Parametric design and structural analysis of bridge beams

(Dynamo-Sofistik-Revit link)

Juan González Cavero

Contents

A video tutorial is also available: https://www.youtube.com/watch?v=T_vnRhejjgo&feature=youtu.be

1. System settings (first time)

The user must have installed in his computer the following software:

- Revit 2020 or above (BIM modeler)
- Dynamo 2.1 + some Dynamo packages (Visual programming)
- Sofistik 2018 or above (Structural analysis)

* It might work for older versions, but it has not been checked.

Dynamo is included in Revit. To open it, go to the **Manage** tab in Revit, **Visual Programming**:

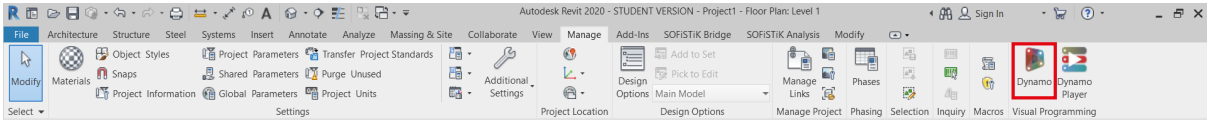

Figure 1.1 Revit toolbar

Dynamo allows creating custom nodes, which can then be grouped in packages. There is a big community that shares its own developed packages, so any user can download and use them for free. There are also companies that creates these specific packages for profit.

Dynamo version 2.1 or above is required. You can check the version within Dynamo selecting **About** under the **Help** toolbar. An option to Update will appear if it is not Up-to-date.

To be able to use these scripts, we need to download a couple of packages.

- SofiLink 2
- Clockwork
- Data-Shapes

The easiest way to install a package is by using the **Packages** toolbar in your Dynamo interface.

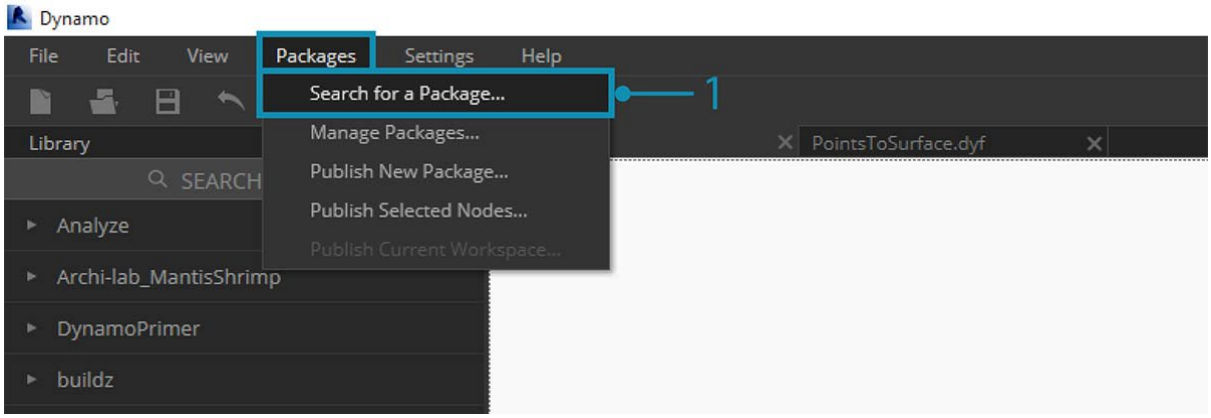

Figure 1.2 Dynamo Packages

Search for the aforementioned packages and download them. If any Dialog Box appears, press OK. Once they are downloaded and automatically installed. You can see these new packages at the bottom of the **Library** (at the left by default).

At this point we have all the necessary nodes (contained in the packages) to run the script we want. But where is the script? It is located in the installation directory of SofiLink.

The easiest way to get there is by using again the **Packages** toolbar and selecting **Manage Packages…**, go to **SofiLink 2** and select **Show Root Directory**. Then, go to **extra**.

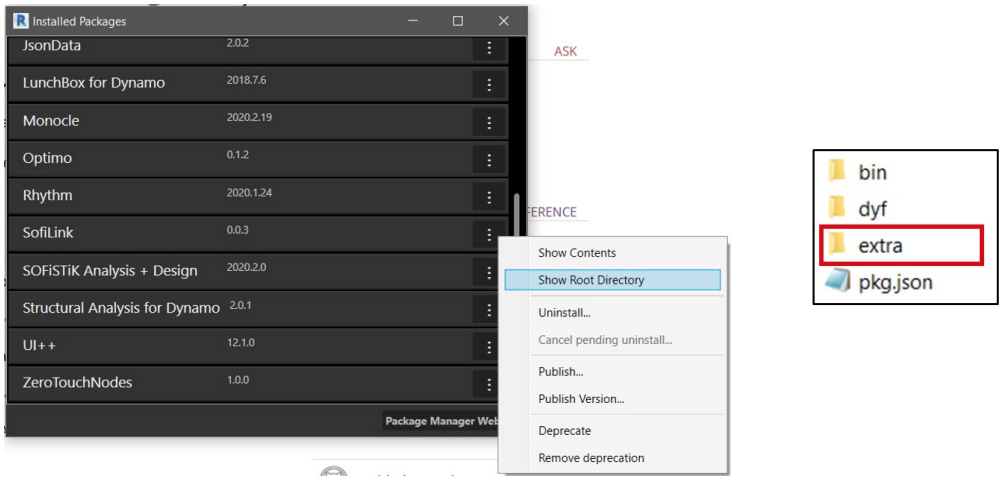

Figure 1.3 Package Root Directory

The files contained in this folder should not be modified. Though, you can copy them to your workspace in order to work with them.

Moreover, in order to automatize Sofistik by means of a batch file. Sofistik's solver executable *sps.exe* must be set as an *Environment Variable.* This is just copying the installation directory patch where this sps.exe file is located and pasting it into the Path variable. Open *Enviromental variables* > *Edit…* > *New* > and paste the path there. Here it is shown for Sofistik 2020. For Sofistik 2018, it would be the same process.

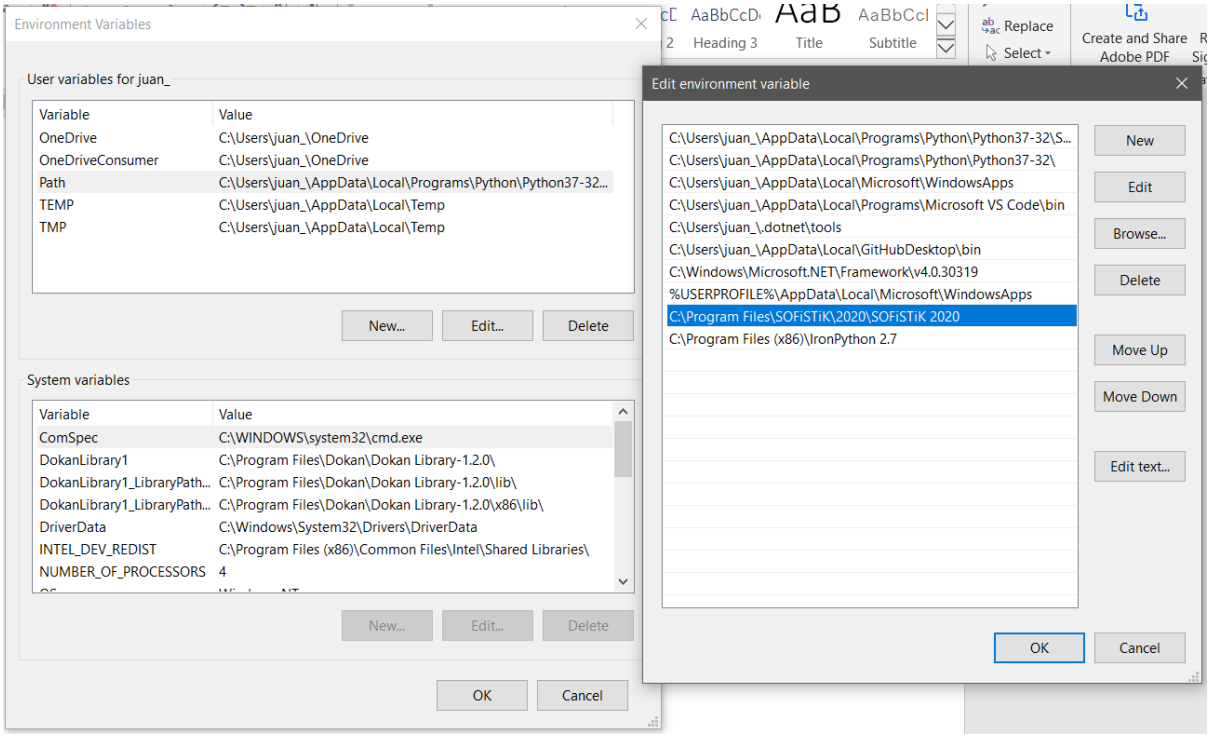

Figure 1.4 Environment variables

2. Basic user

The user does not need to open Dynamo, just Dynamo Player.

A straight workflow allows to quickly define inputs and parameters and see the resulting analysis:

1. Open an empty template in Revit.
2. Open Dynamo Player (Manage > Visual Programming).
3. Select your workspace folder where you located the Dynamo scripts, the Revit family and the required images (these files can be obtained from the Package Root Directory).

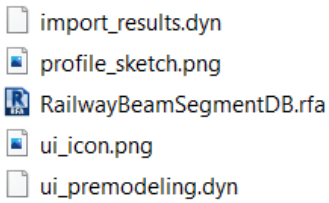

Figure 2.1 Workspace 1

4. Run *ui_premodeling.* Another windows form will open. You can specify here some parameters of the bridge. Press OK. Once it is finished, you are able to see the modeled bridge in Revit (select 3D view in the upper bar).

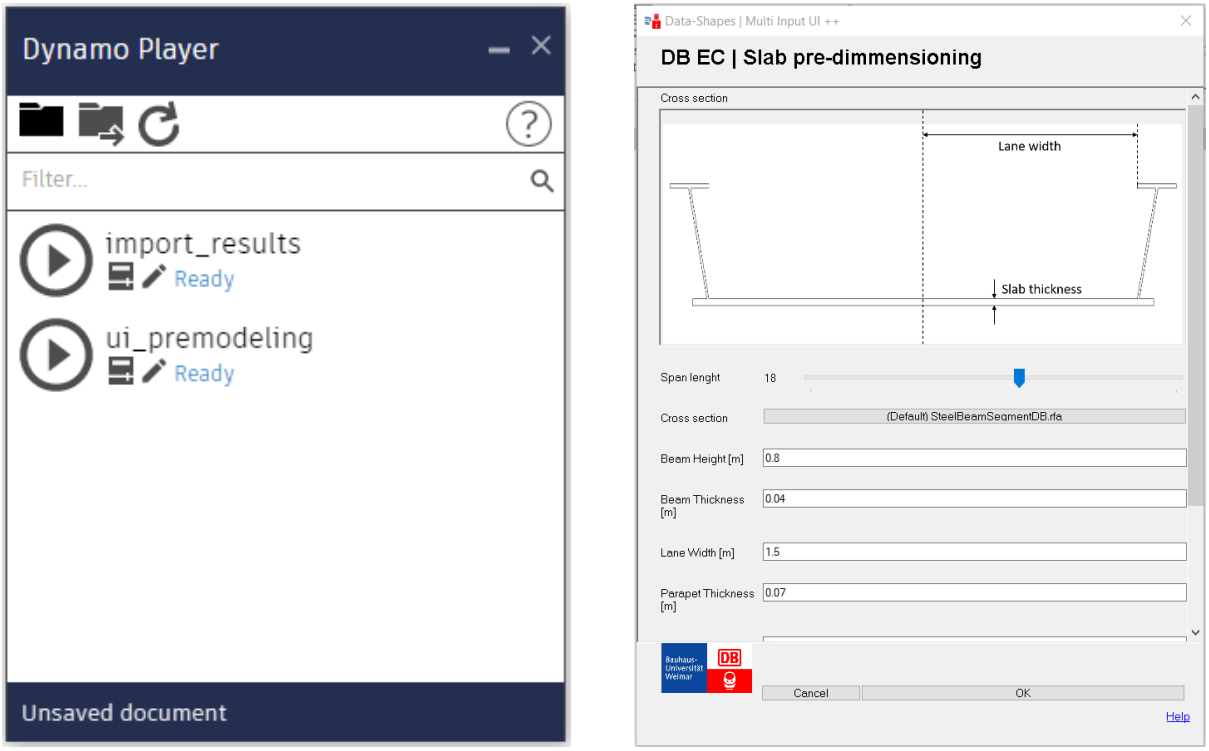

Figure 2.2 Dynamo Player and Data Shape's UI (User Interface)

5. Now, return to the Workspace (in Dynamo Player, press *View current folder)*. You can see new files have been created.

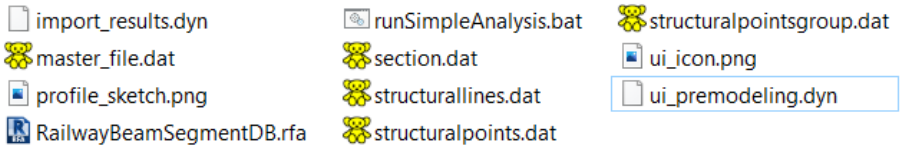

Figure 2.3 Workspace 2

6. By executing *runSimpleAnalysis.bat,* Sofistik will automatically analyze the bridge. Resulting files from the analysis are stored in the folder. Specific results are written to an excel file, *data.xlsx*, and *master_file.plb* contains the complete analysis report.

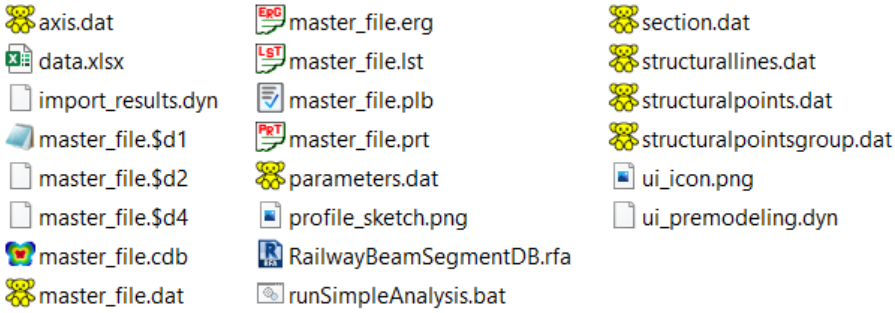

Figure 2.4 Workspace 3

7. Go to Dynamo Player again and run the other script, *import_results.dyn*. The script asks you to select the model element. You need to select the bridge and click again on the run or play button. You will see then that the analysis results have been associated to the model.

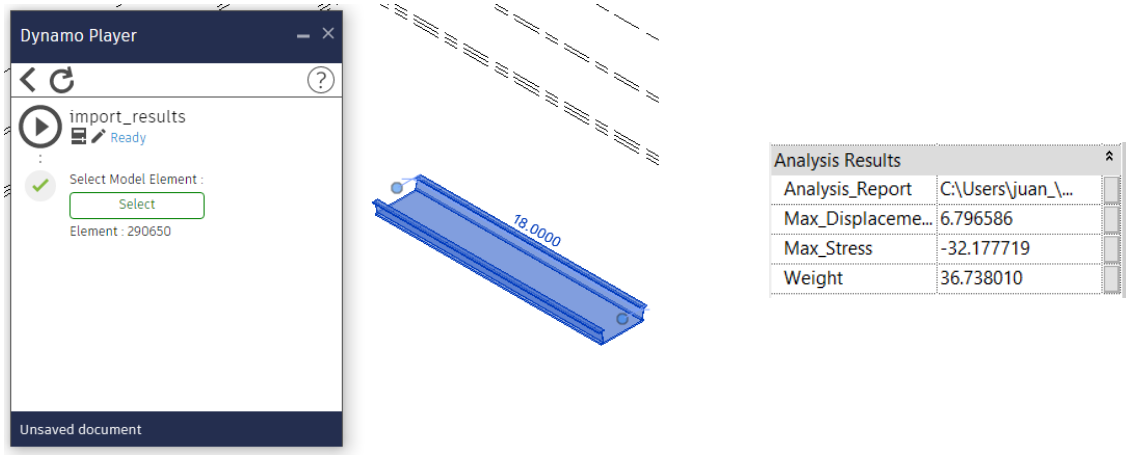

Figure 2.5 Dynamo Player, Revit model and model properties.

3. Revit family

The link of sections from Revit to Sofistik must be manually done but just the first time we want to link a new section.

Sofistik allows parametric design as well as Revit. However, the way it defines sections is not the same. In Sofistik this is done by means of vertices that forms a close polygon. Revit parameters are used to define the coordinates of vertices in Sofistik.

All this code, used to define the section in Sofistik is manually written and then copied to a text parameter of the family.

Here we can see the code that defines the cross section in Sofistik. The OPZ code in the sixth line indicates that the section is symmetric with respect to the Z axis. Therefore, it is enough defining one side of the section. In case the section is not symmetric, the section must be entered completely.

```
$prog aqua
HEAD Section SteelBeamDB
$ Revit defined Cross Section
SECT 1 BTYP CENT TITL 'DB_Profile' MNO 1
$ Polygonal Shape
POLY TYPE OPZ
VERT NO Y                                                    Z
VERT  1 0                                                    =#Parapet_Thickness+#Beam_Height
VERT  2 =#Lane_Width                                         =#Parapet_Thickness+#Beam_Height
VERT  3 =#Lane_Width+#Parapet_Width/2-#Beam_Thickness/2      =#Parapet_Thickness
VERT  4 =#Lane_Width                                         =#Parapet_Thickness
VERT  5 =#Lane_Width                                         0
VERT  6 =#Lane_Width+#Parapet_Width                          0
VERT  7 =#Lane_Width+#Parapet_Width                          =#Parapet_Thickness
VERT  8 =#Lane_Width+#Parapet_Width/2+#Beam_Thickness/2      =#Parapet_Thickness
VERT  9 =#Lane_Width+#Beam_Thickness                         =#Parapet_Thickness+#Beam_Height
VERT 10 =#Lane_Width+#Beam_Thickness+#Parapet_Width/2        =#Parapet_Thickness+#Beam_Height
VERT 11 =#Lane_Width+#Beam_Thickness+#Parapet_Width/2        =#Parapet_Thickness+#Beam_Height+#Slab_Thickness
VERT 12 0                                                    =#Parapet_Thickness+#Beam_Height+#Slab_Thickness
```

Figure 3.1 Cross section definition in Sofistik Teddy (CADINP)

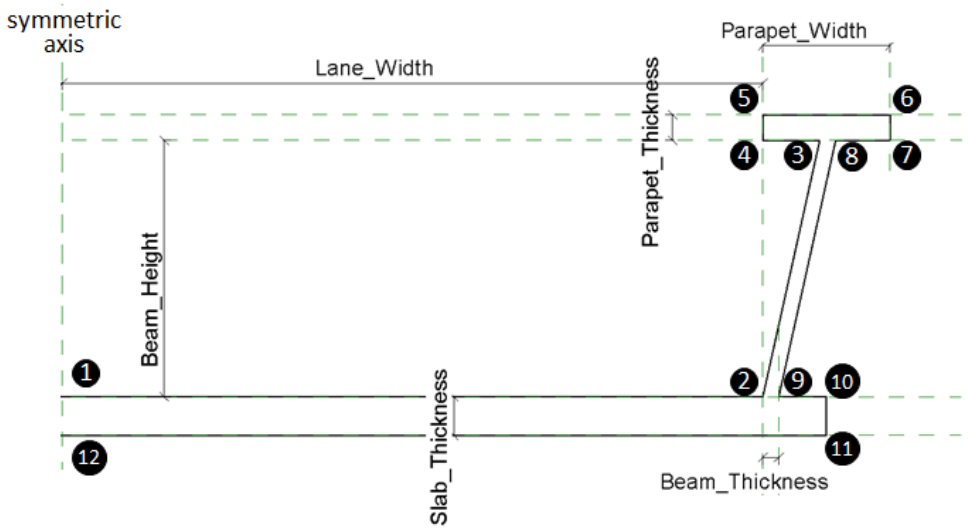

Figure 3.2 Cross section (right side) in Revit

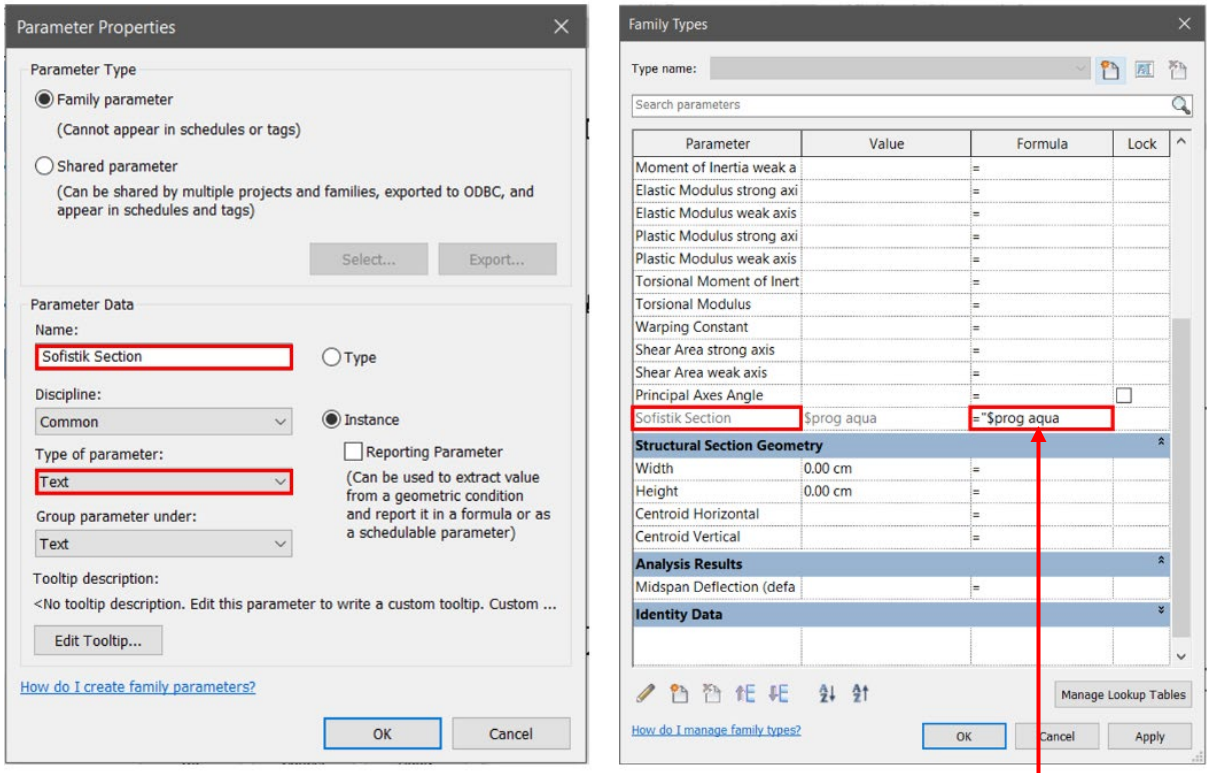

Figure 3.3 Text parameter for section definition in Sofistik

```
$prog aqua
HEAD Section SteelBeamDB
$ Revit defined Cross Section
SECT 1 BTYP CENT TITL 'DB_Profile' MNO 1
$ Polygonal Shape
POLY TYPE OPZ
VERT NO Y                                                    Z
VERT  1 0                                                    =#Parapet_Thickness+#Beam_Height
VERT  2 =#Lane_Width                                         =#Parapet_Thickness+#Beam_Height
VERT  3 =#Lane_Width+#Parapet_Width/2-#Beam_Thickness/2      =#Parapet_Thickness
VERT  4 =#Lane_Width                                         =#Parapet_Thickness
VERT  5 =#Lane_Width                                         0
VERT  6 =#Lane_Width+#Parapet_Width                          0
VERT  7 =#Lane_Width+#Parapet_Width                          =#Parapet_Thickness
VERT  8 =#Lane_Width+#Parapet_Width/2+#Beam_Thickness/2      =#Parapet_Thickness
VERT  9 =#Lane_Width+#Beam_Thickness                         =#Parapet_Thickness+#Beam_Height
VERT 10 =#Lane_Width+#Beam_Thickness+#Parapet_Width/2        =#Parapet_Thickness+#Beam_Height
VERT 11 =#Lane_Width+#Beam_Thickness+#Parapet_Width/2        =#Parapet_Thickness+#Beam_Height+#Slab_Thickness
VERT 12 0                                                    =#Parapet_Thickness+#Beam_Height+#Slab_Thickness
```

Figure 3.4 Cross section definition in Sofistik Teddy (CADINP)

For more information regarding the Dynamo script, check the Thesis appendix.