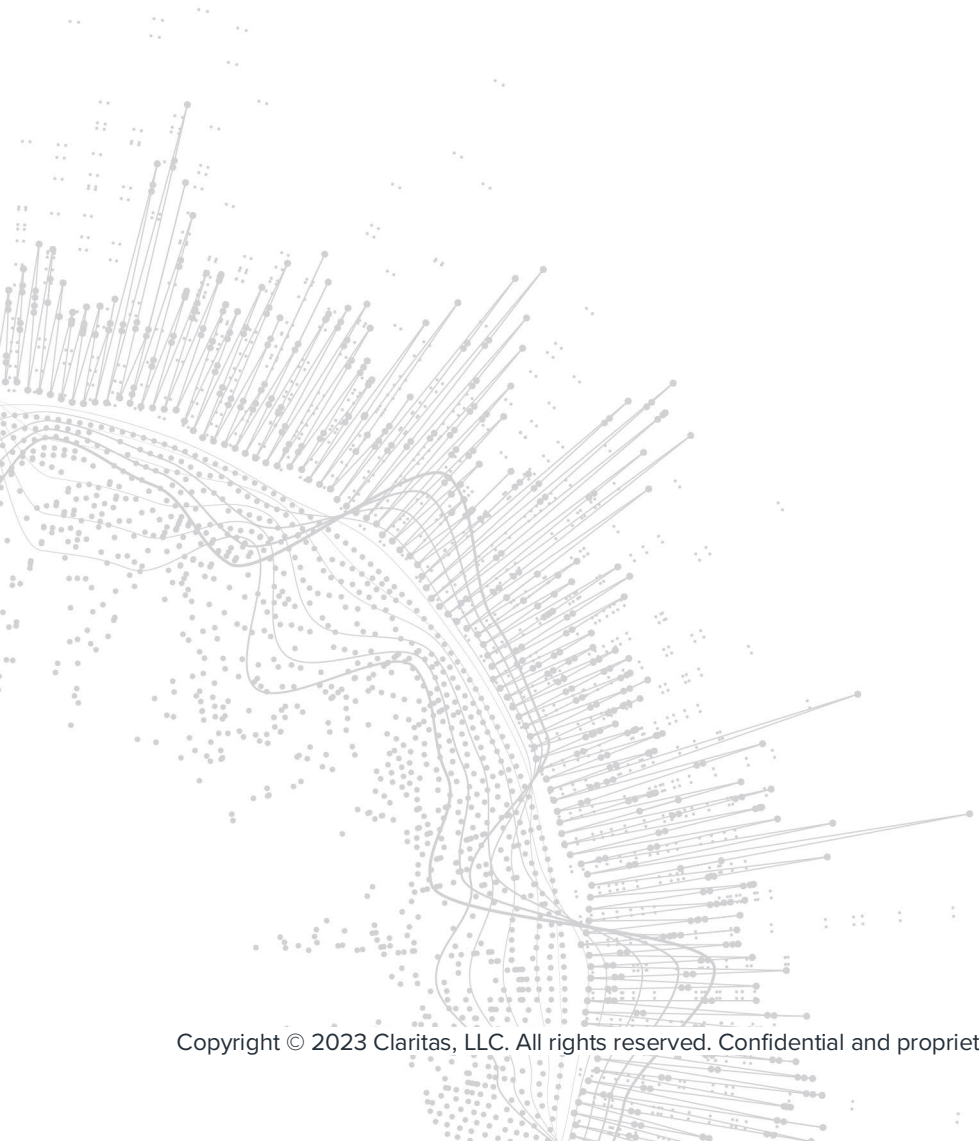




PrecisionCode 6.2 2023 Q2 Update

User Guide



Claritas Precision Code 6.2 2023 Q2 Update

User Guide

Contents

Introduction	3
Product Overview	3
How to Use This Guide	4
Technical Support	4
Start PrecisionCode	4
Understand the Main Screen	6
Close PrecisionCode	6
What's Next.....	6
Customize Your System	8
Set Your User and Data Preferences	8
Load Your Previous Job Settings	10
View Your License Information and Licensed Data/Scores	10
What's Next.....	11
Import Files and Database Tables.....	13
Import a Source File	13
Import a Database Table	17
What's Next.....	20
Work With Imported Files And Database Tables.....	21
Access the Options Screen	21
Append Geographic Data	22
Append Segmentation Codes.....	23
Append Claritas Scores.....	25
Create Summary Data for a File.....	26
Create National Profiles	27
Create Regional Profiles.....	28
Filter a File or Database Table	29

Select Output Formats.....	30
What's Next.....	33
Schedule A Job Through The Command Line Function.....	34
Generate Output.....	34
Work With Output Files And Schedules.....	37
Generate Output.....	37
View the Generated Output	40
Manage Job Schedules	44
Access the Scheduler Screen.....	44
View the Status of Scheduled Jobs	45
Stop a Scheduled Job	45
Delete a Scheduled Job.....	46
What's Next.....	47
Look Up Information For A Location.....	48
Access the Single Location Lookup Screen.....	48
Specify a Lookup Method for Your Location Data	50
Process and View Lookup Information	52
Copy and Paste Lookup Information	53
Print Lookup Information.....	53
What's Next.....	53
Technical Support.....	53
Legal Notifications	54

INTRODUCTION

Product Overview

Claritas PrecisionCode is an easy-to-use, wizard-driven data coding application. With PrecisionCode, you can take your existing customer lists and prepare them for a variety of uses, such as data analysis, spatial mapping, and direct mailing. PrecisionCode allows you to perform the following:

- **Assign latitude and longitude coordinates** based on an exact address or intersection. With the Address and Intersection Lookup features, you can quickly and easily determine the latitude and longitude coordinates of any address or intersection in your customer list. This information can be used in a variety of mapping packages to show the location of customers, prospects, competitors, or retail locations.
- **Include consumer lifestyle information with every address.** PrecisionCode includes the Claritas PRIZM Premier, Claritas P\$YCLE Premier, and Claritas ConneXions segmentation databases. These databases categorize customers into specific lifestyle groups, each of which exhibits unique purchasing behaviors. This type of information enhances your ability to track and understand your customers and their spending behavior.

If you are licensed for any of these databases, PrecisionCode appends marketing information to each address in your customer list. This information can be appended at varying levels of geography, including ZIP+6, ZIP+4, ZIP Code, and block group.

- **Append geography codes to the locations in your customer list.** PrecisionCode can include state, county, census tract, and block group codes with every address. These geography codes can help you determine the concentration of customers or prospects in a market area.
- **Include Claritas scores to the locations in your customer list.** PrecisionCode allows you to include Claritas scores, such as Claritas Income Producing Assets Indicators, Claritas Net Worth Indicators, Claritas Consumer Score - Technology Adoption and Heavy Technology Usage, with every address. These scores can help you determine the further analyze your customers or prospects in an area.
- **Automatically create profiles.** PrecisionCode comes with an automatic profile creation feature. This feature allows you to create National and/or Regional profiles that can be uploaded into the Claritas Segmentation & Market Solutions platform for further analysis.
- **Standardize the addresses in your customer list.** PrecisionCode automatically corrects and updates the street, city, and state names contained in your customer lists to conform to U.S. Postal Service standards. This feature can reduce your mailing costs and improve delivery speed.

How to Use This Guide

The User Guide contains instructions on how to use PrecisionCode. It contains the following chapters:

- **Customize Your System**, which explains how to customize your system according to your preferred user settings
- **Import Files and Database Tables**, which explains how to prepare and import your source files for coding
- **Work with Imported Files and Database Tables**, which explains how to append data to your input file and define your output
- **Work with Output Files and Schedules**, which explains how to generate and save your output or processed files, and maintain your processing schedules
- **Look up Information for a Location**, which describes how to get information for any address, intersection, ZIP/ZIP+4, or latitude/longitude

Technical Support

If you need further assistance, not provided in online Help or in this guide, please contact the Claritas Solution Center between 9:00 a.m. and 8:00 p.m. (Monday through Friday, EST) at 800.866.6511.

Start PrecisionCode

After you have successfully installed PrecisionCode, a PrecisionCode icon displays on your desktop. Follow the steps in this section to access or start PrecisionCode.

Note: It is advised to restart your workstation prior to launching PrecisionCode for the first time.

Additionally, the application must be Run as Administrator when processing any imported files that would also generate an output file. To do this:

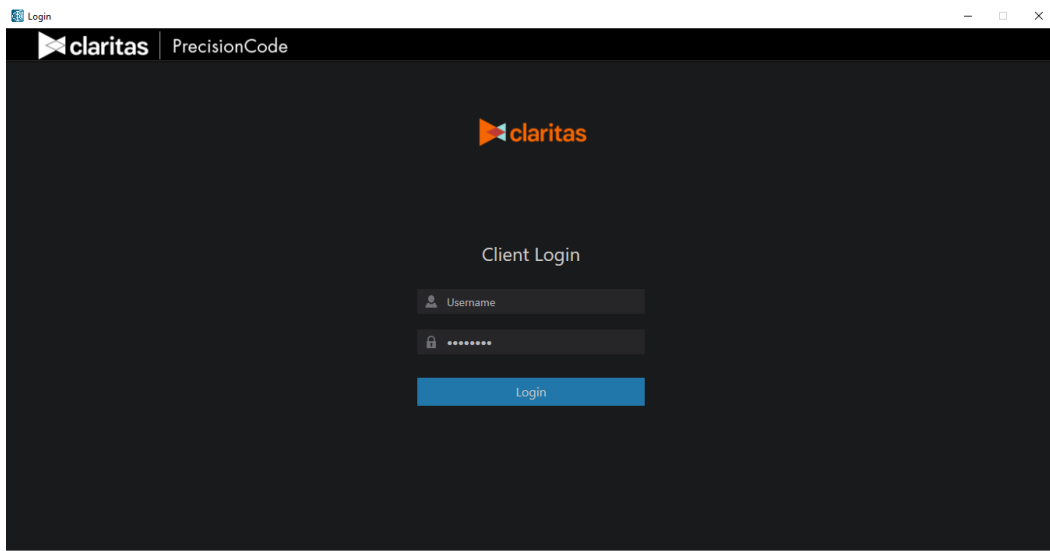
- Right click on the PrecisionCode desktop shortcut
- Select Properties > Compatibility Tab
- Check the 'Run as Administrator' checkbox.

1. On your desktop, double-click the **PrecisionCode** icon on your desktop.



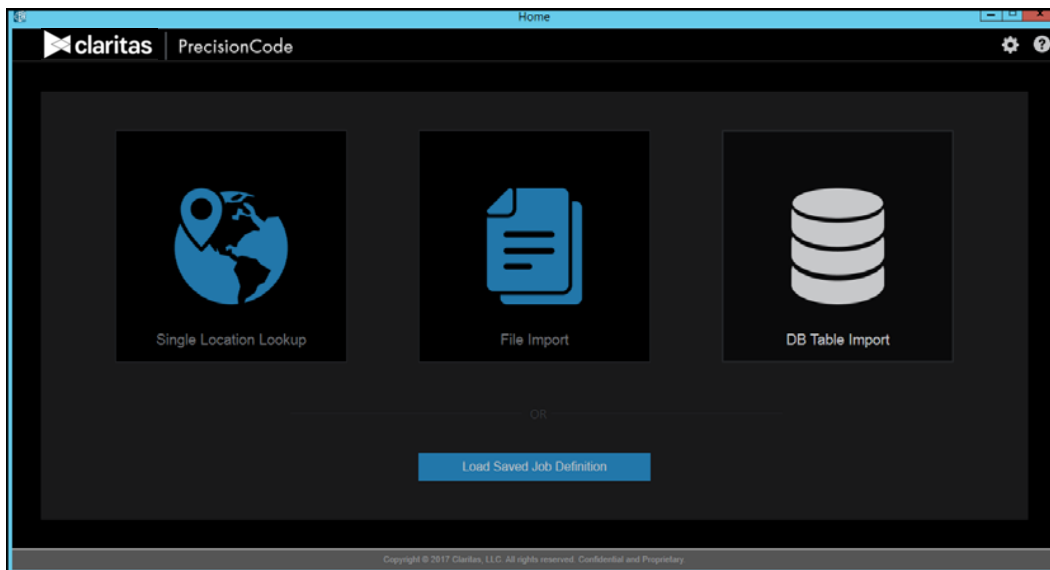
PrecisionCode icon

The Client Login screen appears.



Client Login screen

2. Type your user ID and password in the appropriate text boxes, and then click **Login**. The PrecisionCode main screen appears.








PrecisionCode main screen

Note: If you cannot remember your password, contact the Claritas Solution Center between 9:00 a.m. and 8:00 p.m. (Monday through Friday, EST) at 800.866.6511.

Understand the Main Screen

The main interface consists of the following icons:

ICON	DESCRIPTION
 <p>Single Location Lookup</p>	Lets you search for geographic and/or segmentation information about an address/intersection, a ZIP/ZIP+4 Code, a combination of latitude and longitude coordinates, or a block group.
 <p>File Import</p>	Allows you to import a .CSV, .XLS, .XLSX, or .TXT file and process the file according to the options you select
 <p>DB Table Import</p>	Lets you import an Oracle [®] or Microsoft [®] SQL Server [®] database table and process it.
	Opens and sets your previously saved PrecisionCode job settings
	Allows you to set and manage your system preferences, user account, job or process schedules, and licensed data

Close PrecisionCode

To close the application after use, simply click the Close button (that is, the button with an X mark) located in the upper-right corner of the screen.

What's Next

You are now ready to use PrecisionCode.

To customize your system prior to use, follow the procedures in the “Customize Your System” chapter.

To import files and database tables, follow the procedures in the “Import Files and Database Tables” chapter.

To work on imported files for process, follow the procedures in the “Work With Imported Files And Database Tables” chapter.

To work on output files and task schedules, follow the procedures in the “Work With Output Files And Schedules” chapter.

To look up information about a location, follow the procedures in the “Look Up Information For A Location” chapter.

CUSTOMIZE YOUR SYSTEM

Before you begin using Claritas PrecisionCode, it is recommended that you customize your system according to your preferences. You can set the following preferences for your system:

- Your Web and directory access and protocols
- Your preferred data settings such as default input and output directory, country location, and latitude/longitude format
- Your preferred job settings

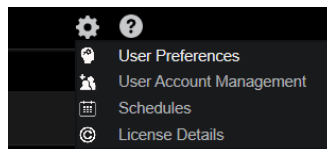
You may also view details regarding your licensed segmentation data and Claritas scores before you use PrecisionCode.

Set Your User and Data Preferences

You can set your default input directory for your source files, default output directory, the country where you are located, your preferred longitude and latitude format, your output file column names, and your preferred spatial refinement process on the User Preferences screen. The system uses your user preferences when processing scheduled jobs as it automatically gets files from your specified input directory and saves the output to your specified output directory.

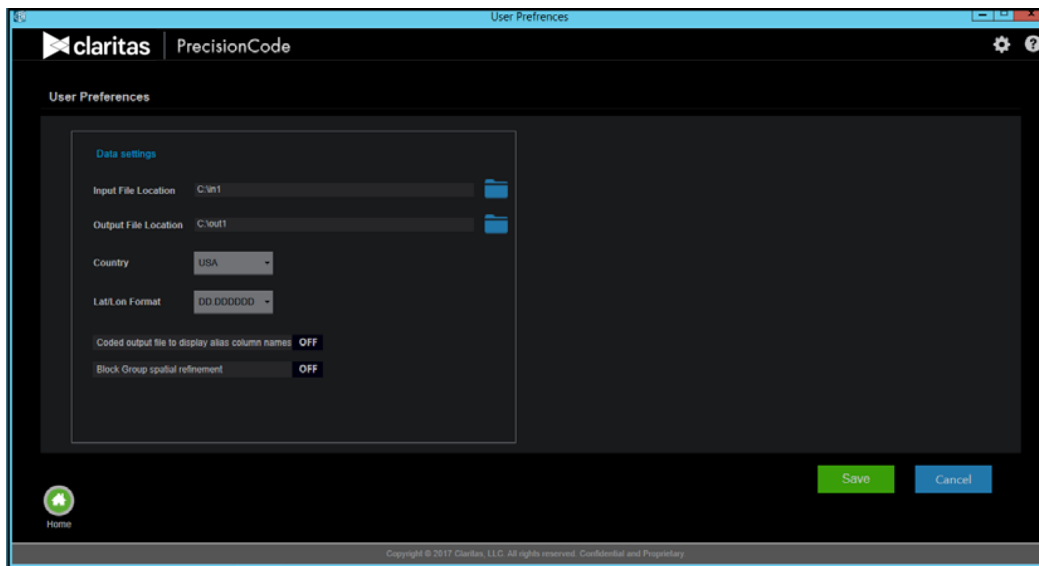
Follow the steps in this section to set your preferences.

1. On the PrecisionCode main screen, click the **System Tools** icon in the upper-right corner of the screen. A list of options appears.



System Tools options

2. Click **User Preferences**. The User Preferences screen appears.



User Preferences screen

3. In the **Data settings** box, in the **Input File Location** text box, specify the path where your input source files are located.

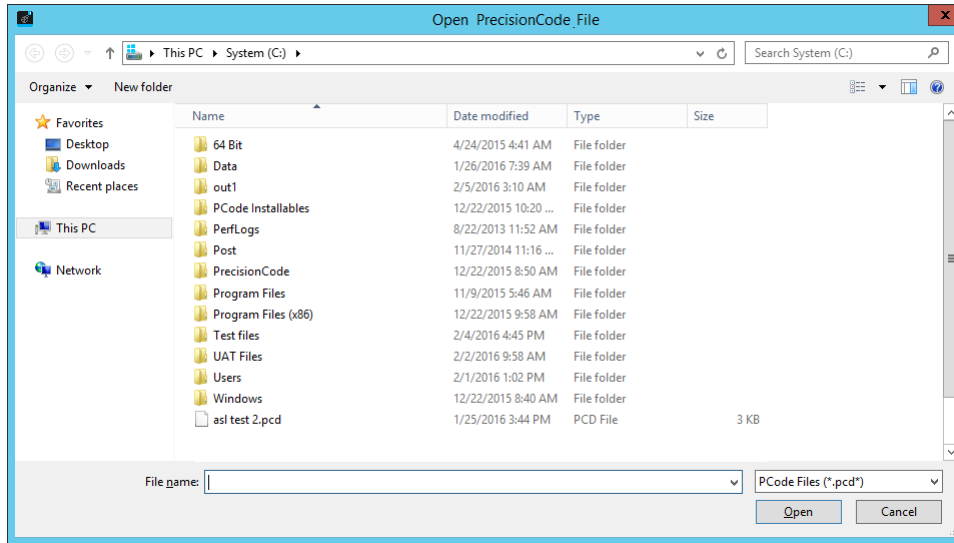
Note: It is recommended that you put your input source files in a folder under My Documents or on a secondary drive, and then select that path.
4. In the **Output File Location** text box, specify the path where you want to save your output files or processed files.

Note: It is recommended that you select a folder under My Documents or on a secondary drive to store your output files.
5. In the **Country** drop-down list, select the abbreviation for the country in which you are located.
6. In the **Lat/Long Format** drop-down list, select your desired format for latitude and longitude.
7. To set your coded output file to show different names for your columns, click the **OFF** button next to **Coded output file to display alias column names** to set it to **ON**.
8. To perform geographic optimization of Block Group assignment to ensure the best match possible to the standard roster, click the **OFF** button next to **Block Group spatial refinement** to set it to **ON**.
9. Click the **Save** button. A message box appears indicating that the settings have been updated.
10. Click **OK** to close the message box.
11. Click the Home button to return to the main screen or click the System Tools icon to select your next action.

Load Your Previous Job Settings

You can load your previous job definitions saved using the new version of PrecisionCode. Follow the steps on this section to load your previous job settings.

1. On the PrecisionCode main screen, click the **Load Saved Job Definition** button. The Open PrecisionCode File dialog box appears.



Open PrecisionCode File dialog box

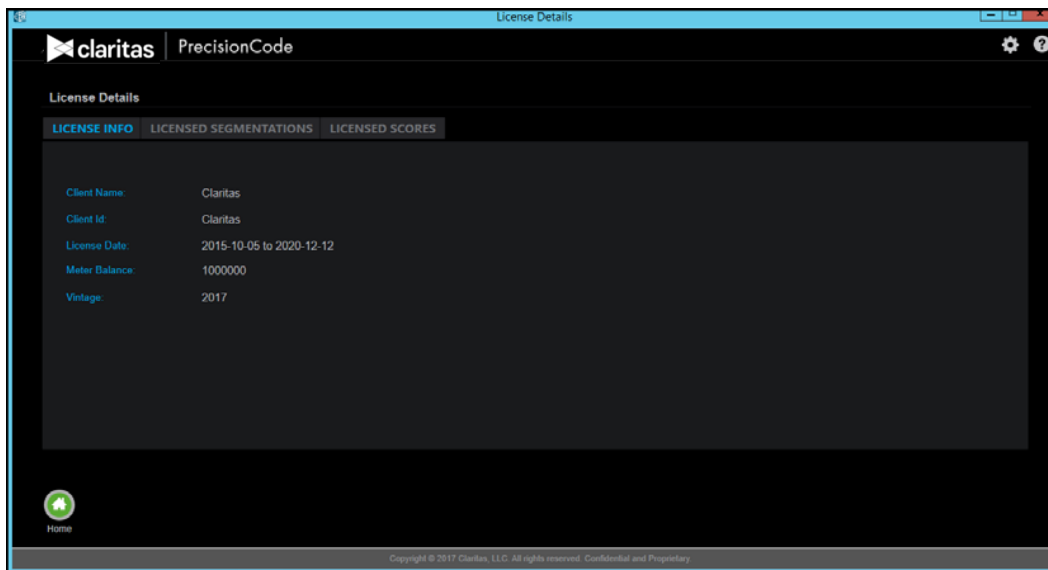
2. Navigate to and select the file that contains the job settings that you want to load, and then click **Open**. The system loads the saved job settings.

View Your License Information and Licensed Data/Scores

You can have detailed information about your license and your licensed data and cores, including the appropriate levels of coding for each data on the License Details screen.

Follow the steps in this section to view your license information and details about your licensed data and scores.

1. In the **System Tools** list of options, click **License Details**. The License Details screen appears.



License Details screen

2. Do any of the following:

TO VIEW YOUR	CLICK
License information	License Info. Your name, ID, license date, and other details appear on the screen.
Licensed segmentation data	Licensed Segmentations. A list of segmentation data that you have licensed, including the level of coding, appears on the screen.
Licensed Claritas scores	Licensed Scores. A list of Claritas scores that you have licensed, including the level of coding, appears on the screen.

Note: If this screen does not show your correct license information, data, or Claritas scores, or if you have any questions about your license, contact your account representative or the Claritas Solution Center between 9:00 a.m. and 8:00 p.m. (Monday through Friday, EST) at 800.866.6511.

3. Click the **Home** button to return to the main screen or click the **System Tools** icon to select your next action.

What's Next

You are now ready to use PrecisionCode.

To import files and database tables, follow the procedures in the “Import Files and Database Tables” chapter.

To work on imported files for process, follow the procedures in the “Work with Imported Files and Database Tables” chapter.

To work on output files and task schedules, follow the procedures in the “Work with Output Files and Schedules” chapter.

To look up information about a location, follow the procedures in the “Look up Information for a Location” chapter.

IMPORT FILES AND DATABASE TABLES

Import a Source File

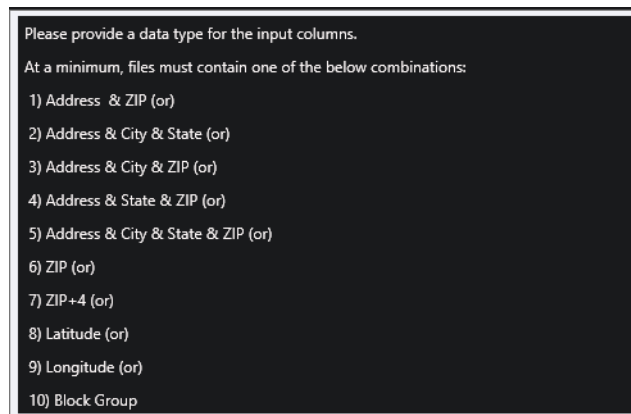
The first step in processing output from a source file—or “input” file—is to import your source file into Claritas PrecisionCode. You can import the following types of files:

- A comma-separated values (.CSV) file
- A Microsoft® Excel® (.XLS or .XLSX) file
- ASCII Text files (fixed-format or delimited)

Important! Before beginning to import your data from the source file, you should verify that the file name for the data source does not include spaces and the same column names.

If your data source is stored in a very deep directory structure or one with very long folder names, your batch process could fail. Before beginning to import your data, you should move your data source file to a directory location with a file path shorter than 255 characters.

At a minimum, make sure that your file contains one of the combinations listed below.



Importing a file requirement as shown in Help

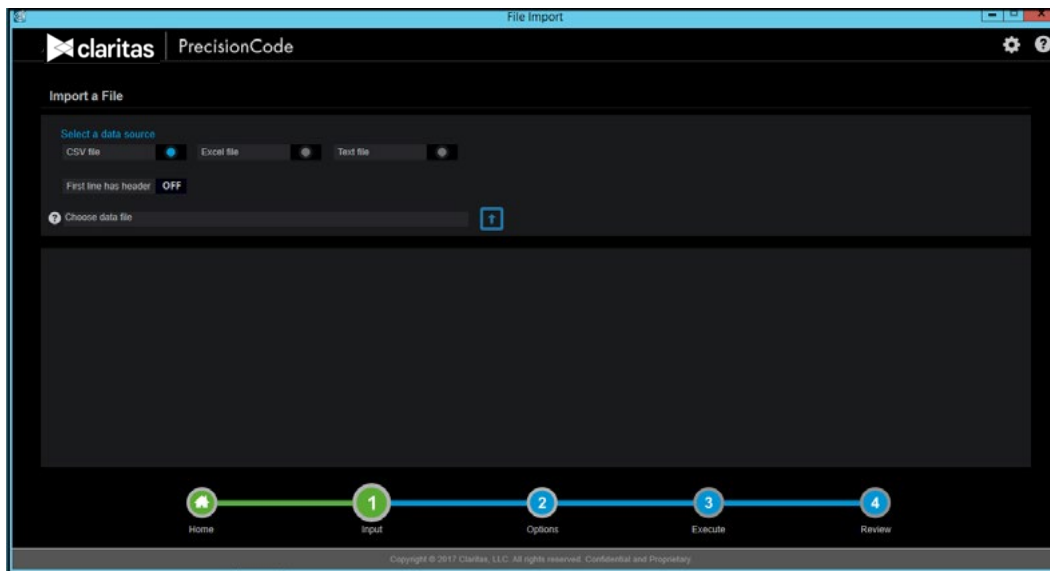
After you import a file, you can then append data; get segmentation information, data scores, and profile data; create summary output; select output formats; and create screening criteria before you process the file. For additional information about completing these tasks, see the “Work with Imported Files and Database Tables” chapter.

Follow the procedure in this section to import a file.

1. On the PrecisionCode main screen, click File Import. The Import a File screen appears.



File Import icon



Import a File screen

2. Below **Data source**, click to select the file type of your source file.

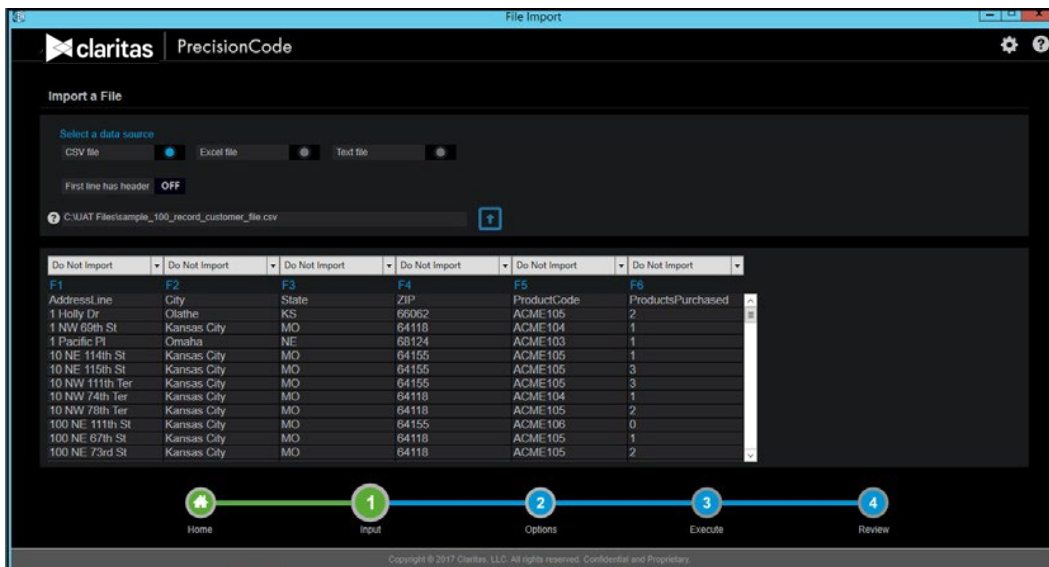
If you select the **Text file option**, the Field Delimiter options appear. Click to select the delimiter (that is, the character that separates the columns of data) included in your source file.



File Input screen with the Text File option selected and the Field Delimiter options displayed

3. Click the folder with arrow button next to the **Choose data file text** box, and then navigate to and select the location of the file that you want to import. The selected path appears in the text box and contents of the selected file appears on the screen.

Note: There is no limit to the number of records in the file that you want to import; however, the system can only show the first 250 records in your file on the screen.



File Input screen showing the contents of a .CSV source file

4. Do either of the following:

IF YOUR SOURCE FILE	THEN
Has first rows as headers and you want to use the headers	Click the OFF button next to First line has header to set it to ON .
Does not include headers or has headers but you do not want to use them	Leave the First line has header setting to OFF

5. By default, the text box on top of each column is set to **Do Not Import**. This setting excludes the column from processing. Do either of the following:

IF	THEN
You select to enable the headers in your source file	Verify the information displayed in the header text box located on top of each column and make sure that displayed information matches the column data type that you want to include in your output and there are no similar column headers. If the text box shows Do Not Import for those columns that you want to include, click the drop-down arrow and select the appropriate column header to display.
Your source file does not include headers or want to assign new headers	Click the drop-down arrow for the header information of each column that you want to import and then select the appropriate header for the column data type.

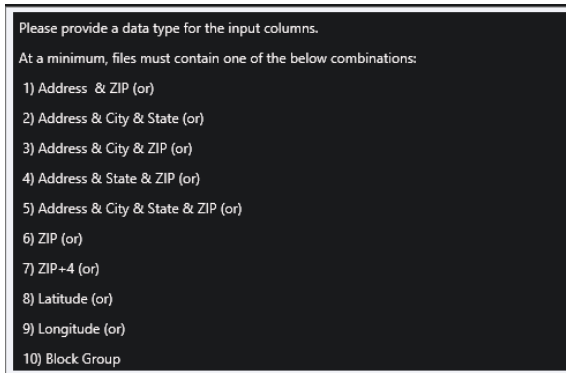
Note: Selecting **Data** as column header allows you to bring in the column data into the coded file as is. You can use this option as column header multiple times. Selecting **Usage/Consumption** as column header requires that the column data contains numeric values.

6. View and verify the contents of your file using the scroll bar located to the right of the contents box.

You have completed the locating and defining your import file. Proceed to the “What’s Next” section.

Import a Database Table

You can import an Oracle or a Microsoft SQL Server database table into PrecisionCode. Similar to file import, prior to importing a database table, make sure that at a minimum, your table contains one of the combinations listed below.



Importing a file requirement as shown in Help

Follow the procedure in this section to import a database table.

1. On the PrecisionCode main screen, click **DB Table Import**. The Import a Table screen appears.

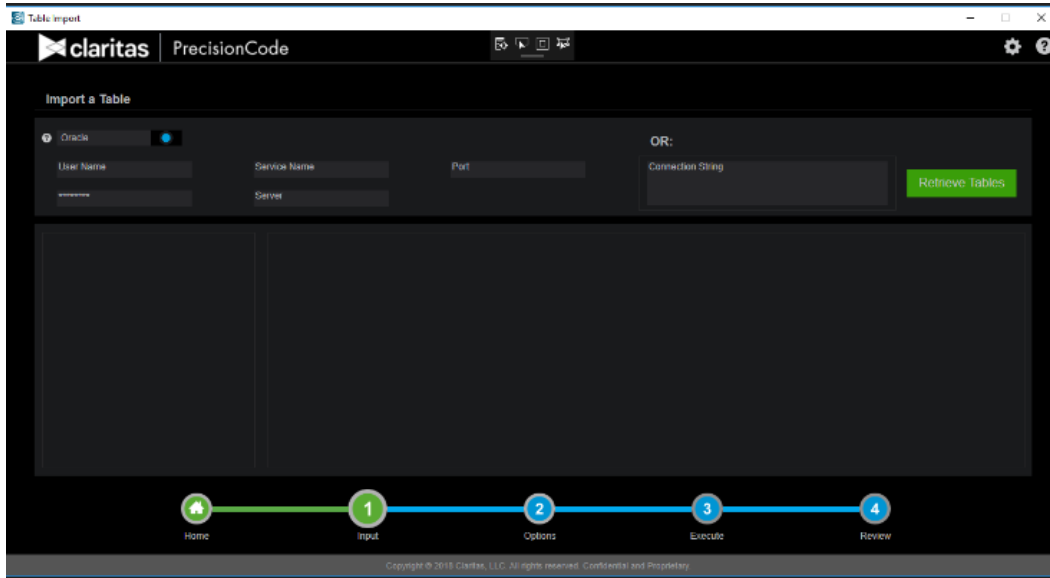


DB Table Import icon



Select a Database screen

- Click to select the type of the database table that you want to import. The options are **Oracle** and **SQL Server**. The appropriate fields for importing the selected database table type appears.



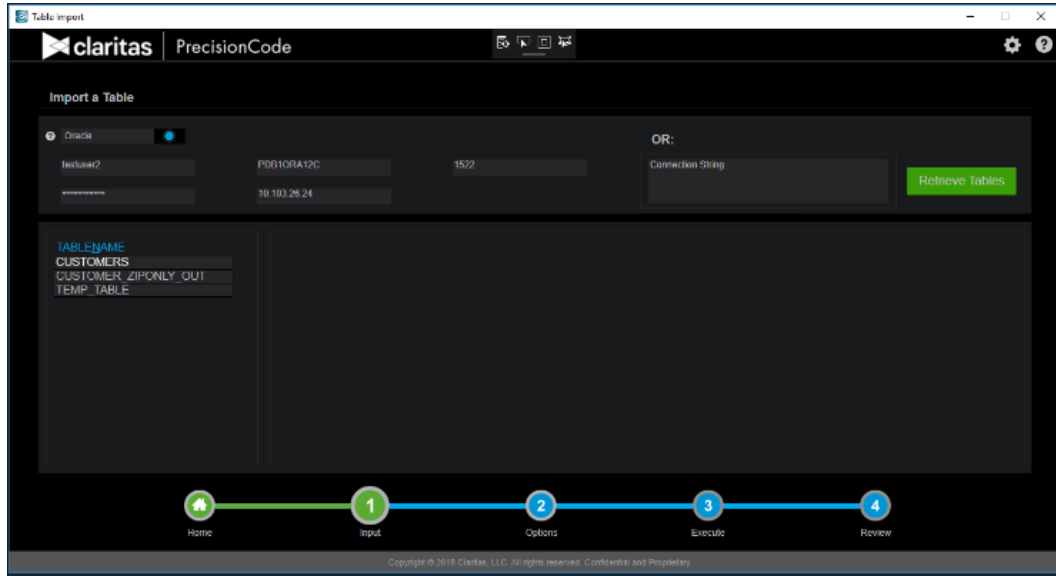
Import a Table screen showing the options importing an Oracle database table

- Type your user name, password, service name, server name, and port in the appropriate text boxes. Alternatively, you can type your server connection path in the **Connection String** text box.
- Verify your settings and then click **Retrieve Table**. The Schema Viewer appears showing a list of available schemas.



Import a Table screen showing the Schema Viewer with a list of schemas

- Use the scroll bar to locate the schema that contains the table that you want to import, double-click to select it, and then click **Retrieve Tables** within the **Schema Viewer**.



Import a Table screen showing the available tables for import

- Use the scroll bar to locate the table that you want to import, and then double-click it. The contents of the selected table appear on the right pane.



Import a Table screen showing the contents of the selected table

- By default, the system automatically matches the column header with the original column headers in the table. However, it only detects common column headers such as Address, City, and State. The rest of the column headers are set to **Do Not Import**. Click the **Do Not Import** drop-down arrow and select the appropriate column header for the displayed column information.

Note: Selecting **Data** as column header allows you to bring the column data into the coded file as is. You can use this option as column header multiple times. Selecting **Usage/Consumption** as column header requires that the column data contains numeric values.

Also, make sure that you assign a column header only once. Otherwise, you will get an error message when you proceed to the next step.

You have completed the locating and defining your import database table. Proceed to the “What’s Next” section.

What’s Next

You are now ready to start preparing your input file or database table for processing. See the next chapter, “Work with Imported Files and Database Tables” for the next steps.

Note: The sequence or progress bar located on the bottom of the screen shows the current step that you are performing in the process by highlighting the number in green. You can use this bar to navigate back to the Home or main screen, or move to the next step. However, you must complete the actions required for the current step before you can move to the next step.

WORK WITH IMPORTED FILES AND DATABASE TABLES

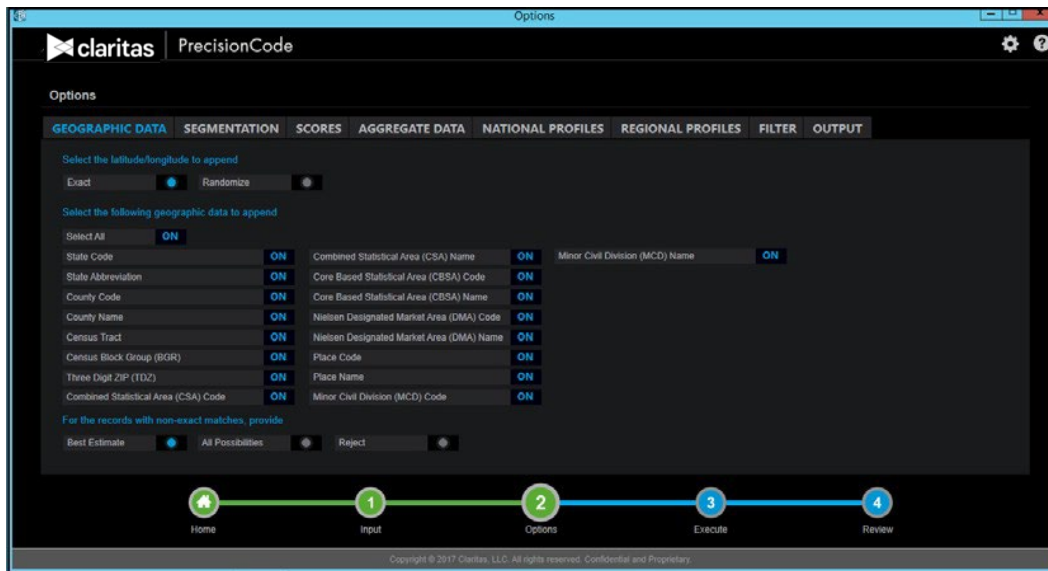
After you have located and defined your input source file or database table, you are now ready to prepare it for process via the Option screen. This chapter explains what to do with a file or database table after you imported it into Claritas PrecisionCode. The following topics are covered:

- Access the Options Screen
- Append Geographic Data
- Append Segmentation Codes
- Append Claritas Scores
- Create Summary Data for a File
- Create National Profiles
- Create Regional Profiles
- Filter a File or Database Table
- Select Output Formats

Access the Options Screen

Notice that there is a sequence or progress bar located on the bottom of the screen. This bar shows the current step that you are in by highlighting the number in green. You can use this bar to navigate back to the Home or main screen, or move to the next step. However, you must complete the actions required for the current step before you can move to the next step.

To begin working on your input file or database table for processing, click number **2 - Options** in the sequence bar. The Options screen appears showing the **GEOGRAPHIC DATA** tab as the default tab.



Options screen showing the GEOGRAPHIC CODES tab

The available tabs on this screen depend on the data you licensed.

Append Geographic Data

Appending data involves adding data to each record of your file or database table based upon the information that already exists for each record.

Note: A record is a single “line” in a file containing information about a single address, customer, intersection, or business location.

You can append geographic codes (for example, census, postal, and Nielsen Media Research (DMA) codes) to your imported file or table.

Follow the steps in this section to append geographic data.

1. On the GEOGRAPHIC DATA tab, below Select the latitude/longitude to append, click one of the following:

CLICK	TO APPEND
Exact	The exact latitude and longitude coordinates of your location
Randomize	Randomly selected latitude and longitude coordinates to make your exact location untraceable. The system anonymizes the latitude and longitude for each data point by up to 0.10 of a mile. However, this does not affect the segmentation and scores assigned to the data point on the geocoded file. The output should only be used for customer analysis.

2. By default, all geographic data options are selected. To select specific options, perform the following:
 - a. Below **Select the geographic data to append**, click the **ON** button next to **Select All** to set it to **OFF**. This automatically resets all geography level buttons to **OFF**.
 - b. Select the geography data that you want to add and then click the **OFF** button for each data to set it to **ON**.
3. Below **For records with non-exact matches, provide**, click one of the following to specify how you want to the system to handle records with no exact geographic code match:

CLICK	TO
Best Estimate	Get the closest code that the system can find or locate for each non-matched record
All Possibilities	Get all possible codes for each non-matched record
Reject	Exclude or disregard all non-matched records

4. Verify your selections and then click the available tab to proceed preparing your file or database table for process.

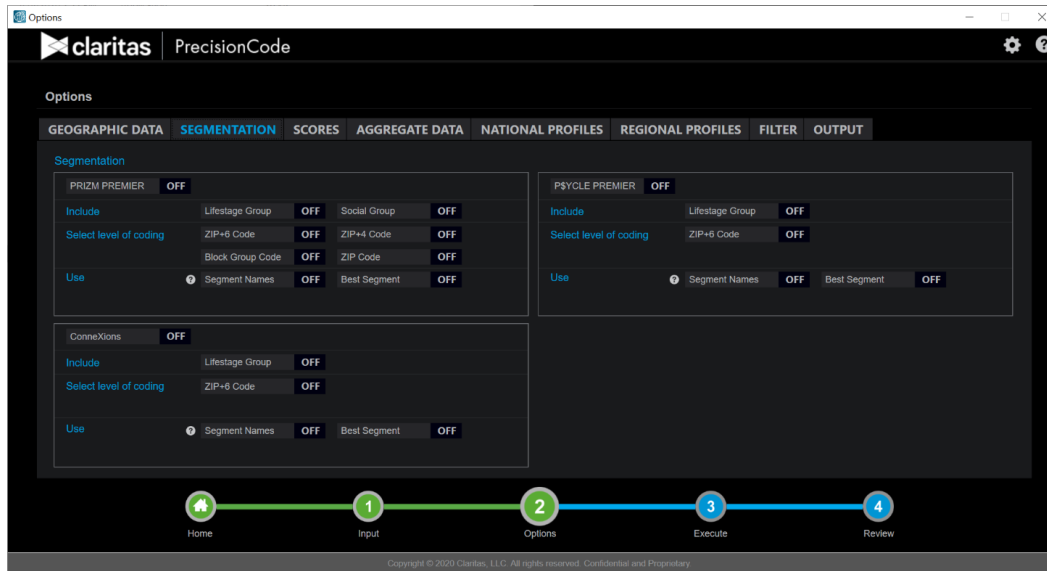
Append Segmentation Codes

Important! Perform the procedure in this section only if you have licensed segmentation data and the SEGMENTATION tab is available.

In addition to geographic codes, you can also append segmentation codes to your input file or database table.

Follow the steps in this section to append segmentation codes.

1. On the **Options** page, click the **SEGMENTATION** tab. The segmentation data that you licensed and the available options for appending segmentation codes appear.



Options screen showing the SEGMENTATION tab

2. Click the **OFF** button next to your desired segmentation data to set it to **ON**. This automatically sets the **Segment Names** and **Best Segment** options to **ON**, which means that the system automatically adds the segment names and best segments for the selected segmentation system. You can click the **ON** button to reset it to **OFF**.
3. Next to **Include**, select the group that you want and then click the **OFF** button next to the group name to set it to **ON**.
4. Next to **Select level of coding**, select the geographic levels that you want and then click the **OFF** button next to each desired geographic level to set it to **ON**.
5. Repeat **Steps 3 and 4** to select additional segmentation data and specify the group names and geographic levels that you want to include.
6. Verify your selections and then click the available tab to proceed preparing your file or database table for process.

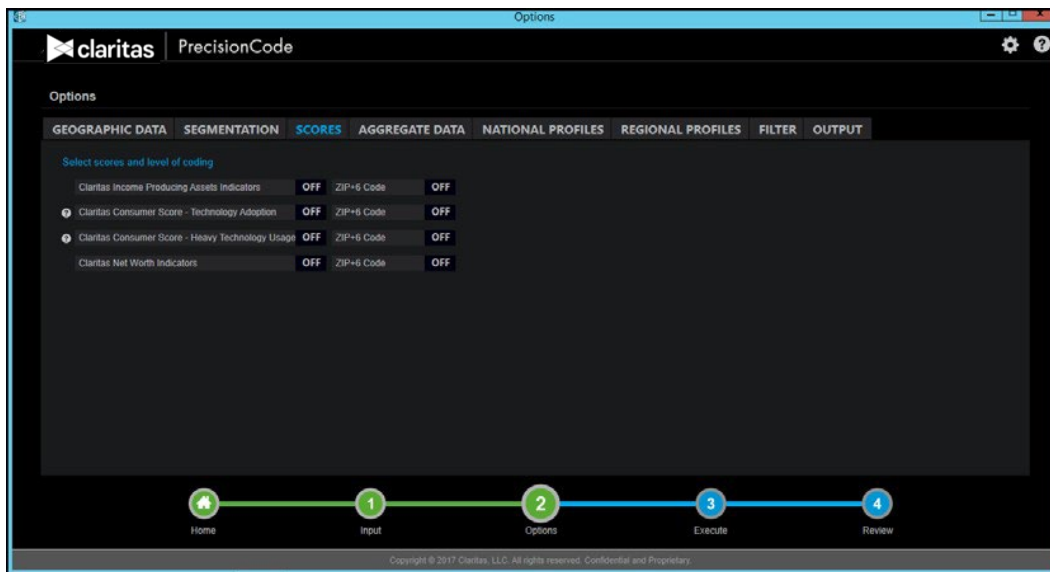
Append Claritas Scores

Important! Perform the procedure in this section only if you are licensed to a Claritas score and the SCORES tab is available.

In addition to geographic and segmentation codes, you can also append Claritas scores to your input file or database table.

Follow the steps in this section to append data scores.

1. On the **Options** page, click the **SCORES** tab. The licensed data that you can score and the available options for appending data scores appear.



Options screen showing the SCORES tab

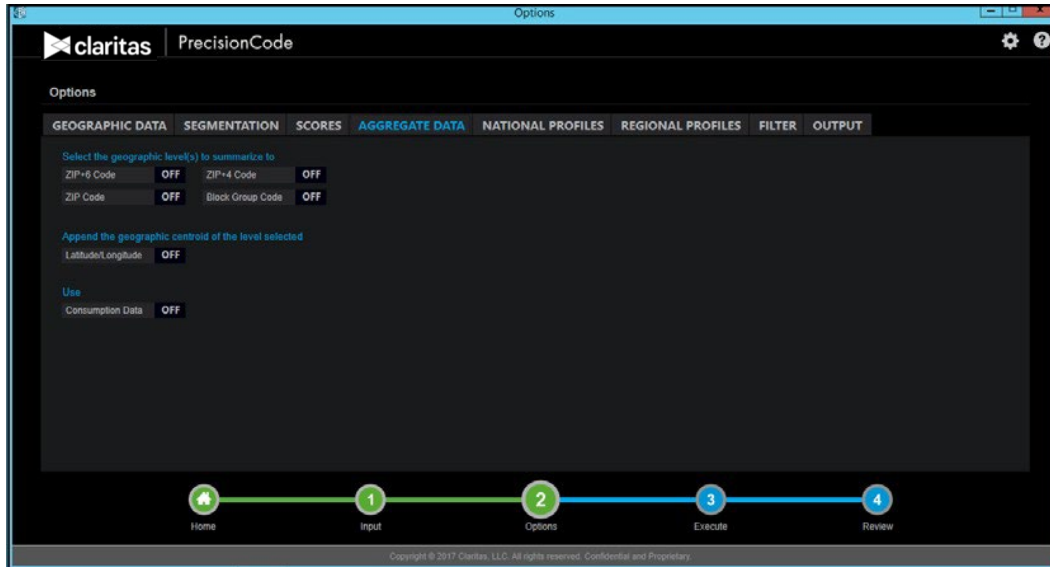
2. Below **Select scores and level of coding**, select the data for which you want to get Claritas scores, and then click the **OFF** button next to the data name to set it to **ON**.
3. To the right of the selected data, select the geographic level to include for coding and then click the **OFF** button next to the geographic level to set it to **ON**. The level options are **ZIP+6** and **ZIP+4**.
4. Repeat **Steps 2 and 3** to select more data and specify your desired geographic level to include for coding.

Verify your selections and then click the available tab to proceed preparing your file or database table for process.

Create Summary Data for a File

You can summarize data for each specific geographic level, and include the data to your output. Follow the steps in this section to create summary data.

1. On the **Options** page, click the **AGGREGATE DATA** tab. The licensed data that you can score and the available options for appending data scores appear.



Options screen showing the AGGREGATE DATA tab

2. Below **Select the geographic level(s) to summarize to**, select the geographic level for which you want to get aggregate or summary data, and then click the **OFF** button next to the desired geographic level(s) to set it to **ON**.
3. To include the latitude and longitude information to the selected geographic levels, click the **OFF** button next to Latitude/Longitude to set it to **ON**.

Note: The Latitude/Longitude option is not available to append if you selected ZIP+6 as one of the geographic levels to summarize.

4. Do either of the following:

IF	THEN
You want to get the summary data for the consumption data included in your input file	Click the OFF button next to Consumption Data to set it to ON .
Your input file does not include consumption data	The option does not appear.

5. Verify your selections and then click the available tab to proceed preparing your file or database table for process.

Create National Profiles

Important! Perform the procedure in this section only if you have licensed a segmentation data and the SEGMENTATION and SCORES tab are available.

You can set the system to automatically create National profiles for the locations in your input file or database table, and add the worksheet to your output.

Follow the steps in this section create National profiles.

1. On the **Options** page, click the **NATIONAL PROFILES** tab. The segmentation system(s) that you have licensed and additional options for generating profile worksheet appear.



Options screen showing the NATIONAL PROFILES tab

2. Below **Select the segmentation system(s)**, select the segmentation system(s) for which you want to create National profiles, and then click the **OFF** button next to the desired segmentation system(s) to set it to **ON**.
3. To generate a full profile worksheet for the selected segmentation system(s), click the **OFF** button next to **Generate Full Profile** to set it to **ON**.
4. Do either of the following:

IF	THEN
You want to use the consumption data included in your input file to generate the profiles	Click the OFF button next to Consumption Data to set it to ON .
Your input file does not include consumption data	The option does not appear.

5. Verify your selections and then click the available tab to proceed preparing your file or database table for process.

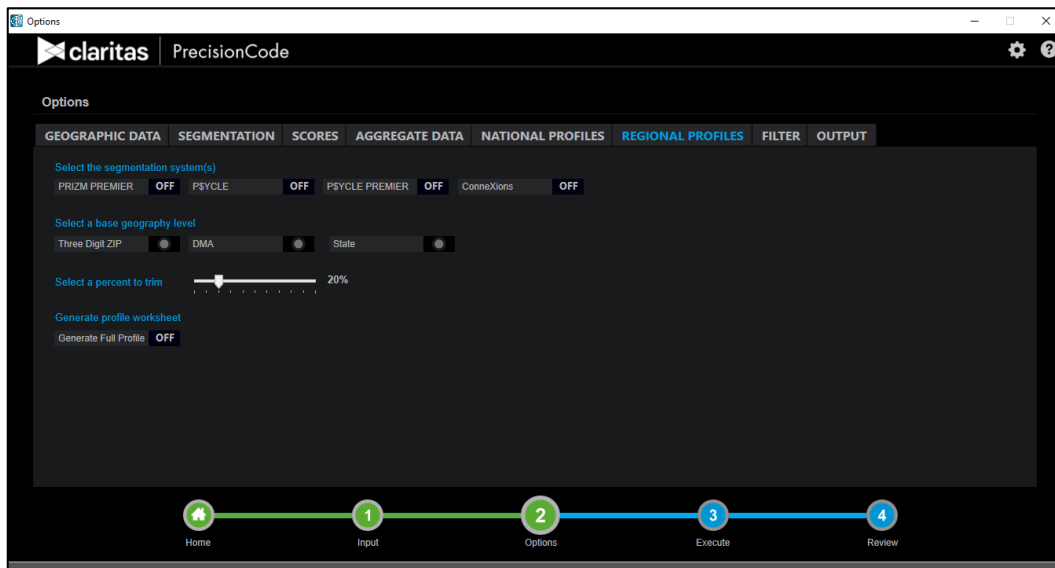
Create Regional Profiles

Important! Perform the procedure in this section only if you have licensed a segmentation data and the SEGMENTATION and SCORES tab are available.

In addition to National profiles, you can create Regional profiles for the locations in your input file or database table, and add the worksheet to your output.

Follow the steps in this section create Regional profiles.

1. On the **Options** page, click the **REGIONAL PROFILES** tab. The segmentation system(s) that you have licensed and additional options for generating profile worksheet appear.



Options screen showing the REGIONAL PROFILES tab

2. Below **Select the segmentation system(s)**, select the segmentation system(s) for which you want to create Regional profiles, and then click the **OFF** button next to the desired segmentation system(s) to set it to **ON**.
3. Below **Select a base geography level**, click to select the desired geography level that you want the system as the base geography. The system uses the base geography to determine the base market that includes the most significant records of your file and locate the bulk of your customers.
4. To the right of **Select a percent to trim**, click and drag the slider to the percentage that you want the system to remove from the original record file every time it sorts the file to determine the base market. The default value is **20%**, which means that the system will automatically include only geographies with top 80% of customer counts and %Penetration (that is, a measure indicating the percentage of a base (for example, population) that engages in a certain behavior or characteristic).

- To generate a full profile worksheet for the selected segmentation system(s), click the **OFF** button next to **Generate Full Profile** to set it to **ON**.
- Do either of the following:

IF	THEN
You want to use the consumption data included in your input file or database table to generate the profiles	Click the OFF button next to Consumption Data to set it to ON .
Your input file or database table does not include consumption data	Option does not appear

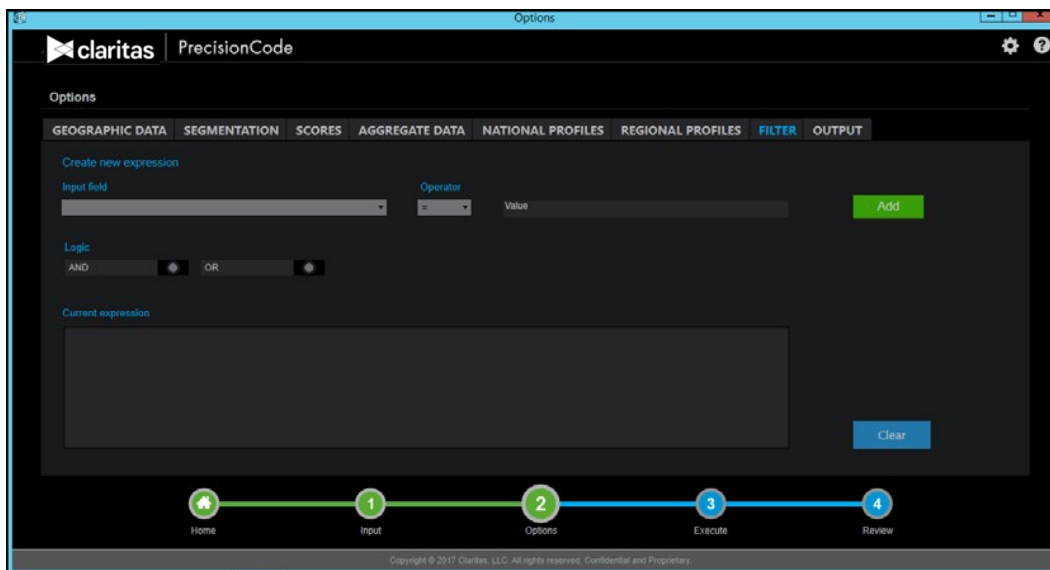
- Verify your selections and then click the available tab to proceed preparing your file or database table for process.

Filter a File or Database Table

As an option, you can filter your file or database table so that the output created by PrecisionCode contains only desired information. Filtering a file or table “screens” its data. Any data matching the screening criteria you specify is included in the output; data which does not meet the screening criteria is not included in the output.

Follow the steps in this section to filter your input file or database table.

- On the **Options** page, click the **FILTER** tab. Available options for creating filter expression appear.



Options screen showing the FILTER tab

2. In the **Create new expression** area, below **Input Field**, click the drop-down arrow and then, in the list, select the field that you want to use in your filter expression. You can select any field to filter the file.
3. Below **Operator**, click the drop-down arrow and then, in the list, select the desired arithmetic or comparison operator.
4. In the **Value** text box, type your desired value.
Note: Values can be numeric or alphabetic. For example, if you selected “ZIP Code” as the data type, you would use a numeric value, but if you selected “City,” you would use an alphabetic value.
5. Click the **Add** button. Your current expression appears in the **Current expression** box.
6. To add an additional statement to your expression, in the **Logic** box, do the following:
 - a. Below **Logic**, click to select either **AND** or **OR**, depending on whether you want to filter in data that meets both parts of the expression or data that meets either one part or the other.
 - b. Repeat Steps 2 through 5 to specify your desired data item, operator, and value for the next filter expression.
7. Repeat **Step 6** to add more statements to your expression.
8. Verify your filter expression and then click the available tab to proceed preparing your file or database table for process.

Select Output Formats

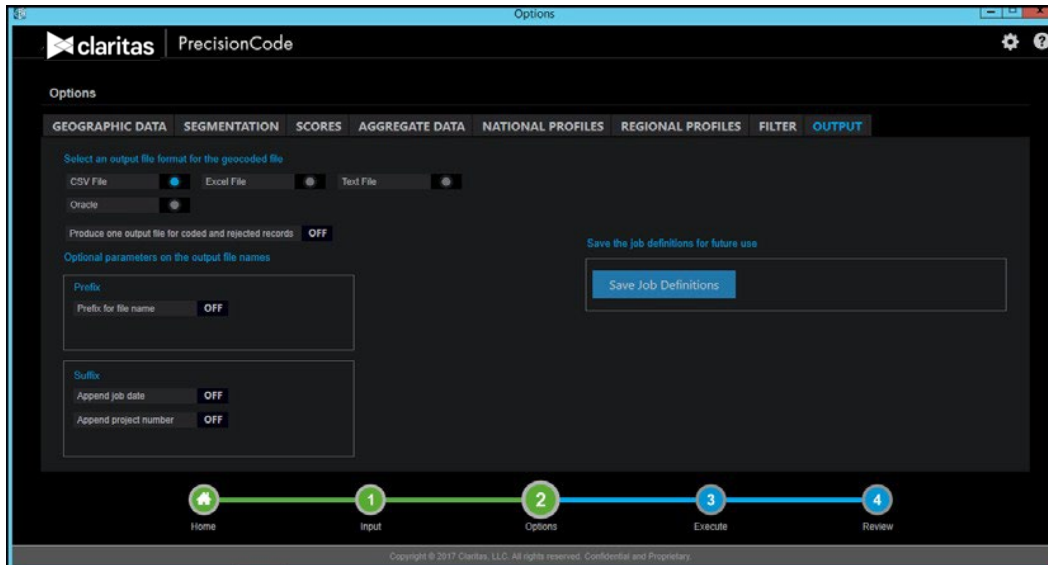
PrecisionCode can create output in any of the following formats:

- Text
- Comma Delimited (.CSV)
- Microsoft® Excel® (.XLSX)
- Oracle or SQL Server (if you have imported a database table)

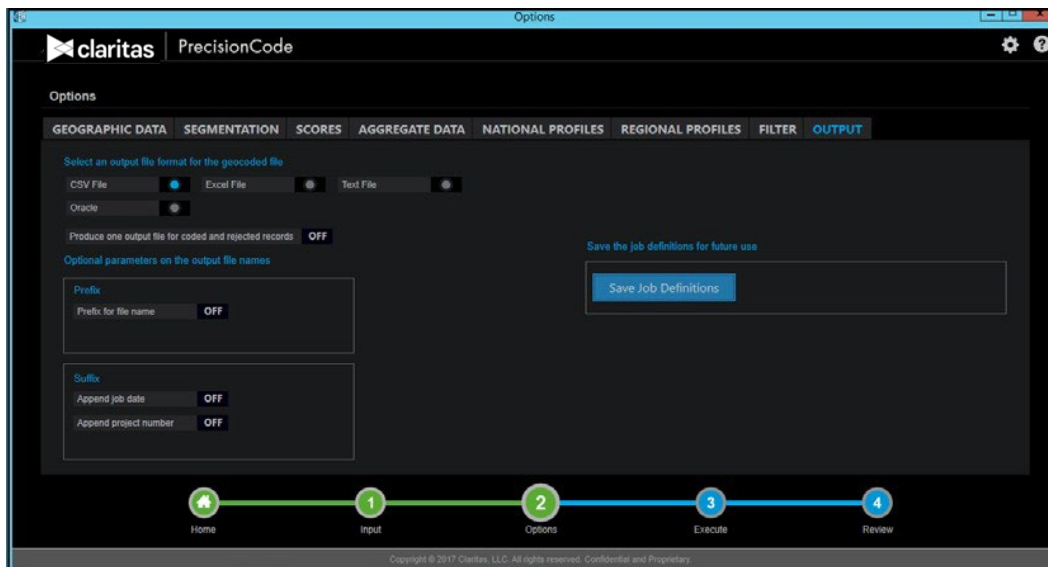
Any output file created by PrecisionCode contains the records you imported from your file or database table, and any data and/or profiles you chose to append to and created for that file or table.

Follow the steps in this section specify a filter.

1. On the **Options** page, click the **OUTPUT** tab. Available options for defining your output file appear.



Options screen showing the OUTPUT tab for processing an input file

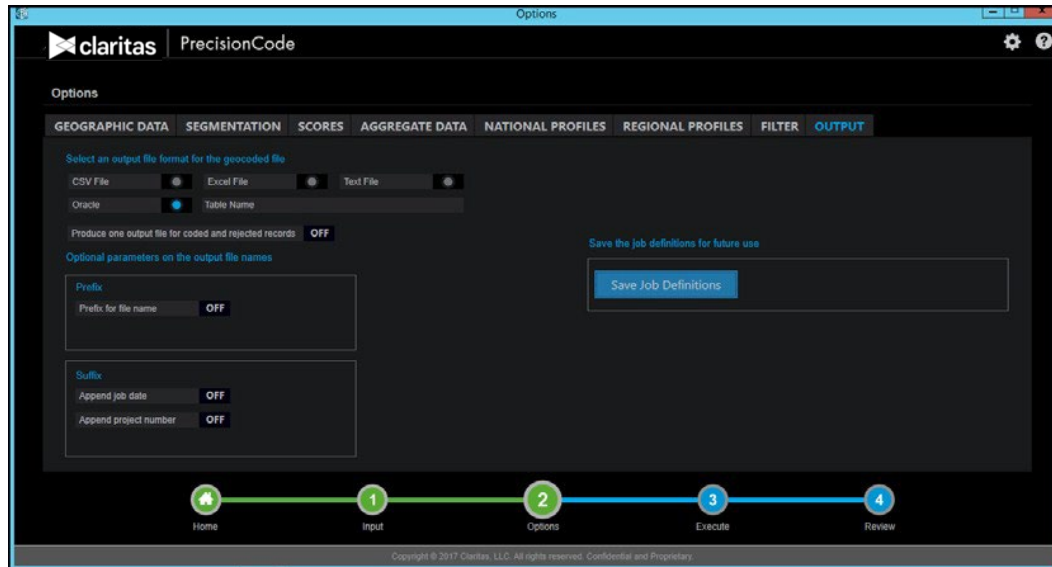


Options screen showing the OUTPUT tab for processing an input Oracle database table

2. Below **Select an output file format for the geocoded file** area, click to select your desired output format.

If you select Oracle or SQL Server as an output format for your imported database table, a Table Name text box appears to the right of the selected option. Type your desired name for the output table in the text box.

Note: The output format option available for processing a database table depends on your input database table type. For example, if your input database table type is Oracle, the system includes Oracle as an output format option; if your input database type is SQL Server, the system includes SQL Server as an output format option.



Options screen showing the Table Name text box on the OUTPUT tab

3. By default, the system creates separate files for coded and rejected records when the output is generated. To set the system to combine the coded and rejected records into one file, click the **Produce one output file for coded and rejected** records button to set it to **ON**.
4. To add a prefix to your output file name when the system creates and saves it, in the **Prefix** box, click the **OFF** button to set it to **ON**, and then type your desired prefix in the provided text box.
5. To add a suffix to your output file name when the system creates and saves it, in the **Suffix** box, do the following:
 - a. To add the job date as suffix, click the **OFF** button next to **Append job date** to set it to **ON**. This option is recommended if you want to set recurring jobs or tasks.
 - b. To add a project number as suffix, click the **OFF** button next to **Append project number** to set it to **ON**.
6. To save your job definitions, click the **Save Job Definitions** button. The Save PrecisionCode job definitions dialog box appears.
7. Navigate to and select the path where you want to save your job settings, type a file name in the **File Name** text box, and then click **Save**. A message box appears indicating that your job definitions have been saved.
8. Click **OK** to close the message box.

What's Next

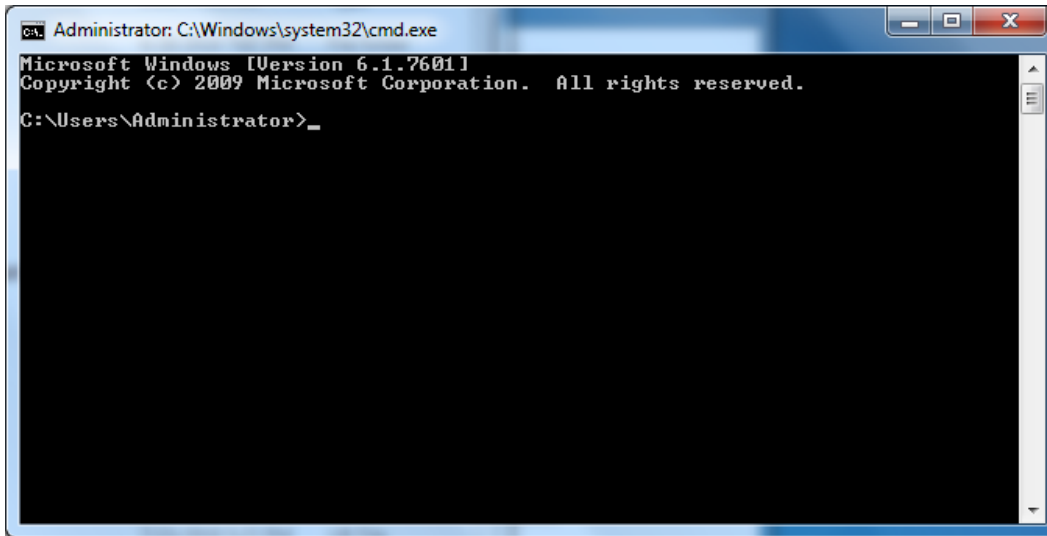
You are now ready to generate your output file. For instructions on how to generate, view, and manage output files, see the next chapter “Work with Output Files and Schedules.”

SCHEDULE A JOB THROUGH THE COMMAND LINE FUNCTION

This section will explain the process of scheduling a batch job through the command line function.

Generate Output

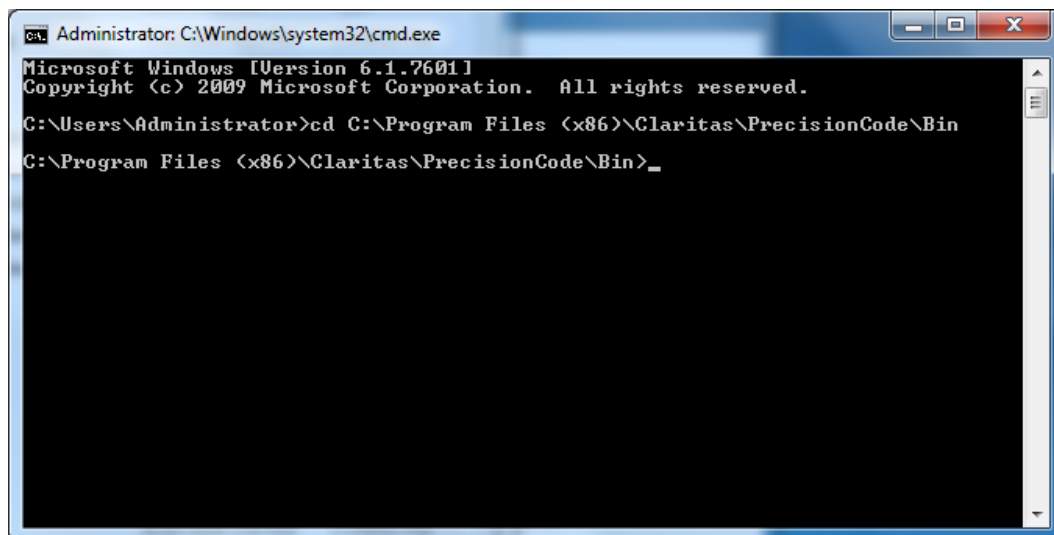
1. Open a command prompt.



Command Prompt

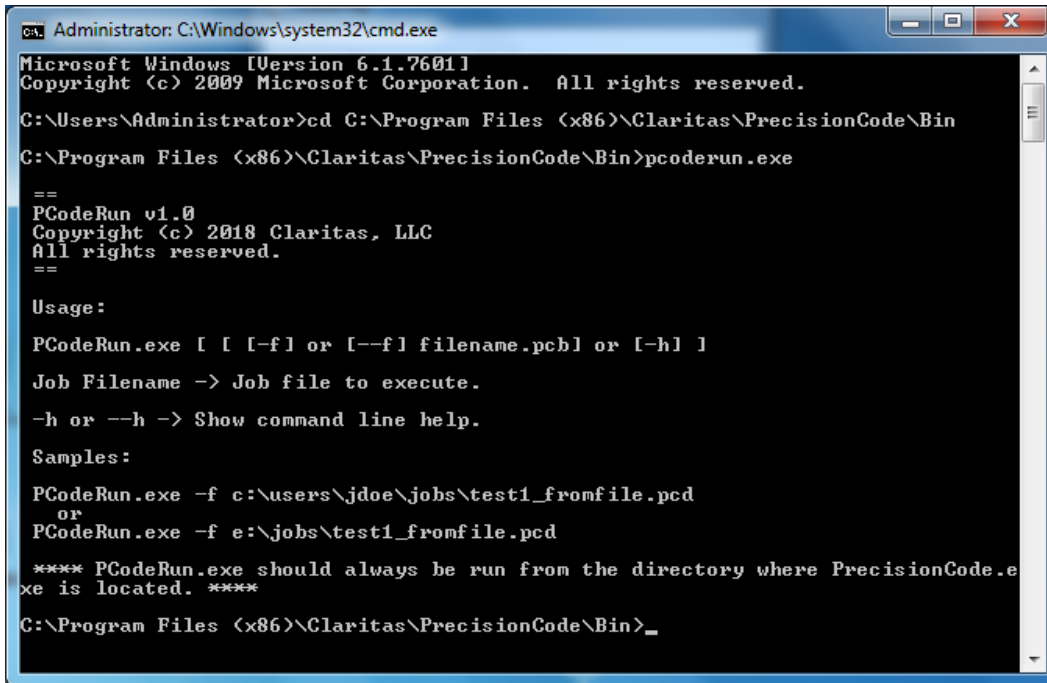
2. Go to bin directory on the PrecisoCode installation dir.

cd "C:\Program Files (x86)\Claritas\PrecisionCode\Bin"



Bin Directory

3. Execute PCodeRun.exe this will print show usage. (optional)



```
Administrator: C:\Windows\system32\cmd.exe
Microsoft Windows [Version 6.1.7601]
Copyright (c) 2009 Microsoft Corporation. All rights reserved.

C:\Users\Administrator>cd C:\Program Files (x86)\Claritas\PrecisionCode\Bin
C:\Program Files (x86)\Claritas\PrecisionCode\Bin>pcoderun.exe

==
PCodeRun v1.0
Copyright (c) 2018 Claritas, LLC
All rights reserved.
==

Usage:

PCodeRun.exe [ [ [-f] or [--f] filename.pcd] or [-h] ]
Job Filename -> Job file to execute.
-h or --h -> Show command line help.

Samples:

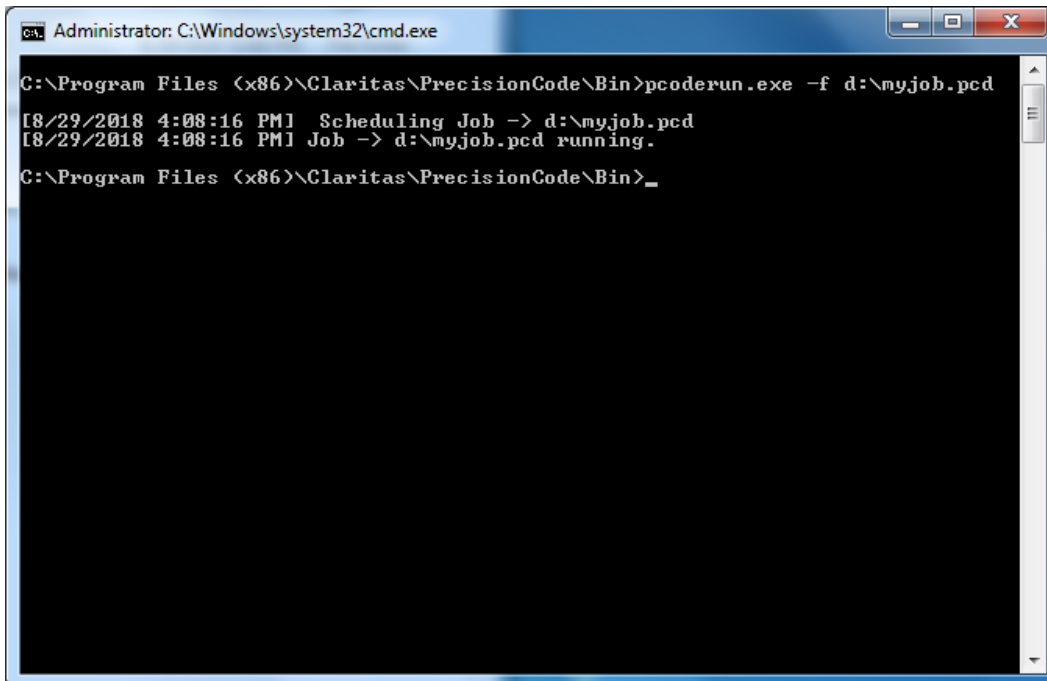
PCodeRun.exe -f c:\users\jdoe\jobs\test1_fromfile.pcd
OR
PCodeRun.exe -f e:\jobs\test1_fromfile.pcd

**** PCodeRun.exe should always be run from the directory where PrecisionCode.exe
is located. ****

C:\Program Files (x86)\Claritas\PrecisionCode\Bin>_
```

Execute PCodeRun.exe

4. To execute your job from command line just type: Pcoderun.exe -f d:\myjob.pcd



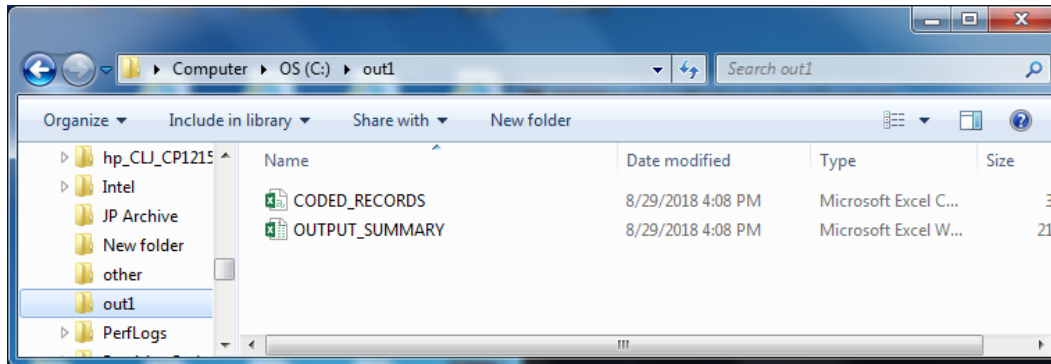
```
Administrator: C:\Windows\system32\cmd.exe

C:\Program Files (x86)\Claritas\PrecisionCode\Bin>pcoderun.exe -f d:\myjob.pcd
[8/29/2018 4:08:16 PM] Scheduling Job -> d:\myjob.pcd
[8/29/2018 4:08:16 PM] Job -> d:\myjob.pcd running.

C:\Program Files (x86)\Claritas\PrecisionCode\Bin>_
```

Execute job from command line

5. When the job finishes results will show up as usual in the output folder.



Output Folder

WORK WITH OUTPUT FILES AND SCHEDULES

After you have worked on your input file, you are now ready to process it. This chapter contains information on how to generate and view the file, as well as manage your scheduled output generation task(s).

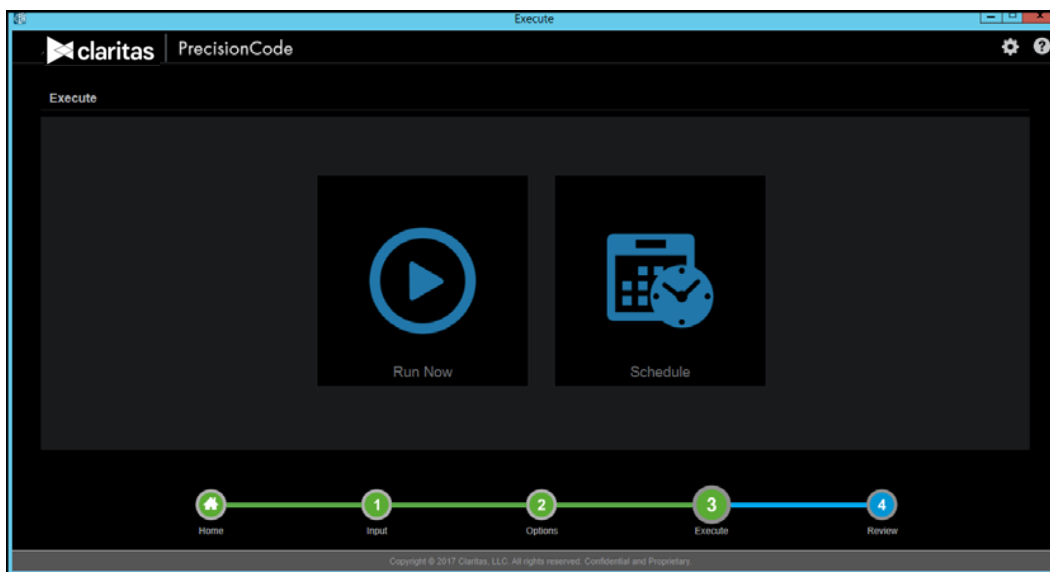
Generate Output

Once you have selected processing options for your file, you can process your file to generate output. During file processing:

- Any data and/or Claritas scores you chose to append is added.
- Any summary output and profiles you selected are created.
- Output files containing the original records from your imported file or table plus any appended data are created in the formats you have selected.

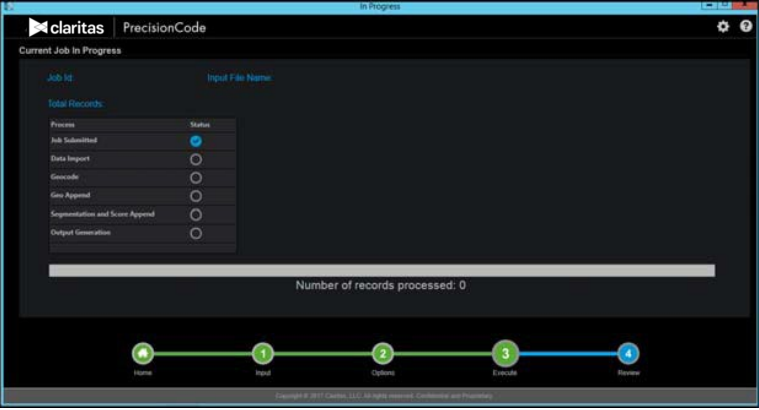
Follow the steps in this section to generate output.

1. Locate the sequence bar on the currently displayed screen, and then click number **3 - Execute** in the sequence bar. The Execute screen appears showing the available output generation options.



Execute screen

2. Perform either of the following:

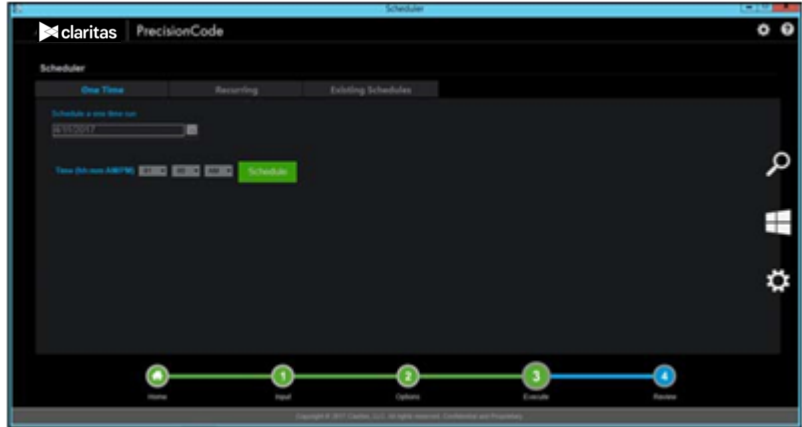
TO GENERATE THE OUTPUT	DO THE FOLLOWING
<p>Immediately or now</p>	<p>a. Click Run Now. A message box appears indicating that a new job has been scheduled.</p> <p>b. Click the OK button. The Current Job In Progress screen appears.</p>  <p>Current Job In Progress screen</p> <p>When the job completes, a Review button appears below the progress bar.</p>

TO GENERATE THE OUTPUT

DO THE FOLLOWING

Later and schedule output generation jobs

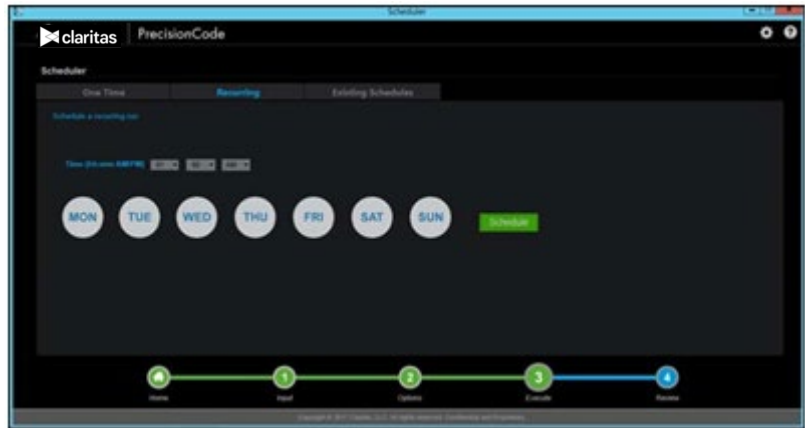
a. Click **Scheduler**. The Scheduler screen appears.



Scheduler screen showing the One Time tab

b. To schedule a one-time output generation process, on the **One Time** tab, specify the date and time that you want the task to run, and then click **Schedule**.

To schedule the output generation process as a recurring task, click the **Recurring** tab and then specify the time and day(s) of the week you want the system to run the task, and then click **Schedule**.



Scheduler screen showing the Recurring tab

A message box appears indicating that a schedule has been created and all schedules can be maintained on the Existing Schedules tab.

c. Click the **OK** button to close the message box.

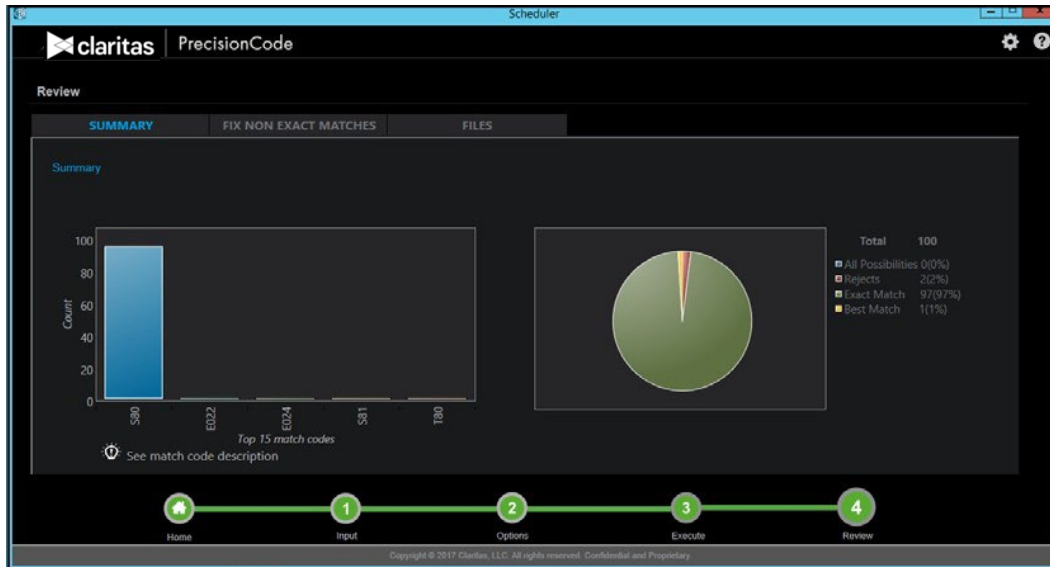
View the Generated Output

You can view the generated output on the Review screen.

- Depending on your specified report output generation settings, perform either of the following:

IF YOU HAVE SELECTED TO...	THEN DO THIS...
Generate the output immediately after importing and working on the file	Click the Review button.
Schedule the output generation task	<ol style="list-style-type: none"> On the Scheduler screen, click the Existing Schedules tab. A list of scheduled tasks appears with the latest task performed or schedule on top of the list. <div data-bbox="704 737 1430 1125" data-label="Image"> </div> Check the status of the desired scheduled task. If it has been completed, click the task, and then click Review on the progress bar.

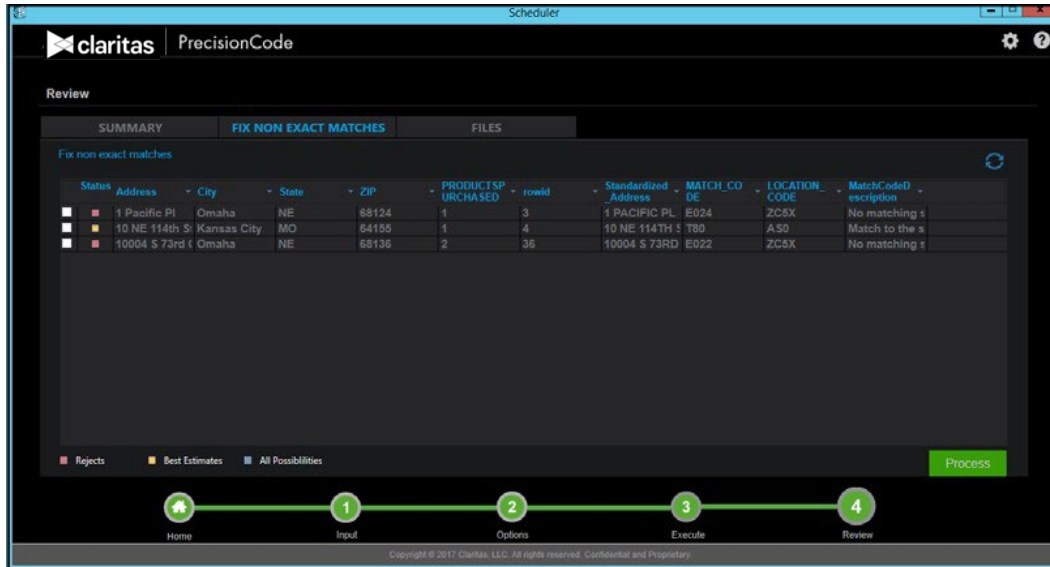
The Review screen appears showing the generated output.



Review screen showing the SUMMARY tab

2. By default, the SUMMARY tab appears showing the top 15 match codes. Summary data is presented in bar graphs (counts vs. match codes) and pie charts. To understand the graph and view the match code descriptions, perform the following:
 - a. Click the See match code description link. Descriptions for each match code on the graph appear and the count for each match code.
 - b. Click the close button (that is, button with an x mark located in the upper-right corner of the screen) to close the descriptions.

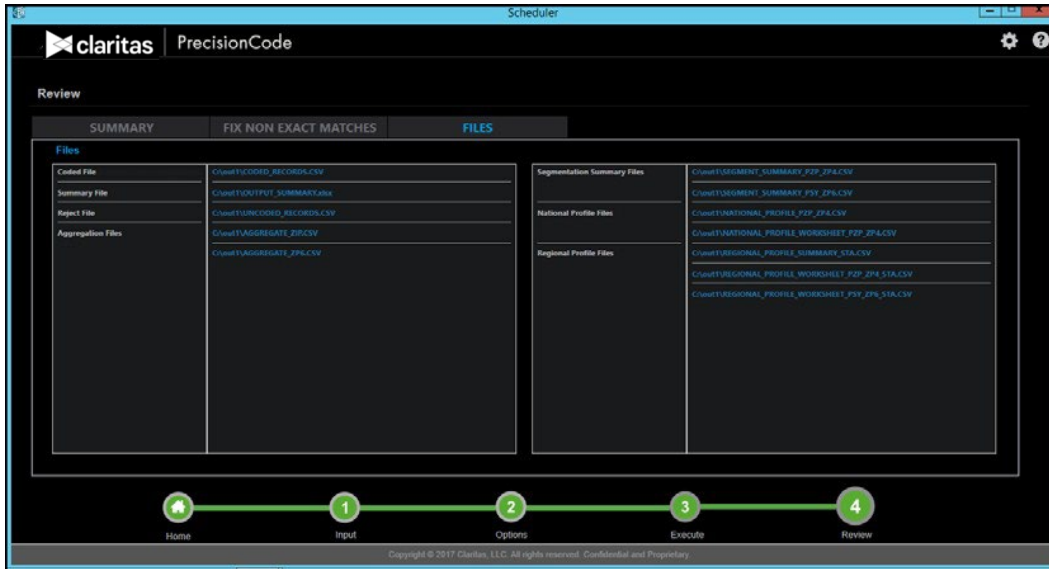
3. To view and fix the non-matched records, perform the following:
 - a. Click the FIX NON-EXACT MATCHES tab. A list of non-matched records appears.



Review screen showing the FIX NON-EXACT MATCHES tab

- b. Click to select the check boxes of the records that you want to fix, and then click the **Process** button. The system re-processes the selected records to find the possible or best matches.
- c. To refresh the list, click the **refresh** button (that is, the button with circular arrows) located in the upper-right corner of the table.

4. To view the generated output files, perform the following:
 - a. Click the **FILES** tab. A list of generated output files with hyperlinked paths appears.



Review screen showing the FILES tab

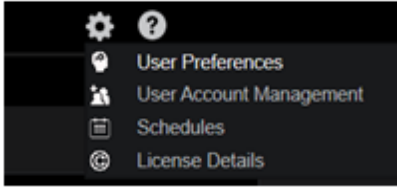
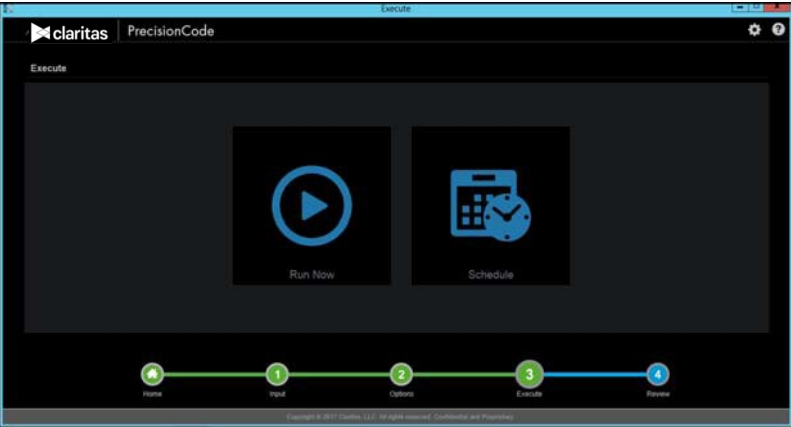
- b. To view each file, double-click the hyperlinked path next to the file that you want to open. The file appears on the screen.
 Note: File enhancement match codes and definitions are available [here](#).
 - c. Close the file after viewing the information. The system returns you to the **FILES** tab so you can select a new file to view.
5. Click the **Home** icon on the progress bar to return to the Claritas PrecisionCode home screen or click the number icon that corresponds to the step that you want to review.

Manage Job Schedules

You can view the status of your job schedules, set a new schedule, stop in progress jobs, or delete a scheduled job on the Scheduler screen.

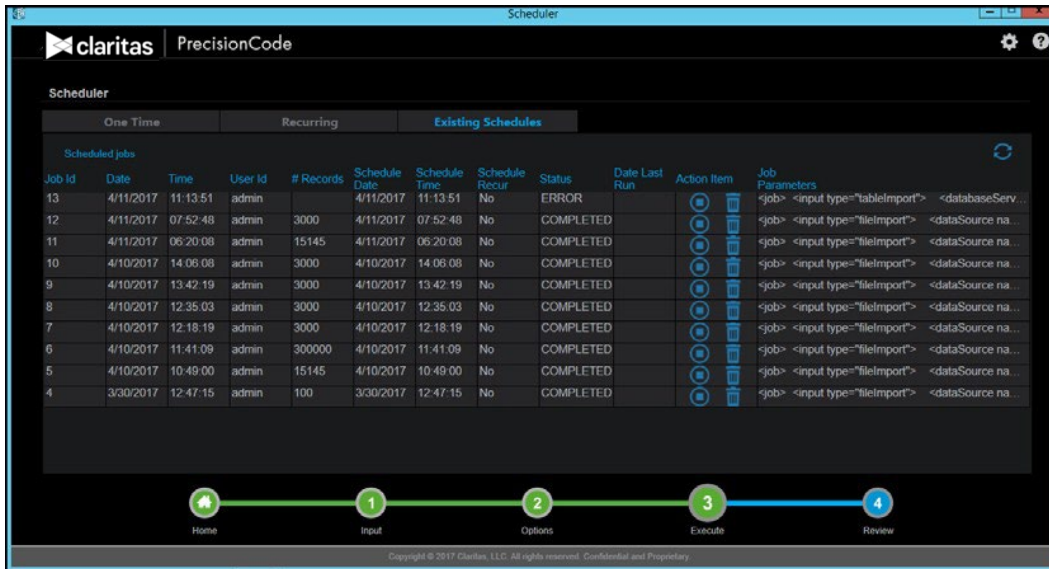
Access the Scheduler Screen

Perform either of the following to access the Scheduler screen:

TO ACCESS THE SCHEDULER SCREEN VIA...	CLICK...
<p>The PrecisionCode main screen</p>	<p>The System Tools icon in the upper-right corner of the screen and then click Schedules.</p>  <p>System Tools icon and options</p> <p>The Scheduler screen appears showing a list of schedules on the Existing Schedules tab.</p>
<p>The Execute screen during the file import process</p>	<p>Schedule</p>  <p>Execute screen showing the Run Now and Execute options</p> <p>The Scheduler screen appears showing the One-Time tab as default tab.</p>

View the Status of Scheduled Jobs

1. On the **Scheduler** screen, make sure that you are on the **Existing Schedules** tab.



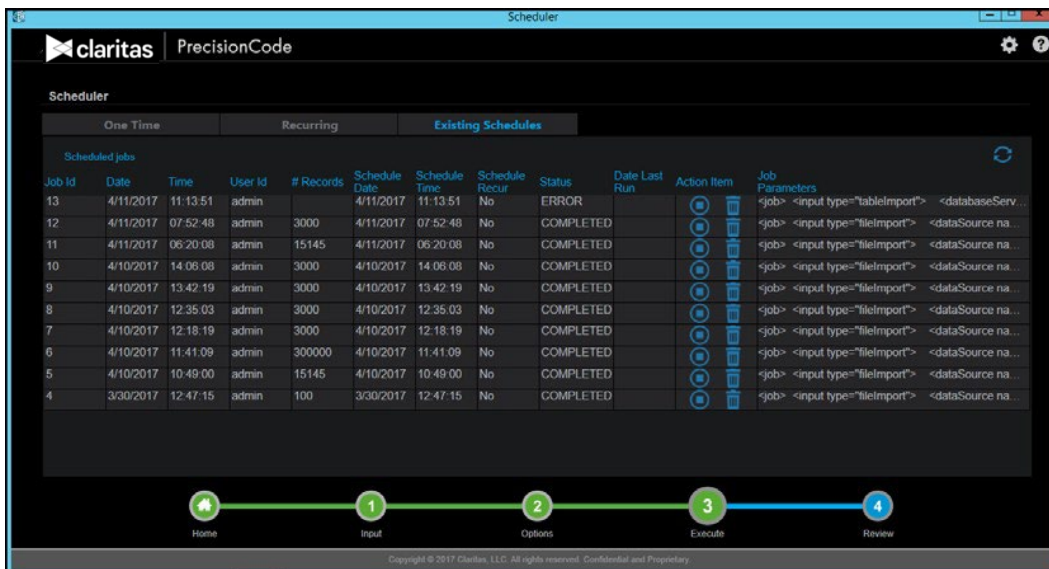
Scheduler screen showing the Existing Schedules tab

2. Locate the **Status** column. The column shows the status of each scheduled job. Status can be **IN PROGRESS**, **ERROR**, or **COMPLETED**.

Stop a Scheduled Job

Important! You can only stop a job that is currently in progress or with IN PROGRESS status.

1. On the **Scheduler** screen, make sure that you are on the **Existing Schedules** tab.



Scheduler screen showing the Existing Schedules tab

2. Locate the job that you want to stop.
3. Locate the **Action Item** column and then click the provided **Stop** button for your selected job. A message box appears specifying that your selected job and all jobs currently in progress will be stopped.

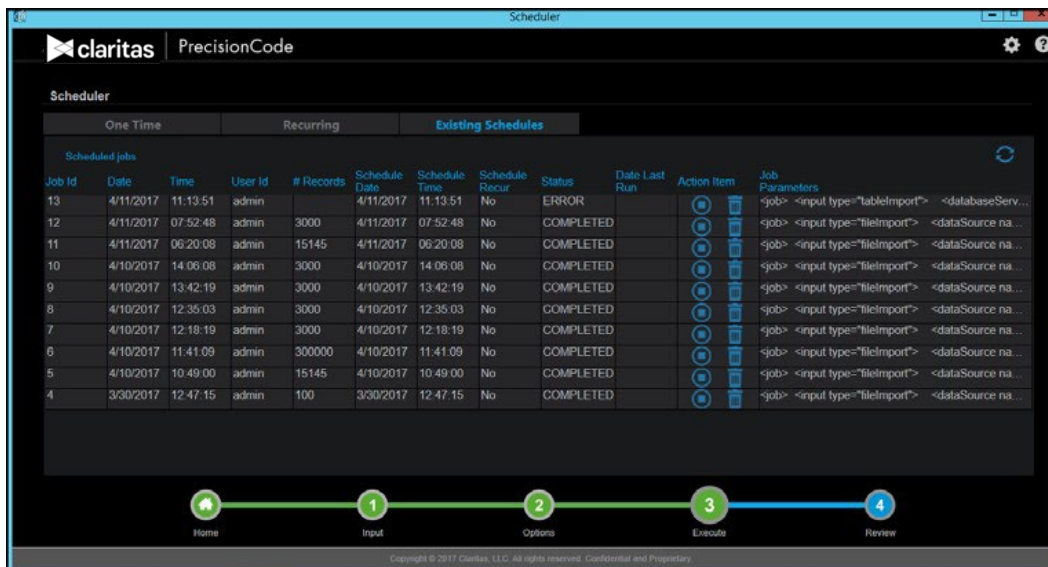


Stop button

4. Click **OK** to confirm the action and close the message box. The system stops the processing all jobs in progress.

Delete a Scheduled Job

1. On the **Scheduler** screen, make sure that you are on the **Existing Schedules** tab.



Scheduler screen showing the Existing Schedules tab

2. Locate the job that you want to delete.
3. Locate the **Action Item** column and then click the provided Delete button for your selected recurring job. A message box appears specifying that your selected job will be deleted.



Delete button

4. Click **OK** to confirm the action and close the message box. The system deletes your selected recurring job and removes the job from the Existing Schedules list. This also permanently removes and stops the recurring job from future occurrence.

What's Next

To change your system settings, follow the procedures in the “Customize Your System” chapter.

To import another file or database table, follow the procedures in the “Import Files and Database Tables” chapter.

To work on imported files for process, follow the procedures in the “Work with Imported Files and Database Tables” chapter.

To look up information about a location, follow the procedures in the “Look up Information for a Location” chapter.

LOOK UP INFORMATION FOR A LOCATION

This chapter explains how to look up geographic codes and/or segmentation codes, including data scores, for an address/intersection, a ZIP/ZIP+4 Code, a combination of longitude and latitude coordinates, or a block group. You can use this information to show and analyze the location of customers, prospects, competitors, or retail locations.

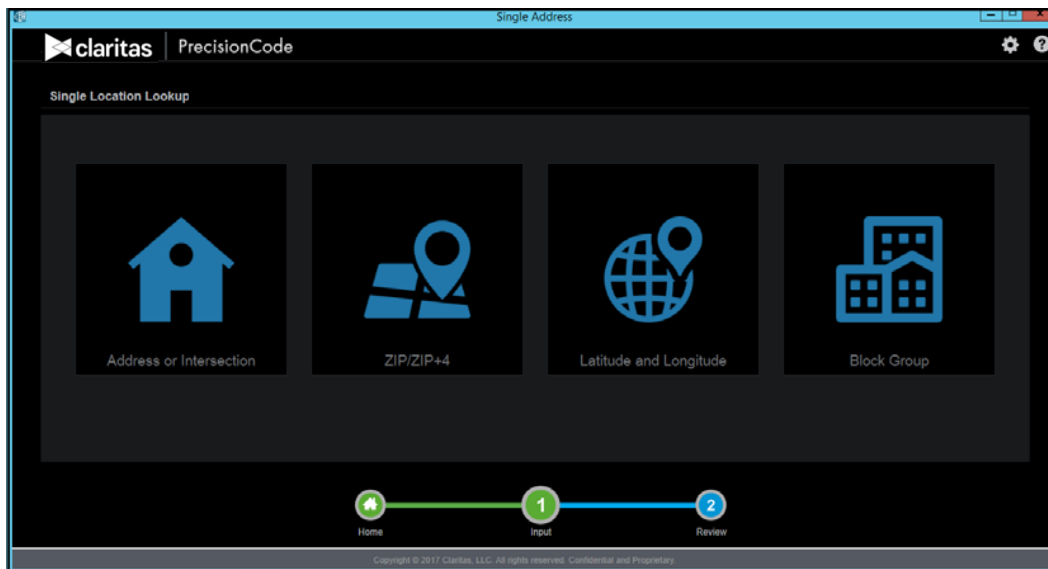
After you retrieve the geographic codes and/or segmentation codes, you can copy and paste the information to another application, or print it so you can share it with other users or save it for future use.

To look up information for a location, you need to do the following procedures:

1. Access the Single Location Lookup screen.
2. Specify the lookup method for the type of location data that you have, and then the location data.
3. Process and view the geographic and/or segmentation data for your specified location.
4. Copy or print the location information.

Access the Single Location Lookup Screen

You can begin your search for location information via the Single Location Lookup screen. To access the screen, click the **Single Location Lookup** icon on the Claritas PrecisionCode main screen.




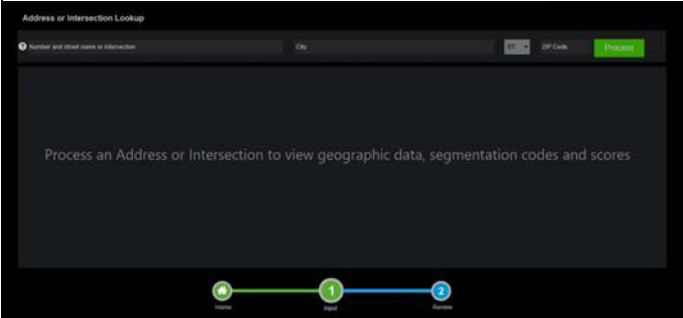
Single Location Lookup screen


Notice that a sequence or progress bar appears below the icons. The application highlights the current step that you are in by showing the number in green. You can use this bar to navigate back

to the Home or main screen, or move to the next step. However, you must complete the actions required for the current step in order to move to the next step.

Specify a Lookup Method for Your Location Data

Your lookup method depends on the type of location data that you have. To select the appropriate lookup method for your location data, refer to the table below.

IF YOU HAVE	CLICK	AND THEN DO THE FOLLOWING
An address or intersection	 <p>Address or Intersection</p>	<p>a. Make sure that your address or intersection includes any of the following combinations:</p> <div data-bbox="776 575 1411 926" style="border: 1px solid black; padding: 5px;"> <p>An address may consist of any of the following:</p> <ol style="list-style-type: none"> 1) P.O.Box, City, State, ZIP Code (PO Box 2782 ,Kensington, MD 20891) 2) Address Suite#, City, State, ZIP Code (200 W. Jackson Blvd Suite#250, Chicago, IL 60606) 3) Address, City, State, ZIP Code (200 W. Jackson Blvd #2500, Chicago, IL 60606) 4) Address, ZIP Code (200 W. Jackson Blvd #2500, 60606) 5) Address, City, State (200 W. Jackson Blvd #2500, Chicago, IL) <p>An Intersection can be in either the following combinations</p> <ol style="list-style-type: none"> 1) Intersection, ZIP Code (Wells & Jackson Blvd, 60606) 2) Intersection, City, State (Wells & Jackson Blvd, Chicago, IL) <p>The intersection can be separated by 'at', '@', '&', and 'and'.</p> </div> <p style="text-align: center;">Address/Intersection formats</p> <ol style="list-style-type: none"> b. On the Address or Intersection Lookup screen, in the first text box, type the number and street name or the intersecting street names. c. In the City text box, type the city where the address or intersection is located. d. Click the ST drop-down arrow and select the state where the address or intersection is located. e. In the ZIP Code text box, type the ZIP Code of the address or intersection. <div data-bbox="764 1257 1443 1572" style="border: 1px solid black; padding: 5px;">  <p style="text-align: center;">Address or Intersection Lookup screen</p> </div>

IF YOU HAVE	CLICK	AND THEN DO THE FOLLOWING
A ZIP or ZIP+4 Code	 <p>ZIP/ZIP+4</p>	<p>On the ZIP/ZIP+4 Lookup screen, type the ZIP or ZIP+4 Code of the location in the ZIP/ZIP+4 Code text box.</p>  <p>ZIP/ZIP+4 Lookup screen</p>
Latitude and longitude coordinates	 <p>Latitude and Longitude</p>	<p>On the Latitude and Longitude Lookup screen, type the latitude and longitude coordinates in the Latitude and Longitude text boxes, accordingly.</p>  <p>Latitude and Longitude Lookup screen</p>
A block group	 <p>Block Group</p>	<p>On the Block Group Lookup screen, type the block group in the Block Group text box.</p>  <p>Block Group Lookup screen</p>

Process and View Lookup Information

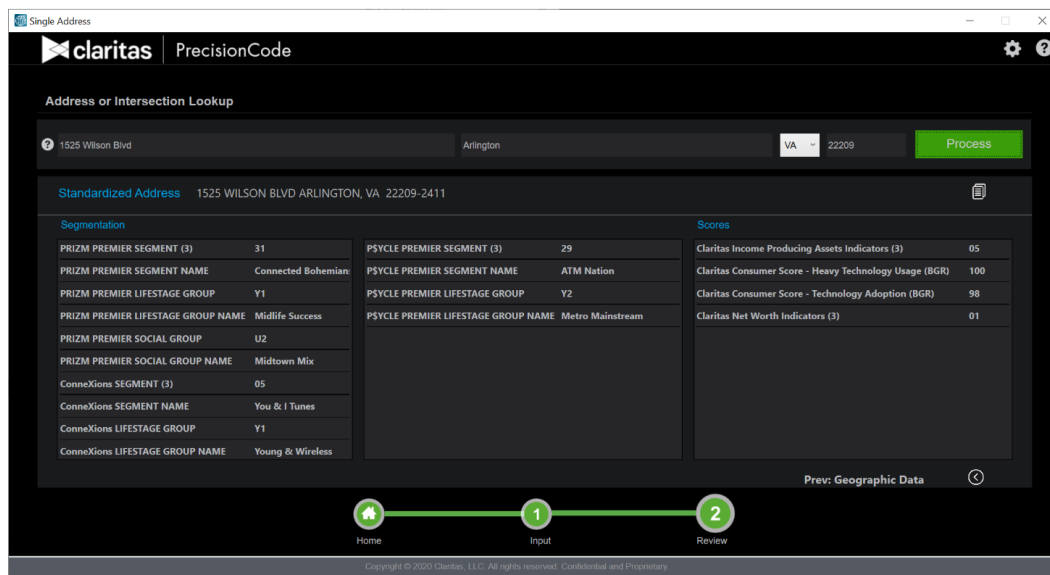
1. After you specified your location, click the **Process** button. The geographic data for the location appears on the screen.



Geographic Codes Screen with Sample Information

2. **If you have not licensed Claritas segmentation data**, the system retrieves only the geographic information for your specified location.

If you have licensed Claritas segmentation data, in addition to the geographic information, the system retrieves the appropriate segmentation codes and data scores for your specified location, and then displays a “Next: Segmentation” text and arrow buttons in the lower-right corner of the Geographic Codes screen. **To view the segmentation codes and data scores**, click the right arrow button.



Segmentation Codes Screen with Sample Information and Scores

3. To return to the Geographic Codes screen, click the left arrow button.

Copy and Paste Lookup Information

You can copy and paste the retrieved information to another application.

1. To copy, click the **Copy** button located on the upper-right corner of the screen. This copies the information to the clipboard.



Copy icon

2. To paste, open the application that you want to use (for example, Microsoft® Word® or Notepad) and then paste the information.
3. To save the information, click the available **Save** icon or **Save** command in the opened application.

Print Lookup Information

To print the lookup information, simply click the Print icon on the upper-right corner of the screen and follow the printing instructions/prompts.



Print icon

What's Next

To change your system settings, follow the procedures in the “Customize Your System” chapter.

To import another file or database table, follow the procedures in the “Import Files and Database Tables” chapter.

To work on imported files for process, follow the procedures in the “Work with Imported Files and Database Tables” chapter.

To work on output files and task schedules, follow the procedures in the “Work with Output Files and Schedules” chapter.

TECHNICAL SUPPORT

If you need further assistance, not provided in the user guide, please contact the Claritas Solution Center between 9:00 a.m. and 8:00 p.m. (Monday through Friday, EST) at 800.866.6511.

LEGAL NOTIFICATIONS

ConneXions, P\$YCLE and PRIZM are registered trademarks of Claritas, LLC. The DMA data are proprietary to The Nielsen Company (US), LLC (“Nielsen”), a Third-Party Licensor, and consist of the boundaries of Nielsen’s DMA regions within the United States of America. Other company names and product names are trademarks or registered trademarks of their respective companies and are hereby acknowledged.

This documentation contains proprietary information of Claritas. Publication, disclosure, copying, or distribution of this document or any of its contents is prohibited, unless consent has been obtained from Claritas.

Some of the data in this document is for illustrative purposes only and may not contain or reflect the actual data and/or information provided by Claritas to its clients.