

The Focus



A Publication for ANSYS Users

Contents

Feature Articles

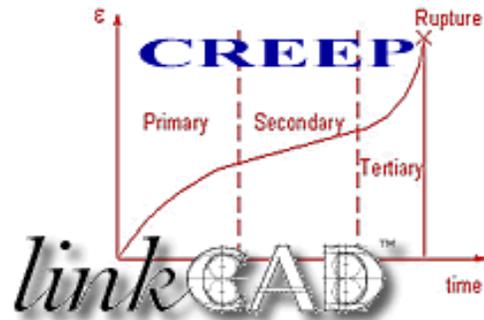
- [Using ANSYS for Creep Analysis](#)
- [“Running Start”: Combining Engineering and Mentoring](#)
- [Get EE Data into ANSYS with Electronic Layout File Reader](#)

On the Web

- [Grupo SSC E-zine \(en Español\)](#)

Resources

- [PADT Support: How can we help?](#)
- [Upcoming Training at PADT](#)
- [About *The Focus*](#)
 - *The Focus* Library
 - Contributor Information
 - Subscribe / Unsubscribe
 - Legal Disclaimer

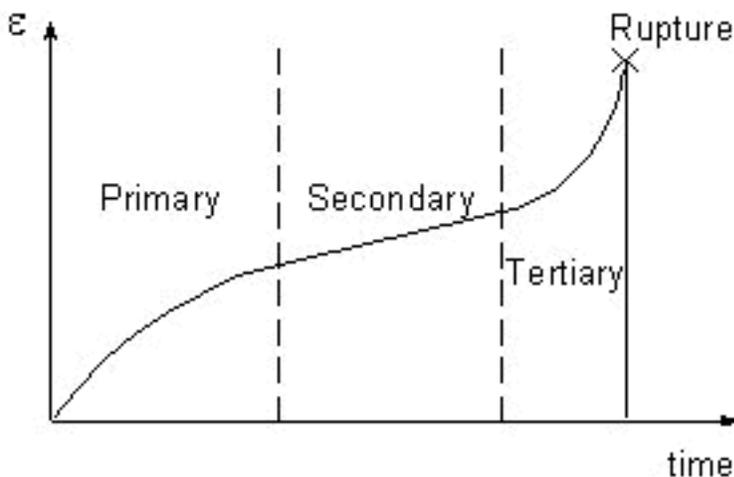


ANSYS
Support Distributor

Using ANSYS for Creep Analysis

by [Carlos Shultz](#) of PADT

Creep is a rate dependent material nonlinearity in which the material continues to deform under a constant load. The three stages of creep are shown in the figure below. ANSYS has the capability of modeling the first two stages (primary and secondary). The tertiary stage is usually not analyzed since it implies impending failure (gross element distortion).



Creep strain due to constant applied stress

ANSYS analyzes creep using two time integration methods. Both are applicable to static or transient analyses.

- *Implicit* creep method is robust, fast, accurate, and recommended for general use.
 - Handles temperature dependent creep constants, as well as simultaneous coupling with isotropic hardening plasticity models.
- *Explicit* creep method is useful for cases where very small time steps are required.
 - Creep constants cannot be dependent on temperature.
 - Coupling with other plastic models is available by superposition only.

The Focus



A Publication for ANSYS Users

Implicit Creep

Implicit creep refers to the use of backward Euler integration for creep strains. This method is numerically unconditionally stable, which means that it does not require as small a time-step as the explicit creep method, so it is much faster overall.

For implicit creep plus rate-independent plasticity, the plasticity correction and creep correction are done at the same time, not independently. Consequently, implicit creep is generally more accurate than explicit creep, but it is still dependent on the time-step size. A small enough time-step must be used to capture the path-dependent behavior accurately.

The following example input shows the use of the implicit creep method. TBOPT=2 specifies that the primary creep equation for model 2 will be used. Temperature dependency is specified using the TBTEMP command, and the four constants associated with this equation are specified as arguments with the TBDATA command.

```
TB, CREEP, 1, 1, 4, 2
TBTEMP, T1
TBDATA, 1, C1, C2, C3, C4
```

You can simultaneously model implicit creep and BISO, MISO, NLISO, BKIN, and HILL plasticity.

RATE command can be used to turn implicit creep on and off. Useful for setting up initial conditions prior to a creep analysis (plasticity is often active during the ramp up phase of a creep analysis).

```
/SOLU
RATE, OFF !Creep calculations turned off
TIME, 1.0E-8 !Time period set to a very small value
SOLVE !First load step
RATE, ON !Creep analysis turned on
TIME, T1 !Time period set to desired value
SOLVE !Second load step
```

Enforce a creep limit ratio using the creep ratio control option in commands CRPLIM or CUTCONTROL, CRPLIMIT. A recommended value for a creep limit ratio ranges from 1 to 10.

The Focus



A Publication for ANSYS Users

Explicit Creep

Explicit creep means that the forward Euler method is used for the calculation of creep strain evolution. The creep strain rate used at each time step corresponds to the rate at the beginning of the time step and is assumed to be constant throughout that time step, Δt . Because of this, very small time steps are required to minimize error.

For explicit creep with plasticity, plasticity correction is performed first followed by creep correction. These two corrections occur at different stress values; therefore, it may be less accurate.

Explicit creep is no longer recommended for creep analysis. If you are learning to use creep in ANSYS, learn to use the implicit creep method.

Life Calculations for Creeping Components

Currently there is not a standard criterion for estimating the life of components subjected to various amounts of stress and temperature. Several different methods have been proposed; here are a few of them:

1. *Time-hardening rule*, total accumulated creep strain is the sum of creep strains at each level of exposure, the state of the component moves along constant time from curve to curve, failure occurs when the part reaches the end of a curve
2. *Strain-hardening rule*, differs from time-hardening in that the state moves along constant strain from curve to curve, failure occurs when the part reaches the end of a curve
3. *Life-fraction rule*, differs from time-hardening in that the state moves from curve to curve by determining the ratio of exposure time to total life which has been used up to that time and then moving to the next curve at that same ratio, failure occurs when the part reaches the end of a curve

The life-fraction rule is considered intermediate to the time- and strain-hardening rules; these bound real life behavior. These rules are sometimes implied by the choice of creep law. If you use a time-hardening creep equation, you are assuming

The Focus



A Publication for ANSYS Users

that the component follows a time-hardening rule. More complex relationships can be implemented through the use of user defined creep routines (please [contact us](#) for additional information regarding user creep routines).

General Recommendations

1. Prevent fictitious stresses from causing convergence issues
 - a. Use broad boundary conditions to hold components, fictitious stresses often occur when fixed displacements are applied
 - b. Use pressures and accelerations to load components, fictitious stresses often occur when fixed forces are applied
 - c. If fictitious stresses are unavoidable, create a layer of linear elements (without the creep material model active) between the high stresses and the rest of the model
2. Test your creep model on a simple test case
 - a. 3D bars with constant stress applied is my preferred test case
 - b. Correlate by checking deflections vs time to verify that your model is correct
 - c. Be aware that creep data is often constant load data that is presented as constant stress data, adjusting for this is not a simply task
3. Solution Controls, turn SOLCON, *on* and use NSUBST, *4, 1e6, 4*
 - a. Keep it simple, only adjust solution parameters if the defaults fail you
4. Deflection limited analyses much easier than analysis of rupture
 - a. Element distortion will cause convergence issues when trying to model failure
 - b. Biasing the mesh, by making the elements thinner in the anticipated primary direction of stress, can delay mesh distortion since the loading will be deforming the elements into a better shape

The Focus



A Publication for ANSYS Users

Running Start Combining Engineering and Mentoring

by [Jeff Strain](#) of PADT

Give a man a fish, and he will eat for a day. Teach him to fish and he will eat for a lifetime.

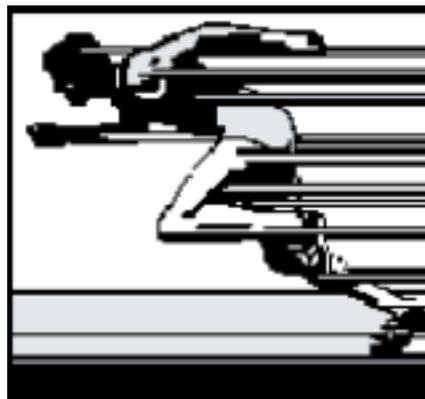
Chinese Proverb



Learning alone can be slow!

To address this concern, PADT offers Running Start, a combination of our engineering, mentoring, and training services. The foundation of this is our engineering service, where we perform a real world analysis to address a concern that is typical of the problems you solve on a regular basis. The deliverables from our engineering services would include pertinent ANSYS files, such as the database and results files, and an analysis report.

However, with Running Start we don't stop there. In addition, we would educate you on the analysis procedure so that you not only would be able to repeat it on your own, but build new skills upon what you have learned. Depending on your desires, this would include ANSYS script files (well organized and heavily commented), a step-by-step presentation, and a discussion with you or your group about the procedure. You may also opt for customized training and follow-up support.



The Focus



A Publication for ANSYS Users

Whether you are new to ANSYS and want to learn analysis procedures for real world problems, or a veteran ANSYS user embarking on a new application, Running Start could be just the thing to get you going. Not only would you have to devote fewer manhours to figuring out specific problems, you would also be provided with ANSYS scripts that could be reused or altered based on your needs.



The lessons you learn from Running Start can also be applied to future analysis challenges you may encounter. Any customized training you request would be targeted toward specific applications and therefore more relevant to your daily tasks.

If you are interested in the Running Start package, please call us at (480) 813-4884 or (800) 293-PADT, or contact us through email at info@padtinc.com.

The Focus



A Publication for ANSYS Users

Get EE Data into ANSYS with Electronic Layout File Reader

by [Eric Miller](#) of PADT

The Cool Add-ons for ANSYS Series

The other day a user mentioned a specific feature that ANSYS did not contain. In looking for a solution, I noticed that there are a lot of cool little programs out there that people are selling that don't seem to get a lot of press. So, since Rod is always e-mailing me about new articles, it seemed *The Focus* would be a good place to highlight a few. This issue will highlight a great little tool for reading the alphabet soup of electronic system layout formats into ANSYS.

LinkCAD"/ANSYS

It may seem like that Electrical Engineer a few cubicles over is always throwing you a new file format that you have to struggle to read. With the release of LinkCAD/ANSYS, [Bay Technologies](#) in Aptos, California (Duuude!) has delivered a tool that can truly remove a lot of pain for many users. Based on their proven LinkCAD translator, this tool adds the ability to output EE layout geometry directly into ANSYS. That's right...no IGES, no importing into a 2D CAD system to clean it up, no custom readers! The table below shows all of the input and output formats supported in this first release.



Format	Import	Export
DXF	Yes	Yes
CIF	Yes	Yes
GDS-II	Yes	Yes
IE3D	Yes	Yes
Gerber RS-274x/RS-274D	Yes	Yes
PostScript		Yes
ANSYS ANF		Yes
Parasolid x_t		Yes
TETIN		Yes

Supported LinkCAD/ANSYS formats.

See What You're Working On!

It's an idea whose time has come. LinkCAD VIEW lets you see the file you're (supposed) to be working on, both big picture and layer by layer. Is that really the file you asked for? Click on the light bulbs to turn individual layers on and off. Zoom and cruise the CAD drawing. Since LinkCAD automatically finds broken polylines before exporting, you can actually see them before you request they be repaired.

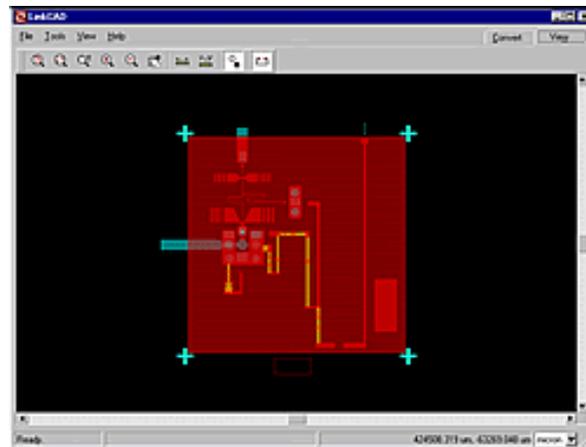
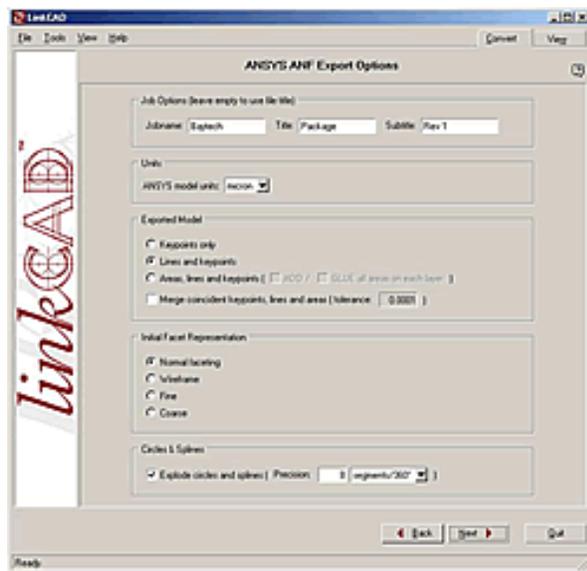
The Focus



A Publication for ANSYS Users

In some early testing, PADT has had a lot of success with this tool, and it is proving to be a much easier way to get geometry for PCBs, chip packages, and MEMS devices. The program has a simple wizard-like interface with lots of useful options and a very nice layer oriented viewing window. You can [download](#) a 15-day demo copy and try it for yourself, or [see](#) the full list of features and screen shots of the interface. At just \$3,500, it can save you a lot of heartache and cut down on the cursing of EE guys.

You can purchase LinkCAD/ANSYS direct from Bay Technology or from most of the ANSYS ASDs in North America.



The Focus



A Publication for ANSYS Users

About *The Focus*

The Focus is a periodic electronic publication published by PADT, aimed at the general ANSYS user. The goal of the feature articles is to inform users of the capabilities ANSYS offers and to provide useful tips and hints on using these products more effectively. *The Focus* may be freely redistributed in its entirety. For administrative questions, please contact [Rod Scholl](#) at PADT.

The Focus Library

All past issues of *The Focus* are maintained in an online [library](#), which can be searched in a variety of different ways.

Contributor Information

Please don't hesitate to send in a contribution! Articles and information helpful to ANSYS users are very much welcomed and appreciated. We encourage you to send your contributions via e-mail to [Rod Scholl](#).

Subscribe / Unsubscribe

To subscribe to or unsubscribe from *The Focus*, please visit the PADT e-Publication [subscriptions](#) management page.

Legal Disclaimer

Phoenix Analysis and Design Technologies (PADT) makes no representations about the suitability of the information contained in these documents and related graphics for any purpose. All such document and related graphics are provided as is without warranty of any kind and are subject to change without notice. The entire risk arising out of their use remains with the recipient. In no event, including inaccurate information, shall PADT be liable for any direct, consequential, incidental, special, punitive or other damages whatsoever (including without limitation, damages for loss of business information), even if PADT has been advised of the possibility of such damages.

The views expressed in *The Focus* are solely those of PADT and are not necessarily those of ANSYS, Inc.