Intralaminar Damage model (IDM)

VUMAT subroutine for Abaqus/Explicit

User guide

Dr. Wei Tan
Queen Mary University London

Prof. Brian Falzon
Royal Melbourne Institute of Technology

Introduction

This documentation is intended as a quick-start guide for users of the IDM subroutine. Detailed information regarding the theory behind this subroutine can be found in other accompanying documentations. This document is version specific. There are four main components in this document:

- Material definition and ordering
- Creating and running the analysis
- Variables outputted by the simulation and their meaning
- Distributed computing information

An example input file for a simple case is provided in the appendix.

IDM traces its origins to a model developed by Falzon and Faggianni for use in the modelling of composite structural behaviour under impact loading. Subsequent developments enable this code to capture the behaviour of composite structures subjected to crush loading. This code introduces a user defined material behaviour within the framework of the Abaqus/Explicit analysis. Below is an overview of the main features of the IDM.

Main features

- Full 3D implementation
- Energy based constitutive relationship
- Non-linear shear
- Load dependent fracture plane
- Mode mixing in transverse loading
- Loading, unloading and load reversal considered
- Damage interaction
- Element deletion

Input

Material Definition

Below is an example of a section of an input file showing how the user material needs to be defined in ABAQUS for the intralaminar damage subroutine to function correctly:

```
 ** MATERIALS
**
*Material, name=HTA
*Density
 1.6e-09,
*Depvar, delete=7
    23,
*User Material, constants=39
145000.,    10300.,    12100.,       0.3,       0.3,       0.3,     5300.,     3950.
     5275.,      2000.,      1600.,        64.,       290.,       0.1,   0.04184,        0.1
   0.04184,        0.1,   0.04184,      5500.,       91.6,       79.9,       0.22,        1.1
       0.7,        0.7,        0.7,      12.86,      12.86,      12.86, 1.3687e+06,    142300.
     5900., 1.3687e+06,    142300.,      5900., 1.3687e+06,    142300.,      5900.
**
```

Although this input file may be created manually, the easiest way is to create the material using the "Create Material" dialogue box in the property module. Inside the "Create material" dialogue box each of the above * keywords can be implemented.

For the above example the inputs to the dialogue box are:

Name:	HTA
Density (required for explicit):	1.6e-9
Number of dependent variables (*depvar):	23
Dependent variable controlling deletion:	7
Number of material properties:	39

Material property ordering

The aim of the code is to provide a predictive power so the provision of correct and accurate material property values is essential. In total, 39 distinct material properties will be required.

Property number	Property	Description
1	E_1	Fibre direction (11) elastic modulus
2	E_2	Transverse direction (22) elastic modulus
3	E_3	Thickness direction (33) elastic modulus
4	$_{12}$	Poisson's ratio
5	$_{23}$	Poisson's ratio
6	$_{13}$	Poisson's ratio
7	G_{12}	Shear strength in the 12 direction
8	G_{23}	Shear strength in the 23 direction
9	G_{13}	Shear strength in the 13 direction

10	X^{T}	Fibre direction tensile strength
11	X^{C}	Fibre direction compressive strength
12	Y^{T}	Transverse direction tensile strength
13	Y^{C}	Transverse direction compressive strength
14	γ_{12}^{0}	Shear damage (legacy)
15	γ_{12}^{max}	Shear damage (legacy)
16	γ_{23}^{0}	Shear damage (legacy)
17	γ_{23}^{max}	Shear damage (legacy)
18	γ_{13}^{0}	Shear damage (legacy)
19	γ_{13}^{max}	Shear damage (legacy)
20	S_{12}	Shear strength in 12 direction
21	G_{1}^{T}	Critical energy release rate (fibre direction, tensile)
22	G_{1}^{C}	Critical energy release rate (fibre direction, compressive)
23	G_{2}^{T}	Critical energy release rate (matrix direction, tensile)
24	G_{2}^{C}	Critical energy release rate (matrix direction, compressive)
25	$G_{12,C}$	Critical energy release rate (shear 12)
26	$G_{23,C}$	Critical energy release rate (shear 23)
27	$G_{13,C}$	Critical energy release rate (shear 13)
28	a_{12}	Shear modulus reduction
29	a_{23}	Shear modulus reduction
30	a_{13}	Shear modulus reduction
31	$c_{1,12}$	Curve-fit parameter for non-linear shear profile (12 direction)
32	$c_{2,12}$	Curve-fit parameter for non-linear shear profile (12 direction)
33	$c_{3,12}$	Curve-fit parameter for non-linear shear profile (12 direction)
34	$c_{1,23}$	Curve-fit parameter for non-linear shear profile (23 direction)
35	$c_{2,23}$	Curve-fit parameter for non-linear shear profile (23 direction)
36	$c_{3,23}$	Curve-fit parameter for non-linear shear profile (23 direction)
37	$c_{1,13}$	Curve-fit parameter for non-linear shear profile (13 direction)
38	$c_{2,13}$	Curve-fit parameter for non-linear shear profile (13 direction)
39	$c_{3,13}$	Curve-fit parameter for non-linear shear profile (13 direction)

Some legacy parameters are kept to enable backwards compatibility.

Analysis

Creating the analysis

It is recommended that Abaqus/CAE be used to create the input files due to the additional requirements of the IDM.

Steps to creating an input file for modelling with IDM subroutine

1. Create part (3D, solid, deformable)
2. Create material by inputting material property according to order described by previous section into a user material (General -> User material in the Edit Material dialogue box)
3. Assign material to section
4. Assign section to part
5. Assign orientation to part
6. Instance part in Assembly
7. Mesh the part, note that mesh must be done on the instance rather than on the part directly.
8. Add analysis step
9. Request outputs
    a. It is recommended that the SDV (dependent variables) be outputted as they contain valuation information such as damage variables, etc. This is done by selecting State/Field/User/Time -> SDV option in the Output variables box under Edit Field Output Request
10. Add initial and boundary conditions
11. Add contacts and constraints
12. Create analysis job
    a. Under the General tab, the User subroutine file entry must be the path pointing to the file containing IDM
    b. Under Precision tab, Abaqus/Explicit precision for both the packager and analysis step must be the same as that specified in IDM (double precision by default)

Running the analysis

There are two ways of running the simulation, either through Abaqus/CAE or through command-line.

Abaqus/CAE

Right click on job created for the model and submit

Command-line

```
abaqus analysis job=job-name input=input-path user=idm-path double=both*
        * default option in IDM
```

Ensure that the free format is selected either within the environment file abaqus_v6.env or within the global environment configuration in Abaqus

Domain level parallelisation offered by Abaqus/Explicit is supported by IDM. This can be invoked either via CAE through Edit job or from the command-line via the “cpus=” attribute.

Output

SDV results

In the previous section the SDVs are requested for output. These values are stored in the .odb file generated by the simulation. Each element has its own set of SDVs that change with time. Below is a list that relates the SDV number to the value being stored.

SDV no.	Value	Description
1	s_{11}	Longitudinal strain
2	s_{22}	Transverse strain
3	s_{33}	Thickness strain
4	s_{12}	12 shear strain
5	s_{23}	23 shear strain
6	s_{13}	13 shear strain
7	Active	Element deletion (0 being a deleted element)
8	d_{11}^{T}	Fibre tensile damage parameter
9	d_{11}^{T}	Fibre tensile damage parameter
10	θ_{fp}	Fracture plane angle
11	d_{mat}	Matrix damage parameter
12	$s_{0,mat}$	Matric damage initiation strain
13	$X_{0,mat}$	Matric damage initiation stress
14	$s_{f,mat}$	Matrix failure strain
15	s_{12}	12 inelastic shear strain
16	s_{23}	23 inelastic shear strain
17	s_{13}	13 inelastic shear strain
18	$l_{c,11}$	Longitudinal direction characteristic length
19	$l_{c,mat}$	Transverse direction characteristic length
20	$\sigma_{E,12}$	12 effective shear stress
21	$\sigma_{E,23}$	23 effective shear stress
22	$\sigma_{E,13}$	13 effective shear stress
23	Bad	Flag for failing element size criteria (1=fail)

SDVs 24 and onwards are used for debugging and checking purposes. SDVs of note are 8,9 and 11 which describe the extent of damage in the material. Furthermore, if SDV 23 has a value of 1 in significant portion of the model, then mesh refinement is recommended.

Miscellaneous information

Usage of IDM on distributed computing

It is requisite that the extension of the vumat.for be changed to vumat.f due to differences in the extension association between Windows

NCI

Job submission line:

    qsub shellscript

Job submission shell script for abaqus 6.11.2, assuming IDM resides in the same folder as the input file

```
#!/bin/bash
#PBS -l vmem=1200mb
#PBS -l walltime=16:50:00
#PBS -l jobfs=4000mb
#PBS -l ncpus=1
#PBS -l software=abaqus
#PBS -wd

module load abaqus/6.11-2

# copy all your input files to the jobfs area.
# In our case this is just inputfile, but you may have several files

cp vumat.f
cp *.inc
cp model.inp $PBS_JOBFS


cd $PBS_JOBFS
#--------------------------------------------------
# set up the environment file
# add any ABAQUS environment details here so that they are
# automatically put into the abaqus environment file.
# See the site userguide section 3 for more parameters for
# controlling abaqus behaviour.
# eg
# explicit_precision=double
# standard recommendation for memory is 90% of available memory.
# --- do not remove these lines ---
cat << EOF > abaqus_v6.env
memory = "$(bc<<<"$PBS_VMEM*90/100") b"
cpus = $PBS_NCPUS
EOF
# end of environment file setup
#--------------------------------------------------
```

```

abaqus analysis job=job-name input=model.inp user=vumat.f double=both scratch=$PBS_JOBFS

# copy all the relevant files back to the submission directory
# at the end of the job

cp * $PBS_O_WORKDIR
```

SGE

Job submission line:

```
        qsub shellscript.sh
```

The environment file abaqus_v6.env is required.

```
#!/bin/bash
#
# Your job name
#$ -N AbaqusJOB
#
# Use current working directory
#$ -cwd
#
# Join stdout and stderr
#$ -j y
#
# pe (Parallel environment) request. Set your number of processors here.
#$ -pe openmpi 1
#
# Run job through bash shell
#$ -S /bin/bash
#
# Mail notifications
#$ -m beas
#$ -M boman
#
# Use this queue
#$ -q lomem.q
## If modules are needed, source modules environment:
#. /etc/profile.d/modules.sh
# Add any modules you might require:
#module add shared sge openmpi/gcc abaqus
. ~/.bashrc
# Define particulars of this run:
ABAQUS_PROGRAM=abaqus
ABAQUS_INPUT=Tension
ABAQUS_JOBNAME=TensionJOB #ABAQUS_${JOB_ID}
ABAQUS_ARGS="-user ../src/idmf/vumat"
ABAQUS_SCRATCH_DIR=/local/ # a verifier! (inutile en explicite)
ln -s ../src/idmf/vumat.for ../src/idmf/vumat.f
#
# To manage abaqus jobs, you need to catch signals
# and use "abaqus terminate" to stop the job
#
exit_gracefully () {
  abaqus terminate job=$ABAQUS_JOBNAME
  echo Abaqus job $ABAQUS_JOBNAME terminated
  exit
}
# The following output will show in the output file. Used for debugging.
echo "Got $NSLOTS processors."
# invoke abaqus in the background on the compute node:
trap exit_gracefully SIGUSR2
# use this for MPI (distributed memory) parallelism
$ABAQUS_PROGRAM cpus=$NSLOTS mp_mode=mpi job=$ABAQUS_JOBNAME
input=$ABAQUS_INPUT scratch=$ABAQUS_SCRATCH_DIR $ABAQUS_ARGS
# Report some useful info
/bin/uname -a
#
# now sleep until lock file disappears
#
sleep 60
while [ -f ${ABAQUS_JOBNAME}.lck ]; do
sleep 5
done
```

Example code

Single element test for fibre compression

```
*Heading
** Job name: c1x Model name: c1x
** Generated by: Abaqus/CAE 6.11-2
*Preprint, echo=NO, model=NO, history=NO, contact=NO
**
** PARTS
**
*Part, name=cube
*End Part
**
**
** ASSEMBLY
**
*Assembly, name=Assembly
**
*Instance, name=cube-1, part=cube
*Node
      1, -0.0500000007, -0.0500000007,  0.100000001
      2, -0.0500000007, 0.0500000007,  0.100000001
      3, -0.0500000007, -0.0500000007,           0.
      4, -0.0500000007, 0.0500000007,           0.
      5, 0.0500000007, -0.0500000007,  0.100000001
      6, 0.0500000007, 0.0500000007,  0.100000001
      7, 0.0500000007, -0.0500000007,           0.
      8, 0.0500000007, 0.0500000007,           0.
*Element, type=C3D8R
1, 5, 6, 8, 7, 1, 2, 4, 3
*Nset, nset=_PickedSet2, internal, generate
 1,  8,  1
*Elset, elset=_PickedSet2, internal
 1,
*Nset, nset=_PickedSet3, internal, generate
 1,  8,  1
*Elset, elset=_PickedSet3, internal
 1,
*Orientation, name=Ori-1
1., 0., 0., 0., 1., 0.
1, 0.
** Section: Section-1
*Solid Section, elset=_PickedSet2, orientation=Ori-1, material=HTA
,
*End Instance
**
*Nset, nset=_PickedSet5, internal, instance=cube-1, generate
 5,  8,  1
*Elset, elset=_PickedSet5, internal, instance=cube-1
 1,
*Nset, nset=_PickedSet7, internal, instance=cube-1, generate
 1,  4,  1
*Elset, elset=_PickedSet7, internal, instance=cube-1
 1,
*Nset, nset=_PickedSet8, internal, instance=cube-1
 3, 4, 7, 8
*Elset, elset=_PickedSet8, internal, instance=cube-1
 1,
*Nset, nset=_PickedSet9, internal, instance=cube-1, generate
 1,  7,  2
*Elset, elset=_PickedSet9, internal, instance=cube-1
 1,
*End Assembly
**
** MATERIALS
**
*Material, name=HTA
*Density
 1.6e-09,
*Depvar, delete=7
    23,
*User Material, constants=39
145000.,     10300.,     12100.,         0.3,         0.3,         0.3,       5300.,       3950.
     5275.,       2000.,       1600.,         64.,        290.,         0.1,  0.04184,         0.1
   0.04184,         0.1,   0.04184,       5500.,        91.6,        79.9,        0.22,         1.1
       0.7,         0.7,         0.7,       12.86,       12.86,       12.86, 1.3687e+06,    142300.
     5900., 1.3687e+06,    142300.,       5900., 1.3687e+06,    142300.,       5900.
**
** BOUNDARY CONDITIONS
**
** Name: BC-3 Type: Displacement/Rotation
*Boundary
_PickedSet7, 1, 1
** Name: BC-4 Type: Displacement/Rotation
*Boundary
_PickedSet8, 3, 3
** Name: BC-5 Type: Displacement/Rotation
*Boundary
_PickedSet9, 2, 2
** ----------------------------------------------------------------
**
** STEP: Step-1
**
*Step, name=Step-1
*Dynamic, Explicit
, 0.001
*Bulk Viscosity
0.06, 1.2
**
** BOUNDARY CONDITIONS
**
```

```
** Name: BC-2 Type: Velocity/Angular velocity
*Boundary, type=VELOCITY
_PickedSet5, 1, 1, -50.
**
** OUTPUT REQUESTS
**
*Restart, write, number interval=1, time marks=NO
**
** FIELD OUTPUT: F-Output-1
**
*Output, field, number interval=200
*Node Output
RF, U, V
*Element Output, directions=YES
EVF, LE, S, SDV, STATUS
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history, variable=PRESELECT
*End Step
```