Workshop

Tennis Racket Simulation using Abaqus

Introduction

In this workshop you will become familiar with the process of creating a model interactively by using Abaqus/CAE. You will create the tennis racket model shown in Figure W–1.
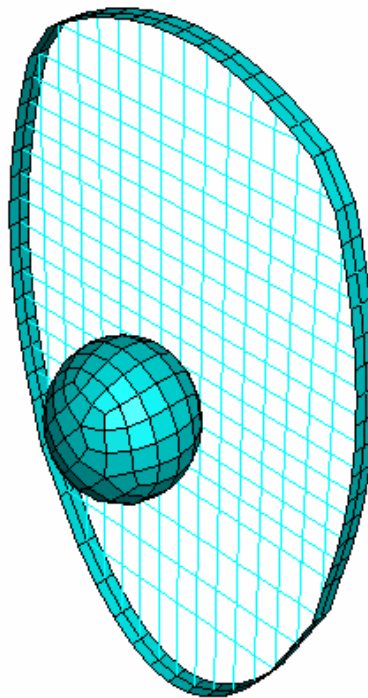

Figure W–1 Tennis racket and ball.

Preliminaries

1. Enter the working directory for this workshop:

   cd Workshops/tennis-demo

2. Start a new session of Abaqus/CAE using the following command:

   abq662se **cae**

   where *abq662se* is the command used to run Abaqus Student Edition.

3. Select **Create Model Database** from the **Start Session** dialog box.

The Model Tree is located to the left of the toolbox area. If the Model Tree is not visible, make sure there is a check mark next to **Show Model Tree** in the **View** menu. If the Model Tree is still not visible, drag the ✛ cursor from the left side of the Abaqus/CAE window to expand the Model Tree. The Model Tree provides a visual description of the hierarchy of items in the model database along with access to most of the model development functionality available in Abaqus/CAE. If you click mouse button 3 on an item in the tree, a menu appears listing commands associated with the item. For example, Figure W–2 shows the menu for the **Models** container. In the **Models** menu, the **Create** menu item appears in bold because it is the default action that will be performed when you double-click the **Models** container.
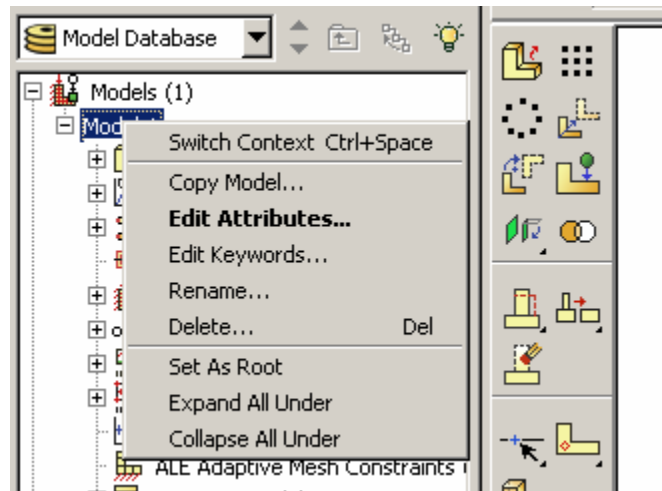
Figure W–2 **Models** menu.

4. In the Model Tree, double-click **Models** to create a new model in the model database.

   The model attributes editor appears.

5. In the **Edit Model Attributes** dialog box that appears, enter the name **TENNIS-DEMO** and click **OK**.

   The model **TENNIS-DEMO** appears in the Model Tree, as shown in Figure W–3. The new model is underlined to indicate that it is a current object (i.e., **TENNIS-DEMO** is the model you are currently working on).

   Tip: To reduce clutter in the Model Tree, collapse the container for **Model-1** by clicking the "−" sign next to the model name or set **TENNIS-DEMO** as the root.
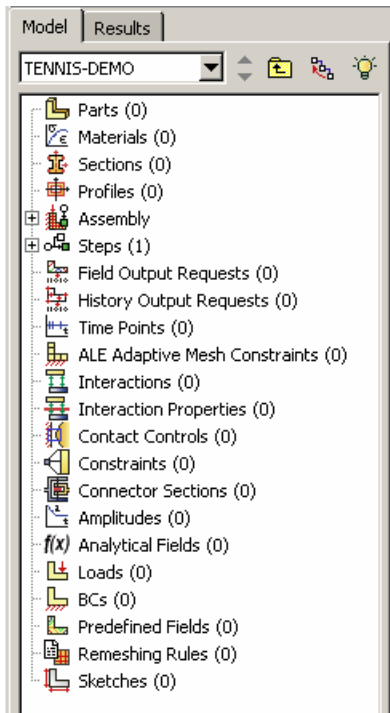
Figure W–3 Model Tree.

6. To save the model database, select **File→Save As** from the main menu bar and type the file name `tennis-demo` in the **Save Model Database As** dialog box. Click **OK**.

    The `.cae` extension is added to the file name automatically.

Creating a part

In this section you will create two three-dimensional parts for frame and ball by first sketching the two-dimensional profile and then extruding/ revolving it.

1. For creating frame we will import already existing sketch. Use **File→ Import→ Sketch** to import `frame-sketch.sat` and `strings-sketch.sat` files.
2. In the Model Tree, double-click **Parts** to create a new part in the model **TENNIS-DEMO**.

    The **Create Part** dialog box appears.

3. In the **Create Part** dialog box, name the part `Frame`, and specify an approximate size of `200`. Accept the default settings of a three-dimensional, deformable body. Change the base feature to shell, extruded base feature (Figure w-4). Click **Continue**.
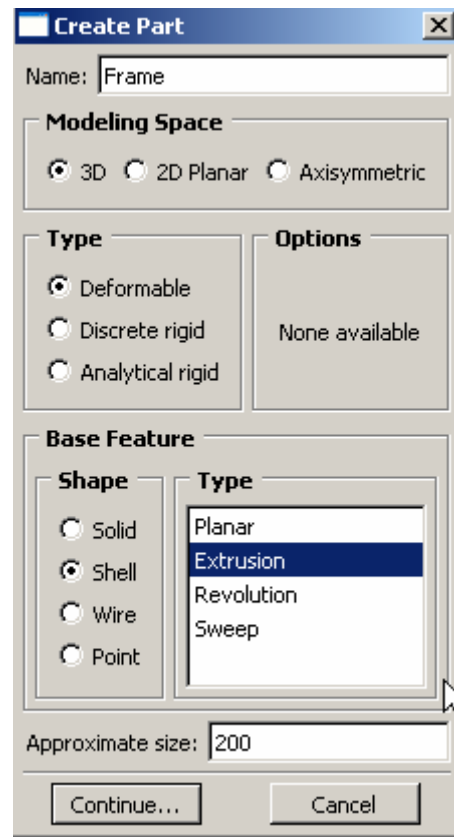
Figure W–4 Create Part area

Abaqus/CAE displays text in the prompt area near the bottom of the window to guide you through the procedure, as shown in Figure W–5. Click the cancel button to cancel the current task; click the backup button to cancel the current step in the task and return to the previous step.
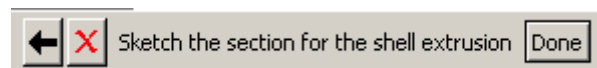

Figure W–5 Prompt area.

The Sketcher toolbox appears in the left side of the main window, and the Sketcher grid appears in the viewport.

4. In this case instead of sketching the profile we will import it from already existing sketch. To import an already existing sketch choose **Add** → **Sketch** and select **frame-sketch** from the **Select Sketch** window (Figure W-6). The frame sketch will appear in the current Viewport.
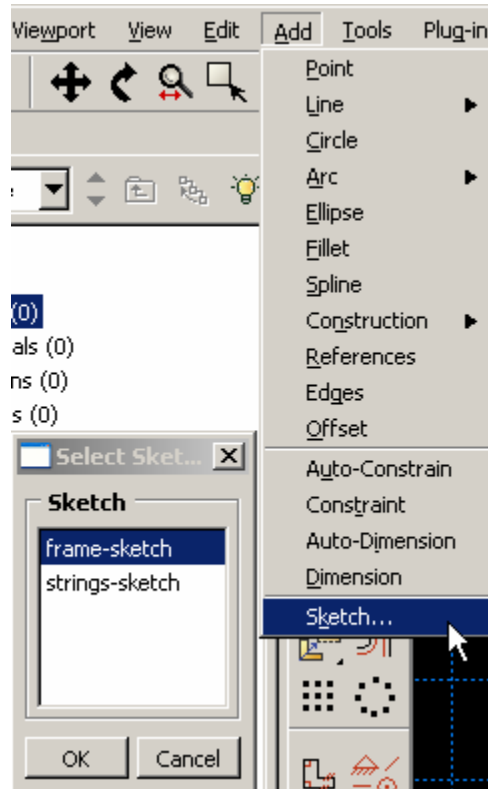
Figure W–6 Add Sketch tool.

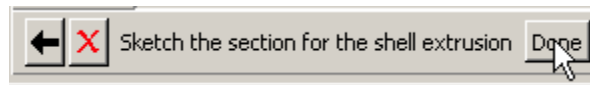5. Click **Done** in the prompt area to exit the sketcher (Figure W-7).

Figure W-7 Prompt area

6. Enter an extrusion depth of `0.5` in the **Edit Base Extrusion** dialog box, and click **OK**.

    Abaqus/CAE displays an isometric view of the new part, as shown in Figure W-8. The **Parts** icon in the Model Tree changes to indicate that the model contains one part, .
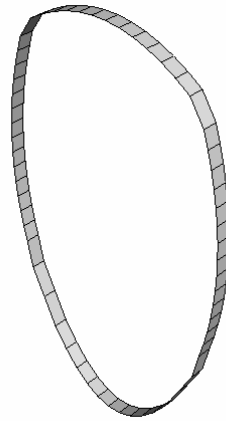
Figure W–8 Extruded part.

7. To create strings we will first create partition at the middle of the frame. We will use a datum plane for partition. So to create a datum plane first choose **Tools→ Datum**. In the **Create Datum** window choose **Plane** with **3 point Method** (Figure W-9). Click **Ok** and Select 3 points at the middle of the frame (Figure W-10). This will create a datum plane at the middle of the frame.
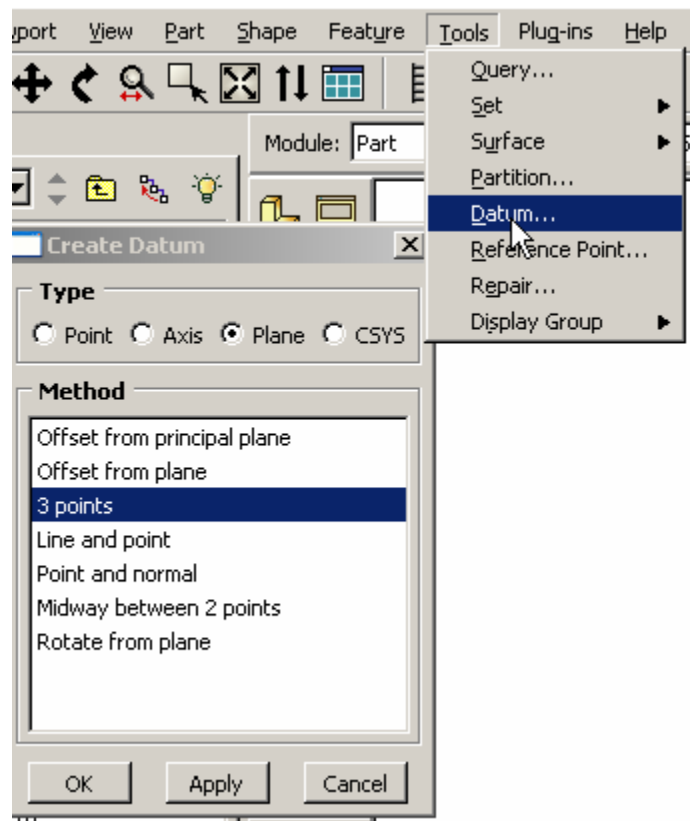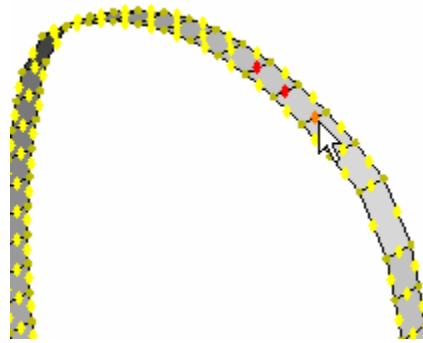

Figure W-9

Figure W-10

8. Choose partitioning tool as shown in Figure W-11 and press **Ok**. Select the datum plane from Viewport to create partition.
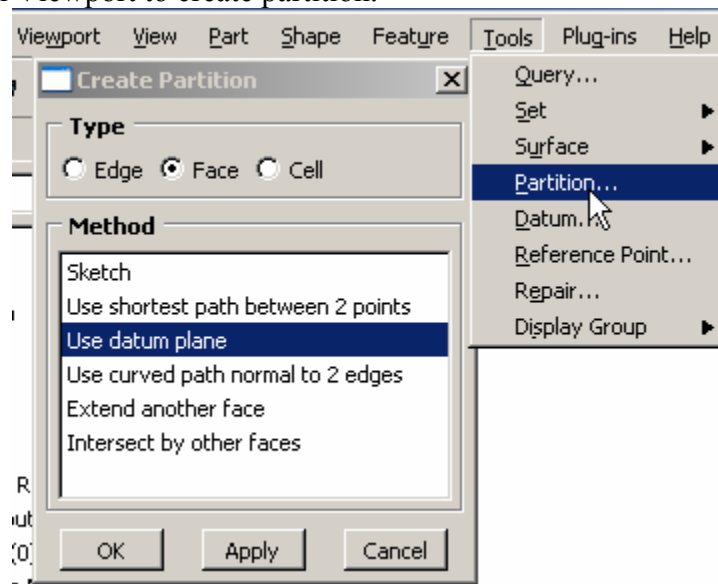

Figure W-11

9. Prompt area will ask to select the faces to partition. To select all the faces of frame create a drag box and press **Done** from the prompt area. Then the prompt area will ask to select the datum plane. Click on the datum plane created in step 7 and select **Create Partition** from the prompt area.
10. Create a set using **Tools→ Set→ Create**. Name the set as `frame` and then select the whole frame by dragging a box.
11. Create a datum axis along Y-Axis by selecting **Tools → Datum → Axis → Principal axis** as shown in figure W-12, click **Ok** and then select **Y-Axis** from the prompt area.

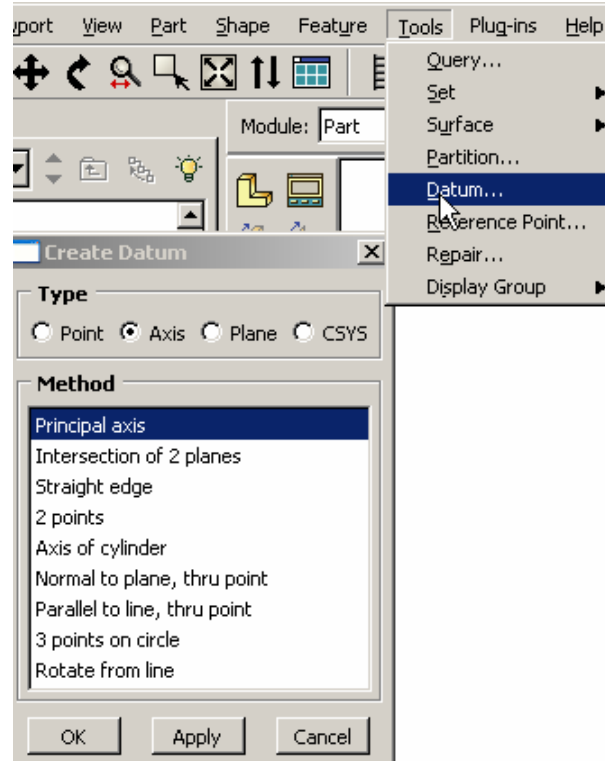Figure W-12

12. Choose Shape→ Wire→ Sketch to create strings. Choose the datum plane and datum axis respectively for the 'plane of wire' and 'vertical axis on right'.
13. In the sketcher select **Add**→ **Sketch** and select `strings-sketch`. Press **Done** to continue. This will result in wires as shown in Figure W-13. Press **Done** to exit out of sketcher.
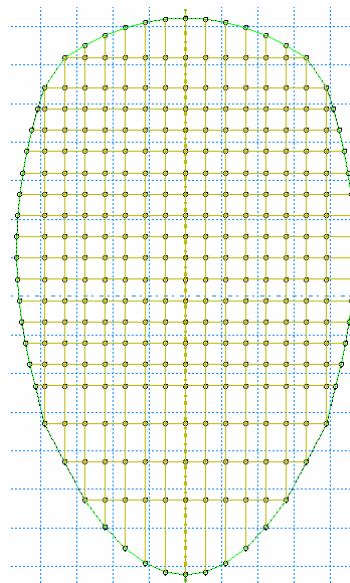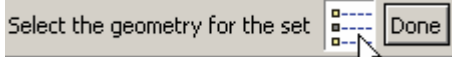

Figure W-13

Introduction to Abaqus

14. Choose [icon] and remove all the faces (i.e. frame) from the Viewport by dragging a box around the whole model and pressing **Done** from the prompt area. Create a set using **Tools→ Set→ Create**. Name the set as `string-elem` and then select the displayed strings by dragging a box.

15. Create another set using **Tools→ Set→ Create**. Name the set as `strings`. Click Show/Hide selection option button [Select the geometry for the set ... Done]. In the Options window change **Select from:** to **Vertices** and change drag-shape to polygon as shown in Figure W-14.
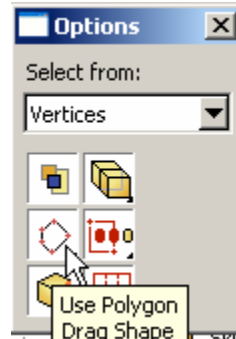

Figure W-14

16. Then draw a polygon shape in the Viewport to include only the internal nodes as shown in Figure W-15. Right click in the Viewport and press **Done** to create a node set which contains only the internal nodes of strings.
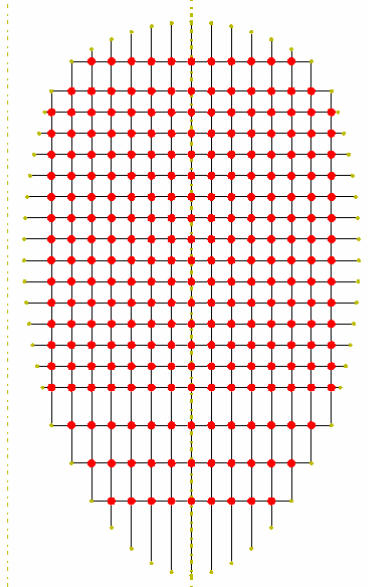

Figure W-15

17. then select the displayed strings by dragging a box

18. Press Replace All button [Initial ... Replace All] to view the complete model.

19. Create a part name **Ball** by double-clicking the Parts container in the Model tree. Choose **3D deformable shell** with type as **revolution**. Specify approximate size as 20. Press continue to sketch a ball with radius **1.5** using create circle tool.
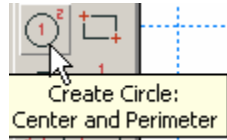
Enter center as **0, 0** and perimeter as **1.5, 0** in the prompt area to complete the definition of this circle.

20. Add a vertical construction line passing through origin using **Add→ Construction → Vertical** and entering **0, 0** as position in the prompt area.

21. Select Edit→ Auto-Trim and select the left half of circle for trimming. Press Done to exit out or sketcher. At this point you will be prompted to select the 'Construction line to act as centerline'. So select the construction line which was created in step 17.

22. In the **Edit Revolution** window enter angle as **360** degrees and press **Ok** to obtain part shown in Figure W-16.
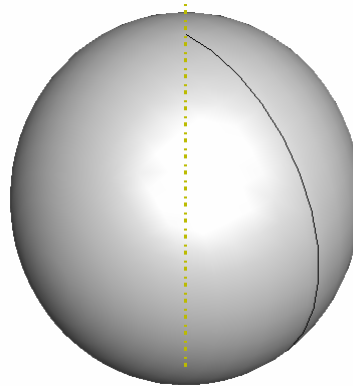
Figure W-16

23. Create a datum plane passing through the edge on Ball part using **Tools→ Datum → Plane → 3 Points**. Partition the ball using this datum plane (**Tools→ Partition→ Face→ Use Datum Plane**).

24. Create **ball** set by selecting complete ball geometry (**Tools→ Set→ Create**).

25. Create **ball-outer** surface using **Tools→ Surface-Create.** Select the whole ball geometry and then choose outer surface (Brown) from the prompt area. Create **ball-inner** surface using **Tools→ Surface-Create.** Select the whole ball geometry and then choose inner surface (Purple) from the prompt area.

Creating a material definition

You will now create three materials, one each for frame, strings and ball.

To define a material:

1. In the Model Tree, double-click **Materials** to create a new material in the model **TENNIS-DEMO**.

    Abaqus/CAE switches to the Property module, and the material editor appears.

2. In the **Edit Material** dialog box, name the material **Frame**. Notice the various options available in this dialog box.
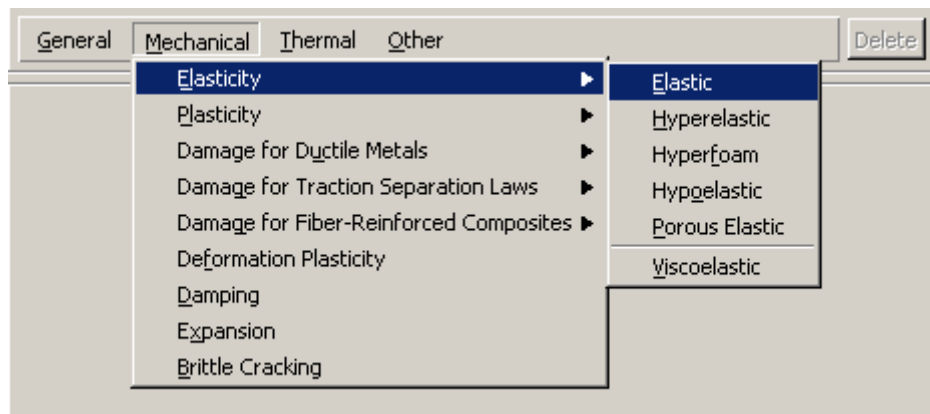
Figure W–17 Pull–down menu.

3. From the material editor's menu bar, select **Mechanical→Elasticity→Elastic,** as shown in Figure W–17.

    Abaqus/CAE displays the **Elastic** data form.

4. Enter a value of **1.0E10** for Young's modulus and a value of **0.0** for Poisson's ratio in the respective fields, as shown in Figure W–18. Use **[Tab]** to move between cells, or use the mouse to select a cell for data entry.

Figure W–18 Material editor.

5. From the material editor's menu bar, select **General→Density** and enter **1.0E-3** as density value.
6. Press **Ok** to complete **Frame** material definition.
7. Similarly create **String** material definition with density= **0.000107**, Young's **Modulus**=**1.0E6** and **Poisson's Ratio**=0.0.

8. Double click on Materials container in the model tree to create a material named **Rubber**. Select **Mechanical→Elasticity→Hyperelastic** to enter hyperelastic material properties as shown in Figure W-19. Also add density of **0.0001** using **General→Density**. Press **Ok** to complete the material definition
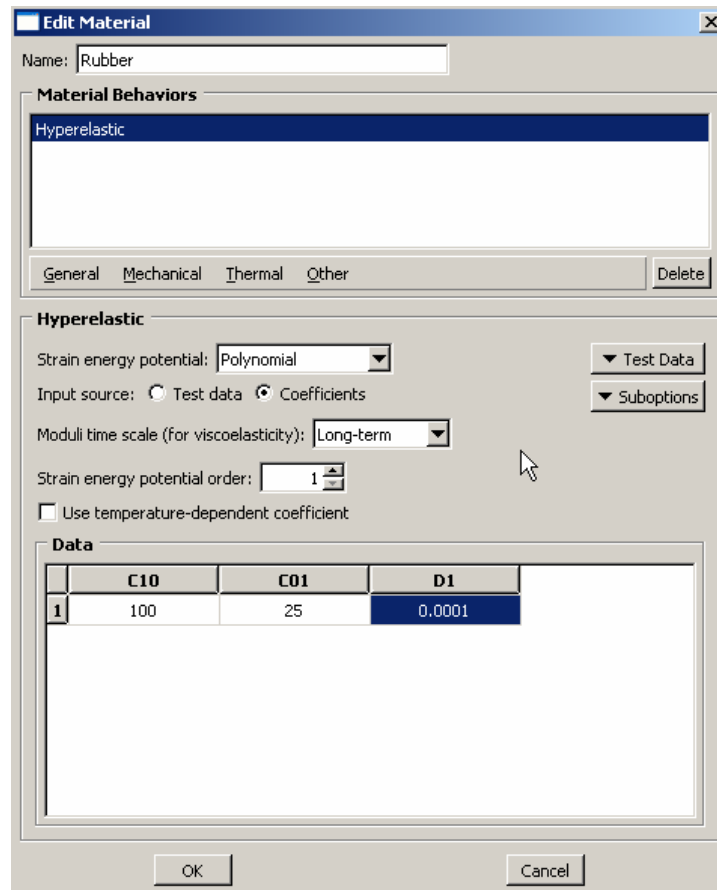
Figure W-19

Defining and assigning section properties

Next, you will create three sections. These sections will refer to the materials **Frame, String,** and **Rubber** that you just created.

To define shell and truss sections:

1. In the Model Tree, double-click **Sections** to create a new section in the model **TENNIS-DEMO**.

   The **Create Section** dialog box appears.

2. In the **Create Section** dialog box:

   a. Name the section **FrameSection**.

   b. Change the category to **Shell** and the default type **Homogeneous**.

   c. Click **Continue**.

The solid section editor appears.

3. In the **Edit Section** dialog box:

    a. Choose Section Integration Before Analysis.

    b. Change the thickness to **0.1**.

    c. Select **Frame** material from the material list.

    d. Click **OK**.

4. Create another section with name **RubberSection**. In the **Create Section** dialog box, change the category to **Shell** and the default type **Homogeneous**. Click **Continue**.

5. In the **Edit Section** dialog box: Choose Section Integration during Analysis. Change the thickness to **0.2.** Select **Rubber** material from the material list. Change the Thickness Integration Points to **3** and click **OK**.

6. Create a third section with name **StringSection**. In the **Create Section** dialog box, change the category to **Beam** and the type to **Truss**. Click **Continue**.

7. In the **Edit Section** dialog box: Select **string** material from the material list. Specify cross-section area of **0.007854** corresponding to 0.1 in diameter and click **OK**.

To assign the section definitions:

1. In the Model Tree, expand the branch for the part **Frame** (click the "+" symbol to expand the **Parts** container and then click the "+" symbol next to the part **Frame**).

2. Double-click **Section Assignments** to assign a section to the set **frame** and **strings**.

    Abaqus/CAE displays prompts in the prompt area to guide you through the procedure.

3. From the prompt area select **Sets**, and select the **frame** set and press **Continue**. From the Section Assignment Editor use toe drop down list to select **FrameSection.** Press **Ok** to complete the definition.

4. Then from the Eligible Sets choose **string-elem** press **Continue**. Select **StringSection** from the Edit Section Assignment window and press **Ok**.

5. Expand the branch for part **Ball.** Double-click **Section Assignments** to assign a section to the set **ball.** Select **ball** from the eligible sets and assign section RubberSection to it.

6. Click **Cancel** to close Eligible Sets window if it is open.

7. Abaqus/CAE colors the ball green to indicate that the section has been assigned.

Assembling the model

The assembly for this analysis consists of a two instances one each of **Frame** and **Ball** parts.

To assemble the model:

1. In the Model Tree, expand the branch for the **Assembly** and double-click **Instances** to create a new part instance.

   Abaqus/CAE switches to the Assembly module, and the **Create Instance** dialog box appears.

2. In the **Create Instance** dialog box, select **Frame** and **Ball** and click **OK**.

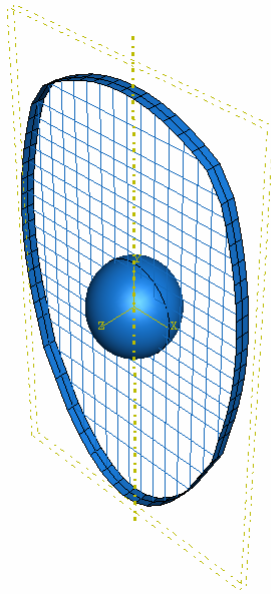   Abaqus/CAE displays the new part instances in the viewport as shown in Figure W-20.

Figure W-20

3. Translate the **Ball** instance using **Instance→Translate**. Select the **ball** instance when prompted and enter the coordinates **0, 0, 0** and **0, 0, 1.8** as start and end point of translation vector respectively.

4. Create a reference point using **Tools→Reference Point** and enter the coordinates as **0.0, 0.0, 0.25**. Create another reference point with the following coordinates: **0.0, 0.0, 1.8**.

5. Create a set named **refnode** for RP-1 and **refnode-cav** for RP-2.

Configuring the analysis

In this simulation we are interested in the dynamic response of the ball and racket when ball hits the racket with a velocity. The frame will be modeled as rigid to reduce the analysis cost for this simulation. This is a single event, so only a single analysis step is needed for the simulation. Consequently, this model will consist of two steps:

- An initial step, in which you will apply a boundary condition that constrains all degrees of freedom of rigid racket frame and initial velocity to the ball.
- A Dynamic, Explicit analysis step, in which a dynamic response will be calculated when ball hits the racket.

Abaqus/CAE generates the initial step automatically, but you must create the analysis step yourself.

To create a dynamic, explicit analysis step:

1. In the Model Tree, double-click **Steps** to create a new step in the model **TENNIS-DEMO**.

   Abaqus/CAE switches to the Step module, and the **Create Step** dialog box appears.

2. In the **Create Step** dialog box:

   a. Name the step **Dynamic**.

   b. From the list of available general procedures in the **Create Step** dialog box, select **Dynamic, Explicit**.

   c. Click **Continue**.

   The step editor appears.

3. In the **Edit Step** dialog box:

   a. In the **Description** field of the **Basic** tabbed page, enter **Dynamic analysis of tennis racket**

   b. Change the step time to **0.015** and keep **NLGEOM** on to include nonlinear geometry effects.

   c. Click the **Incrementation** and **Other** tab to see its contents; you can accept the default values provided for the step.

   d. Click **OK** to create the step and to exit the step editor.

4. Create a history output request for CVOL and PCAV:

   a. In the model tree double-click the **History Output Requests** container. Clock Continue in the Create History dialogue box to create a history output with name **H-Output-2** in step **Dynamic**.

   b. Enter the values shown in Figure W-21 and click **Ok** to complete the definition.
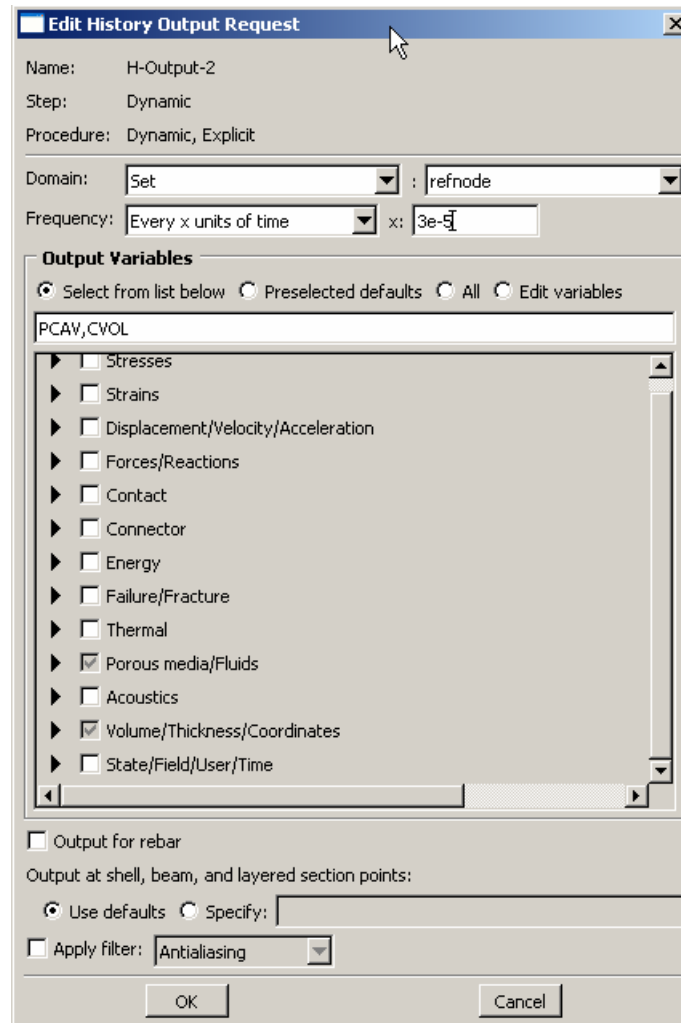
Figure W-21

5. Modify history output request **H-Output-1** by double-clicking it in the model tree and changing frequency of output to every **3e-5** units of time (same as H-Output-2).

Applying contact and rigid body constraints in the model

Next, you will define **frame** as rigid body and constraint it with **refnode**. You will also define contact between **ball** and **strings**.

To create rigid body constraint:

1. In the Model Tree, double-click **Constraints**, select **Rigid Body** and press **Continue**.

   Abaqus/CAE switches to the Interaction module and prompts the **Edit Constraint** dialog box appears.

2. In the **Edit Constraint** dialog box:

   a. Select **Region type** as Body and press **Edit** (Figure W-22).

   b. From the prompt area select **Sets** and select **Frame-1.frame** from the **Eligible Sets** window**,** then press **Continue.**

   c. Press **Edit** to choose the **Reference Point**. From the prompt area again select **Sets**, choose **refnode** and press continue.
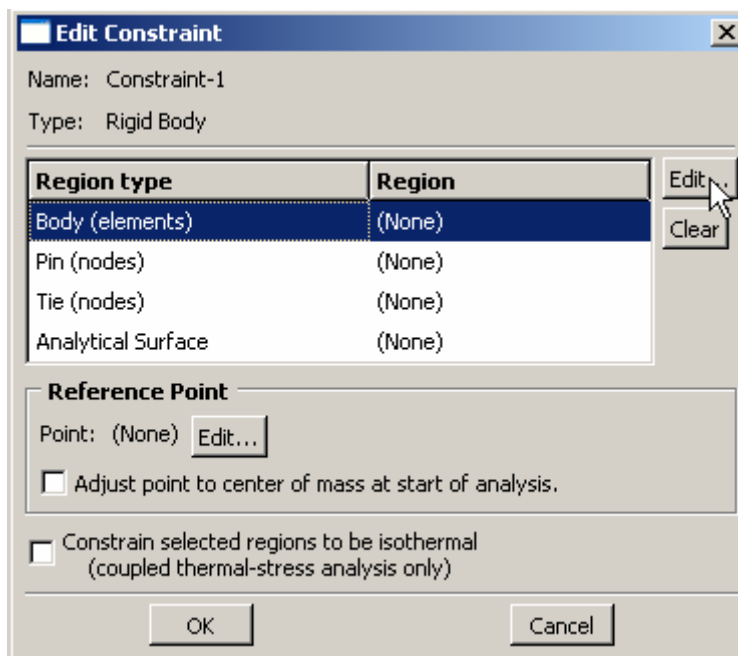
   d. Click **Ok** to complete the rigid body definition.

Figure W-22

To create contact between ball and strings:

1. In the Model Tree, double-click **Interactions**, choose **Surface-to-surface contact (Explicit)** from the Create Interaction window and press **Continue**.

2. CAE will prompt to select the master surface. Select `Ball-1.ball-outer` as master surface from the eligible surface list and press **Continue.**

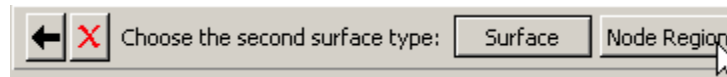3. From the prompt area select **Node Region** (W-23).

Figure W-23

4. Select `Frame-1.strings` and press **Continue** from the **Eligible sets** dialogue box.

5. In the **Edit Interaction** dialog box:

    a. Press **Create** next to **Contact interaction property** to create friction interface (Figure W-24).

    b. Press **Continue** in the **Create Interaction Property** window.

    c. Select **Mechanical→Tengential Behavior** and define friction properties as shown in Figure W-25. Then press **Ok.**

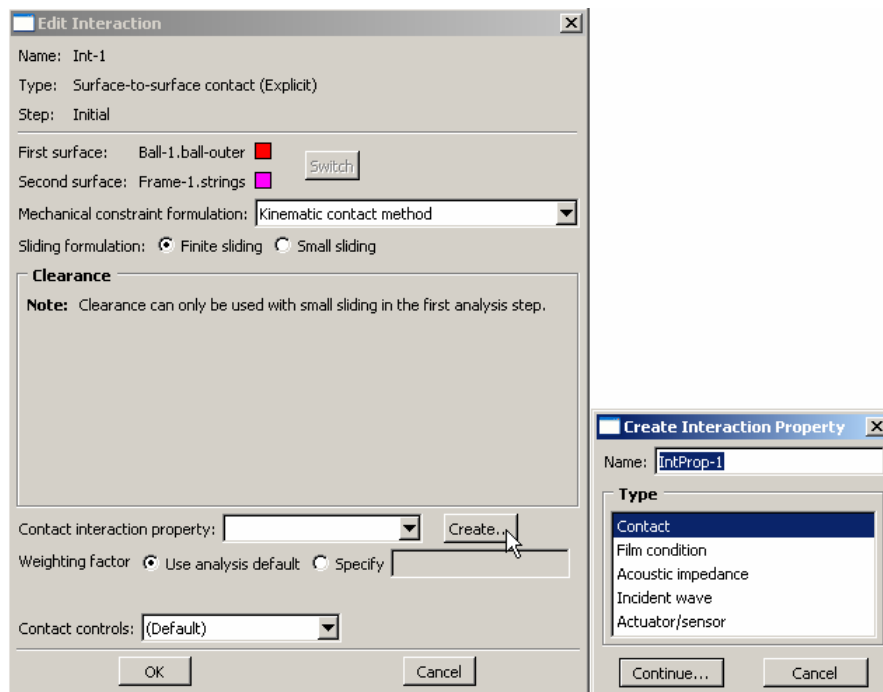   d. Click **Ok** to complete the interaction definition.
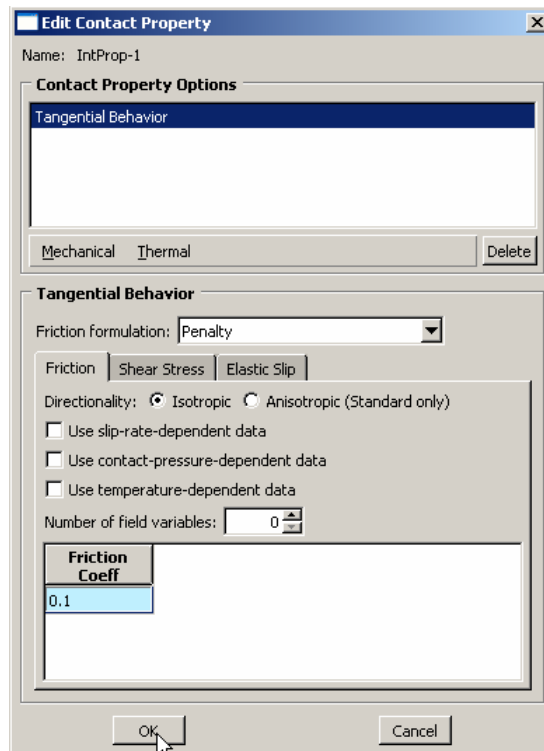

Figure W-24

Figure W-25

Applying a boundary condition and a load to the model

Next, you will define the fixed boundary condition at **refnode** and initial velocity on **ball.**

To apply boundary conditions to refnode:

1. In the Model Tree, double-click **BCs** to create a new boundary condition in the model **TENNIS-DEMO**.

   Abaqus/CAE switches to the Load module, and the **Create Boundary Condition** dialog box appears.

2. In the **Create Boundary Condition** dialog box:

   a. Name the boundary condition **Fixed**.

   b. Select **Initial** as the step in which the boundary condition will be activated.

   c. In the **Category** list, accept the default category selection **Mechanical**.

   d. In the **Types for Selected Step** list, select **Symmetry/Antisymmetry/Encastre** as the type.

   e. Click **Continue**. Abaqus/CAE displays prompts in the prompt area to guide you through the procedure. Select **Sets** from prompt area to choose **refnode** set from the Eligible sets.

    f.   From the **Edit Boundary Condition** dialogue box choose `Encastre` to fix all degrees of freedom of `refnode` and press **Ok.**

To apply a velocity on the ball:

1. In the Model Tree, double-click **Predefined Fields** to create a new load in the model **TENNIS-DEMO**.

   The **Create Predefined Field** dialog box appears.

2. In the **Create Predefined Field** dialog box:

    a.   Name the load `Velocity`.

    b.   Select **Initial** as the step.

    c.   In the **Category** list, accept the default category selection **Mechanical**.

    d.   In the **Types for Selected Step** list, select **Velocity**.

    e.   Click **Continue**.

    f.   Select **Sets** from prompt area to choose `ball` set from the Eligible sets.

    g.   Specify V1=`68.3` and V3=`-255` in the **Edit Predefined Field** dialogue box and press **Ok**

Meshing the model

You use the Mesh module to generate the finite element mesh. You can choose the meshing technique that Abaqus/CAE will use to create the mesh, the element shape, and the element type. Abaqus/CAE uses a number of different meshing techniques. The default meshing technique assigned to the model is indicated by the color of the model when you enter the Mesh module; if Abaqus/CAE displays the model in orange, it cannot be meshed without assistance from the user.

To assign the mesh controls:

1. In the Model Tree, double-click **Mesh** in the branch for the part **Frame**.

   Abaqus/CAE switches to the Mesh module and displays the part **Frame**.

To assign an Abaqus element type:

1. From the main menu bar, select **Mesh→Element Type**.

2. Select `string-elem` from the **Eligible Sets.**

3. In the **Element Type** dialog box, make sure that the following is selected:

    • **Standard** is the default **Element Library** selection.

    • **Linear** is the default **Geometric Order**.

    • Change **Family** to **Truss**.

4. In the lower portion of the dialog box, examine the element shape options. A brief description of the default element selection is available at the bottom of each tabbed page.

A description of the element type T3D2 appears at the bottom of the dialog box. Abaqus/CAE will now mesh strings with T3D2 elements.

5. Click **OK** to assign the element type and to close the dialog box.

To mesh the model:

1. From the main menu bar, select **Seed→Part** to seed the part.
   The **Global Seeds** dialog box appears. The default global element size is based on the size of the part.

2. In the **Global Seeds** dialog box, enter an approximate global size of **1** and click **OK**.

3. From the main menu bar, select **Mesh→Part** to mesh the part.

4. Click **Yes** in the prompt area or click mouse button 2 in the viewport to confirm that you want to mesh the part instance.

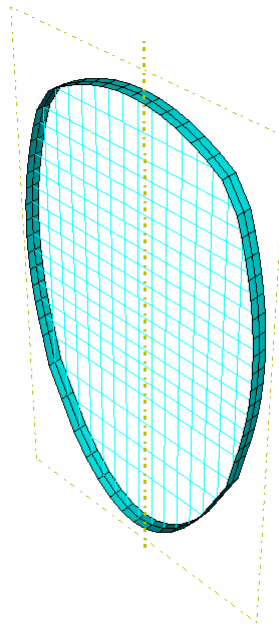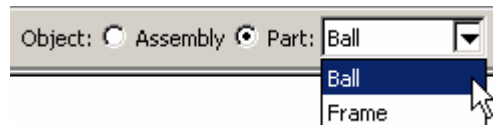   Abaqus/CAE meshes the part instance and displays the resulting mesh, as shown in Figure W–26.

Figure W–26 Part instance showing the resulting mesh

5. To mesh **Ball** part, switch part to **Ball** (Figure W-27). Select **Mesh→Controls.** Select the whole ball region. From the **Mesh Controls** window change the **Algorithm** to **Advancing front** and press **Ok.**

W-27

6. Assign a global seed of **0.5** using **Seed→Part**.

7. From the main menu bar, select **Mesh→Part** to mesh the part. Ball will be meshed with S4R elements by default.

8. Click **Yes** in the prompt area or click mouse button 2 in the viewport to confirm that you want to mesh the part instance.

Editing keywords to define fluid cavity and initial conditions

The definition of the model **TENNIS-DEMO** also contains gas cavity definition for ball and initial stress and pressure condition on strings and ball. These definitions are not available through GUI interface of CAE so you will add then through keyword editor.

To edit keywords:

1. From the main menu bar, select **Model→Edit Keywords** and select **TENNIS-DEMO** model.

   Abaqus/CAE will launch **Edit Keywords** window.

2. In the **Edit Keywords** window, enter the following keywords as shown in Figures W-28, W-29 and W-30. After entering all the keywords click **Ok**.

   Note: Keywords are case-insensitive

*FLUID BEHAVIOR, NAME=GAS

*MOLECULAR WEIGHT

0.1367,

*CAPACITY, TYPE=TABULAR

112.847,

*FLUID CAVITY, NAME=BALL, REF NODE=REFNODE-CAV, SURFACE=BALL-1.BALL-INNER, BEHAVIOR=BALL-1.GAS, AMBIENT PRESSURE=14.7

1.0

*INITIAL CONDITIONS, TYPE=STRESS

FRAME-1.STRING-ELEM, 1273.

BALL-1.BALL, 22.5, 22.5

*INITIAL CONDITIONS, TYPE=FLUID PRESSURE

REFNODE-CAV, 6.0

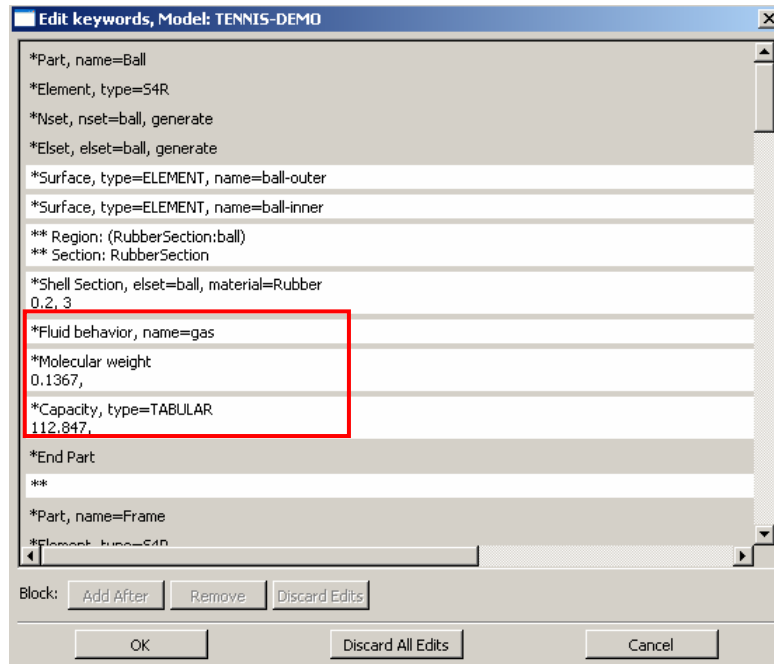*PHYSICAL CONSTANTS, ABSOLUTE ZERO=-273.16, UNIVERSAL GAS CONSTANT=73.562
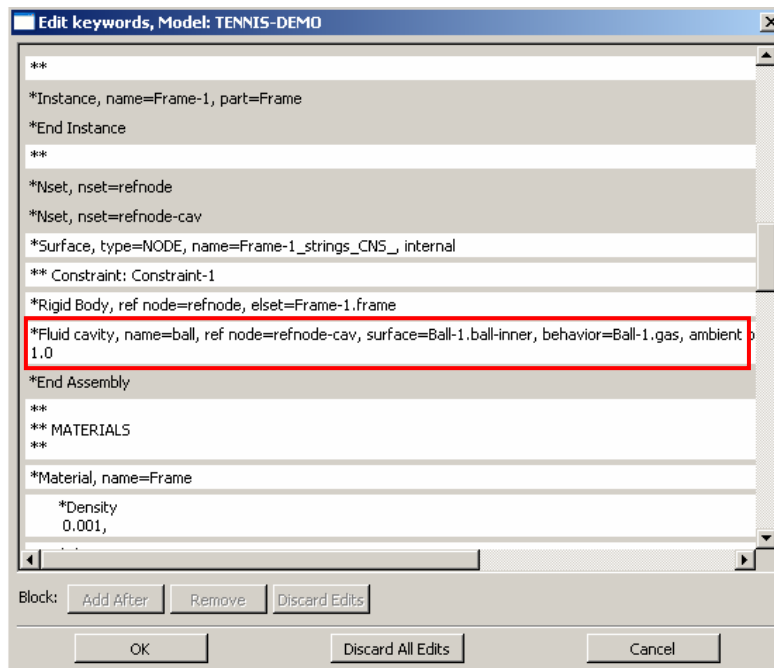
Figure W-28 Keyword Editor window
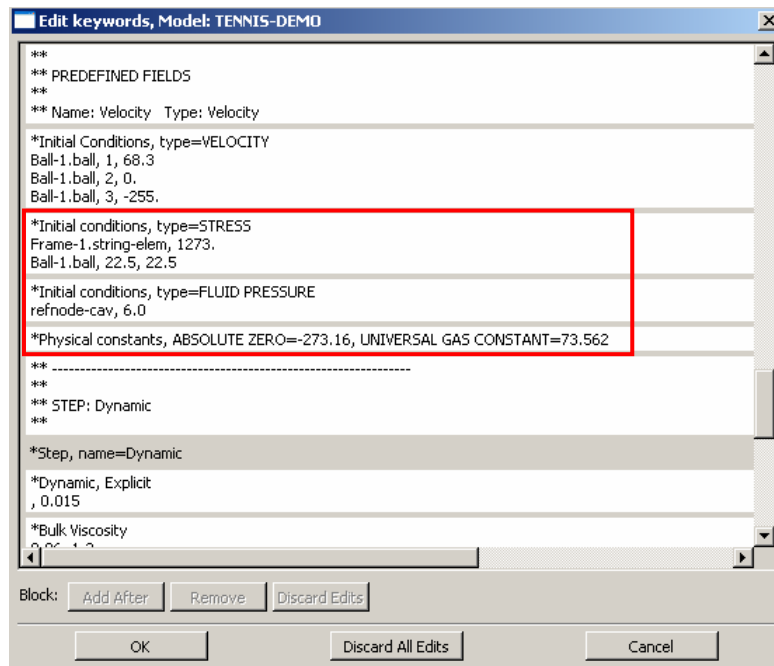

Figure W-29 Keyword Editor window

Figure W-30 Keyword Editor window

Creating and submitting an analysis job

The definition of the model **TENNIS-DEMO** is now complete. Next, you will create and submit an analysis job to analyze the model.

To create and submit an analysis job:

3. In the Model Tree, double-click **Jobs** to create a new analysis job.

    Abaqus/CAE switches to the Job module, and the **Create Job** dialog box appears.

4. In the **Create Job** dialog box, name the job `Tennis-Analysis` and select the model **TENNIS-DEMO**. Click **Continue**.

    The job editor appears.

5. In the **Description** field of the **Edit Job** dialog box, enter `Workshop`.

6. Click the tabs to see the contents of each folder of the job editor and to review the default settings. Click **OK** to accept the default job settings.

7. In the Model Tree, expand the **Jobs** container and click mouse button 3 on the job **Tennis-Analysis**.

    The menu for the job **Tennis-Analysis** appears.

8. From the job menu, select **Submit**.

    The icon for the job will change to indicate the status of the job in parenthesis after the job name. As the job runs the status **Running** will be shown in the Model Tree. When the job completes successfully, the status will change to **Completed.**

Viewing the analysis results

You are now ready to view the results of the analysis in the Visualization module.

1. In the Model Tree, click mouse button 3 on the job **Tennis-Analysis** and select **Results** from the menu that appears.

   Abaqus/CAE switches to the Visualization module, opens the output database created by the job (**Tennis-Analysis.odb**), and displays the undeformed shape of the model, as shown in Figure W–31.
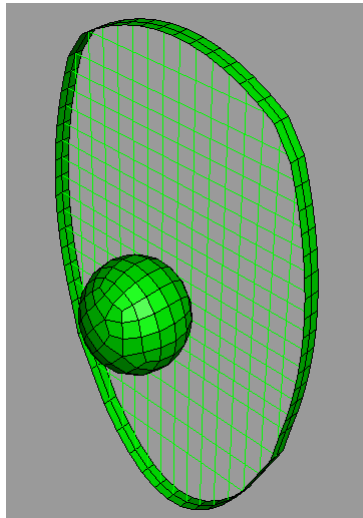

Figure W–31 Undeformed model shape.

2. In the toolbox, click 🏗 (or select **Plot**→**Deformed Shape** from the main menu bar) to view a deformed model shape plot, as shown in Figure W–32.
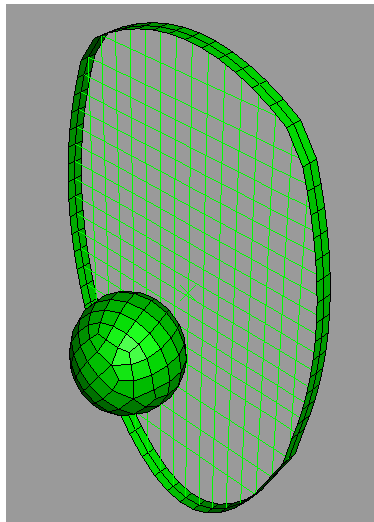

Figure W–32 Deformed model shape.

You may need to use the **Auto-Fit View** tool ⛶ to rescale the figure in the viewport.

3. In the toolbox, click 📐 (or select **Plot→Contours→on Deformed Shape** from the main menu bar) to view a contour plot of the von Mises stress, as shown in Figure W–33. In this figure, the font size for the legend text has been increased to 10 (**Viewport→ Viewport Annotation Options**).
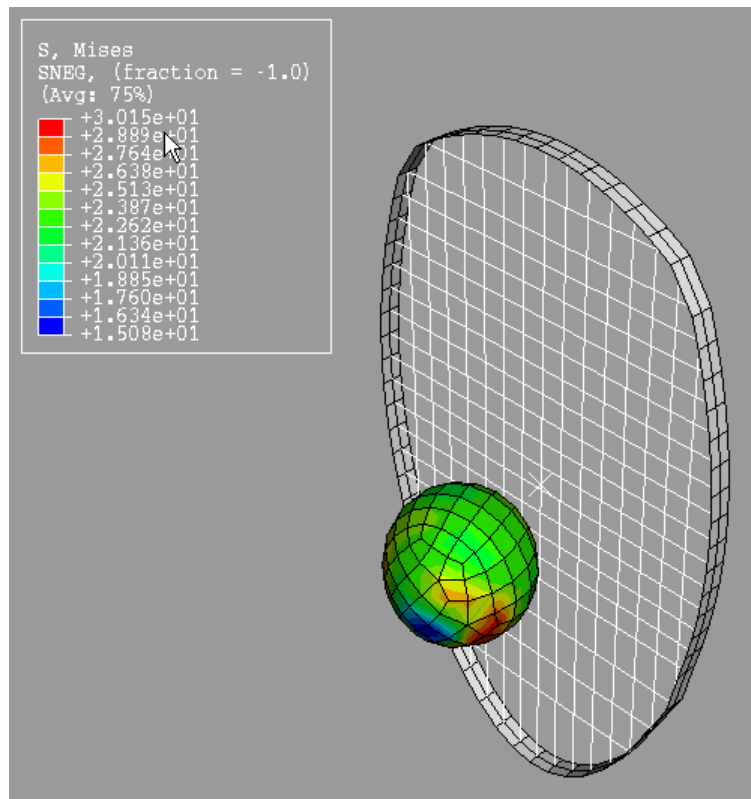

Figure W–33 Mises stress contour plot.

4. In the toolbar, click 💾 (or select **File→Save** from the main menu bar) to save your model in a model database file. Close the Abaqus/CAE session by selecting **File→Exit** from the main menu bar.