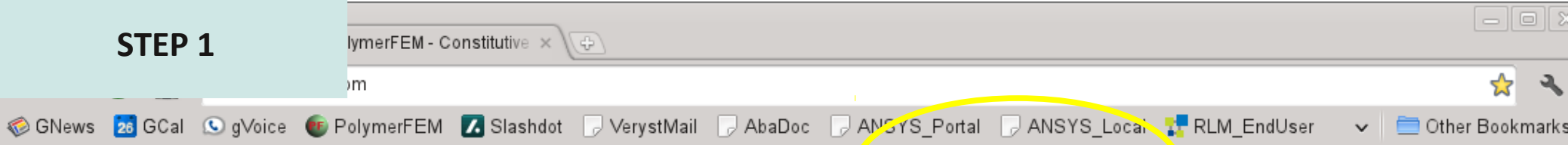


# STEP 1



**PolymerFEM.com**  
MODELING & SIMULATION

Welcome, **Jorgen** Notifications: 2 My Profile Settings Register Log Out

Home Forum Blogs What's New? **First Sign In**

Front Page News About Training Classes MCalibration PolyUMod User-Material Models Experimental Data Advanced Search

Then click on the forums page

Home Front Page

**Menu**

- News
- About
- Training Classes
- MCalibration
- PolyUMod User-Material Models
- Experimental Data
- Case Studies

**Recent Blog Posts**

**ANSYS applied dynamic stress analysis**  
Oh...thanks. I got it answered anyways, but I'll remember that for next time!  
mzintegral 2011-08-03 14:35

**ANSYS applied dynamic stress analysis**  
As your question on the forums page instead.  
-Jorgen  
Jorgen 2011-08-03 14:35

## FRONT PAGE

**New Finite Element Modeling of Polymers Training Class**  
by Jorgen



### New Online Training Class

- Do you perform FEA of polymer components but do not know the tradeoffs, pitfalls, and benefits of different constitutive models, modeling options, and material test methods?
- Do you want to get a *competitive advantage* by using more advanced and accurate FEA techniques?

**The next class will be given on Oct 18 - 19, 2011.**

The following pdf-file gives more information about the upcoming training class, including information about how to register.  
...

[Read More](#)

### FEA of PLLA Interference Screws

by Jorgen Published on 2011-08-22 14:55

### MCalibration Movie Demo

by Jorgen Published on 2010-09-07 23:25



I will be giving a



### Recent Forum Posts

- Applying a Boundary Condition to a node in an Abaqus input file**  
Hi Guys,  
How do you apply a boundary condition to a specific node in an Abaqus input file? When an input file is generated a set is created for each node. Here is an example  
\*\* BOUNDARY...  
slimjimmy 2011-09-26, 08:13
- Dear all,  
I am facing the referred problem? Can anyone help me?  
@ Giulio: Were you able to solve the problem? If so, can you please share what you did...  
Best wishes,  
cardoso  
cardoso 2011-09-26, 01:41
- You do realize that this**

# STEP 2



## Forum

### PolymerFEM - Constitutive Models

Welcome to the PolymerFEM - Constitutive Model...

Select a Topic

#### Finite Element Modeling

Last Post

General discussion and questions about finite element modeling.



**Constitutive Models** (1 Viewing)  
Discussion about constitutive models



Threads: 504  
Posts: 2,119

How to simulate dynamic stiffness of rubber...  
by kunal\_deshmukh55  
2011-09-22, 01:55



**Finite Element Modeling** (8 Viewing)  
FE modeling and analysis



Threads: 999  
Posts: 3,712

Applying a Boundary Condition to a node in...  
by slimjimmy  
Today, 08:13



**Computer Software** (1 Viewing)  
Computer software and tools



Threads: 149  
Posts: 551

welding of two plates in abaqus  
by hessamvalidi  
2011-09-17, 08:23

#### Polymeric Materials

Last Post

Information about polymeric materials



**Thermoplastics**  
Behavior of thermoplastics



Threads: 63  
Posts: 198

which failure model in abaqus for pa66...  
by hust\_cad  
2011-09-14, 23:09



**Elastomers**  
Behavior of elastomers and rubbers



Threads: 111  
Posts: 479

marc mentat simulation of rubber automotive...  
by Jorgen  
2011-08-08, 21:03



**Thermosets**  
Behavior of thermoset polymers



Threads: 13  
Posts: 40

Eccobond 285 Properties  
by Jorgen  
2011-02-20, 19:20



**Biopolymers**



Threads: 11

No creep response from poroelastic

# STEP 3

Constitutive Models

forumdisplay.php?22-Constitutive-Models



Forum Finite Element Modeling Constitutive Models

Click on "Post New Thread"

+ Post New Thread

Threads 1 to 20 of 504 Page 1 of 26 1 2 3 11 ... Last

## Forum: Constitutive Models

Discussion about constitutive models

Forum Tools Administrative Search Forum Inline Mod

Title / Thread Starter	Replies / Views	Last Post By
How to simulate dynamic stiffness of rubber component?! Started by byhyj, 2004-10-24 02:24	Replies: 4 Views: 4,071	kunal_deshmukh55 2011-09-22, 01:55
Constitutive Modeling of PP Started by Mart, 2011-09-21 07:34	Replies: 0 Views: 63	Mart 2011-09-21, 07:34
Extract DDSDE from Abaqus material model library Started by tlesicar, 2011-08-26 04:27	Replies: 4 Views: 287	tlesicar 2011-09-15, 10:14
😊 Constitutive equation umat! Started by wangying, 2011-09-12 08:50	Replies: 0 Views: 95	wangying 2011-09-12, 08:50
Constitutive Material Model for Thermoplastic Elastomer Started by sharathchav, 2011-08-22 20:36	Replies: 3 Views: 214	ml18244 2011-09-11, 15:59
Deformation gradient for finite strains and shell elements in abaqus Started by muvik, 2011-08-16 08:04	Replies: 1 Views: 175	Jorgen 2011-09-11, 09:03
System Error 2 in run UMAT Started by koohsari, 2010-09-05 23:01	★★★★★ Replies: 4 Views: 1,186	lanfeng 2011-09-09, 03:09
updating displacements in umat Started by tlesicar, 2011-07-12 14:49	Replies: 2 Views: 288	tlesicar 2011-08-26, 04:12
Plane Stress UMAT	Replies: 0	dhurst

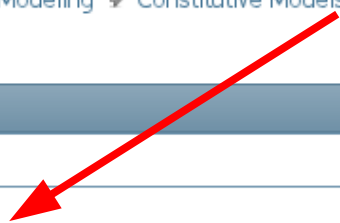
# STEP 4

Forum Finite Element Modeling Constitutive Models Post New Thread

## Post New Thread

Forum: Constitutive Models

**Enter a title and text**



Your Message

Title:

Rich text editor toolbar with icons for font, size, bold, italic, underline, list, link, unlink, image, video, table, code, and other formatting options.

Main text input area for the forum post.



[More]

Post Icons:

