

Introducción a ABAQUS CAE

Método de Elementos Finitos

Máster Ing. de Estructuras, Cimentaciones y Materiales

J. M.^a Goicolea, F. Gabaldón

*Grupo de Mecánica Computacional
Escuela de Ingenieros de Caminos, UPM*

10 de septiembre de 2021



Índice

1 Definición del problema

2 Modelo de EF con ABAQUS

3 Resultados

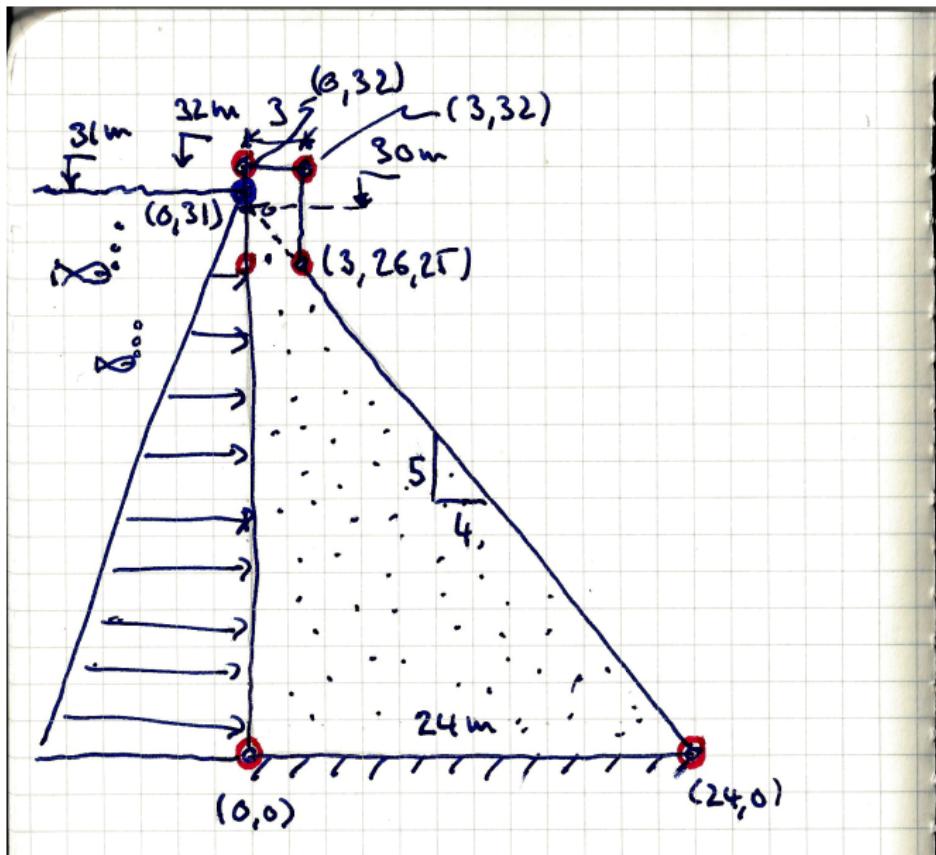
Realización del cálculo

Mallas y mapas de resultados

Resultados numéricos



Presa de gravedad



Formas de crear el modelo de elementos finitos

Opciones

① Interfaz gráfica ABAQUS CAE

- Forma gráfica e interactiva, basada en menús, desplegables, iconos...
- Crea un fichero "modelo.cae" y uno "modelo.inp"
- Permite definir algunos datos con el ratón o de forma numérica
- Permite visualizar el modelo creado
- Permite visualizar los resultados en el mismo marco

② Edición directa del fichero de mandatos (*keywords*)

- Creación de fichero de texto simple "modelo.inp"
- Permite control preciso del modelo
- Adecuado para expertos o que ya conozcan bien el tipo de modelo
- Rápida modificación de parámetros u otros valores
- Modelos paramétricos



Índice

1 Definición del problema

2 Modelo de EF con ABAQUS

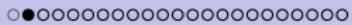
3 Resultados

Realización del cálculo

Mallas y mapas de resultados

Resultados numéricos





Iniciar CAE – Interfaz gráfica

A screenshot of the Abaqus/CAE Student Edition 6.14-2 software interface. The window title is "Abaqus/CAE Student Edition 6.14-2 [Viewport: 1]". The menu bar includes File, Model, Viewport, View, Part, Shape, Feature, Tools, Plug-ins, Help, and ?.

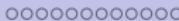
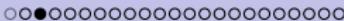
The toolbar contains various icons for file operations, selection, and modeling tools. The main workspace shows a dark blue background with a large, semi-transparent watermark of the SIMULIA logo.

The left sidebar displays the "Model" tab selected, showing the "Model" tree. The tree structure includes:

- Model (1)
 - Model-1
 - Parts
 - Materials
 - Calibrations
 - Sections
 - Profiles
 - Assembly
 - Steps (1)
 - Field Output Request
 - History Output Request
 - Time Points
 - ALE Adaptive Mesh
 - Interactions
 - Interaction Properties
 - Contact Controls
 - Contact Initialization
 - Contact Stabilization
 - Constraints
 - Connector Sections
 - Fields
 - Amplitudes
 - Loads
 - BCs
 - Predefined Fields
 - Remeshing Rules
 - Optimization Task

The right side of the interface shows a large, empty workspace area.

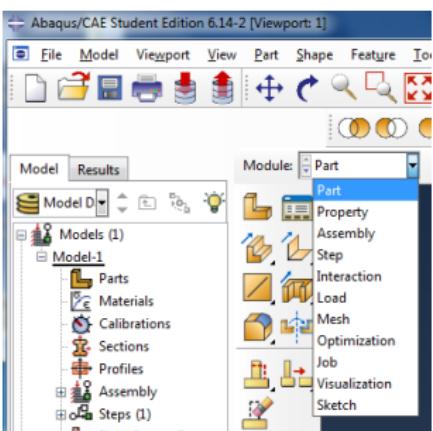
At the bottom left, a message box states: "A new model database has been created. The model 'Model-1' has been created." The bottom right corner features the SIMULIA logo.



Fases (*módulos*) para crear el modelo con CAE

Desplegable Module

- ① Part – Definición geométrica
- ② Property – Materiales, secciones
- ③ Assembly – Colocar, repetir, combinar partes
- ④ Step – Pasos del cálculo
- ⑤ Load – Cargas y condiciones de contorno
- ⑥ Mesh – Malla de elementos y nodos
- ⑦ Job – Ejecución del cálculo
- ⑧ Visualization – Ver y extraer resultados



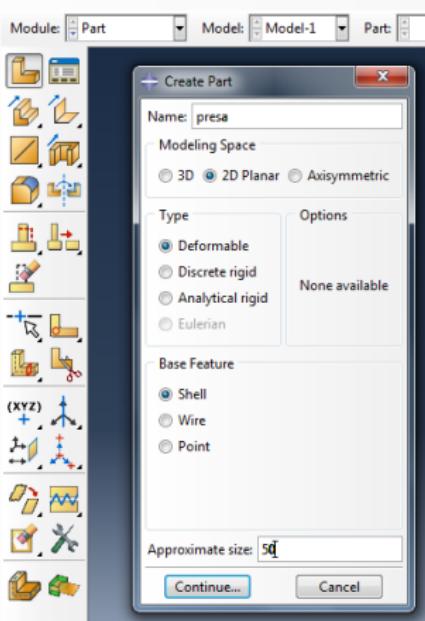
Módulo Part

Entrar en módulo

Part

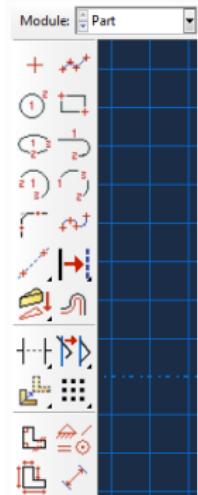


Crear parte (ícono L)



*2D planar, Deformable,
Shell*

Herramientas: Icóno
de poligonal

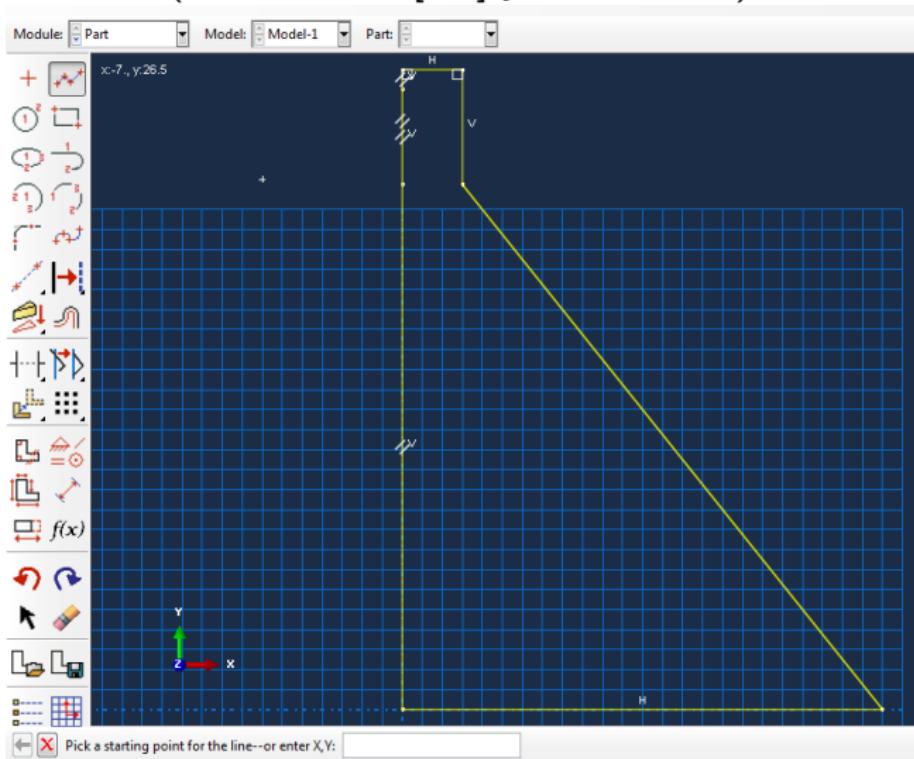


Part

Geometría creada por puntos de una poligonal
(terminar con [esc] y botón *Done*)

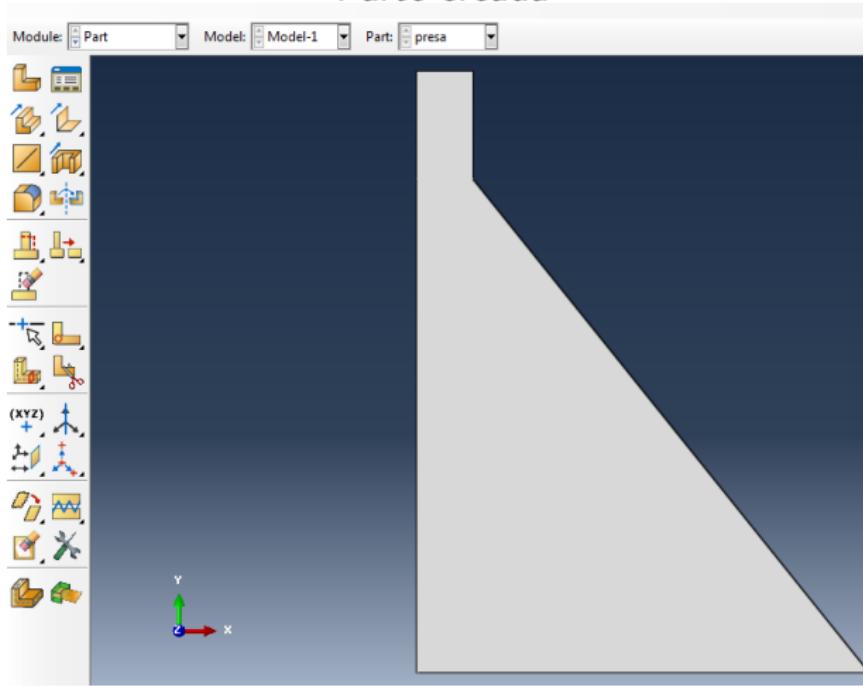
0,0
24,0
3,26.25
3,32
0,32
0,31
0,26.25
0,0

(OJO, punto
como
separador
decimal)



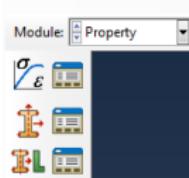
Part

Parte creada



Módulo Property

Entrar en módulo
Property



- Crear material con icono
- Constantes elásticas:
 $E = 30 \text{ Gpa}$,
 $\nu = 0.2$

Propiedades: *Mechanical* → *Elasticity* → *Elastic*

Screenshot of the ABAQUS 'Edit Material' dialog box:

Name: hormigon
Description: hormigon de la presa

Material Behaviors: Elastic

Elastic

Type: Isotropic

Use temperature-dependent data

Number of field variables: 0

Moduli time scale (for viscoelasticity): Long-term

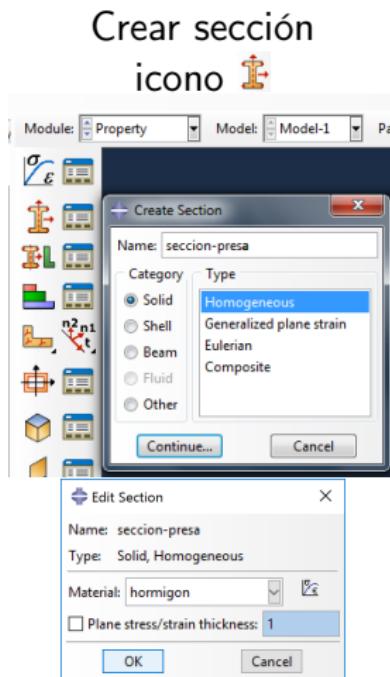
No compression

No tension

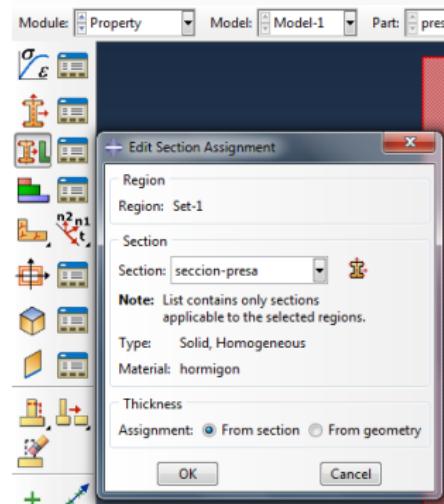
Data

Young's Modulus	Poisson's Ratio
1 30e9	0.2

Property



Asignar sección a la parte icono 



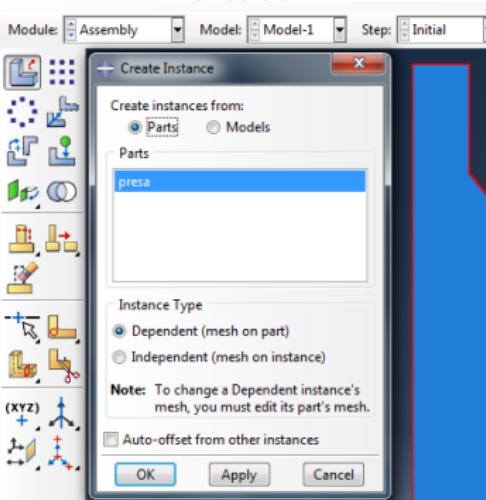
Seleccionar presa para asignar → se colorea verde

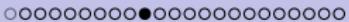
Módulo Assembly

Entrar en módulo



Crear instancia con la parte
creada



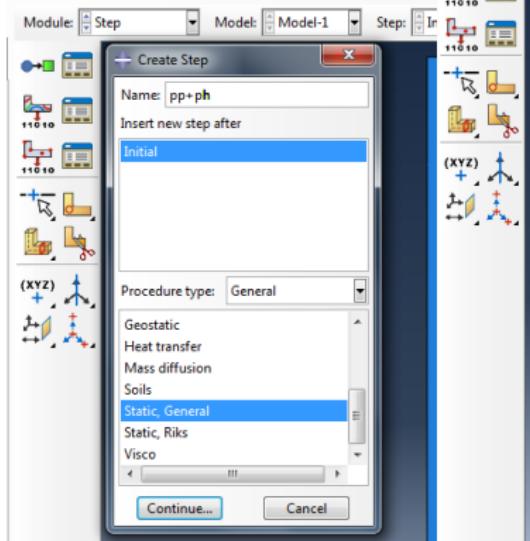


Módulo Step

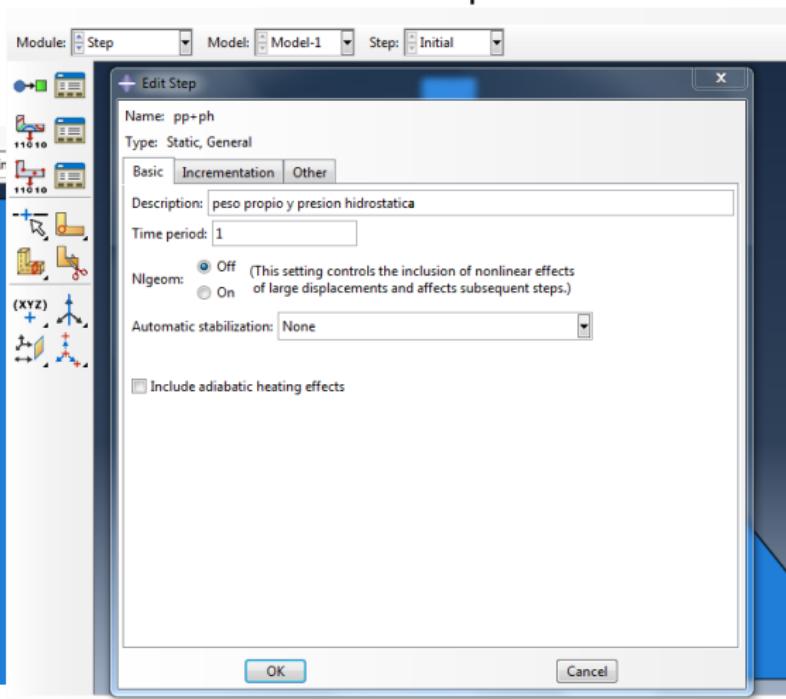
Crear step



Static, General

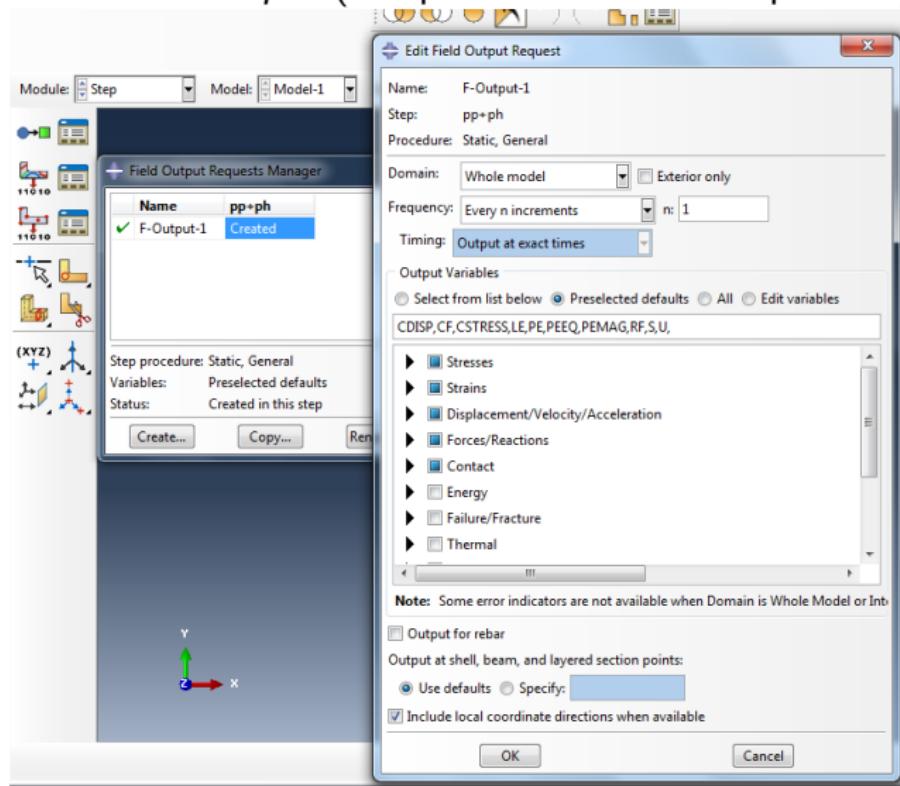


Editar step



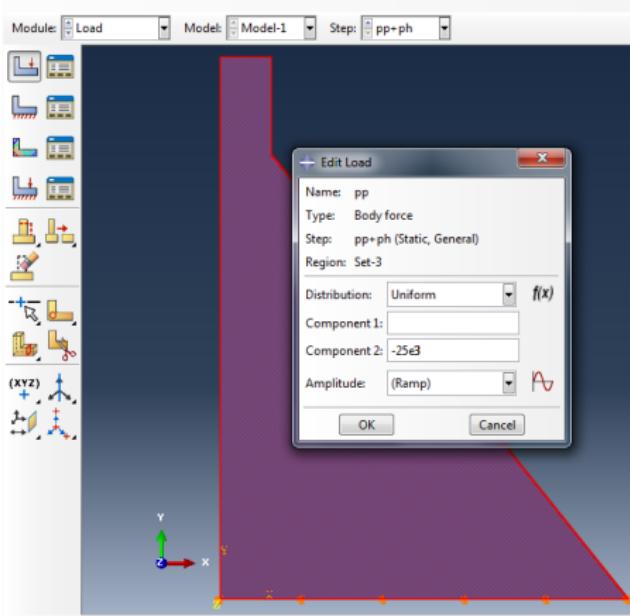
Step

Editar field output (campos de resultados requeridos)

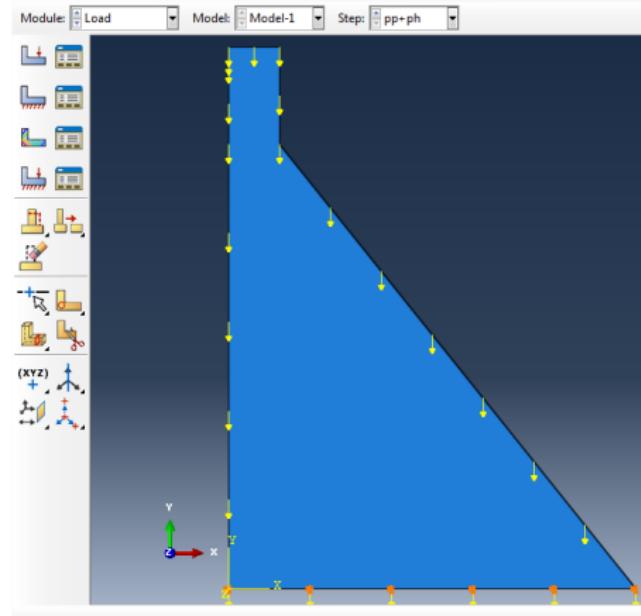


Load

Definir peso propio
ícono  → *Body force*
Seleccionar cuerpo

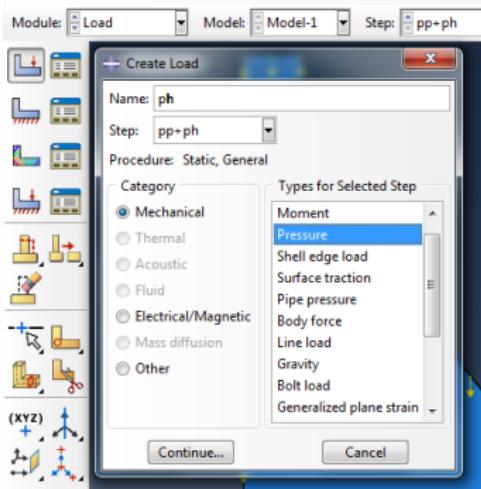


Peso propio creado

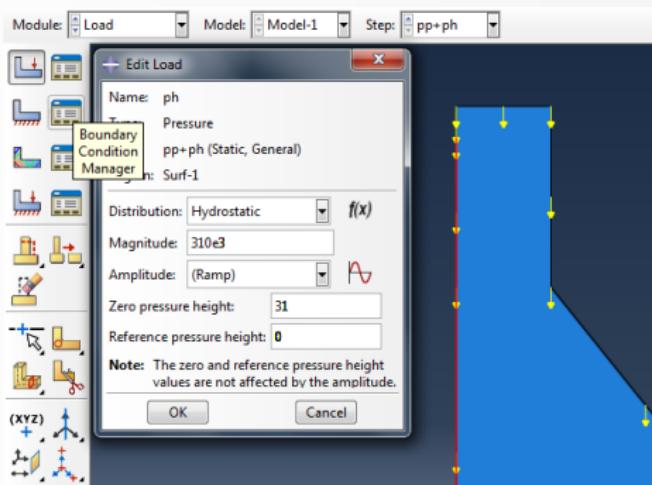


Módulo Load

Definir presión...



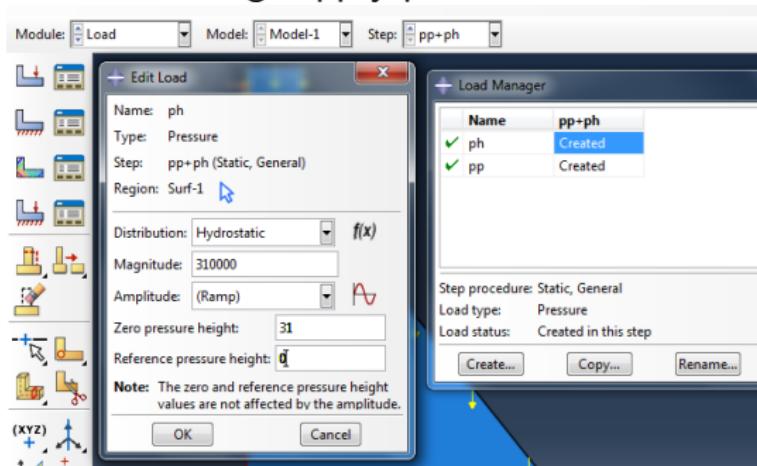
... como hidrostática



La presión hidrostática se aplica en el paramento vertical desde la cota 0 hasta la cota 31, con un valor máximo de 310 kN/m^2 . Se selecciona con el puntero las partes del borde en las que se aplica esta presión, empleando los puntos previamente definidos (0, 0), (0, 26.25) y (0, 31.0).

Load

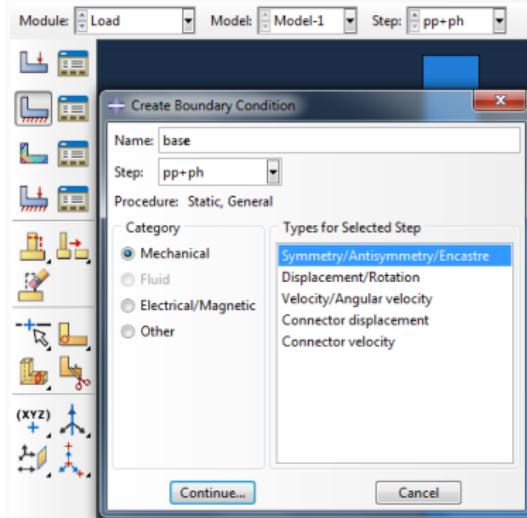
Cargas pp y ph creadas



Load

Crear condición de contorno
ícono 

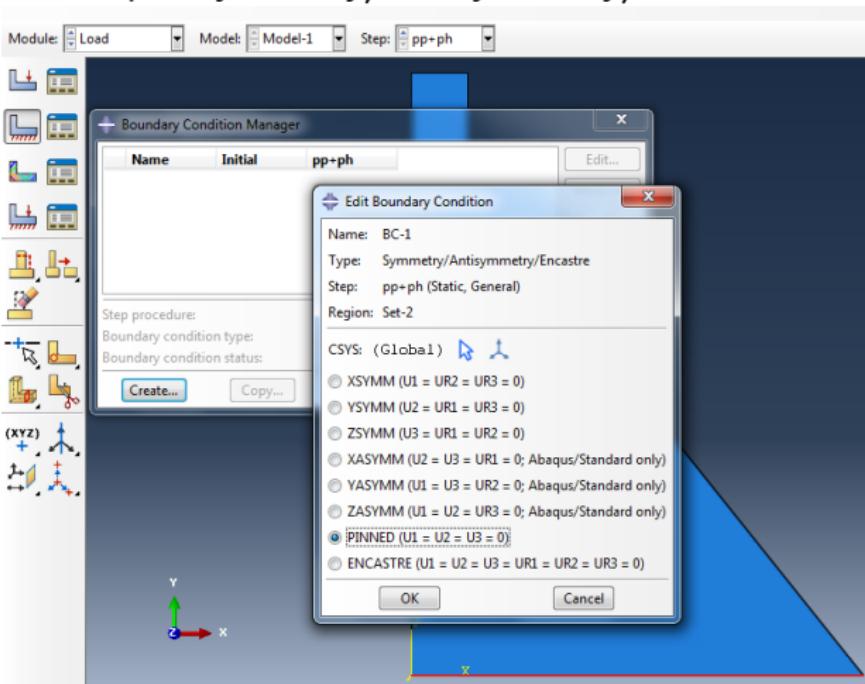
(seleccionar segmentos del borde con el puntero)



Load

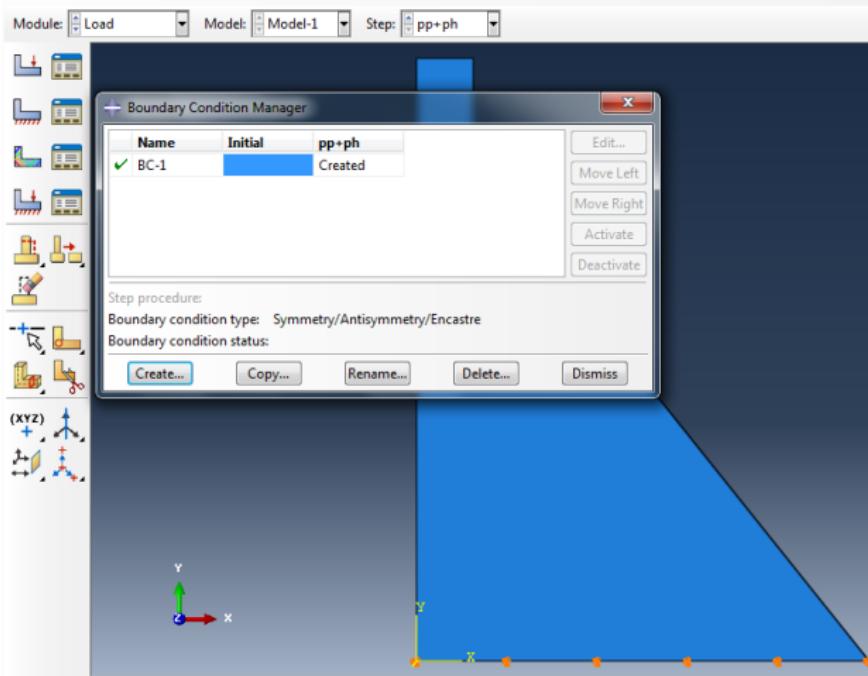
Condición de base fija

Tipo: *Symmetry/Antisymmetry/Encastre*



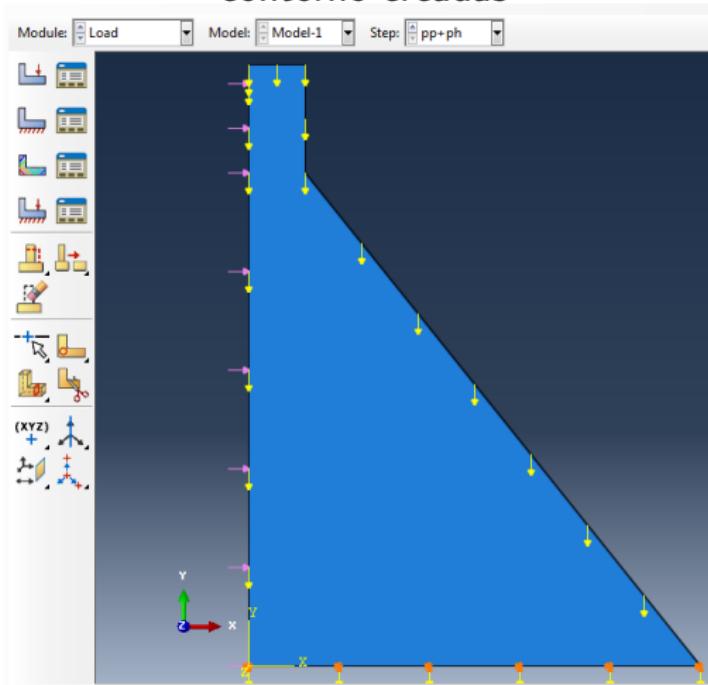
Load

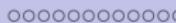
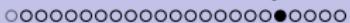
Condición de contorno creada



Load

Cargas de peso propio y presión hidrostática y condiciones de contorno creadas

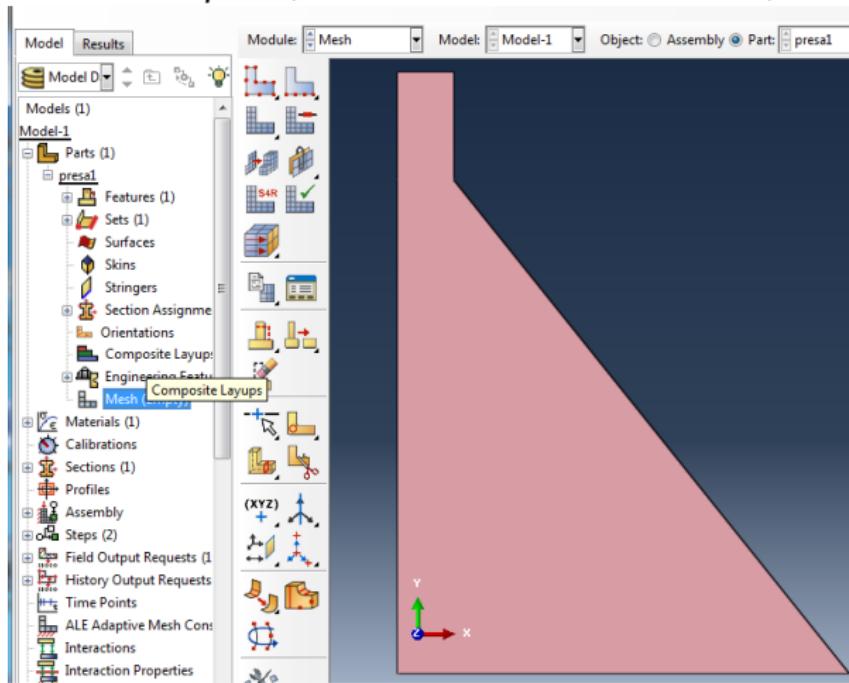




Módulo Mesh

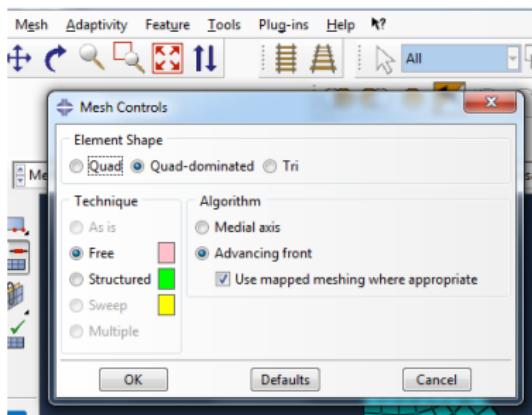
Entrar en Mesh

expandir arbol de parte, seleccionar elemento Mesh, doble-click



Mesh

Menú: *Mesh > controls*



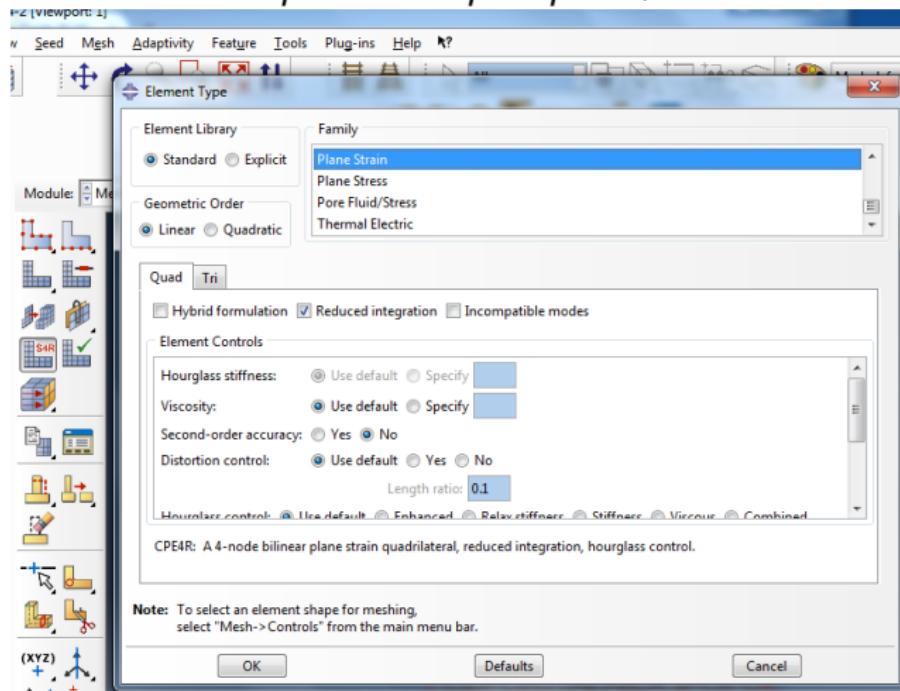
(dejamos las opciones por defecto)



Mesh

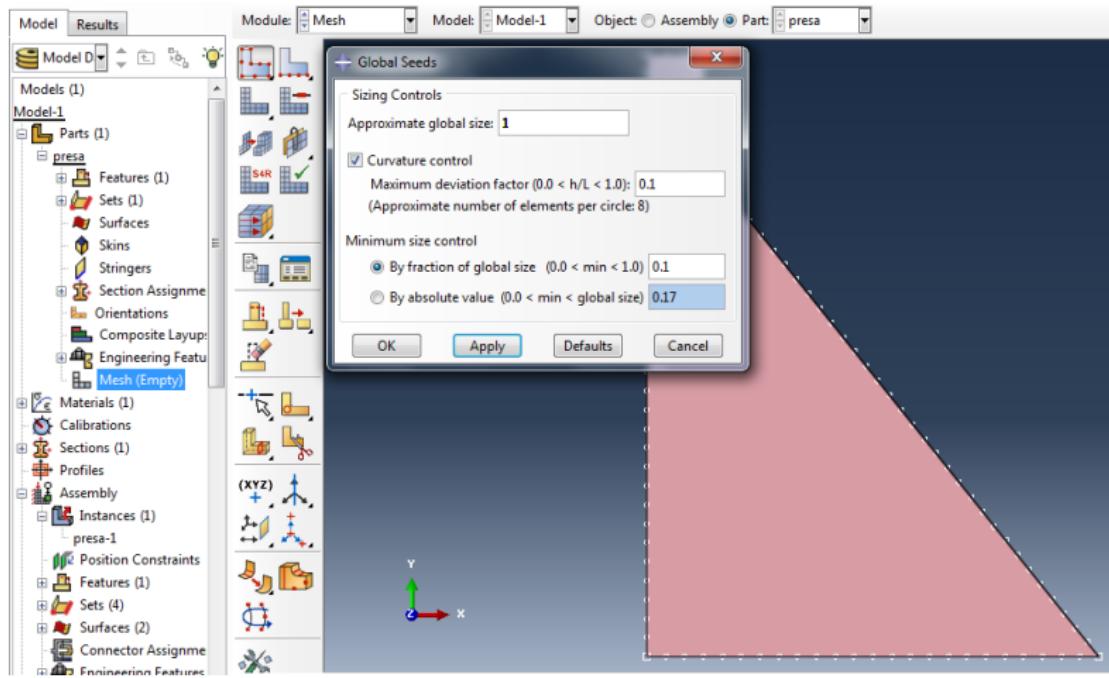
Menú: Seleccionar tipo de elemento *Mesh > Element type > Plane Strain (CPE4R)*

Seleccionar la parte a la que aplica > botón *Done*



Mesh

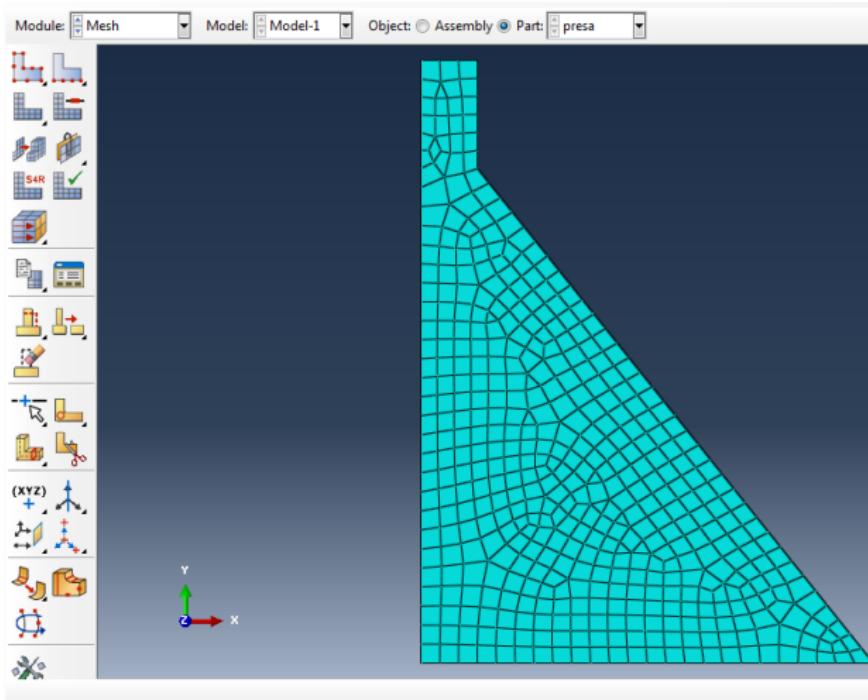
Definir y aplicar semillas en bordes: icono 



Mesh

Crear la malla: icono L

Pregunta: *OK to mesh the part?*



Índice

1 Definición del problema

2 Modelo de EF con ABAQUS

3 Resultados

Realización del cálculo

Mallas y mapas de resultados

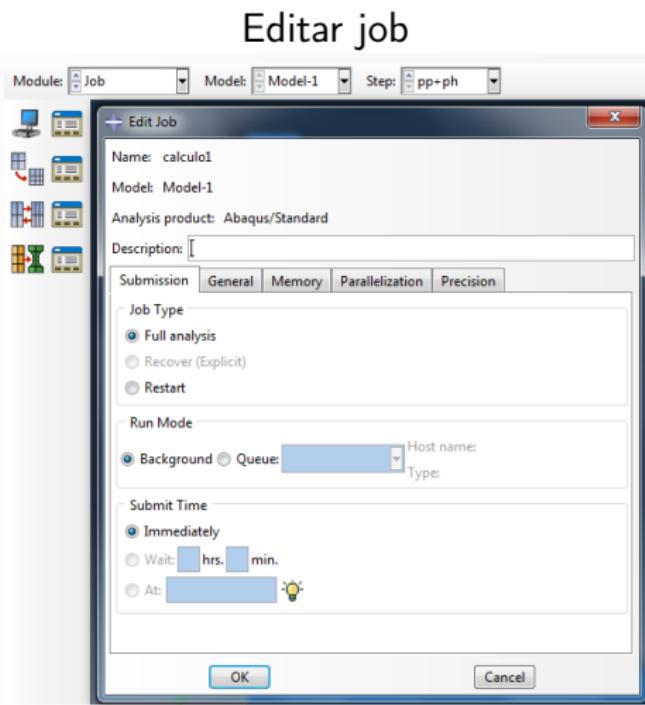
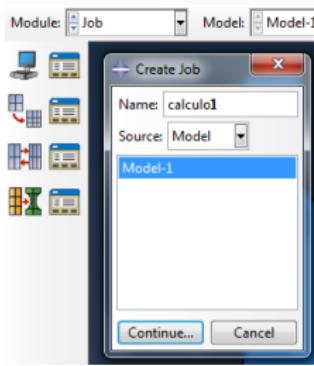
Resultados numéricos





Módulo Job

Crear Job



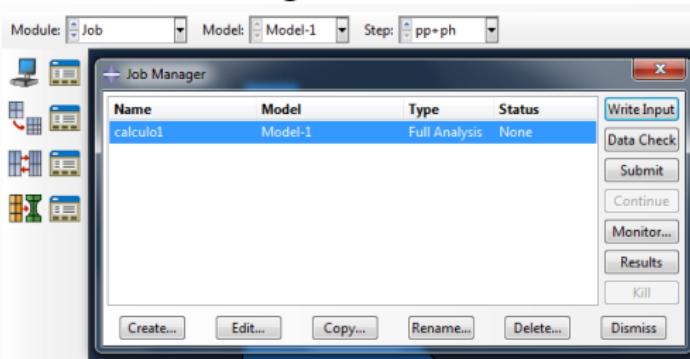
(dejar valores por defecto)



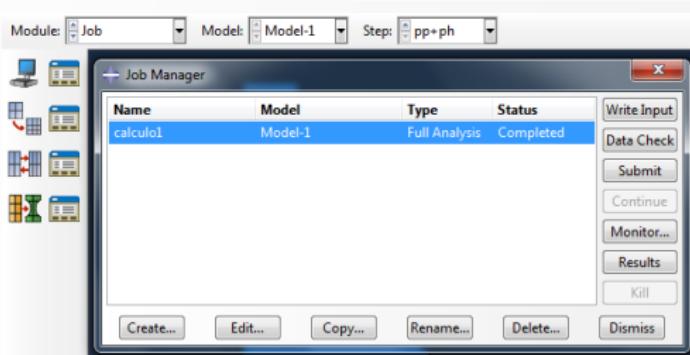


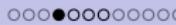
Job

Job manager: botón *submit*

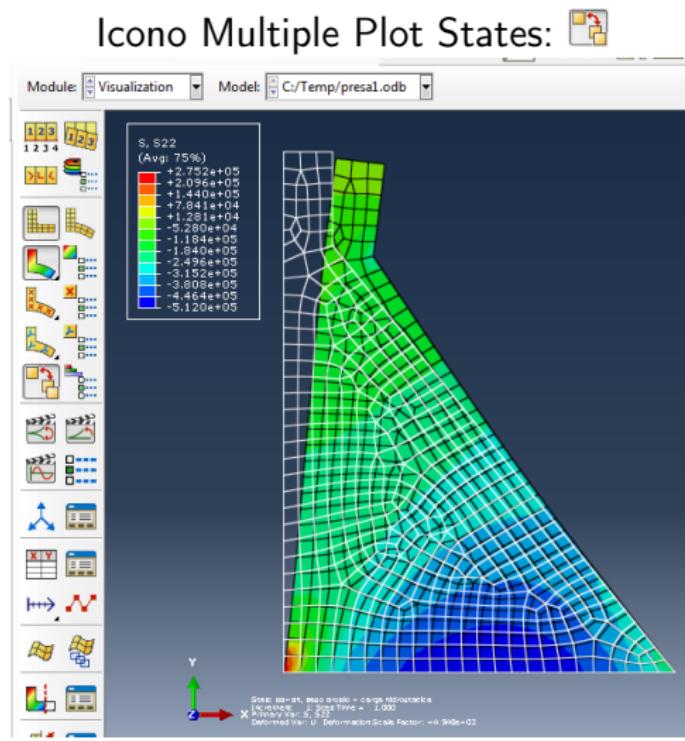


Status cambia: *None* → *Submitted* → *Running* → *Completed*

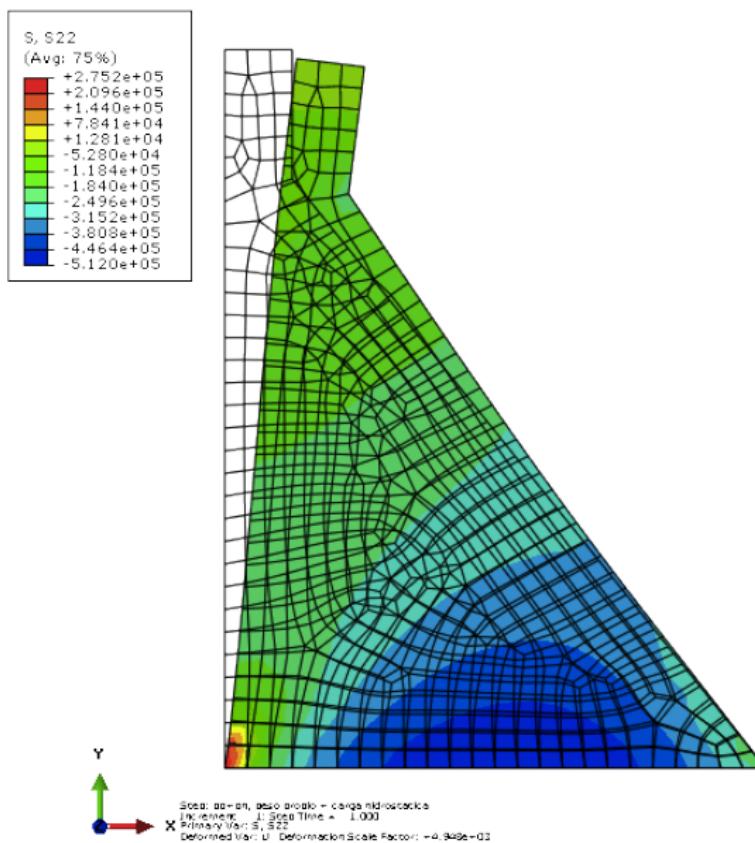




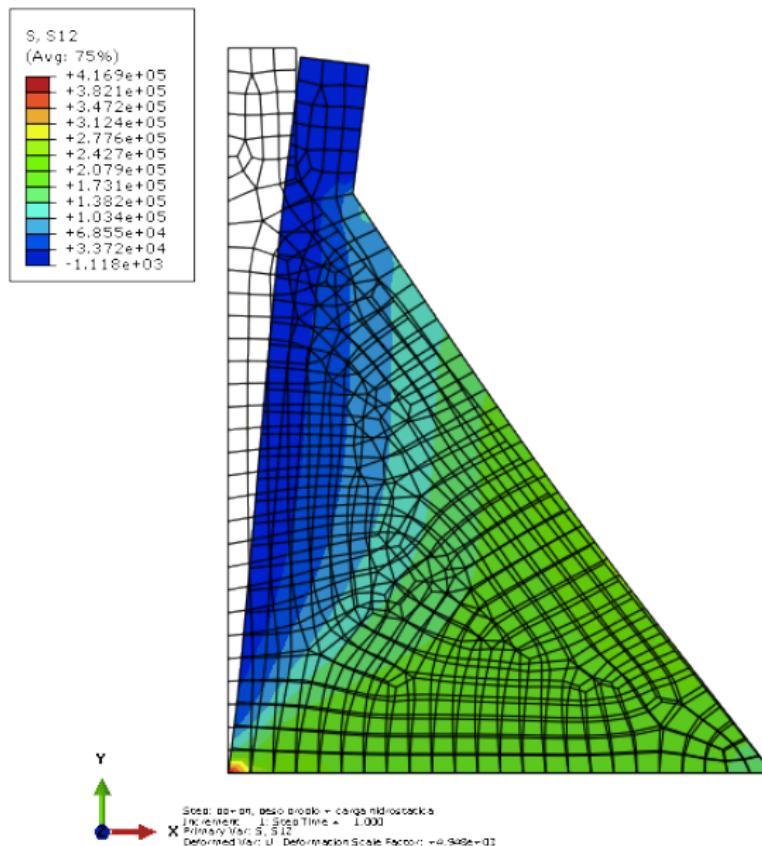
Selección de opciones a visualizar



Tensiones verticales σ_{22}

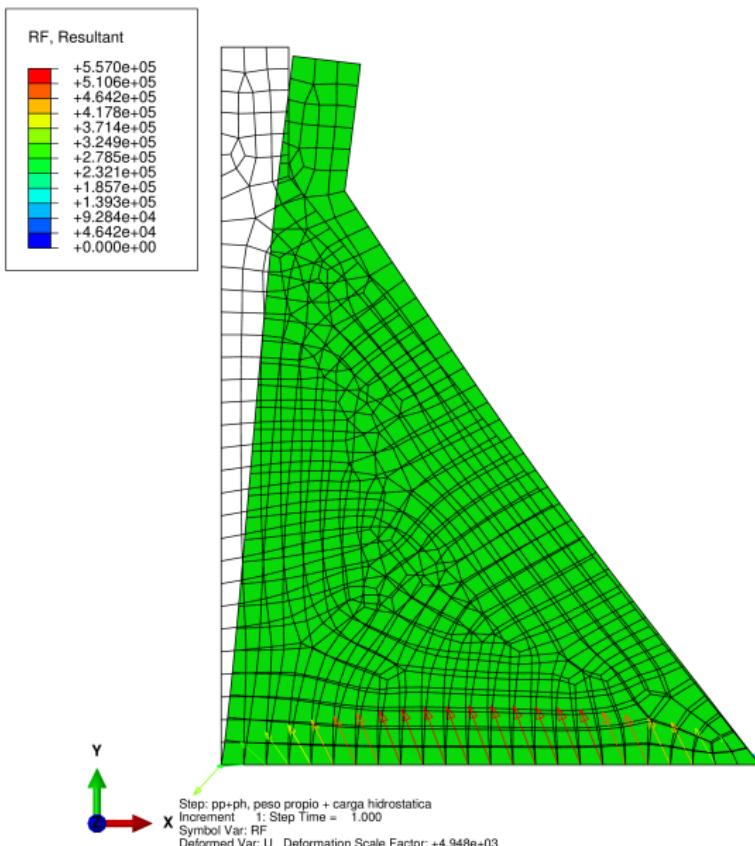


Tensiones tangenciales σ_{12}





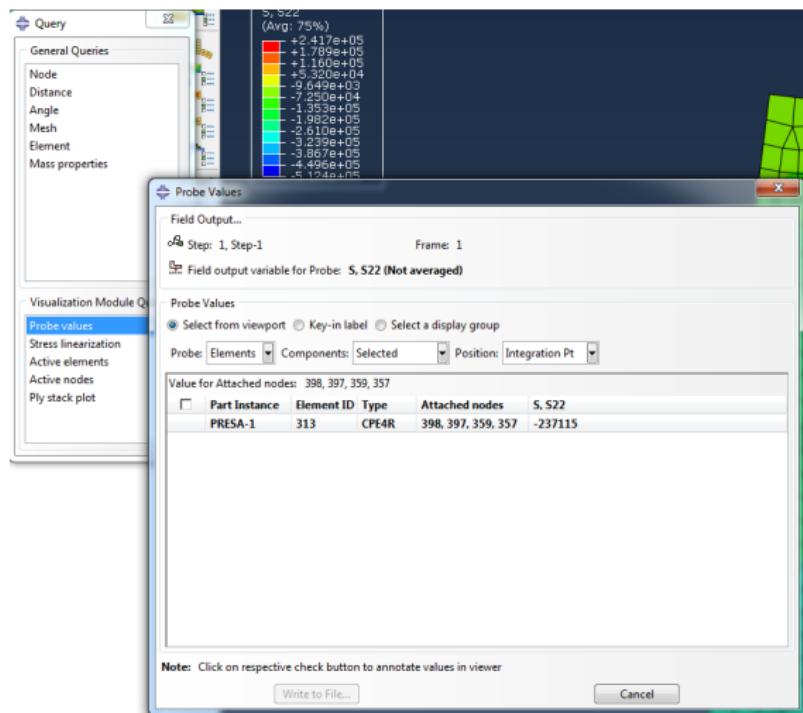
Reacciones en la base RF



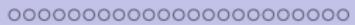
Seleccionar
“Symbol” para
mostrar vectores
con flechas



Datos puntuales: Tools > Query > Probe Values

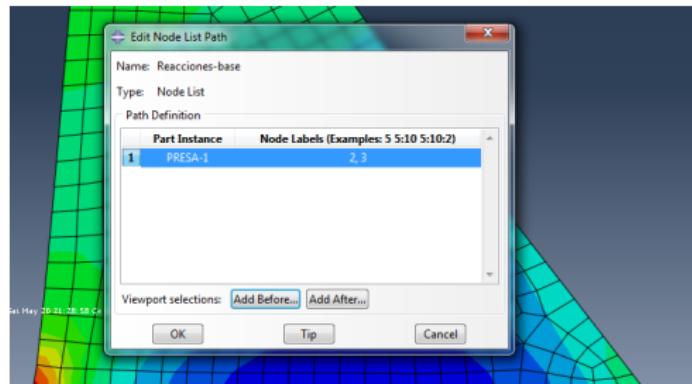
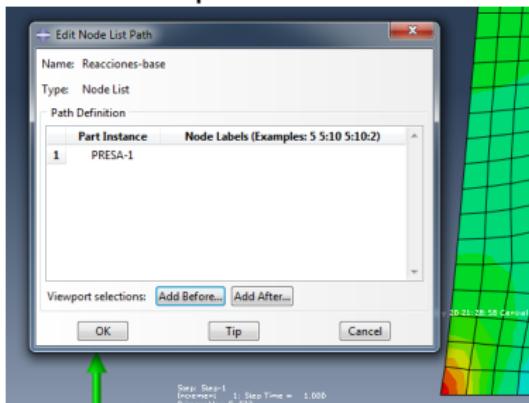
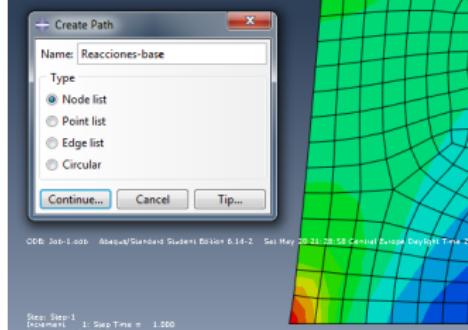


- Seleccionar elementos/nodos individualmente
- Se puede guardar en fichero con *Write to file*

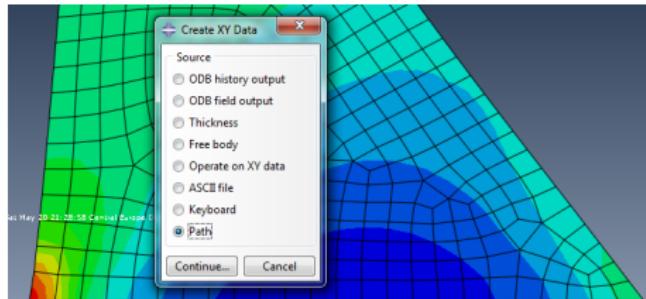


Datos según corte: Tools >Path > Create

Por ejemplo, la base de la presa:



Datos según corte: ícono create XY-data > Plot



OJO: Marcar
Include
intersections

XY Data from Path

Data Extraction

Path: Reacciones-base
Model shape: Deformed Undeformed
Project onto mesh, tolerance: 0

Point Locations:
 Path points
 Include intersections
 Uniform spacing
Number of intervals: 10

X Values:
 True distance X distance
 Normalized distance Y distance
 Sequence ID Z distance

Y Values:
Step: 1, Step-1
Frame: 1 [Step/Frame]
Field output variable: S, S22 (Avg: 75%)
Field Output...
Note: Result option settings will be applied to calculate result values for the current step and frame.

Save As... Plot Cancel

Field Output

Step/Frame
Step: 1, Step-1
Frame: 1 04

Primary Variable	Description (* indicates complex)
RF	Reaction force at nodes
S	Stress components at integration points
U	Spatial displacement at nodes

Invariant
Magnitude

Component
RF1
RF2

Section Points... OK Apply Cancel



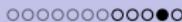


Gráfico según corte: RF2 (create X-Y data)

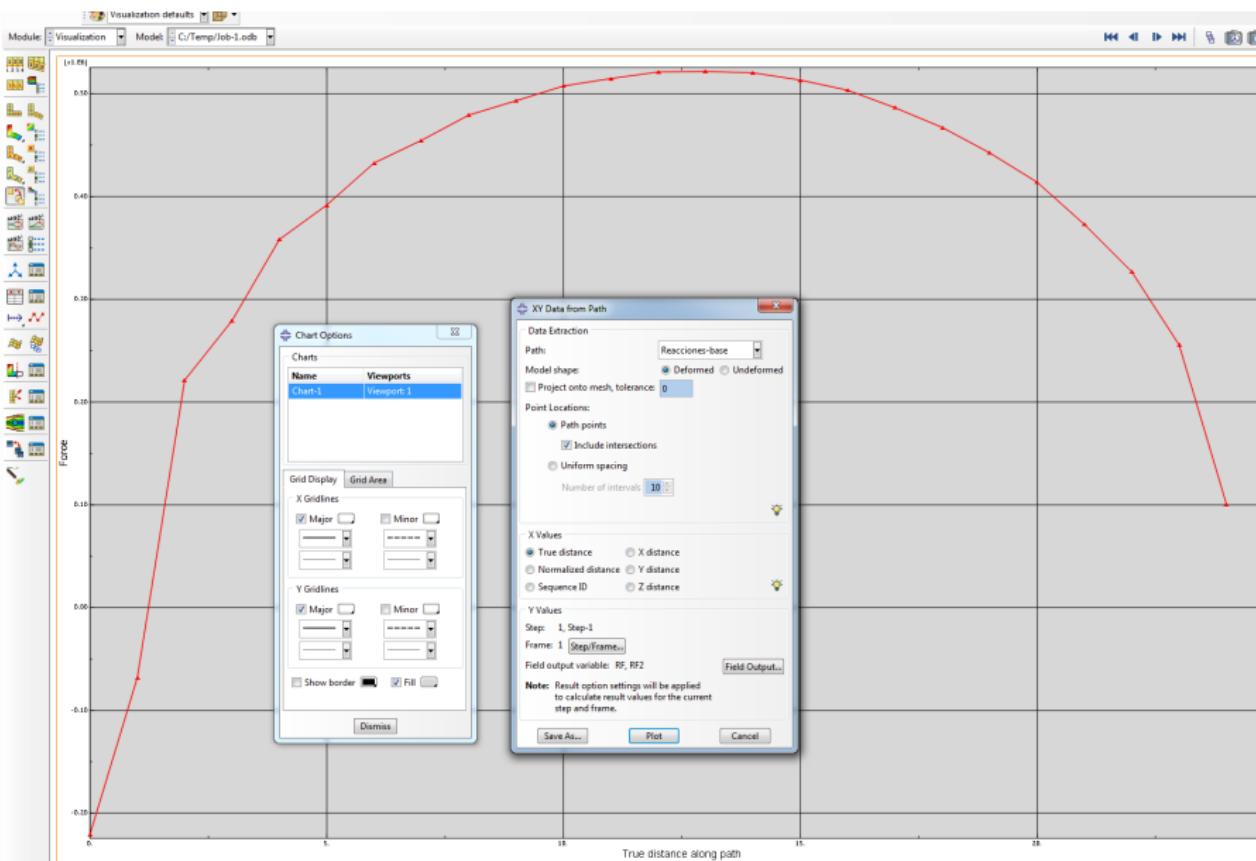




Gráfico según corte: S22

