

Remeshing a Deformed Mesh

When the mesh deformation has become so large that the quality of the mesh is too bad, you can generate a new mesh for the deformed configuration and then continue the solver. You can do this by following these steps:

- 1 Add a stop condition
- 2 View the deformed mesh
- 3 Copy the solution
- 4 Create a deformed configuration
- 5 Remesh the deformed configuration
- 6 Continue solving with the new mesh

The following sections contain details about these steps and some additional information.

ADD A STOP CONDITION

Add a stop condition in the solver to make it stop **when** the mesh quality becomes too bad. If you use the time-dependent solver, do this by right-clicking, for example, **Study 1>Solver Configurations>Solver 1>Time-Dependent Solver 1** and selecting **Stop Condition** from the context menu. If you use the parametric stationary solver, do this by right-clicking for example **Study 1>Solver Configurations>Solver 1>Stationary Solver 1>Parametric 1** and selecting **Stop Condition** from the context menu. In the **Stop expression** edit field enter, for example, `mod1.a1e.minqual-0.1` to stop **before** the **mesh quality** becomes **less than 0.1**. The solver stops when the stop expression becomes **negative**.

VIEW THE DEFORMED MESH

You can view the deformed mesh in a **surface plot** or volume plot by selecting **Wireframe** under **Coloring and Style**, and selecting **No refinement** in **Resolution** under **Quality**. In 3D, it is instructive to use an **Element scale factor** less than 1 under **Shrink Elements**.

COPY THE SOLUTION

To keep the first solution, right-click **Study 1>Solver Configurations>Solver 1** and select **Solution>Copy**. The copied solution appears **as a new solver sequence Copy 2** under **Solver Configurations**.

CREATE A DEFORMED CONFIGURATION

Create a deformed configuration by right-clicking, for example, **Results>Data Sets>Solution 2** and selecting **Remesh Deformed Configuration**. The deformed configuration appears as a **new Deformed Configuration 1** node **under Meshes**. The deformed configuration works as a **new geometry** but with restricted functionality. For example, its context menu only contains **Measurements, Export to File, Mesh, Delete, Rename, Properties**, and **Dynamic Help**. The **Settings** window of the deformed configuration indicates which solution it was constructed from. You can also change the parameter or time for which the deformed configuration was generated. If you do this, click the **Update** button to see the corresponding deformed configuration in the graphics.

REMESH THE DEFORMED CONFIGURATION

Expanding the **Deformed Configuration 1** node shows that a new mesh sequence has been added beneath it. This mesh sequence only contains a **Size** feature and a **Reference 1** feature. The reference feature refers to the **original** mesh sequence. This means that the new mesh sequence uses the same features as the original mesh sequence. You can build the new mesh sequence by selecting **Build All** from its context menu. If you want to make changes to the new mesh sequence before building it, right-click **Reference 1** and select **Expand**. Then the features from the original mesh sequence are copied to the new mesh sequence. You can also add and remove features in the new mesh sequence. If you need several meshes on the deformed configuration (for the multigrid solver, for example) you can add an additional mesh sequence by right-clicking **Deformed Configuration 1** and selecting **Mesh**.

CONTINUE SOLVING WITH THE NEW MESH

- 1 In the study step (for example, **Study 1>Step 1: Time Dependent**), use the **Mesh list** to select the new mesh sequence.
- 2 Change the **Times list** or the **Parameter values** list to include only the time or parameter corresponding to the deformed configuration plus the remaining times or parameters.
- 3 Change the **initial value** to be the last time or parameter of the previous solution. Do this by selecting **Study 1>Solver Configurations>Solver 1>Dependent Variables**. Under **Initial Values of Variables Solved For**, change **Method** to **Solution** and **Solution** to **Solver 1** (for example), and select the appropriate time or parameter value in the list **Time** or **Parameter value**. **Usually the Automatic alternative suffices**, which selects the last time or parameter value. Change the settings under **Values of Variables Not Solved For** similarly.

- 4 Solve for the remaining times or parameters by right-clicking **Study 1** and selecting **Compute**.

REMESHING SEVERAL TIMES

You can do multiple remeshings by iterating the above steps. For each of the solver runs you get a copy of the solution (**Copy 2, Copy 3, Copy 4**, and so on) and a corresponding data set (**Solution 2, Solution 3, Solution 4**, and so on). In the plot group you can select one of these data sets for results analysis and visualization.

ALTERNATIVE USING SEVERAL STUDIES

The above procedure uses a single solver sequence that is modified for each solver run. If you want to recompute the whole sequence of runs, an alternative that uses one study for each run is better. To do that, add a new study after meshing each deformed configuration. The copy solution step is not needed in this case. If you have changed settings in the study or its solver sequence, make these changes also in the new study. For example, the stop condition has to be added.