Lecture 6

Adaptive Meshing and Distortion Control

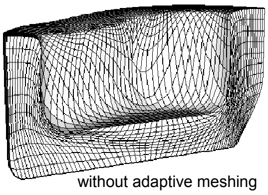
Overview

- Introduction to Adaptive Meshing
- Lagrangian Adaptive Mesh Domains
- Eulerian Adaptive Mesh Domains for Steady-state Analyses
- Output and Diagnostics
- Additional Features of Adaptive Meshing
- Element Distortion Control

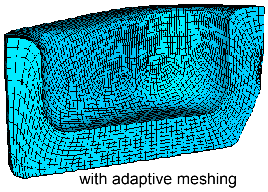
Introduction to Adaptive Meshing

Introduction to Adaptive Meshing

- **Motivation**
  - In many nonlinear simulations the material in the structure or process undergoes very large deformations.
    - These deformations distort the finite element mesh, often to the point where
      - the mesh is unable to provide accurate results
      - or the analysis terminates for numerical reasons.
    - In such simulations it is necessary to use adaptive meshing tools to periodically minimize the distortion in the mesh.



without adaptive meshing

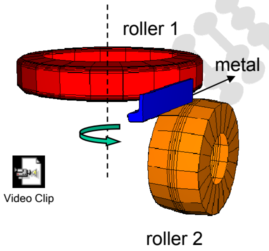


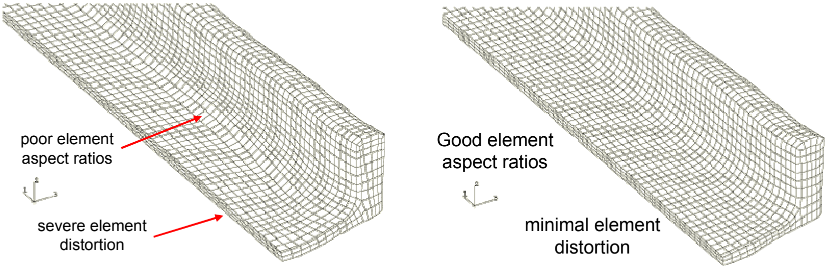
with adaptive meshing

Forming of a steel part

Introduction to Adaptive Meshing

– ABAQUS/Explicit provides a very general and robust adaptive meshing capability for highly nonlinear problems ranging from quasi-static to high-rate dynamic.



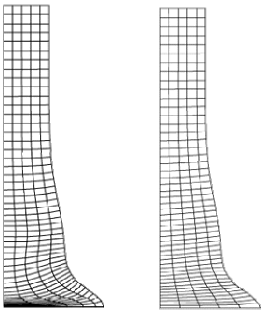


without adaptive meshing **with adaptive meshing**

Transient Rolling analysis

Introduction to Adaptive Meshing

- **Applications**
  - – Can be used as a continuous adaptive meshing tool for transient analysis problems undergoing large deformations, such as:
    - Dynamic impact
    - Penetration
    - Sloshing
    - Forging
  - – Can be used as a solution technique to model steady-state processes, such as
    - Extrusion or rolling
  - – Can be used as a tool to analyze the transient phase in a steady-state process


without adaptive meshing with adaptive meshing

Impact of a copper rod

Introduction to Adaptive Meshing

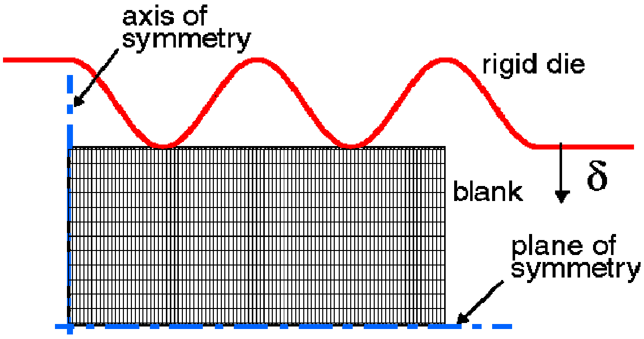
- **Discretization errors**
  - The adaptive meshing algorithm in ABAQUS/Explicit is **not** designed to correct discretization errors in finite element meshes.

Introduction to Adaptive Meshing

- **Pure Lagrangian description**
  - A pure Lagrangian model of a problem is one where the mesh moves with the material.
    - With this approach it is easy to track surfaces and to apply boundary conditions in the problem.
    - The mesh may become very distorted if the material undergoes significant deformation;
      - the quality of the results will deteriorate as the mesh becomes distorted.
  - Most problems in ABAQUS use a pure Lagrangian description.

Introduction to Adaptive Meshing

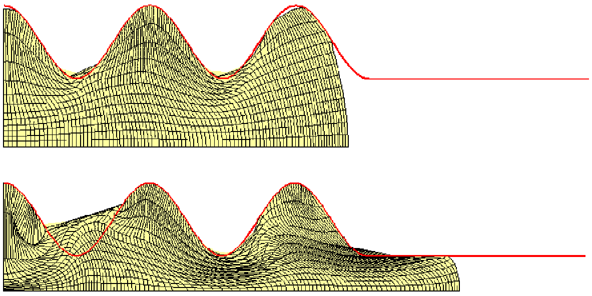
- Some simulations, such as the axisymmetric forging process shown below, cannot be easily performed with a pure Lagrangian description.



Undeformed model

Introduction to Adaptive Meshing

- In this problem, the plastic deformation of the material creates excessive element distortion.


70% of die travel

100% of die travel

Lagrangian simulation deformed shape

- The need for adaptive meshing to reduce mesh distortion during this analysis is clear.

Introduction to Adaptive Meshing

- **Adaptive remeshing is performed in ABAQUS/Explicit using the arbitrary Lagrangian-Eulerian (ALE) method.**
- **The primary characteristics of the adaptive meshing capability are:**
  - A smoother mesh is generated at regular intervals to reduce element distortion and to maintain good element aspect ratios.
  - The same mesh topology is maintained—the number of elements and nodes and their connectivity do not change.
  - It can be used to analyze:
    - Lagrangian (transient) problems in which no material leaves the mesh

      and
    - Eulerian (steady-state) problems in which material flows through the mesh.

Introduction to Adaptive Meshing

- **The adaptive meshing implementation in ABAQUS/Explicit is very general**
  - Adaptive meshing is very cost-effective in an explicit framework.
    - Improving mesh quality increases the stable time increment size, which makes up for the added cost of the adaptive mesh increments.
  - Adaptive meshing is supported for all step-dependent features (contact, mass scaling, etc.).
  - Adaptive meshing can be used with all material models with the exception of the brittle cracking model.
    - However, adaptive meshing cannot occur across material boundaries.
    - Adaptive meshing is not recommended for hyperelastic or hyperfoam materials.
      - See the distortion control section for recommendations on using these materials in analyses with large deformations.

Introduction to Adaptive Meshing

– Once the region of the model that will use adaptive meshing is identified, the algorithm is automatic.

– In ABAQUS/Explicit adaptive meshing is available for all first-order, reduced-integration, continuum elements.

  - Other element types may exist in the model.

Introduction to Adaptive Meshing

- **Relationships between the mesh and underlying material**

  – **Lagrangian** description: nodes move exactly with material points.

    - It is easy to track free surfaces and to apply boundary conditions.
    - The mesh will become distorted with high strain gradients.

  – **Eulerian** description: nodes stay fixed while material flows through the mesh.

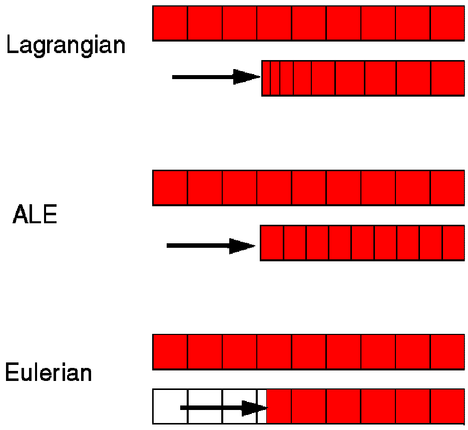
    - It is more difficult to track free surfaces.
    - No mesh distortion because the mesh is fixed.

  – **Arbitrary Lagrangian-Eulerian (ALE)** method: combines the features of pure Lagrangian analysis and pure Eulerian analysis.

    - Mesh motion is constrained to the material motion only where necessary (at free boundaries),
    - Otherwise, material motion and mesh motion are independent.

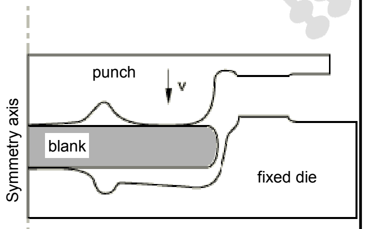
Introduction to Adaptive Meshing

- Motion of mesh and material with various methods:

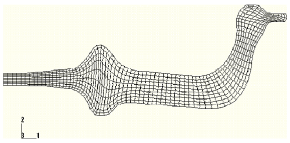


Introduction to Adaptive Meshing

- **Adaptive mesh domains**
  - Adaptive mesh domains define the regions of the model where the mesh can move independently of material deformation.
- **Lagrangian adaptive mesh domains**
  - Lagrangian adaptive mesh domains are usually used to analyze transient or quasi-static problems with large deformations.
    - On the boundary of a Lagrangian domain the mesh will follow the material in the direction normal to the boundary.
    - The mesh covers the same material domain at all times.



undeformed model

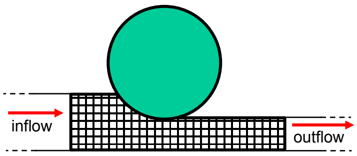


final deformed shape of the blank

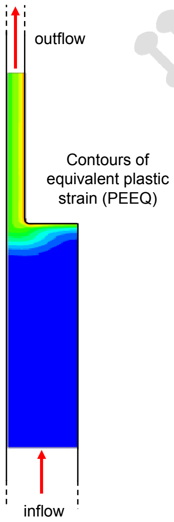
Axisymmetric forging analysis

Introduction to Adaptive Meshing

- **Eulerian adaptive mesh domains**
  - Eulerian adaptive mesh domains are usually used to analyze steady-state processes involving material flow.
    - On certain user-defined boundaries of an Eulerian domain, material can flow into or out of the mesh.



Steady-state rolling



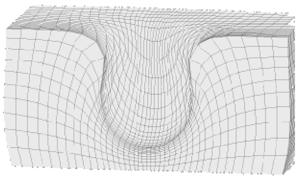
Extrusion analysis

ABAQUS

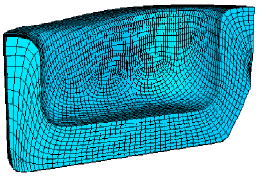
Lagrangian Adaptive Mesh Domains

Lagrangian Adaptive Mesh Domains

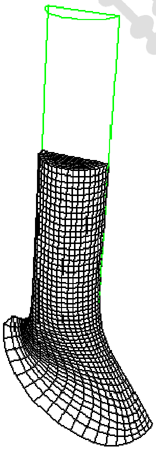
- **With a Lagrangian adaptive mesh domain the mesh represents the same material domain at all times.**
  - On the boundary of a Lagrangian domain the mesh will follow the material in the direction normal to the boundary.
  - This technique is often used to analyze transient or quasi-static problems with large deformations.



Crushable foam indentation



Bulk metal forming



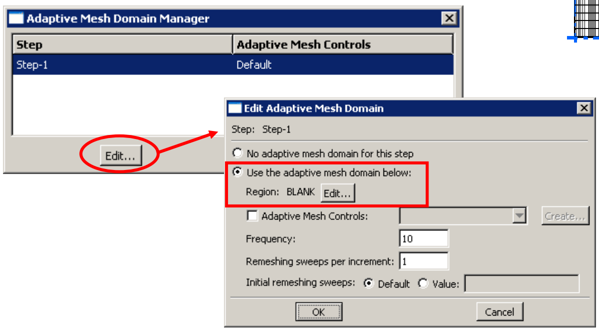
High speed impact

Lagrangian Adaptive Mesh Domains

- **Example: Axisymmetric forging problem with adaptive meshing**

```
*ADAPTIVE MESH,ELSET=BLANK
```

From the main menu bar of the Step module, select **Other → Adaptive Mesh Domain → Manager**



Step	Adaptive Mesh Controls
Step-1	Default

Step: Step-1

○ No adaptive mesh domain for this step

● Use the adaptive mesh domain below:

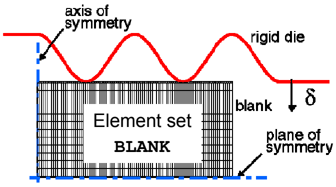
Region: BLANK

☐ Adaptive Mesh Controls:

Frequency: 10

Remeshing sweeps per increment: 1

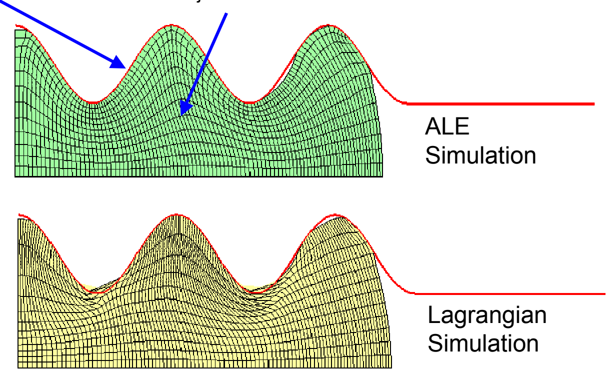
Initial remeshing sweeps: ● Default ○ Value:



Undeformed model

Lagrangian Adaptive Mesh Domains

- **Example (cont'd) : Axisymmetric forging problem with adaptive meshing**

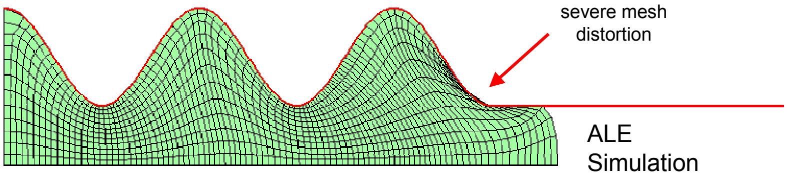


Video Clip

Deformed meshes at 70% of die travel

Lagrangian Adaptive Mesh Domains

- **Example (cont'd) : Axisymmetric forging problem with adaptive meshing**
  - The default adaptive meshing behavior is not effective enough to prevent mesh distortion towards the end of the forging analysis.
    - The default adaptive meshing options are indented for:
      - low- to moderate-rate dynamic problems
      - quasi-static process simulations undergoing moderate deformation.
    - This analysis ends prematurely with an excessive element distortion error.



Deformed mesh at end of analysis (91% of die travel)

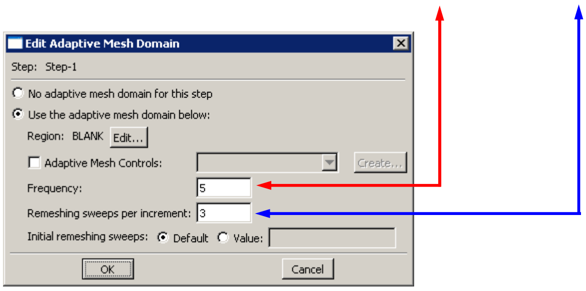
Lagrangian Adaptive Mesh Domains

- **Frequency of adaptive meshing**
  - In most cases the frequency of adaptive meshing is the parameter that most affects the mesh quality and the computational efficiency of adaptive meshing.
    - The default for Lagrangian (transient) problems, is for an adaptive mesh increment to be performed after every 10 “explicit” increments.
    - If the entire model acts as the adaptive mesh domain, each adaptive meshing increment costs about the same as 3–5 “explicit” increments.
  - In an adaptive meshing increment, ABAQUS/Explicit creates a new smoother mesh by sweeping iteratively over the adaptive mesh domain.
    - During each sweep, nodes are adjusted slightly to reduce element distortion.
    - By default, 1 mesh sweep is performed per adaptive mesh increment.

Lagrangian Adaptive Mesh Domains

- **Example (cont’d) : Axisymmetric forging problem with adaptive meshing**
  - Increase the adaptive mesh frequency for the forging example so that:
    - adaptive meshing is performed every 5 increments and
    - 3 mesh sweeps are performed every adaptive mesh increment.

```
*ADAPTIVE MESH,ELSET=BLANK, FREQUENCY=5, MESH SWEEPS=3
```



Edit Adaptive Mesh Domain

Step: Step-1

○ No adaptive mesh domain for this step

● Use the adaptive mesh domain below:

Region: BLANK

☐ Adaptive Mesh Controls:

Frequency: 5

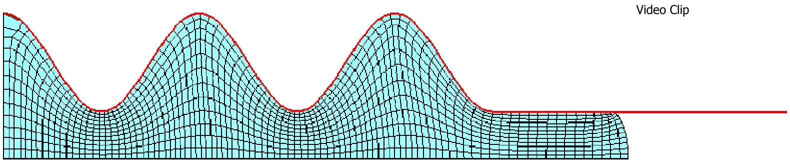
Remeshing sweeps per increment: 3

Initial remeshing sweeps: ● Default ○ Value:

Lagrangian Adaptive Mesh Domains

- **Example (cont'd) : Axisymmetric forging problem with adaptive meshing**
  - With the increased adaptive mesh frequency and more mesh sweeps per adaptive mesh increment, the mesh quality is improved.

Video Clip



Deformed mesh at end of analyses (100% of die travel)

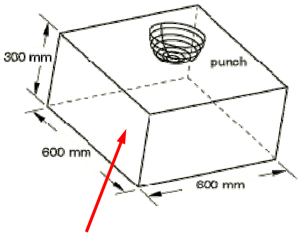
Lagrangian Adaptive Mesh Domains

- **Adaptivity with graded meshes**
  - The two objectives of ABAQUS/Explicit's adaptive meshing algorithm are:
    - to reduce the distortion and
    - to improve the aspect ratios of the elements in the adaptive mesh domain.
  - There are many problems where it is desirable to maintain a graded mesh throughout the analysis.
    - The adaptive meshing capability in ABAQUS/Explicit allows the user to specify that the original mesh gradation should be maintained.

Lagrangian Adaptive Mesh Domains

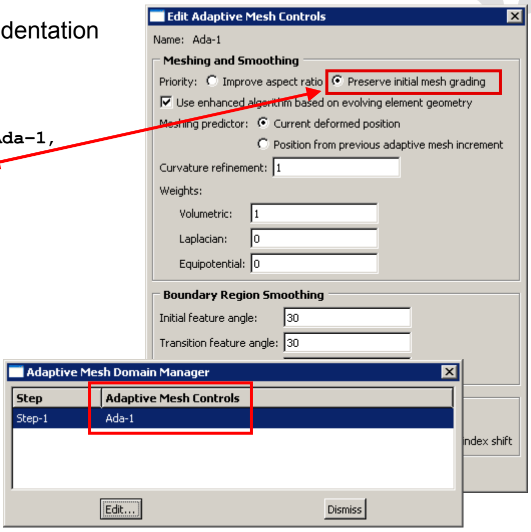
– Example: Crushable foam indentation

```
*ADAPTIVE MESH, ELSET=foam,
 CONTROLS=Ada-1
*ADAPTIVE MESH CONTROLS, NAME=Ada-1,
 SMOOTHING OBJECTIVE=GRADED
```



element set **`foam`**

crushable foam material


Edit Adaptive Mesh Controls

Name: Ada-1

Meshing and Smoothing

Priority: ○ Improve aspect ratio ● Preserve initial mesh grading

☑ Use enhanced algorithm based on evolving element geometry

Meshing predictor: ● Current deformed position
○ Position from previous adaptive mesh increment

Curvature refinement: 1

Weights:

Volumetric: 1

Laplacian: 0

Equipotential: 0

Boundary Region Smoothing

Initial feature angle: 30

Transition feature angle: 30

Adaptive Mesh Domain Manager

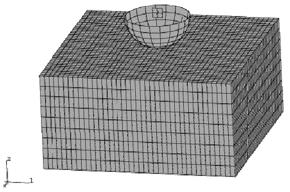
Step	Adaptive Mesh Controls
Step-1	Ada-1

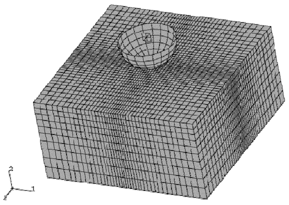
Lagrangian Adaptive Mesh Domains

– Example (cont'd): Crushable foam indentation

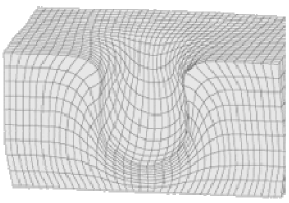
regular mesh

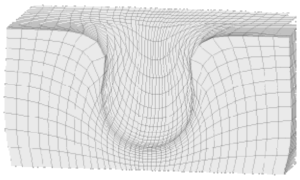
graded mesh





Undeformed mesh





Deformed half mesh
at 100% of die travel

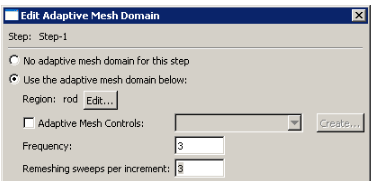
Lagrangian Adaptive Mesh Domains

- **Example: High Speed Rod Impact**
  - Three variations of the rod impact analysis are performed.

① Pure Lagrangian analysis (i.e. no adaptive meshing)

② Lagrangian adaptive meshing analysis with default region boundaries:
  - Nodes move with the material in the direction normal to the material's surface.
  - Nodes are allowed to adapt (adjust their position) tangent to the free surface.

```
*ADAPTIVE MESH, ELSET=rod,
  FREQUENCY=3, SWEEPS=3
```



Step: Step-1

○ No adaptive mesh domain for this step

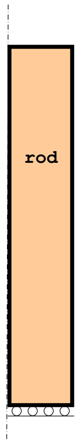
● Use the adaptive mesh domain below:

Region: rod

☐ Adaptive Mesh Controls:

Frequency: 3

Remeshing sweeps per increment: 3



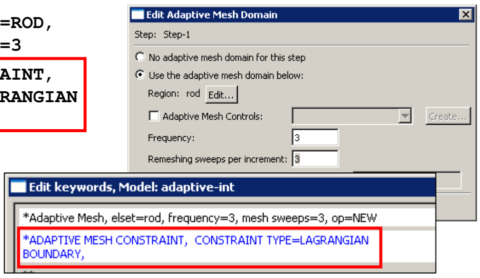
Axisymmetric rod

Lagrangian Adaptive Mesh Domains

- **Example (cont'd): High Speed Rod Impact**

③ Lagrangian adaptive meshing analysis with Lagrangian mesh constraints on the mesh exterior.
  - Nodes move with the material (nonadaptive) on the mesh exterior.
  - Nodes are allowed to adapt (adjust their position) within the rod interior.

```
*ADAPTIVE MESH, ELSET=ROD,
  FREQUENCY=3, SWEEPS=3
*ADAPTIVE MESH CONSTRAINT,
  CONSTRAINT TYPE=LAGRANGIAN
 BOUNDARY,
```



Step: Step-1

○ No adaptive mesh domain for this step

● Use the adaptive mesh domain below:

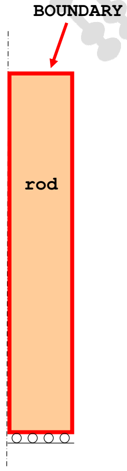
Region: rod

☐ Adaptive Mesh Controls:

Frequency: 3

Remeshing sweeps per increment: 3

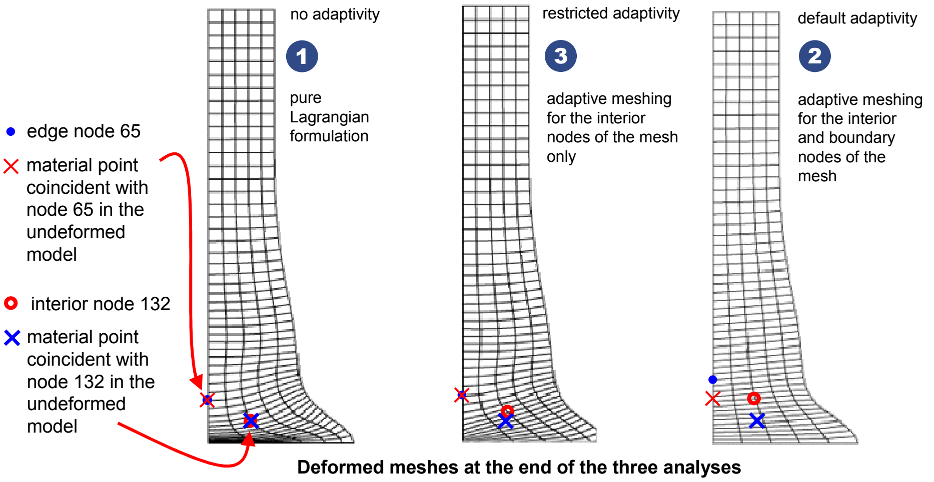
```
*Adaptive Mesh, elset=rod, frequency=3, mesh sweeps=3, op=NEW
*ADAPTIVE MESH CONSTRAINT,  CONSTRAINT TYPE=LAGRANGIAN
BOUNDARY,
```



Axisymmetric rod

Lagrangian Adaptive Mesh Domains

- **Example (cont'd): High Speed Rod Impact**



no adaptivity

①

pure Lagrangian formulation

restricted adaptivity

③

adaptive meshing for the interior nodes of the mesh only

default adaptivity

②

adaptive meshing for the interior and boundary nodes of the mesh

● edge node 65

× material point coincident with node 65 in the undeformed model

○ interior node 132

× material point coincident with node 132 in the undeformed model

Deformed meshes at the end of the three analyses

Lagrangian Adaptive Mesh Domains

- **Timings and peak equivalent plastic strain**

Type of analysis	**CPU time** (Normalized)	**Number of increments** (Normalized)	**Peak equivalent plastic strain**
Pure Lagrangian	1.00	1000	3.00
ALE for interior nodes	0.83	749	2.99
ALE for interior and boundary nodes	0.44	302	2.78

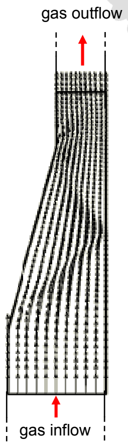
– This example shows that while the cost per increment increases as more nodes are adjusted during adaptive meshing, the overall cost decreases because fewer increments are needed.

  - ABAQUS/Explicit can use larger time increments in the adaptive meshing simulations because the element distortion is minimized (elements remain well-shaped).

Eulerian Adaptive Mesh Domains for Steady-state Analyses

Eulerian Adaptive Mesh Domains for Steady-state Analyses

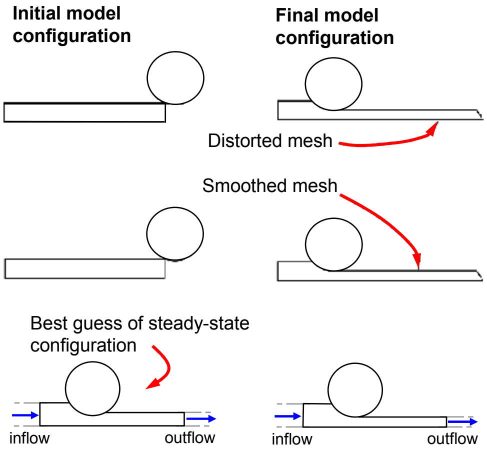
- **An Eulerian description of a problem is one in which the material moves through the mesh—the mesh defines a control volume for the problem.**
  - The adaptive meshing capability in ABAQUS/Explicit can be used to perform simulations of steady-state processes with an Eulerian description.
  - The steady-state conditions for many metal forming processes can be analyzed more readily with an Eulerian description, such as:
    - Rolling
    - Extrusion
    - Drawing
  - Other flow problems can be analyzed, such as a shock wave in a gas traveling with constant velocity through a two-dimensional obstructed channel.



velocity resultant at an intermediate time during shock wave analysis

Eulerian Adaptive Mesh Domains for Steady-state Analyses

- Various methods for modeling a rolling process
  - Pure Lagrangian (transient analysis)
  - Adaptive Meshing with Lagrangian Boundaries (transient analysis)
  - Adaptive Meshing with Inlet and Outlet Eulerian Boundaries (steady-state analysis)

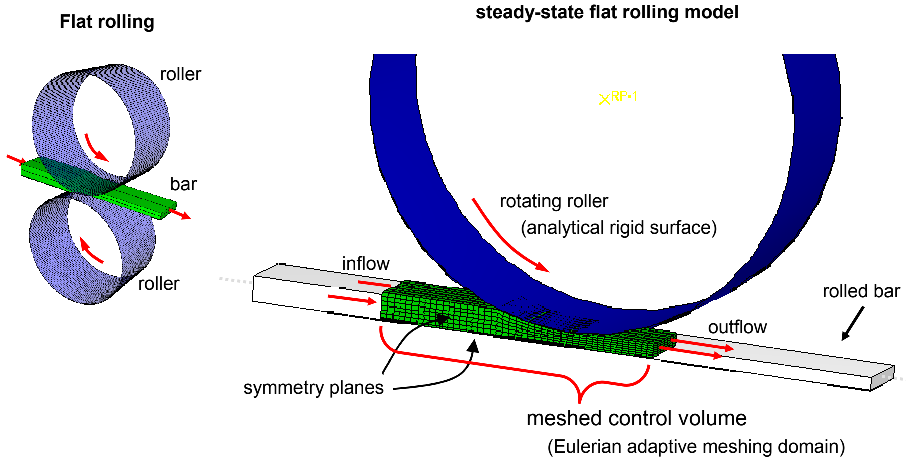


Eulerian Adaptive Mesh Domains for Steady-state Analyses

- **The definition of an Eulerian adaptive meshing problem requires careful consideration.**
  - The following items need to be specified for Eulerian problems:
    - Adaptive mesh domain
      - This is the meshed region that serves as the problem control volume.
      - The mesh must be a reasonable approximation of the steady-state configuration.
    - Inflow and outflow surfaces
    - Mesh constraints
      - To fix the mesh in space so it does not move with the underlying material
    - Material constraints
      - To control material behavior at boundaries, such as the inflow surface.
        - Material conditions at the outflow boundary are typically part of the solution.

Eulerian Adaptive Mesh Domains for Steady-state Analyses

- **Example: Steady-state flat rolling simulation**



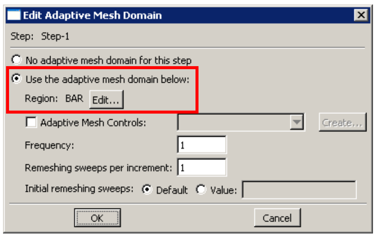
Eulerian Adaptive Mesh Domains for Steady-state Analyses

- **Example (cont'd): Steady-state flat rolling simulation**

① Define the adaptive mesh domain.

  - This adaptive mesh domain definition is the same for Eulerian and Lagrangian adaptive meshing analyses.
  - The default adaptive meshing frequency in Eulerian analyses is one.
    - i.e. adaptive meshing is performed every increment

```
*ADAPTIVE MESH,ELSET=BAR
```



Step: Step-1

○ No adaptive mesh domain for this step

● Use the adaptive mesh domain below:

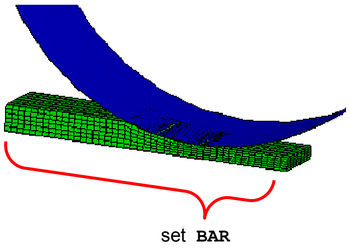
Region: BAR Edit...

☐ Adaptive Mesh Controls: Create...

Frequency: 1

Remeshing sweeps per increment: 1

Initial remeshing sweeps: ● Default ○ Value:

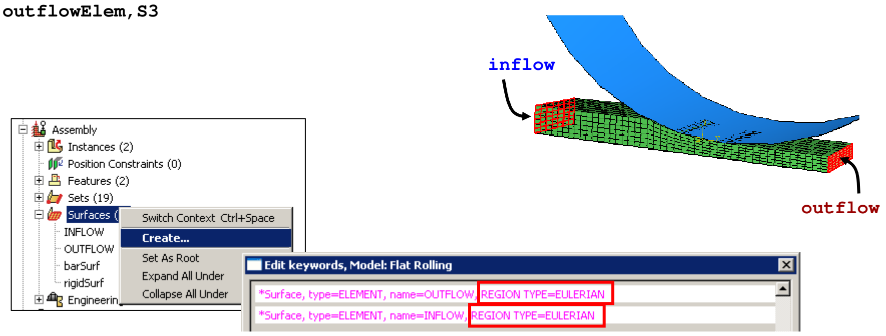


Eulerian Adaptive Mesh Domains for Steady-state Analyses

- **Example (cont'd): Steady-state flat rolling simulation**

  ② Define the Eulerian inflow and outflow surfaces.

```
*SURFACE, NAME=inflow, REGION TYPE=EULERIAN
inflowElem,S5
*SURFACE, NAME=outflow, REGION TYPE=EULERIAN
outflowElem,S3
```



Edit keywords, Model: Flat Rolling

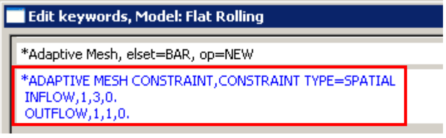
```
*Surface, type=ELEMENT, name=OUTFLOW, REGION TYPE=EULERIAN
*Surface, type=ELEMENT, name=INFLOW, REGION TYPE=EULERIAN
```

Eulerian Adaptive Mesh Domains for Steady-state Analyses

- **Example (cont'd): Steady-state flat rolling simulation**

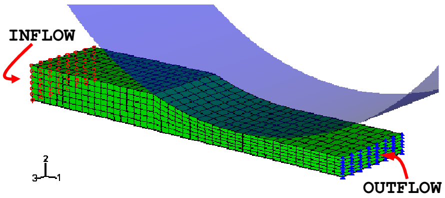
  ③ Constrain the **mesh** motion at the Eulerian surfaces defined in step ②.

  - Constrain the nodes of the inflow and outflow surfaces in the direction normal to the material flow.
    - – Creates a stationary control volume with respect to the material.
  - Constrain the inflow surface in the directions tangent to the flow.
    - – In this example the shape of the inflow boundary is known.



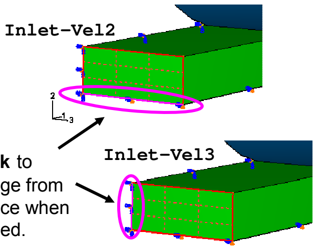
Edit keywords, Model: Flat Rolling

```
*Adaptive Mesh, elset=BAR, op=NEW
*ADAPTIVE MESH CONSTRAINT,CONSTRAINT TYPE=SPATIAL
 INFLOW,1,3,0.
 OUTFLOW,1,1,0.
```



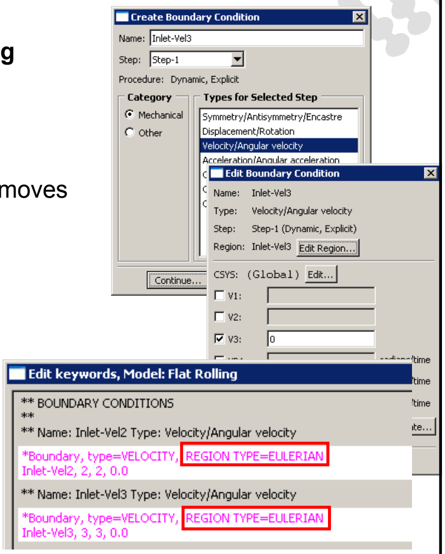
Eulerian Adaptive Mesh Domains for Steady-state Analyses

- **Example (cont'd): Steady-state flat rolling simulation**
  - ④ Constrain the **material** motion at the Eulerian surfaces.
    - At the inflow surface the material moves only normal to the surface.



[Ctrl]+Click to exclude edge from selected face when set is created.

Sets for boundary condition



Name: Inlet-Vel3
Step: Step-1
Procedure: Dynamic, Explicit
● Mechanical
○ Other

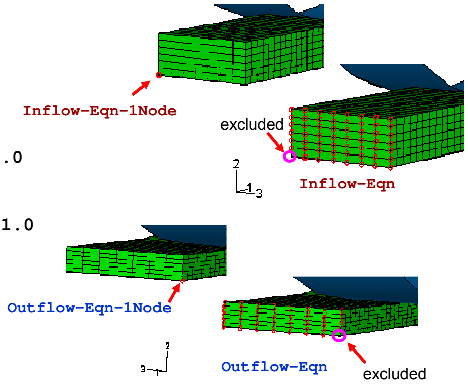
Name: Inlet-Vel3
Type: Velocity/Angular velocity
Step: Step-1 (Dynamic, Explicit)
Region: Inlet-Vel3
CSYS: (Global)
☐ V1:
☐ V2:
☑ V3: 0

```
** BOUNDARY CONDITIONS
**
** Name: Inlet-Vel2 Type: Velocity/Angular velocity
*Boundary, type=VELOCITY, REGION TYPE=EULERIAN
Inlet-Vel2, 2, 2, 0.0
** Name: Inlet-Vel3 Type: Velocity/Angular velocity
*Boundary, type=VELOCITY, REGION TYPE=EULERIAN
Inlet-Vel3, 3, 3, 0.0
```

Eulerian Adaptive Mesh Domains for Steady-state Analyses

- **Example (cont'd): Steady-state flat rolling simulation**
  - ④ (cont'd) Constrain the **material** motion at the Eulerian surfaces.
    - At inflow and outflow boundaries the material velocity is assumed to be uniform.

```
*EQUATION
2,
Inflow-Eqn,1,1.0, Inflow-Eqn-1Node,1,-1.0
*EQUATION
2,
Outflow-Eqn,1,1.0,Outflow-Eqn-1Node,1,-1.0
```

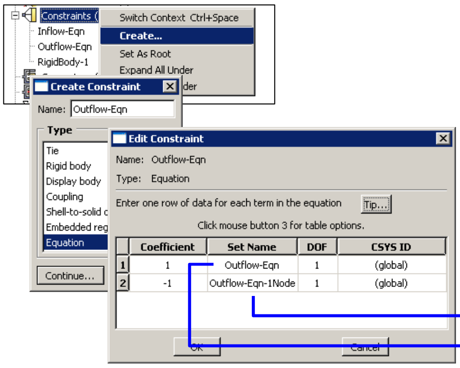


Eulerian Adaptive Mesh Domains for Steady-state Analyses

- **Example (cont'd): Steady-state flat rolling simulation**

  ④ (cont'd) Constrain the **material** motion at the Eulerian surfaces.

  - At inflow and outflow boundaries the material velocity is assumed to be uniform.



Create Constraint — Name: Outflow-Eqn

Edit Constraint

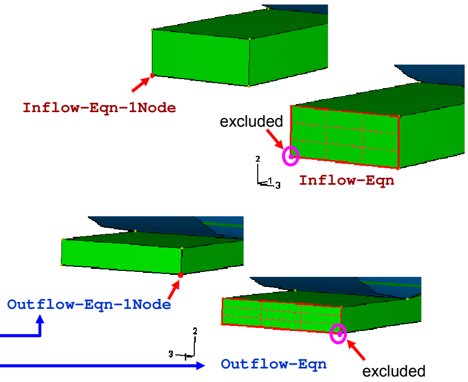
Name: Outflow-Eqn

Type: Equation

Enter one row of data for each term in the equation

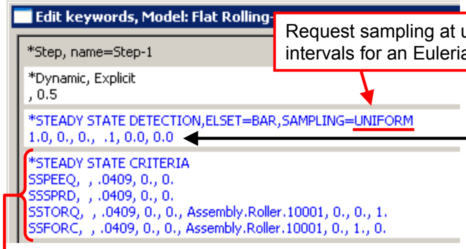
Click mouse button 3 for table options.

	Coefficient	Set Name	DOF	CSYS ID
1	1	Outflow-Eqn	1	(global)
2	-1	Outflow-Eqn-1Node	1	(global)



Eulerian Adaptive Mesh Domains for Steady-state Analyses

- **Example (cont'd): Steady-state flat rolling simulation**
  - Steady-state detection can be used to terminate the ABAQUS/Explicit analysis when specified steady-state criteria are met.
    - This feature is available for uni-directional processes.
    - This feature is not unique to adaptive meshing analyses.



```
*Step, name=Step-1
*Dynamic, Explicit
, 0.5
*STEADY STATE DETECTION,ELSET=BAR,SAMPLING=UNIFORM
1.0, 0., 0.,  .1, 0.0, 0.0
*STEADY STATE CRITERIA
SSPEEQ,  , .0409, 0., 0.
SSSPRD,  , .0409, 0., 0.
SSTORQ,  , .0409, 0., 0., Assembly.Roller.10001, 0., 0., 1.
SSFORC,  , .0409, 0., 0., Assembly.Roller.10001, 0., 1., 0.
```

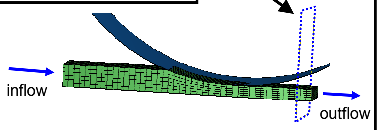
Request sampling at uniform intervals for an Eulerian analysis.

Cutting plane

Analysis terminates when the steady state is detected at the cutting plane.

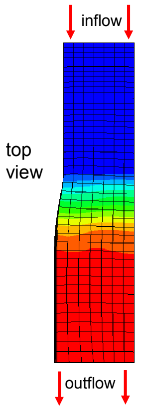
steady-state criteria definitions

Only when all of the criteria specified have been satisfied will the analysis be considered to have reached steady state.

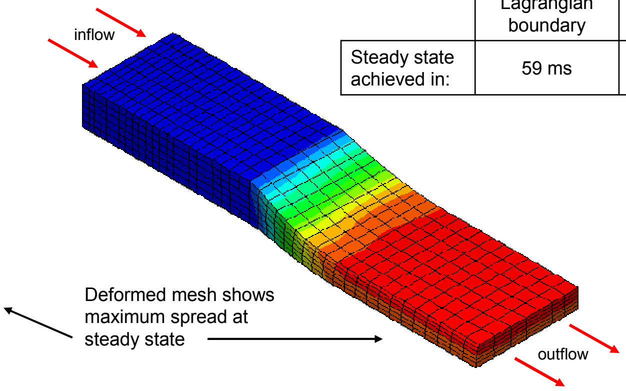


Eulerian Adaptive Mesh Domains for Steady-state Analyses

- **Example (cont'd): Steady-state flat rolling simulation**



	ALE with Lagrangian boundary	ALE with Eulerian boundary
Steady state achieved in:	59 ms	28 ms



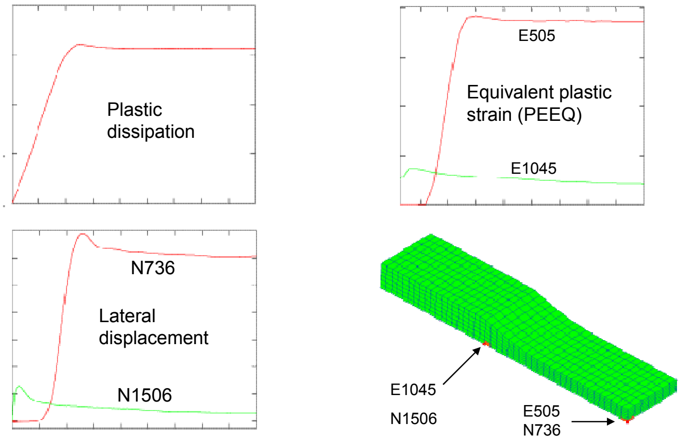
Deformed mesh shows maximum spread at steady state

Contours of equivalent plastic strain in steady-state conditions

---

Eulerian Adaptive Mesh Domains for Steady-state Analyses

- **Example (cont'd): Steady-state flat rolling simulation**



Verifying that a steady-state solution is achieved

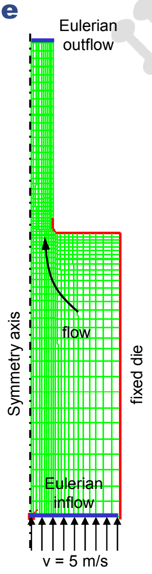
Eulerian Adaptive Mesh Domains for Steady-state Analyses

- **Example: Axisymmetric extrusion analysis**
  - The adaptive meshing definition for this extrusion example is similar to the previous rolling example.
    - Extrusion is different from rolling in that the material is driven by a velocity condition specified at the inflow Eulerian boundary.

  ① All the elements of the bar are included in the adaptive mesh domain.

  ② The Eulerian inflow and outflow surfaces are specified.

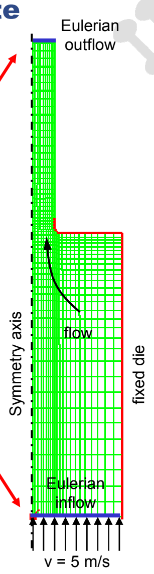
  ③ Adaptive mesh constraints are applied to constrain the mesh at the inflow and outflow surfaces.
    - Both surfaces are constrained vertically.
    - The inflow surface is also constrained horizontally.



Eulerian Adaptive Mesh Domains for Steady-state Analyses

- **Example (cont'd): Axisymmetric extrusion analysis**

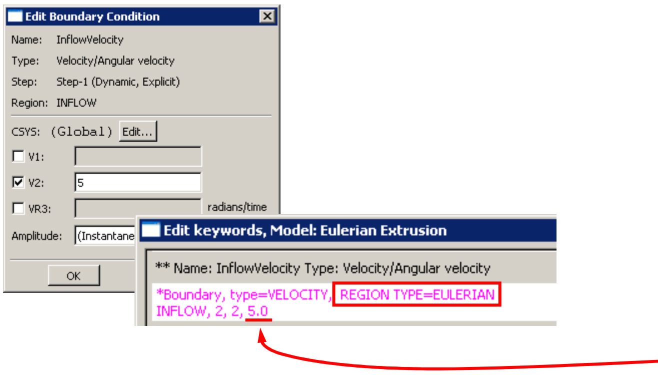
  ④ Constrain the motion of the material passing trough the Eulerian surfaces.
    - An equation constrains the material at the outflow boundary to have uniform velocity.
    - An Eulerian boundary condition prevents the material crossing the inflow surface from moving tangent to the surface.



Eulerian Adaptive Mesh Domains for Steady-state Analyses

- **Example (cont'd): Axisymmetric extrusion analysis**

⑤ Force the material through the mesh with a boundary condition at the inflow surface.



Edit Boundary Condition

Name: InflowVelocity

Type: Velocity/Angular velocity

Step: Step-1 (Dynamic, Explicit)

Region: INFLOW

CSYS: (Global)

☐ V1:

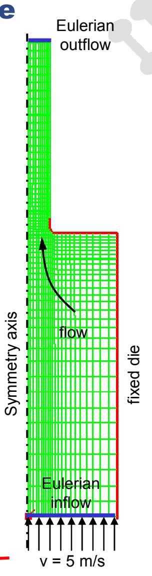
☑ V2: 5

☐ VR3: radians/time

Amplitude: (Instantane

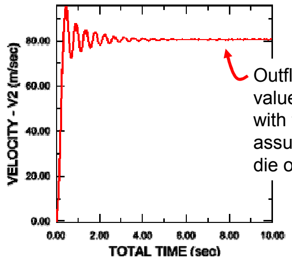
Edit keywords, Model: Eulerian Extrusion

```
** Name: InflowVelocity Type: Velocity/Angular velocity
*Boundary, type=VELOCITY, REGION TYPE=EULERIAN
INFLOW, 2, 2, 5.0
```



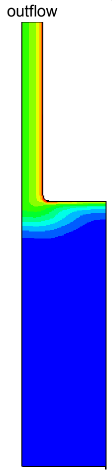
Eulerian Adaptive Mesh Domains for Steady-state Analyses

- **Example (cont'd): Axisymmetric extrusion analysis**
  - The mesh undergoes very little change from the beginning to the end of the analysis because of the accurate initial guess made for the steady-state domain shape.



Outflow velocity reaches a steady value of ~80 m/s, which is consistent with the incompressible material assumption and the 1/16 ratio of the die opening to the billet size.

Outflow velocity

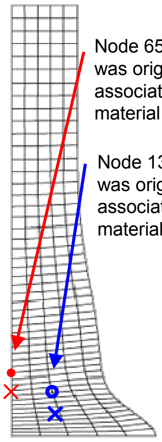


Contours of equivalent plastic strain (PEEQ)

Adaptive Meshing Output and Diagnostics

Adaptive Meshing Output and Diagnostics

- Output for adaptive meshing must be interpreted carefully.
  - Result values at specific locations in the mesh are no longer linked to values at particular material points.
  - A material particle that is coincident with an element integration point at the beginning of a step may not remain so throughout the step.
- Values of displacement and current coordinates represent the motion of the node, not necessarily the motion of the material.
- Contour or vector plots of all other nodal and element variables will show their correct spatial distribution and are, therefore, meaningful.



Node 65 (●) was originally associated with material point ×

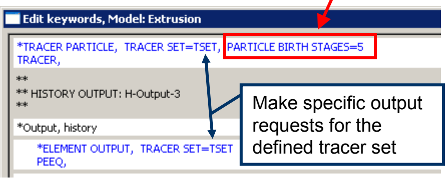
Node 132 (⊙) was originally associated with material point ×

Impact of a copper rod

Adaptive Meshing Output and Diagnostics

- **Tracer particles**
  - Tracer particles can be defined to track material points in an adaptive mesh domain.
    - These particles can also be used to obtain time histories that correspond to the time variation at a specific material point.
  - Output for tracer particles can be written only to the output database file.
    - They can be viewed in ABAQUS/Viewer.
  - The initial location of a tracer particle is defined to be coincident with a node, termed the parent node.
  - Sets of tracer particles can be released from the current locations of the parent nodes at multiple times during the step.

Tracer particles will leave their parent nodes 5 times during the step

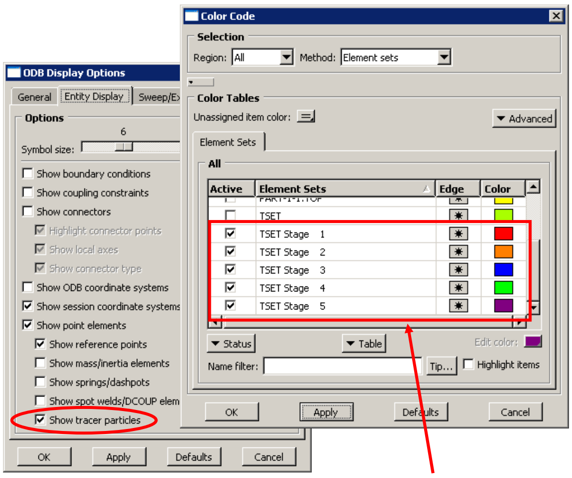


Edit keywords, Model: Extrusion

```
*TRACER PARTICLE,  TRACER SET=TSET, PARTICLE BIRTH STAGES=5
TRACER,
**
** HISTORY OUTPUT: H-Output-3
**
*Output, history
    *ELEMENT OUTPUT,  TRACER SET=TSET
    PEEQ,
```

Make specific output requests for the defined tracer set

Adaptive Meshing Output and Diagnostics



ODB Display Options

Options

Symbol size: 6

☐ Show boundary conditions
☐ Show coupling constraints
☐ Show connectors
☑ Highlight connector points
☑ Show local axes
☑ Show connector type
☐ Show ODB coordinate systems
☑ Show session coordinate systems
☑ Show point elements
☑ Show reference points
☐ Show mass/inertia elements
☐ Show springs/dashpots
☐ Show spot welds/DCOUP elem
☑ Show tracer particles

Color Code

Selection

Region: All Method: Element sets

Color Tables

Unassigned item color:

All

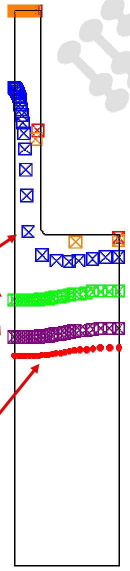
Active	Element Sets	Edge	Color
☐	PART-1-1.TOP	✷	🟨
☐	TSET	✷	🟩
☑	TSET Stage 1	✷	🟥
☑	TSET Stage 2	✷	🟧
☑	TSET Stage 3	✷	🟦
☑	TSET Stage 4	✷	🟩
☑	TSET Stage 5	✷	🟪

Name filter:

☐ Highlight items

color code tracer sets by birth stage number

Video Clip



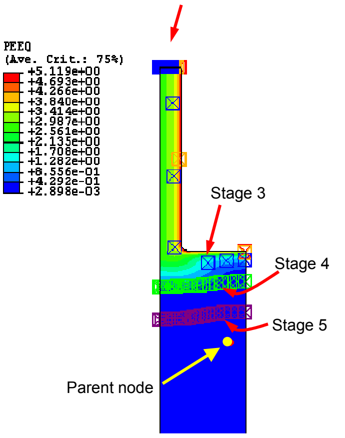
position of tracer particles due to various “birth stages”

location where tracer particles are released (node set **`TRACER`**)

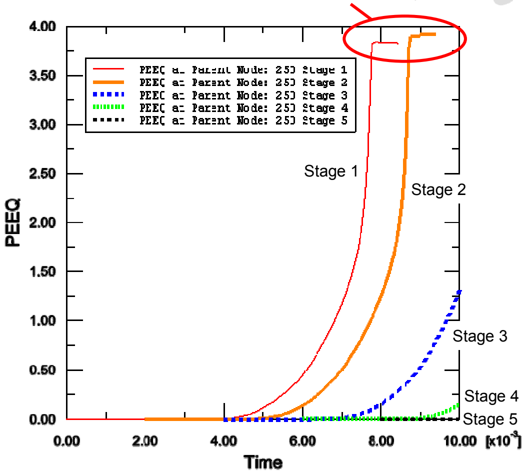
Extrusion analysis

Adaptive Meshing Output and Diagnostics

Stage 1 & 2 tracer particles leave the meshed control volume before the analysis ends



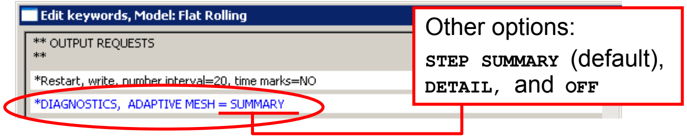
PEEQ contours at end of analysis



PEEQ history for tracer particles from parent node 250

Adaptive Meshing Output and Diagnostics

- **Diagnostic output**
  - To track the efficiency and accuracy of adaptive meshing, diagnostic information can be written to the message (`.msg`) file.
  - By default step summary information is printed at the end of each step, including:
    - The average percentage of nodes moved
    - The maximum percentage of nodes moved
    - The minimum percentage of nodes moved
    - The average number of advection sweeps
  - More (or less) diagnostic information can be requested, for example to obtain a summary for each adaptive mesh increment:



```
** OUTPUT REQUESTS
**
*Restart, write, number interval=20, time marks=NO
*DIAGNOSTICS,  ADAPTIVE MESH = SUMMARY
```

Other options:
`STEP SUMMARY` (default), `DETAIL`, and `OFF`

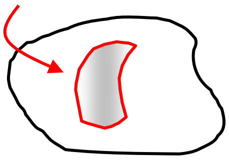
Additional Features of Adaptive Meshing

Additional Features of Adaptive Meshing

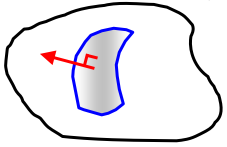
- **Adaptive mesh Boundary Regions**
  - Adaptive mesh boundary regions bound the adaptive mesh domain:
    - Surfaces in three dimensional problems
    - Edges in two-dimensional problems
  - ABAQUS/Explicit will create adaptive mesh boundary regions on:
    - The exterior of a model
    - The boundary between different adaptive mesh domains
    - The boundary between an adaptive mesh domain and a nonadaptive domain
  - You can define adaptive mesh boundary regions using
    - Boundary conditions
    - Loads (concentrated and distributed)
    - Surface definitions

Additional Features of Adaptive Meshing

- Two boundary region types have already been introduced:
  - Lagrangian boundary region:
    - Mesh is constrained to move with the material in the direction normal to the boundary region.
    - Nodes are free to adapt within and along the edges of the region but cannot leave it.
    - Lagrangian boundaries are the default.
      - Exception: the boundary between adaptive and nonadaptive regions is nonadaptive.
  - Eulerian boundary region:
    - Material flows across the boundary,
      - i.e., material flows into or out of the mesh.
    - This region type can only lie on the exterior surface of the model.


Mesh patch follows underlying material

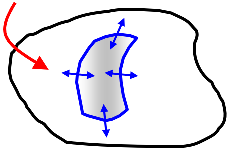
Lagrangian boundary region


Material flows trough the Eulerian boundary

Eulerian boundary region

Additional Features of Adaptive Meshing

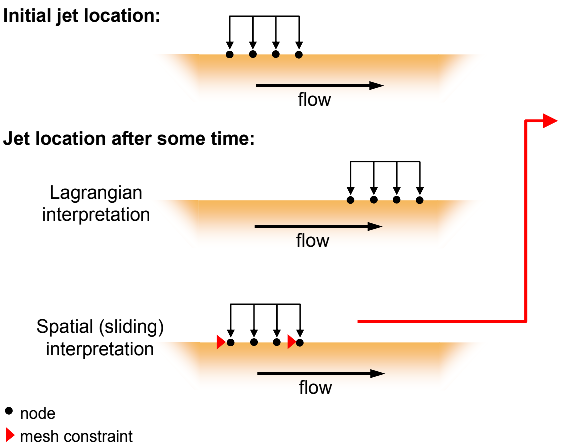
- There is yet another boundary region type.
  - Sliding boundary region:
    - Mesh is constrained to move with the material in the direction normal to the boundary region.
    - The mesh is completely unconstrained in the directions tangential to the boundary region.
      - I.e., the region motion is independent of the underlying material in the tangential directions


Mesh patch slides over the underlying material

sliding boundary region

Additional Features of Adaptive Meshing

- **Example: Surface loading to model a cooling jet**



Spatial (sliding) loading definition:

= convective film condition

```
*FILM, REGION TYPE=SLIDING
 JET, F1, 70., 6.559E-5
```

▶ = adaptive mesh constraint to fix the sliding film condition in space

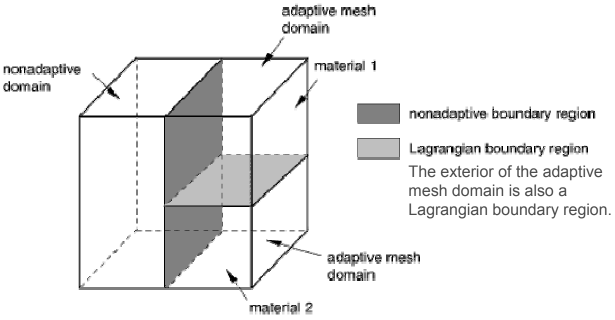
```
*ADAPTIVE MESH CONSTRAINT
 LEFT-JET-NODE, 1,1,0
 RIGTH-JET-NODE,1,1,0
```

Additional Features of Adaptive Meshing

- **Adaptive mesh domains**
  - Multiple adaptive mesh domains can be defined.
    - Element sets used to create adaptive mesh domains cannot overlap.
  - The specified domain will be automatically split into multiple adaptive mesh domains if the specified domain:
    - consists of multiple element types
    - consists of multiple materials
    - spans part instances or regions that are connected by less than a single element face
    - is subject to multiple body force definitions or multiple section control definitions
  - At the boundary between automatically split adaptive mesh domains, the mesh can only adapt along boundary.
  - Element sets (*userelsetname-domain#-step#*) are created for each adaptive meshing domain, including those split automatically.

Additional Features of Adaptive Meshing

– For example, ABAQUS/Explicit automatically divides the adaptive mesh domain defined on the right side of this block at the boundary between two different materials:



The exterior of the adaptive mesh domain is also a Lagrangian boundary region.

user-defined adaptive mesh domain: right half of box

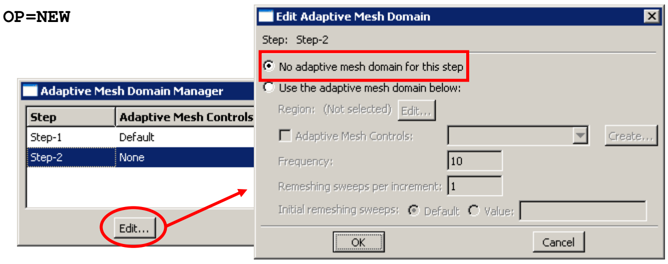
Additional Features of Adaptive Meshing

– By default, all adaptive mesh domains defined in a previous analysis step remain unchanged in the subsequent step.

– Adaptive mesh domains can be added, modified, or removed on a step-by-step basis.

  - For example, to deactivate adaptive meshing in a step:

```
*ADAPTIVE MESH,OP=NEW
```



Step	Adaptive Mesh Controls
Step-1	Default
Step-2	None

Step: Step-2

● No adaptive mesh domain for this step

○ Use the adaptive mesh domain below:

Region: (Not selected)

☐ Adaptive Mesh Controls:

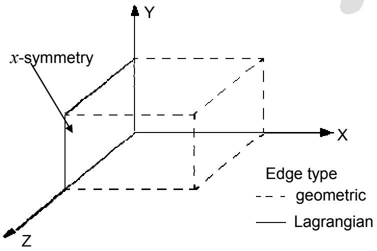
Frequency: 10

Remeshing sweeps per increment: 1

Initial remeshing sweeps: ● Default ○ Value:

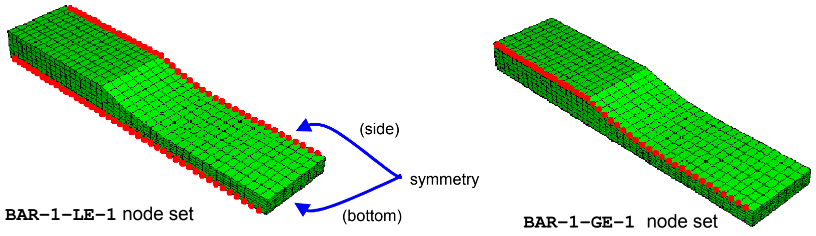
Additional Features of Adaptive Meshing

- **Geometric features**
  - Geometric edges and corners are detected on adaptive mesh domains.
    - Adaptivity is not performed across these geometric features unless they flatten.
  - Geometric edges are “soft” edges.
    - They remain edges until the surfaces flatten.
  - Lagrangian edges are “hard” edges. Adaptive meshing is never performed across these edges.



Additional Features of Adaptive Meshing

- Use automatically created node sets to verify Lagrangian edges, geometric edges, and nonadaptive nodes (*userelsetname-domain#-LE/GE/NA-step#*).
  - LE: Lagrangian edge nodes
  - NA: nonadaptive nodes
  - GE: Geometric edge nodes
- For more information refer to the *ABAQUS Analysis User’s Manual*.



Automatically created node sets for the steady-state flat rolling simulation

Additional Features of Adaptive Meshing

- **Modeling issues with adaptive mesh regions**
  - Combinations of loads, boundary conditions, and surfaces can produce different adaptive mesh regions.
    - Use ABAQUS/Viewer to check for Lagrangian edges, geometric edges, and nonadaptive nodes.
  - If small siding or tied contact is defined in an adaptive mesh domain, all nodes on both surfaces are nonadaptive.
  - All elements other than first-order, reduced-integration, solid elements are nonadaptive.
    - Elements with rebars are not part of adaptive mesh regions.
    - Nodes with spot welds, springs, or dashpots are nonadaptive.
  - Use degenerate quadrilateral/brick elements to define triangular/tetrahedral elements.
    - For example, using the CPE3 element will result in split domains.

Additional Features of Adaptive Meshing

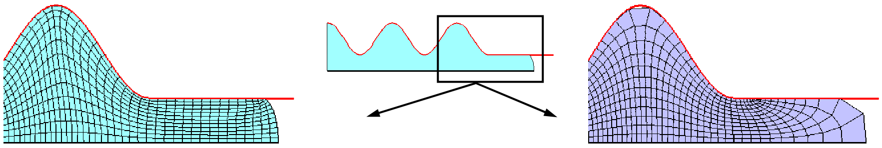
- **Smoothing refers to the remeshing of the domain of interest to smooth element distortion.**
  - ABAQUS/Explicit can use one or more of the following basic smoothing methods:
    - Volume smoothing
    - Laplacian smoothing
    - Equipotential smoothing
  - Volume smoothing is very robust and is the default method
  - For more information on mesh smoothing methods refer to the *ABAQUS Analysis User's Manual*.

Additional Features of Adaptive Meshing

- **After the mesh has been smoothed element variables, nodal variables, and momentum are remapped by advection.**
  - Two advection methods are available in ABAQUS/Explicit:
    - The default second-order advection method improves accuracy during the remapping phase of adaptive meshing.
    - First-order method tends to diffuse any sharp gradients of element variables during the remapping phase.
  - For more information on advection refer to the *ABAQUS Analysis User's Manual*.

Additional Features of Adaptive Meshing

- **Solution-dependent adaptive meshing prevents the reduction of mesh refinement near areas of evolving concave curvature.**
  - Basic smoothing methods reduce the mesh refinement near concave boundaries.
  - With solution-dependent adaptive meshing, mesh gradation is automatically focused toward these areas.
    - The aggressiveness of the meshing is governed by the curvature refinement weight, which has a default value of 1.
  - For more information refer to the *ABAQUS Analysis User's Manual*.



Default curvature refinement

Axisymmetric forging problem

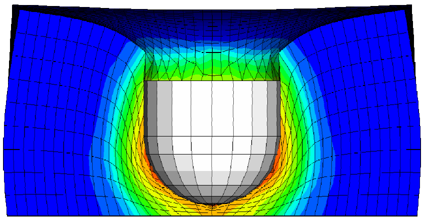
No curvature refinement

Element Distortion Control

Element Distortion Control

- **ABAQUS/Explicit offers distortion control to prevent solid elements from inverting or distorting excessively.**
  - Distortion control is designed to prevent negative element volumes or other excessive distortion from occurring during an analysis.
    - In contrast to the adaptive meshing technique, distortion control does not attempt to maintain a high-quality mesh throughout an analysis.
    - Elements with distortion control can not be included in an adaptive mesh domain.

no-dist-ctrl with dist ctrl

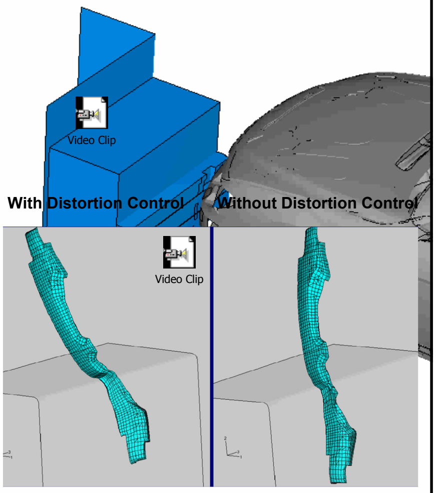


crushable foam indentation

Element Distortion Control

- Many analyses with volumetrically compacting materials (e.g., crushable foams) see large compressive and shear deformations.
  - Especially true when the crushable materials are used as energy absorbers between stiff or heavy components.
- Analyses may fail prematurely when the mesh is coarse relative to strain gradients and the amount of compression.
- Distortion control prevents solid elements from inverting or distorting excessively for these cases.

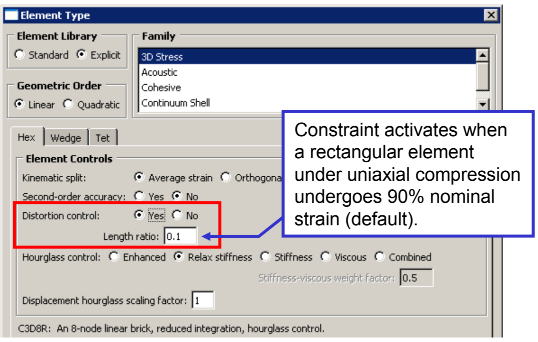
Courtesy of BMW



Element Distortion Control

- **Example: Crushable foam Indentation**

```
*SOLID SECTION, ELSET=BLANK, MATERIAL=FOAM, CONTROLS=DistortionControl
*SECTION CONTROLS, NAME=DistortionControl, HOURGLASS=ENHANCED,
 KINEMATICS=ORTHOGONAL, DISTORTION CONTROL
```



Element Library: ○ Standard ● Explicit

Geometric Order: ● Linear ○ Quadratic

Family: 3D Stress, Acoustic, Cohesive, Continuum Shell

Element Controls

Kinematic split: ● Average strain ○ Orthogona

Second-order accuracy: ○ Yes ● No

Distortion control: ● Yes ○ No

Length ratio: 0.1

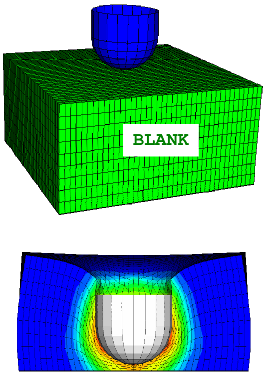
Hourglass control: ○ Enhanced ● Relax stiffness ○ Stiffness ○ Viscous ○ Combined

Stiffness-viscous weight factor: 0.5

Displacement hourglass scaling factor: 1

C3D8R: An 8-node linear brick, reduced integration, hourglass control.

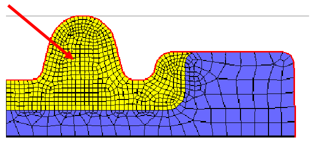
Constraint activates when a rectangular element under uniaxial compression undergoes 90% nominal strain (default).



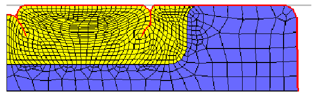
Element Distortion Control

- **Distortion control is activated by default for elements modeled with hyperelastic or hyperfoam materials.**
  - Using adaptive meshing in a domain modeled with hyperelastic or hyperfoam materials is not recommended.
    - Better results are generally predicted using the enhanced hourglass method in combination with element distortion control.

Hyperelastic material



undeformed shape

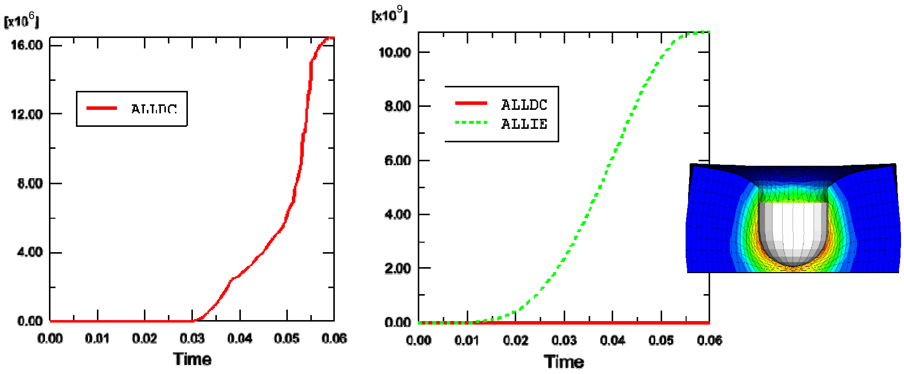


final deformed shape

Compression of a rubber gasket

Element Distortion Control

- **Energy dissipated by distortion control can be output.**
  - Total energy dissipated by distortion control (ALLDC)
  - Energy dissipated in the element by distortion control (ELDC)



Energy output from crushable foam indentation analysis