Isight 3.5

Getting Started Guide

Trademarks and Legal Notices

Offices and Representatives

SIMULIA Worldwide Headquarters	Rising Sun Mills, 166 Valley Street, Providence, RI 02909–2499, Tel: +1 401 276 4400, Fax: +1 401 276 4408, simulia.support@3ds.com, http://www.simulia.com
SIMULIA European Headquarters	Gaetano Martinolaan 95, P. O. Box 1637, 6201 BP Maastricht, The Netherlands, Tel: +31 43 356 6906, Fax: +31 43 356 6908, info.europe@simulia.com

Sales, Support, and Services

North America	Central, Main office, West Lafayette, IN, Tel: +1 765 497 1373, simulia.central.support@3ds.com Central, Cincinnati office, West Chester, OH, Tel: +1 513 275 1430, simulia.central.support@3ds.com Central, Minneapolis/St. Paul office, Woodbury, MN, Tel: +1 612 424 9044, simulia.central.support@3ds.com East, Main office, Warwick, RI, Tel: +1 401 739 3637, simulia.east.support@3ds.com Erie office, Beachwood, OH, Tel: +1 216 378 1070, simulia.erie.info@3ds.com Great Lakes, Main office, Northville, MI, Tel: +1 248 349 4669, simulia.greatlakes.info@3ds.com South, Main office, Lewisville, TX, Tel: +1 972 221 6500, simulia.south.info@3ds.com West, Main office, Fremont, CA, Tel: +1 510 794 5891, simulia.west.support@3ds.com
Argentina	Dassault Systèmes Latin America, Buenos Aires, Tel: +54 11 4345 2360, Horacio.Burbridge@3ds.com
Australia	Dassault Systèmes Australia Pty. Ltd., Richmond VIC, Tel: +61 3 9421 2900, simulia.au.support@3ds.com
Austria	Vienna, Tel: +43 1 929 16 25-0, simulia.at.info@3ds.com
Benelux	Huizen, The Netherlands, Tel: +31 35 52 58 424, simulia.benelux.support@3ds.com
Brazil	SMARTTECH Mecânica, São Paulo SP, Tel: +55 11 3168 3388, smarttech@smarttech.com.br SMARTTECH Mecânica, Rio de Janeiro RJ, Tel: + 55 21 3852 2360, smarttech@smarttech.com.br
Czech Republic	Synerma s. r. o., Psáry, Prague-West, Tel: +420 603 145 769, abaqus@synerma.cz
France	SIMULIA France SAS, Main office, Velizy Villacoublay, Tel: +33 1 61 62 72 72, simulia.fr.support@3ds.com Toulouse office, Toulouse, Tel: +33 5 61 16 99 47 (phone for sales only), simulia.fr.support@3ds.com
Germany	Aachen office, Aachen, Tel: +49 241 474 01 0, simulia.de.info@3ds.com Munich office, Munich, Tel: +49 89 543 48 77 0, simulia.de.info@3ds.com
Greece	3 Dimensional Data Systems, Athens, Tel: +30 694 3076316, support@3dds.gr
India	Abaqus Engineering India Pvt Ltd., Main office, Chennai, Tamil Nadu, Tel: +91 44 43443000, simulia.in.info@3ds.com
Israel	ADCOM, Givataim, Tel: +972 3 7325311, shmulik.keidar@adcomsim.co.il
Italy	Milano (MI), Tel: +39 02 39211211, simulia.ity.info@3ds.com
Japan	Tokyo office, Tokyo, Tel: +81 3 5474 5817, tokyo@simulia.com Osaka office, Osaka, Tel: +81 6 4803 5020, osaka@simulia.com
Korea	Mapo-Gu, Seoul, Tel: +82 2 785 6707/8, simulia.kr.info@3ds.com
Malaysia	WorleyParsons Advanced Analysis, Kuala Lumpur, Tel: +603 2039 9405, abaqus.my@worleyparsons.com
New Zealand	Matrix Applied Computing Ltd., Auckland, Tel: +64 9 623 1223, support@matrix.co.nz
Poland	BudSoft Sp. z o.o., Sw. Marcin, Tel: +48 61 8508 466, budsoft@budsoft.com.pl
Russia, Belarus & Ukraine	TESIS Ltd., Moscow, Tel: +7 495 612 44 22, info@tesis.com.ru
Scandinavia	Västerås, Sweden, Tel: +46 21 15 08 70, info@simulia.se
Singapore	WorleyParsons Advanced Analysis, Singapore, Tel: +65 6735 8444, abaqus.sg@worleyparsons.com
South Africa	Finite Element Analysis Services (Pty) Ltd., 7700 Mowbray, Tel: +27 21 448 7608, feas@feas.co.za
Spain	Principia Ingenieros Consultores, S.A., Madrid, Tel: +34 91 209 1482, abaqus@principia.es
Taiwan	Simutech Solution Corporation, Taipei, R.O.C., Tel: +886 2 2507 9550, ariel@simutech.com.tw
Thailand	WorleyParsons Advanced Analysis Group, Bangkok, Tel: +66 2 689 3000, abaqus.th@worleyparsons.com
Turkey	A-Ztech Ltd., Istanbul, Tel: +90 216 361 8850, info@a-ztech.com.tr
United Kingdom	Dassault Systèmes Simulia Ltd., Main office, Warrington, Tel: +44 1 925 830900, simulia.uk.info@3ds.com Dassault Systèmes Simulia Ltd., Kent office, Nr. Sevenoaks, Tel: +44 1 732 834930, simulia.uk.info@3ds.com

Sales Only

North America	East, Mid-Atlantic office, Forest Hill, MD, Tel: +1 410 420 8587, simulia.east.support@3ds.com Great Lakes, East Canada office, Toronto, ON, Canada, Tel: + 1 416 466 4009, simulia.greatlakes.info@3ds.com South, Southeast office, Acworth, GA, Tel: +1 770 795 0960, simulia.south.info@3ds.com West, Pacific Southwest office, Southern CA and AZ, Tustin, CA, Tel: +1 510 794 5891, Ext. 152, simulia.west.info@3ds.com West, Rocky Mountains office, Boulder, CO, Tel: +1 510 794 5891, Ext. 151, simulia.west.info@3ds.com
Finland	Vantaa, Tel: +358 9 2517 8157, info@simulia.fi
India	Abaqus Engineering India Pvt Ltd, Delhi office, New Delhi, Tel: +91 11 65171877, simulia.in.info@3ds.com Abaqus Engineering India Pvt Ltd, Pune office, Tel: +91 20 30483382, simulia.in.info@3ds.com

China Representative Offices

China	Main office, Chaoyang District, Beijing, P. R. China, Tel: + 86 10 65362345, simulia.cn.info@3ds.com Shanghai office, Shanghai, P. R. China, Tel: + 86 21 5888 0101, simulia.cn.info@3ds.com

Complete contact information is available at http://www.simulia.com/locations/locations.html

Isight

Table of Contents

Isight

Preface

This book is your guide to getting started with Isight.

What is Isight?

Isight provides a suite of visual and flexible tools to set up and manage computer software required to execute simulation-based design processes, including commercial CAD/CAE software, internally developed programs, and Excel spreadsheets. The open API supports the development of custom interfaces to link additional in-house and commercial applications by partners and customers.

The rapid integration of these applications and Isight's ability to automate their execution greatly accelerates the evaluation of product design alternatives.

Using advanced techniques such as optimization, DFSS (Design for Six Sigma), approximations, and DOE, engineers are able to thoroughly explore the design space. Advanced, interactive post processing tools allow engineers to see the design space from multiple points of view. Design trade-offs and the relationships between parameters and results are easily understood and assessed, leading to the best possible design decisions.

The process integration and design optimization capabilities in Isight enable design organizations to reduce design cycle time and manufacturing cost, and significantly improve product performance, quality, and reliability.

When used in conjunction with Fiper, an internet-based distributed framework is created, which supports collaboration among geographically distributed engineering and business partners. In this configuration, Isight acts as the "client" and Fiper acts as the "server".

Documentation

The following manuals are available in the Isight and Fiper library:

- *Isight Component Guide*
- *Isight Development Guide*
- *Isight Getting Started Guide*
- *Isight Runtime Gateway Guide*
- *Isight User's Guide*
- *Fiper Installation and Configuration Guide - WebSphere and DB2*
- *Fiper Installation and Configuration Guide - WebSphere and Oracle*
- *Fiper Installation and Configuration Guide - WebLogic and Oracle*
- *Fiper Express Installation Guide*
- *Fiper Federation (B2B) Guide - WebSphere*
- *Fiper WebTop Guide*

Conventions Used in This Book

The following sections describe the typographic terminology and other conventions used in this book.

Typographical Conventions

This book uses the following typographical conventions:

Convention	Explanation
italics	Introduces new terms with which you may not be familiar, and is used occasionally for emphasis.
bold	Emphasizes important information. Also indicates button, menu, and icon names on which you can act. For example, click **Next**.
UPPERCASE	Indicates the name of a file. For operating environments that use case-sensitive filenames (such as UNIX), the correct capitalization is used in information specific to those environments. Also indicates keys or key combinations that you can use. For example, press the ENTER key.
`monospace`	Indicates syntax examples, values that you specify, or results that you receive.
`monospaced italics`	Indicates names that are placeholders for values that you specify. For example, *`filename`*.
forward slash /	Separates menus and their associated commands. For example, Select File / Copy means that you should select Copy from the File menu. The slash also separates directory levels when specifying locations under UNIX.
vertical rule |	Indicates an “OR” separator used to delineate items.
brackets [ ]	Indicates optional items. For example, in the following statement: SELECT [DISTINCT], DISTINCT is an optional keyword. Also indicates sections of the Windows Registry.
braces { }	Indicates that you must select one item. For example, {yes | no} means that you must specify either yes or no.
ellipsis . . .	Indicates that the immediately preceding item can be repeated any number of times in succession. An ellipsis following a closing bracket indicates that all information in that unit can be repeated.

Mouse Conventions

This action...	Means to...
Click	Point to an object with the mouse pointer and momentarily press the left mouse button.
Double-click	Press the left mouse button twice.
Right-click	Momentarily press the right mouse button.
Drag	Press and hold the left mouse button while dragging item(s) to another part of the screen.
SHIFT+Click	Click an object to select it; then, press and hold the SHIFT key. Click another object to select the intervening series of objects.
CTRL+Click	Press and hold the CTRL key; then, click a selection. This lets you select or deselect any combination of objects.

Keyboard Conventions

Select menu items by using the mouse or pressing ALT+ the key letter of the menu name or item.

Platform Information

For complete details on supported platforms, refer to the following website:

http://www.simulia.com/support/sup_systems_info.html

Support

Both technical engineering support (for problems with creating a model or performing an analysis) and systems support (for installation, licensing, and hardware-related problems) for Isight are offered through a network of local SIMULIA support offices. Contact information is listed in the front of each manual.

SIMULIA Online Support System

The SIMULIA Online Support System (SOSS) has a knowledge database of SIMULIA Answers. The SIMULIA Answers are solutions to questions that we have had to answer or guidelines on how to use Abaqus, Isight, Fiper, SIMULIA SLM, and other SIMULIA products. You can also submit new requests for support in the SOSS. All support incidents are tracked in the SOSS. If you are contacting us by means outside the SOSS to discuss an existing support problem and you know the incident number, please mention it so that we can consult the database to see what the latest action has been.

To use the SOSS, you need to register with the system. Visit the My Support page at www.simulia.com for instructions on how to register.

Many questions can also be answered by visiting the Products page and the Support page at www.simulia.com. The information available online includes:

- Link to the SOSS
- Systems information and computer requirements
- Performance data
- Status reports
- Training seminar schedule
- INSIGHTS Magazine
- Technology briefs

Technical Engineering Support

Technical support engineers are available to assist in clarifying product features and checking errors by giving both general information on using the product and information on its application to specific analyses. If you have concerns about an analysis, we suggest that you contact us at an early stage, since it is usually easier to solve problems at the beginning of a project rather than trying to correct an analysis at the end.

Please have the following information ready before contacting the technical engineering support hotline, and include it in any written contacts:

- The release of Isight that are you using, which can be obtained by accessing the VERSION file at the top level of your Isight installation directory.
- The type of computer on which you are running Isight.
- The symptoms of any problems, including the exact error messages, if any.
- Any log files associated with the error.
- Workarounds or tests that you have already tried.

When contacting support about a specific problem, any available product output files may be helpful in answering questions that the support engineer may ask you.

The support engineer will try to diagnose your problem from the model description and a description of the difficulties you are having. The more detailed information you provide, the easier it will be for the support engineer to understand and solve your problem.

If the support engineer cannot diagnose your problem from this information, you may be asked to supply a model file. The data can be attached to a support incident in the SIMULIA Online Support System (SOSS). It can also be sent by means of e-mail, tape, disk, or ftp. Please check the Support Overview page at www.simulia.com for the media formats that are currently accepted.

If you are contacting us to discuss an existing problem, please give the receptionist the support engineer's name if contacting us via telephone or include it at the top of any e-mail correspondence.

Systems Support

Systems support engineers can help you resolve issues related to the installation and running of the product, including licensing difficulties, that are not covered by technical engineering support.

You should install the product by carefully following the instructions in the installation guide. If you encounter problems with the installation or licensing, first review the instructions in the installation guide to ensure that they have been followed correctly. If this does not resolve the problems, consult the SIMULIA Answers database in the SIMULIA Online Support System for information about known installation problems. If this does not address your situation, please create an incident in the SOSS and describe your problem.

Anonymous FTP Site

To facilitate data transfer with SIMULIA, an anonymous ftp account is available on the computer ftp.simulia.com. Login as user *anonymous*, and type your e-mail address as your password. Contact support before placing files on the site.

Contacting Technical Support

Use the My Support page at www.simulia.com, or obtain local support office contact information from the Locations page at www.simulia.com.

In addition, contact information for offices and representatives is listed in the front of this manual.

Support for Academic Institutions

Under the terms of the Academic License Agreement we do not provide support to users at academic institutions. Academic users can purchase technical support on an hourly basis. For more information, please see www.simulia.com or contact your local support office.

Training

SIMULIA offices offer regularly scheduled public training classes, including classes on Isight. We also provide training seminars at customer sites. All training classes and seminars include workshops to provide practical experience with our products. For a schedule and descriptions of available classes, see www.simulia.com or call your local representative.

Feedback

We welcome any suggestions for improvements to Isight software, the support program, or documentation. We will ensure that any enhancement requests you make are considered for future releases. If you wish to make a suggestion about the service or products, refer to www.simulia.com. Complaints should be addressed by contacting your local office or through www.simulia.com.

Isight

1 Installing Isight

This chapter describes how to install Isight on supported Windows and UNIX/Linux platforms. It also discusses how to migrate previous versions of Isight databases. The chapter is divided into the following topics:

Before You Begin

This document describes the installation and configuration of Isight.

Supported Platforms

For complete details on the supported platforms for Isight 3.5, refer to the following website:

http://www.simulia.com/support/sup_systems_info.html

There are UNIX-specific requirements that must be met prior to using Isight. For information, see “Necessary Changes for Executing on UNIX Systems,” on page 157.

Note: You can install 32-bit and/or 64-bit Isight executables on 64-bit Windows and Linux machines. However, it is recommended that you install only the 64-bit executable on a 64-bit operating system.

System Requirements

The following minimum system requirements are recommended:

- 500 MB of free disk space
- 32-bit Installations: minimum 1 GB RAM (2 GB RAM recommended)
- 64-bit Installations: minimum 2 GB RAM (3 GB RAM recommended)

Removing Old Versions of Isight

It is not necessary to remove a previous version of Isight installed on your system. The new installation is placed, by default, into a new version-based directory.

Important: For more information on upgrading from Isight 3.*x* to Isight 3.5-1, refer to the Isight Release Notes on your Isight DVD.

Installing Your License File

If you have received a license file for Isight, be sure that you know the location of the license file on your local machine or the information for the system acting as a license server. You will have to specify this information during the installation process.

If you are installing Isight in a stand-alone environment, the installer will install and start the license server. If you are installing in a network environment, install the license server once on the host machine; then, in other installs enter the host name and port number (optional).

If you are installing only the license server, you will need to run the applicable FLEXnet script for your platform. For more information, see “Installing Only a License Server,” on page 179.

If you do not yet have a license, you can still install the software, but you will not be able to access it. Contact your SIMULIA representative for details on obtaining a license. Furthermore, if you receive your license file after running the Isight installation program, you will have to manually start the license manager once you receive the license. For more information, see “Installing the Isight License After Installation,” on page 170.

Installing as a Non-administrator User on UNIX/Linux

Typically, Isight is installed by an administrator-type user (a user in the Administrators group on Microsoft Windows or “root” on UNIX/Linux).

Important: You must have Administrator privileges to install Isight on all Windows platforms.

In most cases, it is possible to install as a non-administrator user on UNIX/Linux. A non-root user can install Isight in any directory to which he/she has write access. By default, when installing as a non-root user on UNIX/Linux, the installer will try to install in the user's home directory. When installing as root, the installer defaults to installing in directory /opt. If an administrator creates the directory /opt/SIMULIA and makes it writable, a non-root user can also install in /opt.

The only restriction on a non-administrator installation is that the license server cannot be installed as a service. The license server can be installed and started, it just cannot be set up as a service that is automatically started when the machine reboots.

The license server can be installed as a service separately after the install by an administrator-type user. For more information on this procedure, see "Installing the Isight License After Installation," on page 170.

Installing on a Shared/Network Disk

On Microsoft Windows, Isight must be installed on each machine individually. It is not possible to execute Isight from a shared folder. However, the Isight installation program can be placed on a shared folder. This allows for individual machine installs without having to manually insert the DVD into each machine.

On Unix and/or Linux, it is common to install Isight on a shared disk (i.e., network file system, NFS, or NAS). Therefore, you just install once from a machine that has a DVD drive onto a network file system. When installing on Unix and/or Linux, the installer has an option to install the support for any number of platforms. This option allows the central shared install to be used on all supported machines.

Installing Isight on Windows

This section is divided into the following topics:

Understanding Assumed Default Windows Settings

Windows operating systems have various options that allow you to control the appearance of certain components on your system. Three of these components (the Start menu, the Control Panel, and folder and file extension options) can affect the installation process for Isight. All of the procedures in this book assume that you are using the default Windows settings for each item. If you are using other settings for these components, you may have difficulty following the installation procedures. Follow the instructions below to determine which setting you are currently using.

Important: It is recommended that you change the settings as described in the following procedures, if necessary. Doing so will help you avoid any possible confusion when installing Isight. After installing the software, you can revert back to your old settings.

Determining the Start Menu Setting

To determine the Start menu setting:

1 Right-click the **Start** button; then, select the **Properties** option. The Taskbar and Start Menu Properties dialog box appears. Two menu options are listed on this dialog box.

2. Verify that the Start Menu tab is selected. For installing Isight, it is recommended that you use the **Start menu** option (the first option). This option is the default system setting.

3 Change the current setting, if necessary; then, click **OK**.

Determining the Control Panel Setting

To determine the Control Panel setting:

1 Click the **Start** button; then, click the **Control Panel** option. This step assumes that you are using the default Windows Start Menu option as described in "Determining the Start Menu Setting," on page 21. The Control Panel dialog box appears.

2. Examine the view setting in the upper left corner of the dialog box. One of the following two options will be listed (based on your operating system):

   - Windows XP
     - **Switch to Category View**. If this option is displayed, then the Control Panel is currently in Classic View. For simplicity, it is recommended that you click this option to switch the Control Panel to Category View.
     - **Switch to Classic View**. If this option is displayed, then the Control Panel is configured correctly to correspond with the installation procedures in this book. No changes are necessary prior to installing Isight.
   - Windows Vista
     - **Control Panel Home**. If this option is displayed in bold text with a small dot to its left, then the Control Panel is configured correctly to correspond with the installation procedures in this book. No changes are necessary prior to installing Isight.
     - **Classic View**. If this option is displayed in bold text with a small dot to its left, then the Control Panel is currently in Classic View. For simplicity, it is recommended that you click the **Control Panel Home** option.

3 Proceed to the next section.

Determining Folder and File Extension Options

There are times, such as when deleting the software, that you must remove certain temporary directories that are hidden by default. Also, all of the files in each of the procedures in this book are listed with their file extensions (*.exe, *.bat, etc.). To avoid any confusion, it is recommended that you set your system to show hidden directories and file extensions.

To show hidden directories and file extensions in Windows:

1. Access the Control Panel dialog box as described in "Determining the Control Panel Setting," on page 22.
2. Access the Folder Options dialog box using one of the following methods, based on your operating system:
   - **Windows XP**: On the Control Panel dialog box, click the **Appearance and Themes** link and click the **Folder Options** link.
   - **Windows Vista**: On the Control Panel dialog box, click the **Appearance and Personalization** link and click the **Folder Options** link.
3. Click the **View** tab; then, click the **Show hidden files and folders** radio button.
4. Clear (uncheck) the check box for the **Hide extensions for known file types** option.
5. Click **OK**. The hidden directories and file extensions will now appear when browsing through your system file structure.

Installing the Software

This section describes how to install Isight on a Windows system.

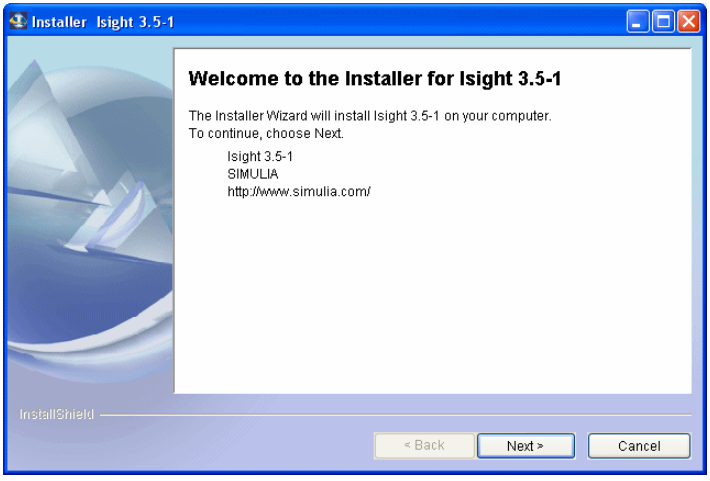
To install Isight:

1. Log in as the Administrator that will install Isight.

   Note: You *must* be the Administrator or have administrative privileges to install on any Windows platform.
2. Insert the Isight DVD.

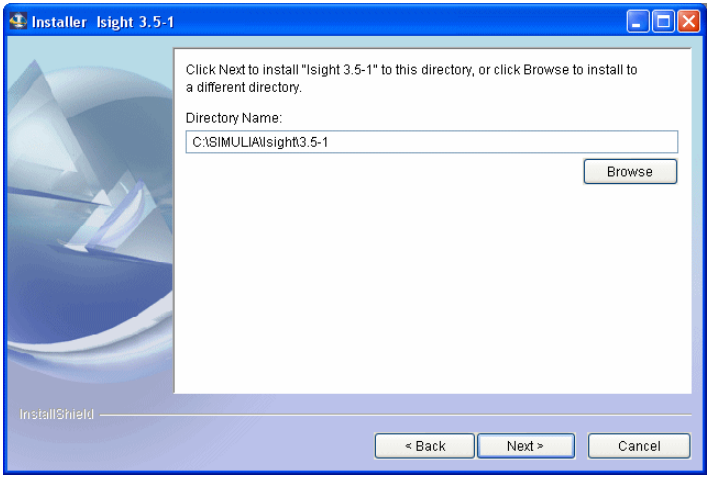
   Note: If installing on Windows Vista, a dialog box appears asking you to confirm the execution of the installation program. Click **Allow** to start the program.

The Isight install program starts and the splash screen appears. The install program will guide you through the installation process. The Welcome screen shows the version of Isight being installed.

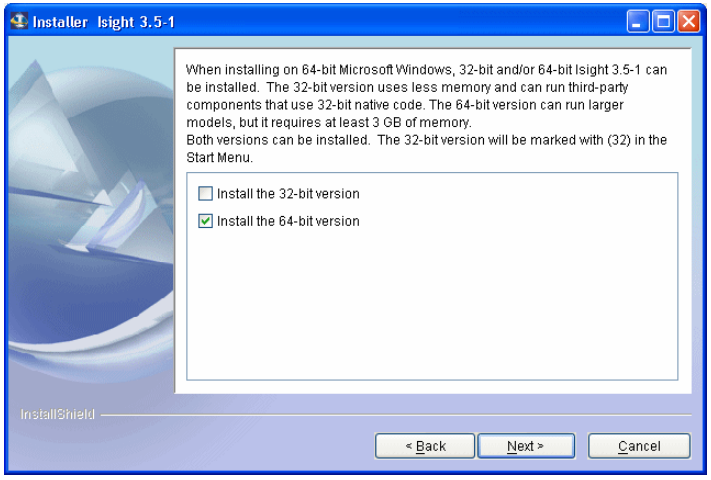


Note: If the installer does not start automatically, execute the `setupwin32.exe` file located on the top level of the Isight DVD. This installation program is used for both the 32-bit and 64-bit versions of the software.

3. Click **Next**. The legal notice agreement screen appears.
4. Read the notice; then, click **Next**. The installation directory screen appears.

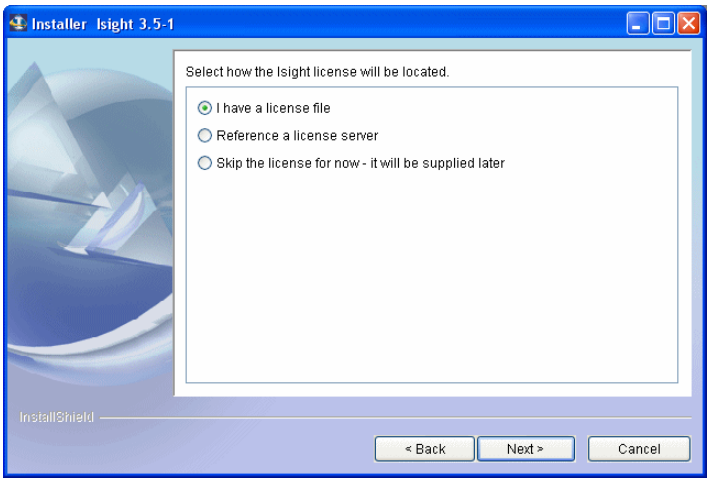


5. Perform one of the following actions:
   - To accept the default directory, click **Next**. Your default directory may vary based on your system settings and permissions.
   - If you wish to place Isight in a different location, click the **Browse** button, and choose a destination folder; then, click **Open**. Remember, through the rest of the installation procedures, the default directory is used for defining installation paths. Be sure to use your specific directory in place of this default directory structure.
6. Click **Next** to continue the installation procedure once you have specified your directory.
7. Perform one of the following options:
   - If you are installing on a 32-bit system, proceed to step 10.
   - If you are installing on a 64-bit system, proceed to step 8.
8. Click **Next**. The bit version screen appears.



9. Determine which bit version of Isight you want to install. The 32-bit version uses less memory and can run third-party components that use 32-bit native code. The 64-bit version can run larger models, but it requires at least 2 GB of memory. You can install one version or both versions. The 32-bit version is designated with "(32-bit)" in the Windows Start menu.

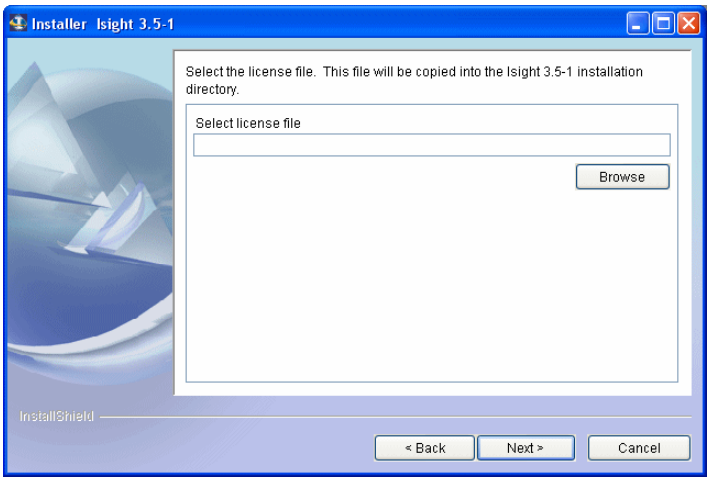
10. Click **Next**. The license location screen appears.



11. Select how the license file will be located. The following options are available:

- **I have a license file**. If you select this option, proceed to step 12.
- **Reference a license server**. If you select this option, proceed to step 15.
- **Skip the license for now - it will be supplied later**. If you select this option, you will be required to manually start your license server once you receive your license file as described in "Installing the Isight License After Installation," on page 170. Proceed to step 17.

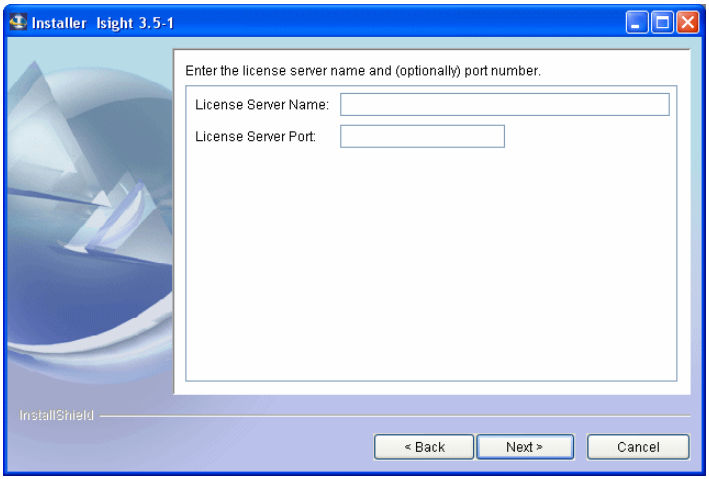
12. Click **Next**. The select license file screen appears.



13. Enter the full path and name of the license file or use the **Browse** button to locate the file.

14. Proceed to step 17.

15. Click **Next**. The license server screen appears.

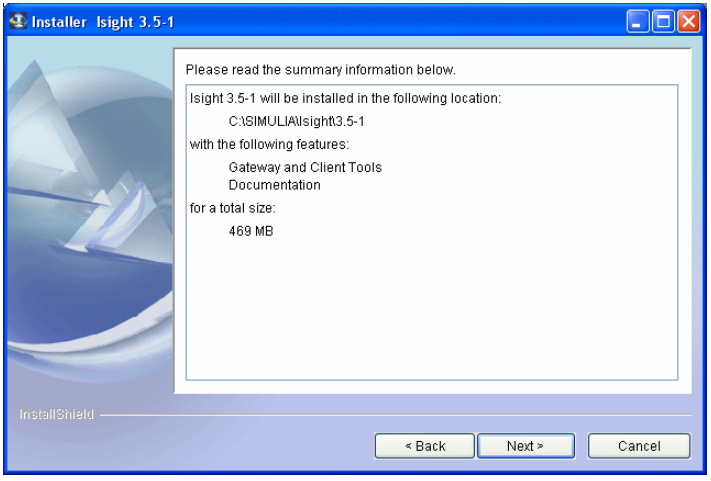


16. Enter the following information. It is needed to contact a license server, and it is used to create a `license.dat` file that references a license server.

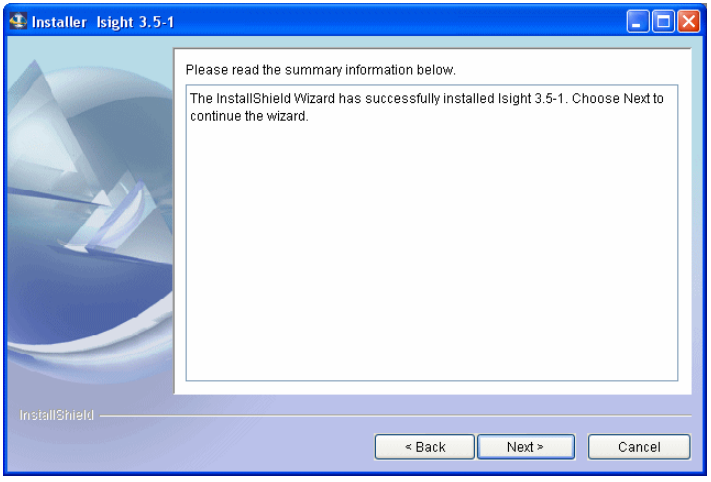
    - **License Server Name**. The host name of the machine running the license server.
    - **License Server Port**. The port is found in the license file. The default is to leave this setting empty (Isight's license manager software selects the port number).

    Important: If the license server you are referencing is behind a Windows Firewall (the firewall supplied with most Windows installations), you will need to manually alter your Isight license. For more information, see "Configuring Your License to Work with Windows Firewall," on page 178.

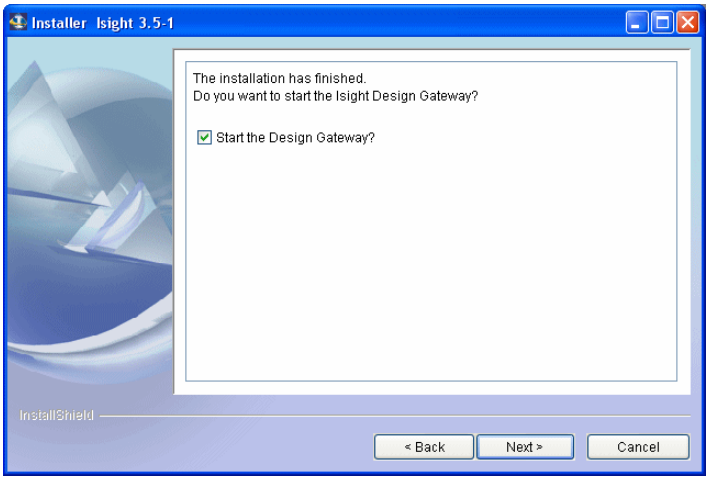
17. Click **Next**. After the installation program calculates the required disk space needed, an installation summary screen appears showing what will be installed and the location.



18. Click **Next**. The installation progress screen appears, and the software is installed. An overall summary message appears when the installation is complete.



19. Click **Next**. If a license file or license server is specified, the Engineous-supplied components are published to the local library. Once the publishing is complete, a message appears stating that the installation is complete.



Note: If you decided to provide Isight with a license at a later time (as specified in step 11), a message appears providing you with your host name and host ID. This information is necessary for creating a license for your system and should be recorded and provided to your SIMULIA representative. Once you have recorded the information, click **Finish**; then, proceed to step 22.

If an earlier version of an Isight database is detected, a database migration program is executed to migrate to the current format. If a Database Migration wizard appears, see "Migrating Isight Databases," on page 38.

20. Determine whether or not you want to start the Isight Design Gateway in Standalone mode as soon as the installation is complete using the corresponding check box.

21. Click **Next**. The installation is complete.

22. If necessary, start the license manager for Isight. Typically, this process is performed automatically. However, there are some cases where it must be accomplished manually, including if you decided to provide Isight with a license at a later time (as specified in step 11). Be sure that you have acquired a license before starting the license manager. For more information, see "Installing the Isight License After Installation," on page 170.

Installing Isight on UNIX/Linux

This section describes how to install Isight on AIX, HP-UX, Solaris, and Linux systems. If your machine does not have a DVD drive, you can install Isight from a shared disk or network folder. For more information, see "Accessing the Installation Files from a Remotely Mounted DVD," on page 155.

Important: It is highly recommended that you review "Necessary Changes for Executing on UNIX Systems," on page 157 for specific requirements which must be met *prior* to installing Isight on UNIX-based systems.

To install Isight on UNIX/Linux:

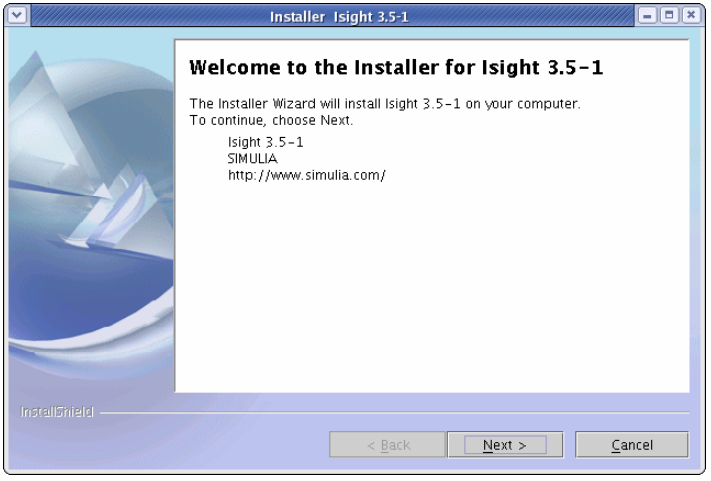
1. Log in as the user that will install Isight. If you are not installing as "root", it is recommended that you review the information in "Installing as a Non-administrator User on UNIX/Linux," on page 19 before beginning your installation.
2. Verify that you have reviewed the pre-installation requirements described in "Necessary Changes for Executing on UNIX Systems," on page 157.
3. Load the Isight DVD and mount it, if necessary. For more information, contact your systems administrator.

   Note: On some UNIX systems, the Isight DVD will automatically mount when it is inserted into the DVD drive, and a file browser window will appear. In this case, you can double-click the correct setup file. These files are listed in step 5.
4. From the terminal window (shell), change to the directory where the Isight DVD is mounted. For example, type:

   ```
   cd /media/dvd
   ```
5. Execute the installation script. Type one of the following commands, based on your operating system:

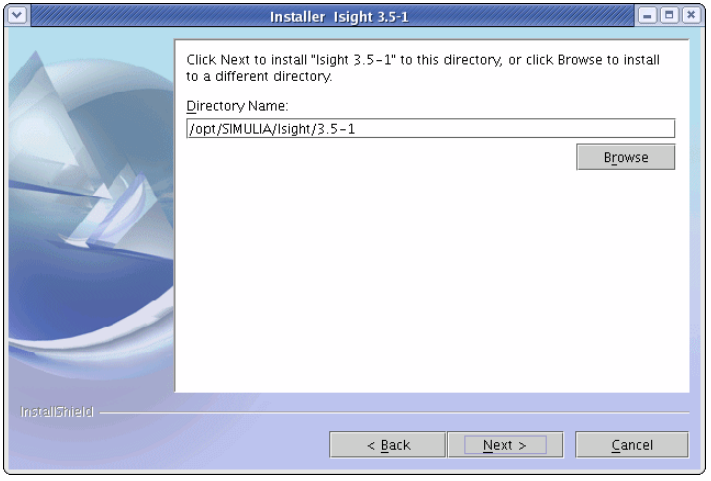
   - `./setupaix`
   - `./setuphpux`
   - `./setuplinux`
   - `./setupsolaris`

The Isight setup program starts. It will guide you through the rest of the installation process. The Welcome screen appears, showing the version of Isight being installed.



6. Click **Next**. The legal notice agreement screen appears.

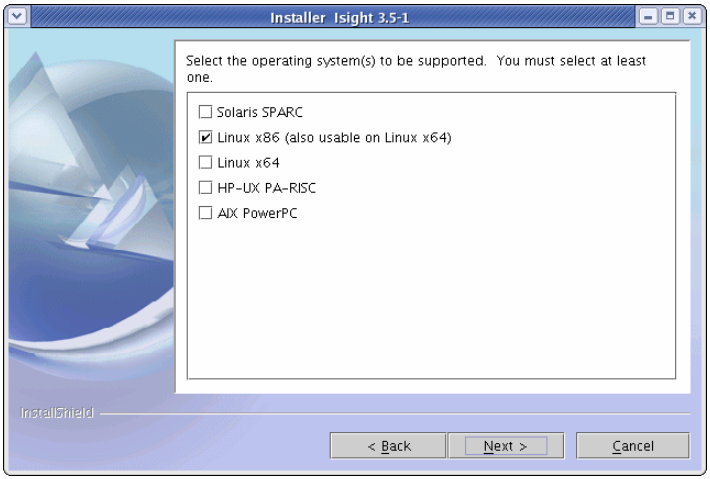
7. Read the notice; then, click **Next**. The installation directory screen appears.The installation directory screen appears.



8. Perform one of the following actions:

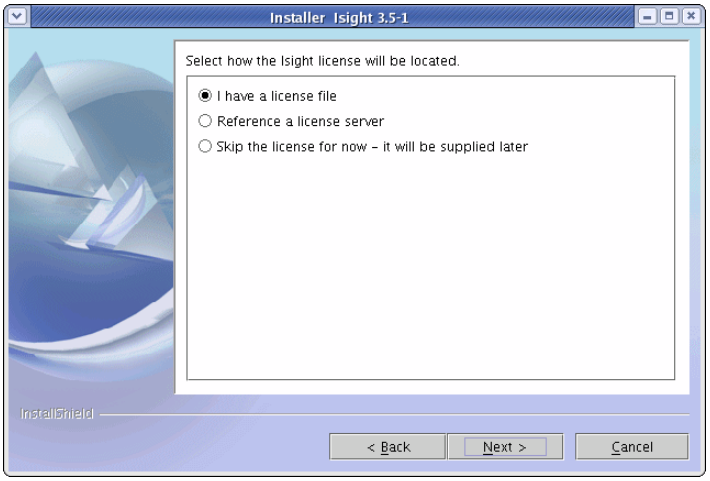
   - To accept the default directory, click **Next**. Your default directory may vary based on your system settings and permissions.
   - If you wish to place Isight in a different location, click the **Browse** button, and choose a destination folder; then, click **Open**. Remember, through the rest of the installation procedures, the default directory is used for defining installation paths. Be sure to use your specific directory in place of this default directory structure.

9. Click **Next**. The operating system selection screen appears.



10. Verify that the appropriate operating system is selected. You can choose to install one or multiple operating systems.

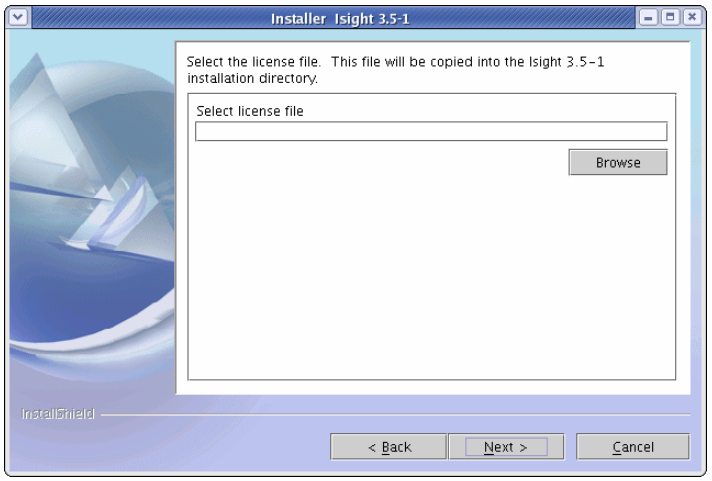
11. Click **Next**. The license location screen appears.



12. Select how the license will be located. The following options are available:

- **I have a license file**. If you select this option, proceed to step 13.
- **Reference a license server**. If you select this option, proceed to step 16.
- **Skip the license for now - it will be supplied later**. If you select this option, you will be required to manually start your license server once you receive your license file as described in "Installing the Isight License After Installation," on page 170. Proceed to step 18.

13. Click **Next**. The license file selection screen appears.

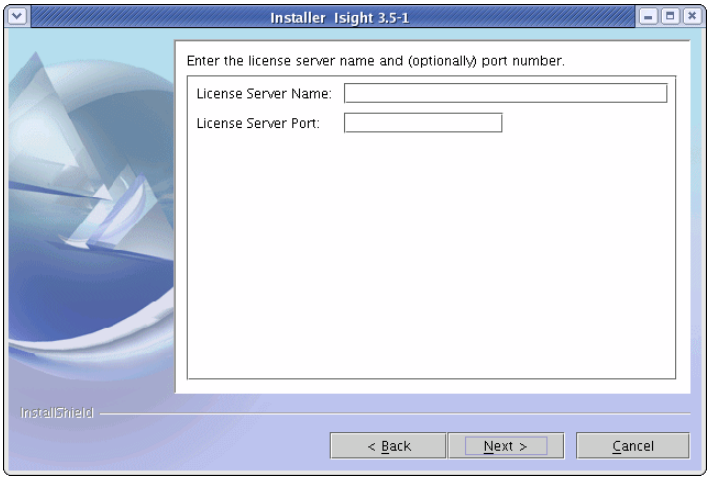


14. Enter the full path and name of the license file or use the **Browse** button to locate the file.

    Note: The license file must be for the computer the installation is running on if the license server is to be installed as a daemon.

15. Proceed to step 18.

16. Click **Next**. The license server screen appears.

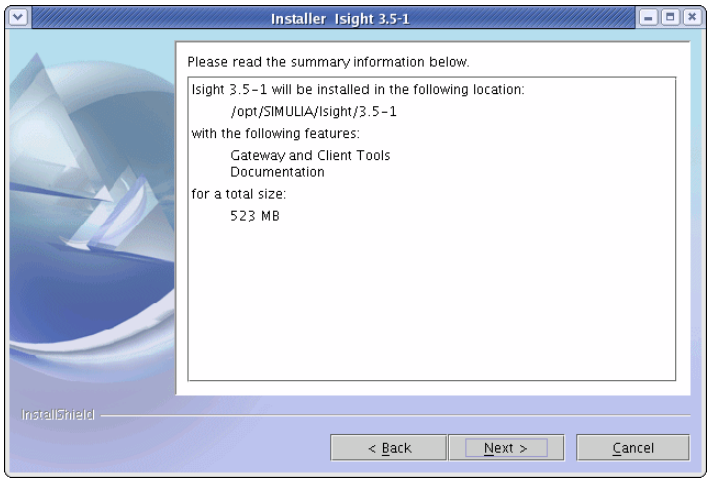


17. Enter the following information. It is needed to contact a license server, and it is used to create a `license.dat` file that references a license server.

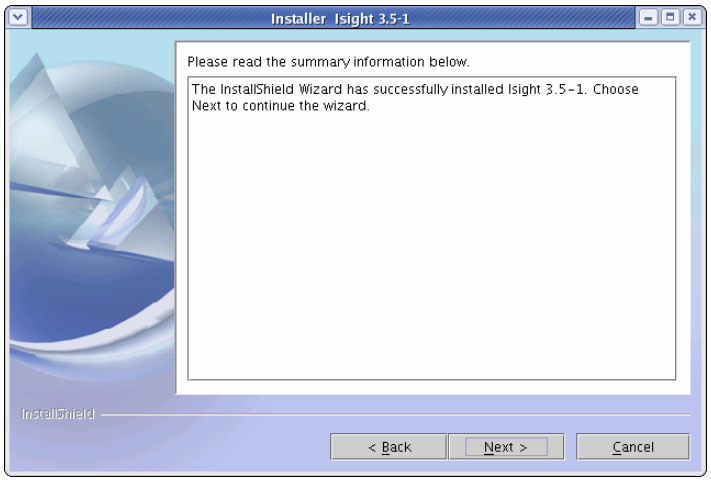
    - **License Server Name**. The host name of the machine running the license server.
    - **License Server Port**. The port is found in the license file. The default is to leave this setting empty (Isight's license manager software selects the port number).

    Important: If the license server you are referencing is behind a Windows Firewall (the firewall supplied with most Windows installations), you will need to manually alter your Isight license. For more information, see "Configuring Your License to Work with Windows Firewall," on page 178.

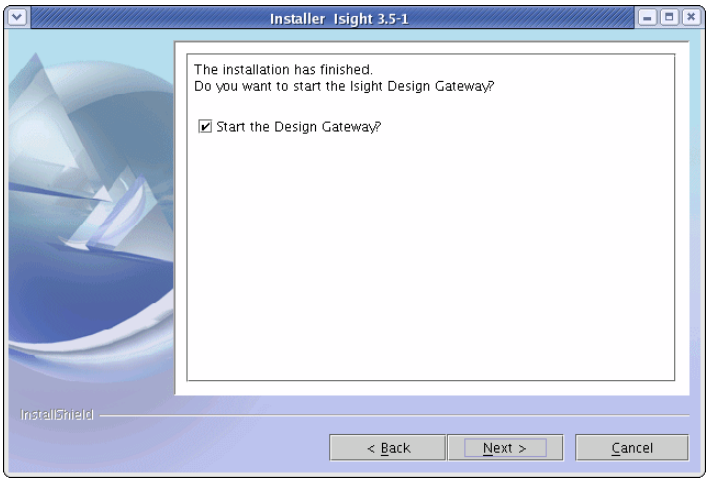
18. Click **Next**. After the installation program checks for required disk space, a summary of the installation appears.



19. Click **Next**. The installation progress screen appears, and the software is installed. An overall summary message appears when the installation is complete.



20. Click **Next**. If a license file or license server is specified, the Engineous-supplied components are published to the local library. Once the publishing is complete, a message appears stating that the installation is complete.

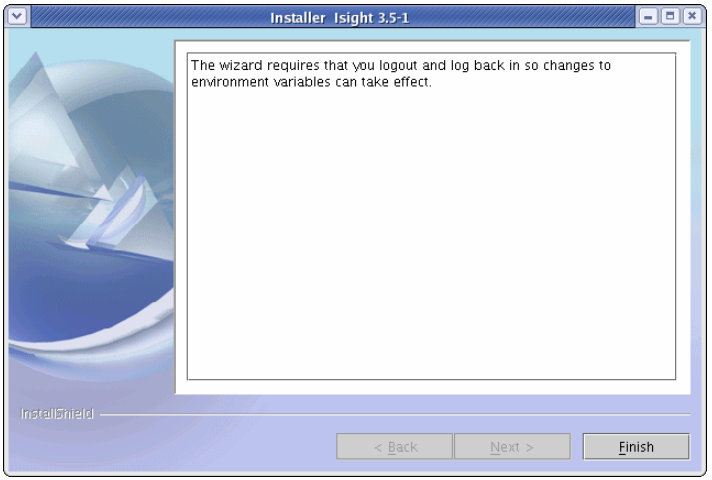


If an earlier version of an Isight database is detected, a database migration program is executed to migrate to the current format. If a Database Migration wizard appears, see "Migrating Isight Databases," on page 38.

Note: If you decided to provide Isight with a license at a later time (as specified in step 12), a message appears providing you with your host name and host ID. This information is necessary for creating a license for your system and should be recorded and provided to your SIMULIA representative. Once you have recorded the information, proceed to step 22.

21. Determine whether or not you want to launch the Isight Design Gateway in Standalone mode as soon as the installation is complete using the corresponding check box.

22. Click **Next** to complete the installation. You are notified when installation is complete.



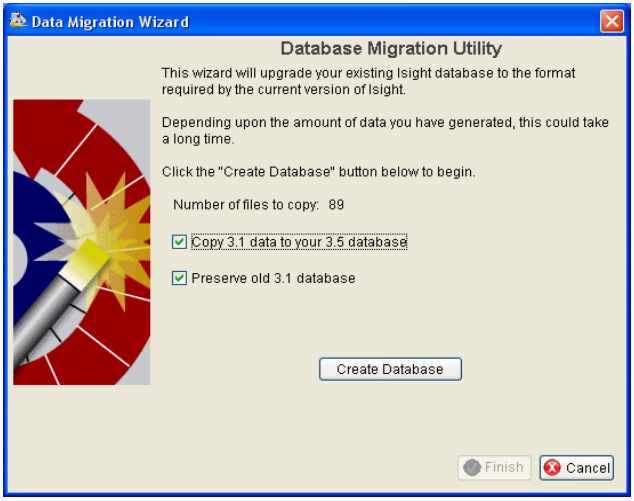
23. Review the final installation message(s).

24. Click **Finish** to complete the installation.

25. If necessary, start the license manager for Isight. Typically, this process is performed automatically. However, there are some cases where it must be accomplished manually, including if you decided to provide Isight with a license at a later time (as specified in step 12) or if you do not have root privileges. For more information, see "Installing the Isight License After Installation," on page 170.

Migrating Isight Databases

If Isight detects a previous version of an Isight database, a migration utility executes, during the installation, to update the database to the Isight 3.5 format. Then, a Data Migration Wizard appears.



The wizard shows the number of files that will be copied. Depending on the number of files, this process may take a long time.

1. Determine if you want to make the data from the previous version of Isight available in 3.5. You may also choose to preserve the previous data in its original format.

   Important: If you choose to preserve the previous data, it is recommended that you not use the earlier version of Isight. Both databases use the same port number which would result in corrupt data.

   Note: You may click Cancel to return to the install process, but upon starting Isight for the first time you will be prompted to migrate the database.

2. Click **Create Database**. Once the migration is complete, a Database completed successfully message appears. You will need to click Create Database even if you choose not to migrate your database and/or preserve 3.1 data.

   Important: If you choose to migrate your previous database, do not click Cancel during the migration. This will result in corrupt data and you will have to manually

delete your previous database. The database is located in the following directory by default:

C:\Documents and Settings*username**fiperdbxx*

Username is the name of the user who installed Isight and *xx* is the version that you want to delete.

3. Click **Finish**; then, return to step 20 in "Installing Isight on Windows" or step 21 in "Installing Isight on UNIX/Linux", depending on the platform on which you are installing Isight.

Isight

2 Getting Acquainted with Isight

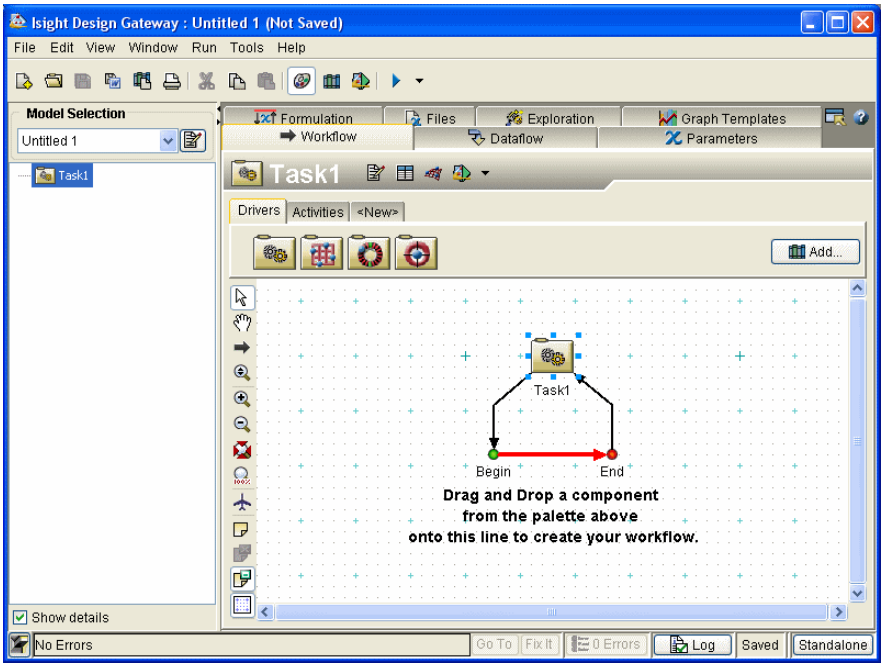
This chapter defines Isight and describes the main interfaces used to create and execute models in the Isight environment. It is divided into the following topics:

What is Isight?

Isight is a service-oriented product development environment, which provides an open, flexible means to incorporate existing analysis and design tools/methods and makes them available to others. It also provides a common, standard way to model your analysis and design process in conjunction with your product data.

The Design Gateway

The Isight Design Gateway is shown below.



This interface is the main Isight interface. It allows you to create models, manipulate components, and perform other functions associated with model design and development.

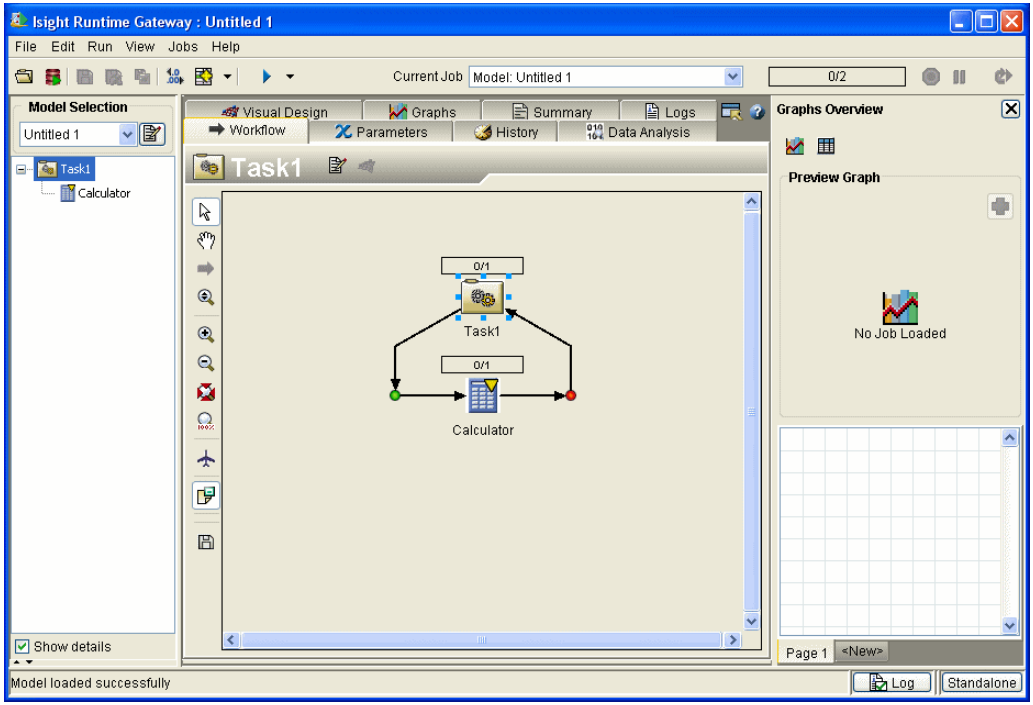
This interface allows you to perform the following basic actions:

- You can drag-and-drop components from the component tabs to the Workflow tab to create a model workflow. The component tabs (Drivers, Activities, etc.) allow you to arrange your components in a way that is convenient for you. You can even create custom tabs using the <New> tab.
- Once added to the workflow, you can double-click any component to access its editor.
- The menu and toolbar buttons allow access to additional features (Publish, Preferences, Run, etc.) as well as other interfaces (Runtime Gateway, Generator, etc.).

For more information on this interface Design Gateway, refer to the *Isight User's Guide*.

The Runtime Gateway

The Isight Runtime Gateway is shown below.



This interface differs from the Design Gateway in that it allows you to control execution and view results, but it only allows limited access to components and the workflow. It also lets you create graphs and tables, view parameter information, create approximations, access data analysis tools, and resubmit models for execution. The following main features are available:

- Workflow tab. Allows you to view the overall model workflow and access component editors.
- Parameters tab and History tab. Allow you to view results information for individual parameters.
- Graphs tab and Data Analysis tab. Provides access to graphs, tables, and post processing tools for viewing results.

- Visual Design tab. Allows you to create and view approximations.
- Logs tab. Provides execution information and may be helpful when errors occur during execution.
- Run menu. Allows you to execute the model workflow repeatedly, without having to return to the Design Gateway.

Using Post Processing Tools

While some of the post processing tools are discussed in the remaining chapters, there are many other tools that can assist you in viewing and analyzing results. These include Problem Formulation (runtime grading) and parameter filtering. Proceed to one of the following sections:

- "Understanding Problem Formulation (Runtime Grading)" on this page
- "Understanding Parameter Filtering," on page 46

Understanding Problem Formulation (Runtime Grading)

Problem formulation allows you to define a problem to be solved for each process component in a model, which by default will be inherited from the parent process component (only for the parameters that exist at that level).

This feature allows you to define the following information for scalar parameters and array elements:

- Lower/upper bounds
- Objective (minimize, maximize, target)
- Scale and weight factors (for objectives and bounds separately)

The problem formulation can be used to grade the designs for any of the components in that subflow. The Runtime Gateway uses this formulation to visually indicate how the runs of any component measure up against the stated objectives/constraints.

Typically, the formulation is defined using the Design Gateway. However, you can also define problem formulation on the Runtime Gateway. For more details about runtime grading, refer to the *Isight Runtime Gateway Guide*. For more information on using

this feature in the Design Gateway, as well as detailed information on how Isight uses the feature, refer to the *Isight User's Guide*.

Understanding Parameter Filtering

You can use the Runtime Gateway Data filter feature to filter the history of runs for the selected component, reducing the amount of information displayed, which may allow you to better analyze the execution data. Filtering can be based on the desired minimum/maximum values you want included for any parameters and/or based on the "grading" of the runs using the problem formulation (lower/upper bounds, objectives). The filtered data set is what is displayed on the Parameters, History, Data Analysis, and Graphs tabs.

Note: Some of the post-processing graphs from design driver component (for example, the Main Effects graphs for DOE) still use the full, unfiltered data set even after filtering.

For more details about parameter filtering, refer to the *Isight Runtime Gateway Guide*.

Components

Components are used as building block for models. Excel, Simcode, and DOE are examples of some of the components that Engineous has developed and can be included with Isight. These components are used in the examples included in this book.

You can also develop your own components to use within the Isight environment. For more information on using all Engineous-provided components, refer to the *Isight Component Guide*. For more information on creating custom components, refer to the *Isight Development Guide*.

Isight

3 Simple Example

This chapter demonstrates a simple Isight model that consists of wrapping an Excel spreadsheet in Isight to create a simple workflow with a single execution. Then, a process component (Loop) is added to further control the execution. The Excel workbook called `SpringCalcs.xls` is used. This file is included in your Isight installation.

Important: Since this example uses an Excel worksheet, it will only work on a Windows operating system. If you are using a UNIX or Linux operating system, proceed to Chapter 4 “Simcode Example”.

The following tasks are demonstrated:

- Creating a new model using the Task component and adding an Excel component
- Configuring an Excel component (specifying a workbook and mapping parameters using different methods)
- Executing the model
- Viewing results using the Summary and History tabs on the Runtime Gateway
- Changing the Task component to a Loop component
- Configuring a Loop component
- Re-executing the model
- Viewing additional results using the Summary and History tabs on the Runtime Gateway

The problem used for this example is a classical tension-compression spring problem. The spring is to be designed for a minimum mass subject to constraints on minimum deflection, shear stress, surge frequency, and limits on the outside diameter and on the design variables, given a 10 lbs. load applied to the spring. The wire diameter, d, coil diameter, D, and the number of coils, n, are the design variables.

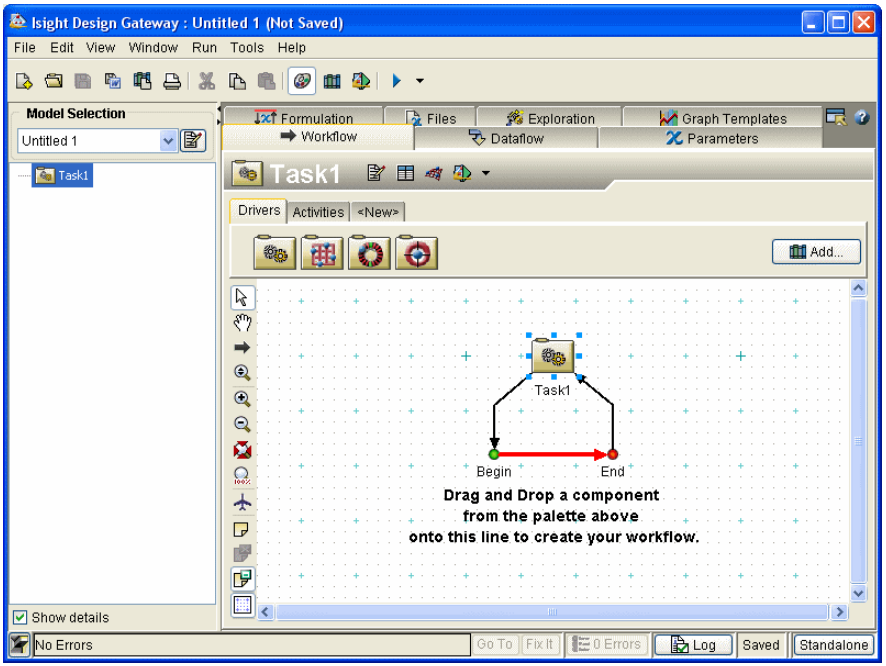
This example will be run using Isight in Standalone mode.

Accessing the Design Gateway

A model is a collection of components combined in such a manner as to "model" a problem to solve and a process to be used in solving that problem, whether it is analysis or analysis and design.

To start Isight and create a model:

1. Click the Windows **Start** button on the task bar at the bottom of your desktop; then, point to **All Programs** / **Isight 3.5** and click the **Design Gateway** option.
2. If the Logon dialog box appears, select the **Standalone** connection profile; then, click **OK**. For more information on logon options, refer to the *Isight User's Guide*. The software continues to load.
3. Click **OK**. The Design Gateway appears with a default Task component already added to the model's workflow.



4. Proceed to "Creating a Model," on page 49.

Creating a Model

The default “starter” workflow is already in place on the Design Gateway, with a Task component added. Now you need to specify additional components to complete the workflow. In this example, you’ll add an Excel workbook using the Excel component editor. This editor allows you to specify a workbook (which is then loaded in the component editor’s emulator) and map parameters from the contents of the workbook.

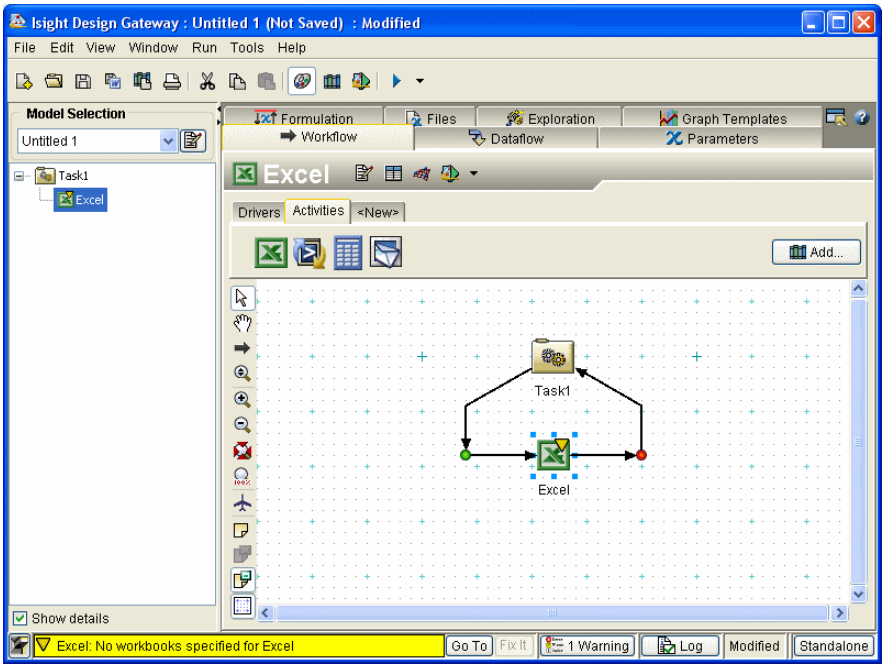
Adding an Excel Component to the Workflow

To add an Excel component to your model workflow:

1. Click the **Activities** tab. This tab is located on the Workflow tab below the Component title bar and toolbar (near the center of the Design Gateway). The component icons change according to the tab that is selected. When this tab is selected, the component palette displays all the currently loaded activity components available in Isight. Activity components take in input parameters, perform some function external to Isight, and provide new values to output parameters.

2. Drag-and-drop the **Excel** icon  onto the red line on the Workflow canvas.

An Excel component is added to the workflow.



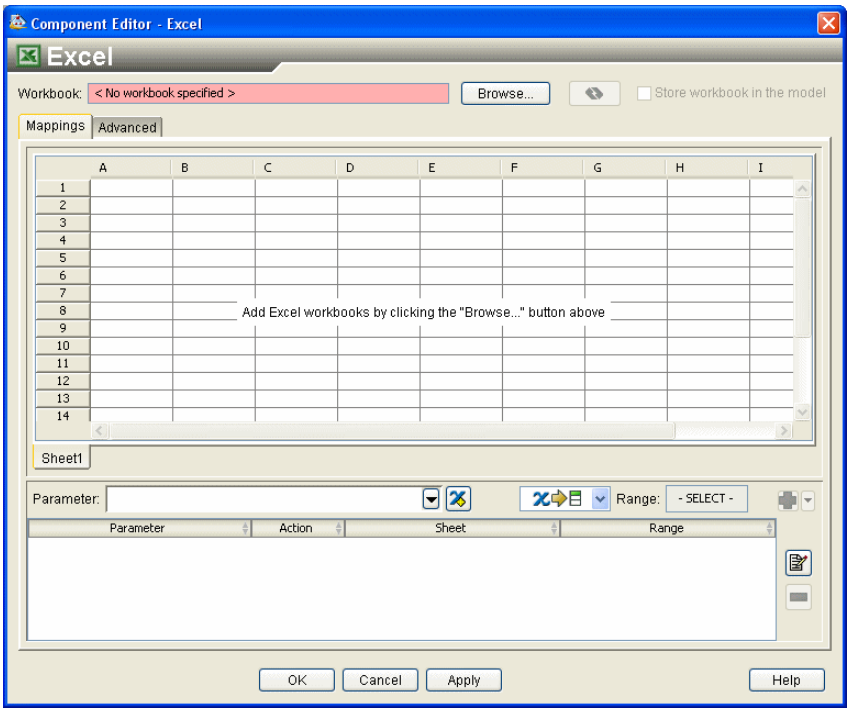
3. Proceed to "Specifying a Workbook" on this page.

Specifying a Workbook

In order to specify which workbook the component will use, you need to access the component editor. Components have editors that allow you to configure them.

To access the component editor and add a workbook:

1. Double-click the **Excel** icon in the workflow. The Excel component editor appears.

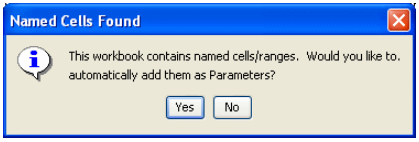


2. Click the **Browse...** button (near the top of the editor) to add an Excel workbook. The Open dialog box appears.

3. Navigate to the following directory:

   <*Isight_install_directory*>\examples\getting_started

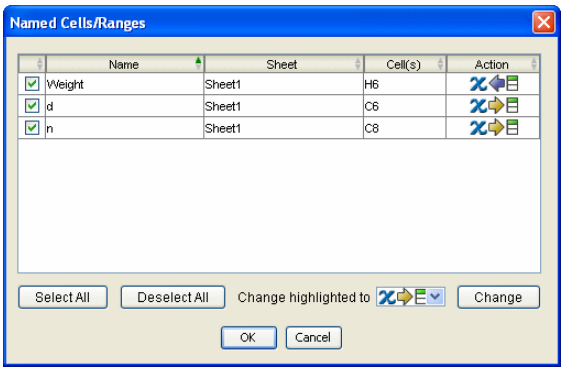
4. Click the **SpringCalcs.xls** file; then, click **Open**. The workbook is loaded into the component editor, and the Named Cells Found dialog box appears.



If you have cells named in the Excel workbook, Isight can automatically create parameters for you. In this example, there are some cells in the Excel workbook

that are already named, and you will use this feature to create some of the parameters automatically.

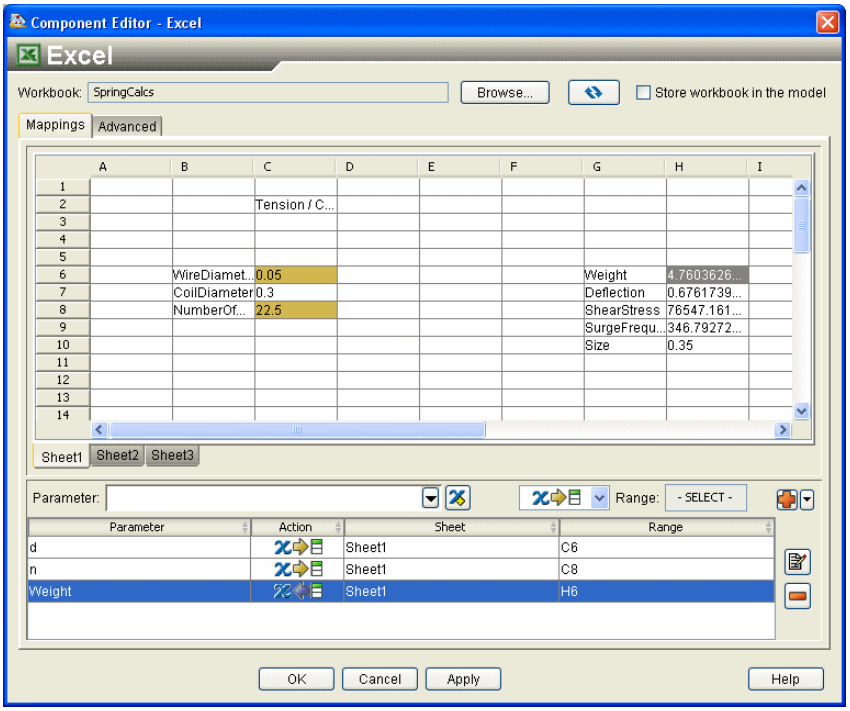
5. Click **Yes.** The Named Cell/Ranges dialog box appears, allowing you to guide Isight through the automatic parameter creation process.



This dialog box shows how Isight will automatically create parameters based on the named cells in Excel. Three of the cells are named *Weight*, *d*, and *n*. The cells that have these names are described in the window. By default, Isight will create parameters *d* (WireDiameter) and *n* (Numberof) as input parameters (as shown in the Action column), and the parameter *Weight* as an output parameter.

6. Click **OK** to accept the defaults.

You are returned to the Excel component editor, and the contents of the workbook are displayed.



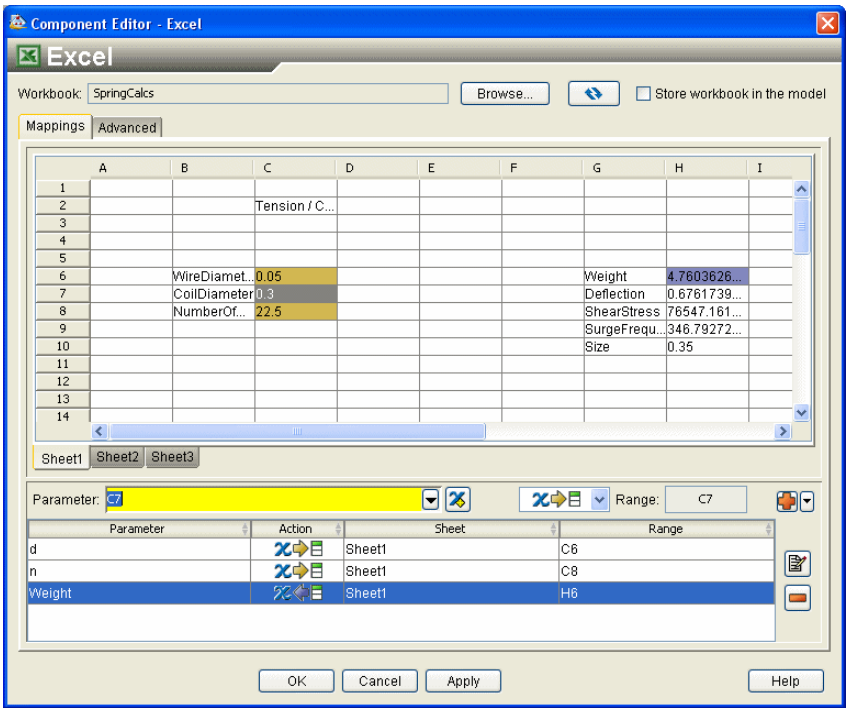
7. Proceed to "Mapping Parameters" on this page.

Mapping Parameters

Now you need to define mappings to the rest of the workbook parameters. In this workbook, there is an additional input parameter that needs to be defined: *Coil Diameter*.

To map the additional parameter:

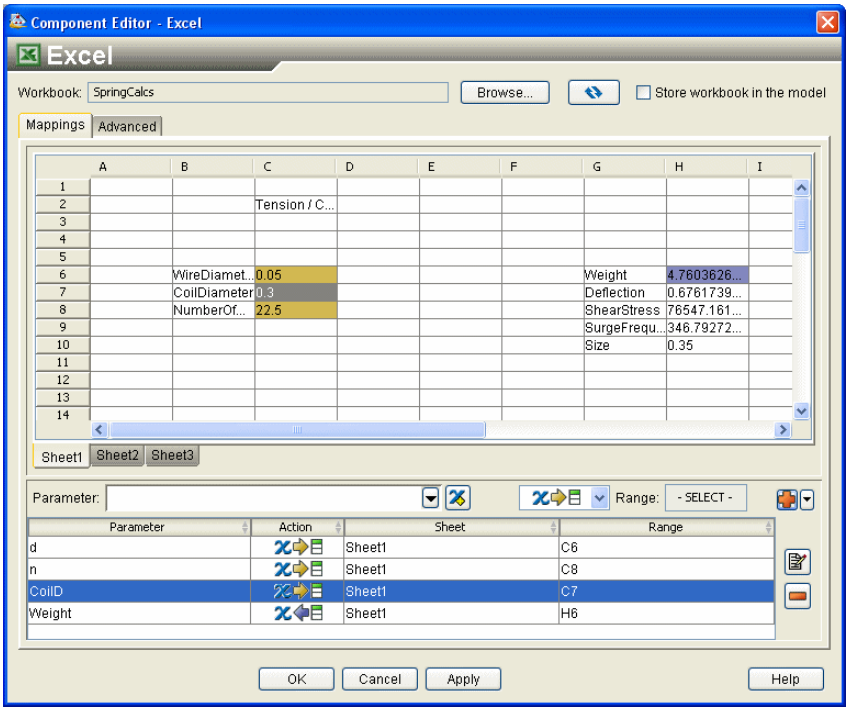
1. Click in the cell that contains the value **0.3** (which is to the right of the cell that is labeled *Coil Diameter*). By clicking in this cell, the value of the cell (C7) is added to the Parameter text box and is highlighted in yellow.



2. Type `CoilD` in the yellow **Parameter** text box. This action assigns a name to the parameter.

   Note: By default, the mapping direction is “input” as shown in the example above. Since the coil diameter is an input parameter, you can leave this setting alone.

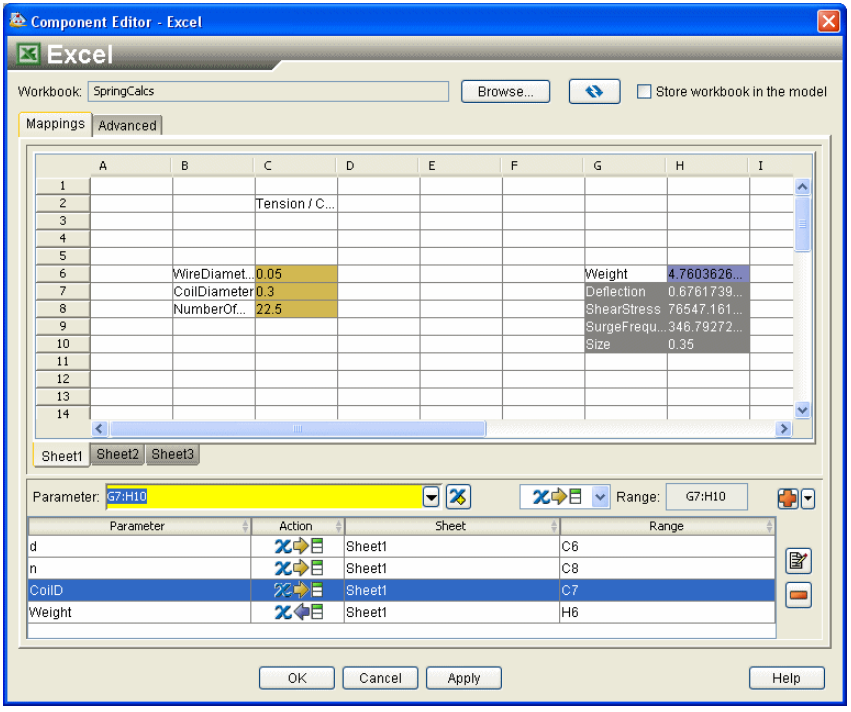
3. Click the  button to add the parameter to the list at the bottom of the editor. Once done, the cell (C7) is highlighted in gold as an input parameter, just like the previous two input parameters.



Instead of typing the names of all the parameters that you want to specify, you can use the name-value mapping option and make use of the fact that some of the cells contain text describing the values in the adjacent cells. You will now use name-value mapping to map the remaining parameters in the workbook.

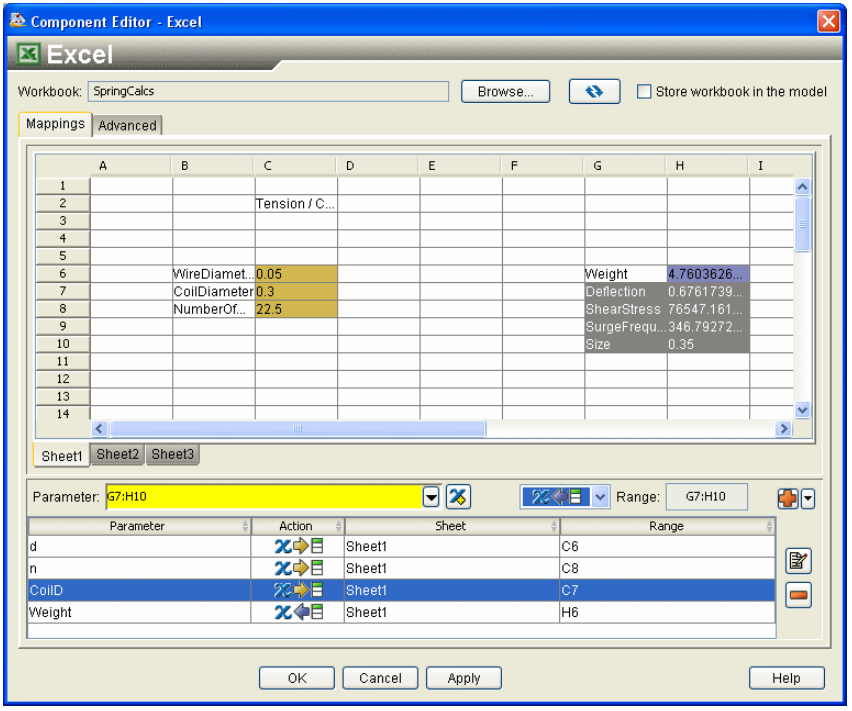
4. Click and drag in the workbook to highlight cells **G7** through **H10**. Do *not* highlight cells G6 or H6, since this parameter is already defined.

Your editor should appear as shown below.



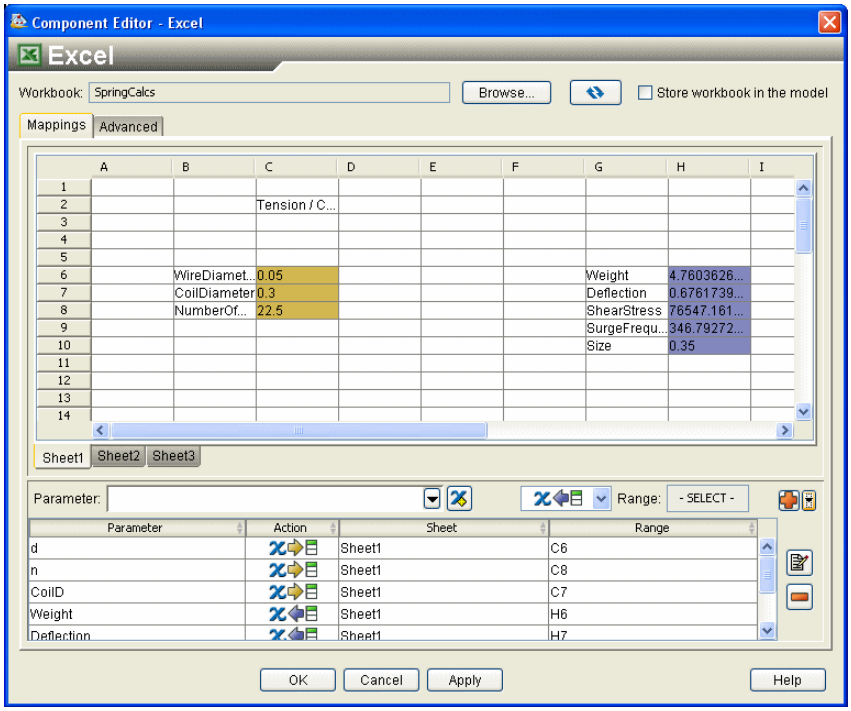
5. Click the  button; then, click the  option. This action changes the mapping direction from input to output.

Your editor should appear as shown below.



6. Click the button next to the button; then, select **Add Name-Value mapping** from the menu that appears.

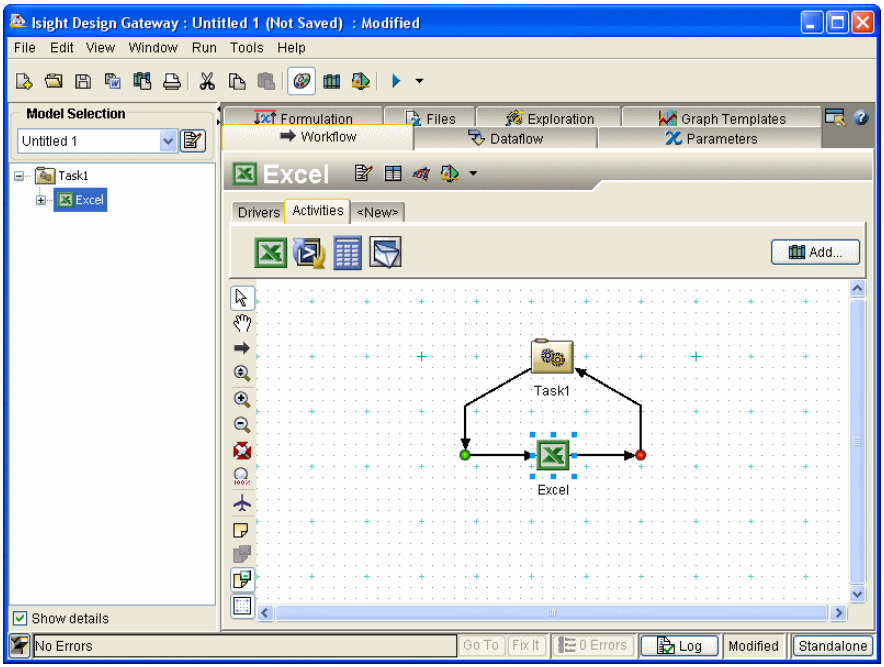
All of the selected output parameters are added to the list at the bottom of the editor. Your editor should appear as shown below, with an updated parameter list.



7. (optional) Click the **Advanced** tab on the Excel component editor; then, clear (deselect) the **Show Excel during execution** check box if you do not want Isight to show the Excel workbook during execution.

8. Click **OK**. Your changes are saved and the component editor is closed.

You are returned to the Design Gateway.



9. Proceed to "Executing the Model and Viewing Results" on this page.

Executing the Model and Viewing Results

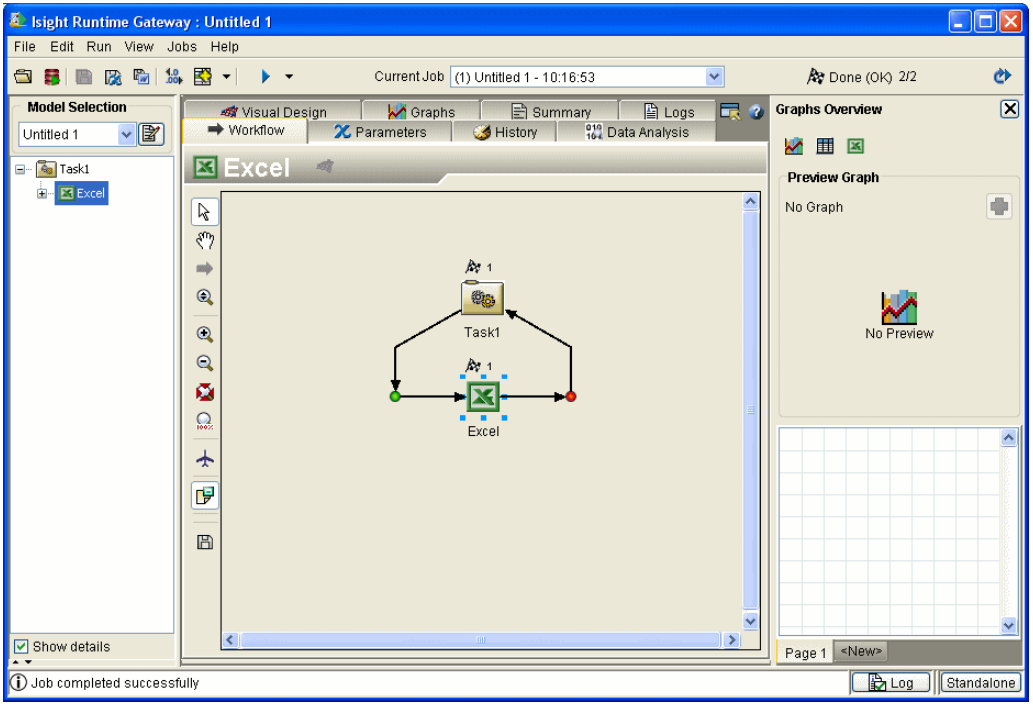
Isight now has enough information to execute the Spring application. Since the parent component of the workflow is the default Task component, the model will only execute one time. This single execution is the essence of the Task component.

To execute the model:

1. Click the button on the Design Gateway toolbar. The Runtime Gateway appears, and the model is executed. The Runtime Gateway automatically loads the Excel workbook. Excel is started on your system, and it displays the selected workbook.

   Note: You can set Isight to not show the Excel workbook during execution by accessing the Advanced tab on the Excel component editor and clearing (deselecting) the **Show Excel during execution** check box. For more information on the other advanced settings, refer to the *Isight Component Guide*.

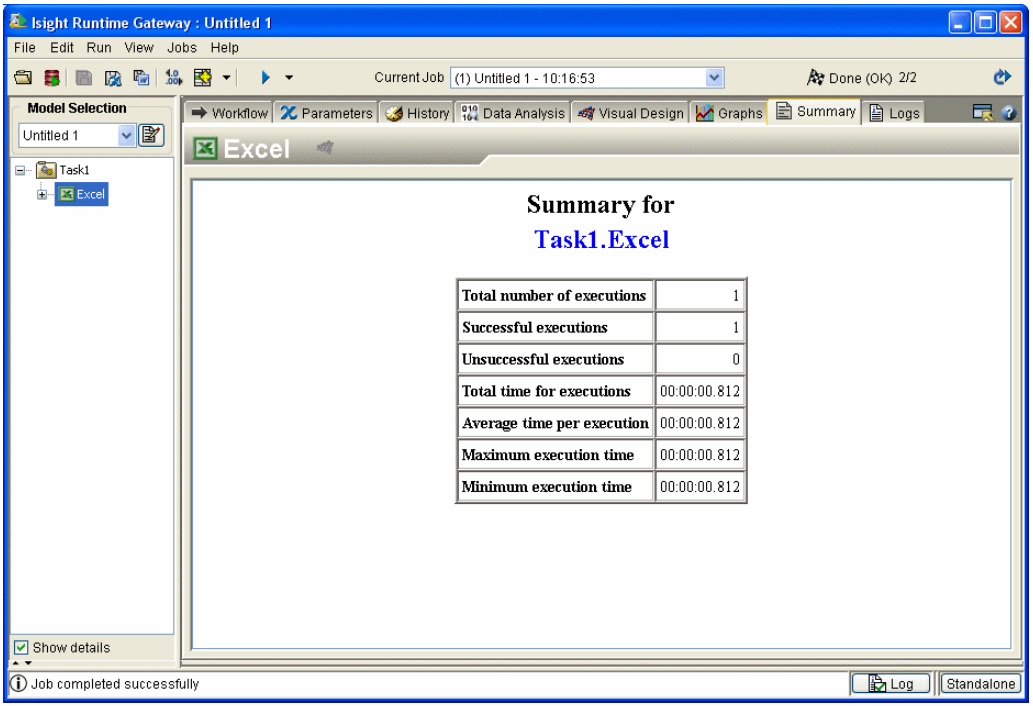
   Once execution is completed, the message "Job completed successfully" appears in the bottom left corner of the interface, and icons appear above each component in the workflow, as shown below.



   You can now view the results of the execution. In this example, you will use the Summary tab and the History tab to view the results. These two tabs display information in different formats.

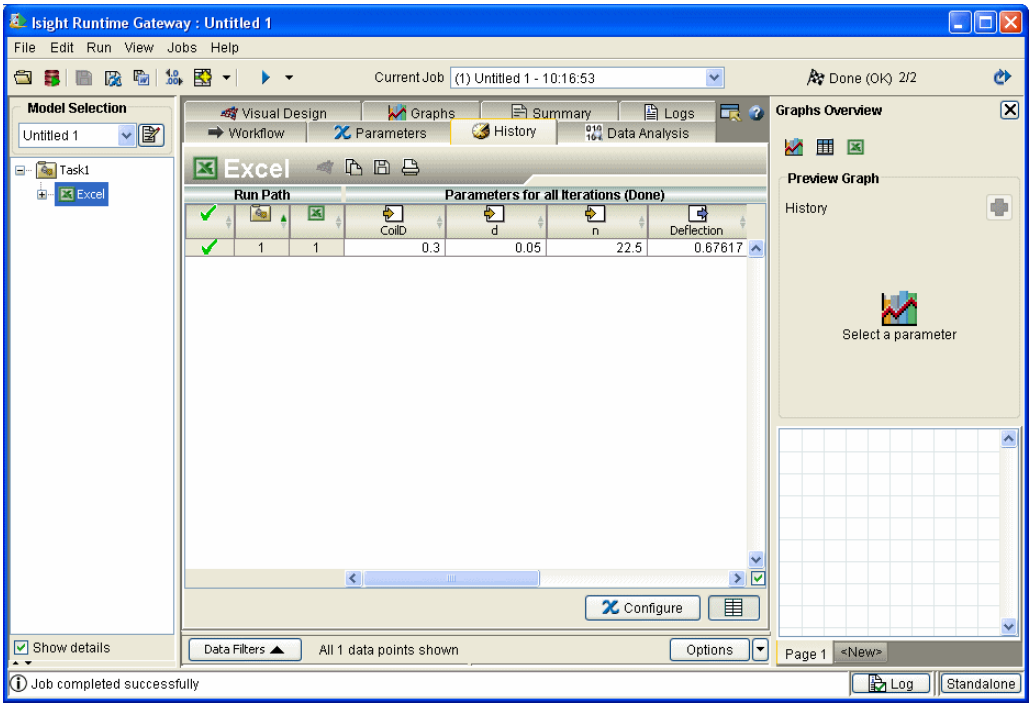
   Note: The content of each tab changes depending on the component selected on the left side of the Runtime Gateway.

2. Click the **Excel** component in the Model Explorer on the left side of the Runtime Gateway; then, click the **Summary** tab on the right side of the Runtime Gateway. A summary of the execution appears. This summary does *not* include individual parameter values.



3. Click the **History** tab.

This tab shows parameter values for each run of the Excel component.



4. Select **Close Window** from the Runtime Gateway **File** menu to close the interface. You are returned to the Design Gateway. Now you'll add a process component to the workflow to facilitate multiple runs of the Excel component during a single execution.

5. Proceed to "Adding a Loop Component to the Model," on page 63.

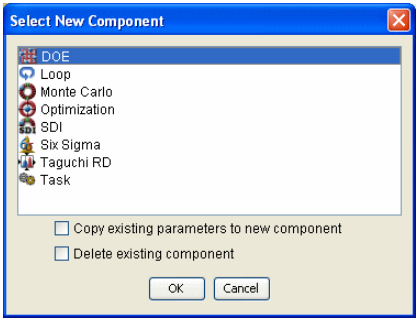
Adding a Loop Component to the Model

As discussed earlier, the Isight Design Gateway starts with a default task. For this example, you will change this to a Loop component. The Loop component is a process component. A process component is a component that contains an Isight workflow within it (also called a subflow). Process components are typically referred to as *design drivers,* which use various algorithms to determine the values of input parameters for each iteration.

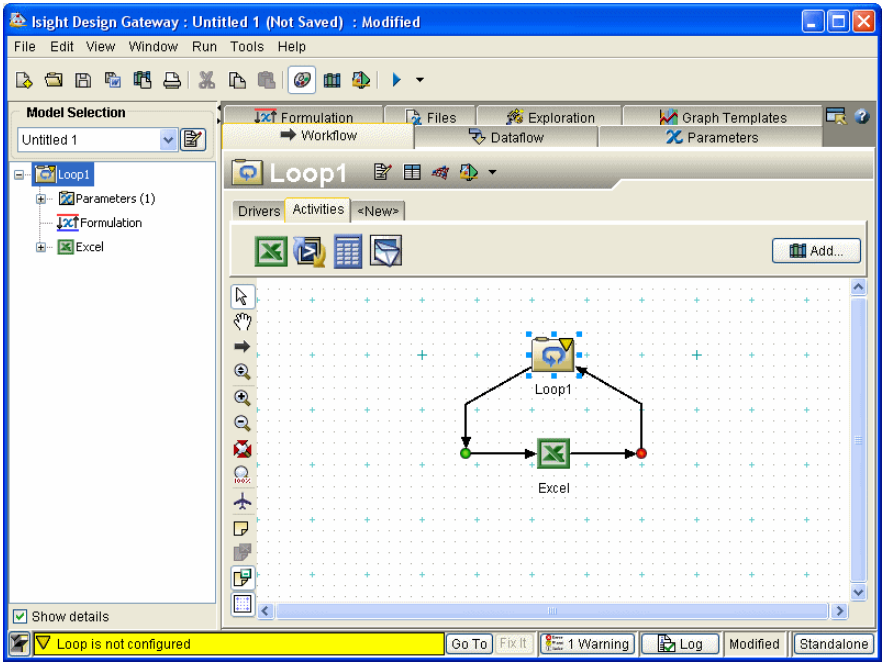
Adding the Loop Component

To add components to the model:

1. Right-click the **Task** component on the **Workflow** tab of the Design Gateway. A menu appears.
2. Point to **Change To** on the menu; then, select the **New...** option. The Select New Component dialog box appears.



3. Click the **Loop** component; then, click **OK**. The Task component has been changed to a Loop component in your workflow.



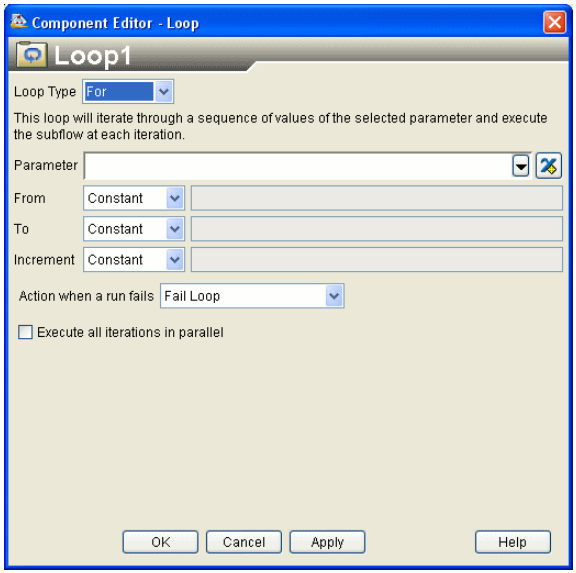
4. Proceed to "Configuring the Loop Component" on this page.

Configuring the Loop Component

At this point, you have fully defined the input parameters and output parameters required in the Excel workbook. Now, you need to configure the Loop process component.

To configure the Loop component:

1. Double-click the **Loop1** icon in the workflow. The Loop component editor appears.

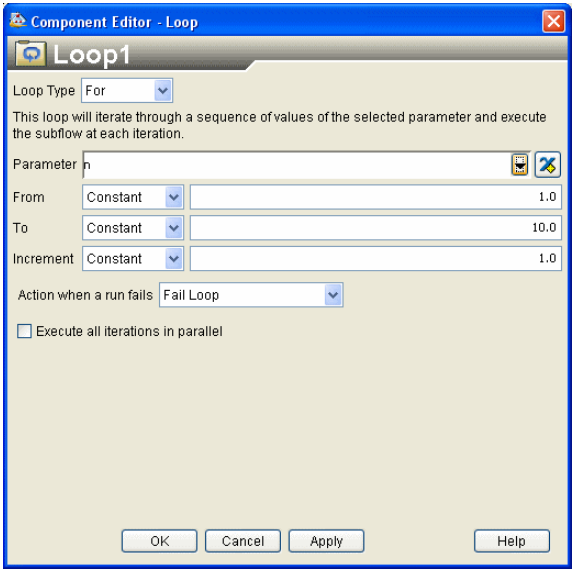


   There are several types of Loops available in Isight, including “for”, “for array”, “for each”, “while”, and “do until”.

2. Verify that **For** is selected from the type from the **Loop Type** drop-down list.

   Using the Parameter drop-down list, you have the option of selecting any of the Isight input or output parameters created earlier in the Excel component.

3. Click the button adjacent to the **Parameter** text box to access the Parameter drop-down list; then, select **n** from the list of parameters. This selection means that you will loop on the value corresponding to the number of coils. Your editor should appear as shown below.

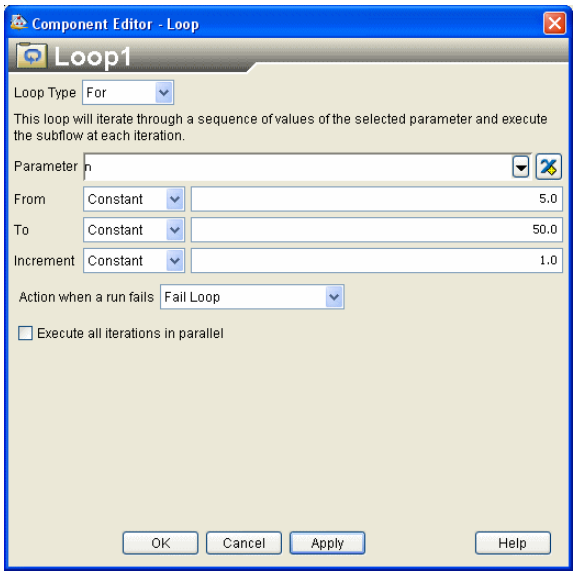


When using the For Loop option, you can also specify a start point, end point, and increment for the parameter you've chosen to loop on. In this case, you are going to vary the number of coils in the spring.

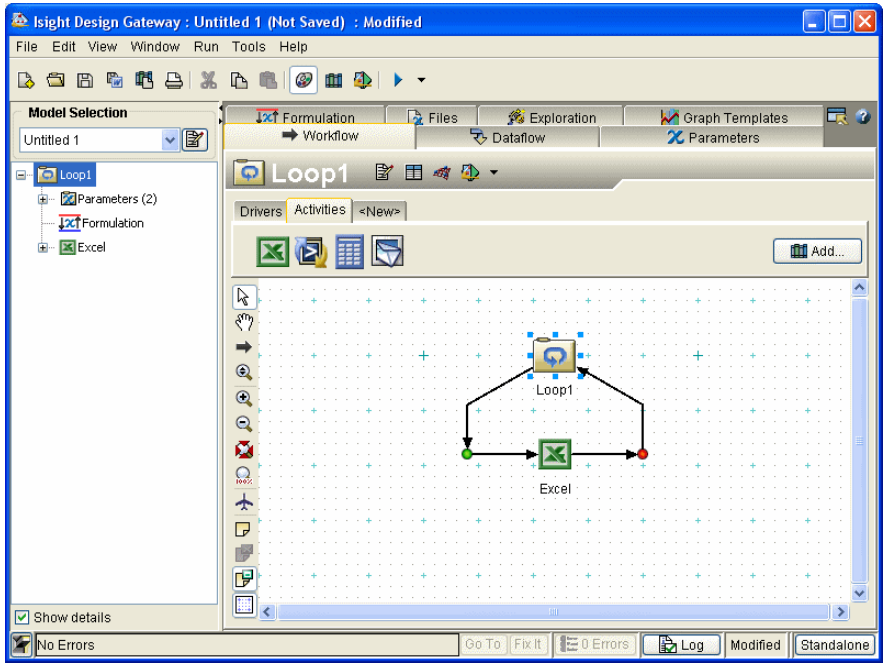
4. Enter the following values in the corresponding text boxes:

   - **From**: `5.0`
   - **To**: `50.0`

5. Verify that the **Increment** option is set to `1.0`. Your editor should appear as shown below.



6. Click **OK**. Your changes are saved, and you are returned to the Design Gateway.



Re-executing the Model and Viewing Updated Results

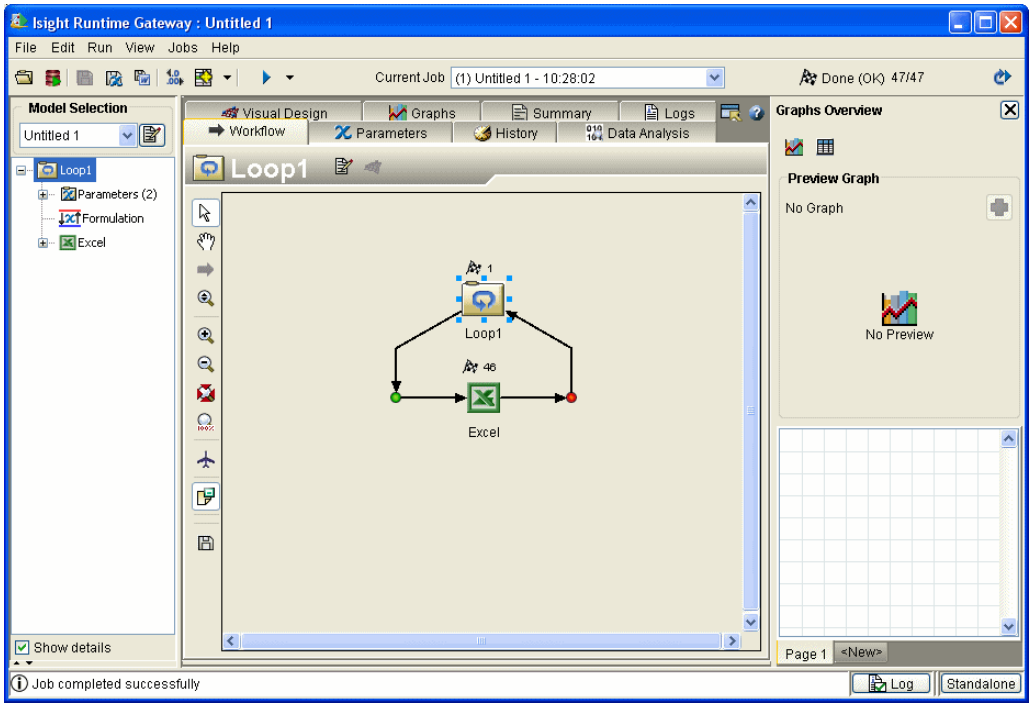
Isight now has enough information to execute the Spring application in a loop.

To execute the model:

1. Click the  button on the Design Gateway toolbar. The Runtime Gateway appears, and the model is executed. The Runtime Gateway automatically loads the Excel workbook and changes the value of the number of coils cell based on the Loop component you defined. You can minimize Excel to view the execution progress in the Runtime Gateway.

   Note: You can set Isight to not show the Excel workbook during execution by accessing the Advanced tab on the Excel component editor and clearing (deselecting) the **Show Excel during execution** check box. For more information on the other advanced settings, refer to the *Isight Component Guide*.

Once execution is completed, the message “Job completed successfully” appears in the bottom left corner of the interface, and icons appear above each component in the workflow, as shown below.



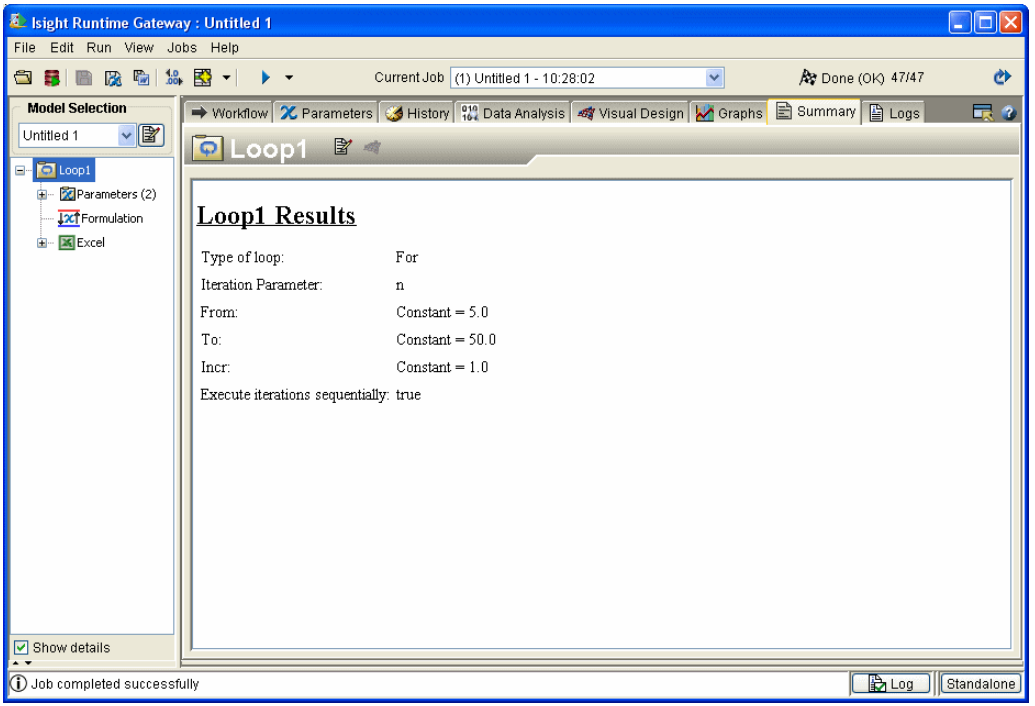
You can now view the results of the execution. In this example, you will use the Summary tab and the History tab to view the results. These two tabs display information in different formats.

Note: The content of each tab changes depending on the component selected on the left side of the Runtime Gateway.

2. Verify that the **Loop1** component is selected in the Model Explorer on the left side of the Runtime Gateway.

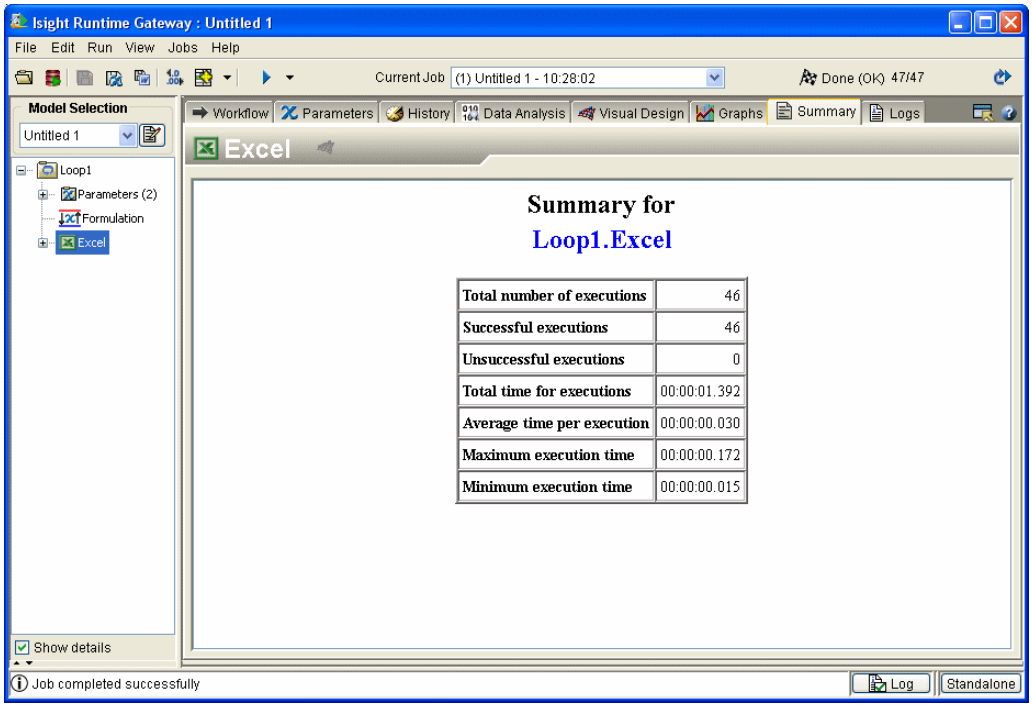
3. Click the **Summary** tab on the right side of the Runtime Gateway.

A summary of the execution for the Loop component appears. This summary does *not* include individual parameter values.



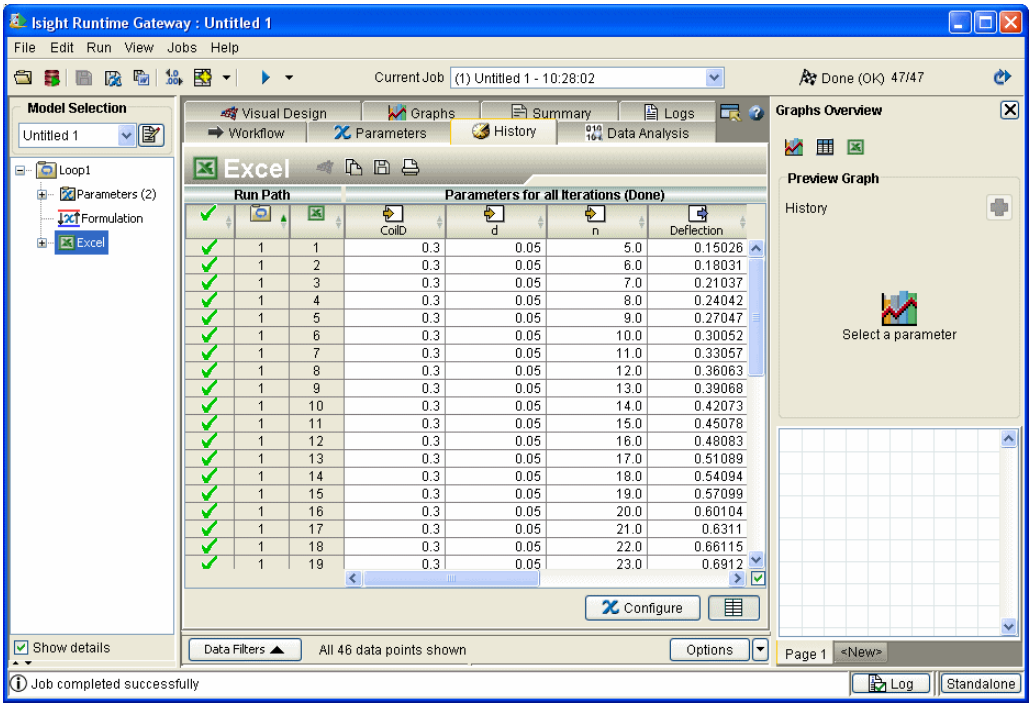
4. Click the **Excel** component in the Model Explorer on the left side of the interface.

Notice that the Summary information changes since the selected component has changed.



5. Click the **History** tab.

This tab shows parameter values for each run of the Excel component.



6. Select **Close Window** from the Runtime Gateway **File** menu to close the interface. You are returned to the Design Gateway.

7. Select **Close Current Model** from the Design Gateway **File** menu to close the model. The Save Model dialog box appears.

8. Save the model, if desired. It is not necessary to save the model to complete any of the other examples in this book.

   Important: Due to naming limitations, do not use the following characters in your model name:

   #, ?, &, %, !, \, $, {

Isight

4 Simcode Example

This chapter demonstrates how to use the Simcode component to build a workflow in the Isight Design Gateway with an existing executable (`aeroarg`) and input and output template (`AeroIn` and `AeroOut`) files. These files are included in your Isight installation. There are separate files for Windows operating systems, several UNIX operating systems (AIX, HP-UX, and Solaris), and Linux. The UNIX and Linux files are located within subdirectories of the directories specified in this procedure.

The following tasks are demonstrated:

- Creating a new model
- Adding a Simcode component to the model's workflow
- Configuring a Simcode component (executable file, input file, and output file)
- Renaming a component
- Executing a component
- Publishing a component
- Viewing a published component in the Library
- Saving a model

Creating the Model and Adding a Component

The first step is to create a model and add the Simcode component. This step is accomplished using the Design Gateway.

To start Isight and create a model:

1. Start the Isight Design Gateway, if necessary. If you are continuing from the previous example (the Simple Example), proceed to the next step.

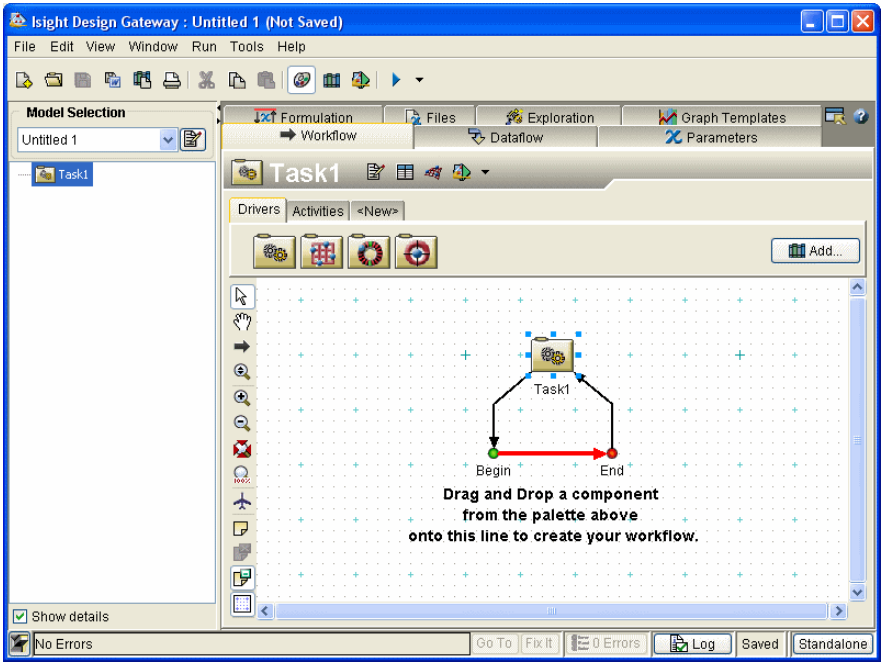
   For more information on how to start the Design Gateway on Windows, see "Accessing the Design Gateway," on page 48.

   To start the Design Gateway on UNIX/Linux, execute the `./gateway` command from the <*isight_install_directory*>/bin directory.

   Note: If you are starting the Design Gateway on a 64-bit Linux system, the 64-bit version of the interface is started by default. However, if you have installed both the 32-bit and 64-bit versions, you can specify the version that will be used by adding `-32` or `-64` to the end of the `gateway` command (be sure to leave a space between the command and the option). Additionally, you can set the default value for the `gateway` command using the FIPER_BITS environment variable described in the *Isight User's Guide*.

2. Select **New (Default)** from the Design Gateway **File** menu.

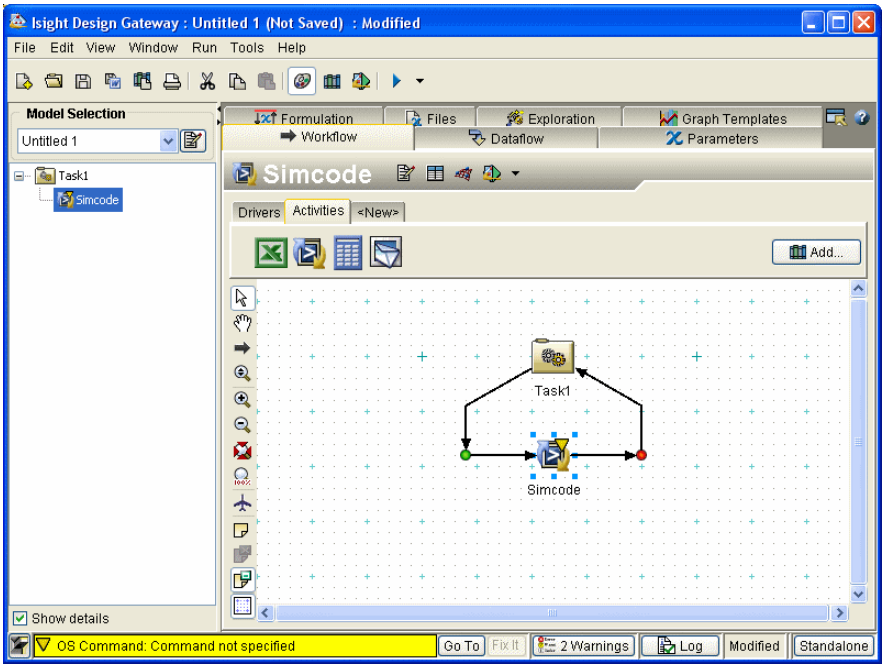
A new model is created with the default (Task) component.



3. Click the **Activities** tab. The currently loaded activity components are shown on the component palette.

4. Drag-and-drop the **Simcode** icon  onto the red line on the Workflow canvas.

A Simcode component is added to the workflow.



Once the component is added to the workflow, it must be configured for your problem.

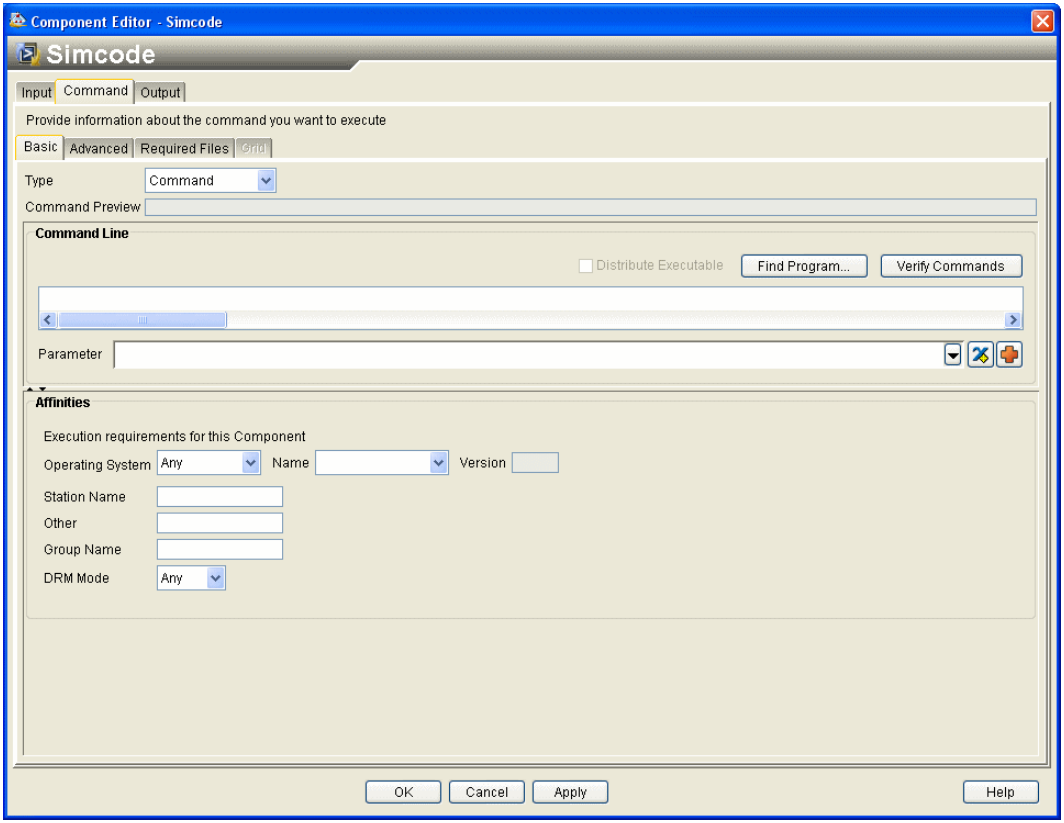
5. Proceed to "Configuring the Executable" on this page.

Configuring the Executable

The next step is to tell Isight how to run the necessary executable file. This task involves entering the name of the program, including the directory in which it resides, and the command line arguments that it takes.

To configure the component:

1. Double-click on the **Simcode** icon  in the workflow. The Simcode Component Editor appears.



   By default, the Command tab is selected. There are four different subtabs available from the Command tab: Basic, Advanced, Required Files, and Grid (if available). The Basic tab is selected by default. The Command Line text box is used to enter the program name, path of the program, name of the input file, and the name of the output file.

2. Click the **Find Program...** button. The Select Program dialog box appears.

3. Navigate to the following directory:

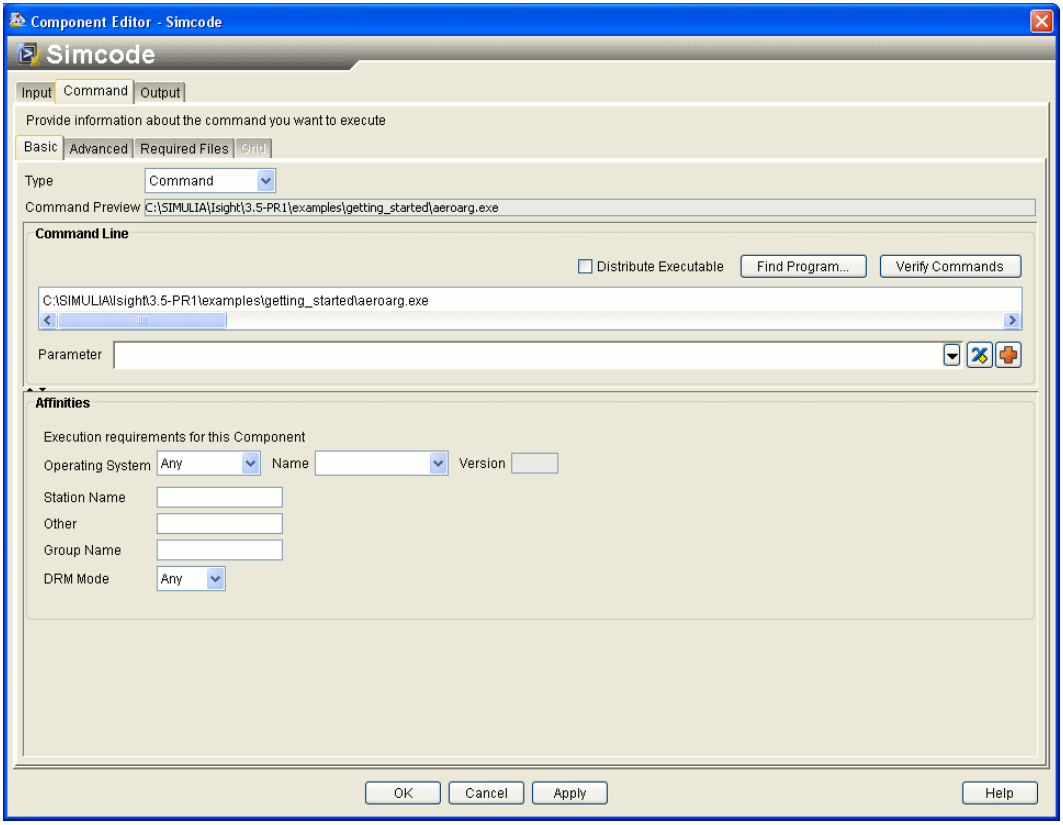
   <*Isight_install_directory*>\examples\getting_started

Note: If you are using UNIX or Linux, navigate into the subdirectory that matches your operating system.

4. Click the **aeroarg.exe** file; then, click **Open**.

   Note: If you are using UNIX or Linux, the name of the file is **aeroarg**.

   You are returned to the Simcode component editor, and the full path of the program is entered into the field.

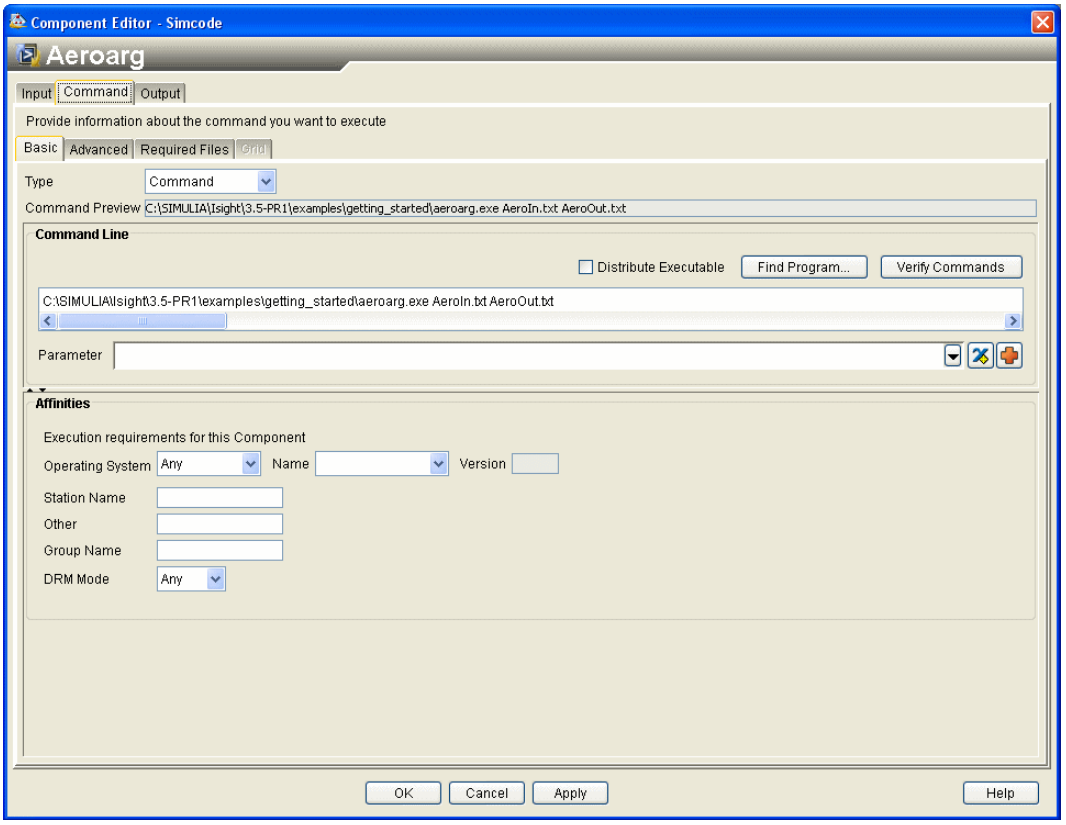


For this example, the input file name is `AeroIn.txt` and the output file name is `AeroOut.txt`. These files need to be added to the Command Line text box.

5. Type `AeroIn.txt AeroOut.txt` following the `...aeroarg.exe` entry in the **Command Line** text box. Be sure to leave a space after the ending text and the information you are typing.

   Note: If you are using UNIX or Linux, you should type `AeroIn` and `AeroOut` following the `aeroarg` entry.

Your component editor should appear as shown below.



The complete syntax for running the `aeroarg.exe` at a command line is as follows (all on one line), based on your operating system:

- **Windows**: <*Isight_install_directory*> `aeroarg.exe AeroIn.txt AeroOut.txt`
- **UNIX/Linux**: <*Isight_install_directory*> `aeroarg AeroIn AeroOut`

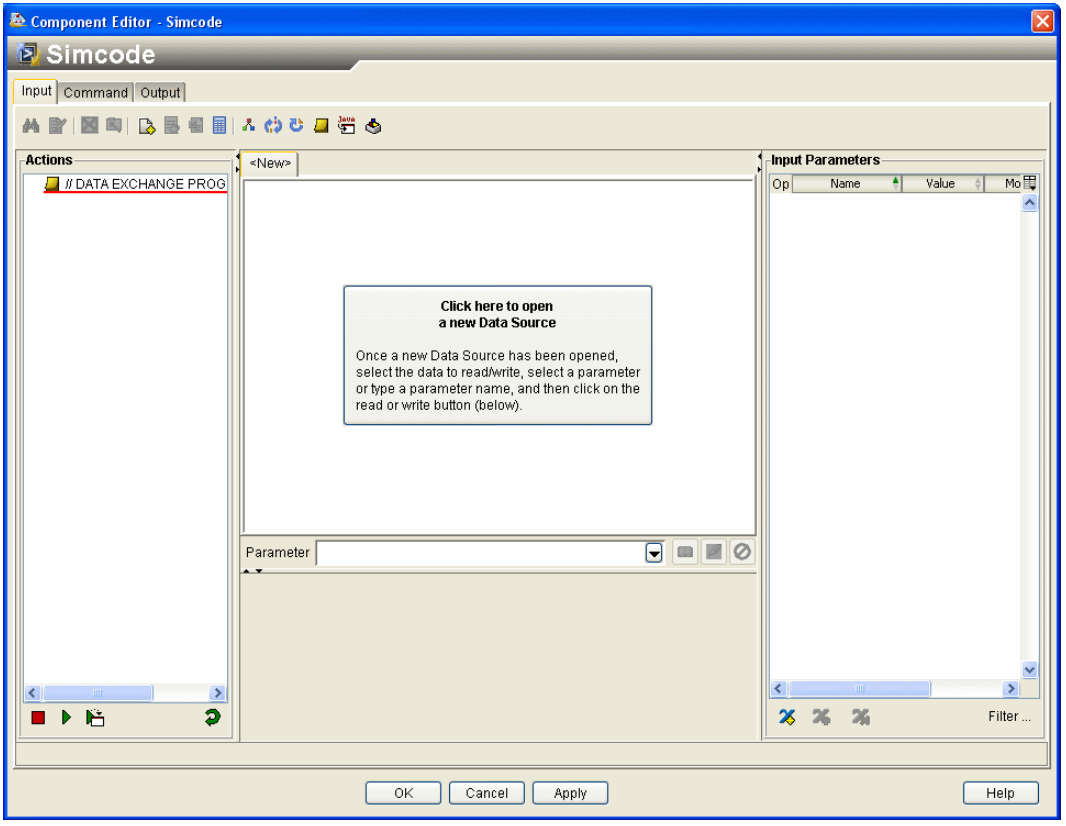
6. Click **Apply** to save your changes to the component. At this point, you have done all that is necessary to define the command line executable to run the `aeroarg.exe` code.
7. Proceed to "Defining the Input Parameters," on page 80.

Defining the Input Parameters

Since Isight will be running this program many times with different values of several input parameters, you must tell Isight how to put the new values you want to investigate into the input files so that Isight can evaluate them.

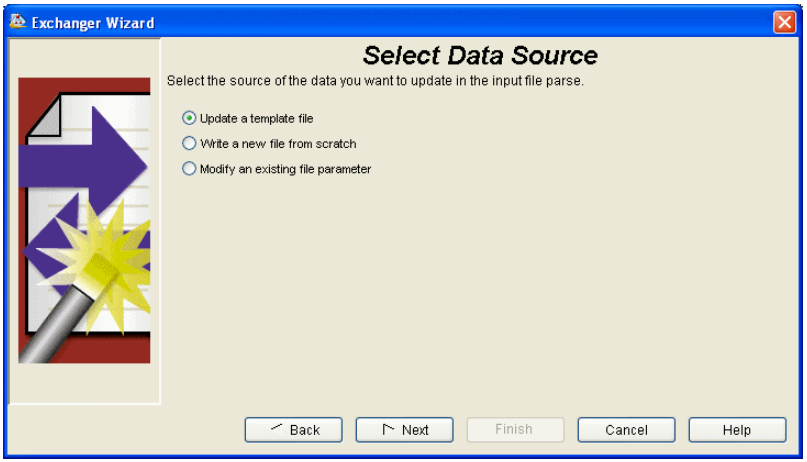
To define input parameters:

1. Click the **Input** tab on the Simcode component editor. The contents of the tab appear.



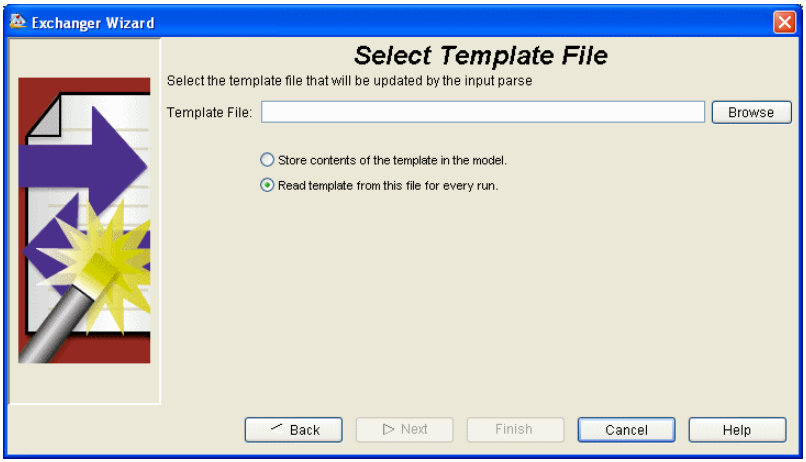
2. Click the large **Click here to open a new Data Source** button in the center of the tab.

The Exchanger Wizard appears.



At this point, you need to select the data source that will be used as the input file and will be updated during execution. In general, it is best to use a template file to define the parsing commands for the input file. This option is selected by default.

3. Click **Next**. The Select Template File screen appears.

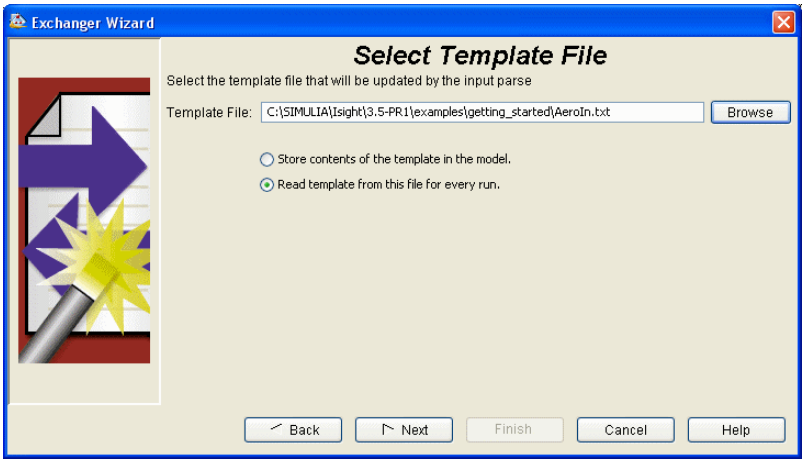


4. Click the **Browse** button. The Open dialog box appears.

5. Click the **AeroIn.txt** file; then, click **Open**.

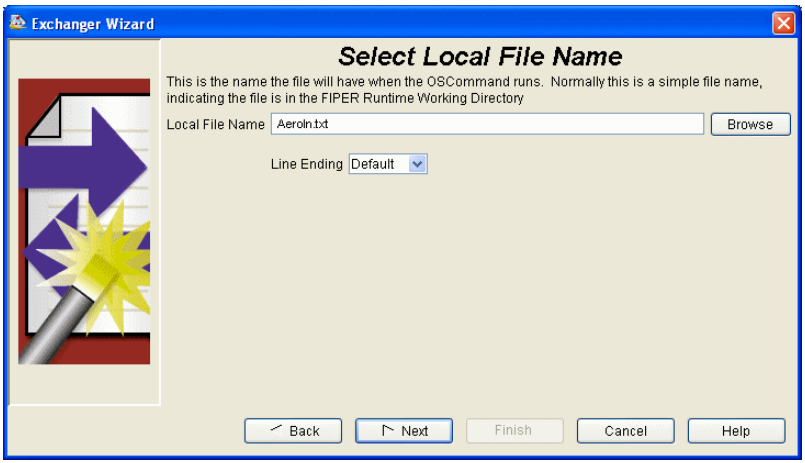
   Note: If you are using UNIX or Linux, you should select **AeroIn** from the subdirectory that matches your operating system.

The full path of the file is added to the Template File text box.



You have the option to store the contents of the template file in the model file or to read from the template file for every run. The latter option is the default.

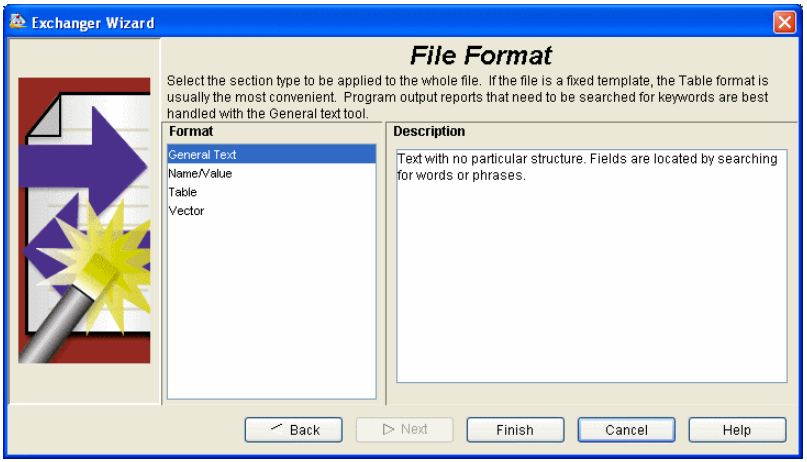
6. Verify that the **Read template from this file for every run** radio button is selected; then, click **Next**. The Select Local File Name screen appears.



By default, the value in the Local File Name text box is `AeroIn.txt` (or `AeroIn` on UNIX or Linux). This value is based on what you previously specified on the Select Template File screen (in step 5).

7. Click **Next**.

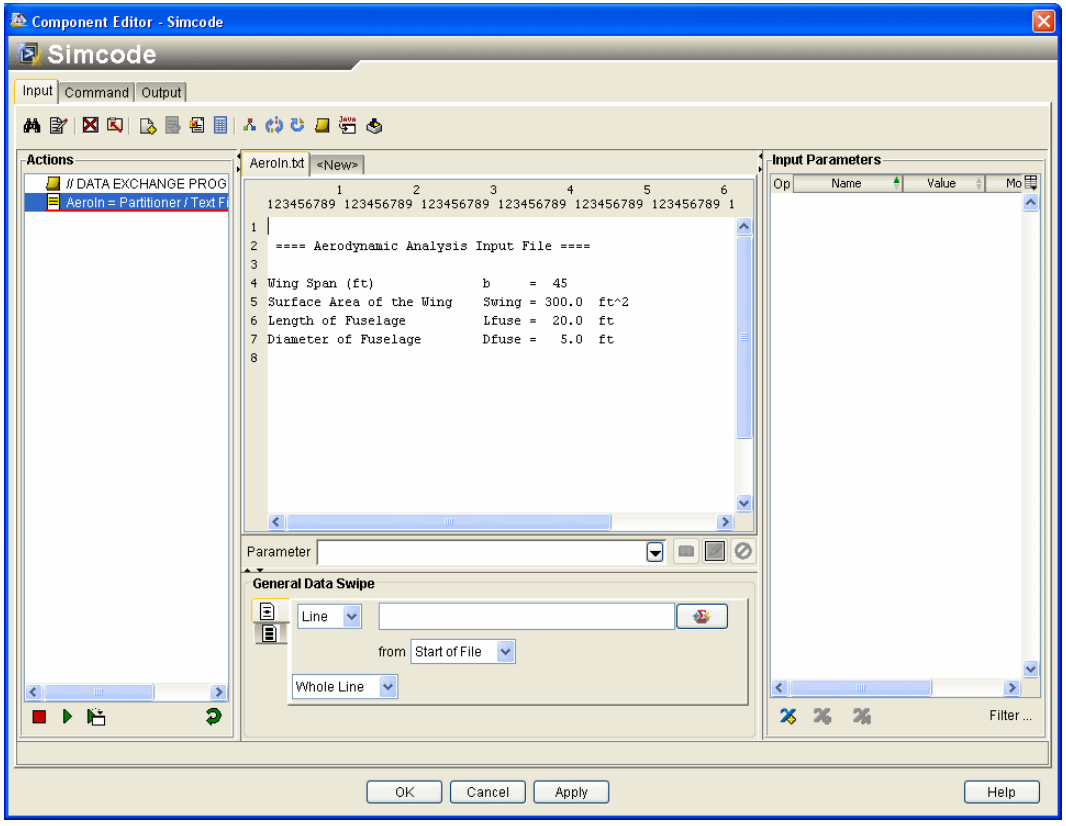
The File Format screen appears.



Isight can do data exchange (or parsing) on four different types of files: General Text, Name/Value, Table, and Vector. For this example, the General Text option is used.

8. Verify that the **General Text** option is highlighted in the Format area; then, click **Finish**.

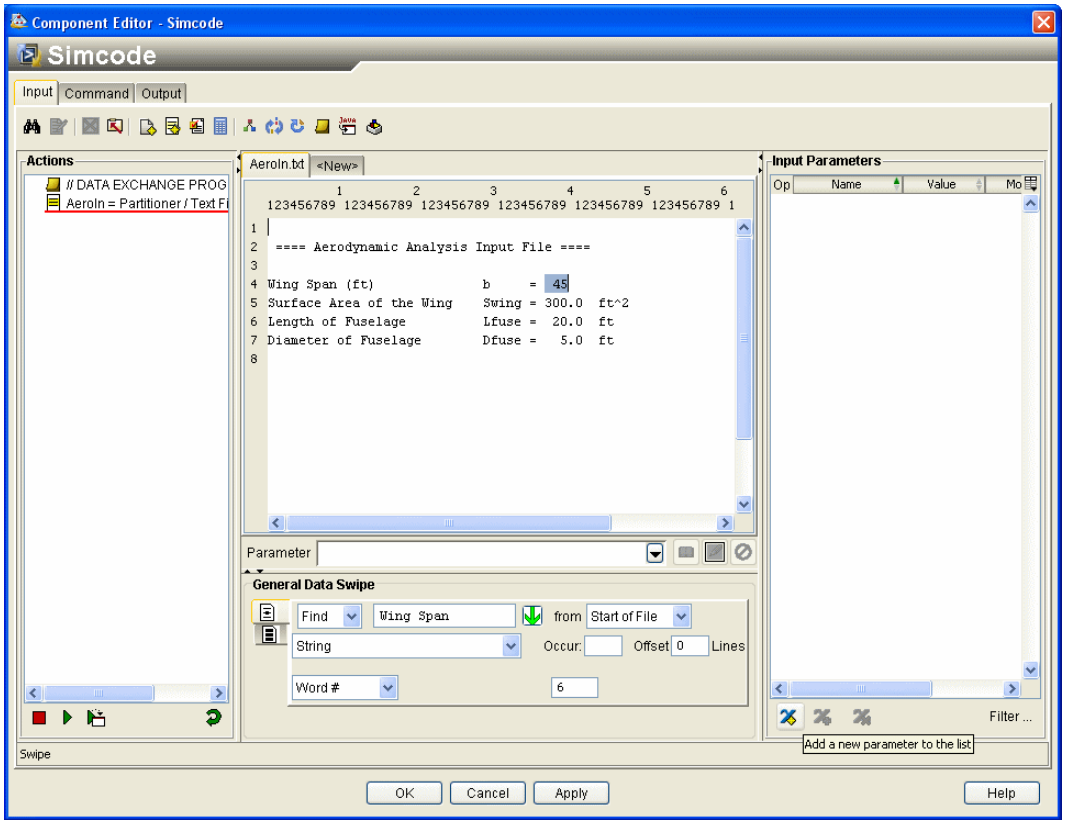
You are returned to the component editor, and the input file information is displayed.



The left side of the editor shows the current list of actions for the file. The center of the editor shows the contents of the template input file used by the simulation code. The right side displays the input parameters with properties such as Mode, Type, etc. Since no parameters have been defined yet, this area is blank.

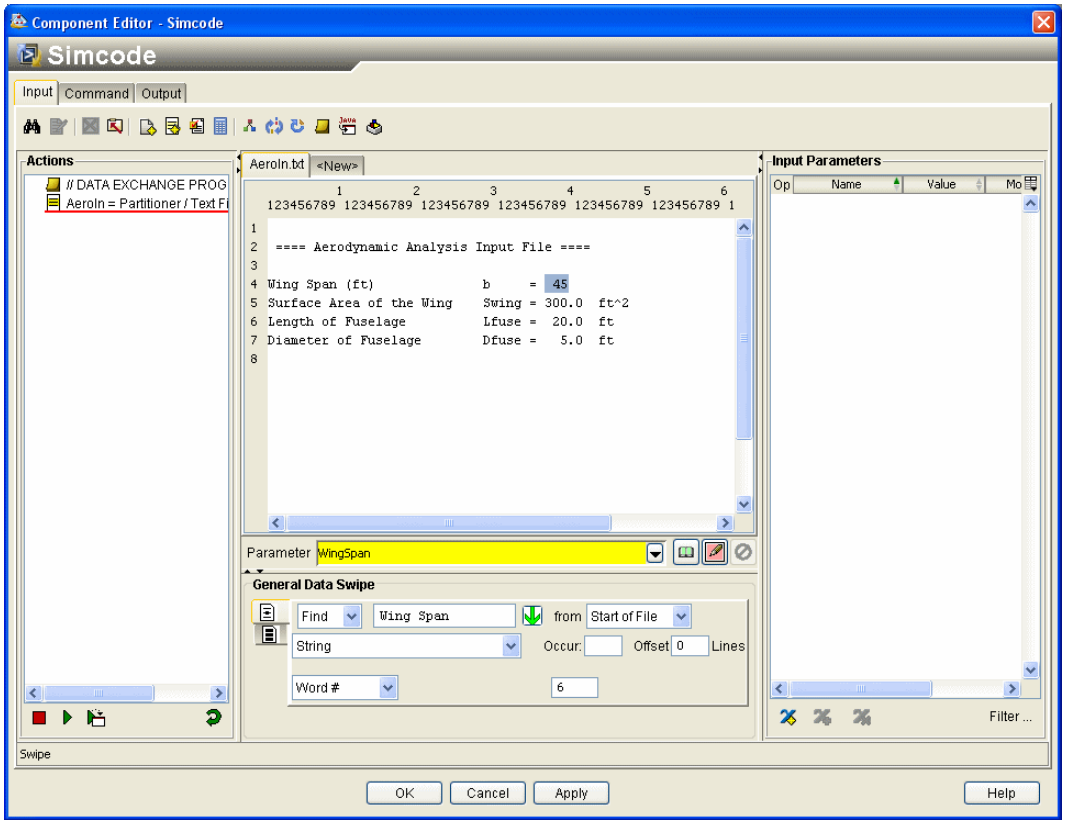
9. Click the value **45** in the center of the editor.

The number is highlighted.



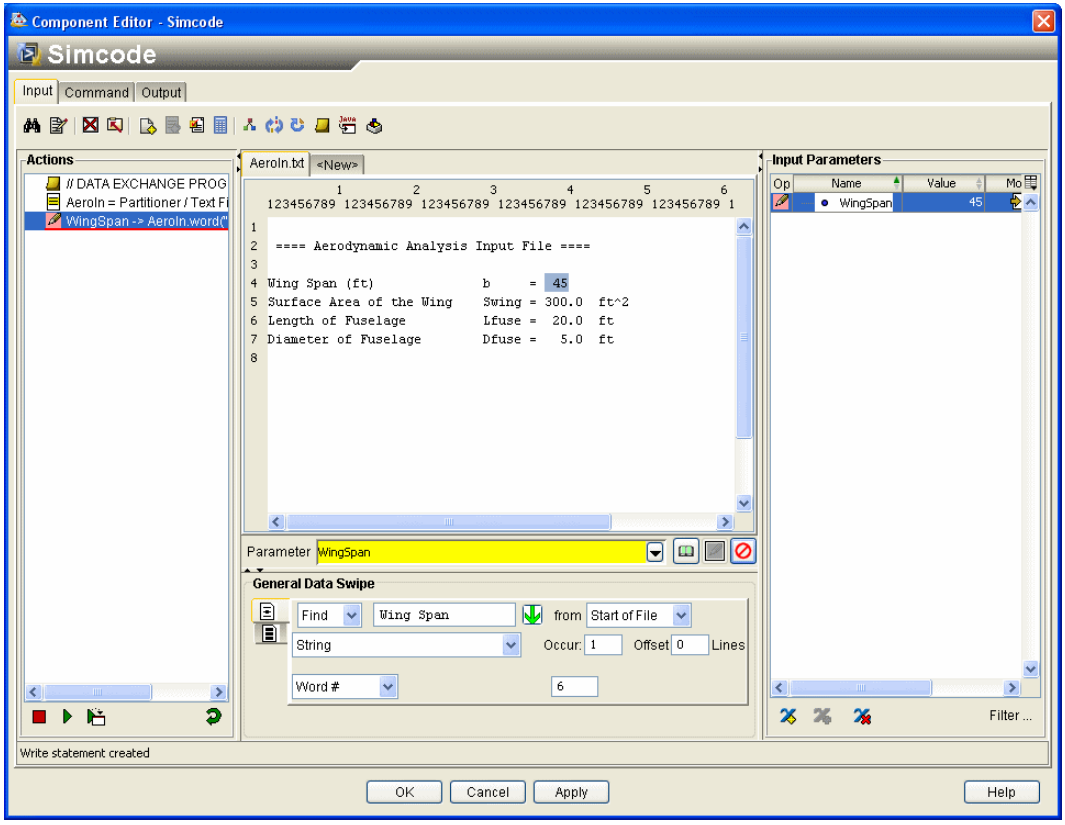
10. Type `WingSpan` in the **Parameter** text box in the center of the editor.

The text box is highlighted in yellow once you begin to enter text.



11. Click the **Write** button  adjacent to the **Parameter** text box. This action tells Isight to write a parameter called “WingSpan” into the input file at the location where the value 45 now resides. For the purpose of file parsing in the Simcode component, Isight typically writes parameters into the input file so that their values may change from run to run. Conversely, Isight reads values from output files after they are run through the Simcode.

The editor is updated with the new action added to the left side and the new parameter added to the right side.

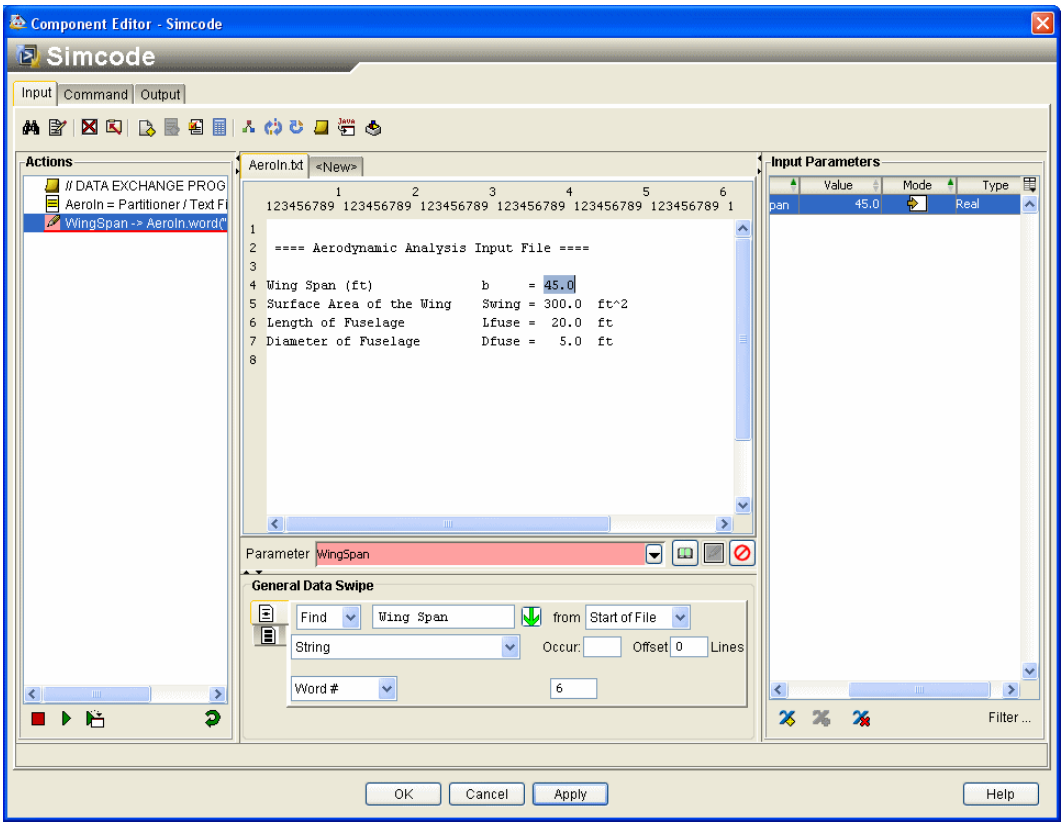


The parameter WingSpan is being declared as an integer, because the number in the input file is written without decimals. However, for this example, the value should be a Real value.

12. Click the cell that contains the word **Integer** in the Input Parameters area of the right side of the editor. You may need to scroll over to see this cell - it is under the **Type** column. A drop-down button appears when the cell is clicked.

13. Select **Real** from the options that appear.

The Type setting for the parameter is updated.



Now you need to define the remaining input parameters using the same procedure.

14. Click the value **300.0** in the center of the editor until the number is highlighted.

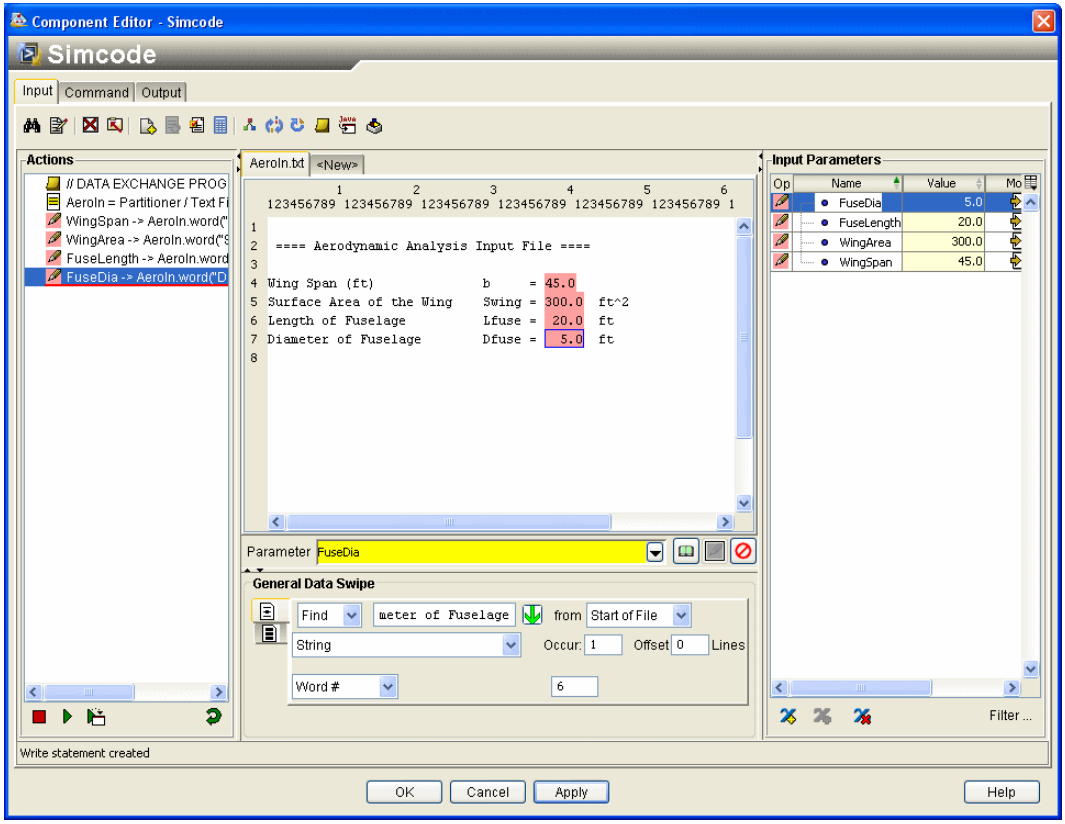
15. Type `WingArea` in the **Parameter** text box; then, click the **Write** button  . The Actions area is updated, and this parameter is added to the list on the right side of the editor. Since the value in the file is in real format, the parameter WingArea is created as a Real value.

16. Click the value **20.0** in the center of the editor until the number is highlighted.

17. Type `FuseLength` in the **Parameter** text box; then, click the **Write** button  . The Actions area is updated, and the parameter is added to the list on the right side of the editor.

18. Click the value **5.0** on the left side of the editor until the number is highlighted.

19. Type `FuseDia` in the **Parameter** text box; then, click the **Write** button  . The Actions area is updated, and the parameter is added to the list on the right side of the editor. When finished, your editor should appear as shown below.



20. Click **Apply** to save your changes to the component.

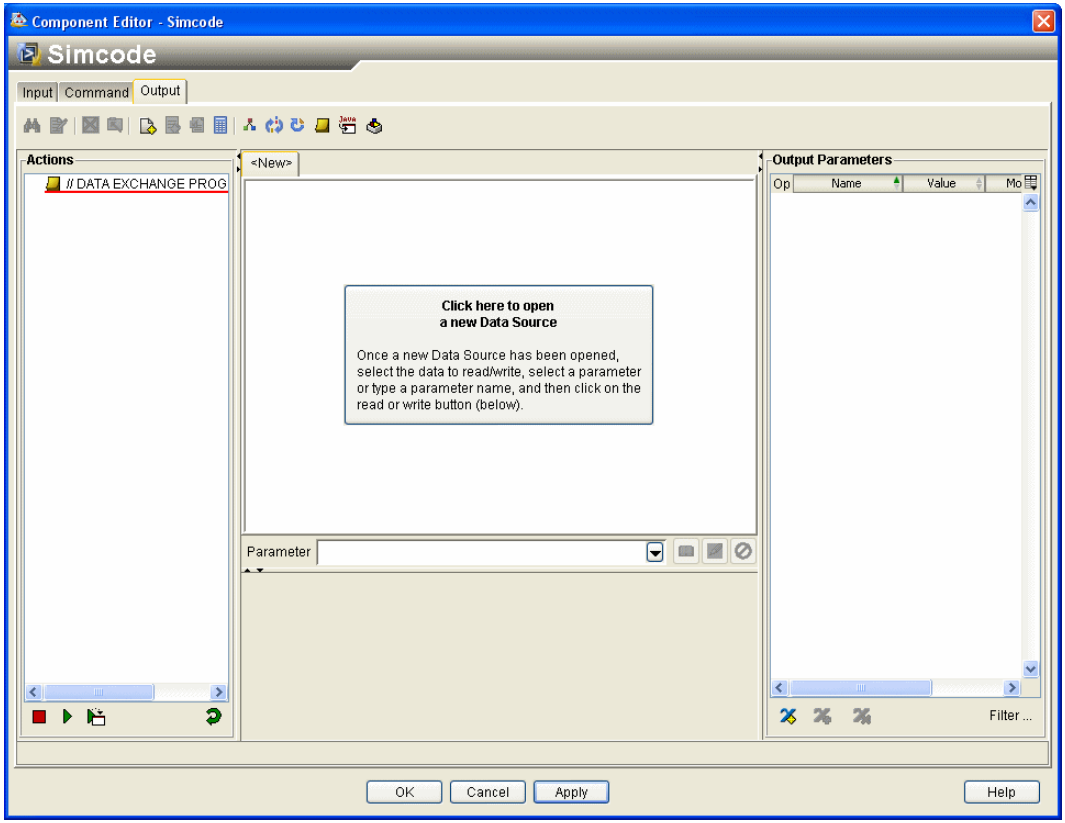
21. Proceed to "Defining the Output Parameters," on page 90.

Defining the Output Parameters

You must now define the values to read after the simulation code has been executed.

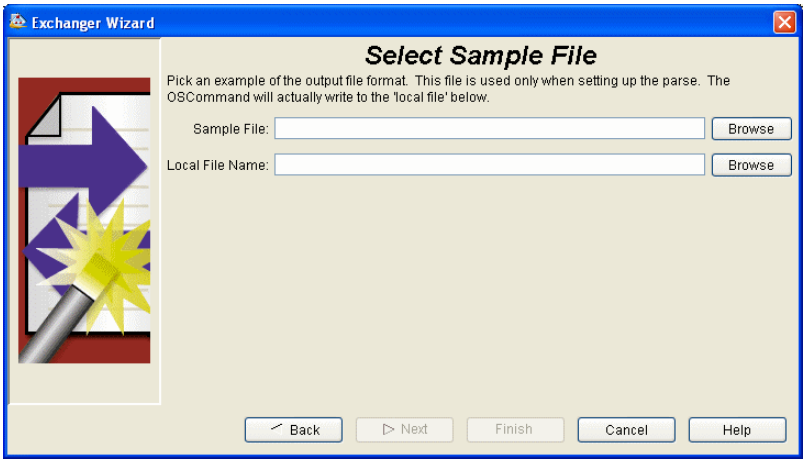
To define output parameters:

1. Click the **Output** tab on the Simcode component editor. The contents of the tab appear.



2. Click the large **Click here to open a new Data Source** button in the center of the tab.

The Exchanger Wizard appears.

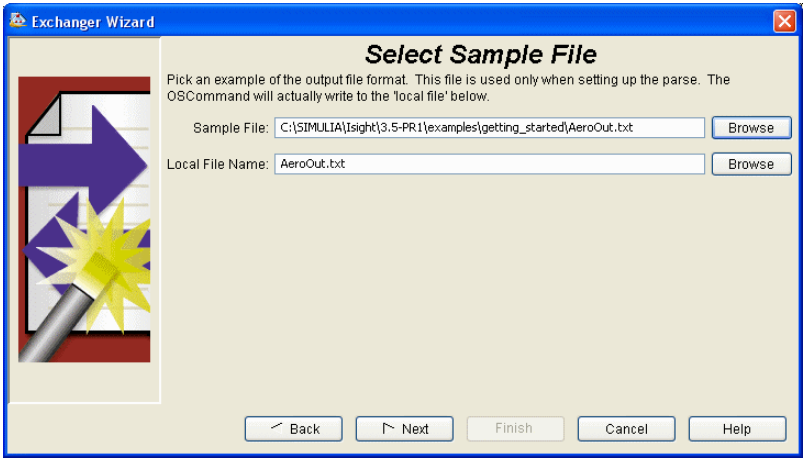


3. Click the **Browse** button adjacent to the **Sample File** text box. The Open dialog box appears.

4. Click the **AeroOut.txt** file; then, click **Open**.

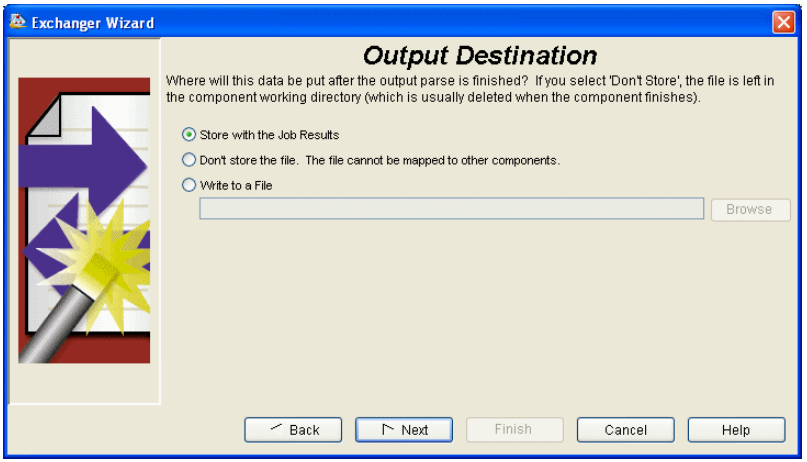
   Note: If you are using UNIX or Linux, you should select **AeroOut** from the subdirectory that matches your operating system.

   The full path of the file is added to the Sample file to use when designing the Data Exchange text box.



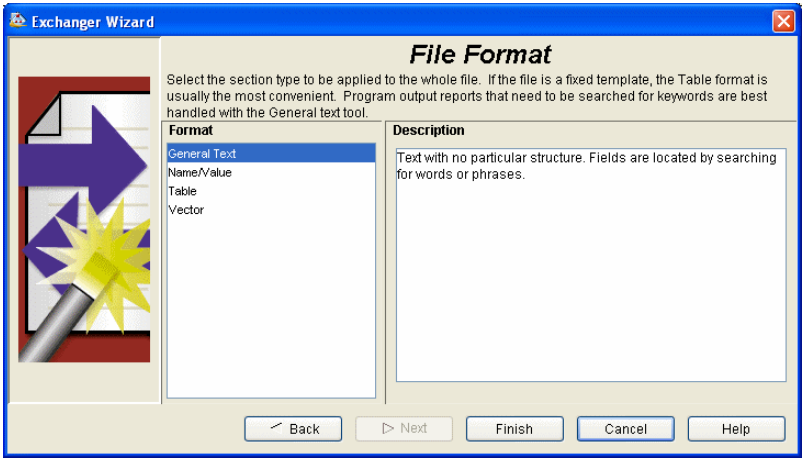
The Local File Name setting is automatically entered as `AeroOut.txt` (or `AeroOut` on UNIX or Linux).

5. Click **Next**. The Output Destination screen appears.

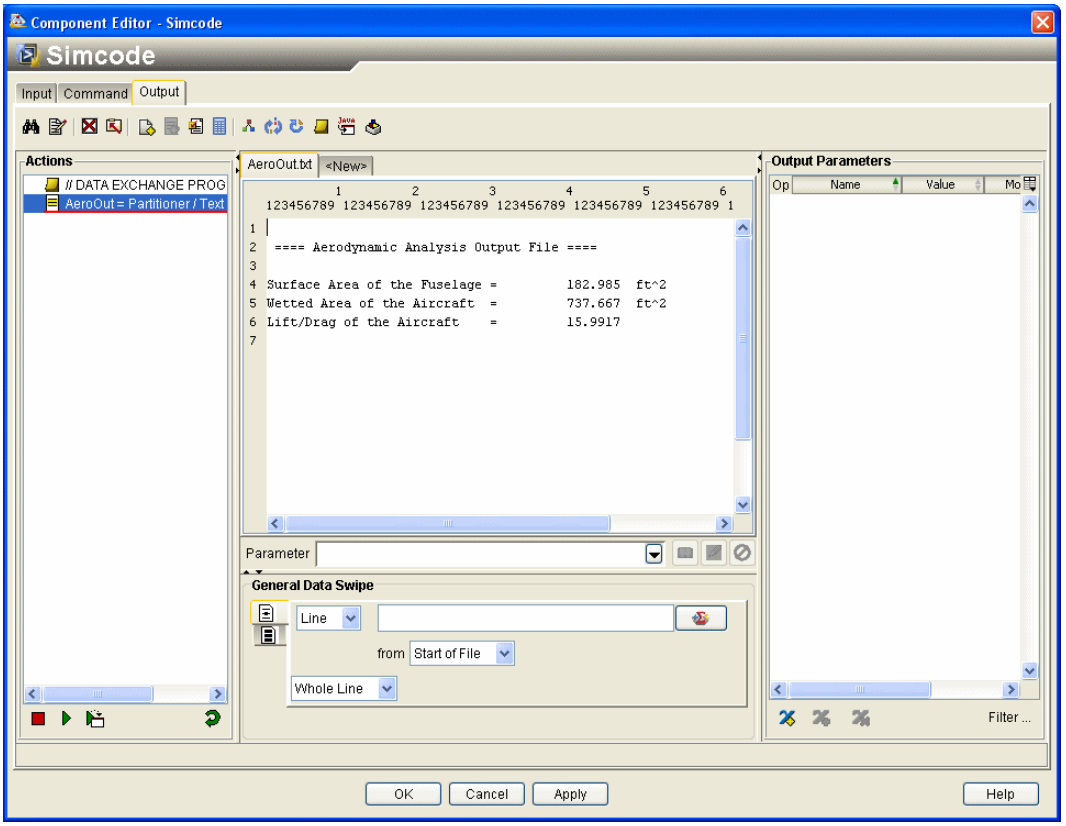


Now you need to decide where the output data will be stored after the output parse is finished.

6. Click **Next** to accept the default option. The File Format screen appears.



7. Verify that the **General Text** option is highlighted in the Format area; then, click **Finish**. You are returned to the component editor, and the output file information is displayed.



The left side of the editor shows the current list of actions for the file. The center of the editor shows a copy of the output template file. The right side displays the output parameters. Now you need to specify the output parameters using the same procedure as used to specify the input parameters.

8. Click the value **182.985** in the center of the editor. The number is highlighted.

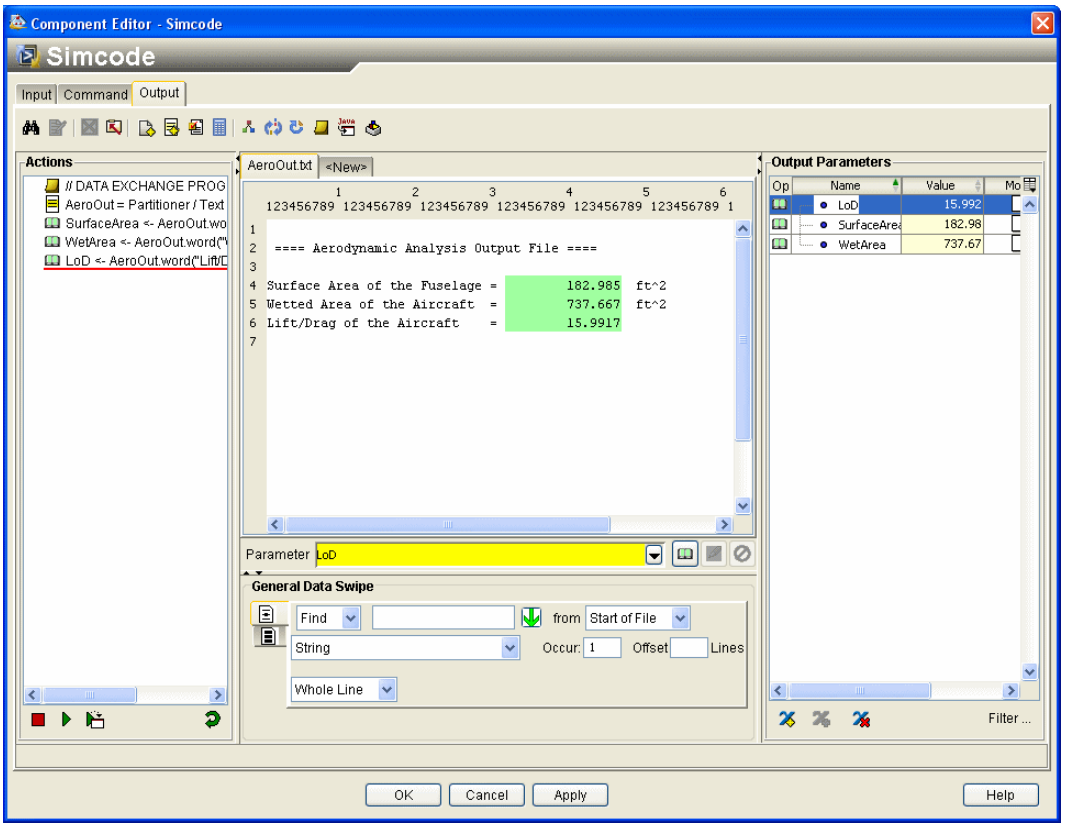
9. Type `SurfaceArea` in the **Parameter** text box; then, click the **Read** button  . The Actions area is updated, and the parameter is added to the list on the right side of the editor.

10. Click the value **737.667** in the center of the editor until the number is highlighted.

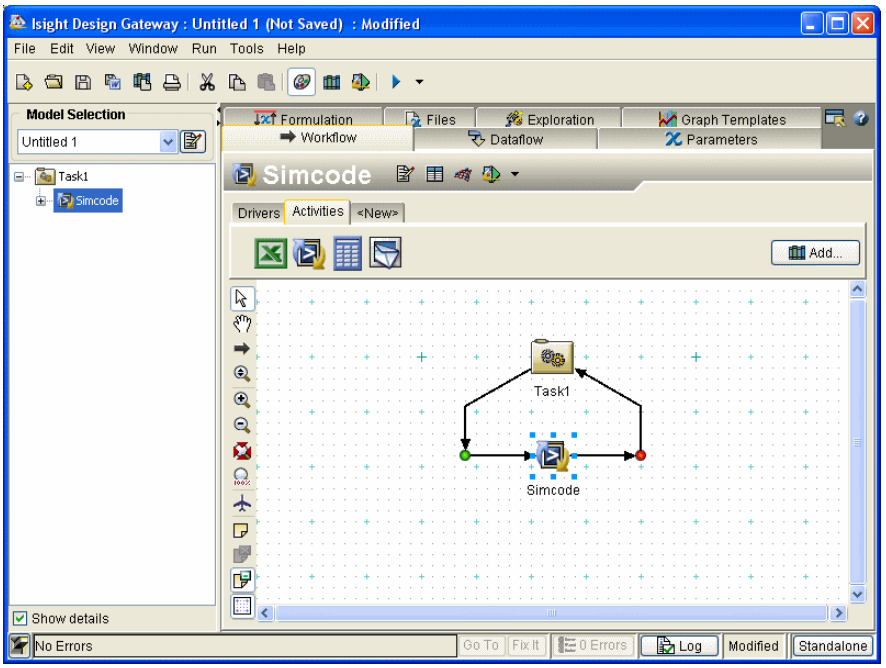
11. Type `WetArea` in the **Parameter** text box; then, click the **Read** button  . The Actions area is updated, and the parameter is added to the list on the right side of the editor.

12. Click the value **15.9917** in the center of the editor until the number is highlighted.

13. Enter `LoD` in the **Parameter** text box; then, click the **Read** button  . The Actions area is updated, and the parameter is added to the list on the right side of the editor. Your editor should appear as shown below.



14. Click **OK** to save your changes and close the Simcode component editor. You are returned to the Design Gateway.



15. Proceed to "Renaming the Component" on this page.

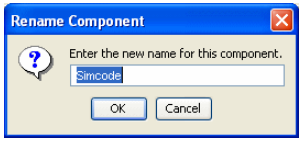
Renaming the Component

In order to make the component's purpose more intuitive, or to help you more easily remember the purpose of the component, you can rename it. This step is completely optional, and does not affect the execution or the model. For information on other options available for customizing Isight, refer to the *Isight User's Guide*.

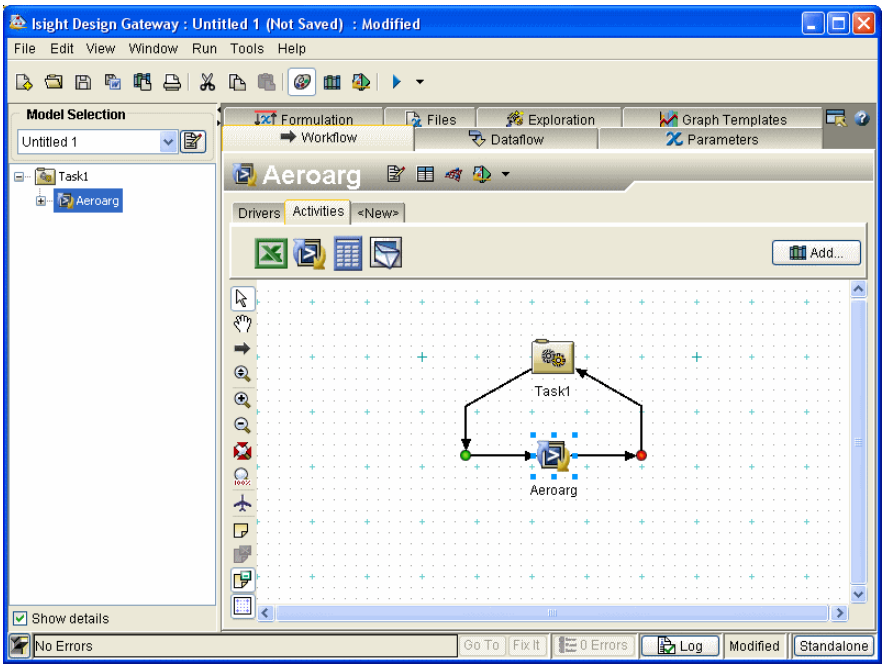
To rename the component:

1. Right-click the **Simcode** icon in the workflow. A menu appears.
2. Click the **Rename** option.

The Rename Component dialog box appears.



3. Type `Aeroarg` in the text box; then, click **OK**. You are returned to the Design Gateway, and the new name now appears below the component, in the Component Title Bar (immediately above the Workflow canvas), and in the Model Explorer on the left side of the interface.



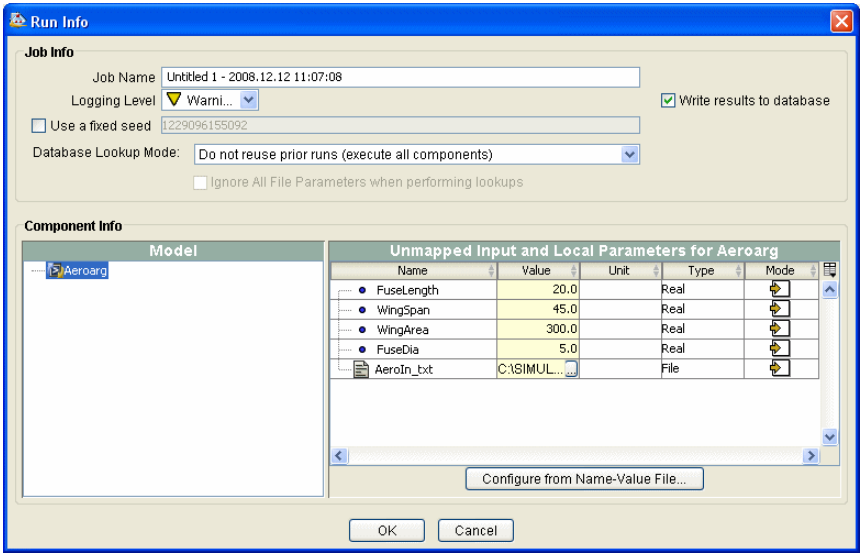
4. Proceed to "Executing the Model," on page 97.

Executing the Model

Isight now has enough information to execute the model.

To execute a model:

1. Right-click the **Aeroarg** icon on the workflow. A menu appears.
2. Point to the **Run** option. Additional options appear.
3. Click the **Configure and Run Component (Aeroarg)** option. The Run Info dialog box appears.

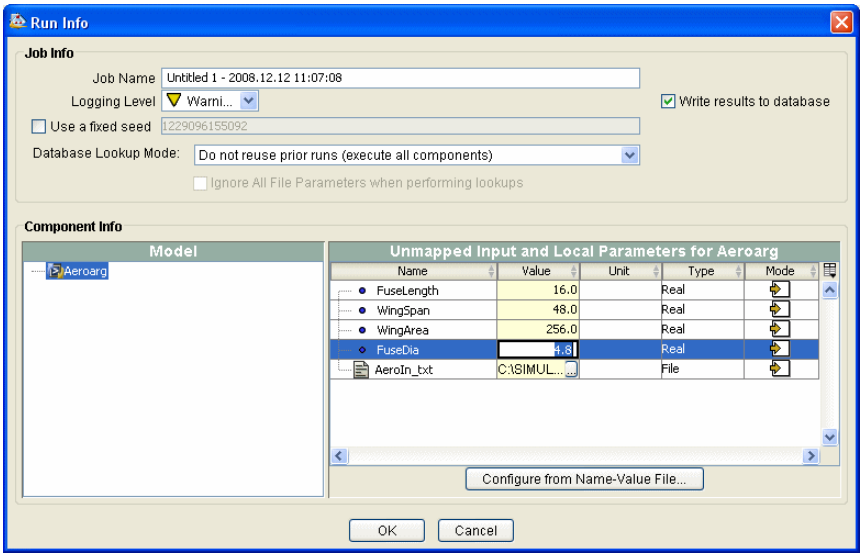


This interface allows you to enter values of interest for the input parameters prior to running the simulation code.

4. Change the entries in the **Value** column for the following input parameters (be sure to hit the Enter key after changing each value):

   - **FuseLength**: `16.0`
   - **WingSpan**: `48.0`
   - **WingArea**: `256.0`
   - **FuseDia**: `4.8`

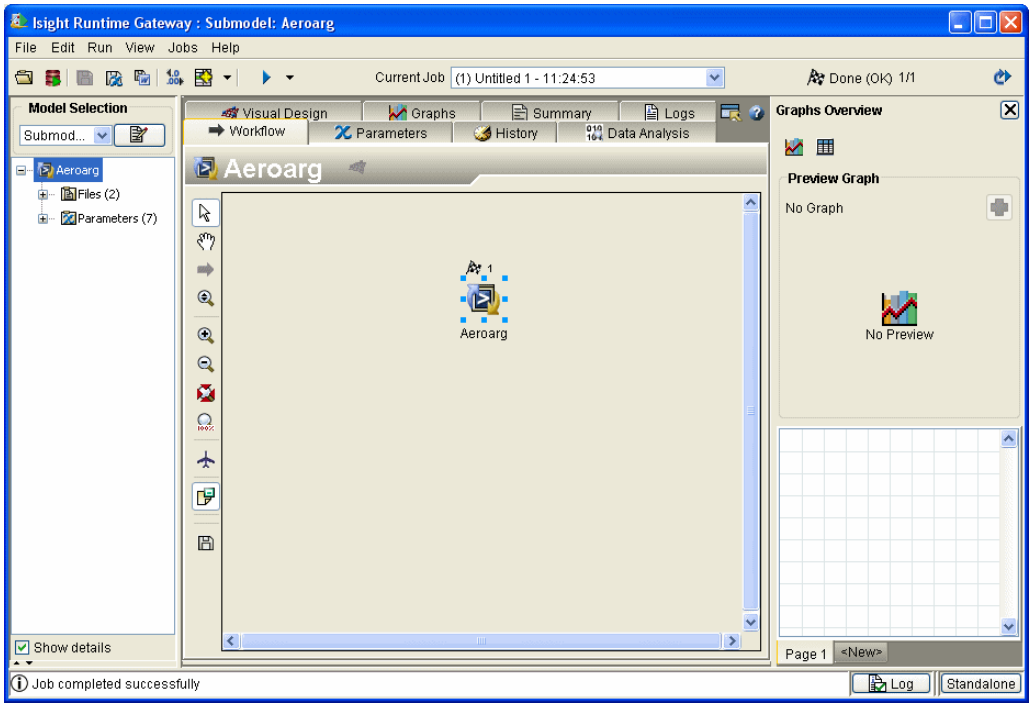
Your dialog box should appear as shown below.



5. Click **OK**. The Runtime Gateway appears, and the simulation code runs with the values specified.

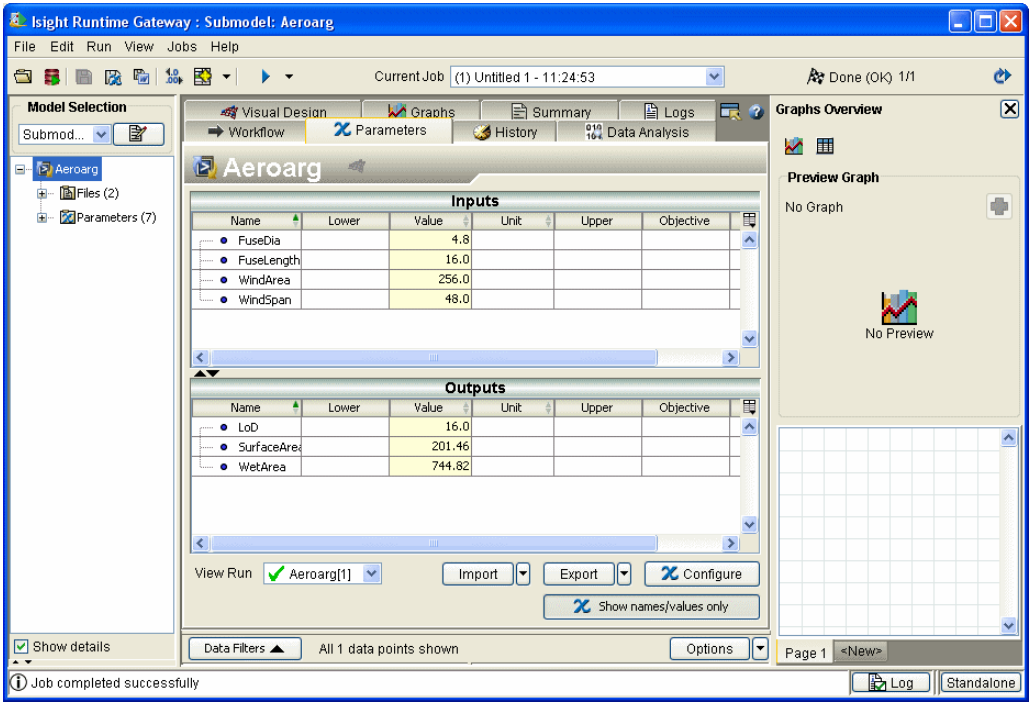
   Note: A dialog box may appear indicating that the current mode of execution only executes the selected component and anything below it. Click **OK** to continue with this example.

Once execution is completed, the “Job completed successfully” message appears in the bottom left corner of the interface, and the icon appears above the component in the workflow, as shown below.



6. Click the **Parameters** tab.

This tab allows you to see the output values for the selected input values.



7. Select **Close Window** from the **File** menu to close the Runtime Gateway. You are returned to the Design Gateway.

8. Proceed to "Publishing a Component" on this page.

Publishing a Component

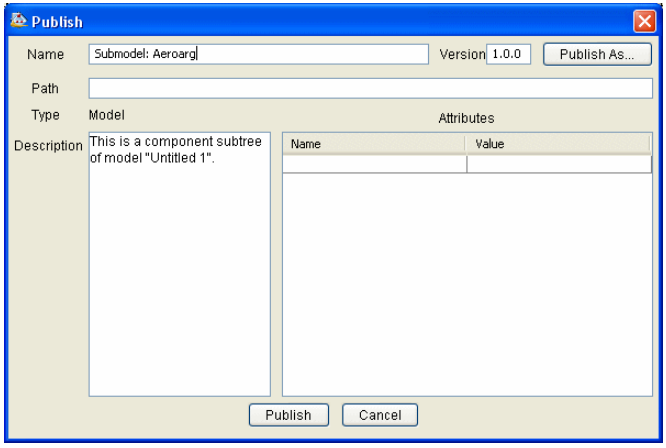
Publishing a component allows it to be accessed by any user who has access to the Library to which the component is published. Whole models can also be published. For more information, refer to the *Isight User's Guide*.

Components and models can be published to either a Standalone Library or an ACS Library (when connected to the Fiper environment), which is accessible to anyone who is connected to that same ACS. The process of publishing a component or model, retrieving a component or model, or deleting a component or model does not differ

based on the type of Library being used. In this example, you will be accessing the local (Standalone) Library.

To publish your component:

1. Right-click the **Aeroarg** component on the Design Gateway Workflow tab. A menu appears.
2. Click the **Publish** option. The Publish dialog box appears.



The component, by default, is named *Aeroarg*. You can change the name and change the description, if necessary, in the corresponding text boxes.

For more information on the other options available on this dialog box, refer to the *Isight User's Guide*.

3. Click **Publish**. The Aeroarg component is published to the Library, and can be accessed at a later time.

Viewing a Published Component

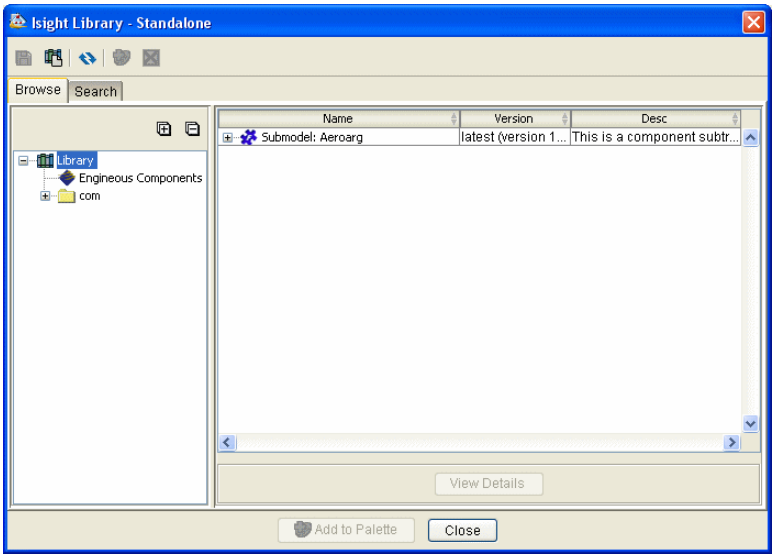
After publishing your component, you can verify that it is viewable in your Library.

To view a published component:

1. Select **Library** from the Design Gateway **View** menu.

   Note: You can also click the button on the Design Gateway toolbar or click the **Add...** button on the component palette.

   The Library dialog box appears.



2. Verify that the component is listed at the top level of the Library. It should appear on the right side of the Library dialog box.

   For more information on using the Library, publishing components, and adding a published component to a model workflow (retrieving them from the Library), refer to the *Isight User's Guide*.

3. Click **Close** to exit the Library dialog box.

Saving the Model

It is recommended that you save this example model, especially if you plan on performing the Isight example contained in the next chapter. The Isight example builds off of the workflow created in this example.

To save the model:

1. Select **Save As...** from the Design Gateway **File** menu. The Save dialog box appears.
2. Navigate to the location where you want to store the model (it is recommended that you save the file in the getting_started directory); then, type a name for the model (such as, `simcode_example`) in the **File name** text box.

   Important: Due to naming limitations, do not use the following characters in your model name:

   #, ?, &, %, !, \, $, {
3. Click **Save**. The file is saved as a *.zmf file (Zipped Model File).

   Notice that the file name you specify is also added to the Model Selector on the left side of the Design Gateway (above the Model Explorer). For more information on this portion of the Design Gateway, as well as setting model properties, refer to the *Isight User's Guide*.
4. Proceed to Chapter 5 "Engineering Example", if desired.

Isight

5 Engineering Example

This chapter guides you through the engineering features of Isight, including design drivers, the Task Plan feature, the Approximation Viewer (Visual Design Driver), and Engineering Data Mining. It also demonstrates how these features can be applied to study different aspects of your problem, and ultimately yield improved designs.

Overview

In Isight, the term “analysis” refers to the portion of your model that, when provided with a specific single set of input parameter values, provides a corresponding single set of output parameter values, which relate directly to performance or quality metrics for the design. The purpose of Isight is to provide tools to help you determine what values of the input parameters will result in more desirable values of the output parameters.

Analysis Definition

A typical usage scenario for Isight is to define the “analysis” portion of your model and then to apply the various Isight tools to that analysis. For the purpose of this section, we will assume that you are starting with a model that already has some Task in it to represent the analysis. Specifically, this section will refer to the aerodynamic analysis described in Chapter 4 “Simcode Example”, but the overall process of using the Isight features as described in the rest of the section applies to any analysis model you have defined.

Design Drivers

Isight provides various components to help study or improve the design by trying alternative sets of input values in a systematic and intelligent manner. These are categorically referred to as “design drivers”. Specifically, Isight offers design driver components for Design of Experiments (DOE), Optimization, and Monte Carlo Simulation. These can be applied to the analysis either individually or in a combined manner using a Task Plan. Each of these modes is described below.

Using the Design Drivers

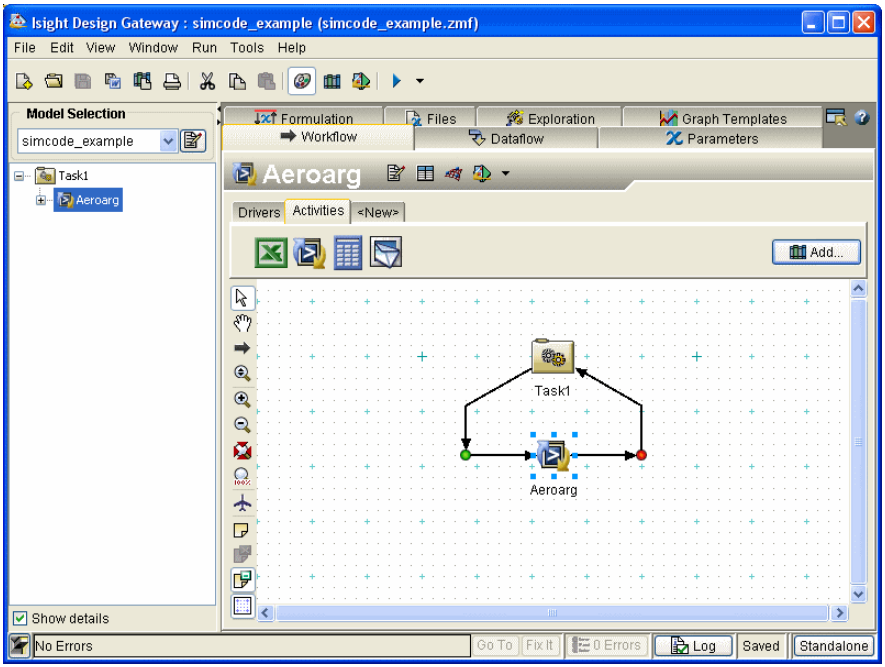
In this example, you will add a DOE component to the model workflow (using the Change To option) and configure it. You will use the example created in Chapter 4 “Simcode Example” as a basis for the procedures described in this chapter. Be sure that you have completed that chapter’s example file before proceeding.

Adding a Design Driver Component to an Existing Model

To use the design drivers:

1. Start the Isight Design Gateway, if necessary. If you are continuing from the previous example (the Simcode Example), proceed to the next step. For more information on how to start the Design Gateway, see “Accessing the Design Gateway,” on page 48.
2. Verify that you have the Simcode Example opened on the Design Gateway. If you have closed this example (and saved it), you can reopen it by selecting **Open from Disk...** from the **File** menu; then, locate the file and click **Open**.

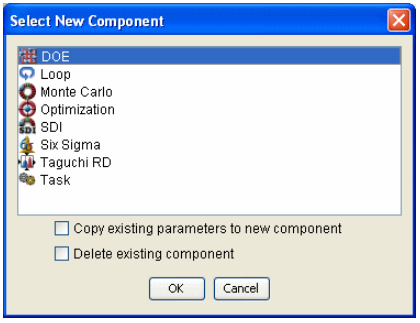
The model is loaded into the Design Gateway.



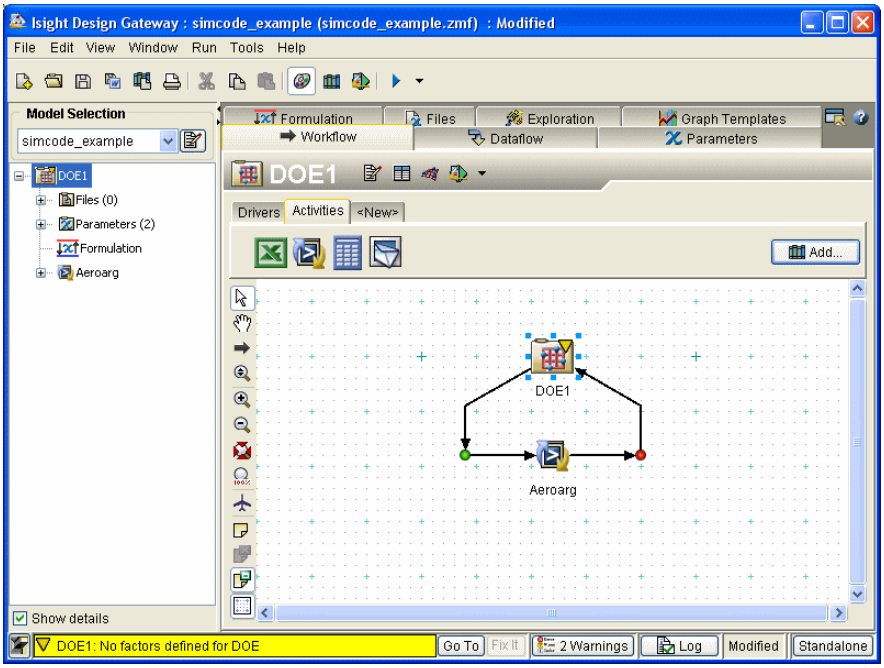
Now you need to change the Aeroarg Task component to a DOE component.

3. Right-click the **Task1** component; then, select **Change To** from the menu that appears. A submenu appears.

4. Click the **New...** option. The Select New Component dialog box appears.



5. Verify that the **DOE** option is selected; then, click **OK**. You are returned to the Design Gateway, and the Task component has been changed to a DOE component.

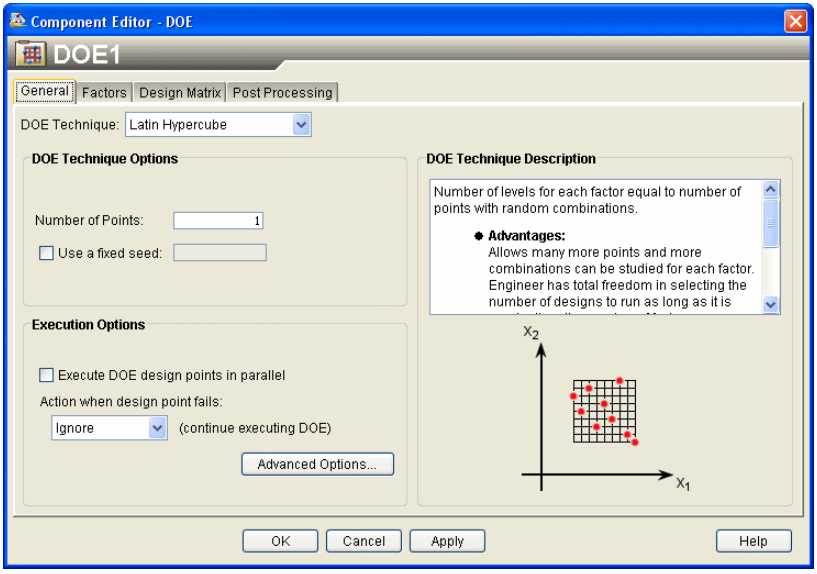


6. Proceed to "Configuring the DOE Component," on page 109.

Configuring the DOE Component

Now you need to configure the DOE component.

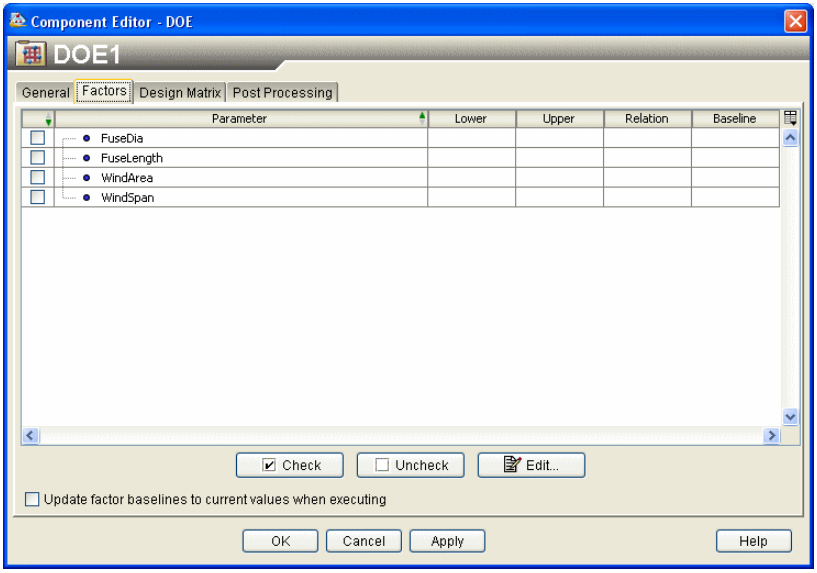
1. Double-click the **DOE** component on the Workflow tab. The component editor appears.



   For this example, you'll use the Latin Hypercube technique and select all available factors and responses.

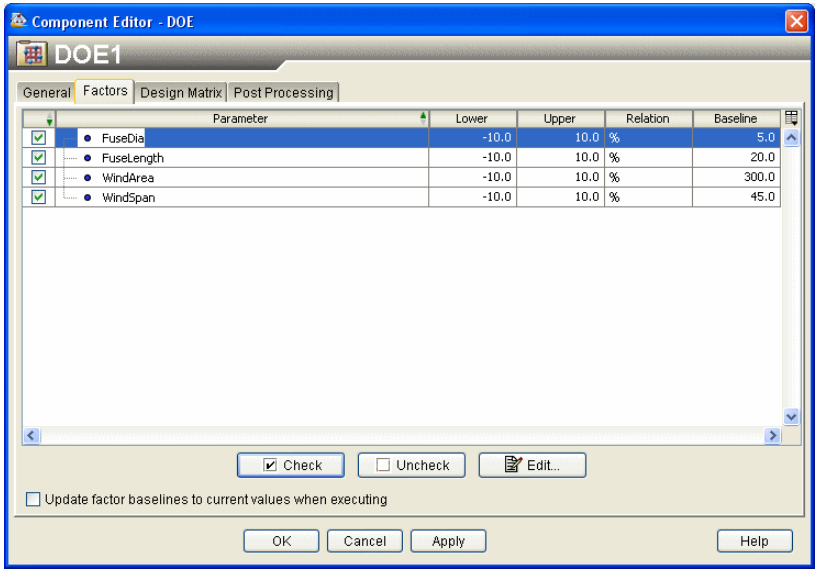
2. Verify that **Latin Hypercube** is selected on the **DOE Technique** drop-down list.

3. Click the **Factors** tab. The contents of the tab appear.

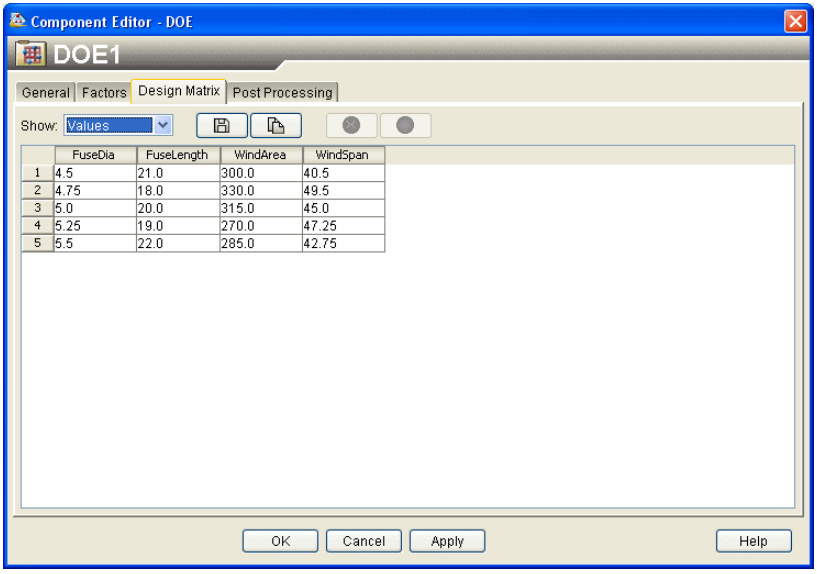


4. Click the **Check** button near the bottom of the tab. A message appears, verifying that you want to select all of the available parameters.

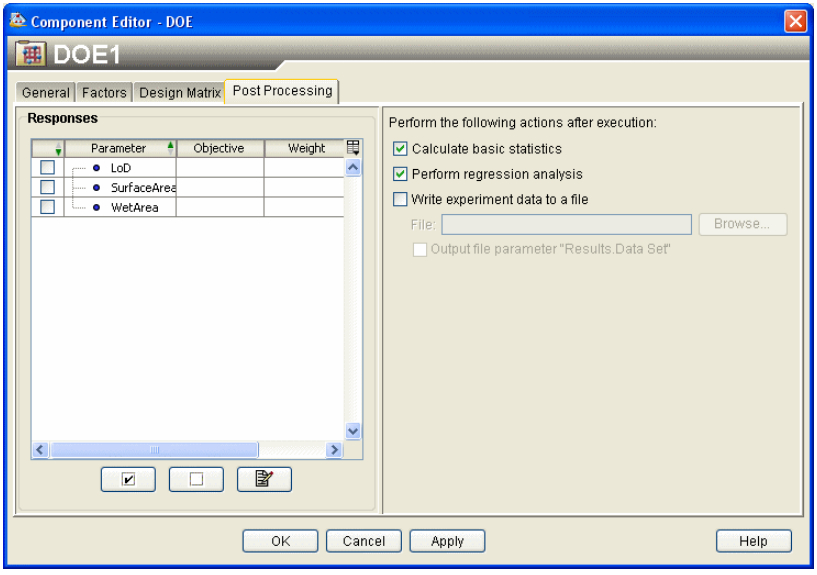
5. Click **Yes**. All of the listed factors are automatically selected.

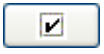


6. Click the **Design Matrix** tab. The contents of the tab appear.



7. Review the Design Matrix; then, click the **Post Processing** tab. The contents of the tab appear.

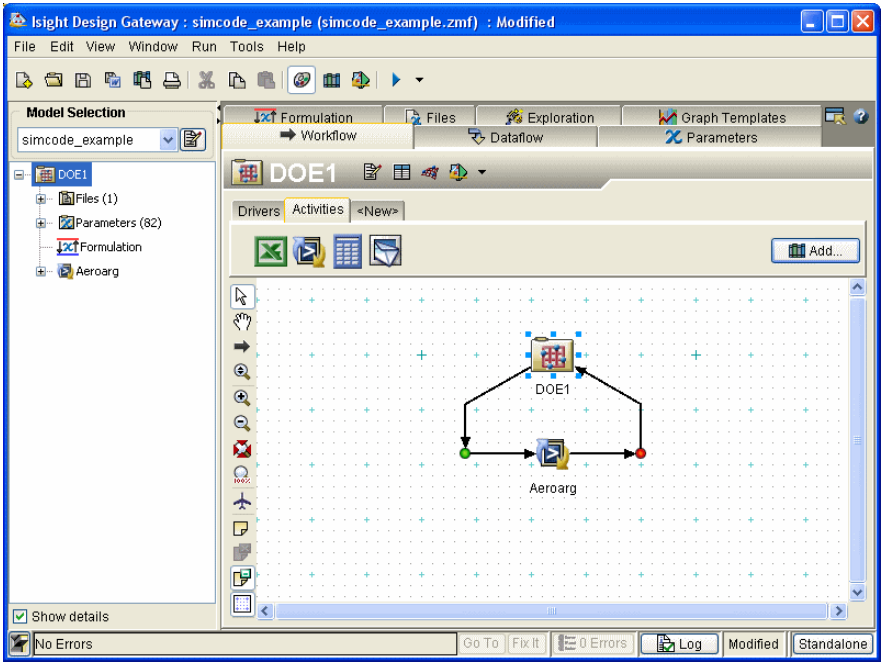


8. Click the  button. A message appears, verifying that you want to select all of the available parameters.

9. Click **Yes**. All of the listed responses are automatically selected.

   For more information on all of the options available on this component editor, refer to the *Isight Component Guide*.

10. Click **OK** to close the component editor and save all of your changes. You are returned to the Design Gateway.



   Notice that the warning message that appears when you first added the DOE component is now gone.

11. Select **Close Current Model** from the Design Gateway **File** menu. The Save Model? dialog box appears.

12. Click **No**; then, proceed to "Using the Task Plan Feature," on page 113.

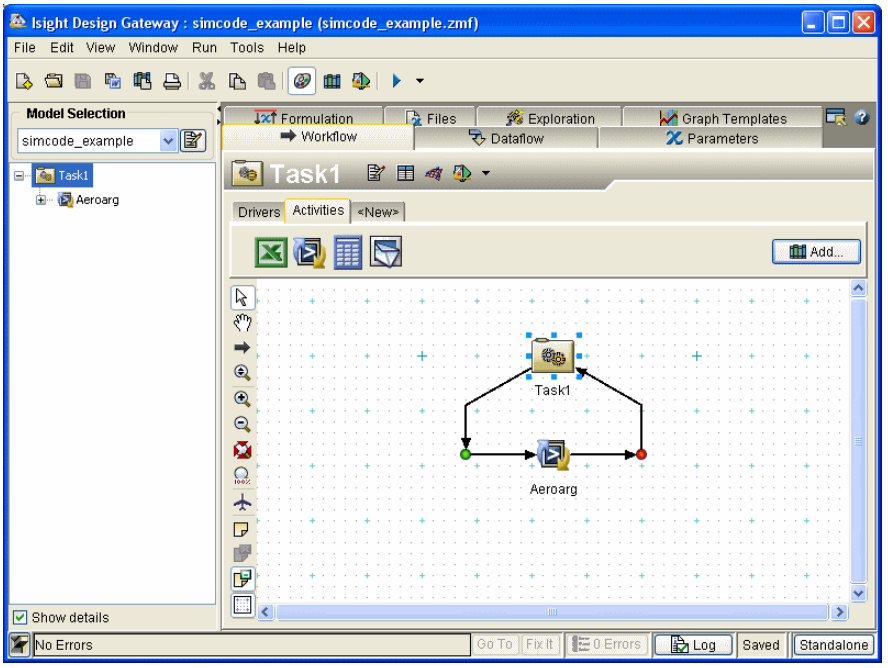
Using the Task Plan Feature

A Task component typically just runs its subflow a single time. However, you can use this component to define a Task Plan comprised of design driver components that will each run in sequence, automatically.

In this example, we will create a Task Plan that will automatically execute a DOE followed by an Optimization.

To create a Task Plan:

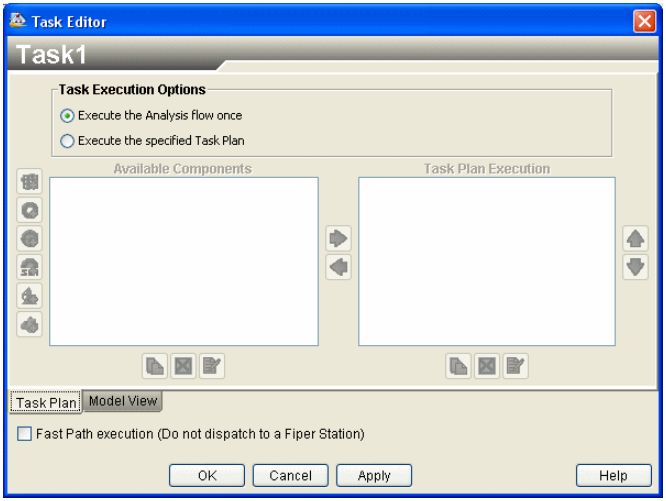
1. Click **Open from Disk...** from the Design Gateway **File** menu; then, select the Simcode example you created in the previous chapter and open it. The model is loaded into the Design Gateway. In essence, this action removes the DOE component you just created, and replaces it with the original Task component (Aeroarg). You'll now recreate the DOE component as part of a Task Plan.



Now you'll access the Task Plan feature.

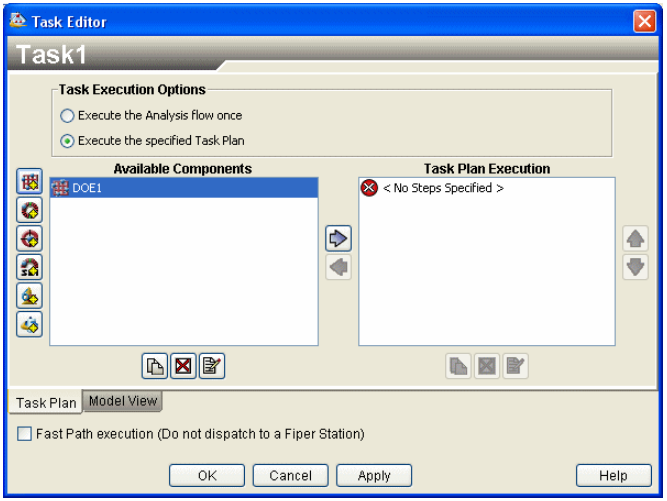
2. Double-click the **Task1** component.

The component editor appears.



3. Click the **Execute the specified Task Plan** radio button. The other options on the lower portion of the editor are activated.

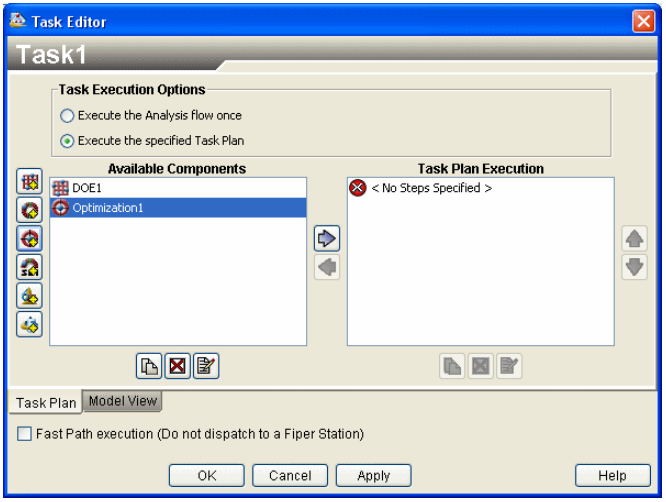
4. Click the  button on the left side of the editor. A DOE component is added to the Available Components list.



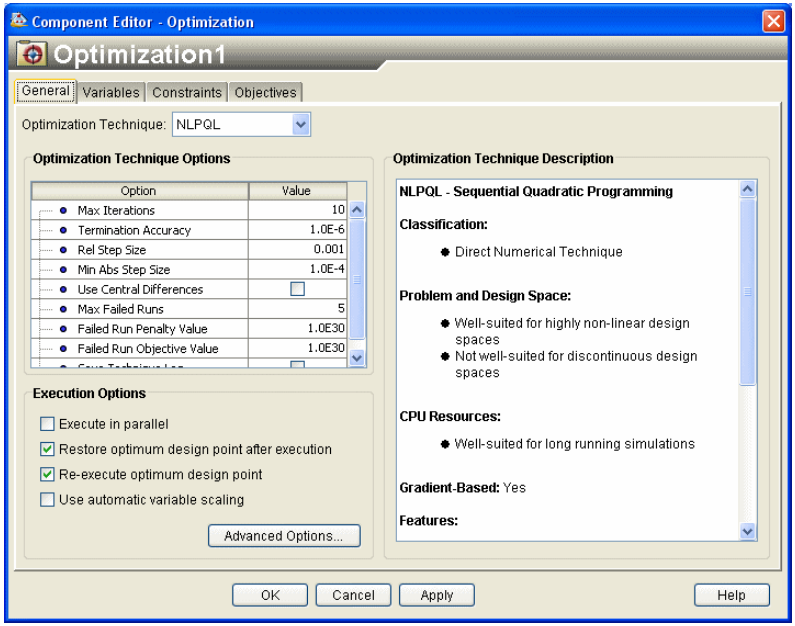
5. Double-click the component in the Available Components list. The DOE component editor appears.

6. Configure the component as described in "Configuring the DOE Component," on page 109, but do not close the model. Proceed to the next step after you exit the DOE component editor.

7. Click the  button (the third button from the top of the left side of the editor). An Optimization component is added to the Available Components list.

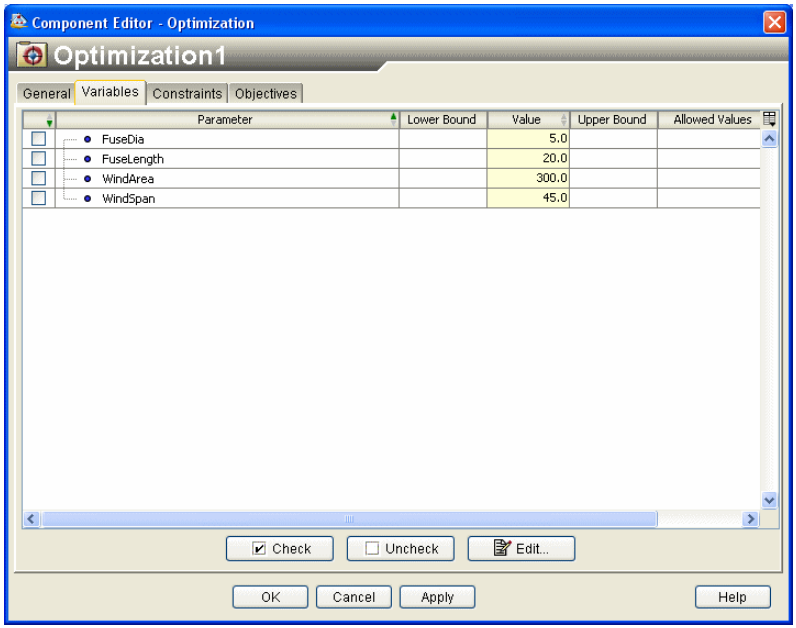


8. Double-click the component in the Available Components list. The component editor appears.



9. Select **NSGA-II** from the **Optimization Technique** drop-down list.

10. Click the **Variables** tab. The contents of the tab appear.



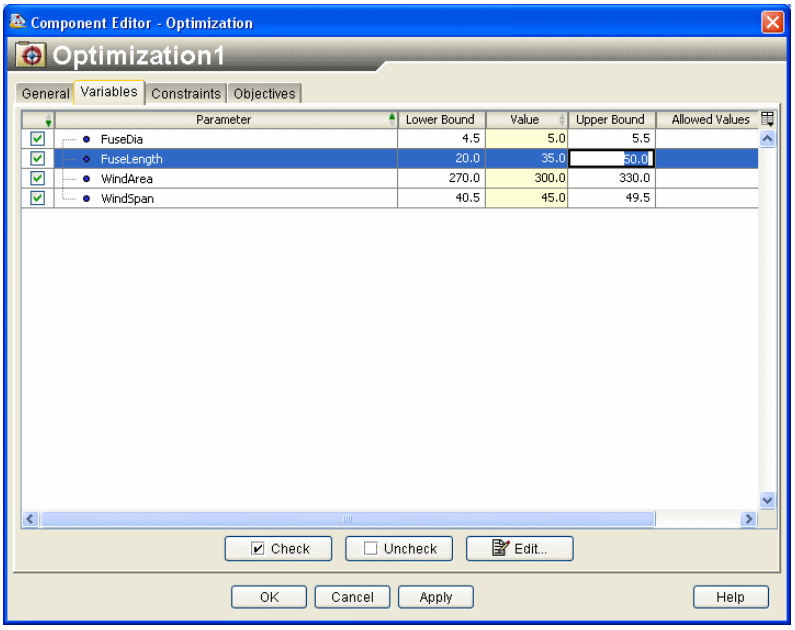
11. Click the **Check** button near the bottom of the tab. A message appears, verifying that you want to select all of the available parameters.

12. Click **Yes**. All of the listed variables are automatically selected, and they will act as the design variables that will be automatically modified during the optimization execution.

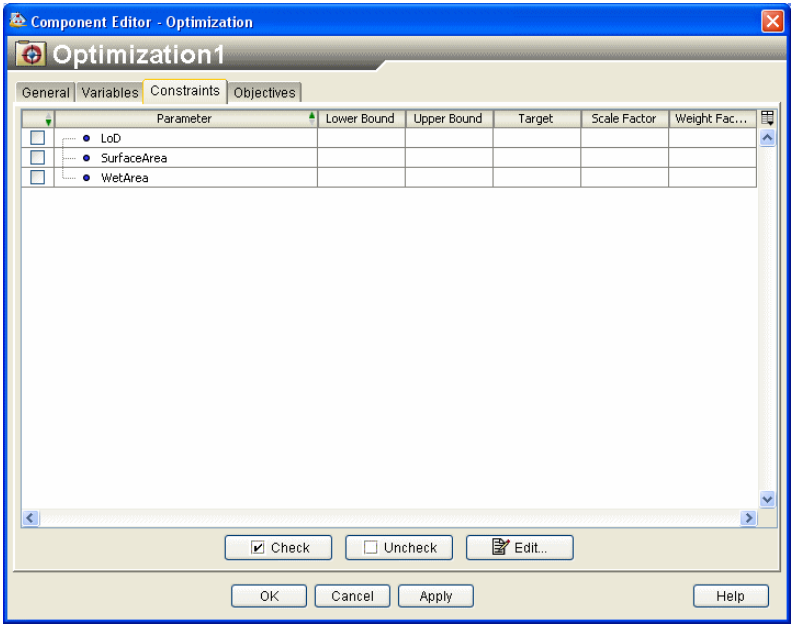
13. Type the following values for the **FuseLength** parameter in the corresponding columns:

    - **Lower Bound**: `20.0`
    - **Value**: `35.0`
    - **Upper Bound**: `50.0`

Your tab should appear as shown below.



14. Click the **Constraints** tab. The contents of the tab appear.

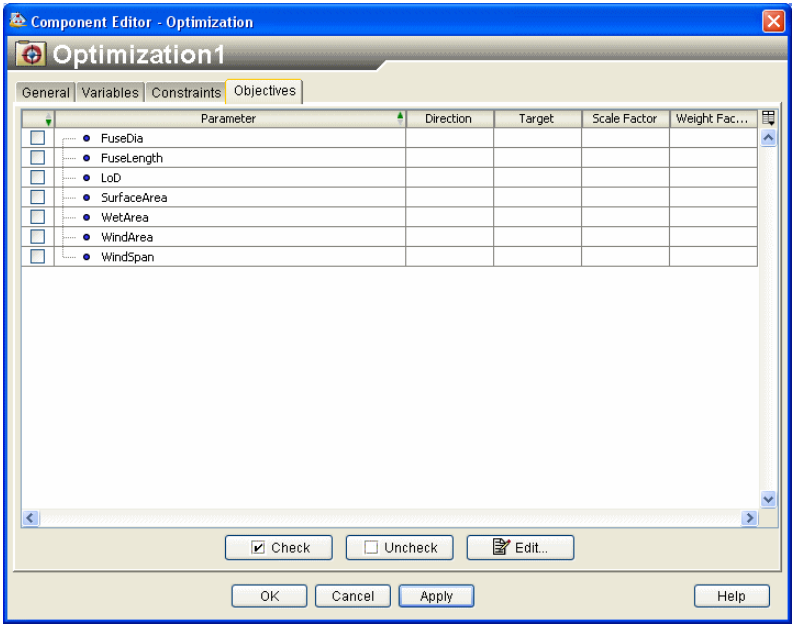


15. Click the **Check** button near the bottom of the tab. A message appears, verifying that you want to select all of the available parameters.

16. Click **Yes**. All of the listed constraints are automatically selected, and they will act as the constraints that will be applied as upper and lower bounds on the design variables (if applicable).

17. Type `4.5` in the **Lower Bound** column for the **SurfaceArea** parameter.

18. Click the **Objectives** tab. The contents of the tab appear.

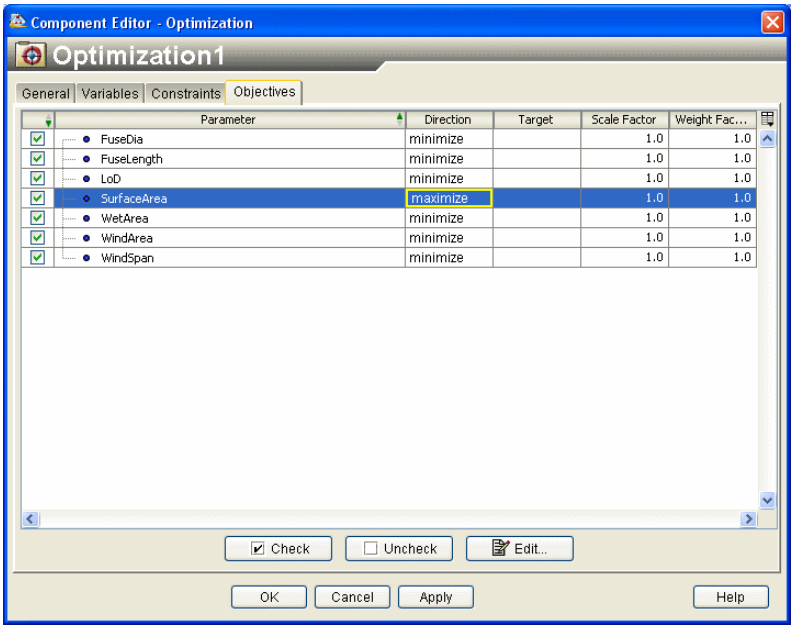


19. Click the **Check** button near the bottom of the tab. A message appears, verifying that you want to select all of the available parameters.

20. Click **Yes**. All of the listed objectives are automatically selected, and they will be applied as objectives (to minimize or maximize output values during optimization), as applicable.

21. Click **minimize** in the **Direction** column for the **SurfaceArea** parameter. A drop-down list appears in the corresponding cell.

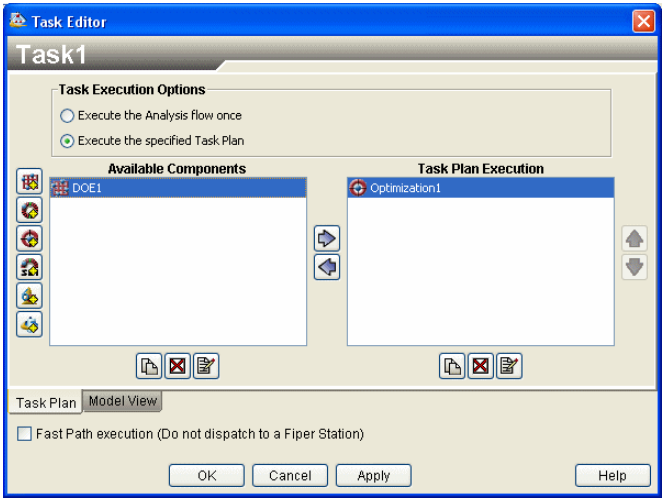
22. Select **maximize** from the options that appear. Your tab should appear as shown below.



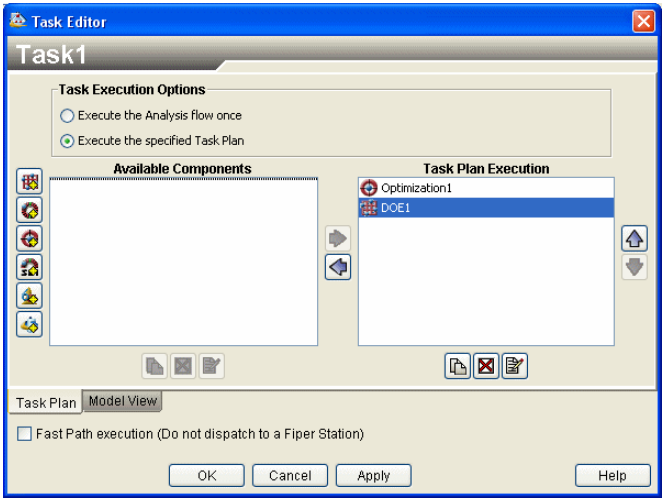
23. Click **OK**. The component editor is closed and you are returned to the Task Editor dialog box.

    Now the components are defined. However, they are not currently used in the model workflow. To designate a component for usage in a model workflow, you need to add it to the Task Plan Execution list.

24. Click the **Optimization** component in the Available Components list; then, click the  button in the center of the editor. The component is added to the Task Plan Execution list, and will be used in model execution.

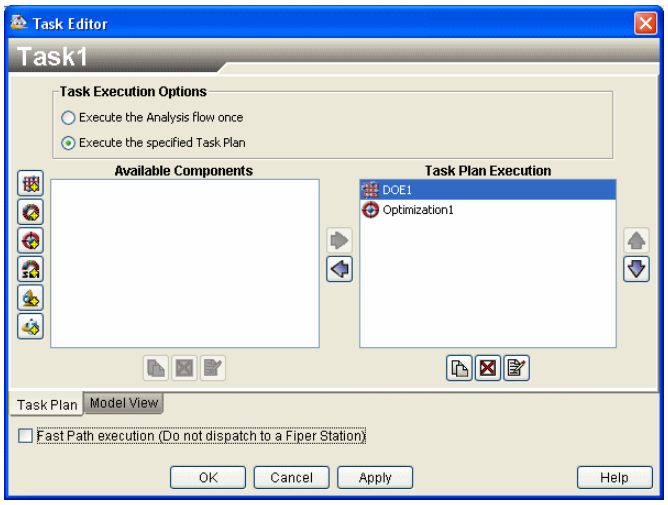


25. Click the **DOE** component in the Available Components list; then, click the  button in the center of the editor. Now both components will be used in model execution.



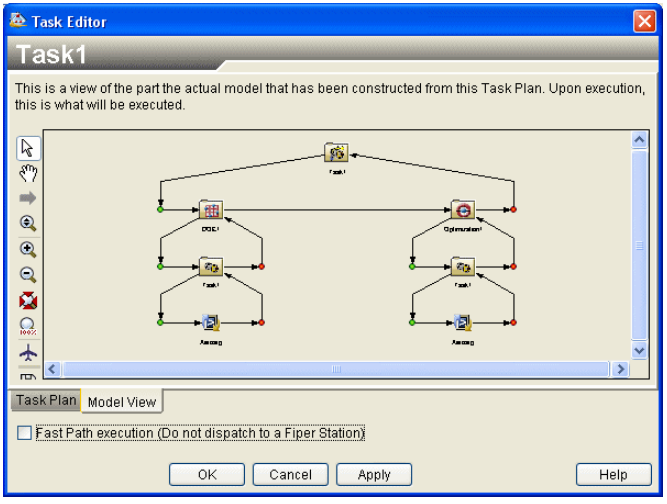
The order of the components in the Task Plan Execution list is important, since this order also determines the execution order of the components.

26. Click the  button on the right side of the Task component editor. The DOE component is moved above the Optimization component.



The Task Plan also gives you the ability to preview the overall model workflow and how it is affected by the Design Plan.

27. Click the **Model View** tab (below the component lists). The model workflow appears.



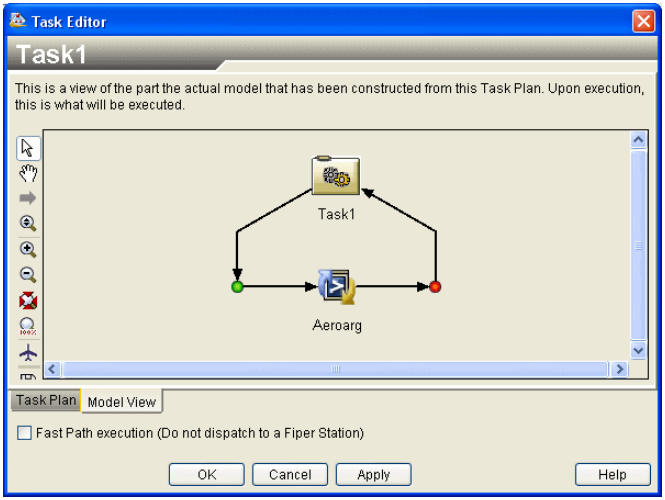
The Task Plan is, in essence, a shortcut for creating the workflow as shown in the model view. Notice that the Aeroarg component is now executed with both of the

defined Task Plan components (DOE and Optimization). For more information on how the workflow is affected by the Task Plan, refer to the *Isight User's Guide*.

You can switch back to analysis workflow only execution at any time by clicking the Execute the Analysis flow once radio button. This action saves the currently defined Task Plan. However, it is disabled and not used during model execution.

28. Click the **Task Plan** tab; then, click the **Execute the Analysis flow once** radio button. The Task Plan options are disabled, but your information is saved.

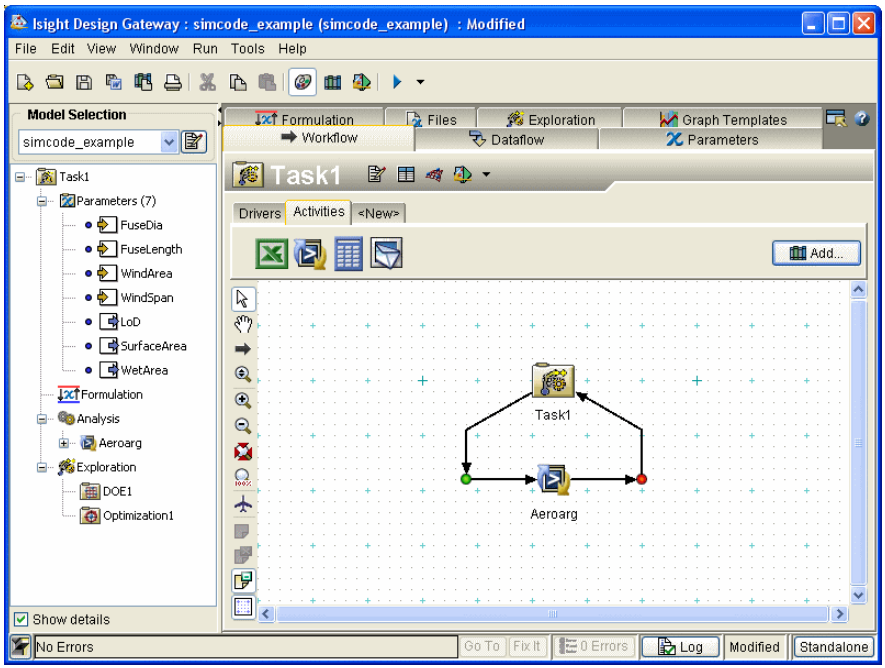
29. Click the **Model View** tab (below the component lists). The model workflow appears.



Notice that only the Aeroarg component appears in the workflow. Both of the Task Plan components are no longer present.

30. Click the **Task Plan** tab; then, click the **Execute the specified Task Plan** radio button to reactivate the Task Plan.

31. Click **OK** to save your Task Plan and close the component editor. You are returned to the Design Gateway. Notice that the Task component icon has changed to show that it now contains Task Plan information.



32. Select **Save As...** from the Design Gateway **File** menu. The Save dialog box appears.

33. Save the model in the desired location with a new name (such as `engineering_example.zmf`). When prompted to change the model name to match the file name, click **Yes**.

    Important: Due to naming limitations, do not use the following characters in your model name:

    #, ?, &, %, !, \, $, {

34. Proceed to "Executing the Model," on page 125.

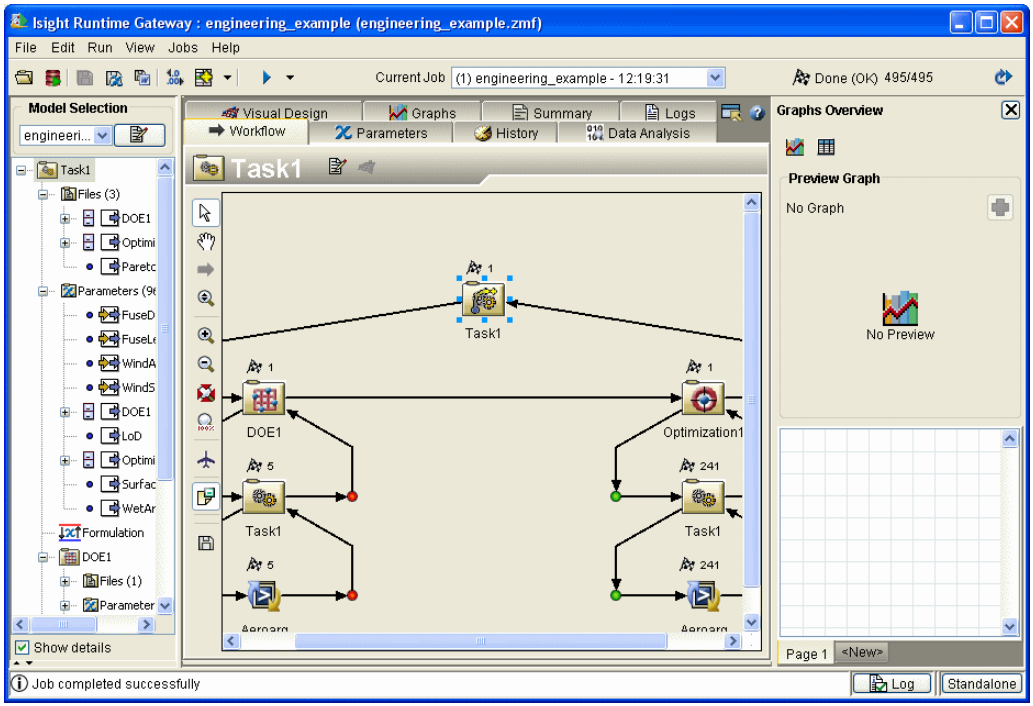
Executing the Model

Now that you have set up your model, it is time to execute it and generate the results.

To execute the model:

1. Click the **Task1** component on the **Workflow** tab to select it; then, click the button on the Component Title Bar. Be sure to *not* click the similar-looking button on the Design Gateway's toolbar. The Runtime Gateway appears and the model is executed.

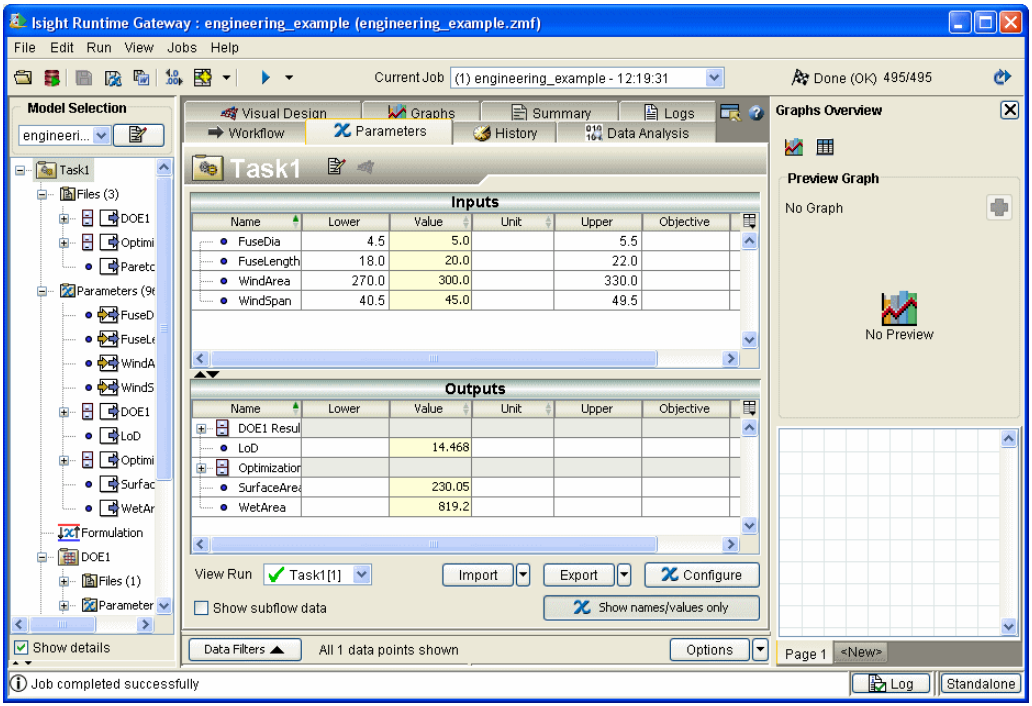
   Once execution is completed, the message "Job completed successfully" appears in the bottom left corner of the interface, and icons appear above each component in the workflow, as shown below.



   The Runtime Gateway provides access to graphs and tables that allow you to view execution information. You can also view data using the tabs on the Runtime Gateway itself.

2. Click the **Parameters** tab.

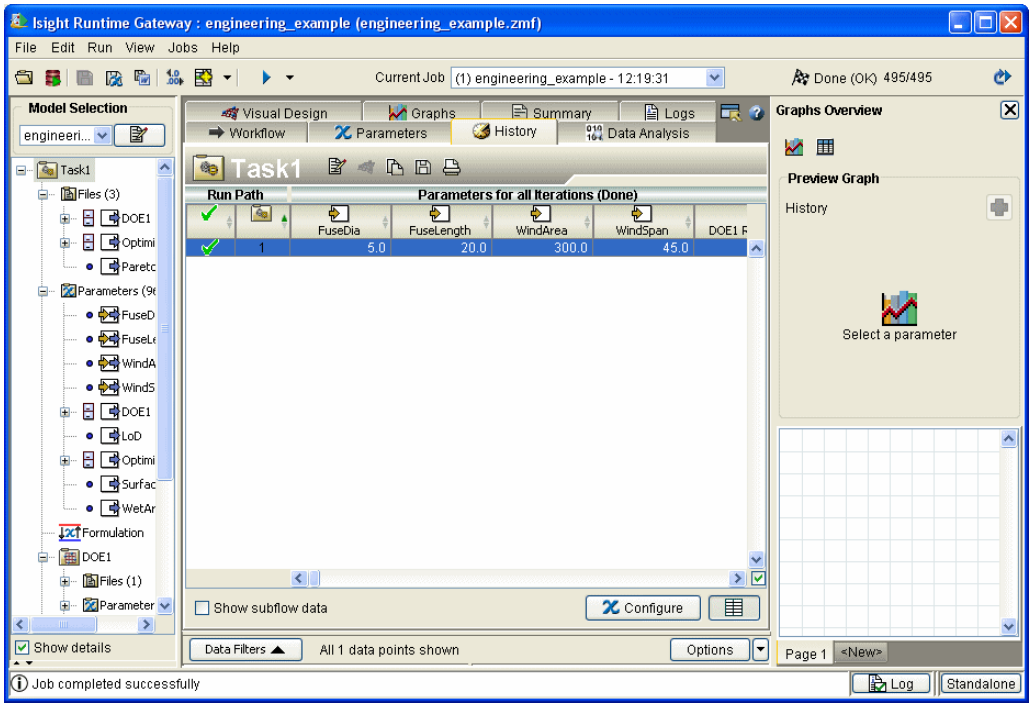
This tab allows you to see the output values for the selected input values.



3. Click the different components listed on the left side of the Runtime Gateway to view their parameter information.

4. Click the **History** tab.

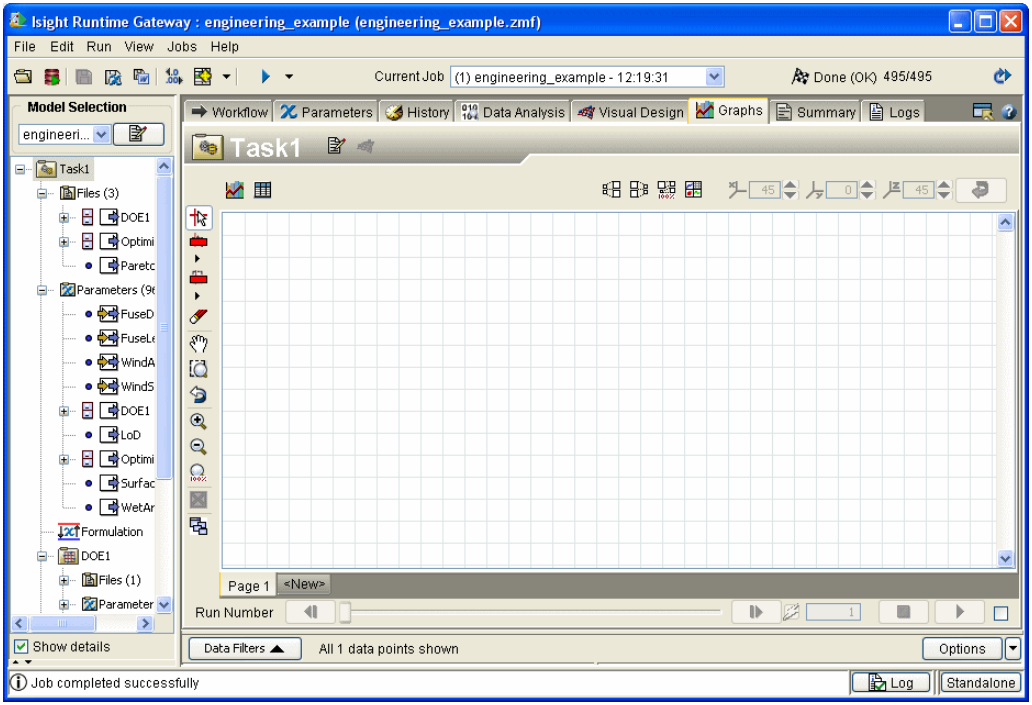
This tab shows parameter values for each run of the selected component.



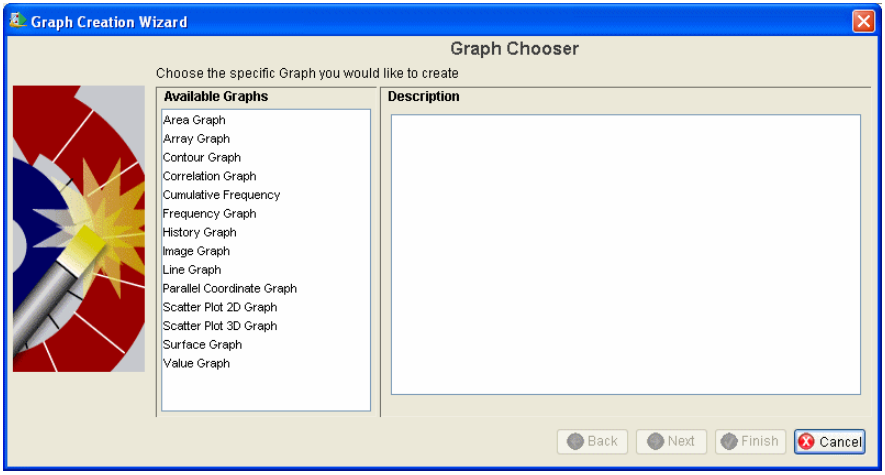
5. Click the different components listed on the left side of the Runtime Gateway to view their history information.

6. Click the **Graphs** tab.

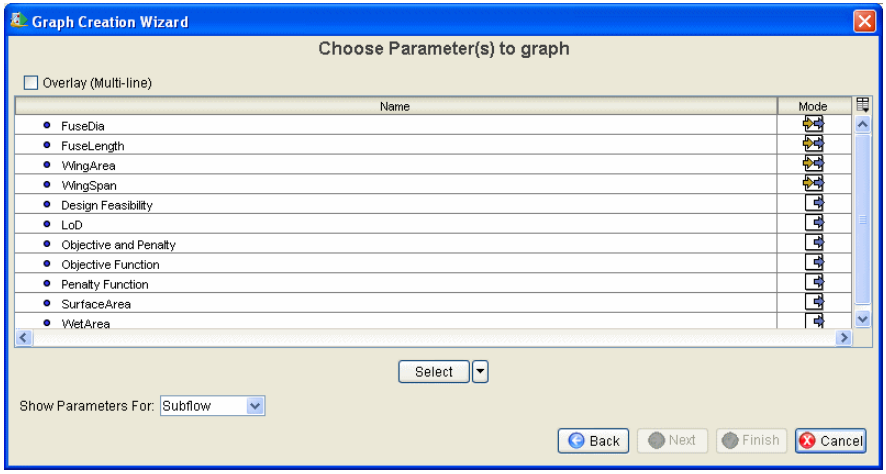
This tab allows you to create graphs and tables of the execution information.



7. Click the **Optimization1** component on the left side of the Runtime Gateway; then, click the button. The Graph Creation Wizard dialog box appears.



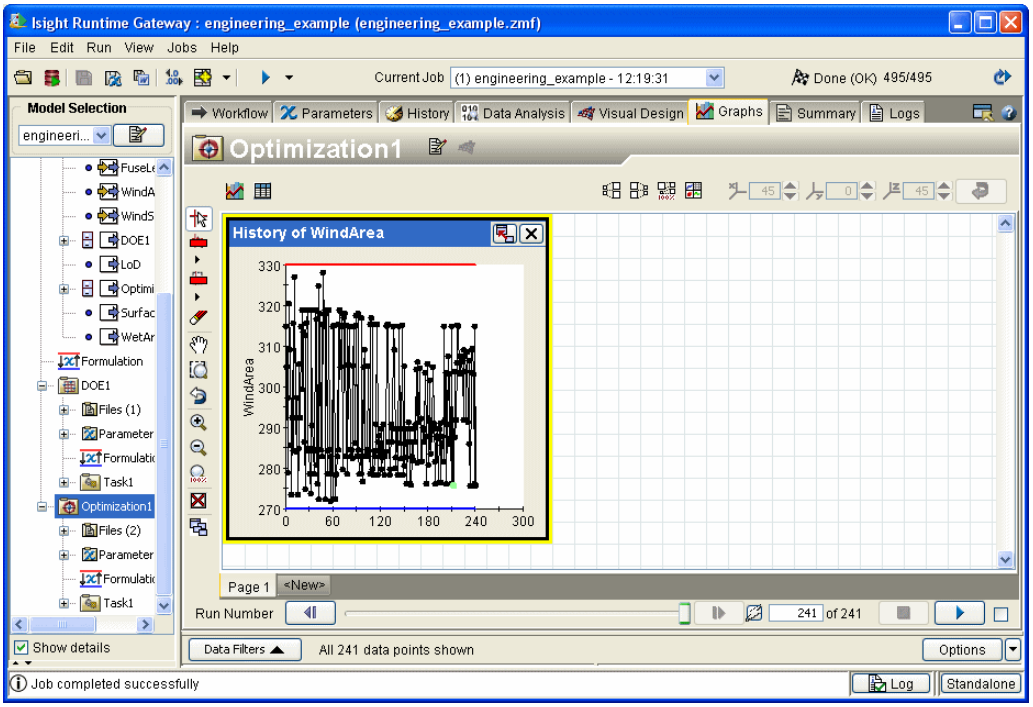
8. Click the **History Graph** option on the left side of the wizard; then, click **Next**. The Parameters screen appears.



9. Click a parameter that will be used to create the graph.

   Note: You can select multiple parameters. However, each parameter selected represents a graph that is created.

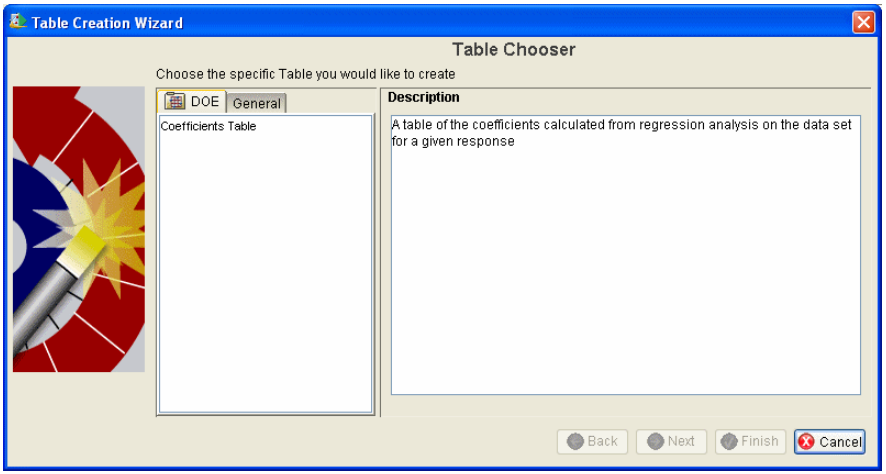
10. Click **Finish**. The graph is created.



For more information on manipulating graphs after they are created, refer to the *Isight Runtime Gateway Guide*.

11. Click the **DOE1** component on the left side of the Runtime Gateway; then, click the  button.

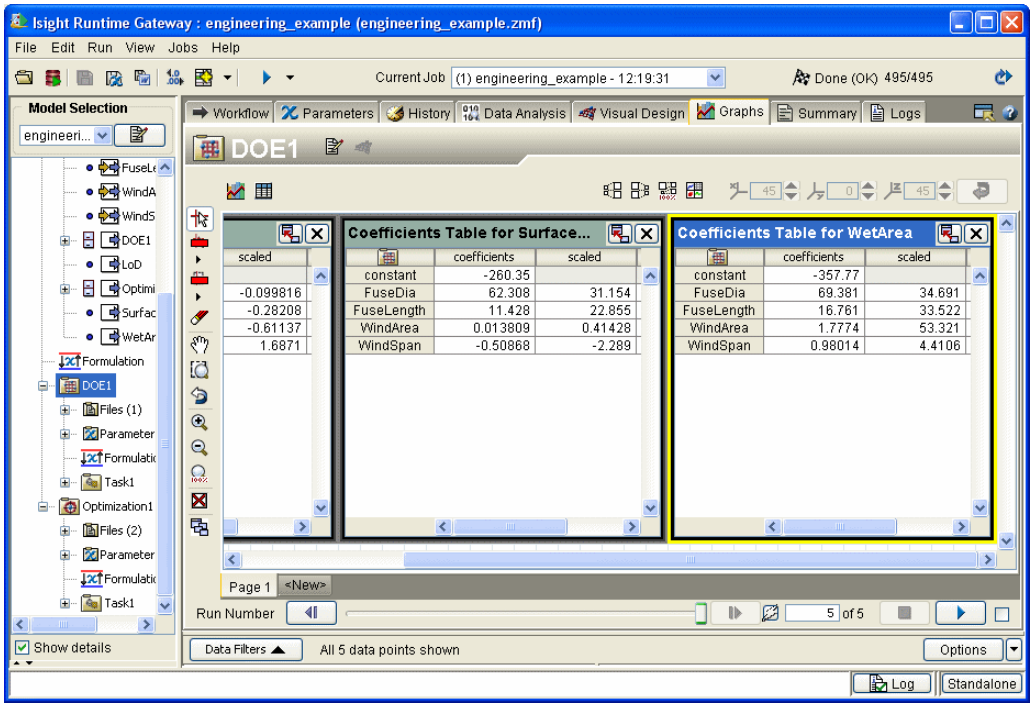
The Table Creation Wizard dialog box appears.



Notice that you can create DOE-specific tables and other tables (tables that are available with every component).

12. Click the **Coefficients Table** option; then, click **Next**.
13. Select all of the available parameters; then, click **Finish**.

Three tables are created and added to the Runtime Gateway.



All of the tables are visible if you use the scroll bars on the Graphs tab. You can also maximize the Runtime Gateway to view all of the tables at the same time.

For more information on manipulating graphs after they are created, refer to the *Isight Runtime Gateway Guide*.

Note: To view the graph you created earlier for the Optimization component, simply click the Optimization component on the left side of the Runtime Gateway. The graph reappears.

14. (optional) Create additional graphs and tables for the components. Some recommendations include Main Effects and Pareto graphs for the DOE component.

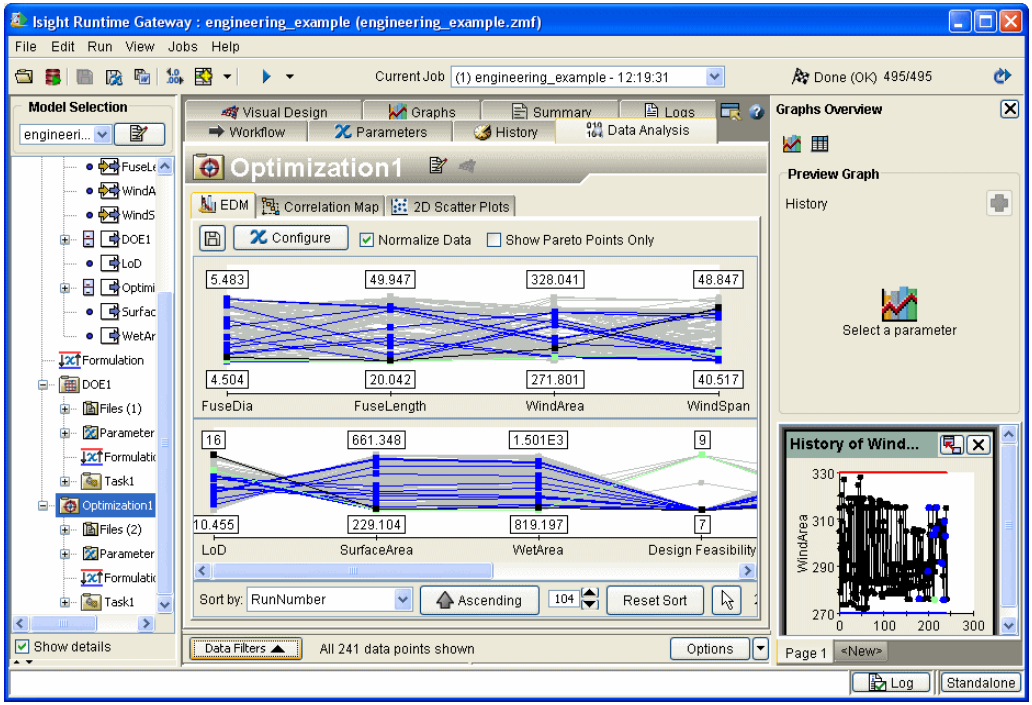
15. Proceed to "Analyzing the Results," on page 133.

Analyzing the Results

The Isight Engineering Data Mining (EDM) tool provides an effective way to view many designs (inputs and corresponding outputs) and view the relationships and trends among those designs. More specifically, the tool provides post-processing functionality for multi-objective optimization by analyzing the results and effectively visualizing Pareto solutions and sorting (screening) the data interactively. The multi-objective techniques provided in Isight are AMGA, NCGA, and NSGA-II. For more information on this technique, refer to the *Isight Component Guide*.

To analyze the results using Engineering Data Mining:

1. Click the **Optimization1** component on the left side of the Runtime Gateway; then, click the **Data Analysis** tab. The contents of the tab appear with the selected component's name displayed at the top of the tab.



This tab is divided into three subtabs, which provide access to three different data analysis tools.

2. Verify that the **EDM** subtab is selected. It should be selected by default. The EDM information is displayed on the subtab. Information for all of the parameters in the selected component is displayed.

3. Proceed to one of the following sections, based on how you would like to use the EDM tool:

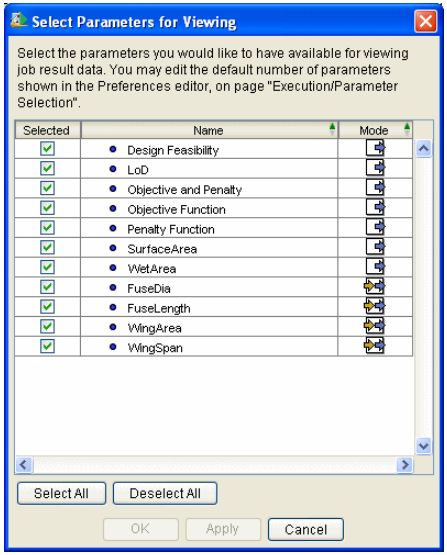
   - To change the parameters that are being used by the EDM tool, proceed to "Changing the Selected Parameters" on this page.
   - To view information for all of the parameters in your component, proceed to "Viewing the Data," on page 135.

Changing the Selected Parameters

By default, all of the parameters in the selected component are used to create the information displayed in the EDM tool. However, you can change which parameters are used by the tool.

To change the parameters used by the EDM tool:

1. Click the **Configure** button near the top of the **EDM** subtab. The Select Parameters for Viewing dialog box appears.



2. Click the parameters that you want to use in the graphs. You can select parameters individually or you can select every listed parameter using the **Select All** button. The Mode column shows you the parameter type (input, output, etc.).
3. Click **OK**. The EDM graphs are updated accordingly.
4. Proceed to "Viewing the Data" on this page for more information on how to read the data displayed on the lower portion of the EDM subtab.

Viewing the Data

The two parallel coordinate graphs on the Engineering Data Mining Tool dialog box display your input and output parameter information for each run of the selected component.

The graph consists of all inputs and outputs (depending on the graph you are viewing) displayed across the x-axis, with each value for each variable displayed in increasing order above that variable. Variable values for each input or output are linked by a sequential line graph to represent each design point. By default, each parameter is displayed on an individual scale, taking the full range of the graph (normalized).

Several options appear at the bottom of the dialog box and allow you to manipulate the order of the displayed parameters.

1. As desired, examine the data displayed on the **EDM** subtab. For more information on the various options available with this tool, refer to the *Isight Runtime Gateway Guide*.
2. Select **Close Window** from the File menu to close the Runtime Gateway. You are returned to the Design Gateway.
3. Proceed to "Approximating the Analysis," on page 136.

Approximating the Analysis

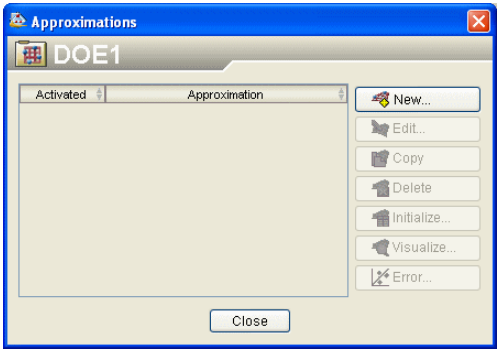
In cases where your analysis task takes a significant amount of time to run, you may want to define an approximation. This approximation will represent the analysis so that the design drivers can quickly evaluate a given design.

Note: The following procedure describes how to create approximations from the Design Gateway. You can also create approximations using the Runtime Gateway. For more information, refer to the *Isight Runtime Gateway Guide*.

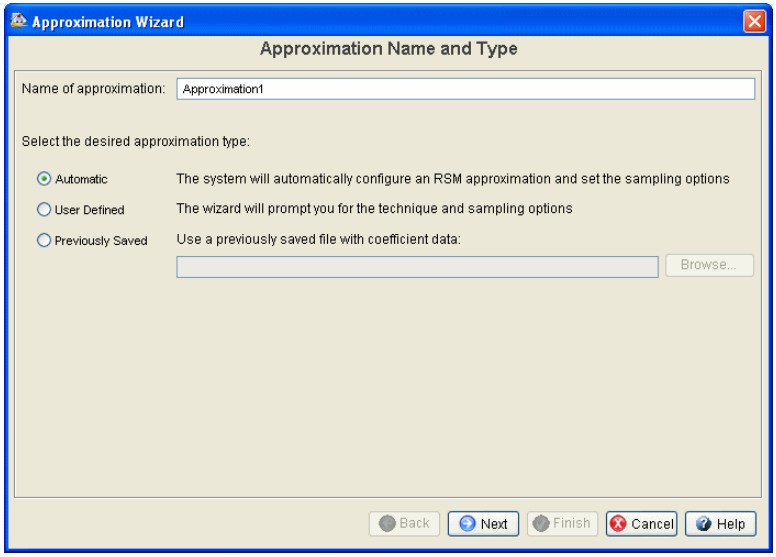
Creating an Approximation

To create an approximation:

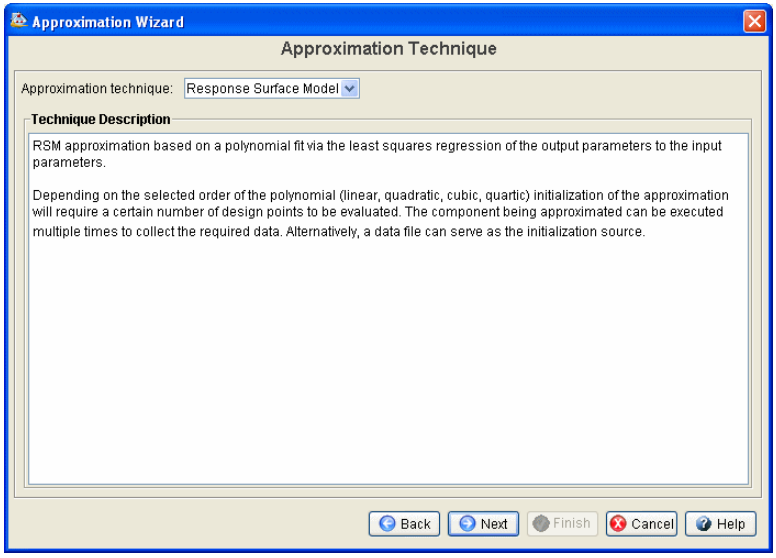
1. Click the **DOE1** component on the left side of the Design Gateway (in the Model Explorer).
2. Right-click the highlighted **DOE1** component; then, select **Approximations** from the menu that appears. You can also click the  button on the Component Title Bar. The Approximations dialog box appears.



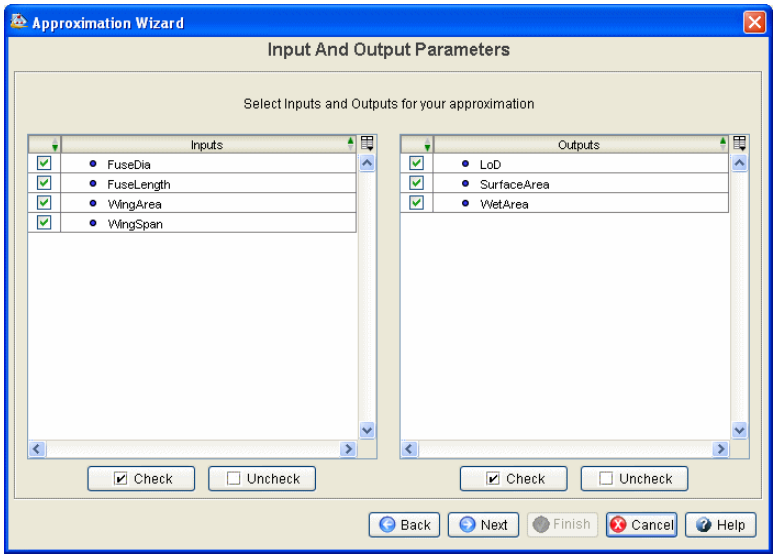
3. Click the **New...** button. The Approximation Wizard appears.



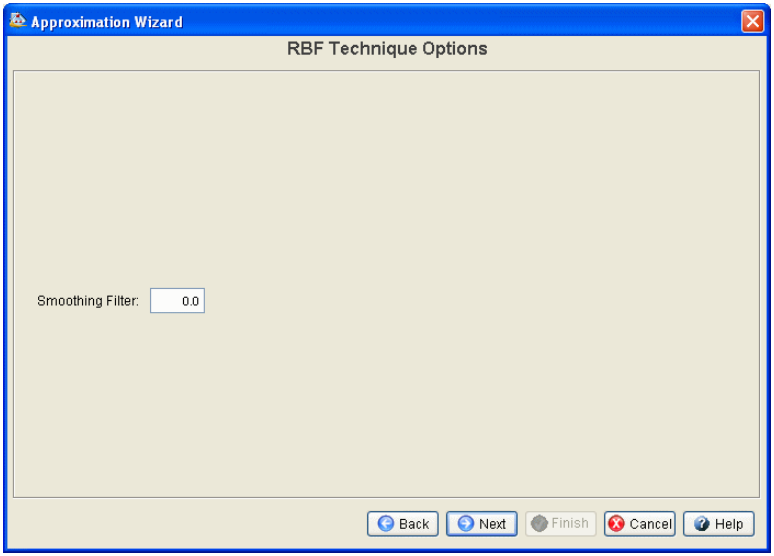
4. Click the **User Defined** radio button; then, click **Next**. The Approximation Technique screen appears.



5. Select **RBF Model** from the **Approximation technique** drop-down list; then, click **Next**. The Input and Output Parameters screen appears.



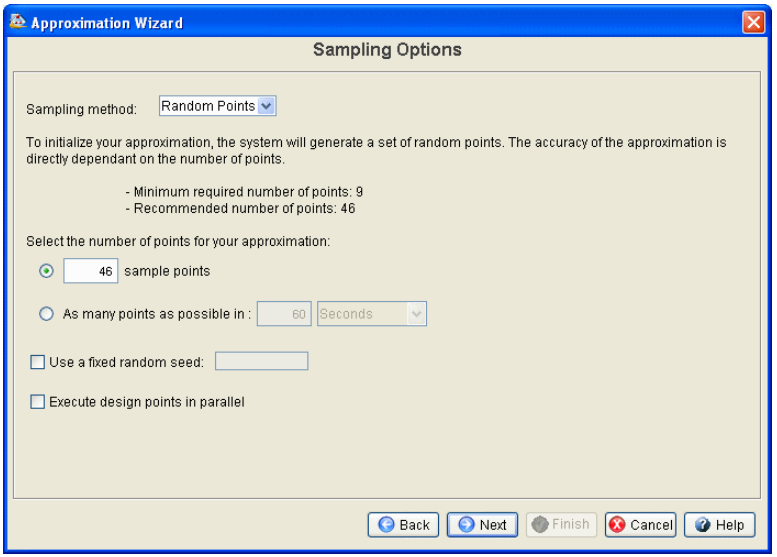
6. Verify that all inputs and outputs are selected; then, click **Next**. The RBF Technique Options screen appears.



7. Set the **Smoothing Filter** option. You can leave the value as 0.0. This option allows you to relax the requirement that the RBF approximation pass through every single data point. Its primary purpose is to smooth out noisy data. By not

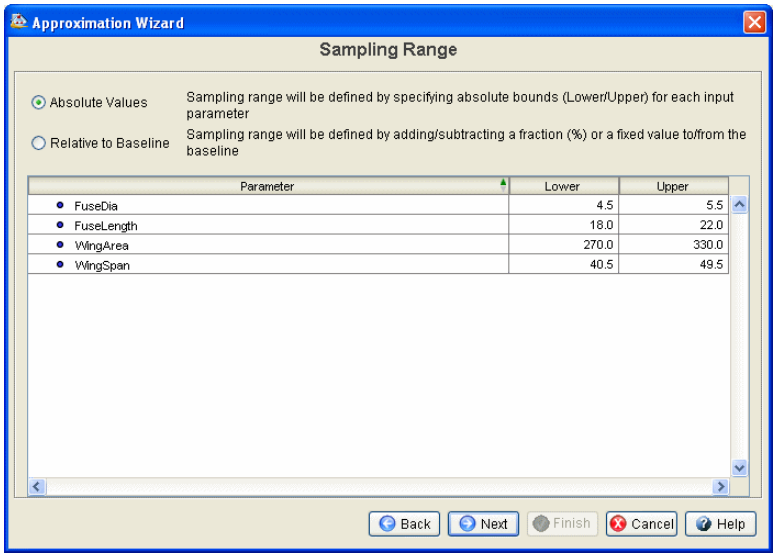
going through every point, Isight can effectively smooth noisy functions and provide an approximation that may be easier to optimize. The value specified by this option averages the output values of points that are clustered in the normalized filter domain. For more information on the Smoothing Filter option, refer to the *Isight User's Guide*.

8. Click **Next**. The Sampling Options screen appears.



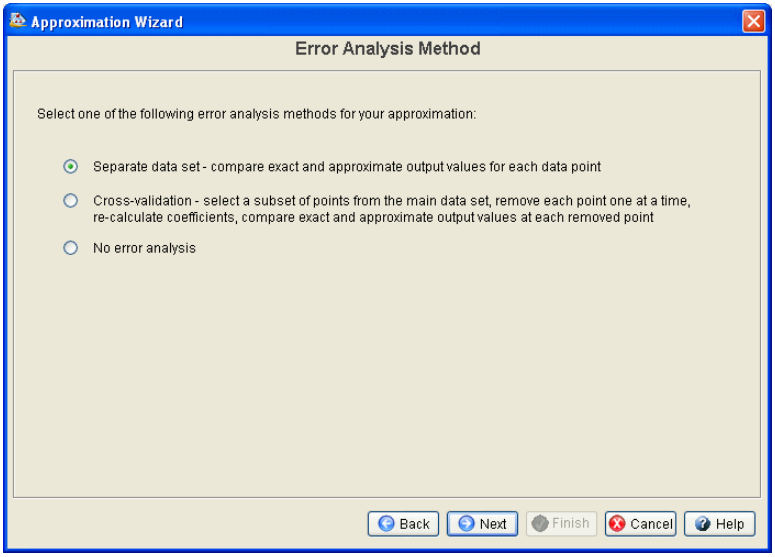
For this example, the default options are sufficient.

9. Click **Next**. The Sampling Range screen appears.



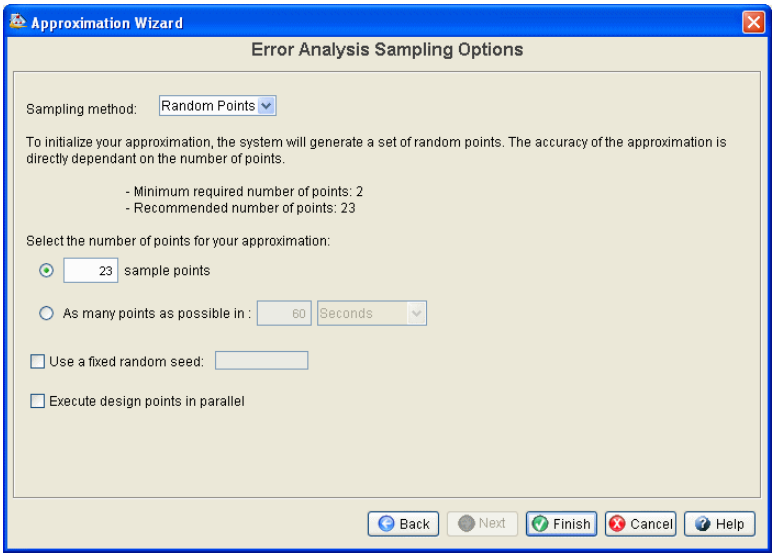
For this example, the default options are sufficient.

10. Click **Next**. The Error Analysis Method screen appears.



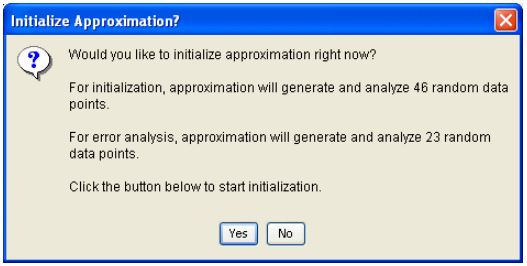
For this example, the default options are sufficient.

11. Click **Next**. The Error Analysis Sampling Options screen appears.

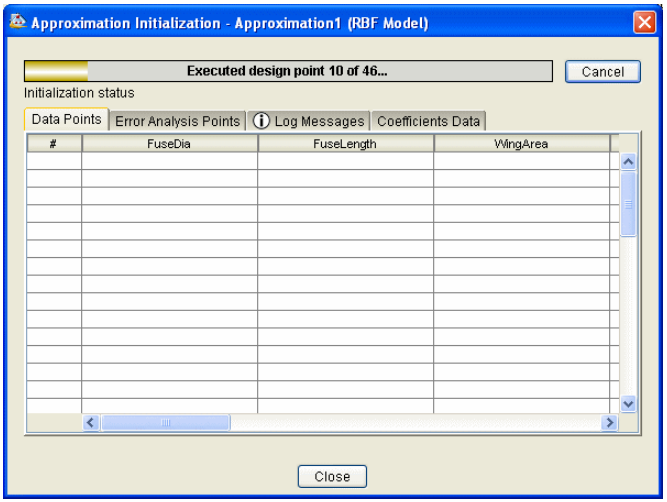


For this example, the default options are sufficient.

12. Click **Finish**. The Initialize Approximation? dialog box appears, asking if you want to initialize the approximation now or wait until a later time.



13. Click **Yes**. The Initialization Status screen appears.



This screen not only displays the status of the initialization (using the bar at the top of the screen), but it also provides access to data points, error analysis points, log messages, coefficients data, and other options once the initialization is complete.

14. Perform any of the following actions, as desired, once the initialization completes:

- View the information on the **Coefficients Data** tab. This tab is initially displayed by default. This tab shows the resulting coefficients calculated from generating the approximation.
- Click the **Data Points** tab to view the values for input and output parameters for each sample point.
- Click the **Error Analysis Points** tab to view the values for input and output parameters for each error analysis point.
- Click the **Log Messages** tab to view all log messages associated with the initialization. You can filter the log messages using the Log Filter drop-down list. For more information on the different log levels, refer to the *Isight User's Guide*.

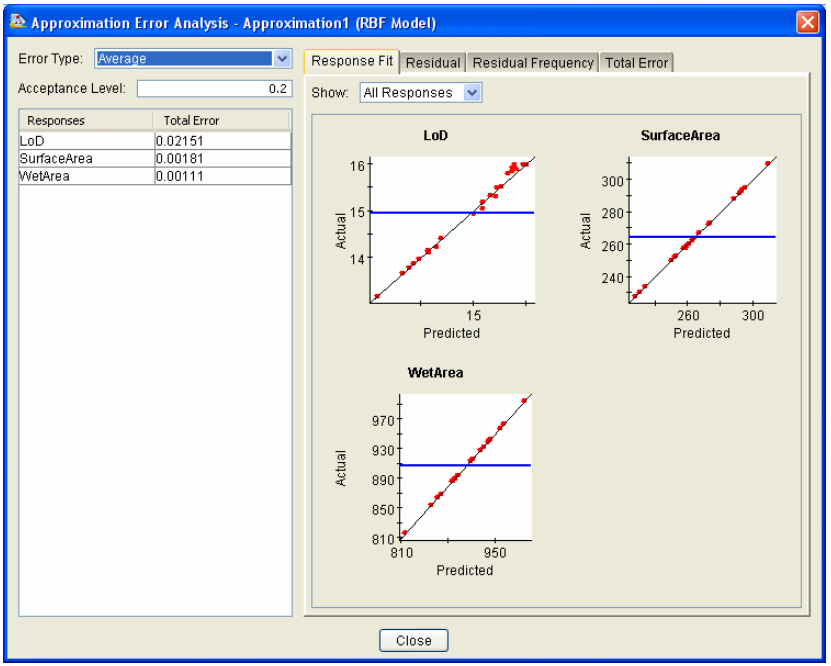
15. Proceed to "Analyzing Approximation Errors," on page 143.

Analyzing Approximation Errors

The Isight approximation error analysis dialog provides a visual representation of the quality of an approximation model for each response. This section provides a brief overview of the tool. For more information on using this interface, refer to the *Isight User's Guide*.

To analyze approximations using the wizard:

1. Click the **Error Analysis...** button on the Approximation Wizard. The Approximation Error Analysis dialog box appears.



Note: You can also access this interface by clicking the button on the Component Title Bar; then, select the appropriate approximation and click the **Error Analysis...** button.

2. Select the desired error type from the **Error Type** drop-down list.

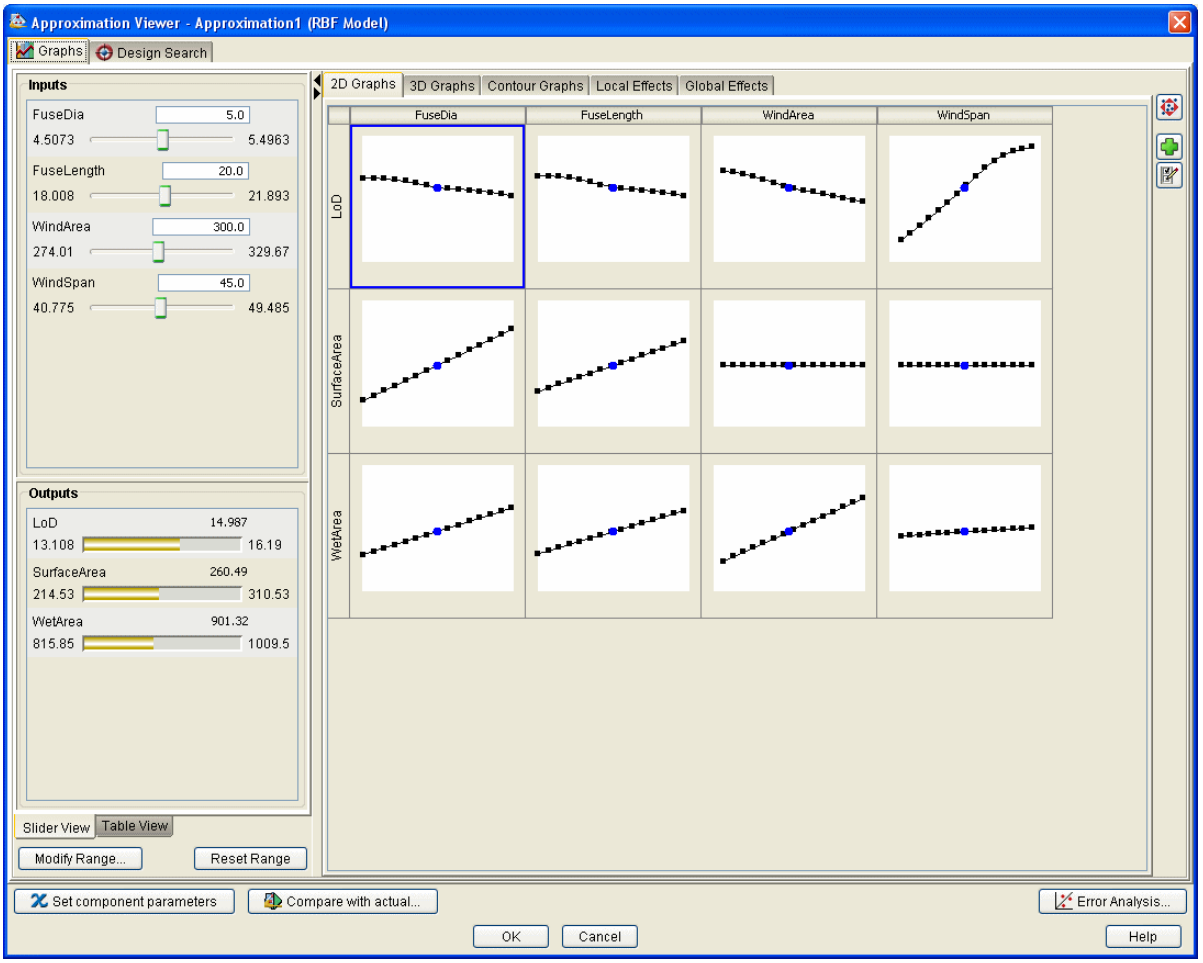
3. Specify the acceptance level for the selected Error Type in the **Acceptance Level** text box. The acceptance level defines the cutoff value for the selected error type that distinguishes a response with acceptable fit (acceptable approximation quality) versus a response with unacceptable fit (unacceptable approximation quality).
4. View the plots and graphs available from the four tabs on the right side of the interface. You can view any plot individually by double-clicking the plot. You can return to the multi-plot view by double-clicking on the single plot.
5. Click **Close** to return to the Approximation Wizard.

Visualizing an Approximation

The Approximation Viewer tool (also known as the Visual Design Driver) allows you to interact with the model and observe, graphically, how changing the input values affects the output values and the shape of the output function.

To visualize an Approximation:

1. Click the **Visualize...** button on the Approximation Wizard. The Approximation Viewer dialog box appears.



Note: You can also access this interface by clicking the button on the Component Title Bar; then select the appropriate approximation and click the **Visualize...** button.

The Approximation Viewer is divided into two tabs: Graphs and Design Search. These tabs provide access to two distinct ways of exploring your design space. The Graph tab allows you to manually view your design and change values. This tab is used in the following example. The Design Search tab allows you to automatically

search your design using certain criteria. For more information on using this tab, refer to the *Isight User's Guide*.

The Graph tab is divided into two panels: a panel on the left that presents the inputs and outputs of the approximation with their corresponding values, and a panel on the right that contains the graphs for visualizing the approximation. These panels are described in more detail below.

If the model has been initialized, the left side of the Approximation Viewer displays the following information (on the Slider View tab):

- Sliders for each model input. You can move the slider bars to change the input values.
- Value bars showing model output values. These values change according to the input values you enter.
- Table View tab. This tab allows you to view input and output information in a table.
- Modify Range button. You can change the input range (lower/upper limits) by typing a new number directly in the dialog box that appears. The number represents a percentage modification from the original range for each input.
- Reset Range. This button allows you to reset the input values to their original values.

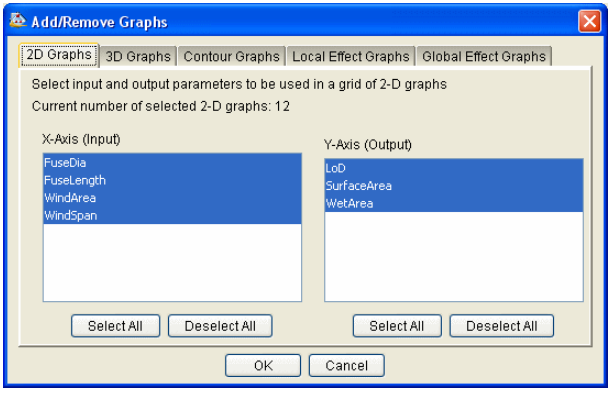
If the model has been initialized, the right side of the Approximation Viewer displays the following information:

- Tabs for viewing and selecting two-dimensional (2D), three-dimensional (3D), Contour Graphs, or Local Effects and Global Effects graphs.
- Thumbnail images of all graphs created at a given time. The selected graph has a border around it.
- When a thumbnail image of a graph is expanded to full-size mode, this side displays the graph in a larger view, and provides graph controls for easier interaction. These options change according to the type of graph selected (two-dimensional, three-dimensional, contour, or effects).
- Drop-down lists to modify the parameters being displayed in the main graph (three-dimensional and effects graphs only), when a graph is expanded to full-size mode.

For more information on using this interface, refer to the *Isight User's Guide*.

Now you'll create a new graph.

2. Click the  button on the right side of the Approximation Viewer. The Add/Remove Graphs dialog box appears.

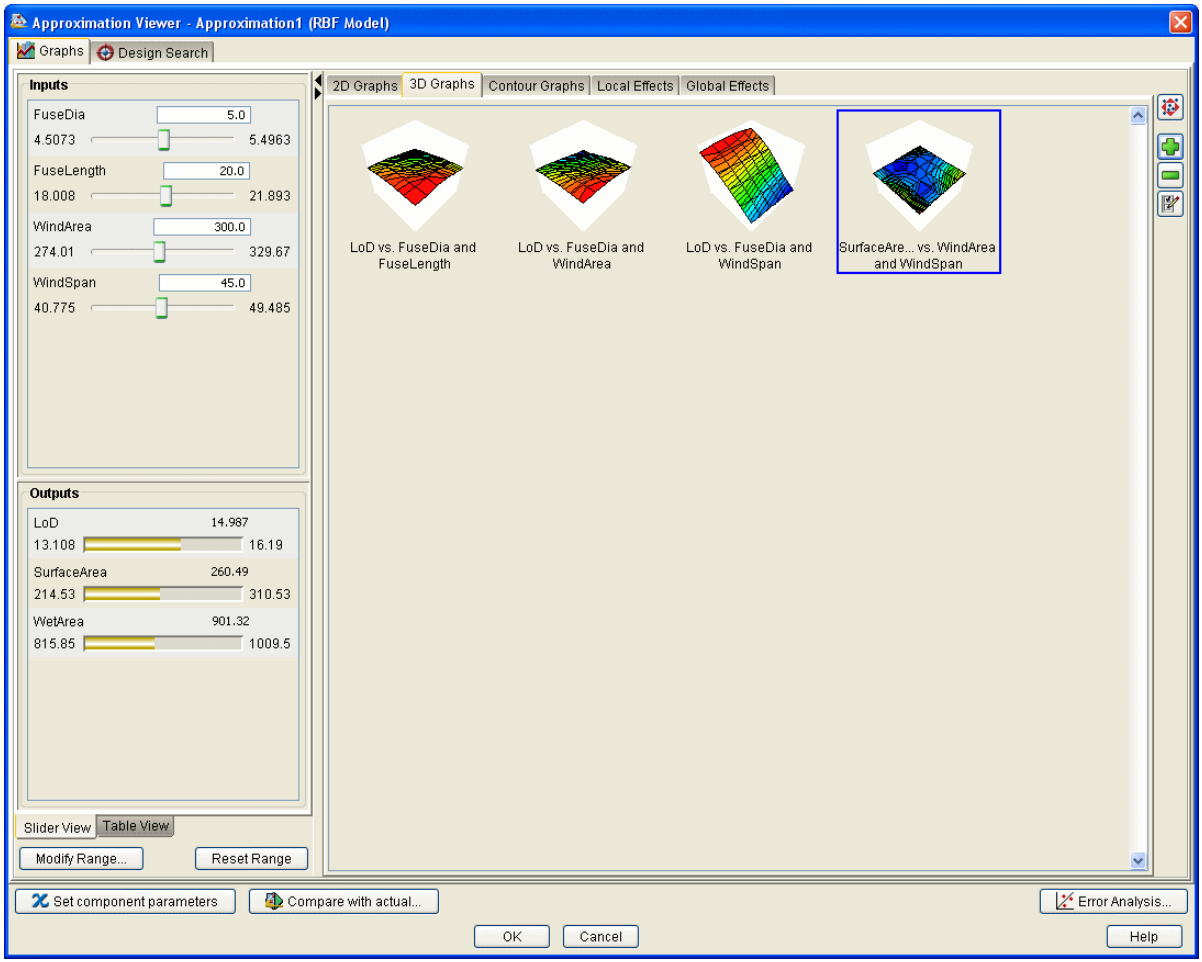


3. Click the **3D Graphs** tab; then, select the following parameters from the corresponding columns:

   - X-Axis (Input): **WingArea**
   - Y-Axis (Input): **WingSpan**
   - Z-Axis (Output): **SurfaceArea**

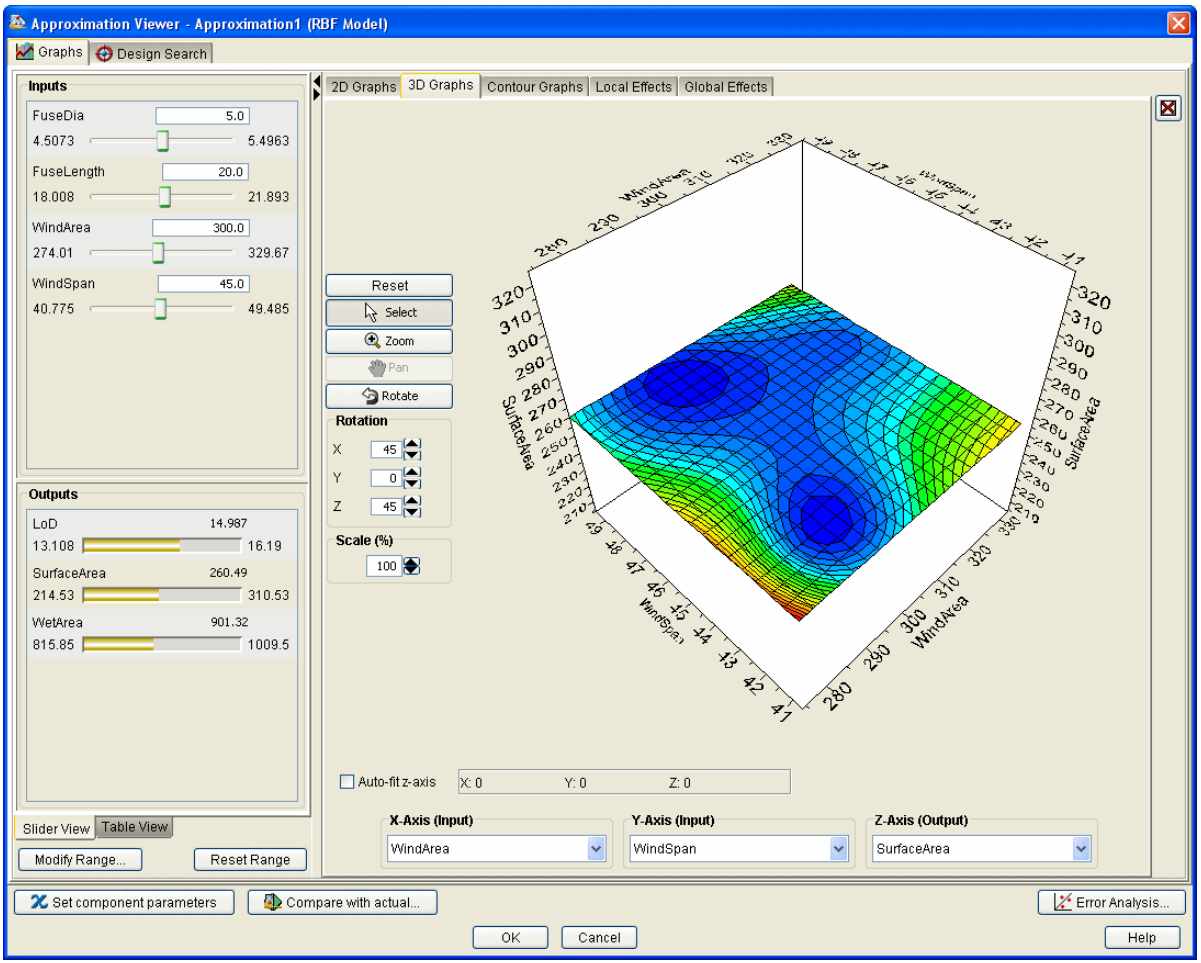
4. Click **OK**.

The new graph is created on the 3D tab of the Approximation Viewer.



5. Double-click the graph.

It is opened in full-size mode.



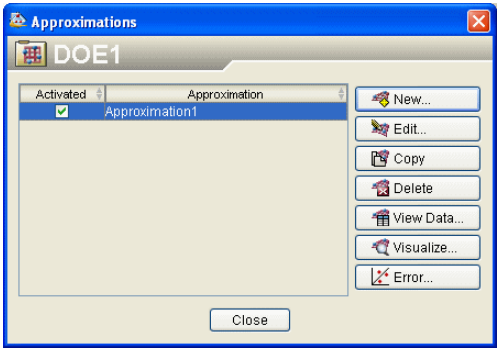
6. Click the **Auto-fit z-axis** check box to change the behavior of the graph when you scale the graph. When this option is selected, the Z-Axis range of the main graph will always match the range of the displayed data. If not selected, the graph's Z-Axis is not modified when it is scaled.

7. Manipulate the graph, as desired using the following tools:

   - Graph controls for working with the large graph to the left of the graph, including zoom, pan, and rotate features. These options change according to the type of graphs selected (three-dimensional vs. two-dimensional).
   - A set of drop-down lists to modify the parameters being displayed in the main graph.
   - Right-click options (when the large graph is clicked) that allow you to modify the appearance of the plot, including floor projections.

8. Click **OK** to close the Approximation Viewer. You are returned to the Approximation Wizard.

9. Click **Close** to exit the Approximation Initialization dialog box. You are returned to the Approximations dialog box, and the approximation you created is now listed on the left side of the dialog box.

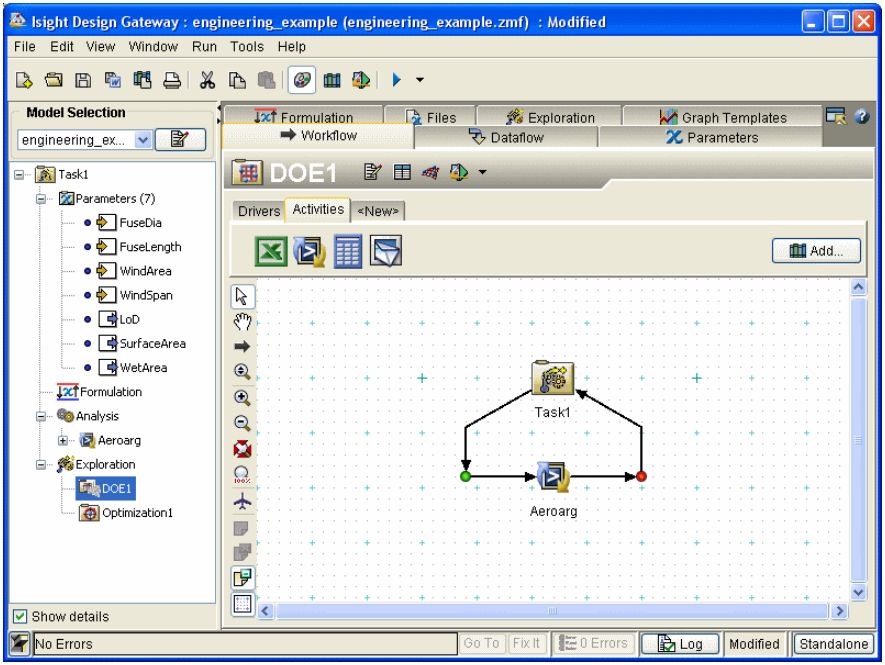


   The approximation with a corresponding check mark in the Activated column is the one that will be used during execution. Only one approximation can be active at any one time for a component.

   You can also clear all of the check boxes in the Activated column to deactivate all of the defined approximations. This option allows you to execute the model without any defined approximations being used.

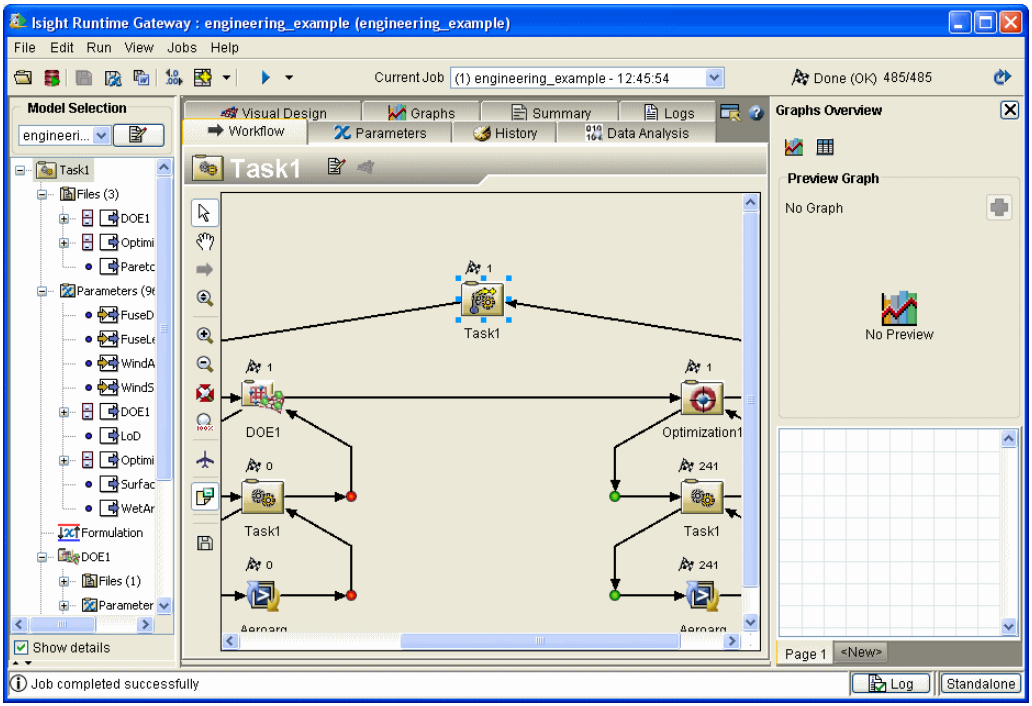
10. Clear (unselect) the check box in the **Activated** column for the approximation you created.

11. Click **Close** to return to the Design Gateway.



Notice that the approximation icon on the DOE component in the Model Explorer is now grayed-out, which is your visual cue that the approximation will not be used during execution.

12. (optional) Reactivate the approximation; then, execute the model and view the results. Pay special attention to how the DOE is executed differently thanks to the new approximation (the subflow doesn't need to execute).



13. Close the Runtime Gateway.

14. Save and close the model, if desired.

15. Close the Design Gateway. The example is complete.

Isight

A Additional Installation Information

This appendix provides additional information that may be useful when installing Isight. It is divided into the following topics:

User Login Names Containing Punctuation

Isight interfaces do not run correctly if they are started using a user login name (user ID) that contains punctuation marks - most notably !, #, ?, and : (exclamation point, pound sign, question mark, and colon). This problem is due to JAVA using URLs internally in order to locate JAR files, and these characters cause the URL to be misinterpreted. The problem is most severe on Windows-based systems, where the default temporary directory is inside a directory named after the user name.

To avoid this problem, you must force the Isight installation and all temporary files into directories that do not contain these characters. First, verify that Isight is not installed in a directory that contains any of these characters. Second, manually set your temporary directory to a directory that doesn't contain these characters. On Windows, set the TEMP environment variable. On UNIX/Linux, set the TMPDIR environment variable. For more information on setting environment variables on your system, contact your local systems administrator.

If the appropriate environment variable cannot be set, the variable FIPER_TEMP can be used.

Finally, if you cannot set either the system environment variables or the FIPER_TEMP variable, create a file called `fiper.bat` (Windows) or `.fiper.sh` (UNIX/Linux) in your home directory. Use this file to set the environment variable FIPER_TEMP to a "safe" directory name. The contents of the new file will be similar to the following examples:

- Windows (`fiper.bat`)

  ```
  set FIPER_TEMP=C:\TEMP\bang-user
  ```

- Unix/Linux (`.fiper.sh`)

  ```
  FIPER_TEMP=/var/tmp/bang-user
  export FIPER_TEMP
  ```

Accessing the Installation Files from a Remotely Mounted DVD

Once you have completed the following steps, you can use the installation procedures to install Isight. For more information, consult with your Systems Administrator.

Installing on UNIX Platforms

1. Locate the Isight DVD.
2. Copy the entire contents of the DVD to the desired location.

   The UNIX utilities cp -r and rcp are convenient for copying these items. When copying files between different platforms (particularly between UNIX and Windows), special care may be needed to preserve the original file permissions and the file name case. In general, read and execute permissions on all files are sufficient for proper execution.
3. Depending on your platform, execute prod_dir/setupaix, prod_dir/setupsolarix, prod_dir/setuphpux, or prod_dir/setuplinux to launch the installation procedure, where prod_dir is the location of the copied installation files.

Installing on Windows Platforms

You can install Isight from a remote DVD device if it has been defined as a shared folder. Setting up the DVD device as a share must be performed by a user with Administrator privileges. To install Isight from a shared remote DVD, map that drive to a local drive and perform the installation as if the drive were local. Universal Naming Convention (UNC) paths are not supported by the installation procedures.

Accessing the Documentation

The Isight installation automatically installs all of the Isight manuals onto your system. These manuals are in PDF format. This documentation consists of all available Isight books, including this book.

These files are located in the following directory:

<*Isight_install_directory*>\docs

You can also access the documentation directly from the Isight DVD (prior to an installation).

In order to read the PDF files, you must have the Adobe Acrobat Reader installed on your system. For more information on obtaining this reader, see Adobe's website:

http://www.adobe.com/products/acrobat/readstep2.html

For more information on using the PDF files provided with Isight, refer to the *Isight User's Guide*.

Necessary Changes for Executing on UNIX Systems

The following section describes steps that must be taken on certain UNIX operating systems prior to using Isight. Proceed to one of the following topics, based on your operating system:

Necessary Changes for AIX

If you are installing Isight on an AIX-based system, you need to verify that you have the FORTRAN runtime file (`libxlf90.a`) installed in the /usr/lpp/xlf directory. This file is an optional add-on to the AIX operating system and may need to be manually installed.

Installations of Isight on the AIX platform require the following libraries:

- xlfrte 8.1.0.0 XL Fortran Runtime Environment
- xlfrte.aix50 8.1.0.0 XL Fortran Runtime Environment AIX 5.0 Libraries

For more information on determining if you have these libraries installed, or instructions on how to install the libraries, refer to your local systems administrator.

Necessary Changes for HP-UX

The following sections describe HP-UX-specific changes that are necessary to ensure that Isight functions properly.

Operating System Requirements

In order to ensure that execution on the HP-UX operating system is successful, you need to take the following steps:

- Increase the number of threads per process from the default setting of 64 to at least 256. Failure to do so will cause non-reproducible errors during execution.

  For more information, contact your local systems administrator or refer to your HP-UX administration documentation concerning how to increase the kernel parameter `max_thread_proc` from the default of 64 to at least 256.

- Tune the kernel according to the recommendations of the HPjune configuration tool. You can download and run the HPjconfig tool from the Hewlett-Packard website:

  http:/www.hp.com/products1/unix/java/java2/hpjconfig/index.html

For more information, contact your local systems administrator.

Java Runtime Environment Requirements

On HP-UX 11.11 PA RISC, use the patches for Java 5.0 as described on the following website:

http://docs.hp.com/en/HPUXJAVAPATCHES/index.html

Warning: Failing to install all required patches to HP-UX will result in Java program problems. Typically, the only way to reset the machine after such a problem is to restart it. This issue presents a serious problem that will affect all users of the machine, not just those running Isight. Furthermore, this problem seems to be present in all versions of HP-UX as initially installed. The patches listed on the web site must be applied *before* Isight are installed.

HP-UX users should download the necessary tool (from http://www.hp.com/go/java), set for Java 5.0, change the kernel parameters as recommended, and install the patches as recommended prior to running Isight.

Necessary Changes for Solaris

If you are installing Isight on a Solaris-based system, you must verify that the system has swap space equal to three or four times the main memory. For more information on determining your system's current specification, contact your local systems administrator.

The large amount of swap space is needed because Isight can grow to over 500 MB. Additional swap space equal to the process size is allocated for a short time every time the program creates a sub-process to execute an external program. This space is never used but must be available. Insufficient swap space will cause Isight jobs to fail sporadically with messages about “insufficient disk space”. Other versions of UNIX/Linux use a different scheme for allocating swap space to processes and can run Isight with swap space equal to twice main memory.

Environment Variables

Several environment variables are set automatically by the Isight installer. They are mainly used for those creating custom Isight add-ons and do *not* need to be set for normal operation. The following environment variables are configured:

- FIPER_HOME_35=<*Isight_install_directory*>
- FIPER_HOME=<*Isight_install_directory*>
- PATH (UNIX only)
  - UNIX: PATH=$FIPER_HOME/bin:$PATH

Note: If the installer is run by a user in the Admin group (on Windows) or with root access (on UNIX), then the environment variables are set for all users (by editing /etc/profile on UNIX). If the user is not in the Admin group or does not have root access, the environment variables are set only for that user (by editing $HOME/.profile on UNIX).

Using the Online Help

The Isight online help is a HTML-based system that runs inside of your default web browser. You can set your default web browser from the Isight Preferences dialog box. For more information on Isight preferences, refer to the *Isight User's Guide*.

In order to view the online help, you must have one of the following browser and operating system combinations installed:

- FireFox 2.2 or higher (all platforms where FireFox is available)
- Internet Explorer 7.0 (Windows) or higher
- Mozilla 2.0 or higher (Solaris and AIX)
- Mozilla 1.2.1 or higher (Linux)

FireFox is available for free download from the following website:

http://www.mozilla.org

If you do not have a supported web browser installed on your system (or on your network), the Isight online help may not function correctly. Browsers not listed above have not been fully tested with the online help. For more information on whether or not your system or network is configured in an acceptable manner, contact your local systems administrator. For more information on using the online help, refer to the *Isight User's Guide*.

Installing Isight Non-Interactively

This section describes how to install Isight with minimal interaction, instead of specifying the numerous options available using the installation script. There are two ways to install Isight non-interactively:

- "Installing Using the Default Setting" on this page. This type of installation involves issuing a single command, and installing Isight using all of the default settings.
- "Installing With Pre-determined User Responses," on page 162. This type of installation allows you to create a file that stores your installation settings. Once created, this file can be used to automatically configure any additional installations.

Installing Using the Default Setting

To install Isight with the default user responses:

1. Open a Command Prompt dialog box (Windows) or a terminal window (UNIX).
2. Enter one of the following commands, based on your operating system:
   - **Windows**: D:\setupwin32.exe -silent (where D: is your DVD drive letter)
   - **UNIX/Linux**: /*dvd_mount_point*/setup*platform*.bin -silent (where dvd_mount_point is specific to your system, and platform specifies the type of UNIX operating system you are using)

   The software is installed.

   Note: A default install does not install a license file. You will need to copy a *license.dat* file from the *license* directory of an existing install to the *license* directory of the new install.

Installing With Pre-determined User Responses

To install Isight with pre-determined user responses:

1. Open a Command Prompt dialog box (Windows) or a terminal window (UNIX).
2. Enter one of the following commands, based on your operating system:
   - **Windows**: `D:\setupwin32.exe -options-record ABS_PATHNAME_TO\RESPONSE_FILE_TO_AUTOGENERATE`

     Note: In this command, D: is your DVD-ROM drive letter. Also, the command should be entered on a single line at the command prompt.
   - **UNIX/Linux**: `/`*`dvd_mount_point`*`/setup`*`platform`*`.bin -options-record /ABS_PATHNAME_TO/RESPONSE_FILE_TO_AUTOGENERATE`

     Note: In this command, *dvd_mount_point* is specific to your system, and *platform* specifies the type of UNIX operating system you are using. This command should be entered on a single line at the terminal prompt.
3. When the installation interface appears, perform the installation procedures as normal. Once completed, the RESPONSE_FILE_TO_AUTOGENERATE is generated automatically.
4. Enter one of the following commands, based on your operating system:
   - **Windows**: `D:\setupwin32.exe -silent -options ABS_PATHNAME_TO\RESPONSE_FILE_TO_AUTOGENERATE`

     Note: In this command, D: is your DVD-ROM drive letter. Also, the command should be entered on a single line at the command prompt.
   - **UNIX/Linux**: `/`*`dvd_mount_point`*`/setup`*`platform`*`.bin -silent -options /ABS_PATHNAME_TO/RESPONSE_FILE_TO_AUTOGENERATE`

     Note: In this command, *dvd_mount_point* is specific to your system, and *platform* specifies the type of UNIX operating system you are using. This command should be entered on a single line at the terminal prompt.

Note: An Absolute path must be used because the installation process performs a “cd” to an unwritable directory.

Uninstalling Isight

This section discusses how to *permanently* remove Isight 3.5 from your system. It is divided into the following topics:

Uninstalling from Windows Platforms

You can permanently remove Isight and the license service at any time, as conditions warrant. This process involves stopping the license server, removing the Isight software, and deleting any temporary directories and files that are left behind.

Typically, the uninstallation process removes the license server on Windows and UNIX/Linux platforms. If you have multiple servers running on an individual machine, you may need to manually uninstall the license server as described in this section.

Important: If you have two versions of Isight 3.5 installed (for example, the initial release and a patch release that followed), you should directly access the uninstaller of the version you want to remove. It is recommended that you *not* use the Windows Control Panel as described in the following procedure. The uninstaller is located in the _uninst subdirectory of the main Isight directory for the version you wish to remove. Simply execute the `uninstall.exe` file.

Note: If you have multiple versions of Isight 3.5 installed on an individual machine using local license files, uninstalling one version of Isight 3.5 may delete the license server used by the other installations. You will need to re-install the license server manually as described in "Installing Only a License Server," on page 179.

Stopping the License Server and Removing the Software

To remove Isight and its license service on Windows:

1. Login as Administrator or a user with administrative privileges.
2. Perform one of the following options, based on your operating system:
   - **Windows XP**: Click **Start**; then, point to **All Programs** / **Isight 3.5-1** and click **FLEXnet Utilities**.
   - **Windows Vista**: Click **Start**, point to **All Programs** / **Isight 3.5-1** and right-click **FLEXnet Utilities**; then, select **Run as administrator** from the menu that appears.

   The LMTOOLs dialog box appears.
3. Click the **Start/Stop/Reread** tab. The contents of the tab appear.
4. Select the Isight license service; then, click the **Stop Server** button. The license server is stopped.
5. Click the **Config Services** tab. The contents of the tab appear.
6. Verify that the Isight service is select from the **Service Name** drop-down list; then, click the **Remove Service** button.
7. Click **Yes** to verify the removal of the service; then, close the LMTools dialog box. Now you need to remove Isight itself.
8. Click **Start / Control Panel** to open the Control Panel dialog box.
9. Perform one of the following options, based on your operating system:
   - **Windows XP**: Click the **Add or Remove Programs** link. The Add or Remove Programs dialog box appears.
   - **Windows Vista**: Click the **Programs** link; then, click the **Programs and Features** link. The list of installed programs appears.
10. Select **Isight 3.5-1** from the list of programs; then, click the **Change/Remove** button (**Uninstall/Change** button on Windows Vista).
11. (*Windows Vista only*) Click the **Continue** button to verify that you want to execute the uninstall program.

The Welcome dialog box appears.

12. Click **Next**. A summary of the uninstallation appears.
13. Click **Next**. The software is removed and a summary of the installation appears.
14. Click **Finish**.

    Note: If a message appears informing you that Isight has been removed and that you must restart your system to complete the uninstallation process, be sure to perform the restart before continuing to the next section.

15. (*Windows XP only*) Close the Add or Remove dialog box
16. Close the Control Panel.
17. Proceed to the next section.

Deleting Temporary Directories and Files

Once you've removed the Isight software and license server, you need to remove the temporary directories and files created by Isight.

To delete temporary directories and files:

1. Navigate to the directory that contains the top level of your Isight installation directory. For example, if you installed Isight in C:\SIMULIA\Isight\3.5-1, navigate to the C:\SIMULIA directory.
2. Delete the Isight directory and all of its contents.
3. Navigate to the following directory, where <*user_name*> is the name of the user who installed and uninstalled Isight:
   - **Windows XP**: C:\Documents and Settings\<*user_name*>
   - **Windows Vista**: C:\Users\<*user_name*>
4. Delete the following items, if they are present:
   - `fiper.preferences` file
   - "fiperdb-35" directory
   - "fiper" directory
   - `Isightinstall.log` file
   - "locallib_3.5" directory

5. Navigate to one of the following directories, based on your operating system, where <*user_name*> is the name of the user who installed and uninstalled Isight:
   - **Windows XP**: C:\Documents and Settings\<*user_name*>\Local Settings\Temp
   - **Windows Vista**: C:\Users\<*user_name*>\AppData\Local\Temp
6. Delete the "fiper" and "fiper0.mmcache" directories.

   The Isight removal process is complete.

Uninstalling from UNIX/Linux Platforms

You can permanently remove Isight and the license service at any time, as conditions warrant. This process involves stopping the license server, removing the Isight software, and deleting any temporary directories and files that are left behind.

Stopping and Removing the License Server

To stop and remove the license server:

1. Perform one of the following actions:
   - Log in as root (obtain root permissions).
   - Verify that you have permission to write in the /etc directory.
2. Change into the following directory:

   <*Isight_install_directory*>/license
3. Execute the following command to stop the license manager:

   ```
   ./flexlm stop
   ```

   A message appears stating that your license manager was shut down successfully.

4. Execute the following command:

   ```
   ./flexlm uninstall
   ```

   A message appears, informing you that the FLEXnet startup script has been removed. The process is complete when you are returned to the prompt.

5. Proceed to the next section.

Removing the Isight Software

To remove the Isight software:

1. Verify that you are logged in as the same user that installed Isight.
2. Navigate to the following directory:

   <*Isight_install_directory*>/_uninst

3. Execute one of the following commands, based on your operating system:

   - `./uninstallaix`
   - `./uninstallhpux`
   - `./uninstalllinux`
   - `./uninstallsolaris`

   The Welcome dialog box appears.

   Important: You must be logged in as the same user that installed Isight or the uninstaller will not be able to completely remove the software.

4. Click **Next**. A summary of the uninstallation appears.
5. Click **Next**. The software is removed. You are informed when the removal is complete.

   Note: A known issue exists on HP-UX that causes an error when the uninstaller attempts to remove some files in the <*Isight_install_directory*>/bin/HP-UX_B.11.00_PA_RISC/jre directory. You will manually delete these files shortly.

6. Click **Next**. A message appears telling you that you have to log out and then log back into your system to update your environment variables.
7. Click **Finish**. The uninstaller is closed.

8. Log out of your system; then, log back into your system.
9. Proceed to the next section.

Deleting Temporary Directories and Files

To delete the temporary directories and files left behind by Isight:

1. Log in as root (obtain root permissions). Although it is not necessary to be root to delete all of the temporary files, it is necessary for some of the files.
2. Navigate to the directory that contains the top level of your Isight installation directory. For example, if you installed Isight in /opt/SIMULIA/Isight/3.5-1, navigate to the /opt/SIMULIA directory.
3. Delete the Isight directory and all of its contents.
4. Navigate to the $HOME directory for the user that installed Isight.
5. Delete the following items, if they are present:
   - `.fiper.preferences` file (notice the leading "." in the file name)
   - "locallib_3.5" directory
   - "fiperdb-35" directory
   - "fiper" directory
   - "FIPER_temp" directory
   - any Isight logs (such as the installation and uninstallation logs)
6. Navigate to the following directory:

   /var/tmp
7. Delete the `flexlm.log` file (if it is present).

   The Isight removal process is complete.

Isight

B License Information

This appendix provides license information that may be useful when installing Isight. It is divided into the following topics:

Installing the Isight License After Installation

If you receive your license file after running the Isight installation program, you will have to manually start the license manager once you receive the license. The Isight license file is generated using your computer information. This information should be given to your SIMULIA representative. SIMULIA then generates the license and e-mails it to the appointed contact at your site.

Proceed to one of the following sections, based on your operating system:

- "Installing the License on Windows" on this page
- "Installing the License on UNIX/Linux," on page 175

Installing the License on Windows

The following are the steps you should follow to correctly install your Isight license. The steps are separated into the following groups:

- "Editing Your Isight License" on this page
- "Configuring the FLEXnet License Manager Software," on page 171

Editing Your Isight License

To edit your license file:

1. Login as Administrator or a user with administrative privileges.
2. Save the `license.dat` file you received via e-mail into the following directory:

   <*Isight_install_directory*>\license

   This overwrites the `license.dat` file that came with Isight, which simply contains a message about setting up your actual license.

   Note: If the machine has old mail software, then you must remove everything before the line beginning with "SERVER".

Configuring the FLEXnet License Manager Software

If a local FLEXnet daemon is required, and FLEXnet is not already configured on your system or network, you must modify the FLEXnet Utilities interface.

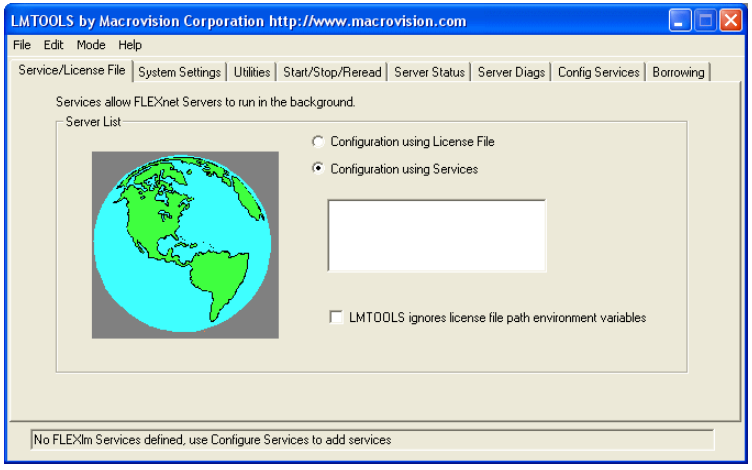
Note: The following procedure assumes you have installed Isight into the default location (C:\SIMULIA\Isight\3.5-1). If you installed the software in a different location, your procedure will differ slightly from the one that follows.

To modify the FLEXnet Utilities interface:

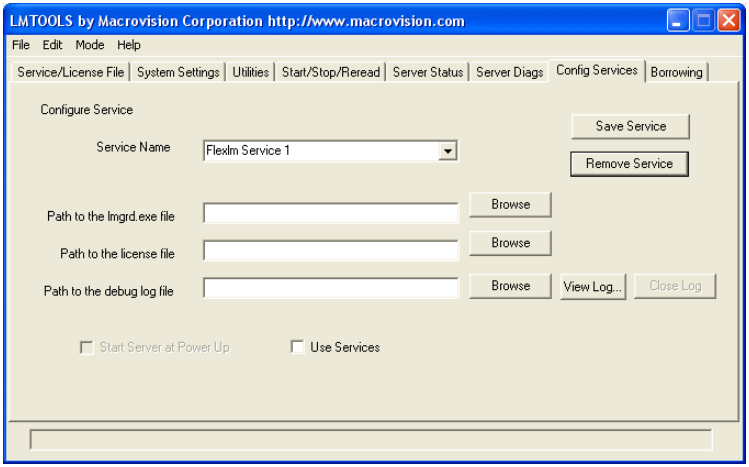
1. Login as Administrator or a user with administrative privileges.
2. Perform one of the following options, based on your operating system:
   - **Windows XP**: Click **Start**; then, point to **All Programs** / **Isight 3.5-1** and click **FLEXnet Utilities**.
   - **Windows Vista**: Click **Start**, point to **All Programs** / **Isight 3.5-1** and right-click **FLEXnet Utilities**; then, select **Run as administrator** from the menu that appears.

   The LMTOOLS dialog box appears with the Service/License File tab selected.

   Note: You can also execute the `lmtools.exe` file from the command prompt to start the FLEXnet Utilities interface. This file is located in the <*Isight_install_directory*>\bin\win32 directory. Be sure to run it using the "Run as administrator" option if using Windows Vista.



3. Verify that the **Configuration using Services** radio button is selected; then, click the **Configure Services** tab. The contents of the tab appear.



4. Click the **Browse** button adjacent to the **Path to the lmgrd.exe file** text box. An Open dialog box appears.

5. Navigate to the following directory:

   <*Isight_install_directory*>\bin\win32

6. Select the **lmgrd.exe** file; then, click the **Open** button. The path appears in the corresponding text box in the Config Services tab.

7. Click the **Browse** button adjacent to the **Path to the license file** text box. An Open dialog box appears.

8. Navigate to the following directory:

   <*Isight_install_directory*>\license

9. Select **License Files(*.dat)** from the **Files of type** text box. The license file appears in the Open dialog box.

10. Select the **license.dat** file; then, click the **Open** button. The path appears in the corresponding text box in the Config Services tab.

11. Click the **Browse** button adjacent to the **Path to the debug log file** text box. An Open dialog box appears.

12. Navigate to any directory in which you wish to have the log file stored. For example:

    <Isight_install_directory>\license

13. Type the name of the log file in the **File name** text box on the Open dialog box. For example:

    `debug.log`

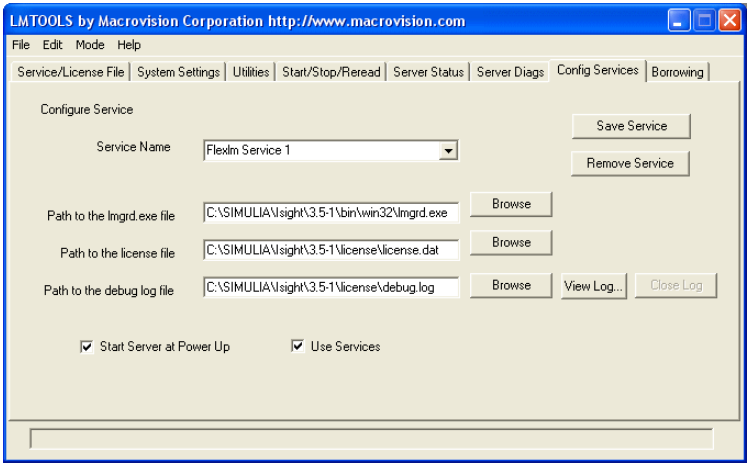
14. Click the **Open** button. The path appears in the corresponding text box in the Config Services tab.

15. Verify that the two check buttons at the bottom of the tab are selected:

    - **Start Server at Power Up**
    - **Use Services**

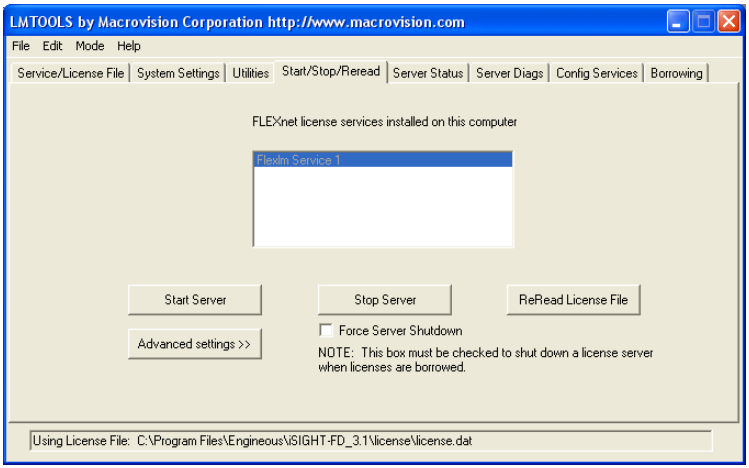
    Note: By selecting the Start Server at Power Up option, you do not have to manually start the license server. It is done automatically when your computer is turned on.

    Your dialog box should appear as shown below.



16. Click the **Save Service** button; then, click **Yes** to verify your settings.

17. Click the **Start/Stop/Reread** tab. The contents of the tab appear.



18. Click the **Start Server** button. The FLEXnet license manager is running when you see the following message at the bottom of the dialog box:

    ```
    Server Start Successful
    ```

    Note: If you have an existing Isight Service defined, and the server fails to start after updating the service, return to the Configure Service tab, delete the old service, and create a new service. This new service should start without any errors.

19. Select **Exit** from the **File** menu to close the LMTOOLS dialog box. You can now start using Isight.

    Note: It is recommended that you access the Design Gateway to test your license file. Also, you'll be prompted to publish the Engineous-supplied Isight components. Once the components are published, the Design Gateway interface will appear, confirming that your license and license server are functioning correctly.

Installing the License on UNIX/Linux

The following are the steps you should follow to correctly install your Isight license. The steps are categorized in the following groups:

Editing Your Isight License

Your Isight license is basically a text file that contains information about what aspects of Isight you can use following installation. For the most part, you do not need to change anything in the file. However, a few customizations are necessary, as described below.

Once the license file has been received:

1. Save the license.dat file you received via e-mail into the following directory:

   <*Isight_install_directory*>/license

   Note: If the machine has old mail software, then you must remove everything in the license file before the line beginning with "SERVER".

2. If you are using a combined license file with an existing FLEXnet license server, you must add the path to engswd on the VENDOR line (second line) of the license file.

   For example, if you installed Isight in the /opt directory on a AIX machine, you would change the VENDOR from:

   VENDOR engswd

   to the following:

   VENDOR engswd /opt/SIMULIA/Isight/3.5-1/bin/AIX_5.1_ppc/engswd

3. If your license file contains the string XXXX on the SERVER line, replace the XXXX string with the host name of your machine.

   The host name of the machine may be determined by typing the command `uname -n` at the terminal prompt.

4. Save and exit your license file.

Installing and Starting the License Manager

A script is provided with your Isight installation that assists you with installing your license manager.

To install your license manager:

1. Perform one of the following actions:

   - Log in as root (obtain root permissions).
   - Verify that you have permission to write in the /etc directory.

2. Change into the following directory:

   <*Isight_install_directory*>/license

3. Execute the following command:

   ```
   ./flexlm install
   ```

   A message appears, informing you that the FLEXnet startup script is being installed. The process is complete when you are returned to the prompt.

4. Execute the following command to manually start the license manager, based on your operating system:

   ```
   ./flexlm start
   ```

   A message appears stating that your license manager was started successfully. You can now start using Isight.

   Note: If you did not have root privileges when you installed Isight, you'll have to execute this command every time you reboot your system. A warning message stating this necessity appeared during your installation. Otherwise, the license manager will start automatically after a reboot.

Modifying Your Isight License Location

The Isight installation places your license file in the <*Isight_install_directory*>\license folder. If a modification is required because, for instance, your organization uses a central FLEXnet license server, you can edit the file to be a "use_server" file and point to the central server.

To edit your license file:

1. Login as Administrator.
2. Navigate to the following directory:

   <*Isight_install_directory*>\license
3. Open the `license.dat` file with the text editor of your choice.
4. Edit the file as necessary. For example, you can edit the file as shown below (replacing *your_server_name* and *your_server_hostid* with the appropriate information for your license server).

   ```
   SERVER your_server_name your_server_hostid
   USE_SERVER
   ```

You can also combine license files for all products using FLEXnet with appropriate editing. For more information, contact SIMULIA technical support.

Configuring Your License to Work with Windows Firewall

If you specified a license server that is behind a Windows Firewall, you need to perform the following actions to ensure that you can connect to the license server when using Isight:

1. Verify that the Windows Firewall has been updated so that the necessary license ports are open for the license server. For more information, refer to any version of the *Fiper Installation and Configuration Guide*.
2. Navigate to the following directory:

   <*Isight_install_directory*>\license
3. Open the `license.dat` file in the text editor of your choice.
4. Add the port number opened on the license server machine's Windows Firewall to the SERVER line in your license file. For example, if port 1700 was opened on the Windows Firewall, your license SERVER line would appear similar to the example shown below:

   ```
   SERVER acsmachine ANY 1700
   ```

   For more information on what port numbers were opened on your license server's Windows Firewall, contact your local systems administrator.
5. Save and close your license file.
6. Start an Isight interface to verify that the you can communicate with the license server.

Understanding the Automatic License Server Process

The following information may be useful in understanding how Isight's installer automatically starts the license manager:

- If the user executing the installer is in the Admin group (on Windows) or has root access (on UNIX), then the license server is installed as a service (in /etc/rc on UNIX).
- If the user is not in the Admin group or does not have root access, a message is displayed telling the user how to install the license manager as Admin/root. For details on this process, see "Environment Variables," on page 159.

Installing Only a License Server

If you are installing only a license server, you need to run the license server script for your specific platform. The scripts are located on the Isight DVD in the FLEXnet directory. Depending on your operating system, proceed to one of the following sections:

- "Installing Only a License Server on Windows," on page 180
- "Installing Only a License Server on UNIX/Linux," on page 184

Important: If the license server you are referencing is behind a Windows Firewall (the firewall supplied with most Windows installations), you will need to manually alter your Isight license. For more information, see "Configuring Your License to Work with Windows Firewall," on page 178.

Installing Only a License Server on Windows

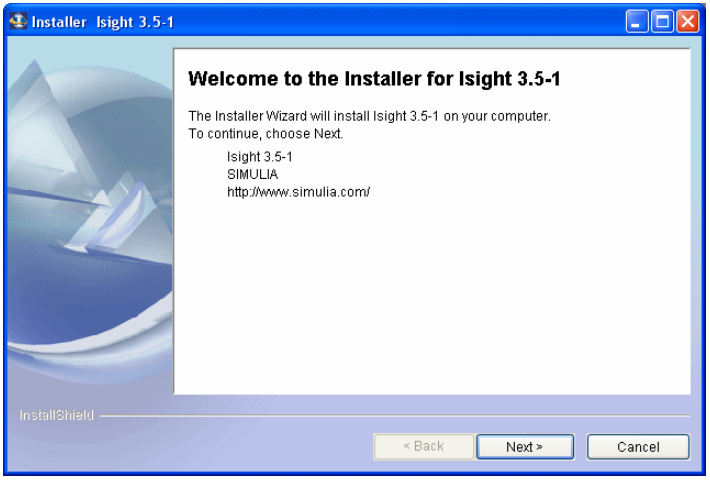
To install only a license server on a Windows platform:

1. Log in as the Administrator that will install the license server.

   Note: You *must* be the Administrator or have administrative privileges to install on any Windows platform.

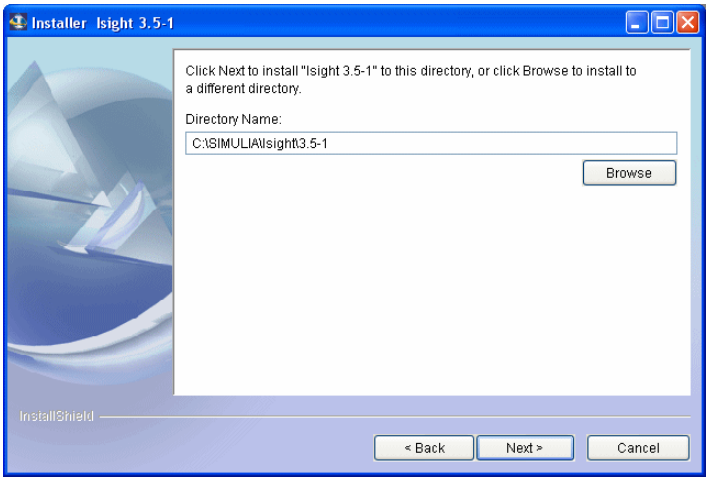
2. Insert the Isight DVD.

3. Navigate to the FLEXnet directory, and then run the flexlmwin32.vbs script. The Isight license server setup program starts. It will guide you through the rest of the installation process. The Welcome screen appears, showing the version of the license server being installed.



4. Click **Next**. The software agreement screen appears.

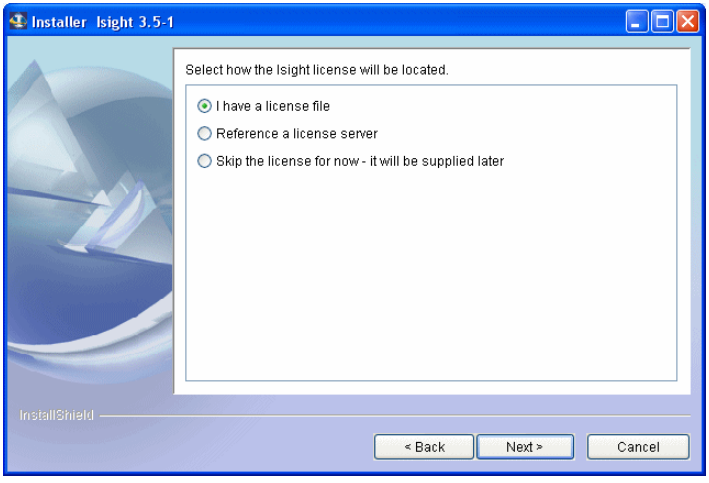
5. Read the agreement; then, click **Next**. The installation directory screen appears.



6. Perform one of the following actions:

   - To accept the default directory, click **Next**. Your default directory may vary based on your system settings and permissions.
   - If you wish to place the license server in a different location, click the **Browse** button, and choose a destination folder; then, click **Open**.

7. Click **Next** to continue the installation procedure once you have specified your directory. The license location screen appears.

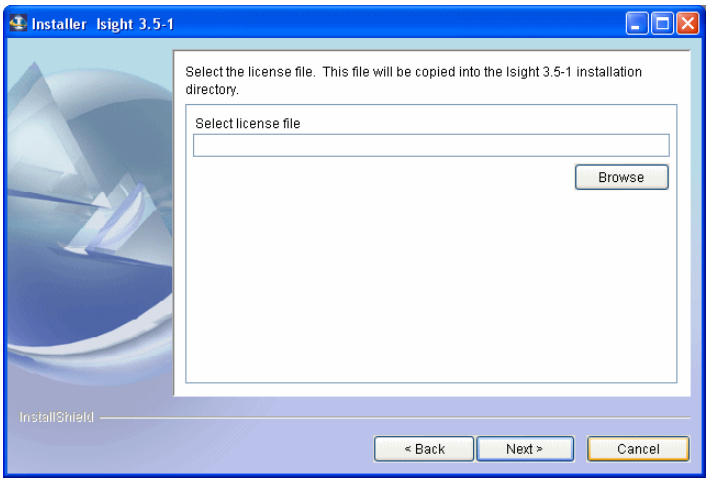


8. Select **I have a license file**.

Note: You can choose the **Skip the license for now - it will be supplied later** option. If you select this option, you will be required to manually start your license server once you receive your license file as described in "Installing the Isight License After Installation," on page 170. Proceed to step 11.

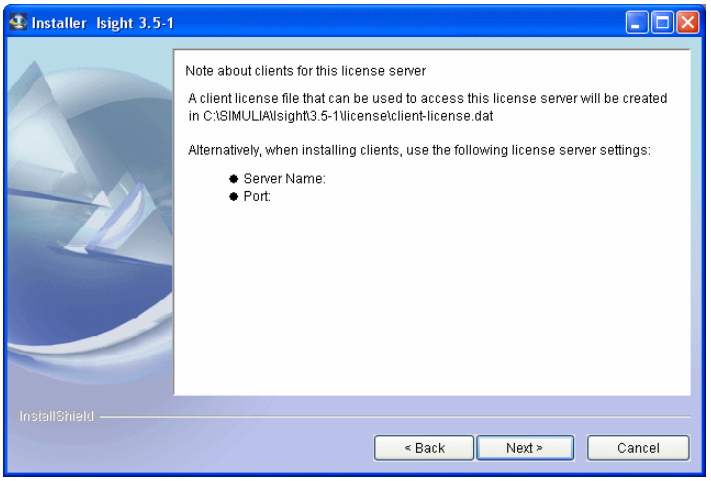
Note: Do not select Reference a license server.

9. Click **Next**. The select license file screen appears.

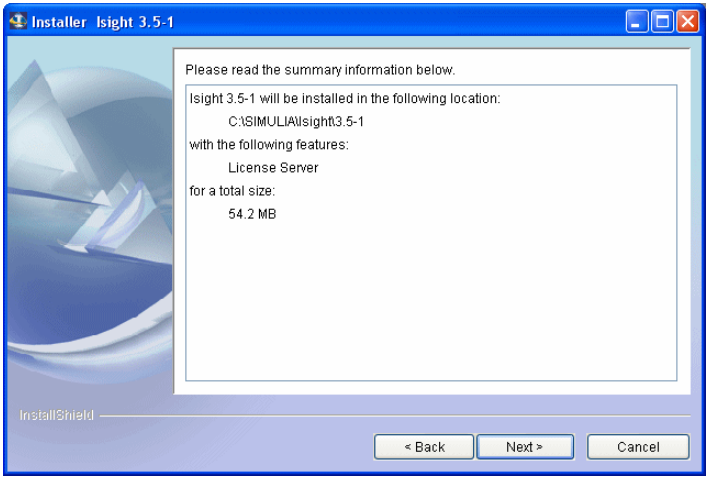


10. Enter the full path and name of the license server file or use the **Browse** button to locate the file.

11. Click **Next**. A screen showing where the license server file will be located and information about installing clients appears.

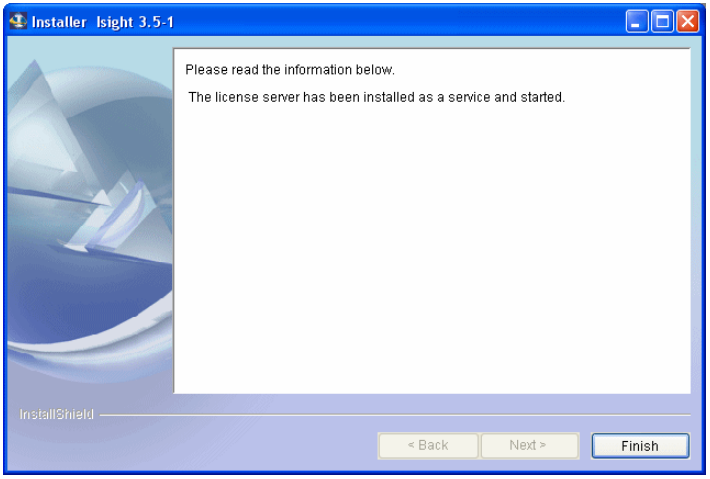


12. Make a note of the server name and port to use for installing clients; then, click **Next**. After the installation program calculates the required disk space needed, an installation summary screen appears showing that only the license server will be installed and the location.



13. Click **Next**. The installation progress screen appears, and the license server is installed. An overall summary message appears when the installation is complete.

14. Click **Next**. A screen showing that the server has been installed and started appears.



15. Click **Finish**.

Installing Only a License Server on UNIX/Linux

To install only a license server on UNIX/Linux:

1. Log in as the user that will install the license server.
2. Load the Isight DVD and mount it, if necessary. For more information, contact your systems administrator. If your machine does not have a DVD drive, you can install the license server from a shared disk or network folder. For more information, see "Accessing the Installation Files from a Remotely Mounted DVD," on page 155.

   Note: On some UNIX systems, the Isight DVD will automatically mount when it is inserted into the DVD drive, and a file browser window will appear. In this case, you can double-click the correct setup file. These files are listed in step 4.
3. From the terminal window (shell), change to the directory where the Isight DVD is mounted. For example, type:

   ```
   cd /media/dvd
   ```

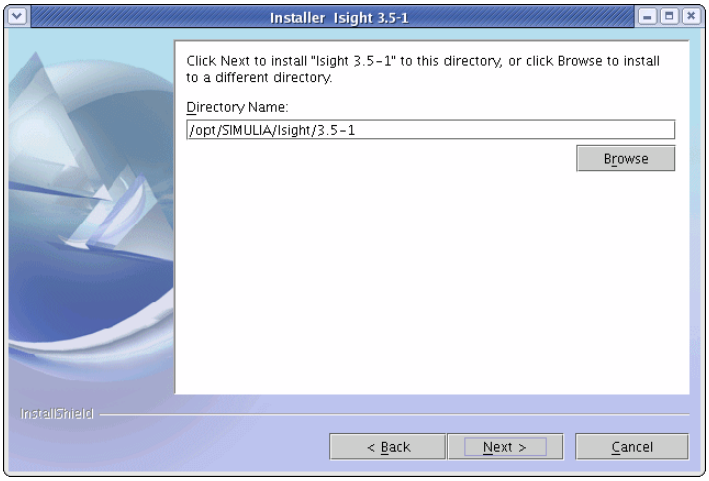
4. Navigate to the FLEXnet directory; then, execute the installation script. Type one of the following commands, based on your operating system:

   - `./flexlmaix`
   - `./flexlmhpux`
   - `./flexlmlinux`
   - `./flexlmsolaris`

   The Isight license server setup program starts. It will guide you through the rest of the installation process. The Welcome screen appears, showing the version of the license server being installed.

5. Click **Next**. The software agreement screen appears.

6. Read the agreement, then click **Next**. The installation directory screen appears.

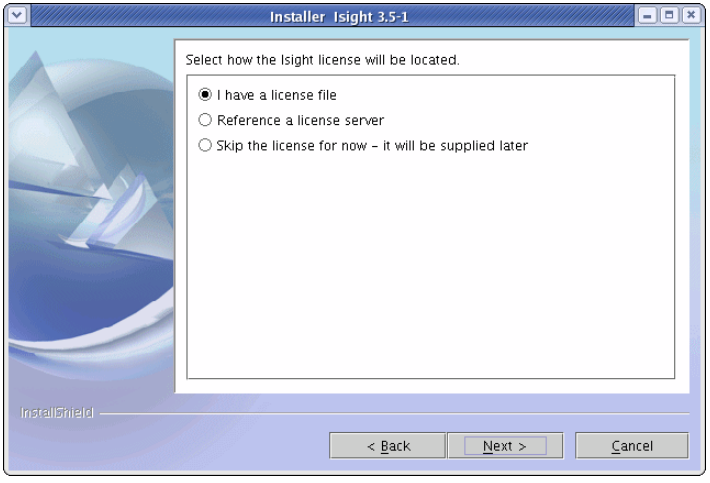


7. Perform one of the following actions:

   - To accept the default directory, click **Next**. Your default directory may vary based on your system settings and permissions.
   - If you wish to install the license server in a different location, click the **Browse** button, and choose a destination folder; then, click **Open**.

8. Click **Next**.

9. Click **Next**. The license location screen appears.

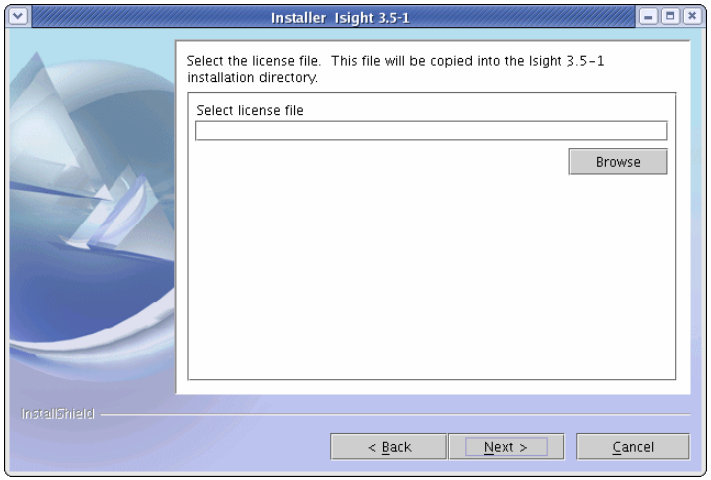


10. Select **I have a license file**.

    Note: You can choose the **Skip the license for now - it will be supplied later** option. If you select this option, you will be required to manually start your license server once you receive your license file as described in "Installing the Isight License After Installation," on page 170. Proceed to step 11.

    Note: Do not select Reference a license server.

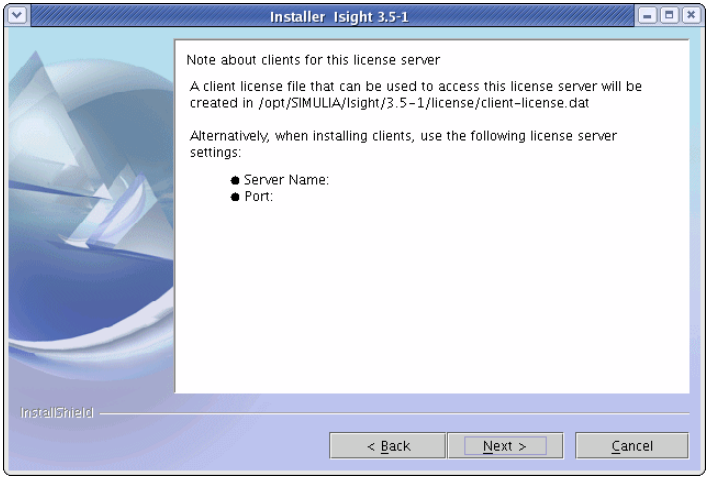
11. Click **Next**. The license file selection screen appears.



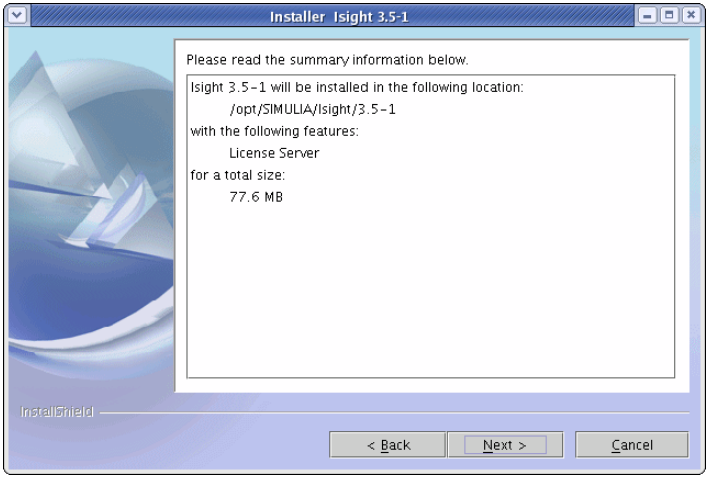
12. Enter the full path and name of the license file or use the **Browse** button to locate the file.

    Note: The license file must be for the computer the installation is running on if the license server is to be installed as a daemon.

13. Click **Next**. A screen showing where the license server file will be located and information about installing clients appears.

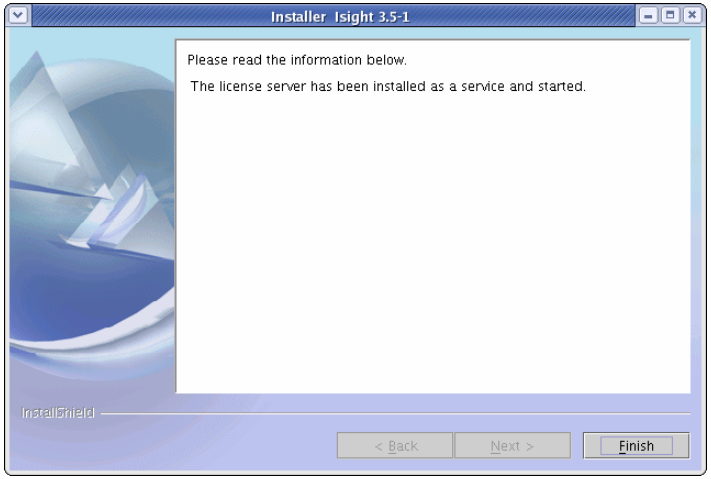


14. Make a note of the server name and port to use for installing clients; then, click **Next**. After the installation program calculates the required disk space needed, an installation summary screen appears showing that only the license server will be installed and the location.



15. Click **Next**. The installation progress screen appears, and the license server is installed.n overall summary message appears when the installation is complete.

16. Click **Next**. A screen showing that the server has been installed and started appears.



17. Click **Finish**.

About SIMULIA
SIMULIA is the Dassault Systèmes brand that delivers a scalable portfolio of Realistic Simulation solutions including the Abaqus product suite for Unified Finite Element Analysis, multiphysics solutions for insight into challenging engineering problems, and SIMULIA SLM for managing simulation data, processes, and intellectual property. By building on established technology, respected quality, and superior customer service, SIMULIA makes realistic simulation an integral business practice that improves product performance, reduces physical prototypes, and drives innovation. Headquartered in Providence, RI, USA, SIMULIA provides sales, services, and support through a global network of regional offices and distributors. For more information, visit www.simulia.com.

About Dassault Systèmes
As a world leader in 3D and Product Lifecycle Management (PLM) solutions, Dassault Systèmes brings value to more than 100,000 customers in 80 countries. A pioneer in the 3D software market since 1981, Dassault Systèmes develops and markets PLM application software and services that support industrial processes and provide a 3D vision of the entire lifecycle of products from conception to maintenance to recycling. The Dassault Systèmes portfolio consists of CATIA for designing the virtual product - SolidWorks for 3D mechanical design - DELMIA for virtual production - SIMULIA for virtual testing - ENOVIA for global collaborative lifecycle management, and 3DVIA for online 3D lifelike experiences. Dassault Systèmes' shares are listed on Euronext Paris (#13065, DSY.PA) and Dassault Systèmes' ADRs may be traded on the US Over-The-Counter (OTC) market (DASTY). For more information, visit http://www.3ds.com.

DASSAULT SYSTEMES