SIMCENTER 12 MOTION
Beta
Contents

Rigid links ................................................................. 3
1. Model review .......................................................... 4
2. Solve the model ......................................................... 8
3. Post-process results .................................................. 13
4. Advanced animation features ...................................... 23

Flexible links ............................................................. 29
1. Create a flexible link .................................................. 30
2. Solve the model ......................................................... 47
3. Post-process results .................................................. 54
4. Advanced animation features ...................................... 58

Coupling with 1D through Mechatronics .............................. 66
1. Create a Mechatronics element ................................. 67
2. Solve the model ......................................................... 83
3. Post-process results .................................................. 86
Rigid links

Demonstrating: solution features of a Simcenter Motion model with rigid links based on CAD geometries

Duration: 25 min

Prerequisites:

- Simcenter 3D Motion build:
  - 12 IP22

- Licenses:
  - Simcenter Engineering Desktop (Bundle or Add-on)
  - Simcenter 3D Motion Modeling and Solver

- Data:
  - Directory: 01-Rigid_Links
  - Backup Package: WheelLoader_ip22_RIGID_FINAL.zip

Description:

This activity shows some of the functionalities available with Simcenter 3D Motion applied on an existing model with rigid links:

- Rigid links creation / editing, component-based or geometry-based links.
- Dynamic solution of a mechanism
- Results post-processing
- CAD-based advanced animation features: clearance measurement.
1. Model review

As a starting point we open the existing wheel loader model.

**Tip:** before loading the Motion model, when starting Simcenter please make sure that the assembly load options are set properly in order to find all required part files when loading the Motion model:

- **Home** tab, then **Assembly Load Options**. Make sure that **From Folder** is set as **Load** option. Close the dialog with the OK button.

- Open the file **WL_Analysis_Completed.sim** included in the Full_Assembly subfolder.

![Assembly Load Options](image)
All rigid links, joints, connections, forces (including 3D contacts) are already defined in the model. Take some time to explore the Motion Navigator where all Motion elements are listed.

Motion links can be defined as referencing existing CAD geometries. Depending on the selected objects, any link can be either component-based or not:

- Component based: selected objects are only CAD components (CAD parts or assemblies, represented by PRT files)
- Generic: one or more selected objects are not components, i.e. other geometries like points, curves, surfaces, solids etc.

Component-based links are displayed in the Motion Navigator with an additional small box in the bottom right side of the link icon: in this model it is easy to verify that most links except the Terrain are component-based, please see the picture below (please note that the icon for the Terrain link also contains a different symbol, showing that the link is fixed to ground).
When selecting any link, all related elements are highlighted in the Motion Navigator.

Select the Bucket link.

Expand the other branches in the Motion Navigator and quickly verify what elements are referencing the Bucket link by detecting those displayed in a different color:
2. Solve the model

A dynamic solution is already available with the name *Rigid*. No results are included in the original package, so we need to launch a solution to verify the dynamic behavior of the system. Solution parameters can be edited by opening the solution item in the Motion Navigator.

Right-click or double-click the *Rigid* branch and check the attributes of the solution (e.g. end time, number of output steps, solver parameters etc.):
Launch the *Rigid* solution (either right-click on the *Rigid* branch and select *Solve...* or hit the *Solve* button in the *Analysis* tab).

When using the original solution attributes, the analysis will take approx. 2-3min to complete.
A set of files are created in the same folder where the Motion simulation file is stored, each one by default with the name of the Motion model + the name of the simulation. In particular:

- `wl_analysis_completed-rigid.minp` (Motion Solver Input file): ASCII file with the description of the model in the Motion Solver format
- `wl_analysis_completed-rigid.mres` (Motion Solver Results file): binary file with all results from the solved analysis
- `wl_analysis_completed-rigid.minf` (Motion Solver Info file): ASCII file with info about the model and the solution, including the total simulation time.

Please note that for each 3D contact couple of links two .OFF files are also created in the working directory, representing the tessellated geometries used to detect and calculate the contact forces during the simulation.

After the completion of the simulation, we’ll animate the results to check if the model works as expected.

- From the Motion group in the Analysis tab, select Animation

- Start animating the mechanism by hitting the Play button in the Animation dialog
**Tip:** move the slider manually to speed up the animation or to go to any desired time step.

**Tip:** for more animation options access the *Results* tab, then click on *Return to Model* when done.
3. Post-process results

In this part of the activity we will see how to create 2D plots of any of the available simulation results.

In particular we will plot the following data (some of them will be compared with other solutions in a later activity):

- Tire forces
- Lift cylinders displacements and forces
- Vehicle chassis trajectory in the XY (horizontal) plane.

Expand the XYZ Result View section at the bottom of the Motion Navigator. The available outputs of the selected element (the front left tire, in this case) will be listed – if not, just make sure the tire is correctly pre-selected.

Expand the Force branch, then right-click on the Normal component and select Plot.
In the Viewport dialog box, select Create New Window (middle button). A new graph with the time-dependent vertical force on the front left tire will be displayed in a separate window. Leave it open for the next step.
Back to the Motion Navigator, select the front right tire (Tire_FR) to display the available outputs in the XY Result View section.

Right-click on the Normal force, then select Overlay: the selected output will be added to the open graph.
To access any desired results more easily from the Motion simulation results, proceed as follows:
Select the front left tire in the Motion Navigator, then right-click on the normal force and select Create Graph Object: this will append a new result item under the XY-Graphing branch, which can be referenced at any time to create a new graph (right-click, then select Plot or Overlay).

Repeat the previous step with the front right tire. When both left and right front tire normal forces are displayed, they can be selected together to create directly a new graph showing both of them.

On an open Graph Window element, expand the upper Toolbar and activate the Probing mode to display a cursor.
Drag the cursor at any desired time step to display the numerical value of any displayed results.

Keep the graph window open, then animate the model again: the cursor position will be in sync with the animation.

Repeat the previous steps to add the following outputs to the XY-Graphing branch of the results set:

<table>
<thead>
<tr>
<th>Motion element name</th>
<th>Motion element type</th>
<th>Component name</th>
<th>Comment</th>
</tr>
</thead>
<tbody>
<tr>
<td>LiftPiston_CYL_Left_LiftCylinder</td>
<td>Cylindrical Joint</td>
<td>Relative Force Magnitude</td>
<td>Left lift cylinder total force</td>
</tr>
<tr>
<td>LiftPiston_CYL_Right_LiftCylinder</td>
<td>Cylindrical Joint</td>
<td>Relative Force Magnitude</td>
<td>Right lift cylinder total force</td>
</tr>
<tr>
<td>LiftPiston_CYL_Left_LiftCylinder</td>
<td>Cylindrical Joint</td>
<td>Relative Displacement Magnitude</td>
<td>Left lift cylinder extension</td>
</tr>
<tr>
<td>LiftPiston_CYL_Right_LiftCylinder</td>
<td>Cylindrical Joint</td>
<td>Relative Displacement Magnitude</td>
<td>Right lift cylinder extension</td>
</tr>
<tr>
<td>----------------------------------</td>
<td>-------------------</td>
<td>---------------------------------</td>
<td>-----------------------------</td>
</tr>
<tr>
<td>RearChassis</td>
<td>Link</td>
<td>Absolute Displacement X</td>
<td>X-coordinate of the Center of Gravity of the RearChassis link</td>
</tr>
<tr>
<td>RearChassis</td>
<td>Link</td>
<td>Absolute Displacement Y</td>
<td>Y-coordinate of the Center of Gravity of the RearChassis link</td>
</tr>
</tbody>
</table>

By default all plots are time-dependent, i.e. the time is used as X-axis. To plot the XY trajectory of the vehicle’s chassis, the X displacement should be used as X axis:
Right-click on the X displacement output under the XY Graphing, then select **Set as X-axis**.

Right-click on the Y displacement, then select **Plot**.
Create a new Graph Window to display the trajectory followed by the rear chassis body of the wheel loader model.
4. Advanced animation features

In this last part of the activity we will calculate and display the minimum distance between two geometries associated to two links of the model, as measured during the simulation. A threshold value will be set, in order to highlight any time steps when this distance becomes smaller than a desired value.

⚠️ From the Motion group in the Analysis tab, select Measure.

⚠️ Set the Type to Minimum Distance

⚠️ As First Set select the face of the front chassis solid body as showed below (and the same face on the other side of the chassis)
As Second Set select the face of the rear chassis solid body as showed below.

Set the Threshold value to 90mm.
Set the Measure Condition as Less Than.

Keep the default tolerance value (0.01 mm).

Make sure both Stop On Event and Make Active check boxes are selected.

Click the OK button to close the dialog. This will create a new Measure element in the Motion Navigator.

Start a new animation, then expand the Packaging Options section and make sure the Measure and the Stop on Event checkboxes are selected.
Animate the results with the Play button.

The minimum distance between the selected geometries will be dynamically displayed at any time step, between the two closest points. The animation will automatically stop at the time (around 17.8s) when the measured distance is smaller than the specified threshold value (90mm).
This completes the tutorial on rigid links with Simcenter 3D Motion.
Flexible links

**Demonstrating**: implementation of flexibility of links in a Simcenter 3D Motion model

**Duration**: 35 min

**Prerequisites:**
- Simcenter 3D Motion build:
  - 12 IP22
- **Licenses**:  
  - Simcenter Engineering Desktop (Bundle or Add-on)  
  - Simcenter 3D Motion Modeling and Solver  
  - Simcenter 3D Motion Flexible Body
- **Data**:  
  - Directory: 02-Flex_Links  
  - Backup-Package: WheelLoader_ip22_FLEX_FINAL.zip

**Description:**
This activity shows some of the functionalities available with Simcenter 3D Motion applied on an existing model with flexible links:
- Flexible links creation / editing
- Dynamic solution of a mechanism with flex links
- Results post-processing: time domain and frequency (modal) domain
1. Create a flexible link

As a starting point we open the existing wheel loader model, as saved in the rigid link tutorial (a copy of that model is already available in the WheelLoader_ip22_RIGID_FINAL.zip file).

To make one of the links in the model as flexible, a modal analysis result file is needed: this can be found in the Flex.zip file, to be extracted anywhere in the file system (in this document it is assumed that the Flex folder will be extracted in the same directory of the Motion model).

Tip: before loading the Motion model, when starting Simcenter please make sure that the assembly load options are set properly in order to find all required part files when loading the Motion model:

- Home tab, then Assembly Load Options. Make sure that From Folder is set as Load option. Close the dialog with the OK button.
Open the file *WL_Analysis_Completed.sim* included in the Full_Assembly subfolder.
This model already includes a set of results as output from a solution called Rigid. Since we are going to add flexibility to one of the links in the model and solve the modified mechanism, we need to add a *Flexible Body* solution:
Home tab, Solution group: drop-down menu, hit the Solution button – or – right-click the top level Motion Navigator branch (WL_Analysis_Completed), then select New Solution.

Set the following attributes:

- Solution Type: Flexible Body
- Analysis Type: Kinematics/Dynamics
- End time: 36s
When creating a new solution, it is possible that some elements in the Motion Navigator are re-enabled. In this case, the `Load_Force` element must be manually disabled to keep the same configuration as the `Rigid` solution.

- Right-click on the `Load_Force` element, then select `Deactivate for all`. 
This will keep this element as inactive for any solution in the model. The new status of the element is also showed in the Motion Navigator.

As next step we will create a new flexible link to take into account the deformation of one part of the model, i.e. the boom link.

ียว Home tab, Flexibility group: hit the Flexible Link button – or – right-click the top level Motion Navigator branch (WL_Analysis_Completed), then select New Flexible Link
In the Flexible Link dialog box, set the following attributes:

- **Type:** Finite Element Method
- **Link:** select the *Boom* link, either from the Graphics Window or directly from the Motion Navigator
- **Flexible Model:** browse for file “boom_sim1-solution_nxnastran.op2” in the Flex subfolder (make sure that *Nastran Results Files (*.op2)* is specified as file type)
Tip: toggle the Preview checkbox to show/hide a separate window with the FE mesh included in the imported OP2 file.
Set Import Units as follows: *(mN)(mm)(kg)* – keep *Infer* selected to get these units automatically from the OP2 file.

- Placement: *Component Position* – check where the FE mesh is placed in the model.
- Name: *Boom_Flex*

- Close the Flexible Link dialog box with the OK button.

The Finite Element mesh is displayed in the model. A new branch is created in the Motion Navigator with the flex link.
Any flexible links are only supported when the active solution is of type Flexible Body. To verify that:

- Double click on the **Rigid** solution (or right-click and select **Make Active**)

The flexible link is still in the Motion Navigator as inactive element.
## Motion Navigator

<table>
<thead>
<tr>
<th>Name</th>
<th>Status</th>
<th>Environment</th>
</tr>
</thead>
<tbody>
<tr>
<td>Full_Assembly</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Links</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Joints</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Couplers</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Connectors</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Markers</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Sensors</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Driver Container</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Load Container</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Load_Force</td>
<td>Inactive for All</td>
<td></td>
</tr>
<tr>
<td>Flexible Links</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Boom_Flex</td>
<td>Inactive</td>
<td></td>
</tr>
<tr>
<td>Vehicle Components</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Contacts</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Rigid</td>
<td>Active</td>
<td>Normal Run</td>
</tr>
<tr>
<td>Results</td>
<td></td>
<td>Results may need update</td>
</tr>
<tr>
<td>Animation</td>
<td></td>
<td></td>
</tr>
<tr>
<td>XY-Graphing</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Load Transfer</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Flex</td>
<td></td>
<td>Flexible Body</td>
</tr>
</tbody>
</table>
Let’s now activate the flexible solution again.

Double click on the Flex solution (or right-click and select Make Active)

Before solving the model, we will edit some of the properties of the flexible link.

First of all, let’s check the status of the connections between the flex link and the mechanism, as available in the Connections tab of the Flexible Link dialog box: the full list is displayed, with some info about each couple of Motion connection element and the corresponding interface node in the FE mesh. In order to get reasonable results from a solution, these items should be coincident or very close to each other: the resulting distance is also reported in the list, so it should be close to zero for any connections. A tolerance can also be set to highlight any connections where the distance between the Motion marker and the FE node is larger than a threshold value. In our case, all connections stay well within the prescribed tolerance (2mm by default).
The next tab to be checked is Mass Invariants: these components take into account all mass properties of the flexible link as calculated from the FE mesh, including the mass distribution and the effects of deformations on mass distribution changes. Some components are always required by the Motion solver to solve the mechanism, whereas some other higher-order components can be disabled to speed up the calculation times, at a cost of lower accuracy in case of large deformations or large rotations of the flexible link, which is not the case with our model:
De-select all selectable Mass Invariants components

In the Advanced tab the user can set some further options usually allowing to reduce the calculation times:

- Frequency Filtering: artificial reduction of the frequency of higher modes, mostly representing local deformations at the attachment points
- Initial transient: artificial increase of modal damping for all modes in an initial transient time.

Enable the Frequency Filtering, then set the following parameters:

- **Cutoff Lower Frequency** = 300Hz
- **Cutoff Upper Frequency** = 400Hz
- **Cutoff Frequency Increment** = 10Hz
- **Cutoff Damping** = 0.5 (50% of critical damping)

**Tip:** in principle, the lower are the cutoff frequencies, the higher is the effect on CPU time reduction. However, to keep a good accuracy of results, a usual best practice is to use as cutoff lower frequency a value which does not affect the first 10-12 modes of the flex link, and in any case no lower than 2 times the highest frequency of interest for the model: in this case accurate results can be expected up to 150Hz. Later in this document we will see how to display in the info file the comparison between the original set of modes and the resulting set from the frequency filtering.

- Enable the Transient Damping, then set the **Transient Time** to 1s
- Close the dialog with the OK button.
As next step we will modify the modal damping of the flexible link. By default a 5% of the critical damping is applied, we will change it to 3%.

- Right-click the flexible link in the Motion Navigator, then select *Edit Damping Factor*
- As %**Viscous** damping value, set 3%.
- Close the dialog box with the OK button.

All relevant info about the modal content of a flex link can be accessed from the *Mode Shape Details* panel below the Motion Navigator: a list of all modes is displayed, with info about active modes (by default all except the first 6 rigid body modes), mode natural frequency, damping values etc.

**Tip:** by right-clicking on any row it is possible to edit the selected mode separately, for instance to enable or disable a mode (not recommended) or set a different damping value only for that mode.

To access the modal content and check the applied damping values:

- Select the flex link in the Motion Navigator
- Enlarge the *Mode Shape Details* panel
- Verify that modal damping is set to 3% for all modes.
2. Solve the model

In this part of the activity we will solve the mechanism with the flexible boom.

Launch the Flex solution (either right-click on the Flex branch and select Solve... or hit the Solve button in the Analysis tab).

When using the original solution attributes, the analysis will take approx. 4-5min to complete.

**Tip:** the total CPU time, as well as several other info on the completed solution, can be easily seen in the Motion Solver Info file, accessible either in the model folder (MINF extension) or directly by right-click the flex solution branch in the Motion Navigator:

The total CPU time is reported as the very last info at the end of the MINF file.
Another interesting info available in the MINF file is the comparison between the original and the modified set of modal frequencies and damping values, easily found by searching for the string “frequency filtering”: 
After the completion of the simulation, we’ll animate the results to check if the model works as expected.

**Tip:** when working with flexible links, before animating the solution it can be helpful to access the Motion Preferences and deactivate the calculation of min/max deformations. This allows to save time during animations, in particular when dealing with large FE meshes and/or several flexible links in the same model:

- **Menu button** – or **File tab** – then **Preferences** – **Motion**
Make sure the *Use Absolute Min/Max values for Flexible Body* is unchecked.

From the *Motion group in the Analysis tab, select Animation*.
Start animating the mechanism by hitting the Play button in the Animation dialog.

Tip: move the slider manually to speed up the animation or to go to any desired time step.

Tip: for more animation options access the Results tab, then click on Return to Model when done.
3. Post-process results

In this part of the activity we will compare some of the simulation results available for the two solutions: rigid and flex.

In particular under the *Rigid* solution the following data are already listed as *Graph Objects*:

- Tire forces
- Lift cylinders displacements and forces
- Vehicle chassis trajectory in the XY (horizontal) plane.

We will verify the effect of the flexibility of the boom on the tire forces.

- Expand the *XY Result View* section at the bottom of the Motion Navigator
- In the Motion Navigator expand the *Vehicle* branch, then select the front left tire (*Tire_FL*)
- Expand the *Force* branch, then right-click on the *Normal* component and select *Create Graph Object*

- Right-click on the new result item created under the *XY-Graphing* branch, then select *Plot*
Keep the graph open, then activate the *Rigid* solution. This might take a few sec.
Right-click on the front left tire force under the XY-Graphing branch, then select Overlay. Both flex and rigid simulation results will be displayed in the same graph.
4. Advanced animation features

In this last part of the activity we will see some additional post-processing and animation features available for flexible links in the *Post Processing Navigator*.

Two main categories of results are available in the Post Processing Navigator:

- Frequency domain: mode shapes
- Time domain: deformations (and any other output types as defined in the modal analysis result file OP2).

In this activity we will focus on mode shape visualization / animation in the frequency domain. The workflow to display time-domain deformations is very similar, so it is not covered by this document.

To display any mode shapes of the flexible link:

- Expand the first item (*boom_sim1-solution_nxnastran*) under the *Imported Results* branch
- Expand the desired mode, then expand the *Displacement – Nodal*
- Right-click on *Magnitude*, then select *Plot*
Mode shapes can be animated from the Results tab – Post Processing group:

- Click on the Play button to start the animation of the mode deformation
Navigate through all available mode shapes by clicking on the Previous Mode/Iteration – Next Mode/Iteration buttons
Click on the **Animation** button to edit the properties of the animation and see the effect of any changes on the current animation:

- **Style**: set to **Modal**
- Check the **Full-cycle** box
- Set **frame delay** to **100ms**

By accessing the **Post View** properties it is possible to modify the visualization of the flexible link.
Hit the Post View button in the Post Processing group of the Results tab

OR

Right-click the Post View branch under the Viewports – Contour Plots branch and select Edit Postview

Try the following changes and check the effects on the current animation (click on the Apply button after each change):

- **Results tab**: navigate through any desired modes from the Result Selection section
- **Display tab**:
  - change the Color Display from Smooth to Banded
  - disable / re-enable the Lighted option
- **Deformation tab**:
  - change the deformation scale by modifying the percentage or the absolute value
This completes the tutorial on flex links with Simcenter 3D Motion.
Coupling with 1D through Mechatronics

**Demonstrating:** implementation of 1D subsystems in a Simcenter 3D Motion model

**Duration:** 30 min

**Prerequisites:**

- **Simcenter 3D Motion build:**
  - 12 IP22

- **Licenses:**
  - Simcenter Engineering Desktop (Bundle or Add-on)
  - Simcenter 3D Motion Modeling and Solver
  - **Simcenter 3D Motion Systems & Controls**
  - LMS Imagine.Lab Amesim

- **Data:**
  - Directory : 03-Mechatronics
  - Backup-Package: WheelLoader_ip22_AMESIM_FINAL.zip

**Description:**

This activity shows some of the functionalities available with Simcenter 3D Motion applied on an existing model coupled with a 1D subsystem:

- Definition of a *Mechatronics* element with control inputs and outputs
- Co-simulation with 1D (Amesim)
- Results post-processing
1. Create a Mechatronics element

Both the rigid and the flexible links wheel loader FINAL models can be used as starting point for this activity (WheelLoader_ip22_RIGID_FINAL.zip or WheelLoader_ip22_FLEX_FINAL.zip file). In case of limited time available, it is recommended to start from the rigid final model. All details and screenshots of this document are related to the flexible link final model.

In order to complete this activity, an Amesim file is needed: this can be found in the Amesim.zip file, to be extracted anywhere in the file system (in this document it is assumed that the Amesim folder will be extracted in the same directory of the Motion model).

Tip: before loading the Motion model, when starting Simcenter please make sure that the assembly load options are set properly in order to find all required part files when loading the Motion model:

- Home tab, then Assembly Load Options. Make sure that From Folder is set as Load option. Close the dialog with the OK button.
Open the file WL_Analysis_Completed.sim included in the Full_Assembly subfolder.

This model already includes two sets of results as output from two solutions (Rigid, Flex). Since we are going to add a mechatronics element in the model and solve the modified mechanism, in order to save the simulation results in a different results set we need to add a new solution. Furthermore, later we will compare the same results from the two solutions, so instead of creating a new solution it can be more convenient to clone an existing one where graph objects are already defined:

Right-click the existing Flex solution and select Clone

Solution Name: Cosim_Amesim. When asked, select 1. Rename Both.
**Tip:** Cloning a solution also keeps the simulation parameters unchanged, as well as the initial status of any active / inactive elements in the Motion Navigator.
As next step we will create the control elements needed to couple our 3D model with a 1D representation of a control and hydraulic actuation system in Amesim, with the following convention:

- **Control inputs**: data measured from 3D and sent as input to 1D (typically displacements, velocities and accelerations)

- **Control outputs**: data calculated from 1D and applied to the 3D model (typically loads).

In our example, the 1D subsystem represents the two lifting cylinders with a desired time-dependent extension and a simple PID control on the cylinders pressure, such that lift forces are applied to the 3D model in order to follow as close as possible the desired values. The Amesim model (.ame) is included in the AMESIM folder.
Procedure to create control inputs (displacements and velocities of the two lift cylinders, in this case):

- **Home** tab, **Control group**: hit the **Control Input** button
As Measurement link, select *Lift_Cyl_Left*

To fill both *Origin* and *Orientation* with one pick, as Origin select the inner or outer circle at one end of the lift cylinder left geometry (make sure that the local Z axis will be oriented along the axial direction of the cylinder).

Tip: to select the desired geometries more easily, hide all links except the one required for the selection.
As Relative and Reference link select Lift_Pist_Left

Select the outer arc at the end of the piston geometry as origin and repeat for orientation (local Z axis along the piston axial direction)

Create a first port by clicking on the Add New Port button in the Ports section
Fill in the *Port Setting* section as follows:

- **Name**: Disp_PL
- **Component**: Z
- **Variable**: Displacement

Without closing the dialog, create a new port with the following properties:

- **Name**: Vel_PL
- **Component**: Z
- **Variable**: Velocity

Set the name as *Cin_Left*.

Close the Control Input dialog with the OK button.

Repeat the previous steps to create a new Control Input element for the lift cylinder on the right side.
**Tip:** in principle, any names can be used for ports. But when using the same names as the corresponding connection signals in the Amesim model the coupling with 1D is done automatically, otherwise a manual mapping must be done when creating the Mechatronics element, as described later in this document.

The two new control input elements are created under a new branch in the Motion Navigator called *Control Input*. 
In the next step of the activity we need to establish the coupling with the Amesim model through a *Mechatronics* element.

- **Home tab, Control group:** hit the *Mechatronics* button

- Set *Type* attribute to *Amesim*

- Set *Import as Parameter*

- Browse to the Amesim file (*HydraulicExcavator_lift_1_PID.ame*) included in the Amesim folder

When importing the Amesim model, an information window will show the automatic associations between the Amesim ports and the Motion ports.

In our example the *Integration Type* attribute will also be automatically set to *Model Exchange*. This is due to the type of interface block defined in the Amesim model.
Tab *Input Ports*:

- Make sure that all ports are automatically set (green check marks). If not, select manually the correct input ports from the control inputs in the Motion Navigator.
- From the *Port Settings* table, select the port *Disp.PL*.
- Set *Offset* to 2403.
- Repeat the previous steps for the *Disp.PR* port.
**Tip:** offsets and scale factors are the easiest way to adapt the 3D model with the sign and units conventions used in the 1D subsystem, without need of applying any modifications to the models.

- **Tab Output Ports:**
  - A list of all output ports as read from Amesim is available
  - Leave all scale factors to 1 and offsets to 0.

- Close the Mechatronics dialog with the OK button.
Now we need to specify where the output signals from Amesim will be applied to the mechanism. This can be done by adding two Control Output elements, to be associated with the Output Ports automatically created in the Mechatronics dialog when importing the Amesim model.
Home tab, Control group: hit the Control Output button

- Set Type to Joint/Constraint
- As Joint/Constraint select the cylindrical joint LiftPiston_CYL_Left_LiftCylinder
- As Output Variable select Force
- As Port selection, expand the Mechatronics element in the Motion Navigator and click on the Force_PL branch to complete the definition of the first control output element
- Change the name to Cout_Left
- Close with the OK button
- Repeat the previous steps to create a new control output on the right lift cylinder.
Before solving the model, we need to make sure that the original drivers on the lift cylinders are disabled, in order to let the 1D subsystem actually drive the model by applying the calculated hydraulic forces.

- Expand the *Driver Container* branch in the Motion Navigator
- Select *Driver_Lift_Left* and *Driver_Lift_Right*, then right-click and select *Deactivate*.
The two drivers applied on the lift cylinders will be left as inactive in the Motion Navigator only for the *Cosim_Amesim* solution.

![Motion Navigator diagram]

**Motion Navigator**

<table>
<thead>
<tr>
<th>Name</th>
<th>Status</th>
<th>Environment</th>
</tr>
</thead>
<tbody>
<tr>
<td>Diff_ShaftOut_RL-DRV-Wheel...</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Diff_ShaftOut_RR-DRV-Wheel...</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Driver_Driveline</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Driver_Dump_Left</td>
<td></td>
<td>Normal Run</td>
</tr>
<tr>
<td>Driver_Dump_Right</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Driver_Lift_Left</td>
<td>Inactive</td>
<td></td>
</tr>
<tr>
<td>Driver_Lift_Right</td>
<td>Inactive</td>
<td></td>
</tr>
<tr>
<td>Steering_Driver</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Load Container</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Flexible Links</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Vehicle Components</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Control Input</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Control Output</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Mechatronics</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Contacts</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Rigid</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Flex</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Cosim_Amesim</td>
<td>Active</td>
<td>Flexible Body</td>
</tr>
</tbody>
</table>
2. Solve the model

In this part of the activity we will solve the mechanism with the lift cylinders driven by the Amesim model.

_launch the _Cosim_Amesim_ solution (either right-click on the _Cosim_Amesim_ branch and select _Solve_... or hit the _Solve_ button in the _Analysis_ tab).

When using the original solution attributes, the analysis will take approx. **4-5min** to complete.
After the completion of the simulation, we’ll animate the results to check if the model works as expected.

- From the Motion group in the Analysis tab, select Animation

Tip: move the slider manually to speed up the animation or to go to any desired time step.

Tip: for more animation options access the Results tab, then click on Return to Model when done.
Animation Sampling Rate:
Samples animation by the number of steps specified.
3. Post-process results

In this part of the activity we will compare the results from the lift cylinders as driven by Motion elements (joint drivers, Flex solution) or by Amesim (Cosim_Amesim solution).

In particular under the cloned Cosim_Amesim solution the following data are already listed as Graph Objects:
- Tire forces (not used for comparison)
- Lift cylinders displacements and forces.

Lift forces were measured from joint drivers, which are disabled in the current solution. Therefore we need to measure these forces as values of the Control Output elements, calculated by Amesim and applied to Motion.

- Expand the XY Result View section at the bottom of the Motion Navigator
- In the Motion Navigator expand the Mechatronics branch, then select the Force_PL port
- In the XY Result View right-click on the Force_PL component and select Create Graph Object
Repeat the previous steps with the right side component *Force_PR*. 
Now we will compare both displacements and lift forces between the Flex and the Cosim_Amesim solutions.

Right-click on any of the new result items created under the XY-Graphing branch (e.g. Force_PR), then select Plot (Create a new window).
Keep the graph open, then right-click on the lift cylinder displacement measure (LiftPiston_CYL_Right_LiftCylinder->MAG,Displacement(rel)) and create a new plot on a separate window.
While keeping both graphs open, activate the *Flex* solution.

Right-click on the right lift cylinder force (`LiftPiston_CYL_Right_LiftCylinder-FM,Force(rel)`) under the XY-Graphing branch, then select *Overlay*.

Select the graph window where the lift cylinder force from the Amesim cosimulation is plotted, in order to display the original driver force on top of the 1D hydraulic force.
Right-click on the right lift cylinder displacement \((\text{LiftPiston\_CYL\_Right\_LiftCylinder\textendash}MAG,\text{Displacement}\text{(rel)})\) under the XY-Graphing branch, then select Overlay.

Select the graph window where the lift cylinder displacement from the Amesim cosimulation is plotted, in order to display the original driver force on top of the 1D hydraulic force.
When zooming on the transient phases it can be clearly seen some time delay between the desired lift displacement (as rigidly imposed by the joint drivers) and the resulting displacement by applying hydraulic forces as output from the 1D control system.

This completes the tutorial on mechatronics simulations with Simcenter 3D Motion.