



000: vncashn_100_m_102a5.odt ABAQUS/Standard Version 6.7.4P1 Fri May 23 14:36:32 GMT Standard Time 2008

File: equi@brite
 Procedure: 3D Step Time = 300.0
 Primary von: USDR2
 Enhanced von: USDR2
 Enhancement Scale Factor: +1.000e+00

Rubber in Engineering

Finite Element Analysis

Rubber Consultants has an advanced Finite Element Analysis capability which can be used to solve a wide range of problems including large strain analysis of elastomeric components and heat transfer problems. This capability is enhanced by our practical experience in development of material models, fracture and fatigue of rubber, and our in-house testing facilities which can be used to obtain data for calculating material model parameters. Analysis is carried out using MARC and ABAQUS software.

- FEA can help with:
 - Design of component to meet stiffness specification
 - Improvement in performance of seals
 - Optimisation of cure cycle
 - Improvement in fatigue life
- Results may include:
 - Deformed geometry
 - Global force-deflection plots
 - Natural frequencies and mode shapes
 - Contour plots of stress, strain, strain energy density,
 - Temperature, state of cure etc
- Advice offered on choice of material model
- Tests performed to provide necessary data
- Assistance in determining thermal constants

Both MARC and ABAQUS are general non-linear FE solvers particularly suited to rubber problems, due in part to their advanced contact algorithms and appropriate material models.

For more information please contact:

Dr Alan Muhr
amuhr@rubberconsultants.com
 +44 (0)1992 584966 ext 2082

Dr Julia Gough
jgough@rubberconsultants.com
 +44 (0)1992 584966 ext 2010

