

ANSYS Tutorial

Creating Load Steps for a Transient Analysis

This tutorial demonstrates how to create load steps in ANSYS for a transient analysis.

(Go to Main Menu)

Preprocessor

Element Type

Add/Edit/Delete

Add

Structural & Solid & Quad 4 node 42 & **OK**

Material Props

Material Models

Structural

Linear

Elastic

Isotropic

EX = 210e9

PRXY = 0.3 & **OK**

Modeling

Create

Areas

Rectangle

By Dimensions

(Click X1 box) 0 (Click X2 box) 20

(Click Y1 box) 0 (Click Y2 box) 5 & **OK**

Meshing

Mesh Tool

Click on the Lines Set button

Pick lines 2 & 4 & **Apply**

NDIV = 10

SPACE = -2 & **Apply**

Pick lines 1 & 3 & **Apply**

NDIV = 30

SPACE = -2 & **OK**

Click the **Mesh Button**

Solution

Analysis Type

New Analysis

Choose **Transient** & **OK** & **OK**

Sol'n Controls**Basic Tab**Time at end of loadstep: 10Click on **Number of substeps**Number of substeps: 10Frequency: Choose **Write N number of substeps**Where N = 10

The screenshot shows the 'Solution Controls' dialog box with the 'Basic' tab selected. The 'Analysis Options' section has 'Small Displacement Transient' selected in the dropdown and 'Calculate prestress effects' unchecked. The 'Time Control' section has 'Time at end of loadstep' set to 10, 'Automatic time stepping' set to 'Prog Chosen', and 'Number of substeps' selected with a value of 10. The 'Max no. of substeps' and 'Min no. of substeps' are both set to 0. The 'Write Items to Results File' section has 'All solution items' selected, and a list of items including 'Nodal DOF Solution', 'Nodal Reaction Loads', 'Element Solution', 'Element Nodal Loads', and 'Element Nodal Stresses'. The 'Frequency' section has 'Write N number of substeps' selected in the dropdown, and 'where N =' set to 10. At the bottom are 'OK', 'Cancel', and 'Help' buttons.

Solution Controls

Basic | Transient | Sol'n Options | Nonlinear | Advanced NL

Analysis Options

Small Displacement Transient

☐ Calculate prestress effects

Time Control

Time at end of loadstep: 10

Automatic time stepping: Prog Chosen

☒ Number of substeps

☐ Time increment

Number of substeps: 10

Max no. of substeps: 0

Min no. of substeps: 0

Write Items to Results File

☒ All solution items

☐ Basic quantities

☐ User selected

Nodal DOF Solution

Nodal Reaction Loads

Element Solution

Element Nodal Loads

Element Nodal Stresses

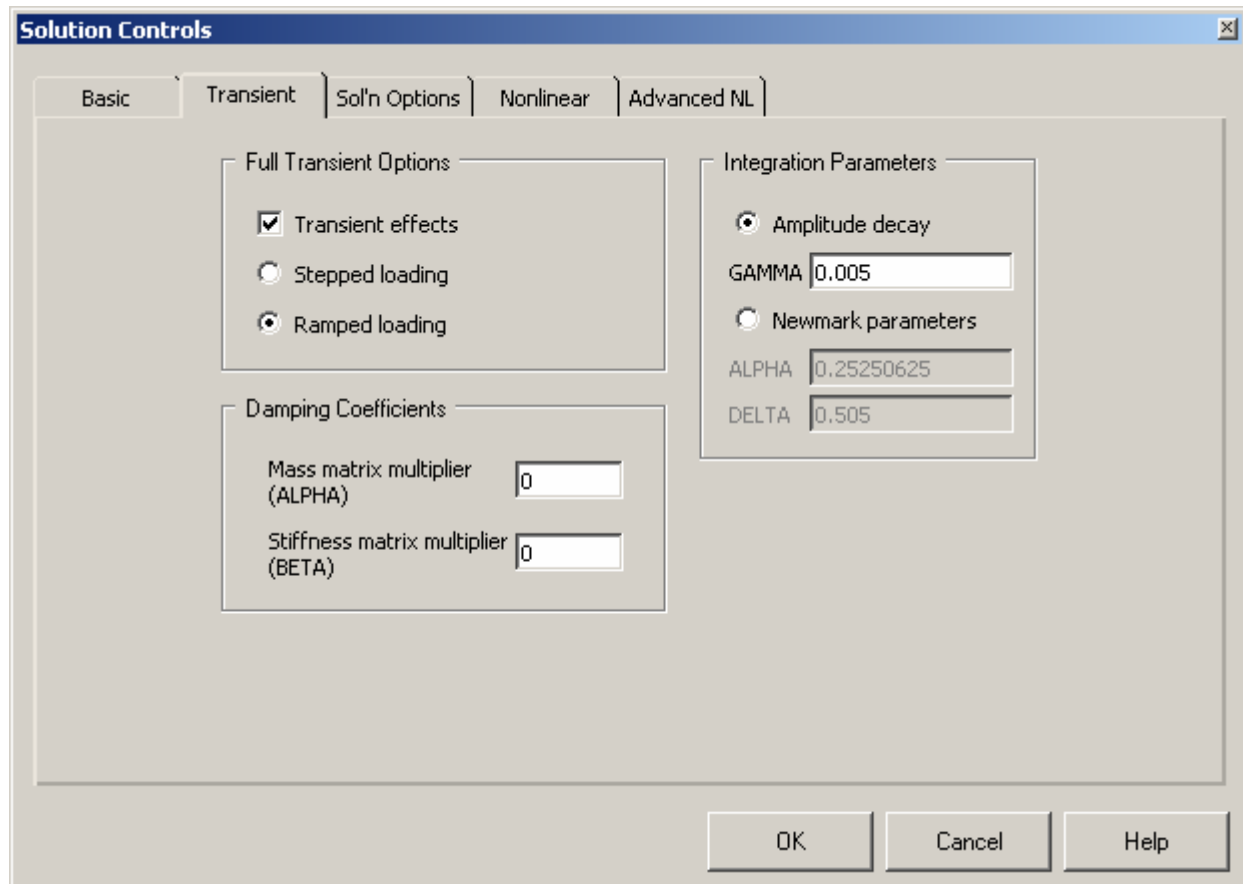
Frequency:

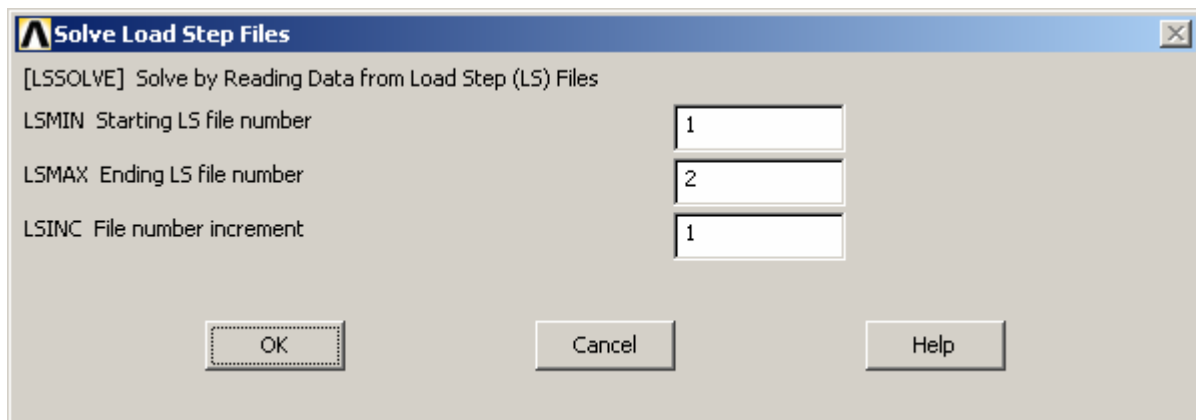
Write N number of substeps

where N = 10

OK Cancel Help

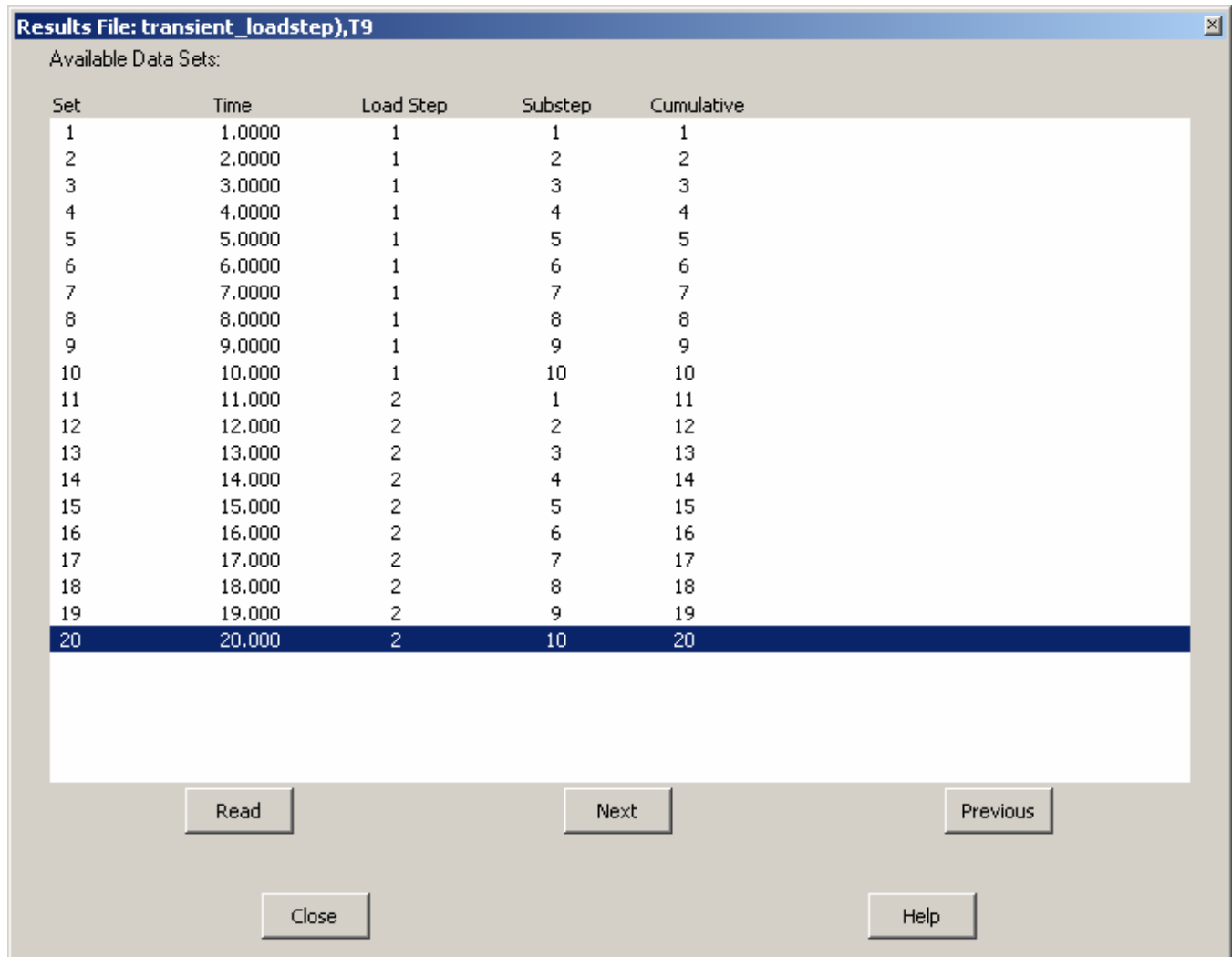
Transient TabChoose **Ramped Loading** & **OK**

**Define Loads****Apply****Structural****Displacement****On Lines**Pick line **2** & **OK**Select **All DOFs**Displacement Value = **0** & **OK****Force/Moment****On Keypoints**Pick keypoints **2** & **3** & **OK**Direction of Force/Moment: **FY**Force/Moment Value: **5000** & **OK****Load Step Opts****Write LS File**Load step file number n: **1** & **OK**

Analysis Type**Sol'n Controls****Basic Tab**Time at end of loadstep: 20Click on **Number of substeps**Number of substeps: 10Frequency: Choose **Write N number of substeps**Where N = 10**Define Loads****Delete****Structural****Force/Moment****On Keypoints****Pick All & OK****Apply****Structural****Force/Moment****On Keypoints**Pick keypoints **2 & 3 & OK**Direction of Force/Moment: **FY**Force/Moment Value: **-5000 & OK****Load Step Opts****Write LS File**Load step file number n: 2 & OK**Solve****From LS Files**Starting LS file number: 1Ending LS file number: 2File number increment: 1 & OK

Reviewing Transient Results

(Go to Main Menu)

General Postproc**Read Results****By Pick**(select the load step and substep of interest) & **Read & Close****Plot Results****Contour Plot**

(Choose the plots of interest)

Animating Results

(Go to Utility Menu)

PlotCtrls**Animate****Over Time ...**Number of animation frames: **20**Model result data: select **Load Step Range**Range Minimum: **1**Range Maximum: **2**Contour data for animation: select **Stress & von Mises****OK**