ME 24-688 – Week 10

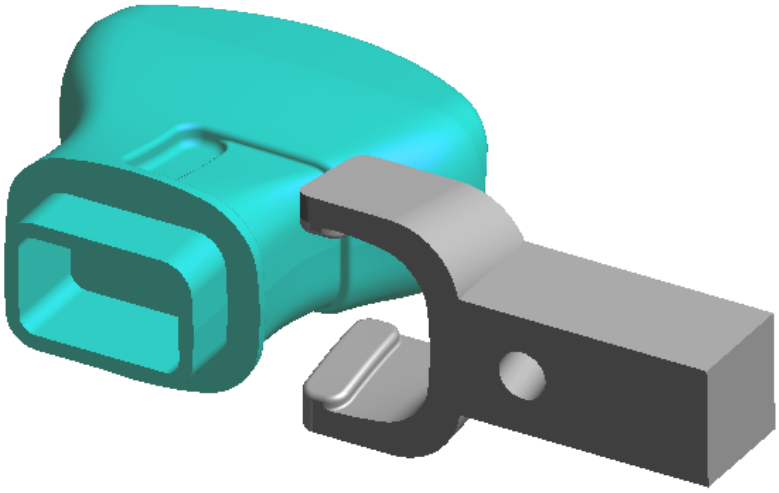
Nonlinear Static Stress

Nonlinear Static Stress

Establishing the correct real world elements and relationships is critical to all simulations. This includes contact relationships and accurate material definition to ensure the proper material behavior is provided. This project will continue from the past projects and expand your knowledge of the Autodesk Simulation product around nonlinear analysis.

1.1 Project 3 – Medical Device Snap Fit Analysis

During this project you will need to simplify the model geometry to improve analysis performance. Then setup the MES with nonlinear materials analysis to simulation the snap click part snapping onto the nose piece part to determine if the part will fail due to stress during deformation. Several contact relationships will be assigned and also mesh refinement will be used to ensure proper results.

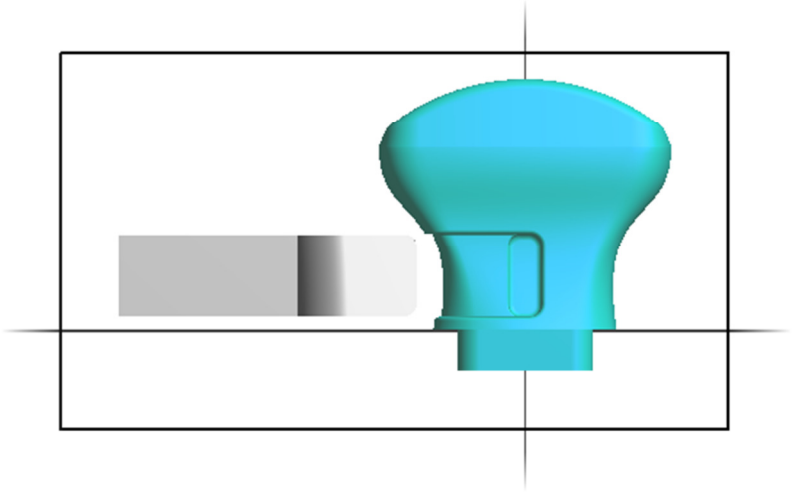


1. Start **Autodesk Inventor 2012** and **Open** the *Medical Device Clip.iam* assembly file.

2. **Orbit** the model around to better understand the shape. The *Snap Clip* part will be pressed over the *Nose Piece* part to produce a snap fit. Each part is made out of plastic and the design is symmetrical about the **XZ** plane.

3. To simplify the simulation and save time since the model is symmetrical we will remove half of the model before sending the parts to Autodesk Simulation for analysis. To begin click **Model tab** | **Sketch panel** | **Create 2D Sketch** to begin a new sketch. Then expand the origin folder in the browser for the assembly and select the **XZ** origin plane.

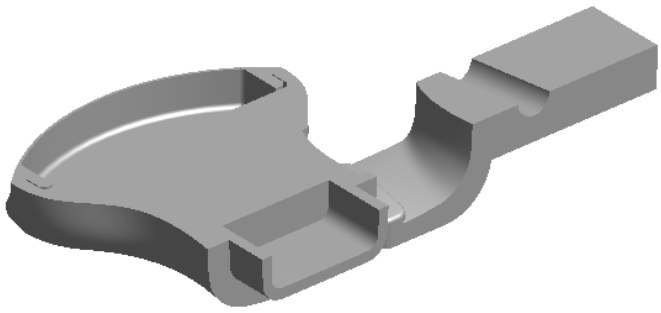
4. Sketch a rectangle by clicking **Sketch tab** | **Draw panel** | **Rectangle** and select two points in the graphics screen to away from the model as shown below.



5. Sketch **Finish Sketch** on the **Exit** panel.

6. Click **Model tab** | **Modify Assembly panel** | **Extrude** to start the Extrude feature. Enter 20 mm for the distance and ensure **Cut** is select and click **OK** to complete.

7. The model will look like the below image now.



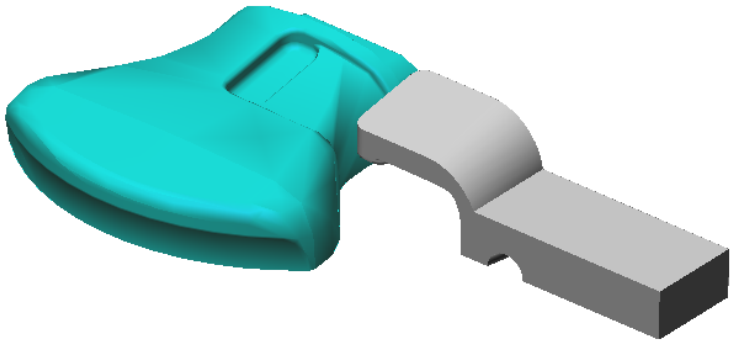
8. **Save** the file and now we are ready to begin the analysis. By removing half of the model we will reduce the amount of elements and nodes the analysis will have to solve saving time.

9. Click **Add-Ins tab** | **Autodesk Simulation panel** | **Start Simulation**. This will open the active model in Autodesk Simulation. If you do not have this Add-In you can open Autodesk Simulation then open the saved Autodesk Inventor assembly (IAM) file.

10. When prompted for to replace existing material property data select **No** at this time. The materials for each part will be assigned in Autodesk Simulation.

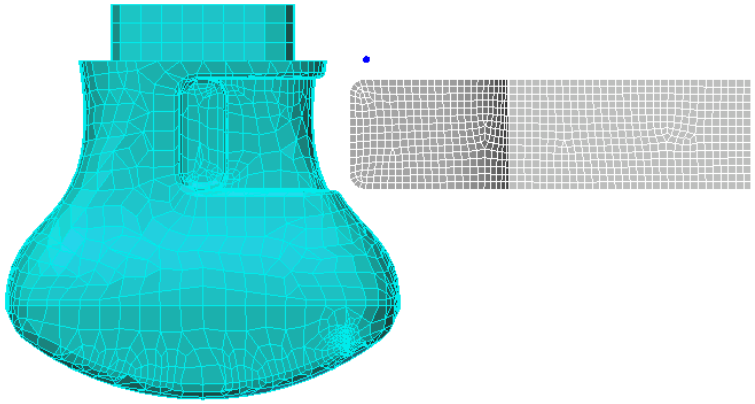
11. Set the **Analysis Type** to **MES with Nonlinear Material Model** in the **FEA Editor** Browser.

12. **Save** the file. The file will be an Autodesk Simulation FEA Model (FEM) file type.



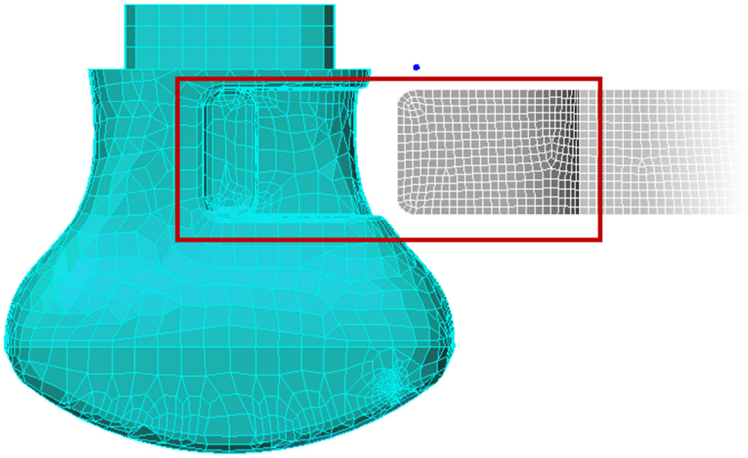
13. Mesh the model using the default setting by clicking **Mesh tab** | **Mesh panel** | **Generate 3D Mesh**.

14. Select **Back** on the **View Cube** to view the model directly from the top.

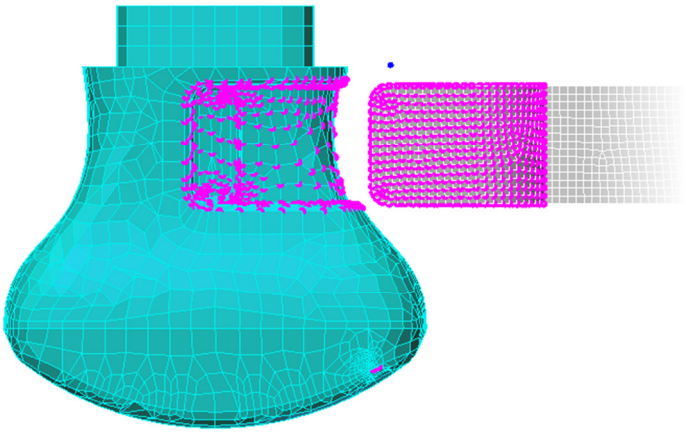


15. Change your selection settings to **Rectangle** for **Shape** and **Surfaces** for **Select**.

16. Draw a rectangle selection as shown below to select all of the surfaces of the primary contact area of the study for the snap-fit.

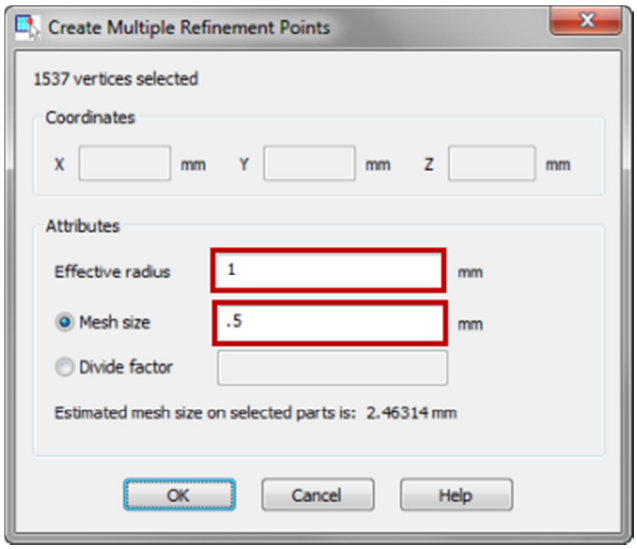


17. Right-Click in the graphic window area and select **Select Subentities** | **Vertices**. This will change the selection to pick all of the nodes on those surfaces.



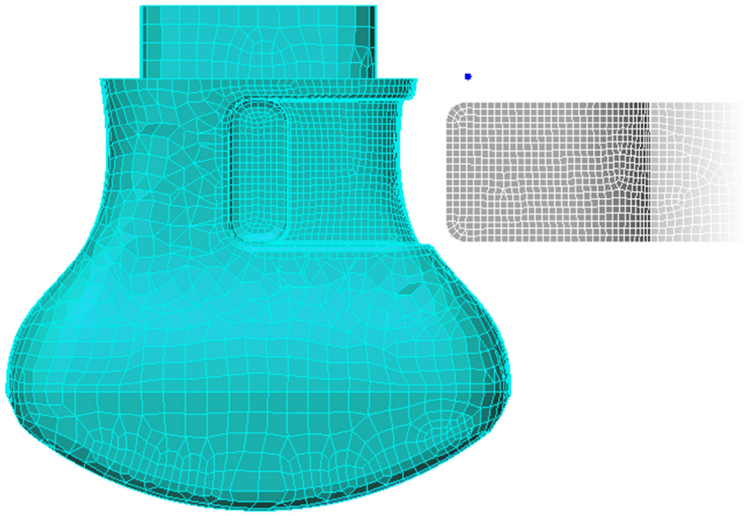
18. To ensure the mesh is controlled in this area we will add mesh refinement points to each of the selected nodes. Click **Mesh tab** | **Refinement Points panel** | **Add to Selected Nodes**.

19. Within the **Create Multiple Refinement Points** dialog enter 1 for the **Effective Radius** and 0.5 for the **Mesh size**. Click **OK** to complete.

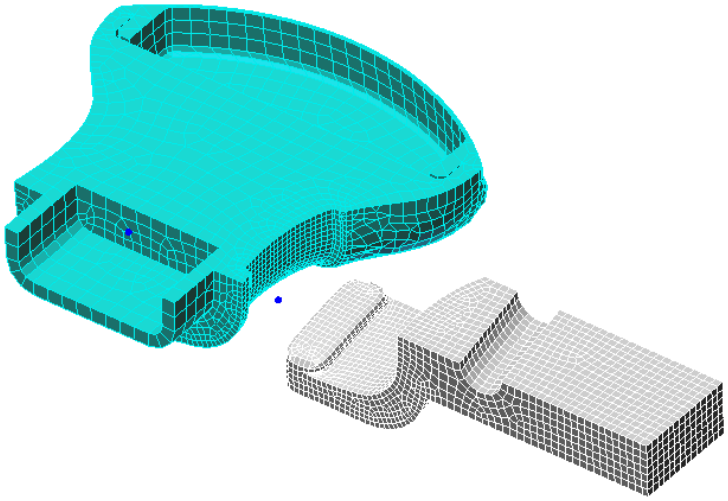


20. Now we will recalculate the mesh of the model. Click **Mesh tab** | **Mesh panel** | **Generate 3D Mesh**.

21. To improve the visualization of the mesh click **Mesh tab** | **Refinement Points panel** | **Visibility** to turn off the visibility of the refinement points. You will notice the nice consistent mesh elements in the area of concern now.

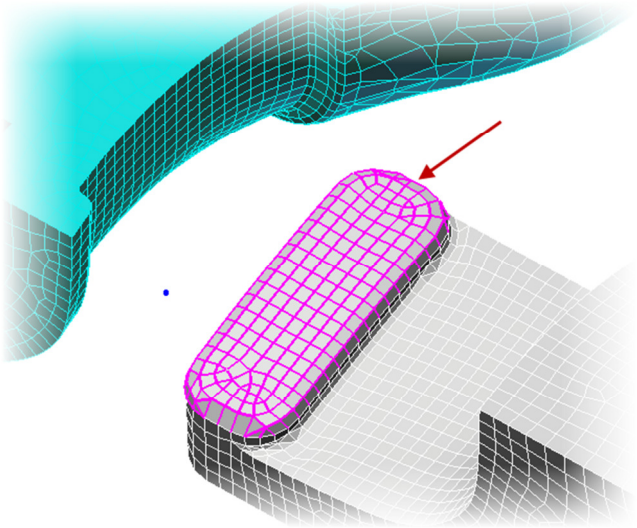


22. **Orbit** the model into a position similar to the image shown below.



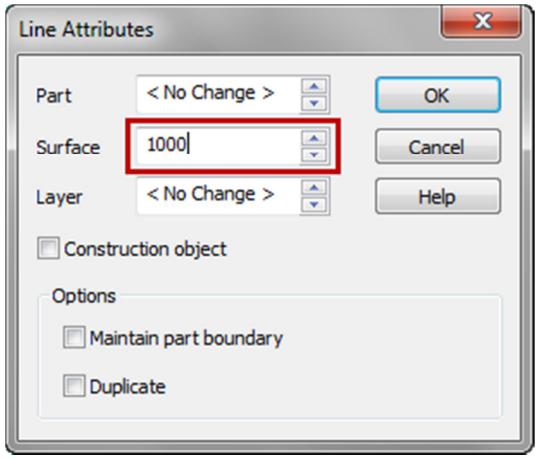
23. Change your selection settings to **Point** for **Shape** and **Surfaces** for **Select**.

24. Zoom in and hold down CTRL and select the 9 surfaces that are shown below that make up the click contact page and edge surfaces.



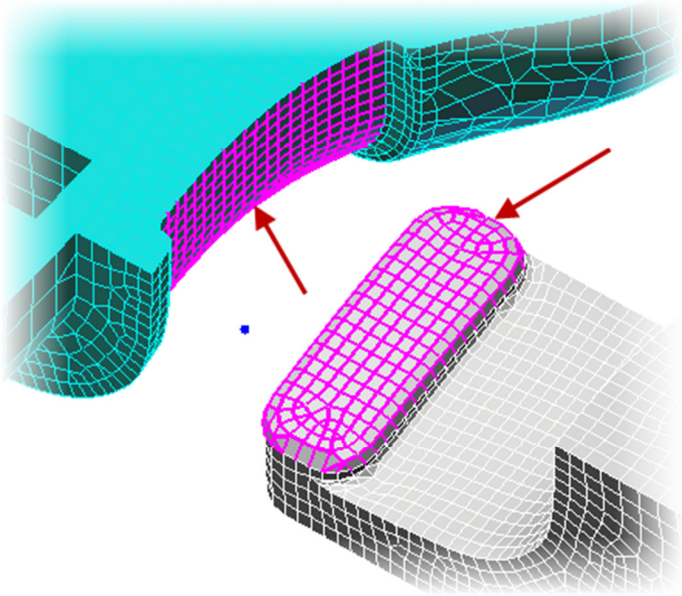
25. Right-Click in the graphics window and select **Select Subentities** | **Lines**. This will select all of the mesh element lines. Then Right-Click in the graphics window again and select **Edit Attributes**.

26. Enter 1000 as the Surface name and click **OK** and then **Yes**. This will group the selected lines into a single surface for easier selection later.

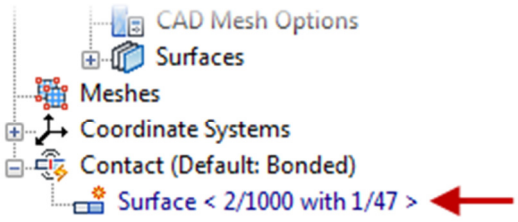


27. Change your selection settings to **Point** for **Shape** and **Surfaces** for **Select**.

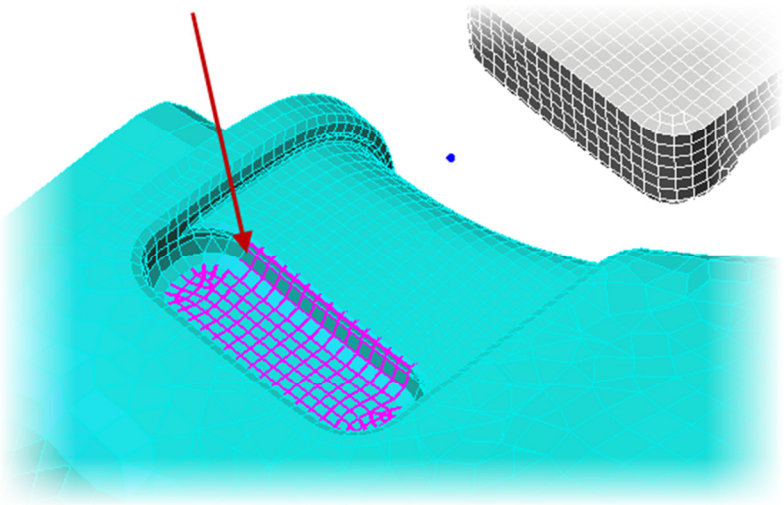
28. Now select the new *1000* single surface and then select the marked contact surface of the *Nose Piece* part.



29. To establish a contact relationship between these two surfaces click **Setup tab** | **Contacts panel** | **Surface-to-Surface Contact**. Then hit **ENTER** to accept the default contact name in the **FEA Editor browser**. This contact will allow the surfaces to touch each other or separate but not interfere.



30. **Orbit** the model and change the selection back to **Point** for **Shape** and **Surfaces** for **Select**. Then select the three surfaces shown below.



31. Right-Click in the graphics window and select **Select Subentities** | **Lines**. This will select all of the mesh element lines. Then Right-Click in the graphics window again and select **Edit Attributes**.

32. Enter 2000 as the Surface name and click **OK** and then **Yes**. This will group the selected lines into a single surface for easier selection later.

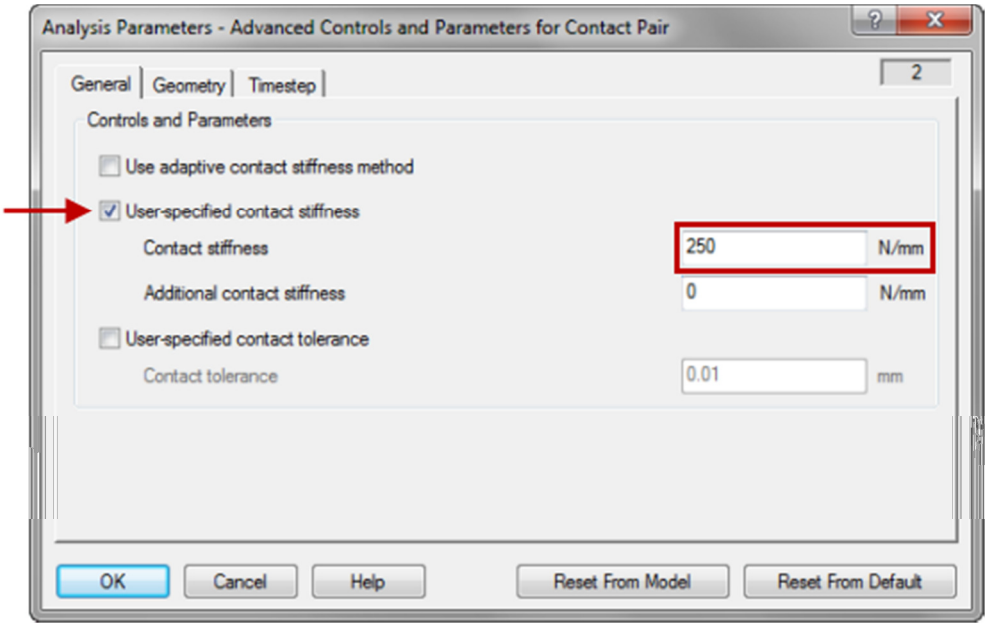
33. Change your selection settings to **Point** for **Shape** and **Surfaces** for **Select**.

34. Select the *1000* and *2000* created surfaces from the browser Surfaces folders or on the model. Then establish a new **Surface-to-Surface Contact** with the default name.

35. Hold down the CTRL and select both of the contacts created in the **FEA Editor** Browser and then Right-Click and select Settings.

36. Click the Advanced button in the **Controls and Parameters for Contact Pair** dialog.

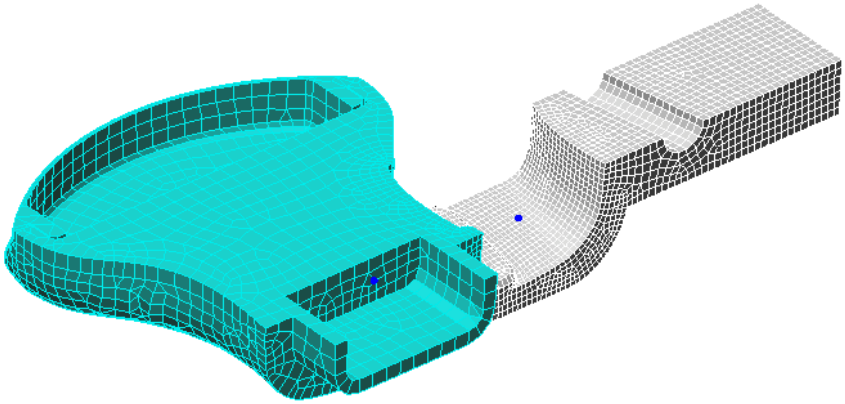
37. Under the General tab uncheck the "**Use adaptive contact stiffness method**" and check the "**User-specified contact stiffness**" checkbox. Enter 250 as the Contact Stiffness.



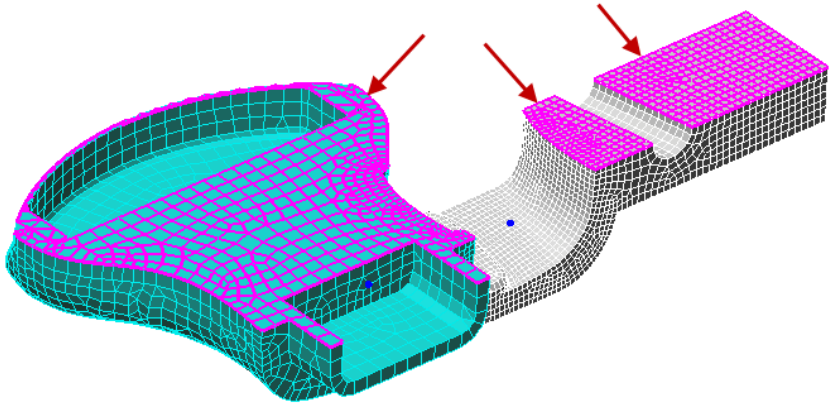
38. Click **OK** and **OK** to close the dialogs

39. Change your selection settings to **Point** for **Shape** and **Surfaces** for **Select**.

40. **Orbit** the model into a position similar to the image below.

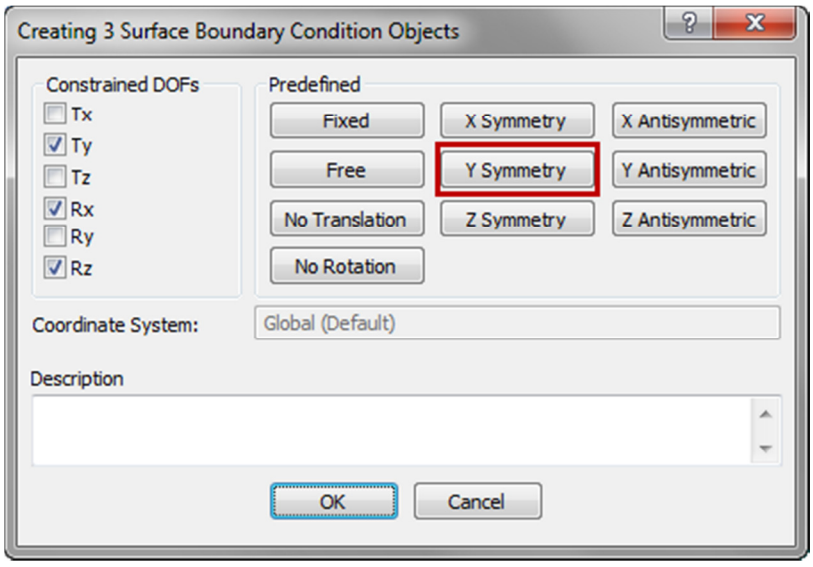


41. Select the three planar surfaces as shown below.

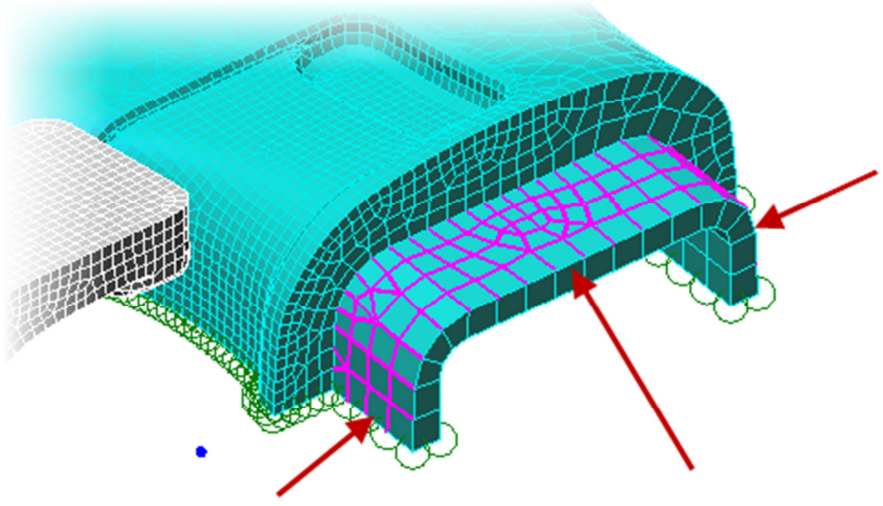


42. With only half of the model imported into Autodesk Simulation we will add a symmetry constraint to the model. Click **Setup tab** | **Constraints panel** | **General Constraint**.

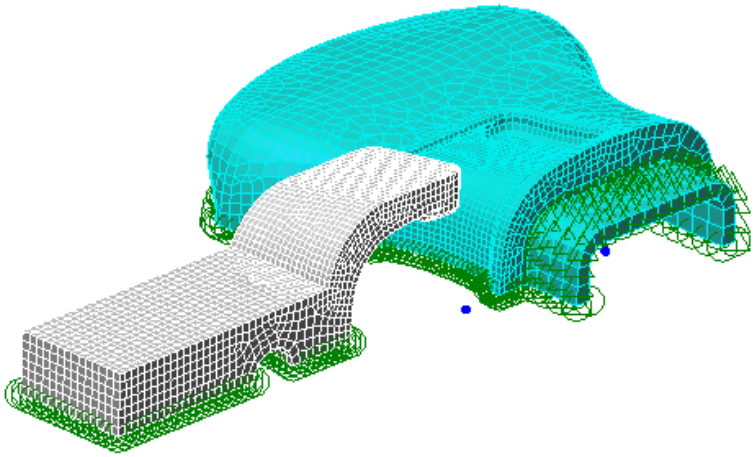
43. Select **Y Symmetry** as the Predefined selection and click **OK**. This will mirror the complete model about the XZ plane.



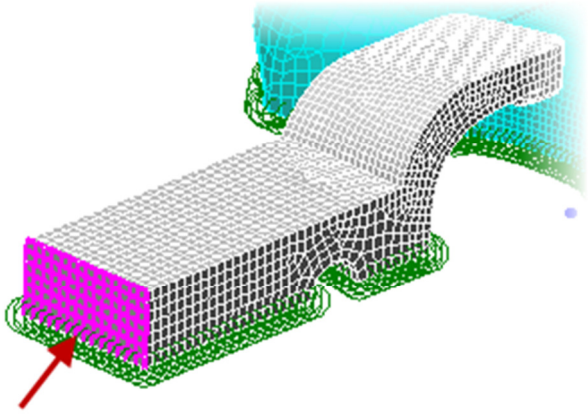
44. Select the five (5) outer surfaces as shown below on the *Nose Piece* part. Add a **Fixed General Constraint** to these surfaces locking all degrees of freedom on their nodes.



45. **Orbit** the model into a position similar to the image shown below.



46. Select the end surface of the *Snap Clip* part as shown below.

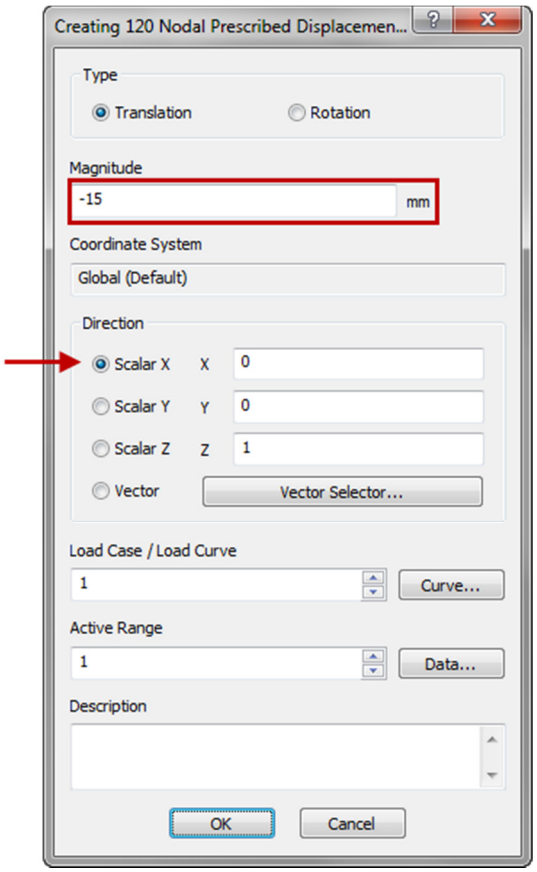


47. Add a **General Constraint** to this surface constraining the **Tz** DOF. This will preview the part from moving in the Z direction.

48. Select the same end surface of the *Snap Clip* part.

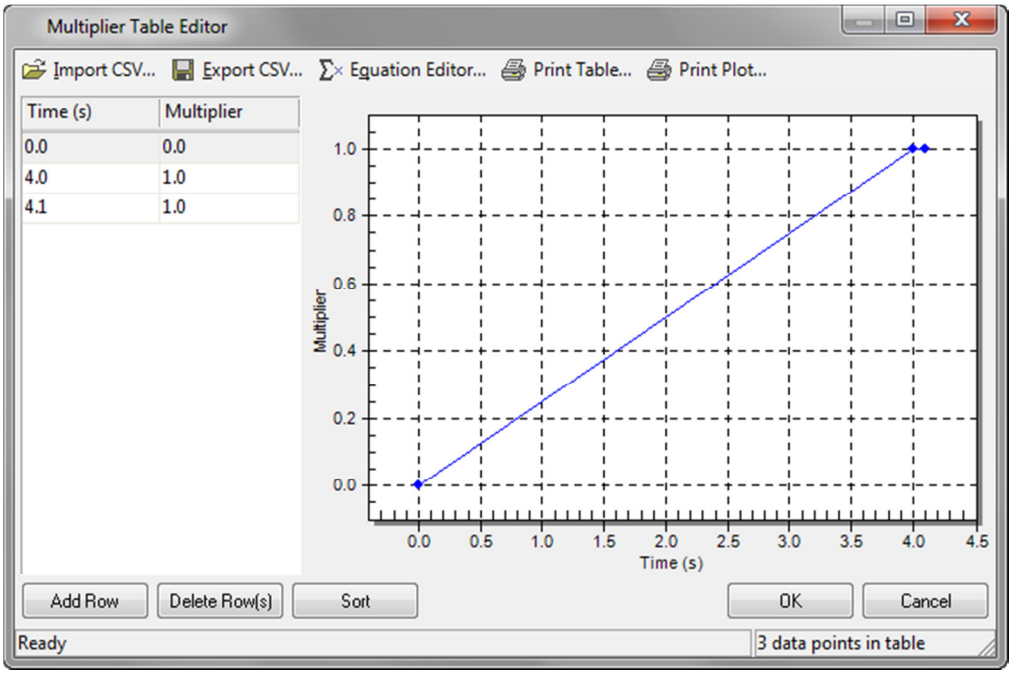
49. Right-Click in the graphics window and select **Select Subentities** | **Vertices**.

50. Now we will assign the displacement of this part to move into the locked position around the Nose Piece. Click **Setup tab** | **Constraints panel** | **Prescribed Displacement** to open the **Prescribed Displacement** dialog. Enter -15 into the Magnitude field and select **Scalar X** as the Direction.



51. Click the **Curve** button.

52. Click **Add Row** to add another time and multiplier entry. Then enter the following values.



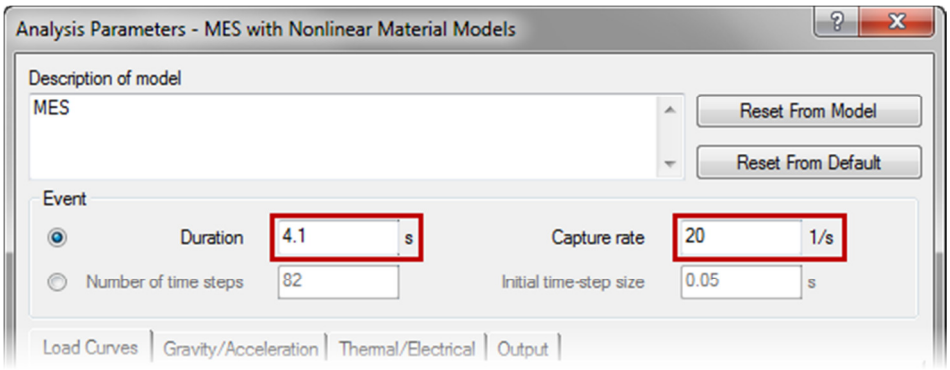
53. Click **OK** to accept the Curve and click **OK** to exit the **Prescribed Displacement** dialog.

54. In the **FEA Editor Browser** multi-select *Part 1* and *Part 2* and then Right-Click and select **Edit | Element Data**. In the **Element Definition** dialog ensure *Large Displacement* is selected as the Analysis Type.

55. Click **OK** to accept values.

56. To setup the overall parameters of the analysis click **Setup tab | Model Setup panel | Parameters**.

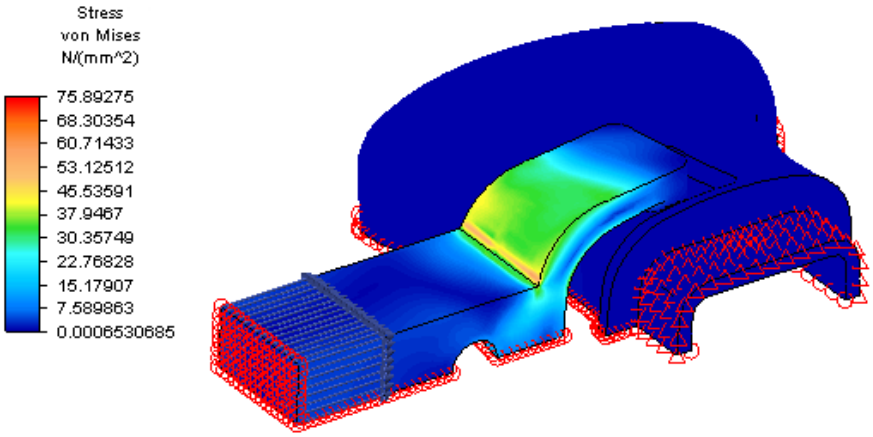
57. Within the **Analysis Parameters** dialog enter 4.1 in the **Duration** field and 20 for the **Capture Rate**.



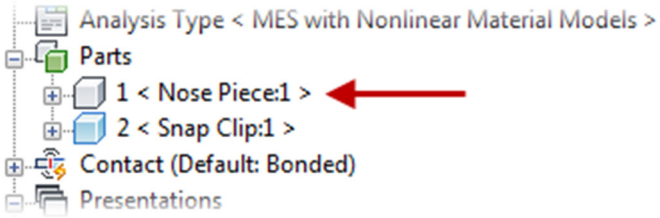
58. Click **OK** to complete.

59. **Save** your model as the solving processing may take some time on your computer.

60. Now you can solve the analysis by clicking **Run Simulation**. This analysis can take up to 30 minutes to complete because of the number of nodes and complexity of the nonlinear analysis. So take some time and enjoy a snack.



61. To focus on the *Snap Clip* part Right-Click on the *Nose Piece* part and turn off the **Visibility**.

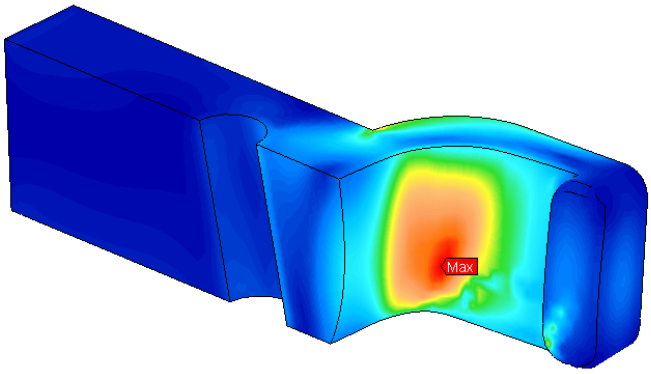


62. Click on **Results Options tab** | **View panel** | **Loads and Constraints**. This will turn off the display of the loads and constraint symbols on nodes.

63. To move the model into a placement where there is contact between the two parts click **Results Contours tab** | **Load Case Options panel** | **Load Case – Middle**. This will set the Load Case to 41 of 82 of the analysis.

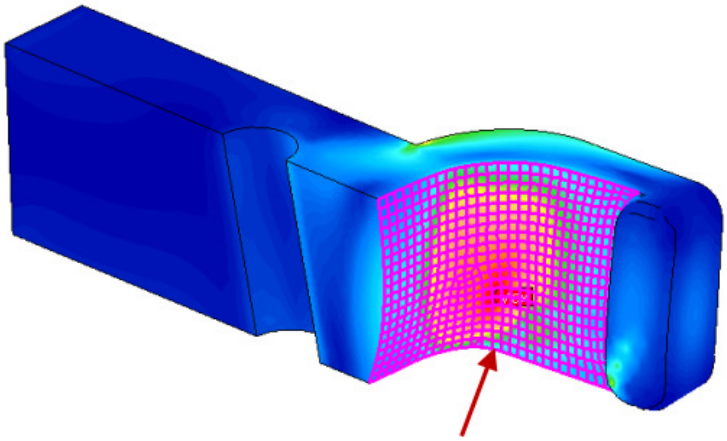
64. Turn on the **Maximum** probe display under the **Results Inquire** tab within the **Probes** panel.

65. **Orbit** the model into a position similar to the image where you can see the inner surface that has the highest stress.



66. Change your selection settings to **Point** for **Shape** and **Surfaces** for **Select**.

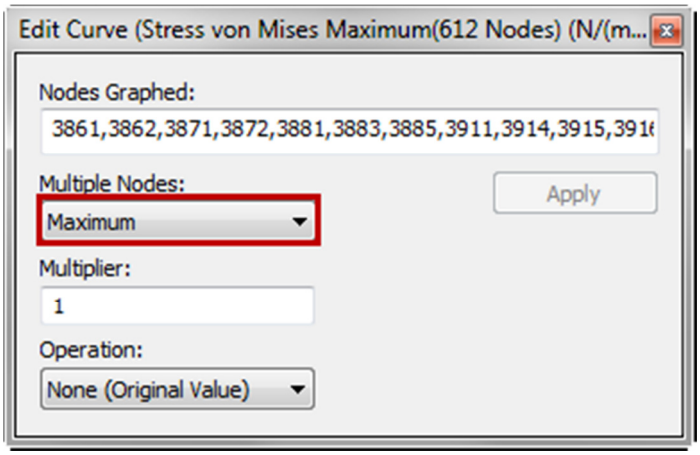
67. Select the surface shown below



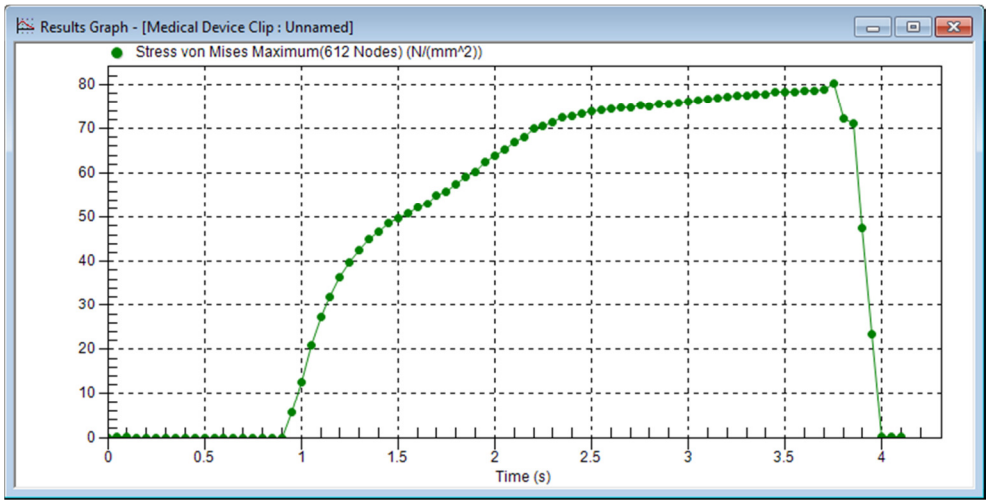
68. Right-Click in the graphics window and select **Select Subentities** | **Nodes**.

69. Now Right-Click in the graphics windows and select **Edit New Graph** to open a graph window and display the **Edit Curve** dialog.

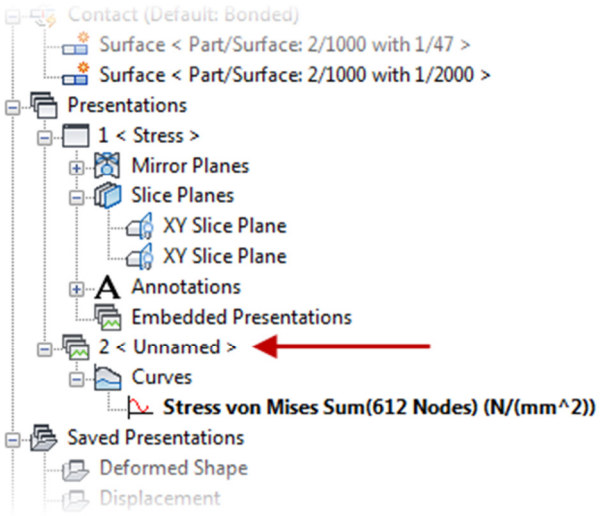
70. With multiple nodes graphed select **Maximum** for **Multiple Nodes** to only graph the highest node at each load case. Then **Close** the dialog.



71. You will notice that the maximum stress is about *81 MPa* and happens at *3.75* seconds.



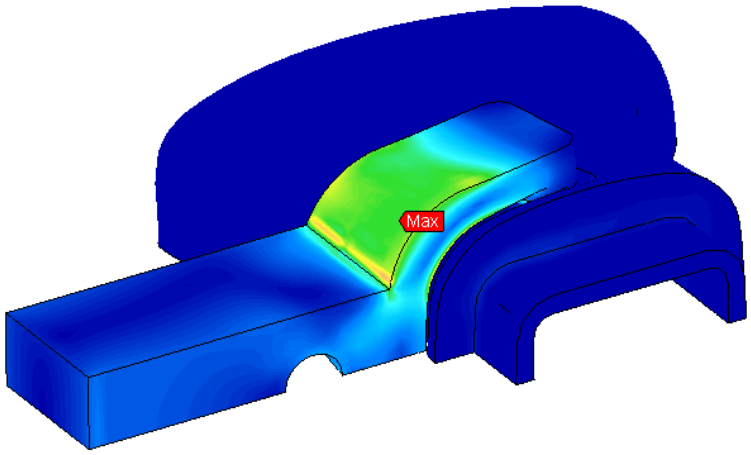
72. Notice the graph is added as *Presentation 2* in the **Results Browser**. You can store multiple presentations within each design scenario.



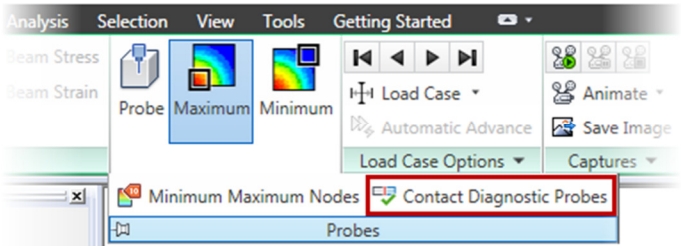
73. Click on the *1 Stress* **Presentation** in the **Results Browser**.

74. Click on the **Results Contours** tab and then set the load case to 75 of 82 which is at the 3.75 second time step. This will place the model in the maximum stress location.

75. Turn the Visibility of the Nose Piece part back on. Then Orbit the model into a position as shown below.



76. Click on the expand arrow of the **Probes** panel and turn off **Contact Diagnostic Probes**. This will not display the chatter probes when the models contact.



77. Click on **Results Contours tab | Settings tab | Legend Properties Settings**. Then click on the **Range Settings** tab.

78. Turn off the "**Automatic calculates value range"** and enter 0 for the Low and 80 for the High. Click **OK** to close.

79. Now to save a movie file of the full analysis. Click on **Captures panel** | **Animate – Save as AVI**.

80. Enter a file name (*Full Analysis Movie*) and click **Save**. The movie will be captures and saves to the computer. When prompted to view the animation click **No**.

81. To prepare a formal report of the analysis click on the Report tab of the Browser. This will provide the default template report.

82. Click **Report tab** | **Setup panel** | **Configure**. This will open the **Configure Report** dialog where you can enter data and configure the presentation of the report before saving and exporting.

83. Click on the **Project Name** in the tree and change the displayed test of the project.

84. Click on the **Title and Author** tree element and enter your name in the **Author** field.

85. Click on the **Executive Summary** tree element and enter a brief description of the results. It would be important to note that the max stress of over 80 MPa so the part would fail based on the material properties.

86. Now we will attach the AVI movie we created to the report. Click on the **Tree** top menu and select **Add AVI File(s)**. Then browse and select the file you created previously and click **Open**. This will insert a new element in the report tress for the movie.

87. Click **Generate Report**.

88. You can now save the report as a PDF by clicking **Save As panel** | **PDF**.