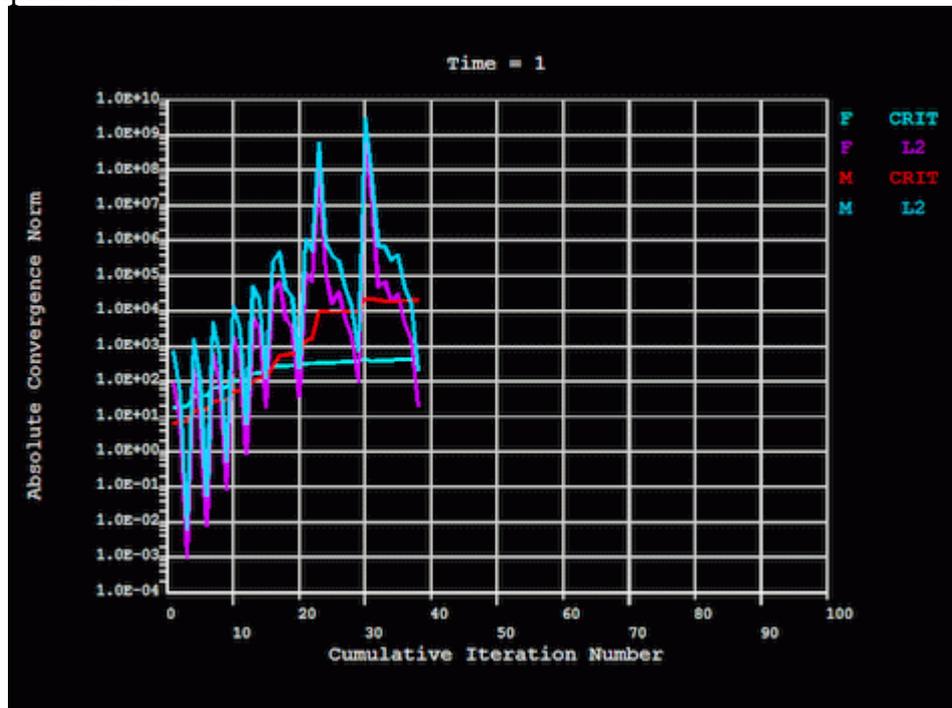


Graphical Solution Tracking

Introduction

This tutorial was completed using ANSYS 7.0 This will act as an explanation of what the Graphical Solution Tracking plot is actually describing. An example of such a plot is shown below and will be used throughout the explanation.



1. Title and Axis Labels

The title of the graph is really just the time value of the last calculated iteration. In this example, the time at the end of the analysis was set to 1. This can be changed with the **Time** command before the Solve command is issued. For more information regarding setting the time value, and many other solution control option, see Chapter 8.5 of the Structural Analysis Guide in the Help file.

The x-axis is labelled `Cumulative Iteration Number`. As ANSYS steps through non-linear analysis, it uses a solver (Newton-Raphson, etc) that iterates to find a solution. If the problem is relatively linear, very few iterations will be required and thus the length of the graph will be small. However, if the solution is highly non-linear, or is not converging, many iterations will be required. The length of the graph in these cases can be quite long. Again, for more information about changing iteration settings, you can see Chapter 8.5 in the help file.

The y-axis is labelled `Absolute Convergence Norm`. In the case of a structural analysis, which this graph is taken from, this absolute convergence norm refers to non-normalized values (ie there are units associated with these values). Some analyses use normalized values. In reality it doesn't really matter because it is only a comparison that is going on. This is what will be explained next.

2. Curves and Legend

As can be guessed from the legend labels, this graph relates to forces and moments. These values are graphed because they are the corresponding values in the solution vector for the DOF's that are active in the elements being used. If this graph were from a thermal analysis, the curves may be for temperature.

For each parameter, there are two curves plotted. For ease of explanation, we will look at the force curves.

- The **F CRIT** curve refers to the convergence criteria force value. This value is equal to the product of **VALUE** \times **TOLER**. The default value of **VALUE** is the square root of the sum of the squares (SRSS) of the applied loads, or **MINREF** (which defaults to 0.001), whichever is greater. This value can be changed using the **CNVTOL** command, which is discussed in the help file. The value of **TOLER** defaults to 0.5% for loads.

One may inquire why the **F CRIT** value increases as the number of iterations increases. This is because the analysis is made up of a number of substeps. In the case of a structural example, such as this, these substeps are basically portions of the total load being applied over time. For instance, a 100N load broken up with 20 substeps means 20, 5N loads will be applied consecutively until the entire 100N is applied. Thus, the **F CRIT** value at the start will be 1/20th of the final **F CRIT** value.

- The **F L2** curve refers to the L2 Vector Norm of the forces. The L2 norm is the SRSS of the force imbalances for all DOF's. In simpler terms, this is the SRSS of the difference between the calculated internal force at a particular DOF and the external force in that direction.

For each substep, ANSYS iterates until the **F L2** value is below the **F CRIT** value. Once this occurs, it is deemed the solution is within tolerance of the correct solution and it moves on to the next substep. Generally, when the curves peak this is the start of a new substep. As can be seen in the graph above, a peak follows everytime the **L2** value drops below the **CRIT** value, as expected.