FEDEM Tutorial: Suspension

Part1: Basic modelling

Prerequisites

Mesh components (Nastran bulk data files):

- lca.nas (sub frame)
- knuckle.nas (right lower control arm)
- uca.nas

(All meshed parts are in units of meters and Newton)

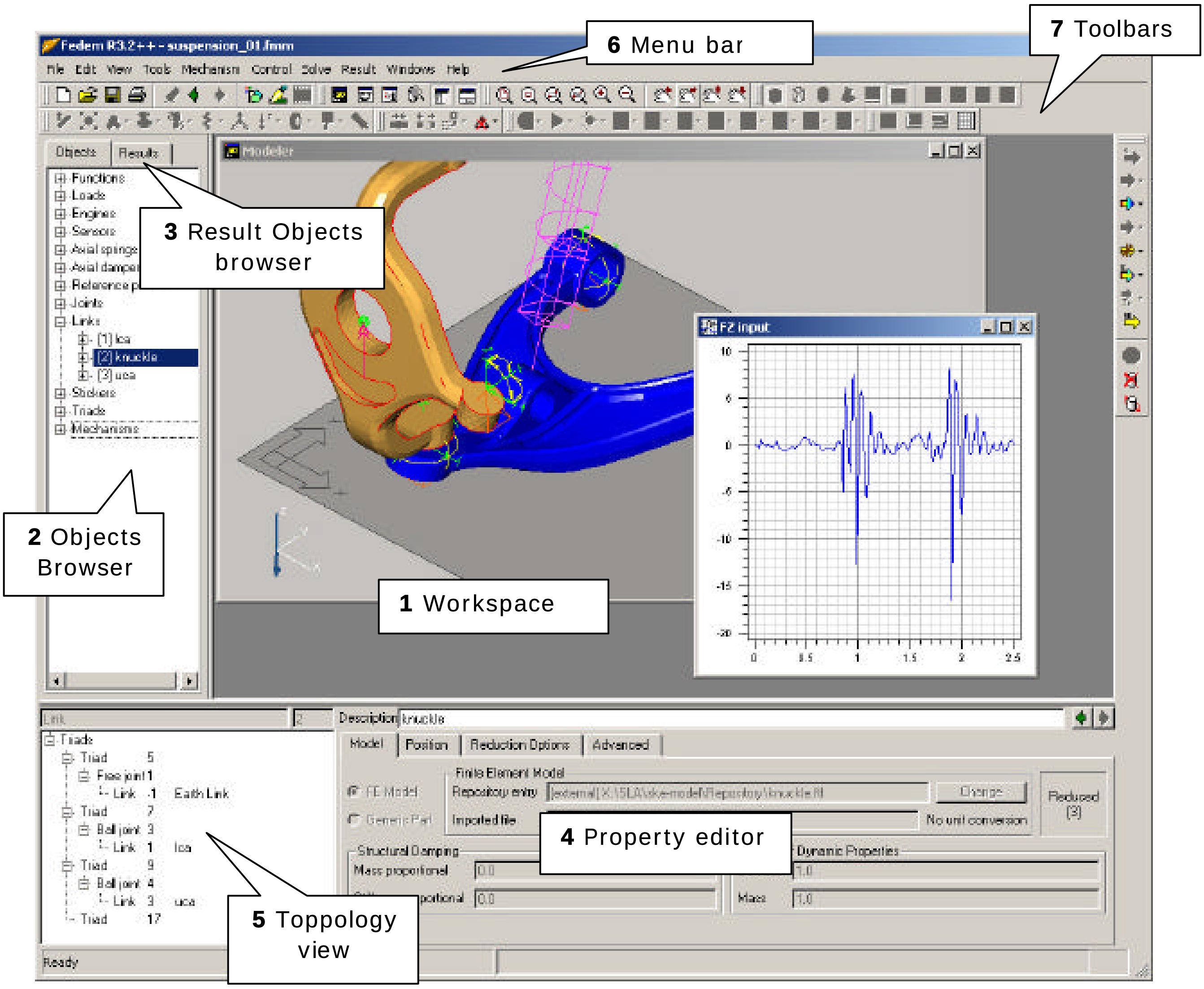
Content of the tutorial

- Building a simplified one-side front suspension
- Apply road load data as driving force input
- Perform a dynamics simulation.
- Perform a stress recovery.

The FEDEM User Interface

The FEDEM User Interface, as shown in the picture below, consists of several areas:

1. The workspace, where the 3D modelling window, the Control system environment window and graph windows are shown.
2. The Objects browser, where all the mechanism elements are listed
3. The Result Objects browser, where all the graphs, curves and animations are listed
4. The Property editor, where you can edit the properties of the model elements
5. The Topology view, showing the topology of the model
6. The Menu bar
7. The Toolbar area, showing several toolbars for quick access to model creation commands



Some of the buttons on the toolbars have several sub-items. This is shown by an arrow in the lower right corner on the button. E.g. the point-to-point joint creation commands can all be found beneath the revolute joint button:

To select a sub-item, click-and-hold the button until the sub-menu shows, and then select the command you want.

Setting up the model

1. Start FEDEM
2. Select **Save As** from the **File** menu; select the directory where you will locate the model file, and type a filename in the file dialog.

3. Select **Set model link repository** from the **File** menu, select the same location as the model file. Select **Create new folder button** in the file dialog and select or create a directory to be used as repository. Mark this directory and click **OK**.
4. Click the **General appearance** button . The General Appearance dialog opens. Set the symbol Size to 0.02
5. In the Object browser, select the reference plane, and set height and width to 0.3.

NOTE: *The use of a model link repository is not required, but it has the advantage that reduced parts (matrices) can be shared between many separate FEDEM models sharing all or some components.*

Additional steps when working in units of mm/N

- Select Model preferences from the File menu. In the dialog select the Mg/mm/N. Change the position tolerance to 0.01 (mm).
- Click the General appearance button . The Generic Appearance dialog opens. Set the symbol Size to 20.
- In the Object browser (left window pane) select Mechanism, and change the gravity field value in the property window to –9810 (mm/s^2).
- In the object browser, select the reference plane, and set height and width to 1000.

NOTE: *FEDEM is unit independent, and it is not <u>required</u> for the user to set units. <u>It is though required that consistent units are being used</u>. However, the setting is beneficial for later use.*

Loading links

1. Click the load link button or select **File -> Load link**. The file dialog opens. Locate the file "lca.nas" which contains the nastran format mesh of the sub frame and click **Open**.

The element model will now be loaded and displayed in the graphics window. If you can't see it press the **Zoom All** button, or select **Zoom All** from the **view** menu.

2. Click the **Item appearance** button, and then click on the lower control arm. The item appearance dialog opens. Select Surface for the polygons and outline for the lines. Adjust the colours to preferred values.

NAVIGATING THE GRAPHICS WINDOW

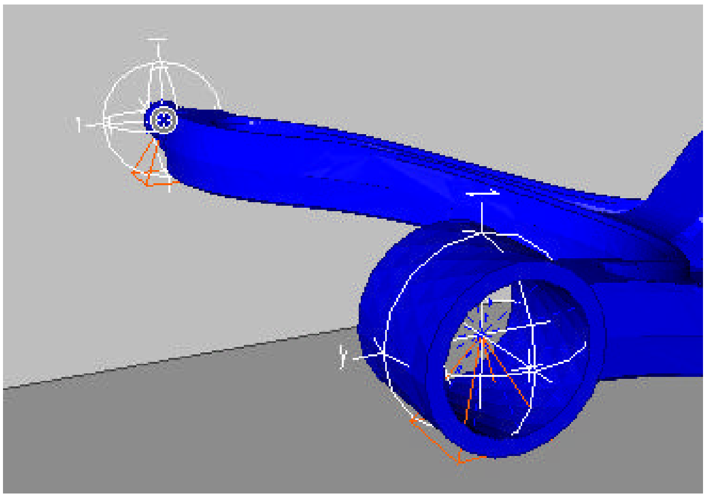
Press F1, F2 or F3 and move the mouse in order to navigate in the graphics window.
F1 + mouse move = pan
F2 + mouse move = zoom in/out
F3 + mouse move = rotate
F4 + mouse click = select centre of rotation

Creating Joints

1. Click and hold the pull down button and select Ball joint or select **Ball joint** from the **Mechanism** menu. Note the instructions in the lower left taskbar field. Zoom in to the centre of the frame mounts of the lower control arm and click at first

centre node. If the preview point marker does not appear to be in the right spot, click again. When the right spot is marked click **middle mouse button** to accept (Done). If you don't have a middle button, use ENTER key. (It may be beneficial to switch to line view in this operation)

2. After pressing DONE in last step, the message bar asks you to pick again for the next joint to position. Pick the second frame mount centre, click DONE when the correct point is pre-marked, and right-click to exit the command cycle. The graphics should now look similar to the picture below.



TIP: *If you're having trouble finding where to put the joints, switch to Line View* .

TIP: *If the graphics window is inactive, activate by clicking at an empty spot.*

TIP: *If you have problem in zooming in, try to change to parallel view or reset the rotation centre by pressing the F4 button and clicking a point.*

3. The joints must now be attached. Click the **Attach** button. The message bar now asks you to select an object to attach. Select the first ball joint. When the ball joint turns red, click the **middle** mouse button and select the **reference plane**. Then click Done (middle button). The master triad should now turn green. The message bar now asks you to select a new object to attach. Select the same joint, accept, select the control arm and accept. Now the ball joint should now turn yellow.
4. Repeat the above step for the other ball joint.

NOTE: GENERAL COMMAND STRUCTURE IN FEDEM
One normally starts by selecting the operation (tool button or an entry in a menu). An instruction text appears in the message board on top of the object browser. Objects are selected by clicking in the graphics window. Repetitive clicking will cycle through coincident objects. Confirm by clicking middle mouse button. Operations are cyclic and restart after completion. Cancel or exit operation modus by clicking the right mouse button.

NOTE: *Each joint in FEDEM has at least 2 triads. The free joint differs from other common joints in having (possible) separate positions for the slave and the master triad. For this reason the user needs to position master and slave triad separately.*

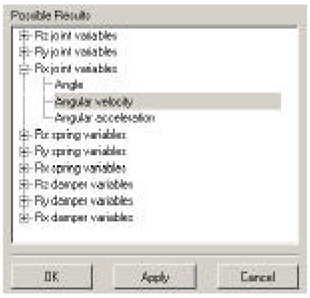
NOTE: *Only a master triad can be attached to ground.*
NOTE: *Triads must be attached to 6 DOF nodes.*

Test-solving the model

1. Reduce the link. Check the output list for successful reduction.
2. Solve the dynamics . Check the output list for successful solving.

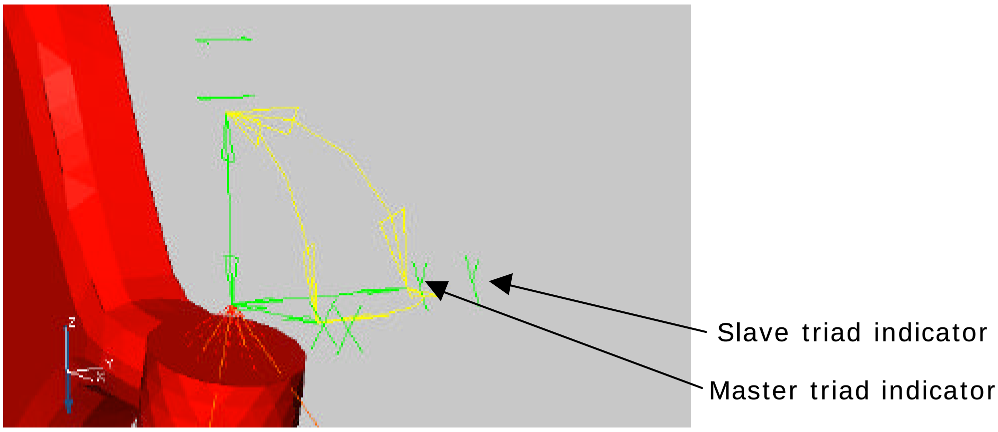
Creating an animation: Right-click in the **Result Objects browse**r and select **New Animation**. Name it 'Dynamics' in the description field. (Remember always to hit Enter after typing in to fields). Load the animation by clicking Load button in its property window. You should see the mechanism swing during action of gravity.

Creating a curve: In the **Result Objects browser**, right-click and select **New Curve**. A graph with a curve is created. In the curve's property box, in the **y-axis frame**, click **Edit**, and the **Result Selector** opens. Click one of the ball joints, and select from the Possible Results pane, Rx joint variables, Angular velocity. Right-click the curve in the Results Objects browser, and select **Show Graph**. The graph window will open in the workspace area.



Continued modelling

1. To continue, delete the results by clicking .
2. Load the knuckle.nas and uca.nas links, and set the link properties to preferred values.
3. Locate the connection point between the tie rod and knuckle (see figure below).
4. Create a free joint as follows: Click the **Free Joint** button or select **Free Joint** from the **Mechanism** menu. Next, you are prompted to give position for the master and slave triad successively. First click the knuckle/rod connection point and accept. Next click the same point for the slave, and accept. The free joint should now appear at the connection.
5. For the free joint, you will have to attach master and slave separately. Click **Attach** , and then click at the outer coordinate letter indicating slave triad (se figure below), and accept. Notice messages in message bar.
6. Next click the knuckle and accept.
7. You are now asked to attach a new object. Select the master triad and accept. Next click the reference plane and accept.
8. The free joint should now be fully coloured, as shown in the picture below, indicating that both the master and the slave triads are attached.
9. For simplicity, we will model the tie rod as a restraint on the knuckle: Select the free joint to display its properties. For the TX direction apply a stiffness of **1E8** N/m.

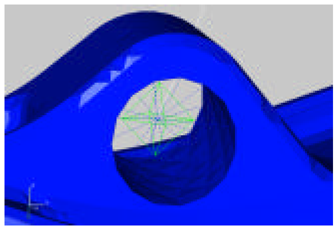


10. Connect the knuckle with the lower control arm by using a ball joint.
11. Connect the upper control arm and the knuckle with a ball joint.
12. Connect the upper control arm's body mounts to ground using ball joints.

Test-solving

This is a good time to stop and do a test solving. Click the Solve Dynamics button. FEDEM will automatically invoke the reducer and thereafter start the dynamics solver. Check the Output list for any messages. The mechanism, which is now fully defined, should move according to gravity.

Applying spring and damper

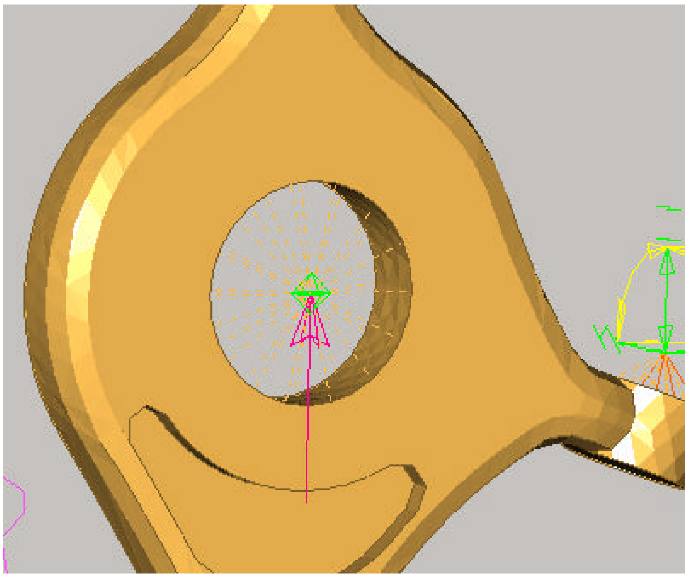


1. Click the **Axial Spring** button, or select **Axial Spring** from the **Mechanism** menu. Select the spring mount in the lower control arm for the first endpoint (see figure). Accept by mid-clicking, and for the next endpoint, type in the following values in the number pane x=0, y=0, z=0.3. Remember to hit Enter after typing the values. Next accept by clicking the middle mouse button, and the spring endpoint should appear between the body mounts of the uca.
2. Attach the triad at the upper spring endpoint to ground.
3. Next, click the **Axial Damper**, located on the spring button drop-down. Place the damper at same endpoints as the spring.
4. In the spring's property window, set the stiffness to **7.5E4.**
5. In the damper's property window, set the damping coefficient to **3E3**.

Creating a vertical force on the knuckle

To include the road characteristics in the model, we apply a force at the wheel centre.

1. Click on the **Force** button  on the Mechanism Creation toolbar.
2. Select the node at the centre of the wheel centre, and accept. Cancel when you are prompted to create another one.
3. Select the force by clicking on it in the 3D modelling view.
4. Edit direction by setting the “From” coordinate to (0, 0, 0), and the “To” coordinate to (0, 0, 1.0).



Drive function
Now we must create a function to use for input to the joint.

1. Right-click in the Objects browser and select **Create → Function**.
2. Change **Type** to **External Function**.
3. Click the **Browse** button, locate and select the FZ.asc file, and click **Open**.
4. Set description to “FZ”.
5. Click the **Create curve** button. A preview of the function now appears in the workspace area. Inspect it and verify that this is the input function you would like to use.
6. Close the curve window.
7. Select the force at the wheel centre by clicking on it in the 3D modelling window or by selecting it in the Objects browser. Use the engine you just created as **Magnitude**.

Solving dynamics

We are finished modelling, and are ready to do our first “real” dynamics solving.

1. Open the Dynamics Solver setup dialog by clicking the Dynamics Solver setup button . Set **Stop time** to 2.5 seconds.
2. Click **Ok**.
3. Run the Dynamics Solver. Look in the Output list for any error messages.
4. Load the animation you created earlier and press Play.

Recovering Stresses

In this section we will look at stresses in the Lower Control Arm. Results from Stress Recovery may take a lot of disc space, so it is beneficial to just look at part of the time history.

1. Open the Stress Recovery Setup dialog . Set values as shown in the picture to the right.
2. We only want to run stress on the lca link. In the Objects browser, right-click on the lca link, and select **Solve → Stresses**. This might take some time.
3. Create a new animation.
4. Toggle on “Load face fringes”, “Load line fringes”, and “Load deformations”.
5. Load the animation. Press Play in the panel that opens in the 3D window. If you toggle on “Show all frames”, you ensure that all frames will be displayed.