An Abaqus UEL implementation of the smoothed finite element method : Supplementary file

Pramod Y. Kumbhar [*1], Amrita Francis [†1], N. Swaminathan[1], R. K. Annabattula[‡1] and S. Natarajan[§1]

[1]Department of Mechanical Engineering, Indian Institute of Technology Madras, Chennai, India - 600036

---

1 Introduction

This file provides a brief overview for the readers on how to utilize the Abaqus input file in conjunction with the provided CSFEM User ELement (UEL) subroutine (viz., CSFEM_2D.f and CSFEM_3D.f). It is presumed that users have some basic knowledge of Abaqus.

2 System requirements

- System with Abaqus installation (for current work Abaqus version 6.12-3 is used)
- Intel fortran compiler linked with Abaqus

3 Input file

As this work involves arbitrary polygons/polyhedra, in the input file, all the elements having same number of nodes are clubbed together followed by the properties associated with the elements(c.f. Listing 1 in the paper). The syntax of the **User element* is followed by the number of nodes associated with the elements. For the current work, the input file generation is automated by MATLAB script. One such input generation MATLAB routine is provided in *Inputfile_generation* folder.

---

[*]Equal contribution
[†]Equal contribution
[‡]Corresponding author
[§]Corresponding author

```
*User element, nodes=4, type=U4, properties=3, coordinates=2, variables=7
1,2
*User element, nodes=5, type=U5, properties=3, coordinates=2, variables=7
1,2
*Element, type=U4,ELSET=four
1,1,2,6,8
3,8,6,5,7
*UEL Property, Elset=four
210000000000, 0.3
*Element, type=U5,ELSET=five
2,2,3,4,5,6
*UEL Property, Elset=five
210000000000, 0.3
```

Figure 1: Clubbing of elements in input file for 2D

As shown in Figure 1, the elements having same number of nodes are clubbed together with suitable type. Followed by that the properties associated with the elements are specified. The above figure is the snippet for the 2D case. Likewise, it can be extended for 3D as provided in *Examples_3D* folder. For further details, interested viewers can refer the Abaqus manual.

4 User ELement(UEL) subroutine

For the current work the UELs are developed in FORTRAN90. The subroutines developed for the current work are provided in *Examples_2D* (UEL file name: *CSFEM_2D.f*) and *Examples_3D* (UEL file name : *CSFEM_3D.f*) folder. Users can generate their own input files and utilize the UEL provided here. For demonstration purpose, one '.inp' file for each 2D and 3D cases are provided.

5 Running the simulation

As Abaqus does not permit the inclusion of UEL in an FE model via Abaqus/CAE GUI, one can run a simulation by following command in the Abaqus terminal

abaqus job=<inp file name >user=<fortran file name>.