Writing User Subroutines with Abaqus

Introduction

The user subroutines in Abaqus allow the program to be customized for particular applications.
For example user subroutine UMAT in Abaqus/Standard and user subroutine VUMAT in Abaqus/Explicit allow constitutive models to be added to the program, while user subroutine UEL in Abaqus/Standard allows the creation of user-defined elements.

Damage and failure for ductile metals

Introduction

Some user subroutines:

- DLOAD for specifying user-defined loading
- UVARM for defining a user output variable
- URDFIL for reading the results file during analyses
- SIGINI for specifying initial stress fields
- USDFLD for defining field variable dependence

Abaqus/Standard subroutines

CREEP: Define time-dependent, viscoplastic behavior (creep and swelling). **DFLOW**: Define nonuniform pore fluid velocity in a _newlineConsolidation analysis.

DFLUX: Define nonuniform distributed flux in a heat transfer or mass diffusion analysis.

DISP: Specify prescribed boundary conditions.

DLOAD: Specify nonuniform distributed loads.

FILM: Define nonuniform film coefficient and associated sink temperatures for heat transfer analysis.

FLOW: Define nonuniform seepage coefficient and associated sink pore pressure for consolidation analysis.

FRIC: Define frictional behavior for contact surfaces.

GAPCON: Define conductance between contact surfaces or nodes in a fully coupled temperature-displacement analysis or pure heat transfer analysis.

GAPELECTR: Define electrical conductance between surfaces in a coupled thermal-electrical analysis.

Abaqus/Standard subroutines

HARDINI: Define initial equivalent plastic strain and initial backstress tensor.
HETVAL: Provide internal heat generation in heat transfer analysis.
MPC: Define multi-point constraints.
ORIENT: Provide an orientation for defining local material directions or local directions for kinematic coupling constraints or local rigid body directions for inertia relief.
RSURFU: Define a rigid surface.
SDVINI: Define initial solution-dependent state variable fields.
SIGINI: Define an initial stress field.
UAMP: Specify amplitudes.
UANISOHYPER_INV: Define anisotropic hyperelastic material behavior using the invariant formulation.
UANISOHYPER_STRAIN: Define anisotropic hyperelastic material behavior based on Green strain.
UCORR: Define cross-correlation properties for random response loading.
UEL: Define an element.
UEXPAN: Define incremental thermal strains.

Abaqus/Standard subroutines

UEXTERNALDB: Manage user-defined external databases and calculate model-independent history information.

UFIELD: Specify predefined field variables.

UFLUID: Define fluid density and fluid compliance for _newlineHydrostatic fluid elements.

UFLUIDLEAKOFF: Define the fluid leak-off coefficients for pore pressure cohesive elements.

UGENS: Define the mechanical behavior of a shell section.

UHARD: Define the yield surface size and hardening parameters for isotropic plasticity or combined hardening models.

UHYPEL: Define a hypoelastic stress-strain relation.

UHYPER: Define a hyperelastic material.

UINTER: Define surface interaction behavior for contact surfaces.

UMASFL: Specify prescribed mass flow rate conditions for a convection/diffusion heat transfer analysis.

UMAT: Define a material’s mechanical behavior.

UMATHT: Define a material’s thermal behavior.

Abaqus/Standard subroutines

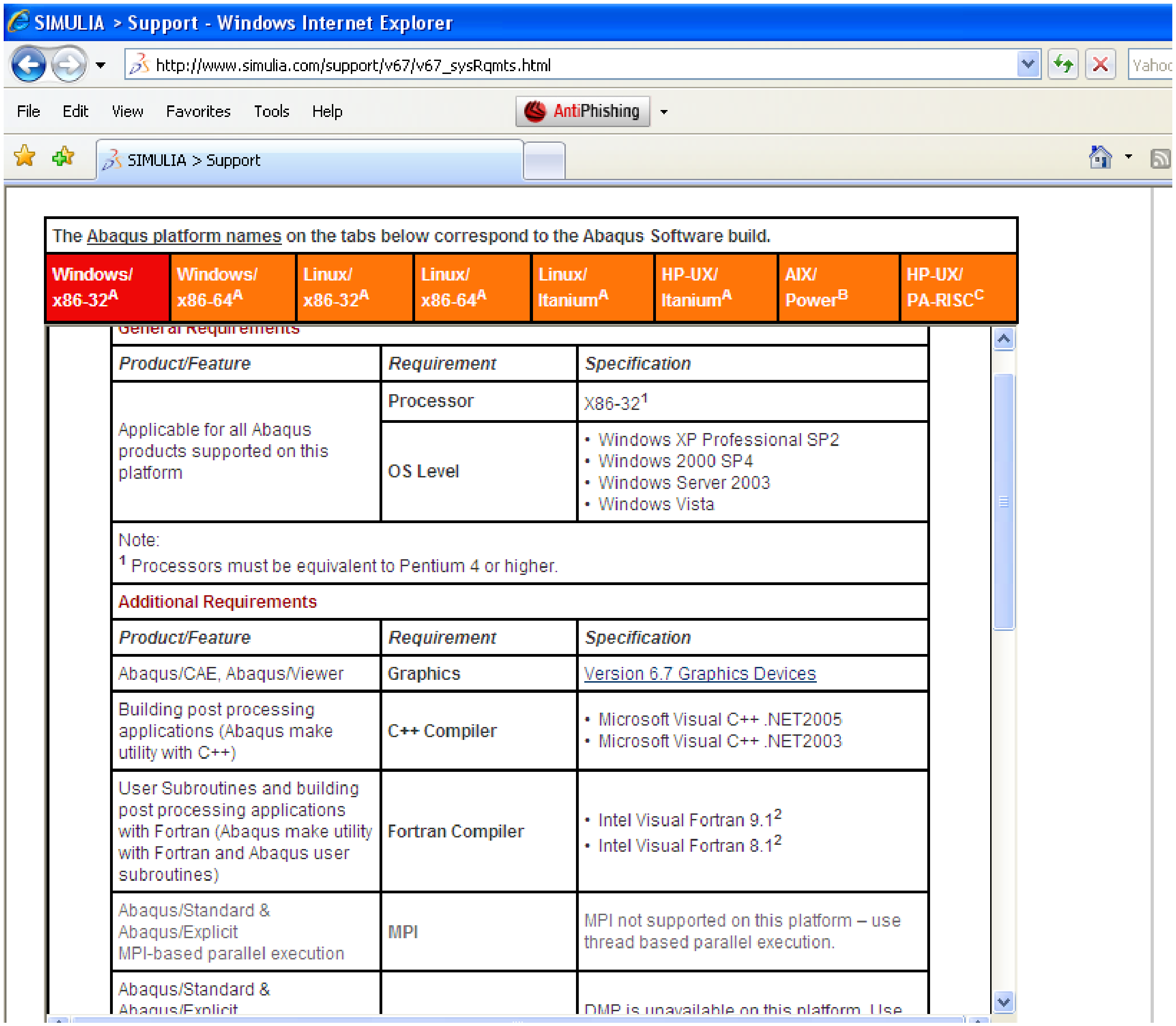
UMAT: Define a material's mechanical behavior.
UMATHT: Define a material's thermal behavior.
UMESHMOTION: Specify mesh motion constraints during adaptive meshing.
UMOTION: Specify motions during cavity radiation heat transfer analysis or steady-state transport analysis.
UMULLINS: Define damage variable for the Mullins effect material model.
UPOREP: Define initial fluid pore pressure.
UPRESS: Specify prescribed equivalent pressure stress conditions.
UPSD: Define the frequency dependence for random response loading.
URDFIL: Read the results file.
USDFLD: Redefine field variables at a material point.
UTEMP: Specify prescribed temperatures.
UTRACLOAD: Specify nonuniform traction loads.
UTRS: Define a reduced time shift function for a viscoelastic material.
UVARM: Generate element output.
UWAVE: Define wave kinematics for an analysis.
VOIDRI: Define initial void ratios.

Abaqus/Explicit subroutines

VDISP: Specify prescribed boundary conditions.
VDLOAD: Specify nonuniform distributed loads.
VFABRIC: Define fabric material behavior.
VFRIC: Define frictional behavior for contact surfaces.
VUAMP: Specify amplitudes.
VUANISOHYPER_INV: Define anisotropic hyperelastic material behavior using the invariant formulation.
VUANISOHYPER_STRAIN: Define anisotropic hyperelastic material behavior based on Green strain.
VUEL: Define an element.
VUFIELD: Specify predefined field variables.
VUHARD: Define the yield surface size and hardening parameters for isotropic plasticity or combined hardening models.
VUINTER: Define the interaction between contact surfaces.
VUMAT: Define material behavior.
VUSDFLD: Redefine field variables at a material point.
VUTRS: Define a reduced time shift function for a viscoelastic material.

Subroutines in Abaqus

- Before you run a job with user subroutine you will need to install C++ and Intel Fortran.



The Abaqus platform names on the tabs below correspond to the Abaqus Software build.

Windows/ x86-32[A]	Windows/ x86-64[A]	Linux/ x86-32[A]	Linux/ x86-64[A]	Linux/ Itanium[A]	HP-UX/ Itanium[A]	AIX/ Power[B]	HP-UX/ PA-RISC[C]

General Requirements

Product/Feature	Requirement	Specification
Applicable for all Abaqus products supported on this platform	Processor	X86-32[1]
	OS Level	• Windows XP Professional SP2 • Windows 2000 SP4 • Windows Server 2003 • Windows Vista

Note:
[1] Processors must be equivalent to Pentium 4 or higher.

Additional Requirements

Product/Feature	Requirement	Specification
Abaqus/CAE, Abaqus/Viewer	Graphics	Version 6.7 Graphics Devices
Building post processing applications (Abaqus make utility with C++)	C++ Compiler	• Microsoft Visual C++ .NET2005 • Microsoft Visual C++ .NET2003
User Subroutines and building post processing applications with Fortran (Abaqus make utility with Fortran and Abaqus user subroutines)	Fortran Compiler	• Intel Visual Fortran 9.1[2] • Intel Visual Fortran 8.1[2]
Abaqus/Standard & Abaqus/Explicit MPI-based parallel execution	MPI	MPI not supported on this platform – use thread based parallel execution.
Abaqus/Standard & Abaqus/Explicit		DMP is unavailable on this platform. Use

Subroutines in Abaqus

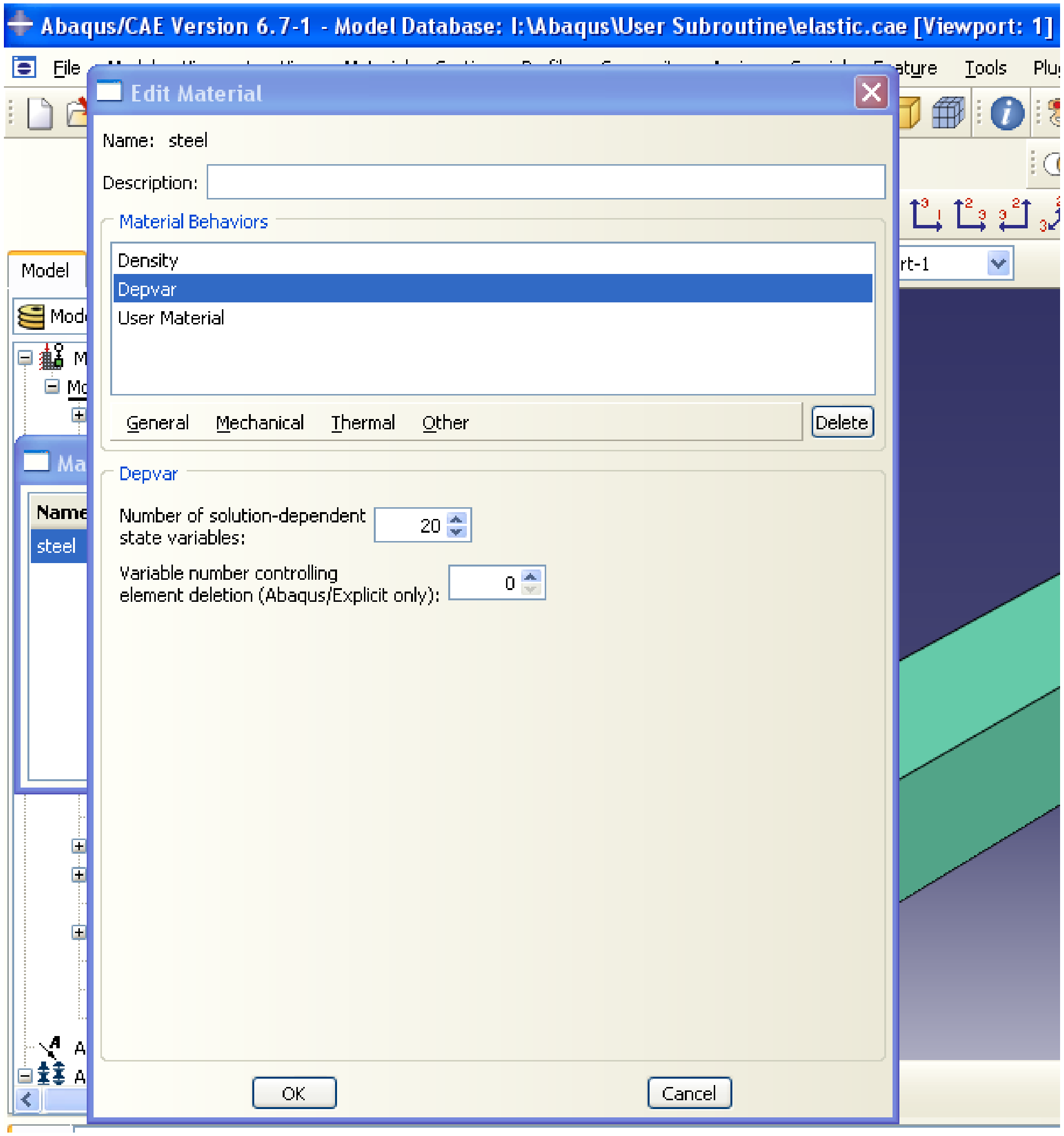
In the ABAQUS CAE environment take the following steps:

- Create a part.
- In the property module, create a material. Use DEPVAR to define number of solution dependent state variables and variable controlling element deletion.

Also define values of material properties that will be interred in the subroutine.

Subroutines in Abaqus

In the ABAQUS CAE environment take the following steps:

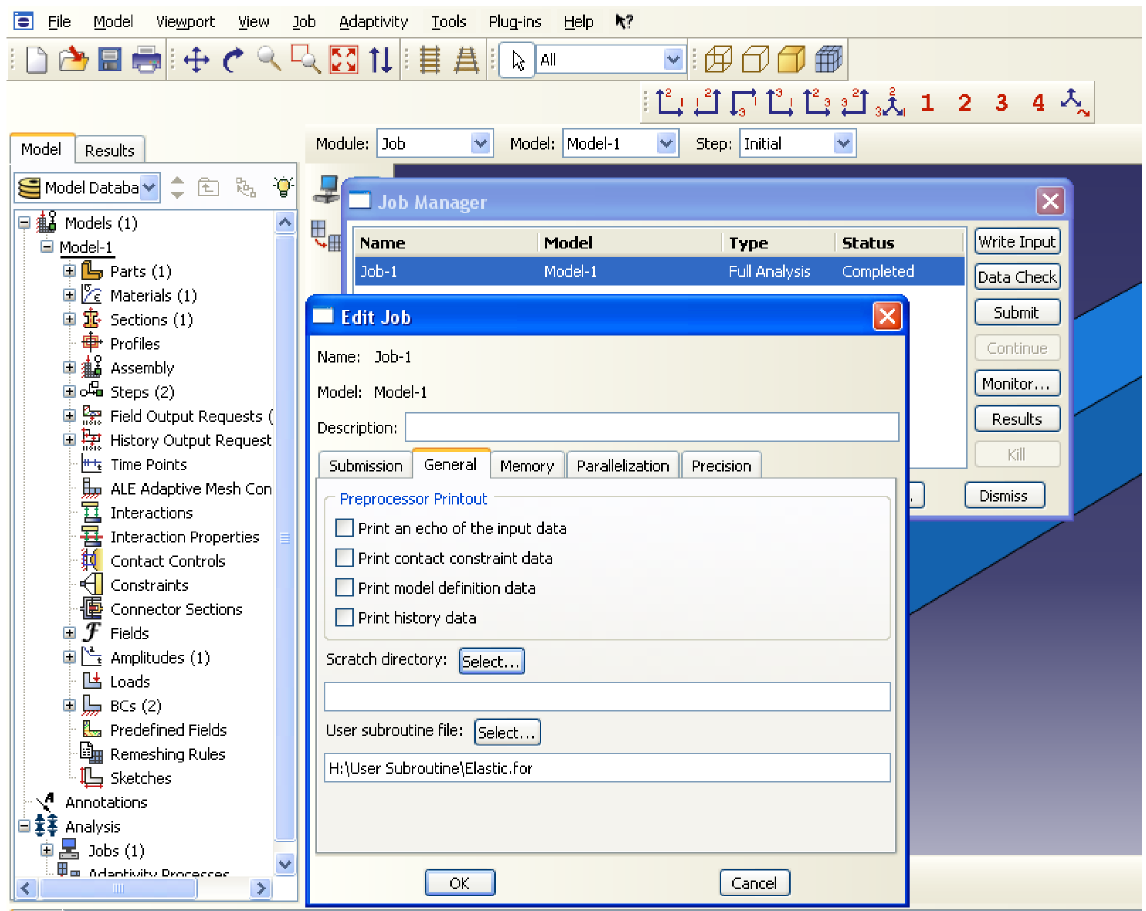


Subroutines in Abaqus

Assembly, step, interaction, load modules are defined the same as ordinary procedure.

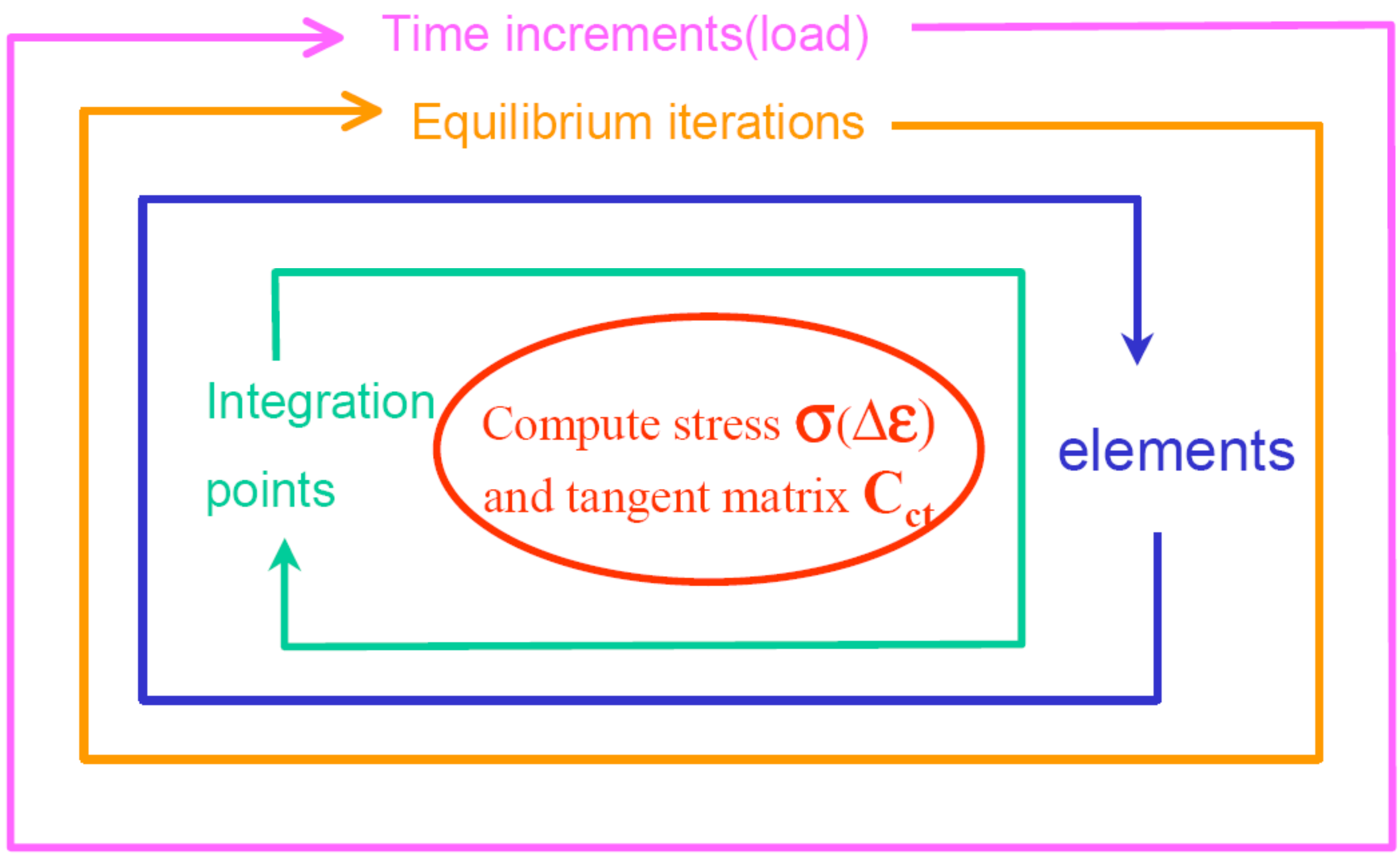
Mesh the part with the required density and element type.

- In the job module, general, define the subroutine you have written.



Non-linear FEA. Implicit method

$$\left[\frac{dF_{int}}{dU}\right] * \{\Delta U\} = \{F_{ext}\} - \{F_{int}\}$$



UMAT & VUMAT

UMAT and VUMAT are the user subroutines for the definition of USER based constitutive models

UMAT ➡ ABAQUS/Standard

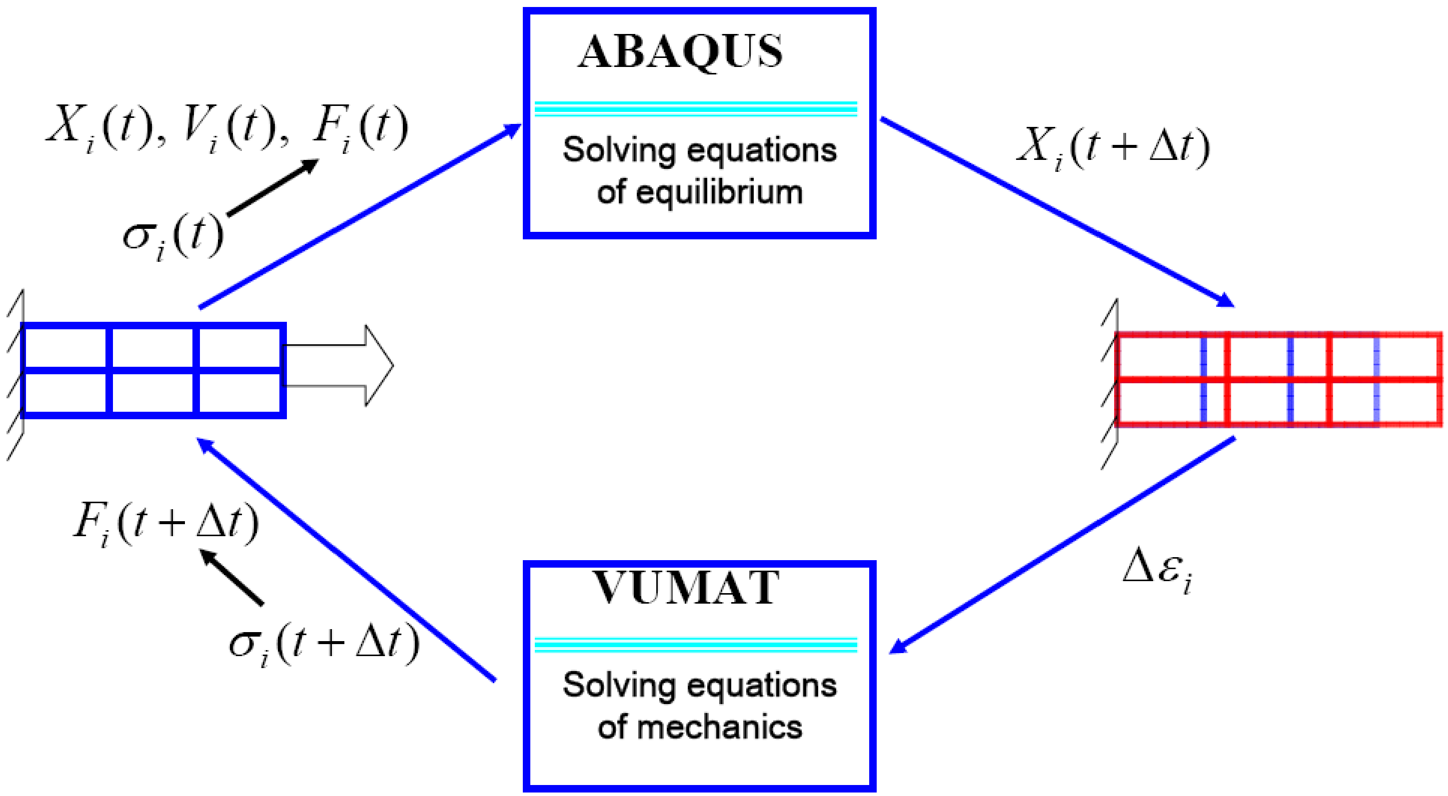
--implicit time integration, must provide "material stiffness matrix" for use in forming the Jacobian of the nonlinear equilibrium equations; $\mathcal{J} = \frac{\partial \Delta \boldsymbol{\sigma}}{\partial \Delta \boldsymbol{\varepsilon}} = \left(\frac{\partial \boldsymbol{\sigma}}{\partial \boldsymbol{\varepsilon}} \right)_{t+\Delta t}$

VUMAT ➡ ABAQUS/Explicit

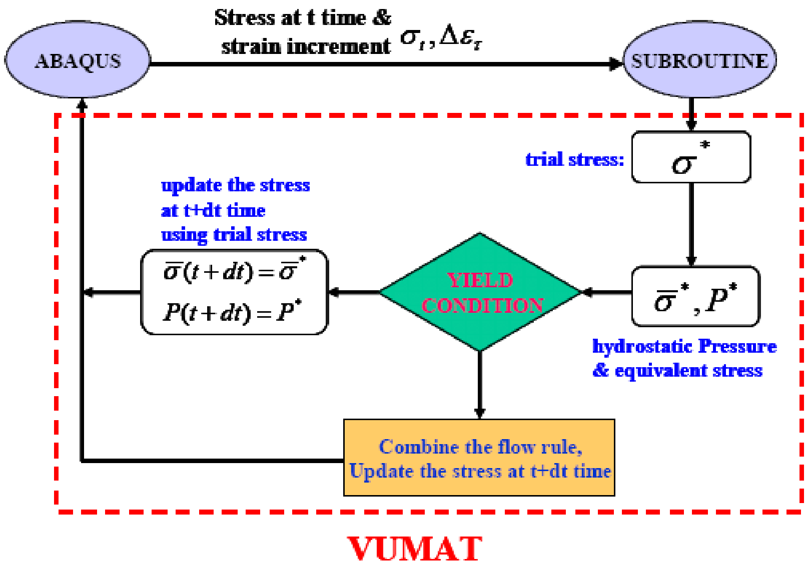
--Explicit time integration, not necessary forming the Jacobian matrix.

Implementation of VUMAT

Where Does VUMAT "Fit" in ABAQUS?



Why Is VUMAT Important?



- ABAQUS constitutive library is extensive BUT;

✓Some models are missing;

✓Some models are not flexible enough;

Example: Gurson Model

*POROUS METAL PLASTICITY, RELATIVE DENSITY=0.95

1. , 1. , 1.

- We can develop our models.

Implementation of VUMAT

Debug and Run a VUMAT

Input File:

```
dpnodev.inp - Notepad
File  Edit  Format  View  Help
*MATERIAL, NAME=METAL
*USER MATERIAL,CONSTANTS=18
9000.D6,0.3D0,120.D6,0.0D0,0.35D0,4,60.0,40,
1.0357D0,-2.172D0,1.3805D0,-4.0185D0,0.5829D0,-2.1954D0,1.
0.598D0,-2.172D0
*DENSITY
1.8E3,
*DEPVAR
3,
*BOUNDARY
1,1,6,0.
2,2,2,0.
4,1,1,0.
*Equation
2
2,1,1.0,3,1,-1.0
*STEP
```

\+

Subroutine (VUMAT):

```
DIGITAL Visual Fortran - [vumat.for]
C  vumat - EXPLICIT
      SUBROUTINE vumat(
C      Read only(unmodifiable)variables-
     1 nblock,ndir,nshr,nstatev,nfieldv,nprops,lanneal,
     2 stepTime,totalTime,dt,cmname,coordMp,charLength,
     3 props,density,strainInc,relSpinInc,
     4 tempOld,stretchOld,defgradOld,fieldOld,
     5 stressOld,stateOld,enerInternOld,enerInelasOld,
     6 tempNew,stretchNew,defgradNew,fieldNew,
C      Write only(modifiable) variables-
     7 stressNew,stateNew,enerInternNew,enerInelasNew)
C
      INCLUDE 'vaba_param.inc'
```

ABAQUS Command:

```
C:\WINDOWS\System32\cmd.exe
Microsoft Windows XP [Version 5.1.2600]
(C) Copyright 1985-2001 Microsoft Corp.

C:\Documents and Settings\wangw>

C:\>abaqus job=test input=dpnodev user=vumat datacheck

C:\>abaqus job=test input=dpnodev user=vumat
```

Implementation of VUMAT