EU-CIRCLE

A pan-European framework for strengthening Critical Infrastructure resilience to climate change

D7.9 Administration and User Manuals for SimICI System final

Contractual Delivery Date: 04/2018	**Actual Delivery Date**: 10/2018
Type: Report	**Version**: v1.0

Dissemination Level [Public] Deliverable

Statement

This deliverable describes the administrative and user manual for the SimICI system. It presents the way that CIRP and flooding models should be operated by the users.

© Copyright by the **EU-CIRCLE** consortium, 2015-2018

EU-CIRCLE is a project that has received funding from the European Union's Horizon 2020 research and innovation programme under grant agreement No 653824. Please see http://www.eu-circle.eu/for more information.

⚠ DISCLAIMER: This document contains material, which is the copyright of EU-CIRCLE consortium members and the European Commission, and may not be reproduced or copied without permission, except as mandated by the European Commission Grant Agreement no. 653824 for reviewing and dissemination purposes.

The information contained in this document is provided by the copyright holders "as is" and any express or implied warranties, including, but not limited to, the implied warranties of merchantability and fitness for a particular purpose are disclaimed. In no event shall the members of the EU-CIRCLE collaboration, including the copyright holders, or the European Commission be liable for any direct, indirect, incidental, special, exemplary, or consequential damages (including, but not limited to, procurement of substitute goods or services; loss of use, data, or profits; or business interruption) however caused and on any theory of liability, whether in contract, strict liability, or tort (including negligence or otherwise) arising in any way out of the use of the information contained in this document, even if advised of the possibility of such damage.

Preparation Slip			
	Name	Partner	Date
From	A.S. Chen / A. Kostaridis	UNEXE/STWS	10/2018
Reviewer			
Reviewer			
For delivery	A. Sfetsos	NCSRD	09/2018

Document Log			
Issue	Date	Comment	Author / Organization
V0.1	08/2018	ToC	UNEXE
V0.2	10/2018	CIRP Admin and User Manual for SIMICI	STWS
		List here the changes and their rational for each release	

Executive Summary

The EU-CIRCLE climate hazard mapping and impact assessment are integrated in the Critical Infrastructure Resilience Platform (CIRP) for end-users applications and outcome visualisation. Deliverable D7.9 summarises the integration and demonstrates the functions of CIRP via the Virtual Data Set described in D7.4.

Contents

1 Introduction

Using CIRP, the users can select the scenarios and dataset for carrying out hazard modelling and data analyses via the built-in map. The user friendly interface allows users to browse and define the study areas, amending the input data to execute various analyses. The high performance simulators such as CADDIES 2D for flood simulation are embedded in CIRP. Users can easily prepare the input data to execute simulations, and visualise modelling results.

The outcomes can be further utilised by other data processing modules to provide in-depth analyses. For example, the hazard information can be combined with fragility/vulnerability functions to determine the risk of critical infrastructure to climate disasters.

2 EU-CIRCLE Critical Infrastructure Resilience Platform

Within EU-CIRCLE, the Critical Infrastructure Resilience Platform (CIRP) was developed to integrate various climate hazard modelling and impact assessment methodologies to demonstrate and visualise the analysing results to the stakeholders. The following section describes the environment setup, section 2.2 describes the administration framework and section 2.3 presents the user guide (the execution of a selected set of CIRP analyses using the SIMICI dataset of the Virtual City).

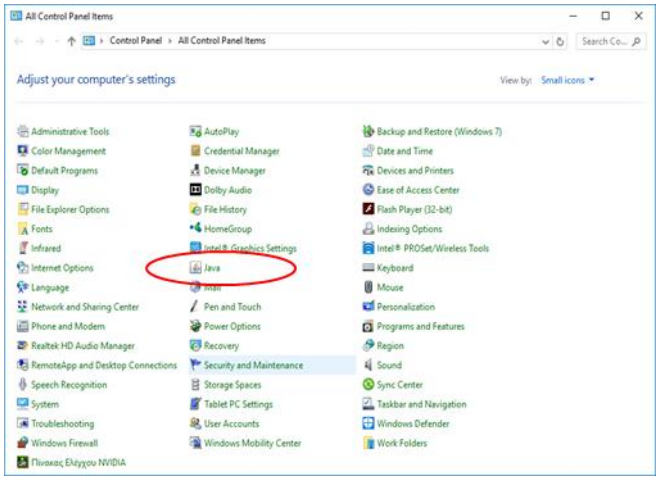
2.1 Environment Setup

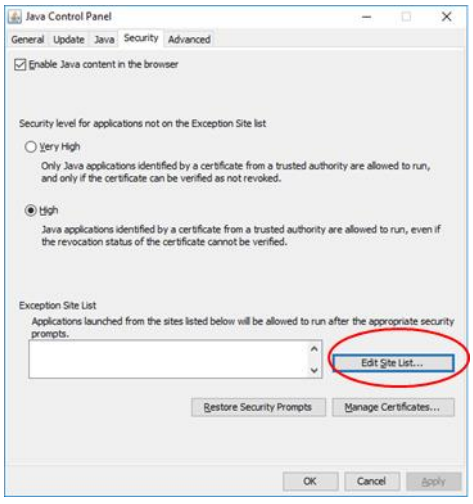
CIRP is a multi-user enterprise software accessible over the Web as a Rich Client software application (Web Start Technology). The following prerequisites are required:

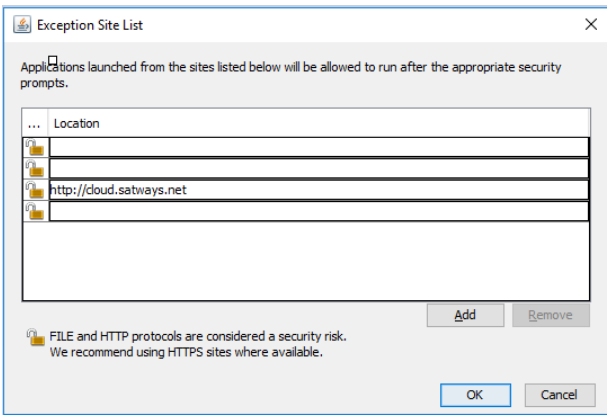
- Windows OS is only supported
- Java Runtime 1.8.0_121 (32 or 64 bit) or later
- An R Interpreter

Configuration Steps:

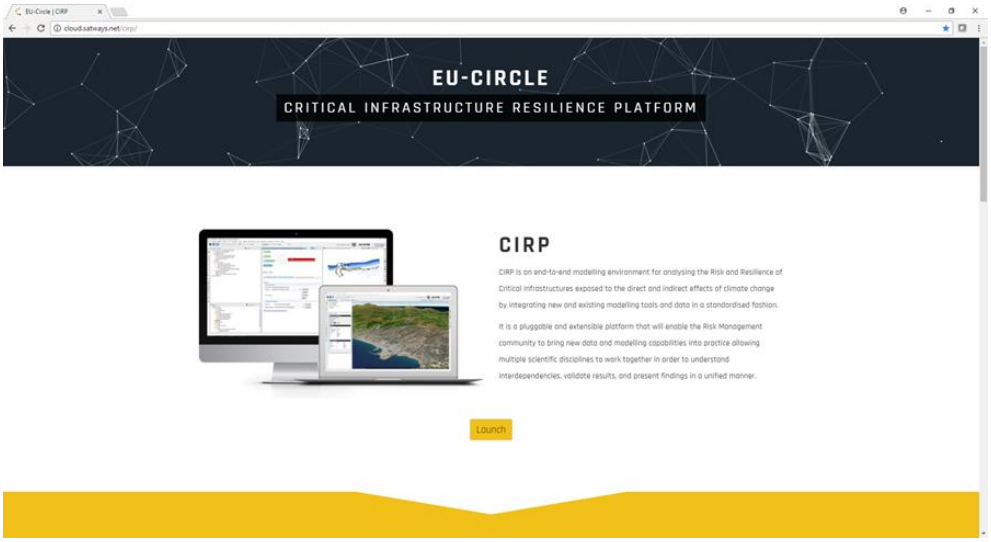
1. Install the Java Runtime environment (if not exist in your system)
2. Open the Windows Control Panel and click the Java icon
3. The Java Control Panel Window will popup. Click the Security Tab and press the “Edit Site List…” button.
4. Add the following entry: “http://cloud.satways.net” (see screenshot below) and press OK



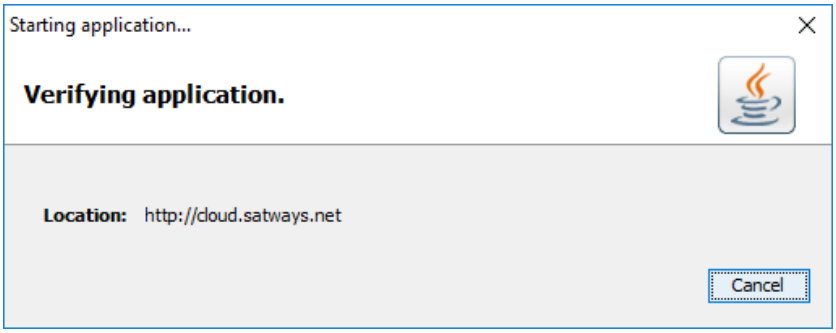




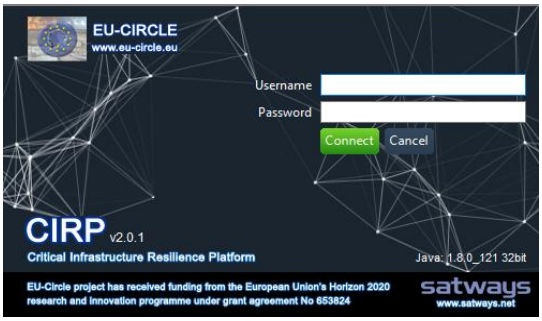
5. Finally make sure that the following ports are not blocked by your organization's network: 1098, 1099, 3873, 4447, 8080 and 61616.
6. Navigate to the URL and download the latest R version **https://cran.r-project.org/bin/windows/base/**
7. Installation path is: C:\Program Files\R\R-3.X.X (depending on the version)
8. Open your Windows Explorer and navigate to the installation path
9. Run the R.exe located in bin folder of R installation as Administrator and type the following command: install.packages('extRemes')
10. Once the prerequisites are installed you can visit the following webpage: **http://cloud.satways.net/cirp**
11. Press the Launch button
12. A file named cirp.jnlp will be downloaded. Double click it.



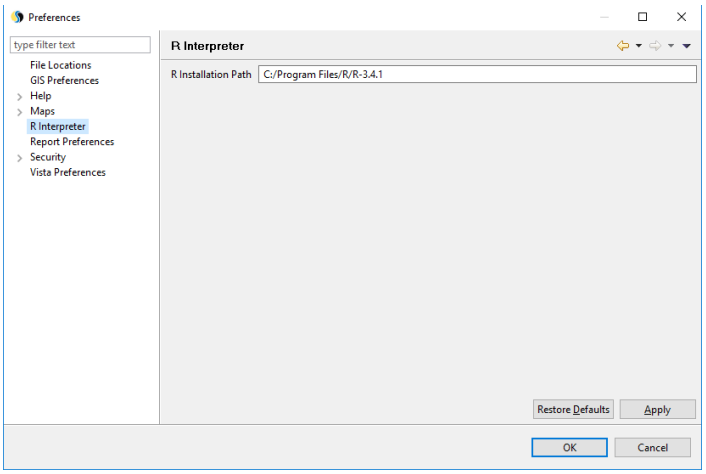
13. The very first time the CIRP application will start the download process. A similar window will appear with a progress bar indicating the download progress.
14. Please note that the initial application download takes place only once



15. Once download is completed the Security Warning window will appear.
16. Press “I accept the risk and want to run this application”
17. After a while login screen will be shown
18. Fill in the user credentials provided by the Administrator (STWS) and press Connect.



19. As a final step and once the GUI is shown press the Menu Preferences
20. In the pop window (see below) press the R interpreter option on the left side
21. Enter C:/Program Files/R/R3.X.X as the R installation path and press OK.



2.2 CIRP Administration

In CIRP each user may possess one or more Roles. A CIRP role is associated with a set of Views and tools.

In general the CIRP User Interface is divided into a set of Perspectives. A Perspective is a set of Views and tools organized into dockable windows. The administrator Role is a pre-defined role that is able to access the Administrator perspective. The latter is accessible via the main toolbar (icon on the left of Figure 1).

Figure 1: Toolbar for perspective selection

The administrator perspective provides the CIRP administration tools for User, Role and Access Rights management. Users are organized into organizations. It consists of three main Views:

- System administration View
- Users View

The System Administration View

Using the options under the User and Roles Management section, the administrator is able to create edit and delete users and user roles.

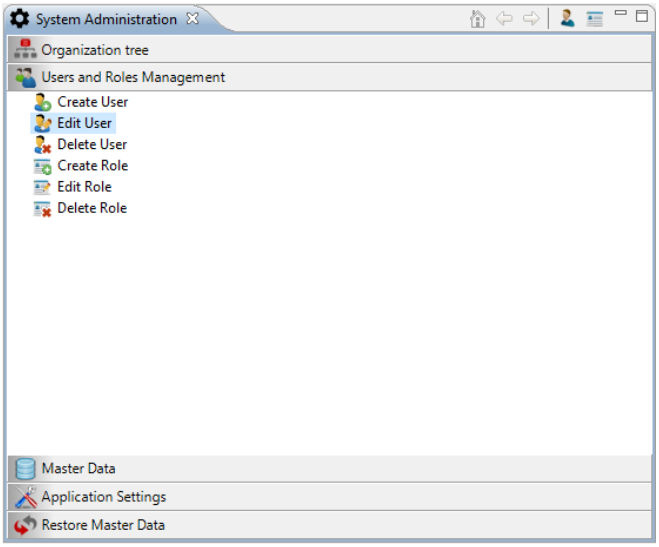


Figure 2: Managing users and roles

For a new user, the administrator must fill in the following information:

- Name
- Surname
- Username:
- Password
- Department
- Etc.

The “Create” button creates the new user. All the compulsory fields are marked with a special icon to help the administrator recognize what information is missing in order to be allowed to continue to the next page of the wizard.

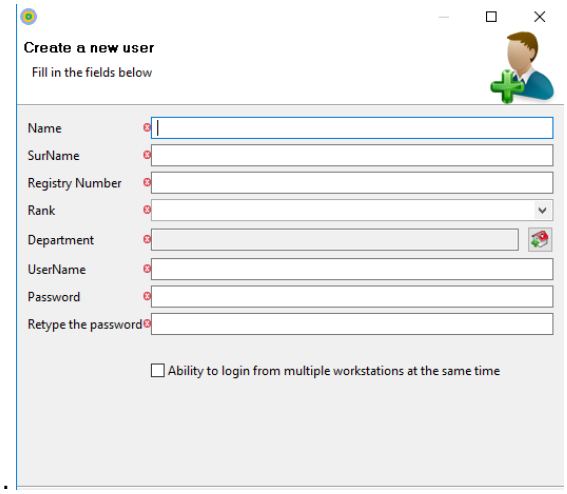


Figure 3: Create New User

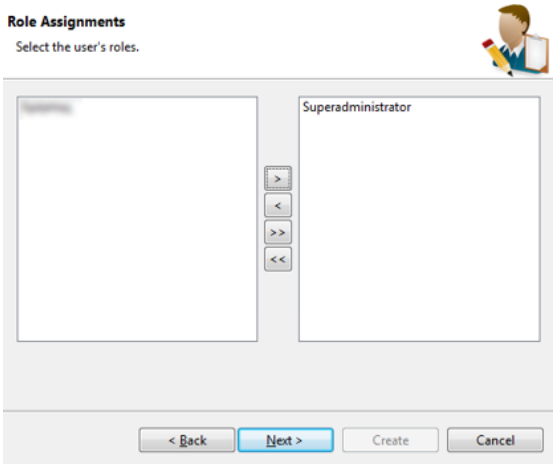


Figure 4: Assign User Roles

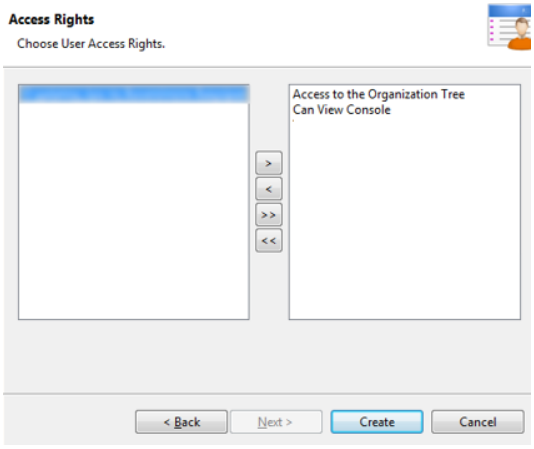


Figure 5: Assign Access Rights

Regarding User Roles the system gives the ability to manage any number of Roles by providing access to the system Perspective, Views and toolbars. A user can be assigned more than one Role.

A user who has the predefined role of Administrator can perform the following functions:

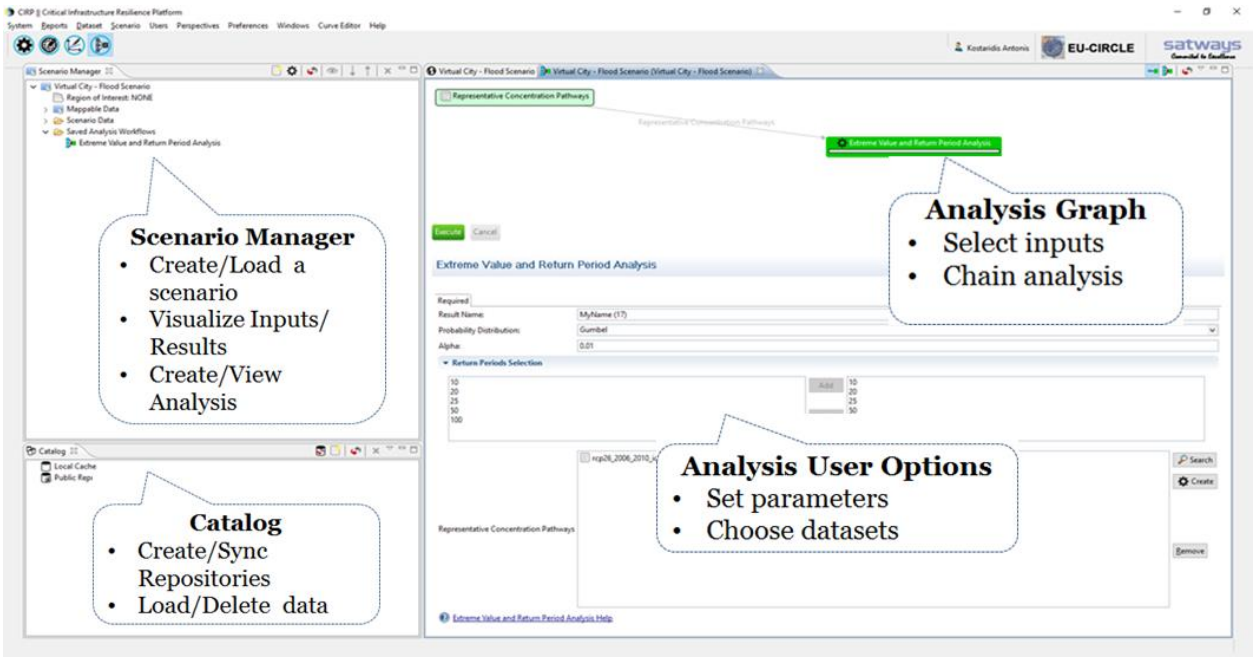
- Create or Delete an Organization's Departments

- Create or Delete Users
- Create or Delete Roles
- Parametrize the System (Application Settings)

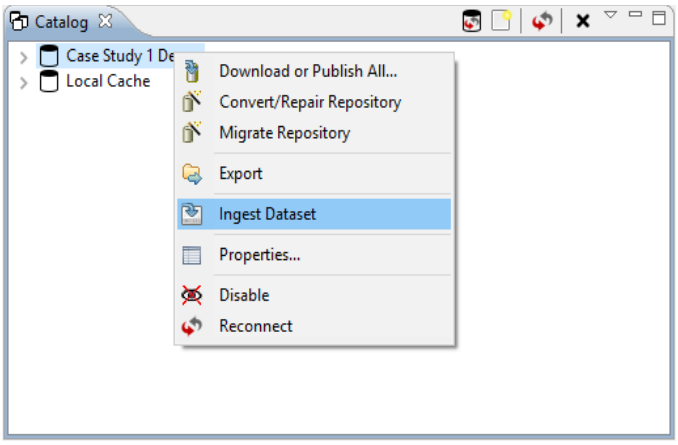
The Administrator can perform organizational management and tune the system according to the requirements of the users and the organization.

2.3 CIRP User manual

The graphical user interface of CIRP consists of different perspectives accessible from buttons of the main toolbar and each perspectives consists of a set of Views (dockable windows). The following Figure presents the main perspective of CIRP where impact assessment scenarios can be executed.



The Catalog View allows the management of repositories (Local or Public) and their datasets.



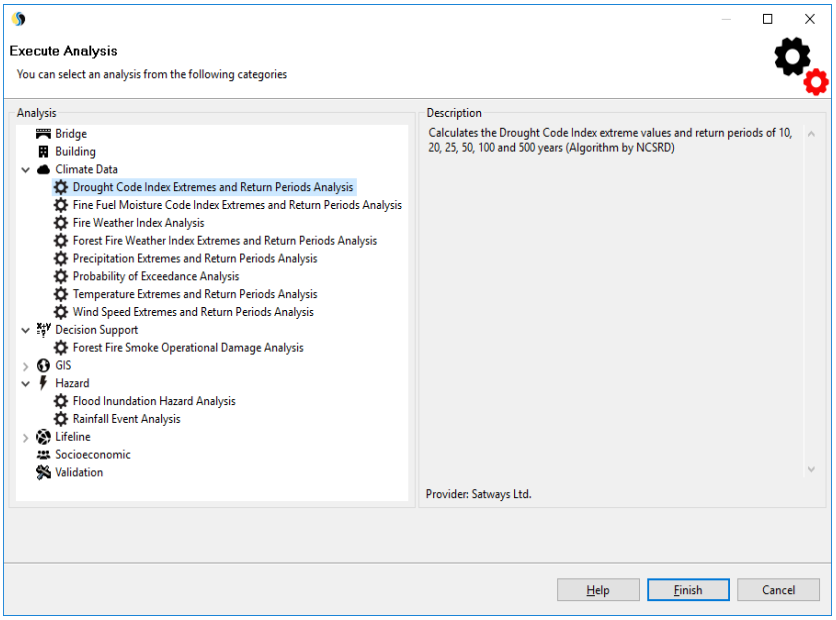
The following dataset types are currently supported:

- Feature Datasets (shapefiles)

- Raster Datasets (Ascii Grid)
- Fragility/Damage Curves (XML)
- Fragility/Damage Curves Mappings (XML)
- Tables (CSV Files)
- Grid Datasets (NetCDF Files)

The analysis toolbox provides the following categories:

- Climate Data
- Building
- Decision Support
- GIS
- Hazard
- Lifeline

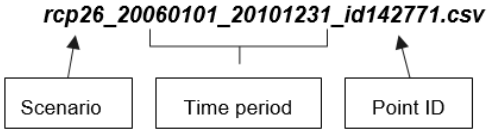


2.3.1 Climate Data Analyses

A Climate data analysis uses CSV datasets as input and performs Extreme Value Theory and Frequency Analysis. The following analysis are currently provided in the Climate Data Analysis category.

- Temperature Extremes and Return Period Analysis
- Wind Speed Extremes and Return Period Analysis
- Precipitation Extremes and Return Period Analysis
- Drought Code Index Extremes and Return Period Analysis
- Fine Fuel Moisture Code Index Extremes and Return Period Analysis
- Forest Fire Weather Index Extremes and Return Period Analysis
- Fire Weather Index Analysis
- Probability of Exceedance Analysis

An input dataset file for the Virtual City has the following filename convention:



And sample content:

TIME	LAT	LON	T_00[C]	WS_00[m/s]	T_06[C]	WS_06[m/s]	T_12[C]	R_D[mm]	WS_12[m/s]	RH_D[%]	SM_D[kg/m2]	T_18[C]	WS_18[m/s]
20060101	35.4012985	24.2626991	4.11439991	1.08889997	3.63080001	1.42990005	16.1812	0.679799974	3.06629992	82.4738007	298.75	13.7228003	7.75969982
20060102	35.4012985	24.2626991	12.1202002	7.9678998	10.5530996	6.88859987	13.3780003	1.02999997	11.8950996	70.2600021	298.5	9.04440022	11.6323996

Where:

T: Temperature

WS: Wind Speed

R_D: Rain Daily

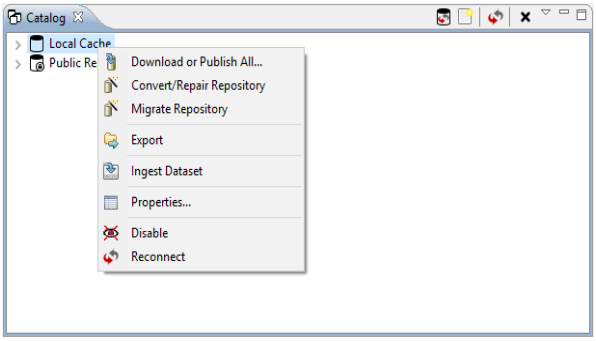
RH_D: Relative Humidity Daily

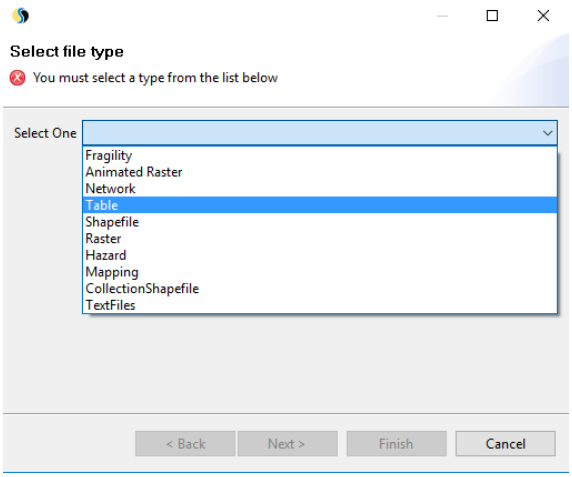
SM_D: Soil Moisture Daily

2.3.2 Ingesting RCP Datasets

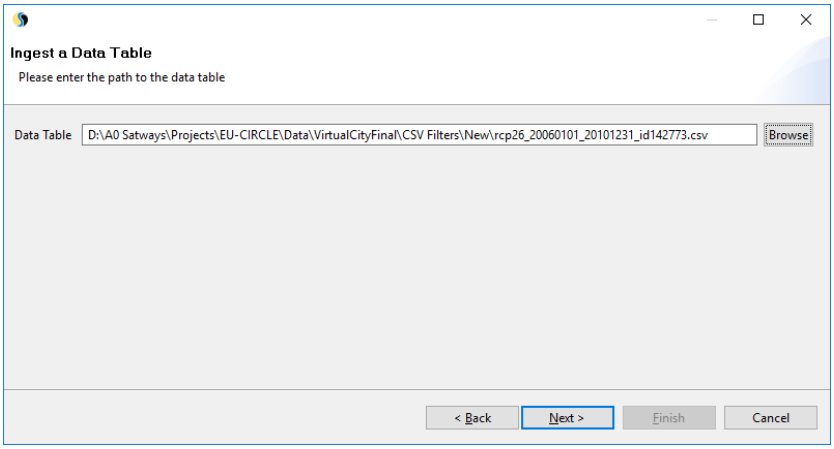
Follow the steps below:

1. Right click the local repository and select the menu option “Ingest”
2. The Ingestion Wizard will popup
3. Select the “**Table**” file type

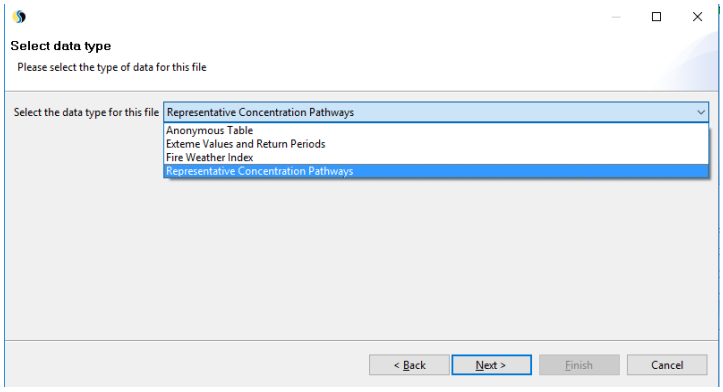




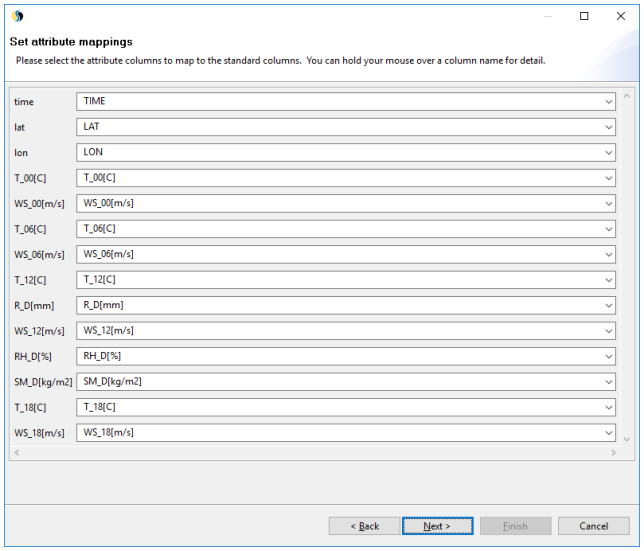
4. Select the RCP CSV file from our local disk and press the Next button



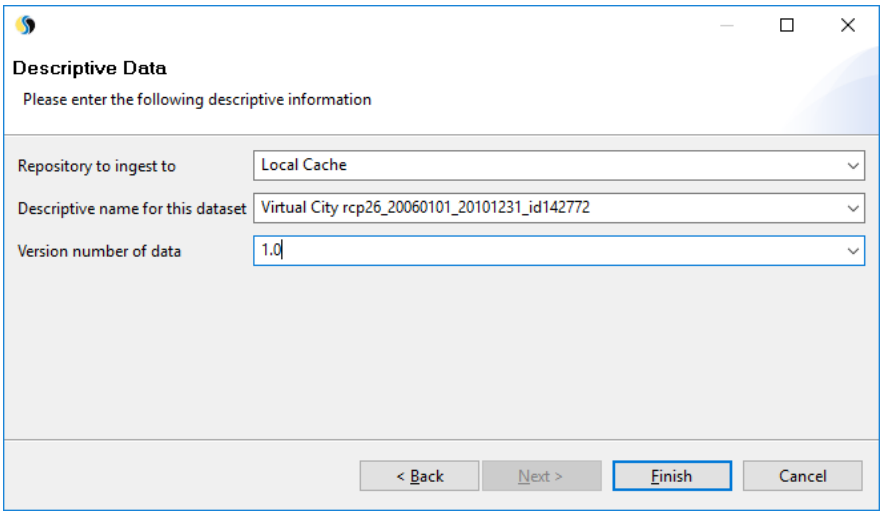
5. In the “Select Data type” wizard page select the “Representative Concentration Pathways” from the drop down and press the Next button.



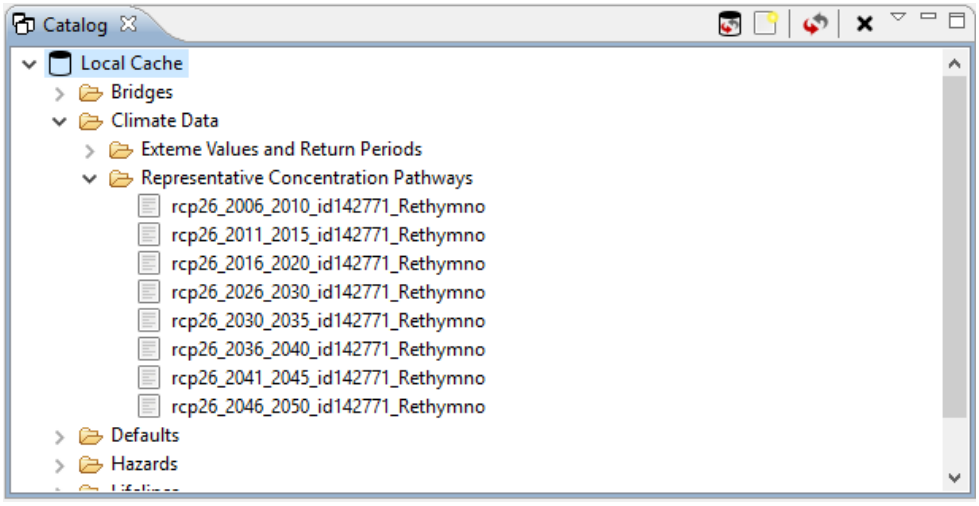
6. In the “Set attribute mappings” page if the ingested CSV file has already the correct column names, CIRP will read them and the mapping is automatic. In this case you can press the Next button.



7. The “Set extra field information” wizard page is empty so you can press immediately the Next button
8. Enter a descriptive name for this dataset and a version number. Then press the **Finish** button.



9. Your ingested dataset will appear in the Climate Data -> Representative Concentration pathways folder in your Local Cache.

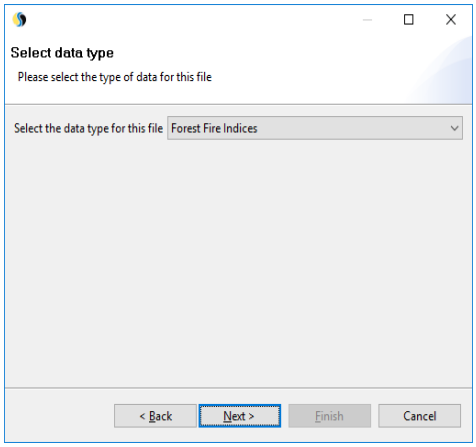


Another method is to ingest multiple RCP Cordex file in one shot

- Right click the local repository and select the menu option “Ingest”
- The Ingestion Wizard will popup
- Select the “MultiTable” file type
- Follow the previous steps without specifying a name for the ingested file

2.3.3 Ingesting Forest Fire Indices Datasets

Forest Fire Indices are CSV datasets that can be ingested in the same way as RCP datasets (one file or multiple files). The difference is the data type wizard page (see following Figure) where you need to choose “Forest Fire Indices).



The Forest Fire Indices datasets have the following naming convention and columns:

FIR_rcp85_20060101_20101231_id549.csv

Scenario | Time period | Point ID

```
TIME , LAT , LON , DC_D    ,FMC_D   ,FWI_D   ,
 20060501,   45.3824692    ,   4.77150583    ,   32.2126999    ,   64.2464981    ,  0.319011986    ,
 20060502,   45.3824692    ,   4.77150583    ,   32.9561996    ,   48.5330009    ,  0.124606997    ,
```

DC_D: Drought Code Daily

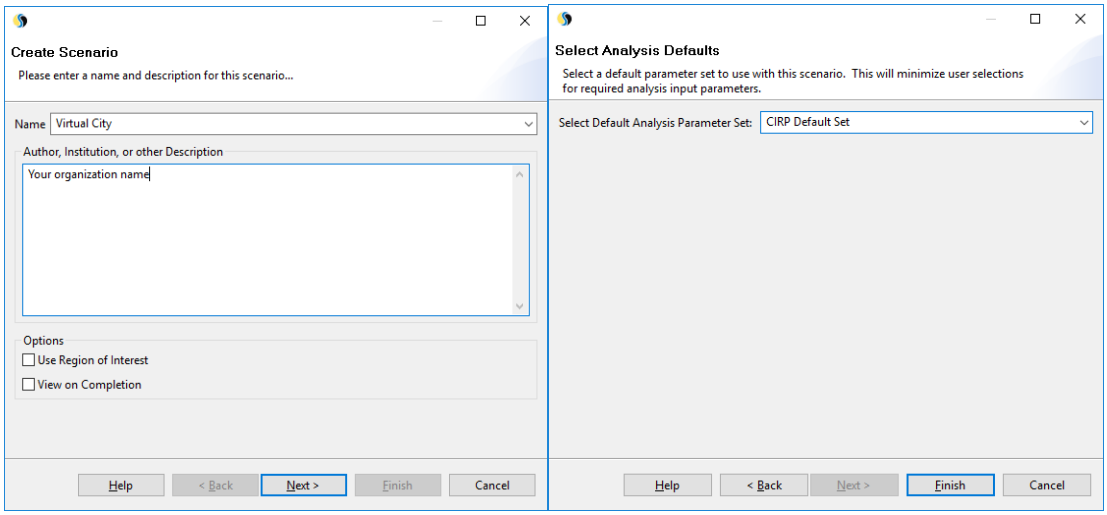
FMC_D: Fine Fuel Moisture Code Daily

FWI_D: Fire Weather Inded Daily

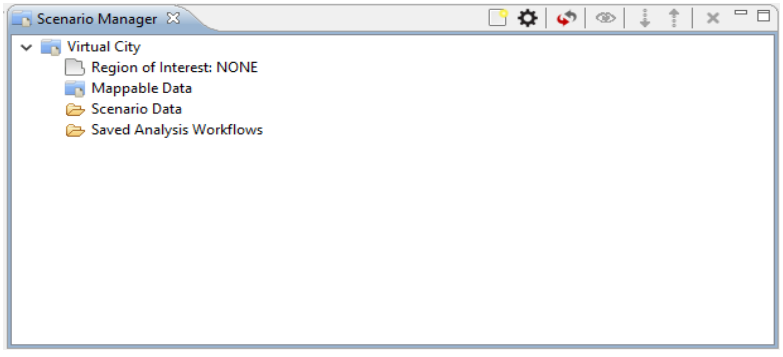
2.3.4 Creating a New Scenario

In order to Create a new Scenario press the “New Scenario” button in the Scenario Manager View toolbar

The "Create Scenario" wizard will appear. Enter a name for your scenario and optionally enter a description. Untick the "Use Region of Interest" checkbox and press the Next button. In the next page select "CIRP Default Set" and press Finish.



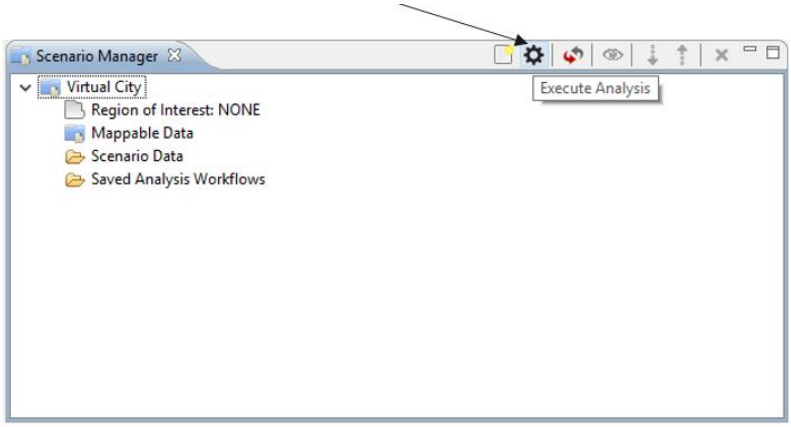
The scenario appears in the Scenario Manager View



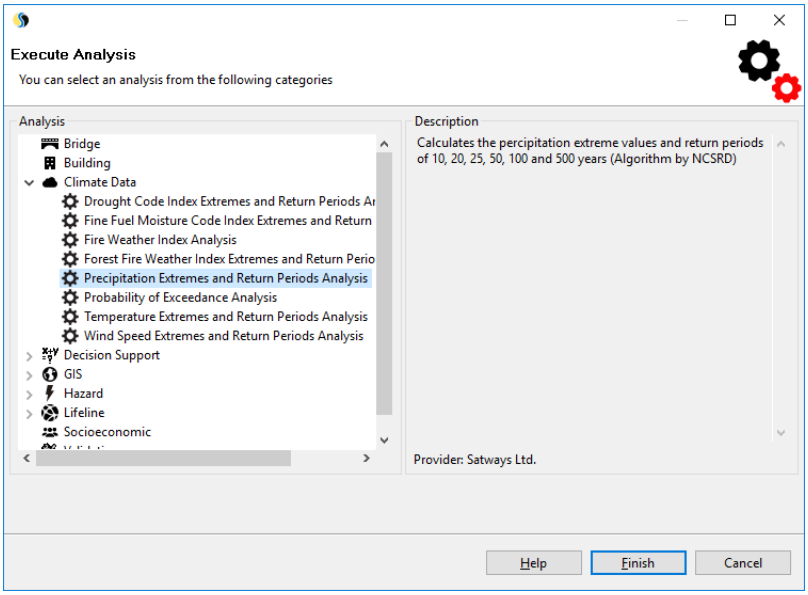
2.3.5 Selection and Execution of Analyses

Precipitation Extreme Values and Return Periods Analysis

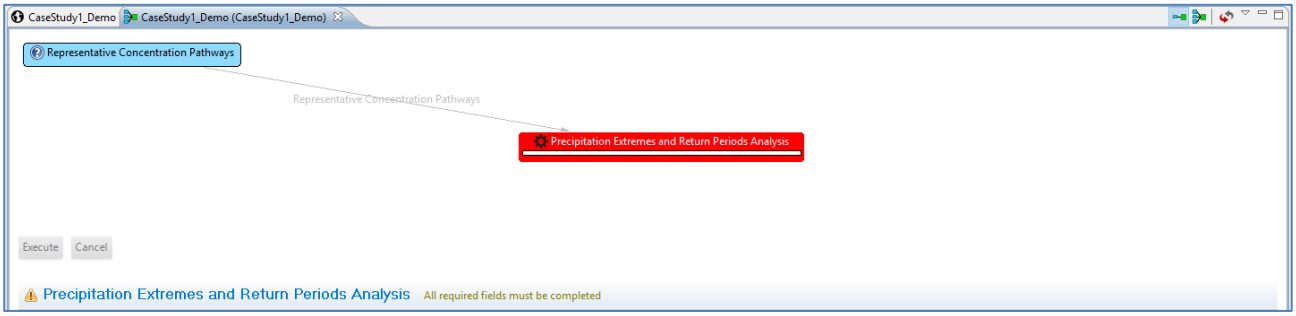
- In order to execute the "Precipitation Extreme Values and Return period analysis", Press the "Execute Analysis" button from the Scenario Manager view



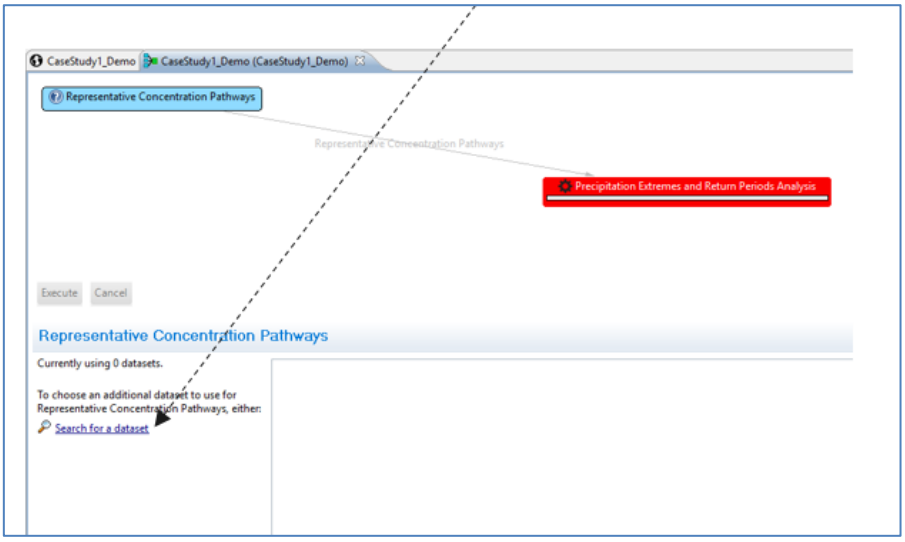
- Unfold the Climate Data category, select the "Precipitation Extreme Value and Return Period Analysis" option and press Finish.

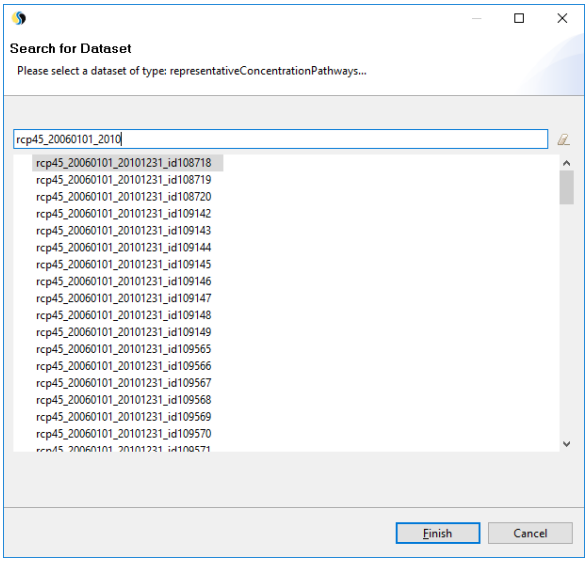


- The analysis graph View will be opened.
- Click on the blue box "Representative Concentration Pathways" in order to select an RCP dataset.

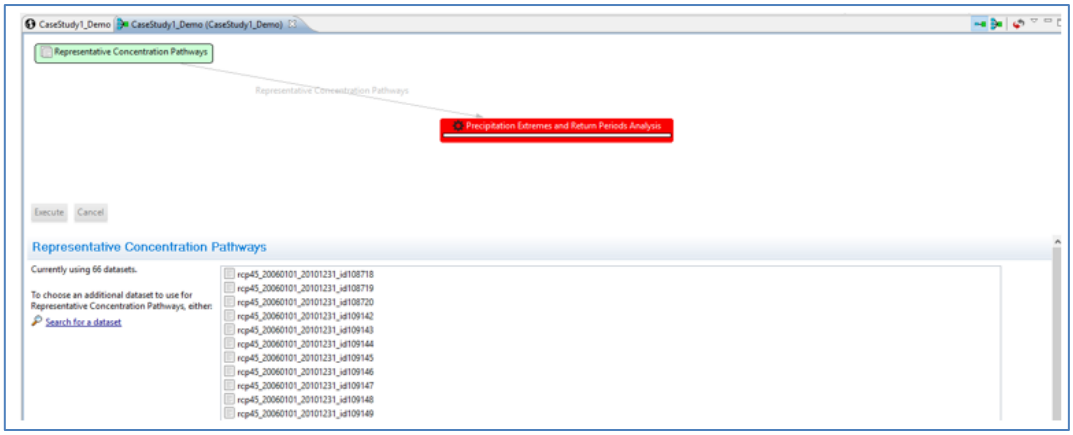


- Press the "Search for a dataset" option and select an ingested dataset from the popup window.
- You can filter the dataset names by typing in the text box. Then press Finish.



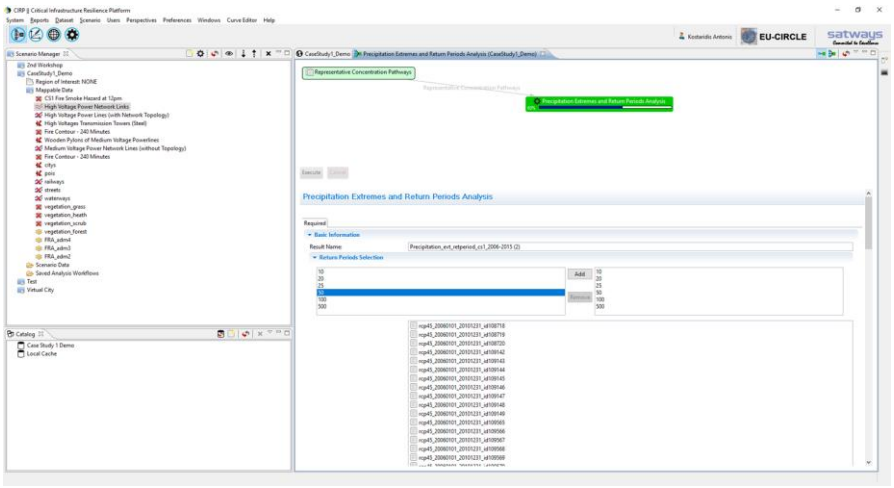


- The blue box will become green as the input dataset has been chosen.
- Click the red analysis box to set the rest of the analysis options.

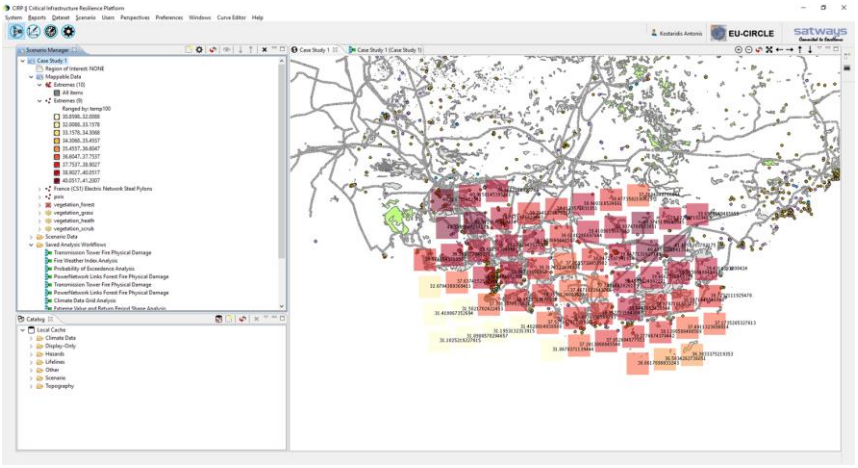


- Select a name for the output file

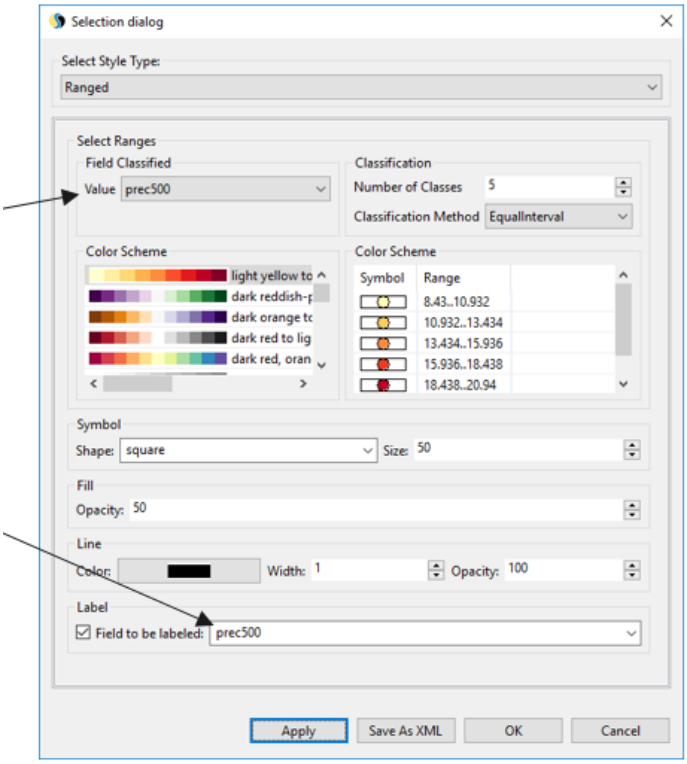
- Select the distribution type and alpha value
- Select the return periods for which you wish to calculate the extreme values
- Once all parameters are set the red box becomes red and the execute button is activated
- Press the Execute button



- After some seconds the analysis execution will be completed.
- A new file will be created in the Scenario tree and appears on the map with the attributes depending on the return periods selected.
- The default layer style is Ranged based on prec100 column data

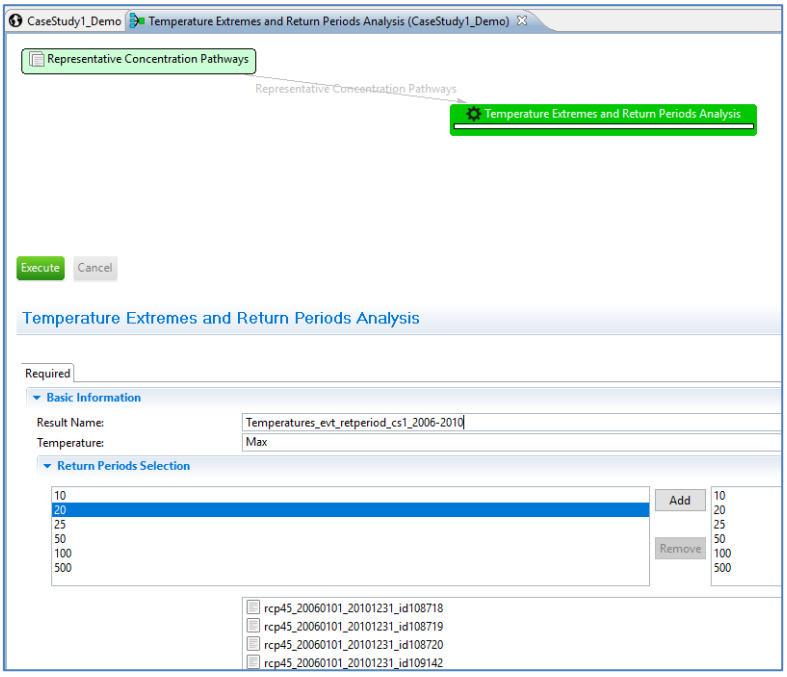


- Change the resulted datasets (feature layer) style in order to visualize any return period on the map
- Select both the Field Classified to be the same with the Label (e.g. prec500)

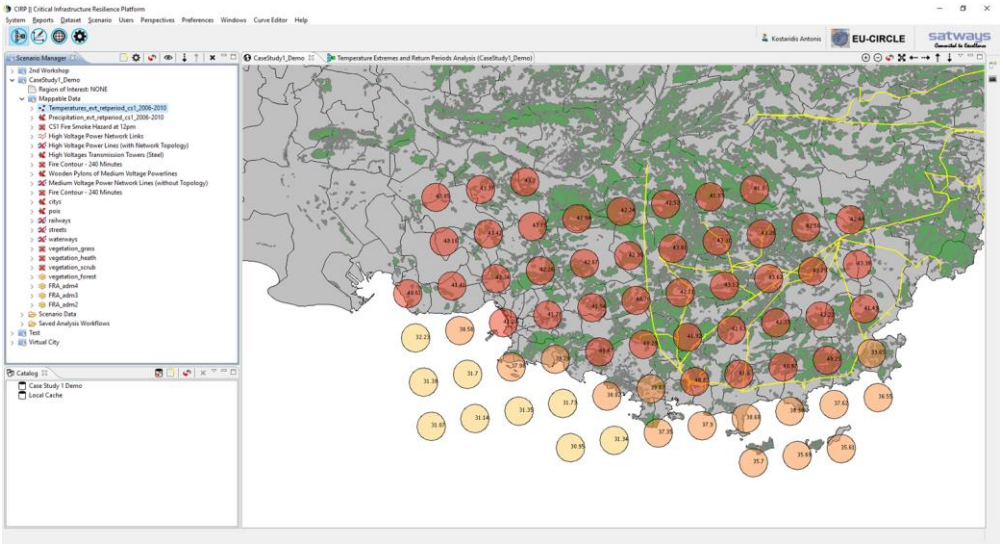


Temperature Extreme Values and Return Periods Analysis

- Select from the Climate Data category the “Temperature Extreme Value and Return Period Analysis” option and press Finish.
- Follow the steps described in the “Precipitation Extreme Values and Return Periods Analysis” in order to load the RCP datasets for the Virtual City
- In the Analysis View:
    - Select a name for the output file
    - Select either Max, Min or Median for the temperature values
    - Select the return periods
    - Optionally select the distribution type and alpha value
    - Once all parameters are set the red box becomes red and the execute button is activated
    - Press the Execute button



- After some seconds the analysis execution will be completed.
- A new file will be created in the Scenario tree and appears on the map with the attributes depending on the return periods selected.
- The default layer style is Ranged based on temp100 column data



- You can change the resulted datasets (feature layer) style in order to visualize any return period on the map
- Select both the Field Classified to be the same with the Label (e.g. temp500)

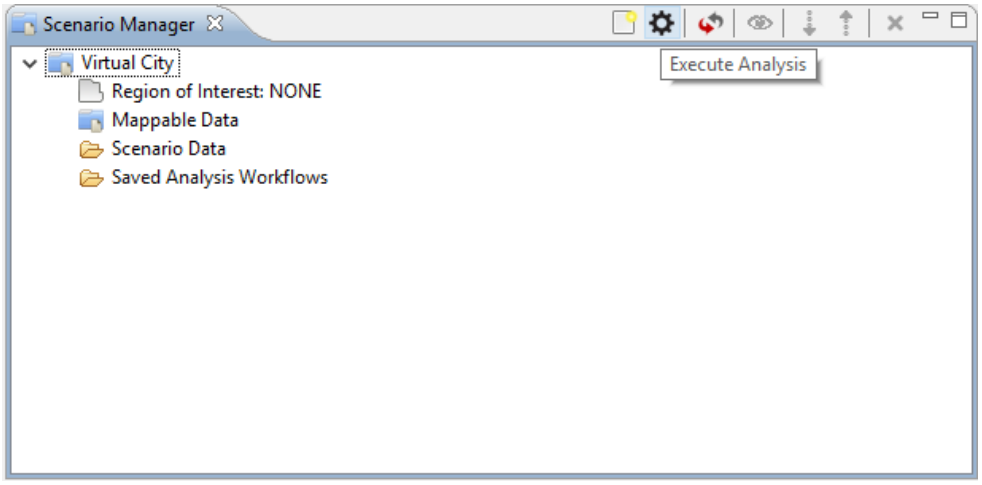
Wind Speed Extreme Values and Return Periods Analysis

- Select from the Climate Data category the “Wind Speed Extreme Value and Return Period Analysis” option and press Finish.
- Follow the steps described in the “Precipitation Extreme Values and Return Periods Analysis” in order to load the RCP datasets for the Virtual City
- In the Analysis View:
    - Select a name for the output file
    - Select the return periods
    - Optionally select the distribution type and alpha value
    - Once all parameters are set the red box becomes red and the execute button is activated
    - Press the Execute button
    - After some seconds the analysis execution will be completed.
    - A new file will be created in the Scenario tree and appears on the map with the attributes depending on the return periods selected.

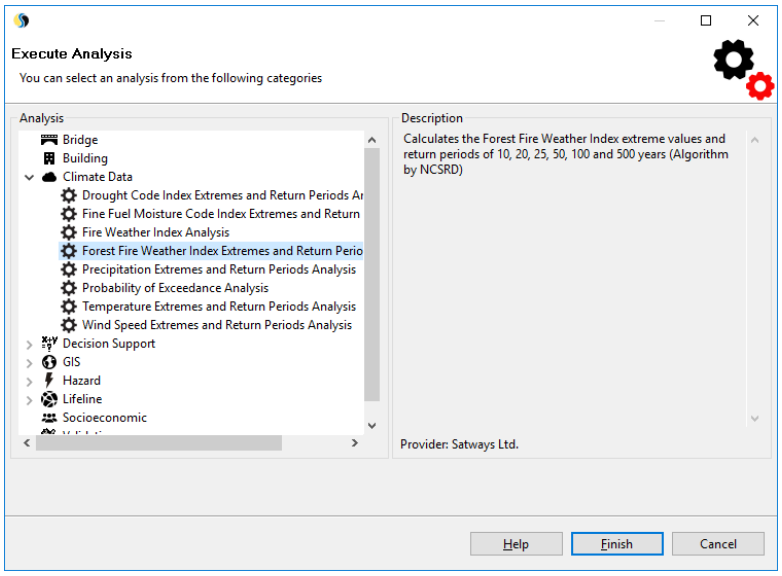
Forest Fire Weather Index (FWI) Extreme Values and Return Periods Analysis

This analysis takes as input Forest Fire Indice grid files (CSVs) and calculates the Fire Weather Index Extreme values per return period.

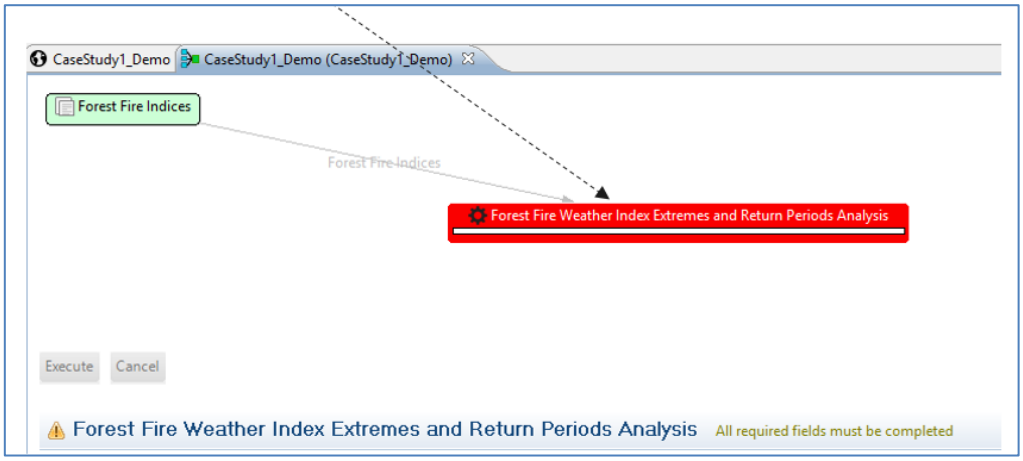
- The output of the analysis is a POINT feature layer (shapefile) with feature per input grid location.
- For each feature the FWI value and associated class for each return period is calculated. Thus the following attributes are produced:
    - fwi10 (10 years FWI value)
    - fwi20 (20 years FWI value)
    - fwi25 (25 years FWI value)
    - fwi50 (50 years FWI value)
    - fwi100 (100 years FWI value)
    - fwi500 (500 years FWI value)
    - fwi10Cl (10 years class)
    - fwi20Cl (20 years class)
    - fwi25Cl (25 years class)
    - fwi50Cl (50 years class)
    - fwi100Cl (100 years class)
    - fwi500Cl (500 years class
- Press the “Execute Analysis” button to select an analysis



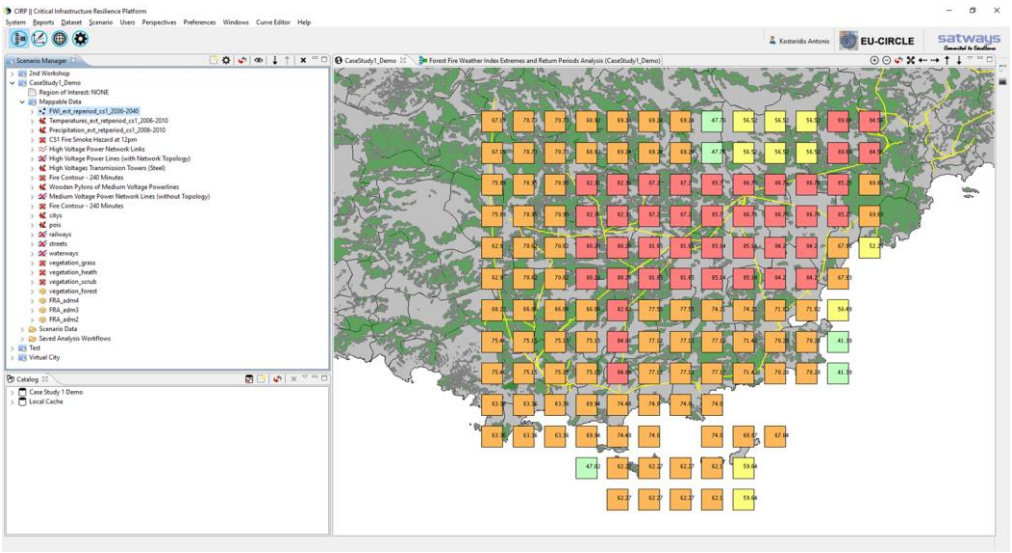
- Unfold the Climate Data category, select the “Forest Fire Weather Index Extreme Value and Return Period Analysis” option and press Finish.



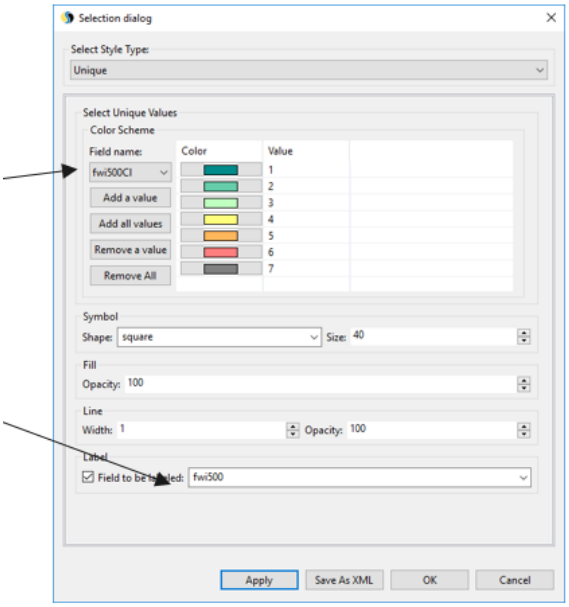
- Click on the blue box “Forest Fire Indices” in order to select a dataset.
- Press the “Search for a dataset” option and select an ingested dataset from the popup window.
- You can filter the dataset names by typing in the text box. Then press Finish.
- The blue box will become green as the input dataset has been chosen.
- Click the red analysis box to set the rest of the analysis options.



- Select a name for the output file
- Select the return periods
- Optionally select the distribution type and alpha value
- Once all parameters are set the red box becomes red and the execute button is activated
- Press the Execute button
- After some seconds the analysis execution will be completed.
- A new file will be created in the Scenario tree and appears on the map with the attributes depending on the return periods selected.
- The default layer style is Ranged based on fwi100Cl column data



- You can change the resulted datasets (feature layer) style in order to visualize any return period on the map
- E.g. select the Field Classified to be fwi500Cl and the same the Label (e.g. fwi500)

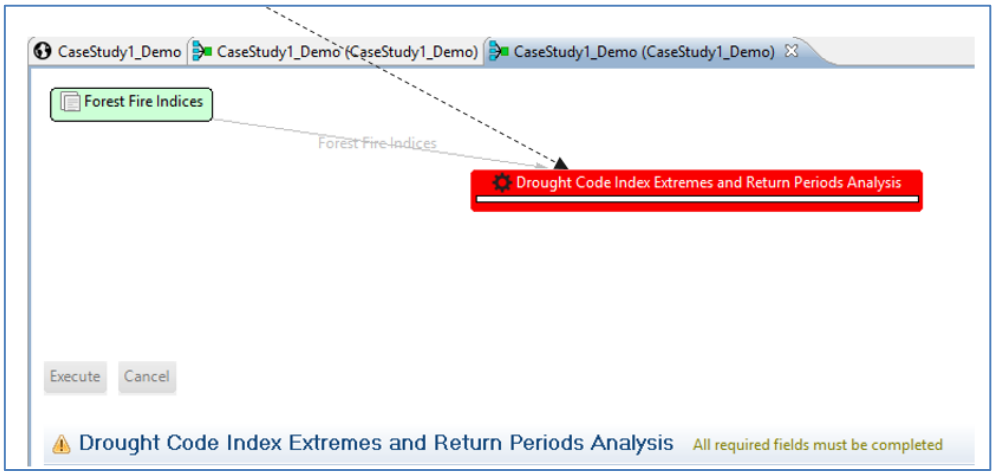


Drought Code Extreme Values and Return Periods Analysis

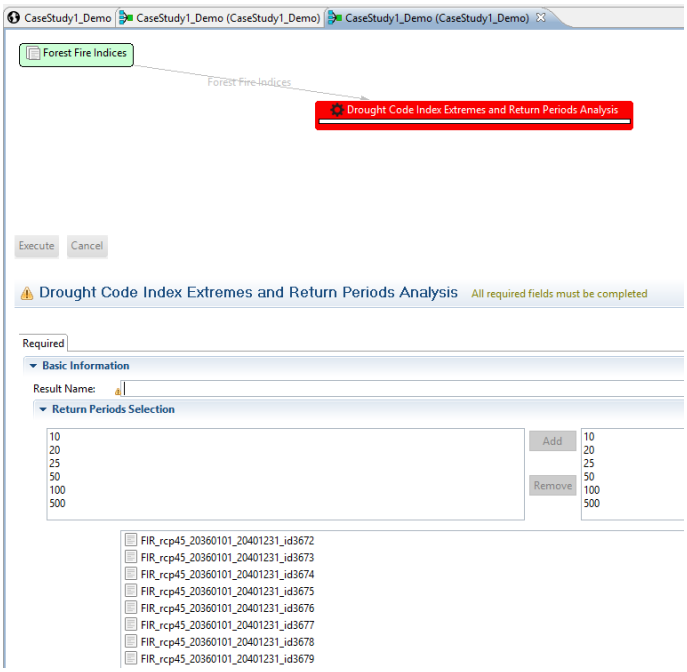
This analysis takes as input Forest Fire Indice grid files (CSVs) and calculates the Drought Code Extreme values per return period.

- The output of the analysis is a POINT feature layer (shapefile) with feature per input grid location.
- For each feature the drought code value and associated class for each return period is calculated. Thus the following attributes are produced:
    - dc10 (10 years drought code value)
    - dc20 (20 years drought code value)
    - dc25 (25 years drought code value)
    - dc50 (50 years drought code value)
    - dc100 (100 years drought code value)
    - dc500 (500 years drought code value)
    - dc10Cl (10 years class)
    - dc20Cl (20 years class)
    - dc25Cl (25 years class)
    - dc50Cl (50 years class)
    - dc100Cl (100 years class)
    - dc500Cl (500 years class)
- Press the “Execute Analysis” button to select an analysis
- Unfold the Climate Data category, select the “Drought Code Index Extreme Value and Return Period Analysis” option and press Finish.
- The analysis graph View will be opened
- Click on the blue box “Forest Fire Indices” in order to select a dataset.
- Press the “Search for a dataset” option and select an ingested dataset from the popup window.

- You can filter the dataset names by typing in the text box. Then press Finish.
- The blue box will become green as the input dataset has been chosen.
- Click the red analysis box to set the rest of the analysis options.



- Select a name for the output file
- Select the return periods
- Optionally select the distribution type and alpha value
- Once all parameters are set the red box becomes red and the execute button is activated
- Press the Execute button



- After some seconds the analysis execution will be completed.
- A new file will be created in the Scenario tree and appears on the map with the attributes depending on the return periods selected.
- The default layer style is Ranged based on dc100Cl column data

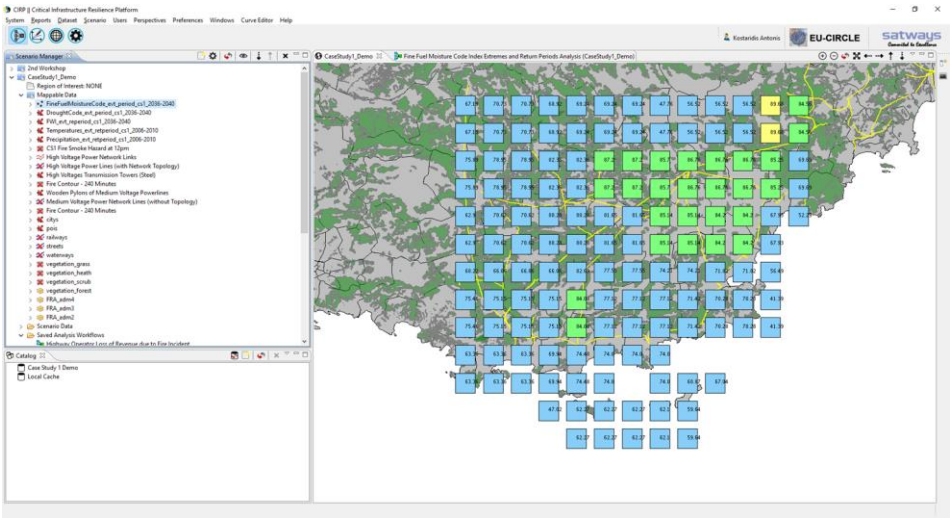
- You can change the resulted datasets (feature layer) style in order to visualize any return period on the map
- E.g. select the Field Classified to be dc500Cl and the same the Label (e.g. dc500)

Fine Fuel Moisture Code (FFMC) Extreme Values and Return Periods Analysis

This analysis takes as input Forest Fire Indice grid files (CSVs) and calculates the Fine Fuel Moisture Code Extreme values per return period.

- The output of the analysis is a POINT feature layer (shapefile) with feature per input grid location.
- For each feature the drought code value and associated class for each return period is calculated. Thus the following attributes are produced:
    - ffmc10 (10 years fine fuel moisture code value)
    - ffmc20 (20 years fine fuel moisture code value)
    - ffmc25 (25 years fine fuel moisture code value)
    - ffmc50 (50 years fine fuel moisture code value)
    - ffmc100 (100 years fine fuel moisture code value)
    - ffmc500 (500 years fine fuel moisture code value)
    - ffmc10Cl (10 years class)
    - ffmc20Cl (20 years class)
    - ffmc25Cl (25 years class)
    - ffmc50Cl (50 years class)
    - ffmc100Cl (100 years class)
    - ffmc500Cl (500 years class)
- Press the “Execute Analysis” button to select an analysis
- Unfold the Climate Data category, select the “Fine Fuel Moisture Code Index Extreme Value and Return Period Analysis” option and press Finish.
- The analysis graph View will be opened
- Click on the blue box “Forest Fire Indices” in order to select a dataset.
- Press the “Search for a dataset” option and select an ingested dataset from the popup window.
- You can filter the dataset names by typing in the text box. Then press Finish.
- The blue box will become green as the input dataset has been chosen.
- Click the red analysis box to set the rest of the analysis options.
- Select a name for the output file
- Select the return periods
- Optionally select the distribution type and alpha value
- Once all parameters are set the red box becomes red and the execute button is activated
- Press the Execute button
- After some seconds the analysis execution will be completed.

- A new file will be created in the Scenario tree and appears on the map with the attributes depending on the return periods selected.
- The default layer style is Ranged based on ffmc100Cl column data



- You can change the resulted datasets (feature layer) style in order to visualize any return period on the map
- E.g. select the Field Classified to be ffmc500Cl and the same the Label (e.g. ffmc500)

Forest Fire Indices Threshold Statistical Analysis

This analysis takes as input Forest Fire Indice grid files (CSVs) and calculates the number of days a Forest Fire Index exceeds or not a threshold value

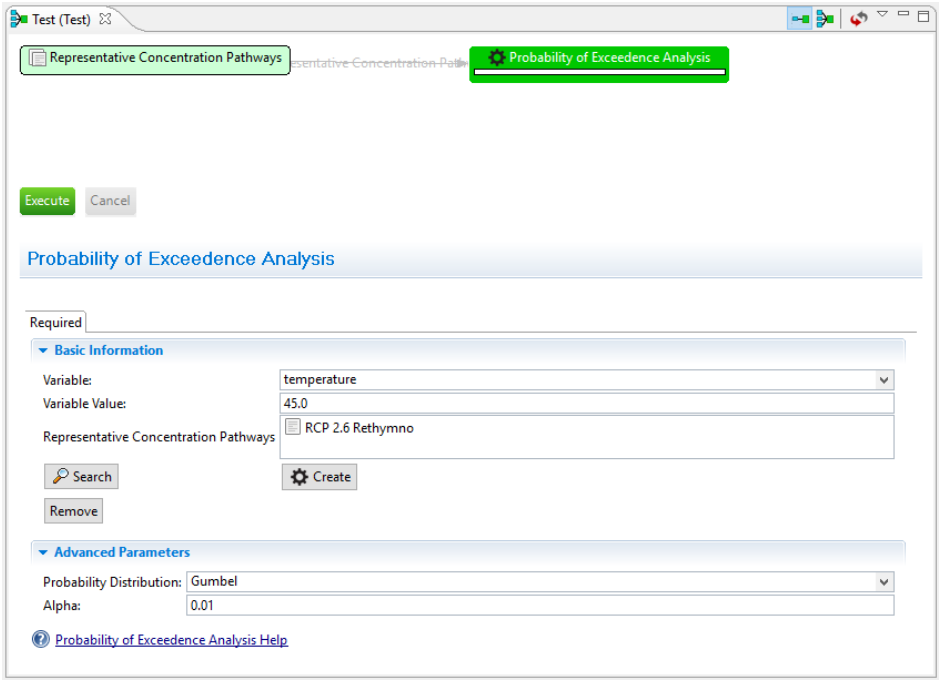
- The output of the analysis is a POINT feature layer (shapefile) with feature per input grid location.
- For each feature the following attributes are calculated:
    - ffi_var (forest fire index variable name. can take values: dc_c, ffmc_d, fwi_d)
    - ffmc20 (20 years fine fuel moisture code value)
    - operator (denotes the comparison against the threshold value)
    - threshold (the threshold value)
    - startdate (starting date of input FFI data)
    - enddate (ending date of input FFI data)
    - no_days (number of days)
- Press the “Execute Analysis” button to select an analysis
- Unfold the Climate Data category, select the “Forest Fire Indices Threshold Statistics Analysis” option and press Finish.
- Click on the blue box “Forest Fire Indices” in order to select a dataset.
- Press the “Search for a dataset” option and select an ingested dataset from the popup window.
- You can filter the dataset names by typing in the text box. Then press Finish.
- Click the red analysis box to set the rest of the analysis options.
- Select a name for the output file

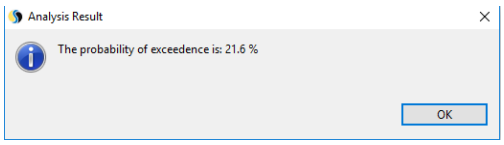
- Select the FFI index
- Once all parameters are set the red box becomes red and the execute button is activated
- Press the Execute button
- After some seconds the analysis execution will be completed.
- A new file will be created in the Scenario tree and appears on the map with the attributes depending on the return periods selected.

Probability of Exceedance Analysis

This analysis calculates probability of exceedance for any given climate variable based on the value set via the Analysis UI.

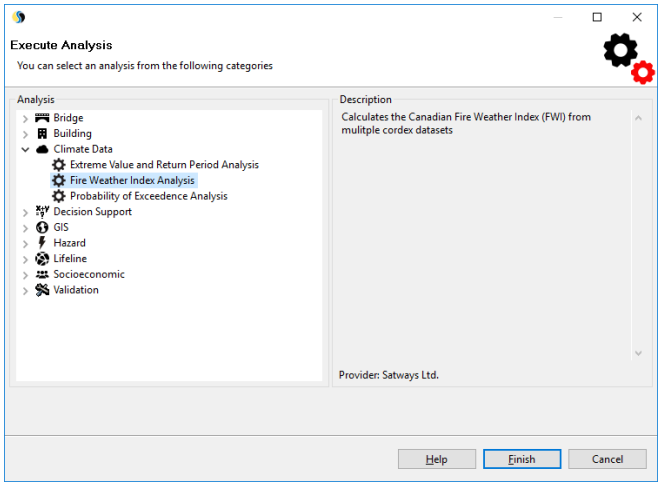
- You can also select the probability distribution and set an alpha value.
- The resulting probability is shown via a popup dialog after the analysis is finished.



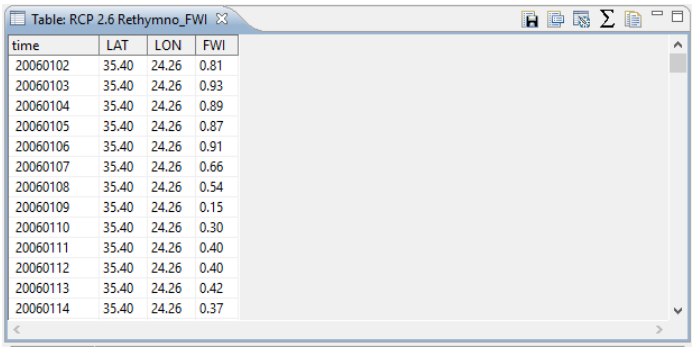


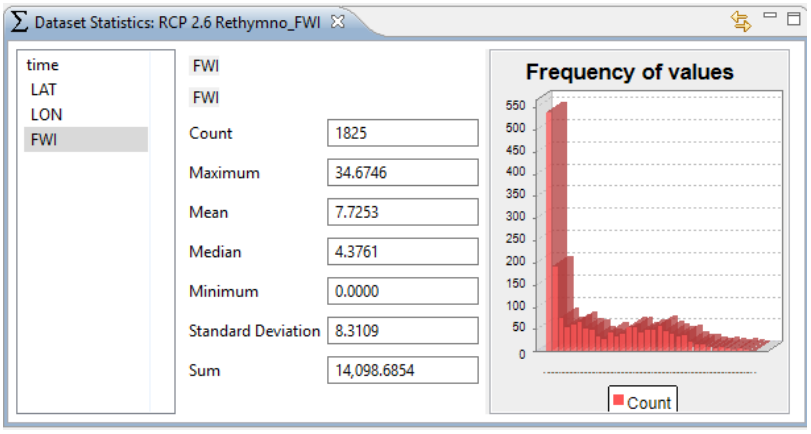
Fire Weather Index Analysis

- This analysis calculates the Calculates the Canadian Fire Weather Index (FWI) from mulitple cordex datasets
- Assuming that cordex files have already been ingested
- Select the new analysis from the “Climate Data” category



- Select one or more cordex datasets from the Repository
- For each of the selected cordex datasets a new file will be created that contains the FWI for Date
- Press the Execute Button
- The resulting files have the same name as the cordex files with the additional suffix "_FWI")



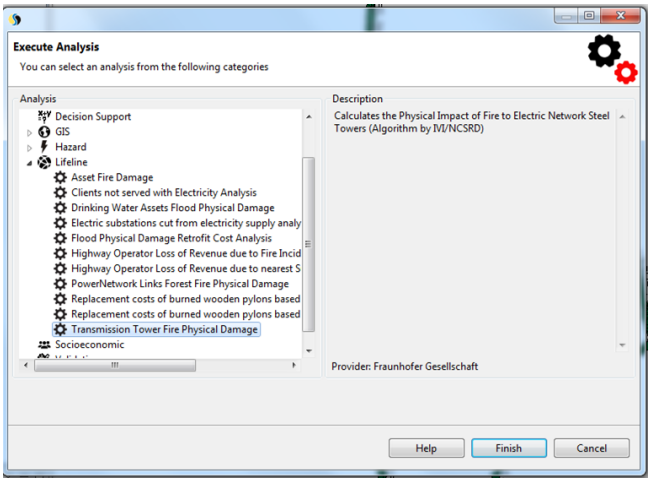


Transmission Tower Fire Physical Damage

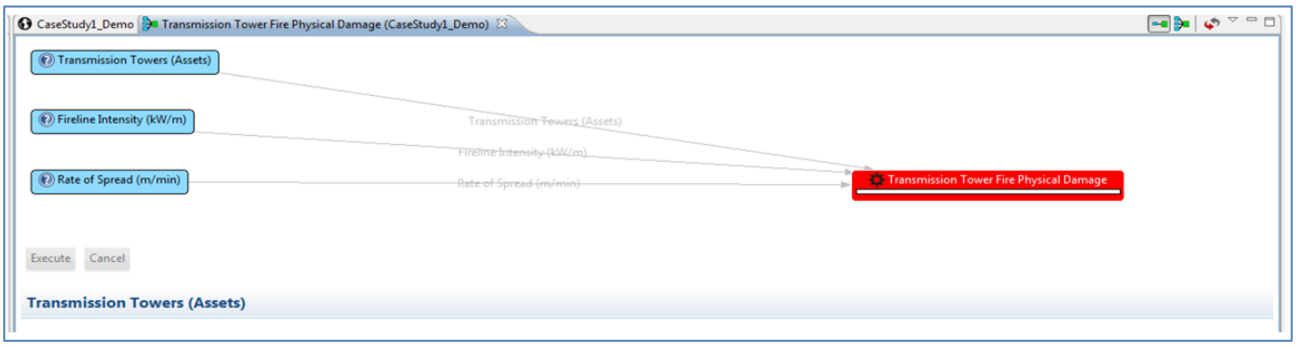
This analysis determines if steel transmission pylons are physically damaged due to fire based on the following steps: get fireline intensity per computational cell, estimate temperature, find strength of structure, select exposure, determine fire duration, and finally find damage value from fragility curve. The result value is added to the column damagefactor of Asset attribute table.

- Assuming that new scenario with unticked "Use Region of Interest" checkbox was created.

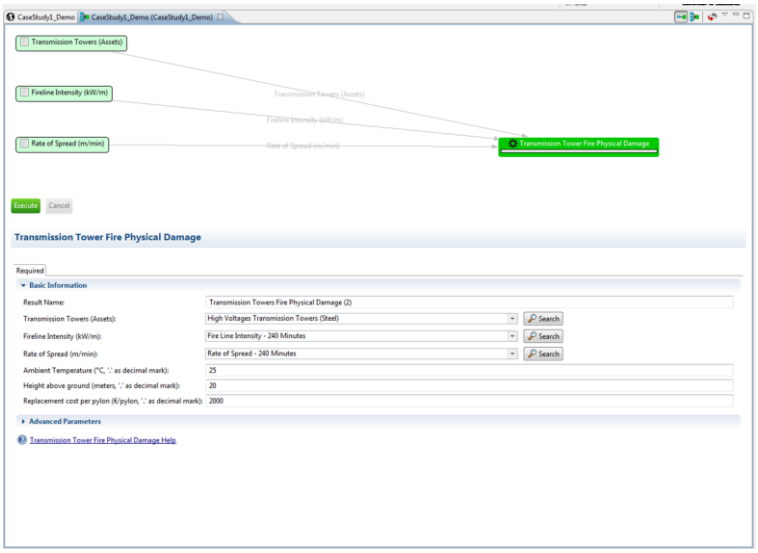
- Unfold the Lifeline category and select the “Transmission Tower Fire Physical Damage” option and press Finish.

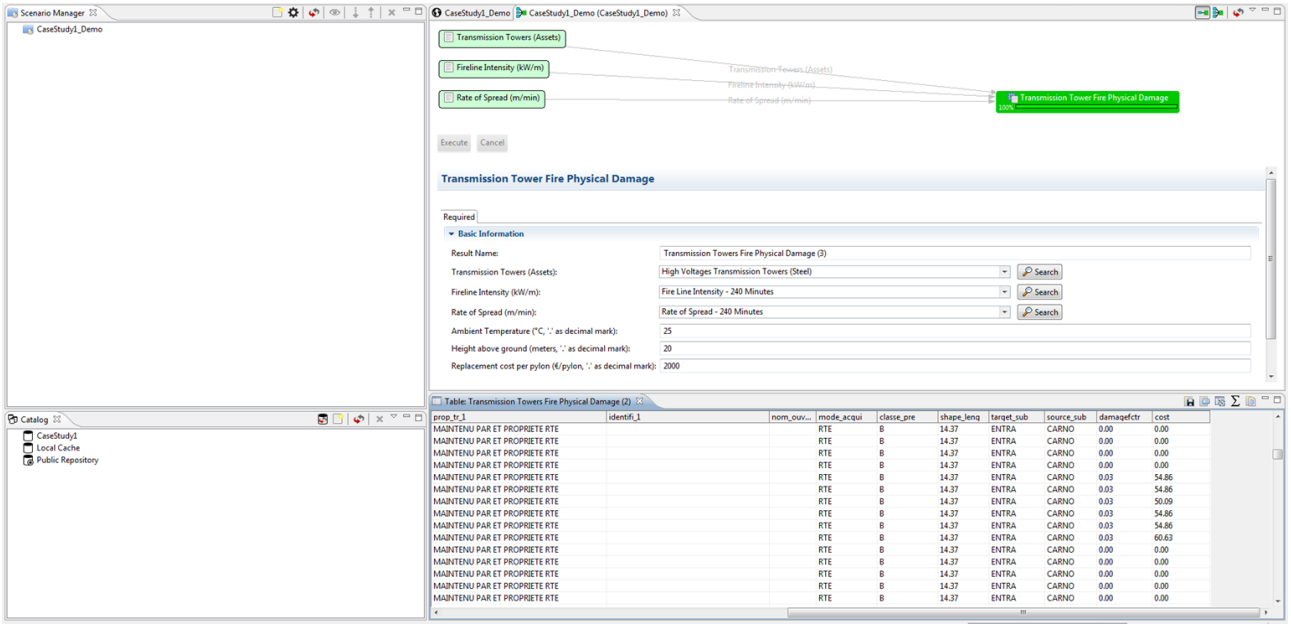


- The analysis graph View will be opened. Click on the blue boxes “Transmission Towers (Assets)”, “Fireline Intensity (kW/m)” and “Rate of Spread (m/min)”in order to select an RCP datasets.



- Press the “Search for a dataset” option and select an ingested dataset from the popup window. Then press Finish.
- The blue box will become green as the input dataset has been chosen
- Select a name for the output file
- Select the Ambient Temperature, Height above ground, and Replacement cost per pylon
- Press the Execute button
- After some seconds the analysis execution will be completed.
- A new file will be created in the Scenario tree under the folder name “Transmission Tower Fire Physical Damage”
- Double click the file to view its contents

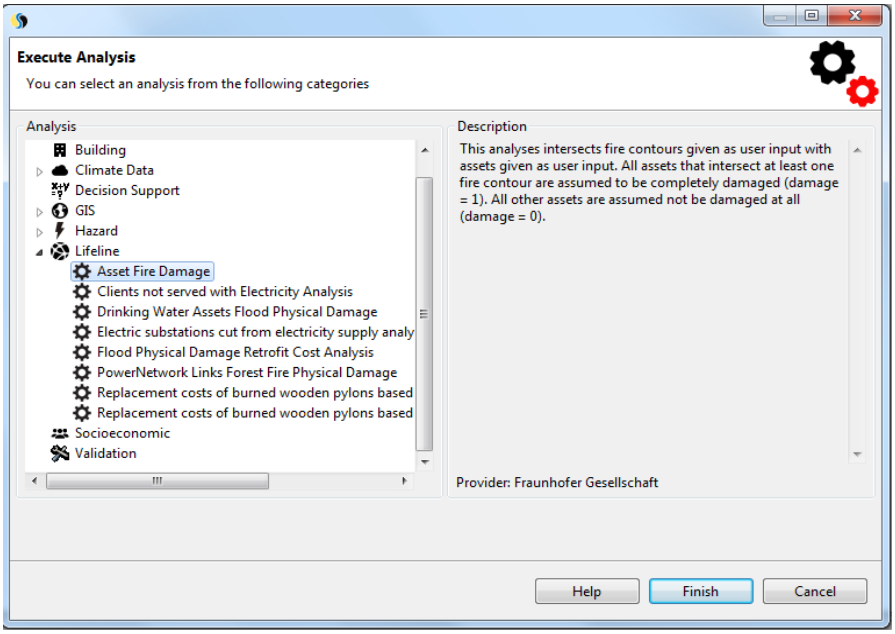




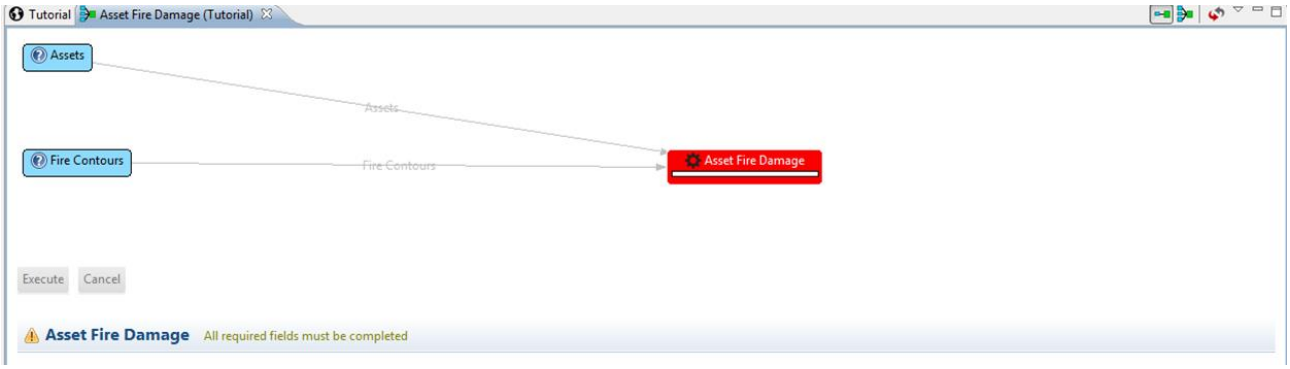
Asset Fire Damage Analysis

This analyses intersects fire contours given as user input with assets given as user input. All assets that intersect at least one fire contour are assumed to be completely damaged (damage = 1). All other assets are assumed not be damaged at all (damage = 0).

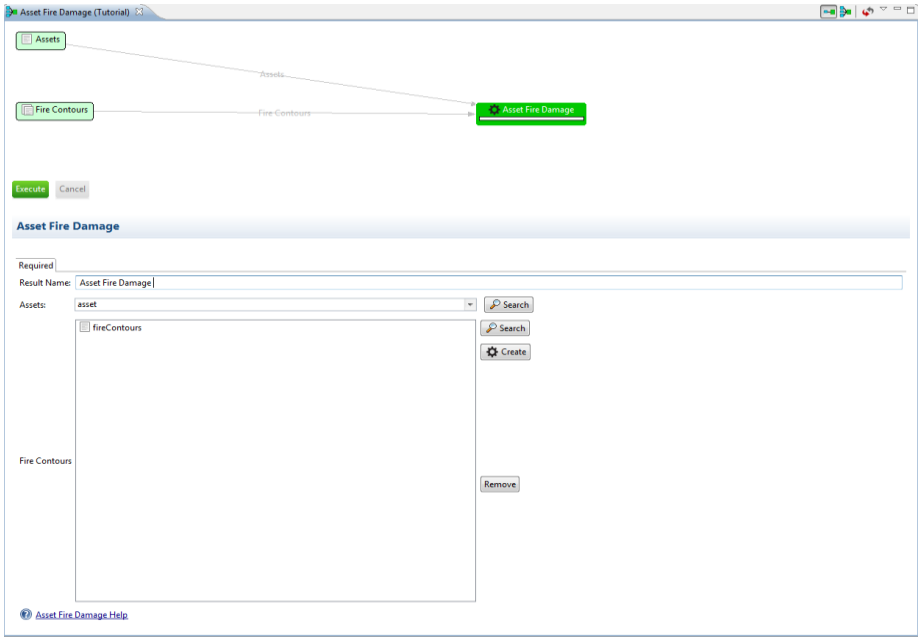
- Assuming that new scenario with unticked “Use Region of Interest” checkbox was created.
- Unfold the Lifeline category and select the “Asset Fire Damage” option and press Finish.

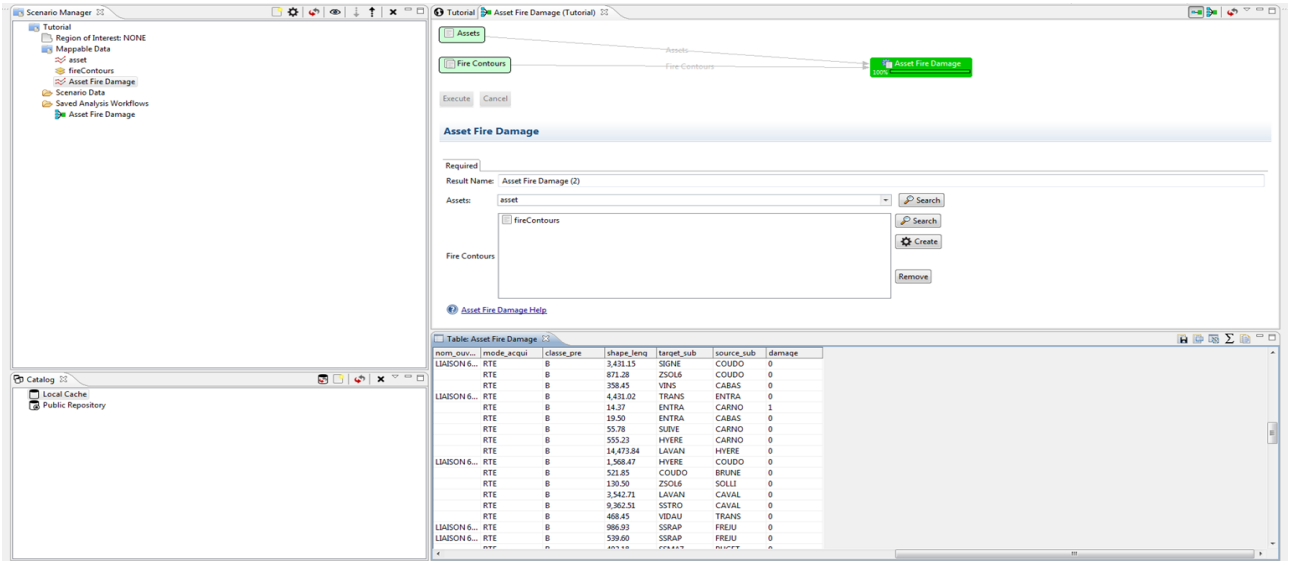


- The analysis graph View will be opened. Click on the blue boxes “Assets” and “Fire Contours” in order to select an RCP datasets.



- The blue box will become green as the input dataset has been chosen.
- Select a name for the output file
- Press the Execute button
- After some seconds the analysis execution will be completed.
- A new file will be created in the Scenario tree under the folder name “Asset Fire Damage”
- Double click the file to view its contents

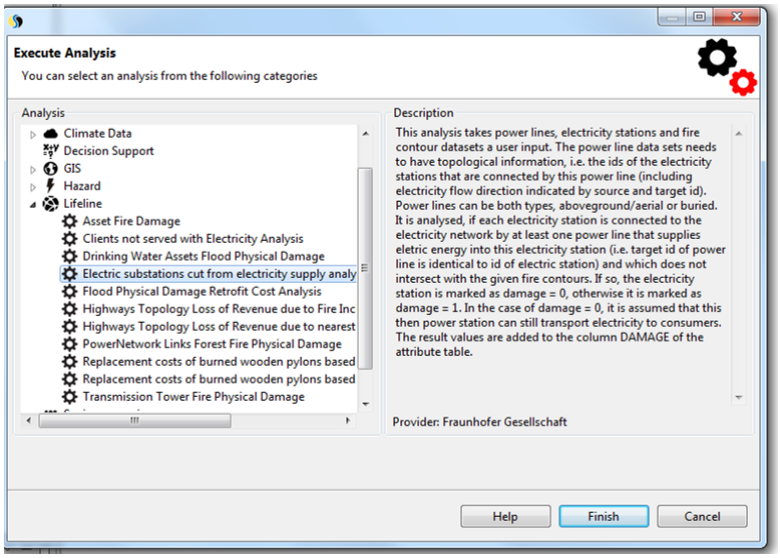




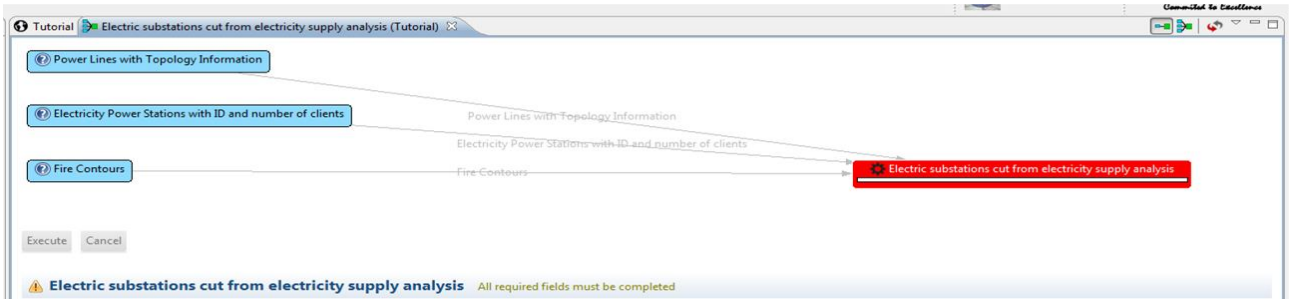
Electric substations cut from electricity supply analysis

This analysis takes power lines, electricity stations and fire contour datasets a user input. The power line data sets needs to have topological information, i.e. the ids of the electricity stations that are connected by this power line (including electricity flow direction indicated by source and target id). Power lines can be both types, aboveground/aerial or buried. It is analysed, if electricity station is connected to the electricity network by power line that is burned during fire incident or that is connected to another electricity substation that is connected to the electricity network by power line that is burned then remaining capacity of current substation is reduced to the value proportional to the whole network capacity loss value and the number of non served clients will consequently increase.

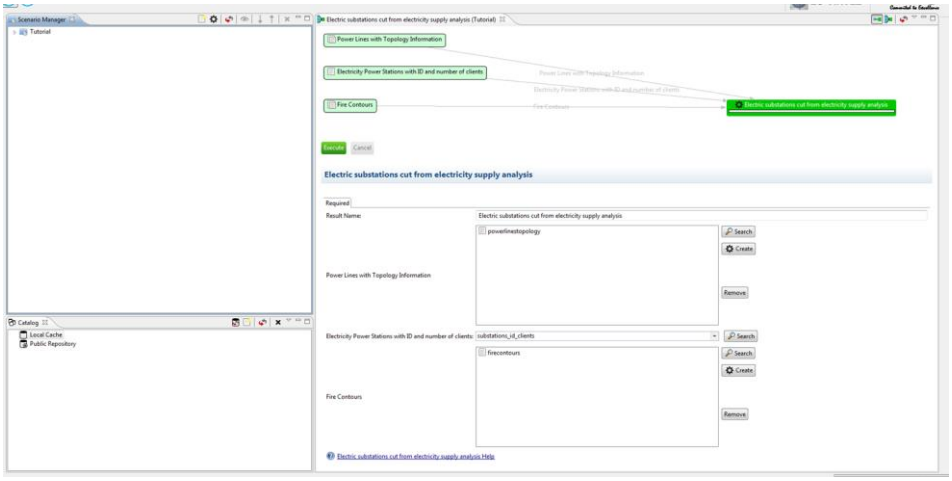
- Assuming that new scenario with unticked “Use Region of Interest” checkbox was created.
- Unfold the Lifeline category and select the “Electric substations cut from electricity supply analysis” option and press Finish.

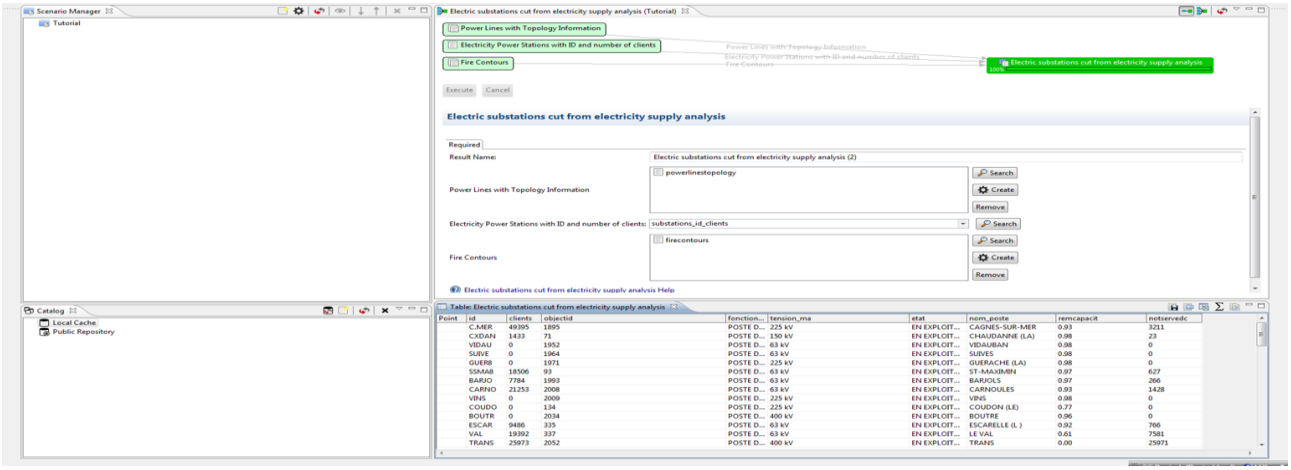


- The analysis graph View will be opened. Click on the blue boxes “Power Lines with Topology Information”, “Electricity Substations with ID and Number of Clients” and “Fire Contours” in order to select an RCP datasets.



- Press the “Search for a dataset” option and select an ingested dataset from the popup window. Then press Finish.
- The blue box will become green as the input dataset has been chosen.
- Select a name for the output file
- Press the Execute button
- After some seconds the analysis execution will be completed.
- A new file will be created in the Scenario tree under the folder name “Electric substations cut from electricity supply analysis”.
- Double click the file to view its contents

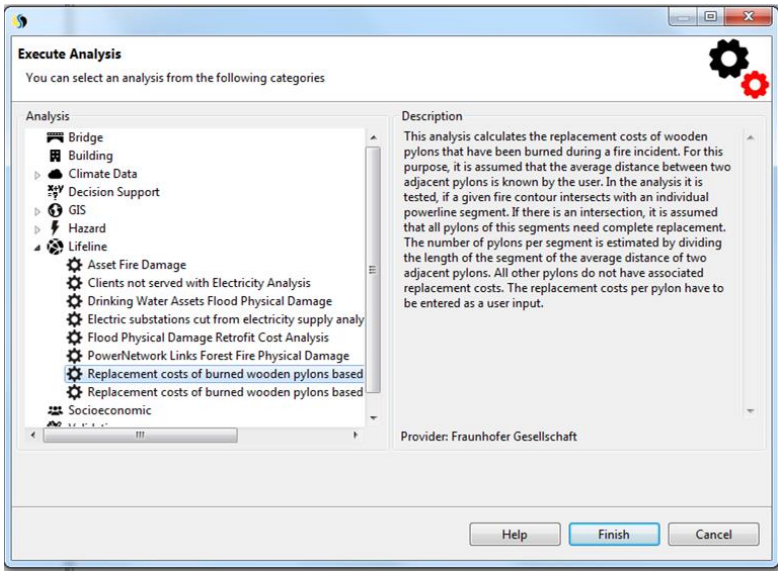




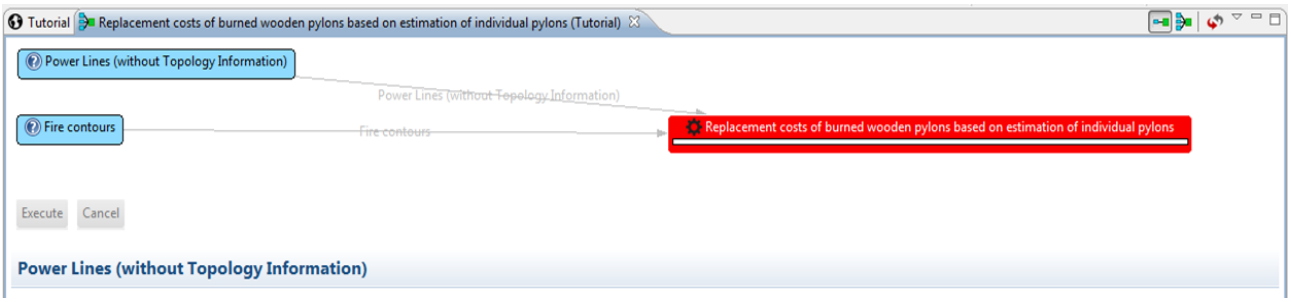
Estimated Number and Replacement Cost of wooden pylons Analysis

This analysis calculates the replacement costs of wooden pylons that have been burned during a fire incident. For this purpose, it is assumed that the average distance between two adjacent pylons is known by the user. In the analysis it is tested, if a given fire contour intersects with an individual powerline segment. If there is an intersection, it is assumed that all pylons of this segments need complete replacement. The number of pylons per segment is estimated by dividing the length of the segment of the average distance of two adjacent pylons. All other pylons do not have associated replacement costs. The replacement costs per pylon have to be entered as a user Input.

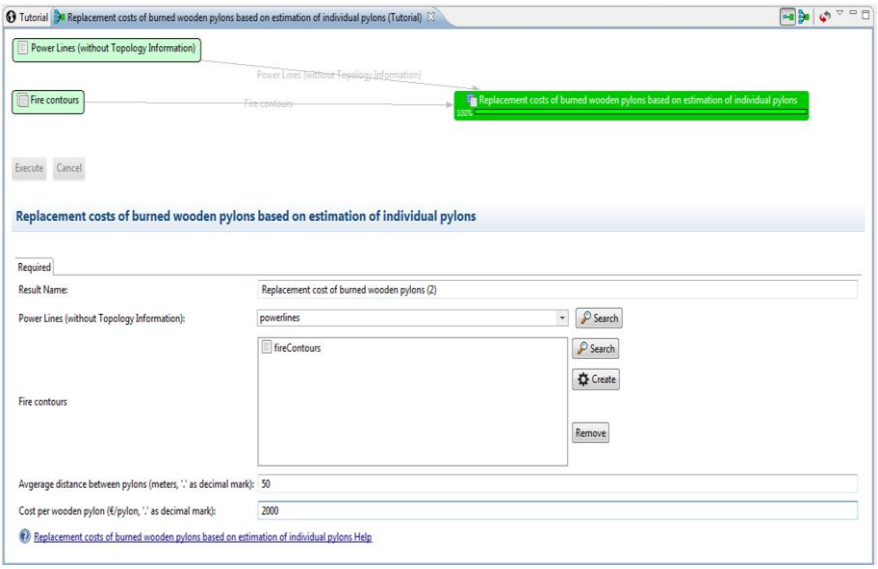
- Assuming that new scenario with unticked “Use Region of Interest” checkbox was created.
- Unfold the Lifeline category and select the “Replacement costs of burned wooden pylons based on estimation of individual pylons” option and press Finish.

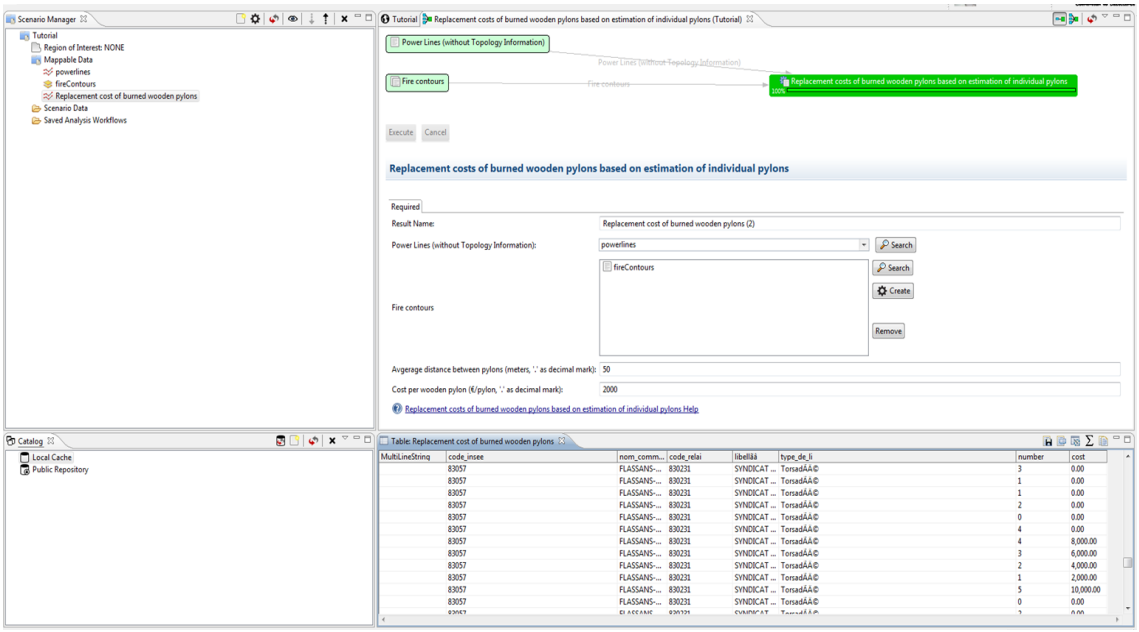


- The analysis graph View will be opened. Click on the blue boxes “Power Lines (without Topology Information)” and “Fire Contours” in order to select an RCP datasets



- Press the “Search for a dataset” option and select an ingested dataset from the popup window. Then press Finish.
- The blue box will become green as the input dataset has been chosen.
- Select a name for the output file
- Select the Average distance between pylons and the Cost per wooden pylon
- Press the Execute button
- After some seconds the analysis execution will be completed.
- A new file will be created in the Scenario tree under the folder name “Replacement cost of burned wooden pylons”.
- Double click the file to view its contents

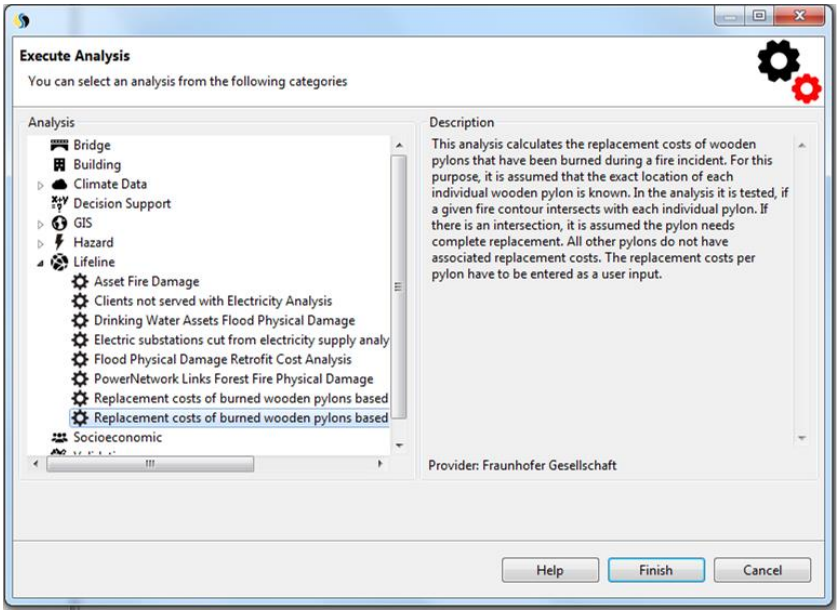




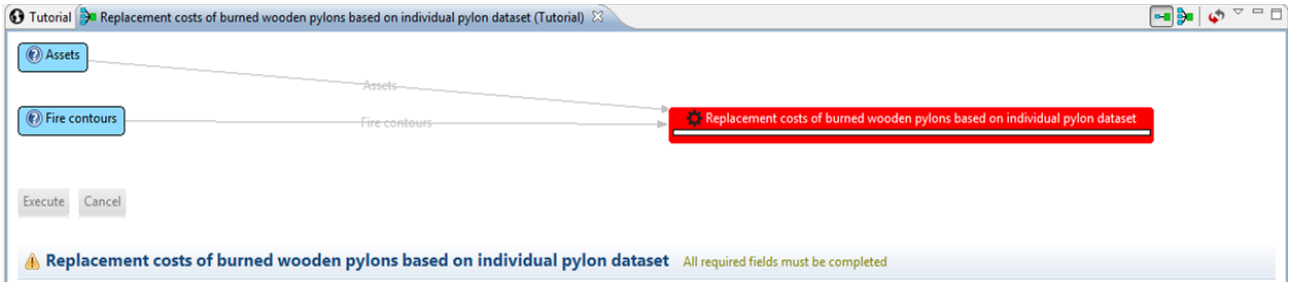
Replacement cost of wooden pylons Analysis

This analysis calculates the replacement costs of wooden pylons that have been burned during a fire incident. For this purpose, it is assumed that the exact location of each individual wooden pylon is known. In the analysis it is tested, if a given fire contour intersects with each individual pylon. If there is an intersection, it is assumed the pylon needs complete replacement. All other pylons do not have associated replacement costs. The replacement costs per pylon have to be entered as a user Input

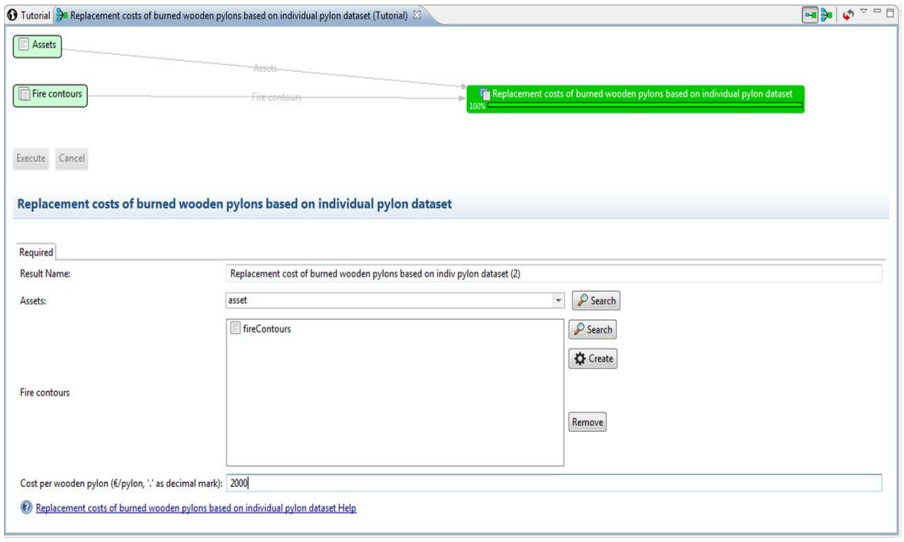
- Assuming that new scenario with unticked “Use Region of Interest” checkbox was created.
- Unfold the Lifeline category and select the “Replacement costs of burned wooden pylons based on individual pylon dataset” option and press Finish.

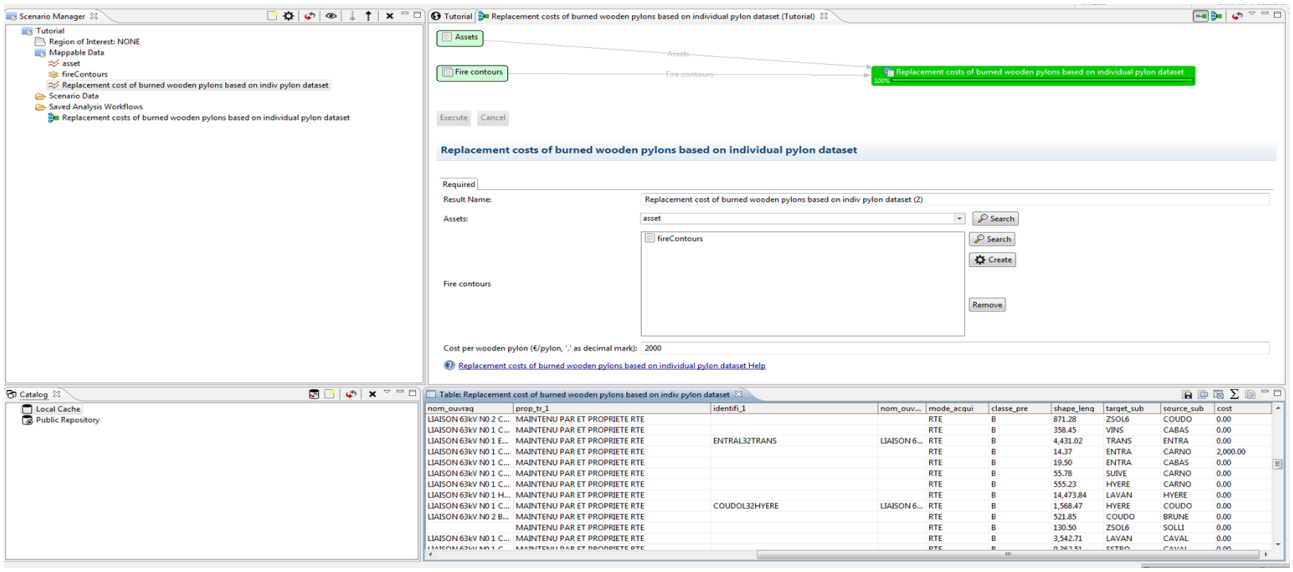


- The analysis graph View will be opened. Click on the blue boxes “Assets” and “Fire Contours” in order to select an RCP datasets.



- Press the “Search for a dataset” option and select an ingested dataset from the popup window. Then press Finish.
- The blue box will become green as the input dataset has been chosen.
- Select a name for the output file
- Select the Cost per wooden pylon
- Press the Execute button
- After some seconds the analysis execution will be completed.
- A new file will be created in the Scenario tree under the folder name “Replacement cost of burned wooden pylons based on indiv pylon dataset”.
- Double click the file to view its contents

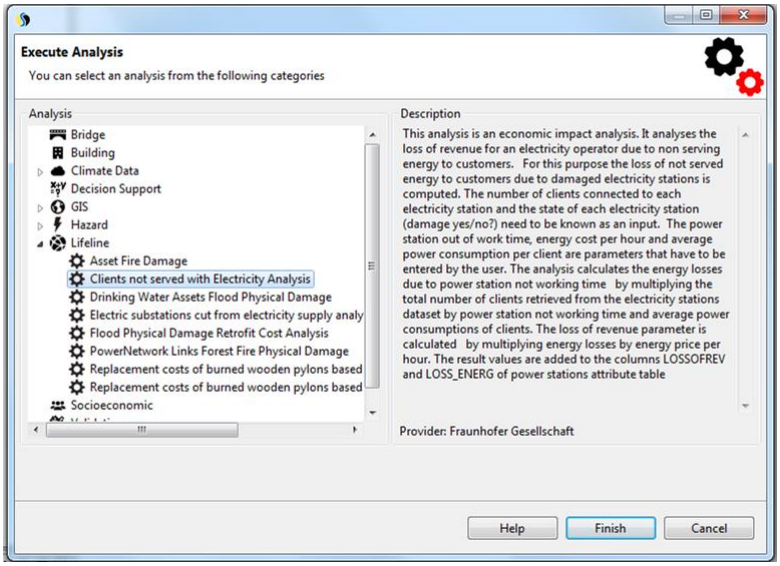




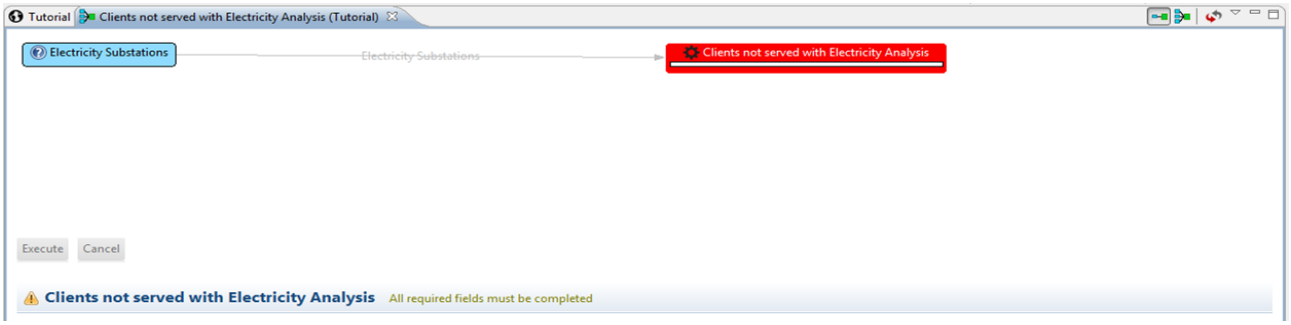
Clients not served with Electricity Analysis

This analysis is an economic impact analysis. It analyses the loss of revenue for an electricity operator due to non serving energy to customers. For this purpose the loss of not served energy to customers due to damaged electricity stations is computed. The number of clients connected to each electricity station and the state of each electricity station (damage yes/no?) need to be known as an input. The power station out of work time, energy cost per hour and average power consumption per client are parameters that have to be entered by the user. The analysis calculates the energy losses due to power station not working time by multiplying the total number of clients retrieved from the electricity stations dataset by power station not working time and average power consumptions of clients. The loss of revenue parameter is calculated by multiplying energy losses by energy price per hour.

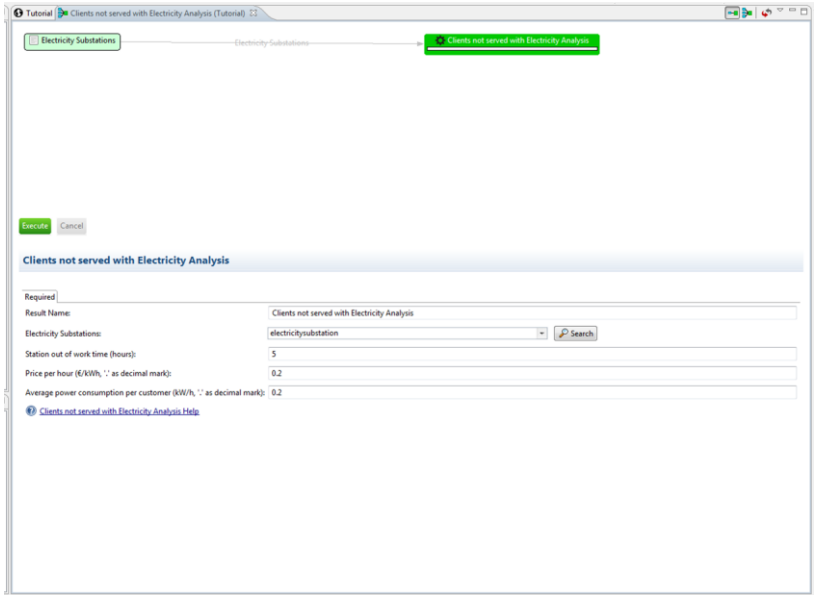
- Assuming that new scenario with unticked “Use Region of Interest” checkbox was created.
- Unfold the Lifeline category and select the “Clients not served with Electricity Analysis” option and press Finish.

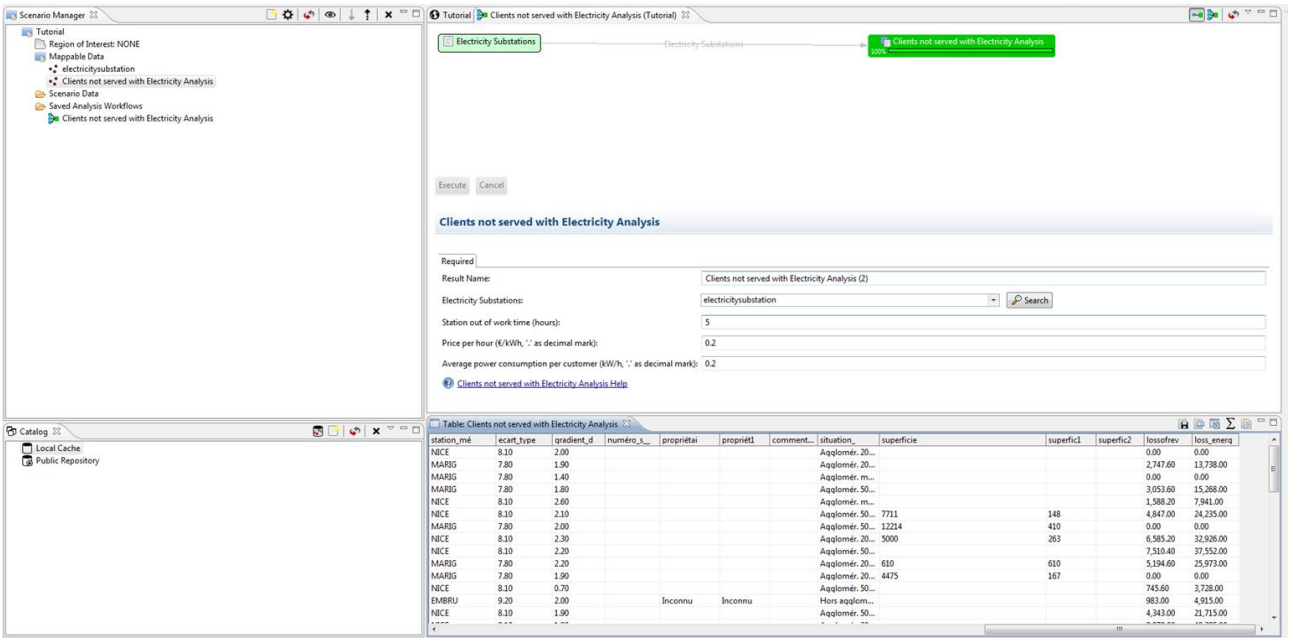


- The analysis graph View will be opened. Click on the blue boxes “Electricity Substations” in order to select an RCP datasets.



- Press the “Search for a dataset” option and select an ingested dataset from the popup window. Then press Finish.
- The blue box will become green as the input dataset has been chosen.
- Select a name for the output file
- Select the Station out of work time
- Select the Price per hour
- Select the Average power consumption per customer
- Press the Execute button
- After some seconds the analysis execution will be completed.
- A new file will be created in the Scenario tree under the folder name “Clients not served with Electricity Analysis”.
- Double click the file to view its contents

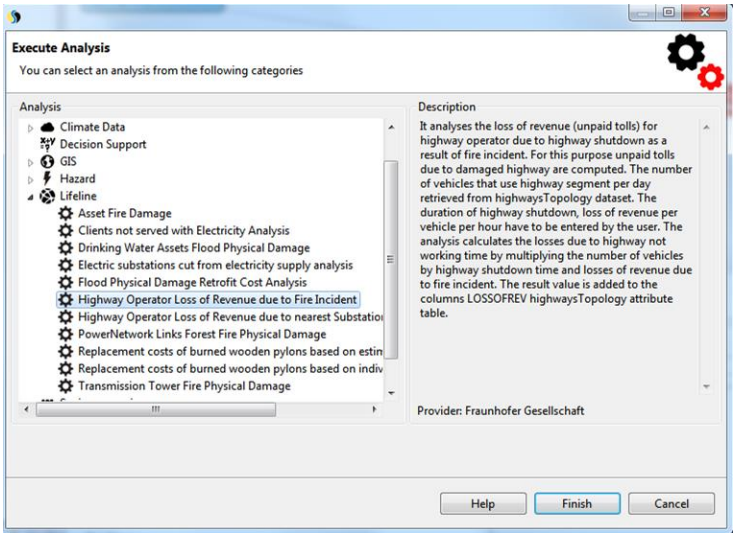




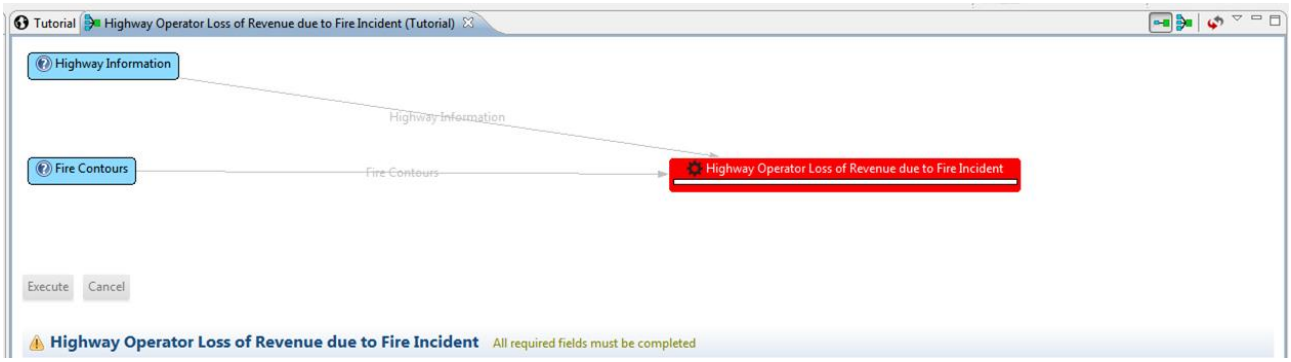
Highways Operator Loss of Revenue due to Fire Incident

It analyses the loss of revenue (unpaid tolls) for highway operator due to highway shutdown as a result of fire incident. For this purpose unpaid tolls due to damaged highway are computed. The number of vehicles that use highway segment per day retrieved from highwaysTopology dataset. The duration of highway shutdown, loss of revenue per vehicle per hour have to be entered by the user. The analysis calculates the losses due to highway not working time by multiplying the number of vehicles by highway shutdown time and losses of revenue due to fire incident.

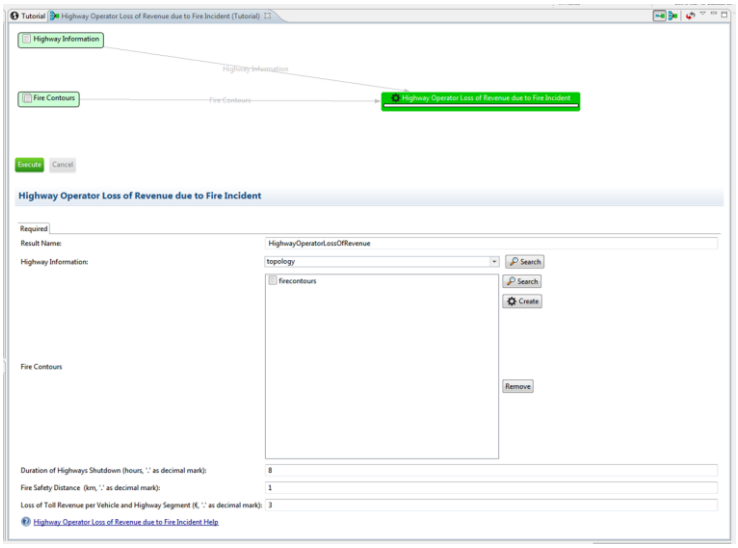
- Assuming that new scenario with unticked “Use Region of Interest” checkbox was created.
- Unfold the Lifeline category and select the “Highways Topology Loss of Revenue due to Fire Incident” option and press Finish.

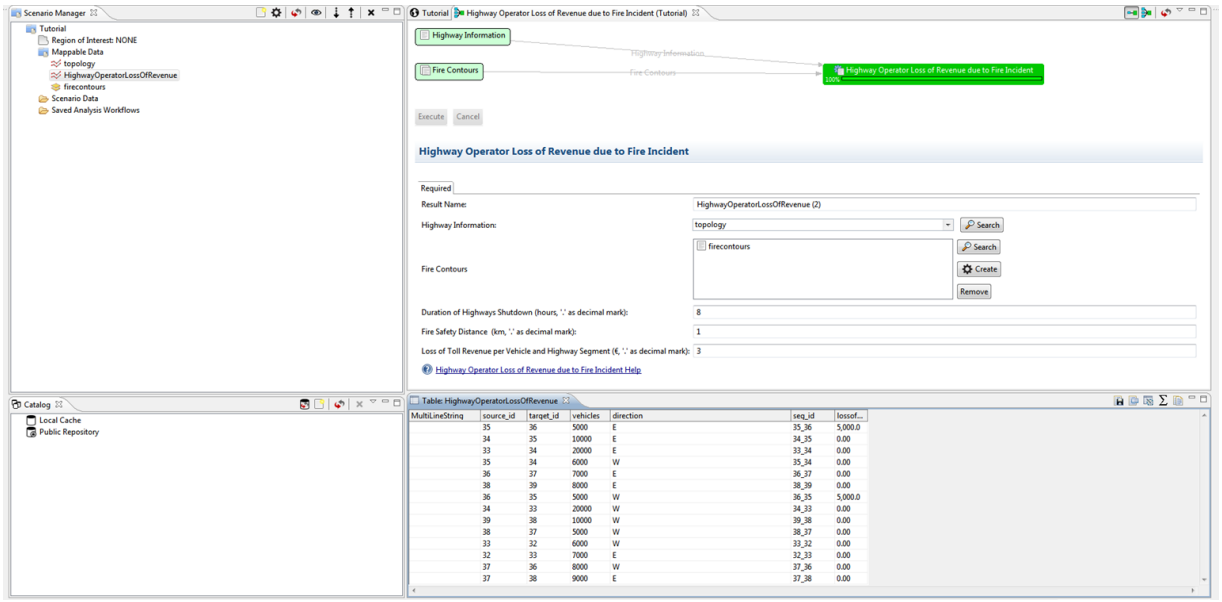


- The analysis graph View will be opened. Click on the blue boxes “Highway Information”, “Fire Contours” in order to select an RCP datasets.



- Press the “Search for a dataset” option and select an ingested dataset from the popup window. Then press Finish.
- The blue box will become green as the input dataset has been chosen.
- Select a name for the output file
- Select the Duration of Highways Shutdown
- Select the Fire Safety Distance
- Select the Loss of Toll Revenue per Vehicle and Highway Segmen
- Press the Execute button
- After some seconds the analysis execution will be completed.
- A new file will be created in the Scenario tree under the folder name “Highways Topology Loss of Revenue due to Fire Incident”.
- Double click the file to view its contents

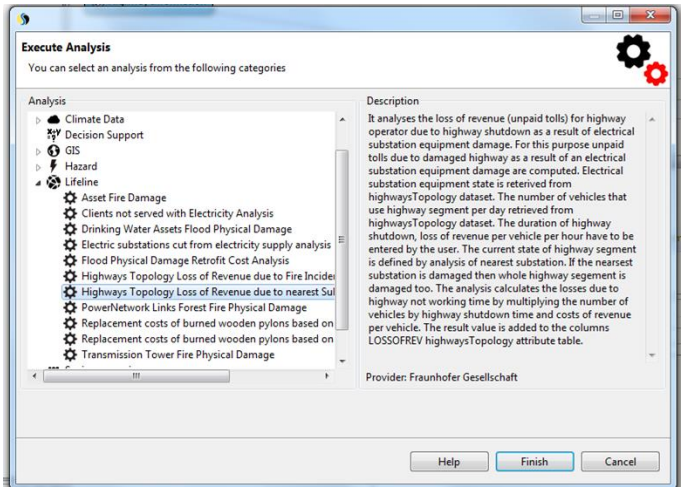




Highway Operator Loss of Revenue as cascading effect of electricity blackout

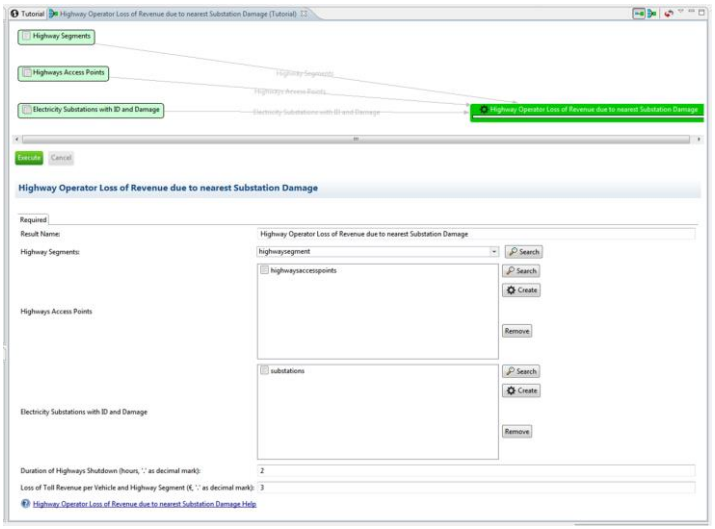
It analyses the loss of revenue (unpaid tolls) for highway operator due to highway shutdown as a result of electrical substation equipment damage. For this purpose unpaid tolls due to damaged highway as a result of an electrical substation equipment damage are computed. Electrical substation equipment state is retrieved from highwaysTopology dataset. The number of vehicles that use highway segment per day retrieved from highwaysTopology dataset. The duration of highway shutdown, loss of revenue per vehicle per hour have to be entered by the user. The current state of highway segment is defined by analysis of nearest substation. If the nearsest substation is damaged then whole highway segment is damaged too. The analysis calculates the losses due to highway not working time by multiplying the number of vehicles by highway shutdown time and costs of revenue per vehicle.

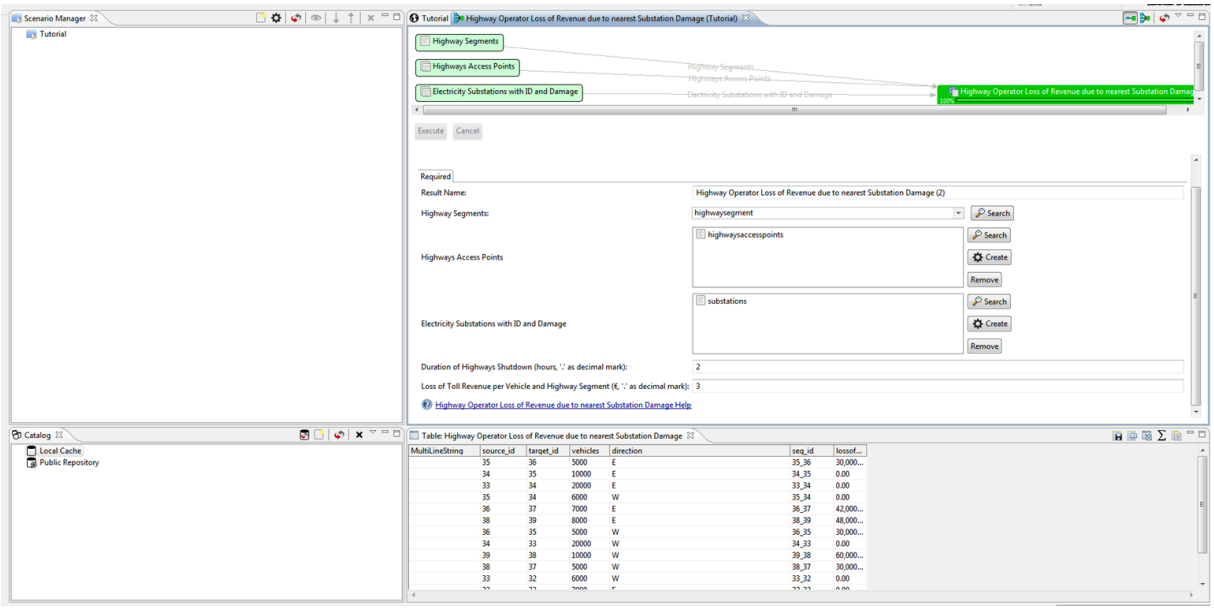
- Assuming that new scenario with unticked “Use Region of Interest” checkbox was created.
- Unfold the Lifeline category and select the “Highways Topology Loss of Revenue due to nearest Substation Damage” option and press Finish.



- The analysis graph View will be opened. Click on the blue boxes “Highway Segments, “Electricity Substations with ID and Damage”, “Highways Access Points” in order to select the datasets.

- Press the “Search for a dataset” option and select an ingested dataset from the popup window. Then press Finish.
- The blue box will become green as the input dataset has been chosen.
- Select a name for the output file
- Select the Duration of Highways Shutdown
- Select the Loss of Toll Revenue per Vehicle and Highway Segment
- Press the Execute button
- After some seconds the analysis execution will be completed.
- A new file will be created in the Scenario tree under the folder name “Highways Topology Loss of Revenue due to nearest Substation Damage”.
- Double click the file to view its contents





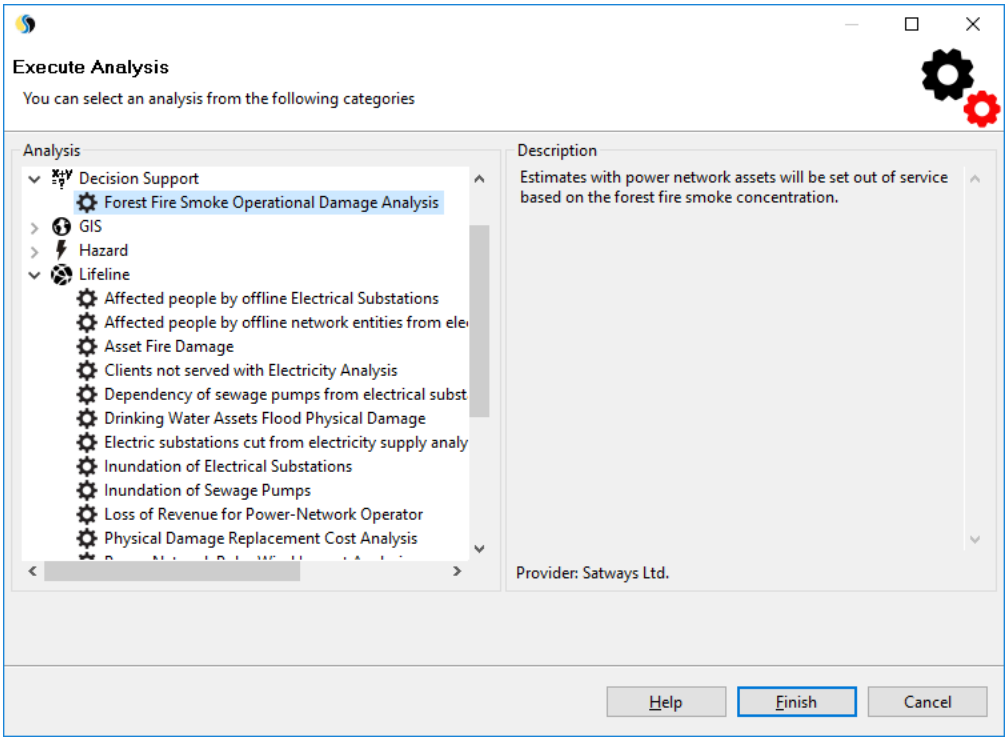
Forest Fire Smoke Operational Damage Analysis

In this analysis the following forest fire impacts are considered:

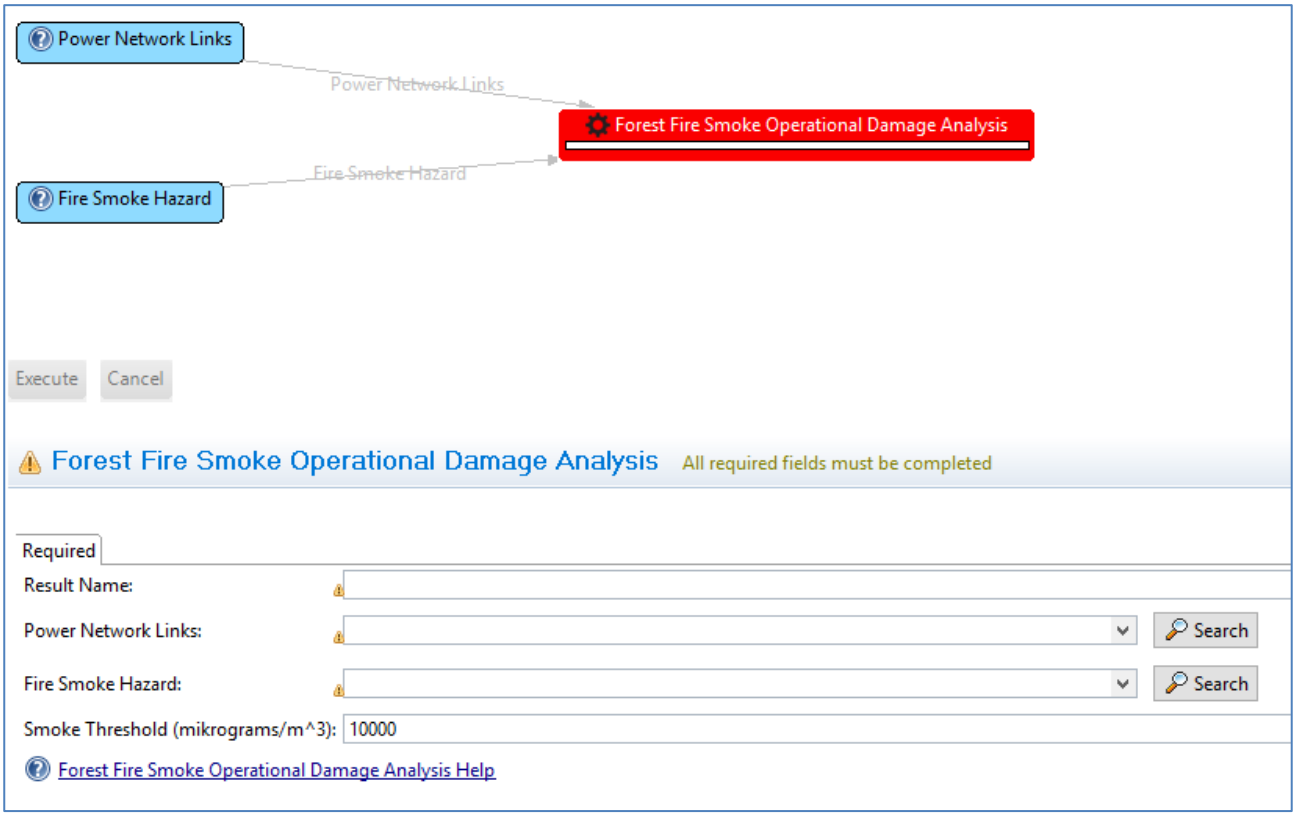
1. Operational damage: If smoke concentration over transmission line (overlay function) exceeds certain threshold then “cut” power
2. Maintenance actions: Smoke deposition (over entire event) exceeds certain threshold (Threshold is user defined)

Assuming that new scenario with unticked “Use Region of Interest” checkbox was created.

- Unfold the Decision Support category and select the “Forest Fire Smoke Operational Damage Analysis” option and press Finish.

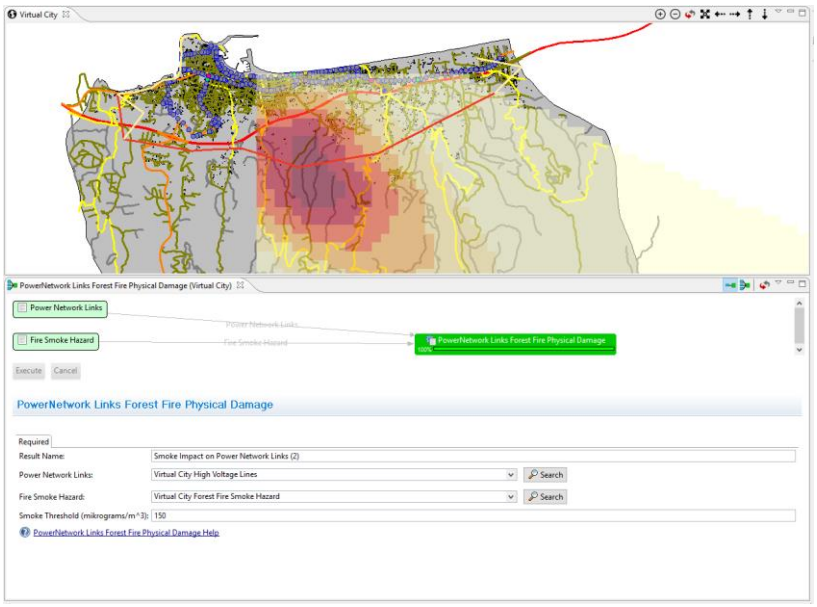


- The analysis graph View will be opened. Click on the blue boxes “Power Network Links” and “Fire Smoke Hazard” in order to select the datasets from your local repository.



- Press the “Search for a dataset” option and select an ingested dataset from the popup window. Then press Finish.
- The blue box will become green as the input dataset has been chosen.
- Select a name for the output file

- Enter the Ambient Temperature value
- Enter the smoke concentration threshold (mikrograms/m^3)
- All graph boxes will become green. Press the Execute button
- After some seconds the analysis execution will be completed and in the map the lines that need to be powered off are depicted in red color.

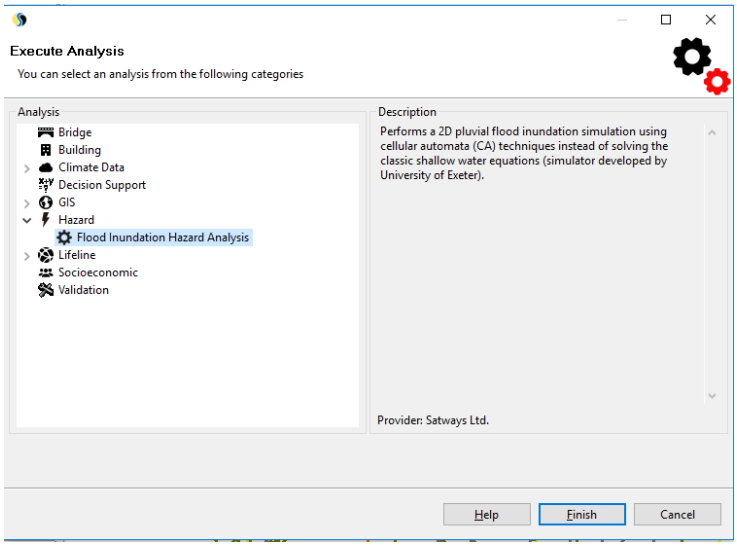


Flood Inundation Hazard Analysis

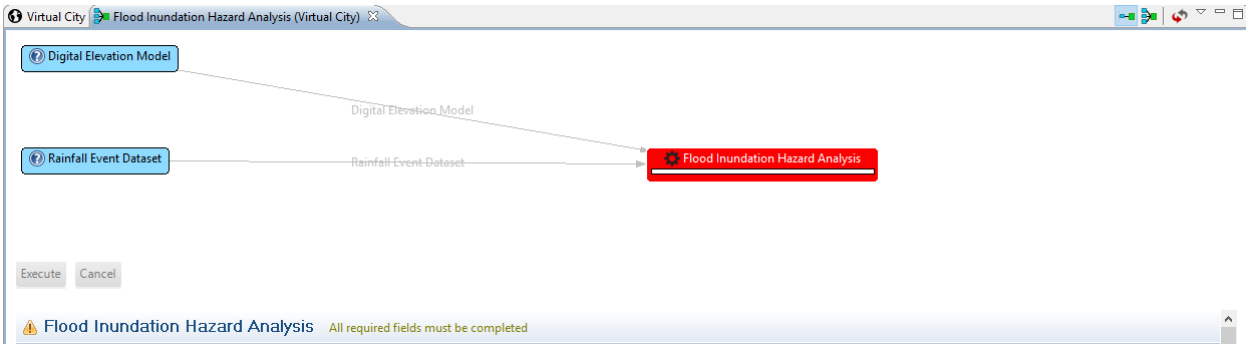
This analysis is based on the caflood simulator. The **caflood** application performs a 2D pluvial flood inundation simulation using cellular automata (CA) techniques instead of solving the classic shallow water equations (SWEs). This simplification dramatically reduces the computational load of a CA model in comparison to a physically based model.

The caflood application is part of the **CADDIES** framework/project which is final aim is to produce faster algorithms for handling dual drainage flood modelling, i.e. where the urban surface flow (major system) is combined with the sewer flow (minor system). This is achieved by using CA techniques together with modern parallel hardware.

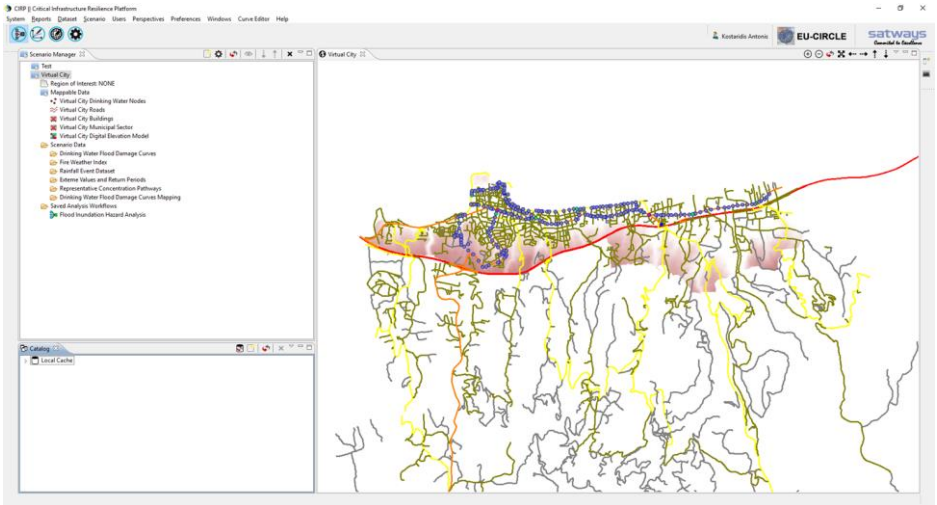
- Click the “New Analysis” option and choose the “Flood Inundation Hazard Analysis” from the “Hazard” analysis category



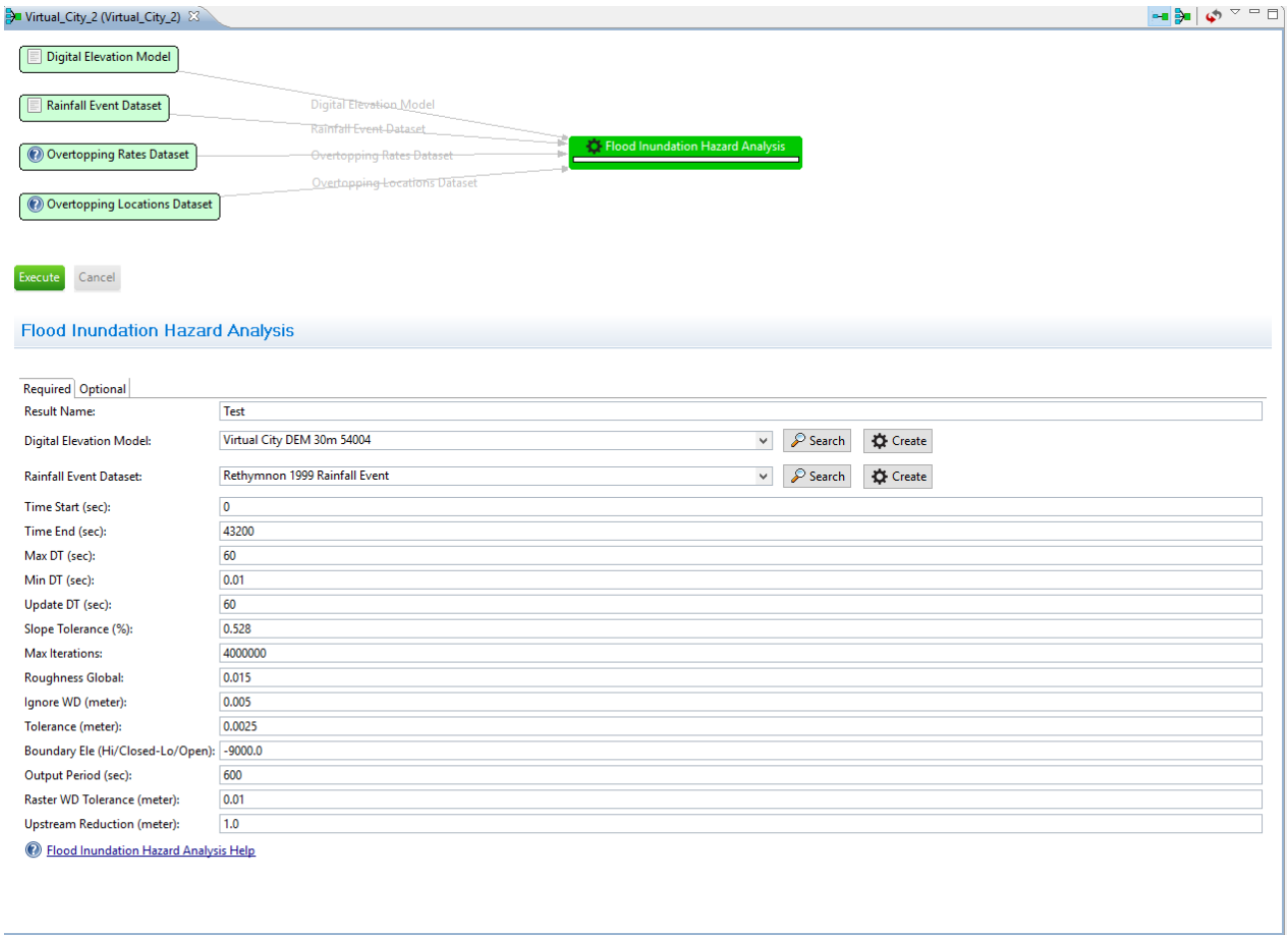
- Click the Digital Elevation Model box and select the DEM of the Virtual City. If no dataset exists then ingest a new dataset as “Raster” -> “Digital Elevation Model” type.



- The image below shows the loaded DEM of the area of interest and the GIS layers on top (road network, drinking water network)



- Click the “Rainfall Event Dataset” box and select the rainfall for your run. If no rainfall exists then ingest a new dataset as “Text File” -> “Rainfall Event Dataset”.
- Click the analysis box and select the parameters of CADDIES:



- Time Start (seconds) is generally equal to zero, meaning the first time step corresponding to the beginning of the event and it can be used to start a simulation from a certain point in time. In future version of the software, this allows the CADDIES-2D to resume a simulation from a previously produced output.
- Time End (seconds) is the timing to end of the simulation.
- Max DT (seconds) sets the upper limit for the time step DT. The DT is defined as $\Delta t = \Delta l / v_{max} \cdot \alpha$ where Δl is the length of a cell and v_{max} is the maximum velocity in the domain (α is introduced later). When velocity is approaching zero, the time step tends to be infinite and thus the computation would stop. Therefore an upper limit of DT has to be defined. The value of Δt is automatically updated by the program every update DT seconds (see later). Generally an upper limit of 60 seconds is recommended by the software developer.
- Min DT (seconds): the CADDIES-2D uses adaptive time step to capture the movement of flow front. The time step reduces as the maximum velocity increases. In contrast to the Max DT, very small time step may occur when the velocity is high and it will affect the model performance significantly. Therefore, it is mandatory to set a minimum value. Generally a lower limit of 0.01 seconds is recommended by the software developer.
- Update DT (seconds): Δt changes according to the velocity. Theoretically it should be update at every time step; however to compute Δt at each and every time step is a time consuming task. The user can set up an interval called update DT in which the same Δt t is used. At the end of each interval the new Δt is computed. In this version of the software the value cannot be over 60 and must be an integer factor of 60 such as 10, 20 or 30

seconds. Software developer recommends using 60 seconds to not affect the performance and to provide good accuracy.

- Slope Tolerance (%): in the WCA2Dv2 model, the time step Δt that the software uses at each update interval is computed using the formula suggested by Hunter et al. (2005):

$$\Delta t=⟦\Delta x⟧\text{^}2/4 \min(2n/R\text{^}(5⁄3)\ \ S\text{^}(1⁄2)\ ),S>\sigma$$

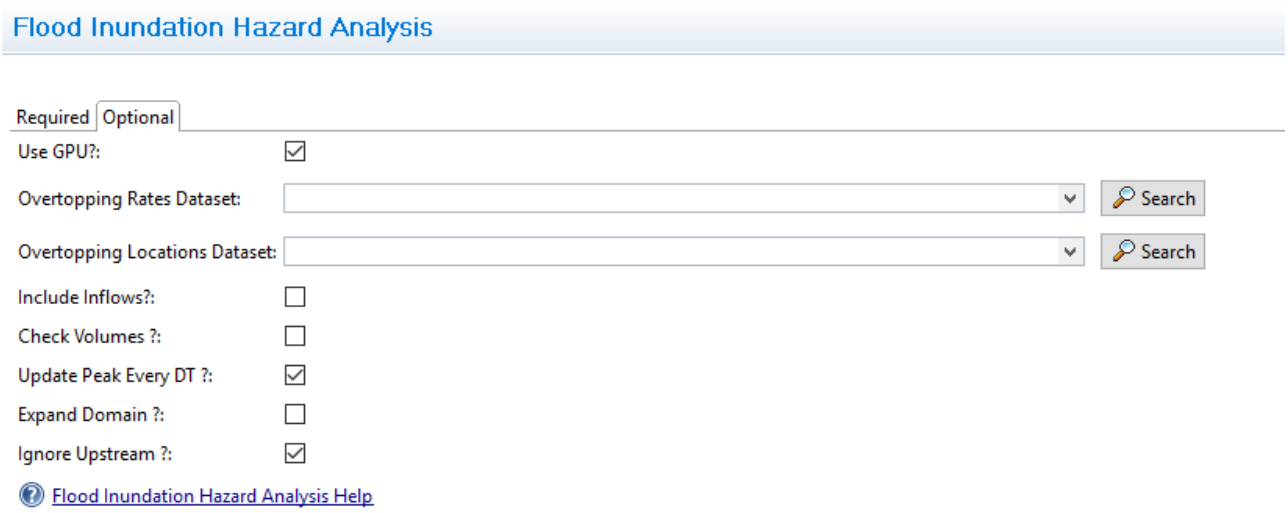
  This formula has the drawbacks that when the slope between two cells tends to be zero, the time step is driven to zero. Thus the slope tolerance σ is used to ignore all the cells where the water slope is less than the tollerance. A rule of thumb is to use an order of magnitude less than the average slope percent of the terrain used. For example, if the average slope is 5.28%, then the value of the slope tolerance should be 0.528.

- Max Iterations It is necessary to set a number just to be sure that the simulation stops in case Δt becomes very small.
- Roughness Global: this is the roughness value which is applied to all the cells of the domain.
- Ignore WD (meter): this parameter is used to speed up the model by ignoring the flow dynamic for cells with very shallow water. During the simulation, if a cell at a given time step has water depth (WD) < 0.005 m, the model assumes there is NO OUTFLOW for the cell and skips the computing. However, there might always be inflow from the surrounding cells. In other words, in order to have outflow from a given cell the condition WD ≥ 0.005 m must be satisfied.
- Tolerance (meter): This is another parameter used to speed up the simulation. During the simulation, if two neighbor cells have a water level difference that is less than the tolerance, then the caflood application assumes there is NO OUTFLOW between the pair and skips the computation.
- Boundary Ele (Hi/Closed-Lo/Open): this parameter sets the boundary condition. If a catchment (or a spatial domain) has very low boundary elements, the flow will be free to leave the domain and the exiting volume is a volume lost. On the contrary, by setting a high value of the boundary cell it is like virtually closing the domain and, at the downstream of the catchment or the exit point of the domain, the water keeps accumulating. For our purposes it is better to set a very low and negative value (e.g., -9000).
- Output Period (s): the output displayed on the screen is updated once every specified seconds from the start of the simulation (in this case 600s, i.e., once every ten minutes in the simulation time, no run-time). The number of outputs depends on the length of the simulation (Time End). For example a 2 hours simulation and an output period of 600 seconds produce twelve outputs to console.
- Output Computation Time: true or false. If the parameter is set as “true” then the computation time is printed on the screen. In the opposite case not.
- Check Volumes: true or false. Check that the mass conservation is respected or at least preserved within a certain error.
- Remove Proc Data (No Pre-Proc): true or false. When this command is activated it removes all the temporary CADDIES-2D data files generated during the simulation. Note: the files generated during the preprocessing phase are not involved by this command.
- Remove Pre-Proc Data: this command complements pt.31. If the parameter is set on “true” then all files created during preprocessing are also deleted.
- Raster VEL Vector Field: true or false. When this command is not activated (false) and there is a Raster Grid output-setup-file for the velocity, two output ASCII grid files are generated

which show respectively the speed and the angle of the velocity for each cell. If this command is activated (true), only a single output CSV file is generated where each line contains the velocity information of one cell. This information are: the X and Y coordinates of the cell centre, the speed, the angle in radian, and the angle in degree. This command is used to plot the velocity of the cells as vectors in GIS software instead than a raster grid.

- Raster WD Tolerance (meter): when caflood has run the simulation it takes the input of this instruction to set a threshold for wet and non-wet cells before saving the output. Referring to the example, all cells with less than 0.01 m of water depth will be treated as dry and therefore no water depth will be exported for those cells. All the raster output are influenced by this parameter, i.e. any eventual depth and velocity outputs. The user has to consider that the value of this parameter strongly depends on the vertical accuracy of digital elevation model, the type of terrain, the grid resolution and the quality of the precipitation and infiltration type of data.
- Upstream Reduction (meter): it is used by Ignore Upstream command to identify the amount of meter to reduce the possible elevation height at each update step.

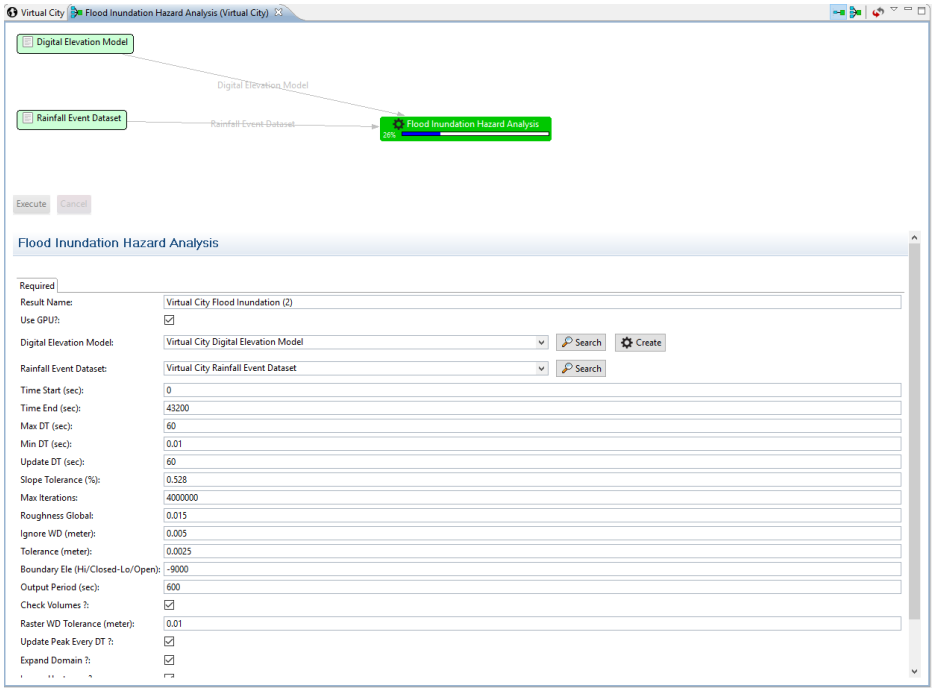
- Select the Optional parameters from the respective Tab



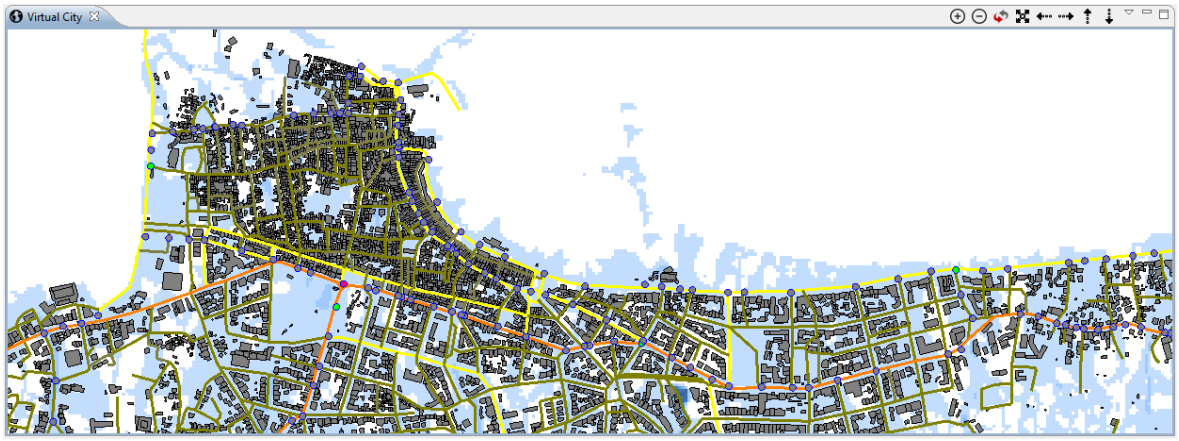
- Use GPU: Check if for GPU acceleration
- Update Peak Every DT: true or false. When this command is activated, the maximum/peak values of a physical variable are saved at every time step. The default is false; the maximum values are saved only every update DT (usually 60 seconds). Empirical tests showed that this command, when activated, has an insignificant impact on the accuracy but a large impact on the run time of the simulation.
- Expand Domain: true or false. When this command is not activated (false), the computational domain is the full domain, i.e. each cell with elevation data is processed. When this command is activated, the computational domain is set to be the same of the given events and it will expand when water reach the border of the computational domain. This command is useful to save run time when a simulation has a limited number of point sources of water like an inflow event from a small area.
- Ignore Upstream: true or false. When this command is activated the model identifies the elevation height where the water is still, i.e. no outflow is generated during an update step. Once a new height is identified all cells with elevation above the height are not considered for computation any more. This can save some run-time depending on the terrain and on

the type of events modeled. At the worst case scenario, it might add around 1% of extra run-time.

- When all input datasets and mandatory parameters have been chosen the blue boxes will become green. Then press the Execute button.



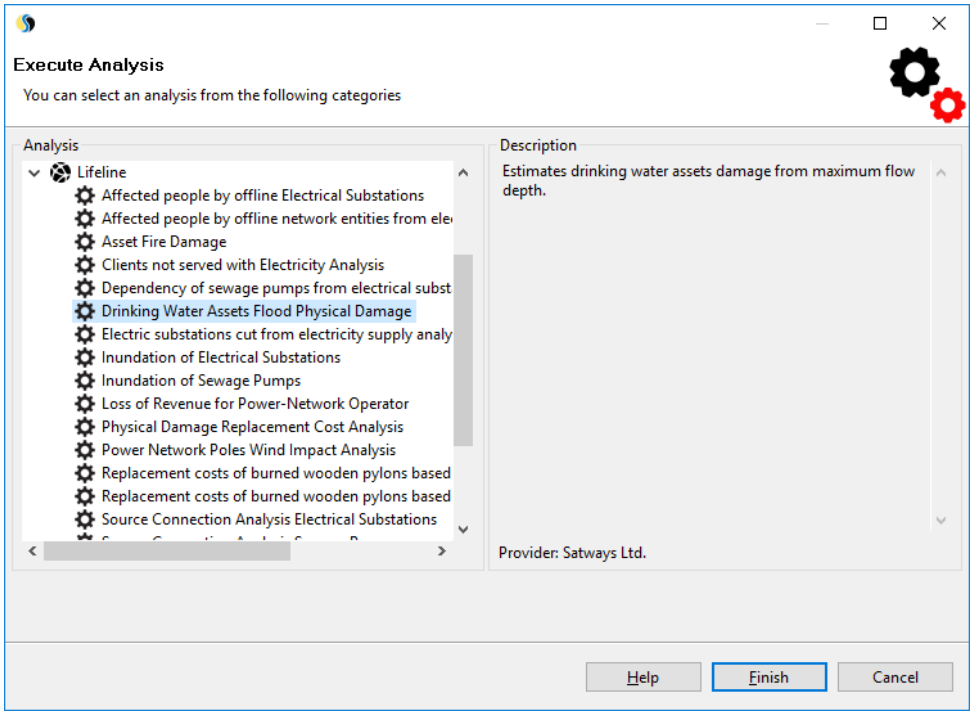
- Upon completion the flood maximum water depth raster is created and automatically ingested into the local repository (see Figure below).



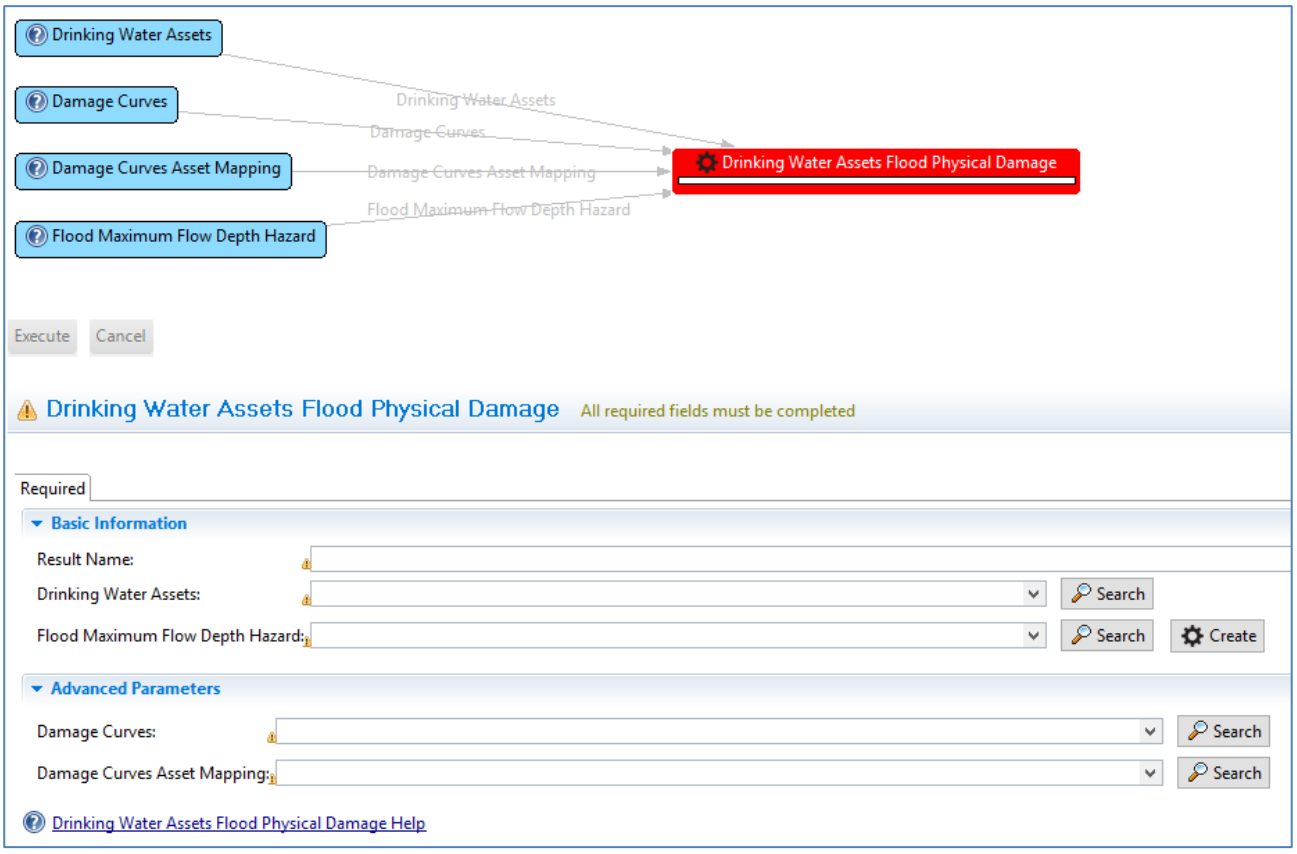
Flood Physical Impact on Drinking Water Assets

This analysis calculates the flood impact on drinking water assets according to the HAZUS methodology. As such is uses Damage curves and associated mapping XML files that represent the physical impact per asset characteristics according to the maximum water depth.

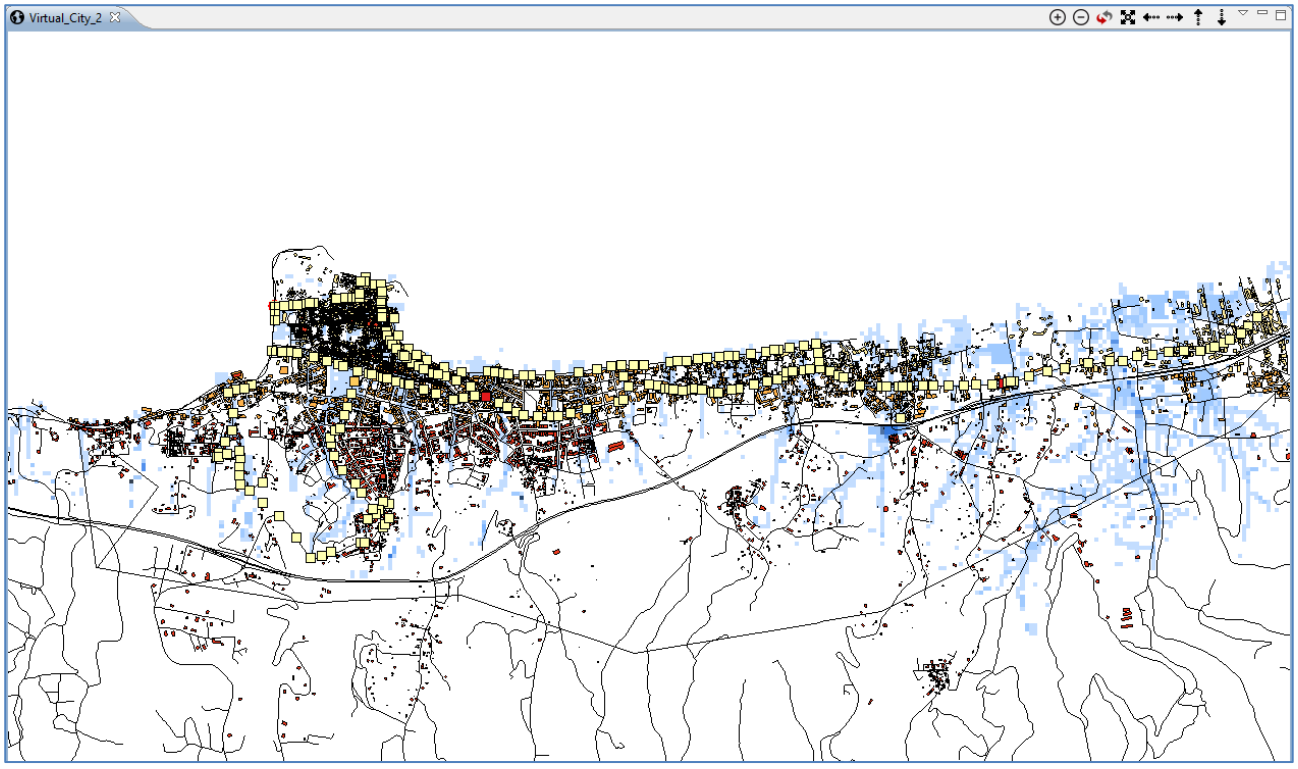
- Click the “New Analysis” option and choose the “Flood Physical Impact on Drinking Water Assets” from the “Lifelines” analysis category



- Click the Drinking Water Assets, Damage Curves, Damage Curves Asset Mapping and Flood Maximum Flow Depth Hazard boxes and select ingested data types from your repositories.



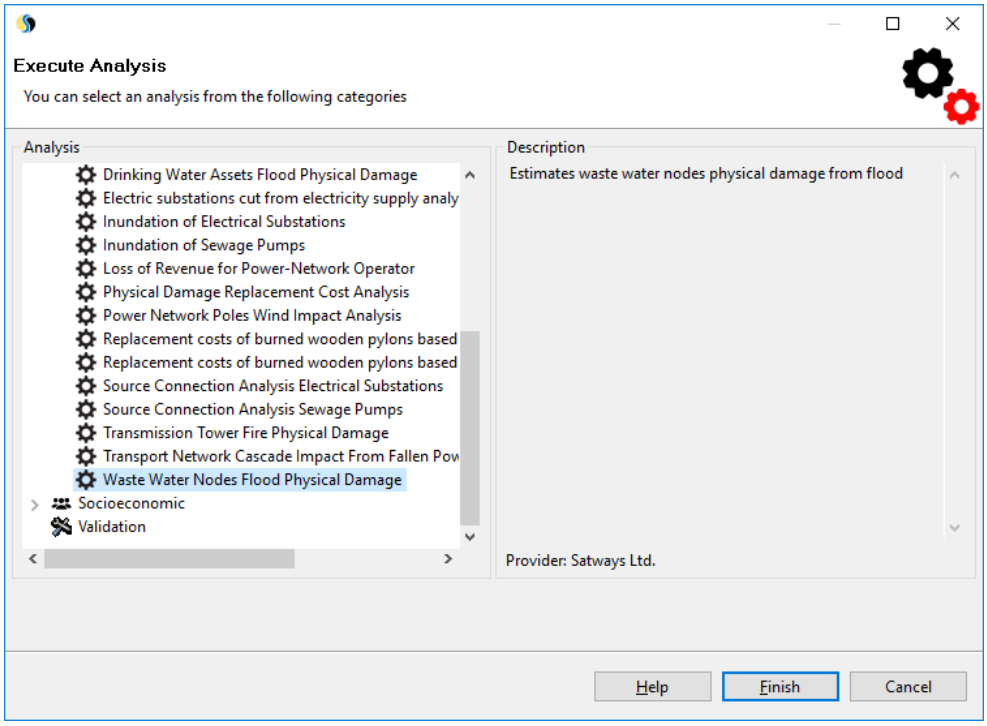
- Enter a name for the resulting file and press the Execute button.
- Upon completion a shapefile is produced based on the input Drinking Water assets shapefile and the dataset is shown on the Map with proper styling depending on the damage percent.



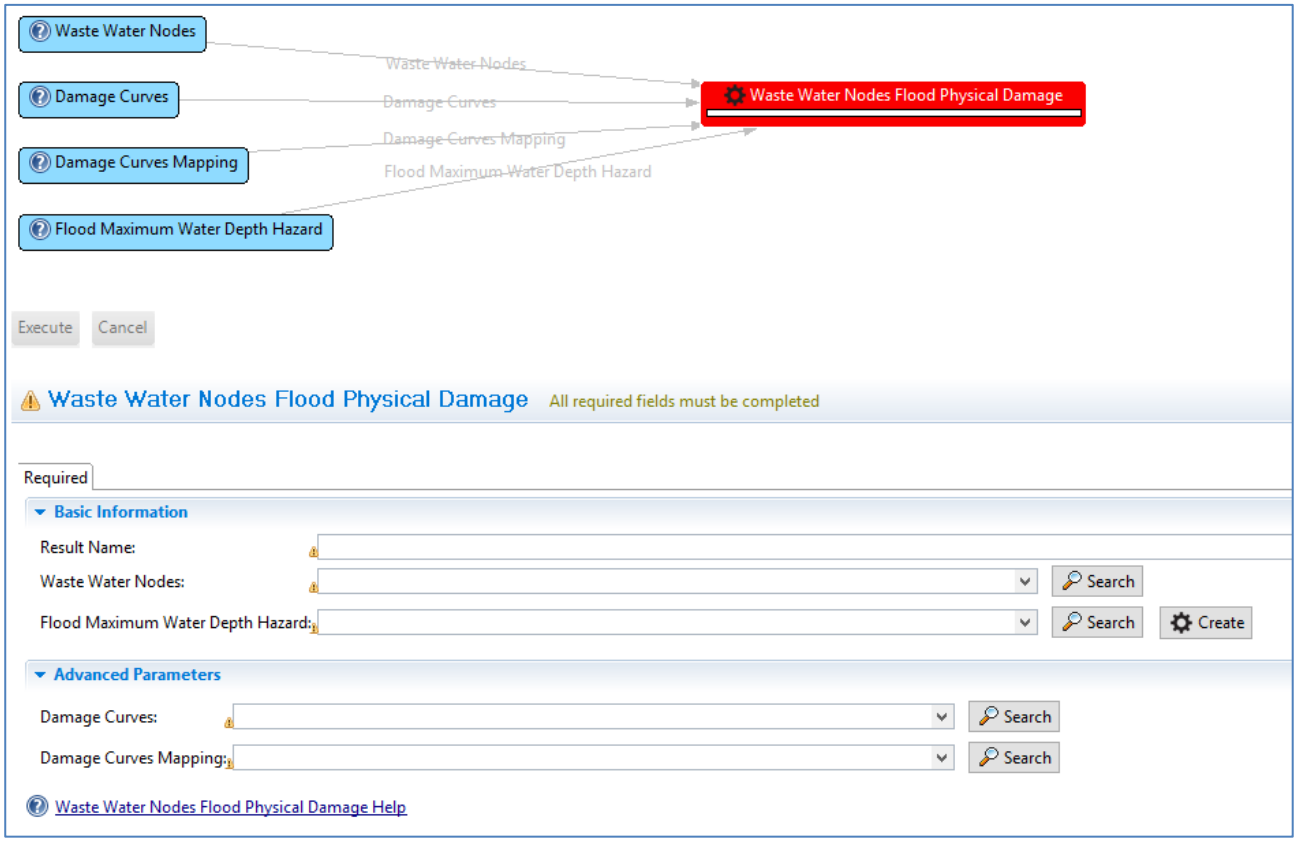
Flood Physical Impact on Waste Water Assets

This analysis calculates the flood impact on waste water assets according to the HAZUS methodology. As such is uses Damage curves and associated mapping XML files that represent the physical impact per asset characteristics according to the maximum water depth.

- Click the “New Analysis” option and choose the “Flood Physical Impact on Waste Water Assets” from the “Lifelines” analysis category



- Click the Waste Water Assets, Damage Curves, Damage Curves Asset Mapping and Flood Maximum Flow Depth Hazard boxes and select ingested data types from your repositories.



- Enter a name for the resulting file and press the Execute button.
- Upon completion a shapefile is produced based on the input Waste Water assets shapefile and the dataset is shown on the Map with proper styling depending on the damage percent.

3 CADDIES 2D user manual

CADDIES 2D flood model has been integrated within CIRP in the EU-CIRCLE project. The model can be executed from CIRP and the results can be displayed within CIRP. It is also possible to run CADDIES as an independent tool and the following is the detailed instruction of using CADDIES.

3.1 Introduction

The **caflood** application performs a 2D pluvial flood inundation simulation using cellular automata (CA) techniques instead of solving the classic shallow water equations (SWEs). The CA technique offers a versatile method for modelling complex physical systems using simple operations (Wolfram 1984). This simplification dramatically reduces the computational load of a CA model in comparison to a physically based model.
CA model usually consists of five essential features: a discrete space, the distribution of the neighbour cells, the state of the cells, the discrete time step and the transition rules (Itami 1994). The transition rules are composed of simple operations that govern the evolution of each cell's state. This makes use of the previous state of the cell itself and those in its neighbourhood. Since computing the new state of a cell does not depend on the state of any other cells at the same time step, CA algorithms are well suited to parallel computation.
The caflood application is part of the **CADDIES** framework/project which is final aim is to produce faster algorithms for handling dual drainage flood modelling, i.e. where the urban surface flow (major system) is combined with the sewer flow (minor system). This is achieved by using CA techniques together with modern parallel hardware.

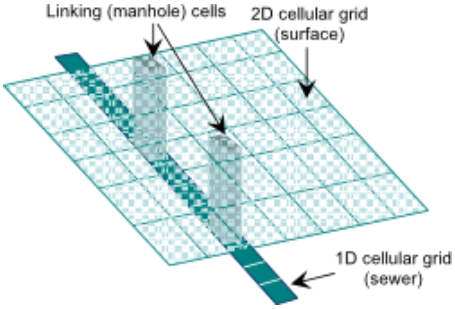


Figure 6. Example of interactions between 2D surface flow and 1D sewer flow via linking elements (manholes)

The caflood application implements the Weighted Cellular Automata 2D model (**WCA2D**) with simplified transition rules to simulate inundation events instead of using complex physically based equations and mathematical operations. This model improves the methodology adopted in the CA2D model (Ghimire et al. 2013) using a weight-based system. The WCA2D is a diffusive-like model that ignores inertia terms and momentum conservation. The model has been designed to work with various general grids, (e.g., rectangular, hexagonal or triangular grid) with different neighbourhood types (e.g., the four cells of the von-Neumann (VN) neighbourhood or the eight cells of the Moore neighbourhood).
The major points of this model are:

- The ratio of water transferring from the central cell to the downstream neighbour cells (intercellular-volume) is calculated using a minimalistic and quick weight-based system;
- The volume of water transferring between the central cell and the neighbour cells is limited by a single equation, which is a composition of a simplified Manning's formula and the critical flow condition.
- Both the adaptive time step and the velocity, as an average velocity, are evaluated within a larger update time step to speed up the simulations.

Additional information about the WCA2D model is available in (Guidolin et al. 2015)
The caflood application uses the **CADDIES Application Programming Interface** (API) to implement the 2D model (Guidolin et al. 2012). The CADDIES API defines a standard set of methods, data structures and

variables that can be used to develop parallel CA algorithms. A developer needs to write the code of the CA model only once. After that, the CADDIES API gives the flexibility to produce the same CA model for any type of CA grid, square/hexagonal/triangular grid, and to use different high performance acceleration techniques without changing the code or with minimum effort.
In caflood, the WCA2D model has been implemented using only a square cell grid with von-Neumann neighbourhood. Furthermore, thanks to the CADDIES API, a simulation can be executed in a multi-core CPU using OpenMP library (Dagum and Menon 1998) and in a multi-core CPU and on a graphics card GPU using the OpenCL library (Munshi and others 2011).

3.2 CAFLOOD EXECUTABLE

The caflood application is composed of a single executable, called '**caflood**', which can be executed from a command line shell, like Windows PowerShell of UNIX Bash. As previously mentioned, caflood can run a simulation on a multi-core CPU or in a graphics card GPU. Furthermore, the simulation can be performed with either single precision or double precision floating point values. These different characteristics are identified at compile time to improve performance; thus, it is possible to have multiple versions of the caflood executable.
The features of the application are:

- Can run on multi-core CPU and on graphics card GPU;
- Can be easily controlled by simple CSV set-up files;
- Uses an adaptive time step;
- Can simulate three types of events (rain fall, inflow and water level rise/fall) in all the domain or in a specific area of the domain;
- Uses a single roughness value for all the domain;
- Can compiled to use single precision or double precision floating point values;
- Takes as input only ASCII/GRID DTM/DEM files with projected coordinate system;
- Can produce ASCII/GRID raster file of the water depth/level and velocity at specific time steps and for the maximum values;
- Can produce CSV files with the time plot of the water depth/level and velocity at specific points in the domain.

This caflood application was tested only on a Windows 7 64 bits operating system (OS) and on a Linux 64 bit OS.

3.3 Execution and configuration file examples

In order to run a single simulation using the compiled executable, **caflood** requires three arguments, which must be given in the following order when the command is executed: 1) an input folder, 2) a setup-file, and 3) an output folder. The user needs to produce a comma separated value (CSV) setup-file which can have any name but must be located inside the input folder. The file is a list of command, files, instructions and parameters that the model need to run a simulation and produce one or more output. In the next paragraph, there is a synthetic example of a setup-file called Sim.csv.

[Sim.csv]

1. Simulation Name , Simple Sloping Domain Simulation
2. Short Name (for outputs) , simple
3. Pre-proc Name , simple
4. Version , 1,0,0
5. Model Type , WCA2Dv2
6. Time Start (seconds) , 0
7. Time End (seconds) , 43200

8. Max DT (seconds) , 60
9. Min DT (seconds) , 0.01
10. Update DT (seconds) , 60
11. Slope Tolerance (%) , 0.528
12. Max Iterations , 4000000
13. Roughness Global , 0.015
14. Ignore WD (meter) , 0.005
15. Tolerance (meter) , 0.0025
16. Boundary Ele (Hi/Closed-Lo/Open) , -9000
17. Elevation ASCII , simple.asc
18. Rain Event CSV , Rain40mm60min.csv
19. Water Level Event CSV ,
20. Inflow Event CSV ,
21. Time Plot CSV , WLpoints.csv, VELpoints.csv
22. Raster Grid CSV , WDraster.csv, VELraster.csv
23. Output Console , true
24. Output Period (s) , 600
25. Output Computation Time , true
26. Check Volumes , true
27. Remove Proc Data (No Pre-Proc) , true
28. Remove Pre-Proc Data , true
29. Raster VEL Vector Field , true
30. Raster WD Tolerance (meter) , 0.01
31. Update Peak Every DT , false
32. Expand Domain , false
33. Ignore Upstream , true
34. Upstream Reduction (meter) , 1.0

This file gives the input instruction for a single simulation. Instructions include the file name, the prefix name which will be used to save the data output files, some model parameters (roughness, time step, input and output setup files etc.). The input and output data needed and produced by the simulation are specified by multiple input-setup-files and output-setup-files (some examples will be presented later), respectively, that must be located in the input folder. The details of each instruction line are described as following:

1. **Simulation Name** is the full name of the simulation.
2. **Short Name** is the short cut of the Simulation Name. The string here defined will be used as a prefix to the names of data output files. The suffix of the data output files is composed of the name of the output-setup-file and one between the time when the data was generated and the name PEAK for the maximum values. In the example case all data output files (velocity, water depth, etc…) will be named automatically simple_WDraster_1800.asc, simple_VELraster_PEAK.asc, simple_WLpoints.csv, etc…
3. **Pre-proc Name** in principle is the same as Short Name. This is used to name the preprocessing files created during the initial phase of the simulation. This parameter is useful when a catchment needs multiple different events to be simulated, so the pre-processed catchment data is loaded faster between simulations.
4. **Version** is used to indicate the version of the simulation executed. It can be used to identify the console output of same simulation when it is executed with different parameters. It does not have any extra impact on the simulation and it is safe to leave it as ‘1.0.0’.
5. **Model Type** is used to indicate the flood model utilized during the simulation. The caflood application is designed to run multiple models. This version can run two models: WCA2Dv1 and WCA2Dv2. The

former is the first version of the weight-based model and is going to be deprecated. The latter is the second version which is the one used in the last journal article (Guidolin et al 2015).

6. **Time Start (seconds)** is generally equal to zero, meaning the first time step corresponding to the beginning of the event and it can be used to start a simulation from a certain point in time. In future version of the software, this allows the CADDIES-2D to resume a simulation from a previously produced output.
7. **Time End (seconds)** is the timing to end of the simulation.
8. **Max DT (seconds)** sets the upper limit for the time step DT. The DT is defined as

$$\Delta t = \frac{\Delta l}{v_{max}} \cdot \alpha$$

Where Δl is the length of a cell and v_{max} is the maximum velocity in the domain (α is introduced later). When velocity is approaching zero, the time step tends to be infinite and thus the computation would stop. Therefore an upper limit of DT has to be defined. The value of Δt is automatically updated by the program every *update DT* seconds (see later). Generally an upper limit of 60 seconds is recommended by the software developer.

9. **Min DT (seconds)**: the CADDIES-2D uses adaptive time step to capture the movement of flow front. The time step reduces as the maximum velocity increases. In contrast to the Max DT, very small time step may occur when the velocity is high and it will affect the model performance significantly. Therefore, it is mandatory to set a minimum value. Generally a lower limit of 0.01 seconds is recommended by the software developer.
10. **Update DT (seconds):** Δt changes according to the velocity. Theoretically it should be update at every time step; however to compute Δt at each and every time step is a time consuming task. The user can set up an interval called update DT in which the same Δt t is used. At the end of each interval the new Δt is computed. In this version of the software the value cannot be over 60 and must be an integer factor of 60 such as 10, 20 or 30 seconds. Software developer recommends using 60 seconds to not affect the performance and to provide good accuracy.
11. **Slope Tolerance (%):** in the WCA2Dv2 model, the time step Δt that the software uses at each update interval (see. Pt. 9) is computed using the formula suggested by Hunter et al. (2005) :

$$\Delta t = \frac{\Delta x^2}{4} min\left(\frac{2n}{R^{5/3}} S^{1/2}\right), S > \sigma$$

This formula has the drawbacks that when the slope between two cells tends to be zero, the time step is driven to zero. Thus the slope tolerance σ is used to ignore all the cells where the water slope is less than the tollerance. A rule of thumb is to use an order of magnitude less than the average slope percent of the terrain used. For example, if the average slope is 5.28%, then the value of the slope tolerance should be 0.528.

12. **Max Iterations** It is necessary to set a number just to be sure that the simulation stops in case Δt becomes very small.
13. **Roughness Global:** this is the roughness value which is applied to all the cells of the domain.
14. **Ignore WD (meter):** this parameter is used to speed up the model by ignoring the flow dynamic for cells with very shallow water. During the simulation, if a cell at a given time step has water depth (WD) < 0.005 m, the model assumes there is NO OUTFLOW for the cell and skips the computing. However, there might always be inflow from the surrounding cells. In other words, in order to have outflow from a given cell the condition WD ≥ 0.005 m must be satisfied.
15. **Tolerance (meter):** This is another parameter used to speed up the simulation. During the simulation, if two neighbor cells have a water level difference that is less than the tolerance, then the caflood application assumes there is NO OUTFLOW between the pair and skips the computation.

16. **Boundary Ele (Hi/Closed-Lo/Open):** this parameter sets the boundary condition. If a catchment (or a spatial domain) has very low boundary elements, the flow will be free to leave the domain and the exiting volume is a volume lost. On the contrary, by setting a high value of the boundary cell it is like virtually closing the domain and, at the downstream of the catchment or the exit point of the domain, the water keeps accumulating. For our purposes it is better to set a very low and negative value (e.g., -9000).
17. **Elevation ASCII:** terrain input file. It must be a DTM ASCII grid file.
18. **Rain Event CSV:** this is an input-setup-file(s) and it indicates a rain event which can be applied to the full domain or to a specific given area. The rain can vary in time (hyetograph) and it is indicated in mm/hr. This input is composed of a single CSV for each event. It is possible to have multiple events by passing multiple files.
19. **Water Level Event CSV:** this is an input-setup-file(s) and it indicates a water level event which can be applied to the full domain or to a specific given area. This event is used to specify a specific water level (in meter) at specific time. This input is composed of a single CSV for each event. It is possible to have multiple events by passing multiple files.
20. **Inflow Event CSV:** this is an input-setup-file(s) and it indicates an inflow event which can be applied to the full domain or to a specific given area. The inflow can vary in time and it is indicated in m3/s. Attention, this differs from the rain event since the inflow value is calculated by interpolation. This input is composed of a single CSV for each event. It is possible to have multiple events by passing multiple files.
21. **Time Plot CSV:** this is an output-setup-file(s) and it indicates the points in the domain where the time varying values of a specific given physical variable (e.g. water depth, velocity, etc.) must be saved. Each time plot is identified by a single CSV file and it can save only one single physical variable over multiple points using a time period. The data output generated is a CSV file. It is possible to have multiple time plot outputs by passing multiple output-setup-files.
22. **Raster Grid CSV:** this is an output-setup-file(s) and it indicates that all the values in the domain of a specific physical variable must be saved for every given time interval. It is also possible to save the maximum values and the final values at the end of the simulation. Each raster grid is identified by a single CSV file and it can save only one single variable. The data output generated is an ASCII-grid file. It is possible to have multiple raster grid outputs by passing multiple output-setup-files.
23. **Output Console:** true or false. If true, the progress of the simulation is displayed to the console (or to a specific console output file in the BATCH Mode) once every specified period (see pt. 28) in seconds. If false, nothing is displayed. The information displayed are: total number of iterations and the number of iterations from the last output, the simulation time, the minimum, maximum, average and last DT from the last output, the maximum velocity in the domain, highest point where water is still moving (if pt. 38 is true), the input and output volumes (if pt. 30 is true) and the run time in seconds from the beginning of the simulation (if pt. 29 is true).
24. **Output Period (s):** the output displayed on the screen is updated once every specified seconds from the start of the simulation (in this case 600s, i.e., once every ten minutes in the simulation time, no run-time). The number of outputs depends on the length of the simulation (Time End). For example a 2 hours simulation and an output period of 600 seconds produce twelve outputs to console.
25. **Output Computation Time:** true or false. If the parameter is set as “true” then the computation time is printed on the screen. In the opposite case not.
26. **Check Volumes:** true or false. Check that the mass conservation is respected or at least preserved within a certain error.
27. **Remove Proc Data (No Pre-Proc):** true or false. When this command is activated it removes all the temporary CADDIES-2D data files generated during the simulation. Note: the files generated during the preprocessing phase are not involved by this command.

28. **Remove Pre-Proc Data:** this command complements pt.31. If the parameter is set on “true” then all files created during preprocessing are also deleted.
29. **Raster VEL Vector Field:** true or false. When this command is not activated (false) and there is a Raster Grid output-setup-file for the velocity, two output ASCII grid files are generated which show respectively the speed and the angle of the velocity for each cell. If this command is activated (true), only a single output CSV file is generated where each line contains the velocity information of one cell. This information are: the X and Y coordinates of the cell centre, the speed, the angle in radian, and the angle in degree. This command is used to plot the velocity of the cells as vectors in GIS software instead than a raster grid.
30. **Raster WD Tolerance (meter):** when caflood has run the simulation it takes the input of this instruction to set a threshold for wet and non-wet cells before saving the output. Referring to the example, all cells with less than 0.01 m of water depth will be treated as dry and therefore no water depth will be exported for those cells. All the raster output are influenced by this parameter, i.e. any eventual depth and velocity outputs. The user has to consider that the value of this parameter strongly depends on the vertical accuracy of digital elevation model, the type of terrain, the grid resolution and the quality of the precipitation and infiltration type of data.
31. **Update Peak Every DT:** true or false. When this command is activated, the maximum/peak values of a physical variable are saved at every time step. The default is false; the maximum values are saved only every update DT (usually 60 seconds). Empirical tests showed that this command, when activated, has an insignificant impact on the accuracy but a large impact on the run time of the simulation.
32. **Expand Domain:** true or false. When this command is not activated (false), the computational domain is the full domain, i.e. each cell with elevation data is processed. When this command is activated, the computational domain is set to be the same of the given events and it will expand when water reach the border of the computational domain. This command is useful to save run time when a simulation has a limited number of point sources of water like an inflow event from a small area.
33. **Ignore Upstream:** true or false. When this command is activated the model identifies the elevation height where the water is still, i.e. no outflow is generated during an update step. Once a new height is identified all cells with elevation above the height are not considered for computation any more. This can save some run-time depending on the terrain and on the type of events modeled. At the worst case scenario, it might add around 1% of extra run-time.
34. **Upstream Reduction (meter):** it is used by **Ignore Upstream** command to identify the amount of meter to reduce the possible elevation height at each update step.

The order of which the instruction lines appear in the setup CSV file is not significant, i.e. they can be of any order. If an instruction line has a default value, it can be omitted. If the model does not recognize an instruction line in the setup file, it will stop the computation and return an error. In the next paragraph, there is an example of output-setup-file for the Raster Grid CSV parameter called WDraster.csv:

[WDraster.csv]

1. Raster Grid Name , Simple Sloping Domain WD Raster Grid
2. Physical Variable , WD
3. Peak , true
4. Period (seconds) , 0

The details of each instruction line are described as following:

1. **Raster Grid Name:** is the full name used to identify the raster grid file.
2. **Physical Variable:** is the physical variable that is going to be saved in the ASCII raster grid. The options are: water depth (WD), water level (WL) and ‘angular’ velocity (VEL).

3. **Peak:** true of false. When this command is activated the simulation memorizes and produces a raster grid with the peak/maximum values of the physical variable chosen.
4. **Period:** a raster grid of the physical variable will be produced every period seconds from the start of the simulation. If, for instance, the user wants to have an output every 10 minutes, he/she have to use 600.

In the next paragraph, there is an example of output-setup-file for the Time Plot CSV parameter called WLpoints.csv:

[WLpoints.csv]

1. Time Plot Name , Simple Sloping Domain WL Points
2. Physical Variable , WL
3. Points Name , 1, 2
4. Points X Coo , 264682, 264538
5. Points Y Coo , 664581, 664665
6. Period (seconds) , 60

The details of each instruction line are described as following:

1. **Time Plot Name:** is the full name used to identify the time plot file.
2. **Physical Variable:** is the physical variable that is going to be saved in the data output CSV file. The options are: water depth (WD), water level (WL) and 'angular' velocity (VEL).
3. **Points Name:** a list of names which identifies the locations where the time varying values of the physical variable will be saved.
4. **Points X Coo:** the X coordinates of the points. The number of coordinates must be the same of the number of names in the 'Points Name' list.
5. **Points Y Coo:** the Y coordinates of the points. The number of coordinates must be the same of the number of names in the 'Points Name' list.
6. **Period:** the value of the various locations of the physical variable will be saved every period seconds from the start of the simulation. If, for instance, the user wants to have an output every 10 minutes, he/she have to use 600.

In the next paragraph, there is an example of output-setup-file for the Rain Event CSV parameter called Rain40mm60minl.csv:

[Rain40mm60min.csv]

1. Event Name , Rain Intensity for An Hour
2. Rain Intensity (mm/hr) , 40, 0
3. Time Stop (seconds) , 3600, 7200
4. Zone (tlx tly w h) ,

The details of each instruction line are described as following:

1. **Event Name:** is the full name used to identify the rain fall event.
2. **Rain Intensity:** is a list of values that indicate the amount of rainfall to fall in mm/hr at specific times.
3. **Time Stop:** is a list of times in seconds when the corresponding values in 'Rain intensity' will be stop to be used in the simulation. This list must be the same length of the list 'rain Intensity'.
4. **Zone:** is the area where the event will be applied. The area is identified by the top-left corner x and y coordinate and the width and height sizes. If this parameter is empty, the event will be applied in the entire domain.

An example of command line execution of the **caflood** program in widows and UNIX are respectively the following:

- `caflood /sim C:\Indir Sim.csv C:\Outdir`
- `caflood -sim C:\Indir Sim.csv C:\Outdir`

Other than the obligatory three arguments previously described (input folder, setup-file, output folder), the program can be launched with multiple optional arguments identified by the prefix '/' in Windows OS and the prefix '-' in UNIX OS. In this case, the 'WCA2D' argument is used to indicate that the model requested is the weighted cellular automata 2D model; without this argument no simulation will be executed. Using the argument 'help' is possible to have a list of optional arguments.

4 Summary

This Deliverable details the functions for EU-CIRCLE CIRP. The hazard modelling and data analaysing modules are embedded in CIRP that users can efficiently use CIRP to simulate various climate scenarios and evaluate the impact of hazards. All the tools were integrated and tested via the Virtual Data Set presented in D7.4. It shows the advantage of CIRP that streamlines the algorithm of analyses and to improve the risk communications with stakeholders.

5 References

Dagum, Leonardo, and Ramesh Menon. 1998. 'OpenMP: An Industry Standard API for Shared-Memory Programming'. *Computational Science & Engineering, IEEE* 5 (1): 46–55.
Ghimire, Bidur, Albert S. Chen, Michele Guidolin, Edward C. Keedwell, Slobodan Djordjević, and Dragan A. Savić. 2013. 'Formulation of a Fast 2D Urban Pluvial Flood Model Using a Cellular Automata Approach'. *Journal of Hydroinformatics* 15 (3): 676. doi:10.2166/hydro.2012.245.
Guidolin, Michele, Albert S. Chen, Bidur Ghimire, Ed Keedwell, Slobodan Djordjevic, and Dragan A. Savic. 2015'A Weighted Cellular Automata 2D Inundation Model for Rapid Flood Analysis'. *Under Review*.
Guidolin, Michele, Andrew Duncan, Bidur Ghimire, Mike Gibson, Edward Keedwell, Albert S. Chen, Slobodan Djordjevic, and Dragan Savic. 2012. 'CADDIES: A New Framework for Rapid Development of Parallel Cellular Automata Algorithms for Flood Simulation'. In Hamburg, Germany: IWA (International Water Association).
Itami, Robert M. 1994. 'Simulating Spatial Dynamics: Cellular Automata Theory'. *Landscape and Urban Planning* 30 (1): 27–47.
Munshi, A., and others. 2011. 'The OpenCL Specification Version 1.1'. *Khronos OpenCL Working Group*.
Wolfram, Stephen. 1984. 'Cellular Automata as Models of Complexity'. *Nature* 311 (October): 419–24.