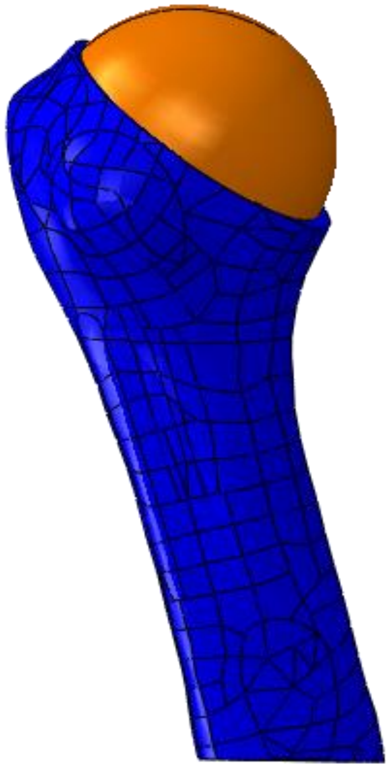
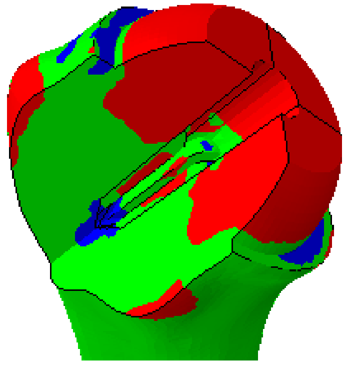
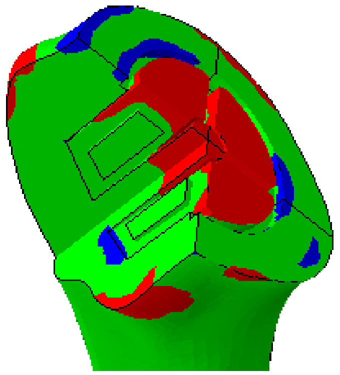


Tissue Biomechanics

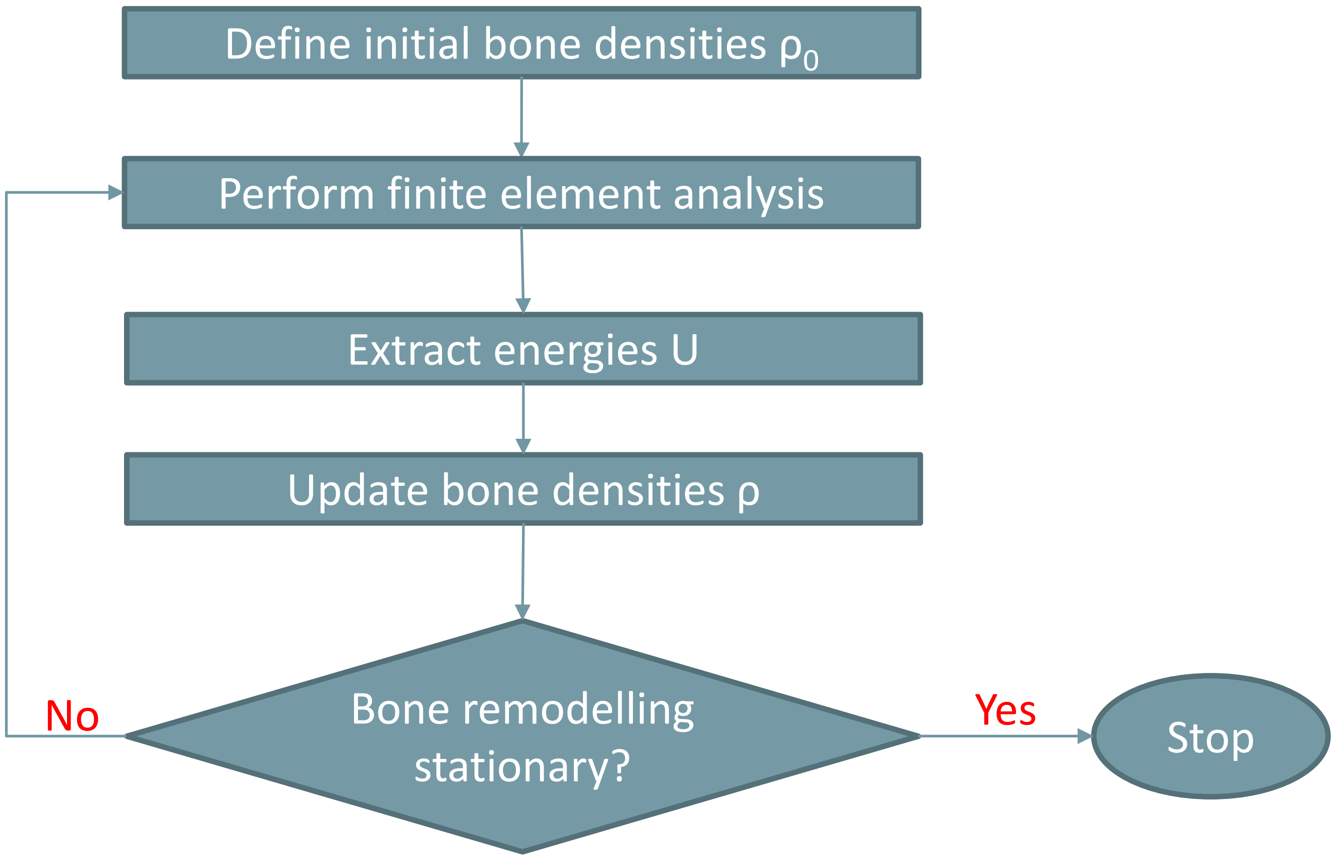
2º semester, 2017-2018

Carlos Quental, Paulo Fernandes, Fernando Simões

Instituto Superior Técnico, Universidade de Lisboa

Bone remodelling model


Define initial bone densities ρ_0
Perform finite element analysis
Extract energies U
Update bone densities ρ
Bone remodelling stationary?
No
Yes
Stop

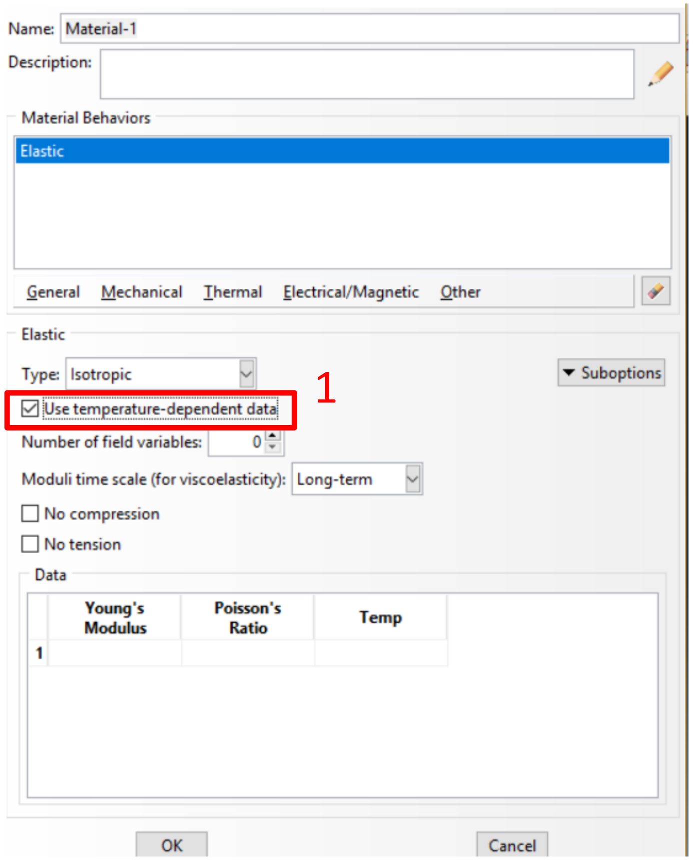
Bone remodelling model

- **Bone densities** are defined at the **nodes** of the finite element mesh as if they were temperatures.

- Defining **bone densities as temperatures simplifies** the definition of the bone material properties as Abaqus allows the definition of **temperature-dependent properties (1)**.

- **A table** describing how the material properties of bone change with bone density (temperature) **needs to be defined**.

$$E = E_0 \rho^p$$


Name: Material-1

Description:

Material Behaviors

Elastic

General Mechanical Thermal Electrical/Magnetic Other

Elastic

Type: Isotropic

☑ Use temperature-dependent data

1

Number of field variables: 0

Moduli time scale (for viscoelasticity): Long-term

☐ No compression

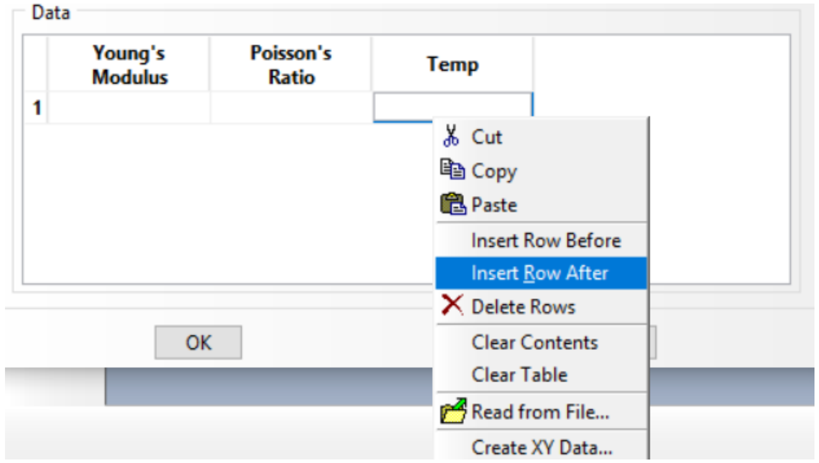
☐ No tension

Data

	Young's Modulus	Poisson's Ratio	Temp
1			

Bone remodelling model

- To create a table, add rows to the data section by **right-clicking the mouse over this section** and selecting **Insert Row After**.

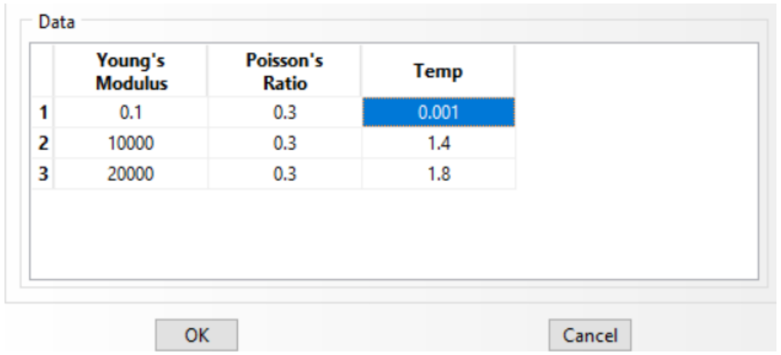


- As the material law for bone ($E = E_0 \rho^p$) cannot be defined analytically in Abaqus, several data points of the law need to be defined to better approximate the function.

- **Between data points**, Abaqus interpolates the data using **linear interpolation**.

Bone remodelling model

- **For example,** if three points were used,

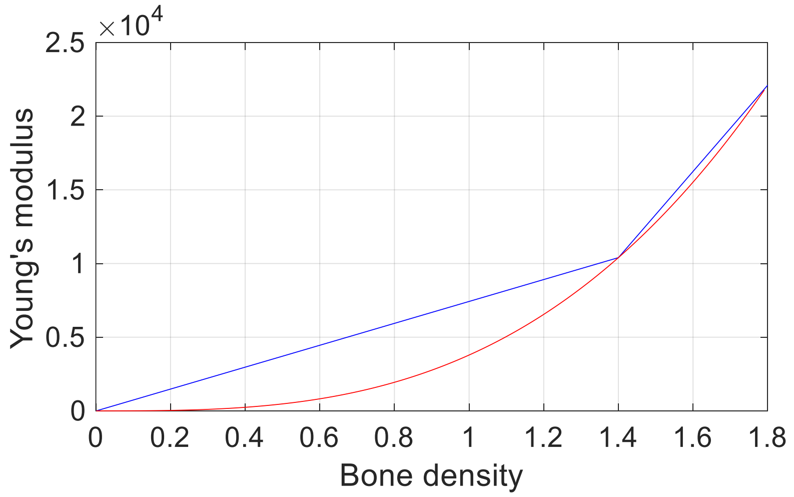


Data

	Young's Modulus	Poisson's Ratio	Temp
1	0.1	0.3	0.001
2	10000	0.3	1.4
3	20000	0.3	1.8

- Abaqus would define the Young's modulus of bone using the blue curve instead of the true curve in red.

- **Notice** that the bone material properties are defined at this point but the **bone densities** throughout the bone **still have to be defined**.

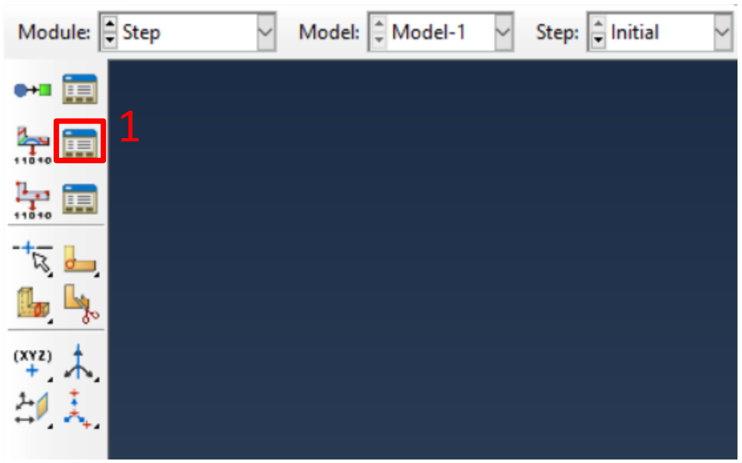


Bone remodelling model

- **Huiskes' bone remodelling law** is given by**:**

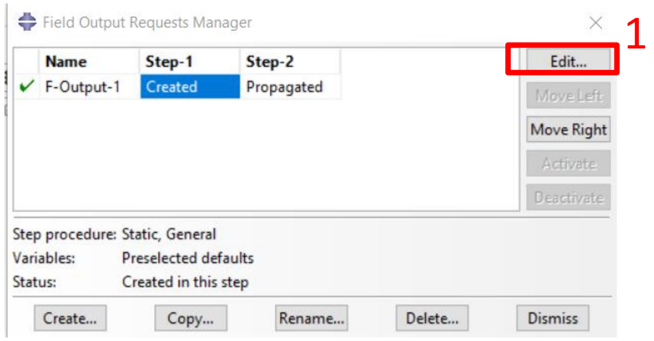
$$\frac{d\rho}{dt} = \begin{cases} B\left(\frac{U}{\rho} - (1-s).k\right), if\ \frac{U}{\rho} < (1-s).k \\ B\left(\frac{U}{\rho} - (1+s).k\right), if\ \frac{U}{\rho} > (1+s).k \\ 0 \qquad ,otherwise \end{cases}$$

- **By default**, Abaqus does not report the **strain energy density U**. To report U, the Field Output needs to be **edited** using the **Field Output Manager (1) in the step module**.


Module: Step Model: Model-1 Step: Initial

Bone remodelling model

- **Click edit (1):**



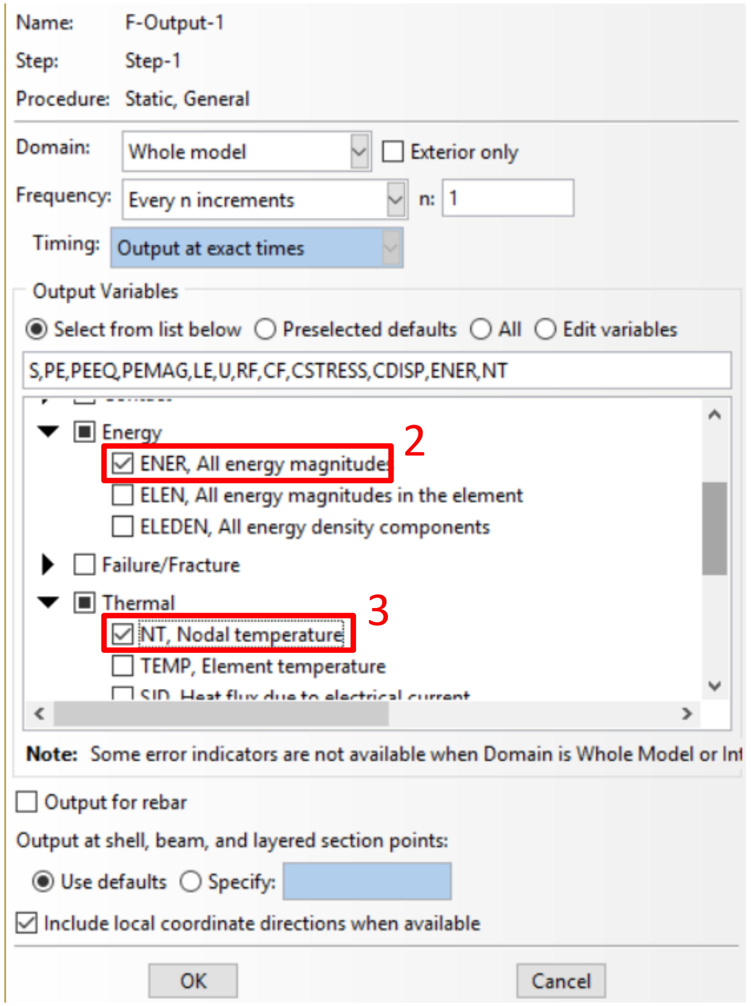
Field Output Requests Manager

	Name	Step-1	Step-2
✓	F-Output-1	Created	Propagated

Step procedure: Static, General
Variables: Preselected defaults
Status: Created in this step

- Select the **ENER, All energy magnitudes (2)** option.

- To be able to see the **bone density distribution (temperature)**, the **NT, Nodal Temperature (3)** option also needs to be selected.



Name: F-Output-1
Step: Step-1
Procedure: Static, General
Domain: Whole model ☐ Exterior only
Frequency: Every n increments n: 1
Timing: Output at exact times
Output Variables
[✓ Select from list below] Preselected defaults All Edit variables
S,PE,PEEQ,PEMAG,LE,U,RF,CF,CSTRESS,CDISP,ENER,NT

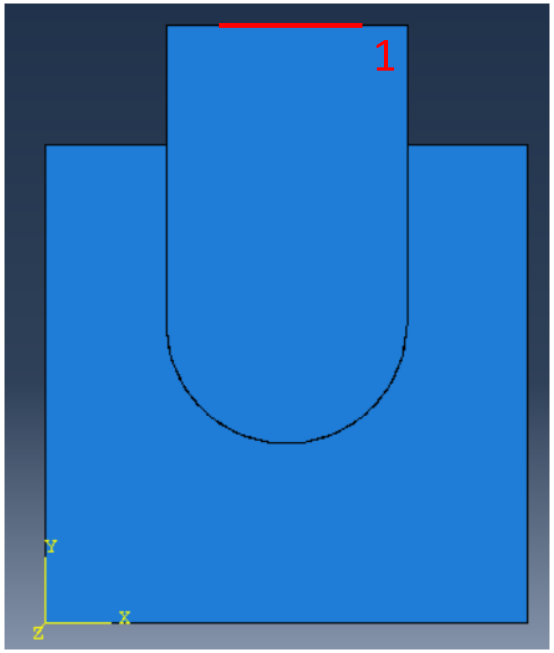
Energy
☑ ENER, All energy magnitudes
☐ ELEN, All energy magnitudes in the element
☐ ELEDEN, All energy density components
☐ Failure/Fracture
Thermal
☑ NT, Nodal temperature
☐ TEMP, Element temperature
☐ SJD, Heat flux due to electrical current

Note: Some error indicators are not available when Domain is Whole Model or Int

☐ Output for rebar
Output at shell, beam, and layered section points:
[✓ Use defaults] Specify:
☑ Include local coordinate directions when available

Bone remodelling model

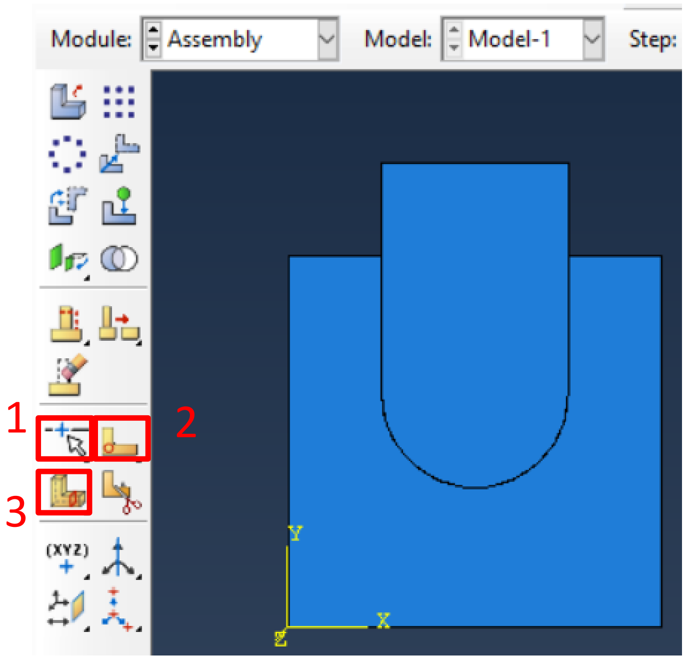
- If a **concentrated force** is to be **distributed over a certain region (for example, region (1))**, the force can be applied on an **attachment point** and be distributed to the region using a **coupling constraint**.



- An **attachment point** is a point that **does not belong to any part** but may transmit force to the model.

Bone remodelling model

- To define the **region for force distribution**, **partitions** may have to be created using the **Partition edge: Specify Parameter by Location (1)** at the assembly module.

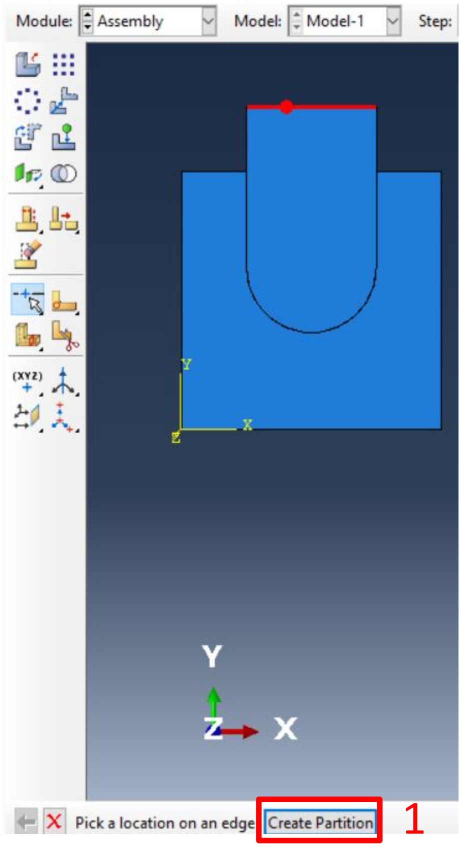


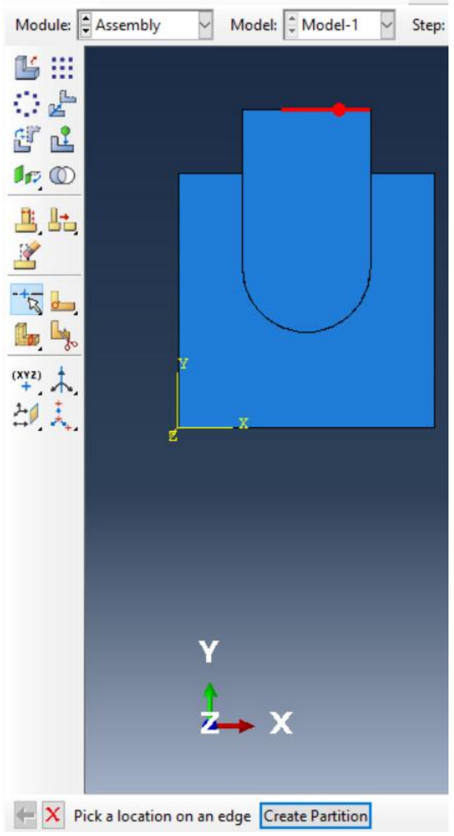
- **Partitions on surfaces and solids** can be created using the features **Partition Face (2)** and **Partition Cell (3)**, respectively.

Bone remodelling model

- Using the **Partition edge: Specify Parameter by Location** tool, select the first point of the region to split the edge into two edges and click on **Create Partition (1)**.

- **Repeat the procedure** for the **opposite extremity** of the region.





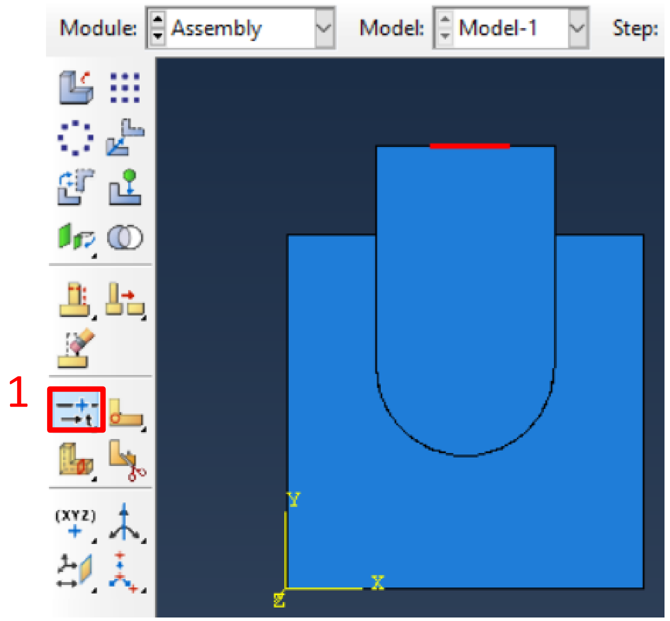
Bone remodelling model

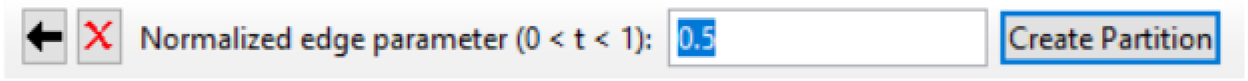
- Considering that the **attachment point** should be **placed at the centroid of the region**, a node at the middle of the region may be created to help define the attachment point.

- For that, select the tool **Partition Edge: Enter Parameter (1)**, which appears after clicking a few seconds on the Partition Edge tool.

- After selecting the edge, click **Done**.

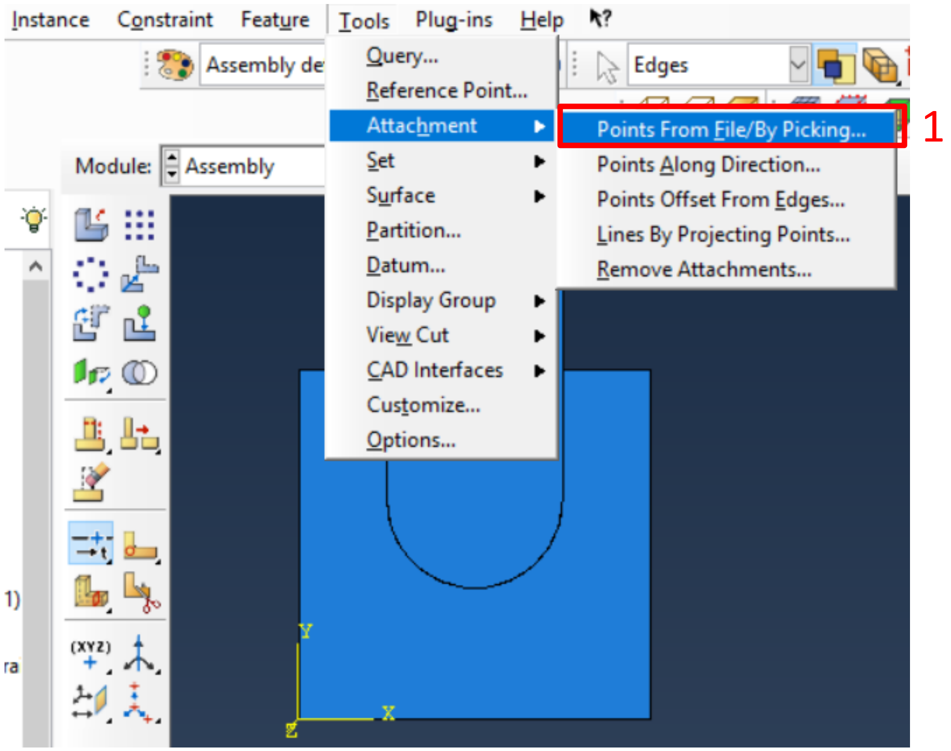
- Then **input 0.5** and click **Create Partition**. Notice that t = 0 is the first point of the edge and t = 1 is the last.




Normalized edge parameter (0 < t < 1): 0.5 Create Partition

Bone remodelling model

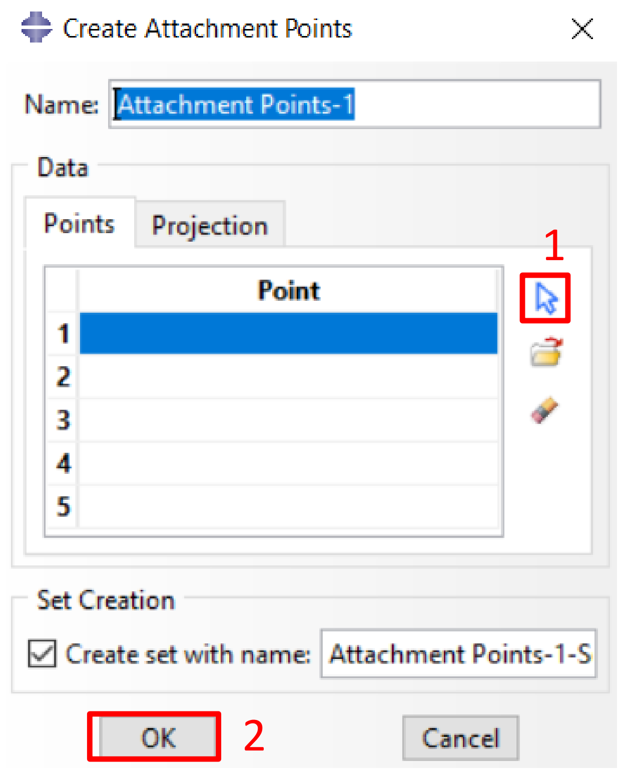
- To create an attachment point, click **Tools -> Attachment -> Points From File / By Picking… (1)**.



- **Other, intuitive, options** are also **available** to define attachment points.

Bone remodelling model

- Then click on the **arrow (1),** select the middle node of the edge, and click **OK (2)**.



Create Attachment Points

Name: Attachment Points-1

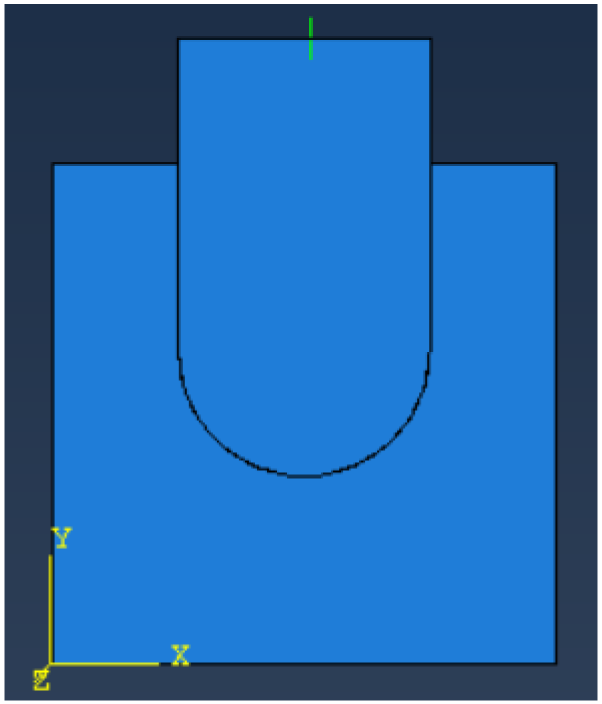
Data

Points | Projection

	Point
1	
2	
3	
4	
5	

Set Creation

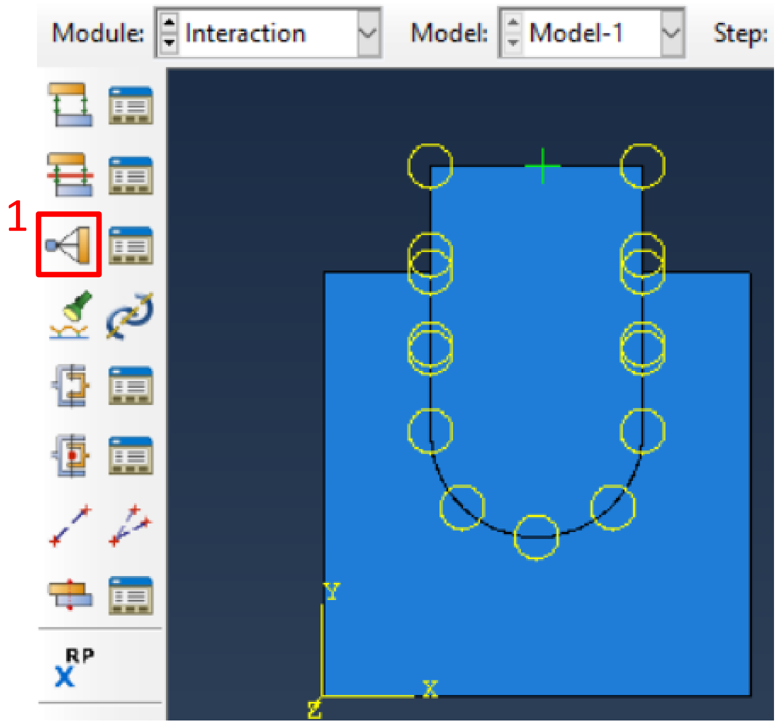
☑ Create set with name: Attachment Points-1-S

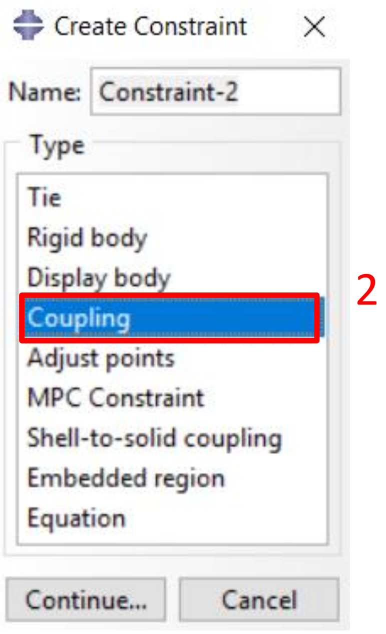


- The **attachment point** appears as a **green plus sign**.

Bone remodelling model

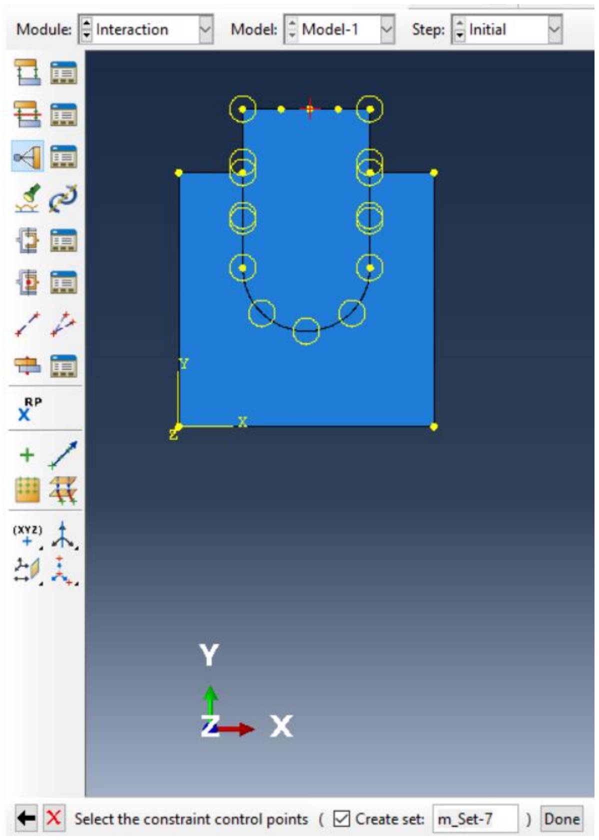
- To define a **coupling interaction** between the attachment point and the region, go to the **Interaction module,** click on **Create Constraint (1)**, and select **Coupling (2)**.



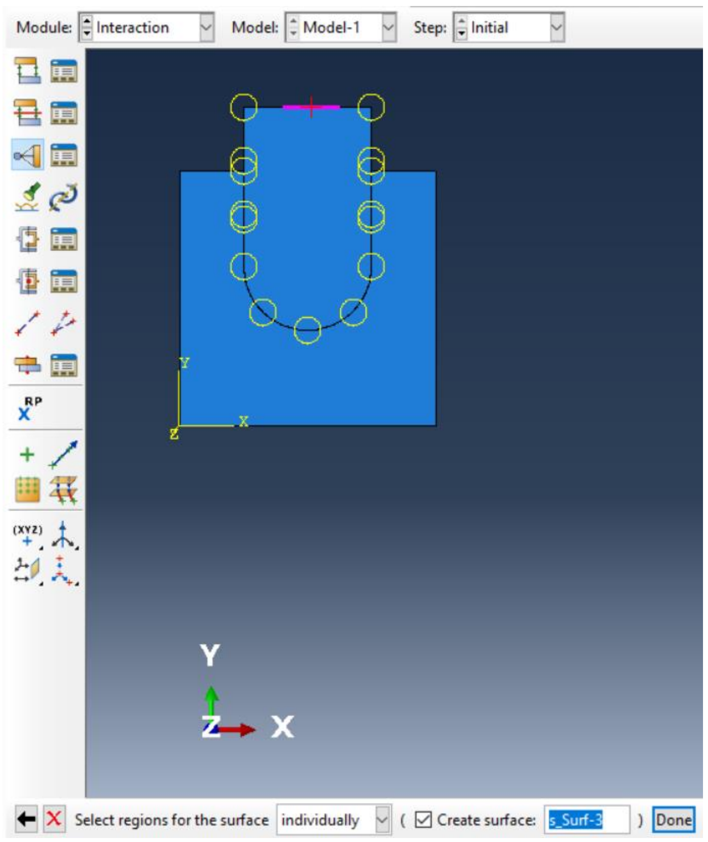


Bone remodelling model

- **Select the attachment point** and click **Done**. Then, **select Surface** as the region type**, identify the region,** and click **Done**.



Select the constraint control points (☑ Create set: m_Set-7) Done

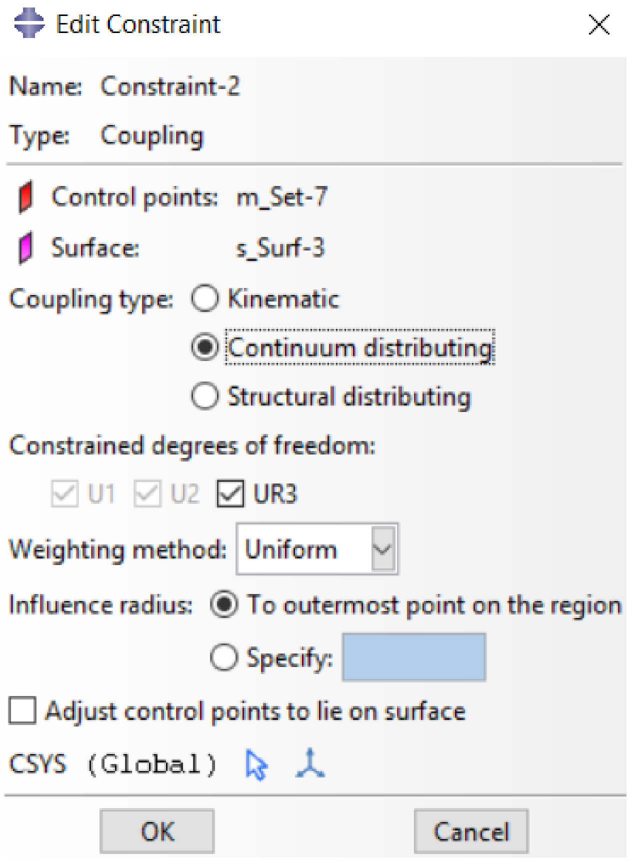


Select regions for the surface individually (☑ Create surface: s_Surf-3) Done

Bone remodelling model

- The **coupling type** must be **Continuum distributing.** The remaining options may be the default.

- The **Weighting method** defines how the force is distributed.
  - **Uniform** means the force is distributed uniformly over the region.
  - The remaining options decrease the magnitude of the forces applied on the nodes according to their distance to the attachment point.



Edit Constraint

Name: Constraint-2

Type: Coupling

Control points: m_Set-7

Surface: s_Surf-3

Coupling type: ○ Kinematic [✓ Continuum distributing] ○ Structural distributing

Constrained degrees of freedom: ☑ U1 ☑ U2 ☑ UR3

Weighting method: Uniform

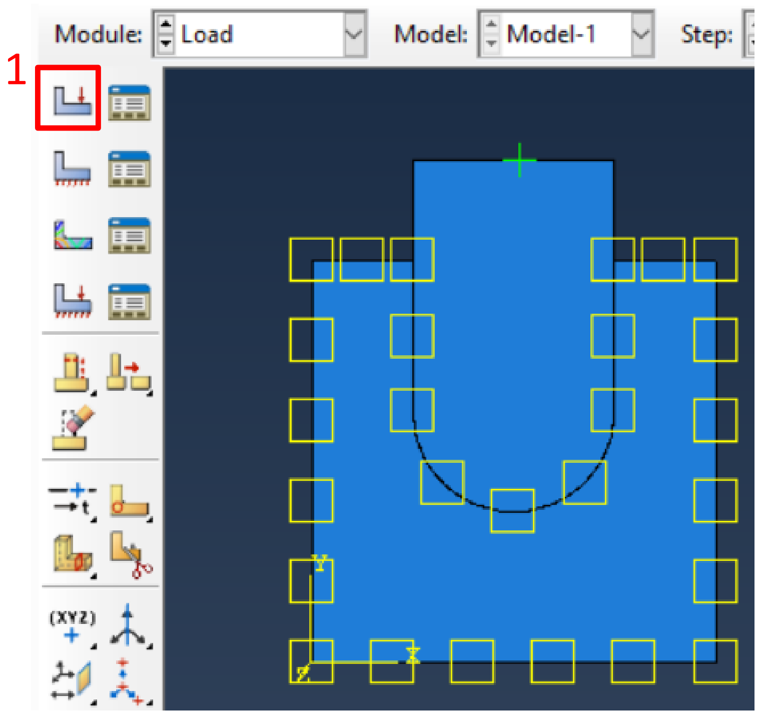
Influence radius: [✓ To outermost point on the region] ○ Specify:

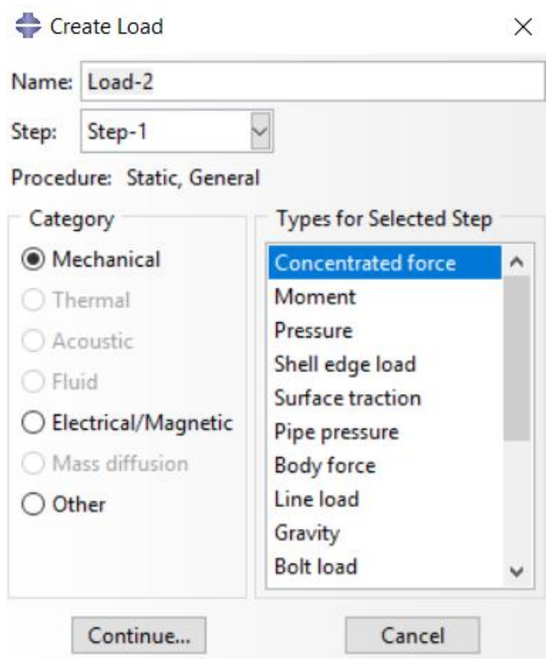
☐ Adjust control points to lie on surface

CSYS (Global)

Bone remodelling model

- Once the coupling interaction between the attachment point and the region is defined, the **concentrated force** should be defined at the **attachment point** using the **Create Load (1)** tool at the **Load module**.




Create Load

Name: Load-2

Step: Step-1

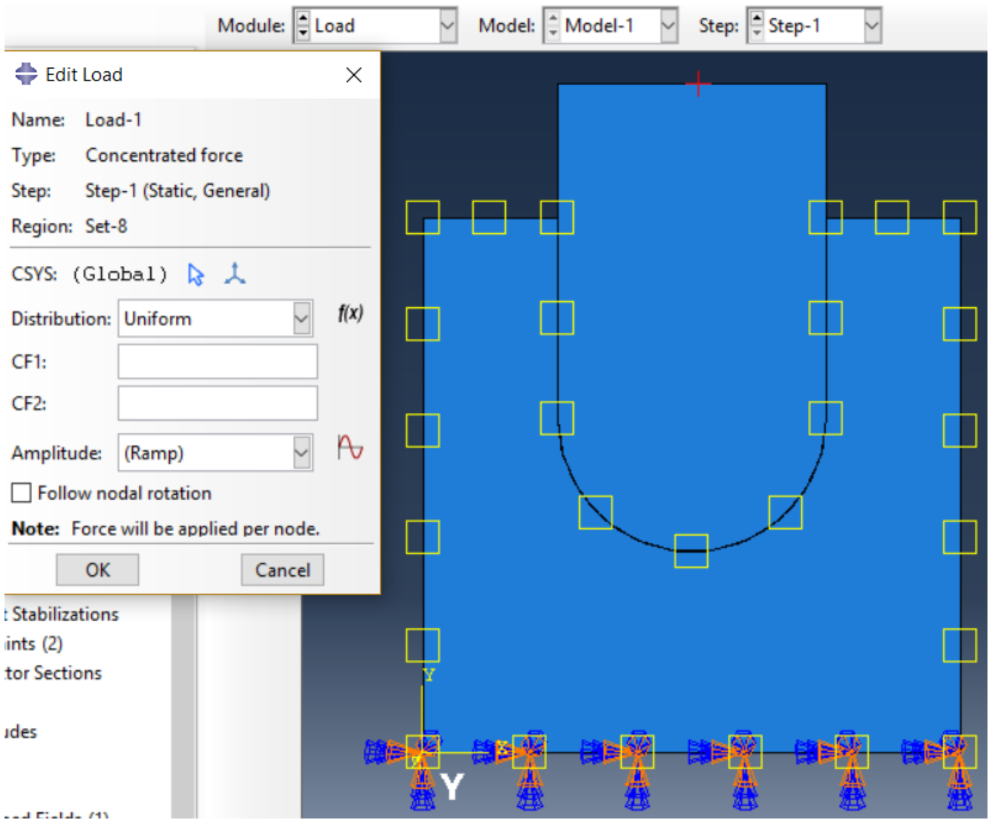
Procedure: Static, General

Category: [✓ Mechanical] Thermal Acoustic Fluid Electrical/Magnetic Mass diffusion Other

Types for Selected Step: [✓ Concentrated force] Moment Pressure Shell edge load Surface traction Pipe pressure Body force Line load Gravity Bolt load

Bone remodelling model

- After selecting the attachment point, input the **components of the force (CF1 and CF2)**.


Module: Load
Model: Model-1
Step: Step-1

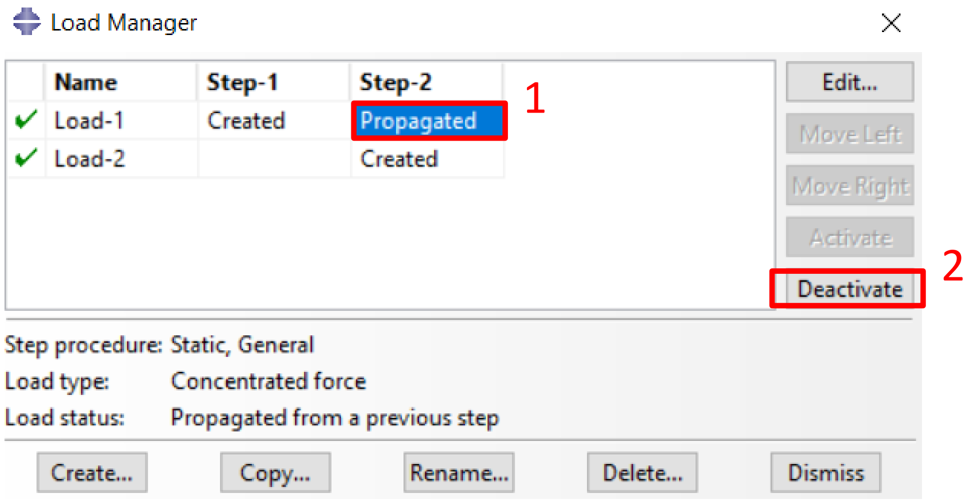
Edit Load

Name: Load-1
Type: Concentrated force
Step: Step-1 (Static, General)
Region: Set-8

CSYS: (Global)
Distribution: Uniform
CF1:
CF2:
Amplitude: (Ramp)
☐ Follow nodal rotation
Note: Force will be applied per node.

Bone remodelling model

- If **more than one load case** exists, you can define **different steps**, one for each load case.



Load Manager

	Name	Step-1	Step-2
✓	Load-1	Created	Propagated
✓	Load-2		Created

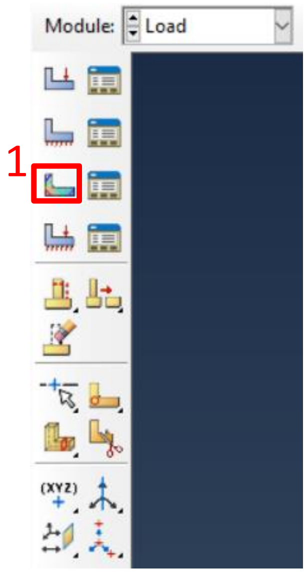
Step procedure: Static, General
Load type: Concentrated force
Load status: Propagated from a previous step

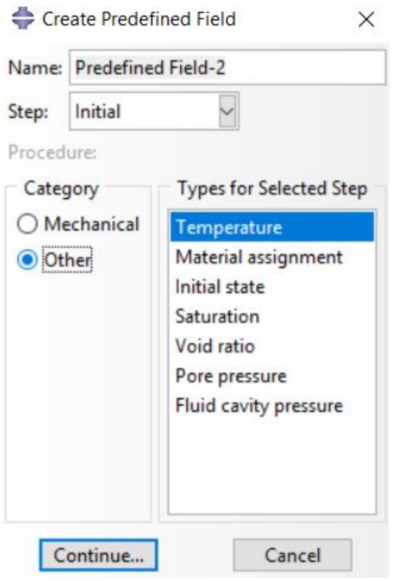
- By default, Abaqus **propagates the loads** from one step to the **following steps** (notice the propagated instruction in (1)).

- To prevent Abaqus from propagating the loads along the steps, click on the first **Propagated instruction** for the load case and click on **Deactivate (2)**.

Bone remodelling model

- To define how the **bone density changes** within the bone, a **predefined field** needs to be defined (1).




Create Predefined Field

Name: Predefined Field-2

Step: Initial

Procedure:

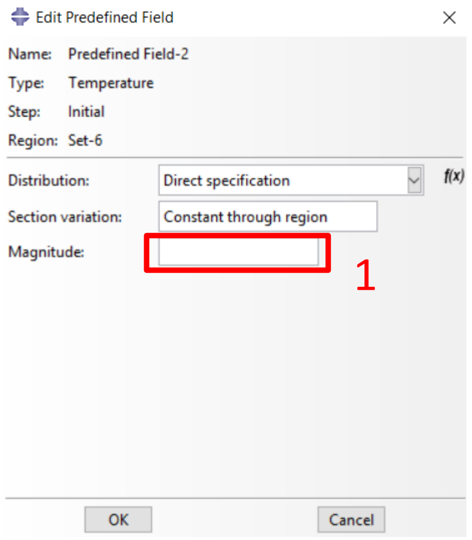
Category: ○ Mechanical [✓ Other]

Types for Selected Step: [✓ Temperature] Material assignment, Initial state, Saturation, Void ratio, Pore pressure, Fluid cavity pressure

- As bone density will be simulated as temperature, select Category **Other** and Type **Temperature**. For the step, select **Initial**.

Bone remodelling model

- After selecting the regions for the field, i.e., the bone, define the bone density (temperature) at (1).



Edit Predefined Field

Name: Predefined Field-2
Type: Temperature
Step: Initial
Region: Set-6

Distribution: Direct specification
Section variation: Constant through region
Magnitude:

- At the **Abaqus interface**, only a **constant bone density** can be defined within the bone.
- A **variable bone density distribution** needs to be **defined later by changing** the instructions of the input file regarding the definition of the constant predefined field.

Bone remodelling model

- To change the input file, search for the keyword **Initial**:

```
** Name: Predefined Field-1   Type: Temperature
*Initial Conditions, type=TEMPERATURE
Set-5, 1.
```

- To define **a variable bone density distribution**, to be read from a text file called **dens.txt**, change the line following the instruction ***Initial**, i.e.:

```
** Name: Predefined Field-1   Type: Temperature
*Initial Conditions, type=TEMPERATURE
*Include, input=dens.txt
```

Bone remodelling model

- The density file should contain **in each line** the bone density for each node of the finite element mesh using the following **structure**:

```
NameOfInstance.NumberOfNode, NodalDensity
```

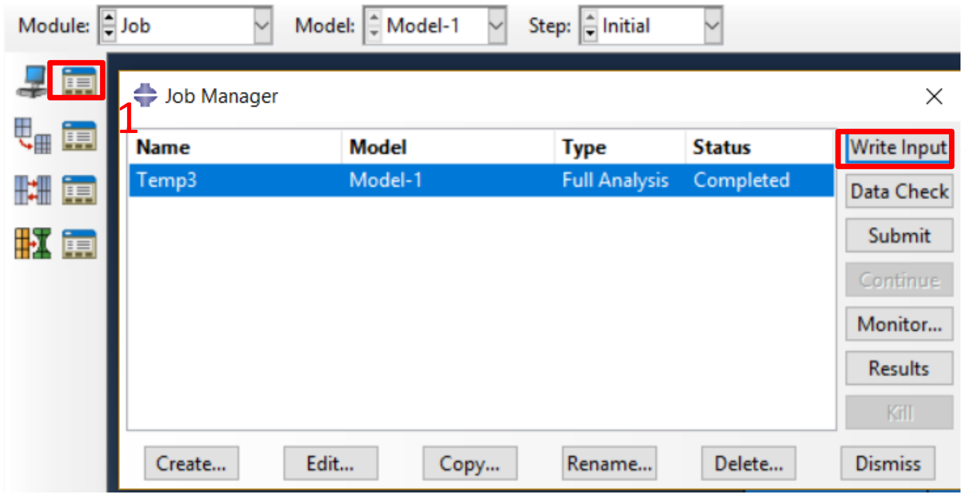
where NameOfInstance is the name of the instance, NumberOfNode is the number of the node, and NodalDensity is the density at the node NumberOfNode.

- **For example**:

```
Part-3-1.1, 0.631353
Part-3-1.2, 0.256001
Part-3-1.3, 0.050299
Part-3-1.4, 0.175944
Part-3-1.5, 0.141576
Part-3-1.6, 0.104506
Part-3-1.7, 1.800000
Part-3-1.8, 0.631713
```

Bone remodelling model

- Once the finite element model is defined, the ***.inp file** needs to be written. For that, click on the **Job Manager (1) -> Write Input (2)** in the Job module.



Name	Model	Type	Status
Temp3	Model-1	Full Analysis	Completed

- To perform a **finite element analysis in Matlab**, use the following instruction:

```
system('abaqus job=NameInpFile inter');
```

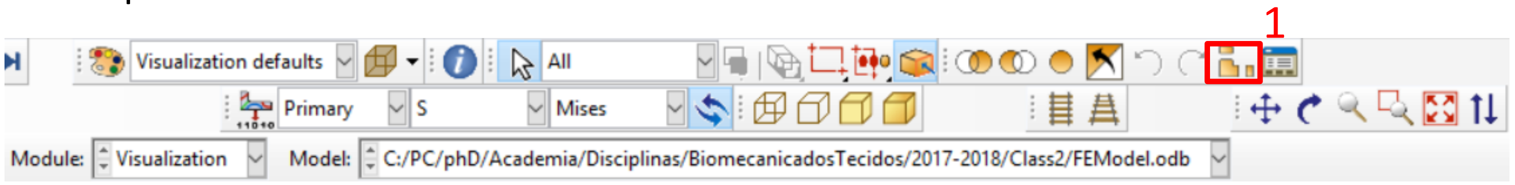
where **NameInpFile** is the **name of the finite element model** (Temp3 in the example above).

Bone remodelling model

- Once a finite element analysis is completed, the **strain energy densities** need to be **extracted from the output file**. For that purpose, a **python script** needs to be created to have the strain energy densities **reported automatically into a text file**.

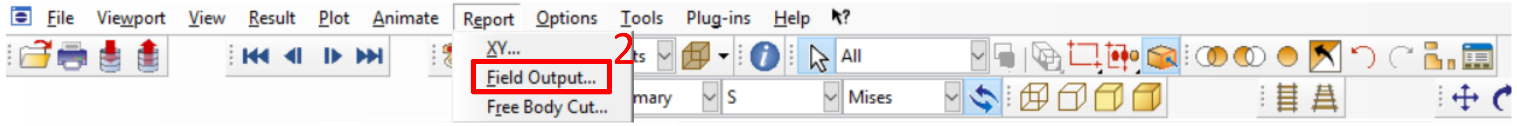
- To create the python script, **complete a finite element analysis** of the model developed **once** and **open** the corresponding ***.odb file**.

- Once the *.odb file is open, **Create a Display Group (1)** to make bone the only part visible in the results. If only the bone is visible, the text file will only contain data for this part.



Module: Visualization Model: C:/PC/phD/Academia/Disciplinas/BiomecanicadosTecidos/2017-2018/Class2/FEModel.odb

- To obtain a text file with the strain energy densities at the nodes, click **Report -> Field Output (2)**.

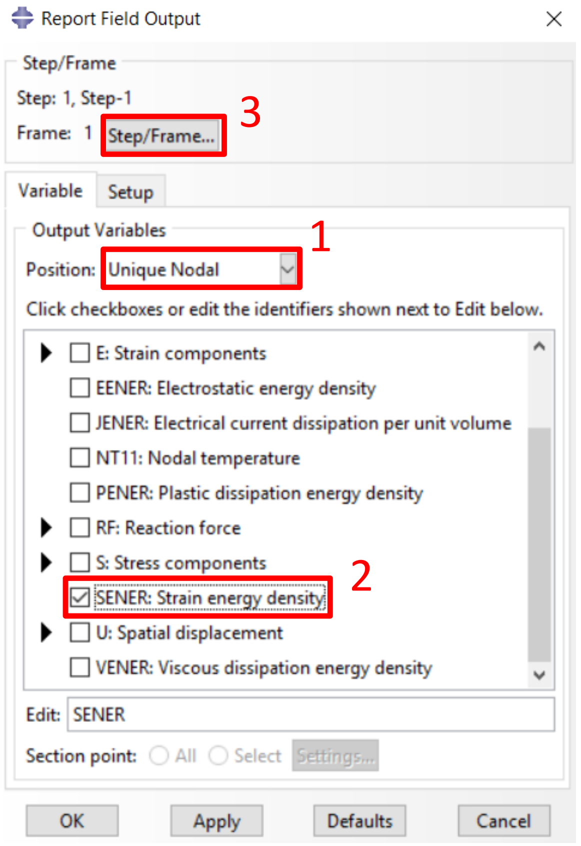


Bone remodelling model

- Then select Position **Unique Nodal (1)** and check the box **SENER: Strain energy density (2)**.

- The results are reported for a specific step. You can **select the step** wanted by clicking on **Step/Frame (3)**.

- At the **Setup** tab, the name of the ouput file can be defined, as well as the option to append the data (or not).



Report Field Output

Step/Frame

Step: 1, Step-1

Frame: 1 Step/Frame... 3

Variable | Setup

Output Variables 1

Position: Unique Nodal

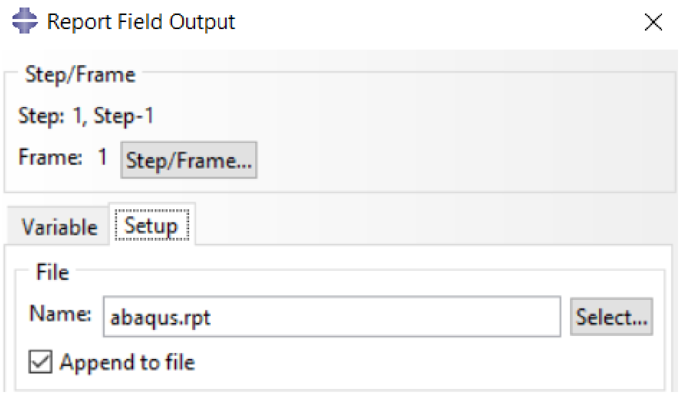
Click checkboxes or edit the identifiers shown next to Edit below.

- ☐ E: Strain components
- ☐ EENER: Electrostatic energy density
- ☐ JENER: Electrical current dissipation per unit volume
- ☐ NT11: Nodal temperature
- ☐ PENER: Plastic dissipation energy density
- ☐ RF: Reaction force
- ☐ S: Stress components
- ☑ SENER: Strain energy density 2
- ☐ U: Spatial displacement
- ☐ VENER: Viscous dissipation energy density

Edit: SENER

Section point: ○ All ○ Select Settings...

OK Apply Defaults Cancel



Report Field Output

Step/Frame

Step: 1, Step-1

Frame: 1 Step/Frame...

Variable | Setup

File

Name: abaqus.rpt Select...

☑ Append to file

Bone remodelling model

- Once the report is created, **close Abaqus**. At the working folder, there should be an **abaqus.rpy** file that contains all the instructions performed.

- Rename the file abaqus.rpy to **OutputExtraction.py**, which will be used in Matlab to **create automatically** an output file with the strain energy densities from the finite element analyses**.**

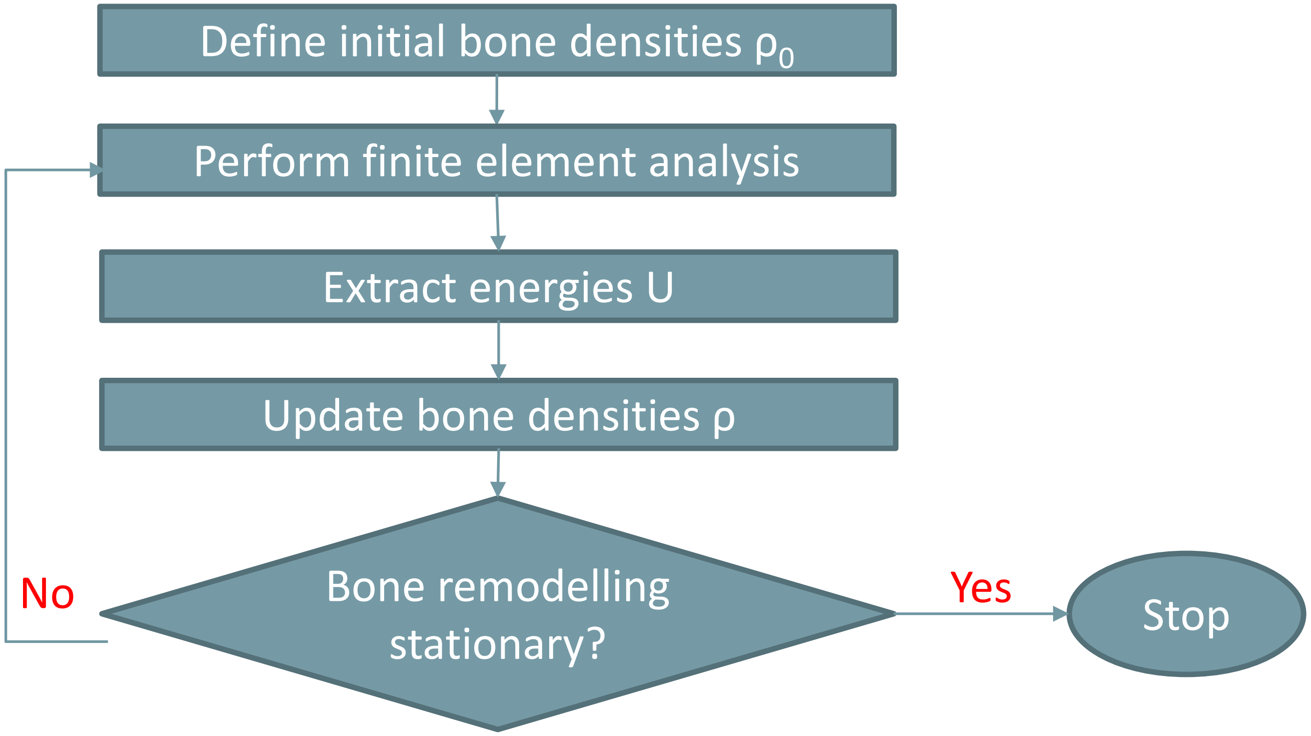
- To run the Python script **in Matlab**, use the following instruction:

```
system('abaqus viewer noGUI=OutputExtraction.py');
```

- Once the output file is created, the strain energy densities need to be read from the file to evaluate the bone remodelling law.

Bone remodelling model

- Using the tools and procedures described, the bone remodelling process needs to be implemented in Matlab.



Define initial bone densities ρ_0
Perform finite element analysis
Extract energies U
Update bone densities ρ
Bone remodelling stationary?
No
Yes
Stop

- If more than one load case exists, caution is needed to extend the procedures described to such conditions.