
Graphing Module.....	105
Optional Cooling Tower and Boiler.....	107
Printing Reports.....	108

Chapter 5: The Horizontal Design Module..... 109

Overview.....	109
General Features.....	109
Opening Projects.....	110
New Projects.....	111
Existing Projects.....	111
Saving Projects.....	111
Typical Operation.....	111
Entering Data into the Tabbed Panels	112
<i>Configuration</i>	112
Trench Layout.....	112
Pipe Configuration in Trench.....	113
Straight Pipe Configurations.....	114
Single Pipe Vertical Alignment.....	114
Two-Pipe Vertical Alignment.....	114
Three-Pipe Vertical Alignment.....	114
Slinky Pipe Configurations.....	114
Vertical Slinky.....	115
Horizontal Slinky.....	115
Modeling Time Period.....	115
<i>Piping</i>	116
Piping Parameters.....	116
<i>Soil</i>	118
Undisturbed Ground Temperature.....	118
Soil Thermal Properties.....	118
Diffusivity Calculator.....	119
Ground Temperature Corrections at Given Depth.....	119
Regional Air Temperature Swing.....	119
Coldest/Warmest Day in Year.....	120
<i>Fluid</i>	120
Design Heat Pump Inlet Fluid Temperatures.....	120
Design System Flow Rate.....	120
Solution Properties.....	121
<i>Results</i>	122
Reporting Section.....	124
Optional Cooling Tower and Boiler.....	124
Printing Reports.....	127

Chapter 6: The Surface Water Design Module..... 128

Overview.....	128
General Features.....	128
Opening Projects.....	129
New Projects.....	130
Existing Projects.....	130
Saving Projects.....	130
Typical Operation.....	130
Before You Begin.....	131
Entering Data into the Tabbed Panels	131
<i>Surface Water</i>	131
Surface Water Temperatures at Average Circuit Pipe Depth.....	132
Surface Water Temperatures at Average Header Pipe Depth.....	132
Primary Header.....	133
Branches.....	133
Details (Reference only).....	133
<i>Piping</i>	133
Circuit Parameters.....	134
Circuit Pipe Size.....	134
Number of Parallel Circuits.....	135
Circuit Style.....	135
Circuit Head Loss per 100 feet.....	135
Extra Equivalent Length per Circuit.....	135
Header Parameters.....	136
Number of Lines.....	137
Pipe Size.....	137
Header Length / Average Branch Length.....	137
Head Loss per 100 feet.....	138
<i>Soil</i>	138
Ground Temperature Corrections at Given Depth.....	139
Depth of Header in Soil.....	139
Soil Type.....	139
Regional Air Temperature Swing.....	139
Coldest/Warmest Day in Year.....	139
Corrected Temperature.....	139
<i>Fluid</i>	139
<i>Results</i>	140
Reporting Section.....	142
Printing Reports.....	143

Chapter 7: Reports..... 144

Overview.....	144
The Report Preview Window.....	144
Project Reports.....	145
Information.....	145
Calculation Results.....	145
Input Parameters.....	146
Loads.....	146
Monthly Inlet Temperatures.....	146
Comments.....	146
Zone Reports.....	146
Detailed Form.....	147
Concise Form.....	147

Equipment List.....	147
Loads List.....	147
Names List.....	148
Finance Reports.....	148
Concise Form.....	148
Detailed Form.....	148
Inputs Forms	148
Financial Analysis Form.....	149
Thermal Conductivity Report.....	149
Fluid Dynamics Report.....	149
Concluding Remarks.....	149

Chapter 8: Tables and Data Reference Files.....150

Overview.....	150
Tables Included with Ground Loop Design.....	151
Fluid Properties.....	151
Soil Properties.....	152
Pipe Properties.....	152
Conversions.....	153
Adding Customized Reference Files.....	153
Original Model.....	153
HTML Files.....	154
Editing Existing Files.....	154
Making a Table.....	155
Adding a Picture, Graph, or Figure.....	155
Taking Care with Updates.....	156
Concluding Remarks.....	156

Chapter 9: The Finance Module.....157

Overview.....	157
General Features.....	158
Theoretical Basis.....	159
Opening Projects.....	159
New Projects.....	159
Importing Data from an Open Heat Exchanger Project.....	160
Existing Projects.....	160
Saving Projects.....	161
Typical Operation.....	161
Entering Data into the Tabbed Panels	161
<i>Incentives</i>	161
<i>Other Costs</i>	162
Emissions Costs.....	163
Average Building Costs.....	164
Equipment Related Costs.....	165
<i>Utility Costs</i>	169
Rates for Common Fuels.....	169
Annual Inflation Rates.....	170
<i>Conventional</i>	170
Alternate Systems.....	171
System Details.....	172
Cooling.....	172
Heating.....	173

<i>Geothermal</i>	174
Power Summary Panel.....	175
The Modeling Time Period.....	176
Project Power Summary.....	176
Geothermal System Details.....	176
Primary Geothermal Tab.....	177
Cooling.....	177
Heating.....	178
Installation Area.....	179
Hybrid Component Tab.....	180
Cooling.....	180
Heating.....	181
<i>Results</i>	182
Annual Costs.....	183
NPV Lifetime Costs.....	183
Printing Reports.....	184
References	184

Chapter 10: The Thermal Conductivity Module..... 185

Overview.....	185
General Features.....	186
Theoretical Basis.....	187
Opening Projects.....	187
New Projects.....	187
Existing Projects.....	187
Saving Projects.....	187
Importing Conductivity Data.....	188
Typical Operation.....	189
Entering Data into the Tabbed Panels	189
<i>Diffusivity</i>	189
<i>Flow</i>	190
<i>Bore</i>	191
<i>Results</i>	192
Calculation Interval.....	194
Calculation Results.....	194
Data Quality.....	194
<i>Graphs</i>	196
Printing Reports.....	197

Chapter 11: The Computational Fluid Dynamics Module.....198

Overview.....	198
Nomenclature.....	200
General Features.....	201
Theoretical Basis.....	202
Opening Projects.....	202
New Projects.....	202
Existing Projects.....	203
Saving Projects.....	203
Typical Operation.....	203
Entering Data into the Tabbed Panels	204

<i>Circulation Pumps</i>	204
Managing Circulation Pumps.....	205
New and Copy.....	205
Remove and Clear.....	205
Re-number.....	205
Summary View Toggle.....	205
Circulation Pump Details.....	208
<i>Automation</i>	209
GHX Module.....	209
Return Pipe Style.....	209
Circuit Info.....	209
Header Info.....	210
Supply Return Info.....	211
Manifold.....	211
Return Pipe Style.....	212
Section Outlet Info.....	213
Supply Return Info.....	213
Ultra Manifold.....	213
Return Pipe Style.....	214
Section Outlet Info.....	214
Supply Return Info.....	215
<i>Fluid</i>	216
Fluid Info.....	217
Solution Properties.....	220
<i>Layout</i>	221
Section 1: Calculate and Results Display Buttons.....	222
Section 2: Layout Manager Workspace	226
Section 3: Flow Type Selection.....	229
Section 4: Properties Window.....	230
Section 5: Circuit Confirmation Calculator.....	232
GLD Piping Language.....	232
Piping Components.....	233
Basic Grammar.....	238
Concept 1: Component Families.....	238
Concept 2: Component Relationships.....	238
Concept 3: Parallel and Serial Flow Paths.....	240
Concept 4: Direct and Reverse Return Headers.....	240
Sample Loopfield Layouts.....	250
Building Piping Systems.....	260
Manual Methods.....	260
Adding Pipe Pairs.....	261
Adding Circuits.....	264
Drag and Drop.....	264
Copy and Paste.....	265
Hide and Display.....	268
Delete.....	268
Using the Property Window.....	271
Using the Pipe/Fitting Manager.....	273
Automatic Methods	276
GHX Module Builder.....	276
Manifold/Vault Builder.....	282
Ultra Manifold/Vault Builder.....	286
Calculating and Reviewing Results	288
Calculating Results.....	288
Reviewing Results	289
Properties Window Results.....	290

CONTENTS

Layout Manager Workspace Results.....	291
Review Panel Results	295
Auto Optimization Tools	297
Purging Flow Rate Auto Optimizer.....	297
GHX Header Design Optimizer.....	299
Adding Circulation Pumps.....	302
Adding a Circulation Pump.....	303
Deleting a Circulation Pump.....	306
Printing Reports.....	306
Exporting Piping Reports.....	307
Exporting Circulation Pump Reports.....	307
Concluding Remarks.....	308

PREFACE

Before You Begin

This chapter describes the typical uses and users of the software. It also describes the installation procedure and hardware and software requirements for the Ground Loop Design (GLD)TM program. Additionally, the chapter introduces the licensing system.

Introduction: Typical Uses and Users

GLDTM 2010 is intended as a Design Studio for professional HVAC designers and engineers working in the area of geothermal applications. It is primarily designed for use with light-commercial or commercial installations, since the calculations take into account a) the long-term thermal effects that often determine the necessary design requirements and b) piping and flow optimization design considerations that can have a significant impact on overall system performance and cost-effectiveness. The program is optimized for hybrid system design that combine boilers, cooling towers, fluid coolers, solar thermal and the like with ground heat exchangers. Additionally, the loads representation employed in GLD's Zone Management system allows for detailed equipment selection and specific load distribution data to maximize calculation accuracy.



The Premier Version of GLDTM includes four design modules: one for vertical borehole ground heat exchanger systems, one for horizontal heat exchanger systems, one for surface water (pond, lake, etc.) installations and the new computational fluid dynamics (CFD) module for piping and flow optimization

design across all types of ground heat exchangers (the Professional/Standard Edition lacks the CFD module). It also includes two loads modules, one for average block loads and one for the more detailed zone model. The loads data can be shared between modules using GLD's unique linking system. In addition, loads data (including design day, monthly and 8760 hourly loads data) from external energy simulation programs as well as from Excel files conveniently can be imported into the loads modules. Note that the Professional/Standard Edition cannot use 8760 hourly loads data. GLD also includes a financial module for conducting financial, CO₂ emissions and energy costs analyses of various HVAC systems. The Thermal Conductivity analysis module, an optional module, performs soil thermal conductivity and borehole thermal response analyses using in-situ thermal conductivity/response test data.

Because of the extensive customization and override features included in the software, GLD is suited ideally for both standard and non-standard applications, which can involve significant variations in equipment, loads, and operational parameters for each zone in the design. The user, who may prefer to add his or her specific images or data sheets, has the freedom to customize the data reference files.

With instant, direct metric/English unit conversions and foreign language capabilities, GLD is a truly international program. With GLD, communicating project parameters, equipment requirements and loads data with coworkers, partners, and vendors anywhere in the world is efficient and easy. The program provides a framework for international standardization.

System Requirements for Running GLD

This section lists the hardware and software requirements for running GLD.

Hardware Requirements

A full installation has the following minimum hardware requirements:

- 1 GB RAM (2 GB recommended)
- 150 MB hard disk space (300 MB recommended)
- Intel Core 2 Duo Processor for optimal simulations

Software Requirements

GLD has the following software requirements:

- System running under Windows[®]
- Netscape Navigator[®] or Internet Explorer[®]

Operating System Requirements

GLD will operate under Windows 9X/ME/NT/2000/XP/VISTA/7. GLD will operate under Apple Macintosh Parallels or Boot Camp running a Windows OS as well.

Internet Browser Requirements

An Internet browser is required only for viewing the data reference files, and not for general program operation. To access the data reference files, at least one of the following browsers is necessary:

- Netscape Communicator Version 5.0 or later.
- Internet Explorer Version 5.0 or later.

Installation Procedure

If you have problems installing GLD, please visit the support page at <http://www.climatemaster.com> or contact your distributor. Note that you can also download GLD from the internet at <http://www.climatemaster.com>. The downloadable version always will be the most recent release.

Initial Installation

For CD versions of GLD, installation should start automatically. If not, the software may be installed by clicking on the **Setup.exe** file included on the disk.

The program is set to install in the folder:

(Main Drive):\Program Files\Gaia Geothermal\GLD2010

If desired, the user can specify a different location during the installation sequence.

Installation of Updated Versions or Re-Installation

GLD2010 will not overwrite a previous version of GLD on the user's computer. If the user needs to uninstall and reinstall GLD1010 for any reason, existing work files, pumps, and zone files will not be affected. However, the pumplist.gld file will be overwritten and any customized data reference files need to be protected (see below).



Note:

The file “**Pumplist.gld**” in the \GLD\pumps folder will be overwritten upon re-installation. If the user has added pumps other than those originally included with the program, *this file should be copied or moved to a backup directory* prior to removal and re-installation. After re-installation, the **Pumplist.gld** file can be returned to the \GLD\Pumps folder *or* the desired contents can be added to the contents of the new **Pumplist.gld** file using a simple text editor like **Notepad.exe**. The format of the file is provided below.

Pumplist.gld

Pump List File
Number of Manufacturers (Integer)
First Manufacturer Name (Text)
Street Address (Text)
City, State, Zip (Text)
Country (Text)
Telephone Number (Text)
Number of Different Series for this Manufacturer (Integer; Example: 2)
Series #1 Name (Text)
Series #1 filename without .hpd extension (Text)
#Date Entered# (Text; Example: #2001-10-05#)
Series #2 Name (Text)
Series #2 filename without .hpd extension (Text)
#Date Entered# (Text; Example: #2001-10-06#)
Second Manufacturer Name (Text)
Street Address (Text)
...

Alternatively, any pump files not included with the setup package may be added from within the program itself using the method described in Chapter 2, under *Adding Pump Sets Obtained from External Sources*.

The actual original heat pump data files (*.hpd), will not be deleted unless their names are identical to those being installed. Thus, all data can be recovered even if the previous version of the **Pumplist.gld** file is overwritten. However, this will either involve editing the **Pumplist.gld** file manually to include the customized data, or identifying those files within the program itself. In general, if there are only a few pump sets to add, working within the program may be best. If there are many, cutting

and pasting from the old file using a text editor may prove to be more efficient (Remember to modify the number of manufacturers if necessary!).

If the user has created customized heat pump sets, it may be wise to make a backup of all data files prior to removal and re-installation.



Additionally, *customized data reference files should be backed up* before any user-modified GLD menu HTML documents are replaced. The linked HTML documents themselves will not be overwritten.

Program Licensing

This section describes the USB dongle and license transfer options available in the GLD 2010 Edition.

Software License Dongle

Your GLD software license is stored on the USB dongle that came with your program. This dongle enables you effortlessly to transfer GLD from one computer to another. Please be careful not to misplace this dongle. Lost dongles can not be replaced without the purchase of a new license.

If the dongle is not attached to your computer, GLD will function as a trial version, which is functional except for a few design parameters that are locked at certain values.

When you insert the dongle into a free USB port on your computer for the first time, your computer most likely will recognize the dongle and after a few seconds, the dongle light will turn on. When it turns on, your license will activate. However, if your computer indicates that the dongle is “new hardware” you have two options for installing the dongle driver:

How to Install the Dongle Driver:

Windows 7, Vista, XP and Windows 2000 users with internet access:

If your computer has access to the internet, your computer can automatically install the drivers. Follow along with the Windows new hardware wizard to install the drivers. The process takes a few minutes. When the installation is complete, the dongle light will turn on.

All other users:

Via Windows Explorer, navigate to:
(Main Drive):\Program Files\Gaia Geothermal\GLD2010\Extras

In the Extras folder you will find a HASPUserSetup.exe program.

Run the program to install the dongle driver. When the installation is complete, the dongle light will turn on.

After Dongle Installation is Complete:

Now that the dongle is installed, you can access the full functionality of the GLD version that you purchased. If you remove the dongle, the program will revert to demo mode. If you reattach the dongle, the program will reactivate again.

How To Transfer the Program Between Computers

The dongle licensing system allows the user to transfer the license from one computer to another. If a user decides to transfer GLD from one computer to another, all he or she has to do is the following:

- Install GLD onto the target computer.
- After the “demo version” of the program is running on the new computer, attach the dongle and follow the above instructions regarding dongle driver installation.

Dongle Activation for Apple Macintosh Computers

Use this command in the Darwin (Unix) window of the Terminal Utility in the Utilities folder (there is no need to restart the computer or Parallels):

```
sudo launchctl unload/Library/LaunchDaemons/com.aladdin.aksusbd.plist
```

CHAPTER 1

GLD Overview

This chapter is an introduction to the GLD Premier 2010 Edition software package. It introduces new features, the Design Studio, the Heat Pump and Loads Modules, the Borehole, Horizontal and Surface Water Design Modules, the Finance Module, the Thermal Conductivity Module, the reporting functions, and the data reference files. There is also an explanation of the theoretical and experimental basis for the program's calculations.

General Program Features

GLD 2010 is a “Geothermal Design Studio” that provides the user with a freedom that single-purpose software cannot offer. The program is modular and permits flexibility in the designing process and customization based on designer preferences. Additionally, it has an English/metric unit conversion option, providing applicability to the widest range of equipment and customers. Because the software is available in different languages, it is truly international in its ability to traverse national borders as well as language and cultural barriers.



New in Premier 2010 Edition

GLD Premier 2010 Edition adds a range of features to the program including:

- The Computational Fluid Dynamics (CFD) module. This new module enables designers to easily model, analyze and optimize the piping systems that comprise the foundation of ground heat exchanger systems. Piping systems consist of a wide possible range of connected components including : Manifolds/Vaults, Supply and Return Runouts, Supply and Return Headers, single and double u-bends, the fittings that connect the

systems together and circulation pumps. Utilizing a new, patent-pending visual interface for viewing and creating a design via a drag and drop methodology, the CFD module can automatically design optimal flow-balanced (direct and reverse return) systems while providing designers with the flexibility they need for standard and non-standard systems. Outputs include flow rates, fluid velocities, Reynold's numbers, pressure drop, fluid volume and the like for every single point in a dynamic geothermal piping system. The GLD CFD Module is the first tool in humanity's toolbox for modeling the complex fluid dynamics in geothermal heat exchanger systems.

- 8760 Hourly energy simulations. With a new g function engine based off of recent heat transfer research advances, GLD can now model the bore and fluid temperatures and equipment performance hour by hour over one or more design years. The hourly simulation provides the highest degree of design accuracy and optimization and enables (for example) advanced solar thermal recharge simulations and more precise average annual consumption and performance (COP/EER) calculations.
- A suite of new visualization tools. A set of new tools enables instant visual review of critical design parameters and results. Now users can visualize loads in 2-D color graphs and graph a variety of monthly and hourly results from within the Design Studio.
- The updated heat pump module and heat pump database can now store recommended and minimum heat pump pressure drop and flow rate data.
- An updated user interface. An updated user interface with the new design dashboard enables a designer to compare designs with greater speed.
- Enhanced industry integration. GLD now features bi-directional integration with leading building energy simulation tools including the IES <Virtual Environment> and the Trane Trace software products.
- Customized logos and report inputs. GLD now enables users to brand reports with their own company logos.
- A number of new professional reports
- Borehole Thermal Resistance (BTR) calculations from in-situ data in the Thermal Conductivity module.
- Export your borehole designs to AutoCAD.

The Design Studio

The studio is the desktop work area in which the designer conducts his or her project analyses and establishes the basis for designs. When additional projects are desired, new windows may be opened or existing projects may be loaded. The Loads modules hold and display the information for the particular installation. Other windows may be opened concurrently. For example, one window may be used to edit or to modify heat pump data, another to calculate equivalent full load hours, and still others to provide easily accessible graphs or charts that may be required repeatedly through the course of a design. Similar design plans can be compared directly, or entirely different designs can be created

and varied. All of the information a designer needs exists in one convenient location within GLD.

Besides opening and closing windows and taking care of file management, the studio desktop menu and toolbar include control features which can be applied to more than one different type of project. For example, the English/metric unit conversion tool can convert a single window without affecting the rest of the open windows. Project reports can also be printed from the studio desktop.

Customization

GLD offers the user a great deal of freedom in how he or she enters and uses information. Rather than conforming designs to the software, this software package allows some modification and variation in its included features.

Some of the most common areas of customization in GLD include the entry of loads and the selection of equipment. Although fully automatic modes are available, the user also has the ability to customize or override the automatic features. For example, detailed load information may be included for precision designs, while extremely limited data is enough for rough calculations. Additionally, if the data are available, the designing engineer can enter his or her own pump sets to take full advantage of the automatic selection procedures. Also, different families of pumps can be used within a single project, and even individual pumps not included in the pre-defined pump sets can be employed as required.

Another area where customization is possible is in the data reference files, which are based on HTML. With a simple HTML editor the user can include any tables, data, pictures, graphs, charts, or any other useful information that meets the user's needs. User-added files can supplement or replace the data reference files already provided with GLD.

Custom Company Information



New in GLD Premier 2010 is the *Settings* dropdown menu. From this menu, users can enter general company and contact information that is repopulated in the various *Information* tabbed panels throughout the program. This feature is available in the Premier Edition of the program.

Metric/English Units

One of the intrinsic features in GLD is the English/metric unit conversion capability. The English/metric option can be used not only to compare values, but it also can be used to quickly make use of specific equipment or loads data supplied in only one format.

Because the reports and data reference files automatically recognize the selected units, users can obtain different reports and data lists depending on the state of the Design Studio. Presentation and comparison of project information between different engineers and designers is now a straightforward process.

Internationalization

Because GLD is multi-language capable, users easily can communicate accurate results and design parameters across borders, even when the designers are not proficient in the technical language of their foreign counterparts.

Currently, Bulgarian, Chinese, Czech, French, German, Greek, Italian, Japanese, Korean, Lithuanian, Romanian, Russian and Spanish versions are available. Figure 1.1 is a screenshot from the Korean version (metric).

The screenshot shows the '존 관리자' (Zone Manager) window in the Korean version of GLD. The window title is '존 관리자' and the file name is 'BoreholeSample.zon'. The interface includes a menu bar with '열펌프' (Heat Pump) and '부하' (Load). A toolbar contains various icons for file operations. On the left, a list of zones (구역) is shown, with '구역 3' (Zone 3) selected. The main area displays a table of heat pump and zone load data for '구역 3 부하의 계획' (Zone 3 Load Planning). The table includes columns for '참조 라벨' (Reference Label), '설계 냉시간 부하' (Design Cooling Load), '연중전체 부하' (Annual Total Load), and '설계 온도' (Design Temperature). The data is organized into sections for '구역 3 부하의 계획' (Zone 3 Load Planning), '설계 냉시간 부하' (Design Cooling Load), and '설계 온도' (Design Temperature). The bottom status bar shows '유량: 11.4 (L/min)/3.5kW' and '인입 온도 (섭씨): 29.4'.

구역	구역 3 부하의 계획	참조 라벨	설계 냉시간 부하	연중전체 부하	설계 온도
1	5.0	8시 - 12시	13.5	11.1	29.4
2	전달	12시 - 16시	10.6	5.9	10.0
3	시간 계산	16시 - 20시	6.2	2.9	
4		20시 - 8시	0.0	3.5	
5		연중전체 부하 등가시간:	1050	220	
6		설계 온도와 유량에서의 열펌프 상세			
7		펌프명	번호	냉방	난방
		EV048	1	13.7	14.0
		용량 (kW)		3.74	3.11
		동력 (kW)		3.7	4.5
		COP		43.5	36.0
		유량 (L/min)		0.98	0.80
		부분 부하 계수			

Figure 1.1 Korean Version of GLD

Heat Pump and Zone/Loads Modules Introduction

The underlying framework of GLD is based on three modules that permit flexibility in the addition and modification of components related to geothermal designs. The first is the heat pump module, which takes a representative amount of data from the heat pump specifications and then uses it for the automatic pump selection features. The second and third are the average block and zone loads modules, which provide organized methods for entering the heat gains or losses

for an installation. Because the heat pump and loads modules are closely related, users can match heat pumps to the loads automatically or manually.

An advantage of this design is that the heat pump selection and the loads modules can be connected directly with the various design modules available in the studio. **Therefore, one type of loads and heat pump data can be used for all designs.**

Heat Pump Module

In GLD 2010, ClimateMaster heat pump data is stored in the heat pump module. This module functions as an input interface to the ClimateMaster heat pump database. Because the ClimateMaster heat pump database is up to date in GLD, the heat pump module is not accessible from the ClimateMaster Ground Loop Design software suite. The heat pump database itself is accessible from the loads modules (where the designer selects heat pumps).

When new ClimateMaster heat pumps become available, users will have the option of updating their heat pump database following instructions from ClimateMaster or its distributors. These updated heat pumps can then be used within the software to take advantage of the automatic equipment sizing features of GLD.

The heat pump model that GLD uses requires the input of a minimum of six data points for both heating and cooling modes. These data points relate capacity and power to the inlet source temperature and are fit using a polynomial line to provide an accurate model for the equipment for any given design parameters. By including additional data from different source flow rates and/or different inlet load temperatures and flow rates, higher levels of accuracy are possible.



New in GLD 2010, the heat pump module can store recommended and minimum flow rate and pressure drop information for each heat pump.

Note that while the Heat Pump Module is not accessible in the ClimateMaster Ground Loop Design software suite, the heat pump database is accessible through the loads modules.

Zones/Loads Modules

GLD employs two different types of load input schemes. With the Zone Manager Loads module, users can perform a detailed analysis. With the Average Block Loads module, users can make quick estimates without performing detailed component design work. In Premier Financial 2010 Edition, users can optionally add monthly and/or hourly loads data to the Average Block Loads and then calculate month-by-month and/or hour-by-hour inlet temperatures in a Borehole Design module. These stand-alone modules are linked to design modules using

the *Studio Link* system (Chapter 3). Both modules can import loads data from commercial loads programs and Excel files.

Zone Manager Loads Module

TIP

Component-style designs often are more appropriate for geothermal installations, particularly when equipment is available in various sizes. The units can be placed near or within the locations to be conditioned. With regard to water source heat pumps, it is often much easier to bring water lines to the equipment instead of providing ductwork or long load lines from a centralized source.

TIP

When considering geothermal applications, the precision of the zone/loads model is crucial because it relates directly to the extent of external heat exchanger installation. Heat exchanger costs impact the overall costs of a project. Additionally, a unit that is called only when necessary or is well matched to a zone will be more efficient than a larger unit that may cycle more often.

Inputs for GLD's Zone Manager Loads module include peak load information for each of the zones in an installation at different periods during the day. These loads data can be matched automatically to heat pumps stored within GLD's Heat Pump Database. Therefore, ideal and rapid sizing is possible. As with the Average Block Loads module, the annual running time also may be included for a buried heat exchanger. This loading information can be simple or complex, depending on the level of detail the designer desires. To facilitate this model, the zones can be viewed either independently or together on the summary panel.

Average Block Loads Module

For quick estimates and general calculations, there is no need to do a full zone analysis for a project. In these cases, designers can quickly enter data and consider approximate designs using the Average Block Loads module.

The average block model takes peak data from up to four time periods during the peak day, and then uses a generalized form of the automatic pump selection sequence to match a particular type of pump to an entire installation. For buried heat exchangers, the model also uses weekly and annual operational time as parameters. The hours can be computed from monthly loads data using the Equivalent Hours Calculator (Chapter 3).

In GLD Premier 2010, users have the option of calculating month-by-month inlet temperatures and/or 8750 hourly inlet temperatures in the borehole design module. Performing these calculations requires detailed

monthly and/or hourly loads data and therefore the average block module in the GLD Premier 2010 Edition now accepts the input of monthly total and peak loads for both heating and cooling as well as hourly peak loads for both heating and cooling. Note that while a user can design a system without these detailed hourly and monthly loads data, he or she cannot perform detailed simulations without the data.

Loads modules are covered in detail in Chapter 3.

Design Modules

The GLD Geothermal Design Studio consists of the following three heat exchanger design modules:

- The **Borehole Design Module** – In fixed temperature mode, this module models the lengths of bore required for a vertical borehole exchanger system. In fixed length mode, it models the inlet temperatures for a user-defined borehole field length. Additionally, the borehole design module can model and graph the monthly and hourly inlet temperatures for the design (if monthly/hourly loads data have been input into the Average Block loads module).
- The **Horizontal Design Module** – This module determines the length of piping required for a horizontal trench/bore/slinky exchanger system.
- The **Surface Water Design Module** - This module determines the length of piping required when a closed loop of pipe inserted into a body of water acts as the heat exchange medium.

All three modules utilize the same loads module formalism, and are linked to loads modules using the *Studio Link* system.

All three modules also include an optional expanded user interface function (available only in the Premier version which can be purchased as an upgrade) as can be seen in figure 1.2. By double clicking on any of the tabbed panels (Results, Fluid, Soil, U-Tube, etc.) an expanded calculation view appears which enables the designer both to see the calculated results immediately after any parameter has been modified and also to access the parameters that are most commonly adjusted during the design optimization process.

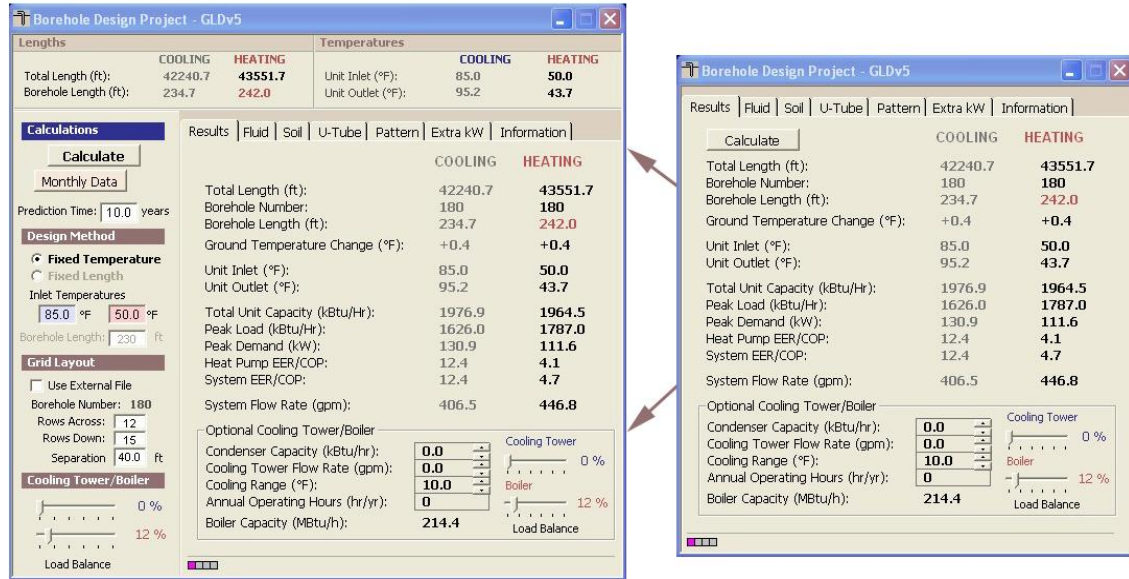


Fig. 1.2 Optional Expanded Interface

Borehole Design Module

Description

The Borehole Design module allows the user to enter various parameters with respect to the desired vertical borehole system. Input is arranged in panels corresponding to the type of input as shown in figure 1.3. Key design parameters can be modified quickly in the expanded user interface as well (see figure 1.2, above).



Fig. 1.3 Borehole Design Panel List

Using these seven panels (*Results, Fluid, Soil, U-Tube, Pattern, Extra kW, and Information*), the user enters the project-specific information. After the user enters all parameters, the software calculates results based on the input data. Within this framework, it is straightforward and easy to make changes and conduct new calculations.

The Borehole Design module allows for two types of design methodologies: **fixed temperature** and **fixed length** designs. Fixed temperature refers to the design process in which users specify target inlet

temperatures (designers set, or fix, the temperatures themselves) and then have the program calculate results such as the required bore length, the outlet temperatures and the coefficient of performance (COP), etc., based on the input data.

With fixed length designs, designers specify the required borefield length by inputting the number of boreholes in the design and then defining the borehole length (fixing the total design length) After entering these data as well as the other design parameters, the software calculates results such as, the inlet and outlet temperatures and the coefficient of performance (COP), etc., based on the input data. The fixed length feature is well suited for designing when land resources are limited, when a designer wishes to quickly reverse engineer a system, etc.

Additionally, when the borehole design module is linked to an Average Block loads module that has monthly or hourly loads data entered, the program can calculate and report monthly and/or hourly inlet temperatures and COP/EER values.

A more complete description about how to enter data and perform calculations in the Borehole Design module is provided in Chapter 4.

Theoretical Basis

To continue providing geothermal system designers with the widest range of flexibility, two separate theoretical models now are included within the GLD framework. The first model, and the original one used exclusively in GLD versions 1-4, is based on the cylindrical source model and allows for quick length or temperature calculations based on limited data input. The second is based on a line source theory, but is more detailed in its ability to generate monthly and/or hourly temperature profiles over time given monthly loads and peak data and/or hourly loads data. This second model is also able to model the impact of balanced and unbalanced loads on loopfield performance and length requirements. This second theory is popular throughout Europe and growing in popularity for its unique strengths. in some academic and institutional circles, and it is now included so that users can directly compare the two models' results using an identical input data set. Although the outputs of the two models do not always agree, they do give the designer more information on which to base a final system design.

The vertical bore length equations used in the primary model in the Borehole Design module are based upon the solution for heat transfer from a cylinder buried in the earth. The method was developed and tested by Carslaw and Jaeger (Carslaw and Jaeger, 1947). The solution yields a temperature difference between the outer cylindrical surface and the

undisturbed far field soil temperature. Ingersoll suggested using the equation and its solution for the sizing of ground heat exchangers in cases where the extraction or rejection occurs in periods of less than six hours (where the simple line source model fails) (Ingersoll, 1954). The borehole module's equations include the suggestions of Kavanaugh and Deerman, who adjusted the methods of Ingersoll to account for U-tube arrangement and hourly heat variations (Kavanaugh and Deerman, 1991). It also employs the borehole resistance calculation techniques suggested by Remund and Paul to account for pipe placement, grout conductivity, and borehole size (Paul, 1997).

Additionally, the software calculates the amount of energy absorbed by or withdrawn from the ground using the load information collected from the individual zones and their relationship to the equipment selected.

The calculations find the conditions for long-term, steady state operation of borehole fields based on the desired heat pump inlet temperatures. In order to provide an optimum design and prevent system failure, the combination of parameters must allow for proper extraction or dissipation of energy from or to the earth at the location of interest.

For the first model, the most complete description of the calculations and input data can be found in Chapter 3 of the book, *Ground Source Heat Pumps - Design of Geothermal Systems for Commercial and Institutional Buildings*, by S.P. Kavanaugh and K. Rafferty, 1997. In extensive tests, this model consistently proved to be the most accurate when compared with calibrated data from actual installations (Hughes and Shonder, 1998).

The second model within the Borehole Design module is based on the solution to the purely heat conductive problem in a homogenous medium, which was solved by approximating the borehole as a finite line sink (Eskilson, 1987). The steady state solution relates to the case where heat is extracted continuously from the borehole without ever exhausting the heat source, making it a fully renewable source of energy.

As implemented in GLD, the difference between the second model and the first is that with the second model, it is possible to when a constant heat extraction rate (Q) is extracted from the borehole. It makes use of a dimensionless "G-function" concept to model the temperature variations, taking into account the ratio of the borehole radius and length and the physical layout of the bore field. Bandos, et al. (2009) recently developed modifications to the G-function which are implemented in GLD 2010. GLD also employs its own internal borehole superposition model, allowing users to define the borehole layout in a gridfile, import the gridfile into the program and then automatically determine the required G-function.

Because of increased data entry requirements for the monthly and peak loads data in the second model, it is only applicable when used in conjunction with the Average Block Loads module (where only one set of monthly loads data is required per installation). Use of the Zone Manager is limited to the original cylindrical source theoretical model. The other design modules currently do not make specific use of the monthly loads data except in the reduced, “equivalent hours” form.

Horizontal Design Module

Description

The Horizontal Design module, similar to the Borehole Design module, allows the user to enter parameters necessary to describe a horizontal buried pipe and trench configuration. Again, the interface is arranged in panels corresponding to the type of input. Key design parameters also can be modified quickly in the expanded user interface as well (see figure 1.2, above).

After the user enters all parameters, the software calculates results such as the required trench and pipe lengths, the inlet and outlet temperatures, the coefficient of performance (COP), etc., based on the input data.

The input information is organized into seven panels, as shown in figure 1.4.



Fig. 1.4 Horizontal Design Panel List

Using these seven panels (*Results, Fluid, Soil, Piping, Configuration, Extra kW, and Information*), the user enters the project-specific information.

A more complete description about how to enter data and perform calculations in the Horizontal Design module is provided in Chapter 5.

Theoretical Basis

The horizontal trench length equations used in the Horizontal Design module are based upon the Carslaw and Jaeger solution for heat transfer from cylinders buried in the earth, as described in the single vertical case

above. Again, this method properly models shorter time periods of heat extraction or rejection, where the simple line source model fails. Since a number of pipes may be buried in close proximity, this model must be modified to account for all mutual pipe interactions. A major benefit derived from using this model, besides its ability to accurately assess heat-transfer, is that both the horizontal and the vertical design modules can operate under the same loads formalism.

In 1948, Ingersoll and Plass demonstrated that the Kelvin line source theory could be used to estimate the change in temperature of a buried pipe in which heat is being absorbed or rejected (Ingersoll and Plass, 1948). In a ground coupling system, an apparent thermal resistance between the circulating fluid and the undisturbed ground dominates the overall resistance. In 1985, in the *ASHRAE Design/Data Manual for Ground-Coupled Heat Pumps*, Parker et. al. outlined a method by which this “field resistance” or “soil resistance” could be estimated and applied to determine piping and trench length requirements for a buried pipe system. In the case of horizontal pipe systems located near the ground surface, the mathematics necessitate the inclusion of “mirror image” pipes into the calculations. These “mirror image” pipes are located the same distance above the surface as the buried pipes are below it. In a multiple pipe system, the soil temperature in the vicinity of any single pipe is determined by both the undisturbed earth temperature and by the thermal interference from other pipes in the same and in adjacent trenches. (Parker, Bose, and McQuiston, 1985).

The current Horizontal Module effectively employs a combination of the cylindrical model of Carslaw and Jaeger and the multiple pipe methodology of Parker et. al. Additionally, as in the Borehole Module, the equations also include modifications suggested by Kavanaugh and Deerman that adjust the methods of Ingersoll to account for physical arrangement and hourly heat variations (Kavanaugh and Deerman, 1991). However, time-step-based rates of rejection and extraction also previously were discussed in some depth by Parker et. al.

The two “Slinky” options available on the Configuration panel partially are based on the above formalism. Because of the complexity of the solution to the heat transfer equation for coiled loops of pipe, the design procedure used for the “Slinky” options is actually only a theoretical approximation. This approximation is recommended in *Closed-loop Geothermal Systems: Slinky Installation Guide* and is based on a specific set of tests conducted on 36” diameter Slinky coils (Jones, 1995). In the approximation, the program first calculates the total trench length required for a single U-Tube buried at the specified trench depth. It then divides the calculated length by 250 ft and multiplies the result by a factor determined from both the run fraction and the Slinky pitch (distance

between adjoining loops). The horizontal Slinky configuration employs the same calculation procedure as that of the vertical. However, in the case of the horizontal Slinky, the U-tube depth is lowered such that the average depth of the vertical Slinky would be equal to that of a flat horizontal Slinky. The pitch and run fraction function is obtained from a two-dimensional interpolation over the surface determined from the experimentally determined data points provided in the Slinky manual.

Surface Water Design Module

Description

The Surface Water Design module allows the user to enter various parameters concerning the body of water (lake, pond, river, etc.) system. As in the Borehole Module, inputs are arranged in panels that relate to the type of input. After the user enters all parameters, the software calculates the required pipe length, the circuit number, the inlet and outlet temperatures and the COP, etc., based on the design specifications. Again, within this framework, it is straightforward to make changes and recalculate results, especially when using the expanded user interface.

The input information is organized into seven panels, shown in figure 1.5.

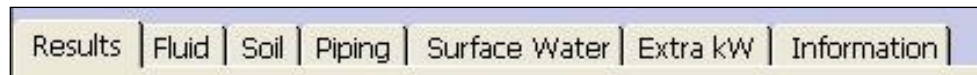


Fig. 1.5 Surface Water Design Panel List

These seven panels include *Results*, *Fluid*, *Soil*, *Piping*, *Surface Water*, *Extra kW* and *Information*. The panel names and many of the panel input parameters differ from those of the Borehole Design module.

A more complete description about how to enter data and perform calculations in the Surface Water Design module is provided in Chapter 6.

Theoretical Basis

To determine the length of pipe necessary for different surface water systems, experiments were conducted for different size pipes in coiled and “slinky” configurations for both heating and cooling modes (Kavanaugh, 1997). GLD uses a polynomial fit of this experimental data to determine the amount of pipe necessary for different loading conditions.

Additionally, coefficients are used to take into account the effect of the heat transfer in the lengths of the header and the branch piping that are in both the water and the soil between the installation and the submerged circuits. The program combines all factors so that the loop system provides the source inlet temperature at the heat pump requested by the designer.

Because the circuit layout is of primary importance to the designer concerned with pumping losses, the head loss estimation feature for different piping configurations is included in the Surface Water Design module. Users can quickly explore different layouts to determine the optimum design in terms of both heat transfer and circulation pump energy losses.

A description of some of the calculations and the input data can be found in Chapter 7 of the book, *Ground Source Heat Pumps - Design of Geothermal Systems for Commercial and Institutional Buildings*, by S.P. Kavanaugh and K. Rafferty, 1997.

Lifecycle Costing and CO₂ Module

The Lifecycle Costing and CO₂ module, similar to the abovementioned design modules, allows the user to enter parameters necessary to calculate both “hard” and “soft” annual and lifecycle Net Present Value (NPV) costs associated with his or her designed geothermal system compared to standard HVAC systems. This enables designers and decision makers to compare simultaneously the financial profiles and benefits of geothermal vs. standard HVAC systems.

The interface is arranged in panels corresponding to the type of input. After the user enters his or her desired parameters, the software calculates results such as the energy consumed by the selected systems, the CO₂ emitted from the systems, the water consumed by the systems (if any), etc.

The input information is organized into six panels, as shown in figure 1.6.

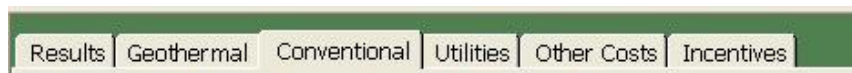


Fig. 1.6 Lifecycle Costing and CO₂ Panel List

Using these six panels (*Results, Geothermal, Conventional, Utilities, Other Costs and Incentives*), the user can enter, describe and compare project-specific financial and emissions estimates.

A more complete description of this module can be found in Chapter 9.

Thermal Conductivity Module

The optional Thermal Conductivity module enables designers to quickly analyze thermal conductivity test data from the GeoCube™, a product from Precision Geothermal LLC, as well as from other test units. Outputs from this analysis include formation thermal conductivity, diffusivity and borehole thermal resistance,.

See Chapter 10 for a full description of the Thermal Conductivity module.

Additional Modules

GLD's Design Studio has the potential for additional modules that may be included in later versions. These modules would also be able to take advantage of the Design Studio's heat pump and loads models.

Reports

GLD's reporting features allow the designer to make hardcopies of both the data entered and the resulting calculations. These reports are design records, and are valuable when communicating the design to others involved in the projects.

Project Reports

Every design module has associated project reports, which can be printed at any time from the Design Studio desktop. The project report contains all the project information, and includes the parameters chosen, the calculation results, and the name of the zone file used. Both concise and detailed versions of the report are available.

Monthly and Hourly Inlet Temperature Reports

Monthly and hourly inlet temperature reports can be printed from the Design Studio desktop after calculating inlet temperatures in the borehole design module. The reports contain heat transfer, power, borehole temperature, T_f (fluid temperature), exiting water temperature, entering water temperature, minimum entering water temperature and maximum entering water temperature for each month or hour of the design. The four reports include a concise temperature report, a detailed temperature report, a report that offers all project parameters, loads data and temperature data and a report that offers project parameters and loads data.

Zone Reports

A 'print' button in the loads modules allows the designer to print the loads-related information in various formats. Because the zones contain information about the zones, the loads, and the equipment, it is often necessary to obtain reports of the information in separate as well as combined documents.

For example, at one time, a designer may want to quickly see all of the zones with their loads and corresponding equipment. At other times, the designer may only need to see a list of the equipment for each zone. GLD offers five different zone report options including:

- A concise zone report
- A detailed zone report
- An equipment list report
- A loads report
- A zone names report

Lifecycle Cost and CO₂ Reports

A 'print' button in the finance module allows the designer to print the finance-related information in various formats.

GLD offers four different finance report options including:

- A concise finance report
- A detailed finance report
- A concise inputs report
- A detailed inputs report
- A financial analysis report

Thermal Conductivity Reports

A 'print' button in the thermal conductivity module allows the designer to print out a detailed, professional report. The designer can also print out large, color graphs to include in the report.

Computational Fluid Dynamics Reports

An 'export' button in the CFD module allows the designer to export user-specified design details into a text file for further processing in a spreadsheet program.

Reports are described in detail in Chapter 7.

Data Reference Files

To access the data reference files the user must have an internet browser present in the GLD-enabled computer. The program will work without the browser, but the data reference files may not be accessible.

Metric and English reference files are included with GLD. These files aid in the correct verification and entry of the various parameters. The three main topics/design aids currently included under the *Tables* menu in the Design Studio are *Fluid Properties*, *Soil Properties*, and *Pipe Properties*. A convenient *Conversions* table with metric/English conversions in two different formats is included for reference as well. Reference files can be opened and left as open windows on the desktop, and the user can refer to them as necessary during the design process.

Realizing that designers and engineers have their own preferred resources, GLD employs the HTML browser model so that the user has ultimate control over the reference files. The designer simply creates a basic HTML file containing customized data, pictures, graphs, charts, etc. and then modifies the included top level HTML files to link to their pages. The system requires a very basic knowledge of HTML, but it offers an extremely flexible system for user customization.

Detailed information on reference files and sample HTML can be found in Chapter 8.

Program Help and Support

GLD contains a comprehensive, searchable database of help topics. Access this feature from the Design Studio *Help* menu. Through the *Help* menu it is also possible to access the latest web resources and updates. If these resources do not answer your question, please contact your vendor for support.

References

Bandos, Tatyana, et. al. (2009). Finite Line-Source Model for Borehole Heat Exchangers: Effect of Vertical Temperature Variations. *Geothermics*. 38, 263-270.

Carslaw, H.S. and Jaeger, J.C. *Conduction of Heat in Solids*. Oxford: Clarendon Press, 1947.

Eskilson, P. *Thermal Analysis of Heat Extraction Boreholes*. Doctoral Thesis. University of Lund, Department of Mathematical Physics. Lund, Sweden. 1987.

Hughes, P.J. and Shonder, J.A. *The Evaluation of a 4000-Home Geothermal Heat Pump Retrofit at Fort Polk, Louisiana: Final Report*. Oak Ridge National Laboratory, TN. ORNL/CON-460. 1998

Ingersoll, L.R. and Plass, H.J. "Theory of the ground pipe heat source for the heat pump." *Heating, Piping, and Air Conditioning*. 20:7 (July). 1948.

Ingersoll, L.R., Zobel, O.J., and Ingersoll, A.C. *Heat conduction with engineering, geological, and other applications*. New York: McGraw-Hill. 1954.

Jones, F.R. *Closed-Loop Geothermal Systems: Slinky Installation Guide*. Rural Electric Research, National Rural Electric Cooperative Association, Oklahoma State University, International Ground Source Heat Pump Association, and Electric Power Research Institute, 1995.

Kavanaugh S.P. and J.D. Deerman. Simulation of vertical U tube ground coupled heat pump system, ASHRAE Transactions, Volume 97, pages 287 - 295, 1991.

Kavanaugh S.P. and Rafferty, K. *Ground Source Heat Pumps - Design of Geothermal Systems for Commercial and Institutional Buildings*, ASHRAE, 1997.

Parker, J.D., Bose, J.E., and McQuiston, F.C. *ASHRAE Design/Data Manual for Ground-Coupled Heat Pumps*. ASHRAE Research Project RP-366, 1985.

Paul, N. *The Effect of Grout Thermal Conductivity on Vertical Geothermal Heat Exchanger Design and Performance*. M.S. Thesis, South Dakota State University. 1996.

CHAPTER 2

Adding/Editing Heat Pumps

To effectively use any of the design modules included with GLD, it is important to understand how the system models heat pump data. For the purpose of adding new or editing existing heat pumps to GLD's Heat Pump Database, the Add/Edit Heat Pumps Module is included as a separate module in the Design Studio. This chapter describes the theory of the module and gives an example of how to enter heat pump data. A more detailed example can be found online, and accessed through the *Help* menu web resources option. Note that while this module is not available in the ClimateMaster edition of the software, the ClimateMaster pump data is of course fully accessible from within the loads modules.

Heat Pump Model

Description

For convenience, the Loads modules in GLD predict how heat pump characteristics will vary with changes in the input design parameters. If the designer changes the inlet source or load temperatures, or the system flow rate, the capacity and power data of the units may also change. The easiest and most accurate way of realizing these changes is to employ an

internal model, which the software uses to update the pump data automatically. Using GLD, the designer can concentrate on the effects of variations without worrying about how the individual pumps in various zones will react to such changes.

The heat pump model employed in GLD reproduces the complete operational data of any particular unit when supplied with a few representative data points selected from across the range of interest. Data for each pump can be entered into the model and grouped together under manufacturer and series headings. The data need only be input once, and then can be used repeatedly for subsequent modeling sessions. Pump data is stored permanently in the '**pumps**' directory. Many popular pumps from major manufacturers already are included with the program.

In both heating and cooling modes, the *minimum* data required is the capacity and power variations with source inlet temperature. To increase the modeling accuracy, these same variations have to be included at a second flow rate. Even more accurate results can be obtained if correction factors are provided for variations in the load inlet temperature and flow rate. The level of accuracy depends both on the amount of data available and the time the designer wants to invest.

Note that GLD's heat pump module allows for both water-to-air and water-to-water pumps.

Theoretical Basis

Capacity and Power

Heat pump capacities and power requirements vary smoothly but significantly for differing source inlet temperatures. Three points taken along both the capacity vs. temperature and power vs. temperature curves are fit to a polynomial equation to model these variations. The resulting calculated coefficients are then used to generate capacity or power values for any given source inlet temperature.

The basic polynomial equation used for fitting has the form:

$$y = a + bx + cx^2,$$

where a, b, and c are the three coefficients calculated from the fitting routine. For the capacity case, 'y' represents the capacity and 'x' is the desired temperature. For the power-input determination, 'y' is the power and 'x' again is the temperature. **Be aware that these coefficients do change for metric and English units.**



The software stores coefficients for each pump, and then uses the coefficients with the source inlet temperatures chosen by the designer to determine the unit capacity and power.

Flow Rate

To model the effect of the source flow rate on the calculated capacity and power, data from a second flow rate are used. Generally speaking, with different flow rates the shape of the capacity and power curves does not change significantly, but is shifted up or down by a constant factor. This factor is determined for each of the three temperature data points and averaged over those input to obtain the linear flow factor, which is shown on the input screen.

Once the flow factor is determined, the linear capacity or power change per flow unit may be calculated. The program then calculates a new capacity or power at any specified flow rate using the initial values already known from the stored data.



If no data points are entered for a second flow rate, the flow factor is assumed to be the constant value of 1.0. This means that the capacity and power will not vary with changes in flow rate.

Considering the size of the variations (generally only a few percent), this simple model is accurate enough for most pumps. A completely accurate model of the flow rate variations for all possible pumps would require significantly more data entry.

Load Side Corrections

The GLD Edit/Add Heat Pumps module also can include corrections to the capacity or power that result from variations in the load side inlet temperature or flow rate. They are entered as correction factors across the desired temperature or flow range. The software again uses the polynomial fitting to model these correction factors. In these cases, a four-coefficient model is used to better model the types of variations that may occur. Three to five points are allowed as data input.



Again, if load side correction data are not included, there will be no capacity or power variations with load temperature or flow, and all correction factors will be 1.0, the standard value.



The load side temperature range will generally be considerably different for water-to-air and water-to-water pumps. GLD suggests different initial

temperature ranges when the user chooses the water-to-air or the water-to-water pump type option.

Entering Data into the Add/Edit Heat Pumps Module

The user opens the Edit/Add Heat Pumps module from the Design Studio Heat Pumps menu. Note that one module can be open at a time.

When the module opens, there are two selection boxes present in the upper pane, while no pump data is displayed in the lower pane. In the left box, the user can choose to select either one of the manufacturers from the list of existing manufacturers or 'New Series'. If a manufacturer is selected, the associated list of pump series available for that particular manufacturer appears in the box on the right. When a series is chosen, the data for that series appears in the lower panel.

Creating a New Series and/or Manufacturer

If the user chooses 'New Series' from the manufacturer list on the left, the lower pane becomes active with another selection box that requests direction as to whether to use an existing manufacturer or to create a 'New Manufacturer'. After the user makes a selection, the panel changes to show information about the manufacturer and series. The manufacturer information will be editable if the series belongs to a new manufacturer. The Edit/Add Heat Pumps module with an open 'Pump Information' panel is shown in figure 2.1.

The screenshot shows a software window titled "Edit/Add Heat Pumps". At the top, there are two dropdown menus: the first is set to "-- New Series --" and the second is labeled "Series Name". Below these is a section titled "Pump Information" which contains a dropdown menu set to "-- New Manufacturer --". Under this, there are two columns of input fields. The left column, titled "Manufacturer Information", includes fields for "* Manufacturer's Name", "Street", "City, State, Zip", "Country", and "Phone". The right column, titled "Series Information", includes fields for "* Series Name", "* Filename (without extension)", and "Entry Date" (which has a dropdown menu showing "7 /27/2001"). At the bottom of the window are two buttons: "Proceed" and "Cancel".

Fig. 2.1 Pump Information Panel



After the user enters all the data and clicks the 'Proceed' button, all of the information for the series being added will be stored in the **Pumplist.gld** file. **Note that the information marked with an asterisk must be included before the user is allowed to proceed.**

Editing Pump Data

Once the new pump series information is entered, or an existing pump series is selected from the upper pane, the Pump Edit pane will appear in the lower pane of the Edit/Add Pumps module, as shown in figure 2.2. There are two sub-panes. The left sub-pane is a list of the pumps already included in the series. The right sub-pane is a series of tabbed panels that contain the data for each pump on the list. In the case of a new series, both the list and the panel section will be empty until a new pump is created. The name of the current manufacturer and series are shown in the selection boxes in the upper pane.

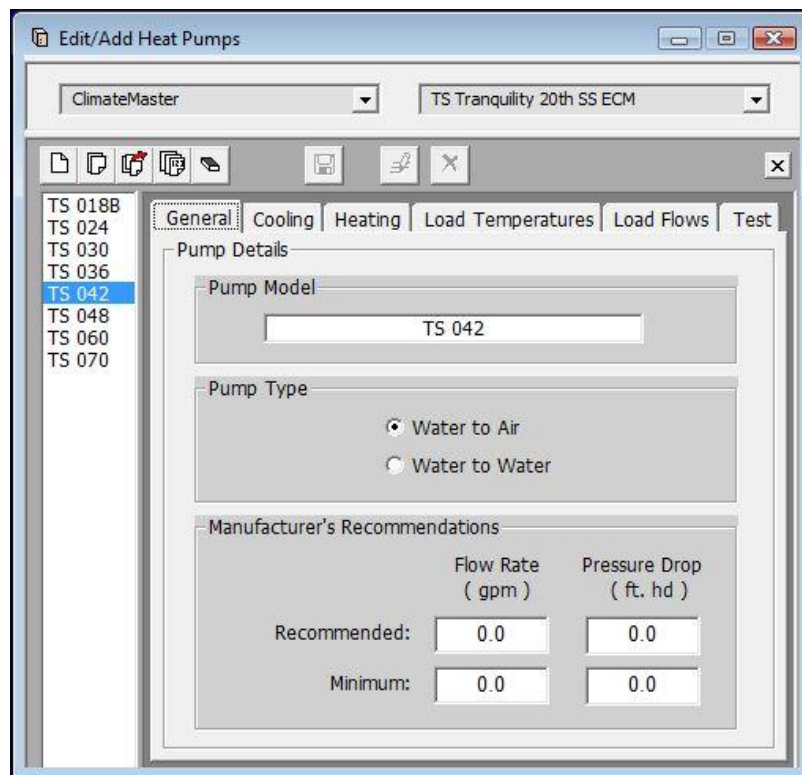


Fig. 2.2 Pump Edit Pane

Pump Series Controls

The Pump Series control buttons, shown in figure 2.3, are found above the list and the pump data panels. They include the Pump Edit controls (*New*,

Copy, Remove, Reorder, and Clear,) the pump *Save* control, the *Edit Pump Information* control, and the *Delete Series* control.

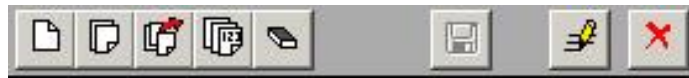


Fig. 2.3 Pump Series Controls

Pump Edit Controls

The Pump Edit Control buttons are designed to work directly with the pump list. New pumps are added by pressing the *New* button. Copies of existing pumps are added with the *Copy* button. *Remove* is used to remove a pump from the list. *Reorder* is pressed to reorganize the list, both alphabetically and numerically. *Clear* is used to delete all pumps from the current list. **Be careful not to accidentally delete pumps.**

Save Control

The *Save* control button can be used at any time to save the current pump information.

Edit Pump Information Control

The *Edit Pump Information* control button allows the user to edit both the series and the manufacturer information for a given pump. **Note, however, that if the manufacturer information is changed, it will change for every series connected to that manufacturer.** 'Proceed' or 'Cancel' will return the user to the Pump Edit Pane.

Delete Series Control

The *Delete Series* control button deletes the current series. If the series is the only series of a manufacturer, the manufacturer also will be deleted automatically.

Note: The actual heat pump file (.hpd) will not be deleted from the **pumps** directory. If necessary, the series can be restored by creating a 'New Series'. The user need only provide the appropriate manufacturer and series name, and use the deleted '.hpd' filename for the pump set 'Filename'. Incomplete fields will be recreated from the *.hpd file. If the original file no longer*

*exists, the program creates a new *.hpd file. Incidentally, the same system can be used to add new pump sets obtained from external sources as described below.*

General Information

The General panel is the first panel a user sees when he or she decides to input data for a new pump. It has an input box for the name of the pump, and in the 'Pump Type' area, the user selects whether the pump should be classified as a water-to-air or a water-to-water pump. An example of the pump *General* panel is shown in the lower right pane of figure 2.2.



The General panel also now has recommended and minimum pressure drop and flow rate input boxes for each heat pump. These data have been included for use in pressure drop calculations performed in an upcoming release of GLD Residential. In a future version of GLD Premier, these data may also be utilized in conjunction with the new CFD module.

Capacity, Power, and Flow Rates

The capacity, power, and flow rate information pertaining to the source side flow for both heating and cooling are entered into the two tabbed panels labeled *Cooling* and *Heating* in the Pump Edit pane. An example of the *Cooling* panel is shown below in figure 2.4. The *Heating* panel follows an identical format, although the temperatures will be different.

Heat Pump Specifications for Cooling - SOURCE					
FLOW RATE 1			FLOW RATE 2		
5.4 gpm			10.0 gpm		
EWT (deg F)	Capacity (MBtu/hr)	Power (kW)	Capacity (MBtu/hr)	Power (kW)	
77.0	33.9	2.81	37.0	2.66	
95.0	31.3	3.19	35.2	3.06	
115.0	28.6	3.59	33.2	3.49	
Coefficients:					
	Capacity	Power	Flow Factor:		
a	46.84028	0.97056	1.13	0.96	
b	-0.18719	0.02614			
c	0.00025	-0.00003			
Calculate Coefficients					

Fig. 2.4 Heat Pump Specifications – Cooling

As can be seen from the figure, the source entering water temperature (EWT) is listed to the left, and the capacity and power requirement of the unit at different flow rates are listed to the right. Once the values are input, the coefficients and flow factor can be calculated from the entered data. The *Calculate Coefficients* button turns red when values are changed, indicating that new coefficients must be calculated before proceeding.



Note: If data for only one flow rate are available, only the first capacity and power requirement data must be included, under the section entitled 'FLOW RATE 1'. The data under 'FLOW RATE 2' can be left as zeroes, and the program will ignore them, leaving the flow factor as 1.0.

Load Side Corrections

Corrections resulting from variations in inlet temperatures and flow rates on the load side can be entered in the *Load Temperatures* and *Load Flows* tabbed panels of the Pump Edit pane. If these corrections are not added, the factors remain at 1.0 and input variations in load temperature or flow rate will have no effect on calculated capacities and/or input power. Time permitting, however, it is best to include as much information as possible from what the manufacturer provides.



Load Temperatures Panel

The *Loads Temperatures* panel is where corrections for variations in the load inlet temperature are input. Both the cooling and heating information (taken at the average or standard source temperature and flow rate, and the average load flow rate) are entered on the same panel, an example of which is shown in figure 2.5.

The factors shown in figure 2.5 were calculated from a manufacturer's list of capacities provided for the different temperatures, using the capacity at the selected temperature as the numerator and the capacity at 67°F for cooling (70°F for heating) as the denominator. The 67°F (70°F) capacity values were those used for the inlet source data on the *Cooling* and *Heating* tabbed panels described previously. Occasionally, manufacturers will provide capacity values at the standard temperature with a table of correction factors that can be entered into the GLD *Load Temperatures* panel directly.

Notice how in figure 2.5 five points of data are included for cooling but only three are included for heating. The software requires a minimum of three data points for its coefficient



calculation. More data may be input if desired. **However, no boxes may be left blank!** Other temperature *and* coefficient values must be set to zero in this case. As a convenience, '0' buttons are included to quickly set rows to zero.

The screenshot shows the 'Heat Pump Load Temperatures Panel' with tabs for General, Cooling, Heating, Load Temperatures, Load Flows, and Test. The 'Load Temperatures' tab is active, displaying 'Temperature Corrections - LOAD'.

COOLING:

EAT-WB (deg F)	Capacity Factor	Power Factor
61.0	0.910	0.960
64.0	0.960	0.980
67.0	1.000	1.000
70.0	1.050	1.020
73.0	1.090	1.040

HEATING:

EAT-DB (deg F)	Capacity Factor	Power Factor
60.0	1.050	0.980
70.0	1.000	1.000
80.0	0.940	1.020
0.0	0.000	0.000
0.0	0.000	0.000

Below the tables are coefficient input fields (a, b, c, d) for both cooling and heating, and a 'Calculate Coefficients' button.

Fig. 2.5 Heat Pump Load Temperatures Panel



Note: If correction factors are unknown or unnecessary, they can all be left at the constant value of 1.0, which is the initial condition that exists when a new pump is first added.

Load Flows Panel

Similar to the *Load Temperatures* panel, the *Load Flows* panel allows the user to enter corrections for variation in load side flow rates. **The system used here is different, however.** Every pump is assigned a *nominal* flow rate, and the data is input as percentages of the nominal flow rate. A sample *Load Flows* panel is shown in figure 2.6.

To get a capacity factor at a flow rate of 80 percent of nominal, for example, the capacity of the unit at 80 percent of nominal would be divided by the capacity at the nominal flow rate. The procedure is identical for the power factors. Data is usually taken at standard source temperatures and flows, and at the standard load

temperature. Quite often, the manufacturer provides lists of these variations that can be input directly.

Once again, a minimum of three points is necessary for the coefficient calculations, and '0' buttons are provided for quickly setting the unused rows to zero. **Remember - boxes must be set to 0 if they are not used!**



Flow Corrections - LOAD

Nominal Flow Rate **1140** CFM

COOLING:

% of Nominal	Capacity Factor	Power Factor
80.0	0.974	0.965
90.0	0.987	0.981
100.0	1.000	1.000
110.0	1.012	1.020
120.0	1.025	1.042

HEATING:

% of Nominal	Capacity Factor	Power Factor
80.0	0.972	1.032
90.0	0.986	1.016
100.0	1.000	1.000
110.0	1.020	0.984
120.0	1.042	0.968

Below the tables are coefficient input fields:

	a	b	c	d
COOLING	0.78510	0.00368	-0.000026	0.0000001
HEATING	0.82300	0.00354	-0.000034	0.0000002

Buttons for setting unused rows to zero (0) are provided next to the 110.0 and 120.0 rows in both tables.

Calculate Coefficients

Fig. 2.6 Heat Pump Load Flows Panel

Testing Input Data

The *Test* panel is provided as a final check after a pump's data has been input into the Heat Pump module. Without testing the data directly, there is no way to know if mistakes were made during the input process.

A sample Test panel is shown in figure 2.7. As can be seen from the figure, both source and load entering water and air temperatures, as well as flow rates, can be edited directly. Clicking the "Test" button performs the calculation to see what capacity, power, and EER/COP result from the chosen input parameters. Average values are used initially, but by varying the parameters the designer can see how well the newly created model matches the data set used for data entry.

The screenshot shows a software window titled 'Test' with tabs for General, Cooling, Heating, Load Temperatures, Load Flows, and Test. The Test tab is active, displaying a table with three columns: SOURCE, LOAD, and RESULTS. The table has two rows of data. The first row is for a cooling test, with SOURCE (EWT) at 95.0 deg F and Flow at 7.7 gpm, LOAD (EAT-WB) at 66.2 deg F and Flow at 1140 CFM, and RESULTS (Capacity, Power, EER/COP) at 33.1 MBtu/hr, 3.12 kW, and 10.6 respectively. The second row is for a heating test, with SOURCE (EAT-DB) at 45.0 deg F and Flow at 7.7 gpm, LOAD (EAT-WB) at 68.0 deg F and Flow at 1140 CFM, and RESULTS (Capacity, Power, EER/COP) at 31.7 MBtu/hr, 2.58 kW, and 3.6 respectively. A 'Test' button is located at the bottom center of the panel.

SOURCE		LOAD		RESULTS		
EWT (deg F)	Flow (gpm)	EAT-WB (deg F)	Flow (CFM)	Capacity (MBtu/hr)	Power (kW)	EER/ COP
95.0	7.7	66.2	1140	33.1	3.12	10.6
EAT-DB (deg F)		EAT-WB (deg F)				
45.0	7.7	68.0	1140	31.7	2.58	3.6

Fig. 2.7 Heat Pump Test Panel

Often, any input errors will be evident immediately from the test (by comparing the test results with the input sheet). Additionally, the user can use this test to make certain that the pump data are accurate over the particular range of temperatures, flows, etc. that he or she typically uses, and then modify the data if necessary.

Exiting the Edit/Add Heat Pumps Module

After editing or adding heat pumps, and calculating all necessary coefficients, the user should make sure that the pumps are saved by clicking the *Save* button on the Pump Series control bar. When the pumps are securely saved, the *Save* button will become disabled.

Clicking the close button in the upper right hand corner of the lower pane closes the Pumps Edit Pane, and clicking the close button in the upper right hand corner of the Edit/Add Heat Pumps window closes the Edit/Add Heat Pumps module. Closing without saving edited data will initiate a dialog box that reminds the user to save the data before closing.

Heat Pump File Descriptions

There are two types of files created by the Edit/Add Heat Pumps module. The first is the **Pumplist.gld** file, which maintains the current master list of manufacturers and the series associated with those manufacturers. The

Pumplist.gld file also includes the filenames (without the '.hpd' extension) of the heat pump data files associated with the individual series.

The second type of file is the '.hpd', heat pump data file, for each individual series of pumps. This file type keeps track of all the data input by the user as well as the pump names and the coefficients calculated within the module. Since '.hpd' files cannot be deleted by the program (unless they are accidentally overwritten), many difficulties usually can be overcome by just adding new pump sets or, if necessary, editing the **Pumplist.gld** file directly. The format of the **Pumplist.gld** file is given in the Preface, page 3.

Adding Pump Sets Obtained From External Sources

To provide the greatest amount of flexibility to the user, GLD allows the user to obtain heat pump data files (*.hpd files) from external sources. For example, a heat pump set may be copied from a fellow designer, or even downloaded from a participating heat pump manufacturer's website.

Since the original **Pumplist.gld** file does not contain a reference to the externally obtained data set, it must be added manually. The procedure for this is as follows:

1. Place the '*.hpd' file into the \GLD\pumps folder.
2. Add a 'New Series'.
 - a. If the series belongs to an existing manufacturer, choose the appropriate manufacturer.
 - b. If the series belongs to an unlisted manufacturer, choose 'New Manufacturer' from the list.
3. Provide the 'Series Name' and 'Manufacturer Name', as required.
4. Under 'Filename', **type the existing filename** of the series to be added. Note-the existing filename is the *.hpd file the user just put into the pumps folder in step 1 above.
5. Click "Proceed".

GLD will open the heat pump file for editing and will include it in its Heat Pump Database. Additionally, if this is a new manufacturer, any included manufacturer information will become visible for this pump set. Since the **Pumplist.gld** file has been modified, it will register the new pumps for use in all modules opened afterwards.

Other Resources

For additional information and specific instructions on how to enter pump data step-by-step, please visit the following website:

<http://www.gaiageo.com/webresources.htm>

CHAPTER 3

Loads and Zones

All of the calculations performed in GLD fundamentally are based on loads provided by the designer. This chapter describes the unique GLD loads system, and how to enter the loads in both the Zone Manager and the Average Block loads modules. In GLD 2010 Professional/Standard Edition, designers can use monthly loads data. Users that upgrade to the Premier edition can use 8760 hourly data in the Average Block loads module for precise design control. A description of how to prepare and use these data are included. Additionally, it explains the pump matching capabilities and operation both in automatic and in manual modes. At the end of the chapter there is an explanation of how to import external loads files as well as a brief review of the program's loads input methodology.

The GLD Loads Model

The intrinsic flexible nature of the GLD Geothermal Design Studio appears again in the loads models the software employs: the user is not limited to a single style of loads input. Similar to the design modules, a designer can choose between different types of loads input schemes based on the level of complexity he or she desires and the time he or she wishes to invest. These loads modules are then 'linked' to one or more design modules using the GLD Studio link system. Currently, two loads modules are available, the Zone Manager Loads module and the Average Block Loads module

The Zone Manager Loads module is provided for designers who desire a full analysis capability. Loads are input as separate zones, and each zone is matched with a particular pump. This mode is more valuable when users require thorough designs.

The Average Block Loads module offers a rapid system of entering whole systems information for users who do not require or desire to input the data for a fully zone-divided installation. Rather than matching specific pumps to each zone, the Average Block Loads module uses a particular, user-defined style of pump (or COP) and matches it in an average way to the entire installation. Although the input scheme is simpler, the design calculations are identical to those of the more complex Zone Manager Loads module. In fact, on average, if identical values could be placed in both the Zone Manager and Average Block loads modules, identical calculated bore lengths would result.



The Average Block Loads module optionally can accept monthly loads (total and peak) data. New in the GLD 2010 Edition, the Average Block Loads module also can accept 8760 hourly loads data. When the user inputs these monthly and/or hourly data, the program provide a number of calculated outputs included monthly/hourly borehole evolution temperatures, heat pump performance on a monthly or hourly basis, graphical representations of the thermal storage effects from balanced loads profiles, etc.

Zone Files

Zone (loads) files are stored as ***.zon** files in the GLD ‘**zones**’ directory. They have a general format that can be read into any loads module, and they **can be used simultaneously** in different design modules. However, if this is done, it may be wise to save any changes under different filenames.

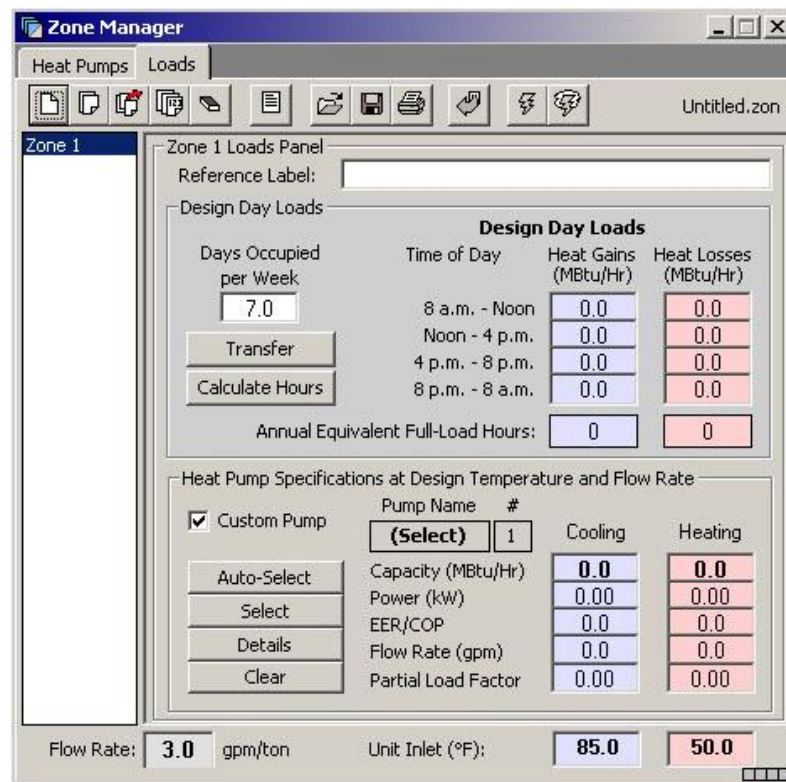
Both loads modules are stand-alone entities. **The files are entirely independent of project design files.** This means that an entire installation loading design can be entered, matched with pumps, optimized, and saved without ever opening a design module. This is valuable for users who wish to keep the loads entry and pump selection completely separate from the studio’s geothermal design modules. Now users can work on designs and load inputs at different times, and can use the same loads files for various projects/styles of project.

New zone files can be created by clicking the ‘*New*’ button in any loads module, or by clearing all of the current loads information with the *Clear* button, followed by the ‘*New*’ button. The designer provides a filename when the zone file is saved.

Zone files can be opened and saved using the *Open* and *Save* buttons on the Loads panel.

The Zone Manager Loads Module

For commercial, non-centralized installations, it is often necessary to divide loads into separate zones that individually are served by specific heat pumps. This type of system has many advantages including lower installation and service costs as well as a highly accurate method of matching the loads to the heat exchanger. From the time-specific loads data that the user provides, GLD determines the maximum heating and cooling loads of the entire system, and then uses these values to calculate the length of heat exchanger required.



Zone Manager

Heat Pumps Loads

Untitled.zon

Zone 1

Zone 1 Loads Panel

Reference Label:

Design Day Loads

Days Occupied per Week:

Transfer

Calculate Hours

Design Day Loads

Time of Day	Heat Gains (MBtu/Hr)	Heat Losses (MBtu/Hr)
8 a.m. - Noon	0.0	0.0
Noon - 4 p.m.	0.0	0.0
4 p.m. - 8 p.m.	0.0	0.0
8 p.m. - 8 a.m.	0.0	0.0

Annual Equivalent Full-Load Hours:

Heat Pump Specifications at Design Temperature and Flow Rate

☒ Custom Pump

Pump Name	#	Cooling	Heating
(Select)	1	0.0	0.0
Capacity (MBtu/Hr)		0.00	0.00
Power (kW)		0.0	0.0
EER/COP		0.0	0.0
Flow Rate (gpm)		0.0	0.0
Partial Load Factor		0.00	0.00

Flow Rate: gpm/ton Unit Inlet (°F):

Fig. 3.1 Zone Manager Loads Module, Main View

The Zone Manager loads module can be opened either from the *Loads* Menu or by clicking the Zone Manager toolbar button. An example of the module opened to the *Loads* tabbed panel is shown in figure 3.1. The *Heat Pumps* tabbed panel will be discussed shortly.

In the *Main View*, Zones in GLD are organized in a list on the left side of the *Loads* tabbed panel. Each zone panel contains information relating to the working zone, including a zone name, the loading information, and the information about any heat pumps selected for that zone. Selecting a different zone name in the zone list changes the working zone.

Using the list, the designer can bring up and modify any particular zone by clicking on its name. An essentially equivalent but more compact summary of the

input data can be obtained in the *Summary View*, obtained by clicking on the *Summary View* toggle button. Different representations of zone data can also be printed as reports.

Managing Zones in the *Loads* Tabbed Panel

The buttons along the top of the *Zone Manager* are used to work with the zones. A closer view is shown in figure 3.2.

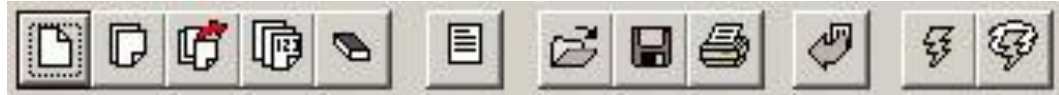


Fig. 3.2 Zone Manager Control Buttons

The five buttons on the left side are zone-editing controls, and they include *New*, *Copy*, *Remove*, *Renumber*, and *Clear*. A *Summary* view of all the zones can be obtained by hitting the sixth, or *Summary View*, toggle button. The next three buttons are the *Open* and *Save* buttons, for opening and saving the zone files, and the *Print* button, for printing various zone reports. The next button is the *Import Loads* button, a description of which can be found towards the end of this chapter under “Importing Loads Data from External Programs.” The final two buttons on the far right are for pump selection across the entire set of zones, and include *Auto-Select All* and *Update/Reselect*, which are discussed in more detail below.

New and **Copy**

A new zone may be created at any time from the *Loads* panel by clicking the *New* button. Identical zones may be created from any existing zone by bringing up that zone’s data window and clicking the *Copy* button.

Remove and **Clear**

Zones also can be deleted from the list. Any zone can be removed from the list by bringing up the zone’s data window and pressing the *Remove* button. To delete all of the zones in the list, press the *Clear* button.

Renumber

If several zones are added or removed from the list, click the *Renumber* button to reorganize the zones. This button rennumbers the existing zones from one, starting with the first zone in the current list.

Summary View Toggle Button

With the *Summary View* toggle button, the user can at any time simultaneously look at the group of zones. This view provides lists of the heat pump data in both cooling and heating modes as well as collective information about the set of chosen pumps. This information includes the peak loads and when they occur, and the total combined capacity, the peak demand, and the average efficiency of the selected equipment. Although individual pumps cannot be added or removed in the *Summary View*, changes made across the entire pump selection are directly observable. A sample *Summary* panel is shown in figure 3.3. Note that more than one type of pump series is listed.

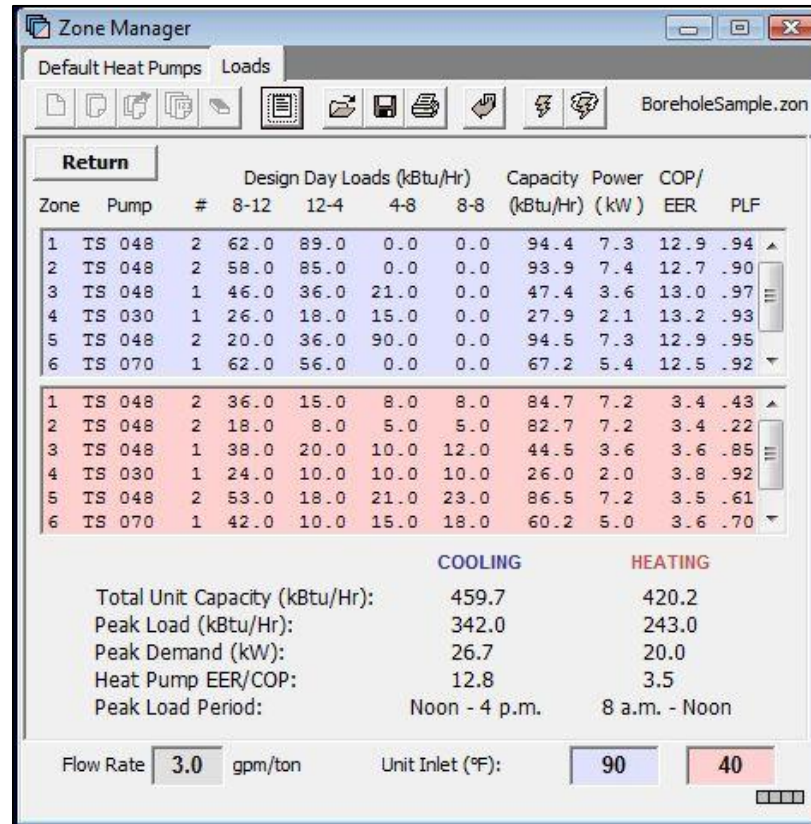


Fig. 3.3 Zone Manager Summary View

Entering Loads

Loads can be entered directly in the individual zone data windows back in the *Main View* of the *Loads* tabbed panel. A sample entry is shown in figure 3.4. The GLD loads input methodology may be new for some designers. Consequently, an additional and alternative description of the methodology can be found at the end of this chapter.

Design Day Loads

According to the model that GLD uses in the Zone Manager, average peak load data for every hour of a twenty-four hour day can be included if desired. However, for simplification, average peak loads for the design day, or the day of heaviest usage in the year for both cooling (heat gains) and heating (heat losses) modes of operation, can be input for up to four separate times of the day. These include morning (8 a.m. to 12 noon), afternoon (12 noon to 4 p.m.), evening (4 p.m. to 8 p.m.) and night (8 p.m. to 8 a.m.). **This method of input not only provides the total load, but also identifies when the equipment will be in use for the heat exchanger calculations.**

Design Day Loads			
Days Occupied per Week	Time of Day	Heat Gains (MBtu/Hr)	Heat Losses (MBtu/Hr)
5.0	8 a.m. - Noon	46.0	38.0
	Noon - 4 p.m.	36.0	20.0
	4 p.m. - 8 p.m.	21.0	10.0
	8 p.m. - 8 a.m.	0.0	12.0
Annual Equivalent Full-Load Hours:		1050	220

Fig. 3.4 Sample Loads Input Data

If only one peak value during the day is provided to the designer, it can be entered into one or several of the time slots, depending on how the loads will be expected to change during the course of a day. Slightly reduced values can be added for off-peak hours if the building still will be in operation but not at full load. Insignificant time slots can be left at zero.



Note: If only one peak load value is provided per zone, the designer will need to be consistent in placing it in the same time slot for every zone. This is because the software loops through all of the zones to determine which time of day has the highest loading requirements prior to performing its calculations.

If only cooling or only heating loads data are to be used, all of the non-used slots **should remain as zeroes**. Only the side with the loads provided will be calculated.

Annual Equivalent Full-Load Hours

The hours entered into the lower section of figure 3.4 are determined from detailed annual loads data for the system being designed. They represent the annual number of hours the system will be running if operating at full load, and are a measure of the system running time.

This system is used both to limit the amount of data the user must enter and to simplify the calculations. It is identical to methods that require input of all the

monthly data but more concise, since it represents the total energy input to the ground in terms of the peak load. Month-to-month variations are not necessary in the annual/monthly/daily pulse model used in GLD.

For example, if a loading report provides the number of Btus required by this zone each month, the hours per month will be obtained by dividing the monthly Btu requirement by the peak Btu/h value. The resulting number will be the monthly equivalent full-load hours. To get the annual full-load hours, the value will need to be obtained for every month that required heating or cooling, and then combined to finally get the annual equivalent heating or cooling hours.



If exact values are not available, an estimate should be made with regard to the expected running time of the unit in each particular zone. Estimates of time must be reduced, of course, from actual running time since the ‘annual equivalent full-load hours’ represents the running time if the system were operating continuously at full load, which is not generally the case.

Equivalent Hours Calculator

To aid in this calculation, GLD includes the *Equivalent Hours Calculator*, found in the *Tools* menu, or obtainable directly by clicking the ‘Calculate Hours’ button. Figure 3.5 shows a view of the Equivalent Hours Calculator.

The screenshot shows a window titled "Equivalent Hours Calculator". Inside, there's a section for "Annual Equivalent Full-Load Hours". It includes a "Peak Hourly Load" field with the value "0.0" and unit "MBtu/hr". Below that is a "Monthly Total Loads" section with a list of months from January to December, each followed by a text box containing "0.0" and the unit "MBtu". At the bottom, there's a "Full-Load Hours" field with the value "0" and unit "hr". There are two buttons at the very bottom: "Close" and "Calculate".

Fig. 3.5 Equivalent Hours Calculator



Remember that although the vertical bore length calculation results are not extremely dependent on the running hours within one zone for multi-zone designs, the total number of running hours across the zones can certainly affect the required bore length. The user should attempt to enter the running hours as accurately as possible.

Equivalent hours are unnecessary for a surface water design, since long-term buildup effects are unimportant. If a loads module is linked to a Surface Water Design module, the hours will not be visible.

Days per Week

This value represents the occupation of the installation, in days per week. The building in the example is only occupied during weekdays, so the value 5.0 was entered. Decimal values can be used for partial occupations, and the amount can vary between zones. If the heat loss calculations embody occupancy data, then days per week can be left at the value 7.0

Again, the occupation is unnecessary for a surface water design, since long-term buildup effects are unimportant. If a loads module is linked to a Surface Water Design module, the days per week will not be visible.

Pump Matching and Selection

Every zone has heat pump equipment associated with it. Equipment matching and selection is done within the zone data window, in the lower section entitled, “Heat Pump Specifications at Design Temperature and Flow Rate”. In this section, the designer has three choices when matching a pump to a zone:

- Automatic selection based on the active heat pump series
- Manual selection from a list of all available pumps
- Custom input of pump data

Once selected, the zone retains all of the information associated with the pump chosen. This information includes the pump name, the number of pumps, and the capacity, power consumption, EER/COP, flow rate, and partial load factor in both cooling and heating modes. If obtained from the list of available pumps, detailed information is also available, including the manufacturer and series name, the pump type, and the inlet load temperatures.

Figure 3.6 shows the pump selection section of the zone data window with sample data matched to the loads data of figure 3.4.

Several buttons can be found in the pump selection section. These include *Auto-Select*, *Select*, *Details*, and *Clear*. A checkbox is also included to indicate when the pump is a ‘custom pump’, or a pump not included in GLD’s internal list of pumps.

Heat Pump Specifications at Design Temperature and Flow Rate				
		Pump Name	#	
<input type="checkbox"/> Custom Pump		TS 048	2	
Auto-Select	Capacity (kBtu/Hr)	94.4		84.7
Select	Power (kW)	7.31		7.19
Details	EER/COP	12.9		3.4
Clear	Flow Rate (gpm)	22.3		9.0
	Partial Load Factor	0.94		0.43

Fig. 3.6 Sample Pump Selection Section with Data

Auto-Select

This option is by far the easiest method of matching a pump to the loads in a particular zone. By clicking the *Auto-Select* button, GLD utilizes the information stored for the active pump series and determines which pump within the list is best suited to the zone in question. If the listed pumps are too small for the zone loads, the software increases the number of pumps of each size until an acceptable match is achieved.

The pump selection process uses information from the Zone Manager loads module. This includes the chosen inlet source temperature, the flow rate, the heat pump series, and the initial inlet load temperatures. The flows and load temperatures can be entered at the bottom of the module, and the active heat pump series and load temperatures may be changed on the *Heat Pumps* tabbed panel.

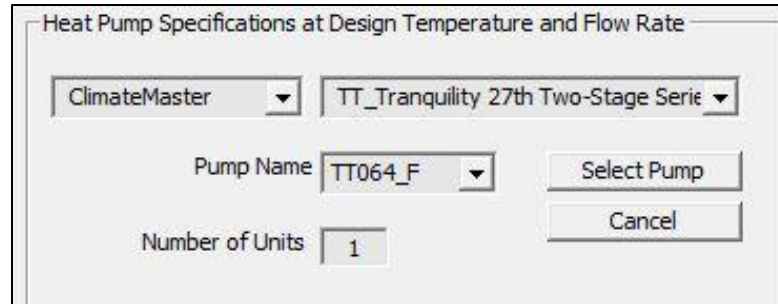
Manual Select

If an automatically selected heat pump is for any reason undesirable, or a different pump series from the same manufacturer, or even from a different manufacturer is required, the *Select* button may be used. This button allows the designer to choose any of the stored pumps. As with the *Auto-Select* button, all of the associated fields are calculated automatically once the pump is selected.

When the *Select* button is pressed, the selection panel appears, as shown in figure 3.7. After a pump is chosen, pressing *Select Pump* will place the pump in the zone and automatically calculate all of the associated parameters. *Cancel* will return the user to the main display without changing any pumps.

Note: Unlike with Auto-Select, a pump that is manually selected may or may not match the loads in the zone. It is the responsibility of the designer to make sure the pumps match the zones.





Heat Pump Specifications at Design Temperature and Flow Rate

ClimateMaster TT_Tranquility 27th Two-Stage Serie

Pump Name TT064_F

Number of Units 1

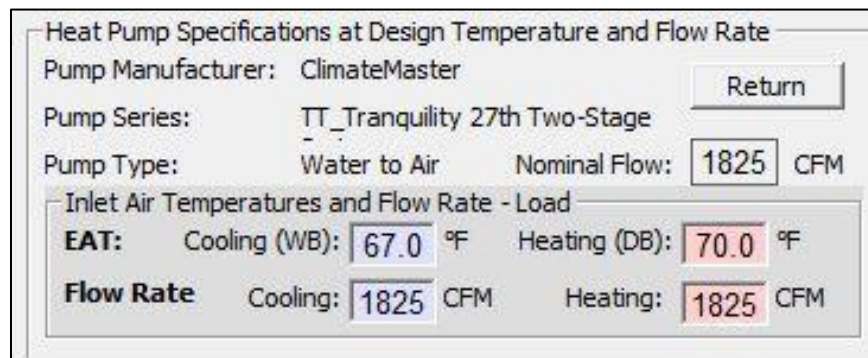
Select Pump

Cancel

Fig. 3.7 Pump Selection Panel

Details

Specific details about a given pump may be obtained by clicking the *Details* button. Additionally, the details panel is where the designer may vary the loads input temperatures or flows for that particular pump. *After the user presses the return button, variations in the input load temperature will affect the pump parameters listed on the main pump selection area.* A sample details panel is shown in figure 3.8.

Heat Pump Specifications at Design Temperature and Flow Rate

Pump Manufacturer: ClimateMaster

Pump Series: TT_Tranquility 27th Two-Stage

Pump Type: Water to Air

Nominal Flow: 1825 CFM

Inlet Air Temperatures and Flow Rate - Load

EAT: Cooling (WB): 67.0 °F Heating (DB): 70.0 °F

Flow Rate Cooling: 1825 CFM Heating: 1825 CFM

Return

Fig. 3.8 Pump Details Panel

Clear

Pressing the *Clear* button clears the current pump in a zone. All values are reset to the initial state, allowing the user to reselect or enter a pump for the zone.

Custom Pump (Customization)

If the designer must include a heat pump unit that is not stored in GLD's Heat Pump Database, he or she may add customized pumps simply by entering values directly into the boxes on the pump selection section of the zone data window. When the user does this and overrides the automatic selection features, a check appears next to the "Custom Pump" label, indicating that the pump information is

from an external source. The details section will no longer contain information about the pump manufacturer, series, or type.

The calculation portion of GLD will require at least the capacity and power data to utilize the pump properly. *The actual COP used in the calculations is determined from the capacity and the power, not the input text box.* Other information may be added for the designer's reference.



*Note: When a custom pump is included, its values will remain unchanged during the designing process. **Variations in inlet source or load temperatures, or system flow rate, will not affect a customized pump's data.***

Automatic Heat Pump Selection Options for the Entire Zone Set

Two controls are included with GLD that allow for an automatic selection of pumps throughout the entire set of zones. This feature is useful when the pump set needs to be compared or changed, or when modifications are required throughout the existing set. These controls are necessary so that large sets of pumps can be changed or updated without having to step through each individual zone.



Auto-Select All Pumps

The *Auto-Select All Pumps* control performs the same function as the *Auto-Select* button in the pump selection section of the zone data window, except it performs the selection sequentially through all of the zones. It uses the active heat pump series selected on the *Heat Pumps* tabbed panel.

Note: Auto-Select All Pumps will overwrite all currently selected pumps, including custom pumps.



Update/Reselect Current Pumps

The *Update/Reselect Current Pumps* control reselects the pumps in all zones after determining the current series used in each particular zone. For example, if most of the pumps belonged to the same water-to-air series, but one was a water-to-water pump, this control would determine the difference and update the pumps accordingly.

Note: Custom pumps are not affected when the Update/Reselect Current Pumps control is activated.

Working Series Selection in the Heat Pumps Tabbed Panel

Figure 3.9 Shows the Zone Manager opened to the *Heat Pump* tabbed panel. This panel is used to specify the working series for all of the automatic selection features described for the *Loads* tabbed panel. In the *Heat Pump* tabbed panel, the user simply selects the pump series that he or she intends to use for the matching session. The selection may be changed at any time without affecting previously automatically selected units. **However, if the ‘Auto-Select All Pumps’ button on the *Loads* panel is pressed, every zone will be replaced with the current working series.** Additionally, in this panel the user may define an inlet load temperature to be used in any automatic selection.

Choosing the Active Series

The active heat pump series is the series of heat pumps used by the *Auto-Select* features in the *Loads* panel.

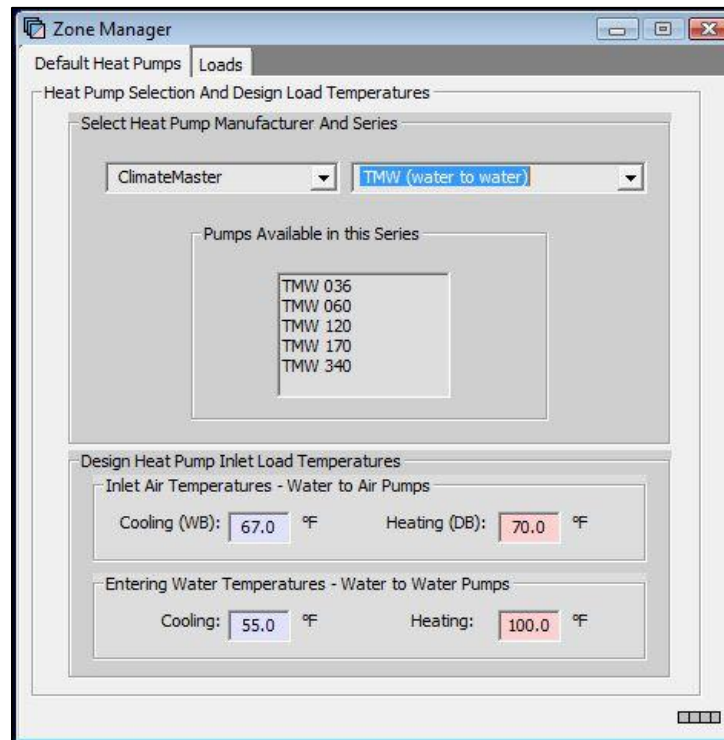


Fig. 3.9 Heat Pumps Tabbed Panel

It represents the primary heat pump family utilized by the designer for a particular project. Although this is the primary series, other pumps may still be selected for certain zones using either the *Select* button or by defining a custom pump. To choose a pump series, select a manufacturer, followed by the desired series of that manufacturer. A list of available pumps appears in the list box.

Inlet Load Temperatures



Values for the initial inlet load temperatures for both water-to-air and water-to-water pumps may be entered in the appropriate boxes. If necessary, these values may be changed for individual pumps in the *Loads* panel. For water-to-air pumps, 'WB' refers to "Wet Bulb" and 'DB' refers to "Dry Bulb" temperatures.



The Average Block Loads Module



If detailed zone-style modeling is unnecessary for an initial calculation, or if information is incomplete for a component-based design, or if the user desires to calculate monthly and/or hourly inlet temperatures, or if the user wishes to estimate the benefits of the thermal recharge "battery" from a system for a borehole design, the Average Block loads module is a better option than the Zone Manager (and in the case of monthly and hourly inlet temperatures, the only option). The required input consists of only a single set of loads, which represents the entire installation. This single set of loads data optionally can be entered in a new month-by-month loads screen for inlet temperature calculations. Hourly loads data can be imported using a .csv file or a proprietary file type from one of several energy simulation tools that now interact bi-directionally with GLD Premier 2010.

The pump matching model in the Average Block module is slightly different from the model for the individual zones. A single pump type is selected from the GLD Heat Pump Database to approximate the average pump characteristics of the installation. For example, if the designer is planning to use the highest efficiency pumps, a pump in a series with a higher coefficient of performance (COP) might be chosen over a lower efficiency pump. If specific pump characteristics are required, they can be input directly, overriding the automatic functions.

Two views of the Average Block Loads Module are shown in figures 3.10 and 3.11. Although it resembles a single zone in the *Loads* tabbed panel of the Zone Manager loads module, it has some differences: there is an hourly data check box, a monthly loads button, there is no list of zones, and the pump matching section has a different format.

Average Block Loads Untitled.zon

Reference Label:

Design Day Loads

7.0 Days / Week

☐ Hourly Data

Transfer

Calculate Hours

Monthly Loads

Time of Day	Heat Gains (kBtu/Hr)	Heat Losses (kBtu/Hr)
8 a.m. - Noon	0.0	0.0
Noon - 4 p.m.	0.0	0.0
4 p.m. - 8 p.m.	0.0	0.0
8 p.m. - 8 a.m.	0.0	0.0

Annual Equivalent Full-Load Hours: 0

Heat Pump Specifications at Design Temperature and Flow Rate

☒ Custom Pump

Pump Name: (Select)

	Cooling	Heating
Capacity (kBtu/Hr)	0.0	0.0
Power (kW)	0.00	0.00
EER/COP	0.0	0.0
Flow Rate (gpm)	0.0	0.0
Partial Load Factor	0.00	0.00

Flow Rate: 3.0 gpm/ton

Unit Inlet (°F): 85.0 50.0

Fig. 3.10 Average Block Loads Module

Average Block Loads Untitled.zon

Monthly Load Data

Update

Cancel

	Cooling		Heating	
	Total (kBtu)	Peak (kBtu/hr)	Total (kBtu)	Peak (kBtu/hr)
January	0	0	0	0
February	0	0	0	0
March	0	0	0	0
April	0	0	0	0
May	0	0	0	0
June	0	0	0	0
July	0	0	0	0
August	0	0	0	0
September	0	0	0	0
October	0	0	0	0
November	0	0	0	0
December	0	0	0	0
Total:	0	3.0	0	3.0

Hours at Peak: 3.0 3.0

Flow Rate: 3.0 gpm/ton

Unit Inlet (°F): 90 40

Fig. 3.11 Monthly Loads Input Boxes in Average Block Module

Managing the Average Block Loads

The buttons along the top of the *Average Block Loads* module are used to work with the single panel of loads information. A closer view is shown in figure 3.12.



Fig. 3.12 Average Block Loads Module Controls

The buttons on the left are zone-editing controls, and include only *New* and *Clear*. To the right are the *Open* and *Save* buttons, for opening and saving the zone files, along with the *Print* button, for printing various zone reports. The last button on the right is the *Import Loads* button which is explained towards the end of this chapter. Unlike the Zone Manager, there are no Auto-Select buttons.

New

A new set of loads data may be created initially by clicking the *New* button. Since only one panel is allowed, this button becomes disabled after a new set appears. It is re-enabled when the set is cleared.

Clear

To delete all of the current information, press the *Clear* button.

Entering Loads

The method of entering loads data into the Average Block is nearly identical to the method used in the Zone Manager. There are two main differences. The first is that the summed loads values may be larger than the smaller values used in individual zones. Refer to the Zone Manager Entering Zones section or the end of this chapter for specific details about the *Design Day Loads*, *Annual Equivalent Full-Load Hours*, and *Days Occupied per Week* sections. Note that the Annual Equivalent Full-Load Hours can be calculated for the entire installation using the *Equivalent Hours Calculator*.

The second difference is that the loads data entry method has been expanded to accept monthly and/or hourly loads data. Monthly and hourly loads data are necessary for calculating monthly and hourly inlet temperatures (and monthly and hourly heat pump performance), respectively, in the borehole design module. **Note that adding monthly and/or hourly loads is necessary only if a designer wishes to calculate monthly and/or hourly inlet temperatures for a vertical borehole heat exchanger.**

Monthly Loads

To access the monthly loads data input panel as seen in figure 3.13, click on the *Monthly Loads* button.

Cooling		Heating	
Total (kBtu)	Peak (kBtu/hr)	Total (kBtu)	Peak (kBtu/hr)
0	0	0	0
0	0	0	0
0	0	0	0
0	0	0	0
0	0	0	0
0	0	0	0
0	0	0	0
0	0	0	0
0	0	0	0
0	0	0	0
0	0	0	0
0	0	0	0
0	0	0	0
Total:	0	3.0	0
Hours at Peak		Hours at Peak	

Fig. 3.13 Monthly Loads Input Boxes

There are three ways to enter the loads data:

- Manually enter the total and peak cooling and heating loads in the appropriate boxes.
- Copy and paste from Excel using the Excel icon button. See later in this chapter for how to format the Excel file. Note that since GLD Version 5 (from 2008), the formatting has changed!
- Import a commercial loads file or excel file using the loads import button (see below)

If necessary, hit the “zero box” at the top of each column to reset all values to “0”.

In the last row of the peak cooling and peak heating columns, there are two input boxes for the number of hours the system is expected to operate at the peak. The initial value is set at **3.0** hours and shown in bold face. Only the Monthly Data, or second model of the Borehole Design module uses this “Hours at Peak” value in its calculations. The number entered here determines the number of sequential hours the peak load is applied to the system. For example, if the peak cooling load is expected to last from 12 noon until 4pm on July 21st, then the user can enter 4.0 cooling hours at peak. Except for a process load, it is probably rarely the case

where the peak load is applied for more than 4-6 hours. The impact of modifying the hours at peak input on final results can be viewed most clearly in the new Graphing Module in GLD 2010, which is described below and in the calculated peak entering and exiting water temperatures shown in the Monthly Design method (see below).

The Design Day heat transfer model utilizes the 4 or 8 hour time period already fixed in the Design Day Loads section.

After entering the loads data, hit the *Update* button to return to the main loads panel screen. Notice that the program automatically converts the monthly loads into the design day format following the calculations described on pages 65-66.

Hourly Loads

Details pertaining to importing hourly loads can be found later in this chapter in the section entitled "Importing Loads Data Into the Average Block Loads Module."



Graphical View of Loads

New in GLD2010, users can graphically view the loads profile by pushing the graph button:



This button can be found in the Average Block Loads module as can be seen at the top of figure 3.13.

Pump Selection

Although the selection process is identical to selection in the Zone Manager loads module, the results are slightly different. Figure 3.16 shows the result after selecting a pump, and then modifying the *partial load factor* to 0.9 on the dominant load (Heat Gains) side.

Design Day Loads

7.0 Days / Week

☐ Hourly Data

Transfer

Calculate Hours

Monthly Loads

Design Day Loads

Time of Day	Heat Gains (kBtu/Hr)	Heat Losses (kBtu/Hr)
8 a.m. - Noon	0.0	300.0
Noon - 4 p.m.	500.0	200.0
4 p.m. - 8 p.m.	200.0	0.0
8 p.m. - 8 a.m.	0.0	0.0

Annual Equivalent Full-Load Hours: 700 600

Heat Pump Specifications at Design Temperature and Flow Rate

☐ Custom Pump

Pump Name: TT026_F

	Cooling	Heating
Capacity (kBtu/Hr)	500.0	443.7
Power (kW)	38.50	33.70
EER/COP	13.0	3.9
Flow Rate (gpm)	125.0	75.0
Partial Load Factor	1.00	0.68

Select

Details

Clear

Fig. 3.16 Average Block Loads Pump Selection

In this case, an average pump was selected for the zone, and that pump was given a partial load factor of 1.00 for the dominant cooling side. Since the partial load factor, the ratio between the peak loads and the total equipment capacity, varies depending on designer preference, it can have any value of 1.0 or less. Additionally, the partial load factor will remain constant as the continuous update feature modifies the pump values due to changes in the temperature or the flow rate. The partial load factor plays a small role in the heat exchanger length determination calculations.

Details and Clear

The Details and Clear buttons and the Details panel operate in the same way as they do in the Zone Manager Loads module. However, one difference is that no variation of the load flow rates is permitted in the Details panel.

Custom Pump (Customization)

Checking the Custom Pump check box allows an override of all automatic pump selection features. The user can input any data desired, although once again the COP used in the calculations is calculated from the capacity and the power, not taken from the text box list.

Pump Continuous Update Feature

The *Update/Reselect Current Pumps* control is called automatically when changes are made to either the inlet source temperature or the system flow rate from within the Zone Manager, the Average Block Loads module, or the design modules. In this way, the designer does not have to worry about updating the pumps already matched to zones in GLD.



However, the designer must be aware that sometimes this may result in a new pump size assignment due to capacity changes related to variations in temperature or flow. If this is problematic, custom pumps may be used to lock pump values into a zone. However, for proper modeling, any customized pumps must be edited separately by the designer after the design parameters have been established.

The Studio Link System

The *Studio Link* system is a powerful feature in GLD that gives users the ability to link or to unlink the loads modules to or from the design modules. When a loads module is linked to a Borehole, Horizontal or Surface Water Design module, all of the data in that loads module is transferred to the design module. Once the connection is established, the pertinent information is stored within the design module, which makes transfers in from or out to the loads module as necessary. Since the information is now held in the design module, it is possible to add multiple design modules with only a single loads module open. When studio links are established, the information shown in the loads module will correspond to the active design project.

As long as a link is active, design modules retain information about the type of link and the filename of the associated zone '.zon' file. This information is stored in saved project '.gld' files, so that the appropriate loads module can be opened and loaded when a project file is opened.



Making a Link

The most direct method of making a link between a loads and a design module is to open both modules to be linked, activate (click on) the design module, and then press the 'Link' button on the toolbar. Another option is to choose "Link" from the GLD *Loads* menu. If there is only one type of loads module open, a link will be established with that module. If more than one type of loads module is open, GLD will query the user for his or her linking preference.

Alternative systems for linking exist, but they are more indirect. For example, if only one unlinked design module is present, a link may be established from any

open loads module, since GLD automatically recognizes the user's intention. If more than one unlinked design module exists, however, pressing the link button from an active loads module will have no effect.

The link status lights in the corners of the modules indicate when a link has been formed. Colors indicate the type of link. Link status lights are described in more detail below.

Unlinking

To break a link between modules, simply activate (click on) the design module to be disconnected and click the 'Unlink' button on the toolbar. Equivalently, the user can choose "Unlink" from the GLD *Loads* menu. The link will be broken, and all related loads information for the design module will be cleared. However, the information still exists in the loads module, and can be recovered by linking again if necessary.

If only one design module is linked to a particular loads module, unlinking from the loads module is also possible. If more than one linked design window is open, however, clicking the unlink button from a loads module will have no effect, since GLD cannot determine which project should be disconnected.

The link status lights in the corners of the modules indicate when links are broken. Link status lights are described in more detail below.

Studio Link Status Lights

Studio Link status lights are used to indicate when links are made, when data transfer occurs, and when links are broken. They are located in the lower left hand corner of the design modules, and the lower right hand corner of the loads modules.

Connection Established - First Light from Left

The light furthest to the left indicates both whether or not a connection is established and the type of connection. If the light is off, no connection is established. Magenta indicates a link to an Average Block Loads module, while light blue indicates a link to a Zone Manager loads module.

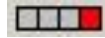
Receiving Data - Second Light from Left

The second light from the left indicates when the module is receiving data from the other module. It is green in color.



Sending Data - Third Light from Left

The third light from the left indicates when the module is sending data to the other module. It is yellow in color.



Broken Connection – Rightmost Light

The light on the right turns red whenever a connection is broken. It turns off again when connections are reestablished.

Importing Loads Data From External Programs

With GLD, users easily can import design day, monthly and/or hourly loads data from both commercial loads programs and Excel files directly into the loads modules.

Importing Loads Into the Average Block Loads Module



The Average Block Loads module can accept the importation of monthly and hourly loads data sets. For monthly and hourly loads sets, users can import loads files from 3rd party building simulation tools or can import .csv files from Excel. For monthly loads, users have the additional option of quickly copying/pasting monthly loads data from Excel and into the Average Block Loads module.

Importing Monthly and Hourly Loads From 3rd Party Programs

To import a file from a commercial loads program, the user can click on the Import button at the top of the Average Block loads module. It looks like this:



Doing so automatically opens the file dialog box in the “Loads Files” folder. There may be several subfolders in which users should store the loads data files that GLD will be using. Users can select one of these folders to display all of the files that can be imported from that particular folder. *Note that in previous versions of GLD, the file dialog box in the "Zones" folder would open up.*

Monthly Loads Data

When the user selects a valid monthly import file, the program automatically transfers the data into the active Average Block Loads module. Note that any previously **existing loads will be overwritten**. If the user is importing a monthly geothermal template (GT) file from the Trane Trace program, an *Import Loads*

window will open, showing the imported data in detail. This window is shown in figure 3.17. The imported monthly and hourly total and peak data also are automatically imported into the monthly loads input boxes as seen in figure 3.13.

The *Import Loads* window displays the imported data, the filename, and the name of the program that generated the file. Total loads and peak demand data are presented on separate screens for cooling and heating. Use the buttons on the bottom of the window to toggle between the two. On the right is the monthly partial load factor, calculated by GLD. The data can be modified directly in the Import Loads window or by hitting the “Modify” button the user can open the file in the Equivalent Hours Calculator, where the data can be edited as well.

The user can transfer the modified data into the Average Block Loads module by pressing the “Transfer” button. When both the Calculator and the Import Loads windows are open, the program first will ask the user from which window, the Calculator or the Import Loads window, he or she wishes to transfer data. The program then prompts the user to decide to which loads, heating or cooling, the data should be transferred.

	Total (kBtu)	Peak (kBtu/hr)	Monthly Load Factor
January	17859.0	27.5	0.87
February	16113.0	27.6	0.87
March	19432.0	140.7	0.19
April	51266.0	372.1	0.19
May	100432.0	447.1	0.30
June	160586.0	552.1	0.40
July	153721.0	536.6	0.39
August	159662.0	543.8	0.39
September	114720.0	477.0	0.33
October	27758.0	298.6	0.12
November	21229.0	258.8	0.11
December	18000.0	120.3	0.20
Total/Max:	860778.0	552.1	
Full-Load Hours:	1559	hr	

Fig. 3.17 Import Loads Window

Since loads calculation programs express results in a number of different ways, GLD edits the input data so that it matches the Design Day formalism used on the main screen of the Average Block Loads Modules. Occasionally, however, the

data from external loads programs do not have the hour-by-hour level of detail that GLD can accept. In these cases, the designer or GLD must make modifications to the imported data to assure that the proper level of detail is retained. In this way the program can be certain to calculate the appropriate heat exchanger size. These modifications are explained at the end of this chapter.

Figure 3.18 is an example of how GLD displays the monthly loads of figure 3.17 in the Design Day formalism found on the main page of the Average Block Loads module.

Time of Day	Heat Gains (MBtu/Hr)	Heat Losses (MBtu/Hr)
8 a.m. - Noon	157.2	201.4
Noon - 4 p.m.	552.1	0.0
4 p.m. - 8 p.m.	157.2	0.0
8 p.m. - 8 a.m.	157.2	0.0

Annual Equivalent Full-Load Hours: 1559 (Heat Gains), 54 (Heat Losses)

Fig. 3.18 Results of Importing

Hourly Loads Data

When the user selects a valid hourly loads import file, the program automatically transfers the data into the active Average Block Loads module. At present time, GLD can accept hourly data files from the IES <VE> and Trane Trace software products as well as from CSV files. If a user wishes to see GLD integrated with other 3rd party simulation tools, please contact GLD support. The hourly loads import and analysis is available as an upgrade from the Standard/Professional version of GLD.

As mentioned previously, when an hourly data file is imported into the Average Block Loads module, the Hourly Data checkbox will be checked as can be seen in figure 3.19 below, indicating that the loads data in the Average Block Loads module is powered by an hourly data set.

Design Day Loads

7.0 Days / Week

☒ Hourly Data

Fig. 3.19 Hourly Data Check Box

Because the hourly data set is so extensive, it is not possible to review the data set hour-by-hour from within GLD. However, it is possible to view the hourly data

organized into a monthly data format by hitting the *Monthly Loads* button on the Average Block Loads module after importing the hourly data. This can be seen in figure 3.15. Note that when viewing the hourly data in the Monthly Data framework, the *Update* button is deactivated, indicating that the hourly data can not be modified from within the GLD framework. If the designer wishes to modify the hourly loads data set, the designer must do so from within his or her energy simulation program.

Importing Monthly and Hourly Loads From Excel and Spreadsheets

Monthly Loads Data

There are three ways to import monthly loads data from Excel or another spreadsheet into the Average Block Loads Module. All three methods require the loads data to be in the following format. Each row of data is for one month of the year with the first populated row representing January loads and the last populated row representing December loads.

Cooling Total (kBtu)	Cooling Peak (kBtu/hr)	Heating Total (kBtu)	Heating Peak (kBtu/hr)
55287	335	382470	1060
46953	345	150525	1105
106020	831	98665	745
194889	1008	37332	325
323767	1066	11014	115
424979	1252	291	22
567918	1325	0	0
516207	1260	0	0
381425	1245	61574	87
204515	938	98623	225
69766	377	144339	200
52249	347	206000	897

The first way to import the loads data is the copy/paste method. *Select ONLY the 12 x 4 block of loads data and copy it (Ctrl-C)*. In the Average Block Loads module, click the *Monthly Data* button and figure 3.13 will appear. Hit the Excel icon as shown in figure 3.13 and the data will be copied automatically into the Average Block Loads module.

The second way to import the loads data is to save the Excel file as a .csv file into the Loads Files/Monthly Data Files folder. To import this .csv file the user can click on the Import button at the top of the Average Block loads module. It looks like this:



Navigate to the .csv file of interest and import it into GLD.

The third way to import monthly data from an Excel file is by using the ‘Import Loads’ command found in the Design Studio *Loads menu*. Select ‘Import Loads’ and an Import Loads window similar to that in Fig 3.17 will appear. GLD expects the Excel data to be in the above order and format.

To import the Excel data, simply highlight the four columns in the Excel spreadsheet and copy them onto the clipboard (Ctrl-C). **Note: highlight only the numeric data. DO NOT highlight the column and row descriptions.** Then in the Import Loads window click on the *Excel* icon. The data will be imported. The data can be modified directly in the Import Loads window or by hitting the “Modify” button the user can open the file in the Equivalent Hours Calculator, where the data can be edited as well.

The user can transfer the modified data into the Average Block Loads module by pressing the “Transfer” button. When both the Calculator and the Import Loads windows are open, the program first will ask the user from which window, the Calculator or the Import Loads window, he or she wishes to transfer data. The program then prompts the user to decide to which loads, heating or cooling, the data should be transferred.

TIP

Note that it is possible to import a single column of data. Following the column order listed above, put the single column of data in the correct position. Fill the remaining columns with zeros and then copy all four columns to the clipboard.

Hourly Loads Data

There is only one way of importing hourly loads data from Excel or another spreadsheet program and into GLD Premier 2010. Loads data must be in the following format:

Chillers load (kBtu/h)	Boilers load (kBtu/h)
0	0

0.002	0
0.09	0
0.188	0
0.987	0
2.38	0
6.586	0
193.683	0

In this format, one hour of cooling/heating data is in each row. ***The three blank rows after the top title row are critical and must be included to ensure import fidelity.***

Save the file as a .csv file into the Loads Files/Hourly Data Files. The .csv file can then be imported using the import button:



Importing Loads Into the Zone Manager Loads Module

Importing loads into the Zone Manager is a simpler proposition because the Zone Manager uses only Design Day and annual energy loads data for its calculations at present time. Monthly and hourly data sets are not used by the Zone Manager for design work.

Users can import commercial loads programs data by clicking on the *Import* button found in the Zone Manager loads modules and import Excel files data by using the 'Import Loads' command from the Design Studio *Loads* menu.

Importing Design Day Loads From 3rd Party Programs

Users can import Design Day and annual energy loads from the Trane Trace program (for example) one zone at time if desired. The user first can select a zone of interest (see description of Zone Manager above) and then can hit the import button:



This automatically opens the file dialog box in the "Loads Files" folder, and displays several subfolders from which files that can be imported. When the user selects a valid import file, the program automatically transfers the data into the current open zone of the Zone Manager Loads module. Note that any previously

existing loads will be overwritten. At the same time the data is transferred into the Zone Manager Loads module, an *Import Loads* window is opened, showing the imported data in detail. This window is shown in figure 3.17, and its corresponding loads entry is shown in figure 3.18.

The Import Loads window in figure 3.17 displays the imported data, the filename, and the name of the program that generated the file. Total loads and peak demand data are presented on separate screens for cooling and heating. Use the buttons on the bottom of the window to toggle between the two. On the right is the monthly partial load factor, calculated by GLD. The data can be modified directly in the Import Loads window or by hitting the “Modify” button the user can open the file in the Equivalent Hours Calculator, where the data can be edited as well.

The user can transfer the modified data into the Zone Manager Loads module by pressing the “Transfer” button. When both the Calculator and the Import Loads windows are open, the program first will ask the user from which window, the Calculator or the Import Loads window, he or she wishes to transfer data. The program then prompts the user to decide to which loads, heating or cooling, the data should be transferred.

Importing Design Day Loads From Excel and Spreadsheets

From Excel Spreadsheets

There are two ways to import Design Day and annual energy loads data from Excel or another spreadsheet into the Zone Manager Loads module. Both methods require the loads data to be in the following format. Each row of data is for one month of the year with the first populated row representing January loads and the last populated row representing December loads.

Cooling Total (kBtu)	Cooling Peak (kBtu/hr)	Heating Total (kBtu)	Heating Peak (kBtu/hr)
55287	335	382470	1060
46953	345	150525	1105
106020	831	98665	745
194889	1008	37332	325
323767	1066	11014	115
424979	1252	291	22
567918	1325	0	0
516207	1260	0	0
381425	1245	61574	87
204515	938	98623	225
69766	377	144339	200
52249	347	206000	897

The first way to import the loads data is the copy/paste method. Find the ‘Import Loads’ command in the Design Studio *Loads* menu. Select ‘Import Loads’ and an Import Loads window similar to that in Fig 3.17 will appear. GLD expects the Excel data to be in the above order and format.

To import the Excel data, simply highlight the four columns in the Excel spreadsheet and copy them onto the clipboard (Ctrl-C). **Note: highlight only the numeric data. DO NOT highlight the column and row descriptions, if any.** Then in the Import Loads window click on the *Excel* icon. The data will be imported. The data can be modified directly in the Import Loads window or by hitting the “Modify” button the user can open the file in the Equivalent Hours Calculator, where the data can be edited as well.

The user can transfer the modified data into the Average Block Loads module by pressing the “Transfer” button. When both the Calculator and the Import Loads windows are open, the program first will ask the user from which window, the Calculator or the Import Loads window, he or she wishes to transfer data. The program then prompts the user to decide to which loads, heating or cooling, the data should be transferred.

The second way to import the loads data is to save the Excel file as a .csv file into the Loads Files/Monthly Data Files folder. To import this .csv file the user can choose the zone of interest and then click on the Import button at the top of the Zone Manager loads module. It looks like this:



Navigate to the .csv file of interest and import it into GLD.

When Imported Data is Not Detailed Enough: How the Program Modifies External Loads Files

In the case of a loads program that generates only total monthly loads and peak monthly demand, nothing is known about the daily hour-by-hour transfer of heat to or from the installation. This information is important in the Borehole and Horizontal Design modules because the hourly data ultimately determines the contributions to the daily and monthly pulses of heat to the ground (GLD performs calculations based on daily, monthly and annual heat pulses). In this type of situation, GLD will use the peak demand and total monthly loads to determine a monthly partial load factor (PLF_m) for the peak design month, where

$$\text{PLF}_m = (\text{actual run time per month}) / (\text{run time if at full load per month}).$$

Once the program calculates the PLF_m, it automatically determines the relationship between off-peak period loads and peak period loads to assure that

the monthly partial load factor matches that of the imported data. The program assumes that the peak demand occurs during the top four-hour period, multiplied by the number of days in the month. If the total heat gains or losses provided for the peak month still exceed this value, the remainder of the total monthly loads are evenly split between the other time periods in the day, making up the remaining 20 hours. If not, the demands of all other periods are set to 0. The peak and its time block will be used for the daily pulse. The monthly pulse utilizes the data in the off-peak periods to recalculate the PLFm. A sample PLFm calculation is presented below.

Assume the monthly calculation gives a total monthly load in January of 10000 KBtu (kWh), and the corresponding peak demand from noon to four p.m. is 30 KBtu/hr (kW). In this case, the monthly partial load factor is:

$$\text{PLFm} = 10000 \text{ KBtu} / (30 \text{ KBtu/hr} * 24\text{hr} * 31 \text{ days}) = 0.448$$

If this value is to be transferred correctly into the Design Day Loads boxes in the loads modules, the 0.448 must remain the same. Noon to four p.m. represents four hours out of twenty-four in a day. Loads not included in that four-hour period must be included in the other twenty hours of the day. The following equation is used to determine the relationship between off-peak loads and peak loads so that the PLFm is maintained (Note that this automatic calculation also assumes that the installation is running 7 days per week, and changes the 'Days per Week' value to reflect this. If other occupation times are desired, the values will need to be changed manually to reflect proper distribution over the course of a month):

PLFm =

$$\begin{aligned} & (\text{Days per Week} / 7 \text{ days}) \times \\ & ((4 \text{ hr} \times [\text{Peak Demand 8-12am}] \\ & + 4 \text{ hr} \times [\text{Peak Demand 12-4}] \\ & + 4 \text{ hr} \times [\text{Peak Demand 4-8}] \\ & + 12 \text{ hr} \times [\text{Peak Demand 8pm-8am}]) / (24 \text{ hr} \times [\text{Top Peak Demand}])) \end{aligned}$$

0.448 =

$$\begin{aligned} & (7 \text{ days per Week} / 7\text{days}) \times \\ & ((4 \text{ hr} \times 30 \text{ KBtu/hr}) \\ & + (4 \text{ hr} \times Y) \\ & + (4 \text{ hr} \times Y) \\ & + (12 \text{ hr} \times Y) / (24 \text{ hr} \times 30 \text{ KBtu/hr})) \end{aligned}$$

or, solving for Y:

$$Y = (((30 \text{ KBtu/hr} \times 24 \text{ hr}) \times 0.448) - (30 \text{ KBtu/hr} \times 4\text{hr})) / 20 \text{ hr}$$

$$Y = (322.56 \text{ KBtu} - 120 \text{ KBtu}) / 20 \text{ hr}$$

$$Y = 10.128 \text{ KBtu/hr}$$

To preserve the partial load factor when transferring into the Design Day Loads, 30 KBtu/hr has to be transferred to the noon to four p.m. block as expected. The 10.128 KBtu/hr needs to be transferred into each of the other three blocks, which represent the other 20 hours of the day.

GLD performs the monthly partial load and the full-load hours calculations automatically when it imports a file containing only monthly and peak loads data. However, if the designer knows more specific details about the installation in question, he or she may want to place those loads more precisely in the actual in-use periods of the day, and consider also the daily occupation of the installation (i.e. not in use on weekends, etc.). However, as long as the peak demand and partial monthly load factor remain the same, the calculated length will also remain the same, no matter what the representation, since the daily and monthly pulses remain unchanged.

Review of Loads Entry in GLD

The loads input methodology in GLD is not as complicated as it first may appear to be. This system has been chosen for two main reasons: First, the advanced mathematical model the program employs allows the loads to be broken into hourly pulses throughout the day of peak demand (the 'Design Day'), which should provide a better overall accuracy in the calculations. Second, GLD uses full-load equivalent hours to reduce the total amount of data entry.

GLD also accepts monthly total and peak loads as well as hourly loads in the average block loads module. These data are only necessary for monthly and hourly inlet temperature calculations in the borehole design module. The mathematical model the program employs for these calculations requires the more detailed monthly and hourly loads data.

Design Day Loads

The 'Design Day' heat gains and losses are simply the average hourly peak demands of the installation over the different periods of the day. Although the program could include all 24 hours of the day separately, it instead uses three 4-hour periods and one 12-hour period to simplify input. These average hourly loads can be entered directly into the corresponding entry box. The soil resistance models employed by the program actually use this data to determine the daily and monthly transfer of energy into the soil. This is because the model assumes that

there are different resistances associated with the annual, monthly, and daily pulses of heat being transferred.

If an installation is not being used at night, for example, the demand for the 12 hour period might be set to 0.

Annual Equivalent Full-Load Hours

Because complete loads entry could be extensive, especially in applications with more than a few zones, GLD limits the necessary data by compacting all of the monthly loads into a single number, the "Annual Equivalent Full-Load Hours". This number effectively represents all of the monthly total loads data (KBtu or kWh), in terms of the peak demand value (KBtu/hr or kW). The advantage is that a single value is used instead of twelve (one for each month of the year).

The full-load hours calculation procedure is straightforward. Simply sum the monthly total loads for all of the months (Kbtu or kWh), and divide by the peak demand (KBtu/hr or kW). The resulting number, the *annual equivalent full-load hours*, then has the units of hours. To put it another way, think of the *annual equivalent full-load hours* as the total number of hours the system would be running in a year if it ran at full capacity the whole time.

To help with this calculation, the program offers the "Equivalent Hours Calculator" as one of the standard tools included in the Geothermal Design Studio. If the designer knows the monthly total loads and peak demand, he or she can simply input them into the boxes provided in the calculator. Pressing 'Calculate' then determines the hours, according to the summation and division described above. When the user presses the 'Transfer' button in any loads module **when the calculator is showing**, the values will be transferred directly into the loads module, as previously described.

Surface Water Design Loads

The Surface Water Design Module does not require the loads input detail of the other design modules. Since there is no long-term build-up of heat in the water, the only values that are actually required are the peak demand of the installation. All other values may be set to zero, or included simply for reference.

CHAPTER 4

The Borehole Design Module

This chapter describes the features and operation of the Borehole Design module. This module is used in the design of vertical borehole systems. It is one of the three design modules included with the Standard/Professional Edition of GLD and one of the four design modules included with the Premier Edition.

Overview

A design is only as good as the quality of the data that goes into it. This is certainly the case with the GLD Borehole Design module. Although GLD utilizes the best theoretical models available today, the most accurate results will naturally result from the most accurate input parameters. Because the calculations conducted here involve the combination of a large number of input parameters, **care must be taken to assure that proper values are verified before use.** Assuming that reasonable values are provided to the software, the software will provide reasonable results.

General Features

To aid in the data entry process, the Borehole Design module in GLD consists of a set of panels, grouped by subject, through which the designer can enter and edit the input variables efficiently. For example, parameters related to the soil are

listed on the *Soil* panel, while piping choices are listed on the *U-tube* panel. The idea is that everything related to a project is presented simultaneously and is easily accessible at any time during the design process. In the expanded user interface mode (available as an upgrade), which can be expanded by double clicking on any of the tabs, the most commonly modified parameters as well as calculation results are always visible, as seen below in figure 4.1.

The screenshot displays the 'Borehole Design Project - 2010-10-01 Bridge Condos by Trow - 20 years with grid file' window. It features several panels and tabs for user interaction.

Lengths Panel:

	COOLING	HEATING
Total Length (ft):	13115.0	13115.0
Borehole Length (ft):	262.3	262.3

Temperatures Panel:

	COOLING	HEATING
Peak Unit Inlet (°F):	83.4	44.6
Peak Unit Outlet (°F):	94.2	37.1

Calculations Panel:

Buttons: Calculate, Monthly, Prediction Time: 10.0 years

Design Method:

☐ Fixed Temperature
☒ Fixed Length

Inlet Temperatures: 90.0 °F, 40.0 °F
 Borehole Length: 262 ft

Grid Layout:

☐ Use External File
 Borehole Number: 50
 Rows Across: 10
 Rows Down: 5
 Separation: 15.1 ft

Cooling Tower/Boiler:

Load Balance: 0 %

Results Panel (Fluid, Soil, U-Tube, Pattern, Extra kW, Information):

	COOLING	HEATING
Total Length (ft):	13115.0	13115.0
Borehole Number:	50	50
Borehole Length (ft):	262.3	262.3
Ground Temperature Change (°F):	N/A	N/A
Peak Unit Inlet (°F):	83.4	44.6
Peak Unit Outlet (°F):	94.2	37.1
Total Unit Capacity (kBtu/Hr):	755.9	810.7
Peak Load (kBtu/Hr):	755.9	810.7
Peak Demand (kW):	111.0	120.0
Average Heat Pump EER/COP:	15.3	4.1
System EER/COP:	13.0	3.5
Avg. Annual Power (kWh):	3.13E+4	2.68E+4
System Flow Rate (gpm):	157.5	168.9

Optional Cooling Tower/Boiler:

Condenser Capacity (kBtu/hr): 0.0
 Cooling Tower Flow Rate (gpm): 0.0
 Cooling Range (°F): 16.0
 Annual Operating Hours (hr/yr): 0
 Boiler Capacity (kBtu/hr): 0.0

Sliders for Cooling Tower, Boiler, and Load Balance are shown at 0 %.

Fig. 4.1 Optional Expanded User Interface

The Borehole Design module includes several additional features:

- Metric and English unit conversion
- Printed reports of all input and calculated data
- Convenient buttons to bring up tables and calculators
- A 'Calculate' button used to refresh the calculations
- A 'Monthly Data' button used to calculate monthly inlet temperatures
- An 'Hourly Data' button used to calculate hourly inlet temperatures
- For monthly and hourly simulations, average annual energy consumption estimates
- A 'Graphing' button used to graph inlet temperature data within the Design Studio
- Boiler and cooling tower hybrids



Opening Projects

There are two ways to open Borehole Design projects. One is by using the ‘New Borehole’ command from the Design Studio *File* menu or toolbar, and the other is by opening an existing Borehole Design project (*.gld) file. Files cannot be opened if other modules with the same name are already open. As many files can be opened as the system’s memory permits.



New Projects

New projects may be opened at any time from the Design Studio by choosing ‘New Borehole’ from either the Design Studio *File* menu or the toolbar. New projects open with standard parameter values that must be edited for new projects. The module opens directly into the Information panel, through which the designer enters information about the new project.

In new projects, no loads files (*.zon) are loaded. The user must create a new loads file or open an existing loads file into one of the loads modules. Links may be established using the Studio Link system described in Chapter 3.



Existing Projects

Existing projects may be opened at any time from the Design Studio by choosing ‘Open’ from the Design Studio *File* menu or toolbar. The file automatically opens into a new Borehole Design Project module.

If a loads file (*.zon) is associated with the loaded project, the loads file automatically will be loaded into the appropriate loads module and opened along with the project file. However, if the associated loads file cannot be found, the user will be notified and the automatic file loading will not occur.

Saving Projects

Projects may be saved at any time using ‘Save’ or ‘Save As’ from the Design Studio *File* menu or by clicking the save button on the toolbar. When the user closes the program or module, the program automatically asks the user if he or she wants to save the project and associated loads files.

Typical Operation

Although each user will have his or her own unique style, the typical operation of the Borehole Design module would include the following steps:



1. Enter Loads and select pump in either the Average Block Loads module or the Zone Manager module
2. Form a link between the loads module and the design module
3. Modify step-by-step the input parameters listed in each panel
4. Perform initial calculation
5. Modify various parameters and recalculate to determine the effects of the modifications
6. Add an optional boiler/cooling tower
7. Run a monthly or hourly energy simulation
8. Establish an optimal system
9. Save and/or print the project and associated zone file

Entering Data into the Tabbed Panels

GLD's innovative tabbed panel system provides for easy organization of and direct access to the relatively large number of design parameters associated with a particular project. This section describes the *Information*, *Extra kW*, *Pattern*, *U-Tube*, *Soil*, *Fluid*, and *Results* panels (See Chapter 3 for a discussion of Loads entry).

Information

The contents of the *Information* panel are shown in figure 4.2. All of the descriptive information related to the project is stored in this panel. This primarily includes the names of the project and designer and the dates. Reference data concerning the client also can be included on this page, so that all relevant project information is in one convenient location.

In addition to generalized project information, specialized comments can be included in the 'Comments' section of the *Information* panel. This area allows the designer to make any notes particular to the specific project that may not necessarily fit under any of the other topics provided.



All of the data in the information panel is optional, but completing the page is recommended for the sake of organization. Reports utilize the project information as a way of distinguishing one project from another.

Except for the dates, the information panel input boxes contain only text, and any desired format may be used when filling out the form.

Note that to reduce repetitive data entry, designer and company information can be entered in the *Settings* dropdown menu at the top of the design studio. This information then automatically populates part of the *Information* panel.

The screenshot shows a software window titled "Borehole Design Project - verticalsampleforManual". It has a tabbed interface with tabs for "Results", "Fluid", "Soil", "U-Tube", "Pattern", "Extra kW", and "Information". The "Information" tab is selected. Below the tabs is a section titled "Project Information" containing the following fields:

- Project Name:** Borehole Design Sample Project
- Designer Name:** D. B. Engineer
- Date:** 10/5/2007 (dropdown menu)
- Project Start Date:** 10/5/2007 (dropdown menu)
- Client Name:** ABC Corp
- Address Line 1:** 1333 Any St.
- Address Line 2:** Suite #2200
- City:** Anytown
- State:** CA
- Zip:** 91711
- Phone:** (555)555-1212
- Fax:** (555)555-1213
- Email:** engineer@abc.com
- Comments:** This is a sample borehole project file for Ground Loop Design.

Fig. 4.2 Information Panel Contents

Extra kW

Additional energy that is utilized by the system can be entered in the *Extra kW* panel. The entry boxes are shown in figure 4.3.

This panel is included for entire system average efficiency calculations. The top entry box, "Circulation Pumps", is for the energy required by the system circulation pumps. The middle entry box, "Optional Cooling Tower", is for the energy required by a cooling tower (if used). The lower entry box, "Additional Power Requirements", is for all other elements (besides the heat pump units) in the system that may require energy input. For example, heat recovery units require additional energy that can be recorded in this box so that it can be used in the overall calculation of the System EER/COP.

In the 'Circulation Pumps' section, the 'Required Input Power' is calculated from the 'Pump Power' required by the pump(s) for the system in question and the average 'Pump Motor Efficiency'. It is not possible to edit the 'Required Input Power' values directly. However, if the pump motor efficiency is set to 100%, the 'Pump Power' and 'Required Input Power' will be the same.

Borehole Design Project - verticalsampleforManual

Results | Fluid | Soil | U-Tube | Pattern | **Extra kW** | Information

Circulation Pumps

Required Input Power: kW

Pump Power: hP

Pump Motor Efficiency: %

Optional Cooling Tower

	Pump	Fan
Required Input Power:	<input type="text" value="0.2"/> kW	<input type="text" value="1.3"/> kW
Power:	<input type="text" value="0.2"/> hP	<input type="text" value="1.5"/> hP
Motor Efficiency:	<input type="text" value="85"/> %	<input type="text" value="85"/> %

Additional Power Requirements

Additional Power: kW

Fig. 4.3 *Extra kW* Panel Contents

If an optional cooling tower is used for hybrid applications, the demands of the pump and fan may be included on this panel. The tower pump is selected based on the water flow and the total head (these also determine the horsepower). The required fan horsepower and motor efficiency may also be entered to include the demand of the fan. *Generally, cooling tower inputs are left at zero initially, and then modified once the program suggests the cooling tower size and flow rate.*

The ‘Additional Power’ may be included as necessary.



Note: To make a kilowatt entry in the ‘Pump Power’ box, switch to metric units, enter the kilowatt value, and then return to English units.

Pump Power Calculator

If the pump efficiency, system flow rate and head loss are known or have been calculated in the CFD module, the *Pump Power Calculator* can be used to determine the pump power. The new CFD module makes it straightforward to calculate system head loss and thereby estimate the pump power with a degree of accuracy. An image of the pump power calculator is shown in figure 4.4.

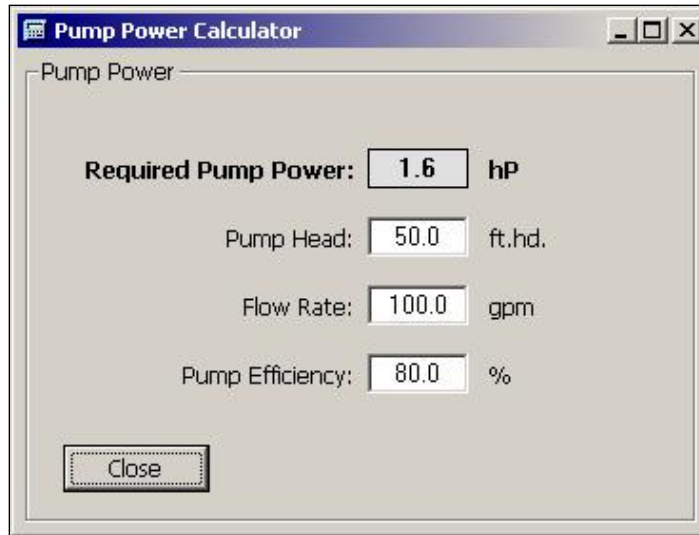


Fig. 4.4 Pump Power Calculator

Pattern

Information pertaining to the ground field arrangement is in the *Pattern* panel. This includes the vertical boreholes pattern, the borehole separation, the optional selection of external grid files, export to AutoCAD, the number of boreholes per parallel loop, and the fixed borehole length design option. The input screen is shown in figure 4.5. Pattern, borehole separation and external grid file data also are visible and adjustable in the expanded user interface, as seen in figure 4.6.



New for GLD Premier 2010 on the *Pattern* tab is a built-in and updated g function generator. GLD Premier 2010 generates a g function on demand for any possible vertical borefield design, just one of many unique features that GLD provides to designers.

Vertical Grid Arrangement

The standard Borehole Design module is configured to accept equally spaced borehole patterns based on an x, y coordinate system. For rectangular systems, users can enter the pattern directly into the *rows across* and *rows down* boxes. For non-rectangular systems, see external grid files, below.

Separation between Vertical Bores

This value is the center-to-center distance between adjacent bores. For optimal use of space, the current calculations allow only one spacing distance between vertical bores in either direction.

Borehole Design Project #1

Results | Fluid | Soil | U-Tube | **Pattern** | Extra kW | Information

Vertical Grid Arrangement

Borehole Number: **100**

Rows Across:

Rows Down:

Borehole Separation: ft

☐ Use External File

Filename: No File

Boreholes per Parallel Circuit

Bores Per Circuit:

1 2 3

Fixed Length Mode

☐ On/Off Borehole Length: ft

Fig. 4.5 *Pattern* Panel Contents

Grid Layout

☐ Use External File

Borehole Number: **30**

Rows Across:

Rows Down:

Separation: ft

Fig. 4.6 *Pattern* Data in Expanded User Interface



External Grid Files

For non-rectangular and potentially non-equally spaced systems, users have the option of creating and then importing an external grid data file that contains the x, y coordinates for each borehole in the system. Using an external grid file offers absolute control over loopfield design. To export a design to AutoCAD, an external grid file is required.

User-designated grid files must follow this format:

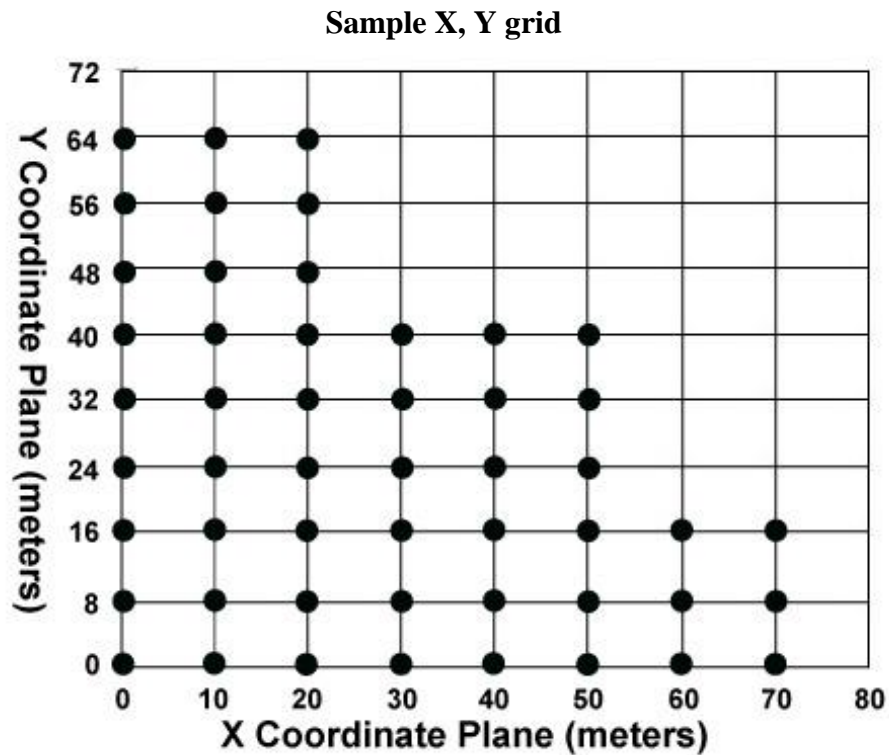
Sample *GridData.txt* created in text editor

```
Metric
0,0
0,10
10,0
10,10
0,20
20,0
20,20
10,20
20,10
30,0
30,10
30,20
30,30
20,30
10,30
0,30
```

This sample grid file was manually created in a text editor. As can be seen, the first line indicates whether a Metric (meters) or English (feet) units coordinate plane is specified. The next line indicates the x, y coordinates of the first borehole. The line after that indicated the x, y coordinates of the second borehole, etc. This particular sample has 16 boreholes spaced at 10 meter intervals.

Below can be found a recommended three step protocol for creating grid files:

- 1) First, lay out your proposed borehole field on a x,y grid similar to the ones shown. Doing so will decrease the chance of errors when creating a grid file.



Note that this grid is in meters and that each black dot represents a borehole.

- 2) Open up a text editor and in the first line of a new text file, specify whether the grid file is in metric or English units. Since this grid is in meters, the first line of the grid file should say “Metric”.
- 3) In the text file begin listing the boreholes in the grid file as can be seen below. Note that the first borehole is at the origin (0,0).

Grid file for sample X,Y grid

```
Metric
0,0
0,8
0,16
0,24
0,32
0,40
0,48
0,56
0,64
10,0
10,8
etc...
```

After a grid file is created, it must be saved in the following folder:

Gaia Geothermal\GLD2010\GridFiles

so that the GLD program can find the grid file.

After creating and saving a grid file, load it into the heat exchanger module by first checking the *Use External File* checkbox and then hitting the 'Select' button to choose the desired file as seen in figure 4.7.

Vertical Grid Arrangement

Borehole Number: 16

Rows Across: 10

Rows Down: 10

Borehole Separation: 100.0 ft

☒ Use External File **Select** Clear Show

Filename: GridData.txt Create

Fig. 4.7 Selecting an External Grid File

The file name will appear directly below the checkbox. In this case, the filename is GridData.txt. When a valid grid file has been selected, the number of boreholes in the file is displayed and the standard *rows across*, *rows down* and *borehole separation* text boxes become inactive. If GLD is unable to read the grid file (for example, if the formatting is incorrect), then GLD will revert to the standard rectangular grid (rows across and rows down). Click the *Show* button to review the selected grid file at any time.

When a gridfile is selected, it is indicated in the expanded user interface, as seen in figure 4.8.

Grid Layout

☒ Use External File

Borehole Number: 16

Filename: GridData.txt

Fig. 4.8 Use of External File Indicated in Expanded User Interface



G Function Calculator

The G Function Calculator is a built-in function that generates a g function on demand for any loopfield configuration. These g functions enable GLD to perform advanced monthly and hourly simulations.



Export to AutoCAD

New in GLD 2010 is the ability to export a loopfield design to AutoCAD. This is an optional upgrade for Professional/Standard Edition users.

To export to AutoCAD, the designer must be using an external grid file. If the designer is designing a non-rectangular loopfield then all the user has to do is the following:

- Confirm that the grid file is selected
- Navigate to the *File* dropdown menu at the top of GLD
- Choose the Export File option and export the AutoCAD file of choice
- GLD will export an .scr file into the GLD2010/CAD Files folder that can be read into AutoCAD.

If the designer is designing a loopfield using the rows across and rows down input boxes (and not using a grid file), the user must first input the loopfield design into the input boxes and then must hit the 'Create' button that can be seen in figure 4.7. Doing so will create a grid file. The user can then export a .scr file following the above bullet point instructions.

Note that if the user is using the standard input boxes for loopfield design, he or she should be sure to deselect "use external file" after completing the export to AutoCAD process.

Boreholes per Parallel Loop



The 'number of boreholes per parallel loop' refers to the piping arrangement within the borehole pattern. The calculation will give slightly different bore lengths depending on whether one, two, or more boreholes are included in one parallel circuit. Remember that pumping costs will increase as the pipe lengths per parallel circuit become longer. Pressure drop impacts can be fully explored in the new CFD module.

Fixed Length Mode

By selecting fixed length mode, the designer can specify the loop field length (*number of boreholes x length per borehole*) and have GLD

calculate the entering water temperatures. When in fixed length mode, it is important to note that both cooling and heating lengths are identical, unlike in the fixed temperature mode (where designers specify temperatures and calculate lengths). The expanded user interface displays the design mode (fixed temperature or fixed length) as well as adjustable parameters associated with each mode. For the fixed temperature mode, the entering water temperatures can be adjusted, while for the fixed length mode, the borehole length can be modified. This can be seen in figure 4.9.

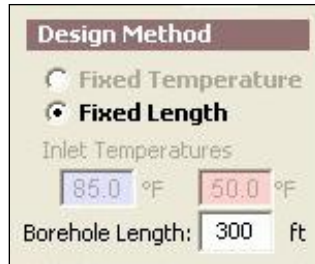


Fig. 4.9 Design Method in Expanded User Interface

U-Tube

The *U-Tube* panel contains information related to the pipe and bore. The main purpose of the panel is to obtain a value for the borehole thermal resistance (BTR). Calculated according to the method of Paul and Remund (Paul, 1996), the thermal resistance calculation takes into account the pipe parameters and positioning, the borehole diameter, and the grout thermal conductivity. If desired, an experimentally determined value of the BTR also may be entered into the textbox, which then overrides all calculations. In GLD Premier 2010, the updated Thermal Conductivity module can calculate BTR from empirical data. The panel contents are shown in figure 4.10.



The screenshot shows a software window titled "Borehole Design Project - verticalsampleform manual". It has a tabbed interface with tabs for "Results", "Fluid", "Soil", "U-Tube", "Pattern", "Extra kW", and "Information". The "U-Tube" tab is active.

Calculated Borehole Equivalent Thermal Resistance

Borehole Thermal Resistance: **0.231** h*ft*°F/Btu

Pipe Parameters

Pipe Resistance: **0.104** h*ft*°F/Btu Check Pipe Tables

Pipe Size: **1 in. (25 mm)**

Outer Diameter: **1.32** in

Inner Diameter: **1.08** in

Pipe Type: **SDR11**

Flow Type: **Turbulent**

U-Tube Configuration

☒ Single

☐ Double

Radial Pipe Placement

☐ Close Together

☐ Average

☒ Along Outer Wall

Borehole Diameter

Borehole Diameter: **5.50** in

Backfill (Grout) Information

Thermal Conductivity: **1.08** Btu/(h*ft*°F)

Fig. 4.10 U-Tube Panel Contents

Pipe Parameters

The pipe parameters are entered in the 'Pipe Parameters' section. They include the pipe resistance and pipe outside diameter, followed by the configuration and placement of the pipe in the bore.

GLD calculates the convective resistance using the Dittus-Boelter correlation for turbulent flow in a circular tube (Incropera and DeWitt, 1990). The calculations use average values of the Reynolds number to represent the different types of flow, with values of $Re = 1600$, 3150 , or 10000 for laminar, transition, and turbulent, respectively. The calculations also use average values of viscosity and the Prandtl number for water, taken at a temperature of 70°F .

Using the standard expression for resistance of a hollow cylinder (Incropera and DeWitt, 1990), the program can calculate an approximate

value for the pipe resistance. It assumes HDPE pipe with a conductivity of 0.225 Btu/h''ft°F.

The pipe resistance varies with the pipe style and flow. The user can select the size and type of pipe from the appropriate selection boxes. If another pipe diameter is required, it can be entered directly into the text boxes as needed.

Note: By pressing the 'Check Pipe Tables' button, the 'Pipe Properties' tables will open.

If the user wants to enter an experimentally determined pipe resistance, or requires more precise calculations, he or she can enter these values directly into the 'Pipe Resistance' text box, overriding all pipe resistance calculations.

The user also selects the U-tube configuration and radial pipe placement for the designed installation. A single U-tube refers to two pipes placed in the bore, while a double U-tube refers to four pipes placed in the bore. The radial pipe placement can be one of the following:

- **Close together** - 1/8" average distance between the pipes
- **Average** - pipes are centered at a point halfway between the wall and the center of the bore
- **Along outer wall** - pipes are against the outer wall

Illustrations are included to clarify the choices.



Note: The 'Double' U-tube configuration at this stage is added more for reference than for practical use. Currently, the values GLD uses are based on experimental data and a new theoretical model accounting for a lower pipe and convective resistance, and a larger displacement of the grout. Designers should be aware of this fact, and remember that a 'single' U-tube is the standard option.

Borehole Diameter and Backfill (Grout) Information



The user can enter the borehole diameter and the grout thermal conductivity directly into their respective text boxes. **If cuttings are used for the backfill, the average soil conductivity should be entered here.**

Soil

Input parameters relating to the soil are located in the *Soil* panel, as shown in figure 4.11. These include the average ground temperature, the soil thermal properties and the modeling time period.

Borehole Design Project #2

Results | Fluid | **Soil** | U-Tube | Pattern | Extra kW | Information

Undisturbed Ground Temperature

Ground Temperature: °F

Soil Thermal Properties

☐ *View Layer Calculator

Thermal Conductivity: Btu/(h*ft*°F)

Thermal Diffusivity: ft²/day

Modeling Time Period

Prediction Time: years

Fig. 4.11 Soil Panel Contents



The undisturbed ground temperature refers to the temperature of the soil below the surface layer, where there is no longer a seasonal swing. This value may be determined from regional data or by recording the actual stabilized temperature of water circulated through pipe in a test bore.



The soil thermal properties are a little harder to define, and care must be taken to provide accurate values, especially for the thermal conductivity. The thermal diffusivity relates to the density of the soil and its moisture content. Typical values of thermal conductivity and diffusivity for sand, clay, and different types of rocks can be found in the 'Soil Properties' tables. However, it is recommended that soil tests are performed to obtain these values. *The thermal conductivity in particular has a large effect on the calculated bore length, and should be determined with care through in-situ tests or comparison with other projects installed in the local vicinity. GLD does not encourage the use of ex-situ data.*



Drilling Log Conductivity Calculator

The layer calculator is a new feature in GLD2010 that enables designers to use a drilling log to produce a quick weighted-average calculation for thermal conductivity and diffusivity. *While some non-published empirical studies indicate that weighted average calculations offer conductivity results that are different from empirically-derived thermal conductivity results,* some designers prefer to estimate conductivity from a drill log. ***For commercial projects, thermal conductivity tests are generally recommended.*** Figure 4.13a is a screenshot of the layer calculator.

The screenshot shows the 'Soil Thermal Properties' dialog box with the '*View Layer Calculator' checkbox checked. At the top, it displays 'Thermal Conductivity: 1.22 Btu/(h*ft*°F)' and 'Thermal Diffusivity: 0.98 ft^2/day'. Below this is a list of layers: 'Layer # 1', 'Layer # 2' (highlighted in blue), and 'Layer # 3'. To the right of the list, there are input fields for 'Layer #2': 'Name: Layer #2', 'Layer Thickness: 75.0 ft', 'Soil Type: Other' (with a dropdown arrow), 'Conductivity: 0.95 Btu/(h*ft*°F)', and 'Diffusivity: 0.86 ft^2/day'.

Fig. 4.13a Drilling Log Conductivity Calculator

Diffusivity Calculator

For the designer's assistance, GLD includes a *Diffusivity Calculator* that can be used to determine the actual diffusivity if all the soil parameters are known. It requires knowledge of the thermal conductivity, the dry specific heat and density, and the moisture level in the soil. An image of the diffusivity calculator is shown in figure 4.13b.

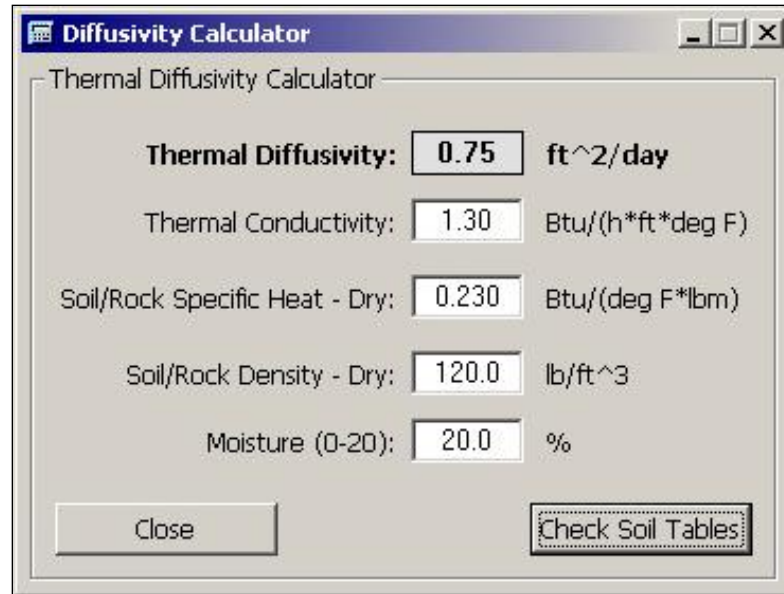


Fig. 4.13b Diffusivity Calculator

Modeling time Period

TIP

In GLD, ten years is used as a standard length of time for the ground temperature to stabilize, although longer time periods may be entered if desired. When excessive ground water movement is known to occur, one year is sometimes used as the modeling time period. In this case, it is assumed that the ground temperature stabilizes in a single year due to the neutralizing effects of the ground water movement.

The modeling time/prediction time period can also be viewed and modified in the expanded user interface, as seen in figure 4.14.

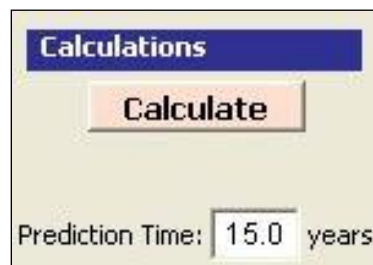


Fig. 4.14 Design Prediction Time in Expanded User Interface

Fluid

The circulating fluid parameters may be entered in the *Fluid* panel. A sample input screen is shown in figure 4.15. Note **that automatic fluid data entry mode** is available as an option in this version of GLD.

Design Heat Pump Inlet Fluid Temperatures

The heat pump inlet fluid temperatures are included in the *Fluid* panel. The designer can input the desired inlet source temperatures for both heating and cooling here. *When changes are made to these values, the heat pumps in all zones are updated automatically.* **Since the new calculated equipment capacities can lead to changes in selected equipment, the designer must be aware of the changes.** Customized pump values must be manually adjusted.



The inlet fluid temperatures also can be viewed and modified in the expanded user interface, as seen in figure 4.9 above. **Note: inlet temperatures can only be modified in fixed temperature mode.**

Design System Flow Rate



The system flow rate per installed ton is included on the *Fluid* panel. *This is the system flow rate per ton of peak load, not installed capacity* (This is because it is assumed that all units will not be running at full load simultaneously, even in the peak load condition). The impact of installed capacity flow rates and purging flow rates on system performance can be explored in the new CFD module.

Borehole Design Project - verticalsampleformanual

Results | **Fluid** | Soil | U-Tube | Pattern | Extra kW | Information

Design Heat Pump Inlet Fluid Temperatures

Cooling: °F Heating: °F

Design System Flow Rate

Flow Rate: gpm/ton

Solution Properties

☒ Automatic Entry Mode

Fluid Type: %

Specific Heat (Cp): Btu/(°F*lbm)

Density (rho): lb/ft³

Fig. 4.15 *Fluid Panel Contents*

TIP

Optimized systems generally operate in the range from 2.5 to 4.0 gpm/ton, while the ideal system flow rate is somewhere around 3.0 gpm/ton. Again, if the flow rate is changed, the selected heat pumps are updated in the loads modules.

Solution Properties

Solution properties are also included in the *Fluid* panel. These include the *specific heat* and *density* of the circulating fluid. Also, a reference label is included so that the designer knows the percentage of antifreeze and antifreeze type; **however, this reference label is not currently linked to the other input parameters.**

The specific heat and density values of the antifreeze are used for the calculation of the heat pump outlet temperature, which in turn is used for the bore length calculation.



Additionally, the viscosity of the solution may affect the flow type in the pipe, which was selected on the *U-Tube* panel. The designer must be aware of any changes made. The new CFD module models the impact of viscosity changes on system performance.

In **automatic entry mode**, the user first selects the fluid type and then selects the desired freezing temperature. GLD automatically displays the specific heat and density for the fluid selection. When the automatic entry mode checkbox is marked, the program is in automatic entry mode.

In **manual entry mode**, the user manually selects and inputs the specific heat and density for the target solution as seen in figure 4.16. When the automatic entry mode checkbox is unmarked, the program is in manual entry mode.

Solution Properties

☐ Automatic Entry Mode

Fluid Type: %

Specific Heat (Cp): Btu/(°F*lbm)

Density (rho): lb/ft³

Fig. 4.16 Manual Entry Mode for Solution Properties

*Note: Since solution properties vary considerably and non-linearly with type and percentage of additive, GLD does not include detailed automatic antifreeze information for all conditions. Generalized tables of data may be found in the 'Fluid Properties' tables. **It is recommended that the designer manually enter the desired values in the input text boxes.***

Results

All results for both the heating and the cooling calculations can be viewed at any time on the *Results* panel. After all data has been entered or any changes have been made, the user can choose from the calculate interim or final results using the 'Calculate' button. New in GLD Premier 2010, the designer can choose one of three calculation methodologies - Design Day, Monthly or Hourly - from the dropdown menu prior to hitting the 'Calculate' button. The 'Calculate' button is also available in the expanded user interface, as seen in figure 4.1. The three methodologies are described briefly below and in more detail in Chapter 1.

Design Day: This calculation methodology works with loads from both the Zone Manager Loads module and the Average Block Loads module. The calculation performed is based on the cylindrical source heat transfer theory as described in Chapter 1. The Design Day model works in both fixed temperature and fixed length mode (described above).

Monthly: This calculation methodology works with loads from the Average Block Loads module if the designer has imported monthly loads data. The calculation performed is based on an advanced heat transfer theory. Incorporating a dimensionless "g function", this methodology calculates the evolution of the borehole wall and fluid temperatures over time. The monthly model works only in fixed length mode.

Hourly: This calculation methodology works with loads from the Average Block Loads module if the designer has imported hours loads data. The calculation performed is based on an advanced computational heat transfer theory. Incorporating a dimensionless "g function", this methodology calculates the evolution of the borehole wall and fluid temperatures over time. The hourly model works only in fixed length mode.

The Calculate panel is divided into two sections. On the top is the reporting section, which presents the calculation results. The lower "*Optional Cooling Tower/Boiler*" section is included to assist in the sizing of a cooling tower and/or boiler. This is a convenient tool for hybrid-type designs, which may be desirable when the cooling length exceeds that of heating or when the heating length exceeds that of cooling. The cooling tower and boiler options are discussed in more detail below.

Design Day Results

Results Subsections: Fixed Temperature Mode

In fixed temperature mode (where the designer selects target EWTs and the program calculates borehole depths), the reporting section is separated into five subsections. A sample screen for fixed temperature design day results can be seen in figure 4.17.

The two lists on the *Results* panel are for heating and cooling. Although all of the numbers shown are valid and respond to changes, the side with the longer required length is printed in bold type so that it stands out. The non-dominant side heat exchanger results are grayed out because in the actual installation, heating and cooling installed lengths are identical. In the results in fixed temperature mode, non-dominant side results are "short looped" (compared to the actual installation), are not applicable to the actual installation and therefore lose

relevance. For example, in figure 4.17 the heating side borehole length is listed at 105.8 ft. In reality, when installed, the boreholes will be 253.3 ft deep (as per the cooling side requirements). The presented results therefore "short loop" the heating side by $253.3 - 105.8$ or 147.5 ft. Consequently, the other results on the heating side relate to a 105.8 ft deep borehole rather than a 253.3 ft borehole, which in reality will not be the case. The results are grayed out as a reminder to the designer.



The first subsection deals with the bores, including the total length, the borehole number, and the borehole length for one bore. A common way to adjust the borehole length to a desired value is to change the borehole number or pattern on the *Pattern* panel.

The second subsection presents the predicted long-term ground temperature change with respect to the average ground temperature of the installation. Remember that in the fixed temperature mode, only the temperature change listed in bold has any relevance. Note that both temperature changes will be equal if the cooling and heating loads to the ground are equal (as in the case where a hybrid system is utilized to balance the loads out).

	COOLING	HEATING
Total Length (ft):	15197.0	6348.2
Borehole Number:	60	60
Borehole Length (ft):	253.3	105.8
Ground Temperature Change (°F):	+2.3	+5.5
Unit Inlet (°F):	90.0	35.0
Unit Outlet (°F):	100.1	29.2
Total Unit Capacity (kBtu/Hr):	763.2	590.7
Peak Load (kBtu/Hr):	730.5	547.4
Peak Demand (kW):	57.3	47.5
Heat Pump EER/COP:	13.4	3.6
System EER/COP:	12.7	3.4
System Flow Rate (gpm):	182.6	136.9

Optional Cooling Tower/Boiler

Condenser Capacity (kBtu/hr):	0.0	Cooling Tower	0 %
Cooling Tower Flow Rate (gpm):	0.0		
Cooling Range (°F):	10.4	Boiler	0 %
Annual Operating Hours (hr/yr):	0		
Boiler Capacity (kBtu/hr):	0.0	Load Balance	

Fig. 4.17 *Results* Panel Contents: Fixed Temperature Design Day

The third subsection of the report lists the heat pump inlet and outlet temperatures of the circulating fluid.

The fourth subsection lists the total unit capacity, the peak loads and demand of all the equipment, and the calculated heat pump and system efficiencies. The peak load is the maximum and is determined from whichever time period across all the zones has the highest load. The peak demand includes all pumps and external energy requirements, including those listed in the *Extra kW* panel.

Finally, the system flow rate is listed in its own subsection. The system flow rate is calculated from the peak load divided by 12,000 Btu/ton, and then multiplied by the flow rate (in gpm/ton) chosen on the *Fluid* panel. It represents the flow rate from the installation out to the buried pipe system.

Calculation results for lengths and temperatures are always available in the expanded user interface as well, as seen in figure 4.18. Calculations can be performed at any time in the expanded user interface, as well.

Lengths			Temperatures		
	COOLING	HEATING		COOLING	HEATING
Total Length (ft):	15197.0	6348.2	Unit Inlet (°F):	90.0	35.0
Borehole Length (ft):	253.3	105.8	Unit Outlet (°F):	100.1	29.2

Fig. 4.18 Calculation Results in Expanded User Interface

Results Subsections: Fixed Length Mode

In fixed length mode (where the designer selects the target borehole depth and the program calculates EWTs and pump performance), the reporting section also is separated into five subsections. A sample screen for fixed temperature design day results can be seen in figure 4.19.

The two lists on the *Results* panel are for heating and cooling. In fixed length mode, **both heating and cooling results are printed in bold type** so that they stand out. This is different from fixed temperature model, above. The reason is that in fixed length mode, performance calculations for both the dominant and non-dominant sides are based on the actual designer-selected length of the heat exchanger. Results for both sides are therefore relevant.

The first subsection deals with the bores, including the total length, the borehole number, and the borehole length for one bore. A common way to adjust the borehole length to a desired value is to change the borehole number or pattern on the *Pattern* panel.

TIP

The second subsection presents the predicted long-term ground temperature change with respect to the average ground temperature of the installation. Both

temperatures are more or less identical, are in bold, and have relevance. The temperatures are identical because they represent the average temperature change for the entire loopfield over the design lifetime. Since there can be only one overall average and the borehole lengths for heating and cooling are defined and equal, the ground temperature change prediction is reported in bold for both heating and cooling.

	COOLING	HEATING
Total Length (ft):	15180.0	15180.0
Borehole Number:	60	60
Borehole Length (ft):	253.0	253.0
Ground Temperature Change (°F):	+2.2	+2.2
Unit Inlet (°F):	90.0	49.6
Unit Outlet (°F):	100.1	43.6
Total Unit Capacity (kBtu/Hr):	763.2	674.3
Peak Load (kBtu/Hr):	730.5	547.4
Peak Demand (kW):	57.3	43.7
Heat Pump EER/COP:	13.4	3.9
System EER/COP:	12.7	3.7
System Flow Rate (gpm):	182.6	136.9
Optional Cooling Tower/Boiler		
Condenser Capacity (kBtu/hr):	0.0	Cooling Tower
Cooling Tower Flow Rate (gpm):	0.0	0 %
Cooling Range (°F):	10.4	Boiler
Annual Operating Hours (hr/yr):	0	0 %
Boiler Capacity (kBtu/hr):	0.0	Load Balance

Fig. 4.19 Results Panel Contents: Fixed Length Design Day

The third subsection of the report lists the heat pump inlet and outlet temperatures of the circulating fluid.

The fourth subsection lists the total unit capacity, the peak loads and demand of all the equipment, and the calculated heat pump and system efficiencies. The peak load is the maximum and is determined from whichever time period across all the zones has the highest load. The peak demand includes all pumps and external energy requirements, including those listed in the *Extra kW* panel.

Finally, the system flow rate is listed in its own subsection. The system flow rate is calculated from the peak load divided by 12,000 Btu/ton, and then multiplied by the flow rate (in gpm/ton) chosen on the *Fluid* panel. It represents the flow rate from the installation out to the buried pipe system.

Calculation results for lengths and temperatures are always available in the expanded user interface, as seen in figure 4.18, above. Calculations can be performed at any time in the expanded user interface, as well.

Optional Cooling Tower/Fluid Cooler and Boiler Section

Cooling towers and boilers can be added to designs via the sliders that are located on both the *Results* panel (figure 4.17) and in the expanded interface, as seen in figure 4.20. The Cooling Towers and Boilers can be run independently or together in order to balance required lengths or temperatures.

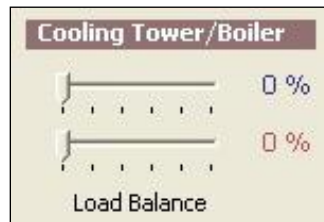


Fig. 4.20 Cooling Tower and Boiler Controls in Optional Expanded User Interface

Cooling Towers

Although typically not recommended because of increased running and maintenance costs, the user may elect to add a cooling tower to a cooling-dominated geothermal system to reduce the total boring lengths, and therefore the total initial installation costs. To facilitate this design choice, GLD offers the cooling tower, or “hybrid”, option.

In any case where the calculated boring lengths for cooling are longer than those for heating, the difference in the lengths can be eliminated through the use of a cooling tower tied in parallel to the geothermal ground loop. This requires that either the cooling tower capacity is chosen such that both the peak load and the annual load to the ground are balanced or if a full balance is unnecessary, a capacity is chosen that allows for downsizing the loop to an acceptable length.

To aid in the sizing process, a *Load Balance* control is provided in the Optional Cooling Tower section of the Calculate panel. Although clicking the slider control can initiate a valid calculation or recalculation, the slider control generally is employed after initial calculations have been conducted. The *Load Balance* is a slider-based control that represents a

percentage of the total cooling load, both instantaneous (peak) and annual. For example, a *100% Load Balance* would be equivalent to saying that the entire cooling load of the system would be handled by the cooling tower. Conversely, a *0% Load Balance* would mean that no cooling tower is employed.

In a typical design it is difficult to predict exactly how much load balance or what size of cooling tower is necessary to match the cooling and heating lengths. However, using the *Load Balance* slider control, the designer can optimize the system to the lengths desired by directly controlling the amount of cooling load to be handled by the cooling tower.

In the case where the designer desires the shortest length possible, the design requires a perfect balance of the heating and cooling loads to the ground. The length from this perfect balance would be the minimum length required to adequately cover the heating load requirement. **To accomplish this, the *Load Balance* slider needs to be adjusted to the percentage value where the calculated cooling and heating bore lengths are approximately equivalent.**

TIP

*Note: As expected, the Long Term Ground Temperature Change for both heating and cooling should be **identical** in a perfectly balanced system.*

TIP

In other cases, the designer just may seek a reduction in the total required system length rather than a perfect balance. Using the slider control, the designer can select the desired length and then note the required cooling tower condenser capacity (as calculated by the program).

As the designer adds cooling tower capacity to a design, the peak load of the geothermal system will decrease such that the total capacity of the geothermal system and the hybrid system equals the peak load as defined in the loads module.

NEW

Once the required cooling tower capacity is determined, the designer can further modify the various cooling tower parameters to match them to his or her own system. The standard equation used in the program (Francis, 1997) is:

Condenser Capacity (Btu/hr)

$$= \text{Flow Rate (gpm)} \times 500 \times \text{Temperature Difference (°F)},$$

where the 500 is used for pure water, and represents a factor derived from:

$$\text{Specific Heat of Water (1.0)} \times 60 \text{ min/hr} \times \text{Density (8.33 lb/gal)} = 500$$

(Note that GLD actually calculates this factor from the input fluid properties on the *Fluids* panel, although pure water is a logical choice for most cooling-dominated applications.)

For example, if the cooling range is increased above the initial minimum value, the capacity of the condenser also is increased, reducing the total number of operating hours. However, in the same case, decreasing the required flow rate is another option which would keep the condenser capacity and operating hours unchanged. The only limitations are the required temperature difference and the minimum condenser capacity needed to meet the chosen design length. With GLD, users have the flexibility to choose the parameters that fit best in their designs.

Boilers

In GLD, boilers are similar to cooling towers except that they are added in order to reduce the overall heating load on the system. In this case, the user may actually reduce the peak and annual heating loads by the flat percentage defined by the slider value. The required boiler capacity and the modified peak loads applied to the loop field are shown on the panel, but no other inclusion electrical or fuel costs for the boiler are included in the calculation report. The expected heat pump power is also reduced by the same percentage, in order to estimate a real system.

Monthly Data Results

Results Subsections: Fixed Length Mode

For Monthly Data calculations, fixed length mode is the only option available. This is because the loopfield geometry must be fully defined (including borehole depth), before the calculations can be performed. As a result, when a designer selects the Monthly Data calculation methodology, the program switches to and locks in to fixed length mode.

GLD calculate monthly inlet temperatures for a user defined modeling time period (see 4.12). Depending on the modeling time period and the computer resources, this calculation may take several seconds to complete.

For the Monthly Data results, the reporting section is separated into five subsections and one Graphing Module. Results that are unique to the Monthly Data results (compared to the Design Day results) are displayed in purple. A sample screen for Monthly Data results can be seen in figure 4.21.

The two lists on the *Results* panel are for heating and cooling. In fixed length mode, both heating and cooling results are printed in bold type so that they stand out. The reason is that in fixed length mode, performance calculations for both

the dominant and non-dominant sides are based on the actual designer-selected length of the heat exchanger. Results for both sides are therefore relevant.

The first subsection deals with the bores, including the total length, the borehole number, and the borehole length for one bore. A common way to adjust the borehole length to a desired value is to change the borehole number or pattern on the *Pattern* panel.

TIP

The second subsection presents the predicted long-term ground temperature change with respect to the average ground temperature of the installation. When calculating Monthly Data results, the average ground temperature change always will be reported as N/A. This is because the updated theory in GLD Premier 2010 used for these calculations is not directly amenable to such soil temperature calculations. Designers that need to estimate the soil temperature change can do so using the Design Day calculation described above.

	COOLING	HEATING
Total Length (ft):	15180.0	15180.0
Borehole Number:	60	60
Borehole Length (ft):	253.0	253.0
Ground Temperature Change (°F):	N/A	N/A
Peak Unit Inlet (°F):	81.6	43.9
Peak Unit Outlet (°F):	90.0	37.8
Total Unit Capacity (kBtu/Hr):	755.9	810.7
Peak Load (kBtu/Hr):	755.9	810.7
Peak Demand (kW):	46.0	60.5
Seasonal Heat Pump EER/COP:	18.7	4.3
Heat Pump EER/COP:	16.4	3.9
Avg. Annual Power (kWh):	2.56E+4	2.51E+4
System Flow Rate (gpm):	189.0	202.7
Optional Cooling Tower/Boiler		
Condenser Capacity (kBtu/hr):	0.0	Cooling Tower
Cooling Tower Flow Rate (gpm):	0.0	0 %
Cooling Range (°F):	8.7	Boiler
Annual Operating Hours (hr/yr):	0	0 %
Boiler Capacity (kBtu/hr):	0.0	Load Balance

Fig. 4.21 Results Panel Contents: Fixed Length Monthly Data Results

Lengths			Temperatures		
	COOLING	HEATING		COOLING	HEATING
Total Length (ft):	15180.0	15180.0	Peak Unit Inlet (°F):	81.6	43.9
Borehole Length (ft):	253.0	253.0	Peak Unit Outlet (°F):	90.0	37.8

Fig. 4.22 Monthly Data Results in Optional Expanded User Interface



The third subsection of the report lists the heat pump inlet and outlet temperatures of the circulating fluid. Note that in GLD Premier 2010, **these purple numbers are absolute peak temperatures and not average peak temperatures.** In previous versions of GLD, reported results for inlet and outlet temperatures were average peak temperatures. *By presenting absolute peak temperatures, it makes it easier for the designer to compare results from the Design Day cylindrical source theory and the Monthly Data line source theory.* Some designers enjoy making this comparison because seeing similar results from two divergent heat transfer theories/calculation methodologies enhances design confidence. On the flip side, results discrepancies between the two theories can enable designers to hone in on potential design issues. Note that absolute peak temperatures are sensitive to the hours at peak input which can be seen in figure 3.13.



The fourth subsection lists the total unit capacity, the peak loads and demand of all the equipment, the calculated seasonal heat pump efficiency, the calculated design day efficiency and the calculated average annual power consumption. The peak load is the maximum and is determined from whichever time period across all the zones has the highest load. The peak demand includes all pumps and external energy requirements, including those listed in the *Extra kW* panel.



In GLD Premier 2010, the calculated seasonal cooling and heating heat pump efficiency values over the design lifetime are quite useful for lifecycle cost and CO₂ emissions analyses in the Finance Module. The design day efficiency is the predicated heat pump performance on the cooling and heating design day. The average annual power consumption is calculated by summing up the monthly heat pump power draw over the design lifetime and dividing by the number of years. Including the system loads, the dynamic fluid temperatures and the dynamic heat pump performance, there is no more accurate way to estimate the power consumption of a geothermal design. Designers may find it interesting to see the impact of borehole spacing changes on average annual power consumption.

Finally, the system flow rate is listed in its own subsection. The system flow rate is calculated from the peak load divided by 12,000 Btu/ton, and then multiplied by the flow rate (in gpm/ton) chosen on the *Fluid* panel. It represents the flow rate from the installation out to the buried pipe system.



The Graphing Module

Users also can view a range of monthly data results using the new Graphing Module. In the expanded user interface, a graphing icon button will appear after hitting the 'Calculate' button as seen in figure 4.23 (Remember: the user can access the expanded user interface by double clicking on any of the tabs).

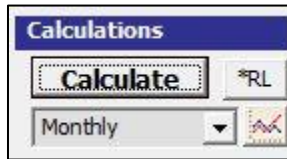


Figure 4.23 Monthly Data Graphing Button



Users can also access the graphs from the *Tools* dropdown menu, selecting the Graph Data option and then importing the data of set of interest into the Graphing Module. Figure 4.24 is a screenshot of the Graphing Module.

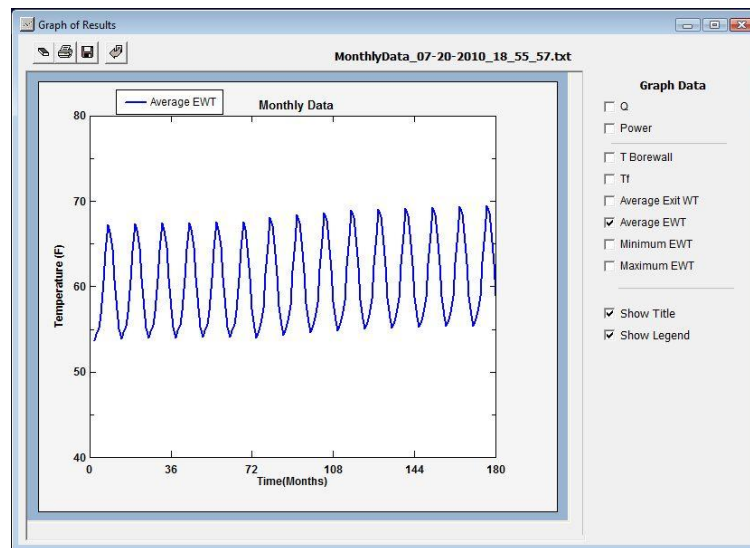


Figure 4.24 The Graphing Module



The new Graphing Module in GLD Premier 2010 is much more powerful than the graphing functions in previous versions of GLD. In the new module, users can left click the mouse and drag a box around an area of interest in the graph. Users can then release the mouse button *to zoom in on the area of interest*. This process can be repeated multiple times. Users can right click the mouse at any time to zoom out to the original view.

Within the graph, the designer can choose which data to view, save, and/or print. Options include: Q (heat transferred to or from the ground), heat pump power

consumption, borehole temperature, T_f (the average temperature of fluid in the borehole calculated as the average of exiting and entering temperatures), average exiting water temperature, average entering water temperature, and minimum (a variation of the average calculated from the application of short term heating loads) and maximum (a variation from the average calculated from the application of short term peak cooling loads) entering water temperatures. The designer can also add a title and legend to the graph. More than one graph can be open at the same time, enabling designers to quickly compare different designs. Saved graphs can be found in the GLD/Graph Images folder.

A dated monthly data text file containing the temperature data is generated and stored in the Monthly Data folder each time the 'Calculate' button is pressed. If necessary, data from this file can be imported into Excel.

Optional Cooling Tower/Fluid Cooler and Boiler Section

Cooling towers and boilers can be added to designs via the sliders that are located on both the *Results* panel (figure 4.16) and in the expanded interface, as seen in figure 4.25. The Cooling Towers and Boilers can be run independently or together in order to balance required lengths or temperatures.

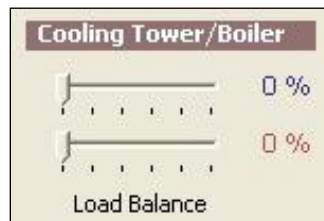


Fig. 4.25 Cooling Tower and Boiler Controls in Optional Expanded User Interface

Cooling Towers

Although typically not recommended because of increased running and maintenance costs, the user may elect to add a cooling tower to a cooling-dominated geothermal system to reduce the total boring lengths, and therefore the total initial installation costs. To facilitate this design choice, *GLD* offers the cooling tower, or “hybrid”, option.

In any case where the calculated boring lengths for cooling are longer than those for heating, the difference in the lengths can be eliminated through the use of a cooling tower tied in parallel to the geothermal ground loop. This requires that either the cooling tower capacity is chosen such that both the peak load and the annual load to the ground are balanced or if a full balance is unnecessary, a capacity is chosen that allows for downsizing the loop to an acceptable length.

To aid in the sizing process, a *Load Balance* control is provided in the Optional Cooling Tower section of the Calculate panel. Although clicking the slider control can initiate a valid calculation or recalculation, the slider control generally is employed after initial calculations have been conducted. The *Load Balance* is a slider-based control that represents a percentage of the total cooling load, both instantaneous (peak) and annual. For example, a *100% Load Balance* would be equivalent to saying that the entire cooling load of the system would be handled by the cooling tower. Conversely, a *0% Load Balance* would mean that no cooling tower is employed.

In a typical design it is difficult to predict exactly how much load balance or what size of cooling tower is necessary to match the cooling and heating lengths. However, using the *Load Balance* slider control, the designer can optimize the system to the lengths desired by directly controlling the amount of cooling load to be handled by the cooling tower.

In the case where the designer desires the shortest length possible, the design requires a perfect balance of the heating and cooling loads to the ground. The length from this perfect balance would be the minimum length required to adequately cover the heating load requirement. **To accomplish this, the *Load Balance* slider needs to be adjusted to the percentage value where the calculated cooling and heating bore lengths are approximately equivalent.**

TIP

*Note: As expected, the Long Term Ground Temperature Change for both heating and cooling should be **identical** in a perfectly balanced system.*

TIP

In other cases, the designer just may seek a reduction in the total required system length rather than a perfect balance. Using the slider control, the designer can select the desired length and then note the required cooling tower condenser capacity (as calculated by the program).



As the designer adds cooling tower capacity to a design, the peak load of the geothermal system will decrease such that the total capacity of the geothermal system and the hybrid system equals the peak load as defined in the loads module.

Once the required cooling tower capacity is determined, the designer can further modify the various cooling tower parameters to match them to his or her own system. The standard equation used in the program (Francis, 1997) is:

Condenser Capacity (Btu/hr)

$$= \text{Flow Rate (gpm)} \times 500 \times \text{Temperature Difference (°F)},$$

where the 500 is used for pure water, and represents a factor derived from:

$$\text{Specific Heat of Water (1.0) x 60 min/hr x Density (8.33 lb/gal)} = 500$$

(Note that GLD actually calculates this factor from the input fluid properties on the *Fluids* panel, although pure water is a logical choice for most cooling-dominated applications.)

For example, if the cooling range is increased above the initial minimum value, the capacity of the condenser also is increased, reducing the total number of operating hours. However, in the same case, decreasing the required flow rate is another option which would keep the condenser capacity and operating hours unchanged. The only limitations are the required temperature difference and the minimum condenser capacity needed to meet the chosen design length. With GLD, users have the flexibility to choose the parameters that fit best in their designs.

Boilers

In GLD, boilers are similar to cooling towers except that they are added in order to reduce the overall heating load on the system. In this case, the user may actually reduce the peak and annual heating loads by the flat percentage defined by the slider value. The required boiler capacity and the modified peak loads applied to the loop field are shown on the panel, but no other inclusion electrical or fuel costs for the boiler are included in the calculation report. The expected heat pump power is also reduced by the same percentage, in order to estimate a real system.

After adjusting either the cooling tower or boiler slider, the designer will notice that the program automatically recalculates the entire design. For larger systems, it may take several seconds to update the results.

For both cooling tower and boiler hybrids it is recommended that a user first uses the fixed temperature Design Day mode to calculate the required lengths based on the hybrid design. Next, the user is recommended to switch to Monthly mode, enter the borehole length calculated in Design Day mode and then add the hybrid system until the desired peak inlet temperatures are reported.

Hourly Data Results

Results Subsections: Fixed Length Mode



For Hourly Data calculations, fixed length mode is the only option available. This is because the loopfield geometry must be fully defined (including borehole depth), before the calculations can be performed. As a result, when a designer

selects the Hourly Data calculation methodology, the program switches to and locks in to fixed length mode. Note that hourly calculations are only available after upgrading to the Premier Edition.

GLD calculate hourly inlet temperatures for a user defined modeling time period (see figure 4.12). *It is highly recommended that a designer changes the modeling time period to one year prior to hitting 'Calculate'.* Extending the modeling time period beyond one or two years results in a geometric increase in required calculations. While a smaller loopfield modeled over a single year could take a few minutes to process, a large loopfield modeled over several years *could take GLD an entire evening to process.* Note that during the calculation process, the Studio Link status lights at the bottom of the module will flash and cycle, indicating that GLD is working.

Because hourly simulations are computationally intensive, it is recommended that the design optimize a design using the Design Day and Monthly Data methodologies described above first. After a designer is comfortable with a design, her or she optionally may run an hourly simulation over a design year to estimate system performance based on the more fine hourly loads data.

Note that after the user starts the hourly calculation, a cancel button will appear that enables the user to end the process if necessary.

For the Hourly Data results, the reporting section is separated into five subsections and one Graphing Module. Results that are unique to the Hourly Data results (compared to the Design Day results) are displayed in green. A sample screen for Hourly Data results can be seen in figure 4.26.

The two lists on the *Results* panel are for heating and cooling. In fixed length mode, both heating and cooling results are printed in bold type so that they stand out. The reason is that in fixed length mode, performance calculations for both the dominant and non-dominant sides are based on the actual designer-selected length of the heat exchanger. Results for both sides are therefore relevant.

The first subsection deals with the bores, including the total length, the borehole number, and the borehole length for one bore. A common way to adjust the borehole length to a desired value is to change the borehole number or pattern on the *Pattern* panel.

TIP

The second subsection presents the predicted long-term ground temperature change with respect to the average ground temperature of the installation. When calculating Hourly Data results, the average ground temperature change always will be reported as N/A. This is because the updated theory in GLD Premier 2010 used for these calculations is not directly amenable to such soil temperature calculations. Designers that need to estimate the soil temperature change can do so using the Design Day calculation described above.

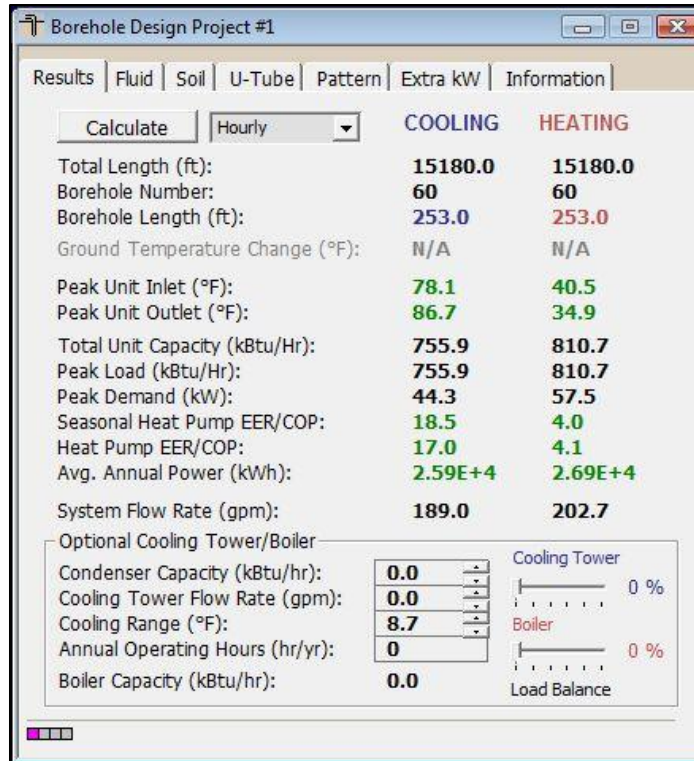


Fig. 4.26 Results Panel Contents: Fixed Length Hourly Data Results

Lengths			Temperatures		
	COOLING	HEATING		COOLING	HEATING
Total Length (ft):	15180.0	15180.0	Peak Unit Inlet (°F):	78.1	40.5
Borehole Length (ft):	253.0	253.0	Peak Unit Outlet (°F):	86.7	34.9

Fig. 4.27 Hourly Data Results in Optional Expanded User Interface



The third subsection of the report lists the heat pump peak inlet and peak outlet temperatures of the circulating fluid. **These green numbers are absolute peak temperatures and not average peak temperatures.** *By presenting absolute peak temperatures, it makes it easier for the designer to compare Hourly Data results with Design Day and Monthly Data Results.* Note that these peak temperatures are not influenced by changes in the hours at peak control which can be seen in figure 3.13. This is because of the inherent detail in hourly loads profiles.

The fourth subsection lists the total unit capacity, the peak loads and demand of all the equipment, the calculated seasonal heat pump efficiency, the calculated design day efficiency and the calculated average annual power consumption. The peak load is the maximum and is determined from whichever time period across all the zones has the highest load. The peak demand includes all pumps and external energy requirements, including those listed in the *Extra kW* panel.



In GLD Premier 2010, the calculated seasonal cooling and heating heat pump efficiency values over the design lifetime are quite useful for lifecycle cost and CO₂ emissions analyses in the Finance Module. The design day efficiency is the predicated heat pump performance on the cooling and heating design day. The average annual power consumption is calculated by summing up the hourly heat pump power draw over the design lifetime and dividing by the number of modeling years. Including the system loads, the dynamic fluid temperatures and the dynamic heat pump performance, there is no more accurate way to estimate the power consumption of a geothermal design. Designers may find it interesting to see the impact of borehole spacing changes on average annual power consumption.



Finally, the system flow rate is listed in its own subsection. The system flow rate is calculated from the peak load divided by 12,000 Btu/ton, and then multiplied by the flow rate (in gpm/ton) chosen on the *Fluid* panel. It represents the flow rate from the installation out to the buried pipe system.



The Graphing Module

Users also can view a range of hourly data results using the new Graphing Module. In the expanded user interface, a graphing icon button will appear after hitting the 'Calculate' button as seen in figure 4.28 (Remember- the user can access the expanded user interface by double clicking on any of the tabs).

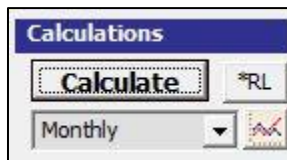


Figure 4.28 Monthly Data Graphing Button



Users can also access the graphs from the *Tools* dropdown menu, selecting the Graph Data option and then importing the data of set of interest into the Graphing Module. Figure 4.29 is a screenshot of the Graphing Module with hourly data.

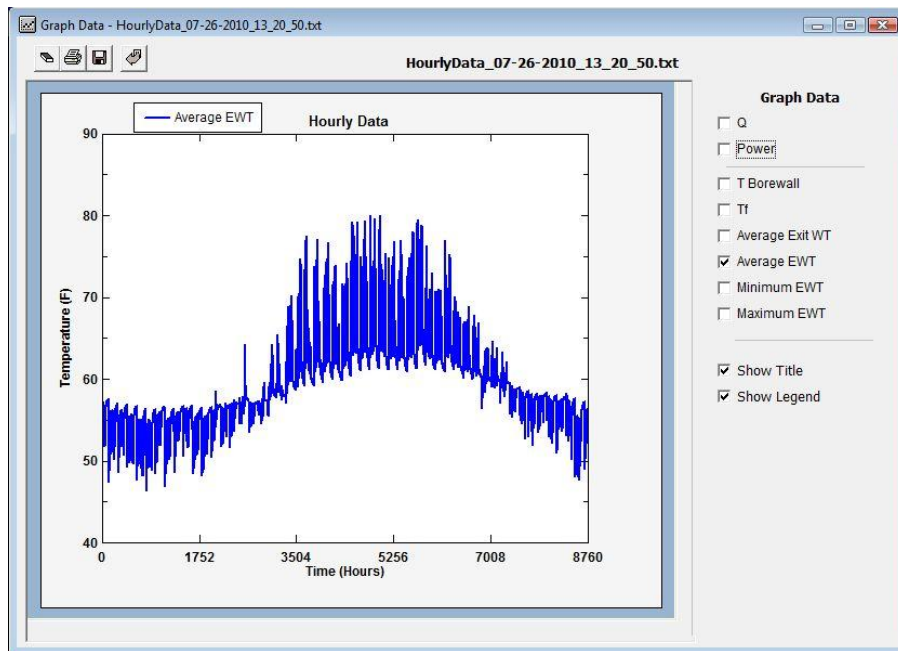


Figure 4.29 The Graphing Module with Hourly Data



The new Graphing Module in GLD Premier 2010 is much more powerful than the graphing functions in previous versions of GLD. In the new module, users can left click the mouse and drag a box around an area of interest in the graph. Users can then release the mouse button *to zoom in on the area of interest*. This process can be repeated multiple times. Users can right click the mouse at any time to zoom out to the original view.

Within the graph, the designer can choose which data to view, save, and/or print. Options include: Q (heat transferred to or from the ground), heat pump power consumption, borehole temperature, *T_f (the average temperature of fluid in the borehole calculated as the average of exiting and entering temperatures), average exiting water temperature, average entering water temperature, and minimum (a variation of the average calculated from the application of short term heating loads) and maximum (a variation from the average calculated from the application of short term peak cooling loads) entering water temperatures*. The designer can also add a title and legend to the graph. More than one graph can be open at the same time, enabling designers to quickly compare different designs. Saved graphs can be found in the GLD/Graph Images folder. Note that since the hour is the shortest modeling time frame and the program outputs results on an hourly time scale, the average, max and min EWT values are identical.

A dated hourly data text file containing the temperature data is generated and stored in the Hourly Data folder each time the 'Calculate' button is pressed. If necessary, data from this file can be imported into Excel.

Optional Cooling Tower/Fluid Cooler and Boiler Section

It is not recommended that a user designs a hybrid system while in Hourly Data mode because each time a user changes the hybrid system size, GLD will recalculate the entire system. This could be very time consuming. It is much faster to optimize a hybrid system in the Monthly Data mode and then run an hourly simulation based on the hybrid design.

More details about hybrid systems can be found above.



The Design Compare Button

The design compare button, also known as the Design Dashboard, enables a user to quickly and simultaneously compare the results from a Design Day, a Monthly and possibly an Hourly simulation. The button only appears after the user has selected the Monthly or Hourly design method. Figure 4.30 is a sample screen shot from a design that shows results from both the Design Day and Monthly simulation. This feature is only available in the Premier Edition.

	COOLING			HEATING		
	Design Day	Monthly	Hourly	Design Day	Monthly	Hourly
Total Length (ft):	15624.0	15624.0	--	15624.0	15624.0	--
Borehole Number:	60	60	--	60	60	--
Borehole Length (ft):	260.4	260.4	--	260.4	260.4	--
Ground Temperature Change (°F):	1.1	--	--	1.1	--	--
Peak Unit Inlet (°F):	82.4	75.5	--	39.8	44.1	--
Peak Unit Outlet (°F):	92.4	83.9	--	33.9	37.9	--
Total Unit Capacity (kBtu/Hr):	941.7	755.9	--	810.7	810.7	--
Peak Load (kBtu/Hr):	755.9	755.9	--	810.7	810.7	--
Peak Demand (kW):	53.4	45.4	--	63.3	60.8	--
Average Heat Pump EER/COP:	14.1	18.1	--	3.8	4.3	--
System EER/COP:	14.1	16.6	--	3.8	3.9	--
Avg. Annual Power (kWh):	--	2.64E+4	--	--	2.54E+4	--
Equip. Flow Rate (gpm):	235.4	189.0	--	202.7	202.7	--
System Flow Rate (qpm):	189.0	189.0	--	202.7	202.7	--

Figure 4.30 The Optional Design Dashboard/Design Compare Window

Printing Reports

Reports of the active project can be printed at any time from the Design Studio using the toolbar print button or from the *File* menu → *Print*.

Two project reports and four monthly/hourly inlet temperature reports are available. In the concise and detailed reports, information printed includes all of the input parameters from the design module, along with the associated results. In the concise and detailed reports, the zone and loads information is not included with the report, and must be printed separately from the *Loads* panel. The filename of the *.zon file associated with the project report is also listed on the reports. The other four inlet temperature reports offer different combinations of input parameters, loads and monthly inlet temperatures that designers can choose among depending on their reporting needs.

More information on reports can be found in Chapter 7.

References

Francis, E., Editor. *Refrigeration and Air Conditioning, 3rd Edition*. Air-Conditioning and Refrigeration Institute. p.186. Prentice Hall, New Jersey, 1997.

Incropera, F. and Dewitt, D. *Introduction to Heat Transfer, 2nd Edition*. p. 456, p. 98. John Wiley and Sons, New York. 1990.

Paul, N. *The Effect of Grout Thermal Conductivity on Vertical Geothermal Heat Exchanger Design and Performance*. M.S. Thesis, South Dakota State University. 1996.

CHAPTER 5

The Horizontal Design Module

This chapter describes the features and operation of the Horizontal Design module. This module is used in the design of near-surface horizontal systems including horizontal trench, bore and pit systems. It is one of the three design modules included in the Professional/Standard Edition of GLD and one of four design modules included in the Premier Edition.

Overview



As with the Borehole and Surface Water Design modules, the calculations made in the Horizontal Design module involve the combination of a large number of input parameters. **Care must be taken to assure that proper values are verified before use.** Assuming that reasonable values are provided to the software, the software will provide a reasonable result.

General Features

The Horizontal Design module in GLD also includes a set of panels, grouped by subject, through which the designer can enter and edit the input variables in a straightforward and efficient manner. For example, parameters related to trench

configuration are listed on the Configuration panel, while piping choices are listed on the *Piping* panel. Everything related to a project is presented simultaneously and easily is accessible throughout the design process. In the expanded user interface mode (available after upgrading to the Premier edition), which can be expanded by double clicking on any of the tabs, the most commonly modified parameters as well as calculation results are always visible, as seen below in figure 5.1.

The screenshot displays the 'Horizontal Design Project - HorizontalSample_usermanual5' window. It features several panels and tabs for configuring a trench layout and calculating results.

Lengths

	COOLING	HEATING
Total Trench Length (ft):	4145.9	8177.4
Single Trench Length (ft):	207.3	408.9

Temperatures

	COOLING	HEATING
Unit Inlet (°F):	85.0	50.0
Unit Outlet (°F):	95.2	44.0

Calculations

Calculate

Prediction Time: 10.0 years

Design Method

☒ **Fixed Temperature**

Inlet Temperatures

85.0 °F 50.0 °F

Configuration

Trench Number: 20

Separation: 10.0 ft

Depth: 24.0 ft

Width: 24.0 in

Cooling Tower/Boiler

0 %

0 %

Load Balance

Results | Fluid | Soil | Piping | Configuration | Extra kW | Information

Trench Layout

Number: 20

Depth: 24.0 ft

Separation: 10.0 ft

Width: 24.0 in

Pipe Configuration in Trench

Five icons showing different pipe arrangements in a trench.

Modeling Time Period

Prediction Time: 10.0 years

Diagram: A schematic showing a trench layout with three pipes. The total number of pipes is 3. The vertical separation [Y] is 24.0 in, and the horizontal separation [X] is 12.0 in.

Fig. 5.1 The Optional Expanded User Interface

The Horizontal Design module includes several additional features:

- Metric and English unit conversion
- Printed reports of all input data and calculated results
- Convenient buttons to bring up tables and calculators
- A 'Calculate' button used to refresh the calculations
- Boiler and cooling tower hybrids

Opening Projects

There are two ways to open Horizontal Design projects. One is by using the 'New Horizontal' command from the Design Studio *File* menu or toolbar, and the other is by opening an existing Horizontal Design project (*.gld) file. Files cannot be opened if other modules with the same name are already open. As many files can be opened as the system's memory permits.



New Projects

New projects may be opened at any time from the Design Studio by choosing 'New Horizontal' from either the Design Studio *File* menu or the toolbar. New projects open with standard parameter values that must be edited for new projects.

In new projects, no loads files (*.zon) are loaded. The user must create a new loads file or open an existing loads file into one of the loads modules. Links may be established using the Studio Link system described in Chapter 3.



Existing Projects

Existing projects may be opened at any time from the Design Studio by choosing 'Open' from the Design Studio *File* menu or toolbar. The file automatically opens into a new Horizontal Design Project module.

If a loads file (*.zon) is associated with the loaded project, the loads file automatically will be loaded into the appropriate loads module and opened along with the project file. However, if the associated loads file cannot be found, the user will be notified and the automatic file loading will not occur.

Saving Projects

Projects may be saved at any time using 'Save' or 'Save As' from the Design Studio *File* menu or by clicking the save button on the toolbar. When the user closes the program or module, the program automatically asks the user if he or she wants to save the project and associated loads files.

Typical Operation

Although each user will have his or her own unique style, the typical operation of the Horizontal Design module would include the following steps:

1. Enter Loads and select pumps in either the Average Block Loads module or the Zone Manager module
2. Form a link between the loads module and the design module
3. Modify step-by-step the input parameters listed in each panel
4. Perform initial calculation
5. Modify various parameters and recalculate to determine the effects of the modifications
6. Add a optional boiler/cooling tower
7. Establish an optimal system
8. Save and/or print the project and associated zone file

Entering Data into the Tabbed Panels

GLD's innovative tabbed panel system provides for easy organization of and direct access to the relatively large number of design parameters associated with a particular project. This section describes the *Information*, *Extra kW*, *Configuration*, *Piping*, *Soil*, *Fluid*, and *Results* panels. The *Information* and *Extra kW* panels are identical to those included in the Borehole Design module described in Chapter 4, so the reader is referred there for detailed information. See Chapter 3 for a discussion of Loads entry.

Configuration

Information pertaining to the trench configuration is in the *Configuration* panel. This includes the trench layout, the pipe configuration in the trenches, and the modeling time. The input screen is shown in figure 5.2. Trench number, separation, depth and width options also are visible and adjustable in the expanded user interface, as seen in figure 5.3.

Trench Layout

This is the section where the user enters all parameters regarding the physical size and placement of the trenches. The number of trenches may be modified at any time using the up-down arrows, and "Separation" refers to the center-to-center distance between adjacent trenches. The program assumes all trenches will be equal in separation, length, depth and width. Note that if the selected piping configuration does not fit into the selected trench size, the program will automatically adjust the size of the trench to accommodate the selection.



In GLD2010, the minimum center-to-center distance between adjacent trenches has been reduced.

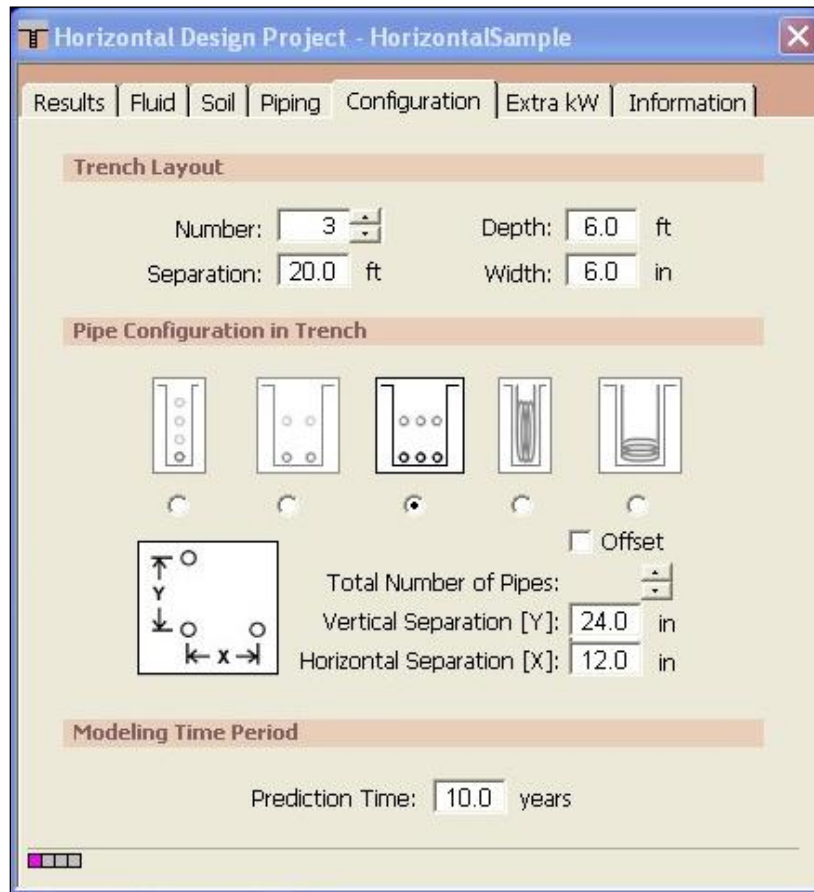


Fig. 5.2 Configuration Panel Contents

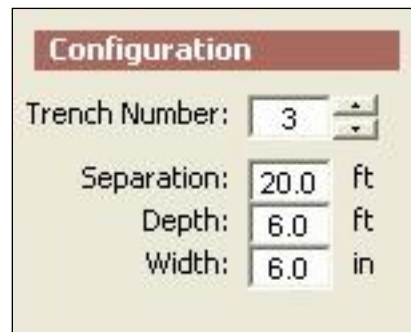


Fig. 5.3 Configuration Controls in Expanded User Interface

Pipe Configuration in Trench

The designer defines the physical arrangement of pipe in the trenches in this section.

STRAIGHT PIPE CONFIGURATIONS

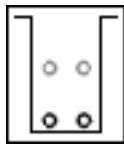
In the case of the three straight-pipe configurations, the user also provides the total number of pipes and the horizontal [X] and vertical [Y] separation of the pipes in the trench. An additional offset, meaning a horizontal shift between adjacent vertical layers, can be included if desired.

Single-Pipe Vertical Alignment



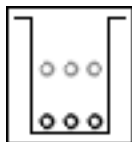
In this arrangement, the user creates a single column of pipes. The number of pipes chosen defines how many layers will be included. Each pipe is separated from its neighbor by the given vertical separation [Y], starting from the bottom of the trench. If the 'Offset' box is checked, each pipe layer will be shifted from the pipe layer below by the given horizontal separation [X].

Two-Pipe Vertical Alignment



In this arrangement, the user creates two-pipe layers. The number of pipes chosen defines how many layers will be included (2, 4, 6, etc.). Each vertical layer is separated from the one above or below by the given vertical separation [Y]. If the 'Offset' box is checked, each pipe layer will be shifted from the pipe layer below by one-half the given horizontal separation [X/2]. *This arrangement can be utilized to model horizontal bores.*

Three-Pipe Vertical Alignment



In this arrangement, the user defines three pipe layers. The number of pipes chosen defines how many layers will be included (3, 6, 9, etc.). Each vertical layer is separated from the one above or below by the given vertical separation [Y]. If the 'Offset' box is checked, every layer will be shifted from the layer below by one-half the given horizontal separation [X/2].

SLINKY PIPE CONFIGURATIONS

In the case of the horizontal and vertical slinky configurations, the user

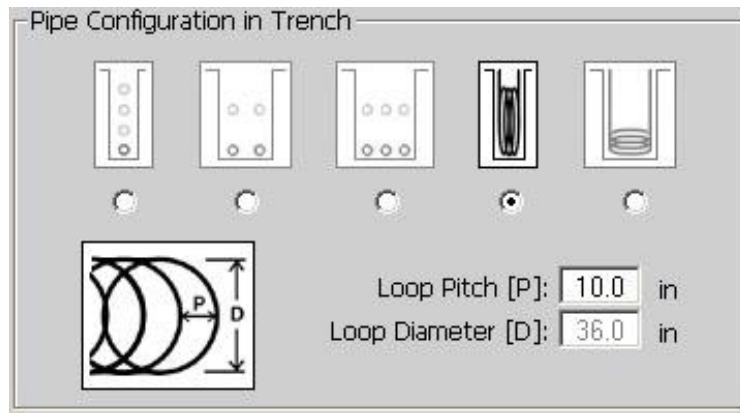


Fig. 5.4 Slinky Variables

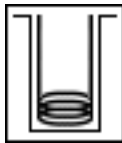
may define the pitch and diameter of the Slinky. Because of the limited model employed, the pitch must be between 10 and 56 inches, and the diameter must be 36 inches. See figure 5.4.

Vertical Slinky



In this arrangement, the slinky is placed vertically within a trench and is resting at the bottom. The trench may be as narrow as the pipe and soil allow.

Horizontal Slinky



In this arrangement, the slinky is placed horizontally at the bottom of the trench. The minimum trench width depends on the slinky diameter.

Modeling time Period

In GLD, ten years is used as a standard length of time for ground temperature stabilization, although longer or shorter time periods may be entered if desired. In the case of horizontal systems, a single year or less is often chosen since the interaction with the atmosphere or sunlight generally reduces the long-term buildup or reduction of soil temperatures. Long-term thermal effects are more commonly associated with vertical bores. The modeling time/prediction time period can also be viewed and modified in the expanded interface as seen in figure 5.5

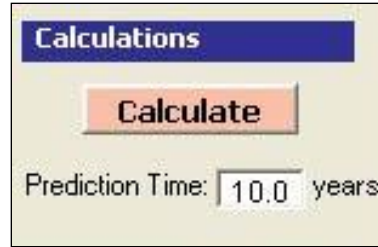


Fig. 5.5 Prediction Time Controls in Optional Expanded User Interface

Piping

The *Piping* panel contains all the information related to the particular pipe chosen for the buried heat exchanger. The program uses information about the pipe size and flow type to determine the associated pipe resistance, which ultimately is used in the length calculations. The input screen for the piping panel is shown in figure 5.6.

Pipe Parameters

The pipe characteristics are entered in the 'Pipe Parameters' section. They include the pipe resistance, the inside and outside pipe diameter, and the pipe and flow type.

As in the Borehole Design module, GLD calculates the convective resistance using the Dittus-Boelter correlation for turbulent flow in a circular tube (Incropera and DeWitt, 1990). The calculations use average values of the Reynolds number to represent the different types of flow, with values of $Re = 1600$, 3150 , and 10000 for laminar, transition, and turbulent, respectively. The calculations also use average viscosity values and the Prandtl number for water, taken at a temperature of 70°F .

The screenshot shows a software window titled "Horizontal Design Project - HorizontalSample". It has a tabbed interface with tabs for "Results", "Fluid", "Soil", "Piping", "Configuration", "Extra kW", and "Information". The "Piping" tab is selected. Below the tabs is a section titled "Pipe Parameters". This section contains several input fields and dropdown menus:

- Pipe Resistance: 0.156 h*ft**F/Btu
- Pipe Size: 1 in. (25 mm)
- Outer Diameter: 1.32 in
- Inner Diameter: 1.08 in
- Pipe Type: SDR11
- Flow Type: Turbulent

Below these fields is a button labeled "Check Pipe Tables". At the bottom left of the window, there are three small colored squares (pink, grey, and grey).

Fig. 5.6 Piping Panel Contents

Using the standard expression for resistance of a hollow cylinder (Incropera and DeWitt, 1990), the program calculates an approximate value for the pipe resistance. It assumes HDPE pipe with a conductivity of 0.225 Btu/h”ft*F.

The pipe resistance varies with the pipe style and flow. The user can select the size and type of pipe from the appropriate selection boxes. **If another pipe diameter is required, it can be entered directly into the text boxes as needed.**

Note: By pressing the ‘Check Pipe Tables’ button, the ‘Pipe Properties’ tables will open.

If the user wants to enter an experimentally determined pipe resistance, or requires more precise calculations, he or she can enter these values directly into the ‘Pipe Resistance’ text box, overriding all pipe resistance calculations.

Soil

Input parameters relating to the soil are located in the *Soil* panel, as shown in figure 5.7. These include the average ground temperature, the soil thermal properties, and the ground temperature corrections at a given depth.

Fig. 5.7 Soil Panel Contents

Undisturbed Ground Temperature

The undisturbed ground temperature refers to the temperature of the soil below the surface layer, where there is no longer a seasonal swing. This value may be determined from regional data or by recording the actual stabilized temperature of water circulated through pipe in a test bore.

TIP

Soil Thermal Properties

The soil thermal properties are a little harder to define, and care must be taken to provide accurate values, especially for the thermal conductivity. The thermal diffusivity relates to the density of the soil and its moisture



content. Typical values of thermal conductivity and diffusivity for sand, clay, and different types of rocks can be found in the ‘Soil Properties’ tables. However, it is recommended that designers perform soil tests to obtain these values. *The thermal conductivity in particular has a large effect on the calculated bore length, and should be determined with care through in-situ tests or comparison with other projects installed in the local vicinity. GLD does not encourage the use of ex-situ data.*

Diffusivity Calculator

For the designer’s assistance, GLD includes a *Diffusivity Calculator* that can be used to determine the actual diffusivity if all pertinent soil parameters, including the thermal conductivity, the dry specific heat and density, and the moisture level in the soil, are known.

Diffusivity Calculator

Thermal Diffusivity Calculator

Thermal Diffusivity: 0.75 ft²/day

Thermal Conductivity: 1.30 Btu/(h*ft*deg F)

Soil/Rock Specific Heat - Dry: 0.230 Btu/(deg F*lbm)

Soil/Rock Density - Dry: 120.0 lb/ft³

Moisture (0-20): 20.0 %

Close Check Soil Tables

Fig. 5.8 Diffusivity Calculator

Ground Temperature Corrections at Given Depth

In a horizontal configuration, the ground temperature around buried pipes can vary significantly simply due to the proximity to the surface. To account for this variation at different depths, the regional ‘Swing’ temperature and phase shift are used in a sinusoidal equation. The program determines the depth of each pipe in the chosen configuration, and then calculates the expected temperature at that depth.

Regional Air Temperature Swing

This is the temperature swing for the location of interest. It is a measure of the average temperature variation of the region during the warmest and

coolest months as compared to the yearly average temperature. Regions with temperate climates have a lower temperature swing than regions that have large differences between summer and winter temperatures.

Coldest/Warmest Day in Year

These are the actual days of the year, on a 365-day scale, when the temperature is usually coldest or warmest. For example, if February 3 is approximately the coldest day of the year, the value entered will be '34' (31 days in January, plus 3 days of February).

Fluid

The circulating fluid parameters may be entered in the *Fluid* panel. A sample input screen is shown in figure 5.9. In the expanded user interface, fluid temperatures can be viewed and modified at any time, as seen in figure 5.10.

Design Heat Pump Inlet Fluid Temperatures

The heat pump inlet fluid temperatures are included in the *Fluid* panel. The designer can input the desired inlet source temperatures for both heating and cooling here. *When changes are made to these values, the heat pumps in all zones are updated automatically.* **Since the new calculated equipment capacities can lead to changes in selected equipment, the designer must be aware of the changes.** Customized pump values must be adjusted manually.



Design System Flow Rate

The system flow rate per installed ton is included on the *Fluid* panel. *This is the system flow rate per ton of peak load, not installed capacity* (This is because it is assumed that all units will not be running at full load simultaneously, even in the peak load condition).



Optimized systems generally operate in the range from 2.5 to 4.0 gpm/ton, while the ideal system flow rate is somewhere around 3.0 gpm/ton. Again, if the flow rate is changed, the selected heat pumps are updated in the loads modules.

Horizontal Design Project - HorizontalSample

Results | Fluid | Soil | Piping | Configuration | Extra kW | Information

Design Heat Pump Inlet Fluid Temperatures

Cooling: 85.0 °F Heating: 50.0 °F

Design System Flow Rate

Flow Rate: 3.0 gpm/ton

Solution Properties

☒ Automatic Entry Mode

Fluid Type: Water

Freezing Point: 32 °F 0.0% by Weight

Specific Heat (Cp): 1.00 Btu/(°F*lbm)

Density (rho): 62.4 lb/ft³

Check Fluid Tables

Fig. 5.9 Fluid Panel Contents

Design Method

☒ Fixed Temperature

Inlet Temperatures

85.0 °F 50.0 °F

Fig. 5.10 Inlet Temperatures in Expanded User Interface

Solution Properties

Solution properties are also included in the *Fluid* panel. These include the *specific heat* and *density* of the circulating fluid. Also, a reference label is included so that the designer knows the percentage of antifreeze and antifreeze type; *however, this reference label is not currently linked to the other input parameters.*

The specific heat and density values of the antifreeze are used for the calculation of the heat pump outlet temperature, which in turn is used for the trench length calculation.





Additionally, the viscosity of the solution may affect the flow type in the pipe, which was selected on the *Piping* panel. The designer must be aware of any changes made.

In automatic entry mode, the user first selects the fluid type and then selects the desired freezing temperature. GLD automatically displays the specific heat and density for the fluid selection.

In manual entry mode, the user manually selects and inputs the specific heat and density for the target solution.



*Note: Since solution properties vary considerably and non-linearly with type and percentage of additive, GLD does not include detailed automatic antifreeze information for all conditions. Generalized tables of data may be found in the 'Fluid Properties' tables. **It is recommended that the designer manually enter the desired values in the input text boxes.***

Results

All results for both the heating and the cooling calculations can be viewed at any time on the *Results* panel. After all data has been entered or any changes have been made, the user can calculate interim or final results using the 'Calculate' button. The 'Calculate' button is also available in the expanded user interface, as see in figure 5.5.

A sample screen for the *Results* panel can be seen in figure 5.11. Results are also displayed in the expanded user interface, as see in Figure 5.12.

Borehole Design Project - HorizontalSample_1

Results | Fluid | Soil | Piping | Configuration | Extra kW | Information

Calculate

	COOLING	HEATING
Total Trench Length (ft):	4145.9	8177.4
Trench Number:	20	20
Single Trench Length (ft):	207.3	408.9
Total Pipe Length (ft):	12437.6	24532.1
Single Trench Pipe Length (ft):	621.9	1226.6
Unit Inlet (°F):	85.0	50.0
Unit Outlet (°F):	95.2	44.0
Total Unit Capacity (kBtu/Hr):	469.6	464.7
Peak Load (kBtu/Hr):	342.0	243.0
Peak Demand (kW):	27.4	18.2
Heat Pump EER/COP:	12.5	3.9
System EER/COP:	12.5	3.9
System Flow Rate (gpm):	85.5	60.8

Optional Cooling Tower/Boiler

Condenser Capacity (kBtu/hr):	0.0	Cooling Tower	0 %
Cooling Tower Flow Rate (gpm):	0.0		
Cooling Range (°F):	10.0	Boiler	0 %
Annual Operating Hours (hr/yr):	0		
Condenser Capacity (kBtu/hr):	0.0	Load Balance	

Fig. 5.11 Results Panel Contents

Lengths			Temperatures		
	COOLING	HEATING		COOLING	HEATING
Total Trench Length (ft):	4145.9	8177.4	Unit Inlet (°F):	85.0	50.0
Single Trench Length (ft):	207.3	408.9	Unit Outlet (°F):	95.2	44.0

Fig. 5.12 Results Display in Optional Expanded User Interface

The two lists on the *Results* panel are for heating and cooling. Although all of the numbers shown are valid and respond to changes, the side with the longer required length is printed in bold type so that it stands out. The longer length determines the installation size and for this reason the shorter-length system results lose relevance.

The Results panel is divided into two sections. On the top is the reporting section, which presents the calculation results. The lower “*Optional Cooling Tower/Boiler*” section is included to assist in the sizing of a cooling tower and/or boiler. This is a convenient tool for hybrid-type designs, which may be desirable when the cooling length exceeds that of heating or when the heating length

exceeds that of cooling. The cooling tower and boiler options are discussed in more detail below.

Reporting Section



The reporting section is further separated into several subsections. The first deals with the trenches, including the total length, the number of trenches, and the length for one trench. A common way to adjust the trench length to a desired value is to change the trench number on the *Configuration* panel.

The associated pipe length, both total and for a single trench, directly follow the reported trench lengths. The pipe lengths are a function of the selected configuration of pipe in the trench, so the length of trench is always less than the length of pipe when anything other than a single pipe configuration is chosen.

The following subsection of the report lists the heat pump inlet and outlet temperatures of the circulating fluid.

The next subsection lists the total unit capacity, the peak loads and demand of all the equipment, and the calculated heat pump and system efficiencies. The peak load is the maximum and is determined from whichever time period across all the zones has the highest load. The peak demand includes all pumps and external energy requirements, including those listed in the *Extra kW* panel.

Finally, the system flow rate is listed in its own subsection. The system flow rate is calculated from the peak load divided by 12,000 Btu/ton, and then multiplied by the flow rate (in gpm/ton) chosen on the *Fluid* panel. It represents the flow rate from the installation out to the buried pipe system.

Optional Cooling Tower and Boiler Section

Cooling towers and boilers can be added to designs via the sliders that are located on both the *Results* panel (figure 4.16) and in the expanded interface, as seen in figure 5.14. The Cooling Towers and Boilers can be run independently or together in order to balance required lengths or temperatures.

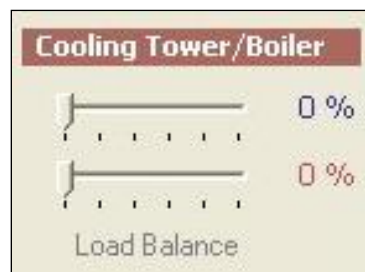


Fig 5.14 Cooling Tower and Boiler Controls in Optional Expanded User Interface

Cooling Towers

Although typically not recommended because of increased running and maintenance costs, the user may elect to add a cooling tower to a cooling-dominated geothermal system to reduce the total boring lengths, and therefore the total initial installation costs. To facilitate this design choice, GLD offers the cooling tower, or “hybrid”, option.

In any case where the calculated trench lengths for cooling are longer than those for heating, the difference in the lengths can be eliminated through the use of a cooling tower tied in parallel to the geothermal ground loop. This requires that either the cooling tower capacity is chosen such that both the peak load and the annual load to the ground are balanced or if a full balance is unnecessary, a capacity is chosen that allows for downsizing the loop to an acceptable length.

To aid in the sizing process, a *Load Balance* control is provided in the Optional Cooling Tower section of the Calculate panel. Although clicking the slider control can initiate a valid calculation or recalculation, the slider control generally is employed after initial calculations have been conducted. The *Load Balance* is a slider-based control that represents a percentage of the total cooling load, both instantaneous (peak) and annual. For example, a *100% Load Balance* would be equivalent to saying that the entire cooling load of the system would be handled by the cooling tower. Conversely, a *0% Load Balance* would mean that a cooling tower is not employed.

In a typical design it is difficult to predict exactly how much load balance or what size of cooling tower is necessary to match the cooling and heating lengths. However, using the *Load Balance* slider control, the designer can optimize the system to the lengths desired by directly controlling the amount of cooling load to be handled by the cooling tower.

In the case where the designer desires the shortest length possible, the design requires a perfect balance of the heating and cooling loads to the ground. The length from this perfect balance would be the minimum length required to adequately cover the heating load requirement. **To accomplish this, the *Load Balance* slider needs to be adjusted to the percentage value where the calculated cooling and heating bore lengths are approximately equivalent.**

TIP

*Note: As expected, the Long Term Ground Temperature Change for both heating and cooling should be **identical** in a perfectly balanced system.*

In other cases, the designer just may seek a reduction in the total required system length rather than a perfect balance. Using the slider control, the

TIP

designer can select the desired length and then note the required cooling tower condenser capacity (as calculated by the program).

Once the required cooling tower capacity is determined, the designer can further modify the various cooling tower parameters to match them to his or her own system. The standard equation used in the program (Francis, 1997) is:

Condenser Capacity (Btu/hr)

$$= \text{Flow Rate (gpm)} \times 500 \times \text{Temperature Difference (}^{\circ}\text{F)},$$

where the 500 is used for pure water, and represents a factor derived from:

$$\text{Specific Heat of Water (1.0)} \times 60 \text{ min/hr} \times \text{Density (8.33 lb/gal)} = 500$$

(Note that GLD actually calculates this factor from the input fluid properties on the *Fluids* panel, although pure water is a logical choice for most cooling-dominated applications.)

For example, if the cooling range is increased above the initial minimum value, the capacity of the condenser also is increased, reducing the total number of operating hours. However, in the same case, decreasing the required flow rate is another option that would keep the condenser capacity and operating hours unchanged. The only limitations are the required temperature difference and the minimum condenser capacity needed to meet the chosen design length. With GLD, users have the flexibility to choose the parameters that fit best in their designs.

Boilers

In GLD, boilers are similar to cooling towers except that they are added in order to reduce the overall heating load on the system. In this case, the user may actually reduce the peak and annual heating loads by the flat percentage defined by the slider value. The required boiler capacity and the modified peak loads applied to the loop field are shown on the panel, but no other inclusion electrical or fuel costs for the boiler are included in the calculation report. The expected heat pump power is also reduced by the same percentage, in order to estimate a real system.

Printing Reports

Reports of the active project can be printed at any time from the Design Studio using the toolbar print button or from the *File* menu → *Print*.

The information printed includes all of the input parameters from the design module, along with the associated results. The zone and loads information is not included with the report, and must be printed separately from the *Loads* panel. The filename of the *.zon file associated with the project report is also listed on the report.

Two different project reports are available: concise and detailed. The concise form includes all of the design parameters, but omits some of the project information and comments. The detailed version includes the project information and comments.

More information on reports can be found in Chapter 7.

References

Francis, E., Editor. *Refrigeration and Air Conditioning, 3rd Edition*. Air-Conditioning and Refrigeration Institute. p.186. Prentice Hall, New Jersey, 1997.

Incropera, F. and Dewitt, D. *Introduction to Heat Transfer, 2nd Edition*. p. 456, p. 98. John Wiley and Sons, New York. 1990.

Paul, N. *The Effect of Grout Thermal Conductivity on Vertical Geothermal Heat Exchanger Design and Performance*.

CHAPTER 6

The Surface Water Design Module

This chapter describes the features and operation of the Surface Water Design module. This module is for the design of systems that use bodies of water, including ponds, rivers, lakes, oceans, etc. It is one of the four design modules included with GLD.

Overview



As with the Borehole and Horizontal Design modules, the calculations made in the Surface Water Design module involve the combination of a large number of input parameters. **Care must be taken to assure that proper values are verified before use.** Assuming that reasonable values are provided to the software, the software will provide a reasonable result.

General Features

The Surface Water Design module in GLD also includes a set of panels, grouped by subject, through which the designer can enter and edit the input variables in a straightforward and efficient manner. For example, parameters related to the body of water are listed on the *Surface Water* panel, while piping choices are listed on the *Piping* panel. Everything related to a project is presented simultaneously and easily is accessible throughout the design process. In the

expanded user interface mode (optional upgrade), which can be expanded by double clicking on any of the tabs, the most commonly modified parameters as well as calculation results are always visible, as seen below in figure 6.1.

Surface Water Design Project - SurfaceWaterSample

Lengths		Temperatures	
	COOLING	HEATING	
Total Length (ft):	4087.2	8187.8	Unit Inlet (°F):
Circuit Length (ft):	371.6	545.9	Unit Outlet (°F):
			COOLING
			HEATING
			55.0
			36.0
			64.2
			30.1

Calculations

Calculate

Design Method

☐ Fixed Temperature

Inlet Temperatures

55.0 °F 36.0 °F

Pipe Layout

Number of Parallel Circuits

11 15

Circuit Style

☐ Coil ☒ Slinky

Results | Fluid | Soil | Piping | Surface Water | Extra kW | Information

Circuit | Header

Circuit Parameters

Circuit Pipe Size: 1 in. (25 mm)

Number of Parallel Circuits

Cooling: 11 Heating: 15

Circuit Style

☐ Coil ☒ Slinky

Primary

Cooling: 0.9 ft.hd. Heating: 1.5 ft.hd.

Extra Equivalent Length per Circuit: 33.1 ft

Fig 6.1 Optional Expanded User Interface

The Surface Water Design module includes several additional features:

- Metric and English unit conversion
- Printed reports of all input and calculated data
- Convenient buttons to bring up tables and calculators
- A ‘Calculate’ button used to refresh the calculations
- A system to monitor header and branch piping head losses

Opening Projects

There are two ways to open Surface Water Design projects. One is by using the ‘New Surface Water’ command from the Design Studio *File* menu and the other

is by opening an existing Surface Water Design project (*.gld) file. Files cannot be opened if other modules with the same name are already open. As many files can be opened as the system's memory permits.

New Projects

New projects may be opened at any time from the Design Studio by choosing 'New Surface Water' from the Design Studio *File* menu or the toolbar. New projects open with standard parameter values that must be edited for new projects.

In new projects, no loads files (*.zon) are loaded. The user must create a new loads file or open an existing loads file into one of the loads modules. Links may be established using the Studio Link system described in Chapter 3.

Existing Projects

Existing projects may be opened at any time from the Design Studio by choosing 'Open' from the Design Studio *File* menu or toolbar. The file automatically opens into a new Surface Water Design Project module.

If a loads file (*.zon) is associated with the loaded project, the loads file will be loaded automatically into the appropriate loads module and opened along with the project file. However, if the associated loads file cannot be found, the user will be notified and the automatic file loading will not occur.

Saving Projects

Projects may be saved at any time using 'Save' or 'Save As' from the Design Studio *File* menu or by clicking the save button on the toolbar. When the user closes the program or module, the program automatically asks the user if he or she would like to save the project file.

Typical Operation

Although each user has his or her own style, the typical operation of the Surface Water Design module would include the following steps:

1. Enter Loads and select pump in either the Average Block Loads module or the Zone Manager module
2. Form a link between the loads module and the design module
3. Modify step-by-step the input parameters listed in each panel
4. Perform initial calculation

5. Modify various parameters and recalculate to determine the effects of the modifications
6. Establish an optimal system
7. Save and/or print the project and associated zone file

Before You Begin

The theoretical model, which is based on experimental data and non-laminar flow, requires a minimum system flow rate of 3.0 gpm/ton in the pipes to achieve proper heat transfer. Minimum flow rates through the circuit piping also are required to maintain the non-laminar flow with different antifreeze solutions. Thus, there is a limit on the maximum recommended number of parallel circuits required in the system, which in turn determines the length of an individual circuit.



Changing the pipe size requires a change in the minimum required flow rates, which can either increase or decrease the maximum recommended number of parallel circuits and their lengths. However, this also can have substantial effects on the piping head losses, which must also be considered in order to reduce the pumping costs.



To fully optimize a system in the Surface Water Design module, **the designer thoroughly must understand the relationship between the system flow rate, the minimum required flow rates, the pipe size, the head loss per length of pipe, and the preferred number of parallel circuits.** GLD can conveniently make all the appropriate calculations, but the designer must first have a grasp of all of the individual inputs required and the relationships among them.

Finally, the surface water designing process actually involves an additional stage of optimization that is not included with the Borehole Design module. The Surface Water module includes a piping calculation component to assist the designer in selecting the best pipe sizes and circuit lengths.

Entering Data into the Tabbed Panels

GLD's innovative tabbed panel system provides for easy organization of and direct access to the relatively large number of design parameters associated with a particular project. This section describes the *Surface Water*, *Piping*, *Soil*, *Fluid*, and *Calculate* panels. The *Information* and *Extra kW* panels are identical to those included in the Borehole Design module described in Chapter 4, so the reader is referred there for detailed information. See Chapter 3 for a discussion of Loads entry.

Surface Water

Use the Surface Water panel to enter data related to the body of water being used as the heat transfer medium. Figure 6.2 shows the associated input screen.

Surface Water Design Project #1

Results | Fluid | Soil | Piping | Surface Water | Extra kW | Information

Surface Water Temperatures at Average Circuit Pipe Depth

Summer: 46.0 °F Winter: 39.2 °F

Surface Water Temperatures at Average Header Pipe Depth

Primary

Summer: 70.0 °F Winter: 35.0 °F

Branches

Summer: 70.0 °F Winter: 35.0 °F

Details (Reference Only)

Surface Water Type: Pond

Surface Area: 4000 ft²

Circuit Pipe Depth: 12.0 ft

Fig. 6.2 Surface Water Panel Contents

Surface Water Temperatures at Average Circuit Pipe Depth

These are the temperatures in the body of water at the depth where the majority of the pipe will reside. The “Circuit Pipe” refers to the main heat exchanger portion of the pipe, and **does not include** the header pipe leading from the surface.

Temperatures in bodies of water naturally change from summer to winter. Both temperatures, at the circuit pipe depth, should be included in this section.

Surface Water Temperatures at Average Header Pipe Depth

TIP

These are the summer and winter temperatures at the average depth in the body of water where the submerged portion of the header pipes reside. “Header Pipe” refers to the section of pipe leading from the surface to the main heat exchanger (circuit) portion of the loop. Further distinctions are described below.

Primary Header

This is the standard “header”, which will most likely come either directly from the installation or from a Manifold that comes from the installation main supply and return lines.

Branches

These will be any branches that split from the primary headers. Generally they will be smaller in size than the primary header.

Details (Reference Only)

The surface water details are not used in any calculations. They are included for the designer’s reference. Several different types of water bodies are included, but the designer can type anything in the selection box.

Piping

The *Piping* panel contains all the information related to the circuit piping and the piping selected for the primary header(s) and up to one level of branching off the primary header(s). The heat exchanger circuits actually dominate the heat transfer, but if the supply and return lines are long or exposed to different design conditions, care must be taken with the header heat transfer. The input screen for the piping circuit panel is shown in figure 6.3. Figure 6.4 is a view of the piping controls in the expanded user interface. Figure 6.5 is the input screen for the piping header panel.

Surface Water Design Project #1

Results | Fluid | Soil | Piping | Surface Water | Extra kW | Information

Circuit | Header

Circuit Parameters

Circuit Pipe Size: 1 in. (25 mm)

Number of Parallel Circuits

Cooling: 11 Heating: 15

Circuit Style

☐ Coil ☒ Slinky

Circuit Head Loss per 100 feet

Cooling: 0.9 ft.hd. Heating: 1.5 ft.hd.

Extra Equivalent Length per Circuit: 33.1 ft

Fig. 6.3 *Piping* Circuit Panel Contents

Pipe Layout

Number of Parallel Circuits

11 15

Circuit Style

☐ Coil ☒ Slinky

Fig. 6.4 *Piping* Controls in Expanded User Interface

Circuit Parameters

Circuit Pipe Size



This is the size of the pipe used in the primary heat transfer circuits. Although larger pipes offer better heat transfer, designers generally prefer smaller sizes (3/4", 1") because of ease of handling and lower pipe costs.

Number of Parallel Circuits



This is the number of parallel circuits required to maintain the required minimum flow rates defined by the designer. If the number of circuits entered here is greater than the allowed number of circuits, **this value will be overwritten automatically with the limiting value when the calculations are performed.**

Even if the circuits are split into equivalent groups (for example, three groups with ten circuits each), the total number of parallel circuits (the smallest unit) will not change.

Circuit Style



Both loose bundled coils and “slinky” (spread out) styles are available. If extensive spacers are used in a coil style arrangement, the slinky model may provide more accurate results, but the loose coil option will provide the more conservative results.

Circuit Head Loss per 100 feet



This is the head loss for the particular style of pipe. *These values are not entered automatically.* Instead, they come from designer’s charts. A chart in English units is included with GLD in the “Pipe Tables” section. **The designer must be aware that this value changes with pipe size, temperature, and flow rate.**

Extra Equivalent Length per Circuit

This is an average pipe length value included per circuit to take into account all fittings (elbows, tees, etc.). It is only necessary for the head loss calculations.

Fig. 6.5 *Piping Header Panel Contents*

Header Parameters

The GLD Surface Water module assumes that a standard supply and return line design will consist of mains, followed by a Manifold that splits the mains into the headers. Headers are generally the first pipes to enter the ground or water. They can then branch off once more if necessary (branch lines). For small systems, the mains may be the headers, and there may not be branches. For larger systems, there may be many headers and multiple levels of branches.

In the *Piping* panel, the model employed allows for multiple headers and multiple first-level branches off of those headers. If further branching is required, the head loss calculations will need to be calculated and added separately. Their effect on the calculated piping length, which cannot be included, will depend on their length. **All headers are assumed to have an identical pipe size, and an approximately equivalent flow.** The same is true for the branch lines.

If there are no branches, the number of branches should be set to zero. The Surface Water Design module starts with only a single primary header.

GLD uses the header information so that the heat transfer losses or gains are taken into account. The software then uses this corrected value iteratively to modify the length of the circuit loop piping, so that the desired entering water temperature for the heat pumps is provided. These calculations depend directly on the header-depth surface water and soil temperatures obtained from the *Surface Water* and *Soil* panels.

Additionally, the program calculates the average head losses of the system when provided with the head losses per 100 ft. for each type of pipe in the system. *These values vary with pipe size, antifreeze, and flow rate.* Several graphs are provided with the program to help determine these values for pure water and standard solutions, **but the designer is ultimately responsible for making sure the appropriate values are entered.** *These head loss calculations also require the one-way length of the header, which is doubled within the program to account for both the supply and return lines.*



Because the inputs to headers and branches are similar, they are described together below.

Number of Lines

This is the number of header or branch lines in the system.

Pipe Size

This is the size of the pipe used in the primary header or branches. For pumping reasons, the size of the primary header is generally larger than the branch and circuit pipe sizes, and branches are generally larger than the circuit pipe size.

Header Length / Average Branch Length

This is the designer-defined **one-way** length of the pipe from the installation to the water line, and then from the water to the circuit pipes. Different heat transfer calculations are used for the header pipe buried in the soil and the header pipe submerged in the water.

If a primary header enters the water, it is automatically assumed that the branches have no soil component. Likewise, if branches enter the soil, it is assumed that the primary header has no water component.

Head Loss per 100 feet

This is the head loss for the particular style of pipe. *These values are not entered automatically.* Instead, they come from designer's charts. A chart



in English units is included with GLD in the “Pipe Tables” section. **As mentioned above, the designer must be aware that this value changes with pipe size, temperature, and flow rate.**

Soil

The *Soil* panel is included only for the heat transfer calculations associated with the portion of the header pipe in the soil. The model uses the undisturbed ground temperature of the soil as well as several other parameters associated with the installation location to determine the temperature at pipe depth on the coolest and warmest days of the year. This temperature then is used to determine how much heat is transferred from the header pipe to the soil or vice versa.

Once the amount of heat transfer from or to the soil is known, the circuit pipe length (calculated from the surface water data) can be modified to provide fluid with the desired inlet source temperature to the heat pumps.

The *Soil* panel input screen is shown in figure 6.6.

Surface Water Design Project #1

Results | Fluid | **Soil** | Piping | Surface Water | Extra kW | Information

Undisturbed Ground Temperature

Ground Temperature: 62.1 °F

Ground Temperature Corrections at Given Depth

Depth of Header in Soil: 4.0 ft

Soil Type: Wet

Regional Air Temperature Swing: 22.0 °F

Coldest/Warmest Day in Year: Winter: 34 Summer: 225

Corrected Temperature (°F): 48.1 76.8

Check Soil Tables

Fig. 6.6 *Soil* Panel Contents

Ground Temperature Corrections at Given Depth

Depth of Header in Soil

This is simply the average depth in the soil between the water's edge and the installation at which the primary header or branches will be buried.

Soil Type

The soil type can have one of three values: wet, dry, or average. GLD uses this to assign an approximate diffusivity value to the soil used in the temperature model.

Regional Air Temperature Swing

This is the temperature swing for the location of interest. It is a measure of the average temperature variation of the region during the warmest and coolest months as compared to the yearly average temperature. Regions with temperate climates have a lower temperature swing than regions that have large differences between summer and winter temperatures.

Coldest/Warmest Day in Year

These are the actual days of the year, on a 365-day scale, when the temperature is usually coldest or warmest. For example, if February 3 is approximately the coldest day of the year, the value entered will be '34' (31 days in January, plus 3 days of February).

The program uses these days to determine the soil temperature at the given depth at these times of the year.

Corrected Temperature

These are the corrected temperatures at the depth specified, calculated automatically from the undisturbed temperature and the other input values provided. These values are used in the heat transfer calculation between the header or branch pipes and the soil.

Fluid

The fluid panel is identical to the one described for the Borehole Design module in Chapter 4 except for one addition. That addition is the minimum required circuit flow rate in the lower 'Minimum Circuit Flow Rate and Solution Properties' section. The added section is shown in figure 6.7. As in the other modules, the inlet temperatures can be viewed and modified from the optional expanded interface, as seen in figure 6.8.

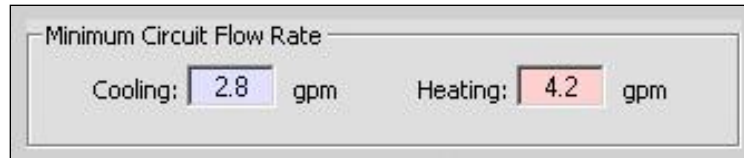


Fig. 6.7 Minimum Circuit Flow Rate Section of the *Fluid* Panel



GLD uses this information in conjunction with the system flow rate to establish the maximum number of parallel circuits. The flow rates required for non-laminar flow for several antifreeze solutions are included as a table in the 'Fluid Properties' set. **Exact values for a particular mixture may need to be determined independently by the designer.**

Note once again that changes in the inlet source temperature or the system flow rate will cause an automatic update of the selected pumps.

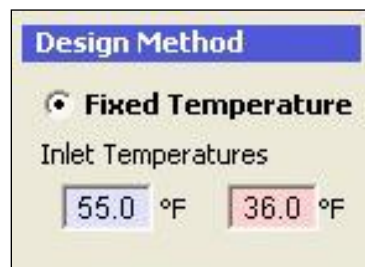


Fig. 6.8 Inlet Temperature Controls in Optional Expanded User Interface

Results

There are several significant differences between the Surface Water Design module's *Results* panel and the Borehole Design module's *Results* panel. These differences relate to the nature of the calculations, as well as to the inclusion of the head loss calculation results. Figure 6.9 shows a typical view of the *Results* panel. Figure 6.10 shows the results display in the expanded user interface. Figure 6.11 shows the 'Calculate' button in the expanded user interface.

Again, there are two lists shown on the *Results* panel, one for heating and one for cooling. Although all of the numbers resulting from both sets of calculations are valid, the side with the longer length is printed in bold type, so that it stands out. The longer length usually determines the installation size, and for this reason the shorter-length system results lose relevance. *However, in cases where the cooling and heating lengths are similar, care must be taken to assure the safest design.*

Surface Water Design Project - SurfaceWaterSample		
Results Fluid Soil Piping Surface Water Extra kW Information		
Calculate	COOLING	HEATING
Total Length (ft):	4087.2	8187.8
Circuit Length (ft):	371.6	545.9
Number of Circuits:	11	15
Max. Parallel Circuits:	11	15
Unit Inlet (°F):	55.0	36.0
Unit Outlet (°F):	64.4	30.1
Approach Temp. (°F):	5.9	-3.1
Total Unit Capacity (kBtu/Hr):	415.5	277.4
Peak Load (kBtu/Hr):	130.6	253.1
Peak Demand (kW):	8.4	2.9
Heat Pump EER/COP:	20.5	3.7
System EER/COP:	14.3	3.2
Total Head Loss (ft.hd.):	8.6	26.3
Header Loss (ft.hd.):	5.0	17.6
Circuit Loss (ft.hd.):	3.6	8.7
System Flow Rate (gpm):	32.6	63.3
Primary Header (gpm):	16.3	31.6
Branch Header (gpm):	8.2	15.8
Circuit (gpm):	3.0	4.2

Fig. 6.9 Calculate Panel Contents

Lengths			Temperatures		
	COOLING	HEATING		COOLING	HEATING
Total Length (ft):	4087.2	8187.8	Unit Inlet (°F):	55.0	36.0
Circuit Length (ft):	371.6	545.9	Unit Outlet (°F):	64.4	30.1

Fig. 6.10 Results as Displayed in Optional Expanded User Interface

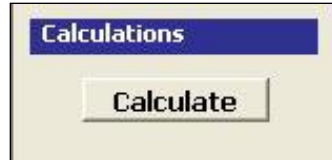


Fig. 6.11 Calculate Button in Optional Expanded User Interface

Reporting Section

TIP

The surface water report has five sections. The first deals with the circuit pipe, and includes the total length, the length for one circuit, the number of circuits, and the maximum allowable number of parallel circuits (shown in red). If the maximum allowable number of parallel circuits exceeds the actual number of circuits, the actual number of circuits may be increased in the *Piping* panel to reduce the individual circuit lengths and thus reduce head losses. However, this type of reduction is not always necessary or desirable. Other ways of increasing the maximum allowable number of parallel circuits include changing the system flow rate or the minimum circuit flow rate for non-laminar flow.

The second section lists different temperature variables. The first two of these are Source inlet and outlet temperatures. The final variable is the approach temperature, which is the difference between the pond temperature and the desired inlet source temperature.

TIP

Note: In surface water heating applications, although the solution within the pipe may not freeze, the freezing temperature of the body of water is generally 32°F. If the heat pump outlet temperature is too far below this value, the water may freeze on the pipe, greatly reducing its heat transfer characteristics and potentially leading to system failure. The designer must always pay attention to the outlet temperature value for this reason.



As with the Borehole Design module, the third section lists the total unit capacity, the peak loads and the demand of all the equipment, followed by the calculated heat pump and system efficiencies. The peak load is the maximum, determined from whichever time period across all the zones has the highest load. The peak demand includes all pumps and external energy requirements, including those listed in the *Extra kW* panel. *Care must be exercised when equipment energy requirements listed in the Extra kW panel refer to only heating or only cooling types of equipment.* In these cases, the pump efficiency could be fine, but the system efficiency might be incorrect.

The fourth section lists the total head loss calculation results, as well as the individual losses for the header and circuit pipe. It does not include any losses for the heat pump equipment, which must be considered separately. This section is convenient for determining the optimum pumping arrangement for the system.

Finally, the system flow rate is listed along with the flow rates in the primary and branch headers, as well as the flow in the individual circuits. The system flow rate is calculated from the peak load divided by 12,000 Btu/ton, and then multiplied by the system flow rate in gpm/ton (as given on the *Fluid* panel). The primary header flow rate is calculated from the system flow rate divided by the number of primary headers, and the branch flow rate is obtained from the primary header flow rate divided by the number of branches (as given on the *Piping* panel). The circuit flow rate is obtained by dividing the system flow rate by the total number of circuits (also provided on the *Piping* panel).

Printing Reports

Reports of the active project can be printed at any time from the Design Studio using the toolbar print button or from the *File* menu → *Print*.

The information printed includes all of the input parameters from the design module, along with the associated results. Zone and loads information can be printed separately from the *Loads* panel. The filename of the *.zon file associated with the project report is also listed on the report.

Three different project reports are available: concise, detailed and detailed with loads. The concise form includes all of the design parameters, but leaves out some of the project information and comments. The detailed form includes the information and comments.

More information on reports can be found in Chapter 7.

CHAPTER 7

Reports

This chapter covers the report creation and printing features of GLD. It includes project, zone and financial reports.

Overview

GLD includes reporting features. These features have been added for professionals who need to keep records of their designs and communicate them to others. There are nine different report styles included within the package, and this chapter provides an explanation of as well as suggested uses for each type of report.

The Report Preview Window

When a particular report is selected, a report preview window opens to show a preview of the report. Report preview windows have a zoom feature that allows adjustment of the magnification. Additionally, reports may be sent to a printer or exported as various file types, including text and html. Multiple reports may be opened simultaneously, even if they originate from the same project.



Report preview windows do not react directly to metric/English unit conversion. Instead, a report opens with the same units used by its parent design module. If another system of units is required, the user must first change the unit system of the design module (using the Design Studio 'Units' menu), and then open a new report.

Project Reports

Project reports may be opened at any time from the Design Studio *File* menu by selecting *Print*. An option dialog box appears displaying the six types of reports that are available: *concise*, *detailed*, *input data with loads*, *concise temperature*, *detailed temperature*, and *full project*. The first two project reports are available in all three heat exchanger design modules. Detailed reports contain full project information, while concise reports limit the project information and exclude any comments. Detailed reports generally require multiple pages while concise reports are designed for single-page printouts. The other four project reports are associated with monthly inlet temperatures and therefore are available only with the borehole module. The user selects a preference and then clicks ‘OK’.

The report does not print automatically, but instead creates the report preview window, in which the report can be reviewed prior to printing. Printing can be done by clicking on the printer icon in the upper left hand corner of the report preview window.

In general, project reports contain several main sections:

- **Information**
- **Calculation Results**
- **Input Parameters**
- **Loads** (borehole only when linked to Average Block module)
- **Monthly Inlet Temperatures** (borehole only when linked to Average Block module)
- **Comments**

Information

This section contains the information from the design module’s *Information* panel. The project and designer’s names, dates, client’s name and address, etc. appear here. This section is included at the top of every report. Concise reports only include the project name and start date.

Calculation Results

This section lists the results of the calculations and essentially is the same information shown on the *Calculate* panel of the design module. The most important results, such as the total length of pipe required, are highlighted and boxed in order to stand out from the background. The report presents results of both the heating and the cooling calculations.

Input Parameters

This section contains all of the parameters entered by the designer during the design process. Parameters are placed into sections with names taken directly from the panels in the heat exchanger design modules. The filename of the zone file associated with the project is listed under the *Loads* heading.

Loads

This section contains all of the loads data entered in the Average Block loads module (peak loads and monthly loads as well, if entered). This section is only available in borehole module reports since only the borehole module is capable of calculating monthly inlet temperatures based on the input loads.

Monthly Inlet Temperatures

This section contains a summary section of the average and peak inlet temperatures followed by the month-by-month temperatures, and other associated data.

Comments

This section, at the end of the report, is reserved for any additional information that the designer would like to include with the project.

Zone Reports

Zone (or loads) reports are printed directly from the *Loads* modules. They include only the project information and data from the zones, presented in different formats. Five different zone reports exist, containing complete or specific information about the zones.

Zone reports work in conjunction with project reports, but are actually a separate entity. They are representative of the actual installation rather than the heat exchanger portion of the system. Zone delineation, loads, and equipment are separate from the heat exchanger system. It is for this reason that the designer would necessarily want to view and consider this information apart from the specific heat exchanger details. For example, if the design is a building, the zone reports will cover everything within the building, while the project report essentially will contain information about everything outside, or external to the building.

A zone report is printed from the *Loads* panel of the Zone Manager or directly from an Average Block Loads module by clicking the printer button in the controls. A dialog window appears, giving the designer the list of available report styles. After the making a choice, click 'OK' to bring up the report window.

There are five different zone reports included with GLD:

- **Detailed Form**
- **Concise Form**
- **Equipment List**
- **Loads List**
- **Names List**

Detailed Form

The *Detailed Form* zone report is the most detailed zone report. It lists all of the information included in every zone, along with full explanations of the listed parameters. The format is open and easy to read. However, as with the project reports, the detailed form produces a much longer printed report than any of the more compact versions.

Concise Form

The *Concise Form* zone report contains most of the detail of the long report, but it is packed into a smaller space. It does not include zone names, occupation days, detailed pump information (manufacturer, series, and type), or full descriptions of the items listed. It does, however, contain important information about the loads and the operational parameters of the equipment matched to those loads.

Equipment List

The *Equipment List* lists only the equipment associated with each zone. It provides detailed pump information, including name, number, manufacturer, series, and type, plus all of the operational data associated with that pump. It is an ideal report for engineers or contractors who require equipment lists but do not necessarily need to know further details about the design.

Loads List

The *Loads List* lists only the loads associated with each zone. It provides the Design Day loads at the different periods during the day in both heating and cooling modes. For the Borehole Design module, the Loads report includes the annual hours and weekly occupation information.

Names List

The *Names List* is just a list of the full reference names of the different zones, combined with the zone number, pump name, and number of pumps required for the zone. It makes a convenient, compact link between zone name and number, and is especially useful when the project consists of many separate zones.

Finance Reports

Finance reports are printed directly from the Finance module. They include the project information and financial data, presented in different formats. Five different finance reports exist.

A finance report is printed directly from the Finance module by clicking the printer button in the controls. A dialog window appears, giving the designer the list of available report styles. After the making a choice, click 'OK' to bring up the report window.

There are five different zone reports included with GLD:

- **Concise Form**
- **Detailed Form**
- **Concise Inputs Form**
- **Detailed Inputs Form**
- **Financial Analysis Form**

Concise Form

The *Concise Form* finance report is the simplest finance report. It lists the fuel/energy usage and costs on an annual and NPV lifetime basis for only the geothermal system.

Detailed Form

The *Detailed Form* finance report is the most detailed finance report. It lists the fuel/energy usage and costs on an annual and NPV lifetime basis for both geothermal and conventional HVAC systems.

Concise and Detailed Inputs Forms

The *Concise/Detailed Inputs Forms* contain lists of all of the inputs used in the financial analysis.

Financial Analysis Form

The *Financial Analysis Form* finance report provides a useful financial comparison of the geothermal and one conventional system. In addition, it provides simple payback summary results.

Thermal Conductivity Report

The Thermal Conductivity report is printed directly from the Thermal Conductivity module. The report includes all inputs and calculated results, including a number of color graphs.

Computational Fluid Dynamics Reports

Reports related to the CFD module are described in chapter 11.

Concluding Remarks

There are no data in GLD that are not expressible in a printed form. The designer can organize and share information both during the developmental stages of a project and after the design is complete.

CHAPTER 8

Tables and Reference Files

This chapter covers the tables and reference files of GLD. It starts with a description of the included files, and then explains how the user may add customized files to the existing set.

Overview

Favorite references are like a comfortable pair of worn-in sneakers. Although this software package provides some useful information in the included tables, it may never replace the old standards. Rather than trying to impose a particular system onto the users of the software, GLD employs a technologically sophisticated system that allows the user to customize the reference files as much as he or she desires. With this system a new pair of shoes feels comfortable immediately.

The reference files included with GLD are minimal, consisting of a few tables and graphs that should aid in the selection of requested parameters. All files are written in open HTML (Hypertext Mark-up Language) files. The designer can edit and add to them as he or she desires to create a customized reference library within the Design Studio environment.

As with the heat pump and loads models, the reference files model is another customizable element of the geothermal Design Studio that the user has the option to control.

Tables Included with GLD

Several tables are included with GLD. They are separated into several broad categories from which most questions will arise. These include:

- **Fluid Properties**
- **Soil Properties**
- **Pipe Properties**
- **Conversions**

The first three sections present a menu screen with hyper-links to various tables that have been included in the package. The fourth section consists of a pair of metric-to-English units conversion tables that answer most common engineering conversion problems. Below is a description of the included files.

Fluid Properties

Fluid properties refer to any data related to the circulation fluid. The five *Fluid Properties* tables in GLD are the following:

Table 1: Densities and Specific Heats of Various Solutions

Table 2: Minimum Required Flow Rate for Non-laminar Flow

(Tables 3-5 included only in English Units)

Table 3: Head Loss in SDR 11 HDPE Pipe - 20% Propylene Glycol

Table 4: Head Loss in SDR 11 HDPE Pipe - 20% Methanol

Table 5: Head Loss in SDR 11 and 17 HDPE Pipe - Pure Water

Some of these charts could have also been placed with the Pipe Properties tables, but because they vary primarily with solution type, they were placed here.

In an ideal world, the *Fluid Properties* tables would include all of the graphs, charts, and tables for all of the parameters of all possible antifreeze combinations. However, because these variations are difficult to predict for specific projects, only partial information has been included. For the most accurate designs, designers are encouraged to seek out their own favorite antifreeze combinations, and determine the specific heat, density, and minimum required flow rate for non-laminar flow.



Soil Properties

Soil properties refer to any data related to the soil. The four reference files are listed below.

Table 1: Thermal Conductivity and Diffusivity of Sand and Clay Soils

Table 2: Thermal Properties of Rocks at 77⁰ F

Table 3: Earth Temperatures, Soil Swing and Phase Constants for U.S. Cities

Table 4: Earth Temperatures, Soil Swing and Phase Constants for Canadian Cities



The first two *Soil Properties* tables included with GLD provide various soil parameters, including ranges for thermal conductivity (k) and thermal diffusivity (α) for various types of soils.

These tables should not be considered accurate for a given location; however, they should provide the designer with a realistic range within which their own measurement results should fall.

The third and fourth tables contain mean earth temperatures and other parameters for U.S. and Canadian cities. These tables particularly may be useful for horizontal designs.

Pipe Properties

Pipe properties refer to any data related to the piping. The *Pipe Properties* tables included with GLD are related to either the borehole thermal resistance or the pipe physical data. They are listed below.

Table 1: Thermal Conductivities of Typical Grouts and Backfills

Table 2: Pipe and Tube Dimensions

Table 3: Required Flow Rates to Achieve 2 ft/s – SDR11 Pipe

The first table provides thermal conductivities for some typical grouts. The second lists the physical dimensions (inner and outer diameter) for common pipe sizes in various types of pipe. The third, although unnecessary for the associated calculations, provides some convenient flow rates required for proper purging of a piping system.

Conversions

The Conversions table has two separate lists of metric to English conversions, placed together in one file. As already mentioned, the user can obtain multipliers for most common metric/English unit changes by going through the listed conversions.

Adding Customized Reference Files

The user can create customized reference files by editing the existing HTML files with the table lists, and making new links. The process is simple and requires only a very basic knowledge of HTML

Original Model

The original model included with GLD consists of these files:

English

FluidTables.html
 FluidTable1.html
 FluidTable2.html
 FluidTable3.html
 FluidTable4.html
 FluidTable5.html

SoilTables.html
 SoilTable1.html
 SoilTable2.html
 SoilTable3.html
 SoilTable4.htm

PipeTables.html
 PipeTable1.html
 PipeTable2.html
 PipeTable3.html

Metric

FluidTablesMetric.html
 FluidTable1Metric.html
 FluidTable2Metric.html

SoilTablesMetric.html
 SoilTable1Metric.html
 SoilTable2Metric.html
 SoilTable3Metric.html
 SoilTable4Metric.html

PipeTablesMetric.html
 PipeTable1Metric.html
 PipeTable2Metric.html
 PipeTable3Metric.html

To add a new file, the **FluidTables.html**, the **SoilTables.html**, or the **PipeTables.html** must be edited. The user must create a link in one of the three aforementioned *.html files to the new file (which contains the table, graph or image that the user would like to have available in GLD).



*Note: GLD requires the **FluidTables.html**, **SoilTables.html**, and **PipeTables.html** files (and their metric counterparts, **FluidTablesMetric.html**, **SoilTablesMetric.html**, and **PipeTablesMetric.html**) as the initial files when opening the associated tables. They can be edited, but if they are deleted the associated tables cannot be opened at all.*

HTML Files

HTML refers to Hypertext Mark-up Language. It is the language used on web pages, and commonly used in software to quickly provide linked information to users. HTML files can be created with an HTML editor (like those distributed with common browsers) or with a simple text editor. They must, however, follow a certain format and have a '.htm' or '.html' extension.

Editing Existing Files

Existing files may be edited by simply opening up the original file into a text editor or HTML editor, making changes, and then saving the file again. For example, if a user wishes to add a new pipe table to the list, he or she first will create the table (i.e. PipeTable4.html) and then will add a link to it on the PipeTables.html file.

Additionally, if the user wishes to add additional information to an existing table or figure, he or she only has to open the appropriate HTML file in a text editor or HTML editor and make and save the desired changes. For example, if adding a new link, **PipeTables4.html**, to the **PipeTables.html** file, one might add this new link with the name "Table 4: New Pipe Table" by typing the new link at the end of the **PipeTables.html** file into a text editor as follows (the added section is in **bold** type):

```
.  
.   
.   
<li>  
<a href="PipeTable3.html">Table 3: Required Flow Rates to Achieve  
2ft/s – SDR 11 Pipe</a></li>  
</ul>  
  
<li>  
<a href="PipeTable4.html">Table 4: New Pipe Table</a></li>  
</ul>  
  
</body>  
</html>
```

PipeTables.html (edited version)

Making a Table

A new table can be made at any time by creating one as an HTML file. The easiest way to do this is to use an HTML editor. It is much more difficult to make a table using plain HTML in a text editor.

Although any name is valid for a table, tables can be added to the appropriate group by just extending the naming sequence already being used. For example, the name **PipeTable4.html** could be used as the name for a new file.

Adding a Picture, Graph, or Figure

If an image is stored as either ***.jpg** or ***.gif**, it can be imported into an HTML page. The HTML page can be linked directly to the GLD reference files.

As an example, let's assume that an engineer scans an image of his favorite density vs. percent solute graph for Calcium Chloride and saves it in the Help Files directory as a *jpeg* image, called **CaCl2Density.jpg**. A very simple HTML file can be created with a text editor, and called **FluidTable6.html**. The entire **FluidTable6.html** file would be as follows:

```
<html>
<head></head>
<body>
<img SRC="CaCl2Density.jpg" >
</body>
</html>
```

FluidTable6.html

Remember, the **FluidTables.html** file would have to be edited to include the new link to the **FluidTable6.html** file, similar to the example given in *Editing Existing Files*, above.

If everything is done properly, when 'Fluid Properties' is selected from the *Tables* menu in the Design Studio, 'Table 6' will appear as a link in the list of available tables. By clicking on the link, the CaCl2 density image, **CaCl2Density.jpg**, will appear, and can be used as a convenient internal reference.

Taking Care with Updates

Updated versions of GLD may have new reference files and new versions of **FluidTables.html**, **SoilTables.html**, or **PipeTables.html**. If this is the case, then any custom changes to these files made by the user may be overwritten during a new installation. *Although the linked files will remain, the user is advised to make backup files of all customized reference files before new GLD installations or updates.*

Concluding Remarks

The reference files in GLD are added entirely for the user's convenience. Designers should find the customizable geothermal Design Studio an ideal and familiar environment in which they can conduct their work with the highest levels of efficiency and confidence.

CHAPTER 9

The Finance Module

This chapter describes how to use the GLD finance module, a module that models both “hard” and “soft” costs associated with geothermal and standard HVAC systems. All of the calculations fundamentally are based on data provided by the designer, providing for the greatest range of flexibility and accuracy.

Overview

When designers, architects and building owners are deciding whether or not to install a ground source heat pump system, they must consider a variety of factors including cost. Cost means different things to different people. Some think of the “hard” costs –the first costs associated with the design and installation of an HVAC system. Others think of lifecycle operating costs. In an increasingly green-focused world, still others think of environmental costs. Finally, some percentage think of the “soft” costs associated with HVAC systems- the opportunity costs associated with large vs. small mechanical rooms, the varying maintenance costs associated with one system vs. another, and even the water consumption costs associated with some types of systems, such as geothermal-cooling tower hybrid systems.

The Ground Loop Design financial module allows designers to model and estimate all of the aforementioned costs- from expected future CO₂ emissions costs to the annual and lifetime operating costs of geothermal, hybrid, and more standard HVAC systems. Furthermore, it enables decision makers to compare simultaneously the financial profiles of multiple systems.

The finance module either can be used on a standalone basis or in conjunction with a heat exchanger system designed in GLD. On a standalone basis, users can enter minimal data for a quick energy cost and emissions estimate or can enter detailed data for a more comprehensive financial analysis.

Users also can model the financials of a heat exchanger system designed in GLD. The program automatically transfer the applicable parameters into the finance module and reports the financial and emissions analysis. As with the other modules in Ground Loop Design, it is important to remember that the calculated results are only as good as the quality of the user-defined inputs. Assuming that reasonable values are provided to the software, the software will provide reasonable results. It is also important to note that the finance module is only an estimation tool and for a variety of reasons, installed HVAC systems may have costs and emissions that vary significantly from the estimates.

General Features

To aid in the data entry process, the Finance module in Ground Loop Design consists of a set of panels, grouped by subject, through which the designer can enter and edit the input variables efficiently. For example, parameters related to the utility costs are listed on the *Utility Costs* panel, while conventional system comparison choices are listed on the *Conventional* panel. The idea is that everything related to a single financial project is presented simultaneously and is easily accessible at any time during the design process. The tabbed panels can be seen in figure 9.1, below.



Fig. 9.1 Financial Model Panel List

The Financial module includes several additional features:

- Analyses and comparisons are based on:
 - Energy usage costs
 - CO₂ emissions costs
 - Water usage costs
 - Maintenance costs
 - Mechanical room lease value opportunity costs
 - Installation costs
 - Salvage values/costs
 - Tax Incentives
 - Adjustable inflation and discount rates

- Metric and English unit conversions
- Printed reports of all input and calculated data
- A 'Calculate' button used to refresh the calculations
- Quick importation and modeling of systems designed in the vertical, horizontal and pond modules
- Stand alone financial analysis capabilities
- Comparison of a geothermal system with up to four alternative systems

Theoretical Basis

The financial module analyzes a number of hard and soft costs associated with geothermal and other HVAC systems. It models these costs both for a single year and for the building lifetime.

Many of the factors required for these analyses are user-definable and the level of analysis depends on the needs of the user. If the user enters only some of the cost factors, then some costs can not be calculated or displayed. If the user enters all of the cost factors, then all of the costs can be calculated and displayed.

For the single year costs, the program sums up the various costs for a single year of operation and displays the results. For the lifetime costs, the program uses a net present value (NPV) analysis that incorporates an overall discount rate as well as inflation rates associated with different fuel types.

Opening Projects

There are two ways to open Finance projects. One is by using the 'New Finance' command from the Design Studio *File* menu or toolbar, and the other is by opening an existing Finance project (*.fin) file from within the finance module. In the design studio, only one financial module can be open at a time.



New Projects

New financial projects may be opened at any time from the Design Studio by choosing 'New Finance' from either the Design Studio *File* menu or the toolbar. New projects open with standard cost values that the user can modify as necessary for new projects. The module opens directly into the *Results* panel.

New financial projects can be for a stand-alone financial analysis or for use in conjunction with an existing heat exchanger design project. For use in conjunction with an existing heat exchanger design project, see below.

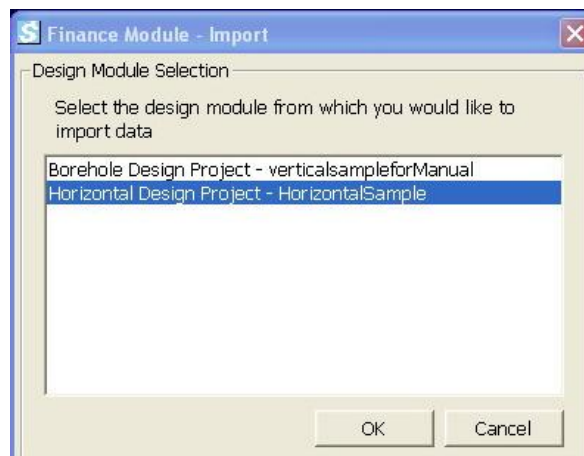
Importing Data from an Open Heat Exchanger Design Project

If a designer wishes to perform a financial and emissions analysis of a vertical, horizontal or pond project that he or she designed with GLD, he or she can do so by following these steps:

- 1) Open the project file (vertical, horizontal, or pond) of interest and make sure that the loads file (zone manager or average block) that is linked to the project file is open as well.
- 2) Push the import button on the toolbar at the top of the finance module. It looks like this:



- 3) A window similar to the image below will appear.



- 4) Select the project/design module of interest and click 'Ok'.
- 5) The relevant design parameters automatically will be loaded into the finance module.

Please note that if a user imports a surface water project, the user must manually enter equivalent full load hours into the geothermal tab. This is because in GLD, the surface water loads modules neither have nor require full load hours inputs (see Chapter 6 and page 156).

Existing Projects

Existing Finance projects may be opened at any time from within the Finance module by choosing 'Open' from the Finance module toolbar.

Saving Projects

Finance projects may be saved at any time by clicking the save button on the Finance module toolbar. When the user closes the program or module, the program automatically asks the user if he or she wants to save the finance project.

Typical Operation

Although each user will have his or her own unique method, the typical operation of the Finance module would include the following steps:

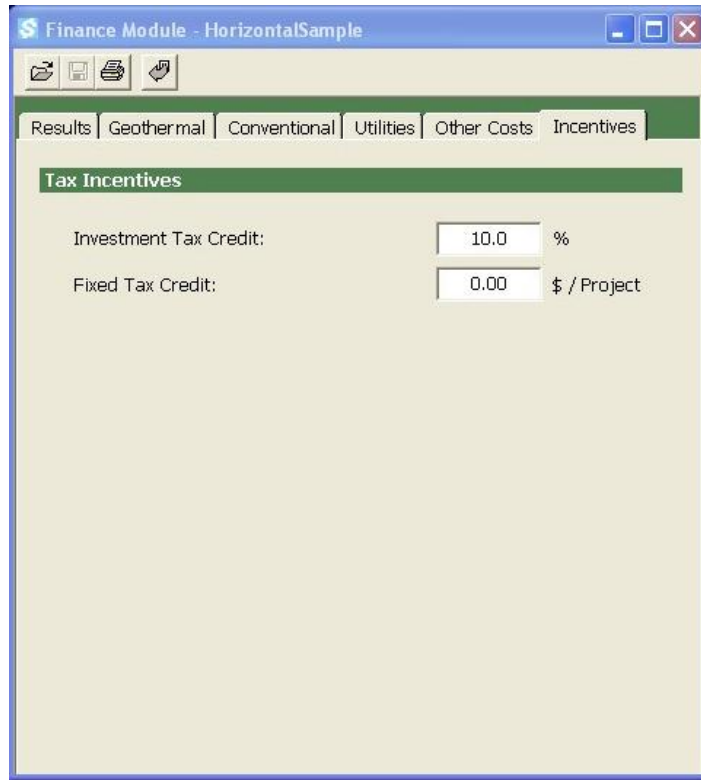
1. Open a new finance module
2. Choose metric or English units
3. If necessary, enter/modify project specific financial data in the incentives, other costs and utility costs tabbed panels
4. Either link to an open heat exchanger design file or manually enter the geothermal project data
5. In the conventional tabbed panel, choose up to 4 conventional systems to compare to the geothermal system
6. In the results tabbed panel, hit the calculate button to view the financial analysis
7. Make modifications as necessary
8. Save and/or print the finance project reports

Entering Data into the Tabbed Panels

Ground Loop Design's innovative tabbed panel system provides for easy organization of and direct access to the relatively large number of design parameters associated with a particular project. This section describes the *Incentives*, *Other Costs*, *Utility Costs*, *Conventional*, *Geothermal* and *Results* panels.

Incentives

Information pertaining to financial incentives for geothermal systems can be found in the *Incentive* panel as seen in figure 9.2, below. If incentives are available, users can enter the incentive as an investment tax credit percentage or as an absolute tax credit. For example, in late 2008, the U.S. Congress passed H.R. 1424 which authorizes up to \$2000 in federal tax credits for residential systems and 10% federal tax credits for commercial systems. For commercial projects in the US, users can enter 10% in the investment tax credit text box. For residential projects in the US, users can enter up to \$2000 in the fixed tax credit text box. Incentives are subtracted from the installation costs and reported in detail in the reports.



Finance Module - HorizontalSample

Results | Geothermal | Conventional | Utilities | Other Costs | Incentives

Tax Incentives

Investment Tax Credit: 10.0 %

Fixed Tax Credit: 0.00 \$ / Project

Fig. 9.2 *Incentives* Panel Contents

Other Costs

Information pertaining to a variety of hard and soft costs can be found in the *Other Costs* panel. This includes all of the baseline data for non-utility costs including CO₂ emissions costs, average building costs and equipment-related costs. The contents of the *Other Costs* panel are shown in figure 9.3.



All of the data entry options in the *Other Costs* panel are optional, but by entering the data, the program is able to calculate many of the hard and soft costs associated with HVAC systems.

Note that calculating the soft cost benefits of geothermal systems may help designers convince clients of the important, yet oftentimes overlooked benefits of geothermal HVAC systems.

The *Other Costs* panel is divided into three sections: emissions costs, average building costs and equipment-related costs.

Finance Module - HorizontalSample

Results | Geothermal | Conventional | Utilities | **Other Costs** | Incentives

Emissions Costs

CO₂ Emission Rate: 1.4 lbs / kWh

CO₂ Emissions Cost: 30.00 \$ / ton

Effective Initiation Delay: 3 yr

Average Building Costs

Total Structure Floor Space: 14,000 ft²

Average Building Construction Cost: 100.00 \$ / ft²

Lease Value: 3.00 \$ / (ft² * yr)

Equipment-Related Costs

System Type: Geothermal Heat Pump

Fuel Type: Electricity

Installation Costs: 6.50 \$ / ft²

Maintenance Costs: 0.00 \$ / ft²

Salvage Value: 0.00 \$ / ft²

Fig. 9.3 Other Costs Panel Contents

Emissions Costs

As the global response to climate change intensifies, CO₂ emissions regimes will likely become the norm. These regimes may include cap and trade mechanisms, taxes, and other to-be-determined processes for incentivizing emissions reductions. In some countries, designers already are looking at geothermal HVAC systems both as a source of emissions credits that they can sell in the growing carbon markets for a profit as well as an attractive application of the Kyoto Protocol Clean Development Mechanism (CDM). It is likely that the Kyoto Protocol will be superseded in the next few years by a new agreement that requires even more stringent CO₂ emissions regimes. Additionally, local, regional and national level emissions control regimes are becoming more common. For these reasons, the finance module enables designers to determine the CO₂ emissions reductions associated with a geothermal system compared to a more traditional HVAC solution.

The **CO₂ emissions rate** is the carbon intensity per kWh of electricity generated. This rate is based on the fuel mix (coal, hydro, nuclear, etc) used to generate the electricity that will power the electrical geothermal

HVAC systems. This intensity data can be found fairly easily on state, provincial, national and NGO environmental protection websites. As the emissions rate can vary greatly, it is recommended that the designer spend a few minutes finding the appropriate rate for the project's region. In the USA, the national average is 1.34 lbs of per kWh. Detailed information on each state or province can be found on the following websites:

<http://www.eia.doe.gov/oiaf/1605/ee-factors.html> (USA)

http://www.ec.gc.ca/pdb/ghg/inventory_report/2005_report/a9_eng.cfm (CANADA)

The finance module enables designers to specify other energy sources besides electricity. Options include fuel oil (#2), natural gas, propane/LPG, coal (weighted average of anthracite, bituminous and semibituminous), wood and biomass. Because the CO₂ emissions from these fuel types tend not to vary as much as the CO₂ emissions associated with electrical power generation, the program uses standard CO₂ emissions coefficients for these other fuel types. These emissions coefficients are from the 2006 IPCC Guidelines for National Greenhouse Gas Inventories, Volume 2.

The **CO₂ emissions cost** enables GLD to put a price tag on the CO₂ emitted from a range of HVAC systems. While at present time many consider this a soft cost, soon it may be a hard cost that influences investment decisions. As a result, it is especially important to consider the CO₂ emissions costs over a project's lifetime. While emissions costs estimates vary within the range of \$5 USD/ton to \$80 USD/ton, it is likely that the cost will hover somewhere around \$30 USD/ton in the future.

The **effective initiation delay** is included because while at present time there are few enforced CO₂ emissions regimes, eventually CO₂ emissions will have a financial cost associated with them. This delay allows designers to estimate the lifetime CO₂ emissions costs starting at a point in the future that is defined by the effective initiation delay. For example, if the designer anticipates that 5 years from now, the CO₂ emissions from the HVAC system will be taxed, the designer can enter "5" into this box. When the program performs the Lifetime NPV costs of the system, it will begin including CO₂ emissions costs starting at year five. The effective initiation delay enables designers to maximize the accuracy of the program's calculations.

Average Building Costs

When a designer is considering the financial costs of one HVAC system versus another, it is important to remember and include overall building costs. A well designed, decentralized geothermal system may have no

need for a mechanical room while a central chiller/boiler plant may require a thousand or more square feet of space. This additional space costs money to construct. In addition, the space used for a central plant has an opportunity cost- its lost rental or lease value. For these reasons, the finance module enables designers to ascertain a) the building construction cost reductions (if any) and b) the revenue generated from the additional available square footage (if any) of a geothermal system compared to a more traditional HVAC solution.

The **total structure floor space** enables designers to enter the total square footage (floor space) of the to-be-conditioned space. Please be sure not to enter the square footage of any unconditioned floor space.

The **average building construction cost** enables designers to enter the per square foot construction costs of the building.

The **lease value** is the market lease value of the floor space in \$ per square foot per year terms.

Equipment Related Costs

Equipment related costs are a key ingredient in estimating overall system installation and maintenance costs as well as salvage values (if any). Typically, designers and engineers specify these costs on a per square foot (of the total conditioned space square footage) basis. The finance module can take these values into account in conjunction with the average building costs when calculating the annual and NPV lifetime costs associated with different HVAC systems.

The **system type** dropdown menu enables designers to select one of eight different types of systems including geothermal heat pumps, boilers, furnaces, air-source heat pumps, gas-fired heat pumps, air-cooled chillers, water-cooled chillers and unitary air conditioners.

The **fuel type** dropdown menu enables designers to select one of several different fuel types for the system type that the designer selected above.

Each of these system type-fuel type combinations has three optional cost parameters associated with it. These parameters include **installation costs** (per square foot), **maintenance costs** (per square foot per year) and **salvage value** (per square foot), all on a per square foot basis.

Experienced HVAC engineers oftentimes have a good rule-of-thumb estimate for the per square foot **installation costs** (including capital equipment) for a variety of HVAC systems. For geothermal systems, costs vary greatly depending on the geology, drilling conditions, type of heat

exchanger utilized, etc. This makes it a bit challenging to have a “rule of thumb” for geothermal installation costs. That being said, some research has been published comparing commercial geothermal system installations costs to those of more standard systems. In particular, data collected and analyzed by Bloomquist suggest the following for vertical closed loop commercial systems:

Average installation cost	Maximum installation cost	Minimum installation cost
\$100/m ² (\$9.3/ft ²)	\$135/ m ² (\$12.5/ft ²)	\$36/ m ² (\$3.3/ft ²)

Bloomquist further suggests that horizontal, closed loop systems have installation costs that are less than 50% of the cost of vertical, closed loop systems.

Below is a table adapted from Bloomquist that indicates average installation/capital costs for more standard HVAC systems:

HVAC System Type	Installation/Capital Costs
Rooftop DX with electric heating	\$52/ m ² (\$4.8/ft ²)
Rooftop DX with gas heating	\$61/ m ² (\$5.7/ft ²)
Air-source heat pump	\$74/ m ² (\$6.9/ft ²)
Rooftop variable air volume (VAV)	\$86/ m ² (\$8.0/ft ²)
Water-source heat pump with gas boiler/cooling tower	\$133/ m ² (\$12.4/ft ²)
Central VAV with chiller, cooling tower and gas perimeter heat	\$162/ m ² (\$15.0/ft ²)
Four-pipe fan coil unit with electric chiller and gas boiler	\$171/ m ² (\$15.9/ft ²)

The above data are included in this manual as a convenience and general reference for Ground Loop Design software users. It is of course the responsibility of the designer to determine the exact installation cost parameters for use in the financial module.

After having selected a system type and an appropriate fuel type for the system, the user can then enter the per square foot installation cost value. If for example a user wishes to have the program perform comparisons involving both natural gas and fuel oil boilers, the user must be sure to enter the installation cost data for both types of boilers.

Please note that the finance module breaks the conventional system analysis into separate heating and cooling systems analyses. If a user wants the program to estimate installation costs for a roof DX/gas boiler system, the user must first select the unitary air conditioner, choose a fuel type and then enter the installation costs associated with the DX system. The user must then select a boiler, choose the appropriate fuel type and

then enter the installation costs associated with the gas boiler. This system, while slightly more labor intensive for the user, provides for the highest degree of analysis flexibility.

Experienced HVAC engineers also oftentimes have a good rule-of-thumb estimate for the per square foot per year **maintenance costs** for a variety of HVAC systems. Once again, some research has been published comparing commercial geothermal system maintenance costs to those of more standard systems. Because younger systems have lower maintenance costs than older systems, maintenance costs increase over time. Data collected and analyzed in several studies by Bloomquist, Cane, et al., Hughes, et al. and Dohrmann and Alereza suggest the following range of maintenance costs for geothermal and conventional HVAC systems:

Below is a table on maintenance costs adapted from Hughes, et al.

HVAC System Type	Maintenance Costs
Air cooled chiller/gas fired water boiler	\$0.94/ m ² /yr (\$0.088/ft ² /yr)
Geothermal system	\$0.99/ m ² /yr (\$0.093/ft ² /yr)
Water cooled chiller/gas fired steam boiler	\$1.45/ m ² /yr (\$0.135/ft ² /yr)
Water cooled chiller/gas fired water boiler	\$2.01/ m ² /yr (\$0.187/ft ² /yr)

Below is a table on maintenance costs adapted from Cane, et al.:

System Type	Average Age	Mean Maint Costs in 1997 dollars
Geothermal system	5	\$1m ² /yr (\$0.093/ft ² /yr)
Water source heat pump	18	\$3.3m ² /yr (\$0.31/ft ² /yr)
Packaged air-to-air	2	\$5m ² /yr (\$0.47/ft ² /yr)
Split air-to-air	24	\$4m ² /yr (\$0.37/ft ² /yr)
Reciprocating chiller	2	\$4.40m ² /yr (\$0.4/ft ² /yr)
Centrifugal chiller	20	\$5.5m ² /yr (\$0.52/ft ² /yr)
Absorption chiller	29	\$8m ² /yr (\$0.75/ft ² /yr)

Below is a third table with data based off of an analysis conducted by Dohrmann and Alereza:

System Type	Age of System (years)				
	0	2	5	10	20
Geothermal system	\$2.2m ² /yr (\$0.208/ft ² /yr)	\$2.3m ² /yr (\$0.215/ft ² /yr)	\$2.4m ² /yr (\$0.226/ft ² /yr)	\$2.6m ² /yr (\$0.243/ft ² /yr)	\$2.96m ² /yr (\$0.277/ft ² /yr)
WLHP	\$3.84m ² /yr (\$0.36/ft ² /yr)	\$3.92m ² /yr (\$0.367/ft ² /yr)	\$4.03m ² /yr (\$0.378/ft ² /yr)	\$4.21m ² /yr (\$0.395/ft ² /yr)	\$4.58m ² /yr (\$0.429/ft ² /yr)
DX-cooling/Electric heating	\$4.08m ² /yr (\$0.382/ft ² /yr)	\$4.14m ² /yr (\$0.388/ft ² /yr)	\$4.26m ² /yr (\$0.399/ft ² /yr)	\$4.38m ² /yr (\$0.416/ft ² /yr)	\$4.8m ² /yr (\$0.45/ft ² /yr)
2-pipe fan coil w/ boiler and chiller	\$5.58m ² /yr (\$0.523/ft ² /yr)	\$5.66m ² /yr (\$0.530/ft ² /yr)	\$5.77m ² /yr (\$0.541/ft ² /yr)	\$5.95m ² /yr (\$0.558/ft ² /yr)	\$6.3m ² /yr (\$0.592/ft ² /yr)
VAV w/ boiler and chiller	\$6.77m ² /yr (\$0.634/ft ² /yr)	\$6.84m ² /yr (\$0.641/ft ² /yr)	\$6.95m ² /yr (\$0.651/ft ² /yr)	\$7.13m ² /yr (\$0.668/ft ² /yr)	\$7.5m ² /yr (\$0.703/ft ² /yr)
4-pipe fan coil with boiler and chiller	\$7.32m ² /yr (\$0.686/ft ² /yr)	\$7.4m ² /yr (\$0.693/ft ² /yr)	\$7.5m ² /yr (\$0.703/ft ² /yr)	\$7.7m ² /yr (\$0.720/ft ² /yr)	\$8.1m ² /yr (\$0.755/ft ² /yr)

The above data are included in this manual as a convenience and general reference for Ground Loop Design software users. It is of course the responsibility of the designer to determine the exact maintenance cost parameters for use in the financial module.

After having selected a system type and an appropriate fuel type for the system, the user can then enter the per square foot per year maintenance cost value. If for example a user wishes to have the program perform comparisons involving both natural gas and fuel oil boilers, the user must be sure to enter the maintenance cost data for both types of boilers.

Please note that the finance module breaks the conventional system analysis into separate heating and cooling systems analysis. If a user wants the program to estimate maintenance costs for a roof DX/gas boiler system, the user must first select the unitary air conditioner, choose a fuel type and then enter the maintenance costs associated with the DX system. The user must then select a boiler, choose the appropriate fuel type and then enter the maintenance costs associated with the gas boiler. This system, while slightly more labor intensive for the user, provides for the highest degree of analysis flexibility.

Experienced HVAC engineers also may have a good rule-of-thumb estimate for the per square foot **salvage value** for a variety of HVAC systems. Users can enter per square foot salvage values for the different systems following the methods outlined above.



If a designer is doing a financial comparison and not getting any output results for installation costs, maintenance costs or salvage values, it is worthwhile confirming that the baseline cost data have been entered. If the data have not been entered, results can not be calculated.

Utility Costs

Input parameters relating to utility costs are located in the *Utility Costs* panel, as shown in figure 9.4. These include summer and winter utility costs for a range of fuel types, the expected annual inflation rates for each fuel type and an overall discount rate that is used in the NPV calculations.

The *Utility Costs* panel is divided into two sections: rates for common fuels and annual inflation rates.

Finance Module - HorizontalSample

Results | Geothermal | Conventional | Utilities | Other Costs | Incentives

Rates for Common Fuels

Energy Source	SUMMER	WINTER
Electricity:	0.10 \$ / kWh	0.10 \$ / kWh
Fuel Oil:	4.00 \$ / Gallon	4.00 \$ / Gallon
Natural Gas:	0.012 \$ / ft ³	0.012 \$ / ft ³
Propane:	3.50 \$ / Gallon	3.50 \$ / Gallon
Wood:	300.00 \$ / ton	300.00 \$ / ton
Coal:	40.00 \$ / ton	40.00 \$ / ton
Biomass:	300.00 \$ / ton	300.00 \$ / ton
Water:	0.0015 \$ / Gallon	0.0015 \$ / Gallon

Annual Inflation Rates

Fuel Inflation Rate: Electricity 3.0 %

Discount Rate: 5.0 %

Fig. 9.4 *Utility Costs* Panel Contents

Rates for Common Fuels

The rates for common fuels can be entered in the 'Rates for Common Fuels' section. These fuels include electricity, fuel oil, natural gas, propane, wood, coal, biomass (excluding wood and wood pellets) and water. Although water is not a fuel, it is consumed in some HVAC systems (such as cooling towers) and therefore is included here for financial modeling purposes.



Note that it is essential that users enter rates for both summer and winter rates, even if they are identical. Failure to do so will result in underestimated cost results.

Annual Inflation Rates

In this section, users can enter the expected **fuel inflation rates** for each fuel type as well as the overall discount rate. Because inflation rates vary depending on fuel type, users can enter the inflation rate for each fuel type for the highest levels of modeling accuracy.

To enter the fuel inflation rate for each fuel type, the user first selects a fuel type from the dropdown menu and then enters the appropriate inflation rate. The user can then repeat the process for the other fuel types. Note that if the user selects a HVAC system but does not include a fuel inflation rate appropriate for that system, then an accurate NPV analysis cannot be performed. For example, if the user does not enter the inflation rate for fuel oil, yet selects a fuel oil boiler as part of a geothermal hybrid system, then the program will not be able to take into account the fuel oil inflation rate when performing the NPV calculations. This could lead to an inaccurate final cost analysis.

Enter the **discount rate** in the other text box. This discount rate is used in the overall NPV calculations.

Conventional

The finance module enables designers to compare the costs of a geothermal system with up to five different conventional systems. These conventional systems can include any combination of heating (boiler, furnace, air source heat pump, water source heat pump) and cooling (air cooled chiller, water cooled chiller, unitary air conditioner) equipment. After the users first selects a conventional system number (1-5) and then defines the equipment type and performance characteristics for that system number, the program determines the energy requirements and operating costs for the system. The parameters relating to the conventional system options are located in the *Conventional* panel as shown in figure 9.5.

The *Conventional* panel is broken up into two sections: alternate systems and system details.

Finance Module - HorizontalSample

Results | Geothermal | **Conventional** | Utilities | Other Costs | Incentives

Alternate Systems

System: 1 ◀ ▶

	COOLING	HEATING	TOTAL
Total Annual Power:	5198.1 kWh	4023.0 kWh	9221.1 kWh
Water:	0.0 Gallons	0.0 Gallons	0.0 Gallons
Other:	None	2438773.0 ft ³ Natural Gas	

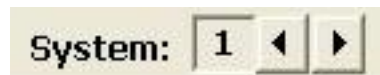
System Details

	COOLING	HEATING
Eqv Full-Load Hours:	255 hr	4023 hr
Equipment Type:	Unitary Air Conditioner	Boiler
Power Source:	Electricity	Natural Gas
Installed Capacity:	252.0 kBtu/hr	500.0 kBtu/hr
Efficiency:	13.0 EER	80.0 %
Extra Power:	1.0 kW	1.0 kW
Installation Area:	300.0 ft ²	300.0 ft ²
Water Usage Rate:	0.00 gpm/ton	0.00 gpm/ton

Fig. 9.5 *Conventional* Panel Contents

Alternate Systems

In this section, users can scroll through the alternate (standard HVAC) systems and see the summary of each system's energy and fuel/fuel type consumption. Users can scroll through and review each system by clicking on the left and right arrows as seen below:



Please note that if a system has not been defined (see below), power and fuel type information can not be displayed.

In general, a user will select a system (system 1, for example) and then proceed to define the system in the system details section. After defining system one, the user can choose to define another system by selecting system two and then entering the relevant information for it. The user can repeat the procedure for up to five alternate systems.

System Details

In this section, the user can enter details about the system he or she selected (system 1-5) in Alternate Systems, above. Please note that some of the details are locked out. For example, the equivalent full-load hours values cannot be changed by the user. Full load hours are entered in the *Geothermal* tabbed panel and transferred automatically into the conventional panel. This is to ensure that the comparison between the geothermal system and the conventional systems is based off of an equal number of full load hours.

Cooling

In this column, the user can enter details about the alternate cooling system(s).

Equivalent Full Load Hours

These values are transferred from the *Geothermal* panel and can not be changed by the user. If the user wishes to change these values, he or she must do so from the *Geothermal* tabbed panel.

Equipment Type

The user can select from among three cooling systems: air-cooled chillers, water-cooled chillers and unitary air conditioners. Please note that since these systems have different efficiency rating systems, the efficiency units change depending on the system selected.

Power Source

Users can select from among electricity, natural gas and propane.

Installed Capacity

Here users can enter the cooling system's *installed* capacity. Note that in general, the installed capacity for conventional systems exceeds the peak capacity of geothermal systems. This is because conventional mechanical equipment is usually significantly oversized compared to the equipment in a well designed geothermal system.

Efficiency

Here users enter the expected overall system efficiency for the selected cooling equipment. Note that the measurement units vary depending on

the selected system (i.e. kW/ton for water-cooled chillers and EER for unitary air-conditioners).

Extra Power

Here users enter extra power requirements for the system (such as circulation pumps, etc).

Installation Area

In this section, users enter the floor space square footage required by the selected cooling equipment. For example, if a water-cooled chiller is selected and it requires 1000 ft² of mechanical room space, the user can enter 1000 ft² here.

Water Usage Rate

If the selected cooling equipment consumes water, the user can enter the water usage rate here.

Heating

In this column the user can enter details regarding the alternate heating system(s).

Equivalent Full Load Hours

These values are transferred from the *Geothermal* panel and can not be changed by the user. If the user wishes to change these values, he or she must do so from the *Geothermal* tabbed panel.

Equipment Type

The user can select from among four heating systems: boilers, furnaces, air-source heat pumps and gas-fired heat pumps. Please note that since these systems have different efficiency rating systems, the efficiency units change depending on the system selected.

Power Source

Users can select from among electricity, fuel oil, natural gas, propane, wood, coal or biomass (biomass excluding wood).

Installed Capacity

Here users can enter the heating system's *installed* capacity. Note that in general, the installed capacity for conventional systems exceeds the peak capacity of geothermal systems. This is because conventional mechanical equipment is usually significantly oversized compared to the equipment in a well designed geothermal system.

Efficiency

Here users enter the expected overall system efficiency for the selected heating equipment. Note that the measurement units vary depending on the selected system (i.e. % efficiency for boilers and COPs for air-source heat pumps).

Extra Power

Here users enter extra power requirements for the system (such as circulation pumps, etc).

Installation Area

In this section, users enter the floor space square footage required by the selected heating equipment. For example, if a central boiler system is selected and it requires 1000 ft² of mechanical room space, the user can enter 1000 ft² here.

Water Usage Rate

If the selected heating equipment consumes water, the user can enter the water usage rate here.

Geothermal

In this section, users enter parameters and values pertaining to the geothermal system. As mentioned previously, users have the option of importing relevant data for the financial analysis from an open heat exchanger design project. Conversely, users can manually enter the geothermal project data directly into the financial module. An overview of the *Geothermal* panel is shown in figure 9.6.

The *Geothermal* panel is divided into two sections: a summary panel and two details tabbed sub-panels.

Finance Module - HorizontalSample

Results | **Geothermal** | Conventional | Utilities | Other Costs | Incentives

Geothermal System

20 years ☐ Import ☒ Manual

	COOLING	HEATING	TOTAL
Geothermal Power:	2608.4 kWh	120890.5 kWh	123499.0 kWh
Hybrid Power:	0.0 kWh	0.0 kWh	0.0 kWh
Total Annual Power:	2608.4 kWh	120890.5 kWh	123499.0 kWh
Water:	0.0 Gallons	0.0 Gallons	0.0 Gallons
Other:	None	None	

Primary Geothermal | Hybrid Component

	COOLING	HEATING
Eqv Full-Load Hours:	255 hr	4023 hr
Peak Capacity:	197.7 kBtu/hr	373.5 kBtu/hr
Average Heat Pump Efficiency:	22.0 EER	3.8 COP
Circulation Pump Input Power:	1.2 kW	1.2 kW
Pump Power:	1.5 hP	1.5 hP
Motor Efficiency:	90 %	90 %
Additional Power:	0.0 kW	0.0 kW
Installation Area:	300.0 ft ²	

Fig. 9.6 Geothermal Panel Contents

Geothermal Project Power Summary Panel

The top third of the *Geothermal* panel displays several features of the geothermal system including the modeling time period, the energy usage and fuel type for the geothermal system. This can be seen below in figure 9.7.

Geothermal System

20.0 years ☐ Import ☒ Manual

	COOLING	HEATING	TOTAL
Geothermal Power:	2608.4 kWh	120890.5 kWh	123499.0 kWh
Hybrid Power:	0.0 kWh	0.0 kWh	0.0 kWh
Total Annual Power:	2608.4 kWh	120890.5 kWh	123499.0 kWh
Water:	0.0 Gallons	0.0 Gallons	0.0 Gallons
Other:	None	None	

Fig. 9.7 Geothermal Project Power Summary Panel

The modeling time period

The **modeling time period** is necessary for calculating the NPV lifetime costs of the design.

When a user imports a design project into the finance module, the modeling time period automatically is set to match the modeling time period use in the heat exchanger design. In such a case, the modeling time period is grayed out, indicating that the value was imported. If a user wishes to override this value, he or she can do so by first selecting the “Manual” option and then entering the time period of interest.

Geothermal Project Power Summary

Below the modeling time period, the user can see the program’s annual energy usage estimate for the geothermal system. Results are listed in ‘heating’, ‘cooling’ and ‘total’ columns for ease of review.

Energy usage is divided up into the following constituent parts:

- geothermal power (the power consumed by the geothermal system)
- hybrid power (the power consumed by the hybrid system, if any)
- total annual power (a summation of geothermal and hybrid power)
- water (the amount of water, if any, consumed by the system on an annual basis)
- other (other fuel sources (natural gas, fuel oil, etc) and the annual amount consumed).

Calculated energy usage values are updated as soon as a change is made in system parameters (see below).

Geothermal System Details

The bottom two-thirds of the *Geothermal* panel displays system details related to the geothermal and hybrid system. Details of the geothermal and hybrid system (if any) can be found on separate tabbed panels. The geothermal system tabbed panel can be seen below in figure 9.8 and the hybrid component tabbed panel can be seen in figure 9.9.

Primary Geothermal		Hybrid Component	
COOLING		HEATING	
Eqv Full-Load Hours:	255 hr	4023 hr	
Peak Capacity:	197.7 kBtu/hr	373.5 kBtu/hr	
Average Heat Pump Efficiency:	22.0 EER	3.8 COP	
Circulation Pump Input Power:	1.2 kW	1.2 kW	
Pump Power:	1.5 hP	1.5 hP	
Motor Efficiency:	90.0 %	90.0 %	
Additional Power:	0.0 kW	0.0 kW	
Installation Area:	300.0 ft ²		

Fig. 9.8 Geothermal System Tabbed Panel

Primary Geothermal		Hybrid Component	
COOLING		HEATING	
Eqv Full-Load Hours:	255 hr	4023 hr	
Hybrid Type:	Cooling Tower	Boiler	
Fuel Type:	Electricity	Electricity	
Hybrid System Capacity:	0.0 kBtu/hr	0.0 kBtu/hr	
Hybrid Unit Efficiency:	0.0 %	0.0 %	
Additional Power:	0.0 kW	0.0 kW	
Installation Area:	0.0 ft ²	0.0 ft ²	
Water Usage Rate:	0.00 gpm/ton	0.00 gpm/ton	

Fig. 9.9 Geothermal System Tabbed Panel

Primary Geothermal Tab

Cooling

In this column, the user can enter details about the geothermal cooling system (s).

Equivalent Full Load Hours

The user can enter the equivalent full load hours here if the user has not imported the data automatically from a heat exchanger project design.

Peak Capacity

The user can enter the peak capacity (*note that this is the peak load covered by the equipment and not the installed equipment capacity*) here if the user has not imported the data automatically from a heat exchanger project design.

Average Heat Pump Efficiency

Here the user enters the expected EER for the cooling side of the system if the user has not imported the data automatically from a heat exchanger project design.

Note that if the user has imported the data from a vertical heat exchanger project that has monthly data calculated (see chapter 4), then the imported EER is the average EER over the system lifetime and not the peak conditions EER. Generally, using the monthly data provides for a higher EER (and lower costs) since average fluid temperatures tend to be less extreme than the fluid temperatures during peak load conditions.

Circulation Pump Input Power, Pump Power and Motor Efficiency

The circulation pump input power automatically is calculated from the pump power and motor efficiency. These values can be imported from a heat exchanger design project or manually entered.

Additional Power

The user can enter power for all other elements (besides the heat pump units) in the system that may require energy input. Again, these data can be imported from a heat exchanger project (if the data are in the project) or can be entered manually.

Geothermal Heating

In this section, the user can enter details about the geothermal heating system (s).

Equivalent Full Load Hours

The user can enter the equivalent full load hours here if the user has not imported the data automatically from a heat exchanger project design.

Peak Capacity

The user can enter the peak capacity (*note that this is the peak load covered by the equipment and not the installed equipment capacity*) here if the user has not imported the data automatically from a heat exchanger project design.

Average Heat Pump Efficiency

Here the user enters the expected COP for the heating side of the system if the user has not imported the data automatically from a heat exchanger project design.



Note that if the user has imported the data from a vertical heat exchanger project that has monthly data calculated (see chapter 4), then the imported COP is the average COP over the system lifetime and not the peak conditions COP. Generally, using the monthly data provides for a higher COP (and lower operating costs) since average fluid temperatures tend to be less extreme than the fluid temperatures during peak load conditions.

Circulation Pump Input Power, Pump Power and Motor Efficiency

The circulation pump input power automatically is calculated from the pump power and motor efficiency. These values can be imported from a heat exchanger design project or manually entered.

Additional Power

The user can enter power for all other elements in the system (besides the heat pump units) that may require energy input. For example, heat recovery units require additional energy that can be recorded in this box so that it can be used in the overall calculation of the System *COP*. Again, these data can be imported from a heat exchanger project (if the data are in the project) or can be entered manually.

Installation Area

In this section, the user enters the floor space square footage required by the geothermal mechanical equipment. For example, if a school geothermal system is decentralized and the heat pumps are located in the ceilings above the classrooms, the user might leave this value at “0”. Conversely, if the geothermal equipment is located in a small closet in each classroom, the designer could multiply the square footage of each closet by the number of closets in the school to calculate a cumulative value for entry in this text box.

Hybrid Component Tab

Cooling

In this column, the user can enter details about the hybrid component of the geothermal cooling system (s).

Equivalent Full Load Hours

The user can enter the equivalent full load hours here if the user has not imported the data automatically from a heat exchanger project design. Note that by default, the equivalent full load hours value in the hybrid panel matches the full load hours in the *geothermal* panel. If the user changes the value in the *geothermal* tabbed panel, the value in the hybrid component panel changes as well. The user does have the option though of changing this value in the hybrid tab so that it does not match the value in the *geothermal* tabbed panel.

Hybrid Type

At present time, the user has the option of selecting a cooling tower.

Fuel Type

Electricity is the only option for the cooling tower at this time.

Hybrid System Capacity

Here the user can enter the installed capacity of the hybrid system. This value automatically is entered when the user imports a heat exchanger design project (that has a hybrid component) into the finance module.

Hybrid Unit Efficiency

This value is not applicable to the cooling tower selection and is grayed out.

Additional Power

Here the user enters extra power requirements for the system (such as fans, circulation pumps, etc).

Installation Area

In this section, the user enters the floor space square footage required by the selected cooling equipment. For example, if a cooling tower requires

400 ft² of rooftop space, the user can enter 400 ft² here. Of course, if the rooftop space has no commercial value *per se*, it would be reasonable to decrease the input square footage value.

Water Usage Rate

The user can enter the water usage rate for the cooling tower here. 0.3 gpm/ton is a reasonable starting point for many systems.

Heating

In this section, the user can enter details about the hybrid component of the geothermal heating system (s).

Equivalent Full Load Hours

The user can enter the equivalent full load hours here if the user has not imported the data automatically from a heat exchanger project design. Note that by default, the equivalent full load hours value in the hybrid panel matches the full load hours in the *geothermal* panel. If the user changes the value in the *geothermal* tabbed panel, the value in the hybrid component panel changes as well. The user does have the option though of changing this value in the hybrid tab so that it does not match the value in the *geothermal* tabbed panel.

Hybrid Type

At present time, the user has the option of selecting a boiler.

Fuel Type

The user can select from among seven fuel options.

Hybrid System Capacity

Here users can enter the installed capacity of the hybrid system. This value automatically is entered when the user imports a heat exchanger design project (that is a hybrid system design) into the finance module.

Hybrid Unit Efficiency

The user can enter the boiler's thermal efficiency here.

Additional Power

Here the user enters extra power requirements for the system (such as fans, circulation pumps, etc).

Installation Area

In this section, the user enters the floor space square footage required by the selected heating equipment. For example, if the boiler requires 600 ft² of floorspace, the user can enter 600 ft² here.

Water Usage Rate

The user can enter the water usage rate, if any, for the boiler.

Results

All of the cost/emissions results for both the geothermal and alternate systems can be viewed at any time on the *Results* panel. After all data have been entered or any changes have been made, the user can calculate interim or final results using the 'Calculate' button. A sample screen for this panel can be seen in figure 9.10.

The Calculate panel is divided into two sections. On the top is the Annual Cost section. On the bottom is the Net Present Value (NPV) Lifecycle Costs section.

In each of the two sections, results are presented in two columns: the first is for the geothermal system and the second is for the alternate system(s). When more than one alternate system has been defined, users can scroll through the different alternate systems using the arrows.

Note that the presented costs are the summations of heating, cooling and hybrid system costs. Data are broken down into their constituent parts and displayed in the reports (see below).

The screenshot shows the 'Finance Module' window with the 'Geothermal' tab selected. The 'Estimated Cost Results' section displays a comparison between the 'Geothermal' system and an 'Alternate' system (labeled 'Air-cooled Chiller' and 'Boiler'). The 'Calculate' button is visible. The results are presented in two tables: 'Annual Costs (\$)' and 'NPV Lifecycle Costs (\$) - 10 years'.

Estimated Cost Results		
	Geothermal	Alternate 1 Air-cooled Chiller Boiler
Annual Costs (\$)		
Energy	10,297.83	19,676.14
CO2 Emissions	1,255.41	1,698.51
Water	554.63	0.00
Maintenance	3,000.00	5,000.00
Mechanical Room Lease	2,050.00	3,077.05
Annual Total	17,157.87	29,451.70
NPV Lifecycle Costs (\$) - 10 years		
Energy	109,523.55	219,211.77
CO2 Emissions	8,452.39	11,435.60
Water	5,546.25	0.00
Maintenance	26,198.23	43,663.72
Mechanical Room Lease	17,902.13	26,871.09
Installation	100,800.00	93,032.00
Salvage	(2,107.76)	(913.36)
Lifecycle Total	277,514.79	393,300.83

Fig. 9.10 Geothermal System Tabbed Panel

Annual Costs

The annual costs section presents costs associated with running the geothermal system and alternate systems over a single year. Costs include the energy costs, CO₂ emissions costs, water usage costs, maintenance costs and mechanical room lease opportunity costs. Costs are reported as “0.00” if one or more of the required (and user-defined) variables used in the calculations have not been set. For example, if the user has selected a natural gas boiler as an alternate heating system but has not specified the maintenance costs for such a system, then the maintenance costs will be reported as “0.00”. Upon seeing the “0.00” the user can go back to the *Other Costs* panel, input the maintenance costs, return to the *Results* page and then hit ‘Calculate’ again to recalculate the results.

NPV Lifecycle Costs

The NPV Lifecycle Costs section presents costs associated with running the geothermal system and alternate system over the time frame specified on the *Geothermal* tab. Costs include lifetime energy costs, lifetime CO₂ emissions costs,

TIP

lifetime water usage costs, lifetime maintenance costs, lifetime mechanical room lease opportunity costs, installation costs and salvage value (if any) at the end of the building lifecycle. All of these lifecycle costs are calculated using the inflation and discount rates that the user has specified on the *Utilities* panel. Once again, the report will output a value of “0.00” if the user has not input the pricing parameters necessary for performing the calculation. Also please note that if any incentives have been entered into the *Incentives* tabbed panel, these values are subtracted from the overall installation costs and the net result is displayed.

Printing Reports

Financial reports can be printed at any time using the toolbar print button in the finance module.

A total of five reports, including two finance reports, two inputs reports and one financial summary report are available. The concise finance report has information related to geothermal financials and energy usage. The detailed finance report has information related to geothermal and conventional system financials and energy usage. The concise inputs report has a truncated list of all the data inputs used in the financial calculations. The detailed inputs report has a full list of the data inputs used in the financial calculations.

More information on reports can be found in Chapter 7.

References

Bloomquist, R.G., 2001. The Economics of Geothermal Heat Pump Systems for Commercial and Institutional Buildings, *Proceedings of the International Course on Geothermal Heat Pumps*, Bad Urach, Germany.

Cane, D., et al. 1997. Survey and Analysis of Maintenance and Service Costs in Commercial Building Geothermal Systems. Caneta Research Inc., for the Geothermal Heat Pump Consortium, *RP-024*.

Chiasson, A. 2006. Final Report Life Cycle Cost Study of a Geothermal Heat Pump System BIA Office Bldg., Winnebago, NE, *Feasibility Studies and Life-Cycle Cost Analysis*, Oregon Institute of Technology.

Dohrmann, D.R. and Alereza, T. 1986. Analysis of Survey Data on HBVAC Maintenance Costs, ADM Associates, Inc. for *ASHRAE Transactions* 92(2A).

CHAPTER 10

The Thermal Conductivity Module

This chapter describes how to use the GLD thermal conductivity module, a module that enables designers to quickly analyze test data collected from thermal conductivity test units such as the GeoCube™. This module is an optional upgrade.

Overview

For commercial vertical and horizontal heat exchangers, an accurate assessment of soil thermal properties (soil temperature, conductivity and diffusivity) is essential. Even a small percentage error in thermal property estimates can lead to either excessive installation costs or system underperformance. Consequently, in many cases, *in situ* thermal conductivity tests are well worth the investment.

Until recently, two modalities have dominated conductivity testing. In the first, designers and engineers have outsourced conductivity testing and data analysis to third parties. In the second, they have conducted tests on rented equipment and then sent the data to third parties for analysis. These modalities are costly and time consuming. In addition, these modalities limit the control designers have over the test and the analysis. As this industry continues to grow, more and more companies desire to have in-house testing and analysis capabilities. In light of

this evolving situation, GLD now includes a comprehensive Thermal Conductivity analysis module.

The Ground Loop Design Thermal Conductivity module allows designers to import conductivity test data (in CSV format) collected by a thermal conductivity test unit data logger. Users can then quickly input the borehole depth and calculation interval (hours 12 to 40, for example) and then calculate the thermal conductivity and estimated diffusivity. The module includes a suite of automated data analysis tools that assess the raw conductivity data quality. The module also enables designers to optimize the modeling period via an auto-graphing overlay function. A professional report output is included as well. With the Thermal Conductivity module, conductivity analysis is accurate, easy and nearly instantaneous.



The Thermal Conductivity module in GLD Premier 2010 now provides Borehole Thermal Resistance (BTR) results from in-situ test data and offers an enhanced graph visualization module that accelerates the data analysis process.

General Features

To aid in the analysis process, the Thermal Conductivity module in Ground Loop Design consists of a set of panels, grouped by subject, through which the designer can enter and edit the input variables efficiently. For example, parameters related to the diffusivity estimation are listed on the *Diffusivity* panel while bore-specific information can be viewed on the *Bore* panel. The idea is that everything related to a single conductivity analysis is presented simultaneously and is easily accessible at any time. The tabbed panels can be seen in figure 10.1, below. In GLD Premier 2010, graphs appear in a separate and flexible module when data are imported into the Thermal Conductivity module.

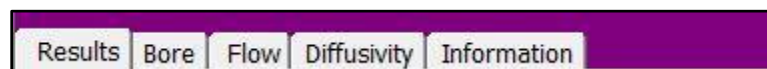


Fig. 10.1 Thermal Conductivity Panel List

The Thermal Conductivity module includes several additional features:

- Diffusivity estimator based on user-defined soil conditions
- Borehole Thermal Resistance (BTR) calculated results
- Flow rate test based on calibrated unit data and/or flow sensors
- Metric and English unit conversions
- Adjustable calculation intervals for conductivity analysis
- A range of color graphs in a stand-alone module

- Graph overlays for calculation interval optimization
- Automatic data quality analysis/error checks
- Printed report of all input and calculated data
- A 'Calculate' button used to refresh the calculations

Theoretical Basis

The Thermal Conductivity module uses the line source theory, the most commonly used theory, for the evaluation of conductivity test data. This analysis methodology requires a constant rate of heat injection (a stable power supply) and a thermal conductivity unit that is well insulated from the ambient air temperature.

Opening Projects

There are two ways to open Thermal Conductivity projects. One is by using the 'New Thermal Conductivity' command from the Design Studio *File* menu or toolbar, and the other is by opening an existing Thermal Conductivity project (*.gtc) file from within the Thermal Conductivity module. In the design studio, only one Thermal Conductivity module can be open at a time.



New Projects

New Thermal Conductivity projects may be opened at any time from the Design Studio by choosing 'New Thermal Conductivity' from either the Design Studio *File* menu or the toolbar. The module opens directly into the *Results* panel.



Existing Projects

Existing Thermal Conductivity projects may be opened at any time from within the Thermal Conductivity module by choosing 'Open' from the Thermal Conductivity Module toolbar.

Saving Projects

Thermal Conductivity projects may be saved at any time by clicking the save button on the Thermal Conductivity module toolbar. When the user closes the program or module, the program automatically asks the user if he or she wants to save the project.

Importing Conductivity Data

The Thermal Conductivity module can import CSV (comma separated value) files generated by thermal conductivity test unit data loggers. The new version of the Thermal Conductivity module provides a robust CSV file reading capability. If the module has trouble reading in a data set, it will provide the user with guidance and instructions.

At present time, the module is designed to read in CSV files that follow the format output by the GeoCube™, a test unit manufactured by Precision Geothermal, LLC. If a user wishes to import CSV files from another test unit, the user should make sure that the column format matches that of the GeoCube™ test unit. The basic format is as follows:

Format for Geocube Units with Flow Sensors:

Plot Title: Sample TC Test, Geothermal Town, USA							
#	Date Time, GMT-06:00	Voltage, VAC()	Current, A()	Temp, °F()	Temp, °F()	Counts, #()	Batt, V
1	1/6/2010 10:55	235.233	0.8822	54.169	55.168	16	12.64
2	1/6/2010 10:57	235.374	0.908	54.43	54.734	16	12.64

Format for Geocube Units with Pressure Transducers:

Plot Title: Sample TC Test, Geothermal Town, CA									
#	Time, GMT-06:00	Voltage, VAC()	Current, A()	Temp, °F()	Temp, °F()	Pressure, PSIG()	Pressure, PSIG()	Batt, V	
1	3/17/2009 16:02	211.526	0.9432	59.616	59.529	-0.009	12.404	12.1	
2	3/17/2009 16:04	211.386	0.9525	59.572	59.529	0.092	12.227	12.21	

Note that some conductivity units do not provide this much data (for example, many units do not record pressure). In such a case, make sure that the CSV file has the column title and then just populate the data rows with 0.0.

To import the CSV file, first save it in the following folder:

(Main Drive):\\Program Files\\GLD2010\\ThermalConductivity\\Thermal Conductivity Data Files

Next, in the Bore tab, enter the borehole length.

Next, click the following import button that is found on the module toolbar:



Finally, select the CSV file of interest.

Typical Operation

The typical operation of the Thermal Conductivity module would include the following steps:

1. Open a new Thermal Conductivity module
2. Choose metric or English units
3. Enter soil parameters in the Diffusivity Panel if desired
4. On the Bore Panel, enter the borehole depth
5. Import the in-situ data set
6. Review the graphs that appear in the Graphing Module
7. On the Results Panel hit the Calculate button
8. Modify calculation interval if necessary to maximize overlap of raw data and best fit lines in Temp vs. LN (Time) graph
9. Save and/or print the conductivity test report

Entering Data into the Tabbed Panels

Ground Loop Design's tabbed panel system provides for easy organization of and direct access to the relatively large number of design parameters associated with a particular project. This section describes the *Diffusivity*, *Flow*, *Bore* and *Results* panels.

Diffusivity

Information pertaining to diffusivity estimate calculations can be found in the *Diffusivity* panel as seen in figure 10.2, below. Users can enter estimates of soil specific heat, density and moisture content. By pressing the Check Soil Tables button, users can quickly access reference files that may contain some of these data. After the user calculates the thermal conductivity on the *Results* tab, the program automatically estimates the diffusivity based on the calculated conductivity and user-input soil values. The estimated diffusivity is displayed on this panel and the *Results* panel.

If the user wishes to manually enter an estimated diffusivity, the user can unselect the automatic estimator mode check box and then manually enter a diffusivity value.

Thermal Conductivity Calculation Project

Project File: None
Import Data File: American_Swedish_Institute_#2.csv

Results | Bore | Flow | **Diffusivity** | Information

Thermal Diffusivity Calculator

☒ Automatic Estimator Mode

Thermal Diffusivity: 0.91 ft²/day

Thermal Conductivity: 0.97 Btu/(h*ft*°F)

Soil/Rock Specific Heat - Dry: 0.200 Btu/(°F*lbm)

Soil/Rock Density - Dry: 100.0 lb/ft³

Moisture (0-20): 8 %

Check Soil Tables

Fig. 10.2 Diffusivity Panel Contents

Flow

Information pertaining to a particular conductivity test unit's flow-pressure coefficients can be selected, viewed and calculated in this tab. See figure 10.3 below.

Some conductivity test units have the capacity to collect pressure drop information data during a conductivity test. If a particular unit has this capacity and the flow-pressure relationship has been calibrated, then these calibration data can be entered (or in the case of preloaded data, viewed and selected) here. These data are useful because it enables GLD to monitor the raw test data for flow rate stability throughout the conductivity test.

If a user needs to enter calibration data for a conductivity test unit that is not already included in the module, the user will need to manually enter calibration data.

The calibration process does not have to be complex. To collect these data, a user can monitor the pressure drop for three flow rates (a flow rate can be determined with a stop watch and a bucket of defined volume). After these data are collected, the user can select 'Other' under the TC Unit Model Name, enter the pressure

drop/flow rate data, and hit the *Calculate Coefficients* button. Hit the save button to store the data for the current analysis.

Calibration data are stored in the following file:

**(Main Drive):\Program
Files\GLD2010\ThermalConductivity\ModelCoefficients.txt**

In GLD 2010, the Thermal Conductivity automatically determines flow rates for units that have flow meters rather than pressure sensors. For those units with flow meters, the flow calibration process does not apply.

Thermal Conductivity Calculation Project

Project File: None
Import Data File: None

Results Bore **Flow** Diffusivity Information

Flow Rate Information

TC Unit Model Name: GeoCube Standard

Flow-Pressure Coefficients

a: 12.632
b: -0.645
c: 0.013

Pressure Drop lb/in ²	Flow Rate gpm
1.00	5.0
2.00	10.0
3.00	15.0

Calculate Coefficients

Fig. 10.3 Flow Panel Contents

Bore

Input parameters related to the test borehole are located in the *Bore* panel, as shown in figure 10.4. While the bore length and ground temperature are integral to the calculation, all other parameters are for reference only and are included in the report.

Users must enter the appropriate bore length to ensure accurate results.

In "Automatic Estimator Mode", The program automatically estimates the undisturbed ground temperature from the in situ data set as it is imported. When "Automatic Estimator Mode" is not selected, the user must manually enter the undisturbed ground temperature. It is recommended that users manually determine the undisturbed ground temperature usual industry-accepted standards.

Note that the undisturbed ground temperature impacts the Borehole Thermal Resistance (BTR) calculations. Therefore for those designers wishing to use the BTR in a design, it becomes critical to have an accurate undisturbed ground temperature.

Fig. 10.4 Bore Panel Contents

Results

All of the results for the conductivity analysis can be viewed at any time on the *Results* panel and in the Graphing Module. After all data have been entered or any changes have been made, the user can calculate interim or final results using the 'Calculate' button. Each time the user hits the 'Calculate' button the graphs will be automatically updated. A sample screen for this panel and the Graphing Module can be seen in figures 10.5 and 10.6.

The Calculate panel is divided into three sections. On the top is the Calculation Interval input section. In the middle are the Calculation Results. At the bottom is the Data Quality section.

Immediately to the right of the Calculate button is a 'Save Calculated Graph Data' button. If the user checks this box before hitting the 'Calculate' button, then all of the raw data used in the graphs is exported as a text file into the **Thermal Conductivity/Thermal Conductivity Report Data Files** folder. The designer can import these data into Excel to create his or her own graphs if so desired.

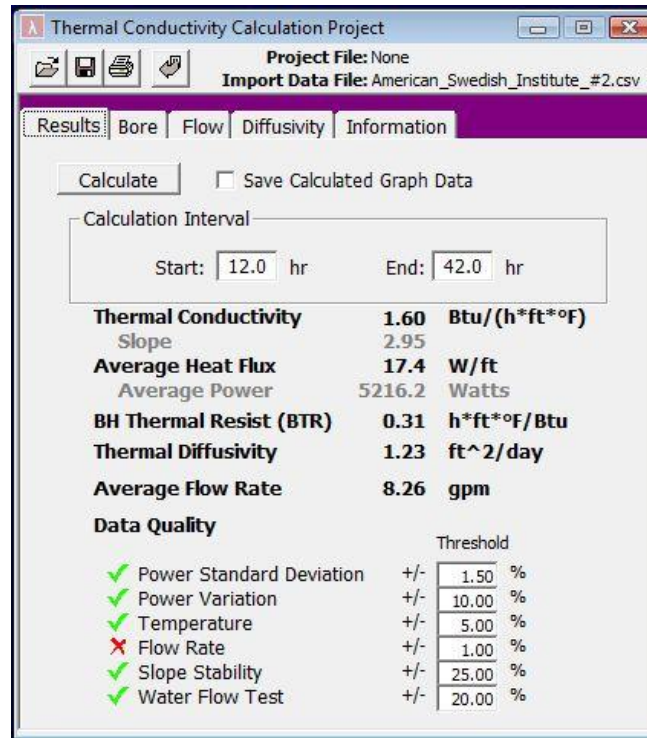


Fig. 10.6 Results Tabbed Panel

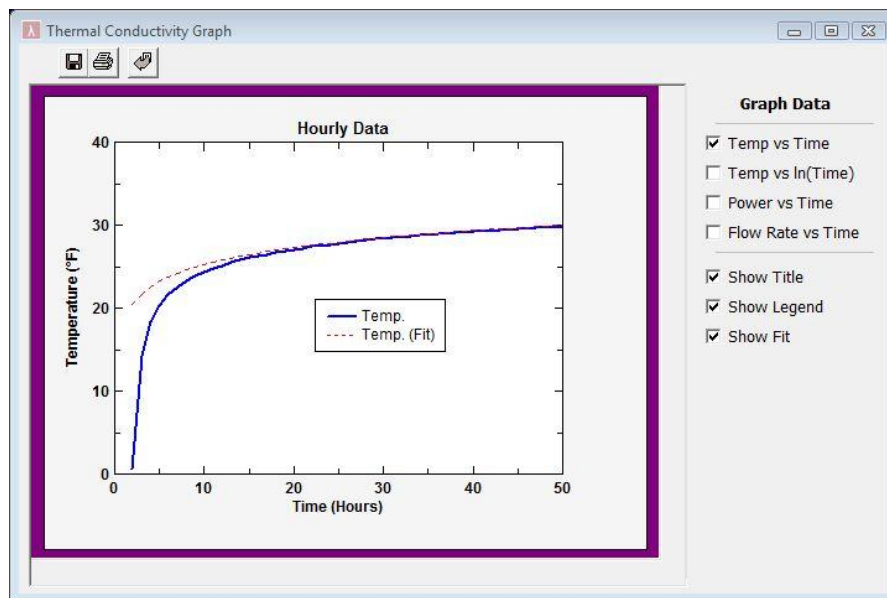


Fig. 10.7 Graphing Module Results

Calculation Interval

The calculation interval is a key factor in the data analysis. Typically, conductivity tests are run for approximately 48-72 hours and the 12 to 40 hour data range are used in calculations.

In this section, the user can input their desired interval range prior to calculating (or recalculating) the conductivity and estimating the diffusivity. See the graphs section below for more information on the relationship between this interval and the graphed data.

Calculation Results

The calculation results section displays the calculated thermal conductivity (and slope of the line), the average heat flux (and average power), the calculated borehole thermal resistance (BTR), the estimated thermal diffusivity (estimate based off of calculated conductivity and user-input soil values in the Diffusivity panel), and the average flow rate (note that if flow rate calibration data are not entered in the *Flow* tab, the flow rate result may not be applicable).

If users adjust the calculation interval and hit the Calculate button again, these results will be updated.

Note that the BTR calculation is extremely sensitive to the undisturbed ground temperature. Designers are encouraged to determine the undisturbed ground temperature with maximum accuracy prior to conducting the TC test and then manually enter the undisturbed ground temperature in the Bore tabbed panel. If the user does not enter the temperature manually, the module will automatically estimate the undisturbed ground temperature from the first two minutes of temperature data in the imported csv file.

Data Quality

The data quality section reports on whether or not certain aspects of the data meet user-defined thresholds. These analyses are useful for determining the overall reliability of the conductivity test data. A green check indicates that the entire data set remains within the threshold range. A red “x” indicates that at least one data point extends beyond the threshold range. Details for each test are described below:

Power Standard (Deviation)

The power standard deviation test checks the standard deviation of all points compared to the average value and sees if the deviation falls within the user-defined acceptable range. The default value is +/- 1.5%.

Power Variation

The power variation test checks to see if any point goes over a predefined limit, which is a percentage of the average value. The default is +/- 10%.

Temperature

The temperature test checks to see if the temperature decreases from its maximum measured value to a point below a defined threshold for more than 1% of the entire range. It assumes that the temperature should not decrease with constant heat input. The default is +/- 5%.

Flow Rate

The flow rate test checks the standard deviation of the flow rate. If the standard deviation exceeds the threshold as a percentage of the average, it fails. The default is +/- 5%.

Slope Stability

The slope stability test divides the total range of interest into 5 sections and then calculates the slope for each section, where each section is defined as the range from the starting point to each section's end point. With an ideal data set, all slopes would be the same. The slopes of each section are compared to the overall slope. If the slope varies by more than the user defined threshold, the test fails. The default value is +/- 25%.

Water Flow Test

The water flow test is similar to the above test but checks to see if consecutive slopes are decreasing, which would suggest water movement and an invalid thermal conductivity measurement. The default is +/- 10%.



Graphs

The Thermal Conductivity module automatically graphs conductivity test data after the CSV data file from a conductivity test is imported into the module. These graphs are displayed in a new stand-alone Graphing Module that enables designers to review graphs and calculated results simultaneously. Note that in the new Graphing Module, users can left click the mouse and drag a box around an area of interest in the graph. Users can then release the mouse button *to zoom in*

on the area of interest. This process can be repeated multiple times. Users can right click the mouse at any time to zoom out to the original view.

There are four types of graphs: Temperature vs. Time, Temperature vs. LN (time), Power vs. Time and Flow vs. Time. An overview image can be seen in figure 10.7.

The Temperature vs. Time graph and the Temperature vs. LN (time) graph are graphed according to the requirements of the Line Source analysis methodology. The Power vs. Time and Flow vs. Time graphs are included for test quality control purposes.

Upon initial CSV data file importation, only the raw data are graphed as seen below in figure 10.8. After the *Calculate* button is pushed in the Results tab, the data are analyzed and the calculated line is graphed as an overlay. This can be seen in figure 10.9.

The overlay/calculated line depends on the user-specified calculation interval (specified in the Results tab). Users can adjust this calculation interval and recalculate as necessary to bring the raw data and calculated lines as close together as possible. This is useful for determining the optimal calculation interval. For example, if a user finds that over the 12 to 40 hour time interval the two lines do not overlap closely, the user might view the power vs. time graph. If the power vs. time graph indicated a power supply instability between hours 35 and 38, the user could change the calculation interval to 12 to 34 hours, recalculate the line and then compare the raw data with the new line. If the newly calculated line better matches the raw data, then the user might reasonably use the calculated conductivity value for the 12 to 34 hour time interval rather than for the 12 to 40 hour time interval.

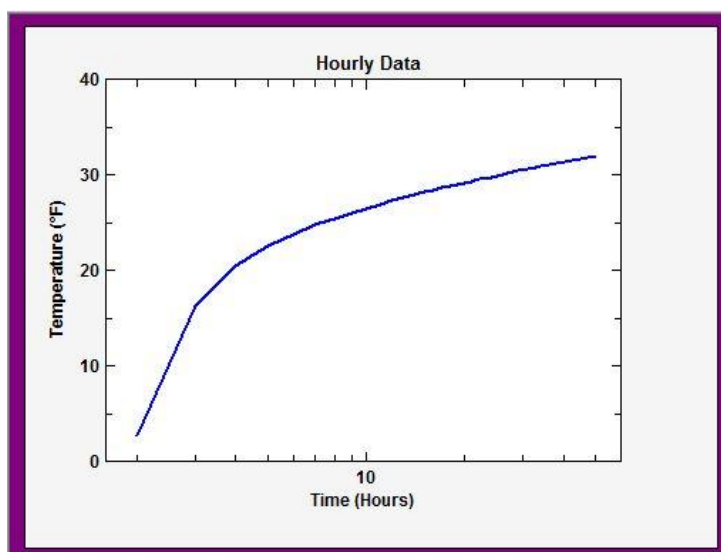


Fig. 10.8 Raw Test Data: Temp vs. LN (Time)

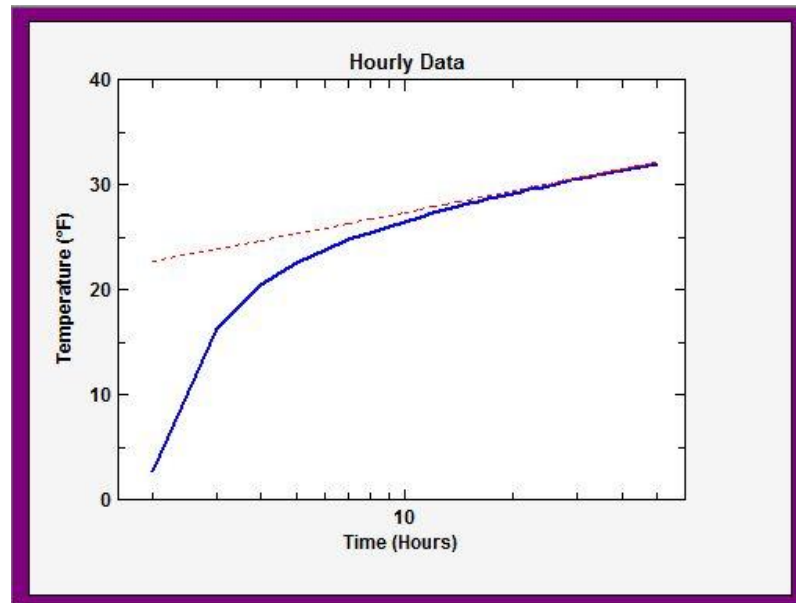


Fig. 10.9 Raw and Modeled Data: Temp vs. LN (Time)

Printing Reports

The Thermal Conductivity report can be printed at any time using the toolbar print button in the conductivity module.

More information on reports can be found in Chapter 7.

CHAPTER 11

The Computational Fluid Dynamics (CFD) Module

This chapter describes how to use the new Computational Fluid Dynamics (CFD) module, a module that answers the sometimes difficult question, "How should I set up my geothermal piping systems so that they maximize performance, minimize operational costs and are easily purged of air after installation and before start up?" This module is a powerful and optional upgrade.



Overview

Piping optimization is an essential and oftentimes overlooked component in competent geothermal loop design. When designed correctly, a piping system will be easy to purge and provide the flow characteristics essential for efficient heat transfer all while minimizing pumping and operational costs.

Up until now, piping optimization has been a time consuming, difficult and iterative process. The present state of the art for geothermal piping design is based on homegrown spreadsheets, rule of thumb estimates, and piping specification sheets. Indeed, a mid-sized commercial design could easily take an experienced designer a half day or more to try to optimize. Modeling reverse return systems with any accuracy is particularly difficult when using spreadsheets and hand calculations.

The new Computational Fluid Dynamics (CFD) module in GLD 2010 Premier changes all this. The CFD module provides designers and engineers with a level of design control and power that until now has not existed in our collective toolbox. This module specialized in designing the loopfield piping systems. The module can be used in conjunction with other piping software solutions that specialize in building internal piping systems.

The patent-pending CFD module utilizes a new and easy-to-learn visual "drag and drop" language for describing any possible loopfield configuration including direct and reverse return ground heat exchangers, supply and return runouts, manifolds, vaults, the fittings that connect everything together and circulation pumps. The user has the ability to define each and every part of the entire design down to the smallest detail if he or she so desires. Doing so of course could take some time. This is why the CFD module also includes a suite of intelligent algorithms that optimize piping design automatically and nearly instantly (fittings in the 2010 version require manual selection).

These algorithms efficiently calculate for example the proper supply and return header piping reductions to ensure that user defined purging flow rates are maintained throughout any piping design. They also make it very easy to have a flow-balanced system. After the CFD module has auto-sized the system (excluding the fittings), the user can look at a variety of fluid dynamics characteristics for each and every part of the design. These characteristics include pipe length, pipe size, flow rate, velocity, fluid volume, Reynolds number and pressure drop among others. If the user needs to make a minor or major modification to the auto-calculated system (such as manually changing the diameter of a particular pipe section) he or she is able to do so easily and then view the impact of the change on the overall system. Below are two more representative examples of how the CFD module can be used:

- If a user wishes to see what happens to the overall pressure drop in a GHX Module and the Reynolds number in a single GHX Circuit if he or she switches from 20% to 10% propylene glycol, he or she can do so easily.
- If a designer has a system that is unbalanced in design (for example a Vault with five 10 circuit "ground heat exchanger modules" and one 6 circuit "ground heat exchanger module") the CFD module can help determine the piping arrangements that provide the most balanced flow.

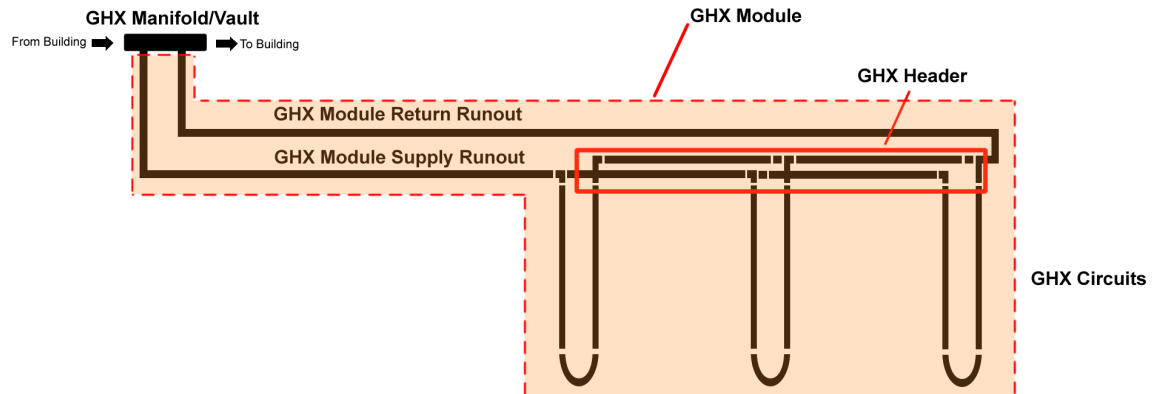
The CFD module either can be used on a standalone basis or in conjunction with a heat exchanger system designed in GLD. On a standalone basis, users can create piping systems using a variety of wizards and tools. On an integrated basis, users can build a design in CFD based off of a designed heat exchanger system.

As with the other modules in Ground Loop Design, it is important to remember that the calculated results are only as good as the quality of the user-defined

inputs. Assuming that reasonable values are provided to the software, the software will provide reasonable results.

Nomenclature

Within the global geothermal industry standard nomenclature is sorely lacking. After having polled and interviewed several dozen active designers, we are adopting the following nomenclature for use in this manual and the software:



GHX: Refers to a ground heat exchanger, and may include vertical, horizontal trenching, horizontal boring, pond or lake heat exchanger buried in the ground or submerged in a body of water.

GHX Circuits: HDPE pipe buried in the ground in horizontal or vertical orientation designed to transfer energy to and from the ground. Typically a number of GHX Circuits are fusion welded to a GHX Header that is in turn fusion welded to a Supply-Return Runout. Heat transfer fluid is circulated through the assembly to a building.

U-Tube: An assembly of two lengths of HDPE pipe connected on one end with a molded, purpose built U-bend.

GHX Header: Connection points between Supply-Return Runout piping and GHX Circuits. GHX Headers are buried in the ground adjacent to the GHX Field and are comprised of an assembly of fusion welded fittings and pipe. Fittings and pipe are manufactured using HDPE resin and are connected using heat fusion (butt fusion, socket fusion or electro-fusion).

Supply-Return Runout : Supply-Return Runout refers to the high-density polyethylene (HDPE) piping installed to connect the GHX Circuit piping to the Pump House header. The Supply-Return Runout has both a supply pipe and a return pipe.

GHX Manifold: Connection point for Supply-Return Runout piping from GHX field. A GHX Manifold is typically located inside a building or in a geothermal Vault located away from the building.

GHX Module: Completed assembly of GHX components, including GHX Supply and Return Runouts, GHX header and GHX Circuits.

GHX Field: Assembly of all GHX Modules connected to a single building or group of buildings via GHX Manifold(s)/Vault(s).

General Features

To aid in the piping optimization process, the CFD module in Ground Loop Design consists of a set of panels, grouped by subject, through which the designer can enter and edit the input variables efficiently. For example, parameters related to fluids are listed on the *Fluid* panel, while options related to the automation of the pipe building process are listed in the *Automation* panel. The idea is that everything related to a single piping optimization project is presented simultaneously and is easily accessible at any time during the design process. The tabbed panels can be seen in figure 11.1, below.



Fig. 11.1 CFD Module Panel List

The CFD module includes several additional features:

- GHX AutoBuilder:
 - Direct or reverse return systems
 - More than one bore per parallel circuit option
 - Double GHX Circuit systems
 - Manifold/Vault builder
 - Ultra Manifold/Vault builder
 - Pipe size exclusion control
 - Flow rate determination for purge velocity
 - Auto supply/return headering for purge velocity optimization
 - Integration with design modules
 - Circulation pumps
- Fluids database
- Fittings database for manual fittings selection
- A range of wizards for design and modification
- Customizable design workspace
- Customizable results displays

- Detailed properties window
- Metric and English unit conversions
- Printed reports
- A 'Calculate' button used to refresh the calculations
- Quick importation and modeling of systems designed in the vertical, horizontal and pond modules

Theoretical Basis

The CFD module applies an innovative approach to finding the solution to complex fluid dynamics problems associated with a nearly unlimited range of GHX field designs. This modular and patent-pending approach involves building up entire piping systems from the following two foundational components:

- a GHX Circuit with supply/return pipes and one or more fittings (inlet/end/outlet)
- a supply/return Pipe Pair with one or more fittings on each pipe

These units can be linked together via drag and drop methods in the two dimensional Layout Manager Workspace. As a designer links these components together, the piping system expands in size and complexity. Regardless of the complexity of the designed system, the CFD module understands the relationship between individual components, families of components and the overall GHX field. The CFD module can then calculate a diverse range of fluid dynamics results or can auto-size the system to satisfy a designer's requirements (such as auto-sizing a supply and return headering system that has a 2 ft/s flow rate throughout it for purging effectiveness).

For a designer to competently engineer a system with the CFD module, he or she will benefit from a familiarity with both of the above mentioned components as well as the simple "grammar" that describes the relationships between and among them. This language is introduced later in this chapter.

Opening Projects

There are two ways to open CFD projects. One is by using the 'New Piping' command from the Design Studio *File* menu or toolbar, and the other is by opening an existing CFD project (*.pip) file from within the CFD module. In the design studio, only one CFD module can be open at a time.



New Projects

New CFD projects may be opened at any time from the Design Studio by choosing 'New Piping' from either the Design Studio *File* menu or the

toolbar. New projects open with several default values that the user can modify as necessary for new projects. The module opens directly into the *Layout* panel.

New CFD projects can be for a stand-alone analysis or for use in conjunction with an existing heat exchanger design project. For use in conjunction with an existing heat exchanger design project, see below.



Existing Projects

Existing CFD projects may be opened at any time from within the CFD module by choosing 'Open' from the CFD module toolbar.

Saving Projects

CFD projects may be saved at any time by clicking the save button on the CFD module toolbar. When the user closes the program or module, the program automatically asks the user if he or she wants to save the CFD project.

Typical Operation

Although each user will have his or her own unique method, the typical operation of the CFD module would include the following steps:

1. Open a new CFD module
2. Choose metric or English units
3. If necessary, enter/modify automation details in the *Automation* Panel
4. If necessary, modify flow rate details in the *Fluid* Panel
5. Modify fluid type as necessary
6. In the *Layout* tab, design a new one use wizards or manual building techniques
7. Adjust the size of the CFD module to maximize viewing flexibility
8. Hit the '*Calculate*' button to analyze the system
9. Hit the '*Display*' button to choose which results to review
10. Select to have the program automatically determine the purging flow rate and/or auto size the piping systems to ensure the user defined purging flow rate
11. Make modifications as necessary
12. Add circulation pumps
13. Save and/or print the CFD piping report

Entering Data into the Tabbed Panels

Ground Loop Design's innovative tabbed panel system provides for easy organization of and direct access to the relatively large number of design parameters associated with a particular project. This section describes the *Circulation Pumps*, *Automation*, *Fluid* and *Layout* panels.

Circulation Pumps

General information pertaining to a piping system's circulation pumps can be entered and found in the *Circulation Pumps* panel as seen in figure 11.2, below.

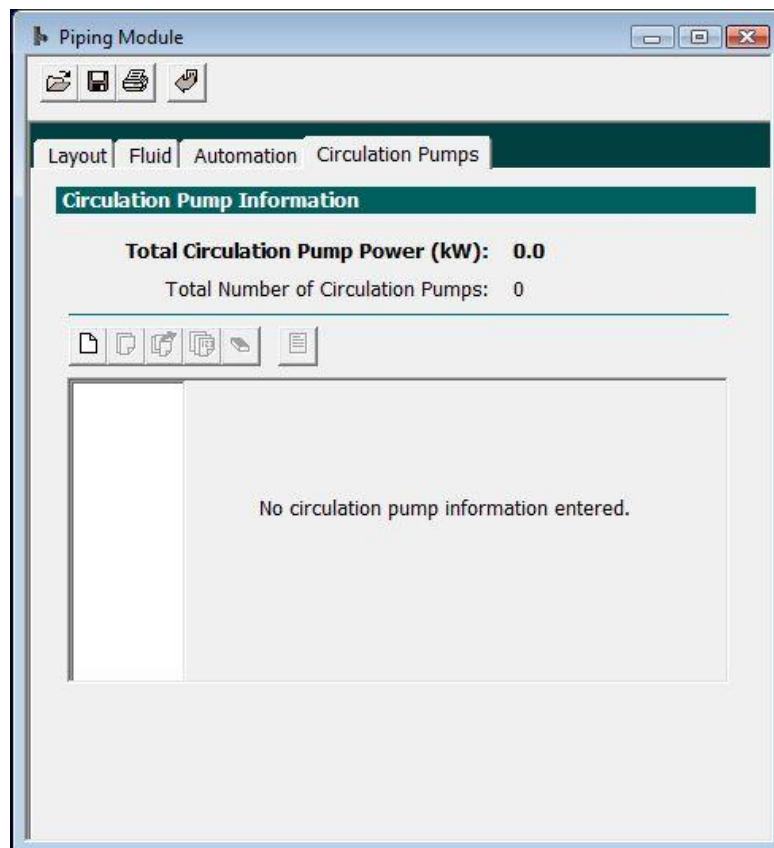


Fig. 11.2 *Circulation Panel Contents*

After a designer has finalized a piping system in the *Layout* Panel, the user can either add circulation pumps in the *Circulation Pumps* panel or can add circulation pumps directly in the *Layout Panel* as required to cover the pressure drop in the piping system. *Remember that these calculations in version 2010 do not include heat pump pressure drops. They may be added in a future version.*

At the top of the panel is a summary of the total circulation pump power and the total number of circulation pumps. These numbers update automatically as the user adds/removes circulation pumps from his or her design.

Managing Circulation Pumps

The buttons along the top of the Circulation Pump Manager are used to add and modify circulation pumps. A closer view is shown in figure 11.3.



Fig. 11.3 Circulation Pump Control Buttons

The five buttons on the left side are circulation pump-editing controls, and they include *New*, *Copy*, *Remove*, *Renumber*, and *Clear*. A *Summary* view of all the pumps may be obtained by hitting the sixth, or *Summary View*, toggle button (this feature not available in all versions).

New and **Copy**

A new circulation pump may be added at any time by clicking the *New* button. Identical pumps may be created from any existing pump by bringing up that pump's data window and clicking the *Copy* button.

Remove and **Clear**

Pumps also can be deleted from the list. Any zone can be removed from the list by bringing up the pump's data window and pressing the *Remove* button. To delete all of the pumps in the list, press the *Clear* button.

Renumber

If several pumps are added or removed from the list, click the *Renumber* button to reorganize the pumps. This button rennumbers the existing pumps from one, starting with the first pump in the current list.

Figure 11.4 shows a system with three pumps added along with the total circulation pump power requirements listed at the top.

Summary View Toggle Button

With the *Summary View* toggle button, the user can at any time simultaneously review all of the circulation pumps. A sample *Summary* panel is shown in figure 11.4A.

Circulation Pump Information

Total Circulation Pump Power (kW): 7.9

Total Number of Circulation Pumps: 3

Pump 1
 Pump 2
 Pump 3

Total Circulation Pump Power

Pump Name: Pump 1

Linked Element: Supply-Return Runout

Required Pressure Drop (ft. hd): 6.8

Required Flow Rate (gpm): 30.0

Required Input Power (kW): 2.6

Pump Power (hP): 3.0

Pump Motor Efficiency: 85

Fig. 11.4 A Circulation Pump System

Circulation Pump Information			
Total Circulation Pump Power (kW): 6.8			
Total Number of Circulation Pumps: 3			
<input type="button" value="File"/> <input type="button" value="Copy"/> <input type="button" value="Paste"/> <input type="button" value="Print"/> <input type="button" value="Help"/>			
Pump Name	Pressure Drop (ft. hd)	Flow Rate (gpm)	Pump Power (kW)
Pump 1	9.0	22.0	1.8
Pump 2	17.0	44.0	2.6
Pump 3	15.0	37.0	2.4

Fig. 11.4A Summary View of Circulation Pumps

Circulation Pump Details

This section stores information related to each individual circulation pump

Pump Name

In this section, the designer specifies the pump name.

Linked Component

In this section, the name of the component in the *Layout* panel that has the circulation pump associated with it is displayed here. If the designers adds a pump directly from the *Circulation Pumps* tabbed panel and not through the *Layout* panel, the linked component will be left blank. Note that most designers find it more efficient to add pumps from within the *Layout* panel.

Required Pressure Drop

In this section, the designer specifies the required pressure drop for the pump. Again, if the designer adds the circulation pump from the *Layout* panel, the required pressure drop automatically will be transferred from the *Layout* panel results (which store the calculated fluid dynamics results).

TIP

Note that when the designer adds a circulation pump in the *Layout* panel, the details of the pump are stored and updated dynamically in the *Circulation Pump* panel. When the fluid dynamics are updated in the *Layout* panel, the results are dynamically updated in the *Circulation Pump* panel as well.

Required Flow Rate

In this section, the designer specifies the required flow rate for the pump. Again, if the designer adds the circulation pump from the *Layout* panel, the required flow rate automatically will be transferred from the *Layout* panel results (which store the calculated fluid dynamics results).

Note that when the designer adds a circulation pump in the *Layout* panel, the details of the pump are stored and updated dynamically in the *Circulation Pump* panel. When the fluid dynamics are updated in the *Layout* panel (such as well selecting a different flow rate), the results are dynamically updated in the *Circulation Pump* panel as well.

Required Input Power

The required input power is calculated automatically from the user-defined pump power and pump motor efficiency. It is anticipated that a future version of GLD will include a comprehensive circulation pump database that automatically calculates required input power.

Automation

Input parameters relating to piping system design automation are located in the *Automation* panel, as shown in figure 11.5. These parameters are divided into several sub tabbed panels including those related to individual GHX Modules, Manifolds, Ultra Manifolds and Pipe Size options available for use by the auto-building algorithms. The combined information stored in the *Automation* panel is used by the CFD module's algorithms to build and/or auto-size piping systems. Figure 11.5 is an overview of the entire *Automation* panel.

The screenshot shows the 'Piping Module' window with the 'Automation' tab selected. The panel is organized into several sections:

- Manifold and GHX Module Automation Presets**: This section contains four sub-tabs: 'GHX Module' (selected), 'Manifold', 'Ultra Manifold', and 'Pipe Sizes'.
- Return Piping Style**: A section with a 'Return Type' dropdown menu set to 'Reverse Return'.
- Circuit Information**: This section is divided into 'Circuits' and 'Headers' sub-tabs. The 'Circuits' sub-tab is active, showing the following parameters:

Number of Circuits:	8	Extra	
One-way Circuit Length (ft):	300.0	Extra	0.0
Circuit Pipe Size:	SDR11	1 in. (25 mm)	
Circuits Per Parallel Loop:	1		
Circuits Per One-Way Length:	1		
- Supply-Return Runout Information**: This section contains parameters for the supply and return runouts:

One Way Length (ft):	200.0	Extra	0.0
Pipe Size:	SDR11	2 in. (50 mm)	

Fig. 11.5 Automation Panel Contents

GHX Module

Details related to an individual GHX Module can be seen in the GHX Module sub-tabbed panel in figure 11.5

Return Piping Style

This section stores information related to direct and reverse return systems.

Return Type

In this section, the designer specifies whether the GHX Module has direct or reverse return routing. Note that this specification has a potentially large impact on both the calculated results as well as the visual representation of the design in the *Layout* panel. This will be discussed in more detail later on in this chapter.

Circuit Information

The Circuit tabbed panel stores parameters related to the GHX Circuits in each GHX Module. Circuit information is broken into Basic and Details tabbed panels. The Basic tabbed panel can be seen in detail in figure 11.6.

Circuit Information		
Circuits	Headers	
Number of Circuits:	8	Extra
One-way Circuit Length (ft):	300.0	0.0
Circuit Pipe Size:	SDR11	1 in. (25 mm)
Circuits Per Parallel Loop:	1	
Circuits Per One-Way Length:	1	

Fig. 11.6 Basic Circuit Information Panel Contents

Number of Circuits

Here the user enters the number of GHX Circuits desired in a single GHX Module.

Circuit Separation

Here the user enters the center-to-center separation between GHX Circuits. This separation distance dictates the GHX Header length between adjacent GHX Circuits.

One Way Circuit Length

Here the user enters the one way length of a single GHX Circuit.

Circuit Pipe Size

Here the user enters the circuit pipe size for a single GHX Circuit.

Circuits Per Parallel Loop

Here the user enters the number of GHX Circuits he or she desires per parallel loop. Put another way, if a designer wants to have two GHX Circuits in series on one parallel loop, the designer can enter a "2" here. Figure 11.30 is an example of 2 circuits per parallel loop.

Circuits Per One-Way Length

Here the user enters the number of GHX Circuits he or she desires per one way length. Put another way, if a designer wants to have two GHX Circuits in parallel or in series in the same borehole, the designer can enter a "2" here.

Headers Information

The Header tabbed panel stores parameters related to the basic headering system including circuit separation (ie- the headering piping length between boreholes) and the headering pipe type and size. This can be seen in figure 11.7.

Circuit Information		
Circuits	Headers	
Circuit Separation (ft):	20.0	Extra 0.0
Header Pipe Size:	SDR11 ▼	2 in. (50 mm) ▼

Fig. 11.7 Details Information Panel Contents

Supply-Return Runout Information

This section stores information regarding the Supply-Return Runout pipe pair (supply pipe and return pipe) that links the GHX Header with a Manifold/Vault, etc.

One Way Length

Here the user can enter the one way length from the Manifold/Vault to the first GHX Circuit. The return pipe will default to the same length. These lengths can be modified later as necessary.

Pipe Size

Here the user enters the Supply-Return Runout pipe size. Both the Supply and Return Runout will be the same size but they can be adjusted independently if necessary (an explanation of how to do this comes later).

Manifold

Details related to an individual Manifold can be seen in the Manifold tabbed panel in figure 11.8. Note that a Manifold also can be thought of as being a Vault.

Manifold and GHX Module Automation Presets

GHX Module **Manifold** Ultra Manifold Pipe Sizes

Return Piping Style

Return Type: Direct Return

Section Outlet Information

Section Outlet Number: 5

Section Outlet Separation (ft): 6.0 Extra 0.0

Section Outlet Pipe Size: SDR11 2 in. (50 mm)

Supply-Return Runout Information

One Way Length (ft): 200.0 Extra 0.0

Pipe Size: SDR11 3 in. (80 mm)

Fig. 11.8 Manifold Information Panel Contents

Return Piping Style

This section stores information related to direct and reverse return systems.

Return Type

Return type is locked at direct return since reverse-return Manifolds are rarely if ever used.

Section Outlet Information

The section outlet information refers to how the outlets in the Manifold connect to GHX Modules via the GHX Module Supply-Return Runouts.

Section Outlet Number

Here the user enters the number of outlets there are in the Manifold/Vault that connect to GHX Modules via the GHX Module Supply-Return Runouts.

Section Outlet Separation

Here the user enters the distance separating the section outlets in the Manifold/Vault.

Section Outlet Pipe Size

Here the user enters the outlet size connecting to GHX Modules via the GHX Module Supply-Return Runout(s).

Supply-Return Runout Information

The Manifold Supply-Return Runout information refers to the pipe pair that is the parent of the Manifold/Vault. For example, in an in-building Manifold system, the Supply-Return Runout information would likely pertain to the pipe pair going to/from the Manifold and to/from the circulation pump house or equivalent.

One Way Length

Here the user enters the one way length of the supply pipe. The return pipe will default to the same length.

Pipe Size

Here the user enters the Supply-Return Runout pipe size. Both the Supply and Return Runout will be the same size but they can be adjusted independently if necessary (an explanation of how to do this comes later).

Ultra Manifold

Details related to an individual Ultra Manifold can be seen in the Ultra-Manifold tabbed panel in figure 11.9. Note that in the CFD module, an Ultra-Manifold is defined as a Manifold or a Vault that is connected to other "child" Manifolds/Vaults. Ultra-Manifolds are only applicable for use in the largest of commercial projects that require nested levels of Vaults/Manifolds.

Manifold and GHX Module Automation Presets

GHX Module Manifold **Ultra Manifold** Pipe Sizes

Return Piping Style

Return Type: Direct Return

Section Outlet Information

Section Outlet Number: 5 Extra

Section Outlet Separation (ft): 6.0 0.0

Section Outlet Pipe Size: SDR11 3 in. (80 mm)

Supply-Return Runout Information

One Way Length (ft): 200.0 Extra 0.0

Pipe Size: SDR11 4 in. (100 mm)

Fig. 11.9 Ultra Manifold Information Panel Contents

Return Piping Style

This section stores information related to direct and reverse return systems.

Return Type

Return type is locked at direct return since reverse-return Manifolds are rarely if ever used.

Section Outlet Information

The section outlet information refers to how the outlets in the Ultra Manifold connect to Manifolds/Vaults via the Manifold Supply-Return Runouts.

Section Outlet Number

Here the user enters the number of outlets there are in the Ultra Manifold/Vault that connect to child Manifolds/Vaults via the Manifold Supply-Return Runout(s).

Section Outlet Separation

Here the user enters the distance separating the section outlets in the Ultra Manifold.

Section Outlet Pipe Size

Here the user enters the outlet size connecting to the Supply-Return Runouts of the child Manifolds/Vaults.

Supply-Return Runout Information

The Ultra Manifold Supply-Return Runout information refers to the pipe pair that is the parent of the Ultra Manifold/Vault. For example, in an in-building Ultra Manifold system, the Supply-Return Runout information would likely pertain to the pipe pair to/from the circulation pumps and to/from the Ultra Manifold.

One Way Length

Here the user enters the one way length of the supply pipe. The return pipe will default to the same length.

Supply Pipe Size

Here the user enters the supply pipe size. The return pipe will default to the same size.

Pipe Sizes

Details related to the pipe sizes available for auto-building and auto-optimization of piping systems can be seen and selected in the Pipe Sizes tabbed panel in figure 11.10.

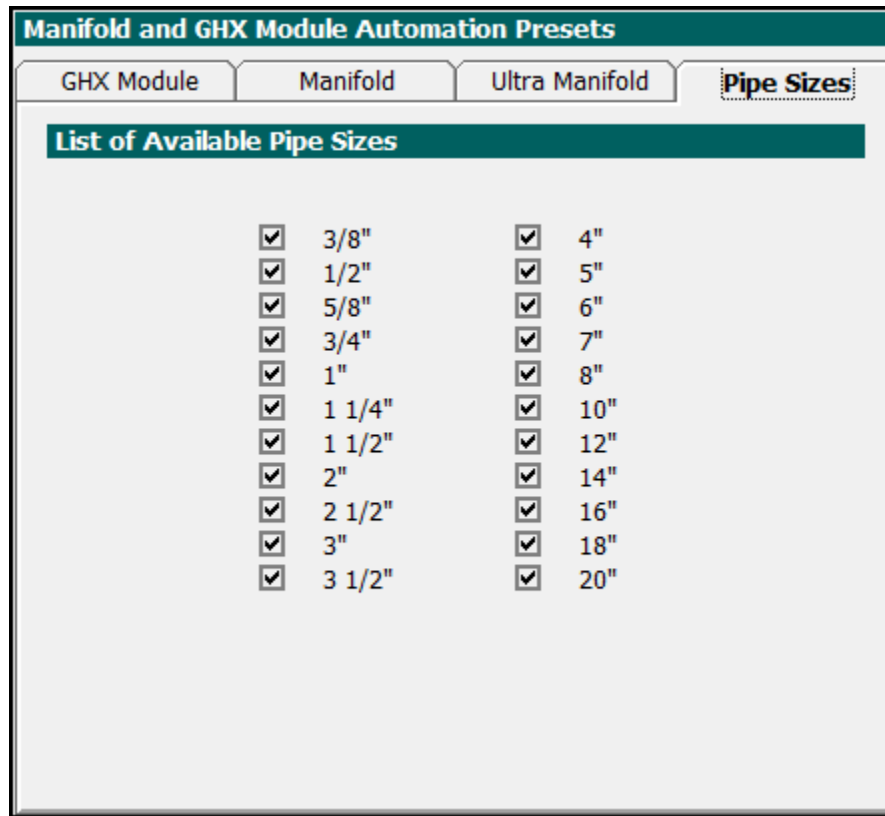


Fig. 11.10 Pipe Sizes Selection Panel

List of Available Pipe Sizes

This section contains a list of available pipe sizes in the CFD module. If there are certain pipe sizes a designer does not wish to use in a design, he or she can deselect them. The optimization algorithms in the CFD module will only use the selected pipe sizes for designing systems.

By selecting pipe sizes of interest and deselecting (for example) pipe sizes that are unavailable in a particular designer's region or market, the designer helps ensure that the system designed by the CFD module actually can be built by a construction team using readily available pipes.

Fluid

All parameters relating to fluid flow rates and fluid properties are listed in the *Fluid* panel as shown in figure 11.11. In addition, this panel contains top level controls for some of the auto design features associated with optimizing systems for appropriate purging fluid velocities.

Piping Module

Layout | **Fluid** | Automation | Circulation Pumps

Fluid Information

Peak Load Flow Rate (gpm): 30.00

Installed Capacity Flow Rate (gpm): 60.00

Purging Flow Rate (gpm): 90.00

☒ Auto-Flow
☒ Auto-Size

Purging Target Velocity (ft/s): 2.00 (Minimum) 100.00 (Maximum)

Solution Properties

☐ Automatic Entry Mode ☒ % by Weight ☐ % by Volume

Fluid Type: 100 % Water

Design Temperature: 32.0 °F

Specific Heat (Cp): 1.00 Btu/(°F*lbm)

Density (rho): 62.4 lb/ft³

Dynamic Viscosity (u): 3.738E-5 lbf*s/ft²

Check Fluid Tables

Fig. 11.11 *Fluid Panel Contents*

Fluid Information

In this section users may enter flow rates for peak load, installed capacity and purging. These three flow rate options, handy for comparison purposes, are used to calculate fluid dynamics results in the Layout tab. Note that users can switch between the three entered flow rates in the Layout tab and so do not have to keep returning back and forth to the Fluid tab during the piping optimization process.

Peak Load Flow Rate

The peak load flow rate is the flow rate necessary to cover the peak heating or cooling load. The peak flow rate, typically calculated in the heat exchanger design modules, is based on the peak load and the flow rate in GPM/ton or its metric equivalent. The user can enter the peak flow

rate here or it can be automatically transferred in when a user imports a design project into the CFD module.

Installed Capacity Flow Rate

Some designers desire to see the fluid dynamics performance of their piping system under the installed capacity flow rate for circulation pump sizing purposes (for example). Typically, the installed capacity flow rate is higher than the peak load flow rate.

Purging Flow Rate



Knowing the appropriate purging flow rate is essential for proper purging of a GHX Module or a GHX field prior to start-up. Failure to purge a system properly can result in decreased system performance. Calculating a purging flow rate for a particular system can be a time intensive process. In addition, many designers prefer to engineer a GHX Module headering system to ensure ease-of-purging. Such engineering can require significant effort. As a result, many engineers design the same system over and over again without exploring potentially more efficient design scenarios. The new CFD module can automatically size the headering system, save time and provide designers with a new way to experiment and innovate.

Manual Entry of Purging Flow Rate

In the default configuration that can be seen in figure 11.12, the user can enter a purging flow rate of interest. After doing so, the user can see how the designed system performs in the *Layout* panel. In general, for purge flow rates, designers will be looking at the velocity in the GHX Circuits, headering sections, and run out pairs.

Purging Flow Rate (gpm):	90.00	<input checked="" type="checkbox"/> Auto-Flow
	Minimum	<input type="checkbox"/> Auto-Size
Purging Target Velocity (ft/s):	2.00	Maximum
		100.00

Fig. 11.12 Purging Flow Rate Data Entry

Automatic Purging Flow Rate Calculations

Auto Adjust Option

If the designer wishes to have the CFD module calculate an appropriate purging flow rate *for the GHX Circuits* based on the system shown in the *Layout* panel, all he or she has to do is check the *Auto Adjust* box and

input a target minimum purging velocity. Doing so will activate the *Purging Flow Rate Auto-Optimizer* tool which is described at the end of this chapter.

As can be seen in figure 11.13, when the user selects the *Auto Adjust* box, the purging flow rate input box deactivates and the target velocity flow rates activate.

When only the Auto Adjust box is checked (and the Auto Size box is unchecked), the maximum purge velocity has no impact on the calculations and therefore is deactivated.

TIP

Note that for purging a system with water (water is the standard fluid for purging and the fluid utilized automatically by the CFD module), a minimum velocity of 2 ft/s throughout the system to be purged is optimal.

After a user hits the *Calculate* button in the Layout panel, the calculated required purging flow rate will update in the purging flow rate box (in figure 11.13 the flow rate is calculated to be 68.3gpm based on the system in the *Layout* panel).

Purging Flow Rate (gpm):	68.3	<input checked="" type="checkbox"/> Auto-Flow
		<input type="checkbox"/> Auto-Size
	Minimum	Maximum
Purging Target Velocity (ft/s):	2.00	100.00

Fig. 11.13 The Auto Adjust Option

Auto Adjust and Auto Size Option

Designers have an even more advanced control at their disposal in the CFD module: the auto size option. With this option, both the minimum and maximum target velocities impact the final result.

When a user selects the *Auto Adjust* option as mentioned above, the *Auto Size* checkbox also becomes available. This can be seen in figure 11.14. When a user checks both boxes, adjusts the minimum and maximum flow rates as necessary, returns to the Layout panel and hits *Calculate*, the CFD module will automatically adjust the pipe sizes across the GHX Module runouts and GHX headers to minimize the purging flow rate and pressure drop while ensuring that the target purging velocity range is achieved throughout the piping system.

In other words, when a user selects both *Auto Adjust* and *Auto Size*, the CFD module designs a new piping system or modifies an existing piping system for optimal purging flow. The tool that performs this calculation is

called the *GHX Header Design Optimizer* and it is explained in great detail at the end of this chapter.

Some designers wonder about the purpose of the maximum target velocity. The maximum target velocity impacts the auto piping sizing selections in the following way: if the user specifies a low maximum target velocity (say 5 ft/s) the auto sizing function has flexibility to choose a larger pipe diameter that offer slower flow rates and lower pressure drops. If the user specifies a higher maximum target velocity (say 50 ft/s), the auto sizing function will tend to be limited to smaller pipe diameters that enable faster velocities and also higher head loss. This will be explored in more detail later in this chapter.

Purging Flow Rate (gpm):	90	<input checked="" type="checkbox"/> Auto-Flow
	Minimum	<input checked="" type="checkbox"/> Auto-Size
Purging Target Velocity (ft/s):	2.00	Maximum
		100.00

Fig. 11.14 The Auto Size Option

Solution Properties

Solution properties are also included in the *Fluid* panel. These include the *design temperature (which impacts viscosity)*, *specific heat*, *density* and *dynamic viscosity* of the circulating fluid. Also, a reference label is included so that the designer knows the percentage of antifreeze and antifreeze type; ***however, this reference label is not currently linked to the other input parameters.***

In **automatic entry mode**, the user first selects the fluid type and then selects the desired freezing temperature. GLD automatically displays the specific heat, density and viscosity for the fluid selection. When the automatic entry mode checkbox is marked, the program is in automatic entry mode.

In **manual entry mode**, the user manually selects and inputs the specific heat and density for the target solution as seen in figure 11.15. When the automatic entry mode checkbox is unmarked, the program is in manual entry mode.



Fig. 11.15 Solution Properties Data Entry



*Note: Since solution properties vary considerably and non-linearly with temperature, type and percentage of additive, GLD does not include detailed automatic antifreeze information for all conditions. Generalized tables of data may be found in the 'Fluid Properties' tables. **For all designs, it is recommended that the designer manually enter the desired values in the input text boxes to ensure that fluid properties match the design requirements.***

Layout

The *Layout* panel is the heart of the CFD module. This is the panel in which the designer builds piping systems, explores their fluid dynamics implications and then modifies the design manually or automatically as necessary. The *Layout* panel can be seen in figure 11.16.

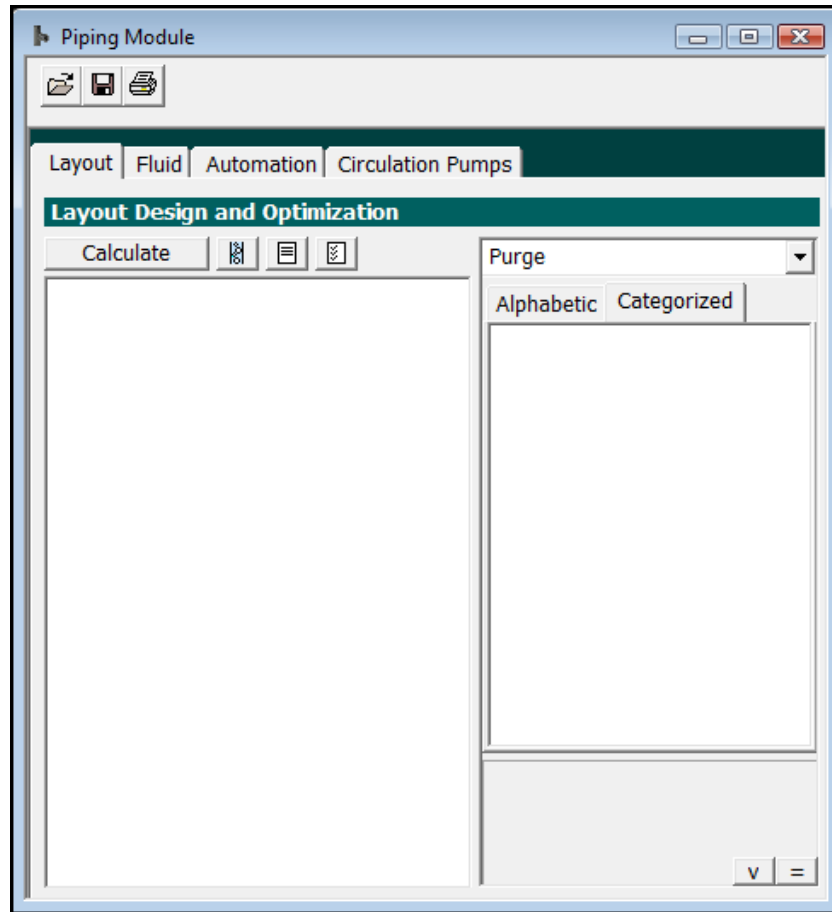
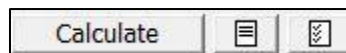


Fig. 11.16 Layout Panel Contents In Purge Mode

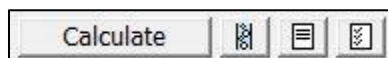
The *Layout* panel is broken into five primary sections including the Calculate and Results Display Buttons, the Layout Manager Workspace (the big white space on the left), the Flow Type Drop Down Menu, the Properties Window on the right and the Circuit Confirmation Calculator (invisible in figure 11.16).

Section One: Calculate and Results Display Buttons

At the top are the 'Calculate' and two or three types of results display buttons as can be seen below.



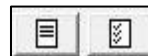
Three Buttons in Peak Load and Equipment Flow Rate Modes



Four Buttons in Purge Flow Rate Mode

After the user has created or modified a design he or she can hit the 'Calculate' button to see the updated fluid dynamics results.

The other results display buttons fulfill a special role in the *Layout* panel. The CFD module produces a large range of results. At certain times in the design process one subset of results may be applicable. At another stage in the design process a different subset of results may be applicable. At the end of the design process, the designer may wish to view all results. Because of the diverse processes involved in the design process, users have the flexibility of selecting which specific results they wish to see at a particular time and how they wish to see them. Users can do so via the two results display buttons:



When the user hits the left button (the "Review" button), the Review panel appears. The Review panel is well-suited for quickly reviewing a design. It is explained in more detail later in this chapter.

When the user hits the right button (the 'Display' button), a pop-up window will appear as can be seen in figure 11.17.

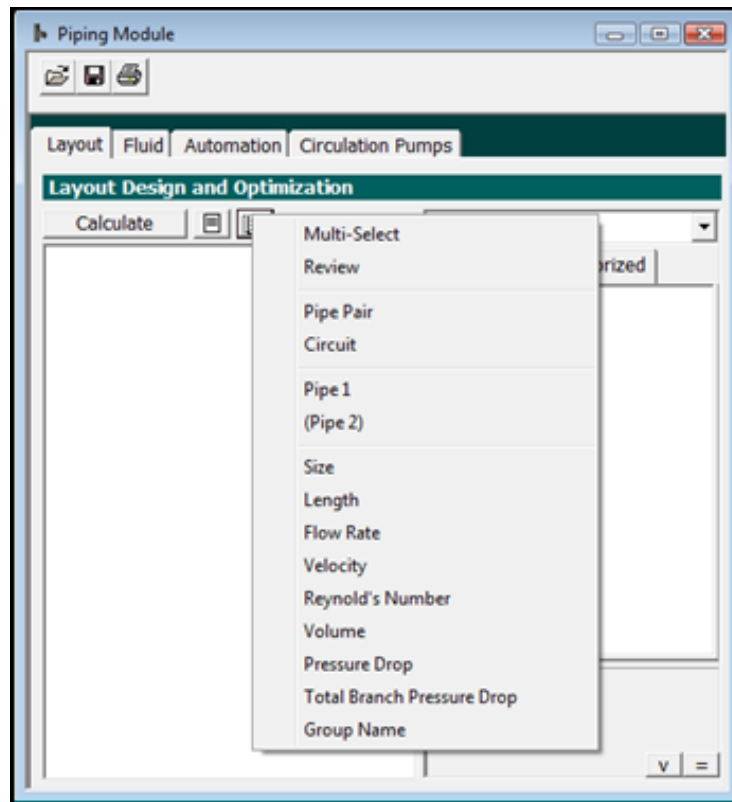


Fig. 11.17 Display Options Pop Up Window

As can be seen in figure 11.17, display options are broken into four groups:

- Multi-Select/Review
- Component types: Pipe Pair and Circuit
- Supply and Return Pipes: Pipe 1 and (Pipe 2)
- Details: Size, Length, Flow Rate, Velocity, Reynold's Number, Volume, Pressure Drop and Group Name

Each group is explored below.

Multi-Select and Review

Multi-Select: Selecting this option opens up a new window that enables a designer to select multiple parameters at the same time.

Review: Selecting this option opens up the Review panel (explained below)

Pipe Pair and Circuit (ie, the component types)

Pipe Pair: When a user selects Pipe Pair, all of the Pipe Pairs in the piping system design are enabled to display the selected fluid dynamics results

Circuit: When a user selects Circuit, all of the GHX Circuits in the piping system design are enabled to display the selected fluid dynamics results

Pipe 1 and (Pipe 2)

Pipe 1: When a user selects Pipe 1, all of the supply-side pipes in the piping system design (for both pipe pairs and GHX Circuits) are enabled to display the selected fluid dynamics results

(Pipe 2): When a user selects (Pipe 2), all of the return-side pipes in the piping system design (for both pipe pairs and GHX Circuits) are enabled to display the selected fluid dynamics results. Note that the CFD modules uses the () notation to indicate return-side flow.

Length, Size, Flow Rate, Velocity, Reynold's Number, Volume and Pressure Drop

Size: When a user selects Size, all of the pipe diameters for the selected component types (pipe pair and/or circuit) and pipes (pipe 1 and/or (pipe 2)) are displayed.

Length: When a user selects Length, all of the pipe lengths for the selected component types (pipe pair and/or circuit) and pipes (pipe 1 and/or (pipe 2)) are displayed.

Flow Rate: When a user selects Flow Rate, all of the flow rates within the selected component types (pipe pair and/or circuit) and pipes (pipe 1 and/or (pipe 2)) are displayed.

Velocity: When a user selects Velocity, all of the velocities within the selected component types (pipe pair and/or circuit) and pipes (pipe 1 and/or (pipe 2)) are displayed.

Reynold's Number: When a user selects Reynold's Number, all of the Reynold's Numbers within the selected component types (pipe pair and/or circuit) and pipes (pipe 1 and/or (pipe 2)) are displayed.

Volume: When a user selects Volume, all of the volumes within the selected component types (pipe pair and/or circuit) and pipes (pipe 1 and/or (pipe 2)) are displayed.

Pressure Drop: When a user selects Pressure Drop, all of the pressure drops within the selected component types (pipe pair and/or circuit) and pipes (pipe 1 and/or (pipe 2)) are displayed.

Total Branch Pressure Drop: When a user selects this, the program displays pressure drops across entire systems of elements.

Group Name

Group Name: Group name is a "meta" property that applies to all components in a particular component family (such as a GHX Module, a Manifold, etc). The group name is used to sort and organize components when viewing results in the Review panel.

Note that in using this system, a user must select at least one component type (pipe pair/circuit), one pipe (Pipe 1/(Pipe 2)) and one characteristic (length, size, etc) before results will be actively displayed. Also note that a user has to build a piping system before piping system results can be seen.

Notice in figure 11.18a which options are selected from the Display button (Pipe Pair, Circuit, Pipe 1, Length and Reynold's Number). Next, notice that lengths and Reynold's Numbers for the supply side pipe (Pipe 1) for both the Pipe Pairs and Circuits are displayed in figure 11.18b.

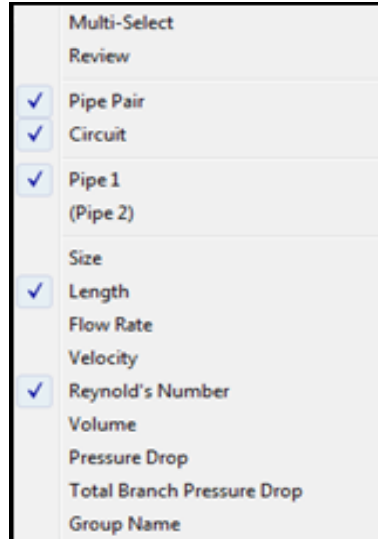


Fig. 11.18a Selected Results for Display

Layout Design and Optimization			
Calculate			
GHX Module - Supply-Return Runout	200.0 ft	4461	
U Circuit #01	300.0 ft	4034	
GHX Header Section #01	20.0 ft	2228	
U Circuit #02	300.0 ft	4024	

Fig. 11.18b Sample of Displayed Results

In purge mode, a third button becomes visible as seen below:



This third button provides ready access to automated controls such as auto-adjusting the purging flow rate and auto-sizing the headering system. These features are described in more detail below.

Section Two: The Layout Manager Workspace

The left side section of the *Layout* panel is the Layout Manager Workspace as can be seen in figure 11.16. The Layout Manager Workspace is the area in which the designer builds, modifies and reviews piping systems. The Layout Manager Workspace is in many ways the heart of the enter CFD module. Consequently, it provides the user with significant flexibility and customization. These customization features will now be introduced.

Customizing the Layout Panel

Because piping systems can be quite large, sometimes the designer will want to adjust the size and position of the Layout Manager Workspace to optimize its functionality.

Layout Manager Workspace Sizing Control

There are three ways to optimize the Layout Manager Workspace area.

- 1) The designer can adjust the size of the entire CFD module in one of two ways. First, the designer can maximize the module by hitting the maximize button in the upper right corner of the CFD module. Second, the designer can move the mouse cursor to one of the edges of the CFD module and then click-hold-drag to expand the CFD module. An expanded CFD module can be seen in figure 11.19. Compare this to figure 11.16.

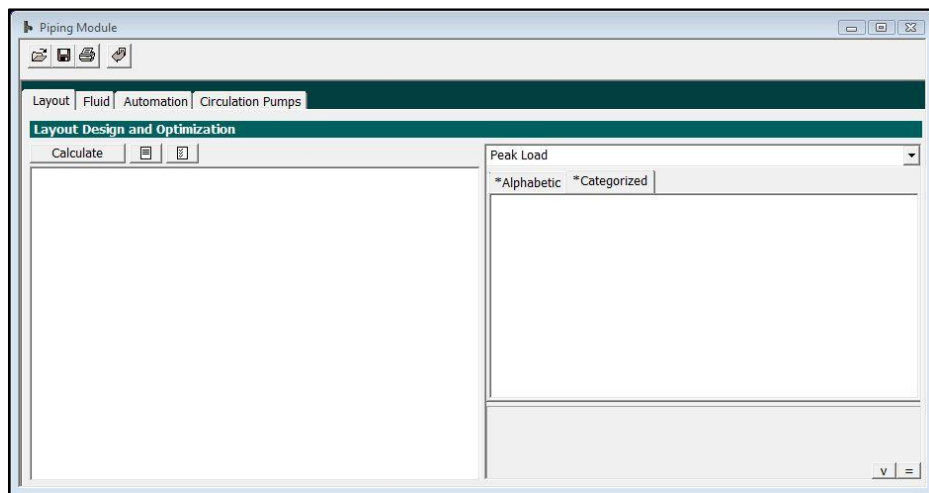


Fig. 11.19 An Expanded Layout Screen Provides More Room to Work

- 2) The designer can move the Properties Window (the window on the right side in the default configuration) to the bottom of the *Layout* tab to provide more horizontal room for the Layout Manager Workspace. The designer can do this by hitting the *Toggle View* button which can be found in the bottom right corner of the *Layout* panel:



When the user hits the *Toggle View* button the screen will shift as can be seen in figure 11.20. In this view, the Layout Manager Workspace extends to the right edge of the CFD module and the Properties Window is

underneath the Layout Manager Workspace. For systems with large GHX Modules, this is an excellent technique for viewing the entire GHX Module on one screen.

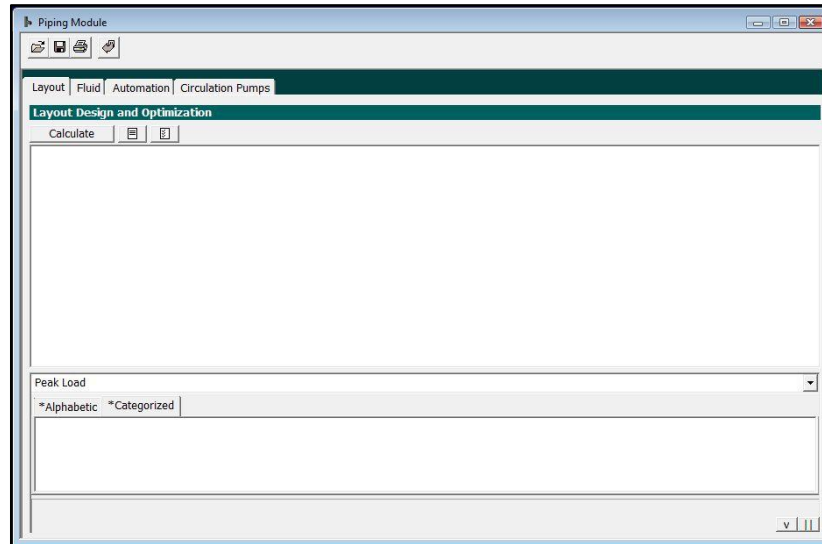


Fig. 11.20 The Toggle View Button Extends the Layout Manager Workspace

The user can hit the *Toggle View* button at any time to switch back and forth between the two views.

3) The designer can drag the Property Window to the right or left to provide more or less Layout Manager Workspace area as needed. To do so, the user needs to be in the primary Toggle View as can be seen in figure 11.19. The user can then move the mouse to the vertical bar that separates the Layout Manager Workspace from the Property window and click and drag to move the bar either left or right. An example of such an adjustment can be seen in figure 11.21.

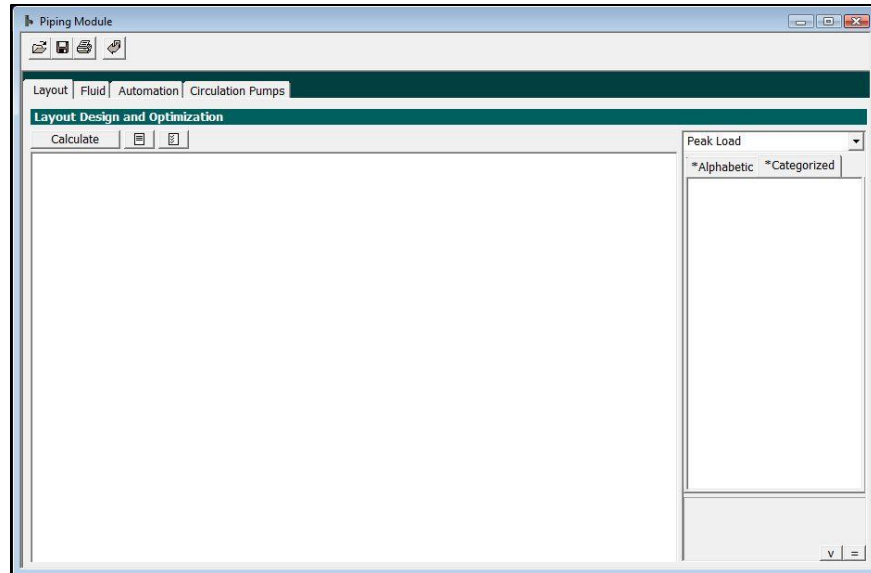


Fig. 11.21 Adjust the Relationship between the Two Main Windows

Section Three: Flow Type Selection

At the top right is a drop down menu from which users can select to see the piping design performance results under three flow scenarios: peak load, equipment (installed capacity) and purge. This can be seen in figure 11.22.

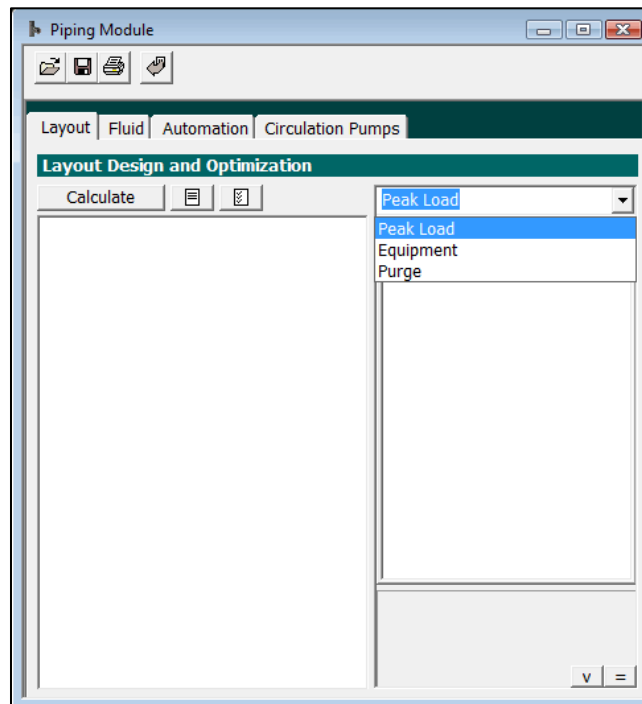


Fig. 11.22 Flow Scenario Selection

Remember that the flow rates for each can be entered in the *Fluid* panel.

Section Four: The Properties Window

The fourth section is the Properties Window. When a designer selects a piping design component in the Layout Manager Workspace, a wide range of details pertaining to the component can be viewed and modified in the Properties Window. A Property Window can be seen in figure 11.23.

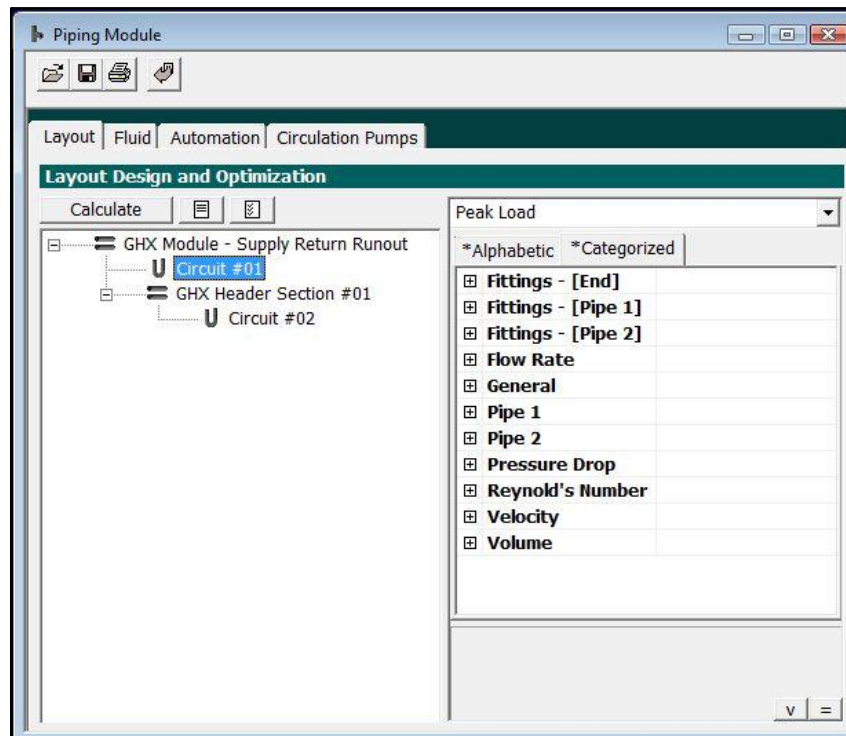


Fig. 11.23 The Properties Window

In figure 11.23, in the Layout Manager Workspace circuit #1 has been selected. Details regarding circuit #1 can be seen in the Properties Window. Properties for all GHX Circuits include the following:

- Fittings (end or bottom)
- Fittings (on pipe 1, the supply side pipe of the GHX Circuit)
- Fittings (on pipe 2, the return side pipe of the GHX Circuit)
- Flow Rate
- General Information
- Pipe 1 (supply pipe) details
- Pipe 2 (return pipe) details
- Pressure Drop
- Reynold's Number

- Velocity
- Volume

Properties for Pipe Pairs (Supply-Return Runouts, GHX Deader sections) are identical except that they have only two fittings (by default) rather than the three (by default) of GHX Circuits.

Users can explore the details of each property section by clicking on the "+" to expand the view. In figure 11.24 the Pipe 1 (supply pipe) property details have been expanded.

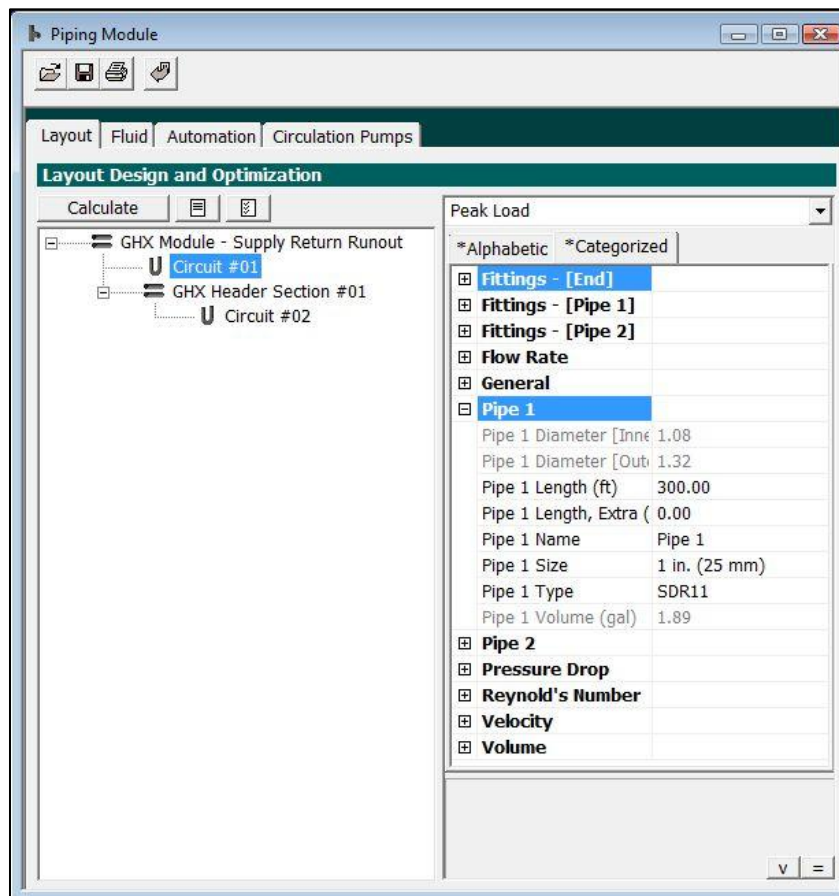


Fig. 11.24 Pipe 1 Properties for GHX Circuit #1 Expanded

Pipe 1 has a number of user-definable and modifiable properties including the pipe length, an extra pipe length, the pipe name as it is displayed in the Layout Manager Workspace, the pipe size and the pipe type. Grayed out properties (such as the fluid volume in the pipe) are calculated by the program automatically and not adjustable.

Note that the Properties Window is only one of several ways of entering and modifying pipe information. The Properties Window provides the most detail but

it also offers the slowest entry method. Other, faster entry methods are described below.

Section Five: The Circuit Confirmation Calculator

The fifth section is the Circuit Confirmation Calculator panel. When a designer is designing a larger commercial system, the Circuit Confirmation Calculator keeps track of both the number of GHX Circuits and the total length of circuit pipe.

The Circuit Confirmation Calculator by default is hidden. To view the Calculator, a user can push the following button which can be found in the bottom right corner of the *Layout* Panel:



When a user hits the above button the Circuit Confirmation Calculator will appear at the top of the *Layout* panel as can be seen in figure 11.25.

Layout Design and Optimization			
	Required	Included	Difference
Total Circuit Length (ft):	0.0	0.0	0.0
Total Circuit Number:	0	0	0

Fig. 11.25 The Circuit Confirmation Calculator is Visible

The user can enter the total circuit length and total circuit number required at the start of a design and the Calculator will count down towards 0 as the user adds circuits. This calculator ensures that a designer does not have too many or too few circuits and that the total calculated circuit length equals the total expected circuit length.

The GLD Piping Language

Now that the primary components of the CFD module as well as the user interface have been introduced, it is possible to begin understanding the GLD piping language. The GLD piping language consists of two fundamental components and a grammar that describes how the components interact. This section describes the fundamental components and explains via several practical examples how to use the visual piping "grammar" to model piping systems including direct and reverse return systems. Please note that while the below description is quite detailed, it is not necessary to remember everything because the CFD module automatically handles nearly all of these features and functions. However, having a basic understanding of the system will give a designer the power to modify and adjust his or her systems quickly and effortlessly.

TIP

Piping Components

As mentioned previously, two components power the entire CFD module. They are:

- a **GHX Circuit** with at least three fittings (inlet/end/outlet)
- a **Supply/Return Pipe Pair** with at least two fittings: one supply side fitting (generally before the supply side pipe) and one return side fitting (generally after the return side pipe)

U

The GHX Circuit

An individual GHX Circuit consists of the following five subcomponents:

- A_f - Fitting for attachment to parent header pipe (optional)
- A - Supply side pipe
- A_{fu} - End fitting that connects Pipe A and Pipe A' (optional)
- A' - Return side pipe (usually length $A = \text{length } A'$)
- A'_f - Fitting for attachment to child header pipe (optional)



Note: arrows indicate supply/return flow directions. Also, the space between sections is intentional to illustrate the individual subcomponents.

Each of these subcomponents has a large number of user-definable characteristics associated with it including:

Fittings A_f , A_{fu} and A'_f

- Fitting type (socket tee branch, butt tee branch, etc.)
- Fitting pipe size
- Fitting equivalent length
- Fitting name
- Fitting volume

Pipe A and A':

- Pipe size
- Pipe type
- Pipe inner diameter
- Pipe outer diameter
- Pipe length
- Extra pipe length
- Pipe name
- Pipe volume

Note that each section can have multiple fittings in case a design requires a series of reducing fittings or butt fusions. Also note that while the range of control can seem overwhelming, in automatic mode, most of these variables are selected automatically for the designer by the CFD algorithms. In this version of the software, note that the fittings are not automatically selected by the CFD algorithms.

Within the CFD Layout Manager Workspace, a single GHX Circuit appears in figure 11.26. Note that the Workspace is on the left side of the screen and the right side contains a Properties Window. The properties window can be expanded as necessary to view all of the characteristics for all five subcomponents of each GHX Circuit. Note that the Properties Window also contains fluid dynamics results for each circuit. These will be reviewed later.

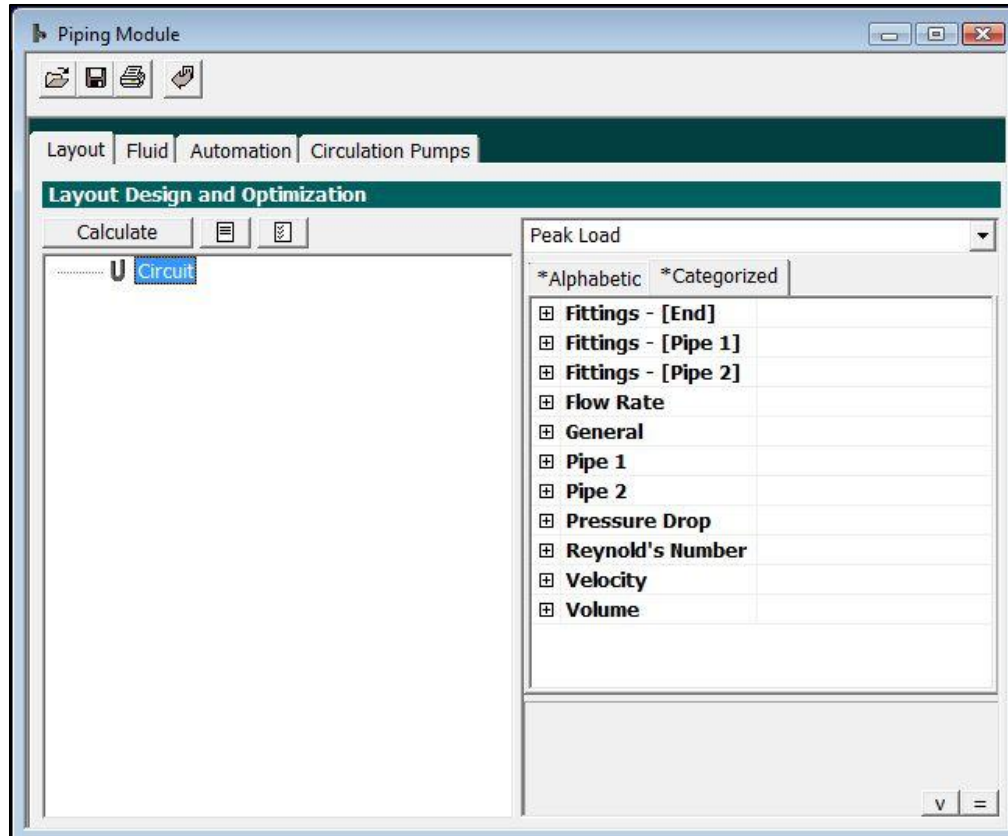


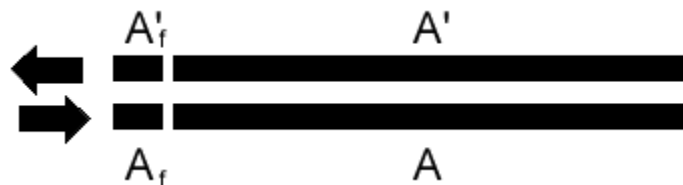
Fig. 11.26 The basic GHX Circuit



The Supply/Return Pipe Pair

An individual Supply/Return Pipe Pair consist of the following four subcomponents:

- A_f - Supply side fitting (generally before the supply side pipe)
- A - Supply side pipe
- A' - Return side pipe
- A'_f - Return side fitting (generally after the return side pipe)



Note: arrows indicate supply/return flow directions. Also, the space between sections is intentional to illustrate the individual subcomponents.

Each of these subcomponents has a large number of user-definable characteristics associated with it including:

Fittings A_f and A'_f :

- Fitting type (socket tee branch, butt tee branch, etc.)
- Fitting pipe size
- Fitting equivalent length
- Fitting name
- Fitting volume

Pipe A and A':

- Pipe size
- Pipe type
- Pipe inner diameter
- Pipe outer diameter
- Pipe length
- Extra pipe length
- Pipe name
- Pipe volume

Note that each section can have multiple fittings in case a design requires a series of reducing fittings. Also note that while the range of control can seem overwhelming, in automatic mode, most of these variables are selected automatically for the designer by the CFD algorithms. In this version of the software, note that the fittings are not automatically selected by the CFD algorithms.

Within the CFD Layout Manager Workspace, a single supply/return pipe pair appears in figure 11.27. Note that the workspace is on the left side of the screen and the right side contains a properties window. The properties window can be expanded as necessary to view all of the characteristics for all five subcomponents of each GHX Circuit. Note that the properties window also contains fluid dynamics results for each pipe pair. These will be reviewed later.

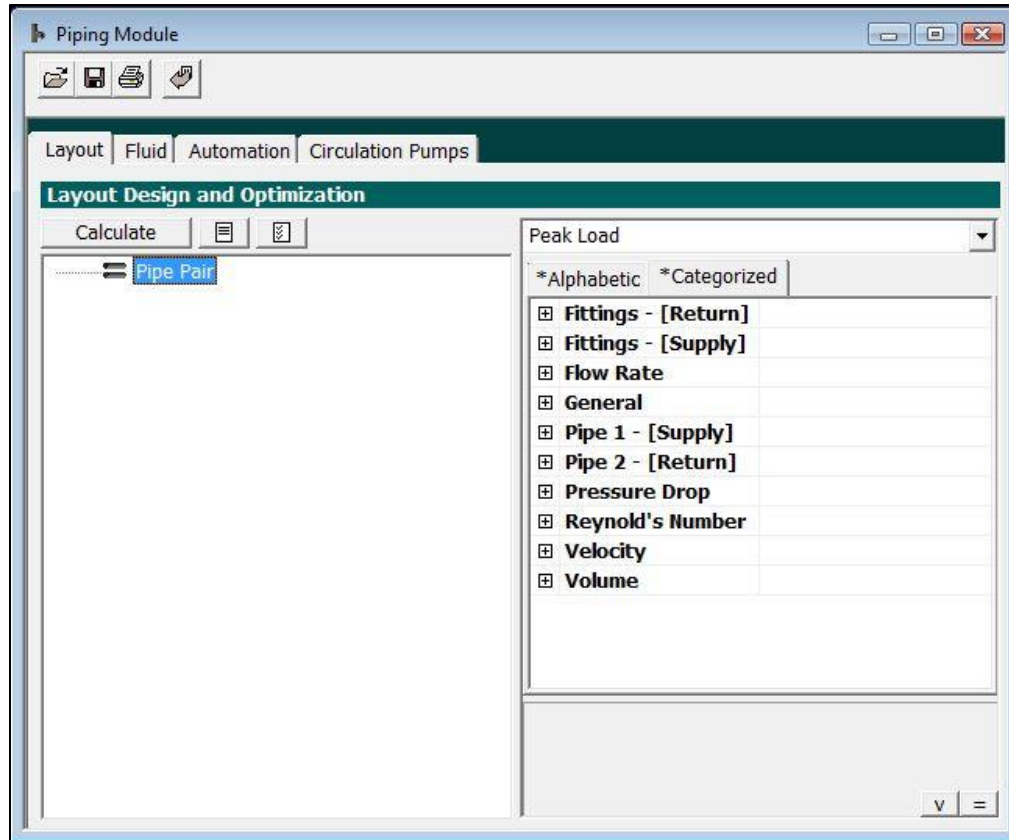


Fig. 11.27 The basic pipe pair.

Direct return piping pairs consist in the CFD module are symbolized by the following image:



Reverse return pipe pairs are symbolized by the following image:



Differences in how the CFD module models direct and reverse return piping systems are described below.

Piping Components: Summary

Each of the two basic components, the Pipe Pair and the GHX Circuit, consist of a pipe pair and two or more connection fittings. Each of these two basic components in turn are comprised of multiple sub-components. Each of these subcomponents is fully controllable by the designer. For example, in a Pipe Pair, the supply side pipe can be a different length or diameter than the return side pipe. This fine-grained control is critical for optimizing direct return and reverse return

GHX headers (for example). Also, the Pipe Pair and GHX Circuit components are not limited to two and three fittings, respectively. Users can add as many fittings as necessary to each component. This provides designers with unlimited flexibility and modeling accuracy.

Basic Piping Grammar

Now that the two basic components have been introduced, the basic piping grammar can be described. The most straightforward way of doing so is by providing a general introduction via four core concepts and then studying several basic loopfield layouts.

FOUR CORE CONCEPTS

CONCEPT ONE: Component Families

The first basic concept behind the grammar is that individual components are connected to one another piece by piece to form a cohesive and comprehensive system. As individual components are strung together they form "component families." An example of a component family is a GHX Module. Because the details of each piece in a component family are well understood, complex systems consisting of numerous "nested" component families can be analyzed effortlessly by the CFD module algorithms. The term "nested" is used because of how the component families are displayed in the Layout Manager Workspace: they appear "nested". This will become clear shortly.

CONCEPT TWO: Parent, Child and Sibling Component Relationships

Parent, Child and Sibling Definitions are as follows (note that a single component can fulfill multiple roles):

Parent: a component that has one or more directly connected "downstream" components. All components except the last component in a flow path play the role of **parent** component.

Child: a component that has a directly connected "upstream" parent component. Most **child** components (except the last one in a flow path), play the role of **parent** as well to one or more "downstream" components. Fluid flow from **parent** to **child** is in *series*.

Sibling: a component that along with one or more other components shares the same parent. Fluid flow from **parent** to two or more **siblings** is in *parallel*.

This "parent/child/sibling" nomenclature can be explained through the below figures. Figure 11.28 is a schematic drawing of a direct return three GHX Circuit GHX Module. Figure 11.29 is a screenshot of the same design in the Layout Manager Workplace with component relationships added.

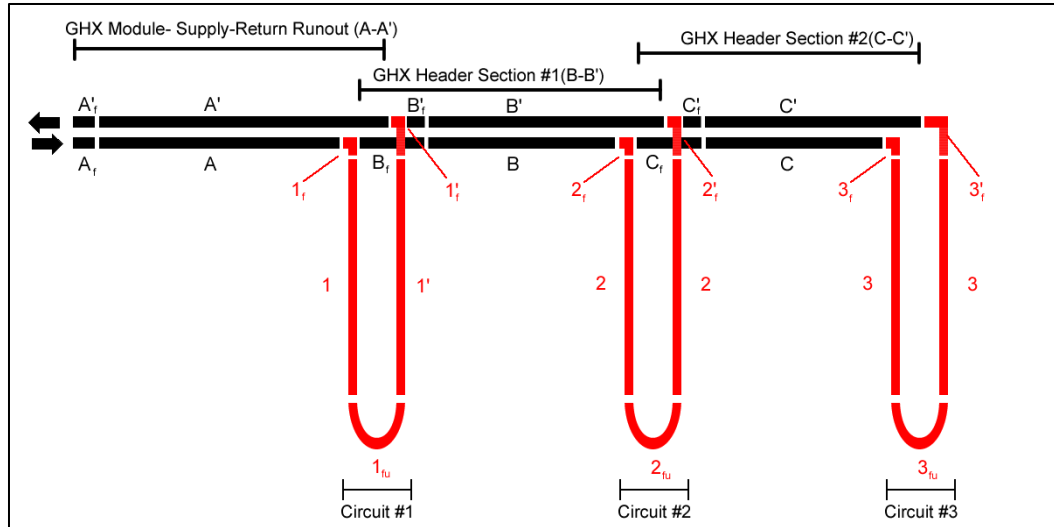


Fig. 11.28 A Sample Direct Return GHX Module For Reviewing Nomenclature

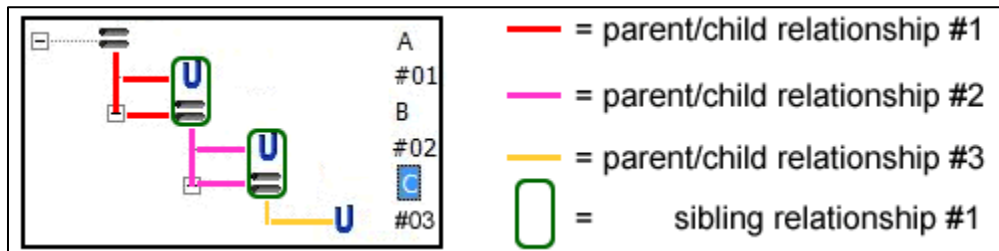


Fig. 11.29 Direct Return Component Relationships in the Layout Manager Workplace

The components in the system in figure 11.28 have the following titles which can be seen graphically in figure 11.29:

Pipe Pair A: Parent to Circuit #1 and Pipe Pair B
 Circuit #1: Child to Pipe Pair A, Sibling of Pipe Pair B
 Pipe Pair B: Child of Pipe Pair A, Sibling of Circuit #1
 Circuit #2: Child of Pipe Pair B, Sibling of Pipe Pair C
 Pipe Pair C: Child of Pipe Pair B, Sibling of Circuit #2
 Circuit #3: Child of Pipe Pair C

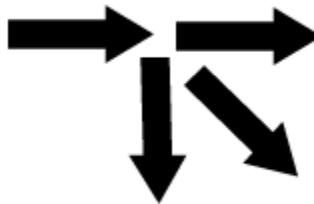
This parent/child nomenclature will be referred to from time to time throughout the rest of this manual.

Note that for reverse return systems, this nomenclature is modified (see below).

CONCEPT THREE: Parallel and Serial Flow Paths

Parallel Flow Paths

A parallel flow path is defined as one in which a flow path (and component) divides into two or more parallel flow paths (and components). Note that parallel does not mean equal. It merely means that the flow branches off in two or more directions. When a **parent** is attached to two or more **children**, the flow splits off in parallel. Visually, a parallel flow across four components can be thought of as looking like this:



Series Flow Paths

A series flow path is defined as one in which a flow path continues in one direction from one component to another component. When a **parent** has one **child**, the flow travels from **parent** to **child** in series. Visually, a series flow across two component elements can be thought of as looking like this:



As long as the designer recognizes that **parallel flow** involves three or more component elements (a **parent** and at least two **children**) and two or more flow directions and that **series flow** involves two component elements (a **parent** and a **child** for example or in the case of reverse return systems, a **sibling** and a **sibling**) and one flow direction, he or she is ready to proceed to the next section.

CONCEPT FOUR: Direct and Reverse Return GHX Headers

There are two general types of GHX Modules: those with direct return headers (figure 11.30 below) and those with reverse return headers (figure 11.34 below). GHX Header design is of critical importance because a poorly designed system will be very difficult, if not impossible, to purge properly. The two GHX header types are explained and compared below in regards to how they are represented in the CFD module.

Direct Return Systems

General Description

Direct return GHX Headers generally are easier to design, easier to build and can require less total pipe (and hence offer a lower total pressure drop) compared to reverse return GHX Headers. The return pipe of the GHX Module Supply-Return Runout *may* be shorter in the direct return case compared to the reverse return case. This is easily visualized: look at the A' return pipe of the GHX Module Supply-Return Runout in both figures 11.30 and 11.34: return pipe A' of the GHX Module Supply-Return Runout in the direct return case is much shorter.

Of course, if a reverse return GHX Header system follows the "horseshoe" approach, the length of return pipe A' of the GHX Module Supply-Return Runout in the reverse return system could be nearly the same as return pipe A' of the GHX Module Supply-Return Runout in the direct return system, thereby reducing the lower pressure drop benefit associated with direct return systems. It all depends on the particular design.

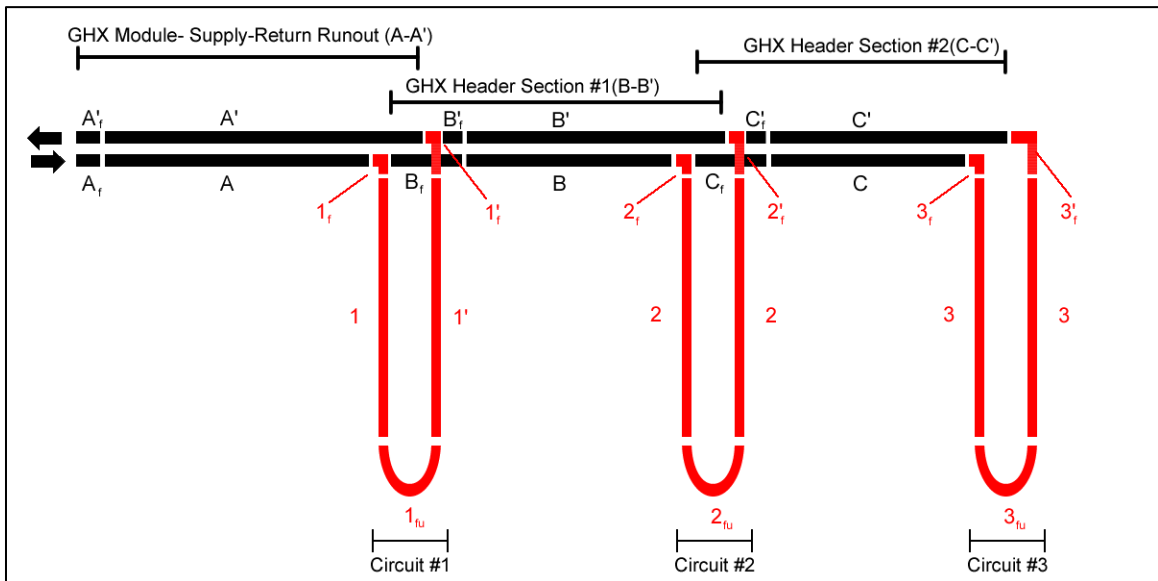


Fig. 11.30 A Direct Return GHX Module

The direct return system in the above figure 11.30 has three flow paths. The three flow paths are:

1. Fluid circulates from supply pipe A of GHX Module Supply-Return Runout AA' through Circuit #1 and back through return pipe A' of GHX Module Supply-Return Runout AA'
2. Fluid circulates from supply pipe A of GHX Module Supply-Return Runout AA' to supply pipe B of GHX Header Section BB' through Circuit #2 and

back through return pipe B' of GHX Header Section BB' and back through return pipe A' of GHX Module Supply-Return Runout AA'

3. Fluid circulates from supply pipe A of GHX Module Supply-Return Runout AA' to supply pipe B of GHX Header Section BB' to supply pipe C of GHX Header Section CC' through Circuit #3 and back through return pipe C' of GHX Header Section CC' and back through return pipe B' of GHX Header Section BB' and back through return pipe A' of GHX Module Supply-Return Runout AA'

In other words, in a direct return system, the flow paths get longer and longer as the GHX Circuits go out farther and farther. In figure 11.30 above, it is clear that a molecule of water flowing through circuit #1 travels a shorter distance and returns faster to the circulation pump than a molecule of water flowing through circuit #2 or circuit #3.

Figure 11.31 is a direct return, three GHX Circuit GHX Module in the Layout Manager Workspace that is identical to the GHX Module in figure 11.30. It shows how the Layout Manager Workspace displays direct return systems. The flow paths have been added to enhance understanding.

As can be seen in figure 11.31, direct return GHX Header sections and pipe pairs are represented by this symbol:

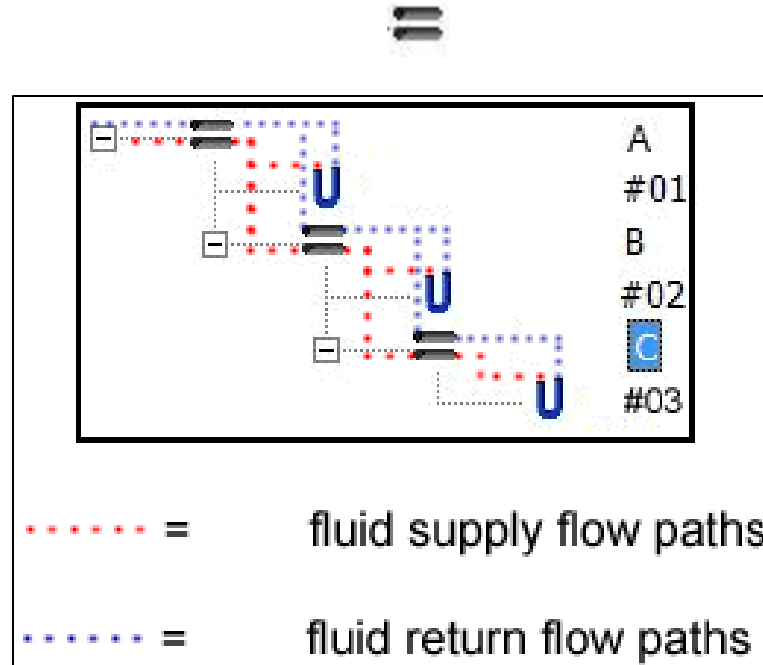


Fig. 11.31 Fluid Flow Paths of the Direct Return GHX Module

The progressively lengthening flow paths can be seen via the dotted fluid supply and return flow paths. The flow loop that reaches its end in Circuit #1 before working its

way back up to return pipe A' of the GHX Module Supply-Return Runout A is shorter than the full flow loop that ends in Circuit #2 before working its way back up through return pipe B' of the GHX Header B before finally reaching the return pipe A' of the GHX Module Supply-Return Runout A.



In summary, a simple way to remember how direct return systems model and visualize flow is as follows: the GHX Circuit (which looks like a "u") is like a U-Turn that sends the fluid flow "back up" to the top.

Design for Purging

When a direct return system is optimized for purging, the GHX Header system is composed of a series of reducing header pipe pairs. Reducing header pipe pairs maintain the flow velocity (ft/s) necessary to purge air effectively. In direct return systems GHX Header pipes reduce identically all the way down on both the supply and return side. This can be seen in figure 11.32, which is a sample auto-sized GHX Module with eight GHX circuits and reducing headers (these figures are part of the CFD Module display controls which are explained in great detail later in this chapter. For now they are included for illustrative purposes). Note that there are seven GHX Header Sections (for 8 GHX circuits which are not displayed). Figure 11.32 clearly shows that the supply side (Pipe 1) and return side (Pipe 2) pipes reduce symmetrically from GHX Header Section #1 all the way down to GHX Header Section #7: 2" pipe reduces to 1 1/2" pipe which reduces to 1 1/4" pipe and finally down to 1" pipe on the GHX Header sections.

Name	Group Name	Pipe 1 Size	Pipe 2 Size
GHX Module - Supply-Return Runout	GHX Module #01	2"	2"
GHX Header Section #01	GHX Module #01	2"	2"
GHX Header Section #02	GHX Module #01	2"	2"
GHX Header Section #03	GHX Module #01	2"	2"
GHX Header Section #04	GHX Module #01	2"	2"
GHX Header Section #05	GHX Module #01	1 1/2"	1 1/2"
GHX Header Section #06	GHX Module #01	1 1/4"	1 1/4"
GHX Header Section #07	GHX Module #01	1"	1"

Fig. 11.32 Optimized Direct Return Reducing Headers System

Name	Group Name	Pipe 1 Size	Pipe 2 Size	Pipe 1 Reynold's Number	Pipe 2 Reynold's Number
U Circuit #01	GHX Module #01	1"	1"	5801	5801
U Circuit #02	GHX Module #01	1"	1"	5486	5486
U Circuit #03	GHX Module #01	1"	1"	5246	5246
U Circuit #04	GHX Module #01	1"	1"	5072	5072
U Circuit #05	GHX Module #01	1"	1"	4956	4956
U Circuit #06	GHX Module #01	1"	1"	4756	4756
U Circuit #07	GHX Module #01	1"	1"	4570	4570
U Circuit #08	GHX Module #01	1"	1"	4403	4403

Fig. 11.33 Imbalanced Reynolds Numbers in an Optimized Direct Return Reducing Headering System

For the GHX Header system described in figure 11.32, the GHX circuits are described in figure 11.33. Notice how the Reynold's Numbers decrease from 5801 in circuit #1 down to 4403 in circuit #8, a reduction of nearly 25%. It is quite clear that the different length flow paths in direct return systems result in unbalanced systems.

Reverse Return Systems

Reverse return GHX Headers generally are more complex to design (although not with the CFD module), take more effort to build and can require more total pipe (and hence offer a higher total pressure drop) compared to direct return GHX Headers. The return pipe of the GHX Module Supply-Return Runout *may* be longer in the reverse return case compared to the direct return case.

This is easily visualized: look at return pipe A' of the GHX Module Supply-Return Runout in both figure 34 and figure 30: it is longer in the reverse return case. Of course, if a reverse return GHX Header system follows the "horseshoe" approach, the length of return pipe A' of the GHX Module Supply-Return Runout in it could be more or less the same length as return pipe A' of the GHX Module Supply-Return Runout in the direct return system. This would reduce the lower pressure drop benefit associated with shorter direct return systems. It all depends on the particular design.

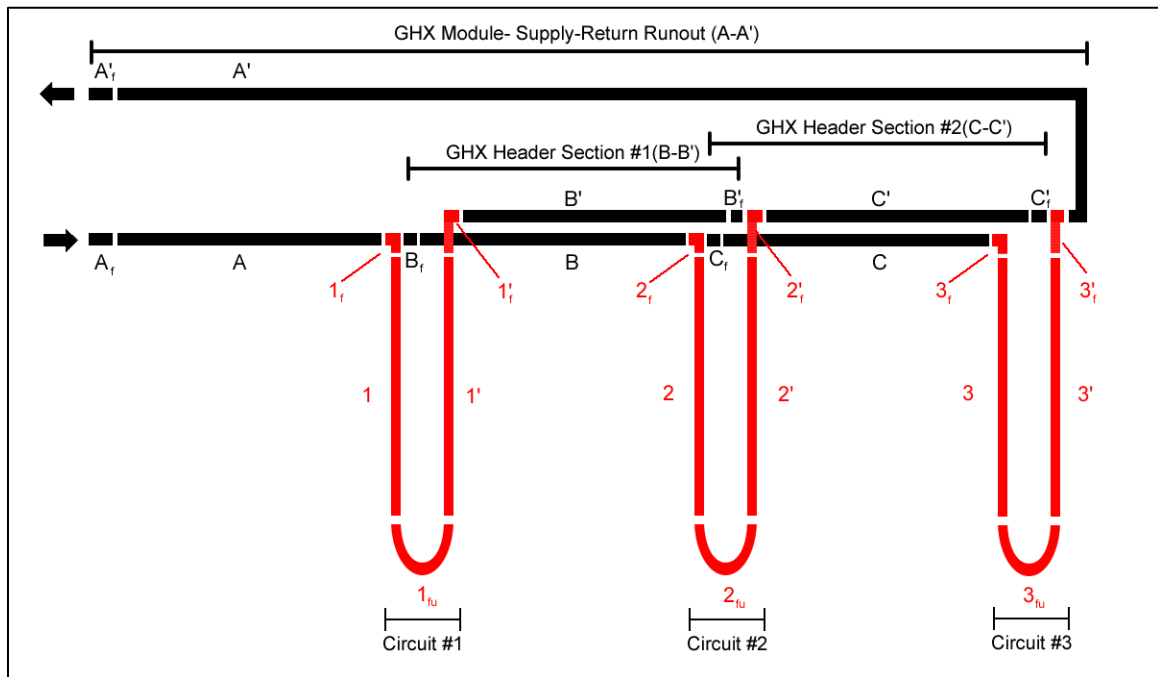


Fig. 11.34 A Reverse Return GHX Module

The reverse return system in the figure 11.34 has three flow paths. The three flow paths are:

1. Fluid circulates from supply pipe of Pipe Pair A through Circuit #1 and then continues on into the return pipe of Pipe Pair B and then into the return pipe of Pipe Pair C and finally into the return pipe of Pipe Pair A
2. Fluid circulates from supply pipe of Pipe Pair A to the supply pipe of Pipe Pair B through Circuit #2 and then continues on into the return pipe of Pipe Pair C and finally into the return pipe of Pipe Pair A
3. Fluid circulates from supply pipe of Pipe Pair A to the supply pipe of Pipe Pair B to the supply pipe of Pipe Pair C to Circuit #3 and then continues on into the return pipe of Pipe Pair A

In other words, in a reverse return system, the flow paths stay pretty much the same length for all the GHX circuits. This can be seen even in the descriptions of the three path flows above: they are all about the same length (compare this to the descriptions of the three path flows in the direct return section and notice how those get progressively longer).

In a reverse return system, the flow paths within the GHX Module section are the same length for each molecule of water, regardless of whether the molecule goes through Circuit #1 or through Circuit #3. In these systems, the return pipe of the GHX Module Supply-Return Runout (pipe A' in figure 11.34) is connected to the farthest GHX Circuit. For comparison's sake, in direct return systems (as can be seen in fig 11.30 above), the return pipe of the GHX Module Supply-Return Runout (A') is connected to the closest GHX Circuit. Reverse return systems are inherently flow balancing which has made them the standard in the geothermal industry.

Figure 11.35 is a reverse return, three GHX Circuit GHX Module in the Layout Manager Workspace that is identical to the GHX Module in figure 11.34. It shows how the Layout Manager Workspace displays direct return systems. The flow paths have been added to enhance understanding.

As can be seen in figure 11.35, reverse return GHX Header sections and pipe pairs are represented by this symbol:



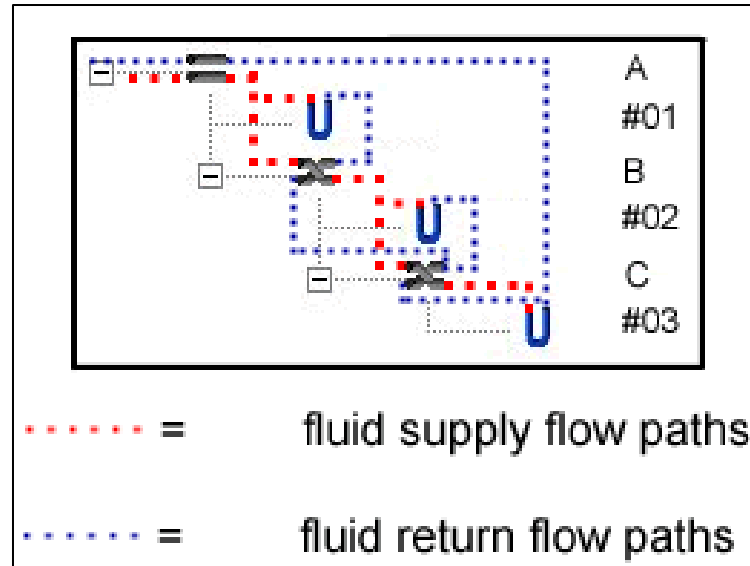


Fig. 11.35 Fluid Flow Paths of Reverse Return GHX Module

The colored and dotted fluid supply and return flow paths for the reverse return system can be seen in figure 11.35. Note that why the supply flow path identical to that in the direct return system, the return flow path is very different. Recall that in the direct return systems, the GHX Circuit (which looks like the letter "u") is like a "U Turn" that receives a "downward flowing" supply flow and shifts it into a "upward flowing" return flow (note that "down" and "up" refer to the top and bottom of figure 1135 and not to physical directions. In reverse return systems, the GHX Circuit is more like a relay that sends the flow cascading farther down all the way to the last final reverse return GHX Header section (C in figure 11.35). To explain this difference, additional reverse return-specific terminology is required.

Parent, Child and Sibling Component Relationships Revisited

The "**parent/child/sibling**" nomenclature needs to be modified and augmented for reverse return systems. While reverse return header components have **parent/child** and **sibling** relationships (like they do in direct return systems as can be seen in figure 11.29), reverse return systems have two unique additional relationships based on the same components. These can be seen in figure 11.36 and are described below.

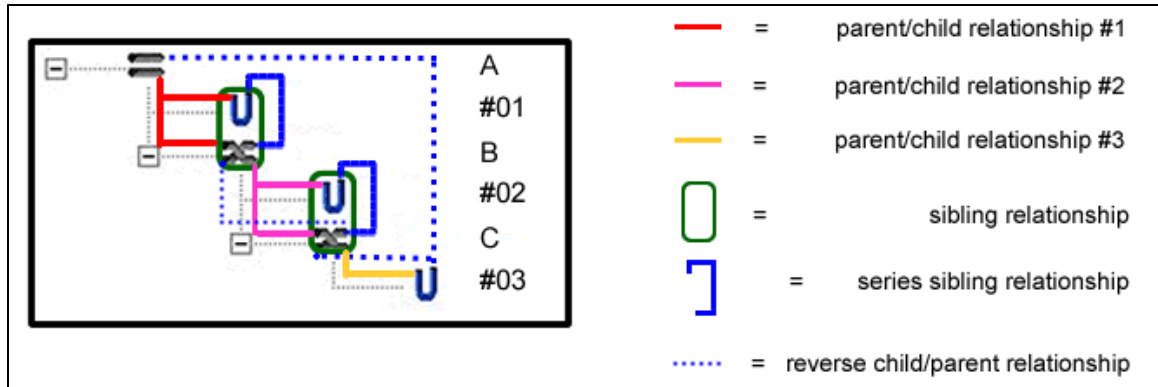


Fig. 11.36 Reverse Return Component Relationships in the Layout Manager Workspace

Series Sibling Relationships: In reverse return systems (like the one seen in diagram 11.36 above), the supply pipe of Pipe Pair A is the **parent** of both Circuit #1 and the supply pipe of Pipe Pair B. Circuit #1 and the supply pipe of Pipe Pair A are **siblings**. As such, the flow into the **siblings** from the **parent** is in parallel, just like it is in direct return systems. However, between these two **siblings** there is another flow path. This is one in which the return pipe of Circuit #1 flows into the return pipe of Pipe Pair B. In other words, even though Circuit #1 and Pipe Pair B are **siblings**, there is a series flow from one **sibling** to another. With reverse return systems, **sibling** relationships are hybrids - they have both parallel and serial flow characteristics. This relationship is called the **series sibling relationship**.

If you will recall, within the Layout Manager Workspace, siblings are *vertically stacked*.

In direct return systems *vertically stacked* siblings are always in parallel flow.



*In reverse return systems however, **vertically stacked** siblings are in both parallel and series flow.* This series flow aspect in the **series sibling relationship** is responsible for the other relationship that is unique in reverse return systems, the **reverse child-parent relationship**.

Reverse Child-Parent Relationships: In reverse return systems, fluid flows from the return pipe of one GHX Header pair to the return pipe of another GHX Header pair. This is identical in the direct return systems except that in reverse return systems, visually the return flow path is heading "down" rather than up. In the direct return systems the return flow path is heading "up". As a result, components in reverse return systems that are connected for return flow but actually flow in the "down"

direction are like the reverse of Parent-Child relationships. Hence, they are termed "**reverse child-parent relationships.**"

In the Layout Manager Workspace, these reverse return child-parent relationships become apparent in figure 11.35. Notice how the supply and return flows are more or less in parallel and in the same direction ("down"). Only at the last GHX Circuit does the return flow actually begin flowing in the return ("up") direction to return pipe A' of the GHX Module Supply-Return Runout.

Again, this can be seen in figure 11.34 in which the return pipes B' and C' of the GHX Headers flow in parallel with the "supply" flow in supply pipes B and C until the last GHX Circuit #3 at which point the return flow reverses course and flows through the return pipe A' of the GHX Module Supply-Return runout and heads into the return direction.

A straightforward way to think about how CFD Module models reverse return systems is as follows:

TIP

In reverse return systems, GHX Circuits are like relays that send the flow farther down the GHX Module. It is only at the last GHX Circuit, where the flow heads back up to where it started.

Remember that direct returns are different:

In the direct return systems, the GHX Circuit is like a U turn that sends the flow back up to where it started.

Design for Purging

When a reverse return system is optimized for purging, the GHX Header system is composed of a series reducing header pipe pairs. Reducing header pipe pairs maintain the flow velocity (ft/s) necessary to purge air effectively. Reverse return systems are very different from direct return systems when it comes to the design of the GHX Header reductions. GHX Header pipes reduce all the way down on the supply following a calculated optimal pipe reduction profile. However, on the return side the pipe reduction is reversed. Indeed, on the reverse side, pipe diameters increase in size as the return pipes of the GHX Header get closer and closer to the return pipe of the Supply-Return Runout.

This can be seen via an example in figure 11.37 which is a sample auto-sized, reverse return GHX Module with eight GHX circuits and reducing headers (this figures are part of the CFD Module display controls and are explained in great detail later in this chapter. For now they are included for illustrative purposes). Notice how the pipe sizes reduce down from 2' all the way to 3/4" on the pipe 1 (supply) side of the GHX Header system. On the pipe 2 (return) side of the system, the pipe sizes *expand* in

TIP

diameter as they get closer and closer to the return pipe of the GHX Module Supply-Return Runout. In this example, the supply (pipe 1) and return (pipe 2) sides of the GHX Header system are palindromes. Optimized reverse return systems also can be called "Palindromic Reverse Returns."









Name	Group Name	Pipe 1 Size	Pipe 2 Size
 GHX Module - Supply-Return Runout	GHX Module #01	2"	2"
 GHX Header Section #01	GHX Module #01	2"	3/4"
 GHX Header Section #02	GHX Module #01	2"	1 1/4"
 GHX Header Section #03	GHX Module #01	2"	1 1/2"
 GHX Header Section #04	GHX Module #01	2"	2"
 GHX Header Section #05	GHX Module #01	1 1/2"	2"
 GHX Header Section #06	GHX Module #01	1 1/4"	2"
 GHX Header Section #07	GHX Module #01	3/4"	2"

Fig. 11.37 Optimized Direct Return Reducing Headers System

For comparison, in a direct return system, the header pipes reduce identically all the way down the cascade on both the supply and return side as can be seen above in figure 11.32



Name	Group Name	Pipe 1 Size	Pipe 2 Size	Pipe 1 Reynold's Number	Pipe 2 Reynold's Number
 Circuit #01	GHX Module #01	1"	1"	4604	4604
 Circuit #02	GHX Module #01	1"	1"	5123	5123
 Circuit #03	GHX Module #01	1"	1"	5188	5188
 Circuit #04	GHX Module #01	1"	1"	5229	5229
 Circuit #05	GHX Module #01	1"	1"	5229	5229
 Circuit #06	GHX Module #01	1"	1"	5188	5188
 Circuit #07	GHX Module #01	1"	1"	5123	5123
 Circuit #08	GHX Module #01	1"	1"	4604	4604

Fig. 11.38 Balanced Reynolds Numbers in an Optimized Reverse Return Reducing Headering System

As mentioned previously, reverse return GHX Header systems are inherently flow balanced. This can be seen in figure 11.38 above, which shows the eight GHX Circuits that branch off of the seven GHX Header Sections in figure 11.37 above. Notice how the Reynold's Number drop off symmetrically from the central GHX Circuits (Circuits #4 and #5). Also note how the Reynold's Numbers vary only by 12% between the center (Circuits #4 and #5) and outer (Circuits #1 and #8) GHX Circuits. Compare this to the 25% difference in Reynold's Numbers in the direct return case above and it becomes clear that reverse return systems provide significant flow balancing benefits (notice that the direct and reverse return designs, fluid types and flow rates are identical in all regards except for the return piping style and therefore the calculated difference is a valid theoretical result).

Modeling reverse return GHX Header systems mathematically in a non-trivial task. As a result, reverse return GHX Header systems in the CFD module in the current version of GLD have certain requirements including:

- Reverse return systems must include *at least* two reverse return pipe pairs and three GHX Circuit in one nested family of components

Remembering this reverse return requirements will enable a designer to design more quickly.

Now that the four core concepts have been reviewed - component families, component relationships, parallel and series flow and direct/reverse returns, we will examine five loopfield designs. Four are direct return and one is reverse return.

SAMPLE LOOPFIELD LAYOUTS

BASIC DIRECT RETURN LOOPFIELD LAYOUT 1

Figure 11.39 is an illustration of a direct return, two GHX Circuit GHX Module. The Supply-Return Runout, GHX header section and their associated fittings are in black. The GHX Circuits and their associated fittings are in red. *Note that pipe and fitting lengths within a single component do now have to be the same length. For example, the return pipe of the Supply-Return Runout (A') is longer than the supply pipe of the Supply-Return Runout (A).*

Between each connection a space has been added to visibly separate different sections of the system for easy comparisons with the layout structure in the CFD module. Figure 11.40 is the identical layout in the CFD module.

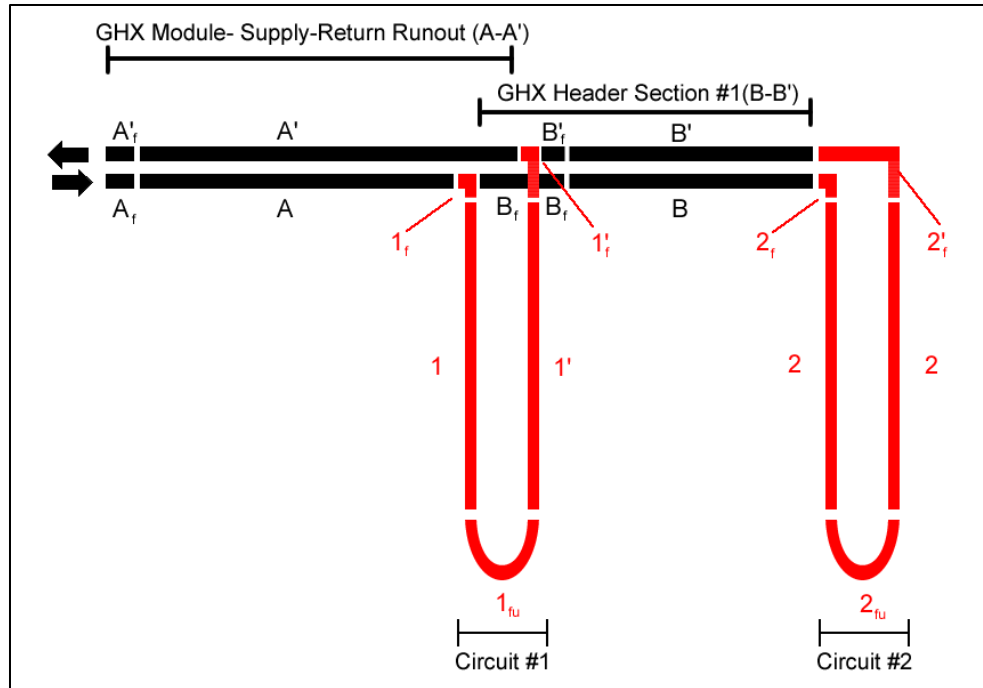


Fig. 11.39 Basic Direct Return Loopfield Layout 1

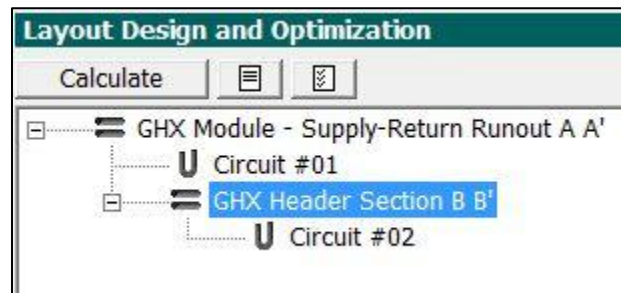


Fig. 11.40 Basic Direct Return Loopfield Layout 1 in Layout Manager Workspace

Note that in figure 11.39, the layout consists of a combination of the two components: the pipe pair and the GHX Circuit. Also note that for both circuits 1 and 2, the fittings for attachment to the return header pipes (1'_r and 2'_r) are longer than those attached to the supply header pipes (1_f and 2_f). There is no requirement that each subcomponent in a particular component has a uniform length. Indeed, the designer has as much control as he or she desires.

As introduced before, it is important to understand how parallel and serial flow paths are displayed in the Layout Manager Workspace.

Parallel Flow Paths



In figure 11.39, a parallel flow path occurs where supply pipe A of the GHX Module Supply-Return Runout branches into two components: Circuit #1 and supply pipe B of GHX Header Section BB'. The fluid flows in Circuit #1 and supply pipe B of GHX Header Section BB' therefore are in parallel. Circuit #1 and supply pipe B of GHX Header Section BB' are **siblings** because they share the same parent. In the CFD Layout Manager Workspace, parallel flow paths or siblings are **vertically stacked** directly above one another. This can be seen in figure 11.40 where Circuit #1 is directly above GHX Header Section BB'.

Series Flow Paths



In figure 11.39 a series flow path occurs where supply pipe B of GHX Header Section BB' continues into Circuit #2. The fluid flow from supply pipe B of GHX Header Section BB' to Circuit #2 is therefore in series. Supply pipe B of GHX Header Section BB' is the **parent** of Circuit #2 and Circuit #2 is the **child** of supply pipe B of GHX Header Section BB'. In the CFD Layout Manager workspace, serial flow paths or parent/child relationships are **stacked with indentation**. This can be seen in figure 11.40 where Circuit #2 is one level below the GHX Header Section B and indented. What this means is that Circuit #2 is connected to GHX Header Section B in series.

As long as the designer recognizes that **parallel flow** involves three or more component elements and two or more flow directions and that **series flow** involves two component elements and one flow direction, he or she is ready to proceed to the next section.

To solidify our understanding of how the Layout Manager Workspace diagrams direct return systems, we will follow the fluid flow in figure 11.40. Because there are two GHX Circuits, there are two major flow paths. The flow path(s) are as follows :

(note that supply flowpaths use this symbol (►) and return flowpaths use this symbol (◄)).

The first flow path:

- Fluid flows from supply pipe A of GHX Module Supply-Return Runout into Circuit #1
- ◄ Fluid flows from Circuit #1 into return pipe A' of GHX Module Supply-Return Runout

The second flow path:

- Fluid flows from supply pipe A of GHX Module Supply-Return Runout into supply pipe B of GHX Header
- Fluid flows from supply pipe B of GHX Header into Circuit #2
- ◄ Fluid flows from Circuit #2 into return pipe B' of GHX Header
- ◄ Fluid flows from return pipe B' of GHX Header into return pipe A' of GHX Module Supply-Return Runout

BASIC DIRECT RETURN LOOPFIELD LAYOUT 2

Figure 11.41 is an illustration of a four circuit, two "circuit per parallel loop" direct return GHX Module. While the figure looks somewhat complex because each individual piece is labeled/numbered, it will soon become clear that in the CFD module, the layout is quite straightforward. The GHX Module Supply-Return Runout, the GHX Header pipe pairs and their associated fittings are in black. The GHX Circuits and their associated fittings are in red. Between each connection a space has been added to visibly separate different sections of the system for easy comparisons with the layout structure in the CFD module. To ensure clarity, each individual component in the figure is listed below:

Supply/Return Pipe Pairs:	GHX Circuits
A _f -A-A'-A' _f	1 _f -1-1 _{fu} -1'-1' _f
B _f -B-B'-B' _f	2 _f -2-2 _{fu} -2'-2' _f
C _f -C-C'-C' _f	3 _f -3-3 _{fu} -3'-3' _f
D _f -D-D'-D' _f	4 _f -4-4 _{fu} -4'-4' _f

Note that each supply/return pipe pair consists of four subcomponents and each GHX Circuit consists of five subcomponent as mentioned above in the basic description of the two components (the pipe pair and the GHX Circuit).

Figure 11.42 is the identical layout in the CFD module.

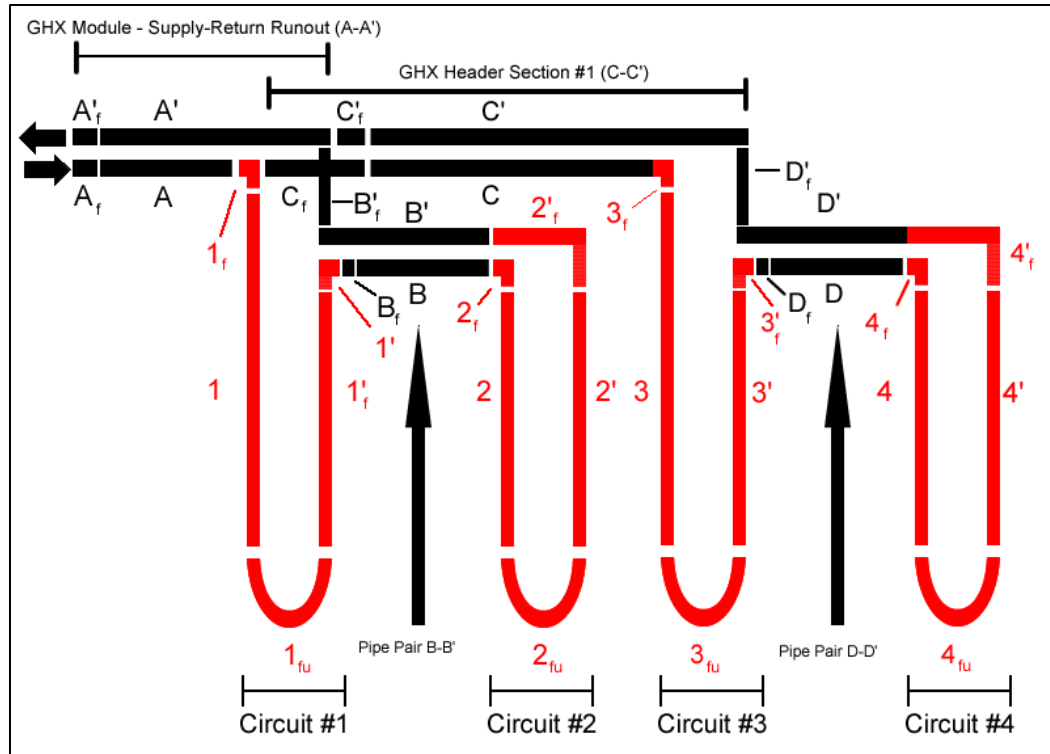


Fig. 11.41 Basic Direct Return Loopfield Layout 2

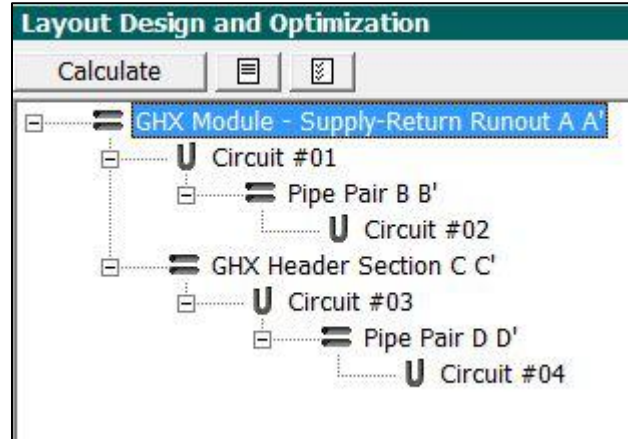


Fig. 11.42 Basic Direct Return Loopfield Layout 2 in Layout Manager Workspace

Note that although figure 11.41 looks somewhat complicated, the layout consists of nothing more than a combination of the two components: the Pipe Pair and the GHX Circuit. In this example however, the two components are hooked up in a different way: two GHX Circuits in series (Circuit #1 and Circuit #2 for example) are connected to each other by supply pipe B of Pipe Pair BB'. The return pipe B' of Pipe Pair BB' brings the entire series of two parallel circuits back into return pipe A' of the GHX Module Supply Return Runout AA'.

Can you find the parallel flow paths in figure 11.42? Remember- parallel flow paths are *vertically stacked and have one parent and at least two children (or at least two siblings looking at it from the child's perspective)*. This means that Circuit #1 and the supply pipe C of GHX Header Section CC' are parallel flow paths. Circuit #1 and supply pipe C of GHX Header Section CC' are **siblings** and share supply pipe A of the GHX Module Supply-Return Runout AA' as a **parent**. This becomes very clear in figure 11.41 where the flow branches from supply pipe A of the GHX Module Supply-Return Runout Pipe AA' and into Circuit #1 and supply pipe C of the GHX Header Section CC'.

After looking at figure 11.42, one might ask if pipe pair BB' and Circuit #3 are in parallel/are siblings as well since they are vertically stacked. The answer is no. Remember that a parallel flow path is defined as one in which a flow path (and component) divides into two or more parallel flow paths (and components). Pipe pair BB' and circuit #3, although vertically stacked, do not branch out from the same predecessor component. Therefore they cannot be siblings and cannot be in parallel.

Can you find the serial flow flows in figure 11.42? Remember- serial flow paths are *stacked with indentation and each parent can have only one child*. This means that supply pipe A of the GHX Module Supply-Return Runout AA' - Circuit #1 - supply pipe B of Pipe Pair BB' - Circuit #2 are in series since they are *stacked with indentation and each component has only one child and one parent*. Can you describe the other major series flow path in figure 11.42? It is supply pipe A of the GHX Module Supply-Return Runout AA'- supply pipe C of the GHX Header Section CC'- Circuit #3- supply pipe D of Pipe Pair DD'- Circuit #4. Hopefully by now you are feeling comfortable with the Layout Manager workspace and how it displays parallel and series flow for a variety of conditions. We will look at two more basic direct return examples to solidify our understanding.

It is important to note that the visual "grammar" that the CFD module uses is not to scale. The graphics used to describe the pipes and their relationships are identical in size even if the underlying pipe properties are different. For example, in figure 31, circuit #1 could be 100 ft deep and circuit #2 could be 200 feet deep.

BASIC DIRECT RETURN LOOPFIELD LAYOUT 3

Figure 11.43 is the layout of a four circuit, two GHX "circuits per bore" GHX Module. The two "circuits per bore" (or double U tubes according to some nomenclature) are in parallel (they are *vertically stacked*).

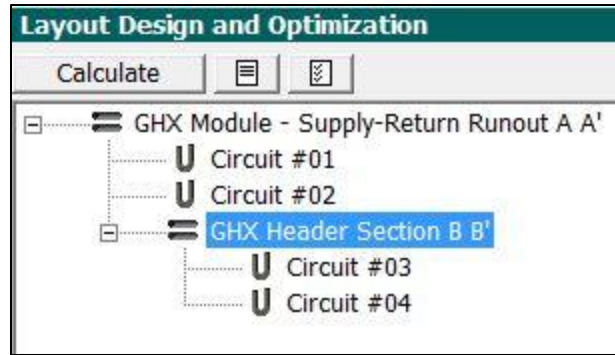


Fig. 11.43 Basic Direct Return Loopfield Layout 3 in Layout Manager Workspace

Can you find the parallel flow paths in figure 11.43? Remember- parallel flow paths are **vertically stacked**. The following paths are in parallel:

- Circuit #1 and Circuit #2 are in parallel coming out of supply pipe A of the GHX Module Supply Return Runout AA' and returning to return pipe A' of the GHX Module Supply-Return Runout AA'. Circuit #1 and Circuit #2 are **siblings** that share the GHX Module Supply-Return Runout AA' as a **parent**.
- Similarly, Circuit #3 and Circuit #4 are in parallel coming out of supply pipe B of the GHX Header Section BB' and returning to the return pipe B' of the GHX Header Section BB'.
- Circuit #1, Circuit #2 and supply pipe B of the GHX Header Section BB' (for this parallel flow path system, flow comes from supply pipe A of the GHX Module - Supply-Return Runout and then branches in three directions to Circuit #1, Circuit #2 and supply pipe B of the GHX Header Section. Remember that parallel flow means that the flow branches off in two or more directions. In this case, flow is branching off in three directions. Three **siblings** (Circuit #1, Circuit #2 and supply pipe B of the GHX Header Section BB') share the parent supply pipe A of the GHX Module Reverse Return Runout. The **siblings** are **vertically stacked**.

Can you find the serial flow paths in figure 11.43? Remember- serial flow paths are **stacked with indentation**. The following paths are in series:

- Supply pipe A of the GHX Module Supply-Return Runout AA' -Circuit #1
- Supply pipe A of the GHX Module Supply-Return Runout AA'-Circuit #2
- Supply pipe A of the GHX Module Supply-Return Runout AA' - Supply Pipe B of GHX Header Section BB'-Circuit #3
- Supply pipe A of the GHX Module Supply-Return Runout AA' - Supply Pipe B of GHX Header Section BB'-Circuit #4

Each of these flow paths consists of a linear series of **parent-child** connections.

BASIC DIRECT RETURN LOOPFIELD LAYOUT 4

Figure 11.44 is the layout for another four circuit, two "GHX circuits per bore" GHX Module. The two "circuits per bore" (or double U tubes according to some nomenclature) are in series.

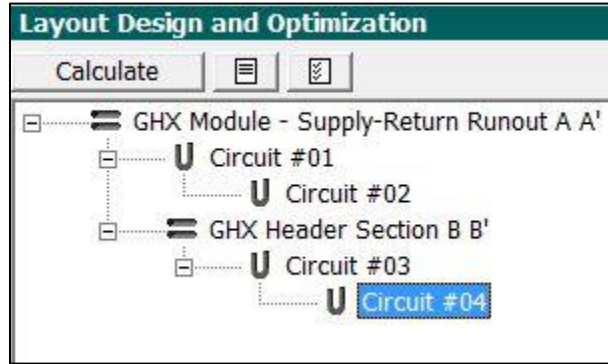


Fig. 11.44 Basic Direct Return Loopfield Layout 4 in Layout Manager Workspace

Can you find the parallel flow paths in figure 11.44? Remember- parallel flow paths are **vertically stacked**. The following paths are in parallel:

- Circuit #1 and supply pipe B of the GHX Header Section BB' are in parallel as they both come out of supply pipe A of the GHX Module-Supply-Return Runout AA'. Circuit #1 and supply pipe B of the GHX Header Section BB' are **vertically stacked**, are **siblings** and share the **parent** supply pipe A of the GHX Module- Supply-Return Runout AA'.

Can you find the serial flow paths in figure 11.44? Remember- serial flow paths are **stacked with indentation**. The following paths are in series:

- Supply pipe A of the GHX Module- Supply-Return Runout AA' -Circuit #1- Circuit #2
- Supply pipe A of the GHX Module- Supply-Return Runout AA' -supply pipe B of the GHX Header Section BB'- Circuit #3- Circuit #4

BASIC REVERSE RETURN LOOPFIELD LAYOUT 1

Reverse return systems and how they are modeled in the CFD Module were introduced before. Key features are presented again here as a review.

Reverse return systems in the CFD Module have some special features and requirements which are described below.



- Reverse return systems must include *at least* two reverse return pipe pairs and three GHX Circuit in one nested family of components.
- Reverse return GHX Module systems do not follow the standard layout formalism that was presented for direct return systems. Recall that with direct return systems, when the GHX Circuit returns to its piping GHX Header section, it returns to its parent piping GHX Header section that is one level above it and left justified. This can be seen clearly in figure 11.31. Although the reverse return pairs themselves do represent matched supply and return header sections and the circuits and fittings are standard, the GHX Circuit, when it returns to its piping header section, it returns to the piping header component that is ***directly beneath it***. This is called the **series sibling relationship** as described above.

To counteract this discrepancy and maintain flow consistency, the final GHX Circuit at the end of the system links back directly to the return pipe of the GHX Module Supply-Return Runout. This is best illustrated through an example. This can be seen in figure 11.45 where the final GHX Circuit (Circuit #3) links to the return pipe A' of the GHX Module Supply-Return Runout AA'.

- On the supply pipe side, fittings come before the pipe (see A_f and A in figure 11.34 above. On the return pipe side, fittings come after the pipe (see C' and C'_f in figure 11.34 above.) This same formalism applies to both direct and reverse return systems: in general, supply side fittings come before the supply side pipe and return side fittings come after the return side pipe. In the direct return example, this can be seen in figure 11.28 where the return side fitting B'_f comes after pipe B'.
- The GHX Module Supply-Return Runout does not use the reverse return symbol because the reverse return system technically begins with the first GHX Circuit and ends with the last GHX Circuit.

To understand more clearly how the CFD Module displays reverse return systems, an example will be explored. Figure 11.45 is a three GHX Circuit reverse return system. Figure 11.46 is an example of how the system is modeled in the Layout Manager Workspace.

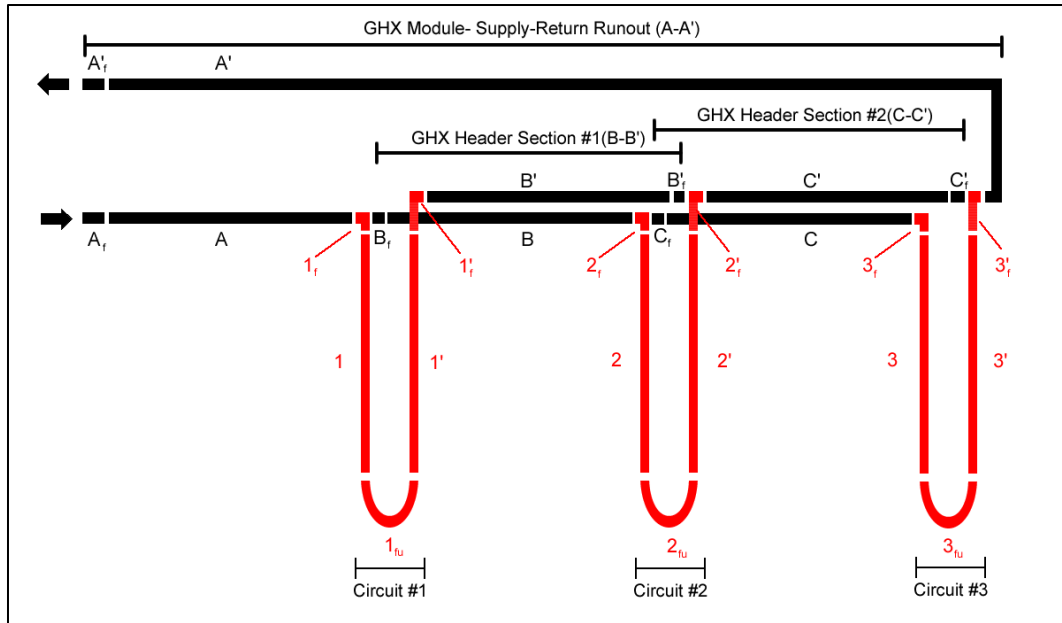


Fig. 11.45 Basic Reverse Return Loopfield

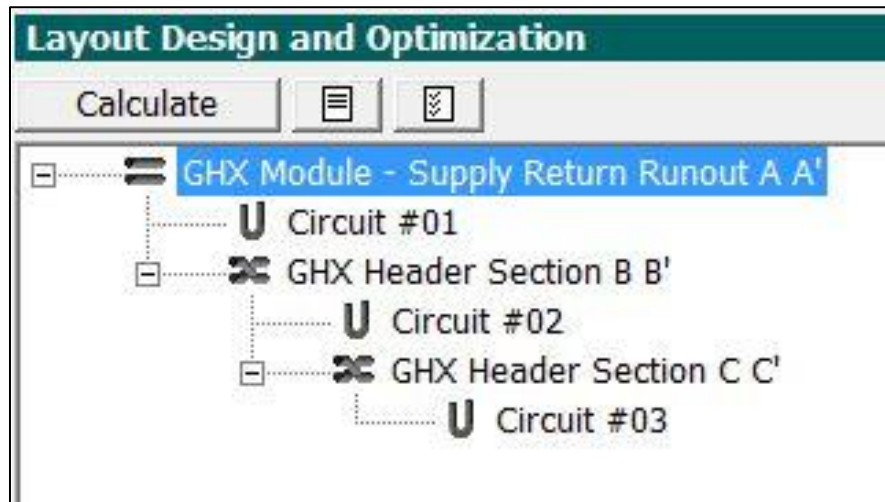


Fig. 11.46 Basic Reverse Return Loopfield in Layout Manager Workspace

Can you find the parallel flow paths in figure 11.46? Remember- parallel flow paths are **vertically stacked**. The following paths are in parallel:

- Circuit #1 and supply pipe B of GHX Header Section BB' are in parallel coming out of supply pipe A of the GHX Module Supply Return Runout AA'. Circuit #1 and supply pipe B of GHX Header Section BB' are **siblings** that share the supply pipe A of GHX Module Supply-Return Runout AA' as a **parent**.
- Circuit #2 and supply pipe C of GHX Header Section CC' are in parallel coming out of supply pipe B of the GHX Header Section BB'. Circuit #2

and supply pipe C of GHX Header Section CC' are **siblings** that share the supply pipe B of the GHX Header Section BB' as a **parent**.

Can you find the serial flow paths in figure 11.46? Remember- serial flow paths are *stacked with indentation and for reverse return systems, can be found in the series sibling relationships*. The following paths are in series:

- Supply pipe A of the GHX Module Supply-Return Runout AA' -Circuit #1
- Supply pipe A of the GHX Module Supply-Return Runout AA' - Supply Pipe B of GHX Header Section BB'-Circuit #2
- Supply pipe A of the GHX Module Supply-Return Runout AA' - Supply Pipe B of GHX Header Section BB'-Circuit #2-Supply Pipe C of the GHX Header Section CC'-Circuit #3
- Return Pipe of Circuit #1-Return pipe B' of GHX Header Section BB'-Return pipe C' of GHX Header Section CC'-Return pipe A' of the GHX Module Supply-Return Runout AA'
- Return Pipe of Circuit #2-Return pipe C' of GHX Header Section CC'- -Return pipe A' of the GHX Module Supply-Return Runout AA'
- Return Pipe of Circuit #3- Return pipe A' of the GHX Module Supply-Return Runout AA'

When looking at the details, reverse return system are quite complicated. Luckily does not need to remember much of this. Indeed, the intuitive systems employed by the CFD Module make it quite easy to build piping systems, perform simulations and review results.

Now we will learn how to build piping systems in the Layout Manager Workspace.

Building Piping Systems

In this section we will explore how to build a GHX Field using both manual and automatic tools and techniques. A number of these tools and techniques can be utilized in both the manual and automatic design modes. Rather than introduce the tools twice in both the manual methods subsection and the automatic methods subsection, many are described only in the manual methods subsection. Therefore, designers that intend to use only the automatic methods still will benefit from reading the entire section.

Manual Methods

The CFD module offers a range of techniques and tool for the designer who desires to build manually a piping system from the ground up. These techniques and tools include how to:

- Add a new pipe pair
- Add a new GHX Circuit
- Drag and drop pipe pairs and circuits
- Copy and paste pipe pairs and circuits
- Hide and display nested component families
- Delete pipe pairs and circuits
- Modify parameters with the Properties Window
- Modify parameters with the Pipe and Fitting Manager

Each of these will be explored in detail below.

Adding a New Pipe Pair

Adding a new pipe pair is typically the first step a designer will take when manually designing a system. He or she can do so from the *Layout Panel*. On the panel, the user can move the mouse into the Layout Manager Workspace (see figure 11.16) and then *right click* to bring up window as see in figure 11.47.

Note that the reverse return pipe pair option is inaccessible at this preliminary design stage. Manually building a reverse return flow system requires that a standard pipe pair act as a parent to the first reverse return pipe pair. Reverse return systems are described in some detail above.

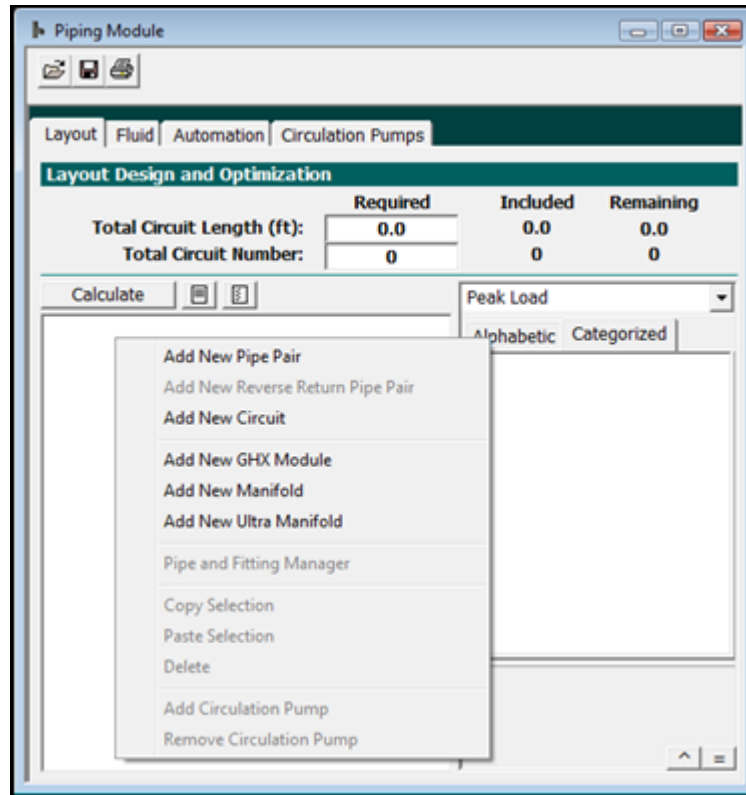


Fig. 11.47 Manually Adding a Pipe Pair

The user can select '*Add New Pipe Pair*' and then a pipe pair will appear at the top of the Layout Manager Workspace as can be seen in figure 11.48.

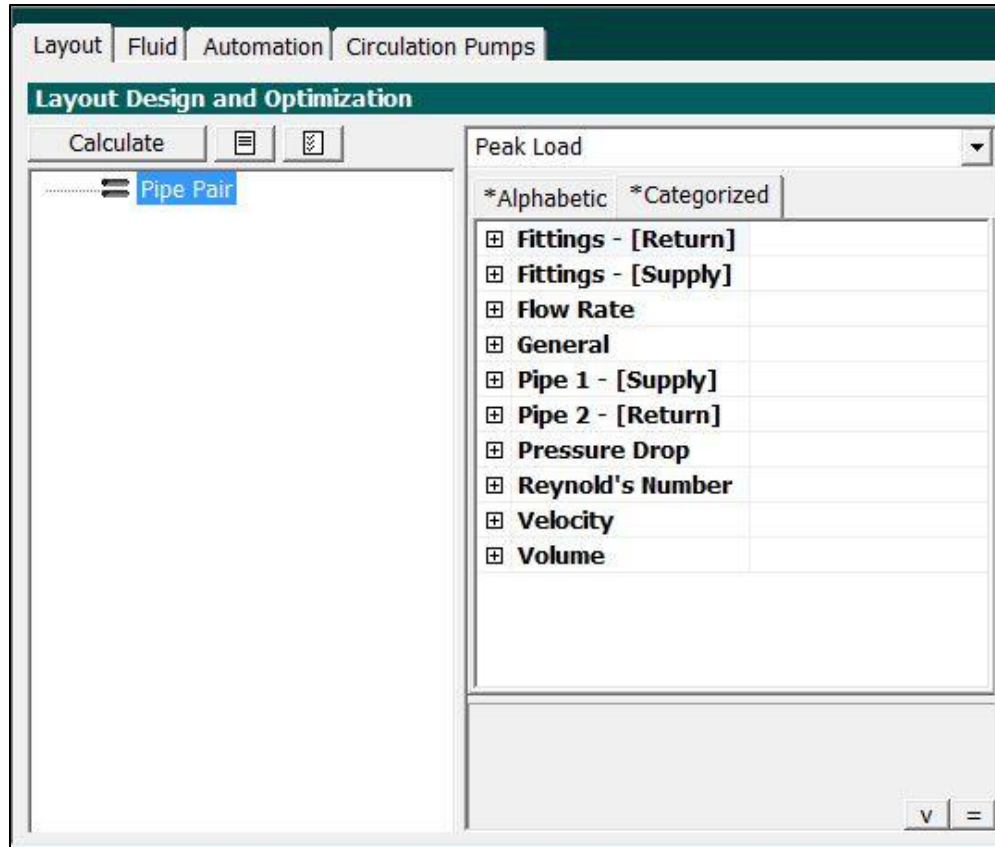


Fig. 11.48 A New Pipe Pair Component Has Been Added

The user can proceed to add another pipe pair by repeating the process. The result will look like figure 11.49

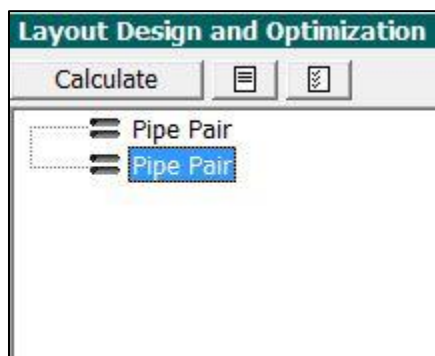


Fig. 11.49 Manually Adding a Second Pipe Pair

Adding a New GHX Circuit

Following the methods outlined for adding new pipe pairs, the user can add a new GHX Circuit independent of other piping components as can be seen in figure 11.50.

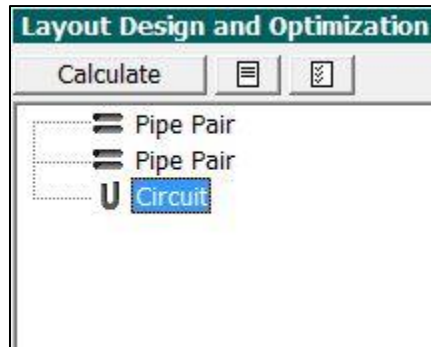


Fig. 11.50 Manually Adding a GHX Circuit

Conversely, if a user wishes to add a new GHX Circuit as a "child" of another piping component he can do so by moving the mouse over the "parent" component of interest, right clicking and then adding a new circuit. The result of such of an effort can be seen in figure 11.51.

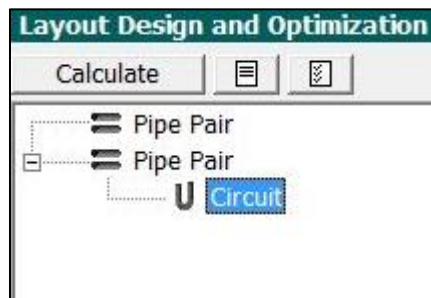


Fig. 11.51 Manually Adding a GHX Circuit As a Child to the Pipe Pair

Dragging and Dropping Pipe Pairs and Circuits

After having added components into the Layout Design Manager, users can quickly move components or nested families of components via the standard drag and drop methodology. For example, the user can select the GHX Circuit in figure 11.50 and then drag it onto the second pipe pair so that it looks like figure 11.51.

The user can then select the second pipe pair (and in doing so select all of the "child" components which in this case consist of only the circuit) and drag and drop the entire nested component family onto the first pipe pair. The result can be seen in figure 11.52.

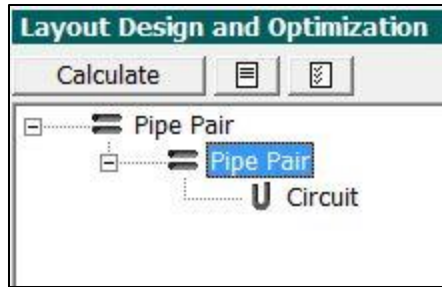


Fig. 11.52 The New Nested Component Family System After Dragging and Dropping the Second Pipe Pair/GHX Circuit Nested Component Family onto the First Pipe Pair.

The drag and drop process is very flexible and enables designers to quickly design and adjust systems.

Copying and Pasting Pipe Pairs and Circuits

A user also has the option of copying and pasting individual components, nested families of components or partial nested families of components. For example,

In figure 11.53, the user can right click on the circuit and choose *Copy*.

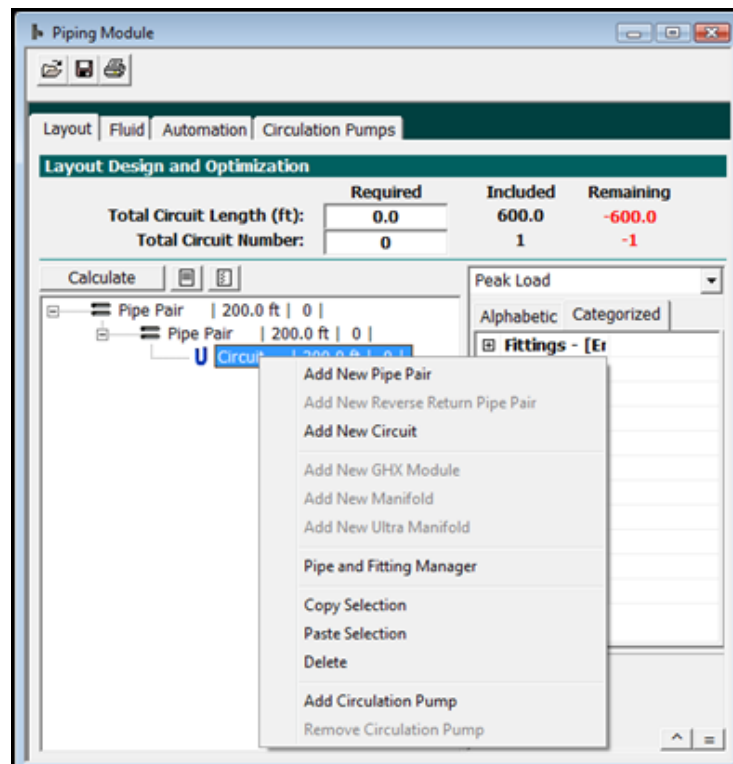


Fig. 11.53 Right Click on the Component of Interest and Choose 'Copy Selection'

After doing so, the user can paste the circuit in a variety of places in the tree: either as a child in a preexisting component/nested component family (figures 11.54 and 11.55) or as an independent circuit (figure 11.56).

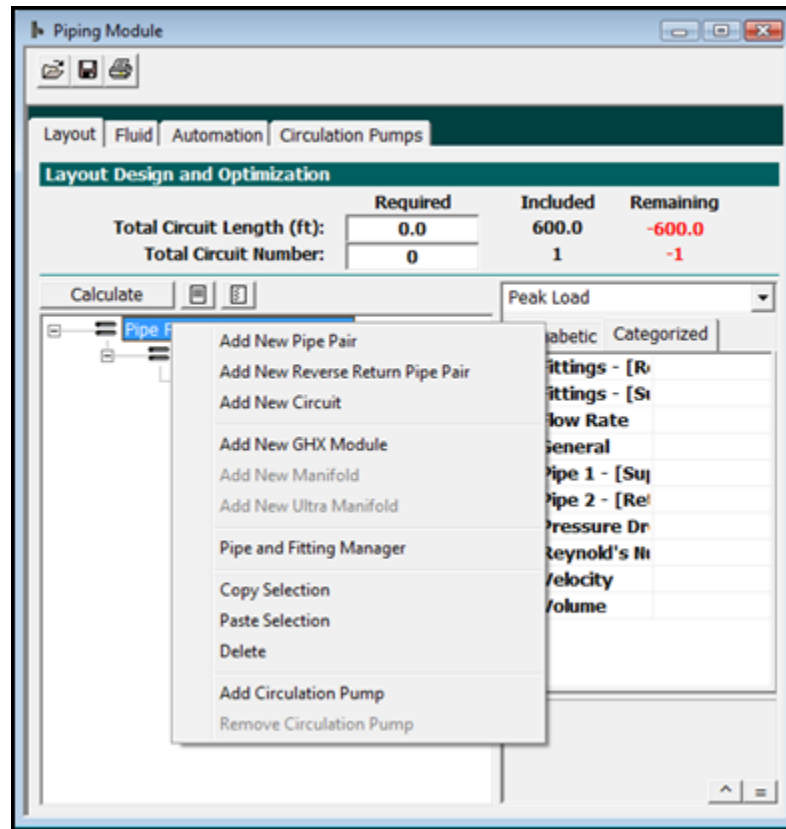


Fig. 11.54 Select the Component of Interest, Right Click and Choose Paste Selection

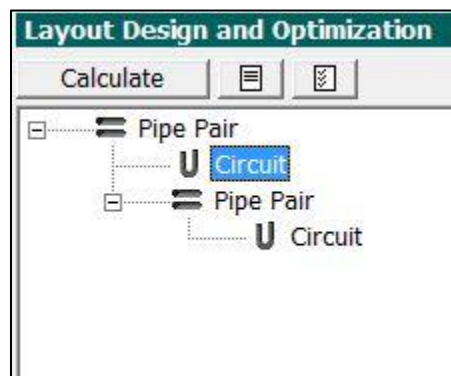


Fig. 11.55 The Circuit Has Been Copied and Pasted as a New Child to a Parent Pipe Pair

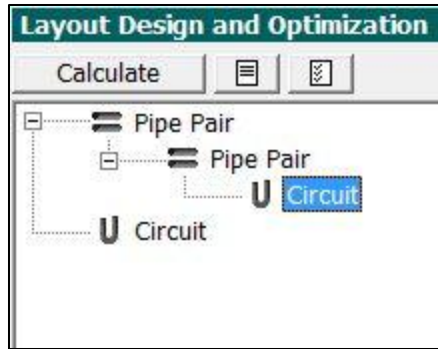


Fig. 11.56 The Circuit Has Been Copied and Pasted As An Independent "Parent"

Note that copying and pasting is not limited to individual components. Entire nested component families and partial nested component families can be copied and pasted as well. This is very convenient if a designer spends some time building (for example) a GHX Module with 10 GHX Circuits and wants to have several of these GHX Modules. After the designer finishes one, all he or she has to do is select the component at the top of the GHX Module (the top "parent" component) and then copy and paste. The entire GHX Module will be instantly replicated.

Hiding and Displaying Nested Component Families

Nested component families consist of two or more components that are in parent/child relationships. Figure 11.57 is an example of a nested component family. The nested components combine to form one GHX Module made up of 8 GHX Circuits, seven GHX Header sections and one GHX Module Supply-Return Runout. As can be seen through the GHX Module, there are a number of boxes with minus (-) signs. The user can click on any of these boxes to hide all the components that are children of the selected box. When a user closes the top box in a nested component family, the entire family is hidden. This can be seen in figure 11.58.

TIP

This option to hide and display components and nested component families can be useful when a designer is working with a large system and wishes to focus on one area of the system without distraction.

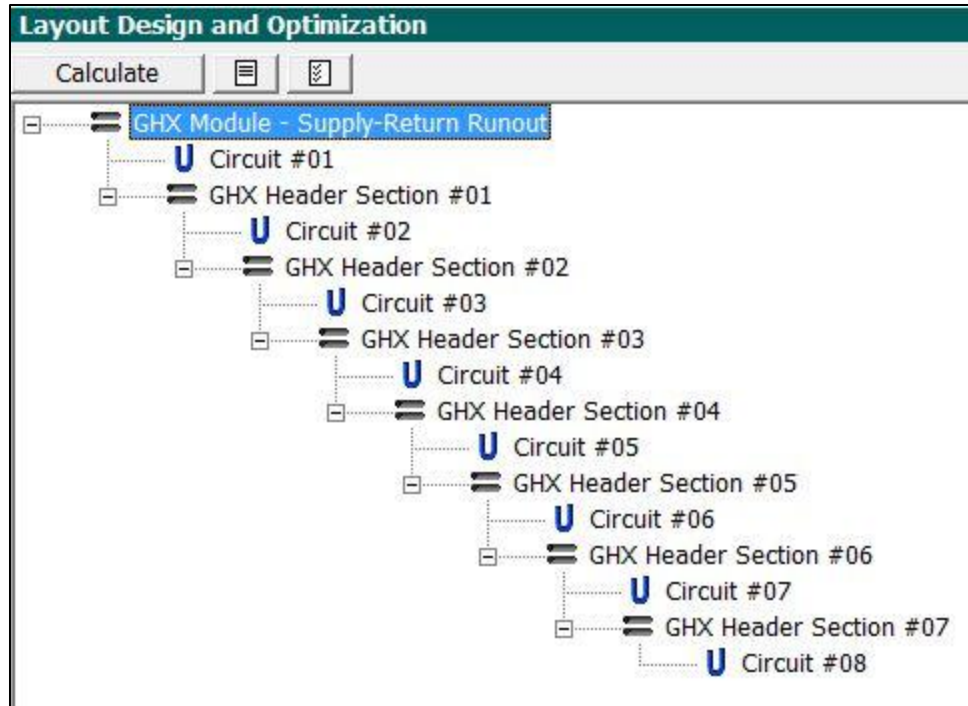


Fig. 11.57 Click on the Minus Box To Hide The Entire GHX Module

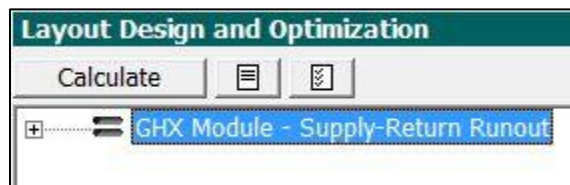


Fig. 11.58 The Entire GHX Module is Now Hidden

Deleting Pipe Pairs and Circuits

Deleting components and nested component families is as easy as creating them. To delete an individual component the user merely has to select the component, right click and select delete. This can be seen in figure 11.59 and figure 11.60.

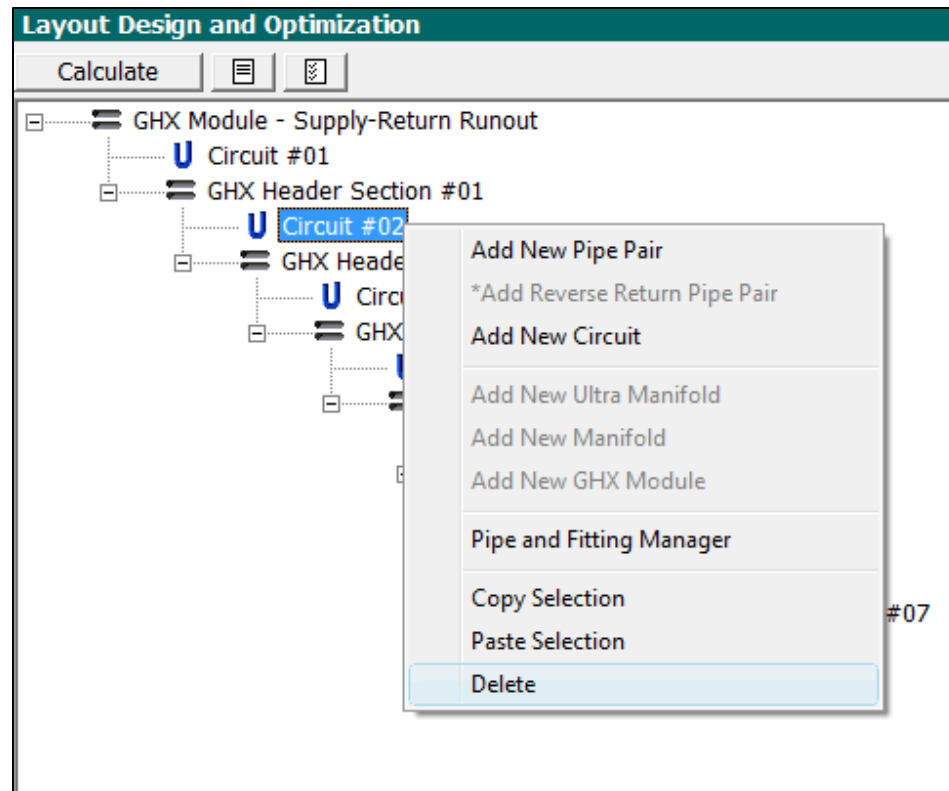


Fig. 11.59 Select the Component of Interest, Right Click and Choose Delete

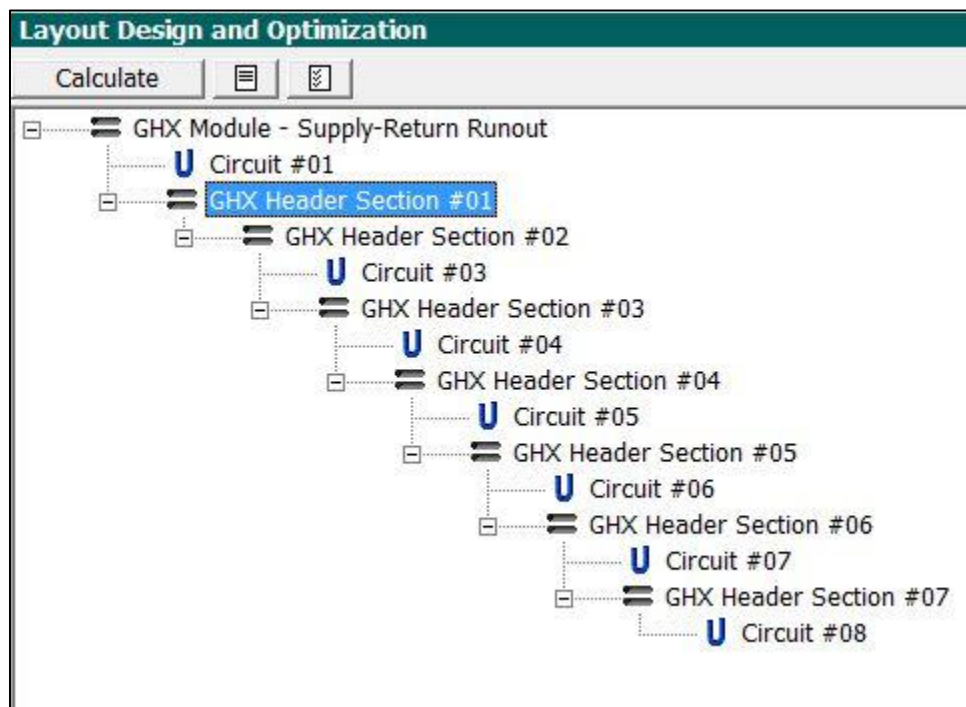


Fig. 11.60 Circuit #2 Has Been Deleted From Its Position as a Child of GHX Header Section #1

To delete an entire nested component family or part of a nested component family, the user has to select the highest level component the user wishes to delete (everything below the selected component will be deleted), right click and select delete. This can be seen in figure 11.61 and figure 11.62.

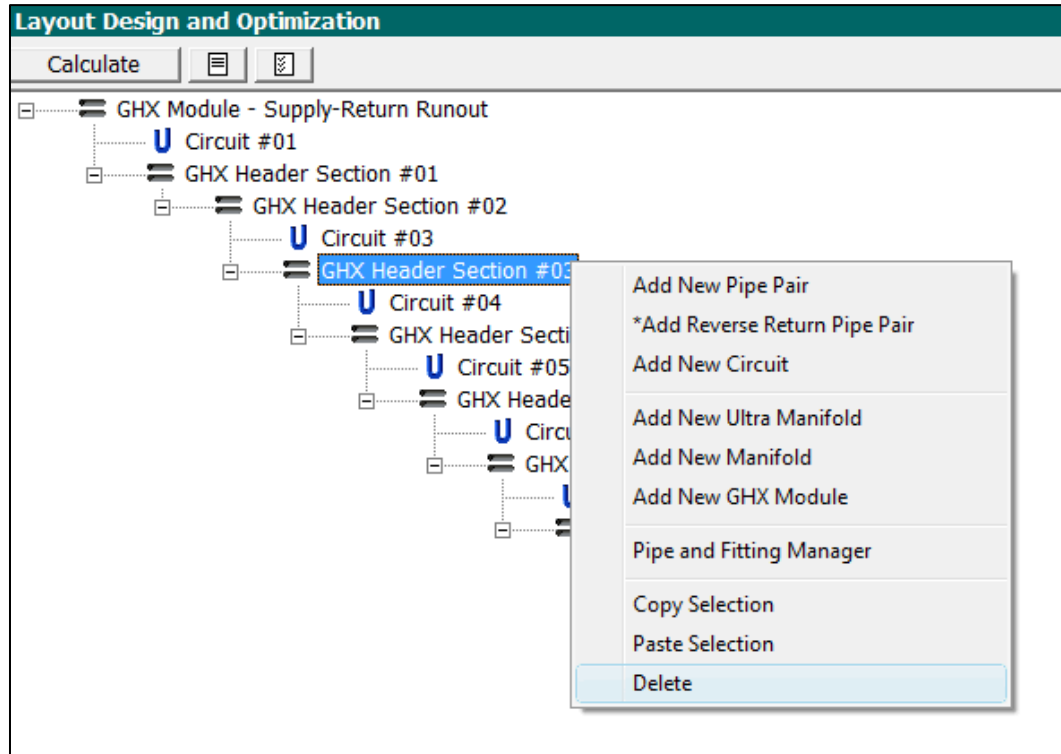


Fig. 11.61 Deleting Part of a GHX Module/Nested Component Family

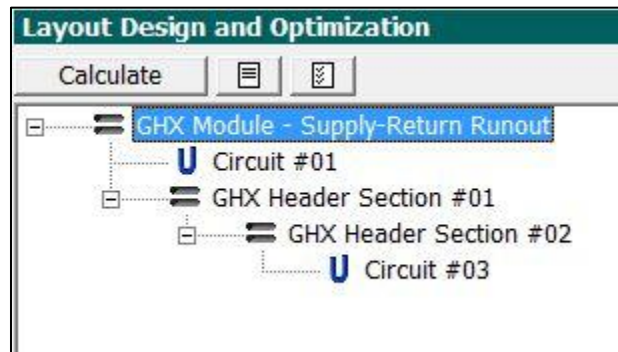


Fig. 11.62 All that remains of the GHX Module after deleting GHX Header Section #3 and Everything Below It

Modifying Parameters with the Properties Windows

After manually creating a piping system or at any time in the creation process, the user can modify the properties of any of the components. One way to do so is via the Properties Window, which is usually found to the right of the Layout Manager Workspace.

To modify a particular component the user first clicks on the component with the mouse and then adjusts the parameters in the Properties Window.

For example, in figure 11.63 below, the first pipe pair has been selected. As can be seen in both the Layout Manager Workspace and the Properties window, the pipe pair name has been changed from the default name, "GHX Module Supply-Return Runout" to "Main Supply-Return Runout Pipe Pair." In addition, the Pipe 1 (supply) and Pipe 2 (return) lengths have been set to 100 feet with 3" diameter pipe.

When setting up a system manually, users have control over name (group name and individual pipe name; group name will be described later), length, extra length, size, type, as well as a similar level of control over pipe fittings. The user is encouraged to explore the fittings as well as other options in the Properties Window. Note that properties that display calculated results (such as velocity) will be set to 0.0 until the 'Calculate' button is pushed.

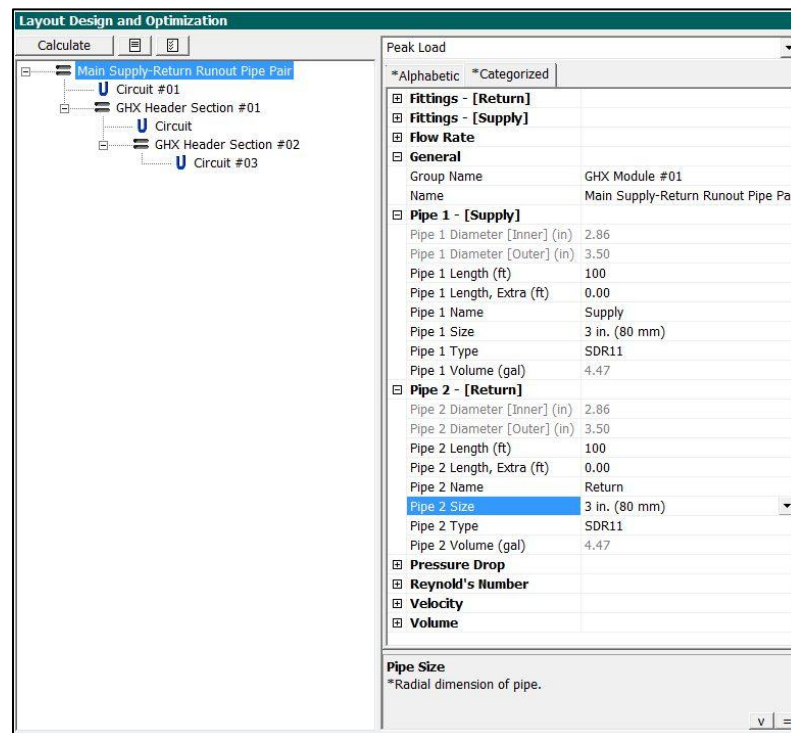


Fig. 11.63 Properties Can Be Modified With the Properties Window

The designer can also control details about the Supply Side and Return Side fittings in the Properties Windows for the same pipe pair (the Supply-Return Runout from the above figure).

TIP

It is important to remember where these fittings are located in the model (see the GLD Piping Language section of this chapter). In figure 11.64 below, some of the fitting options can be seen in the drop down menu. Note that if a user wishes to manually enter a fitting equivalent length, he or she must first choose *Other* and then can manually enter the equivalent length.

The fittings database built into the program will be growing over time as designers and manufacturers provide our firm with more accurate and detailed data pertaining to different types of fittings.

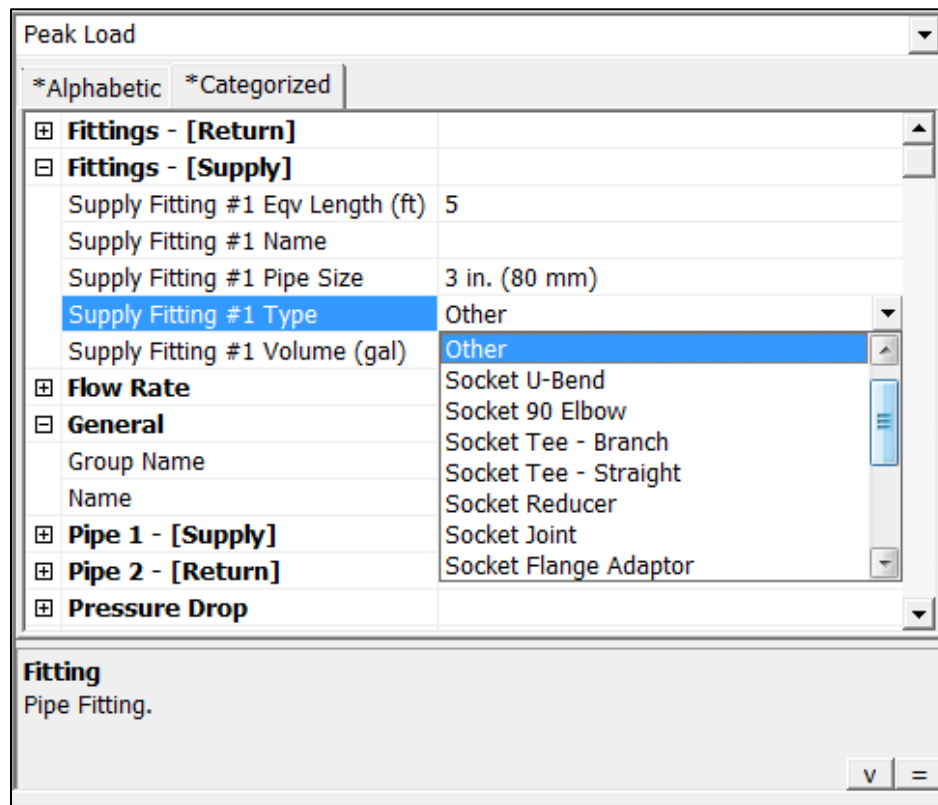


Fig. 11.64 Fittings Can Be Selected In the Properties Window

Modifying Parameters with the Pipe and Fitting Manager

The Pipe and Fitting Manager provides an entirely different system component customization experience. A user can access the Pipe and Fitting Manager by right clicking on the component of interest in the Layout Design Manager and selecting the Pipe and Fitting Manager. This can be seen in figure 11.65.

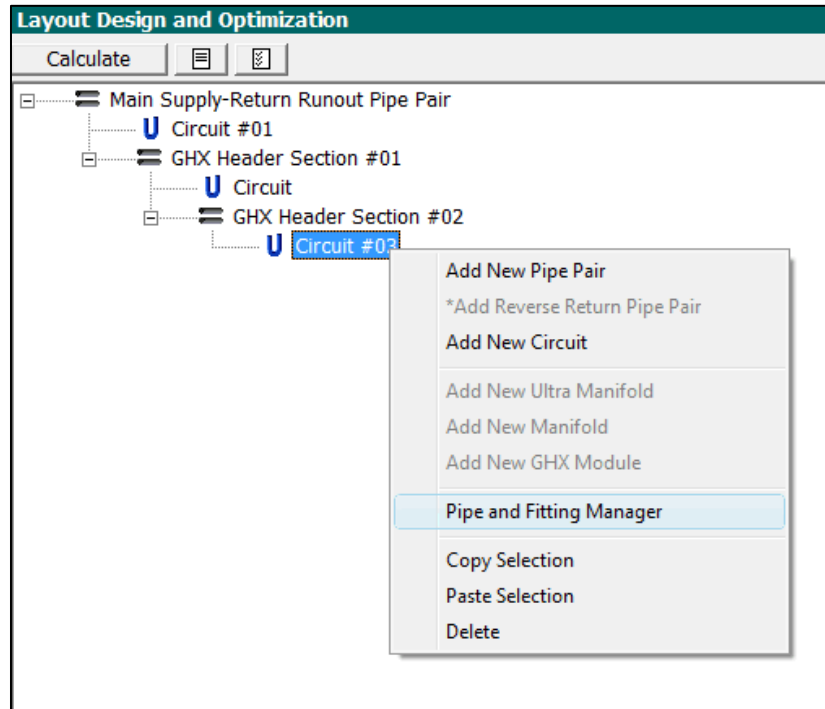


Fig. 11.65 Accessing the Pipe and Fittings Manager for Circuit #2

After doing so, the Pipe and Fitting Manager for the selected component (a GHX Circuit in this case) will open. The Manager can be seen in figure 11.66.

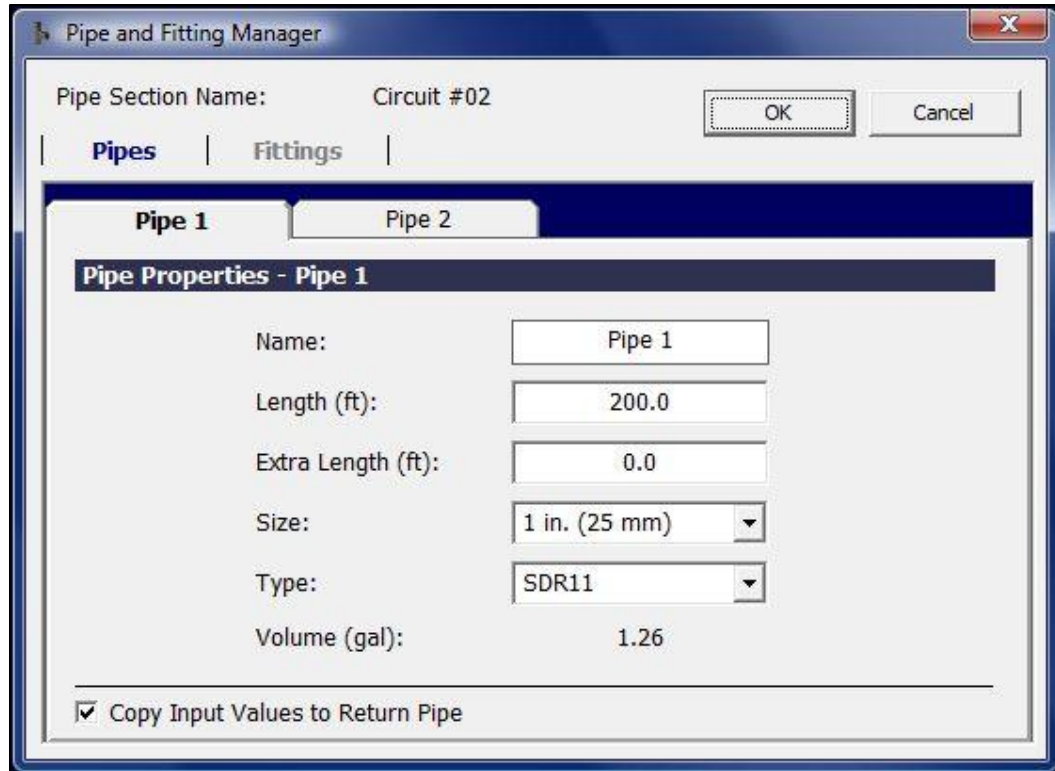


Fig. 11.66 The Pipe Section in the Pipe and Fitting Manager

The Pipe and Fitting Manager is divided into two sections. In the upper section the user can choose between adjusting "Pipes" or "Fittings" properties by clicking on one or the other. The lower section has detailed input and options boxes for the selected item (pipe or fittings).

Pipes

The pipes section is broken into two identical tabbed panels: Pipe 1 and Pipe 2. In the Pipe 1 panel, the user can enter the Pipe 1 name, length, extra length, pipe size and pipe type. The volume is auto-calculated from the pipe geometry and reported as well.

Beneath the Pipe 1 properties is a option box entitled "copy input values to return pipe". This option, if selected, will apply changes made to Pipe 1 to Pipe 2 automatically. In most cases, designers want to make the same changes to both pipes and this feature saves some time.

Fittings

The fittings section can be seen in figure 11.67 below.

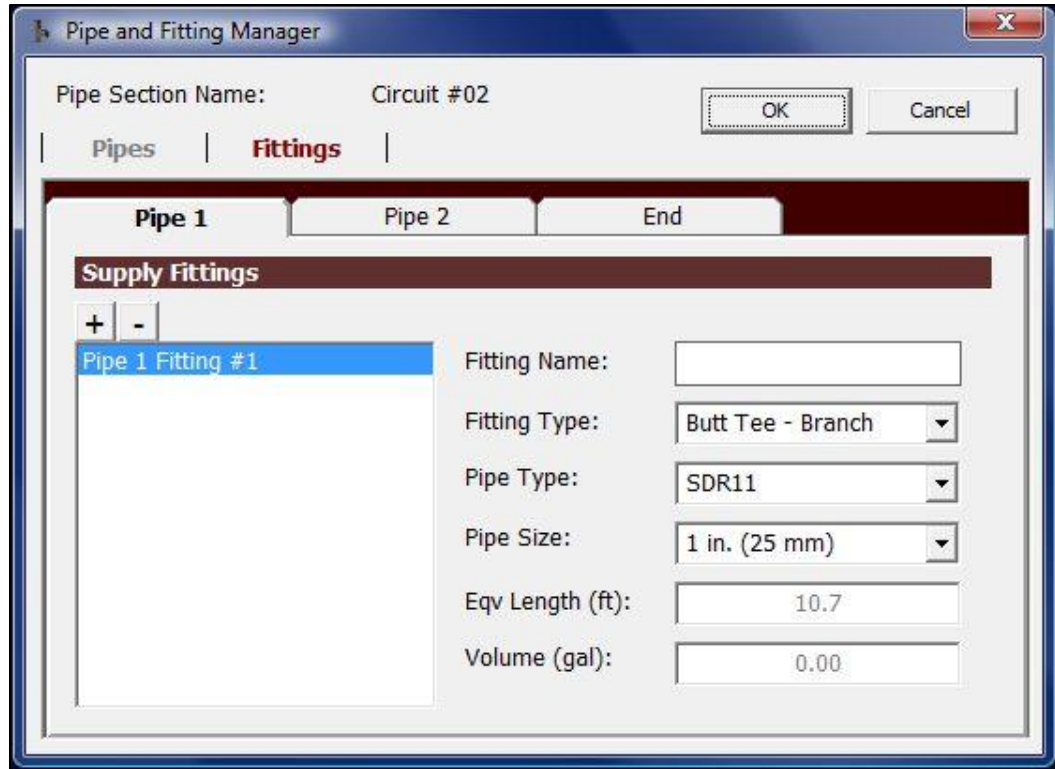


Fig. 11.67 The Fittings Section in the Pipe and Fitting Manager

The fittings tab is broken into either two or three tabbed panels, depending on the selected component. If a pipe pair has been selected, there are two available fittings tabbed panels: Pipe 1 and Pipe 2. If a circuit has been selected, three fittings tabbed panels will be available: Pipe 1, Pipe 2 and End. In figure 11.67, 3 tabbed panels are available, indicating that a circuit was selected (as indeed was the case as can be seen in figure 11.65 above).

For some designs, engineers will use two or more fittings in series at a single piping connection point. The CFD module enables a designer to add more than one fitting as necessary. When additional fittings are added, they are added to the Properties Window and of course included in the calculations.

As can be seen above, a single pipe fitting- a 1" SDR11 Butt-Tee branch fitting- has been selected. A designer can add or remove fittings as necessary via the +/- buttons which can be seen below:



If a user desires a fitting that is not included in the database, the user can choose 'Other' and enter his or her required parameters.

After the designer has completed his or her selection/modification of fittings and pipes using the Pipe and Fitting Manager, the designer can hit 'OK' to save the

updates into the design. If the designer hits 'Cancel', all updated information will be lost.

All updates made in the Pipe and Fitting Manager can be seen in the Properties Window.

Using the techniques and tools described in this Manual Techniques section, a user can design and build a near infinite range of geothermal GHX fields. After the design is complete, the user can see how it performs. Calculations and performance will be addressed later in this chapter

Automatic Methods

The CFD module also offers a range of tools for the designer who desires to have the module automatically build a wide range of piping systems. These automatic methods provide the designer with tremendous power, flexibility and time savings. While manually building and optimizing a GHX Module using the manual methods described above could take anywhere from a couple of minutes to an hour to complete, the automatic methods described below can complete nearly any task in a matter of seconds. Automated system building tools include:

- The GHX Module Builder (direct and reverse return)
- The Manifold/Vault Builder
- The Ultra Manifold/Ultra Vault Builder

The GHX Module Builder

The GHX Module Builder is a powerful tool that automatically builds flow balanced GHX Modules of any size and complexity.

The GHX Module Builder can be accessed from within the Layout Manager Workspace in the *Layout* Panel.

The user can *right click* the mouse while inside the Layout Manager Workspace to see the menu in figure 11.68 appear.

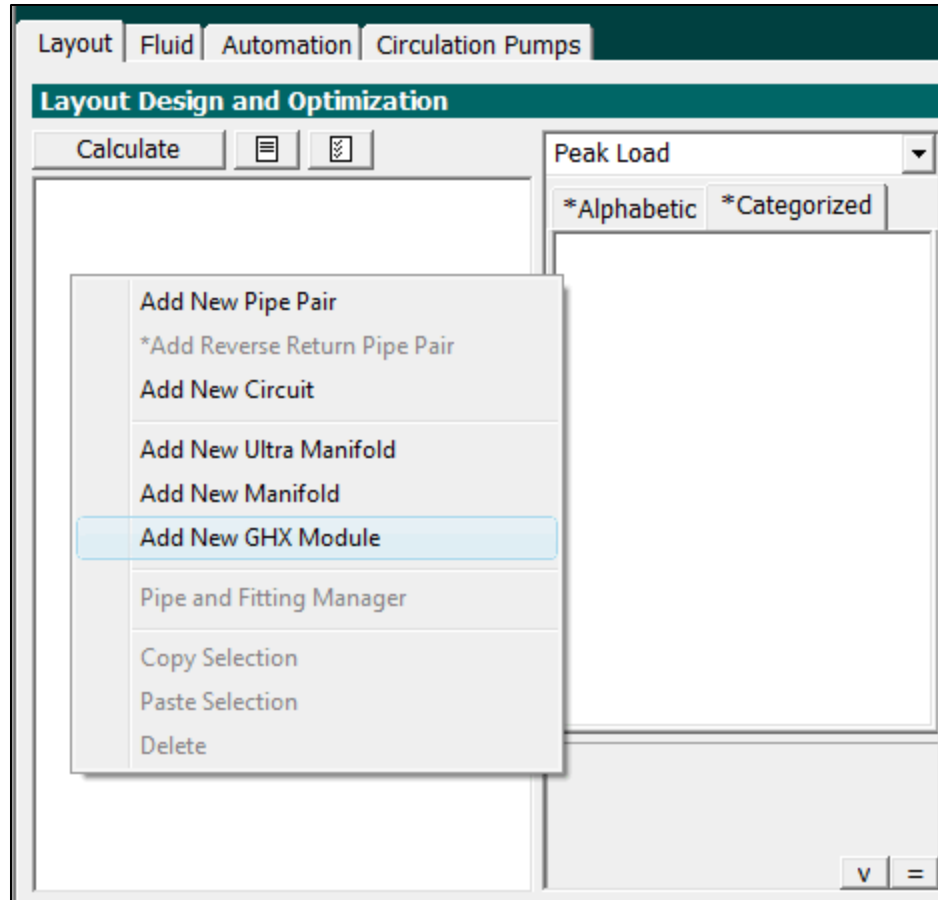


Fig. 11.68 Right Click to Access the GHX Module Builder

After the user selects New GHX Module, the GHX Module Builder will appear as it does in figure 11.69.

GHXModule and Manifold Builder

Group Name

Group Name:

Return Piping Style

Return Type:

Circuit Information

Number of Circuits:	<input type="text" value="8"/>	Extra
Circuit Separation (ft):	<input type="text" value="20.0"/>	<input type="text" value="0.0"/>
One-way Circuit Length (ft):	<input type="text" value="300.0"/>	<input type="text" value="0.0"/>
Circuit Pipe Size:	<input type="text" value="SDR11"/>	<input type="text" value="1 in. (25 mm)"/>
Header Pipe Size:	<input type="text" value="SDR11"/>	<input type="text" value="2 in. (50 mm)"/>
Circuits Per Parallel Loop:	<input type="text" value="1"/>	
Circuits Per One-Way Length:	<input type="text" value="1"/>	

Supply-Return Runout Information

One Way Length (ft):	<input type="text" value="200.0"/>	Extra
Pipe Size:	<input type="text" value="SDR11"/>	<input type="text" value="2 in. (50 mm)"/>

Fig. 11.69 The GHX Module Builder

The GHX Module Builder is broken into five sections: Group Name, Return Piping Style, Circuit Information, Supply-Return Pipe Information and the OK/Cancel buttons. Each section is addressed below.

Group Name

The group name is a parameter applied to every component in a design. The user can use the default group name or select one of his or her choosing. The group name becomes important during the design review process so it is therefore critical that each GHX Module has a unique group name.

Return Pipe Style

The designer can choose to build a reverse return GHX Module or a direct return GHX Module. Because the flow characteristics of the two options are very different, it is critical that the designer selects the correct return piping style.

Circuit Information

In this section, the user enters details pertaining to the circuits themselves. Details include the number of GHX Circuits, the separation between circuits, the one way length of each circuit, the circuit pipe size, circuits per parallel loop and circuits per one way length. Because these last two parameters (circuits per parallel loop and circuits per one way length) may sound unfamiliar they are described below:

Circuits Per Parallel Loop

Here the user enters the number of GHX Circuits he or she desires per parallel loop. Put another way, if a designer wants to have two GHX Circuits in series on one parallel loop, the designer can enter a "2" here. A sample system with 2 circuits per parallel loop can be seen below in figures 11.70 and 11.71.

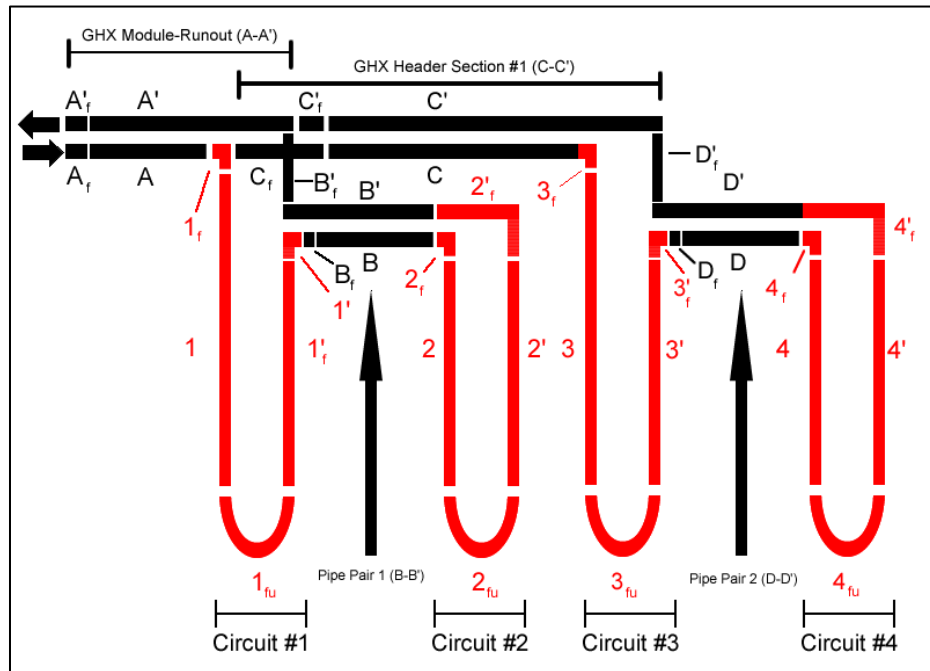


Fig. 11.70 An Example of Two Circuits Per Parallel Loop

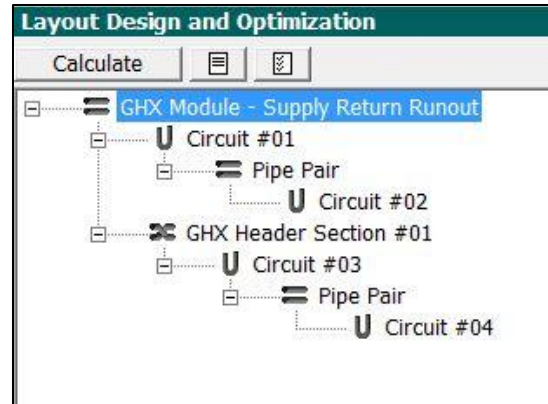


Fig. 11.71 An Example of Two Circuits Per Parallel Loop

Circuits Per One-Way Length

Here the user enters the number of GHX Circuits he or she desires per one way length. Put another way, if a designer wants to have two GHX Circuits in parallel or in series (for example in a single borehole), the designer can enter a "2" here. Examples of GHX Circuits in parallel and series can be seen in figures 11.72 and 11.73 below.

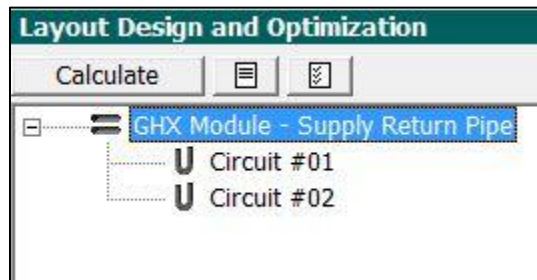


Fig. 11.72 Two Circuits Per One Way Length: Parallel Flow

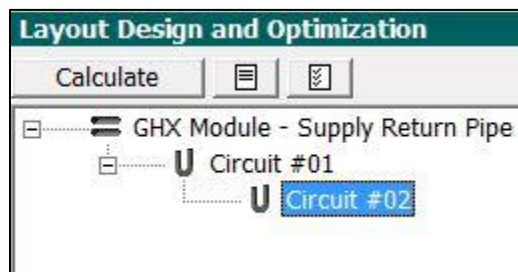


Fig. 11.73 Two Circuits Per One Way Length: Series Flow

Supply-Return Pipe Information

In this section the designer enters the one way length of the supply/return pipe pair that connects the GHX Module with its parent component in the design (typically a Manifold, Vault or circulation pump house). The user also enters the supply/return pipe diameter here.

Note that the GHX Module Builder is pre-populated with design parameters. These default parameters can be updated/modified as necessary in the Automation Panel and on the GHX Module subpanel.

OK and Cancel Buttons

After the designer has reviewed and modified the parameters, he or she can hit the "OK" button and the GHX Module will be auto-built in the Layout Manager Workspace. An example of an auto-built reverse return GHX Module can be seen in figure 11.74.

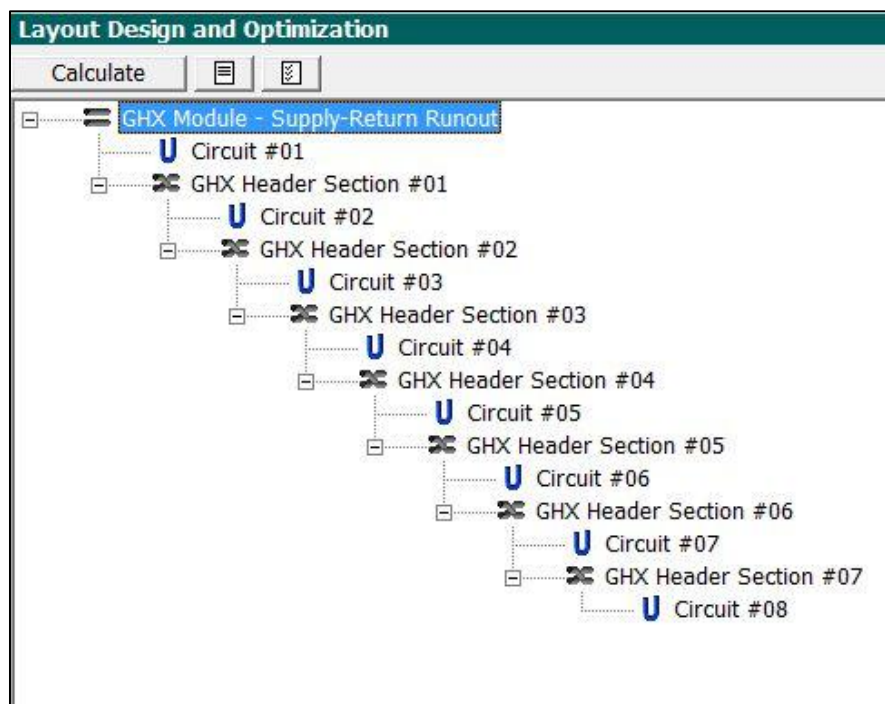


Fig. 11.74 An Auto-Built Reverse Return GHX Module

At this point, the designer can make copies of the GHX Module if so desired using the copy-paste functionality. In addition, the designer can add to or change the module using the manual techniques outlined above.

The Manifold/Vault Builder

The Manifold/Vault Builder is a powerful tool that automatically builds Manifolds/Vaults. Note that conceptually in the CFD module, Manifolds and Vaults are identical.

The Manifold/Vault Builder can be accessed from within the Layout Manager Workspace in the *Layout Panel*

The user can *right click* the mouse while inside the Layout Manager Workspace to see the menu in figure 11.75 appear

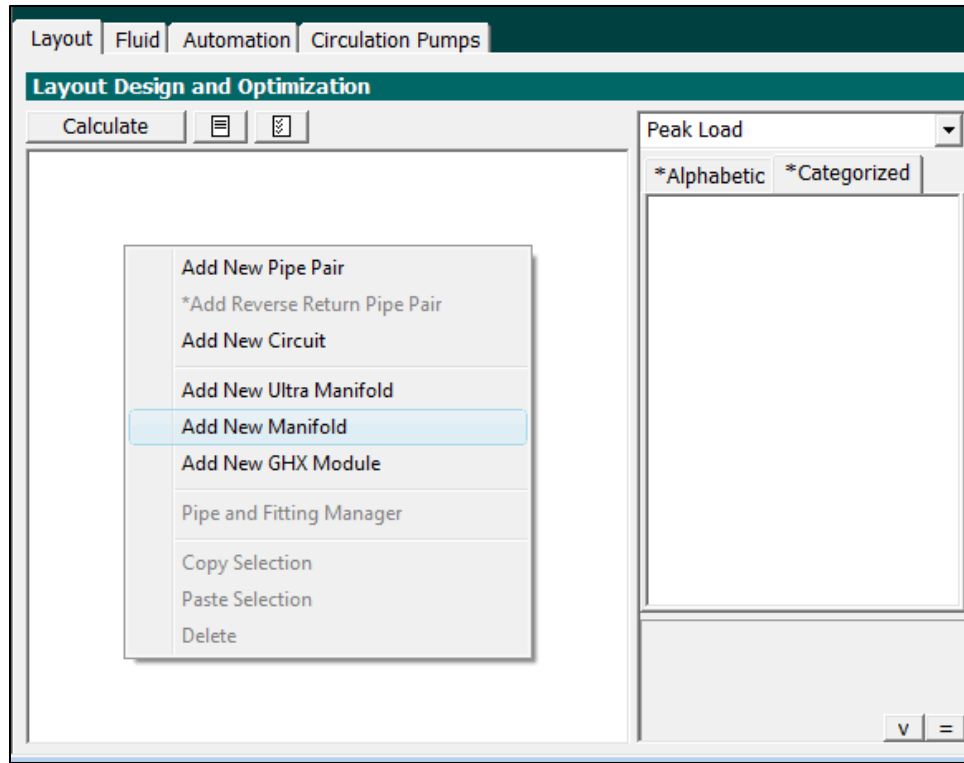


Fig. 11.75 Accessing the Manifold/Vault Builder

After the user selects New Manifold, the Manifold/Vault Builder will open as can be seen in figure 11.76.

GHXModule and Manifold Builder

Group Name

Group Name:

Return Piping Style

Return Type:

Section Outlet Information

Section Outlet Number:	<input type="text" value="5"/>	Extra
Section Outlet Separation (ft):	<input type="text" value="6.0"/>	<input type="text" value="0.0"/>
Section Outlet Pipe Size:	<input type="text" value="SDR11"/>	<input type="text" value="2 in. (50 mm)"/>

Supply-Return Runout Information

One Way Length (ft):	<input type="text" value="200.0"/>	Extra
Pipe Size:	<input type="text" value="SDR11"/>	<input type="text" value="3 in. (80 mm)"/>

Fig. 11.76 The Manifold/Vault Builder

The Manifold/Vault Builder is broken into five sections: Group Name, Return Piping Style, Section Outlet Information, Supply-Return Pipe Information and the OK/Cancel buttons. Each section is addressed below.

Group Name

The group name is a parameter applied to every component in a design. For the Manifold/Vault, the user can use the default group name or select one of his or her choosing. The group name becomes important during the design review process so it is therefore critical that each Manifold/Vault in a system has a unique group name.

Return Pipe Style

For Manifold/Vault systems, the return piping style is locked at direct return since Manifolds/Vaults are always direct return systems.

Section Outlet Information

A Manifold/Vault will have two or more outlets connecting typically to GHX Modules via the GHX Modules' Supply-Return Runouts. The Manifold/Vault is a

parent to the GHX Modules. Details pertaining to these outlets including the number of outlets, the separation between each outlet in the Manifold/Vault and the section outlet pipe size (which is likely to be identical to the Supply-Return Runout pipe size coming in from the GHX Modules).

Supply-Return Pipe Information

In this section the designer enters the one way length of the supply/return pipe pair that connects the Manifold/Vault with its **parent** component in the design. In many cases, the parent component of a Manifold or Vault will be a circulation pump system. Therefore the user may choose to enter the pipe distance from the Manifold/Vault to the circulation pump. The user also enters the supply/return pipe pair diameter here.

Note that the GHX Module Builder is pre-populated with design parameters. These default parameters can be updated/modified as necessary in the Automation Panel and on the GHX Module subpanel.

OK and Cancel Buttons

After the designer has reviewed and modified the parameters, he or she can hit the "OK" button and the Manifold/Vault will be auto-built in the Layout Manager Workspace. An example of an auto-built direct return Manifold can be seen in figure 11.77.

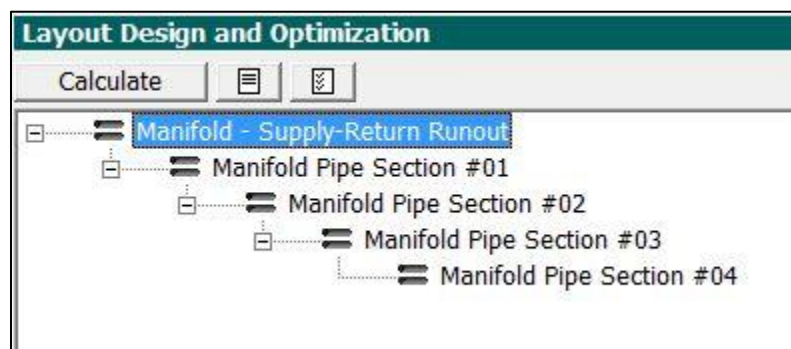


Fig. 11.77 A Manifold with Five Outlets

At this point, the designer can use the GHX Module Builder if so desired and auto-build four GHX Modules and attach them to the four Manifold pipe sections. An example of a completed system can be seen in figure 11.78.

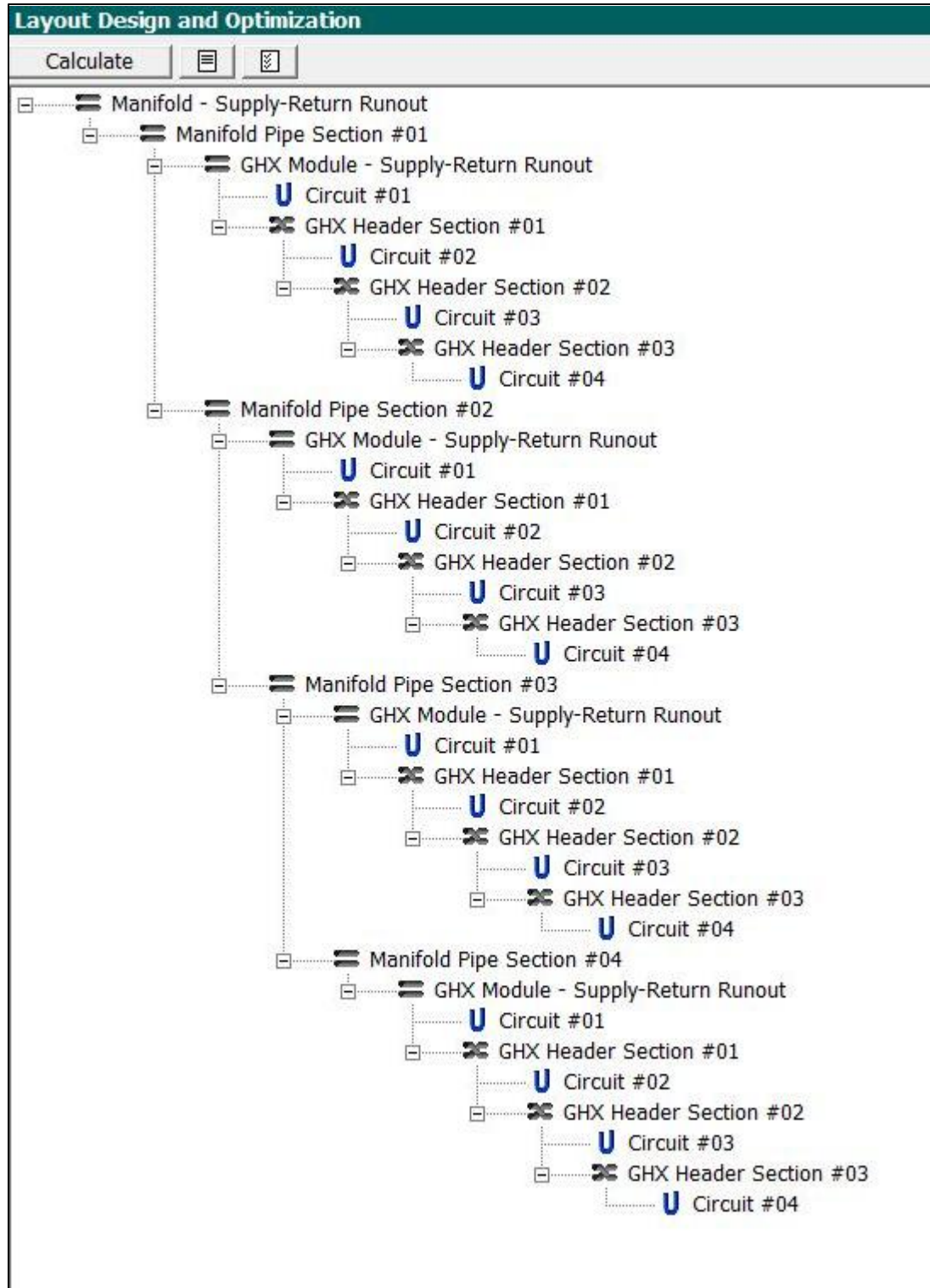


Fig. 11.78 A Manifold with Four Outlets Hooked Up to Four GHX Modules

Note that each of the four GHX Modules has four GHX Circuits with reverse return headering. The small, four GHX Circuit GHX Modules are for illustrative purposes. Real world Manifold systems would likely have more than four GHX Circuits per GHX Module.



Remember that systems in the Layout Manager Workspace are not drawn to scale. The GHX Module Supply-Return Runout coming out of Manifold Pipe Section #1 could be twice as long or half as long (for example) as the GHX Module Supply-Return Runout coming out of Manifold Pipe Section #2. Of course, great variations in length could impact flow balancing. However, even physically imbalanced systems can be flow balanced using the automatic and manual controls in the CFD Module.

The Ultra Manifold/Ultra Vault Builder

The Ultra Manifold/Ultra Vault Builder is a powerful tool for very large commercial systems that require nested tiers of Manifolds and/or field Vaults. The highest level Manifold or Vault is defined as an "Ultra" Manifold or "Ultra" Vault. Coming into an Ultra Manifold or Ultra Vault from the child side are supply/return runouts from two or more Manifolds or Vaults. Coming out of the Ultra Manifold or Ultra Vault and heading in the parent direction are a supply/return runout pair. Systems of this size are quite rare but the CFD module is flexible enough to handle them.

The Ultra Manifold/ Ultra Vault Builder can be accessed from within the Layout Manager Workspace in the *Layout* Panel

The user can *right click* the mouse while inside the Layout Manager Workspace to see the menu in figure 11.79 appear.

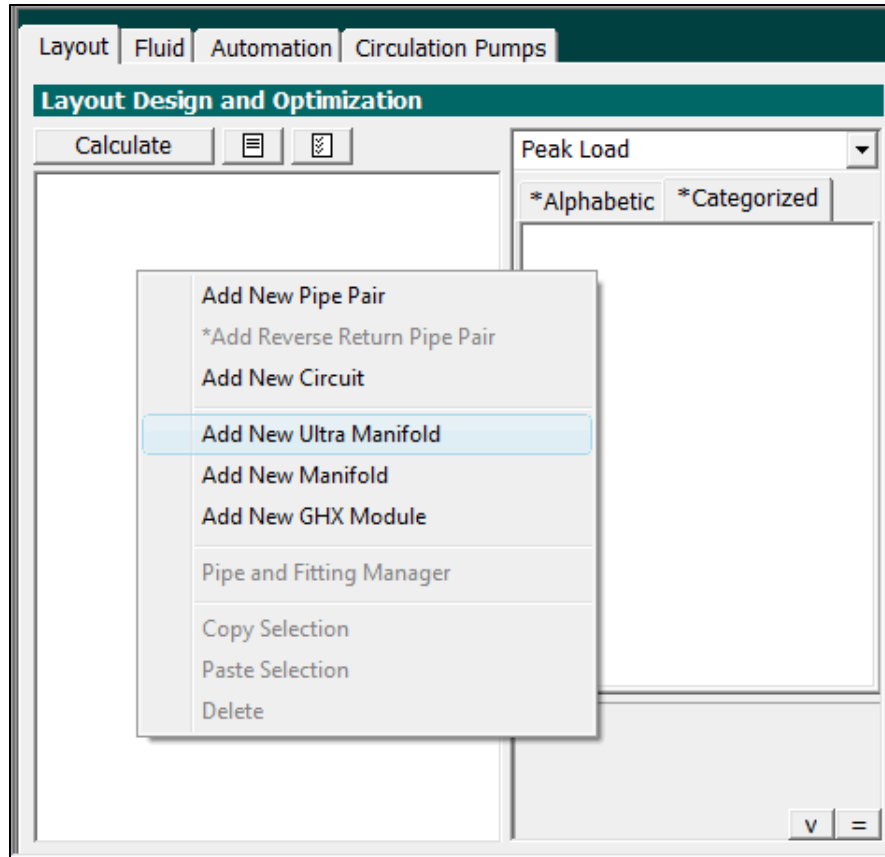


Fig. 11.79 Accessing the Ultra Manifold/Vault Builder

After the user selects New Ultra Manifold, the Ultra Manifold/Ultra Vault Builder will open as can be seen in figure 11.80.

GHXModule and Manifold Builder

Group Name
Group Name: Ultra Manifold #01

Return Piping Style
Return Type: Direct Return

Section Outlet Information
Section Outlet Number: 5
Section Outlet Separation (ft): 6.0
Section Outlet Pipe Size: SDR11 3 in. (80 mm)

Supply-Return Runout Information
One Way Length (ft): 200.0
Pipe Size: SDR11 4 in. (100 mm)

Extra: 0.0

OK Cancel

Fig. 11.80 The Ultra Manifold/Vault Builder

Readers are referred to the Manifold/Vault Builder section above for a description since the Ultra Manifold Builder and the Manifold Builder are nearly identical.

Using the techniques and tools described in both the Automatic and Manual Techniques sections, users can design and build a near infinite range of geothermal GHX loopfields. After the design is complete, the user can see how it performs. Calculations and performance will be addressed later in this chapter.

Calculating and Reviewing Results

In this section we will explore how to calculate and review results.

Calculating Results

After a user has built a piping system (manually/automatically/a combination of both), the designer can hit the 'Calculate' button. Figure 11.81 is an 8 GHX Circuit reverse return GHX Module that will be used as an example for understanding results.

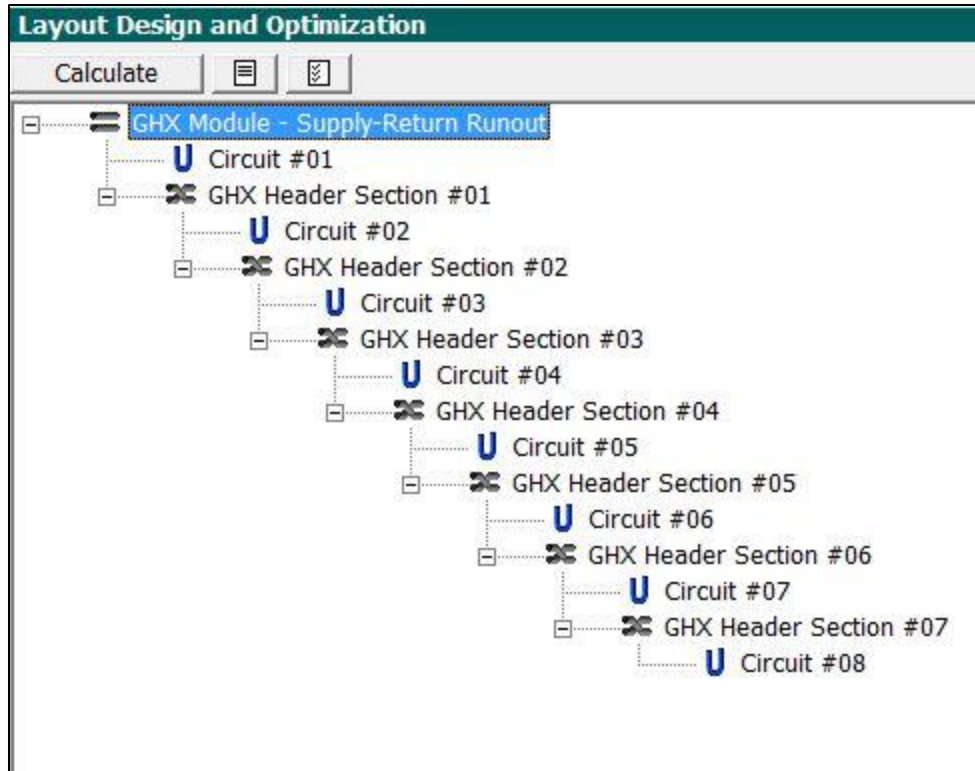


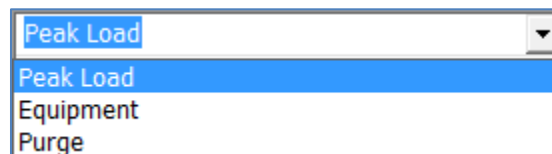
Fig. 11.81 A Sample Eight GHX Circuit Reverse Return GHX Module Example for Understanding Calculated Results

Reviewing Results

Reviewing the initial results of a design is an enjoyable step in the piping design process. First, the user has to choose which results he or she wishes to see. There are three general types of results:

- Peak Load Flow Rate Results
- Installed Equipment Capacity Flow Rate Results
- Purge Flow Rate Results

The user can switch between the three types of results by selecting from among them with the dropdown menu as can be seen below:



Note that all three results types are calculated when the user hits the 'Calculate' button.

It is important for the designer to confirm that he or she is using the appropriate flow rate selection.

Besides having three general types of results, there are also three primary ways of reviewing results. These include:

- Properties Window results
- Layout Manager Workspace results
- Review Panel results

Each is described in turn below.

PROPERTIES WINDOW RESULTS

After the fluid dynamics for a particular system have been calculated, the user can review the results for every component one at a time in the Properties Window. For example, in figure 11.82 below, the Peak Load Flow Rate (set at 30 gpm in the *Fluid* tab) has been selected and results for the GHX Circuit #4 Pressure Drop, Reynold's Number and Velocity are displayed as follows:

Pressure Drop: 2.5 ft. hd
Reynold's Number: 4997
Velocity: 0.88 ft/s



Remember that these results are for the Peak Load Flow Rate and for the selected fluid (which is water in this example). If a user changes the flow rate or the fluid type, results will change as well.

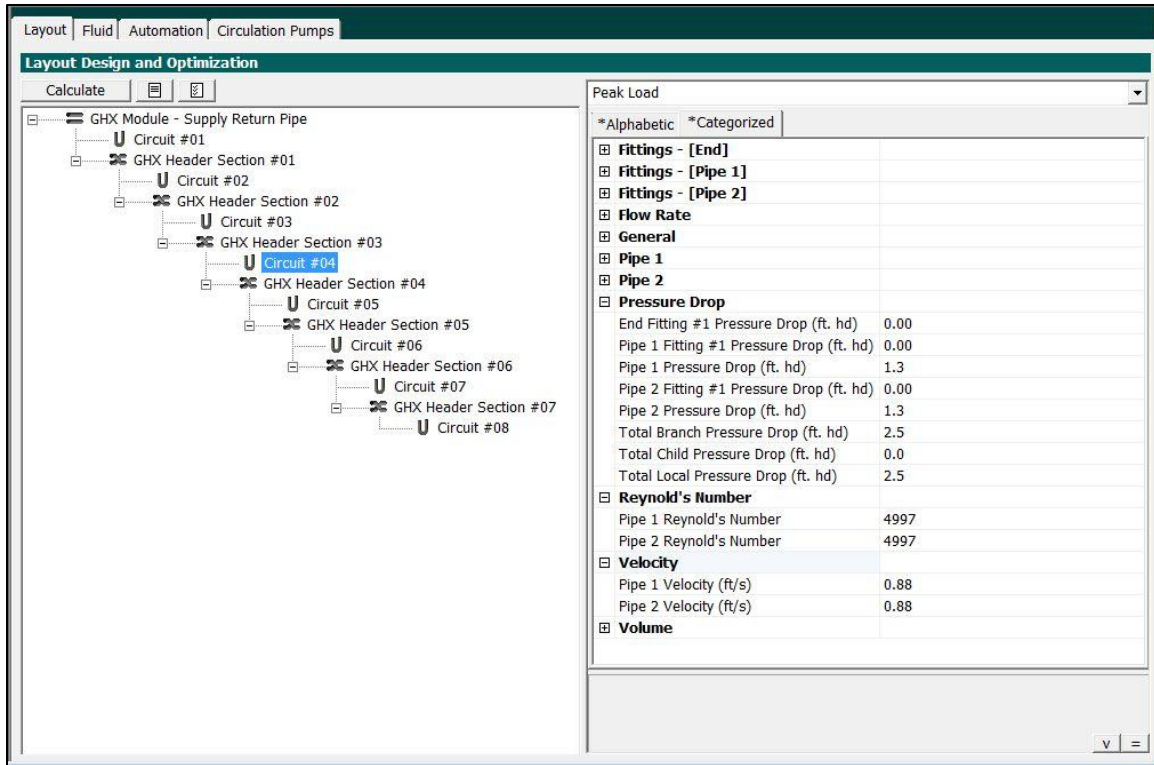


Fig. 11.82 A Sample Property Window Results For An Eight GHX Circuit Reverse Return GHX Module

Note that the user can expand and contract the details in the Properties Windows as necessary to optimize his or her interpretive space.



Also note that total pressure drop for an entire nested component family (such as a GHX Module) can be seen by clicking on the top parent component (GHX Module Supply-Return Pipe in fig X) and viewing the 'Total Branch Pressure Drop' in the Pressure Drop section of the Properties Window.

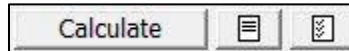
Also note that while the detail available in the Properties Window is unsurpassed, it is not the fastest way to review a system. For a faster system review, designers can use the Layout Manager Workspace results option.

LAYOUT MANAGER WORKSPACE RESULTS

The Layout Manager Workspace, in addition to displaying the visual piping design, also displays results that are matched to the individual components. Viewing results in the Layout Manager Workspace is a faster way to review results across an entire system as well as to compare different components from a variety of perspectives.

Because the results data are comprehensive and too much to absorb at one time, designers have control over which results they wish to see at any one time. The

user can select which information to view using the *Display* icon button which is the **far right** button in the image below:



When a user pushes this button an window similar to that in figure 11.83 will appear.

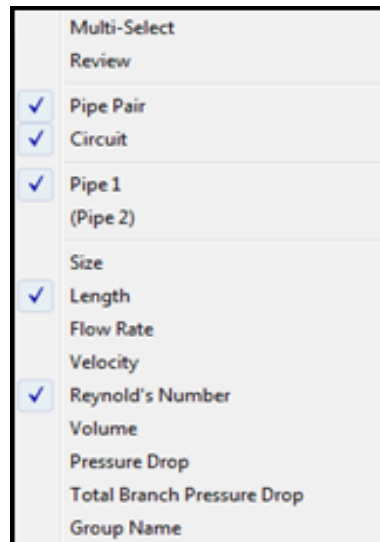


Fig. 11.83 Selecting Which Results to View in the Layout Manager Workspace

The user can then proceed to select data sets of interest.



Note: the user MUST select at least one option from each of the following two categories for results to display:

- Pipe Pair and/or Circuit
- Pipe 1 and/or (Pipe 2)

If the user does not select at least one option from each of the above two categories, results will not be displayed.

If the user selects categories of information as can be seen in figure 11.84, then results will appear as they do in figure 11.85. Viewing this combination of results is useful for looking quickly at the overall piping structure of the GHX Module headering system.

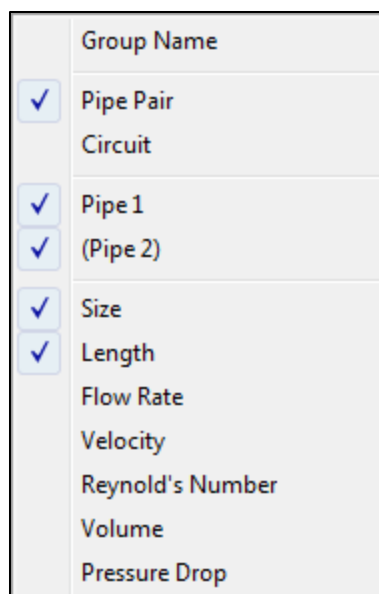


Fig. 11.84 Desired Results to View Have Been Selected

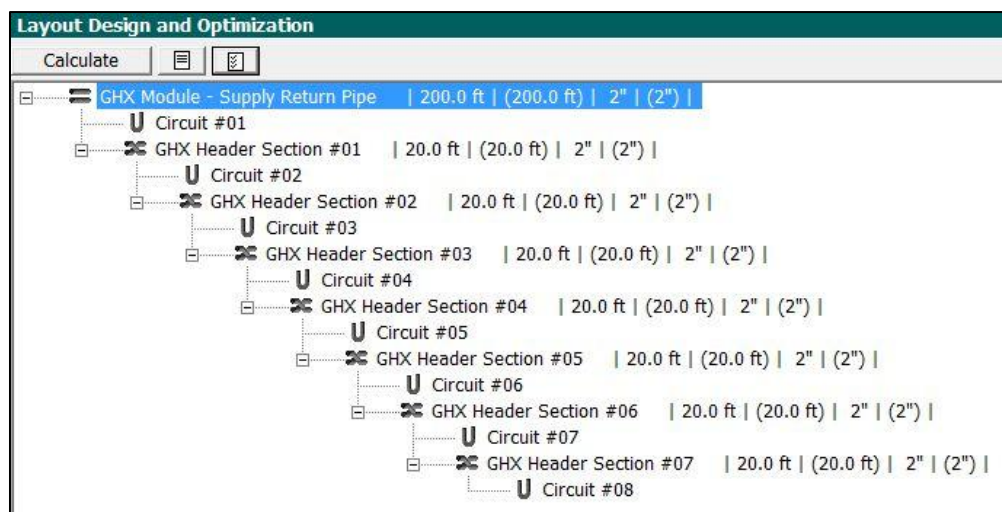


Fig. 11.85 Selected Results Are Now Visible in the Layout Manager Workspace

For another example, if a user selects categories of information as can be seen in figure 11.86, then results will appear as they do in figure 11.87. Viewing this combination of results is useful for looking quickly at circuit flow characteristics and ensuring turbulent flow throughout the design. It is also very useful for understanding the details of reverse return system performance (notice the flow rate and Reynold's Number symmetry).

TIP

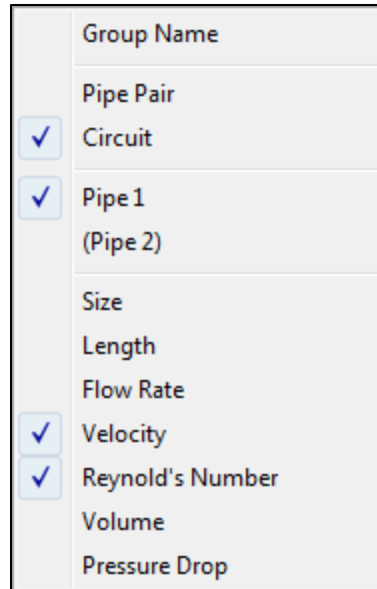


Fig. 11.86 A Different Set of Desired Results Has Been Selected

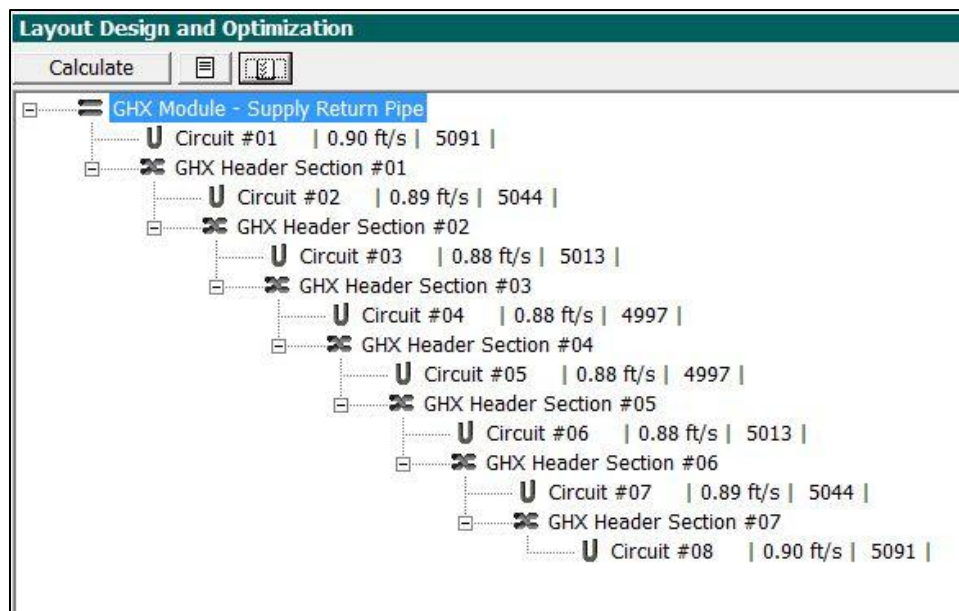
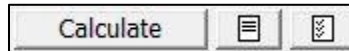


Fig. 11.87 The New Set of Selected Results Are Now Visible

While looking at the results in the Layout Manager Workspace offers many benefits, in some cases it is easier to look at results in a non-nested format. If such results are desired, the user can select the Review Panel Results.

REVIEW PANEL RESULTS

The Review Panel displays results in a "classic" view that is quick and easy to understand. Users can access Review Panel results by using the *Review* icon button which is the **middle** button in the image below:



When a user pushes the middle button a screen similar to the one in figure 11.88 will appear (results expanded manually for ease of viewing).

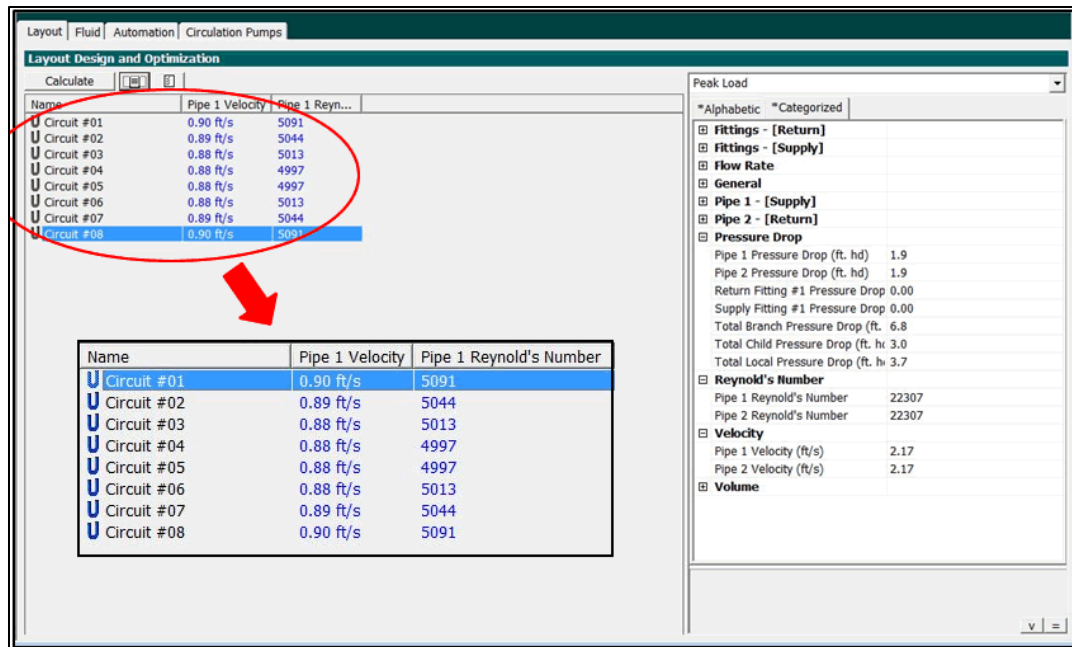


Fig. 11.88 Viewing Results in the Review Panel

TIP

In the Review Panel, users can view the same results that can be viewed in the Layout Manager Workspace. Some designers find the results in the Review Panel to be easier to review because of the vertical column format.

Users are able to calculate updated results from within the Review Panel, review detailed results for a particular component in the Properties Window, and adjust the viewed results as necessary using the same 'Display' button. Figure 11.89 below for example shows a variety results (pipe size, fluid velocity and Reynold's Number) that would be interesting when reviewing design issues related to purging.

Layout Design and Optimization						
Calculate						
Name	Pipe 1 Size	Pipe 2 Size	Pipe 1 Velocity	Pipe 2 Velocity	Pipe 1 Reynold's Number	Pipe 2 Reynold's Number
GHX Module - Supply Return Pipe	2"	2"	2.17 ft/s	2.17 ft/s	22307	22307
U Circuit #01	1"	1"	0.90 ft/s	0.90 ft/s	5091	5091
GHX Header Section #01	2"	2"	1.90 ft/s	0.27 ft/s	19489	2819
U Circuit #02	1"	1"	0.89 ft/s	0.89 ft/s	5044	5044
GHX Header Section #02	2"	2"	1.63 ft/s	0.55 ft/s	16696	5611
U Circuit #03	1"	1"	0.88 ft/s	0.88 ft/s	5013	5013
GHX Header Section #03	2"	2"	1.36 ft/s	0.82 ft/s	13921	8387
U Circuit #04	1"	1"	0.88 ft/s	0.88 ft/s	4997	4997
GHX Header Section #04	2"	2"	1.09 ft/s	1.09 ft/s	11154	11154
U Circuit #05	1"	1"	0.88 ft/s	0.88 ft/s	4997	4997
GHX Header Section #05	2"	2"	0.82 ft/s	1.36 ft/s	8387	13921
U Circuit #06	1"	1"	0.88 ft/s	0.88 ft/s	5013	5013
GHX Header Section #06	2"	2"	0.55 ft/s	1.63 ft/s	5611	16696
U Circuit #07	1"	1"	0.89 ft/s	0.89 ft/s	5044	5044
GHX Header Section #07	2"	2"	0.27 ft/s	1.90 ft/s	2819	19489
U Circuit #08	1"	1"	0.90 ft/s	0.90 ft/s	5091	5091

Fig. 11.89 Viewing A Set of Results Well Suited for Purging Design

Users have the option of sorting data by column by clicking on the top of any of the columns. Reorganizing the data back to its original state can be accomplished by hitting the 'Calculate' button again.

When more than one nested component family (ie- more than one GHX Module, a GHX Module and a Manifold, etc) is present in a design, the designer will benefit from displaying the "Group Name" option. The "Group Name" option allows a designer to sort and resort a large system by Group Name when he or she clicks on the Group Name column. The Group Name column can be seen in figure 11.90.

TIP

Layout Design and Optimization							
Calculate							
Name	Group Name	Pipe 1 Size	Pipe 2 Size	Pipe 1 Velocity	Pipe 2 Velocity	Pipe 1 Reynold's Number	Pipe 2 Reynold's Number
GHX Module - Supply Return Pipe	GHX Module #01	2"	2"	2.17 ft/s	2.17 ft/s	22307	22307
U Circuit #01	GHX Module #01	1"	1"	0.81 ft/s	0.81 ft/s	4604	4604
GHX Header Section #01	GHX Module #01	2"	3/4"	1.92 ft/s	1.27 ft/s	19758	5766
U Circuit #02	GHX Module #01	1"	1"	0.90 ft/s	0.90 ft/s	5123	5123
GHX Header Section #02	GHX Module #01	2"	1 1/4"	1.65 ft/s	1.07 ft/s	16922	7706
U Circuit #03	GHX Module #01	1"	1"	0.91 ft/s	0.91 ft/s	5188	5188
GHX Header Section #03	GHX Module #01	2"	1 1/2"	1.37 ft/s	1.26 ft/s	14049	10323
U Circuit #04	GHX Module #01	1"	1"	0.92 ft/s	0.92 ft/s	5229	5229
GHX Header Section #04	GHX Module #01	2"	2"	1.09 ft/s	1.09 ft/s	11154	11154
U Circuit #05	GHX Module #01	1"	1"	0.92 ft/s	0.92 ft/s	5229	5229
GHX Header Section #05	GHX Module #01	1 1/2"	2"	1.26 ft/s	1.37 ft/s	10323	14049
U Circuit #06	GHX Module #01	1"	1"	0.91 ft/s	0.91 ft/s	5188	5188
GHX Header Section #06	GHX Module #01	1 1/4"	2"	1.07 ft/s	1.65 ft/s	7706	16922
U Circuit #07	GHX Module #01	1"	1"	0.90 ft/s	0.90 ft/s	5123	5123
GHX Header Section #07	GHX Module #01	3/4"	2"	1.27 ft/s	1.92 ft/s	5766	19758
U Circuit #08	GHX Module #01	1"	1"	0.81 ft/s	0.81 ft/s	4604	4604
GHX Module - Supply Return Pipe	GHX Module #02	2"	2"	2.17 ft/s	2.17 ft/s	22307	22307
U Circuit #01	GHX Module #02	1"	1"	0.87 ft/s	0.87 ft/s	4923	4923
GHX Header Section #01	GHX Module #02	2"	1"	1.91 ft/s	0.87 ft/s	19581	4923
U Circuit #02	GHX Module #02	1"	1"	0.89 ft/s	0.89 ft/s	5072	5072
GHX Header Section #02	GHX Module #02	2"	1 1/2"	1.63 ft/s	0.84 ft/s	16773	6918
U Circuit #03	GHX Module #02	1"	1"	0.89 ft/s	0.89 ft/s	5082	5082
GHX Header Section #03	GHX Module #02	2"	2"	1.36 ft/s	0.81 ft/s	13959	8348
U Circuit #04	GHX Module #02	1"	1"	0.89 ft/s	0.89 ft/s	5067	5067
GHX Header Section #04	GHX Module #02	2"	2"	1.09 ft/s	1.09 ft/s	11154	11154
U Circuit #05	GHX Module #02	1"	1"	0.89 ft/s	0.89 ft/s	5067	5067
GHX Header Section #05	GHX Module #02	2"	2"	0.81 ft/s	1.36 ft/s	8348	13959
U Circuit #06	GHX Module #02	1"	1"	0.89 ft/s	0.89 ft/s	5082	5082
GHX Header Section #06	GHX Module #02	1 1/2"	2"	0.84 ft/s	1.63 ft/s	6918	16773
U Circuit #07	GHX Module #02	1"	1"	0.89 ft/s	0.89 ft/s	5072	5072
GHX Header Section #07	GHX Module #02	1"	2"	0.87 ft/s	1.91 ft/s	4923	19581
U Circuit #08	GHX Module #02	1"	1"	0.87 ft/s	0.87 ft/s	4923	4923

Fig. 11.90 Using the Group Name to Sort Larger Systems

Auto-Optimization Tools

The CFD module provides one more invaluable set of tools for the designer who desires to have the CFD module automatically optimize a piping system of interest. These auto-optimization tools provide the designer with tremendous power and will likely change the way engineers design GHX Fields in the future. The auto optimization tools include:

- The Purging Flow Rate Auto-Optimizer
- The GHX Header Design Optimizer

Both of these tools will be explored in detail below. Note that both of these tools can be activated from the following button when in Purge mode:



The Purging Flow Rate Auto-Optimizer

Calculating the purge rate for a GHX Module, etc. is critical to ensure that an appropriately sized purge pump is available to properly purge a system. Performing these calculations in the past has been time intensive (and sometimes nearly impossible) and required the use of charts, diagrams and a healthy dose of engineering knowhow and experience. Now, the Purging Flow Rate Auto-Optimizer instantly calculates the optimal flow rate (in gpm or L/s) to ensure a user-defined target flow rate is maintained throughout the *GHX Circuits* during the purging process. Note that issues related to the headering pairs are covered below in the Supply/Return Header Design Optimizer.

The designer must first have designed a system in the *Layout* panel before he or she can use the Purging Flow Rate Auto-Optimizer. After the designer has laid out a satisfactory first draft system, the user can activate the Purging Flow Rate Auto-Optimizer from the *Fluid* Panel. The user has to select the 'Auto Adjust' check box (indicating that the CFD module will auto adjust the purging flow rate to achieve the user defined purging target velocity) and then define the minimum purging target velocity. In figure 11.91, the minimum target velocity is set to 2 ft/s, the standard for purging with water. Also note that the purging flow rate is deactivated when the designer chooses 'Auto Adjust'. This is because the program will automatically calculate and display the necessary purging flow rate here after the calculation is completed.

Note that when performing purging calculations, the CFD module *always* uses water properties and ignores the fluid properties selected in the lower half of the *Fluid* panel.

Fluid Information			
Peak Load Flow Rate (gpm):	<input type="text" value="30.00"/>		
Installed Capacity Flow Rate (gpm):	<input type="text" value="60.00"/>		
Purging Flow Rate (gpm):	<input type="text" value="90.00"/>		
	<input type="checkbox"/> Auto-Flow		
	<input type="checkbox"/> Auto-Size		
Purging Target Velocity (ft/s):	Minimum	Maximum	
	<input type="text" value="2.00"/>	<input type="text" value="100.00"/>	

Fig. 11.91 Activating the Purging Flow Rate Auto-Optimizer

Calculating Results with the Purging Flow Rate Auto-Optimizer

The designer may now return to the Layout tab, select the **Purge Results Type** from the dropdown menu and hit the 'Calculate' button again. Results from the 8 GHX Circuit GHX Module described in figure 11.87 above are available for view in figure 11.92 below. Notice how the GHX Circuit (and not the GHX Header section) velocities are all at 2 ft/s or higher. Compare these circuit velocities to those in figure 11.87 above.

Layout Design and Optimization						
Calculate <input type="button" value="Print"/> <input type="button" value="Export"/>						
Name	Pipe 1 Size	Pipe 2 Size	Pipe 1 Velocity	Pipe 2 Velocity	Pipe 1 Reynold's Number	Pipe 2 Reynold's Number
GHX Module - Supply Return Pipe	2"	2"	4.95 ft/s	4.95 ft/s	50787	50787
U Circuit #01	1"	1"	2.04 ft/s	2.04 ft/s	11590	11590
GHX Header Section #01	2"	2"	4.32 ft/s	0.63 ft/s	44369	6417
U Circuit #02	1"	1"	2.02 ft/s	2.02 ft/s	11483	11483
GHX Header Section #02	2"	2"	3.70 ft/s	1.24 ft/s	38012	12775
U Circuit #03	1"	1"	2.01 ft/s	2.01 ft/s	11412	11412
GHX Header Section #03	2"	2"	3.09 ft/s	1.86 ft/s	31693	19094
U Circuit #04	1"	1"	2.00 ft/s	2.00 ft/s	11377	11377
GHX Header Section #04	2"	2"	2.47 ft/s	2.47 ft/s	25393	25393
U Circuit #05	1"	1"	2.00 ft/s	2.00 ft/s	11377	11377
GHX Header Section #05	2"	2"	1.86 ft/s	3.09 ft/s	19094	31693
U Circuit #06	1"	1"	2.01 ft/s	2.01 ft/s	11412	11412
GHX Header Section #06	2"	2"	1.24 ft/s	3.70 ft/s	12775	38012
U Circuit #07	1"	1"	2.02 ft/s	2.02 ft/s	11483	11483
GHX Header Section #07	2"	2"	0.63 ft/s	4.32 ft/s	6417	44369
U Circuit #08	1"	1"	2.04 ft/s	2.04 ft/s	11590	11590

Fig. 11.92 The Purging Flow Rate Has Been Calculated to Provide 2 ft/s Velocities

To see what Purging Flow Rate provides the 2 ft/s minimum velocity, the user may return to the *Fluid* tab. From figure 11.92 it is clear that a flow rate of 68.3 gpm covers the minimum 2 ft/s velocity required for purging air out of the GHX Circuits.

Fluid Information			
Peak Load Flow Rate (gpm):	<input type="text" value="30.00"/>		
Installed Capacity Flow Rate (gpm):	<input type="text" value="60.00"/>		
Purging Flow Rate (gpm):	<input type="text" value="68.3"/>	<input checked="" type="checkbox"/> Auto-Flow	<input type="checkbox"/> Auto-Size
	Minimum	Maximum	
Purging Target Velocity (ft/s):	<input type="text" value="2.00"/>	<input type="text" value="100.00"/>	

Fig. 11.92 68.3 gpm Will Purge the GHX Circuits at 2 ft/s

Properly purging a system of air also requires that the supply and return headering pairs are properly purged. This more complex engineering challenge is addressed by the GHX Header Design Optimizer

The GHX Header Design Optimizer

Properly purging a GHX Header system is more difficult than purging individual GHX Circuits because the GHX Header pairs are of larger diameter and therefore require higher flow rates to ensure a particular purging target velocity is achieved. Higher flow rates require larger and more expensive purging pumps.

To avoid these higher pumping costs, designers usually design and build reducing headers that gradually shrink in diameter across the GHX Module. As the headers shrink in diameter, the velocity is boosted. As a result, the required purging flow rates for a system with reducing headers is lower and costs less than for a system that has uniform diameter pipes across the entire headering system. Therefore, designers in the know design reducing headers for both direct and reverse return systems.

The calculations necessary for determining the predicted flow rates and velocities under different piping design systems is a complex and in some cases monumental challenge. As a result, most loopfield designers understand a few basic systems that work and then use them over and over again. It is very difficult to experiment on paper with a variety of systems because the calculations are onerous. Furthermore, reverse return calculations are impossible to perform by hand or calculator and therefore, flow rate, velocity and Reynold's Number predictions are just that- predictions.

TIP

The GHX Header Design Optimizer solves all of these problems.

Note that while this tool is called the "GHX Header Design Optimizer" it also has the capacity of optimizing the design of Manifolds, Vaults, etc. and does so automatically

To use the GHX Header Design Optimizer, the designer must first return to the *Fluid* Panel and select the 'Auto Adjust' and 'Auto Size' check boxes. As was seen previously, the 'Auto Adjust' check box option enables the CFD module to automatically adjust the purging flow rate to ensure the user defined minimum purging target velocity through the *GHX Circuits*. The 'Auto Size' check box takes this a step further. The 'Auto Size' check box automatically redesigns the Supply-Return headering system by changing pipe diameters as necessary to ensure that the flow rates stay within the user-defined minimum and maximum flow target velocities. At the same time the program is analyzing the pipe diameters, it is analyzing the flow rate as well to ensure an optimal, comprehensive solution.

While the minimum target velocity is a familiar design parameter for many designers, the maximum target velocity may be a new tool in the designer's arsenal. The maximum target velocity impacts the GHX Header Design Optimizer in the following way: if the user specifies a low maximum target velocity (say 5 ft/s) the auto-sizing function has flexibility to choose a larger pipe diameter that offer slower flow rates and lower pressure drops. If the user specifies a higher maximum target velocity (say 50 ft/s), the auto-sizing function will tend to be limited to smaller pipe diameters that enable faster velocities and their concomitant higher head losses.

In figure 11.93, both the 'Auto Adjust' and 'Auto Size' boxes are checked and the minimum and maximum purging target velocities are 2 ft/s and 5 ft/s respectively.

Fluid Information			
Peak Load Flow Rate (gpm):	30.00		
Installed Capacity Flow Rate (gpm):	60.00		
Purging Flow Rate (gpm):	68.3	<input checked="" type="checkbox"/> Auto-Flow	<input checked="" type="checkbox"/> Auto-Size
	Minimum	Maximum	
Purging Target Velocity (ft/s):	2.00	5.0	

Fig. 11.93 Activating the Supply-Return Header Design Optimizer

Pipe Type Controls

There is one more optional step a designer can take before having the CFD module auto design the Supply-Return headering system. Many designers have certain pipe size preferences based on previous experience/ease of purchase, etc. The designer can specify in the CFD module which types of pipe he or she does *not* want the program to use when auto optimizing the piping system.

For example, if a designer cannot use 2 1/2" pipe for whatever reason he or she can navigate to the *Automation* tab and then click on the Pipe Sizes tab. The designer can then de-select any pipes he or she wants to exclude from the auto

designer's database. For example, in figure 11.94, pipe sizes 2 1/2" and 3 1/2" have been de-selected and therefore will not be used in any of the auto designs.

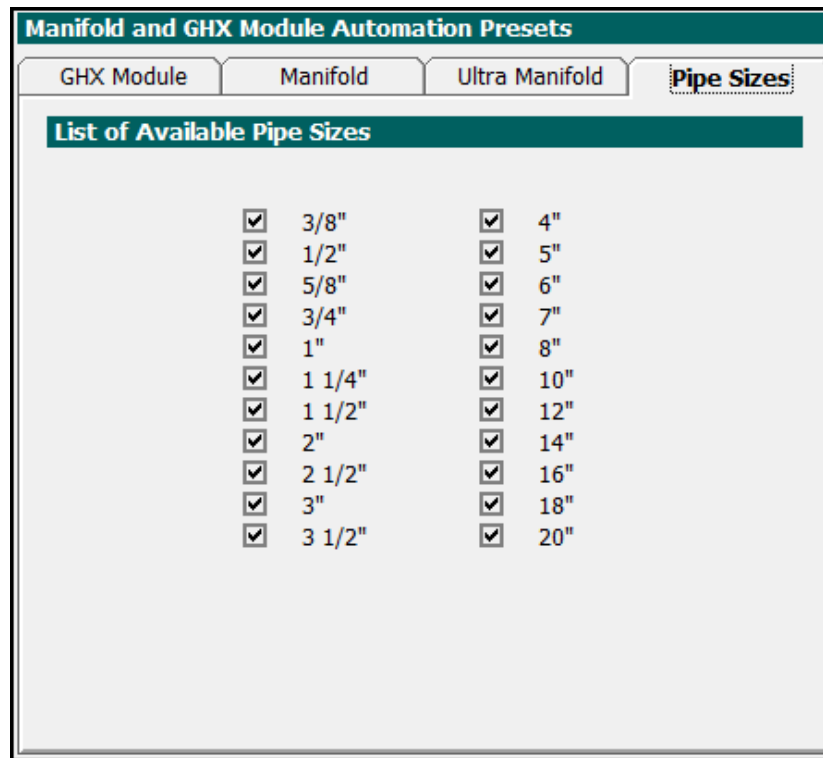


Fig. 11.94 Selecting Pipes to Exclude From the Design Optimizer

Calculating Results with the GHX Header Design Optimizer

The designer may now return to the Layout tab, select the **Purge Results Type** from the dropdown menu and hit the 'Calculate' button again. Results from the 8 GHX Circuit GHX Module described in figure 11.87 above are available for view in figure 11.95 below.

Notice how now the circuits and header sections all have velocities that are at 2 ft/s or higher. Also notice that the header pipe sizes have changed. In the previous example and before the GHX Header Design Optimizer tool was used, all the header pipes were 2" on the supply and return side. Now, the optimized reverse-return headering system has reducing headers that start at GHX Header Section #5 (highlighted) with a 1 1/2" pipe and reduce down to a final 3/4" header section.

Notice also the Header sizes in the Pipe 2 (return pipe) column. They start out at the top with a 3/4" diameter pipe in GHX Header Section #1 and gradually increase until reaching a steady state 2" diameter at GHX Header Section #4. In the Header Sections, the Pipe 1 and Pipe 2 sizes are exact opposite/palindromes.

This is because they are a reverse return system which necessitates such a setup. If the system had been direct return, the layout would appear quite different.



Layout Design and Optimization						
Calculate  						
Name	Pipe 1 Size	Pipe 2 Size	Pipe 1 Velocity	Pipe 2 Velocity	Pipe 1 Reynold's Number	Pipe 2 Reynold's Number
GHX Module - Supply Return Pipe	2"	2"	5.37 ft/s	5.37 ft/s	55098	55098
U Circuit #01	1"	1"	2.00 ft/s	2.00 ft/s	11372	11372
GHX Header Section #01	2"	3/4"	4.75 ft/s	3.14 ft/s	48802	14242
U Circuit #02	1"	1"	2.23 ft/s	2.23 ft/s	12654	12654
GHX Header Section #02	2"	1 1/4"	4.07 ft/s	2.65 ft/s	41795	19033
U Circuit #03	1"	1"	2.25 ft/s	2.25 ft/s	12814	12814
GHX Header Section #03	2"	1 1/2"	3.38 ft/s	3.10 ft/s	34701	25497
U Circuit #04	1"	1"	2.27 ft/s	2.27 ft/s	12916	12916
GHX Header Section #04	2"	2"	2.68 ft/s	2.68 ft/s	27549	27549
U Circuit #05	1"	1"	2.27 ft/s	2.27 ft/s	12916	12916
GHX Header Section #05	1 1/2"	2"	3.10 ft/s	3.38 ft/s	25497	34701
U Circuit #06	1"	1"	2.25 ft/s	2.25 ft/s	12814	12814
GHX Header Section #06	1 1/4"	2"	2.65 ft/s	4.07 ft/s	19033	41795
U Circuit #07	1"	1"	2.23 ft/s	2.23 ft/s	12654	12654
GHX Header Section #07	3/4"	2"	3.14 ft/s	4.75 ft/s	14242	48802
U Circuit #08	1"	1"	2.00 ft/s	2.00 ft/s	11372	11372

Fig. 11.95 An Optimized and Auto-Sized Reverse Return Headering System

The designer can now return to the *Fluid* panel to view the required purging flow rate for this now auto-optimized system. The flow rate of 74.1 gpm can be seen in figure 11.96 below.

Fluid Information	
Peak Load Flow Rate (gpm):	<input type="text" value="30.0"/>
Installed Capacity Flow Rate (gpm):	<input type="text" value="60.0"/>
Purging Flow Rate (gpm):	<input type="text" value="74.1"/>
	<input checked="" type="checkbox"/> *Auto Adjust <input checked="" type="checkbox"/> *Auto Size
Purging Target Velocity (ft/s):	<div>Minimum</div> <div><input type="text" value="2.0"/></div> <div>Maximum</div> <div><input type="text" value="5.0"/></div>

Fig. 11.96 74.1 gpm Will Purge The Optimized System

Adding Circulation Pumps

The designer has the option of adding one or more circulation pumps into his or her piping system. By adding circulation pumps, the CFD module can not only keep track of them individually but can keep track of their cumulative pump power (kW) requirements. This is useful because the circulation pumps for an optimized piping system should ideally consume no more than 10% of the total power consumed by the full system.

Remember that in GLD 2010 the piping system calculations do not include heat pump pressure drop.

In this section we will explore how to add circulation pumps to a design in the Layout Manager Workspace.

Adding A Circulation Pump

TIP

To add a circulation pump, the designer should already have built and tested his or her piping system. After the designer is satisfied with the system, he or she can add a circulation pump by right clicking on the component that will be attached to the circulation pump. A screen similar to the one in figure 11.97 below will appear.

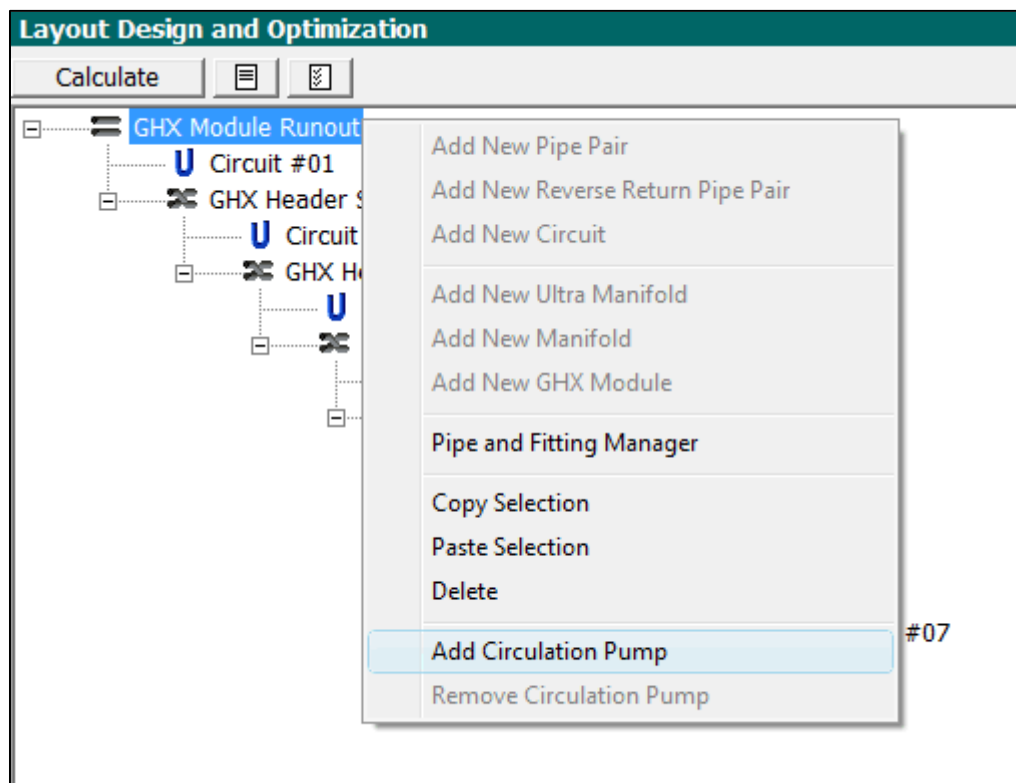


Fig. 11.97 Adding a Circulation Pump

After the user selects "Add Circulation Pump" a circulation pump will appear on the appropriate component as can be seen in figure 11.98 below. Notice the red arrow that appears. This red arrow is the CFD Module symbol for a circulation pump.

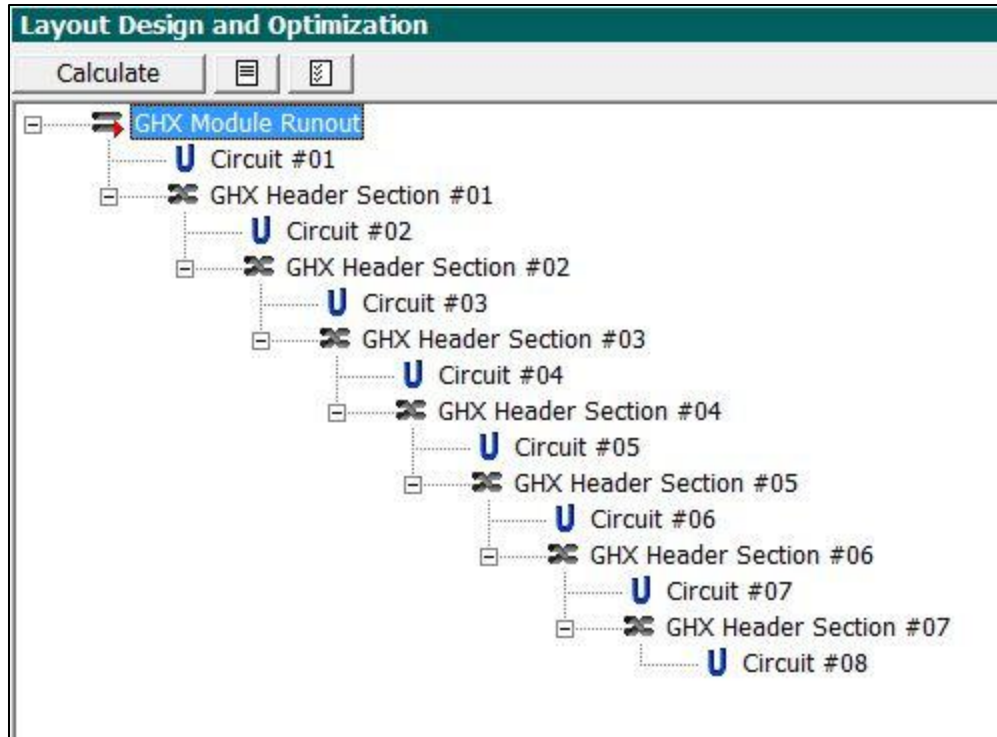


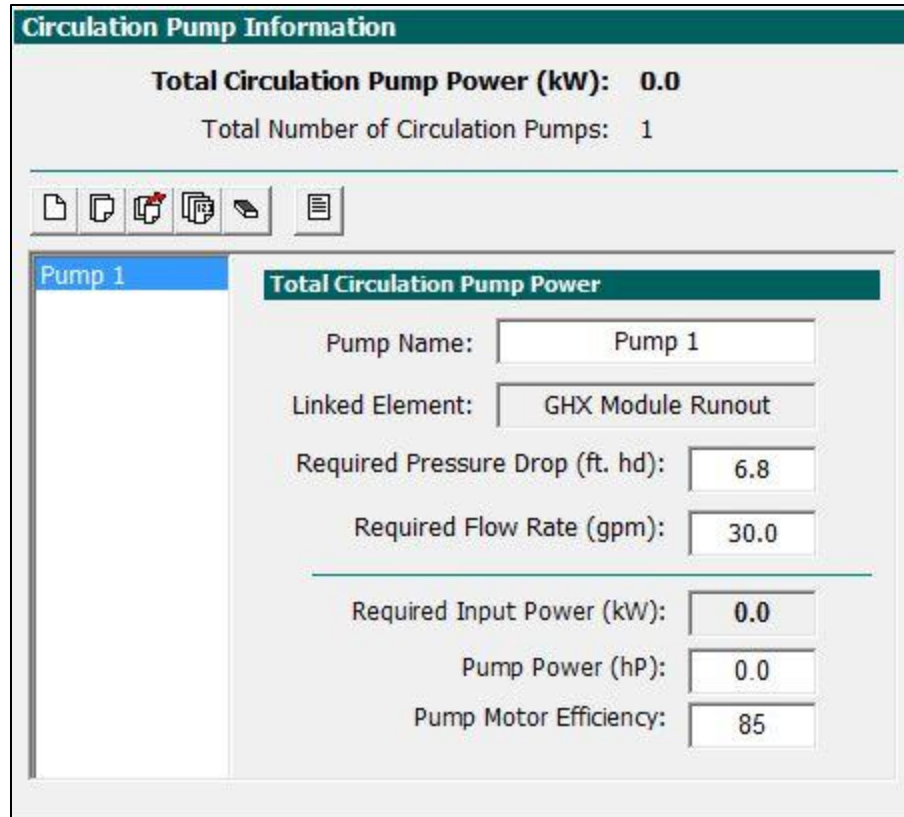
Fig. 11.98 A Circulation Pump Has Been Added

After adding a circulation pump, details regarding the circulation pump (including flow rate and pressure drop) are added automatically to a new circulation pump record in the *Circulation Pumps* panel (see figure 11.99 below)

Notice how the linked component name (the name of the component that has the circulation pump) appears as well as the associated pressure drop and flow rate.



It is important that in the Layout Manager Workspace the designer has selected the flow type of interest (peak, equipment or purge) prior to viewing the circulation pump details in the Circulation Pumps panel. If a designer wishes to have a circulation pump sized for the equipment flow but has selected peak flow, the displayed pump details will be for peak flow and not equipment flow.



Circulation Pump Information

Total Circulation Pump Power (kW): 0.0

Total Number of Circulation Pumps: 1

Pump 1

Total Circulation Pump Power

Pump Name: Pump 1

Linked Element: GHX Module Runout

Required Pressure Drop (ft. hd): 6.8

Required Flow Rate (gpm): 30.0

Required Input Power (kW): 0.0

Pump Power (hP): 0.0

Pump Motor Efficiency: 85

Fig. 11.99 Details of the Added Circulation Pump

In this version of GLD, the user may add the pump power and pump motor efficiency for each circulation pump and then the program will calculate the required input power for the pump.

Future versions of the software likely will have a dynamic circulation pump performance engine included to do this final calculation automatically.

Note that if a user manually enters a pump in the *Circulation Pumps* tabbed panel it will not be associated with a component in the Layout Manager Workspace.

TIP

Note that if a user modifies a piping design in the Layout Manager Workspace after having added a circulation pump, as the fluid dynamics results update in the Layout Manager Workspace, they will also update automatically in the *Circulation Pumps* tab as well.

Note that circulation pumps cannot be added to reverse return pipe-pair components.

Deleting a Circulation Pump

To delete a circulation pump, the designer can right click on a component that has a circulation pump. A screen similar to the one in figure 11.100 below will appear.

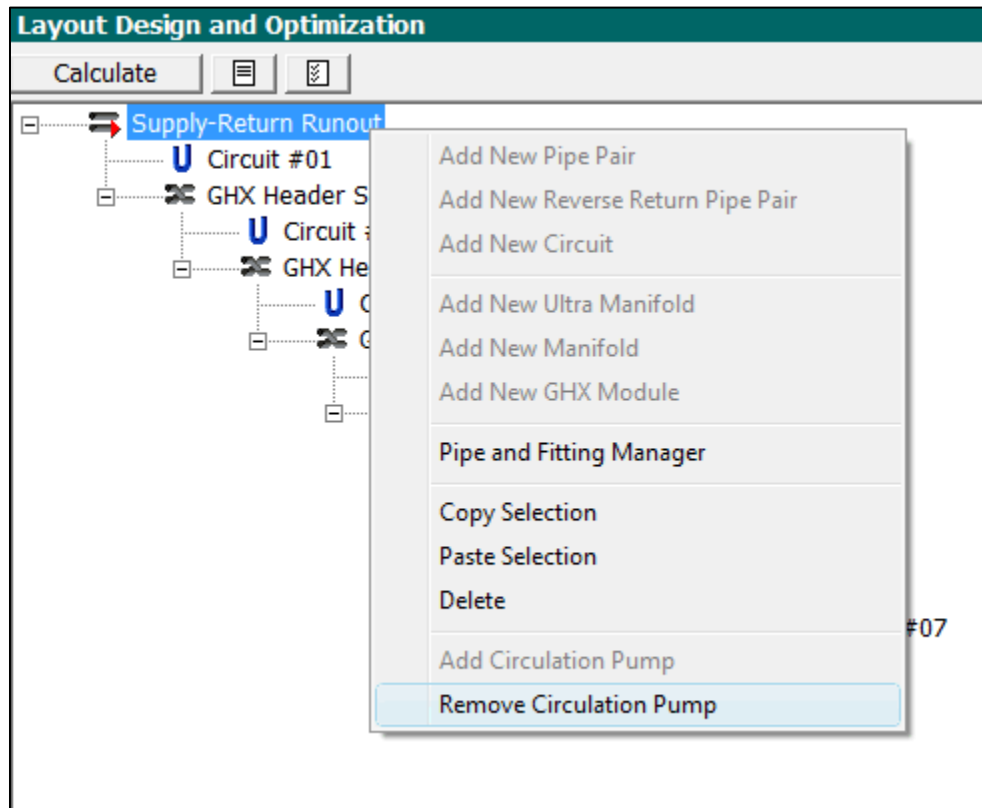


Fig. 11.100 Deleting a Circulation Pump

TIP

After the user deletes the circulation pump, the record for the particular circulation pump will be deleted from the *Circulation Pumps* tabbed panel as well.

Note that if the user manually added a circulation pump directly into the *Circulation Pumps* panel (and not through the Layout Manager Workspace) , then the user must manually delete the circulation pump from the *Circulation Pumps* panel as well.

Printing Reports

Both a CFD piping report and a circulation pump report can be exported from GLD and into a csv file format for easy review and subsequent use in spreadsheet programs.

The process for doing so is as follows:

Exporting a Piping Report

To export a piping report, the user must first complete a design and go to Review mode. An example can be seen in figure 11.101 below.





















Layout Design and Optimization						
Calculate    						
Name	Pipe 1 Size	Pipe 2 Size	Pipe 1 Length	Pipe 2 Length	Pipe 1 Flow Rate	Pipe 2 Flow Rate
 GHX Module - Supply-Return Runout	2"	2"	200.0 ft	200.0 ft	90.00 gpm	90.00 gpm
 Circuit #01	1"	1"	300.0 ft	300.0 ft	11.37 gpm	11.37 gpm
 GHX Header Section #01	2"	2"	20.0 ft	20.0 ft	78.63 gpm	11.37 gpm
 Circuit #02	1"	1"	300.0 ft	300.0 ft	11.27 gpm	11.27 gpm
 GHX Header Section #02	2"	2"	20.0 ft	20.0 ft	67.36 gpm	22.64 gpm
 Circuit #03	1"	1"	300.0 ft	300.0 ft	11.20 gpm	11.20 gpm
 GHX Header Section #03	2"	2"	20.0 ft	20.0 ft	56.16 gpm	33.84 gpm
 Circuit #04	1"	1"	300.0 ft	300.0 ft	11.16 gpm	11.16 gpm
 GHX Header Section #04	2"	2"	20.0 ft	20.0 ft	45.00 gpm	45.00 gpm
 Circuit #05	1"	1"	300.0 ft	300.0 ft	11.16 gpm	11.16 gpm
 GHX Header Section #05	2"	2"	20.0 ft	20.0 ft	33.84 gpm	56.16 gpm
 Circuit #06	1"	1"	300.0 ft	300.0 ft	11.20 gpm	11.20 gpm
 GHX Header Section #06	2"	2"	20.0 ft	20.0 ft	22.64 gpm	67.36 gpm
 Circuit #07	1"	1"	300.0 ft	300.0 ft	11.27 gpm	11.27 gpm
 GHX Header Section #07	2"	2"	20.0 ft	20.0 ft	11.37 gpm	78.63 gpm
 Circuit #08	1"	1"	300.0 ft	300.0 ft	11.37 gpm	11.37 gpm

Fig. 11.101 Preparing to Export a Design

The user must first display the results he or she wishes to export using the display controls described earlier in this chapter. After the desired results are visible the user can hit this button:



and then name a csv file. By default, the file will be exported to the Piping folder.

Exporting a Circulation Pump Report

To export a circulation pump report, the user must first add pumps and then go to the Summary panel in the circulation pumps tab. An example can be seen in figure 11.102 below.








Circulation Pump Information			
Total Circulation Pump Power (kW): 4.4			
Total Number of Circulation Pumps: 2			
      			
Pump Name	Pressure Drop (ft. hd)	Flow Rate (gpm)	Pump Power (kW)
Pump 1	120.3	90.0	1.8
Pump 2	34.0	65.0	2.6

Fig. 11.102 Preparing to Export a Circulation Pump Design

The user can then hit the following button:



and then name a csv file. By default, the file will be exported to the Piping folder.

Concluding Remarks

The new CFD module is a powerful program. We appreciate your feedback and suggestions for the module so that we can continue to improve it over time.