AUTOTURN

ESSENTIALS AND ADVANCED TRAINING MANUAL

Transoft Solutions Inc.

350 - 13700 International Place
Richmond, BC CANADA V6V 2X8
TEL | 1.604.244.8387
FAX | 1.604.244.1770
EMAIL | info@transoftsolutions.com
WEB | www.transoftsolutions.com

Disclaimer

Transoft Solutions Inc. makes no warranty or guarantee, expressed, implied or statutory, as to the accuracy, reliability, suitability, functioning, or results derived from the AutoTURN® software program. Nor shall the licensing of the program and related program materials or manuals constitute any warranty or guarantee. Transoft Solutions Inc. shall have no liability or responsibility to the licensee or any other person or entity with respect to any liability, loss or damage caused or alleged to be caused directly or indirectly by the AutoTURN software program and/or related program materials, including, but not limited to, any interruption of service, loss of business or anticipated profits, or consequential damages resulting from the use or operation of this product. All computer program results require professional interpretation, and Transoft Solutions Inc. makes no warranty of results obtained by using the AutoTURN software program. Any liability of Transoft Solutions Inc. is limited exclusively to product replacement.

This software and related materials and manual are sold "as is". Transoft Solutions Inc. is not responsible for typographical or graphical errors that may appear in this document. Any and all warranties for merchantability and/or fitness for a particular purpose, express, implied or statutory, are expressly excluded. Portions of the program are protected by copyright laws, Common Law Copyright, and/or trade secret protection by Transoft Solutions Inc. All programs and related program materials are subject to proprietary rights of Transoft Solutions Inc.

Copyrights

TABLE OF CONTENTS

AUTOTURN

Overview and Benefits

SOFTWARE OVERVIEW AND BENEFITS

How the Software Works

AutoTURN is interactive user-friendly software that simulates swept paths for most vehicle combinations and for any tangent / circular curve combinations. The AutoTURN model uses two basic algorithms to simulate turning vehicles. The main algorithm is based on the graphical method developed by Vaughan and Sims[1] and the UMTRI model[2], and is commonly referred to as the incremental method of analysis or the constant pursuit method. The algorithm consists of an analytical means of duplicating the operation of the tractrix integrator and using CADD routines to simplify input for the analysis and to facilitate in the plotting of the desired vehicle paths. The incremental position of the steering point is set at a fixed distance equal to a small number relative to real world vehicle and turning dimensions. This method was selected because in most practical situations it does not significantly affect off tracking. Furthermore, through coordinate geometry calculation, the coordinates of the incremental points are easily defined. See Exhibits 1 and 2.

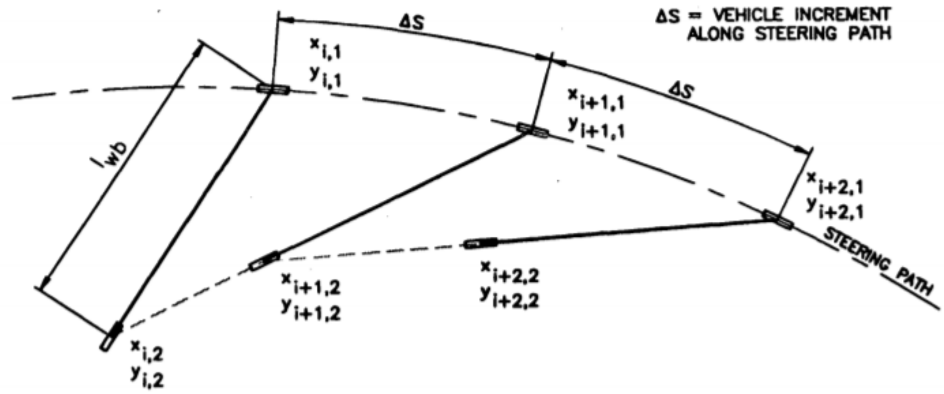


Exhibit 0. - Application of the algorithm to single-unit vehicle

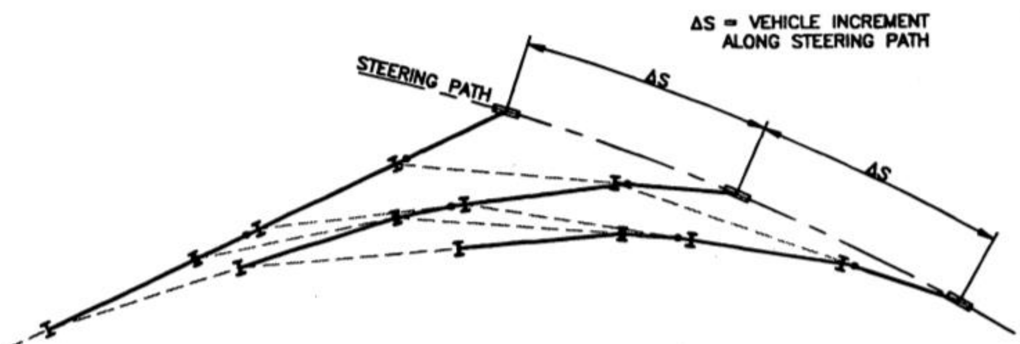


Exhibit 0 - ***Application of the algorithm to multiple unit vehicles***

The model needs to be enhanced for determining tracking and swept paths. To determine a vehicle tracking path, a new vehicle position, including the tire coordinates, is calculated for each incremental step. The coordinates for the body of the vehicle are also calculated for each incremental step.

In addition to the constant pursuit algorithm, AutoTURN incorporates a second key algorithm that provides a general but 'intelligent' approach to represent driver maneuverability. The algorithm incorporates steering lock-to-lock times for low-speed simulations and minimum AASHTO criteria for speeds above 15 km/h. AutoTURN does

[1] "Determination of Swept Paths of Vehicles", Traffic Accident Research Unit, Department of Motor Transport, N.S. Wales, 1970

[2] "Vehicle Offtracking Models", The UMTRI, Transport Research Record 1054, 1985

not provide specific simulations for speeds in excess of 60 km/h because research shows that for higher speeds, vehicle off-tracking is significantly lower due to centripetal forces.

Software Limitations

The user is responsible for applying engineering judgment in interpreting and using program results. Familiarity with the software and its limitations is therefore important to ensure that the results produced by the software are applied appropriately.

Use of Design Guidelines: AutoTURN is not intended to be a source for design information. The software must be used in conjunction with the most currently available design guidelines for the local jurisdiction.

Interpretation of Program Results: AutoTURN is primarily a kinematic model, meaning that it does not account for the forces resulting from vehicle inertia or road surface conditions other than dry pavement. AutoTURN produces geometrically idealized results that may be difficult to replicate exactly under actual field driving and road conditions. A successfully run AutoTURN simulation does NOT suggest that all drivers can follow the defined path in real conditions, NOR does it imply that other drivable paths between the desired start and end positions are not possible. For the above reasons, AutoTURN results should be used conservatively, with ample allowances added for clearances between vehicle tire tracks or swept path envelopes and road edges or obstructions.

Turn Speeds and Minimum Turning Radii: Turn limitations for standard design vehicles are generally based on the minimum turn radius as specified in the design guidelines for each jurisdiction. These are generally intended for speeds of less than 15 km/h (about 10 mph).

Unusual Conditions: If non-standard vehicle types, roadway surfaces other than dry pavement, or other unusual or non-standard conditions are used, AutoTURN may produce inaccurate results.

Caution is advised when dealing with the following conditions for which program results have not been verified:

- Speeds greater than 15 km/h (about 10 mph) where tire slippage and inertial forces become significant
- Icy or otherwise slippery road surfaces because they tend to produce less tire tracking than dry road surfaces
- Custom vehicles with multiple-axle bogeys consisting of three or more axles per bogey (as in heavy load movers)
- Vehicles where the ratio of the spacing between the fixed axles in a group to the wheelbase (the distance between axle groups) is 0.5 or more
- Vehicles having more than four tires per axle
- Vehicle combinations with multiple tractors
- Vehicle combinations consisting of tractors with four or more trailing units.

It is highly recommended that the user obtains as much turning information as possible for non-standard vehicles from manufacturers and other sources to be able to verify and draw conclusions about AutoTURN's results.

Tight Maneuvers with Standard Vehicles: The standard design vehicles provided with AutoTURN have turn limitations derived from the turning templates contained in the design manuals published by the corresponding jurisdictions. The primary purpose of these design manuals is to provide guidance for typical road design conditions. Consequently, in some cases, the standard vehicles may have turning circles that are too large for situations where tight maneuvering is required, such as in loading areas or parking facilities. In these cases, it may be necessary to create user-defined vehicles and provide turn limitations that allow the tighter, more realistic minimum turning circles. In AutoTURN this can be accomplished by copying the design vehicle and setting higher values for the steering lock and articulating angles.

CAUTION: The user must set steering lock angles that reflect the turn limitations of the specific vehicle being considered. Actual field measurements or manufacturers' data should be used. The use of arbitrary angles will lead to misleading results.

Acceleration: AutoTURN does not consider vehicle acceleration or deceleration. When simulations are generated from a stopped position, or to a stopped position, the vehicle is assumed to reach its final speed in a short enough distance that it can be ignored when determining the turn radius.

Benefits

The design of transportation facilities is generally based on the turning and swept paths of design vehicles. Until recently, engineers, architects, CAD technicians, and designers worldwide had to rely on plastic vehicle turning templates to develop the layout of these facilities (i.e. road intersections, off-street parking lots, loading dock ramps, and other facilities that involve vehicle accesses). This method has several drawbacks. The increasing use of computers in the design and drafting of highways and other transportation facilities has necessitated the need for a computerized method to simulate the turning paths of specific design vehicles, make changes, finalize the design and prepare final drawings of the same paths more efficiently.

The AutoTURN software has significantly simplified the task of analyzing the movement of vehicles in the CAD environment. AutoTURN is a software program that allows for the simulation of low-speed turn maneuvers for roadway vehicles in CAD drawings within AutoCAD®, MicroStation®, BricsCAD®, or ZwCAD® platforms. This interactive program allows quick reviews of alternative paths, different vehicle types, and configurations, analyzing sightlines and oversized loads, and results in the expedient development of design and construction drawings.

Starting the Software

Before you begin the AutoTURN Training Lessons, there are a few items that you should set:
If you are using AutoCAD Release 2015 or later

1. Open AutoCAD 2015 or later.
2. Start a drawing from scratch and use the default English (feet and inches) settings.
   To start AutoTURN, select **Load AutoTURN** from the AutoTURN menu.

OR
If available, use the desktop shortcut for AutoTURN.
If you are using MicroStation V8i (SS10) or later

1. Open MicroStation V8i (SS10) or later.
2. Start a new drawing and set **Design File Settings** –> **Working Units** –> **Master Units** to **Feet**.
3. To start AutoTURN, click **Utilities** on the menu, select **MDL Applications**,
4. Click **Browse**, and then select the atv8.ma/atcn.ma file from the AutoTURN program folder.

OR
If available, use the desktop shortcut for AutoTURN.

NOTE:
Throughout the training, AutoTURN exercises are maintained only within AutoCAD drawing files and all command references have been formatted using standard AutoCAD terminology (i.e. layers, turn-off, etc.) Users of MicroStation should import supplied AutoCAD drawing files for all exercises and lessons.

Once you have set AutoCAD or MicroStation drawing settings, you are now ready to follow the exercises.

AUTOTURN

Toolbar Overview

HIGHLIGHTS OF SOFTWARE FEATURES

Toolbar Overview

The new AutoTURN features were added as a direct response to suggestions made by current users of earlier versions of the software and with research input supplied by Transoft Solutions Product Management team. The enhancements were developed to provide users with superior control and flexibility in their designs.
The following is a review of all features available in the latest AutoTURN version release.

Program Settings

Sets various general program settings, such as general simulation settings, program permissions, as well as provides functions for upgrading a license and for moving a program license between computers.

Properties

Sets the various properties for turn simulations (i.e. update, max. width, various vehicle envelopes, etc.), and hatching of vehicle simulations.

Vehicles

Sets the current design vehicle. This tool is also used to create new vehicles; new vehicle types and groups; to add loads and/or sightlines to vehicles; and calculate the steering lock angle for custom vehicles.

IntelliPath

Evaluates multiple routes that avoid specified exclusion zones (e.g. buildings, curbs, etc.) and then generates movements with different vehicles on selected routes.

Generate Arc Path

Generates a path that follows an arc from the start position to the position of the mouse.

Generate Corner Path

Generates a corner-type maneuver, by setting the turn radius and the amount of turn sweep or selecting sweep angle item.

Oversteer Corner Path

Generates an oversteer corner maneuver, as a result of setting: the turn radius, the amount of turn sweep, and available area to oversteer.

Place Adaptive Simulation

Places a turn simulation using the current vehicle on a selected path.

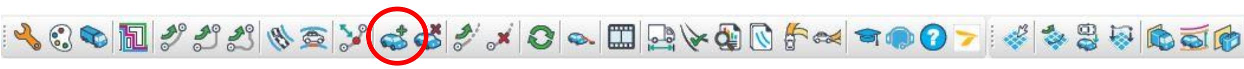
Place Vertical Simulation

Places a forward vertical simulation using the current vehicle on selected vertical path elements.

Path Control

Edits 2D turn simulations by adding, moving, or deleting nodes or by making small offset, radius, or sweep angle adjustments to corner path or oversteer corner path simulations.

Place Vehicle



Places plan view representations of the current vehicle anywhere along a selected path or turn simulation.

Delete Vehicle

Deletes vehicles in simulations that were placed using the Place Vehicle tool.

Continue Simulation

Continues generating the selected 2D turn simulation.

Delete Last Selection

Deletes the last selection of a selected 2D turn simulation.

Regenerate Simulation

Recreates the selected turn simulation taking into account any changes to the elements drawn on the specified Obstacles Layer/Level that interact with the turn simulation envelopes.

Analyze Sight Lines

Draws sightlines analysis on horizontal alignments, 3D terrains, and vertical simulations.

Run Animation

Runs an animation of the current vehicle on the selected path elements or runs an animation of the vehicle used in the selected simulation.

Insert Profile

Inserts a dimensioned profile of the current vehicle (i.e. vehicle name, units, all vehicle dimensions, lock to lock time, steering lock angle, and articulating angle).

Inspect Simulation

Reports the steering lock angle, input speed, maximum speed, proximity to elements on specific layers/levels, turn wheels from stop condition, and minimum turn radius for a simulation.

Generate Report

Generate a Speed Profile Report or Turning Report.

Recall Simulation

Recalls previously-stored turn simulations by either placing the simulations or running their animations. You can also rename simulations to archive them and delete unwanted simulations.

Generate Template

Generates a turning template of the current vehicle complete with all relevant vehicle dimensions and turning characteristics

Generate Grade Template

Generates a grade template for the current vehicle showing the vehicle's maximum upward and downward grade capabilities.

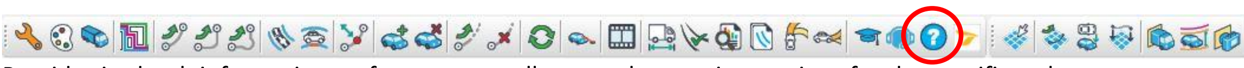
Tutorials

Allows you to play, record, and delete interactive tutorials.

Technical Support

Provides answers to Frequently asked questions and/or request technical support assistance.

Help



Provides in-depth information on features as well as step-by-step instructions for the specific task.

Transoft Solutions Online

Short cut to www.transoftsolutions.com

Select Terrain

Sets the current terrain used by AutoTURN Pro's 3D tools to a selected surface or mesh element or the XY Plane in the drawing.

Generate 3D Arc path

Generates a 3D simulation that follows an arc from the start position to the final position of the mouse on a selected terrain.

Convert 2D to 3D

Converts a selected 2D turn simulation into a 3D turn simulation on a terrain.

Place 3D Simulation

Places a 3D turn simulation using a 3D capable [current vehicle] on a selected path on a terrain.

Analyze Punch Through

Draws a cross-section of a selected 3D turn simulation at a cutting plane line and creates an annotated section detail.

Analyze Vehicle Clearance

Creates a graph showing the longitudinal profile of a selected 3D turn simulation and crossing elements.

View punch Through

Sets up a sectional view for analyzing conflicts. This tool aids in visualizing a 3D turn simulation and how it interacts with obstacles and the terrain by moving the vehicle back and forth along the simulation at the punch through.

Transoft Solutions Online

Short cut to www.transoftsolutions.com

AUTOTURN

ESSENTIALS TRAINING

EASY STEPS TO USING THE SOFTWARE

Overview and Getting Started

This lesson will illustrate the basic steps in using all AutoTURN software features through simple examples.

How to Preset Program Settings

1. Click **Program Settings** on the AutoTURN toolbar.
2. Select the **General** category settings as illustrated in Exhibit 1.

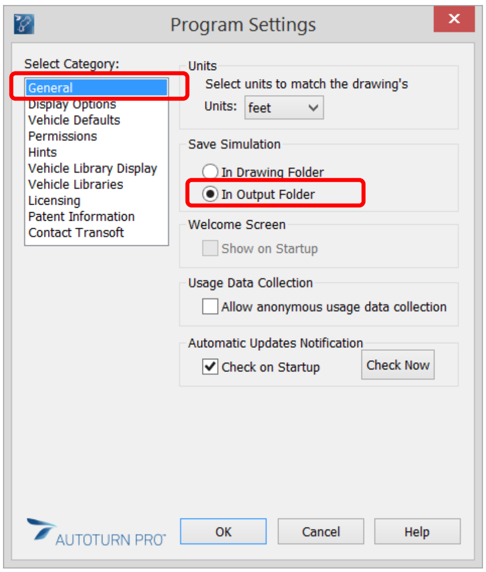


Exhibit 1 –*Program Settings Dialog Box – General Category*

3. Set the drawing's units to *feet*.
4. In the **Save Simulation** section, select the **In Output Folder** checkbox. See Exhibit 1.
5. Now select **Display Options** from the category list within the **Program Settings** dialog box.
6. Within this category, select the **Steering Pointer** checkbox and click **OK**.

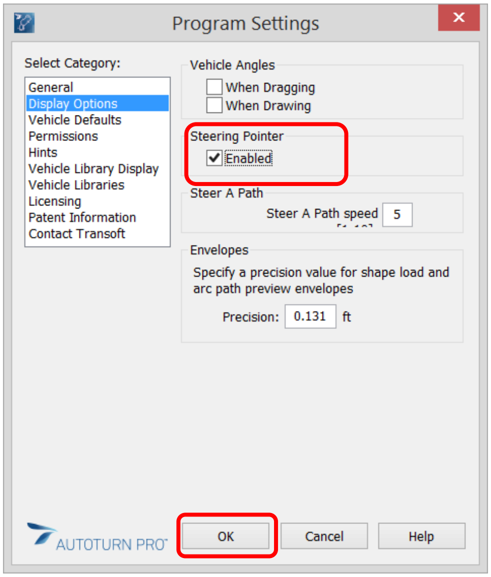


Exhibit 2 – *Program Settings – Display Options Category*

7. You can filter the type(s) of Standard Vehicle Library that will be displayed in the Design Vehicle Library.

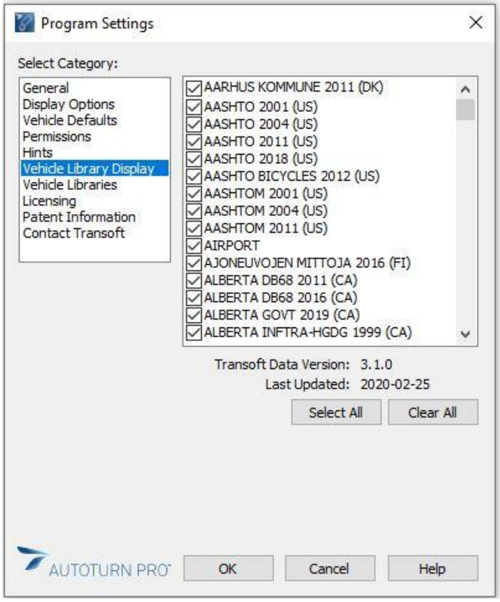


Exhibit 3 – *Program Settings – Vehicle Library Display*

How to Preset Simulation Properties

The ability to define the different properties for the simulation, allows the designer to define how the simulation and its components will be displayed, improving your presentations, or following your company's drafting standards for your simulations.

▶ Properties Dialog Box Explained

The properties dialog box allows the user to define the display details of each of the properties category making it possible to fine-tune the needs of the simulation. See Exhibit 1.

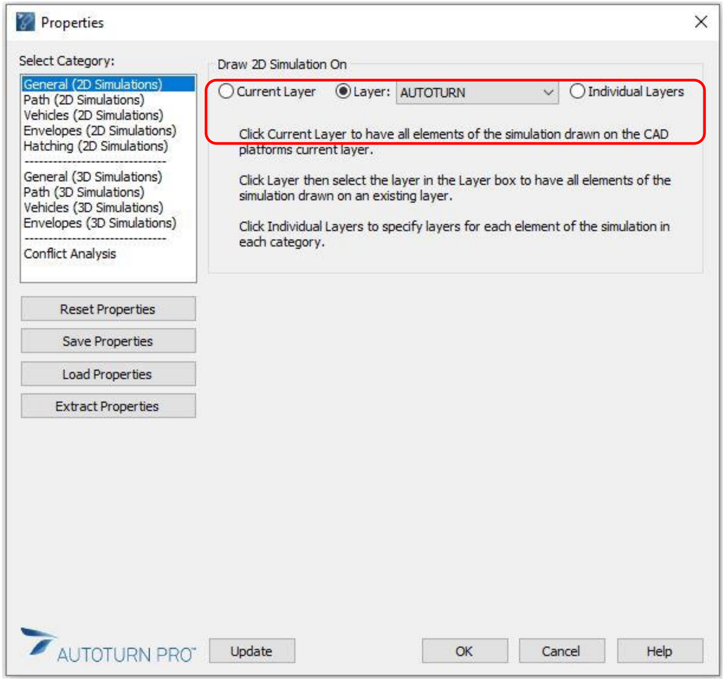


Exhibit 1 – *Properties Dialog Box*

The categories presented in the Select category section allow you to configure the display of the different components of your simulations. There are 8 different property categories and they are:

General (2D and 3D simulations): Define a Layer/Level or Individual layers/levels for the different elements of the swept path generated.

Path (2D and 3D simulations): Define Individual Layers/Levels, Colors, Line Styles, and Transparency for the centerline path generated.

Vehicles (2D and 3D simulations): Define how vehicles will appear in your simulations by setting up its characteristics.

Envelopes (2D and 3D simulations): Define all the envelope details (Layers/Levels, Colors, Line Styles, and Transparency) that are relevant for the project that you've been working on in the Envelope Draw Options.

Hatching: Define layers/levels and Hatching Draw Options for the forward and reverse envelopes.

Conflict Analysis: Set the layers/levels that contain elements to be evaluated for conflicts with various envelopes of the turn simulations. Also, set the color and transparency for the different conflict elements under Conflict Analysis Draw Options.

The Reset Properties, Save Properties, Load Properties, Extract Properties options allow you to save and share different sets of properties with other users. It helps provide a consistent appearance or a standardized presentability for the project.

Use the Reset Properties option to restore the default settings (settings the software is shipped with) for properties.

Use the Save properties option to save the current set of properties to a custom file that can be reused or passed to other colleagues. In the **Save Properties** dialog box enter the desired name for the file and then click OK. See Exhibit 2.

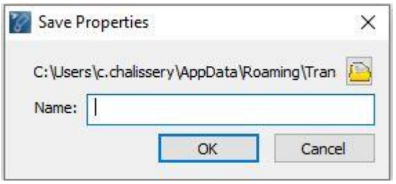


Exhibit 2 – *Save Properties Dialog Box*

By default, the exported set of properties will be stored at the following location in your machine: C:\Users\Your System name\Appdata\Roaming\Transoft Solutions\AutoTURN 11 \Properties User. Or you may provide a new folder location as desired.

Use the **Load Properties option** to use/load the previously saved Properties Styles. When you click on this button, the Load Properties dialog box will display. Select the desired properties set file from the list, and then click OK. See Exhibit 3.

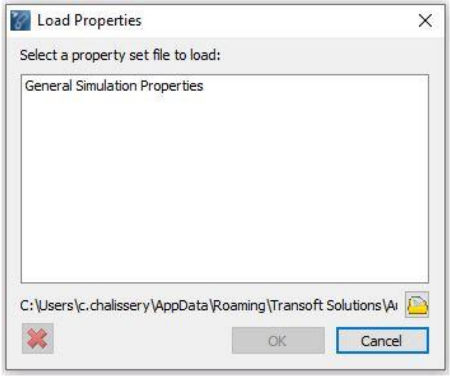


Exhibit 3 –*Load Properties Dialog Box*

Previously saved simulation properties will be imported and will use the layer/level, colors, envelopes, hatching, etc. as defined within the saved file.

Use **Extract Properties** and then select an existing turn simulation to replace the current set of properties, with the properties from the selected simulation.

The **Update** button, allows you to change the properties of a previously created simulation without having to recreate the simulation.

▶ How to Preset Properties

1. Click  **Properties** on the toolbar.

2. Select the **General (2D Simulations)** category.

3. Set to draw 2D Simulation on a **Layer/Level** “AUTOTURN”. See Exhibit 1.

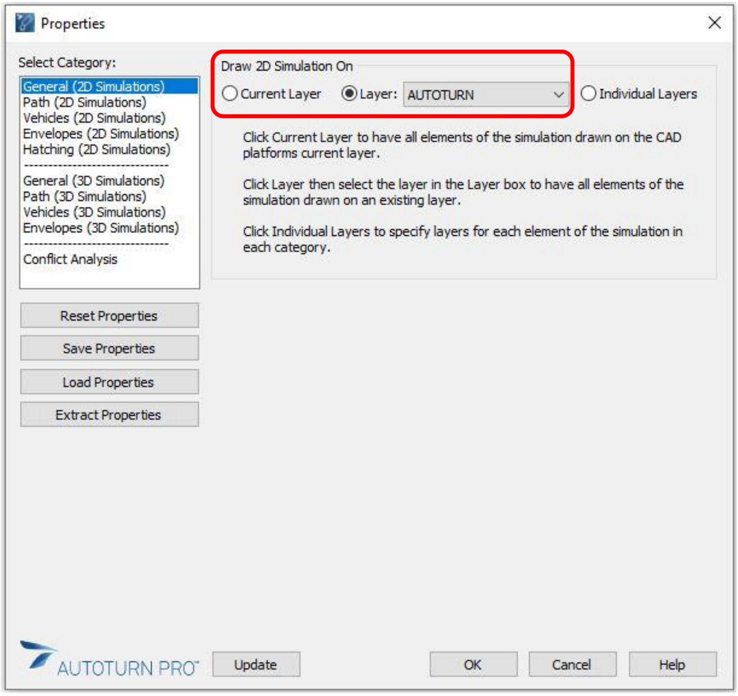


Exhibit 1 – *Properties Dialog Box – General Category*

4. After setting the **Path (2D Simulations)** category, select the **Vehicles (2D Simulations)** category.

5. Select draw vehicles at “Paths Ends”.

6. Enable the Vehicle element display as illustrated in Exhibit 2.

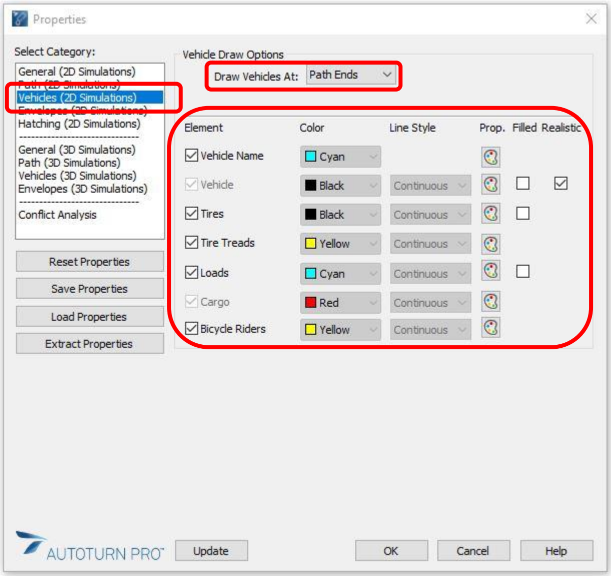


Exhibit 2 – *Properties – Vehicles Category*

7. Once completed, select the **Envelopes (2D Simulations)** category.

8. Enable the display envelopes as illustrated in Exhibit 3.

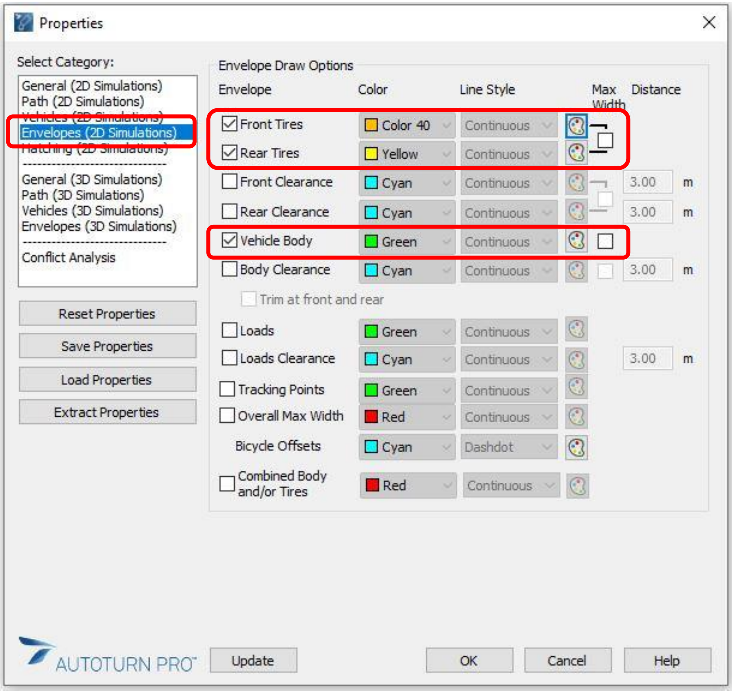


Exhibit 3 – *Properties* - *Envelopes Category*

9. Next, select the **Hatching (2D Simulations)** category

10. Set the draw hatching on **Layer/Level** "AUTOTURN".

11. In the **Select an Envelope to hatch** list, select **Vehicle Body**.

12. Select the **NET** hatch pattern, **Color:** Red, **Scale:** *50*, and **Angle:** *0*.

13. Select Separate Reverse hatching option, Color: Blue, Scale: 50, and Angle: 0 and click **OK**. See Exhibit 4.

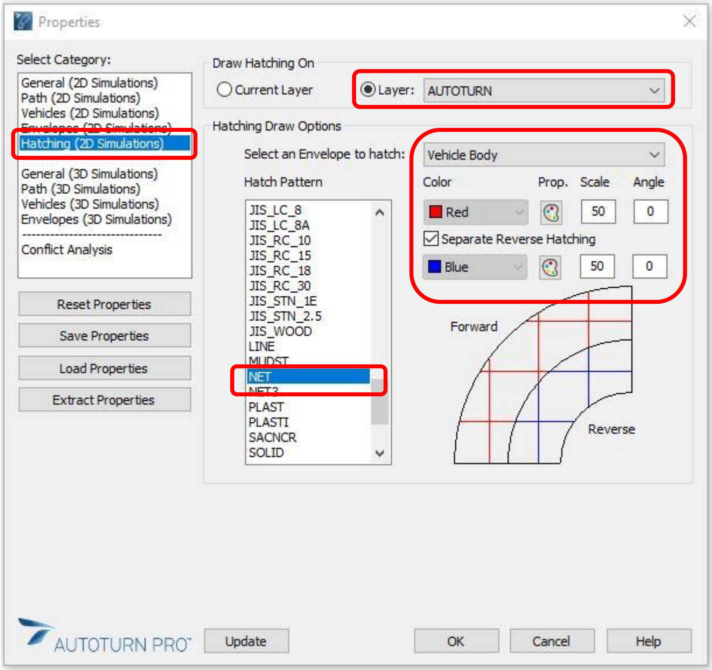


Exhibit 4 – *Properties – Hatching Category*

▶ How to Preset 3D Simulation Properties

NOTE:

The following content refers only to AutoTURN Pro features.

1. Click  **Properties** on the standard AutoTURN Pro toolbar.
2. In the **Properties** dialog box, select **Vehicles (3D Simulations)**. Ensure that **Tires** and **Realistic** options are enabled. See Exhibit 1.

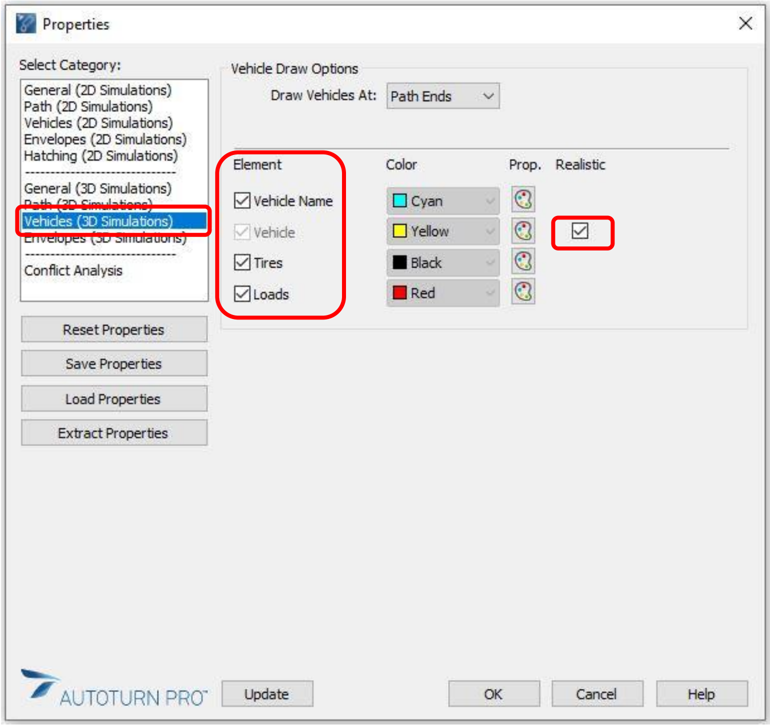


Exhibit 1 – *Properties - Vehicles (3DSimulations) Category*

3. In the **Properties** dialog box, select **Envelopes (3D simulations),** and enable **Vehicle Body**. See Exhibit 2.

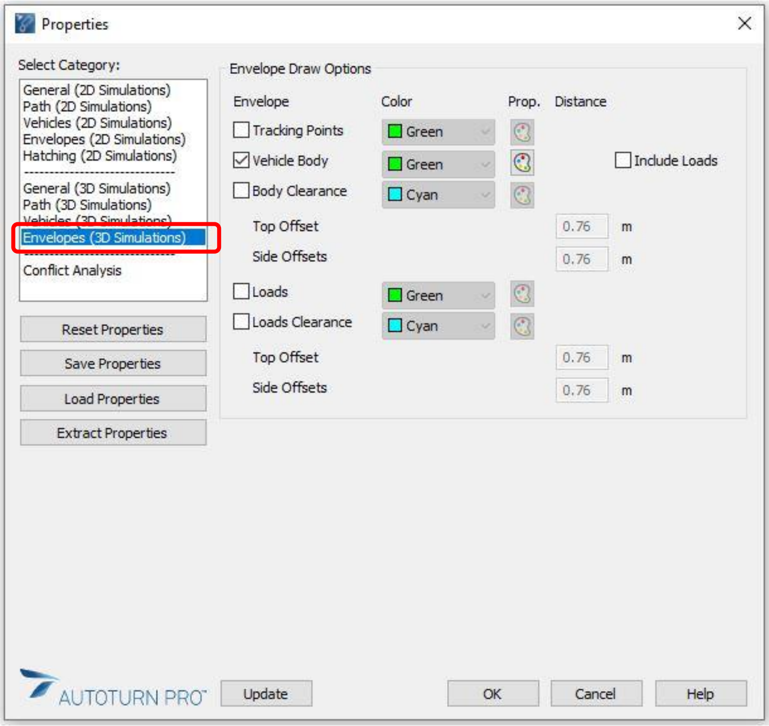


Exhibit 2 – *Properties – Envelopes (3D Simulations) Category*

❑ Now that you have set the AutoTURN program settings and simulation properties, you are ready to continue the lesson by defining a design vehicle.

Additional Notes:

How to Select a Design Vehicle

1. To begin, click **Vehicles** on the toolbar.

> **NOTE:**
> *This allows you to access the standard vehicle libraries within AutoTURN (i.e. AASHTO, TAC, Austroads, etc.).*

2. In the **Select Current vehicle** dialog box, from the list of libraries displayed, select the 'Clear All' option, and then select the **AASHTO 2018** library.
3. Scroll through the library and select the **WB-62** vehicle.
4. Click **OK**. See Exhibit 1.

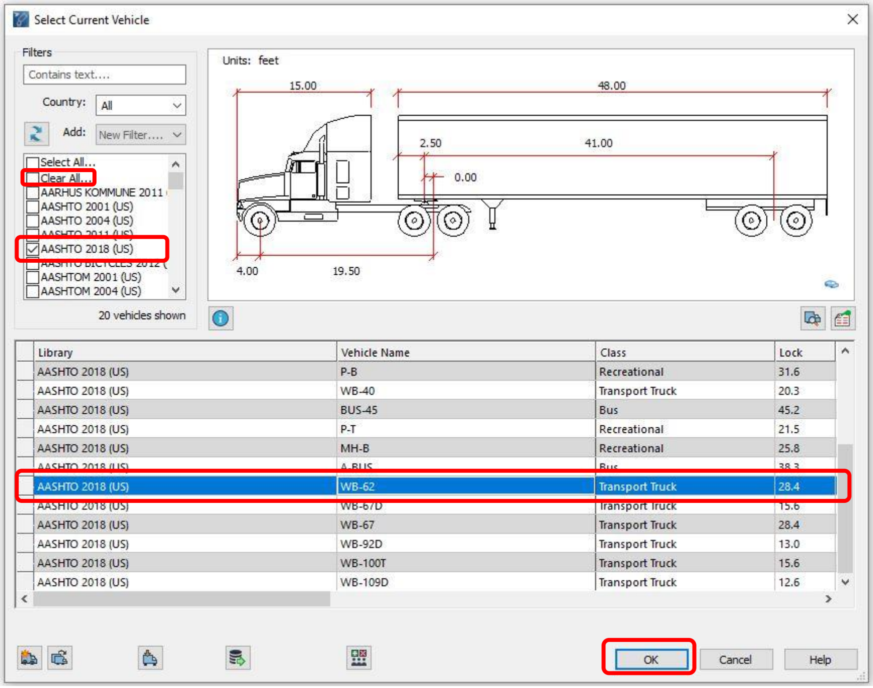


Exhibit 1 – *Select Current Vehicle Dialog Box - AASHTO 2018 WB-62*

❑ **You have now selected a design vehicle that will be used to generate maneuver**

Additional Options to select a design vehicle:

1. **Search Option**: In the **Select Current vehicle** dialog box, under **Filters**, type **AASHTO 2018** in the search box and press the **Enter/TAB** key. See Exhibit 2.

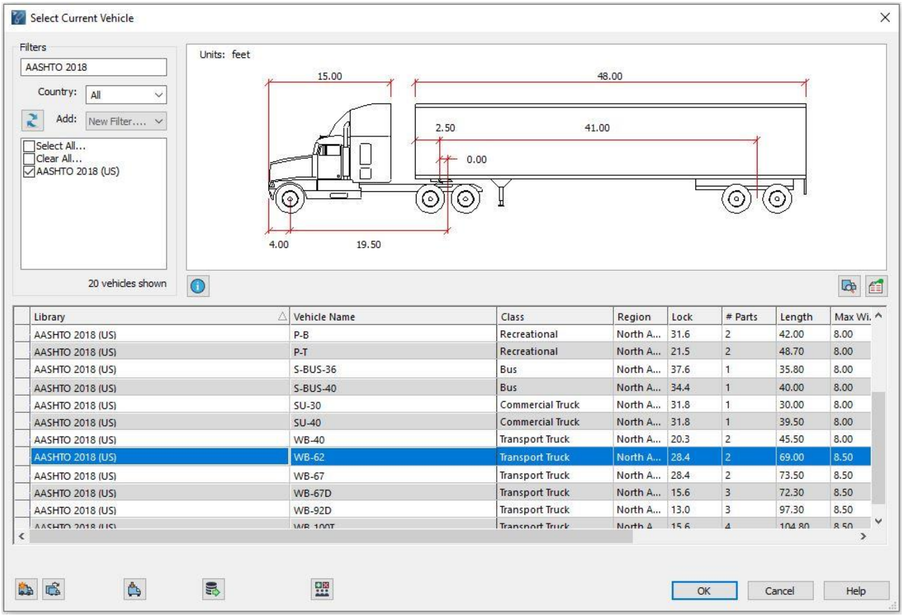


Exhibit 2: Search vehicles

2. **Filter Options**: In the **Select Current vehicle** dialog box, under **Filters** – Add dropdown to view library filter options. Select one or more filter options to filter the vehicle/library list according to **Country, Parts, Class, Features, Guideline, Length, Level, Region, User groups.** See Exhibit 3.

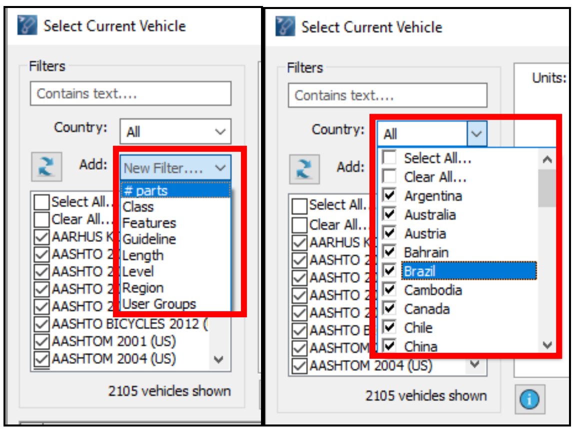


Exhibit 3: Filter options

3. **User Groups**: In the **Select Current Vehicle** dialog box, select the **AASHTO 2018 WB-62** vehicle and then click  **Add/Remove from User Groups** to display the **User Groups** dialog box. See Exhibit 4.

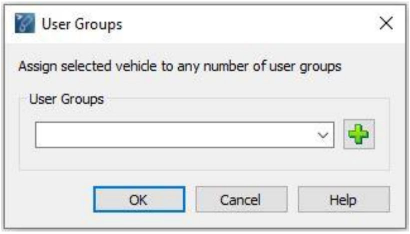


Exhibit 4: User Groups

Click **Add user groups** and in the **New User Group window,** provide the name **PROJECT 1** for the group. Click **OK**. See exhibit 5.

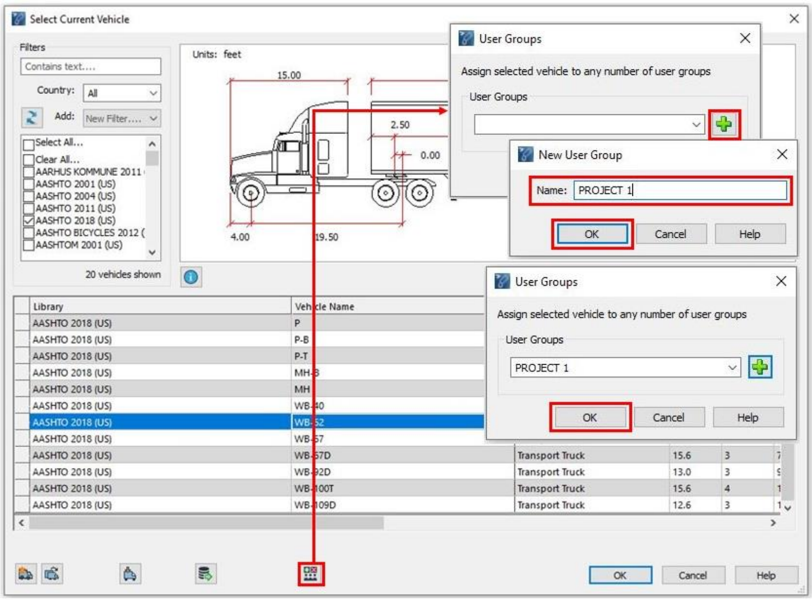


Exhibit 5: User groups – Add vehicle

From the **Select Current vehicle** dialog box, **Add** New Filter – **User Groups** – Select **PROJECT 1.** See Exhibit 6.

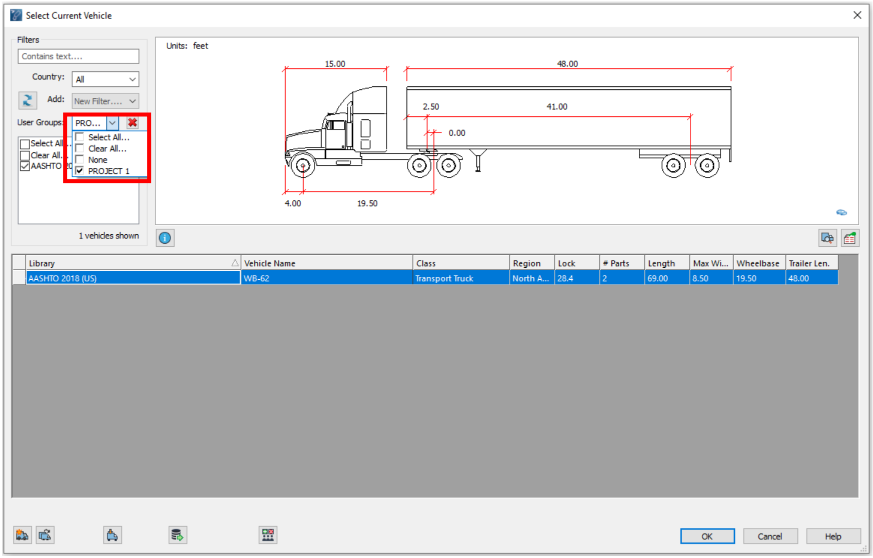


Exhibit 6: User groups – vehicle display

The **user group** named **PROJECT 1** is displayed containing vehicle **WB-62** from **AASHTO 2018** library.

Additional Notes:

How to Place an Adaptive Simulation

The **Place Adaptive Simulation** tool places a turn simulation using the current vehicle based on following selected path elements with the center path of the steering axle group or by offsetting the turn simulation a specified distance relative to the selected turn simulation envelopes (an offset turn simulation).

In this next exercise, Song & Chan Architects Inc. has been commissioned to design the new Burger Joint restaurant at the corner of No. 3 Road and 6th Avenue. The objective of this exercise is to use the Large Passenger Car and WB-40 design vehicles from the AASHTO 2018 library to test the width tolerance of the drive-thru lane.

1. Select Passenger Car from the AASHTO 2018 (US) library. See Exhibit 1.

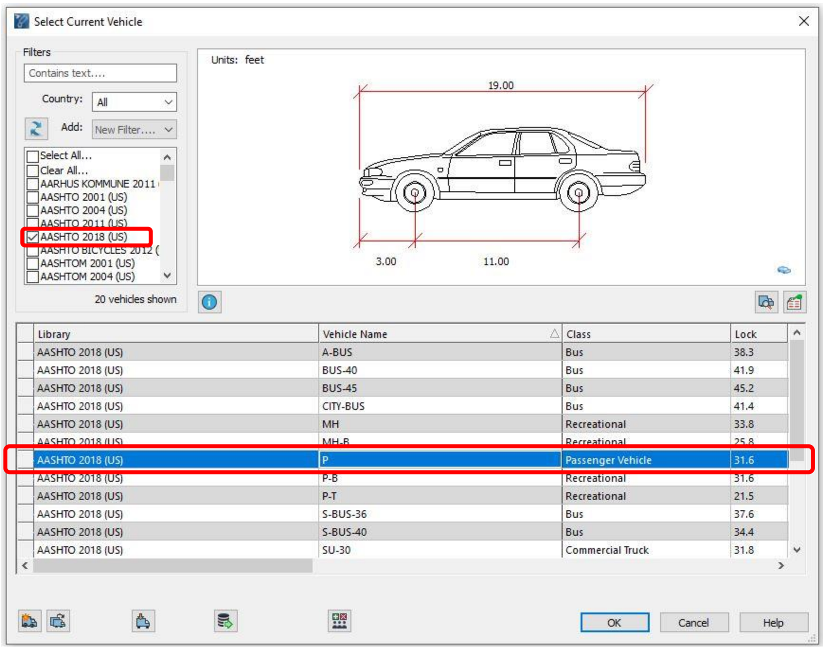


Exhibit 1 – *Select Current Vehicle – Passenger Car*

2. Click  **Place Adaptive Simulation** on the toolbar.

TRANSOFT SOLUTIONS

3. The **Place Adaptive Simulation** tool will prompt you to pick the reference line(s).
4. Select the following reference lines in the order you expect the vehicle to follow the path (i.e.1 to 5 assuming the vehicle starts traveling at 1 and ends at 5). See Exhibit 2.

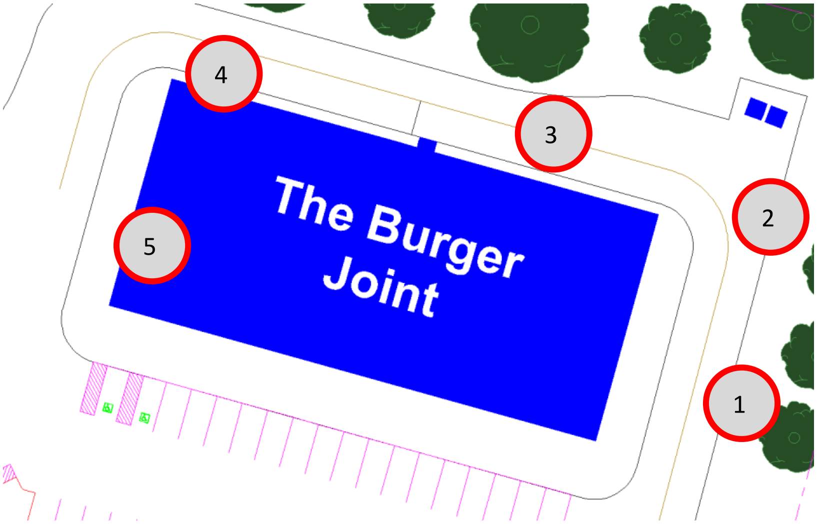


Exhibit 2 – *Reference Lines*

5. Pick any blank area in the drawing after finishing selecting the desired path. See Exhibit 3.

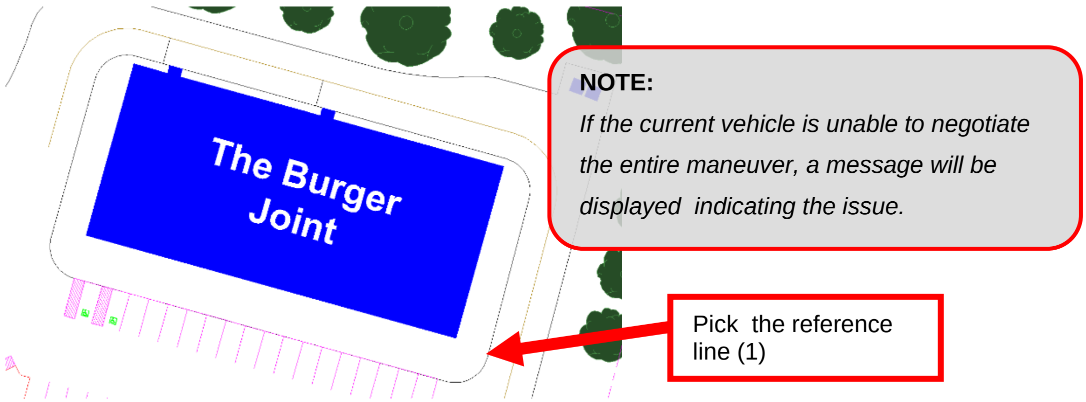


Exhibit 3 – *Starting Point*

6. Immediately after indicating the simulation starting point, the **Place Adaptive Simulation** dialog box appears. Set the offset simulation to Offset Right, and enter an offset value of *2* feet. See Exhibit 4.

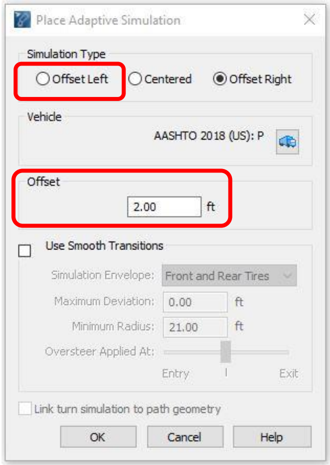


Exhibit 4 – *Place Adaptive Simulation Dialog Box*

7. The vehicle path that is generated is a low-speed simulation. Notice that the "Turn Wheels from Stop" operation is applied. See Exhibit 5. The vehicle path is based on the selected geometry. If the selected geometric radius selected is less than the vehicle's minimum turning radius, the simulation will end at the point where the radius exceeds.

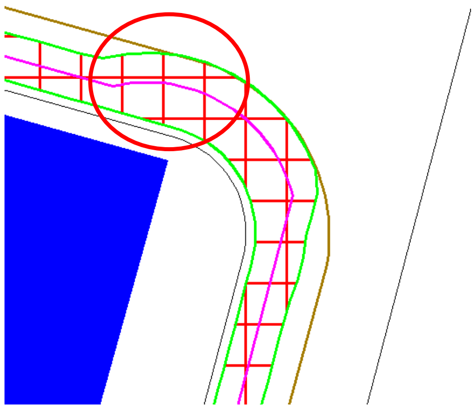


Exhibit 5 – *Path Based on Reference Geometry*

8. Select **Use Smooth Transitions**. Notice that the vehicle minimum turning radius is used to calculate the Entry and Exit of the simulation. The vehicle minimum turning radius is used to calculate the entry and exit offsets of the vehicle's path. See Exhibit 6.

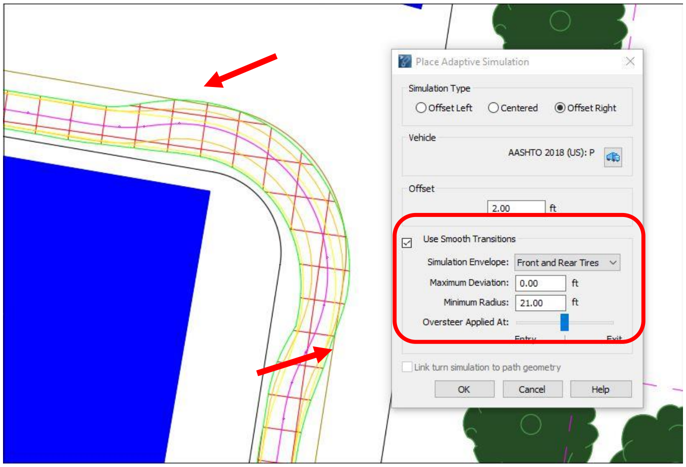


Exhibit 6 – *Effect of Use Smooth Transitions on Entry and Exit Offsets*

9. The simulation has been created. See Exhibit 7.

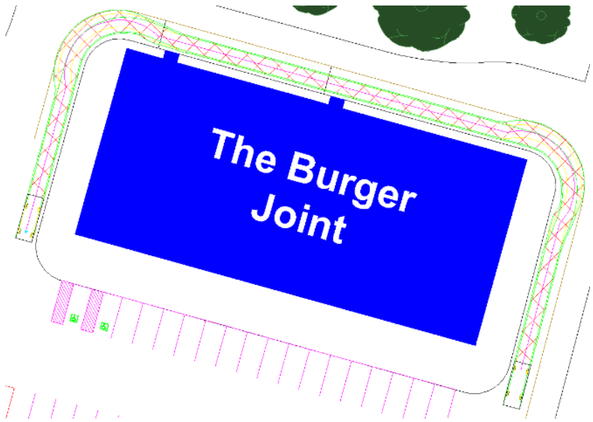


Exhibit 7 – *Place Adaptive Simulation*

10. Repeat the previous steps for the WB-40 design vehicle. See Exhibit 8.

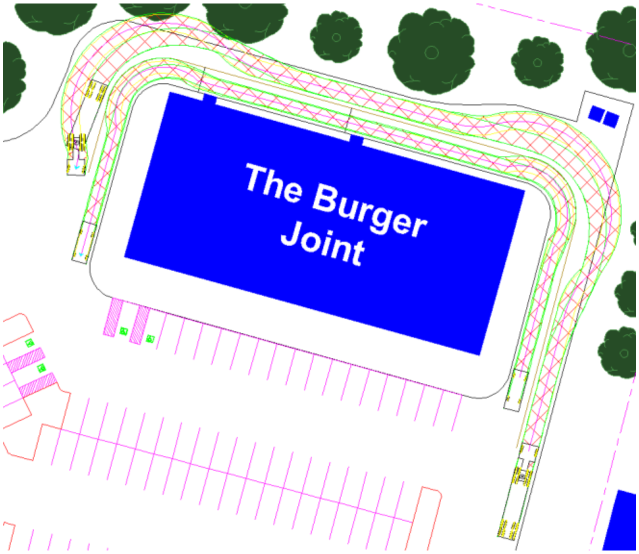


Exhibit 8 – *WB-40 Offset Simulation Results*

Based on the results of the Place Adaptive Simulations of the two design vehicles, to allow the WB-40 to service this Burger Joint location, the architects will have to widen the back corridor.

Additional Notes:

How to Check Geometric Conflicts

The **Conflict Analysis** tool allows designers to evaluate portions of user drawn elements that conflict with the turn simulation.

1. Update the Properties → Conflict Analysis → Obstacles Layers to the layer/level containing the following lines. See Exhibit 1.

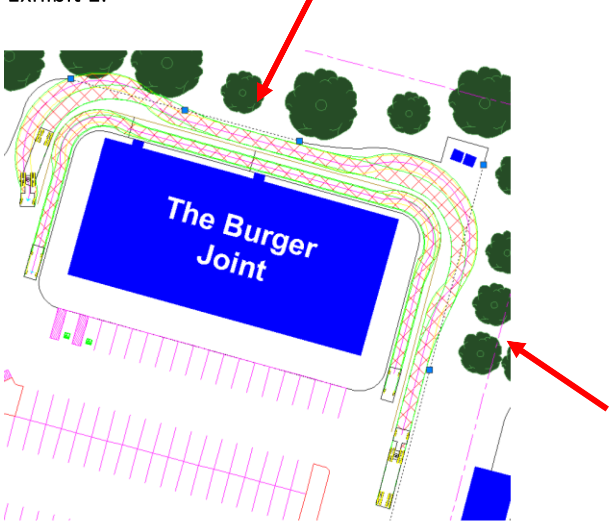


Exhibit 1 – *Conflict Analysis – Adding Obstacle layer.*

2. Click **Regenerate Simulation** and select the WB-40 turn simulation. See Exhibit 2.

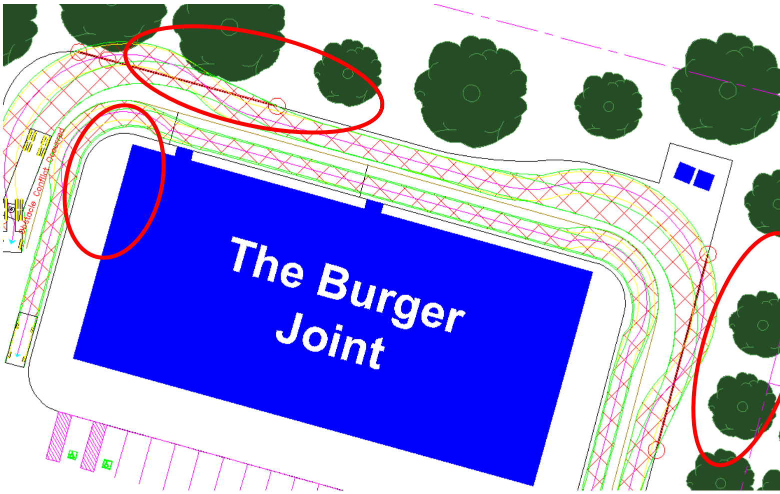


Exhibit 2 – *WB-40 Conflict Analysis display*

An 'Obstacle Conflict Occurred' callout is displayed, and the conflict areas are denoted in red.

Additional Notes:

How to Update Simulation Properties

In the following exercise, we will learn how to differentiate between two simulations by changing their display properties.

The architects at Song & Chan are preparing the Burger Joint site drawing for a client presentation. To differentiate the display properties of the WB-40 from the Passenger vehicle, the display properties for these vehicles will need to be changed.

1. Click  **Properties** on the toolbar.

2. In the **Properties** dialog box, select the **Envelopes** category. Change the color of the vehicle body and the front and rear tire envelopes as illustrated in Exhibit 1.

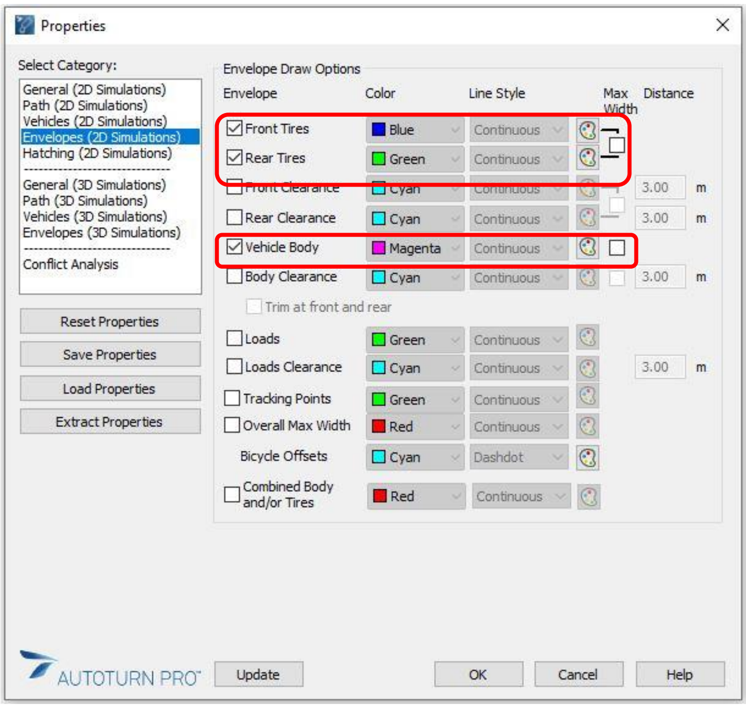


Exhibit 1 – *Properties Dialog Box – Envelopes Category Settings*

3. Select the **Hatching** category. Change the hatch pattern to **Solid** and change the color as indicated in Exhibit 2, set the transparency to be 70%.

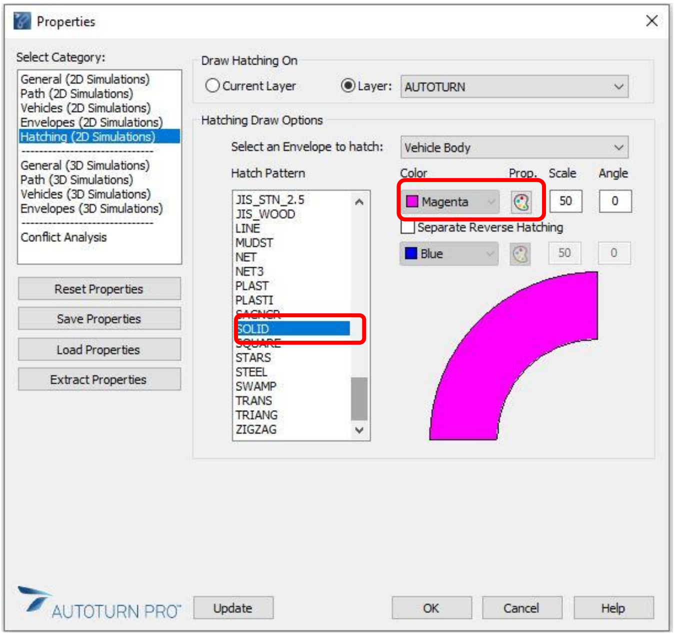


Exhibit 2 – *Properties Dialog Box – Hatching Category Settings*

4. Click **Update** and select the Passenger Car simulation. Click **OK** to exit the **Properties** dialog box. See Exhibit 3.

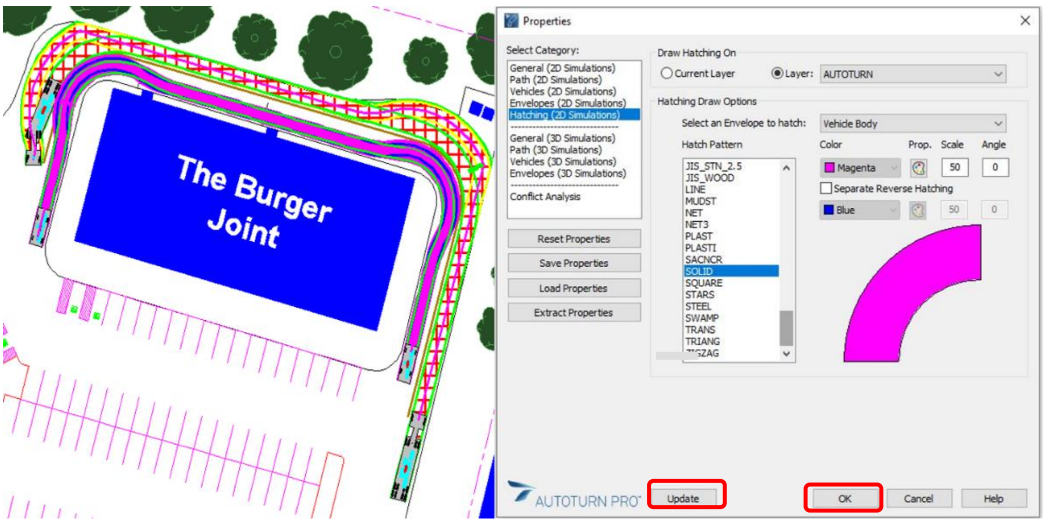


Exhibit 3 – *Updated Display Properties*

NOTE:
New simulations will have the new display properties.

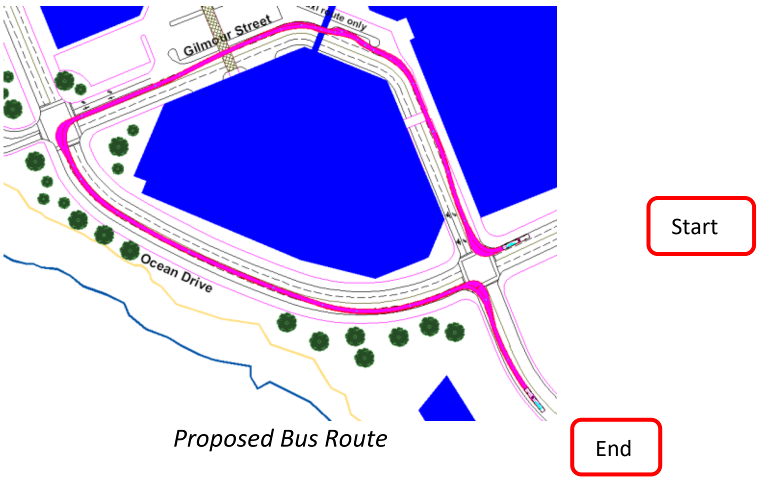
Additional Notes:

How to Generate a Low-Speed Corner Simulation

In the following exercise, we will explore the **Generate Corner Path** tool, and learn how to apply this knowledge to real-world design challenges.

The **Generate Corner Path** tool generates a turn simulation using an entrance tangent, arc, and exit tangent, with input for the turn radius and the amount of turn sweep. The sweep can also be set by selecting an existing element. The tangents can be lengthened or shortened by moving the mouse in the direction of the tangent.

Airport Express
Oceans Center Mall has recently expanded the mall due to the thriving tourist boom. The Transit Corporation is planning to introduce an Airport Express bus route that will service between TRC Int'l Airport and Oceans Center Mall. The proposed transit route will allow buses to enter at the four-way intersection of Ocean Drive, Gilmour Street, and Campbell Street. The bus will make a right turn and make its way towards the proposed bus stop. Once the bus has picked up and dropped off passengers, it will make its way towards Ocean Drive, where it will make a left turn and travel towards Campbell Street. The objective of this exercise is to use the Generate **Corner Path** and 2D **Arc Path** tools to navigate the A-BUS (AASHTO 2018) in and out of Oceans Center Mall.



Proposed Bus Route

1. Select the A-BUS from the AASHTO 2018 (US) Design Vehicle Library. See Exhibit 1.

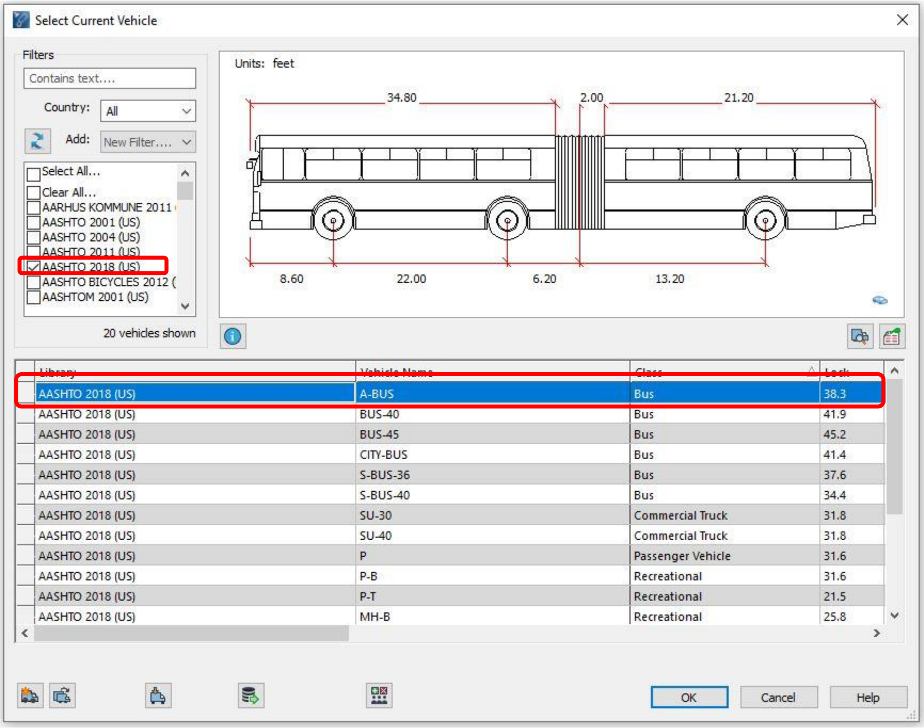


Exhibit 1 – *Select Current Vehicle Dialog Box – AASHTO 2018 (US) A-BUS*

2. Click **Generate Corner Path** on the toolbar.

3. Place the A-BUS on the right lane at the intersection of Ocean Drive and Gilmour Street. Set the vehicle's starting position. See Exhibit 2.

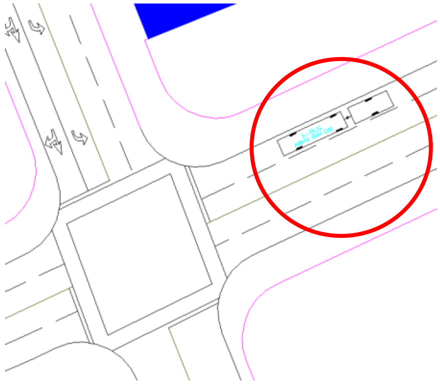


Exhibit 2 – *A-BUS Starting Position*

4. Set the approach speed to 10 mph and select the sweep path angle. Select a geometry line to reference the sweep angle. See Exhibit 3.

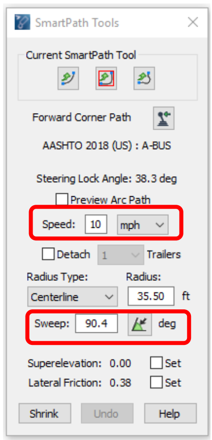


Exhibit 3 – *Smart Path Tools – Speed and Sweep Settings*

5. Drag the A-BUS forward to make the right turn onto Gilmour Street. Click to place the second tangent. Switch over to the 2D **Arc Path** tool, and navigate the design vehicle along Gilmour Street towards Ocean Drive. See Exhibit 4.

TRANSOFT SOLUTIONS

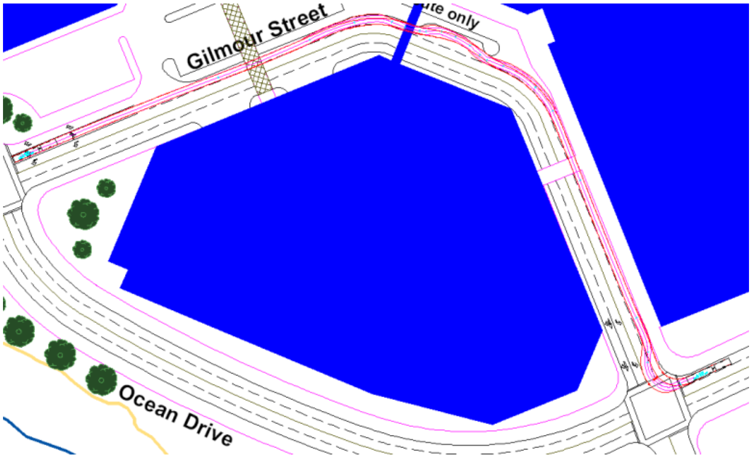


Exhibit 4 – *Arc Path Route*

6. Once you have reached the intersection of Gilmour and Ocean Drive, switch to the **Corner Path** tool. See Exhibit 5.

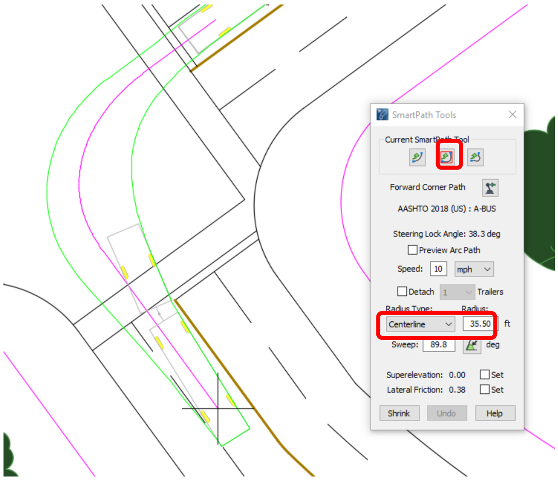


Exhibit 5 – *Corner Path Tool*

How to Generate Reverse Maneuvers

In this next exercise, we will use the 2D **SmartPath** tools to maneuver a WB-62 into the Snail-Mail Shipping depot located at the TRC International Airport. The objective of this exercise is to explore AutoTURN's vehicle reverse maneuvers. You can create reverse maneuvers using the 2D **Arc Path** and Generate **Corner Path** tools. Each one has its unique characteristics and will yield different results. To enable reverse in AutoTURN, just simply drag the vehicle backward.

1. Select the WB-62 design vehicle from the AASHTO 2018 (US) Vehicle Library. See Exhibit 1.

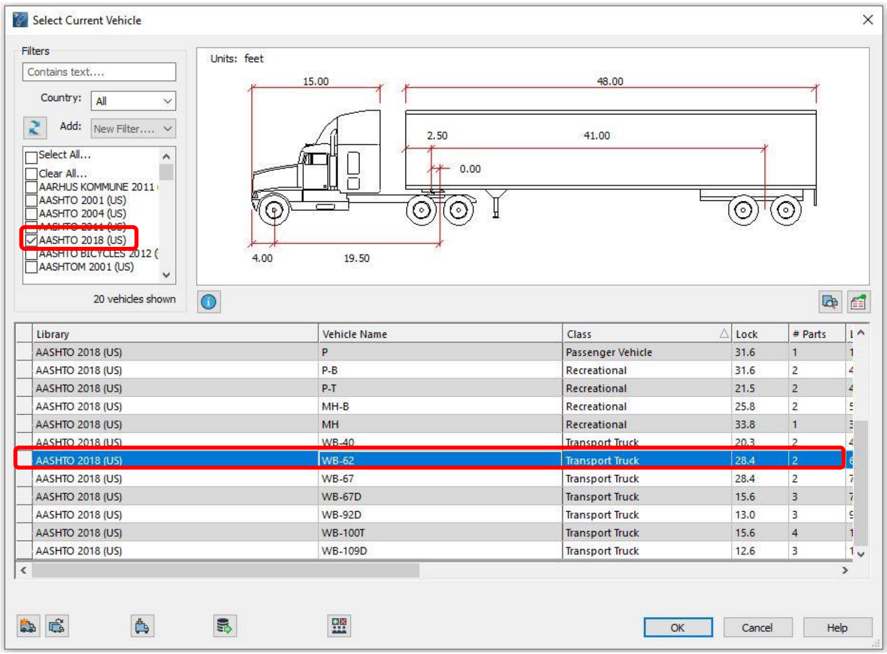


Exhibit 1 – *Select Current Vehicle – AASHTO 2018 (US) WB-62*

2. Click  **Generate 2D Arc Path** on the toolbar.

3. Place the WB-62 as illustrated. See Exhibit 2.

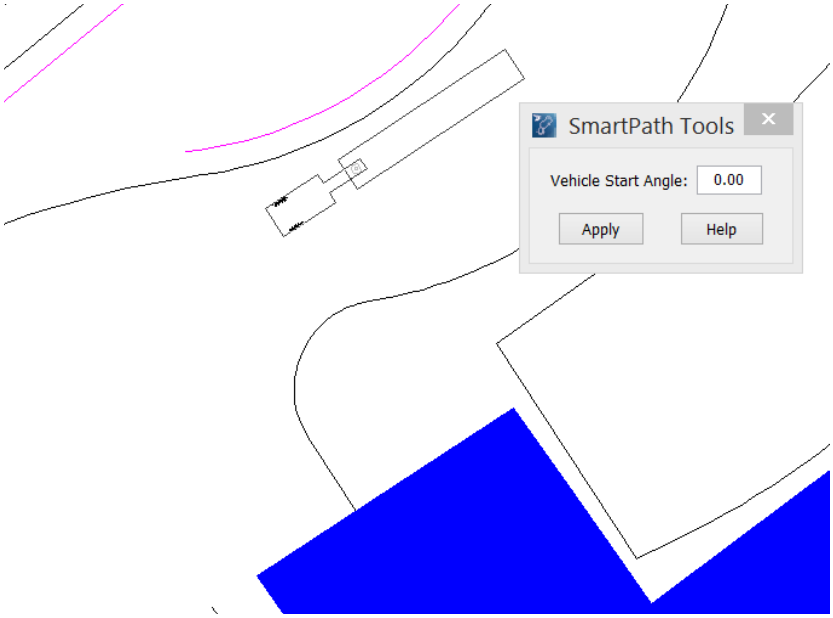


Exhibit 4 – *Place vehicle*

4. Perform a 180-degree turn, and right-click to end the simulation. See Exhibit 5.

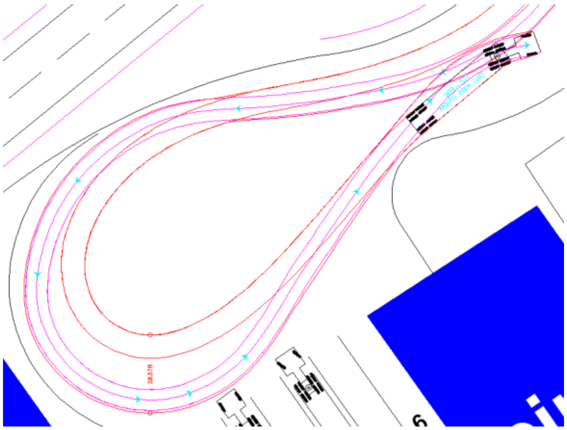


Exhibit 5 – *180-degree Turn*

How to Continue a Simulation

In the following exercise, we will learn how to use the Continue Simulation command in AutoTURN. We will create and save various reverse simulations on specified layers through the Continue Simulation command.

1. Click **Continue Simulation** on the toolbar to continue the simulation.
2. In the **Continue Simulation** dialog box, select Continue as a new simulation and New Layer. Type *Bay 1* in the New Layer edit box. See Exhibit 1.

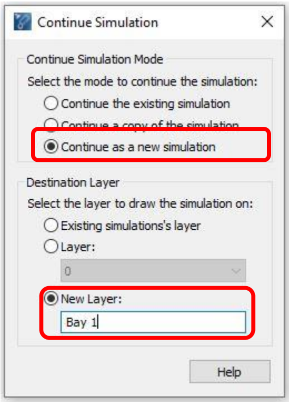


Exhibit 1 – *Continue Simulation – New Layer*

NOTE:

Continue Simulation options:

Continue the existing simulation *– continues from the end of an existing simulation.*

Continue *a copy of the simulation - continues a copy of an existing simulation, leaving the original simulation intact.*

Continue as a new simulation *- starts a new simulation from the end of an existing one.*

3. Drag the WB-62 backward to enable the reverse function. The **Reverse** dialog box will appear for the **Arc Path**, **Corner Path,** and **Steer A Path** tools. See Exhibit 2.

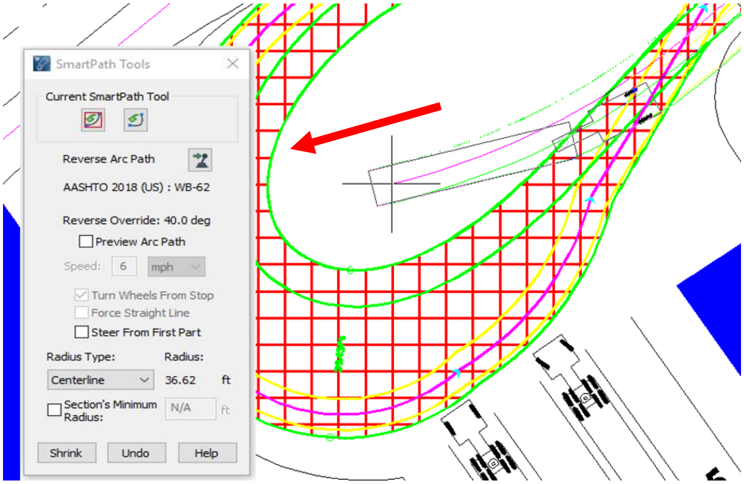


Exhibit 2 – *Dragging the Vehicle Backwards to Engage Reverse*

4. Click the **Reverse Corner Path** tool, click **Select Sweep Angle**, and click the reference curb geometry near Bay 1. See Exhibit 3.

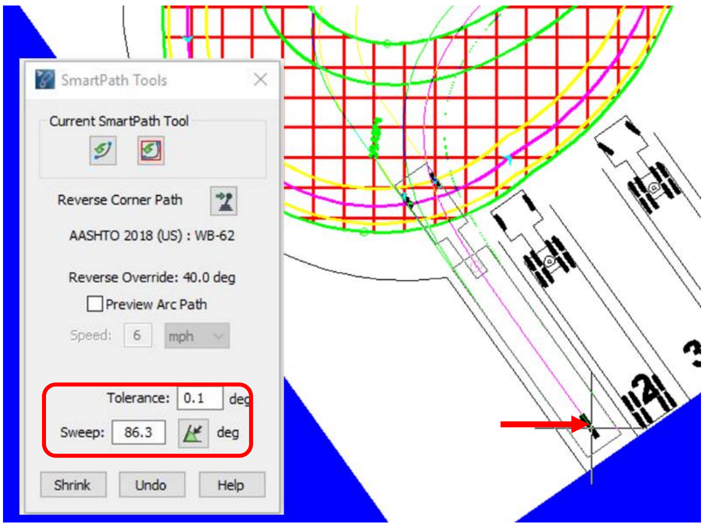


Exhibit 3 – *Select Sweep Angle*

NOTE:

*When you are using **Reverse Corner Path**, you can specify a sweep angle or you can reference an object's angle for the sweep angle.*

5. Hide the Bay 1 layer.
6. Click **Continue Simulation** on the toolbar and select **Continue as a new simulation**. Place the simulation on a New Layer called *Bay 5*. See Exhibit 4.

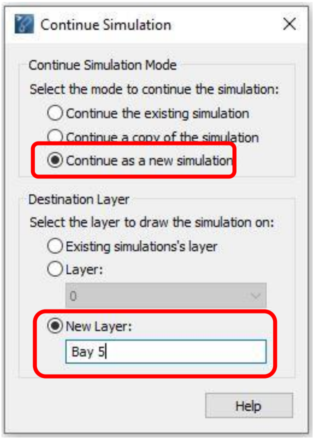


Exhibit 4 – *Continue Simulation – New Layer*

7. Reverse the WB-62 into Bay 5 using the **Reverse Arc Path** tool. See Exhibit 5.

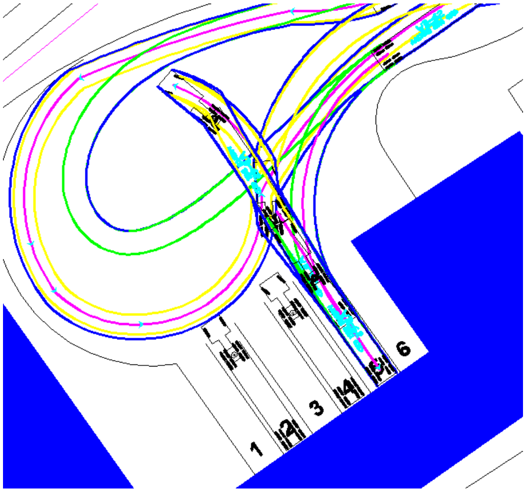


Exhibit 5 – *Reverse Arc Path tool*

8. Enable the Layers Bay 1, Bay 3, and Bay 5 to compare.

Three-Part Reverse

Performing reverse maneuvers for vehicles with three parts is similar to the two-part vehicle reverse. The following three-part design vehicles can perform a three-part reverse.

THREE-PART DESIGN VEHICLES				
ALBERTA INFRTRA-HGDG (CA)	**AUSTROADS 2013 (AU)**	**STANDARDS 2018 (AU)**	**CROW Advieslijst LZV**	**TAC 2017 (CA)**
Log Haul Truck	B-Double 26M	A-Double	LZV Configuratie E	BTD
WB23	B-Double 25M	B-Double		ATD

How to Generate Oversteer Path

The **Generate Oversteer Corner Path** simulates an oversteer maneuver performed by a design vehicle. This maneuver is similar to the **Generate Corner Path** tool; except entry and exit offsets are used to control the amount of oversteer a design vehicle will require to complete a maneuver. This tool is useful for maneuvering a design vehicle into a tight intersection where real estate is limited.

Right-Turn Headache

There is a major traffic problem near Keeven Road and Shihundu Drive. This non-signalized intersection was originally designed to allow medium-sized trucks to service this industrial corridor. Recently, the old intersection at Spears Boulevard and Keeven Road was replaced with a roundabout. This roundabout was designed to service a WB-62 that will allow more goods to be delivered and picked up from the local businesses. With the increased presence of larger sized transport vehicles in this area, local drivers are complaining about the traffic congestion at the intersection of Keeven Road and Shihundu Drive.

The objective of this exercise is to create a simulation of a WB-62 negotiating a right turn and analyze the results of the simulation.

1. Select the AASHTO 2018 (US) WB-62 design vehicle from the Vehicle Library. See Exhibit 1.

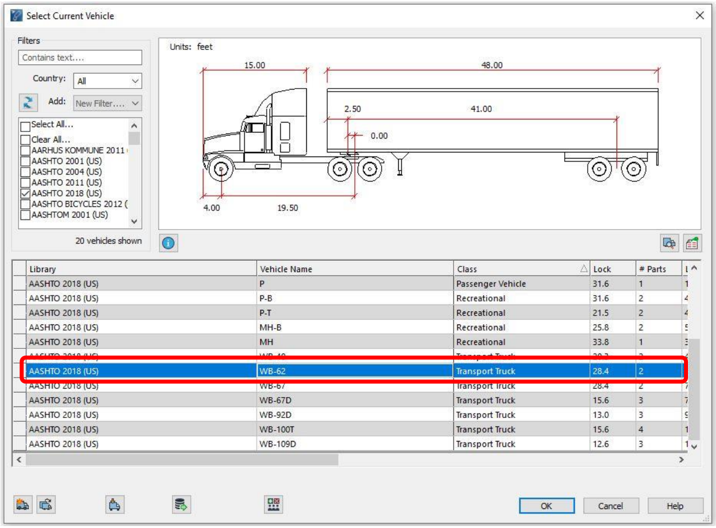


Exhibit 1 – *Select Current Vehicle – AASHTO 2018 – WB-62*

2. Click  **Oversteer Corner Path** on the toolbar.

TRANSOFT SOLUTIONS

3. Place the WB-62 near the intersection of Shihundu Drive and Keeven Road. Set the appropriate starting angle. See Exhibit 2.

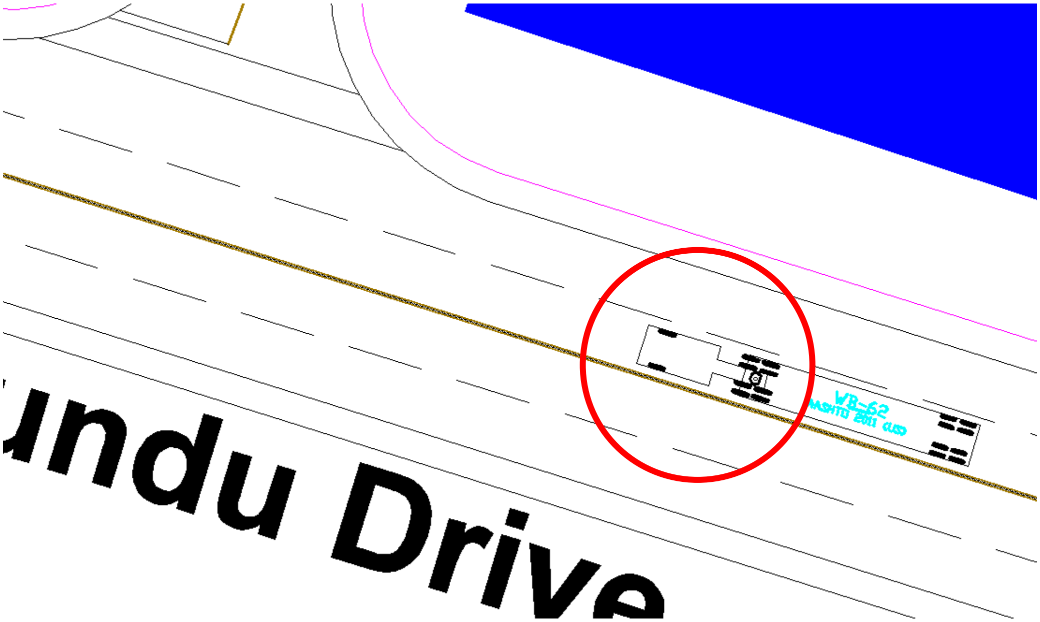


Exhibit 2 – *Vehicle Placed Near Intersection*

▶ How to Reposition the Vehicle's Starting Position

1. Place the mouse pointer near the front cab of the design vehicle. See Exhibit 3.

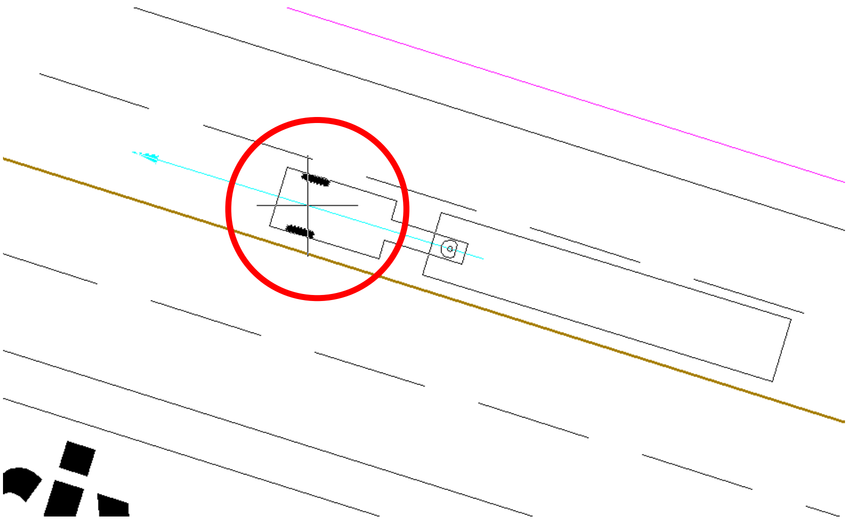


Exhibit 3 – *Vehicle Repositioning*

2. Click to pick up the vehicle and reposition the vehicle's starting position in the right lane. See Exhibit 4.

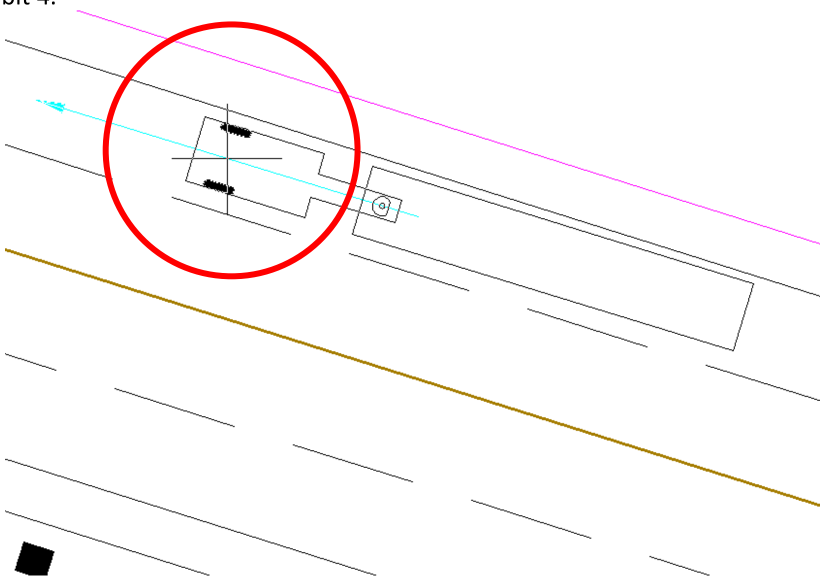


Exhibit 4 – *Vehicle Repositioning*

3. Set the approach speed to 6 mph and enter *95* degrees for the vehicle sweep angle. Set the **Entry Offset** to *20* feet and select the **Use Minimum Exit offset** checkbox. See Exhibit 5.

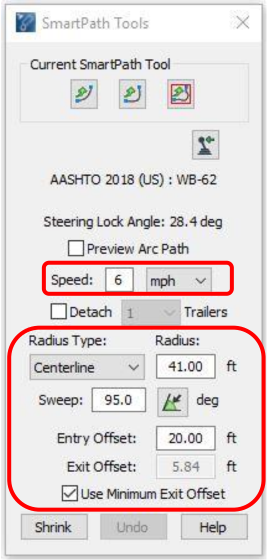
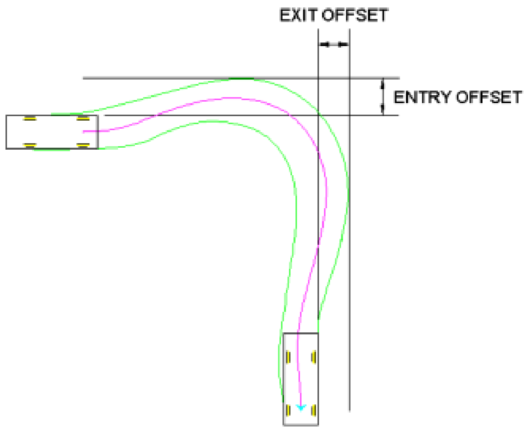


Exhibit 5 – *SmartPath Tools – Forward Oversteered Path Settings*

> **NOTE:**
> *The entry and exit offsets are measured from the body of the vehicle's first part perpendicular to the extents of the vehicle's first part envelope.*



Oversteer Corner Path - Offset Dimensions

A high Entry and Exit offset* value will generate a wider turn. The **Use Minimum Exit** option provides the minimum Exit value of that design vehicle.

***There are no predetermined offset values. Use your engineering judgment to determine the optimum offset values.**

4. Click to complete the first segment of the right turn, and right-click to end the command. See Exhibit 6.

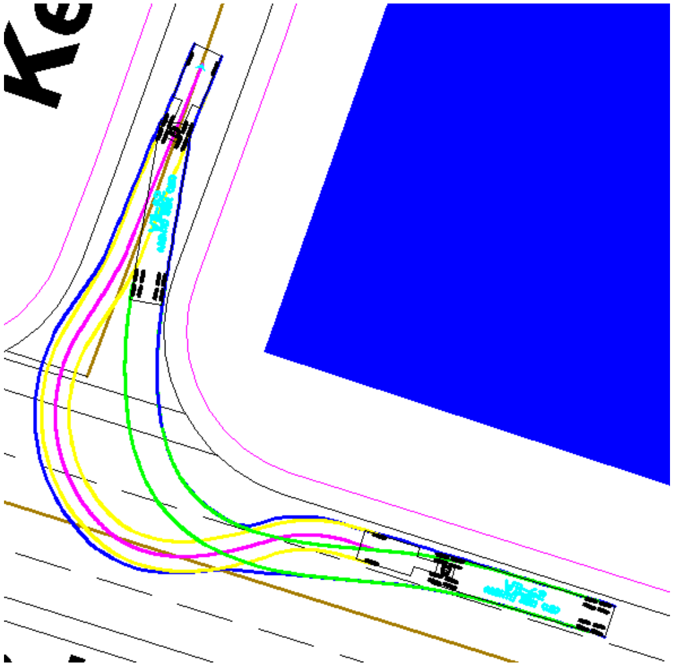


Exhibit 6 – *Oversteered Path*

From this exercise, we can see the problems that exist at this intersection, and we can propose possible solutions that will allow WB-62 sized vehicles to execute the right turn onto Keeven Road.

Summary: Items Covered in this Section

- Generate Oversteer Corner Path
- Specify a Sweep Angle
- Entry Offset
- Exit Offset
- Use Minimum Exit offset
- How to reposition the vehicle's starting point

How to use the Inspect Simulation (AutoTURN Pro only)

The Inspect Simulation tool helps review simulations created with AutoTURN to ensure they meet the project's design criteria and use realistic parameters. The tool reports on selected key parameters such as steering lock angle, input speed, maximum speed, proximity to elements on specific layers/levels, turn wheels from stop condition, and minimum turn radius. The ranges for several of these key aspects can be set and inspected depending on the characteristics of the design scenario.

Analyzing simulations

In this example, we will use the Inspect Simulation tool to review two simulations created using AutoTURN to analyze that key parameters were properly set when creating them, and how these elements can influence the outcome and its impact on the project.

Exhibit 1. Inspect simulation – opposing turns

4. Open the drawing **Inspect Simulation.dwg or .dgn** depending on your CAD platform. Note the 3 simulations that are already in the drawing, visually all the simulations appear to be valid and in the case of the opposing turns, there seems to be sufficient clearance to accommodate both turns safely.

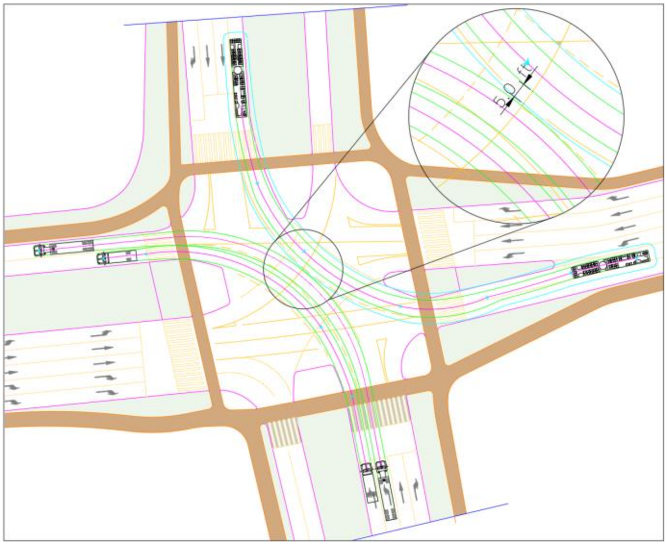


Exhibit 2. Opposing turn clearances

Besides the visual appearance of the simulations, there is uncertainty about the settings or parameters used to create the simulation that could have a significant impact on the design scenario such as the creation speed and the use of the turn wheels from stop, which is only recommended for very low-speed maneuvers (below 6 mph 10 km/h) at very tight situations.

5. Under the reports section, click on the **Inspect Simulation** tool from the ribbon.

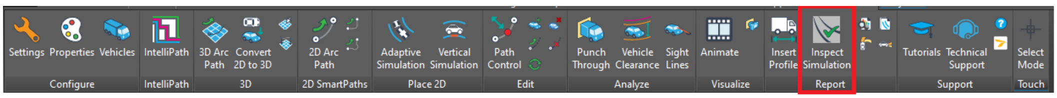


Exhibit 3. Launch the Inspect Simulation tool

6. The dialog will allow you to select the different design criteria or parameters to inspect simulations according to the project or design needs

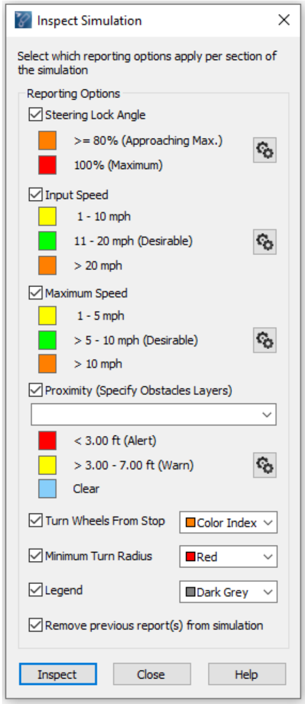


Exhibit 4. Inspect Simulation dialog

In this particular exercise we will be looking at 3 particular conditions:

a. The use of turn wheels from stop: This capability is only recommended for tight situations where the vehicle needs to come to a complete stop and perform drastic changes to the steering, good examples of these are loading docks and parking lots where the vehicles have limited space and will be moving at very low speeds.

b. The creation speed: At speeds, over 10 mph the minimum turning radius of the vehicle is a function of the speed, friction coefficient, and superelevation.

c. Proximity to objects and other simulations: ensure vehicles have sufficient clearance to road elements as well as other vehicles is essential to ensure the safety and efficiency of the design.

7. Select the input speed and click on the Settings button:

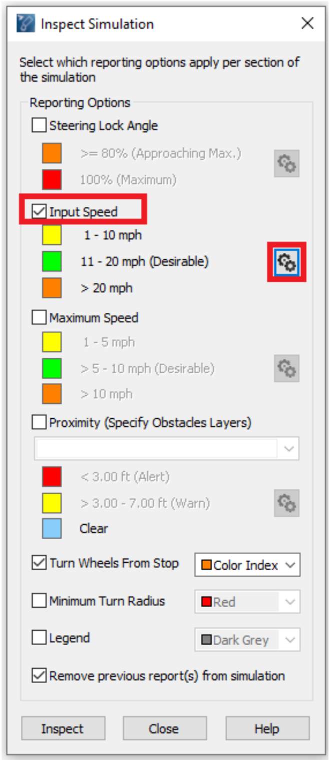


Exhibit 5. Select input speed and settings criteria

Configure the ranges for the different input speed criteria as per Exhibit 6 below. Whilst a speed below 10 mph (16km/h) are feasible for vehicles making these turns if they are starting to move after a red light, however, a more critical condition would be if they are already moving at a higher speed. If the simulation was created at lower than desirable speed, in reality, drivers may turn wider if already in motion when approaching the intersection. Hence, configure the tool to highlight in yellow if the input speed is lower or equal to 10 mph (16 km/h), green if the input speed is between 11 mph (18 km/h) and 20 mph (32 km/h), and orange fr speeds over 21 mph (34 km/h) and click OK.

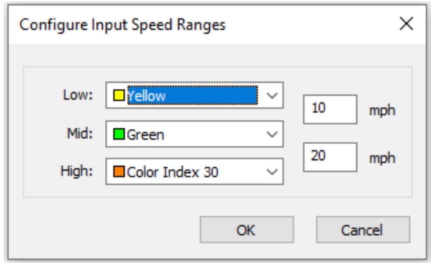


Exhibit 6. Configure the input speed ranges

8. Select the turn wheels from stop option and Inspect:

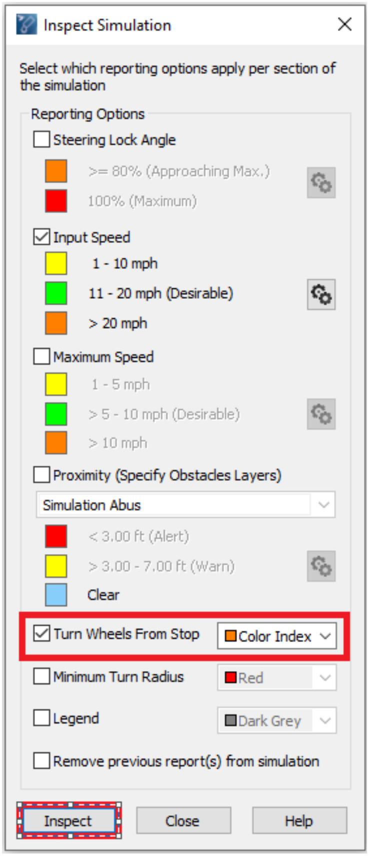


Exhibit 7. Select the Turn Wheels From Stop

9. Select the articulated bus simulation and the semi-trailer simulation, and observe the two reports:

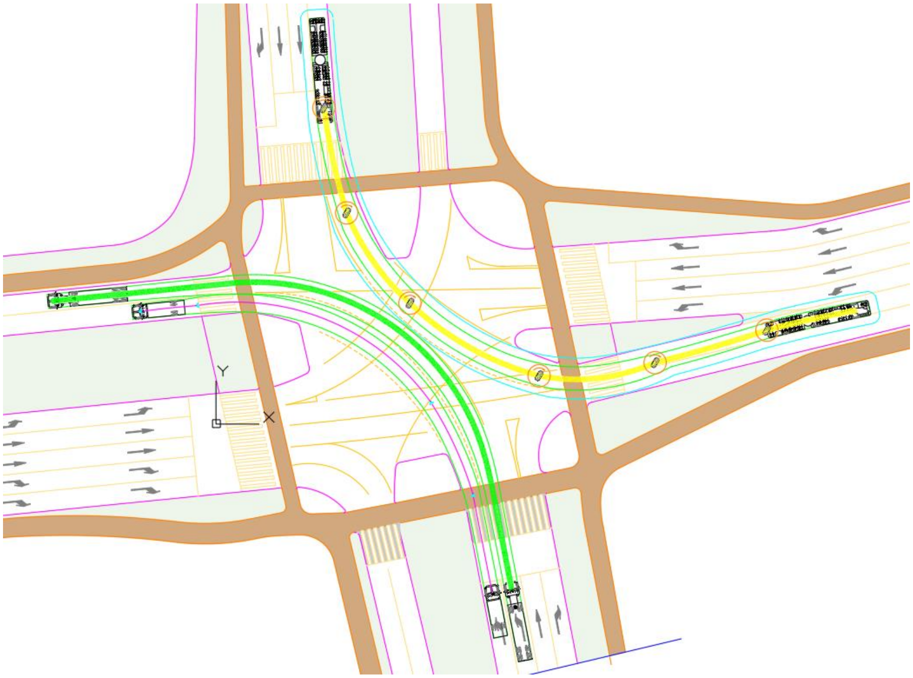


Exhibit 8. inspect opposing turn simulations

Whilst the semi-trailer simulation matches the project's design criteria, note how the input speed in the case of the Articulate bus is lower. More so, note how the "Turn Wheels from Stop" symbol is displayed for every section of the simulation which would be very unlikely in a realistic situation.

10. Click on properties and under the general category select the layer/level named: "Simulation Abus" and click OK:

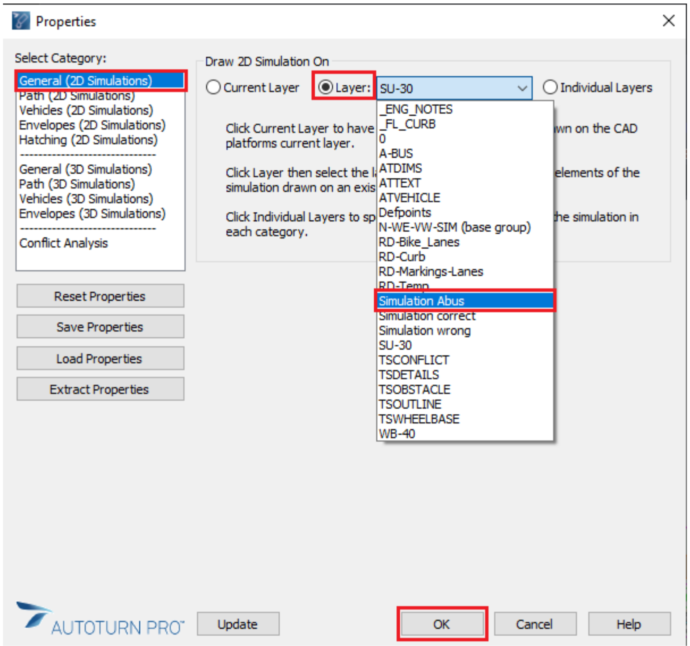


Exhibit 9. Change the simulation layer

11. Under the Envelopes (2D Simulations) Category verify that the Body Clearance envelope is not checked:

TRANSOFT SOLUTIONS

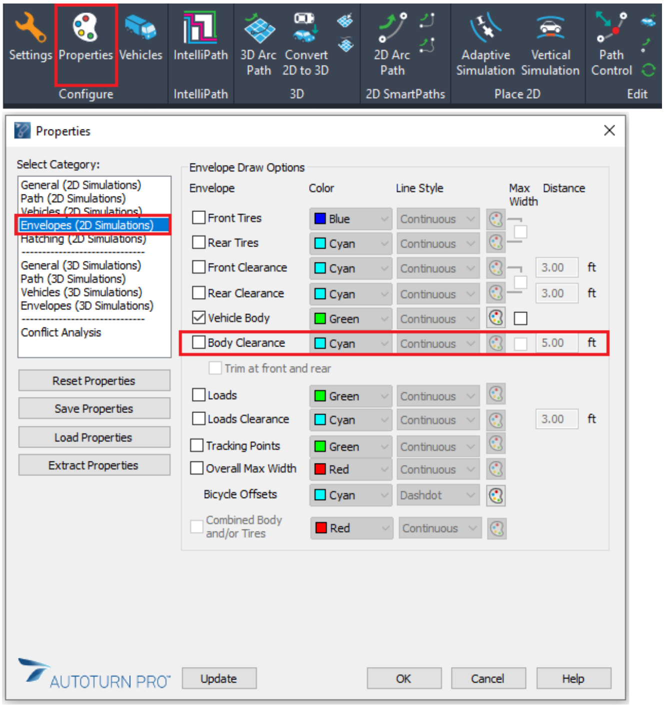


Exhibit 10. Uncheck the Clearance Envelope option

12. Click Vehicles and select the Articulated Bus from the 2018 AASHTO vehicle library and click OK

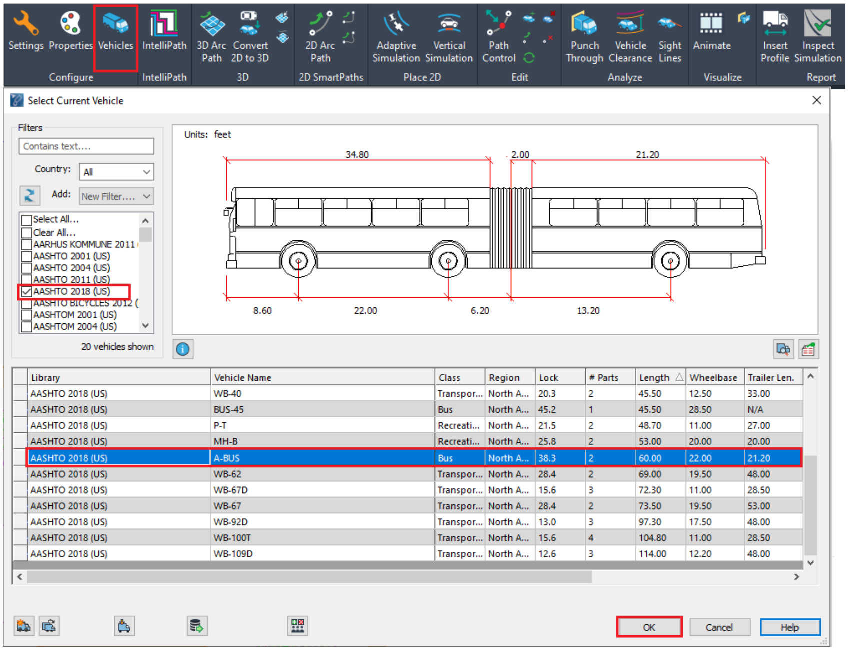


Exhibit 11. Select the Articulated Bus from the AASHTO 2018 library

13. Delete the articulated bus simulation, and click the Corner Path tool, to create a new simulation for the Articulated bus. Set the speed to be 20 mph (32 km/h), use the Select Sweep Angle option to pick the lane line to align the vehicle with it at the of the movement.

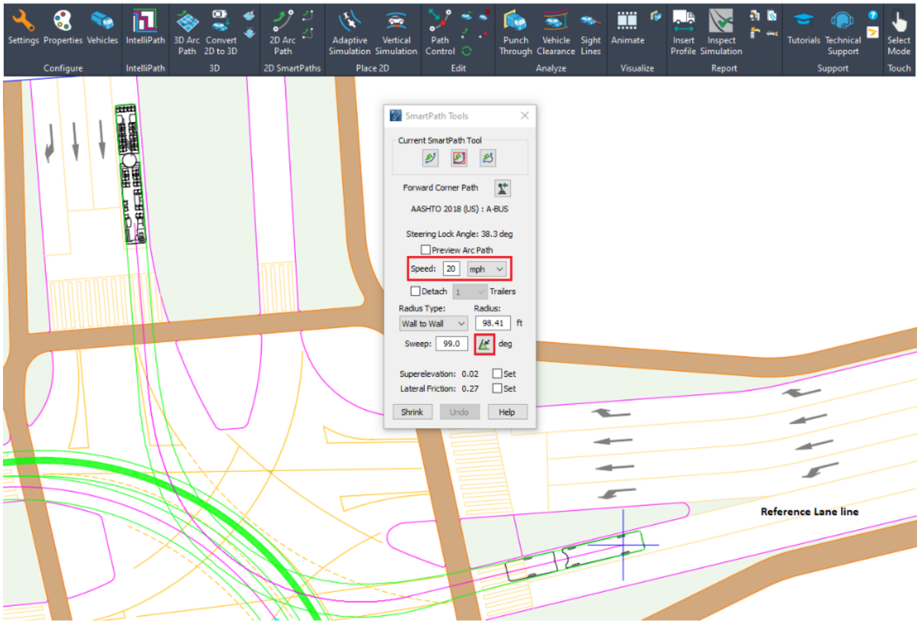


Exhibit 12. Corner Path, Input Speed and Sweep Angle from geometry

14. Before placing the section change the radius to 160 ft and place the simulation as shown in Exhibit 13 below:

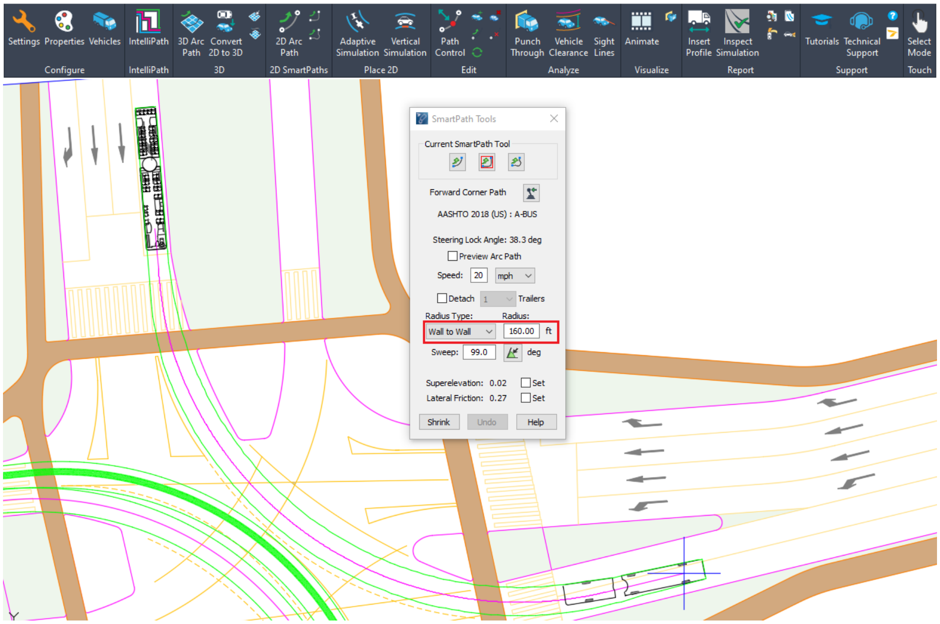


Exhibit 13. Set the simulation's Corner radius

Place the simulation as shown in exhibit 13.

15. Click the inspect Simulation button and in addition to the previous 2 parameters, select the Proximity option. In the dropdown list select the WB-40 layer/level:

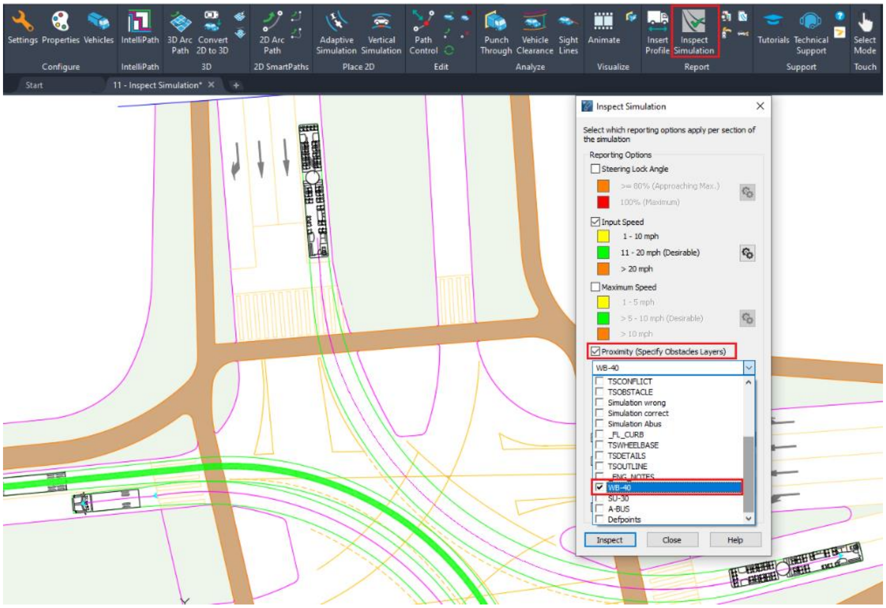


Exhibit 14. Inspect Simulation

16. Click the Configure Proximity settings button and set the proximity warnings to be red from 0 to 5 ft and yellow from 5 to 7ft and click ok

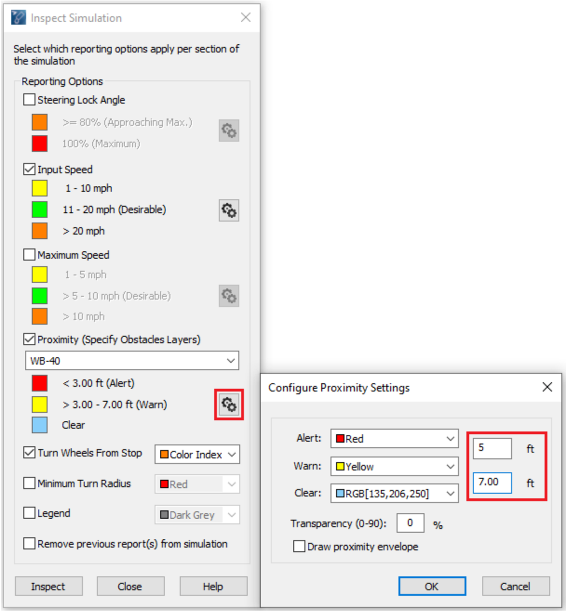


Exhibit 15. Inspect Simulation Proximity settings

17. Click Inspect and select the simulation created for the Articulated bus:

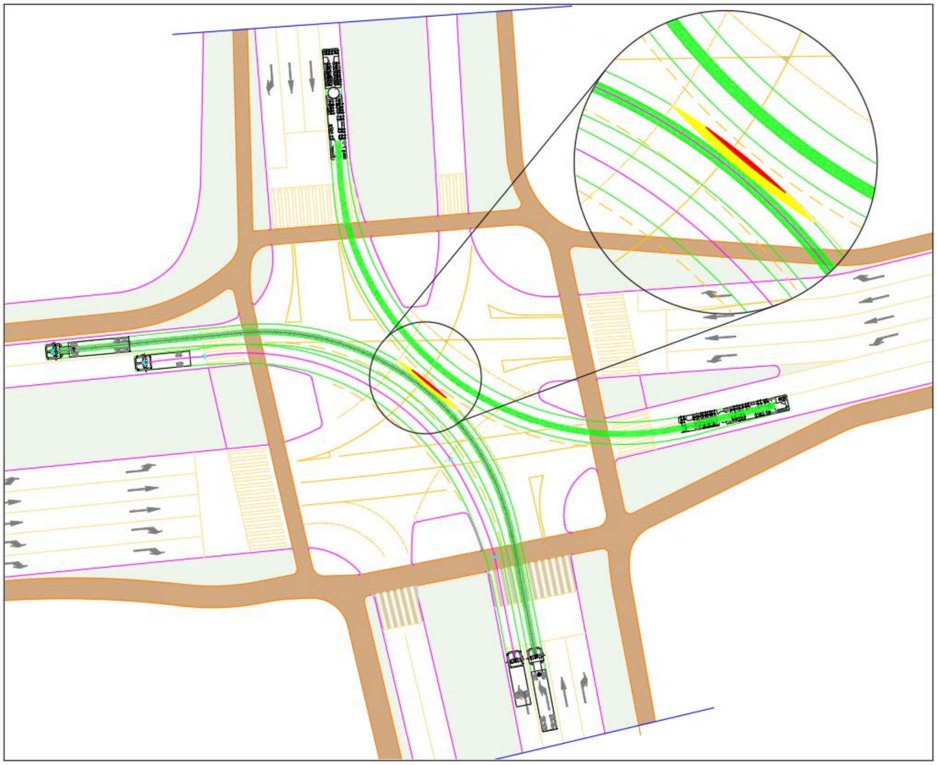


Exhibit 16. Inspect Simulation - Proximity report

Note how the report highlights a violation of the minimum required distance in red and proximity warnings in yellow.

Path Control for Corner Path simulations

The Path Control tool is intended to help you perform adjustments to the simulations. For example, in cases where the vehicle simulation passes too close to an object or goes slightly over a curb line; this tool will allow you to make the necessary corrections to one or multiple sections without having to recreate the entire simulation.

The small adjustments can be performed by adding or deleting nodes, or simply by dragging the nodes to control the desired position, offset for oversteering, radius, or even sweep angle in the case of a corner path simulation (or oversteer corner path simulation). For vehicles with rear steering capabilities, the Path Control also offers a special node for adjusting the path. Generally, nodes are displayed for every section of the simulation as well as at the start and endpoints.

NOTE:

For significant changes to the path trajectories, it is recommended that other editing tools such as Delete Last Section and Continue Simulation be used.

Adjusting a 2D Turning Simulation

The following scenario illustrates a **WB-40** from the **AASHTO2018 (US) library** performing a 2D simulation. The simulation is created by using a few combinations of the different SmartPath tools including the 2D Arc Path for the first portion and Corner Path to negotiate the turn. Note that the vehicle envelope of the simulation crosses over the nose median of the 7th Avenue. (Refer to the highlighted conflicts)

Since the conflicts could likely be avoided if the vehicle oversteered slightly, the Path Control tool can be applied to make the small adjustments. This approach is more suitable than having to redo the simulation.

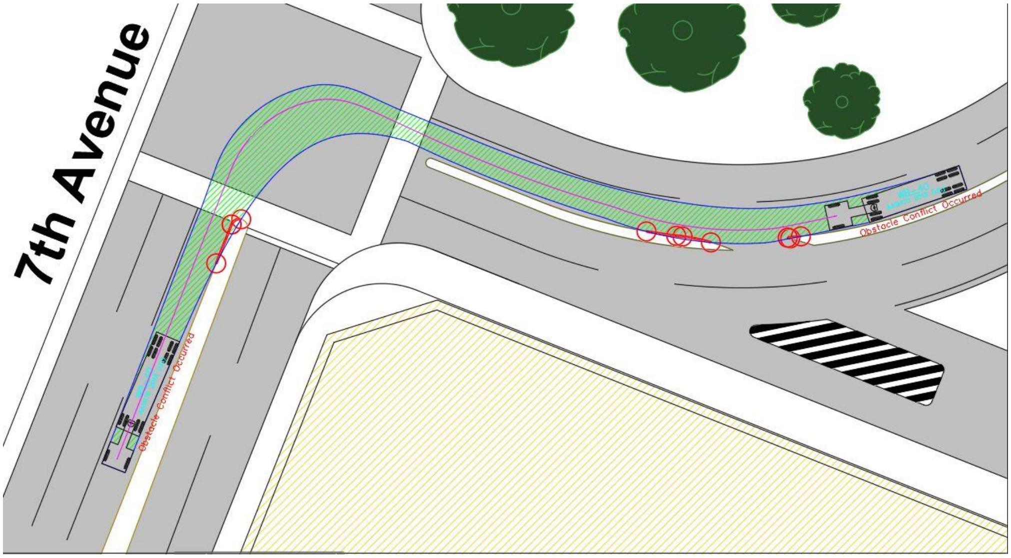


Exhibit 1 – *Turning Simulation*

▶ How to edit a 2D Simulation

1. Click  Path Control on the toolbar and pick the simulation you wish to edit, once it is selected, the different nodes associated with the simulation will be made available for controlling the path. See Exhibit 2.

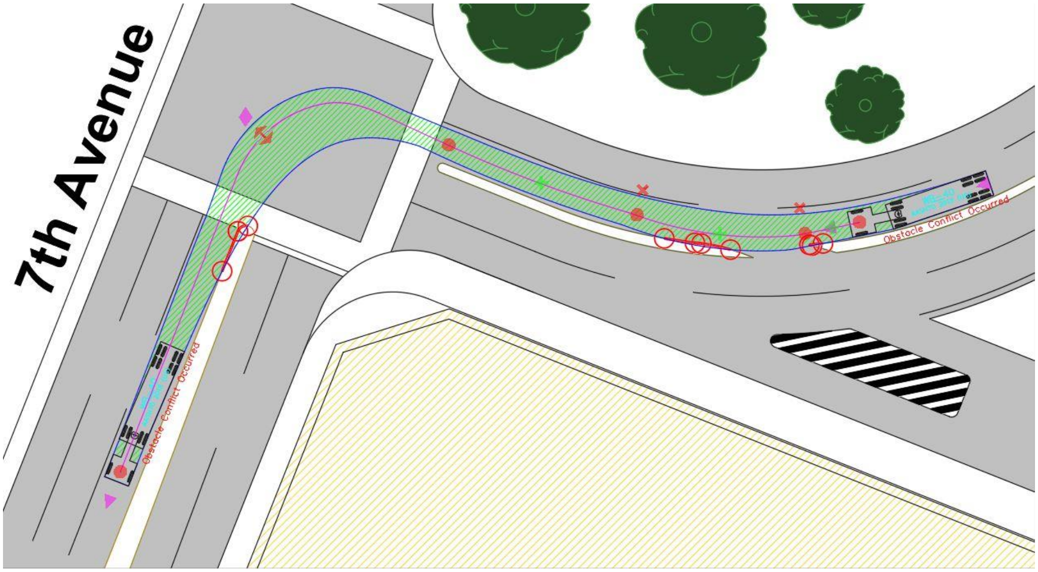


Exhibit 2 – *Path Control nodes*

2. Depending on the node, different editing capabilities will be available. The following table describes the various actions that can be performed depending on the node type.

Symbol	Tooltip	Description
	Move Node	Moves the selected node to a new location, adjusting the simulation accordingly.
	Add Node	Adds a node to the simulation.
	Delete Node	Deletes a node from the simulation.
	Adjust Start Angle	It allows you to adjust the start angle for the first vehicle position at the start of the simulation. This node is only available in the first vehicle position of 2D arc, corner, or oversteer corner path turn simulations and only for single part vehicles and the first part of multi-part vehicles. This node is not available for simulations generated with vehicles using rear steering.
	Adjust Articulating Angle	It allows you to adjust the articulating angle of the second or subsequent part of the first vehicle position at the start of the simulation. It requires a simulation that was generated with a multi-part vehicle. This node is not available for simulations generated with vehicles with rear steering.
	Adjust Radius	It allows you to adjust the radius of a corner in a corner or oversteer corner path simulation. Entry and exit tangents must be of sufficient length to support modifying the radius. In other words, if the radius needs to be increased, the length of the entry and/or exit tangent must first be increased. This node is not available for reverse corner or reverse oversteer corner path simulations that were generated with multi-part vehicles.
	Adjust Offsets	It allows you to adjust the entry and exit offsets of a corner in a corner or oversteer corner path simulation or section. Entry and exit tangents must be of sufficient length to support modifying entry and exit offsets. In other words, if the exit offset needs to be increased, the length of the exit tangent must first be increased. This node is not available for reverse corner or reverse oversteer corner path simulations that were generated with multi-part vehicles.
	Adjust Position and Angle	It allows you to adjust the exit tangent length and sweep angle of corner or oversteer corner path simulation.
	Adjust Rear Steering	It allows you to adjust the rear steered part's path in the simulation. It requires a simulation that was generated with a vehicle with rear steering. Corner and oversteer corner path simulations are not supported.

Table 1 – *Path Control Nodes' Actions*

3. Start adjusting the simulation by first dragging the node to control the articulating angle (angle between the tractor and trailer) (1), then adjust the start angle (2) and then move the position node (3) to adjust the starting position for the vehicle. See Exhibit 3. With this step, we have reached a better positioning of the vehicle along the lane as well as avoided conflicts with the median.

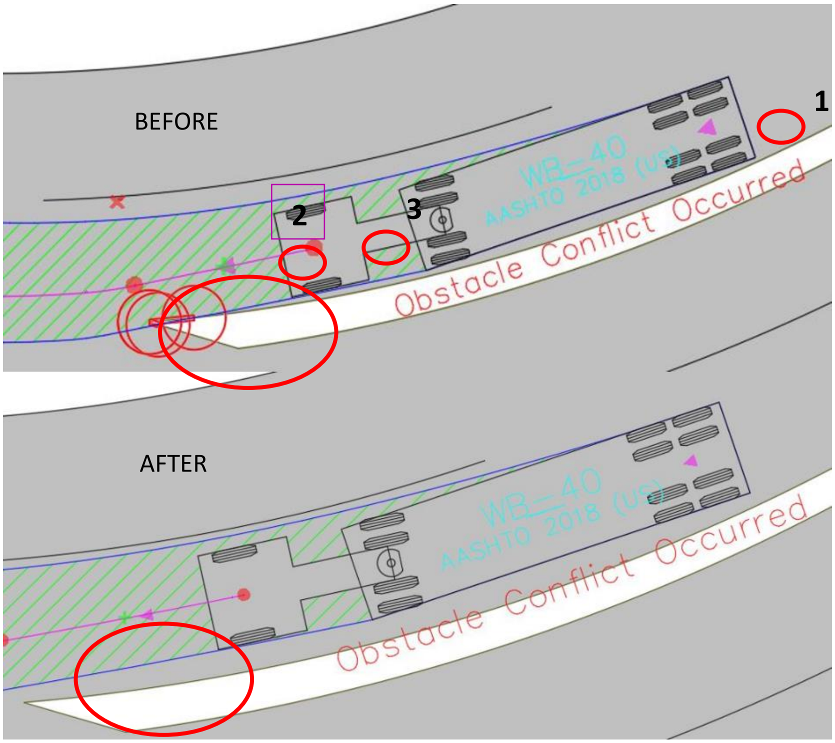


Exhibit 3 – *Adjusting Starting Position*

4. To maintain the vehicle within the left lane, add a new node, and drag it towards the median to situate the vehicle to stay within the lane. See Exhibit 4.

TRANSOFT SOLUTIONS

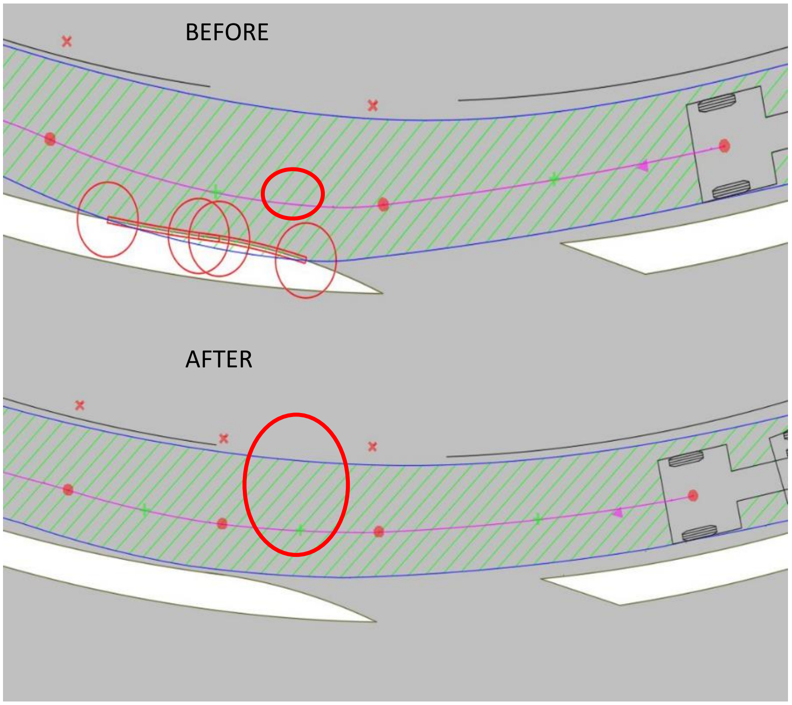


Exhibit 4 – *Adding a Node*

5. Finally, use the Adjust Offsets node to control the offsets for the Corner Path section of the turn to avoid the conflict reported at the nose median. By dragging the Adjust Offsets node, it is possible to create an oversteer corner turn from a corner path simulation. See exhibit 5.

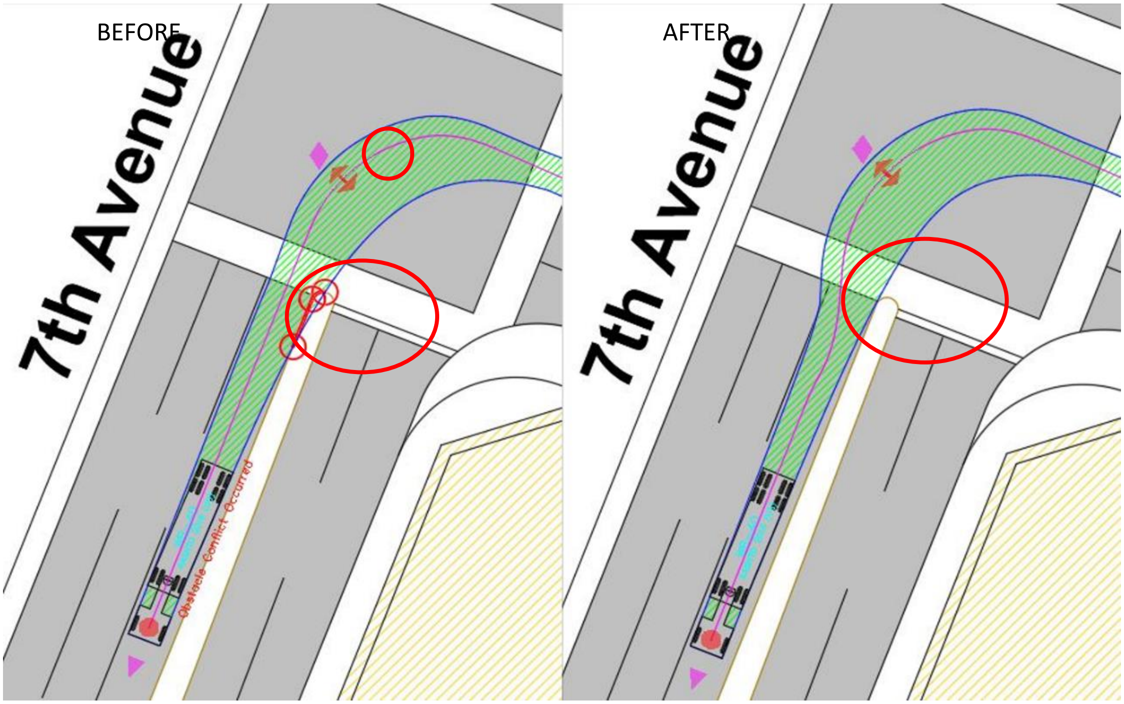


Exhibit 5 – *Adjusting Offsets*

6. The results of these adjustments using Path Control tools in the simulation are presented below. See exhibit 6.

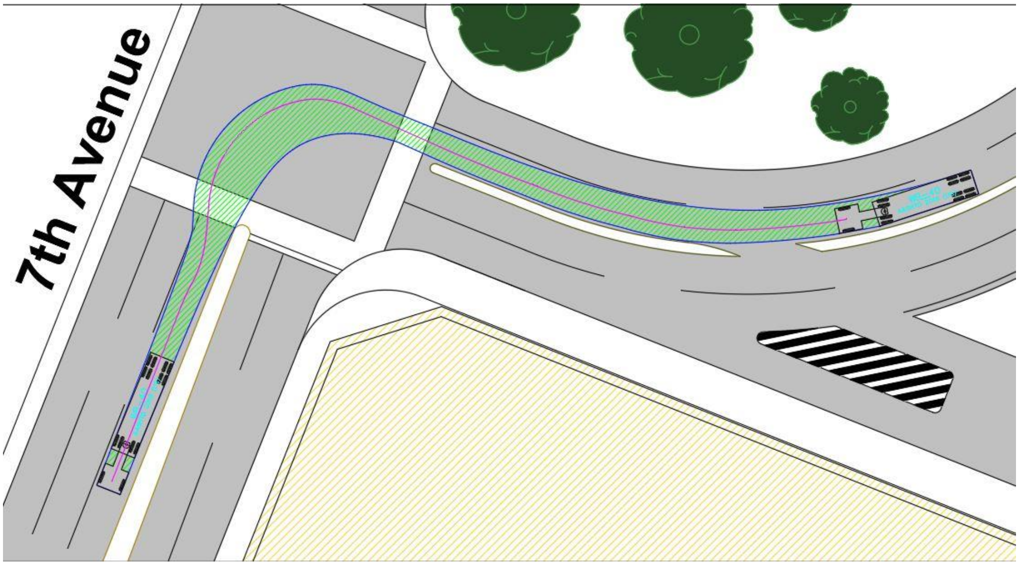


Exhibit 6 – *Final simulation*

How to Generate a Grade Template and Place a Vertical Simulation

The Generate Grade Template and Place Vertical Simulation tools will help assess ground clearance issues in 2D for the design Vehicle. Whether it is for evaluating the maximum uphill or downhill slopes based on the vehicles' ground clearances (i.e. at the front overhang, wheelbase, or rear overhang), or pinpointing potential conflicts along a vertical profile, these two tools can be applied depending on the information available.

The Generate Grade Template tool will compute the vehicle's maximum upward and downward grades achievable to a tolerance of 5 mm such that the vehicle's underside will not conflict with the terrain. The template is generated for the vehicle selected in the Select Current Vehicle dialog box.

The Vertical Simulation tool places a forward simulation in the profile view, using the current vehicle on selected vertical path elements. This is ideal for scenarios where the vertical profile of the ground has been defined (e.g. profile along a driveway with a ramp).

> **NOTE:**
>
> *These tools are meant for approximation purposes and to offer some general guideline as they do not consider the effects of a laden vehicle, suspension effects, or flat tires etc.*

Determining maximum slopes and evaluating vertical profiles

For the first part of this exercise, we will determine the maximum upward and downward grades for a design vehicle given its ground clearances. The second part of the exercise will evaluate the vehicle profile vs the vertical profile representing a tunnel section using the place vertical simulation tool to identify possible conflicts between the vehicle's underside and the vertical profile, as well as possible conflicts with overhead constraints.

▶ How to Place a Grade Template

The Grade Template displays the maximum up and down slopes achievable by the selected design vehicle. This tool can help at the early stages of a design when the designer is trying to determine the maximum slopes for example for entry and exit ramps to a parking facility or driveway.

This tool uses the varying vehicle ground clearances for the underside of the vehicle (e.g. wheelbase, and the front and rear overhangs) as defined in the Vehicles dialog to determine the maximum upward and downward slopes achievable. It would consider chamfering if it was applied in the vehicle definition. See Exhibit 1.

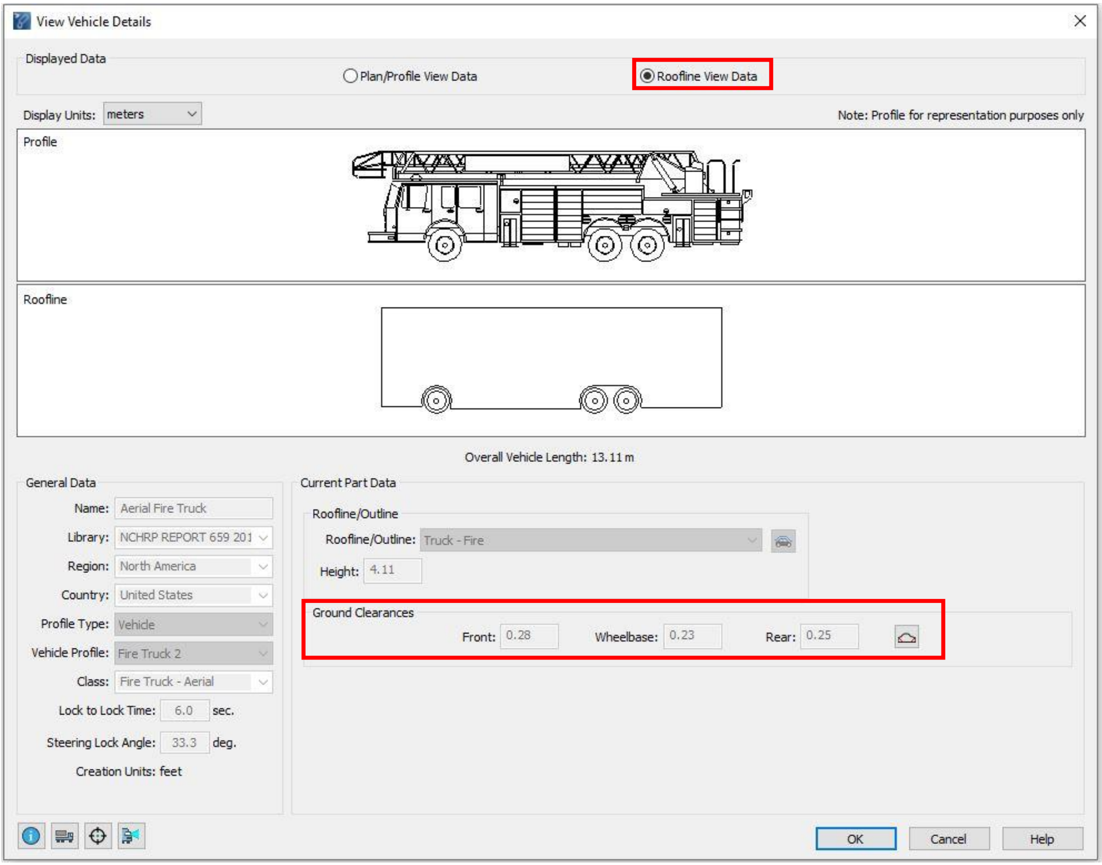
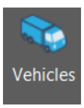


Exhibit 1 – *Vehicle Ground Clearances*

18. Click 

Vehicles and select the NCHRP Report 659 – Aerial Fire Truck. See Exhibit 2.

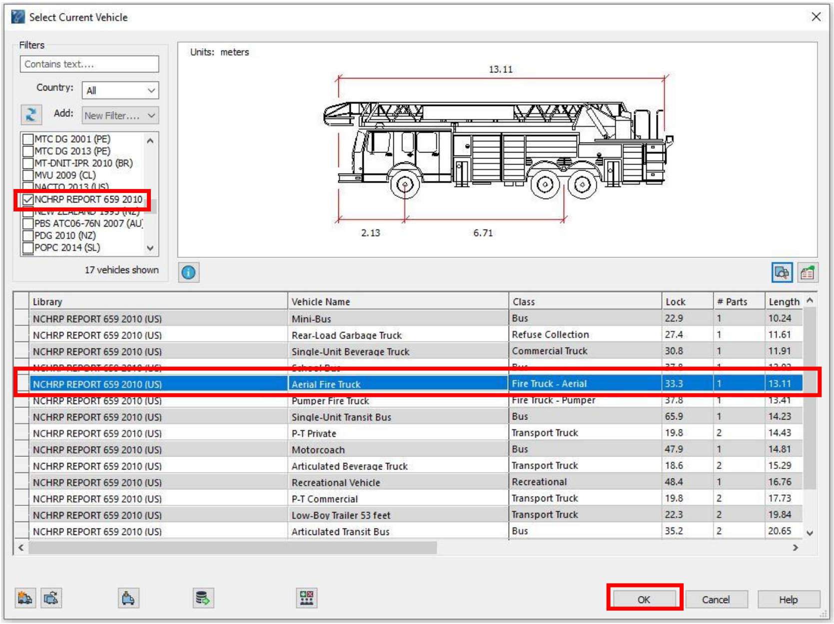


Exhibit 2 – *Vehicles Dialog Box*

19. Click in  **Generate Grade Template** in the Toolbar. In the Generate Grade Template dialog box, it is possible to indicate the starting grade either positive (uphill) or negative (downhill), and the tool will calculate the maximum upward and downward grades based on the starting grade and the vehicle's ground clearances.

Enter 3% in the starting grade and press TAB in your keyboard, see how the tool is calculating the maximum grades based on the corresponding starting grade defined. For the purpose of this example, enter 0% as the starting grade and press tab so that the tool updates the calculations.

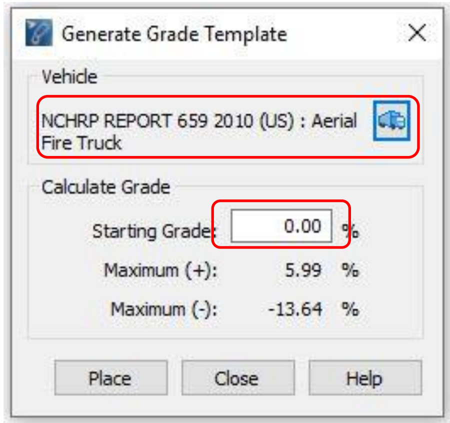


Exhibit 3 – *Generate Grade Template Dialog Box*

20. Pick a point in the drawing to place the template as indicated in Exhibits 4 and 5.

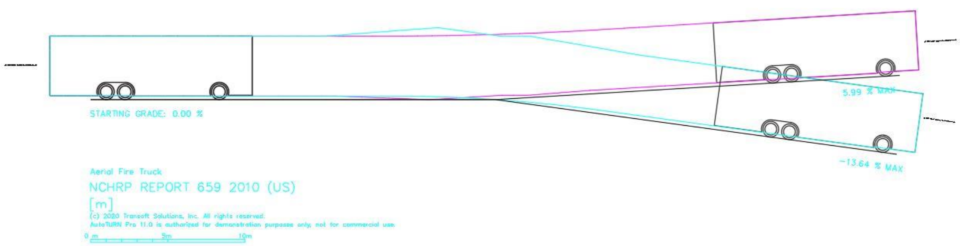


Exhibit 4 – *Grade Template of Recreational Vehicle – Starting Grade at 0%*

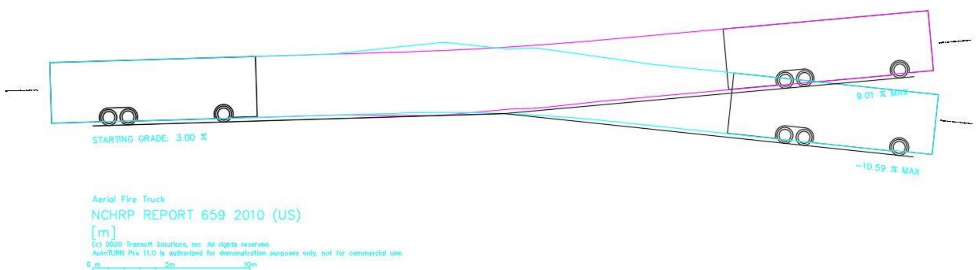


Exhibit 5 – *Grade Template of Recreational Vehicle – Starting Grade at 3%*

The Starting grade as well as the maximum upward and downward grades are displayed by the grade template.

NOTE:

The grade break is abrupt as it does not consider a vertical curve. If you have a vertical profile already defined, it is more appropriate to use the Vertical Simulation tool.

▶ How to Generate a Vertical Simulation

The place vertical simulation tool can be used to perform simplified vertical clearance analysis in 2D. For cases when the vertical profile is known, it can help assess possible conflicts between the vehicle and the vertical profile or overhead obstacles. Note: This is a simplified analysis as compared to a full 3D analysis because the vehicle is assumed to be on a 0% cross grade.

The vertical profile may comprise of connected lines and arcs as well as complex chains and polylines. It should not have any vertical exaggeration for performing vertical simulation analysis.

1. Open the **Vertical Simulation_Scenario** drawing. In the properties, make sure that in the Conflict Analysis Category, the Obstacles Layer is updated with the layer(s) the overhead conflicts are on.. See Exhibit 1.

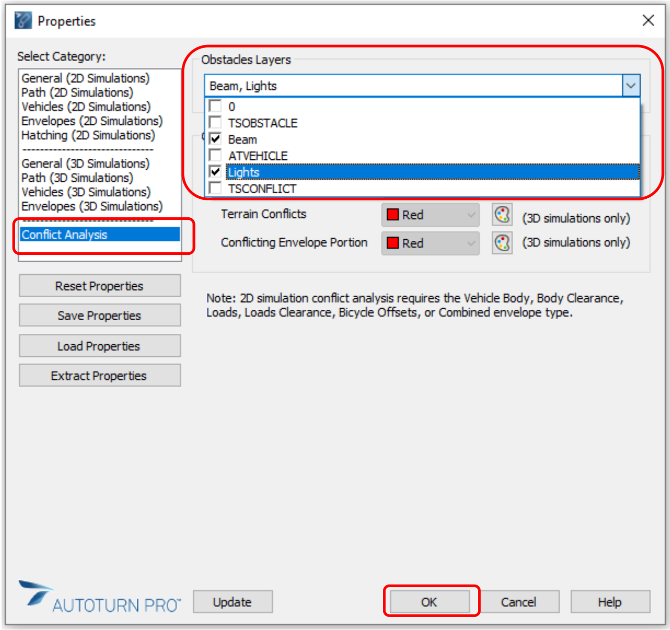
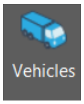


Exhibit 1 – *Setup properties for conflict analysis.*

2. Click 

Vehicles and select the NCHRP Report 659 – Recreational vehicle. See Exhibit 2

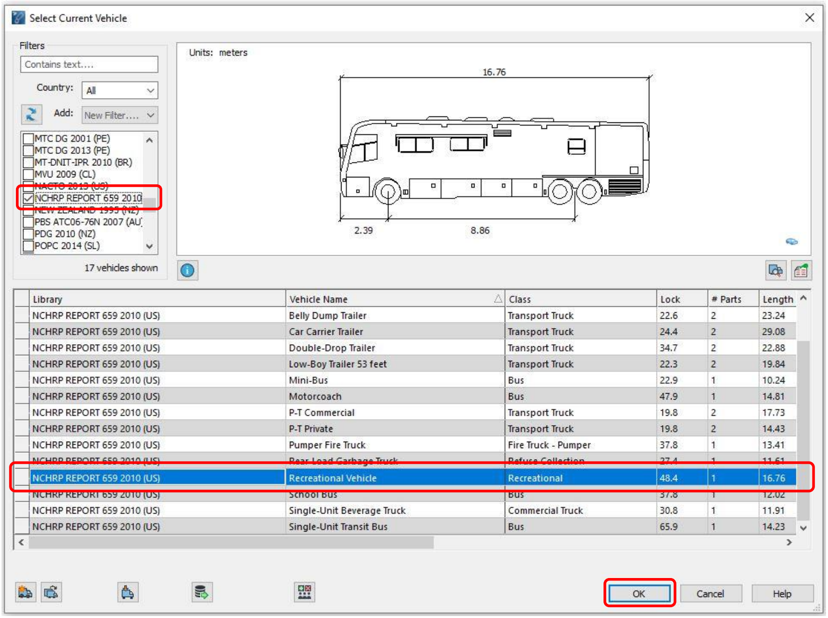
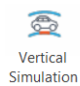


Exhibit 2 – *Select the design vehicle.*

3. Click 

Vertical Simulation in the toolbar and select the tunnel's vertical profile of the ground to generate the simulation.

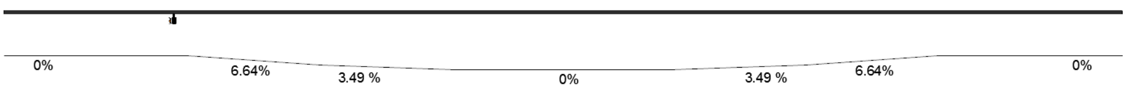


Exhibit 2 – *Tunnel Grade*

4. The Vertical Simulation is generated first, and the dialog box will allow you to control the different envelopes and clearances that will be displayed. Under the Vertical Simulation Draw Options, select Body Envelope and clear the Top and Bottom Body Clearance checkboxes. See Exhibit 3.

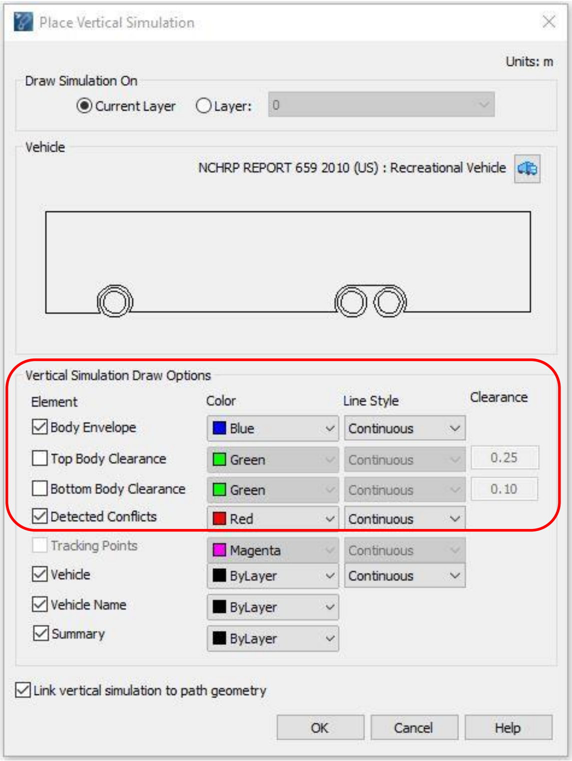


Exhibit 3 – *Place Vertical Simulation Dialog Box*

NOTE:

When using complex chains or polylines, ensure that "Link vertical simulation to path geometry" is selected. This enables the simulation to be adjusted dynamically when the ground profile is edited in case corrections are required.

5. Based on the simulation, it can be observed that the design vehicle can navigate without bottoming out or hitting obstacles, as no conflicts have been reported. See Exhibit 4.

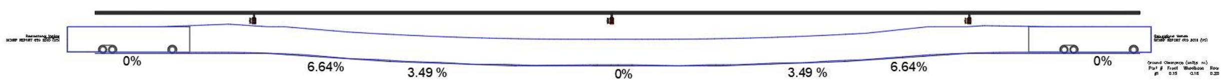


Exhibit 4 – *Vertical Simulation – Body Envelope*

6. As the next step for our analysis, we would like to verify there is enough safety buffer to account for taller vehicles (e.g. vehicles carrying luggage on top). Under Vertical Simulation Draw Options, select the Top Body Clearance checkbox and type a value of 0.3 m (1ft). See Exhibit 5.

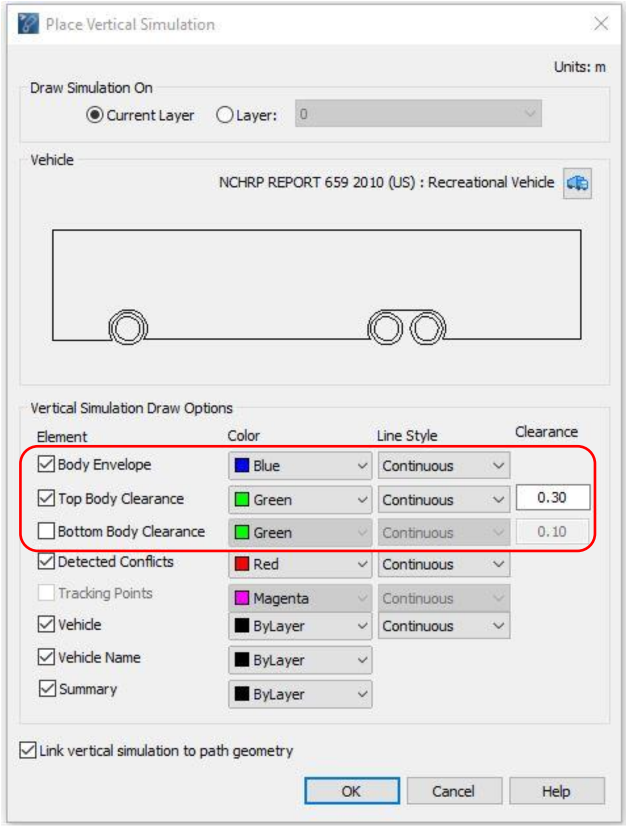


Exhibit 5 – *Place Vertical Simulation Dialog Box*

7. Notice how the tool now highlights a conflict with the overhead sign in red, and the text message indicating that a conflict has occurred is displayed at the first and last vehicle positions. See Exhibit 6.

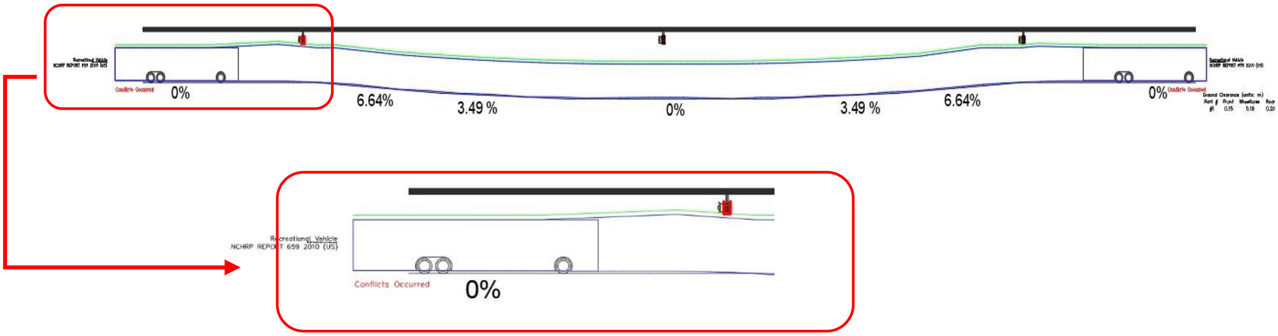


Exhibit 6 – *Vertical Simulation – Top body clearance*

Using the Place Vehicle or the Run Animation tools in AutoTURN, it is possible to place or move the vehicle along the simulation, allowing you to analyze the vehicle at any step of the simulation.

▶ Place Vehicle

The Place vehicle tool places a representation of the current vehicle anywhere along a selected path or, in the case of a selected turn simulation, the vehicle used in the turn simulation. If there is a realistic vehicle drawing for the vehicle, it can be placed in the drawing by selecting the Realistic checkbox in the Vehicle Draw Options section of the Vehicles (2D Simulations) category of the Properties dialog box.

1. Click Place Vehicle on the AutoTURN Ribbon pick the vehicle in the 2D Vertical Simulation and move the pointer along the path to drag the vehicle. Use a left-click to place the vehicle at any desired position along the simulation and right-click to end the place vehicle tool. See Exhibit 1.

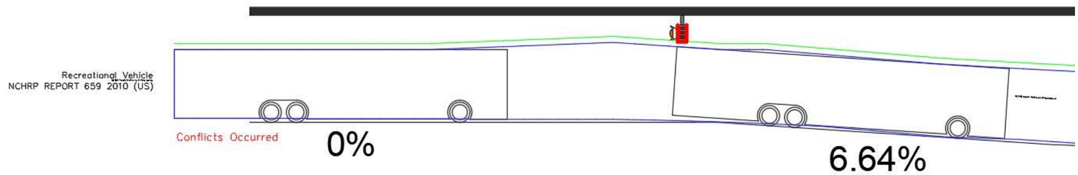


Exhibit 1 – *Place vehicle*

How to Insert Vehicle Profile

The **Insert Profile** tool inserts a dimensioned view of the current or the active vehicle from the active simulation.

1. To insert the profile of the vehicles used from the Burger Joint exercise, navigate to the simulation created, click  **Insert Profile** on the toolbar.
2. Select the AASHTO Passenger car simulation created.
3. Place the vehicle profile as illustrated in Exhibit 1.

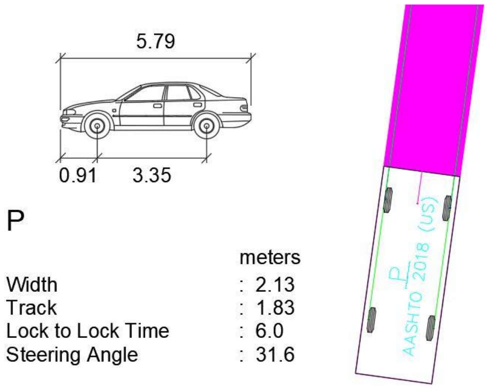


Exhibit 1 – *Passenger Car Profile*

4. Click  **Insert Profile** on the toolbar.
5. Select the WB-40 simulation, and place the WB-40 profile next to the turn simulation. See Exhibit 2.

TRANSOFT SOLUTIONS

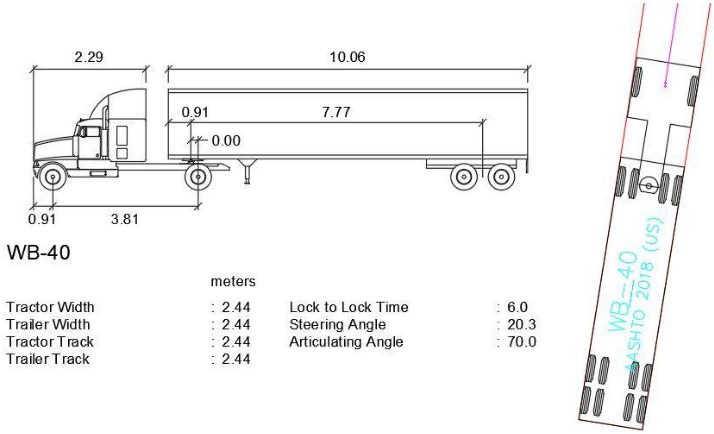


Exhibit 2 – *WB-40 Profile*

Note: The profile inserted is cosmetic and not to scale. It can only be used for representation purposes.

You can choose to insert the profile in the current active layer/level or a different layer/level in the drawing.

How to Generate Electronic Turning Templates

The **Generate Template** tool generates a turning template of the current vehicle complete with all relevant vehicle dimensions and turning characteristics. The template is generated based on the vehicle set in the **Select Current Vehicle** dialog box and settings specified in the **Properties** dialog box such as colors and line styles, fills, envelopes, etc. Templates can be generated for any vehicle.

NOTE:

Turning Templates can be used for presentations or used as a reference for the creation of geometry.

In this next exercise, we are going to include the turning template of the WB-40 for the Burger Joint client presentation.

1. Select the WB-40 design vehicle from the AASHTO – 2018 (US) library.
2. Click **Generate Template** on the toolbar.
3. The **Generate Template** dialog box will appear. See Exhibit 1.

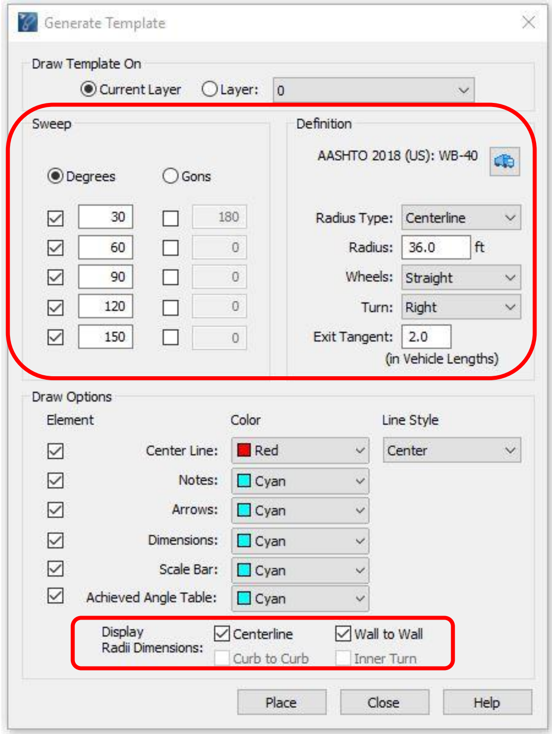


Exhibit 1 – *Generate Template Display Options*

Note The curb to curb and inner turn options are grayed out in the window because the tire envelopes in the ***Properties – Envelopes (2D simulation)*** *category are not selected.*

4. Under the **Sweep** section, select **Degrees** or **Gons** as the desired angle units. You can check the boxes for a specified sweep angle that you want to generate.
5. Under **Definition**, you can select the desired radius type from the **Radius Type** list. You can also select a value for the radius from the **Radius** list. If the current vehicle is unable to negotiate a specific sweep turn, a message is displayed stating the sweep turn that could not be generated in the template drawing. You can also select the desired value for the exit tangent from the **Exit Tangent** list.

6. Under **Draw Options**, you can select the checkboxes for each element that will be drawn and radii dimensions that will be displayed.

TRANSOFT SOLUTIONS

7. Click **Place** to place the template. See Exhibit 2.

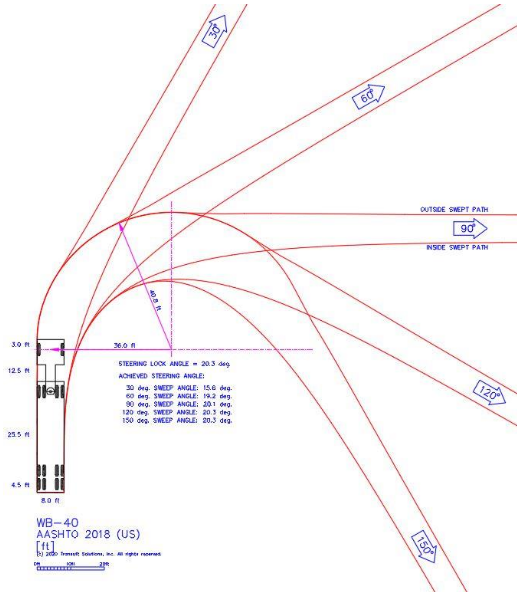


Exhibit 2 – AASHTO 2018 (US) *WB-40 Turning Template*

Generate Template vs. SmartPath Tools

Standard templates published by most jurisdictions consist of simple tangent and arc paths. Rate of wheel turn is not usually considered. The **SmartPath Tools** such as **Generate Arc Path** and **Generate Corner Path** takes into account the rate at which the wheel can be turned and incorporate spiral transition sections into the path. For this reason, the **SmartPath Tools** cannot be used to create standard turning templates.

How to View a Simulation Report

The **Generate Report tool** displays a graphical representation of the vehicle's steering angle (and articulating angles for multi-part vehicles) over the length of the picked turn simulation. Simply click on the **Generate Report** button on the toolbar. The graphical report can be placed in the drawing to accompany the simulation and a summary report can be generated and saved to a file.

1. To place a graphical report of the WB-40 from the Burger Joint exercise, navigate to the simulation created, click **Generate Report** on the toolbar.
2. Pick the WB-40 simulation and in the **Generate Report** dialog box select Turning Report, and click on the Create button. See Exhibit 1.

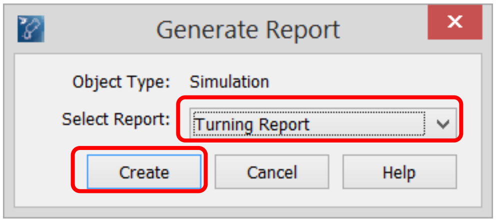


Exhibit 1 – *Select Turning Report*

3. Select the **Show Formatting Options** checkbox. In the format options, you can select the information that you would like to have reported. See Exhibit 2.

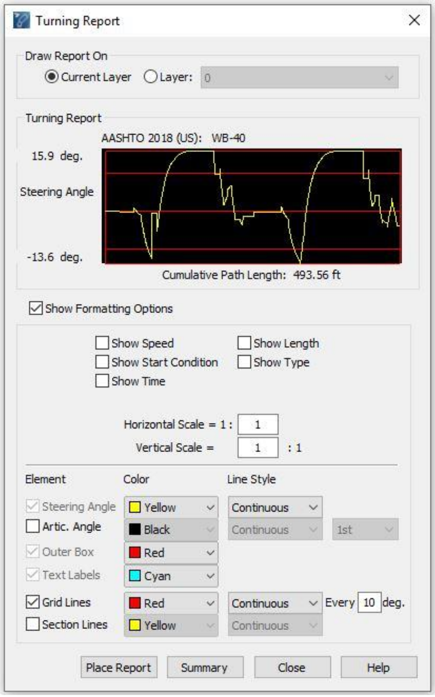


Exhibit 2 – *Turning Report Dialog Box – Formatting Options*

Format Options

Show Speed
Select this option to display the Vehicle speed for each section of the simulation when the report is placed in the drawing.

Show Length
Select this option to display the length of each section when the report is placed in the drawing.

Show Start Condition
Select this option to display the start condition for each section when the report is placed. The start condition is either "in motion" if the Turn Wheels From Stop checkbox was not selected or "stopped" if the **Turn Wheels From Stop** checkbox was selected in the **SmartPath Tools** dialog box when the section was generated.

Show Type
Select this option to display the maneuver type for each section when the report is placed.

Show Time
Select this option to display the elapsed time for each section when the report is placed.

Horizontal Scale
This option allows the report to be stretched out or shortened to accommodate report labels when the report is placed in the drawing.

Steering Angle
Sets the color and line style for the line representing the steering angle.

Articulating Angle
Sets the color and line style for the lines representing the articulating angles of each trailer of the multi-part vehicle.

Outer Box
Sets the color for the outermost line of the report.

Text Labels
Sets the color for the text labels that appear below the report when placed in the drawing.

Origin Circle
Select this option to display and set the color for the circle designating zero (0) degrees.

Grid Lines
Select this option to display and set the color, line style, and spacing interval of the horizontal grid lines.

Section Lines
Select this option to display and set the color and line style for the vertical lines designating change of sections in the simulation.

4. Click **Place Report** to place the turning report below the WB-40 profile. See Exhibit 3.

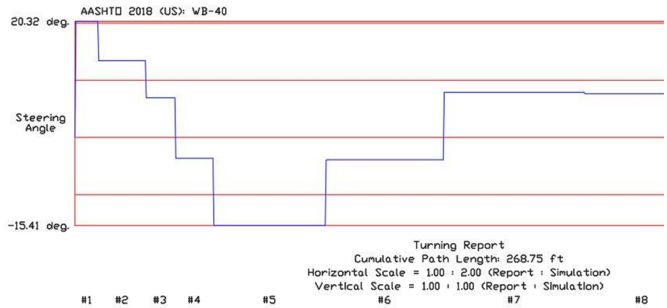


Exhibit 3 – *Graphical Turning Report*

Alternatively, you can create a detailed text summary of the simulation. To create the text summary of the simulation, repeat the previous steps. Instead of **Place Report**, click **Summary** to generate the text file. See Exhibit 4.

```
File  Edit  Format  View  Help
AutoTURN Pro Turning Report

Date:                           2020-Mar-24-17-42-00
Vehicle:                        WB-40
Library:                        AASHTO 2018 (US)
Analysis Line Length:           268.75 feet
Total time:                     0 minute(s), 25 seconds

--------------------------------------------------
Section:                        1
--------------------------------------------------
Type:                           Arc
Orientation:                    Forward
Trailer(s) Detached:            No
Length:                         7.86 feet
Start:                          -8013.83, -3291.54
End:                            -8006.79, -3288.07
Start angle:                    380.0 deg.
End angle:                      392.5 deg.
Sweep angle:                    12.5 deg.
Radius:                         36.00 feet

Start condition:                Stopped
Speed (Constant):               6 mph
Elapsed time:                   0 minute(s), 0 seconds
Superelevation:                 Not considered
Lateral Friction:               Not considered

Vehicle angles (deg.)
         Start:     0.0, 0.0
           End:     20.3, 10.7
           Max:     20.3, 10.7
```

Exhibit 4 – *Text Summary*

Analyzing Sight Lines

Sightline analysis is one method to evaluate roadway safety. It can help analyze or establish the design speed based on the driver's ability to see, react, or stop. It can identify potential roadway obstructions whether it is from the vertical curves or roadside features (i.e. terrain, embankment, median barrier, etc.)

The Analyze **Sightlines** tool is designed for evaluating sightlines and stopping sight distances for horizontal alignments, vertical alignments, or the 3D model of the road corridor based on the driver's eye position and the object's height to help determine terrain obstructions, or determine areas that have to be clear of obstacles. While this analysis is typically performed on vertical profiles and horizontal alignments, it can be applied to a 2D or 3D simulation. The drive path will be evaluated.
C
Information about the stopping sight distances for different design guidelines for reference can be found in the Appendix section of the help file:

Country	Design or Operating Speed (km/h)													Driver Eye Height (m)		
	20	30	40	50	60	70	80	90	100	110	120	130	140	Passenger Car	Truck	Object Height
Australia	Stopping Sight Distance (m)															
(Normal design)							115	140	170	210	250	300				
(Normal design)				45	65	85	105							1.15	1.80	0.20
(Restricted situations)				40	55	70										
Austria			35	50	70	90	120		185		275		380	1.00		0.00 - 0.19
Britain				70	90	120			215		295			1.05		0.26
Canada			45	65	85	110	140	170	200	220	240			1.05		0.38
France		25	35	50	65	85	105	130	160					1.00		0.35
Germany					65	85	110	140	170	210	255			1.00	2.50	0.00 - 0.45
Greece					65	85	110	140	170	205	245			1.00		0.15 - 0.60
South Africa			50	65	80	95	115	135	155	180	210			1.05	1.80	
Sweden		35		70		165				195				1.10		0.20
Switzerland			35	50		95	120	150	195	230	280			1.00	2.50	0.15

Source: Transportation Research Board of the International Academies - Transportation Research Circular, Issue E-C003, 1998, p. 32:1-23.

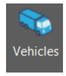
	Design or Operating Speed (km/h)														Driver Eye Height (m)	
United States Metric	20	30	40	50	60	70	80	90	100	110	120	130			Passenger Car	Object Height
	Stopping Sight Distance Design (m)															
	20	35	50	65	85	105	130	160	185	220	250	285			1.08	0.6
	Design or Operating Speed (mph)														Driver Eye Height (ft)	
United States Imperial	15	20	25	30	35	40	45	50	55	60	65	70	75	80	Passenger Car	Object Height
	Stopping Sight Distance Design (ft)															
	80	115	155	200	250	305	360	425	495	570	645	730	820	910	3.5	2

Source: American Association of State Highway and Transportation Officials - A Policy on Geometric Design of Highways and Streets 2004. Exhibit 3-1. Stopping Sight Distance, pp. 112, 113.

Performing sightline analysis on Alignments and Simulations

The first example will illustrate the way to perform the analysis using a vertical simulation after the simulation was checked on the vertical profile.

▶ Analyze Sight Lines – Vertical Profile

21. Open the drawing file, **2D Sight Lines_Scenario.dwg**. Click 

Vehicles and select the AASHTO 2018 (US) SU – 30 Vehicle and click OK. See exhibit 1.

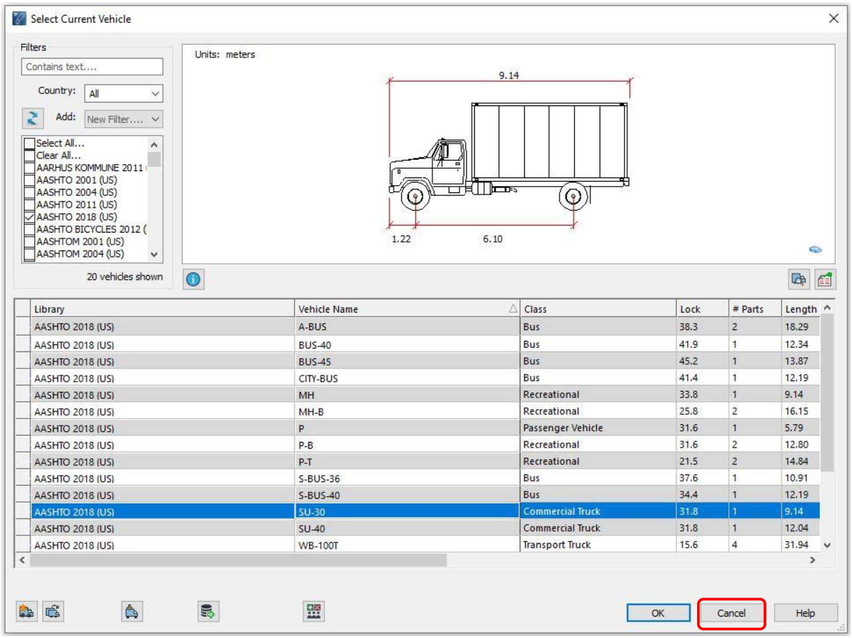


Exhibit 1 – *Vehicles Dialog Box – AASTHO 2018 (US) SU - 30*

22. Click Place Vertical Simulation 

on the toolbar and then click to select the Vertical Profile. In the Place Vertical Simulation dialog box, select only the Vehicle option and clear all the other display options as the focus of this exercise is not on ground clearance checking. See Exhibit 2.

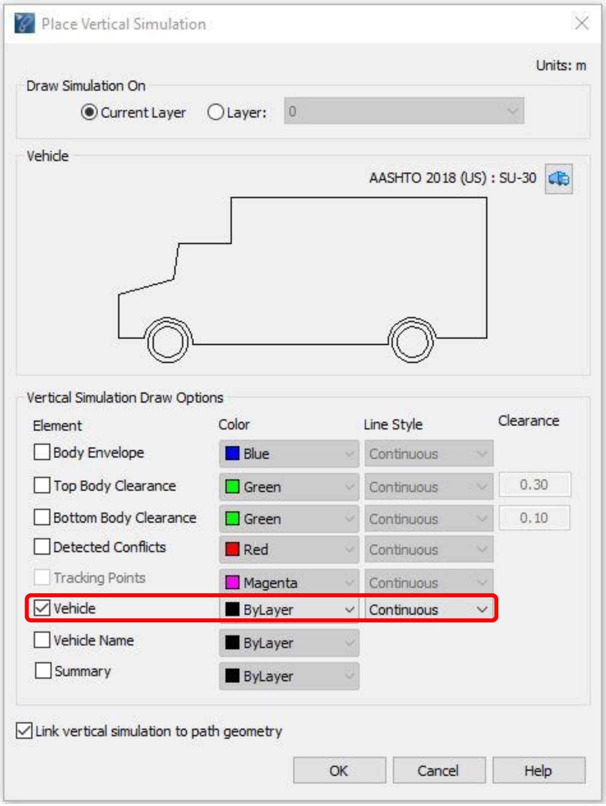


Exhibit 2 – *Generate Grade Template Dialog Box*

23. Click OK to place the simulation. See Exhibit 3.

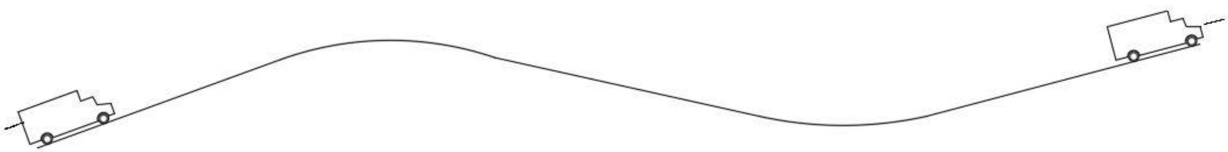
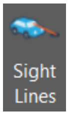


Exhibit 3 – *Define 2D Loads Dialog Box*

NOTE:

It is not necessary to apply Analyze Sight Lines on a Vertical Simulation. It can work directly on the vertical profile by selecting the geometry (esp. if checking ground clearances is not important). If the vertical profile is exaggerated, this needs to be specified in the dialog (Refer to Exhibit 5).

24. Click 

Analyze Sight Lines on the toolbar. In the Analyze Sight Lines dialog box, select the Vertical Profile option. See Exhibit 4.

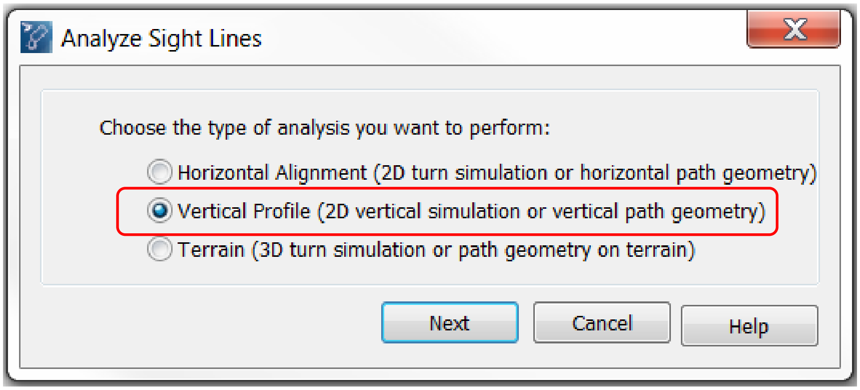


Exhibit 4 – *Analyze Sight Line dialog box.*

25. Pick the vehicle in the 2D Vertical Simulation. In the Analyze Sight Lines dialog box, enter the corresponding values for the Driver's eye height, the object height, stopping sight distance, and the interval distance as depicted in Exhibit 5.

You can enter different values for the stopping sight distance and see how the tool updates the drawing dynamically. For this example, use a stopping sight distance of 50 meters.

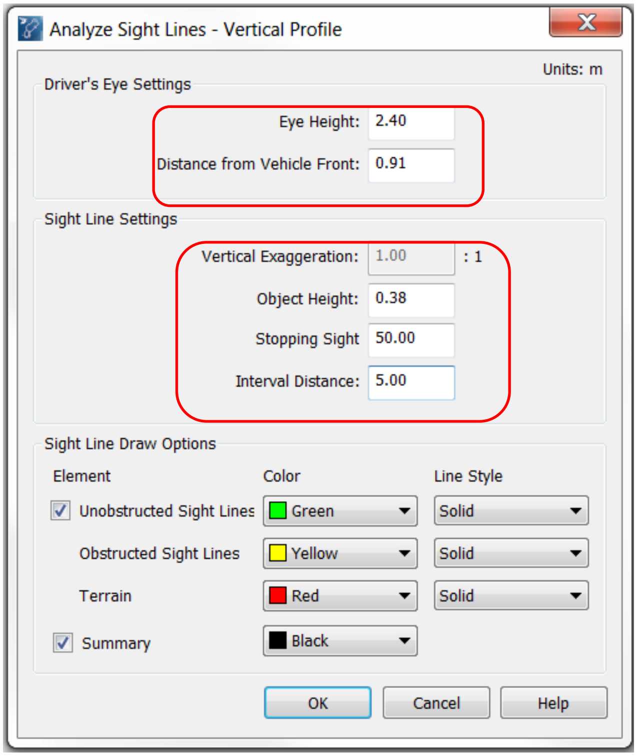


Exhibit 5 – *Analyze Sight Line – Vertical profile dialog box*

26. Click OK to place the sightlines in the drawing. See Exhibit 6.

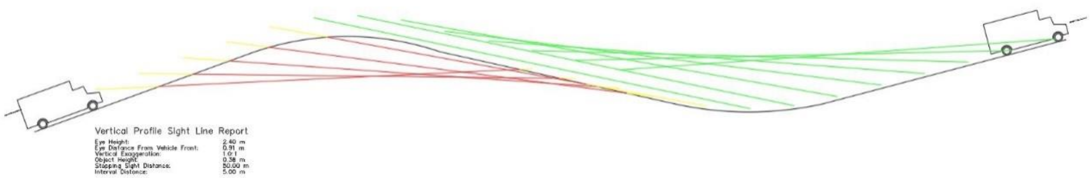
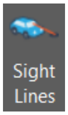


Exhibit 6 – *Profile View with Vertical Simulation and Sight Lines*

Note that the obstructed sightlines are displayed in Yellow and Red (e.g. yellow depicts the portion that above the vertical profile while the red shows the portion below the profile).

▶ Analyze Sight Lines – Horizontal Alignment

Using Analyze Sight Line on a horizontal alignment will help evaluate the horizontal sight line offset. This analysis will help identify the needed space beyond the roadway at the curve sections.

1. Click 

Analyze Sight Line in the toolbar. In the Analyze Sight Lines dialog box, select the Horizontal Alignment option, and click Next. See Exhibit 1.

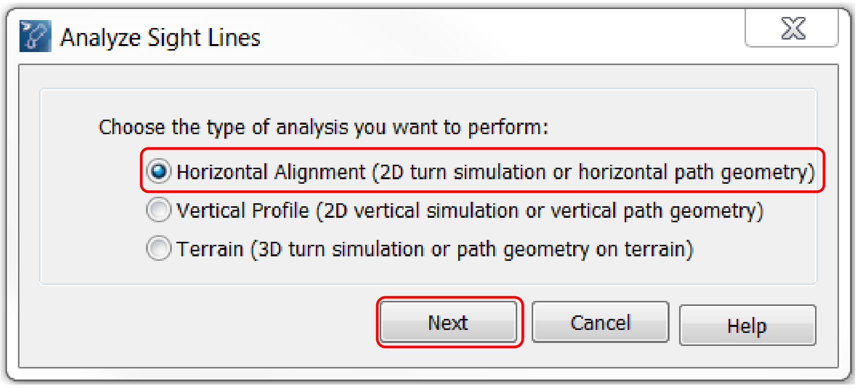


Exhibit 1 – *Analyze Sight Line dialog box.*

2. Pick the horizontal path geometry (i.e. the magenta line represents the centerline of a lane). See Exhibit 2.

Exhibit 2 – *Horizontal Path Geometry*

3. In the Analyze Sight Lines - Horizontal Alignment dialog box, under Sight Line Settings, specify a Stopping Sight Distance of 80 meters and an Interval Distance of 10 meters.

Note that while the vertical profile allowed you to set the driver's eye height and the object height location, working with horizontal alignments and 2D simulations provides you with a different set of options, in accordance with the scenario and analysis performed.

Click OK to finish the Sight Lines Analysis. See Exhibit 3.

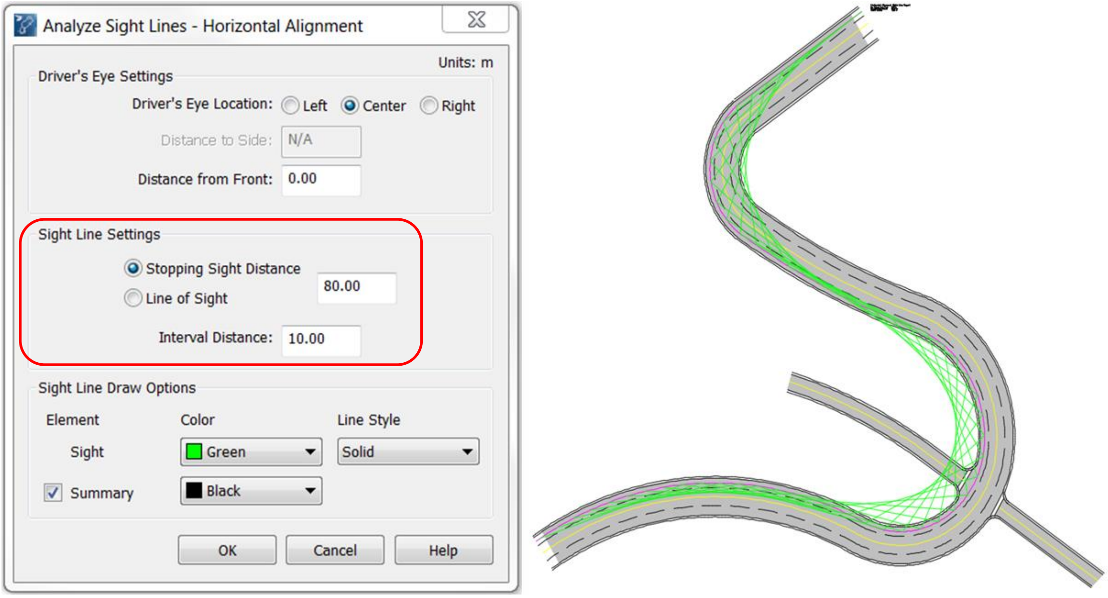
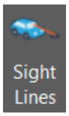


Exhibit 3 – *Sight Lines – Horizontal Alignment*

▶ Analyze Sight Lines – 3D Alignment (AutoTURN Pro Only)

Sightline analysis can also be performed on a roadway corridor were the 3D terrain model is available. This example will show you how to perform sightline analysis and stopping sight distance using a 3D simulation to determine the sightlines obstructed by the terrain. This analysis will also work directly on an Autodesk Civil 3D Alignment.

1. Click 

Analyze Sight Line from the toolbar. In the Analyze Sight Lines dialog box, select the Terrain option, and click Next. See Exhibit 1.

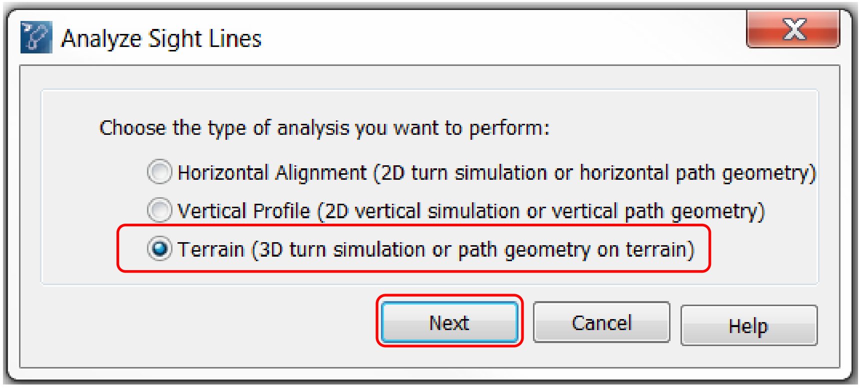


Exhibit 1 – *Analyze Sight Lines dialog box*

2. Pick the 3D Simulation (created using AutoTURN Pro). See Exhibit 2.

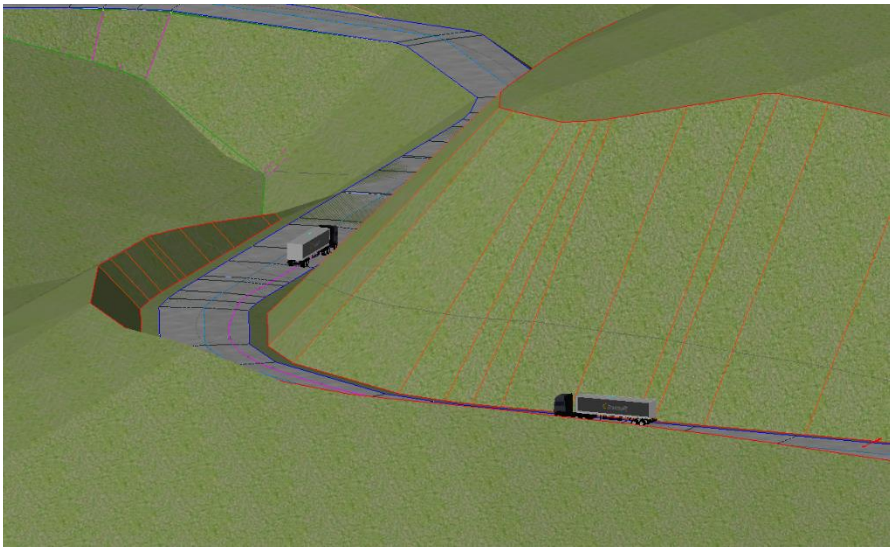


Exhibit 2 – *3D Simulation*

3. In the Analyze Sight Lines – Terrain dialog box, under Sight Line Settings, specify the values for the different parameters as indicated in Exhibit 3, and click OK to place the corresponding stopping sight distance. Similar to the previous 2 examples, the stopping sight distance or the line of sight in the drawing are updated dynamically as you enter new values in the dialog. The colors for displaying the sightlines can also be controlled and changed in the dialog.

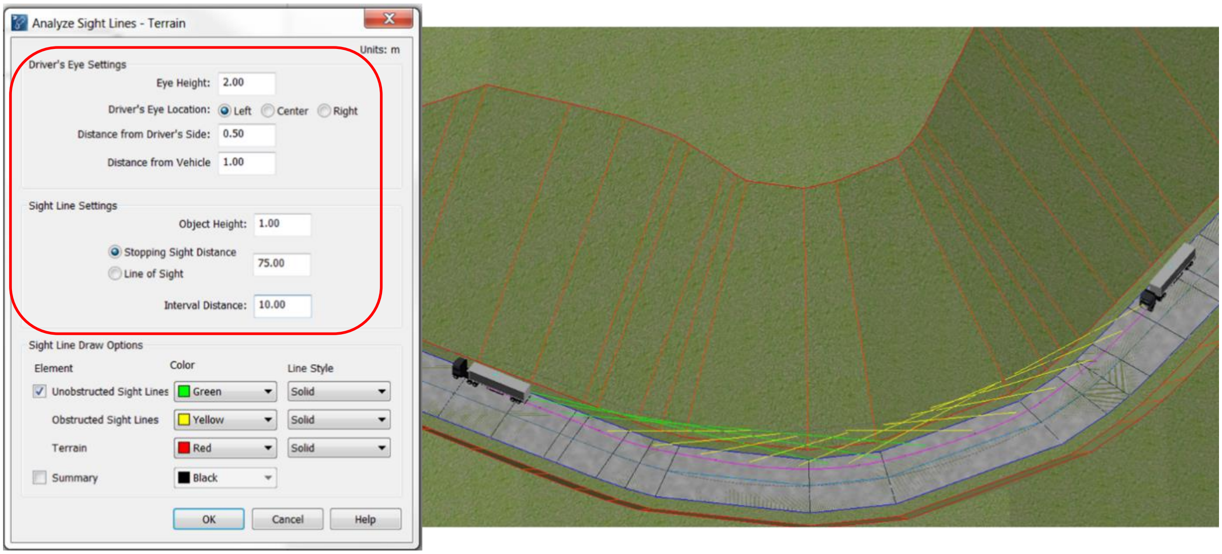


Exhibit 3 – *Sight Lines on terrain*

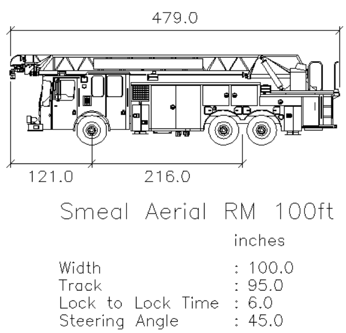
Vehicle Creation - Customization

AutoTURN can be used to create almost any type of custom vehicle for the library. In AutoTURN, there are four methods for creating custom vehicles. They are:

- ☐ Copy and customize a standard vehicle
- ☐ Create a new vehicle from predefined vehicle types
- ☐ Create and save new vehicle types
- ☐ Create realistic custom vehicle types

▶ How to Copy and Customize a Standard Vehicle

The Transoftville Fire Department is planning to purchase new fire trucks to replace their aging fleet. They have decided to purchase the Smeal 100’ Rear Mount Aerial truck to service Fire Hall #8.



Smeal 100’ Rear Mount Aerial

The objective of this exercise is to create a custom Smeal 100’ Rear Mount Aerial truck and test if the vehicle can maneuver into the Fire Hall #8 facility for servicing.

1. Click  **Vehicles** on the toolbar.
2. Select the NCHRP REPORT 659 2010 (US) library, select the Aerial Firetruck, and click the **Copy Vehicle** button. See Exhibit 1.

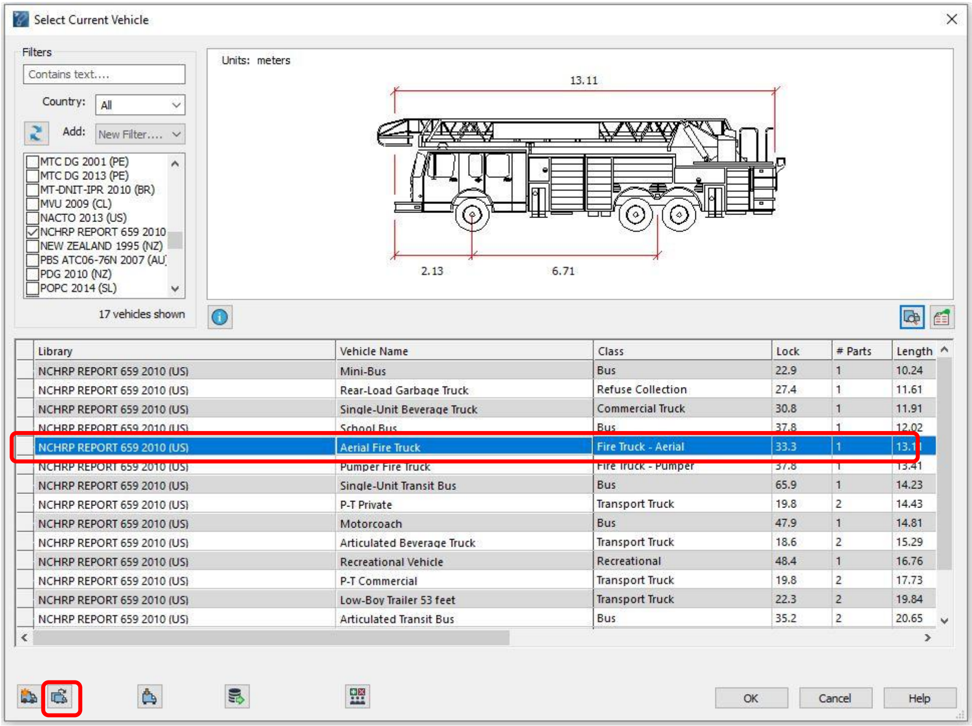


Exhibit 1 – *Select Current Vehicle Dialog Box*

3. The **Copy Vehicle** dialog box will appear. In the **General** tab, change the creation units to inches. Enter all of the required General dimensions as illustrated in Exhibit 2.

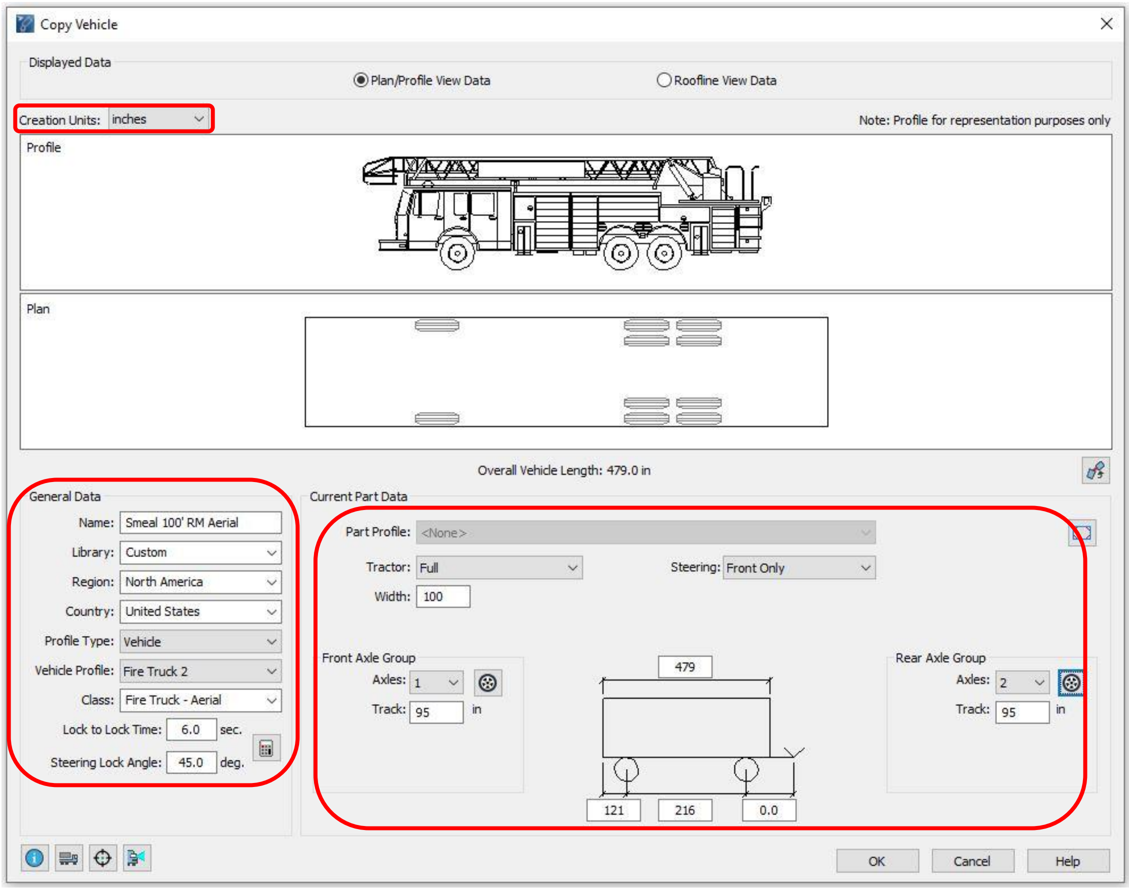


Exhibit 2 – *Copy Vehicle Dialog Box – Smeal 100' Rear Mount Aerial General Dimensions*

4. In cases where the vehicle corners are rounded rather than squared, you can use this option to modify it. Click the **Corners** button. See Exhibit 3.

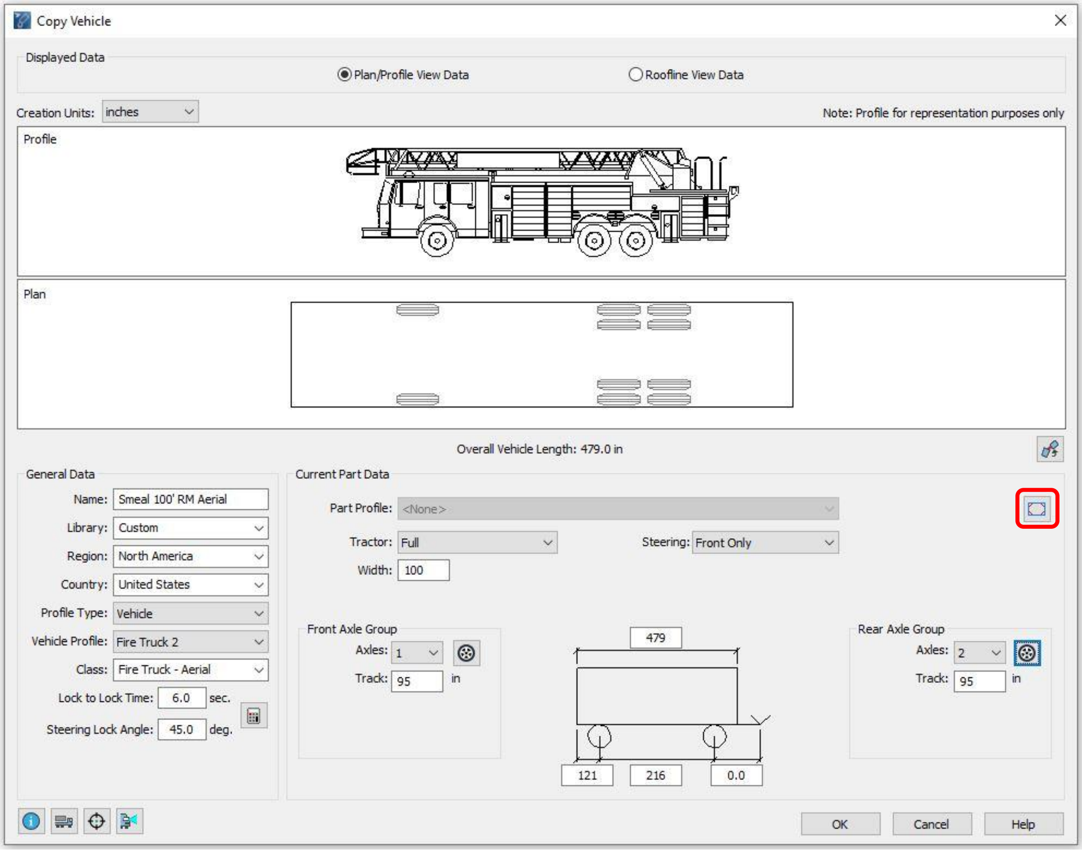


Exhibit 3 – *Copy Vehicle – Parts View Tab*

5. In the **Front Corners** list, select **Filleted** and enter *24* inches for the front radius. Under **Rear Corners**, enter *12* inches for the rear radius. Click **OK** to close the dialog box. See Exhibit 4.

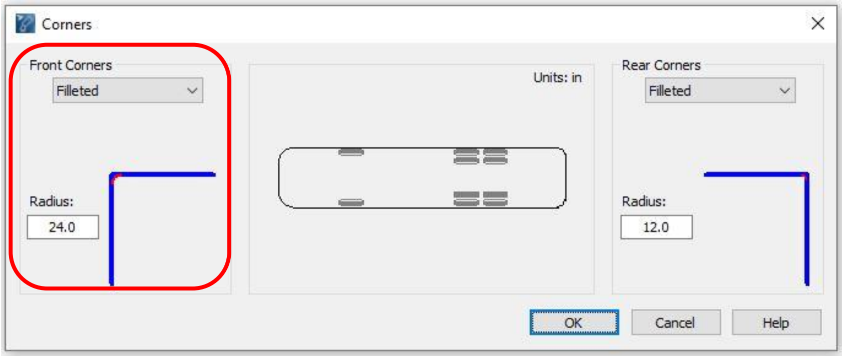


Exhibit 4 – *Corners Dialog Box – Front and Rear Fillets*

6. Click **OK** to save the vehicle in the Custom Library group. See Exhibit

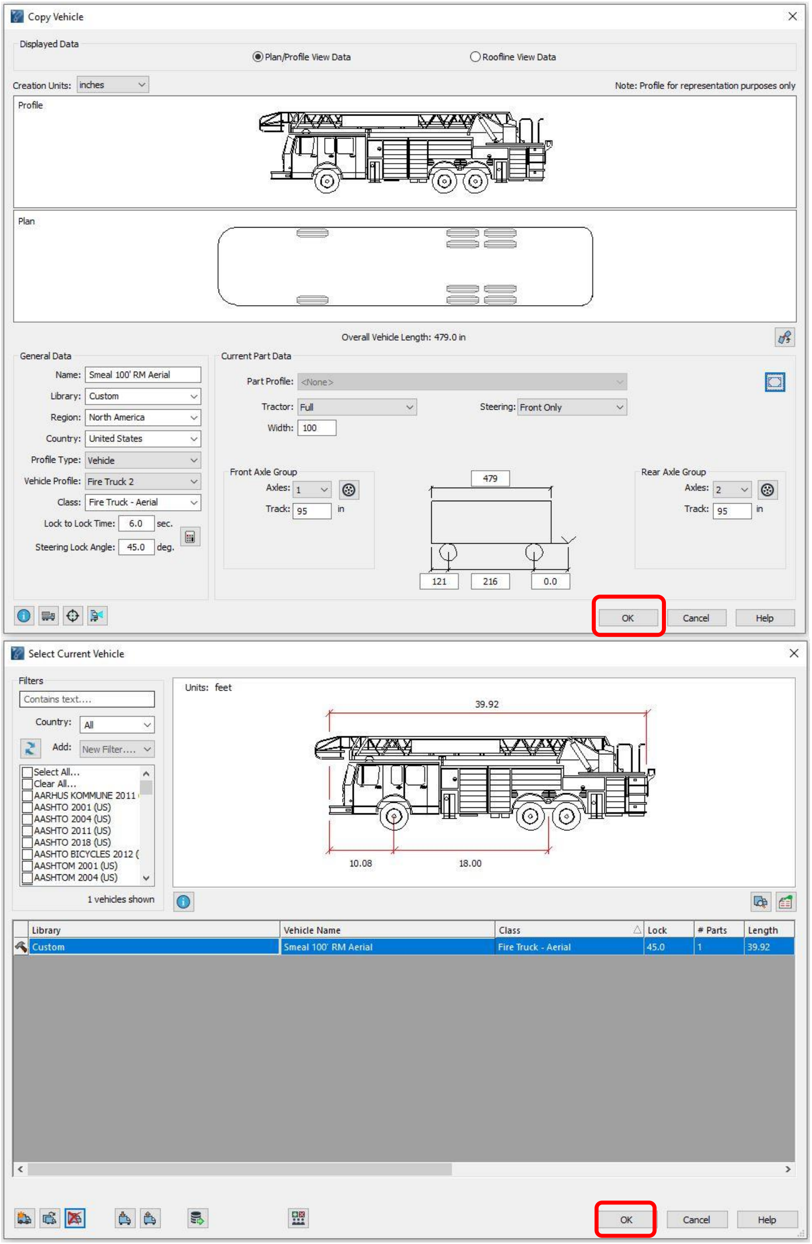


Exhibit 5 – *Select Current Vehicle Dialog Box – Vehicle Saved to the Custom Library*

7. Select the Smeal Aerial Fire Truck and click **Generate Arc Path** on the toolbar.

8. Place the Smeal fire truck at the corner of Bathgate Road and Beach Side Street. Navigate the vehicle into the back parking lot of Fire Hall #8, and perform a 360-degree turn as illustrated in Exhibit 6.

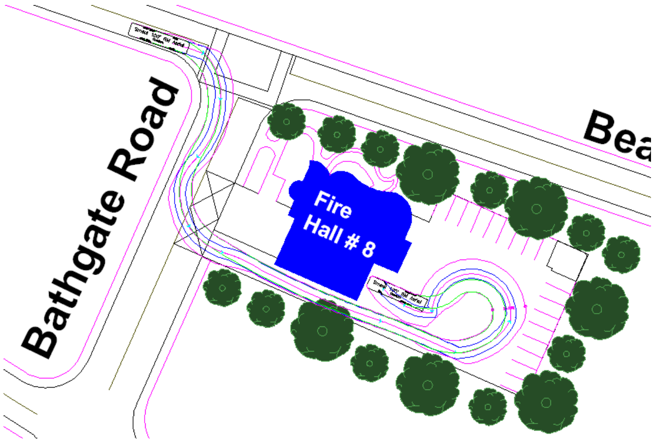


Exhibit 6 – *Test Maneuver*

By navigating the custom Smeal 100' RM Aerial truck, we can conclude that there is sufficient space within which the vehicle can easily maneuver.

▶ How to Create a Custom Vehicle from scratch

In this lesson, you will learn how to create and customize a vehicle from scratch. You will also learn how to display a side view, or profile view of the vehicle in the **Vehicles** dialog box. The profile drawing must be available to AutoTURN for each vehicle type.

NOTE:

Creating a profile drawing is optional, since AutoTURN will automatically display a plan view of the vehicle if no profile drawing exists. However, the Insert Profile tool requires a profile drawing.

In the following exercise, the Sanitation Department of Transoftville is planning to purchase new refuse trucks from the Heil Refuse Trucking Company. The new trucks will replace their aging fleet of Heil Formula 5000 trucks. The objective of this exercise is to create a custom vehicle type and create a profile drawing of the new refuse truck.

Creating a Custom Profile Drawing

The following steps will help guide you in creating a custom profile.

1. Open the drawing Heil DuraPack 7000.dwg (.dgn for Microstation). See Exhibit 1.

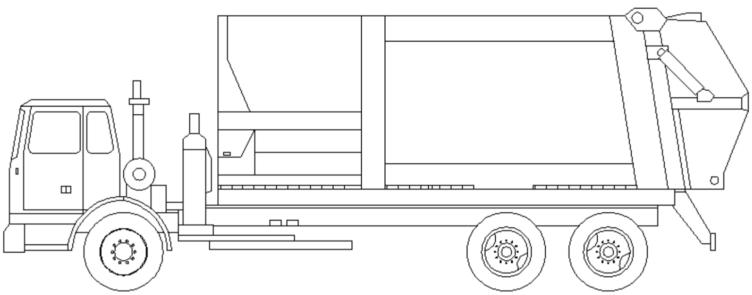


Exhibit 1 – *Heil DuraPack 7000*

2. Create three layers (levels) – ATDIMS, ATVEHICLE, and ATTEXT.
3. Add dimension lines to the vehicle as shown in Exhibit 2 for the following:
   - Body Length
   - Body Front Overhang

- Wheelbase
- Width

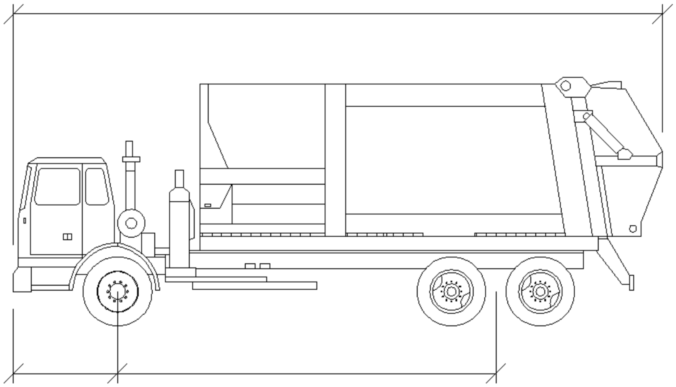


Exhibit 2 – *Dimension Lines*

4. Set the vehicle details to the ATVEHICLE layer (level), and the dimension lines to the ATDIMS layer (level).

5. We must then place Profile Tags, see Exhibit 3, against the various dimensions that we have placed. These tags enable AutoTURN to create editable dimensions for the vehicle profile in question and display the dimensions when the **Insert Profile** command is used. In this case, the Profile Tags to be used are as follows:

- BF1 - Body Front Overhang
- WB1 - Wheelbase
- BL1 - Body Length

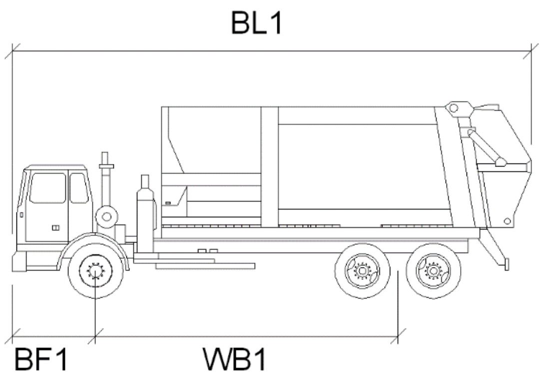


Exhibit 3 – *Profile Tags*

NOTE:

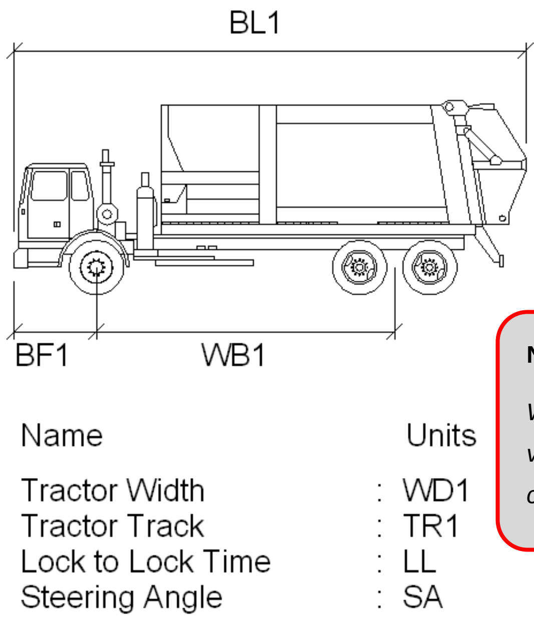
Profile Tags – Profile drawings must contain text elements called *tags* that tell AutoTURN where particular vehicle dimensions or other parameters are to be placed. The tags are text strings of 2 or 3 characters that represent the various possible dimensions that can be applied to each vehicle part and some that apply to the vehicle as a whole.

For each vehicle part, use the following tags followed by a number indicating the part number:

- BF - Body Front Overhang
- WB - Wheelbase
- BL - Body Length
- CR - Axle to Connector
- WD - Width
- TR - Track

6. Next, place a set of additional text and profile tags, see Exhibit 4, beneath the profile drawing which detail any additional vehicle parameters which will require definition, in this case the profile tags we will be adding are:

- WD1 - Tractor Width
- TR1 - Tractor Track
- LL - Lock to Lock Time
- SA - Steering Angle



NOTE:

When dealing with articulated vehicles, an additional profile tag can be used.

Exhibit 4 – *Text and Profile Tags*

7. Set the text and profile tags to layer (level) ATTEXT.
8. Select all of the elements of the profile drawing and re-position the drawing so that the midpoint of the wheelbase dimension line, WB1, is at co-ordinate point 0,0.
9. Close the drawing and save the file as Heil DuraPack 7000.dwg (.dgn for MicroStation) into the Types User folder of your AutoTURN application, the typical file path would be: ***You will need to create a subfolder called "Custom" at C:\Users\YourUsername\AppData\Roaming\TransoftSolutions\Common\ProfilesUser\Vehicles\Custom (Windows 7 or higher)**

NOTE:

The file name must match the name used for the custom vehicle when created, as AutoTURN will search for an identically named DWG (DGN) file to use as a profile view when accessing the library dialogs or the ***Insert Profile*** *command.*

General Guidelines

- Orient drawings so that the vehicle points to the left and the middle of the vehicle on the wheelbase dimension line at coordinate 0,0.
- A vehicle profile is a drawing of the vehicle side view.
- AutoCAD: each profile drawing must exist as separate elements (i.e. not as a block) in a .dwg file.
- MicroStation: Once the drawing of the custom vehicle profile has been created, MicroStation users can save the drawing as a DWG drawing file. Alternatively, MicroStation users can create a cell file of the custom profile. AutoTURN operated in MicroStation can read either DWG or CEL file formats.
- Any amount of detail can be included to represent the vehicle's profile as long as all lines representing the vehicle are drawn on the ATVEHICLE layer.
- The profile drawing (either the .dwg file or the cell library) must have the same name as the vehicle type and must be located in the Profile Types folder (e.g. C:\Users\Your Username\AppData\Roaming\Transoft Solutions\Common\Profiles User\Vehicles\Custom.
- Profile Layers/Levels - Drawings should be created with the following layer/level setup:

Layer/Level	**Description**
ATVEHICLE	All lines pertaining to the vehicle
ATDIMS	All dimension lines
ATTEXT	All dimension labels and text

▶ How to Create Realistic Vehicles

In the following exercise, we are going to create a custom realistic vehicle that will then be placed within our **Custom** library group for use within simulations.

Creating a Realistic Vehicle Drawing

AutoTURN generates parametric plan views for custom vehicles based on the part configuration and vehicle dimensions. However, since a generic plan view may not provide an accurate enough representation of the vehicle it is recommended that users create their own drawings for custom vehicles.

Rogers CR35 Gooseneck

The Crane Construction Company specializes in equipment operation of crane bulldozers. They use the Rogers CR35 Gooseneck trailer to transport their bulldozers to construction sites or for client delivery. In the following exercise, we will create a Rogers CR35 Gooseneck transport trailer for Crane Construction Co.

Rogers CR35 Gooseneck

1. Open the Rogers CR35 Front Cab.dwg file from the folder of training drawings. See Exhibit 1.

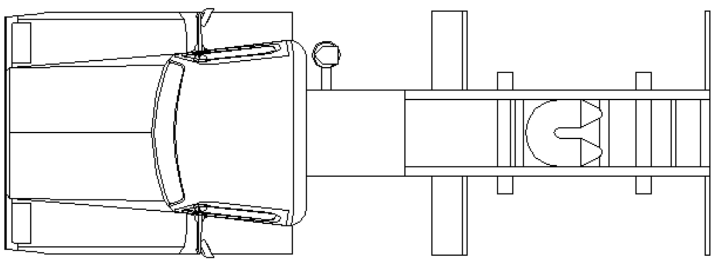


Exhibit 1 – *Rogers CR35 Front Cab Drawing*

2. Select all of the elements and rotate the drawing so that the vehicle faces to the right and the front center of the vehicle is positioned at exactly co-ordinate 0,0. See Exhibit 2.

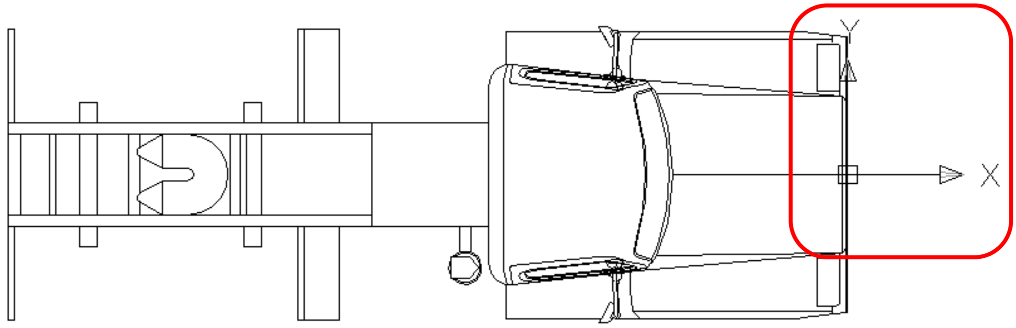


Exhibit 2 – *Vehicle Positioned at Co-ordinate 0,0*

3. Create three individual layers onto which we can place the various elements of the vehicle called:

- TSOUTLINE
- TSDETAILS
- TSWHEELBASE

4. Select the exterior line of the vehicle, a single closed polyline that encompasses the entire plan view profile of the vehicle, and place it onto the TSOUTLINE layer.

5. Select all of the interior detail lines of the vehicle and place them onto the TSDETAILS layer.

6. Draw a line underneath the vehicle with the following parameters.

- Length 15.08 feet.
- Start point 3.74 feet back from the front of the vehicle.

7. This line, as displayed in Exhibit 3, should then be placed onto the TSWHEELBASE layer.

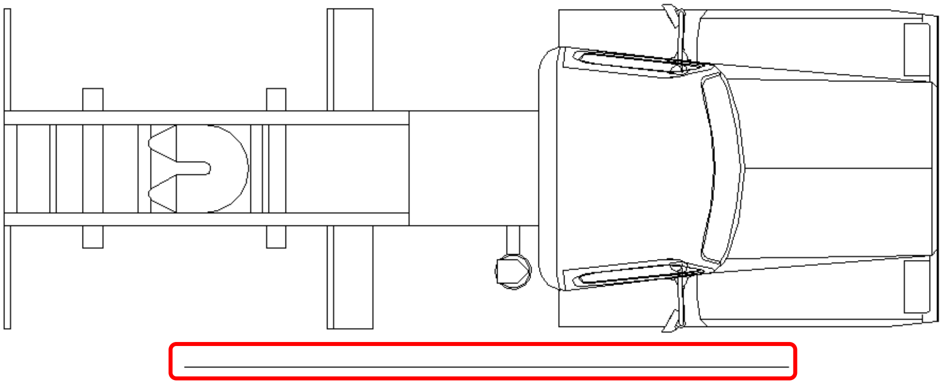


Exhibit 3 – *Wheelbase Line*

> **NOTE:**
>
> *The line that is drawn and placed on the TSWHEELBASE layer is used by AutoTURN to scale the drawing when required.*

8. Close the drawing and save it into the Vehicles User folder of your AutoTURN application in a new folder named Custom, the typical file path would be: C:\Users\Your Username\AppData\Roaming\Transoft Solutions\Common\vehicles user\Custom Rename the file so that it is now called *Rogers CR35 Gooseneck*. *Create a sub folder called "Custom"
9. Repeat the previous steps for the trailer portion of the vehicle.
10. Create a wheelbase line with a length of 36.75 feet and place it 3.74 feet back from the front of the trailer.
11. Close the drawing and save it as Rogers CR35 Gooseneck@1.dwg. Place the saved file into the same directory as the front cab.

NOTE:

*The file name must match the name used for the custom vehicle when created, as AutoTURN will search for an identically named .dwg file to use as a plan view during simulations. Each part of a multi-part custom vehicle must be drawn individually. The first drawing's name is the same as the n**ame** of the custom vehicle (e.g. the name of the first drawing of a two part vehicle would be: Semi Trailer.dwg). Each subsequent drawing must be named the same as the first drawing with the suffix @ followed by the trailer number (e.g. the second drawing of the two part vehicle would be: Semi Trailer@1.dwg).*

NOTE:

Creating the Drawing – MicroStation

Once the drawing of the custom vehicle has been created, MicroStation users can save the drawing as a DWG drawing file. Alternatively, MicroStation users can create a cell file of the custom vehicle as outlined below. AutoTURN operated in MicroStation can read either DWG or CEL file formats.

1. Start a new design file.
2. Create a new level named TSOUTLINE and draw the extents of the vehicle on this level.
3. Create a new level named TSDETAILS. Draw all the details inside the basic outline of the vehicle on this level. This will improve performance by allowing AutoTURN to separate the drawing elements and only display the vehicle outline while dragging and running animations.
4. Create a level named TSWHEELBASE. Draw a line that exactly matches the wheelbase value of the custom vehicle on this level. It can be located anywhere in the drawing and will not display in the vehicle preview dialog boxes.
5. Create a new cell library named the same as the name of the custom vehicle.
6. Create a cell and add it to the new library. The cell can be given any name. There must be only the one cell in this library.
7. Create a folder in the **Vehicles User** folder to save the drawing to. The folder name must be the same as the name of the **Library** of the custom vehicle.
8. Save the cell library to the folder created in step 7.

General Guidelines

- Orient drawings so that the vehicle points to the right and the front center of the vehicle part is at exactly coordinate 0,0.
- **Wheels** - Do not draw wheels. AutoTURN draws them automatically.
- Any amount of detail can be included to represent the vehicle's plan view as long as the vehicle's outline is continuous and goes on its own layer and a wheelbase line is drawn the same length as the wheelbase value of the custom vehicle. The drawing must be drawn full-scale.
- Each part of a multi-part custom vehicle must be drawn individually. The first drawing's name is the same as the name of the custom vehicle (e.g. the name of the first drawing of a two part vehicle would be: Semi Trailer.dwg). Each subsequent drawing must be named the same as the first drawing with the suffix @ followed by the trailer number (e.g. the second drawing of the two part vehicle would be: Semi Trailer@1.dwg).
- Drawings should be created with the following layer/level setup:

Layer/Level	Description
TSOUTLINE	All lines pertaining to the extents of the vehicle
TSDETAILS	All detail lines inside the vehicle (windows, doors, contours, etc.)
TSWHEELBASE	This single line must exactly match the length of the custom vehicle's wheelbase. This line is used to scale the drawing when it is used by AutoTURN.

Creating a Custom Profile for a Two-part Vehicle

The next stage of our custom vehicle creation is to produce a custom side profile drawing of the Rogers CR35 Gooseneck. The following steps explain how to create the profile for a two-part vehicle.

1. Open the **Rogers CR35 Profile.dwg** file from the folder of training drawings. See Exhibit 1.

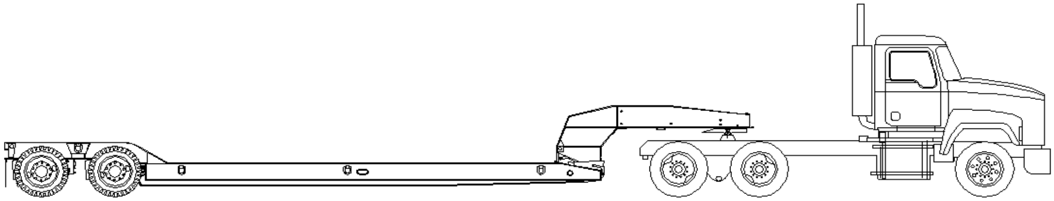


Exhibit 1 – *Rogers CR35 Side Profile*

2. Reorient the vehicle so that it points to the left.
3. Add dimension lines to the vehicle as shown in Exhibit 2 for the following:
   - Body Length
   - Body Front Overhang
   - Wheelbase
   - Axle Connector
   - Width
   - Track

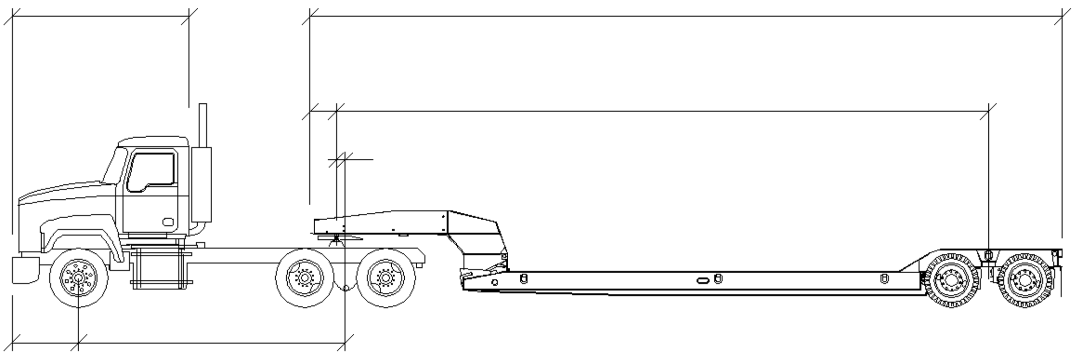


Exhibit 2 – *Dimension Lines*

4. We must then place Profile Tags, see Exhibit 3, against the various dimensions that we have placed. In this case, the Profile Tags to be used are as follows:

   - BF1 - Body Front Overhang
   - WB1 - Wheelbase
   - BL1 - Body Length

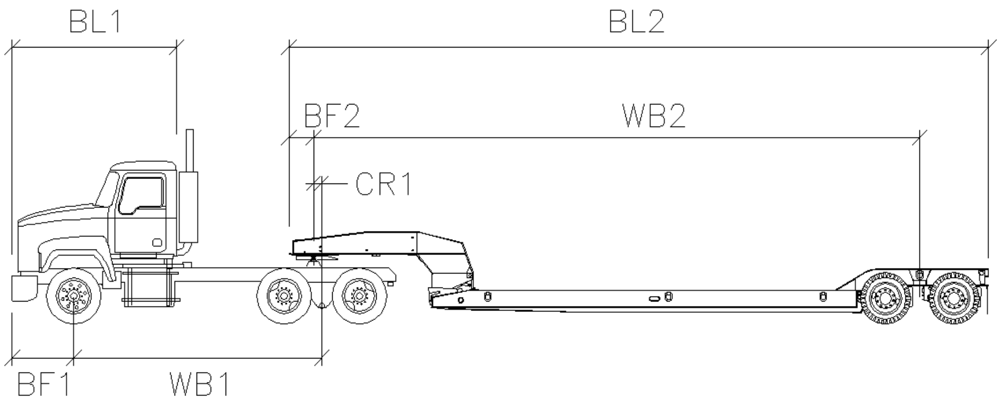


Exhibit 3 – *Profile Tags*

5. Next, we will need to place a set of additional text and profile tags. Since the Rogers CR35 Gooseneck is a two-part vehicle, we will need to add the tag for the Articulating Angle. See Exhibit 4.

- WD1 - Tractor Width
- WD2 - Trailer Width
- TR1 - Tractor Track
- TR2 - Trailer Track
- LL - Lock to Lock Time
- SA - Steering Angle
- AA - Articulating Angle

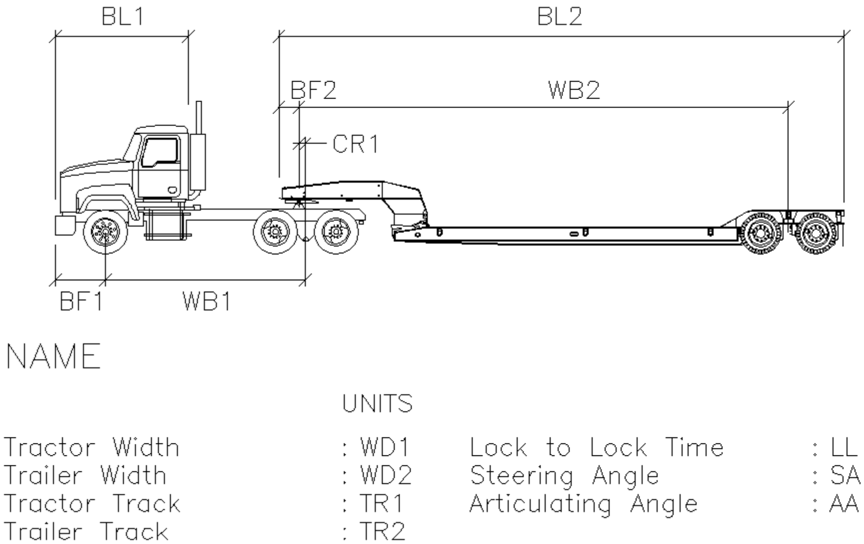


Exhibit 4 – *Text and Profile Tags*

6. Create three individual layers onto which the various elements of the vehicle profile can be placed, these layers are as follows.
   - ATVEHICLE
   - ATDIMS
   - ATTEXT

7. Select all of the dimension lines and place them onto the layer ATDIMS.
8. Select all of the dimension labels and profile tags and place them onto the layer ATTEXT.
9. Select all of the drawing detail related to the vehicle profile and place them on the layer ATVEHICLE.
10. Select all of the elements of the profile drawing and re-position the drawing so that the midpoint of the wheelbase dimension line, WB1, is at co-ordinate point 0,0.
11. Close the drawing and save it into the Types User folder of your AutoTURN application, the typical file path would be:
12. C:\Users\YourUsername\AppData\Roaming\TransoftSolutions\Common\ProfilesUser\Vehicles\Custom
    Rename the file so that it is now called *Rogers CR35 Gooseneck*.

Creating the Custom Vehicle Definition

Before we can complete the creation of our custom realistic vehicle, we need to create a new vehicle type. This is outlined in the following steps.

1. Click  Vehicles on the toolbar.
2. Click the Create New Vehicle button in the dialog box. See Exhibit 1.

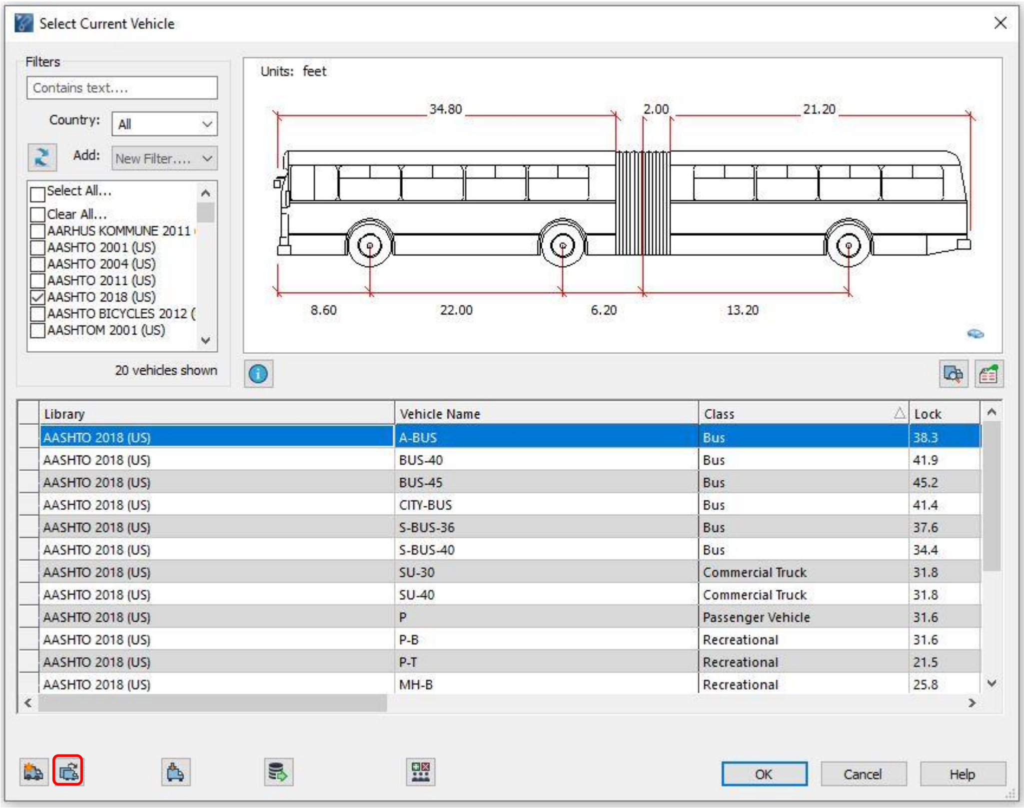


Exhibit 1 – *Types Dialog Box – Create New Type Button*

3. Enter the following data in the Create New Vehicle dialog box as shown in Exhibit 2.
   - Name - Rogers CR35 Gooseneck
   - Class - Construction
   - Tractor - Pin Ahead
   - Rear Axles - 2

- Rear Wheels - 4

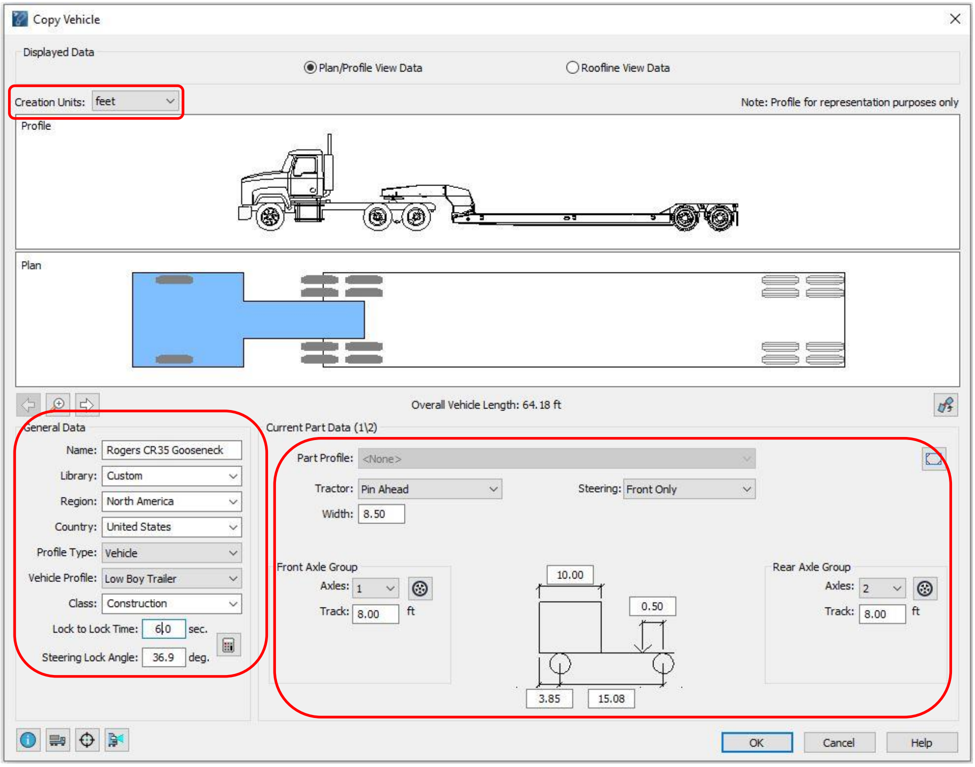


Exhibit 2 – *Create New Vehicle Dialog Box*

4. Change the Creation Units to feet and enter the following values in the dialog box.

- Body Length - 10.00 feet
- Body Front Overhang - 3.85 feet
- Wheelbase - 15.08 feet
- Body Front - 1.25 feet
- Axle to Connector - 0.50 feet
- Body Length - 42.17 feet
- Wheelbase - 36.75 feet
- Width - 8.00 feet
- Front Track - 8.00 feet

- Rear Track - 8.00 feet
- Trailer Track - 8.50 feet
- Name - Rogers CR35 Gooseneck

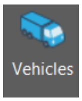
5. Select the Calculate Steering Lock tool and enter a Centerline Turn Radius of 25.12 feet, then click Apply. A new Steering Lock Angle of 36.9 degrees will be displayed in the dialog box. See Exhibit 2.

Vehicle Creation - Axle configurations

▶ How to Create a New Vehicle

The ability to Create New Vehicles allows you to customize vehicles by defining the dimensions and turning characteristics to match manufacturers' specifications. This ranges from creating a new vehicle from scratch, adding a new part (e.g. a trailer) to an existing vehicle, or modifying the number of axles, tires, and even changing the vehicle dimensions.

In the first example, we will show how easy it is to add an axle and reconfigure this vehicle from tandem and tridem axles.

1. Click 

Vehicles on the toolbar.

2. Select the AUSTROADS 2013 (AU) standard vehicle library. Select the SU TRUCK and click the **Create New Vehicle** button. See Exhibit 1.

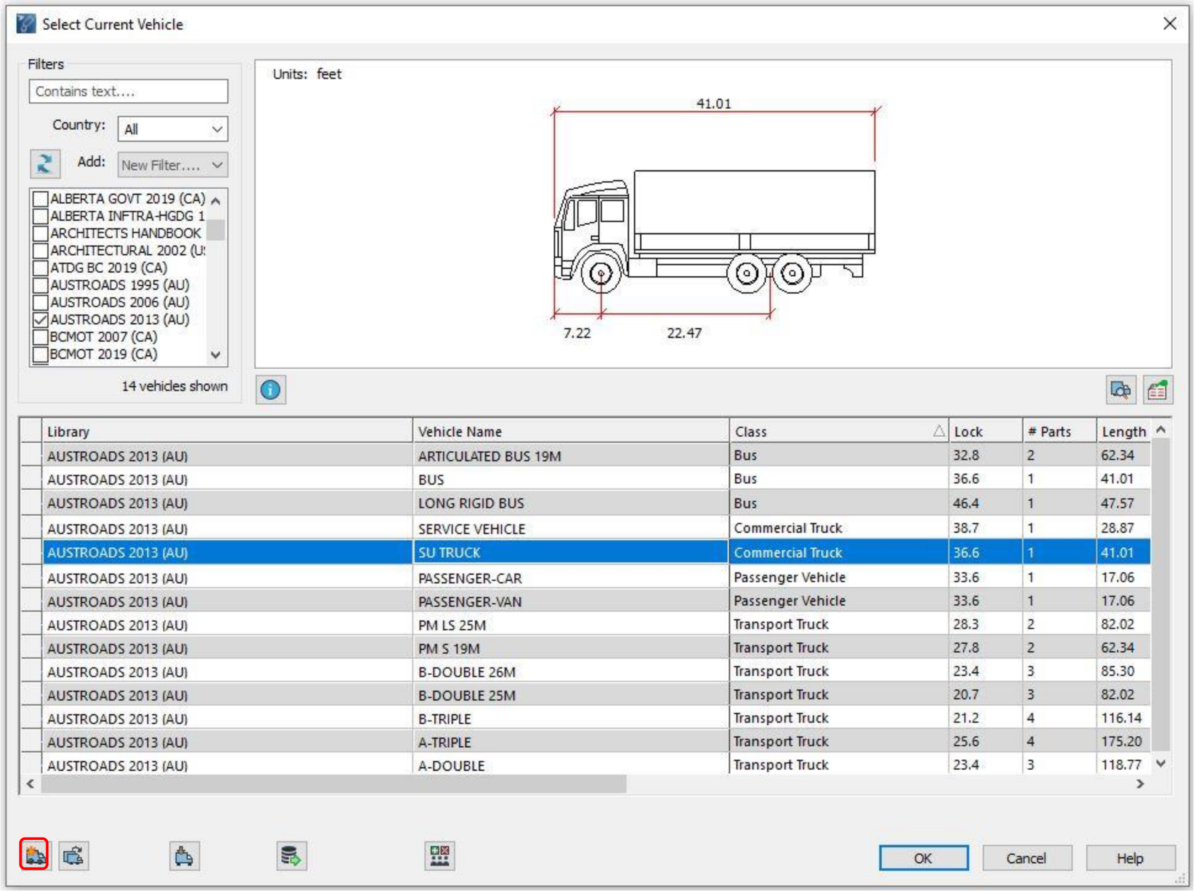


Exhibit 1 – *Select Current Vehicle Dialog Box*

3. In the **Create New Vehicle** window, start by naming the new vehicle, change the Creation Units to inches and add a new axle to the Rear Axle Group as shown in Exhibit 2.

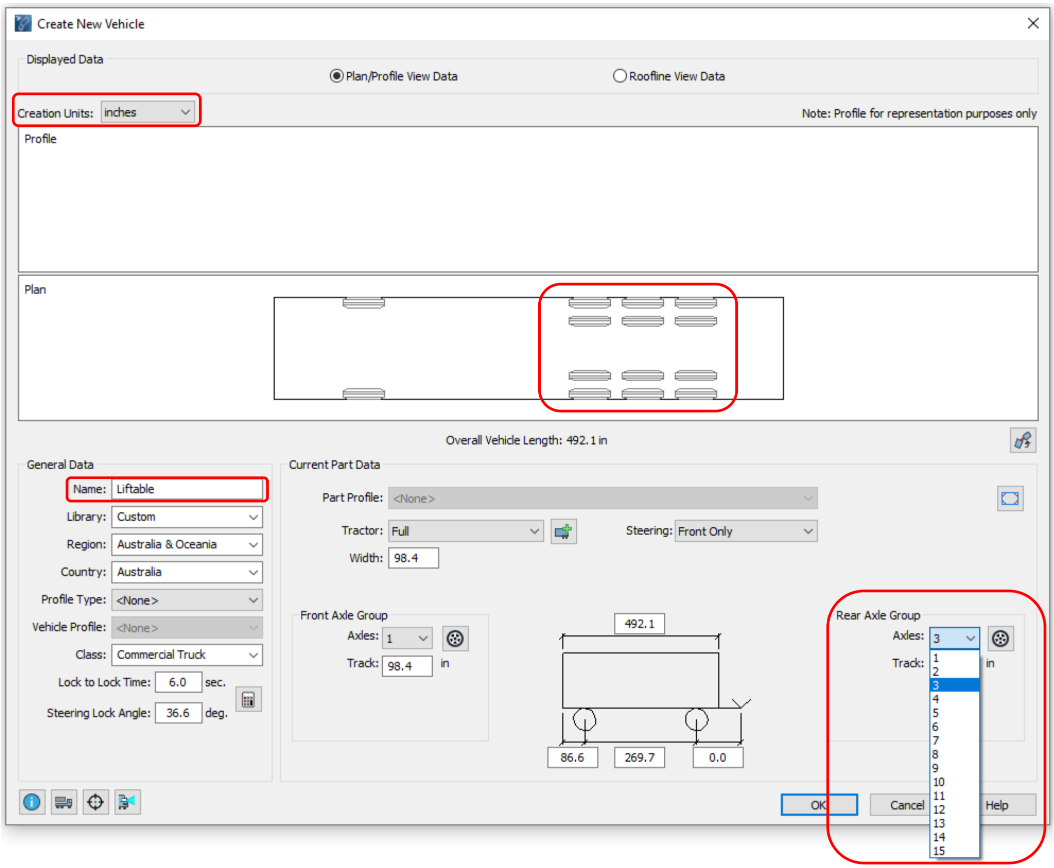


Exhibit 2 – *Create New Vehicle Dialog Box*

4. For this example, we want to analyze a vehicle using an axle that can be lifted or lowered to help the vehicle's axle weight distribution. Although these axles are intended to help with axle loading purposes, it has a direct impact on the vehicle turning behavior as the effective wheelbase of the vehicle changes depending on whether the axle is lifted or lowered. These vehicles can be defined and the "liftable" axles can be raised or lowered when generating different sections of the simulation.

In the **Create New Vehicle** Dialog Box click the **Rear Axle Group Details,** change the number of Liftable Axles to 1 and the position to Front, this will define the first axle of the rear axle group as the liftable axle. If the rear option is selected from the Position dropdown list, the axles will be counted from the rear to the front. Hence the rearmost axle will be defined as the liftable axle. For the purpose of this example, leave the option for the Position to Front and select 1 for the Liftable axles dropdown. See Exhibit 3.

Then click **Customize** Axle Spacing and define the distance between the first and second axles of 70 inches. Click Ok to close both dialog boxes and observe the results in the Plan View Data. See Exhibit 3.

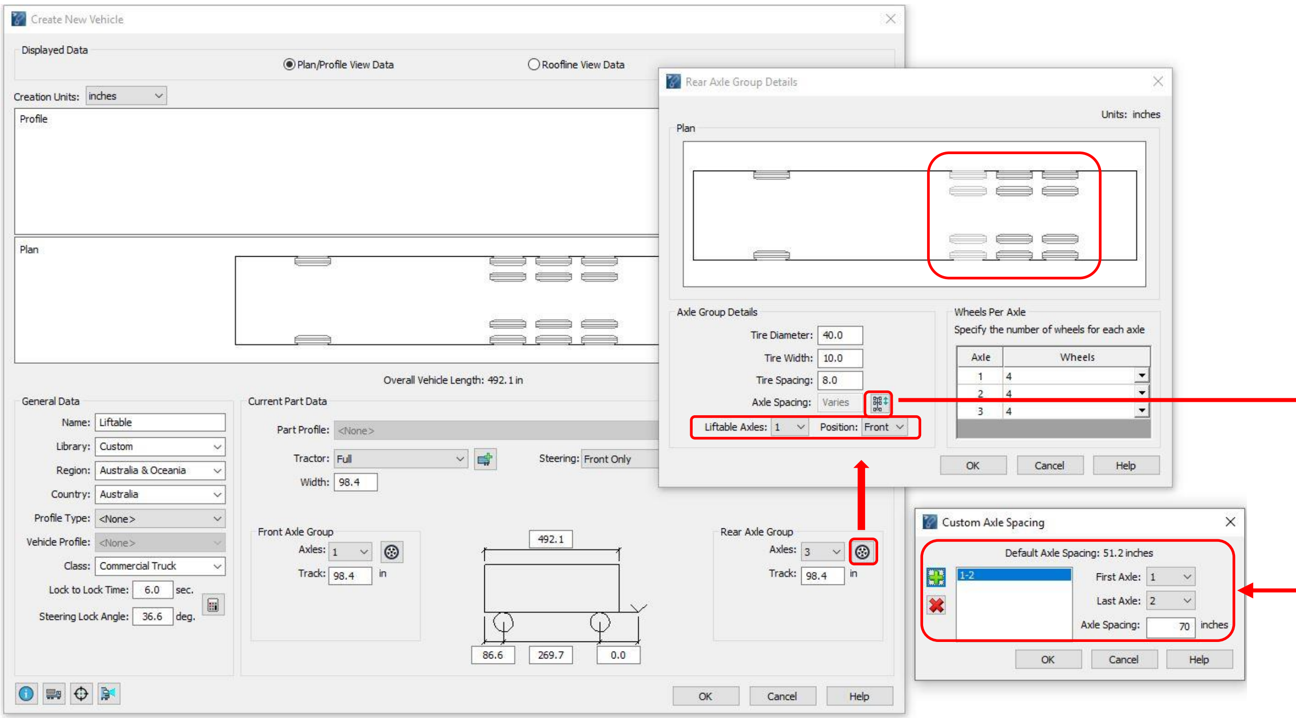


Exhibit 3 – *Customizing Axles*

5. Click OK to close the **Create New Vehicle** Dialog Box and Ok again to select the vehicle. To better illustrate the effect of the liftable axles when it comes to the turning behavior of the

vehicle, let's create two minimum turn simulations using the same speed, one with all three axles on the ground and the second one where the first axle of the rear axle group is lifted.

6. In the Ribbon, bar click 

2D Arc Path. Click in the drawing area to define a start point for the simulation and enter 90 degrees for the start Angle and click apply. Then, in the **Smart Path Tools,** set the speed to 6 mph and perform a turn simulation keeping the smallest turning radius possible. See Exhibit 4.

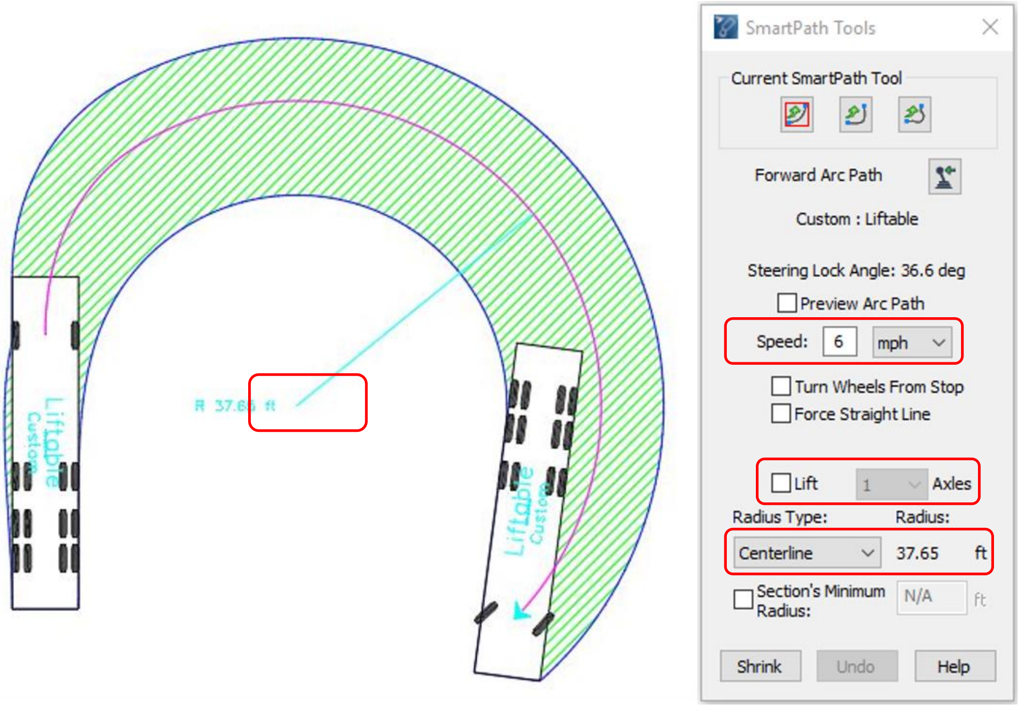


Exhibit 4 – *2D Arc Path*

7. For comparison purposes, draw markers in the beginning and in the end of the simulation, then copy and place these markers beside the first simulation. See Exhibit 5.

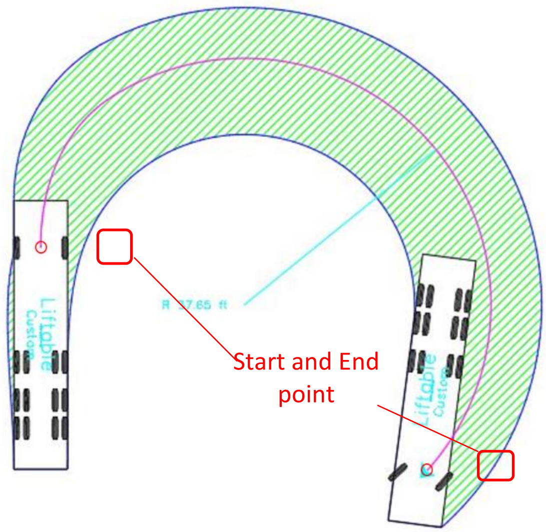


Exhibit 5 – *Turning Simulation Start and End Points*

8. Create a new simulation starting in the copied Start Point. In the Smart Path Tools dialog, ensure the same speed and the start angle is set, but check the **Lift** box. Once again try to produce a minimum turn to reach the second marker on the right. Note how the minimum radius is bigger than the one in the first simulation. See Exhibit 6.

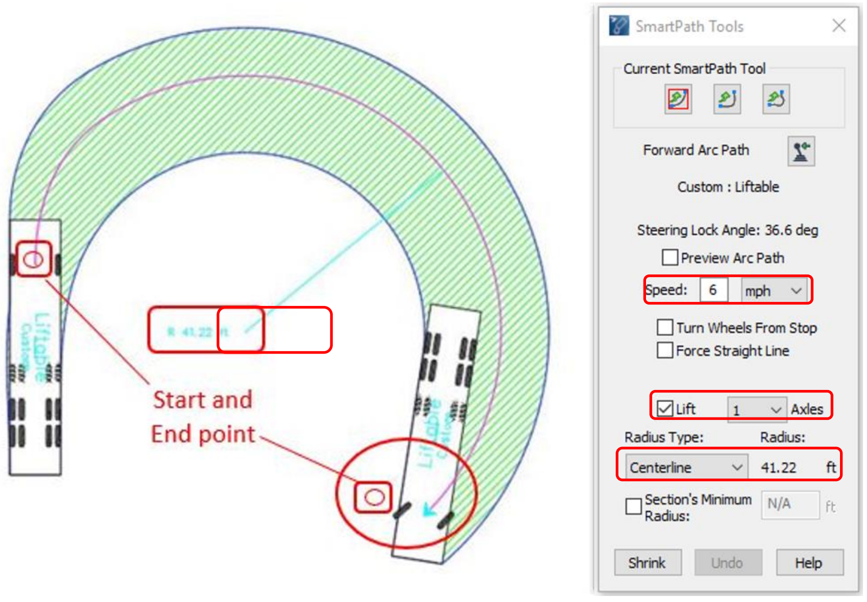


Exhibit 6 – *2D Arc path – Lifted Axles*

9. The Radius is higher because when the axle is lifted the effective wheelbase of the vehicle is longer. This is the expected behavior for vehicles with longer wheelbases. See exhibit 7.

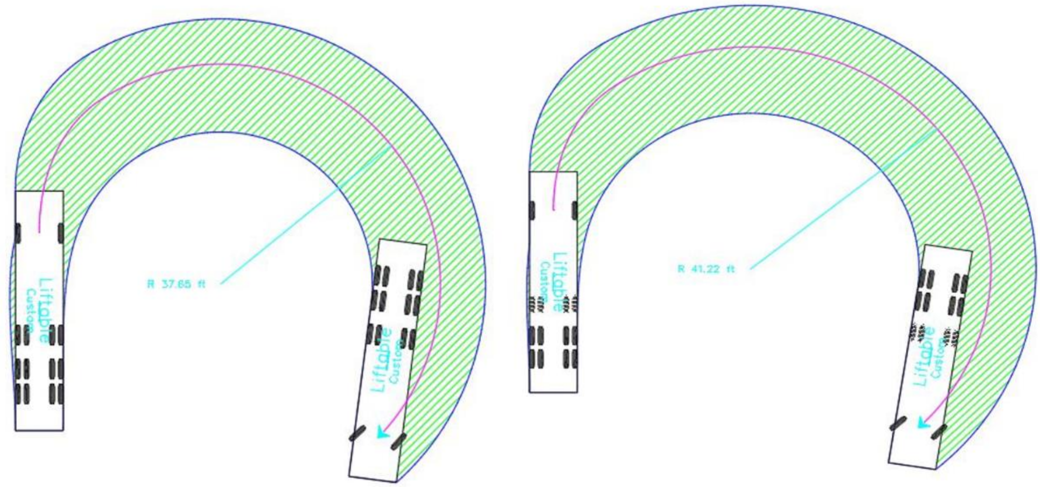
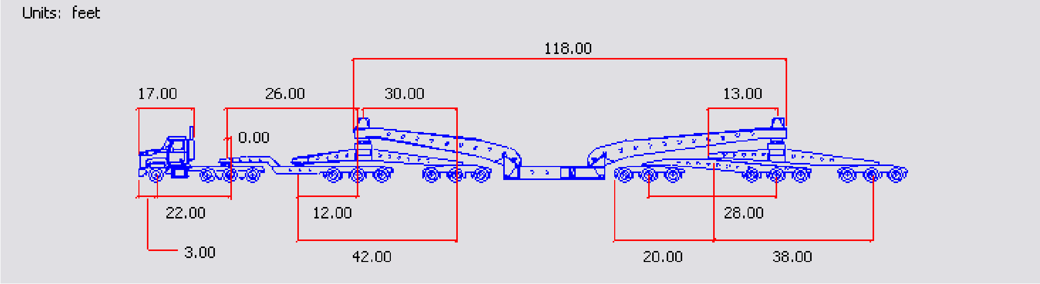


Exhibit 7 – *Turning Simulations Comparison*

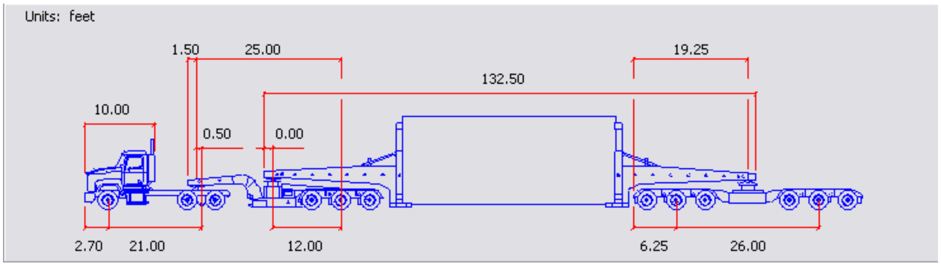
How to Maneuver Special Transport Vehicles

In AutoTURN, the Special Transport Vehicles are a set of custom vehicles that are used to deliver heavy, oversized and elongated loads to construction sites. These are independent rear steering vehicles that have either an independent or an axle steering system. In AutoTURN, there are six types of Special Transport Vehicles, and they are:

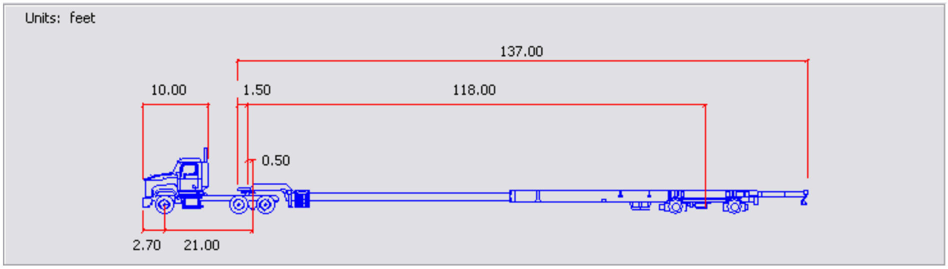
19 Axle Heavy Haul Trailer:
This vehicle is used to transport large heavy-duty items such as large printing presses, turbine generators, etc.



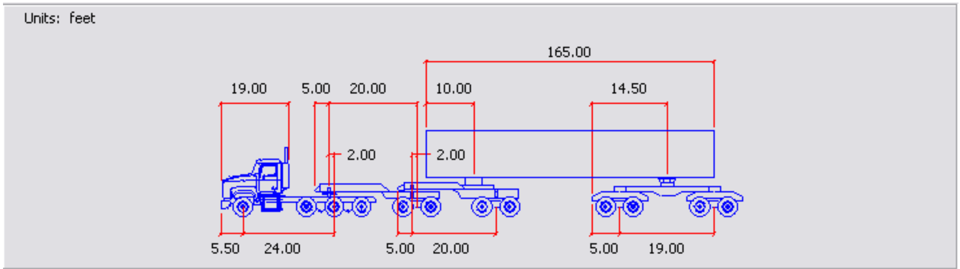
Wind Tower Trailer ("Schnabel" trailer):
This vehicle is used to haul tower segments (wind towers). The tower segments themselves act as the bodies of the trailer, while the jeeps are attached at both ends.



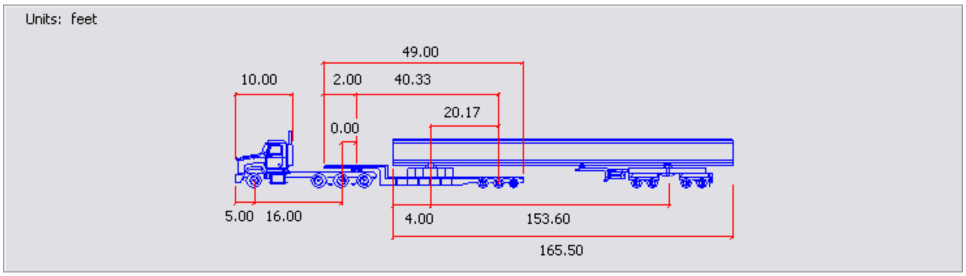
Wind Blade Trailer:
This vehicle is used to transport the blades for the wind turbines.



Beam Transporter 1 & 2:
This vehicle is used to transport precast concrete sections for buildings and bridges.

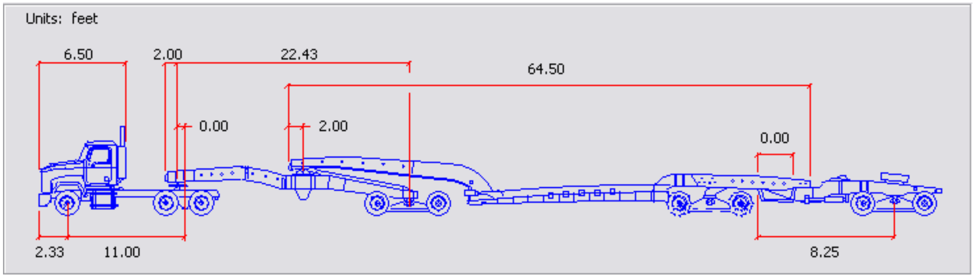


Beam Transporter 1



Beam Transporter 2

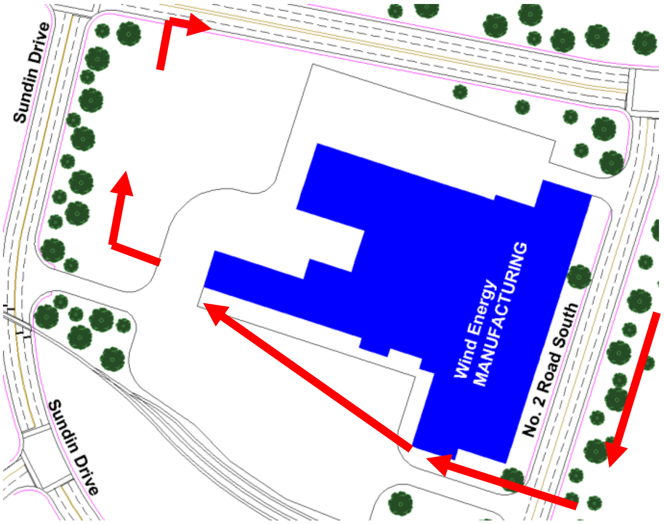
Booster Trailer:
This vehicle is used to move heavy construction machinery such as bulldozers, etc.



The size, length and articulating angle of the specialty transport can easily be customized to your project need. This is covered in the AutoTURN Advanced tutorials.

How to Control the Path Angle of a Trailer

In this exercise, Wind Energy Manufacturing – an industry leader in wind turbine technology will need to transport a large section of a wind tower for customer delivery. They have contracted the Flores Heavy Moving Company to transport this large complex tower section to the TRC Rail Shipping yards. The objective of this exercise is to navigate the Wind Tower transport into the Wind Energy Manufacturing facilities to pick up the wind tower section. The Wind Tower transport will enter the facilities from the south entrance on No. 2 Road, and exit on to Sundin Drive. The transport will then proceed north and make a right turn on to 7th Avenue and drive to the TRC Rail Shipping yards.



Wind Tower Transport Proposed Pick-up Route.

1. Select the Wind Tower Trailer from the Vehicle Library. See Exhibit 1.

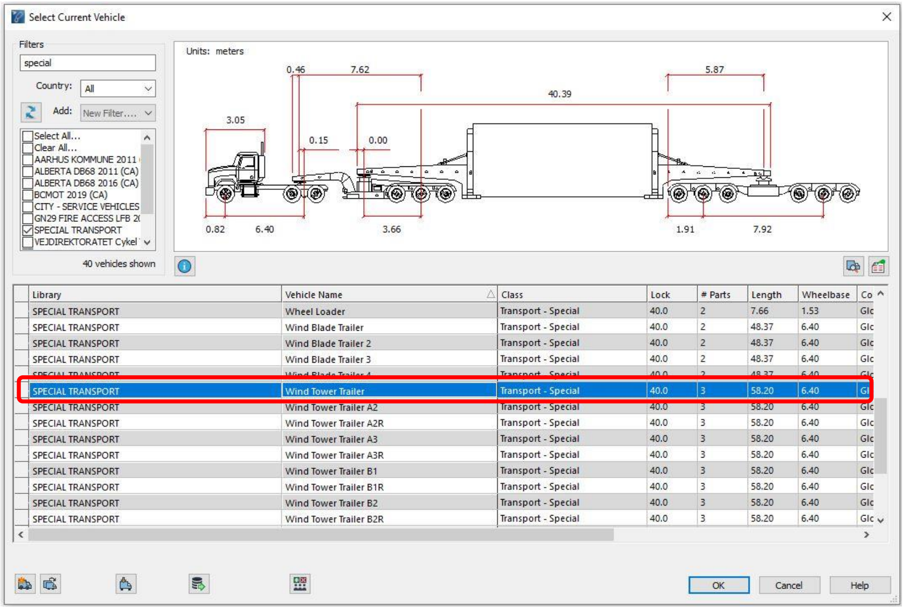


Exhibit 1 – *Select Current Vehicle – Wind Tower Trailer.*

2. Click **Generate Arc Path** on the toolbar.
3. Place the Wind Tower vehicle near the south entrance of the Wind Energy Manufacturing entrance. Set the appropriate starting angle. See Exhibit 2.

Exhibit 2 – *Wind Tower Trailer Starting Position*

4. Set the speed of the vehicle to 6 mph and select the **Turn Wheels from Stop** checkbox. See Exhibit 3.

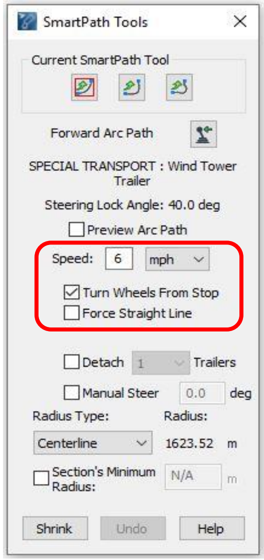


Exhibit 3 – *SmartPath Tools – Turn Wheels from Stop Enabled*

5. Navigate the Wind Tower transport into the facility. Enabling the **Manual Steer** allows you to control the path of the rear bogey. The path angle of the rear bogey is represented by a red line. See Exhibit 4.

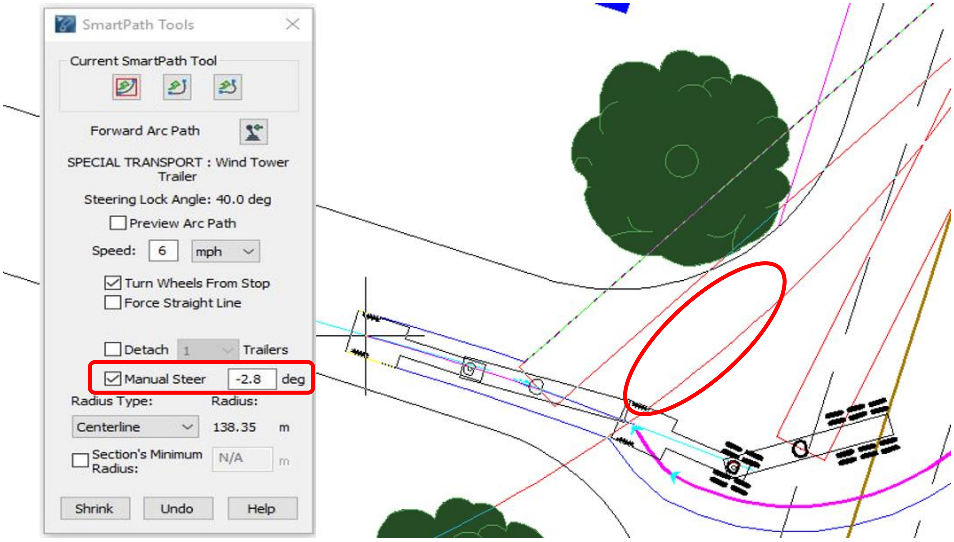


Exhibit 4 – *Manual Steering*

NOTE:

*You can press CTRL and simultaneously scroll the middle mouse wheel to increase/decrease the **Override Angle** or **Path Angle** value by intervals of 0.2 degrees (for MicroStation users: If the mouse wheel settings (Workspace -> Preferences -> Mouse Wheel) were specified prior to the installation of AutoTURN 7.0, you should set the **Ctrl+Wheel** option in **Set Mouse Wheel Preference** to none to be able to use this functionality).*

6. Navigate the vehicle forward, while holding the CTRL key and simultaneously scrolling the mouse wheel to control the path angle of the rear bogey. See Exhibit 5.

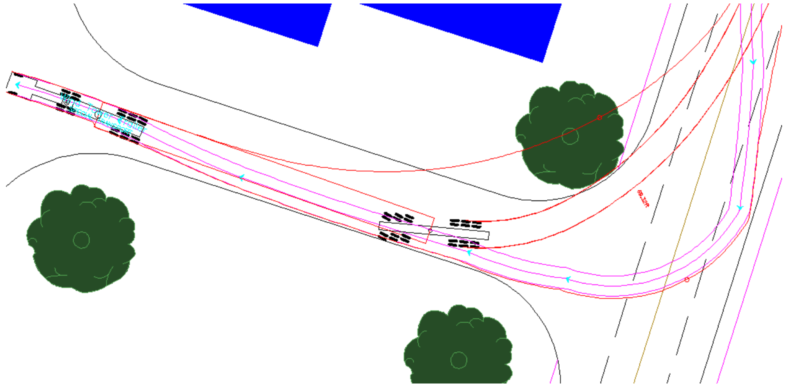


Exhibit 5 – *The swept path of the wind tower encroaches upon the tree line during the turn.*

7. When the rear bogey has cleared the obstacle, clear the **Manual Steer** checkbox. See Exhibit 6.

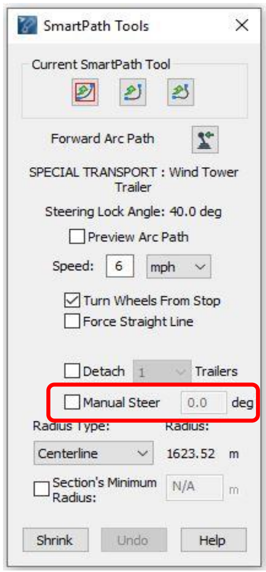


Exhibit 6 – *SmartPath Tools –Manual Steer option disabled*

8. Navigate the Wind Tower Vehicle towards the west entrance onto Sundin Drive and then onto 7th Avenue. See Exhibit 7.

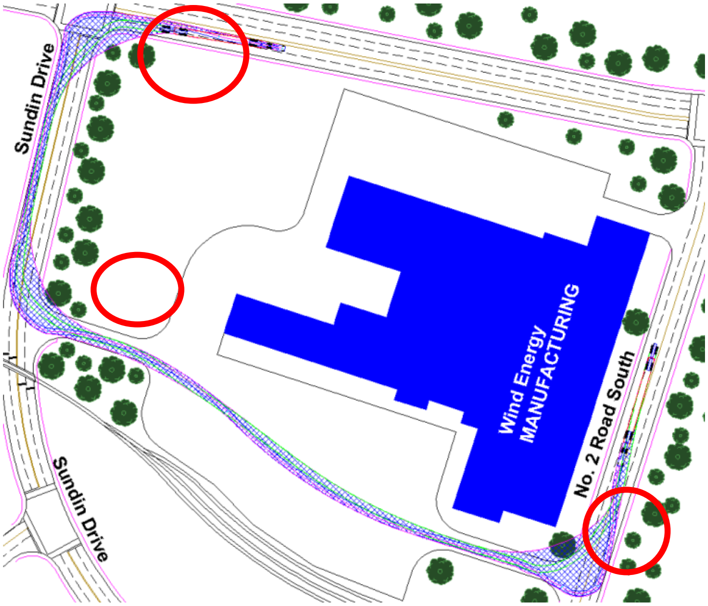


Exhibit 7 – *Route Obstacles*

▶ Customizing Special Transport Vehicles' Axle Groups

It is possible to customize the Axle Groups for Special transport vehicles by Copying or Creating a new one.

1. Click Vehicles on the toolbar or ribbon, select the SPECIAL TRANSPORT – Booster Trailer C1 and click the Copy Vehicle button. See Exhibit 1.

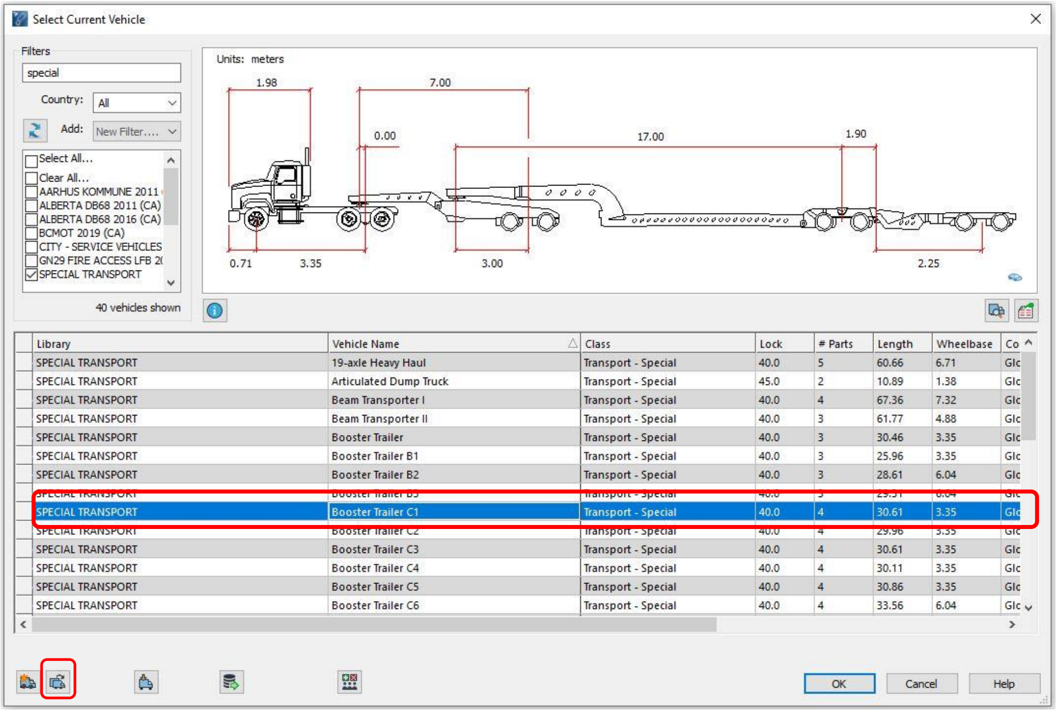


Exhibit 1 – *Copy Vehicle – Booster Trailer C1.*

2. Give a name to the copied vehicle and navigate to the last part of the vehicle. Note that it is possible to make changes to the vehicle's dimensions, Axle groups, choose the tractor connection and trailer type. Change the Steering option to Rear Steerable. See Exhibit 2.

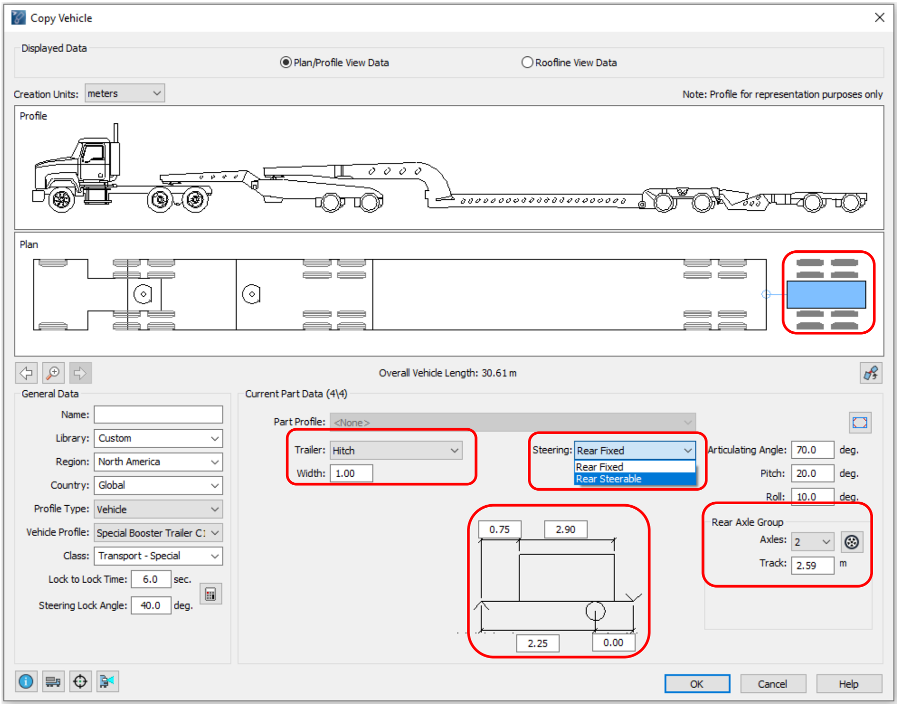


Exhibit 2 –*Wind Tower Trailer rear steering.*

NOTE:

See below all the Steering Types available:

Part Type	Description
Tractor	
Front Only	Steering axles in the front group only.
Rear Only	Steering axles in the rear group only.
Front and Rear	Steering axles in the front and rear groups.
Front Fixed Only	Axles in the front group only and none are steering axles.
Trailer	
Rear Fixed	Axles in the rear group only and none are steering axles.
Rear Steerable	Fixed and steering axles in the rear group.

How to Configure Steering Linkages

The **Steering Linkages** sets the articulating angle to rear steering angle relationship for a rear steered part in custom vehicles. The rear steered part's steering angle can be linked to a previous part's articulating angle. A steering linkage can also be defined for single part custom vehicles when front and rear steering is configured. In this case, the angle of the front steering angle affects the rear steering angle. The steering linkage is defined by specifying the angle pairs that define the steering behavior between the two axle groups. See below the steps to configure Steering Linkages for a vehicle.

1. Click Vehicles on the toolbar or ribbon, select the SPECIAL TRANSPORT – Wind Blade Trailer and click the Copy Vehicle tool. See Exhibit 1.

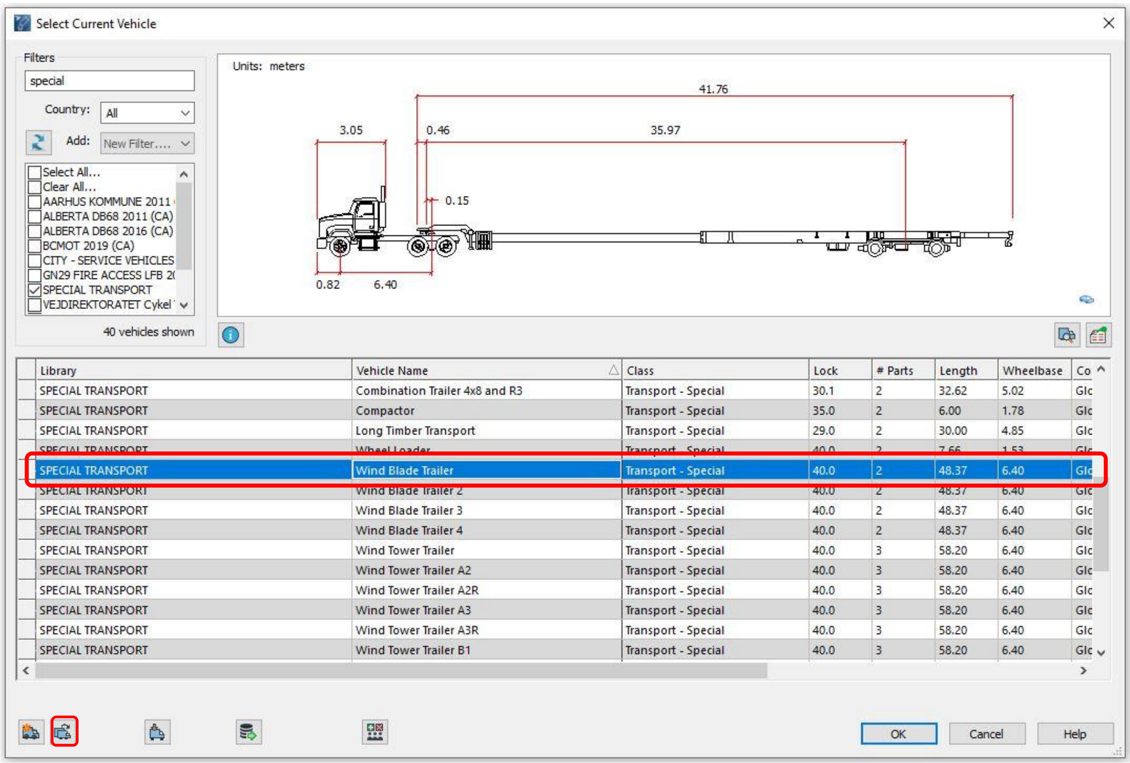


Exhibit 1 – *Copy Vehicle – Wind Tower Trailer.*

2. Give a name to the copied vehicle and navigate to the last part of the vehicle and click in Steering Linkages. See Exhibit 2.

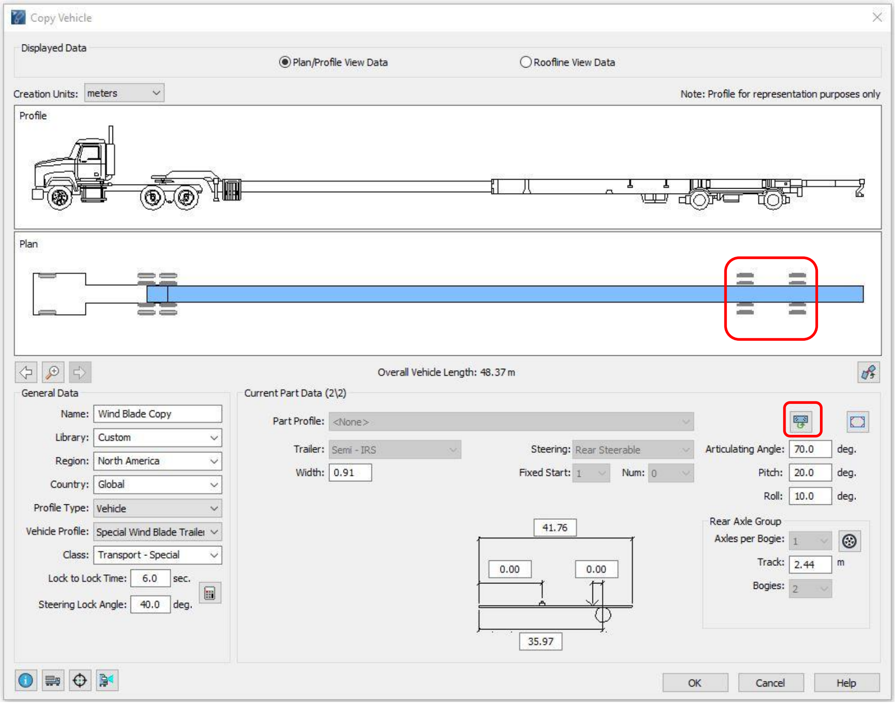


Exhibit 2 –*Wind Tower Trailer rear steering.*

3. If the multi-part vehicle has more than one articulating angle in the vehicle, under Steering Linkage Relationship, select the part to establish a relationship with. Under Steering Linkage Data, specify the desired angle pairs by clicking Insert, and then typing the desired values in the columns. As changes are made in the table, the Steering Linkage Graph updates accordingly. Click OK to close the Steering Linkages dialog box. See Exhibit 3.

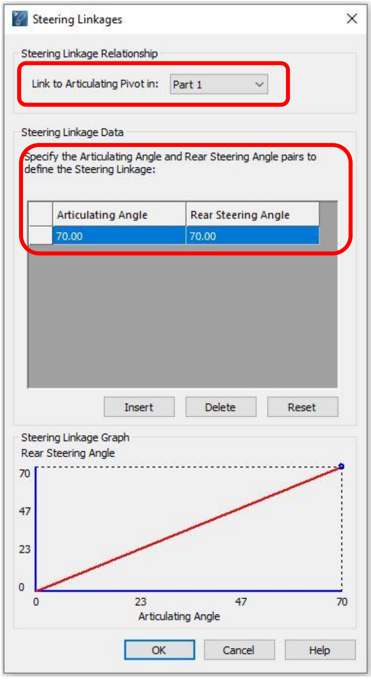


Exhibit 3 – *Steering Linkages - multi-part vehicle*

4. Set the Rear Steering Angle to 25.00 (note the changes in the Steering Linkage Graph) click OK and select the vehicle. See Exhibit 4.

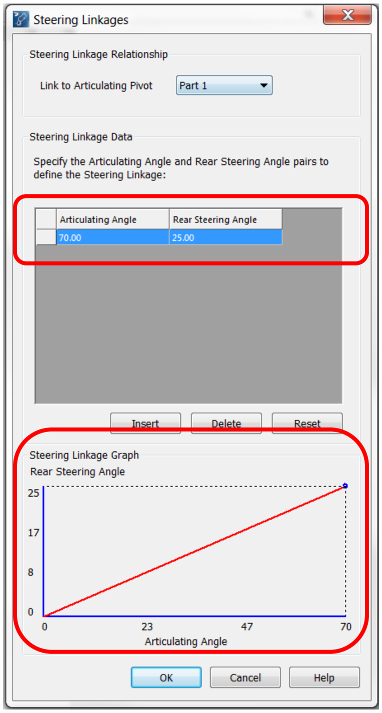


Exhibit 4 – *Change the Rear Steering Angle*

5. Now, let's place two turning simulations using the vehicle with customized Rear Steering Angle and the regular Wind Blade Trailer and then compare the Rear Steering Angles when the Articulating Angle is maximum (70 degrees). See Exhibit 5.

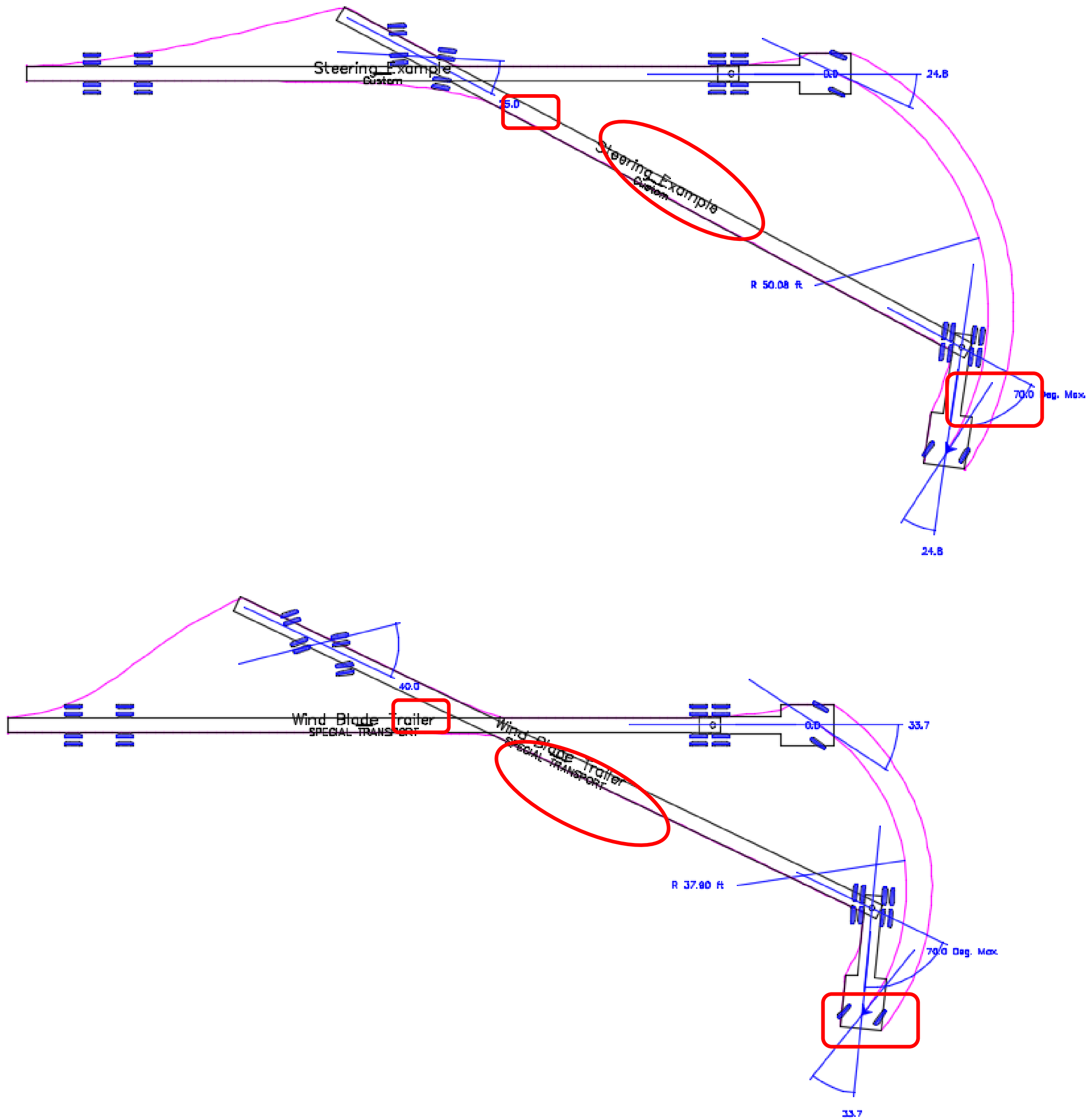


Exhibit 5 – *Special Transport Vehicles – Rear Steering Angle comparison*

NOTE:

Despites the Wind Blade Trailer have a Rear Steering Angle of 70 degrees as default, in the simulation above it performs the maximum Axle Group Angle (Located in Rear Axle Group Details) which in this case is 40 degrees.

How to use IntelliPath to perform Circulation Analysis

Circulation analysis is one of the key tasks when trying to establish how vehicles will access or circulate through a site or a road network. The use of **IntelliPath** to perform this type of analysis involves defining the scenario; comprising of an analysis area, exclusion lines/zones (non-drivable areas), and routes the vehicle has to travel, after which the drive paths can be evaluated for one or multiple vehicles.

Note: **IntelliPath** is better suited for scenarios where the designer is trying to determine whether a route can accommodate one or multiple vehicles. Vehicle swept paths are automatically created by the software once the scenario and the routes are defined.

Analyzing traffic calming measures

In this example, we will use the IntelliPath tool to illustrate its application on evaluating the effects of implementing traffic calming measures for service vehicles.

For the following downtown scenario (i.e. street network scenario), the existing routes for a fire truck and a commercial truck will be defined first. Then, vehicle movements will be generated using the IntelliPath tool for both vehicles. Next, the curb extensions representing the chicane to cause the s-shaped drive path will be introduced in the model. Finally, the effects on the drive path and speed for both vehicles will be observed.

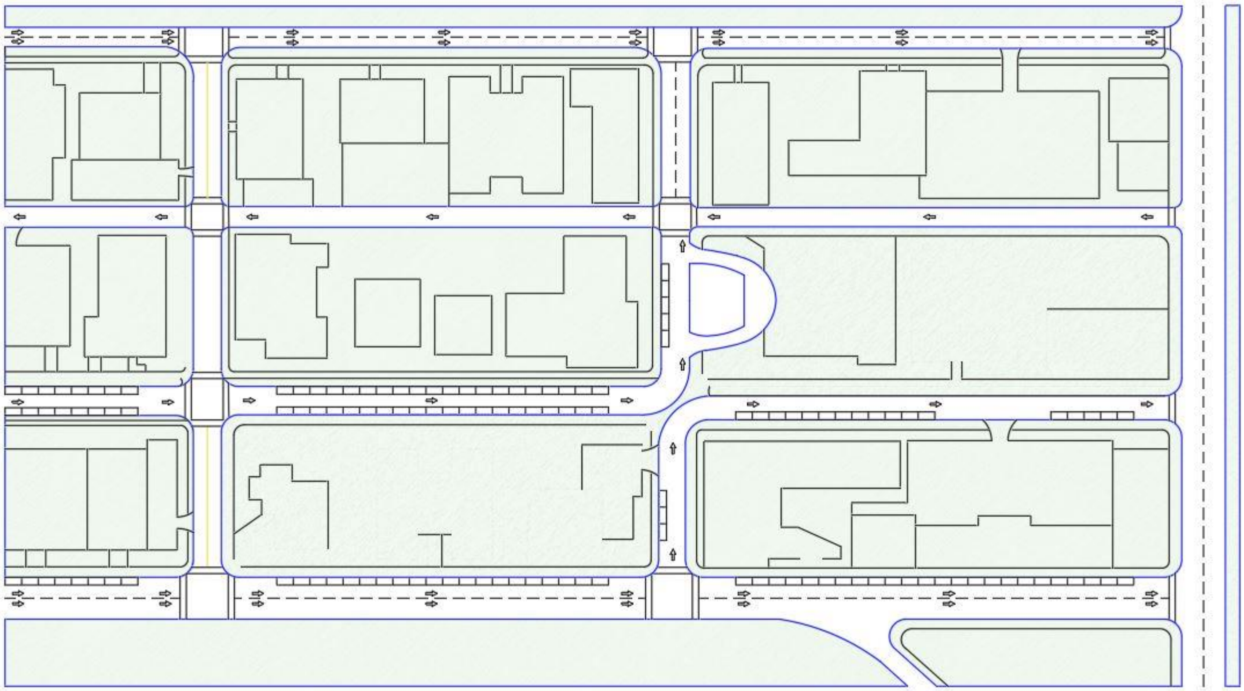


Exhibit 1 – Analysis Area of the Street Network Scenario

1. The first step is to define the analysis area. This area defines the extent of where all the vehicle movements will be generated. This area can be defined in the drawing by clicking points to create a closed shape (requires at least 3-points) . The route analysis area can be any shape or size.

   Click  **IntelliPath** on the main toolbar to begin. In the New Route Analysis Area dialog box, type “MY AREA 1” to define the name for the area. See Exhibit 2. Note: Multiple analysis areas can be defined for the same design scenario.

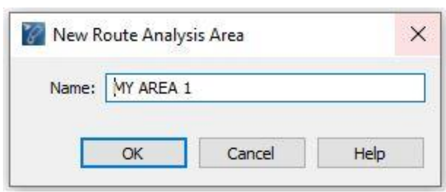


Exhibit 2 – Define the name for the new route analysis area

In the IntelliPath dialog box, under the Route Analysis Area section, click Define Route Analysis Area. Then pick 4 points (*as shown below in exhibit 3)* enclosing the entire area to be analyzed, Right-click the mouse to exit the selection process. The enclosed area will be highlighted in cyan. See exhibit 3.

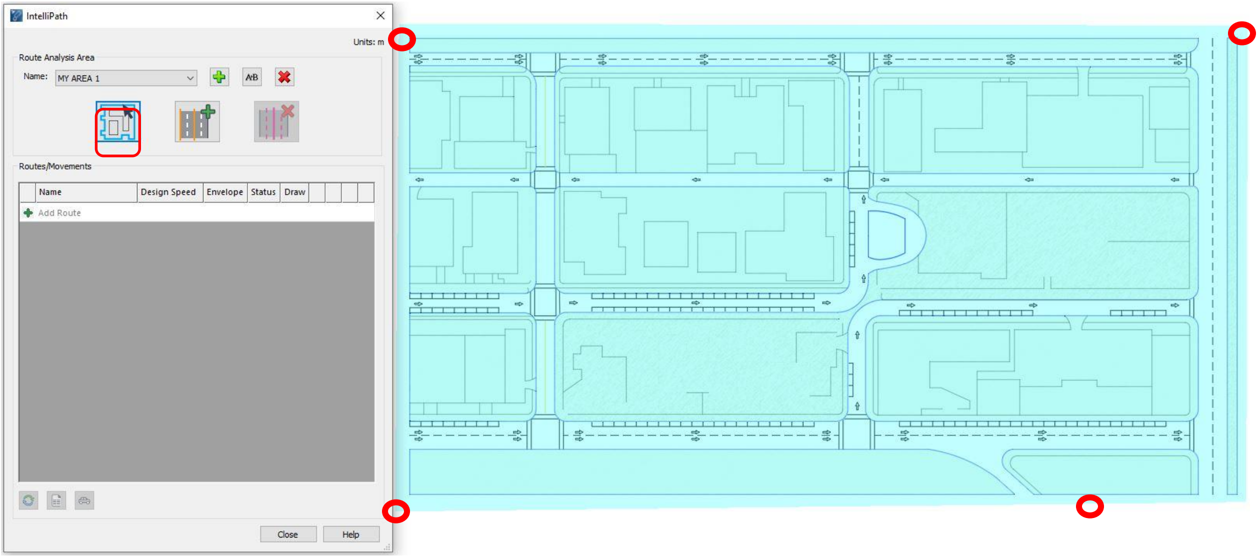


Exhibit 3 – *Route Analysis Area*

2. The next step is to indicate the non-drivable areas in the drawing, this is achieved by selecting the exclusion areas (road/building boundaries). These areas can be lines, arcs, polylines, closed complex chains, closed combinations of lines, arcs, and polylines the vehicle can't cross. Note: The drivable areas dictate the extends of the vehicle's tire tracks or the vehicle body envelopes when the movements are generated.

3. In the IntelliPath dialog box, click the **Pick Exclusion Lines** button and select the non-drivable areas by clicking on the individual objects representing all the zones that are not supposed to have traffic or routes passing through them or drag from left to right to select all objects that are entirely enclosed in the selection rectangle or lasso (*window* selection) or drag from right to left to select all objects that are crossed by the selection rectangle or lasso (*crossing* selection) The objects selected will be highlighted in yellow as depicted in Exhibit 4.

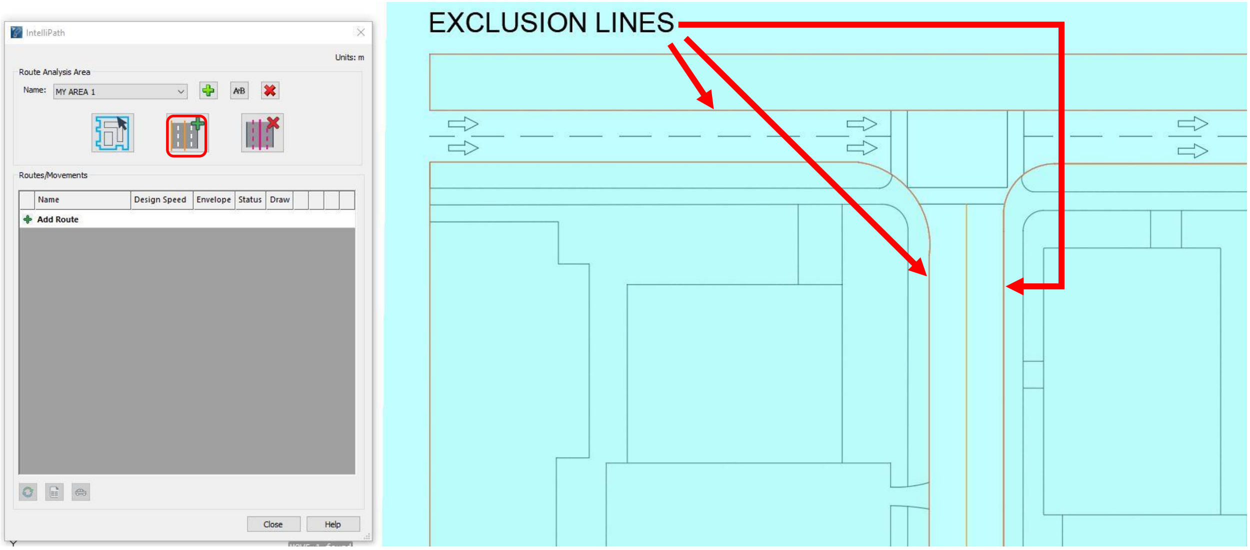


Exhibit 4 – *Define exclusion Zones*

4. After defining the Exclusion areas, we can proceed to perform the circulation analysis. To define the different routes click Add Route in the IntelliPath dialog box, under the Routes/Movements section. Click on the upper-left corner of the road to set the route's starting point, then move the cursor to the desired endpoint and do a left-click to place the route and right-click to finish defining the route. This will trigger the Editing route options to fine-tune the route. The irregular edges of the route are to be modified/smoothened into a straight line or an arc or can be aligned to an existing element, using the pick element option. At any point, you can click on the **Edit Route** button to fine-tune the routes even after creation. Generate the 2 routes (yellow and magenta routes) as shown in Exhibit 5.

Note: To edit the route in parts, click on the Edit Route button, click on the start point and endpoint of the section on route to be modified/fine-tuned and select how to redraw the selected boundary. You can select to draw as a Corner (line), an Arc, or Pick an element.
For cases where multiple routes are possible between the desired start and endpoints, you will need to do intermediate clicks (left clicks) along the desired route. If you have miss-clicked, you'll have to delete the route and add a new one.

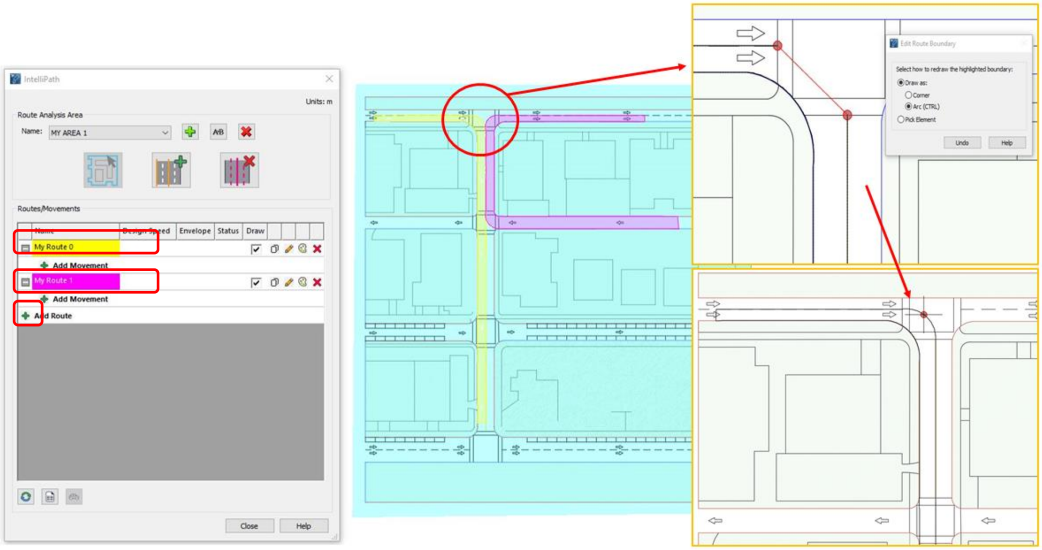


Exhibit 5 – *Adding a Route*

5. The routes define the intended vehicle circulation through the site. Once the routes have been defined, one or multiple vehicle movements can be created for each route to preliminary assess whether the vehicles will be able to use the route or not.
6. Under My Route 0, Click Add Movement to open the Generate movement dialog box. The design vehicle and the movement speed can be defined in this dialog. For this example, select the NCHRP REPORT 659 (US) – Aerial Fire Truck, and set the speed to 10 km/h set. See Exhibit 6.

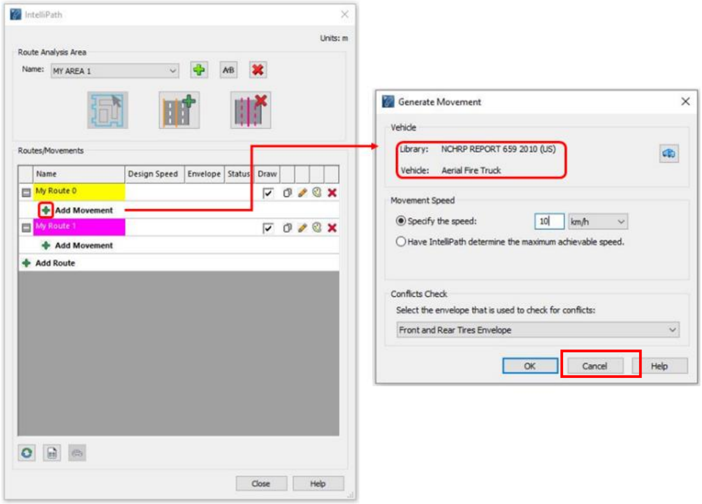
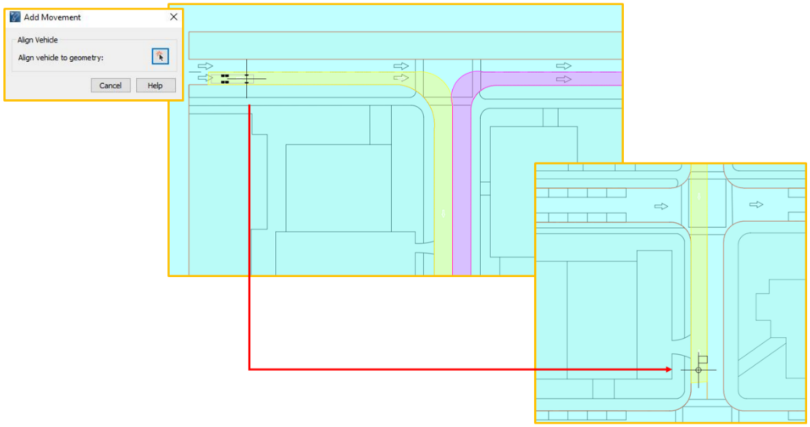


Exhibit 6– *Select Design vehicle and speed*

7. In the drawing, do a left click to define the desired starting point for the simulation and click again to define the vehicle's orientation. For the start position, choose a point close to the beginning of the route (i.e. North West region). Click a second time to define the starting orientation of the vehicle. Note that, as you move the cursor along the route the cursor will be displayed in green, but if the vehicle goes over one of the exclusion zones or outside the analysis area, the cursor will turn red. After defining the orientation, move your cursor to the desired endpoint for the movement, do a left click to select the point and a right-click to place or generate de movement as depicted in Exhibit 7.



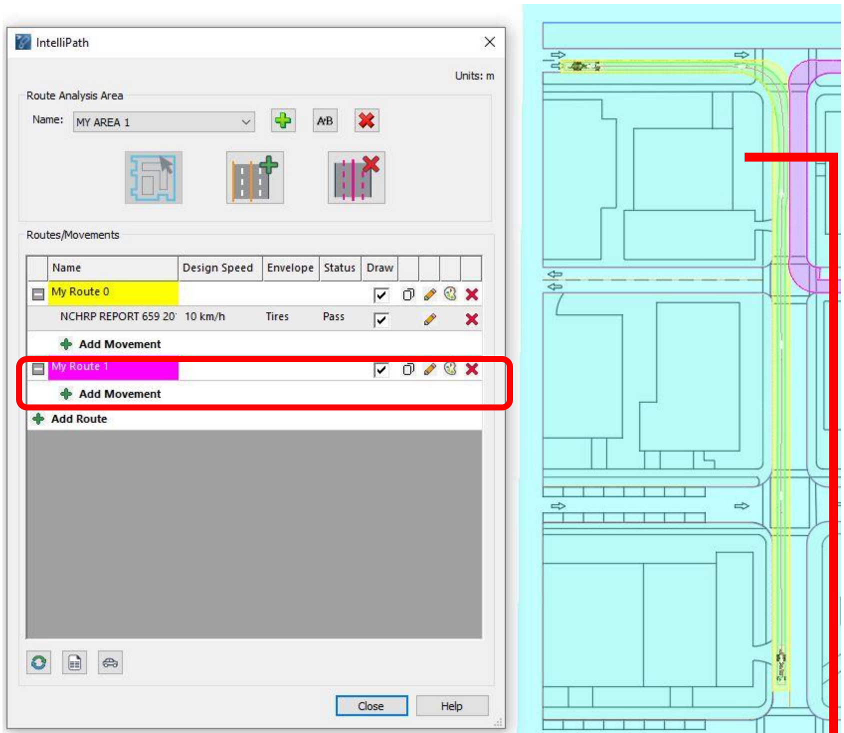


Exhibit 7– *Generate Movement*

8. To generate a movement for the second route, under the My Route 1 in the IntelliPath dialogue box, Click Add Movement and select the AASHTO 2018 (P) vehicle Similarly, set the speed to 10km/h. Define the starting point and orientation, and the endpoint for the vehicle movement as indicated in Exhibit 8.

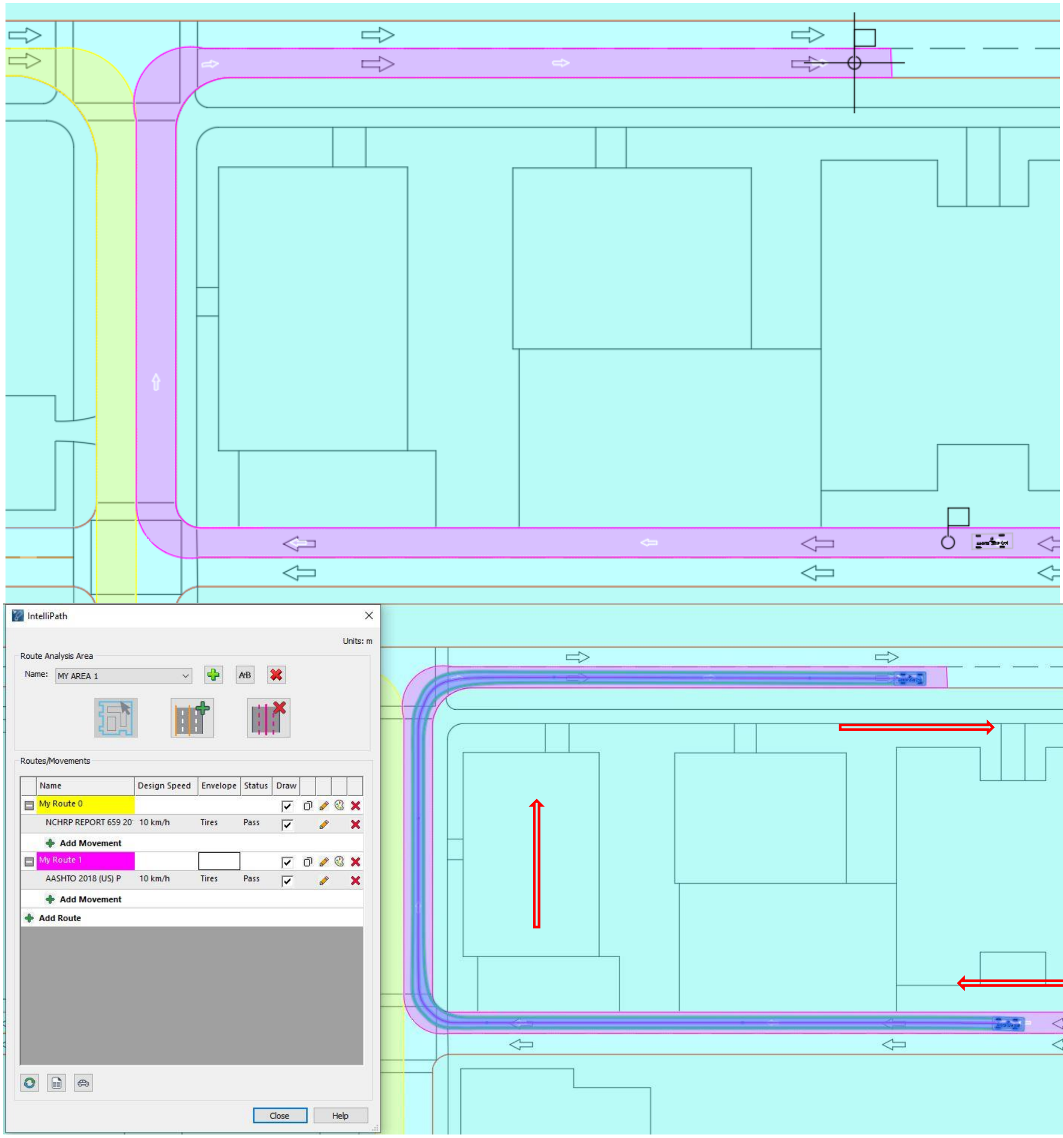


Exhibit 8 – *Generate Movement*

Note: The start and endpoints defined will have a significant impact on the drive paths.

9. After generating both movements, you can observe that both vehicles can use the routes at 10km/h without problems.
Note: The speed, combined with the vehicle's lock-to-lock time (in the case of low-speed maneuvers), is used to calculate the steering rate to generate the movement under comfortable driving behavior.

10. As the second part of this example, we would like to evaluate the maximum achievable speed that a vehicle can make for a particular route. Turn ON the layer/level, "Chicanes" in AutoCAD/MicroStation drawing. Create a new route as shown in Exhibit.9. Edit the route to match the geometry of the curb extensions.

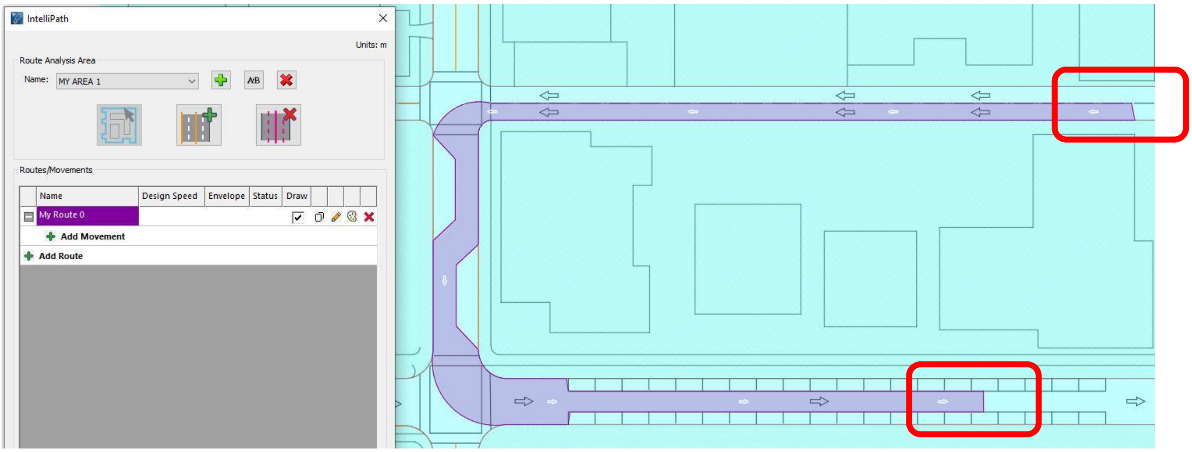


Exhibit.9 – *Create routes*

11. Use the **Add movement** button to generate a movement in the defined route by using the AASHTO 2018 (US) P vehicle. Under the Movement Speed, select the "**Have IntelliPath determine the maximum achievable speed**" option as shown in exhibit 10 and click OK. See exhibit 10

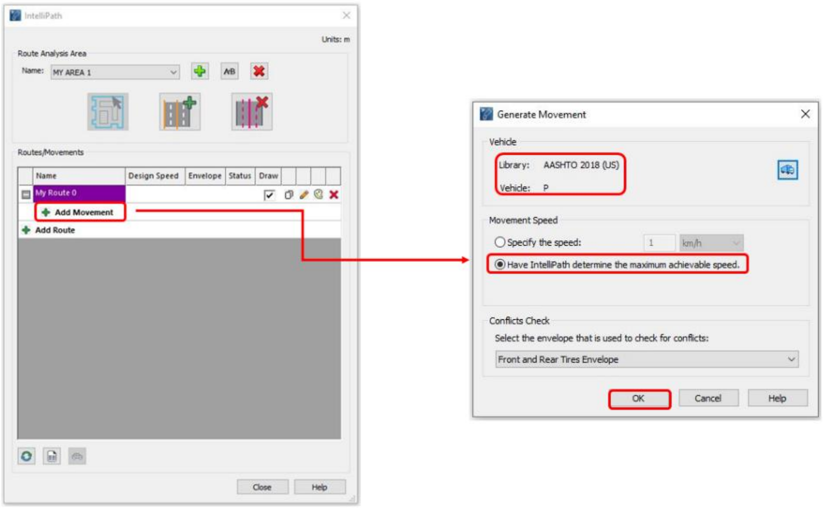


Exhibit 10: IntelliPath – *Maximum achievable speed calculation*

12. Click the **Align vehicle to geometry** option from the **Add Movement** window and pick any pre-existing geometry (line, arc, or polyline) in the drawing and drag the mouse to set the orientation of the vehicle in the desired direction. This aligns the vehicle along the selected geometry pointing to the selected direction. See exhibit 11.
13. Left-click at the desired vehicle starting point and endpoint inside the route. Right-click anywhere to exit the process. The IntelliPath window pops up showing the maximum speed the vehicle can travel within the route. See exhibit 12.

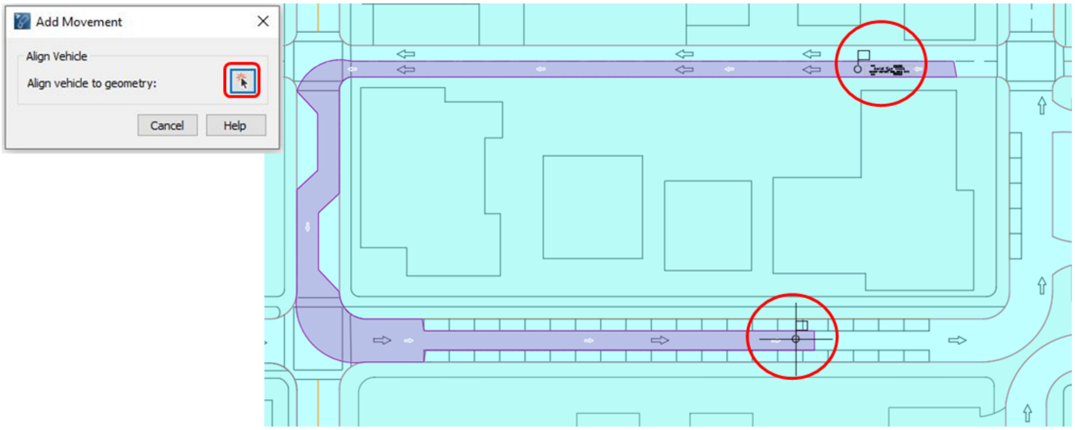


Exhibit 11: Align vehicle to geometry

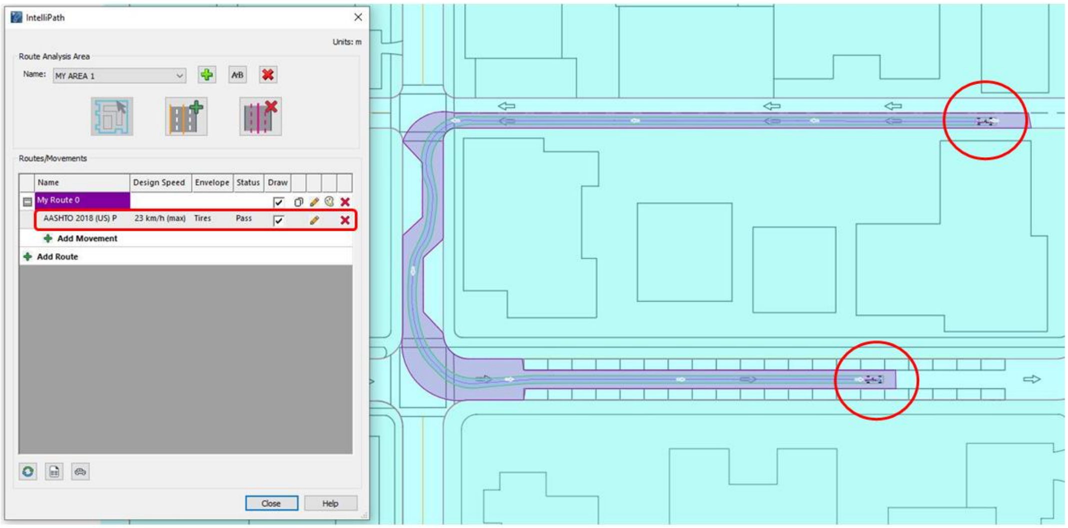


Exhibit 12: IntelliPath – Maximum achievable speed calculation

2D & 3D (AutoTURN Pro Only) Loads

There are cases where oversized loads or vehicle components that are wider or longer than the vehicle body such as wind blades, plows, plows, construction equipment, etc. need to be included as part of the swept path analysis since they will be key to determine the space requirements for both vehicle and cargo.

The ability to place one or more elements or loads to a vehicle part, and produce the envelope described by the load (s), will allow you to assess design scenarios or verifications such as the ones presented below:

Vehicles attachment

- Snowplowing Vehicles (i.e. model the shovel)
- Agricultural machinery

http://www.government-fleet.com/article/story/2012/01/north-dakota-finds-purchasing-solution-for-complex-snow-plow-truck-delivery.aspx
https://pixabay.com/en/agriculture-agricultural-vehicles-1401587/

Oversized Loads

- Wind Blades Transport
- High or Wide Loads

NOTE:

The Add Loads to the Vehicle procedure requires existing CAD geometry in the drawing to represent the load. The geometry needs to be a polyline in AutoCAD or a complex chain, shape, complex shape in MicroStation. The load needs to be drawn in relation to the vehicle part (pointing left) that the load is to be associated with. The load datum (reference point) is the same as the part datum by default (the front, center of the vehicle part) when importing the load.

TRANSPORTING A HYDRAULIC EXCAVATOR

The following scenario will illustrate how to use AutoTURN to create simulations that involve the modeling of loads carried by a vehicle and the envelopes they describe. In this case, we will select the vehicle **NCHRP Report 659 (US) – Low-Boy Trailer** to transport a Hydraulic Excavator. Subsequently, a simulation will be generated to evaluate whether the vehicle and its load, successfully execute the turn without having any conflicts with the existing infrastructure.

More so, to ensure that the vehicle and the load have enough space and no damage is caused to the existing infrastructure, a 3 ft. clearance is required between the load and the adjacent building. See exhibit 1.

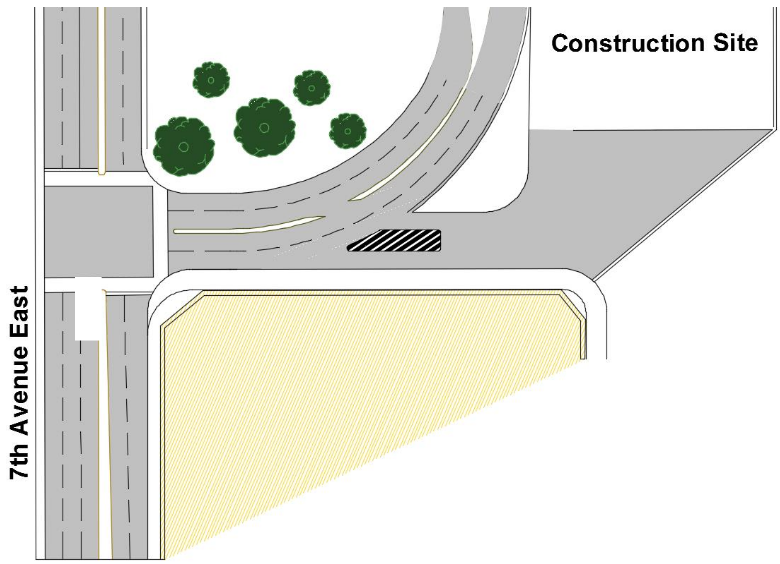
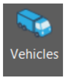


Exhibit 1 – *Transporting a Hydraulic Excavator Scenario*

▶ How to Add a 2D Load

1. Click 

Vehicles and select the **NCHRP Report 659 (US) – Low-Boy Trailer 53 feet**, then click in **View/Edit Vehicle Details**. See exhibit 2.

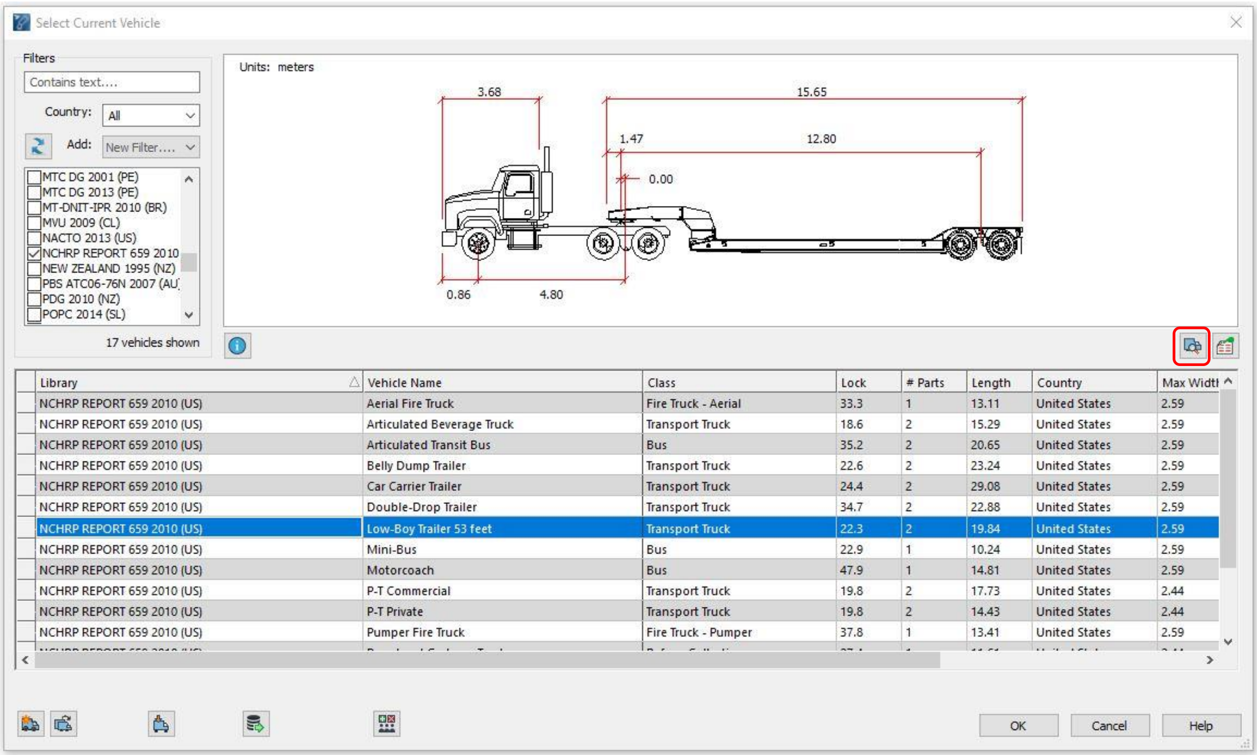


Exhibit 2 – *Vehicles Dialog Box*

2. In the **View Vehicle Details Dialog Box**, click in **Define Loads**. See Exhibit 3.

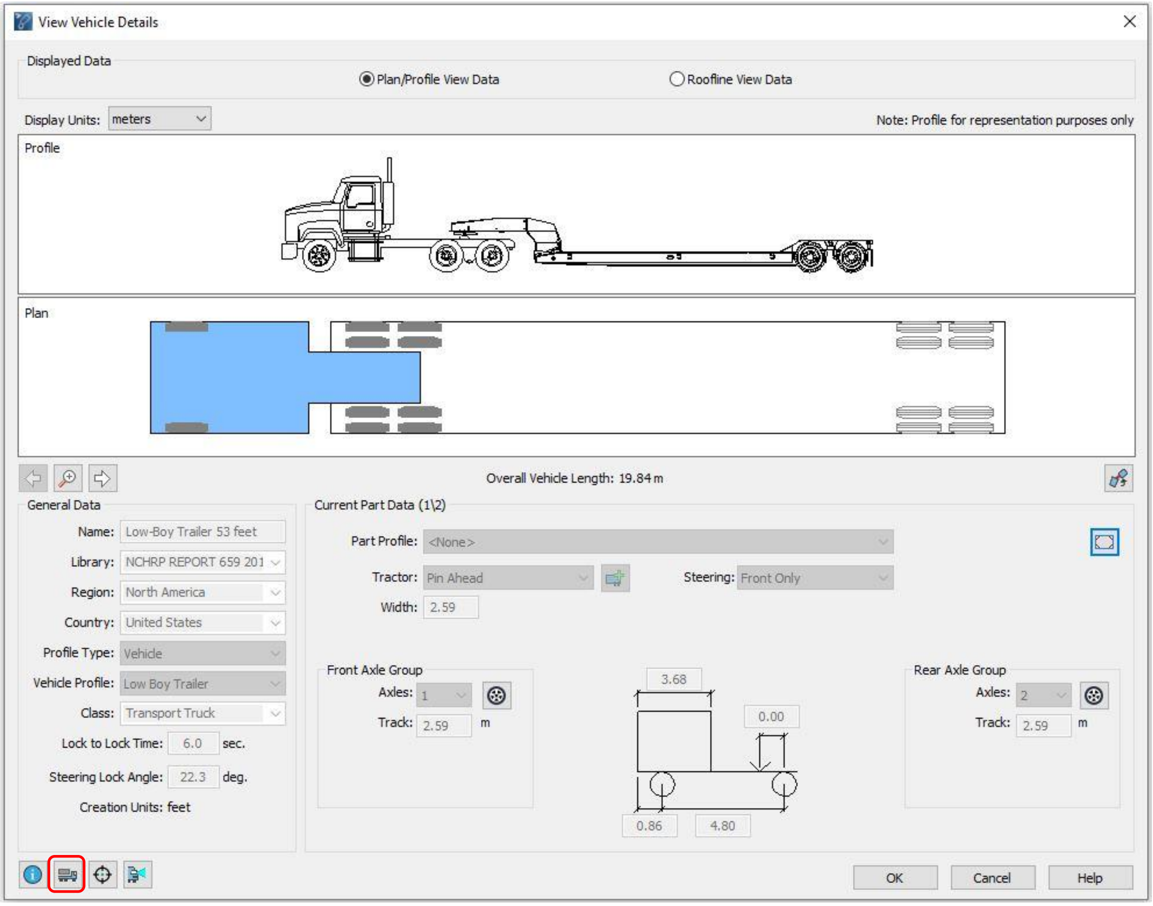


Exhibit 3 – *View Vehicle Details Dialog Box*

3. In the **Define Loads Dialog Box** use the arrows to select the second part of the vehicle, the current vehicle part is indicated by the red triangle located above the center of the vehicle part. Then click in **Import load**. See Exhibit 4.

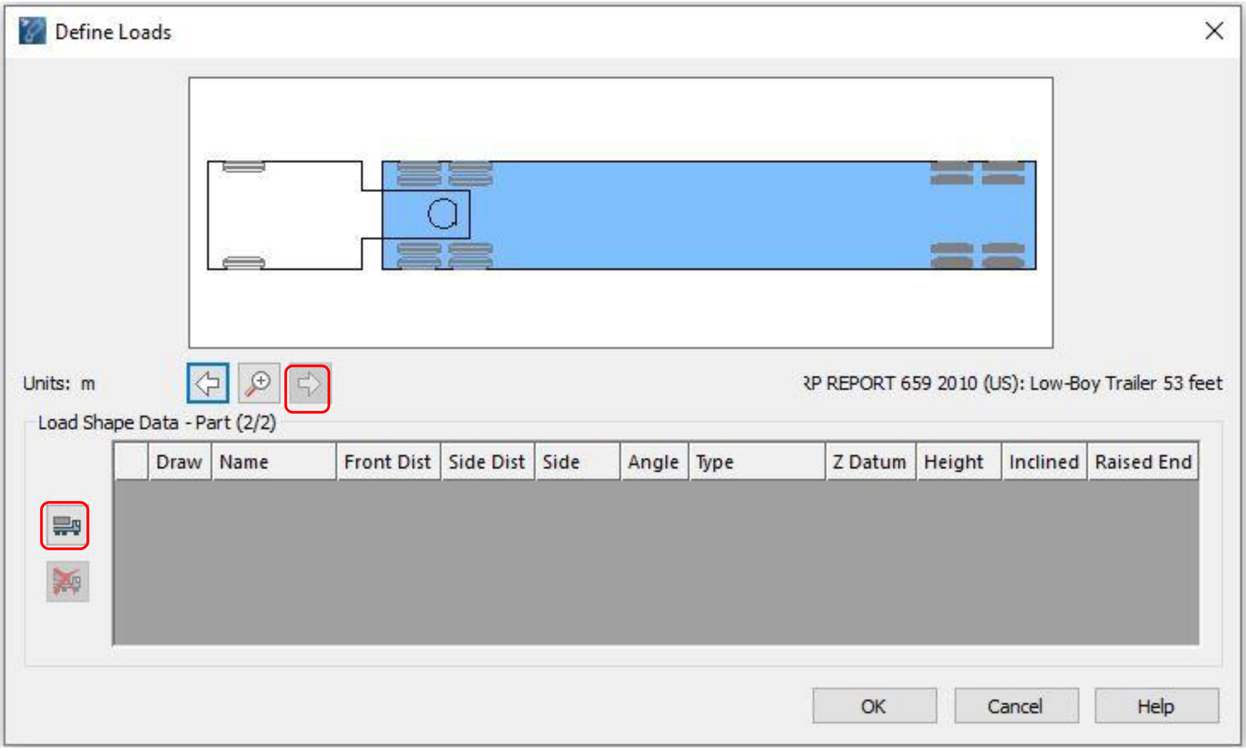


Exhibit 4 – *Define 2D Loads Dialog Box*

4. The outline of the load or the vehicle component should be represented in the drawing by a closed complex chain or polyline depending on the CAD platform. In this example, the geometry represents the outline of a Hydraulic Excavator. See Exhibit 5.

Exhibit 5 – *2D Load - CAT 336D L Hydraulic Excavator*

5. Select/pick the closed polyline representing the load.
6. After selecting the load, you will be asked to indicate (click) a Reference point that will define the position of the load to be attached to the vehicle. See Exhibit 6.

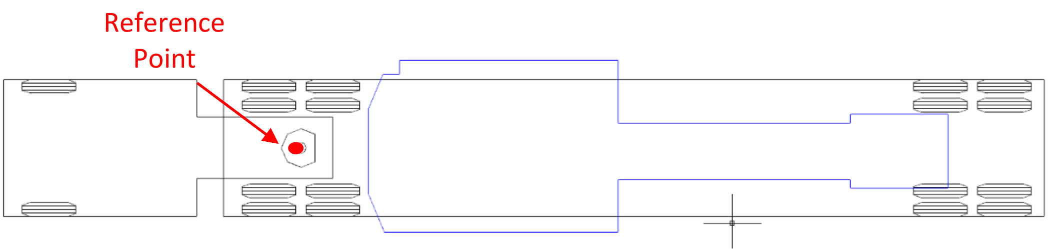


Exhibit 6 – *Define 2D Loads Dialog Box*

7. To use the load to generate envelopes, we need to keep the Draw checkbox selected. Rename the load to "Excavator" and click **ok** to close the Define 2D Loads Dialog Box. See Exhibit 7.

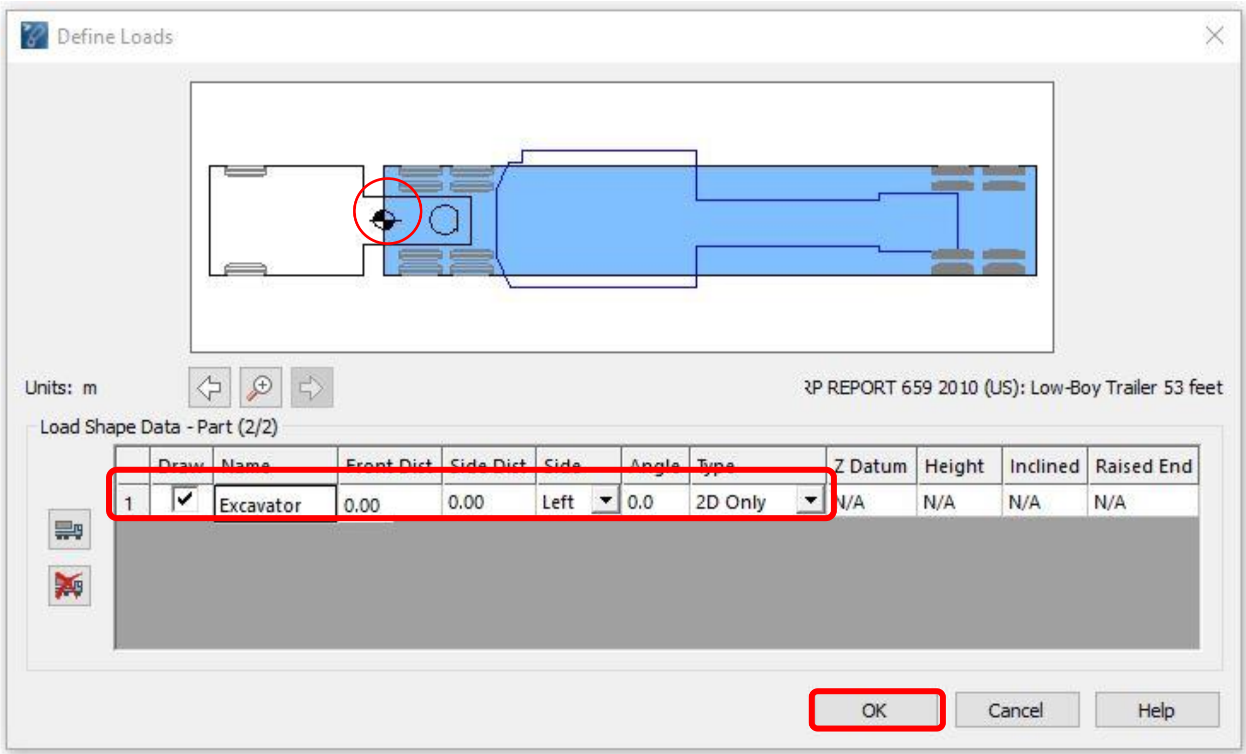


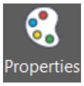
Exhibit 7 – *Define 2D Loads Dialog Box*

NOTE:

It's possible to modify the location of the load by entering the desired values in the Front Distance, Side Distance, and Angle boxes, and specify the desired location of the load relative to a side of the vehicle by selecting the desired side in the Side list.

▶ How to Simulate a vehicle carrying an oversized 2D Load

1. Once the load has been attached properly to the vehicle, it is possible to proceed with the analysis. Note that all the buildings in the drawing are under the 'Buildings' level or layer.

2. Click on the 

Properties tool, and select the Conflict Analysis category. Select and add the 'Buildings' and 'Vegetation' level or layer to be included in the obstacles layer and Click OK. AutoTURN will now report any conflicts between the vehicle envelopes and geometry using these levels/layers. See Exhibit 1.

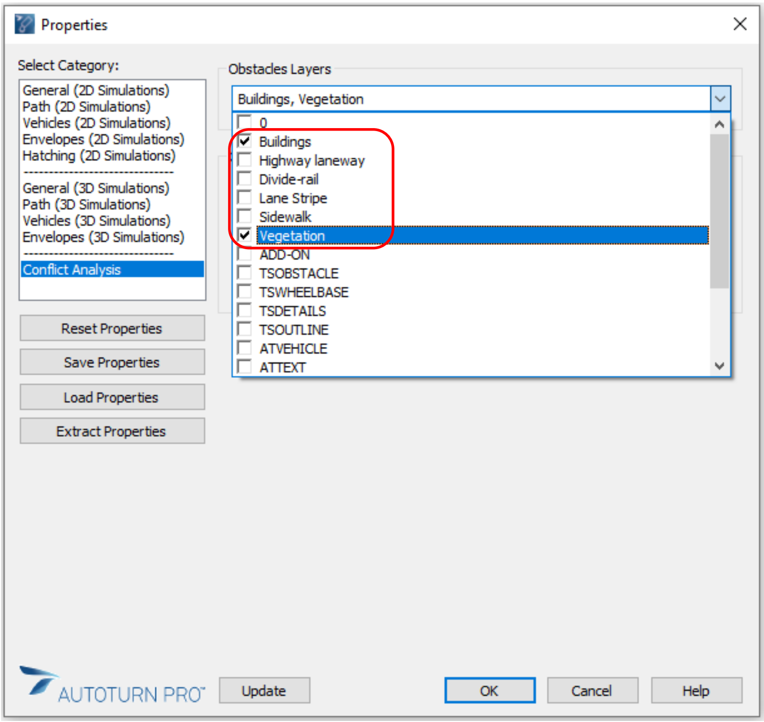
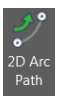


Exhibit 1 – *Obstacles Layer - Buildings*

3. Click 

Generate Arc Path in AutoTURN Pro Ribbon and Click on the 'Align vehicle to geometry' option, select any geometry (line, arc, polyline) in the North-East travel lane on 7th

Avenue in the drawing, and drag the mouse to the North direction to align the vehicle as shown.

4. Click at the bottom of the road to place the vehicle facing north-east. See Exhibit 2.

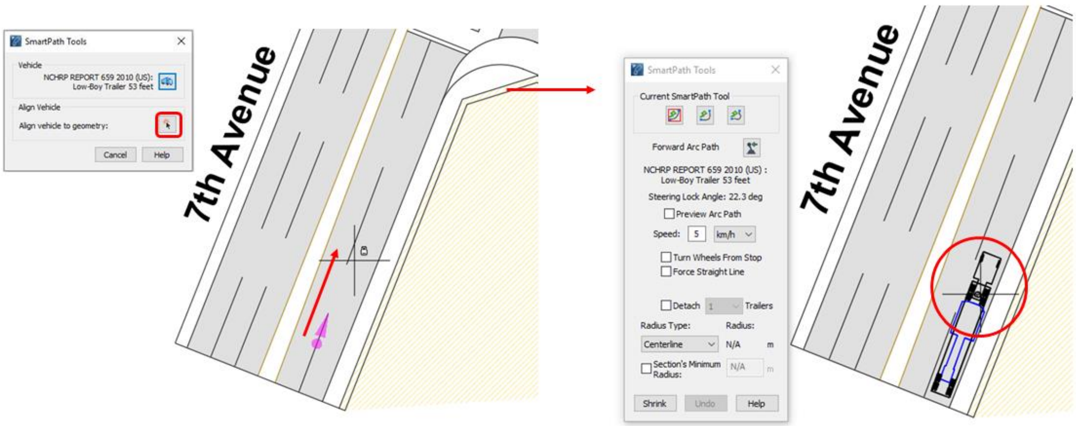


Exhibit 2 – *Placing simulation*

5. In the SmartPath tools, set the speed to 15km/h, generate the first segment of the simulation by moving your mouse in a straight line towards the intersection, and doing a left click before the vehicle gets to the corner. See Exhibit 3.

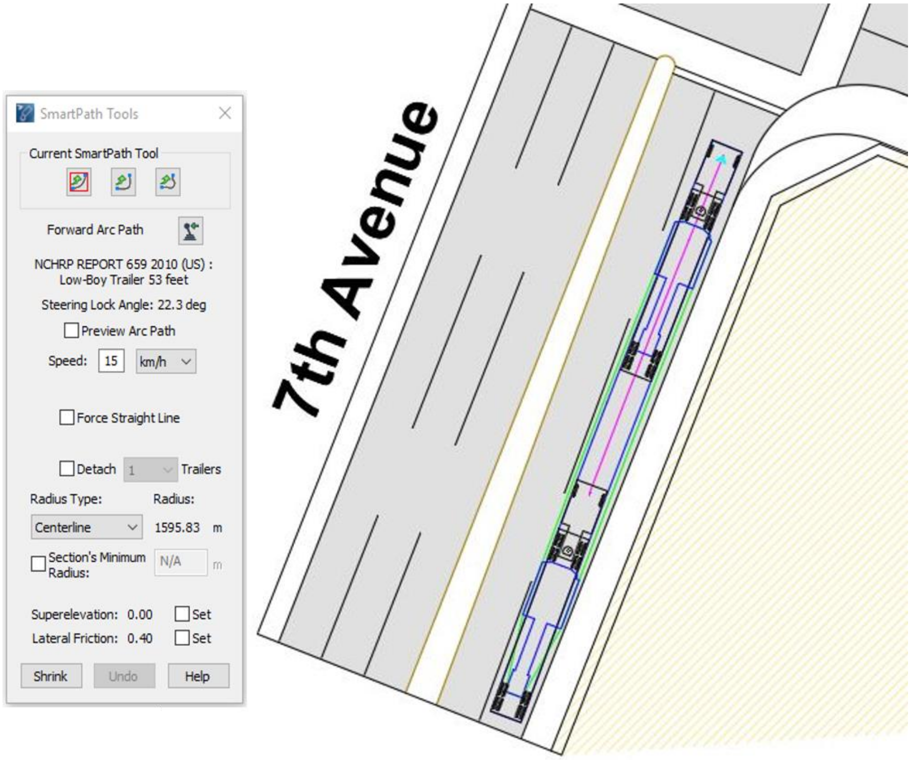


Exhibit 3 – *Generate Arc Path – First segment.*

6. In the dialog, click the Oversteer Corner Path tool. Change the speed to 10km/h, check the "Use Minimum Offset" box, and click on "Select Sweep Angle" to use reference geometry in the drawing to define the desired sweep angle for the simulation. Select one of the lines forming the curb that are at a 90-degree angle with the 7th Avenue. Move your mouse towards the desired end position for the vehicle (South-East) and do a left-click to place the section and a right-click to finish. Note that the software detects an Obstacle Conflict. See Exhibit 4.

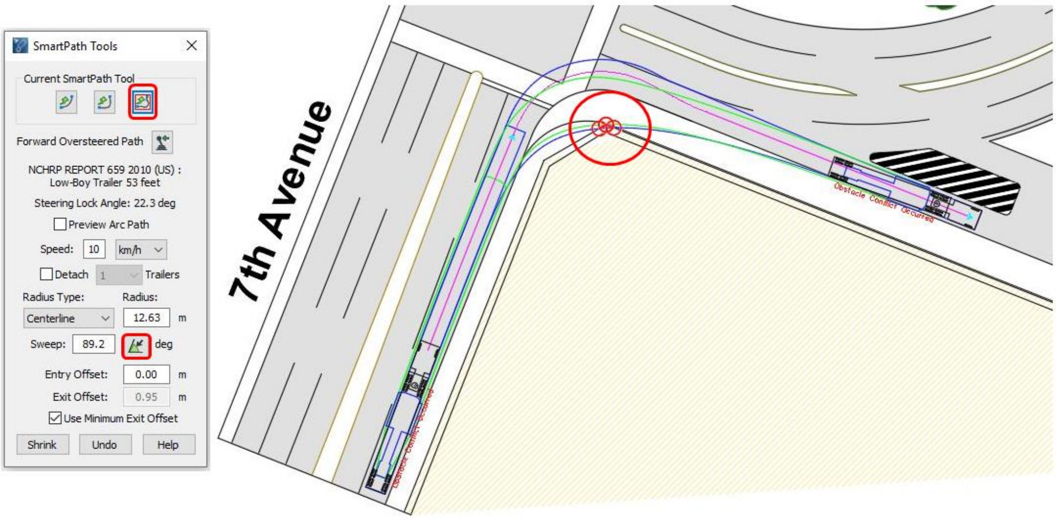
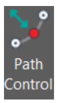


Exhibit 4 – *Oversteer Corner Path*

7. In AutoTURN it is possible to perform small adjustments to the vehicle path that can help overcome similar situations when an obstacle conflict is detected with the envelope. . The Path Control tool will allow you to perform these adjustments without exceeding the turning characteristics of the vehicle nor the minimum turn radius based on the selected speed for each section.

8. Click 

Path Control then select the simulation to adjust the section where the conflict takes place. It is possible to edit the simulation manipulating the nodes.

9. Click the diamond node to adjust the entry and exit offsets of the simulation by simply dragging them in the required direction, so that the clearance envelope set for the load is away from the building. This operation will cause the vehicle to oversteer. See Exhibit 5

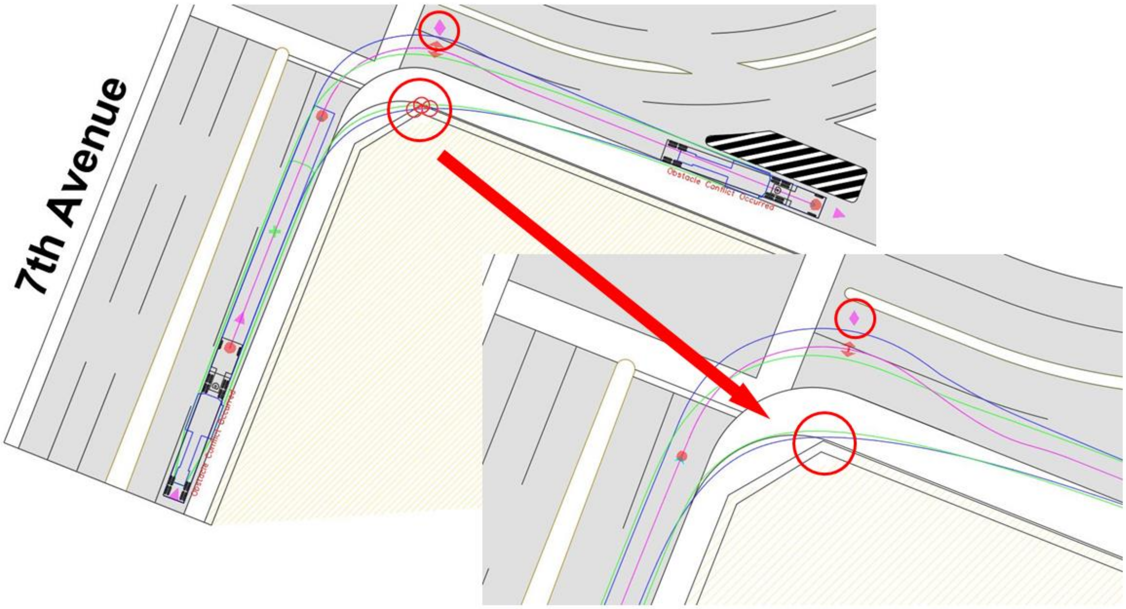


Exhibit 5 – *Adjusting Offsets*

10. Once the envelope is away from the building make a right-click to finish the command. As can be observed both the vehicle and the load can access the construction site while keeping enough clearance to the existing infrastructure. See Exhibit 6.

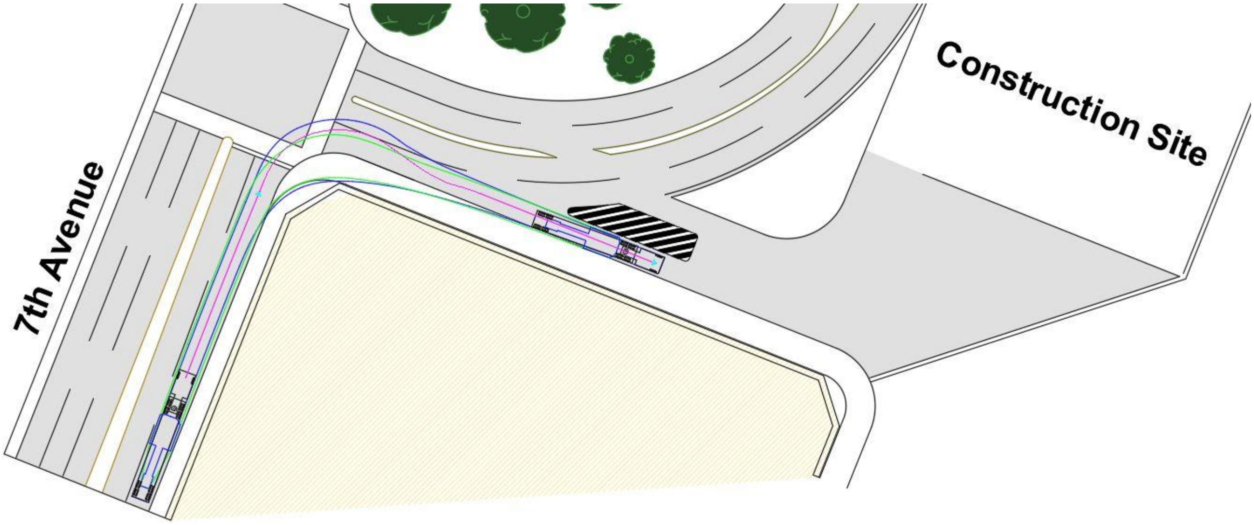


Exhibit 6 – *End of Simulation*

▶ How to simulate a 3D Load

AutoTURN Pro allows you to model 3D loads by extruding an existing 2D load. AutoTURN Pro will then generate the 3D envelope described by the vehicle and its load allowing you to perform a comprehensive clearance analysis and identify potential issues between the vehicle and the terrain or the load and overhead items such as traffic lights, structures, etc.

1. Repeat the steps 1 to 5 from “How to Generate a 2D Load” exercise and then change the type for “2D&3D”. in the **Define Loads** dialog box.

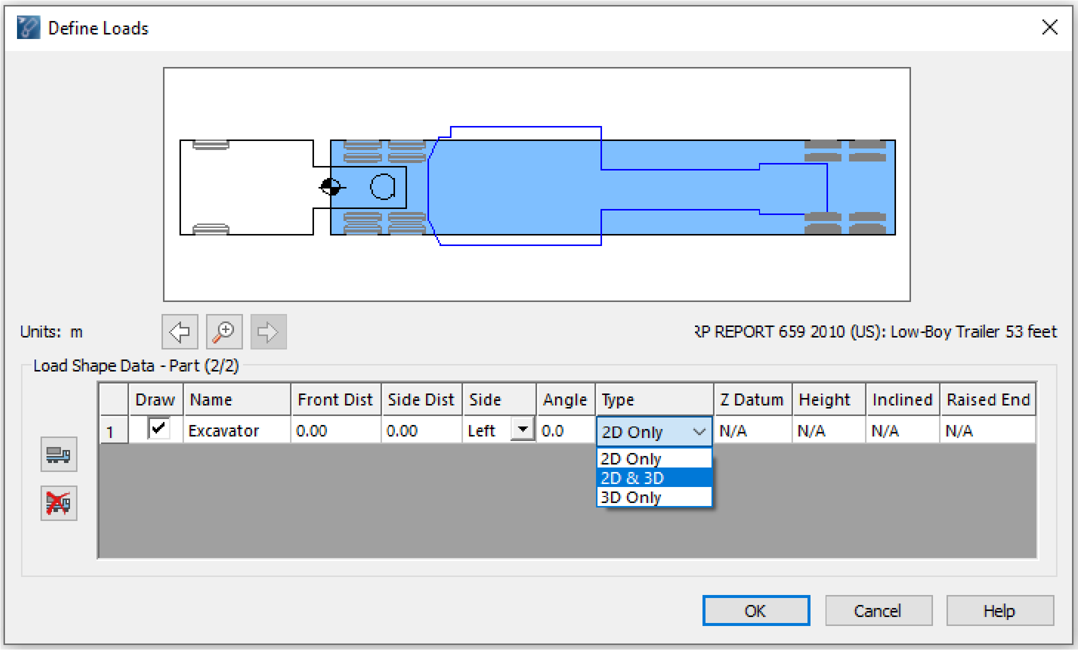


Exhibit 1 – *Select desired options for the loads*

2. The **2D&3D** and **3D only** options enable the Z Datum and Height fields. The Z Datum is the location of the lowermost point of the load (i.e. the distance from the ground to the bottom of the load). To modify the Z Datum, type the desired value in the cell, for this example we will be using 1.6m. Also, type the desired Height for your load (5.0m in the example). You can also provide an inclined height and point of inclination for the load in case the loads rest on the vehicle at an angle. For this example, leave the values as 0.00.

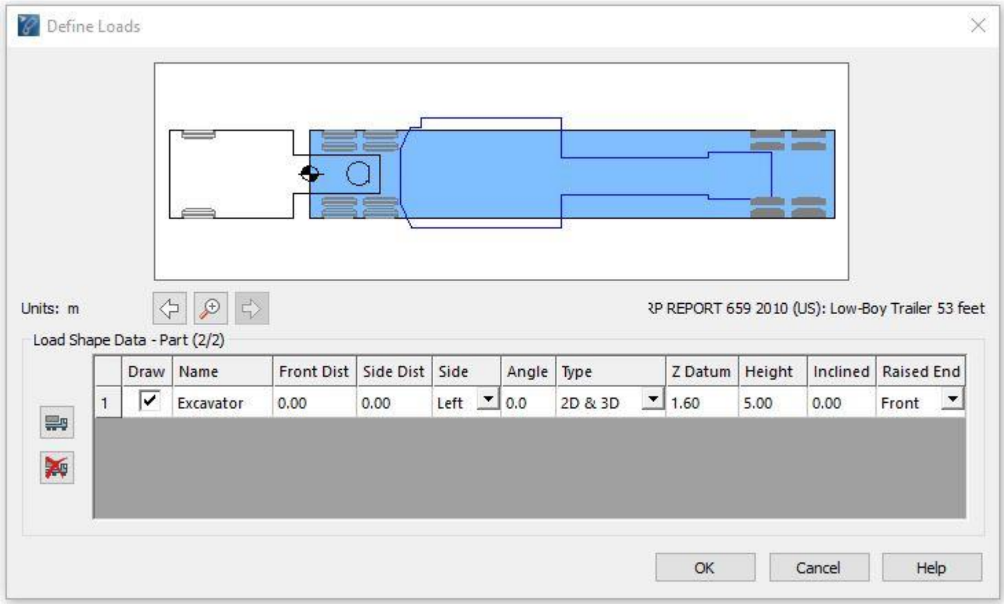


Exhibit 2 – *2D and 3D load option*

3. Click ok to close all dialog boxes and Place a 3D simulation to see the result. The envelope generated now includes the vehicle, the load, the vehicle envelope, and the load envelope. See Exhibit 3.

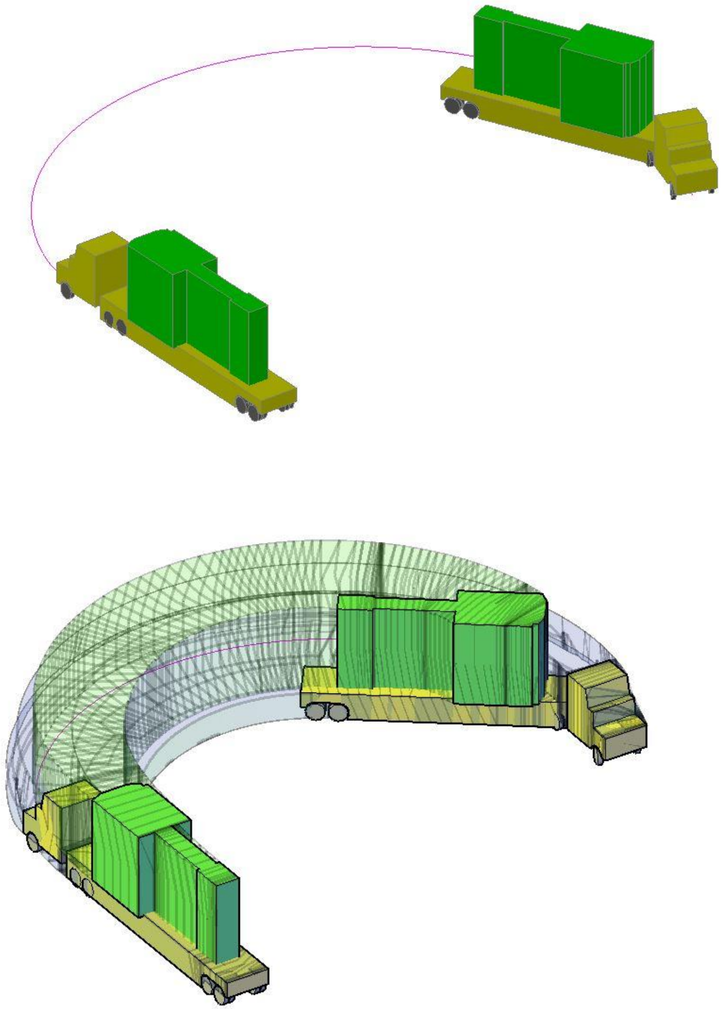


Exhibit 3 – *2D and 3D load extrusion and 3D envelope*

NOTE:

3D Loads feature is only available for AutoTURN Pro users

Summary

Items covered in this section:

- ☐ Add a 2D Load
- ☐ Path Control
- ☐ Add a 3D Load

AUTOTURN

ADVANCED TRAINING

TRANSOFT SOLUTIONS

ADVANCED TRAINING EXERCISES

3D Tracking Points

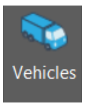
The modeling of complex loads in 3D can be time-consuming if the purpose is primarily for analytical reasons (not visualization purposes). For cases where the modeling of these loads is required, the ability to define 3D tracking points and generate its corresponding envelopes in AutoTURN Pro could allow you to track the most critical points of the load, without the need to model the full vehicle in 3D.

Fire Truck Appliances

In this example, we will simulate the basket attached to the ladder of a Fire Truck by setting up tracking points representing the key points (outermost) of the basket. Afterward, we will perform clearance analysis in 3D to evaluate possible conflicts for the vehicle entering the Fire Department Facilities.

http://www.rosenbaueramerica.com/fire-trucks/aerials/cobra-aerial-platforms

▶ Define Tracking Points

1. Click 

Vehicles and select the NCHRP Report 659 (US) – Aerial Fire Truck, then click in View/Edit Vehicle Details. See Exhibit 1.

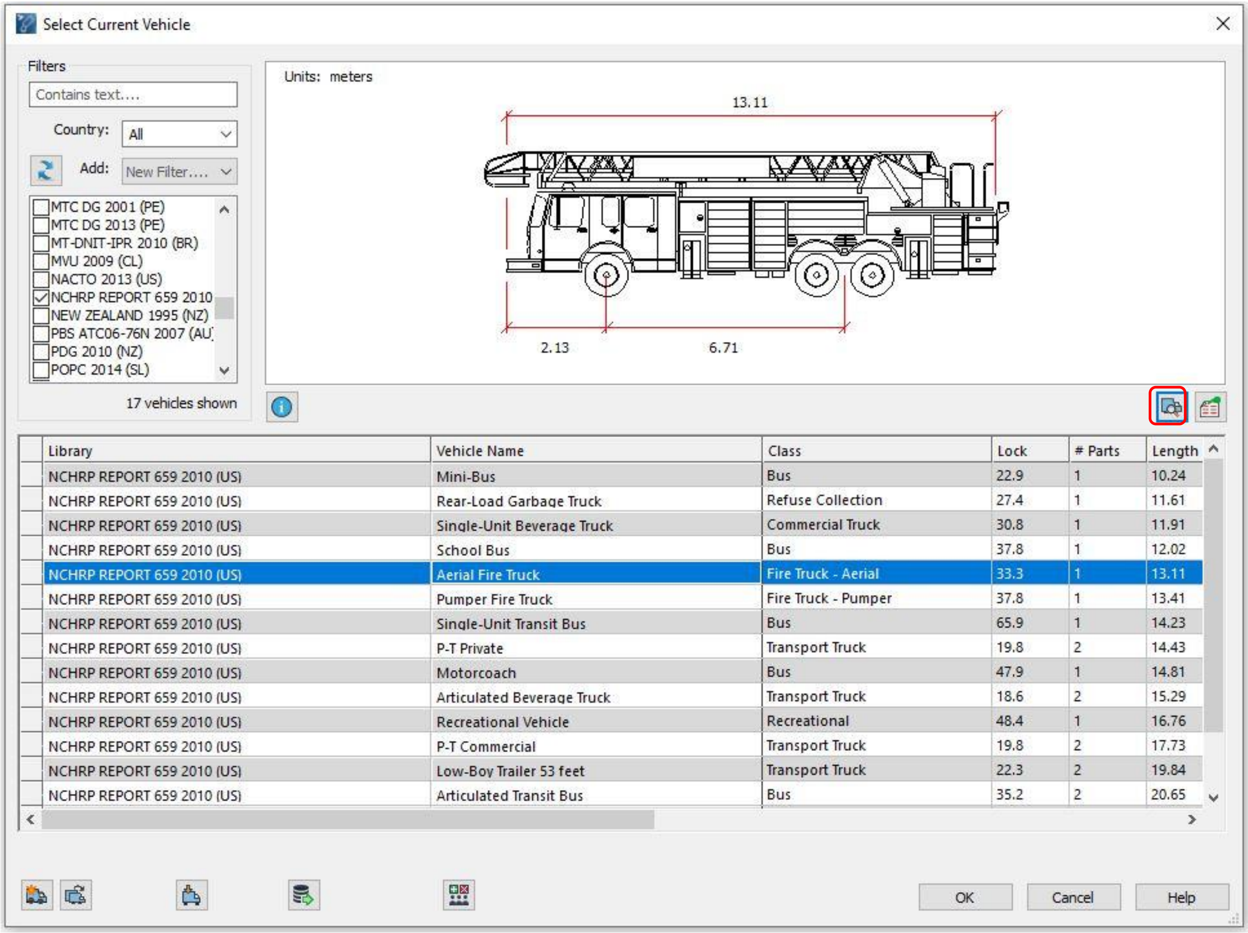


Exhibit 1 – *Vehicles Dialog Box*

2. In the View Vehicle Details dialog box, click in Define Tracking Points. See Exhibit 2.

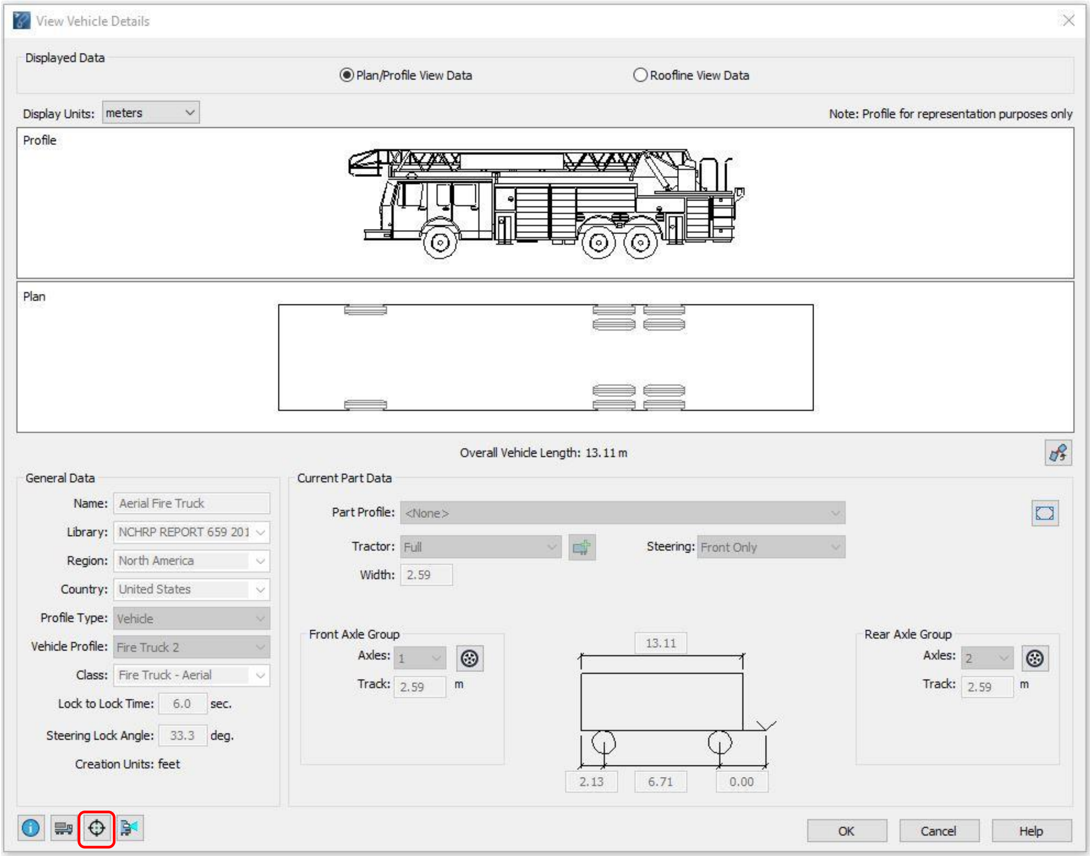


Exhibit 2 – *View Vehicle Details*

3. In the Tracking Points dialog box click in Add Point. See Exhibit 3.

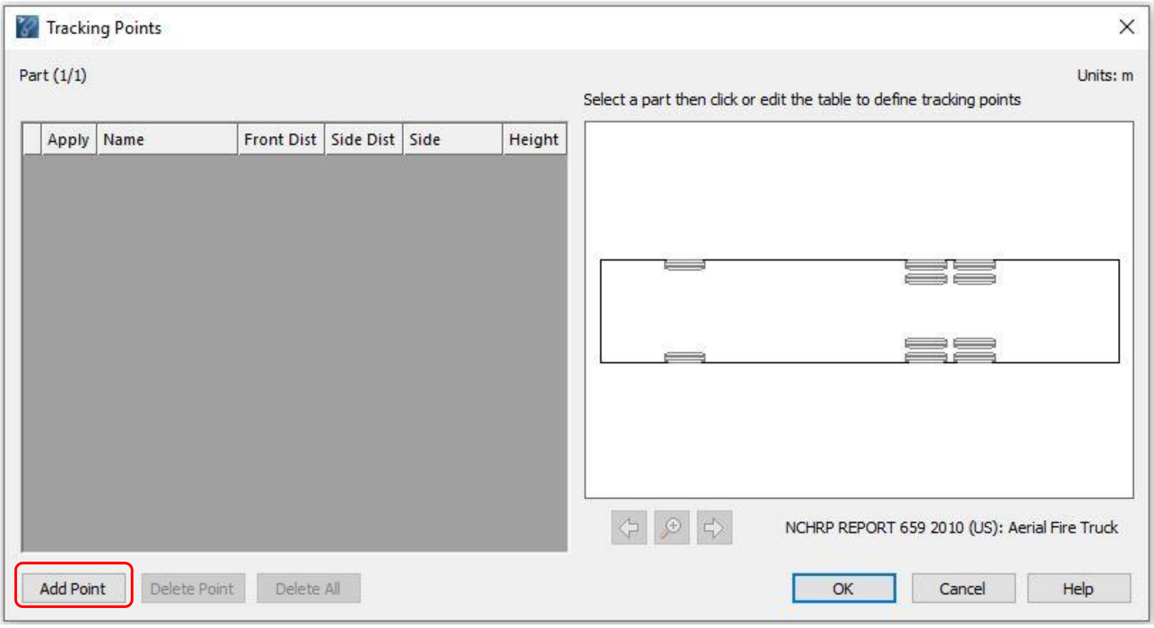


Exhibit 3 – *Tracking Points dialog box*

4. The location of the tracking points, relative to the vehicle part, is defined by entering their corresponding distances concerning the front and side of the vehicle and its height to the ground. The side option will define the position of the tracking point in relation to the longitudinal axle at the center of the vehicle part. The tracking points can also be named by entering it in the name section of the dialog.
   3D tracking points can be displayed or hidden by checking the apply checkbox, as depicted in Exhibit 4.

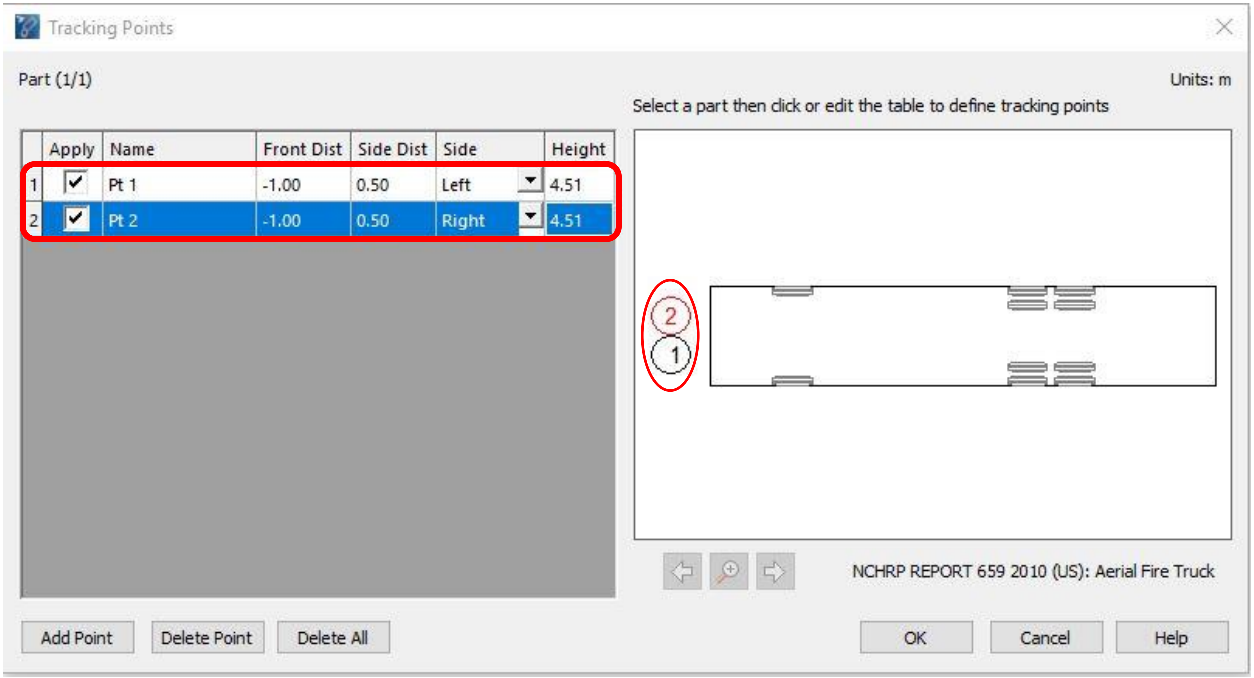
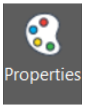


Exhibit 4 – *Add Tracking Points*

▶ How to Perform clearance analysis using 3D Tracking Points

1. In  **Properties**, select the Conflict Analysis category. Select Door Lintel as the "Obstacles Layer". See Exhibit 1.

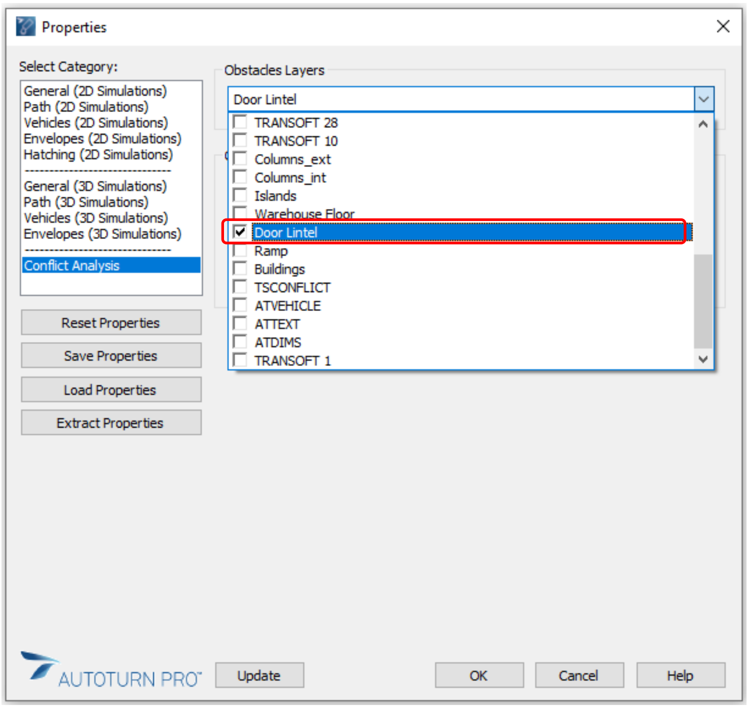


Exhibit 1 – *Obstacles Layer – Door Lintel*

2. Select the Envelopes 3D category, check the Tracking Points box only, and click ok to close. See Exhibit 2.

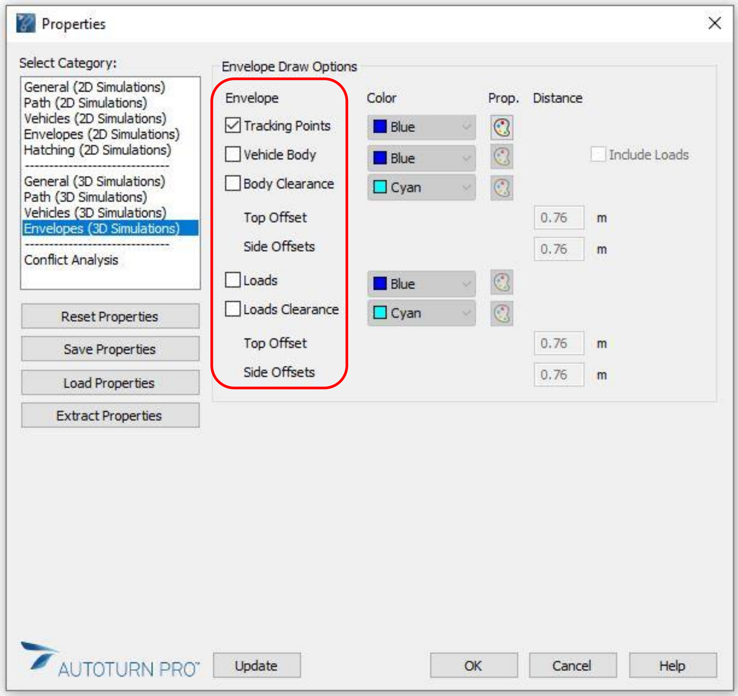


Exhibit 2 – *Envelope draw options – Tracking Points*

3. Click in **Select Terrain** button in AutoTURN Pro Ribbon bar, and select the terrain to generate the simulation on top of it. See Exhibit 3.

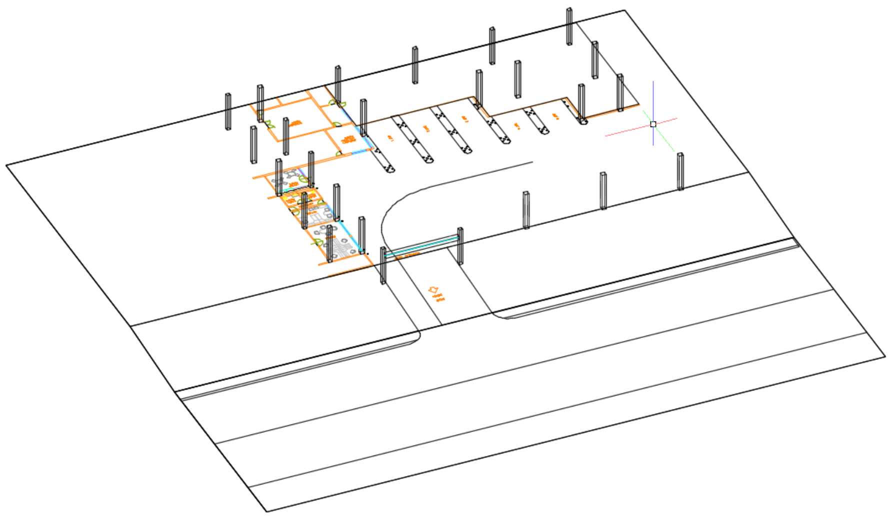


Exhibit 3 – *Select Terrain*

4. Then, click the **Place 3D Simulation** button and select the indicated path. Note the conflicts occurring at the lintel due to the basket height, and on the surface as the rear overhang of the vehicle hits the surface due to the ramp's inclination. See Exhibit 4.

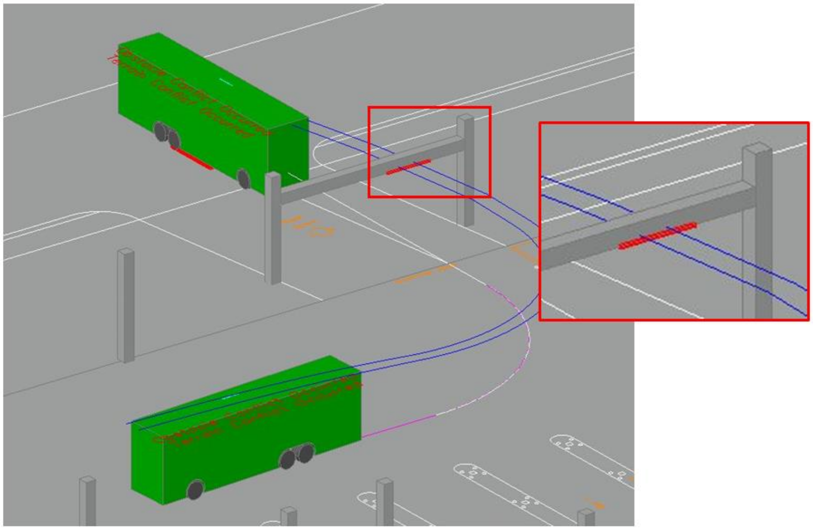


Exhibit 4 – *Place 3D Simulation*

AutoTURN Pro 3D Creation Tools

Vehicle clearance analysis is one of the key verifications to help prevent costly damages to vehicles and infrastructure. The unique 3D vehicle envelope generated by the program will help you to quickly identify any potential issues between the vehicle and the terrain and or overhead elements protruding the envelope.
In AutoTURN Pro there are 3 different methods for generating 3D simulations.

Method 1 – Generate 3D Arc Path
This method allows you to generate a 3D path on a terrain that follows an arc from the start position to the position of the mouse.

Method 2 – Place 3D Simulation
This method allows the user to project lines, arcs, and polylines onto the 3D terrain. This path represents the vehicle's centerline path.

Method 3 – Convert 2D to 3D
This method allows the user to generate the vehicle's swept path in 2D, and then project the simulation onto the 3D terrain.

▶ Generate 3D Arc Path

This method allows you to create the turn simulation directly in 3D. The following exercise explains how to generate a 3D vehicle swept path simulation using Method 1.

1. Click  **Vehicles** on the standard toolbar and select the **City Bus** from the **TRANSOFT** design vehicle library Note the 3D vehicle icon on the bottom right corner. This icon only appears for the vehicles that are supported in 3D. See Exhibit 1.

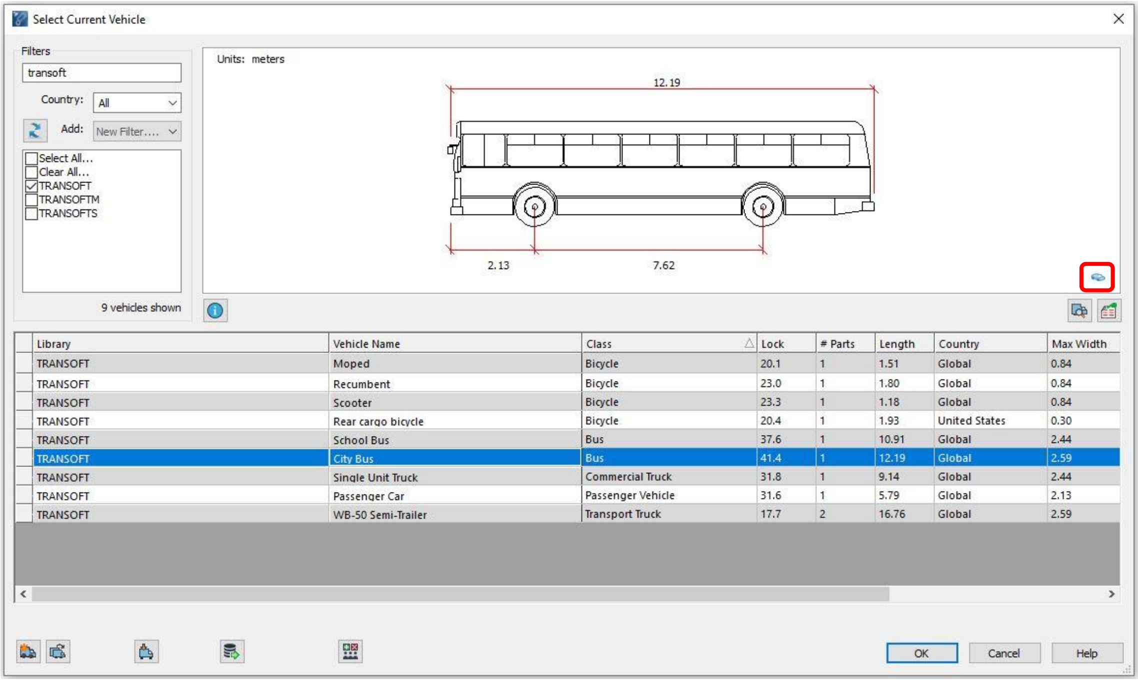


Exhibit 1 – *Select Current Vehicle Dialog Box*

2. Click  **Select Terrain**, and then select the 3D terrain.

3. Click  **Generate 3D Arc Path**, and then place the 3D City Bus starting position on W 120th Street. See Exhibit 2.

Exhibit 2 – *Design Vehicle Starting Position*

4. Drag the vehicle forward to generate the 3D Simulation. Left-click to define each arc segment.

5. Navigate the bus into the roundabout and circulate the vehicle towards N 88th Ave, and then enter onto N 88th Ave. Complete the turn and right-click the mouse to end the simulation. See Exhibit 3

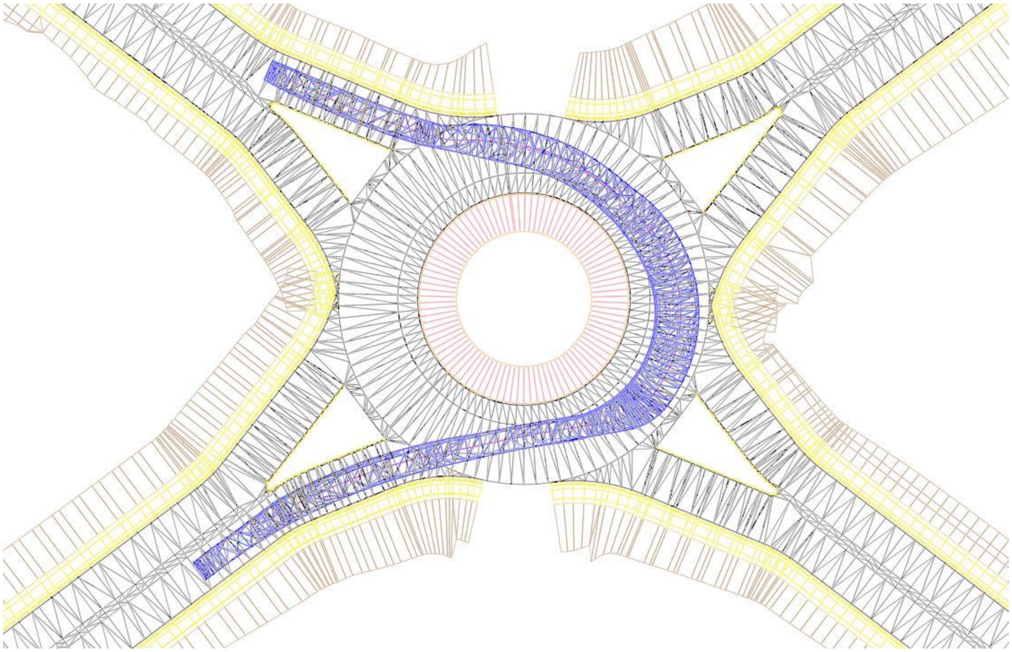


Exhibit 3 – *3D Arc Path/Swept Path Simulation*

6. Change the window view to an SE isometric – realistic view to accurately view the 3D simulation created. See Exhibit 4.

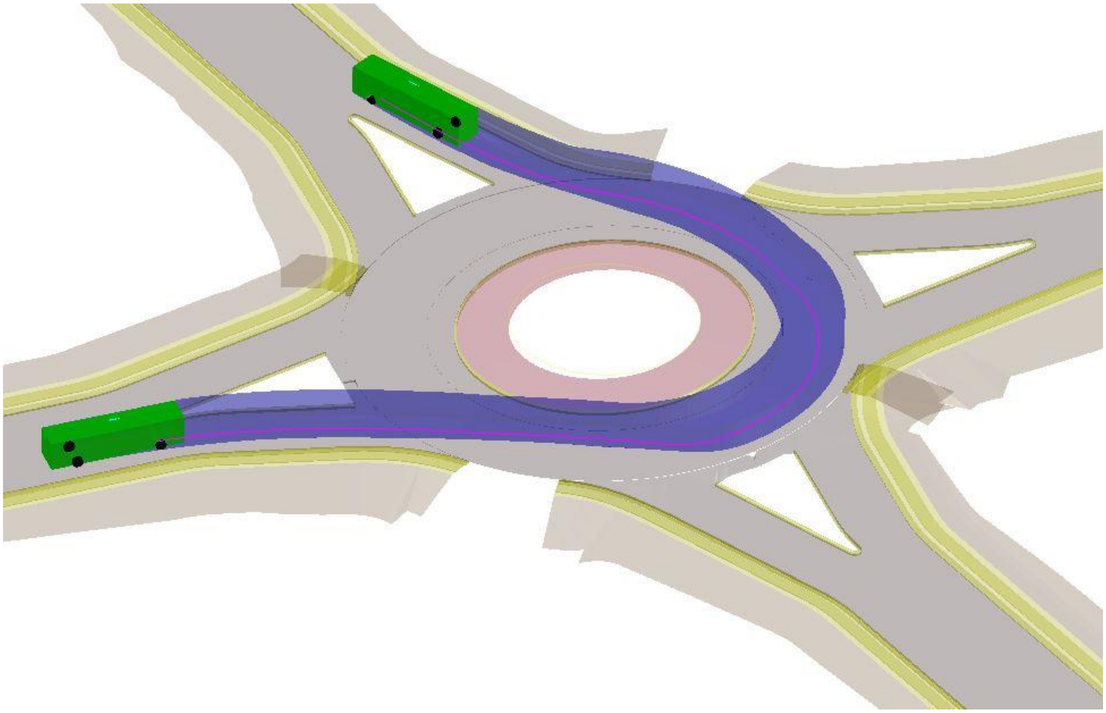


Exhibit 4 – *SW Isometric Realistic View*

Place 3D Simulation

The Place 3D simulation method will allow you to use a 3D geometry as a reference to produce 3D simulations. This tool generates the movement at a very low speed. Only forward maneuvers can be created using this tool.

The following exercise describes how to generate a 3D swept path by using the Place 3D Simulation tool. The selected element(s) represents the vehicle's centerline path.

1. Change the current view to the SE Isometric view.
2. Click **Place 3D Simulation**, and select the geometry (magenta polyline) near the point (1) as shown in Exhibit 1 and 2

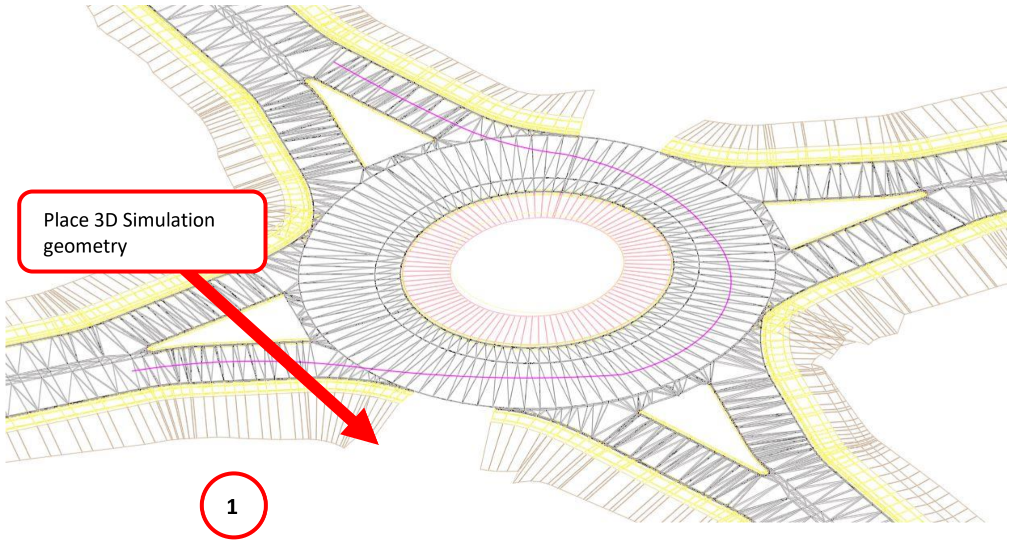


Exhibit 1 –*Starting Position of the Simulation*

TRANSOFT SOLUTIONS

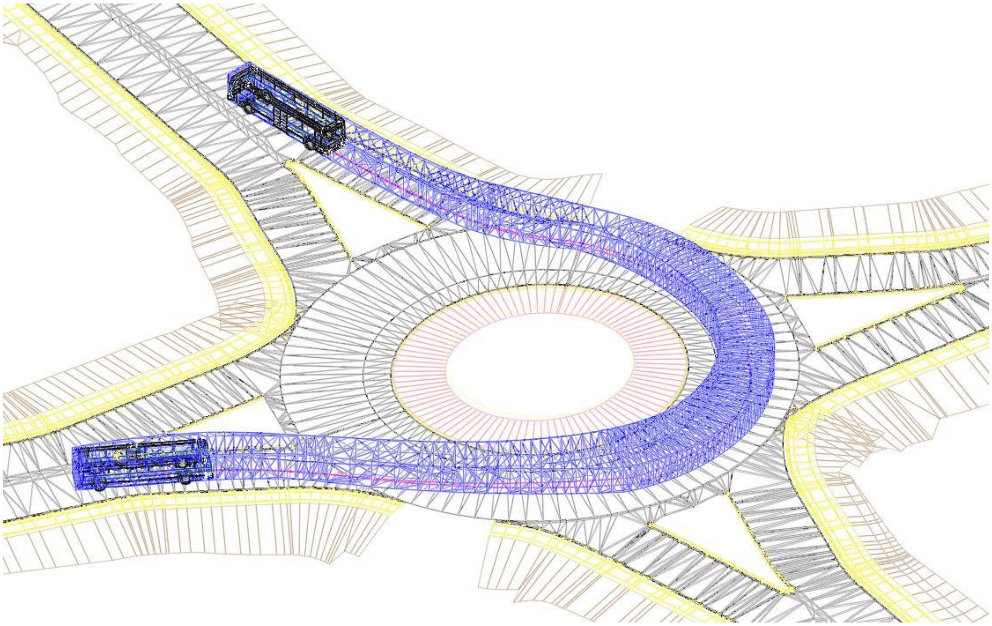


Exhibit 2: Completed Simulation

▶ Convert 2D to 3D

Users can use existing 2D simulations to generate 3D simulations (no need to recreate the simulations in 3D) as well as have all the power of the smart path tools (e.g. corner & oversteer, path control, etc.) to generate the turning maneuver. The following exercise illustrates how to convert 2D swept path simulations into a 3D simulation using the Convert 2D to 3D tool.

1. Click **Generate Arc Path** on the standard toolbar. Place the Bus vehicle on the right turn lane on W 120th Street and using the **Arc Path** tool, navigate the Bus towards the roundabout, and make a left circulatory turn to N 88th Avenue. See Exhibit 1

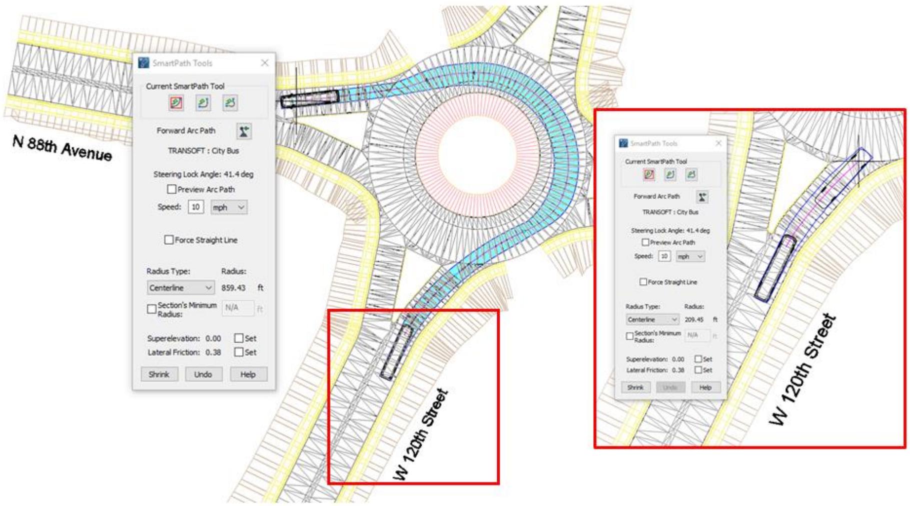


Exhibit 1– *Design Vehicle Navigation*

2. Change the view to a SE Isometric view.
3. Click **Convert 2D to 3D** on the AutoTURN Pro 3D toolbar and select the 2D Simulation.

4. Once the 2D simulation has been converted to 3D, a prompt will appear. Click **Yes** to delete the 2D simulation. See Exhibit 2.

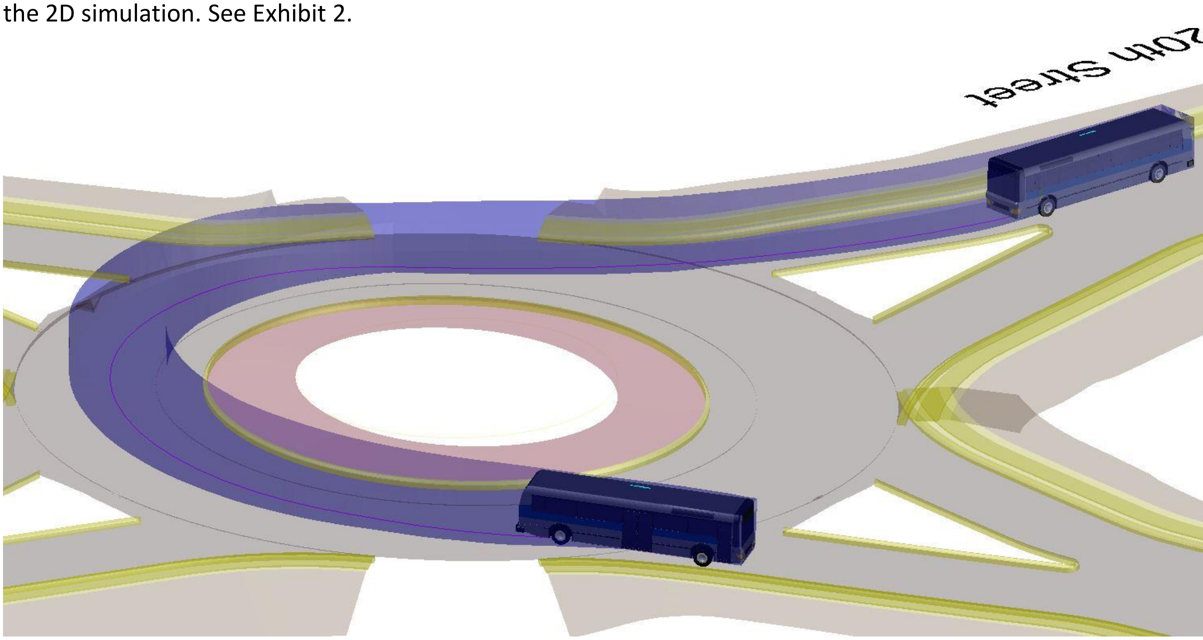


Exhibit 2 – *3D simulation created using Convert 2D to 3D tool.*

NOTE:
Once the vehicle 3D swept path has been generated, AutoTURN Pro 3D will end the simulation at a point if the vehicle's pitch, roll (two-part vehicles) has been exceeded.

Summary

Topics covered in this section:

- ☐ 3D vehicle selection and understanding vertical profile information
- ☐ Defining 3D terrain

Generating a 3D Swept Path Simulation

- ☐ Generate 3D Arc path
- ☐ Place 3D Simulation
- ☐ Convert 2D to 3D

Visualization Tools

In Addition to the 3D reporting tools, visualization tools in AutoTURN will help your presentations to clients or other stakeholders. These tools are **View Punch Through, Place 3D Vehicles,** and **Run Animation**.

▶ View Punch Through

The **View Punch Through** tool sets up a sectional view for analyzing conflicts. This tool is designed to be used in conjunction with the **Run Animation** and **Place 3D Vehicle** tools to aid in visualizing the 3D turn simulation and how it interacts with obstacles and the terrain.

▶ Configuring the Punch Through View

NOTE:

- This tool requires an existing 3D turn simulation and crossing line element (where the punch through is desired) in the drawing.
- Ensure that at least two viewports/windows are configured:
  - In AutoCAD, use the VPORTS command.
  - In MicroStation, use Window -> Views.
- To name and save views for later use:
  - In AutoCAD, use View -> Named Views.
  - In MicroStation, use Utilities -> Saved Views

In the following exercise, we will use the **View Punch Through** function to view a 3D simulation and its interaction with the terrain. Once the viewports are configured, proceed with the following steps.

1. Open the **Road-Rail Crossing 3D** drawing.
2. Click  **View Punch Through** on the AutoTURN Pro 3D toolbar.
3. Select the 3D simulation that was generated with the **Generate 3D Arc Path** tool. Once selected, AutoTURN will prompt for the user to select the line element that crosses the simulation.
4. Select the line element that was used for the previous exercise. See Exhibit 1.

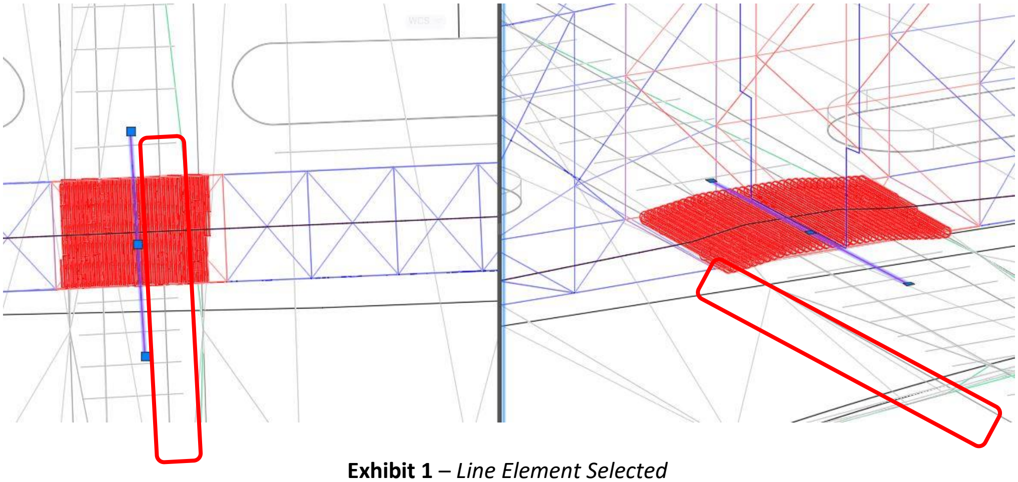
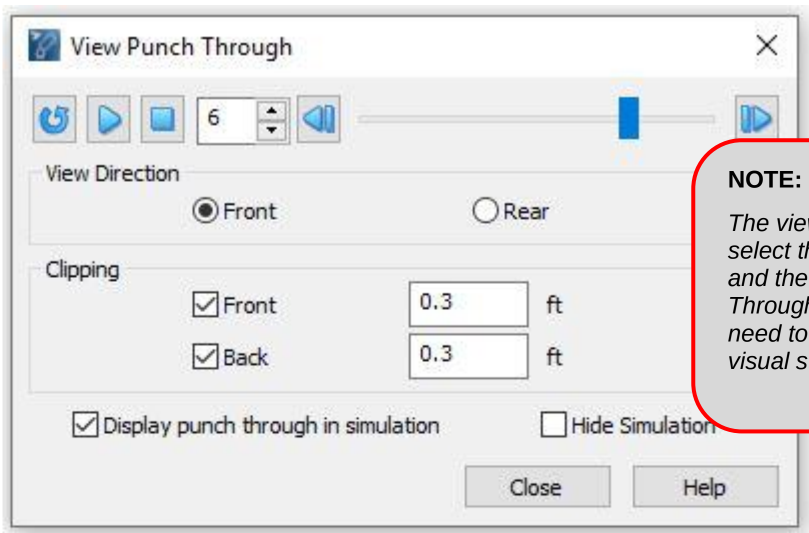


Exhibit 1 – *Line Element Selected*

5. Configure the options in the **Configure Punch Through** dialog box as desired and visualize the simulation using the animation's buttons. See Exhibit 2.

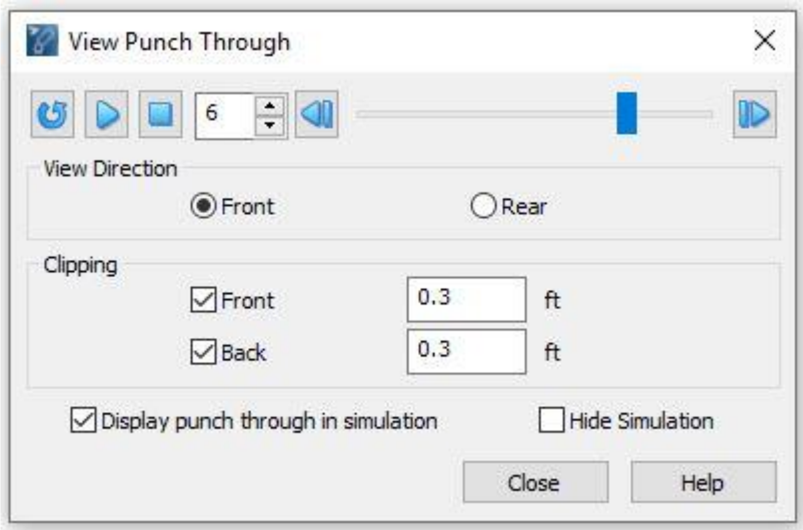


NOTE:

The view in which you select the simulation and the Punch Through view both need to be using a 3D visual style.

Exhibit 2 – *Configure Punch Through View Dialog Box*

Configure Punch Through View Dialog Box Explained



Configure Punch Through View Dialog Box

View Direction

Select **Front** to view the punch through from the end of the simulation (the direction towards the last vehicle position in the simulation). Conversely, click **Rear** to view the punch through from the beginning of the simulation (the direction towards the first vehicle position in the simulation).

Clipping Section

Front

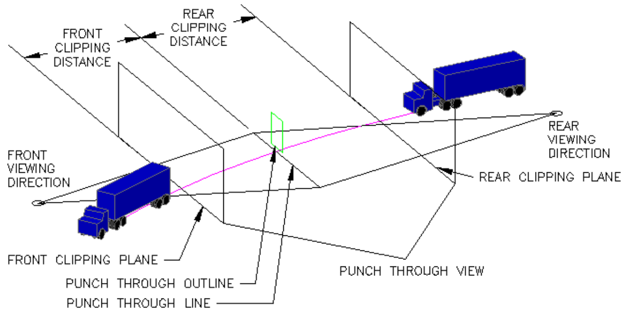
Select this checkbox, and then specify the distance from the selected punch through-line to the front clipping plane.

Back

Select this checkbox, and then specify the distance from the selected punch through-line to the rear clipping plane.

Display Punch Through in Simulation

Select this checkbox to draw the punch through the outline in the simulation at the location of the selected punch through-line.



Punch Through View Configuration Options Illustration

▶ Place 3D Vehicle

The **Place 3D Vehicle** tool makes it possible to place the vehicle at intermediate positions along the simulation. This will allow you to visualize the vehicle at specific positions that might be key or relevant for your analysis. If there is a realistic 3D vehicle drawing for the vehicle, it will be used if the selected 3D turn simulation was generated/placed with the **Realistic** option selected.

The following exercise describes how to place 3D vehicles along a selected 3D turn simulation. For this exercise, we will use the simulation that was generated using the **Convert 2D to 3D** tool example.

Convert 2D to 3D Simulation

1. Click **Place Vehicle** on the AutoTURN Pro toolbar.
2. Select the simulation that was generated with the Convert 2D to 3D tool

3. Move the 3D vehicle along the path, and then click to place the vehicle roughly at the locations shown in Exhibit 1.

4. Right-click to end the command. See Exhibit 1.

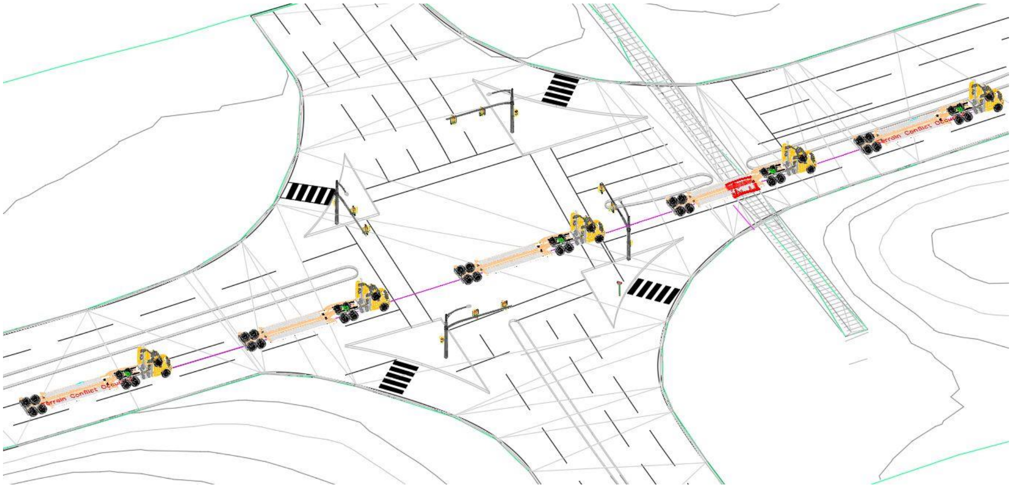


Exhibit 1 – *3D Vehicle Placement*

NOTE:

This tool can be used as an alternative to the ***Run Animation*** *tool for visualizing vehicle movement along a 3D turn simulation. Simply move the pointer backwards and forwards along the 3D turn simulation to visualize the movement, and then right-click when you have finished.*

Delete 3D Vehicle

The **Delete 3D Vehicle** tool deletes 3D vehicles in 3D turn simulations that were placed using the **Place 3D Vehicle** command.

The following exercise describes how to delete placed 3D vehicles from the previous exercise.

1. Click  **Delete 3D Vehicle** on the AutoTURN Pro 3D toolbar.
2. In the 3D simulation, pick a point inside the wheelbase of any 3D vehicle part to delete it.
3. Continue deleting 3D vehicles as desired.
4. Right-click to end the command. See Exhibit 1.

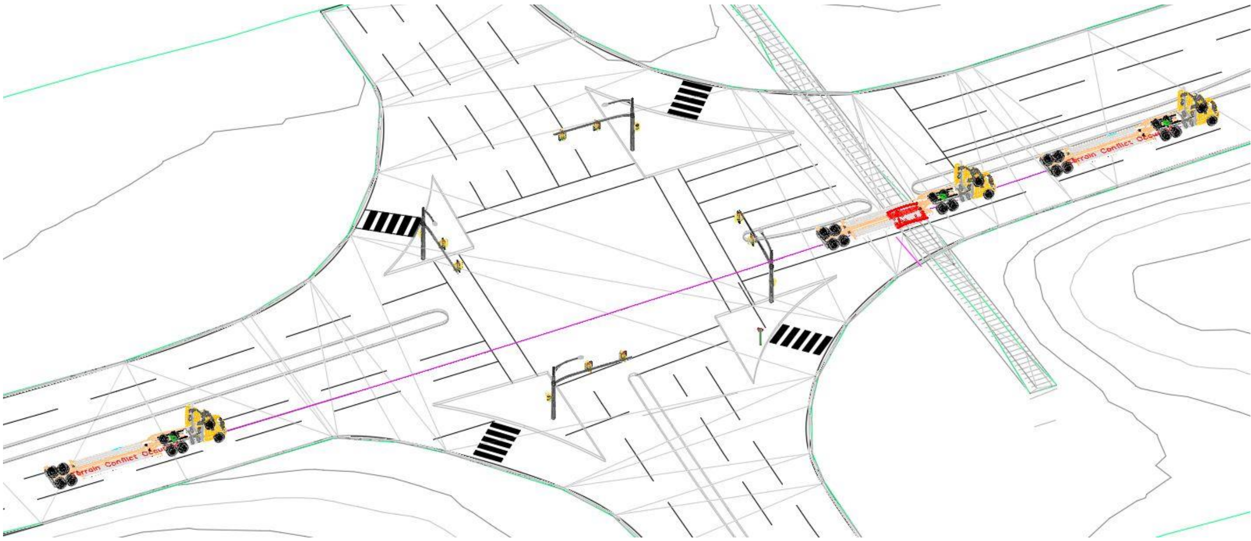


Exhibit 1 – *Deleted 3D Vehicles*

Run Animation

The **Run Animation** tool runs an animation of the current vehicle on the selected path elements or runs an animation of the vehicle used in the selected simulation.

1. Click **Run Animation** on the standard toolbar.
2. Select the 3D simulation from the previous exercise.
3. Click **Play Animation** to simulate the vehicle movement.
4. Click **Close** to end the command.

NOTE:

The animation playback speed does not reflect the actual speed of the design vehicle.

Summary

Using Visualization Tools

- ☐ View Punch Through
- ☐ Place 3D Vehicles/ Delete 3D Vehicles
- ☐ Run 3D Animation

AutoTURN Pro 3D Reporting and Visualization Tools

Viewing 3D Envelopes

To perform vehicle clearance analysis, the unique 3D vehicle envelope in AutoTURN Pro allows you to quickly visualize potential issues with the terrain and overhead elements. Vehicle Body envelopes as well as Clearance envelopes can be set and displayed by clicking **Properties**. In the **Properties** dialog box, select the **Envelopes (3D Simulation)** category. In the **Envelopes (3D)** category, users can select 3D envelopes for the Vehicle body, 3D Loads and 3D clearance envelopes See exhibit 1

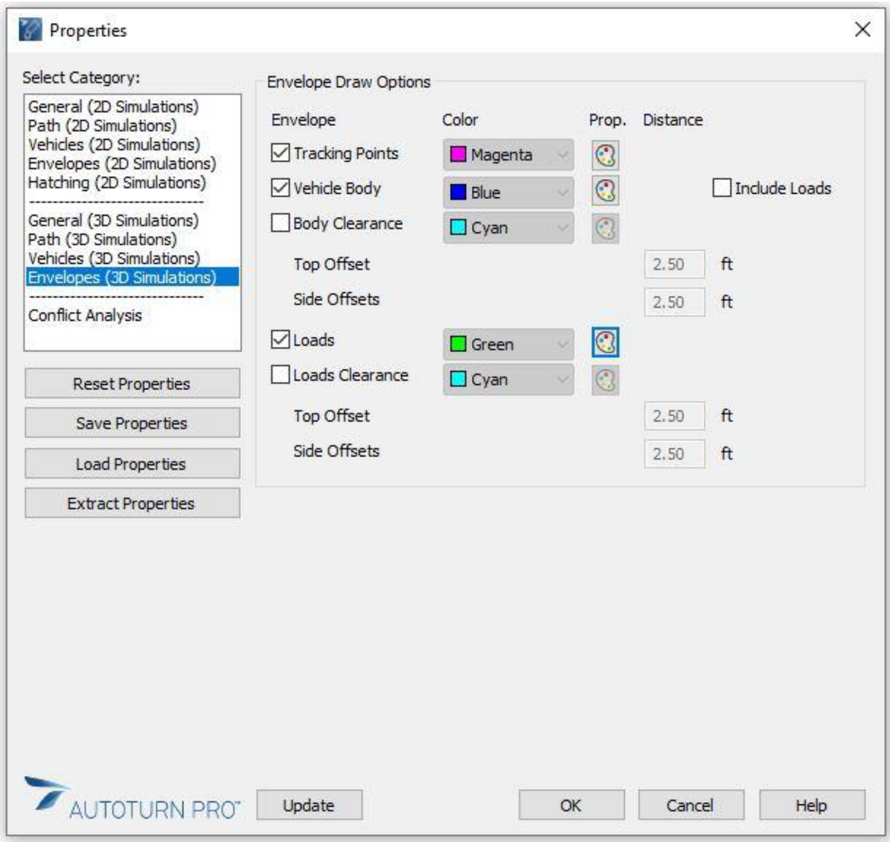


Exhibit 1: Properties Dialog Box – Envelopes (3D Simulation) Category

▶ Vehicle Body

Select this checkbox to have the 3D turn simulation drawn with a 3D vehicle body or 3D swept path envelope. The 3D swept path envelope is the widest path swept out by the sides and overhangs of the 3D vehicle. See exhibit 2

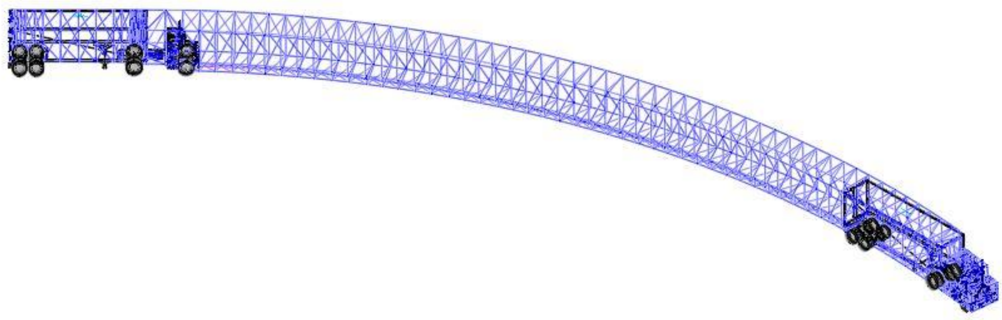


Exhibit 2: 3D Vehicle Body Envelope

▶ Body Clearance

Some scenarios require designers to ensure certain clearance to structures is maintained. By selecting the Body clearance option, Top and Side Offsets can be specified for the 3D envelope. See exhibit 3.

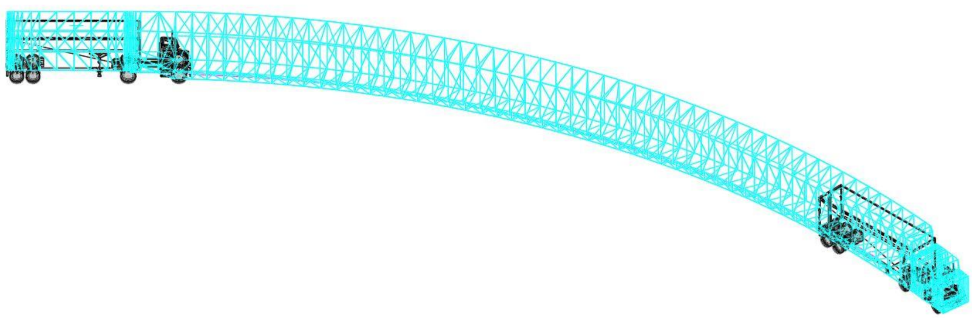


Exhibit 3: Vehicle Body Clearance Envelope

▶ Update

The envelopes displayed, the colors and clearances can be changed at any time. Click **Update**, and then select an existing simulation to update the simulation to the properties currently displayed. This is useful for making changes to existing simulations without having to recreate or place them again.

1. Click **Properties** on the standard toolbar.
2. Select the **Envelopes (3D Simulation)** category and enable **Vehicle Body**.
3. Click **Update**, and then select the **Generate 3D Arc** simulation. See Exhibit 4.

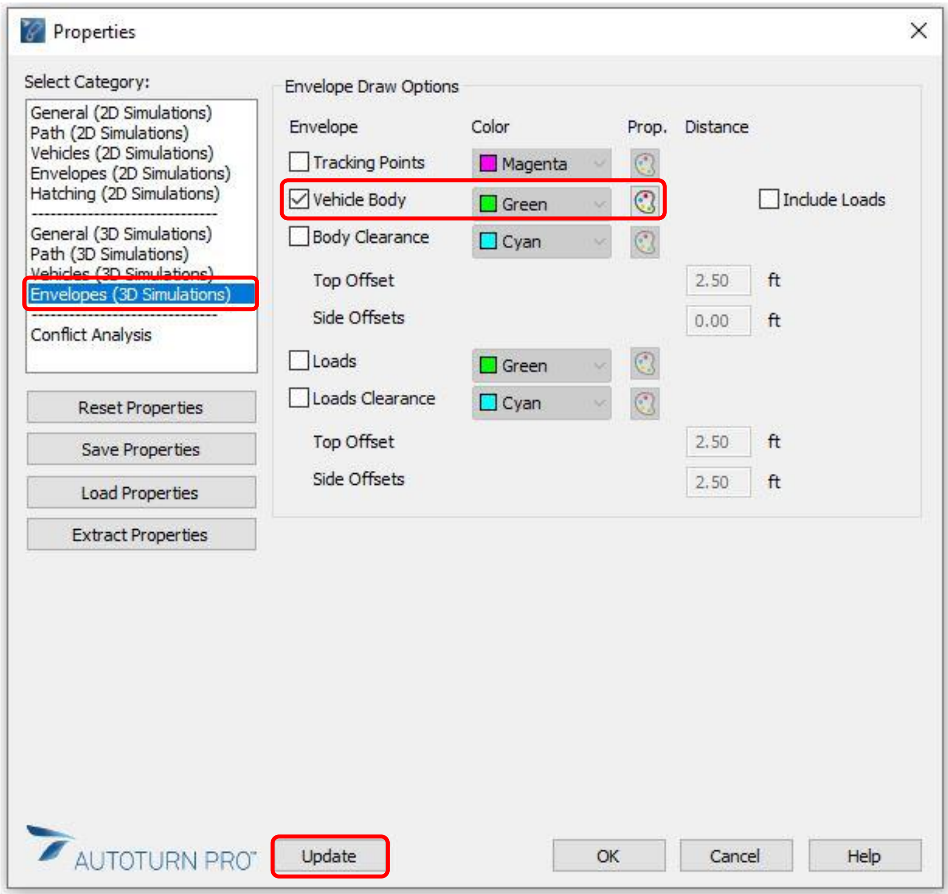


Exhibit 4 – *Properties Dialog Box Settings*

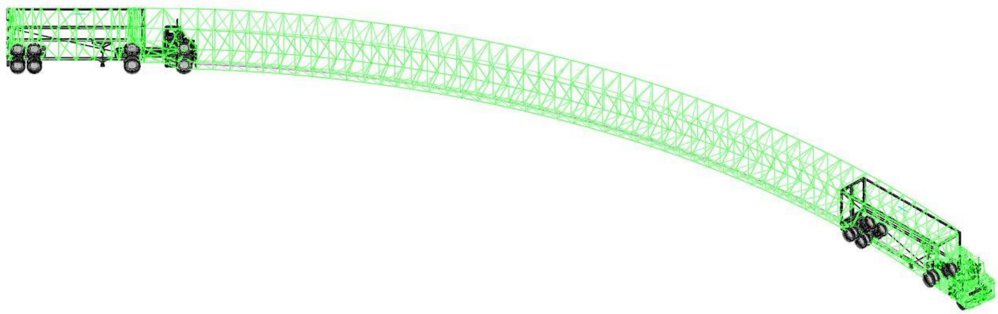


Exhibit 5: Simulation vehicle body envelope updated

▶ Analyze Vehicle Clearance

In addition to the 3D vehicle envelope, the reporting tools in AutoTURN Pro provide you with more detailed information of the simulations, and the potential conflicts. These tools are **Analyze Punch Through** and **Analyze Vehicle Clearance**.

The analysis vehicle clearance report generates a longitudinal profile of the 3D and the crossing elements. The profile can be created along the centerline path, the center of the rear axle group or the left, right, and center of the 3D swept path envelope.

The following exercise explains how to generate an **Analyze Vehicle Clearance** report for the simulation that was created using the **3D Arc Path** tool.

1. Open the drawing 'Intersection – Rail crossing' and create a  2D Arc path simulation using the 'Low-boy Trailer 53 feet' from the 'NCHRP Report 659 2010 (US) library as shown in Exhibit 1.

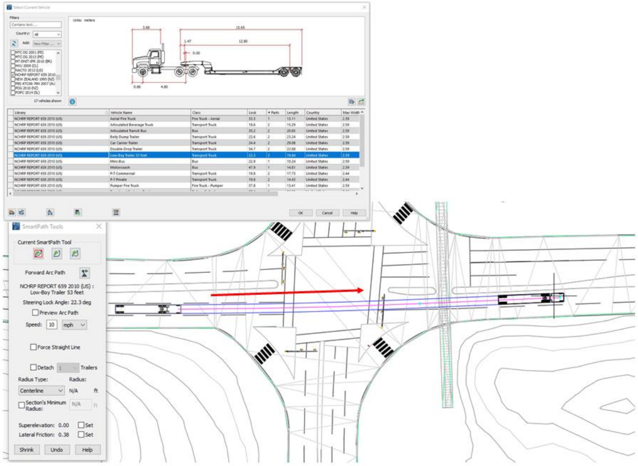


Exhibit 1 – Create 2D simulation

2. Click  **Convert 2D to 3D** on the AutoTURN Pro 3D toolbar and select the 3D terrain and the 2D Simulation. Exhibit 2.
(Once the 2D simulation has been converted to 3D, a prompt will appear. Click **Yes** to delete the 2D simulation)

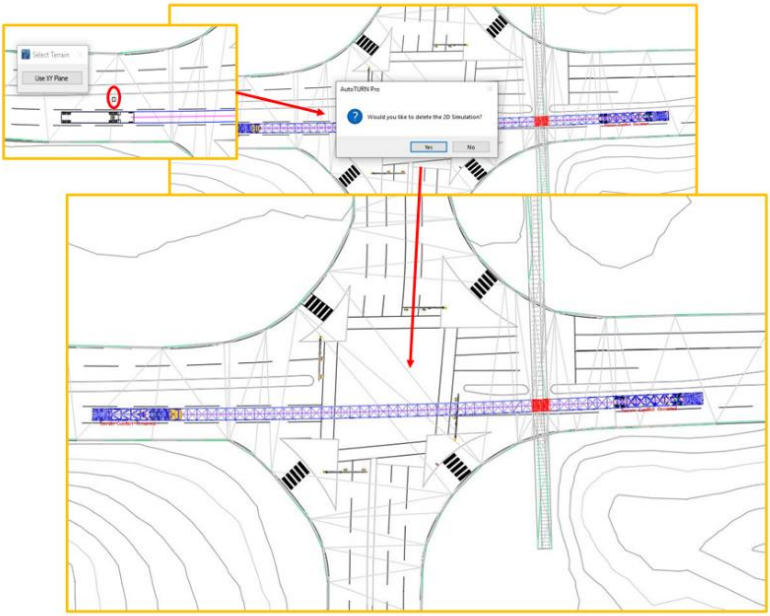


Exhibit 2: Convert 2D to 3D simulation

3. Click  **Analyze Vehicle Clearance** on the AutoTURN Pro 3D toolbar.
4. Select the 3D simulation that was generated in the **previous** step.
5. Once the simulation has been selected, the **Analyze Vehicle Clearance** dialog box allows the user to set color and line styles for the vehicle clearance report. The report can be placed using the current layer or any other layer in the drawing, or it can be exported to a .CSV file, by clicking the **Summary** button. The summary report will allow you to set the station intervals along the path, to have a reference of where the conflicts take place. See Exhibit 3

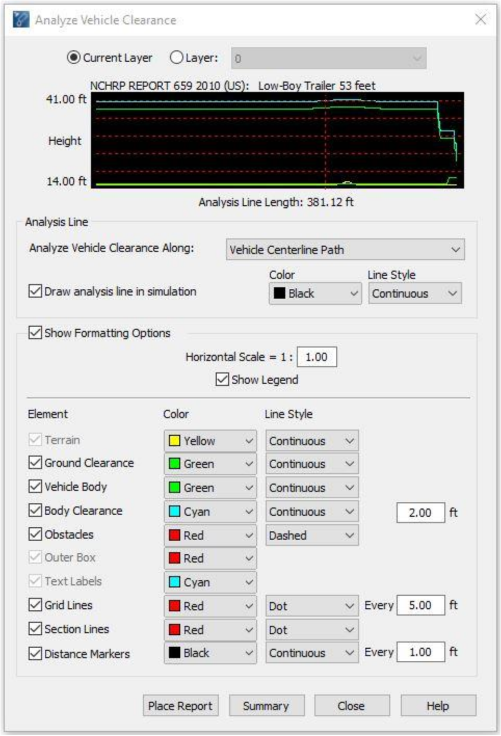
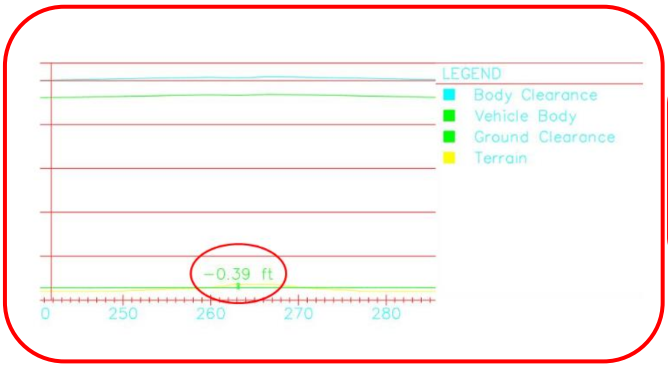


Exhibit3 – *Analyze Vehicle Clearance Dialog Box*

6. Click **Place Report**, and then place the graph into the drawing. See Exhibit 4



NOTE:

The report indicated the conflict of the vehicle body and terrain. See Exhibit 4

Exhibit 4 – *Graphical View of Conflict*

7.

▶ Analyze Punch Through Report

The **Analyze Punch Through** tool, draws one or multiple cross-sections for the selected 3D simulation at a cutting plane line and creates an annotated section detail.

> **NOTE:** *This tool only supports lines, not polylines.*

The following exercise describes how to generate an **Analyze Punch Through** report for the simulation that was generated using the **Generate 3D Arc Path** tool.

1. Draw a line (magenta) that extends across to both sides through the region of conflict between the vehicle's ground clearance and terrain. See Exhibit 1.

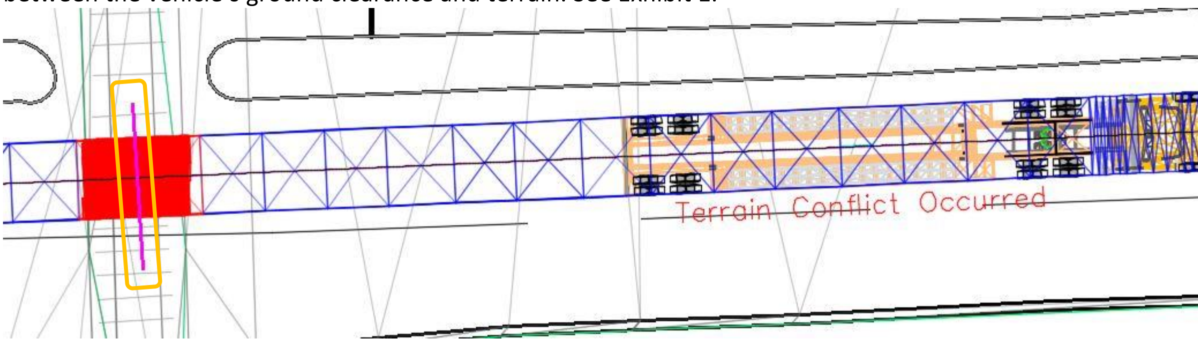


Exhibit 1 – *Line crossing the Region of Conflict*

2. Click  **Analyze Punch Through** on the AutoTURN Pro 3D toolbar.
3. Select the 3D Arc path simulation, and then select the line that crosses the 3D simulation.
4. Pick a point in the drawing to insert the cross-section. Once the cross-section has been inserted into the drawing, the **Analyze Punch Through** dialog box is made available for the user to change display options. See Exhibit 2.

Exhibit 2 – *Analyze Punch Through Dialog Box and Cross Section*

5. Now check the **Multiple Cross Sections** box to generate additional punch through sections. Select the **Distance, Interval, and Direction** (*as shown below in exhibit 3*) for creating additional cross-sections. Click **OK.** See exhibit 3.

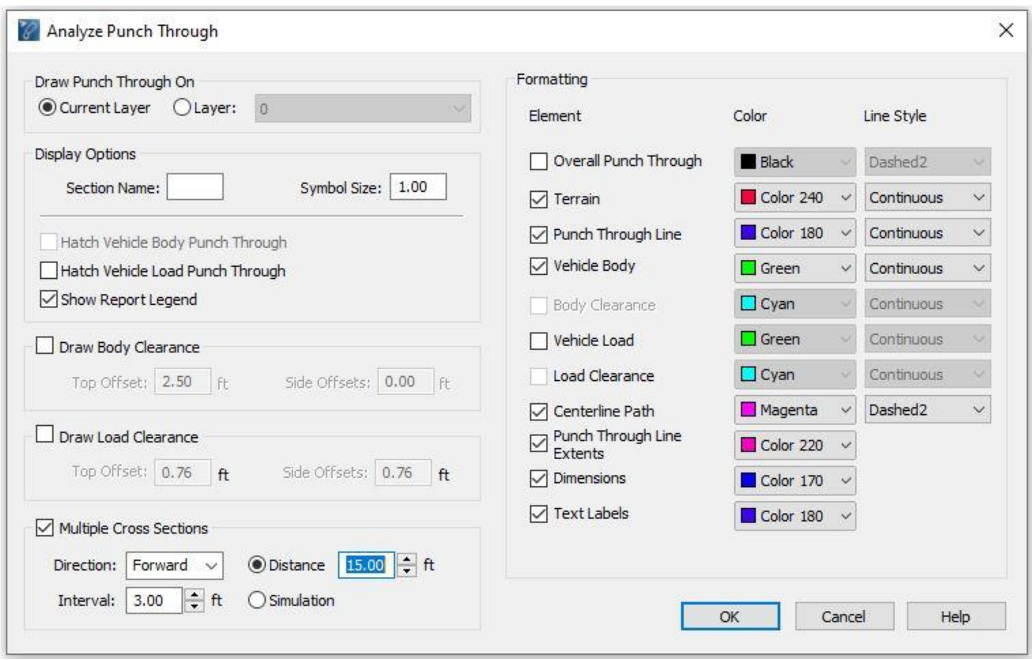


Exhibit 3 – Analyze Punch Through: Multiple Cross Sections

6. The generation of multiple cross-sections will let you pinpoint where the conflict with the surface is more severe. See exhibit 4.

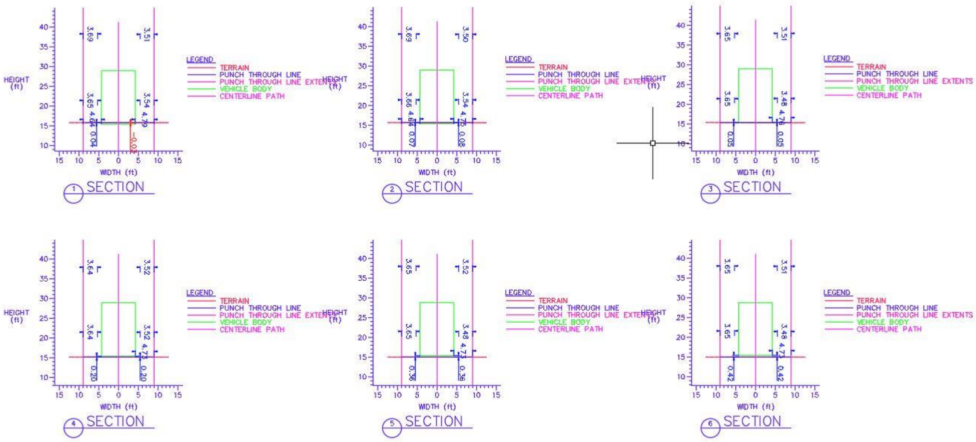


Exhibit 4: Multiple cross-sections showing the severity of conflicts

Summary

Topics covered in this section:

Using 3D Reporting Tools

- ☐ Analyze Vehicle Clearance
- ☐ Analyze Punch Through
- ☐ 3D Envelopes

3D Vehicle Customization

Similar to the process of creating custom 2D vehicles in AutoTURN, users can create 3D vehicles with custom vehicle profiles or import 3D custom realistic vehicles.

The modeling of 3D vehicles in AutoTURN involves 3 key aspects: 1 the modeling of the vehicle's underside or ground clearances, 2. the vehicle height, and 3 the vehicle's rooflines. These 3 components are used to define the 3D model and to generate the corresponding 3D envelope

Profile Dialog Box Explained

The profile of the design vehicle is accessed through the vehicle library by clicking **View/Edit Vehicle Details**. Within the **Edit Vehicle Details** dialog box, select the **Roofline View Data** option. See Exhibit 1.

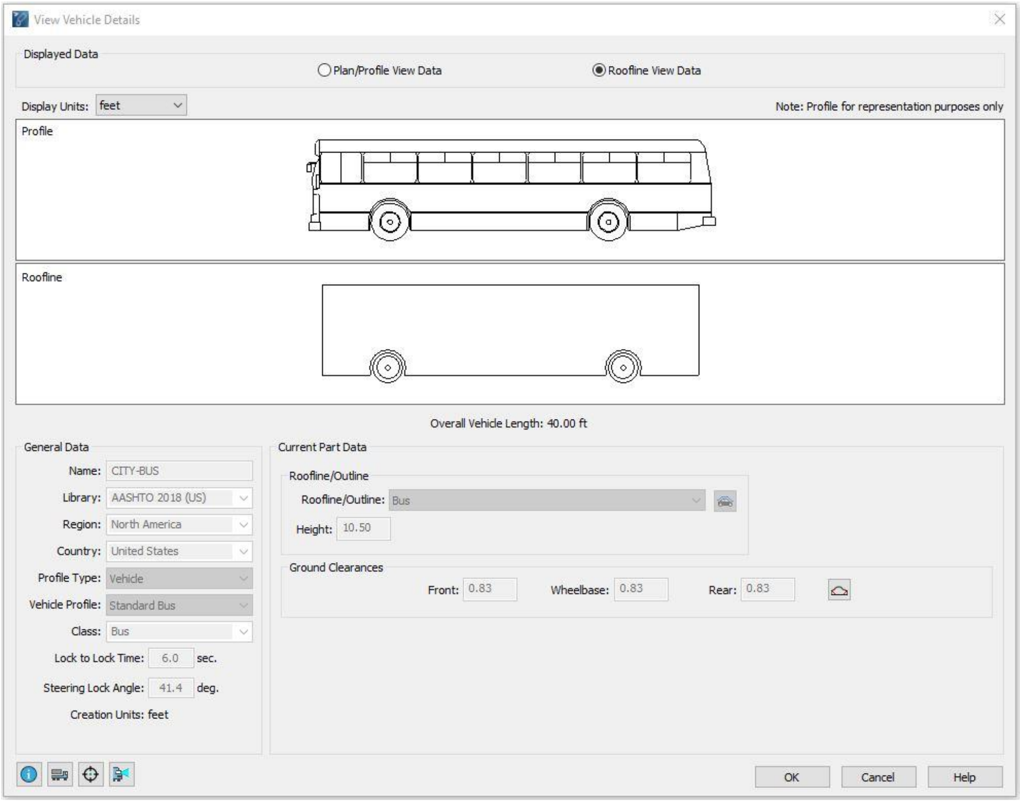


Exhibit 1 – *Edit Vehicle Details dialog box.*

Roofline/Outline – Allows the user to select a predefined roofline shape already defined in the software or import a pre-drawn (custom) roofline (complex chain or polyline) or outline by clicking **Import Outline or Roofline**.

Height – Indicates the overall vehicle height.

Connector Heights - Allows connector heights in multi-part vehicles to be configured. Connector heights may need to be changed when a vehicle's ground clearance values are changed.

Ground Clearances – Allows the user to adjust the vehicle's front, wheelbase, and rear ground clearances for the vehicle's undercarriage, when using a roofline.

Chamfers – Allows the user to adjust the front and rear profiles, so that a vehicle may travel up and down slight inclines without damaging the vehicle.

▶ Creating a Custom Roofline

The following exercise illustrates how to create a custom CITY BUS with a custom 3D roofline.

1. Load the drawing file **CITY BUS Roofline**.
2. Click **Vehicles** on the standard toolbar.
3. Select the **AASHTO 2018 (US)** library and make a copy of the **CITY-BUS** design vehicle. See Exhibit 2.

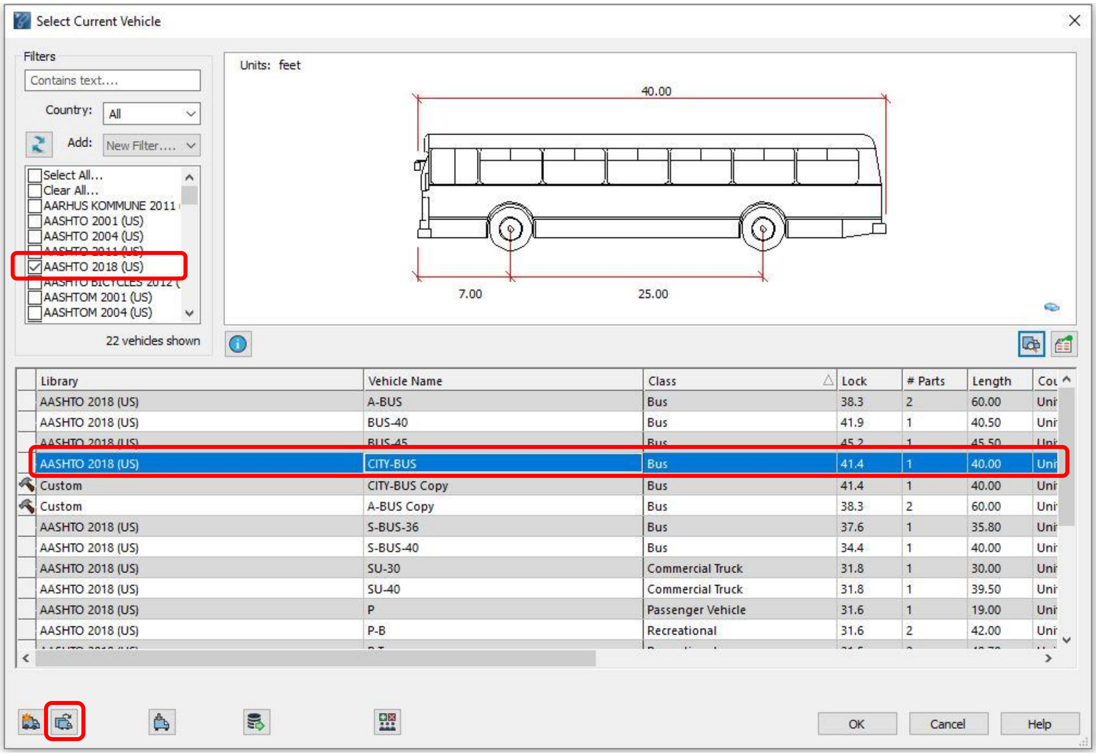


Exhibit 2 – *Select Current Vehicle dialog box*

4. Name the vehicle **CITY BUS Copy**. Click the **Roofline View Data**, and then click  **Import Outline or Roofline**. See Exhibit 3.

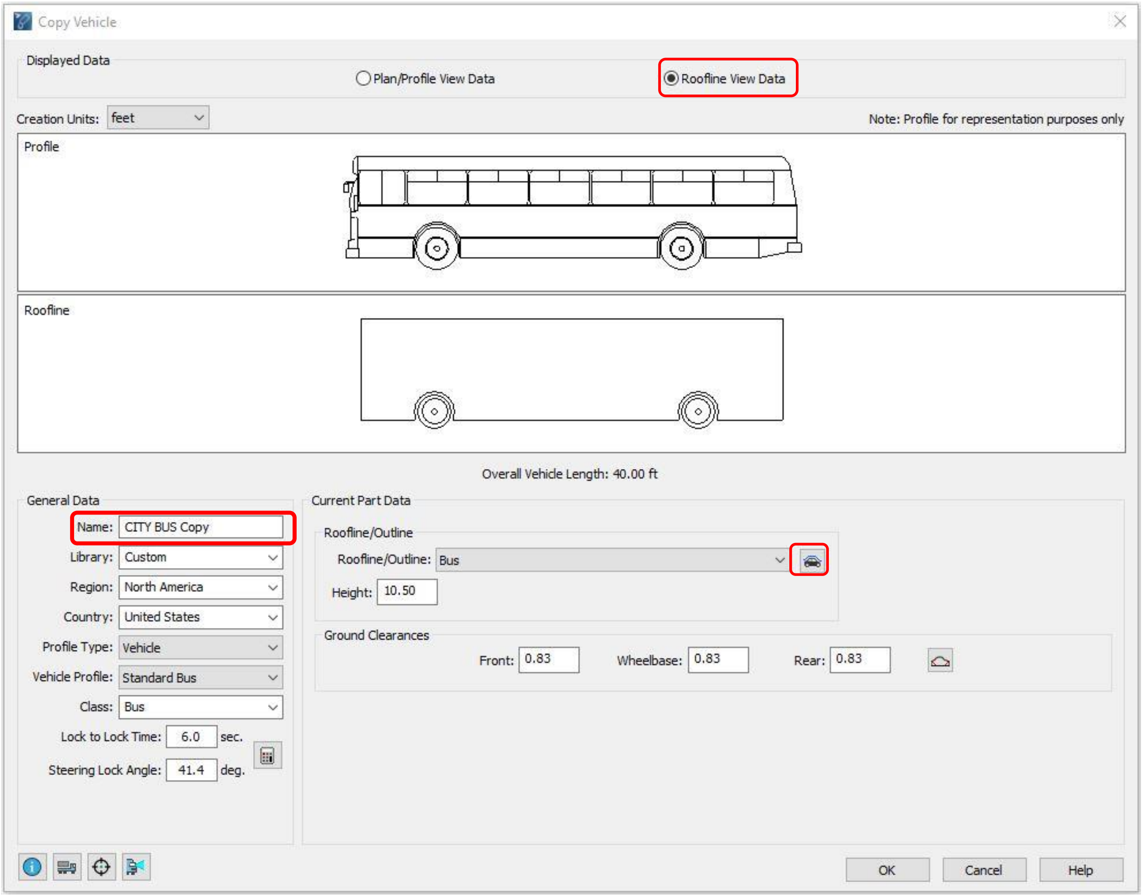


Exhibit 3 – *Copy Vehicle Dialog Box – General Tab*

5. Select the polyline from the CITY BUS Roofline drawing that resembles the side profile of the bus. Right-click for (0,0) and to accept. See Exhibit 4.

Exhibit 4 – *Polyline from CITY BUS Roofline Drawing*

6. Click **OK** to save the CITY BUS Copy vehicle into the **Custom** library. See Exhibit5.

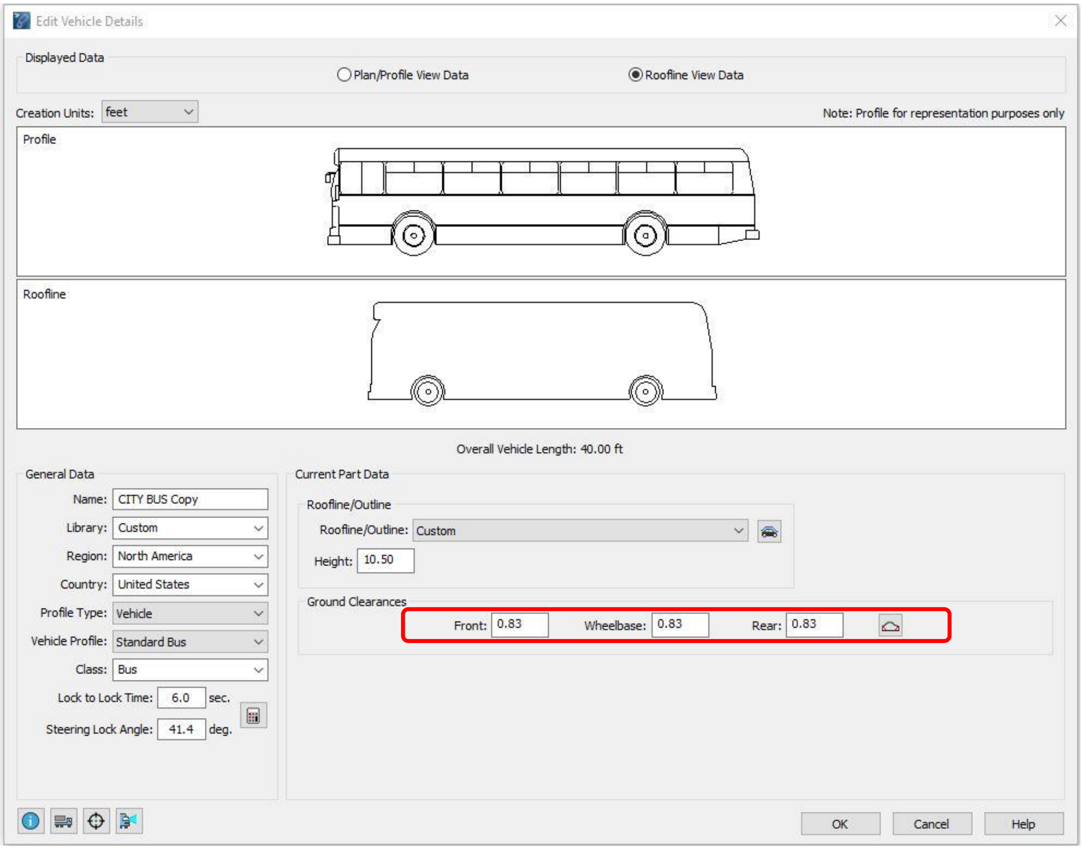


Exhibit 5 – *Copy Vehicle Dialog Box – 3D Profile Tab*

7. The design vehicle now appears in the custom library and is ready for use. See Exhibit 6

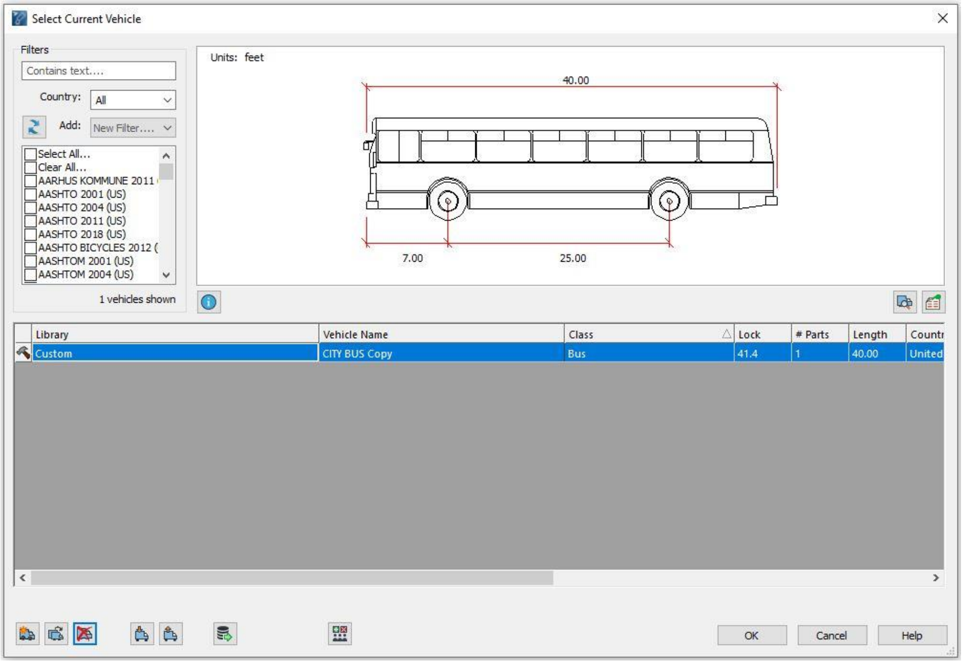


Exhibit 6 – *Select Current Vehicle Dialog Box – Custom Library List*

Creating a Custom Outline

For cases where more level of detail is required to model the vehicle in 3D, AutoTURN allows you to define the entire profile for the vehicle.

To do this, you will need to have the vehicle profile defined as a closed complex chain or a closed polyline. The following exercise illustrates how to create a custom 3D Garbage Truck with a custom 3D outline.

1. Load the drawing file **Garbage Truck Outline**.
2. Click **Vehicles** on the standard toolbar.
3. Select the NCHRP Report 659 (US) library and make a copy of the Rear-Load Garbage Truck. See Exhibit 1.

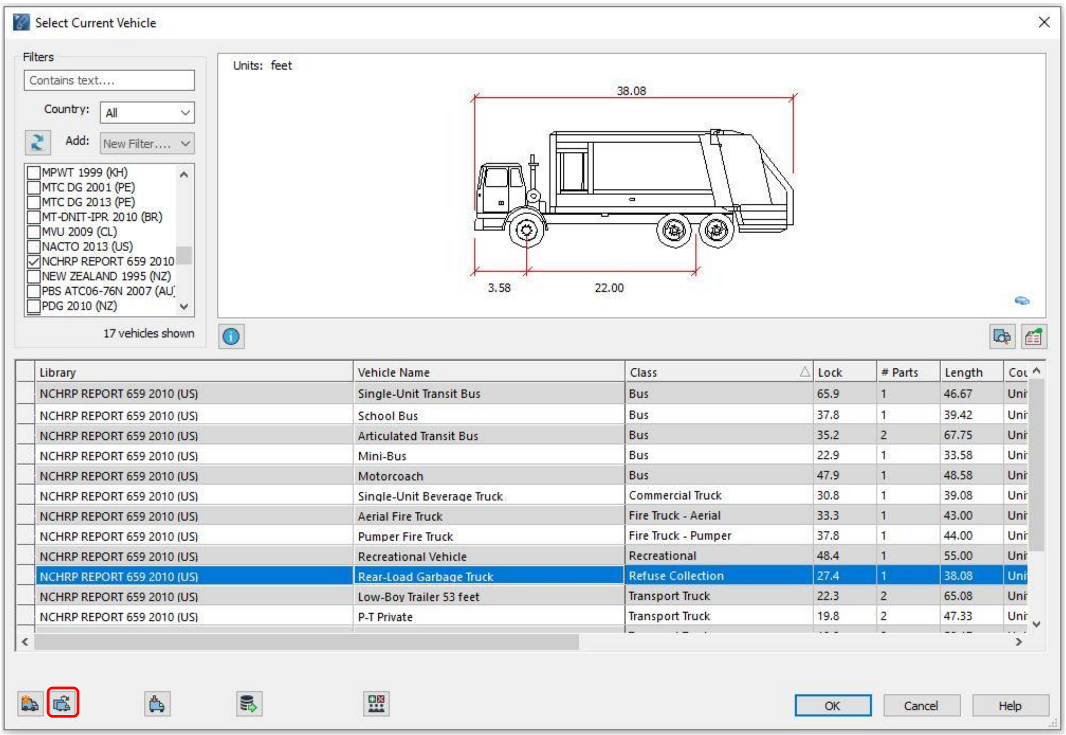


Exhibit 1 – *Select Vehicle*

4. Name the design vehicle **Garbage Truck Copy**. Click the **Roofline View Data**, and then click **Import Outline or Roofline**. See Exhibit 2.

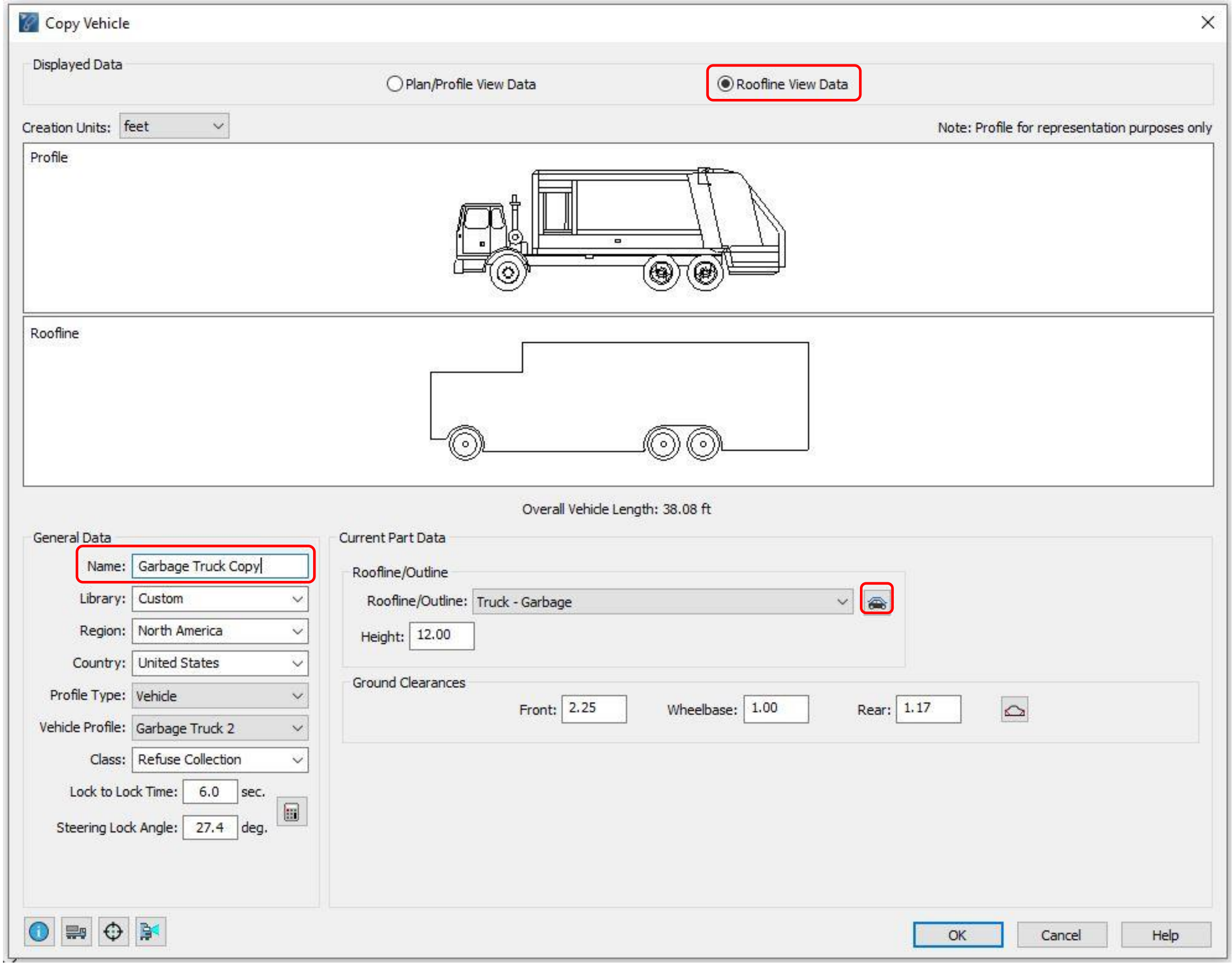


Exhibit 2 – *Import Roofline/Outline*

5. From the **Garbage Truck Outline** drawing, select the closed polyline that represents the outline of the vehicle then, pick a reference point for the ground or right-click for (0,0). See Exhibit 3.

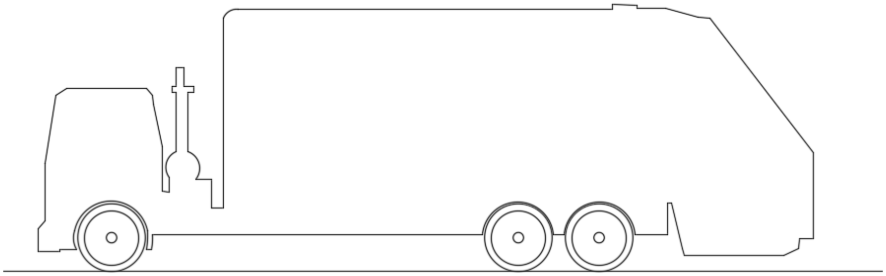


Exhibit 3 – *Garbage Truck Outline*

6. Now, it is possible to visualize the custom outline of the vehicle in the Roofline view data dialog box. See Exhibit 4.

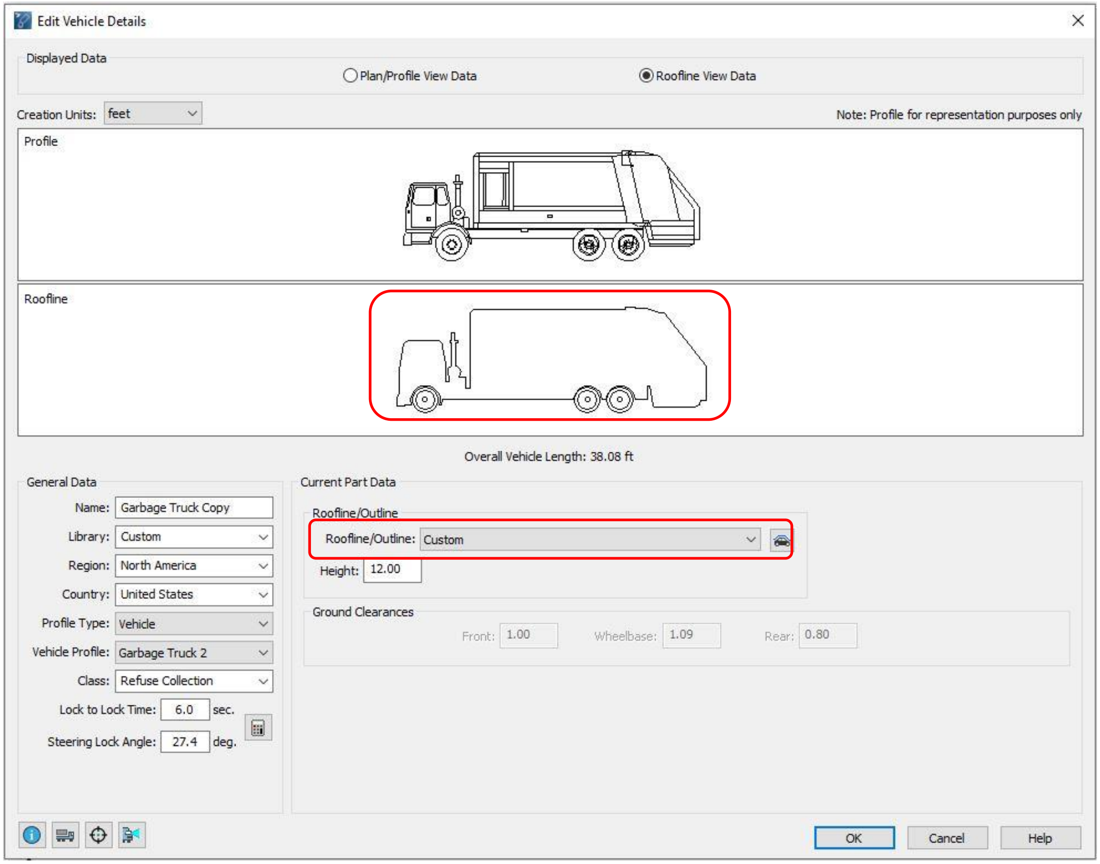


Exhibit 4 – *Adding Custom Outline*

Note that the ground clearance information is no longer required since the 3D model will use the outline of the vehicle and the datum point indicated in step 5 as reference for the vehicle definition in 3D.

7. The new vehicle will now use the outline to simulate the vehicle in 3D and produce the corresponding 3D envelope. See Exhibit 5.

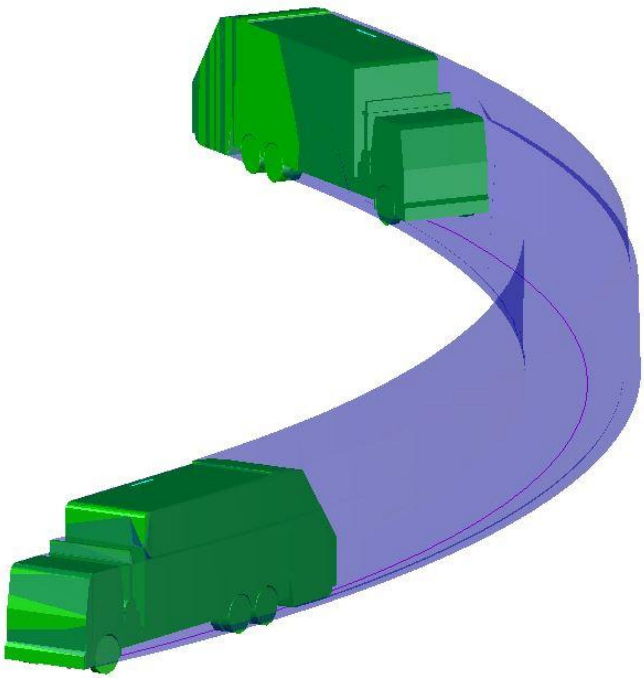
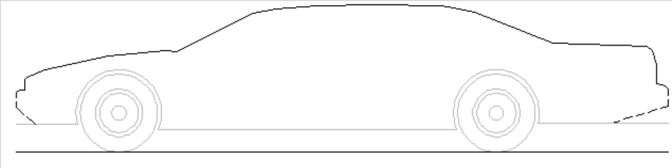


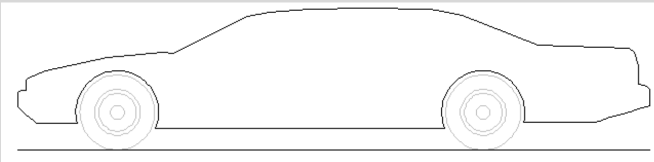
Exhibit 5 – *3D Simulation*

General Guidelines for Creating Pre-Drawn Rooflines or Outlines:

- The roofline/outline of the vehicle must be a polyline or complex chain.
- The roofline/outline must be drawn to scale
- The roofline should not be drawn such that the front and rear ends of the vehicle touch the ground clearance heights. In order for the chamfers to be configured optimally, it is best if the roofline ends at extents of the vehicle's length as in the following illustration:



Example of a roofline drawing



Example of an outline drawing

In the illustration, the solid lines represent the roofline and its relation to the ground (drawn to the actual size of the vehicle); the light grey lines represent the ground clearance values and tires; the dashed lines represent the configured chamfers.

AutoTURN Pro 3D generates a parametric 3D analytical model for all vehicles based on the part configuration and vehicle dimensions. This analytical model is used for calculations and analysis functions. However, since a generic 3D profile based on overall dimensions may not provide an aesthetically pleasing representation of the vehicle, it is recommended that users create their 3D drawings for custom vehicles. To replace the analytical model with a custom realistic 3D model, select the Realistic checkbox in the General(3D Simulations) category of the Properties dialog box.

General Guidelines for Creating 3D Realistic Vehicles

- All drawings must be drawn in feet units regardless of the vehicle's creation units setting.
- Draw vehicle body to match the roofline's height and ground clearance values as measured from z=0.
- Orient drawings so that the vehicle points to the right and the front center of the vehicle part is at exactly coordinate 0,0.
- Do not draw wheels - they are drawn automatically by AutoTURN Pro.
- Any amount of detail can be included to represent the vehicle in 3D as long as it is drawn full scale.
- Each part of a multi-part custom vehicle must be drawn individually, and each file name must include a "_3D" suffix. The first drawing's name is the same as the Name field of the custom vehicle (e.g. the name of the first drawing of a two-part vehicle would be: Semi Trailer_3D.dwg). Each subsequent drawing must be named the same as the first drawing with the suffix comprised of @ followed by the trailer number and then _3D (e.g. the second drawing of the two part vehicle would be: Semi Trailer@1_3D.dwg).
- MicroStation users must create a Cel file of the 3D vehicle.
- 3D Vehicles are saved into the following directory:

  Windows 10
  C:\Users\<your username>\AppData\Roaming\Transoft Solutions\Common\vehicles3d user\<library name>\
- When creating the custom vehicle, the name of the custom vehicle must match the file name of the 3D vehicle.
- Keep the poly count low for the 3D models.

How to create realistic 3D vehicles

The following exercise describes how to create realistic 3D vehicles.

1. Copy the files ***WB-50 Tutorial_3D*** and ***WB-50 Tutorial@1_3D*** in the following directory:

   Windows 10:
   C:\Users\<your username>\AppData\Roaming\Transoft Solutions\Common\vehicles3d user\<library name>\Custom
   (Create the Custom folder in the above directory)

2. Click **Vehicles** on the standard toolbar.

3. In the **Select Current Vehicle** dialog box select the **Transoft** library and highlight the **WB-50 Semi-Trailer** vehicle. See Exhibit 1.

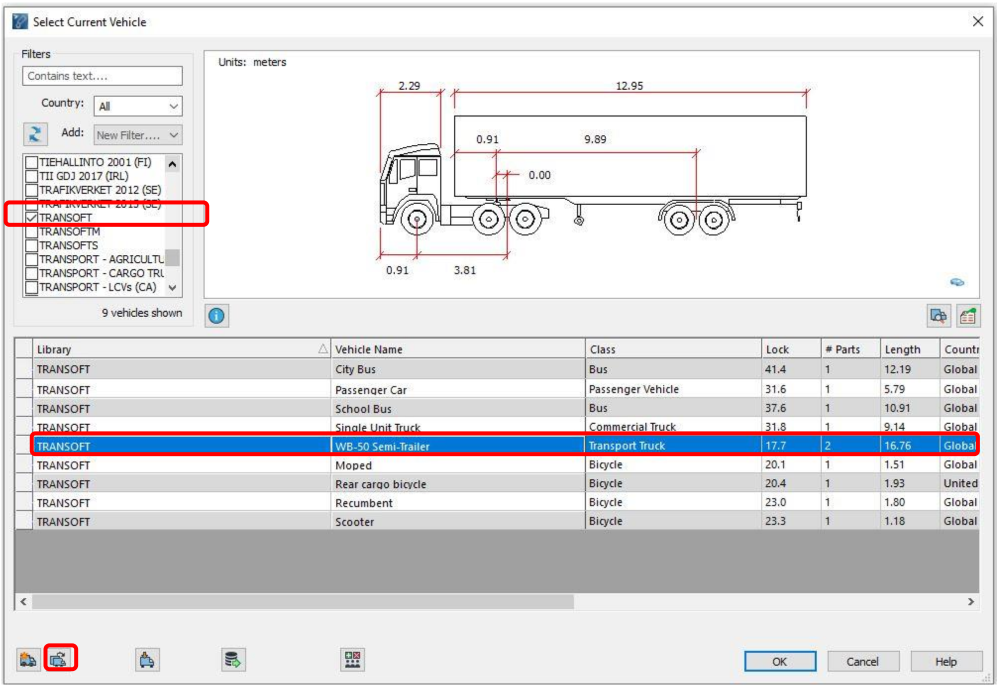


Exhibit 1 – *Select Current Vehicle Dialog Box – TRANSOFT library*

4. Click **Copy Vehicle**.
5. Check the **Plan/Profile View Data**, name the vehicle: **WB-50 Tutorial**, and change the Steering Lock Angle to 20 degrees. See Exhibit 2.

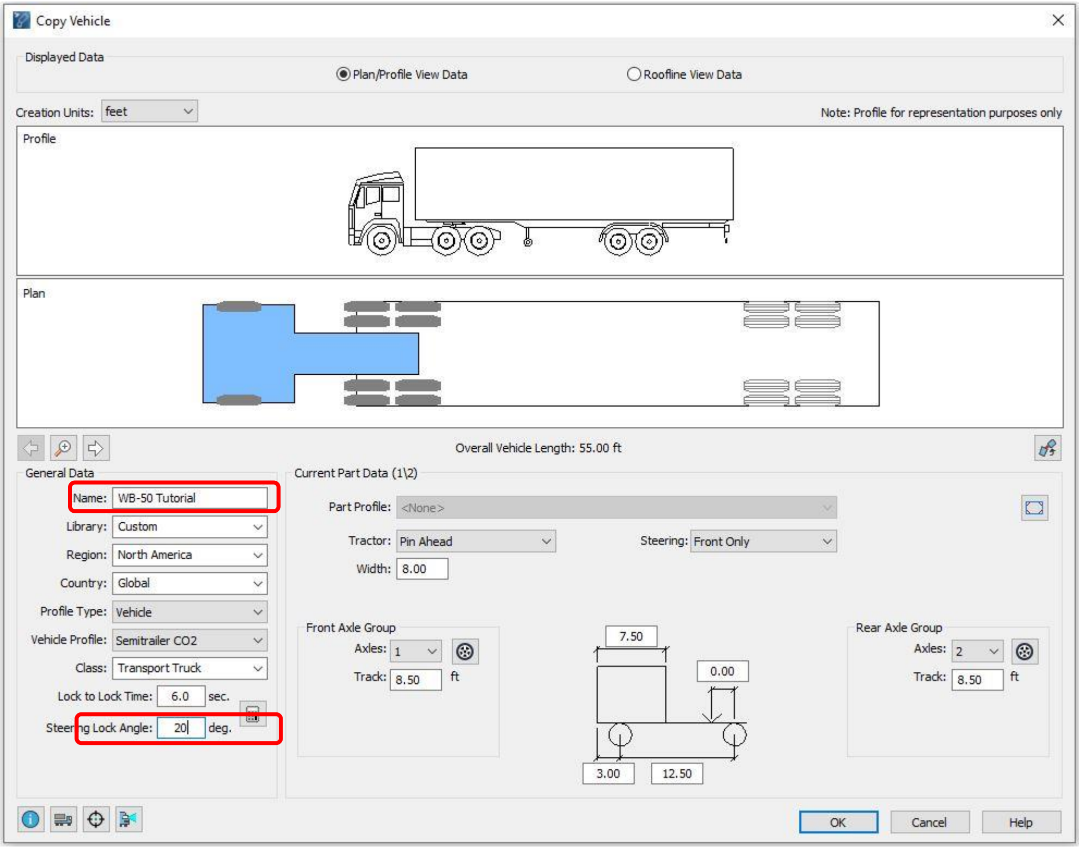


Exhibit 2 – *Copy Vehicle Dialog Box – General Tab Settings*

6. Click **OK** to save the customized WB-50 vehicle.
7. Select the WB-50 vehicle from the Custom library.
8. Create a new drawing file, and then click **Select Terrain** on the AutoTURN Pro 3D toolbar. Click **Use XY Plane** at the prompt. See Exhibit 3

Exhibit3– *Select Terrain Dialog Box*

9. Click **Generate 3D Arc Path** on the AutoTURN Pro 3D toolbar and generate an arc path simulation. See Exhibit 4.

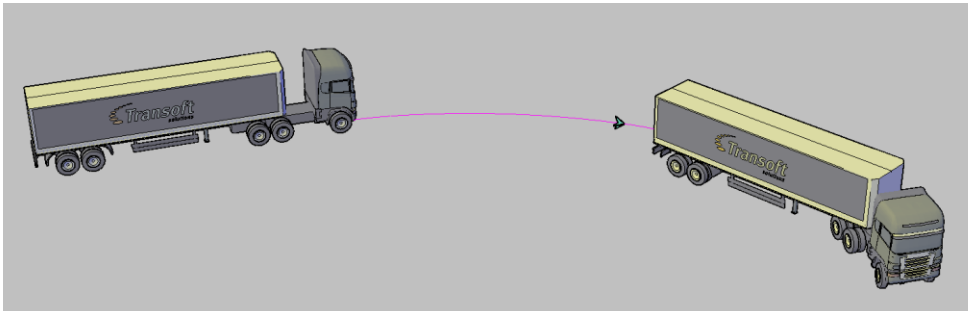


Exhibit 4– *Custom WB-50 and 3D Arc Path*

- NOTE: Navigate to **Properties -> Vehicles (3D Simulations)** and make sure to check/select the **Realistic** option/box. .

You can enter different values for the stopping sight distance and see how the tool updates the drawing dynamically. For the purpose of this example, use a stopping sight distance of 50 meters.

How to Simulate bicycle maneuvers (AutoTURN Pro only)

The unique ability to simulate bicycle maneuvers in AutoTURN Pro will help you assess the space requirements for various types of cycles from standard bicycles to recumbents, scooters, and more.

Cycles and non-motorized vehicles differ from motorized vehicles in regards to their turning characteristics, speed, overall shape, and the envelopes they generate since most of these vehicles lean to steer.

Simulating bicycle maneuvers

In this example, we will evaluate the space requirements for different types of design cycles to make sure they can safely and efficiently use the bikeway infrastructure

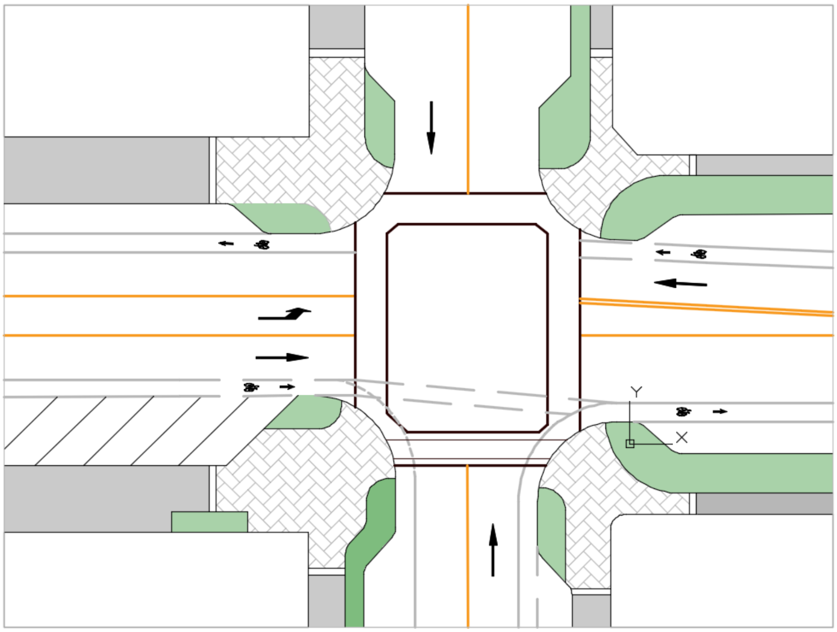


Exhibit 1. Simulate bicycle maneuvers

1. Open the drawing **28 – Simulate Bicycle Maneuvers .dwg or .dgn** depending on your CAD platform.

2. Click the Vehicles button and in the search box type “bicycle” and press either the key enter key or the tab key on your keyboard; this will filter AutoTURN Pro’s vehicle libraries and

display only the cycle class (or non-motorized) vehicle libraries, select the standard AASHTO bicycle and click the view vehicle details button:

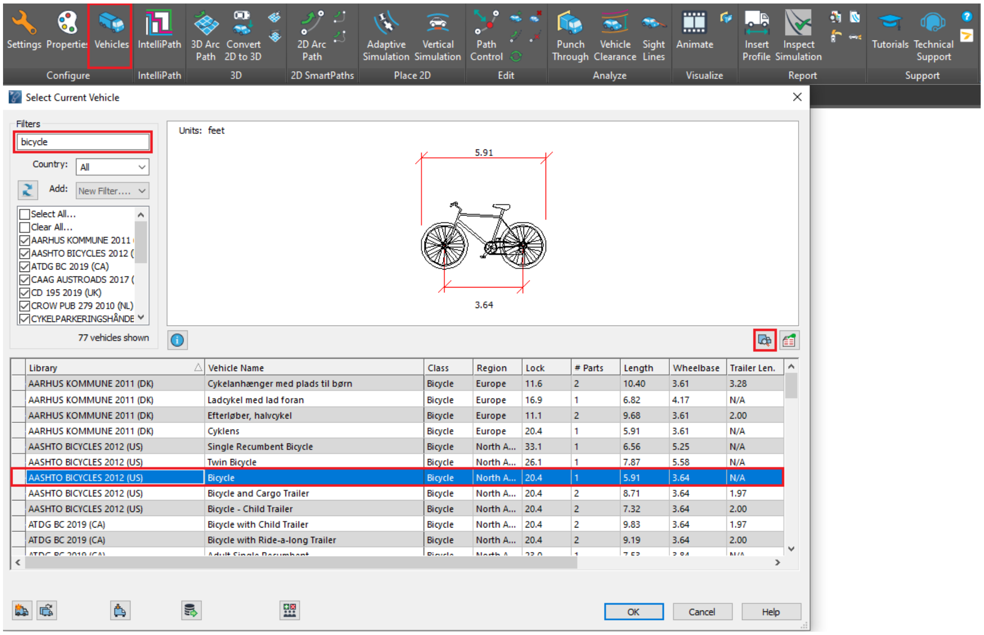


Exhibit 2. Filter and select standard bicycle

3. Check the Show Rider box and note the different parameters that are taking into account in the modeling of these types of vehicles:

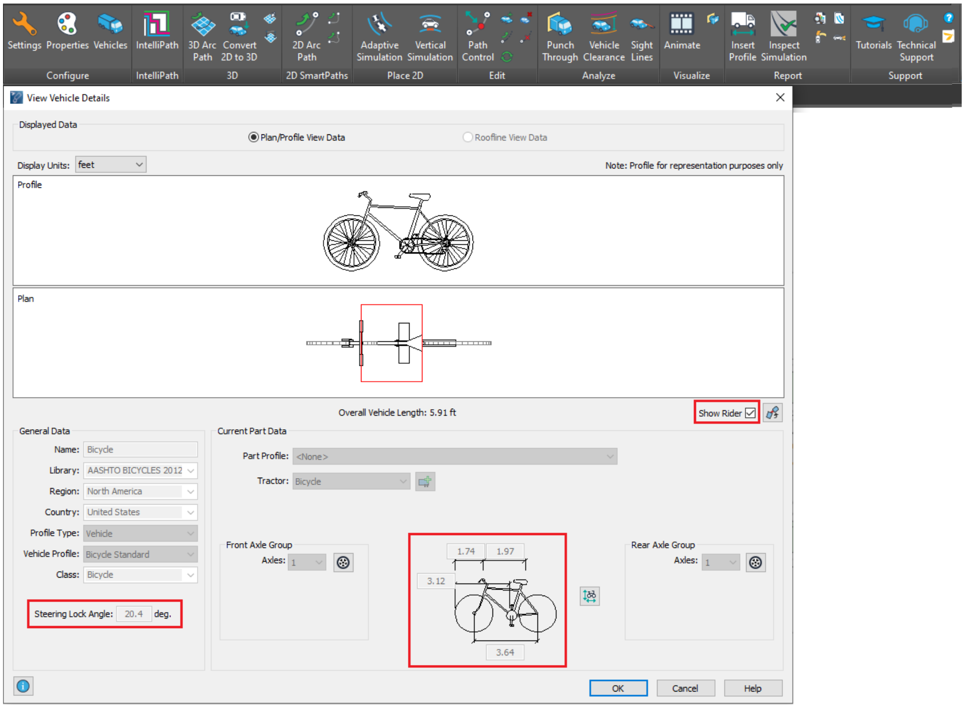


Exhibit 3. Key parameters to create custom bicycles

4. Click on the Vehicle's Heights and Widths button:

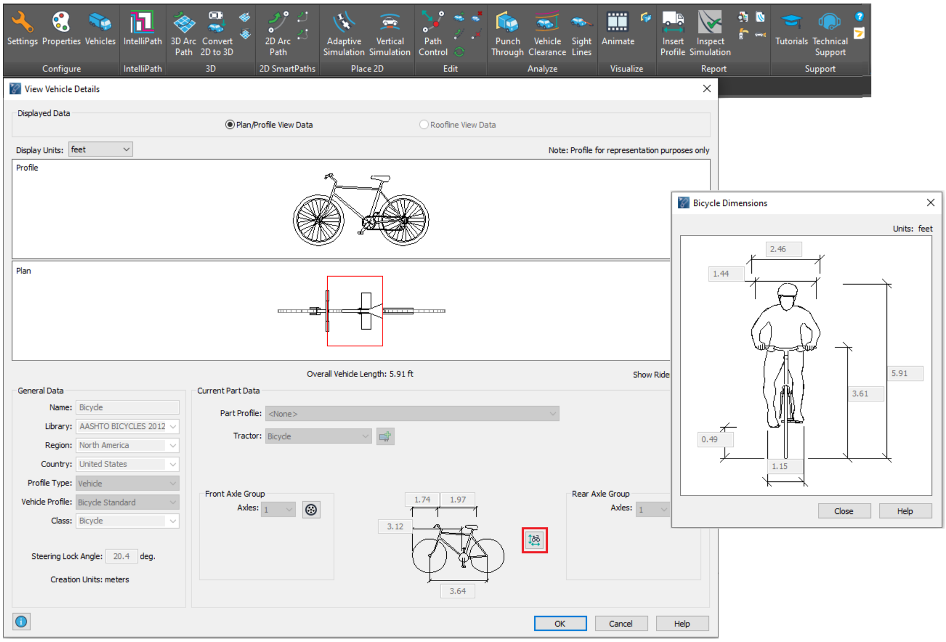


Exhibit 4. Key parameters to create custom bicycles

In the case of standard bicycles, parameters such as the operational widths and heights are predefined as per the corresponding design guides. However, when creating custom bicycles all the different aspects and key heights and widths can be defined for the custom cycle. A key element to the modeling of the bicycle and its envelopes is the rider's box; defined based on the rider's width, height, and the distance between the handlebar and the seat. A similar concept is applied for bicycles such as the twin bike. Click close, and ok to select the standard bicycle.

Click the 2D Arc Path button to create a simulation for the selected bicycle, click the align vehicle to geometry option, and pick on of the lines that define the bike lane going West to East. Then click to define the starting point of the simulation as per Exhibit 5 below:

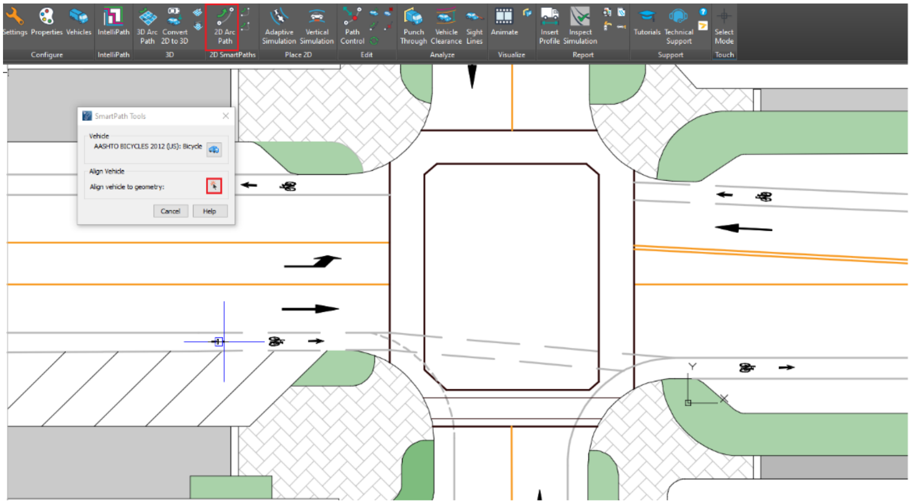


Exhibit 5. Create simulation for standard bicycle

5. Change the speed to 10 mph and generate a first straight section until the point where the rear tire is ahead of the bike lane's arrow:

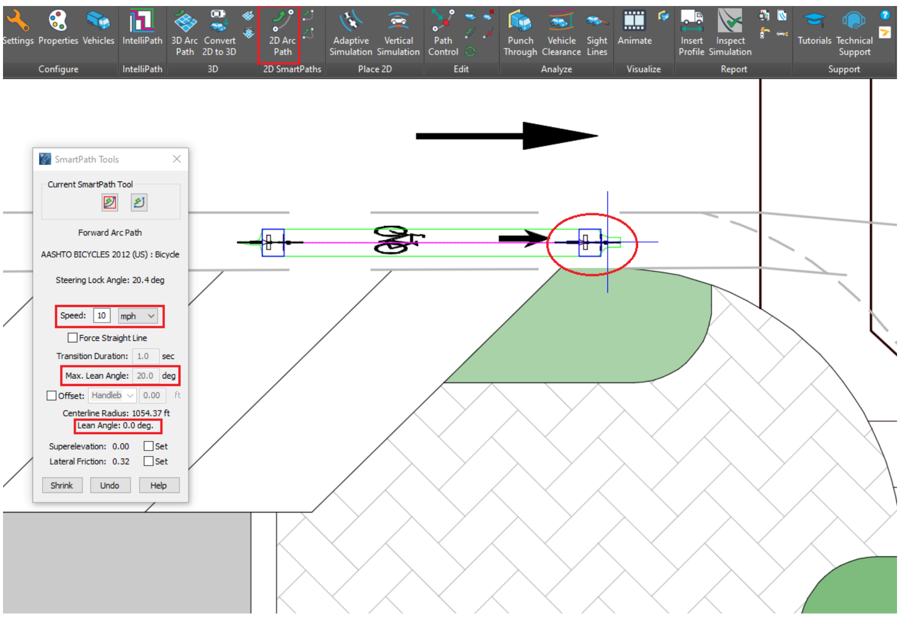


Exhibit 6. Create simulation for standard bicycle, speed, and initial lean

Note that there are 2 lean angles reported by the SmartPath dialog. The maximum lean angle (fixed for standard cycles) is set to 20 degrees as recommended by the AASHTO guideline. This can be changed for custom cycles to a maximum of 25 degrees, the angle at which the guideline highlights the risk of pedal strike.

In the SmartPath dialog, check the offset box and use the dropdown to select the pedals' offset, by leaving the offset value at 0 the tool will display the envelope produced by the pedals given the pedal's width specified at the vehicle's definition. The pedal's envelopes will be displayed from this point until you select a different element for the offset or uncheck the dialog's box:

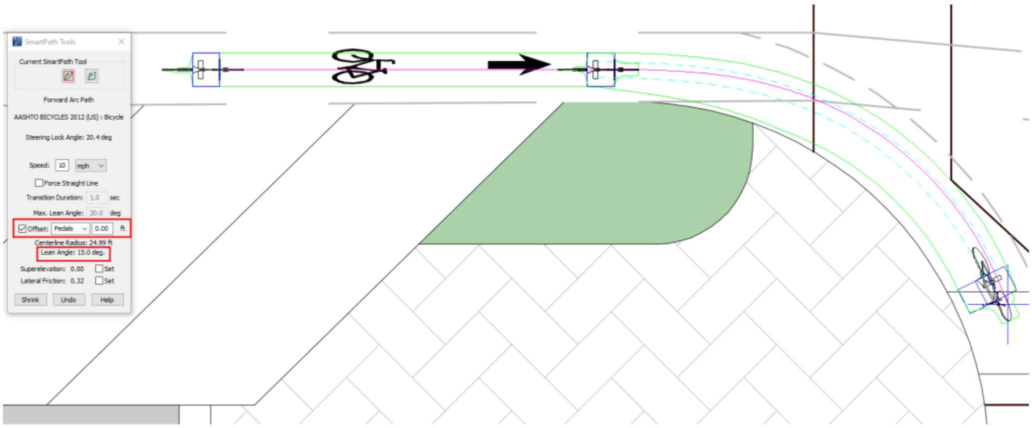


Exhibit 7. Create simulation for standard bicycle, pedals' offset, and reported lean

Note how the achieved lean is reported for the turn maneuver based on the radius and the input speed.

6. Change the offset to the handlebars and note how the offset envelopes automatically change for the next section of the simulation. Depending on the design scenario analyzing clearances to different elements of the bicycle may be required, for instance, pedals will be more relevant for checking clearances against curbs, whilst handlebars would be more relevant in the case of railings, etc.

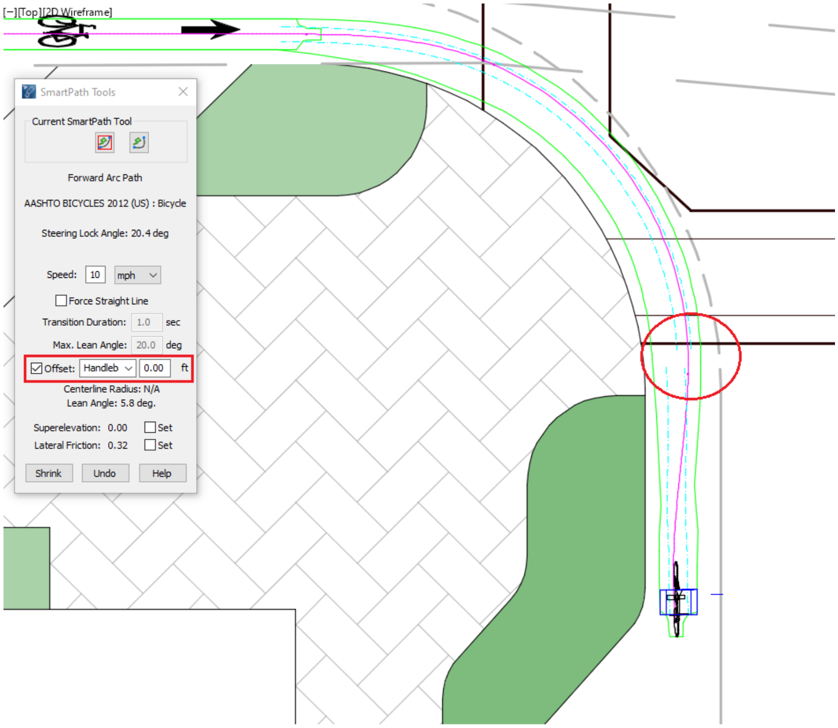


Exhibit 8. Create simulation for standard bicycle, handlebar's offsets, and reported lean

7. Click on the Vehicles button and select the Bakfiets 3 cycle from the CROW PUB 279

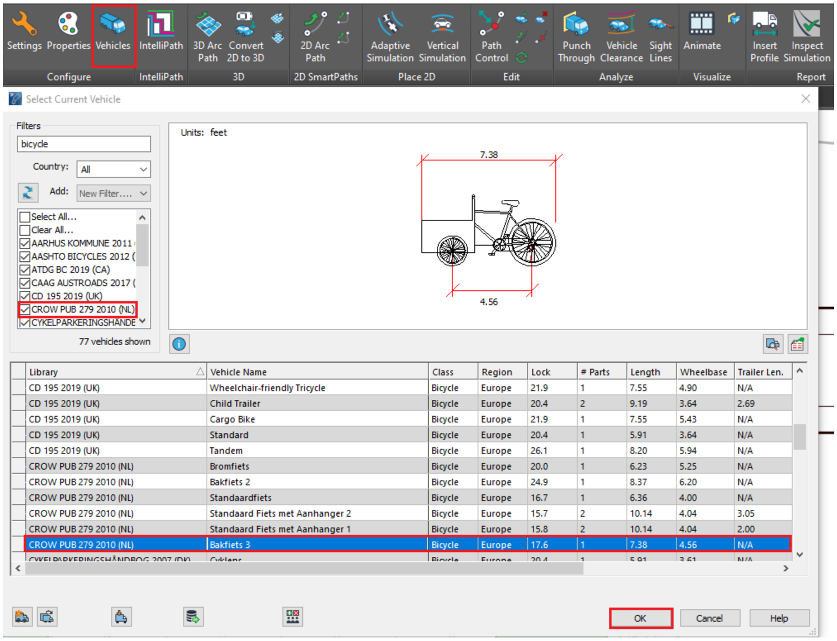


Exhibit 9. Select front cargo bicycle

8. Click Corner Path and place the vehicle pointing north at the East side of the intersection

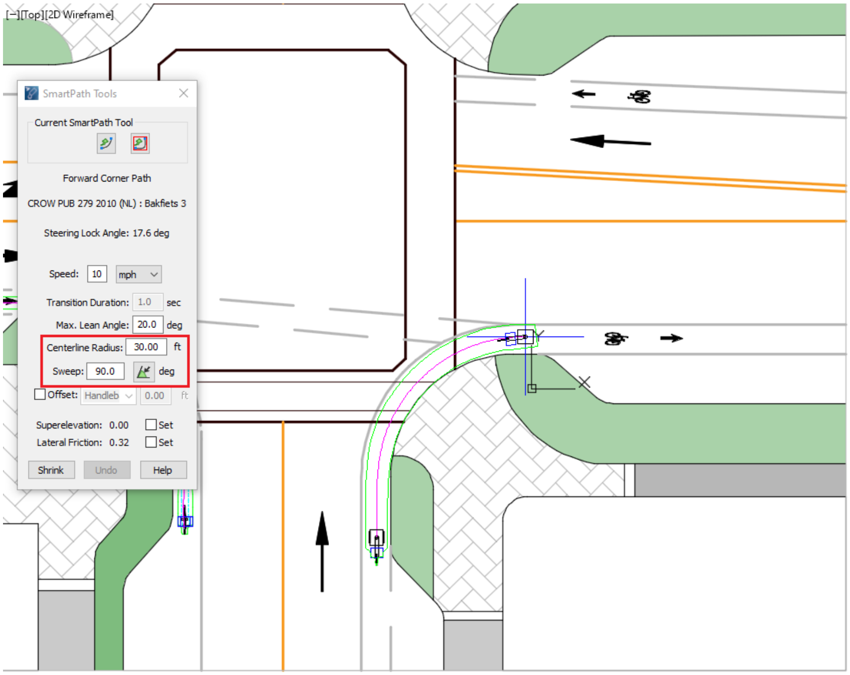


Exhibit 10. Generate a Corner Path simulation

9. Define the Centerline Radius to be 30 ft and the sweep angle to be 90 degrees and generate the simulation.

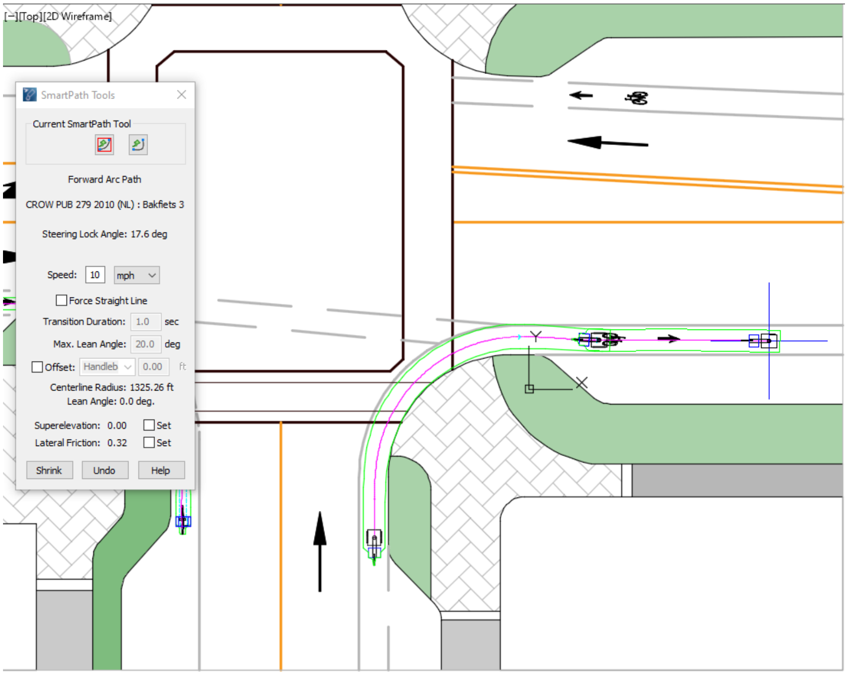


Exhibit 11. Alternating between the SmartPath tools

After the Corner turn is made switch to the Arc Path tool, and place another section for the simulation. Note that this type of cycle won't lean yet will be able to use the same options and capabilities of the other cycles.

TECHNICAL APPENDICES

Steering Angles vs. Minimum Turn Radius

The maximum allowable steering angle is defined in AutoTURN as the average of the angles of the two steering wheels in the lock or maximum angle position. The steering angle as a function of the minimum centerline turning radius can be calculated directly from the simplified formula:

SA = arcsin (WB/ Rc)

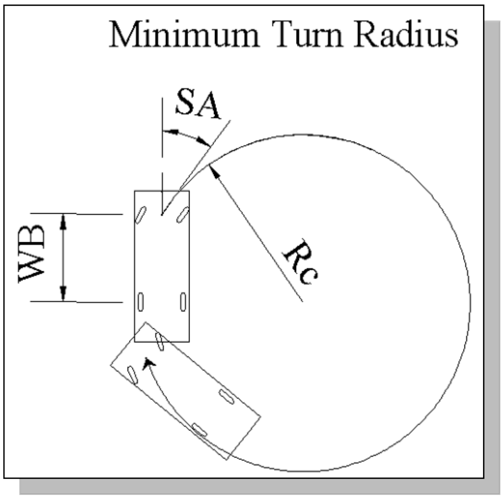
Where:

WB = Wheelbase of first unit (to center of rear axle group)

Rc = Centerline minimum turning radius

SA = Maximum steering angle (in degrees)

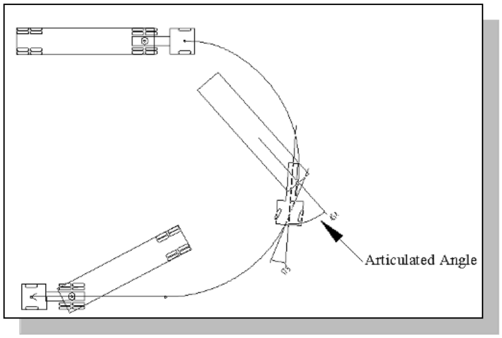
The circular arc formed by the turning path radius of the front outside tire of the vehicle is called the turning radius.



This method assumes the minimum radius is valid for a continuous turn (180 degree turn). In addition, the minimum radius is a limiting value of curvature for a given design speed and is determined from the maximum rate of superelevation and the maximum side friction factor.

Articulating Angle

The angle between adjoining units of a tractor/trailer when the combination unit is placed into a turn is measured between the longitudinal axes of the tractor and trailer as the vehicle turns. The maximum tractor/trailer angle occurs when a vehicle makes a 180 degree turn at the minimum turn radius. This angle is reached slightly beyond the point where the maximum swept path width is achieved.



Angles of as much as 90 degrees between a tractor and a trailer or between trailers may, in some cases, be possible before physical contact between units is made. However, side forces on tires are usually the limiting factor. A typical maximum between unit angles ranges from 60 degrees to 75 degrees.

While the steering angle will usually be the limiting factor in tight turn situations, keep in mind that for very long trailers, tractor/trailer and trailer/trailer angles can reach larger values.

Lock-to-Lock Times and the Effect on Simulations

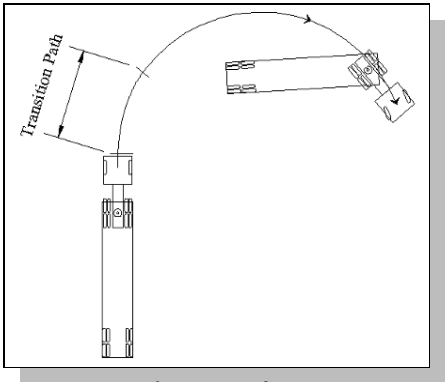
Lock-to-lock time is the time in seconds that an average driver would take under normal driving conditions to turn the steering wheel of a vehicle from the lock position on one side to the lock position on the other side. A driver turning at an intersection follows a transition path as it enters or leaves a circular horizontal curve. A curve inserted to link the tangent section of a road with the circular arc is known as a transition curve. Its radius reduces from infinity at the end of the tangent to a circular arc at the beginning of the arc.

A form of curve that approximates these requirements is alternatively referred to as the Euler spiral or clothoid. The clothoid is used to define a transition path where the radius at any point of the spiral varies inversely with the distance measured along the spiral.

$L = 0.0702V^3/RC$

Where:
L → Minimum length of spiral, m
V→ Velocity, km/h
R→ Curve radius, m
C→ Rate of increase of centripetal acceleration, m/s^3

The value C ranges from 0.75 m/s^3 for a speed of 80 km/h to 1.2 m/s^3 for 30 km/h as per AASHTO *Green Book*.



The advantage of providing a natural, easy-to-follow path for drivers is that it ensures less encroachment and promotes more uniform speeds.

Friction and Superelevation

The side (lateral) friction factor represents the lateral acceleration that acts on the vehicle. The coefficient of friction (f) is the friction force divided by the mass perpendicular to the pavement and is expressed as the following formula:

$f = (V^2/127R) – (e/100)$

Where:
f → side (lateral) friction factor
V → Velocity, km/h
R → Turn Radius, m
e → Superelevation

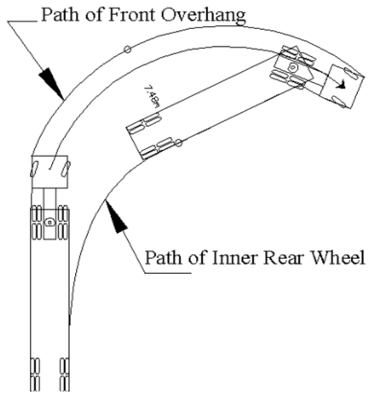
The maximum allowable side (lateral) friction factor is the point at which the side (lateral) force causes the driver to experience a feeling of discomfort when maneuvering a curve. The side friction factor at which skidding is imminent depends on the speed of the vehicle, the type and conditions of the pavement surface, and the type and conditions of the tires. The design values of the side (lateral) friction factor are based on an acceptable degree of discomfort and safety against skidding under normal driving conditions and vary with the speed from 0.32 at 20 mph (30 km/h) to 0.165 at 45 mph (70 km/h). Moreover, curves should not be designed directly on the basis of the maximum available side (lateral) friction factor.

Superelevation is the cross fall of the roadway surface in a horizontal curve which runs up to the outer extreme of the curve. Superelevation shall extend uniformly from the flow line of the gutter on the high side of the street to the lip of the gutter on the low side of the street, keeping the standard slope of the gutter on the low side unchanged.

Superelevation is used to minimize the effect of centripetal force on driver/passenger comfort and maximize the adhesion of the tire to the road when cornering. This is achieved by tilting the pavement towards the center of the curve so that centripetal forces are offset by the pavement crossfall. The maximum superelevation rates are controlled by climate conditions, terrain conditions, the type of area, and the frequency of slow-moving vehicles.
Road authorities adopt various methods of calculating the amount of superelevation required to achieve comfort and cornering ability for the traffic that is traveling at design speed. Some authorities use an empirically derived table of maximum friction demand for various design speeds to choose the amount of superelevation needed. It has been found that drivers/passengers are more tolerant of higher friction demand when traveling at low speed. As the speed increases, their tolerance of friction demand decreases. The acceptable values fall in a fairly narrow range and some authorities have adopted a fixed value for friction.

Vehicle Dimensions Affecting Swept Path

Swept path width is the amount of roadway width that a vehicle covers in negotiating a turn. It is equal to the amount of off tracking plus the width of the vehicle. The boundaries of the turning (swept) path of a design vehicle for its sharpest turn are established by the outer trace of the front overhang and the path of the inner rear wheel.



The vehicle dimensions that affect horizontal alignment and cross sections are the vehicle wheelbase, width, and minimum turn radius.

Compliance with AASHTO *Green Book*

The AutoTURN software generates a minimum turning path that is based on following AASHTO formula:

$$R = V^2 / 127 (e + f)$$

Where:

R → Radius, m

V → Velocity, km/h

e → Superelevation (m/m, e.g. 0.02 = 2/100)

f → Friction (e.g. 0.38)

Default values of superelevation and side (lateral) friction factor are based on AASHTO *Green Book* 1994 and 2001 edition.

Glossary of Terms

GLOSSARY

■ Constant Pursuit

An approximate numerical method used to calculate the positions and orientations of the body of a vehicle and any trailing articulating components as it moves forward, given a starting position and the path to be followed by the vehicle's front axle.

■ Design Vehicle

A set of dimensions for a common vehicle that defines a standard used by a particular jurisdiction for typical road design conditions.

■ Lock To Lock Time

The time in seconds that an average driver would take under normal driving conditions to turn the steering wheel of a vehicle from the lock position on one side, to the lock position on the other side.

■ Maximum Articulating Angle

The maximum angle that can occur between adjacent vehicle units, such as between the tractor and trailer of a semitrailer. This angle is measured as zero degrees when the units are aligned.

■ Minimum Turn Radius

This is defined as the radius of the circle a vehicle will turn with the steering wheels at the steering lock position. This circle passes through the centerline of the front axle for forward maneuvers and through the center of the rear axle for reverse maneuvers.

■ Pre-drawn Path

The vehicle path is drawn in the CAD Platform (AutoCAD or MicroStation) using the CAD Platform's drawing tools. This path defines a known or best guess of the path that will be followed by the center of the front axle of the current vehicle.

■ Section

A section is a single maneuver in a turn simulation.

■ Steering Lock Angle

The maximum angle the steering wheels can be turned. It is defined as the average of the angles made by the left and right steering wheels with the longitudinal axis of the vehicle when the wheels are turned to their maximum angle.

- **Superelevation**

The angle or bank of a road surface. It is expressed as a slope. For example, a superelevation of 0.01 is a one-percent slope, which rises 1 unit for every 100 units of horizontal distance.

- **Swept Path**

The swept path of a vehicle is the envelope swept out by the sides of the vehicle body, or any other part of the structure of the vehicle. A swept path determines whether the vehicle will make contact with vertical obstructions. Create a swept path simulation by selecting the Vehicle body envelope in Simulation Properties.

- **Tire Tracks**

The paths taken by the front and rear-most tires. Create a tire track simulation by selecting Front and Rear tire envelopes in Simulation Properties.

- **Track**

The track of a vehicle is defined as the distance between the outside edges of the outermost tires on the left and right sides.

- **Turn Simulation**

A connected series of one or more vehicle maneuvers. The simulation comprises the path, the vehicle and the tire track or vehicle body envelopes.

- **User-Defined Vehicle**

A set of dimensions for a common or user-created vehicle.

- **Width**

The distance between adjacent sides of a vehicle.